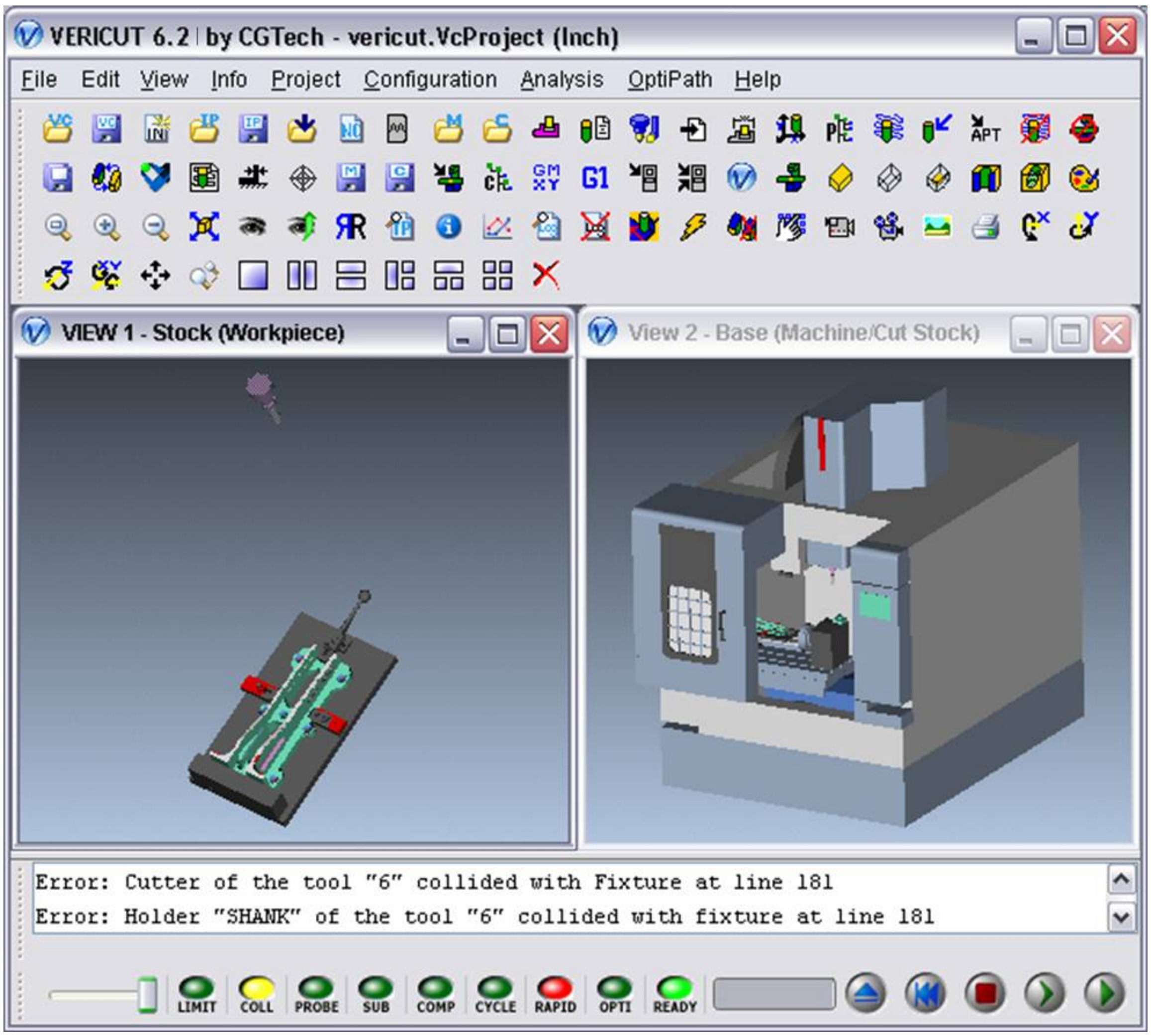
VERICUT HELP



Getting Started with VERICUT

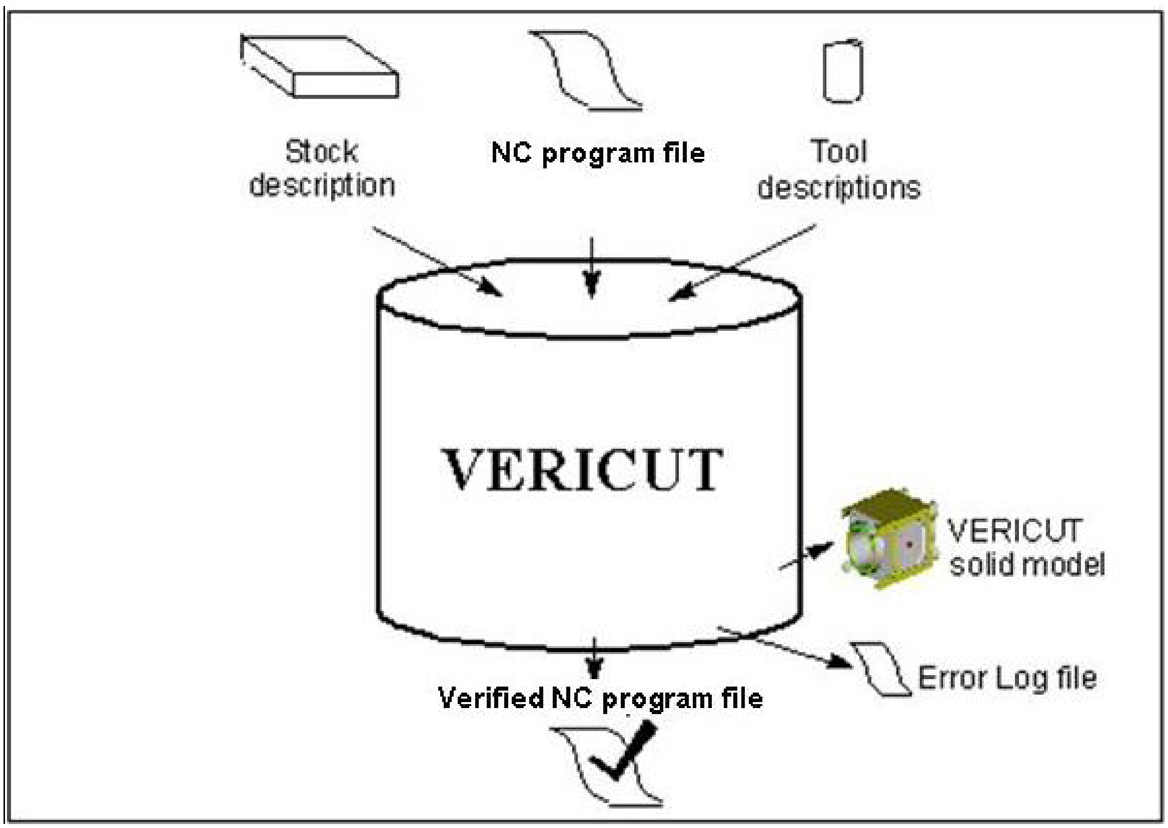
Introduction

Introduction to VERICUT

VERICUT simulates or replaces costly steps of the NC data verification process, plus optimizes the efficiency of material removal. By simulating NC programs, VERICUT can reduce or eliminate the time-consuming and expensive steps of the traditional data prove-outs. The program also optimizes program feed and speeds to increase the efficiency of the machining process. This all adds up to lower costs and higher profits for your business.

Need training? Visit the CGTech web site at www.cgtech.com and click **Training**, in the **Services** pull-down menu, for training course descriptions, schedules, and online registration.

NC program verification process with VERICUT:



VERICUT interactively simulates the material removal process of NC program data. Instead of messy wax or foam models, or costly prove-outs that waste valuable production time, a simulated VERICUT part is graphically produced on your computer screen. VERICUT then verifies the accuracy of the NC program and makes certain the finished part matches the design model. Before any machine time is spent, VERICUT catches discrepancies that could corrupt the cutting process and identifies the responsible NC program records for fast and easy adjustment.

The input into VERICUT is NC program data from almost any source. G and M code data as well as APT type CL-files are directly processed by VERICUT. Similar to the requirements of machining a real part, VERICUT needs the NC program data, a description of the raw stock material to be machined, and descriptions of the cutting tools used to machine the part. The result of the verification process is a solid model of the machined part and an error Log file reporting any machining errors detected during the simulation. The VERICUT model can be inspected, saved, or used as the stock material for another NC program.

In addition to streamlining the verification process, VERICUT dramatically boosts productivity on the shop floor. Automated and user-selectable optimization settings in VERICUT let you maximize the material removal process. The feed rate and cutting speed of each NC program automatically adjust based on the depth, width, and angle of each cut. In fact, you can even set VERICUT to compensate for dull or custom cutters or other elements unique to your operation.

VERICUT simulates up to 5-axis milling, drilling and wire EDM operations, as well as turning and combination mill/turn machining. With a complete line of specialization modules for added features and functionality, VERICUT is truly the total simulation, verification, and optimization solution for unleashing NC productivity.

Also see "**Overview of Using VERICUT**", in the *Using VERICUT* section, in the *CGTech Help Library*.

Installing VERICUT

In most cases installing VERICUT is a straight-forward and easy task, requiring only a few minutes of time. Prior to installing any software, review the system requirements and the installation procedure for the computer type on which you intend to operate VERICUT.

Detailed step-by-step instructions are provided in the installing_cgtech_products.pdf included on the VERICUT software CD, and online in the **Installing CGTech Products** section of the *CGTech Help Library*. Follow all applicable steps to ensure smooth VERICUT installation and successful operation.

Accessing VERICUT

The method of accessing VERICUT is different, depending on the environment in which it is operated. All methods described below are for accessing VERICUT as installed using instructions provided by CGTech. Your VERICUT installation may be different.

Access VERICUT on a Windows computer:

From the Taskbar, click **Start > All Programs > CGTech VERICUT 6.x > VERICUT 6.x** where "6.x" is the desired VERICUT version number.

For example, to run VERICUT 6.2 you would click: Start > All Programs > CGTech VERICUT 6.2 > VERICUT 6.2

Access VERICUT on a UNIX computer:

Type "vericut". If the command is not found, include the full path, e.g. "/<cgtech6x>/<computertype>/commands/vericut" where <cgtech6x> represents the directory path to where CGTech products were installed, and <computertype> is the type of computer on which VERICUT is being operated, for example: hp, rs, or solaris.

For example, to run VERICUT installed in "/server/apps/cgtech62" on an HP700 computer, you would type: /server/apps/cgtech60/hp/commands/vericut

Hint: Copy the "vericut" command file to the /usr/bin directory and the next time you log in, you should be able to just type "vericut".

Access VERICUT from CATIA:

Assuming you have CGTech's **CATIA-to-VERICUT Interface** installed, choose the "**CATV**" function. Use CATV functions to select the CATIA data to use in the verification session (geometry models, NC programs, etc.) and transfer the data to VERICUT. Then access VERICUT via choosing **PROCESS menu > VERICUT**.

Access VERICUT from NX:

Assuming you have CGTech's **NX-to-VERICUT Interface** installed, access VERICUT from NX as follows:

Click on the **Verification** icon in the NX CAM icon bar.

Access VERICUT from Pro/Engineer:

Access VERICUT from ProE as follows:

Click **NC Sequence Play Path menu > NC Check** => simulates the current selected NC sequence.

Click **CL Data menu > NC Verification** => enables users to select a single NC sequence, multiple sequences, or an entire NC operation for simulation.

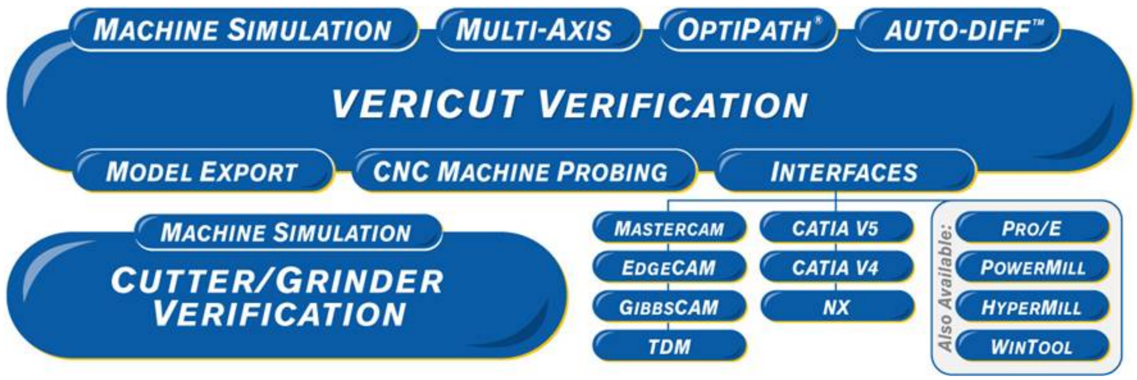
NOTE: The NC data must be selected prior to selecting starting the verification process.

Access VERICUT from Mastercam-to-VERICUT Interface:

Access VERICUT from Mastercam as follows:

Hold down the Alt key and click the C key (referred to as **Alt-C**). You will see a file selection dialog offering all the installed C-Hooks. Pick "**VERICUT.dll**" and click on the **Open** button.

VERICUT Licenses and Options



VERICUT licenses and capabilities differ, depending on the environment in which VERICUT is being operated. A modular approach provides the flexibility to purchase only the capabilities you need. As your needs change, you can add the appropriate licenses to increase VERICUT's functionality. The terms "licenses" and "options" are synonymous, since each option for VERICUT is licensed by CGTech. It is not necessary to install any additional software to add an option. CGTech provides a license that allows immediate access. See Product Line, on the CGTech website, for complete details on available options.

CGTech software licenses allow you to share VERICUT over a network. Only one user may access a given license at a time. CGTech's license system supports Windows 2000, XP, XP64, HP, RS and Solaris. See VERICUT System Requirements, on the CGTech website, for more detailed information. The server and networked workstations can be different computer types.

Licensing for VERICUT

VERICUT 6.2 licensing is required. Pre-6.2 licensing is incompatible with the 6.2 CGTech Network License Server. If you do not have a 6.2 license, obtain licensing from CGTech as described below.

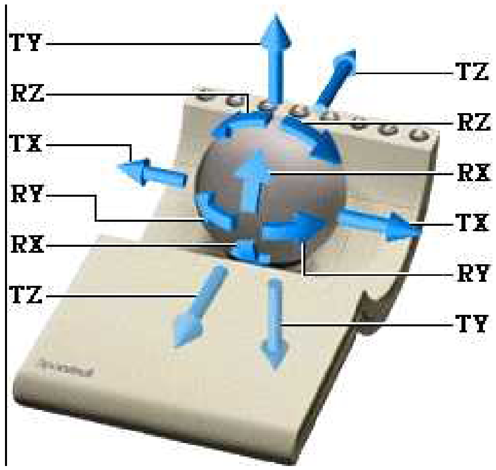
6.2 licenses are issued via email only. Your 6.2 license will be emailed to the primary VERICUT user at your company. If you do not have licensing, submit an application via CGTech website.

NOTE TO NEW CUSTOMERS — You will need to know the machine ID (ethernet address) of the machine on which the CGTech Network License Server is to operate. To determine the machine ID, see the steps described in the **Installing CGTech**

Products section of the *CGTech Help Library*. (ref. **Determine Your Host ID and Request a License**).

VERICUT with SpaceBall/SpaceMouse

VERICUT now supports 3Dconnexion 6 axis input devices (SpaceBall and SpaceMouse) on Windows. Use of these devices requires a driver known as 3DxWare.



These devices have at least 8 buttons, some of which we have designated as follows:

1. Toggles translations on and off.
2. Toggles rotations on and off (you can't have both of these off together, so turning one off will turn the other on if necessary).
3. Toggles the dominant axis filter on and off (when on, only one axis moves at once).
4. Fit.
5. Decreases sensitivity.
6. Increase sensitivity.
7. Reverse
8. Refine

With the exception of buttons 4, 7 and 8, these functions are a de-facto standard for software that supports the devices. PTC uses button 4 as "Fit" in Pro/E. A configuration file (VERICUT.scg) is required and is accessed by the 3DxWare driver on startup. This file is distributed by 3Dconnexion with the most recent version of their driver.

While beta testing has shown that VERICUT generally works correctly with various older models and older versions of the device drivers, CGTech recommends using the latest driver version. It can be downloaded from www.3dconnexion.com. The VERICUT configuration file was added to 3DxWare version 2.3 build 5, September 4, 2002.

NOTE: 3Dconnexion motion control devices currently work with VERICUT on Windows platforms only.

Performance Considerations

This section describes ways to help increase VERICUT productivity.

System considerations

Computer speed — VERICUT runs faster on faster computers. When comparing computers made by different manufacturers, you can get a general idea how VERICUT will perform by comparing the processor speed for floating point computations.

Multiple processors — VERICUT runs 10-20% faster on computers with dual processors over a comparable computer with a single processor.

Memory — The amount of memory available to VERICUT can make a difference. See "**VERICUT System Requirements**" for memory recommendations. If you intend to process large tool paths, complex cutting operations, or use **AUTO-DIFF**, you may need more memory. VERICUT uses virtual memory (hard disk) when physical memory (RAM) is exhausted. Best performance is obtained when sufficient physical memory is available to VERICUT. Excessive memory does not make VERICUT run faster, however, insufficient memory will slow it down.

Running concurrently with other applications — Even though your system may run VERICUT or other large applications (e.g. CAD/CAM packages) comfortably, running them concurrently may cause competition for resources: memory, disk space, etc. Under these conditions the computer begins "paging" or "swapping" memory (a.k.a. "disk thrashing"), which adversely affects performance. Symptoms of disk thrashing are: constant hard drive activity or noise, active applications are slow, "Out of memory" error messages or warnings appear. In the worst case you can run out of all available memory which can cause your computer to crash or hang. When determining the amount of memory the system needs to run your applications, be sure to consider the memory requirements of ALL applications that will be run concurrently.

Graphics cards — VERICUT uses 2-D raster (pixmap) graphics. It does not use triangle shading (such as with a 3-D API like OpenGL or Direct3D), texture mapping, or much (if any) vector drawing. All graphics cards (even the cheapest) do pixmap graphics. The card needs enough display or video memory to drive the display resolution at the desired number of colors. For example, 8 mb of display memory may be enough to do a 1024 x 768 display at True Color, but may only be able to do 64k colors at 1600 x 1024 resolution. VERICUT 5.0 or higher performs best in a True Color display environment. More expensive does not necessarily mean "better for VERICUT". Lower cost cards frequently outperform expensive cards when running VERICUT.

Tolerances

Cutting Tolerance — The "**Cutting Tolerance**" value (ref. **File menu > Properties, Tolerance tab**) controls the accuracy of cuts applied to the workpiece. This value directly affects the speed and quality of the results from comparing or exporting models, optimizing tool paths, checking for holder collisions, "Fast Feed" errors, dynamic rotation speed, and zoom speed. A larger cutting tolerance value makes VERICUT go faster, but the resulting cut model and collision checking will be less accurate. A smaller cutting tolerance value may slow the simulation, but provides the high level of accuracy needed to perform detailed model analysis or export, optimization, etc.

- A larger cutting tolerance value is probably okay for quick visual checks, or when only X-Caliper will be used to analyze the model (no AUTO-DIFF).
- Use a smaller cutting tolerance value if you expect to use AUTO-DIFF, Model Export, OptiPath, or examine very small model details.

How VERICUT uses Cutting Tolerance to determine model accuracy — The "**Base Cut Tolerance on**" option on the **File menu > Properties: Tolerance tab** controls how VERICUT applies the **Cutting Tolerance** value to set cut model accuracy.

- Choose "**Stock Size**" to achieve consistent performance on similar sized parts, regardless of the variety of cutting tools used. However, cuts made by very small cutters or cutters with complicated shapes should be evaluated closely to determine if sufficient accuracy has been maintained.
- Choose "**Tool Size**" to consider sizes of cutting tools in addition to the stock model size, or when optimal results must be achieved for all cutting tools. Be aware that with this option, very small cutters such as tiny drills, scribe end mills, etc. can severely impact overall performance.

Model file tolerances — Model files, such as Stereo lithography (STL) or VERICUT model files, exported from CAD systems should be created using tolerances appropriate for how they will be used in VERICUT. For example, models that will be used as stock workpieces, holding fixtures, or NC machine components probably do not need to be as accurate as models that will be used by AUTO-DIFF to check for gouges or excess material. Outputting CAD models with excessively small tolerance values can result in very large model file sizes that cause VERICUT to use large amounts of memory. Similar considerations apply to tolerances for converting IGES model files that will be used in VERICUT. Assuming you have enough memory, model file size/complexity does not affect cutting speed in VERICUT.

Other tips and tricks

Axes display — Axes displayed in the graphics area (e.g. via **View menu > View Axes** or right-click shortcut menu) slow cutting speed. For best performance clear all axes before cutting.

Fixture display — Fixtures displayed in the "Workpiece" view slow cutting speed. For best performance, set Visibility to Machine View on the Component Attributes tab of the Modeling window. (e.g. via **Project menu > Setup Models > Define: Component Attributes tab** or double-click on the fixture component in the Component Tree)

Skip Cut and Animation Speed control — These features on the **Project menu > Processing Options > Motion** window directly affect cutting speed. Skipping cuts can make VERICUT run faster by updating the display less frequently, especially when multi-axis cuts, large cutters, or tools with holders are involved. Also, for best performance ensure the **Animation Speed** slider, located at the bottom of the VERICUT main window, is positioned as far right as it will go ("Fast").

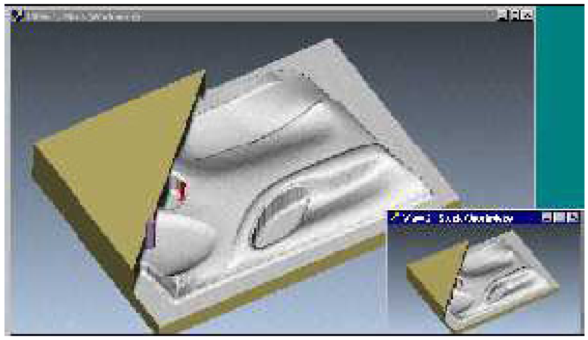
No Animation — This feature on the **Project menu > Processing Options > Motion window** enables you to reduce processing time. When **No Animation** is toggled "on", the graphics display is not updated until either processing is complete or you **Stop** the processing. At that time the cut model is displayed in it "final" state or the state that it was in when processing was stopped. You can also toggle **No Animation** On/Off using the **No Animation** icon, , in the VERICUT toolbar.

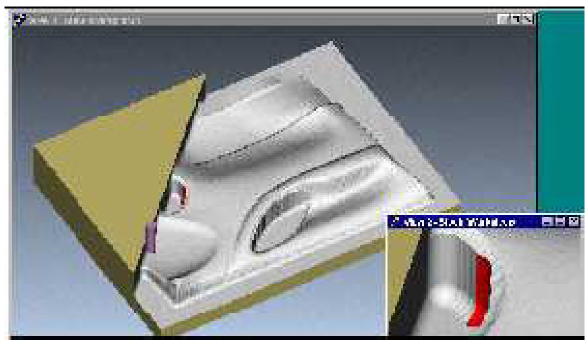
Tool holders — Using tool holders enables VERICUT to detect when non-cutting portions of the tool assembly remove material or crash into clamps/fixtures holding the workpiece. You can minimize this effect by modeling only portions of the tool assembly that can be involved in a collision. For example, don't bother modeling the taper of a milling tool holder (portion that seats inside the spindle). If you don't have concerns about tool holder collisions, VERICUT will run faster if tool holders are not displayed, e.g.: **Project menu > Processing Options > Motion**: clear the "**Display Holders in Workpiece View**" checkbox.

Status and NC Program windows — Opening these windows via the Info menu provides valuable information about the machining process. However, since VERICUT updates them with each tool path record processed, having them open while cutting slows performance. For best performance, close them before cutting. You can reopen them later, as needed for reference.

Address unsupported codes in G-Code tool paths — When processing G-Code tool paths, "unsupported" codes cause messages to be written to the VERICUT Log file and the VERICUT main window message area. Reporting numerous unsupported codes can slow performance dramatically. Avoid writing "false" errors and the corresponding time loss by addressing unsupported codes via the NC control configuration. To ignore codes that are unimportant to the simulation, use **Configuration menu > Word/Address** to add groups that will have unsupported codes call the macro, **IgnoreMacro**.

Faster zooming — Following is a technique to improve the zoom and display speed, and eliminate the post zoom delay between pressing **Play** and resuming cutting. Create a second, smaller view window (**View menu > Layout > Add View**) and position it to the side. Cut the part. When you want to see an area in more detail, use the smaller window for zooming. This protects the refined display in the larger window. Because VERICUT needs only to update the smaller image, you get zoom images much faster than performing the operation in the larger window. Also, when you restart the simulation there is virtually no delay before VERICUT starts cutting again.





Disable Report — This feature on the **Analysis menu > AUTO-DIFF window: Settings tab** enables you to turn off the **AUTO-DIFF Report** feature to reduce AUTO-DIFF processing time. For large NC program files, generating the report can take a significant amount of time. When toggled "off", you will still see the AUTO-DIFF results in the graphics area.

Want more VERICUT tips and tricks from the experts? Visit the CGTech web site at www.cgtech.com and click "*SUPPORT PAGE*", then look for "*Tips and Tricks*".

VERICUT Files

Summary of VERICUT File Types

The sections that follow provide information about the files VERICUT uses and typical file extensions.

Types of VERICUT files

Files provided by CGTech to assist with simulating NC program and machine tool motions can be divided into two categories:

Library files — General purpose files for configuring VERICUT to simulate NC program and machine motions. Library files are included in every installation, and include:

VERICUT default files — Default files opened by VERICUT. The files are configured to perform a short simulation when Play is pressed. Two sets of files provide demonstrations for inch (vericut.VCProject) and metric (vericutm.VCProject) environments.

VERICUT initialization files — Initialization files opened by VERICUT via File menu > New Project. Two sets of files provide initialization for inch (init.VCProject) and metric (initm.VCProject) environments.

Machine and Control files — NC machines and controls intended to help you configure VERICUT for simulating G-Code NC programs that run on popular NC machining centers. See "**Library Machine files**" and "**Library Control files**" for details.

For information about these files, see "**Library Files**".

Sample files — Files used to demonstrate VERICUT capabilities and options. Sample files are included with CGTech software on the CD, and are available to all users who choose to load them. For information about these files, see **Sample-Demo Files** in the *CGTECH Help Library*.

VERICUT File Descriptions

The types of files used by VERICUT to load and save various types of data are listed below in alphabetical order. See the appropriate section for a description of file contents and formats, and how to open and save them.

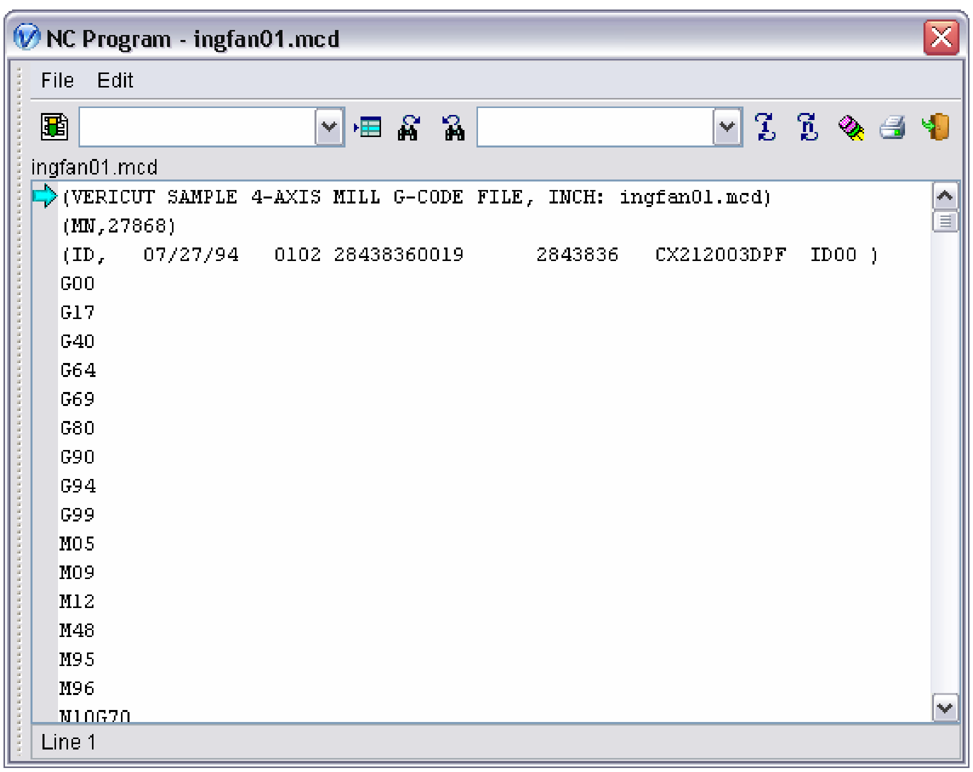
APT Output File

Typical file extension: .apt

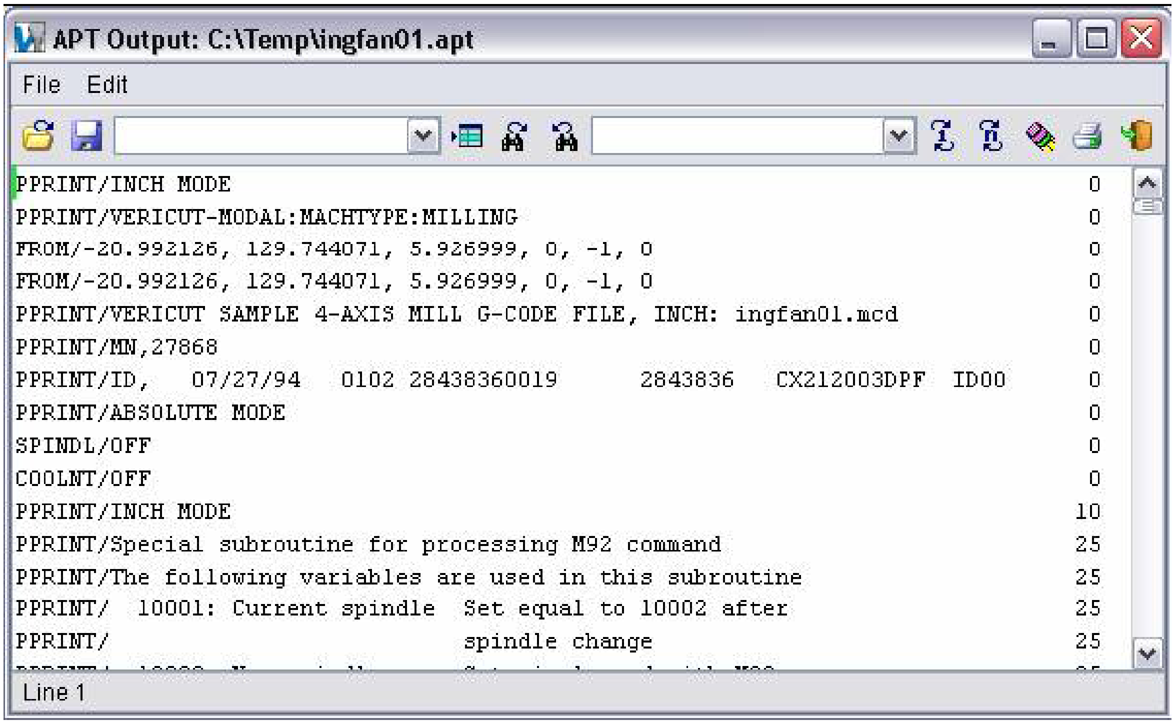
An ASCII text file containing APT NC program records resulting from reverse post-processing by VERICUT. By default when sequenced G-Code data is converted, columns 73-80 in the output file indicate the block numbers of the G-Code data block where the converted records came from. Sequence number output is controlled via the Output Block Sequence Numbers feature on the **Project menu > Processing Options > G-Code > Process Options: Messages tab**.

An APT Output file is created when a G-Code NC program is processed with **Create APT Output File** feature on the **Project menu > Processing Options > G-Code > Process Options: G-Code Output Files tab** is active. You can view this file using the **View APT Output File** feature on the **Project menu > Processing Options > G-Code > Process Options: G-Code Output Files tab** or the **View** feature, under **APT Output File**, on the **Project menu > Output: G-Code Files tab**.

Sample G-Code NC Program file:



Sample of resulting APT Output file:



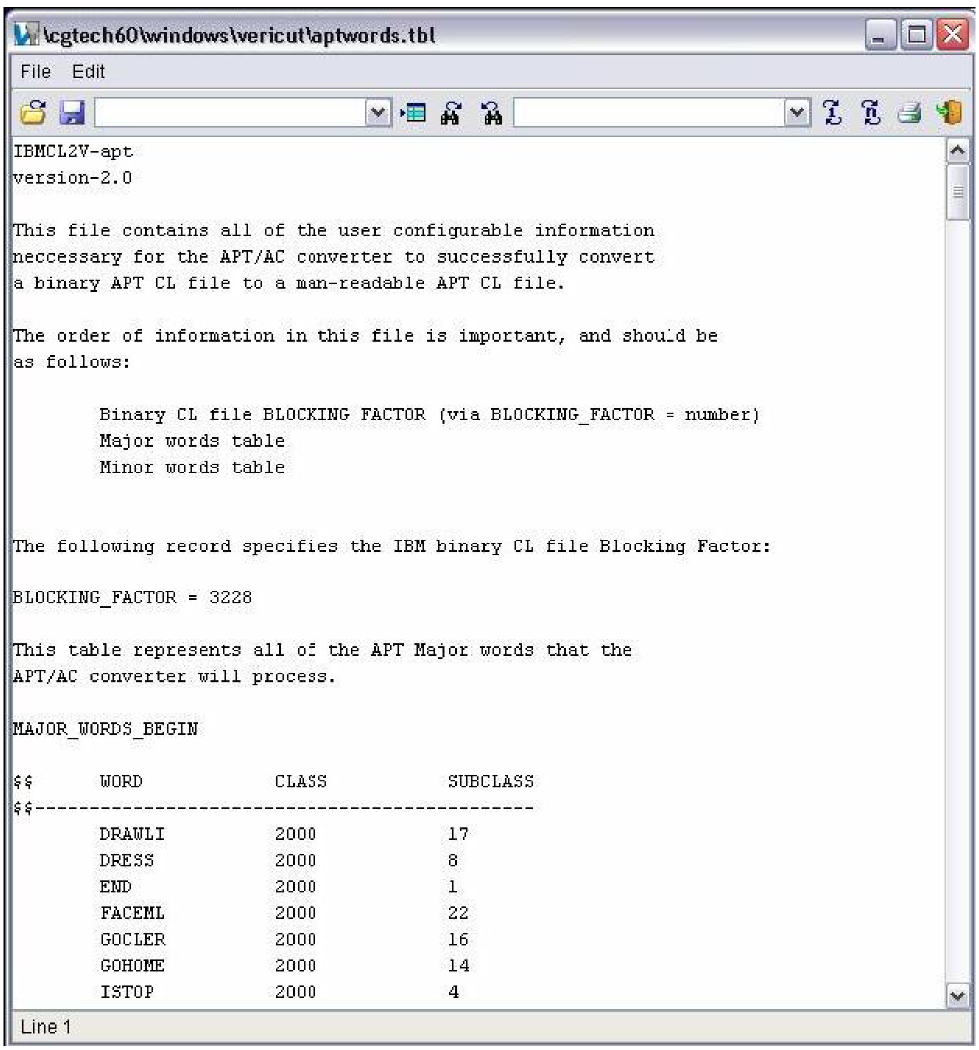
APT Table File

Typical file extension: .tbl

An ASCII text file used by the Binary CL Converter that contains information necessary to convert binary APT data into ASCII APT NC program records. Such information includes: blocking factor, major and minor word classes, sub-classes, etc.

APT Table files are loaded into VERICUT via the **File menu > Convert > Binary CL** function. APT Table files can also be specified on the command line when the Binary CL Converter is executed, such as during batch processing to simulate NC programs without user interaction.

Sample APT Table file "aptwords.tbl" (located in the \vericut\ directory of your VERICUT installation):




```
IBMCL2V-apt
version-2.0

This file contains all of the user configurable information
neccessary for the APT/AC converter to successfully convert
a binary APT CL file to a man-readable APT CL file.

The order of information in this file is important, and should be
as follows:

        Binary CL file BLOCKING FACTOR (via BLOCKING_FACTOR = number)
        Major words table
        Minor words table


The following record specifies the IBM binary CL file Blocking Factor:

BLOCKING_FACTOR = 3228

This table represents all of the APT Major words that the
APT/AC converter will process.

MAJOR_WORDS_BEGIN

$$      WORD            CLASS           SUBCLASS
$$------------------------------------------------
        DRAWLI          2000            17
        DRESS           2000            8
        END             2000            1
        FACEML          2000            22
        GOCLER          2000            16
        GOHOME          2000            14
        ISTOP           2000            4
```

Tips:

1. The default blocking factor for interpreting binary CL data is 3228. If this does not produce favorable results, use any text editor to change the blocking factor. Other typical blocking factors you can try are: 512, 4628, 15476.
2. If the ASCII NC program file output by the converter has major or minor words other than what you expected, try editing the APT Table file to change the "Word" output for a specific APT class/subclass.

AUTO-DIFF Report File

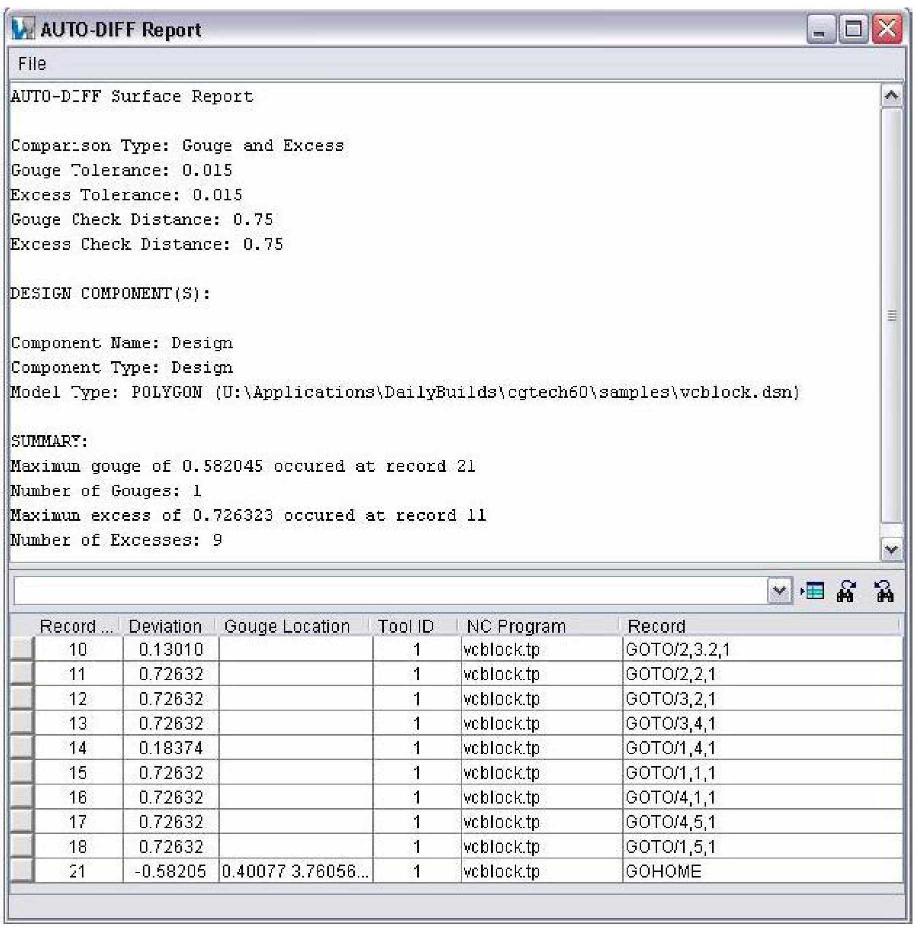
Typical file extension: .rpt

An ASCII text file that contains information about an AUTO-DIFF model comparison operation, such as identification of all detected errors, and the NC program history related to each error. This information is valuable when determining the correctness of the VERICUT simulated cut model, and NC program records responsible for discrepancies detected by AUTO-DIFF.

An AUTO-DIFF Report file is created when an AUTO-DIFF comparison is performed, and can be viewed via the "Report" feature. (Ref. **Analysis menu > AUTO-DIFF**)

The exact format of the report varies, but generally includes a summary of the results, information about tolerances and models used in the comparison, error types (gouge/excess) and amounts, NC program history associated with errors, etc.

Sample AUTO-DIFF Report file:




```
AUTO-DIFF Surface Report

Comparison Type: Gouge and Excess
Gouge Tolerance: 0.015
Excess Tolerance: 0.015
Gouge Check Distance: 0.75
Excess Check Distance: 0.75

DESIGN COMPONENT(S):

Component Name: Design
Component Type: Design
Model Type: POLYGON (U:\Applications\DailyBuilds\cgtech60\samples\vcblock.dsn)

SUMMARY:
Maximun gouge of 0.582045 occured at record 21
Number of Gouges: 1
Maximun excess of 0.726323 occured at record 11
Number of Excesses: 9
```

Record ...	Deviation	Gouge Location	Tool ID	NC Program	Record
10	0.13010		1	vcblock.tp	GOTO/2,3.2,1
11	0.72632		1	vcblock.tp	GOTO/2,2,1
12	0.72632		1	vcblock.tp	GOTO/3,2,1
13	0.72632		1	vcblock.tp	GOTO/3,4,1
14	0.18374		1	vcblock.tp	GOTO/1,4,1
15	0.72632		1	vcblock.tp	GOTO/1,1,1
16	0.72632		1	vcblock.tp	GOTO/4,1,1
17	0.72632		1	vcblock.tp	GOTO/4,5,1
18	0.72632		1	vcblock.tp	GOTO/1,5,1
21	-0.58205	0.40077 3.76056...	1	vcblock.tp	GOHOME

AVI File

Typical file extension: .avi

Also known as an Audio-Video Interleaved file, this file contains recorded images of VERICUT. AVI files are saved (recorded) via the **File menu > Images > Record Movie** function, and replayed via **File menu > Images > Playback AVI Movie**.

Batch Script Files

Multiple VERICUT batch sessions can be run via a single executable command by including VERICUT batch commands in an executable script file. The script file is then submitted to the operating system for processing. After each batch execution terminates, the operating system then executes the next command in the script file.

Batch files for different operating systems differ slightly in format. Details will follow later in this document. In general, each batch command in the script file appears as it would when entered by itself on the command line. Other operating system commands, such as those that delete or rename files can also be included in the script file.

Tip: VERICUT batch files can be easily created using VERICUT's Batch Wizard.

Windows batch script files

A batch file to be run on Windows is named with a ".bat" extension, for example "batchfile.bat". Windows NT batch execution lines in the script file are "called" via the "call" command to return control to the script file after each VERICUT execution. The examples that follow show two sample script files.

Windows XP batch script file examples:

Example 1 executes two batch sessions, each processing a part unrelated to the other.

```
call vericut batch "vcp=part1.VcProject"
call vericut batch "vcp=part2.VcProject"
```

Example 2 represents a part machined in two setups: the first is configured to automatically save an In-process file named "setup1.ai" via the **AutoSave** function. The second uses the ip=<ipfile> option to load the "setup1.ai" In-process file so cutting can be continued with the session configuration saved in the "setup2.VcProject" project file.

```
call vericut batch "vcp=setup1.VcProject"
call vericut batch "vcp=setup2.VcProject" "ip=setup1.ai"
```

UNIX batch script files

A batch script file to be executed in UNIX must have "executable" permissions. Use the "chmod" command to give the file executable permissions, for example the following command makes a file named "batchfile" executable by all users: chmod 777 batchfile The "batchfile" file is executed via typing the file name on the command line, for example type: batchfile. The examples that follow show two sample script files.

UNIX batch script file examples:

Example 1 executes two batch sessions, each processing a part unrelated to the other.

```
vericut batch vcp=part1.VcProject
vericut batch vcp=part2.VcProject
```

Example 2 represents a part machined in two setups: the first is configured to automatically save an In-process file named "setup1.ai" via the **AutoSave** function. The second uses the ip=<ipfile> option to load the "setup1.ai" In-process file so cutting can be continued with the session configuration saved in the "setup2VcProject" project file.

```
vericut batch vcp=setup1.VcProject
vericut batch vcp=setup2.VcProject ip=setup1.ai
```

CGTECH Macro Executable File

Typical file extension: .cme

A platform-independent binary file also known as a "CME file", that is created using the **C Macro Extension – Application Programming Interface**, or **CME–API** (ref. **CME-API** in the VERICUT Development Tools section, in the *CGTech Help Library*) and contains custom macros that control processing G-Code data. In addition to macros, this file can also establish modals which allow users to further customize how machine code data is interpreted, and provide access to custom tables containing job and machine related data. The CME file used to perform these tasks is specified via the Advanced Control Option window: Dev Kit CME tab (ref. **Advanced Control Option window: Dev Kit CME tab**, in the Configuration Menu section of *VERICUT Help*).

Control File

Typical file extension: .ctl

An ASCII text file that contains data describing how the NC control processes G-Codes. Data in this file includes: a list of interpretable machine codes, machine code format information, rules for grouping machine codes, and rules for calling macros. This file is required to interpret G-Code data via the G-Code Data NC program type. When the **C Macro Extension – Application Programming Interface**, or **CME–API** (ref. **CME-API** in the VERICUT Development Tools section, in the *CGTech Help Library*) is used to create a custom CME file, the Control file can also specify a custom CME file to assist with the simulation.

By default, VERICUT is configured with a "Fanuc" control. Many control configurations are available in the library that is installed with VERICUT software- see "**Library Control Files**" for more information.

A different Control file can be loaded into VERICUT via the **Configuration menu > Control > Open** function. NC control configurations are changed via other "**Control**" functions in the **Configuration menu**. Save the new file via **Configuration menu > Control > Save As**.

Encrypted Control Files

A "standard" encrypted (X-File) control file:

- has a .xctl extension
- works with a standard VERICUT license.
- is a compressed binary file.
- contains all files referenced by the Control file (no external subroutines).
- simulates exactly the same as a standard control file (.ctl)
- disables the **Configuration menu** features when used in a setup.
- can be created by any user.
- ***can not*** be decrypted/expanded except by CGTech so retain your original control (.ctl) file

A "limited" encrypted (X-File) control file:

- has a .xctl extension
- works only for a specific customer with a VERICUT Limited license.
- is a compressed binary file.
- contains all files referenced by the Control file (no external subroutines).
- disables the **Configuration menu** features when used in a setup.
- can only be created by CGTech.
- ***can not*** be decrypted/expanded except by CGTech.

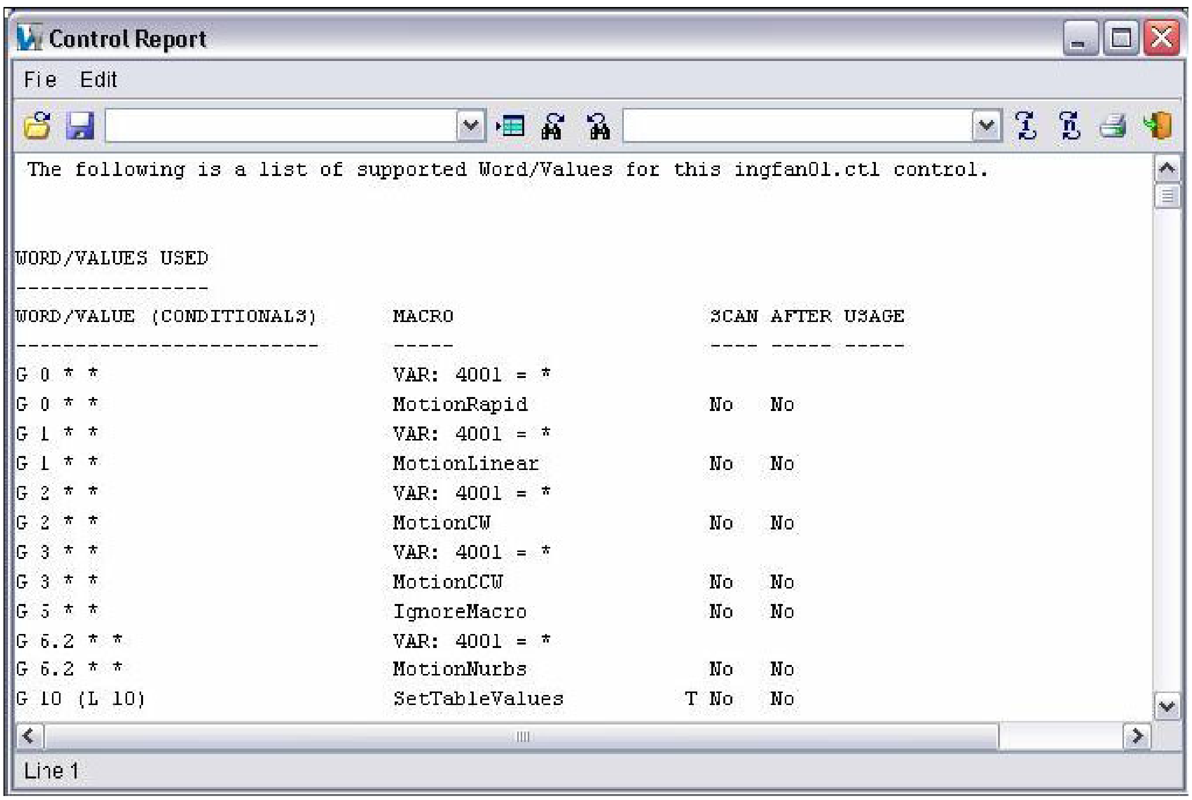
Control Report File

Typical file extension: .rpt

An ASCII text file that contains information about how the current NC control configuration will interpret various codes. Reported information includes:

G-Codes present in the NC program and macros (actions) the control will perform, variables used (if any), cutting tools used, subroutines defined and referenced, and more. Control Report files are created via the **Info menu > Control Report** function.

Sample Control Report file:



CSYS File

An ASCII text file containing information necessary to create a VERICUT Coordinate System using the CSYS from File feature in the **Coordinate System window**. The file contains a collection of APT CL matrices, possibly with different syntaxes in a single file. The CSYS File is also used by the csys_file command line argument.

The CSYS file may contain one or all combinations of:

CATIA syntax:

```
$$*CATIA0
$$*AXS1
$$     1.00000 ...
```

Use "$$*AXS xx" to name the coordinate system, for example "AXS1" in the above snippet.

NX syntax:

```
TOOL PATH/P1
.
.
MSYS/ ...
```

Use "TOOL PATH/ xx" to name the coordinate system, for example "P1" in the above snippet.

PTC syntax:

```
$$-> FEATNO / 102
.
.
TRANS / ...
$$-> CSYS / ...
```

Use "FEATNO / xxx" to name the coordinate system, for example "102" in the above snippet.

VERICUT-MATRIX syntax:

Modify the syntax to allow an optional name. The code needs to support both with a quoted name string and without.

```
PPRINT/VERICUT-MATRIX "name" 1,0,0 ...
```

Design Points File

Typical file extension: .pts

An ASCII text file, also known as an "inspection point file", that contains point locations used to inspect the VERICUT model when performing an AUTO-DIFF Point comparison operation.

Design Point files are loaded into VERICUT via the **Project menu > Setup Models > Define** function. Point record formats support specifying surface normals, tolerances, and check distances with any or all inspection points. Example point record formats follow.

Design Points file record formats:

GOTO/X,Y,Z — where "X,Y,Z" represents the design point location, for example: "GOTO/1,2,3".

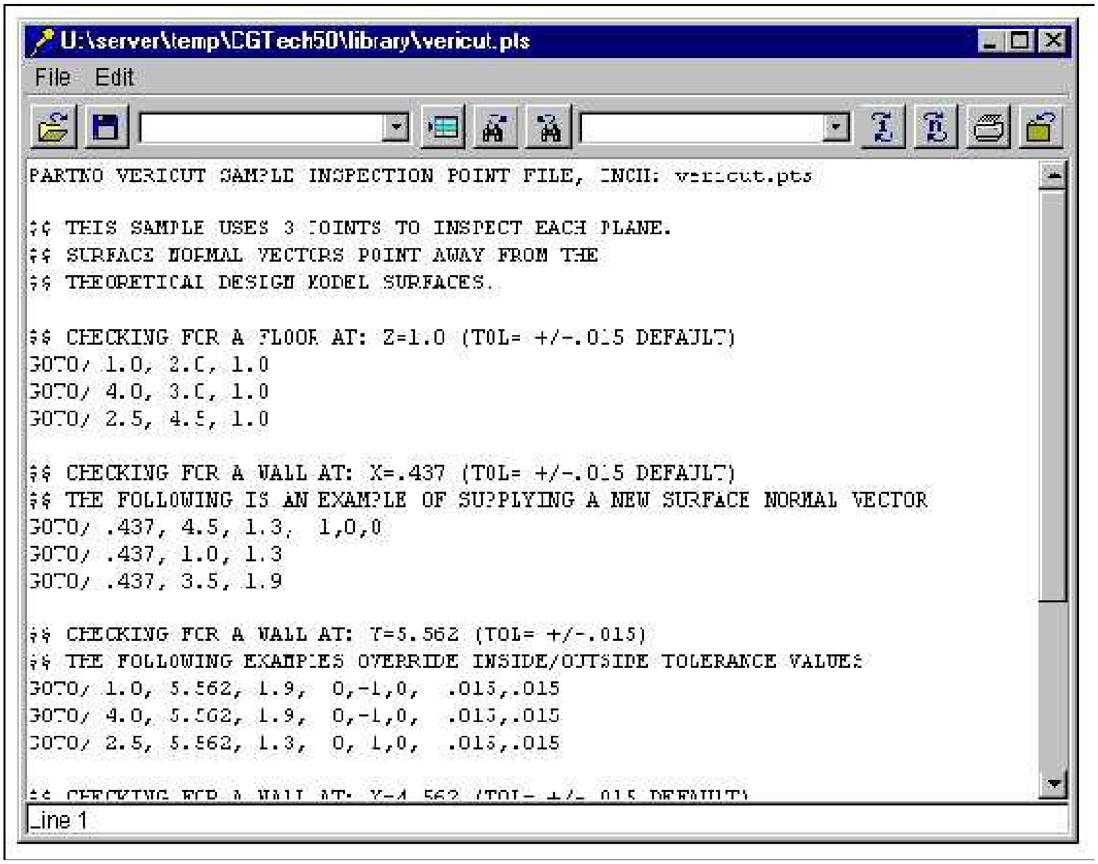
GOTO/X,Y,Z,I,J,K — as above plus "I,J,K" represents the surface normal vector, for example: "GOTO/1,2,3,1,0,0".

GOTO/X,Y,Z,I,J,K,intol,outtol — as above plus "intol" and "outtol" override the Gouge Tolerance and Excess Tolerance values, for example: "GOTO/1,2,3,1,0,0,.0005,.015". The overrides are only applied to the point where they are listed.

GOTO/X,Y,Z,I,J,K,intol,outtol,incheck,outcheck — as above plus "incheck" and "outcheck" override the Gouge Check Distance and Excess Check Distance values, for example: "GOTO/1,2,3,.1,0,0,.015,.015,.300,.300". The overrides are only applied to the point where they are listed.

Any combination of the above formats can be included in the Design Points file. There is no limit to number of records that the file can contain. "$$" comment records and spaces are permitted.

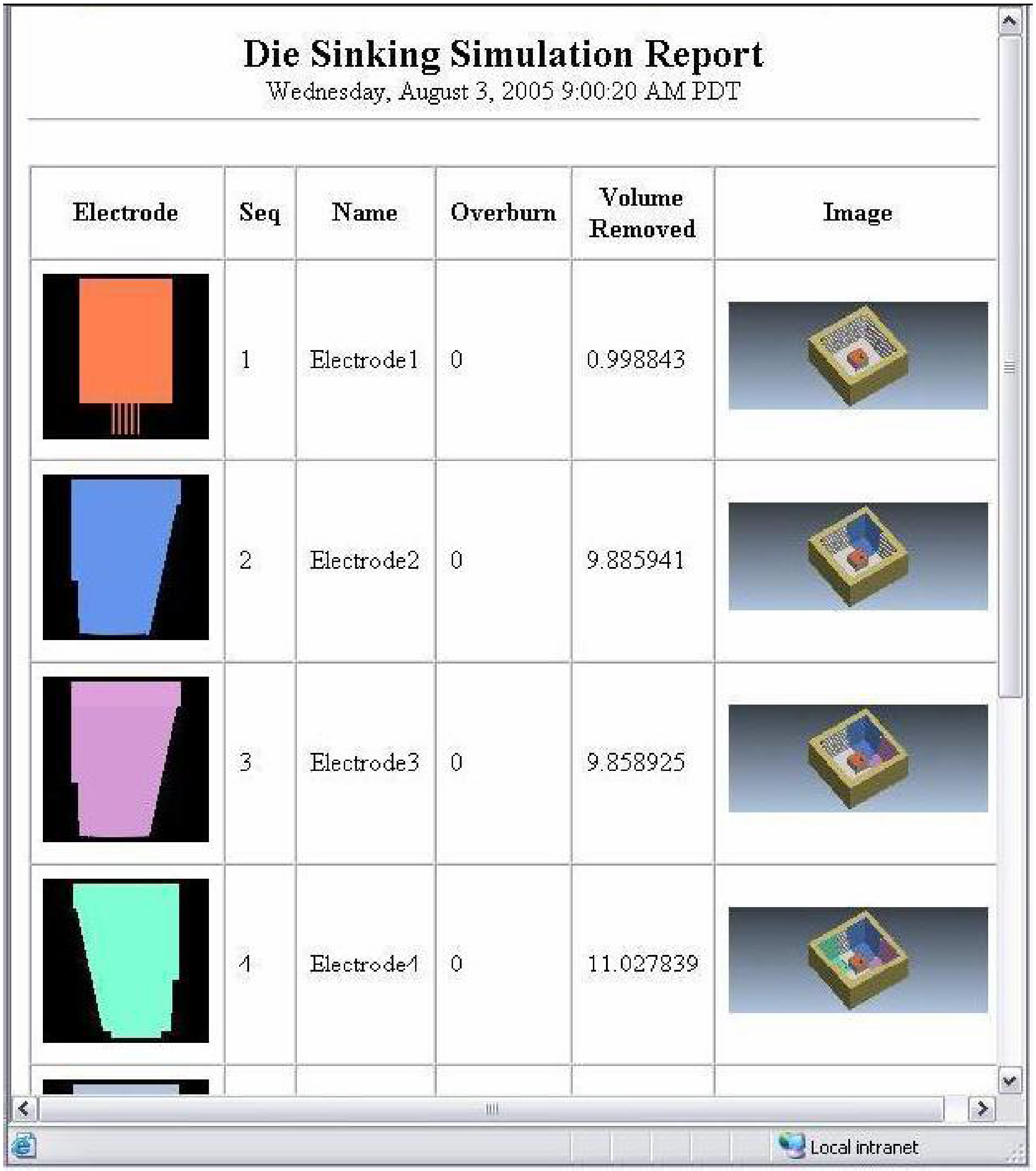
Sample Design Points file:



Die Sinking Simulation Report File

Typical file extension: .htm

An HTML file that contains data describing a particular die sinking process.



Die Sinking Simulation Report

Wednesday, August 3, 2005 9:00:20 AM PDT

Electrode	Seq	Name	Overburn	Volume Removed	Image
	1	Electrode1	0	0.998843	
	2	Electrode2	0	9.885941	
	3	Electrode3	0	9.858925	
	4	Electrode4	0	11.027839	

G-Code Log File

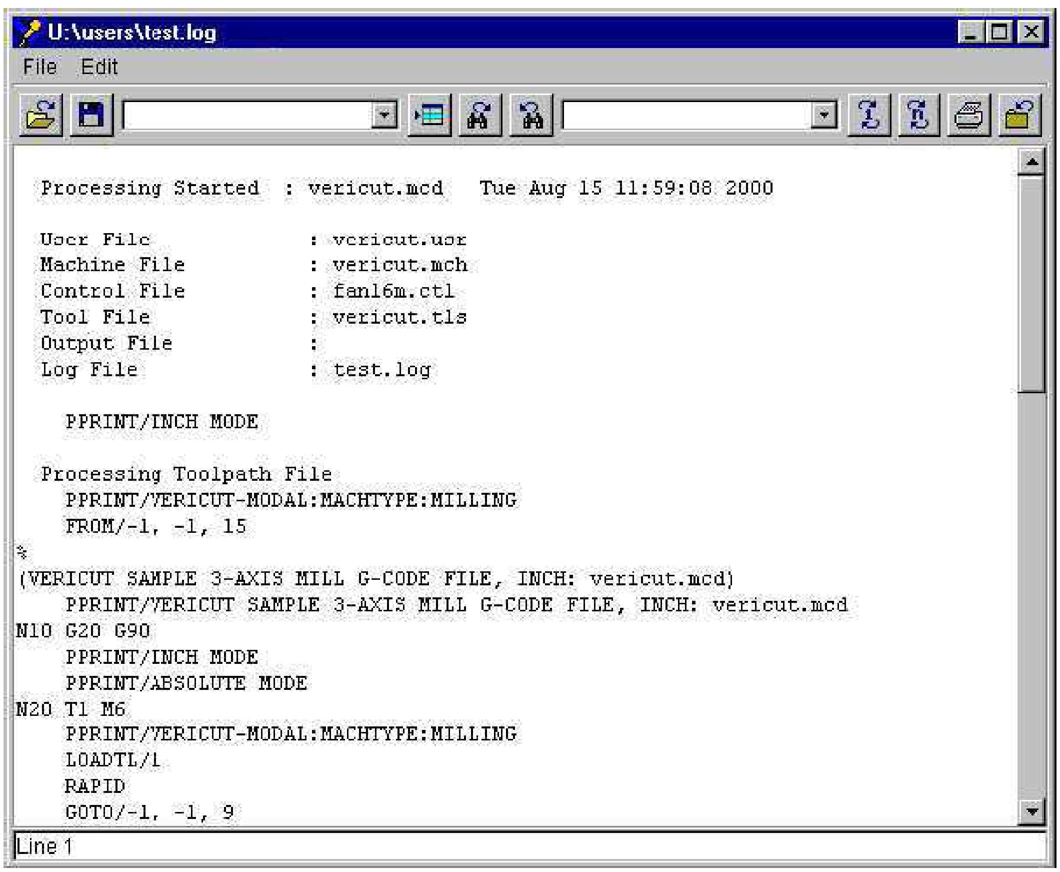
Typical file extension: .log

An ASCII text file that contains error, warning and informational messages about G-Code processing. One of the most valuable pieces of information this file provides is a copy of each input G-Code data block followed by the converted ASCII APT NC program record indented by 4 spaces. This allows you to see a direct translation of G-Code versus equivalent ASCII APT output for all NC program records processed.

G-Code Log files are created when a name is entered in the **Project menu > Processing Options > G-Code > Process Options: G-Code Output File tab**, **Log File** field, and are viewed via **Project menu > Processing Options > G-Code > Process Options: G-Code Output Files tab**, **View G-Code Log File**.

For help with interpreting error and warning messages in this file, visit the [VERICUT Users' Forum] or contact CGTech technical support via our [website], just click on the support link.

Sample G-Code Log file:



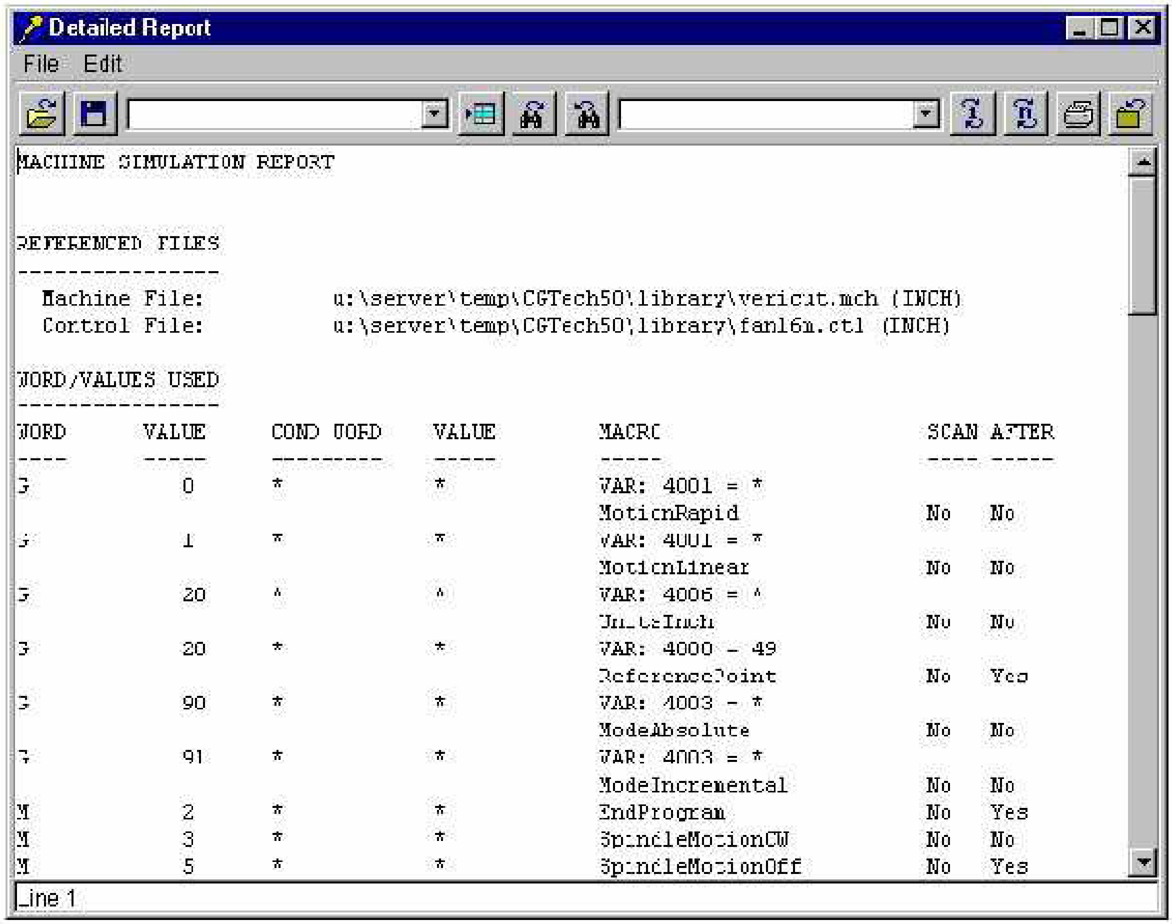

```
 Processing Started  : vericut.mcd   Tue Aug 15 11:59:08 2000

 User File             : vericut.usr
 Machine File          : vericut.mch
 Control File          : fan16m.ctl
 Tool File             : vericut.tls
 Output File           :
 Log File              : test.log

    PPRINT/INCH MODE

 Processing Toolpath File
    PPRINT/VERICUT-MODAL:MACHTYPE:MILLING
    FROM/-1, -1, 15
%
(VERICUT SAMPLE 3-AXIS MILL G-CODE FILE, INCH: vericut.mcd)
    PPRINT/VERICUT SAMPLE 3-AXIS MILL G-CODE FILE, INCH: vericut.mcd
N10 G20 G90
    PPRINT/INCH MODE
    PPRINT/ABSOLUTE MODE
N20 T1 M6
    PPRINT/VERICUT-MODAL:MACHTYPE:MILLING
    LOADTL/1
    RAPID
    GOTO/-1, -1, 9
```

G-Code Report File

Typical file extension: .rpt

An ASCII text file that contains information about how the current NC control configuration will interpret a G-Code NC program file. Reported information includes: G-Codes present in the NC program and macros (actions) the control will perform, variables used (if any), cutting tools used, subroutines defined and referenced, and more. G-Code Report files are created via the **Info menu > G-Code Report** function.

Sample G-Code Report file:




```
MACHINE SIMULATION REPORT


REFERENCED FILES
----------------
  Machine File:         u:\server\temp\CGTech50\library\vericut.mch (INCH)
  Control File:         u:\server\temp\CGTech50\library\fan16m.ctl (INCH)

WORD/VALUES USED
----------------
WORD     VALUE     COND WORD     VALUE       MACRO                     SCAN AFTER
----     -----     ---------     -----       -----                     ---- -----
G          0       *             *           VAR: 4001 = *
                                             MotionRapid               No   No
G          1       *             *           VAR: 4001 = *
                                             MotionLinear              No   No
G         20       *             *           VAR: 4006 = *
                                             UnitsInch                 No   No
G         20       *             *           VAR: 4000 = 49
                                             ReferencePoint            No   Yes
G         90       *             *           VAR: 4003 = *
                                             ModeAbsolute              No   No
G         91       *             *           VAR: 4003 = *
                                             ModeIncremental           No   No
M          2       *             *           EndProgram                No   Yes
M          3       *             *           SpindleMotionCW           No   No
M          5       *             *           SpindleMotionOff          No   Yes
```

IGES Model File

Typical file extension: .igs

An IGES (Initial Graphics Exchange Specification) model file, also known as an "IGES file", is an ASCII text file which describes virtually anything that can be modeled in a CAD system. IGES data is widely accepted as an industry standard for transferring model data. While IGES model files containing only 2-D data (lines and arcs) are not useful in VERICUT, files containing 3-D surfaces and solids can be used to describe stock, fixture, or design models, as well as machine component shapes.

IGES model files are imported into VERICUT via the **Project menu > Setup Models > Define** function, or can be converted into STL or VERICUT Polygon model files using the IGES converter.

Image File

Typical file extension: .img

A platform-independent binary file that contains recorded images of a NC program simulation. Image files can contain any combination of snapshot (single still image) and animation images (similar to a movie). These files can be recorded and replayed on any supported computer platform.

Two types of VERICUT images can be recorded. Both types can be recorded into the same file. Machining status is automatically recorded with either image type.

Snapshot — single-frame image. These images are also used to mark stop points in an animation.

Animation — multiple images that when replayed, appear like a simulation (like a movie).

Image files are saved (recorded) via the **File menu > Images > Record Movie** function, and replayed via **File menu > Images > Playback VERICUT Movie**.

NOTE: Image files are not intended for printing. Use **File menu > Print View** or **File menu > Images > View Capture** to print VERICUT images.

In-Process File

Typical file extension: .ip

A part ASCII text-part binary file also known as an "IP file" that contains the data necessary to re-establish a VERICUT session, including the current VERICUT model (with cuts), user interface settings, and a copy of the Log file. IP files are also used to "backup" the verification session, safeguarding against lost work if the session is interrupted by computer or power failure.

Save an In-process file interactively via **File menu > In Process > Save As**, or automatically via the **File menu > AutoSave** function. You can also save an IP file using the SAVE_IP, VERICUT-COMMAND record (ref. to VERICUT-COMMAND Record in *Automating VERICUT*, in the *CGTech Help Library*). Due to differences in math processing on various computer types, IP files are usable on computers of the same type on which they were saved.

In-Process (IP) files are by default compressed when they are saved. VERICUT can read these compressed files directly.

To disable writing compressed IP files, set the environment variable CGTECH_COMPRESS=IP (ref. **Environment Variables** in the *VERICUT Help* section, in the *CGTech Help Library* for additional information).

Load IP files via **File menu > In Process > Open** to re-establish a VERICUT session or provide a VERICUT model for additional machining and analysis. IP files can also be specified on the command line when VERICUT is executed, such as during batch processing to simulate NC programs without user interaction.

NOTE: Environment variables can be used to affect VERICUT's use of IP files. For more information, see **Environment Variables** in the *VERICUT Help* section, in the *CGTech Help Library* for additional information.

Inspection Sequence Custom Tolerance File

Typical file extension: .txt

A custom inspection tolerance file is an ASCII text file that contains default tolerances that are automatically used when creating inspection record in an Inspection Sequence data table.

Inspection Sequence File

Typical file extension: .VcInspect

An inspection sequence file is an XML (Extensible Markup Language) text file used to retain all the information needed to reproduce the Inspection Sequence data table and the corresponding inspection markers on the workpiece for a single inspection report.

Log File

Typical file extension: .log

An ASCII text file that contains information about a VERICUT session, such as: names of NC programs processed, errors, warnings, simulated machining times, and other informational messages.

When VERICUT is run, it must always be able to open a Log file. When an existing Log file of the same name is found VERICUT responds differently, depending on the run mode:

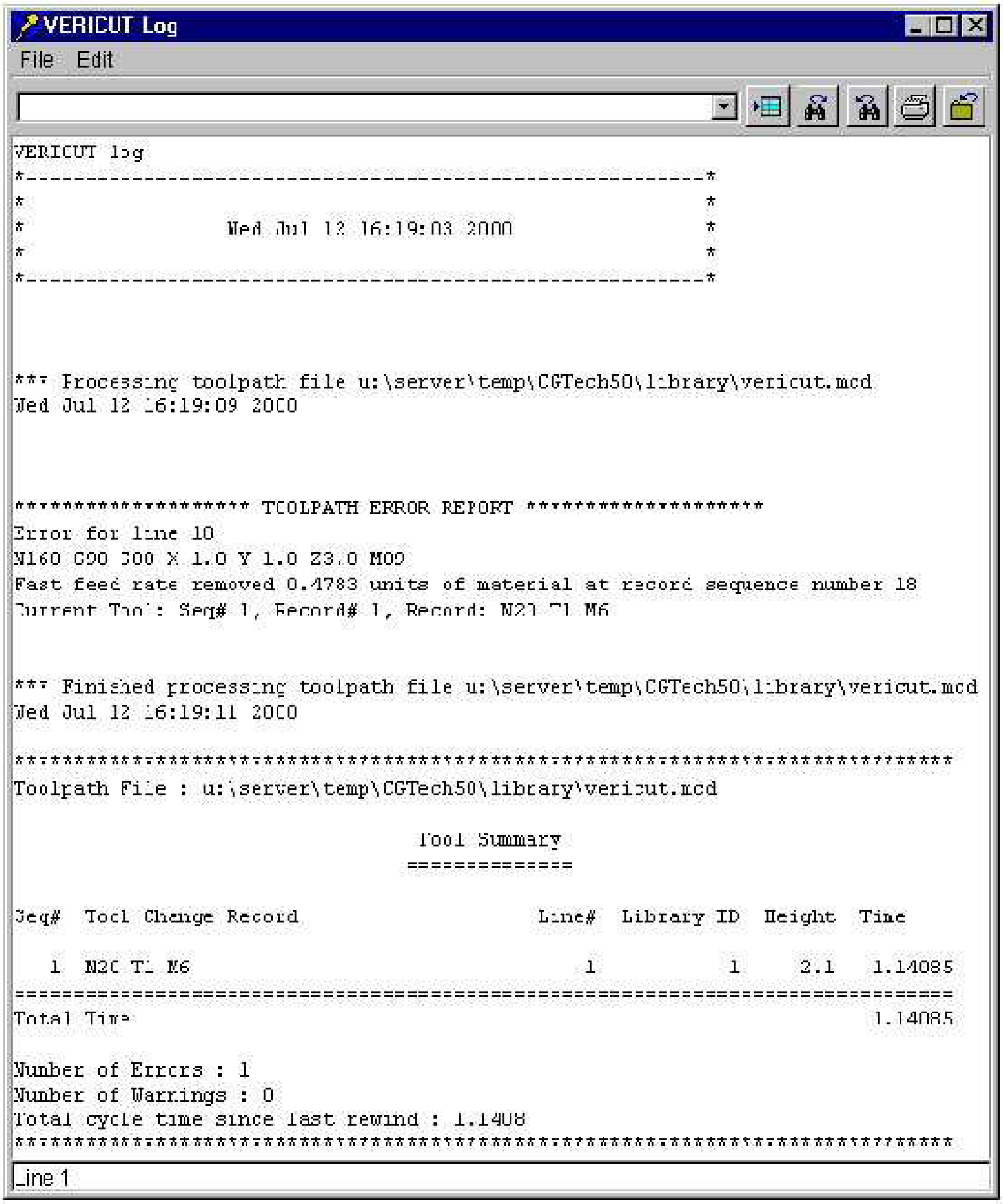
VERICUT run in interactive mode — Log file is automatically reset, unless the "append log" command line option is used.

VERICUT run in batch mode — Log file is automatically reset, unless the "reset log" command line option is used.

For more information on command line options used with VERICUT, see "**Command line options for VERICUT**".

Using the **Info menu > VERICUT Log** function, Log file contents can be viewed, edited, or the file name changed. For help with interpreting error and warning messages in this file, visit the VERICUT Users' Forum or contact CGTech technical support via our website, just click on the support link.

Sample VERICUT Log file:




```
VERICUT log
*----------------------------------------------------------*
*                                                          *
*                  Wed Jul 12 16:19:03 2000                *
*                                                          *
*----------------------------------------------------------*



*** Processing toolpath file u:\server\temp\CGTech50\library\vericut.mcd
Wed Jul 12 16:19:09 2000



******************** TOOLPATH ERROR REPORT ********************
Error for line 10
N160 G90 G00 X 1.0 Y 1.0 Z3.0 M09
Fast feed rate removed 0.4783 units of material at record sequence number 18
Current Tool: Seq# 1, Record# 1, Record: N20 T1 M6


*** Finished processing toolpath file u:\server\temp\CGTech50\library\vericut.mcd
Wed Jul 12 16:19:11 2000

*******************************************************************************
Toolpath File : u:\server\temp\CGTech50\library\vericut.mcd

                                 Tool Summary
                                ==============

Seq#  Tool Change Record                   Line#  Library ID  Height  Time

   1  N20 T1 M6                                1           1     2.1   1.14085
===============================================================================
Total Time                                                             1.14085

Number of Errors : 1
Number of Warnings : 0
Total cycle time since last rewind : 1.1408
*******************************************************************************
```

Machine File

Typical file extension: .mch

An ASCII text file that contains data describing the construction, kinematics, and other properties of an NC machine tool. By default, VERICUT is configured with a 3-axis vertical mill machine. Many machine configurations are available in the library that is installed with VERICUT software- see **Library Machine Files** in the *VERICUT Files section* for more information.

A different Machine file can be loaded into VERICUT via the **Configuration menu > Machine > Open** function. NC machine configurations are changed via other "Machine" functions in the Configuration menu. Save the new file via **Configuration menu > Machine > Save As**.

Encrypted Machine Files

A "standard" encrypted (X-File) machine file:

- has a .xmch extension
- works with a standard VERICUT license.
- is a compressed binary file.
- contains all files referenced by the Machine file (no external models).
- simulates exactly the same as a standard machine file (.mch)
- disables the **Configuration menu** features when used in a setup.
- can be created by any user.
- ***can not*** be decrypted/expanded except by CGTech so retain your original machine (.mch) file

A "limited" encrypted (X-File) machine file:

- has a .xmch extension
- works only for a specific customer with a VERICUT Limited license.
- is a compressed binary file.
- contains all files referenced by the Machine file (no external models).
- disables the **Configuration menu** features when used in a setup.
- can only be created by CGTech.
- ***can not*** be decrypted/expanded except by CGTech.

NC Program File

Typical file extension: .mcd (G-Code), .tp (APT), many other extensions depending on CAM system

NC program, or "NC program" files are ASCII text files that contain data describing cutting tool positions, machine information, and other information required to operate NC machine tools.

NC program files to be simulated or optimized are referenced in VERICUT via the **Project menu > NC Programs** function. VERICUT does not actually keep a copy of the NC program files, but instead stores NC program file names and keep tracks of which record in the file is being processed.

NC programs files can be divided into two general types: APT-CLS NC and G-Code. A general discussion of each type follows.

APT-CLS NC program files

APT and CLS NC program files are generic file formats output by CAM systems that are intended to be easily read by the NC programmer. They are intermediate file formats that typically are not used directly by the NC machine. Before the data can be used by an NC machine these files must be post-processed to a G-Code NC program file format containing codes specific to that machine tool (see "G-Code NC program files" below).

VERICUT processes "Simple ASCII APT" NC programs-meaning all macros, motion commands, and motion copy routines are pre-processed into an ASCII centerline file containing GOTO, CIRCLE, CYCLE, and miscellaneous machine commands. No geometry definitions, symbolic substitutions, macros, or complex motions (such as GORGT, GOLFT, etc.) are present.

Common APT-CLS NC program variations that VERICUT can process are listed below.

ACL
CADRA CL data
CATIA workstation APT
CV APT
ProManufacturing APT
UG CLS

To process one of the APT-CLS NC program variations listed above, be sure to specify the appropriate NC Program Type under **Project menu > NC programs**.

G-Code NC program files

G-Code NC program files are formatted for use directly by the NC machine. These files typically originated as APT-CLS NC program files (see "APT-CLS NC program files" above), and have been post-processed to contain specific codes for a given machine tool.

To process G-Code NC program file, be sure to specify NC program Type=G-Code Data under **Project menu > NC programs**. While VERICUT easily processes G-Code data that conforms to the EIA Standard RS-274 format, other variations can also be simulated with a little configuration effort. Configuration menu functions provide most of the configurability needed to support various NC machine and control combinations.

Operations File

Typical file extension: .VcTmp

Introduction

An Operations File, or Ops File, can be used to assemble a project with multiple setups from several VERICUT project files, which typically have the extension ".VcProject". This assembly process can only be triggered from the command line that invokes VERICUT. The necessary command line syntax is;

For Windows ... "ops=D:\My\Path\MyOperationsFile.VcTmp" ...

For UNIX ... "ops=/My/Path/MyOperationsFile.VcTmp" ...

Any extension can be used in the file name, but by convention ".VcTmp" denotes a temporary file.

An Operations File is a man-readable text file. Each line of the file is independent and references a VERICUT project file. A line requests that information from its project file should be transferred to the project being assembled. A line can request that the entire project and all its setups be transferred, or that information from one setup be transferred, or that all the setup-independent information, which we will refer to as project information, be transferred. It is suggested that the first line of the file should always request at least the project information from a ".VcProject" file that was saved by VERICUT, so that the new project will have a good foundation.

Syntax

In the following explanation the Windows path delimiter, "\", is used. For a UNIX platform you should substitute the "/" character. Names should be enclosed in double quotation characters ("). Fields are separated by one or more spaces.

To request that the entire project, with all its setups, be transferred to the new project being assembled, the syntax of the required line is;

```
ALL   FILE="My\Path\MyProjectTemplate.VcProject"
```

To request that the project information from a file be transferred to the new project being assembled, the syntax of the required line is;

```
PROJECT   FILE="My\Path\MyProjectTemplate.VcProject"
```

Setup independent information in a project file pertains to the following VERICUT menu items;

- Model Export
- Colors
- Image Recording and Playback
- View Capture
- AutoSave
- View Select/Store
- Status
- Graphs
- Cycle Definitions
- AUTO-DIFF

To request that setup information from a file be transferred to the new project, the syntax of the required line is;

```
SETUP  NAME="Setup A"  INDEX=1 SETUP=1  MERGE_TOOLS FILE="\Path\Tmp.VcProject"
```

The NAME field specifies the name that the setup will have in the new project. The INDEX field selects which of the setups in the referenced file will be transferred. An index value of "1" selects the first setup in the file, "2" would use the second setup, etcetera. The INDEX field is optional and defaults to 1 if not present. The SETUP=n field specifies the index of the setup in the merged file that this line refers to. It is optional and is only required if the MERGE_TOOLS field is present. The MERGE_TOOLS field specifies that the tool library associated with the setup being imported should be merged with the current tool library rather than overwriting it. This field is optional. The FILE field references the VERICUT project file that contains the required setup.

Note that more than one SETUP line in the Operations File can have the same NAME field. The first such line would cause a new setup to be appended to the project being assembled, and subsequent lines could adjust the parameters of that same setup. Setup specific information in a project file pertains to the following VERICUT menu items;

- Properties
- Views
- Sections
- Components and Models
- NC Programs and Filters

Tools and Tool Changes
APT and G-Code Settings
Coordinate Systems
Control
Machine
Inspection
Die Sinking
OptiPath

Project Naming

The name and location of the assembled project file will be derived from the name and location of the Operations File. The extension ".VcProject" will replace any extension that the Operations File's name has, and the assembled project will be placed in the same folder. This assumes that the user saves the project before explicitly renaming it and before exiting VERICUT. To remind the user that he needs to save the project before exiting, you can include another keyword on the command line. Thus you may have;

```
...   "ops=D:\My\Path\MyOperationsFile.VcTmp"   usr_changed   ...
```

Optimized File

Typical file extension: .opti, .mco (G-Code NC program file) or .tpo (APT NC program file)

The Optimized NC program file is created from optimizing NC program feed rates via OptiPath.

For details about using OptiPath and sample optimized NC program files, see the OptiPath menu functions.

OptiPath Library File

Typical file extension: .olb

An ASCII text file that contains optimization data for use by VERICUT's OptiPath feed rate optimization module. Specific optimization data values are defined per tool intended to be used in specific machining conditions, e.g. workpiece material, NC machine, etc.

NC program optimization can performed in a variety of ways, including: configuring cutting tools-to-optimization library records, storing optimization information with tools in a VERICUT Tool Library, or interactive prompting.

Post-Processor File

Typical file extension: .VcPost

A Post-Processor file is an ASCII text file, created using the CGTech Post Processor, containing all the information required to read machine independent NC Program data and convert it to G-Code files for a specific machine/control combination.

Preferences File

Typical file extension: .prefs

Also known as a "Prefs file", this ASCII text file is automatically saved when you exit VERICUT and stores such user preferences as:

- VERICUT user interface **Look & Feel**
- size and locations of VERICUT windows
- state of menu toggles in any window that has them, for example: **View menu > Toolbar**, **View Axes**, etc.
- filters for VERICUT file selection windows
- recent User/Project files opened in VERICUT

The Preferences file is unique for each VERICUT version and each user, based on your login. The file is named "cgtech_user.prefs" for VERICUT 5.0, and follows the naming convention "cgtech_<ver#>_user.prefs" for subsequent versions. For example, a Prefs file for VERICUT 5.1 has the name "cgtech_51_user.prefs". The Prefs file is automatically stored in your home directory and are updated each time you exit VERICUT. When you run a new VERICUT version for the first time (no Prefs file exists for that version), it will attempt to use the Prefs file from the previous version.

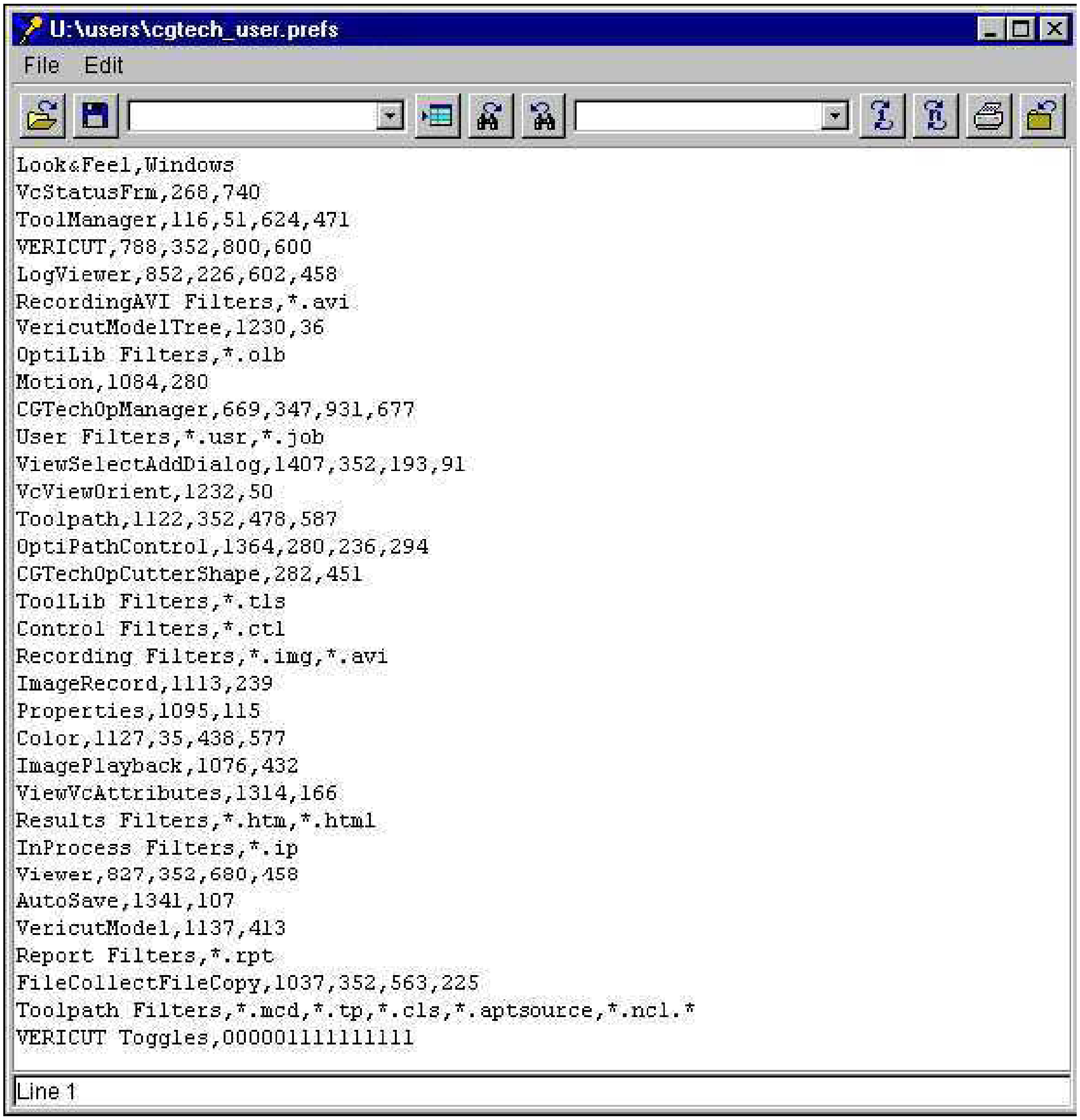
When users share a machine (or an account), CGTech recommends that each user set a unique CGTECH_USERNAME environment variable to prevent overwriting the Preferences file. See "**Environment Variables**" for more information.

To reset Preferences file settings:

Windows — Choose Reset VERICUT Preferences file from the CGTech programs group.

UNIX — Delete the "cgtech_user.prefs" from your home directory, or run the "delprefs" command file located in the ...<computertype>/commands directory of your VERICUT installation.

Sample Preferences file "cgtech_user.prefs":




```
Look&Feel,Windows
VcStatusFrm,268,740
ToolManager,116,51,624,471
VERICUT,788,352,800,600
LogViewer,852,226,602,458
RecordingAVI Filters,*.avi
VericutModelTree,1230,36
OptiLib Filters,*.olb
Motion,1084,280
CGTechOpManager,669,347,931,677
User Filters,*.usr,*.job
ViewSelectAddDialog,1407,352,193,91
VcViewOrient,1232,50
Toolpath,1122,352,478,587
OptiPathControl,1364,280,236,294
CGTechOpCutterShape,282,451
ToolLib Filters,*.tls
Control Filters,*.ctl
Recording Filters,*.img,*.avi
ImageRecord,1113,239
Properties,1095,115
Color,1127,35,438,577
ImagePlayback,1076,432
ViewVcAttributes,1314,166
Results Filters,*.htm,*.html
InProcess Filters,*.ip
Viewer,827,352,680,458
AutoSave,1341,107
VericutModel,1137,413
Report Filters,*.rpt
FileCollectFileCopy,1037,352,563,225
Toolpath Filters,*.mcd,*.tp,*.cls,*.aptsource,*.ncl.*
VERICUT Toggles,0000011111111111
```

Project File

Typical file extension: .VcProject

A VERICUT Project file is an XML (Extensible Markup Language) text file containing VERICUT session settings, also known as "user configuration values" or "user values", including: measurement units, colors, choices, data field values, directory paths and file names, etc. These files are relatively small in size, yet can control the entire cutting process and minimize the effort required to configure the system for processing NC program files.

Project files are saved via the **File menu > Save Project** function, and loaded via File menu > Open. User files can also be specified on the command line when VERICUT is executed, such as during batch processing to simulate NC programs without user interaction.

Report Template File

Typical file extension: .VcTemplate

A Report Template file is an XML (Extensible Markup Language) text file containing all of the formatting information required to generate VERICUT reports.

Status File

Typical file extension: .txt

An ASCII text file that contains whatever data is sent to the Status window during the simulation (ref. **Info menu > Status**). A Status file is created when the **CGTECH_STATUS** environment variable is set to a file name before VERICUT is executed. The file is overwritten when the VERICUT model is reset.

For each NC program record processed, the following status data is written:

REC — Indicates a NC program record processed.

REC fields — NC program Record Line Number, NC program Record

DATA — Specifies most of the status values separated by commas. Fields not displayed in the Status window are represented by commas.

DATA fields — Mch Axis Loc- X, Y, Z, A, B, C, U, V, W, Tool Tip Loc- X, Y, Z, I, J, K, Errors, Warnings, Time, Time%, Dist, - Dist%, Coolant, Feedrate, OP Feedrate, Spindle, OP Spindle

TOOLCHANGEREC — The record which caused the last tool change (present only if the Status window is configured to show this field).

TOOLDESC — The current tool description (present only if the Status window is configured to show this field).

TOOLGEOMETRY — The current tool geometry (present only if the Status window is configured to show this field).

APT example:

```
REC,12,GOTO/3,2,1
DATA,,,,,,,,,,,,,,,3,,,,0,0,0.5355,,,,ON ,10 IPM,,500,CLW ,
TOOLCHANGEREC,CUTTER/1.125,.25,0,0,0,0,2.1
```

G-Code example:

```
REC,20,N0130G0X-.2Y-.3Z2.S1200M3
DATA,,,,,,,,,,,,,,19,,,4,0,0,1.6979,,,,OFF,2 IPM,,1200,
TOOLCHANGEREC,N0120T4M6
```

Stereolithography (STL) Model File

Typical file extension: .stl

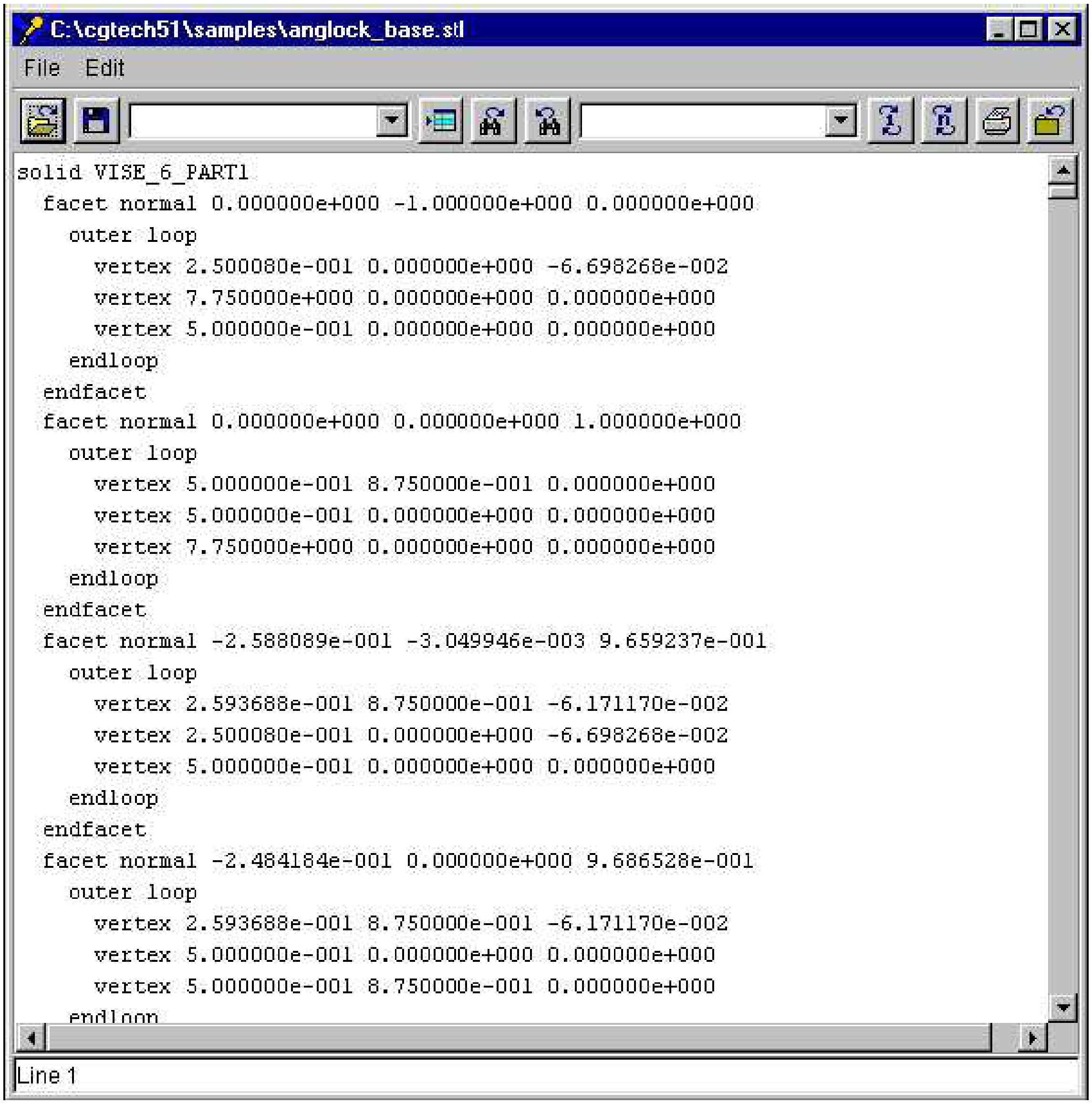
A Stereolithography model file, also known as an "STL" or "SLA" file, is an ASCII text or binary file which describes virtually any surfaced or solid model shape. The STL data representing the model shape is composed of three-sided facets with associated surface normals. STL files typically originate from a CAD system and represent an enclosed model used by an SLA machine, such as developed by 3D Systems.

Load Stereolithography model files into VERICUT via the **Project menu > Setup Models > Define** function. For an STL file to be used in VERICUT as a solid model, the STL data must represent a fully enclosed (watertight) model and have normals that point consistently outward. (STL files used as surface models do not have to be watertight.) If normals are inconsistent, gaps exist between facets, or facets overlap, solid model display results are unpredictable. When cutting is started or the model is analyzed, these symptoms indicate problems exist in the model file data:

- model is not displayed at all
- model is displayed with holes, gaps, or seams
- portions of the model are missing

Tip: If any of the above symptoms are experienced, use the PolyFix Converter and/or the Stock Consistency Check feature (ref. **File menu > Properties: General tab**) to repair the model before cutting. Similar advice applies for models added to Design components for use with AUTO-DIFF, except the Design Consistency Check feature is used (ref. **Analysis menu > AUTO-DIFF: Options tab**).

Sample STL file:




```
solid VISE_6_PART1
  facet normal 0.000000e+000 -1.000000e+000 0.000000e+000
    outer loop
      vertex 2.500080e-001 0.000000e+000 -6.698268e-002
      vertex 7.750000e+000 0.000000e+000 0.000000e+000
      vertex 5.000000e-001 0.000000e+000 0.000000e+000
    endloop
  endfacet
  facet normal 0.000000e+000 0.000000e+000 1.000000e+000
    outer loop
      vertex 5.000000e-001 8.750000e-001 0.000000e+000
      vertex 5.000000e-001 0.000000e+000 0.000000e+000
      vertex 7.750000e+000 0.000000e+000 0.000000e+000
    endloop
  endfacet
  facet normal -2.588089e-001 -3.049946e-003 9.659237e-001
    outer loop
      vertex 2.593688e-001 8.750000e-001 -6.171170e-002
      vertex 2.500080e-001 0.000000e+000 -6.698268e-002
      vertex 5.000000e-001 0.000000e+000 0.000000e+000
    endloop
  endfacet
  facet normal -2.484184e-001 0.000000e+000 9.686528e-001
    outer loop
      vertex 2.593688e-001 8.750000e-001 -6.171170e-002
      vertex 5.000000e-001 0.000000e+000 0.000000e+000
      vertex 5.000000e-001 8.750000e-001 0.000000e+000
    endloop
```

Tool Library File

Typical file extension: .tls

An ASCII text file that contains descriptions of cutting tools, shanks and holders, as well as other tool data used by VERICUT. Specific tool data is stored and retrieved via a tool "ID".

This file is created and maintained via the **Project menu > Tools** function. The Tool Library file used by VERICUT when the **Project menu > NC programs** function is configured to do so.

NOTE: If errors occur when attempting to load the file, visit the [VERICUT Users' Forum]() or contact CGTech technical support via our [website](), just click on the support link for assistance.

Sample VERICUT Tool Library file:

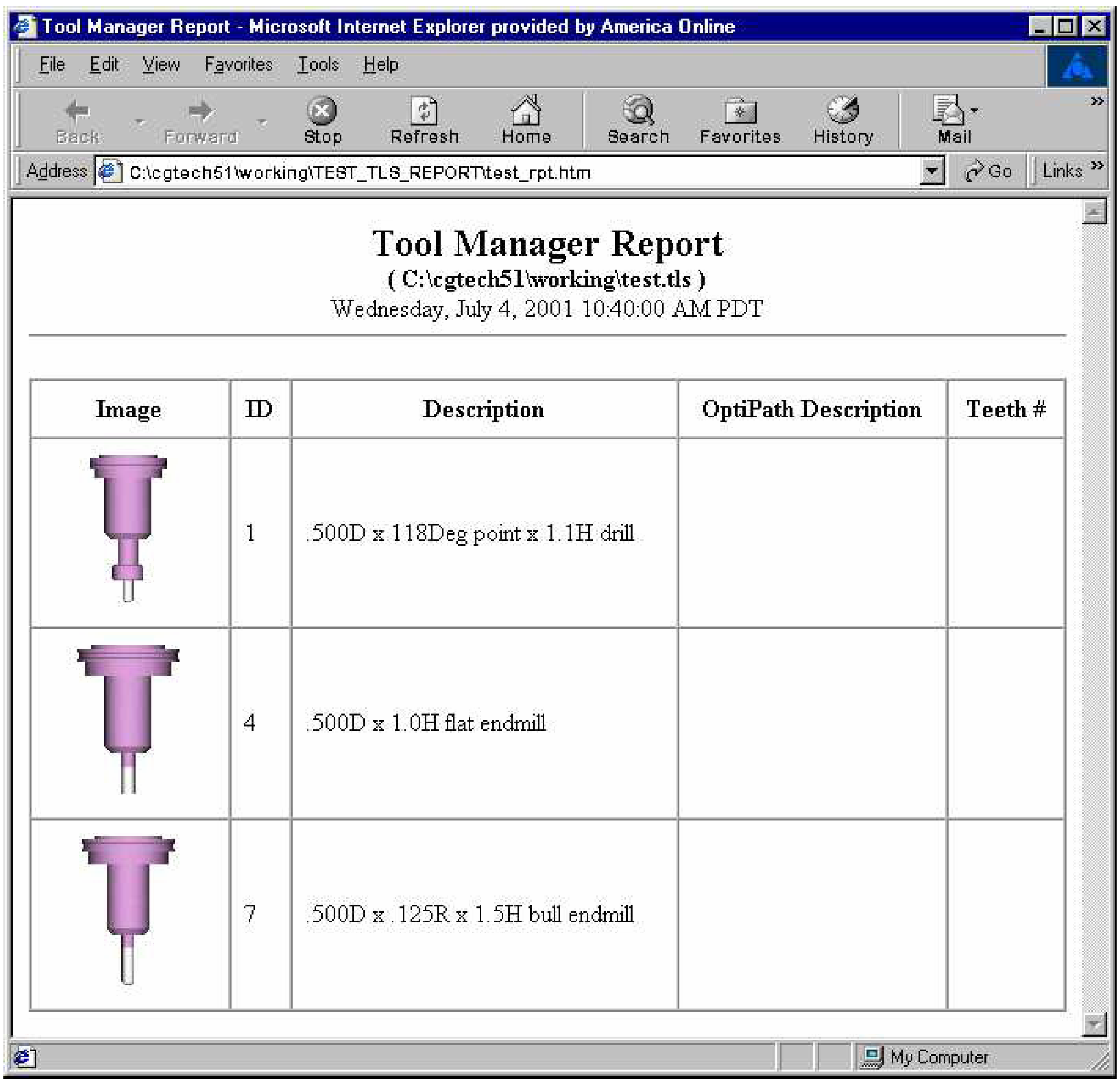
```
<?xml version="1.0"?>
<CGTechToolLibrary Version="6.0">
 <Tools>
  <Tool ID="1" Units="Millimeter">
   <Description>80 DEG, 0.03 (0.762 MM) RAD INSERT - OD</Description>
   <SpinAxis>SpindleAxis</SpinAxis>
   <Type>Turning</Type>
   <Cutter>
    <Insert ID="Cutter1" Type="GENERAL INSERT C">
     <P1>12.8959</P1>
     <P2>0</P2>
     <P3>0.8</P3>
     <P4>5</P4>
     <P5>0</P5>
     <P6>0</P6>
     <P7>0</P7>
     <Thickness>5</Thickness>
    </Insert>
   </Cutter>
   <Holder>
```

Tool Manager Report File

Typical file extensions: .htm (HTML version), .txt (test version)

An HTML or text file that contains information about tools in a VERICUT Tool Library file, such as: the Tool Library name, listing of tools, tool images (HTML version only), and optimization properties, if defined. Tool Manager Report files are created and viewed from the Tool Manager window via clicking **File menu > Create Report** features.

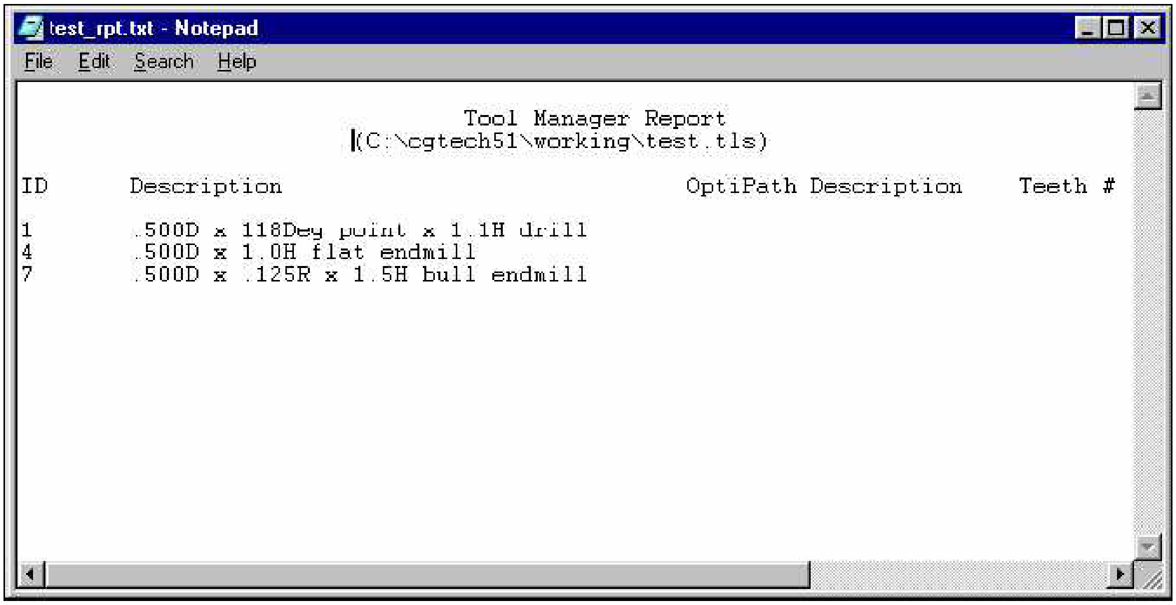
Sample HTML version Tool Manager Report file:



Tool Manager Report
(C:\cgtech51\working\test.tls)
Wednesday, July 4, 2001 10:40:00 AM PDT

Image	ID	Description	OptiPath Description	Teeth #
	1	.500D x 118Deg point x 1.1H drill		
	4	.500D x 1.0H flat endmill		
	7	.500D x .125R x 1.5H bull endmill		

Sample text version Tool Manager Report file:




```
                              Tool Manager Report
                         (C:\cgtech51\working\test.tls)

ID      Description                                 OptiPath Description    Teeth #

1       .500D x 118Deg point x 1.1H drill
4       .500D x 1.0H flat endmill
7       .500D x .125R x 1.5H bull endmill
```

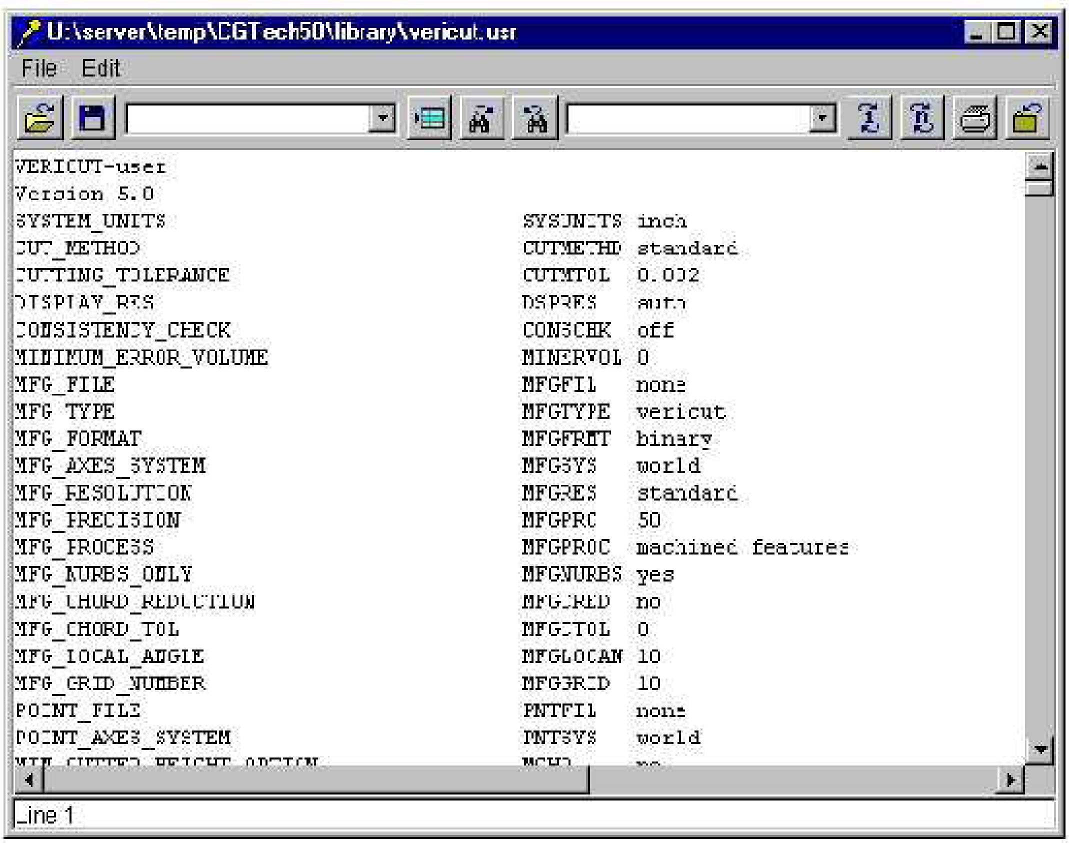
User File

Typical file extension: .usr

An ASCII text file that contains VERICUT session settings, also known as "user configuration values" or "user values", including: measurement units, colors, choices, data field values, directory paths and file names, etc. These files are relatively small in size, yet can control the entire cutting process and minimize the effort required to configure the system for processing a NC program file.

User files are loaded via **File menu > Open**. User files can also be specified on the command line when VERICUT is executed, such as during batch processing to simulate NC programs without user interaction.

Sample User file:



Within the User file, the left column contains descriptive information about user variables, the center column lists the user variable names, and the right column lists saved values for corresponding user variables.

Use the user variable names listed in the center column when referencing via command line options. For more information, see "**Command Line Options**".

NOTE: Environment variables can be used to affect VERICUT's use of User files. For more information, see "**Environment Variables**".

VERICUT Linear Sweep File

Typical file extension: .swp

A VERICUT Linear-sweep file, or "Linear-sweep file" for short, is an ASCII text file that describes an extruded shape. A 2-D profile is defined in the XY plane, then swept (extruded) along the model's Z axis.

Linear-sweep files are loaded into VERICUT via the **Project menu > Setup Models > Define** function, and contain the following data:

File type record — Specifies the type of VERICUT file, for example "VERICUT-model".

Version number record — Software version in which the file is to be used, for example "version-3.0".

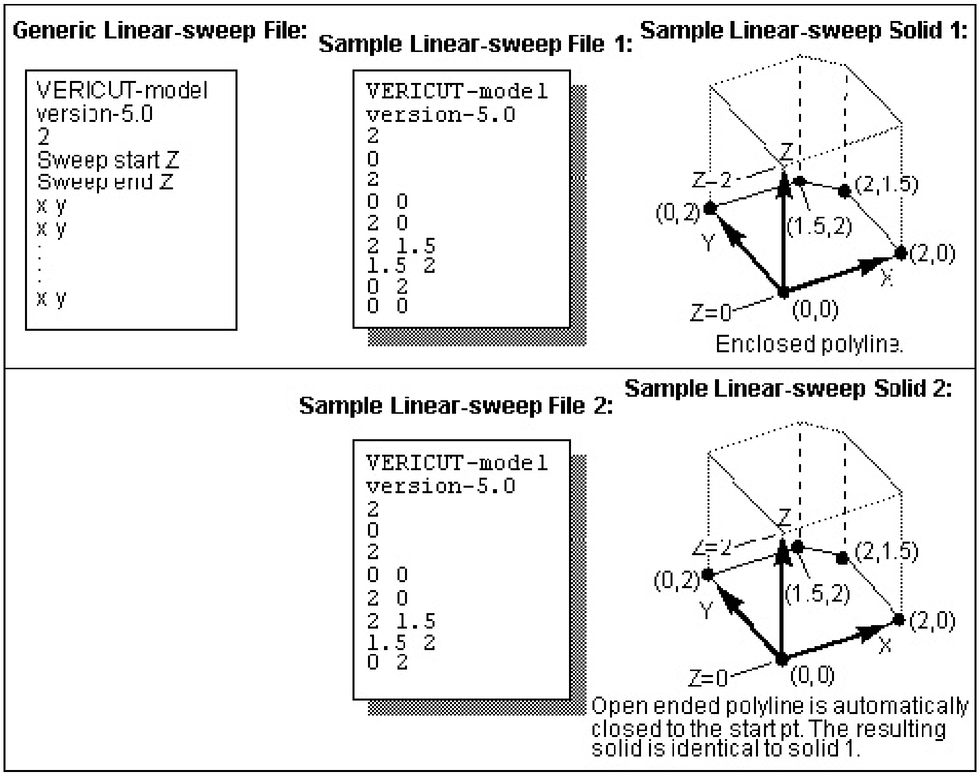
Model file identifier record — Model file type: "2" identifies a Linear-sweep file.

Sweep start value — Z value of the linear sweep start.

Sweep end value — Z value of the linear sweep end.

Polyline data — The remaining records in the file are the XY point pairs which define the polyline to extrude. Each X and Y value must be on a separate line with a space between values. There is no limit on the number of point pairs allowed, but there must be at least 3 pairs. The polyline can be completely enclosed or open ended, but can not cross over itself. Open ended polylines are automatically closed to the start point.

Sample Linear-sweep files:



VERICUT Polygon File

Typical file extensions: design- .dsn, fixture- .fix, stock- .stk, any- .ply

A VERICUT Polygon file, or "Polygon file" for short, is an ASCII text file that describes virtually any shape open surface or enclosed shape. The polygon (faceted) data is composed of three or four sided polygons with associated surface normals that represent the true geometric model shape.

Load Polygon model files into VERICUT via the **Project menu > Setup Models > Define** function. For a Polygon file to be used in VERICUT as a solid model, the polygon data must represent a fully enclosed (watertight) model and have normals that point consistently outward. (Polygon model files used as surface models do not have to be watertight.) If normals are inconsistent, gaps exist between facets, or facets overlap, solid model display results are unpredictable. When cutting is started or the model is analyzed, these symptoms indicate problems exist in the model file data:

- model is not displayed at all
- model is displayed with holes, gaps, or seams
- portions of the model are missing

Tip: If any of the above symptoms are experienced, use the PolyFix Converter and/or the Stock Consistency Check feature (ref. **File menu > Properties, General tab**) to repair the model before cutting. Similar advice applies for models added to Design components for use with AUTO-DIFF, except the Design Consistency Check feature is used (ref. **Analysis menu** > **AUTO-DIFF, Options tab**).

Polygon file format:

File type record — Specifies the type of VERICUT file, for example "VERICUT-model".

Version number record — Software version in which the file is to be used, for example "version-3.0".

Model file identifier record — Model file type: "1" specifies a Polygon file.

Sweep start value — Z value of the linear sweep start.

Sweep end value — Z value of the linear sweep end.

Polygon format record — Describes the surface normal data and vertex data formats. This record contains three integer numbers separated by blanks: n1 n2 n3 (e.g. "0 0 1") where:

n1 (normal vector type) — Specifies the type of surface normal vector information present in the polygon file.

0 - each polygon has one normal vector
1 - each vertex has a normal vector

n2 (normal vector direction) — Specifies the surface normal vector direction relative to the model surface.

0 - normal vectors point inward
1 - normal vectors point outward
2 - normal vector directions are inconsistent (should ONLY be used after both the "0" and "1" values have been attempted with unfavorable results; requires significantly more processing time and computer resources)

n3 (Polygon format type) — Specifies the polygon data format.

0 - binary data format (recommended- requires less storage space and is processed faster, however, binary formatted files are computer-specific)
1 - ASCII data format

Polygon data — The remaining records in the file are the polygon data records which define the model shape. Each model facet (or polygon) is represented by a series of records describing its number and location of vertices, as well as surface normal vector direction(s). There is no limit on the amount of data allowed.

Two polygon data formats are supported. The polygon data type must be consistent for the entire file.

Format 1 - 1 normal per polygon:

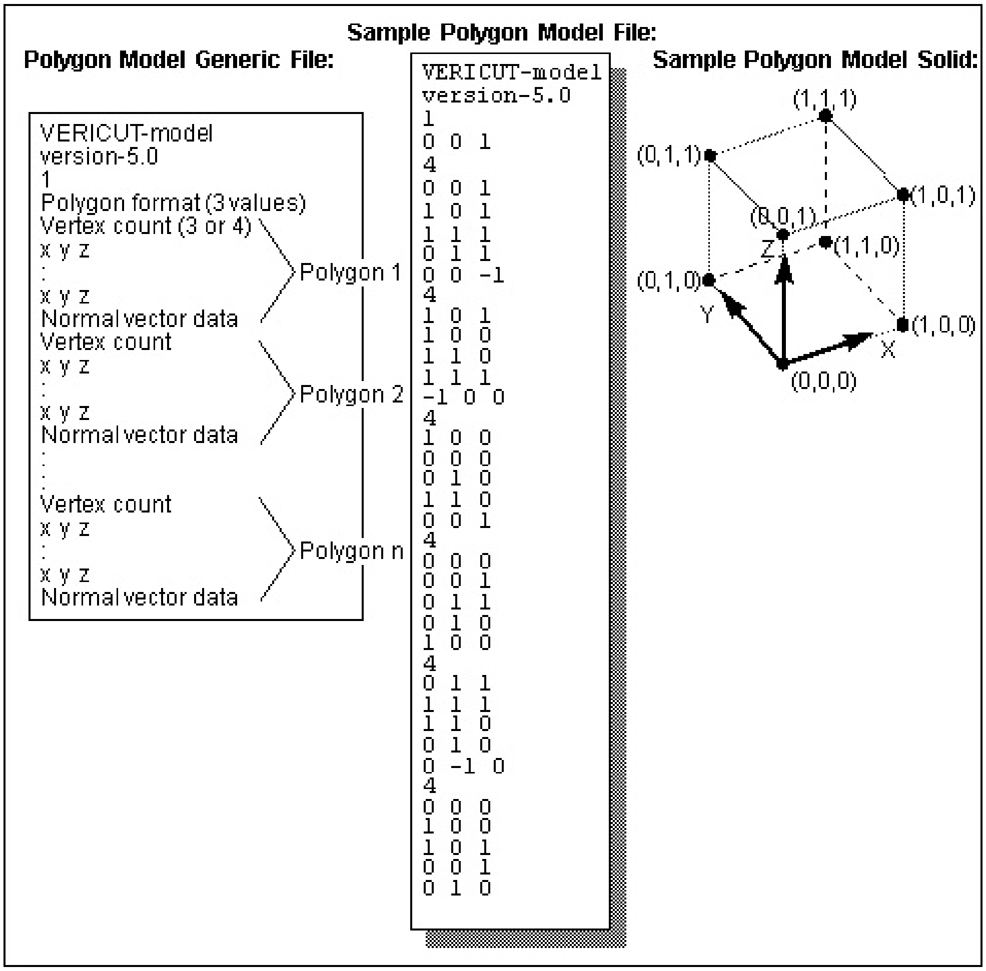
vertex count (3 or 4)
X Y Z (vertex 1)
X Y Z (vertex 2)
X Y Z (vertex 3)
I J K (normal)

Format 2 - 1 normal per vertex:

vertex count (3 or 4)
X Y Z (vertex 1)
X Y Z (vertex 2)
X Y Z (vertex 3)
I J K (normal 1)

I J K (normal 2)
I J K (normal 3)

Sample Polygon file:



VERICUT Solid File

Typical file extension: .vct

A VERICUT Solid file is a binary file containing data representing a cut stock model. The VERICUT Solid enables you to use a previously created cut stock model in a new VERICUT session as a stock, fixture or design model. You can also use a VERICUT Solid as a design model for Constant Gouge Check.

A cut stock model can be saved as a VERICUT Solid using either **File menu > Save Cut Stock > VERICUT Solid** in the VERICUT Menu Bar, or using the **File > Save Cut Stock** option in the **Configuration menu > Component Tree window**, or automatically via the **File menu > AutoSave** function. You can also save a VERICUT Solid file using the SAVE_VCT, VERICUT-COMMAND record (ref. to VERICUT-COMMAND Record in *Automating VERICUT*, in the *CGTech Help Library*).

VERICUT Solid (.vct) files are by default compressed when they are saved. VERICUT can read these compressed files directly.

To disable writing compressed VERICUT Solid files, set the environment variable CGTECH_COMPRESS=NO (or IP). (ref. **Environment Variables** in the *VERICUT Help* section, in the *CGTech Help Library* for additional information).

A cut stock model can be saved as a VERICUT Solid file "with features" or "without features". When the cut stock model is saved "with features", the cut database and the full history database are stored in the VERICUT Solid file in addition to the geometry data.

Original cut and stock colors are also preserved if the "**Save With Features**" option is used. When the file is loaded into VERICUT, the color options in the Model Definition window are ignored and the colors specified in the Shade Color list (ref. **Color window: Define tab**, also in the *VERICUT Help* section) are used. Only indices into the Shade Color list are stored. The actual colors displayed after loading the VERICUT Solid file into a new VERICUT session will only be the same if the Shade Color list is the same. Otherwise, the colors will be re-assigned according to the colors in the new Shade Color list. Pre-6.0 VERICUT Solid files are compatible and should correctly reproduce cut colors when the file is loaded into a V6.0, or later, VERICUT session. The original stock colors were not saved in the pre-6.0 VERICUT Solid files. When such a file is loaded into V6.0, its uncut stock colors will be assigned according to the Model Definition color setting for the model.

When the cut stock model is saved "without features", only the geometry data is saved in the VERICUT Solid file.

Toggle the **Save with Features** option, on the Save Cut Stock window, **On** (the default) or **Off**.

VERICUT Solid files are loaded into VERICUT using the **Project menu > Setup Models > Define function.**

When a VERICUT Solid file, saved with features, is loaded into a new VERICUT session:

Before cutting:

Modeling window > Position tab >Assemble tab features, **Mate** and **Align**, recognize the original features saved in the VERICUT Solid.

X-Caliper treats the VERICUT Solid in an approximated (triangulated) way, the same as any other primitive (Swept Solid, Solid of Revolution (SOR), etc.).

When cutting begins:

If the VERICUT Solid model was saved "with features", then the history database is used to create the new cut stock from VERICUT Solid model with the new settings. This is usually is much faster and more accurate than using a VERICUT Solid saved "without features", and restores all features of the original cut stock saved into the VERICUT Solid file.

X-Caliper:

Once the new cut stock is created, all X-Caliper measurements on a VERICUT Solid model, saved "with features", are exact.

NOTES:

1. VERICUT Solid models are not recognized in FastMill Cut Mode or when using AUTO-DIFF Surface.
2. AUTO-DIFF and Die Sinking models are always saved without features (features are destroyed by the tolerance offsets).

VERICUT Solid-of-Revolution File

Typical file extension: .sor

A VERICUT Solid-of-revolution file, or "Solid-of-revolution file" for short, is an ASCII text file that describes a revolved shape. A 2-D profile is defined in the ZX plane, then revolved around the model's Z axis.

Solid-of-revolution files are loaded into VERICUT via the **Project menu > Setup Models > Define** function, and contain the following data:

File type record — Specifies the type of VERICUT file, for example "VERICUT-model".

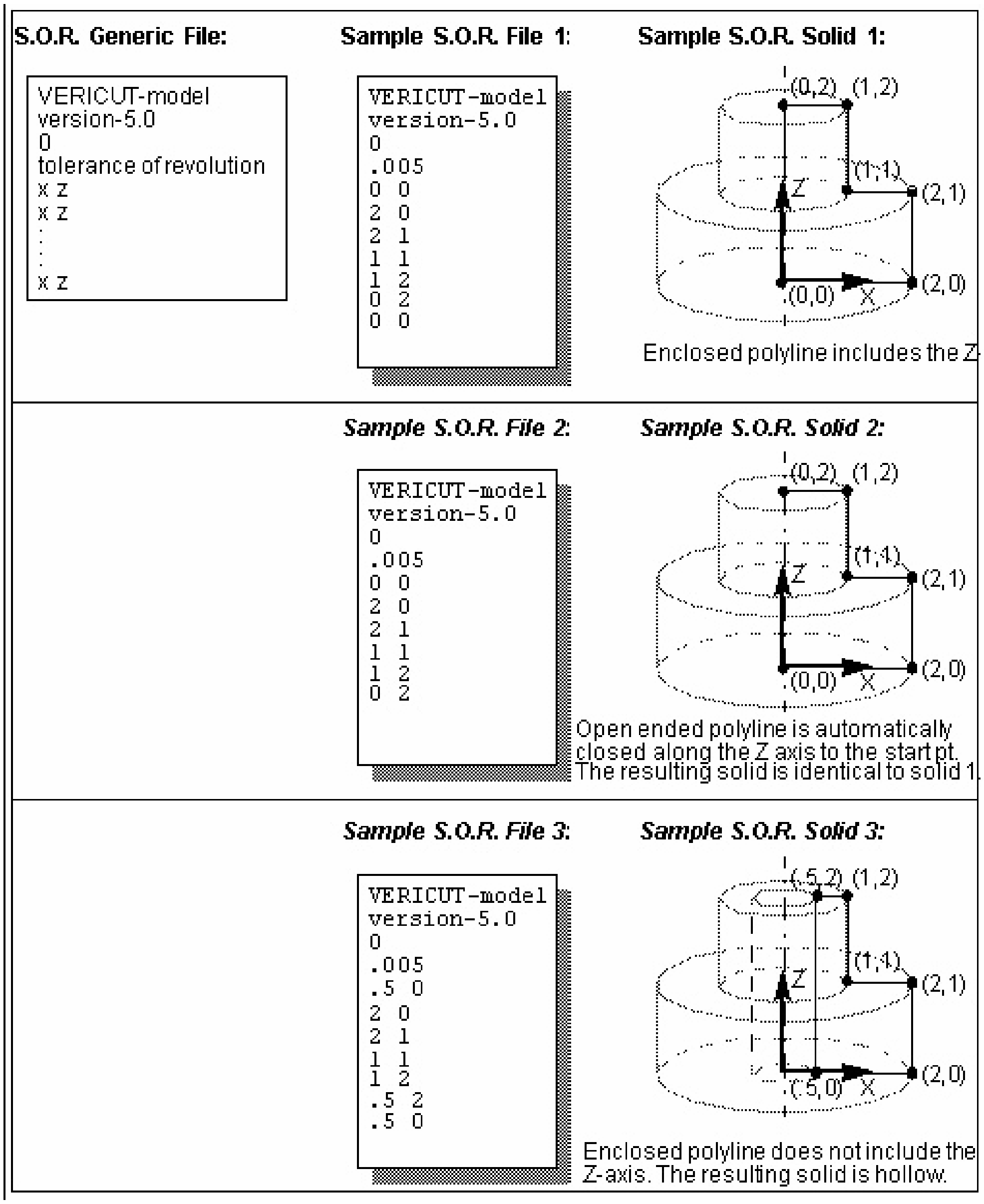
Version number record — Software version in which the file is to be used, for example "version-5.0".

Model file identifier record — Model file type: "0" identifies a Solid-of-revolution file.

Tolerance of revolution — Chordal tolerance (intol) of the revolved surface, controls the S.O.R.'s degree of roundness for the model database.

Polyline data — The remaining records in the file are the XZ point pairs which define the polyline to revolve. Each X and Z value must be on a separate line with a space between values. There is no limit on the number of point pairs allowed, but there must be at least 3 pairs. X values must be greater than or equal to zero. The polyline can be completely enclosed or open ended, but can not cross over itself. Open ended polylines are automatically closed to the start point.

Sample Solid-of-revolution files:



Library Files

Library File Overview

This section provides information about the library files installed with VERICUT ("library" directory in your VERICUT installation). Library files are included in every installation, and include:

General Purpose Library Files

The following types of General Purpose Library Files are included:

VERICUT default files — Default files opened by VERICUT. The files are configured to perform a short simulation when Play is pressed. Two sets of files provide demonstrations for inch (vericut.*) and metric (vericutm.*) environments.

VERICUT initialization files — Initialization files opened by VERICUT via File menu > New Project. Two sets of files provide initialization for inch (init.*) and metric (initm.*) environments.

Library Control Files

Library Control Files are example NC machine controls intended to help you configure VERICUT for simulating G-Code NC Programs that run on popular NC machining centers.

Library Machine Files

Library Machine Files are example NC machines intended to help you configure VERICUT for simulating G-Code NC Programs that run on popular NC machining centers.

General Purpose Library Files

The general purpose library files are listed below in alphabetical order.

Library file name:	Usage:	Notes:
2_axis_lathe_template_inch.vcproject **2_axis_lathe_template_metric.vcproject** **3_axis_mill_template_inch.vcproject** **3_axis_mill_template_metric.vcproject** **4_axis_mill_template_inch.vcproject** **4_axis_mill_template_metric.vcproject**	These files are sample template files to be used with the CAM Interfaces (ref. to the **Converters and CAD/CAM Interfaces** section, in the *CGTech Help Library*)	These files are intended to be used as a starting point for creating your own template files. Change the machine and control files, and any settings as appropriate for your environment.
init.VcProject (in), **initm.VcProject** (mm)	Initialization files for VERICUT- clears settings from the prior simulation, then configures VERICUT with default settings for inch (init.*) or metric (initm.*) environments.	Automatically loaded by VERICUT via selecting **File menu > New Project > Inch** (Millimeter).
vericut.VcProject (in), **vericutm.VcProject** (mm)	Default files- files configured to perform a short simulation of a three setup project, using three different machines, when Play is pressed.	Demonstrates how VERICUT identifies many common NC tool path problems.

Library Control Files

This section describes Control files available in the CGTech library ("library" directory in your VERICUT installation). These files enable you to quickly configure VERICUT to simulate how features of various NC controls affect the machining process. A Control file is loaded into VERICUT via the **Configuration menu > Control > Open** function, or by opening a Project file configured to use the control.

Available NC controls are listed below in alphabetical order, by control manufacturer and model. See also the accompanying notes file for details (same base file name with .txt extension-located in the "library" directory).

NOTE: Customized machines and controls can be found in the "samples" directory in your VERICUT installation. See "**Sample-Demo Files**" in the *CGTech Help Library* for more information.

Control Manufacturer	Control Model	Control File Name
Acramatic (by Cincinnati)		
	850MC	acr850mc.ctl
	950MC	acr950mc.ctl
	2100	acr2100.ctl
	CNC PC	acrcncpc.ctl

Allen-Bradley		
	7320	ab_7320.ctl
	8200	ab_8200.ctl
	8600	ab_8600.ctl
	9260	ab_9260m.ctl

Bosch		
	CC300	bshcc300.ctl

Boston Digital		
	BDC 3200 (inch)	bdc3200_in.ctl
	BDC 3200 (mm)	bdc3200_mm.ctl

Charmilles		
	200 Wire EDM	chr200.ctl

Cincinnati - see "Acramatic"		

Fadal		
	CNC 88, Format 1 ("Fadal" mode)	fad88a.ctl
	CNC 88, Format 2 ("Fanuc" mode)	fad88.ctl

Fanuc		
	Series 0	fan0m.ctl (mill) fan0t.ctl (turn)
	Series 3	fan3t.ctl (turn) fan3tt.ctl
	Series 5	fan5m.ctl (mill)
	Series 6	fan6m.ctl (mill) fan6t.ctl (turn)
	Series 9	fan9t.ctl (turn)
	Series 10	fan10m.ctl (mill) fan10t.ctl (turn)

	Series 11	fan11m.ctl (mill) fan11t.ctl (turn)
	Series 12	fan12m.ctl (mill) fan12t.ctl (turn)
	Series 15	fan15im.ctl (model "I" mill) fan15it.ctl (model "I" turn) fan15m.ctl (mill) fan_15m_custom_optimizable.ctl fan15t.ctl, (turn) fan15t_t.ctl (turn-turret) fan15t_t_ws.ctl
	Series 16	fan16im.ctl (model "I" mill) fan16it.ctl (model "I" turn) fan16m.ctl (mill) fan16t.ctl, (turn) fan16w.ctl (wireEDM) mori_seiki_mt2000sz_fan16it.ctl
	Series 18	fan18im.ctl (model "I" mill) fan18it.ctl (model "I" turn) fan18m.ctl (mill) fan18t.ctl (turn)
	Series 21	fan21im.ctl (model "I" mill) fan21it.ctl (model "I" turn) fan21m.ctl (mill) fan21t.ctl (turn)
	Series 30	fan30im.ctl (model "I" mill)
	Series 31	fan31im.ctl (model "I" mill)
	Series 32	fan32im.ctl (model "I" mill)

	Series 150	fan150im.ctl (model "I" mill) fan150it.ctl (model "I" turn) fan150m.ctl (mill) fan150t.ctl (turn)
	Series 160	fan160im.ctl (model "I" mill) fan160it.ctl (model "I" turn) fan160m.ctl (mill) fan160t.ctl (turn)
	Series 180	fan180im.ctl (model "I" mill) fan180it.ctl (model "I" turn) fan180m.ctl (mill) fan180t.ctl (turn)
	Series 210	fan210im.ctl (model "I" mill) fan210it.ctl (model "I" turn) fan210m.ctl (mill) fan210t.ctl (turn)
	Series 300	fan300im.ctl (model "I" mill)
	Series 310	fan310im.ctl (model "I" mill)
	Series 320	fan320im.ctl (model "I" mill)
	Series 3000	fan3000c.ctl

Fidia		
	M30	fidia_m30.ctl

GEMINI **(by Kearney & Trecker)**		
	GEMINI-C	kt_c.ctl
	GEMINI-D	gemini_d.ctl
	GEMINI-E	gemini_e.ctl

General Electric (GE)		
	1050MC CNC	ge_1050mc.ctl
	2000MC CNC	ge_2000mc.ctl

Generic (Fanuc-like) controls		
	decimal	generic.ctl (in) genericm.ctl (mm)
	leading zeros	genlead.ctl (in) genleadm.ctl (mm)
	trailing zeros	gentrail.ctl (in) gentrailm.ctl (mm)

Giddings & Lewis (G&L) **see "NumeriPath"**		

HAAS		
	CNC	hascnc.ctl
	MiniMill	haas_minimill.ctl

Heidenhain		
	Mill Plus	heimplus.ctl
	TNC 407	hei407.ctl (conversational) hei407g.ctl (ISO)
	TNC 415b	hei415b.ctl (conversational) hei415bg.ctl (ISO)
	TNC 415c	hei415c.ctl (conversational) hei415cg.ctl (ISO)

	TNC 425	hei425.ctl (conversational) hei425g.ctl (ISO)
	TNC 426	hei426.ctl (conversational) hei426g.ctl (ISO)
	TNC 430	hei430.ctl (conversational) hei430g.ctl (ISO)
	TNC530	hei530.ctl

Kearney & Trecker - see "GEMINI"		

Lumonics		
	Laserdyne System 94	laserdyne_94.ctl

Maho - see "Phillips"		

Makino		
	Professional Series 3	makpro3.ctl
	Professional Series 5	makpro5.ctl makpro5_mag3.ctl

Mazak		
	Matrix	mazak_matrix_nexus510c.ctl
	Mazatrol Fusion 640	maz640m.ctl (mill) maz640mpro.ctl (mill) maz640t.ctl (turn) maz640t_t.ctl maz640mpro_mazak_e650h_2000u.ctl maz640m_mazak_nexus410a.ctl maz640m_variaxis.ctl mazak_e650h_2000u_640mpro.ctl mazak_nexus410a_maz640m.ctl

	Mazatrol M-2	mazm2.ctl
	Mazatrol M-3	mazm3.ctl
	Mazatrol M-32	mazm32.ctl
	Mazatrol T-2	mazt2.ctl
	Mazatrol T-3	mazt3.ctl
	Mazatrol T-32	mazt32.ctl

NUM		
	1020T	num1020t.ctl
	1040T	num1040t.ctl
	1060M	num1060m.ctl
	1060T	num1060t.ctl

NumeriPath (by G&L)		
	800M	nph800m.ctl
	8000L	nph8000l.ctl
	8000M	nph8000m.ctl

Okuma - see "OSP"		

Olivetti		
	MC8600	oli8600.ctl

OSAI		
		osai_10.ctl

OSP (by Okuma)		
	OSP700L	osp700l.ctl
	OSP5000L	osp5000l.ctl
	OSP5000M	osp5000m.ctl
	OSP7000L	osp7000l.ctl osp7000l_t.ctl (turret)
	OSP7000M	osp7000m.ctl

PaR Systems		
	Cimroc 5000	cimroc_5000.ctl

Phillips		
	CNC 432	phi432.ctl
	CNC 532	phi532.ctl

Sinumerik (by Siemens)		
	3T, 3TT	sin3t.ctl, sin3tt.ctl
	810	sin810d.ctl (810D)
	840	sin840c.ctl (840C) sin840d.ctl (840D) index_sin840c.ctl (840C) sin840d_frames.ctl
	880M	sin880m.ctl (880M)
	FM-NC	sinfmnc.ctl

Sundstrand		
	Micro SWINC LM	swilm.ctl
	Micro SWINC M200	swim200.ctl

Thermwood		
	91000 Super Control	thermwood_91000.ctl

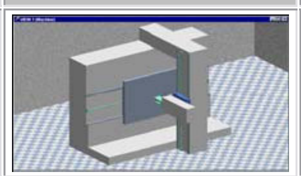
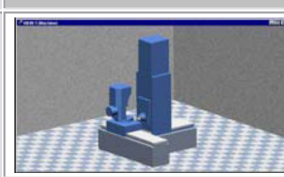
Tosnuc (by Toshiba)		
	888	tos888.ctl

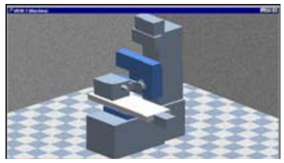
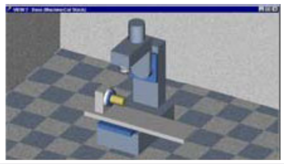
Yasnac		
	i80m	yasi80m.ctl
	MX-3	yasmx3.ctl

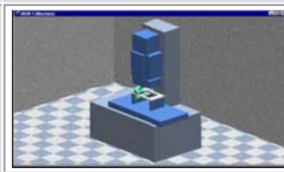
Library Machine Files

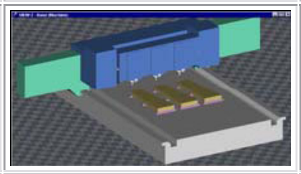
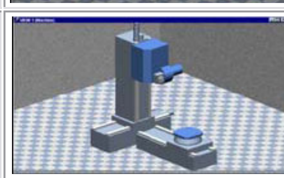
This section describes Machine files available in the CGTech library ("library" directory in your VERICUT installation) and CGTech sample files ("samples" directory). These files enable you to quickly configure VERICUT to simulate motions on various NC machines. Note that while library machines are generally considered ready to use, many of the sample machines have job-related data that may interfere when configuring for use with new NC tool paths. A Machine file is loaded into VERICUT via the **Configuration menu > Machine > Open** function, or by opening a Project/User file configured to use the machine.

Available NC machines are listed below in alphabetical order, by manufacturer and model. All machines reside in the "library" directory of your VERICUT installation unless otherwise noted to be in "samples" or "showroom". See also the accompanying notes file for details (same base file name with .txt extension-located in the "library" directory). See "**Sample-Demo Files**" in the *CGTech Help Library* for more information.

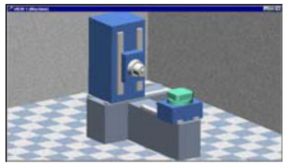
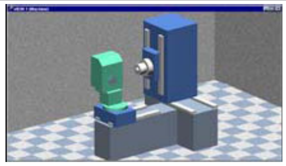
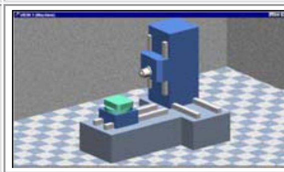
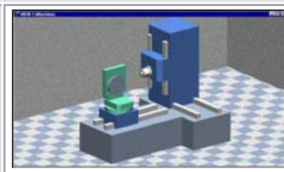
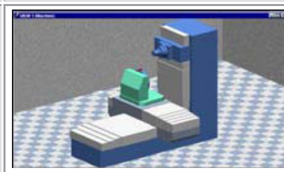
Machine Manufacturer	Machine Model	Thumbnail	Machine File Name
Aerostar			
	H2200 5ax horiz. mill		aerh2200.mch
Bohle			
	BW120S 5ax horiz. mill		bohbw120.mch

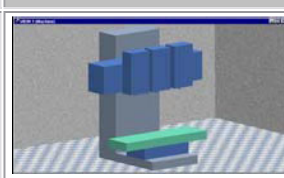
Bostomatic			
	405 5ax vert. mill		bos405.mch
	5ax vert. mill		samples - bos5vm01.mch

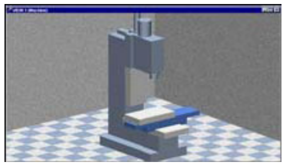
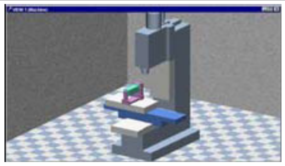
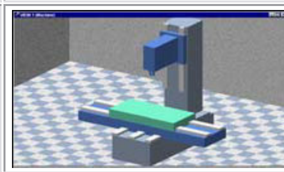
Charmilles			
	Robofil 200 4ax wire EDM		chr200.mch

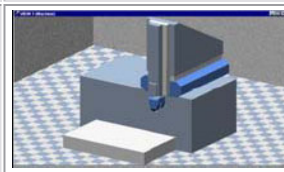
Cincinnati			
	3-spindle 5ax gantry mill		samples - cinci_3spindle_5ax.mch
	5ax dual- tilting head vert. mill		samples- cin5vm01.mch
	T30 5ax horiz. mill		cint30.mch (inch) cint30m.mch (mm), cint30_2.mch,

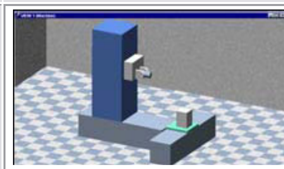
Deckel Maho - see "Maho"			

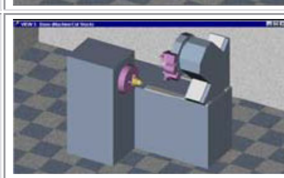
Dixi			
	100 4ax horiz. mill		dix100m4.mch
	100 5ax horiz. mill		dix100m5.mch
	150 4ax horiz. mill		dix150m4.mch
	150 5ax horiz. mill		dix150m5.mch
	350 TCA50S 5ax horiz. mill		dix350m.mch
	DPH-80 4ax horiz. mill		dixi_dph80.mch

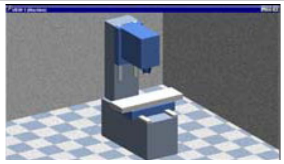
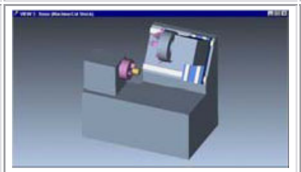
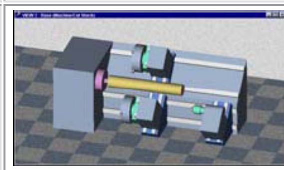
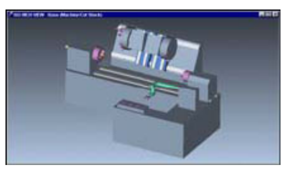
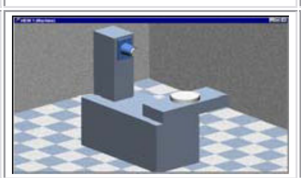
Droop and Rein			
	DR969 vert. 3 spindle mill		dr_dr969.mch

Fadal			
	VMC 15 3ax vert. mill		fadvmc15.mch
	VMC 4020 4ax vert. mill		fadal_vmc4020.mch
	VMC 6030 3ax vert. mill		fadal_vmc6030.mch

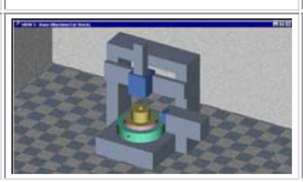
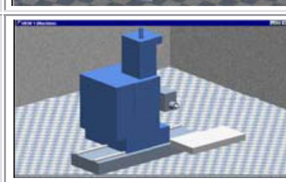
Fidia			
	218 5ax vert. mill		fidia_218.mch

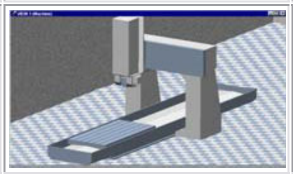
Forest			
	Flexiax 510 5ax horiz. mill		forfx510.mch

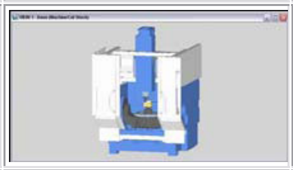
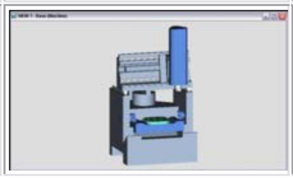
Generic machines			
	2ax horiz. lathe		generic_2ax_turn.mch
	2ax horiz. turret lathe		samples-2hl02.mch (shown), 2hl03.mch

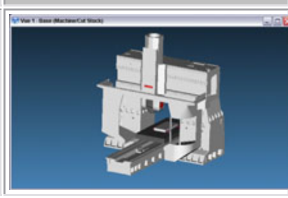
	3ax vert. mill		g3vm.mch generic_vmill_3ax.mch generic_vmill_3ax_3d.mch
	4ax vert. mill-table A	Not 3-D	g4vmta.mch
	3ax vert. mill with robot gripper arm		samples - bars_robot.mch
	3ax horiz. mill/turn-table C		samples - 3ht.mch
	4ax horiz. dual turret lathe		samples - 4hl02.mch (steadyrest), 4hl03.mch (no steadyrest), 4hl04.mch (steadyrest)
	4ax horiz. dual turret lathe with pick-off spindle		samples - mcdturn6.mch (robot arm part-grabber)
	4ax horiz. mill-table B		library - generic_vmill_4ax_table_b.mch samples - 4hmtb.mch (shown)
	4ax horiz. mill/turn-table C		samples - 4ht.mch (in), 4htm.mch (mm)
	4ax vert. mill-table A	Not 3-D	generic_vmill_4ax_table_a.mch

	5ax horiz. mill-head A, table B	Not 3-D	generic_5ax_hmill_heada_tableb.mch
	5ax horiz. mill-table A on table B	Not 3-D	generic_5ax_hmill_tablea_tableb.mch
	5ax horiz. mill-table B on table A	Not 3-D	generic_5ax_hmill_tableb_tablea.mch
	5ax vert. mill-head A on head B	Not 3-D	generic_5ax_vmill_heada_headb.mch
	5ax vert. mill-table A, head B	Not 3-D	generic_5ax_vmill_tablea_headb.mch
	9ax cutter grinder		samples - 9v_grind.mch

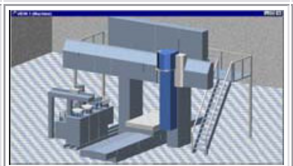
Giddings & Lewis (G&L)			
		Not 3-D	gl_2ax_turn.mch
	5ax vert. mill/turn-table C		samples- gl_5vt01.mch
	G60 4ax horiz. boring mill		gl_g60.mch

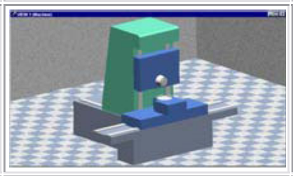
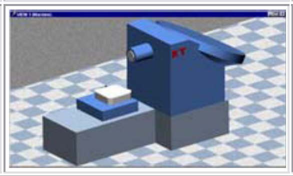
HAAS			
	MiniMill 3ax vert. mill		haas_minimill.mch
	VB-1 5ax vert. mill		hasvb1.mch

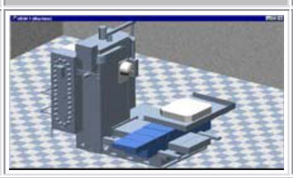
Hermle			
	C30 5ax mill		hermle_c30.mch
	C800U 5ax mill		hermle_c800u.mch

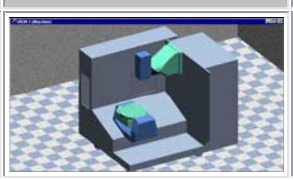
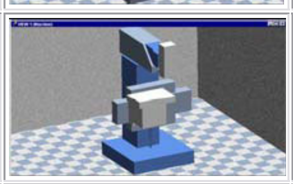
Huron			
	KX 100 5ax mill		huron_kx100.mch

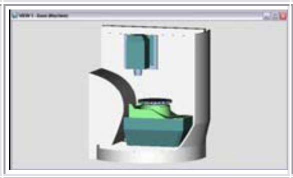
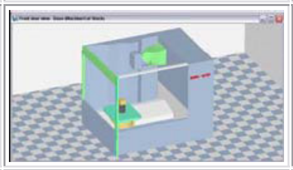
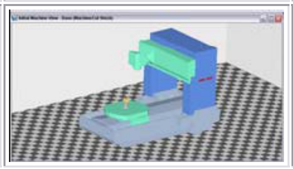
Index			
	G200 4ax mill/turn		index_g200.mch

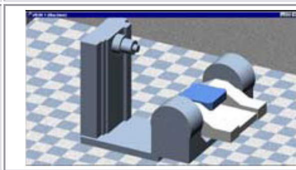
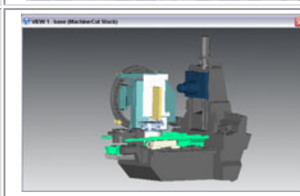
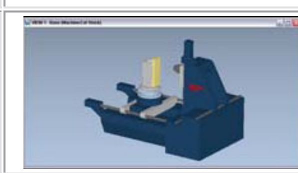
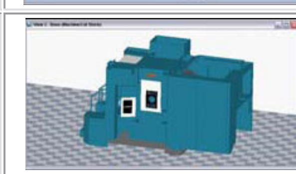
Ingersoll			
	Mastercenter 4ax Multi-head gantry mill-detailed		ingmast1.mch

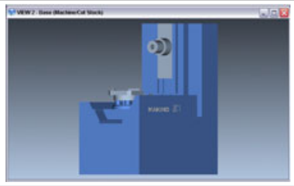
Kearney & Trecker (K&T)			
	HB4 4ax horiz. mill		kt_hb4.mch
	Model 180 4ax horiz. mill		kt_180.mch

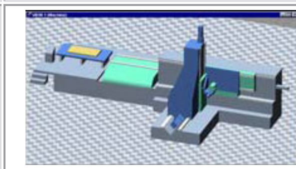
Kuraki			
	KBM11 5ax horiz. boring mill		kurkbm11.mch

Maho			
	DMU 80 P 5ax horiz. mill		mahdmu80.mch
	MH700 3ax vert. mill		mah700m.mch (mm), mahmh700.mch (in)
		Not 3-D	mahvrt.mch

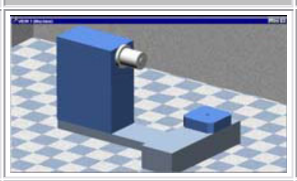
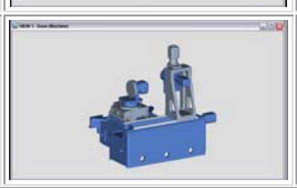
	DMU 50V 5ax vert. mill		dmg_dmu50v.mch
	DMU 60 t 5ax vert. mill		dmg_dmu60t.mch
	DMU 125 P 5ax vert. mill		dmg_dmu125p.mch
	DMU 200 P 5ax vert. mill		dmg_dmu200p.mch

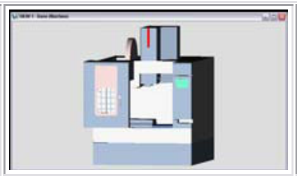
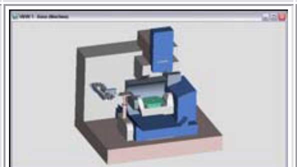
Makino			
	MC1010 5ax horiz. mill		mak1010.mch
	A51 4ax horiz. mill		makino_a51.mch
	A55 4ax horiz. mill		makino_a55.mch
	A66 5ax horiz. mill		makino_a66.mch
	A77 4ax horiz. mill		makino_a77.mch

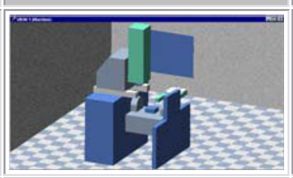
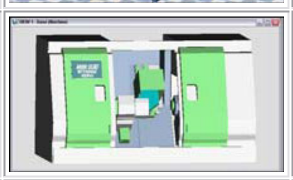
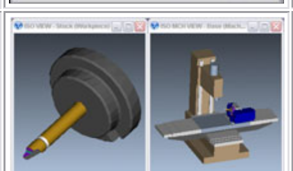
	A71 3ax horiz. mill		makino_a71.mch

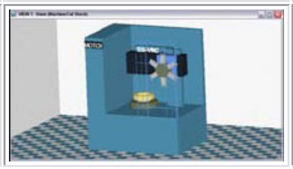
Marwin			
	Advanced Manufacturing Cell 5ax horiz. mill		mrwamc.mch

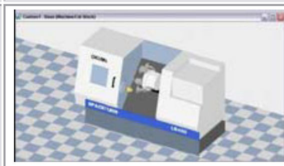
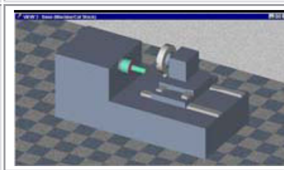
Matsuura			
	FX-5 4ax vert. mill	Not 3-D	matfx5.mch
	MAM 600HF vert. mill	Not 3-D	mat600hf.mch

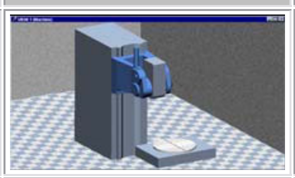
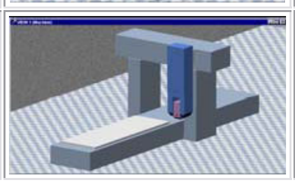
Mazak			
	H500 4ax horiz. mill		mazh500.mch
	Super 400H 4ax horiz. mill		maz400h.mch
	E650H horiz. mill/turn		mazak_e650h_2000u_3_jaw.mch
	HTC 4000 5ax horiz. mill		mazak_htc_4000.mch

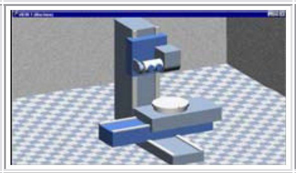
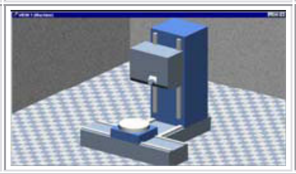
	Nexus 410A vert. mill		mazak_nexus410a.mch
	Nexus 510C 3ax vert. mill		mazak_nexus_510c.mch
	QTN250MY		mazak_qtn250my.mch
	Variaxis 500 5ax vert. mill		mazak_variaxis500.mch
			mazak_variaxis730.mch

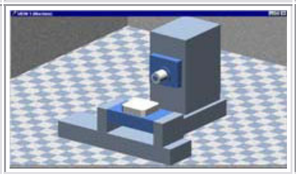
Mori Seiki			
	Partner M-400C1 5ax vert. mill		ms_m400.mch
	MT 2000 SZ multi-axis turning center		mori_seiki_mt2000sz_alpha1.mch
	Mori Seiki SV500 3ax vert. machining center (with Tsukakoma TRNCB 201 double rotary table)		mori_seiki_sv500.mch

Motch			
	135 VNC vert. turning center		motch_135vnc.mch

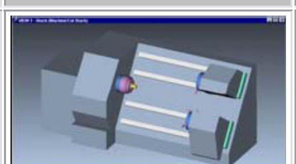
Okuma			
	LB400 2ax horiz. lathe		Okuma_LB400.mch
	LB25 2ax horiz. lathe		samples - okulb25.mch
	MB-46V 3ax vert. mill		okuma_mb46v_osp7000m.mch

SNK			
	FSP-100V 5ax horiz. mill		snkfsp01.mch
	SH-200 5ax gantry mill		snksh200.mch

Sundstrand			
	Omnimil OM3 5ax vert. mill		sunom3.mch
	Omnimil OM80 5ax horiz. mill		sunom80.mch

Toyoda			
	FH80 5ax horiz. mill		tydfh80.mch

Tsugami			
	MA3 4ax horiz. mill	Not full 3-D	tsuma3.mch

Warner & Swasey			
	Titan 4ax horiz. dual turret lathe		ws_titan.mch

Sample–Demo Files

See the Sample-Demo Files section in the *CGTech Help Library*.

Environment Variables

You can set environment variables to help customize the VERICUT user interface, alter VERICUT performance, and perform other various actions. This section describes environment variable formats and the available environment variables that can affect the VERICUT simulation.

Environment variables are typically set via CGTech command files located in the CGTech "commands" directory, under the directory representing your computer type. For example, the command files or VERICUT 6.1 installed on the "C:" drive of a Windows NT computer are located in the C:\CGTech61\winnt\commands directory. Environment variables can also be set in any other system file used to execute VERICUT.

Formats

Formats used to set environment variables differ, based on the computer's operating system. Consult the operating system documentation, or refer to the CGTech-installed command files on your computer for specific examples.

Operating System	Environment Variable Format	Examples of Use
Windows	set ENV_VAR_NAME=value	set CGTECH_DEFAULT_UNITS=inch set CGTECH_LIBRARY=C:\CGTech50\library
UNIX	ENV_VAR_NAME=value export ENV_VAR_NAME (2 records required to set each variable)	CGTECH_DEFAULT_UNITS=inch export CGTECH_DEFAULT_UNITS CGTECH_LIBRARY=C:\CGTech50\library export CGTECH_LIBRARY

"CGTECH" Environment Variables

Available environment variables that can affect the VERICUT simulation are listed below in alphabetical order. For simplicity, all examples of use below are in Windows format. See above for UNIX format examples.

CGTECH_CEILING_IMAGE — Specifies the JPEG file to use as texture for the ceiling in OpenGL machine views.

Example of use:

set CGTECH_CEILING_IMAGE=C:\path\ceiling.jpg

CGTECH_CODEC_FILTER — Enables you to remove invalid codecs from the Select a CODEC pull-down list in the Image Record window (**File > Images > Record Movie**, **Output Format** set to **AVI**)

Example of use:

set CGTECH_CODEC_FILTER=CVID Y411

The above removes the CVID and Y411 options from the list. Use a blank space to separate options.

CGTECH_COMPRESS — By default, In Process (IP) files and VERICUT Solid (.vct) files are compressed when they are saved. VERICUT can read these compressed files directly. CGTECH_COMPRESS enables you to disable writing compressed In Process files and/or VERICUT Solid files.

Valid settings are:

NO = no compression

IP = compress IP files only

VCT = compress VCT files only

ALL = compress both IP and VCT files (Default)

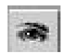
Example of use:

set CGTECH_COMPRESS=ALL

CGTECH_DEFAULT_UNITS — Sets the default units for VERICUT. Specify either INCH or MILLIMETER

Example of use:

set CGTECH_DEFAULT_UNITS =MILLIMETER

CGTECH_DISPLAY — Sets the animation display method for tool motion in the Workpiece view. The display is updated for animation when you start cutting or the refine the display (e.g. press Refine Display ). Changing the animation display method can affect simulation speed and display quality, but does not affect model accuracy. Results are different, based on the types of parts you cut and specified Cutting Tolerance.

Options:

REFINE1 — This display method creates the best quality animation image for all-around use, but may take slightly longer to simulate with than REFINE2 (see below).
REFINE2 — Alternate display method-fastest when a large amount of model detail is visible. This method is more sensitive to Cutting Tolerance than REFINE1, and is typically very good when the cut tolerance is small enough.
ULTRA — Another alternate display method-this method can be faster than the others, but has lower display quality when compared to the default method.

Example of use:

set CGTECH_DISPLAY=REFINE2

CGTECH_FLOOR_IMAGE — Specifies the JPEG file to use as texture for the floor in OpenGL machine views.

Example of use:

set CGTECH_FLOOR_IMAGE=C:\path\floor.jpg

CGTECH_FONT_SIZE — Sets the font size for text appearing in the VERICUT user interface. Size can be given an absolute point size, or incremental value from the default value of "12".

Examples of use:

set CGTECH_FONT_SIZE=14 (sets 14 pt. font)
set CGTECH_FONT_SIZE=+2 (also sets 14 pt. font: default 12 +2= 14)

CGTECH_FSB_ICONS — Environment variable CGTECH_FSB_ICONS is used to permit suppression of attempts to get individual icons for each branch of the file system tree. With "set CGTECH_FSB_ICONS=No" (or Non, Nein, Nyet or anything that starts with an "N") each branch will use a simple folder icon.

Examples of use:

set CGTECH_FSB_ICONS=No

This environment variable was added to resolve a situation where VERICUT hangs when opening a File Selection Box (FSB) when a Samba drive is mounted on a network.

- If the Samba drive is mounted in Windows via a UNC path, then VERICUT's FSB will always hang while opening. The hang occurs the second time a VERICUT FSB is opened. Thus you can successfully browse to set your working dir, but then hang when opening the FSB to get the next file (IP, project, etc).
- If the Samba drive is mounted in Windows via a drive letter, then the hang is more sporadic. But it still hangs while the FSB is opening.

It is VERY IMPORTANT to realize that the user does not need to access the Samba drive for the problem to happen. It simply needs to be mounted on his computer.

Use of this environment variable eliminates going thru Samba to get each folder's "desktop.ini" file.

CGTECH_INSPECTION_TOLERANCES — Specify a fully qualified /path/filename of a "custom" inspection tolerance file" containing default tolerances to be used for Inspection Sequence.

Example of use:

set CGTECH_INSPECTION_TOLERANCES=C:\mydir\inspect_tolfile.txt

CGTECH_LIBRARY — Specifies the directory location of the CGTech Library of machines and controls accessed via choosing the "CGTECH_LIBRARY" shortcut option on file selection windows. Setting this environment variable can be useful for sites that use their own library VERICUT machines and controls instead of that supplied on CD with VERICUT. The full directory path can be included.

Example of use:

set CGTECH_LIBRARY=C:\mylibrarydir

See also: "Where VERICUT looks for files" in the *Introduction* section

CGTECH_MACHINE_CONFIG — Enables you to turn off the display of the **Configuration** menu, in the VERICUT menu bar. Set to FALSE to remove the display.

Example of use:

set CGTECH_MACHINE_CONFIG=FALSE

CGTECH_MATRIX_FORMAT — Enables you to view the Matrix Tables in the **Coordinate System window: Matrix tab**, the **Machine Offsets window**, and the **Modeling Window: Position tab: Matrix tab** with the I, J, K values along the vertical axis and the X, Y, Z along the horizontal axis rather than the "default" horizontal orientation.

Horizontal Orientation (default):

	I	J	K	D
X	1.00000000	0.00000000	0.00000000	0.00000000
Y	0.00000000	1.00000000	0.00000000	0.00000000
Z	0.00000000	0.00000000	1.00000000	3.00000000

Example of use:

set CGTECH_MATRIX_FORMAT=VERTICAL

Vertical Orientation:

	X	Y	Z	D
I	1.00000000	0.00000000	0.00000000	0.00000000
J	0.00000000	1.00000000	0.00000000	0.00000000
K	0.00000000	0.00000000	1.00000000	0.00000000

CGTECH_OLD_FSB — Enables you to use the more common Windows File Selection Box (FSB) rather than the "standard" VERICUT FSB.

Example of use:

set CGTECH_OLD_FSB=YES

CGTECH_REF_CUTTER_LIB — Enables you to specify an external Tool Library file to be used as a reference library for the Milling Tool Wizard. All cutters in the specified tool library will be listed, by Tool ID, in the Milling Tool Wizard's Cutter pull-down list.

Example of use:

set CGTECH_REF_CUTTER_LIB=C:*path**filename.tls*

CGTECH_REF_EXTENSION_LIB — Enables you to specify an external Tool Library file to be used as a reference library for the Milling Tool Wizard. All extensions in the specified tool library will be listed, by Tool ID, in the Milling Tool Wizard's Extension pull-down list.

Example of use:

set CGTECH_REF_EXTENSION_LIB=C:*path**filename.tls*

CGTECH_REF_HOLDER_LIB — Enables you to specify an external Tool Library file to be used as a reference library for the Milling Tool Wizard. All holders in the specified tool library will be listed, by Tool ID, in the Milling Tool Wizard's Holder pull-down list.

Example of use:

set CGTECH_REF_HOLDER_LIB=C:*path**filename.tls*

CGTECH_SLM_DEBUG — Enables you to write out "client side" license problem debug information to a file. This must be defined before VERICUT is run.

Example of use:

set CGTECH_SLM_DEBUG=FILE:*path**filename*

CGTECH_STATUS — Creates a Status file, an ASCII text file that contains whatever data is sent to the Status window during the simulation. The full directory path can be included with the file name.

Example of use:

set CGTECH_STATUS=C:\mydir\statusfile.txt

CGTECH_TPEDITOR — Causes Edit Toolpath to issue a system command using the string content of the variable, with the current tool path file as an argument. Thus you can choose to run an alternate tool path editor, such as any one of the clever NC code editors available today.

When you press **Edit > NC Program > 'c:\path\filename.mcd'**, or **Project menu > NC Program >(list item) > right mouse click > Edit**, or the Edit NC Program icon  , then VERICUT issues the system command:

c:\program files\cooltoolpatheditor\editor.exe c:\path\filename.mcd

This will only work with tool path editors that can take the tool path file name as an argument on the command line that runs the editor. Depending on the editor's command syntax, it may be necessary to point CGTECH_TPEDITOR at a batch or script file that intercepts the tool path file name and re-formats the command as necessary.

Example of use:

set CGTECH_TPEDITOR=c:\program files\cooltoolpatheditor\editor.exe

CGTECH_USERNAME — Specifies the user name to include in the name of the Preferences file that is automatically saved when you exit VERICUT. This environment variable is recommended to prevent users that share a machine (or an account) from overwriting the Preferences file.

Example of use:

set CGTECH_USERNAME=bob (Preferences file saved => "cgtech_bob.prefs")

CGTECH_VCPROJECT — Specifies the default Project file for VERICUT. The full directory path can be included.

Example of use:

set CGTECH_VCPROJECT=C:\mydir\myprojectfile.VcProject

CGTECH_VCUSR — Specifies the default User file for VERICUT. The full directory path can be included.

Example of use:

set CGTECH_VCUSR=C:\mydir\myuserfile.usr

CGTECH_VIEWER — Enables you to select an external editor/viewer.

Example of use:

set CGTECH_VIEWER=C:*path**editor*.exe

CGTECH_X_WALL_IMAGE — Specifies the JPEG file to use as texture for the two walls having normals along plus or minus X in OpenGL machine views.

Example of use:

set CGTECH_X_WALL_IMAGE=C:\path\x_wall.jpg

CGTECH_Y_WALL_IMAGE — Specifies the JPEG file to use as texture for the two walls having normals along plus or minus Y in OpenGL machine views.

Example of use:

set CGTECH_Y_WALL_IMAGE=C:\path\y_wall.jpg

Adding custom environment variables for shortcuts

You can add your own environment variables to the beginning section of the VERICUT command file and have them appear in the shortcut list of file selection windows. VERICUT lists all custom environment variables that begin with "CGTECH_" in the shortcut list, and have at least one file in the directory that is recognized as a CGTech file type, for example a file with any of the following extensions: *.ctl, *.ip, *.job, *.mcd, *.mch, *.olb, *.stl, *.tls, *.tp, *.usr, or *.VcProject

Note that the contents of the file are not verified unless loaded by VERICUT. Just the existence of a file with an extension that qualifies as one of the valid CGTech file types (see above) is enough to make the shortcut appear in the list.

Example 1 — The following environment variable creates a shortcut to a directory containing VERICUT 6.1 sample files.

set CGTECH_61SAMPLES=C:\CGTech61\samples\vericut

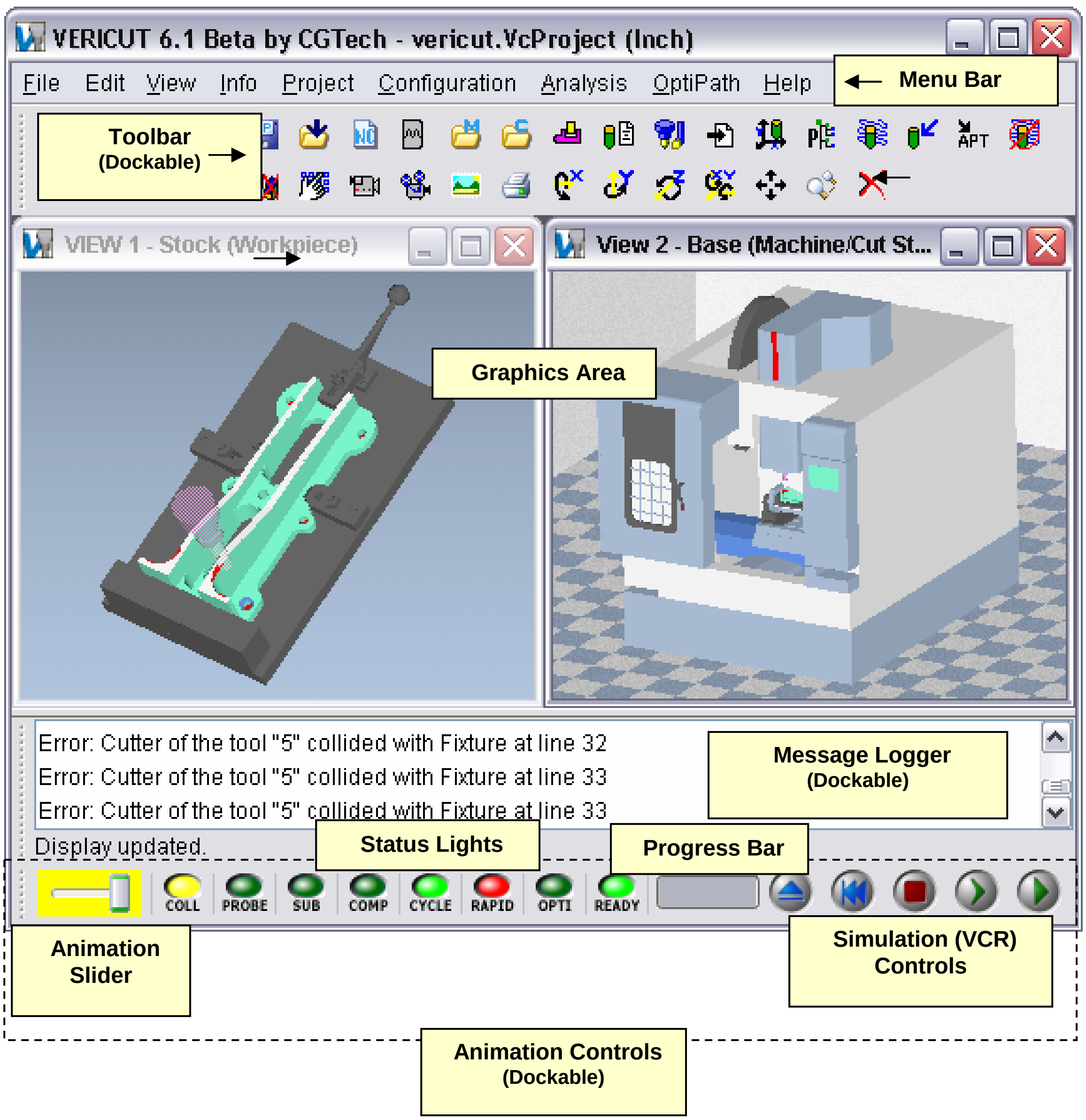
Example 2 — The following environment variable creates a shortcut to a directory containing a user's custom VERICUT files. The directory contains a file named "dummy.usr".

set CGTECH_MYVERICUTFILES=E:\myvericutfiles

Interacting with VERICUT

VERICUT main window

The VERICUT main window is composed of distinct areas, each with different user interaction. The window header displays the last Project file loaded and the current session units (inch or millimeter). This window can be resized like most other window, via dragging the window header, sides or corners.

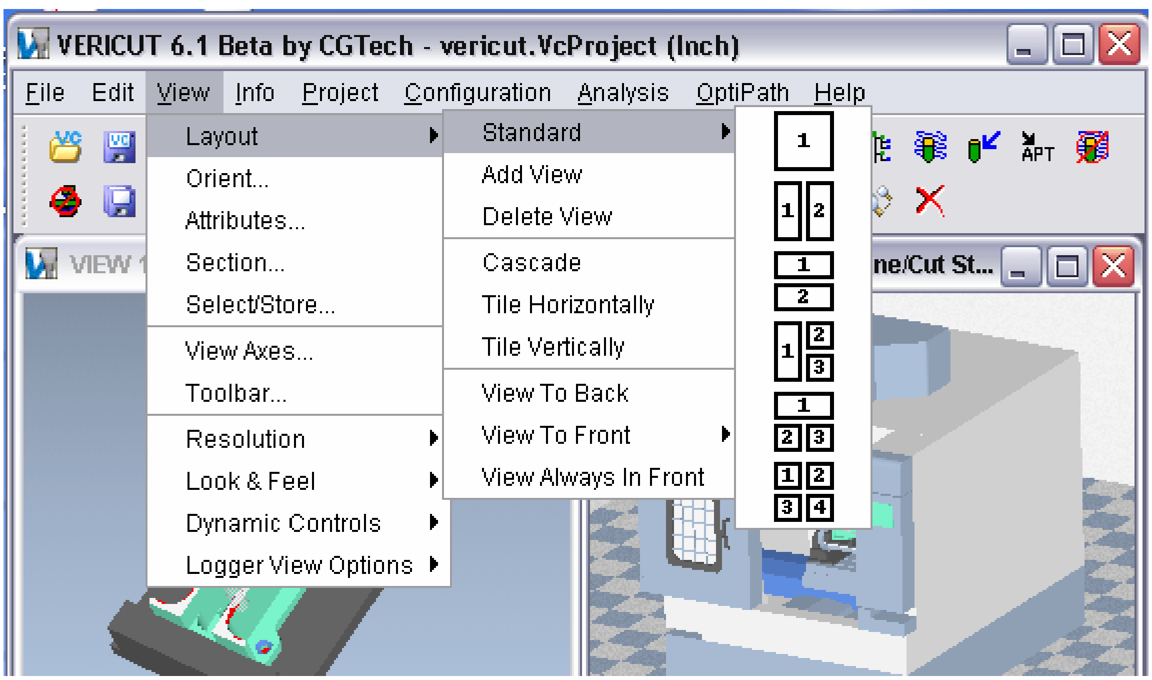


Hint: Press "F1" in any VERICUT window to receive help on that window.

Also see "**Overview of Using VERICUT**", in the *Using VERICUT* section, in the *CGTech Help Library*.

Menu Bar

The menu bar, located across the top of the VERICUT window, provides easy access to VERICUT functions. Each menu in the menu bar contains groups of related functions. Left-click on any menu name to expose the list of functions available in that menu. If an arrow appears to the right of a function in a menu, move the mouse over the arrow to expose a sub-menu of additional functions. Click on the function in the menu you want to use.



VERICUT menu descriptions

The menus which provide access to all VERICUT functions are listed below, followed by a brief description of what the menu functions do. In VERICUT Help, click VERICUT Menus & Functions, then open the desired menu and select the function for more information about each function.

File menu — Functions in this menu open and save the most common files used in VERICUT, access converter products, control properties, export cut model data, and generate VERICUT reports.

Edit menu — Functions in this menu enable you to edit text files, NC programs and control display colors.

View menu — Functions in this menu set up the number of views, orientation (angle and distance) for each view, and store or select commonly used views, and set up other VERICUT display characteristics.

Info menu — Functions in this menu access session information, such as the files currently being used, machining status, machine offsets, and log files.

Project menu — Functions in this menu provide the functions required for "job" setup including the display of the Project Tree. Functions include selection of machines, controls, NC programs, tools, and processing options. Its features also enable the selection and setup of output files created by VERICUT during processing and customized reports.

Configuration menu — Functions in this menu are used to build or configure NC machines and controls.

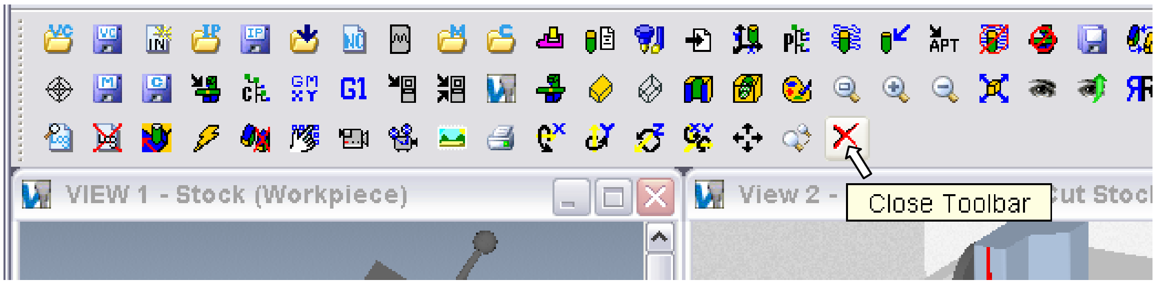
Analysis menu — Functions in this menu are used to analyze and inspect the VERICUT model. It provides access to the following features: X-Caliper, AUTO-DIFF, Comparator, NC Program Review, Inspection and Die Sinking Simulation.

OptiPath menu — Functions in this menu access OptiPath-the feed rate and spindle speed optimization software that creates "optimized" tool paths that cut parts in the least amount of time.

Also see the "**Using VERICUT**" section, in the *CGTech Help Library*.

Tool Bar

VERICUT's **Toolbar** provides quick and easy access to the most commonly-used functions/features.



To see what is associated with a **Toolbar** icon, simply position the cursor over the icon and a tip appears, as shown with the Close Toolbar icon in the illustration above. You can close/open the Toolbar at any time. Using **View menu** > **Toolbar** you can also customize the **Toolbar** to suit your needs. The **Toolbar** is also one of the dockable features enabling you to be able to re-position it if you choose. See "**Dockable Windows**" for more information.

To close the Toolbar:

From the Toolbar select  (**Close Toolbar**).

To open the Toolbar:

Select **View menu** > **Toolbar**.

Select **Display Toolbar**.

To add/remove groups in the Toolbar:

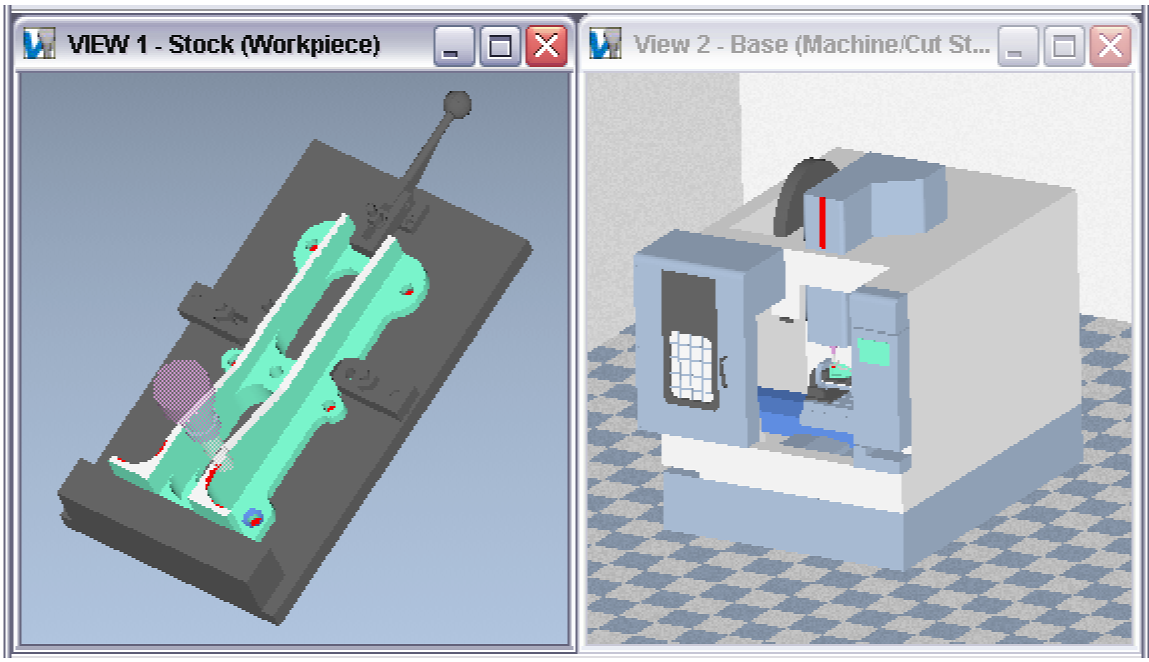
Select **View menu** > **Toolbar**.

Clear the check, by clicking on it, to remove groups you do not want displayed. Add a check, by clicking in the box, for groups that you want to display.

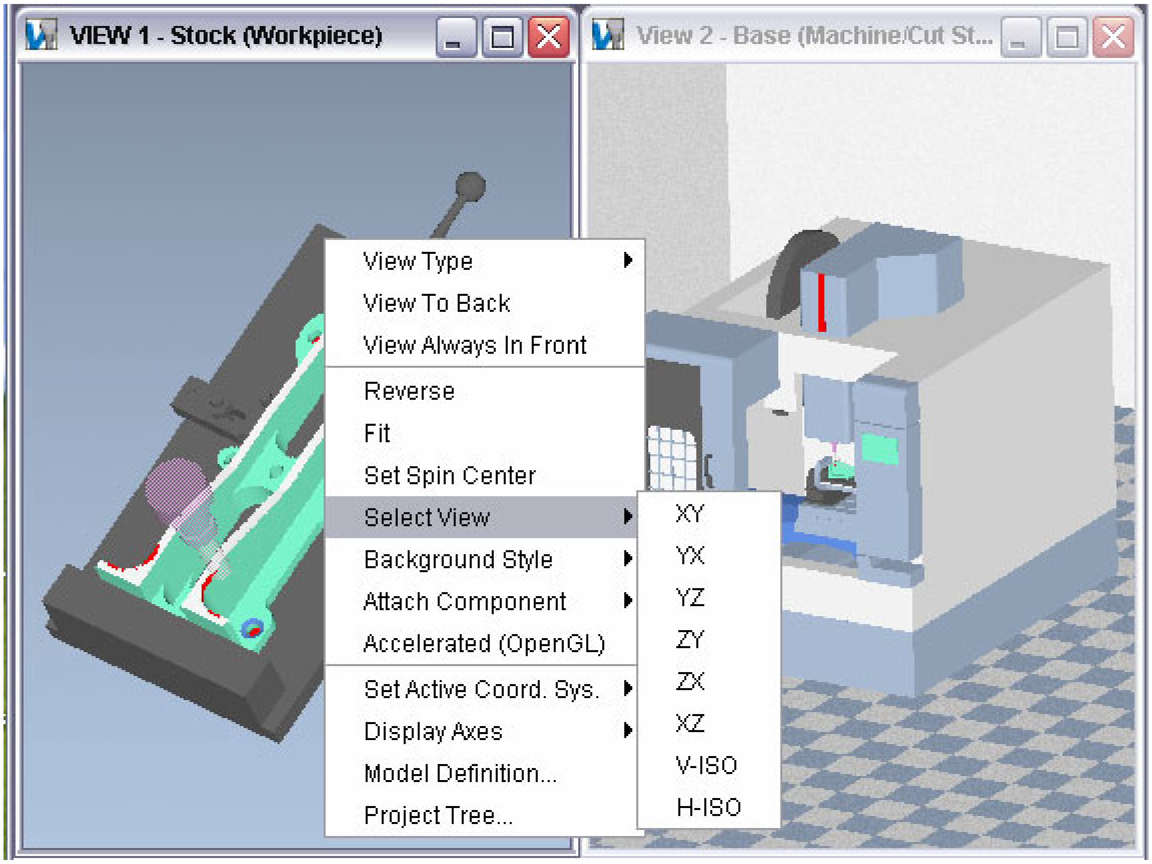
Tip: You can affect all **Toolbar** groups at once using the **Set All** / **Clear All** buttons.

Graphics area

The graphics area is where solid models of the workpiece, fixtures, etc. are displayed, and where the simulation takes place. By default, two views are seen: a Workpiece view and a Machine Cut/Stock view. Using **View menu > Layout** you can add more views displaying the workpiece or NC machine (if a machine is defined). All views are contained within the VERICUT main window.



Right-click in a view to display Shortcuts menu, as shown in the illustration below, enabling you to modify its attributes, such as: view type, standard modeling views, background, etc. Use dynamic and static view options on the **Tool Bar** to rotate, zoom, or fit the model.



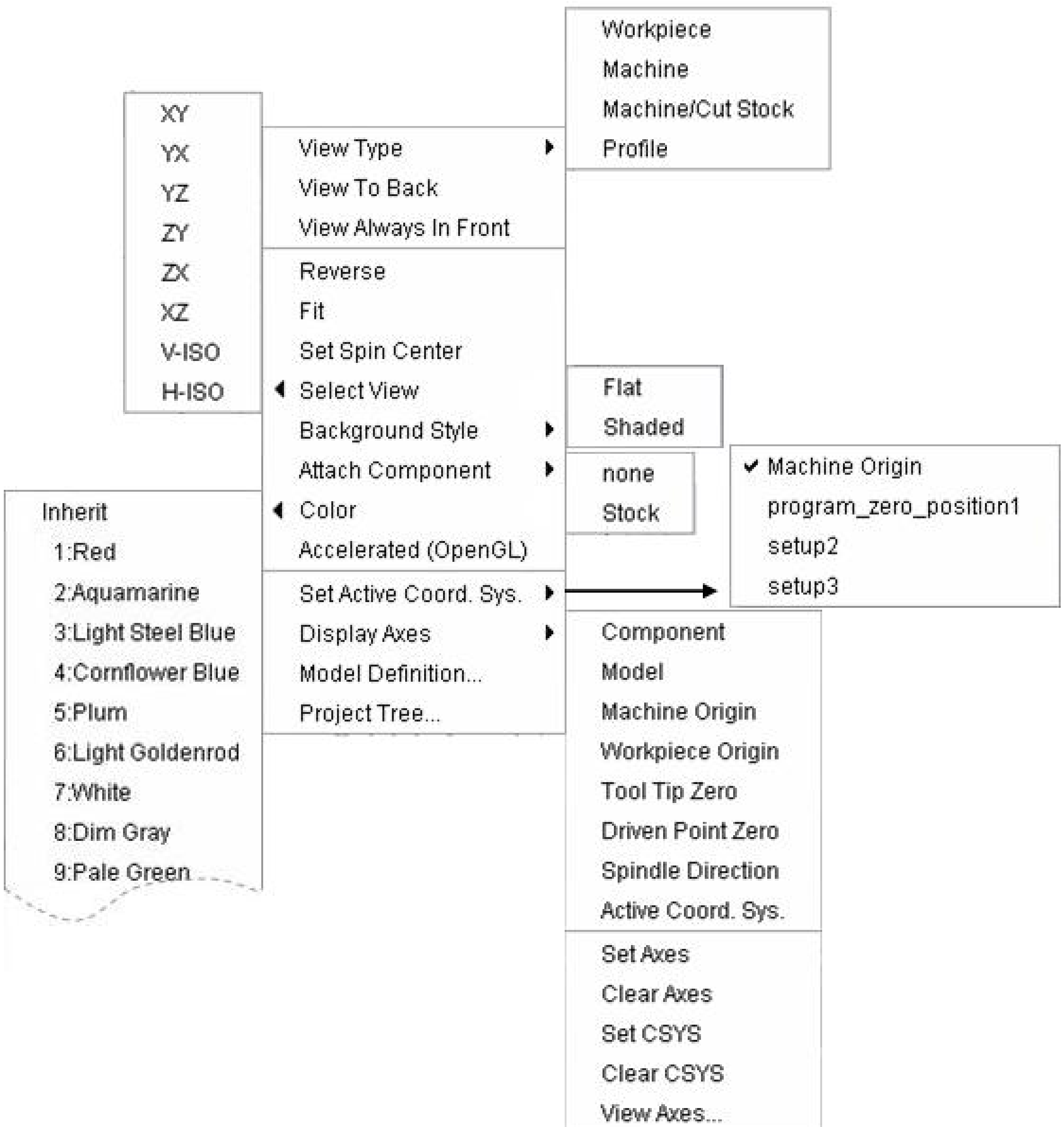
Many other VERICUT functions enable user interactions in this area, including: picking models or surfaces to be measured, determining tool path records responsible for specific cuts, and more.

See the **Right Mouse Button Shortcut Menus** section, also in *VERICUT Help* for a complete list of windows where Shortcut menus are available.

Graphics Area Right Mouse Button Shortcut Menus

Workpiece View

Right-click in a **Workpiece View** to display a menu with the following features:



View Type — Use to change the view type. (ref. **View Attributes window, General tab**, in the View Menu section of *VERICUT Help*)

View to Back — Moves the view to the back. (ref. **Layout (view)**, in the View Menu section of *VERICUT Help*)

View Always in Front — Makes the view remain "always in front". (ref. **Layout (view)**, in the View Menu section of *VERICUT Help*)

Reverse — Reverses viewing direction. (ref. **View Orient window**, in the View Menu section of *VERICUT Help*)

Fit — "Fits" the objects in the view. (ref. **View Orient window**, in the View Menu section of *VERICUT Help*)

Set Spin Center — Use to specify a point to be used as the center of rotation during dynamic rotations. (ref. **View Orient window**, in the View Menu section of *VERICUT Help*)
Select View — Orients the objects in the view to the selected view. The **Select View** feature list will contain all available standard and custom views.

Background Style — Use to specify the background style for the view. (ref. **View Attributes window, General tab**, in the View Menu section of *VERICUT Help*)

Attach Component — Attaches the view point and line of sight for a view to the selected component. (ref. **View Attributes window, General tab**, in the View Menu section of *VERICUT Help*)
Color — Use to change the color of the selected component or model. This feature is only available when the Project Tree, Component Tree, or the Modeling window is displayed. With the Project Tree or the Modeling window open, only the color of components and models in the Attach Component branch can be changed. When the Component Tree is opens all component and model colors can be changed.

Accelerated (Open GL) — Toggles Hardware Graphics Acceleration (OpenGL) "On" and "Off". A check indicates that OpenGL is toggled "On". (ref. **View Attributes window**, in the View Menu section of *VERICUT Help*)

Set Active Coord. Sys. — Use to designate the "active"coordinate system. (ref. **Set Active Coordinate System**, in the Project Menu section of *VERICUT Help*.

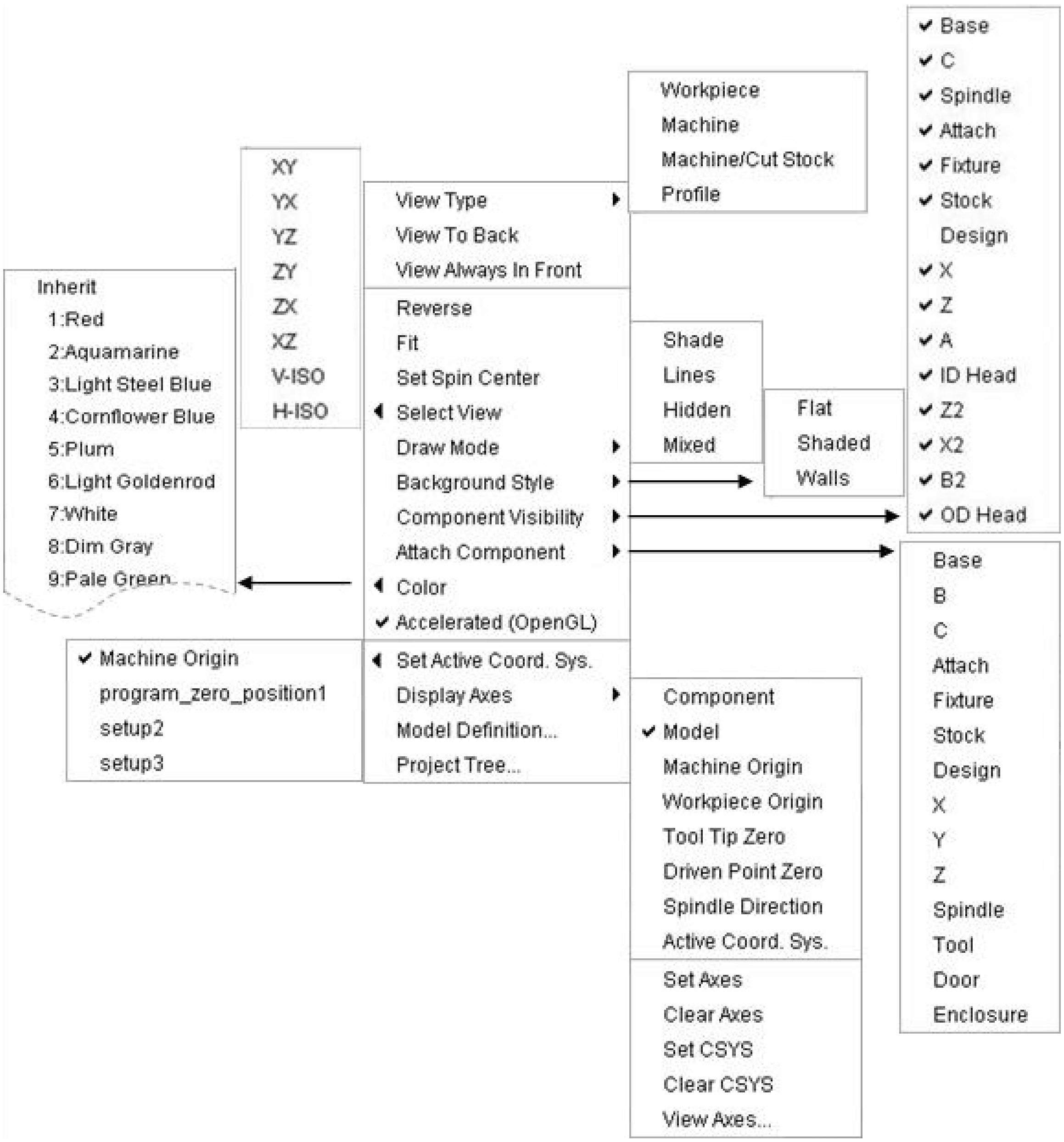
Display Axes — Use to control when various axes and coordinate systems are displayed. A check next to any of the **Display Axes** features, indicates that the feature is toggled "On". (ref. **View Axes window**, in the View Menu section of *VERICUT Help*)

Model Definition — Opens the Modeling window. (ref. **Modeling window**, in the Project Menu section of *VERICUT Help*)

Project Tree — Opens the Project Tree. (ref. **Project Tree**, in the Project Menu section of *VERICUT Help*)

Machine View

Right-click in a **Machine View** to display a menu with the following features:



View Type — Use to change the view type. (ref. **View Attributes window, General tab**, in the View Menu section of *VERICUT Help*)

View to Back — Moves the view to the back. (ref. **Layout (view)**, in the View Menu section of *VERICUT Help*)

View Always in Front — Makes the view remain "always in front". (ref. **Layout (view)**, in the View Menu section of *VERICUT Help*)

Reverse — Reverses viewing direction. (ref. **View Orient window**, in the View Menu section of *VERICUT Help*)

Fit — "Fits" the objects in the view. (ref. **View Orient window**, in the View Menu section of *VERICUT Help*)

Set Spin Center — Provides the same functionality as the **Set Spin Center** feature in the View Orient window. (ref. **View Orient window**, in the View Menu section of *VERICUT Help*)

Select View — Orients the objects in the view to the selected view. The **Select View** feature list will contain all available standard and custom views.

Draw Mode — Use to specify how machine components are displayed. (ref. **View Attributes window, General tab**, in the View Menu section of *VERICUT Help*)

Background Style — Use to specify the background style for the view. (ref. **View Attributes window, General tab**, in the View Menu section of *VERICUT Help*)

Component Visibility — Use to make the selected component visible, or not visible, in machine views (Machine or Machine/Cut Stock) in the VERICUT graphics area. A check indicates that the component is visible.

This feature is especially useful when using "encrypted" machine files and you do not have access to the Component Tree.

If the component list contains more than 32 items, VERICUT will automatically break up the list into sub-lists. Each sub-list will be identified by the first, and last, component in the list.

Attach Component — Attaches the view point and line of sight for a view to the selected component. (ref. **View Attributes window, General tab**, in the View Menu section of *VERICUT Help*)

Color — Use to change the color of the selected component or model. This feature is only available when the Project Tree, Component Tree, or the Modeling window is displayed. With the Project Tree or the Modeling window open, only the color of components and models in the Attach Component branch can be changed. When the Component Tree is opens all component and model colors can be changed.

Accelerated (Open GL) — Toggles Hardware Graphics Acceleration (OpenGL) "On" and "Off". A check indicates that OpenGL is toggled "On". (ref. **View Attributes window**, in the View Menu section of *VERICUT Help*)

Set Active Coord. Sys. — Use to designate the "active"coordinate system. (ref. **Set Active Coordinate System**, in the Project Menu section of *VERICUT Help*.

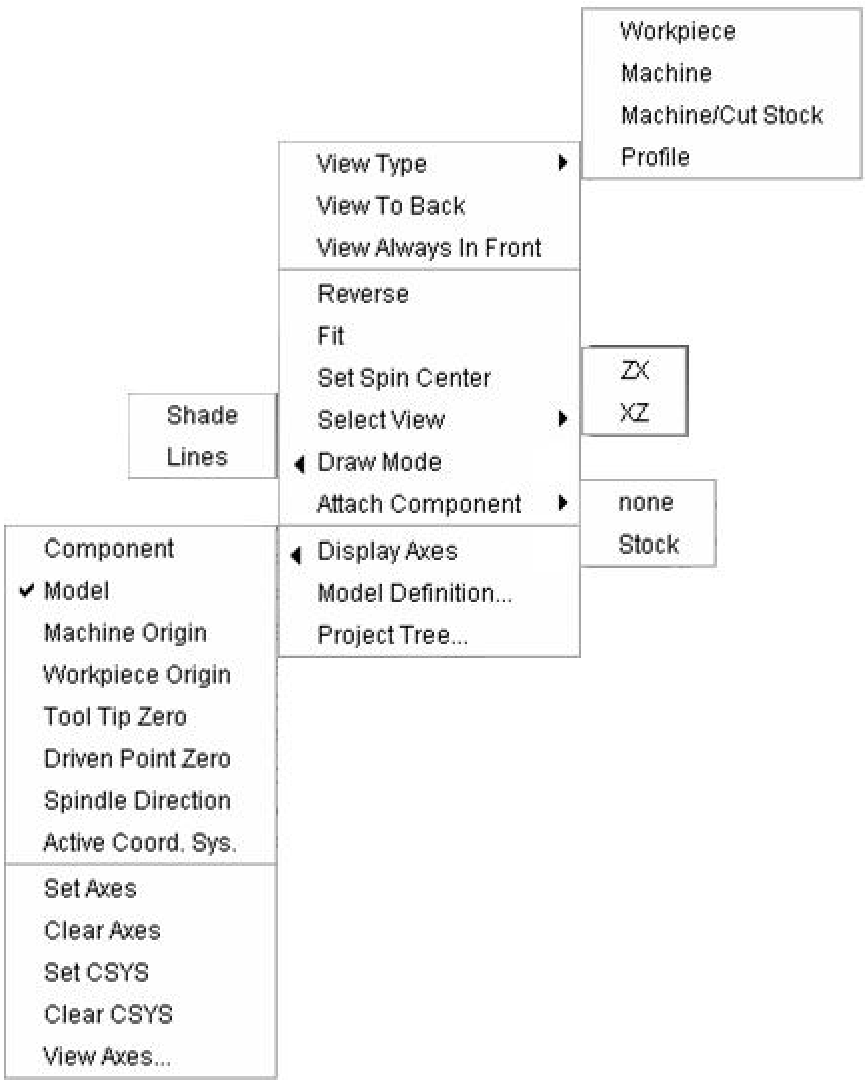
Display Axes — Use to control when various axes and coordinate systems are displayed. A check next to any of the **Display Axes** features, indicates that the feature is toggled "On". (ref. **View Axes window**, in the View Menu section of *VERICUT Help*)

Model Definition — Opens the Modeling window. (ref. **Modeling window**, in the Project Menu section of *VERICUT Help*)

Project Tree — Opens the Project Tree. (ref. **Project Tree**, in the Project Menu section of *VERICUT Help*)

Profile View

Right-click in a **Profile View** to display a menu with the following features:



View Type — Use to change the view type. (ref. **View Attributes window, General tab**, in the View Menu section of *VERICUT Help*)

View to Back — Moves the view to the back. (ref. **Layout (view)**, in the View Menu section of *VERICUT Help*)

View Always in Front — Makes the view remain "always in front". (ref. **Layout (view)**, in the View Menu section of *VERICUT Help*)

Reverse — Reverses viewing direction. (ref. **View Orient window**, in the View Menu section of *VERICUT Help*)

Fit — "Fits" the objects in the view. (ref. **View Orient window**, in the View Menu section of *VERICUT Help*)**Set Spin Center —** Provides the same functionality as the Set Spin Center feature in the View Orient window. (ref. **View Orient window**, in the View Menu section of *VERICUT Help*)

Select View — Orients the objects in the view to the selected view. The **Select View** feature list will contain all available standard and custom views.

Draw Mode — Use to specify how machine components are displayed. (ref. **View Attributes window, General tab**, in the View Menu section of *VERICUT Help*)

Attach Component — Attaches the view point and line of sight for a view to the selected component. (ref. **View Attributes window, General tab**, in the View Menu section of *VERICUT Help*)

Display Axes — Use to control when various axes and coordinate systems are displayed. A check next to any of the **Display Axes** features, indicates that the feature is toggled "On". (ref. **View Axes window**, in the View Menu section of *VERICUT Help*)

Model Definition — Opens the Modeling window. (ref. **Modeling window**, in the Project Menu section of *VERICUT Help*)

Project Tree — Opens the Project Tree. (ref. **Project Tree**, in the Project Menu section of *VERICUT Help*)

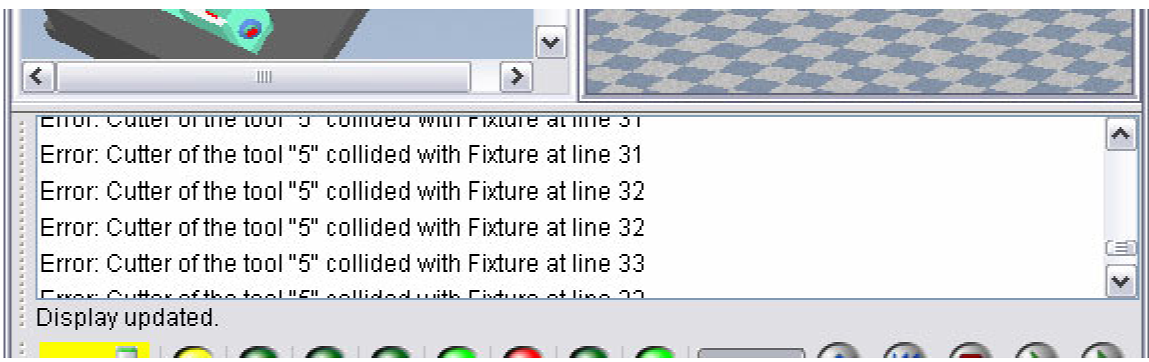
Message Area (Logger)

The message area, also known as the "message logger" or "Logger", is located below the graphics area, and by default displays the most recent message VERICUT has provided. VERICUT provides error, warning, and informational messages to assist you during a simulation session. You can easily determine the type of message by the following:

Information messages have no preceding label.
Warning messages are preceded by "**Warning:**".
Error messages are preceded by "**Error:**".

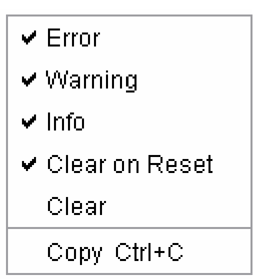
You can control the number of messages displayed in the Logger at any one time by clicking on the divider between the Logger and the Graphics Area and either dragging up to increase the number message lines displayed , or down to reduce the number message lines displayed.

You can always use the scroll bar to view messages that are not currently visible in the Logger window.



The **Message Area/Logger** is also one of the "dockable" features enabling you to be able to re-position it if you choose. See "**Dockable Windows**" for more information.

Shortcut: Right-click in the **Message Area (Logger)** to display a menu with the following features:



NOTES:

1. These features provide the same functionality described under Logger View Options for controlling what is displayed in the Logger. (ref. **Logger View Options**, in the View Menu section of *VERICUT Help*)
2. A check next to any of the features indicates that the feature is toggled "On".

Tip: Clicking on an error in the Logger will highlight the NC program record in the NC Program window (**Info menu > NC Program**) that caused the error.

NOTES:

1. All messages can be seen along with other valuable information in the **Log file**.
2. For information and tips about using VERICUT and troubleshooting problems, visit the VERICUT Users' Forum or contact CGTech technical support via our website, just click on the support link.

Animation Controls

The **Animation Controls** features, located in the VERICUT main window, enable you to control and visually monitor certain events related to the simulation. The **Animation Controls** consist of the following four features.

Animation Speed Slider — Enables you to control the speed of the simulation.

Status Lights — Enables you to visually monitor certain events associated with the simulation (status of travel limit checking, collisions, probing, subroutines, cutter compensation, cycles, motion, OptiPath, and when VERICUT is processing).

Progress Bar — Enables you to monitor the progress of the current task.

Simulation (VCR) Controls — Enables you to start, single step, stop, rewind, and reset the VERICUT simulation.

Each of these **Animation Controls** features is described in detail in the following sections.

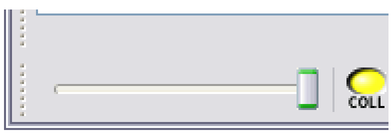
Animation Controls is also one of the "dockable" features enabling you to be able to re-position it if you choose. See "**Dockable Windows**" for more information.

Animation Speed Slider

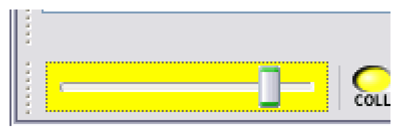
The **Animation Speed Slider**, located on the left end of the **Animation Controls** features, controls the speed at which VERICUT animates material removal in workpiece views. Moving the slide bar left slows animation by adding intermediate tool display positions between motion start and end points. Moving the slide bar right reduces the number of intermediate tool display positions to speed animation display.

The **Animation Speed Slider** will be displayed in one of two ways:

Animation Speed Slider in full right position



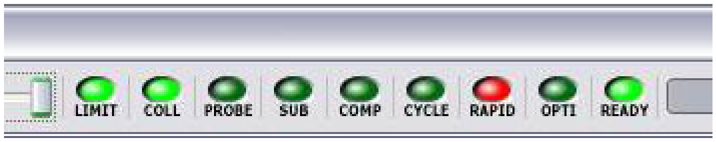
Animation Speed Slider moved off of full right position



Use **Min. Motion Dist.** and **Max. Motion Dist.**, in the **Project > Processing Options > Motion window**, to define the range for the **Animation Speed Slider**.

Tip: To group cuts together for even faster animation display, use the **Skip Cut** feature located on the **Motion window**.

Status Lights



The "**Status Lights**", located between the **Animation Speed Slider** and the **Progress Bar** in the **Animation Controls** features, provide constant visible feedback over when VERICUT is processing certain events, busy performing a task, or optimizing an NC program file. The function and conditions represented by the color of each status light is described below.

LIMIT — Indicates the status of travel limit checking.

Red – indicates that a travel limit has been exceeded on the current block.

Yellow - indicates that a travel limit has been exceeded during the current session.

Green – indicates when Overtravel Detection On is toggled "**On**".

Dark Green – indicates that Overtravel Detection On is toggled "**Off**".

COLL — Indicates the status of collision checking.

Red – indicates that a collision occurred on the current block.

Yellow - indicates that a collision has occurred during the current session.

Green – indicates that Collision Detection is toggled "**On**".

Dark Green – indicates that Collision Detection is toggled "**Off**".

PROBE — Indicates Probe status.

Green — indicates when Probe is "**Armed**".

Dark Green — indicates when Probe is "**Not Armed**".

SUB — Indicates the status of Subroutine processing.

Green — indicates when VERICUT is processing a project subroutine.

Dark Green — indicates that no project subroutine is being processed.

COMP — Indicates the status of Cutter Compensation.

Green — indicates that Cutter Compensation is "**Active**".

Dark Green — indicates that Cutter Compensation is "**Not Active**".

CYCLE — Indicates CYCLE status.

Green — indicates when CYCLE is "**Active**".

Dark Green — indicates when CYCLE is "**Not Active**".

FEED / RAPID — Indicates the current Motion status.

Red — indicates a "**Rapid**" motion.

Green — indicates a "**Linear**" motion.

Dark Green — indicates some "**Other**" type of motion.

OPTI — Indicates OptiPath status.

Green — indicates OptiPath is "**On**".

Yellow — indicates OptiPath is in "**Learn Mode**".

Dark Green — indicates that OptiPath is "**Off**".

READY — Indicates VERICUT Status.

Red — indicates that VERICUT is "**Busy**" processing.

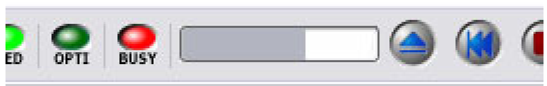
Green — indicates that VERICUT is "**Ready**" for user commands.

TIP: Place the cursor over any of the **Status Lights** to see a summary of its function and conditions.

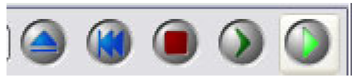
Progress Bar

The **Progress bar**, located between the **Status Lights** and the **Simulation (VCR) Controls** in **Animation Controls** features, provide constant visible feedback over the progress of a CPU-intensive tasks, such as: comparing models using AUTO-DIFF, exporting model data, and checking the consistency of the stock model(s). The progress bar moves towards the right as the task nears completion. To interrupt the task, press  (**Stop**).

Sample progress bar part way through a task:

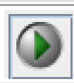
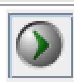
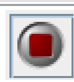
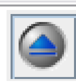


Simulation (VCR) controls



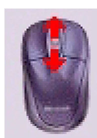
The simulation controls, also known as VCR buttons, located on the right end of the **Animation Controls** features, control interactive tool path simulation. Once VERICUT is configured, you will use these controls to start and stop the simulation, as well as begin with a new, uncut workpiece. To see what is associated with a Simulation control icon, simply position the cursor over the icon and a tip appears.

Simulation controls:

Icon	Name	Function
	**Play to End**	Start or re-start tool path processing
	**Single Step**	Process one tool path record ("single block")
 (or press Escape key <**Esc**>)	**Stop**	Stop processing (after the current record is processed)
	**Rewind NC Program**	Rewind the NC program file to the beginning (leave the model as is)
	**Reset Model**	Display a new VERICUT model and rewind the tool path file

Dynamic Zoom, Pan, and Rotate

The **Dynamic Zoom, Pan and Rotate** features enable you to manipulate the displayed image of a view using a single mouse button.

Rotate		Click with the left mouse button in a view and drag the mouse to rotate the display.
Pan		Click with the right mouse button in a view and drag the mouse to rotate the display.
Zoom		Rotate the thumb wheel while the cursor is in a view to zoom in or out on the display. Move the wheel toward you to make the displayed image larger. Move the wheel away from you to make the displayed image smaller.
Zoom to Box		Press the thumb wheel down in a view and drag the mouse to define the box to zoom to. Release the thumb wheel to start the zoom. You can also click the thumbwheel to define the first corner of the box, and then move the mouse to the position defining the other corner of the box and click again.

NOTES:

1. Dynamic Controls (**View menu > Dynamic Controls)** must be set to VERICUT.
2. The Shift key, Ctrl key, and mouse button combinations used in pre-V6.1 for view manipulation is still available.
3. The Shift key, Ctrl key, and Arrow key combinations are no longer available.
4. In NC Program Review, the dynamic rotate with the left button is briefly suspended when picking a rectangle for **Settings > Display to Box**.

5. In AUTO-DIFF, when using **Compare By Region**, the dynamic rotate with the left button is suspended as long as the **Drag Region** button is active.

Look & Feel

Location: **View menu > Look & Feel**

The **Look & Feel** features change the appearance of VERICUT's user interface. All windows are affected. The look and feel of VERICUT can be changed at any time.

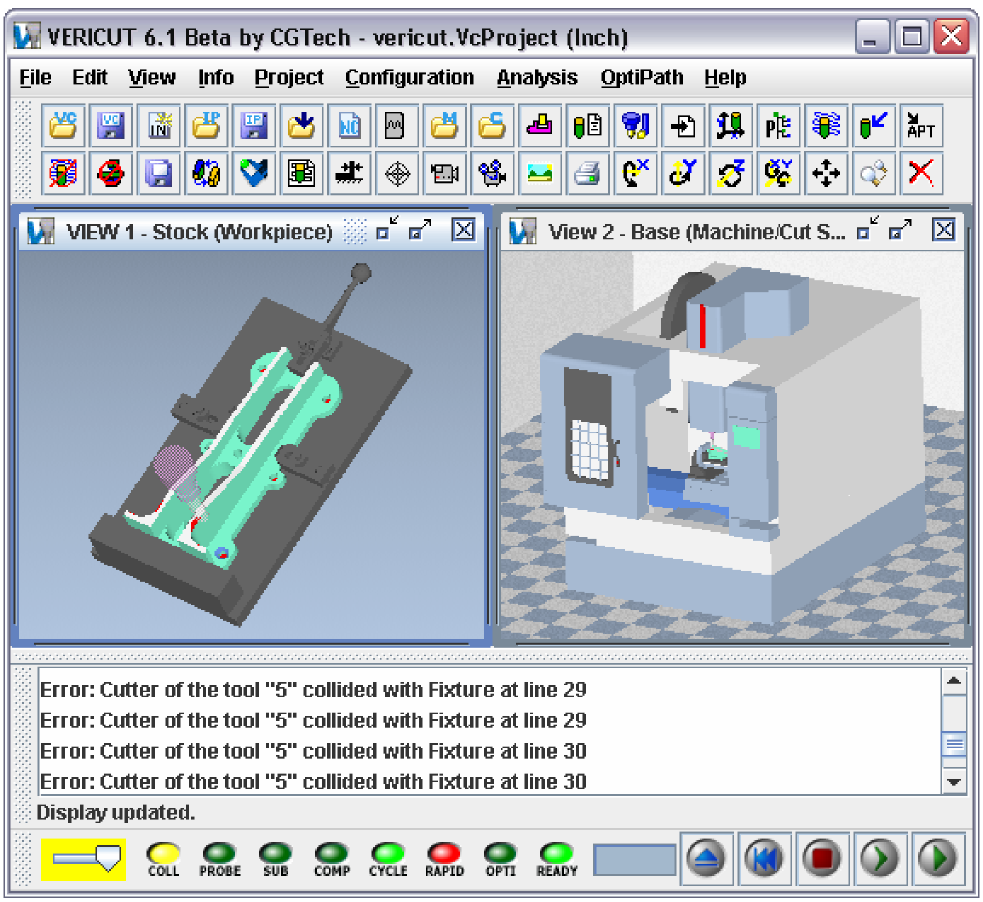
To change the user interface look and feel:

1. In the **View** menu, select **Look and Feel**.
2. Choose the option that provides the desired appearance.

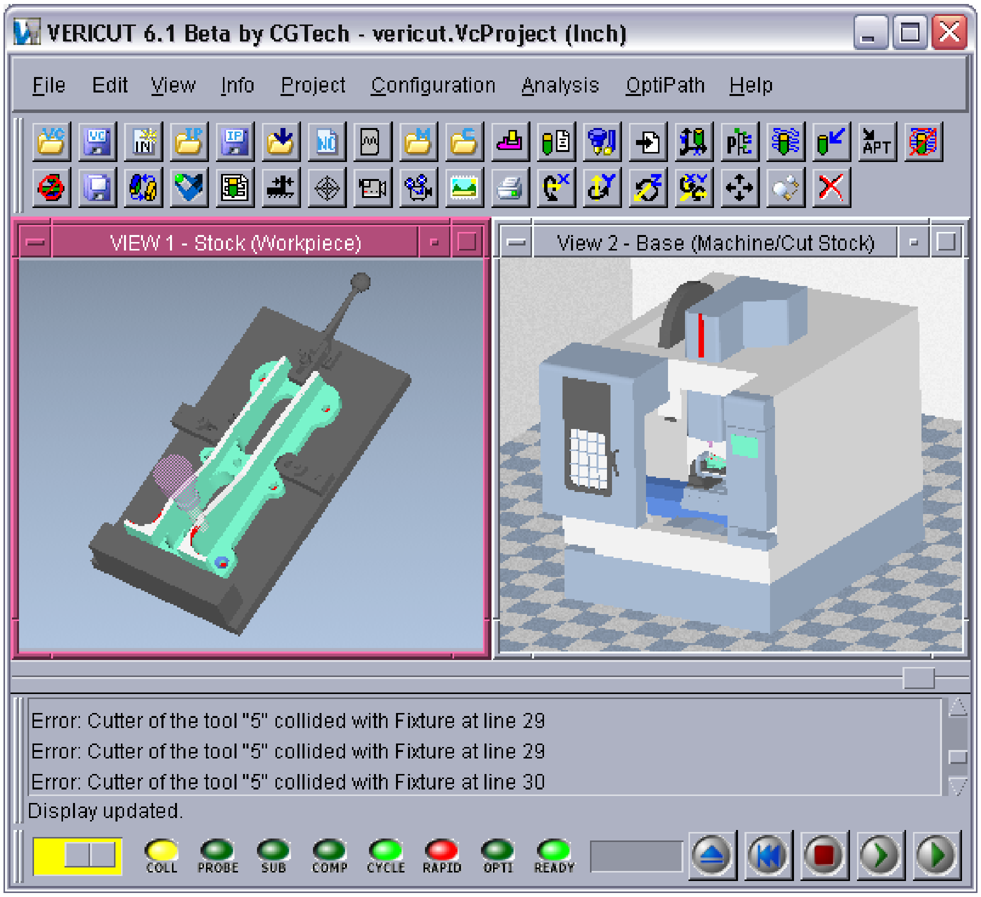
NOTE: Not all options are available on all systems.

Options:

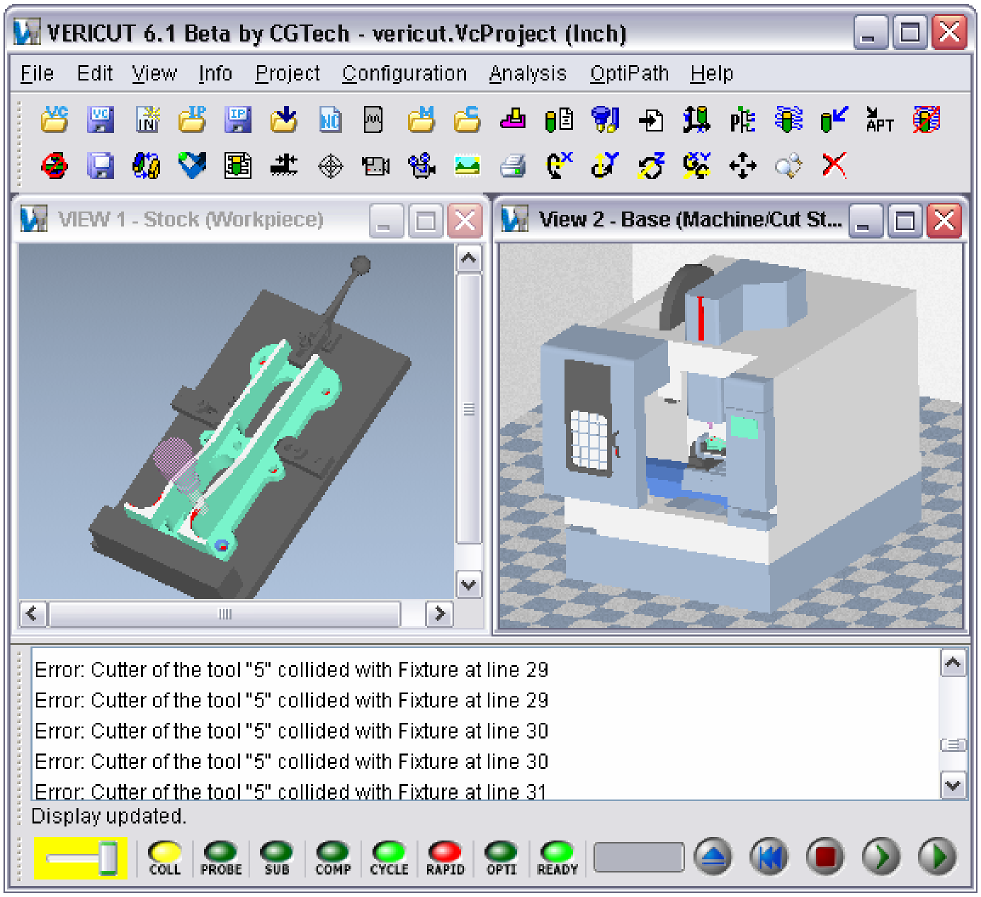
Metal —



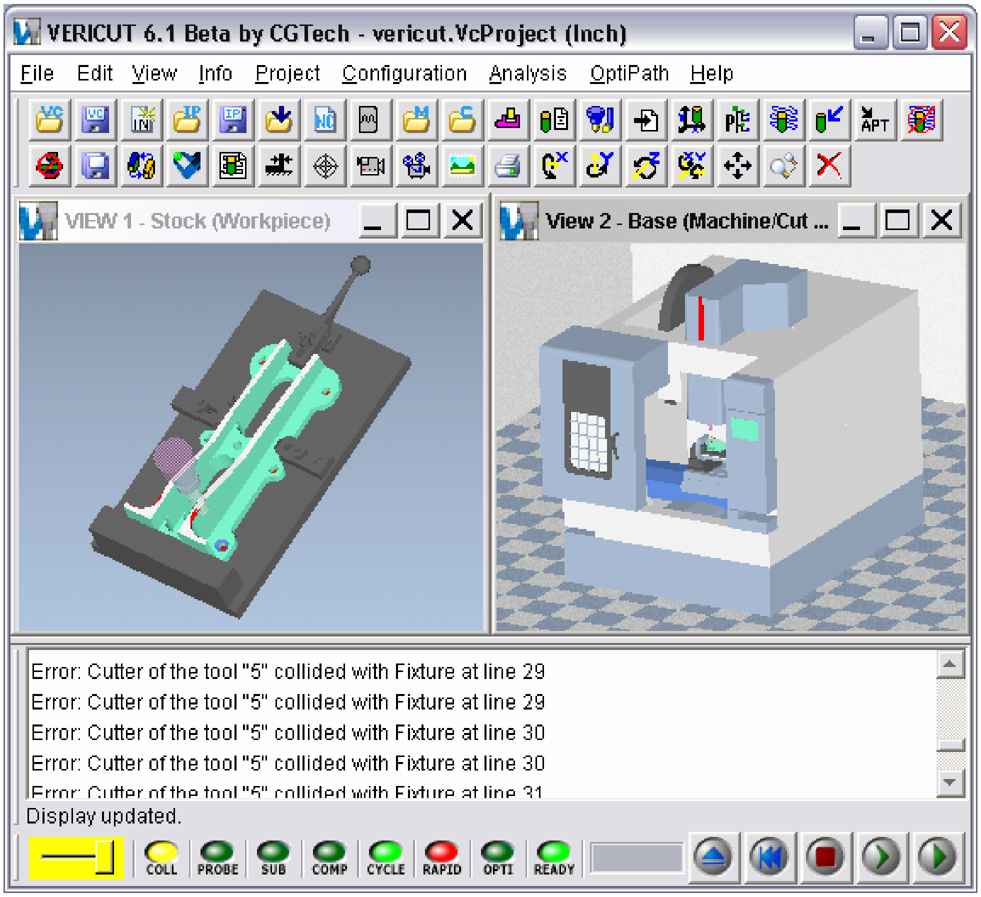
Motif —


VERICUT 6.1 Beta by CGTech - vericut.VcProject (Inch)
File Edit View Info Project Configuration Analysis OptiPath Help
VIEW 1 - Stock (Workpiece)
View 2 - Base (Machine/Cut Stock)
Error: Cutter of the tool "5" collided with Fixture at line 29
Error: Cutter of the tool "5" collided with Fixture at line 29
Error: Cutter of the tool "5" collided with Fixture at line 30
Display updated.
COLL
PROBE
SUB
COMP
CYCLE
RAPID
OPTI
READY

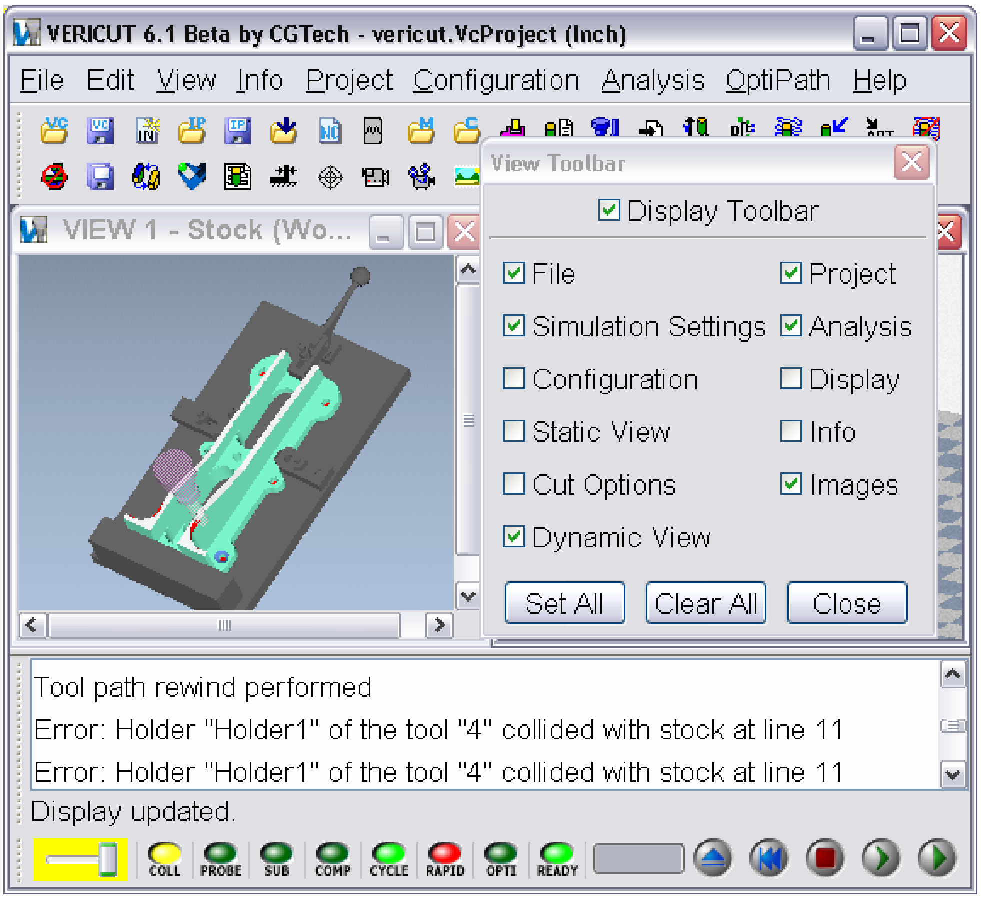
Windows —


VERICUT 6.1 Beta by CGTech - vericut.VcProject (Inch)
File Edit View Info Project Configuration Analysis OptiPath Help
VIEW 1 - Stock (Workpiece)
View 2 - Base (Machine/Cut St...
Error: Cutter of the tool "5" collided with Fixture at line 29
Error: Cutter of the tool "5" collided with Fixture at line 29
Error: Cutter of the tool "5" collided with Fixture at line 30
Error: Cutter of the tool "5" collided with Fixture at line 30
Error: Cutter of the tool "5" collided with Fixture at line 31
Display updated.
COLL PROBE SUB COMP CYCLE RAPID OPTI READY

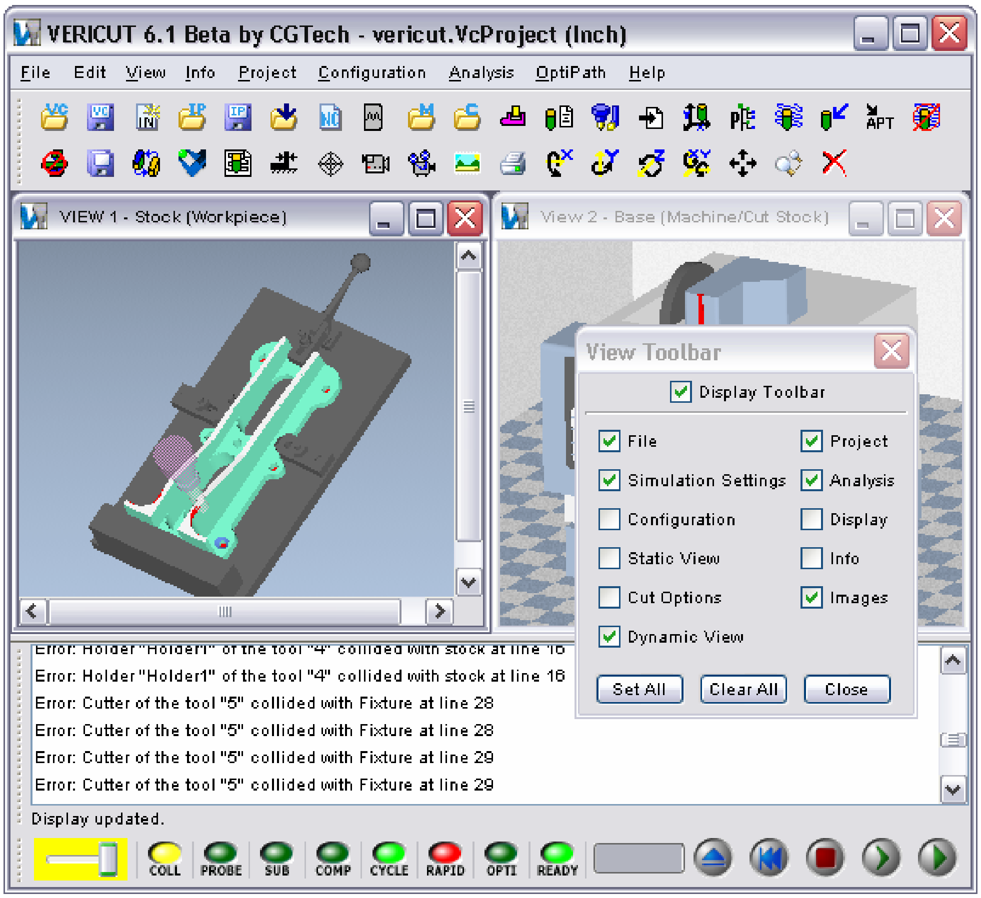
Windows Classic —



Text Larger — Use to make the text in VERICUT menus, message logger, windows, etc. larger. Each time you click on "Text Larger", the text gets incrementally larger to a maximum font size of "14 point".

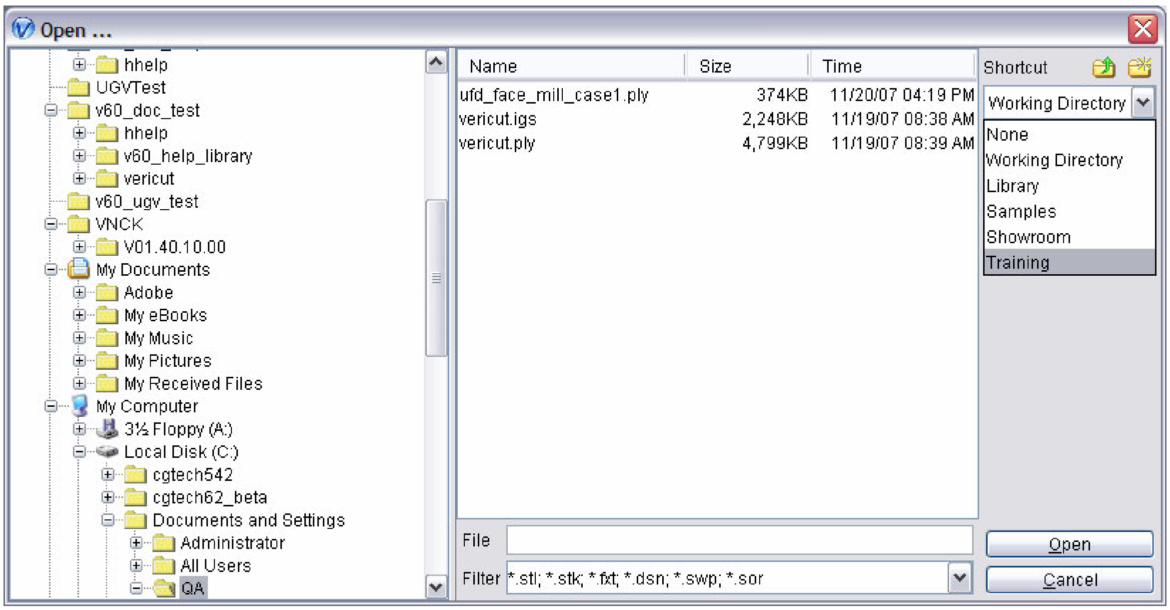


Text Smaller — Use to make the text in VERICUT menus, message logger, windows, etc. smaller. Each time you click on "Text Smaller", the text gets incrementally smaller to a minimum font size of "6 point ".



Introduction to VERICUT File Selection Windows

File selection windows are used throughout VERICUT to open and save various VERICUT file types. They are also accessed by using the **Browse** buttons found on many VERICUT windows to assist you in specifying required /path/filenames. Most features on this window are standard file selection window features that enable you to navigate through directories, filter files, and type, or select, /path/filenames. Each window may vary slightly depending on its intended purpose. This section describes the features common to all VERICUT file selection windows. Features specific to a particular window are described in the documentation for the window.



Directory Tree — Displays your computer's directories (folders) in a tree type structure.

File List — Displays the files contained in the directory highlighted in the Directory Tree, filtered according to the setting of the **Filter** feature described below.

File — Use this text field to manually enter /path/filename information. VERICUT will fill in this field as you use the **Directory Tree** and **File List** to specify the /path/filename. You can also edit the /path/filename entered by VERICUT.

Filter — Use this feature to specify the types of files to be displayed in the **File List**.

Shortcut — The **Shortcut** option list on file selection windows provides quick access to files in your Working Directory, or CGTech Library, CGTech Sample, Showroom, and Training files. Choosing a shortcut automatically filters for files in that directory. You can also type new shortcuts into the list, but they are only remembered during the current VERICUT session. See **Adding Custom Environment Variables for Shortcuts** in the

Environment Variables topic, in *VERICUT Help* for information on adding additional permanent shortcuts.

 **(Up One Level) —** Enables you to quickly move up one level in the directory structure.

 **(New Folder) —** Displays the **New** folder window enabling you to create a new directory/folder.

Enter the name for the new folder in the **New sub-folder name** text field, and then select **OK** to create the new folder inside the folder highlighted in the Directory Tree.

Use **Cancel** to close the **New** folder window without creating the folder.

Open / Save — Use to open/save the specified file in the specified directory.

Cancel — Use **Cancel** to close the file selection window without opening/saving a file.

Right Mouse Button Shortcut Menus

Many of the VERICUT windows have shortcut menus available, by clicking with the right mouse button, that contain features specific to that window. The following provides a summary of the VERICUT windows that have right mouse button shortcut menus. Refer to the VERICUT Help section for each window for specific information about the features contained in these shortcut menus.

VERICUT Main Window

- Workpiece View
- Machine View
- Profile View
- Message Area (Logger)

Text File (edit) window

NC Program (edit) window

View Section window

Select/Store View window

NC Program (Info) window

G-Code Report window

Control Report window

Profile Sketcher window

- Revolve Profile
- Sweep Profile

NC Program window

Tool Manager Window

- Tool Table
- Tool Display Area

G-Code Settings Window

- Subroutines tab
- Collision Detect tab

Variables window

Cycles window

Report Template window
- Page Layout tab
- Styles tab
- User Defined Tags tab

Setup Plan window

MDI window

Project Tree Window
- Project Branch
- Setup Branch
- CNC Machine Branch
- Control Branch
- Machine Branch
- Attach Component
- Component Branch
- Model
- Coordinate Systems Branch
- Coordinate System
- Tooling
- NC Programs
- NC Program file
- NC Subroutines
- NC Subroutine file
- Saved IP Files
- IP file

Machine Settings window
- Collision Detect tab

Component Tree Window
- component
- model

Word Format window
- Word Format tab

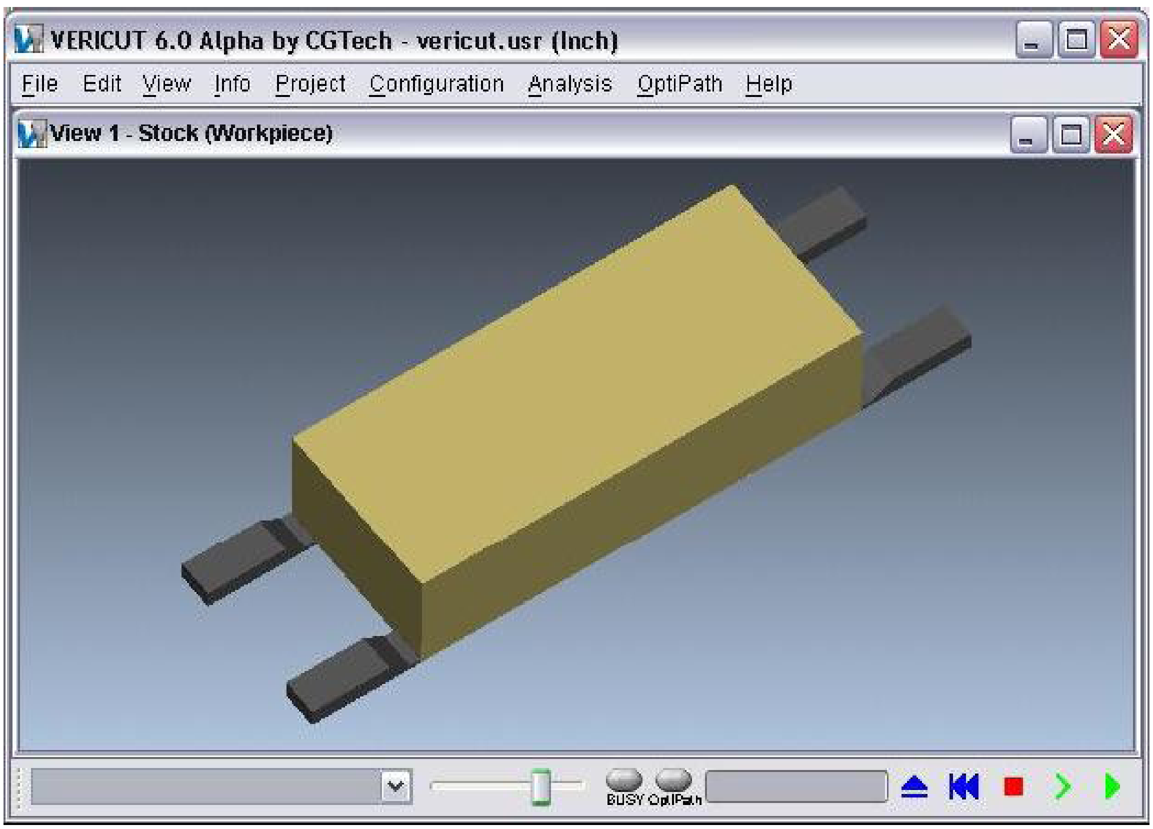
Word/Address window

Control Settings window
- Sync tab

Personalizing the VERICUT Main Window

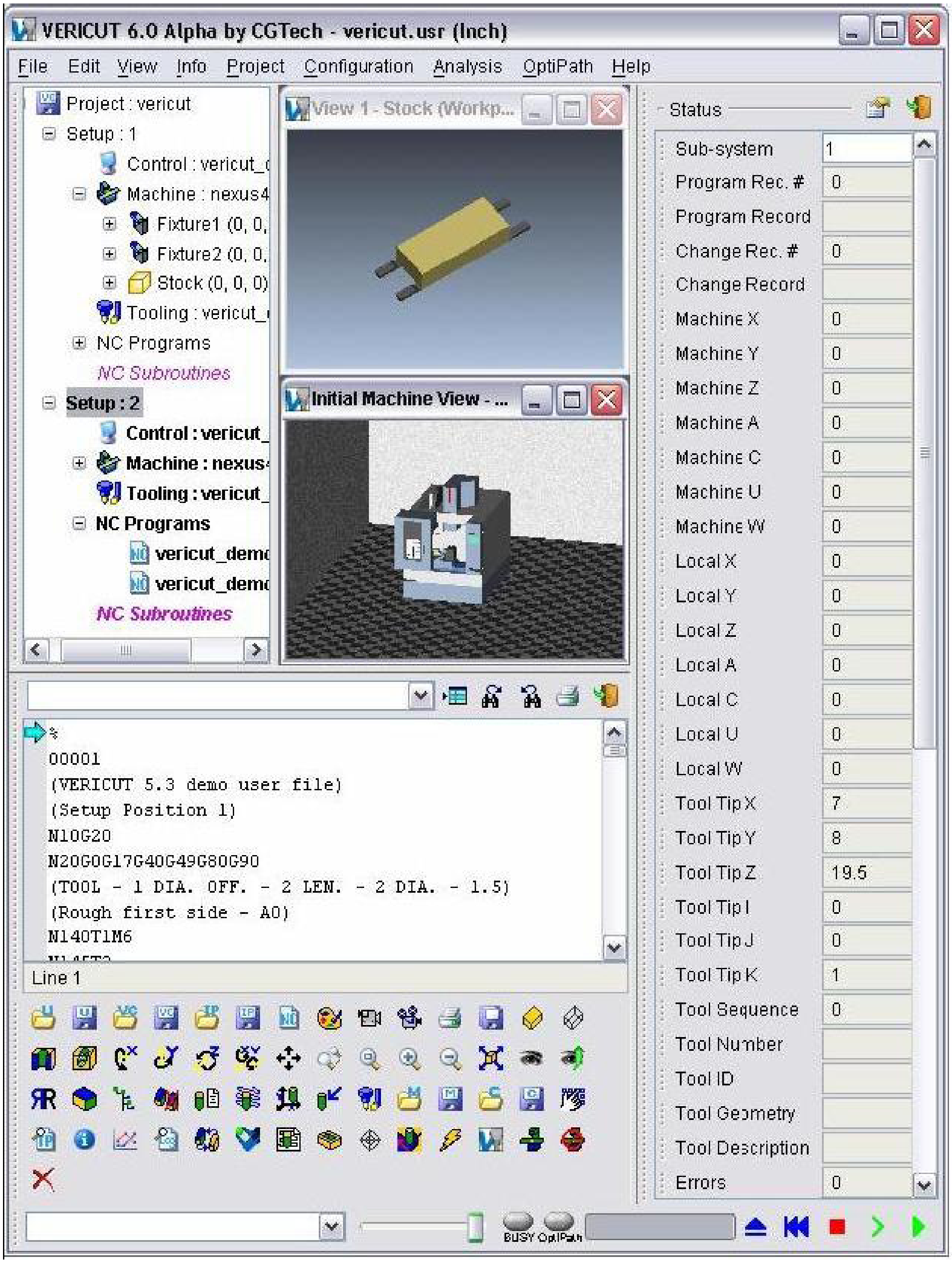
VERICUT enables you to personalize your workspace to match the process by which you use VERICUT's many features, as well as incorporating your own personal preferences. Through the use of "dockable" dialog windows, you can set up your VERICUT workspace as simply or as complex as you choose. Each of these windows can be docked inside the VERICUT main window, or un-docked to be displayed as a stand alone window like previous versions of VERICUT. By docking or undocking windows, you can tailor the VERICUT workspace to meet your specific needs. The "Simple VERICUT Workspace" illustrates the most basic workspace.

Simple VERICUT Workspace:



The "Complex VERICUT Workspace" is shown with nearly all of the available "dockable" windows docked to the VERICUT main window. While it appears cluttered, and is not likely an efficient workspace, the intent is to show the flexibility that you have for setting up your workspace. Your optimum workspace layout will likely fall somewhere between these two extremes. The main point is that you decide the most effective workspace layout based on how you use VERICUT and how you prefer to interact with its many features.

Complex VERICUT Workspace:



Dockable Windows

The following summarizes the dialogue windows that can be docked inside VERICUT's main desktop:

Dockable Window	Location	Docking Positions
Toolbar	VERICUT main window	Left, Right , Top, Bottom, Standalone
Progress bar	VERICUT main window	Top, Bottom, Standalone
NC Program window	Info menu > NC Program	Left, Right , Top, Bottom, Standalone
NC Program Review window	Analysis menu > NC Program Review	Left, Right , Top, Bottom, Standalone
Status window	Info menu > Status	Left, Right, Standalone
Project Tree window	Project menu > Project Tree	Left, Right, Standalone
X-Caliper window	Analysis menu > X-Caliper	Left, Right, Standalone

NOTE: When any of the above "**windows**" is docked, make sure that you click in the window so that it becomes the "active" window before using F1 to get help specific to the window. Otherwise F1 will go to the CGTech Help Library.

Docking and Un-docking windows

The procedure for docking/undocking any of the dockable windows is the same. All dockable windows have a dotted line on the left side of the window as indicated by the arrow in the Toolbar in the picture below. This dotted line is used for both docking and undocking these windows. To undock a currently docked window, left click on the dotted line on the left side of the window, then while keeping the button depressed, drag the rectangle representing the window (see Figure 1) to the desired location outside the VERICUT main window. When you release the mouse button, the window moves and is displayed as a stand alone window.

Figure 1

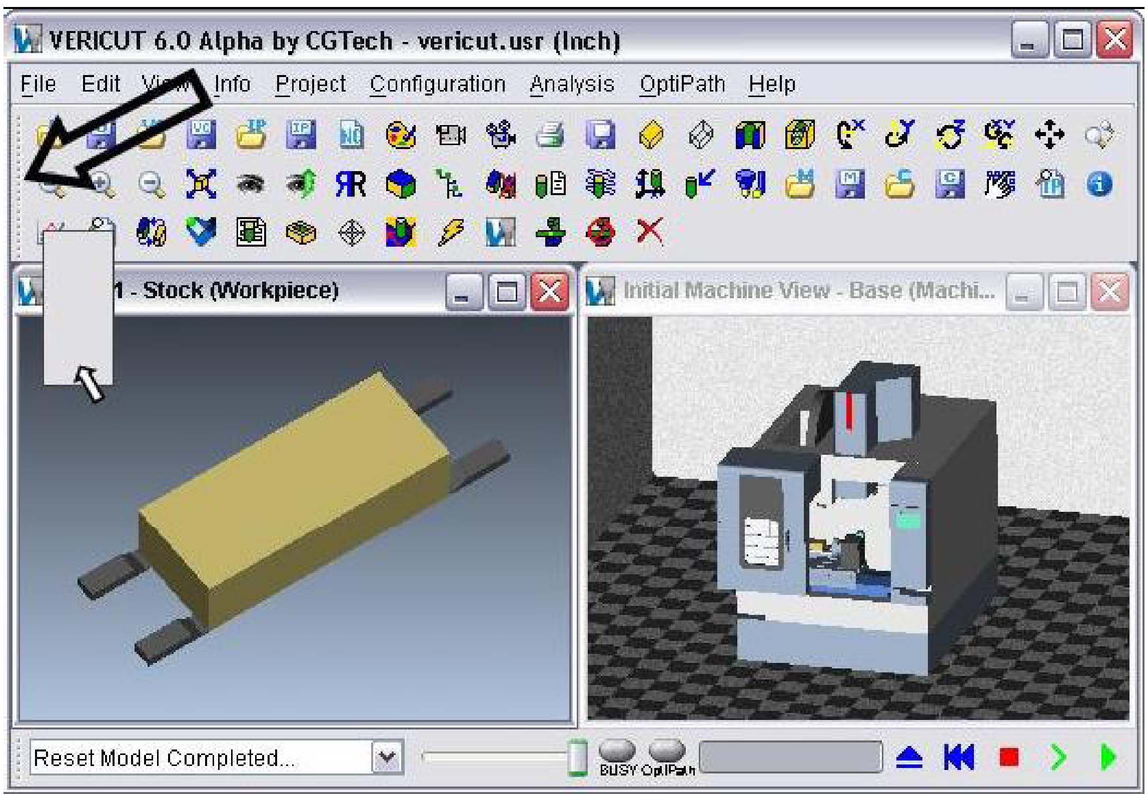
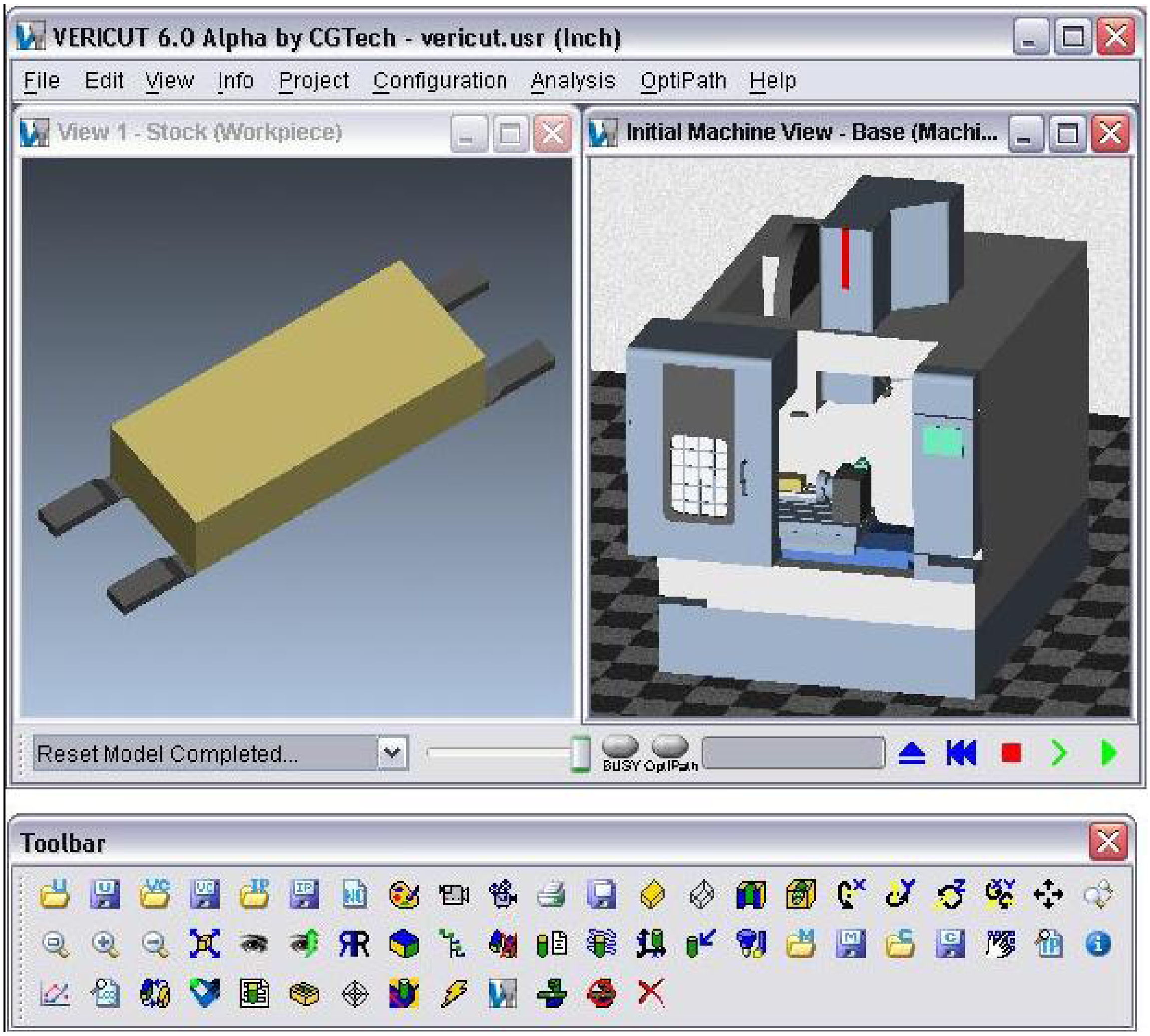


Figure 2 shows the result of dragging the Toolbar window to a position below the VERICUT main window.

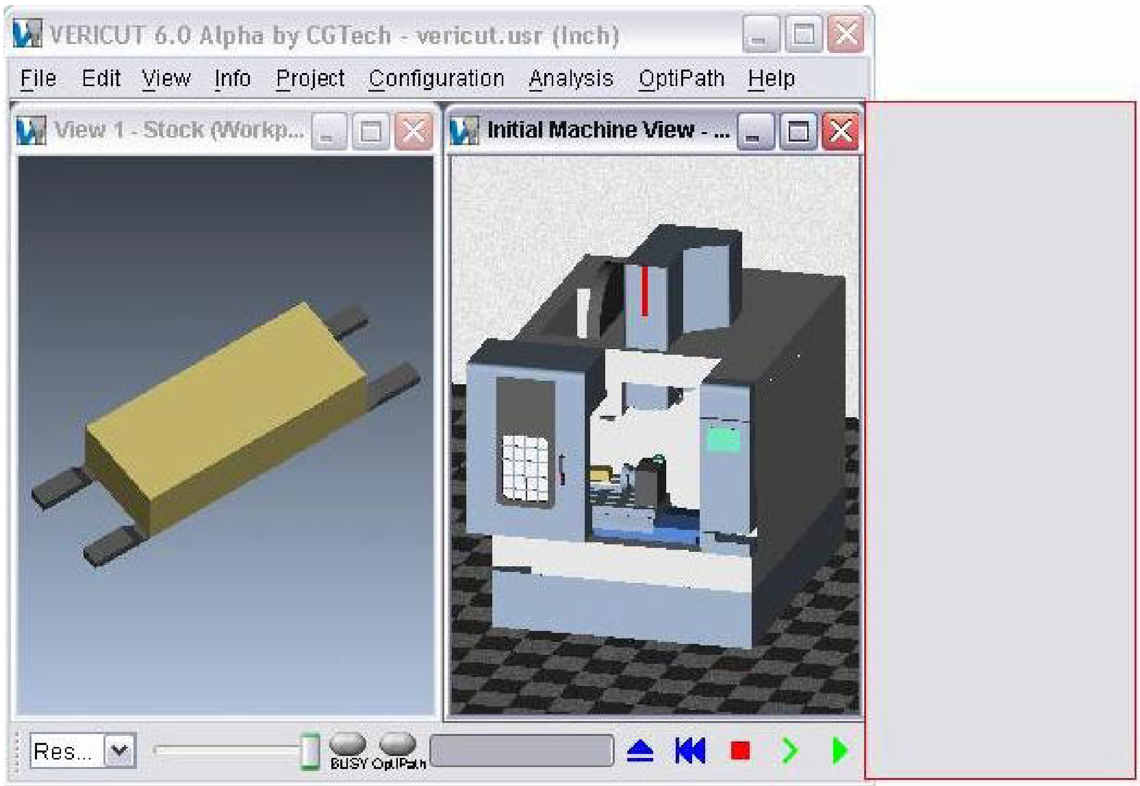
Figure 2



The procedure for docking a stand alone window or moving a currently docked window to a different docking location is the same. Starting with the stand alone Toolbar window shown in Figure 2, left click on the dotted line in the Toolbar window, then drag the rectangle representing the window to one of the docking positions available for the particular window.

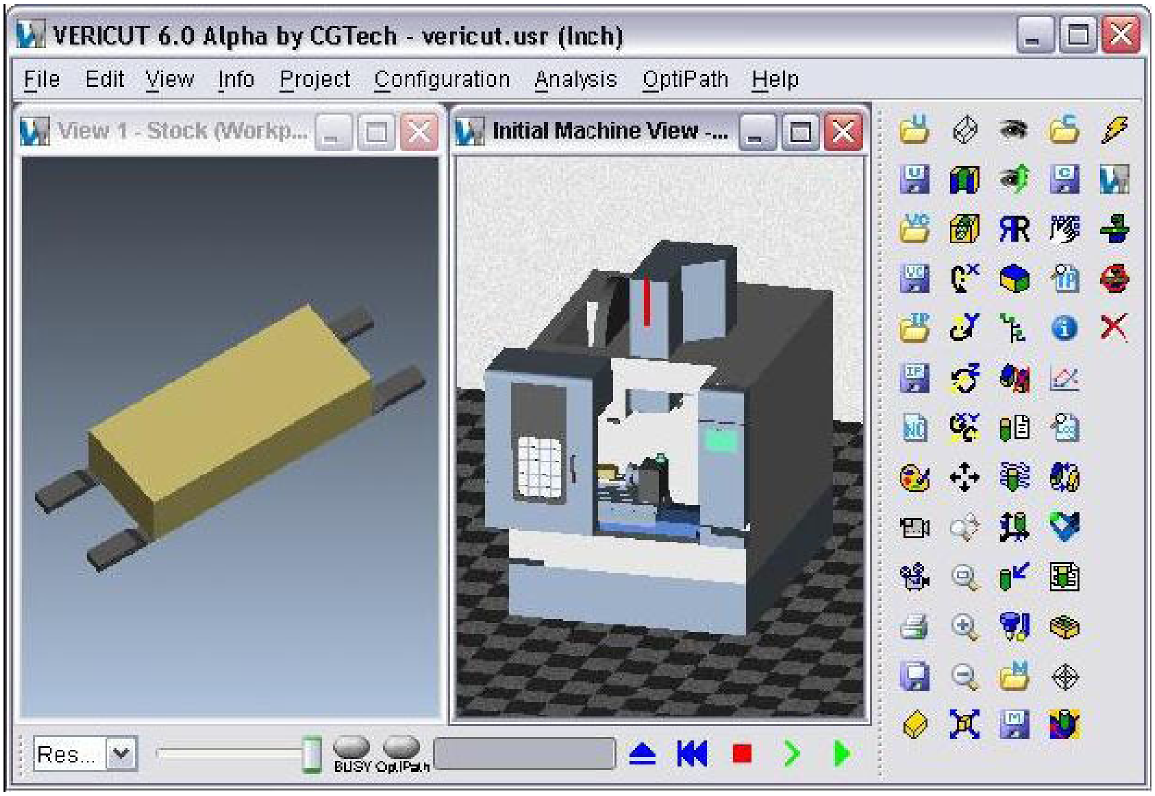
In this case we are docking the Toolbar window on the right side of the VERICUT main window. When the rectangle representing the window is dragged to a dockable position, the shape of the rectangle will change and the border will be displayed red as shown in Figure 3.

Figure 3



Releasing the left mouse button moves the window to the docked position as shown in Figure 4.

Figure 4



Introduction to VERICUT Reports

VERICUT Report provides the tools needed modify/create report templates enabling you to generate "custom" reports.

Generate reports that only include information important to your particular work environment, in a format that is most meaningful to the users of the report. Eliminate the need to search through the wide range of information that is included in a more generic "standard" report.

VERICUT Reports:

- Enables you to generate reports using one of the "standard" report templates supplied. The following "standard" report templates are included and can be found in the "library" directory of your VERICUT installation:

  inspection.VcTemplate - Use this template to create a basic inspection report that includes the inspection sequence table and graphic showing the location of the inspection points on the part. This is the default report template that Inspection Sequence will use if no other template has been specified.

  NOTE: Report template inspection.VcTemplate should only be used for creating reports from the **Inspection Sequence window**. Using it to create reports from other VERICUT applications will result in incorrect data being displayed in the report.

  vericut.VcTemplate - This is the default report template that VERICUT will use if no other template has been specified.

  vericut_basic.VcTemplate - Use this report template to create a report containing overview information about the VERICUT session. Includes a file summary, and a Tool Summary table and Tool Use graph for each NC program file.

  vericut_optipath.VcTemplate - Use this report template to create a report similar to the "basic" report but also includes OptiPath information.

  vericut_shadecopy.VcTemplate - Use this report template to create a report similar to the "basic" report but also includes View Capture images specified with **File > AutoSave: View Capture tab**.

  vericut_full.VcTemplate - Use this report template to create a report similar to the "View Capture" report but also includes OptiPath information.

  NOTE: The above five report templates should only be used for creating reports using **File menu > Create Report > ...** in the main VERICUT window. Using them to create reports from other VERICUT applications will result in incorrect data being displayed in the report.

vericut_tools.VcTemplate - Use this template to create a tool library report. This is the default report template that Tool Manager will use if no other template has been specified.

NOTE: Report template vericut_tools.VcTemplate should only be used for creating reports from the **Tool Manager window**. Using it to create reports from other VERICUT applications will result in incorrect data being displayed in the report.

- Enables you to modify a "standard" report template by changing the content, layout and/or format of the included information and save it as a "custom" report template.
- Enables you to produce "custom" report templates from "scratch". You decide the format, layout and content that best meet your needs.
- Provides the tools to include a wide variety of information from different sources. Include text, tables or pictures. Extract information from the VERICUT user file; include information from external files, or "generic tags" that will be updated with information specific to a particular user file at the time that the report is created.

What is involved with creating a custom VERICUT report?

Creating and deploying a custom report for use by VERICUT users involves the following steps.

1. Set up "styles" to be used for the report template
2. Define "user defined tags" that will be included in the template
3. Create the page layout for the report template
4. Using the report template

For more information on creating and using custom report templates, see the following:

1. See the **Report Template** topics in the **Project menu** section of *VERICUT Help*.
2. See the **Working with VERICUT Reports**, in the *Using VERICUT* section, in the *CGTech Help Library* for additional information.

VERICUT Menus and Features

File Menu

New Project

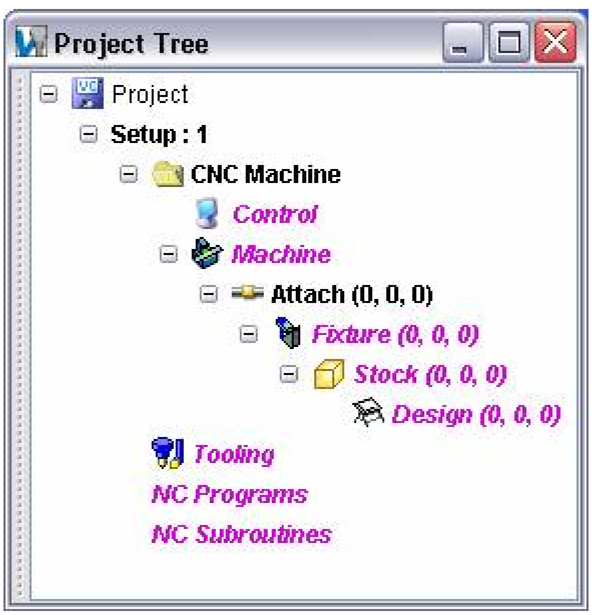
Location: **File menu > New Project > Inch / Millimeter**

Toolbar short cut for creating a New Project file: (**Inch**), or (**Millimeter**). Right-click on the icon to switch between the two modes.

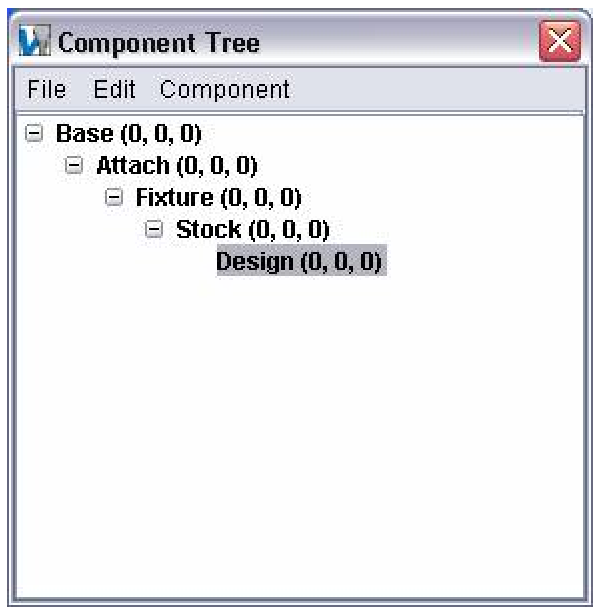
Sets VERICUT to a "new" condition, as if no files have been loaded. Afterwards, configure VERICUT by adding models, tool paths, descriptions of cutting tools, etc.

To establish the new condition, VERICUT purges all data from memory, and then opens the init.VcProject (inch project) or initm.VcProject (millimeter project) Project file from the CGTech library. The Project Tree and Component Tree shown below are automatically defined, ready for you to configure VERICUT by adding models, tool paths, descriptions of cutting tools, etc.

Project Tree after a new condition:



Component Tree after a new condition:



Tips:

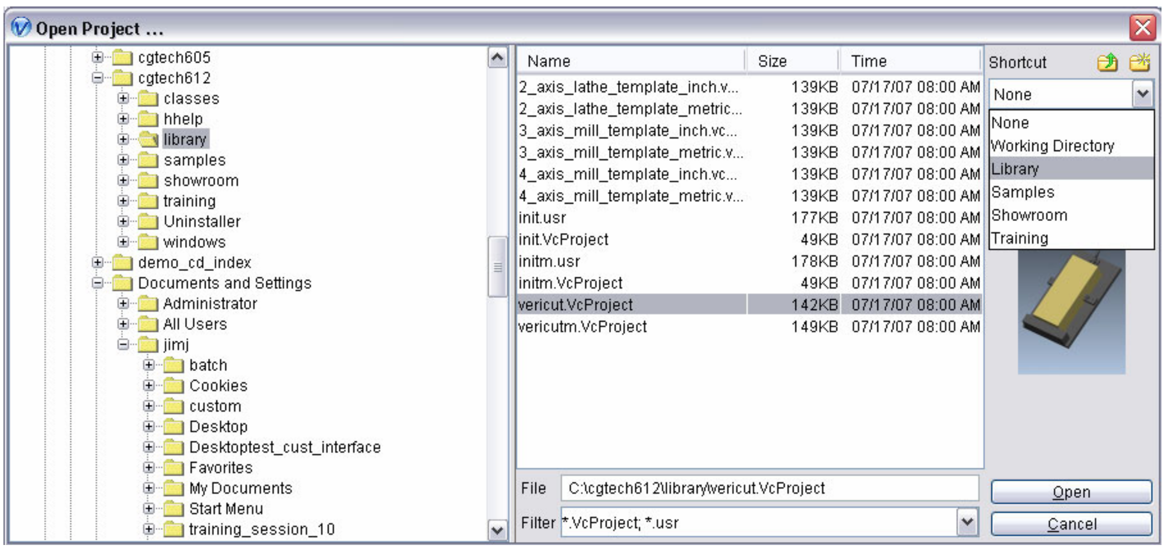
1. You can customize how VERICUT establishes the new condition by making changes to the init.VcProject or initm.VcProject files. Store the changed files in your working directory, or the CGTech library directory.
2. If needed, change the units for the new session as follows:
    A. Right-click in the graphics area and select **Project Tree** in the menu that displays.
    B. In the project tree, right click on the "project" branch, and select **Unit > Inch** or **Millimeter** in the menu that displays.

Open (Project)

Location: **File menu > Open**

Toolbar short cut for opening Project files: 

Use to open an existing Project file with the current VERICUT session settings. Use the **Open Project** file selection window that displays to select the desired project file. Most features on this window are standard file selection window features that enable you to navigate through directories, filter files, and type, or select, /path/filenames. A description of features specific to VERICUT can be found in the **Introduction to VERICUT File Selection Windows** in the Getting Started section of *VERICUT Help*.



Save Project

Location: **File menu > Save Project**

Toolbar short cut for saving Project files: 

Tip: Right-click on the icon to toggle between  (Save Project) and  (Save Project As) modes.

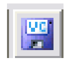
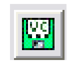
Saves (updates) an existing Project file with the current VERICUT session settings. VERICUT will save the project file if you have sufficient permissions to save the file in its present directory. Otherwise, the Save Project File As window will display enabling you to specify a location to save where you have write permissions.

Save As

VERICUT Users:

VERICUT Location: **File menu > Save As**

Toolbar short cut for saving Project files as: 

Tip: Right-click on the icon to toggle between  (Save Project) and  (Save Project As) modes.

Mold and Die Users:

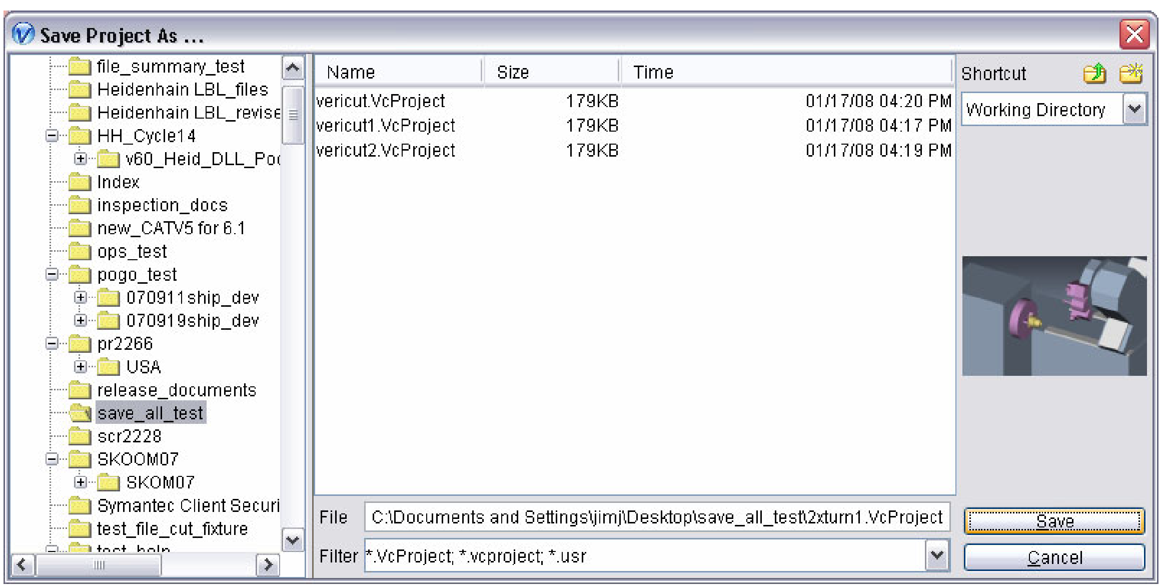
Mold and Die Location: **Setup page > Save the Setup** or **Preview and Optimize page > Save the Setup**

Notebook Feature: 

Cutter Grinder Users:

Cutter Grinder Location: **Setup page > Save the Setup** or **View Simulation page > Save the Setup**

Notebook Feature: 



Opens the **Save Project As** window enabling you to save Project files (ref. Project Files in the Getting Started section of *VERICUT Help*). Most features on this window are standard file selection window features that enable you to navigate through directories, filter files, and type, or select, /path/filenames. A description of features specific to VERICUT can be found in the **Introduction to VERICUT File Selection Windows** in the Getting Started section of *VERICUT Help*.

Save All

Location: **File menu > Save All**

Saves (updates) the Project file, Machine Files and Control Files associated with the current setup. It provides the combined functionality of **File menu > Save Project**, **Configuration menu > Control > Save** and **Configuration menu > Machine > Save**. In the event that a file cannot be saved (for example not having write permissions in the specified directory), a window will open enabling you to **Save As**.

Single Setup Project File:

When you select **Save All** for a single setup project file, the following will happen:

1. VERICUT will save the project file if you have sufficient permissions to save the file in its present directory. Otherwise, the **Save Project As** window will display enabling you to specify a location to save where you have write permissions.
2. VERICUT will save the machine file if you have sufficient permissions to save the file in its present directory. Otherwise, the **Save Machine File** window (ref. **Save As (Machine File)** in the Configuration menu section of *VERICUT Help*) will display enabling you to specify a location to save where you have write permissions.
3. VERICUT will save the control file if you have sufficient permissions to save the file in its present directory. Otherwise, the **Save Control File** window (ref. **Save As (Control File)** in the Configuration menu section of *VERICUT Help*) will display enabling you to specify a location to save where you have write permissions.

Multiple Setup Project File:

When you select **Save All** for a multiple setup project file, the following will happen:

1. VERICUT will save the project file if you have sufficient permissions to save the file in its present directory. Otherwise, the **Save Project As** window will display enabling you to specify a location to save where you have write permissions.
2. VERICUT will save the machine file associated with the "current" setup (ref. **Current Setup**, in the Project menu section of *VERICUT Help*) if you have sufficient permissions to save the file in its present directory. Otherwise, the **Save Machine File** window (ref. **Save As (Machine File)** in the Configuration menu section of *VERICUT Help*) will display enabling you to specify a location to save where you have write permissions.
3. VERICUT will save the control file associated with the "current" setup if you have sufficient permissions to save the file in its present directory. Otherwise, the

Save Control File window (ref. **Save As (Control File)** in the Configuration menu section of *VERICUT Help*) will display enabling you to specify a location to save where you have write permissions.

In Process

Open (In Process file)

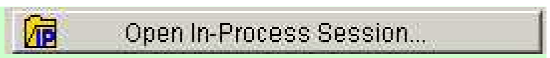
VERICUT Users:

VERICUT Location: **File menu > In Process > Open**

Toolbar short cut for opening IP files: 

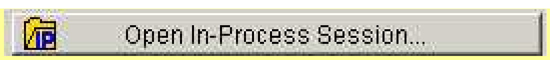
Mold and Die Users:

Mold and Die Location: **Setup page > Open In-Process Session**

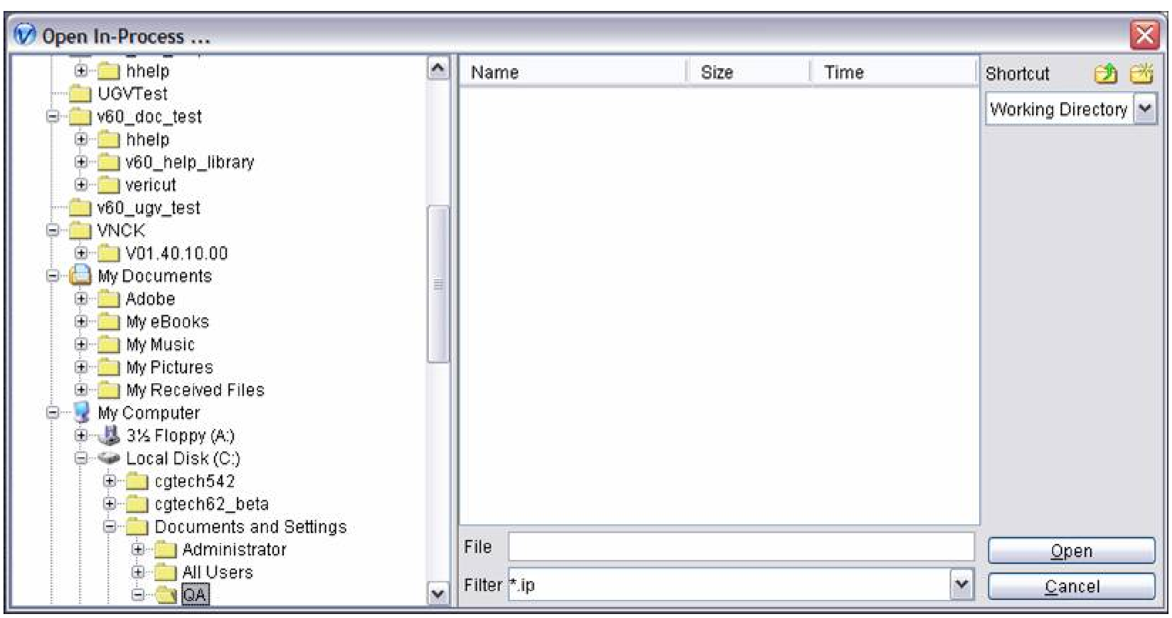
Notebook Feature: 

Cutter Grinder Users:

Cutter Grinder Location: **Setup page > Open In-Process Session**

Notebook Feature: 

Opens the **Open In-Process** window enabling you to open (load) an In Process file, or "IP file" (ref. **In-Process File** in the Getting Started section of *VERICUT Help*). IP files contain all the data necessary to re-establish a VERICUT session, including the current VERICUT model (with cuts), user interface settings, and a copy of the Log file.

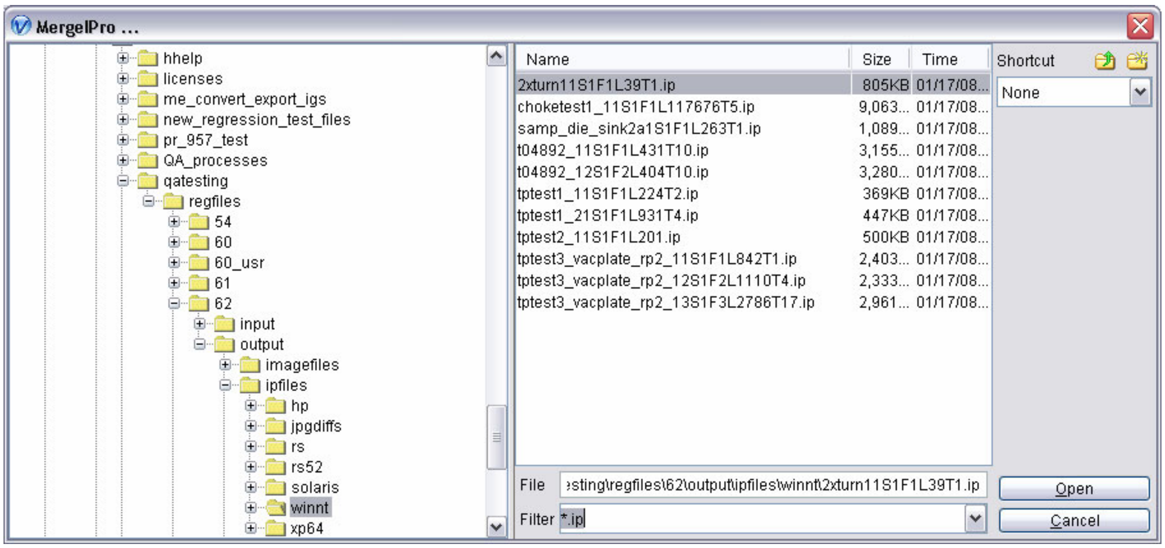


Most features on this window are standard file selection window features that enable you to navigate through directories, filter files, and type, or select, /path/filenames. A description of features specific to VERICUT can be found in the **Introduction to VERICUT File Selection Windows** in the Getting Started section of *VERICUT Help*.

Merge (In Process file)

VERICUT Location: **File menu** > **In Process** > **Merge**

Displays the **MergeIPro** window enabling you to select the In-Process (IP) file (ref. **In-Process File** in the Getting Started section of *VERICUT Help*) to be merged.



Most features on this window are standard file selection window features that enable you to navigate through directories, filter files, and type, or select, /path/filenames. A description of features specific to VERICUT can be found in the **Introduction to VERICUT File Selection Windows** in the Getting Started section of *VERICUT Help*.

Use the Merge feature to load an In-Process (IP) file without removing the current project from VERICUT session. This feature enables you to create multi-setup project files, one setup at a time without the need to reprocess previous setups. This is especially useful for setups that take a long time to process.

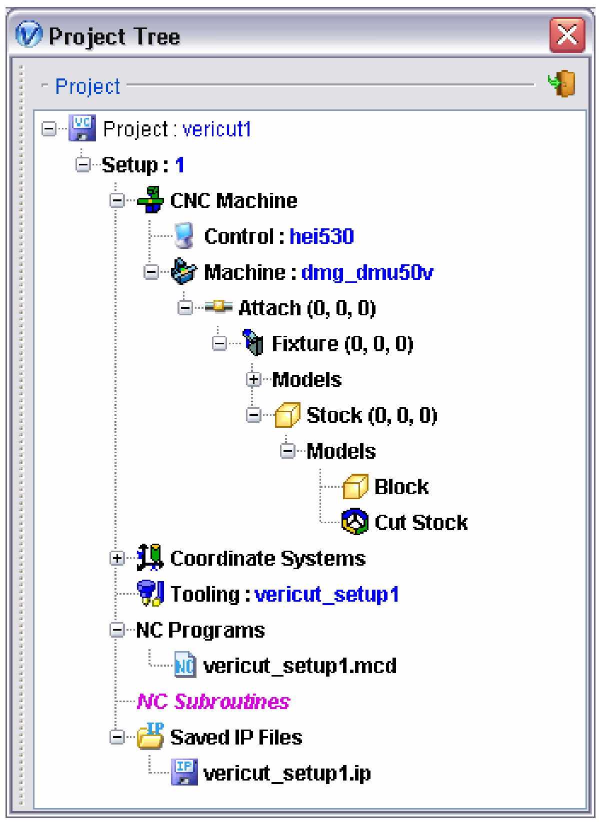
IP files can be successfully "merged" as long as they contain fewer processed setups than the number of setups in the project file that they are being merged into. If the IP file has an equal number, or greater number of setups than the number of setups in the project file that they are being merged into, then the Merge operation is equivalent to **File menu** > **In Process** > **Open**.

When a "merge" operation is successful, the current VERICUT project data remains unchanged, but the setups in the IP file, replace an equal number of setups in the project file. Any setups in the project file, as shown in the Project Tree, below the setups merged from the IP file remain unchanged.

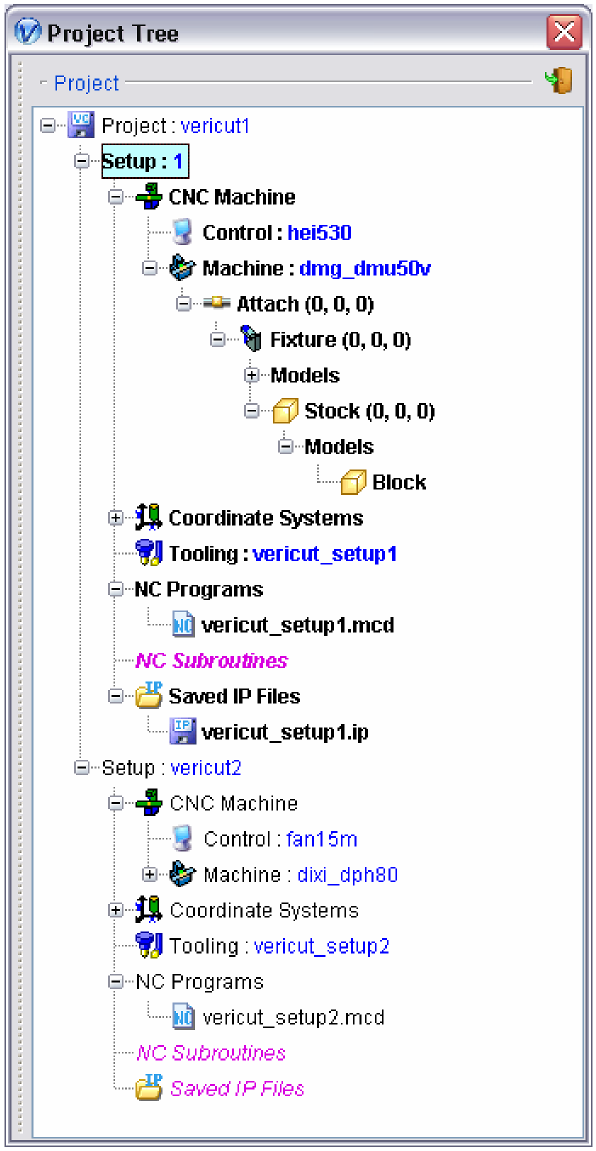
NOTE: There are no checks, or validation, to determine whether or not the IP file is related to current project.

Simple Merge IP file example:

1. Create the project file containing the first setup.
2. Process the first setup. If the results are OK, save an IP file (**File menu > In Process> Save**). Save the project file (**File menu > Save Project**). The Project Tree should now look something like the picture below.



3. Add the next setup to the project file. The Project Tree should now look something like the picture on the next page.

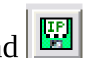


4. Before processing the second setup, merge the IP file you saved in step 2. (You can either use **File menu** > **In Process** > **Merge** in the VERICUT main menu, or in the Project Tree, right click on the IP file shown in Setup 1 and select **Merge IP File** from the pull-down menu.)
5. Once the "merge" is complete, you are ready to process Setup 2 without re-processing Setup1. Setup 1 is restored to the state it was in when you saved the IP file, Setup 2 is unchanged.

Save (In Process file)

Location: **File menu > In Process > Save**

Toolbar short cut for saving IP files: 

TIP: Right-click on the icon to toggle between  (Save In-Process) and  (Save In-Process As) modes.

Saves (or updates an existing) In-Process file, or "IP file" (ref. **In-Process File** in the Getting Started section of *VERICUT Help*), with the current VERICUT model, including all machine cuts. VERICUT will save the In-Process file if you have sufficient permissions to save the file in its present directory. Otherwise, the Save In-Process As window will display enabling you to specify a location to save where you have write permissions.

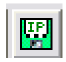
In Process (IP) files are by default compressed when they are saved. VERICUT can read these compressed files directly.

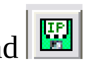
To disable writing compressed IP files, set the environment variable CGTECH_COMPRESS=IP (ref. **Environment Variables** in the *VERICUT Help* section, in the *CGTech Help Library* for additional information).

Save As (In Process file)

VERICUT Users:

VERICUT Location: **File menu > In Process > Save As**

Toolbar short cut for saving IP files: 

TIP: Right-click on the icon to toggle between  (Save In-Process) and  (Save In-Process As) modes.

Mold and Die Users:

Mold and Die Location: **Preview and Optimize page > Save In-Process Session**

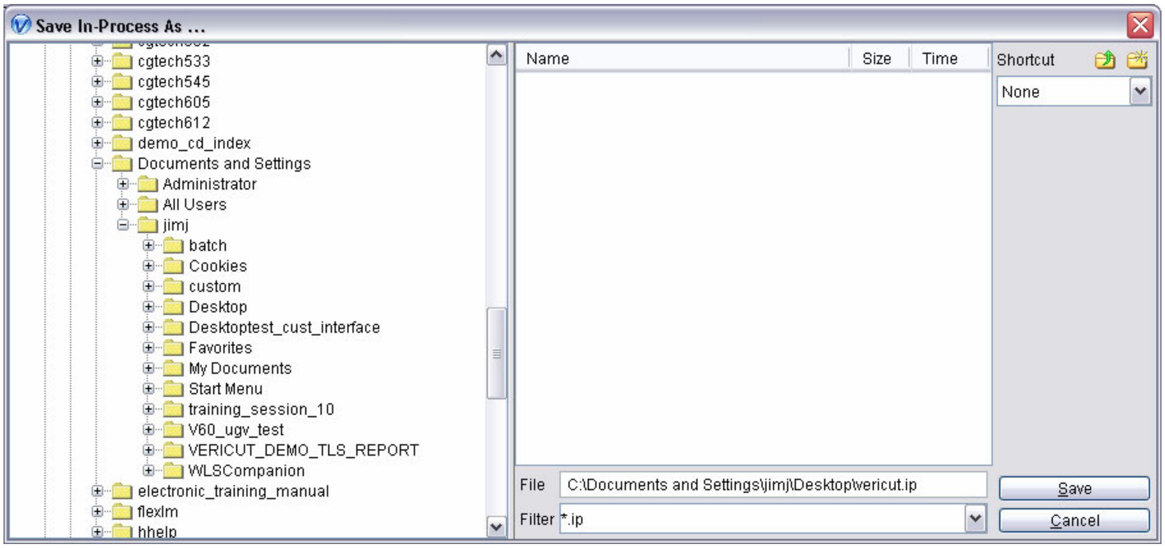
Notebook Feature: 

Cutter Grinder Users:

Cutter Grinder Location: **View Simulation page > Save the In-Process Session**

Notebook Feature: 

Opens the Save In-Process As window enabling you to save (or update an existing) In-Process file, or "IP file" (ref. **In-Process File** in the Getting Started section of *VERICUT Help*), with the current VERICUT model, including all machine cuts. IP files contain all the data necessary to re-establish a VERICUT session, including the current VERICUT model (with cuts), user interface settings, and a copy of the Log file (ref. **Log File** in the Getting Started section of *VERICUT Help*).



Most features on this window are standard file selection window features that enable you to navigate through directories, filter files, and type, or select, /path/filenames. A description of features specific to VERICUT can be found in the **Introduction to VERICUT File Selection Windows** in the Getting Started section of *VERICUT Help*.

In Process (IP) files are by default compressed when they are saved. VERICUT can read these compressed files directly.

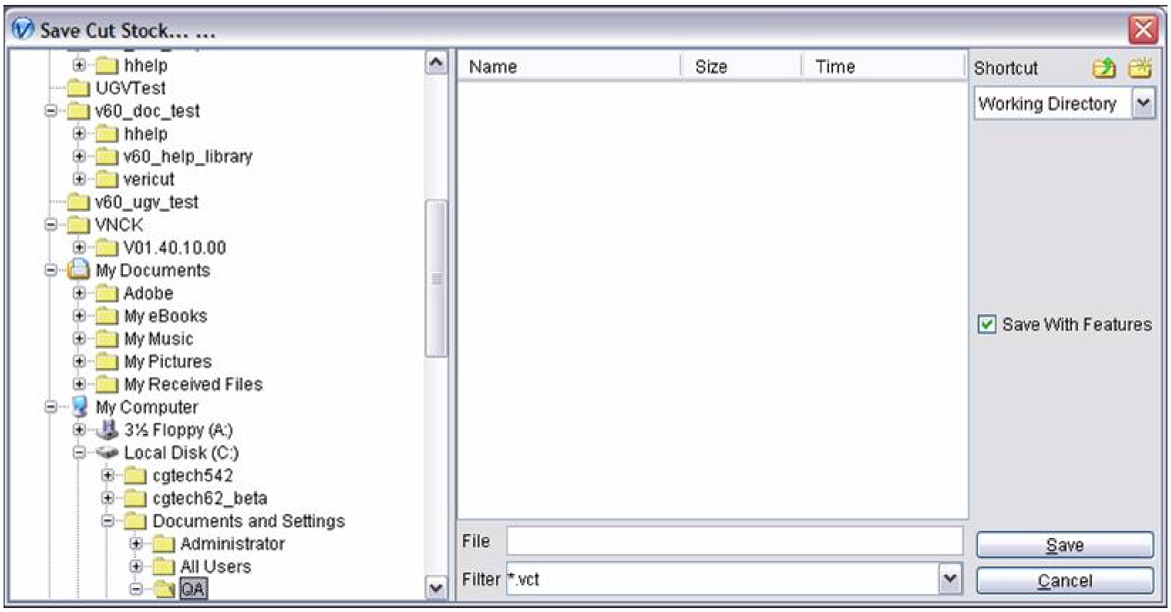
To disable writing compressed IP files, set the environment variable CGTECH_COMPRESS=IP (ref. **Environment Variables** in the *VERICUT Help* section, in the *CGTech Help Library* for additional information).

Save Cut Stock

VERICUT Solid

VERICUT Location: **File menu > Save Cut Stock > *stock name***

Save Cut Stock opens the **Save Cut Stock** window enabling you to save the selected Cut Stock model as a VERICUT Solid file (.vct). This option is only active after a Cut Stock model has been created by VERICUT. The default file extension is .vct.



Most features on this window are standard file selection window features that enable you to navigate through directories, filter files, and type, or select, /path/filenames. A description of features specific to VERICUT can be found in the **Introduction to VERICUT File Selection Windows** in the Getting Started section of *VERICUT Help*.

A cut stock model can be saved as a VERICUT Solid file "with features" or "without features". Toggle the **Save with Features** option, in the Save Cut Stock window, **On** (the default) or **Off**.

When the cut stock model is saved "with features", the cut database, the full feature (cylinder, plane, torus, etc.) database, cut and stock colors are stored in the VERICUT Solid file in addition to the geometry data. When the cut stock model is saved "without features", only the geometry data is saved in the VERICUT Solid file.

VERICUT Solid (.vct) files are by default compressed when they are saved. VERICUT can read these compressed files directly.

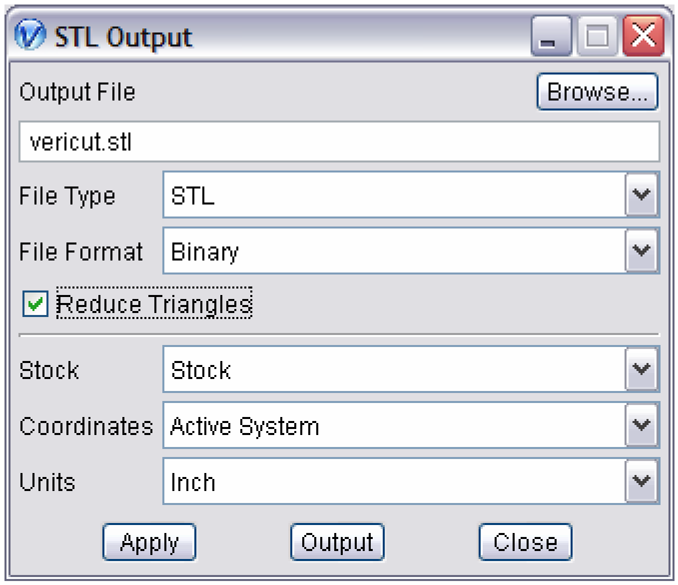
To disable writing compressed VERICUT Solid files, set the environment variable CGTECH_COMPRESS=VCT (ref. **Environment Variables** in the *VERICUT Help* section, in the *CGTech Help Library* for additional information).

This feature provides access to the same functionality as **File > Save Cut Stock** in the in the **Component Tree window**.

See **VERICUT Solid File**, in the *Getting Started with VERICUT* section, of *VERICUT Help* for more information.

STL

VERICUT Location: **File menu** > **Save Cut Stock** > **STL**



Output File — Enter the *path**file* name in the text field of the file to receive exported model data or click on **Browse** and use the Export Model File selection window to specify the file.

File Type — Type of file to export.

Options are: **VERICUT** and **STL**

File Format — Format in which STL or VERICUT model data is exported.

Options are: **ASCII** or **Binary**.

Reduce Triangles — When unchecked (the default), the STL file is created using the original STL output (as found under Model Export), with no triangle reduction. A Model Export license is not required. When checked, the STL file is created using the triangle reduction method as in V6.1. The state is saved in the project file as a global project setting (not a setup setting).

Stock — When multiple Stock components are defined, this option enables you to select which one to export.

Coordinates — Coordinate system in which model data is exported.

Options are: **Stock** (stock component origin) or **Active System** (See **Project menu** > **Coord. System**).

Units — Unit measurement system in which model data is exported.

Options are: **Inch** or **Millimeter**.

Apply — Saves the STL Output window settings.

Output — Processes and exports the VERICUT model according to the STL Output window settings.

Close — Closes the STL Output window.

CAD Model

Export Cut Stock window

Location: **File menu > Save Cut Stock > CAD Model**

Opens a window enabling you to export the VERICUT model. Export formats include: Stereo lithography (STL) or VERICUT Polygon model files, or IGES data. A cut VERICUT model is exported with machine cuts intact. During the export operation a Model Export Boundaries window may open enabling you to manually close ambiguous open boundaries that VERICUT was unable to resolve, prior to completing the export. When the model has been exported, a message informs you how many IGES surfaces (or model file triangles) were created.

Tips on exporting models:

1. Make sure that Model Export Cut Mode in the **File menu > Properties > General tab** is toggled "On".
2. Make sure that Display Holders in Workpiece View in the **Project menu > Processing Options > Motion window** is toggled "Off".

3. Make sure that Visibility is set such that models of fixture components are not visible in the Workpiece View. See **Model menu > Model Definition > Component Attributes tab** for more information.
4. When you cut the model a message similar to "For best AUTO-DIFF, OptiPath and Model Results, set Cut Tolerance to 0.XXXX, due to small tool features" will appear in the logger. Use this information as a starting point for setting Cutting Tolerance and Base Cut Tolerance on values on the **File menu > Properties > Tolerance tab**.
5. To export a higher quality model, reduce the Cutting Tolerance.
6. You can section cut models, then export what remains in the sectioned view. This can be helpful to export portions of complex models on computers with limited resources.
7. Also see **Notes about Using Model Export**, in the *Notes about Special Topics* section, in the *CGTech Help Library*.

Settings tab **—** Features on this tab establish general properties for the model export.

Options tab **—** Features on this tab are used specify properties related to specific methods of processing VERICUT model geometry.

Apply — Saves the Export Model window settings.

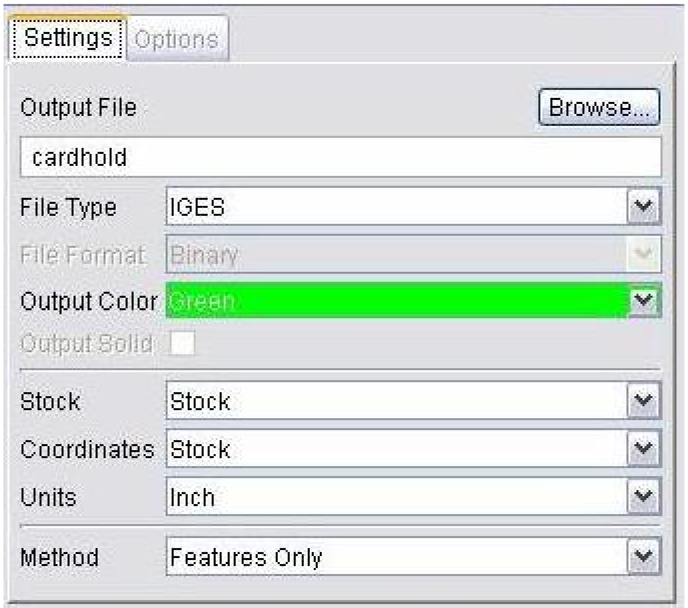
Output — Processes and exports the VERICUT model according to the Export Model window settings. During the export operation a **Model Export Boundaries window** opens providing control over small surfaces that may cause problems in your CAD system. When the model has been exported, a message informs you how many IGES surfaces (or model file triangles) were created.

Close — Closes the Export Model window.

Export Cut Stock window, Settings tab

Location: **File menu > Export Cut Stock**

Features on this tab establish general properties for the model export.



Output File — Enter the \path\file name in the text field of the file to receive exported model data or click on Browse and use the Export Model File selection window to specify the file.

File Type — Type of file to export. Options are: VERICUT, STL, IGES, CATIA V5, ACIS, CATIA V4 and STEP.

File Format — Format in which STL or VERICUT model data is exported. Options are: ASCII or Binary.

Output Color — Use to specify the color that is stored in the IGES file for the exported model. (Only active when File Type = IGES)

Output Solid — Use to export a solid model. (Only active when File Type = CATIA V5, ACIS, CATIA V4, or STEP)

Stock — When multiple Stock components are defined, this option enables you to select which one to export.

Coordinates — Coordinate system in which model data is exported. Options are: Stock (stock component origin) or Active System (See **Project menu > Coord. System**).

Units — Unit measurement system in which model data is exported. Options are: Inch or Millimeter.

Method — Controls the method of processing VERICUT model geometry into the specified data format. Features and Patches, Features Only and Turned Features Only are only active when File Type = IGES, CATIA V5, ACIS, CATIA V4, or STEP. Slices is only active when File Type = IGES

Options:

Features and Patches — Builds synthetic features from several small adjacent features that are difficult to resolve in 'Features Only' mode. Analyzes the cut model using UV grid logic and group machined features by grid region. Grid Count specifies the number of grid squares in U and V space, spread over the cut model surface. VERICUT attempts to create one surface within in each grid, or as few surfaces as possible. This option is recommended for cut models with smooth or transitioning surfaces, for example: mold and die cavities and patterns, blended or sculptured surfaces, etc.

Features Only — Groups similar machined features into like IGES data entities. This option is recommended for cut models with distinct differences between machined features, for example: aerospace structural components, pocketed parts, parts with strengthening ribs/walls and stepped floors, etc.

Turned Features Only — Exports the turned, or "revolved", representation of the cut model. Turned models are exported as an IGES file and contain the 2-D turned profile and surface of revolution.

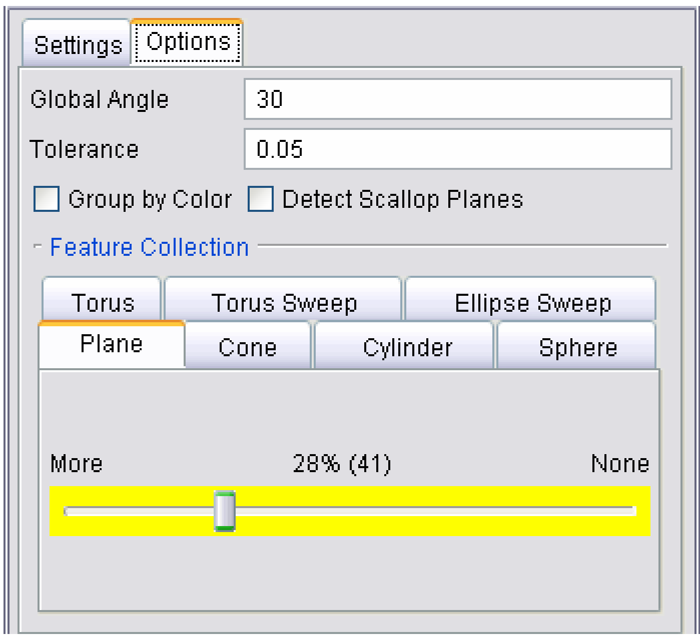
Slices — Exports the cut model such that resulting IGES output is a set of composite curves representing slice profiles of the cut stock.

To Export Cut Stock window

Export Model window, Options tab

Location: **File menu > Export Cut Stock**

Features on this tab are used specify properties related to specific methods of processing VERICUT model geometry. The options displayed will vary depending on the method of processing (either **Features and Patches** or **Slices**) selected on the **Settings** tab.



Process = Features and Patches

Global Angle — Angle which, if exceeded, divides the machined surface into multiple IGES surfaces. Enter the angle in degrees. Increasing the angle decreases output file size and decreases IGES data accuracy. On "Grid" processed models Global Angle affects the output of all machined surfaces.

NOTE: When you increase the Grid Count (more grid sections), the Global angle can also typically be increased since fewer features are considered for each grid.

Tolerance — Use to specify the amount that the exported model surface can deviate from the cut model surface. The smaller the value, the closer the output model will approximate the cut stock.

NOTE: This tolerance value should be equal to, or greater than, the cut tolerance.

Group by Color — When toggled "On" (checkmark visible), patches will be created to only contain data related to a single color.

Detect Scallop Planes — Toggle "On" to activate "scallop plane logic" to process. Processing is faster when toggled "Off".

Feature Collection — Use to specify the size of the area to be taken into account when differentiating between a "feature" and a "surface". Select the appropriate "feature" tab, (**Plane**, **Cone**, **Cylinder**, **Sphere**, **Torus**, **Torus Sweep**, or **Ellipse Sweep**) then use the slider to specify the size of the area. The percent value, shown above the slider, represents the percentage of the total slider distance of the current setting, where the **More** end of the slider is 0% and the **None** end of the slider is 100%. The number in parenthesis, shown above the slider, represents the actual size of the area (specified in $inches^2$, or $millimeters^2$) to be taken into account for the feature. This value is comparable to those shown in the Size column in the Preview and Combine window. The slider displays yellow as soon as it is moved off the **None** position.

The **More** end if the scale is approximately equal to .0001 of the part size. **None** end of the scale is approximately equal to .1 of the part size.

Process = Slices

Slice Direction — Use to specify the direction of the slices. Select X, Y, or Z based on the axes of the cut stock model.

Number of Slices — Use to specify the total number of slices to create.

NOTE: If the specified number is greater than the number of grid planes internally stored in the cut stock database, the number of slices output will automatically be reduced to this smaller value.

Local Angle — determines the max angle between 2 adjacent pieces of one slice when they still can be combined in one segment.

Tolerance — Use to specify the amount that the exported curves can deviate from the cut model surface. The smaller the value, the closer the output curves will approximate the cut stock surface.

NOTE: This tolerance value should be equal to, or greater than, the cut tolerance.

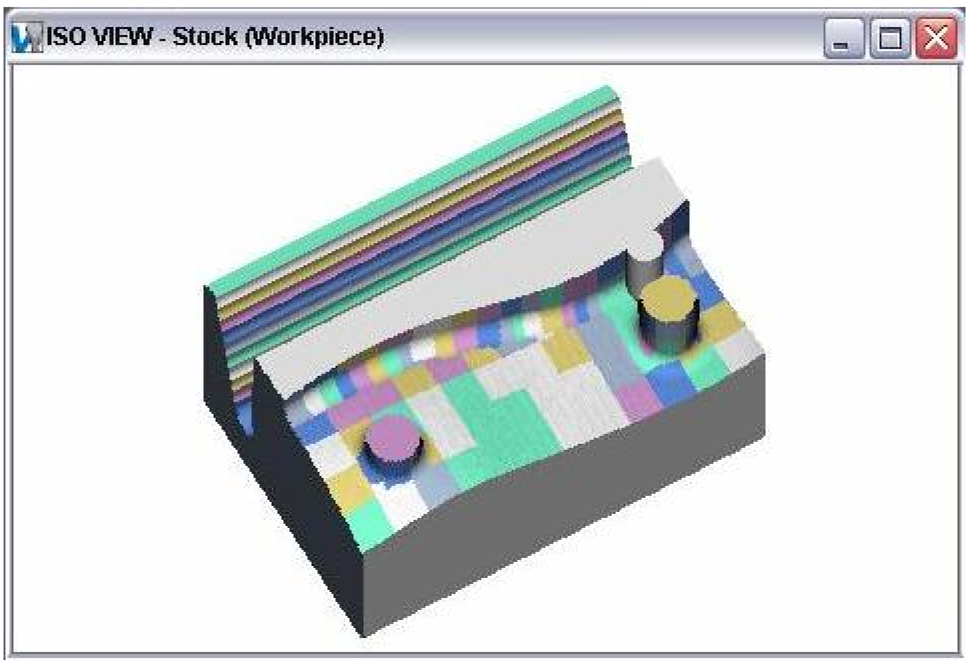
To Export Cut Stock window

Export Cut Stock, Preview and Combine window

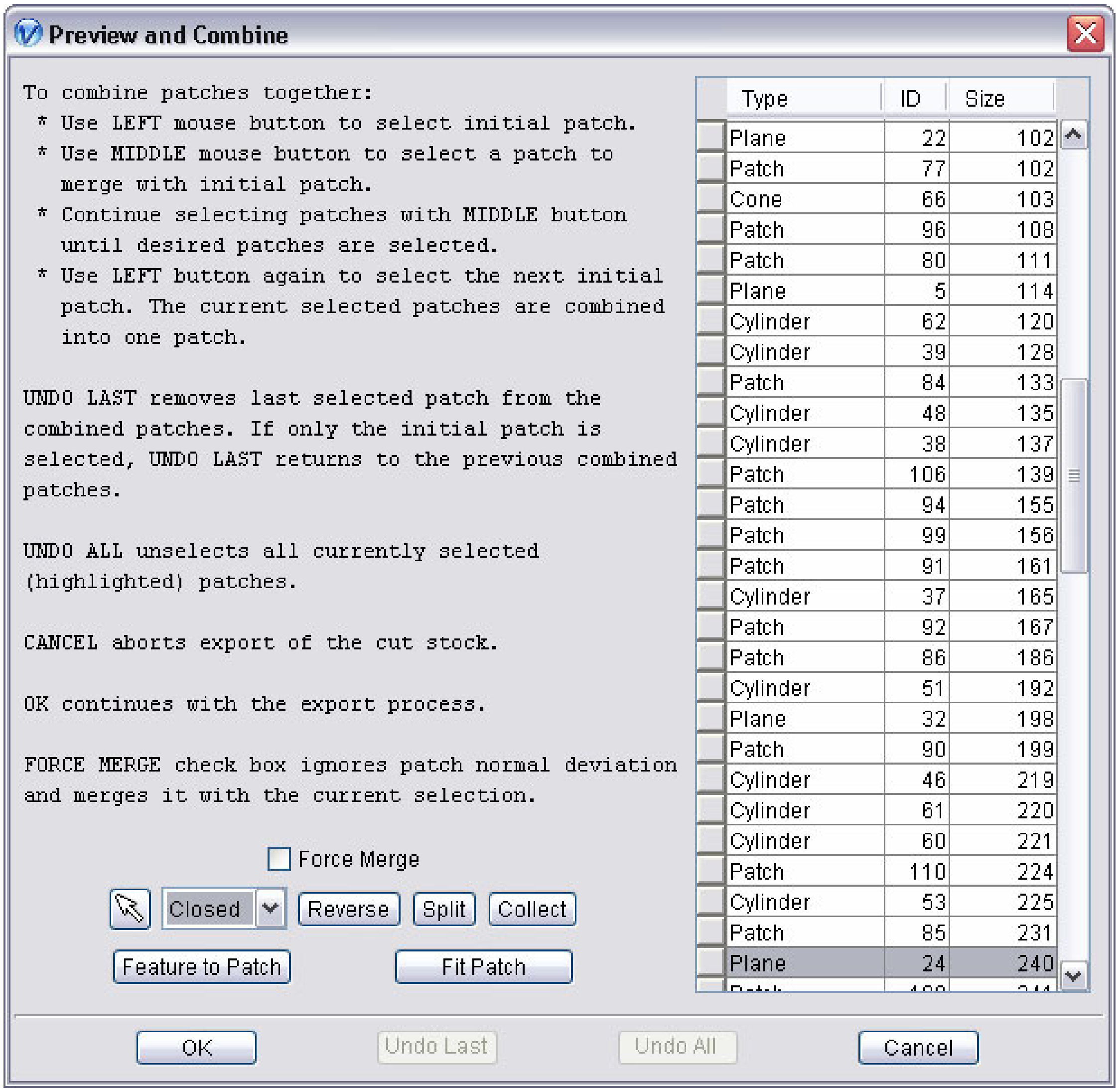
After VERICUT completes a preliminary analysis of the model, the model display changes to show multi-colored patches (see below), and the Preview and Combine window displays enabling you to:

- Preview the model before going through the entire model export process. This enables you to cancel the current export and update settings before proceeding.
- Merge adjacent patches for smoother output, faster model export, and smaller model sizes.
- Split patches for more efficient merges.

VERICUT display for Preview and Combine Patches



Most of the features available, and how to use them, are described right in the window. The others are described below.



 (Pick mode) **—** Use to toggle "pick mode" on and off.

Opened/Closed — Use to specify whether the points selected while in pick mode create an open or closed boundary. Pick mode must be toggled "off" in order to select **Opened** or **Closed.**

Reverse — Use to reverse the order of the points that are picked in pick mode. The direction of in which the points are picked will affect which part of a patch is retained and which part will be removed. It also has an affect on the direction of the resulting surface normal.

Split — Use this feature split a patch. Use either an **Opened** or **Closed** pick mode boundary to specify the location of the split.

Collect — Use this feature to enable the collection of patches based on **Opened/Closed** boundary construction.

Feature to Patch — Use to convert the selected feature to a patch.

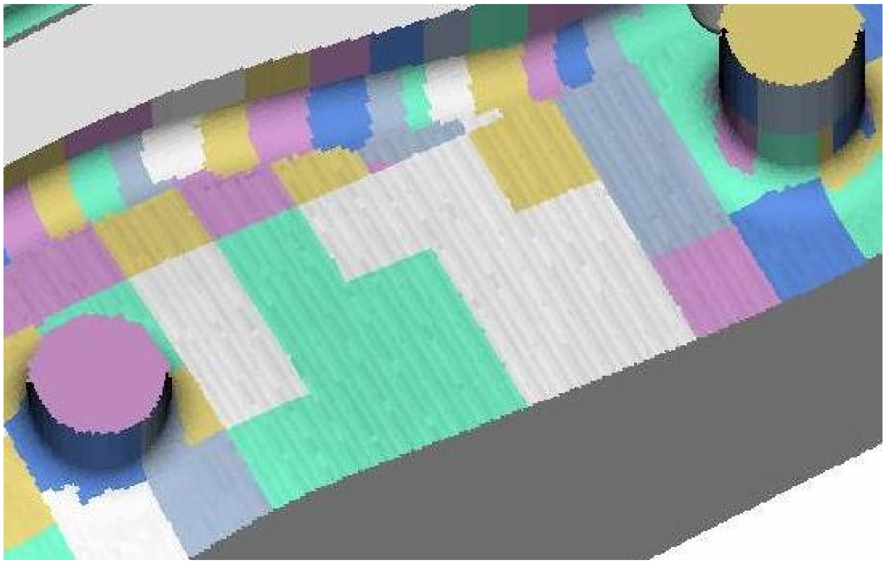
Fit Patch — Use to resize and center the graphics window display around the selected feature/patch. Select the feature/patch in the table or by clicking on the feature/patch in the graphics area.

Tip: Click on the column headings in the Feature Table to sort the table by that characteristic. For example, sorting by **Size** enables small features to be quickly identified.

Export Cut Stock, "Combine" Example

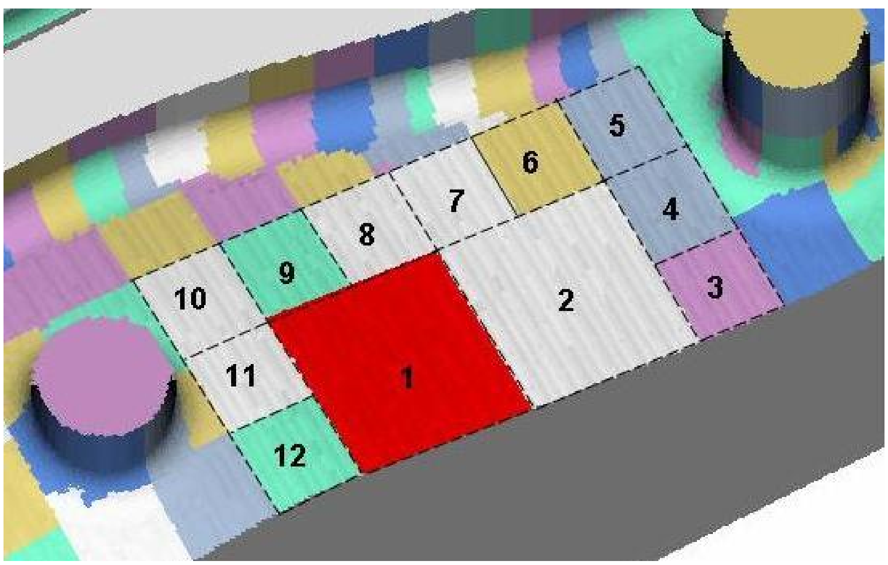
The following example shows the process of combining a number of smaller patches into a single patch.

When Preview and Combine Patches is toggled "On", VERICUT completes a preliminary analysis of the model, then displays the model as multi-colored patches as shown below.

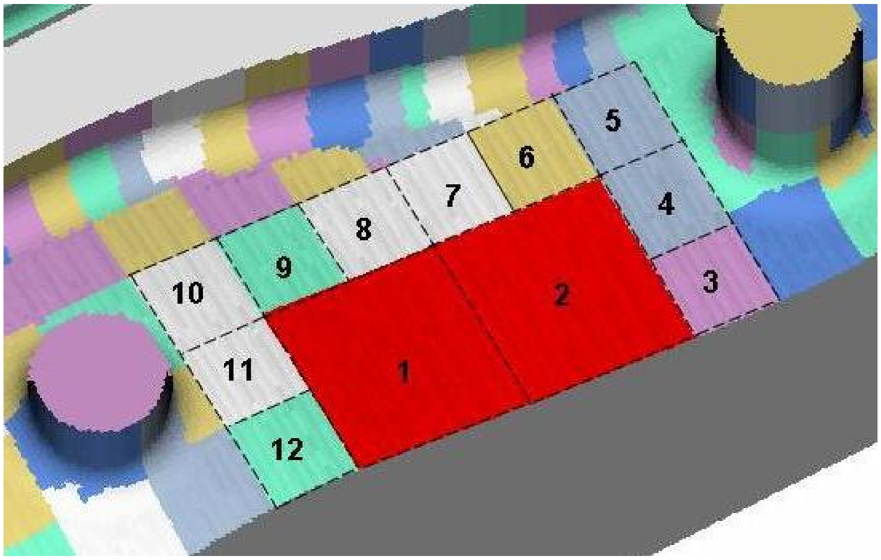


To combine patches, start by selecting the "initial" patch (in this case patch # 1) using the left mouse button. The initial patch will display as shown below.

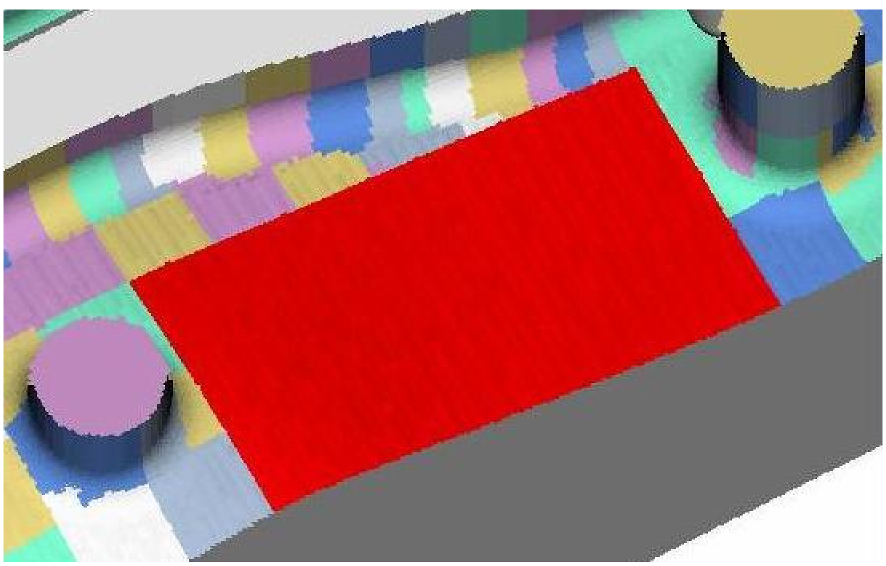
NOTE: Patch boundaries and numbers have been added for illustration purposes and will not be seen in the VERICUT display.



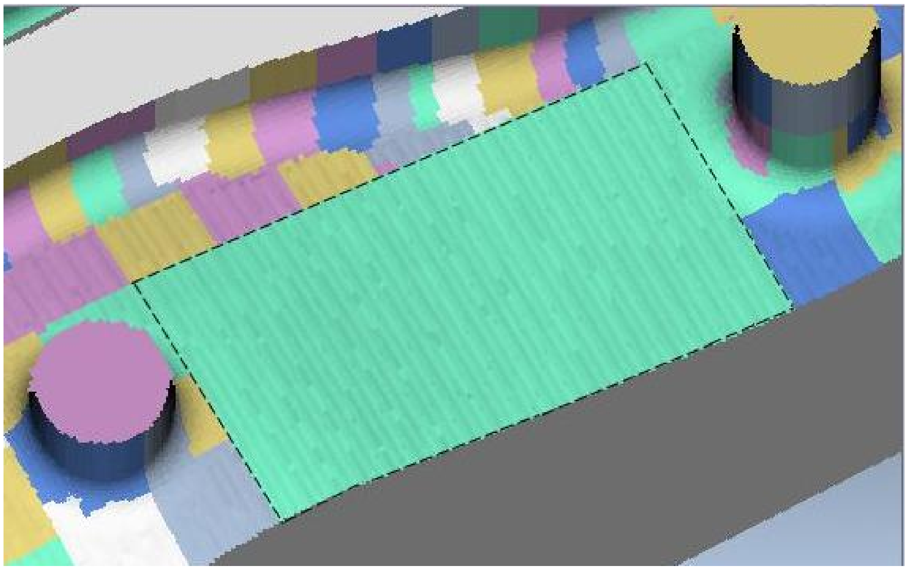
Select the first patch to be merged (in this case patch #2) with the middle mouse button. Any patch adjacent to the "initial" patch could have been selected for merging. The display updates as shown.



Continue selecting adjacent patches to merge using the middle mouse button. The picture below shows the display after all 12 of the numbered patches have been selected for merging.



When finished selecting patches to be merged, click on the part with the left mouse button to combine the selected patches into a single patch as shown in the picture below.

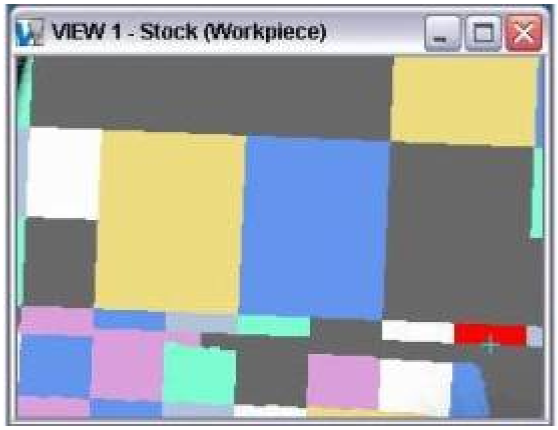


Export Cut Stock, Preview and Combine Techniques

The following techniques are provided to assist you in maximizing the quality of the models you create using the Export Cut Stock features.

1. Avoid long narrow strips.

The first image shows original patches.



The second image shows a resultant long narrow patch which should be avoided.

The third image shows an ideal resultant patch.

2. Consider neighboring node propagation.

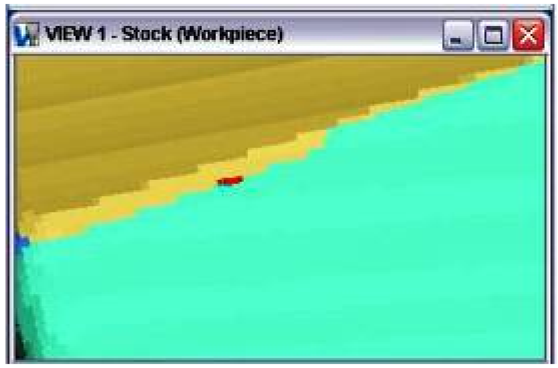
The first image shows original patches.

The second image shows a resultant patch which will be split do to neighboring nodes.

The third image shows what should be done to minimize unnecessary splits.

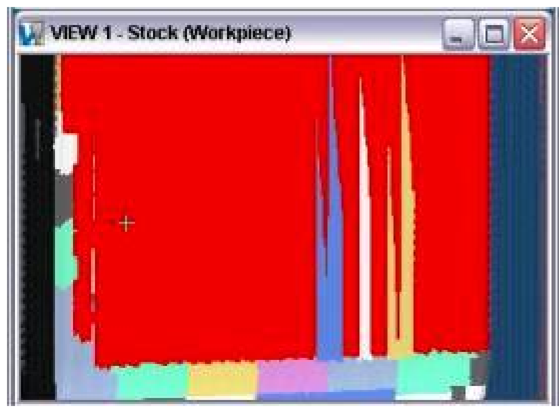
3. Merge small patches.

Use the table to identify small patches by sorting by the Size column. Once small patches have been identified combine small patch with larger patch or combine a group of small patches to create one large patch. The image at the right shows a small patch (in red) that should be merged with one of the two larger patches.

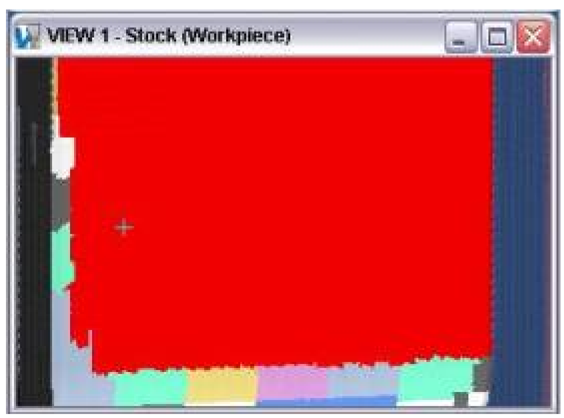


4. Merge all small plane anomalies.

All small anomalies within a plane should be combined with the plane. The first image shows a plane (in red) with several small anomalies within the plane.

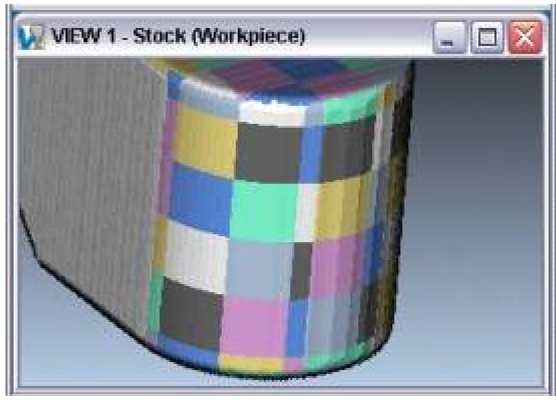


The second image shows the plane with the small anomalies combined with the plane.

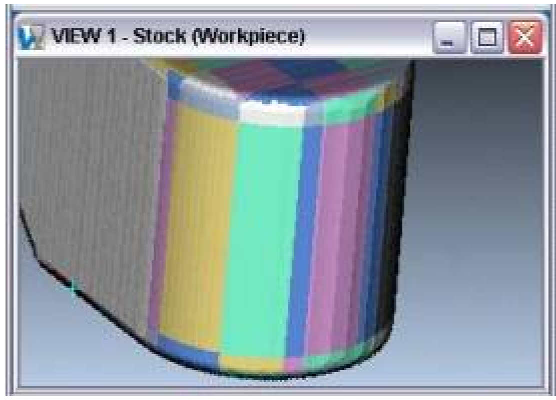


5. Consider initial design intent.

When combining patches take into consideration intended design feature. The initial design intent of the images below is a cylinder.
The first image shows the original patches.

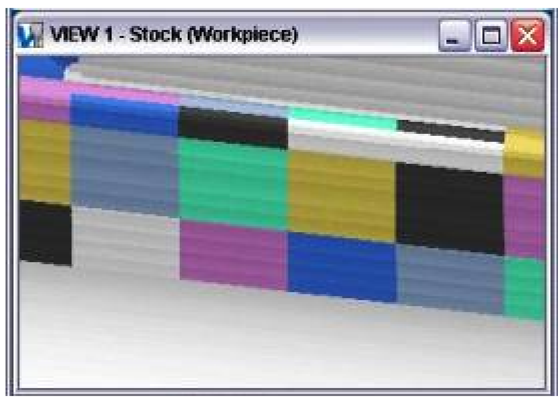


The second image shows the combined patches considering the initial design intent.



6. Tabulated cylinder representation.

Patches should be combined to represent a tabulated cylinder. The first image shows the original patches.

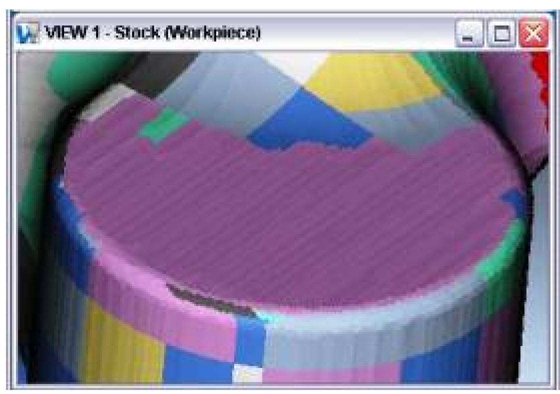


The second image shows what should not be done.

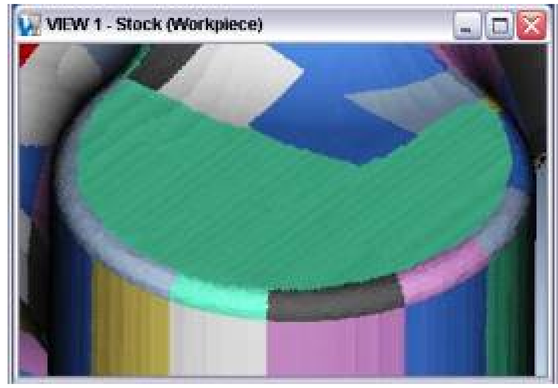
The third image shows the ideal patch combination abiding by the tabulated cylinder rule.

7. Use Split and Merge to improve fillets.

The first image shows the patches from the original collection.

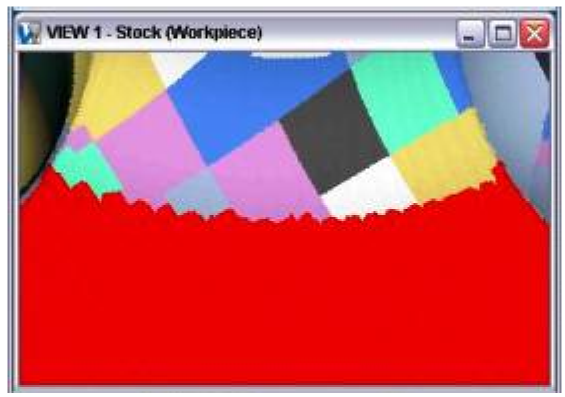


The second image shows the patches after using split and merge to improve fillet areas.

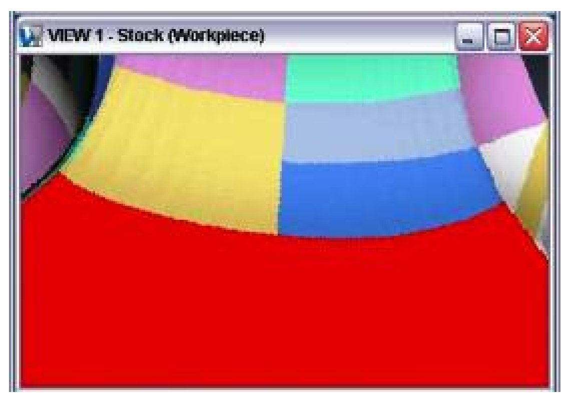


8. Use Split and Merge to improve jagged boundaries.

The first image shows the original patch boundary.

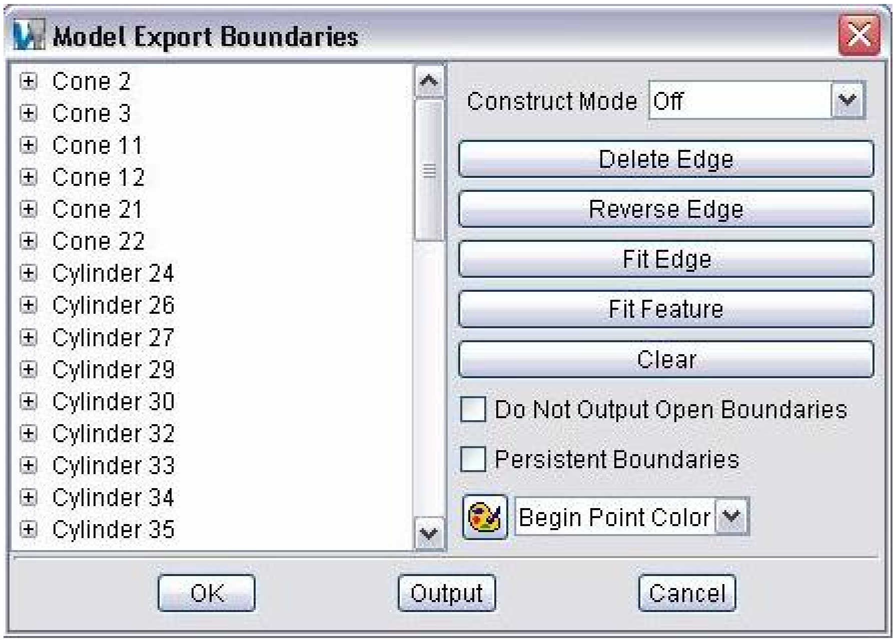


The second image shows the patch, after using split and merge to improve boundary.



Model Export Boundaries window

The features in the Model Export Boundaries window enable you to close difficult or poorly defined feature trimming edges before exporting the model.



Feature/Boundary Table — Contains a list of features. Each "feature" contains one or more open edges that VERICUT was not able to close automatically. Click on the "+" sign to the left of the feature to display a list of edges associated with it.

Construct Mode — These options enable you to manually close open feature edges prior to exporting the model. To activate Construct Mode, select any option in the list other than "Off". A red and white "cross-hairs" cursor indicates when Construct Mode is active.

Options:

Off — Turns off Construct Mode. (You can also use the "escape" key on the keyboard)

Project Edge — Use to extend an edge along a feature/feature intersection, one point at a time by projecting the "Begin / End" point of the highlighted edge to the selected point on the intersection.

Trim Edge — Use to eliminate overlapping boundary curves by trimming the "Begin / End" point of the highlighted boundary to the selected location.

NOTE: It is better to trim the boundary short, leaving a small gap that can then be closed than to have an "overlap" condition in the boundary.

Add Edge — Use to connect two edges within the same feature. The "begin/end" point of the highlighted edge is connected to the "end" or "begin" point on the selected edge.

NOTE: Highlight the edge by selecting it in the Feature/Edge table or by clicking on the edge in the graphics area with the center mouse button.

Construct Patch — Use to construct a patch to fill an open area between features.

Delete Edge — Use to delete the highlighted edge.

Reverse Edge — Use to reverse the direction of the highlighted edge.

Fit Edge — Use to resize and center the graphics window display around the highlighted edge. Select the edge in the Feature/Edge table or by clicking on the edge with the center mouse button in the graphics area.

Fit Feature — Use to resize and center the graphics window display around the selected feature. Select the feature in the Feature/Edge table or by clicking on the feature in the graphics area.

Clear — Use to clear the graphics display of "persistent boundaries".

Do Not Output Open Boundaries — Toggle "on" to prevent the export of open boundaries.

Persistent Boundaries — When toggled "on", enables you to display multiple feature boundaries. Use to display the feature boundaries adjacent to the edge that you are trying to close. Use Clear, described above, to remove "persistent boundaries" from the graphics display.

Color Pallet — Select an object from the list then click on the color pallet icon to display the color pallet. Select the color for the selected object. You can specify colors for the following objects:

Begin Point Color — Use to specify the color to display the "begin" point of an open feature boundary.

End Point Color — Use to specify the color to display the "end" point of an open feature boundary.

Boundary Color — Use to specify the color to display "open" feature boundaries.

Projection Color — Use to specify the color to display the projected feature curve that displays when using Construct Mode options Project, Trim or Extend are used.

Selected Color — Use to specify the color to display "selected" open boundary color (the highlight color of the active boundary).

Closed Color — Use to specify the color to display "closed" feature boundaries.

Node Color — Use to specify the color to display "node" points (the point where two adjacent edge segments connect on a feature boundary).

OK — Creates the export file and closes the Model Export Boundaries window.

Output — Creates the export file and leaves the Model Export Boundaries window open.

Cancel — Cancels the export of the model.

Working Directory

VERICUT Users:

VERICUT Location: **File menu > Working Directory**

Toolbar short cut for setting a working directory: 

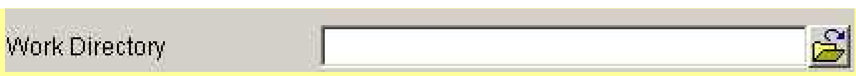
Mold and Die Users:

Mold and Die Location: **Setup page > Work Directory**

Notebook Feature: 

Cutter Grinder Users:

Cutter Grinder Location: **Setup page > Work Directory**

Notebook Feature: 

Opens a window enabling you to set the working directory where you can open and save VERICUT files. After setting the working directory, VERICUT will look here by default whenever files are saved or opened. You can also quickly access files in this directory via the Working Directory entry in the file selection window Shortcut list.

Most features on this window are standard file selection window features that enable you to navigate through directories, filter files, and type or select file names. In this case, all that is required is to select a directory.

Properties window

VERICUT Users:

VERICUT Location: **File menu > Properties**

Mold and Die Users:

Mold and Die Location: **Other Settings page > Session Properties**

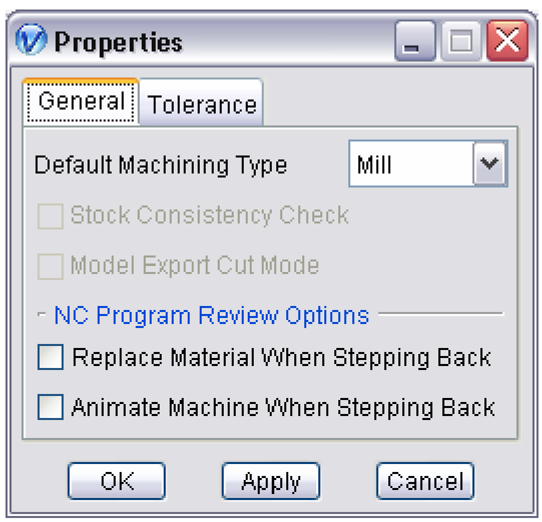
Notebook Feature: 

Cutter Grinder Users:

Cutter Grinder Location: **Other Settings page > Session Properties**

Notebook Feature: 

Opens a window to set properties for the verification session, such as: cut mode, resolution and tolerance values that affect cut model display, accuracy, and motion simulation.



General tab — Features on this tab establish general properties for the VERICUT simulation session.

Tolerance tab — Features on this tab set tolerances for the overall simulation session. These values are used by VERICUT to construct the solid database for machining, simulate circle motions, identify errors related to material removal, and work with conic solid models.

OK — Saves the window settings and closes the Properties window.

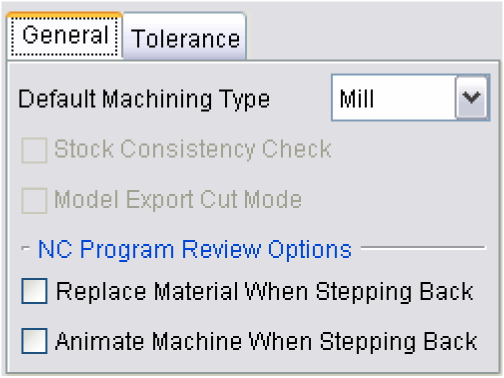
Apply — Saves the current window settings and leaves the Properties window open.

Cancel — Closes the Properties window without saving changes.

Properties window, General tab

Location: **File menu > Properties**

Features on this tab establish general properties for the VERICUT simulation session.



Default Machining Type — Sets the machining mode for VERICUT.

Options:

Mill — A spinning tool removes material when in contact with Stock/Fixture.

Turn — A Spindle spins the Stock/Fixture assembly and a stationary tool (not spinning) removes material when in contact with Stock/Fixture.

Wire EDM — An electrically charged wire removes material when in contact with Stock/Fixture.

The machining type can be changed during the simulation using a VERICUT-MODAL record (APT or G-Code tool paths) or via the **VC_ModeMilling/VC_ModeTurning NC** control macros (G-Code tool paths only).

Stock Consistency Check — When selected, VERICUT checks the consistency of stock models, including: check for "watertight" solid- repair improperly trimmed surfaces (overlaps and gaps), and reconstruct insignificant missing surfaces. This option is highly recommended for stock models creating from importing IGES data, and other model files where portions of the stock model disappear, or gaps/seams appear in the model when cutting is started.

Model Export Cut Mode — When toggled "**On**", causes VERICUT to cut with a more accurate internal cutter representation for better results when using modules like Model Export or OptiPath. Only toggle "**On**" when the additional accuracy is required as it can have an impact on processing speed.

NC Program Review Options
The following options only apply to NC Program review and what is displayed as you step back through a toolpath.

Replace Material When Stepping Back — When toggled "**On**", removed material will be replaced as you step back through the toolpath.

NOTE: **Replace Material When Stepping Back** and **FastMill** mode (ref, **FastMill**, on the Motion window, in the Project menu section, also in the *VERICUT Help* section) are mutually exclusive.

When **Replace Material When Stepping Back** is toggled "On", VERICUT will turn **FastMill** "Off" Conversely, if **FastMill** is toggled "On", VERICUT will turn **Replace Material When Stepping Back** "Off. VERICUT will display a pop-up notification when these situations occur.

Animate Machine When Stepping Back — When toggled "On", the machine will animate as you step back through the toolpath.

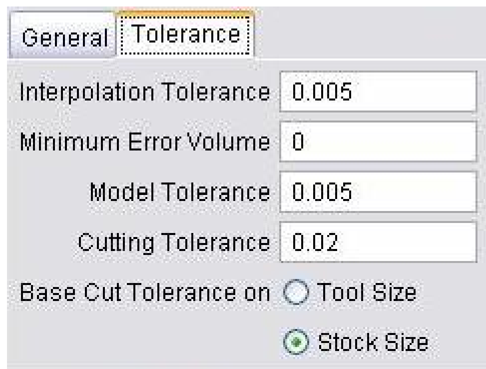
NOTE: These two options must be set ***before*** simulating the toolpath in VERICUT to have the feature available in NC Program Review.

To Properties window

Properties window, Tolerance tab

Location: **File menu > Properties**

Features on this tab set tolerances for the simulation. The tolerance values control the accuracy of the cut stock model, simulated circular motion and other interpolated motions, whether or not certain errors get written to the error log, and the accuracy of cylinders, cones and other revolved models and tool assemblies for collision checking.



Interpolation Tolerance — This value is used to interpolate intermediate positions during NURBS, circular, and helical motions. This tolerance is also used by the Detect 4-Axis Rotary Motion APT setting to determine if sequential GOTO locations can be performed as a single 4-axis motion- ref. **Project menu > Processing Options > APT Settings: Rotary tab**.

Minimum Error Volume — This value sets a threshold for the minimum volume of material that must be removed by a fast feed motion or holder collision in order for an error to be reported. If the volume removed by an error motion is less than this value then no error is reported in the log file or logger for the motion and the error count is not incremented.

However, any volume of material removed by a fast feed motion or holder is always shaded in the error color (typically red), regardless of the volume. Thus small volume errors may be shaded red but will not be reported in the log file or logger if the volume is less than Minimum Error Volume.

The Minimum Error Volume value is in cubic units (inch or millimeter). Use this setting to filter very small fast feed and holder collisions from VERICUT's error reports. The default value "0" causes VERICUT to report all fast feed and holder/stock collision errors.

Model Tolerance — This value is used when displaying revolved models such as cylinders, cones, and revolved profiles. It is also used when displaying the revolved image of a turning stock model created from an IGES profile. Accuracy of cylindrical tool assemblies displayed in a machine view are also affected by this value.

This value has a small effect on collision checking accuracy for revolved models used to represent machine components or fixtures.

Cutting Tolerance — This value sets the accuracy of VERICUT's cut stock model. The Base Cut Tolerance on choices below determine how the value is used to establish the cut stock accuracy. Increasing the Cutting Tolerance value increases the speed of the simulation and various other operations, and uses less memory, but reduces the accuracy of the cut stock model. Decreasing the value improves accuracy of the cut stock model, but reduces speed and increases the memory required by the cut stock model.

The cutting tolerance affects the accuracy and speed of AUTO-DIFF, Model Export, and OptiPath. A smaller Cutting Tolerance produces more accurate results but takes longer to process. VERICUT's ability to detect small holder collisions and fast feed errors are also affected. The speed of dynamic rotate/pan/zoom operations and the time required to create the refined image in the workpiece view are also affected.

NOTE: This feature is only available when the first setup in a project is active. For subsequent setups, this feature is grayed out.

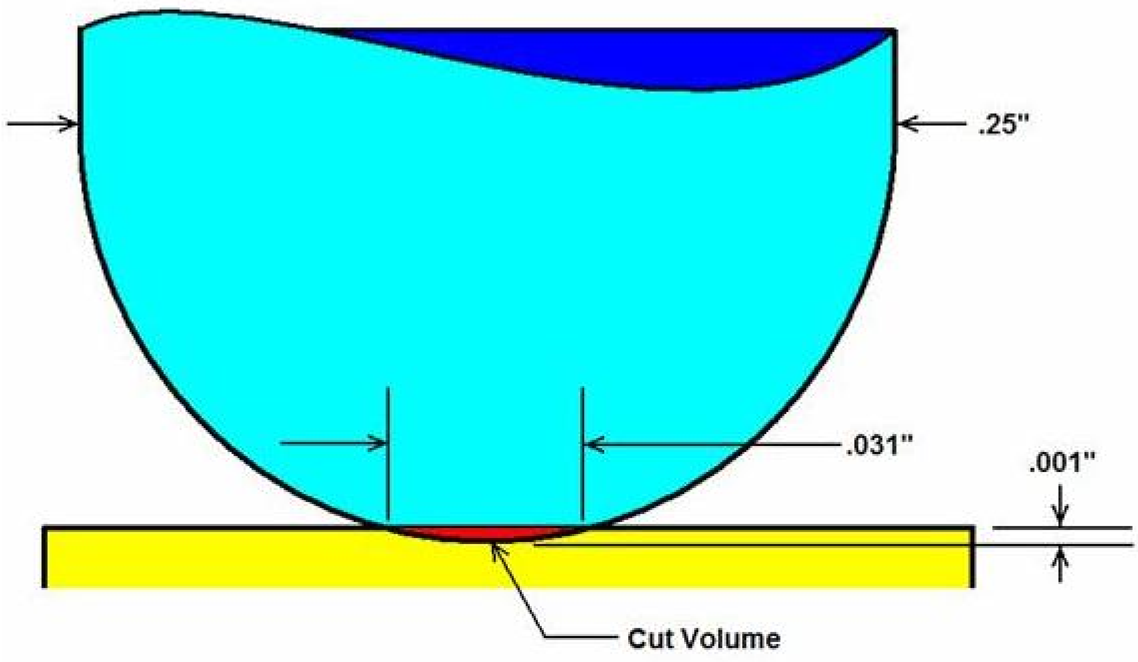
Base Cut Tolerance on — Controls how VERICUT applies the Cutting Tolerance to set the accuracy of the cut stock model.

Tool Size — When selected, VERICUT scans the current tool path files and analyzes the geometry of all cutters used by the NC programs. The Cutting Tolerance value is applied as a chordal deviation on curved tool shapes (diameter, corner radius, etc). The smallest calculated chord length or cutter profile segment is then used to set the accuracy of VERICUT's cut stock model in order to reliably create cut features with the tools used by the current NC programs. This option is recommended when faster simulation speed is important. This is the default setting.

Stock Size — When selected, VERICUT applies the Cutting Tolerance value directly, creating the cut stock model at the specified accuracy. The tool path files are not scanned to detect the cutters used. The value represents the size of the smallest cut feature VERICUT can detect.

NOTE: It is very important to understand that the value represents the size of a cut's volume measured in any direction. Thus, cuts that are larger than the Cutting Tolerance in at least one direction will be detected by VERICUT.

For example, a .001" axial depth cut by a .25" ball end mill produces a volume with the following shape:



As a general practice, setting Cut Tolerance Based on Stock Size to one-fourth (1/4) the size of a cut volume's span will detect the cut. Thus in this case, to detect a .001" deep cut by a .25" ball end-mill, set Cut Tolerance to .007" or smaller. However, results can vary depending on the shape of the cut volume.

The Stock Size option is recommended when an exact value is desired for VERICUT's cut stock model accuracy, independent of the tools used.

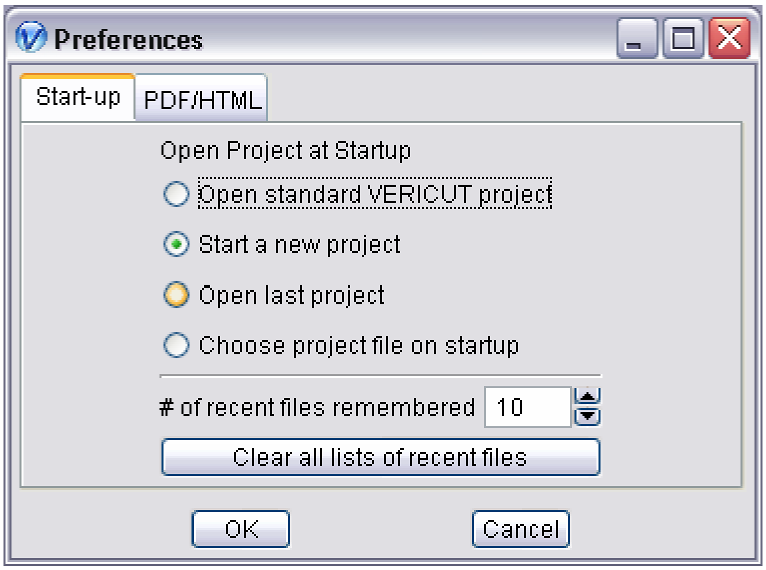
NOTE: This feature is only available when the first setup in a project is active. For subsequent setups, this feature is grayed out.

To Properties window

Preferences window

Location: **File menu > Preferences**

Opens a window enabling you to specify the VERICUT project file (.VcProject) that you want displayed when you start the next VERICUT session, the number of "recent files" that you want VERICUT to remember, and the PDF reader/web browser that you want VERICUT to use for online Help.



Start-up tab — Use the features on this tab to specify the VERICUT project file (.VcProject) that you want displayed when you start the next VERICUT session, the number of "recent files" that you want VERICUT to remember.

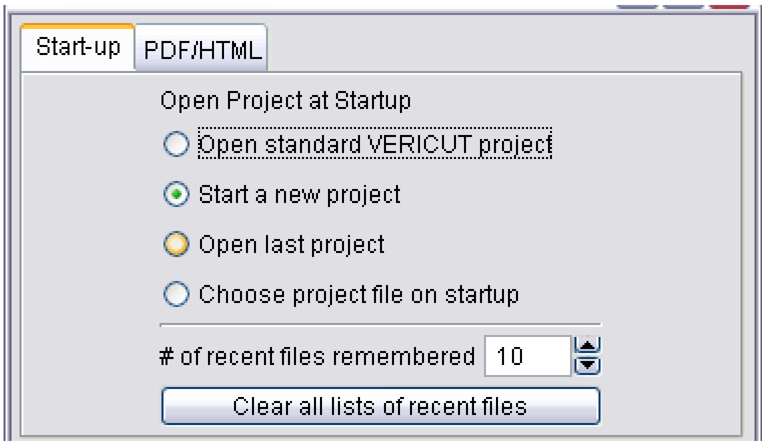
PDF/HTML tab — Use the features in this tab to specify the Adobe Reader and/or Internet Browser that you want VERICUT to use to display online Help.

OK — Saves the window settings and closes the Preferences window.

Cancel — Closes the Preferences window without saving the window settings.

Preferences window, Start-up tab

Use the features on this tab to specify the VERICUT project file (.VcProject) that you want displayed when you start the next VERICUT session, the number of "recent files" that you want VERICUT to remember.



Open standard VERICUT project — When toggled "On", the standard VERICUT project is displayed upon startup. The standard VERICUT project is either vericut.VcProject, or vericutm.VcProject, in the "library'" directory of your VERICUT installation depending on the way environment variable, **CGTECH_DEFAULT_UNITS**, is set.

Start a new project — When toggled "On", the VERICUT session opens ready to start a new project.

Open last project — When toggled "On", the new VERICUT session opens with the same project file that was open when you closed the previous VERICUT session.

Choose project file on startup — When Toggled "On", VERICUT displays the Open Project window at startup enabling you to select a project file.

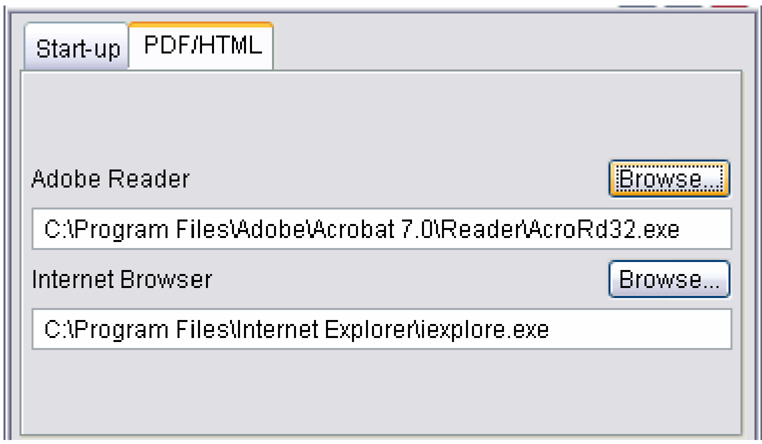
# of recent files remembered — Use this feature to specify the number of "Recent Files" that you want VERICUT to remember and display.

Clear all lists of recent files — Use this feature to clear ***all*** "Recent Files" lists. This includes the "Recent Files" lists in the File menu in the main VERICUT window, Report Template window, and in the Inspection Sequence window.

To Preferences window

Preferences window, PDF/HTML tab

Use the features in this tab to specify the Adobe Reader and/or Internet Browser that you want VERICUT to use to display online Help.



Adobe Reader — Enter the *path*/*file name* in the **Adobe Reader** text field, or use the **Browse** button and use the file selection window that displays, to specify the Adobe Reader that VERICUT is to use to display online Help.

NOTE: This feature is only available on Windows machines. On UNIX machines, the Adobe Reader will be accessed through the internet browser.

Internet Browser — Enter the *path*/*file name* in the **Internet Browser** text field, or use the **Browse** button and use the file selection window that displays, to specify the internet browser that VERICUT is to use to display online Help.

Both of the above settings are stored in the Preferences File. For more information on the Preferences File, see **Summary of VERICUT File Types**, in the **Getting Started with VERICUT** section.

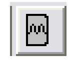
To Preferences window

Convert

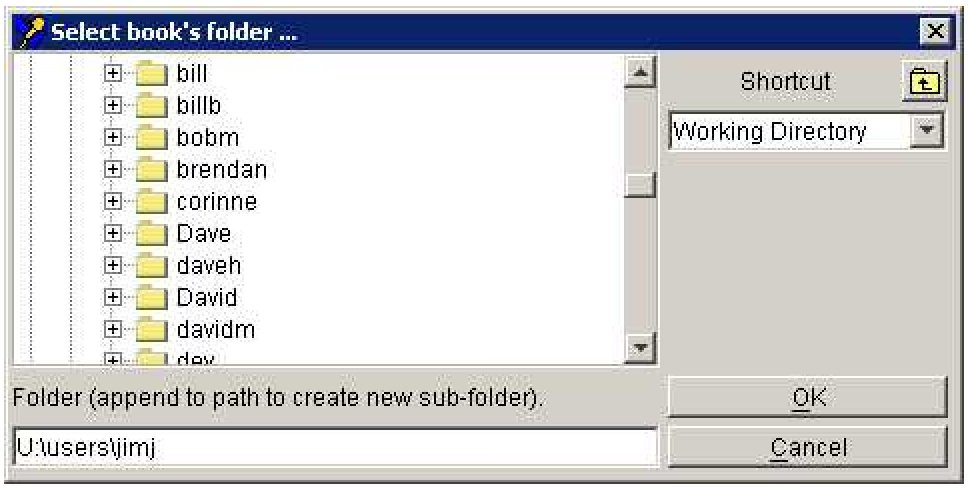
See Converters and CAD/Cam Interfaces in the *CGTech Help Library*.

Custom Interface

Location: **File menu > Custom Interface**

Toolbar short cut for accessing Custom Interface: 

Opens a file selection window that enables you to select an existing custom VERICUT User Interface for use; or for editing an existing, or creating a new custom interface with VERICUT Customizer.



Folder "tree"— Enables you to specify the location for new custom interface folder or select an existing custom interface folder. All files related to the notebook and custom interface are stored in this folder.

Shortcut — Displays the Shortcut option list providing quick access to files in your Working Directory, or CGTech library and sample files. Use the "Folder Up" icon to move to the next higher level in the Folder "tree".

Folder — Use this text field to enter a /path/folder name to open an existing custom interface notebook, or to create a new custom interface folder and open a new "empty" custom interface notebook. You can also use the Folder "tree" to specify the path then append the name for the new custom interface folder in the text field,.

OK — Opens an existing custom interface notebook or creates the new custom interface folder and displays new "empty" custom interface notebook.

Cancel — Ends book selection and returns to VERICUT.

Introduction to VERICUT Customizer

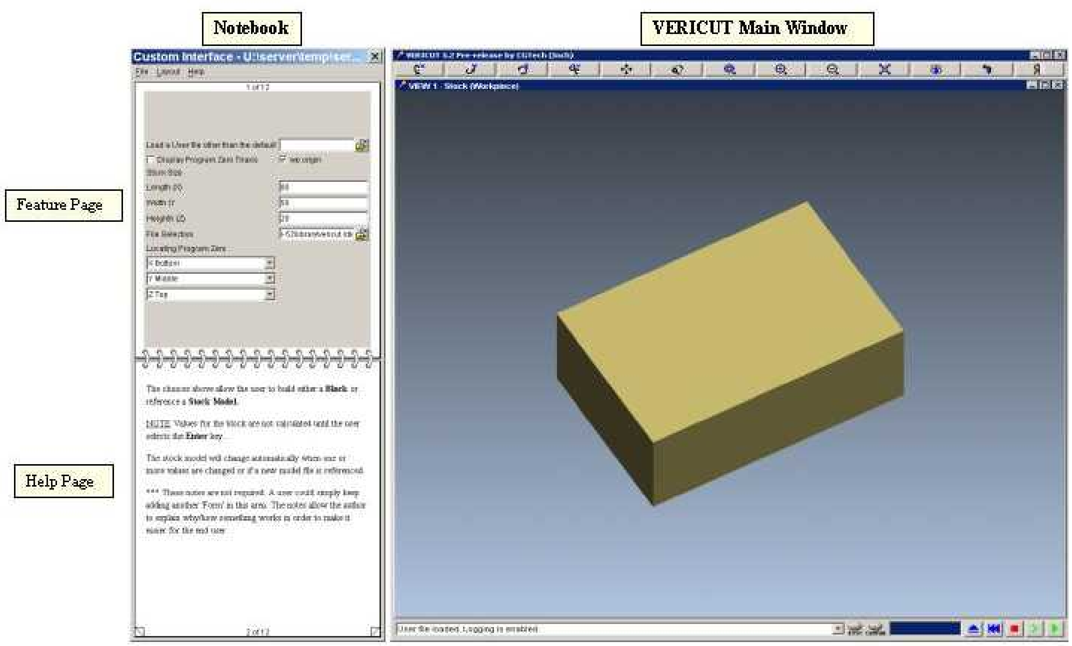
The VERICUT Customizer, or "Customizer", is a standard VERICUT feature that provides the tools to create a custom simplified VERICUT User Interface. VERICUT Customizer is designed to simplify VERICUT usage for specific jobs or classes of jobs.

Include only the VERICUT functionality that you need in a custom interface and eliminate the need to search through the wide range of features that are presented in VERICUT's "full function" standard user interface to find the ones they need to use.

VERICUT Customizer:

- Enables you to graphically arrange and label buttons, choice lists, check boxes, etc. in a way that is most meaningful in your particular work environment.
- Enables you to include instructions and pictures within the interface to prompt or explain to the user how to use VERICUT to accomplish their job.
- Enables you to produce a simplified user interface that will make "new" or "occasional" VERICUT users productive with minimal training.

What does a custom VERICUT interface look like?



A custom VERICUT interface consists of two main parts, the "notebook" and the VERICUT main window.

Notebook

This is a simplified VERICUT user interface. It has been designed for a particular type or class jobs. It contains only the functionality essential to the particular job type without the complexity of the standard "full functionality" VERICUT user interface. A notebook can be created with both "feature" and "help" pages.

Feature Pages -

Contain features (buttons, text fields, choice lists, etc.) that enable you to pass necessary information to VERICUT for processing. Each of the features selected for use on the feature pages can be labeled in the language and/or terminology that is most appropriate for the intended end user. Feature pages use XML files to store feature information.

Help Pages -

Used to include instructions, explanations and pictures within the custom interface to assist the user in providing the necessary information to VERICUT. Help pages are HTML files and GIF or JPEG images to illustrate the instructions.

VERICUT Main Window

The VERICUT main window in a custom user interface is similar to the standard VERICUT main window without the main menu bar and a lot less icons. This makes using VERICUT much easier to use for those who do not need the full range of functionality provided by the standard VERICUT user interface.

What is involved with creating a custom VERICUT interface?

Creating and deploying a custom interface for use by VERICUT users involves the following seven steps. Each of the steps listed below is described in the following section.

1. Set up a default user file.
2. Creating the notebook.
3. Determining the size of the notebook.
4. Adding custom interface features.
5. Adding custom interface help.
6. Test the custom interface.
7. Make the custom interface available for use.

Setting up the Custom Interface Default Project File

Each customized interface notepad references a default project file. This project file is retrieved by VERICUT each time the notebook is opened. It is advisable to preset as many parameters as possible in the default project file. Doing this will minimize the number of features that will be required in the customized interface notebook and make it much simpler to use.

It cannot be stressed too much that the key to creating a concise, easy to use custom interface is attention to detail in the design of the default project file. While there are seven steps involved in creating a custom interface, it is estimated that as much time will be spent on this first step as the other six put together. If you get the default user file right, the remaining steps to create the custom interface will be much easier and the resulting notebook will be much simpler to use.

The following topics should be considered when setting up a default user file for use with a specific notebook:

- Use of colors
- Cut mode (Standard or FastMill)
- Units
- Tolerances
- Conditions for AutoSave
- Layout, orientation and attributes of Views
- Configuration of status window
- Report, log and result file names
- Machine and fixture models
- Toolpath type
- Tool change technique
- Motion, G-Code and APT settings
- Location of stock model within stock component
- Tool and OptiPath library
- Control parameters
- Machine parameters

Many of the parameters associated with the above topics can be set using features displayed on the feature pages of the notebook, but if they are not going to change, there is no reason to include them in the notebook. Just set them in the default project file.

After all, the purpose of the custom interface is to simplify the use of VERICUT.

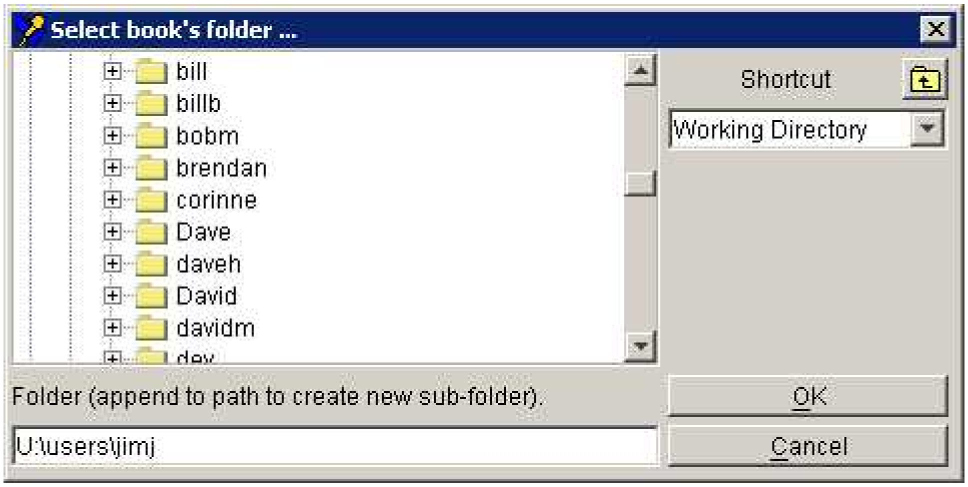
In many instances, creating multiple notebooks that reference different default user files can simplify the use of a custom interface. For example, if you are creating a notebook to lead a machine operator through processing a toolpath in VERICUT before running it on the machine, it could be beneficial to create a notebook and corresponding default user file for each machine/controller combination on the shop floor. That way the operator does not have to enter machine/controller information every time. Just call up the appropriate notebook enter the toolpath information, then process.

Notebooks should be created in such a way that the user is can only save the user file with their input under a different name. Default project files should also have permissions set as "read only" and stored in "write protected" directories and folders so they do not accidentally get modified.

Now that you have tailored a project file for the notebook you intend to create, it's time to create the notebook.

Creating the Custom Interface Notebook

Creating a custom interface notebook:
Select **File > Custom Interface** in the VERICUT main menu to display the Select book's folder window shown below.



Folder "tree"— Enables you to specify the location for the custom interface folder. All files related to the notebook and custom interface are stored in this folder.

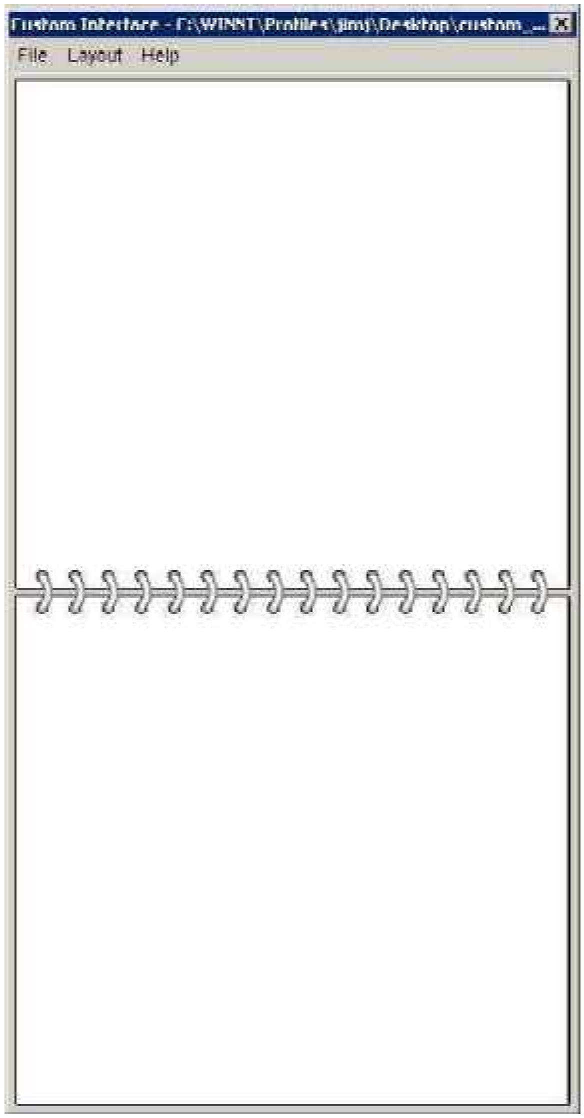
Shortcut — Displays the Shortcut option list providing quick access to files in your Working Directory, or CGTech library and sample files. Use the "Folder Up" icon to move to the next higher level in the Folder "tree".

Folder — Use this text field to enter a /path/folder name, or use the Folder "tree" to specify the path and append the name in the text field, for the new custom interface folder.

OK — Creates the folder and displays the new custom interface notebook.

Cancel — Ends book selection and returns to VERICUT.

The notebook:



Notice that when the notebook is created it contains no pages. Adding pages will be discussed later. The custom interface notebook header shows the folder that the notebook is associated with. The main menu bar contains the following options:

File:

Open — Displays the Select book's folder window allowing you to open another custom interface notebook for editing or create a new notebook.

Recent — Displays a list of recently opened files. To open a file in the list, select the desired file.

Close Book — Closes the VERICUT custom interface and displays the standard VERICUT interface.

Exit VERICUT — Ends the VERICUT session.

Layout:

 — Positions the notebook on the left side of the VERICUT main window.

 — Positions the notebook on the right side of the VERICUT main window.

 — Overlays the notebook on top of the VERICUT main window. Left click on the notebook's header and drag to any position.

Minimize — Reduces the notebook size to only display the custom Interface header and main menu bar.

One Page — Use to display the custom interface notebook as a single page.

Two Pages — Use to display the custom interface notebook as two pages.

Help:

On VERICUT — Starts VERICUT Help.

About VERICUT — Opens a window that displays information about the current VERICUT session.

Notebook modes:
User mode:

When a notebook is in "user" mode it can only be used to interact with VERICUT. It cannot be modified or edited in any way. When a notebook is opened, it is always in "user" mode, even when it is new and empty. If password protection for "author" mode is in effect, the following window will be displayed when the F7 key is used to enter "author" mode:

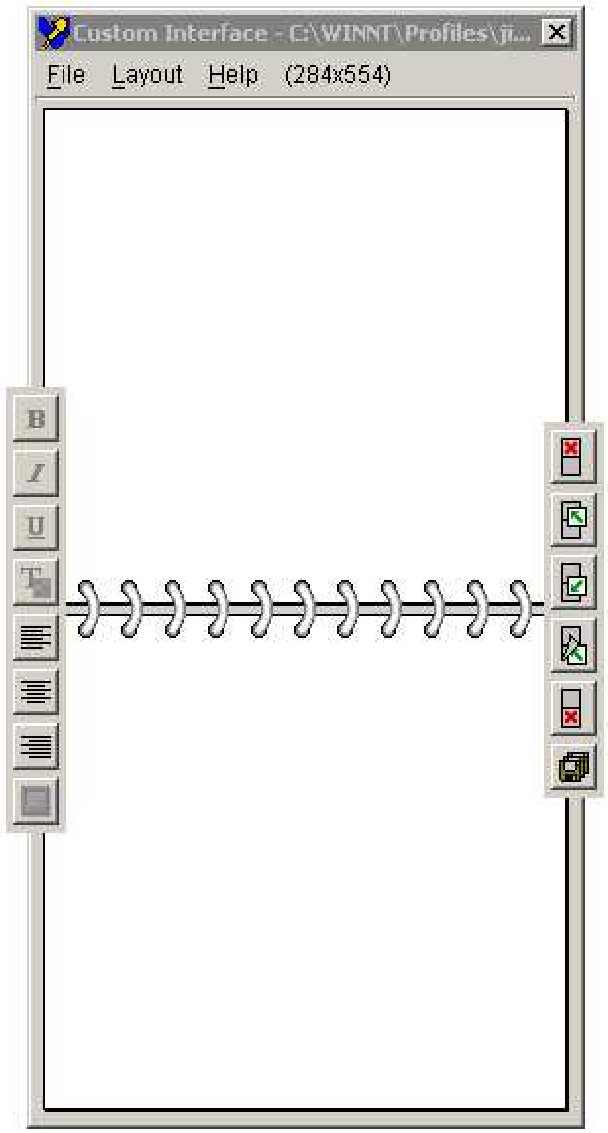
Password — Enter the correct password in the text field and select Unlock to allow entry to "author" mode.

Unlock — Use with password above, to allow entry to "author" mode.

Cancel — Use to dismiss the access window.

Author mode:

When a custom interface notebook is in "author" mode, pages and features can be added or removed, text and labels can be edited and the size of the notebook can be changed. Use the F7 key on the keyboard to switch between "user" and "author" modes. When you are in "author" mode, the notebook changes as seen in the illustration below.



Notice that "(**284x554**)" has been added to the end of the menu bar. This number represents the current size of the notebook in pixels. While you are in "author" mode you can change the size of the notebook. Click on one of the corners or edges and drag to change the notebook size like any other window. The size of the notebook that is set in "author" mode is what the user will have to work with. The size of the notebook is fixed for the user. The position of the notebook can be changed by the user but not the size. See **Sizing the Custom Interface Notebook** below**,** for more information on determining the appropriate size for the notebook.

Notice also that a row of icons has been added to the sides of the notebook. The icons on the right are used to add or remove pages and save the notebook. The icons on the left are used to edit the text on notebook help pages. They remain grayed out and inactive until help pages are added to the notebook.

Finally, the following options have been added to the File menu:

User File — Enables you to specify the default .usr file that is associated with the custom interface notebook.

Protect — This item enables the author to prevent the user from entering "author" mode without entering a password. Selecting Protect displays the Protection window which enables you to specify whether or not to use password protection for the notebook while in "user" mode and to specify a password.

Password — Enter a password in the text field that will allow entry to "author" mode.

Lock — Use to turn on password protection for "author" mode.

Unlock — Use to turn off password protection for "author" mode.

Cancel — Use to exit the Protection window without making changes.

To add password protection for entering "author" mode:

1. Type a password in the text field
2. Select the **Lock** button.
3. Save the notebook using the Save all changes icon.

The next time that the notebook is opened, if the user uses the **F7** key to enter "author" mode, they will be prompted to enter the password before being allowed into "author" mode.

Use the Unlock button while in "author" mode to remove an existing password protection.

Allow Close — This item enables the author to prevent the user from exiting the custom VERICUT interface and returning to the standard VERICUT interface. This item toggles "on" and "off".

If Allow Close is toggled "on" (indicated by a checkmark), the Close Book option will appear in the File menu when in "user" mode allowing the user to exit the custom VERICUT user interface and return to the standard VERICUT user interface.

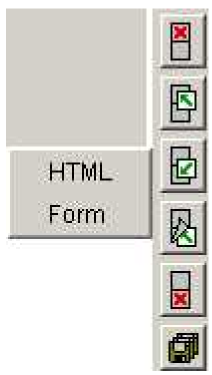
If Allow Close is toggled "off" (indicated by no checkmark), the Close Book option will not appear in the File menu when in "user" mode preventing the user from exiting the custom VERICUT user interface and returning to the standard VERICUT user interface.

Adding pages to the notebook:

Before you can add features or help text to the notebook you need to add pages. Add any combination of feature pages and help pages. Keep in mind that two pages are displayed at once. It makes a lot of sense to have alternate feature pages and help pages. One advantage of this user interface style is that it can lead the user through a specific sequence of steps.

Use the icons illustrated below to add pages to or remove pages from the notebook.

NOTE: If you do not see the icons below on the right side of the notepad, you are not in "author" mode. Use the **F7** key on the keyboard to switch to "author" mode.



Remove upper page — Use to remove the displayed upper page from the notebook.

Insert upper page — Use to insert a page into the displayed upper position of the notebook. Select HTML or Form from the pop-up menu to specify the type of page to create.

HTML — Use to create a help page.

Hint: You can tell a help page by its white background.

Form — Use to create a feature page.

Hint: You can tell a feature page by its gray background.

Insert lower page — Use to insert a page into the displayed lower position of the notebook. Select HTML or Form (described above) from the pop-up menu to specify the type of page to create.

Append page — Use to add a page to the end of the notebook. Select HTML or Form (described above) from the pop-up menu to specify the type of page to create.

Remove lower page — Use to remove the displayed lower page from the notebook.

Save all changes — Use to save all changes made to the notebook. This includes notebook size changes, pages added or removed, and page content added or edited.

NOTE: There is no "undo" capability in "author" mode so saving often is highly recommended.

Once you have added some pages, use the following to move around in the notebook.

Click on one of the "dog-eared" corners in the upper page (or use the Page Up key on the keyboard) to turn back one page in the notebook.

Click on the one of the "dog-eared" corners in the lower page (or use the Page Down key on the keyboard) to turn forward one page in the notebook.

Use the Ctrl and Home keys on the keyboard at the same time to move directly to the first page of the book.

Use the Ctrl and End keys on the keyboard at the same time to move directly to the last page of the book.

Now that you have created the notebook, it's time to decide on the size of the notebook.

Sizing the Custom Interface Notebook

A customized interface notebook retains the size set while in "author" mode and cannot be changed by user. This is because information on one page does not reposition to the next page if the size of the notebook is reduced, nor are scroll bars used if a page contains more information than will fit. To do so would be contrary to the idea of a simplified user interface.

You need to decide on the size of the notebook, which in turn determines the size of its pages, before you start adding content. The size of the notebook can be changed later, but if you make it smaller you may be spending a lot of time cutting and pasting between pages.

To help determine the appropriate size for the notebook, try to answer the following questions:

1. What is the resolution of my target user's screen?
2. How large does the VERICUT main window need to be?
3. How much space to the left or right of the VERICUT main window remains for the notebook? Positioning the notebook on either of the main window is clearly preferable.

   If the answer to this last one is "Not enough!", is it reasonable to overlay the notebook on top of the VERICUT main window?

As a starting point, you might try allocating 3/8ths of the screen's width for the notebook. You can adjust the notebook size by dragging its edges or corners as you would any other window. The numbers on the right side of the notebook's menu bar will continuously update as you re-size the notebook. If your target screen resolution is 1280 x 1024, and you have decided to allocate 3/8ths of the width to the notepad, you could manipulate the window until you see (480 x 990), leaving some space for the Windows taskbar. If your target user has a Windows based platform, you do not want to use screen's full height or the notebook will be partially hidden behind the Windows taskbar.

Use the Save all changes icon () located on the right side of the notebook to save the notebook with its size and position.

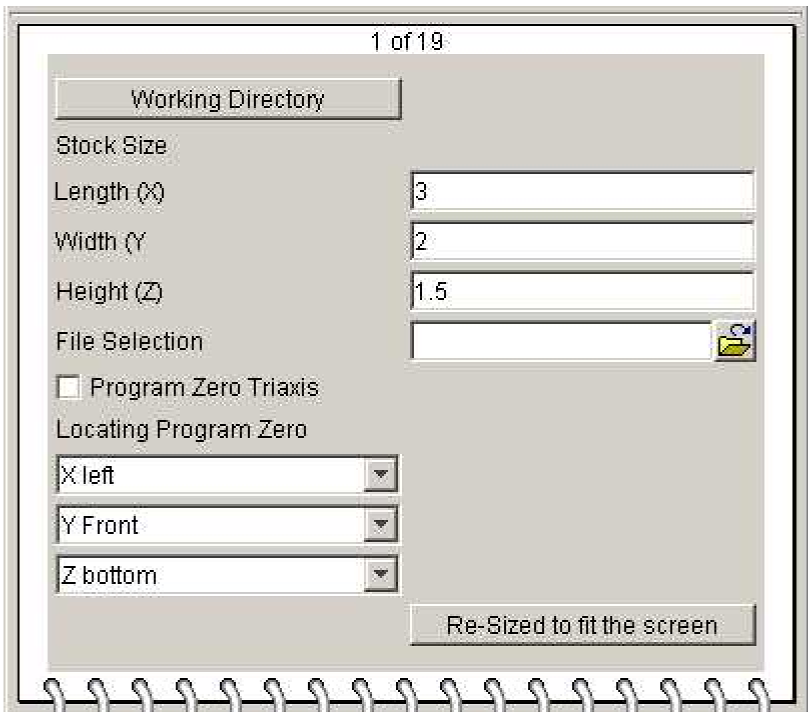
Now that you have established the size of the notebook, and thus the size of its pages, you can add features to the pages.

Creating Custom Interface Feature Pages

Now that you have created and sized the notebook, it is time to add features to the feature pages. The user interacts with features in the notebook to enter numbers, select files, make choices from a limited set of options and see results. As the author of a custom user interface notebook it is your task to arrange a collection of features so the user can specify all the parameters needed to get his or her job done. The way that you lay out the features within the notebook, and use help pages, enables you to lead the user through a logical sequence of actions to accomplish the required task.

If your notebook does not yet contain feature pages, or you need additional ones, use the procedure described in **Adding Pages to the Notebook** to add them.

Feature Page Layout:



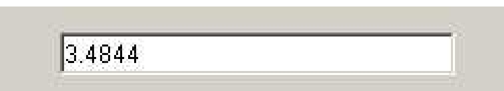
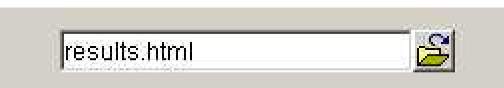
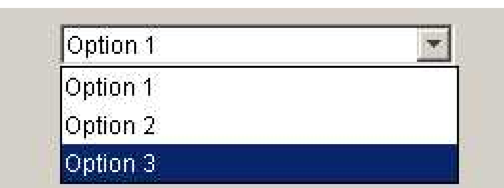
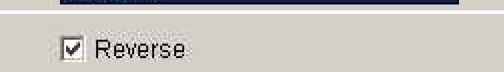
A "feature" page is easily distinguished from a "help" page by its gray background. Features can only be added to "feature" pages. They cannot be added to "help" pages. In order to add features to the page you must be in "author" mode.

Within a page, features are laid out in two columns. The left column will only be as wide as the widest feature that it contains. All features placed in the right column will expand to fill the remaining width. Use blank spaces within features in the left column to control sizing of the two columns. Adding blank spaces to the longest feature in the left column will increase the width of the left column, thereby decreasing the width of the right column. Add or remove blank spaces to the longest feature in the left column until the desired column balance is achieved.

Typically the left column will contain labels explaining the purpose of the corresponding feature in the right column, although you are not restricted to this convention. The first feature that you place in a form will always go to the left column.

What Features Are Available?

The following feature types are available for use in any combination on the feature pages of your notebook.

FEATURE	DESCRIPTION
	Labels the purpose of adjacent features.
	Text Fields are used to enable the user to input character information to, or receive results from VERICUT.
	Numeric Fields are used to input one or more numeric values. Can also be used to input numeric data requiring calculations like "3+31/64".
	File Selection Boxes are used to input file names or browse for the file using the icon.
	Choice Lists are used to present a limited set of options in a drop-down list
	Check Boxes are used for options requiring only a simple on/off condition.
	Radio Buttons are used as an alternate method of presenting a limited set of options. There can only be one set of radio buttons used on each feature page. Only one of the radio buttons will be active at a time. When one is selected, all others in the set become deactivated.
	Buttons are used to trigger actions like file conversion or display a secondary window.
	MDI Buttons are used to input machine control functions.

MDI Numeric Fields are used to input numeric data to machine control functions.

Adding Features:

While in "author" mode (use F7 key on your keyboard if necessary if necessary), move the cursor to an unoccupied area of the page and click the right mouse button. You will see a pop-up menu like the one shown on the left, enabling you to select one of the various feature types described above. As each feature is added to the page you will notice that initially they contain question marks "???" for text. Type over the "???" with the desired text. You can edit the text of an existing feature by clicking in it and making the changes. Remember that features on the page are positioned in just two columns. The first feature is added in the left column. Initially it appears in the center of the page. As soon as you add another feature to the right of it, it repositions the first one from the center to the left edge of the page.
Continue adding the desired features to the page, or until the page is full. At that point, add another feature page and continue. Note that you can insert a feature between two existing rows by carefully positioning the cursor between them.

You may have noticed that the pop-up menu of feature types has another choice, Paste, that we have not discussed yet. If you right-click on one of the features that you have already put on the page, you will see the pop-up menu shown on the left. Select either the Cut or Copy options, then right-click on an empty area of the page. The feature type pop-up menu discussed above will display. Selecting the Paste enables you to move or copy the feature. This is also the way that you can move or copy features to another page if necessary. Delete enables you to remove an existing feature from the page.

Connecting the Features to VERICUT:

To this point the features that you have added to your notebook may look great but they are not yet functional. They need to be connected to the parameters and functions of VERICUT so that they produce the desired result when put to use by the user. These connections to VERICUT are accomplished by using the Details option that appears when you right-click on a feature. This is the same pop-up menu that contained the Cut and Copy options discussed above.

The interaction that follows is dependent on the feature type, although some feature types require similar interaction. Each of the topics listed below is described in detail on the following sections.

- Labels and Text Fields
- Numeric Fields
- File Selection
- Choice Lists and Radio Buttons
- Check Boxes
- Buttons
- MDI Buttons and Numeric Fields

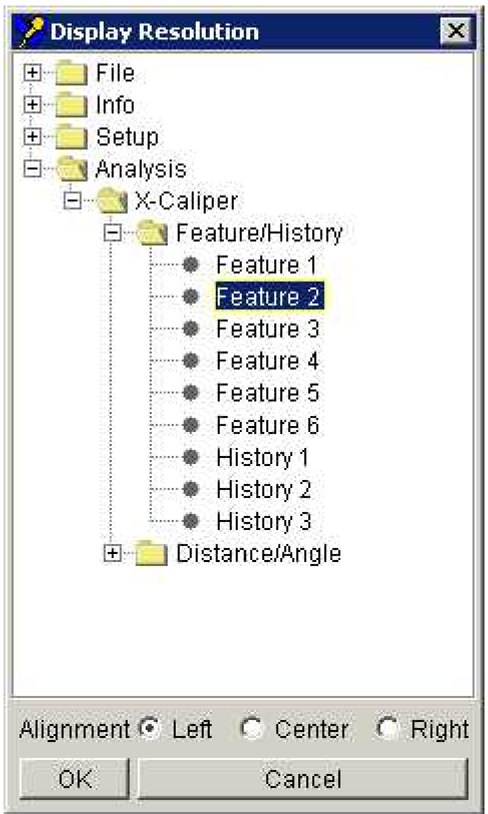
Once you have finished adding functional feature pages to the notebook, it's time to add some help pages.

Custom Interface - Labels and Text Fields

While in "author" mode, right-clicking on either a label or a text field on a feature page and selecting Details, displays essentially the same window. The only difference is that the one for label features has a set of radio buttons that enable you to adjust its alignment. The tree that dominates these windows is identical. It is organized to reflect VERICUT's menu structure. Each "leaf" in the tree represents a text parameter. When selecting between using a label or a text field, keep in mind that a user can only read a label, but can edit the content of a text field. Most label features in a notebook will not need to be associated with a VERICUT parameter. They will simply be used to describe other features on the feature page.

To connect a feature to a parameter, simply expand the tree till you find the parameter you are looking for, select it and click OK. You can also use these windows to check on an existing association, in which case you would use the Cancel button afterwards.

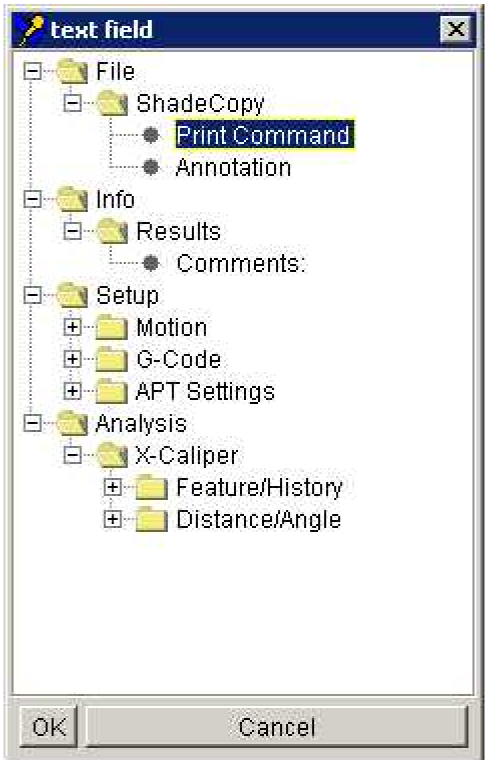
Label Feature Window



Label features can be used to display the text output of certain VERICUT parameters. The label feature window to the left illustrates how a label can be used for this purpose. The label is associated with the second line on X-Caliper's Feature/History tab, which presents the coordinates of a selected geometry feature's datum (such as a sphere's center). When the user selects a geometry feature in the VERICUT graphics area, the text of the label will change to display the resulting text, just as it would be displayed in the X-Caliper window.

It is also possible to associate a text field feature with such a parameter, but it is probably not a good idea to do so unless the user will need to copy and paste the text to another application.

Text Feature Window



The text feature window to the left illustrates how a text field feature can be associated with the operating system's print command for use by VERICUT's ShadeCopy function.

Custom Interface - Label/Text Field Parameter List

File	Shade Copy	**Print Command**	
		Annotation	
Setup	Motion	**Start At**	
		Stop At	
	G-Code	**Block Skip Character**	
	APT Settings	**Rotation Center**	
		Goto Format	
Analysis	X-Caliper	Feature/History	**Feature 1**
			Feature 2
			Feature 3
			Feature 4
			Feature 5
			Feature 6
			History 1
			History 2
			History 3

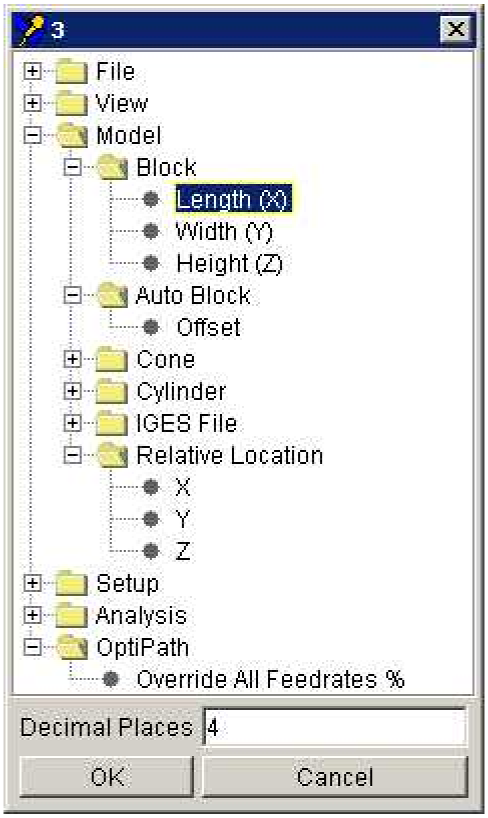
Custom Interface - Numeric Fields

While in "author" mode, right-clicking on a numeric field on a feature page and selecting Details, displays the window illustrated below. The title of the window is the current text from the numeric field. Note that you can also specify the maximum number of decimal places used to display the parameter. For an integer, such as the number of cuts between automatically saved ShadeCopy images, use zero decimal places.

The tree that dominates the window is organized to reflect VERICUT's menu structure. Each leaf in the tree corresponds to a numeric parameter that appears in one or more of VERICUTS's standard user interface windows. Each of them can be placed on a feature page independently, though some will almost always appear in sets. A good example is the direction of the light that illuminates each view, represented by an (I, J, K) vector. The three parameters that control the location of a stock model relative to the stock component's datum are another, but they can be combined with three choice lists to provide some extra functionality that is not currently available in the standard VERICUT user interface.

To connect a feature to a parameter, simply expand the tree until you find the parameter you are looking for, select it and click OK. You can also use these windows to check existing associations, in which case you would use the Cancel button afterwards.

Numeric Field Feature Window



Defining the three dimensions of a stock block is another good example.

In the numeric field feature window displayed on the left, a numeric field feature is associated with the Length (X) dimension of a stock model block. When combined on the feature page with numeric fields associated with the Width (Y) and Height (Z) values, the user can quickly and easily define a stock block

Custom Interface - Numeric Field Parameter List

File	Convert	IGES		**Tolerance**
		Surface		**Tolerance** **Offset** **Base Location**
		VDAFS		**Tolerance**
		Binary CL		**Minimum Cutter Height**
	Model Export	**Grid Count** **Global Angle** **Tolerance** (Grid) **Plane Size** **Number of Slices** **Local Angle** **Tolerance** (Slice)		
	Properties	**Circle Radial Tolerance** **Interpolation Tolerance** **Minimum Error Volume** **Model Tolerance** **Cutting Tolerance**		
	AutoSave	In Process	**# of Cuts**	
		Shade Copy	**# of Cuts**	
			Margins	**Left** **Right** **Top** **Bottom**

			Render	
				Width
				Height
			View Angle	
		VERICUT Solid	**# of Cuts**	
View	Attributes	**Light (IJK)**		
	Section	**Cuts**		
		Delay		
Model	Block	**Length (X)**		
		Width (Y)		
		Height (Z)		
	Auto Block	**Offset**		
	Cone	**Height (Z)**		
		Base Radius		
		Top Radius		
	Cylinder	**Height (Z)**		
		Radius		
	IGES File	**Tolerance**		
		Offset Amount		
		Base Location		
	Location Relative tp Parent	**X**		
		Y		
		Z		
Setup	Motion	Start At	**Line Number**	

	Stop At	**Num of Cuts** **Line Number**	
	Stop At Max Errors **Stop At Max Warnings** **Fast Feed** **Skip Cut** **Min. Cutter Height** **Holder Clearance**		
G-Code	Settings		**Ignore Data after Column** **Rotary Feed Motion Tol** **Rotary Rapid Motion Tol**
	Turning		**CSS Maximum RPM**
	WireEDM		**Wire Diameter** **Work Table XY Plane** **XY Plane to UV Plane**
APT Settings	**Max Tool Axis Angle** **Tool Axis Reset Angle** **Default CLEARP** **GOHOME R Dist**		
Machine	**Default Near Miss**		

Analysis	AUTO-DIFF	Solid	**Gouge Tolerance** **Excess Tolerance**
		Point	**Gouge Tolerance** **Excess Tolerance** **Gouge Check Distance** **Excess Check Distance** **Default Normal (IJK)**
		Profile	**Gouge Tolerance** **Excess Tolerance**
		Constant Gouge Check	**Tolerance**
		Compare by Region	**Minimum XYZ** **Maximum XYZ**
OptiPath	**Override All Feedrates %**		

Custom Interface - File Selection

While in "author" mode, right-clicking on a file selection feature on a feature page and selecting Details, displays the window illustrated below. The title of the window is the current text from the feature, in this case the file's path and name.

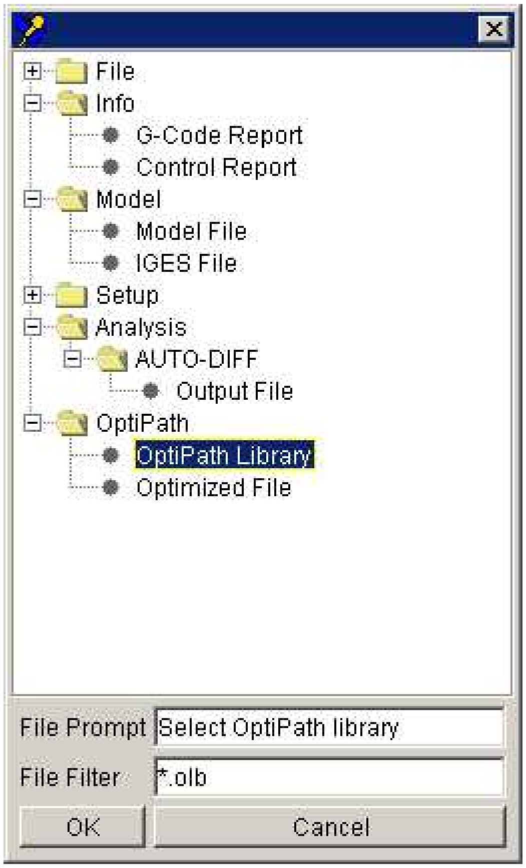
The tree that dominates the window is organized to reflect VERICUT's menu structure. Each "leaf" in the tree represents an input or output file.

To connect a file selection feature to a file type, simply expand the tree till you find the file type you are looking for and select it. To connect a widget to a file type, simply expand the tree till you find the one you are looking for and select it.

You can also supply a prompt which will be used in the file selection window when the user clicks on the browse icon () to the right of the feature. If the file type has one or more commonly used extensions, you can specify them as a filter for this window also. In this case we have just used "*.olb" to narrow the choices to OptiPath libraries, but for toolpaths you could use a compound filter, such as "*.tp;*.apt;*.cls". The user can always override the filter.

When you are satisfied with the properties of the file selection feature, click on the "OK" button. You can also use these windows to check existing associations, in which case you would use the Cancel button afterwards.

File Selection Feature Window



In the example window on the left, a file selection feature is being associated with the OptiPath library.

Custom Interface - File Selection Parameter List

File	**User File**		
	Working Directory		
	Convert	IGES	**Input File**
			Output File
		PolyFix	**Input File**
			Output File
		Surface	**Input File**
			Output File
		VDAFS	**Input File**
			Output File
		Binary CL	**Input File**
			Output File
			APT Table File
		Export Model File	
		Edit	
	ShadeCopy	**Output File**	
	AutoSave	In Process	**Auto Save**
			Auto Error
		Shade Copy	**Auto Save**
			Auto Error
		VERICUT Solid	**Auto Save**
			Auto Error

Info	**G-Code Report**	
	Control Report	
Model	Stock	**Model File**
		IGES File
	Design	**Model File**
		IGES File
	Cut Stock	
Setup	**Toolpath**	
	G-Code	**APT Output File**
		G-Code Log File
	Tool Library	
	Control	
	Machine	
OptiPath	**OptiPath Library**	
	Optimized File	

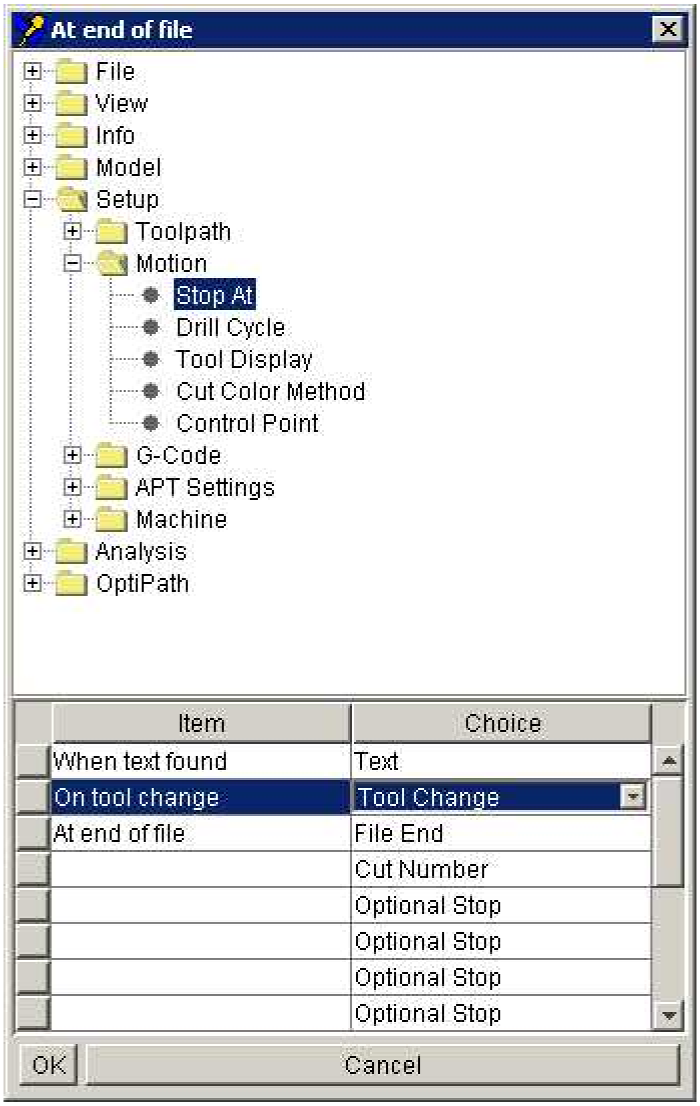
Custom Interface - Choice Lists and Radio Buttons

While in "author" mode, right-clicking on either a choice list or radio button on a feature page and selecting Details, displays essentially the same window. The only difference is that the one for choice list features has a table of choices at the bottom of the window and the one for a radio button feature has a single drop-down list. The title of the window is either the text of the current choice in the choice list or the caption of the radio button.

The tree that dominates these windows is identical. It is organized to reflect VERICUT's menu structure. Each "leaf" in the tree represents a set of choices for a single parameter.

To connect a feature to a parameter, simply expand the tree till you find the parameter you are looking for and select it. Select an option from those available in the Choice pull-down for each Item, and then click OK. The interaction is slightly different for choice lists and radio buttons and described in more detail below. You can also use these windows to check on an existing association, in which case you would use the Cancel button afterwards.

Choice List Feature Window



For a choice list feature, selecting a parameter in the tree, causes the **Choice** column of the table at the bottom of the window to be filled with the set of options that are appropriate for the selected parameter. Each cell of the Choice column is itself a choice list containing the set of options that are appropriate for the selected parameter.

The text that will appear to the user in the choice list appears in the Item column of the table. Double-click on the item to edit the text. You can also edit the text for each item in the choice list from the feature page while in "author" mode.

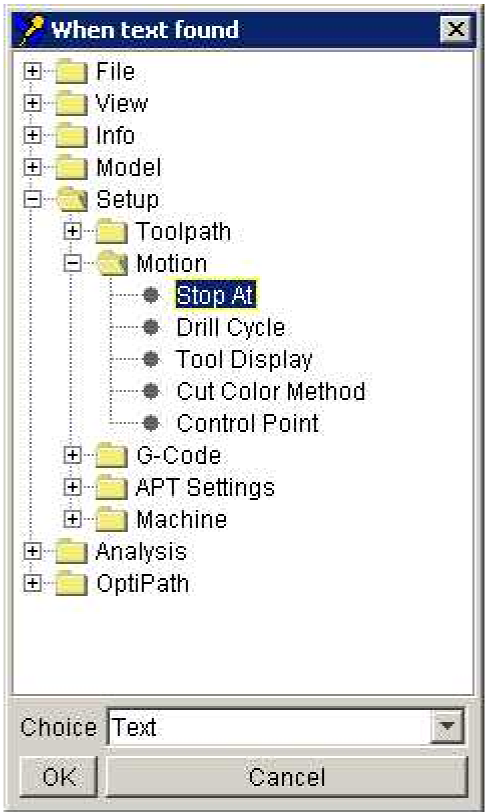
To associate each item with a particular option:

1. Click on an item in the Item column so that it becomes highlighted.
2. Click on the corresponding row in the Choice column to expand the pull-down of available options.
3. Click on the option in the pull-down to associate it with the item.
4. When you have finished associating the items with options, click on the OK button.

You can provide the user with a more limited set of options than is available in the standard VERICUT user interface by simply including fewer items in the Item column.

The items that you include can be in any order. To change the order of items, use the small blank buttons on the left edge of the table to drag rows to different positions.

Radio Button Feature Window



For radio buttons, selecting a parameter in the tree, causes the Choice pull-down at the bottom of the window to be filled with the set of options that are appropriate for the selected parameter. Keep in mind that there can only be one set of radio buttons on each feature page so when picking a parameter for one radio button, you are actually picking it for all the others on the page. The same is true if you change the parameter associated with a radio button. It will change for all the other radio buttons on the page also.

In either case you will need to associate a particular option for each individual radio button.

To associate an option with a radio button, simply click on the Choice pull-down to expand it and click on the desired option. Click **OK**.

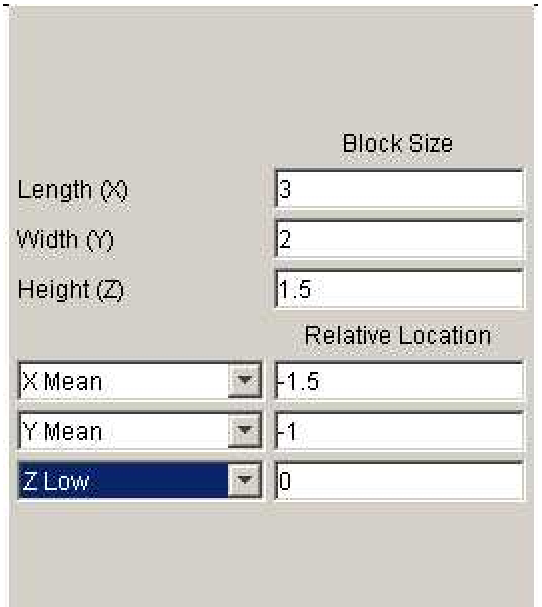
The text for radio buttons can only be edited on the feature page while in "author" mode.

As with choice lists, you do not need to include a radio button for all of the available options. Include fewer radio buttons on the feature page to simplify the interaction for the user.

Most of the parameters in the tree correspond to functionality that is available in VERICUT's standard user interface. The exceptions are the three parameters located under **Model > Relative Location**. These parameters control the location of a stock block model relative to the stock component's datum and when combined on a feature page with certain other features provide some extra functionality for automating the placement of a stock block that is not currently available in the standard VERICUT user interface.

Example - Automatic Stock Block Placement

The following example shows how you can use a custom user interface to automate the placement of a stock block at the machine origin. This cannot be accomplished using the standard VERICUT user interface.



The upper three numeric field features are connected to the X, Y and Z parameters respectively, located under **Model** > **Block** in the numeric field "Details" tree representing the length, width and height of a stock block. When the user changes one of the values, all models in the stock component are replaced by a single block with the specified dimensions.
The lower three numeric field features are connected to the X, Y, and Z parameters respectively, located under **Model** > **Relative Location** in the same tree. When the user changes one of these values, the origin of the block (the X min, Y min, Z min corner) is positioned at the specified offsets.
The three choice lists in the lower left corner are connected to the parameters located under **Model** > **Relative Location** in the choice list "Details" tree. These features are unique to a custom interface and do not correspond to any of the features in the standard VERICUT user interface. They can be used to locate any of the block's corners, mid-edges or mid-faces to the specified location. As shown in the sample at the left, the center of the bottom face of the block is designated as the block's origin.
If a dimension of the block is changed, the position of the block remains unchanged. For example if block width is increased to 3, it would grow by 1/2 a unit in the +Y and -Y directions, thereby keeping the center of the bottom face position unchanged.

If you decide to use one or more of these choice lists, it is possible to limit the options available to the user. Each list can contain up to four choices but not all choices need to be included in the list. For example, the list for the Z axis has options for "Z", "Z Low", "Z Mean" and "Z High". You may decide that "Z Mean" makes no sense in your application and could omit that choice from appearing the list.

However, you should never omit the "X", "Y" or "Z" choices from the choice lists. These are needed to position the block by specifying values directly in one of the numeric entry fields because none of the predetermined positions is appropriate. For example, if the block is 1.5 units high and the user enters a Z location of -0.5, the choice list should not display "Z Low", "Z Mean" or "Z High". Instead it should be able to display just "Z".

You can of course alter the text of the choices, so instead of using "Z Low" and "Z High", you could display "Z Minimum" and "Z Maximum", or "Bottom Face" and "Top Face" or any other suitable text.

NOTE: The three parameters for automating the location of a stock block are not applicable to other types of primitives, such as cylinders, cones, polygon files or IGES files.

Custom Interface - Choice List/Radio Button Parameter List

File	Convert	IGES	**Output File Type** **Output File Format** **Normal Direction** **Surface Input Mode**
		PolyFix	**Input File Type** **Output File Type** **Output File Format**
		Surface	**Input File Type** **Output File Type** **Output File Format**
		VDAFS	**Output File Type** **Output File Format** **Normal Direction**
		Binary CL	**Input File Type**
	Export Model	**File Type** **File Format** **Output Color** **Coordinates** **Units** **Process** **Slice Direction**	
	Colors	**Brightness**	
	Properties	**Default Units** **Display Resolution**	

		Default Machining Type	
		Base Cut Tolerance On	
	Shade Copy	**Type**	
		Method	
		Shading By	
		Orientation	
		Paper Size	
		Light Location	
View	Attributes	**View Type**	
		Draw Mode	
		Background	
Model	Location Relative to Parent	**X**	
		Y	
		Z	
	Normals		
	Units		
	Visibility		
	Mixed Mode		
Setup	Toolpath	**Toolpath Type**	
		Tool Change By	
	Motion	**Start At**	
		Stop At	
		Drill Cycle	
		Tool Display	
		Cut Color Method	
		Control Point	
	G-Code	Settings	**Programming Method**
		Block Skip	**Apply Switch Value**

		4-Axis Sync Method
	Apt Settings	**Default Toolpath Units** **RAPID Motion** **GOHOME Motion** **ROTATE** **ROTABL** **ROTHED** **Philosophy** **Angle Value** **Direction**
	Machine	**Floor/Wall Orient** **Ignore Collision between Cutter and Stock**
Analysis	AUTO-DIFF	**Stock Display** **Design Display** **Comparison Method** **Comparison Type** **Normal Direction**
OptiPath	**OptiPath Control**	

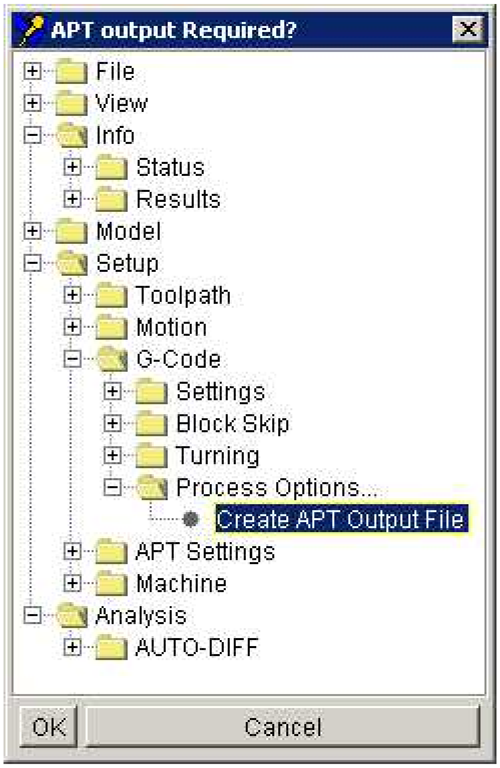
Custom Interface - Check Boxes

While in "author" mode, right-clicking on a checkbox feature on a feature page and selecting Details, displays the window illustrated below. The title of the window is the current text associated with the check box.

The tree that dominates the window is organized to reflect VERICUT's menu structure. Each "leaf" in the tree represents an on/off parameter available in the standard VERICUT user interface.

To connect a feature to a parameter, simply expand the tree till you find the on/off parameter you are looking for, select it and click **OK**. You can also use these windows to check existing associations, in which case you would use the Cancel button afterwards.

Check Box Feature Window



In the example window on the left a check box feature is being associated with the on/off flag that determines whether an APT output file is generated corresponding to the processed G-Code toolpath.

Custom Interface - Check Box Parameter List

File	Convert	Surface	**Project Rectangle**
		Binary CL	**Output Circles**
	Export Model	**Binary**	
		Millimeter	
		Group By Color	
		Detect Scallop Planes	
	Colors	**Recycle Cut Color**	
	Properties	**Millimeter**	
		Stock Consistency Check	
		Model Export Cut Mode	
	AutoSave	In Process	**Cutter Change**
			# of Cuts
			File End
			Auto Error
		Shade Copy	**Print Command**
			Annotation
			Cutter Change
			# of Cuts
			File End
			Auto Error
			Color
			Portrait
			Add output file to Report
			Render
			Specify image size
			Shadows

		VERICUT Solid	**Cutter Change** **# of Cuts** **File End** **Auto Error**
View	Orient	**Zoom creates new View** **Standard Rotation Image**	
	Section	**Section on mouse pick**	
	Axes	**Component** **Model** **Machine Origin** **Workpiece Origin** **Coordinate System** **Tool Tip Zero** **Driven Point Zero** **Active Coordinate System**	
Info	Status	**SubSystem**	
		Position	**Record Number** **Toolpath Record** **Machine Axes** **Tool Tip**
		Tool	**Record Number** **Change Record** **Sequence** **Geometry** **Description** **Number** **ID**
		Status	**Errors**

			Warnings **Time** **Time %** **OP Time** **Distance** **Distance %** **Coolant**
		Feed	**Feedrate** **OP Feedrate**
		Speed	**Spindle** **OP Spindle**
		OP Info	**Volume Removal Rate** **Chip Thickness**
		Graph	**Tool Use** **Cutting Conditions**
Model	Model File	**Outward** **Millimeter**	
	Component Attributes	**Disable Auto Direction** **Reverse Direction**	
	Import	**Offset Surface** **Extend to Rectangle**	
	Delete Material	**Update While Simulating**	
	Save Cut Stock	**Save With Features**	
Setup	Motion	**Stop At Max Errors**	

	Stop At Max Warnings	
	No Animation	
	Check for Spindle Off	
	FastMill	
	Ignore Undercuts	
	Display Holder in Workpiece View	
	Tool/Fixture Collision Only (no cutting)	
	Calculate Min. Cutter Extension	
G-Code	Settings	**Process Cutter Comp.**
		Scan Toolpath Files
		Ignore Data After Column
		Apply Acceleration to Cycle Time
	Block Skip	**Immediate**
		Switch 1
		Switch 2
		Switch 3
		Switch 4
		Switch 5
		Switch 6
		Switch 7
		Switch 8
		Switch 9
	Turning	**RPM changes with RAPID and CSS**
	Process Options	**Create Apt Output File**
Apt Settings	**Millimeter**	
	MULTAX	
	Process Circles	
	Reverse Circles	

		Detect Chordal GOTO's	
		Reset Tool Change	
		Process NX or CATIA Matrix	
		Ignore unrecognized Major words	
		Rapid Motion	**Square**
		GOHOME Motion	**Retract**
		Angle Value	**Absolute**
		Detect 4-Axis Rotary Motion	
	Machine	**Machine Simulation On**	
		Transparent Stock	
		Collision Detect	
		Overtravel Detection On	
		Axis Priority	**Pos/Neg**
			Negative
Analysis	AUTO-DIFF	Settings	**Stock Display**
			Design Display
		Options	**Design Consistency Check**
			Keep Design Solid
			Disable Report
			Report Uncut Differences
			Show Point Vectors
			Constant Gouge Check
		Compare by Region	**On**
			Apply Regions Over

Entire Stock

Improve Cut Stock Tolerance

Custom Interface - Buttons

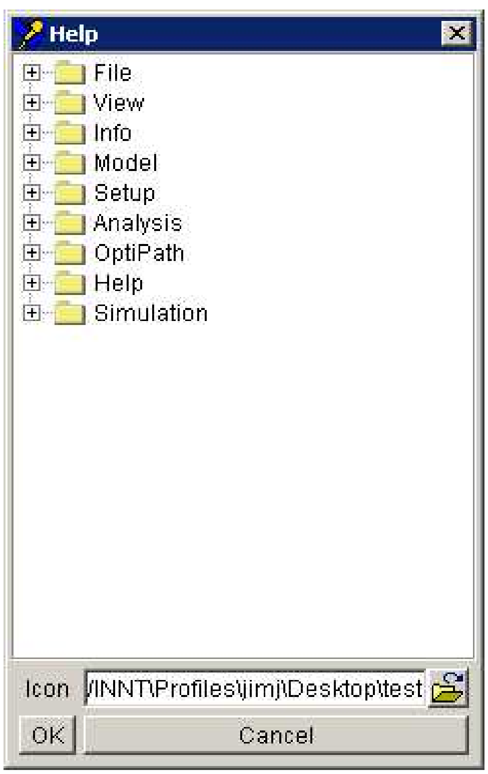
While in "author" mode, right-clicking on a button feature on a feature page and selecting Details, displays the window illustrated below. The title of the window is the current text from the button.

The tree that dominates the window is organized to reflect VERICUT's menu structure. Each "leaf" in the tree represents an action available in the standard VERICUT user interface.

To connect a feature to a parameter, expand the tree until you find the parameter you are looking for, select it and click OK. You can also use these windows to check existing associations, in which case you would use the Cancel button afterwards.

A button can contain an icon in addition to, or instead of, the caption. An icon is derived from a GIF or JPEG file which you select while in this dialog. An icon's height is limited to 16 pixels. If you select a larger icon, it will be shrunk to fit within the button's height, but its aspect ratio is maintained. The GIF or JPEG file will be copied to the book's folder if necessary, so that the entire book can be handled as a unit. Use the Icon field, at the bottom of the Button Feature window, to enter or select the image file to use.

Button Feature Window



In the example window on the left a button feature is being associated with a request to browse VERICUT's help files. When the user presses the button, VERICUT Help will display.

Custom Interface - Button Parameter List

File	User File	**New**
		Open
		Save
		Save As
	In Process	**Open**
		Save
		Save As
	Working Directory	
	Convert	**IGES**
		PolyFix
		Surface
		VDAFS
		Binary CL
	Export Model	
	Edit	**Toolpath**
		File
	Colors	
	Properties	
	Report	**Create Text Report**
		Create HTML Report
		Create PDF Report
		Report Template
		User-Defined Tags
	Print	
	Shadecopy	

Close Book
Exit VERICUT

View	Layout	**One View**
		Left and Right
		Top and Bottom
		1 Left, 2 Right
		1 Top, 2 Bottom
		Four Views
		Add View
		Delete View
		Cascade
		Tile Horizontally
		Tile Vertically
		View to Back
		Always in Front
	Orient	**Dialog**
		XY
		YZ
		ZX
		YX
		ZY
		XZ
		H-ISO
		V-ISO
		Zoom In
		Zoom Out
		Fit
		Fit All
		Refine
		Last Refine Display
		Reverse
		Reset
	Attributes	

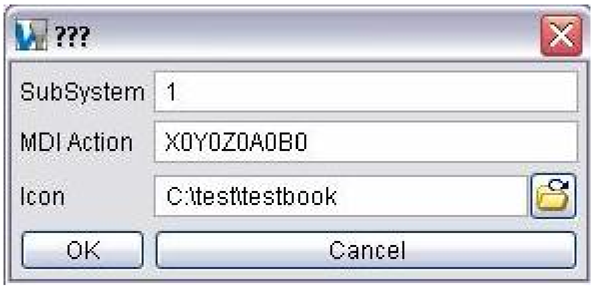
	Section	
	Select/Store View	
	Axes	
Info	**Toolpath**	
	Status	
	Machine Offsets	
	File Summary	
	VERICUT Log	
	Clear Logger	
Model	**Model Definition**	
	Auto Block	
	Component Tree	
	Load Stocks	
	Delete Material	
	Revolve Profile	
	Sweep Profile	
Setup	**Toolpath**	
	Motion	
	G-Code	**Settings**
		Variables
	Apt Settings	
	Cycle Definitions	
	Coordinate Systems	
	Tool Manager	
	Control	**Open**
		Save
		Save As
		Words
		Word/Address
		Mdi
		Settings

		Advanced Control Options
	Machine	**Open**
		Save
		Save As
		Settings
Analysis	**X-Caliper**	
	AUTO-DIFF	**Dialog**
		Apply
	Comparator	
	Toolpath Review	
	Inspection Sequence	
	Die Sinking Simulation	
OptiPath	**OptiPath Control**	
	OptiPath Manager	
	Compare Toolpath Files	
Help	**Browse**	
	About	
Simulation	**Reset Model**	
	Rewind Toolpath	
	Single Step	
	Play to End	

Custom Interface - MDI Buttons and Numeric Fields

While in "author" mode, right-clicking on an MDI button feature on a feature page and selecting Details, displays the window illustrated below. The title of the window is the current text from the MDI button.

MDI Button Feature Window



In the "SubSystem" field, enter the index number of the subsystem (not the subsystem name) that the MDI Action command is to be applied to.

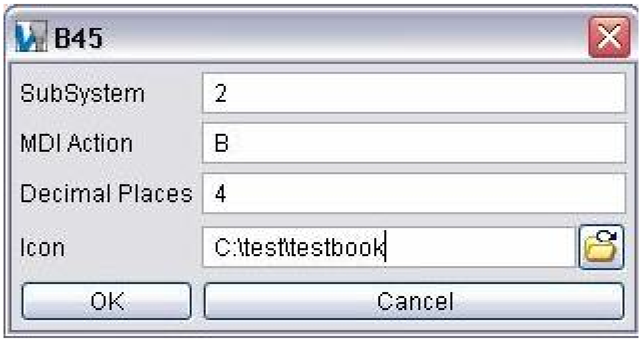
In the "MDI Action" field, enter the command(s) to be executed when the MDI button is pressed by the user, then click **OK**.

You can also use the window to check what command(s) are currently associated with the MDI button, in which case you would use Cancel afterwards.

A default MDI button will contain the icon (), in addition to the caption. But you can provide another icon from a GIF or JPEG file which you select while in this dialog. An icon's height is limited to 16 pixels. If you select a larger icon, it will be shrunk to fit within the button's height, but its aspect ratio is maintained. The GIF or JPEG file will be copied to the book's folder if necessary, so that the entire book can be handled as a unit.

While in "author" mode, right-clicking on an MDI numeric feature on a feature page and selecting Details, displays the window illustrated below. The title of the window is the current MDI command combined with the value from the MDI numeric field.

MDI Numeric Feature Window



In the "SubSystem" field, enter the index number of the subsystem (not the subsystem name) that the MDI Action command is to be applied to. In the "MDI Action" field, enter the command to be executed when the value in the MDI numeric feature field is changed or when the MDI icon () is pressed, then click **OK**. Use the Decimal Places field to specify the maximum number of decimal places to display. For an integer, use zero decimal places. Once you have entered the desired MDI Action command and specified decimal places, click OK.
You can also use the window to check what command and decimal place value is currently associated with the MDI numeric feature, in which case you would use Cancel afterwards.

As with an MDI button, you can change the icon used with an MDI numeric field by specifying an appropriate GIF or JPEG file.

Creating Custom Interface Help Pages

Now that you have created and sized the notebook and added some features to the feature pages, it is time to add some help pages to assist the user in using the custom user interface to accomplish their job. Use help pages in the notebook to instruct the user on how to use the features or guide then through a sequence of actions to accomplish the required task.

A "help" page is easily distinguished from a "feature" page by the white background. Only text and illustrations can be added to "help" pages. In order to add, remove or modify text or add, remove or replace illustrations to a help page you must be in "author" mode (F7 key).

If your notebook does not yet contain any help pages, or you need additional ones, use the procedure described in **Adding Pages to the Notebook** to add them.

Help Page Layout:

Each help page is created blank. Click on the help page and begin to type your instructions. Use the column of icons located on the right side of the notebook (when in "author" mode) to position, place emphasis on portions of text or add "style" to your help page. Use the bottom icon in the column to add illustrations to your help page. Each of the icons is described in more detail below.

The first four icons are used to control the appearance of the text. They can be used in two ways. As you type you can change the current "mode" by clicking on one of the icons. Subsequent text will exhibit the change until you click the same icon again to revert to regular characters. Or you can highlight an existing portion of the text and update its appearance by clicking on one or more of the icons. A double click will highlight a word. A triple click will highlight a line. You can also apply multiple style types to your text as illustrated here using bold, italics and underline on the same text.

Bold
Italics
Underline
Text Color
Align Left
Center
Align Right
Add image
Insert Hyperlink

Bold — Use to make selected text bold.

Italics — Use to make the selected text in italics.

Underline — Use to underline the selected text.

Text Color — Displays a color pallet enabling you specify a color for text. Pick one of the colored squares to proceed. The darker colors show up best on the white background of a help page.

The next three icons are used to control the position of the text displayed on the page.

Align Left — Use to align the selected text to the left side of the page. This is the default.

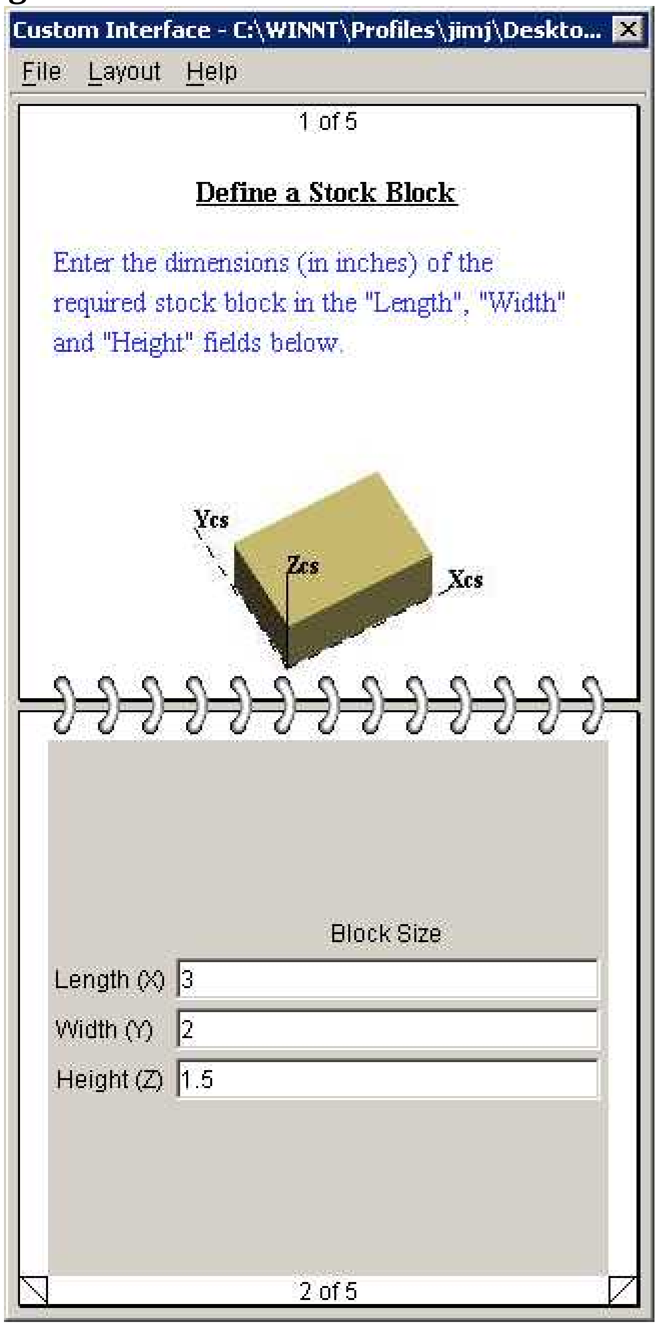
Center — Use to center the selected text on the page.

Align Right — Use to align the selected text to the right side of the page.

Add Image — Use to add illustrations to your help page. The images used for illustrations can be in GIF or JPEG format. When you select an image file it is copied to the notebook folder so that all elements of the notebook are kept together. Each image will be centered in the page and will be at its stored size. If you need to adjust the size of an illustration, do so with an image editing software before using on your help page.

Insert Hyperlink — Use create a hyperlink between the highlighted text and an HTML file. Highlight the desired text, select the Insert Hyperlink icon, then use the file selection box that displays to specify the file to create the hyperlink to.

Sample Help Page:



The figure above illustrates a simple example of how you can use a help page to instruct the user on how to use the features contained on the feature page below.

Now that you have a complete custom VERICUT interface notebook, it's time to do some testing.

Testing the Custom Interface

While a notebook is in "author" mode, (size displayed adjacent to the menu items and columns of icons displayed on both sides) any changes to the content of features are not communicated to VERICUT. To test that all features are connected to the correct parameters and function as intended, the notebook needs to be in "user" mode. Toggle between these modes with the F7 key on the keyboard

Unlike many of VERICUT's windows, the notebook window does not use the concept of an Apply or OK button to invoke sets of changes. When the user changes the content of a feature on a notebook's feature page, that change is sent to VERICUT immediately. For many parameters, a change in value does not cause any immediate response. For example, changing the number of cuts to be processed when the "Play" button is hit has no effect until the toolpath is processed.

On the other hand, some feature changes will cause an immediate response from VERICUT. For example, using a file selection feature associated with the user file will cause VERICUT to retrieve the selected file immediately. Similarly, if you have numeric fields associated with the dimensions of a block, changing one of the values will cause VERICUT to immediately apply the change. With experience you will learn which parameters will prepare for a future response and which ones will result in an immediate response from VERICUT.

As you test the notebook try to view it as the targeted user will. Is the purpose of each feature clear? Would some different wording in the labels make more sense, or perhaps some extra labels? Could an illustration on a help page replace a bunch of words? Do the views presented by the notebook's default user file display the process effectively or could they be improved? Above all, is it simpler to use and save time for the user?

If you decide that refinements are needed in the default user file, closing the notebook while in "author" mode causes VERICUT's standard user interface to be displayed so that you can make those changes.

When you are satisfied with the notebook's behavior, it is time to make your custom VERICUT user interface available for use.

Making the Custom Interface Available for Use

Notebook folder and file descriptions:

All of the files that define a notebook are stored in a single folder, the name of the folder being the name of the notebook. Some or all of the following file types may be present:

- HTML files for the help pages.
  Their names will be of the form "Pagexxx.html", where "xxx" is the page number.
- GIF and JPEG files referenced by the HTML pages.
- XML files for the notebook and it's feature pages.
  Their names will be of the form "Pagexxx.xml", where "xxx" is the page number.
- "Page000.xml" which contains information pertinent to the whole notepad.

Notably absent from the above list is the default user file for the notebook and any files referenced by it. It may be beneficial to store the default user file and any associated files in the notebook folder to simplify data management. That way, if a notebook's folder location is changed, all of the files associated with it move with it.

Making the notebook available:

Deploying a customized notebook to a target user involves making the folder available. This can be done by establishing a network mapping to the folder, or by copying the folder to the user's machine. If the default user file and the files which it needs are not in the same folder, you may find that the relocated notebook is unable to access the default user file. If this is the case, on the target machine use the F7 key to briefly get into "author" mode again, switch to the required user file and re-save the notepad. The user file that is displayed at the time you save becomes the notebook's default.

You can also launch VERICUT each time with a custom user interface instead the standard user interface by using the file customizer. bat to start the VERICUT session. This file can be found in the **\cgtech6x\windows\commands** folder. The **customizer .bat** file needs to be edited as follows to specify which custom interface notebook to use at startup.

In the customizer.bat file locate the "start" command, shown below, at the end of the file.

%start_cmd% "%CGTECH_JRE%\bin\javaw" -Xms16m -Xmx64m -Xss4m -classpath"%CGTECH_CLASSES%;%CGTECH_CLASSES%\CGTech.jar" VeriBook %argstr%

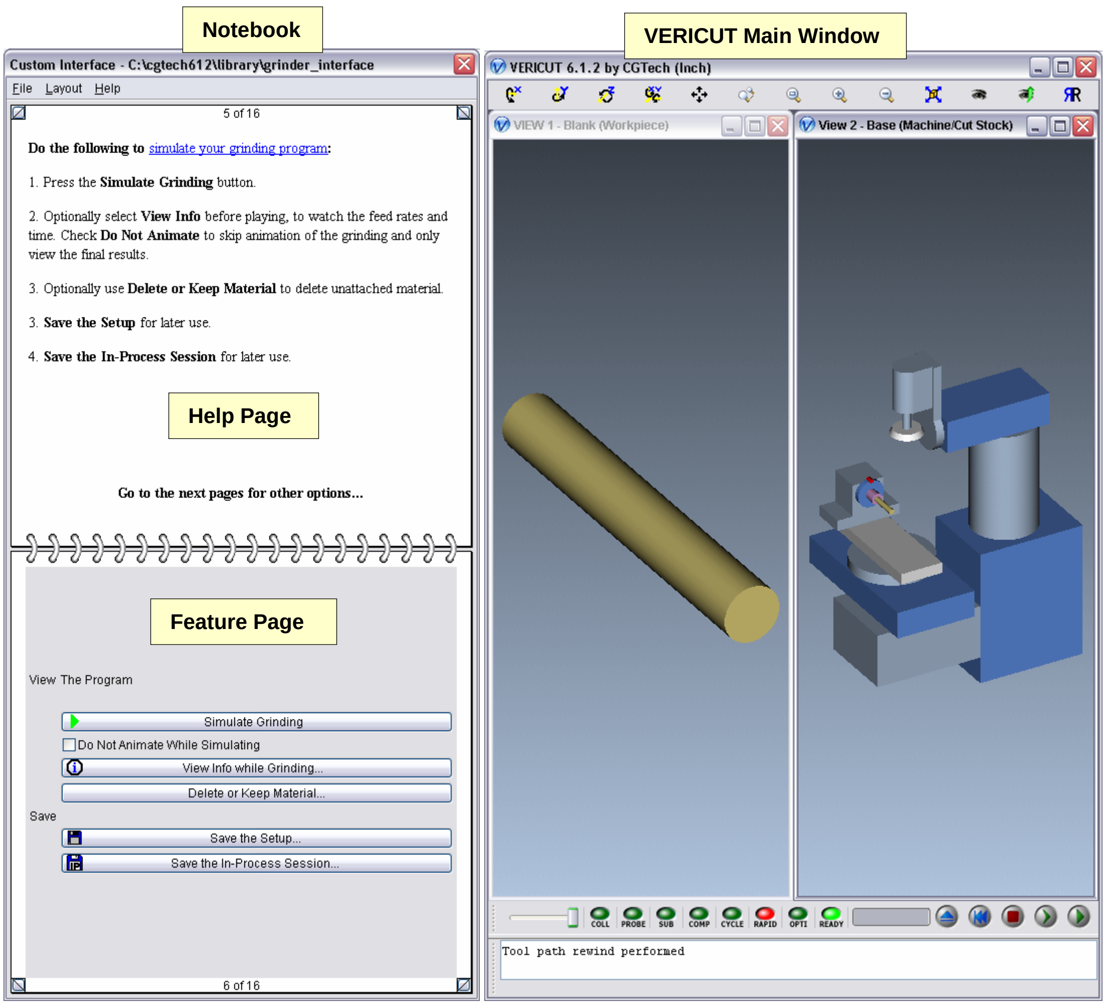
Make sure that the next to the last parameters is **VeriBook** (not Vericut) indicating that VERICUT should be started with a custom user interface instead of the standard user interface.

Replace the last parameter "**argstr**" with the path to the desired custom interface notebook folder.

Now simply use **customizer.bat** to start the VERICUT session with the specified notebook instead of with the standard VERICUT user interface.

Using a VERICUT Custom User Interface

The following user interface is applicable to the VERICUT Mold & Die and VERICUT Grinder products, as well as to any "custom" interface created with VERICUT Customizer.



The user interface consists of two main parts, the "notebook" and the VERICUT main window.

The Notebook

The notebook consists of a menu bar and two different types of pages. Click on the upper page corners to page back in the notebook. Click on the lower page corners to page forward in the notebook.

Menu Bar

File:

Open — Displays the Select book's folder window allowing you to open another custom interface notebook for editing or create a new notebook.

Recent — Displays a list of recently opened files. To open a file in the list, select the desired file.

Exit VERICUT — Ends the VERICUT session.

Layout:

 — Positions the notebook on the left side of the VERICUT main window.

 — Positions the notebook on the right side of the VERICUT main window.

 — Overlays the notebook on top of the VERICUT main window. Left click on the notebook's header and drag to any position.

Help:

On VERICUT — Starts VERICUT Help.

About VERICUT — Opens a window that displays information about the current VERICUT session.

Help Pages

Help pages contain instructions, explanations and pictures to assist you in providing the necessary information to VERICUT. Help pages are easily recognized by the white background.

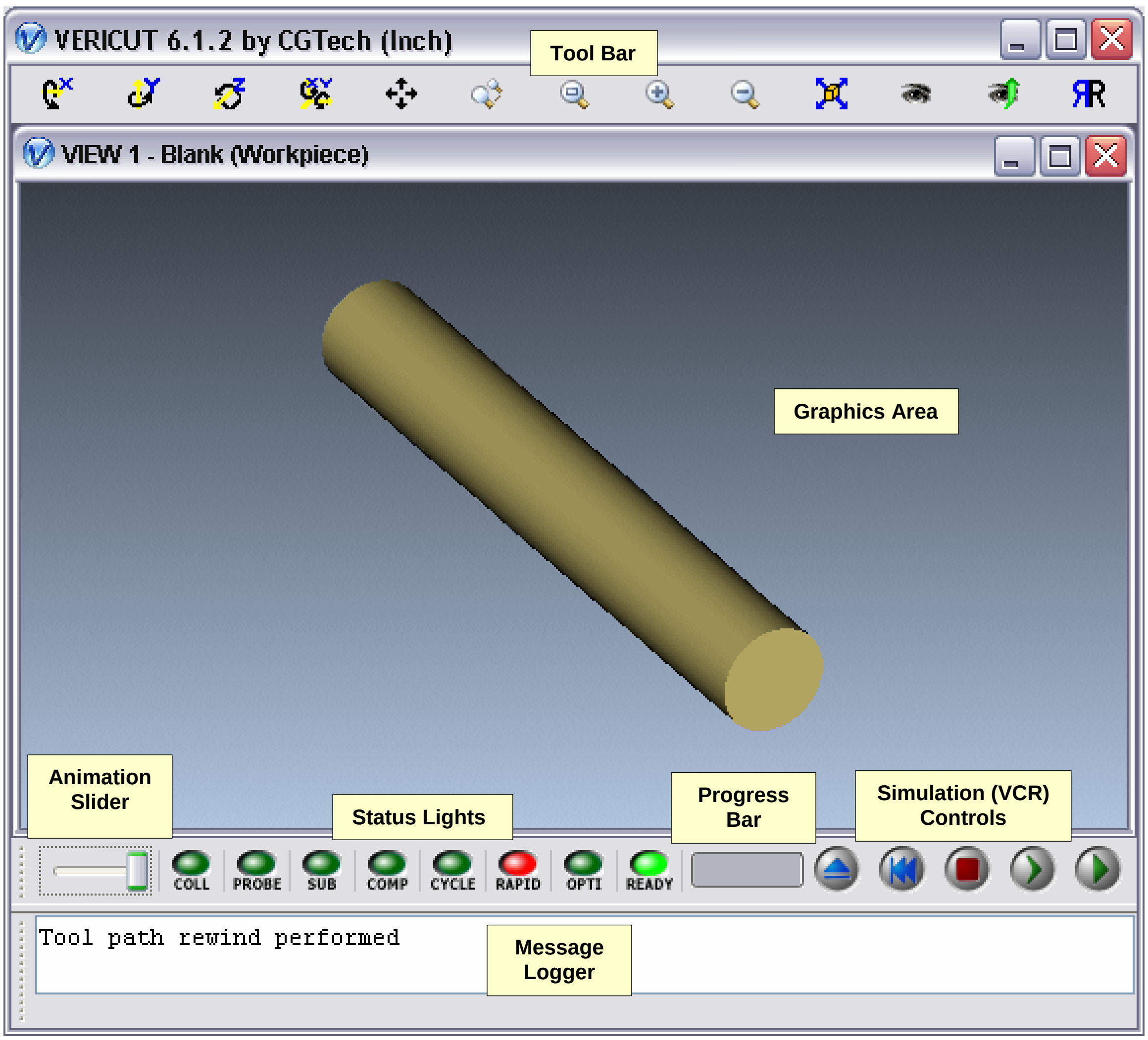
Feature Pages

Feature pages contain features (buttons, text fields, choice lists, etc.) that enable you to pass necessary information to VERICUT for processing. Each of the features on a "feature" page is described on the previous "help" page. Feature pages are easily recognized by the gray background.

Some features display another window enabling you to provide additional information to VERICUT. Use the F1 key to display more information about the displayed window.

VERICUT Main Window

The VERICUT main window is composed of six distinct areas, each with different user interaction. The window header displays the VERICUT version and the current session units (inch or millimeter). This window can be resized and/or moved like most other windows, via dragging the window header, sides or corners.



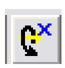
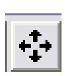
Toolbar

VERICUT's Toolbar provides quick and easy access to the view manipulation functions/features.

To see what is associated with a Toolbar icon, simply position the cursor over the icon and a tip appears.

Dynamic viewing options

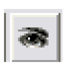
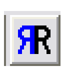
These options use mouse actions to dynamically orient the object in view. Mouse actions are different, depending on the active option-see the table below for details. Dynamic viewing options:

Icon:	Name:	Action:
	Dynamic X Rotation	Horizontal rotation- drag mouse up/down in direction to rotate
	Dynamic Y Rotation	Vertical rotation- drag mouse left/right in direction to rotate
	Dynamic Z Rotation	Screen plane rotation- drag mouse left to rotate CCLW, right for to rotate CLW
	Dynamic XY Rotation	Horizontal/vertical combined rotation- drag mouse up/down/left/right in direction to rotate
	Dynamic Pan	Pan/translate- drag mouse in the direction pan
	Dynamic Zoom	Zoom/magnify- drag mouse up to zoom in, down to zoom out

Shortcut: The following keys provide instant access from the keyboard to dynamic viewing options (press and hold keys while dragging): Dynamic Zoom — <Ctrl>, Dynamic Pan — <Shift>, Dynamic XY — <Ctrl> + <Shift>. Mouse actions are the same as above.

Static viewing buttons

These buttons use various mouse actions to zoom, fit, and reverse the object in view. See the table below for details.

Icon:	Name:	Action:
	Zoom to Box (standard mode)	Left-click and move mouse to trap area to zoom (press <Esc> to interrupt rubberbanding)
 	Zoom In / Zoom Out	Zooms in/out approx. 20% each time clicked
	Fit	Fits object in active view
	Refine Display	Refines the display to improve image quality
	Last Refine Display	Restores the last refined display
	Reverse	Reverses viewing direction, as if you stood behind the object

To learn more about interacting with VERICUT through the main window components, review the following topics under **Interacting with VERICUT**, in the **Getting Started section** of *VERICUT Help*.

Create Report

Location:

File menu > Create Report, in the main VERICUT window.

File > Create Report, in the Inspection Sequence window.

File > Create Report, in the Tool Manager window.

Opens the Save VERICUT Report window enabling you to specify the /*path*/*filename* for the report file to be created. If "User-Defined" information has not been previously defined, (either using the User-Defined Tag Values window or during a previous use of Create Report) VERICUT will then prompt you for any unspecified User-Defined Tag values, then generate the report using the "current" report template. Choose one of the following report formats:

Text — Use this option to create a report in text format.

HTML — Use this option to create a report in HTML format.

PDF — Use this option to create a report in PDF format. (PDF report generation based in part on iText (www.lowagie.com/iText).

See **Introduction to VERICUT Reports** , in the Getting Started section of *VERICUT Help* for information on report template files supplied with VERICUT.

See **Create a Report Using a Template**, in the *Using VERICUT* section, in the *CGTech Help Library* for additional information.

Print View

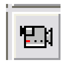
Location: **File menu > Print View**

Toolbar shortcut: 

Opens a window enabling you to print images of VERICUT view ports. Each view port is automatically fit to the printable area and printed on a separate page. All view ports are printed. Features on this window are standard print window features that enable you to access printer settings, specify number of copies, etc. Consult your printer documentation for more information.

Record Movie

Location: **File menu > Images > Record Movie**

Toolbar short cut for recording images: 

Opens a window enabling you to record images as a VERICUT Image file, or an AVI file. Images in all view ports are recorded. Images can be recorded interactively in VERICUT, or via batch processing. After recording is stopped, you can view the file via the **File menu > Images > VERICUT Playback** or **File menu > Images > AVI Playback** function, depending on the file type.

This window appears differently, depending on the **Output Format** choice. See the **VERICUT Image Record window** and **AVI Image Record window** topics below for additional information.

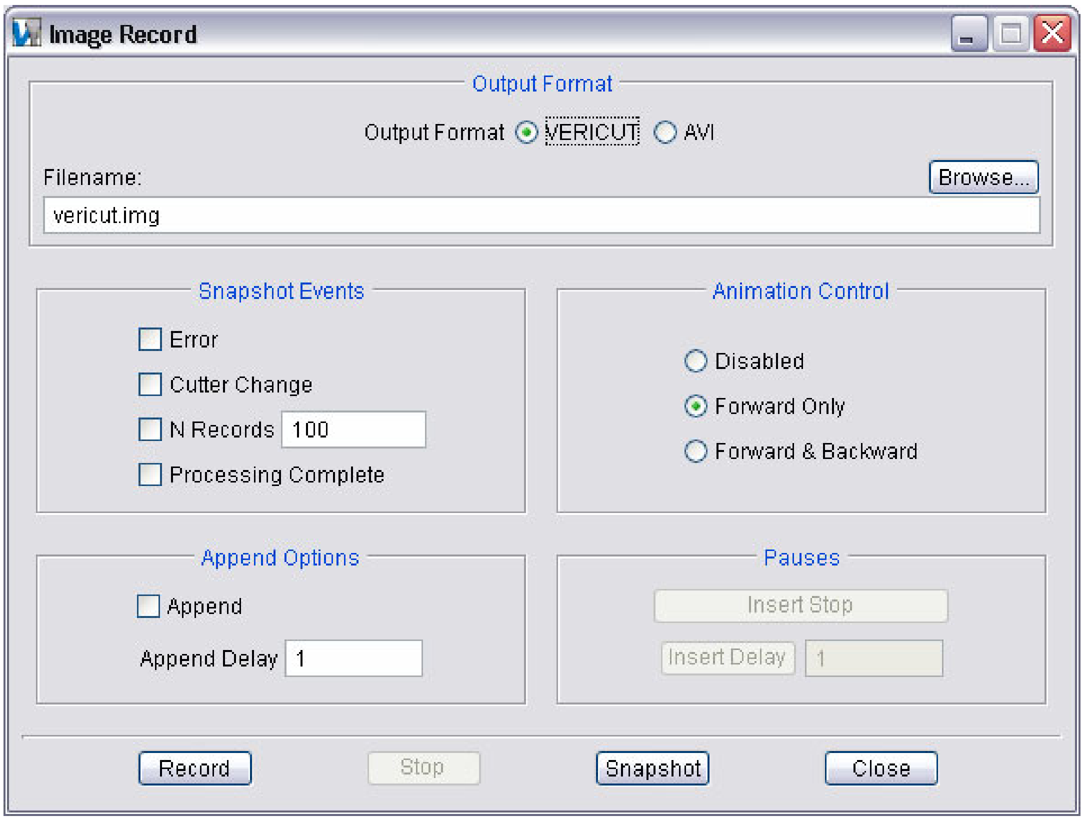
NOTES:

1. The size of the graphics area directly affects the display size of the recorded Image file. Make the VERICUT window larger to record larger images.
2. Recording animations captures all changes in the graphics area. Recorded tool paths with lots of small motions can generate large Image files sizes. Consider using the Animation Slider, located at the bottom of the VERICUT main window to reduce Image file size.
3. AVI record is not intended for use with cascaded views.
4. AVI record does not support capture of the Profile view.
5. The AVI record feature is not available on UNIX platforms.

VERICUT Image Record window

Location: File menu > Images > Record Movie

These features are available when recording images in a VERICUT Image file.



Output Format group:

These options determine the output format and the /path/filename of the recording file.

Output Format = VERICUT — Format of the output image file is VERICUT Image file.

Filename — Name of the file to receive recorded images.

Snapshot Events group:

These options control when a snapshot image is recorded. Each selected event causes a snapshot to be recorded when that event occurs.

Error — Record a snapshot when VERICUT detects an error.

Cutter Change — Record a snapshot when the cutter has been changed.

N Records — Record a snapshot after a specified number of tool path records have been processed.

Processing Complete — Record a snapshot when tool path processing has been completed.

Animation Control group:

These options determine when animation images are recorded, and how they will be replayed.

Disabled — Do not record animation images.

Forward — Record animation images to play in the forward direction only.

Forward & Backward — Record animation images to play in forward or backward direction.

Append Options group:

These options control appending images to an existing Image file.

Append — When active, appends recorded images to the end of an existing Image file.

Append Delay — When appending images, inserts the specified number of seconds of delay prior to appending new images.

Pauses group:

These features control the recording process, and insert delays into the VERICUT Image file.

Insert Stop — Inserts a stop point into the Image file. The stop point is used by the Stop at Inserted Stop play back feature.

Insert Delay — Inserts the specified seconds of delay.

Record — Starts recording images based on current window settings, VERICUT window size, shape, and layout. Recording must be stopped before the Image file can be viewed.

Stop — Stops recording. The Image file can be viewed via **File menu > Images > VERICUT Playback**.

Snapshot — Records a single snapshot image.

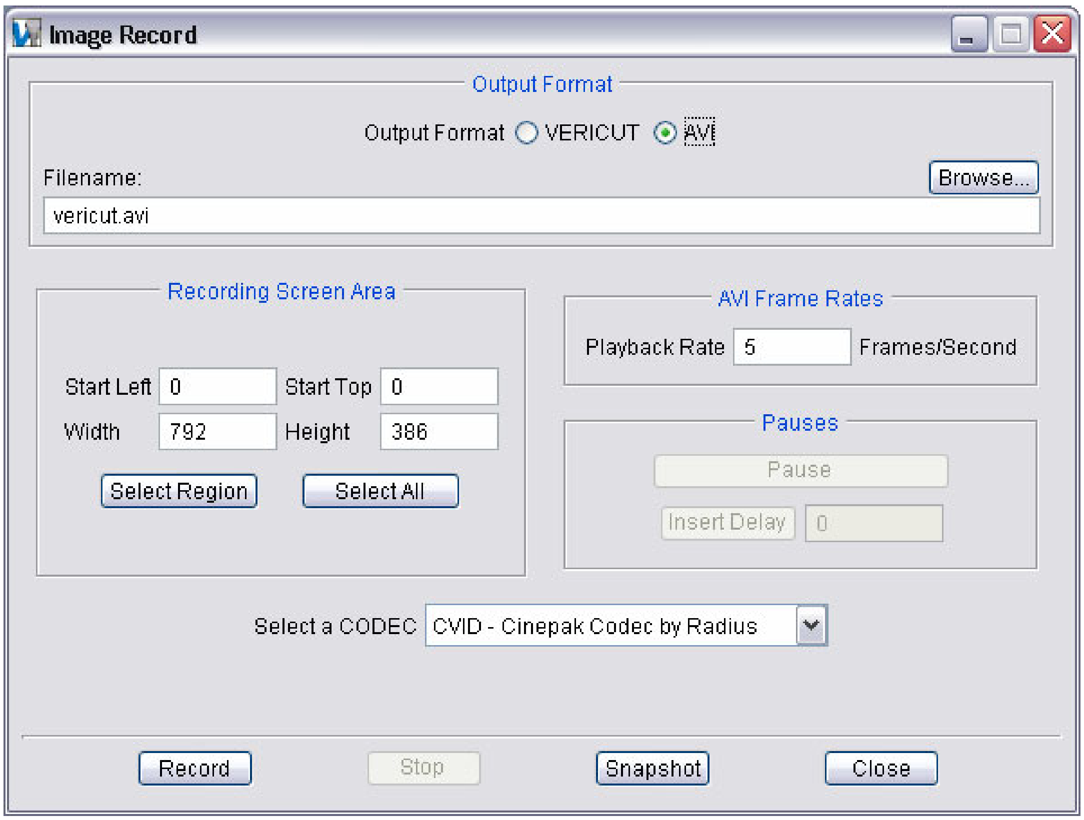
Close — Closes the Image Record window.

AVI Image Record window

Location: File menu > Images > Record Movie

NOTE: This feature is not available on UNIX platforms.

These features are available when recording images in an AVI file.



Output Format group:

These features define the format and /path/filename for the recorded file.

Output Format = AVI — Format of the output image file is an AVI file.

Filename — Name of the file to receive recorded images.

Recording Screen Area group:

These features control the screen area to record.

Start Left / Start Top — Identifies the start point (top-left location) of the screen area to record.

Width / Height — Width and height (in pixels) of the screen area to record, as measured from the top-left start point.

Select Region — Uses a rubber-banding box to indicate region to record. Left-click to define the start point of the region, then move the mouse to trap the area to record. Press <Esc> to interrupt rubber-banding.

Select All — Records all views.

AVI Frame Rates group:

These features set the rate at which to record image frames. Frame rate controls the speed at which the AVI file plays back.

Playback Rate — Number of frames to record per second. Increasing the recording frame rate causes the AVI file to play back faster.

Pauses group:

These features control the AVI recording process, and insert delay into the AVI file.

Pause / Continue — Pauses (or continues) AVI file recording. VERICUT motions simulated after pausing are not recorded.

Insert Delay — Inserts the specified seconds of delay.

Select a CODEC — Select the desired CODEC from the pull-down list. Use the CGTech environment variable, **CGTECH_CODEC_FILTER**, to remove invalid CODECs from the list.

See **Environment Variables,** in the **Getting Started** section for additional information.

Record — Starts recording animation images based on current window settings, VERICUT window size, shape, and layout. Recording must be stopped before the Image file can be viewed.

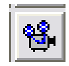
Stop — Stops recording. The AVI file can be viewed via **File menu > Images > AVI Playback**.

Snapshot — Records a single frame image.

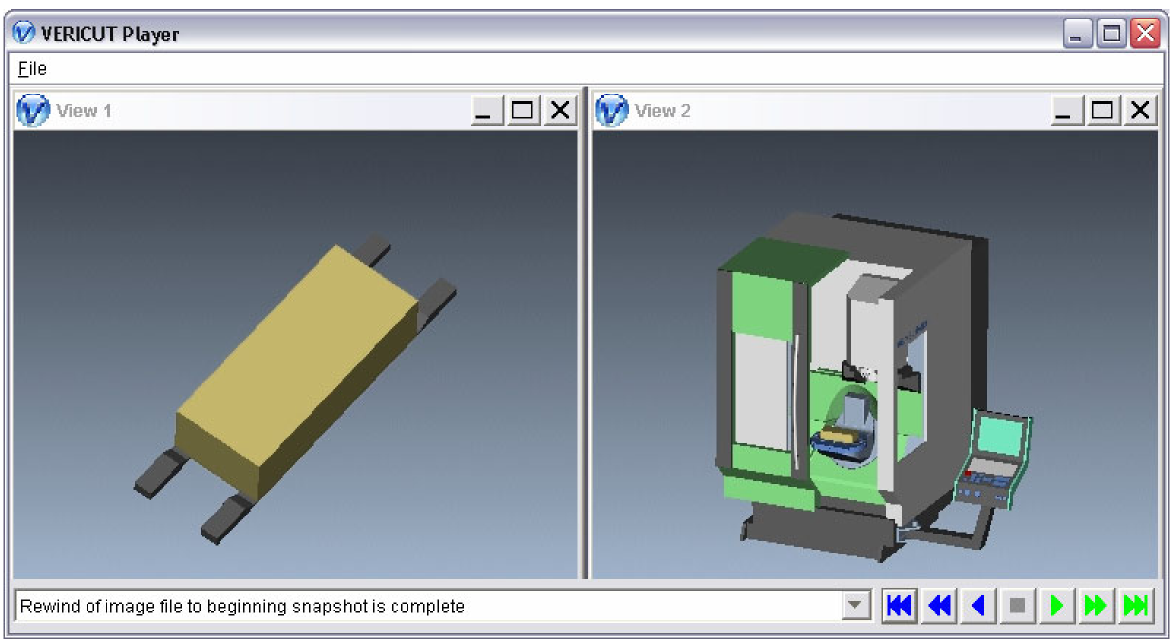
Close — Closes the Image Record window.

Playback VERICUT Movie

Location: **File menu > Images > Playback VERICUT Movie**

Toolbar short cut for playing back VERICUT images: 

Opens a "VERICUT Player" window to load and play back an Image file recorded by the **File menu > Images > Record Movie** function. Images are played in a separate "image player" window enabling simultaneous access to the VERICUT main window and its functions. The size and view layout of the image player window is as recorded. However, this window can be moved, or minimized using standard window controls. Recorded machining status is also seen when the **Status window** is open while playing.



File menu

Functions in this open new Image files, compare existing Image files, and provide control over playing back recorded images.

Open — Open an existing Image file.

Compare Files — Opens a window to configure the player to compare two Image files. Features enable you to play both Image files simultaneously, and identify differences between them.

Playback Options — Opens the Image Playback Options window for controlling playing back recorded images. See the **Playback Options window** topic below for additional information.

Close — Closes the VERICUT Player window.

Playback (VCR) icons

These icons control playing and rewinding recorded images. To see what is associated with an icon, position the mouse cursor over the icon and a tip appears. Playback VCR icons:

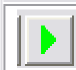
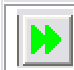
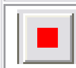
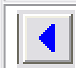
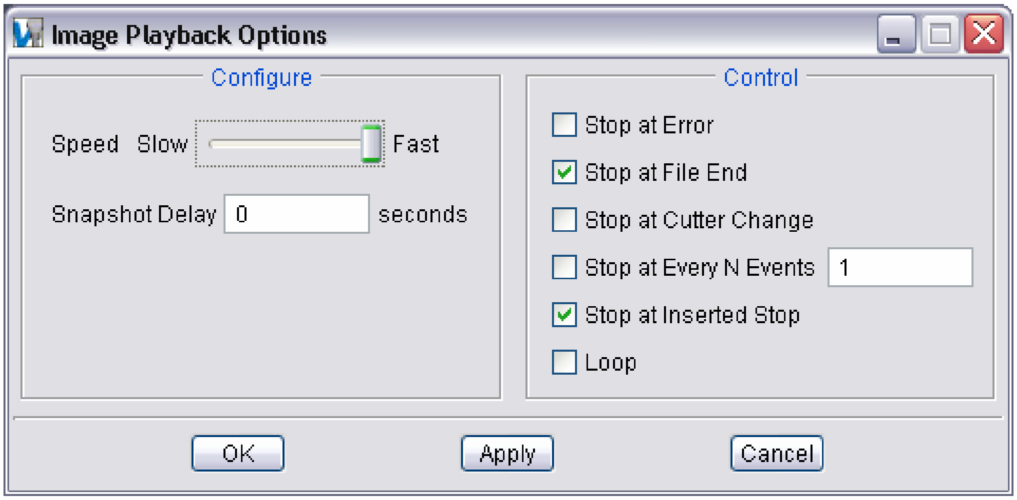
Icon:	Name:	Function:
	**Play**	Play images in the forward direction
	**Next Snapshot**	Advance to the next snapshot
	**Skip to End**	Advance to file end
 (or press Escape key <Esc>)	**Stop**	Stop playing back images
	**Play Backwards**	Play images in the backward direction
	**Previous Snapshot**	Rewind to the previous snapshot
	**Skip to Start**	Rewind to the Image file beginning

Image Playback Options window



Configure group:

Speed — Controls how fast images are played back. Actual speed varies, depending on computer hardware. Stop the play back to access this feature.

Snapshot Delay — Number of seconds to pause after displaying a snapshot.

Control group:

These options provide additional stop control by event, or the ability to play in a continuous loop (without stopping). Options:

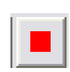
Stop at Error — Stop replay at an error detected by VERICUT.

Stop at File End — Stop replay when the end of a toolpath file end has been reached.

Stop at Cutter Change — Stop replay when the cutter has been changed.

Stop at Every N Events — Stop replay after a specified number of events (frames) have been recorded.

Stop at Inserted Stop — Stop replay on a stop inserted by pressing "Insert Stop" on the **File menu > Images > Record Movie** window.

Loop — Play in a continuous loop until interactively stopped via pressing  (**Stop**), or pressing the **Escape** key <**Esc**>.

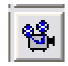
OK — Applies the settings and closes the Image Playback Options window.

Apply — Applies the settings and leaves the Image Playback Options window open.

Cancel — Cancels the current setting changes and closes the Image Playback Options window.

Playback AVI Movie

Location: **File menu > Images > Playback AVI Movie**

Toolbar short cut for playing back VERICUT images: 

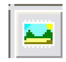
NOTE: This feature is not available on UNIX platforms.

Opens an "AVI Player" window to play back an AVI files recorded by the **File menu > Images > Record Movie** function. Assuming your system has an AVI player installed, the AVI images are played in a separate "image player" window enabling simultaneous access to the VERICUT main window and its functions. The size and view layout of the image player window is as recorded.

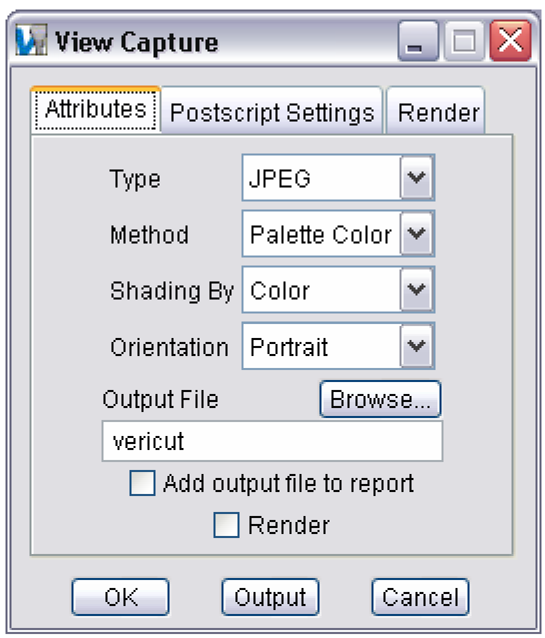
View Capture window

Location: **File menu > Images > View Capture**

File menu > AutoSave > View Capture tab: Properties

Toolbar short cut for accessing View Capture: 

Opens a window to capture VERICUT images and format them for printing or importing into desktop publishing applications. View Capture can also output printer-formatted files that you can move to other computers or printing devices. Images can be captured and output interactively or automatically (via AutoSave), or via batch processing. When configuring this window for future use or batch processing, press OK (instead of Output) to save the window configuration without outputting a View Capture image file. Hint: Avoid having to re-configure this window by creating a master Project file containing the modified View Capture values.



Attributes tab — Features on this tab are used to configure the attributes for formatting captured VERICUT images.

Postscript Settings tab — Features on this tab provide information about centering and annotating PostScript format images on the printed page.

Render tab — Features on this tab enable you to specify the output size of "rendered" images.

OK — Saves the View Capture window settings and closes it without outputting an image file.

Output — Outputs the current VERICUT image to the specified file. File format is determined by the View Capture window configuration.

NOTE: Output is not applicable when setting up **File menu > AutoSave > View Capture tab: Properties**.

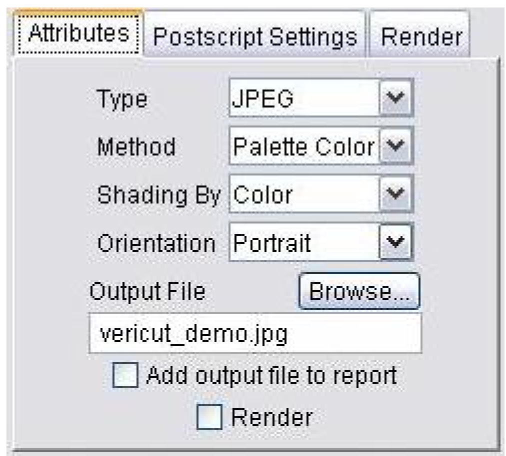
Cancel — Closes the View Capture window without saving setting changes or outputting an image file.

View Capture window, Attributes tab

Location: **File menu > Images > View Capture**

Toolbar short cut for accessing View Capture: 

Features on this tab are used to configure the attributes for formatting captured VERICUT images.



Type — Type of format in which to capture VERICUT images. Printer-ready and electronic formats are available. Options:

- **PostScript —** Postscript format.
- **EPSF —** Encapsulated PostScript File.
- **TIFF —** Tag Image Format File.
- **JPEG —** JPEG compressed format. (JPEG image support based in part on the work of the Independent JPEG Group.)

Method — Method of coloring the captured image. Options:

- **Standard —** black and white only
- **Grayscale —** black and white with shading
- **RGB Color —** Red-Green-Blue values for each pixel
- **Palette Color —** Red-Green-Blue color tables
- **CMYK Color —** Cyan-Magenta-Yellow-Black color tables

Shading By — Method of shading black and white images. Options:

- **Intensity —** Shade by light intensity only, for example: the brightest yellow color is the same shade of gray as the brightest blue color.
- **Color —** Shade by color differences and light intensity.

Orientation — Controls image orientation on the printed page: Landscape or Portrait.

Output File — Use to specify the name of the file to receive image data when the Output button is pressed. Enter the \path\file name in the text field or click on Browse and use the Save File selection window to specify the file.

NOTE: When images are output via **File menu > AutoSave**, the file names specified on the AutoSave, View Capture tab are used instead.

Add output file to report — When toggled "On", and Output is pressed, the output JPEG images are available for use in "standard" VERICUT reports.

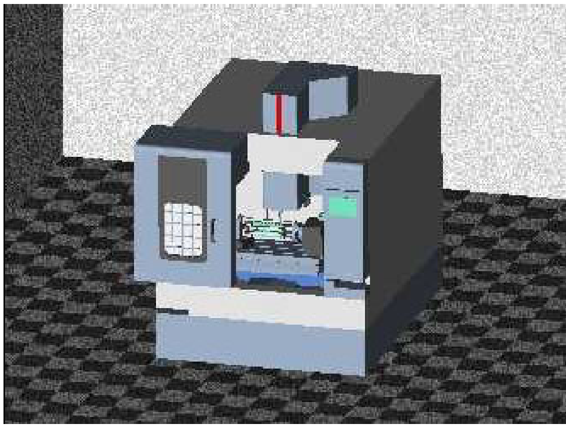
NOTE: **Add output file to report** is not available when setting up **File menu > AutoSave > View Capture tab: Properties**.

Render — Use to toggle the Render feature on and off. When **Render** is toggled "on" and the **Output** button is pressed, the active view will be rendered and a JPEG file will be output with the name specified in the **Output File** field. Pressing the "**Stop**" button during rendering will stop the process.

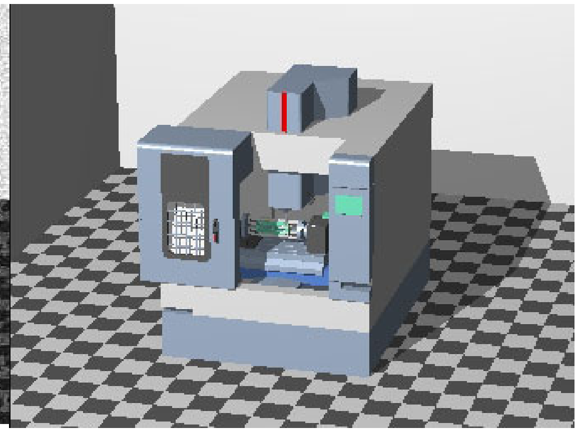
By default, the width and height of the active view is used for the size of the rendered image. Use **Specify image size** on the Render tab to specify a different width and/or height for the rendered image.

- Currently, **JPEG** is the only supported output.
- **Method, Shading By** and **Orientation** are not applicable to **Render**.
- AUTO-DIFF models, Travel Axis, design components/models and Temp Stock components are not supported by the Render feature.

NOTE: Generating a "rendered" image can take considerably longer than generating a standard View Capture image.



Standard View Capture image

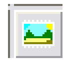


Rendered image

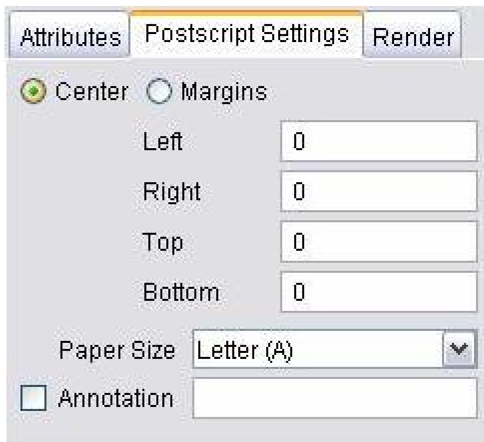
To View Capture window

View Capture window, Postscript Settings tab

Location: **File menu > Images > View Capture**

Toolbar short cut for accessing View Capture: 

Features on this tab provide information about centering and annotating PostScript format images on the printed page.



Center / Margins options — Controls image placement on the printed page. Options:

- **Center —** Center image in the middle of the page.
- **Margins —** Place image according to specified margin dimensions (see below).

Left / Right / Top / Bottom — Margin dimensions, as measured from the corresponding edges of the paper.

Paper Size — Size of paper to print.

Annotation — When active, includes annotated text with the image. Annotation includes the current date, tool path file name, tool path record number being processed, and user specified text.

Tip: If you want to add a note that uses special characters add an "escape" backslash character ("\") before each special character.

Example:

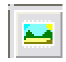
To enter this annotated text => Part Number (1234) ((due date 12/01/00))

Type this in the **Annotation** field => Part Number \(1234\) \(\(due date 12\/01\/00\)\)

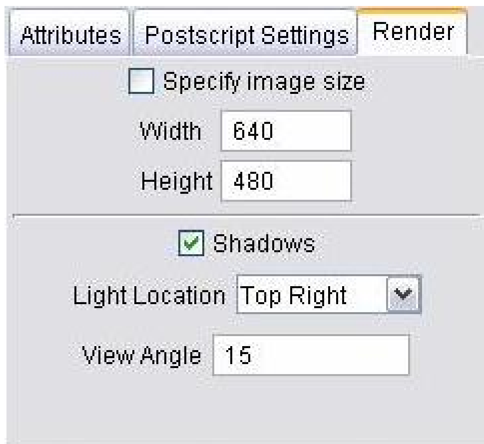
To View Capture window

View Capture window, Render tab

Location: **File menu > Images > View Capture**

Toolbar short cut for accessing View Capture: 

Features on this tab enable you to specify the output size, whether or not to use shadows, and light source location for "rendered" images.



Specify image size — Toggle "On" to use the Width and Height values specified below for creating an image using Render. By default, the width and height of the active view is used for the size of the rendered image.

Width — Use to specify the width for a rendered image.

Height — Use to specify a height for a rendered image.

Shadows — Toggles Shadows On/Off.

Light Location — Use to specify one nine fixed light locations. Choose from: Top Left, Top, Top Right, Left, Center, Right, Bottom Left, Bottom, and Bottom Right.

View Angle — This field enables you too define the view angle. Valid range is from 1 to 80 degrees.

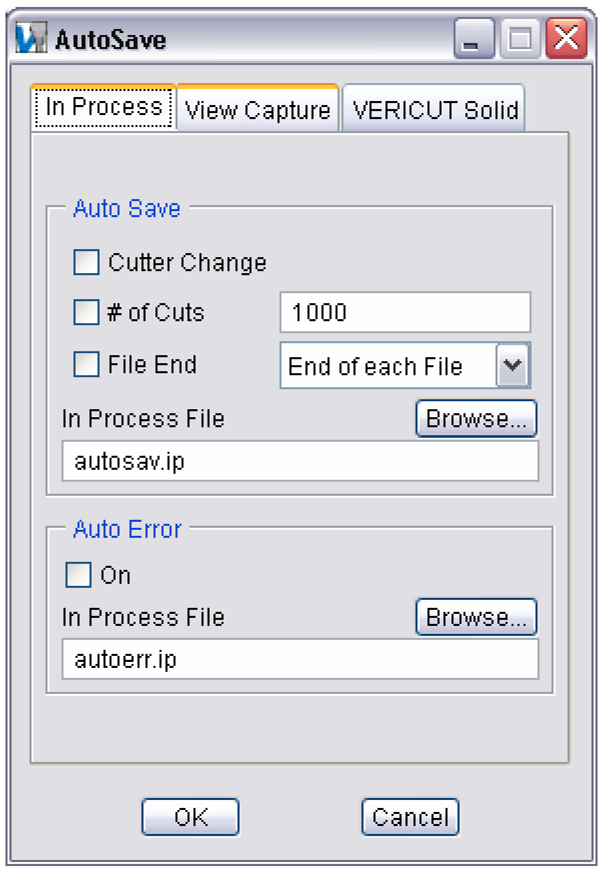
To View Capture window

AutoSave window

Location: **File menu > AutoSave**

Toolbar short cut for accessing AutoSave: 

Opens a window to configure VERICUT to automatically save In Process files, View Capture image files, (such as JPEG, PS, EPSF, or TIFF files) or VERICUT Solid (.vct) files. Saving is based on user specified events or when VERICUT detects errors during tool path processing. Auto-saving is especially useful to save data during batch processing.



[In Process tab](#) — Features on this tab control automatic saving of In Process files.

[View Capture tab](#) — Features on this tab control automatic saving of ShadeCopy image files.

VERICUT Solid tab **—** Features on this tab control automatic saving of VERICUT Solid files.

OK — Saves all tab settings and closes the AutoSave window.

Cancel — Closes the AutoSave window without saving settings.

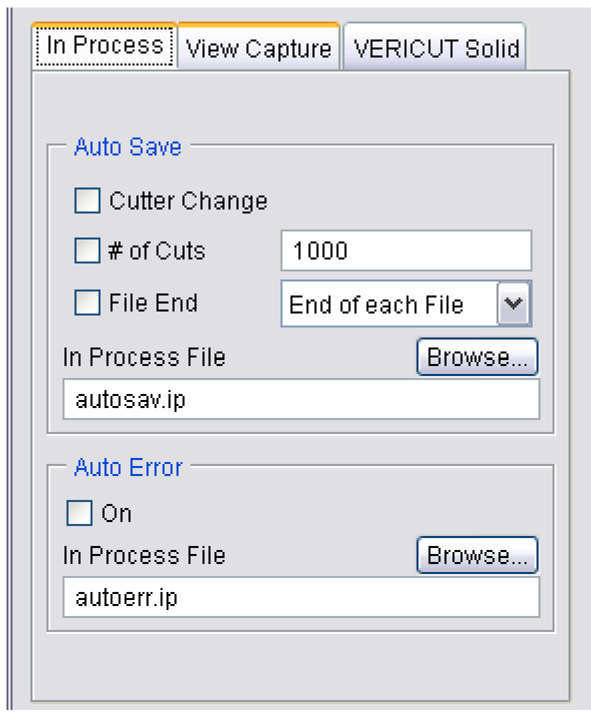
AutoSave window, In Process tab

Location: **File menu > AutoSave window**

Project menu > Output window

VERICUT toolbar short cut: 

Features on this tab control automatic saving of In Process files, or "IP files".



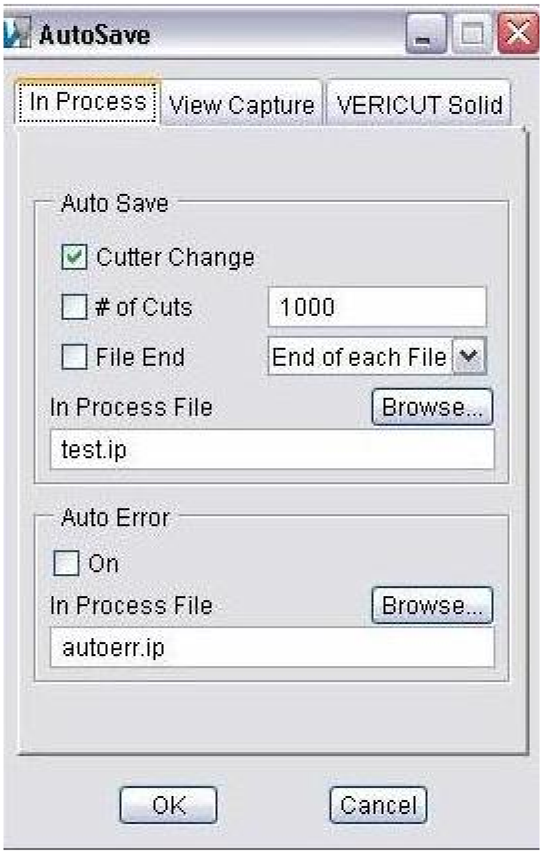
Auto Save options — These options control when VERICUT automatically saves IP files. Click on the box to the left of the event to toggle On/Off. Each selected event causes a file to be saved when that event occurs. Options:

- **Cutter Change —** Save an IP file when the cutter has been changed.
- **# of Cuts —** Save an IP file after a specified number of cuts. Enter the number of cuts in the **# of Cuts** text field.
- **File End —** Save file at the end of tool path processing. Choose either **End of each File** (end of each NC program file), or **End of each Setup**, or **End End** (end of the program) from the pull-down list.

In Process File — Use to specify the base name for IP files saved. Enter the *\path\filename* in the text field or click on **Browse** and use the selection window to specify the file.

Unique file names are maintained by appending the tool path number and line number of the record causing the event to the end of the base file name. Naming convention for auto-saving files is: <IPfilename>n1_n2.ip where "<IPfilename>" is the name specified in the **In Process File** field, "n1"=sequential tool path number, and "n2"=number of the record causing the auto-save event.

The following examples assume the AutoSave (or Output) window configuration shown below.



Example 1: One tool path processed, tool change on record #10 => test1_10.ip

Example 2: Two tool paths processed, tool change on record #10 in both tool paths => test1_10.ip (from tool change in tool path #1), test2_10.ip (from tool change in tool path #2)

Example 3: Two tool paths processed, tool path one tool change on record #10, tool path two tool change on record #5 => test1_10.ip (from tool change in tool path #1), test2_5.ip (from tool change in tool path #2)

Auto Error On — When active, VERICUT automatically saves an IP file when an error is detected.

In Process File — Specifies the base name for IP files saved due to errors. Similar to **In Process File** for the **Auto Save** options (see above), unique file names are maintained by appending the line number of the record causing the error to the end of the specified IP file name.

To AutoSave window

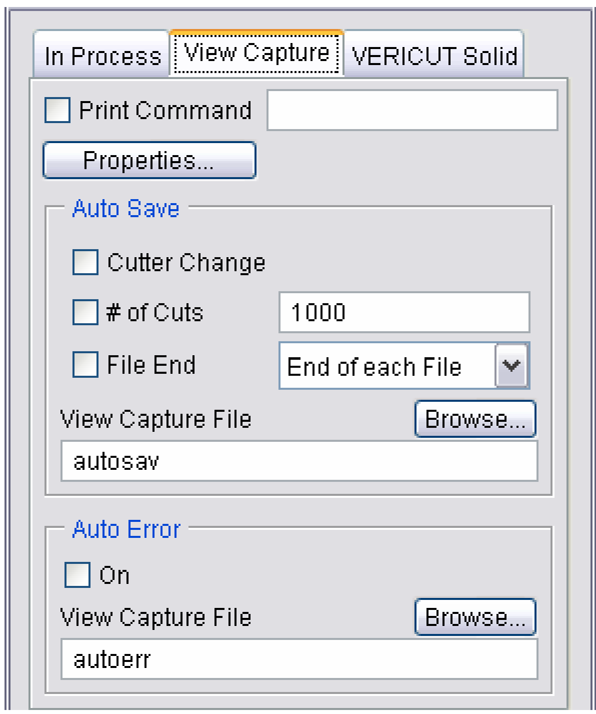
AutoSave window, View Capture tab

Location: **File menu > AutoSave**

Project menu > Output

VERICUT toolbar short cut: 

Features on this tab control automatic saving of View Capture image files (JPEG, PS, EPSF, or TIFF).



Print Command — When toggled "Off", sends graphical data to the specified View Capture image file. When toggled "On", executes the print command that sends raw graphical image data to the printer.

By default, the "prshade" command executes the "prshade" command file containing operating system commands typically used to print images on your computer. If printing fails, correct the entry in the **Print Command** field, or use an ASCII text editor (NOT a word processor) to edit the "prshade" command file to have the proper print command for your computer/printer.

Properties — Opens the **View Capture window** to access settings for file formatting and printing.

Auto Save options

These options control when VERICUT automatically saves View Capture image files. Click on the box to the left of the event to toggle On/Off. Each selected event causes a file to be saved when that event occurs. Options:

- **Cutter Change —** Save a View Capture image file when the cutter has been changed. This feature can also be used to capture images of electrodes used for Die Sinking Simulation when they are changed during the burn process.
- **# of Cuts —** Save a View Capture Image file after a specified number of cuts. Enter the number of cuts in the **# of Cuts** text field.
- **File End —** Save a View Capture image file at the end of tool path processing. Choose either **End of each File** (end of each NC program file), or **End of each Setup**, or **End** (end of the program) from the pull-down list.
- **View Capture File —** Use to specify the base name for View Capture image files saved as a result of Auto Save options described above, when the **Print Command** checkbox is toggled "Off". Enter the *\path\file name* in the text field or click on **Browse** and use the selection window to specify the file. The naming convention used for saved files is as described for IP files-see **In Process tab** for details.

Auto Error options

These options control when VERICUT automatically saves Vie Capture image files due to errors.

- **Auto Error On —** When toggled "On", VERICUT automatically saves a View Capture image file when an error is detected.
- **View Capture File —** Use to specify the base name for View Capture image files automatically saved as a result of errors detected when **AutoError On** is toggled "On", and the **Print Command** checkbox is toggled "Off". Enter the *\path\file name* in the text field or click on **Browse** and use the Save File selection window to specify the file. The naming convention used for saved files is as described for IP files-see **In Process tab** for details.

To AutoSave window

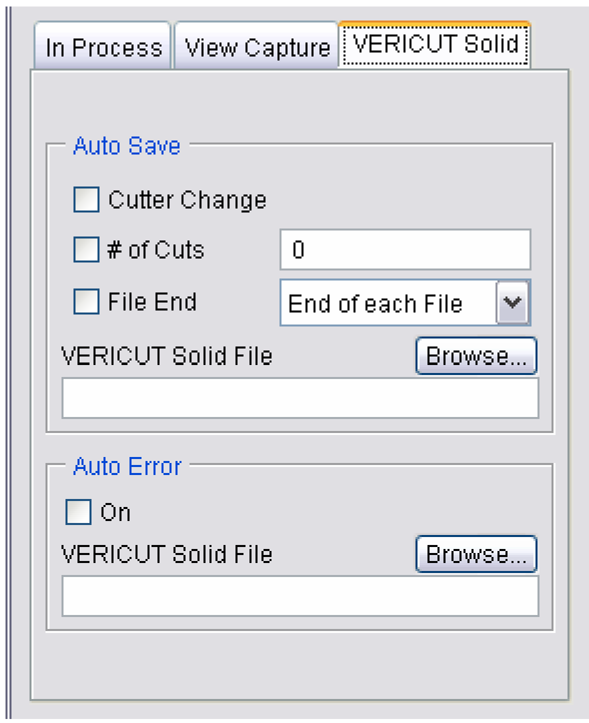
AutoSave window, VERICUT Solid tab

Location: **File menu > AutoSave**

Project menu > Output

VERICUT toolbar short cut: 

Features on this tab control automatic saving of VERICUT Solid (vct.) files.



Auto Save options

Use these options to specify the events when VERICUT should automatically save VERICUT Solid files. Click on the box to the left of the event to toggle On/Off. Each selected event causes a file to be saved when that event occurs. Options:

Cutter Change — Save a VERICUT Solid file when the cutter has been changed.

# of Cuts — Save a VERICUT Solid file after a specified number of cuts. Enter the number of cuts in the # of Cuts text field.

File End — Save a VERICUT Solid file at the end of tool path processing. Choose either **End of each File** (end of each NC program file), or **End of each Setup**, or **End** (end of the program) from the pull-down list.

VERICUT Solid File — Use to specify the base name for VERICUT Solid files saved due to Auto Save options. Enter the *\path\file name* in the text field or click on **Browse** and use the selection window to specify the file.

Auto Error options

Use these options to specify whether or not VERICUT automatically saves VERICUT Solid files when an error is detected.

On — Click on the box to the left to toggle On/Off. When active, VERICUT automatically saves a VERICUT Solid file when an error is detected.

VERICUT Solid File — Use to specify the base name for VERICUT Solid files saved when **AutoError** is active. Enter the \path\file name in the text field or click on Browse and use the selection window to specify the file.

NOTE: The naming convention used for saved VERICUT Solid files is the same as described for IP files-see **In Process tab** for details.

To AutoSave window

Recent Files

Location: **File menu > Exit**

Provides a list of recently opened Project files or User files. Select the desired file from the pull-down list.

List contents are stored in the VERICUT Preferences file.

Exit

Location: **File menu > Exit**

Exits (quits) the VERICUT program. Any files that were opened during the VERICUT session, such as the Log file, Image file, etc. are automatically closed. You can also exit VERICUT by clicking "X" at the top right corner of the VERICUT main window.

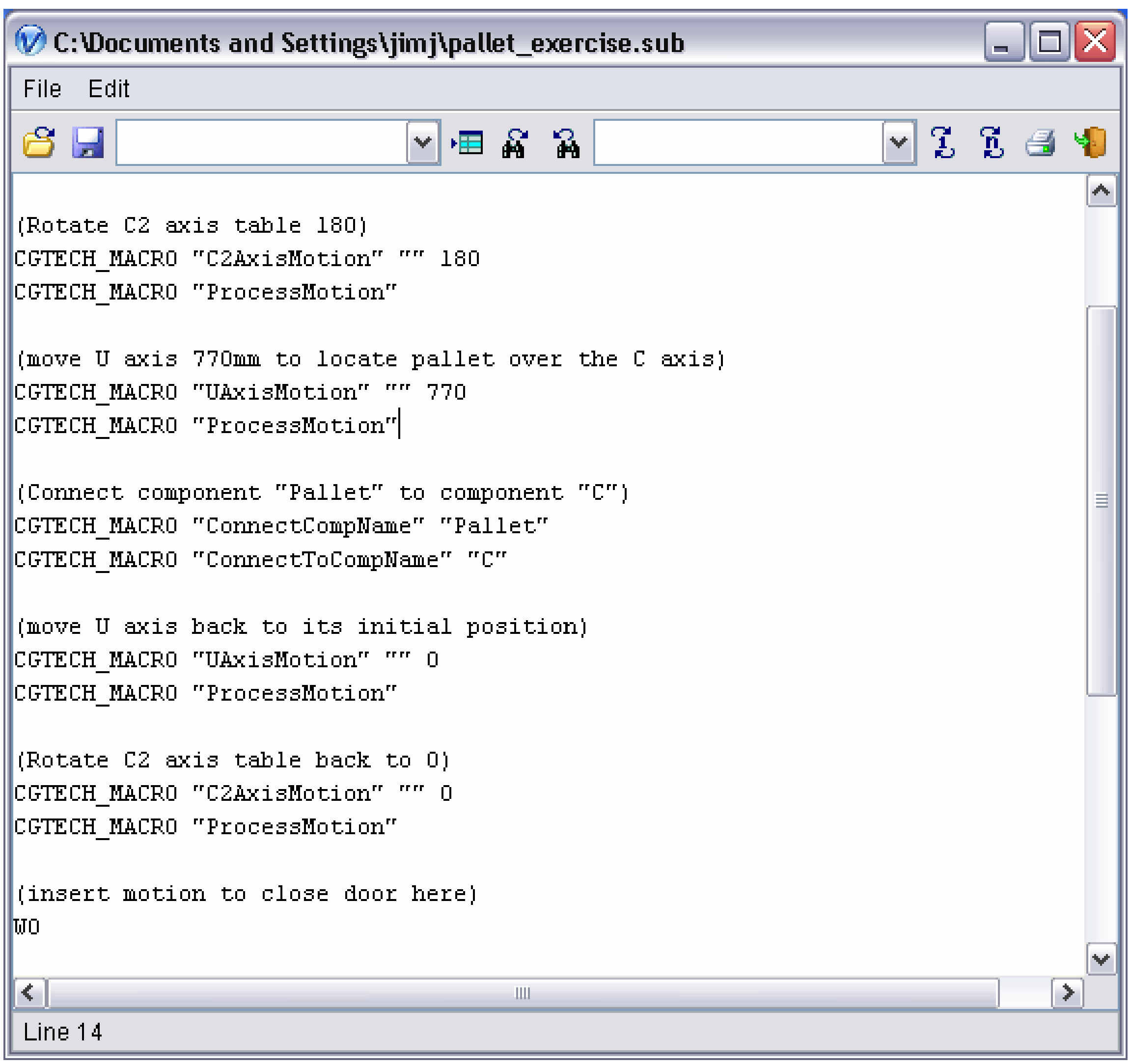
NOTE: Session information ***is not*** automatically saved by VERICUT upon exiting. It is the responsibility of the user to decide if session information should be saved before exiting.

Edit menu

Text File (edit)

Location: **Edit menu > Text File**

Opens a text editor window - just load any ASCII text file and start editing. The following is a general description of the features available for searching and editing text and tool path files throughout VERICUT. The features available in a particular window may vary depending on the purpose of the window that they appear in.



Menu Bar:

The menu bar located at the top of the window provides easy access to major functions. Each menu contains groups of related functions. Left-click on any menu name to display the list of functions available in that menu. Click on the function in the menu you want to use. The name of the current file is also displayed in this area.

File:

Open — Open a file.

Save — Save the current file.

Save As — Save the current file under a different name.

Print — Print the file.

Exit — Close the window.

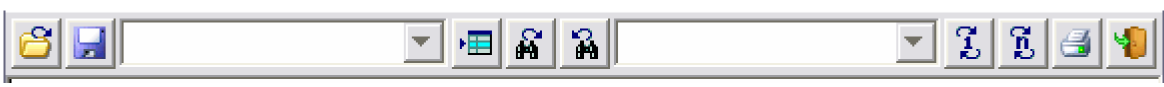
Edit:

Cut — Cuts the highlighted text in the file listing and puts it in the paste buffer.

Copy — Copies the highlighted text in the file listing to the paste buffer.

Paste — Puts the contents of the paste buffer at the location of the cursor in the file listing.

Icon Bar:

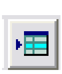


The items in the Icon Bar enable you to search for, and/or replace, specific items in the file listing, print the file, or exit the window. Moving the cursor over the icon will display name of the option. Each feature (from left to right) is described below.

 **Open File —** Opens a file selection window enabling you to select another file.

 **Save File —** Save the current file.

Line Number or Search Text — Use this text field to enter a line number, or a string of text, to search for.

 **Goto Line Number —** Moves the cursor in the file listing to the line number specified in the Line Number or Search Text field.

Search Forward — Searches forward in the file listing for the text string specified in the Line Number or Search Text field.

Search Backward — Searches backward in the file listing for the text string specified in the Line Number or Search Text field.

Replacement Text — Use this text field to enter a "replacement text" string.

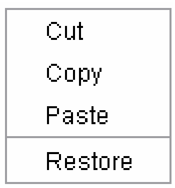
Replace One — Replace one occurrence of text string in the Line Number or Search Text field (or the highlighted text in the file listing) with the text string in the Replacement Text field.

Replace All — Replace all occurrences of the text string in the Line Number or Search Text field with the text string in the Replacement Text field.

Print — Print the file.

Close Window — Closes the window.

Shortcut: Right-click in the Text File Edit window to display the following menu:

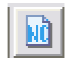


These features provide the same functionality as those available under Edit in the menu bar.

NC Program (edit)

VERICUT Users:

VERICUT Location: **Edit menu > NC Program**

Toolbar short cut for accessing the NC program editor: 

Selecting **Edit menu > NC Program** displays a list of all NC programs and subroutines in the project file. Choosing one of the NC program files/subroutines opens a text editor window with the selected NC program file/subroutine file loaded and ready for editing. Standard text editing features are provided, such as: copy/cut, paste, search, etc.

Mold and Die Users:

Mold and Die Location: **Analyze, View Files, Print page > Edit the Tool Path File**

Notebook Feature: 

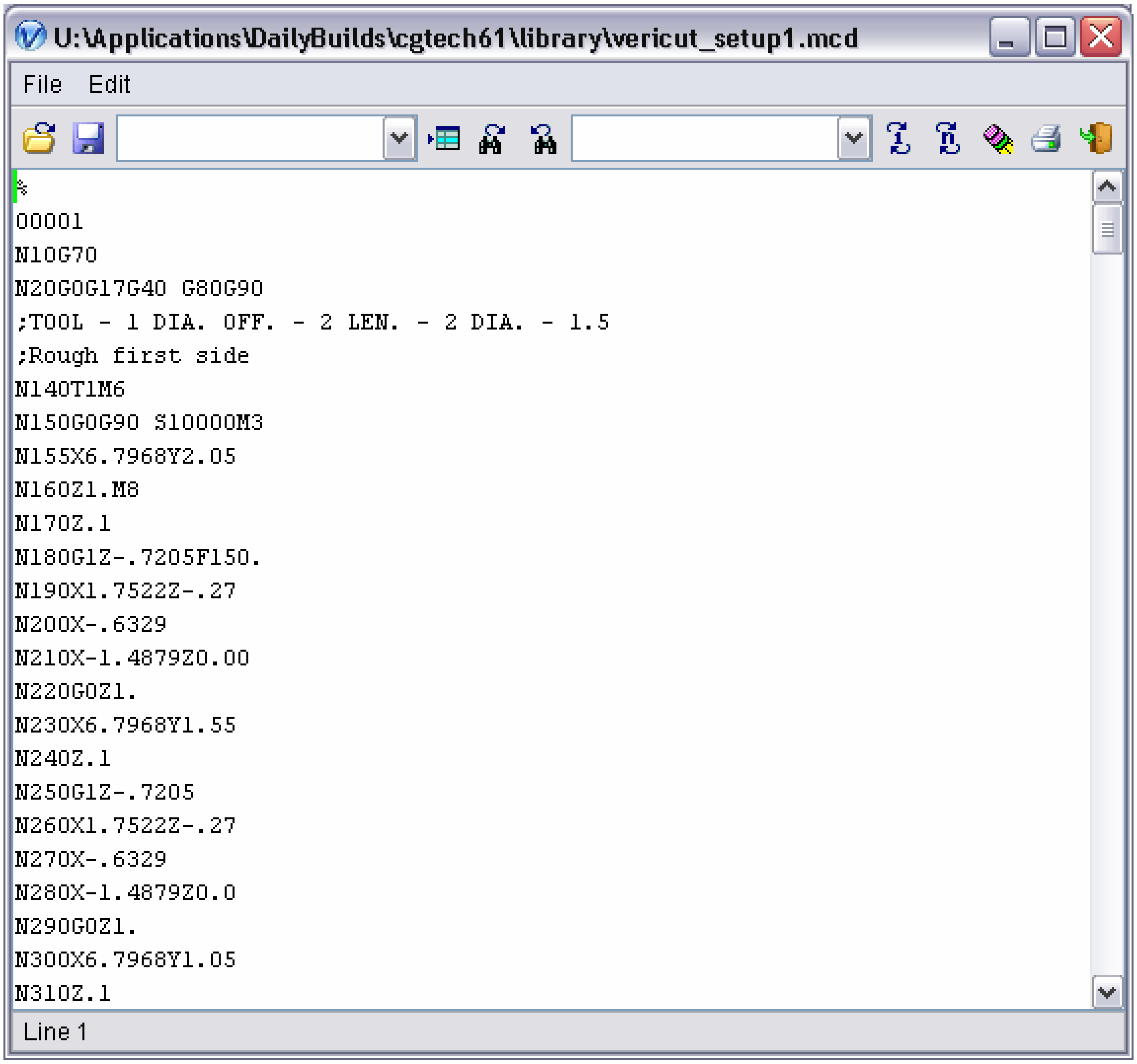
Edit the Tool Path File opens a text editor window with the "current" tool path file loaded and ready for editing.

Cutter Grinder Users:

Cutter Grinder Location: **Analyze, View Files, Print page > Edit Grinding Program**

Notebook Feature: 

Edit Grinding Program opens a text editor window with the "current" grinder program file loaded and ready for editing.



WARNING: Editing the NC program file can cause undesirable results, especially when edits are made to portions of the tool path which have already been processed by VERICUT. These edits can cause the tool path record number pointer to be out of sync with the edited tool path file. In these cases it is recommended to rewind the tool path file after editing, and restart the simulation with the last unedited record prior to the first addition/deletion.

Menu Bar:

The menu bar located at the top of the window provides easy access to major functions. Each menu contains groups of related functions. Left-click on any menu name to display the list of functions that are available in that menu. Click on the function in the menu you want to use. The name of the current file is also displayed in this area.

File:

Open — Open a file.

Save — Save the current file.

Save As — Save the current file under a different name.

Print — Print the file.

Exit — Close the window.

Edit:

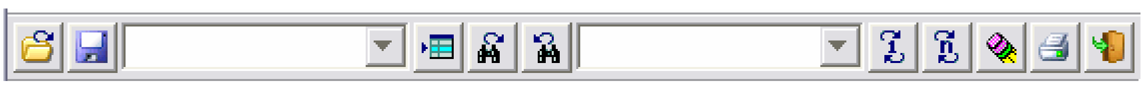
Cut — Cuts the highlighted text in the file listing and puts it in the paste buffer.

Copy — Copies the highlighted text in the file listing to the paste buffer.

Paste — Puts the contents of the paste buffer at the location of the cursor in the file listing.

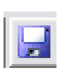
Restore — Restores the NC program to its un-edited state.

Icon Bar:

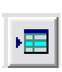


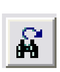
The items in the Icon Bar enable you to search for, and/or replace, specific items in the file listing, print the file, or exit the window. Moving the cursor over the icon will display name of the option. Each feature (from left to right) is described below.

 **Open File —** Opens a file selection window enabling you to select another file.

 **Save File —** Save the current file.

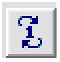
Line Number or Search Text — Use this text field to enter a line number, or a string of text, to search for.

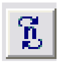
 **Goto Line Number —** Moves the cursor in the file listing to the line number specified in the Line Number or Search Text field.

 **Search Forward —** Searches forward in the file listing for the text string specified in the Line Number or Search Text field.

 **Search Backward —** Searches backward in the file listing for the text string specified in the Line Number or Search Text field.

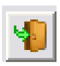
Replacement Text — Use this text field to enter a "replacement text" string.

 **Replace One —** Replace one occurrence of text string in the Line Number or Search Text field (or the highlighted text in the file listing) with the text string in the Replacement Text field.

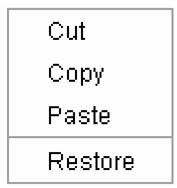
 **Replace All —** Replace all occurrences of the text string in the Line Number or Search Text field with the text string in the Replacement Text field.

 **Undo Highlighted Changes —** Restores the highlighted changes to the original state.

 **Print —** Print the file.

 **Close Window —** Closes the window.

Shortcut: Right-click in the NC Program Edit window to display the following menu:



These features provide the same functionality as those available under Edit in the menu bar.

Color window

VERICUT Users:

VERICUT Location: **Edit menu > Colors**

Toolbar short cut for accessing the Color window: 

Mold and Die Users:

Mold and Die Location: **Other Settings page > Colors**

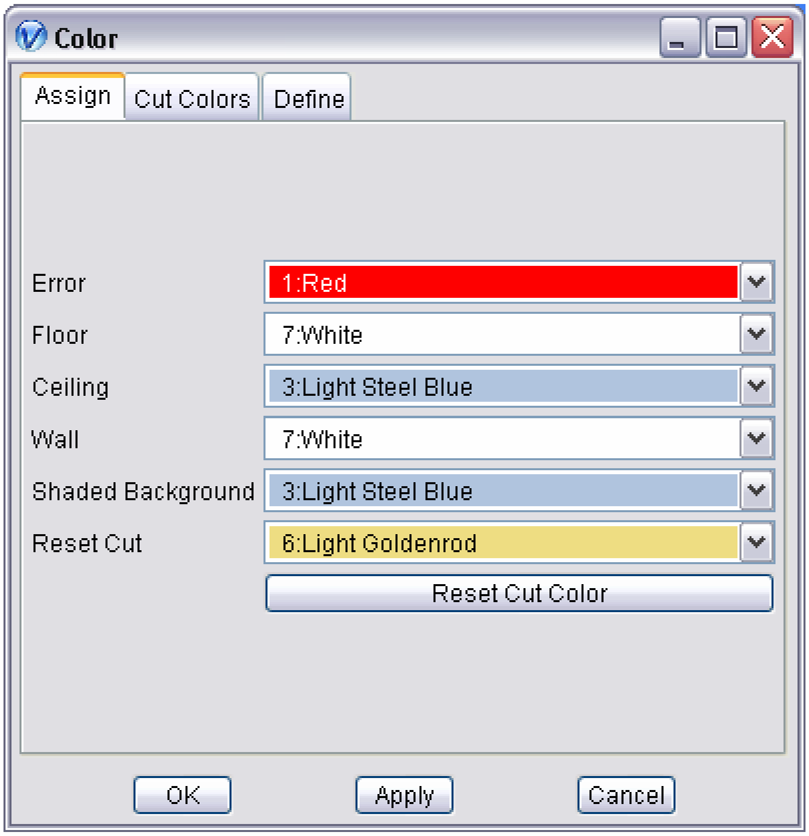
Notebook Feature: 

Cutter Grinder Users:

Cutter Grinder Location: **Other Settings page > Colors**

Notebook Feature: 

Colors opens a window to define the colors seen in VERICUT. Colors are changeable at any time. The affect of changing colors is immediate.



Assign tab — Features on this tab are used to assign Error, Reset Cut and Background colors.

Cut Colors tab — Features on this tab assign colors to machine cuts made in the workpiece.

Define tab — Features on this tab define colors seen in VERICUT, and control light brightness.

OK — Saves tab settings and closes the VERICUT Color window.

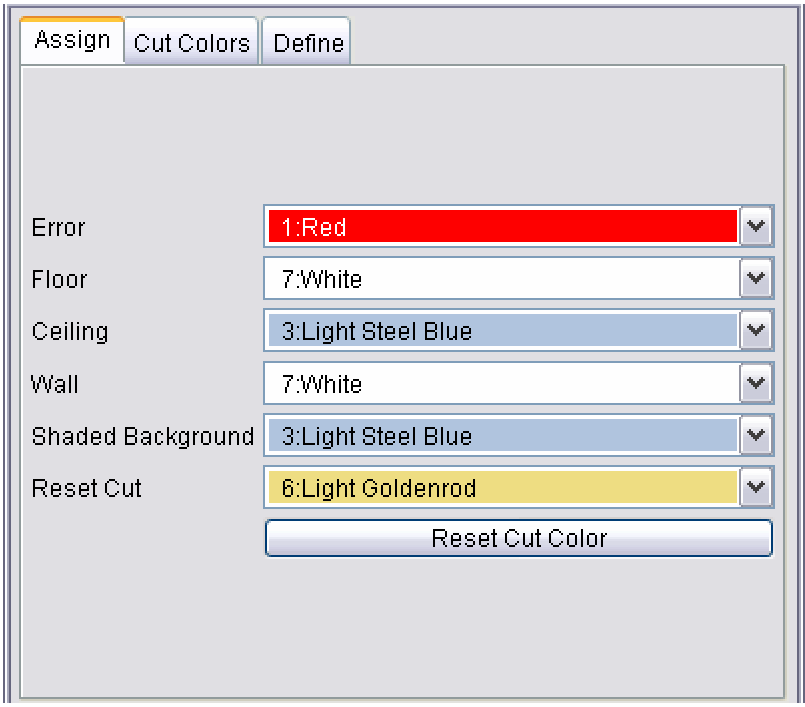
Apply — Saves tab settings and leaves the VERICUT Color window open.

Cancel — Closes the VERICUT Color window without saving tab settings.

Also see "**Using Colors in VERICUT**", in the *Using VERICUT* section, in the *CGTech Help Library*.

Color window, Assign tab

Features on this tab are used to assign Error, Reset Cut and Background colors.



Error — Color for highlighting unsafe machining conditions (default: "Red"). The "error color" will help you identify the following potentially dangerous situations:

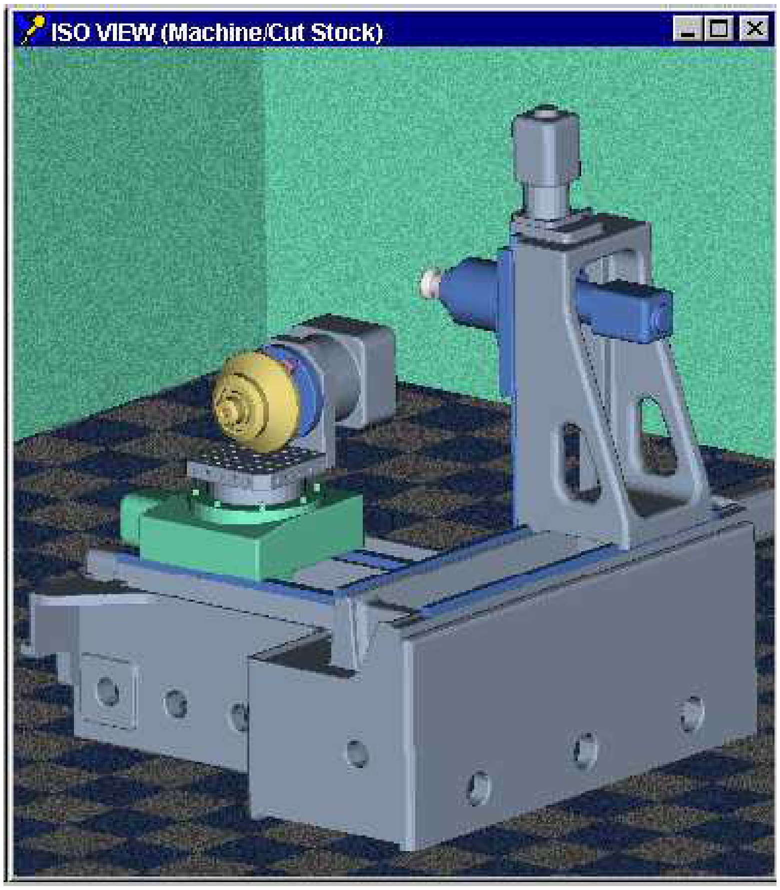
Material removed while cutting too fast (known as a "fast feed" error)

Cuts in clamps, fixtures, bolts, and other hardware holding the workpiece

Material removed by non-cutting portions of the tool, such as: tool holders, extensions, etc.

Gouges in a design model (via AUTO-DIFF)

Floor, Ceiling, Wall — Colors of the floor, ceiling and walls displayed in a machine view when **Background** is set to **Walls** in the **View menu > Attributes window**. A sample machine is shown below with the floor, ceiling and walls in the background. The tile affect on the floor is created by alternating the **Floor** and **Ceiling** colors.



Shaded Background — Use to specify the color that will be displayed when the view Background attribute is set to Shaded.

Reset Cut — Color applied to all cuts in the workpiece when a **Reset Cut Color** operation is performed (see below). By default this color is set the same color as the stock model to give the appearance of uncut stock, however, any defined color can be selected.

Reset Cut Color — When selected, the colors of all cuts in the workpiece are changed to the single color specified by **Reset Cut** (see above).

To Color window

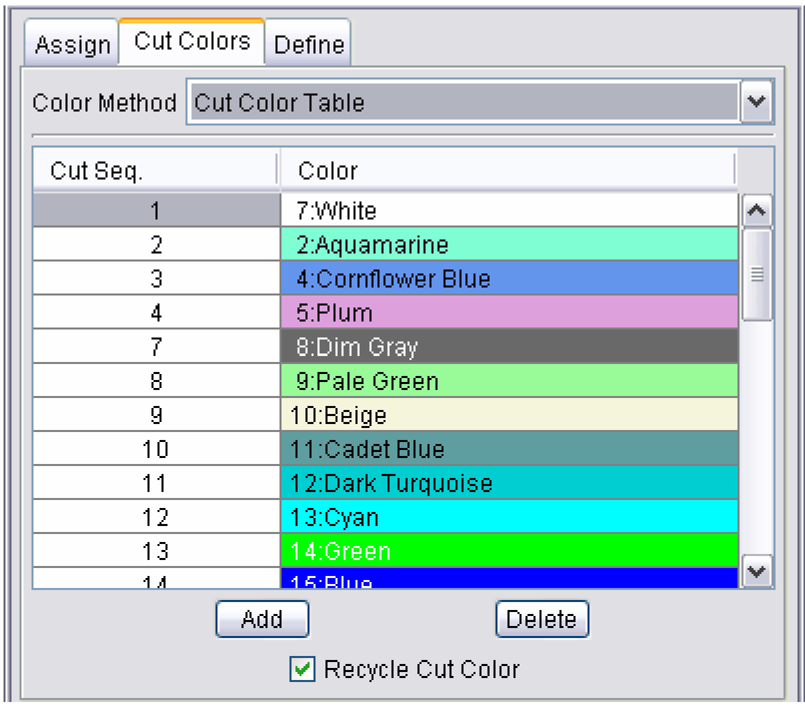
Color window, Cut Colors tab

Features on this tab assign colors to the machine cuts made in the workpiece. The items displayed on this tab will vary depending on the **Color Method** selected.

Color Method

Method of determining colors used to shade machined cuts. Options are:

Cut Color Table — Cut color is determined by the Cut Sequence/Color list defined in the Cut Color Table.



Cut Seq. / Color list — List of colors used for material cut with a safe cutting feed rate (less than the Fast Feed threshold). The cut color list associates colors with cut sequences in the NC program file. A new cut sequence is assumed when a tool with a different shape is loaded for cutting. All portions of the cutting tool shape are considered for differences. When the number of cut sequences exceeds the number of cut colors defined in the table, **Recycle Cut Color** determines the cut color for the remaining tools (see below).

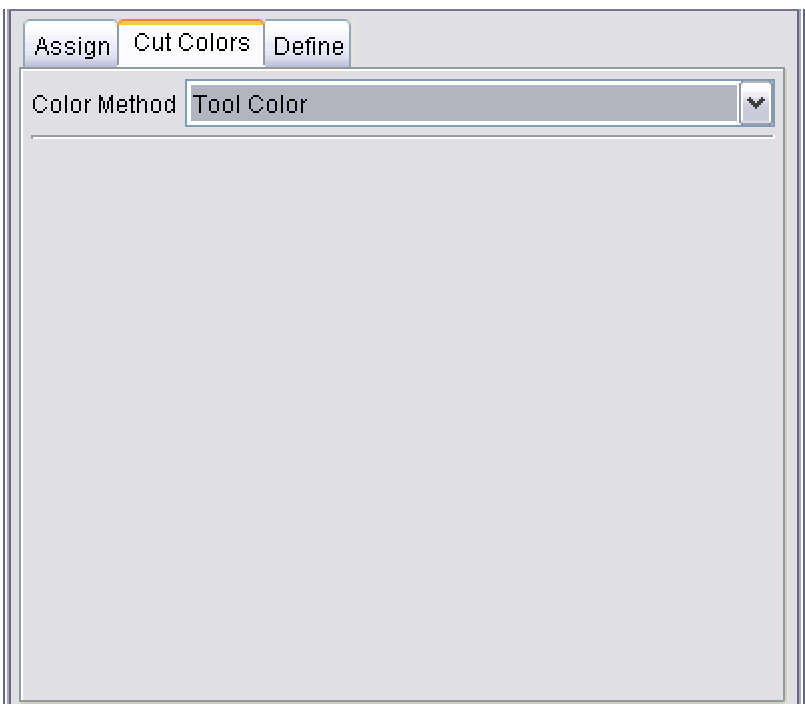
Double-click on the cut sequence number in the table to edit it. Double-click on a color in the table and select from the pull-down color list to change a color. Add or remove colors from the pull-down list using the Shade Colors list feature on the **Color window, Define tab**.

Add — Use to add Cut Sequences/Colors to the list.

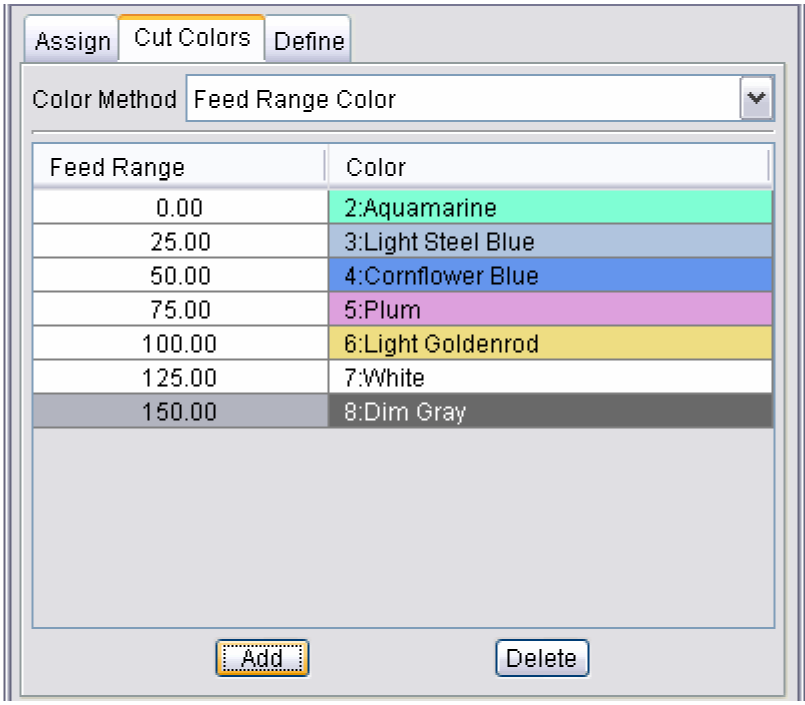
Delete — Use to delete Cut Sequences/Colors from the list.

Recycle Cut Color — When selected, recycles colors in the cut color list when the tool path contains more tools than cut colors defined. For example, assume that 4 cut colors are defined. When tool 5 is encountered VERICUT recycles cut color 1, tool 6 uses cut color 2, and so on. When this option cleared, the last defined cut color is used for cut sequences that follow the last defined cut sequence color. Using the example above, tool 4, 5 and 6 would receive the cut color defined for cut sequence 4.

Tool Color — Cut color is determined by the tool color property stored in a VERICUT Tool Library.



Feed Range Color — Cut color is determined by the feed rate programmed in the NC program file.



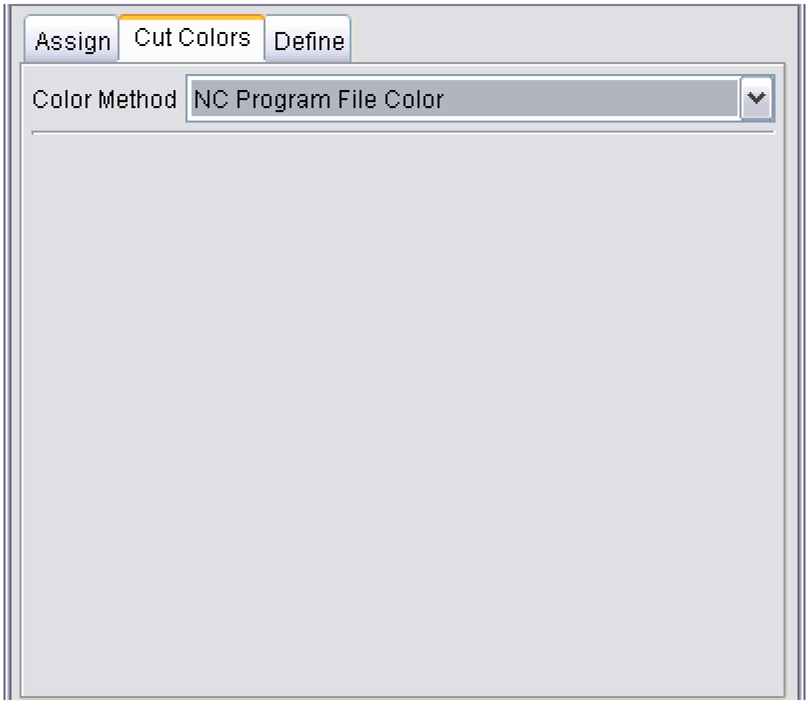
Feed Range / Color list — List of colors used for material cut within a specified feed rate range. The cut color list associates colors with feed rates in the NC program file. Using the sample table shown above, cuts using a feed rate between 0 and 25 will be displayed Aquamarine. Cuts using a feed rate between 25 and 49 will be displayed Light Steel, and so on.

Double-click on the Feed Range number in the table to edit it. Double-click on a color in the table and select from the pull-down color list to change a color. Add or remove colors from the pull-down list using the **Shade Colors** list feature on the **Color window, Define tab**.

Add — Use to add Feed Ranges/Colors to the list.

Delete — Use to delete Feed Ranges/Colors from the list.

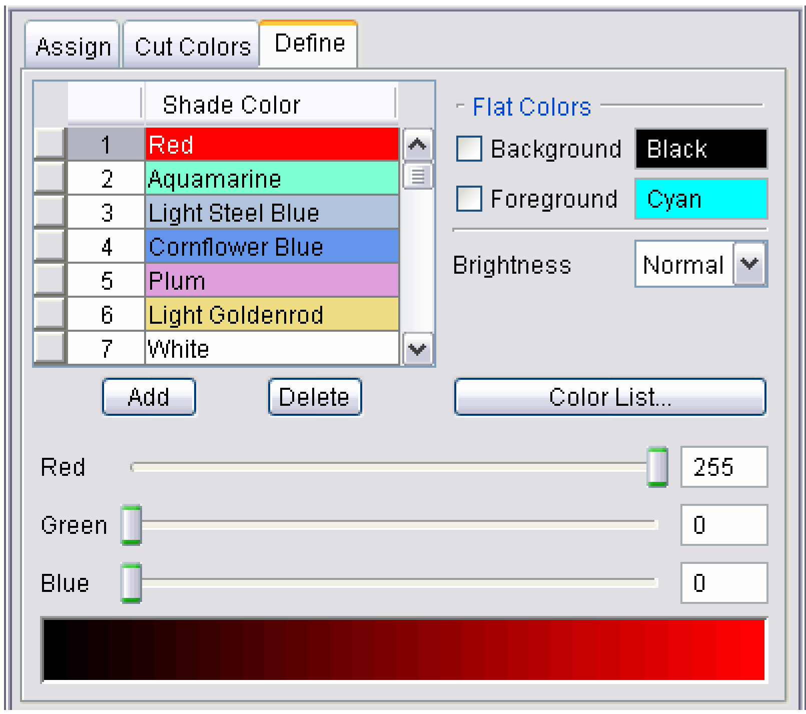
NC Program File Color — Cut color is determined by the Cut Sequence/Color list defined in the Cut Color Table method described above except that the Cut Sequence changes as the NC program file changes instead of when the tool changes. If a Project file only uses one NC program file, then all cuts will be the same color.



To Color window

Color window, Define tab

Features on this tab define colors seen in VERICUT, and control light brightness.



Shade Color list — List of colors that can be used to shade solid objects seen in VERICUT, such as: models, cutting tools, and machined cuts (maximum 128 colors). The number of shade colors also affects shade quality. The color band at the bottom of the window displays the shade variations of the selected color in the list, as impacted by the total number of defined colors. Once defined, assign the colors to solid objects using the features on the **Assign tab** and the **Cut Colors tab**.

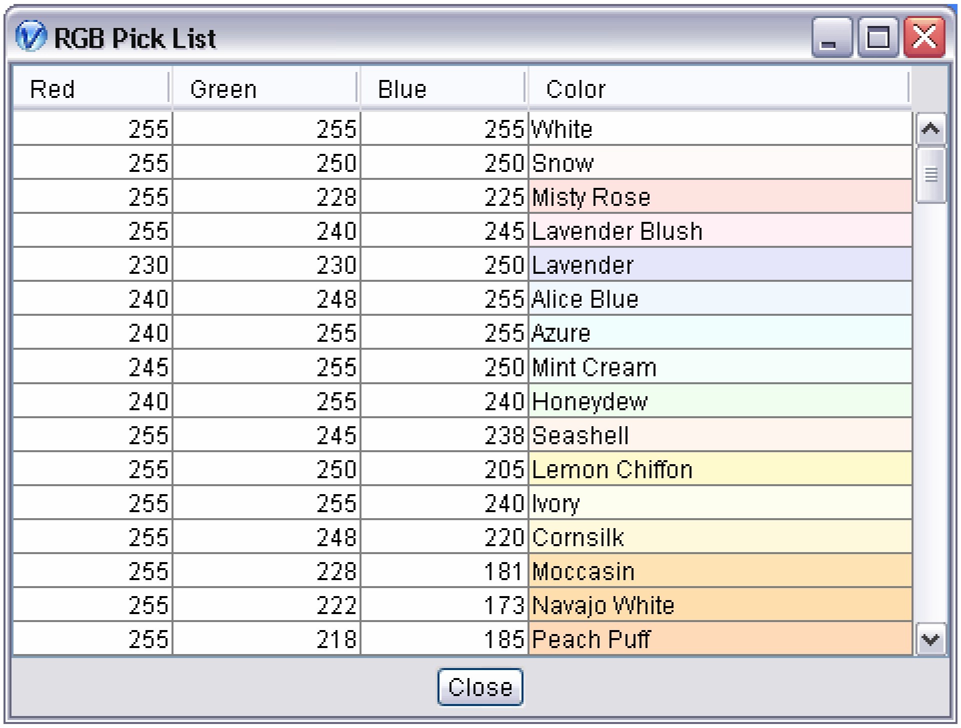
Add — Use to add colors to the Shade Color list.

Delete — Use to delete colors from the Shade Color list.

Red-Green-Blue slidebars — Control the amount of red, green, and blue color components mixed to define a color. The exact value is shown in the field to the right of each slidebar (0=none to 255=maximum).

Color List — Opens a window of pre-mixed colors selectable for background, foreground, and shade colors. Click on the desired color.

RGB Pick List window



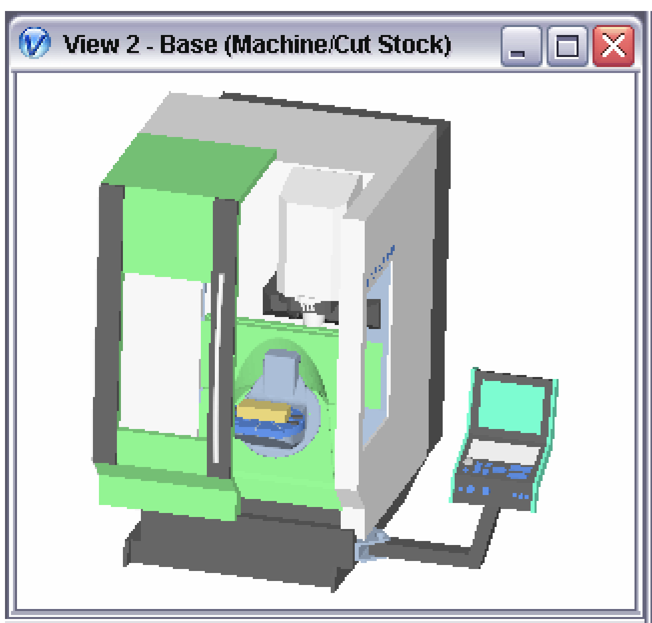
RGB Pick List

Red	Green	Blue	Color
255	255	255	White
255	250	250	Snow
255	228	225	Misty Rose
255	240	245	Lavender Blush
230	230	250	Lavender
240	248	255	Alice Blue
240	255	255	Azure
245	255	250	Mint Cream
240	255	240	Honeydew
255	245	238	Seashell
255	250	205	Lemon Chiffon
255	255	240	Ivory
255	248	220	Cornsilk
255	228	181	Moccasin
255	222	173	Navajo White
255	218	185	Peach Puff

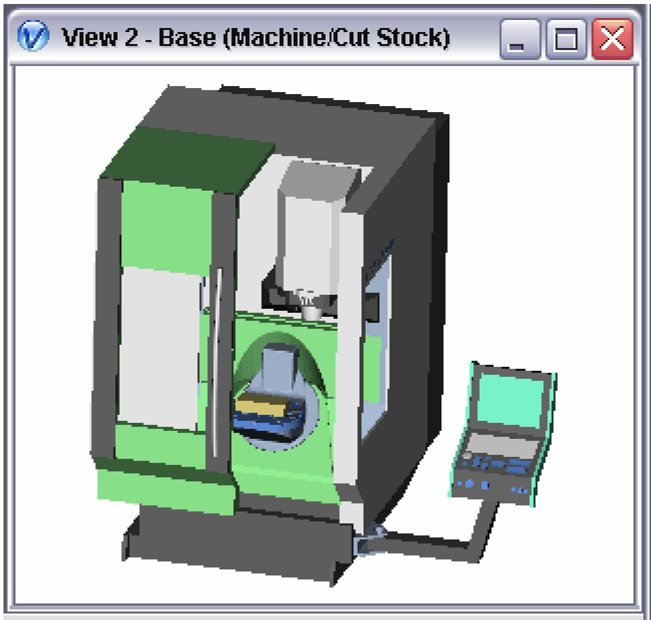
Close

Brightness — Controls the amount of light for shading solid objects. Options are:

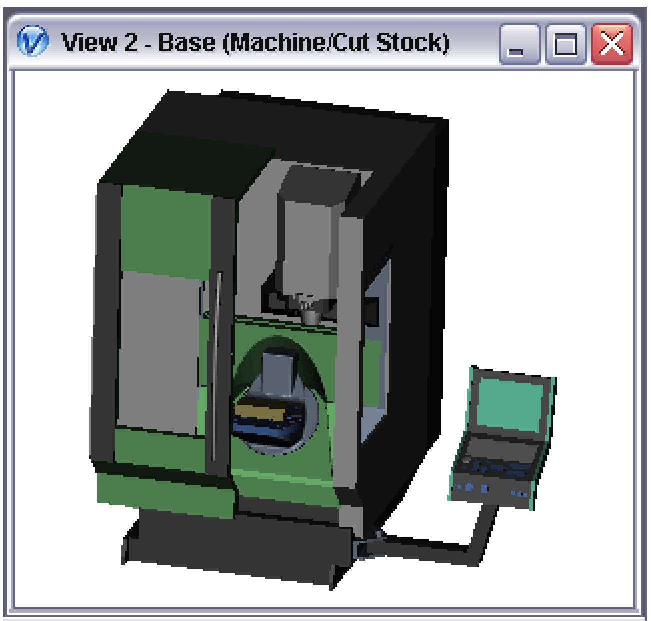
High



Normal



Low



Flat Colors group:

Background — Select this option to change the color of the graphics area background when **View menu > Attributes: Background** is set to "Flat". You can also change the background to display a shaded color, or machine views can display ceiling, floor, and walls.

Foreground — Select this option to change the color of items drawn on top of the VERICUT display, for example: coordinate system axes, lines indicating the bounding region of selected models/components, etc.

The Background and Foreground colors should be different, or foreground-colored items may "blend in" with the background.

To Color window

View Menu

Also see "**Working with VERICUT Views**", in the *Using VERICUT* section, in the *CGTech Help Library*.

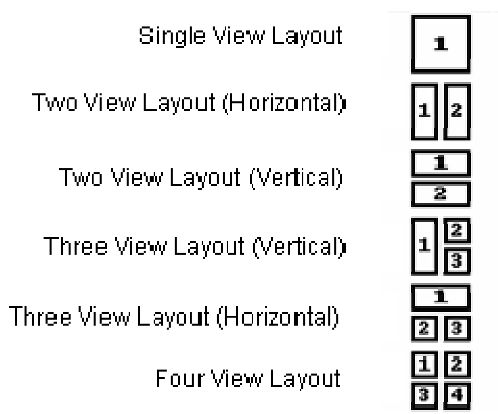
Layout (view)

Location: **View menu > Layout**

The options in the **Layout menu** control how many views of the VERICUT model are seen, and how they are arranged. All views are contained within the VERICUT main window. The size and orientation of the model in each view is individually controlled. View port size and layout are saved in the user file.

One view is considered by VERICUT to be the "active view". A view is made active by clicking in it, and is designated by a colored window header. Many VERICUT functions are effective only in the active view, including most viewing functions.

Standard — Sets views of the VERICUT model to one of the standard view layout choices. By default standard view layouts are tiled, but you can change this using other functions in the Layout menu, or via standard window management features, such as dragging, resizing, etc.



Tip: You can also use the corresponding view layout icons in the Toolbar.

Add View — Adds a view. You can add as many views as desired. What is seen in a view (workpiece, machine, etc.) is determined by the view attributes (ref. View Attributes window, also in the *VERICUT Help* section.

Shortcut: You can quickly change the attributes or orientation of a view by right-clicking in the view, and selecting from the displayed menu. See the **Graphics Area Right Mouse Button Shortcut Menus**, in the Getting Started with VERICUT section of VERICUT Help for more information.

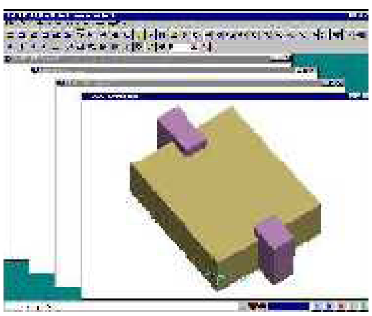
Delete View — Deletes a view. You can also delete a view via clicking "X" at the top right corner of the view window.

Cascade — Cascades views in an overlapping arrangement. Note that cascaded view layouts are not saved by VERICUT.

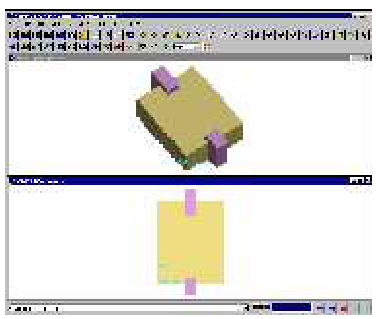
Tile Horizontally — Tiles views in a horizontal arrangement.

Tile Vertically — Tiles views in a vertical arrangement. Examples follow.

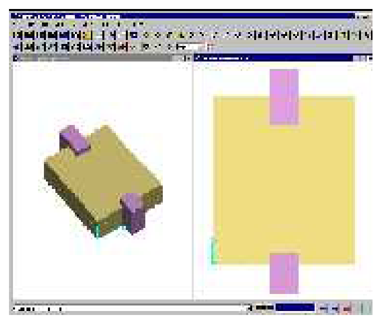
Cascaded



Tiled horizontally



Tiled vertically



View to Back — Moves the "active" view to the back. This function can also be accessed using the right mouse button in the active view.

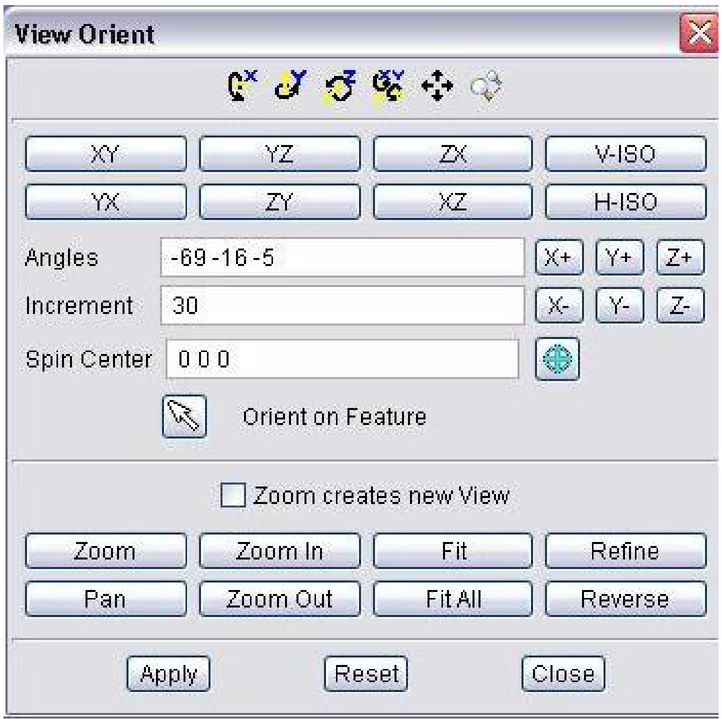
View to Front — Moves the selected view to the front. Selecting "View to Front" displays a list of all available views allowing you to select the desired view to be moved to the front even if it is obscured by other views in the graphics area.

Always in Front — Makes the active view remain "always in front" until it is specifically moved to the back or until another view is specified as "always in front". This function can also be accessed using the right mouse button in the active view.

View Orient window

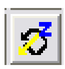
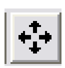
Location: **View menu > Orient**

Opens a window to orient the view, including: rotate, zoom, pan, reverse, etc. You can view machining from any angle or distance, and change the view at any time during the simulation. Views of the workpiece, NC machine, or cutting tools seen in the Tool Manager are oriented in the same manner, as described by the sections that follow. In general, click in a view to make it active, then change view orientation as required. View rotations are described with respect to the "view coordinate system".



Dynamic Viewing Options:

These options use mouse actions to dynamically orient the object in view. Mouse actions are different, depending on the active option-see the table below for details. Dynamic viewing options are also available on the VERICUT Toolbar.

Toolbar Icon:	Name:	Action:
	**Dynamic X Rotation**	Horizontal rotation- drag mouse up/down in direction to rotate
	**Dynamic Y Rotation**	Vertical rotation- drag mouse left/right in direction to rotate
	**Dynamic Z Rotation**	Screen plane rotation- drag mouse left to rotate CCLW, right for to rotate CLW
	**Dynamic XY Rotation**	Horizontal/vertical combined rotation- drag mouse up/down/left/right in direction to rotate
	**Dynamic Pan**	Pan/translate- drag mouse in the direction pan
	**Dynamic Zoom**	Zoom/magnify- drag mouse up to zoom in, down to zoom out

Shortcuts:

1. Use the following mouse features to manipulate the view with a single mouse button.

   Dynamic Rotate — Left mouse button, drag.

   Dynamic Pan — Right mouse button, drag.

   Dynamic Rotation — Rotate the thumbwheel toward you to make the displayed image larger, away from you to make the displayed image smaller.

   See **Dynamic Zoom, Pan and Rotate** in the *Getting Started with VERICUT* section for additional information.

   NOTE: Dynamic Controls (**View menu** > **Dynamic Controls)** must be set to VERICUT.

2. The following keys provide instant access from the keyboard to dynamic viewing options (press and hold keys while dragging):

   Dynamic Zoom — <Ctrl>, drag

   Dynamic Pan — <Shift>, drag

   Dynamic XY — <Ctrl> + <Shift>, drag

   Mouse actions are the same as described above.

Standard drawing view buttons- XY, YX, YZ, ZY, ZX, XZ, V-ISO, H-ISO — Orients the object in the selected standard drawing view and automatically fits the object in the view.

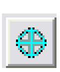
Rotation Options:

Angles — Absolute rotation about the view coordinate system X Y Z axes, respectively. The three values are separated by spaces.

Increment — Degrees of incremental rotation to apply when a rotation button is pressed (see below).

Rotation buttons (X+/X-, Y+/Y-, Z+/Z-) — axis and direction in which to apply incremental rotation specified by the Increment value.

Spin Center — Specify a point to be used as the center of rotation during dynamic rotations. Click in the **Spin Center** field so that it becomes active, then either enter XYZ values (separated by spaces) in the field and click on the **Apply** button or click on a position in the view to specify the **Spin Center** point. The values are applied to the "active" view. If a workpiece view is the active view, the coordinates are relative to the origin of the workpiece coordinate system. If the "active" view is a machine view, the coordinates are relative to the origin of the machine coordinate system.

Press  to display the **Spin Center** marker and activate the point for the active view. To de-activate Spin Center point, press the button again. You can specify a different rotation point for each view. When you close the **View Orient** window, the **Spin Center** markers will no longer be displayed but the activated Spin Centers will remain active. When you use one of the dynamic rotation features described above in a view with an active Spin Center, the Spin Center point will be used as the center of rotation instead of the view coordinate system origin.

Shortcut: You can quickly change the spin center location by right-clicking in the view, select Spin Center from the displayed menu, and then select the desired position in the graphics area. See the **Graphics Area Right Mouse Button Shortcut Menus**, in the Getting Started with VERICUT section of VERICUT Help for more information.

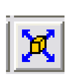
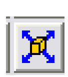
Orient on Feature — Resets the view, normal to the feature selected in the graphics area. The modified view is centered on the position of the selection vector.

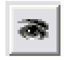
Zoom creates new View — When toggled on, creates a new view for the zoomed area instead of modifying the current view. Also changes the icon in the toolbar to the Zoom (create new view mode) icon.

Shortcut: You can quickly toggle between the two Zoom modes (standard or create new view) by right-clicking on the Zoom to Box icon  , or , in the toolbar.

Static Viewing Options:

These buttons use various mouse actions to zoom, fit, and reverse the object in view. See the table below for details. Static viewing options are also available on the VERICUT Toolbar.

Button name:	Toolbar Icon:	Action:
Zoom (standard mode)	 (Zoom to Box)	Left-click and move mouse to trap area to zoom (press <Esc> to interrupt rubber-banding)
Zoom (with Zoom creates new View toggled On)	 (right click on Zoom to Box)	Similar to Zoom (standard mode) described above except a new view is created to display the zoomed area.
Zoom In / Zoom Out	 	Zooms in/out approx. 20% each time clicked
Fit	 (left click)	Fits object in active view
Fit All	 (right click)	Fits objects in all views

Refine	 (Refine Display)	Refines the display to improve image quality
Reverse		Reverses viewing direction, as if you stood behind the object (see also: "**reverse**" VERICUT command record)
none	 (Last Refine Display)	Restores the last refined display

Shortcuts:

1. You can quickly reverse or orient the model in one of the defined views by right-clicking in the view and selecting from the displayed menu. See the **Graphics Area Right Mouse Button Shortcut Menus**, in the Getting Started with VERICUT section of VERICUT Help for more information.
2. Use the thumb wheel on the mouse to **Zoom to Box**. Click on the thumb wheel and drag the mouse to define the box to zoom to. The zoom takes place when you release the thumb wheel.

   See **Dynamic Zoom, Pan and Rotate** in the *Getting Started with VERICUT* section for additional information.

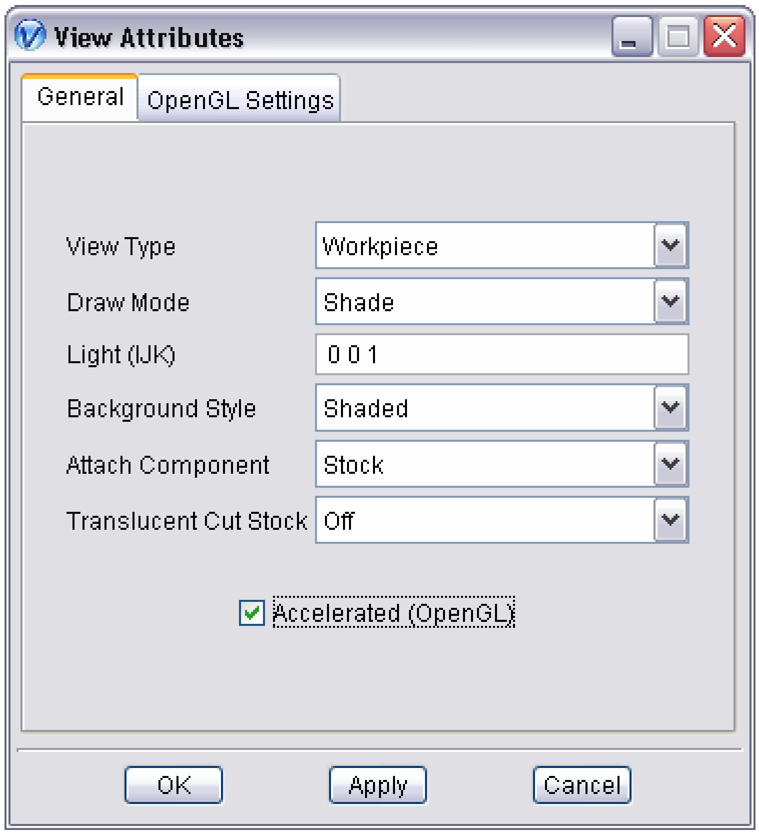
   NOTE: Dynamic Controls (**View menu > Dynamic Controls)** must be set to VERICUT.

Reset — Resets the view to the last orientation accepted by pressing Apply, or as it was when the Orient window was opened.

View Attributes window

Location: **View menu > Attributes**

Opens a window to control the attributes of a view, such as: what is seen in a view, light source, shading, etc. In general, click in a view to make it active, then change attributes as required. Each view has its own view attributes.



General tab **—** The features on this tab enable you to control the display characteristics of a view.

OpenGL Settings tab **—** The features on this tab enable you to control the display characteristics of an Accelerated (OpenGL) view. This tab is only available when **Accelerated (OpenGL)** is toggled "On".

OK — Applies the changes and closes the View Attributes window.

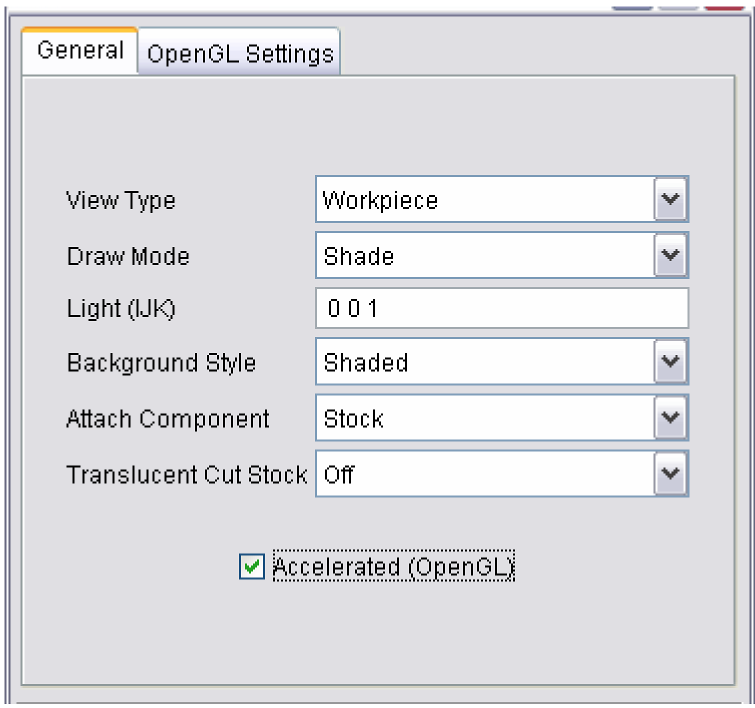
Apply — Applies the changes and leaves the View Attributes window open.

Cancel — Closes the View Attributes window without applying changes.

View Attributes Window, General tab

Location: **View menu > Attributes**

The features on this tab enable you to control the display characteristics of a view.



View Type — Controls what is seen in a view.

Options are:

Workpiece — Displays the stock workpiece and machining that occurs on it. This view supports full inspection capabilities on the machined part, including X-Caliper measurements and AUTO-DIFF model comparisons.

Machine — Displays a 3-D NC machine, when defined. The uncut stock can be seen, however, material removal does not occur in this view type.

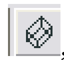
Machine/Cut Stock — Same as above, except material removal does occur. (Does not support machined part inspection-use a Workpiece view type for this activity.)

Profile — Displays a 2-D profile view of a turned workpiece in a G-Code tool path simulation, as if it were spinning and sectioned along the turning axis. The profile is created when cutting begins.

NOTE: Ensure a Spindle component is defined in the machine and turned on by codes in the G-Code tool path file.

Shortcut: You can quickly change the **View Type** by right-clicking in the view, and selecting from the displayed menu. See the **Graphics Area Right Mouse Button Shortcut Menus**, in the Getting Started with VERICUT section of VERICUT Help for more information.

Draw Mode — Controls how machine components are displayed.

Toolbar short cuts for **Draw Shaded** , **Draw Lines** , and **Draw Mixed**  draw modes.

Options:

Shade — Shaded solids

Lines — Wireframe

Hidden — Wireframe with back-facing lines hidden

Mixed — shade, lines or hidden, as defined by the component's Mixed Mode attribute (Ref. **Project menu > Model Setup > Define: Component Attributes tab**)

Shortcut: You can quickly change the **Draw Mode** by right-clicking in the view, and selecting from the displayed menu. See the **Graphics Area Right Mouse Button Shortcut Menus**, in the Getting Started with VERICUT section of VERICUT Help for more information.

Light — I J K vector direction in which the light source points. The three values are separated by spaces, as described in the view coordinate system. A second light source shines on the opposite side of the model.

Background Style — Controls the background seen in a view. Colors for backgrounds are defined via the **Edit menu > Colors** function.

Options:

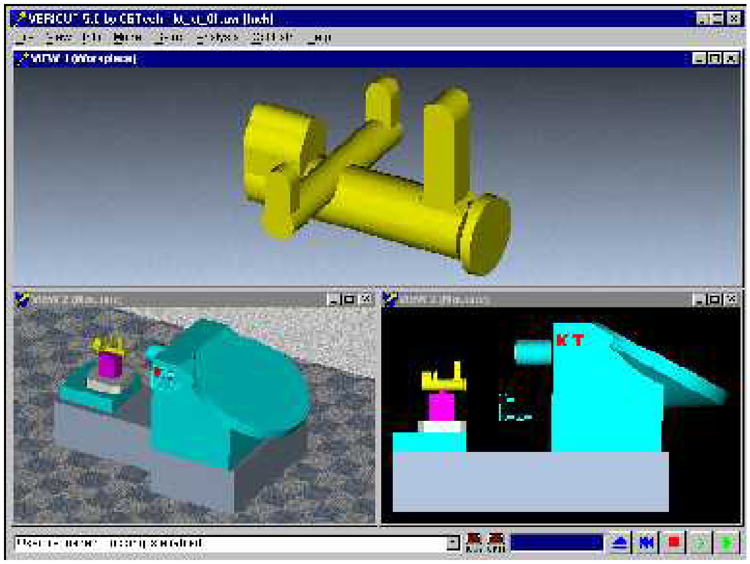
Flat — Monochromatic.

Shaded — Light-to-dark shading.

Walls — Walls, floor, and ceiling-machine views only.

Sample backgrounds:

Shaded



Walls **Flat**

Shortcut: You can quickly change the **Background Style** by right-clicking in the view, and selecting from the displayed menu. See the **Graphics Area Right Mouse Button Shortcut Menus**, in the Getting Started with VERICUT section of VERICUT Help for more information.

Attach Comp. — Attaches the view point and line of sight for a view to the selected component. By default, a Workpiece view is attached to a Stock component, thus the workpiece remains stationary while the tool moves about it. Similarly, by default in a Machine view the view is attached to the machine's Base component, thus the machine base appears stationary while its axes move. Whatever component is selected as the attach component becomes stationary in the simulation, while all other objects are simulated as moving relative to that component.

Shortcut: You can quickly change the component the view is attached to by right-clicking in the view, and selecting **Attach Component** in the menu that displays. See the **Graphics Area Right Mouse Button Shortcut Menus**, in the Getting Started with VERICUT section of VERICUT Help for more information.

Translucent — Controls when the stock model is displayed as translucent (can be seen through).

Translucent can only be applied to an active workpiece view. Once the attribute is set, more than one workpiece view can be displayed "translucent". It is only valid with a "refined display".

All VERICUT functions can be used with a translucent view. However, when the model is rotated, sectioned, or zoomed, the resulting view is a solid. Use **Refine Display** to re-display the translucent view. Almost any model feature can be seen using combinations of **Translucency**, **Reverse**, **Section**, and **Zoom**.

NOTE: Translucent cutters may intermittently disappear while cutting a translucent model. Display the tool as a solid to eliminate this "flashing" affect.

Toolbar short cut: 

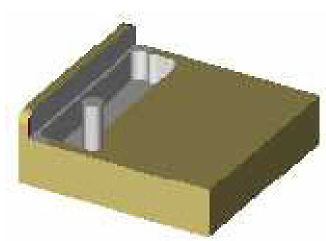
Options are:

Off — Translucency is off and the workpiece is displayed as a solid (can not be seen through).

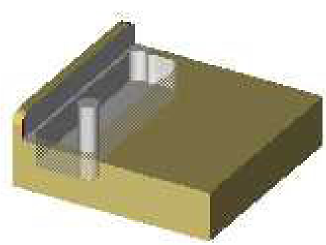
Normal — Causes material in front of a machined surface to become translucent. When a machined surface is not present behind material, the material is displayed in solid.

Extended — Similar to **Normal**, except when a machined surface is not present behind material, the material is still displayed with translucency.

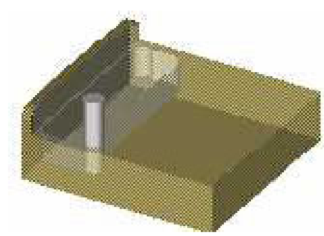
Translucent "Off"



Translucent "Normal"



Translucent "Extended"



Shortcut: You can quickly change the **Translucent** setting by using the toolbar short cut:  Use the left mouse button on the icon to toggle **Translucent** "**On**" or "**Off**". When the **Translucent** icon is toggled "**On**", use the right mouse button on the icon to quickly toggle between **Translucent "Normal"** and **Translucent "Extended"** modes. The change is immediately applied to the "active" workpiece view.

Accelerated (OpenGL) — Toggles Hardware Graphics Acceleration "**On**" and "**Off**".

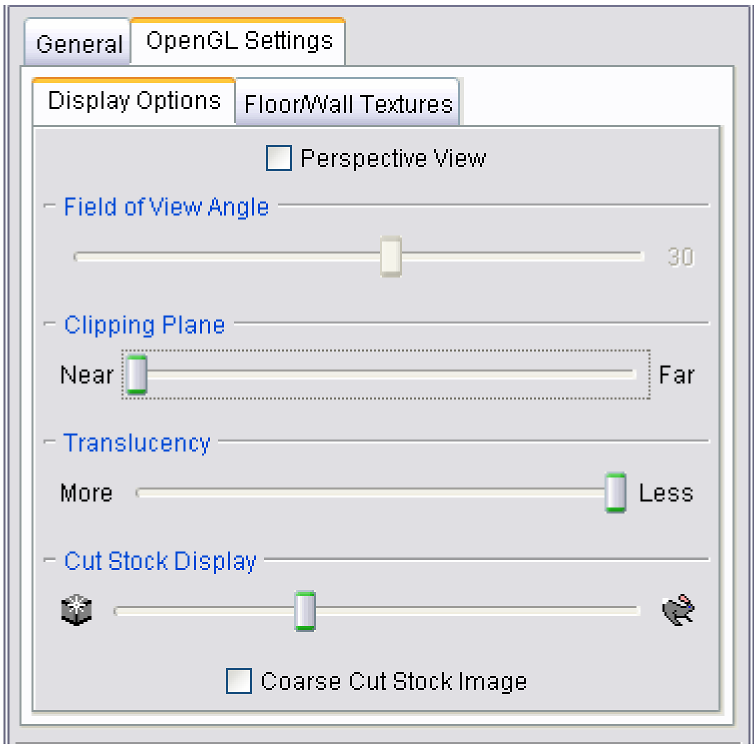
Shortcut: You can also right-click in the graphics area and use **Accelerated (OpenGL)** in the displayed menu to toggle Hardware Graphics Acceleration "**On**" and "**Off**". See the **Graphics Area Right Mouse Button Shortcut Menus**, in the Getting Started with VERICUT section of VERICUT Help for more information.

To View Attributes window

View Attributes window, OpenGL Settings tab

Location: **View menu > Attributes**

The features on this tab enable you to control the display characteristics of an Accelerated (OpenGL) view. This tab is only available when **Accelerated (OpenGL)** is toggled "**On**".



Display Options tab — The features on this tab enable you to control the display characteristics of an OpenGL Workpiece, Machine, or Machine/Cut Stock View.

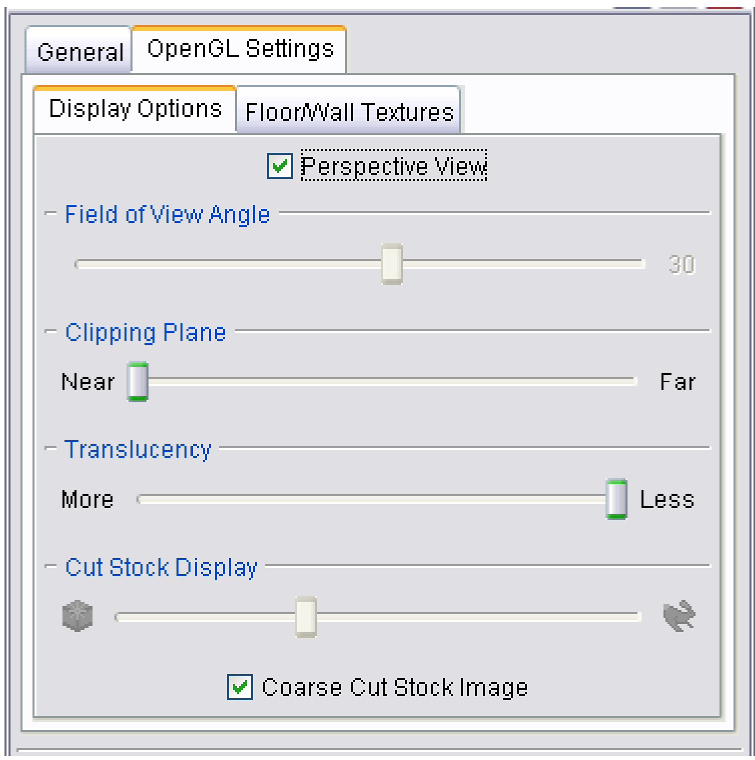
Floor/Wall Textures tab — The features on this tab enable you to specify custom texture files to be used for floor, ceiling and walls of an OpenGL Machine view.

To View Attributes window

View Attributes window, OpenGL Settings tab, Display Options

Location: **View menu > Attributes**

The features on this tab enable you to control the display characteristics of an OpenGL Workpiece, Machine, or Machine/Cut Stock View. The features on this tab are only available when **Accelerated (OpenGL)** is toggled "On".



Perspective View — When **Perspective View** is toggled "**On**", a perspective view is displayed in Machine and Machine/Cut Stock views.

Field of View Angle — This feature is only available when **Perspective View** is toggled "**On**". It enables you to control the Field of View Angle of the perspective view. Move the slider to the left to decrease the view angle and to the right to increase it.

Clipping Plane — This feature is only available when **Perspective View** is toggled "**On**". It enables you to control the position of the "clipping plane" in the perspective view. Move the slider to the left to move the "clipping plane" closer to the plane of the screen. Move it to the right to move the "clipping plane" further into the view (away from the plane of screen). All objects between the "clipping plane" and the plane of the screen are blanked from the display. For example, this feature could be used to blank parts of the machine (like doors) to better view the part on the machine.

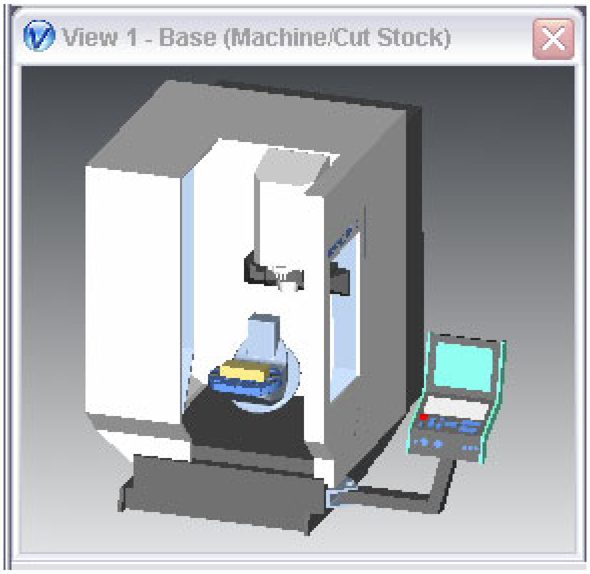
Translucency — This feature enables you to control the translucency characteristic of selected components. For example, this feature could be used to make certain parts of the machine (like doors, or enclosures) to better view the part on the machine.

Before the **Translucency** slider will affect the display, the following conditions ***must*** be met:

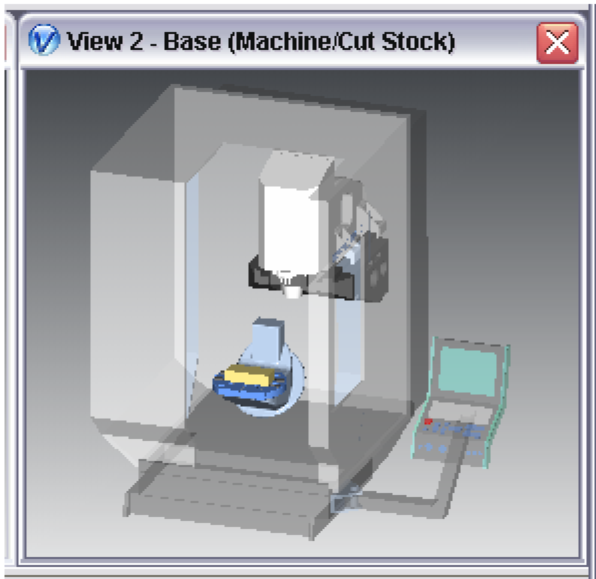
1. The component(s) must have the **Mixed Mode** attribute must be set to **Translucent**. (refer to **Modeling window: Component Attributes tab** in the *VERICUT Help* section, in the *CGTech Help Library*.)
2. **Draw Mode** must be set to **Mixed** (refer to **View Attributes window: General tab** above).

The **Translucency** slider can now be used to vary the translucency of the selected components. The following examples have the machine enclosure component set for using the **Translucency** slider.

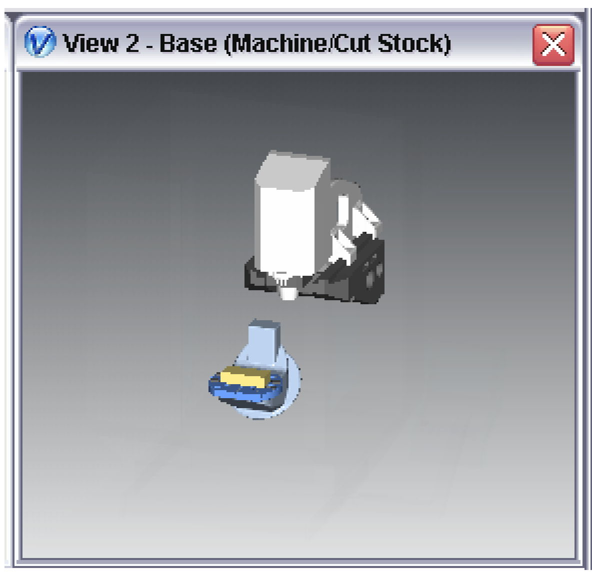
Example of Translucency slider all the way to the right (Less)



Example of Translucency slider at the midpoint



Example of Translucency slider all the way to the left (More)



Cut Stock Display — This feature enables you to control the way that the Cut Stock is displayed (and therefore the processing speed). Moving the slider to the left will display a "sharper" image of the Cut Stock but will result in longer processing times. Moving the slider to the right results in a Cut Stock display with a less sharp image, but processing time will be faster. This feature is only available when the **View Type** is either Workpiece, or Machine/Cut Stock. The **Cut Stock Display** slider is inactive when **Coarse Cut Stock Image** (described below) is toggled "On".

Coarse Cut Stock Image — This feature is only available when the **View Type** is either Workpiece, or Machine/Cut Stock. When toggled "On" (checkmark displayed), the cut stock image in the Machine/Cut Stock view is displayed as an approximate "block" image (also called the "lego" display). This image is only updated at the end of each setup, so that the subsequent setup has an approximate image of the cut stock. The "lego" image moves from setup-to-setup the same as a normal cut stock image.

When toggled "On" in a Workpiece view, the cut stock image is displayed as an approximate "block" image during OpenGL motions (rotate, zoom, pan, etc.). The cut stock display during these actions is in the same state of cutting as the cut stock display in the Machine/Cut Stock view.

When **OK**, or **Apply**, is pressed in the View Attributes window, the cut stock image changes depending on the change in the check-box state, and the **Cut Stock Display** slider becomes inactive. The check-box state is saved in the project setup's view attributes. The Coarse Cut Stock Image feature can be toggled On/Off at any time.

Turning this feature "On" has no effect on the content of the cut stock model, collision checking, or project processing. This is a graphics display change only, for performance reasons.

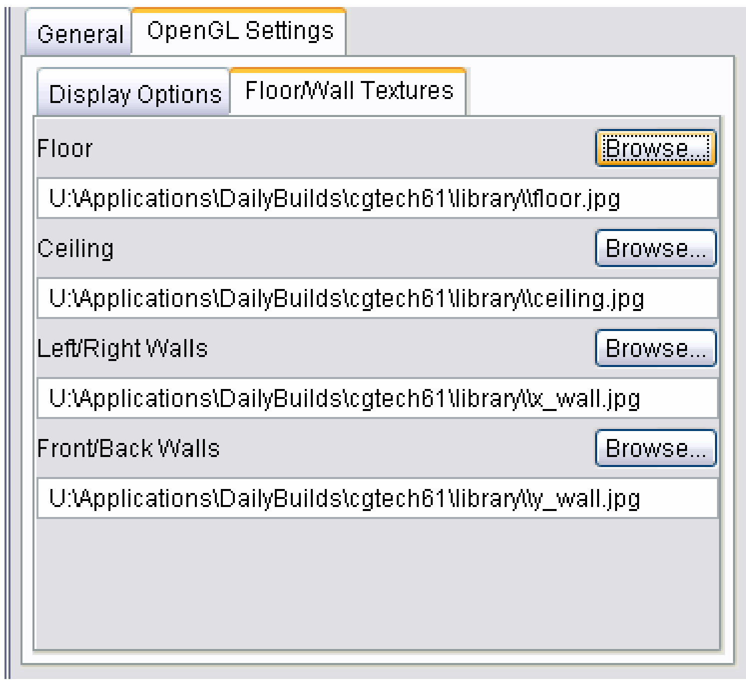
VERICUT features that query the cut stock via graphical picks in a Workpiece, or Machine/Cut Stock, view (such as X-caliper, Modeling, etc) continue to work, regardless of the image.

To OpenGL Settings tab

View Attributes window, OpenGL Settings tab, Floor/Wall Textures tab

Location: **View menu > Attributes**

The features on this tab enable you to specify custom texture files to be used for floor, ceiling and walls of an OpenGL Machine view. These custom textures are only visible in the VERICUT Graphics Area in Machine Views, when **Accelerated (OpenGL)** is toggled "**On**" *and* **Background Style** is set to "**Walls**".



Floor — Enter the *path**file* name in the text field of the JPEG file containing the texture to be used for the floor, or click on **Browse** and use the Texture JPEG File selection window to specify the file.

Ceiling — Enter the *path**file* name in the text field of the JPEG file containing the texture to be used for the ceiling, or click on **Browse** and use the Texture JPEG File selection window to specify the file.

Left/Right Walls — Enter the *path**file* name in the text field of the JPEG file containing the texture to be used for the left and right walls, or click on **Browse** and use the Texture JPEG File selection window to specify the file.

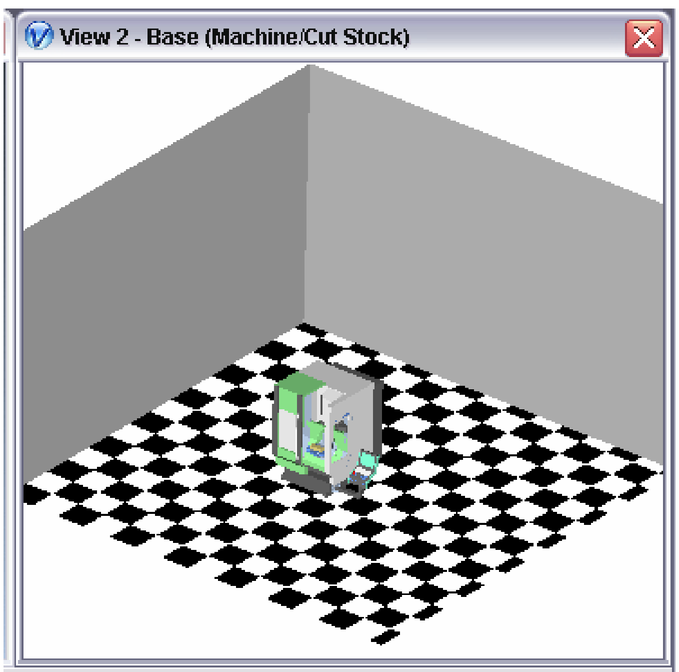
Front/Back Walls — Enter the *\path\file* name in the text field of the JPEG file containing the texture to be used for the front and back walls, or click on **Browse** and use the Texture JPEG File selection window to specify the file.

NOTES:

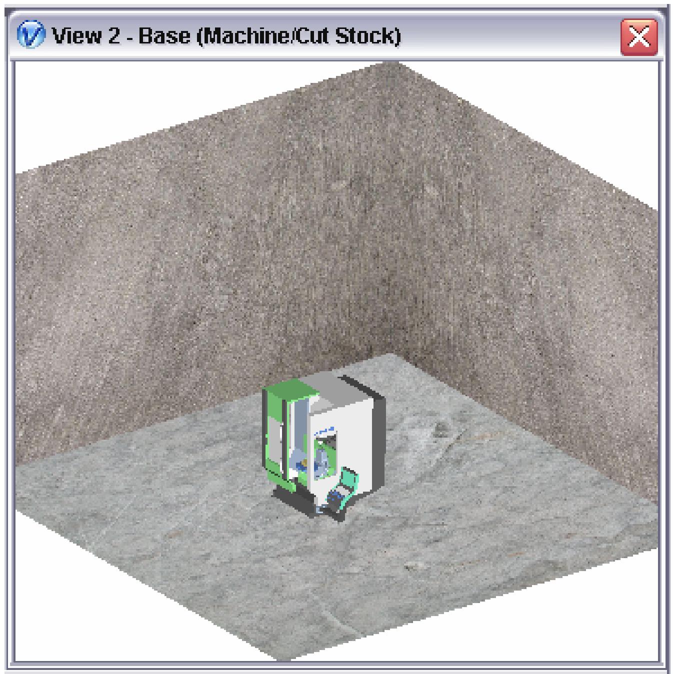
The following information is related to how VERICUT uses the Floor, Ceiling, and Wall JPGs specified above.

1. All images (floor, ceiling, and walls) use a 1:1 aspect ratio.
2. VERICUT cuts given JPEG file images to a rectangle with sizes which are powers of two. For instance, image 1500x760 will be cut to 1024x512. Image 760x760 will be cut to 512x512.
3. Floor/ceiling/walls sizes are calculated based on the machine size in corresponding direction which is extended by some open space around the machine which is also calculated based on the overall machine size.

Sample "standard" OpenGL Machine view:



Sample "custom" OpenGL Machine view:



The custom texture files, specified using the View Attributes window: Floor/Walls Texture tab, only apply to the **current setup** and the specified texture files are stored with the setup in the project file. All OpenGL Machine views in the setup use the same texture files.

Texture files for drawing floor, ceiling, and walls in an OpenGL Machine view can also be specified using the following Environment Variables as in V6.0:

CGTECH_FLOOR_IMAGE
CGTECH_CEILING_IMAGE
CGTECH_X_WALL_IMAGE
CGTECH_Y_WALL_IMAGE

When texture files are specified using Environment Variables, they apply to all OpenGL Machine views with Background Style=Walls, not specific to a particular setup. For more information on these, and all VERICUT environment variables, see **Environment Variables** in the *VERICUT Help* section in the *CGTech Help Library*.

The custom texture files, specified using the View Attributes window: Floor/Walls Texture tab will over-ride one set using one if the Environment Variables. If a custom texture file is not specified with either of these methods, standard VERICUT images will be used.

Consider the following example:

1. No **Floor** texture file is specified on the View Attributes window: Floor/Walls Texture tab.
2. VERICUT will then use any texture file specified using the CGTECH_FLOOR_IMAGE environment variable.
3. If the environment variable has not been specified, VERICUT will use the default chess-board image will be used for the floor.

The same hierarchy is true for the ceiling and the walls.

To OpenGL Settings tab

View Section window

VERICUT Users:

VERICUT Location: **View menu > Section**

VERICUT toolbar short cut: 

Mold and Die Users:

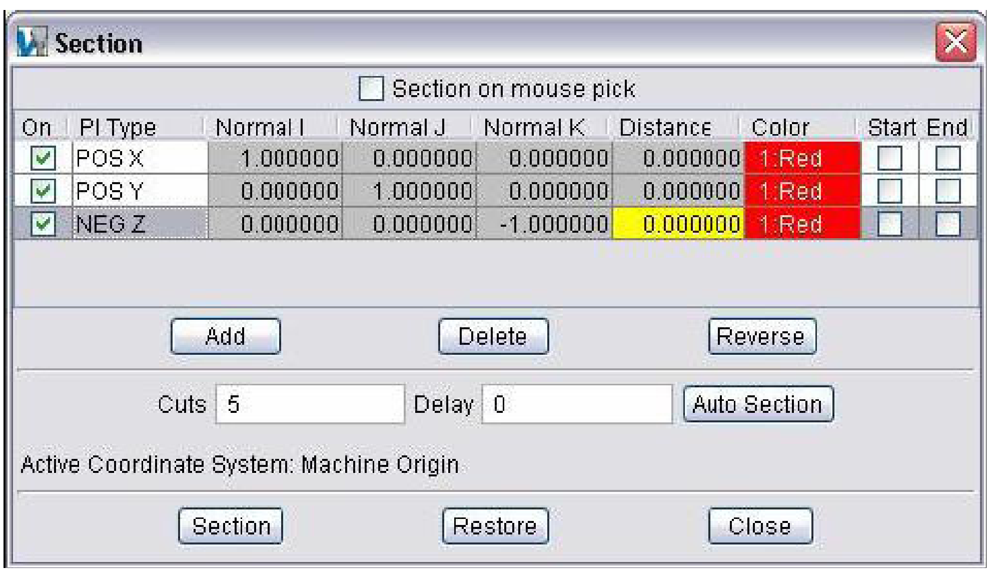
Mold and Die Location: **Views and View Layouts page > Display Section**

Notebook Feature: 

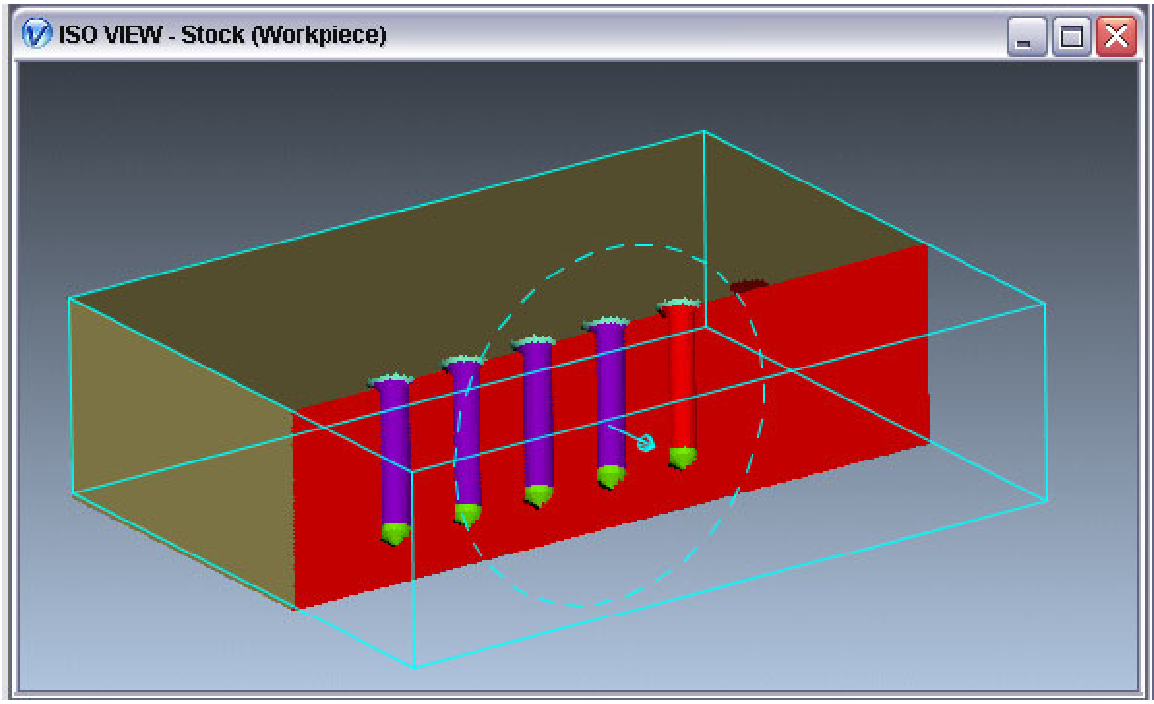
Cutter Grinder Users:

Cutter Grinder Location: Change Views and View Layouts page > Display Section

Notebook Feature: 



Opens the Section window enabling you to define section planes through a VERICUT model in a workpiece view. You can define as many section planes as desired, in any orientation. Section plane orientation and distance are described with respect to the active coordinate system. Restore the original model by clicking anywhere off the part in the workpiece view, or click Restore on the Section window. You can continue cutting on the sectioned model, or restore the original (un-sectioned) model. Machine cuts are applied to the entire model regardless of how it is displayed after sectioning.



Section on mouse pick — Immediately sections the model through the point selected with mouse. The section plane orientation is controlled by the characteristics of the plane highlighted in the Section planes table.

Section planes table:

On — Toggles on and off to activate/deactivate defined section planes in the table. Only the active planes will be used to determine the portion of the model that will be displayed after the section planes are applied.

NOTE: The on/off state of this toggle is only used in conjunction with the Section button below. It has no effect on which plane/direction vector is displayed in the active workpiece view. The plane /direction vector that is displayed in the active workpiece view is determined by the plane that is highlighted in the section planes table.

Pl Type — Specifies section plane orientation. Standard options include six orientations that are perpendicular to axes of the active coordinate system (**POS X**, **POS Y**, **POS Z**, **NEG X**, **NEG Y** and **NEG Z**). POS and NEG define the direction of the normal vector of the section plane with respect to the coordinate system axis. The normal vector points away from the portion of the part that will be displayed when it is sectioned. Using the Custom option, you can define a section plane at any orientation by entering the I, J, and K components of the desired normal vector in the Normal I J K fields (see below).

Normal I, Normal J, Normal K — Specifies the I J K coefficients of a vector perpendicular to the desired section plane. These values are automatically set when Pl Type is set to one of the six standard options and cannot be changed. When Pl Type is set to Custom, the values can be changed to define the direction vector of the desired section plane. Values entered do not have to define a unit vector (vector length=1). Any combination of values can be entered to define the normal vector.

Distance — Distance along the normal vector (see above), measured from the active coordinate system origin to the desired location of the section plane. Key in a value or pick a point on the model with the mouse. A positive value is measured along the specified normal vector, a negative is measured in the reverse direction.

Start / End — When selected, defines the starting and ending planes for automatic sectioning. (See "**Automatic sectioning features**" below).

The following actions can be accomplished by either selecting the associated button in the Section window or selecting it from the pull-down that results from using the right mouse button while the cursor is in the section planes table.

Add — Use to add additional section planes to the table.

Delete — Use to delete the highlighted section plane from the table.

Reverse — Use to reverse the direction vector of the highlighted plane to display the opposite side of the model after sectioning.

Shortcut: Right-click in the Section Planes Table to display the following menu:

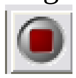
These features provide the same functionality as those described above.

Automatic sectioning features:

This group of features displays a series of animated sectioned views, beginning with the plane identified as the "Start" plane (Start selected), and ending with the "End" plane (End selected). The start plane direction vector must point away from the end plane and the end plane vector must point away from the start plane for proper automatic sectioning to take place. Automatic section cuts are equally spaced.

Cuts — The number of equally spaced section planes that will automatically be applied between the start and end section planes.

Delay — Seconds of delay to pause between each automatic section cut display.

Auto Section — Begins the automatic sectioning sequence. To interrupt an automatic sectioning, press  (**Stop**), or click in the VERICUT main window and press the **Escape** <Esc> key.

Section — Applies the active section plane(s) in the Section planes table to the model. The resulting "sectioned" model is displayed in all workpiece views.

Restore — Restores the solid model from the sectioned display. The sectioned model is for display purposes only. When cutting has been performed with a sectioned view displayed, the results are the same as if the entire model was displayed..

Close — Closes the Section window.

Select/Store View window

VERICUT Users:

VERICUT Location: **View menu > Select/Store**

Mold and Die Users:

Mold and Die Location: **Change Views and View Layouts page > Custom Views**

Notebook Feature: 

Cutter Grinder Users:

Cutter Grinder Location: **Change Views and View Layouts page > Custom Views**

Notebook Feature: 

Opens the Select/Store View window enabling you to store, modify and apply custom views. Defined views are shown in the "View List". Each custom view is stored with the View Type and orientation of the view that was active at the time it was stored. There is no limit to the number of views you can store.

VERICUT Shortcut: You can quickly reverse the orientation, or change the view to one of the standard or stored custom views by right-clicking on the view in the VERICUT graphics window, then selecting the desired view name using Select View option in the displayed menu. This shortcut is not available for **Mold and Die**.

View List — List of defined views.

Shortcut: Right-click in the View List to display the following menu:

These features provide the same functionality as described below.

Activate — Applies the View Type and orientation of the stored view highlighted in the View List to the active view in the VERICUT graphics window.

Modify — Modifies the View Type and orientation of the highlighted view in the View List to match those of the active view in the VERICUT graphics window.

Reset — Resets the active view in the VERICUT graphics area to the View Type and orientation that it had when the Select/Store View window was opened.

Add — Adds a new view, after the highlighted view, to the View List. The new view will have the same View Type and orientation as the active view in the VERICUT graphics window. Double click on the view name to rename the view. Hit <enter> when you are through editing.

Delete — Deletes the highlighted view from the View List.

Delete All — Deletes all views from the View List.

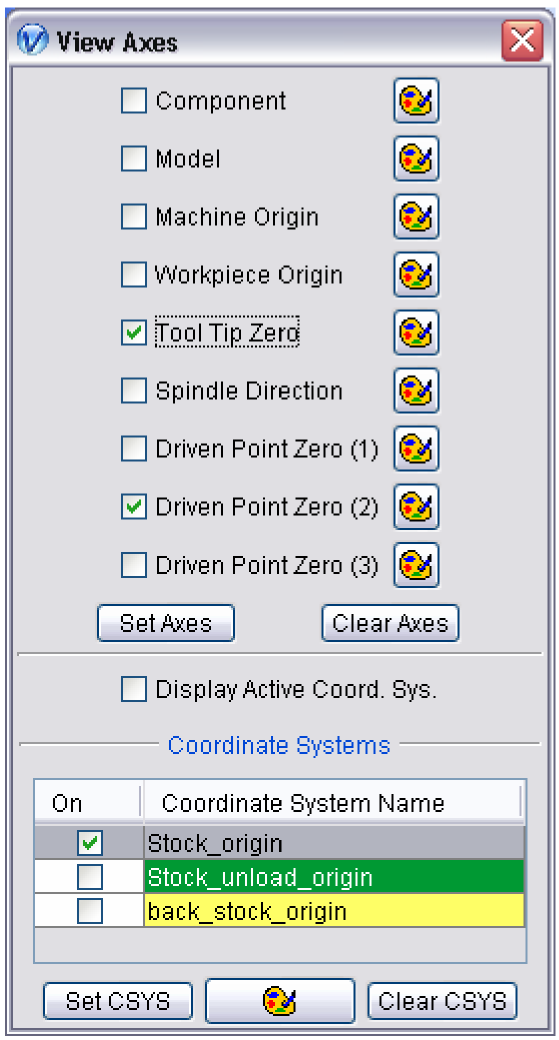
Close — Closes the Select /Store View window.

VERICUT Shortcut: You can quickly **Add**, **Delete**, or **Activate** views in the View List by right-clicking on a view and selecting desired action from the displayed menu. See the **Graphics Area Right Mouse Button Shortcut Menus**, in the Getting Started with VERICUT section of VERICUT Help for more information. This shortcut is not available for **Mold and Die**.

View Axes window

Location: **View menu > View Axes**

Opens a window to control when various axes and coordinate systems are displayed. Axes are created by VERICUT and are stored in the Preferences file. Coordinate Systems (CSYS) are user defined and are stored in the "setup". Once displayed, axes and coordinate system symbols remain displayed until toggled "Off" in the View Axes window. Solid lines indicate that an axis is parallel or pointing out of the screen. Dashed lines indicate that an axis is pointing into the screen.



Shortcut: You can also change axes displayed by right-clicking in the view, and selecting from the Display Axes list in the menu that displays. See the **Graphics Area**

Right Mouse Button Shortcut Menus, in the Getting Started with VERICUT section of VERICUT Help for more information.

AXES

Use the Color Pallet icons , to specify a "global" display color for each type of axis described below.

Component — When selected, displays **XcYcZc** axes that representing the coordinate system of a component. Each component has its own local coordinate system.

Model — When selected, displays **XmYmZm** axes that represent the coordinate system of a model. Each model has its own local coordinate system.

Machine Origin — When selected, displays **XmcYmcZm**c axes that represent the coordinate system in which an NC machine is defined.

Workpiece Origin — When selected, displays **XwpYwpZwp** axes that represent the coordinate system to which Stock, Fixture, and Design components are connected.

- In a simulation where a machine ***IS NOT*** defined, such as when processing APT-CLS tool path files, the workpiece coordinate origin is the origin of the non-moving "Base" component to which Stock, Fixture, and Design components are connected. In previous VERICUT versions, this was known as the "World coordinate system" (XwYwZw axes).
- In a simulation where a machine ***IS*** defined, such as when processing G-Code tool path files, the workpiece coordinate origin is the origin the machine component responsible for carrying the Stock, Fixture, and Design components.

Tool Tip Zero — When toggled "On", displays the **XToolTipYToolTipZToolTip** axes that represent where the tool tip (the VERICUT control point) of the "active" tool would be located, relative to the "active" stock, if all linear axes were positioned at zero. The display is based on the actual Machine/Control configuration and therefore may be displayed as a right hand axis, a left hand axis or possibly even a non-orthogonal axis. If no tool has been loaded, VERICUT assumes a zero length tool.

The following should be considered when using this feature:

- Tool Tip Zero is only applicable when processing G-Code Data.
- The Tool Tip Zero display is based on the "active" tool. If the machine being used has more than one tool component, the correct tool component must be set as active before valid results can be achieved.
- The Tool Tip Zero display is based on the "active" stock. If more than one stock component on the machine, the correct stock component must be set as active before valid results can be achieved.
- The Tool Tip Zero display shows where the tool tip is and therefore may not be particularly useful when programming a 5-axis machine in "gage-length" mode.

Driven Point Zero — When toggled "On", displays the "driven point" (identified by the  symbol) and the **XDrivenPointYDrivenPointZDrivenPoint** axes.

The "driven point" is only displayed in the Machine View. Depending on the machine configuration, more than one Driven Point Zero axis may be displayed in the View Axes window as shown in the picture above.

The initial "driven point" position is determined based on the following:

- If there are no rotary/turret components attached to the "active" tool component, the initial driven point is the origin of the "active" tool component.
- If there is 1 rotary/turret component attached to the "active" tool component, the initial driven point is the origin of the rotary/turret component.
- If there are 2 rotary/turret components attached to the "active" tool component, the initial driven point is the intersection of the two rotary axes.
- If there are more than 2 rotary/turret components attached to the "active" tool component, the initial driven point is the origin of the highest level rotary/turret component.

The following offsets are then applied to the initial "driven point" (adjusted as needed for RTCP):

- Gage Offset
- Tool Nose Compensation Offset
- Tool Length Compensation Offset
- Turret Offset
- Gage Pivot Offset
- Pivot Offset (RTCP)
- 3D Tool Length Comp Offset (Fanuc)

The Driven Point Zero axes represent where the driven point of the "active" tool would be located, relative to the "active" stock, if all linear axes were positioned at zero. The display is based on the actual Machine/Control configuration and therefore may be displayed as a right hand axis, a left hand axis or possibly even a non-orthogonal axis. The Driven Point Zero axes are displayed in Workpiece, Machine and Machine/Cut Stock views.

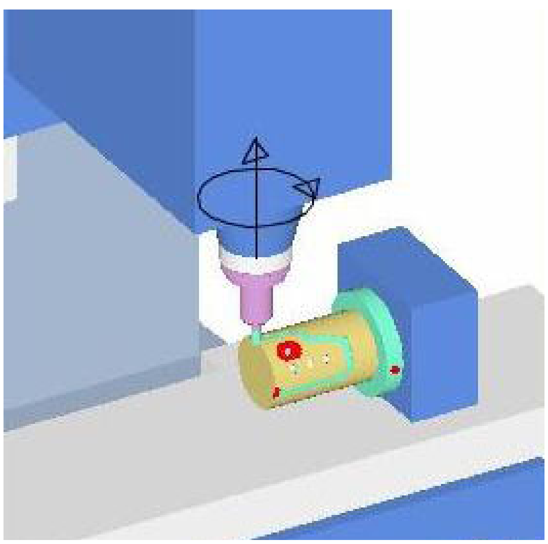
The following should be considered when using this feature:

- Driven Point Zero is only applicable when processing G-Code Data.
- The Driven Point Zero display is based on the "active" tool. If the machine being used has more than one tool component, the correct tool component must be set as active before valid results can be achieved.

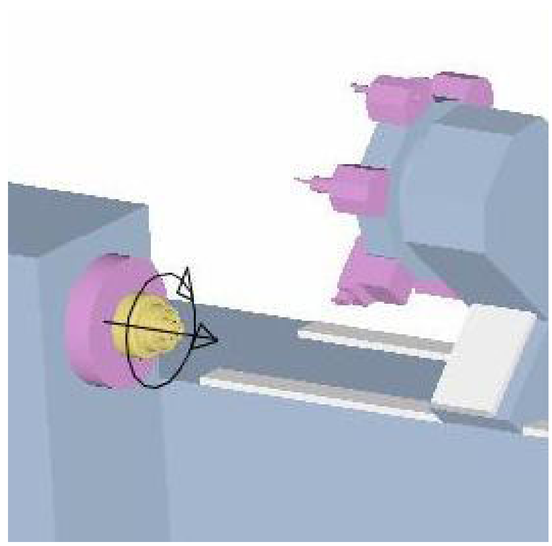
- The Driven Point Zero display is based on the "active" stock. If more than one stock component on the machine, the correct stock component must be set as active before valid results can be achieved.
- Some machine configurations do not conform to initial "driven point" rules described above and therefore will not produce useful results. An example of this would be Maho-Phillips machines using a two position head. The initial driven point could not be calculated correctly resulting in invalid results.
- Dual turret machines typically will not have the active tool defined for both turrets, therefore the initial driven point can not be calculated correctly resulting in invalid results.
- Some machine/controller combinations, like the one in the sample file **bos5vm01.VcProject**, may not produce the results that you might expect, but based on the above definitions, the displayed results are accurate.

Spindle Direction — When toggled "On", a graphic is displayed indicating direction that the spindle is turning.

Mill Example



Lathe Example



The Spindle Direction indicator can also be turned On/Off by right-clicking in the graphics window and selecting **Display Axes > Spindle Direction** from the pop-up menu that displays.

Set Axes — When selected, all "axes" are toggled "On" and will be displayed.

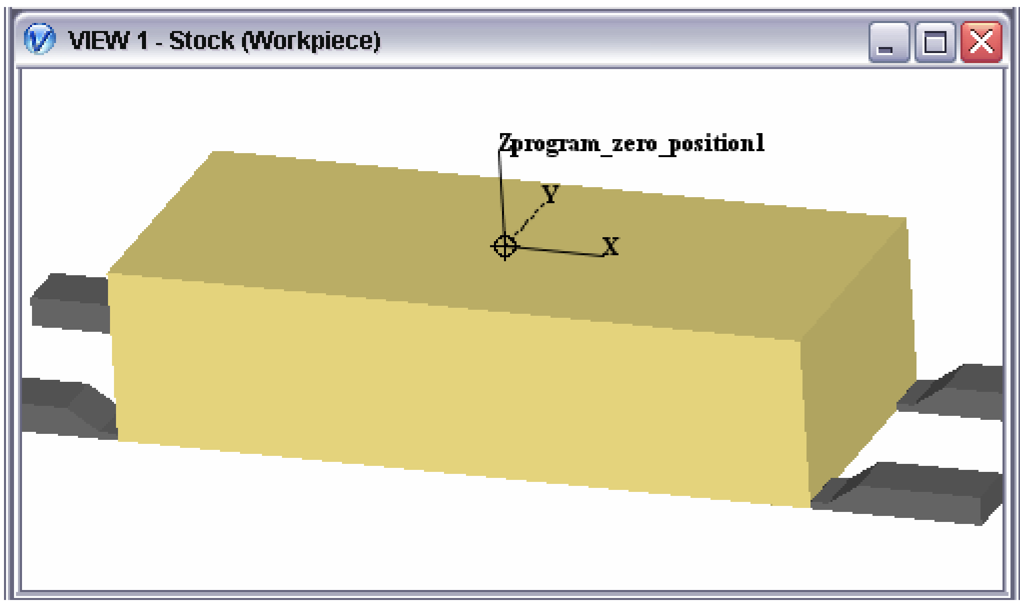
Clear Axes — When selected, all "axes" are toggled "Off" and will not be displayed.

NOTES:

1. Axes display status is stored in the Preferences file. Axes that are displayed when exit a VERICUT session are displayed the next time you run VERICUT.
2. Axes are drawn in the Foreground color by default. Use the Color Pallet icon , for each axis type to change the display color of the individual axes.

COORDINATE SYSTEMS (CSYS)

Display Active Coord. Sys. — When toggled "On", a coordinate system with the name of the active coordinate system will display in the VERICUT Graphics area as shown below. Only the "active" coordinate system will display with the marker at its origin as shown.



Coordinate System Table — The Coordinate System Table displays all coordinate systems defined for the "current setup". Each CSYS in the table has a checkbox that toggles "On" and "Off", indicating whether it is displayed or not. By default, CSYS are drawn in the Foreground color. The color of the CSYS can be changed using the **Color Pallet button**, described below. The background color of each **Coordinate System Name** field indicates the color that the CSYS will be displayed.

Set CSYS — When selected, all "coordinate systems" are toggled "On" and will be displayed.

 (**Color Pallet button) —** Use to define the display color for a "coordinate system". To specify a color for a particular CSYS, click on the **Coordinate System Name** field, in the **Coordinate System Table**, so that it becomes highlighted. The

background color of the **Coordinate System Name** field indicates the current color of the CSYS. Click on the **Color Pallet button** and select the desired color from the color pallet that displays.

Clear CSYS — When selected, all "coordinate systems" are toggled "Off" and will not be displayed.

NOTES:

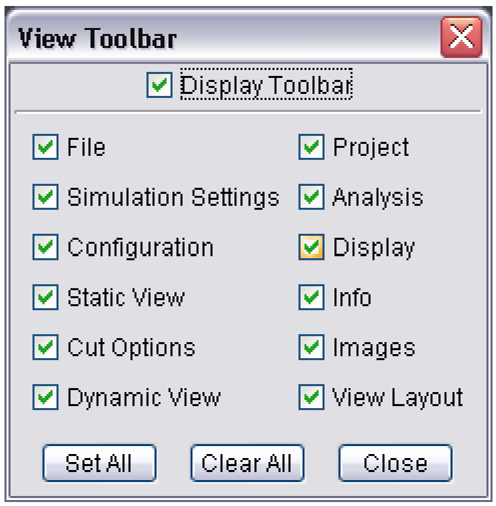
1. Coordinate System display status is stored in the "setup".
2. Axes are drawn in the Foreground color by default. Use the **Color Pallet button** , in the **Coordinate Systems** section of the **View Axes** window to change the display color of individual coordinate systems.

See also "**VERICUT axes and coordinate systems**" in the *Using VERICUT* section. *Using VERICUT* can be found in the *CGTech Help Library*.

View Toolbar window

Location: **View menu > Toolbar**

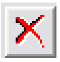
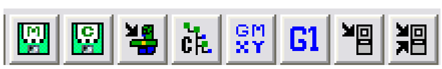
Opens a window to control when the Toolbar is displayed and the icons it contains. Toolbar icons provide quick and easy access to the most commonly-used VERICUT functions/features. The Toolbar can be closed, opened, or customized at any time.

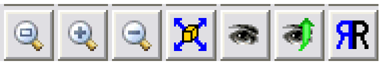


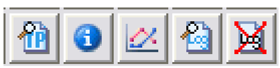
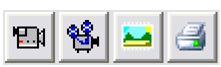
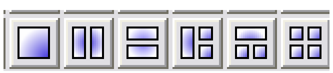
Display Toolbar — When selected, displays Toolbar groups on the VERICUT main window.

Set All, Clear All — Turns on (or off) display of ***all*** Toolbar icons. Corresponding options are all selected (or cleared).

Toolbar groups:

Toolbar Icons:	**Toolbar name:**	**Functions (click for more info)**
	**Close**	Close Toolbar
 or 	**File**	Open Project Save Project / Save Project As (See NOTE 8) New Project (See NOTE 1) Open In Process File Save In-Process / Save In-Process As (See NOTE 9) Working Directory Edit NC Program Custom Interface
	**Simulation Settings**	Motion G-Code Settings APT Settings No Animation No Machine Simulation AutoSave
 or 	**Configuration**	Save Machine /Save Machine As (See NOTE 10) Save Control / Save Control As (See NOTE 11) Machine Settings Component Tree Word Format Word Address Control Settings Control Advanced Options

	**Static View**	Zoom to Box / Zoom creates new View (See NOTE 2) Zoom In Zoom Out Fit / Fit All (See NOTE 12) Refine Display (See NOTE 3) Last Refine Display (See NOTE 4) Reverse
	**Cut Options**	OptiPath Standard/FastMill Cut Modes Delete Detached Stock MDI
	**Dynamic View**	Dynamic X/Y/Z/XY Rotation Dynamic Pan Dynamic Zoom
	**Project**	Open Machine Open Control Setup Models NC Program Tools Output Setup Plan Coordinate Systems Project Tree
	**Analysis**	AUTO-DIFF X-Caliper Toolpath Review

		Die Sinking Simulation Comparator
	**Display**	Workpiece View Machine/Cut Stock View Draw Shaded Draw Lines Draw Mixed Section Translucent (See NOTE 7) Color
	**Info**	View NC Program Status Graphs VERICUT Log Clear Log File and Logger (See NOTE 6)
	**Images**	Image Record Image Playback / AVI Playback (See NOTE 5) View Capture Print View
	**View Layout**	Use to set VERICUT views to the standard view layout shown on the icon. Single View Layout Two View Layout (Horizontal) Two View Layout (Vertical) Three View Layout (Vertical) Three View Layout (Horizontal) Four View Layout

NOTES:

1. Right-click on the New Project icon to toggle between (Inch) and (Millimeter) mode.
2. Right-click on the icon to toggle between (Zoom to Box) and (Zoom creates new View) modes.
3. **Refine Display —** When selected, refines the display to be a high-quality image. Refining the display can useful to see model details after model manipulation, such as zooming, rotating, panning, etc. This action occurs automatically any time cutting is started.
4. **Last Refine Display —** When selected, returns a workpiece view to the last refined display image. This action is useful to quickly start cutting after manipulating the workpiece for viewing: zooming, rotating, panning, etc.
5. **Image Playback/AVI Playback —** The active icon is set automatically based on the active image record output format (ref. **File menu > Images > Record Movie**).
6. **Clear Logfile and Logger —** Clears both the VERICUT Log File and the Logger (Message Logger).
7. Use the left mouse button on the **Translucent** icon to toggle **Translucent** "**On**" or "**Off**". When the **Translucent** icon is toggled "**On**", use the right mouse button on the icon to quickly toggle between **Translucent "Normal"** and **Translucent "Extended"** modes. The change is immediately applied to the "active" workpiece view.
8. Right-click on the icon to toggle between (Save Project) and (Save Project As) modes.
9. Right-click on the icon to toggle between (Save In-Process) and (Save In-Process As) modes.
10. Right-click on the icon to toggle between (Save Machine) and (Save Machine As) modes.
11. Right-click on the icon to toggle between (SaveControl) and (Save Control As) modes.
12. Click the icon with the left mouse button to "fit" the active view. Click the icon with the right mouse button to "fit" all views.

For additional information about the VERICUT functions accessed by Toolbar icons, refer to the function names in the **VERICUT Menus and Features** section of *VERICUT Help*.

Resolution

Location: **View menu > Resolution**

Controls the quality of cut model display. Increasing resolution displays a more accurate representation of the cut model, and requires more computer resources and processing time to display and manipulate the model. A coarse display resolution is automatically used when dynamically manipulating the model, such as rotation, zooming, etc. When the dynamic action is finished, the display is upgraded to the previous resolution.

Options:

Manual — A display resolution which allows most models to be manipulated with acceptable performance and display quality.

Auto — Automatically sets display resolution based on viewing distance from the model. Resolution automatically increases when you zoom closer to see details on the model, and decreases as you zoom out.

Look & Feel

Location: **View menu > Look & Feel**

The **Look & Feel** features change the appearance of VERICUT's user interface. All windows are affected. The look and feel of VERICUT can be changed at any time.

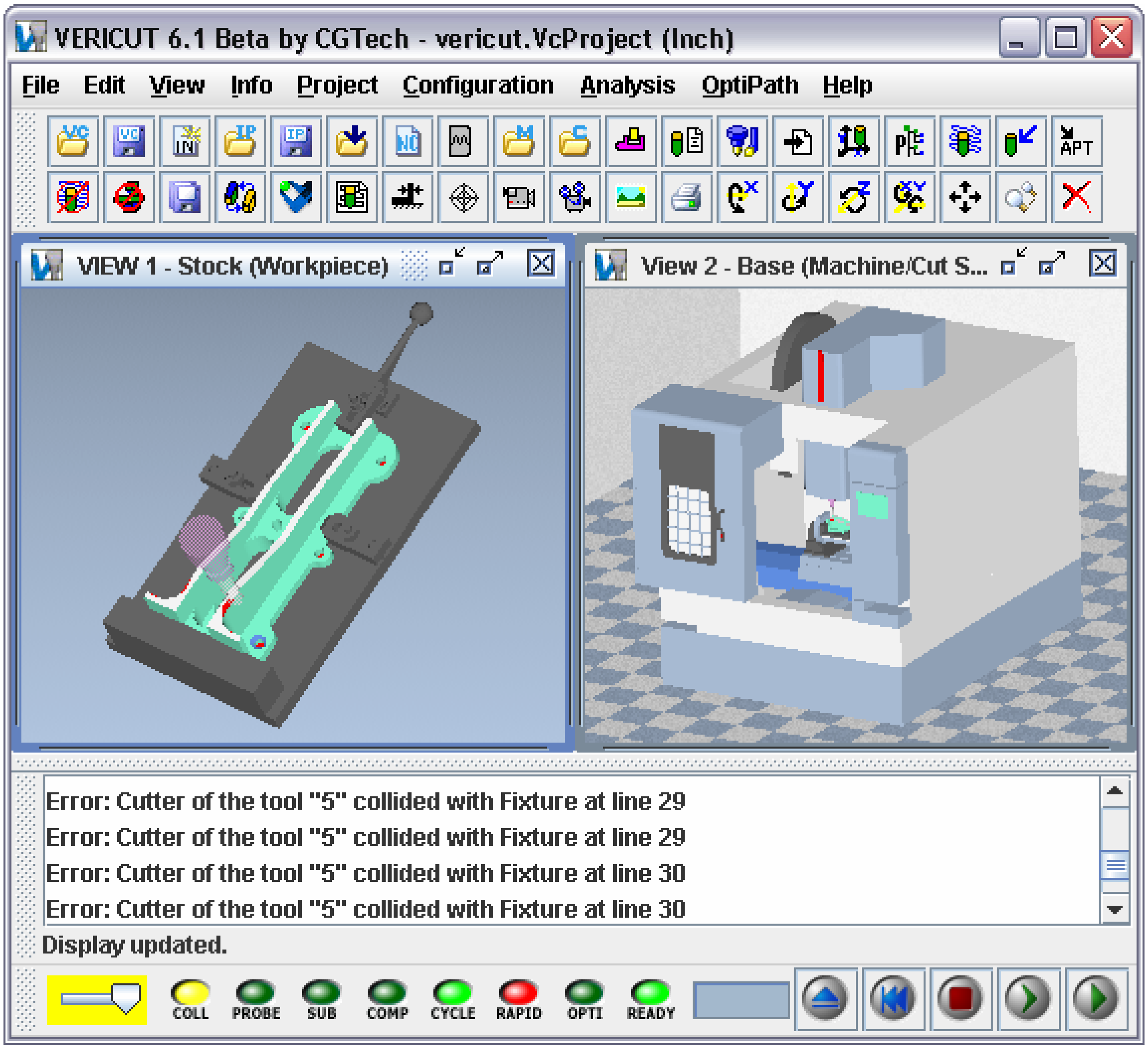
To change the user interface look and feel:

1. Select **View menu > Look and Feel**.
2. Choose the option, from the pull-down list, that provides the desired appearance.

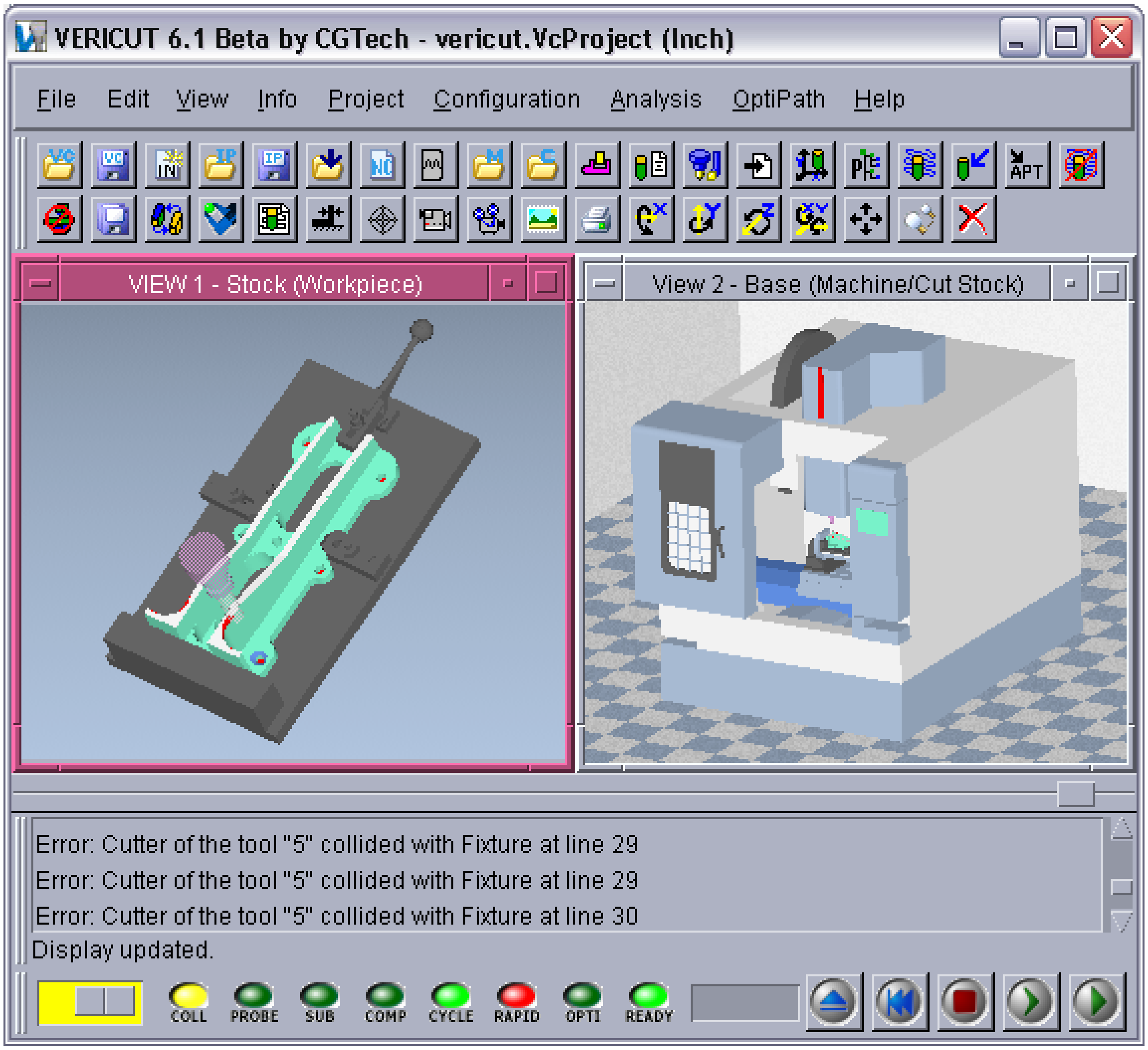
NOTE: Not all options are available on all systems.

Options:

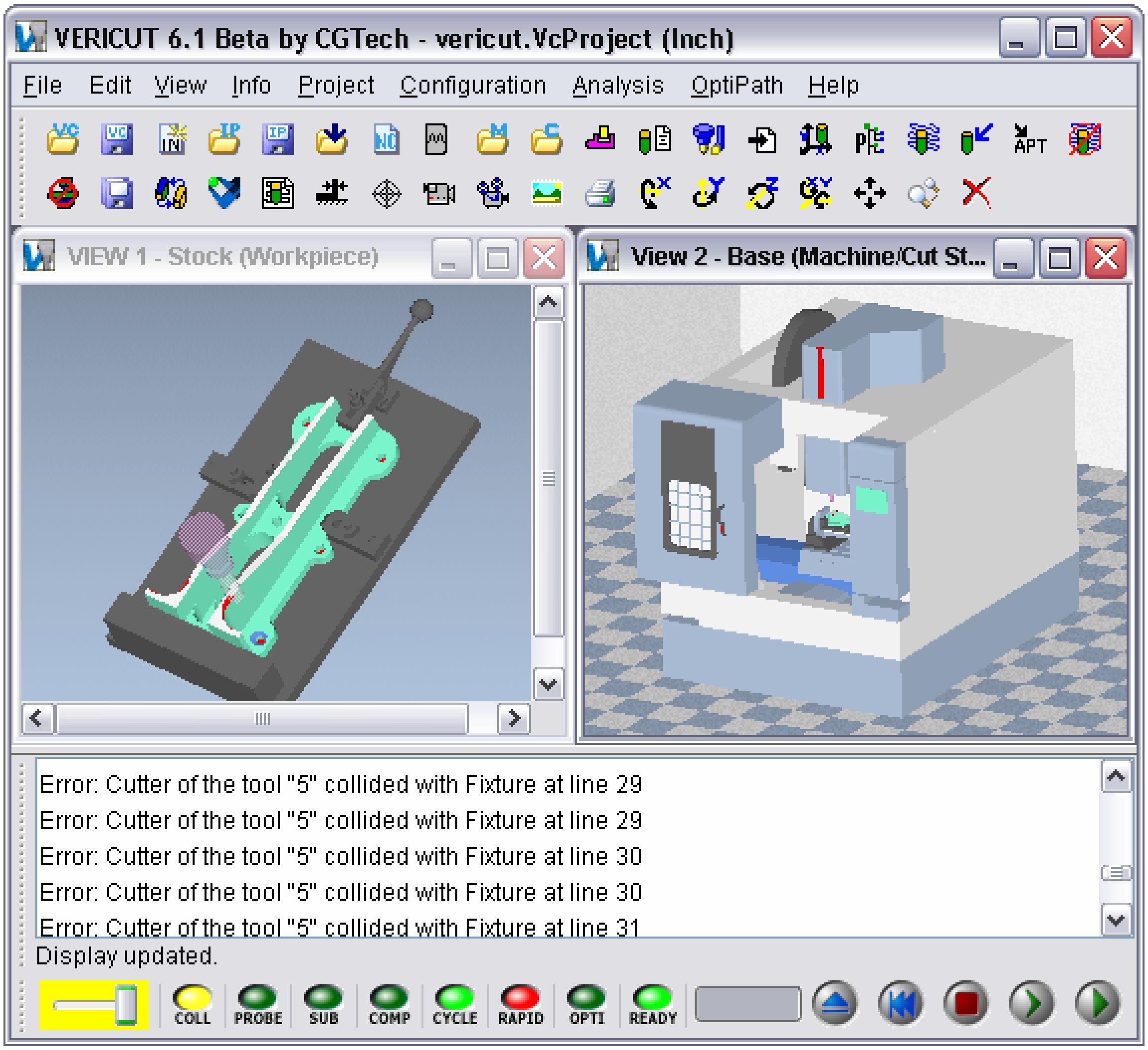
Metal —



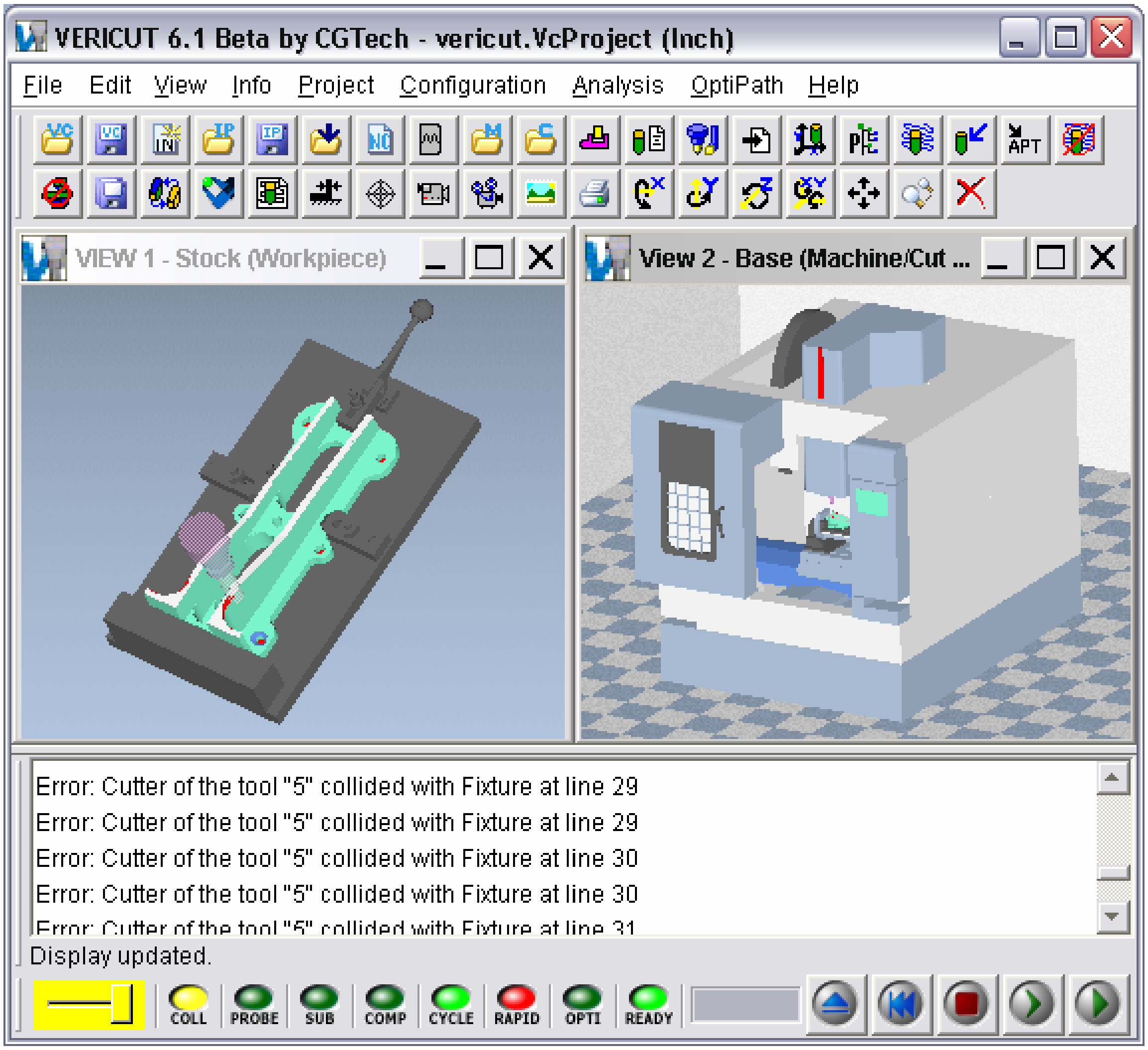
Motif —


VERICUT 6.1 Beta by CGTech - vericut.VcProject (Inch)
File Edit View Info Project Configuration Analysis OptiPath Help
VIEW 1 - Stock (Workpiece)
View 2 - Base (Machine/Cut Stock)
Error: Cutter of the tool "5" collided with Fixture at line 29
Error: Cutter of the tool "5" collided with Fixture at line 29
Error: Cutter of the tool "5" collided with Fixture at line 30
Display updated.
COLL PROBE SUB COMP CYCLE RAPID OPTI READY

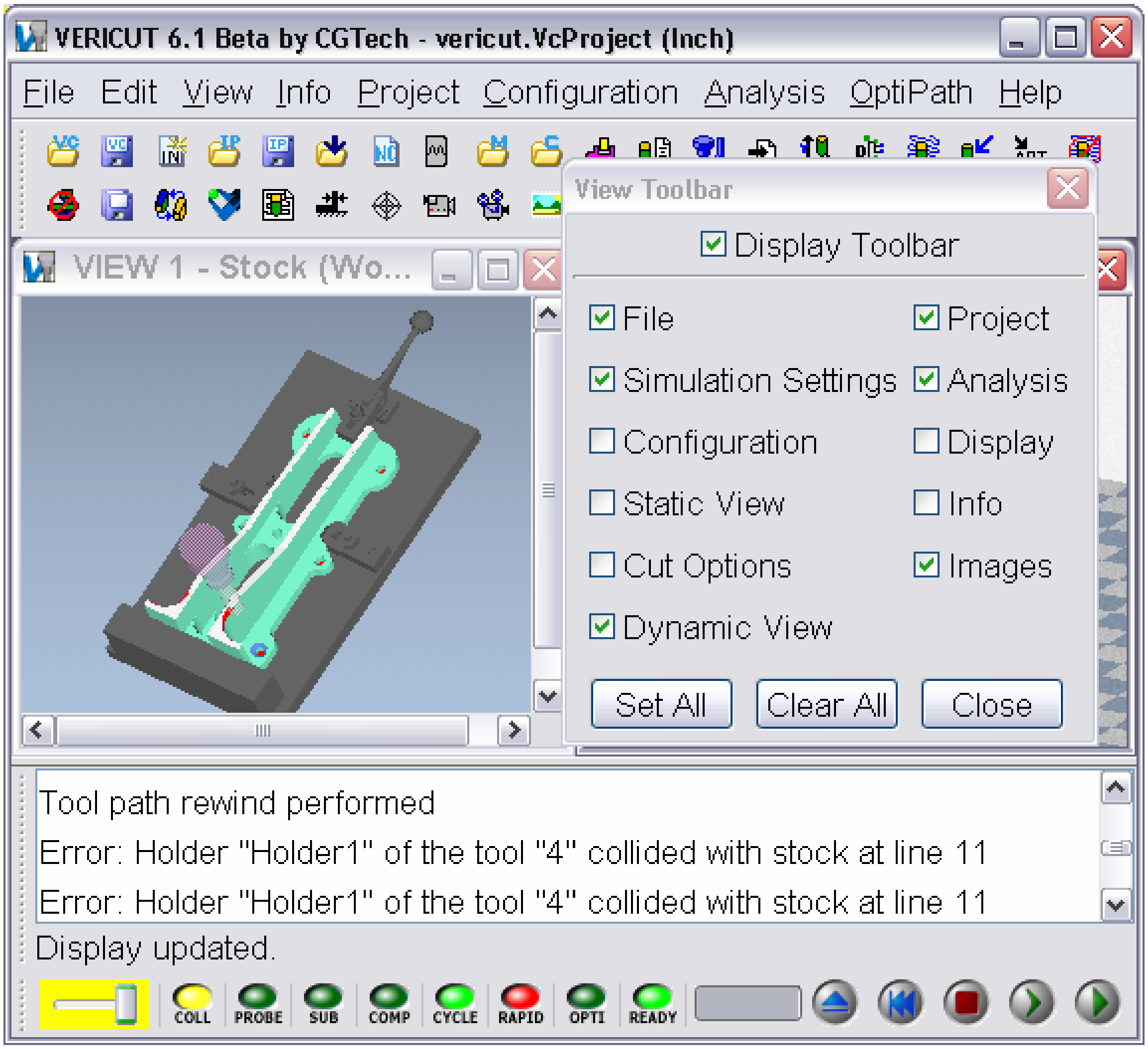
Windows —


VERICUT 6.1 Beta by CGTech - vericut.VcProject (Inch)
File Edit View Info Project Configuration Analysis OptiPath Help
VIEW 1 - Stock (Workpiece)
View 2 - Base (Machine/Cut St...
Error: Cutter of the tool "5" collided with Fixture at line 29
Error: Cutter of the tool "5" collided with Fixture at line 29
Error: Cutter of the tool "5" collided with Fixture at line 30
Error: Cutter of the tool "5" collided with Fixture at line 30
Error: Cutter of the tool "5" collided with Fixture at line 31
Display updated.
COLL
PROBE
SUB
COMP
CYCLE
RAPID
OPTI
READY

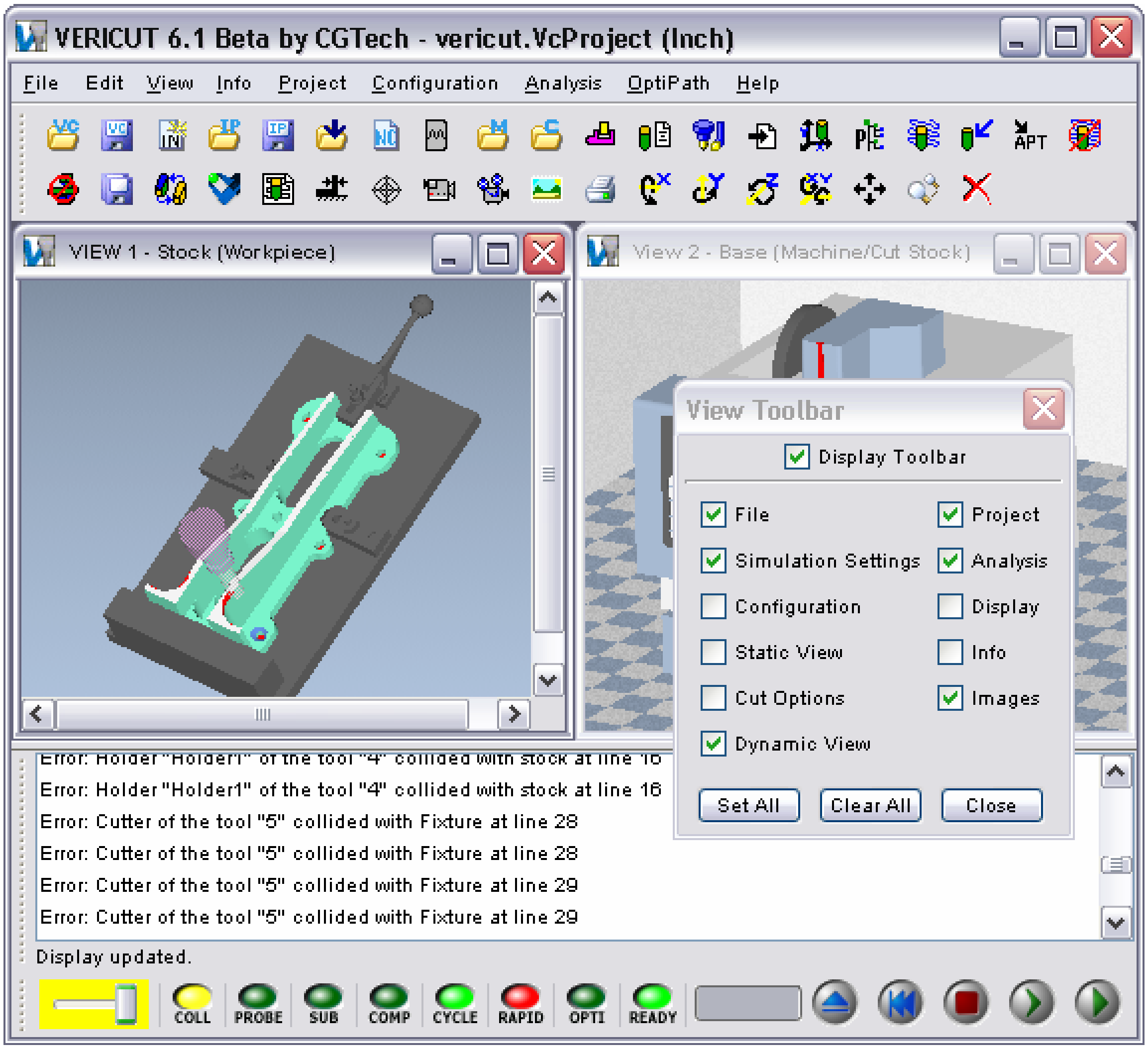
Windows Classic —



Text Larger — Use to make the text in VERICUT menus, message logger, windows, etc. larger. Each time you click on "Text Larger", the text gets incrementally larger to a maximum font size of "14 point".



Text Smaller — Use to make the text in VERICUT menus, message logger, windows, etc. smaller. Each time you click on "Text Smaller", the text gets incrementally smaller to a minimum font size of "6 point ".



Dynamic Controls

Location: **View menu > Dynamic Controls**

View menu > Dynamic Controls options enable you to use the same mouse actions while in VERICUT, that are required to perform dynamic pan, zoom or rotate in each of the supported CAD systems. This parameter is saved in the Preferences file.

CATIA — Use CATIA's mouse action conventions for dynamic pan, zoom and rotate.

Pro/E — Use Pro Engineer's mouse action conventions for dynamic pan, zoom and rotate.

Unigraphics — Use Unigraphics' mouse action conventions for dynamic pan, zoom and rotate.

VERICUT — Use VERICUT's standard mouse action conventions for dynamic pan, zoom and rotate. See **Dynamic Zoom, Pan and Rotate** in the *Getting Started with VERICUT* section for additional information.

Wildfire — Use Pro Engineer Wildfire's mouse action conventions for dynamic pan, zoom and rotate.

Reverse Mouse Wheel — Toggle On/Off to reverse the action of mouse wheel motion.

The following table summarizes how each dynamic feature was implemented for each CAD system.

	CATIA	Pro/Engineer	Unigraphics	VERICUT	Wildfire
Pan	Middle mouse button, drag	Hold Ctrl key, Right mouse button, drag	Hold Shift key, Middle mouse button, drag (or, Middle & Right mouse button, drag)	Right mouse button, drag (or, hold Shift key, any mouse button, drag)	Hold down SHIFT, the middle mouse button, and drag the mouse.
Zoom	Hold Middle mouse button, click Left or Right mouse button, drag:	Hold Ctrl key, Left mouse button, drag: Down or Left	Hold Ctrl key, Middle mouse button, drag (or Left & Middle mouse button,	Rotate the thumbwheel: Toward you for larger, away for	Hold down CTRL, the middle mouse button, and drag the

	Up for larger Down for smaller	for larger Up or Right for smaller	drag	smaller (or, hold Ctrl key, any mouse button, drag: Up for larger Down for smaller)	mouse vertically.
Rotate	Hold Middle mouse button, hold left or Right mouse button, drag	Hold Ctrl key, Middle mouse button, drag	Middle Mouse button, drag	Left mouse button, drag (or, hold Shift & Ctrl keys, any mouse button, drag)	Hold down the middle mouse button and drag the mouse.

To change the dynamic controls behavior:

1. Select **View menu > Dynamic Controls**.
2. Choose the CAD System, from the pull-down list, that you want VERICUT to emulate.

Logger View Options

View menu > Logger View Options

Use the **Logger View Options** to specify what information that you want displayed in the logger, when you want to clear the Logger, or copy a Logger message to paste in another document.
When these features are set before the simulation, only the selected type of information will be written to the Logger. These features can also be used after the simulation add or remove information from the Logger.

Error — When toggled "On" (check displayed), error messages will be written out to the Logger.

Warning — When toggled "On" (check displayed), warning messages will be written out to the Logger.

Info — When toggled "On" (check displayed), informational messages will be written out to the Logger.

Clear on Reset — When toggled "On" (check displayed), the contents of the Logger will be cleared whenever you do a Reset.

Clear — Selecting this feature will immediately clear the Logger. Using this feature only clears the Logger window, not the VERICUT Log file.

Copy Ctrl + C — Use this feature to copy the highlighted message(s) so that it can be pasted to a document.

Pick a number of individual messages for copying by holding the **Ctrl** key down while picking.

Pick a range of consecutive messages by selecting the first message in the range, and then hold the **Shift** key down while selecting the last message in the desired range. All of the messages between two messages that you selected will become highlighted.

Info Menu

NC Program (Info)

VERICUT Users:

VERICUT Location: **Info menu > NC Program**

VERICUT toolbar short cut: 

Mold and Die Users:

Mold and Die Location: **Analyze, View Files, Print page > Display Tool Path Text**

Notebook Feature: 

Cutter Grinder Users:

Cutter Grinder Location: **Analyze, View Files, Print page > Display Grinding Program Text**

Notebook Feature: 

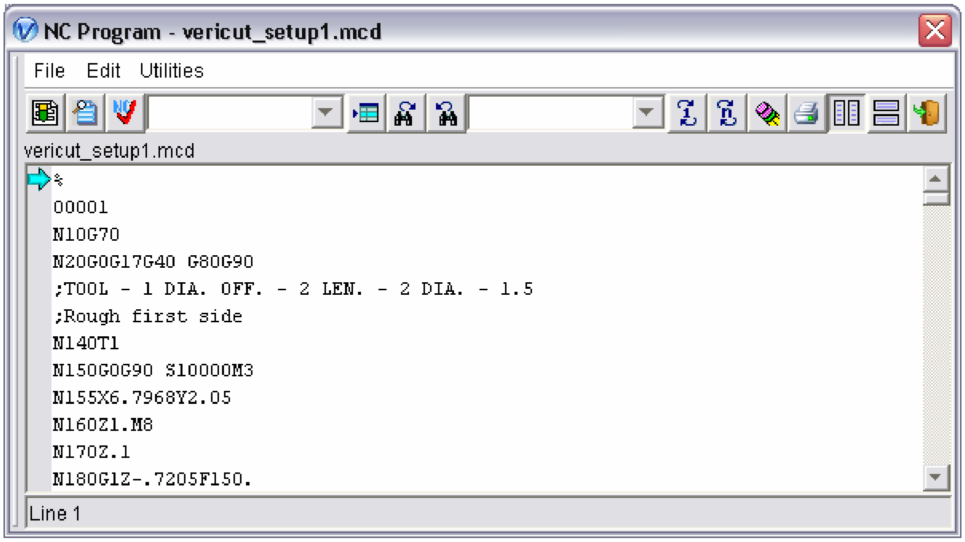
Opens a window displaying the NC program file being simulated. In this window, you can scroll through the entire file to see all the records, search for text forward and backward, print, and more. An arrow pointer indicates the current tool path record being processed. The window is updated when a new NC program file is processed, such as when a list of NC program files are simulated. (Ref. **Project menu > NC Programs**).

The **NC Program** window is also one of the dockable windows enabling you to dock it inside the VERICUT main window if you choose. See **Dockable Windows** in the *Getting Started with VERICUT* section of *VERICUT Help* for additional information.

NOTE: When the **NC Program** window is docked, make sure that you click in the window so that it becomes the "active" window before using F1 to get help specific to the window. Otherwise F1 will go to the CGTech Help Library.

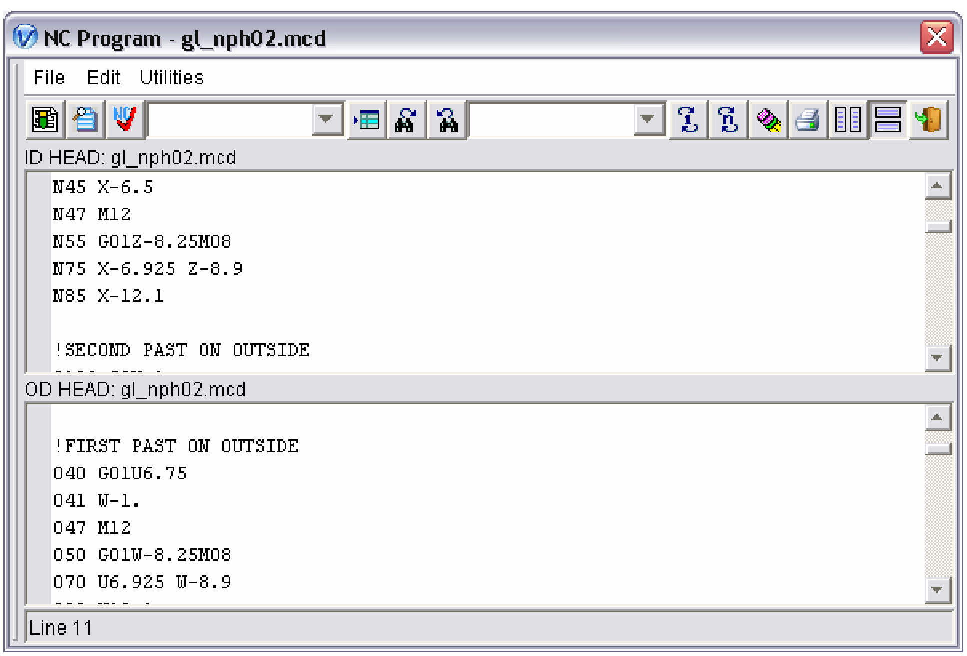
Tip:

You can edit the tool path file using the **Edit menu > NC Program** function in VERICUT, or **Analysis and Viewing page > Edit the Tool Path File** in **Mold and Die**.

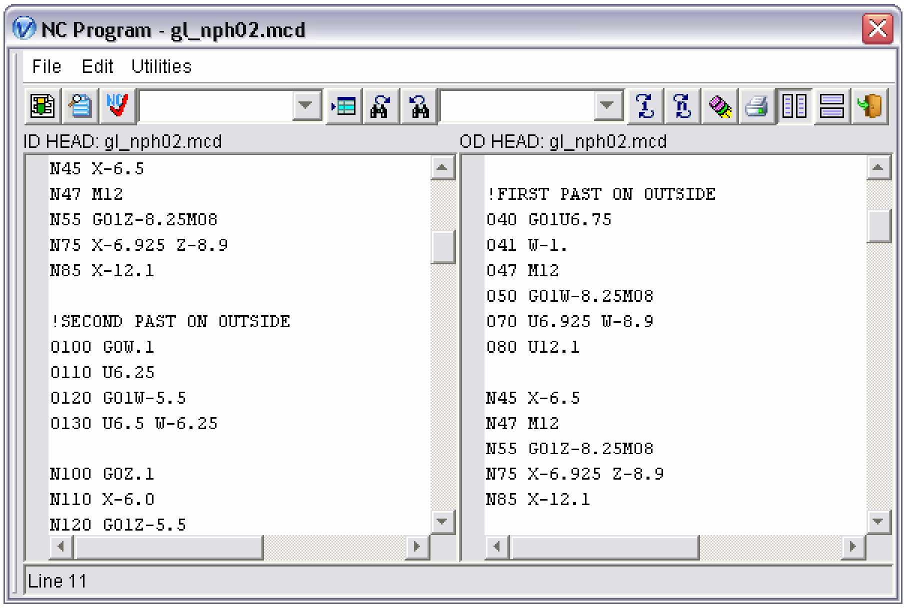


The NC Program window will display as shown above for most types of programs. The NC Program window will display with a separate area for each channel for sync'd programs as shown below. You have the option of displaying these areas either vertically or horizontally.

Tiled Vertical



Tiled Horizontal



Main Menu

The menu located across the top of the NC Program (Info) window provides easy access to major NC Program (Info) functions. Each menu contains groups of related functions. Left-click on any menu name to display the list of functions in that menu group. Click on the function in the menu you want to use. The name of the current NC program file is also displayed in this area.

File

Save As — Save the current NC program file under a different name.

Print — Displays a window enabling you to specify print characteristics and print the NC program.

Mode — Use the features in this list to specify the "mode" that you want VERICUT to be in.

NC Program — Standard VERICUT mode.

NC Program Review — Puts VERICUT in NC Program Review mode. See **NC Program Review**, in the Analysis menu section, in *VERICUT Help* for more information.

NC Program Preview — Puts VERICUT in NC Program Review mode and displays the tool trace represented by the NC progam and the design model. This feature can be used without processing the NC Program first, as required by NC Program Review. See **NC Program Preview**, in the Analysis menu Section of VERICUT *Help* for more information.

Exit — Closes the **NC Program window**.

Edit

Insert From MDI — Enables you to insert NC blocks from the NC Blocks List, in the MDI window, into the NC program displayed in the NC Program (Info) window.

Selected Lines — Use to insert one, or more, highlighted blocks in the MDI window, NC Blocks List, after the selected block in the NC Program (Info) window.

All Lines — Use to insert all blocks in the MDI window, NC Blocks List, after the selected block in the NC Program (Info) window.

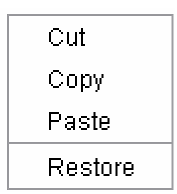
Cut — Cuts the highlighted text in the NC program listing and puts it in the paste buffer.

Copy — Copies the highlighted text in the NC program listing to the paste buffer.

Paste — Puts the contents of the paste buffer at the location of the cursor in the NC program listing.

Restore — use to remove all edits from the NC program listing restoring it to its pre-edited state.

Shortcut: Right-click in the NC Program Info window to display the following menu:



These features provide the same functionality as those described above under Edit in the main menu.

Utilities

Calculator — Opens the Calculator window enabling you to do mathematical calculations and conversions inside VERICUT.

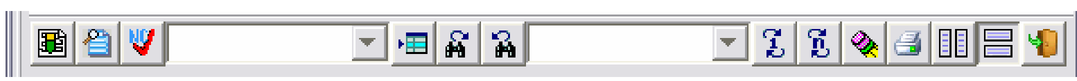
Colors — Opens the NC Program Color window enabling you specify colors for specific features (comments, variable, macros, etc.) in the NC Program listing.

Block Renumber — Opens the Block Renumber window enabling you to renumber one, or more blocks, in the NC program and specify format characteristics to be used for block numbering. The formats specified must conform to the characteristics defined in the current control file.

Check Syntax — Displays the Check Syntax window enabling you to check the displayed NC program for syntax errors.

NOTE: The Utilities features are only available for G-Code NC programs.

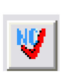
Icon Bar



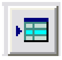
The items in the Icon Bar enable you to search for specific items in the NC program listing, print the NC program listing, or enter, or exit, NC Program Review mode. Moving the cursor over the icon will display name of the option.

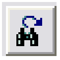
 **NC Program Review —** Use this icon to switch between standard VERICUT mode and NC Program Review mode (ref .**NC Program Review**, in the Analysis menu section, in *VERICUT Help* for more information).

 **NC Program Preview —** Use this icon to put VERICUT in NC Program Review mode and displays the tool trace represented by the NC progam and the design model. This feature can be used without processing the NC Program first, as required by NC Program Review. See **NC Program Preview**, in the Analysis menu Section of VERICUT *Help* for more information.

 **NC Program Syntax Check —** Displays the Check Syntax window enabling you to check the displayed NC program for syntax errors.

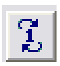
Line Number or Search Text — Use this text field to enter a line number or a string of text to search for.

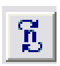
 **Goto Line Number —** Moves the cursor in the NC program listing to the line number specified in the **Line Number or Search Text** field.

 **Search Forward —** Searches forward in the NC program listing for the text string specified in the **Line Number or Search Text** field.

 **Search Backward —** Searches backward in the NC program listing for the text string specified in the **Line Number or Search Text** field.

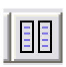
Replacement Text — Use this text field to enter a "replacement text" string.

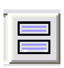
 **Replace One —** Replace one occurrence of text string in the **Line Number or Search Text** field (or the highlighted text in the NC program listing) with the text string in the **Replacement Text** field.

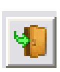
 **Replace All —** Replace all occurrences of the text string in the **Line Number or Search Text** field with the text string in the Replacement Text field.

 **Undo Highlighted Changes —** Use this feature to undo changes made to the NC program. Highlight the change and then click on the icon to "undo" the change.

 **Print —** Prints the NC program listing.

 **Tile Horizontal —** Use to specify that multi-channel NC program listings display tiled horizontally. See the "Tiled Horizontal" picture above.

 **Tile Vertical —** Use to specify that multi-channel NC program listings display tiled vertically. See the "Tiled Vertical" picture above.

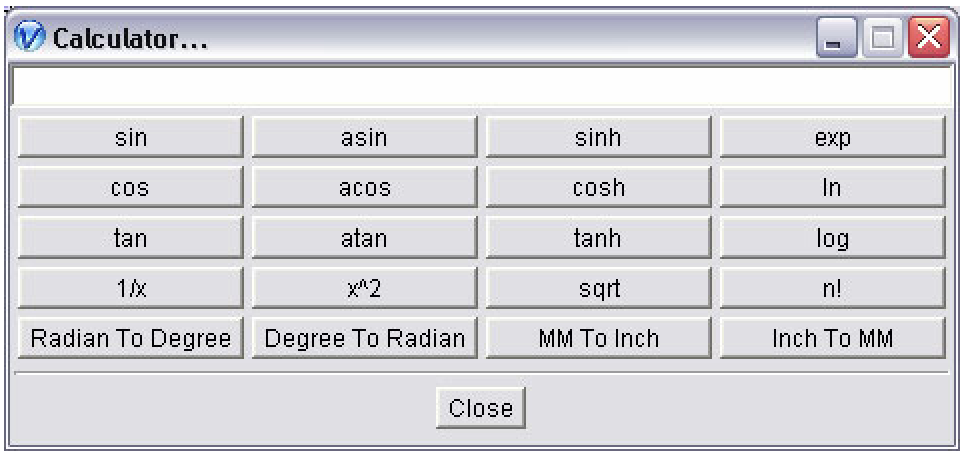
 **Close Window —** Closes the **NC Program** window.

Calculator window

The Calculator window is accessed from the NC Program (Info) window by selecting **Calculator**, in the **Utilities** menu.

The features in this window enable you to do mathematical calculations and conversions from inside VERICUT. Simply enter the value in the text field, and then click on the desired function button.

You can also use the Calculator window to determine the current value of a variable. Enter #*variable*, where *variable* is the variable name, in the text field and VERICUT will return the current value of "*variable*".



Functions

sin — Returns the sin of the angle specified. Enter the angle, in degrees, in the text field.

asin — Returns the arc sine of the value entered in the text field. The specified value must be in the range of -1.0 to 1.0. The returned value is in degrees, and is in the range of -90.0 to 90.0.

sinh — Returns the hyperbolic cosine of the given angle. Enter the angle, in degrees, in the text field.

exp — Returns the natural logarithm "e" raised to the specified power specified in the text field.

cos — Returns the cosine of the specified angle. Returns the hyperbolic tangent of the given angle specified in degrees.

acos — Returns the arc cosine of the value entered in the text field. 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 0.0 to 180.0.

cosh — Returns the hyperbolic cosine of the specified angle. Enter the angle, in degrees, in the text field.

ln — Returns the natural logarithm of the value entered in the text field.

tan — Returns the tangent of the specified angle. Enter the angle, in degrees, in the text field.

atan — Returns the arc tangent of the value entered in the text field. 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 -90.0 to 90.0.

tanh — Returns the hyperbolic tangent of the specified angle. Enter the angle, in degrees, in the text field.

log — Returns the base 10 logarithm of the value entered in the text field. The specified value must be positive otherwise zero is returned.

1/x — The inverse key replaces the variable **x** with the value entered in the text field and returns the answer in the format of a decimal point fraction.

x^2 — Returns the square of the value entered in the text field.

sqrt — Returns the square root of the value entered in the text field.

n! — Returns the factorial value ($n! = 1 \cdot 2 \cdot 3 \cdot \cdot \cdot (n-2)(n-1)n$) of the number entered in the text field.

Radian To Degree — Returns the "degree" equivalent of the "radian" angle value entered in the text field.

Degree To Radian — Returns the "radian" equivalent of the "degree" angle value entered in the text field.

MM To Inch — Returns the "inch" equivalent of the "millimeter" value entered in the text field.

Inch To MM — Returns the "millimeter" equivalent of the "inch" value entered in the text field.

Close — Closes the Calculator window.

NC Program Colors window

The NC Program Colors window is accessed from the NC Program (Info) window by selecting **Colors**, in the **Utilities** menu.

The NC Program Colors window enables you to specify colors for specific features (comments, variable, macros, etc.) in the NC Program listing in the NC Program (Info) window. VERICUT uses the definitions in the Word Format table, for the current control file, to determine what words go into each of the NC program feature categories. The following are the Word Format settings that VERICUT uses.

Comments: Anything that falls between "Begin Comment" and "End Comment"

Variables: Type "Special", Sub Type "Variable Name" or "Variable Tag"

Macros: Type "Macro" or Type "Conditional" with "Word" related Subtype

Operators: Type "Math" or Type "Condition" with a math related Subtype, or Type "Special" with SubType "Separator", or Type "Special" with SubType "Begin/End Type II"

Function: Type "Function", or Type "Type II"

Specials: Type "Special" with SubType = "Skip", "End of block", "Begin/End Data", "Console Message", "Ignore", "Sin84D_*", and "Num VAR Define"

 — Use the color pallet icon next, to each NC program feature, to display a color pallet and use it to select a color for the particular NC program feature. When you have

finished with color selection, use the **OK** button, described below, to apply the colors to the NC program listing in the NC Program (Info) window.

Reset — Use the **Reset** button next to each NC program feature to return the color for the particular feature back to its default color.

OK — Applies the color settings to the to the NC program text displayed in the NC Program (Info) window and closes the NC Program Colors window.

Reset All — Use the **Reset All** button to return the color for ***all*** NC program features back to its default color.

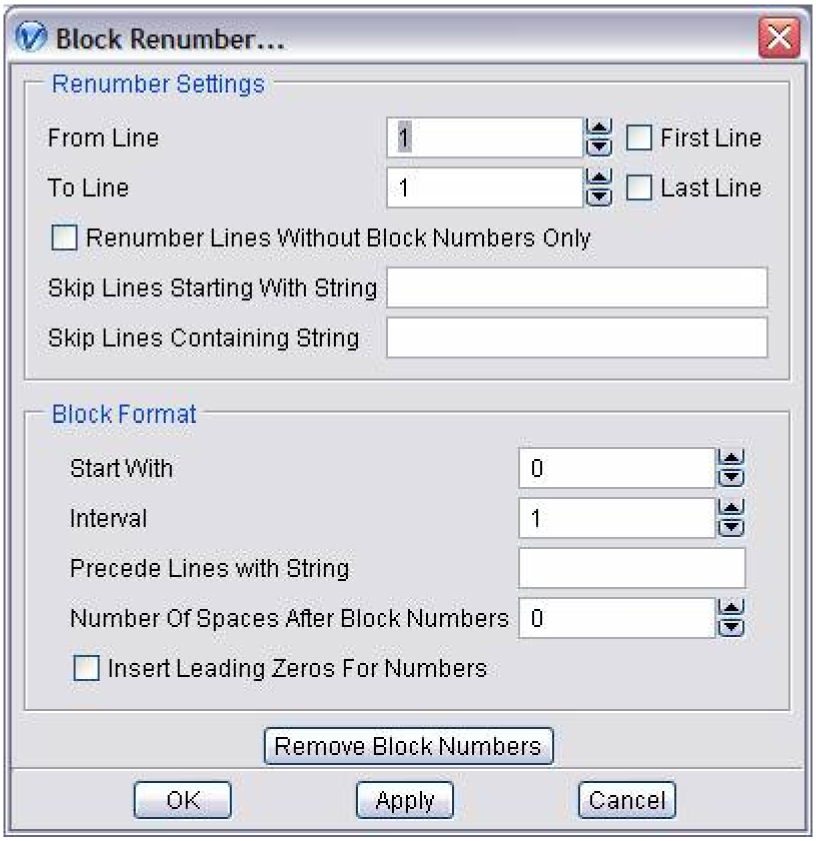
Cancel — Ignores your color setting changes and closes the NC Program Colors window.

VERICUT uses the definitions in the Word Format table, for the current control file, to determine what words go into each of the NC program feature categories. The following are the Word Format settings that VERICUT uses.

Comments:	Anything that falls between "Begin Comment" and "End Comment"
Variables:	Type "Special", Sub Type "Variable Name" or "Variable Tag"
Macros:	Type "Macro" or Type "Conditional" with "Word" related Subtype
Operators:	Type "Math" or Type "Condition" with a math related Subtype, or Type "Special" with SubType "Separator", or Type "Special" with SubType "Begin/End Type II"
Function:	Type "Function", or Type "Type II"
Specials:	Type "Special" with SubType = "Skip", "End of block", "Begin/End Data", "Console Message", "Ignore", "Sin84D_*", and "Num VAR Define"

Block Renumber window

The Block Renumber window is accessed from the NC Program (Info) window by selecting **Block Renumber**, in the **Utilities** menu.



Renumber Settings

Use the Renumber Settings options to specify where to start, and stop, renumbering.

From Line — Use this feature to specify the line in the NC program to begin the renumbering.

First Line — When toggled "On" renumbering will begin at the first line in the NC program.

To Line — Use this feature to specify the line in the NC program to stop the renumbering.

Last Line — When toggled "On" renumbering will stop at the last line in the NC program.

Renumber Lines Without Block Numbers Only — When toggled "On", only blocks within the specified range that do not currently have block numbers will be renumbered.

Skip Lines Starting with String — Enables you to specify a string of characters, that when found at the start of the block, will cause VERICUT to skip the block during the renumbering process.

Skip Line Containing String — Enables you to specify a string of characters, that when found anywhere within the block, will cause VERICUT to skip the block during the renumbering process.

Block Format

Use the Block Format options to specify the format, and number sequence, to use for block numbers. The formats specified must conform to the characteristics defined in the current control file.

Start With — Use this feature to specify the starting number to use for the renumbering sequence.

Interval — Use this feature to specify the increment between numbers to be used for the renumbering sequence.

Precede Lines With String — Use this feature to specify a character, or string of characters, to be used before the block number.

Number Of Spaces After Block Numbers — Use to specify the number of blank spaces to be used between the block number and the next item in the block.

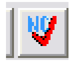
Insert Leading Zeros For Numbers — When toggled "On" leading zeros will be used in the block number.

Remove Block Numbers — Removes all block numbers from the NC program file. Block numbers are identified by the characteristics defined in the current control file.

OK — Applies the new block numbers, as specified, to the NC program file and closes the Block Renumber window.

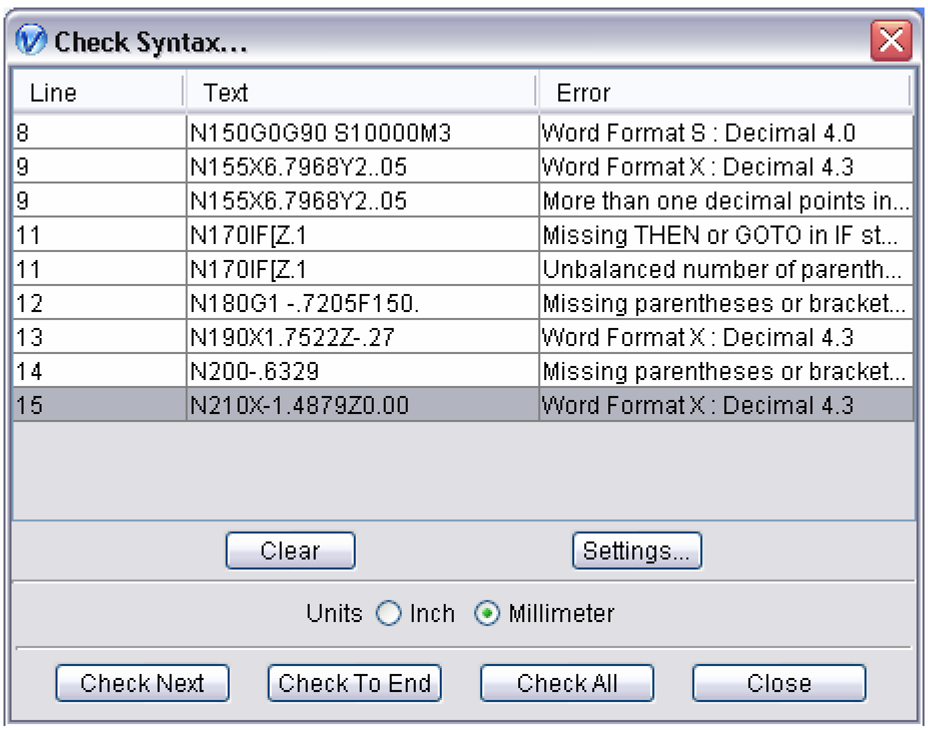
Cancel — Closes the Block Renumber window without changing the NC program file.

Check Syntax window

The Check Syntax window is accessed from the NC Program (Info) window, or from the NC Program Review window: NC Program listing area, by selecting **Check Syntax**, in the **Utilities** menu, or using  (**NC Program Syntax Check**) in the icon bar in either of the windows.

The features in this window enable you to check the NC program displayed in the NC Program (Info) window for syntax errors.

NOTE: This feature is not applicable to APT NC programs.



Check Syntax Error List

Line — This column contains the line in the NC Program listing that contains the syntax error. Click on the Line column header to sort The Syntax Error list by line number.

Text — This column contains the text of the block in the NC program listing that contains the syntax error.

Error — This column contains a description of the kind of syntax error found. Click on the Error column header to sort The Syntax Error list by error type.

Tip: Clicking on any syntax error in the Check Syntax error list moves the cursor to the corresponding block in the NC program displayed in the NC Program (Info) window.

Clear — Clears ***all*** errors from the Check Syntax error list.

Settings — Displays the Word Format window: Syntax Check tab enabling you to view the current syntax checking settings and modify them if necessary. See **Word Format window: Syntax Check tab**, in the Configuration Menu section of *VERICUT Help* for more information.

Units — Use to specify which word format string to check against in the Word Format table. The default is the units value specified in the current control. Select either, **Inch**, or **Millimeter**.

Check Next — Starting at the current cursor location, VERICUT searches the NC Program for the next syntax error and adds it to the Check Syntax error list.

Check To End — Starting at the current cursor location, VERICUT searches to the end of NC Program and adds each syntax error that it finds to the Check Syntax error list.

Check All — VERICUT checks the entire NC program and adds each syntax error that it finds to the Check Syntax error list.

Close — Closes the Check Syntax window.

Status window

VERICUT Users:

VERICUT Location: **Info menu > Status**

VERICUT toolbar short cut: 

Mold and Die Users:

Mold and Die Location: **Preview and Optimize page > View Optimization Info While Cutting**

Notebook Feature: 

Cutter Grinder Users:

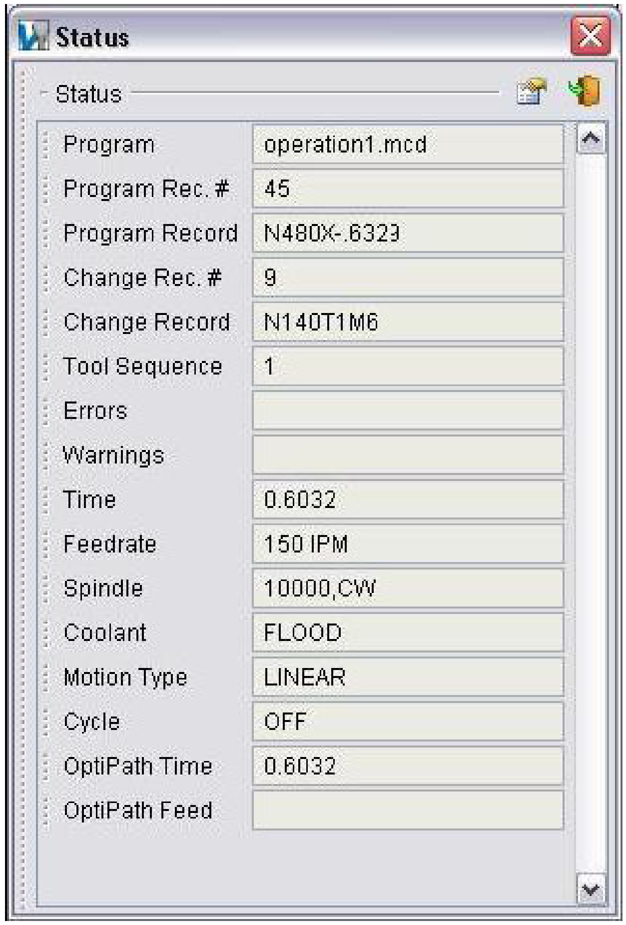
Cutter Grinder Location: **View Simulation page > View Info While Grinding**

Notebook Feature: 

Opens a window that provides status information about the simulation, and what would be occurring on the NC machine. Status information includes: tool path record being processed, machine and tool tip locations, information about the cutting tool, machining conditions, errors and warnings detected by VERICUT. The **Status** window can be left open for monitoring, minimized, closed and opened at any time. Information displayed is configurable, and is updated after each tool path record is processed. You can re-configure the window at any time.

The **Status** window is also one of the dockable windows enabling you to dock it inside the VERICUT main window if you choose. See **Dockable Windows** in the *Getting Started with VERICUT* section of *VERICUT Help* for additional information.

NOTE: When the **Status** window is docked, make sure that you click in the window so that it becomes the "active" window before using F1 to get help specific to the window. Otherwise F1 will go to the CGTech Help Library.



 **Configure —** Displays the Status window in configuration mode enabling you to specify status features that you want displayed in the Status window. Click on the icon to enter configuration mode. Click the icon again to return to the normal display mode.

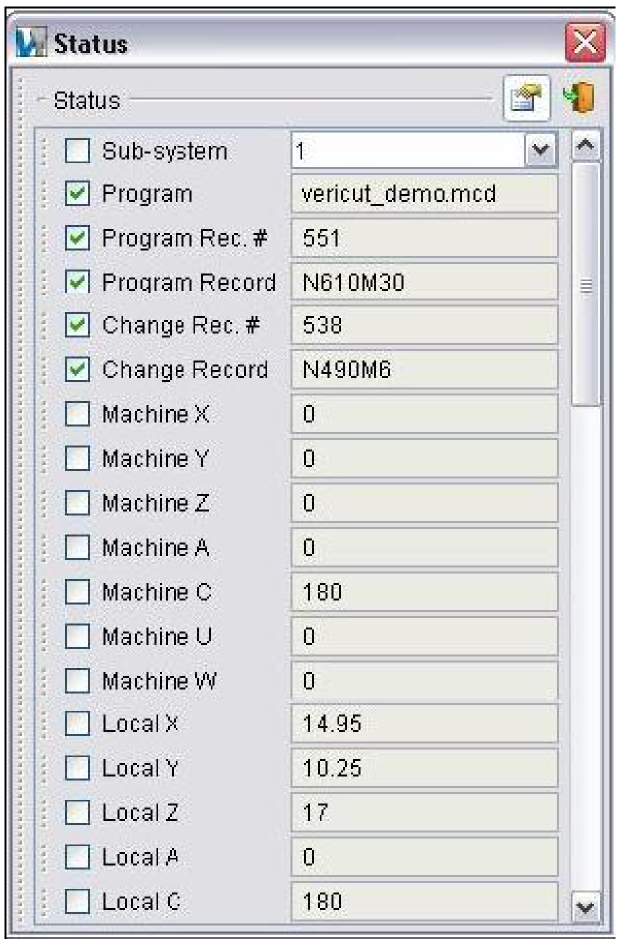
 **Close Window —** Closes the Status window.

Tips:

1. Status is recorded by the **File menu** > **Images** > **Record** function, and can be seen in the Status window when playing back the Image file.
2. You can send status information to a text Status file. To accomplish this, set the **CGTECH_STATUS** environment variable to the name of the file before or when VERICUT is executed.

Status window, configuration mode

The Status window, in configuration mode, enables you to select the items you want displayed in Status window. Click in the box to the left of each feature you want included in the Status window display. When toggled "On", a checkmark is displayed. Click on the checkmark to toggle "Off" the feature.



The following features are available for display in the Status window:

SubSystem — When a machine with more than one subsystem is in use, this option controls which subsystem status to show.

Program — The name of the current NC program.

Program Rec.# — The line number of the current NC program record.

Program Record — Current NC Program record being processed. Both the record text and sequential line number in the file are shown.

Change Rec.# — The line number of the last tool change record.

Change Record — The last tool change record processed. Both the record text and sequential line number in the file are shown.

Machine Axes — Locations of machine axes resulting from processing G-Code NC Program data.

Local Axes — The display will be similar to the Machine Axes (only showing the axis that exists for that subsystem). The contents of the "Local Axes" fields will represent the current X, Y, Z, A, B, C, U, V, W, A2, B2, C2 axis within the local coordinates. Typically, this will correspond to the values entered in the MCD file (except when specifying machine coordinates, or incremental values).

Tool Tip — Current X Y Z tool tip location and I J K tool axis orientation with respect to the active tool path coordinate system.

Tool Sequence — Sequential number of the last tool change event.

Tool Number — Tool number in use (if specified). Examples follow.

Tool path type:	Sample tool chg record:	Status window "Number":
G-Code data	T10M6	10
APT	CUTTER/...	none
APT	PPRINT/VERICUT-TC...	none
APT	LOADTL/1	1
UG CLS	LOAD/TOOL,4,...	4

Tool ID — ID of a tool retrieved from a VERICUT Tool Library file, if any. A blank field indicates the current tool did not come from a tool library. "OVERRIDE" indicates that the current tool has been specified via the **Override** feature. If the ID differs from the **Number** field value described above, the tool was specified via a tool list.

Tool Geometry — Cutter shape geometry. The method used to define the cutter shape is shown in square brackets "[ ]" followed by the description. "PROFILE" indicates a profile type of cutter definition.

Tool Description — Description of a tool retrieved from a VERICUT Tool Library file. A blank field indicates a description for this tool was not defined in the library, or the tool did not come from a tool library.

Errors — Quantity of errors detected by VERICUT. A blank field indicates no errors detected. Consult the Log file for details about the errors.

Warnings — Similar to **Errors** above, except applies to warnings detected.

Time — Time anticipated to machine the part (as simulated by VERICUT). To calculate anticipated machining time, VERICUT records time for tool motions based on specified cutting feed rates and rapid rates.

Time % — Percentage of time in feed rate mode. This value can be helpful when judging tool path efficiency. For example, a high value suggests an efficient tool path, since a value less than 100 percent means that the tool is moving at rapid feed and is probably not removing material.

Distance — Tool movement distance in inches or millimeters, depending in the VERICUT session configuration.

Distance % — Percentage of tool movement distance in feed rate mode. Similar to the **Time** versus **Time%** relationship described above, this value can also be helpful when judging tool path efficiency.

Feedrate — Current programmed feed rate. A blank field indicates no records that control feed rate that have been processed.

Spindle — Current programmed spindle speed. A blank field indicates no records that control spindle speed that have been processed.

Coolant — Current programmed coolant condition. A blank field indicates no records that control coolant that have been processed.

Motion Type — Rapid / Linear / CW / CCW / Thread / Nurbs / Poly

Abs/Inc — Absolute / Incremental

Motion Plane — XY / YZ / ZX

Units — Inch / Metric

Spindle Mode — CSS / RPM

Compensation — On / Off

Cycle — Off / Drill / Face / Deep / Tap /Bore / Mill / Thru / Bore Orient / Bore Drag

Interpolation — None / Polar / Cylindrical

OptiPath Time — Time anticipated to machine the part with an optimized tool path file. This field is only updated when the current tool path if being optimized by OptiPath. To estimate machining time saved by OptiPath, compare this value against the Time field value.

NOTE: OP Time ***DOES NOT*** consider feedrates output as a result of Acceleration/Deceleration optimization adjustments.

OptiPath Feed — Optimized feed rate, as calculated by OptiPath.

OptiPath Spindle — Optimized spindle speed, as calculated by OptiPath.

The following features are in effect when OptiPath Mode is set to **On**, **Prompt While Cutting** or **Learn From NC Program**. They report maximum volume removal rate and chip thickness for each cut calculated by OptiPath for the programmed feedrate.

Volume Removal — Volume Removal Rate is calculated for any **Optimization Method** and for any tool valid for OptiPath.

Chip Thickness — Chip Thickness is calculated only if the **Optimization Method** for the current tool is either **Chip Thickness**, or **Chip Thickness** and **Volume Removal**. **Chip Thickness** is not calculated when OptiPath optimizes by any other method. When **OptiPath Mode** is set to **Learn From NC Program**, **Chip Thickness** is calculated only for tools, which have a convex profile.

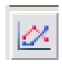
Cutter Comp Value — Current cutter compensation value.

Air Time % — The percent of time spent cutting air.

Polar Coordinates — Display the status of polar coordinate input (ON/OFF).

Graphs window

VERICUT Location: **Info menu > Graphs**

VERICUT toolbar short cut: 

The Graphs window enables you to display a **Tool Use** graph showing information related to tool use and/or the **Cutting Conditions** graph showing selected cutting conditions. Both the **Tool Use** graph and the **Cutting Conditions** graph are refreshed after each setup.

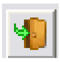
Common Features:

Time Interval — Use to specify the time interval represented by the **Tool Use** graph display. Use the **Time Interval** text field to specify the time interval represented by the **Tool Use** graph.

/2 — Use to reduce the displayed time interval by a factor of 1/2.

X2 — Use to increase the displayed time interval by a factor of 2.

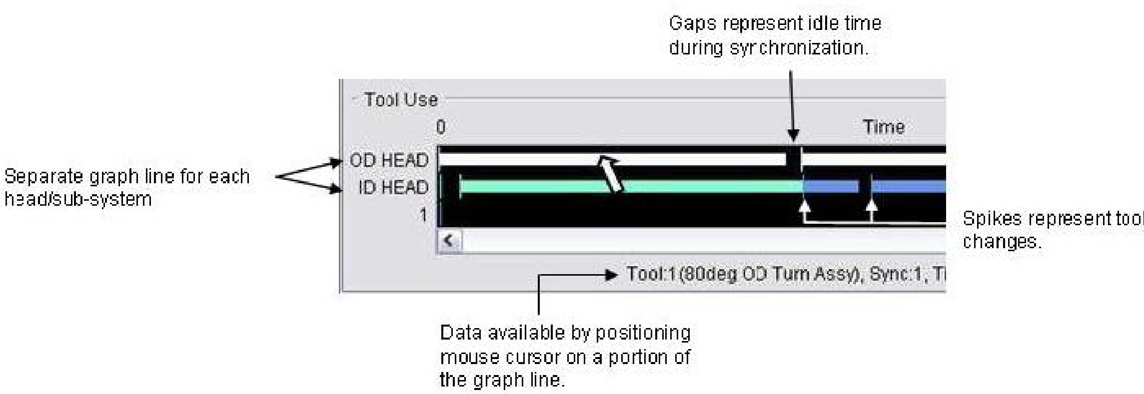
 **Configure —** Displays the Graphs window in configuration mode enabling you to specify graph display characteristics and features. Click on the icon to enter configuration mode. Click the icon again to return to the normal display mode.

 **Close Window —** Closes the Graphs window.

Tool Use Graph:

When toggled "On", the **Tool Use** graph is displayed which shows the machining time spent by each simulated tool. Each tool that cuts the part is shown in the graph with its cut color. Tool changes are indicated by a spike in the graph line. Positioning the mouse cursor on or above a graph line displays data about that tool. A 4-axis turned part has 2 graph lines; one for each turret. Machines with additional heads/sub-systems will have a graph line displayed for each. A gap in a graph line indicates idle time spent waiting for a turret/sub-system to synchronize. Positioning the mouse cursor above a gap in the graph displays the amount of idle time.

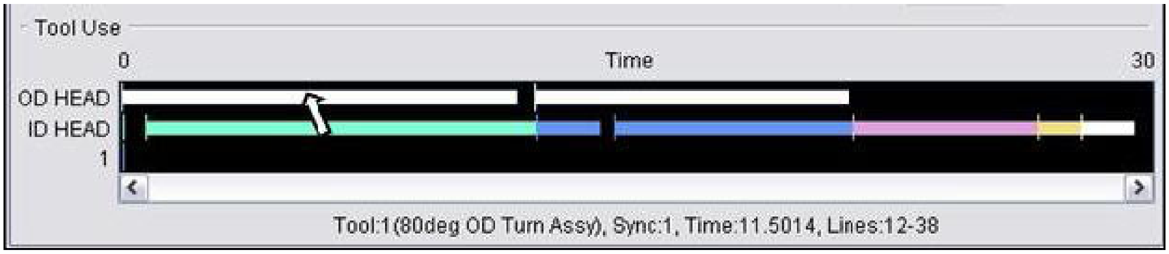
Tool Use graph features:



Sample Tool Use graphs:

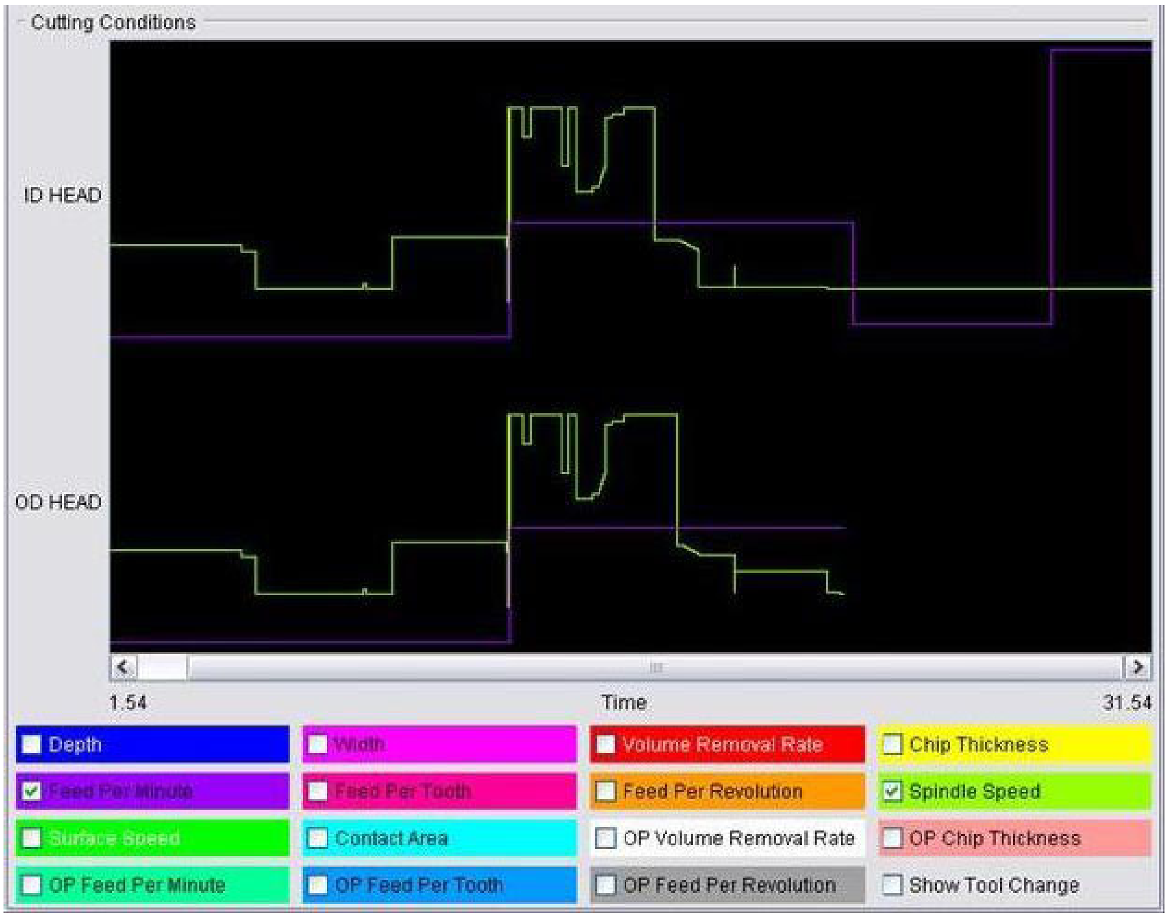
Tool Use graph from a milling NC Program:

Tool Use graph from 4-axis merged turning NC Program:



Cutting Conditions Graph:

When toggled "On", the **Cutting Conditions** graph is displayed which shows changes in the selected cutting condition features. The picture below shows the **Cutting Conditions** graph display for 4-axis merged turning NC Program, with the Feed Per Minute and Spindle Speed features toggled "On".



Cutting Condition graph features:

Select any of the following "cutting condition" features to be displayed in the **Cutting Conditions** graph. Each feature can be toggled On/Off at any time and the graph will be immediately updated.

Depth — Depth of cut.

Width — Width of cut.

Volume Removal Rate — Volume removal rate based on programmed values.

Chip Thickness — Chip thickness based on programmed values.

Feed Per Minute — Feed per minute based on programmed values.

Feed Per Tooth — Feed per tooth based on programmed values.

Feed Per Revolution — Feed per revolution based on programmed values.

Spindle Speed — Spindle speed.

Surface Speed — Surface speed.

Contact Area — The area of the tool that is in contact with the material.

OP Volume Removal Rate — Optimized volume removal rate, as calculated by OptiPath.

OP Chip Thickness — Optimized chip thickness, as calculated by OptiPath.

OP Feed Per Minute — Optimized feed per minute, as calculated by OptiPath.

Op Feed Per Tooth — Optimized feed per tooth, as calculated by OptiPath.

OP Feed Per Revolution — Optimized feed per revolution, as calculated by OptiPath.

Show Tool Change — Use to show tool changes in the Cutting Conditions graph. Tool changes are represented by a vertical line, in the cut color, at the point of the tool change.

Graphs window, configuration mode

The Graphs window, in configuration mode, enables you to specify graph display characteristics and features to be graphed. Many of the features are identical to the ones in the Graphs window and are described there.

Tool Use — When toggled "On" (check mark displayed), displays the **Tool Use** graph which shows the machining time spent by each simulated tool. Each tool that cuts the part is shown in the graph with its cut color. Click in the box to the left of **Tool Use** to toggle the graph On/Off.

Cutting Conditions — When toggled "On" (check mark displayed), displays the **Cutting Conditions** graph which shows changes in the selected cutting condition features. Click in the box to the left of **Cutting Conditions** to toggle the graph On/Off.

Select any of "cutting condition" features to be displayed in the **Cutting Conditions** graph from the list below the graph area.

Color Pallets — Use the color pallet associated with each cutting condition feature to specify the color that is to represent the feature in the graph. Click on the icon to display the color pallet for your color selection.

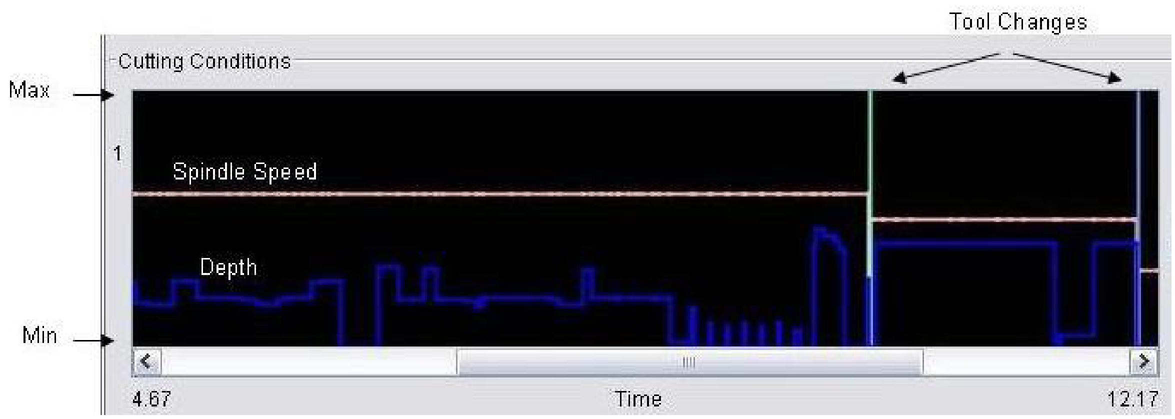
Min/Max values — Use the Min/Max values associated with each cutting condition feature to specify the lower/upper limits of the range of values representing the vertical component of the graph (see the picture below).

Show Tool Change — Use to show tool changes in the **Cutting Conditions** graph. Tool changes are represented by a vertical line, in the cut color, at the point of the tool change (see the picture below).

Set All — Select all available features to be displayed in the **Cutting Conditions** graph.

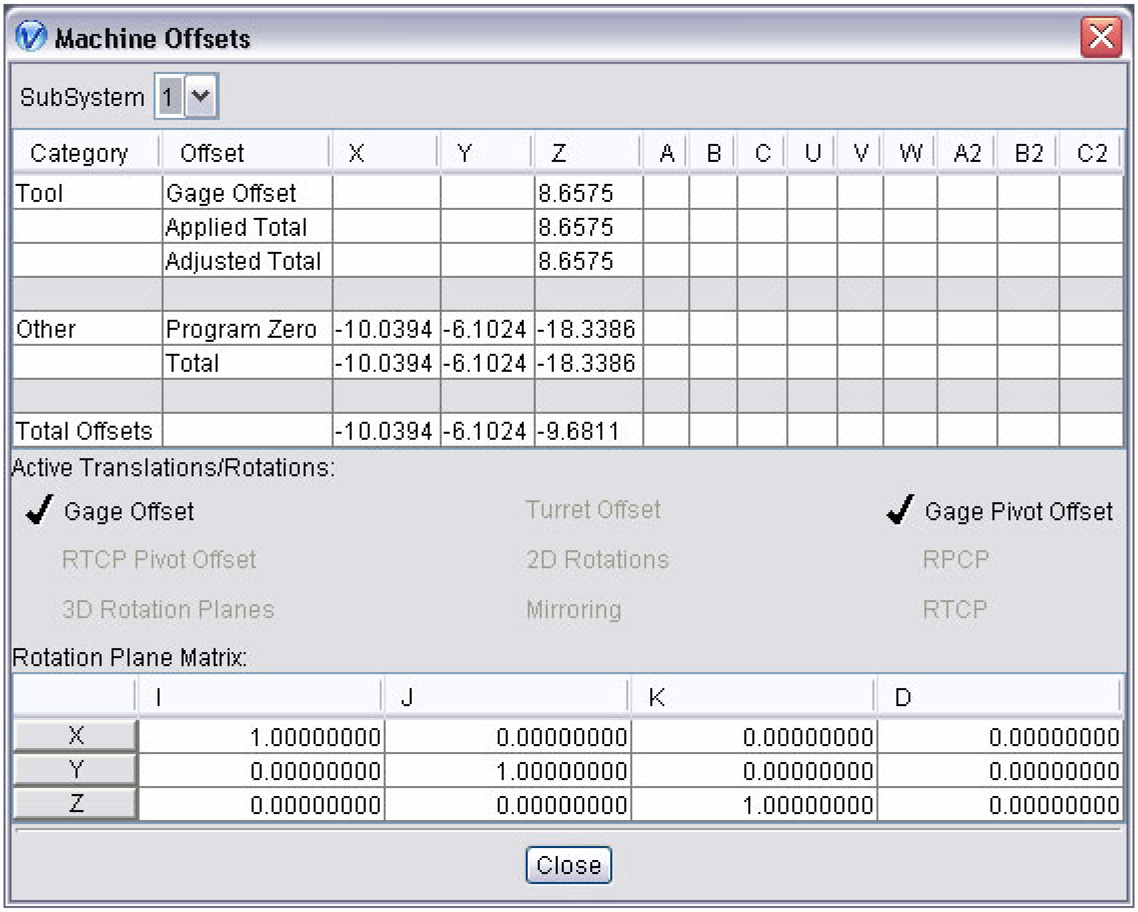
Clear All — Clear all selected features.

Sample Cutting Conditions graph:



Machine Offsets window

VERICUT Location: **Info menu > Machine Offsets**



Opens a window that displays the values that are set for each machine offset and whether the particular machine offset is currently active. If you "**Single Step**" through the toolpath, the displayed values update at the end of each motion. When you are using "**Play to End**", the displayed values are updated each time you "**Stop**" processing or when processing stops at the end of the NC program. The Machine Offsets window is intended for debugging purposes only. The window displays current values set for each offset and whether the offset is currently active. You cannot modify the values directly in the Machine Offsets window. The values must be edited in the table or macro in which they were originally set.

The offsets are grouped into 4 categories: **Work& Shift**, **Tool**, **Other**, and **Total Offsets**, as shown in the **Category** column in the window above. Each Category can consist of a number of different machine offsets and are displayed in the **Offsets** column. The remaining columns indicate the offset values that are set/applied for each machine axis. Each particular offset will only displayed if it contains non-zero values. If all offsets within a Category are zero, then the Category will not be displayed. The exception to this rule is the Total Offsets Category which will always be displayed.

This window also contains a series of flags associated with offsets and transformations.

SubSystem — All values displayed in the Machine Offsets window are specific to a particular subsystem. If the machine configuration has more than one subsystem, use this feature to select the desired subsystem.

Offset Categories

Work and Shift:

These offsets are in a separate category because the **DynamicWorkOffsets** feature applies to these offsets.

Base Work Offset — Base Work Offset is affected by the following macros:

SetBaseWorkOffset
CancelWorkOffsets
EnableWorkShiftOffsets
DisableWorkShiftOffsets
CancelAllWorkOffsets
BaseWorkOffsetValues

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

NOTE: This offset is automatically applied during reset.

Work Offset — Work Offset is affected by the following macros:

WorkCoord
AdditionalWorkCoord
SetBaseWorkOffset
CancelWorkOffsets
EnableWorkShiftOffsets
DisableWorkShiftOffsets
CancelWorkOffsets
CancelAllWorkOffset

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

NOTE: This can also be set from the **Configuration menu > Control Settings: Offsets tab**.

Secondary Work Offset — Secondary Work Offset is affected by the following macros:

SecondaryWorkCoord
SecondaryRefWorkCoord

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

Shift 1 — Shift 1 is affected by the following macros:

ChangeWorkCoord
AbsoluteShift, IncrementalShift
EnableWorkShiftOffsets
DisableWorkShiftOffsets
CancelShiftOffsets
CancelAllWorkOffsets
SiemensShiftOffsetA
SiemensL137COffset
SiemensL137CoffsetCancel

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

Shift 2 — Shift 2 is affected by the following macros:

AbsoluteShiftNum
OkumaCopyStart
OkumaCopyEnd
OkumaShiftRotate
OkumaCancelShiftRotate
SiemensShiftOffsetB

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

Shift 3 — Shift 3 is affected by the following macros:

NumLengthCompX
NumLengthCompY
NumLengthCompZ

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

Total — The combined total of all of the above offsets.

Adjusted Total — The resulting total after the corresponding rotations have been applied.

Tool:

In many cases, an offset will be applied to the machine based on the tool offset and the orientation of the tool. The tool offset can actually be composed of several pieces; some of which are mutually exclusive.

Gage Offset — This value is typically calculated from the tls file. It is the "Gage Offset" minus the "Ctrl Pt". You can override the value from the tls file with an entry in the Gage Offset Table. This value is typically set with a tool change. Its value can also be updated with a call to the **ToolOffsetUpdate** macro

Tool Length Offset — This value is typically only used if the Programming Method is set to Tool Length Compensation. This value can, however, be used in all 3 Programming Methods. The value will be set based on the corresponding entry in the Tool Length Compensation table. If no entry exists, and the Programming Method is set to Tool Length Compensation, this value will be set to the Z value of the Input Program Zero + the Z value of the Gage Offset. Tool Length Offset is affected by the following **ToolLengthComp...** macros:

ToolLengthCompUpdate
ToolLengthCompPos
ToolLengthCompNeg
ToolLengthCompOff
ToolLengthCompValue ToolLengthCompToolnum
SiemensPlaneSelectionArg

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

Tool Nose Comp — Tool Nose Comp is affected by the following **ToolNoseComp...** macros:

ToolNoseCompApply
ToolNoseCompCancel
ToolNoseCompToolNum
ToolNoseCompValue

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

Turret Offset — The turret offset is defined to be the XYZ distance from the active tool to the center of its corresponding Turret. This offset can be turned on and off. When turned on, this offset will be set with calls to either **TurretActivateTool** or

TurretToolChange macros. It can also be reset to zero if the current gage offset is set to zero and the apply turret offset flag is set accordingly. This flag is set with the **ApplyTurretOffset** macro.

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

GagePivotOffset — This offset is used as an alternative approach to RTCP (Dynamic Tool Offsets). RTCP has been based on the concept of the spindle face being the driven point when there is no tool in the spindle. When a tool is loaded, and then rotated, both the gage offset and the pivot offset needs to be adjusted. When driving the center of the pivot, and then adjusting for the gage and pivot offset, the calculation is different. This offset is used for this latter approach. Its functionality is currently limited. The capability is turned on with the **ApplyGagePivotOffset** macro, and its value is updated whenever the tool offsets are updated. This capability is mutually exclusive with RTCP.

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

Current RTCP Pivot Offset — The pivot offset can set be set in the **RTCP Pivot Offset Table**. If the corresponding entry does not exist, it will be auto-calculated based on the active tool component and the position of the corresponding parent rotary components. The application of this offset can be turned on, or off, using the **ApplyPivotOffset** macro. When applied, it will be adjusted based on the orientation of the tool. It assumes that in its initial state, no offsets are to be applied. The initial state is set at start of processing, and can be updated with a call to the **SetPivotOffset** macro.

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

Applied Total — This is the sum of the Tool offsets, which are currently being applied.

NOTE: Not all of the offsets listed will be applied. For example: **Gage Offset** values might be specified, but may not be applied because the **Programming Method** is set to **Gage Length**.

Adjusted Total — This value takes the applied total, possibly adjusts it based on the orientation of the tool, and then applies it to the corresponding axis. See the following macros:

XToolOffsetCompName
YToolOffsetCompName
ZToolOffsetCompName

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

Other:

Dynamic 3D Tool Offset — See **Tool3dOffset** in the *VERICUT Macros* section in the *CGTech Help Library*.

3D Tool Length Comp — The Tool Length Offset as describe above, is an offset, which applies only to Z. There is special handling of this value within the software, and has therefore not been grouped with the Tool offsets. This offset can be applied to any axis. See **FanucToolLengthComp** and **ToolLengthCompOff** in the *VERICUT Macros* section, in the *CGTech Help Library*.

Input Program Zero — This offset is typically set in the **Input Program Zero (Special Z) table**. The values are automatically activated during initialization, and can be updated with calls to **UpdateIPZ** and **AutoUpdateIPZ** macros. When the **Programming Method** is set to **Tool Length Compensation**, the Z value of the IPZ will not be applied. In this programming method, the Z value might later be used in the calculation of the Tool Length Offset.

Input Machine Zero — This offset is set in the **Machine Zero table** and is automatically activated during initialization. Typically, this offset is not used.

Miscellaneous — Other offsets that might be applied – intentionally not defined.

Total — Sum of the above offsets.

Total Offsets:

Sum of all offsets being applied.

NOTE: This does not include rotation planes.

Active Translations/Rotations:

The following become highlighted when the particular translation/rotation is active. **RTCP Pivot Offset** is shown in its "active" state in the sample Machine Offsets window shown at the start of this topic.

Gage Offset — If the Programming Method is set to Tool Tip, the Gage Offset will be applied.

RTCP Pivot Offset — This flag is set if in RTCP mode and the pivot offset is applied. See macros:

RtcpOn
RtcpOff, **RotaryControlPointOnOff**
ApplyPivotOffset

This flag is also set if in a special Fanuc Tool Length Comp Mode and the apply pivot offset flag is set to 2. See macros:

FanucToolLengthCompAxisOn
ToolLengthCompOff

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

3D Rotation Planes — This flag is set if Rotation Planes are currently being applied. See macros:

RotationPlane
RotationPlane2
RotationPlaneCancel
RotationPlaneCancel2
FidiaRotate
FidiaRotate2
FidiaRotateCancel
FidiaRotateCancel2
NumPlane
VirtualYAxis

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

Turret Offset — This flag is set with the **ApplyTurretOffset** macro.

2D Rotations — 2D rotations are defined within the active plane (XY, YZ or ZX). The 2D rotation flag is set with calls to the following macros:

RotateXYZ
RotateCancel
RotateOrigin
RotateMacro

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

NOTES:

1. Optimization is suspended when 2D Rotation is turned on. The following warning message is output: "Optimization turned off while 2D Rotation is active" when optimization is suspended for this reason.
2. If Debug Optimization is turned on, a DEBUG message will be printed on every line which is suspended for this reason.

Mirroring — If mirroring about any axis is set, this mirror flag will be set. Mirroring is activated with calls to the following **Mirror...** macros:

MirrorX
MirrorY
MirrorZ
MirrorA
MirrorB
MirrorC
MirrorA2
MirrorB2
MirrorC2
MirrorXValue
MirrorYValue
MirrorZValue
MirrorAValue
MirrorBValue
MirrorCValue
MirrorA2Value
MirrorB2Value
MirrorC2Value
MirrorCancel
MirrorXCancel
MirrorYCancel
MirrorZCancel
MirrorACancel
MirrorBCancel
MirrorCCancel
MirrorA2Cancel
MirrorB2Cancel
MirrorC2Cancel

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

Gage Pivot Offset — This feature is turned on with the **ApplyGagePivotOffset** macro.

RPCP — This feature can be set using the **Configuration menu > Control Settings: Rotary tab**. It can also be set with the following macros:

RpcpOn
RpcpOff
RotaryControlPointOnOff

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

RTCP — This feature can be set using the **Configuration menu > Control Settings: Rotary tab**. It can also be set with the following macros:

RtcpOn
RtcpOff

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

Rotation Plane Matrix:

The Rotation Plane Matrix represents the cumulative effect of all translations and rotations currently applied to the rotation plane (also known as the working plane). Its twelve parameters reveal the geometric attributes of the local (transformed) coordinate system (CSYS) relative to the machine origin. This representation only includes translations and rotations resulting from using the **RotationPlane..**. macros. The matrix values ***do not*** include the effect of using the **Rpcp...** macros or the **VirtualXAxis** / **VirtualYAxis** macros.

For information about these, and all, VERICUT macros, see the *VERICUT Macros* section, in the *CGTech Help Library*.

The format of the matrix table is as follows:

	I	**J**	**K**	**D**
X	I1	J1	K1	D1
Y	I2	J2	K2	D2
Z	I3	J3	K3	D3

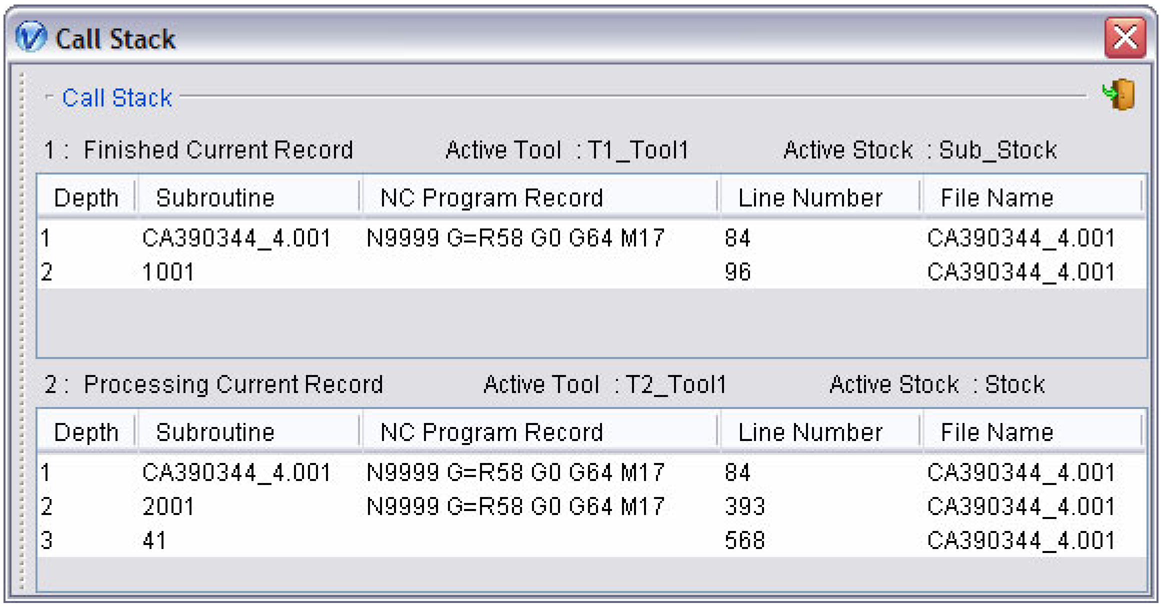
Each row represents an axis of the local CSYS. The first three columns represent the vector associated with each axis: I1, J1, K1 as the positive X-axis vector; I2, J2, K2 as the positive Y-axis vector; and I3, J3, K3 as the positive Z-axis vector. The fourth column values D1, D2, D3 represent the coordinates of the origin point of the local CSYS.

NOTE: If you prefer to see the Matrix Table displayed with the I, J, K along the vertical axis and the X, Y, Z along the horizontal axis, set the environment variable, **CGTECH_MATRIX_FORMAT=VERTICAL**.

Call Stack window

VERICUT Location: **Info menu > Call Stack**

The features in the Call Stack window enable you to track information like subroutine calls, and which subsystem is currently active. The picture below shows the current status of two subsystems.



The following information is displayed for each subsystem:

The processing status of the current record.

Active Tool — Displays the name of the active tool component.

Active Stock — Displays the name of the active stock component.

NOTE: The active stock defaults to the first stock in the component tree. If multiple stocks exists, the active stock needs to be set.

Also see: **ActivateToolSubsystem**, and **SetActiveStockName**, in the *VERICUT Macros* section, in the *CGTech Help Library*.

Each record displays the following information:

Depth — Shows the depth, or level of subroutine calls. For example, in the picture above, Subsystem "2" shows 3 levels of subroutine calls. Depth 1 represents the main program, Depth 2 represents a subroutine (2001) which was called from the main program, and Depth 3 represents a subroutine (110) called from the Depth 2 subroutine.

Subroutine — Shows either the name of the main program, or subroutine, represented by the record.

NC Program Record — Shows the text of NC program record calling the subroutine.

Line Number — Shows the number of the line in the file represented by the record.

File Name — Shows the name of the file that the NC program, or subroutine, is in.

 **(Close Call Stack) —** Closes the Call Stack window.

Tip:

The records in the Call Stack window are linked to the NC Program (Info) window. Clicking on the "calling" record in the Call Stack window will highlight the corresponding block in the NC Program (info) window. Clicking on a subroutine record in the Call Stack window, highlights the first line in the subroutine in the NC Program (Info) window.

File Summary window

VERICUT Users:

VERICUT Location: Info menu > File Summary

Mold and Die Users:

Mold and Die Location: **Analyze, View Files, Print page > Show Files Used**

Notebook Feature: 

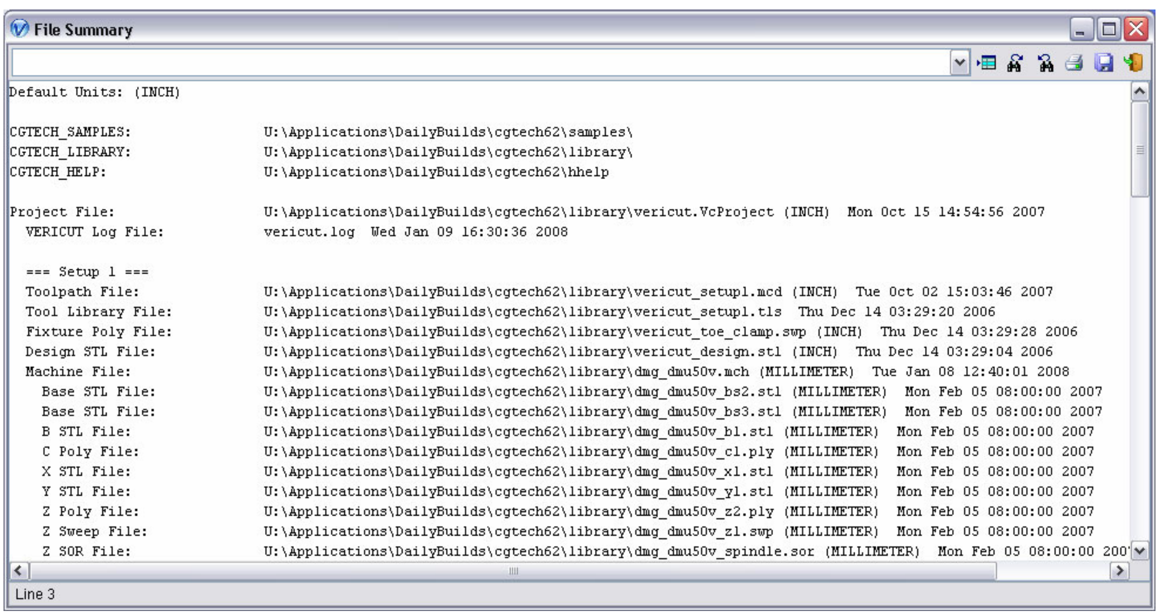
Cutter Grinder Users:

Cutter Grinder Location: **Analyze, View Files, Print page > Show Files Used**

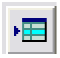
Notebook Feature: 

Opens a window that provides a summary of files with their creation dates, and environment variables, currently being used by VERICUT. All standard text viewing features are supported, such as: search forward/backward, go to line number, print, etc.

File Summary window:



Line Number or Search Text — Use this text field to enter a line number, or a string of text, to search for.

 **Goto Line Number —** Moves the cursor in the file listing to the line number specified in the Line Number or Search Text field.

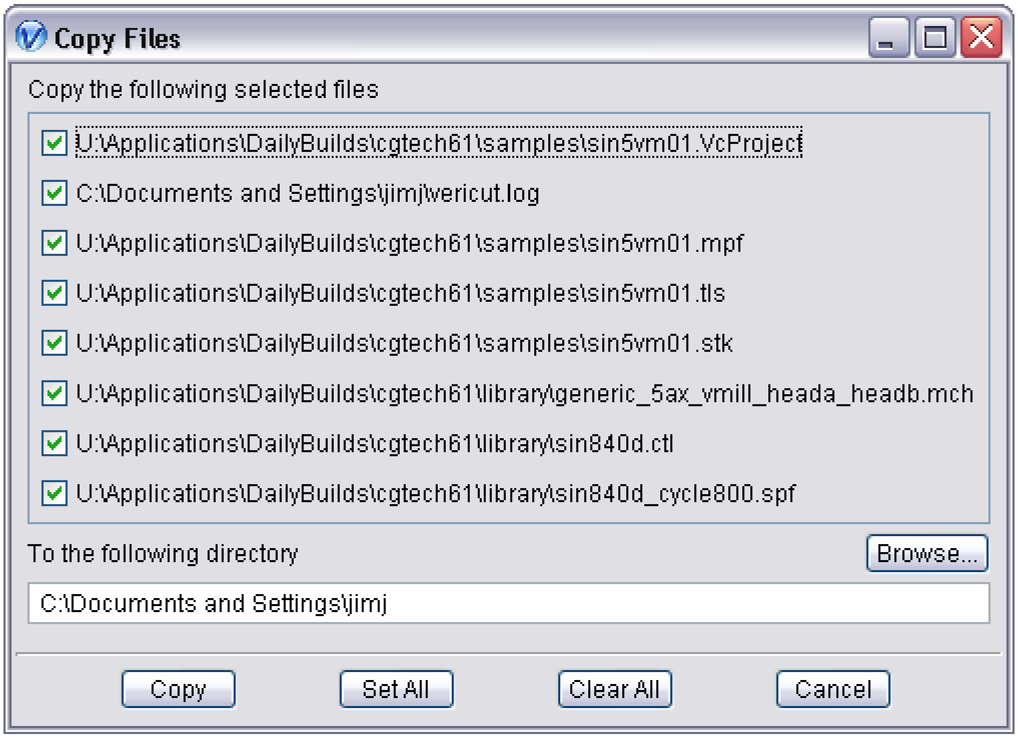
 **Search Forward —** Searches forward in the file listing for the text string specified in the **Line Number or Search Text** field.

 **Search Backward —** Searches backward in the file listing for the text string specified in the **Line Number or Search Text** field.

 **Print —** Print the file.

 **Copy All Files —** Displays the **Copy Files** window enabling you to copy any, or all, of the listed files to a specified directory. Click on the check boxes to select or deselect the files you want copied.

Copy Files window



Use the "**To the following directory**" text field to specify the path to the directory, or use the **Browse** button and use the window that displays to select the directory, to receive the copied files.

Copy — Use to complete the copy when you have finished selecting the files.

Set All — Use to select all of the files in the list for copying.

Clear All — Use to clear the check boxes for all of the files in the list.

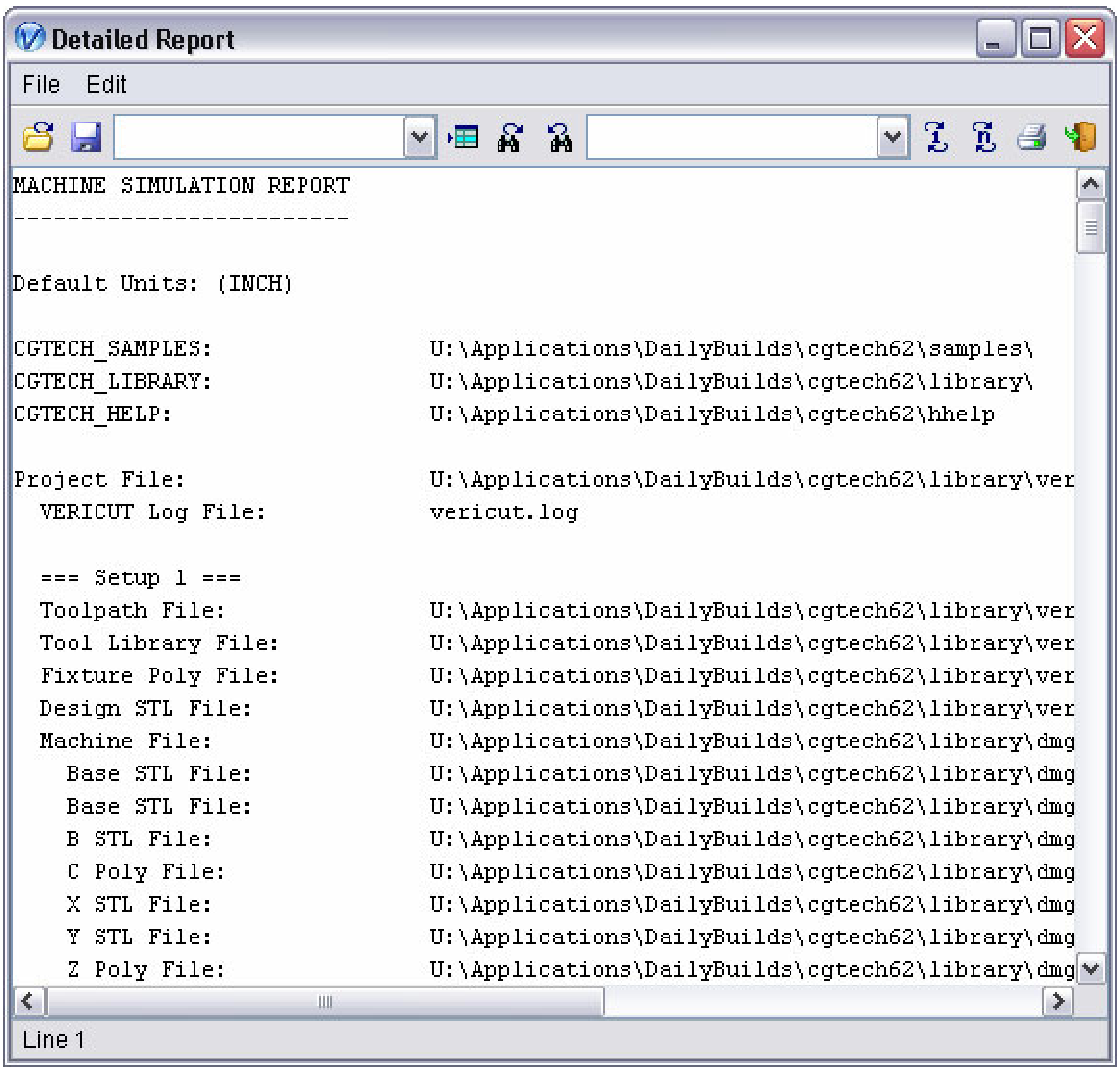
Cancel — Use to close the **Copy Files** window without copying any of the files.

 **Close Window —** Closes the **File Summary** window.

G-Code Report

Location: **Info menu > G-Code Report**

Creates a G-Code Report file containing information about how the current NC control configuration will interpret a G-Code NC program file. Reported information includes: machine codes present in the NC Program and macros (actions) they will call, variables used (if any), cutting tools used, subroutines defined and referenced, and more.



[Click here to see a sample G-Code Report file.]

Menu Bar:

The menu bar located at the top of the window provides easy access to major functions. Each menu contains groups of related functions. Left-click on any menu name to display the list of functions in that menu. Click on the function in the menu you want to use. The name of the current file is also displayed in this area.

File:

Open — Open a file.

Save — Save the current file.

Save As — Save the current file under a different name.

Print — Print the file.

Exit — Close the window.

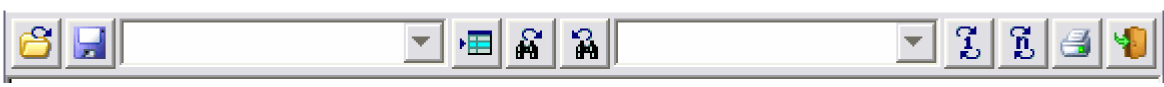
Edit:

Cut — Cuts the highlighted text in the file listing and puts it in the paste buffer.

Copy — Copies the highlighted text in the file listing to the paste buffer.

Paste — Puts the contents of the paste buffer at the location of the cursor in the file listing.

Icon Bar:

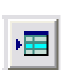


The items in the Icon Bar enable you to search for, and/or replace, specific items in the file listing, print the file, or exit the window. Moving the cursor over the icon will display name of the option. Each feature (from left to right) is described below.

 **Open File —** Opens a file selection window enabling you to select another file.

 **Save File —** Save the current file.

Line Number or Search Text — Use this text field to enter a line number, or a string of text, to search for.

 **Goto Line Number —** Moves the cursor in the file listing to the line number specified in the Line Number or Search Text field.

Search Forward — Searches forward in the file listing for the text string specified in the Line Number or Search Text field.

Search Backward — Searches backward in the file listing for the text string specified in the Line Number or Search Text field.

Replacement Text — Use this text field to enter a "replacement text" string.

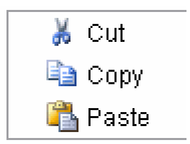
Replace One — Replace one occurrence of text string in the Line Number or Search Text field (or the highlighted text in the file listing) with the text string in the Replacement Text field.

Replace All — Replace all occurrences of the text string in the Line Number or Search Text field with the text string in the Replacement Text field.

Print — Print the file.

Close Window — Closes the window.

Shortcut: Right-click in the Text File Edit window to display the following menu:

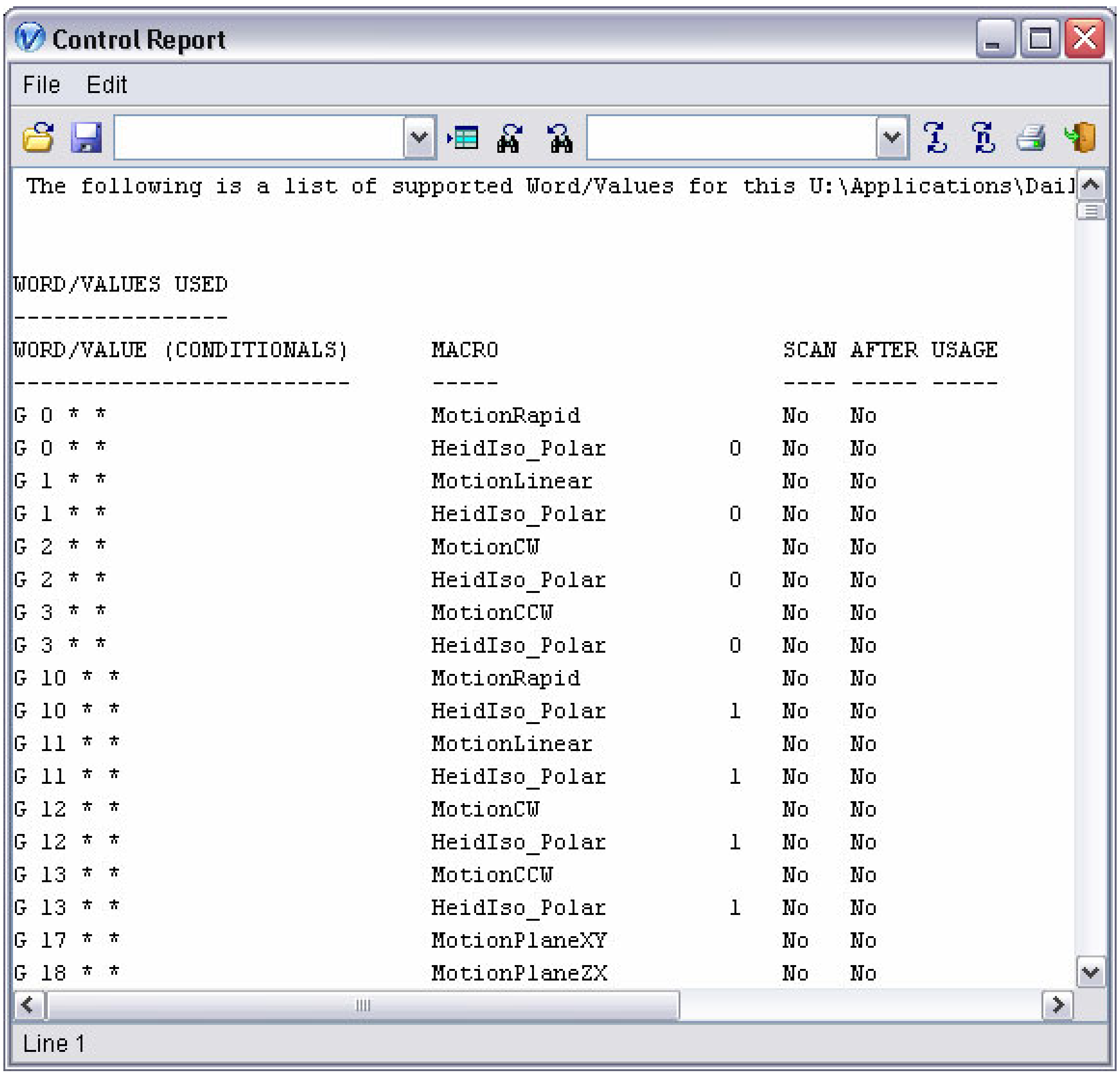


These features provide the same functionality as those described under Edit in the menu bar.

Control Report

Location: **Info menu > Control Report**

Creates a Control Report file containing information about how the current NC control configuration will interpret machine codes. Reported information includes all the G-Codes the control is configured to handle and macros (actions) they will call, subroutines, and other supported control features.



Click here to see a sample Control Report file.

Menu Bar:

The menu bar located at the top of the window provides easy access to major functions. Each menu contains groups of related functions. Left-click on any menu name to display the list of functions in that menu. Click on the function in the menu you want to use. The name of the current file is also displayed in this area.

File:

Open — Open a file.

Save — Save the current file.

Save As — Save the current file under a different name.

Print — Print the file.

Exit — Close the window.

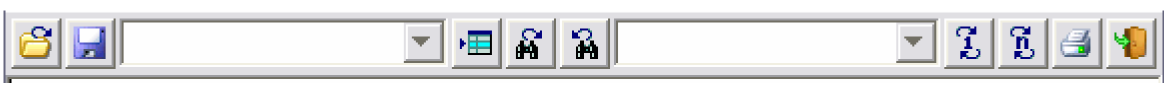
Edit:

Cut — Cuts the highlighted text in the file listing and puts it in the paste buffer.

Copy — Copies the highlighted text in the file listing to the paste buffer.

Paste — Puts the contents of the paste buffer at the location of the cursor in the file listing.

Icon Bar:

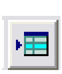


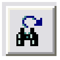
The items in the Icon Bar enable you to search for, and/or replace, specific items in the file listing, print the file, or exit the window. Moving the cursor over the icon will display name of the option. Each feature (from left to right) is described below.

 **Open File —** Opens a file selection window enabling you to select another file.

 **Save File —** Save the current file.

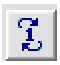
Line Number or Search Text — Use this text field to enter a line number, or a string of text, to search for.

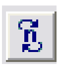
 **Goto Line Number —** Moves the cursor in the file listing to the line number specified in the Line Number or Search Text field.

 **Search Forward —** Searches forward in the file listing for the text string specified in the Line Number or Search Text field.

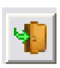
 **Search Backward —** Searches backward in the file listing for the text string specified in the Line Number or Search Text field.

Replacement Text — Use this text field to enter a "replacement text" string.

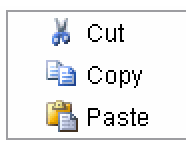
 **Replace One —** Replace one occurrence of text string in the Line Number or Search Text field (or the highlighted text in the file listing) with the text string in the Replacement Text field.

 **Replace All —** Replace all occurrences of the text string in the Line Number or Search Text field with the text string in the Replacement Text field.

 **Print —** Print the file.

 **Close Window —** Closes the window.

Shortcut: Right-click in the Text File Edit window to display the following menu:



These features provide the same functionality as those described under Edit in the menu bar.

VERICUT Log

VERICUT Users:

VERICUT Location: **Info menu > VERICUT Log**

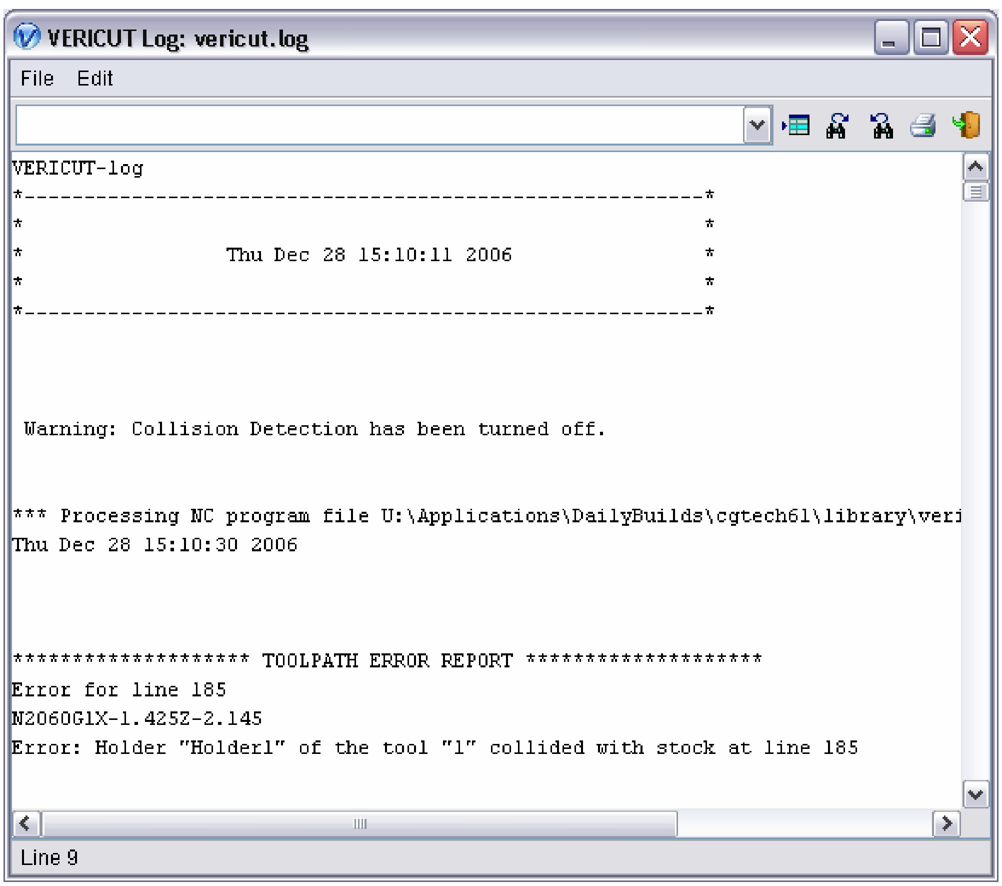
Toolbar shortcut: 

Cutter Grinder Users:

Cutter Grinder Location: **Analyze, View Files, Print page > View Log File**

Notebook Feature: 

Opens a window containing the VERICUT Log file. This file contains session information, such as error, warning and informational messages about the verification session. In this window, you can view, print, change then name, or reset Log file contents.



File menu

Open — Opens a file selection window enabling you to open an existing log file.

Save As — Opens a file selection window enabling you to save the VERICUT Log file.

Print — Opens a window enabling you specify page setup preferences, select a printer and print the VERICUT Log file.

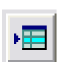
Reset Log File — Use to clear the contents of the VERICUT Log file.

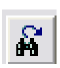
Exit — Closes the VERICUT Log window.

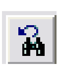
Edit menu

Copy — After highlighting the desired text in the VERICUT Log file, use this feature to copy the highlighted text to a paste buffer which can then be pasted to a text file, email, etc.

Line Number or Search Text — Use this text field to enter a line number, or a string of text, to search for.

 **Goto Line Number —** Moves the cursor in the file listing to the line number specified in the Line Number or Search Text field.

 **Search Forward —** Searches forward in the file listing for the text string specified in the **Line Number or Search Text** field.

 **Search Backward —** Searches backward in the file listing for the text string specified in the **Line Number or Search Text** field.

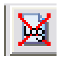
 **Print —** Print the VERICUT Log file.

 **Close Window —** Closes the **VERICUT Log** window.

[Click here to see a sample VERICUT Log file.]

Clear Log File and Logger

Location: **Info menu > Clear Log File and Logger**

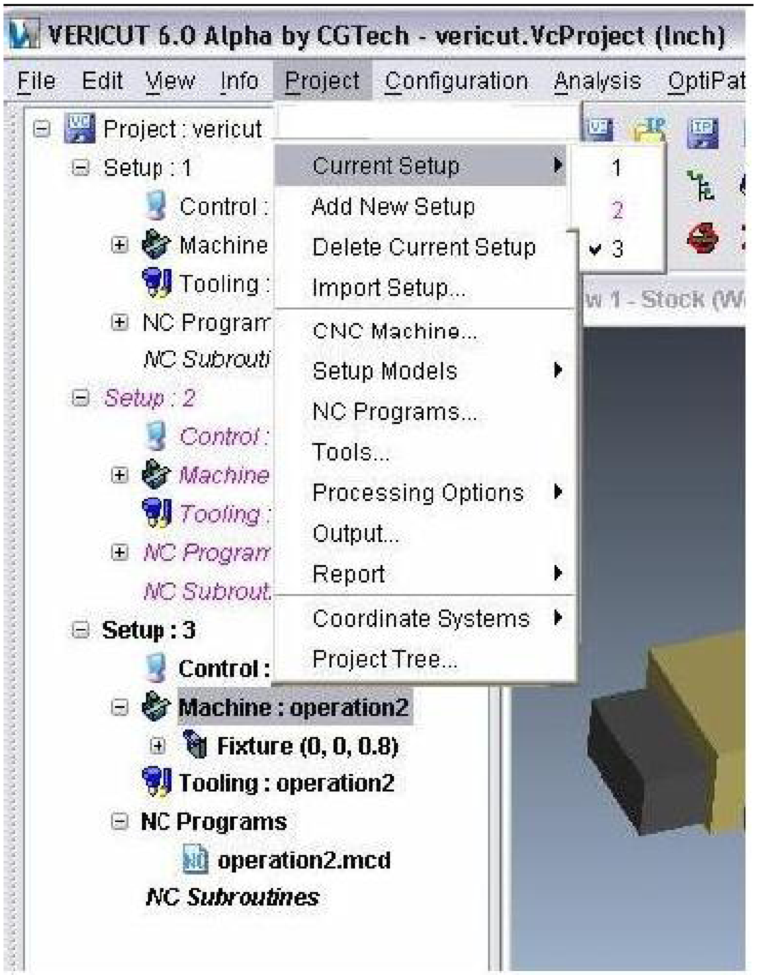
Toolbar shortcut: 

This feature enables you to clear both the Message area, also known as the "message logger", and the VERICUT Log file.

Project Menu

Introduction to the Project Setup menu

The features on the Project Setup menus provide the tools required for the day to day use of VERICUT for setting up jobs for verification and simulation. The following basic concepts are needed to understand the process of setting up a job for VERICUT.



Project — A project consists of one or more setups (see **Setup** below) enabling you to simulate multiple operations in a single VERICUT session. Multiple Setups within a project can use the same, or different, machines. A Project with a single Setup is logically equivalent to the User file of past VERICUT releases.

Setup — A Setup is a collection of VERICUT settings and files (machine, control, tool library, etc.), NC program files, and Setup models (Stock, Fixture, Design) that define the machine operation to be simulated. A Setup is logically equivalent to the User file of past VERICUT releases.

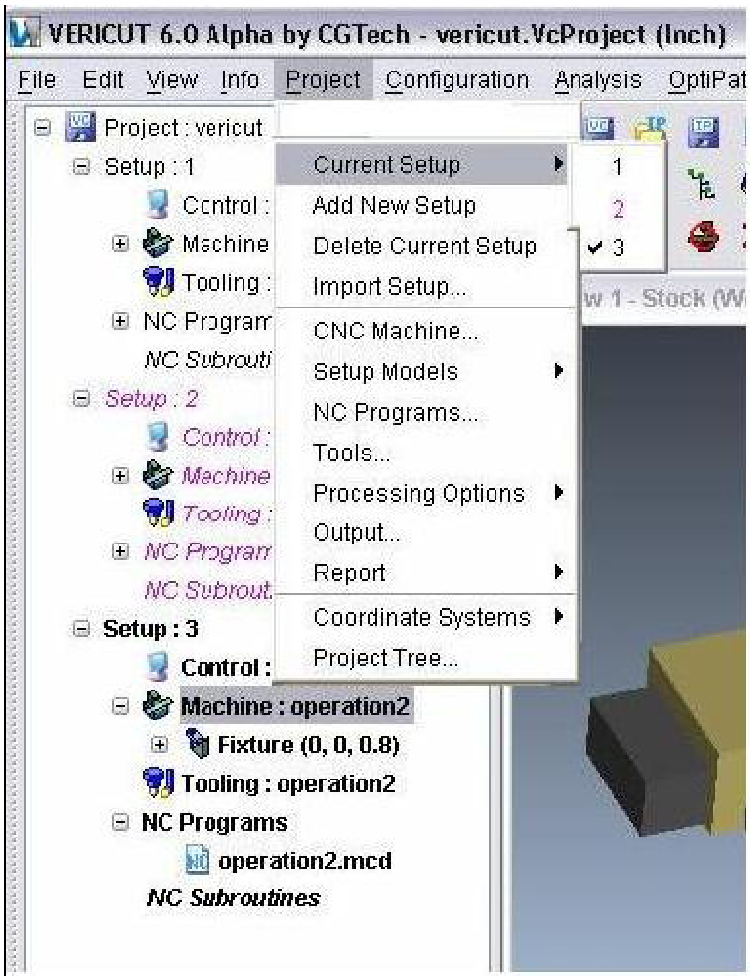
Current Setup — The Setup menu choices such as CNC Machine, NC Programs, etc., apply only to the Current Setup. A setup can be designated "current" either by selecting it using the Current Setup option in the Project menu, or by selecting it in the Project Tree. The "current" setup is indicated by a checkmark in the Current Setup menu and is displayed as bold in the Project Tree (See Setup 3 in the picture above).

Active / Inactive Setup — A Setup can be designated as being in either an Active, or Inactive state. When a Setup is designated as Inactive, it is not simulated, as if it were not present in the overall process. An Inactive setup can however be designated as the "current" setup and modified. Inactive Setups are indicated by red text in both the Current Setup menu and in the Project Tree (See Setup 2 in the picture above).

Current Setup

Location: **Project menu > Current Setup**

Selecting **Current Setup** displays a pull-right menu showing a list of available Setups. A check mark is displayed next to the current setup (See Setup 3 in the picture below). Picking a setup on the list makes it the current one. Inactive setups are displayed in red text (See Setup 2 in the picture below).



Add New Setup

Location: Project menu > Add New Setup

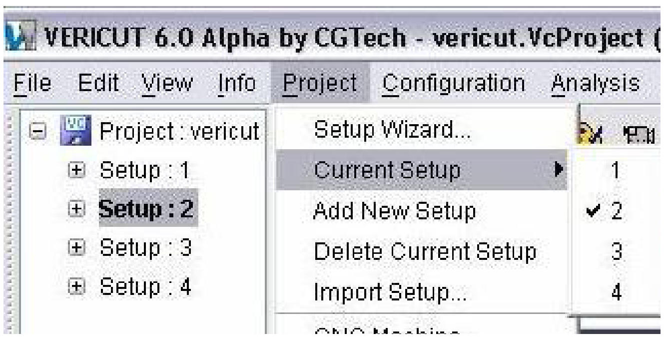
Add New Setup appends a new setup after the current setup. All settings from the current setup are copied to the new setup. Once the new setup is added, it becomes the current setup.

Delete Current Setup

Location: **Project menu > Delete Current Setup**

Delete Current Setup deletes the current setup and makes the next setup in the list the current setup (Example 1). If the setup at the end of the list is deleted, then the previous setup becomes the current setup (Example 2). There must always be at least one setup in a project. Therefore the last remaining setup cannot be deleted.

Example 1:

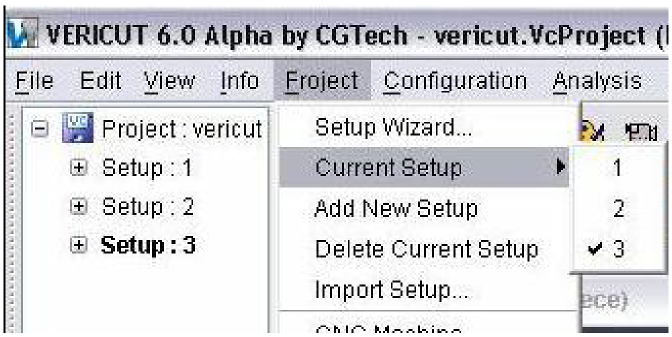


Before using **Delete Current Setup**, Setup: 2 is the "current" setup.

After using **Delete Current Setup**, Setup:2 is deleted and Setup:3 (the next in the list) becomes the "current setup.

Example 2:

Before using **Delete Current Setup**, Setup: 4 (the last in the list) is the "current" setup.

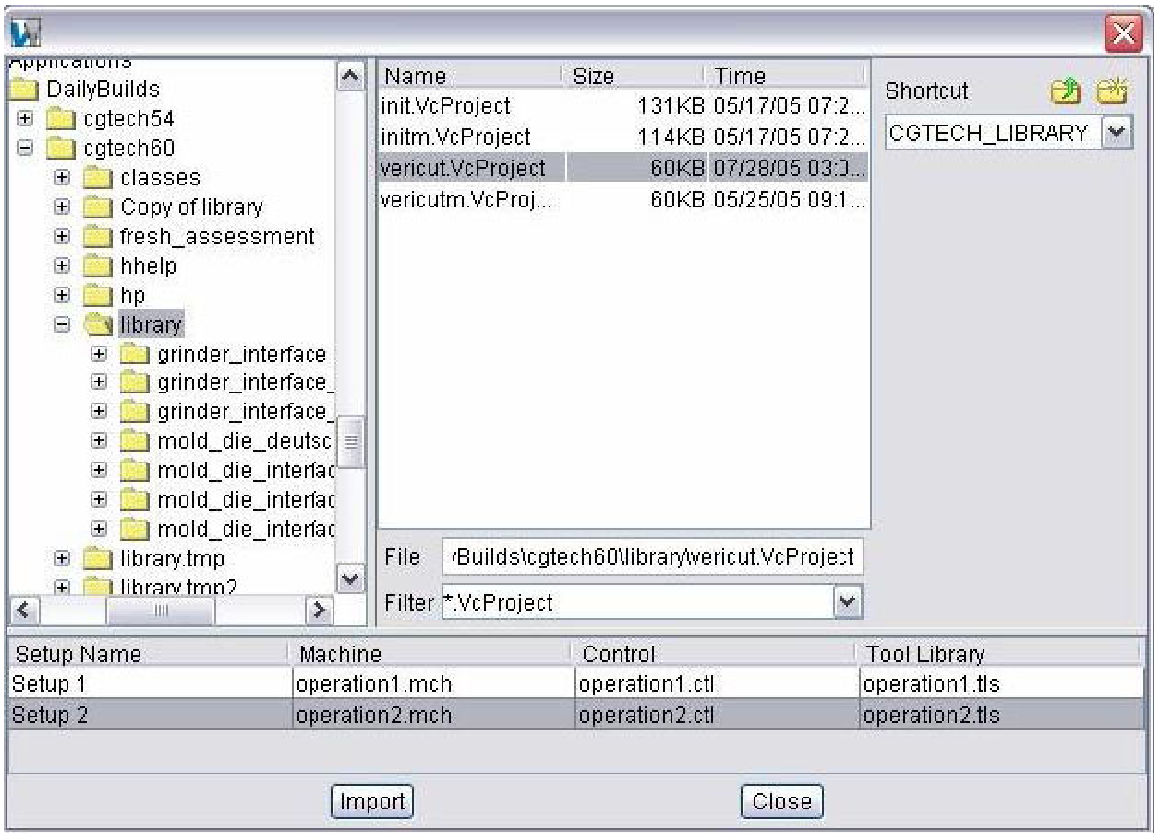


After using **Delete Current Setup**, Setup:4 is deleted and Setup:3 (the previous in the list) becomes the "current setup.

Import Setup

Location: **Project menu > Import Setup**

Copies a setup from either a pre-V6.0 user file or from a V6.0 project file and appends it after the current setup.



Project File selection window — Use the Project file selection window to specify the project file that the setup is to be imported from.

Setup List — After selecting the project file, a list of setups contained in the project file. Select one or more setups to be imported. Use the Shift and Ctrl keys on the keyboard to select multiple setups.

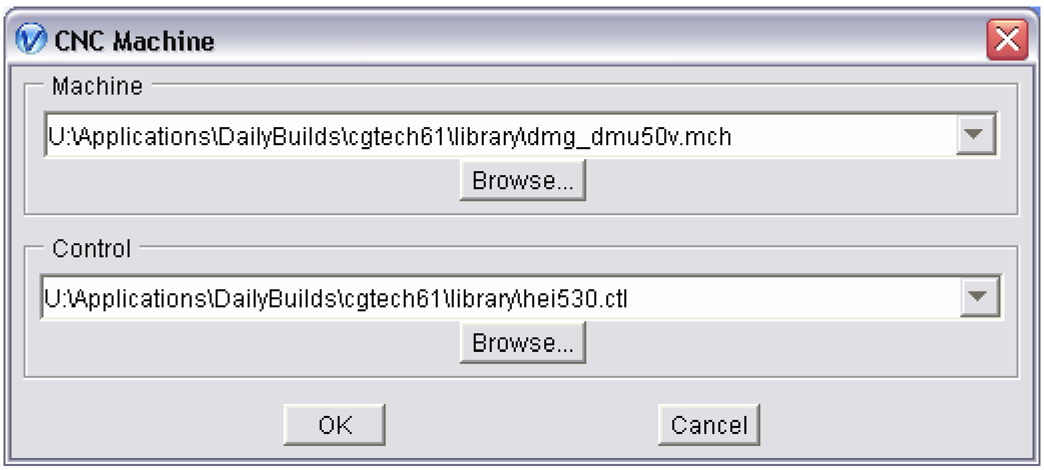
Import — Use to import the highlighted setups into the "current" project file.

Close — Closes the Import Setup window without importing any setups.

CNC Machine window

Location: **Project menu > CNC Machine**

CNC Machine opens a dialogue for selecting the Machine and Control file to use for the current setup.



Machine

Machine files contain data that describes NC machine construction, kinematics, and other properties.

Enter the \path\file name in the text field of the Machine file to be used or click on the Browse button and use the file selection window that displays to specify the machine file.

You can also click on the arrow button at the end of the Machine text field to display a list of all the Machine files contained in the "library" directory of your VERICUT installation and select the desired Machine file from the list.

Control

Control files contain data that describes how the NC control processes machine code data.

Enter the \path\file name in the text field of the Control file to be used or click on the Browse button and use the file selection window that displays to specify the control file.

You can also click on the arrow button at the end of the Control text field to display a list of all the Control files contained in the "library" directory of your VERICUT installation and select the desired Control file from the list.

Setup Models

Define (Modeling window)

VERICUT Users:

VERICUT Location: **Project menu > Setup Models > Define**

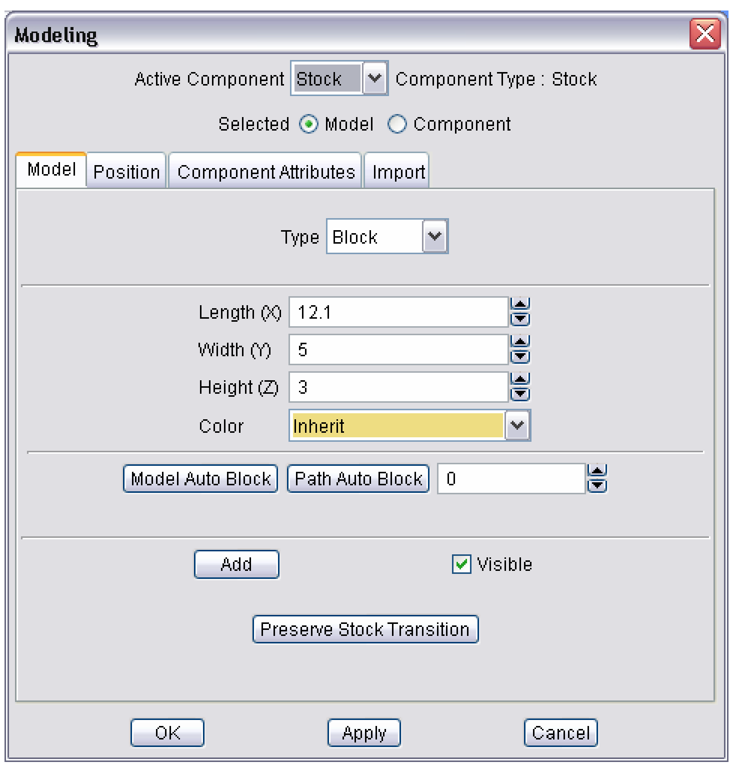
VERICUT toolbar short cut: 

Mold and Die Users:

Mold and Die Location: **Analysis and Viewing page > Specify a Design Model**

Notebook Feature: 

Opens the Modeling window to define or import the models that represent the solid objects used in the simulation. VERICUT uses "components" to describe the function of various solid objects. Models provide 3-D size and shape for material removal and collision checking. To learn more, see "About modeling in VERICUT" in the *Using VERICUT* section. *Using VERICUT* can be found in the *CGTech Help Library*.



Active Component — Component active for editing. The active component is automatically set when you select items from the graphics area or Component Tree window. However, you can select a different component via picking it from the option list.

Component Type — The active component's type is shown in an un-editable field for reference purposes. To change the component type, use the Component Tree window Edit menu.

Selected — Identifies what is selected for editing: a Model or the Component. This option is automatically set when you select items from the graphics area or Component Tree window. However, you can change what is selected for editing via clicking the other option.

Model tab **—** Features on this tab add and modify models attached to components to provide 3-D shape. The models can be simple parametric shapes, or refer to external model files.

Position tab **—** Features on this tab move a component or model.

Component Attributes tab **—** Features on this tab define the attributes of a component.

Import tab **—** Features on this tab import IGES model files for use as stock, fixture, design, or machine component models.

OK — Applies the changes and closes the Modeling window.

Apply — Applies the changes and leaves the Modeling window open.

Cancel — Closes the Modeling window without applying changes.

Tips:

1. Right-click shortcut menu: Features used to manipulate components are displayed in a shortcut menu when a component is selected and the right mouse button is clicked in the graphics area. Similarly, features used to manipulate models are displayed when a model is selected and the right mouse button is clicked.
2. Delete components or models by selection the object in the graphics area, then use the <Delete> key on your keyboard
3. Click repeatedly on a model in the graphics area to toggle selection between the model and its component. Refer to the Component Tree window to review what is selected.
4. Using the center mouse button of a three button mouse, click repeatedly on a model in the graphics area to toggle selection between different models along the

line of sight from the pick point. Use this technique to select models hidden from view by an object in the foreground.

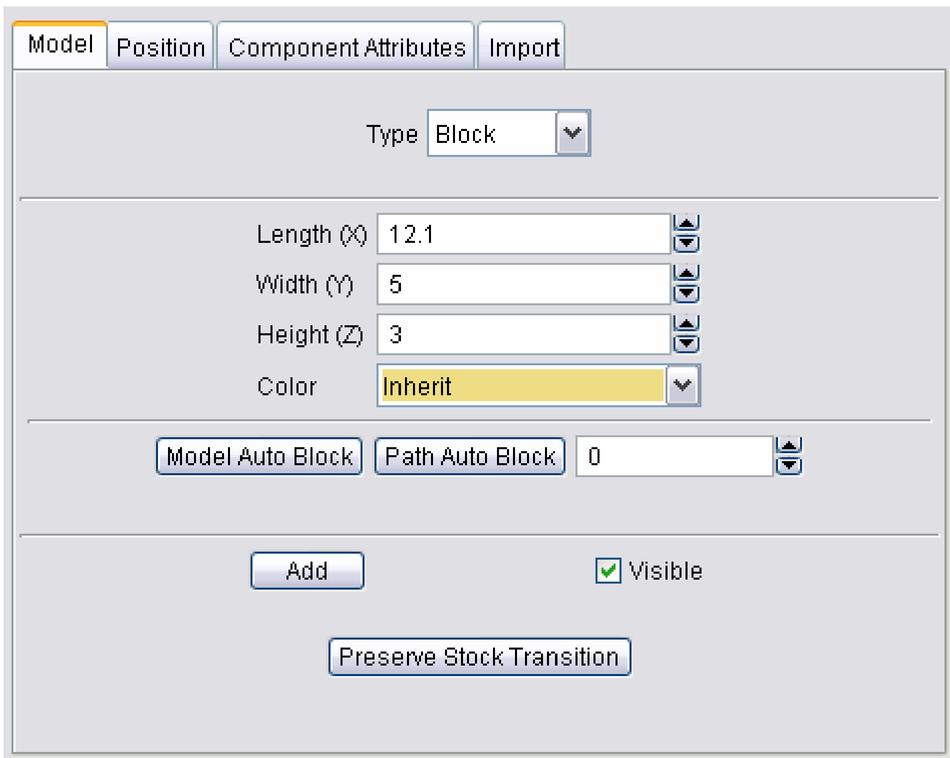
Also see "**Modeling in VERICUT**", in the *Using VERICUT* section, in the *CGTech Help Library*.

Modeling window, Model tab

Location: **Project menu > Setup Models > Define**

Toolbar short cut: 

Features on this tab are used to add and modify models attached to components to provide 3-D shape. The models can be simple parametric shapes, or refer to external model files.



Type — Type of model being defined. The active choice configures the model description area below this option with the values needed to define the selected model. The table below lists the available model types and corresponding descriptions.

Model type:	Appears in the Component Tree:	Description
Block	Block	Length (X) Width (Y) Height (Z) Color
Cone	Cone	Height (Z) Base Radius Top Radius Color
Cylinder	Cylinder	Height (Z) Radius Color
Model File	STL VERICUT Polygon VERICUT Solid Sweep SOR	Model File, Sketcher, Normals*, Units, Color

Block specific features:

Length (X), Width (Y), Height (Z) — Dimensions of the block.

Model Auto Block — This feature enables you to select a model file representing the design model and have VERICUT create a stock block enclosing the selected model. Selecting **Model Auto Block** displays the Model Auto Block file selection window enabling you to select the model file. Use the **Units** pull-down to specify the units (Inch or Millimeter) to be used.

Path Auto Block — When defining a **Block** type model, this feature automatically defines and locates a stock block based on tool positions in the current tool path file(s). Rapid and APT-CLS "FROM" tool positions are ignored during this calculation. The block size and location guarantee that the tool path will cut the block.

Use the text field to specify the offset to be applied to the block created using **Model Auto Block** or **Path Auto Block** (see above). The offset is applied to all dimensions.

Cone specific features:

Height (Z), Base Radius, Top Radius — Dimensions of the cone (height, radius at the base of the cone and the radius at the top of the cone).

Cylinder specific features:

Height (Z), Radius — Dimensions of the cylinder.

Model File specific features:

Model File — Enter the /*path*/*filename* of the model file or use the **Browse** button to display a file selection window and use it to select the model file.

Sketcher — Displays the **Profile Sketcher window** enabling you to edit or create a "Sweep solid (.swp)" or "Revolve (.sor)" model file. This feature is only available when the selected file is a Revolve or Sweep file type. To create a new Revolve or Sweep model file, clear the Model File text field and press **Sketcher**, select the profile type (**Revolve** or **Sweep**) from the window that displays, then **OK** to display the Profile Sketcher window. Cancel dismisses the Profile Type window.

Normals — This feature applies to STL file models only and indicates the direction in which surface normals defined in the STL data point, relative to the model. Select **Inward**, **Outward**, or **Computed** (let VERICUT determine the direction of the surface normals).

Units —Units (**Inch** or **Millimeter**) of the model file.

Common Features

Color — Color of the model. Available colors are the **Shade Colors** defined on the **Edit menu > Colors: Define tab**. Models connected to the component have the option to "Inherit" or override the component's defined color.

Add — Adds the model defined by the current window values to the active component.

Visible — Toggles "on" or "off" to indicate whether or not the model file should be displayed in the graphics area.

Preserve Stock Transition — Use to create the coordinate systems required for transitioning the cut stock(s) from one setup position to another. **Preserve Stock Transition** can be used to create the coordinate systems for each **Stock** component. **Preserve Stock Transition** is only active when a stock component model is selected.

When **Preserve Stock Transition** is pressed, two coordinate systems named "*Previous Setup name* **:** *Current Setup name* **:** *Active Stock component name*" are created and added. *Previous Setup name*, *Current Setup name* and *Active Stock component name* are replaced by the actual names of these entities used in the project file, for example "Rough:Finish:Stock1". One CSYS is added to the current setup, attached to the Stock

Component parent of the selected model. It is created with **Use for Cut Stock Transition** toggled "On" (see the **Define (Coordinate System window)** section) , designating it as a coordinate system used for transitioning the cut stock from one setup position to another. The other CSYS is added to the first setup in the project, and is located at the origin of the stock component whose name matches current setup stock component. The following illustrates how to use **Preserve Stock Transition**:

1. Configure Setup #1.
2. Simulate.
3. Configure Setup #2.
4. Move cut stock into position using cut features and assembly modeling to position the cut stock
5. Press **Preserve Stock Transition**.
6. Simulate.
7. Configure Setup #3.
8. Move cut stock into position.
9. Press **Preserve Stock Transition**.
10. Simulate.
11. and so on ...

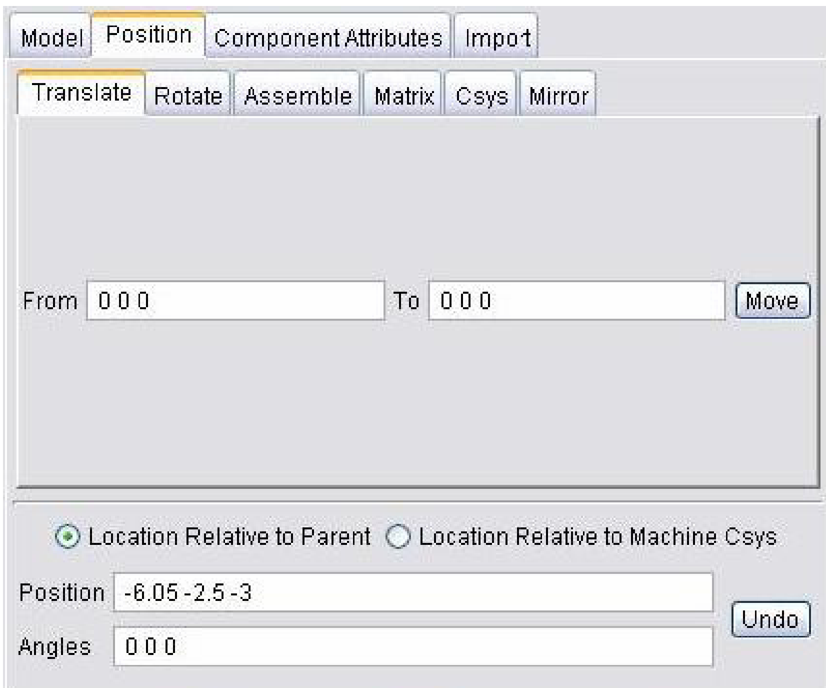
To Modeling window

Modeling window, Position tab

Location: **Project menu > Setup Models > Define**

Toolbar short cut: 

Features on this tab move an object: component or model.



Translate tab — Features on this tab translate the selected object via indicating "from" and "to" points to move the object.

Rotate tab — Features on this tab rotate the selected object about a rotation center point.

Assemble tab — Features on this tab move the object by assembling (mating or aligning) it with other objects.

Matrix tab — Features on this tab move the selected object via a twelve parameter transformation matrix.

CSYS tab — Features on this tab enable you to move a selected component/model from one coordinate system to another.

Mirror tab — Features on this tab enable you to mirror a model about a specified axis (plane).

Location features

The Location features are common to all of the above tabs and show the selected object's position and angle, and can be used to move the object or verify its current location. Values are shown relative to the parent component or machine origin, whichever is selected: Relative Location or Machine Location, respectively.

Location Relative to Parent — When selected, it indicates that the object location is shown relative to the parent component origin. Examples follow.

> *Positioning a component* **—** component is moved relative to the parent component origin.
>
> *Positioning a model* **—** model is moved relative to the component origin.

Location Relative to Machine Csys — When selected, it indicates that the object location is shown relative to the machine origin.

Position — Specifies the absolute XYZ position of the object, separated by spaces.

Angles — Specifies the absolute XYZ rotation of the object, separated by spaces.

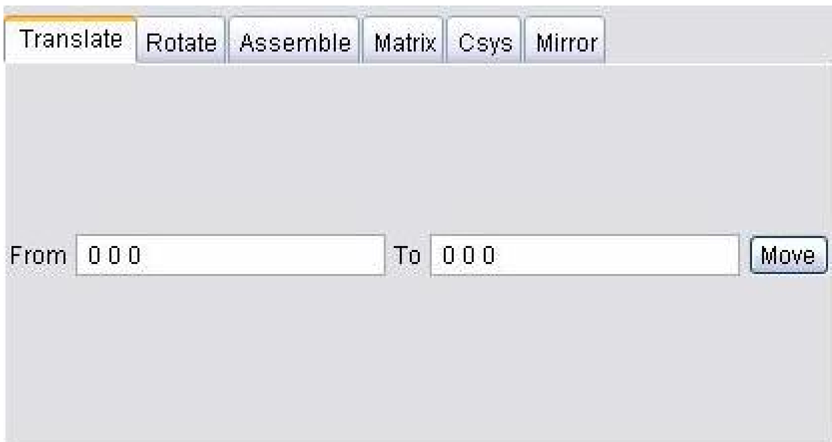
Undo — Returns the object to its previous location, or as it was when the Modeling window was opened.

To Modeling window

Modeling window, Position tab, Translate tab

Location: **Project menu > Setup Models > Define: Position tab**

Features on this tab translate the selected object via indicating "from" and "to" points to move the object. Movement occurs each time you press the **Move** button. If the applied movement is incorrect, press **Undo** (on the Position tab) to return the object to its previous location.



Translate tab features

From / To — Specifies the locations to move the object from and to, relative to the machine origin. XYZ values can be entered (separated by spaces), or selected by clicking in the field then clicking on a model. As you move the mouse over the VERICUT model, a crosshair and vector show you the pending pick-point location. Graphical selection supports picking corner points and midpoints of uncut model geometry, or virtually any point on machined features.

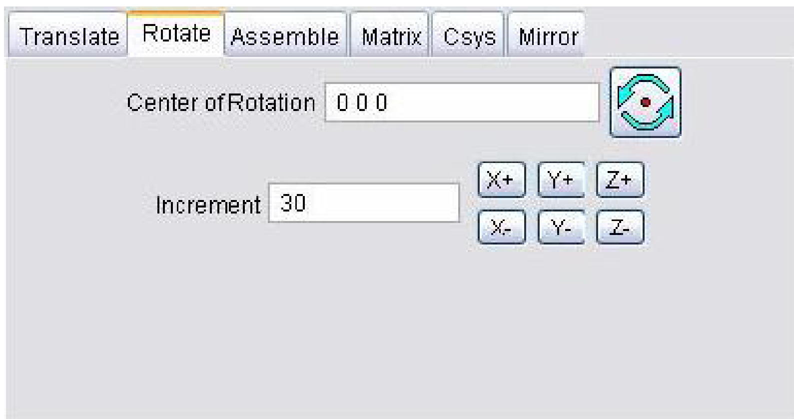
Move — Moves the selected object by the incremental distance, as calculated from the "**From**" point to the "**To**" point location.

To Position tab

Modeling window, Position tab, Rotate tab

Location: **Project menu > Setup Models > Define: Position tab**

Features on this tab rotate the selected object about a rotation center point. Movement occurs each time you press one of the rotation direction buttons: X+/X-, Y+/Y-, Z+/Z-. If the applied movement is incorrect, press **Undo** to return the object to its previous location.



Rotate tab features

Center of Rotation — Specifies XYZ point location about which to rotate the object. XYZ values can be entered (separated by spaces), or selected by clicking in the field then clicking a position on a model. To see the center of rotation, press . To remove the center of rotation symbol press the button again, or close the window.

Increment — Specifies incremental degrees of rotation to apply when one of the rotation direction buttons are pressed.

Rotation direction buttons — (**X+/X-**, **Y+/Y-**, **Z+/Z-**) When pressed, applies the incremental rotation specified in the **Increment** field. Rotation occurs about the **Center of Rotation**, relative to the machine origin.

To Position tab

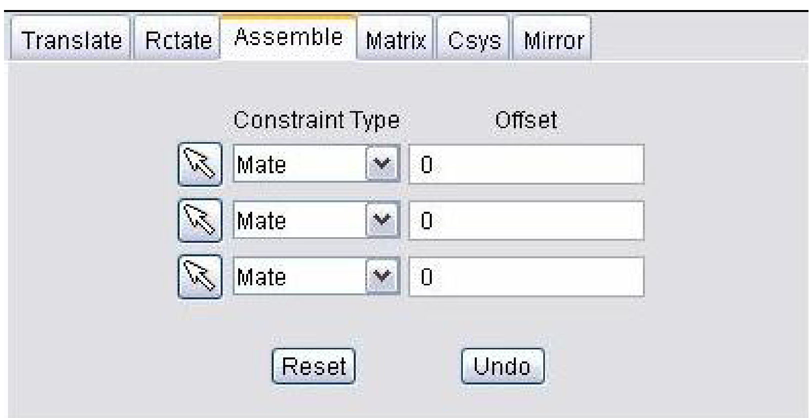
Modeling window, Position tab, Assemble tab

Location: **Project menu > Setup Models > Define: Position tab**

Features on this tab move the selected object by assembling (mating or aligning) it with other objects. Objects are assembled by mating or aligning one to three planar surfaces with surfaces on other models. If a non-planar surface is selected, VERICUT constructs a tangent plane at the pick point. The relationship of surfaces being mated or aligned is known as a "constraint".

Follow these general steps to define a constraint for assembly:

1. Choose the constraint type.
2. Select a surface on the object to move.
3. Select the surface to move the object relative to.



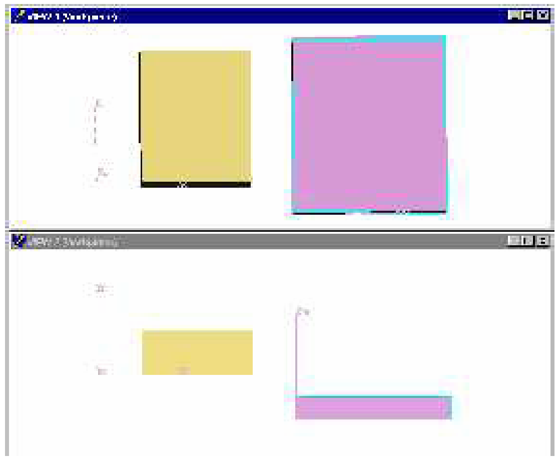
Assemble tab features

Constraint Type — Specifies how to constrain selected surfaces during object movement. After selecting two surfaces to define a constraint, VERICUT moves the object and highlights the satisfied constraint with a checkmark. Options:

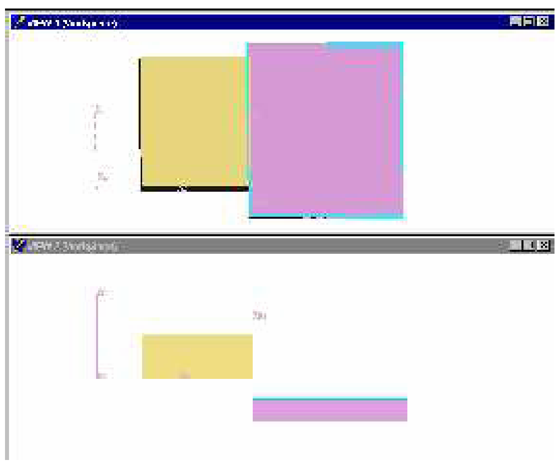
Mate — Moves the object so the selected surface opposes the surface selected on the second object (surface normals oppose each other).

Align — Moves the object so the selected surface is aligned with the surface selected on the second object (surface normals point in the same direction).

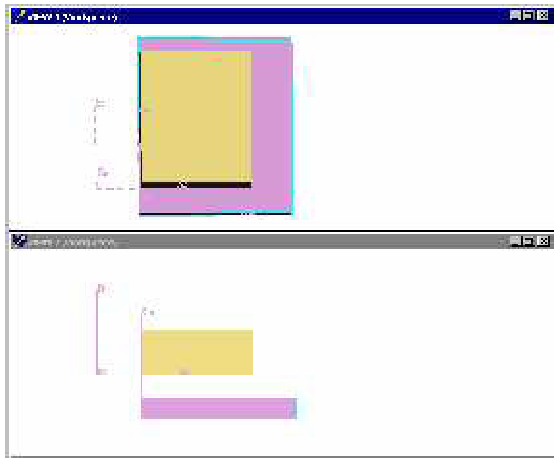
Consider the starting condition below for the examples that follow.



Mated surfaces — the left side of the magenta model is mated with the right side of the yellow model.



Aligned surfaces — the left side of the magenta model is aligned with the left side of the yellow model.



Offset — Distance and direction in which to offset constrained surfaces, applied normal to the surface.

Reset — Resets constraints to receive new data.

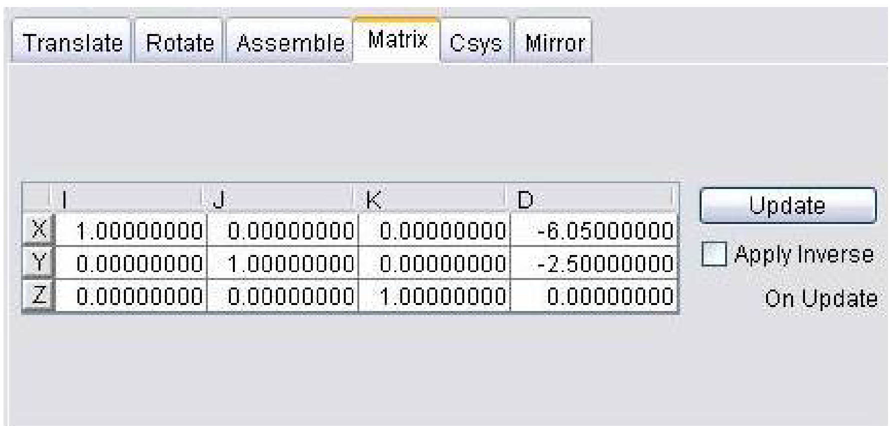
Undo — Returns the object to its previous location, or as it was when the window was opened.

To Position tab

Modeling window, Position tab, Matrix tab

Location: **Project menu > Setup Models > Define: Position tab**

Features on this tab move the selected object via a twelve parameter transformation matrix. If the applied movement is incorrect, press **Undo** (on the Position tab) to return the object to its previous location.



Matrix tab features

Matrix table — The transformation matrix table is similar to the matrix used in programming APT tool paths. Its twelve parameters reveal the geometric attributes of the local (transformed) coordinate system (CSYS) in terms of the machine origin.

The format of the matrix table is as follows:

	I	**J**	**K**	**D**
X	I1	J1	K1	D1
Y	I2	J2	K2	D2
Z	I3	J3	K3	D3

Each row represents an axis of the local CSYS. The first three columns represent the vector associated with each axis: I1, J1, K1 as the positive X-axis vector; I2, J2,K2 as the positive Y-axis vector; and I3, J3, K3 as the positive Z-axis vector. The fourth column values D1, D2, D3 represent the coordinates of the origin point of the local CSYS.

NOTE: If you prefer to see the Matrix Table displayed with the I, J, K along the vertical axis and the X, Y, Z along the horizontal axis, set the environment variable, **CGTECH_MATRIX_FORMAT=VERTICAL**.

Update — Updates the object location to reflect the matrix table transformation. After updating, press OK or Apply to move the object.

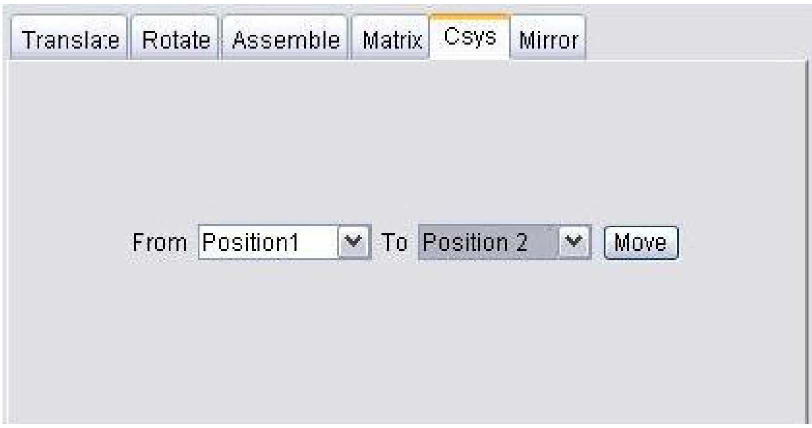
Apply Inverse On Update — When selected, inverts the matrix so that its twelve parameters reveal the geometrical attributes of the machine origin in terms of the local (transformed) coordinate system.

To Position tab

Modeling window, Position tab, CSYS tab

Location: **Project menu > Setup Models > Define: Position tab**

Features on this tab enable you to translate the selected object from one coordinate system (CSYS) to another. Select the "From" CSYS and the "To" CSYS to move the object. Movement occurs each time you press the **Move** button. If the applied movement is incorrect, press **Undo** (on the Position tab) to return the object to its previous location.



CSYS tab features

From / To — Specifies the coordinate system to move the object from, and to. Select the appropriate CSYS from each of the pull-down lists.

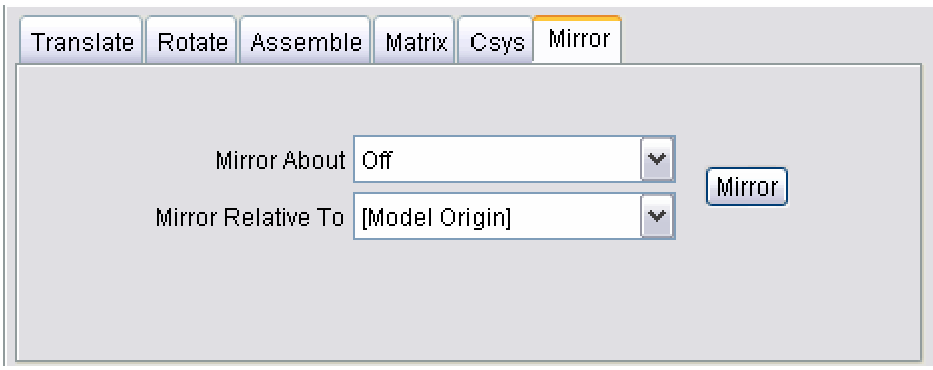
Move — Moves the selected object from the "**From**" CSYS to the "**To**" CSYS orientation.

To Position tab

Modeling window, Position tab, Mirror tab

Location: **Project menu > Setup Models > Define: Position tab**

Features on this tab enable you to mirror a model about a specified plane, of a specified coordinate system. If the applied movement is incorrect, press **Undo** (on the Position tab) to return the object to its previous location.

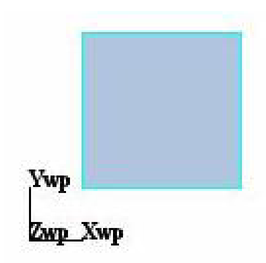
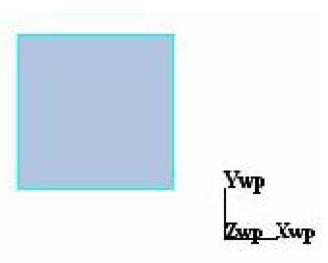


Mirror tab features

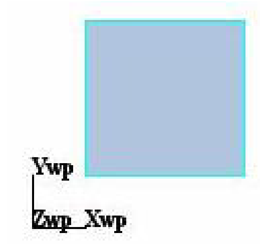
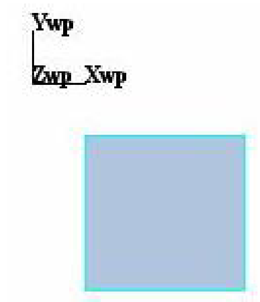
Mirror About — Use the features in the pull-down list to specify the plane that the model is to be mirrored about.

Off — The Mirror attribute is turned off.

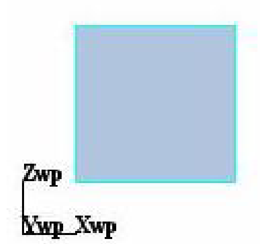
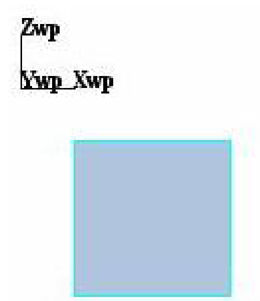
YZ Plane (X direction) — The Mirror attribute is set to mirror about the YZ plane.

Off	**YZ Plane (X direction)**
	

ZX Plane (Y direction) — The Mirror attribute is set to mirror about the ZX plane.

Off	
ZX Plane (Y direction)	

XY plane (Z direction) — The Mirror attribute is set to mirror about the XY plane.

Off	
XY Plane (Z direction)	

Mirror Relative To — Use to specify the coordinate system that the plane that you want to mirror about is related to. For example, the XY plane of the model origin. The pull-down list will contain the model origin (the default) and all available coordinate systems.

Mirror — After specifying the plane, and the coordinate system that it is relative to, press **Mirror** to make the mirror take place.

NOTES:

1. The following model types can be mirrored:

   VERICUT polygon files (polygon, sweep solid or solid of revolution (SOR)

   STL files

   VRML files
2. The model is mirrored in its local coordinate system (not the active coordinate system)
3. The mirror attribute is part of the model structure and is saved with the model data when saved to the project file or machine file.
4. When VERICUT reads a Project file or machine file, the models that have the mirror attribute set will be mirrored when loaded.

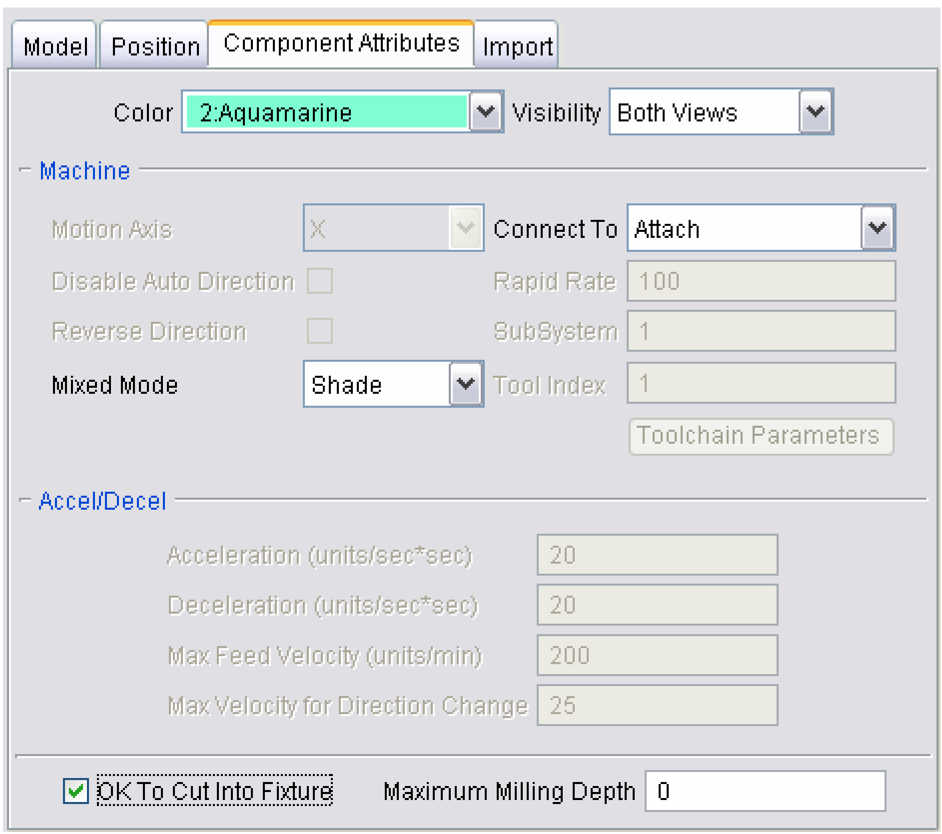
To Position tab

Modeling window, Component Attributes tab

Location: **Project menu > Setup Models > Define**

Toolbar short cut: 

Features on this tab define the attributes of a component. VERICUT handles components and their solid models differently depending on the component's type and attributes. Most attributes apply to components used in building NC machines.



Color — Color of the component. Available colors are determined by the colors defined on the **Edit menu > Colors: Define tab**.

Visibility — Controls the views in which the component is visible. Options:

Blanked — Component is blanked from all views and cannot be seen.

Workpiece View — Component is seen in the workpiece view.

Machine View — Component is seen in the machine view.

Both — Component is seen in both the workpiece and machine views.

See also: VERICUT command records that control displaying/loading/removing model

Machine group:

VERICUT uses a "moving tool philosophy" to describe motion directions, and reflect how the machine maintains the tool-to-workpiece relationship. This philosophy is used regardless of whether the Tool or Stock component is actually moving. Examples follow.

Consider a standard 3-axis vertical milling machine where the Z-axis moves the tool up/down, and the X or Y-axis moves the workpiece.

Example 1:

Z+ command causes the Z-axis to carry the tool in the Z+ direction relative to the workpiece- therefore using the moving tool philosophy describes positive motion direction as "Z+".

Example 2:

X+ command causes the X-axis to carry the workpiece in the X- direction relative to the tool. In this case, the tool is considered to move in the X+ direction relative to the workpiece- therefore using the moving tool philosophy describes positive motion direction as "X+", even though the workpiece is moving in the X- direction.

Motion Axis — Specifies the axis of motion for a linear, rotary or turret type component, relative to the component coordinate system. Use a "moving tool philosophy" to specify the motion axis and direction, even though the component may actually move on the machine instead of the tool.

Linear motion component: the motion axis is the axis in which linear motion occurs.

Rotary and turret components: the motion axis is the axis about which rotation occurs. The positive rotation direction is counterclockwise when viewed from the positive end of the motion axis towards the component origin.

Tool and spindle components: the motion axis is the aligned with the tool axis, typically "Z".

Disable Auto Direction — When toggled "On" (checked), disables VERICUT's sense of the "moving tool philosophy" for that particular component.

When **Disable Auto Direction** is toggled "Off" (not checked), VERICUT checks to determine whether or not the component is "part side". A component is considered "part side", when one of its child components is a Stock component. If the component is "part side", VERICUT automatically reverses the direction of motion.

This feature is intended for use in situations where a component starts connected to one machine component , then later gets disconnected and reconnected to a different machine component (for example, moving a "stock" component from one "fixture" component to a different "fixture" component). If moving the component changes the "part side/not part side" relationship of the components, **Disable Auto Direction** should be toggled "On" (checked) causing the component motion direction to remain constant regardless of where the component is connected.

If the relationship between two components is sometimes "part side", and sometimes "not part side", then the **Disable Auto Direction** should be used, and the component motion direction set explicitly with **Reverse Direction** feature (described below).

In the example below, the machine is configured like:

Base

Y

X

A

Fixture1

U

A2

Fixture2

Sometimes the Stock is connected to Fixture1, and sometimes it is connected to Fixture 2. Components A, U, and A2 should have **Disable Auto Direction** toggled "On", and **Reverse Direction** should be used to control the component's motion direction.

Reverse Direction — When toggled "On" (checked), reverses the component motion direction.

Mixed Mode — Controls how the component is displayed when displayed in Mixed draw mode.

Connect To — Component to which the active component is connected. The component being connected to is known as the "connect component". Components must be connected in the same way as they are in the real world. Improper connections cause incorrect NC machine movement and simulation, especially when rotary components are involved.

Rapid Rate — Feed rate for motion-type components moving in rapid positioning mode (e.g. G0). For linear motion components the value entered is units/minute where "units" reflect the units of the tool path file. For rotary motion components the value entered is degrees/minute.

SubSystem — Subsystem which commands a linear or rotary motion type component to move. Any alpha-numeric sub-system name can be entered (may include spaces) and is always considered as text. Enter "*" (without the quotes) for motion axes included in all machine subsystems. Subsystems are used in machines with multiple motion components that are commanded to move via the same G-Code word, for example "X". By default, a machine uses one subsystem for all motion components, assuming they are commanded to move via unique G-Code words.

When multiple motion components are driven by the same G-Code word, such as can occur in 4-axis lathes or mills with multiple independently controlled heads, define different subsystems to control which component moves when the motion command is processed. Then, configure the control so that other codes in the tool path activate the appropriate subsystem for movement. A full set of tables are available to support each machine subsystem. See the **Sample-Demo Files** section, in the *CGTech Help Library* for examples of machines that use multiple subsystems.

Tool Index — Index number of a Tool component. When defining machines with multiple tool load positions, such as a multi-spindle mill or a turret lathe machine with tools loaded in the turret, define additional Tool components with different index numbers. Values entered are integer numbers. Use the "ActiveTool" macros to activate each Tool component for tool changes. For example, call "ActiveTool2" to activate the Tool component with Tool Index=2, and so on.

Toolchain Parameters — Displays the Toolchain window enabling you to specify toolchain component characteristics. Toolchain Parameters is only active when the Active Component is Tool Chain.

Turret Aid — Displays the Turret Aid window enabling you to specify turret component characteristics. **Turret Aid** is only active when the Active Component is A Turret, B Turret, or C Turret.

Accel/Decel group:

Acceleration / Deceleration — Velocity at which the NC machine component can accelerate (or decelerate). Values entered are units per second per second, where units could be inches or millimeters for linear axes, depending on the machine units, or degrees for rotary axes.

Max Feed Velocity — Maximum speed axes can travel while in feed mode. The value entered is in units per minute, where units could be inches or millimeters for linear axes, depending on the machine units, or degrees for rotary axes.

Max Velocity for Direction Change — Maximum feed rate for motions that cause an axis to decelerate to zero velocity, such as occurs when turning at a corner.

NOTE: These Accel /Decel settings are only used when calculating the **Time** displayed in the **Status window**. They are unrelated to the Accel / Decel settings in the OptiPath window.

Machinable Fixture group:

OK To Cut Into Fixture — Use this feature to specify when it is acceptable to cut into a fixture. This feature is only active when Component Type is Fixture. When toggled "On" (checked), cutter/fixture collisions are reported as described below:

Milling and Tapping Cuts

If **Maximum Milling Depth** (described below) is set to 0, cutter/fixture collisions are not reported regardless of collision depth.

If **Maximum Milling Depth** (described below) is set to any value other than 0, cutter/fixture collisions are only reported if the collision depth is greater than the specified value.

Turning Cuts

For turning cuts, cutter/fixture collisions are not reported regardless of collision depth.

When **OK To Cut Into Fixture** is toggled "Off" (not checked), any cutter/fixture contact produces an error.

Maximum Milling Depth — Use this feature to specify the maximum depth that a mill or tap cutter can cut into the fixture. This feature is only available when **OK To Cut Into Fixture** is toggled "On".

The **Maximum Milling Depth** feature is only applicable to milling, or tapping cuts. For turning cuts, the **Maximum Milling Depth** setting is ignored.

NOTES:

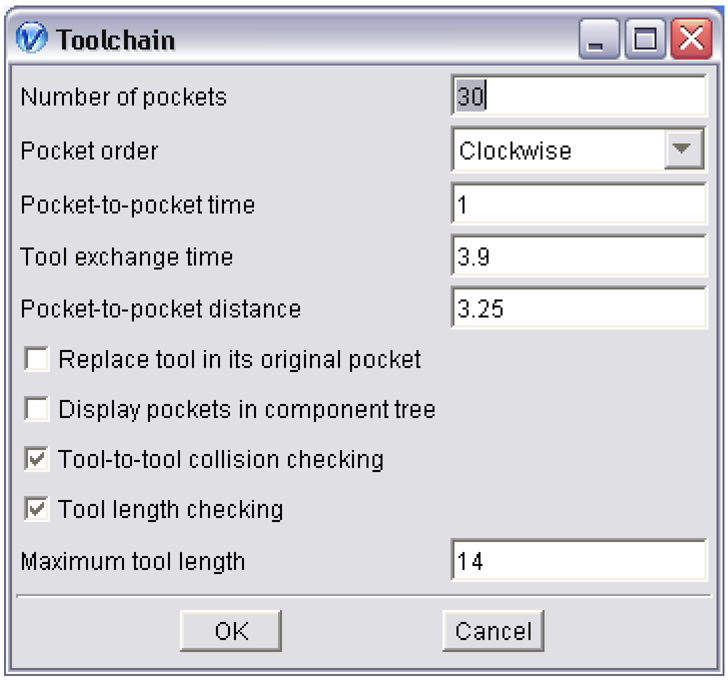
1. The **OK To Cut Into Fixture** feature is only relevant for valid milling, tapping, or turning cuts. For instance, if a milling cutter collides with the fixture while in an invalid spindle state, this option will be ignored and the collision will be reported as usual.
2. The behavior described above applies only for the cutter. Holder collisions and other machine component collisions are not affected by the check box state.
3. Both settings are saved in the fixture component.

To Modeling window

Toolchain window

Location: **Project menu > Setup Models > Define: Component Attributes tab**

The features on this window are used to define toolchain characteristics.



Number of Pockets — Use to define the number of tools in the toolchain. The VERICUT will automatically create this number of equally spaced tool components along the toolchain's perimeter geometry. The toolchain perimeter is defined by the first sweep model file attached to the toolchain component. This file is normally not displayed except for debugging purposes. Other models may be used to define the associated toolchain machine structure.

These tool components are named Pocket1, Pocket2, ..., PocketN. They are created each time the machine definition file is loaded, are not saved, and may not be edited. The position of Pocket1 is defined by the start point of the first boundary geometry segment (line or arc). Pocket1 is also defined as the "select" or "exchange" position. The perimeter geometry is scaled about Pocket1 to achieve the pocket-to-pocket spacing entered in the GUI. Normally this adjustment should be small.

Pocket order — Use to define the direction for pocket numbering. Select **Clockwise** or **Counter Clockwise** from the pull-down list.

Pocket-to-pocket time — Defines the time (in seconds) required to move the toolchain one position.

Tool exchange time — Use to define the time (in seconds) required to move the selected tool into the machine's spindle and return the unloaded tool to the toolchain.

Pocket-to-pocket distance — Defines the distance between adjacent tool centers along the toolchain's perimeter.

Replace tool in its original pocket — Use to specify where the tool being unloaded from the spindle is to be placed in the toolchain. When toggled "On", the unloaded tool is returned to its original pocket position. Otherwise, the tool being unloaded is placed in the pocket emptied by the tool being loaded.

Display pockets in component tree — Use to specify whether or not the toolchain's associated tool components are displayed in the component tree. When toggled "On", tool components will be displayed in the component tree. The default is not to display the tools components since large toolchains can contain a hundred or more tools.

Tool-to-tool collision checking — Use to specify collision checking when a tool is placed in the toolchain during a tool exchange. When toggled "On", the tool being unloaded and returned to the toolchain is checked for collisions with the tools in adjacent pocket positions.

Tool length checking — When toggled "On", tool lengths are checked against the Maximum tool length value when they are initially put in the toolchain. If the tool length exceeds the Maximum tool length value, an error is output.

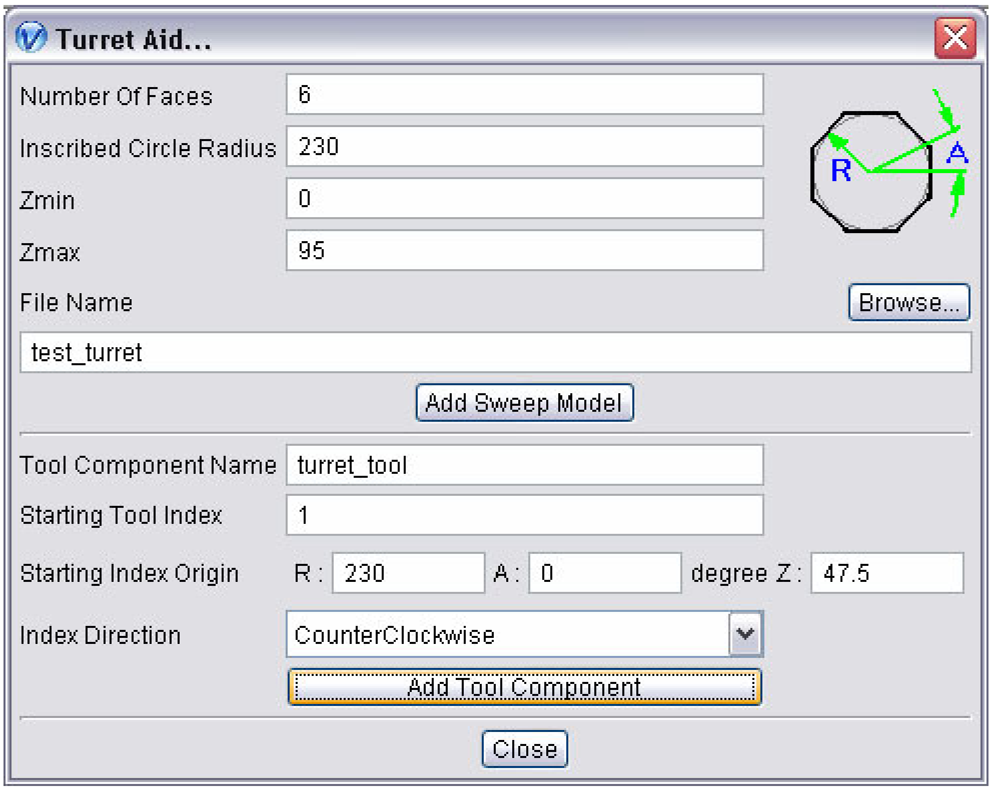
Maximum tool length — Use to specify the maximum tool length value used for Tool length checking.

See **Toolchain Operation**, in the *Using VERICUT* section , in the *CGTech Help Library* for more information.

Turret Aid window

Location: **Project menu > Setup Models > Define: Component Attributes tab**

The features on this window are used to define turret characteristics.



The features in the upper portion of the Turret Aid window enable you to easily create a swept solid model of a turret.

Number of Faces — Use to specify the number of faces on the perimeter of the turret.

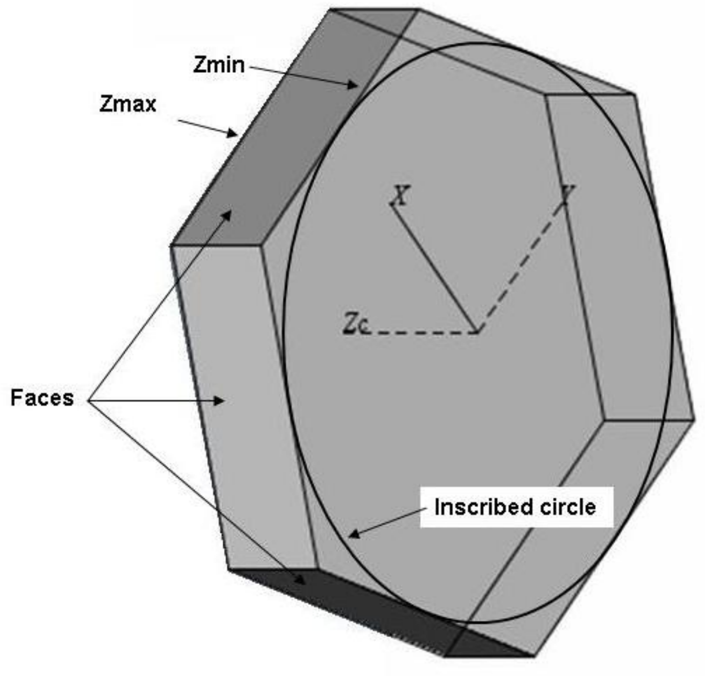
Inscribed Circle Radius — Use to specify the radius of the inscribed circle that will be used to determine the size of the turret.

Zmin — Used to specify the starting location, along the Z-axis, to be used when creating a "swept" solid model of the turret. The distance between **Zmin** and **Zmax** determine the thickness of the turret.

Zmax — Used to specify ending location along the Z-axis to be used when creating a "swept" solid model of the turret. The distance between **Zmin** and **Zmax** determine the thickness of the turret.

File Name — Use to specify the file name for swept solid model file to be created. Enter the *\path\filename* in the **File Name** text field, or click on **Browse** and use the file selection window that displays to specify the file.

Add Sweep Model — Creates a swept solid model using the above information and adds it to the Component Tree.



The features in the lower portion of the Turret Aid window enable you to easily create tool components for the turret.

Tool Component Name — Use to specify the base name for tool components to be created for the turret.

Starting Tool Index — Use to specify the starting index number to be used for the tool components created.

Starting Index Origin — The following features are used to describe the position of the first tool component on the turret. Think of the Starting Index origin as a polar location.

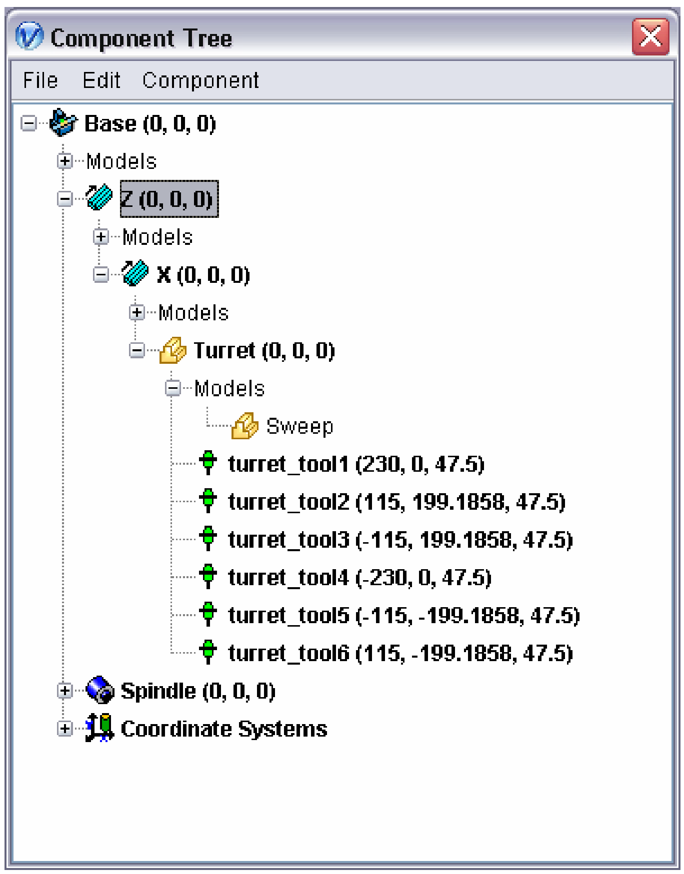
> **R —** Use to specify the radial distance from the turret origin that the tool components will be placed.
>
> **A degrees—** Use to specify the starting location polar angle, at which the first tool component is positioned. By default, if the number of faces is even, the starting location polar angle defaults to 0. If the number of faces is odd, the polar angle is .5(360/number of faces). That puts the default tool location at the center of a face in XY.

Z — Use to specify the Z location for the tool components. The default Z location is .5(Zmax-Zmin).

Index Direction — Use to specify the direction around the turret that subsequent tool components will index from the first position. Choose **Clockwise** or **Counterclockwise**.

Add Tool Component — Creates the tool components and adds them to the Component Tree. A tool component will be created for each face on the turret. The tool component positions will be rotated around the turret origin by "number of faces"/360, and positioned the radial distance (R) away from turret origin and indexed in the specified direction from the Starting Index Origin angle (A). Components will be named "tool component name"1, "tool component name"2, etc., depending on the Starting Tool Index number chosen.

The turret model and tool components in the following Component tree were created using the information in the Turret Aid window shown at the top of this section.

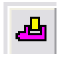


Close — Close the Tool Aid window.

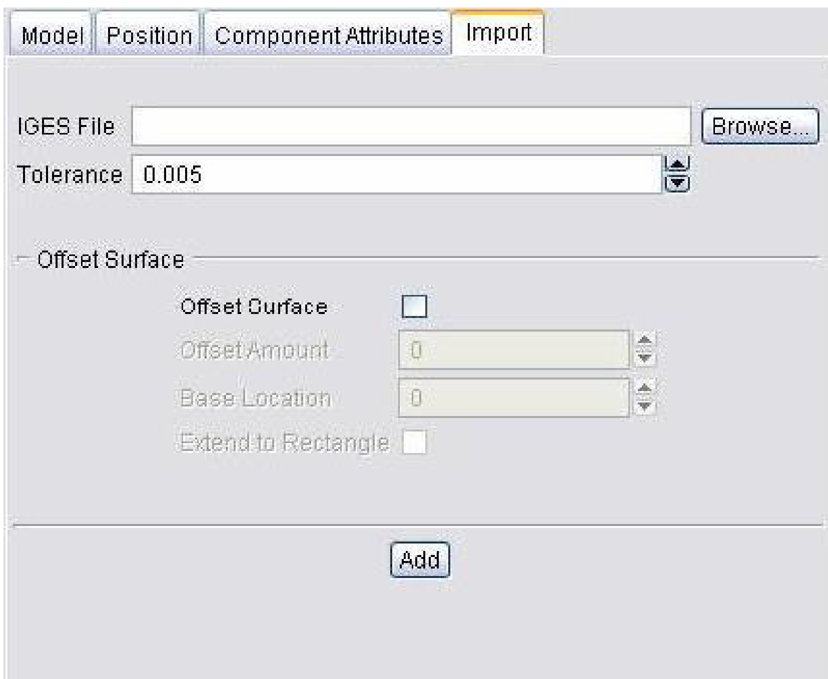
To Component Attributes tab

Modeling window, Import tab

Location: **Project menu > Setup Models > Define**

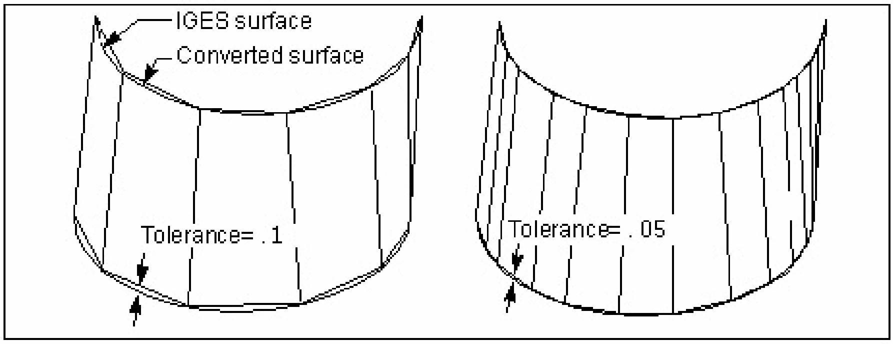
Toolbar short cut: 

Features on this tab import IGES model files for use as stock, fixture, design, or machine component models. The component type determines how the converted IGES model will be used in VERICUT. An option is also available to create a solid model from an IGES surface. The surface can be used as is, or offset before converting it into a solid. During import, VERICUT converts the IGES model data into a VERICUT Polygon file (.ply extension) and automatically refers the converted file. This window functions similar to the IGES Converter window-ref. **File menu > Convert > IGES**.



IGES File — Name of the file containing the IGES data to import. Either enter the */path/filename* in the IGES File text field, or click on the **Browse** button and use the file selection window that displays to select the file.

Tolerance — Specifies the amount of chordal deviation allowed in 3-D space from the IGES surface when creating the converted surface. The converted surface is approximated using "facets", or triangles.



Offset Surface group:

Offset Surface — When active, creates a solid model from an open IGES surface.

Offset Amount — Distance and direction in which to offset the design surface, as applied normal to the input surface. For example: ".100" applies an offset of .100 to the surface in the Z+ direction. A value like "-.100" applies the offset in the Z- direction, effectively "shrinking" the model.

Base Location — Specifies the Z value of the plane to become the base of the solid model. The surface is projected along the Z-axis to this plane.

Extend to Rectangle — When active, expands surface edges to create a rectangular base. The size of the rectangular base is determined by the X-Y bounding region of the surface data.

Add — Adds the VERICUT polygon model defined to the active component.

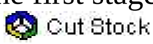
To Modeling window

Load Stocks

Location: **Project menu > Setup Models > Load Stocks**

After cutting, selecting this option causes VERICUT to load new stock models for Stock components having models defined, but a cut stock model does not exist. This capability is used when simulating a series of parts being machined through multiple machining stages. This type of manufacturing is often referred to as "staged machining".

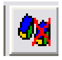
In general, the following actions take place in staged machining:

1. Simulate the first stage cutting operation. (Creates a "Cut Stock" workpiece that appears as  in the **Component Tree window**)
2. Move (e.g. cut and paste) the cut stock to a different Stock component defined to represent the next cutting stage.
3. Select **Load Stocks** to load a new uncut stock model in the first cutting stage.
4. Continue cutting-typically cutting the first and second stages, but may cut all stages even though workpieces have not yet been placed in subsequent cutting stages.
5. Repeat actions as required to cut a workpiece through all cutting stages required to complete the final part configuration.

Delete Detached Stock

VERICUT Users:

VERICUT Location: **Project menu > Setup Models > Delete Detached Stock**

Toolbar short cut: 

Cutter Grinder Users:

Cutter Grinder Location: **View Simulation page > Delete or Keep Material**

Notebook Feature: 

Opens the Delete Material window enabling you to delete or keep pieces of material. This feature is typically used to delete pieces of excess stock that have been cut free from the workpiece. However, pieces of other models can also be deleted.

Items are selected for **Delete/Keep** in the Workpiece View. Any number of items can be selected. Selected items are immediately highlighted with the Error color. Once you click **Delete** (or **Keep**) the items are removed from all views. However, the selected items are not actually deleted until you click **OK**.

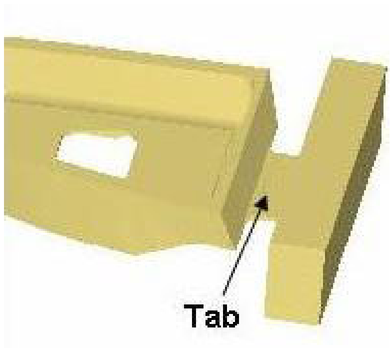
NOTE: Once model pieces are deleted, model pieces can not be restored.

Update While Simulating — When toggled "On", VERICUT checks during the simulation for detached material based on the stock's relationship to all sibling and parent fixture models. If the detached stock does not intersect, or is not tangent to, a parent or sibling fixture model, then the piece is considered "unattached" and is automatically deleted. This logic applies to both milling and turning operations.

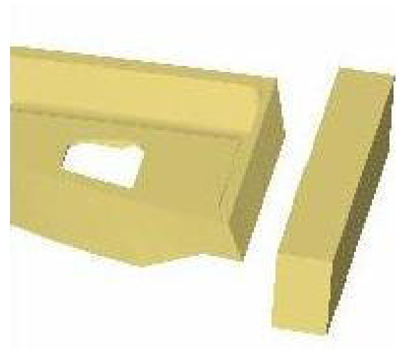
For turning mode, VERICUT checks at the end of each block. For milling mode, VERICUT checks at each tool change, at the end of each file, whenever VERICUT is stopped (except for "single step"), at Clamp/Unclamp events, and whenever the **CheckForLooseMaterial** macro is called. See the example in the next section.

Tab Removal — Use this feature to remove tabs left on the machined part (used for holding the part during the machining process) from a VERICUT cut model. VERICUT will project the cutter used to cut the selected machined surface, through the bottom of the part, removing the tab area. More than one position on the machined surface may need to be selected to remove the entire tab area.

Before Using Tab Removal



After Using Tab Removal



OK — Use this feature to permanently delete the items selected for **Delete**/**Keep**, or activate **Update while Simulating** and close the Delete Material window.

Delete — After selecting the items to be deleted (they will be highlighted), click **Delete** to remove selected items from all views. Click **OK**, described above, to permanently delete the items.

You can also use **Delete** to delete the un-displayed portion of a sectioned view. Without selecting any items, click on **Delete**. You will be prompted with the message "The cut stock is sectioned. Delete the sectioned (invisible) material?" Yes/No". A "**Yes**" response permanently deletes all un-displayed (invisible) sectioned material. A "**No**" response does not. Using the **Keep** in this manner will produce the same results.

Keep — After selecting the items to keep (they will be highlighted), click **Keep** to remove all un-selected items (not highlighted) from all views. Click **OK**, described above, to permanently delete the items.

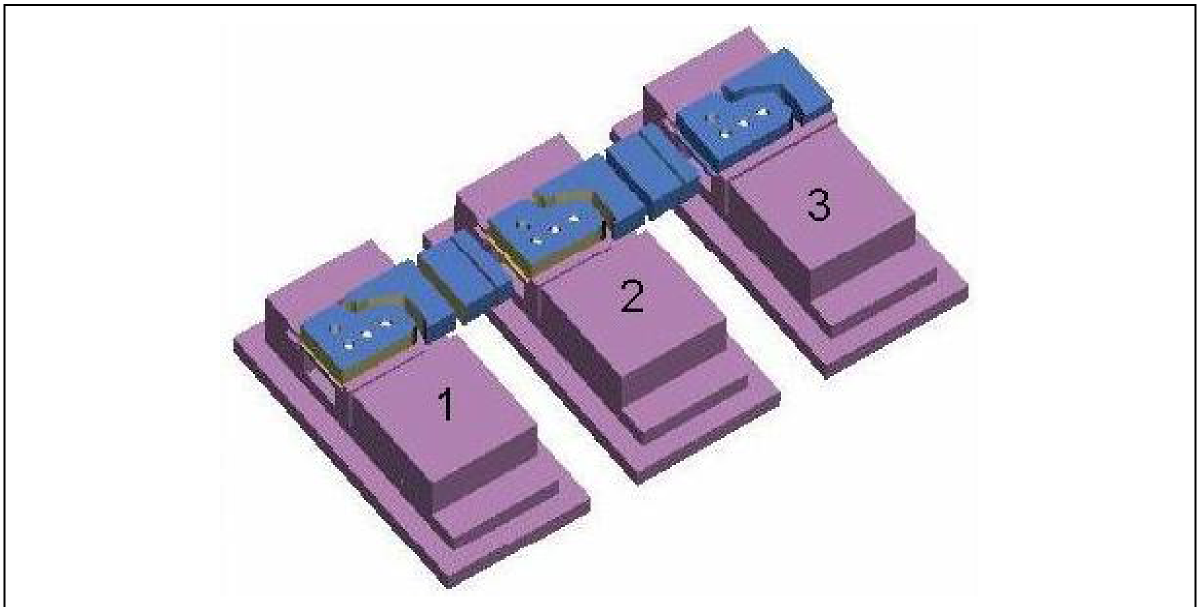
Unselect — Use this feature any time while selecting items for **Delete**/**Keep** to un-select all selected items (they will no longer be highlighted). This feature becomes inactive once you click on **Delete**/**Keep**.

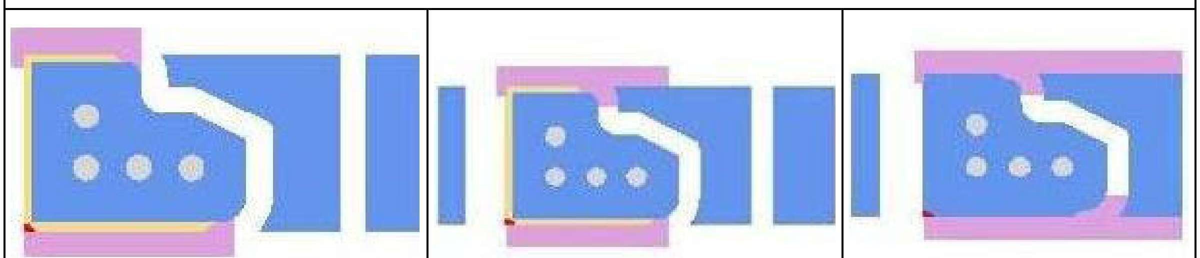
Cancel — Use to un-select all items, cancel any **Delete**/**Keep** action, and close the Delete Material window.

Delete Detached Stock - Update While Simulating Example

Update While Simulating Toggled "Off":

Notice that when **Update While Simulating** is toggled "**Off**", the pieces of excess stock that have been cut free from the workpiece remain displayed, as shown in the pictures below.

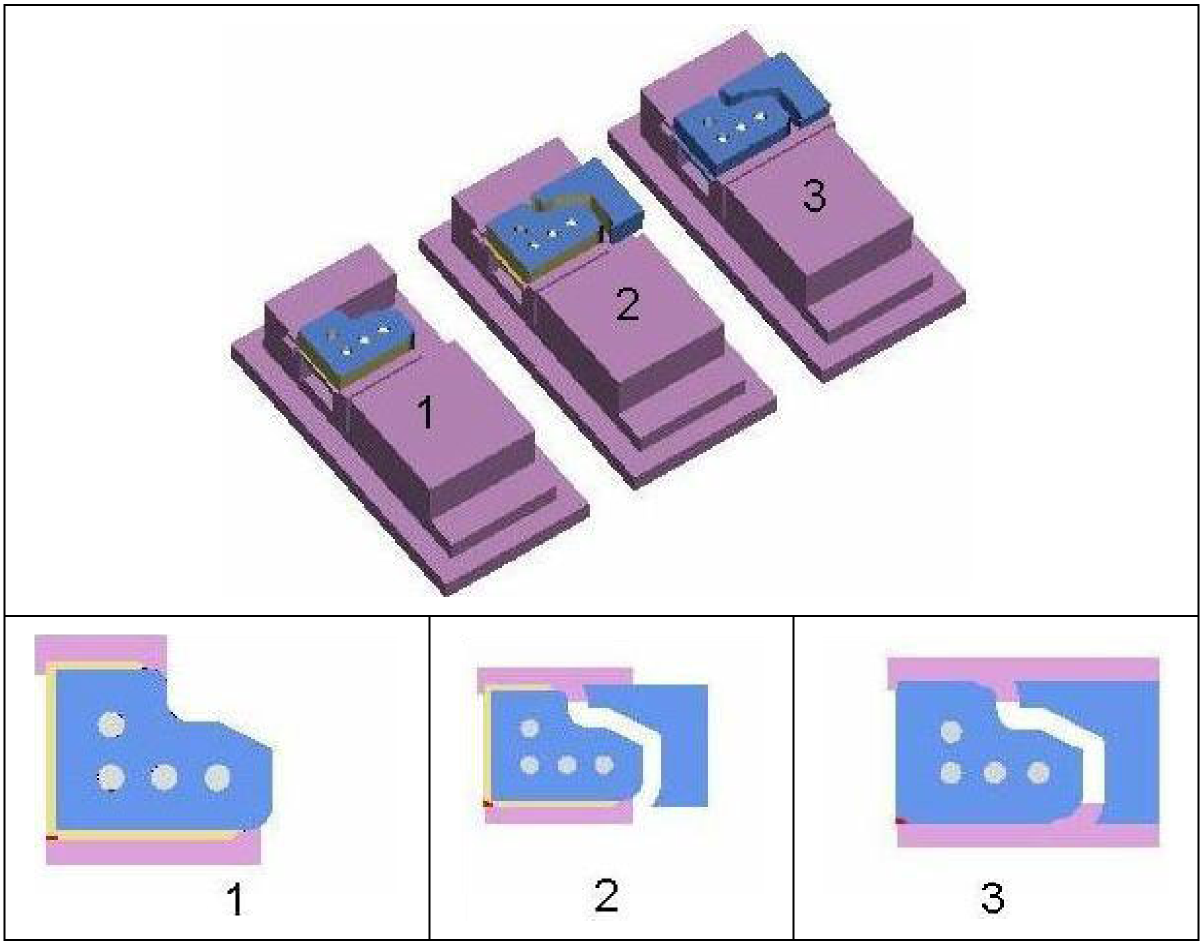




1	2	3

Update While Simulating Toggled "On":

Notice that when **Update While Simulating** is toggled "**On**", most of the pieces of excess stock that have been cut free from the workpiece have been automatically deleted, as shown in the picture below. The reason that they were not all automatically deleted is because of slight differences in the holding fixtures. In setups 2 and 3, the remaining excess stock pieces are at least partially in contact with the fixture so they were not automatically deleted.



Create Model File

VERICUT Location: **Project menu > Setup Models > Create Model File > Revolve / Sweep**

Opens the **Profile Sketcher window** enabling you to define a profile to be used to create a solid model either by rotating the profile about the Z-axis (Revolve), or by sweeping the profile along the Z-axis (Sweep).

Revolve — Click this option to open the Profile Sketcher window in "solid of revolution" mode.

Sweep — Click this option to open the Profile Sketcher window in "sweep solid" mode.

Profile Sketcher Features

Locations:

Project menu > Setup Models > Create Model File > Revolve

Project menu > Setup Models > Create Model File > Sweep

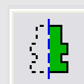
Project menu > Setup Models > Define > Model tab: Type=Model File > Sketcher > Revolve

Project menu > Setup Models > Define > Model tab: Type=Model File > Sketcher > Sweep

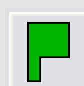
Project menu > Tools > Add(Modify) Cutter >Tool Component tab:

Project menu > Tools > Add(Modify) Insert >Tool Component tab:

Project menu > Tools > Add(Modify) Holder >Tool Component tab:

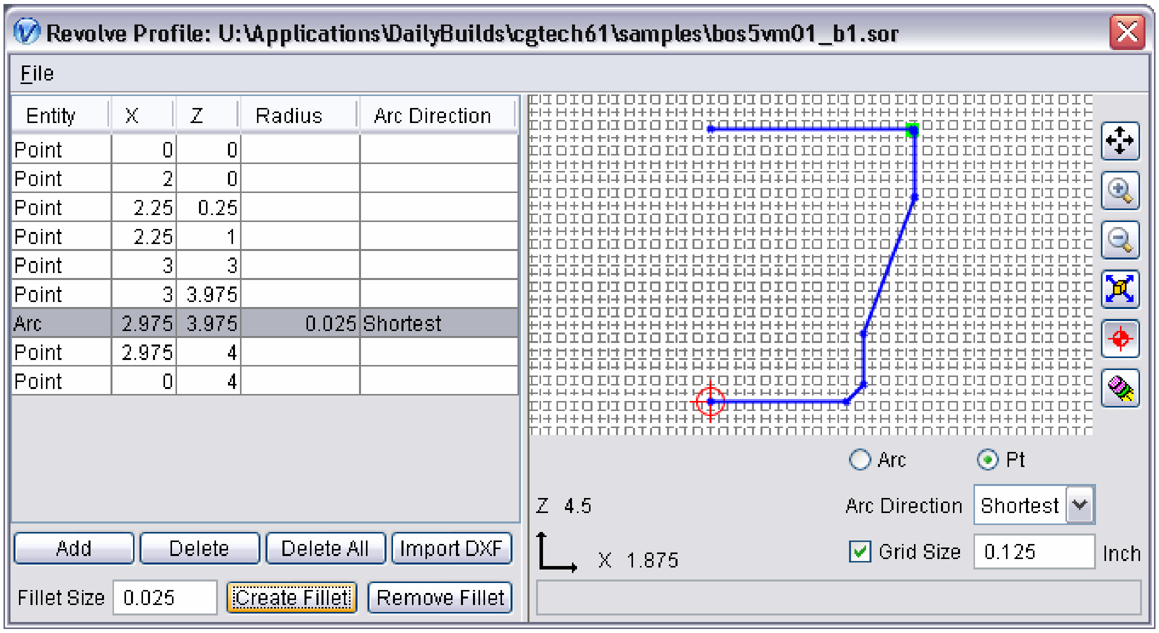
 (Revolve Profile)

Project menu > Tools > Add(Modify) Holder >Tool Component tab:

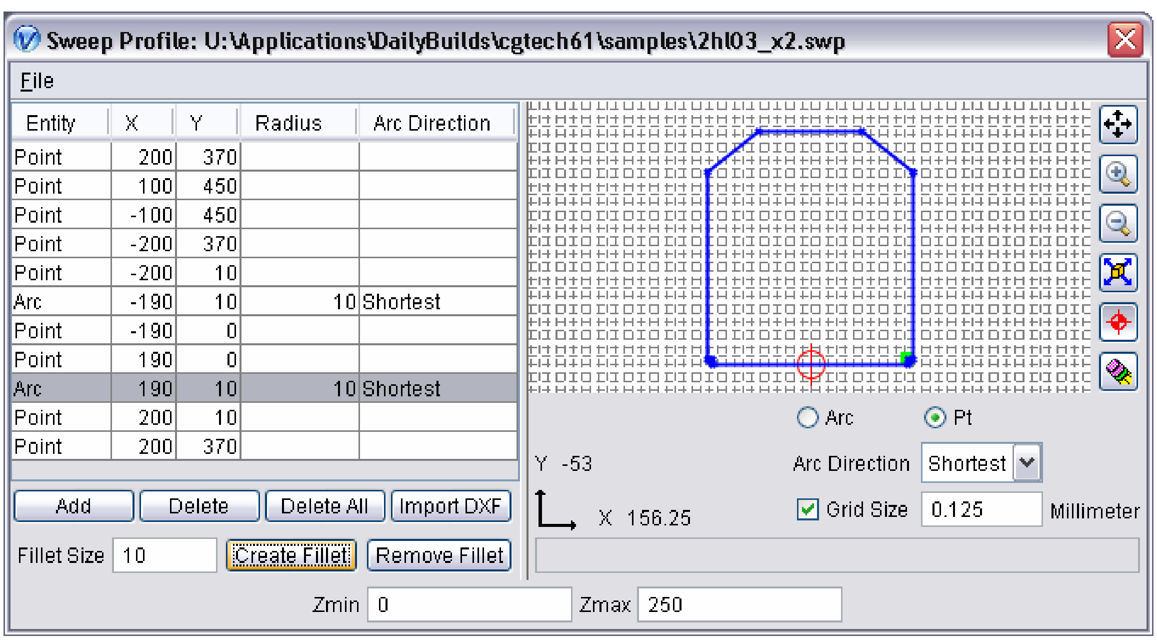
 (Sweep Profile)

The Profile Sketcher provides an easy graphical means of describing a profile of in 2-D drawing space. Profiles are sketched by connecting a series of lines and arc segments. You can select points, enter point/arc data into the Profile entity list, or pick then edit entity values in the list. You can also import the 2-D geometry from a DXF file. The profile displayed in the sketcher window is constantly updated regardless of how you supply profile entity data, making it easy to see what you've defined.

The Profile Sketcher can be opened in either "**Revolve**" mode, where an "open" 2D profile is rotated about the Z-axis to create a solid model or cutter/holder,



or in "**Sweep**" mode, where a "closed" 2-D profile is "swept" a specified distance along the Z-axis to create a solid model or a cutter insert/holder.



The Profile Sketcher is accessible from **Project menu > Setup Models > Create Model File > Revolve/Sweep** and from **Project menu > Setup Models > Define: Model tab** to create profiles that can be rotated about an axis, or swept along an axis to create solid models. It is also accessible in **Tool Manager** to Add/Modify cutters, inserts and holders.

The options available in the Profile sketcher will vary slightly depending on whether it was accessed from one of the Model menu windows or from Tool Manager. Most of the features are common regardless of where the window was accessed from and have identical functionality. Those features that are specific to a particular function are located at the bottom of the window are described in the **Special Features** section below.

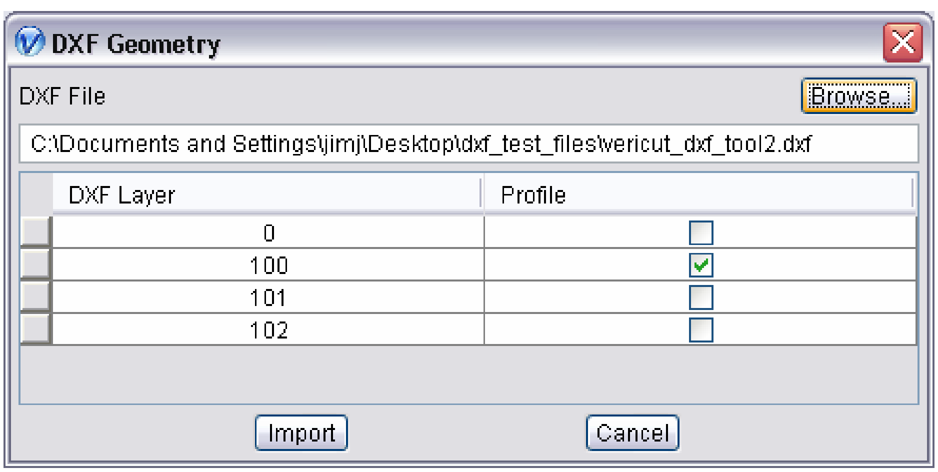
Common Features

Entity list — Lists the point/arc entities that define the profile. The list is created by graphical selection, or by pressing Add to add entities to the list. Existing data can be edited directly in the Entity list..

Add / Delete — Adds (or deletes) a single entity to/from the Entity list. Once added, the entity type (**Point** or **Arc**) can be changed, or data values edited.

Delete All — Deletes all entities in the Entity list.

ImportDXF — Opens the **DXF Geometry window** enabling you import two dimensional tool geometry from files which comply with a de facto CAD system standard, the Data eXchange Format (DXF) to add (or modify) individual cutter (or holder) components in existing tool records.



DXF File — Enter the */path/filename* in the text field, or select **Browse** and use the file selection window that displays, to select the DXF file containing the desired profile.

Import — Use to import the selected profile into the profile sketcher.

Cancel — Use to close the DXF Geometry window without importing the profile.

Fillet Size — Use this text field to specify the size of the fillet to be created. Use with **Create Fillet** described below.

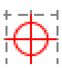
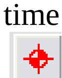
Create Fillet — Use to create a fillet at the selected point. Use **Fillet Size** described above, to specify the size of the fillet.

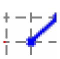
Remove Fillet — Use to remove the selected fillet.

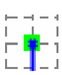
Shortcut: Clicking with the right mouse button in the Sketcher window (Revolve Profile or Revolve Sweep) displays the following shortcut menu:

These features provide the same functionality as those described above under Common Features.

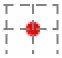
Sketcher window — Displays the current defined tool profile (blue line), based on the entity list. The profile in the window is constantly updated to reflect current list data. The following symbols may be seen in the sketcher window:

 **Tool origin —** The tool origin is assumed by VERICUT to be the control point, or driven point, on the tool. Typically, tool profiles are sketched relative to the tool origin to provide proper cutting by the tool. However, the tool origin can be relocated at any time via dragging the origin point. Tool origin display is controlled by the Origin  feature on the sketcher Toolbar (see below).

 (blue dot) **Entity point —** This symbol indicates a point on the profile or an arc center point. You can change the profile shape by dragging entity points to different locations. The corresponding entity in the list is automatically updated.

 (green dot) **Selected entity point —** This symbol indicates the entity selected for editing. The corresponding entity in the list is also highlighted for editing.

NOTE: Selecting a line segment in the Sketcher window selects both end points for editing. Drag the line segment to the desired location and both end points are automatically updated in the Sketcher window as well as in the Profile Entity List.

 (red dot) **Target selection point —** This symbol follows the mouse and is the target for graphical selection. Values displayed with the Sketcher axes below the sketcher window reflect the target selection point location.

Sketcher axes — Axes that depict the coordinate system in which the tool profile is being constructed. Values displayed with the axes reflect the target selection point location, relative to the tool origin.

Arc / Pt — Controls the profile entity type being defined via graphical selection.

Arc Direction — Controls the direction of an Arc being defined via graphical selection.

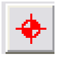
Grid Size — When selected, displays a grid in the sketcher window. When a grid is displayed, target and selected points snap to the nearest grid line intersection. Grid spacing is specified in the data field located right of the GridSize label, and can be changed at any time. Clear the checkbox to remove the grid and select or drag points anywhere in the 2-D drawing space.

Sketcher Message area — Located below the sketcher window, this un-editable text field displays error and informational messages to assist you with defining tool profiles. If actions in the Tool Manager, and/or sketcher, window are not as expected, check the VERICUT main window message area for additional relevant messages.

Sketcher Toolbar — The icons on this Toolbar zoom in/out, fit, and pan the tool profile in the sketcher window. See the table below for details. Sketcher icons:

Shortcut: The following keys provide instant access from the keyboard to dynamic viewing options (press and hold keys while dragging): **Dynamic Zoom —** <**Ctrl**>, **Dynamic Pan —** <**Shift**>.

Icon:	Name:	Action:
	**Pan**	Pan/translate- drag mouse in the direction pan
	**Zoom In**	Zooms in approximately 20% each time it is clicked
	**Zoom Out**	Zooms out approximately 20% each time it is clicked
	**Fit**	Fits the tool in the sketcher window

	**Origin**	When selected, displays the tool origin point
	**Undo**	Select to "undo" the previous action(s).

Special Features

The following are only available in the Sweep Profile window:

Zmin / Zmax — Used to specify the starting and ending location along the Z-axis when creating a "swept" solid model.

The following features are only available when the Sweep or Revolve Profile window is accessed from the Tool Manager. The specific features available will vary depending on the tool type (Mill, Turn, or Probe) and the tool component type (Revolved Cutter, Insert Cutter, or Holder) and the profile window type (Sweep or Revolve).

Color — Color that the tool component is to be displayed in VERICUT. For additional information, see Tool Display Colors in the Tool Add/Modify window, Common Features section.

Do Not Spin with Spindle — Toggle On, or Off, to specify whether or not the tool component spins with the spindle. For example, a milling tool holder would spin with the spindle, while a turning tool holder would not.

Flute Length — Used to specify the length of the cutter having flutes, or teeth, that can remove material.

Spindle Direction — Use to specify spindle direction for a specific revolved cutter. Choose **CW** (clockwise) or **CCW** (counterclockwise).

Thickness — Used to specify the thickness of the insert or holder (sweep profile only).

NOTE: For additional information on these Tool Manager features, see the specific tool component in the Tool Add/Modify window.

DXF Geometry window

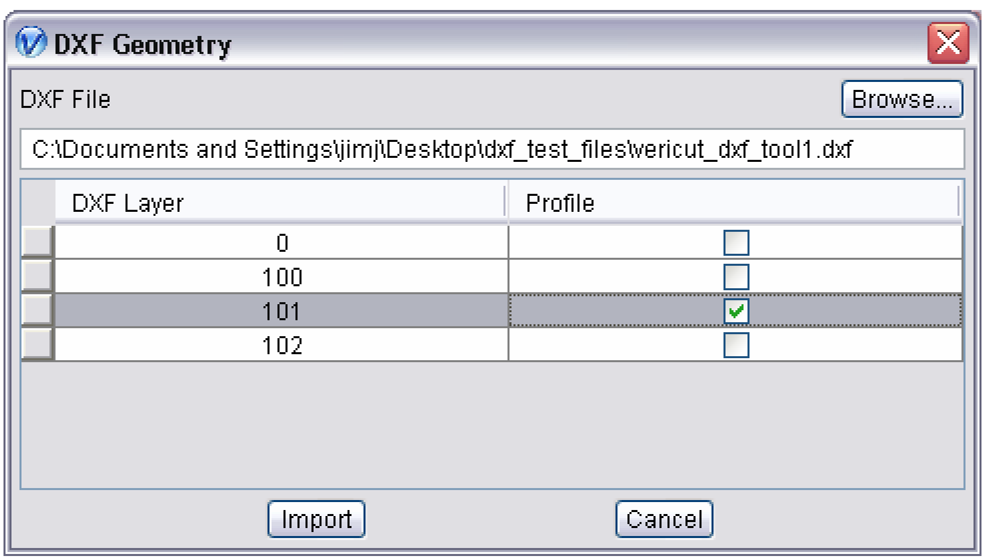
Locations: **Profile Sketcher window > Import DXF**

Tool Manager window > File > Import > DXF Tool

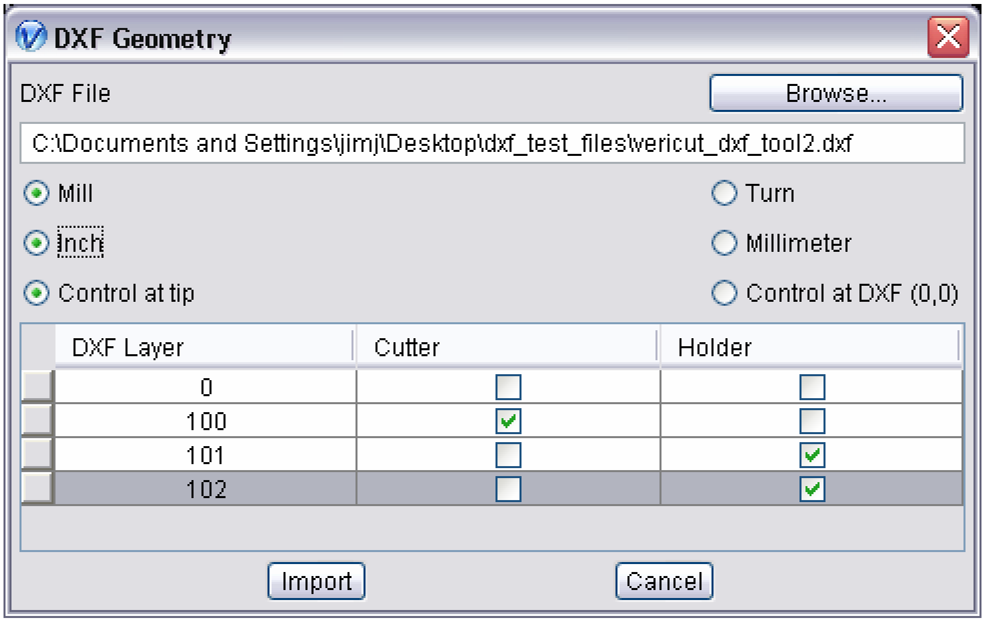
Opens the **DXF Geometry window** enabling you to import two dimensional geometry from files which comply with a de facto CAD system standard, the Data eXchange Format (DXF).

The Profile Sketcher window is accessible **Project menu > Setup Models > Create Model File > Revolve/Sweep** and from **Project menu > Setup Models > Define: Model tab** to create profiles that can be rotated about an axis, or swept along an axis to create solid models. It is also accessible in **Tool Manager** to Add/Modify cutters, inserts and holders.

The options available in the DXF Geometry window will vary slightly depending on whether it was accessed from one of the Model menu windows,



or from Tool Manager.



Many of the features are common regardless of where the window was accessed from and have identical functionality, Those features that are specific to either of the Model menu windows or Tool Manager windows are noted in the feature descriptions below.

Importing DXF Profile Data:

DXF File — Use to specify the DXF file containing the tool data. Enter the /path/file name in the text field or click on Browse and use the Select DXF File window to specify the file.

Mill / Turn — (Tool Manager only) Select **Mill**, or **Turn** to specify, the "type" of tool data being imported.

Inch / Millimeter — (Tool Manager only) Select **Inch**, or **Millimeter**, to specify the "units" of the tool data being imported.

Control at tip/ Control at DXF (0,0) — (Tool Manager only) Select **Control at tip**, or **Control at DXF (0,0)**, to specify the "control point" to be used for the tool data being imported. **Control at tip** puts the control point at the tip of the tool (the point on the profile with the lowest Z value is assumed to be the tip of the tool). **Control at DXF (0,0)** puts the control point at the 0,0 point of the profile.

Component Table —

After specifying the DXF file, a record is created in the Component Table for each layer contained in the file. You can re-order the records in the table by clicking on the button in the first column of the record and dragging it to the desired position.

DXF Layer — This column displays the name, or number, identifier of the layer.

Profile — (Create Model File only) This column is used to identify the profile that is to be used.

Cutter — (Tool Manager only) This column is used to identify the "cutter" geometry data to be imported. Click on the box in the appropriate layer row to activate it (checkmark is visible).

Holder — (Tool Manager only) This column is used to identify "holder" geometry data to be imported. Click on the box in the appropriate layer row(s) to activate it (checkmark is visible).

NOTE: When the DXF Geometry window is accessed from the Tool Manager, only layers that have been indicated as "cutter" or "holder" will be imported. If the DXF window was accessed using Add > DXF Tool in the Tool Manager, only one "cutter" component can be imported, but multiple "holder" components can be imported for each tool record.

Import — Imports the geometry data into the appropriate Profile Sketcher window, or in the case of Add > DXF Tool in the Tool Manager, into the appropriate tool record/tool component.

Cancel — Use to close the DXF Tool Geometry window.

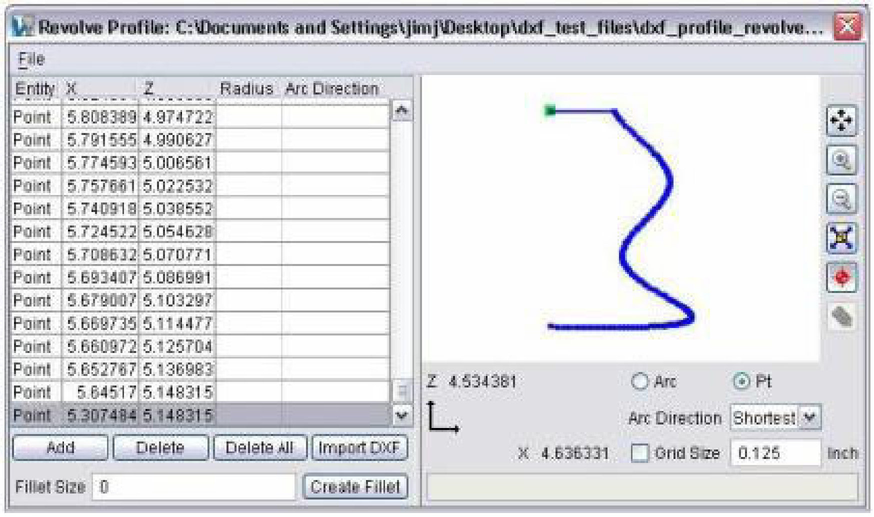
DXF File Requirements:

Only the ENTITIES section of a DXF file is scanned by VERICUT. The BLOCKS section is ignored, so BLOCK references within the ENTITIES section will not be expanded. REPEAT entities are also ignored. The only geometric entity types processed are LINE, ARC, CIRCLE, POLYLINE, LWPOLYLINE, TRACE, SOLID, 3DLINE and 3DFACE .

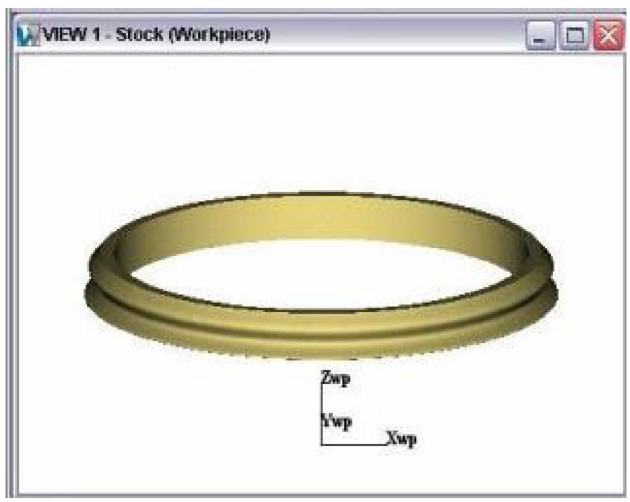
For Creating Solid Models

Import "open" 2D profiles to be rotated about the Z-axis to create "revolved" solid models. Putting the start and end points of the profile off of the Z-axis will result in a solid model that is open in the center like the one shown in the "Revolved Solid Model" picture below. To create the model with a solid center, make sure that the start and end points of the profile are on the Z-axis.

Open Profile

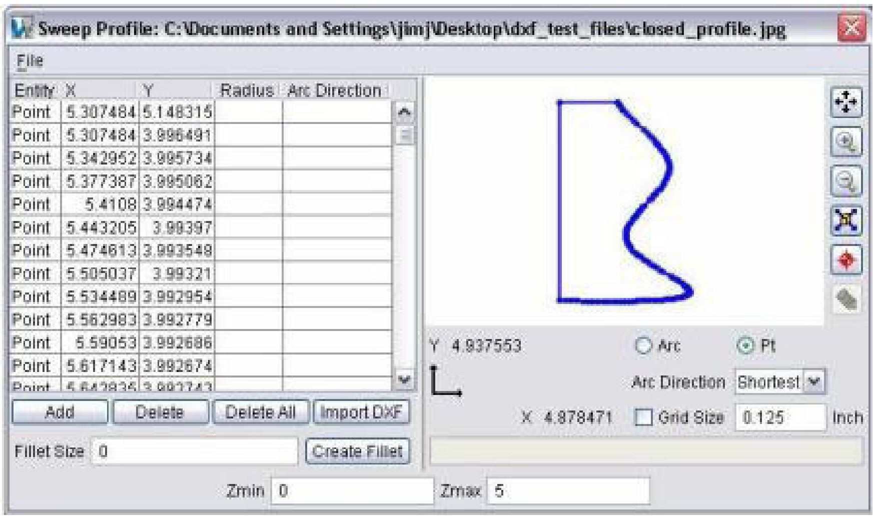


Revolved Solid Model

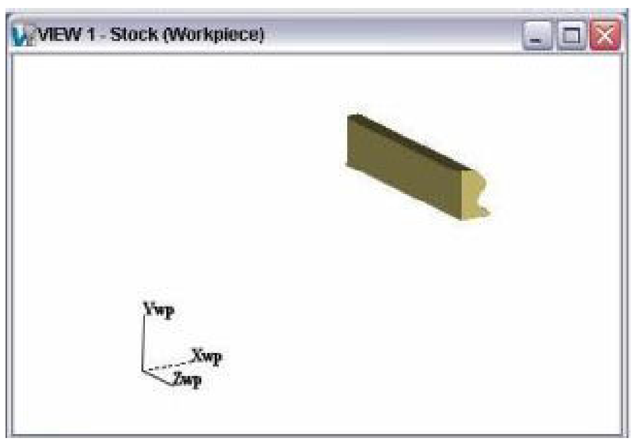


Import "closed" 2-D profiles to be "swept" a specified distance along the Z-axis to create a solid model or a cutter insert/holder.

Closed Profile



Swept Solid model

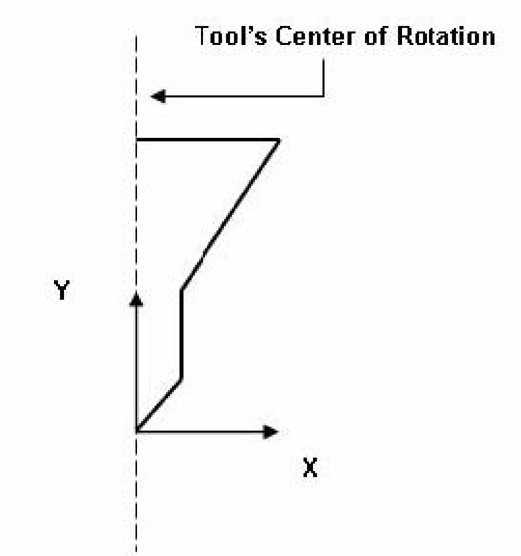
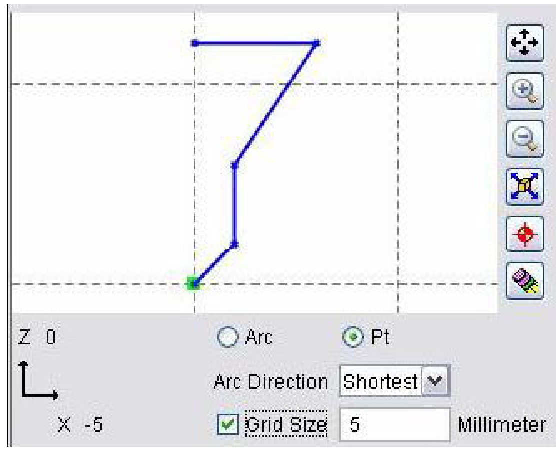
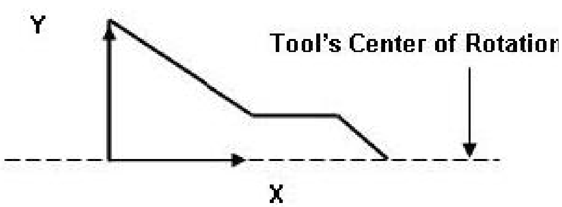
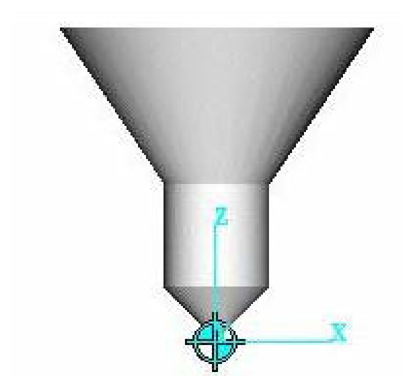


For Creating Cutters, Inserts and Holders

VERICUT expects only one tool per DXF file. Each component (cutter or holder) of the tool should be on its own named or numbered layer. There can be no more than one cutter component, but multiple holder components are permitted. The DXF file may contain other layers which are unrelated to the definition of the tool's geometry. Within each component layer, LINEs, ARCs, CIRCLEs, POLYLINEs, LWPOLYLINEs, TRACEs, SOLIDs, 3DLINEs and 3DFACEs can be used to define the profile of the tool.

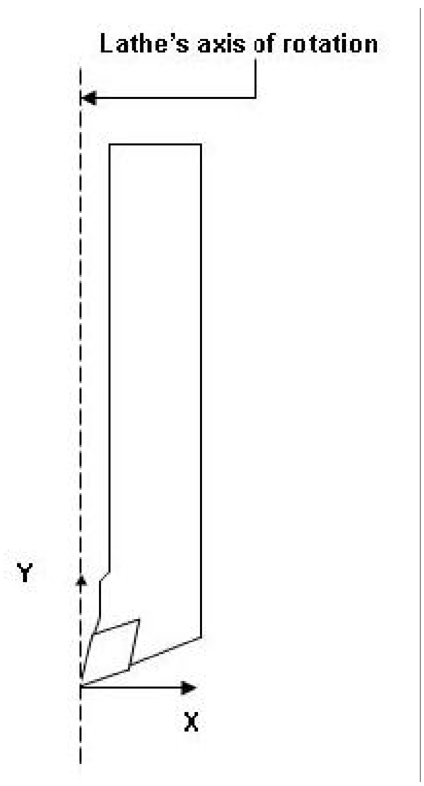
For a milling tool, the Y coordinate of the DXF data corresponds to the tool's axis of rotation, known as the Z axis in VERICUT's Tool Manager. The X coordinate of the DXF data corresponds to the Tool Manager's X axis. An exception to this is a DXF file recognized by its use of layer names as being generated by TDM Systems', Tool Data

Management (TDM) system. In such a file, the X-axis is the tool's axis of rotation, with the tool tip on the right, and the Y coordinate is the radial distance from the rotation axis.

DXF Data	**In VERICUT**
	
DXF Data from TDM 	

For a turning tool, the X coordinate of the DXF data corresponds to the lathe's axis of rotation, known as the Z axis in VERICUT's Tool Manager. The Y coordinate of the DXF data corresponds to the Tool Manager's X axis.

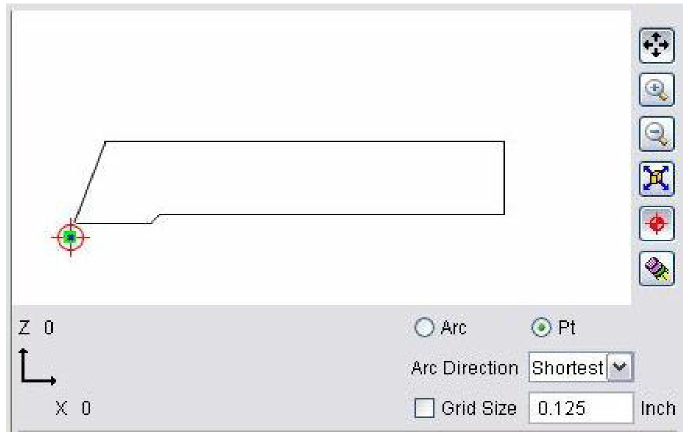
<table>
<tr><th>DXF Data</th><th>In VERICUT</th></tr>
<tr><td>

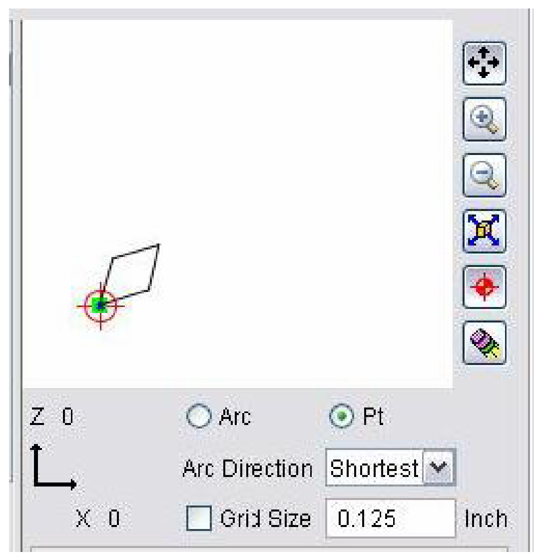


Cutter on Layer 1

Holder on Layer2

</td><td>





</td></tr>
</table>

NC Programs (NC Program window)

VERICUT Users:

VERICUT Location: **Project menu > NC Programs**

Toolbar short cut: 

Mold and Die Users:

Mold and Die Location: **Setup page > Select Tool Path Files and Tools**

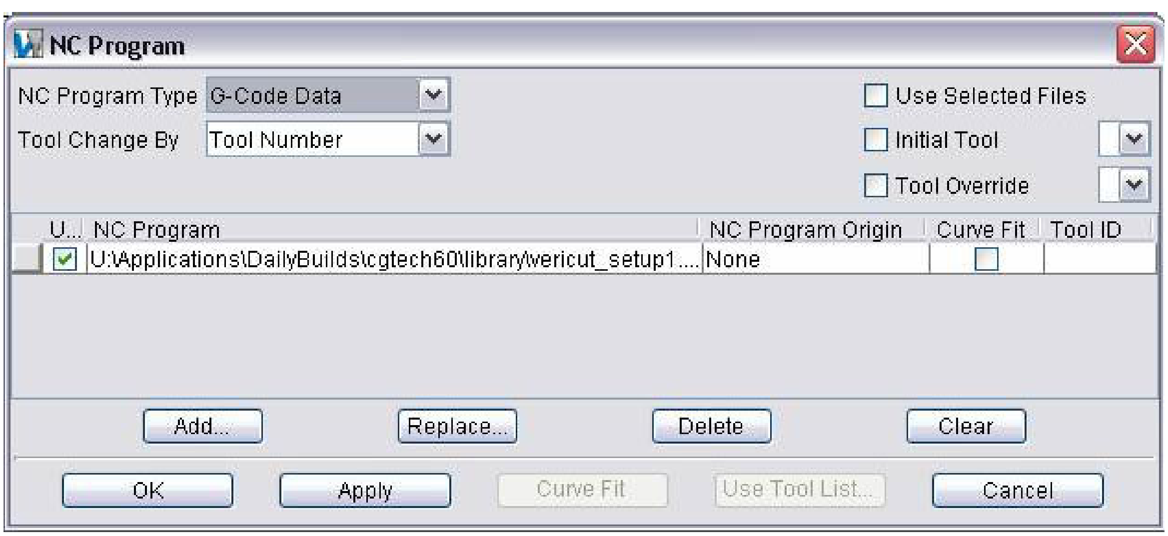
Notebook Feature: 

Cutter Grinder Users:

Cutter Grinder Location: **Setup page > Select Grinding Programs**

Notebook Feature: 

Opens a window to specify the NC program file(s) to simulate, and controls how VERICUT receives descriptions of cutting tools. In general, VERICUT can obtain descriptions of cutting tools by processing parametric cutter description records found in the NC program file, or can be configured to retrieve them from a VERICUT Tool Library file.



NC Program Type — Type of NC program file for VERICUT to simulate. By default, VERICUT simulates G-Code or "machine code data" NC programs destined for a 3-axis mill machine with a "Fanuc-like" control. Using this option, you can quickly configure VERICUT to simulate NC programs from all popular CAM systems. Options:

G-Code Data — G-Code NC program file. With this choice, VERICUT uses a Machine file, Control file, and G-Code settings (ref. **Project menu > Processing Options > G-Code > Settings**) to interpret codes present in the file.

VERICUT APT — Generic ASCII APT NC program file from virtually any source, including ACL format.

NX CLS — NX CLS NC program file.

CATIA APT — CATIA APT source NC program file. This file must be processed from CATIA using the CATUTIL function and typically has a ".APTSOURCE" extension.

CV APT — ComputerVision APT NC program file.

CADRA APT — CADRA APT NC program file.

APT (RevPost) — ASCII APT NC program file created from reverse post-processing G-Code data Refer to "**Reverse post-process G-Codes to APT**" in the **Configuring For Toolpath Simulation** section of *Using VERICUT* for more information. *Using VERICUT* can be found in the *CGTech Help Library*.

Pro/MFG APT — Pro/Manufacturing APT NC program file.

NCL APT — NCL APT NC program file.

Siemens VNC 840D — Displays the **VNCK Control and Main Toolpath** window.

See **840D Virtual NC Kernel and Interface (VNCK)** in the *Converters and CAD/CAM Interfaces* section of the *CGTech Help Library*.

In addition to the NC program types above, VERICUT's converters enable you to simulate NC programs from various CAM systems. See the following sections for more information:

- **File menu > Convert > Binary CL.** See the "**Binary Converter**" topic in the *Converters and CAD/Cam Interfaces* section. *Converters and CAD/Cam Interfaces* can be found in the *CGTech Help Library*.
- "**Reverse post-process G-Codes to APT**" in the **Configuring For Toolpath Simulation** section of *Using VERICUT* for more information. *Using VERICUT* can be found in the *CGTech Help Library*.

Tool Change By — Controls how VERICUT receives descriptions of cutting tools.

Cutter Desc. — Process parametric cutter descriptions in the NC program file. Examples: CUTTER and VERICUT-TC comment records.

VERICUT TC — Process by PPRINT/VERICUT-TC cutter statements in the toolpath file.

Tool Number — Process NC program records that reference tool or pocket numbers and retrieve associated tools from a Tool Library file. Examples: TnM6 (G-Code NC programs), LOADTL or LOAD TOOL, and TURRET. When using Tool Change by Tool Number, The tool number is matched with the pocket number (ex: tool #5 in pocket #5).

List — Refer to a list of tool change event-to-tool ID references that retrieve associated tools from a Tool Library file. The list is accessed via the **Use Tool List** feature, as described later in this section.

File Name — Use a tool from a Tool Library file as specified for each NC program file in the list.

Tool Name — This option is only applicable when **NC Program Type** is set to **NX CLS**. Tool Name refers to the tool identifier in the TOOL PATH/ statement in the NX CLS file.

Use Selected Files — When used in conjunction with the **Use** checkboxes in the Toolpath list, enabling you to select only specific toolpath files from the list for VERICUT processing. If this option is not checked on, all toolpaths in the list will be processed in the order that they appear in the list.

Initial Tool — When active, VERICUT loads an initial tool from a Tool Library file. This action occurs automatically when the Project file is loaded, the NC program is rewound, or the model is reset. Choose the ID of desired tool using the option list next to this feature. This feature is useful when simulating NC programs for machines that have a tool loaded at the time tool processing is started, therefore do not have information about the first tool.

You can also use this feature to preload multiple tools in a milling machine (e.g. tool change carousel), via selecting Initial Tool, but leaving the corresponding field blank. If you specify a tool number, then that tool is forced in to all tool component positions.

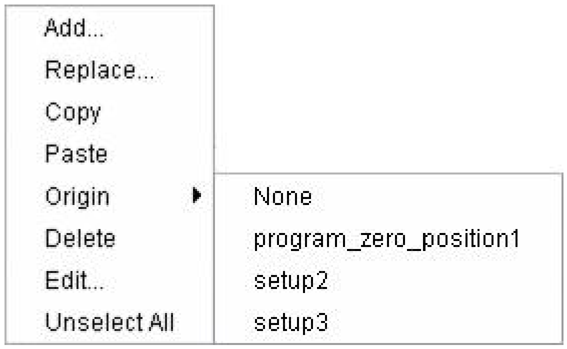
NOTES:

1. This feature should not be used if you are using multiple toolpaths.
2. This action also occurs when a "Turret" type component is present, such as commonly used with turret lathe setups.

Tool Override — When active, overrides the current cutter description with a specified tool from a Tool Library file. This action occurs immediately. Choose the ID of desired tool using the option list next to this feature. VERICUT uses the override tool until you select a different tool ID or clear the **Tool Override** checkbox. When cleared, the tool that would have been in use becomes the active tool.

NC Progam list — Also known as the "toolpath list", displays all of the NC program files available in the project file. The NC Program list consists of some, or all, of the following columns of information depending on the way that the NC Program Type and Tool Change By options are set.

Shortcut: Right-click in the NC Program List to display the following menu:



NOTES:

1. The Add, Replace and Delete options are the same as the buttons described below.
2. **Copy** / **Paste** enable you to copy the highlighted NC program and paste it at the end of NC Program List.
3. The **Origin** option is the same as the NC Program Origin option described below. The Origin pull-down list will contain all available coordinate systems.
4. The **Edit** option displays the **NC Program Editor window** with the highlighted NC program ready for editing. (ref. **NC Program (edit)** also in the *VERICUT Help* section, for additional information.
5. If **Use Selected Files** is toggled "On", **Unselect All** appears in the menu to enable you to uncheck all boxes in the **Use** column with a single pick.

List Order buttons — NC programs are processed by VERICUT in listed order. List order is changed via left-dragging the List Order button in the first column of each toolpath entry in the list. Only NC programs that have not been simulated by VERICUT can be reordered. Reset the model to reorder NC programs that have been already been simulated.

Use — This column is only displayed when the Use Selected Files option, described above, is toggled "On". Use the checkboxes in this column to select certain files in the toolpath list for VERICUT processing. By default all toolpaths are selected (checked) for processing.

NC Program — displays the path and file name of each toolpath currently available for VERICUT processing.

Tip: Double clicking on the NC program file name displays the **NC Program Editor window** with the NC program file ready for editing.

NC Program Origin — Use to specify the coordinate system that corresponds to each toolpath. Change the CSYS by clicking on current value in the Toolpath Origin column then select the desired CSYS from the pull-down list of available coordinate systems. The option None makes the toolpath origin relative to the Workpiece Origin for CL Data, and relative to the Machine Origin for G-Code data.

To change multiple Toolpath Origins at the same time, select the desired toolpaths using one of the following methods:

> *Select multiple files in sequence* — Click first file name in the sequence, then press and hold the <**Shift**> key while clicking the last file name. You can also click on the first file name in the sequence, while still holding down the mouse button, drag the cursor to the last file name in the sequence.
>
> *Select additional files* — Press and hold the <**Control**> key while selecting each additional file.
>
> With either method, selecting a file name a second time while holding down the <**Control**> key unselects the file.

Then right-click in the list area and select **Origin** from the shortcut menu. Select the desired CSYS from the pull-down list to update the NC program Origin for all highlighted NC programs.

Curve Fit — This column is only visible when Toolpath Type is set to G-Code Data. Use the check boxes in this column to select specific toolpaths for Curve Fitting. When you check one or more toolpaths for Curve Fit processing, the **Curve Fit** button, described below, becomes active enabling you to access the **NC Program Curve Fitting Control window** to set curve fit parameters and start the curve fit process.

Tool ID — This column is only visible when **Tool Change By** is set to **File Name**. It specifies the ID of a tool from the current Tool Library file to use with each NC program file.

Add — Opens a file selection window enabling you to add one or more NC program files to the list.

Replace — Displays a file selection window enabling you to select a toolpath file to replace the selected (highlighted) toolpath file in the list. If there is only one toolpath in the list, you do not need to select (highlight) it. Just click on **Replace** to display the file selection window and pick the replacement file. VERICUT will replace the existing NC program file with the new one. You can also double click on an NC program file name in the list to display the "file selection box". To select multiple NC program files for replacement, use any combination of the techniques below.

Select multiple files in sequence — Click first file name in the sequence, then press and hold the <**Shift**> key while clicking the last file name-all files in between those selected are also selected.

Select additional files — Press and hold the <**Control**> key while selecting each additional file. Selecting twice de-selects the file.

Delete — Deletes the selected NC program file from the list. To select multiple NC program files for deletion, use any combination of the techniques described above under Replace.

Clear — Clears the list of all NC program files.

Curve Fit — Displays the **NC Program Curve Fitting Control window** that enables you to set parameters for use during curve fitting. See the **NC Program Curve Fitting Control window** topic, also in this section.

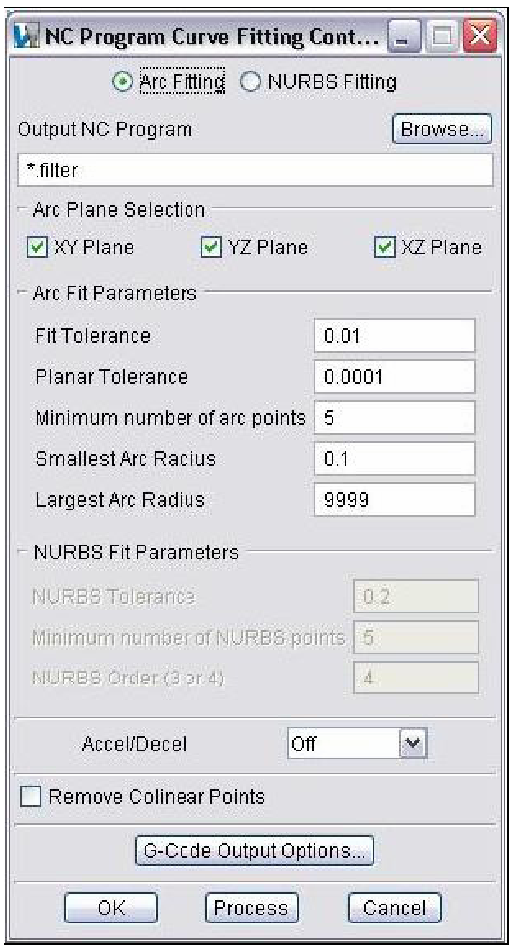
Use Tool List — When **Tool Change By=List**, this feature opens a window to create and maintain a list of references that link tool change events to tools stored in a Tool Library file. You can use this list to supply tools from the library for sequential tool changes, or to replace tools specified for use in the NC program file. See the **Tool Change List window** topic, also in this section.

NC Program Curve Fitting Control window

The curve fitting option is targeted at 3-axis G-Code NC programs containing long sequences of relatively short cutting motions. VERICUT detects sequences of planar motion and fits arcs (or NURBS) whenever the motion can conform to a specified tolerance. Arcs are only replaced along the machine's major axes (XY, YZ and XZ). The resulting NC program files may be smaller, produce an improved surface finish, and process faster in VERICUT than the original NC program file.

Curve fitting makes a single pass through the G-Code NC program(s) and writes new NC program file(s) containing the modified motions. The original NC program files are not changed. Curve fitting may be selectively applied to NC programs by checking the curve fit box in the NC Program list.

The "Curve Fit" button brings up the NC Program Curve Fitting Control window enabling parameters to be set and the curve fitting process to be executed.



Arc Fitting — Select this feature to fit arcs during Curve Fit processing. Selecting this feature also activates the **Arc Fit Parameters** described below.

NURBS Fitting — Select this feature to fit NURBS during Curve Fit processing. Selecting this feature also activates the **NURBS Fit Parameters** described below. NURBS Curve Fitting is supported for the Fanuc G6.2 (Generic control), the Siemens (BSPLINE format), and the Toshiba G6 machine controls.

Output NC Program — Specifies the name of the new G-Code NC program file containing the results of the curve fitting process. The Output NC Program file name must be different from that of the original NC program file name. Curve Fitting will not permit the original NC program file to be overwritten. When curve fitting multiple NC program files, a modified NC program file is output for each NC program file processed. Each output file has an ascending sequence number appended to the end of the base file name. For example: assume Output NC Program ="path.mcd" is entered (without the quotes). Curve Fit processing of 2 NC programs outputs the following files:

path001.mcd => modified NC program from NC program 1

path002.mcd => modified NC program from NC program 2

If a dot "." is the last character of the output NC program file name, then three digits are appended to the file name as the extension. Continuing with the above example: assume Output NC Program ="path." is entered. Curve Fit processing of the same 2 NC programs outputs the following files:

part.001 => modified NC program from NC program 1

part.002 => modified NC program from NC program 2

If an asterisk "*" is used in the output NC program file name, the original NC program file name is used as the output file name in place of the asterisk. If characters follow the asterisk, the input NC program file name extension is replaced with the characters following the asterisk.

Examples follow:

Original NC program file:	**Output NC Program file name:**	**Output file name:**
dir1/filename.mcd	dir2/*	dir2/filename.mcd
dir1/filename.mcd	dir2/*.filter	dir2/filename.filter
dir1/filename.mcd	dir2/*cf	dir2/filenamecf
dir1/filename.mcd	dir2/*cf.mcd	dir2/filenamecf.mcd
filename.mcd	*.cf	filename.cf

Only a single asterisk is acted on in the optimized NC program file name, with one exception: asterisk dot asterisk ("*.*"). This case behaves exactly like the single asterisk file name described above. This is ***NOT*** a regular expression replacement, therefore any characters before the asterisk are ignored and lost. When there is more than one asterisk, only the first one is replaced

Examples follow:

Original NC program file:	**Optimized NC Program file name:**	**Output file name:**
dir1/filename.mcd	dir2/*.*	dir2/filename.mcd
dir1/filename.mcd	dir2/cf*	dir2/filename.mcd
dir1/filename.mcd	dir2/cf*.ext	dir2/filename.ext
dir1/filename.mcd	dir2/*cf.*	dir2/filenamecf.*

Arc Plane Selection — Check boxes for major planes where arc fitting is requested.

Arc Fit Parameters:

The following options are only active when **Arc Fitting** has been selected above.

Fit Tolerance — Maximum deviation allowed for the fitted arc from the NC program points being fitted. All points replaced by a circular motion must be within this distance from the arc. In addition, the midpoint of all motion segments being replaced must be within this distance from the arc.

Planar Tolerance — Maximum deviation from the plane formed by the motions being tested. Points within this tolerance are adjusted to the plane defined by the sequence.

Minimum number of arc points — Lower limit on the number of points required to define an arc.

Smallest Arc Radius — Lower limit on radius of arcs created. Smaller arcs will remain a sequence of points.

Largest Arc Radius — Upper limit on radius of arcs created. Larger arcs will remain a sequence of points.

NURBS Fit Parameters:

The following options are only active when **NURBS Fitting** has been selected above.

NURBS Tolerance — Maximum amount that the NURBS interpolated curve can deviate from the from the linear cut segments being replaced.

Minimum number of NURBS points — Minimum number of linear cut segments required before a NURBS curve fit is attempted.

NURBS Order (3or4) — Use to specify the "order of the NURBS. Select 3 or 4 depending on what "order of NURBS" is supported by the machine controller.

Accel/Decel — This feature applies Accel/Decel settings from the NC machine configuration to adjust optimized feed rates. VERICUT looks ahead in the NC program file and slows the machine prior to direction changes, as well as changes feed rates based on the machine axes ability to accelerate and decelerate. For more information on the individual options, see **Accel/Decel** options on the **OptiPath window**.

Remove Colinear Points — When toggled "On", this feature eliminates colinear points during Curve Fit processing.

G-Code Output Options — Displays the **General** and **OptiPath & Curve Fit tabs** of the **Control Settings window**.

NOTES:

1. If you happen leave the 12 tab **Configuration menu > Control Settings** window open and it gets hidden behind other windows then use G-Code Output Options you may see all 12 tabs instead of just 2 tabs. This is because it is the same window, displayed differently depending on where it is called from.
2. When you use F1 Help on the 2 tab G-Code Output Options window, the documentation for the 12 tab **Configuration menu > Control Settings** window will display. The information for the General and OptiPath & Curve fit tabs is the same for both cases.

OK — Updates the settings saved with the job and dismisses the NC Program Curve Fitting Control window.

Process — Updates the parameters and processes the files selected for curve fitting. Processing multiple NC programs may require running the Curve Fit process more than once if the NC programs require different Curve Fit tolerances. For example, one pass might be used for roughing NC programs and use a larger tolerance. A second pass would then be used for finish NC programs using a smaller tolerance.

NOTE: When using Curve Fit, you must take into account the tolerances used by the CAM system to create the original NC program(s). Using large Curve Fit tolerances may cause the tool to deviate an unacceptable amount from the intended surface.

Upon completion of processing, the curve fit results are displayed in the following message:

Number of arcs created = xxx

Number of moves replaced = yyy

Blocks read = nnn

Blocks output = zzz

Do you wish to use the new file(s) as the active NC program file(s)?

A "**Yes**" answer ***will*** replace the list of NC programs for the current job with the modified NC program files produced by Curve Fit. A "**No**" answer ***will not*** replace the NC programs in the list.

Cancel — Dismisses the NC program Curve Fitting Control window without executing Curve Fitting or saving Curve Fit parameters.

Curve Fitting with a Tool List

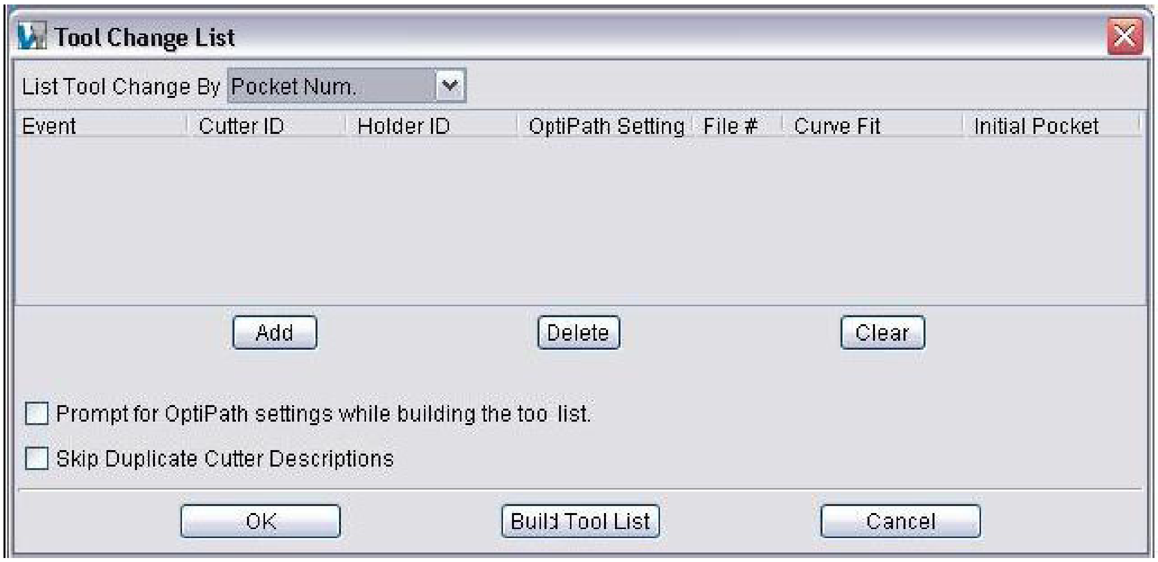
When **Tool Change By** is set to **List**, the **Use Tool List** button is becomes active. The Tool Change List window will contain a Curve Fit column of check boxes. One or more of these boxes must be checked for the appropriate curve fitting to be processed. If the Curve Fit column is hidden, adjust the width of the visible columns.

Arc Quadrant Output

Some machines require that circular blocks do not cross quadrant boundaries. To output this type of circular blocks, the macro **CircleCurveFitQuadrants** should be used as a Start of Processing Event macro. The default allows blocks to cross quadrant boundaries.

Tool Change List window

Opened via pressing **Use Tool List** on the **Project menu > NC Programs** window, the features on this window maintain a list of references that link tool change events to tools stored in a Tool Library file. You can use this list to specify tools to use with sequential tool changes, or to replace tools specified for use in the tool path file.



List Tool Change By — Specifies the tool change event used to build the tool list from. Select an option, and then press **Build Tool List** to have VERICUT automatically create the tool change event list. Options are:

Cutter Desc. — Changes tools when parametric cutter description records in the tool path file are encountered, for example: CUTTER and VERICUT-TC comment records.

VERICUT TC — Changes tools when VERICUT-TC statements in the toolpath file are encountered.

Pocket Num. — Changes tools when tool path records that reference tool or pocket numbers are encountered, for example: TnM6 (G-Code tool paths), LOADTL or LOAD TOOL, and TURRET.

Prog. Stop — Changes when "program stop" codes are encountered, for example: M0 (G-Code tool paths) and STOP.

Pocket Num per File — Similar to Pocket Num., except creates additional entries in the list for duplicate tool numbers that appear in different tool path files.

Tool Change Event/Tool ID list — List of tool change events expected to occur during the simulation as indicated by the active **List Tool Change By** option (see above), and IDs of the tools to use for each. Event text must match the tool change record text exactly as it appears in the NC program file. Type or left-click and choose the tool ID from an option list. Right-click to add and delete tool list entries. For the **Cutter Desc.** and **Prog. Stop** options, the list represents the sequential order of tool changes found in the tool path file(s), and tools to use for each. A **Pocket Num.** list is a cross-reference index between tool or pocket numbers referenced in the tool path file(s) to tool IDs in a VERICUT Tool Library. A **Pocket Num per File** list is similar to **Pocket Num.**, except creates additional entries in the list for duplicate tool numbers that appear in different tool path files.

The **Tool Change List** consists of the following columns of information.

Event — displays the tool change event.

Cutter ID — the ID of the tool.

Holder ID — the ID of the holder associated with the tool.

Optipath Settings — the OptiPath settings associated with the tool.

File # — the file in the Toolpath List that the tool is associated with.

Curve Fit — this column is only visible when Toolpath Type in the Toolpath window is set to G-Code Data and one or more of the NC programs in the NC Programs list has Curve Fit toggled on.

Initial Pocket — this column is related to the use of tool chains. It enables you to specify the pocket that the tool will be initially located.

When using Tool Change by List, the default initial pocket numbers are assigned as follows. The first called tool is assigned to pocket 1, regardless of the tool number. The second called tool is assigned to pocket 2, and so on. You can then change the pocket number for each tool as necessary. When you do a Reset, the tool will display in the assigned pocket. This feature is useful when using large tools that could interfere with each other if located in adjacent pockets. Change the pocket location to ensure that all the tool changes are collision-free. Another use would be to optimize the tool change time for very short operations by minimizing the tool selection time.

Add — Adds a new tool change event to the tool change event list. Event data can be entered, or in the case of tool component IDs and OptiPath settings-can be selected from option lists.

Delete — Deletes the selected tool change event from the tool change event list.

Clear — Clears all tool change events from the list.

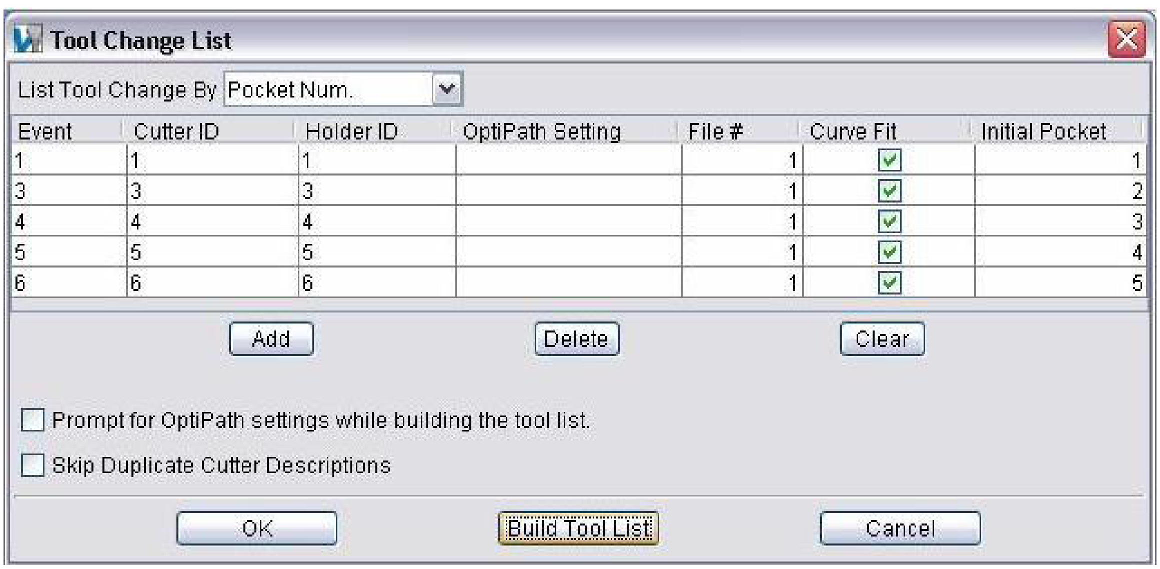
NOTE: Always reset VERICUT (press  (**Reset Model**) on the VERICUT main window) after making changes.

Prompt for OptiPath Settings while building the tool list — When active, causes VERICUT to prompt for OptiPath settings during interactive tool path optimization. At each tool change event an Optimization Settings window opens. Configure this window as desired to optimize the cutting sequence, or choose "**Do not optimize this tool**". An OptiPath Library file is created as you go containing the optimization settings defined for each optimized tool.

For more information about optimizing tool paths, see "**About optimizing tool paths**" in the *Using VERICUT* section. *Using VERICUT* can be found in the *CGTech Help Library*.

Skip Duplicate Cutter Descriptions — When active, causes VERICUT to skip duplicate cutter descriptions when building the tool change event list. This feature is useful for suppressing redundant tool changes often output by CAM systems for different cutting operations by the same cutter.

Build Tool List — Builds a tool list by scanning the files in the tool path list and looking for the tool change events specified by the **List Tool Change By** feature (see above). When multiple tool path files are listed, a comprehensive tool change events list is created with all tool change activity from all listed tool paths. Sample output is shown in the following image.



Tools (Tool Manager window)

VERICUT Users:

VERICUT Location: **Project menu > Tools**

Toolbar shortcut to access the Tool Manager: 

Mold and Die Users:

Mold and Die Location: **Setup page > Work Directory**

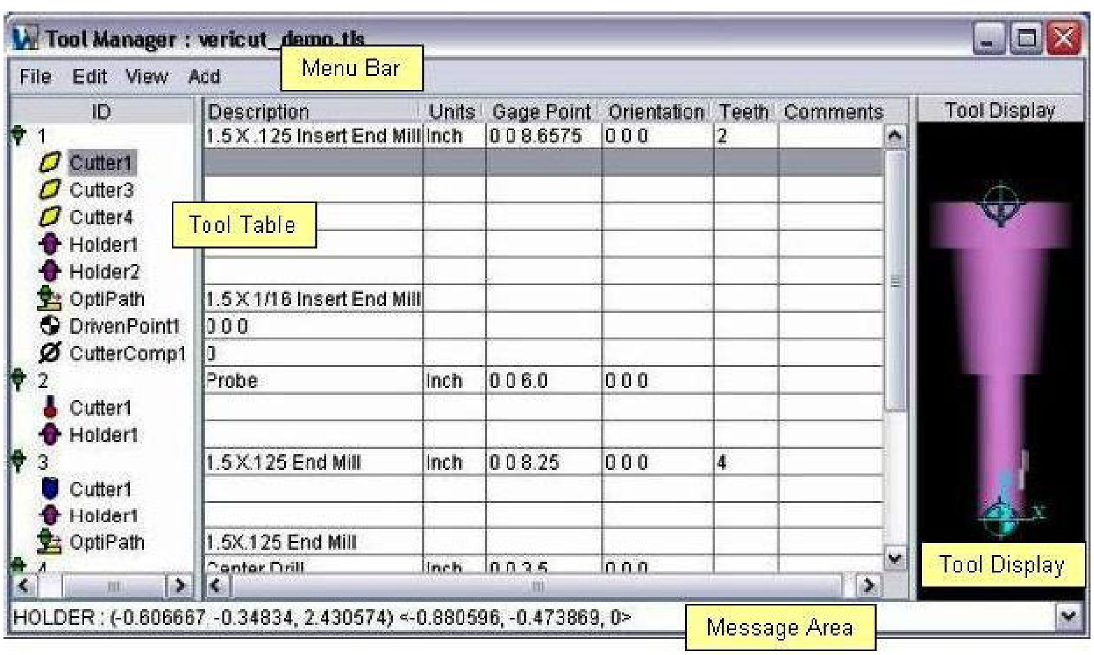
Notebook Feature: 

Cutter Grinder Users:

Cutter Grinder Location: **Setup page > Create Wheels**

Notebook Feature: 

Opens a window to create and maintain VERICUT Tool Library files containing descriptions of cutting tools, or tool assemblies. Library tools are used by VERICUT when the **Project menu > NC Program** function is configured to do so.



In general, double-click on a tool component to modify, or key in supporting values into the associated data fields. Right-click to access shortcut menus for defining and modifying tools. Tools can be defined interactively in VERICUT, or outside of VERICUT by running the Tool Manager "toolman" command file.

The Tool Manager main window is composed of 4 distinct areas (Menu Bar, Tool Table, Tool Display, and Message area), each with different user interaction. Each of these areas is discussed in detail in the topics that follow.

The window header displays the current tool library file. This window can be resized like most other window, via dragging the window header, sides or corners. These areas are discussed in detail in the following topics.

Tips:

1. **Right-click shortcut menus**: Features used to manipulate tools, or tool assemblies, are displayed in a shortcut menu when the right mouse button is clicked in the Tool Table portion of Tool Manager window.

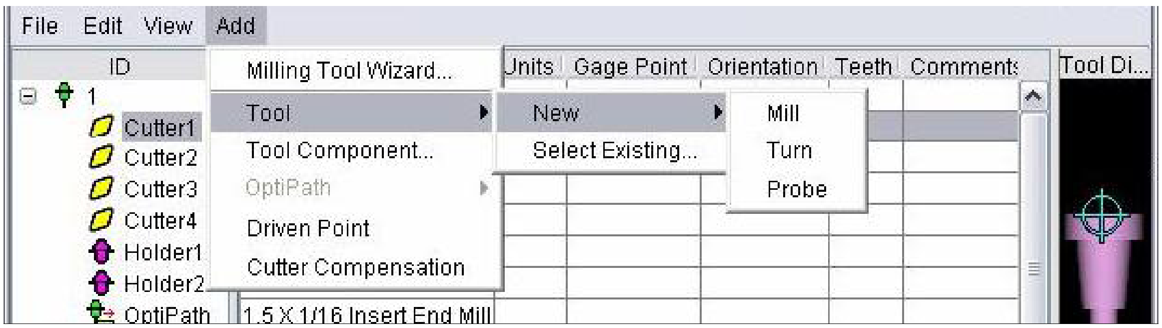
   Features used to manipulate the tool graphics are displayed in a shortcut menu when the right mouse button is clicked in the Tool Display area of Tool Manager window.

2. You can move objects in the Tool Manager window via dragging them from one parent tool to another, or copy them via holding down the <**Ctrl**> key and dragging.

Also see "**Describing Cutting Tools**", in the *Using VERICUT* section, in the *CGTech Help Library*.

Tool Manager window, Menu Bar

The menu bar, located across the top of the Tool Manager window, provides easy access to Tool Manager functions. Each menu item contains groups of related functions. Left-click on any menu name to display the list of functions available for that menu item. If an arrow appears to the right of a function in a menu, move the mouse over the arrow to display a sub-menu of additional functions. Click on the function, in the menu, that you want to use. An expanded function list, for the Add menu is shown below.



File menu

Features in this menu are used to create new, open, or save tool libraries, generate Tool Manager Report files, and close the Tool Manager window.

New — Use to create a new Tool Library file.

Open — Use to open an existing Tool Library file.

Merge — Opens the **Merge Tool Library windo**w enabling you to merge two tool libraries into one.

Save — Use to save the current Tool Library file.

Save As — Use to save the current Tool Library file with a new name/location.

Save Selected Tool — Use to save the selected tool to a specified tool library file. Selecting **Save Selected Tool** displays a file selection window enabling you to specify a */path/filename* for the tool library file.

Import —

OptiPath Library — Opens the **Open OptiPath library file window**, enabling you to import an existing OptiPath Library file.

DXF Tool — Opens the **DXF Tool Geometry window** enabling you to import two dimensional tool geometry from files which comply with a de facto CAD system standard, the Data eXchange Format (DXF).

Export Selected Tool — Use to export highlighted tool in the specified format (**STL** or **VRML**). Displays a file selection window enabling you to specify a /*path*/*filename* for the file.

Create from APT NC Program — Causes the Tool Manager to scan the tool path files for cutter statements (including CUTTER, PPRINT-VERICUT TC, etc), and create a tool library file named <user filename>.tls. The APT tool path tools replace the current tool library contents. Tools are numbered sequentially, beginning at 1.

Create Report — Use to create and view a tool manager report.

Choose one of the following report formats:

Text — Create an inspection sequence report in text format.

HTML — Create an inspection sequence report in HTML format.

PDF — Create an inspection sequence report in PDF format.

See **Introduction to VERICUT Reports** , in the Getting started section of *VERICUT Help* for information on report template files supplied with VERICUT.

See **Create a Report Using a Template**, in the *Using VERICUT* section, in the *CGTech Help Library* for additional information.

Report Template — Displays the Report Template window enabling you to select and/or modify an existing tool manager report template or create a new one.

User-Defined Tag Values — Opens the User-Defined Tag Values window enabling you to assign/edit user-defined tag values used by the current tool manager report template.

Close — Closes the Tool Manager window.

Edit menu

Features in this menu are used to edit tools in the library and their cutter/holder components. (Functions are also available via right-clicking on a tool component.)

Turret Setup — Opens the **Turret Setup window** to enable you to load or change tools, or change tool positions in a turret.

Modify — Opens the **Tool Add/Modify window** enabling you to modify the selected tool component (cutter, insert, probe tip, holder). The features displayed on the window will vary depending on the selected tool component. This option is not available when a tool is highlighted.

Rename — Enables you to rename the highlighted tool/tool component.

Delete — Deletes the selected tool/tool component.

Cut / Copy / Paste — Enables you to cut, copy or paste the selected tool/tool component within the same tool library file.

Search Tool — Displays the **Search Tool window**.

Search OptiPath — Displays the **Search OptiPath Record window**.

Stack — Use to automatically stack tool components on top of each other.

Automatic Gage Offset (Z) — When toggled "On" (checkmark is visible), VERICUT calculates the highest point on the Z-axis of the tool and uses it for the Gage Point whenever a tool component is modified. This feature only applies to milling tools. The state (On or Off) of **Automatic Gage Offset (Z)** is stored in the VERICUT Preferences file.

View menu

Features in this menu provide control over viewing tools in the display area, such as: view orientation, lighting, and symbols used to indicate driven point, gage point, etc.

Orient — Opens the **View Orient window** enabling you to orient the view of a tool. Features include: rotate, zoom, pan, reverse, etc.

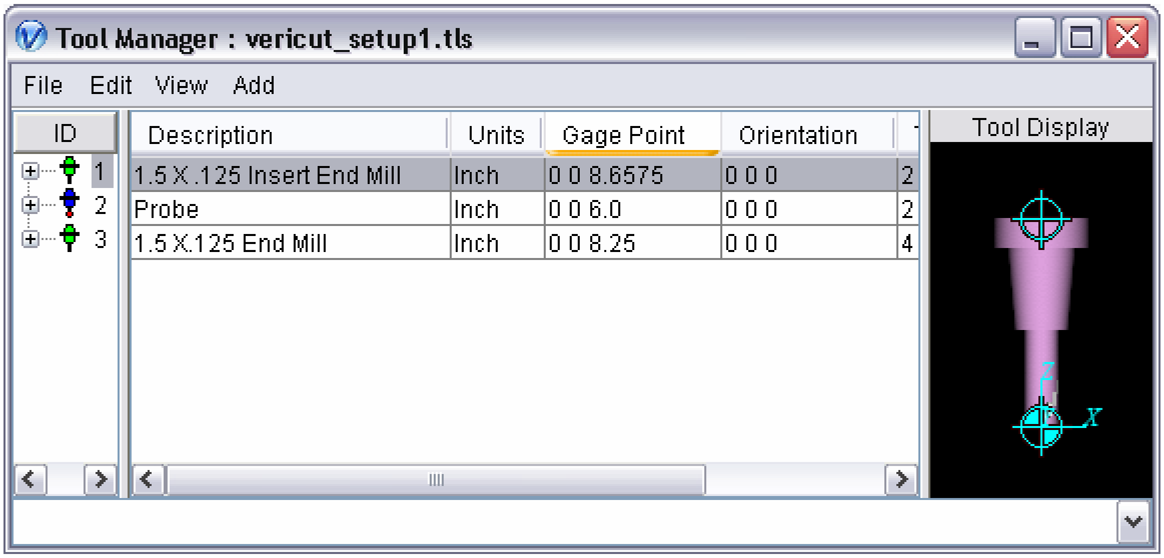
Attributes — Opens the **View Attributes window**, enabling you to display/not display **Driven Point**, **Gage Point**, and **Axes** symbols, as well as define a light source direction vector for the Tool Display graphics window.

Expand All — Expands all tools in the ID list displaying the tool components associated with each tool.

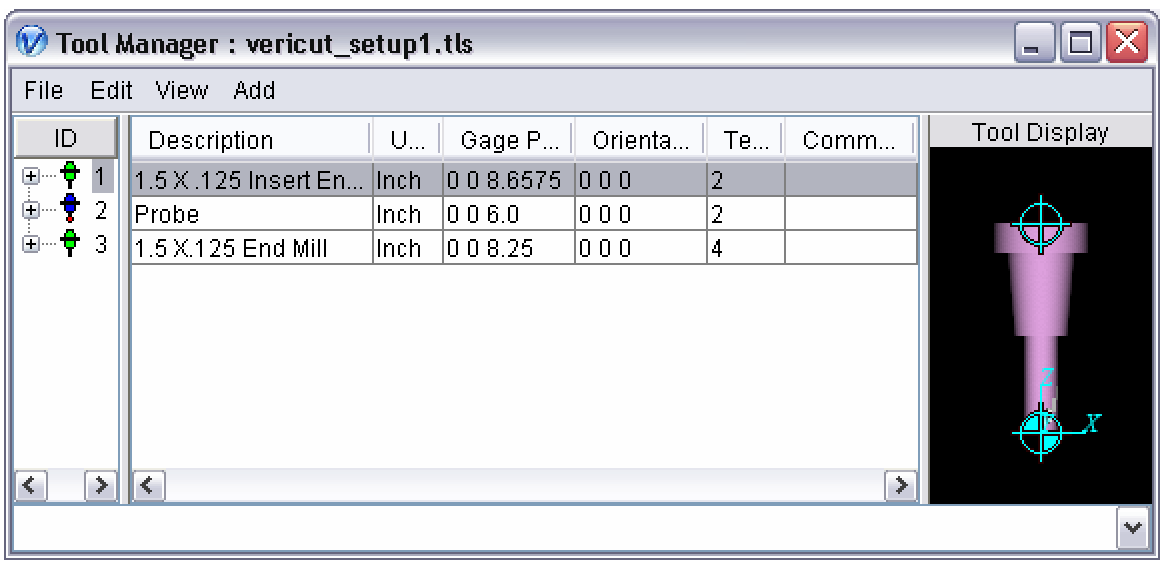
Hide Tool Components — Closes all tools in the ID list displaying only the tools in the list but none of the tool components.

Allocate Width to Columns — Adjusts the column with so that all of the columns are visible.

In the following picture the **Comments** column and most of the **Teeth** column are hidden behind the **Tool Display** area.



Selecting **Allocate Width to Columns** causes VERICUT to adjust column width so that at least some of each column is visible in the **Tool Table** area as shown in the next picture.



Now that all columns are visible, you can easily adjust the size of the **Tool Manager** window, and column widths, as needed.

Add menu

The features in this menu are used to add a new tool, or to add cutter, insert or holder tool components to the selected tool. The features available will vary depending on what is selected in the Tool table. These features are also available via right-clicking on a tool and selecting from the shortcut menu.

Milling Tool Wizard — Displays the Milling Tool Wizard window to assist you in defining Milling tools.

Tool —

New — Adds a new tool to the library. Select the tool type (**Mill**, **Turn**, **Probe**, **Tap**, or **Water Jet**), then key in a unique ID to start the new tool definition.

Select Existing — Displays the **Search Tool window**.

Tool Component — Displays the **Tool Add/Modify window** enabling you to add tool components to the highlighted tool.

OptiPath —

New — Displays the **OptiPath window** enabling you to define a new OptiPath record for the highlighted tool.

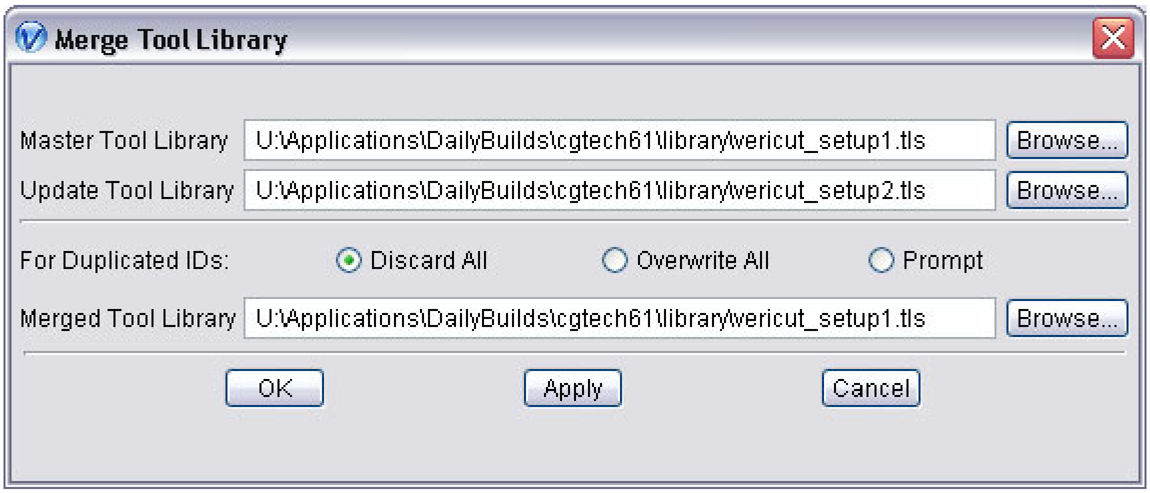
Select Existing — Displays the **Search OptiPath Record window**.

Driven Point — Use to add one or more a Driven Point records to the highlighted tool.

Cutter Compensation — Use to add one or more cutter compensation records to the highlighted tool.

Merge Tool Library window

Accessed from the **Tool Manager: File menu**, the features in the **Merge Tool Library** window enable you to quickly and easily merge two tool library files.



Master Tool Library — Enter the */path/filename,* of the master tool library file, in the **Master Tool Library** text field or use the **Browse** button to display a file selection window and use it to select the master tool library file.

Update Tool Library — Enter the */path/filename*, of the tool library file that will be merged into the master tool library file, in the **Update Tool Library** text field, or use the **Browse** button to display a file selection window and use it to select the update tool library file.

For Duplicated IDs — Use to select one of the options to specify how duplicate tool IDs are to be handled during the merge operation.

> **Discard All —** When toggled "On", duplicate tool IDs in the Update Tool Library will be discarded.
>
> **Overwrite All —** When toggled "On", duplicate tool IDs in the Master Tool Library will be overwritten by the ones in the Update Tool Library.
>
> **Prompt —** When toggled "On", a window will display when a duplicate tool ID is encountered, enabling you to specify whether to **Discard** the duplicate tool ID in the Update Tool Library**,** or **Overwrite** duplicate tool ID in the Master Tool Library, or to **Quit** the merge operation.

Merged Tool Library — Enter the */path/filename*, for the merged tool library file, in the **Merged Tool Library** text field, or use the **Browse** button to display a file selection window and use it to select the merged tool library file.

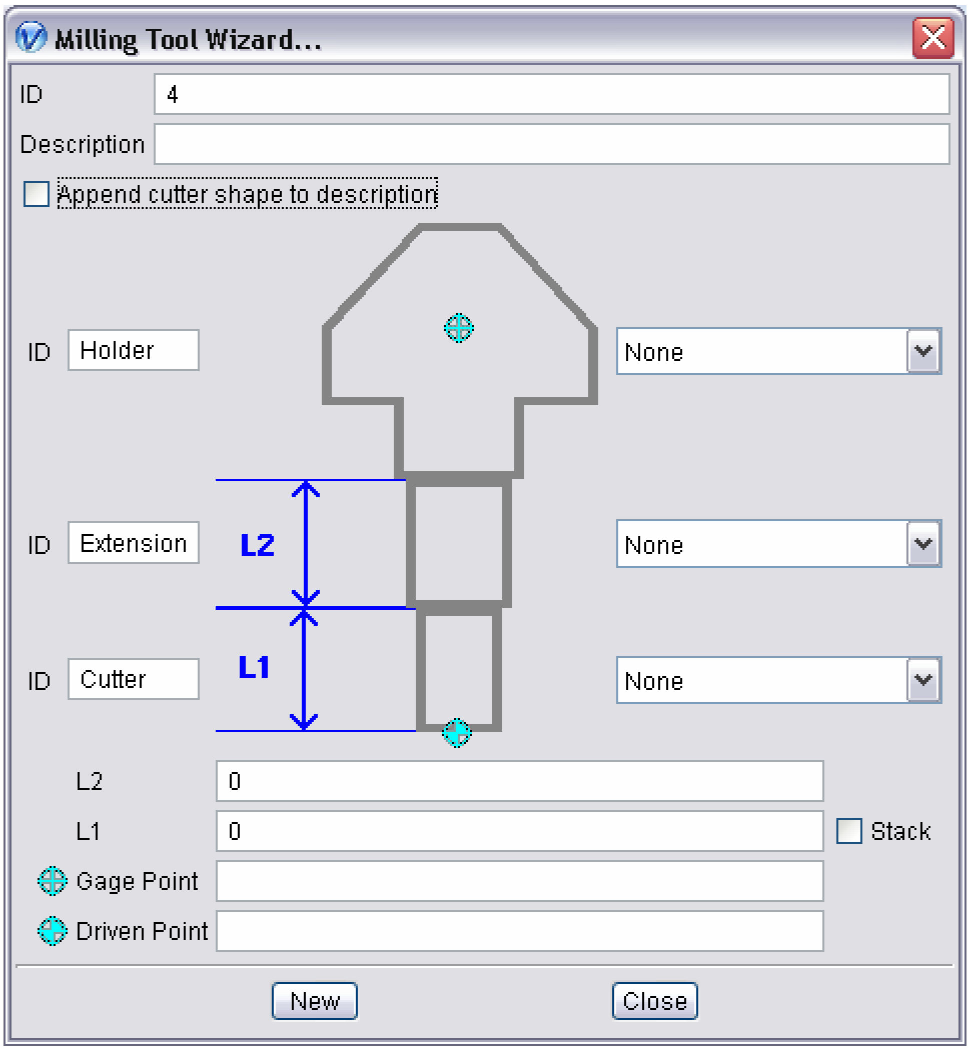
OK — Starts the merge operation and closes the **Merge Tool Library window**.

Apply — Starts the merge operation and leaves the **Merge Tool Library window** open for additional merges.

Cancel — Closes the **Merge Tool Library window** without doing the merge operation.

Milling Tool Wizard

Accessed from the **Tool Manager: Add menu**, the features in the **Milling Tool Wizard** enables you to quickly and easily define tool assemblies for your tool library.



ID (tool) — Use to key in a unique ID for the new milling tool.

Description — Use to add a description for the milling tool / tool assembly.

Append cutter shape to description — When toggled "on", adds the cutter shape description for the cutter defined below to the above description.

ID (holder) **—** Use the text field to enter an ID for the "holder" component.

Holder Type

None — Use to specify no "holder" component in the tool assembly.

New — Displays a window enabling you to add an "extension" component to the tool assembly. See **Tool Add/Modify window: Component tab, in Holder mode** for details.

Search Holder — Displays the **Search Tool window** enabling you to search for existing holders.

Tip: You can also use the CGTECH_REF_HOLDER_LIB environment variable to specify a reference Tool Library file. All holders in the file will be listed in the pull-down list. See **Environment Variables** in the *Getting Started with VERICUT* section of *VERICUT Help*.

ID (extension) **—** Use the text field to enter an ID for the "extension" component.

Extension Type

None — Use to specify no "extension" component in the tool assembly.

New — Displays a window enabling you to add an "extension" component to the tool assembly. See **Tool Add/Modify window: Component tab, in Holder mode** for details.

Search Holder — Displays the **Search Tool window** enabling you to search for existing extensions/holders.

Tip: You can also use the CGTECH_REF_EXTENSION_LIB environment variable to specify a reference Tool Library file. All extensions in the file will be listed in the pull-down list. See **Environment Variables** in the *Getting Started with VERICUT* section of *VERICUT Help*.

ID (cutter) **—** Use the text field to enter an ID for the cutter component.

Cutter Type

None — Use to specify "no" cutter component in the tool assembly.

New Revolved Cutter — Displays a window enabling you to add a revolved cutter component to the tool assembly. See **Tool Add/Modify window: Component tab, in Revolved Cutter mode** for details.

New Insert — Displays the **Milling Insert window** enabling you to describe inserts and a cutter body to be added to the tool assembly.

Search Revolved Cutter — Displays the **Search Tool window** enabling you to search for existing revolved cutters.

Search Insert — Displays the **Search Tool window** enabling you to search for existing mill insert cutters.

Tip: You can also use the CGTECH_REF_CUTTER_LIB environment variable to specify a reference Tool Library file. All holders in the file will be listed in the pull-down list. See **Environment Variables** in the *Getting Started with VERICUT* section of *VERICUT Help*.

L2 — Use the text field to enter the exposed length of the "extension" component. This distance is used to position the "holder" component along the tool's Z-axis.

L1 — Use the text field to enter the exposed length of the "cutter" component. This distance is used to position the "extension/holder" component along the tool's Z-axis.

Gage Point — Use the text field to enter the XYZ coordinates of the gage point. If only one value is entered, it will be applied to Z.

Driven Point — Use the text field to enter the XYZ coordinates of the cutter's driven point. If only one value is entered, it will be applied to Z.

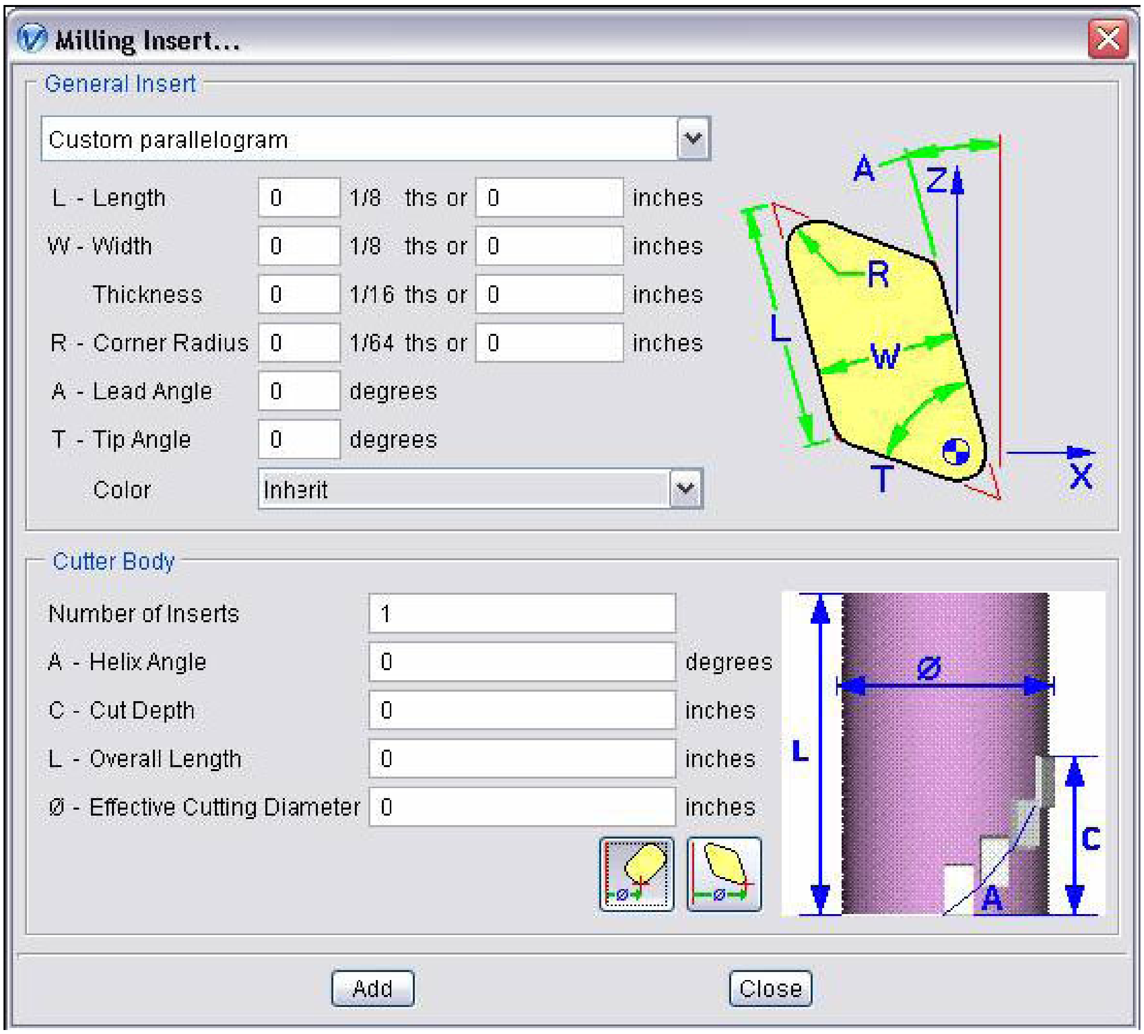
Stack — When toggled "on", automatically stacks tool components on top of each other.

New — Creates the new tool record and leaves the **Milling Tool Wizard** window open to create the next milling tool.

Close — Use to close the **Milling Tool Wizard** window.

Milling Insert window

Accessed from the **Milling Tool Wizard window**, the **Milling Insert window** enables you to quickly and easily define inserted milling tools for your tool library.



General Insert

The General Insert features enable you to easily define "ISO standard", as well as "custom" tool insert shapes. The parameters and diagram will vary depending on the Insert Type selected from the pull-down list. Only the parameters required to describe a particular insert type will be displayed. Use the diagram to identify what measurement each parameter represents.

The following insert types are available in the pull-down list:

A - 85 degree parallelogram

B - 82 degree parallelogram

C - 80 degree diamond

C - 100 degree diamond

D - 55 degree diamond

E - 75 degree diamond

H - Hexagon

K - 55 degree parallelogram

L - Rectangle

M - 86 degree diamond

O - Octagon

P - Pentagon

R - Round

S - Square

T - Triangle

V - 35 degree diamond

W - 80 degree trigon

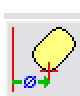
Custom diamond

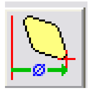
Custom parallelogram

For additional information on these insert types, see **General Insert features** in the Tool Add/Modify Window, Common Features section.

Cutter Body

The Cutter Body features enable you to easily define the cutter body and position the inserts. Enter the desired value in the text field for each parameter. Use the diagram to identify the measurement that each parameter represents. Use the **Location Aids** described below to position the inserts.

Locate by nominal diameter  **—** Positions the center of the insert corner radius on the tool diameter.

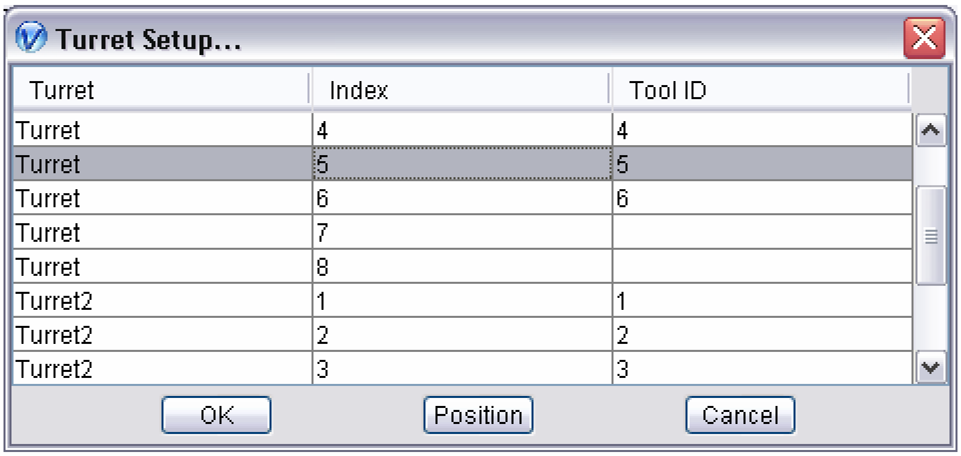
Locate by outside diameter  **—** Positions the outside edge of the insert on the tool diameter.

Add — Adds the defined inserts and cutter body to the tool library and closes the Milling Insert window.

Close — Closes the Milling Insert window without adding inserts/cutter body to the tool library.

Turret Setup window

Accessed from the **Tool Manager: Edit menu**, the features in the **Turret Setup window** enable you to load or change tools, or change tool positions in a turret.



Turret Setup Table

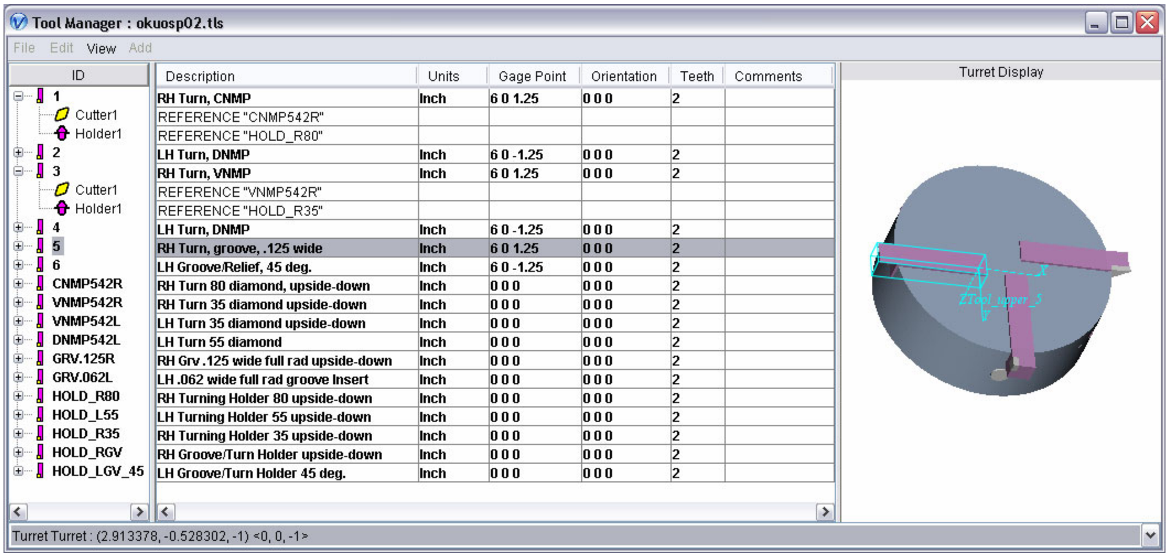
Each record in the Turret Setup Table represents a position on a turret.

Turret — Indicates the name of the turret component. Records for all turrets in the machine configuration will be displayed in the Turret Setup Table.

Index — Indicates the Index number of a turret tool position.

Tool ID — Indicates the Tool ID of the tool loaded in that position in the turret. A blank field indicates that no tool is currently loaded in the position.

When you select a record in the Turret Setup Table, the record becomes highlighted. The Tool Display area in Tool Manager updates to display the turret with all tools currently loaded in it. The tool represented by the selected record is marked in the turret display as well as becomes highlighted in the Tool Manager Tool Table. See the picture that follows.



To add, or change, a tool associated with the highlighted turret record, click in the Tool ID field for that record and select the desired tool from the pull-down list. The list will contain all tools in the current Tool Library and a blank field.

Use the **Position** button, described below, to display the tool positioning window and use it to orient the tool correctly on the turret.

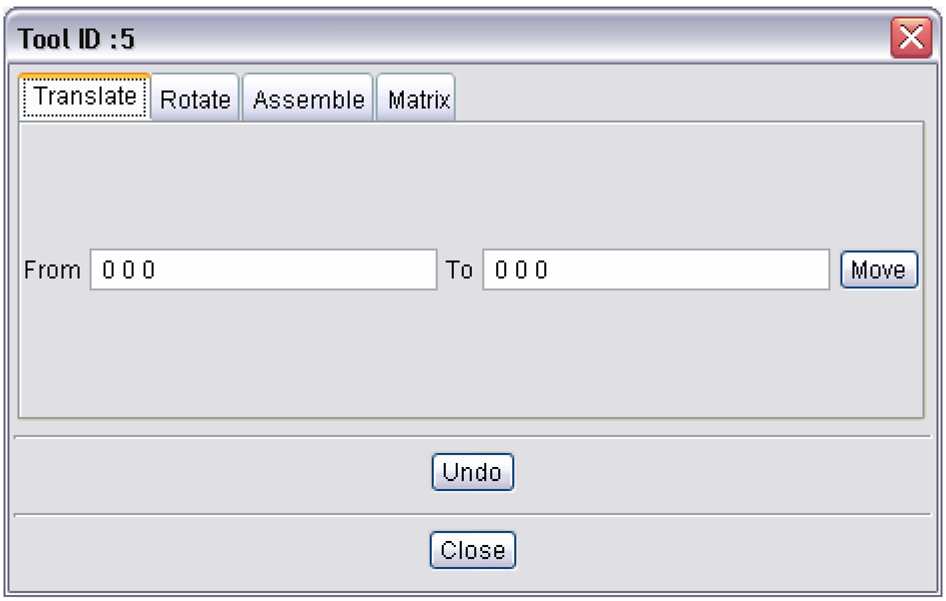
OK — Use OK to accept your changes and close the Turret Setup window.

Position — Displays the Turret Tool Positioning window enabling you to correctly position a tool on the turret.

Cancel — Closes the Turret Setup window without accepting your changes.

Turret Tool Positioning window

Accessed from the **Turret Setup window: Position button**, the features on the Turret Tool Positioning window enable you to correctly orient a tool (or tool assembly) on a turret, similar to orienting models. The features on the Turret Setup window tabs work the same as the features on the corresponding tab on the **Tool Add/Modify window: Assembly tab**.



Translate tab **—** Features on this tab are used to translate the selected tool via indicating "from" and "to" points to move the tool.

Rotate tab **—** Features on this tab are used to rotate the selected tool about a rotation center point.

Assemble tab **—** The features on this tab enable you to assemble tools on a turret, by mating, or aligning, one to three planar surfaces with surfaces on the turret, similar to assembling tool assemblies or models.

Matrix tab **—** Features on this tab move the selected tool via a twelve parameter transformation matrix.

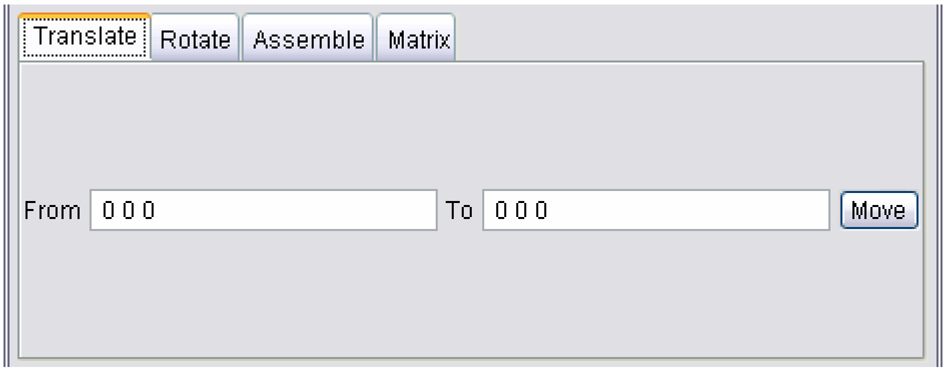
Undo — Returns the tool to its previous orientation, or as it was when the window was opened.

Close — Closes the **Turret Tool Positioning window**.

Turret Tool Positioning window, Translate tab

Features on this tab enable you to translate the selected tool (or tool assembly) via indicating "from" and "to" points to move the tool, similar to translating models. Movement occurs each time you press the **Move** button. If the applied movement is incorrect, press **Undo** (in the Turret Positioning window) to return the object to its previous location.

Features on this tab work the same as the **Translate** tab features on the **Tool Add/Modify window: Assembly tab**.



Translate tab features:

From / To — Specifies the locations to move the tool "from", and "to", relative to the tool origin. XYZ values can be entered (separated by spaces), or selected by clicking in the field then clicking on the tool in the Turret Display area. As you move the mouse over the tool, a crosshair and vector show you the pending pick-point location.

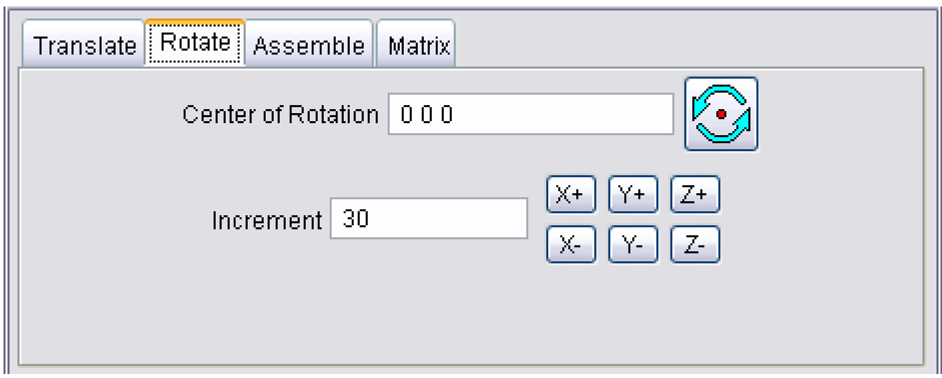
Move — Moves the selected object by the incremental distance, as calculated from the "**From**" point to the "**To**" point location.

To Turret Tool Positioning window

Turret Tool Positioning window, Rotate tab

Features on this tab are used to rotate the selected tool (or tool assembly) about a rotation center point, similar to rotating models. Movement occurs each time you press one of the rotation direction buttons: **X+**/**X-**, **Y+**/**Y-**, **Z+**/**Z-**. If the applied rotation is incorrect, press **Undo** (in the Turret Positioning window) to return the object to its previous location.

Features on this tab work the same as the **Rotate** tab features on the **Tool Add/Modify window: Assembly tab**.



Rotate tab features:

Center of Rotation — Specifies XYZ point location about which to rotate the tool. XYZ values can be entered in the **Center of Rotation** text field (separated by spaces), or selected by clicking in the **Center of Rotation** text field, then clicking on a tool in the Turret Display area. To see the center of rotation, press . To remove the center of rotation symbol press the button again, or close the window.

Increment — Specifies incremental degrees of rotation to apply when one of the rotation direction buttons are pressed.

Rotation buttons — (**X+**/**X-**, **Y+**/**Y-**, **Z+**/**Z-**) Applies the incremental degrees of rotation specified in the **Increment** field. Rotation occurs about the **Center of Rotation** relative to the tool origin.

To Turret Tool Positioning window

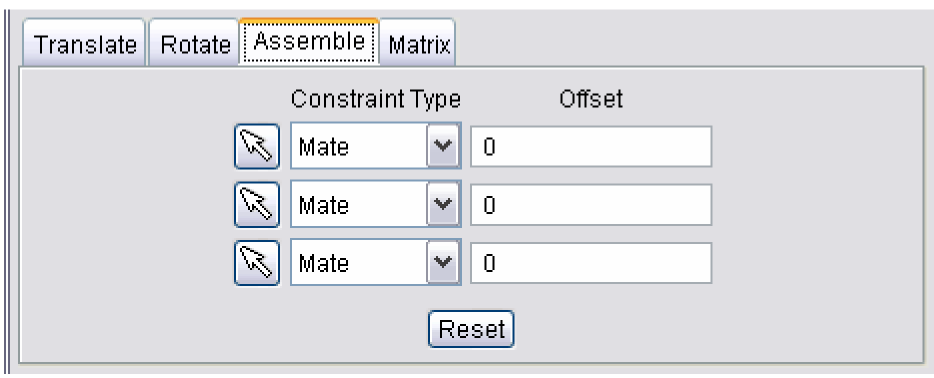
Turret Tool Positioning window, Assemble tab

The features on this tab enable you to orient a tool (or tool assembly) on a turret, by mating, or aligning, one to three planar surfaces with surfaces on the turret, similar to assembling models. If a non-planar surface is selected, VERICUT constructs a tangent plane at the pick point. The relationship of surfaces being mated or aligned is known as a "constraint". If the applied movement is incorrect, press **Undo** (in the Turret Positioning window) to return the tool to its previous location.

Features on this tab work the same as the **Assemble** tab features on the **Tool Add/Modify window: Assembly tab**.

Follow these general steps to define a constraint for assembly:

1. Choose the constraint type.
2. In the Turret Display area, select a surface on the tool to move.
3. Select the surface to move the tool relative to on the turret.



Assemble tab features:

Constraint Type — Specifies how to constrain selected surfaces during tool movement. After selecting two surfaces in the Turret Display area to define a constraint, VERICUT moves the tool and highlights the satisfied constraint with a checkmark.

Options:

Mate — Moves the tool so the selected surface opposes the surface selected on the turret (surface normals oppose each other).

Align — Moves the tool so the selected surface is aligned with the surface selected on the turret (surface normals point in the same direction).

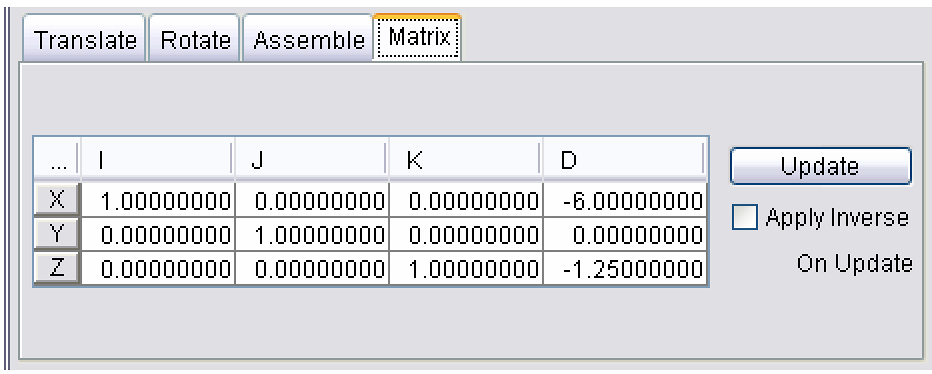
Offset — Distance and direction in which to offset constrained surfaces, applied normal to the surface.

Reset — Resets constraints to receive new data.

To Turret Tool Positioning window

Turret Tool Positioning window, Matrix tab

Features on this tab enable you to orient the selected tool (or tool assembly) via a twelve parameter transformation matrix, similar to orienting models with a matrix. If the applied movement is incorrect, press **Undo** (in the Turret Positioning window) to return the object to its previous location.
Features on this tab work the same as the **Matrix** tab features on the **Tool Add/Modify window: Assembly tab**.



Matrix tab features:

Matrix table — The transformation matrix table is similar to the matrix used in programming APT tool paths. Its twelve parameters reveal the geometric attributes of the local (transformed) coordinate system (CSYS) in terms of the machine origin.

The format of the matrix table is as follows:

	I	**J**	**K**	**D**
X	I1	J1	K1	D1
Y	I2	J2	K2	D2
Z	I3	J3	K3	D3

Each row represents an axis of the local CSYS. The first three columns represent the vector associated with each axis: I1, J1, K1 as the positive X-axis vector; I2, J2, K2 as the positive Y-axis vector; and I3, J3, K3 as the positive Z-axis vector. The fourth column values D1, D2, D3 represent the coordinates of the origin point of the local CSYS.

NOTE: If you prefer to see the Matrix Table displayed with the I, J, K along the vertical axis and the X, Y, Z along the horizontal axis, set the environment variable, **CGTECH_MATRIX_FORMAT=VERTICAL**.

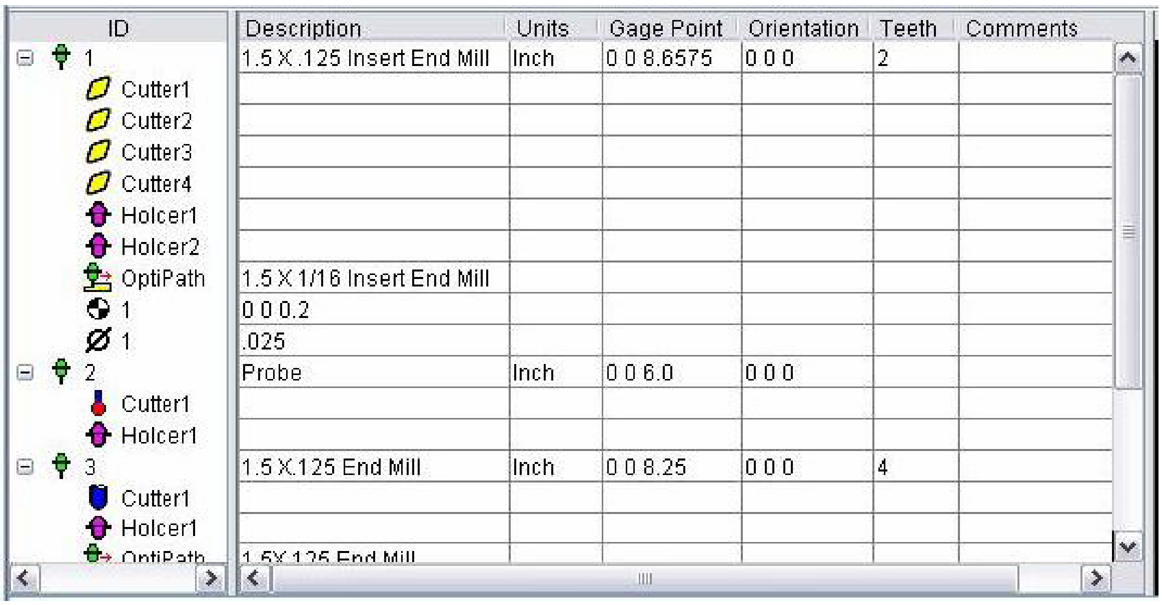
Update — Updates the tool location to reflect the matrix table transformation.

Apply Inverse On Update — When selected, inverts the matrix so that its twelve parameters reveal the geometrical attributes of the machine origin in terms of the local (transformed) coordinate system.

To Turret Tool Positioning window

Tool Manager window, Tool Table

Table of tools defined in the Tool Library file. Tools listed in the table are arranged by ID, but can be sorted by any attribute via clicking on the table header. You can also search for tools in the library using the Search feature. Double-click the tool ID or click the + / - symbols to expand or collapse branches of a tool assembly to see its cutter/holder components. Stretch the window or use the scroll bars to see all the table data.



ID	Description	Units	Gage Point	Orientation	Teeth	Comments
1	1.5 X .125 Insert End Mill	Inch	0 0 8.6575	0 0 0	2	
Cutter1						
Cutter2						
Cutter3						
Cutter4						
Holcer1						
Holcer2						
OptiPath	1.5 X 1/16 Insert End Mill					
1	0 0 0.2					
1	.025					
2	Probe	Inch	0 0 6.0	0 0 0		
Cutter1						
Holcer1						
3	1.5 X.125 End Mill	Inch	0 0 8.25	0 0 0	4	
Cutter1						
Holcer1						
OptiPath	1.5X.125 End Mill					

Tool Table Data:

ID — Alpha-numeric text that identifies the tool in the Tool Library. IDs typically correspond to tool or pocket numbers referenced by the tool change record that will call the tool from the library. For example, ID " 1" would normally be used for a tool called by a G-Code data block "T1M6", or an APT tool change record "LOADTL/1". However, you can use a tool list to cross-reference tools having IDs in a library that are different from those in the tool change record. See **Project menu > NC Programs** for details.

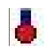
The following symbols are used in the Tool ID tree:

 **Tool**

Tool Components:

 **Insert Cutter**

 **Revolved Cutter**

 **Probe Tip**

 **Holder**

 **OptiPath Record**

 **Driven Point Record**

 **Cutter Compensation Record**

Hint: Use IDs and descriptions that make it easy for others to find tools they need.

Description — Describes the tool or tool assembly. Any alpha-numeric text can be entered in this field. Special characters are not recommended.

Units — Controls the unit measurement system (Inch or Millimeter) in which to describe the tool. If the tool units are different from the session units when loaded, then tool description values are converted appropriately.

Gage Point — Specifies the location of the tool assembly gage point. By default, the gage point is located at the tool origin. Values entered move the gage point relative to the tool origin, or you can select a gage point by clicking in the Gage Point field then clicking on the tool in the Tool Display. As you move the mouse over the tool, a point and normal vector show you the pending pick-point location.

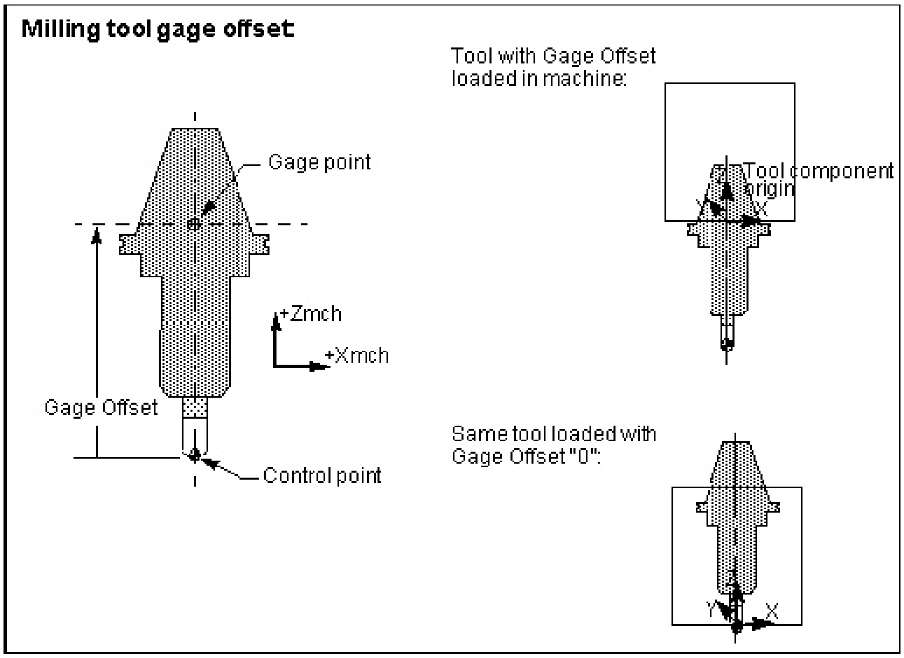
When processing G-Code tool paths, VERICUT uses the gage point in the following two ways:

Controls how the tool loads in the NC machine — When a tool is loaded for cutting, the gage point is located at the NC machine's Tool component origin.

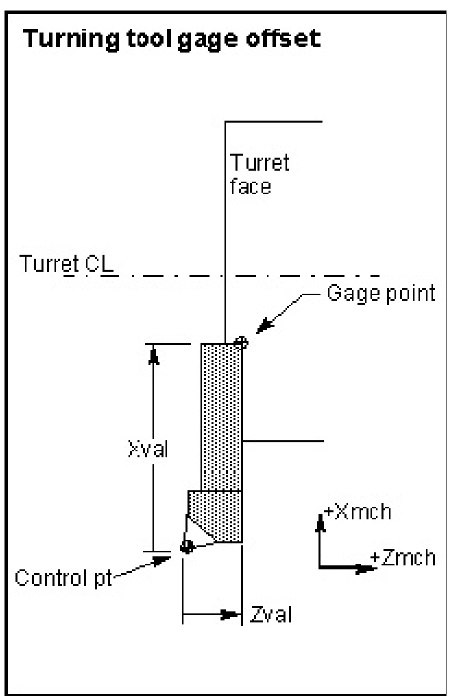
Sets the adjustment needed to drive the tool in "Tool Tip" and "Tool Length *Compensation" programming methods* — These programming methods enable an NC programmer to assume the positions in the tool path are controlling the tool, instead of NC machine axis positions. VERICUT uses the gage point to determine where the driven point is (a.k.a. "control point") relative to the gage point.

Missing or incorrect gage point values causes a tool to load incorrectly on a 3-D machine, or causes improper cutting.

Sample gage points for milling tools:



Sample gage points for turning tools:



The "Gage Point" view attribute displays a hollow target symbol , at the gage point location (ref. **Tool Manager: View menu > Attributes**).

NOTE: Gage Offset table values can be used to override Tool Manager-defined Gage Offset values for setting the adjustment to calculate the driven control point (condition #2 above). However, they do not affect how the tool is loaded in the NC machine (condition #1 above).

When the Gage Offset table is used to adjust tool motions, the NC control must be configured to call the **ToolOffsetAptAdj** macro when a tool change occurs, or when the tool control point changes, to adjust the motions seen in Workpiece views. The **ToolOffsetAptAdj** macro allows the machine simulation to be driven with multiple tool control points, when a single tool control point was defined in the Tool Manager.

If this macro is not called on or after the tool change command, or when the tool control point changes, then Workpiece views may not reflect the adjustment to the Tool Manager-defined Gage Offset.

Orientation — Specifies the orientation of the tool assembly when loaded for cutting. Three values are entered (separated by spaces) to define the X, Y, Z rotation angles, respectively. Angle values are in degrees, relative to the tool origin. Rotation occurs about the tool's gage point.

NOTE: This feature is only applicable when **NC Program Type** is **G-Code Data**.

Teeth — Automatically indicates the number of teeth of the tool referenced by the Optipath record selected from the OP Description pull-down list.

NOTE: You have the ability to manually edit the number of teeth (either by typing in a value or selecting a value from the pull-down list) but be aware that if the number of teeth entered does not match the number of teeth specified in the OptiPath record, the tool will not be optimized.

Comments — This field enables you to add comments related to the tool. The comment text is stored with the tool ID.

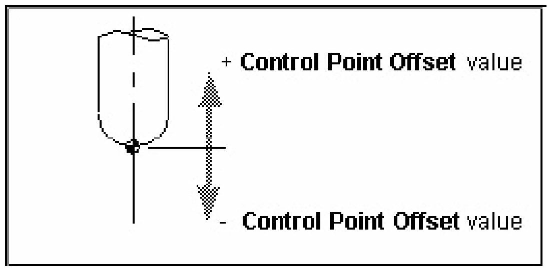
Driven Point record — Specifies an offset for the tool control point, or "driven point". Material is removed based on motion commands and the tool shape relative to the control point. For milling tools, a positive value moves the control point in the positive tool axis direction (closer to the spindle), while a negative value moves the control point in the opposite direction.

NOTE: VERICUT material removal for milling tools does not support driven points with X- or Y- offsets, only Z-offset is allowed. If you wish to drive the machine with an X, or Y, offset driven point, and to have the correct material removal, you should define

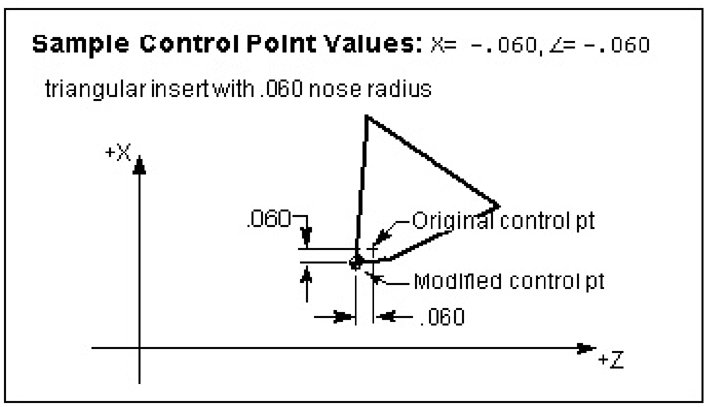
2 driven points for the tool. For example, to drive the machine with an X offset, create the 1st driven point (0,0,0) and the 2nd (6,0,0).

For turning tools, XYZ values (separated by spaces) are entered relative to the spindle axis (Z), and cross-slide axis (X).

Sample Driven Point values for a milling tool:



Sample Driven Point values for a turning tool:

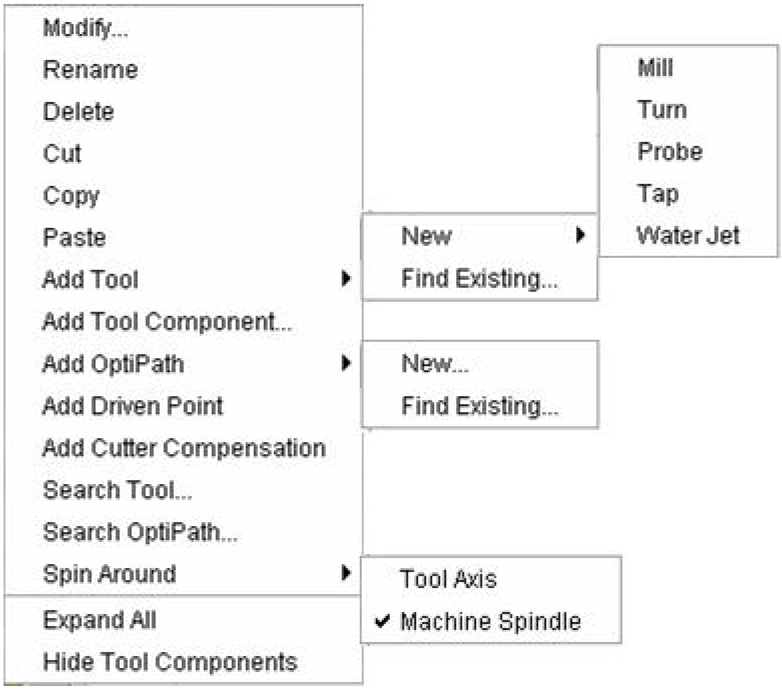


When defined, the "Driven Point" view attribute displays as a hollow target symbol  at the control point location. (Ref. **Tool Manager: View menu** > **Attributes**)

Cutter Compensation record — Specifies the cutter compensation value for a particular tool.

Shortcuts:

1. Double click a tool component in the Tool Table to display the **Tool Add/Modify window** to enable modification of the tool.
2. Right-click in the **Tool Manager Tool Table** to display a menu with the following features:

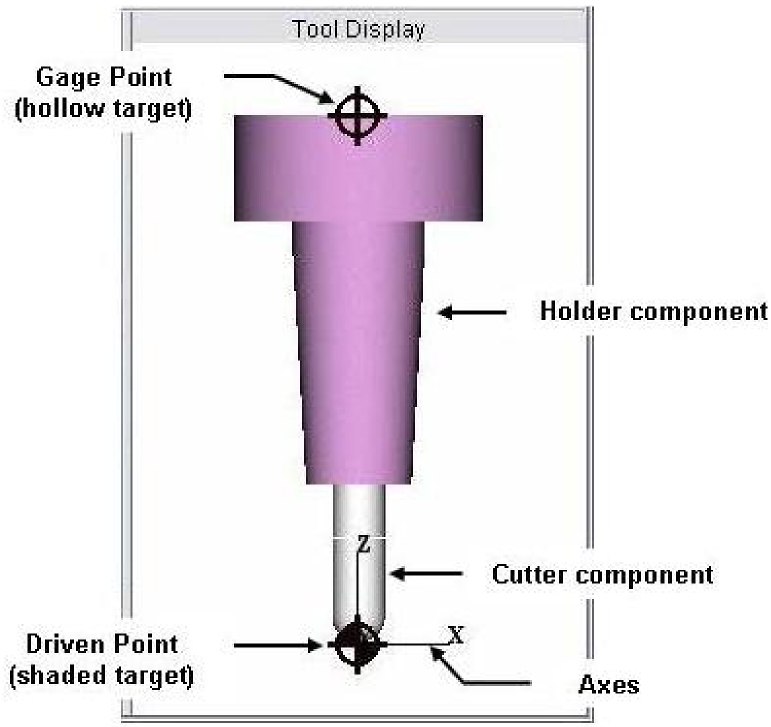


NOTES:

1. Features **Modify**, **Rename**, **Delete**, **Cut**, **Copy**, **Paste**, **Search Tool**, and **Search OptiPath** provide the same functionality described above under Tool Manager window, Edit menu.
2. Features **Add Tool**, **Add Tool Component**, **Add OptiPath**, **Add Driven Point**, and **Add Cutter Compensation** provide the same functionality described above under Tool Manager window, Add menu.
3. The **Spin Around** feature is only available for inserted tools. It enables you to specify whether to spin the tool around the Tool Axis (for example a right angle holder), or around the Machine Spindle axis. Features **Expand All** and **Hide Tool Components** provide the same functionality as described under Tool Manager window, View Menu.

Tool Manager window, Tool Display

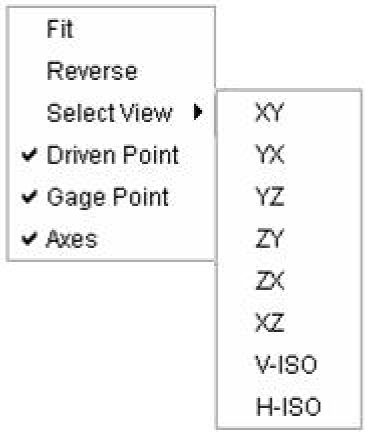
Tool Display area — Displays the tool selected in the Tool Table. Colors distinguish holders from cutter components. Target symbols indicate the tool's driven point and gage point. Use the **View Orient** functions to visually manipulate the tool in the display area.



Shortcuts:

1. Double click a tool component in the **Tool Display area** to display the Tool Add/Modify window to enable modification of the tool.

2. Right-click in the **Tool Manager Tool Display area** to display a menu with the following features:



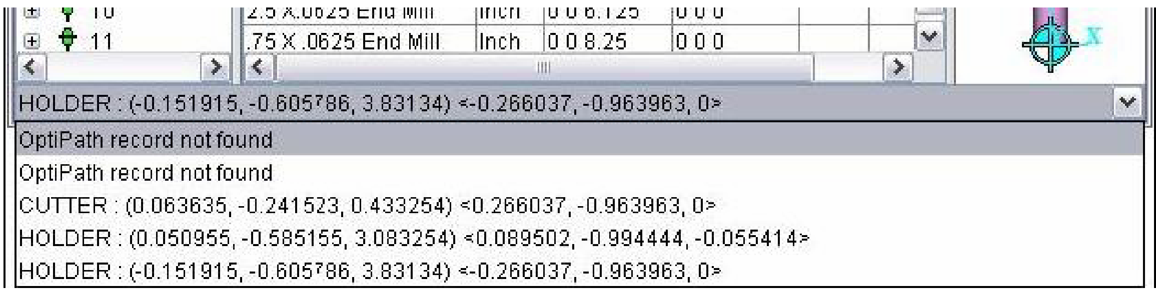
NOTES:

1. See **View Orient window**, also in the VERICUT Help section for information on **Fit**, **Reverse** and **Select View**.
2. A check next to **Driven Point**, **Gage Point**, or **Axes**, indicates that the feature is toggled "On" and the corresponding target marker, or axis, will be displayed in the Tool Display area.

Tool Manager window, Message Area

The message area, also known as the "message logger", is located at the bottom of the Tool Manager window, and by default displays the most recent message VERICUT has provided. VERICUT provides error, warning, and informational messages to assist you while working in the Tool Manager.

To review previous messages, click on the message line-the most recent message is listed last. Click again to close the message logger.



Tool Manager - Tool Add/Modify window

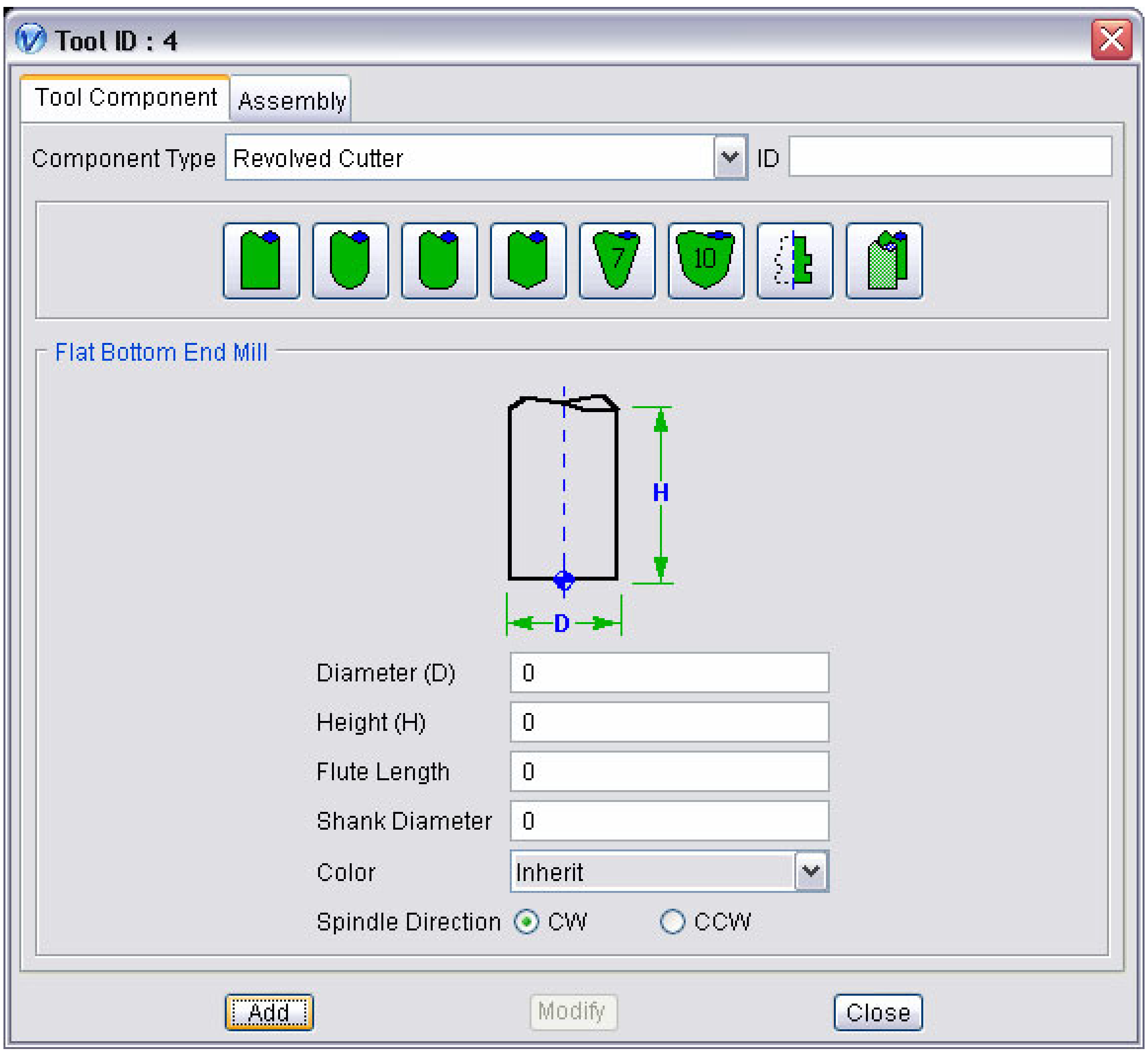
Location:

Project menu > Tools: **Add menu > Tool > New > Mill** (or **Turn**, or **Probe**), or

Add menu > Tool Component

Edit menu > Modify

Opened via adding or modifying tool/tool components, this window is used to describe tool component shape, color, location, and other attributes. Features displayed on the Components tab of this window will vary, depending on the tool/tool component type that you selected to add/modify. The Tool Component tab for a Revolved Cutter component type is shown below.



Tool Component tab **—** Features on this tab are used to define the shape of a tool component in a tool assembly. Features displayed on the Tool Component tab vary, depending on the tool/tool component type that you selected to add/modify.

Assembly tab **—** Features on this tab are used translate or rotate a component in a tool assembly, or to move the selected component by assembling (mating or aligning) it with other objects in the tool assembly.

Add — Use to apply the specified data to the new tool component.

Modify — Use to apply the modified data to an existing tool component.

Close — Close the Tool Add/Modify window without applying the data.

Also see: Tool Add/Modify window, Common Features

Tool Add/Modify window, Tool Component tab

Accessed from the Tool Add/Modify window, the features on this tab are used to define the shape of a tool component in a tool assembly. Features displayed on the Tool Component tab vary, depending on the tool/tool component type that you selected to add/modify.

Revolved Cutter
Mill Insert
Turn Insert
Probe Tip
Tap
Water Jet
Holder

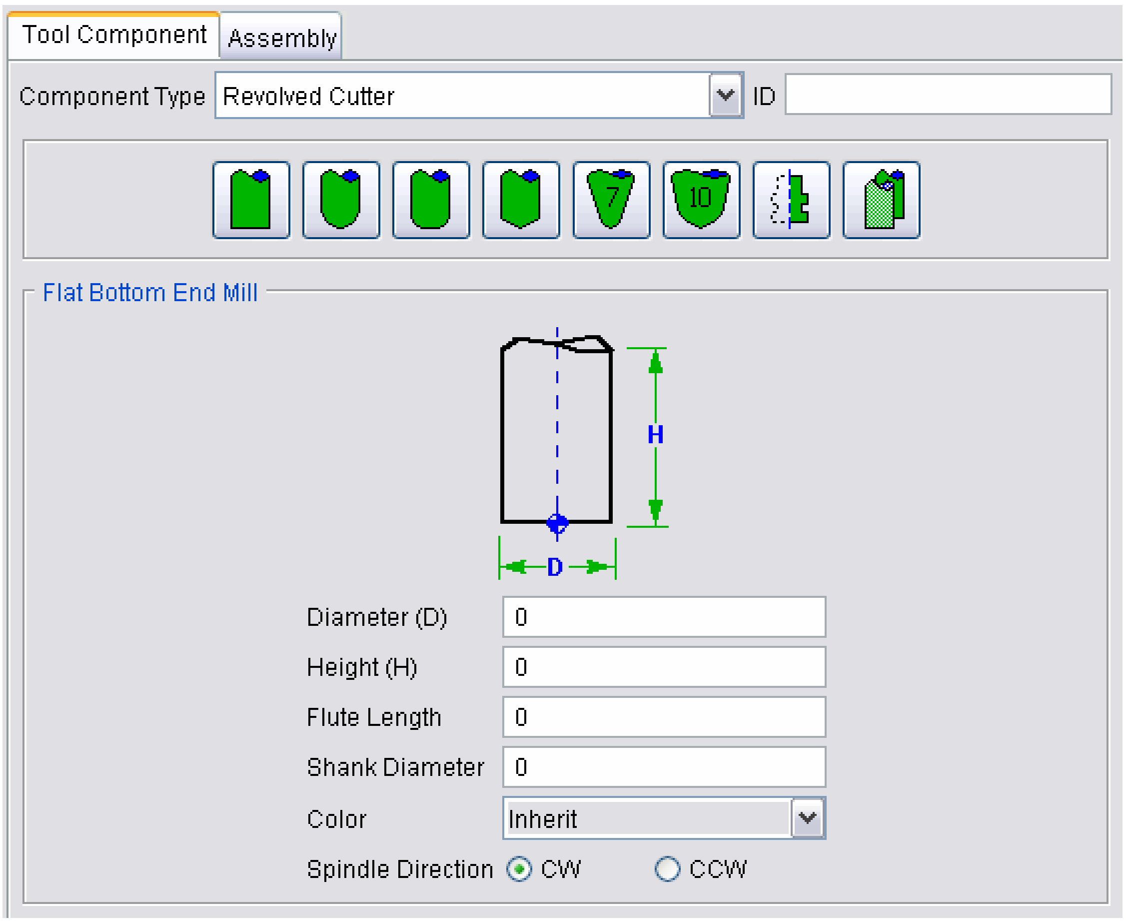
Also see: Tool Add/Modify window, Common Features

To Tool Add/Modify window

Tool Add/Modify window, Tool Component tab (Revolved Cutter)

Opened via adding, or modifying, a Revolved Cutter tool component, this tab is used to describe the shape of a selected "Cutter" component in a mill tool assembly. Options are available to define the cutter via parametric shapes, profile sketcher, or reference a cutter in another tool assembly.

See Tool Add/Modify window, Assembly tab, also in this section, for information on positioning, and orienting, a revolved cutter component in a milling tool assembly.

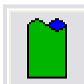
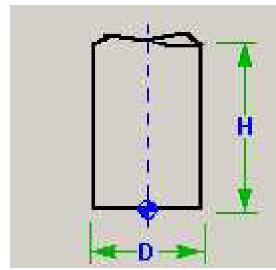
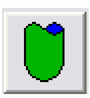
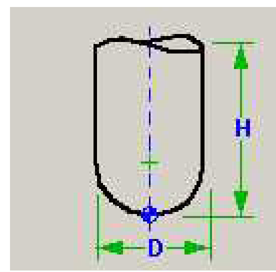
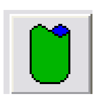
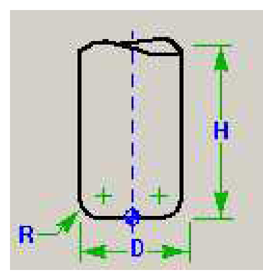
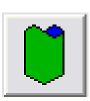
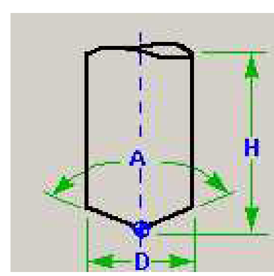


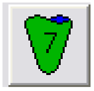
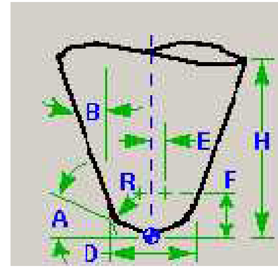
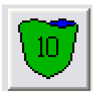
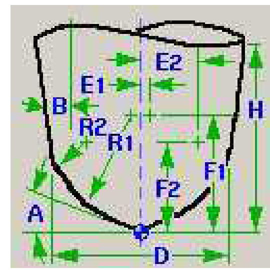
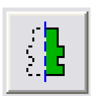
Component Type — Use to specify the type of component you are creating. In this case a Revolved Cutter.

ID — Use to specify the ID for the component you are creating. If not specified, VERICUT will assign a unique default ID.

Tool shape icons — Selecting an icon configures the bottom half of the window to define the selected cutter shape.

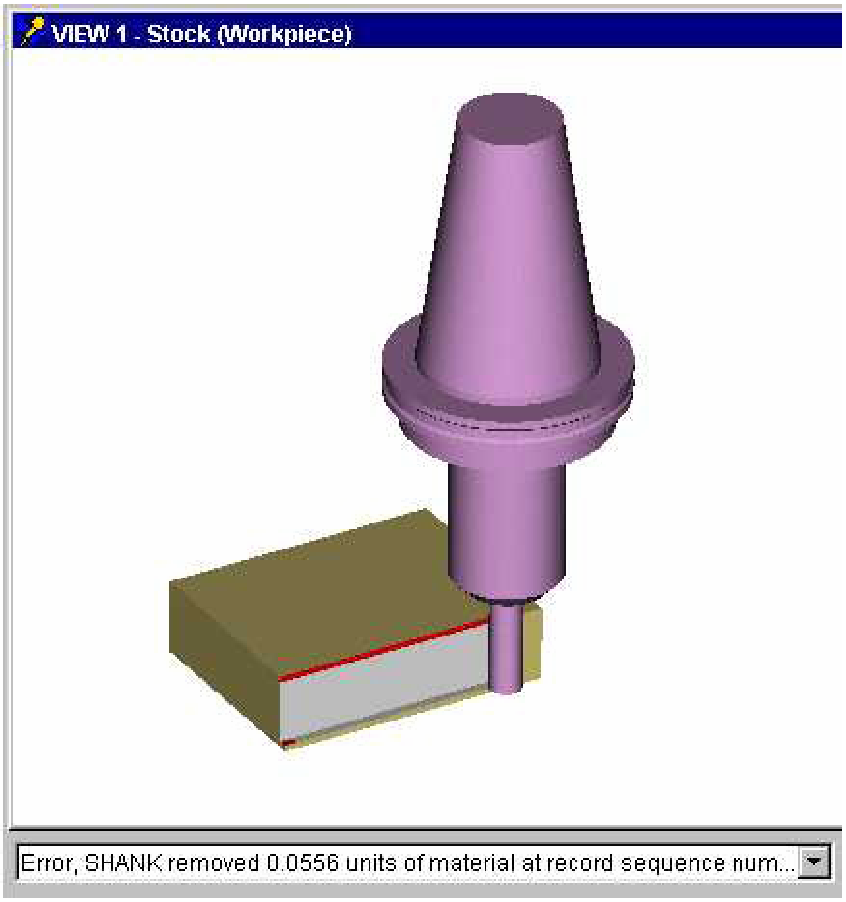
Options:

Icons	Name	Description	
	Flat Bottom End Mill	Diameter (D) Height (H)	
	Ball Nose End Mill	Diameter (D) Height (H)	
	Bull Nose End Mill	Diameter (D), Corner Radius (R), Height (H)	
	Drill	Diameter (D), Drill Point Angle (A), Height (H)	

	7 Parameter	D, R, E, F, A, B, H	
	10 Parameter	D, R1, E1, F1, A, B, H, R2, E2, F2	
	Profile	Opens the Profile Sketcher in "SOR" mode.	
	Reference	Displays the Reference features enabling you to reference a mill cutter in another tool assembly.	

The following options are common to all Tool Add/Modify mill cutter windows except Reference:

Flute Length — Length of the cutter having flutes, or teeth that can remove material. This value is measured from the bottom-most portion of the cutter. Zero assumes the entire cutter length has flutes, or teeth that can cut material. An error similar to "SHANK removed material..." is reported when the non-cutting portion of the cutter removes material. Material removed is shaded using the red Error color.



Shank Diameter — Diameter of the tool shank.

Color — Color that the tool component is to be displayed in VERICUT. For additional information, see Tool Display Colors in the **Tool Add/Modify window, Common Features** section.

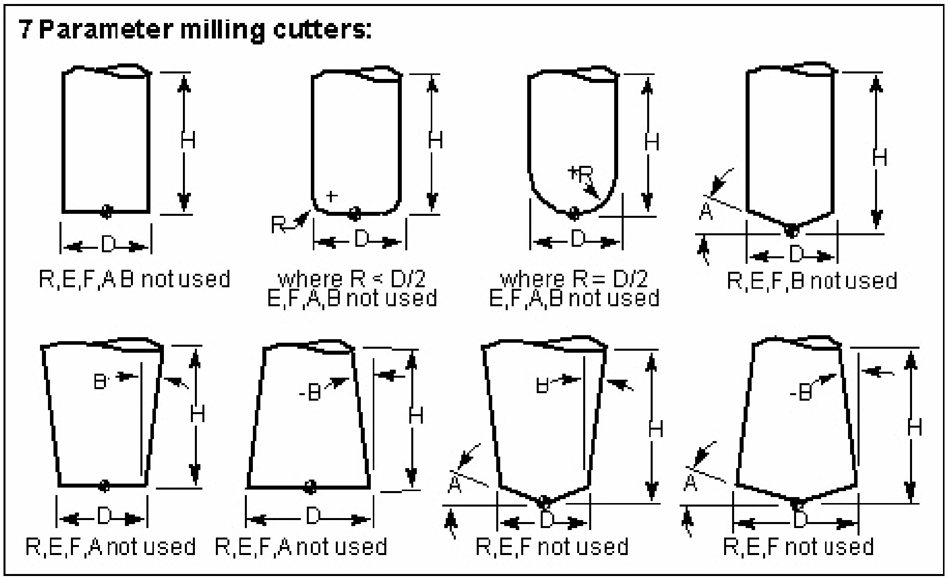
Spindle Direction — Use to specify the appropriate spindle rotation direction for the tool. Choose either **CW** (clockwise) or **CCW** (counterclockwise).

NOTE: For revolved milling cutters, **CW**/**CCW** is defined looking down (negative Z) the tool axis.

Tips for defining milling cutters:

1. Diameter is measured from the intersection points of the cutter side with the bottom flat or angle.
2. D is automatically calculated when D=0, and the corner radius (R) and side angle (B) are specified. (Good tip for defining tapered endmills.)
3. E and F are automatically calculated when diameter and corner radius are specified.

Examples of milling cutter shapes that can be defined via the 7 Parameter option:

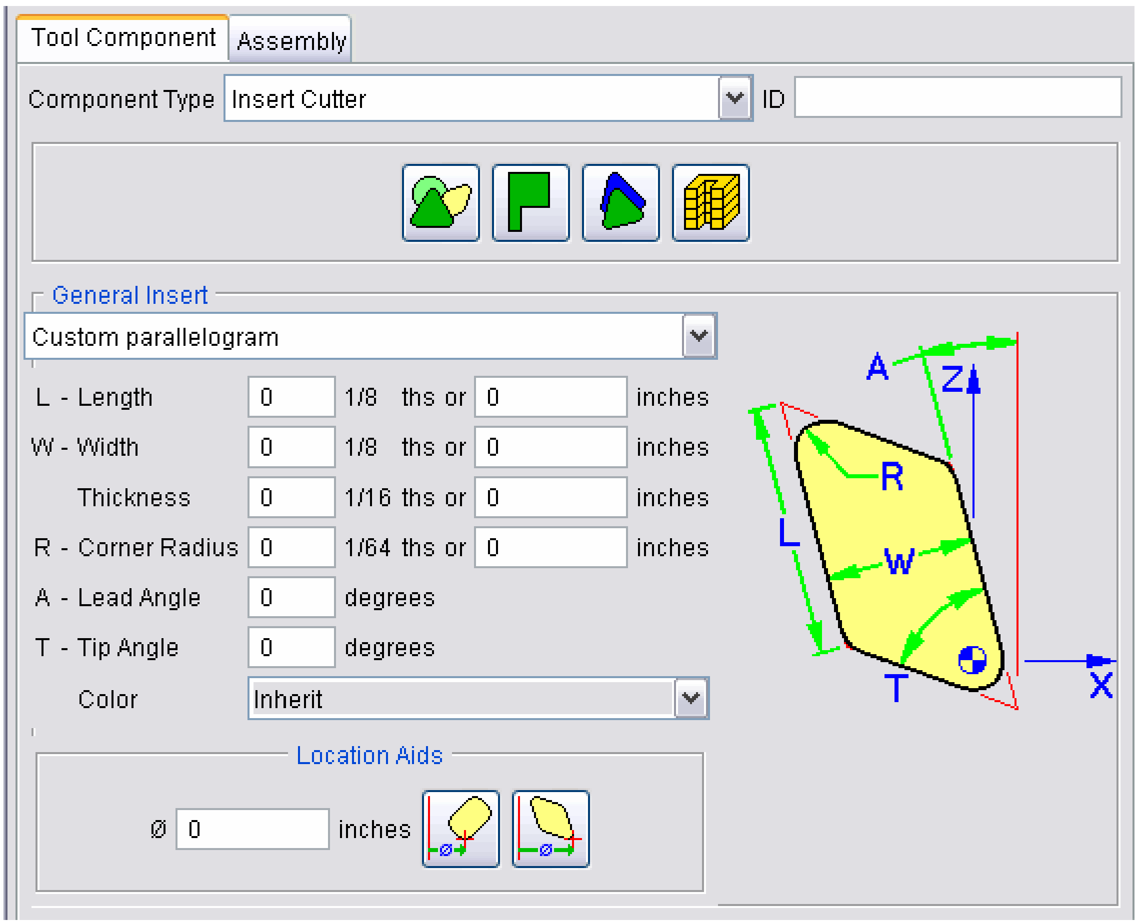


[To Tool Add/Modify window]

Tool Add/Modify window, Tool Component tab (Mill Insert)

Opened via adding or modifying a milling Insert Cutter tool component, this tab is used to describe the insert shape for the "Cutter" component in a mill tool assembly. Options are available to define the cutter insert via ISO standard shapes, profile sketcher, or reference a cutter insert in another tool assembly.

See Tool Add/Modify window, Assembly tab, also in this section, for information on positioning, and orienting, a mill insert component in a milling tool assembly.

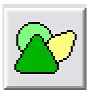
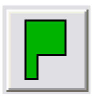
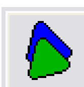
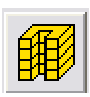


Component Type — Use to specify the type of component you are creating. In this case an Insert Cutter for a milling tool.

ID — Use to specify the ID for the component you are creating. If not specified, VERICUT will assign a unique default ID.

Tool shape icons — Selecting an icon configures the bottom half of the window to define the selected cutter shape.

Options:

Icons	Name	Description
	General Insert	Displays the General Insert features enabling you to define ISO standard insert shapes.
	Profile (sweep)	Opens the Profile Sketcher in "sweep" mode.
	Reference	Displays the Reference features enabling you to reference an insert in another tool assembly.
	Model File	Displays the Model File features enabling you to specify a model file to be used as an insert.

NOTES:

1. For inserted milling cutters, tool spinning direction is determined by the insert thickness direction.
2. VERICUT uses a pre-defined "Relief Angle" of 5 degrees for all tool inserts created with Tool Manager.

To Tool Add/Modify window

Tool Add/Modify window, Tool Component tab (Turn Insert)

Opened via adding or modifying a turning Insert Cutter tool component, this tab is used to describe the insert shape for the "Cutter" component in a turning tool assembly. Options are available to define the cutter insert via ISO standard shapes, profile sketcher, or reference a cutter insert in another tool assembly.

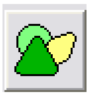
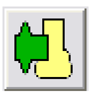
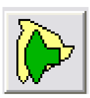
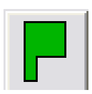
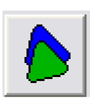
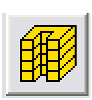
See Tool Add/Modify window, Assembly tab, also in this section, for information on positioning, and orienting, a turn insert component in a turning tool assembly.

Component Type — Use to specify the type of component you are creating. In this case an Insert Cutter for a turning tool.

ID — Use to specify the ID for the component you are creating. If not specified, VERICUT will assign a unique default ID.

Tool shape icons — Selecting an icon configures the bottom half of the window to define the selected cutter shape.

Options:

Icons	Name	Description
	General Insert	Displays the General Insert features enabling you to define ISO standard insert shapes.
	Groove Insert	Displays the Groove Insert features enabling you to define ISO standard groove insert shapes.
	Thread Insert	Displays the Thread Insert features enabling you to define ISO standard thread insert shapes.
	Profile (sweep)	Opens the Profile Sketcher in "sweep" mode.
	Reference	Displays the Reference features enabling you to reference an insert in another tool assembly.
	Model File	Displays the Model File features enabling you to specify a model file to be used as an insert.

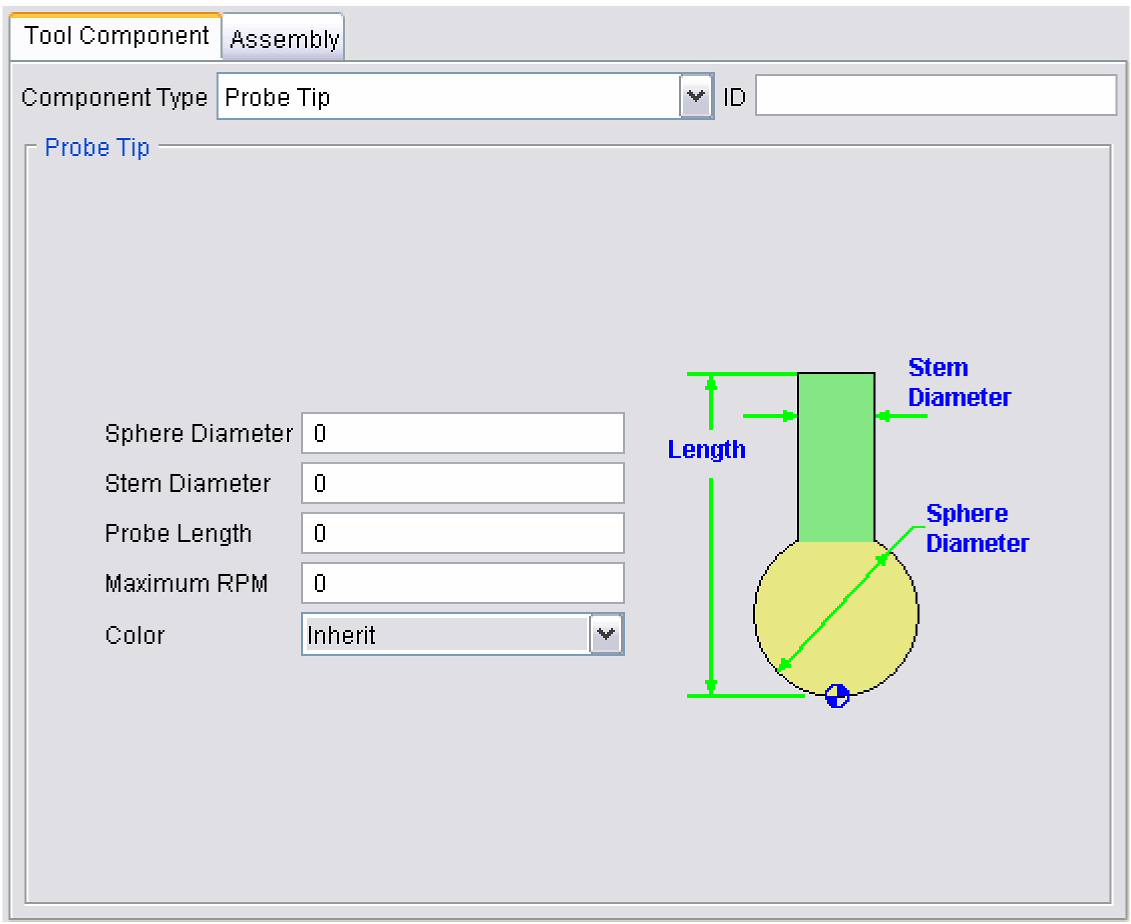
NOTE: VERICUT uses a pre-defined "Relief Angle" of 5 degrees for all tool inserts created with Tool Manager.

To Tool Add/Modify window

Tool Add/Modify window, Tool Component tab (Probe Tip)

Opened via adding or modifying a probe tip tool component, this tab is used to describe the shape of a probe tip component in a probe tool assembly.

See Tool Add/Modify window, Assembly tab, also in this section, for information on positioning, and orienting, a probe tip component in a probe tool assembly.



Component Type — Use to specify the type of component you are creating. In this case a Probe Tip.

ID — Use to specify the ID for the component you are creating. If not specified, VERICUT will assign a unique default ID.

Sphere Diameter — Diameter of the sphere at the end of the probe tip.

Stem Diameter — Diameter of the probe tip stem.

Probe Length — Length of the probe tip.

Maximum RPM — The maximum speed that a probe tool can spin before a spindle On/Off error is output. The default value is 0.

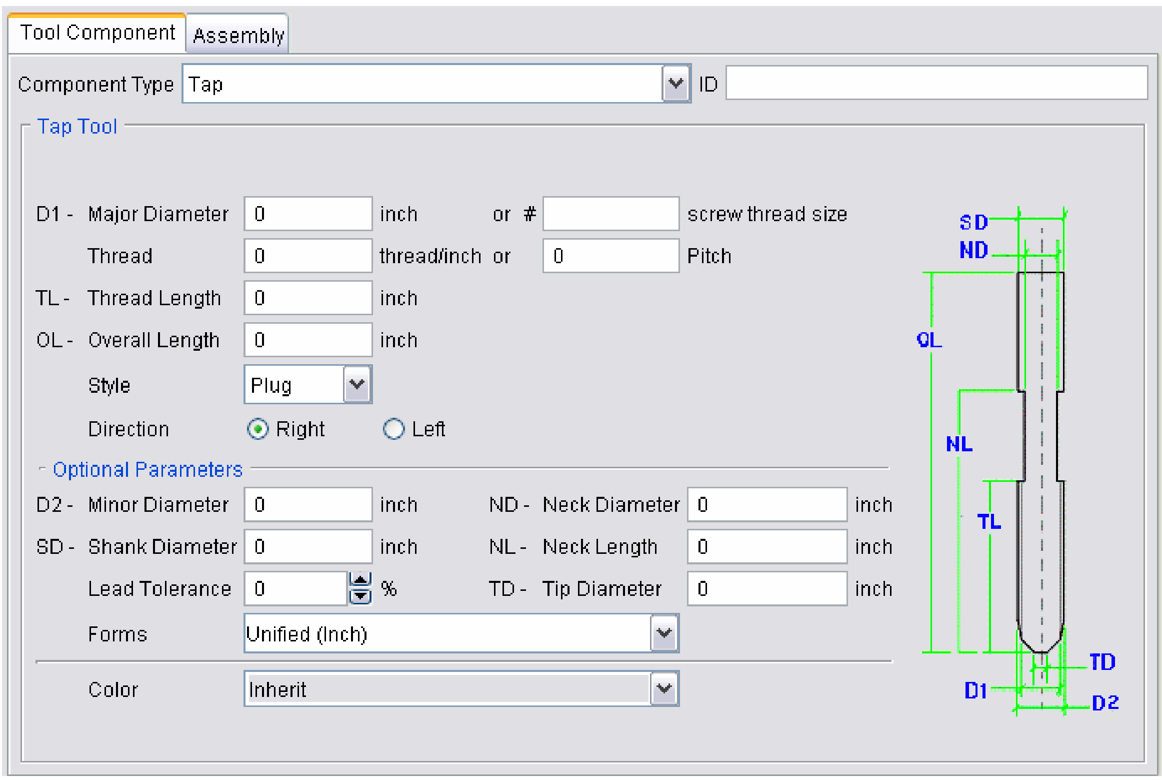
Color — Color that the tool component is to be displayed in VERICUT. For additional information, see Tool Display Colors in the **Tool Add/Modify window, Common Features** section.

To Tool Add/Modify window

Tool Add/Modify window, Tool Component tab (Tap)

Opened via adding, or modifying, a tap tool component, this window is used to describe the shape, position and orientation of a tap component in a tap tool assembly.

See Tool Add/Modify window, Assembly tab, also in this section, for information on positioning, and orienting, a tap component in a tap tool assembly.



Component Type — Use to specify the type of component you are creating. In this case a Tap.

ID — Use to specify the ID for the component you are creating. If not specified, VERICUT will assign a unique default ID.

Required Parameters

Major Diameter — Major diameter of the tap. Specify the diameter in either inches or millimeters, or by the screw thread size (i.e, #4, #6, #8, etc.).
Thread — Thread pitch of the tap. Specify in threads/inch, or threads/millimeter, or by thread pitch value.
Thread Length — Distance from tip to last full thread. Specify in inches or millimeters.
Overall Length — Overall Length of the tap. Specify in inches or millimeters.

Style — Specify the tap style. Select **Taper**, **Plug**, or **Bottom** from the pull-down list.
Direction — Direction of rotation for the tap. Choose **Right** or **Left**.

Optional Parameters

Minor Diameter — Diameter at the root (valley) of thread. Specify in inches or millimeters.
Shank Diameter — Diameter of shaft above full thread. Specify in inches or millimeters.
Lead Tolerance — Lead tolerance expressed as a percentage of the thread's lead value.
Neck Diameter — Reduced diameter between thread length and neck length. Specify in inches or millimeters.
Neck Length — Distance from tip to top of reduced diameter. Specify in inches or millimeters.
Tip Diameter — Diameter of tip of the cutter. Specify in inches or millimeters.
Forms — Use to specify the thread for. Select **Unified (Inch)**, **ISO (Metric)**, **Whitworth (British Standard)**, or **Buttress** from the pull-down list.

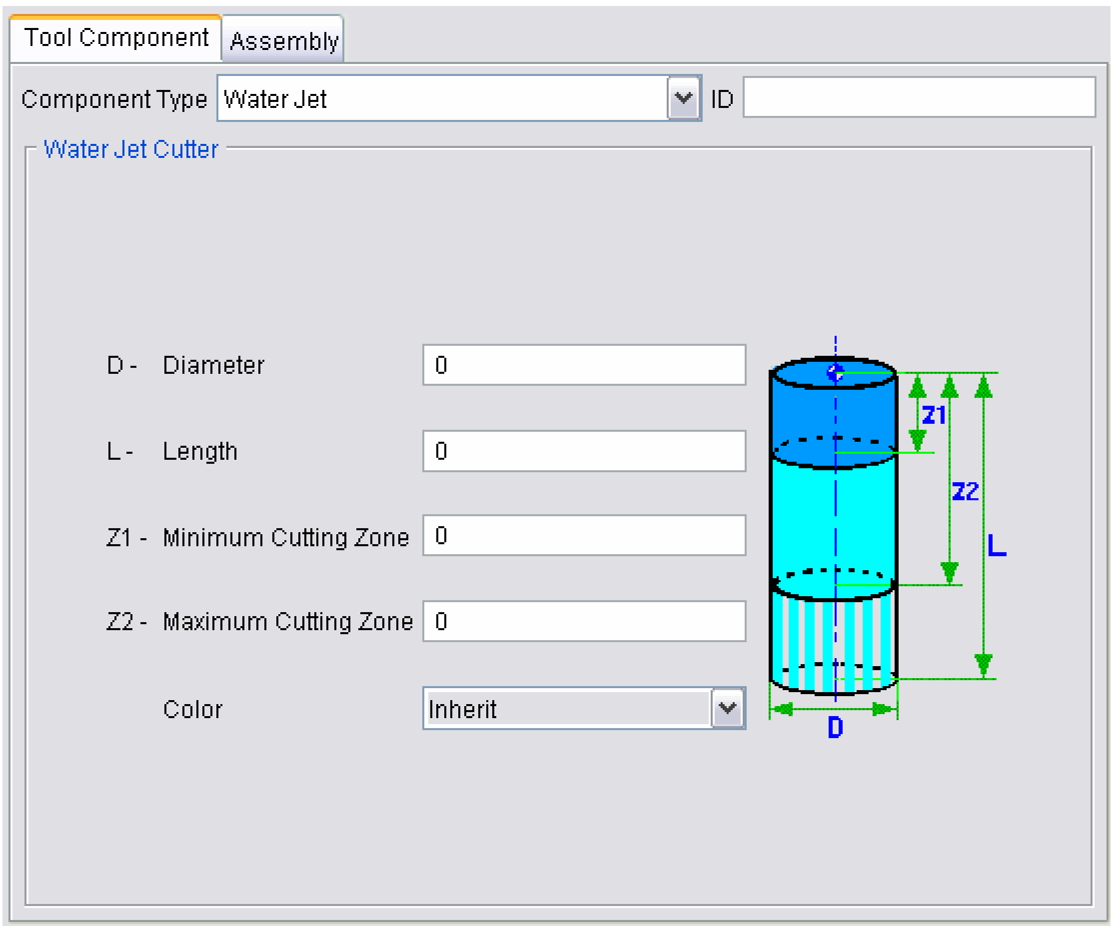
Color — Color that the tool component is to be displayed in VERICUT. For additional information, see Tool Display Colors in the **Tool Add/Modify window, Common Features** section.

To Tool Add/Modify window

Tool Add/Modify window, Tool Component tab (Water Jet)

Opened via adding or modifying a water jet tool component, this tab is used to describe the shape of a waterjet component in a water jet tool assembly.

See Tool Add/Modify window, Assembly tab, also in this section, for information on positioning, and orienting, a water jet component in a water jet tool assembly.



Component Type — Use to specify the type of component you are creating. In this case a Water Jet.

ID — Use to specify the ID for the component you are creating. If not specified, VERICUT will assign a unique default ID.

Diameter — Use to specify the cylindrical diameter of the water jet when the jet is on.

Length — Use to specify the maximum length of the water jet.

Minimum Cutting Zone — Use to specify the minimum distance from the Water Jet component origin, along the component's Z axis, where cutting is desired.

Maximum Cutting Zone — Use to specify the maximum distance from the Water Jet component origin, along the component's Z axis, where cutting is desired.

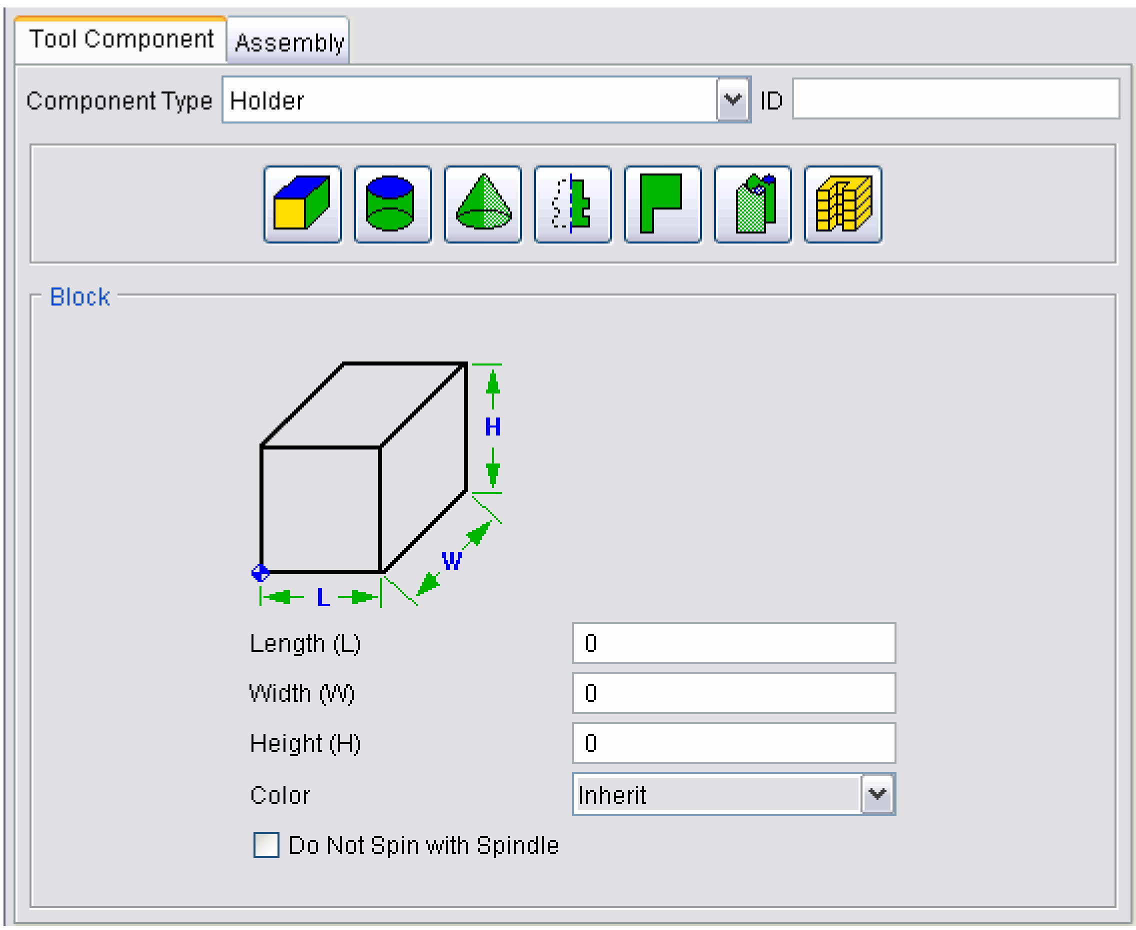
Color — Color that the tool component is to be displayed in VERICUT. For additional information, see Tool Display Colors in the **Tool Add/Modify window, Common Features** section.

To Tool Add/Modify window

Tool Add/Modify window, Tool Component tab (Holder)

Opened via adding or modifying a Holder tool component, this window is used to describe the shape of a selected "Holder" component in a tool assembly. Options are available to define the holder via parametric shapes, profile sketcher, or reference a holder in another tool assembly. More than one holder can be used in a tool assembly. An error similar to "HOLDER removed material..." is reported when the non-cutting portion of the cutter removes material. Material removed is shaded using the red Error color.

See Tool Add/Modify window, Assembly tab, also in this section, for information on positioning, and orienting, a holder component in a tool assembly.

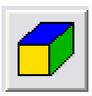
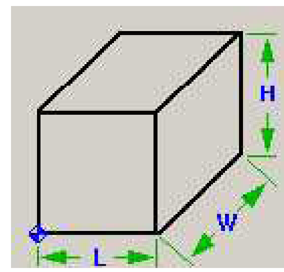
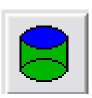
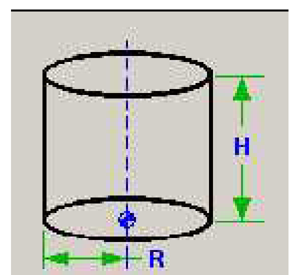
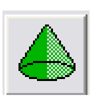
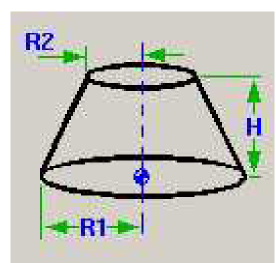
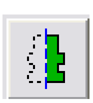
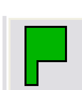


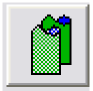
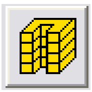
Component Type — Use to specify the type of component you are creating. In this case a Holder.

ID — Use to specify the ID for the component you are creating. If not specified, VERICUT will assign a unique default ID.

Holder icons — Selecting an icon configures the bottom half of the window with the parameters required to define the holder shape.

Options:

Icons	Name	Description	
	Block	Length (L) Width (W) Height (H)	
	Cylinder	Radius (R) Height (H) Tolerance	
	Cone	Bottom Radius (R1) Top Radius (R2) Height (H) Tolerance	
	Profile - SOR	Opens the Profile Sketcher in "SOR" mode.	
	Profile - Sweep	Opens the Profile Sketcher in "Sweep" mode.	

	Reference	Displays the Reference features enabling you to reference a holder in another tool assembly.
	Model File	Displays the Model File features enabling you to select a model file to define the holder.

The following option is common to all Tool Add/Modify Holder windows:

Do Not Spin with Spindle — When toggled "On", indicates that the holder component does not spin.

The following option is common to all Tool Add/Modify Holder windows except Reference:

Color — Color that the tool component is to be displayed in VERICUT. For additional information, see Tool Display Colors in the the **Tool Add/Modify window, Common Features** section.

To Tool Add/Modify window

Tool Add/Modify window, Common Features

Profile Sketcher Features

See Profile Sketcher Features in the **Setup Models** section above.

Reference feature

Tool Add/Modify, Reference feature enables you to "reference" a tool component in another tool library file for use in the current tool library file.

Reference features:

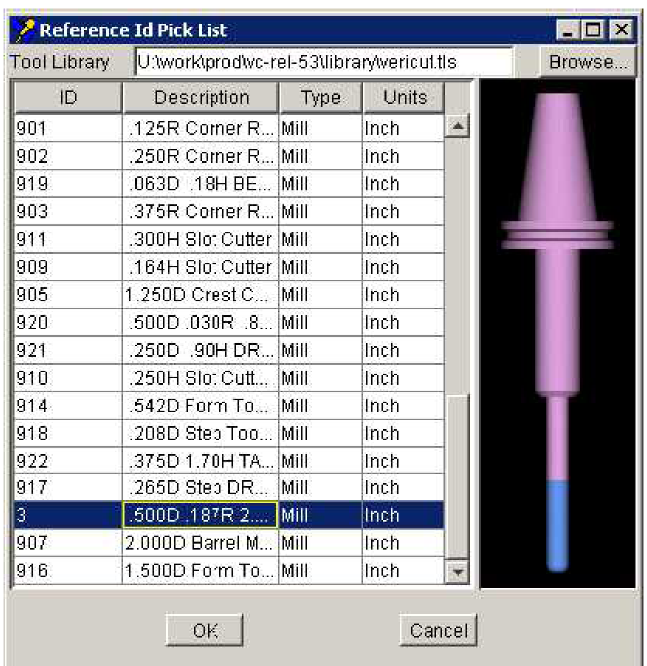
Reference File — Enter the */path/filename* of the tool library file containing the tool component to be referenced in the text field or use the Pick button to display the Reference Id Pick List window and use it to select the tool library file.

Reference ID — Enter tool ID of the tool component to be referenced in the text field or use the **Pick** button to display the Reference Id Pick List window and use it to select the tool component to be referenced.

Pick — Displays the Reference Id Pick List window enabling you to select the tool component to be referenced from the list. The first time that **Pick** is selected during a VERICUT session, a standard file selection window is displayed so that you can select a tool library file. After the tool library file has been selected, the Reference ID Pick List window is displayed so that you can select the tool component.

On subsequent selections of the Pick button during a session, the Reference Id Pick List window is displayed with the previously selected tool library file. Enter the */path/filename* in the **Tool Library** text field or use the **Browse** button to display a file selection window and use it to change the tool library file.

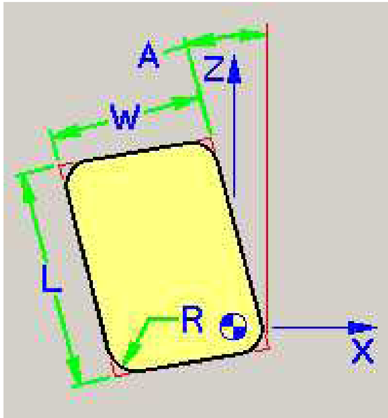
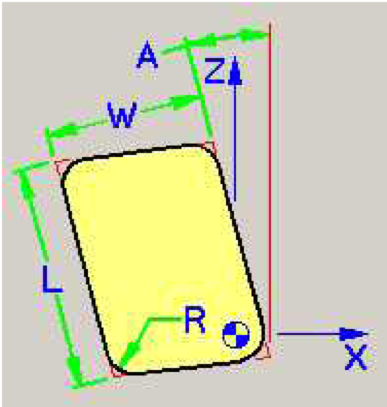
Sample Reference ID Pick List window:

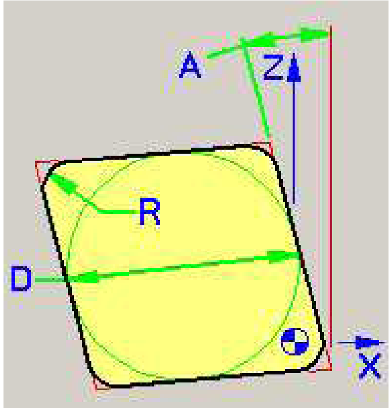
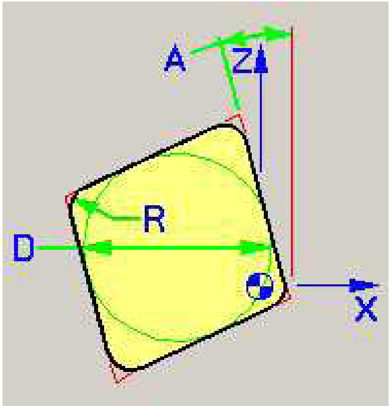
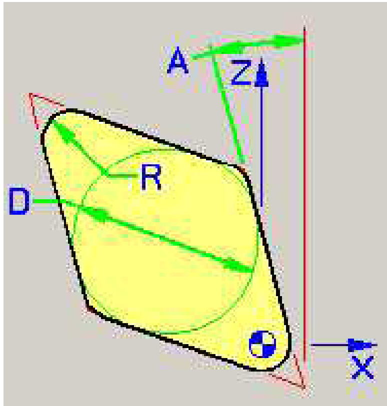


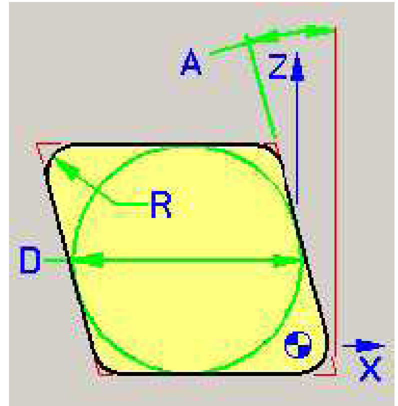
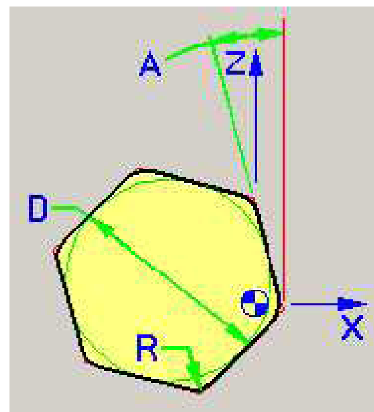
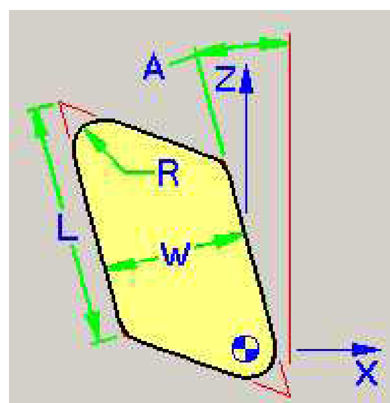
General Insert features

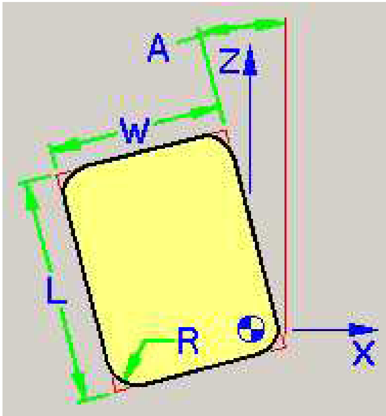
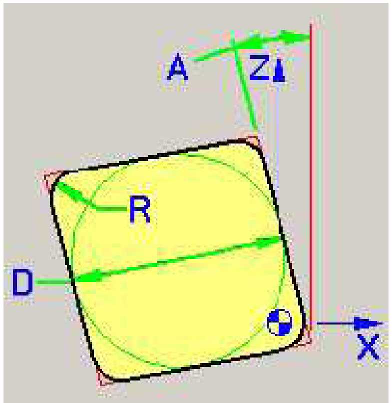
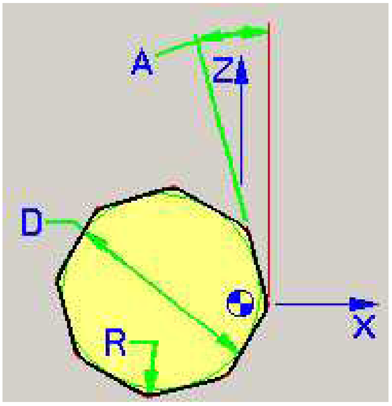
General Insert features enables you to define "ISO standard", as well as "custom" tool insert shapes. The parameters and diagram will vary depending on the Insert Type selected from the list. Only the parameters required to describe a particular insert type will be displayed.

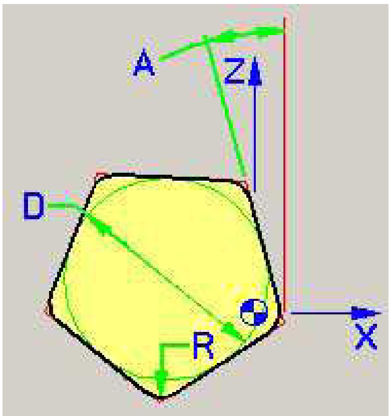
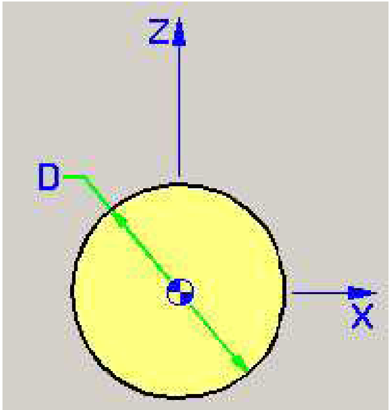
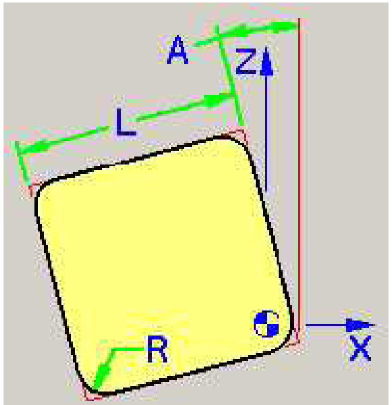
NOTE: VERICUT uses a pre-defined "Relief Angle" of 5 degrees for all tool inserts created with Tool Manager.

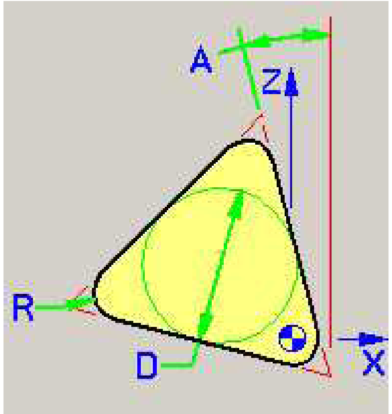
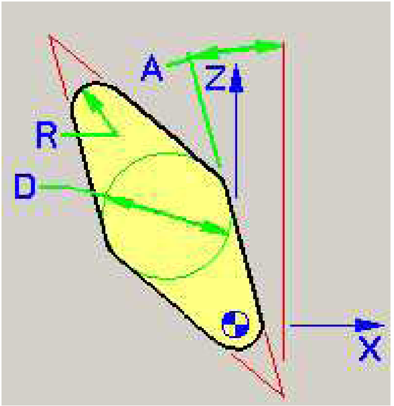
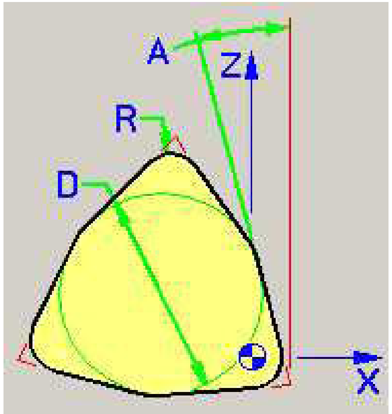
Insert Type	Parameter Description	
A - 85 degree parallelogram	L - Length W - Width Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	
B - 82 degree parallelogram	L - Length W - Width Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	

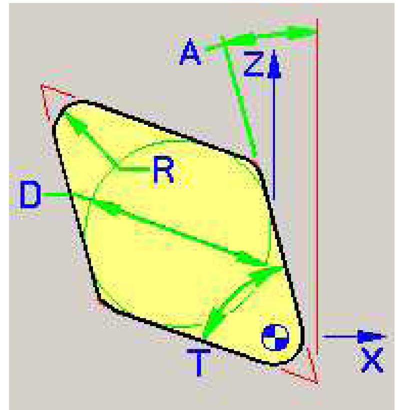
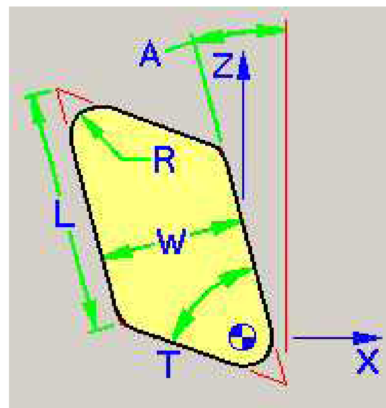
C - 80 degree diamond	D - Inscribed Diameter Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	
C - 100 degree diamond	D - Inscribed Diameter Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	
D - 55 degree diamond	D - Inscribed Diameter Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	

E - 75 degree diamond	D - Inscribed Diameter Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	
H - Hexagon	D - Inscribed Diameter Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	
K - 55 degree parallelogram	L - Length W - Width Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	

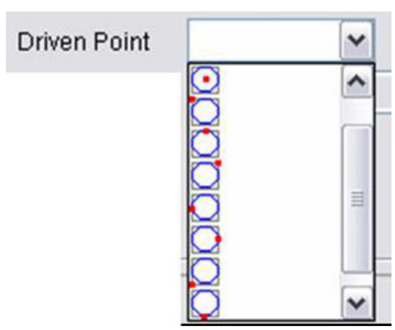
L - Rectangle	L - Length W - Width Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	
M - 86 degree diamond	D - Inscribed Diameter Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	
O - Octagon	D - Inscribed Diameter Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	

P - Pentagon	D - Inscribed Diameter Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	
R - Round	D - Inscribed Diameter Thickness Driven Point* Color *	
S - Square	L - Length Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	

T - Triangle	D - Inscribed Diameter Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	
V - 35 degree diamond	D - Inscribed Diameter Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	
W - 80 degree trigon	D - Inscribed Diameter Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	

Custom diamond	D - Inscribed Diameter Thickness R - Corner Radius A - Lead Angle T - Tip Angle Driven Point* Color *	
Custom parallelogram	L - Length W - Width Thickness R - Corner Radius A - Lead Angle Driven Point* Color *	

***Driven Point —** Use to specify the driven point for the insert. Select the desired driven point from the pull-down list.

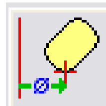


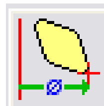
*** Color* —** Color that the tool component is to be displayed in VERICUT. For additional information, see Tool Display Colors in the **Tool Add/Modify window, Common Features** section.

Location Aids

These features are only available for mill Inserts. They enable you to easily position the insert with respect to the tool diameter.

Ø **(Diameter) —** Use this text field to specify the diameter of the tool.

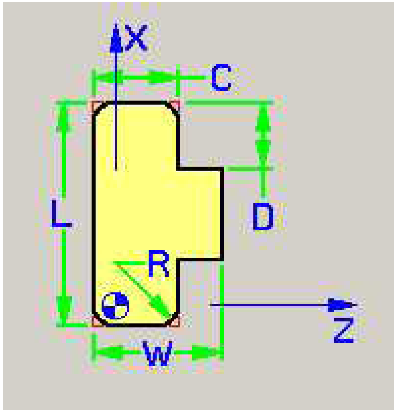
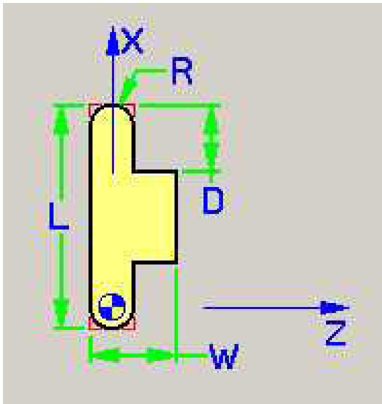
Locate by nominal diameter  **—** Positions the center of the insert corner radius on the tool diameter.

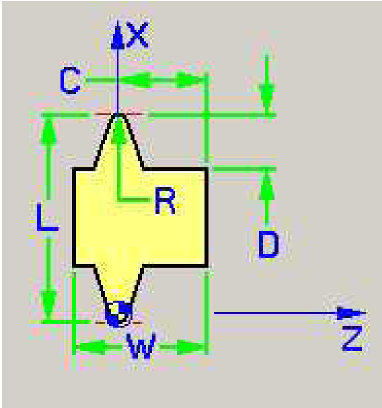
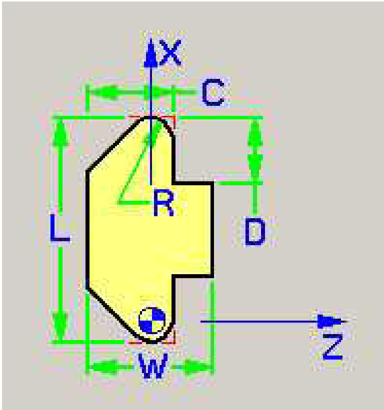
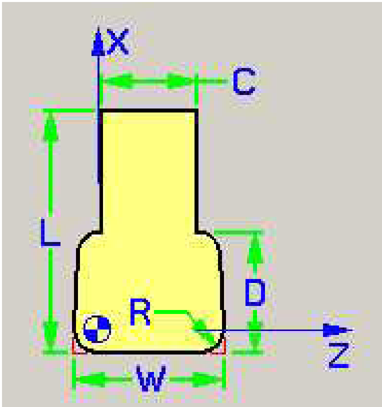
Locate by outside diameter  **—** Positions the outside edge of the insert on the tool diameter.

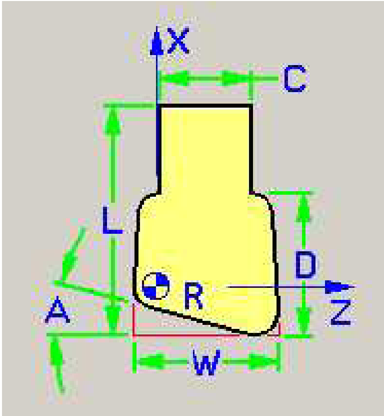
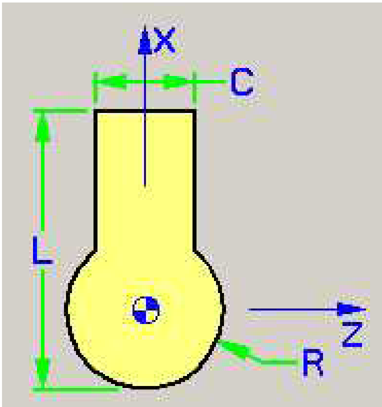
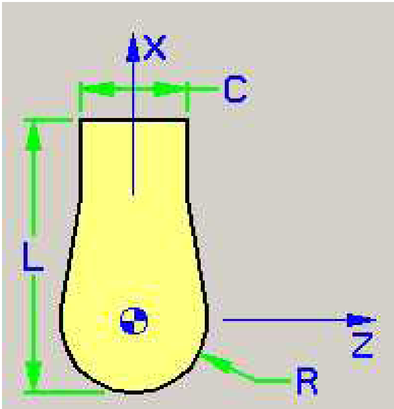
Groove Insert features

Groove Insert features enables you to define "ISO standard" groove insert shapes. The parameters and diagram will vary depending on the Insert Type selected from the list. Only the parameters required to describe a particular insert type will be displayed. Groove Insert features are only available for defining turning cutters.

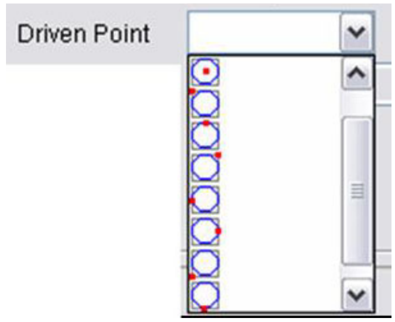
NOTE: VERICUT uses a pre-defined "Relief Angle" of 5 degrees for all tool inserts created with Tool Manager.

Insert Type	Parameter Description	
Square, double end	W - Width L - Length Thickness R - Corner Radius C D Driven Point* Color *	
Radius, double end	W - Width L - Length Thickness R - Corner Radius D Driven Point* Color *	

40 degree V, double end	W - Width L - Length Thickness R - Corner Radius C D Driven Point* Color *	
45 degree chamfer, double end	W - Width L - Length Thickness R - Corner Radius C D Driven Point* Color *	
Square, single end	W - Width L - Length Thickness R - Corner Radius C D Driven Point* Color *	

Chamfer, single end	W - Width L - Length Thickness R - Corner Radius C D A - Chamfer Angle Driven Point* Color *	
Round, single end	L - Length Thickness R - Corner Radius C Driven Point* Color *	
Radius, single end	L - Length Thickness R - Corner Radius C Driven Point* Color *	

***Driven Point —** Use to specify the driven point for the insert. Select the desired driven point from the pull-down list.

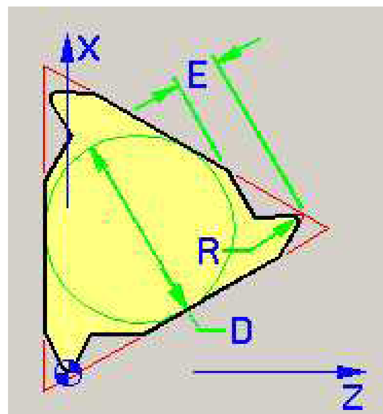
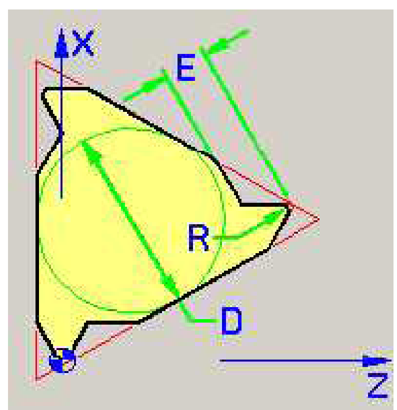


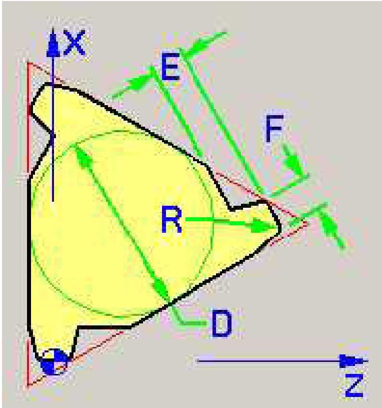
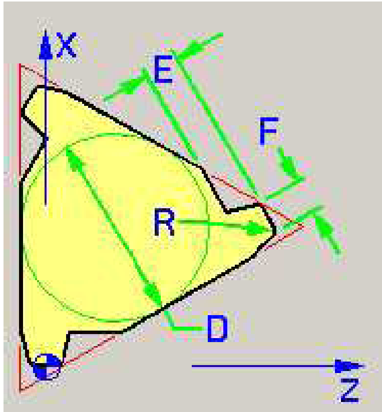
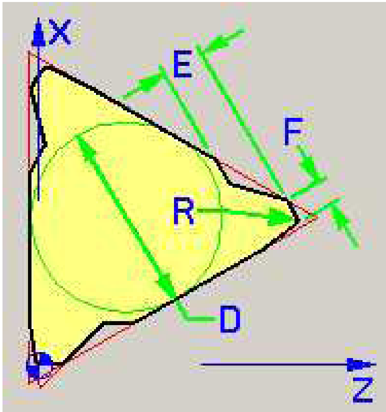
*** Color —** Color that the tool component is to be displayed in VERICUT. Color that the tool component is to be displayed in VERICUT. For additional information, see Tool Display Colors in the **Tool Add/Modify window, Common Features** section.

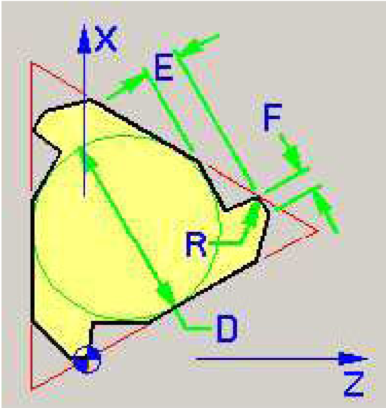
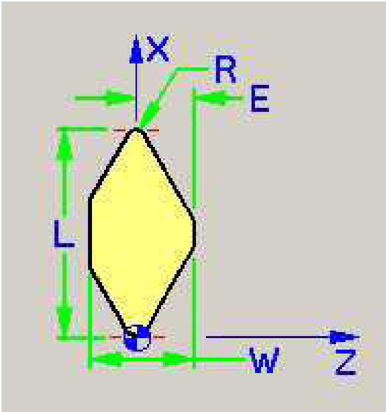
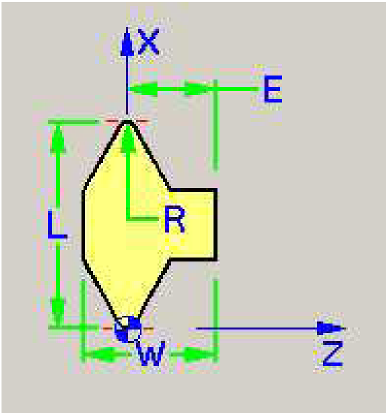
Thread Insert features

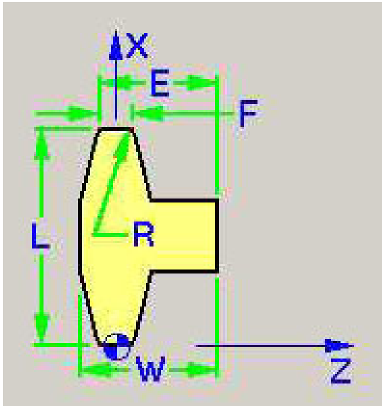
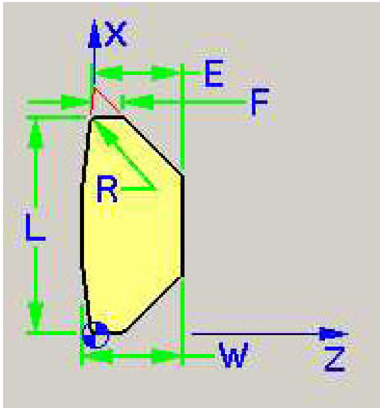
Thread Insert features enables you to define "ISO standard" thread insert shapes. The parameters and diagram will vary depending on the Insert Type selected from the list. Only the parameters required to describe a particular insert type will be displayed. Thread Insert features is only available for defining turning cutters.

NOTE: VERICUT uses a pre-defined "Relief Angle" of 5 degrees for all tool inserts created with Tool Manager.

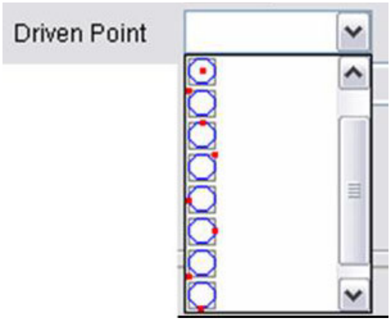
Insert Type	Parameter Description	
55 degree, triangle	D - Inscribed Diameter Thickness R - Corner Radius E Driven Point* Color *	
60 degree, triangle	D - Inscribed Diameter Thickness R - Corner Radius E Driven Point* Color *	

30 degree, trapeze triangle	D - Inscribed Diameter Thickness R - Corner Radius E F – Flat Driven Point* Color *	
ACME, triangle	D - Inscribed Diameter Thickness R - Corner Radius E F – Flat Driven Point* Color *	
Buttress, triangle, right hand	D - Inscribed Diameter Thickness R - Corner Radius E F – Flat Driven Point* Color *	

Buttress, triangle, left hand	D - Inscribed Diameter Thickness R - Corner Radius E F – Flat Driven Point* Color *	
60 degree, double end, asymmetric	W - Width L - Length Thickness R - Corner Radius E Driven Point* Color *	
60 degree, double end, symmetric	W - Width L - Length Thickness R - Corner Radius E Driven Point* Color *	

ACME, double end	W - Width L - Length Thickness R - Corner Radius E F – Flat Driven Point* Color *	
Buttress, double end	W - Width L - Length Thickness R - Corner Radius E F – Flat Driven Point* Color *	

***Driven Point —** Use to specify the driven point for turning inserts. It is not applicable to milling inserts. Select the desired driven point from the pull-down list.



*** Color —** Color that the tool component is to be displayed in VERICUT. Color that the tool component is to be displayed in VERICUT. For additional information, see Tool Display Colors in the **Tool Add/Modify window, Common Features** section.

Model File features

Model File — Enter the */path/filename* of the model file or use the **Browse** button to display a file selection window and use it to select the model file.

Sketcher — Displays the Profile Sketcher window enabling you to edit or create a "swept solid (.swp)" or "solid of revolution (.sor)" model file. This feature is only available the selected file is a SOR or SWEEP file type. To create a new SOR or SWEEP model file, clear the Model File text field and press **Sketcher**.

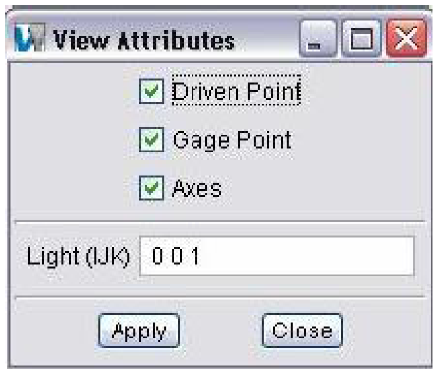
Normals — This feature applies to STL file models only and indicates the direction in which surface normals defined in the STL data point, relative to the model. Select **Inward**, **Outward**, or **Computed** (let VERICUT determine the direction of the surface normals).

Color — Color that the tool component is to be displayed in VERICUT. Color that the tool component is to be displayed in VERICUT. For additional information, see Tool Display Colors in the **Tool Add/Modify window, Common Features** section.

Add DXF Tool window

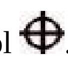
See DXF Geometry window in the **Setup Models** section above.

Tool Manager View Attributes window



The following features enable you to control view attributes related to the Tool Manager, Tool Display area.

Driven Point — Use to display/not display the driven point symbol .

Gage Point — Use to display/not display the gage point symbol .

Axes — Use to display/not display the axes representing the tool origin.

Light (IJK) — I J K vector direction in which the light source points. The three values are separated by spaces.

Tool Display Colors

The display color for tools (cutters and holders) in all VERICUT views is determined by the conditions described below:

If a tool component (cutter or holder) has a specific color defined in Tool Manager, then this color is used to display the component in all views. The cuts made by the cutter are also shaded with this color when Color Method, on the **Color window: Cut Colors tab**, is set to "**Tool Color**". Available colors are defined in the **Shade Colors** list on the **Edit menu > Color window: Define tab**.

If a tool component (cutter or holder) is defined with "Inherit" as the color in Tool Manager, then it will be displayed using a "default" color. The "default" color is defined as the color of the tool's first cutter component or, if the color of the tool's first cutter component is defined as "Inherit", the "default" color is defined as the current cut color from the Cut Color Table on the **Color window: Cut Colors tab**.

An inserted milling cutter, created by spinning several inserts, is displayed in the "default" color.

Turrets and Tool Chains

Tools in turrets and tool chains are displayed according to the following:

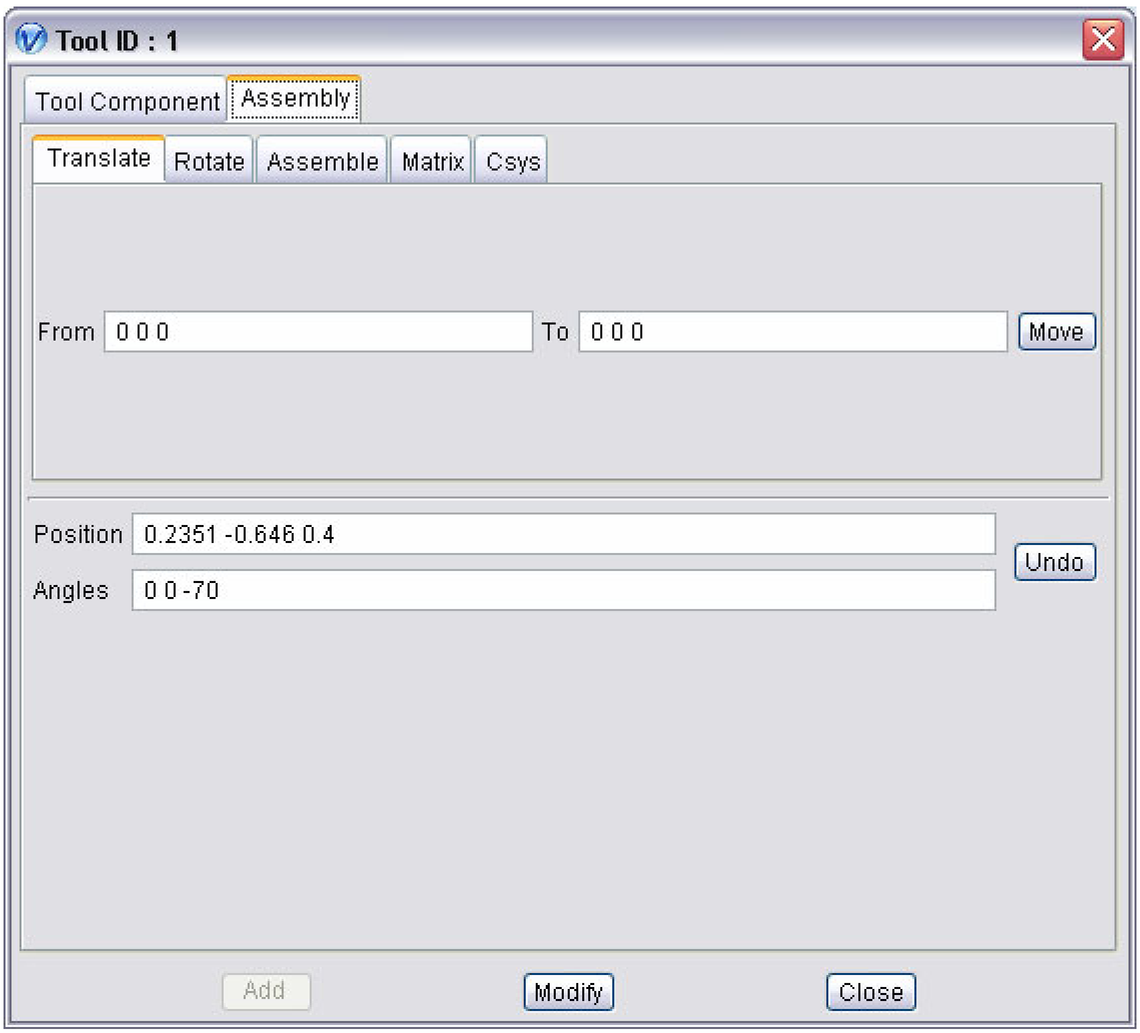
1. Tool holders with color set to "inherit" will always be colored with color index #5 in the Shade Color list (ref. **Color window: Define tab**, also in the *VERICUT Help* section). The same color displayed in Tool Manager.
2. Inserts with color set to "inherit" will be colored with color index #7 in the Shade Color list when initially loaded into a turret or tool chain. The same color displayed in Tool Manager.
3. Inserts with color set to "inherit" will be colored from the Cut Color Table (ref. **Color window: Cut Colors tab**, also in the *VERICUT Help* section) when the tool is used to cut. VERICUT does not know the tool's sequence until it is used to cut. The tools will then this color if on a turret or tool chain.
4. Tool components with specified color will always be displayed the specified color.

Tool Add/Modify window, Assembly tab

Location: **Project menu > Tools: Add menu > Add Tool Component**, or

Project menu > Tools: Edit menu > Modify

Features on this tab are used translate or rotate a component in a tool assembly, or to move the selected component by assembling (mating or aligning) it with other objects in the tool assembly.



Translate tab **—** Features in this group translate the selected tool component via indicating "from" and "to" points to move the tool component.

Rotate tab **—** Features on this tab rotate the selected tool component about a rotation center point.

Assemble tab **—** Features on this tab enable you to assemble tools by mating or aligning one to three planar surfaces with surfaces on other tool components, similar to assembling models.

Matrix tab **—** Features on this tab move the selected tool component via a twelve parameter transformation matrix.

Csys tab **—** Features on this tab enable you to move a selected tool component from one coordinate system to another.

Position — Specifies the absolute XYZ position of the tool component relative to the tool origin. Values in this field are separated by spaces.

Angles — Specifies the absolute XYZ rotation of the tool component relative to the tool origin. Values in this field are separated by spaces.

Undo — Returns the tool component to its previous location, or as it was when the window was opened.

Add — Adds the defined tool component to the **Tool Manager** and closes the **Tool Add/Modify window**.

Modify — Adds the modified tool component to the **Tool Manager** and closes the **Tool Add/Modify window**.

Close — Closes the **Tool Add/Modify window** without adding/ modifying the tool.

To Add/Modify window

Tool Add/Modify window, Assembly tab, Translate tab

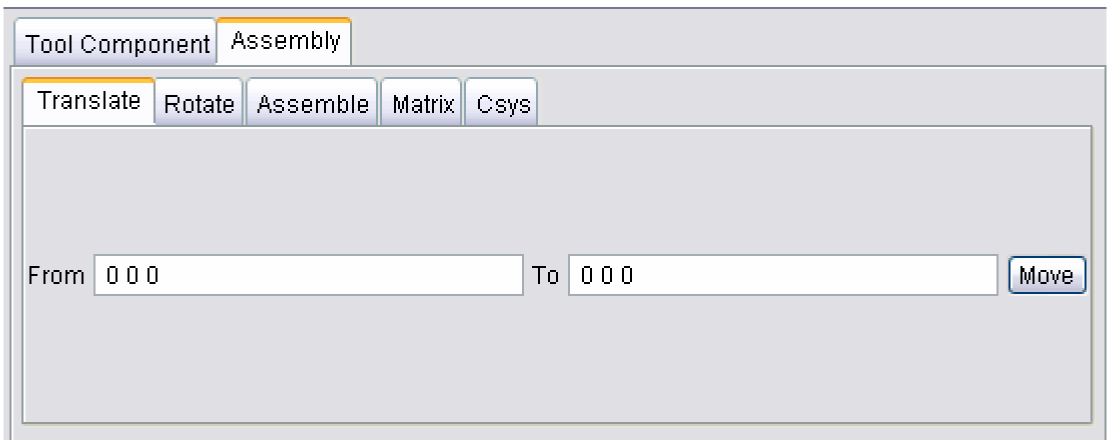
Location: **Project menu > Tools: Add menu > Add Tool Component: Assembly tab**

or

Project menu > Tools: Edit menu > Modify: Assembly tab

Features on this tab are used to translate the selected tool component via indicating "from" and "to" points to move the object. Movement occurs each time you press the **Move** button. If the applied movement is incorrect, press **Undo** (on the Assembly tab) to return the object to its previous location.

Features on this tab work the same as the **Translate** tab features for modeling.



Translate tab features:

From / To — Specifies the locations to move the tool component from and to, relative to the tool origin. XYZ values can be entered (separated by spaces), or selected by clicking in the field then clicking on a tool component shape. As you move the mouse over the tool, a crosshair and vector show you the pending pick-point location.

Move — Moves the selected object by the incremental distance, as calculated from the "**From**" point to the "**To**" point location.

To Assembly tab

Tool Add/Modify window, Assembly tab, Rotate tab

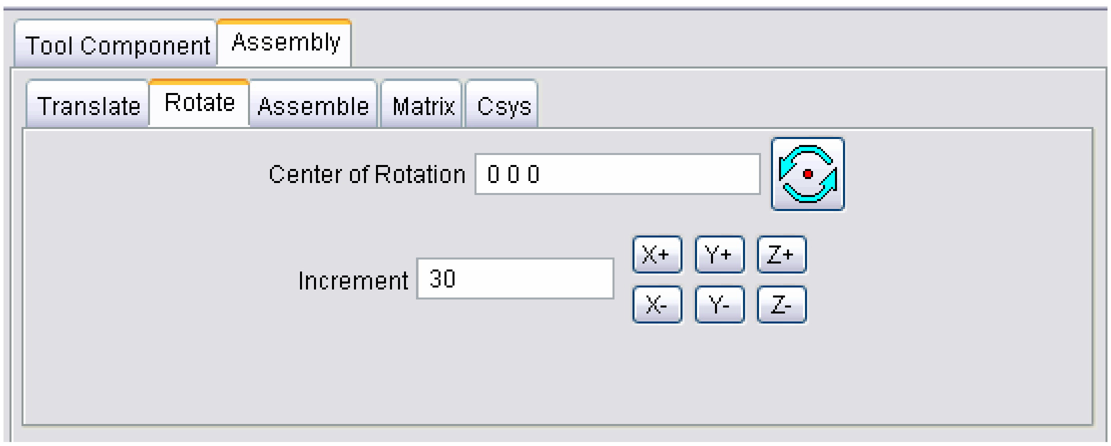
Location: **Project menu > Tools: Add menu > Add Tool Component: Assembly tab**

or

Project menu > Tools: Edit menu > Modify: Assembly tab

Features on this tab are used to rotate the selected tool component about a rotation center point. Movement occurs each time you press one of the rotation direction buttons: **X+/X-**, **Y+/Y-**, **Z+/Z-**. If the applied rotation is incorrect, press **Undo** (on the Assembly tab) to return the object to its previous location.

Features on this tab work the same as the **Rotate** tab features for modeling.



Rotate tab features:

Center of Rotation — Specifies XYZ point location about which to rotate the tool component. XYZ values can be entered in the **Center of Rotation** text field (separated by spaces), or selected by clicking in the **Center of Rotation** text field, then clicking on a tool component in the **Tool Display** area. To see the center of rotation, press . To remove the center of rotation symbol press the button again, or close the window.

Increment — Specifies incremental degrees of rotation to apply when one of the rotation direction buttons are pressed.

Rotation buttons — (**X+/X-**, **Y+/Y-**, **Z+/Z-**) Applies the incremental degrees of rotation specified in the **Increment** field. Rotation occurs about the **Center of Rotation** relative to the tool origin.

To Assembly tab

Tool Add/Modify window, Assembly tab, Assemble tab

Location: **Project menu > Tools: Add menu > Add Tool Component: Assembly tab**

or

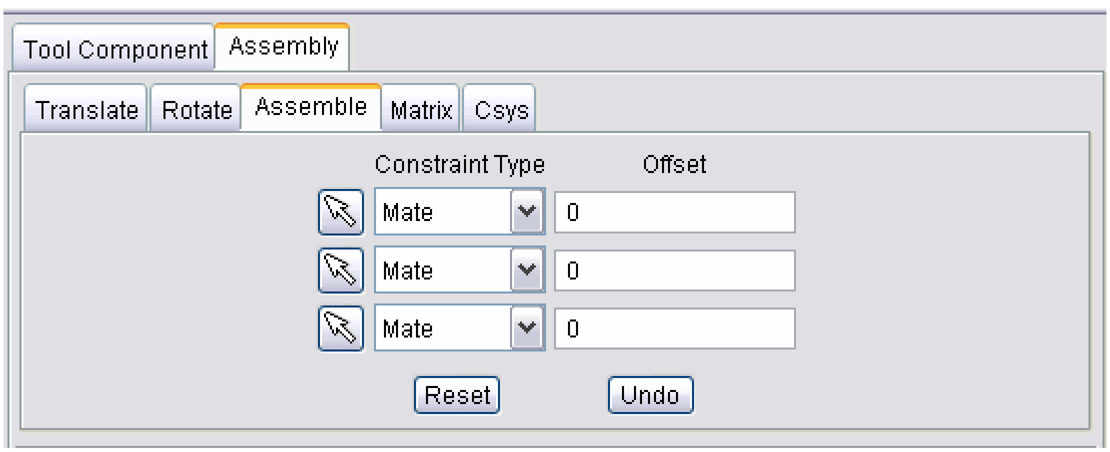
Project menu > Tools: Edit menu > Modify: Assembly tab

The features on this tab enable you to assemble tool components, by mating or aligning one to three planar surfaces with surfaces on other tool components, similar to assembling models. If a non-planar surface is selected, VERICUT constructs a tangent plane at the pick point. The relationship of surfaces being mated or aligned is known as a "constraint". If the applied movement is incorrect, press **Undo** (on the Assembly tab) to return the object to its previous location.

Features on this tab work the same as the **Assemble** tab features for modeling.

Follow these general steps to define a constraint for assembly:

1. Choose the constraint type.
2. Select a surface on the object to move.
3. Select the surface to move the object relative to.



Assemble tab features:

Constraint Type — Specifies how to constrain selected surfaces during tool component movement. After selecting two surfaces to define a constraint, VERICUT moves the tool component and highlights the satisfied constraint with a checkmark.

Options:

Mate — Moves the tool component so the selected surface opposes the surface selected on the second tool component (surface normals oppose each other).

Align — Moves the tool component so the selected surface is aligned with the surface selected on the second tool component (surface normals point in the same direction).

Offset — Distance and direction in which to offset constrained surfaces, applied normal to the surface.

Reset — Resets constraints to receive new data.

Undo — Returns the object to its previous location, or as it was when the window was opened.

To Assembly tab

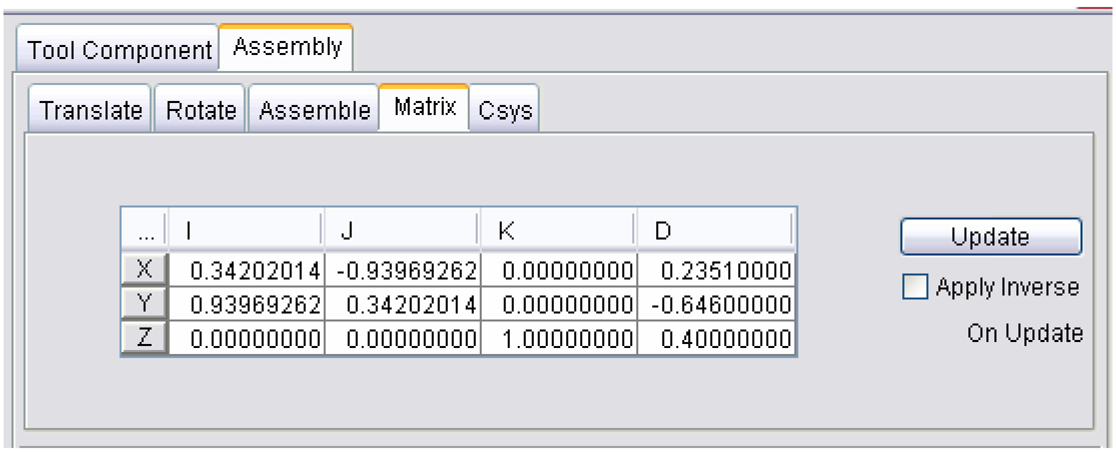
Tool Add/Modify window, Assembly tab, Matrix tab

Location: **Project menu > Tools: Add menu > Add Tool Component: Assembly tab**

or

Project menu > Tools: Edit menu > Modify: Assembly tab

Features on this tab move the selected component via a twelve parameter transformation matrix. If the applied movement is incorrect, press **Undo** (on the Assembly tab) to return the object to its previous location.
Features on this tab work the same as the **Matrix** tab features for modeling.



Matrix tab features:

Matrix table — The transformation matrix table is similar to the matrix used in programming APT tool paths. Its twelve parameters reveal the geometric attributes of the local (transformed) coordinate system (CSYS) in terms of the machine origin.

The format of the matrix table is as follows:

	I	**J**	**K**	**D**
X	I1	J1	K1	D1
Y	I2	J2	K2	D2
Z	I3	J3	K3	D3

Each row represents an axis of the local CSYS. The first three columns represent the vector associated with each axis: I1, J1, K1 as the positive X-axis vector; I2, J2, K2 as the positive Y-axis vector; and I3, J3, K3 as the positive Z-axis vector. The fourth column values D1, D2, D3 represent the coordinates of the origin point of the local CSYS.

NOTE: If you prefer to see the Matrix Table displayed with the I, J, K along the vertical axis and the X, Y, Z along the horizontal axis, set the environment variable, **CGTECH_MATRIX_FORMAT**=**VERTICAL**.

Update — Updates the tool component location to reflect the matrix table transformation.

Apply Inverse On Update — When selected, inverts the matrix so that its twelve parameters reveal the geometrical attributes of the machine origin in terms of the local (transformed) coordinate system.

To Assembly tab

Tool Add/Modify window, Assembly tab, CSYS tab

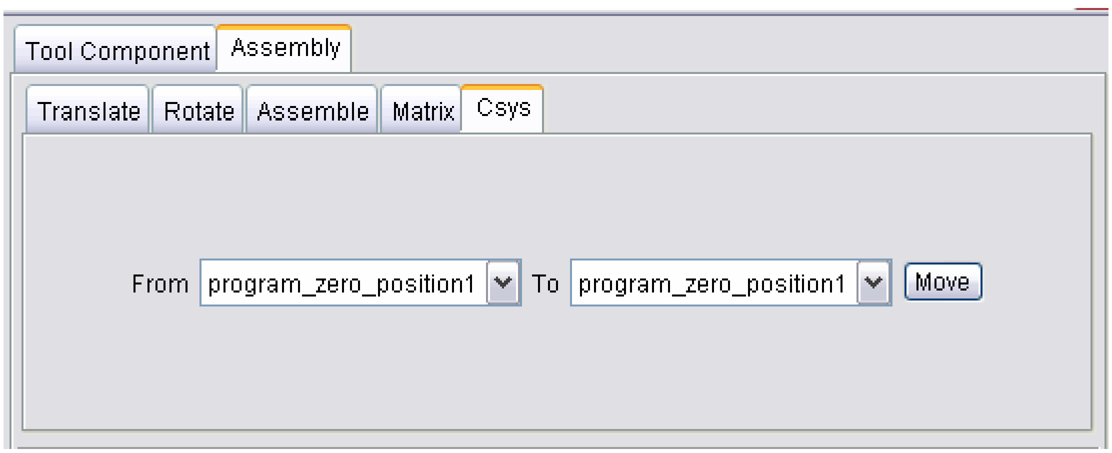
Location: **Project menu > Tools: Add menu > Add Tool Component: Assembly tab**

or

Project menu > Tools: Edit menu > Modify: Assembly tab

Features on this tab enable you to translate the selected tool component from one coordinate system (CSYS) to another. Select the "**From**" CSYS and the "**To**" CSYS to move the object. Movement occurs each time you press the **Move** button. If the applied movement is incorrect, press **Undo** to return the tool component to its previous location.

Features on this tab work the same as the **CSYS** tab features for modeling.



CSYS tab features:

From / To — Specifies the coordinate system to move the tool component from, and to. Select the appropriate CSYS from each of the pull-down lists.

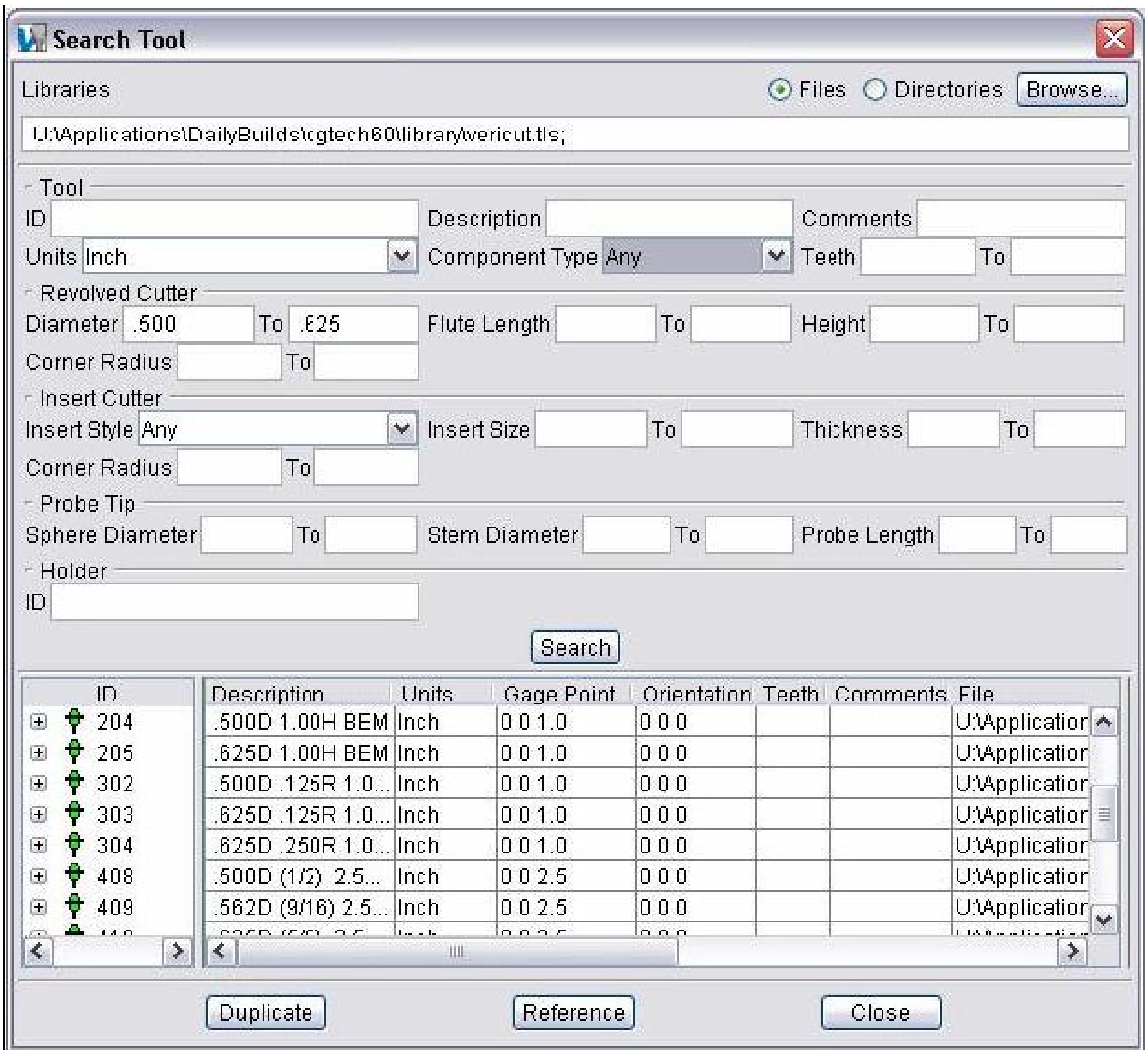
Move — Moves the selected tool component from the "**From**" CSYS to the "**To**" CSYS orientation.

To Assembly tab

Search Tool window

Location: **Project menu > Tools:** **Add menu > Tool > Find Existing** or
Edit menu > Search Tool

Search Tool enables you to search existing tool libraries for tools with the specific attributes.



Libraries group:

Files — Use to search one or more tool library files for tools meeting the specified search criteria.

Directories — Use to search one or more directories of tool library files for tools meeting the specified search criteria.

Browse — Displays a selection window to assist you in selecting the files/directories to be searched.

Tool group:

ID — Search for the specified tool ID.

Description — Search for a specified string of text within the tool's Description field.

Comments — Search for a specified string of text within the tool's Comment field.

Units — Select Any, Inch or Millimeter from the pull-down list.

Component Type — Use to search for tools that include a specific tool component type. Select Any, Revolved Cutter, Insert Cutter, Probe Tip, or Holder from the pull-down list.

Teeth — Use to search for tools containing cutters with a specific number (or within range of) of teeth.

Revolved Cutter group:

Diameter — Use to search for revolved cutters with a specific diameter (or within a diameter range).

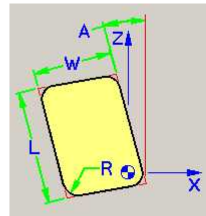
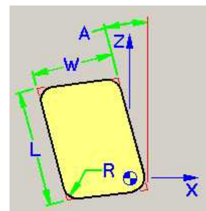
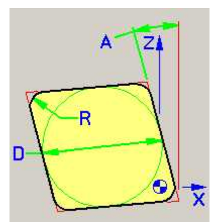
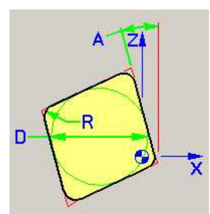
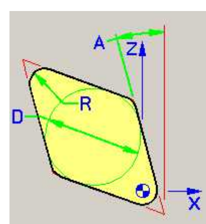
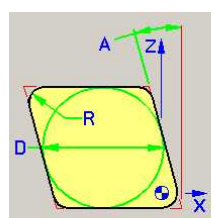
Flute Length — Use to search for revolved cutters with a specific flute length (or within a flute length range).

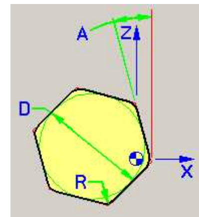
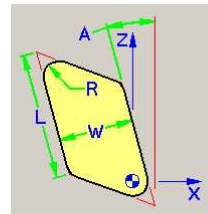
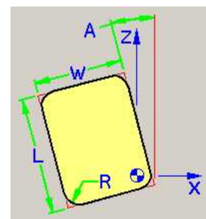
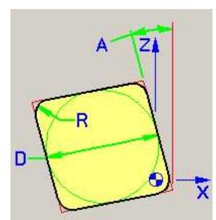
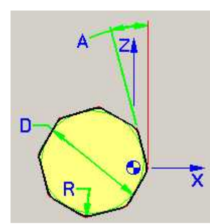
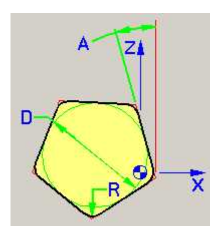
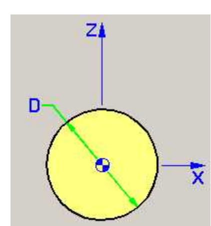
Height — Use to search for revolved cutters with a specific height (or within a height range).

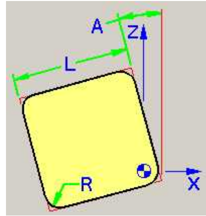
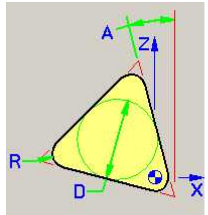
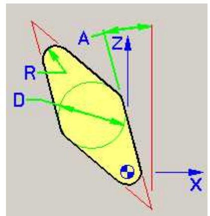
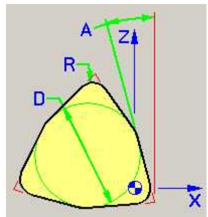
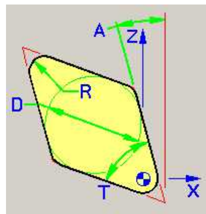
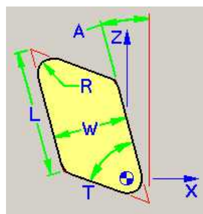
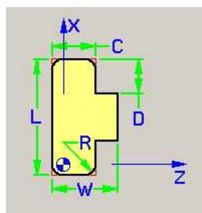
Corner Radius — Use to search for revolved cutters with a specific corner radius (or corner radii within a range).

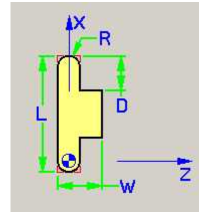
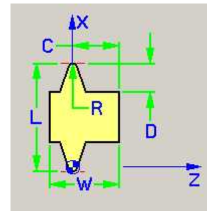
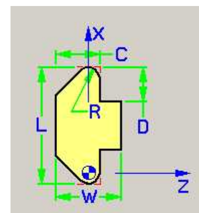
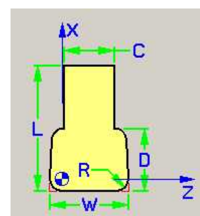
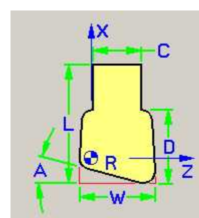
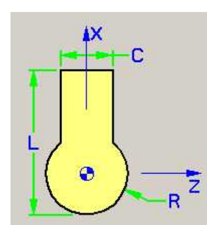
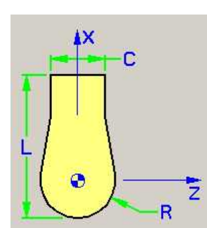
Insert Cutter group:

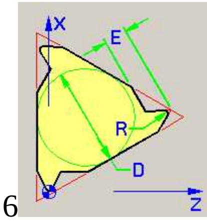
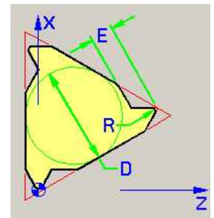
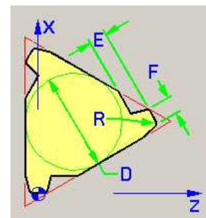
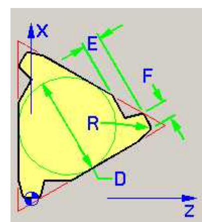
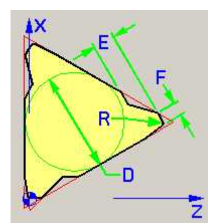
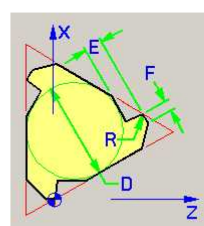
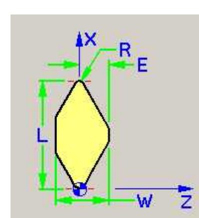
Insert Style — Use to search for a specific insert style (General, Groove, or Thread). Select one of the following insert styles from the pull-down list"

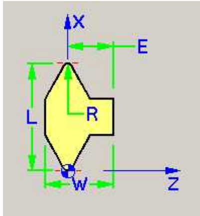
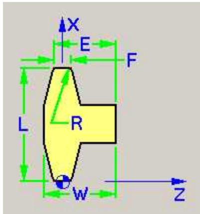
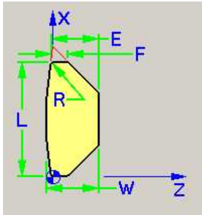
Insert Style	Description	
Any		
GENERAL INSERT A	85 Degree Parallelogram	
GENERAL INSERT B	82 Degree Parallelogram	
GENERAL INSERT C	80 Degree Diamond	
GENERAL INSERT C 100	100 Degree Diamond	
GENERAL INSERT D	55 Degree Diamond	
GENERAL INSERT E	75 Degree Diamond	

GENERAL INSERT H	Hexagon	
GENERAL INSERT K	55 Degree Parallelogram	
GENERAL INSERT L	Rectangle	
GENERAL INSERT M	86 Degree Diamond	
GENERAL INSERT O	Octagon	
GENERAL INSERT P	Pentagon	
GENERAL INSERT R	Round	

<table>
<tr><td>GENERAL INSERT S</td><td>Square</td><td>
</td></tr>
<tr><td>GENERAL INSERT T</td><td>Triangle</td><td>
</td></tr>
<tr><td>GENERAL INSERT V</td><td>35 Degree Diamond</td><td>
</td></tr>
<tr><td>GENERAL INSERT W</td><td>80 Degree Trigon</td><td>
</td></tr>
<tr><td>GENERAL INSERT CD</td><td>Custom Diamond</td><td>
</td></tr>
<tr><td>GENERAL INSERT CP</td><td>Custom Parallelogram</td><td>
</td></tr>
<tr><td>GROOVE INSERT S2</td><td>Square, double end</td><td>
</td></tr>
</table>

GROOVE INSERT R2	Radius, double end	
GROOVE INSERT V2	40 Degree V, double end	
GROOVE INSERT C2	45 Degree Chamfer, double end	
GROOVE INSERT S1	Square, single end	
GROOVE INSERT C1	Chamfer, single end	
GROOVE INSERT O1	Round, single end	
GROOVE INSERT R1	Radius, single end	

THREAD INSERT T55	55 Degree, triangle	6 
THREAD INSERT T60	60 Degree, triangle	
THREAD INSERT TT	30 Degree, trapeze triangle	
THREAD INSERT TA	ACME, triangle	
THREAD INSERT TBR	Buttress, triangle, right hand	
THREAD INSERT TBL	Buttress, triangle, left hand	
THREAD INSERT A60	60 Degree, double end, asymmetrical	

THREAD INSERT S60	60 Degree, double end, symmetric	
THREAD INSERT DA	ACME, double end	
THREAD INSERT TDB	Buttress, double end	

Insert Size — Use to search for insert cutters with a specific insert size (or within a range of insert sizes).

Thickness — Use to search for insert cutters with a specific thickness (or within a thickness range).

Corner Radius — Use to search for insert cutters with a specific corner radius (or corner radii within a range).

Probe Tip group:

Sphere Diameter — Use to search for probe tips with a specific sphere diameter (or within a diameter range).

Stem Diameter — Use to search for probe tips with a specific stem diameter (or within a diameter range).

Probe Length — Use to search for probe tips with a specific length (or within a length range).

Holder group:

ID — Use to search for a holder with a specific ID.

Search — Start the search of the specified tool library (libraries) based on the attributes specified above.

Tool List:

Displays a list of the tools found during the search. The Tool List display and interaction is the same as the Tool Table in the Tool Manager, with the one exception that the tool library file name is also displayed here.

Duplicate — Duplicates the selected tool in the current tool library.

Reference — Creates a reference to the selected tool in the current tool library.

Close — Closes the Search Tool window.

OptiPath window

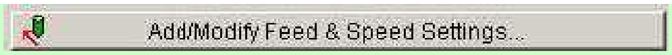
VERICUT Users:

Location: **Project menu > Tools:** **Add menu > OptiPath > New**, or

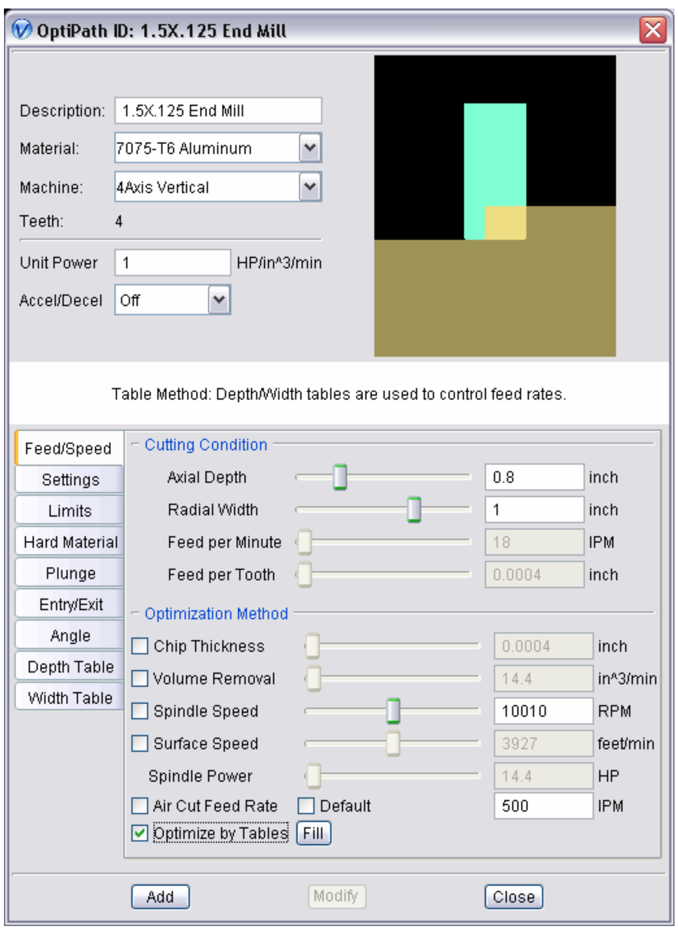
Edit menu > Modify (when an OptiPath tool component is highlighted)

Mold and Die Users:

Mold and Die Location: **Other Settings page > Add/Modify Speed and Feed Settings**

Notebook Feature: 

Opens a window enabling you to create and maintain OptiPath records that describe how to optimize feed rates and/or spindle speeds for various tools and cutting conditions. The OptiPath data is used by VERICUT during tool path optimization. The OptiPath window shows the optimization values for the selected tool.



OptiPath record data

Description — Use to enter a description for the OptiPath record.

Material — Stock material to be cut. Enter a new material name, or click on the arrow to select from a list of materials defined in the library. Any combination of alpha or numeric characters can be used.

Machine — NC machine that will be used to cut the workpiece. Enter a new machine name, or click on the arrow to select from a list of machines defined in the library. Any combination of alpha or numeric characters can be used.

Teeth — Displays the number of teeth that the tool has.

Unit Power — Power that is required to remove one cubic unit of material per minute. The power specification differs, depending on the units set for the OptiPath record (ref. Settings tab below).

Accel/Decel — When active and a G-Code tool path is being optimized, this feature applies Accel/Decel settings from the NC machine configuration to adjust optimized feed rates. VERICUT looks ahead in the tool path file and slows the machine prior to direction changes, as well as changes feed rates based on the machine axes ability to accelerate and decelerate.

Options:

Off — Does not adjust optimized feed rates for acceleration/deceleration.

Exact Stop — Assumes all axes must stop and be positioned exactly at the programmed location before the next block is processed. This option is recommended when sharp corners are required between motion blocks.

Continuous — Allows small errors in positioning in order to maintain a more constant velocity during corners or direction changes.

NOTES:

1. The block preceding an un-optimized block is not processed through Accel/Decel logic.
2. Accel/Decel settings apply to all linear and circular motions, including those in air.
3. These Accel / Decel setting are used to calculate OptiPath feedrates and cycle times. They are unrelated to the Accel / Decel settings in the in the Modeling window: Component Attributes tab.

OptiPath Cutter Shape Graphic — The cutter shape automatically displays for the tool in the Tool Library that the OptiPath record is associated with. VERICUT refers to the cutter shape to calculate material removal volume and other machining related data via features on the OptiPath Manager Feed/Speed tab.

Feed/Speed tab — Features on this tab are used to describe and test various cutting conditions to optimize, as well as control how optimization will be performed for the selected tool.

Tip: Watch the white message area in the middle of the **OptiPath window** for hints on using Optimization Method features as you select them on the Feed/Speed tab.

Settings tab — Features on this tab specify important settings for the optimization process, such as: the OptiPath record units, when to add (interpolate) more cuts, resolution, minimum/maximum optimized feed rates, how to optimize circle and NURBS motions, and more.

Limits tab — The features on this tab enable you to specify limits for cutting characteristics (feedrate, cut depth, volume removal rate, chip thickness, surface speed and RPM) for milling cutters and drills. It ***does not*** apply to turning tools.

Hard Material tab — The features on this tab enable you to adjust the optimized feed rate for 'hostile' cutting conditions (non-climb, thin radial width, and side-loaded cuts) when cutting hard materials.

Plunge tab — The features on this tab control the optimized feed rate for motions along the tool axis that plunge into or retract from material.

Entry/Exit tab — The features on this tab control the optimized feed rate for entering and exiting material.

Angle tab — The features on this tab adjust the optimized feed rate for 3-axis linear cuts ramping in material.

Depth Table tab — The features on this tab controls how optimized feed rates are calculated for cutting at various depths during **Optimize by Tables** optimization.

Width Table tab — The features on this tab adjust the optimized feed rate for cuts that are less than full cutter width during **Optimize by Tables** optimization.

Add — Adds the OptiPath data and optimization parameters to the newly created OptiPath record.

Modify — Applies the modified OptiPath data and optimization parameters to an existing OptiPath record.

Close— Closes the OptiPath Manager window.

See "**Overview of Optimizing NC Programs**" in the *Using VERICUT* section, in the *CGTech Help Library* for information about using OptiPath to optimize your NC programs.

OptiPath window, Feed/Speed tab

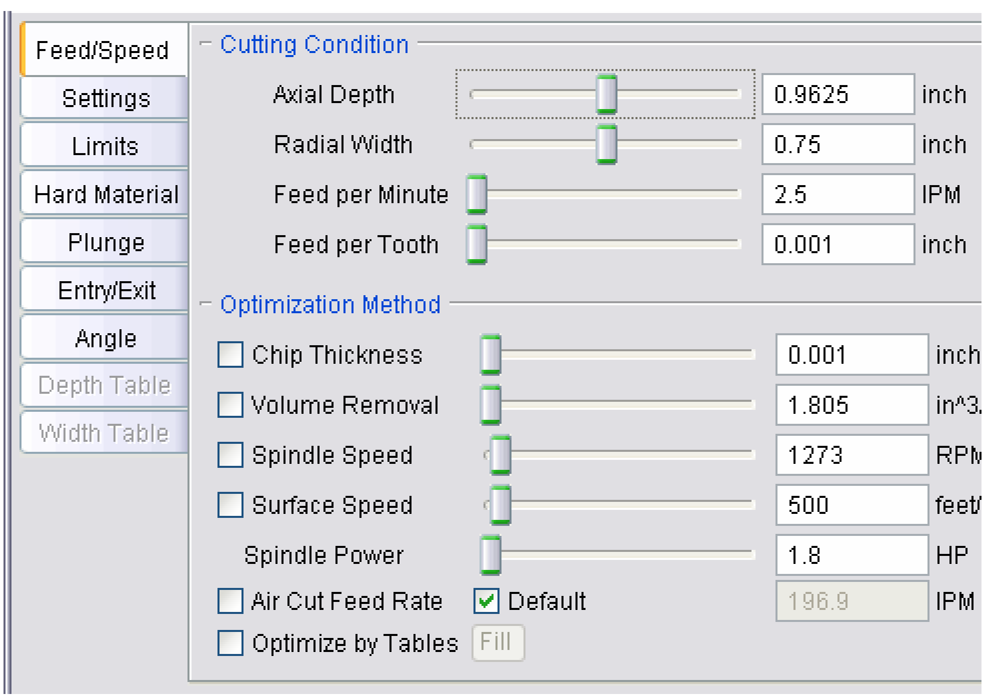
Location: **Project menu** > **Tools:** **Add menu** > **OptiPath** > **New**, or

Edit menu > **Modify** (when an OptiPath tool component is highlighted)

Features on this tab are used to describe and test various cutting conditions to optimize, as well as control how optimization will be performed for the selected tool. As methods are selected, some of the features become locked (unavailable for editing). However, they respond to show you the relationships between feed per tooth, RPM, cutter shape, and feed rate through various cutting conditions. In general, this is a three step process:

1. Configure the features in the **Cutting Condition group** to represent a known successful cutting condition which is typical for the tool.
2. Select the desired optimization method(s).
3. Again use the **Cutting Condition group** to test how optimization will be performed.

After configuring the features on this tab, be sure to review the other tabs in the OptiPath window for proper optimization values.



Cutting Condition group:

This group of features is used to describe and test various machining conditions to optimize. You can enter or adjust the sliders to set the desired values. As values change, the cutter graphic is updated to reflect the current cutting condition. The unit of measure for the values is determined by the units selected on the Settings tab.

NOTE: If the sliders don't go far enough to reach the desired value or the value in the corresponding data field turns red, then another optimization setting is limiting input. If all settings on the **Feed/Speed** tab look okay, then check values on the **Settings** tab, such as **Maximum/Minimum Cut** feed rates.

Axial Depth / Radial Width — Depth and width of cut.

Feed per Minute — Feed rate calculated from feed per tooth, spindle speed, and number of cutting teeth on the cutter.

Feed per Tooth — Thickness of material or "chip load" removed by each tooth on the cutter.

Optimization Method group:

This group of features is used to control how optimization will be performed for the selected tool. Note that all methods optimize air cuts (tool motions not in contact with material). However, the **Air Cut Feed Rate** method optimizes only the cuts entirely in air.

Tips:

1. Watch the white message area in the middle of the **OptiPath window** for hints on using Optimization Method features as you select them on the Feed/Speed tab.
2. When **Chip Thickness** and **Volume Removal** methods are combined (both selected), VERICUT calculates optimized feed rates for each, then uses the feed rate having the smaller value. This has the affect of protecting you against excessive chip thickness or volume conditions that can occur using only one of these optimization methods.

Chip Thickness — When selected, optimizes cutting by flat, ball, and bull nose endmills based on maintaining a specified chip thickness. This method increases the feed rate when width of cut is less than 50% cutter diameter, or when depth of cut is less than the tool corner radius. Good for optimizing high speed machining and for semi-finishing and finishing operations using carbide cutting tools where the danger of "chip thinning" exists. It is especially effective for mold and die applications machining hard materials. Use the slidebar or enter the chip thickness in the data field. If unknown, unselect all optimization methods and use the **Cutting Condition** sliders to calculate it. For more information on chip thinning, see "**The Chip Thinning Problem**" in the *Using VERICUT* section. *Using VERICUT* can be found in the *CGTech Help Library*.

Volume Removal — When selected, optimizes based on a constant volume of material to be removed. This option is good for optimizing tools that encounter widely varying cutting conditions, such as hogging or roughing operations that cut at different cut depths and widths. Use the slide bar or enter the volume removal rate (cubic units) in the data field. If unknown, unselect all optimization methods and use the **Cutting Condition** sliders to calculate it.

Spindle Speed — When selected, outputs spindle speed values that support optimized feed rates. This value is also considered for **Cutting Condition** calculations (see above). Use the slide bar or enter the spindle speed (revolutions per minute) in the data field.

Surface Speed — When selected, optimizes feed rates and spindle speeds based on constant surface speed (CSS). Good for optimizing finish machining by ball or bull-nosed endmills and high speed machining, this option varies spindle speeds to maintain a constant surface speed at the maximum tool contact diameter. Feed rates are adjusted to maintain a constant feed per tooth ("chip load") in material. Use the slide bar or enter the CSS value in feet/minute (FM) for inch tool paths, meters/minute (MM) for metric tool paths. If unknown, unselect all optimization methods and use the **Cutting Condition** sliders to calculate it. Use features on the **Spindle tab** to limit RPM fluctuations and specify the feed per tooth load to be maintained.

NOTE: CSS optimization is for use on high speed NC machines capable of varying spindle speed quickly without harming spindle motor hardware.

Spindle Power — Power required that is required to remove material. The value entered differs depending on the active units for the OptiPath record:

Inch units => horsepower required to remove 1 cubic inch of material

Metric units => kilowatts required to remove 1 cubic centimeter of material per second

Air Cut Feed Rate — This option is useful for reducing time of proven tool paths, without affecting cutting feed rates and resulting surface finishes. When the checkbox is cleared, this feature controls the feed rate used by all other optimization methods to optimize air cuts.

Default option — when selected, the air cut feed rate is determined as follows:

G-Code tool path = Max Feed Velocity of the machine's X-axis.

APT tool path = Fast Feed value.

To specify an air cut feed rate value, clear the **Default** checkbox and enter a value in the data field. The feed rate entered should reflect the maximum speed at which machine axes can move simultaneously in feed mode (e.g. G01).

Optimize by Tables — When selected, optimizes based on specified feed rates for various cut depths, widths, and angles. This option is good for optimizing tools used in hogging or roughing operations where more control over cutting feed rates for various cut depths, widths, or angles is desired. Use the features on the **Depth Table**, **Width Table**, and **Angle** tabs to configure settings for this method.

Fill — Automatically fills table data fields with values calculated using the **Cutting Condition** sliders. The following values are supplied:

> **Depth Table** => defines five equally spaced cut depths based on cutter height and a cutting feed rate for each, based on volume of material removed. Also supplies the calculated volume removal rate for the value for **Max.Volume Removal Rate**.
>
> **Width Table** => calculates a percentage of calculated feed rate for each **% Width** value based on volume of material removed.

To OptiPath window

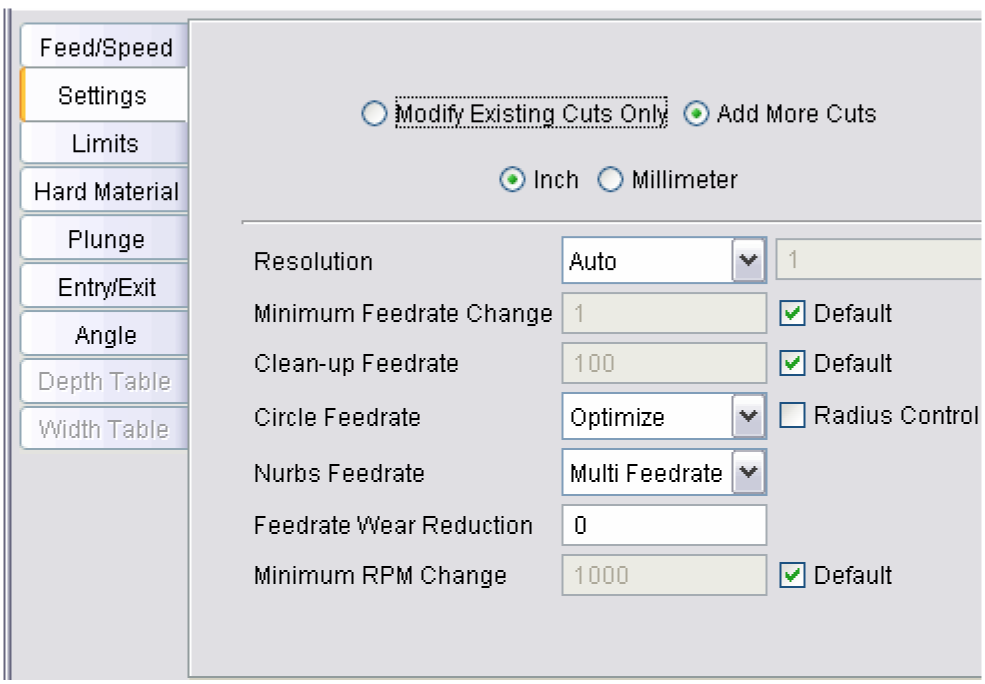
OptiPath window, Settings tab

Location: **Project menu > Tools:** **Add menu > OptiPath > New**, or

Edit menu > Modify (when an OptiPath tool component is highlighted)

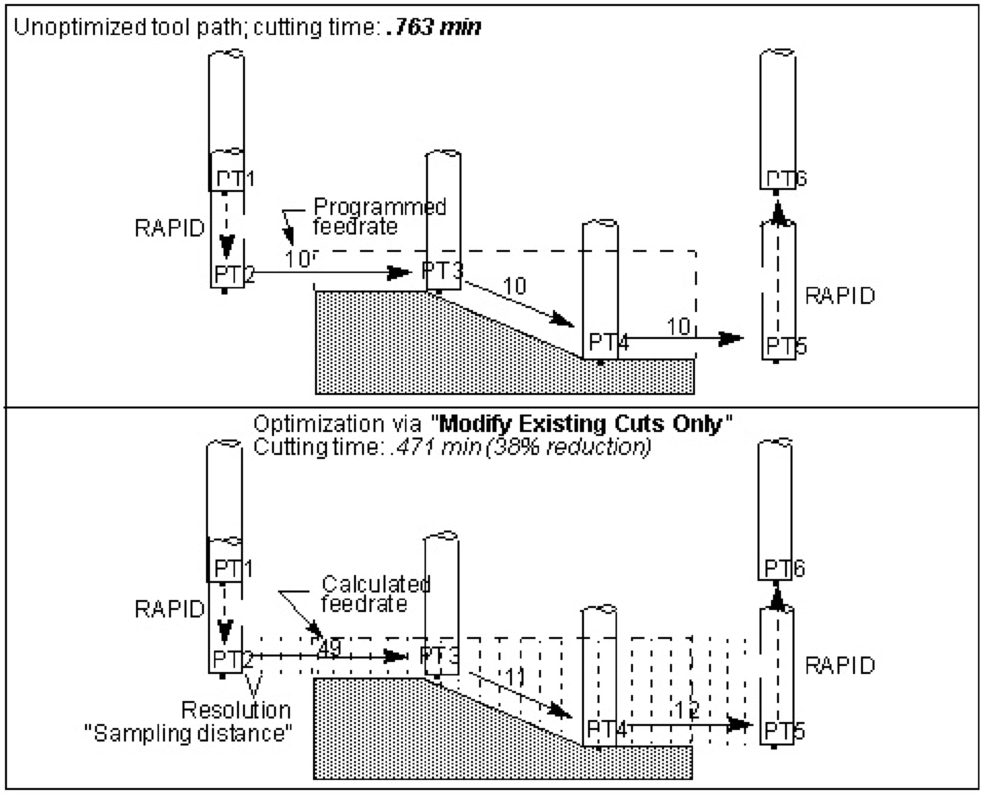
Features on this tab specify important settings for the optimization process, such as: the OptiPath record units, when to add (interpolate) more cuts, resolution, minimum/maximum optimized feed rates, how to optimize circle and NURBS motions, and more.

After configuring the features on this tab, be sure to review the other tabs in the OptiPath Manager window for proper optimization values.



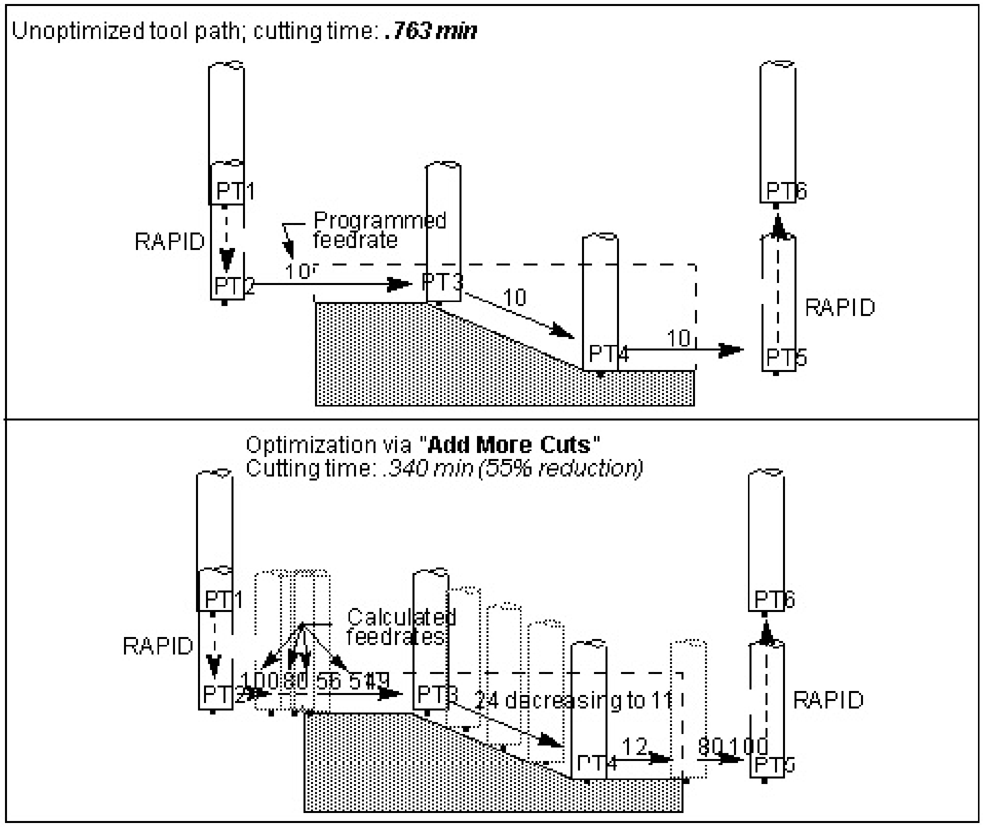
Modify Existing Cuts Only — When selected, this option calculates the most efficient feed rate for each motion. VERICUT uses the **Resolution** value (see below) to divide each motion into samples and calculates an optimum feed rate for each sample. The feed rate calculated for the sample with the most severe cutting conditions is retained as the optimized feed rate for that motion. This option is recommended when disk space is limited for storing optimized tool path data, or when tool positions are relatively close together. A tool path optimized with this choice has more efficient feed rates and the same or slightly more records than the original.

Sample optimization results using the Modify Existing Cuts Only option:



Add More Cuts — Similar to **Modify Existing Cuts Only** except motions are divided up when cutting conditions warrant feed rate changes. A tool path optimized with this choice has very efficient feed rates, but many more records than the original. The **Minimum Feedrate Change** value (see below) controls the quantity of feed rates output to the optimized tool path file. This option is recommended when disk space is ample, or when optimizing widely varying cutting conditions, such as: intermittent cutting, deep versus shallow cuts, etc.

Sample optimization results using the Add More Cuts option:



Inch / Millimeter options — Controls the unit measurement system in which optimization data is entered. If the OptiPath record units are different from the session units, the record values are converted appropriately when loaded.

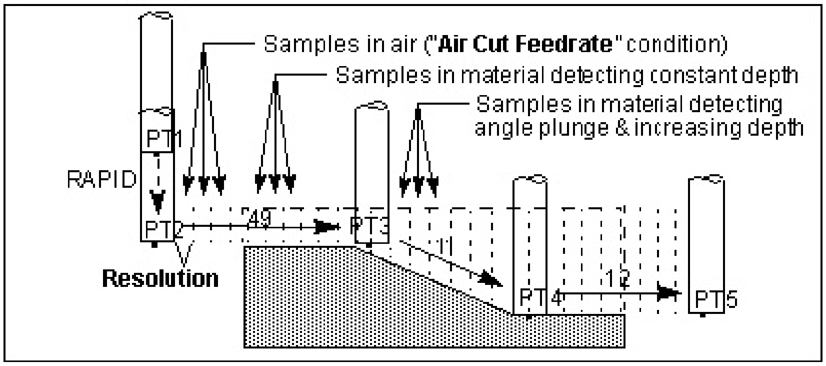
Resolution — Controls the frequency or "sampling distance" used to analyze each tool path motion. Each feed rate controlled motion is partitioned into samples based on this distance. The samples are then analyzed during feed rate calculation.

Options:

Distance — Uses the sampling distance in the corresponding data field. Distance is measured along the tool motion. Using a resolution equal to 1/2 to 1/4 of the tool diameter is recommended.

Auto — Automatically calculates the resolution distance based on the size of the cutting tool relative to the displayed model. This choice is recommended when little or no knowledge of cutting methods or feed rates is available.

Example of Resolution applied when optimizing tool path motions:



Minimum Feedrate Change — Specifies the minimum change from the current optimized feed rate that will cause a new optimized feed rate to be output. This feature controls the quantity of feed rates output to the optimized tool path file. A small value causes more optimized feed rates to be output than when a larger value is entered.

Default option — When selected, sets the default Minimum Feedrate Change = 1 IPM or 25 MMPM, whichever applies.

Clean-up Feedrate — Feed rate used when the tool is adjacent to, but not removing material. This condition is commonly referred to as a "spring pass".

Default option — When selected, sets the default clean-up feed rate as follows:

G-code tool path => 50% of the Max Feed Velocity of the machine's X-axis.

APT tool path => 50% of the Fast Feed value.

Tip: Set the **Clean-up Feedrate** to be less than the **Air Cut Feedrate**, since actual cutting leaves small amounts of material from tool deflection.

Circle Feedrate — Controls when and how feed rates are optimized for circular motions (e.g. G02-3). Circle feed rate optimization produces a single optimized feed rate for the entire circular motion regardless of the active Modify Existing Cuts Only or Add More Cuts option.

Options:

Programmed — Uses the original programmed feed rate.

Fixed — Uses the feed rate in the corresponding data field.

Optimize — Calculates an optimized feed rate based on the maximum material removed throughout the circular motion.

Break-up — OptiPath will attempt to break up circles only if both this option and the **Add More Cuts** option has been selected.

For APT, only one output format is supported:

CIRCLE/ xc, yc, zc, i, j, k, rad - defines circle center, plane and radius

GOTO/x,y,z,ijk - defines the ending point of the motion

Circle feed rates are not optimized under the conditions listed below. These circles receive the original programmed feed rate.

- helical (spiral) circle motions
- circle motion where the tool axis is not perpendicular to the circle axis
- G-code data circle motion programmed in inverse time feed rate mode (e.g. G93)

Radius Control — When toggled On, OptiPath adjusts the feedrate for circular motions. During circle motions, OptiPath determines the maximum contact point away from the tool center line and adjusts the feed rate so the desired feed rate is achieved at that point. **Radius Control** is only available when **Circle Feedrate** is set to **Optimize** or **Break-up**.

NOTE: If the effective cutting radius, R, is bigger than the circle radius, r, OptiPath adjusts (lowers) the feedrate by the factor r/R which is < 1. If R is less than r (r/R > 1), no adjustment is done (OptiPath will not increase the feedrate calculated based on the tool center).

NURBS Feedrate — Controls when and how to optimize NURBS motion feed rates (e.g. G06, G06.2). The active **Modify Existing Cuts Only** or **Add More Cuts** option does not affect feed rates calculated by this feature.

Options:

Programmed — Uses the original programmed feed rate.

One Feedrate — Calculates the most efficient feed rate for NURBS motions. The feed rate calculated for the most severe cutting condition used as the optimized feed rate.

Multi Feedrate — Calculates multiple feed rates for NURBS motions when cutting conditions warrant feed rate changes.

Feedrate Wear Reduction — Reduces the calculated feed rate based on the amount of material that has been removed. The value entered is the percentage of reduction to be applied after each cubic unit of material removed. Zero indicates no adjustment for wear.

Example: assume "**Feedrate Wear Reduction**=.2". The calculated feed rate is reduced .2% after each cubic unit of material has been removed. After five cubic units removed the calculated feed rate is reduced 1% (.2% x 5).

Minimum RPM Change — Specifies the minimum change from the current optimized spindle speed required to output a new spindle speed. This feature controls the quantity of spindle speed changes output to the optimized tool path file. Small **Minimum RPM Change** values, cause more spindle speed changes to be output than larger values.

Tip: Start with a value such that only 3-4 changes can occur across the RPM range.

Default option — When selected, sets the default minimum RPM change = 1000.

To OptiPath window

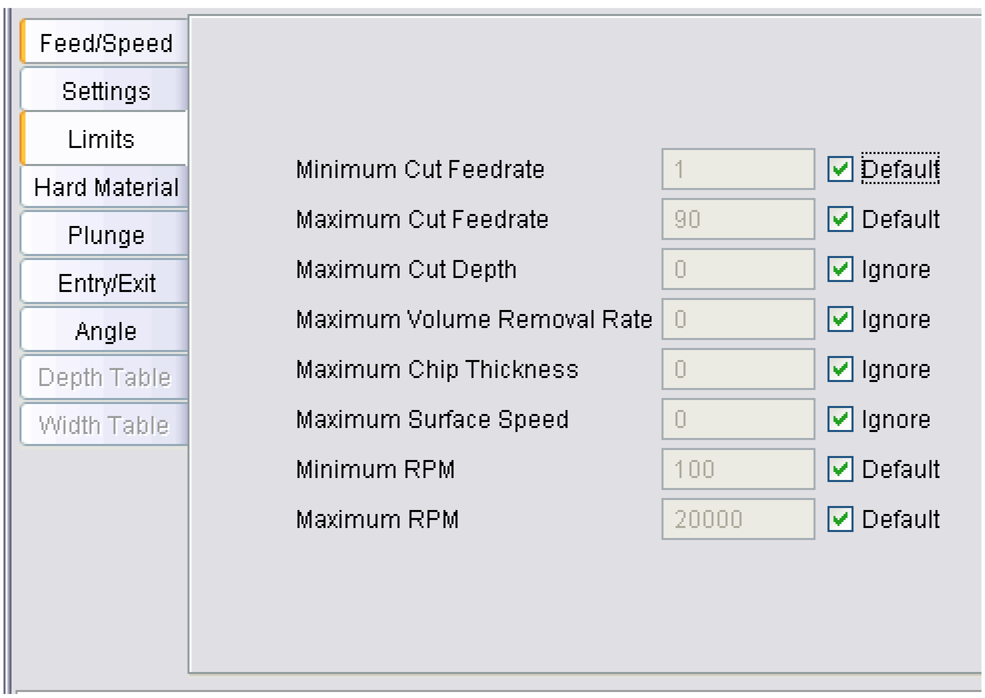
OptiPath window, Limits tab

Location: **Project menu > Tools:** **Add menu > OptiPath > New**, or

Edit menu > Modify (when an OptiPath tool component is highlighted)

The features on this tab enable you to specify limits for cutting characteristics (feedrate, cut depth, volume removal rate, chip thickness, surface speed and RPM) for milling cutters and drills. It ***does not*** apply to turning tools.

After configuring the features on this tab, be sure to review the other tabs in the **OptiPath Manager window** for proper optimization values.



Minimum Cut Feedrate — Specifies the minimum optimized feed rates that can be output when removing material.

Default option — When selected, sets the default minimum feed rate = 1 IPM or 25 MMPM.

Maximum Cut Feedrate — Specifies the maximum optimized feed rates that can be output when removing material.

Default option — When selected, sets the default maximum feed rate as follows:

G-Code tool path => 45% of the Max Feed Velocity of the machine's X-axis.

APT tool path => 45% of the Fast Feed value.

Maximum Cut Depth — Specifies the maximum cut depth that can be output when removing material.

Ignore — When selected, the **Maximum Cut Depth** limit is ignored.

Maximum Volume Removal Rate — Specifies the maximum volume removal rate that can be output when removing material.

Ignore — When selected, the **Maximum Volume Removal Rate** limit is ignored.

Maximum Chip Thickness — Specifies the maximum chip thickness that can be output when removing material.

Ignore — When selected, the **Maximum Chip Thickness** limit is ignored.

Maximum Surface Speed — Specifies the maximum surface speed rate that can be output when removing material.

Ignore — When selected, the **Maximum Surface Speed** limit is ignored.

Minimum RPM — Specifies the minimum spindle speed that can be output during optimization.

Default option — When selected, sets the default minimum RPM as follows:

G-Code tool path => machine spindle minimum set by **SpindleSpeedMin** macro, if not set then 100 RPM.

APT tool path => 100 RPM.

Maximum RPM — Specifies the maximum spindle speed that can be output during optimization.

Default option — When selected, sets the default maximum RPM as follows:

G-Code tool path => machine spindle maximum set by **SpindleSpeedMax** macro, if not set then 20000 RPM.

APT tool path => 20000 RPM.

To OptiPath window

OptiPath window, Hard Material tab

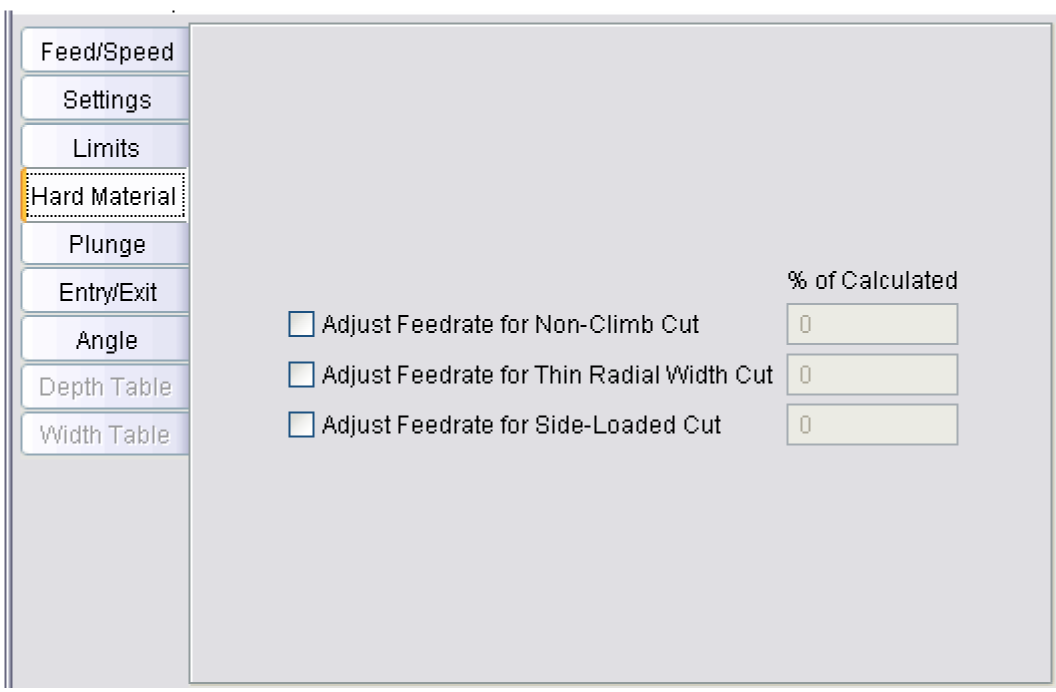
Location: **Project menu > Tools:** **Add menu > OptiPath > New**, or
Edit menu > Modify (when an OptiPath tool component is highlighted)

Features on this tab enable you to specify feedrate adjustment factors for special cutting conditions encountered during the optimization process

Feedrates may need to be adjusted (typically slower than what OptiPath would normally calculate) based on the following 'hostile' conditions when cutting hard materials.

These hostile conditions generally occur when cutting hard or difficult to machine materials such as tool steel, stainless steel, and high temperature alloys. Since there is no good scientific or empirical information to apply to these hostile conditions, the following features enable you to adjust the feed rates under each of these Hard Material conditions.

After configuring the features on this tab, be sure to review the other tabs in the **OptiPath Manager window** for proper optimization values.



Each feature consists of a checkbox to toggle the feature On/Off, and a numeric field enabling you to enter a factor to be applied to the calculated feed rate for each of these conditions. The number represents a percentage of the calculated feed rate.

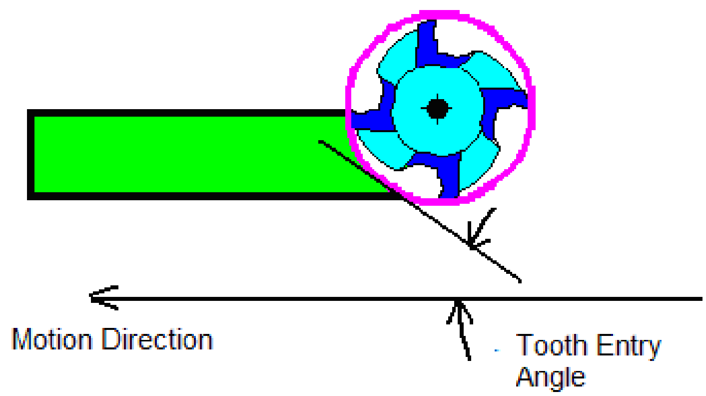
When a feature is toggled "On", the factor is applied to the calculated feedrate each time that the particular condition is detected.

Since these cutting conditions are not mutually exclusive, any combination (or all) of the features can be toggled "On" at the same time and any, or all, of the factors may be applied to the same calculated feed rate as appropriate for the cutting conditions.

After configuring the features on this tab, be sure to review the other tabs in the OptiPath Manager window for proper optimization values.

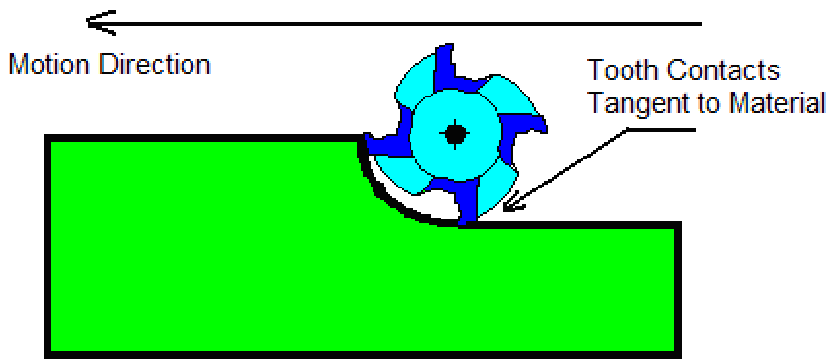
Adjust Feedrate for Non-Climb Cut —

Non-Climb, or "Conventional", cutting occurs when the cutter tooth enters material close to parallel to the direction of motion. See the following diagram:



The cutting condition worsens as the tooth entry angle decreases. When the tooth entry angle is near zero the cutting edge contacts material parallel to the direction of motion and requires excessive force to push the edge into material. The tooth finally enters material when pressure between the cutter and material increases and the tooth rotates past the tangent contact.

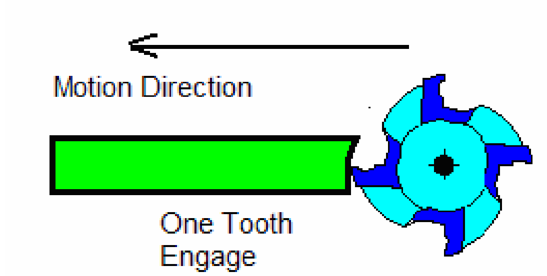
The picture below shows the worst condition.



Adjust Feedrate for Thin Radial Width Cut —

A thin radial width cut, typically only has one tooth engaged in material. When the width of material becomes so small that only one tooth of the cutter is engaged, the tool is no longer well supported and an otherwise reasonable feed rate becomes too aggressive. As the spindle spins the tool transitions between a tooth engaged and cutting, and no tooth engaged, causing the tooth to "bang" into material each time it enters. This "banging" causes premature cutting edge breakdown and reduces tool life.

Determining the number of teeth engaged depends on the number of teeth in the cutter, the material contact axial depth, the width and location of the contact, and the flute helix angle.



Adjust Feedrate for Side-Loaded Cut —

A side-loaded cut, is a condition where material is exclusively on the "climb" side of the cutter

When the tool is cutting with all material on the right of the tool center (for a clockwise rotating spindle), the cutting pressure forces the tool to deflect in the opposite direction. The deflection seriously affects cutting performance, increases cutting forces, and causes premature edge breakdown.

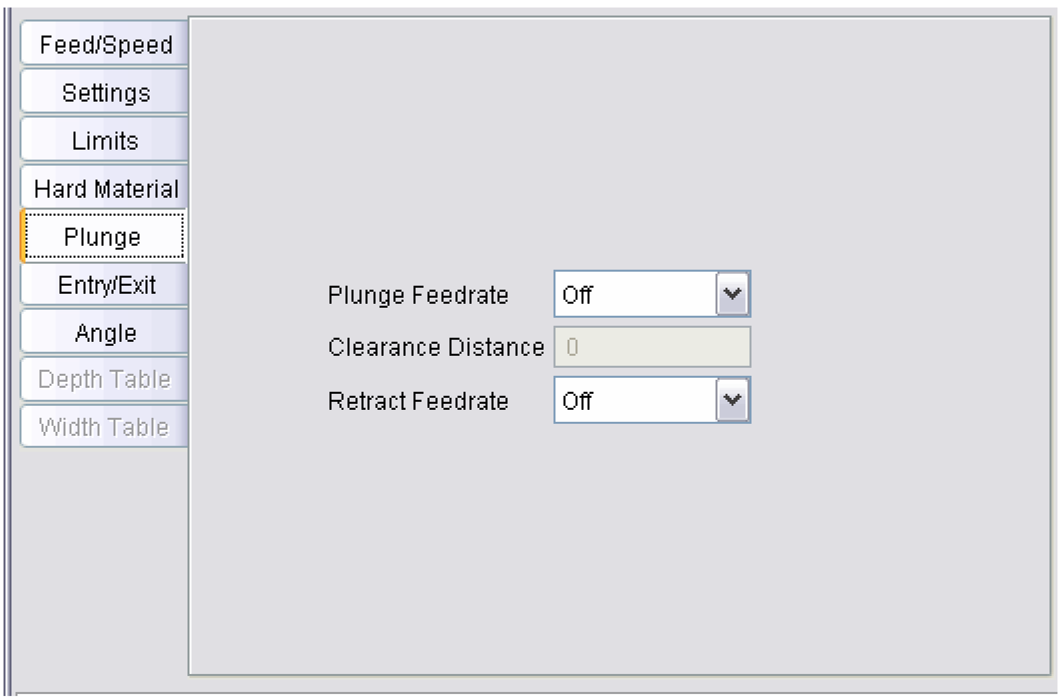
To OptiPath window

OptiPath window, Plunge tab

Location: **Project menu** > **Tools:** **Add menu** > **OptiPath** > **New**, or **Edit menu** > **Modify** (when an OptiPath tool component is highlighted)

The features on this tab control the optimized feed rate for motions along the tool axis that plunge into or retract from material.

After configuring the features on this tab, be sure to review the other tabs in the OptiPath Manager window for proper optimization values.



Plunge Feedrate — Controls when and how feed rates are calculated for plunging along the tool axis into material.

Options:

Off — Disable plunge feed rate control.

Feed/Minute — Uses the feed rate entered in the corresponding data field.

% of Prog. — Adjusts programmed feed rates based on the percentage entered in the corresponding data field. 100% uses the programmed feed rate as is, 50% cuts the feed rate in half, etc.

NOTE: Plunge Feedrate overrides **Angle table** feed rate for 90 degree cutting angle (plunge motion), or the **Entry Feedrate** for entry on plunge motions.

Clearance Distance — Specifies the distance from material to instate the feed rate for plunging.

Retract Feedrate — Controls when and how the feed rates are calculated for retracting along the tool axis away from material.

Options:

Off — Disable retract feed rate control.

Feed/Minute — Uses the feed rate entered in the corresponding data field.

% of Prog. — Adjusts the programmed feed rate based on the percentage entered in the corresponding data field. 100% uses the programmed feed rate as is, 50% cuts feed rate in half, etc.

NOTE: Retract Feedrate overrides **Angle table** feedrate for -90 degree cutting angle (retract motion).

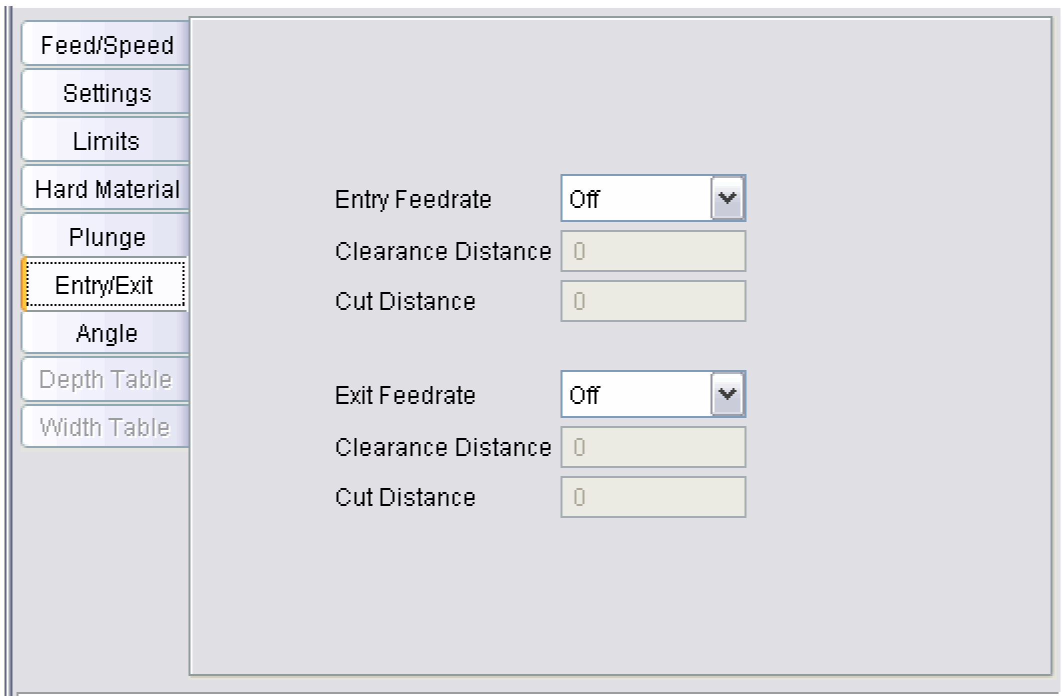
To OptiPath window

OptiPath window, Entry/Exit tab

Location: **Project menu > Tools:** **Add menu > OptiPath > New**, or
Edit menu > Modify (when an OptiPath tool component is highlighted)

The features on this tab control the optimized feed rate for entering and exiting material. The feed rate takes affect at a specified distance from material contact and remains in affect until the tool has cut a specified distance into material, or the entry motion ends.

After configuring the features on this tab, be sure to review the other tabs in the OptiPath Manager window for proper optimization values.



Entry Feedrate — Use to specify when, and how, feed rates are calculated for entering material.

Options:

Off — Turns off **Entry** (or **Exit**) **Feedrate** calculation and application during optimization.

Feed/Minute — Uses the feed rate entered in the corresponding data field.

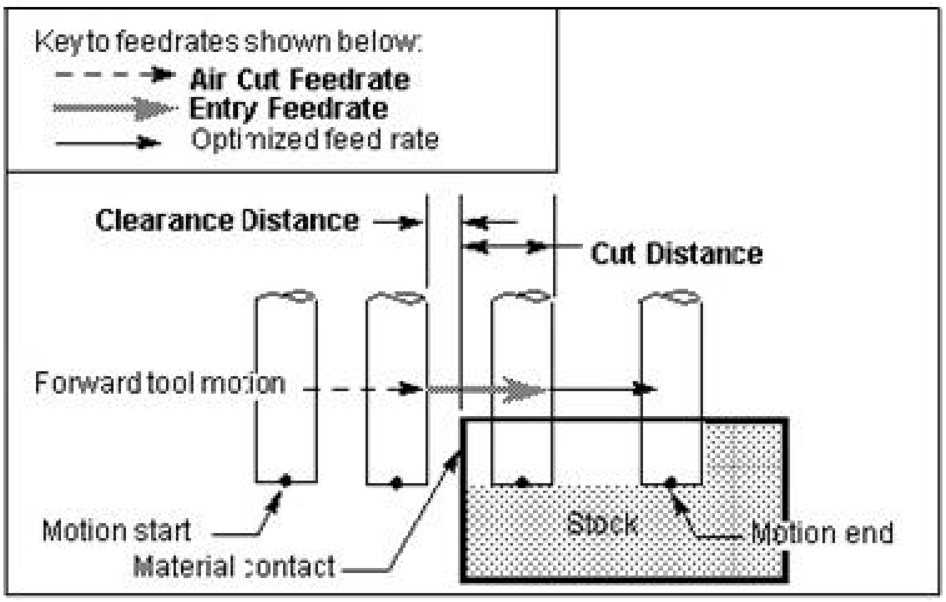
% of Prog. — Adjusts the programmed feed rate based on the percentage entered in the corresponding data field. 100% uses the programmed feed rate as is, 50% cuts feed rate in half, etc.

% of Calc. — Similar to **% of Prog.**, except adjusts the calculated feed rate.

Clearance Distance — Use to specify the distance before entering material to start applying the specified **Entry Feedrate**.

Cut Distance — Use to specify the distance to cut into material, using the **Entry Feedrate**, before normal feed rate optimization resumes.

Effect of Entry Feedrate applied on a motion entering material:

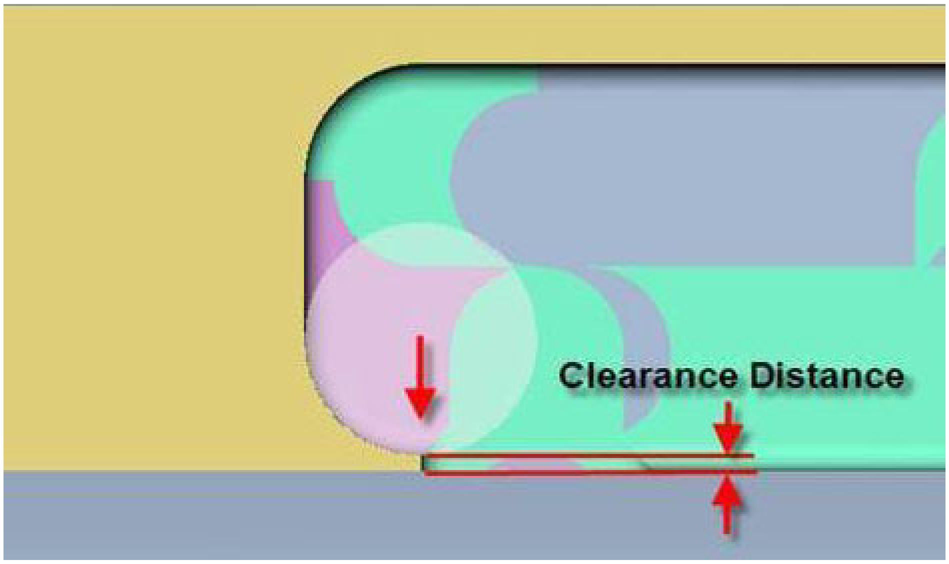


NOTES:

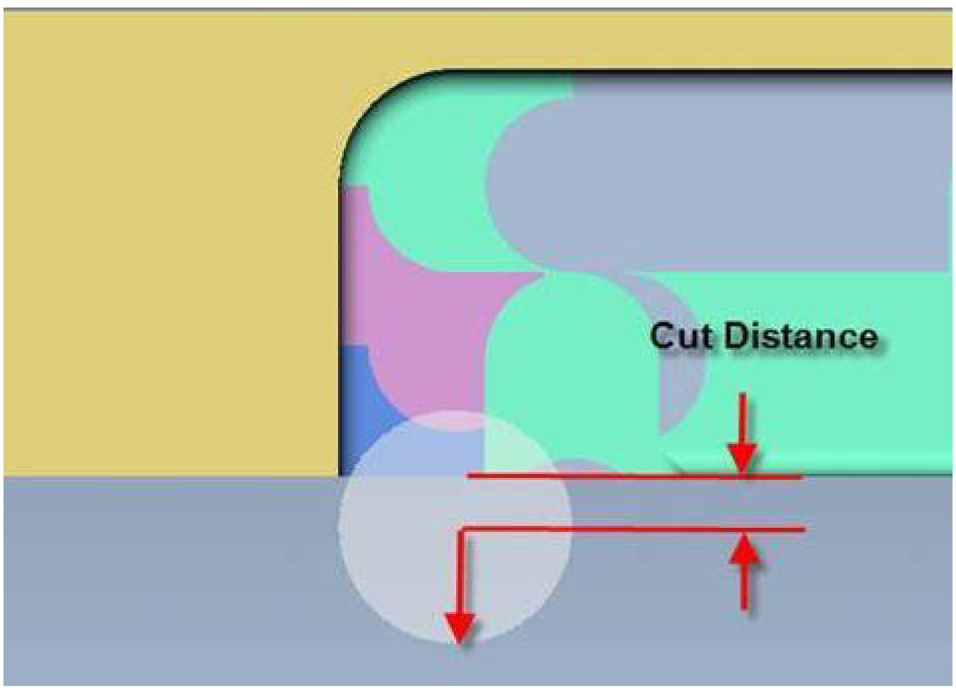
1. When entry motions are optimized with the **Modify Existing Cuts Only** option, calculated entry feed rates are considered among other cutting conditions to determine the appropriate feed rate for motions which enter material.
2. **Plunge Feedrate** overrides entry controlled feed rate for entry on plunge motion.

Exit Feedrate — Use to specify when, and how, feed rates are calculated for exiting material. Options are same as described above for Entry Feedrate.

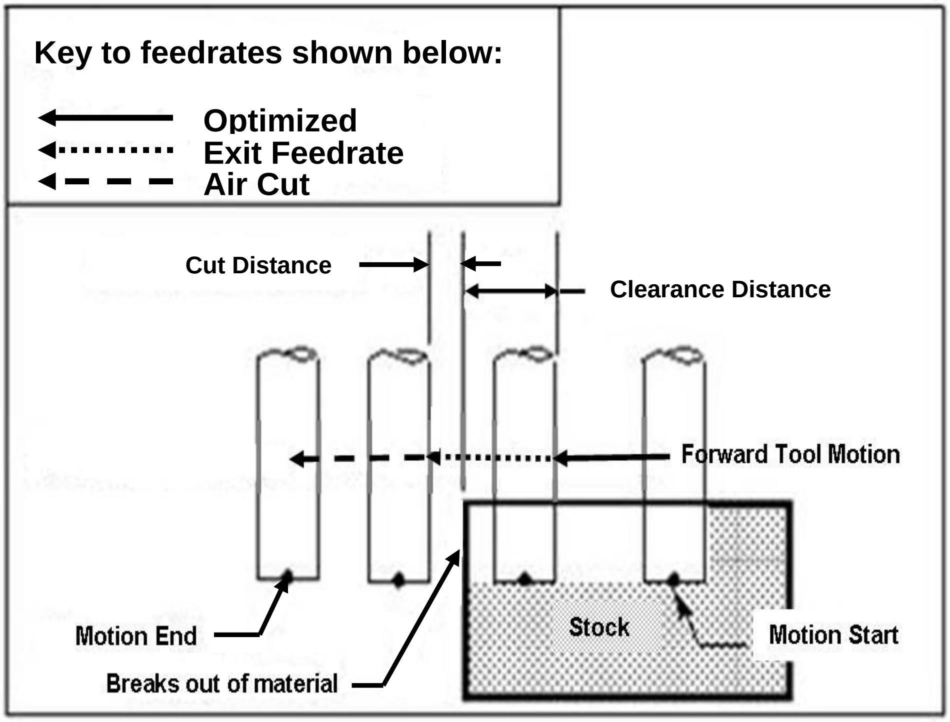
Clearance Distance — Specifies the distance before breaking out of material to start using the **Exit Feedrate**.



Cut Distance — Specifies the distance after breaking free from material to continue using the **Exit Feedrate**, after which the **Air Cut Feedrate** begins.



Effect of Exit Feedrate applied on a motion exiting material:



[To OptiPath window](#)

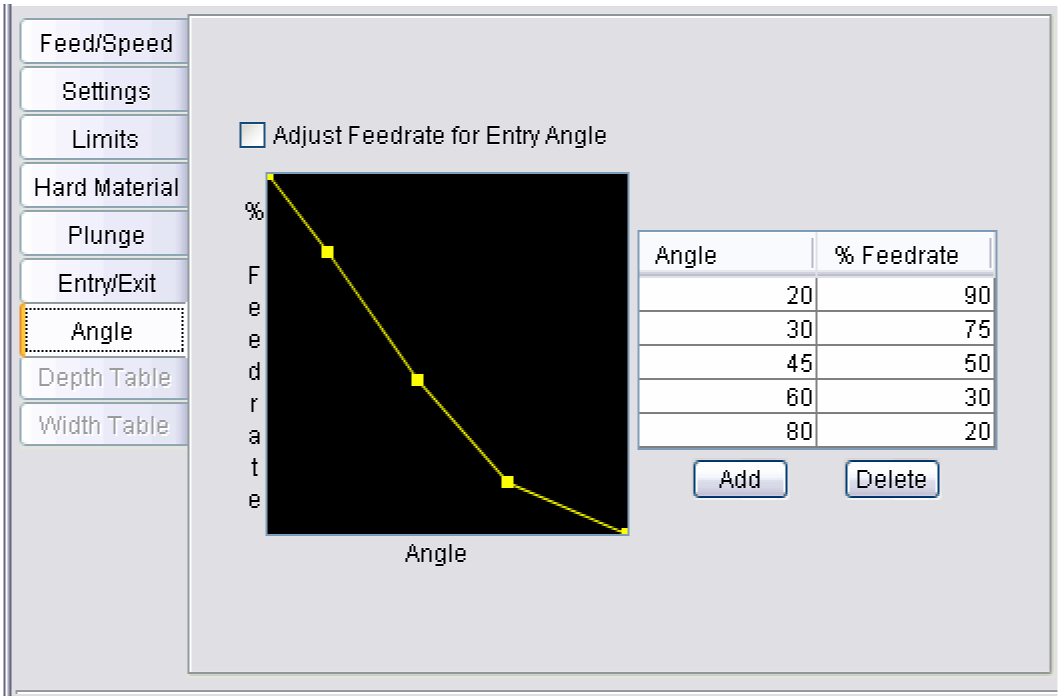
OptiPath window, Angle tab

Location: **Project menu > Tools:** **Add menu > OptiPath > New**, or

Edit menu > Modify (when an OptiPath tool component is highlighted)

The features on this tab adjust the optimized feed rate for 3-axis linear cuts ramping in material. While these features are normally used to reduce the feed rate for ramping downward, they can also be used to affect feed rates for upward ramping motions.

After configuring the features on this tab, be sure to review the other tabs in the OptiPath Manager window for proper optimization values.



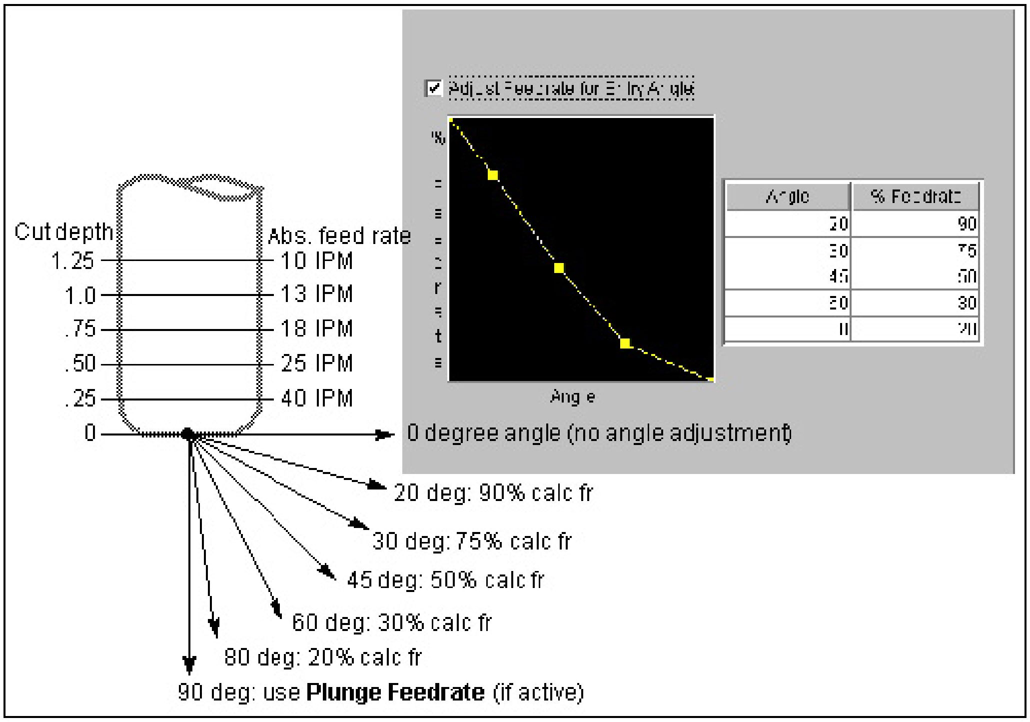
Adjust Feedrate for Entry Angle — When active, adjusts feed rates for ramping motions based on the table configuration.

Angle table — Table of cut angles and corresponding % feed rate adjustments to apply when the tool ramps into material. Values can be entered or supplied via dragging the control points on the graph. Add after, or delete table entries via right-clicking on them and choosing the appropriate option (or use the **Add** / **Delete** buttons below the table). Cuts at angles less than the minimum listed in the table receive the adjustment specified for the minimum angle. Similarly, when the maximum specified cut angle is exceeded, the adjustment specified for the maximum angle is used.

Angle column — Lists up to five cut angles (in descending order) that describe the range of expected cut angles to adjust. Values can be from -90 to 90, where "0" represents horizontal milling, "90" represents plunge motion along the tool axis, and "-90" represents retract motion.

% Feedrate column — Lists % feed rate adjustments to apply to calculated feed rates when ramping in material at the corresponding angle. 100% uses the calculated feed rate as is, 50% cuts feed rate in half, etc.

Sample Angle table:



NOTES:

1. Angle adjusted feedrate for 90 degree cutting angle is overridden by Plunge Feedrate if active.
2. Planar milling motions (0 degree angle) are not adjusted.

To OptiPath window

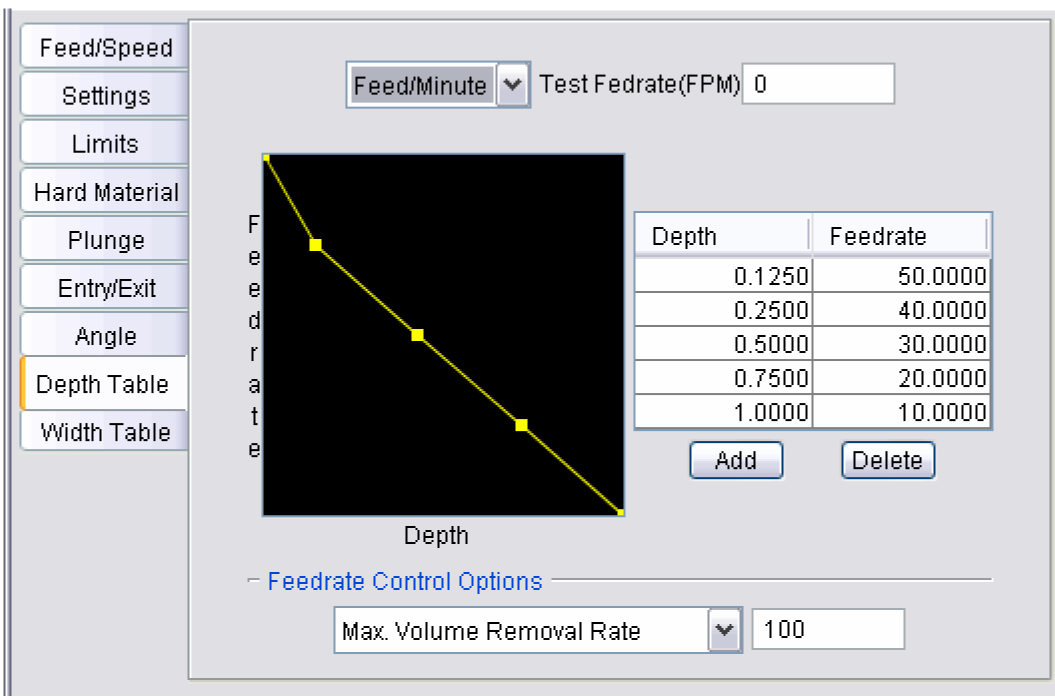
OptiPath window, Depth Table tab

Location: **Project menu > Tools:** **Add menu > OptiPath > New**, or

Edit menu > Modify (when an OptiPath tool component is highlighted)

The features on this tab controls how optimized feed rates are calculated for cutting at various depths during Optimize by Tables optimization. The calculated feed rate can be used as calculated, or adjusted using the **Angle** and/or **Width** tables, as well as features on other tabs in this window.

After configuring the features on this tab, be sure to review the other tabs in the OptiPath Manager window for proper optimization values.



Feedrate value type option list — Controls the how to interpret the feed rates in the Depth table.

Options:

Feed/Minute — Calculates optimized feed rates based on absolute feed rates specified for various cut depths. Specify feed in units specified on the Settings tab.

% of Prog. — Calculates optimized feed rates based on percentages of programmed feed rates specified for various cut depths. "100" uses the programmed feed rate as is, "50" cuts the feed rate in half, etc.

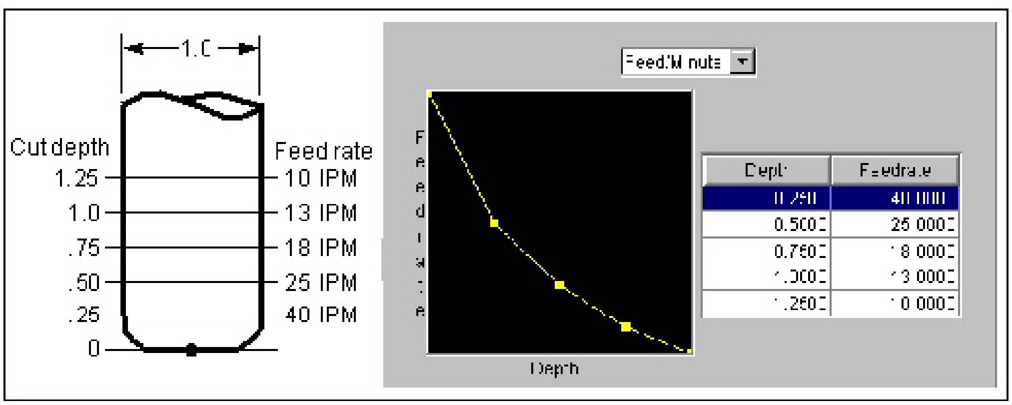
Feed/Tooth — Calculates optimized feed rates based on feed per tooth, or chip load specified for various cut depths. Specify feed per tooth in units used by the tool path file (IPR or MMPR).

Depth table — Table of cut depths and corresponding feed rates to apply when cutting at a specified depth. Values can be entered or supplied via dragging the control points on the graph. Add after, or delete table entries via right-clicking on them and choosing the appropriate option (or use the Add / Delete buttons below the table). Cuts at depths less than the minimum listed depth in the table receive the feed rate specified for the minimum depth.

Depth column — Lists up to five cut depths (in descending order) that describe the range of cutting by the tool.

Feedrate column — Lists feed rates to apply when the tool is cutting at the corresponding depths. Feed rate values are interpreted differently based on the active feed rate value type option (see "Feedrate value type option list" above).

Sample Depth table:



Tip: Un-select all optimization methods and use the **Cutting Condition** sliders to determine appropriate feeds and speeds for a known successful cutting condition, then select **Optimize by Tables** and press **Fill** to automatically enter Depth Table values.

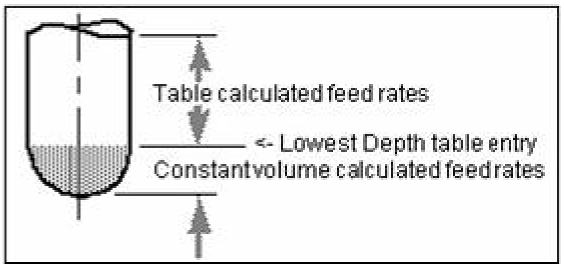
Feedrate Control Options — Controls when and how volume removal rate is considered in feed rate calculations during optimization by tables.

Options:

Off — Disable volume removal rate considerations.

Const. Volume Below Min. Depth — Specifies a constant volume of material is to be removed when the depth of cut is less than the lowest Depth table entry. The volume removal rate is expressed in cubic units, where “units” refers to the units specified on the **Settings tab**. If unknown, un-select all optimization methods and use the Cutting Condition sliders to calculate it.

The example below shows the affect of **Constant Volume Below Minimum Depth** on feed rate calculations for a ball-nose endmill when the lowest Depth Table entry is equal to the radius of the tool.



Max. Volume Removal Rate — Specifies the maximum volume removal for the tool. Optimized feed rates are bounded such that the rate at which material is removed never exceeds this value. The volume removal rate is expressed in cubic units, where "units" refers to the units specified on the Settings tab. If unknown, un-select all optimization methods and use the Cutting Condition sliders to calculate it.

To OptiPath window

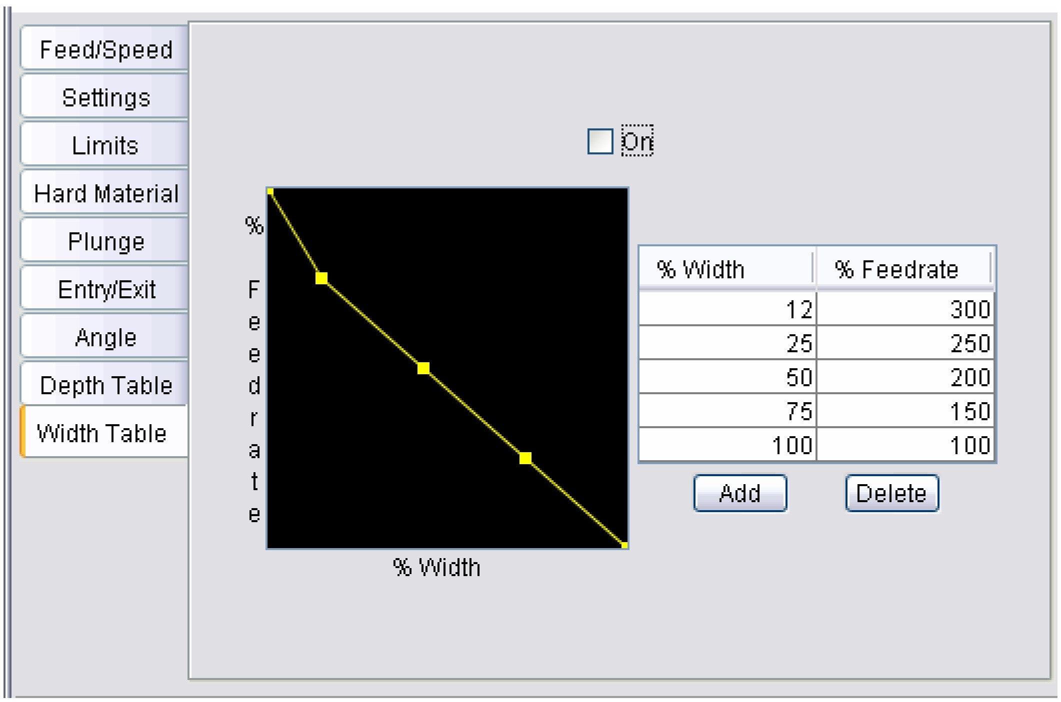
OptiPath window, Width Table tab

Location: **Project menu > Tools:** **Add menu > OptiPath > New**, or

Edit menu > Modify (when an OptiPath tool component is highlighted)

The features on this tab adjust the optimized feed rate for cuts that are less than full cutter width during Optimize by Tables optimization. When the tool is cutting with less than full diameter the feed rate can typically be higher than for a full width cut, also know as a "slot cut".

After configuring the features on this tab, be sure to review the other tabs in the OptiPath Manager window for proper optimization values.



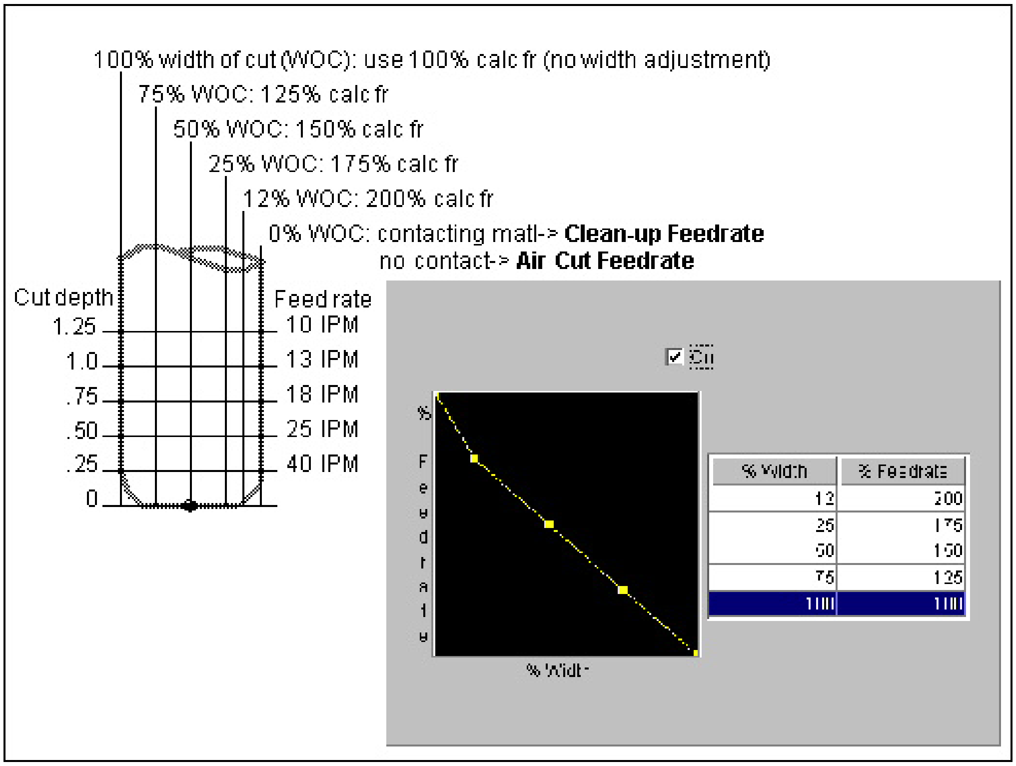
On — When selected, adjusts feed rates for cuts of varying width based on the **Width table** configuration.

Width table — Table of cut widths and corresponding % feed rate adjustments to apply when removing material with the specified cutter width. Values can be entered or supplied via dragging the control points on the graph. Add after, or delete table entries via right-clicking on them and choosing the appropriate option (or use the **Add** / **Delete** buttons below the table). Cuts at widths less than the minimum listed in the table receive the adjustment specified for the minimum width.

% Width column — Lists up to five cut widths (in increasing order) that describe the range of expected cut widths to adjust.

% Feedrate column — Lists % feed rate adjustments to apply to calculated feed rates when removing material with the corresponding cut width. 100% uses the calculated feed rate as is, 50% cuts feed rate in half, etc.

Sample Width table:

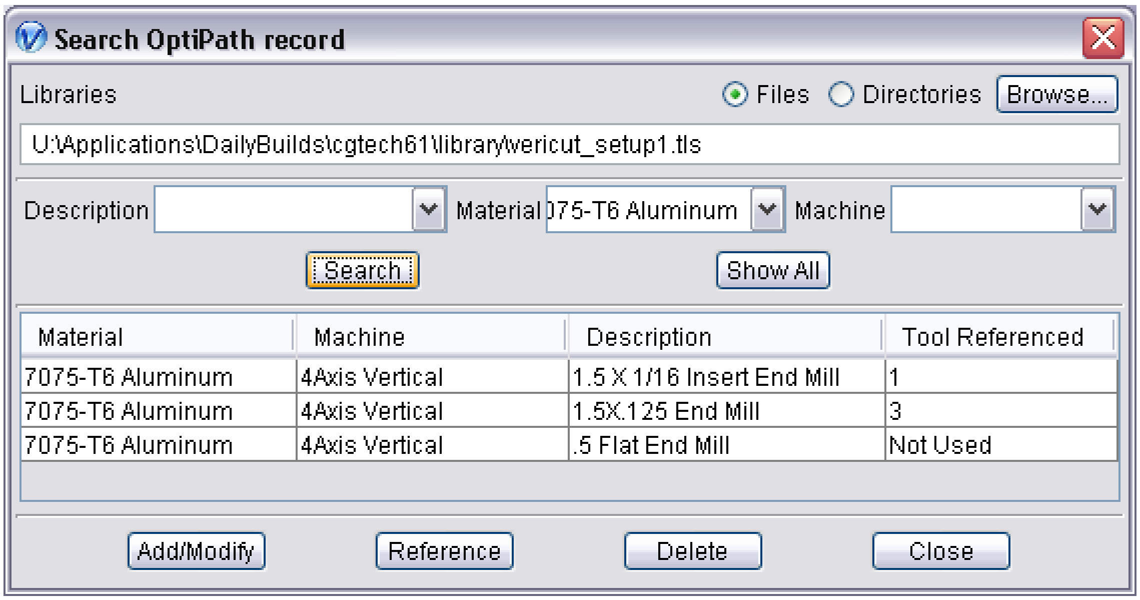


To OptiPath window

Search OptiPath Record window

Location: **Project menu > Tools:** **Add menu > OptiPath > Find Existing** or
Edit menu > Search OptiPath

Search OptiPath Record enables you to search existing tool/OptiPath libraries for records with specific attributes.



Libraries:

Files — Use to specify one, or more, tool/OptiPath library files to be searched for OptiPath Records meeting the specified search criteria.

Directories — Use to specify one, or more, directories of tool/OptiPath library files for OptiPath records meeting the specified search criteria.

Browse — Displays a file selection window to assist you in selecting the files/directories to be searched.

NOTE: By default, if no library file(s)/directories are specified, VERICUT searches the "current" Tool Library file.

Search Attributes:

Description — Search for existing OptiPath records containing a specified string of text within the record's Description field.

Material — Search for existing OptiPath records containing a specific material type.

Machine — Search for existing OptiPath records containing a specific machine type.

Search — Start the search of the specified library files/directories, based on the search attributes specified above. OptiPath records found matching the search attributes are written to the OptiPath Record List.

Show All — Displays all OptiPath records found in the specified library file(s)/directories in the OptiPath Record List.

OptiPath Record List:

Displays a list of Optipath records found during the search. For each record in the list, the **Material**, the **Machine** description, and the tool **Description**, are displayed. The **Tool Referenced** column is only used for OptiPath records in the "current" tool library.

Add/Modify — Opens the OptiPath window, populated with the data from the selected OptiPath record, ready for modifying. If no changes are required, select **Add** in the Optipath window to add the OptiPath record to the "current" tool library.

Reference — Creates a reference to the selected OptiPath record in the "current" tool library. This feature is grayed out for OptiPath records in external files.

Delete — Enables you to delete the selected OptiPath record in the "current" tool library. This feature is grayed out for OptiPath records in external files.

Close — Closes the Search OptiPath Record window.

Processing Options

Activate

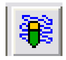
Location: **Project menu > Processing Options > Activate**

Activate enables you to set the state of the current setup to active/inactive. An active state is indicated by a check mark.

Motion window

VERICUT Users:

VERICUT Location: **Project menu > Processing Options > Motion**

VERICUT toolbar short cut: 

Mold and Die Users:

Mold and Die Location: **Advanced Options page > Simulation Settings**

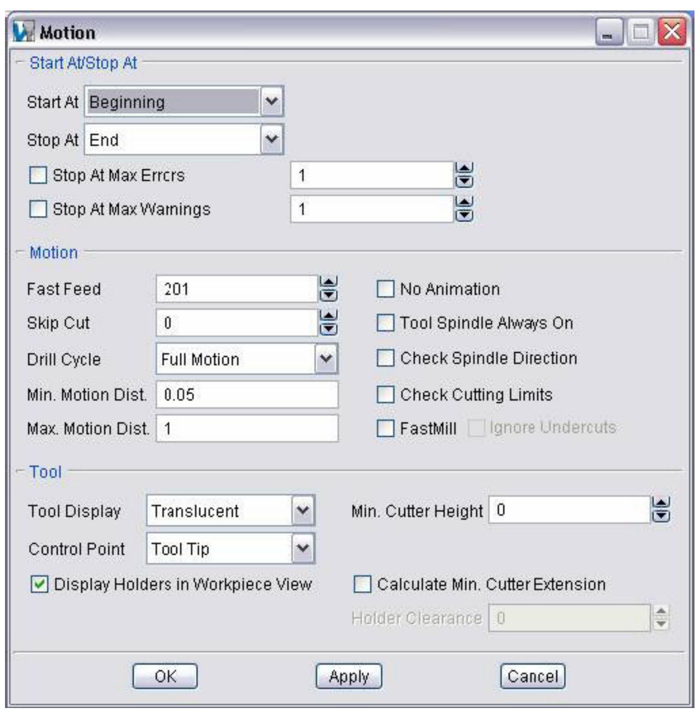
Notebook Feature: 

Cutter Grinder Users:

Cutter Grinder Location: **Advanced Options page > Simulation Settings**

Notebook Feature: 

Opens a window used to control the simulation of tool path and machine motions.



Start At/Stop At group:

Start At — Specifies the line, or tool path record at which cutting simulation starts. VERICUT fast forwards from the beginning of the NC program to the specified line/record while internally processing intermediate tool path records to store values for the current feed rate, spindle speed, coolant, tool description, and tool location.

Options are:

Beginning — Start processing at the beginning of the first file.

Line Number — Start processing at the specified line number of selected toolpath file.

NOTE: A list of all NC programs and job subroutine files defined for the current job will display enabling you to select the toolpath file.

Line Number (Directly) — Start processing at the specified line number without internally processing intermediate tool path records. This line number only applies to the first toolpath (or the first active toolpath if **Use Selected Files**, in the **Project >NC Program window**, is being used to select only specific toolpath files from the list for VERICUT processing).

Text — Start processing on the line that contains the specified text.

Tool ID — Start processing when the specified tool is loaded.

Stop At — Provides options for stopping tool path processing. When applicable, enter supporting "stop at" text or values in the field next to this feature. Text entered is not case sensitive.

End — Stop processing at the end of the last toolpath file.

Num of Cuts — Stop processing after the specified number of cuts.

Line Number — Stop processing at the specified line number of the selected toolpath file.

NOTE: A list of all toolpath and job subroutine files defined for the current job will display enabling you to select the toolpath file.

Text — Stop processing on the line that contains the specified text.

Tool Change — Stop processing when a tool change occurs.

Program Stop — Stop processing when a "program stop" record is encountered in the tool path. Examples of such records include: M0 (G-Code tool paths) or STOP.

Optional Stop — Stop processing when an "optional stop" record is encountered in the tool path. Examples of such records include: M1 (G-Code tool paths) or OPSTOP.

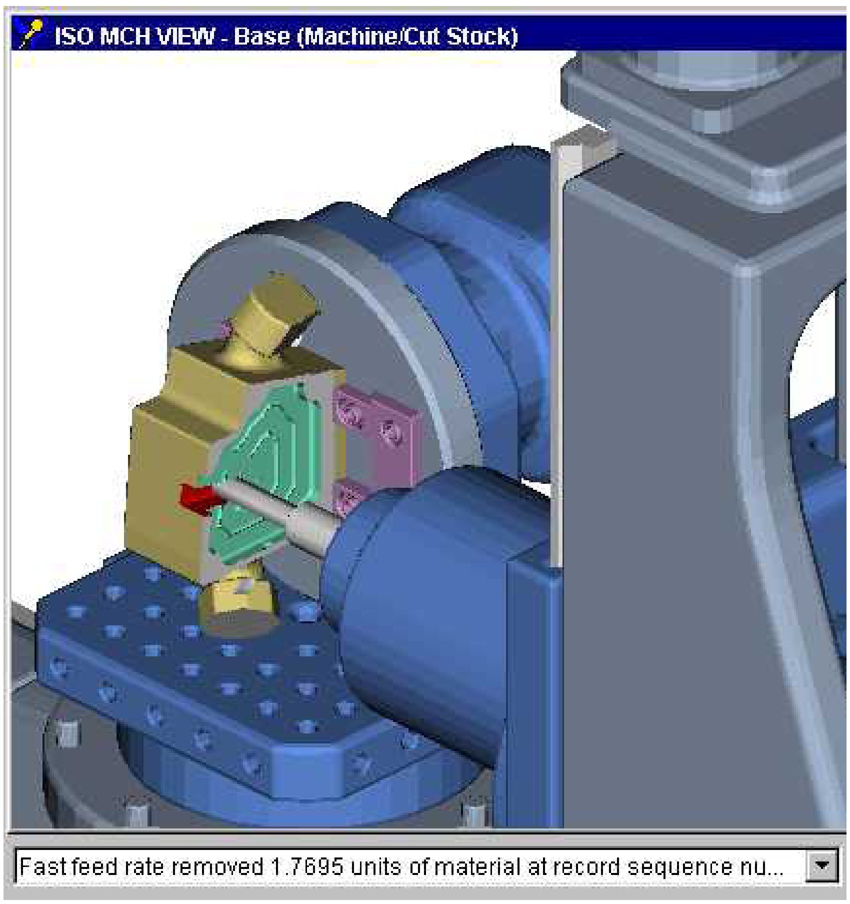
End of each File — Stop processing at the end of each toolpath file.

End of each Setup — Stop processing at the end of each setup.

Stop At Max Errors / Stop At Max Warnings — Number of errors (or warnings) which, if detected, stops tool path processing. Toggle "On" and specify the Max Error/Max Warning value.

Motion group:

Fast Feed — Feed rate threshold at which material removal is unsafe (likely to cause tool breakage, machine malfunction, or part damage). For example, if the maximum feedrate allowed is 180, Fast Feed should be set to 181 or greater. Material removed under these conditions is shaded in the Error color (typically red) and causes a "Fast feed rate" error.



No Animation — Enables you to reduce processing time. When toggled "On", the graphics display is not updated until either processing is complete or you **Stop** the processing. At that time the cut model is displayed in it "final" state or the state that it was in when processing was stopped. You can also toggle **No Animation** On/Off using the **No Animation** icon, , in the VERICUT toolbar.

Skip Cut — Quantity of animated tool motions to skip before updating the display. Enter "0" to display each tool motion. Skipping cuts increases animation speed since the display is not constantly updated. The cut model is unaffected.

Tool Spindle Always On — When toggled On, VERICUT will assume that the spindle is always on and will not check for spindle status (on/off) or output error messages related to cutting with the spindle off.

Drill Cycle — Controls when and how to simulate canned tool axis cycles (e.g. G8n, "CYCLE"). While VERICUT simulates most popular cycle motions, there are also capabilities to add more and change how existing cycles are interpreted. See "**About configuring for cycle simulation**" in the *Using VERICUT* section for more information. *Using VERICUT* can be found in the *CGTech Help Library*.

Options:

Ignore — Ignores cycle motions- the tool moves to each cycle position without performing cycle motions.

Bottom Only — Simulates cycle motions for a hole with one full depth motion- pecking or chip-breaking motions are not simulated.

Full Motion — Simulates all cycle motions, including pecking or chip-breaking motions.

NOTE: When the "**Bottom Only**" or "**Ignore**" options are used, certain motions (and checks) are ignored. This means that there will be no RAPID checks, no material removal, no limit checks, no collision checks, no redraws, etc. Because of it's importance, checks related to "RAPID to the R-plane" motions are included when "**Bottom Only**" is selected, but not when "**Ignore**" is selected.

Need help troubleshooting cycle motion errors? For assistance with diagnosing error messages and conditions, visit the VERICUT Users' Forum or contact CGTech technical support via our website, just click on the support link.

Check Spindle Direction — When toggled On, VERICUT check for spindle direction against the "valid" direction of the tool.

For turning tools, "valid" direction is based on the turning tool orientation. Spindle direction, (**CW** or **CCW**), is determined looking down the Z-axis of the spindle. It is assumed that revolving stock material should come onto the insert in the direction of positive Y-axis of the tool component. If spindle direction is wrong, or spindle speed is not defined, an error is reported to VERICUT logger. Only one error per spindle is reported for each operation turning the spindle On or Off.

The error is reported only if turning tool removes material from stock. For air moves, no errors are reported. Also, if an insert removes material in a motion that crosses the spindle Z-axis, no error is reported (in this case, one part of the insert is always on the correct side, while the other part is on the wrong side, therefore there is no preferable spindle direction).

For milling tools, the direction of the tool spindle (either spindle component or tool component attribute) is checked with the "valid" direction of the tool in the spindle. Tool spindle direction (**CW** / **CCW**) is defined looking down (negative Z) the tool axis. For revolved milling cutters, "valid" direction is based on the setting of Spindle Direction (clockwise or counterclockwise) in the tool definition. For inserted milling cutters, "valid" direction is based on the insert thickness direction.

If the direction of the spindle does not match the direction of the tool, then an error is reported whenever the tool removes material. The tool removes material in the error color (similar to the behavior for turning spindle direction error). The check is done and the error reported only when the tool removes material. Motion that does not remove material does not report an error.

The check is done for G-Code only.

Min. Motion Dist. / Max. Motion Dist. — Enables you to specify a minimum and maximum value, defining the range of the **Animation Speed slider**. The animation speed is interpolated between these two values.

Check Cutting Limits — Toggle On, to have VERICUT do the cutting limit checks during simulation. Cutting limit values are set in the **OptiPath window, Settings tab**.

FastMill — Toggle "On" to process the toolpath in FastMill mode. You can also use the FastMill icon , in the Toolbar to turn on FastMill mode.

NOTES:

1. FastMill:
   - ***does*** "fixed axis" material removal in any tool orientation.
   - ***does not*** do "undercuts". FastMill automatically switches to standard material removal when required.
   - can be turned On/Off at any time.
   - can be used with/without tool animation. Toggle **No Animation**, described above, "On" for fastest processing.
   - is compatible with OptiPath, AUTO-DIFF, Model Export, as well as X-Caliper and any other feature that uses the cut model database.
2. FastMill ***is not*** effective when:
   - The tool axis changes frequently.
   - The cutter "undercuts", such as when using a dovetail or wheel cutter.
   - Cutter height is less than the cut depth resulting in an undercut condition.
3. **Replace Material When Stepping Back** (ref. **Replace Material When Stepping Back,** on the Properties window: General tab, in the File menu section, also in the *VERICUT Help* section) and **FastMill** mode are mutually exclusive.

When **Replace Material When Stepping Back** is toggled "On", VERICUT will turn **FastMill** "Off" Conversely, if **FastMill** is toggled "On", VERICUT will turn **Replace Material When Stepping Back** "Off. VERICUT will display a pop-up notification when these situations occur.

Ignore Undercuts — Works in conjunction with FastMill, described above. When toggled "On" (default), VERICUT does not check each motion for "undercut" conditions allowing for faster processing in those cases where you are sure that there are no "undercut" conditions in the toolpath. Toggle "Off" when you want VERICUT to check for "undercut" conditions as it processes each motion.

NOTE: This feature is only active when **FastMill** is toggled "On".

TIP: Place the cursor over the text field, pull-down menu, or checkbox, for any of the options in the **Motion Group** to see a summary of its function.

Tool group:

Tool Display — Controls when and how the tool is displayed. Options are: **Off** (do not display the tool), **Solid**, or **Translucent**. Material removal is not affected by this choice.

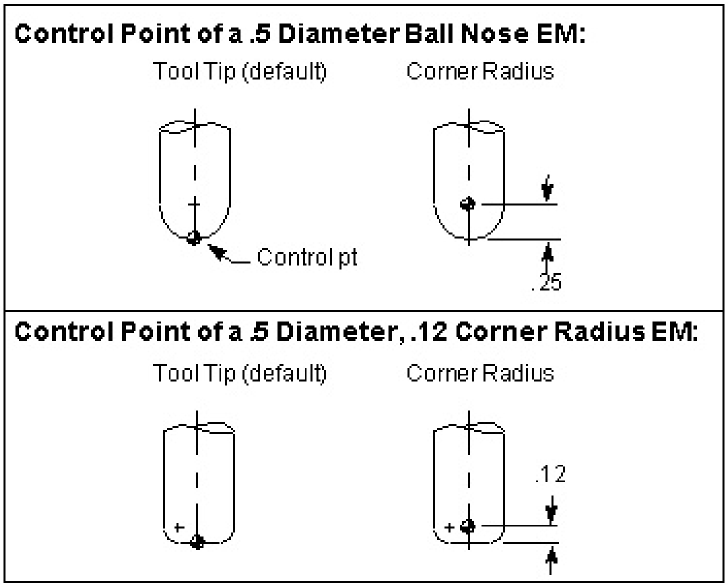
Min. Cutter Height — Default height used when the current tool description does not specify height, or when specified height is less than this value. Affects milling cutters only.

Control Point — Controls how milling tools are driven. By default, VERICUT drives the tool tip. Material is removed based on tool path motion commands and the tool shape, relative to the control point.

Options:

Tool Tip — Point located at the bottom-center of the cutter.

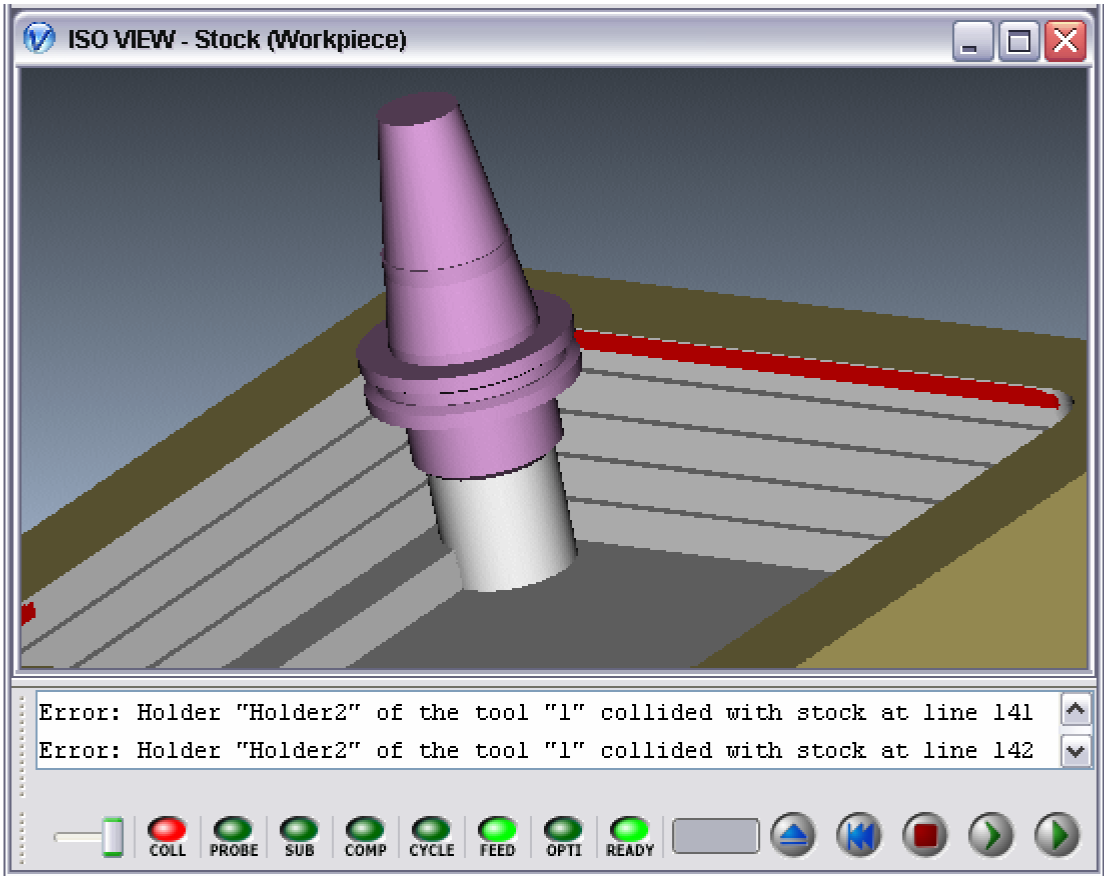
Corner Radius — Point located a distance equal to the cutter corner radius up from the tool tip. With ball nose endmills, this point is commonly referred to as the "ball center".



NOTES:

1. If Corner Radius is used with non-ball nosed endmills, the offset is calculated along the tool axis, from the tool tip to the center of the first arc encountered in the tool profile.
2. To specify a milling tool control point other the tool tip of corner radius, define tools via the Tool Manager and specify a control point offset via the tool's Properties.
3. This feature is not applicable to turning tools. Instead, define tools via the Tool Manager and specify a control point offset via tool Properties.

Display Holders in Workpiece View — When toggled "On", instructs VERICUT to display defined tool holders and detect collisions they may have. Holders are non-modal and are applied only to the active cutter. An error similar to "HOLDER "Holder2" of the tool "1" collided..." is output for tool holder collisions, the Collision status light displays red, and the collision area is shaded using the red Error color in the graphics area.



Calculate Min. Cutter Extension — When active, calculates the minimum height required to avoid shank/holder collisions with the stock and all fixtures in a Workpiece view, for both standard VERICUT, and FastMill mode.

When toggled on, all holders move down to the bottom of the cutter. The first holder and any holders above it are processed independently, so one sees both the first and subsequent holders move down to the same location and the cutter will not be seen. As cutting starts, the holders move up as needed to avoid a collision with stock.

If potential holder collisions are detected during the simulation, VERICUT extends the cutter to avoid the collision and adds this amount to the calculated height. The calculated extension amount is automatically applied to the existing Gage Point Z value.

Note that when these calculations are being performed, tool holder collisions are not reported since VERICUT extends the cutter to avoid the collision.

If desired, you can include additional clearance in the calculation via specifying **Holder Clearance** (see below). The final calculated height for each tool is displayed in the Log file Tool Summary section under the "Min" column heading.

Holder Clearance — Specifies clearance to add when calculating minimum cutter extension (see above). The clearance value is also included in the Log file minimum height values.

NOTE: This feature is only active when **Calculate Min. Cutter Extension** is toggled "On".

OK — Applies the changes and closes the Motion window.

Apply — Applies the changes and leaves the Motion window open.

Cancel — Closes the Motion window without applying changes.

G-Code (G-Code Settings window)

VERICUT Users:

VERICUT Location: **Project menu > Processing Options > G-Code > Settings**

VERICUT toolbar short cut: 

Mold and Die Users:

Mold and Die Location: **Advanced Options page > Job Settings**

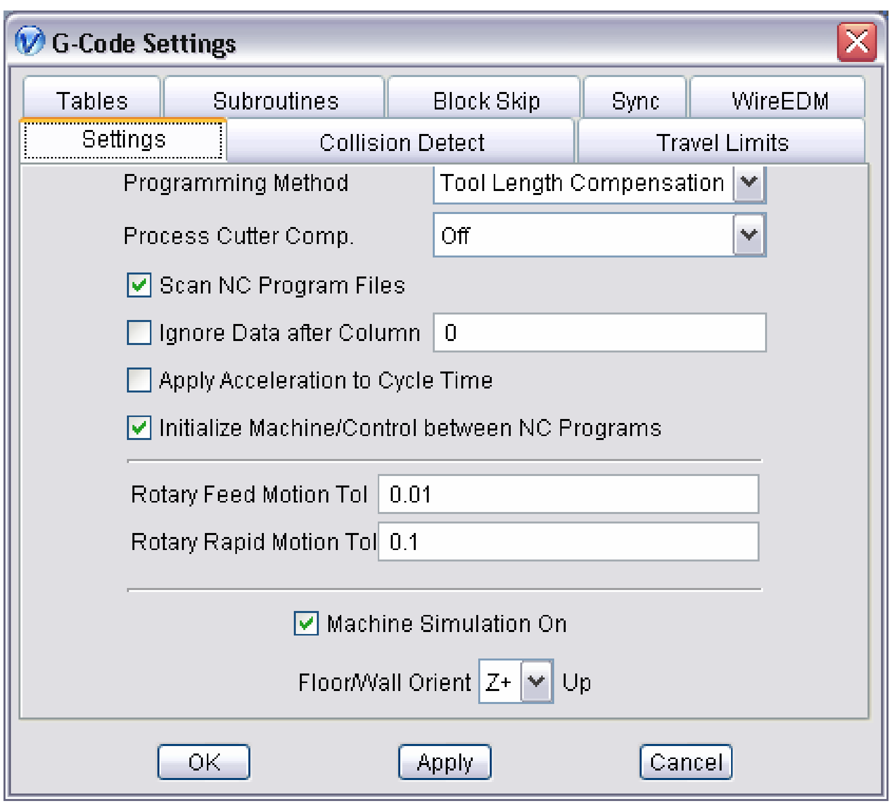
Notebook Feature: 

Cutter Grinder Users:

Cutter Grinder Location: **Advanced Options page > Job Settings**

Notebook Feature: 

Opens a window to configure settings to support processing G-Code NC program files.



Settings tab — Features on this tab provide important settings for G-Code simulation, such as: the G-Code programming method, when cutter compensation (CDC) codes are processed, tool path scanning, and tolerances used for simulating rotary motions.

Collision Detect tab — Features on this tab control when collisions between machine components are detected, which components are protected, and tolerances used for detecting collisions.

Travel Limits tab — Features on this tab define how far each machine axis can go, and control when travel limit errors are detected.

Tables tab — Features on this tab are used to specify work offsets and machine locations specific to the current tool path file(s) and to store tool-related offset and compensation register data.

Subroutines tab — The features on this tab are used to specify names of external files containing subroutines that can be accessed by VERICUT.

Block Skip tab — Features on this tab control which levels of block skipping are recognized, and specify the block skip character.

Sync tab — The features on this tab enable you to select which Sync Sub-system is to be used. This setting is only applicable if **Sync Method** on the Control Settings window, Sync tab is set to something other then **None**. This feature is intended as a debugging tool enabling you to easily select which sync program they want to run rather then running them together at the same time.

WireEDM tab — Features on this tab configure VERICUT for wire EDM simulations.

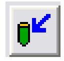
OK — Applies the changes and closes the G-Code Settings window.

Apply — Applies the changes and leaves the G-Code Settings window open.

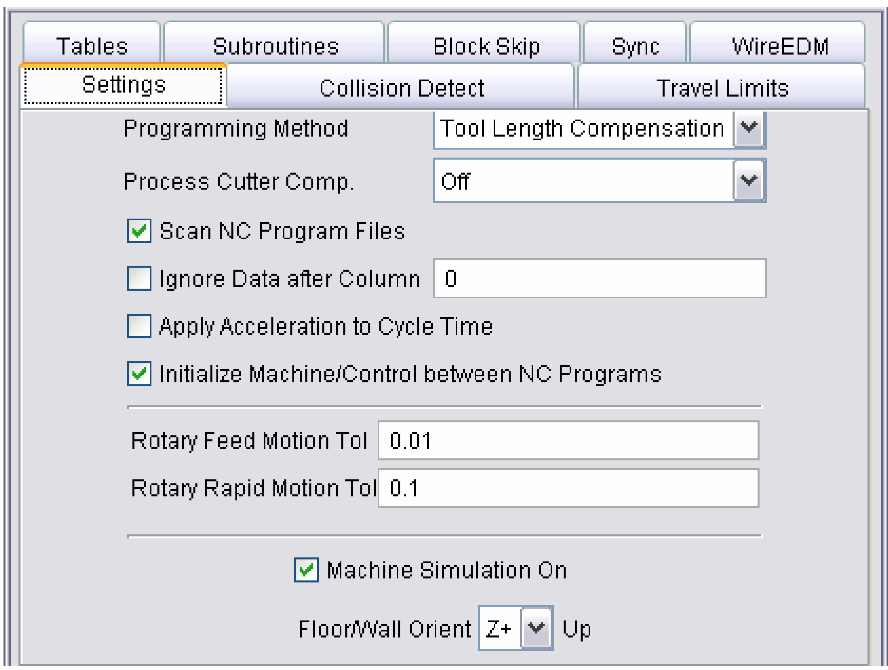
Cancel — Closes the G-Code Settings window without applying changes.

G-Code Settings window, Settings tab

VERICUT Location: **Project menu > Processing Options > G-Code > Settings**

VERICUT toolbar short cut: 

Features on the Settings tab provide important settings for G-Code simulation, such as: the G-Code programming method, when cutter compensation (CDC) codes are processed, tool path scanning, and tolerances used for simulating rotary motions.



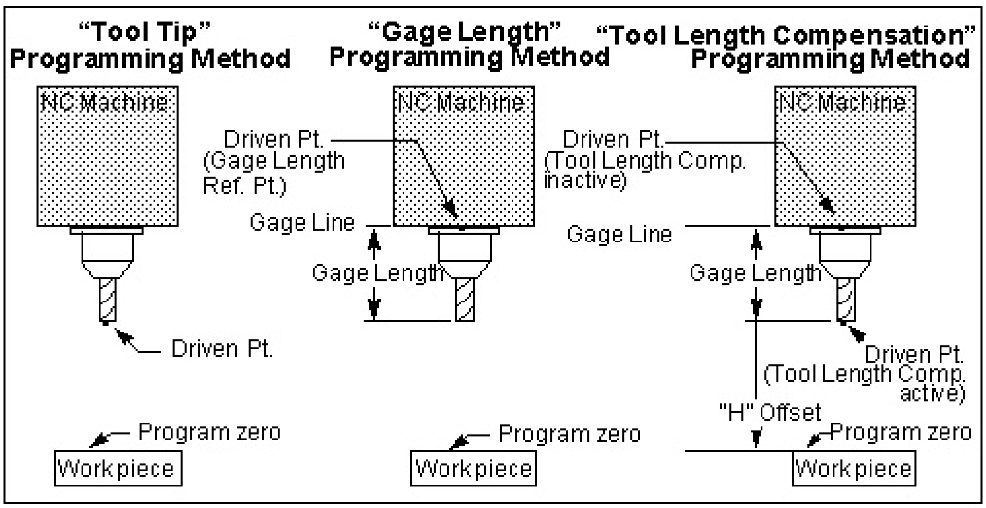
Programming Method — Describes the relationship between cutting tools and what is being driven by tool path file motion commands. (Take care ***NOT*** to describe the relationship between cutting tools and the driven point controlled by a *CAM system*.)
Options:

Tool Tip — XYZ values represent tool tip positions (center of the tool tip) at all times.

NOTE: Tool length compensation codes (G43ZnHn/G49) do not appear in tool tip programmed tool paths. If these codes are present, use the **Tool Length Compensation** programming method.

Gage Length — XYZ values represent gage point positions. The gage point is typically the spindle face, where the centerline of the tool intersects the machine gage line. Gage length values must be supplied using **Gage Offset** in **Tool Manager**, or via entries in a **Gage Offset table**.

Tool Length Compensation — With comp. active (e.g. G43ZnHn) XYZ values represent tool tip positions relative to program zero. With comp. cancelled XYZ values represent machine gage point positions relative to machine zero. Use an **Input Program Zero table** to describe where the part zero is. VERICUT automatically calculates "H" offset values, or enters values in a **Tool Length Compensation table**. For proper tool loading on a 3-D machine supply gage offset values as described above for **Gage Length** programming method.



Process Cutter Comp. — When active, applies cutter compensation, or "CDC" offsets to the programmed tool path. Enter CDC offset values in a **Cutter compensation table**, as described under **Cutter Compensation Table** in the *Tables for Processing G-Codes* section in the *CGTech Help Library*. **Output Precision** (refer to **Control Settings window: Motion tab** for more information) is used to specify the accuracy for calculating CDC offset tool positions.

Options:

Off — Ignore CDC related codes in the NC program file.

On – Default to Zero — Process CDC related codes in the NC program file. Default to zero if no Cutter compensation value is specified.

On – Default to Full Radius — Process CDC related codes in the NC program file. Default to the full tool radius value if no cutter compensation value is specified.

NOTE: Tool paths can not be optimized when CDC offsets are being simulated.

Scan NC Program Files — When active, performs a "scan pass" on NC program files to identify jump destinations for looping-branching logic. Tool paths with this type of logic must be scanned for proper simulation in VERICUT.

Ignore Data after Column — When toggled "On", specifies the G-Code data column number to begin ignoring remaining data in the block. Use this feature when unmarked comments are permitted in designated columns of the G-Code data. Enter the number of the last valid G-Code data column in the text field next to this feature.

Apply Acceleration to Cycle Time — When toggled "On", uses the **Accel/Decel** parameters, as set on the Modeling window: Component Attributes tab (ref. **Modeling window: Component Attributes tab**, also in the *VERICUT Help* section) when calculating the **Time** displayed in the **Status window**.

Initialize Machine/Control between NC Programs — Enables you to specify whether or not to reset the control's states, variables, etc. between NC program files during processing. When toggled "On" (the default), VERICUT resets these values between NC program files. This condition is consistent with prior releases. Toggle "Off" to retain these values between NC program files.

Rotary Feed Motion Tol — Tolerance used to calculate intermediate tool positions resulting from processing rotary motion when **Output Intermediate Points** is active (ref. **Configuration menu > Control Settings: Rotary tab**). Decreasing this value increases the quantity of intermediate points calculated, and decreases simulation speed.

Rotary Rapid Motion Tol — Similar to **Rotary Feed Motion Tol**, except applies to rapid rotary motions.

Machine Simulation On — When toggled "On", simulates machine tool motions when a 3-D machine is displayed in a machine view.

Shortcut: You can quickly toggle **Machine Simulation On**, "On" or "Off" by clicking on the **No Machine Simulation** icon  in the toolbar.

Floor/Wall Orient Up — Use to specify which way is "up" (towards the ceiling) for a machine when a machine view displays walls in the background. (Ref. **View menu > Attributes: Background**). The up direction is relative to the machine origin. Select the appropriate axis direction from the pull-down list.

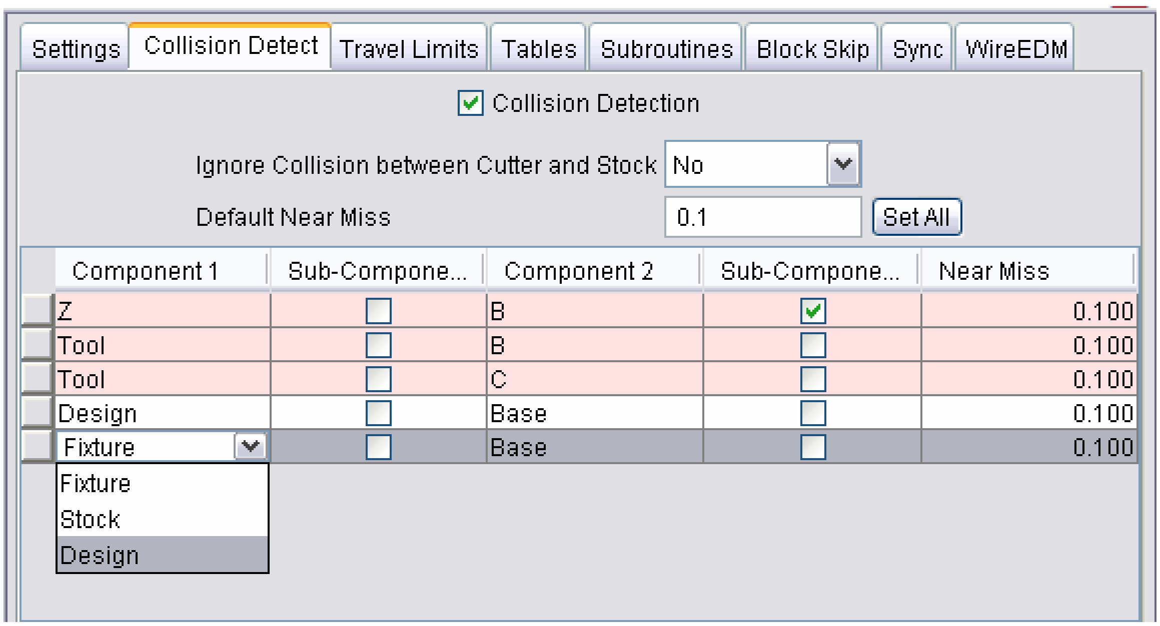
To G-Code Settings window

G-Code Settings window, Collision Detect tab

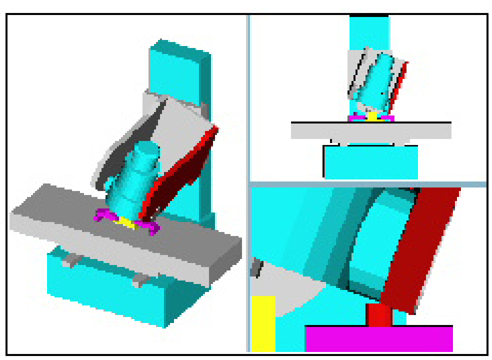
VERICUT Location: **Project menu > Processing Options > G-Code > Settings**

VERICUT toolbar short cut: 

Features on this tab control when collisions between machine components are detected, which components are protected, and tolerances used for detecting collisions.



Colliding components are highlighted using the red Error color, and errors are issued to the Log file identifying collision causing block(s) and machine components.



Collision Detection — When toggled "On", detects collisions between specified components.

Ignore Collision between Cutter and Stock — Controls when collisions between the cutter and the Stock component are ignored. This feature is useful when collision detection is desired between the stock and shank or holder portions of the tool assembly in the machine view, but not with the cutter.

Options are:

> **No —** (default) ***Does not*** ignore cutter-stock collisions. All collisions are reported.
>
> **All Tools —** Ignores cutter-stock collisions for all tools, even inactive tools in multi-tool machines.
>
> **Active Tool —** Ignores cutter-stock collisions for the active tool. However, collisions between stock and inactive tools are detected.

Default Near Miss — Specifies the default collision tolerance applied to all collision cases when **Set All** is pressed (see below).

Set All — Sets the default collision tolerance for all collision cases to the **Default Near Miss** value. You can edit the supplied tolerance for individual cases.

Component/Component collision list

Lists the component-to-component collision cases that are checked when collision detection is turned on.

NOTE: Records displayed in pink come from the machine file and ***can not*** be edited. Records displayed in white are part of the setup and can be edited.

> **Component1/Component 2 —** These features are used to specify the component-to-component collision cases to check. The **Near Miss** value controls how close the components are permitted to be before reporting a collision. Clicking on a component field in a record displays a pull-down list of component to choose from.
>
> NOTE: Do not configure for collision detection between components that move (slide or rotate) against each other, such as connected motion axes. In these cases, errors may occur each time the components move.
>
> **Sub-Components —** Toggle On/Off to include Sub-Components of **Component1/Component2** during collision checking.
>
> **Near Miss —** Specifies the tolerance to use for collision checking. Enter a positive value to be alerted if components come near each other within the specified clearance, zero to indicate components may not touch, or a negative value if components are expected to collide by the specified value.
>
> NOTE: "Near Miss" tolerances are not supported for collision checking against the cut model. The accuracy of collisions with the cut stock is dependent on the "Cutting Tolerance".

Add — Adds a new collision case record to the list. The new record is added after the highlighted record.

Delete — Deletes the selected collision case record from the list. Records displayed in pink come from the machine file and ***can not*** be deleted.

Shortcut: You can right-click in the Component/Component collision list to display a menu containing **Add** and **Delete**. These provide the same functionality described above.

To G-Code Settings window

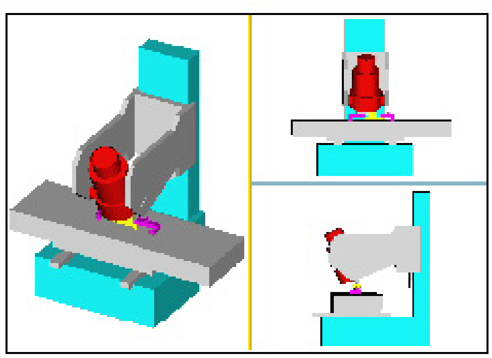
G-Code Settings window, Travel Limits tab

VERICUT Location: **Project menu > Processing Options > G-Code > Settings**

VERICUT toolbar short cut: 

Features on this tab define how far each machine axis can go, and control when travel limit errors are detected.

Overtravel errors cause the violating machine component to "light up" in the Overtravel Color and errors are output to the Log file identifying the problem component.



Log Error for Over Travel — When active, turns on overtravel detection (travel limit checking). Components that move beyond specified limits are highlighted in the **Overtravel Color** and errors are issued to the Log file identifying error causing block(s) and machine components.

Allow Motion Beyond Limit — When toggled "On", axes are allowed to freely travel beyond defined limits. When toggled "Off", an axis is never allowed to travel beyond the defined limit.

Overtravel Color — Use to specify the color in which components that move past their specified limits are shaded. Available shade colors are defined on the **Edit menu > Colors: Define tab**.

Travel Limit Record Table

List of travel limit records in which each record consist of a group number, motion component (and its axis limits), travel limit condition and an ignore switch. These records come from the machine file and ***can not*** be edited.

Group — Indicates the travel limits record group that is currently active A specific group is of travel limit records is activated by using the **TravelLimitsGroup** macro. For information on the **TravelLimitsGroup** macro, and all VERICUT macros, see *VERICUT Macros* in the *CGTech Help Library*.

Component — Indicates a motion component for travel limit checking.

Minimum / Maximum — Indicates the minimum, and maximum, travel limits for the motion axis specified in the record.

The next three columns in the **Travel Limit Record Table** indicate conditions when the travel limit record is used. The condition can be set to "OFF" indicating no special condition for this record or set to a motion component on the machine.

When a motion component is specified, then the travel limit record is used if the axis of the conditional component is within the conditional minimum and maximum limit values.

Component (C) — Indicates a condition when the travel limit record is used. It is set to "OFF" to indicate no special condition for this record, or to a conditional motion component on the machine.

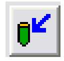
Minimum (C) / Maximum (C) — Indicates the minimum, and maximum, travel limits for the conditional motion component (Component (C)) that will cause the travel limit record to be used.

Ignore — When toggled "On", over travel checking is ignored for the corresponding axis.

To G-Code Settings window

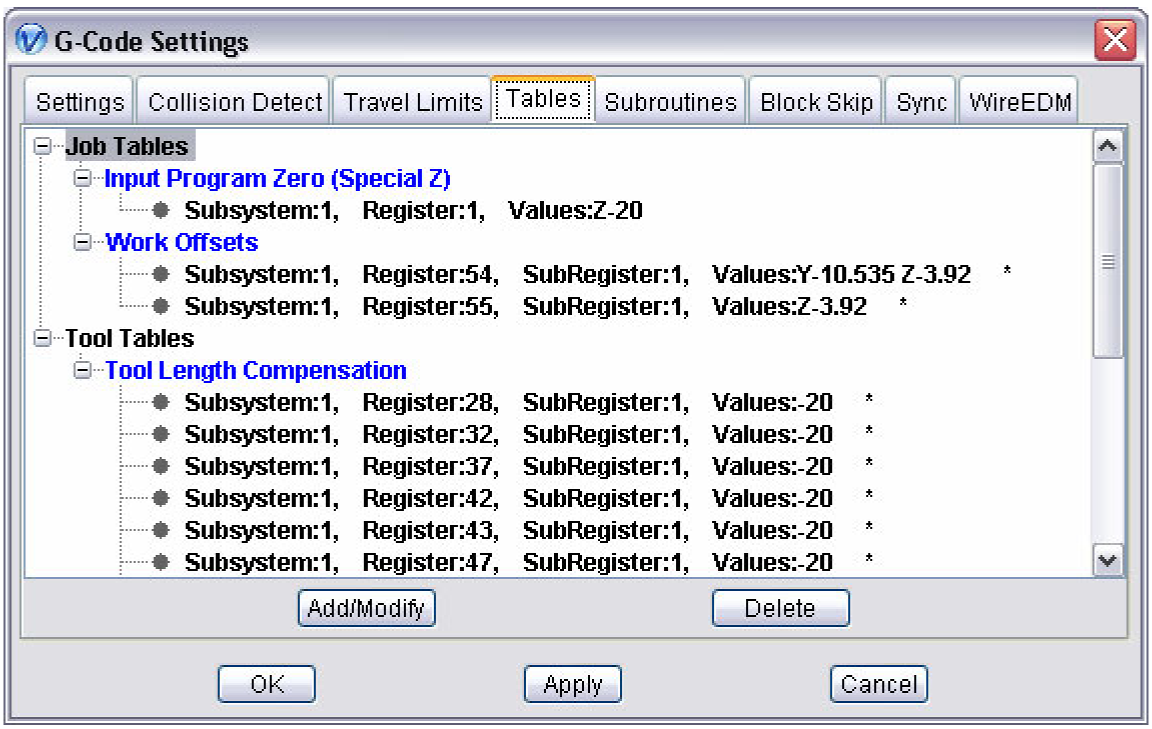
G-Code Settings window, Tables tab

VERICUT Location: **Project menu > Processing Options > G-Code > Settings**

VERICUT toolbar short cut: 

Features on the Tables tab are used to specify "**Job Tables**" to store work offsets and machine locations specific to the current tool path file(s) and "**Tool Tables**" to store tool-related offset and compensation data.

NOTE: The values set in a "**Job Table**" are used to override "standard" values that were set in a **Machine Table** for the current NC program file(s).



Tables list — A list of tables that are currently defined. The Machine Settings tables list will contain only "machine" tables. The G-Code Settings tables list will contain both "job" tables and "tool" tables.

Add/Modify — Use to access the Add/Modify Tables window enabling you to add additional tables or modify existing tables. To **Add** a table, simply click on **Add/Modify**. To modify an existing table, select the table to be modified from the Tables list and click on **Add/Modify**. (You can also "double-click" on the table in the Tables list.)

Delete — Deletes the selected table entry from the Tables list. If it is the last entry for a particular table, then the table is also deleted.

Job Table and **Tool Table** values are stored in the Project file.

The following **Job Tables** can be added or modified with the **G-Code Settings window, Tables tab**:

Base Work Offset
Initial Machine Location
Machine Zero (obsolete, formerly Input Machine Zero)
Machine Reference Location
Tool Change Location
Tool Change Retraction
Turret Rotations
Work Offsets
Program Zero
Input Program Zero (Special Z) (formerly Input Program Zero)
RTCP Pivot Offset
RPCP Pivot Offset

The following **Tool Tables** can be added or modified with the **G-Code Settings window, Tables tab**:

Cutter Compensation
Gage Offset
Probe Offset
Tool Length Compensation
Tool Nose Compensation

See the **Tables for Processing G-Codes** section, in the *CGTech Help Library*, for additional information on VERICUT tables.

Also see "**Building NC Machines**", in the *Using VERICUT* section, in the *CGTech Help Library*.

To G-Code Settings window

Add/Modify Tables window

The following discussion applies to both the **Add/Modify Machine Table window**, accessed from the **Machine Settings window: Tables tab** and to the **Add/Modify G-Code Table window**, accessed from the **G-Code Settings window: Tables tab**.

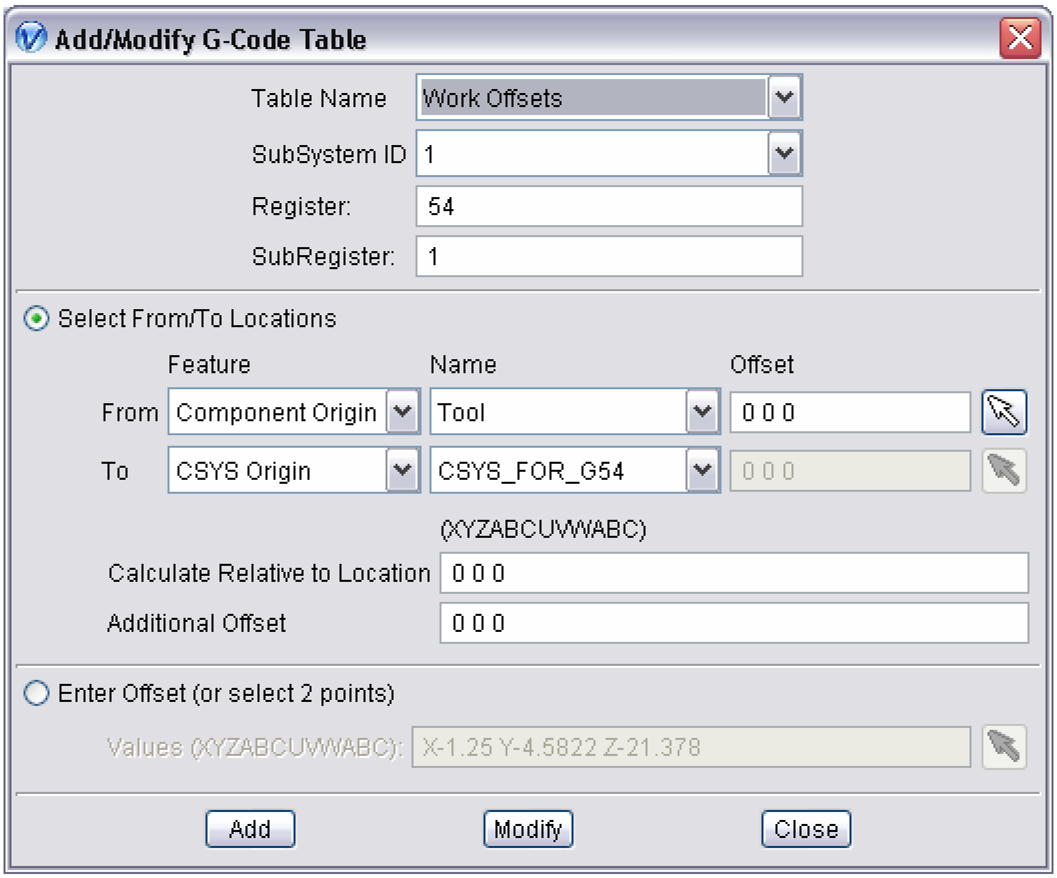


Table Name — Use to identify the table to be added, or modified, if it already exists. The **Table Name** list in the **Add/Modify Machine Table** window will only contain "machine" tables. The **Table Name** list in the **Add/Modify G-Code Table** window will contain both "job" tables and "tool" tables.

SubSystem ID — Use to specify ID of the machine subsystem for which the table is being defined.

Register — The Register number that will be used by VERICUT to access corresponding table data. For "tool" tables, this number typically corresponds to a tool or offset register number. For "job" and "machine" tables, the Register number may correspond to an offset register number, or an integer value, as required by a particular table.

SubRegister — Use of this feature enables you to access multiple sub-values for the same tool from tool related tables. For example, groove tools often have multiple "driven" points, which then correspond to multiple gage offsets, and possibly multiple Cutter Comp values, Tool Nose Comp values, and Tool Length Comp values.

This feature can also be used for Work Offset Tables for controls, like Yasnac, that support the sub-register feature.

SubRegister is only active when the selected "Table Name" identifies a "tool" related table or for Work Offset tables.

Select From/To and **Enter Values**:

These features are only active for the following tables:

Input Program Zero
Input Machine Zero
Base Work Offset
Work Offsets
RTCP Pivot Offset
RPCP Pivot Offset

For the above tables, VERICUT provides two ways to enter table values.

Select From/To Locations — Enables you to specify table values based on a "relational" offset between a "from" point and a "to" point. The "from" and "to" points are designated by specifying a particular component or CSYS. VERICUT will use the origin of the component/CSYS as the point. You can also specify an offset from the specified component/CSYS origin point. Once this relationship has been established, you can change the machine configuration or the location of the CSYS origin and the table values will updated automatically during the initial **Single Step** or during **Play to End**.

For example: After Input Program Zero has been defined to be the offset from the Tool Component to the "Program Zero" CSYS, you can change the machine configuration, or change the location of "Program Zero" CSYS, and the Input Program Zero offset values will be updated automatically during the initial **Single Step** or during **Play to End**.

NOTE: The offsets are calculated based on where the corresponding origin points are located when all linear axes are driven to machine zero (zero with no offsets in place).

When **Select From/To Locations** is toggled "On", the **From/To Feature/Name** lists and the **Offset** text fields become activated. Use the **From/To Feature** lists to specify whether the point is associated with a **Component** or a **CSYS** (coordinate system). Use the **From/To Name** lists to specify a specific component or coordinate system.

NOTE: Only the coordinate systems that have been defined with reference to a machine component (i.e., visible in a Machine or Machine/Cut Stock view when **Coordinate System** axis is toggled "on") will appear in the **CSYS/Names** lists. See the discussion for

Project menu > Coordinate Systems for more information on VERICUT Coordinate Systems.

Use the **Offset** text fields to enter 3 values, separated by spaces, representing the X, Y, and Z offset from the specified origin point. You can also click on the "select" button , then pick a point in the graphics display area. You can pick the point in either a workpiece or machine view. VERICUT will calculate the **Offset** value between the selected point and the designated **Feature**/**Name** origin point. **Offset** is only active when **Feature** is set to **Component**.

Calculate Relative to Location — This feature enables you to have a relational offset recalculated in the machine position where the offset will be used. The new position is immediately calculated and stored and therefore is not dependent on the machine position when the offset is activated.

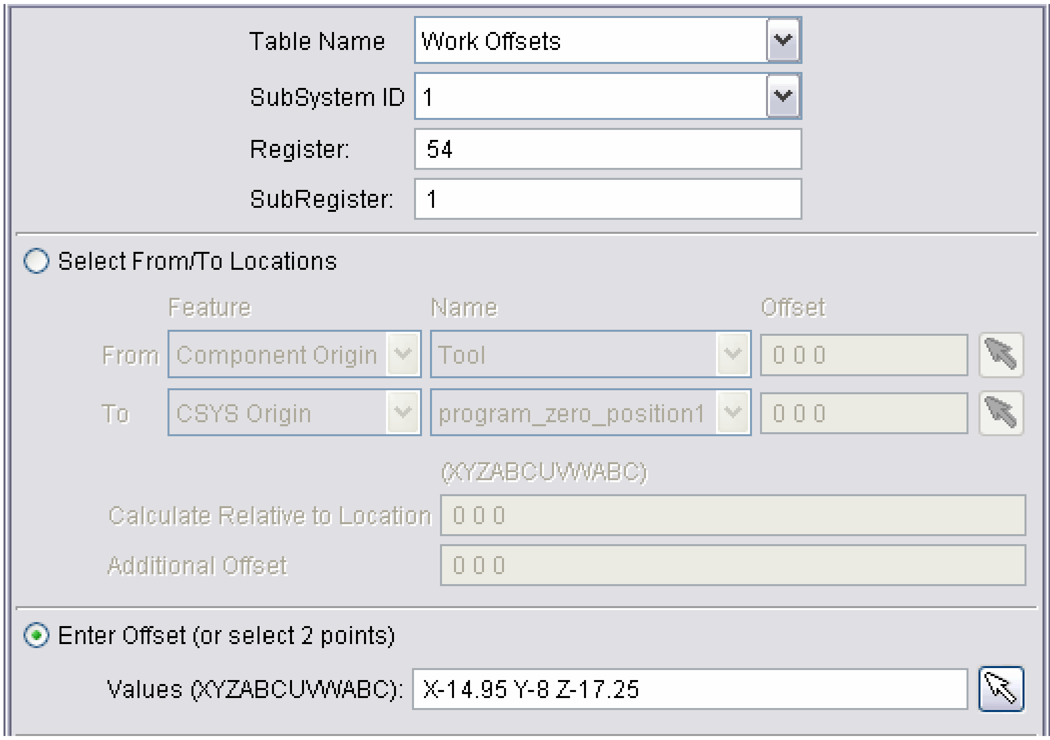
TIP: See **tombstone_work_offsets_single_part.vcproject** and **tombstone_work_offsets_multiple_parts.vcproject** in the *Sample-Demo Files* section, in the *CGTech Help Library* for examples of using this feature.

Additional Offset — Use this text field to specify an additional offset value.

Enter Offset (or select 2 points) — when toggled "On", the **Values (XYZABCUVWABC)** text field is activated enabling you to enter one to twelve numeric values as required by the specific table. Multiple value entries require spaces between the values.

You can also click on the "select" button , then pick two points in the graphics display area. You can pick the points in either a workpiece or machine view. VERICUT will calculate the **Offset** value between the selected points.

For all other tables, **Select From/To** and **Enter Offset (or select 2 points)** are both inactive. Enter the values in the **Values (XYZABCUVWABC)** text field as described for **Enter Offset (or select 2 points)** above.



In the sample window shown above, the current table is the **Work Offsets** table. The current table entry is register "54", and contains the values "-14.95 -8 -17.25". The table is applicable to subsystem "1" of the machine.

The following tables support "axis values" (up to 12 values – XYZABCUVWABC):

Base Work Offset
Initial Machine Location
Machine Zero
Machine Reference Location
Tool Change Location
Tool Change Retraction
Work Offsets
Program Zero
Input Program Zero

VERICUT supports two ways to input values into these tables from the **Add/Modify Table window**. Both use the **Values (XYZABCUVWABC)** text field, but use a different syntax):

Traditional format: values are entered in the order specified (XYZABCUVWABC) and are separated by spaces.

WORD/VALUE format (X10 C45)

In either case, the resulting display of the axis values will be in the WORD/VALUE format. Only non-zero values will be displayed. In the case of all zero's, X0 Y0 Z0 will be displayed.

The WORD/VALUE format has the following rules:

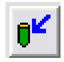
1. Valid Words are: X, Y, Z, A, B, C, U, V, W, AA, BB, CC
2. Mixed formats are not allowed. You can use ***either*** the Traditional format, ***or*** the WORD/VALUE format.
3. The Words can be entered in any order.
4. You only need to specify words with non-zero values.
5. If a word is specified twice, the last value specified will be used.
6. If all values are zero, you can leave the field blank. In this case, the display will be X0 Y0 Z0.

See the **Tables for Processing G-Codes** section, in the *CGTech Help Library*, for additional information on VERICUT tables.

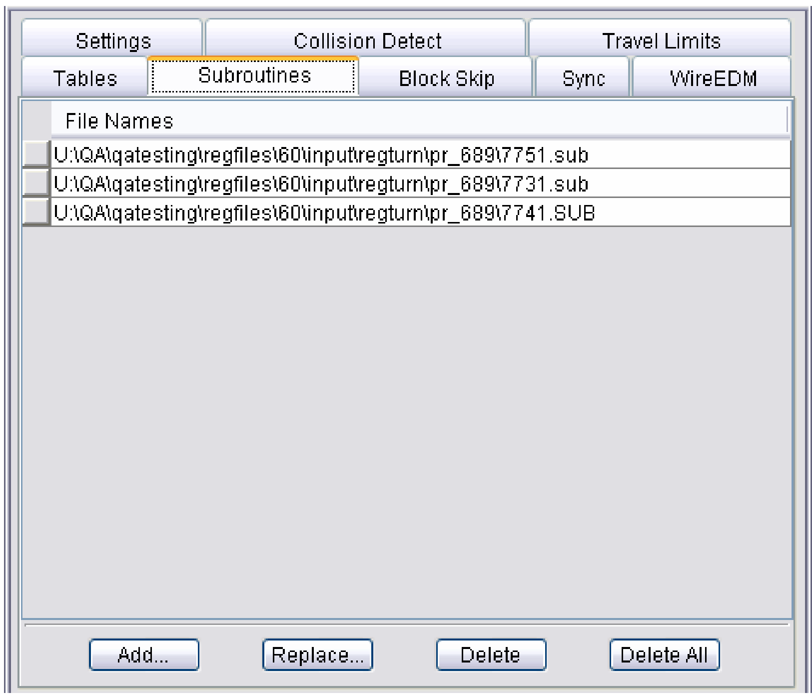
Also see "**Building NC Machines**", in the *Using VERICUT* section, in the *CGTech Help Library*.

G-Code Settings window, Subroutines tab

VERICUT Location: **Project menu** > **Processing Options** > **G-Code** > **Settings**

VERICUT toolbar short cut: 

The features on the Subroutines tab are used to specify names of external files containing subroutines that can be accessed by VERICUT.



File Names list — List of external files containing subs accessible to VERICUT.

Add — Opens the **Job Subroutine Files** selection box enabling you to add external subroutine files to the File Names list.

Replace — Opens the **Job Subroutine Files** selection box enabling you to replace the highlighted subroutine file in the File Names list with another.

Delete — Deletes the highlighted subroutine file from the File Names list.

Delete All — Deletes all subroutines files from the File Names list.

Shortcut: Right-click in the File Names list area to display a shortcut menu with the following features:

Add — Opens the **Job Subroutine Files** selection box enabling you to add external subroutine files to the File Names list. (Same as **Add** described above.)

Replace — Opens the **Job Subroutine Files** selection box enabling you to replace the highlighted subroutine file in the File Names list with another. (Same as **Replace** described above.)

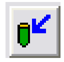
Delete — Deletes the highlighted subroutine file from the File Names list. (Same as **Delete** described above.)

Edit — Displays the of the highlighted subroutine file in a text editing window. Standard text editing features are provided, such as: copy/cut, paste, search, etc. (ref. Text File (edit) on the Edit Menu section of VERICUT Help for more information on using the editing features)

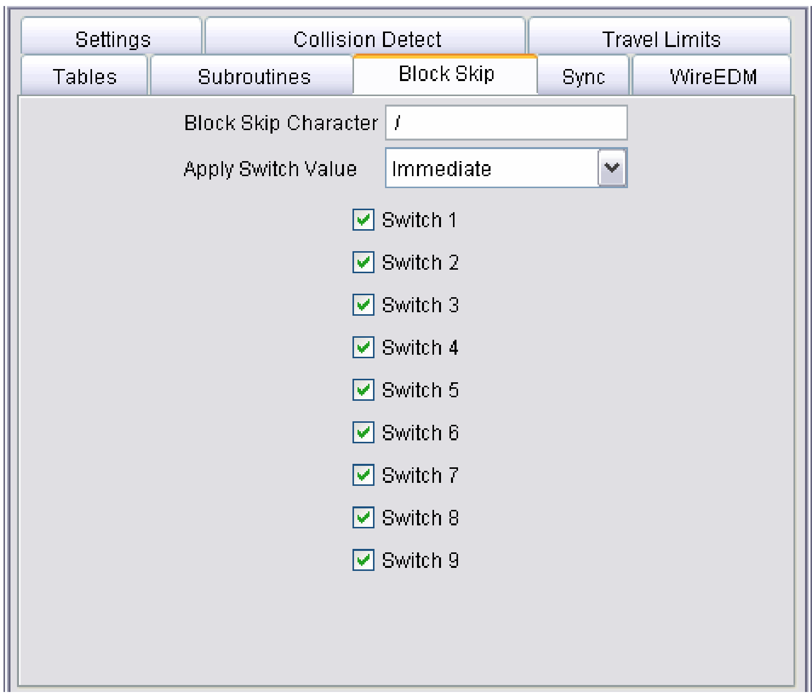
To G-Code Settings window

G-Code Settings window, Block Skip tab

VERICUT Location: **Project menu > Processing Options > G-Code > Settings**

VERICUT toolbar short cut: 

Features on the Block Skip tab control which levels of block skipping are recognized, and specify the block skip character.



Block Skip Character — Character which causes the block to be skipped. This character must be the first character in the data block for skipping to occur.

Apply Switch Value — Controls when block skip switch values are referenced by VERICUT.

Options:

Immediate — Reference switch settings when VERICUT is reset, upon loading a new Project file, and any time tool path processing is started.

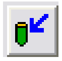
On Reset — Reference switch settings only when VERICUT is reset, or upon loading a new Project file.

Switch 1-9 — When active, processing is skipped for blocks beginning with the designated block skip switch. For example, "Switch 2" corresponds to blocks beginning with "/2", "Switch 3" corresponds to blocks with "/3", and so on.

To G-Code Settings window

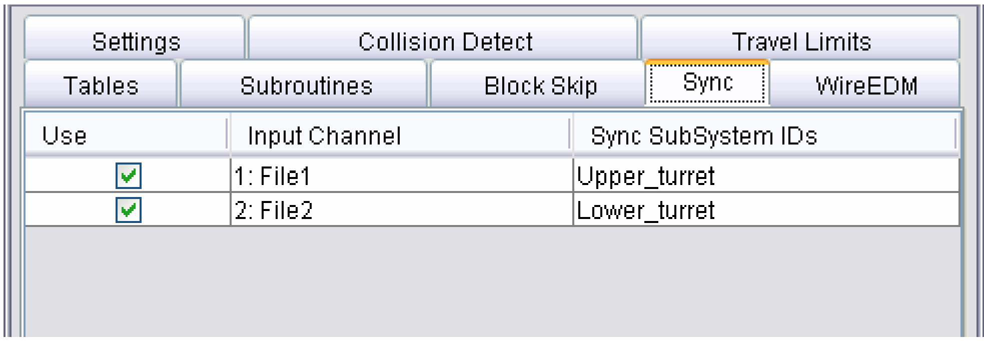
G-Code Settings window, Sync tab

VERICUT Location: **Project menu > Processing Options > G-Code > Settings**

VERICUT toolbar short cut: 

The features on this tab enable you to select which Sync Sub-system is to be used. This setting is only applicable if **Sync Method** on the **Control Settings window, Sync tab** is set to something other then **None**. The features on the **Control Settings window, Sync tab** are used to set up the sync environment.

The features on this tab are intended as a debugging tool enabling you to easily turn off one or more sync'd subsystems rather then running them all at the same time. VERICUT still runs through the "Sync" logic but a subsystem that is turned off, behaves like it is in a permanent "Wait" state.



Use — The checkbox in this field enables you to toggle a particular Sync Subsystem on, or off. When toggled "Off", VERICUT runs through the "Sync" logic but behaves like it is in a permanent "Wait" state.

Input Channel — This field shows what is driving each channel. The information displayed here will be different depending on the Sync Method being used. The above picture illustrates a Fanuc control where each channel is driven by a separate NC program.

Sync Subsystem IDs — This field shows the ID of the sync subsystem that the record represents.

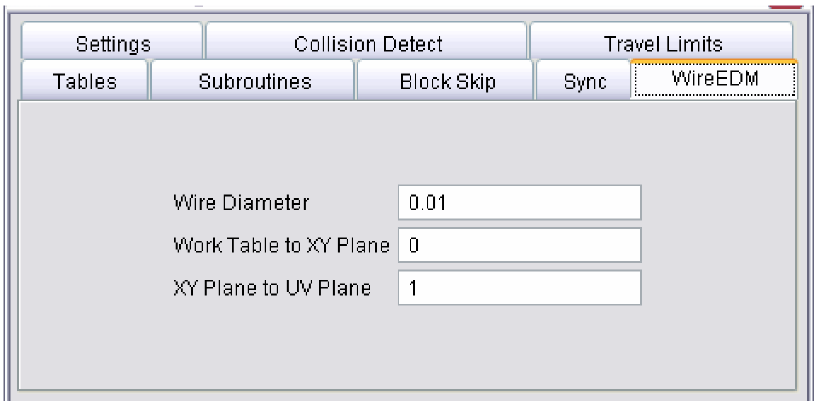
To G-Code Settings window

G-Code Settings window, WireEDM tab

VERICUT Location: **Project menu > Processing Options > G-Code > Settings**

VERICUT toolbar short cut: 

Features on the WireEDM tab configure VERICUT for wire EDM simulations.



Wire Diameter — Specifies the effective cutting diameter of the wire.

Work Table to XY Plane — Distance from the top of the work table (machine Z zero) to the XY reference plane, as measured along the machine Z-axis. This value is usually zero, but can be a positive or negative value. The XY reference plane is the Z-axis height at which the tool path file X and Y values are driven.

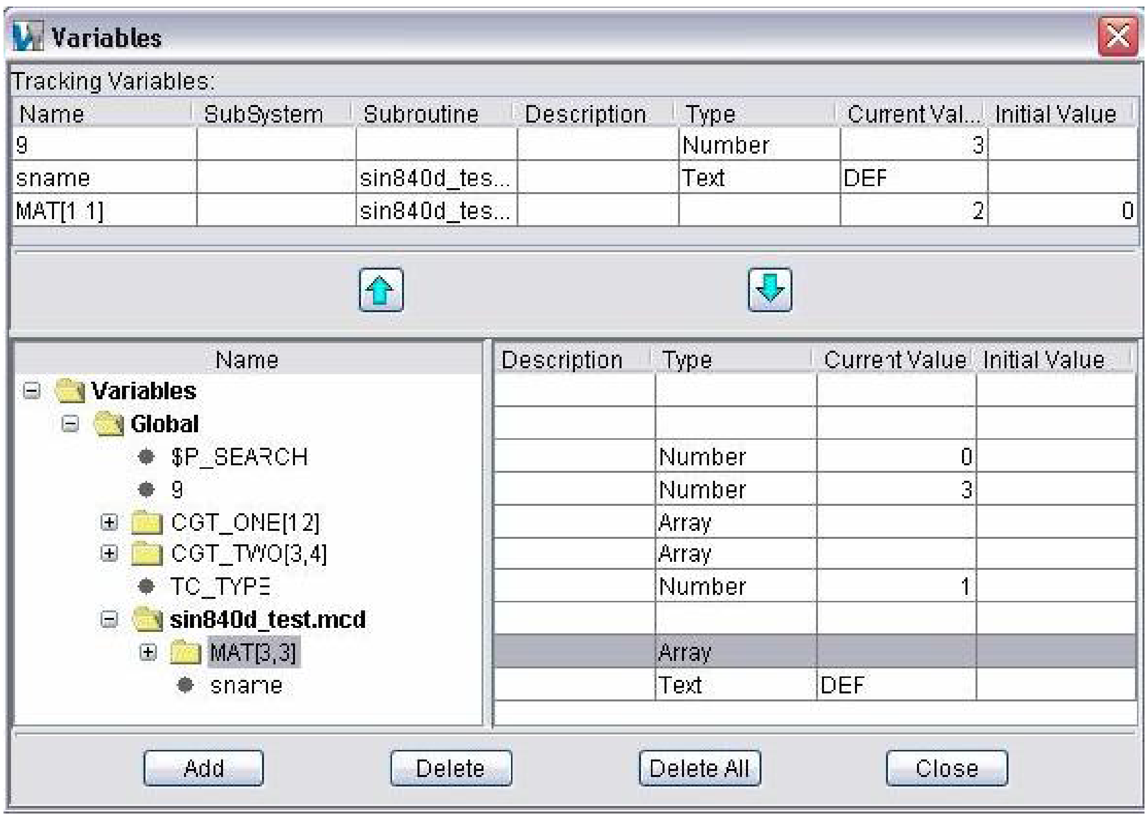
XY Plane to UV Plane — Distance from the XY reference plane to the UV reference plane, as measured along the machine Z-axis. This value must be a positive value. This setting is only used when the tool path file contains U and V-axis values to control the wire angle. The UV reference plane is the Z-axis height at which the tool path file U and V values are driven.

To G-Code Settings window

G-Code (Variables window)

Location: **Project menu > Processing Options > G-Code > Variables**

Features on the Variables window, are used to initialize, maintain, and monitor G-Code variables. Variables are assumed to have zero for a default value. However, you can initialize variables with any numeric or text value. When this window is open during processing, it gives immediate feedback when variables are set or changed by the tool path file. Variables are saved to the .usr file if an initial value or a description is entered, or if the variable is included in the Tracking Variables list.



Tracking Variables:

The Tracking Variables portion of the Variables window enables you to easily track variables of special interest without having to scroll through all variables in the Variables List. The following information is displayed for all variables selected for tracking:

Name — Name of the variable being tracked.

SubSystem — Indicates the subsystem (if any) that the variable being tracked is associated with.

NOTE: **SubSystem** in this case pertains to the subsystems defined in the Variables window variable tree, NOT the subsystems defined by the machine.

Subroutine — Subroutine (if any) that the variable being tracked is associated with.

Description — Description of the variable being tracked.

Type — Type of variable (Number, Text or Array)

Current Value — The current value of the variable.

Initial Value — The initial value of the variable.

 — Use to add the highlighted variable in the Variables List to the Tracking Variables list.

 — Use to remove the highlighted variable(s) from the Tracking Variables list.

TIP: Use one of the following methods to highlight multiple variables in the Tracking Variables list.

1. Use the "**Shift**" key to select a consecutive range of blocks. Click on the first block in the range so that it becomes highlighted, and then while holding down the "**Shift**" key, click on the last block in the range. All of the blocks between the first and last should now be highlighted.

   You can also click on the first variable in the list , then while keeping the mouse button depressed, drag the cursor to the last variable in the range.

2. Use the "**Ctrl**" key to select multiple individual blocks. Click on the first block so that it becomes highlighted, and then while holding down the "**Ctrl**" key, select additional blocks. Each block will become highlighted when it is selected.

Variables List:

The Variables List, located in the lower portion of the Variables window, displays all variables initialized and encountered during tool path processing.

Name — The Name column displays a tree structure showing a list of all variables, and how each variable relates to the overall structure of the NC program (subsystem, toolpath, subroutines, etc.). Tree is constantly updated as the toolpath is processed.

Description — Description of the variable. This field enables you to enter a description for the NC Variable. This feature allows you to document variables for later use, as well as simplifying the tracking of states. Instead of just seeing 4001, a more meaningful "Motion Type" description can be added. Descriptions that end with an " * " has been automatically generated by VERICUT. Automatically generated descriptions can result from any of the following:

Setting a variable on the **Events window**.

Setting a variable on the **Word/Address window**.

Calling one of the **Autoset…** macros.

Calling the **SetDynamicVars** macro.

Type — Type of variable (**Number**, **Text**, **Number Array**, **String Array**, **Frame**, or **Frame Array**); also indicates the type of data that can be assigned to the variable.

Current Value — The current value of the variable.

Initial Value — The initial value of the variable.

NOTE: It is possible to create a variable with no defined values. This is done by adding a variable with either a description, or by adding the variable to the tracking variable list. This is significant because VERICUT treats variables differently depending on whether the variable exists (defined within VERICUT) versus whether the variable has an undefined value.

If you try to access a non-existing variable (a variable that does not show up in variables window), VERICUT outputs an error message saying; "un-initialized variable, defaulting to zero".

When you call a G65 subroutine, variables 1-33 get initialized to "undefined" or "vacant" except those, which are set on the call to the subroutine. If you try to access an "undefined" variable, then VERICUT does not output an error message, or default to zero.

See "**Notes about un-initialized variables**" in the *Using VERICUT* section for additional information. *Using VERICUT* can be found in the *CGTech Help Library*.

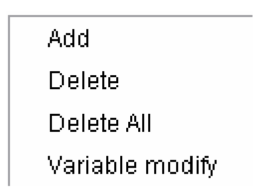
Add — Opens the **Variable Add window** enabling you specify the characteristics of a new variable and add it to the variable list.

Delete — Deletes the highlighted variable from the variable list.

Delete All — Deletes all variables in the variable list.

Close — Close the Variables window.

Shortcut: Right-click in the Variables List to display the following menu:



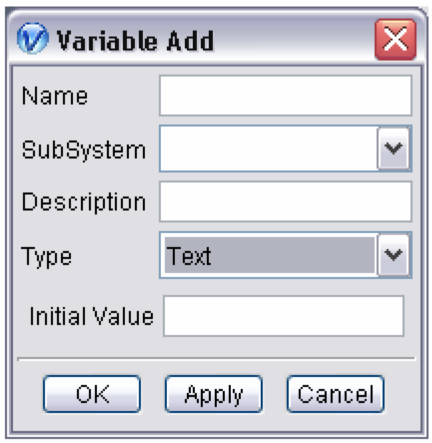
NOTES:

1. Features **Add**, **Delete**, and **Delete All** provide the same functionality as above under Variables List.

2. Feature **Variable modify** opens the Variable Modify window enabling you to modify the characteristics of array variables. This feature is only active when an array variable is highlighted.

Variable Add window

Accessed by selecting **Add** in the Variables window, the features in this window enable you to specify the characteristics of a new variable and add it to the variable list. The Variable Add window will look different depending on the "type" of variable being defined.



Name — Name of the variable.

SubSystem — Use this feature to specify the subsystem that the variable is associated with.

NOTE: **SubSystem** in this case pertains to the subsystems defined in the Variables window variable tree, ***NOT*** the subsystems defined by the machine.

Description — Use this field to add descriptive information about the variable

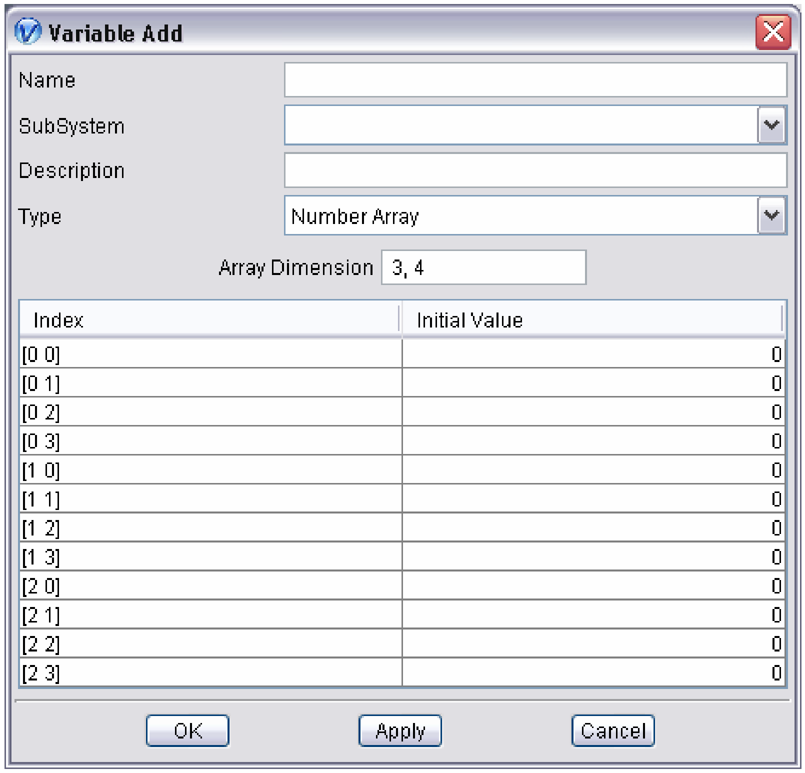
Type — Select the Type of variable from the pull-down list; also indicates the type of data that can be assigned to the variable.

Options:

Number — the variable is real number. For example, value ="0010" => the numeric value "10" is used.

Text — the variable is a text string. For example, value ="0010" => the text string "0010" is used.

Number Array — The variable is an array. When **Number Array** is selected as the variable type, an **Array Dimension** field and an array table are added to the Add Variable window.



Array Dimension — Use to specify the dimensions of the array that the variable represents, for example "3, 4", for a 3x4 array.

Array Table — Use the array table to initialize specific array values if desired. VERICUT automatically adjusts the size if the array based on the dimension set with Array Dimension and all values are defaulted to 0.

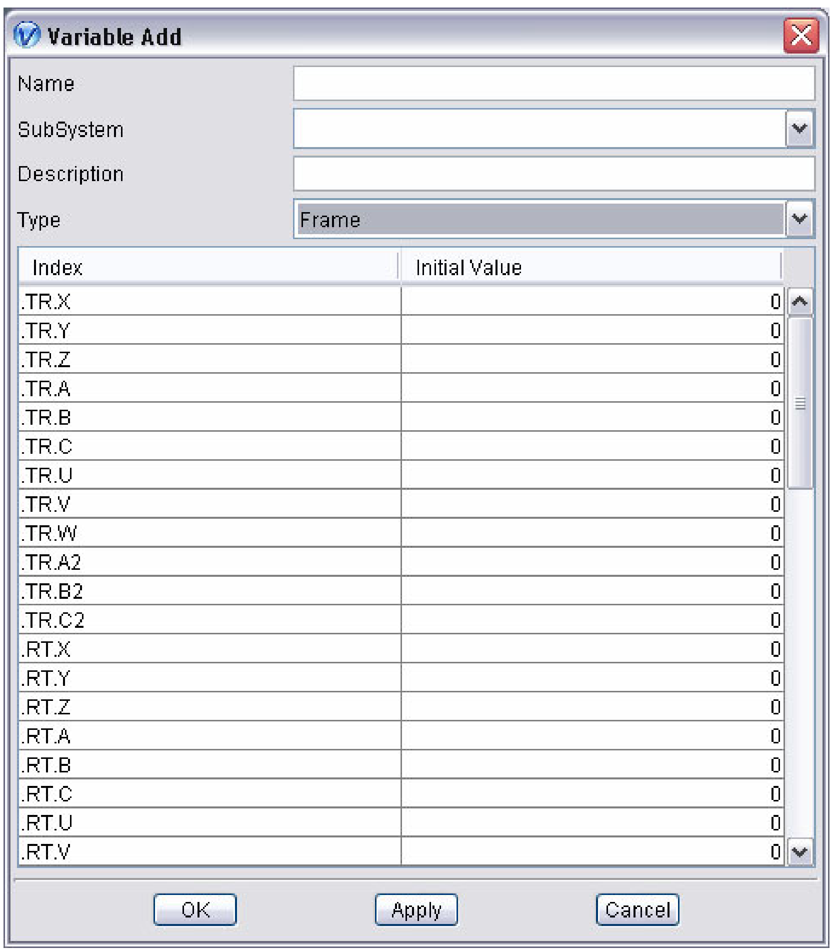
NOTES ON USING ARRAY VARIABLES:

1. Non-numeric variables must be defined in the **Words window** to enable proper parsing.
2. In the **Variables window**, **Type** must be set to **Array**, and **Array Dimension** must also be defined.
3. In the MCD file, the syntax must be of the form: name[2,1]=5 where the square brackets are left and right precedence, and indexes are separated by commas. Each index can also be defined as an expression. For example: name[#7+1,1]=3.
4. Array variables are also supported in the **Word Address window** as a variable to be set. Non-numeric variables can not be used with override values.

String Array — The variable is an array. When **String Array** is selected as the variable type, an **Array Dimension** field and an array table are added to the Add Variable window.

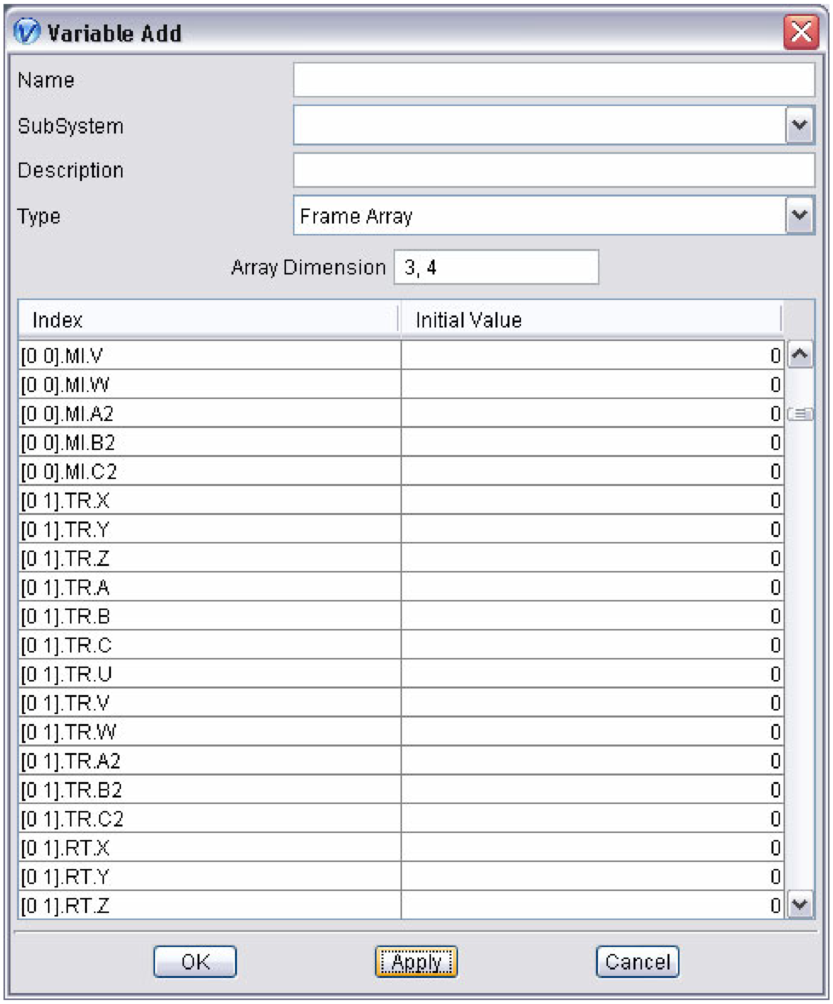
> **Array Dimension —** Use to specify the dimensions of the array that the variable represents, for example "3, 4", for a 3 x 4 array.
>
> **Array Table —** Use the array table to initialize specific array values if desired. VERICUT automatically adjusts the size if the array based on the dimension set with Array Dimension and all values are defaulted to 0.

Frame — When **Frame** is selected as the variable type, a frame table is added to the Add Variable window.



> **Frame Table —** Use the array table to initialize specific frame values if desired.

Frame Array — The variable is an array. When **Frame Array** is selected as the variable type, an **Array Dimension** field and a frame array table are added to the Add Variable window.



Array Dimension — Use to specify the dimensions of the array that the variable represents, for example "3, 4", for a 3x4 array.

Array Table — Use the array table to initialize specific array values if desired. VERICUT automatically adjusts the size if the array based on the dimension set with Array Dimension and all values are defaulted to 0.

OK — Adds the variable to the Variables List in the G-Code Variable window and dismisses the Add Variable window.

Apply — Adds the variable to the Variables List in the G-Code Variable window and leaves the Add Variable window open enabling you to specify additional variables.

Cancel — Dismisses the Add Variable window without adding the variable to the Variables List in the G-Code Variable window.

Variable Modify window

The Variable Modify window is accessed by selecting **Variable Modify** in the Variables window, right mouse button shortcut menu. The features in this window enable you to modify the characteristics of an existing array variable.

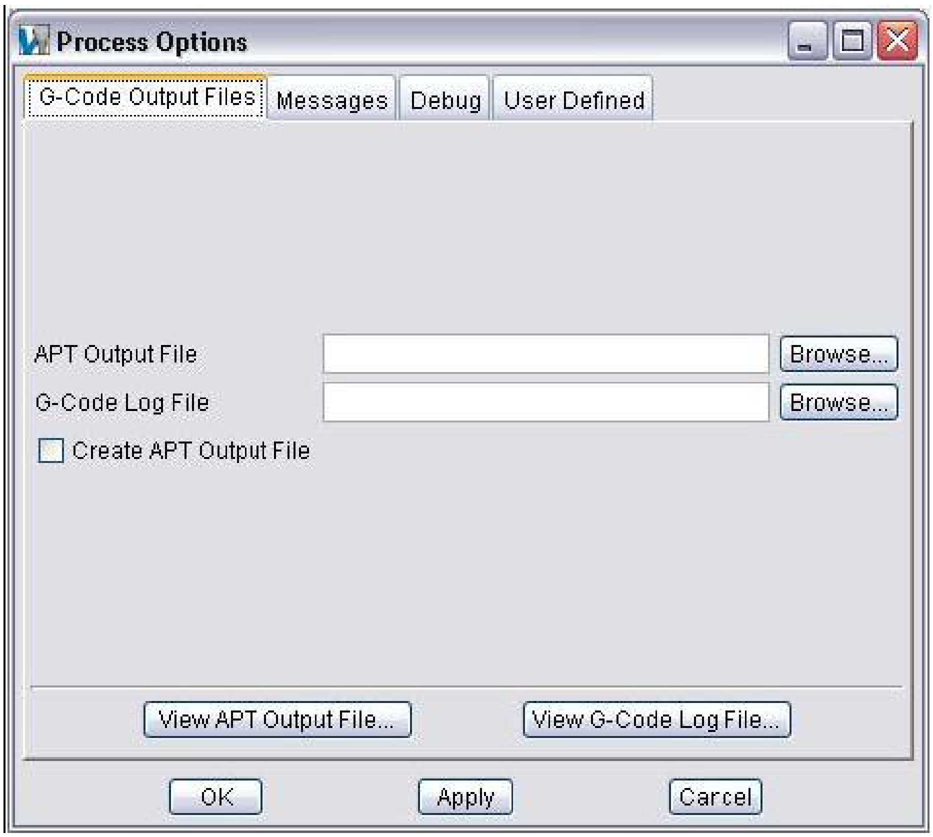
The Variable Modify window will look different depending on the "type" of array variable (Number Array, String Array, or Frame Array) being modified.

The features available in the Variable Modify window are identical to the ones described above under Variable Add window for **Types**: Number Array, String Array , and Frame Array.

G-Code (Process Options window)

Location: **Project menu > Processing Options > G-Code > Process Options**

Opens a window enabling you to configure the settings required to support processing of G-Code NC program files.



G-Code Output Files tab — Features on this tab are used to control conversion, or "reverse post-processing" of G-Code NC program files, and specify the names of files resulting from this process.

Messages tab — Features on this tab are used to set the destination and format of messages output during a simulation.

Debug tab — Features on this tab are used to select what debug messages to send to output.

User Defined tab — Features on this tab are used to configure user defined control settings supplied via a custom CME file.

OK — Applies the changes and closes the Process Options window.

Apply — Applies the changes and leaves the Process Options window open.

Cancel — Closes the Process Options window without applying changes.

Process Options window, G-Code Output Files tab

Location: **Project menu > Processing Options > G-Code > Process Options**

Features on this tab are used to control conversion, or "reverse post-processing" of G-Code NC program files, and specify the names of files resulting from this process.

APT Output File — Name of the file to receive ASCII APT tool path records resulting when Create APT Output File is active (see below). If blank, the file is not created.

G-Code Log File — Name of the G-Code Log file to receive error, warning and informational messages about G-Code processing. If blank, the file is not created.

Create APT Output File — When active, converts the G-Code NC Program data into ASCII APT NC Program records. Converted data is stored in the specified APT Output file (see above).

View APT Output File — Opens a window containing the APT Output file with ASCII APT tool path records created when Create APT Output File is active (see above).

For information about APT NC Program records and formats, see "**Supported APT-CLS records**".

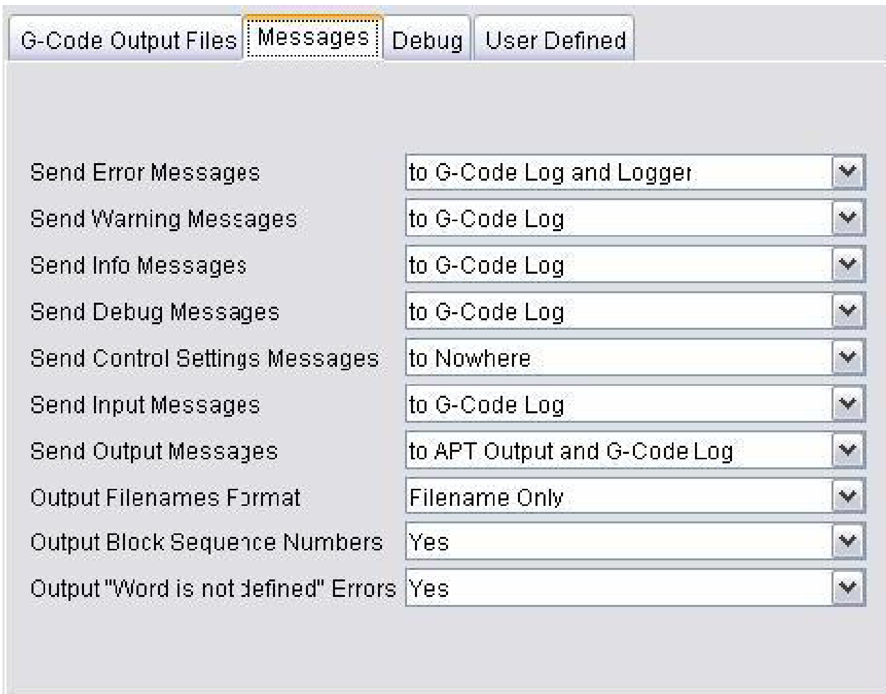
View G-Code Log File — Opens a window containing the G-Code Log file. This file contains information, such as error, warning and informational messages about G-Code processing. In this window, you can view, edit, or reset the G-Code Log file contents.

To Process Options window

Process Options window, Messages tab

Location: **Project menu > Processing Options > G-Code > Process Options**

Features on this tab are used to set the destination and format of messages output during a simulation.



Send Error Messages — Destination of messages about errors-potentially severe problems that will most likely produce incorrect results on the machine tool.

Send Warning Messages — Destination of messages about possible problems. Scrutinize warnings to decide if they will actually impact the machining process.

Send Info Messages — Destination of messages providing information about files used, the time and date that the process started, etc.

Send Debug Messages — Destination of debug messages providing additional information related to internal calculations, conditions, and variable values. The types of debug messages which appear are controlled via **Process Options window: Debug tab**.

Send Control Setting Messages — Destination of messages showing the control settings in effect when the tool path file was processed.

Send Input Messages — Destination of messages showing the input tool path file records processed.

Send Output Messages — Destination of messages showing the converted tool path records resulting from reverse-postprocessing. When **Create APT Output File** is active on the **Process Options window: G-Code Output Files tab**, output messages are sent to the APT Output File specified on the same window. By default, these records are in ASCII APT format. However, using the **C Macro Extension – Application Programming Interface**, or **CME–API** (ref. **CME-API** in the VERICUT Development Tools section, in the *CGTech Help Library*), other output file formats are obtainable.

Output Filenames Format — Controls how names of files used by VERICUT are shown.

Options:

Filename Only

Full Path Names — includes full directory path).

Output Block Sequence Numbers — Controls when block sequence numbers are included with APT output records resulting from conversion. The sequence numbers appear in columns 73-80, and are useful for general reference or debugging purposes.
Options:

Yes — output block sequence numbers

No — don't output block sequence numbers

Output "Word is not defined" Errors — When set to Yes, outputs the "Word is not defined" error message when G-Code words are encountered that the control is not set up to process. To prevent these errors from appearing, ensure this feature is set to No.

Messages can be sent to the destinations listed below:

- to APT Output
- to G-Code Log (not available for Send Output Messages)
- to Logger (not available for Send Output Messages)
- to APT Output and G-Code Log
- to APT Output and Logger
- to G-Code Log and Logger (not available for Send Output Messages)
- to APT Output, G-Code Log and Logger
- to Nowhere (not available for Send Error Messages, Send Warning Messages or Send Output Messages)

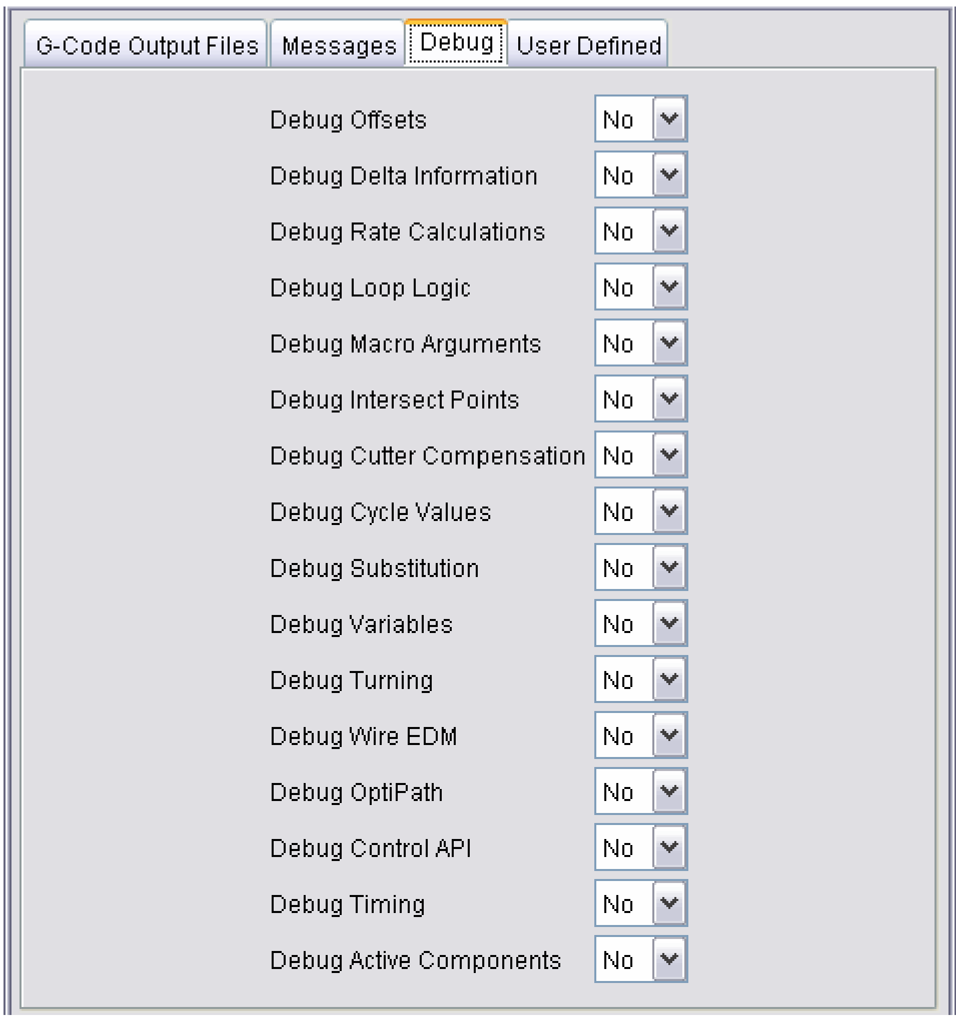
NOTE: Not all destination choices are available for all message types.

To Process Options window

Process Options window, Debug tab

Location: **Project menu > Processing Options > G-Code > Process Options**

Features on this tab are used to select what debug messages to send to output. Debug messages can provide additional information that may be helpful debugging development of files used to simulate G-Code NC programs and simulate machine tool motions. Set each Debug Message to either Yes (send to output) or No (don't send to output). The output destination for debug messages is controlled by "Send Debug Messages" on the **Process Options window: Messages tab**.



Debug Offsets — When selected, outputs debug messages about offset value changes, including: coordinate system offsets/shifts, compensation values in use, etc.

Debug Delta Information — When selected, outputs debug messages about rotating the tool position.

Debug Rate Calculations — When selected, outputs debug messages about rate calculations related to tool movement.

Debug Loop Logic — When selected, outputs debug messages about values calculated by looping or branching logic.

Debug Macro Arguments — When selected, outputs the macro name, the word, the text string value, and the numeric value for the macro being called.

Debug Intersect Points — When selected, outputs debug messages related to intersect points calculated by CDC calculations.

Debug Cutter Compensation — When selected, outputs debug messages about CDC tool offset positions.

Debug Cycle Values — When selected, outputs debug messages related to processing canned tool axis cycles.

Debug Substitution — When to selected, outputs debug messages related to string substitutions (ref. **Configuration menu > Adv. Options: Substitute tab**).

Debug Variables — When selected, outputs debug messages about changes to variable values. See also: **Project menu > Processing Options > G-Code > Variables window**.

Debug Turning — When selected, outputs debug messages associated with turning, for example the distance from the turning axis to the tool control point and calculations related to the use of constant surface speed.

Debug Wire EDM — When selected, outputs debug messages associated with wire EDM machining.

Debug OptiPath — When this is turned on, a descriptive message will be printed whenever machine simulation determines a block is non-optimizable.

Debug Control API — When turned on, debug messages related to the API Interface for the Heidenhain iTNC Controller are output.

See **API Interface for the Heidenhain iTNC Controller**, in the *VERICUT Development Tools* section, in the *CGTech Help Library*.

Debug Timing — When turned on, time related debug messages [Delta time (time for this block), Total time, and Time Attributes (information used to calculate Delta time)] are output for each block.

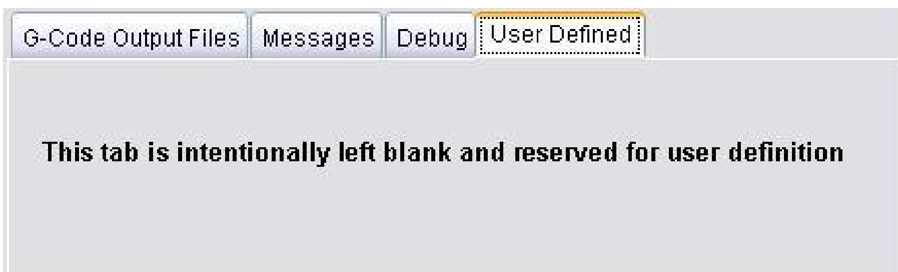
Debug Active Components — When turned on, the Active Subsystem, Active Tool Component name, and Active Stock Component name will be printed on each block. Also, the Active Spindle Component name will be printed whenever it is used or set.

To Process Options window

Process Options window, User Defined tab

Location: **Project menu > Processing Options > G-Code > Process Options**

Features on this tab are used to configure user defined control settings supplied via a custom CME file. This window will appear differently, depending on the CME file in use. CME files are created using the **C Macro Extension – Application Programming Interface**, or **CME–API** (ref. **CME-API** in the VERICUT Development Tools section, in the *CGTech Help Library*), and specified for use in VERICUT via **Dev Kit CME tab** of the **Advanced Control Options window** (ref. **Advanced Control Option window: Dev Kit CME tab**, in the Configuration Menu section of *VERICUT Help*).



NOTE: Developer's Kit-defined settings can also appear on the U**ser Defined (CTRL) tab** of the **Control Settings window**, as determined by the developer.

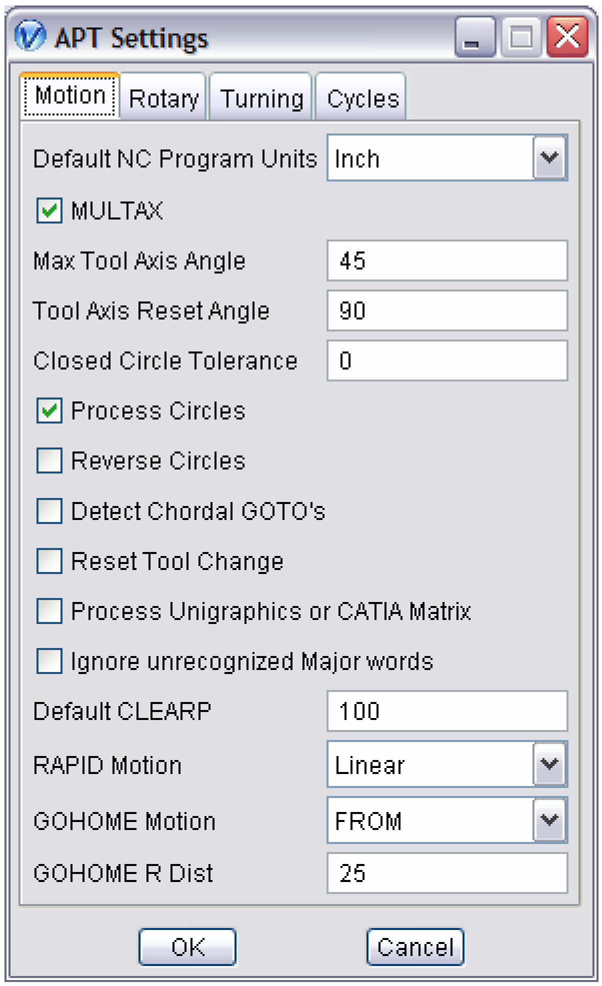
To Process Options window

APT Settings window

Location: **Project menu > Process Options > APT Settings**

VERICUT toolbar short cut: 

Opens a window to configure settings to support processing APT-CLS tool path files.



Motion tab — Features on this tab provide important settings for APT-CLS tool path simulation, such as: how circle and rapid motions are simulated, default tool path units, and more.

Rotary tab — Features on this tab provide control over rotary motions simulated for APT-CLS tool paths.

Turning tab **—** Features on this tab provide control over lathe turning motions simulated for APT-CLS tool paths.

Cycles tab **—** Features on this tab enable you to change how VERICUT simulates APT or CLS tool path canned cycles, also known as "fixed tool axis cycles".

OK — Saves the settings and closes the APT Settings window.

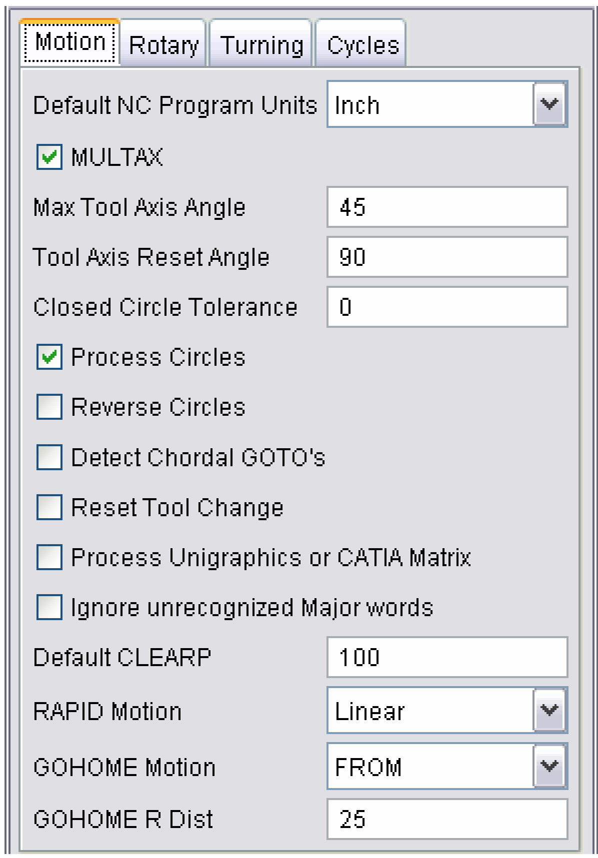
CANCEL — Closes the APT Settings window without saving setting changes.

APT Settings window, Motion tab

Location: **Project menu > Process Options > APT Settings**

VERICUT toolbar short cut: 

Features on this tab provide important settings for APT-CLS tool path simulation, such as: how circle and rapid motions are simulated, default tool path units, and more.



Default Toolpath Units — Default unit measurement system for tool path files that do not contain a "UNITS" record. If the tool path file units differ from the session units, values in the tool path are converted appropriately while being processed.

MULTAX — When selected, interprets six parameter GOTO records as GOTO/x,y,z,i,j,k where i,j,k is tool axis orientation (superseded by MULTAX/OFF). Clearing this checkbox interprets these records as GOTO/x1,y1,z1,x2,y2,z2 - two point locations (superseded by MULTAX/ON).

Max Tool Axis Angle — Maximum angle (in degrees) along a multi-axis motion to display intermediate tool positions. A value of "0" (default) does not display intermediate positions on multi-axis motions. The cut model is unaffected by this feature.

Tool Axis Reset Angle — Minimum angle (in degrees) which is considered a positioning move. Multi-axis motions having angles equal to or greater than the reset angle do not remove material. Instead, the tool display is turned off through the multi-axis motion, then turned on after the multi-axis destination is reached. Cutting resumes at the new tool position.

Closed Circle Tolerance — The Closed Circle Tolerance value is used by VERICUT to determine whether a circular motion should go around the whole circle, or just around a small arc of the circle.

A value of zero in this field causes VERICUT to check whether a circle is meant to be a complete loop using the same method it has used for many years.

A non-zero value causes VERICUT to use a new method for determining whether a circle is meant to be a complete loop. VERICUT compares the specified Closed Circle Tolerance value with the distance between a circle's start, and end, points. If the distance is less than the specified tolerance, VERICUT assumes that the tool must move round the whole circle. If the distance is greater than the specified tolerance, then VERICUT moves the tool around a small arc of the circle.

Process Circles — When selected, processes CIRCLE records to simulate the arc motion. Clear this checkbox to ignore "CIRCLE" records, such as necessary when CIRCLE records are accompanied by their representative chordal GOTO points.

NOTE: Simulating circle motions increases performance dramatically over processing chordal GOTO points.

Reverse Circles — When selected, reverses CIRCLE record motions. Clear this checkbox to simulate circle motions in the direction programmed. (Circle IJK values determine circle direction.)

Detect Chordal GOTOs — When this option and **Process Circles** are selected, the chordal GOTO points which follow CIRCLE records motions are processed to simulate an arc, or circle motion. Clear this checkbox to process the GOTOs as linear tool positions.

Reset Tool Change — When selected, resets the tool display following a tool change. This action turns off the tool display until after the tool is changed and the following motion is processed. The new tool appears at the end of the motion following the tool change. This option is recommended when simulating tool paths for NC machines that automatically retract the tool during a tool change. Clear this checkbox to have VERICUT change the tool at its current location when a tool change command is encountered.

Process NX or CATIA Matrix — When selected, processes these tool path matrix records to transform the tool path to its base coordinate system: CATIA0 or MSYS. Clear this checkbox to ignore tool path matrix records. This feature is useful to automatically orient tool paths to models imported from the same CAD/CAM system.

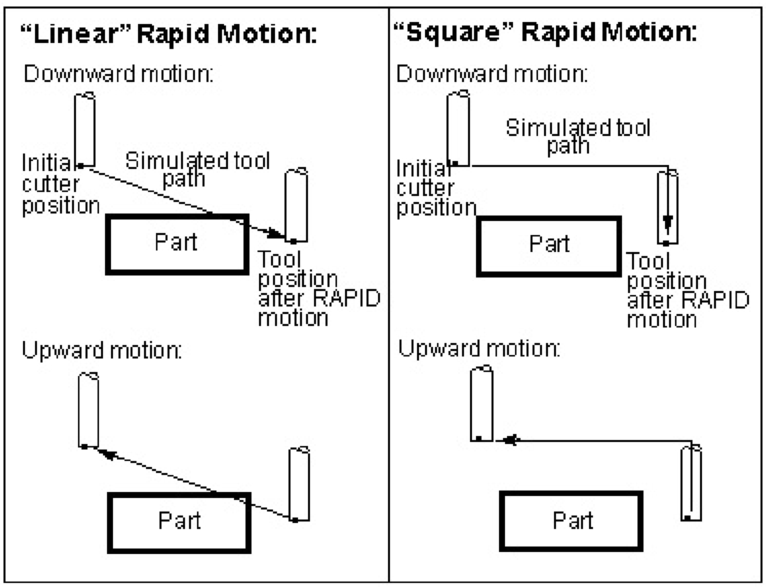
Ignore unrecognized Major words — When toggled on, VERICUT ignores unrecognized APT major words and no warning messages are output to the VERICUT main window message area, or in the Log file.

Default CLEARP — Distance to retract along the tool axis when a RETRCT record is encountered prior to establishing the retract plane. The default CLEARP condition is "off" until a CLEARP record defining a clearance plane is encountered.

RAPID Motion — Controls how to simulate rapid positioning motions. Options:

Linear — Move in a straight line.

Square — Move with "squared off" motion, where Z motions are processed independently from X and Y.



GOHOME Motion — Controls how the tool is returned to the home position when a GOHOME record is processed. Options:

FROM — Move in a straight line to the home position.

Retract — Move with squared off motion, where the tool is first retracted the distance specified by the **GOHOME R Dist** value, then moved to the home position.

GOHOME R Dist — When simulating GOHOME record motions via the "**Retract**" option (see above), this value specifies the distance to retract before going to the home position.

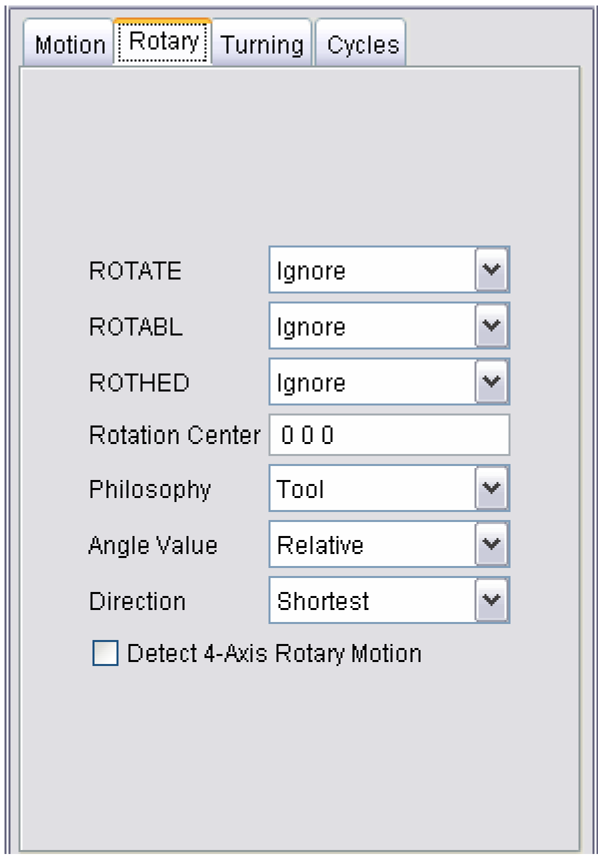
To APT Settings window

APT Settings window, Rotary tab

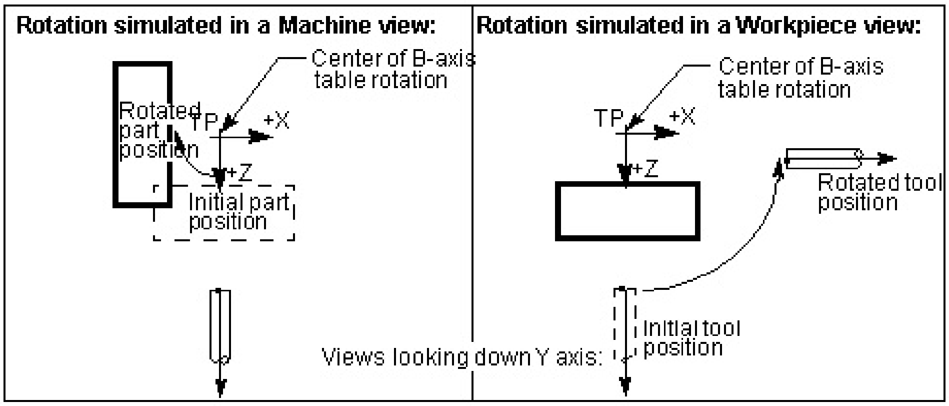
Location: **Project menu > Process Options > APT Settings**

VERICUT toolbar short cut: 

Features on this tab provide control over rotary motions simulated for APT-CLS tool paths. Note that while machines often rotate the workpiece, in a VERICUT Workpiece view the workpiece is fixed and the tool is seen rotating about the workpiece. Both rotary cut paths are identical.



"ROTATE/BAXIS,ATANGL,90" as seen on a machine with a "B" rotary table, and simulated by VERICUT in a Workpiece view:



ROTATE, ROTABL, ROTHED — These features control when and how to simulate motion caused by the corresponding **ROTATE**, **ROTABL**, or **ROTHED** records, respectively. "BAXIS" is the default rotation axis when not specified. The rotary pivot point is defined by the **Rotation Center**. When absolute or incremental rotation is not specified, the **Angle Value** determines how to apply the rotary value.

Options:

Ignore — Ignores the rotary command. (Default)

Rotate — Rotates while cutting. Tool path motions that follow are expected to have the rotation applied.

Rotate/Apply — Similar to **Rotate**, except also applies the rotated coordinate system to subsequent tool path motions. Tool path motions that follow are NOT expected to have the rotation applied.

Position — Rotates without cutting, also known as "positioning". The tool disappears from its initial position, then appears at the rotated position. Tool path motions that follow are expected to have the rotation applied.

Position/Apply — Similar to **Position**, except also applies the rotated coordinate system to subsequent tool path motions. Tool path motions that follow are NOT expected to have the rotation applied.

Rotation Center — Center of rotation for rotary motions simulated via the ROTATE, ROTABL, or ROTHED features (see above).

Philosophy — Describes what is being commanded to rotate for rotary motions simulated via the **ROTATE**, **ROTABL**, or **ROTHED** features (see above): Table (default), or Tool. Changing the rotation philosophy has the affect of reversing rotation direction. Note that ROTHED angle values are always assumed to use a rotating Table philosophy.

Angle Value — Controls how angle values are applied for rotary motions simulated via the **ROTATE**, **ROTABL**, or **ROTHED** features (see above) when the commands do not specify "ATANGL" or "INCR".

Options:

Relative — Incremental angle values.

Absolute — Absolute angle locations. (Default)

Note that ROTHED angle values are always assumed to be Absolute.

Direction — Direction of rotation for simulating for rotary motions simulated via the **ROTATE**, **ROTABL**, or **ROTHED** features (see above).

Options:

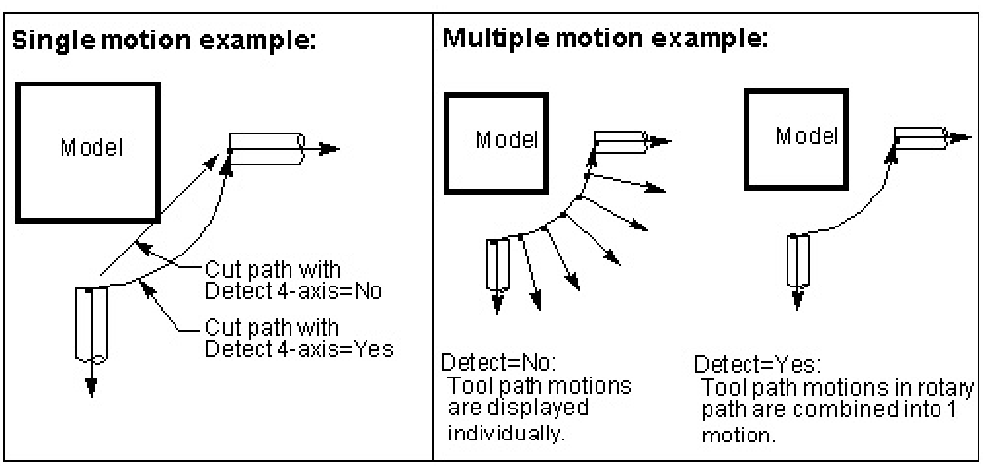
Shortest — Direction which travels the shortest distance. (Default)

CW — Always clockwise.

CCW — Always counterclockwise.

Linear — Refers to angles on a linear axis "wrapped" around the rotary component. Rotary values specify absolute locations along the linear-rotary axis, while the sign (+ / -) controls which end of the linear axis is used.

Detect 4-Axis Rotary Motion — When selected, simulates an arc cut for multi-axis motions that fit within an arc path. Clear this checkbox to simulate each multi-axis motion independently- recommended when simulating 5-axis contouring-type motions. The Interpolation Tolerance value is used to determine which motions are simulated in the 4-axis cut.



To APT Settings window

APT Settings window, Turning tab

Location: **Project menu > Process Options > APT Settings**

VERICUT toolbar short cut: 

Features on this tab provide control over lathe turning motions simulated for APT-CLS tool paths.

GOTO Format — Specifies how turning GOTO record values are to be displayed in a Workpiece view. In this view, VERICUT simulates turning motion in the ZX plane, where Z is the spindle-axis, and X is the cross-slide-axis. This feature acts like a post-processor for turning tool position data.

Example 1- Assume you have a turning tool path programmed in GOTO/x,y format. By default, **GOTO Format** "#zval, #xval" displays the first GOTO value as Z-axis motion, and the second value as X-axis motion.

Example 2- Assume you have a turning tool path that was reversed post-processed from G-Code data such that GOTO records are in GOTO/x,0,z format (center value is "zero"). For VERICUT to process this correctly, set **GOTO Format**= #xval, #yval, #zval.

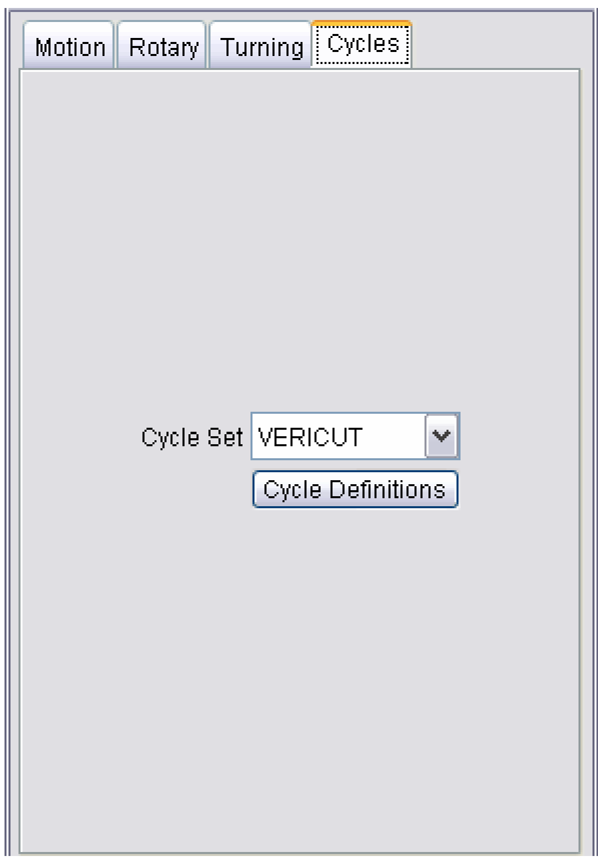
To APT Settings window

APT Settings window, Cycles tab

Location: **Project menu > Process Options > APT Settings**

VERICUT toolbar short cut: 

Features on this tab enable you to change how VERICUT simulates the APT or CLS CYCLE records to simulate tool path canned cycles, also known as "fixed tool axis cycles". The Drill Cycle feature controls when and how these records are simulated.



Cycle Set — Controls which cycle definition set(s) are considered by VERICUT when processing APT tool paths. If you add a new cycle set, for a different flavor of APT or a different post-processor, you can give it any name you wish and it will appear in this list. "ALL" considers all cycle sets defined for use.

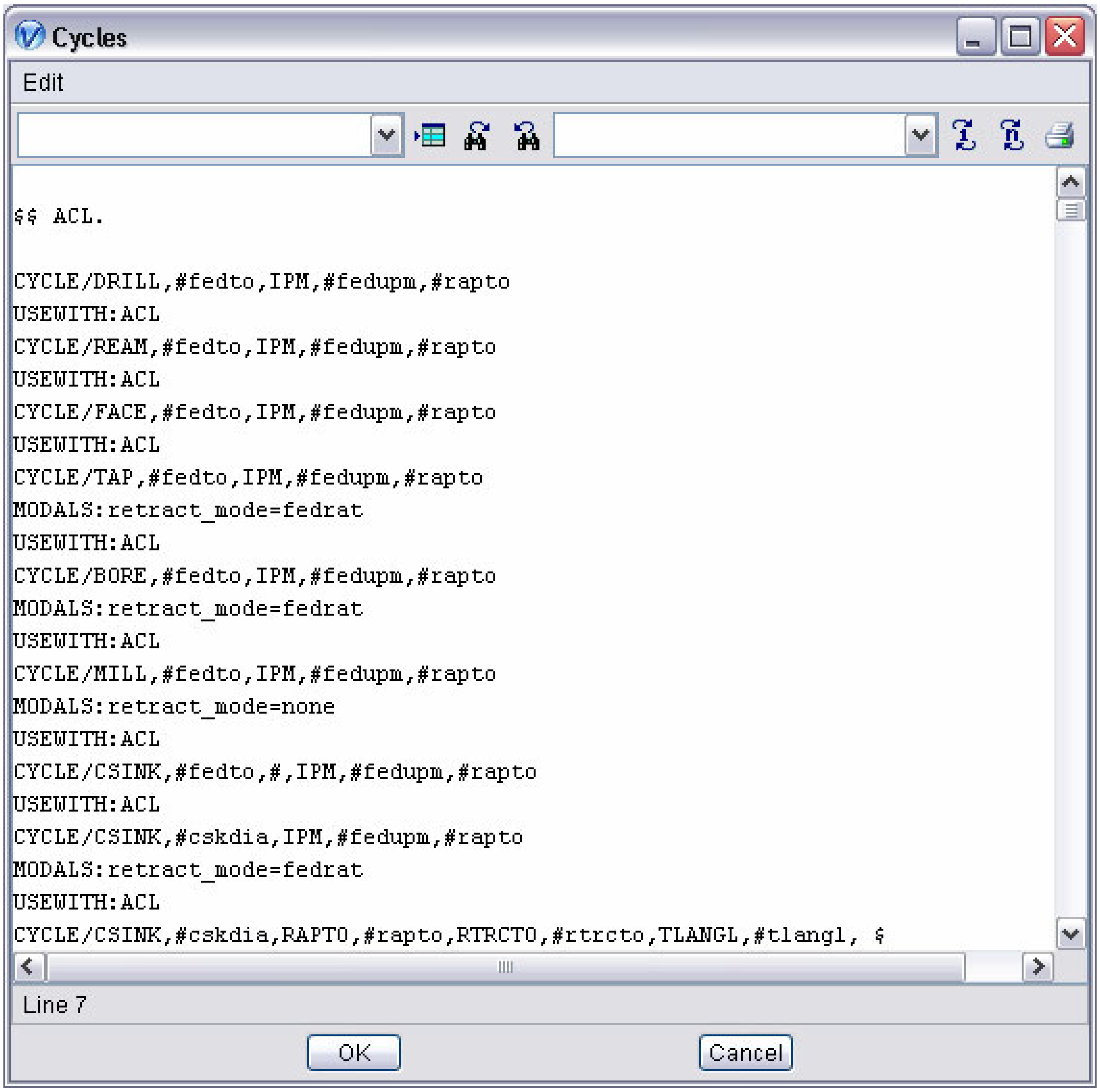
Cycle Definitions — Opens the Cycles window enabling you to maintain cycle and modals definitions used to interpret "CYCLE" records. This feature is especially useful if simulated cycle motions are not displayed as expected. When VERICUT can not find definitions to interpret a CYCLE record, an error similar to "Invalid CYCLE record" error is issued. In these cases, cycle definitions must be modified to interpret the cycle data.

For more information about configuring VERICUT to interpret cycle commands, see "**About configuring for cycle simulation**" in the *Using VERICUT* section. *Using VERICUT* can be found in the *CGTech Help Library*.

To APT Settings window

Cycles window

Accessed using the **Cycle Definitions** button on the APT Settings window, Cycles tab, the Cycles window features enable you to maintain cycle and modals definitions used to interpret "CYCLE" records. This feature is especially useful if simulated cycle motions are not displayed as expected. When VERICUT can not find definitions to interpret a CYCLE record, an error similar to "Invalid CYCLE record" error is issued. In these cases, cycle definitions must be modified to correctly interpret the cycle data.



Menu Bar:

The menu bar located at the top of the window provides easy access to major functions. Left-click on any menu name to display the list of functions in that menu. Click on the function in the menu you want to use.

Edit:

Cut — Cuts the highlighted text in the file listing and puts it in the paste buffer.

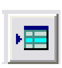
Copy — Copies the highlighted text in the file listing to the paste buffer.

Paste — Puts the contents of the paste buffer at the location of the cursor in the file listing.

Icon Bar:

The items in the Icon Bar enable you to search for, and/or replace, specific items in the file listing, print the file, or exit the window. Moving the cursor over the icon will display name of the option. Each feature (from left to right) is described below.

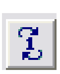
Line Number or Search Text — Use this text field to enter a line number, or a string of text, to search for.

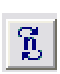
 **Goto Line Number —** Moves the cursor in the file listing to the line number specified in the Line Number or Search Text field.

 **Search Forward —** Searches forward in the file listing for the text string specified in the Line Number or Search Text field.

 **Search Backward —** Searches backward in the file listing for the text string specified in the Line Number or Search Text field.

Replacement Text — Use this text field to enter a "replacement text" string.

 **Replace One —** Replace one occurrence of text string in the Line Number or Search Text field (or the highlighted text in the file listing) with the text string in the Replacement Text field.

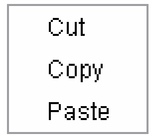
 **Replace All —** Replace all occurrences of the text string in the Line Number or Search Text field with the text string in the Replacement Text field.

 **Print —** Print the file.

OK — Accepts the changes and closes the Cycles window.

Cancel — Closes the Cycles window with out accepting the changes.

Shortcut: Right-click in the Cycles window to display the following menu:



These features provide the same functionality as those available under Edit in the menu bar.

Supported APT-CLS tool path records

APT tool path records typically contain a major vocabulary word followed by a slash ("/"), then a combination of data and/or minor vocabulary words. APT records can be preceded by leading spaces or tabs. Spaces can appear anywhere in the record. APT record data typically ends prior to column 72. However, a "$" at the end of a record continues onto the following line.

The APT tool path records processed by VERICUT are listed below in alphabetical order. See the appropriate section for details. Records not listed are typically ignored by VERICUT.

ARC record

Causes circular motion. The format of this record is similar to CIRCLE record, except specifies the arc end point via an additional set of X, Y, Z values.

CATIA0 record

Specifies a CATIA-style twelve parameter tool path transformation matrix. VERICUT processes this record when **Process Matrix**, on the **APT Settings window: Motion tab**, is active.

Example:

$$*CATIA0

$$*AXS2

$$ .00000 .00000 1.00000 -6.59040

$$ .99360 .11299 .00000 -16.50458

$$ -.11299 .99360 .00000 -3.32025

CHGTOOL record

Specifies the tool number to be changed into the spindle, same as LOADTL record. This record is not processed by default, however, you can use **Tool Change By**, in the **NC Program window**, to process this record and retrieve cutting tool descriptions from a VERICUT tool library.

Format:

CHGTOOL/n[,any text]

CIRCLE record

Causes circular motion. The GOTO point preceding the CIRCLE record defines the start of the circular motion. The GOTO following the CIRCLE defines the circular motion end. Helical motion is generated when the end point measured along the tool axis differs from the start point. This record is processed when **Process Circles**, on the **APT Settings window: Motion tab**, is active.

Formats:

CIRCLE/x,y,z,i,j,k,r[,n,n,n,n] - where "x,y,z" defines the circle center point, "i,j,k" is the axis orientation. "r" is the circle radius, but is not used since the radius is computed from the start, end and center points. The vector defines the positive Z-axis of counter-clockwise motion following the right-hand rule. In the 11 parameter format, VERICUT only uses the first 7 parameters.

Examples:

CIRCLE/1,2,3,0,0,1,5 - CCW circle motion about point 1,2,3, radius of 5

CIRCLE/1,2,3,0,0,-1,5 - as above, except CW arc motion

CLEARP record

Defines a clearance plane used by subsequent RETRCT records. See also: **Default CLEARP** on the **APT Settings window: Motion tab**.

Formats:

CLEARP/OFF

CLEARP/value

CLEARP/XYPLAN(YZPLAN or ZXPLAN),value - establish a clearance plane parallel to the part coordinate XY, YZ, or ZX plane, respectively at "value" distance from the part coordinate system origin. Only one plane may be active at one time. If no plane option is specified, XYPLAN is assumed.

CLEARP/PERPTO,value - establishes a clearance plane perpendicular to the tool axis at "value" distance from the part coordinate system origin. Clearance plane distance is measured in the direction of the tool axis. Changing the tool axis vector changes the clearance plane.

CLRPLN record

Defines a clearance plane, same as CLEARP record.

CLRSRF record

Defines a clearance plane, same as CLEARP record.

CNTRL record

Defines a control point for use with a NURBS record.

Formats:

CNTRL/x,y,z[,i,j,k][,WEIGHT,w] - where "x,y,x" are the control points (or poles) for tool position, optional "i,j,k" values control the tool axis, and optional "w" specifies the weights.

CONT record

GOTO continuation record, similar to GOTO record.

Formats:

CONT/x,y,z

x,y,z

:

CONT/x,y,z,i,j,k

x,y,z,i,j,k

:

COOLNT record

Controls coolant usage. Text following the slash is displayed in the **Status window** as the current coolant condition.

Formats:

COOLNT/any text

COOLNT/ON[,any text]

COOLNT/OFF

CSYS record

Specifies a ProManufacturing-style twelve parameter tool path transformation matrix. VERICUT processes this record when conditions are suitable. A more detailed explanation follows. The "$$->FEATNO" record resets back to the identity matrix, and "$$->END" ends TRANS/ROTATE/CSYS actions.

Example:

$$-> CSYS / 1.0000000000, 0.0000000000, 0.0000000000, 0.0000000000, $

0.0000000000, 1.0000000000, 0.0000000000, 0.0000000000, $

0.0000000000, 0.0000000000, 1.0000000000, 0.0000000000

ProManufacturing APT has combinations of options that determine how **ROTATE**, **TRANS**, and **CSYS** are applied. For VERICUT to determine how to apply these transformations, all records are processed concurrently. The table below describes cases where **ROTATE**, **TRANS**, and **CSYS** are processed. VERICUT ignores these records for cases that are not described in the table.

"ROTATE" record present	**"TRANS" record present**	**"$$->TRANS" present**	**Action**
No	No	No	Ignore CSYS and TRANS
No	No	Yes	Apply TRANS
No	Yes	No	Apply CSYS
Yes	No	Yes	Add TRANS to CSYS, then apply CSYS
Yes	Yes	No	Apply CSYS

CUTCOM record

CUTCOM record supports cutter compensation for ProManufacturing CL data. If VERICUT encounters:

$$-> CUTCOM_GEOMETRY_TYPE / OUTPUT_ON_PROFILE

Then sees a CUTCOM/LEFT, or CUTCOM/RIGHT, it reads ahead to the CUTCOM/OFF. The profile is then offset to the left or right by the cutter radius and VERICUT re-reads the APT, making adjustments as required. The algorithm used is not intended to match any particular postprocessor or controller.

If OptiPath is being used, the OUTPUT_ON_PROFILE gets replaced by OUTPUT_ON_CENTER in the generated file, and the subsequent GOTOs refer to the cutter centerline.

CUTTER record

Describes the shape of the cutting tool.

This record is processed by default, however, you can use **Tool Change By**, in the **NC Program window**, to ignore these and process other records in the tool path file to retrieve cutting tool descriptions from a VERICUT tool library.

Formats:

CUTTER/d,r,e,f,a,b,h

CUTTER/d,r,h

CUTTER/d,r (requires **Min. Cutter Height** > 0)

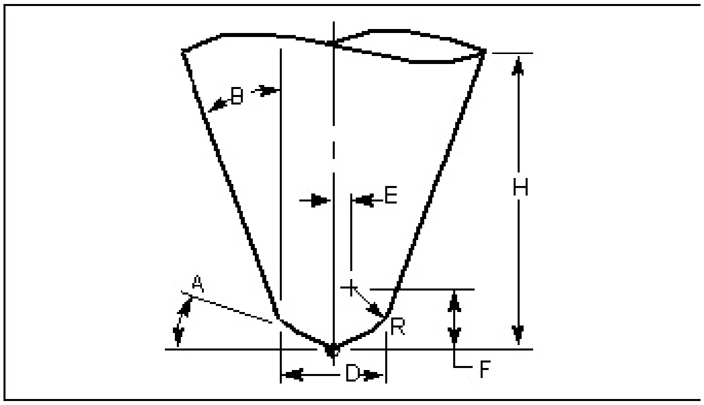
CUTTER/d (requires **Min. Cutter Height** > 0)

CUTTER/XCUT(SHANK or HOLDER),...profile description

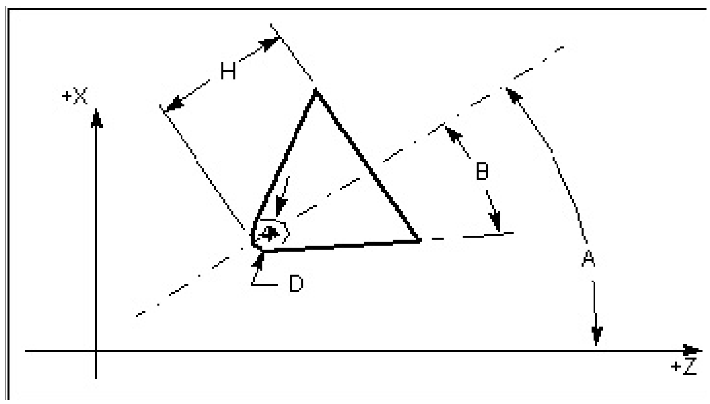
The table below and figures that follow describe how the values "d" through "h" are interpreted, based on if the cutter is used for milling or turning.

Value	Milling Cutter	Turning Cutter
d	diameter	nose diameter
r	corner radius	not used
e	dist. from tool centerline to corner radius center	not used
f	dist. from tool tip to corner radius center	not used
a	base angle	tool angle- degrees from spindle axis
b	side angle	side angle- degrees from tool centerline to its side
h	height	height- measured from tool tip along its centerline

Generic 7 Parameter Mill Cutter Definition: CUTTER/d,r,e,f,a,b,h



Generic 4 Parameter Turn Cutter Definition: CUTTER/d,0,0,0,a,b,h (or CUTTER/d,,,,a,b,h)



CYCLE record

Describes a canned tool axis cycle motion (e.g. drilling, reaming, boring, etc.) that is to be repeated at subsequent motion locations. The **Drill Cycle** feature, in the **Motion window**, controls when and how these records are simulated.

Formats:

CYCLE/<parameters> - where "<parameters>" provide information about how the canned motion is to be applied.

If canned cycle motions are not displayed as expected ensure that **Drill Cycle** is not set to **Ignore**. If cycle simulation is still not correct, use **Cycle Def** on the **APT Settings window: Cycles tab** to configure how VERICUT interprets canned cycle commands.

CYLNDR record

Causes circular motion, same as CIRCLE record.

END record

Ends the tool path transformation matrix caused by a combination of CSYS, ROTATE and TRANS records. VERICUT processes this record when conditions are suitable. See "CSYS record" for details.

FEATNO record

Resets the tool path transformation matrix caused by a combination of CSYS, ROTATE and TRANS records back to the identity matrix. VERICUT processes this record when conditions are suitable. See "CSYS record" for details.

FEDRAT record

This record sets the feed rate, or speed at which the tool moves. Text following the slash is displayed in the **Status window** as the current feedrate condition.

By default, a zero feedrate is used until a FEDRAT record is processed. The active feed rate value is used until another FEDRAT record is encountered. A RAPID motion overrides the active FEDRAT for that motion only.

Formats:

FEDRAT/value[,IPM] also: IPR, MMPM, MMPR - where “value” represents feedrate in the current mode. The feedrate value, and mode if specified, are displayed in the **Status window** Feedrate field.

FROM record

This record establishes where the tool is starting from. There is typically one record of this type per tool path file and it usually precedes any other motion record in the file. If a FROM record is not encountered, then the first GOTO point location is assumed to be the "from" position.

Formats:

FROM/x,y,z[,i,j,k] - where "x,y,z" is the starting point and "i,j,k" represents the starting tool axis vector. If no ijk information is found, then a tool axis of 0,0,1 is assumed.

GODLTA record

Designates incremental cutter movement relative to the current cutter location.

Formats:

GODLTA/x,y,z - specifies the incremental distance along each axis to move the tool from its current location. For example, if the tool is currently at 4, 5, 6 in xyz when GODLTA/1,2,3 is processed, the tool moves to x=5 (4+1), y=7 (5+2), and z=9 (6+3).

GODLTA/value - moves the cutter along the tool axis the specified amount.

If there is a transformation active, such as by a previous rotary motion command or a stock transformation, then the GODLTA values are transformed accordingly.

GOFWD record

Designates forward cutter movement relative to the current cutter location. VERICUT considers this record only when processing motion generated by GOFWD used with CIRCLE or CYLNDR records. Numerous formats are recognized, many of which also consider the INDIRV record to determine the "forward" motion direction.

GOHOME record

Specifies that the cutter return to the home position. The home position is assumed to be the location specified by the last FROM record processed. If a FROM record has not been processed, then 0, 0, 0 is used.

Formats:

GOHOME - move all axes to the home position. The **GOHOME Motion** feature, on the **APT Settings window: Motion tab**, controls if the cutter is moved in a straight line, or is retracted prior to going home.

GOHOME/XAXIS(YAXIS or ZAXIS) - treated as GOTO statements with the specified AXIS element set to the FROM value for that axis. The other two elements are set to the current tool path position values.

GOTO record

Designates a position where the tool is moved to.

Formats:

GOTO/x,y,z[,i,j,k] - where "x,y,z" represent the ending location of the tool motion and optional "i,j,k" values control the tool axis in multi-axis movements. "0,0,1" tool axis orientation is assumed. "GOTO/" text is optional after the first GOTO motion, for example:

GOTO/x,y,z[,i,j,k]

x,y,z[,i,j,k]

:

HEAD record

Found in tool paths for mill-turn and 4-axis lathe machines, the "HEAD" record specifies which machining head is in use.

Formats:

HEAD/1- head 1 active

HEAD/2- head 2 active

HEAD/BOTH- head 1 active

HEAD/OFF- head 1 active

When head 2 is active the tool is changed via a "**LOADTL**" or "**TURRET**" record, VERICUT prepends the tool ID specified by the **LOADTL/TURRET** record with the characters "2_" to retrieve tools defined for use by head 2.

Example 1:

HEAD/1

TURRET/1

Looks for ID "1" in the Tool Library (interpreted as tool 1 for use by head 1).

Example 2:

HEAD/2

TURRET/1

Looks for ID "2_1" in the Tool Library (interpreted as tool 1 for use by head 2).

INDIRV record

Vector coefficients describing the "forward" motion direction. VERICUT considers this record only when processing motion generated by a GOFWD record used with CIRCLE or CYLNDR records.

Format:

INDIRV/i,j,k

KNOT record

Defines a knot parameter for use with a NURBS record.

Format:

KNOT/t1,...,t(n-p+2) - where "t1" - "t(n-p+2)" are the knot parameters.

LOAD/TOOL record

Specifies the tool number to be changed into the spindle, same as LOADTL record. This record is not processed by default, however, you can use **Tool Change By**, in the **NC Program window**, to process this record and retrieve cutting tool descriptions from a VERICUT tool library.

Format:

LOAD/TOOL,n[,any text]

See also HEAD record for information about how the Pro/MFG APT "HEAD" record affects how this record is used to retrieve tool from a VERICUT tool library.

LOAD/WIRE record

For wire EDM machining- causes the wire to be loaded-sets cutting mode in VERICUT for wire EDM machining. See also UNLOAD/WIRE record.

LOADTL record

Specifies the tool number, or pocket number of the tool to be changed into the spindle. Optionally, this record may also specify tool length or other aspects about the tool, however, VERICUT ignores all parameters following the tool number.

This record is not processed by default, however, you can use **Tool Change By**, in the **NC Program window**, to process this record and retrieve cutting tool descriptions from a VERICUT tool library.

Format:

LOADTL/n[,any text] - where "n" is the index number of the tool (number or alpha-numeric). Leading zeros in the tool ID are also ignored, for example "LOADTL/01" is interpreted as "LOADTL/1". Any text following the "/" or number data after tool ID is ignored, for example "LOADTL/2,LENGTH,6.500" is interpreted as "LOADTL/2".

See also HEAD record for information about how the Pro/MFG APT "HEAD" record affects how this record is used to retrieve tools from a VERICUT tool library.

MODE record

Identifies the machining to be performed: mill or turn.

Formats:

MODE/MILL(TURN)

MOVARC record

Causes circular motion, same as CIRCLE record.

MSYS record

Specifies an NX-style nine parameter tool path transformation matrix. VERICUT processes this record when Process Matrix, on the **APT Settings window: Motion tab**, is active.

MULTAX record

Determines if subsequent GOTO motions are multi-axis motions. The multi-axis condition is modal. When multi-axis mode is on it remains on until another MULTAX record turns it off.

Formats:

MULTAX/ON

MULTAX/OFF

NURBS record

Defines a NURBS B-spline motion command. NURBS motion commands also utilize the CNTRL record and KNOT record as part of the B-spline definition.

Format:

NURBS/[p]- where "p" indicates the B-spline order. The "NURBS" record typically spans multiple lines and includes a number of other record types, such as "KNOT", "CNTRL", etc.

Example:

NURBS/

KNOT/1.0000000

CNTRL/61.2424,93.2577,-31.6260

:

CNTRL/59.8775,93.2224,-31.6263

PPRINT record (general purpose)

Comment record that typically has no affect on tool motion. "PPRINT" resides in columns 1-6. Text following "PPRINT" is typically ignored by VERICUT, however, some PPRINT records can perform VERICUT actions. See also "**Controlling VERICUT with comment records**" in the *Using VERICUT* section. *Using VERICUT* can be found in the *CGTech Help Library*.

PPRINT/ATP CBOX record

Defines a stock block via min/max corner points. This record is processed by VERICUT when a stock model is loaded and the model is not yet cut. A new stock block is defined and fit into all displayed views.

Format:

PPRINT/ATP CBOX xmin,ymin,zmin,xmax,ymax,zmax

Example:

PPRINT/ATP CBOX -1,-2,-1,5,7,2 -defines a 6 x 9 x 3 block positioned X= -1, Y= -2, Z= -1

PPRINT TOOL AXIS record

Specifies a new tool axis orientation to be applied to subsequent tool positions. The specified tool axis is modal and remains in effect until another tool axis orientation is specified.

Example:

PPRINT TOOL AXIS= XAX=-1.0000 YAX=.0000 ZAX=.0000

PPRINT/VERICUT record

(or **PPRINTVERICUT**) Sets a VERICUT user value, or performs a VERICUT command or action. PPRINT/VERICUT text must be between columns 1-72. PPRINT data can span multiple PPRINT records via ending each PPRINT to be continued in a dollar sign. The next line must be a PPRINT/VERICUT record with the same keyword as the previous line. Words in PPRINT data can not be split by the dollar sign. Examples are provided in the sections that follow.

See also "**Controlling VERICUT with comment records**" in the *Using VERICUT* section. *Using VERICUT* can be found in the *CGTech Help Library*.

RAPID record

Overrides the current feed rate and causes a machine tool to move as fast as possible. When the RAPID record is processed by VERICUT, the motion following the RAPID moves the tool using the **Fast Feed** setting in the **Motion window**.

For example, assume the file below is processed while **Fast Feed=100**.

RAPID

GOTO/1,2,3

FEDRAT/20

GOTO/2,3,4

RAPID

GOTO/3,4,5

GOTO/4,5,6

The motions are simulated by VERICUT as follows:

feed rate from the current tool location to 1,2,3: 100

feed rate from 1,2,3 to 2,3,4: 20

feed rate from 2,3,4 to 3,4,5: 100

feed rate from 3,4,5 to 4,5,6: 20

In both examples below, motion from 1, 2, 3 to 4, 5, 6 uses a feed rate of "20":

Example 1:

FROM/1,2,3

RAPID

FEDRAT/20

GOTO/4,5,6

In example 1, a FEDRAT record immediately follows the RAPID record before a motion is encountered. Under this condition, the FEDRAT value overrides a RAPID record.

Example 2:

RAPID

FROM/1,2,3

FEDRAT/20

GOTO/4,5,6

In example 2, a FROM record is not considered to be a motion record. Therefore, the RAPID record has no affect.

RETRCT record

Retracts the cutter along the tool axis to a specified clearance plane, or by a specified amount.

Formats:

RETRCT- retract the tool along the tool axis until it reaches the clearance plane defined by the previous CLEARP or CLRSRF record

RETRCT/value- retract along the tool axis the specified amount

RETRCT/ON- retract to the Z level prior to a cycle record.

Example:

GOTO/1,2,3

RETRCT/ON

CYCLE/DRILL,1.0,3,IPM,.1 $$ Retracts to Z=3 between cycle positions.

ROTABL record

Specifies a table rotation. Unless otherwise specified, rotation about the Y-axis is assumed. This record is processed when the **ROTABL** setting, on the **APT Settings window: Rotary tab**, is active. Otherwise, **ROTABL** is ignored.

Formats:

ROTABL/AAXIS(BAXIS or CAXIS),value

ROTABL/AAXIS(BAXIS or CAXIS),ATANGL,value

ROTABL/AAXIS(BAXIS or CAXIS),INCR,value

ROTABL/ATANGL,value

ROTABL/INCR,value

ROTATE record

Similar to the ROTABL record, this record specifies a table rotation. ROTATE is handled differently, depending on the active setting of **NC Program Type**, in the **NC Program window**.

1. Non-ProManufacturing tool path types - when the **ROTATE** setting, on the **APT Settings window: Rotary tab**, is active the rotary action is simulated by VERICUT. Otherwise, **ROTATE** is ignored.
2. **Pro/MFG APT** tool path types- ROTATE is considered along with TRAN and CSYS when determining if the tool path is to be transformed. See "CSYS record" for details. The "$$->FEATNO" record resets back to the identity matrix, and "$$->END" ends TRANS/ROTATE/CSYS actions.

Formats:

ROTATE/AAXIS(BAXIS or CAXIS),value

ROTATE/AAXIS(BAXIS or CAXIS),ATANGL,value

ROTATE/AAXIS(BAXIS or CAXIS),INCR,value

ROTATE/ATANGL,value

ROTATE/INCR,value

ROTHED record

Specifies a head rotation. This record is processed when the **ROTHED** setting, on the **APT Settings window: Rotary tab**, is active. Otherwise, **ROTHED** is ignored.

Formats:

ROTHED/ATANGL,value

ROTHED/value

SET/MODE record

This record specifies the machining to be performed: mill or turn.

Formats:

SET/MODE,MILL

SET/MODE,TURN

SPINDL record

Specifies spindle rotation speed (RPM); direction and gear ranges may also be specified. Text following the slash is displayed in the **Status window** as the current spindle condition.

Formats:

SPINDL

SPINDL/rpm[,any text]

SPINDL/[any text,]rpm

SPNDL record

Specifies spindle rotation speed (RPM), same as SPINDL record.

STOP record

Stops the NC machine at its current location. This record commonly indicates an operator function follows, such as: changing the tool, changing the part-fixture setup, etc.

SURFACE record

Causes circular motion, same as CIRCLE record.

TLAXIS record

Specifies a new tool axis orientation. The new orientation is modal and is applied to following GOTO points that do not have I, J, K values. VERICUT processes this record when **NC Program Type**, in the **NC Program window**, is set to **CATIA APT**.

Example:

TLAXIS/ .000000, 1.000000, .000000

TRANS record

Specifies a three parameter tool path translation. VERICUT processes this record when **NC Program Type**, in the **NC Program window**, is set to **Pro/MFG APT** and conditions are suitable. The presence of ROTATE and TRANS records affect when VERICUT processes a TRANS record. See "CSYS record" for details.

Format:

TRANS / dx, dy, dz

where "dx, dy, dz" are the distances in X, Y, and Z to translate the tool positions that follow. The "$$->FEATNO" record resets back to the identity matrix, and "$$->END" ends TRANS/ROTATE/CSYS actions.

TURRET record

Similar to LOADTL record, this record often appears in tool paths for turning operations to specify the tool number to be changed into location for cutting. This record is not processed by default, however, you can use **Tool Change By**, in the **NC Program window**, to process this record and retrieve cutting tool descriptions from a VERICUT tool library.

The number following "TURRET" is often a composite of two numbers: a tool number and offset number. However, VERICUT simply interprets the entire number as a tool number. For example, while "TURRET/0101" may indicate tool 01 is used with offset 01), VERICUT interprets this to mean tool 0101 (or 101) is to be used from the VERICUT tool library.

Formats:

TURRET/n[,any text]

TURRET/FACE,n[,any text]

See also **HEAD record** for information about how the Pro/MFG APT "HEAD" record affects how this record is used to retrieve tools from a VERICUT tool library.

UNITS record

Sets the unit measurement system of the data in the tool path file: inches or millimeters.

Formats:

UNITS/INCHES

UNITS/MM

UNLOAD/WIRE record

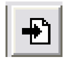
For wire EDM machining- causes the wire to be unloaded-sets non-cutting mode in VERICUT for wire EDM machining. See also LOAD/WIRE record.

VECTOR record

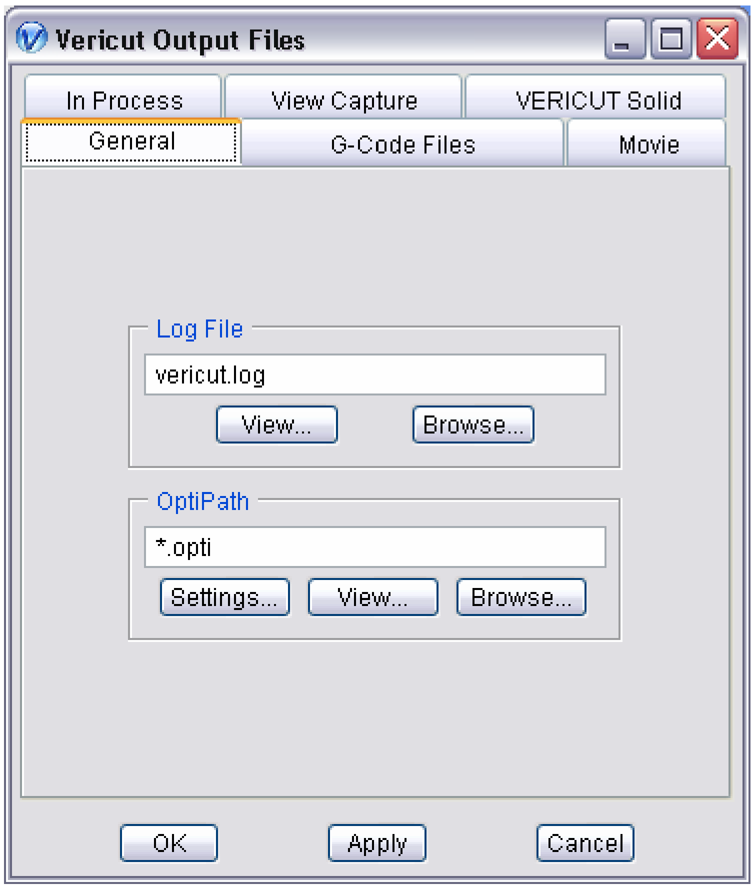
Designates a position where the tool is moved to, same as GOTO record.

Output window

Location: **Project menu > Output**

VERICUT toolbar short cut: 

Opens the VERICUT Output files window to setup, create, or view VERICUT's many output files.



In Process tab — Features on this tab control automatic saving of In Process files, or "IP files". The features on this tab provide the same functionality as the **AutoSave window, In Process tab** found in the *File* section.

View Capture tab — Features on this tab control automatic saving of View Capture image files (JPEG, PS, EPSF, or TIFF files).

VERICUT Solid tab — Features on this tab control automatic saving of VERICUT Solid (vct.) files.

General tab **—** Features on this tab enable you to specify and view VERICUT Log files, and OptiPath modified (optimized) NC Program files. It also provides access to the **OptiPath Control window**.

G-Code Files tab **—** Use the features on this tab to specify and view the G-Code Log file and the APT Output file.

Movie tab **—** Use the features on this tab to specify an output file, record, and view VERICUT Image files or AVI files.

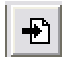
OK — Saves all tab settings and closes the VERICUT Output Files window.

Apply — Saves all tab settings and leaves the VERICUT Output Files window open.

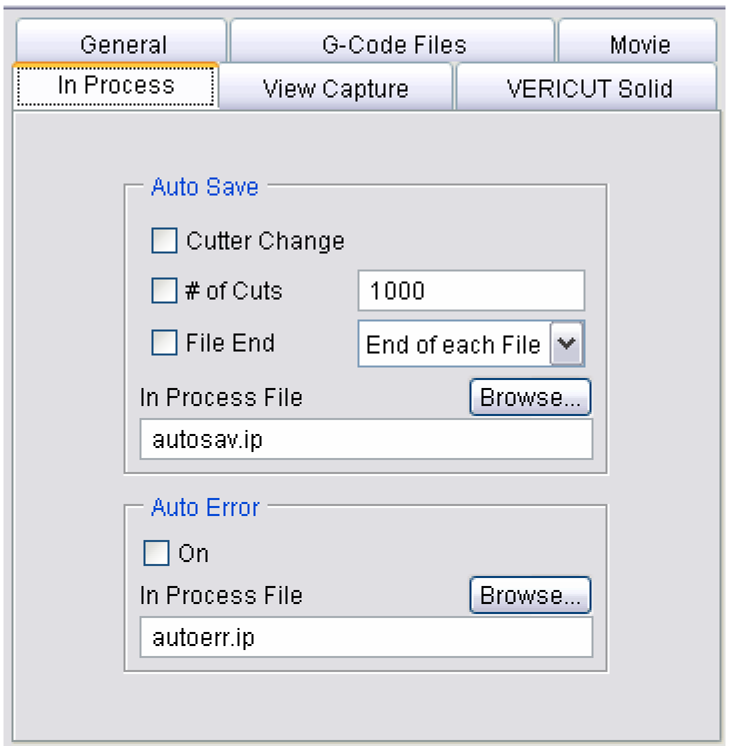
Cancel — Closes the VERICUT Output Files window without saving settings.

Output window, In Process tab

Location: **Project menu > Output window**

VERICUT toolbar short cut: 

Features on this tab control automatic saving of In Process files, or "IP files".



Auto Save options

These options control when VERICUT automatically saves IP files. Click on the box to the left of the event to toggle On/Off. Each selected event causes a file to be saved when that event occurs.

Options:

Cutter Change — When toggled "On", an In Process file is saved whenever the cutter has been changed.

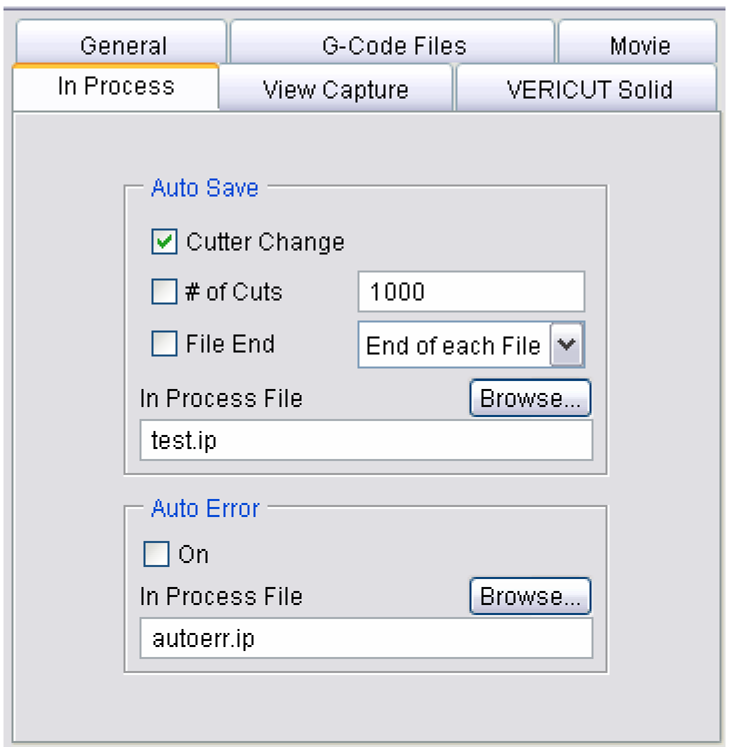
# of Cuts — When toggled "On", an In Process file is saved after a specified number of cuts. Enter the number of cuts in the **# of Cuts** text field.

File End — When toggled "On", an In Process file is saved at the end of tool path processing. Choose either **End of each File** (end of each toolpath file) or **End** (end of the NC program) from the pull-down list.

In Process File — Use to specify the base name for In Process files saved. Enter the *\path\filename* in the **In Process File** text field, or click on **Browse** and use the selection window that displays to specify the file.

Unique file names are maintained by appending the tool path number and line number of the record causing the event to the end of the base file name. Naming convention for auto-saving files is: <IPfilename>n1_n2.ip where "<IPfilename>" is the name specified in the In Process File field, "n1"=sequential tool path number, and "n2"=number of the record causing the auto-save event.

The following examples assume the Output window configuration shown below.



Example 1: One tool path processed, tool change on record #10 => test1_10.ip

Example 2: Two tool paths processed, tool change on record #10 in both tool paths => test1_10.ip (from tool change in tool path #1), test2_10.ip (from tool change in tool path #2)

Example 3: Two tool paths processed, tool path one tool change on record #10, tool path two tool change on record #5 => test1_10.ip (from tool change in tool path #1), test2_5.ip (from tool change in tool path #2)

Auto Error options

Options:

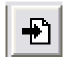
On — When active, VERICUT automatically saves an IP file when an error is detected.

In Process File — Specifies the base name for IP files saved due to errors. Similar to In Process File for the Auto Save options (see above), unique file names are maintained by appending the line number of the record causing the error to the end of the specified IP file name.

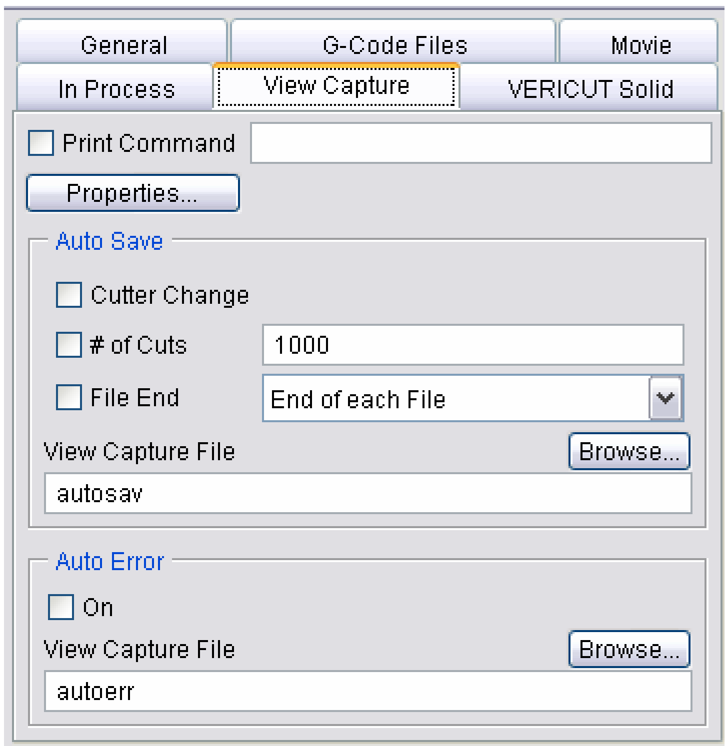
To Output window

Output window, View Capture tab

Location: **Project menu > Output**

VERICUT toolbar short cut: 

Features on this tab control automatic saving of View Capture image files (JPEG, PS, EPSF, or TIFF).



Print Command — When toggled "Off", sends graphical data to the specified View Capture image file. When toggled "On", executes the print command that sends raw graphical image data to the printer.

By default, the "prshade" command executes the "prshade" command file containing operating system commands typically used to print images on your computer. If printing fails, correct the entry in the Print Command field, or use an ASCII text editor (***NOT*** a word processor) to edit the "prshade" command file to have the proper print command for your computer/printer.

Properties — Opens the View Capture window to access settings for file formatting and printing.

Auto Save options

These options control when VERICUT automatically saves View Capture image files. Click on the box to the left of the event to toggle On/Off. Each selected event causes a file to be saved when that event occurs.

Options:

Cutter Change — When toggled "On", a View Capture image file is saved whenever the cutter has been changed.

# of Cuts — When toggled "On", a View Capture image file is saved after a specified number of cuts. Enter the number of cuts in the **# of Cuts** text field.

File End — When toggled "On", a View Capture image file is saved at the end of tool path processing. Choose either **End of each File** (end of each toolpath file) or **End** (end of the NC program) from the pull-down list.

View Capture File — Use to specify the base name for View Capture image files saved as a result of using the **Auto Save** options described above, when the **Print Command** checkbox is toggled "Off". Enter the *\path\file* name in the **View Capture File** text field, or click on **Browse** and use the selection window that displays to specify the file. The naming convention used for saved files is the same as described for In Process files. See the **In Process tab** section for details.

Auto Error options

These options control when VERICUT automatically saves View Capture files due to errors.

Options:

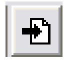
On — When toggled "On", VERICUT automatically saves a View Capture image file when an error is detected.

View Capture File — Use to specify the base name for View Capture image files automatically saved as a result of errors detected when AutoError **On** is toggled "On", and the **Print Command** checkbox is toggled "Off". Enter the *\path\file name* in the **View Capture File** text field, or click on **Browse** and use the **Save File** selection window that displays to specify the file. The naming convention used for saved files is the same as described for In Process files. See the **In Process tab** section for details.

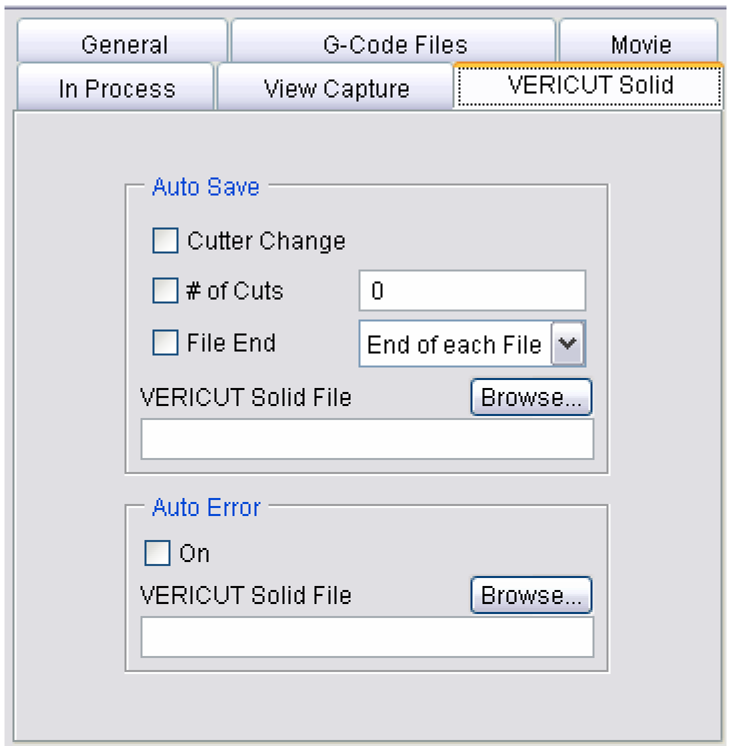
To Output window

Output window, VERICUT Solid tab

Location: **Project menu > Output**

VERICUT toolbar short cut: 

Features on this tab control automatic saving of VERICUT Solid (vct.) files.



Auto Save options

These options control when VERICUT automatically saves VERICUT Solid files. Click on the box to the left of the event to toggle On/Off. Each selected event causes a file to be saved when that event occurs.

Options:

Cutter Change — When toggled "On", a VERICUT Solid file is saved whenever the cutter has been changed.

# of Cuts — When toggled "On", a VERICUT Solid file is saved after a specified number of cuts. Enter the number of cuts in the **# of Cuts** text field.

File End — When toggled "On", a VERICUT Solid file is saved at the end of tool path processing. Choose either **End of each File** (end of each toolpath file) or **End** (end of the NC program) from the pull-down list.

VERICUT Solid File — Use to specify the base name for VERICUT Solid files saved due to **Auto Save** options. Enter the *\path\filename* in the **VERICUT Solid File** text field, or click on **Browse** and use the file selection window that displays to specify the file.

AutoError options

Use these options to specify whether or not VERICUT automatically saves VERICUT Solid files when an error is detected.

Options:

On — When toggled "On", VERICUT automatically saves a VERICUT Solid file when an error is detected.

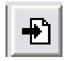
VERICUT Solid File — Use to specify the base name for VERICUT Solid files saved when AutoError **On** is active. Enter the *\path\file name* in the VERICUT Solid File text field, or click on **Browse** and use the file selection window that displays to specify the file.

NOTE: The naming convention used for saved VERICUT Solid files is the same as described for In Process files. See the **In Process tab** section for details.

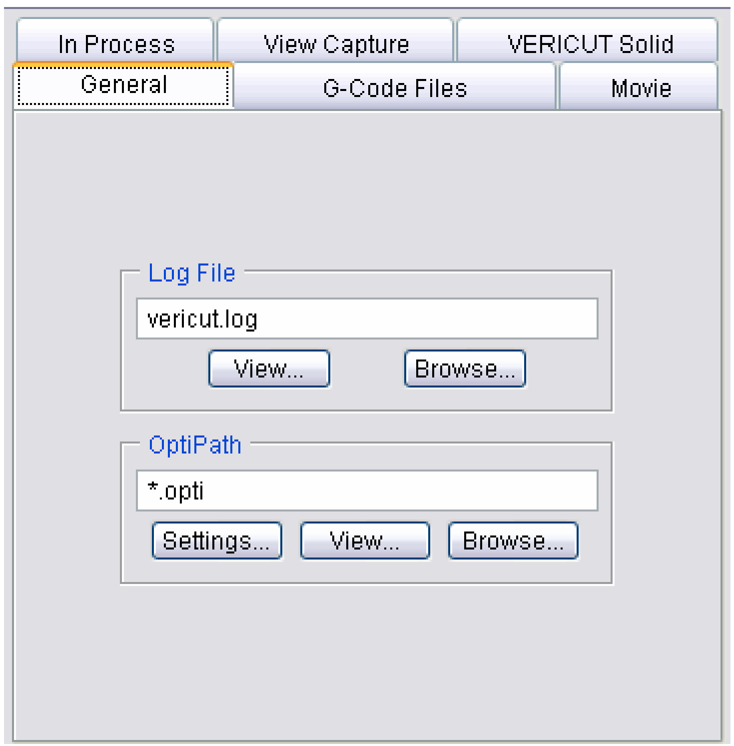
To Output window

Output window, General tab

Location: **Project menu > Output**

VERICUT toolbar short cut: 

Features on this tab enable you to specify and view VERICUT Log files, and OptiPath modified (optimized) NC Program files. It also provides access to the **OptiPath Control window**.



Log File

Use to specify the name of the VERICUT Log file. Enter the *\path\file name* in the **Log File** text field, or click on **Browse** and use the file selection window that displays to specify the file.

View — Opens a window containing the VERICUT Log file. This file contains session information, such as error, warning and informational messages about the verification session. Standard text editing features are provided, such as: copy/cut, paste, search, etc. For more information on using the editing features, see **Text File** under **Edit menu**.

OptiPath

Use to specify the name of the Optimized file. Enter the *\path\filename* in the **OptiPath** text field, or click on **Browse** and use the file selection window that displays to specify the file.

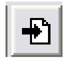
Settings — Displays the OptiPath Control window.

View — Opens a window containing the optimized NC program file. Standard text editing features are provided, such as: copy/cut, paste, search, etc. For more information on using the editing features, see **Text File** under **Edit menu**.

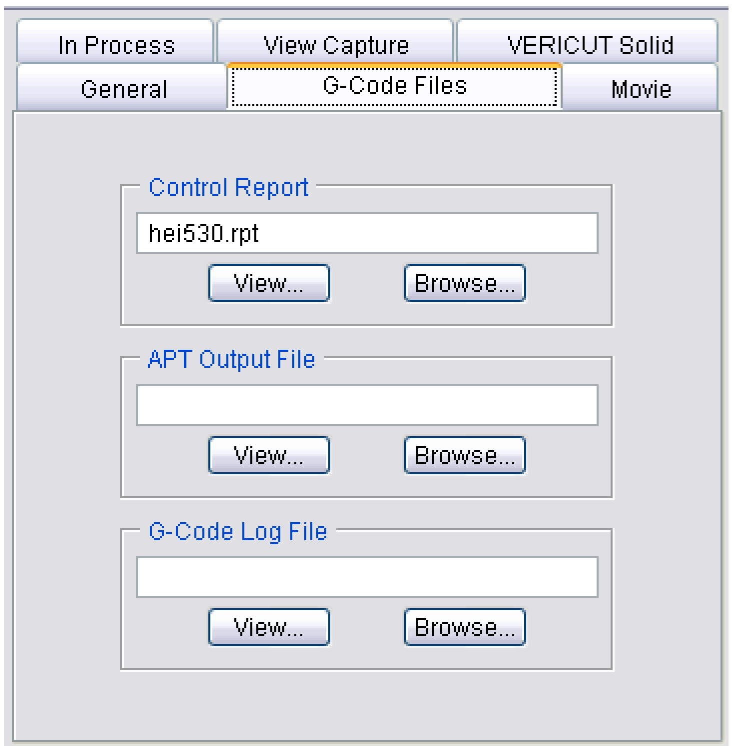
To Output window

Output window, G-Code Files tab

Location: **Project menu > Output**

VERICUT toolbar short cut: 

Use the features on this tab to specify, and view, Control Reports, APT Output Files and the G-Code Log Files.



Control Report

Enter the \ *path\filename* of the Control Report file in the **Control Report** text field, or use the **Browse** button and use the file selection window that displays to specify it. This file contains information about how the current NC control configuration will interpret various codes.

View — Opens a window containing the Control Report file. In this window, you can view, edit, or reset the Control Report file contents. Standard text editing features are provided, such as: copy/cut, paste, search, etc. For more information on using the editing features, see **Text File** under **Edit menu**.

APT Output File

Enter the *path**filename* of the APT Output file in the **APT Output File** text field, or use the **Browse** button and use the file selection window that displays to specify it. This file contains information, such as error, warning and informational messages about G-Code processing.

View — Opens a window containing the APT Output file with ASCII APT tool path records created when Create APT Output File is active (ref. **Process Options window: G-Code Output Files tab**). Standard text editing features are provided, such as: copy/cut, paste, search, etc. For more information on using the editing features, see **Text File** under **Edit menu**.

G-Code Log File

Enter the *path**filename* of the G-Code Log file in the **G-Code Log File** text field, or use the **Browse** button and use the file selection window that displays to specify it. This file contains information, such as error, warning and informational messages about G-Code processing.

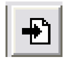
View — Opens a window containing the G-Code Log file. In this window, you can view, edit, or reset the G-Code Log file contents. Standard text editing features are provided, such as: copy/cut, paste, search, etc. For more information on using the editing features, see **Text File** under **Edit menu**.

Click here to see a sample G-Code Log file.

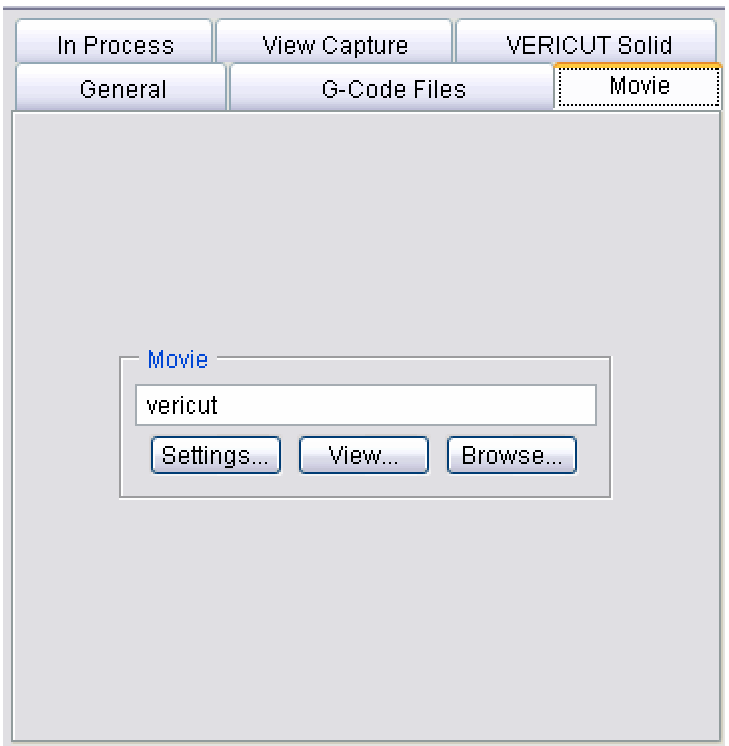
To Output window

Output window, Movie tab

Location: **Project menu > Output**

VERICUT toolbar short cut: 

Use the features on this tab to specify an output file, record, and view VERICUT Image files or AVI files.



Movie

Use to specify the name of the Image/AVI output file. Enter the *path**filename* in the Movie text field, or click on **Browse** and use the file selection window that displays to specify the file.

Settings — Displays the Image Record window.

View — Opens a "VERICUT Player" window (or "AVI Player" window) to load and play back an Image file (or AVI file). The type of "player" window that opens depends on the Output Format setting in the Record Movie window.

To Output window

Report (Report Template window)

Location: **Project menu > Report > Report Template >Edit** in the main VERICUT window.

File > Report Template in the Inspection Sequence window.

File > Report Template in the Tool Manager window.

Opens the Report Template window enabling you to define a new, or modify an existing, report template.

The Menu Bar

File

New — Displays an empty Report Template window enabling you to create a new report template file.

Open — Displays a file selection window enabling you to open an existing report template file. The "current" report template file is stored in the Preferences file.

Save — Opens a window enabling you to save the current report template file.

Save As — Opens a window enabling you to save the current report template file under a different name or path.

Recent Files — Provides a list of recently opened Report Template files. To open a file in the list, select the desired file. List contents are stored in the Preferences file.

Close — Closes the Report Template window without saving the report template file.

Edit

Add — Enables you to add records to a new or existing report template file.

Edit — Enables you edit the content of the highlighted record. This feature is only available for the Page Layout tab.

Delete — Enables you to delete the highlighted record from the report template file.

The Tabs

Page Layout tab **—** Features on this tab enable you to specify or modify the content and page layout of a report template.

Styles tab **—** Features on this tab enable you to specify display characteristics (font, font size, color, etc.) to be used for the report template.

User-Defined Tags tab **—** Features on this tab enable you to specify "generic" records included in a report template that will be updated with information specific to a particular user file at the time the report is created. The user file specific information can be associated with the User-Defined Tag using the **User-Defined Tag Values window**. If all User-Defined Tag values are not defined prior to creating a report, you will be prompted for any missing information at the time the report is created.

Page Setup tab **—** Features on this tab enable you to specify or modify page format characteristics such as margins, orientation and paper size. The features on this tab are intended to be use with PDF format reports, although it may also be useful for HTML format reports with certain web browsers.

Also see "**Working with VERICUT Reports**", in the *Using VERICUT* section, in the *CGTech Help Library*.

Report Template window, Page Layout tab

Location: **Project menu > Report > Report Template >Edit** in the main VERICUT window.

File > Report Template in the Inspection Sequence window.

File > Report Template in the Tool Manager window.

Category	Type	Content	Alignment	Style
First Page Head...	Text	VERICUT Report	Center	Title
First Page Head...	Table	Custom Table	Center	Normal
Body	Table	Custom Table	Left	Normal
Body	Text	File Summary	Center	Heading
Body	Table	File Summary Table	Left	Normal
Body	Text		Left	Normal
Body	Table	User Table	Left	Normal
		PageBreak		
Body	Text	Shade Copy	Center	Heading
Body	Table	Shade Copy Table	Left	Normal
		Page Break		
		Start Toolpath loop		
Body	Table	Custom Table	Left	Normal
Body	Text	Tool Summary	Center	Heading
Body	Table	Tool Summary Table	Left	Normal
Body	Text		Left	Normal
Body	Text	Tool Use Graph	Center	Heading
Body	Text	Time	Center	Normal
Body	Picture	Tool Use Graph	Center	Normal
Body	Table	Custom Table	Center	Normal
Body	Text		Left	Normal
Body	Text	Cutting Graph	Center	Heading
Body	Picture	Cutting Graph	Center	Normal
Page Footer	Text	[Full Page Number]	Center	Normal
		Page Break		
		End Toolpath Loop		
Body	Text	Total Tool Use Graph	Center	Heading
Body	Text	Time	Center	Normal
Body	Picture	Tool Use Graph	Center	Normal
Body	Table	Custom Table	Center	Normal

The Menu Bar

File

See Report Template window, above, for a description of the **File** features.

Edit

Add — Enables you to add records to a new or existing report template file. The following features are available for use in report templates:

NOTE: When starting a new report template, Add will not become active until at least one style record has been added using the Styles tab.

Text

Use to add text related records to a report template. Select the desired option from the pull-down list. The options in the list will vary depending on where you accessed the Report Template window (VERICUT main window, Tool Manager window, or the Inspection Sequence window).

String — Adds a record to write out a specified text string. Enter the text in the "Content" field of the record.

Date — Adds a record to write out the date with the format selected from the pull-down list.

Short — "12/24/03"
Medium — "Dec 24, 2003"
Long — "December 24, 2003"
Full — "Wednesday, December 24, 2003"

Time — Adds a record to write out the time with the format selected from the pull-down list.

Short — "10:49 AM"
Medium — "10:49:48 AM"
Long — "10:49:48 AM PST"

Page Number — Adds a record to write out page numbers with the format selected from the pull-down list.

 # — "1"
Page # — "Page 1"
Page # of # — "Page 1 of 3"

File Name

Adds a record to write out the file name of the file type selected from the pull-down list. Choose either, **Full Path** (to write out the file name in the format /*path*/*filename*), or **File Name** (to write out the file name only), then select the file type from the pull-down list.

Project File — Adds a record to write out the file name of the "current" Project file.
Machine File — Adds a record to write out the file name of the "current" Machine file.
Control File — Adds a record to write out the file name of the "current" Control file.
Tool Library File — Adds a record to write out the file name of the "current" Tool Library file.
Design Model File — Adds a record to write out the file name of the "current" Design Model file.
NC Program — Adds a record to write out the file name of the "current" NC Program File.
Log File — Adds a record to write out the file name of the "current" Log file.
Optimized NC Program File — Adds a record to write out the file name of the "current" Optimized Toolpath file.

Process Data — Adds a record to write out "process" related data of the type selected from the pull-down list.

Material — Adds a record to write out the "material" specified on the **OptiPath Control window, Settings tab**.
Machine — Adds a record to write out the "machine" specified on the **OptiPath Control window, Settings tab**.
Total Time — Adds a record to write out the total time anticipated to machine the part (as simulated by VERICUT).
Total Number of Errors — Adds a record to write out the number of errors detected by VERICUT.
Total Number of Warnings — Adds a record to write out the number of warning detected by VERICUT.
Total Optimized Time —Adds a record to write out the total time anticipated to machine the part (as simulated by VERICUT) using optimized toolpath files.
Total Time Difference— Adds a record to write out the total time difference between **Total Time** and **Total Optimized Time**.
Total Distance — Adds a record to write out the total tool movement distance.
Total Cut Distance % — Adds a record to write out the total percentage of tool movement distance that occurred while in feed rate mode.
Total Volume Removed — Adds a record to write out the total volume of material removed.

Stock Envelope — Adds a record to write out the rough dimensions of the stock.

Tool Change Data — Adds a record to write out "tool" related data, of the type selected from the pull-down list, for the current tool.

Seq — Adds a record to write out the sequential number of the last tool change event.
Record — Adds a record to write out the last tool change record processed. Includes both the sequential line number and the record text.
Tool Description — Adds a record to write out the tool ID and description of a tool retrieved from a VERICUT Tool Library file. A blank field indicates a description for this tool was not defined in the library, or the tool did not come from a tool library.
Cutter Info — Adds a record to write out cutter shape geometry data. Includes both the method used to define the cutter shape (shown in square brackets "[ ]", for example [Profile] indicates a profile type of cutter definition) followed by the parameter values used to describe the cutter.
Cutter Height — Adds a record to write out the height of the cutter.
Flute Length — Adds a record to write out the flute length of the cutter.
Gage Point — Adds a record to write out the cutter's gage point value.
OptiPath Record — Adds a record to write out the OptiPath tool description and the number of teeth for the tool
Optimized By — Adds a record to write out the optimization method used for the tool.
Original Time — Adds a record to write out the anticipated cutting time (as simulated by VERICUT) for the tool.
Optimized Time — Adds a record to write out the anticipated optimized cutting time (as simulated by VERICUT) for the tool.
Time Difference — Adds a record to write out the difference between the optimized and un-optimized cutting times for the tool
Distance — Adds a record to write out the tool movement distance.
Distance % — Adds a record to write out the percentage of tool movement distance while in feedrate mode.
Volume Removed — Adds a record to write out the volume of material removed by the tool.
Errors — Adds a record to write out the number of errors detected by VERICUT for the tool.
Warnings — Adds a record to write out the number of warnings output by VERICUT for the tool.
Min Extension — Adds a record to write out the minimum height required to avoid shank/holder collisions with the tool.
Tool ID — Adds a record to write out the Tool ID of the "current" tool.
Driven Point — Adds a record to write out the driven point of the "current" tool.
Teeth — Adds a record to write out the number of teeth (flutes) of the "current" tool.

Cutter ID — Adds a record to write out the Cutter ID of the "current" tool assembly.
Holder ID — Adds a record to write out the Holder ID of the "current" tool assembly.
Cutter Diameter — Adds a record to write out the diameter of the "current" tool.
Cutter Stick-out — Adds a record to write out the "cutter stick-out" value (distance from the tip of the tool to the lowest point on the holder) of the "current" tool.
Cutter Compensation — Adds a record to write out the cutter compensation value of the "current" tool.
Comments — Adds a record to write out the Tool Manager Comments of the "current" tool.
Air Time % — Adds a record to write out the percent of time that the "current" tool spent cutting air.
Axial Depth — Adds a record to write out the depth of cut.
Radial Width — Adds a record to write out the width of cut.
Maximum Spindle Speed — Adds a record to write out the maximum spindle speed.
Minimum Spindle Speed — Adds a record to write out the minimum spindle speed.
Maximum Feedrate — Adds a record to write out the maximum feedrate.
Minimum Feedrate — Adds a record to write out the minimum feedrate.

User-Defined Tags —

Add User-Defined Tag — Adds a record to write out the value of a "text type" User Defined Tag. Use Add User-Defined Tag, to create a new "text type" User-Defined Tag, or select an existing User-Defined Tag from the pull-down list.

Table — Displays a window enabling you to add a VERICUT generated table, or "custom" table, to a report template. Choose one of the following table types from the Table pull-down list. Each type in detail in a separate section below.

Custom Table — Displays the Custom Table window enabling you to output a table of your own design to a VERICUT report.

File Summary Table — Displays the File Summary Table window enabling you to output a VERICUT generated file summary table to a VERICUT report.

Tool Summary Table — Displays the Tool Summary Table window enabling you to output a VERICUT generated tool summary table to a VERICUT report.

View Capture Table — Displays the View Capture Table window enabling you to output a VERICUT generated table containing View Capture images specified using the features on the **File > AutoSave: View Capture tab**.

Tools Table — Displays the Tools Table window enabling you to output a VERICUT generated tool table to tool library report.

The Tools Table window is only available when the Report Template window was accessed from the Tool Manager menu bar (**File menu > Report Template**).

Inspection Features Table — Displays the Inspection Features Table window enabling you to output a VERICUT generated inspection features table to an inspection report.

The Inspection Features Table window is only available when the Report Template window was accessed from the Inspection window menu bar (**File menu > Report Template**).

or choose one of the existing (if any) "table type" User-Defined Tags at the bottom of the list.

Picture — Displays the Picture window enabling you to add VERICUT generated pictures, or custom pictures from a file, to a report template.

Page Break — Enables you to manually add page breaks to control printed output.

Start Toolpath Loop — Use to mark the "start" of a group of records that will be output for each toolpath in user files containing multiple toolpaths.

End Toolpath Loop — Use to mark the "end" of a group of records that will be output for each toolpath in project files containing multiple toolpaths.

Start Tool Change Loop — Use to mark the "start" of a group of records that will be output at each tool change.

End Tool Change Loop — Use to mark the "end" of a group of records that will be output at each tool change.

Start View Loop — Use to mark the "start" of a multiple image Setup Plan.

End View Loop — Use to mark the "end" of a multiple image Setup Plan.

NOTE: **PageBreak**, **Start Toolpath Loop**, **End Toolpath Loop**, **Start Tool Change Loop**, **End Tool Change Loop**, **Start View Loop**, and **End View Loop** cannot be edited. They can only be deleted.

The Record Table

Each record in the table consists of five columns of information. Click on any column in a row to highlight a record for editing. The order of the records in the table can be changed by clicking on the square button at the beginning of the record and dragging the record to the desired position. Each record in the table consists of the following information:

Category — Use the options in this column to specify what "part" of the report the record is to be included. With the record highlighted, click on the record "category" and select from the pull-down list (**Body**, **Page Header**, **Page Footer**, **First Page Header**, or **First Page Footer**).

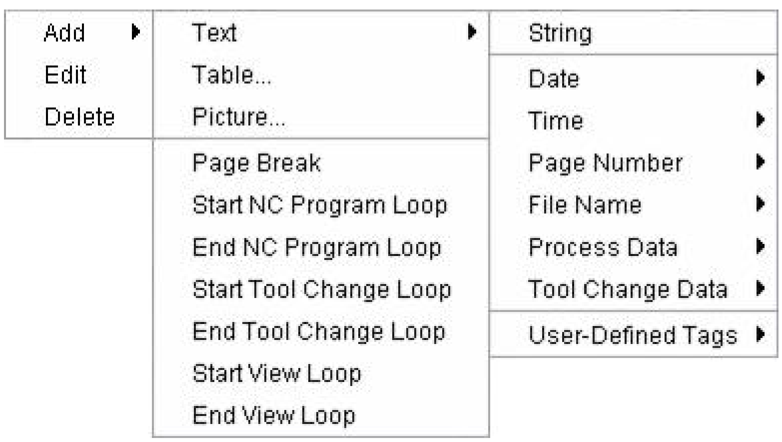
Type — Indicates the record type. With the record highlighted, double click Table or Picture to display the corresponding window for editing the record.

Content — Indicates the "content" of the record. With the record highlighted, double click on the content of a Table or Picture record to display the corresponding window for editing the record. Double click on the "content' of a text string record to add or modify the text.

Alignment — Indicates how the information will be aligned horizontally in the report. With the record highlighted, click on the record "category" and select from the pull-down list (**Body**, **Page Header**, **Page Footer**, **First Page Header**, or **First Page Footer**).

Style — Indicates the "style" to be used for the record. With the record highlighted, click on the record "style" and select from the pull-down list. The list will contain all for the styles that have been defined, using the **Styles tab**, for the current report template.

Shortcut: Right-click in the Record Table to display the following menu:



NOTE: These features provide the same functionality as those described above under Edit in the main menu. The third level features (**Date**, **Time**, **Page Number**, etc.) each have an additional pull-down list of features as described above under Edit.

To Report Template window

Custom Table window

The features in the Custom Table window enable you to output a table of your own design to a VERICUT Report.

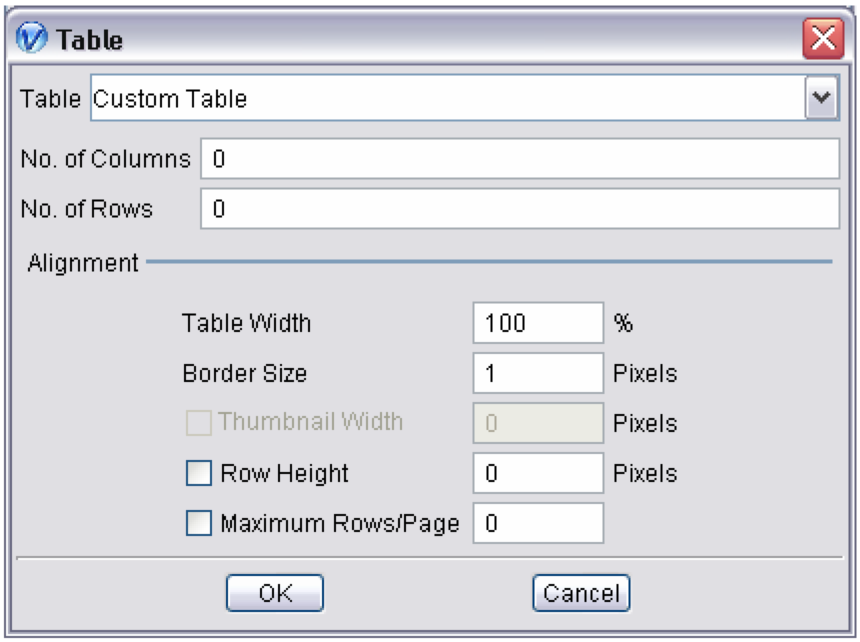


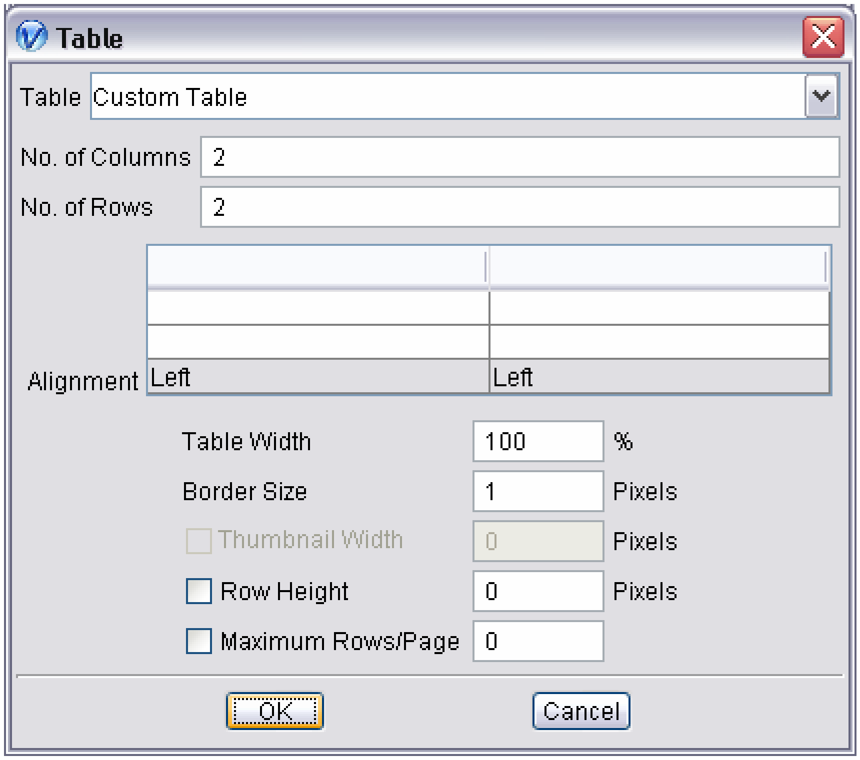
Table — Custom Table

> TIP: The Table field is editable so that you can create a user defined table tag which will enable you to populate the table with the contents of a text file. See **Using Data from a File in a Report**, in the *Using VERICUT* section, in the *CGTech Help Library* for additional information.

No. of Columns — Enter the number of columns that you want in the table.

No. of Rows — Enter the number of rows that you want in the table.

Once you specify the number of columns and rows that you want in the table, pressing **OK**, expands the window as shown below so that you can define the contents of the table.



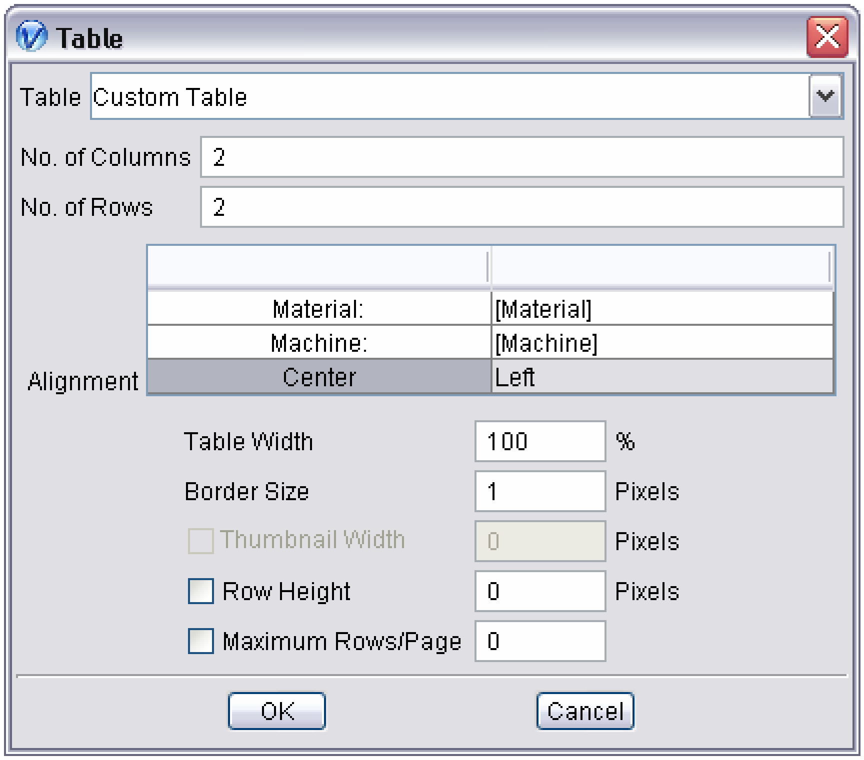
Right-clicking in any of the table cells displays a pop-up menu with the following options:

InsertRow — Use to insert an additional blank row into the table.
Insert Column — Use to insert an additional blank column into the table
Delete Row — Use to delete the row containing the highlighted cell.
Delete Column — Use to delete the column containing the highlighted cell.
Insert Text — Select the desired feature from the pull-right list. Each of the features is described above in the Report Template window, Edit menu bar, under Text.

Repeat to define the contents of each table cell.

Alignment — Left-click in each cell in the alignment row and choose **Left**, **Center**, or **Right** from the pull-down list to specify how you want the data in that column to be positioned.

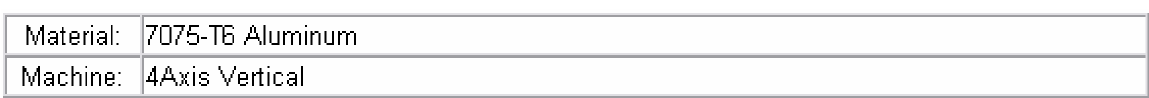
The following picture shows an example of a finished "custom table" record.



The remainder of the window features (**Table Width**, **Border Size**, **Thumbnail Width**, **Row Height**, **Maximum Rows/Page**, **OK**, and **Cancel**) are common to all table windows and are described in the Common Table Features section below.

Example Results

Using a Custom Table defined as shown in the picture above will result in a table in the VERICUT Report similar to the following picture.



Material:	7075-T6 Aluminum
Machine:	4Axis Vertical

File Summary Table window

File Summary Table — Displays the File Summary Table window enabling you to output a VERICUT generated file summary table to a VERICUT report.

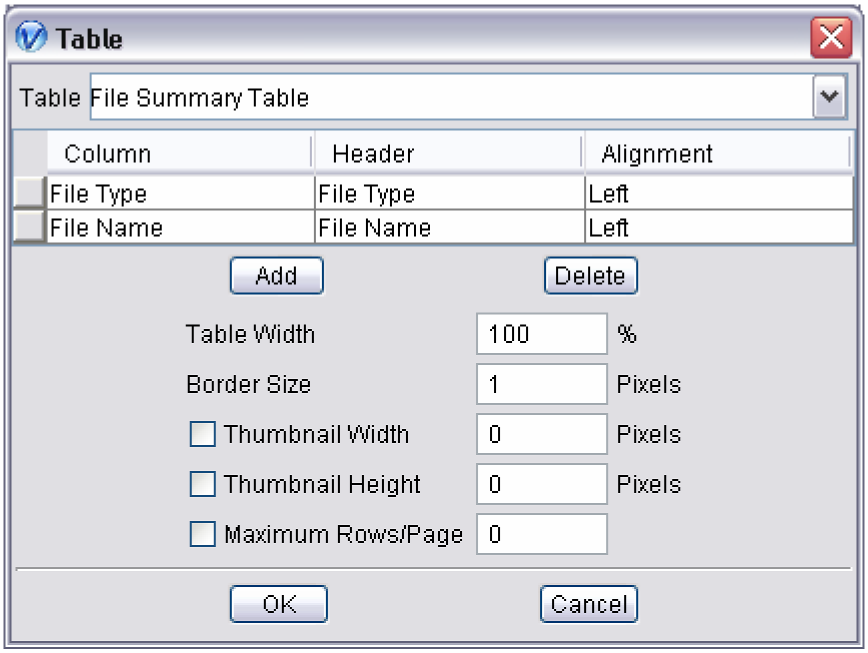


Table — File Summary Table

View Capture Data List

Each row in the File Summary Data List represents a column that will be displayed in a File Summary table in a VERICUT Report. The first row in the File Summary Data List is used to define the first column in the File Summary table. The second row in the File Summary Data List defines the second column in the File Summary table.

The order of the rows in the File Summary Data List can be changed by clicking on square button at the beginning of any row and dragging the row to the desired position in the list.

Column — The features in this column are used to specify the type of data that is to be displayed in this column of the report.

File Type — Adds the file type (Project File, Machine File, Control File, etc.) to the File summary table record.

File Name — Adds the /*path*/*filename* of the file to the File Summary table record.

Header — Use to define the column header to be used for the column in the File Summary table. Clicking on the "Header" item in a highlighted row, will put it in edit mode so you can enter a header label that is most meaningful in your work place.

Alignment — Left-click in the Alignment cell in a highlighted row and choose **Left**, **Center**, or **Right** from the pull-down list to specify how you want the data in that column of the File summary table to be positioned.

Add — Use to add a row after the highlighted row in the File Summary Data List.

Delete — Use to delete the highlighted row from the File Summary Data List.

The remainder of the window features (**Table Width**, **Border Size**, **Thumbnail Width**, **Thumbnail Height**, **Maximum Rows/Page**, **OK**, and **Cancel**) are common to all table windows and are described in the Common Table Features section below.

Example Results

Using a File Summary Table defined as shown in the picture above will result in a table in the VERICUT Report similar to the following picture.

File Type	File Name
Project File	C:\cgtech612\library\vericut.VcProject
Machine File	C:\cgtech612\library\mazak_nexus410a.mch
Control File	C:\cgtech612\library\maz640m_mazak_nexus410a.ctl
Tool Library File	C:\cgtech612\library\vericut_setup3.tls
Design Model File	C:\cgtech612\library\vericut_design.stl
NC Program	C:\cgtech612\library\vericut_setup3.mcd

Tool Summary Table window

Tool Summary Table — Displays the Tool Summary Table window enabling you to output a VERICUT generated tool summary table to a VERICUT report.

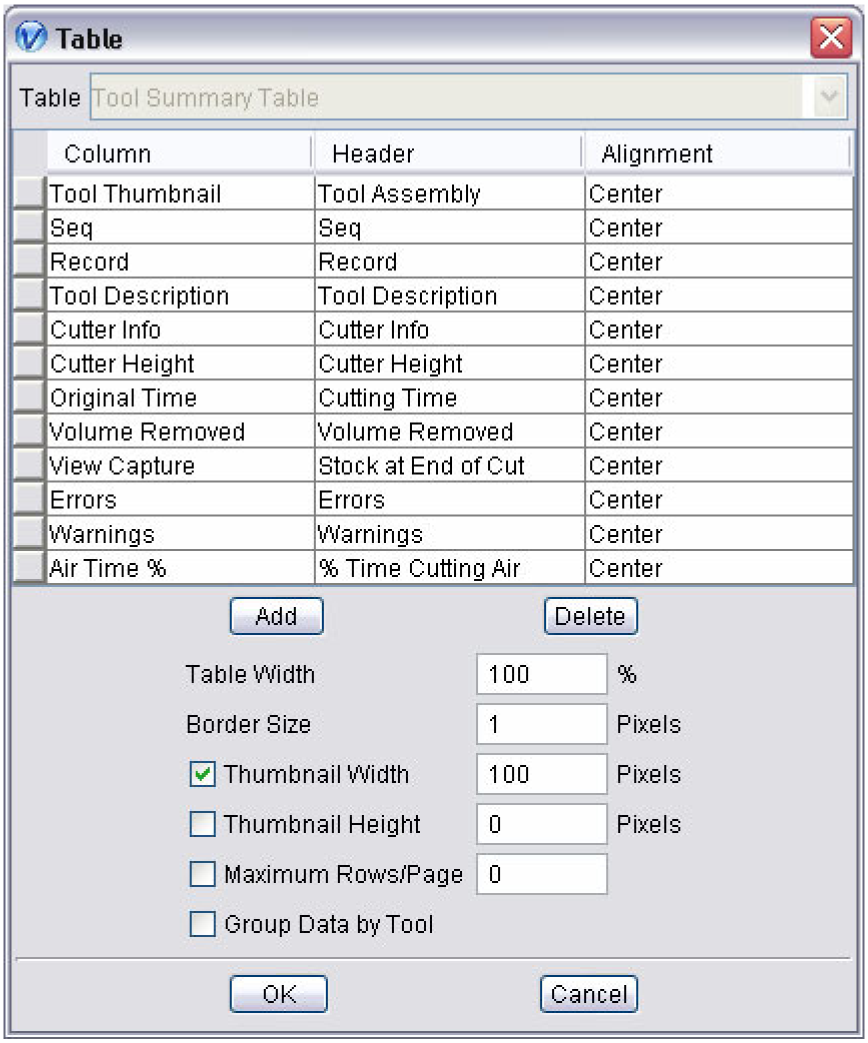


Table — Tool Summary Table

Tool Data List

Each row in the Tool Data List represents a column that will be displayed in the Tool Summary table in a VERICUT Report. The first row in the Tool Data List is used to define the first column in the Tool Summary table. The second row in the Tool Data List defines the second column in the Tool Summary table, and so on.

The order of the rows in the Tool Data List can be changed in a couple of ways. Clicking on one of the column headings (**Column**, **Header**, **Alignment**) in the Tool Data List will sort the table's rows alphabetically by the items contained in the column. Secondly, you can click on square button at the beginning of any row and drag the row to the desired position in the list.

Column — The features in this column are used to specify the type of data that is to be displayed in this column of the report. Clicking on the "Column" item in a highlighted row, will display a pull-down list of data types that you can select from. The following are the data types that are available for use in a Tool Summary table.

Thumbnail — Use to display an image of the tool, in a Tool Summary table record.

View Capture — Use to output an image at the end of each cutting sequence, in a Tool Summary table record. View Capture, in the AutoSave window, must have both **Cutter Change** and **File End : End of Each File** toggled "On".

See **AutoSave window: View Capture tab**, also in *VERICUT Help*, for additional information.

Seq — Use to add the sequence number of the tool change event, to a Tool Summary table record.

Record — Use to add the tool change record processed, to a Tool Summary table record. Includes both the sequential line number and the record text.

Tool Description — Use to add a description of a tool retrieved from a VERICUT Tool Library file, to a Tool Summary table record. A blank field indicates a description for this tool was not defined in the library, or the tool did not come from a tool library.

Cutter Info — Use to add cutter shape geometry data, to a Tool Summary table record. Includes both the method used to define the cutter shape (shown in square brackets "[ ]", for example [Profile] indicates a profile type of cutter definition) followed by the parameter values used to describe the cutter.

Cutter Height — Use to add the height of the cutter to a Tool Summary table record.

Flute Length — Use to add the flute length of the cutter to a Tool Summary table record.

Gage Point — Use to add the cutter's gage point value to a Tool Summary table record.

OptiPath Record — Use to add the OptiPath tool description for the tool to a Tool Summary table record.

Optimized By — Use to add the optimization method used for the tool to a Tool Summary table record.

Original Time — Use to add the anticipated cutting time (as calculated by VERICUT) for the tool to a Tool Summary table.

Optimized Time —Use to add the anticipated optimized cutting time (as calculated by VERICUT) for the tool to the Tool Summary table record.

Time Difference — Use to add the difference between the optimized, and un-optimized, cutting times for the tool to a Tool Summary table record.

Distance — Use to add the tool movement distance to a Tool summary table.

Distance % — Use to add the percentage of the tool movement distance that occurred while in feedrate mode to a Tool Summary table record.

Volume Removed — Use to add the volume of material removed by the tool to a Tool Summary table record.

Errors — Use to add the number of errors detected by VERICUT for the tool to a Tool Summary table record.

Warnings — Use to add the number of warnings output by VERICUT for the tool to a Tool Summary table record.

Min Extension — Use to add the minimum cutter height required to avoid shank/holder collisions with the tool to a Tool Summary table record.

Tool ID — Use to add the Tool ID of the tool to a Tool Summary table record.

Driven Point — Use to add the driven point of the tool to a Tool Summary table record.

Teeth — Use to add the number of teeth (flutes) of the tool to a Tool Summary table record..

Cutter ID — Use to add the Cutter ID(s) of the cutter/inserts used in the tool assembly to a Tool Summary record.

Holder ID — Use to add the Holder ID(s) of the holders used in the tool assembly to a Tool Summary table record.

Cutter Diameter — Use to add the diameter of the tool to a Tool Summary table record.

Cutter Stick-out — Use to add the "cutter stick-out" value (distance from the tip of the tool to the lowest point on the holder) of the tool to a Tool Summary table record.

Cutter Compensation — Use to add the cutter compensation value of the tool to a Tool Summary table record.

Comments — Use to add the Comments from the Tool Manager for the tool to a Tool Summary table record.

Air Time % — Use to add the percent of time that the tool spent cutting air to a Tool Summary table record.

Header — Use to define the column header to be used for the record in the Tool Summary table. Clicking on the "Header" item in a highlighted row, will put it in edit mode so you can enter a header label that is most meaningful in your work place.

Alignment — Left-click in the Alignment cell in a highlighted row and choose **Left**, **Center**, or **Right** from the pull-down list to specify how you want the data in that column of the Tool Summary table to be positioned.

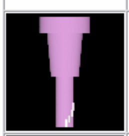
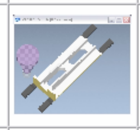
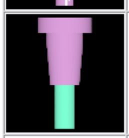
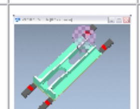
Add — Use to add a row after the highlighted row in the Tool Data List.

Delete — Use to delete the highlighted row from the Tool Data List.

The remainder of the window features (**Table Width**, **Border Size**, **Thumbnail Width**, **Thumbnail Height**, **Maximum Rows/Page**, **OK**, and **Cancel**) are common to all table windows and are described in the Common Table Features section below.

Example Results

Using a Tool Summary Table defined as shown in the picture above will result in a table in the VERICUT Report similar to the following picture.

Tool Assembly	Seq	Record	Tool Description	Cutter Info	Cutter Height	Cutting Time	Volume Removed	Stock at End of Cut	Errors	Warnings	% Time Cutting Air
	1	N140T1	1.5 X .125 Insert End Mill	GENERAL INSERT S: 504 GENERAL INSERT S: 504 GENERAL INSERT S: 504 GENERAL INSERT S: 504	0.625	5.1575	137.3494		6	1	39.1951%
	2	N4500T3	1.5 X .125 End Mill	Bull Nose: 1.5, 0.0625, 3.25	3.25	1.6733	6.6129		4	0	33.0886%
Total						6.8308	143.9623		10	1	37.6992%

View Capture Table window

View Capture Table — Displays the View Capture Table window enabling you to output a VERICUT generated table containing View Capture images, as specified on the **File > AutoSave: View Capture tab**, to a VERICUT Report.

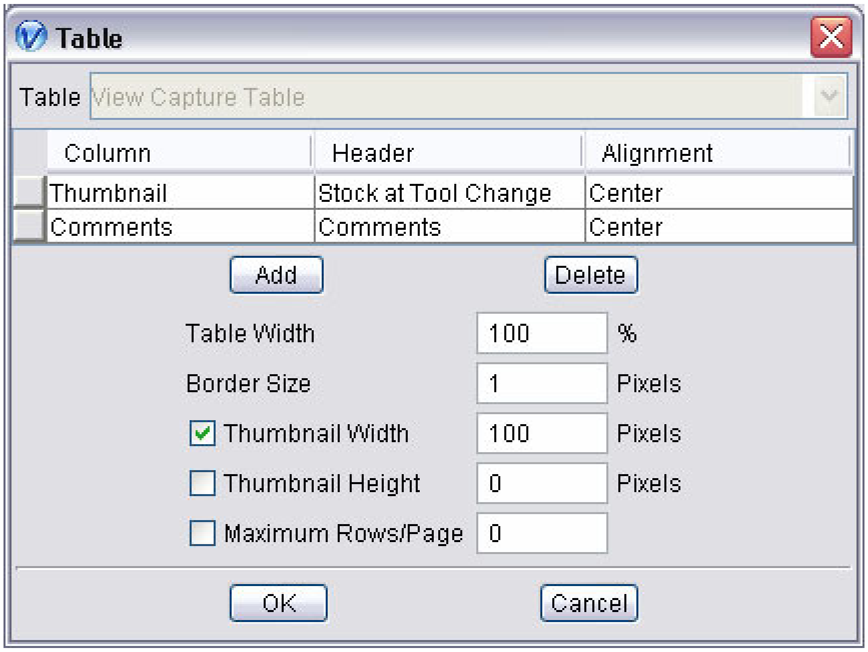


Table — View Capture Table

View Capture Data List

Each row in the View Capture Data List represents a column that will be displayed in the View Capture table in a VERICUT Report. The first row in the View Capture Data List is used to define the first column in the View Capture table. The second row in the View Capture Data List defines the second column in the View Capture table.

The order of the rows in the View Capture Data List can be changed by clicking on square button at the beginning of any row and dragging the row to the desired position in the list.

Column — The features in this column are used to specify the type of data that is to be displayed in this column of the report.

Thumbnail — Displays a View Capture image to a View Capture table record.

Comments — Displays VERICUT generated comments about the image.

Header — Use to define the column header to be used for the column in the View Capture table. Clicking on the "Header" item in a highlighted row, will put it in edit mode so you can enter a header label that is most meaningful in your work place.

Alignment — Left-click in the Alignment cell in a highlighted row and choose **Left**, **Center**, or **Right** from the pull-down list to specify how you want the data in that column of the View Capture table to be positioned.

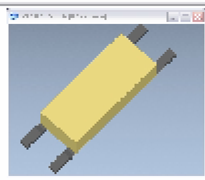
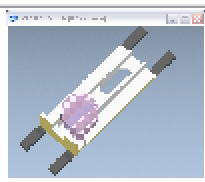
Add — Use to add a row after the highlighted row in the View Capture Data List.

Delete — Use to delete the highlighted row from the View Capture Data List.

The remainder of the window features (**Table Width**, **Border Size**, **Thumbnail Width**, **Thumbnail Height**, **Maximum Rows/Page**, **OK**, and **Cancel**) are common to all table windows and are described in the Common Table Features section below.

Example Results

Using a View Capture Table defined as shown in the picture above will result in a table in the VERICUT Report similar to the following picture.

Stock at Tool Change	Comments
	View Capture: Cutter change; Line number: 7
	View Capture: Cutter change; Line number: 262

Tools Table window

Tools Table — Displays the Tools Table window enabling you to create a tool library report. The Tools Table window is only available when the Report Template window was accessed from the Tool Manager menu bar (File menu > Report Template).

> NOTE: Report templates containing Tools Tables should only be used for creating tool library reports from the **Tool Manager window**. Using them to create reports from other VERICUT applications will result in incorrect data being displayed in the report.

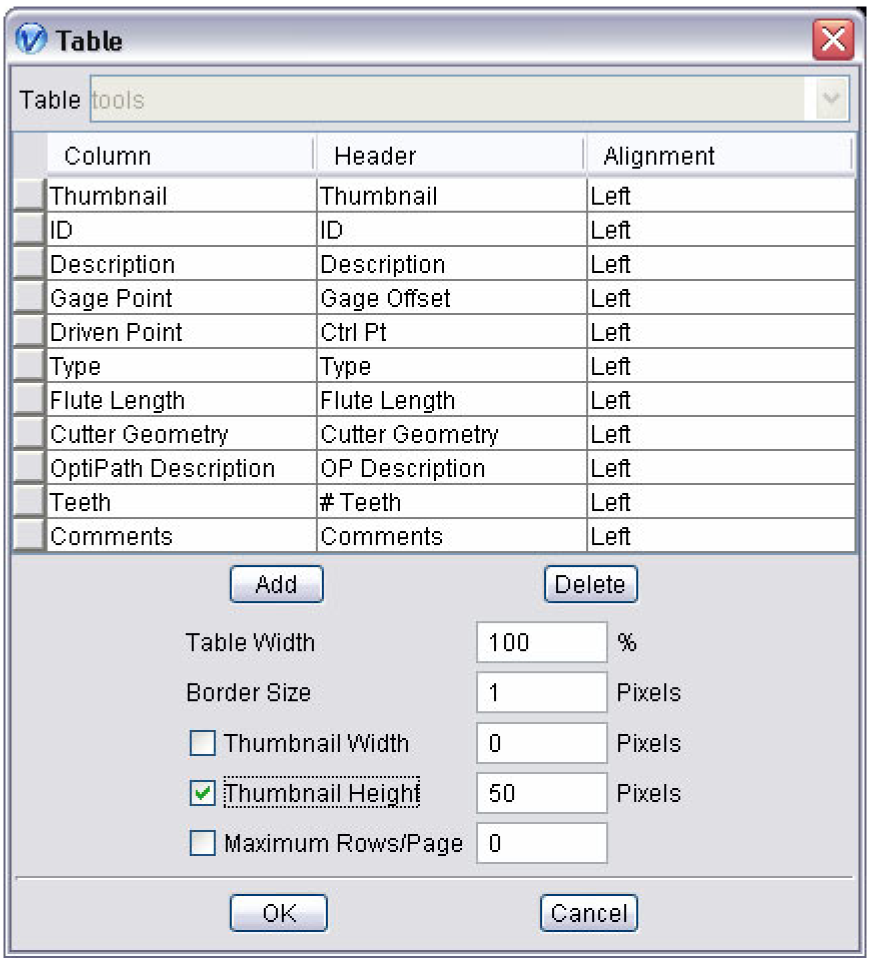


Table — tools

Tool Data List

Each row in the Tool Data List represents a column that will be displayed in the Tool Library Report. The first row in the Tool Data List is used to define the first column in the Tool Library Report. The second row in the Tool Data List defines the second column in the Tool Library Report, and so on.

The order of the rows in the Tool Data List can be changed in a couple of ways. Clicking on one of the column headings (**Column**, **Header**, **Alignment**) in the Tool Data List will sort the table's rows alphabetically by the items contained in the column. Secondly, you can click on square button at the beginning of any row and drag the row to the desired position in the list.

Column — The features in this column are used to specify the type of data that is to be displayed in this column of the report. Clicking on the "Column" item in a highlighted row, will display a pull-down list of data types that you can select from. The following are the data types that are available for use in a Tool Library Report.

Thumbnail — Use to display a thumbnail image of the tool, in a Tool Library Report.

ID — Use to display the alpha-numeric text that identifies the tool in the Tool Library, in a Tool Library Report.

Description — Use to display the description of the tool in a Tool Library Report.

Gage Point — Use to display the location of the gage point of the tool assembly, in a Tool Library Report.

Driven Point — Use to display the offset for the tool control point, or "driven point", in a Tool Library Report.

Type — Use to display the type of tool (Mill, Turn, Probe, etc.), in a Tool Library Report.

Flute Length — Use to display the tool's flute length in a Tool Library Report.

Cutter Geometry — Use to display the cutter geometry (for example, Bull Nose: 1.5, 0.0625, 3.25) in a Tool Library Report.

OptiPath Description — Use to display the description from an OptiPath record associated with the tool, in a Tool Library Report.

Teeth — Use to display the number of teeth that the tool has, in a Tool Library Report.

Comments — Use to display any comments from the tool library tool record, in a Tool Library Report.

Cutter Compensation — Use to display the cutter compensation value of the tool in a Tool Library Report.

Cutter ID — Use to display the ID(s) of the cutter(s) used in the tool assembly, in a Tool Library Report.

Holder ID — Use to display the ID(s) of the holder(s) used in the tool assembly, in a Tool Library Report.

Cutter Diameter — Use to display the diameter of the cutter in a Tool Library Report.

Cutter Height — Use to display the height of the cutter in a Tool Library Report.

Cutter Stick-out — Use to display the "cutter stick-out" value (distance from the tip of the tool to the lowest point on the holder) of the tool, in a Tool Library Report.

See Tool Manager window, also in *VERICUT Help*, for additional information on these data types.

Header — Use to define the column header to be used for the record in the Tool Library Report. Clicking on the "Header" item in a highlighted row, will put it in edit mode so you can enter a header label that is most meaningful in your work place.

Alignment — Left-click in the Alignment cell in a highlighted row and choose **Left**, **Center**, or **Right** from the pull-down list to specify how you want the data in that column of the Tool Library Report to be positioned.

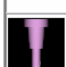
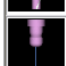
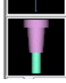
Add — Use to add a row after the highlighted row in the Tool Data List.

Delete — Use to delete the highlighted row from the Tool Data List.

The remainder of the window features (**Table Width**, **Border Size**, **Thumbnail Width**, **Thumbnail Height**, **Maximum Rows/Page**, **OK**, and **Cancel**) are common to all table windows and are described in the Common Table Features section below.

Example Results

Using a Tools Table defined as shown in the picture above will result in a table in the Tool Library Report similar to the following picture.

Thumbnail	ID	Description	Gage Offset	Ctrl Pt	Type	Flute Length	Cutter Geometry	OP Description	# Teeth	Comments
	1	1.5 X .125 Insert End Mill	0 0 8.6575	0 0 0	Mill		GENERAL INSERT S: 504 GENERAL INSERT S: 504 GENERAL INSERT S: 504 GENERAL INSERT S: 504	1.5 X 1/16 Insert End Mill	2	
	2	Probe	0 0 6	0 0 0	Probe		Probe: 0.1875, 0.12, 3		2	
	3	1.5 X.125 End Mill	0 0 8.25	0 0 0	Mill	2.6	Bull Nose: 1.5, 0.0625, 3.25	1.5X.125 End Mill	4	

Inspection Features Table window

Inspection Features Table — Displays the Inspection Features Table window enabling you to create an inspection report. The Inspection Features Table window is only available when the Report Template window was accessed from the Inspection window menu bar (File menu > Report Template).

> NOTE: Report templates containing Inspection Feature Tables should only be used for creating inspection reports from the **Inspection window**. Using them to create reports from other VERICUT applications will result in incorrect data being displayed in the report.

Table — Inspection Features

Inspection Feature List

Each row in the Inspection Features List represents a column that will be displayed in the Inspection Report. The first row in the Inspection Features Table is used to define the first column in the Inspection Report. The second row in the Inspection Feature List defines the second column in the Inspection Report, and so on.

The order of the Inspection Feature List's rows can be changed in a couple of ways. Clicking on one of the column headings (**Column**, **Header**, **Alignment**) in the Inspection Feature List will sort the table's rows alphabetically by the items contained in the column. Secondly, you can click on square button at the beginning of any row and drag the row to the desired position in the list.

Column — The features in this column are used to specify the type of data that is to be displayed in this column of the report. Clicking on the "Column" item in a highlighted row, will display a pull-down list of data types that you can select from. The following are the data types that are available for use in an Inspection Report.

Symbol — Use to display the symbol of the type of measurement that the record represents, for example  for a floor thickness, in an Inspection Report.

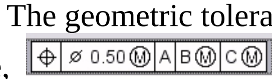
Feature — Use to display a very brief description of the type of feature the record represents, for example "Floor Thickness", in an Inspection Report.

Identifier — Use to display the feature identifier associated with the inspection record, for example, A1, in an Inspection Report.

Instrument — Use to display the type of instrument that is to be used to check the dimension, for example, "Ultrasonic", in an Inspection Report.

Dimension — Use to display the expected dimension in an Inspection Report.

Tolerance — Use to display the acceptable tolerance value, or range of values, in an Inspection Report.

Geo. Tolerance — The geometric tolerance specification associated with the record, for example, 

, in an Inspection Report.

Measurement — Use to display the inspector's actual measurement in an Inspection Report.

Tool ID — Use to display the identifier of the tool that cut the feature, in an Inspection Report.

See **VERICUT Inspection**, in the Analysis menu section of *VERICUT Help* for additional information on these data types.

Header — Use to define the column header to be used for the record in the Inspection Report. Clicking on the "Header" item in a highlighted row, will put it in edit mode so you can enter a header label that is most meaningful in your work place.

Alignment — Left-click in the Alignment cell in a highlighted row and choose **Left**, **Center**, or **Right** from the pull-down list to specify how you want the data in that column of the Inspection Report to be positioned.

Add — Use to add a row after the highlighted row in the Inspection Feature List.

Delete — Use to delete the highlighted row from the Inspection Feature List.

The remainder of the window features (**Table Width**, **Border Size**, **Thumbnail Width**, **Thumbnail Height**, **Maximum Rows/Page**, **OK**, and **Cancel**) are common to all table windows and are described in the Common Table Features section below.

Example Results

Using an Inspection Features Table defined as shown in the picture above will result in a table in the Inspection Report similar to the following picture.

Symbol	Feature	Identifier	Instrument	Dimension	Tolerance	Geo.Tolerance	Measurement	Tool
	Floor Thickness	A1	Ultrasonic	10.0000	±0.20			
	Hole Diameter	A2	Plug Gage	50.0000	±0.30			
	Datum Plane	B						
	Distance	B1	Caliper					

Common Table Features

The following features are common to all table types except where noted below.

Table Width — Use to specify the table width relative to the window/page width. Enter the Table width as a percentage of the window/page width.

Border Size — Use to specify a border size for a "custom" table. Enter the border size in pixels.

Thumbnail Width — Use to specify the width of "images" included in tables. Toggle "On" and enter the width value in pixels.

Thumbnail Height — Use to specify the width of "images" included in tables. Toggle "On" and enter the width value in pixels.

NOTES: The following rules apply to **Thumbnail Width/Thumbnail Height**:

1. If only one of the Thumbnail dimensions is toggled "On", the active dimension will be used and the other direction dimension will be scaled to maintain the image aspect ratio.
2. If neither of the Thumbnail dimensions is toggled "On", the native image size will be used.

Row Height —Use to specify the row height for a "custom" table. Toggle "On" and enter the height value in pixels. This feature is only available in the Custom Table window.

Maximum Rows/Page — Use to specify the maximum number of rows per page when printing a "custom" table.

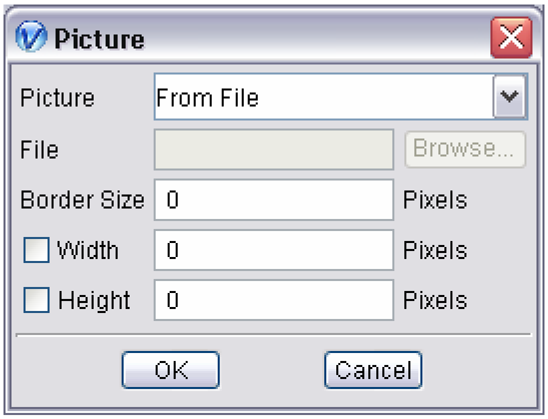
Group Data by Tool — When toggled "On", the Tool Summary Table will group sequences using the same tool. When toggled "Off", the Tool Summary Table will list tools sequence by sequence. This feature is only available in the Tool Summary Table window.

OK — Creates a table record, as specified, in the report template and closes the Table window.

Cancel — Closes the Table window without creating a table record in the report template.

Picture window

Picture — Displays a window enabling you to add VERICUT generated pictures, or custom pictures from a file, to a report template.



Picture — Use to specify the type of picture to add. Select one of the picture types from the pull-down list.

Add a custom image using:

From File — outputs the image from the specified file.

or choose one of the VERICUT generated pictures:

Tool Use Graph — outputs a **Tool Use graph** image.
Cutting Graph — outputs a **Cutting Conditions graph** image.
Start of Process Image — outputs an image at the start of processing. When included within a "tool change loop", an image of the part at the start of cutting with current tool is output. When included within a "toolpath loop", an image of the part at the start of the current toolpath is output.
End of Process Image — outputs an image at the end of processing. When included within a "tool change loop", an image of the part at the end of cutting with current tool is output. When included within a "toolpath loop", an image of the part at the end of the current toolpath is output.

Tool Image — outputs an image of the current tool. When included within a "tool change loop", an image of the current tool is output at each tool change.

Setup Plan — outputs a setup plan image. Multiple image setup plans can be output by using a **Start View Loop** record before the **Setup Plan** record and an **End View Loop** record after.

or choose one of the existing (if any) "picture type" User-Defined Tags at the bottom of the list.

File — Enter the */path/filename* of the image file, or use **Browse** to display a file selection window and use it to pick the image file. This feature is only active when Picture is set to **From File**.

Border Size — Use to add a border to the image. Enter the border size in pixels.

Width — Use to specify the width for the displayed image. Toggle "On" and enter the width value in pixels.

Height — Use to specify the height for the displayed image. Toggle "On" and enter the height value in pixels.

NOTES: The following rules apply to **Width/Height**:

1. If only one of the dimensions is toggled "On", the active dimension will be used and the other direction dimension will be scaled to maintain the image aspect ratio.
2. If neither of the dimensions is toggled "On", the native image size will be used.

OK — Adds the picture record, as specified, to the template and closes the Picture window.

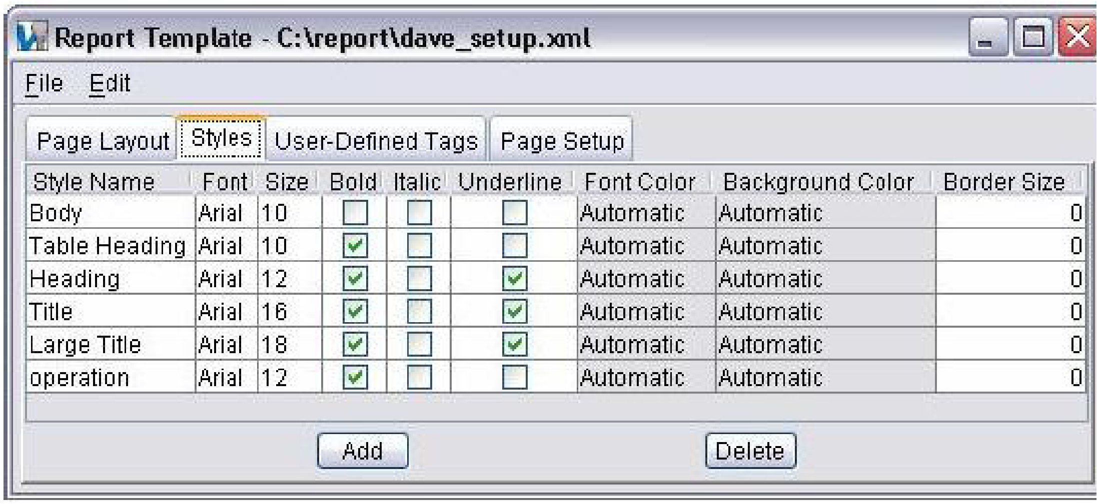
Cancel — Closes the Picture window without adding a picture record to the template.

Report Template window, Styles tab

Location: **Project menu > Report > Report Template >Edit** in the main VERICUT window.

File > Report Template in the Inspection Sequence window.

File > Report Template in the Tool Manager window.



The Menu Bar

File

See **Report Template window** for a description of the File features.

Edit

Add — Enables you to add "style" records to a report template file.

Delete — Enables you to delete "style" records from a report template file.

NOTE: **Add** and **Delete** can also accessed using the **Add** or **Delete** button at the bottom of the **Styles tab** or by right-clicking in the **Style Record table** and selecting **Add** or **Delete**.

The Style Record Table

Each record in the table consists of nine columns of information describing a particular style record. Click on any column in a row to highlight a record for editing. Each record in the table consists of the following information:

Style Name — The name used to identify a particular style record, These are the items that will appear in the **Style** pull-down list, in the record table, on the **Page Layout tab**.

Font — Indicates the font type to be used for the style record. With the record highlighted, click on the Font field and select from the pull-down list.

Size — Indicates the font size to be used for the style record. With the record highlighted, click on the **Size** field and either select from the pull-down list or type in a font size.

Bold — Toggle the "Bold" characteristic "On/Off" for the style record.

Italic — Toggle the "Italic" characteristic "On/Off" for the style record.

Underline — Toggle the "Underline" characteristic "On/Off" for the style record.

Font Color — With the record highlighted, double click on the Font Color field to display a color pallet. (You can also right-click on a "color" field and select **Set Color** to display the color pallet.) Click on the desired font color in the pallet. Click on the **X** in the upper right corner of the color pallet to dismiss the pallet.

Background Color — With the record highlighted, double click on the Background Color field to display a color pallet. (You can also right-click on a "color" field and select **Set Color** to display the color pallet.) Click on the desired background color in the pallet. Click on the **X** in the upper right corner of the color pallet to dismiss the pallet.

Border Size — Indicates the border size to be used for the style record. With the record highlighted, click on the border size and type in a font size (in pixels).

Shortcut: Right-click in the Style Record Table to display the following menu:

NOTES:

1. Feature **Set Color** provides the same functionality as described above for Font Color.

2. Features Add and Delete provide the same functionality as described above under Edit in the menu bar.

To Report Template window

Report Template window, User-Defined Tags tab

Location: **Project menu** > **Report** > **Report Template** >**Edit** in the main VERICUT window.

File > **Report Template** in the Inspection Sequence window.

File > **Report Template** in the Tool Manager window.

User-Defined Tags enable you to define "generic" records in a report template, whose value is specified at the time that the template is used to create a report. User-Defined Tag values can be defined in the **User-Defined Tags window** or by using PPRINT data in the toolpath file.

The Menu Bar

File

See **Report Template window** for a description of the **File** features.

Edit

Add — Enables you to add "User-Defined Tag" records to a report template file.

Delete — Enables you to delete "User-Defined" records from a report template file.

NOTE: Add and Delete can also accessed using the Add or Delete button at the bottom of the User-Defined Tags tab or by right clicking in the User-Defined Tags Record table and selecting Add or Delete.

The User-Defined Tag Record Table

Each record in the table consists of a Tag Name that is used to identify the record and a Type (Text, Table or Picture from File). Click on any column in a row to highlight a record for editing. Each record in the table consists of the following information:

Tag Name — The name used to identify a particular User-Defined Tag record. After adding a User-Defined Tag record, type in a unique tag name.

Type — Indicates the "type" of information represented by the tag. After adding a and naming a User-Defined Tag record, click in the Type field and select the appropriate data type (**Text**, **Table**, or **Picture From File**) from the pull-down list.

NOTE: Tag Name and **Type** can be edited by highlighting the record, then clicking the appropriate field and either editing the text or selecting the desired Type from the pull-down list. Once a Tag Name is referenced in a report template, the User-Defined Tag record can no longer be edited.

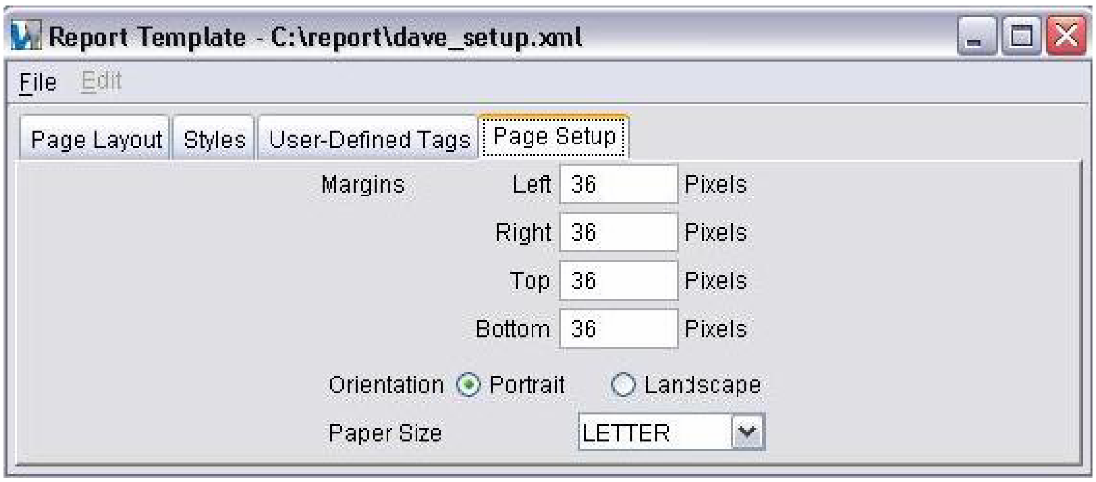
Shortcut: Right-click in the User-Defined Tag Record Table to display the following menu:

NOTE: These features provide the same functionality as those described above under Edit in the menu bar.

To Report Template window

Report Template window, Page Setup tab

Features on this tab enable you to specify or modify page format characteristics such as margins, orientation and paper size. The features on this tab are intended to be use with PDF format reports, although it may also be useful for HTML format reports with certain web browsers.



To Report Template window

User-Defined Tag Values window

Location: **Project menu > Report > User-Defined Tag Values**

File > User-Defined Tag Values in the Inspection Sequence window.

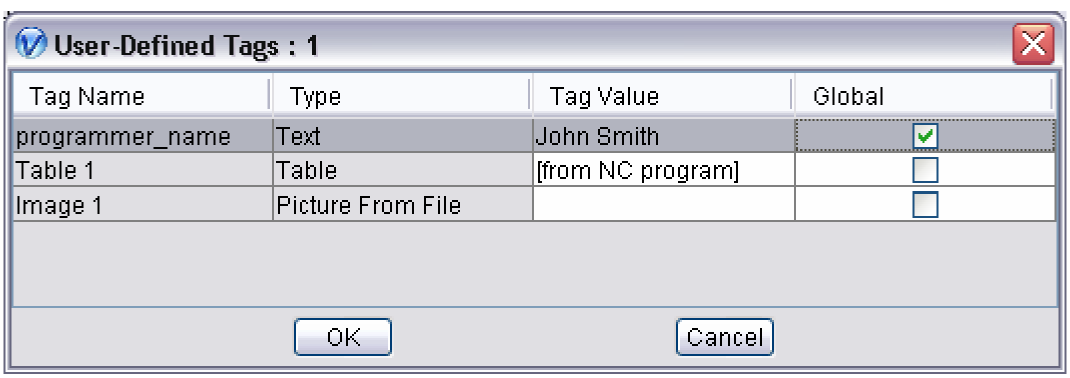
File > User-Defined Tag Values in the Tool Manager window.

Opens the **User-Defined Tags window** enabling you to assign job specific information to "generic" User-Defined Tags (ref. Report Template window: User Defined Tag tab, also in the VERICUT Help section), prior to creating a report. Double click on a **Tag Value** field to enter, or edit, the tag value.

Once User Defined Tag values have been specified, they are stored in the project file. Use the check boxes in the Global column to specify whether a User Defined Tag value applies to the project or to a particular setup. When toggled On (checkmark visible), the User Defined Tag value is saved at the project level. When toggled Off, the User Defined Tag value is saved with the setup.

If User Defined Tag values are not specified prior to creating a report, you will be prompted for them at the time the report is created.

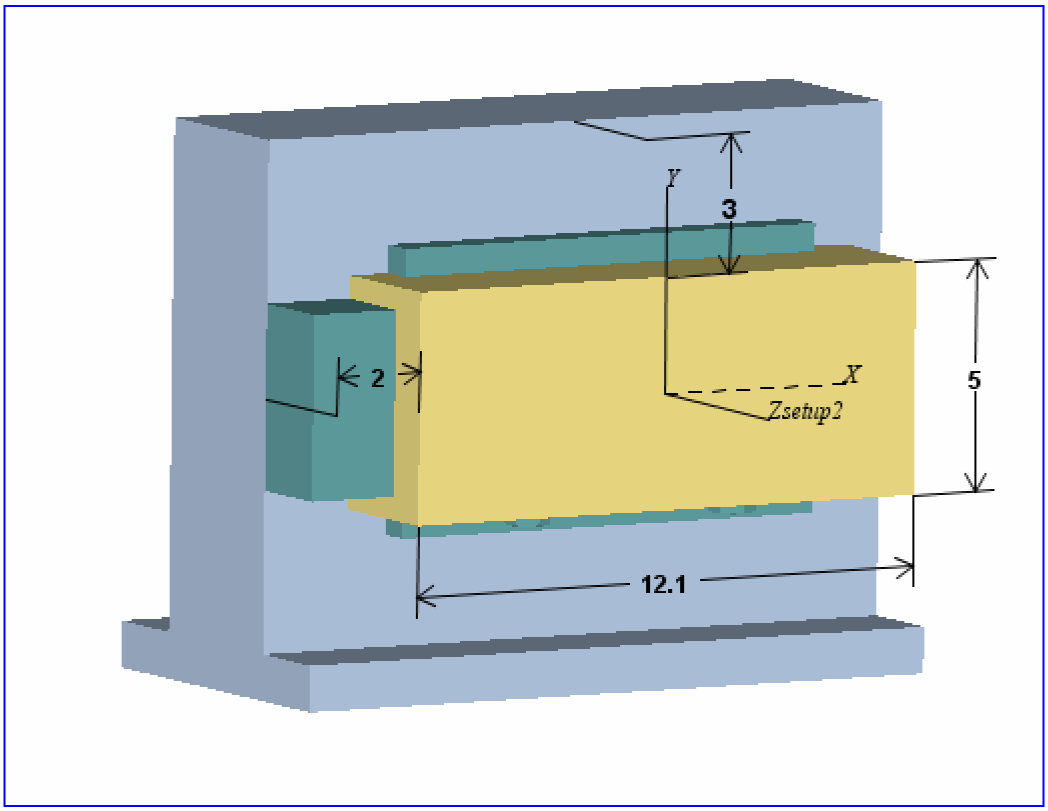
You can also specify User-Defined Tag Values using PPRINT statements in the toolpath file.



Setup Plan

Introduction to Setup Plan

Setup Plan enables you to create a setup plan image, including notes and dimensions, which can be included in a VERICUT report.



Important Concepts

Setup Plan View — The Setup Plan View defines the spatial relationship between the model and paper and affects how the image appears on the paper, i.e. the orientation and position of the model in the jpeg image. It is represented on screen by the spatial relationship between the models and the paper-edge rectangle (also referred to as the "plan rectangle").

Dimension Plane — The dimension plane (also referred to as "the glass") is the plane that all dimensions are projected into. This affects how dimensions display on the screen, how dimension values are calculated, and defines the meaning of "horizontal" (X-axis of the Csys) and "vertical" (Y-axis of the Csys), etc.

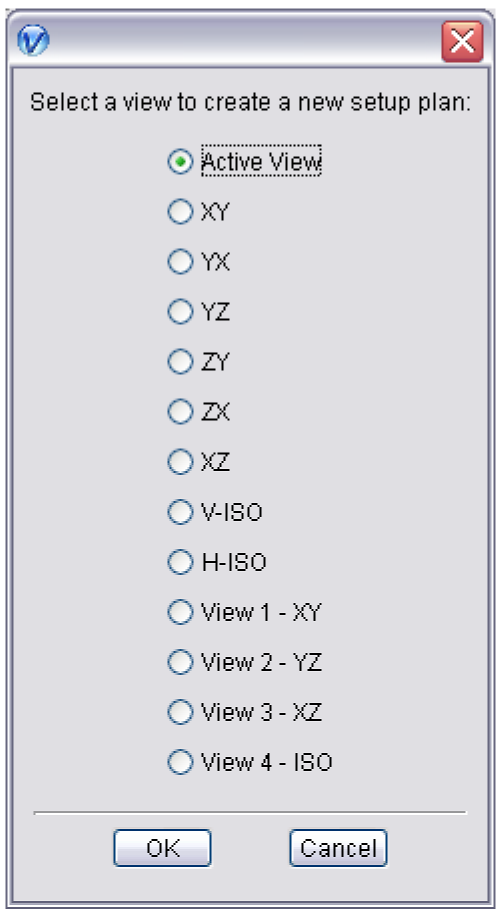
Viewport Image — The viewport image is the orientation of the image in the VERICUT viewport and is manipulated using VERICUT's view controls. The viewport image does not affect the Setup Plan View or the dimension plane.

Select View window

Location: **Project menu > Report > Setup Plan**

The **Select View** window enables you to specify the initial view that you want to create the setup plan in. The view choices in the list will include: **Active View** (The view that is currently active in the VERICUT session. This is the default.); Standard Views **(XY**, **YX**, **YZ**, **ZY**, **ZX**, **XZ**, **V-ISO** and **H-ISO**); and any custom views previously defined in the **Select View** window, or in the **Select/Store View** window, for the current setup. See **Select/Store View window**, also in the *VERICUT Help* section, for additional information.

NOTE: Selecting one of the standard views **(XY**, **YX**, **YZ**, **ZY**, **ZX**, **XZ**, **V-ISO** or **H-ISO**) specifies only the view orientation. The view type (Machine or Workpiece) is determined by the view that is currently active in the VERICUT session.

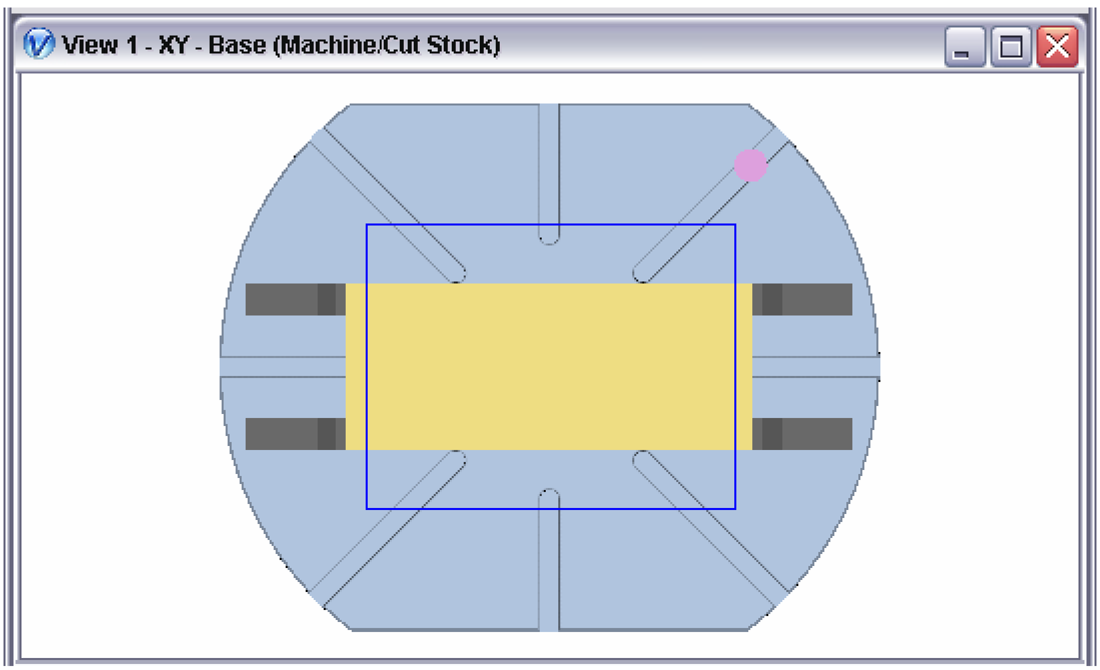


OK — After selecting the initial view for the setup plan, click **OK**. The following will happen:

1. View 1 in VERICUT will change to the selected view.
2. The "plan rectangle", representing the default plan size is displayed on the glass (a plane, referred to as "the glass", parallel to the view that setup plan record is associated with) in the specified view. See the picture below. The "plan rectangle" is drawn on the glass in the **Clip Box** color specified on the Settings tab.
3. Unless you selected an existing custom view previously defined in the Select/Store View window, a new "Setup Plan" view will be created and will be added to the Select/Store View window. If you selected a previously defined view, the original view name will be used in the Setup Plan record.

   New views will be added to both the Select/Store View window and the Select View window, even if a setup plan record is not actually created for the initial view selected (selecting **Cancel** in the Setup Plan window).
4. The Setup Plan window displays, enabling you to continue the process of creating a setup plan.

Setup Plan view with "plan rectangle"



Cancel — Use **Cancel** to close the Select View window without any further action to create a setup plan.

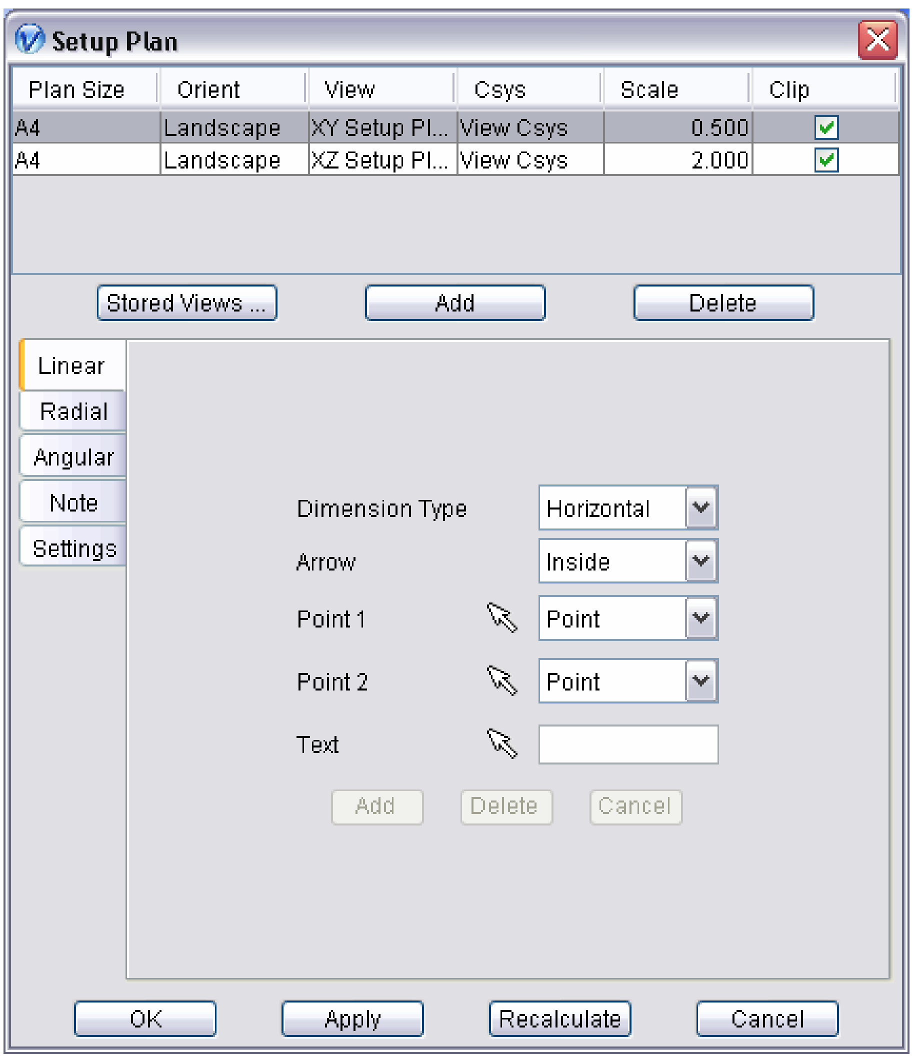
NOTE: The Select View window only displays when you select **Project menu > Report > Setup Plan** when no setup plan records exist for the current setup. If one, or more, setup plan records already exist the Setup Plan window opens immediately.

Setup Plan window

Location: **Project menu > Report > Setup Plan**

VERICUT toolbar short cut: 

The features in the Setup Plan window enable you to add new dimensions or notes to, or edit existing dimensions or notes in, a setup plan. The dimensions are placed on a dimension plane (referred to as "the glass") represented by the XY plane of the specified Csys.
Once the Setup Plan is created it can be included in a VERICUT report. See the Adding a Setup Plan Image to a Report topic below for information about adding Setup Plan information to VERICUT Reports.



Setup Plan Table

Each record in the Setup Plan Table represents a setup plan image that can be added to a VERICUT report. Each record consists of the following columns of data:

Plan Size — Sets the size of the "plan rectangle" drawn on the glass. Choose (**A**, **B**, **C**, **A1**, **A2**, or **A3**) from the pull-down list. (Default = A for inch, A1 for mm)

Orient — Sets the orientation of the "plan rectangle" drawn on the glass. Choose (**Portrait**, or **Landscape**) from the pull-down list. (Default = Landscape)

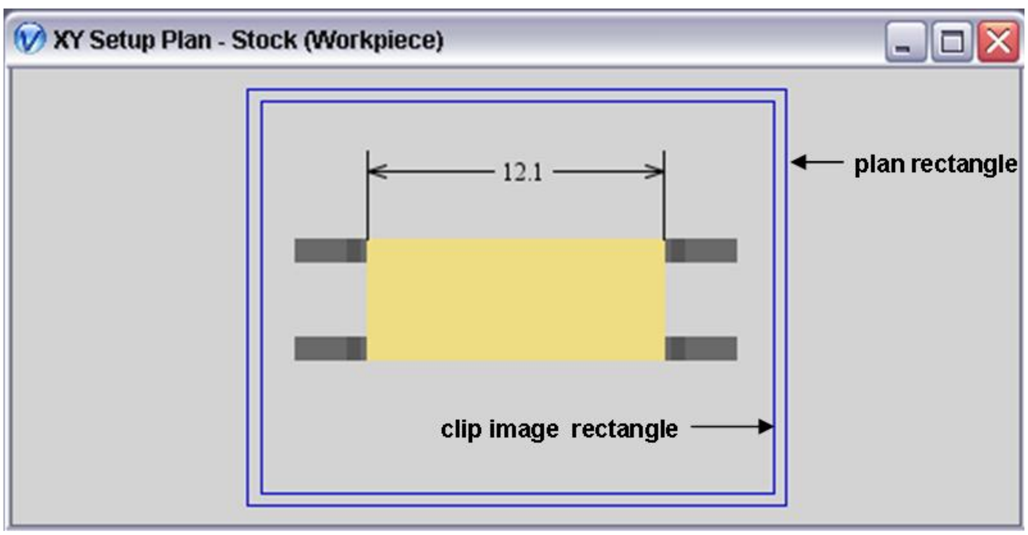
View — Defines the orientation of the setup plan image in VERICUT's 3D space. Select the desired view from the pull-down list. The list contains all views defined in the **View menu** > **Select/Store View** window for the current setup.

Csys — Used to define the dimension plane, represented by the XY plane of the Csys, and the meaning of "horizontal" (X-axis of the Csys) and "vertical" (Y-axis of the Csys). The default is View Csys, the coordinate system that defines the view for the current setup plan record.

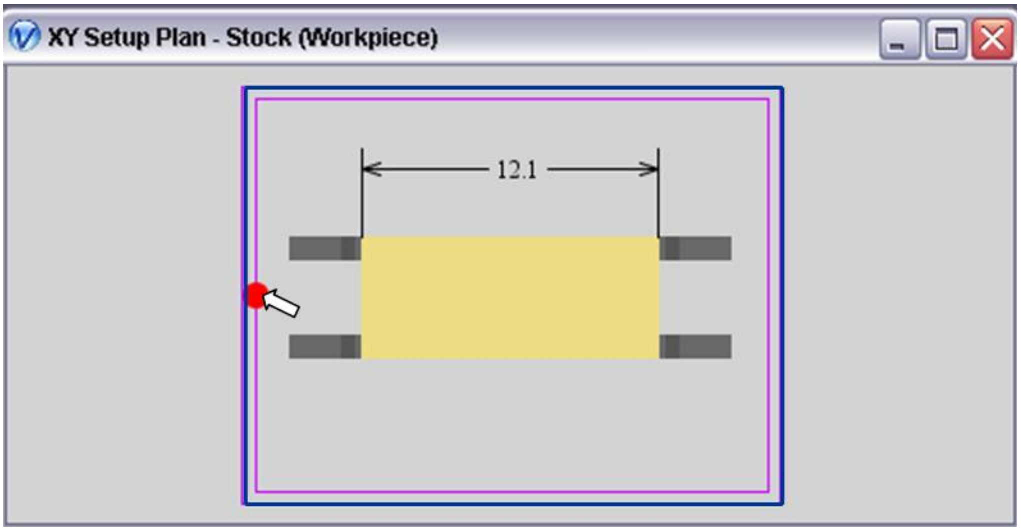
Scale — Sets the size of the "plan rectangle" relative to VERICUT's 3D space. Enter the desired scaling factor in the number field. (Default = 1)

Clip — Use to activate the "clip image rectangle" that will be used to determine what will be displayed in the setup plan image in the setup plan report.

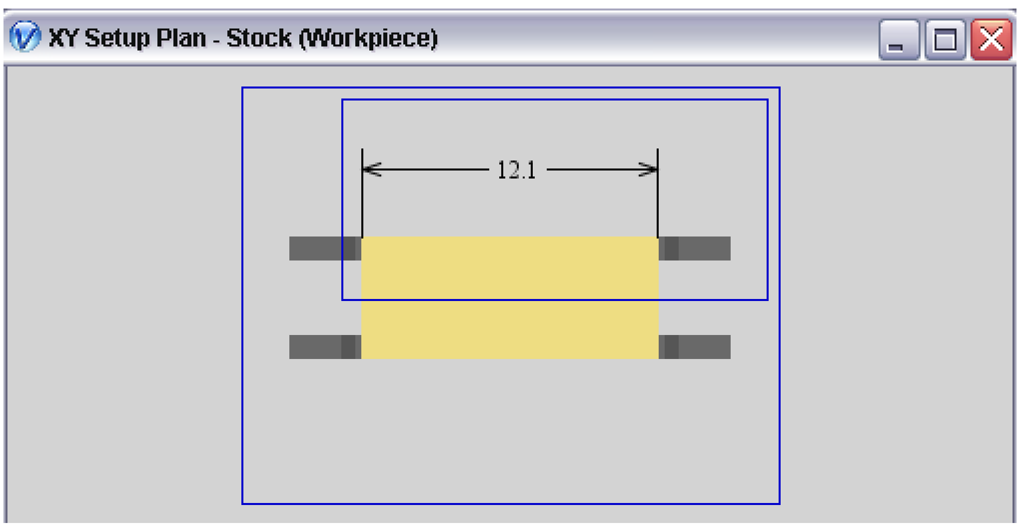
When toggled "On", (checkmark displayed) a re-sizable "clip image rectangle" is displayed on the on the glass is drawn on the glass in the **Clip Box** color specified on the Settings tab. The image that appears in the setup plan report is clipped (contained) by this rectangle.



To change the size and location of the "clip image rectangle, hold the cursor over any of the four corners, or over the midpoint of any of the rectangle's sides so that a red dot displays as shown in the picture below. Also notice that the rectangle changes to the **Selection** color defined on the Settings tab.

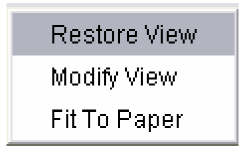


Left click on the dot and drag it to modify the "clip image rectangle". Repeat the "click and drag" using the other available red dot control points until the desired "clip image rectangle" size, shape, and location is achieved.



Setup Plan Table Right Mouse Button Menu

Right-click in the Setup Plan Table to display the following menu:

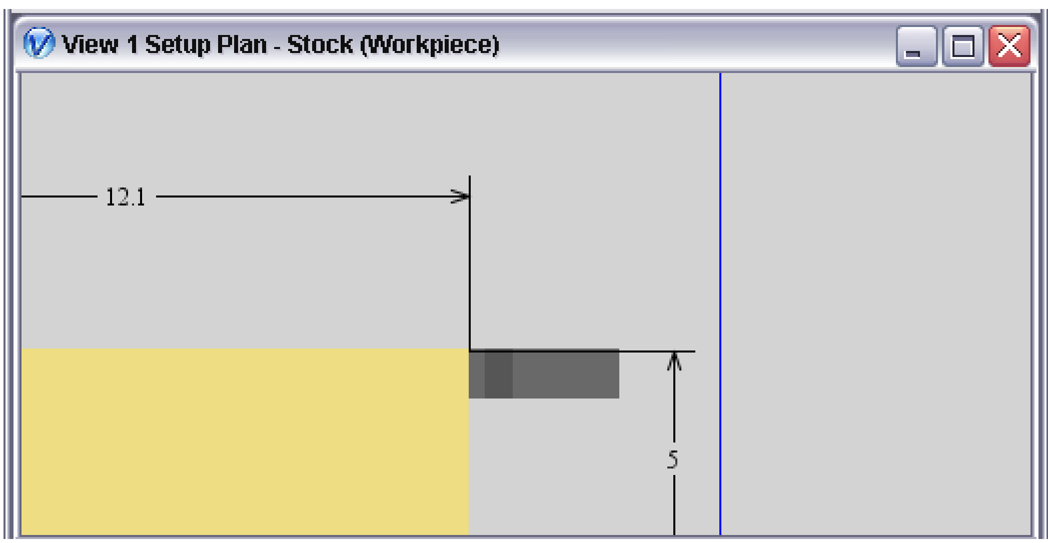


Restore View — This feature enables you to easily return a rotated view in the VERICUT graphics area to the view orientation specified in the highlighted setup plan record.

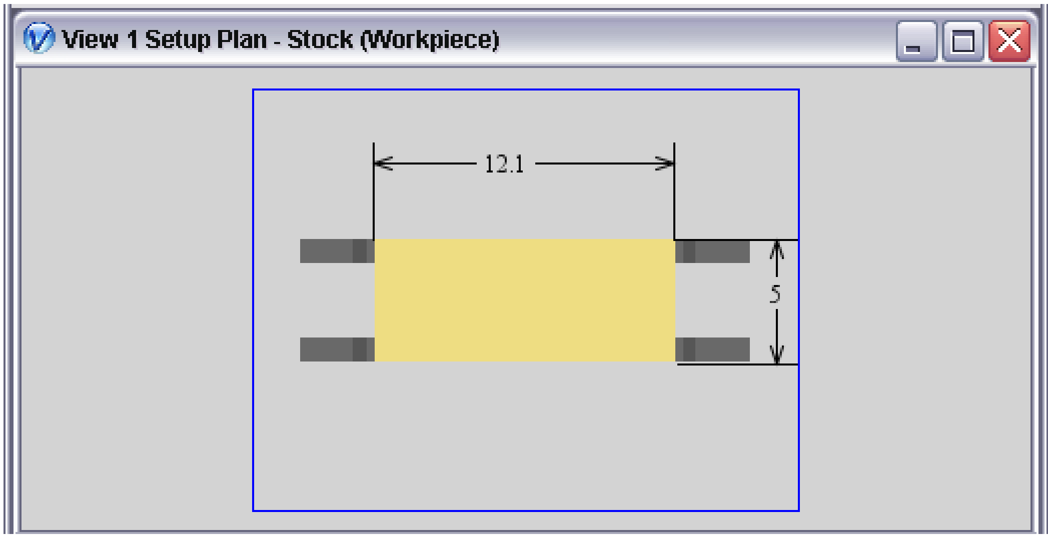
Modify View — Modifies the View Type and orientation of the view in the highlighted record in the Setup Plan Table to match those of View 1 in the VERICUT graphics window. This feature provides the same functionality as **Modify** in the **Select/Store View** window.

Fit To Paper — Use to "fit" the VERICUT view to the size of the setup plan paper. See the pictures below.

VERICUT view before using **Fit To Paper**



VERICUT view after using **Fit To Paper**



When a record is selected in Setup Plan Table:

1. The selected record becomes highlighted.
2. The View 1 display in VERICUT switches to the view associated with the selected record.
3. A rectangle representing the plan size, and its scale in the view, is drawn on the glass in the **Clip Box** color specified on the Settings tab.
4. If the **Clip** checkbox in the in the setup plan record is toggled "On" (checked) a re-sizable "clip image rectangle" is drawn on the glass in the **Clip Box** color specified on the Settings tab.
5. Dimensions/notes associated with the record are drawn on the glass.

Stored Views — Opens the Select /Store View window (ref. **Select/Store View window**, also in the *VERICUT Help* section).

Add — Adds a setup plan record to the Setup Plan Table.

Delete — Deletes the highlighted setup plan record from the table.

Linear tab **—** Use the features on this tab to add, or modify, linear dimensions in the highlighted setup plan record.

Radial tab **—** Use the features on this tab to put radial dimensions on the highlighted setup plan record.

Angular tab **—** Use the features on this tab to put angular dimensions on the highlighted setup plan record.

Note tab **—** Use the features on this tab to put notes on the highlighted plan record.

Settings tab **—** Use the features on this tab to define characteristics to be used for notes and dimensions (number format, font, font size, etc.).

OK — Saves all setup plan records and closes the Setup Plan window.

Apply — Saves all setup plan records and leaves the Setup Plan window open for additional work.

Recalculate — Selecting **Recalculate** causes VERICUT to re-calculate ***all*** dimension values, and apply the "current" settings on the Settings tab to all dimensions and notes in the highlighted setup plan record.

> NOTE: Any dimension text values that were manually entered, for example "not to scale" dimensions will be overwritten by VERICUT calculated values when **Recalculate** is used.

Cancel — Closes the Setup Plan window without saving setup plan records.

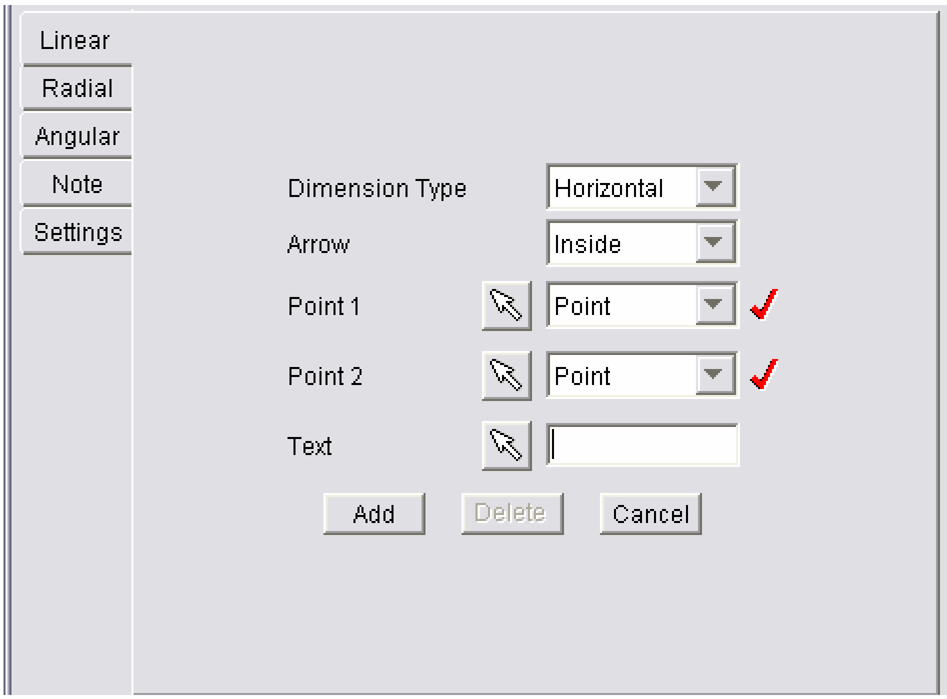
NOTES:

1. Setup Plan data is saved per setup in the project file.
2. Image (not glass) is clipped in HTML/PDF output based on the status (checked/not checked) of the **Clip** checkbox in the setup plan record, and "clip image rectangle" size and position.
3. See the Adding a Setup Plan Image to a Report topic below for information about adding Setup Plan information to VERICUT Reports.

Setup Plan window, Linear tab

Location: **Project menu > Report > Setup Plan**

The features on the Linear tab enable you to add linear dimensions to, or modify existing linear dimensions in, the setup plan record.



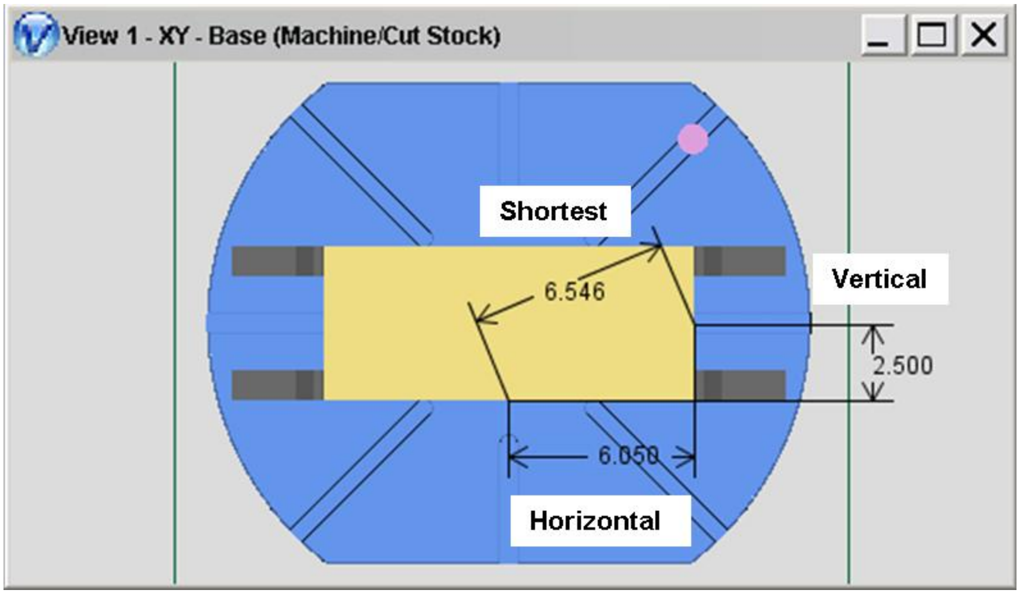
Dimension Type — Use to specify the type of linear dimension that you want to create.

Horizontal — Dimensions the horizontal distance between **Point 1** and **Point 2**.

Vertical — Dimensions the vertical distance between **Point 1** and **Point 2**.

Shortest — Dimensions the straight line distance between **Point 1** and **Point 2**.

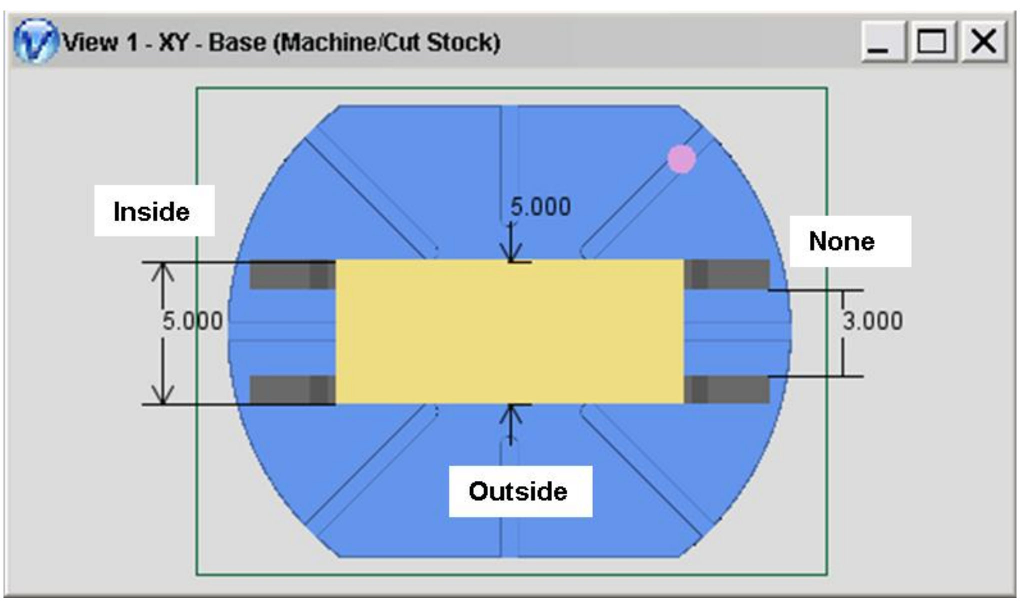
In the following picture, all three dimensions were defined using the same two points (the mid-points of the edges). Each of the three dimensions was created using a different Dimension Type as shown in the picture.



Arrow — Use to specify the type of arrow to be used for the dimension.

- **Inside —** Place the arrows between the two points.
- **Outside —** Place the arrows outside of the measured points.
- **None —** Do not use arrows for the dimension. Display only a line between the two points.

The following picture illustrates the difference between **Inside**, **Outside**, and **None**.



Point 1 — Choose one of the construction methods from the pull-down list, then click the arrow  to enter "pick" mode and follow the prompts below the message area (logger) to define the point. The following construction methods are available:

Point — Use to select a point.

Vector/Plane — Define a point represented by the intersection of a vector with a plane.

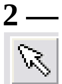
3 Planes — Define a point represented by the intersection of three planes.

Circle — Define a point represented by the center of a circle.

CSYS Origin — Define a point represented by the origin of a coordinate system (CSYS).

Once you have finished picking the geometry required to satisfy the chosen construction method, VERICUT will display the point in View 1 and display a red check next to **Point 1** on the Linear tab, as shown in the Linear tab picture above, to indicate that **Point 1** has been selected. VERICUT will also automatically activate "pick" mode for **Point 2**.

If you want to re-select **Point 1**, repeat the above steps.

Point 2 — Choose one of the construction methods from the pull-down list, then click the arrow  to enter "pick" mode and follow the prompts below the message area (logger) to define the point. The following construction methods are available:

Point — Use to select a point.

Vector/Plane — Define a point represented by the intersection of a vector with a plane.

3 Planes — Define a point represented by the intersection of three planes.

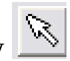
Circle — Define a point represented by the center of a circle.

CSYS Origin — Define a point represented by the origin of a coordinate system (CSYS).

Once you have finished picking the geometry required to satisfy the chosen construction method, VERICUT will display the point in View 1 and display a red check next to **Point 2** on the Linear tab, as shown in the Linear tab picture above, to indicate that **Point 2** has been selected. VERICUT will also automatically activate "pick" mode for **Text**.

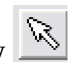
If you want to re-select **Point 2**, repeat the above steps.

Text — VERICUT will automatically fill in the dimension in the **Text** text field after you select **Point 1** and **Point 2**. Pick the position in View 1 where you want the dimension text placed.

If you want to move the text position, click the arrow  to enter "pick" mode, then pick the new location in View 1.

You also have the option of adding to the text, for example adding a tolerance value. If the text that you add is typical of all linear dimensions, use the **Number Label** feature on the Settings tab to have VERICUT automatically add the additional text to all linear dimensions.

You can also override the value that VERICUT enters, for example to create a "not to scale" dimension.

In either case, when you finish editing the text, click the arrow  to enter "pick" mode, then pick the position in View 1 where you want the dimension text placed.

Add — Use to add the dimension to the setup plan record. The dimension will display in View 1 in the **Dimension** color specified on the Settings tab.

Modify — The **Add** button changes to **Modify** when you are in "edit" mode. Use **Modify** to add your changes to the setup plan record.

To enter "edit" mode, simply select an existing dimension or note in View 1 so that it becomes highlighted in the **Edit** color specified on the Settings tab. VERICUT will automatically display the correct tab for editing the selected dimension or note. Use the tab features to change the dimension as desired. When finished with your changes, use the **Modify** button to add the changes to the setup plan record and display the modified dimension in View 1 in the **Dimension** color specified on the Settings tab.

See the Tips on Editing Setup Plan Dimensions topic below for additional information.

Delete — Use when in "edit" mode to delete the selected (highlighted) dimension from the setup plan record.

Cancel — Use to cancel the current selections for the dimension, and return it to its original condition. In "add" mode the dimension returns to the un-defined condition. In "edit" mode the dimension returns to its un-edited condition.

Steps for creating a linear dimension

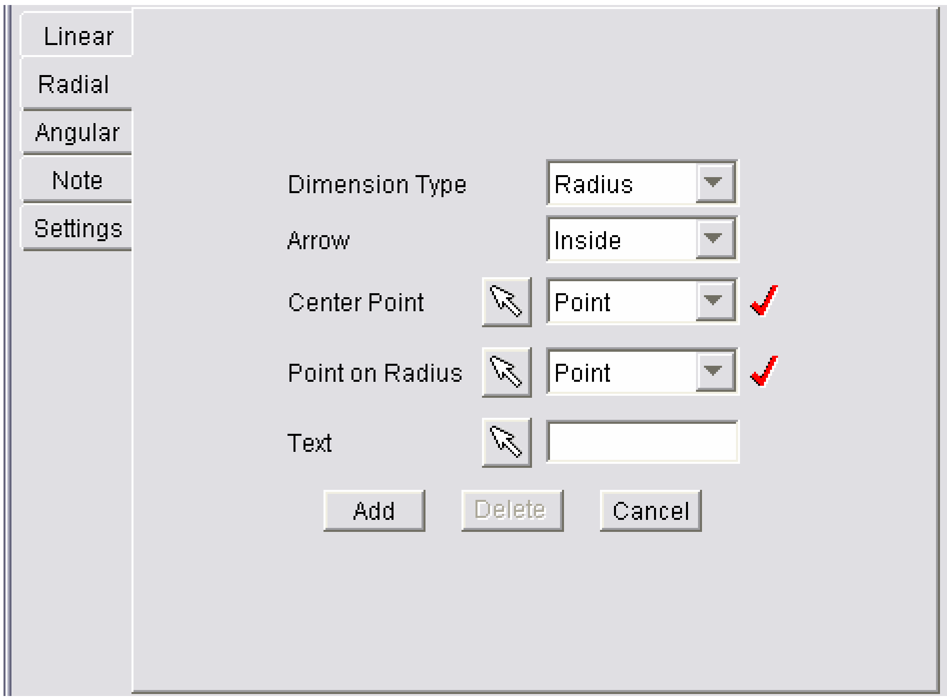
1. Choose the dimension type.
2. Choose the arrow type.
3. Pick the start point of the linear dimension (**Point 1**).
4. Pick the end point of the linear dimension (**Point 2**).
5. Pick the location for the dimension text.
6. Select **Add** to accept the current selections and add the linear dimension to the setup plan record.

To Setup Plan window

Setup Plan window, Radial tab

Location: **Project menu > Report > Setup Plan**

The features on this tab enable you to add new radial dimensions to, or edit existing radial dimensions in, a setup plan record.



Dimension Type — Use to specify the type of radial dimension that you want to create.

Radius — Display the dimension using the radius value.

Diameter — Display the dimension using the diameter value.

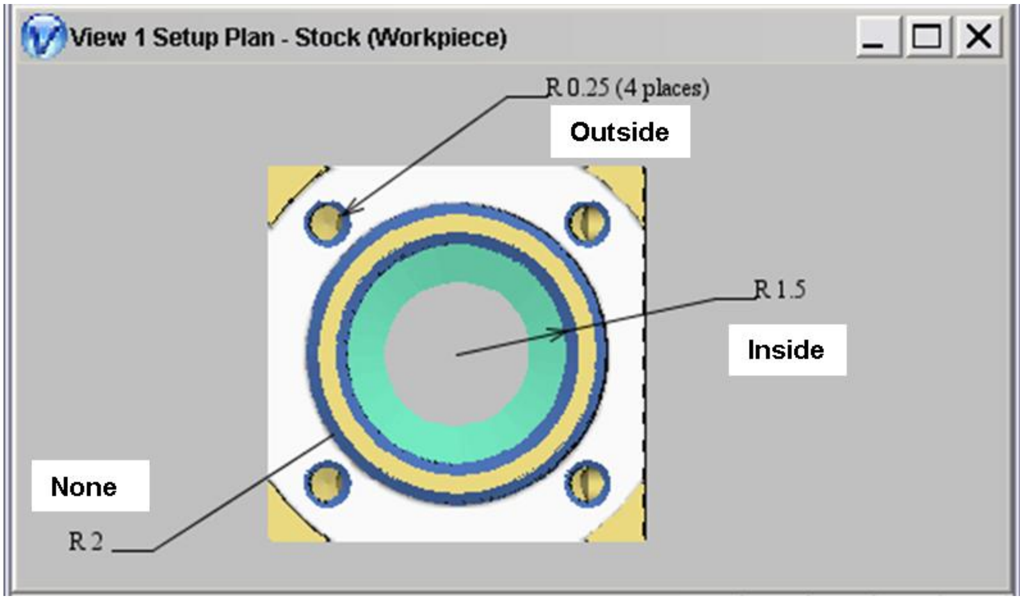
Arrow — Use to specify the type of arrow to be used for the dimension.

Inside — Places the arrow inside of the radial entity being dimensioned.

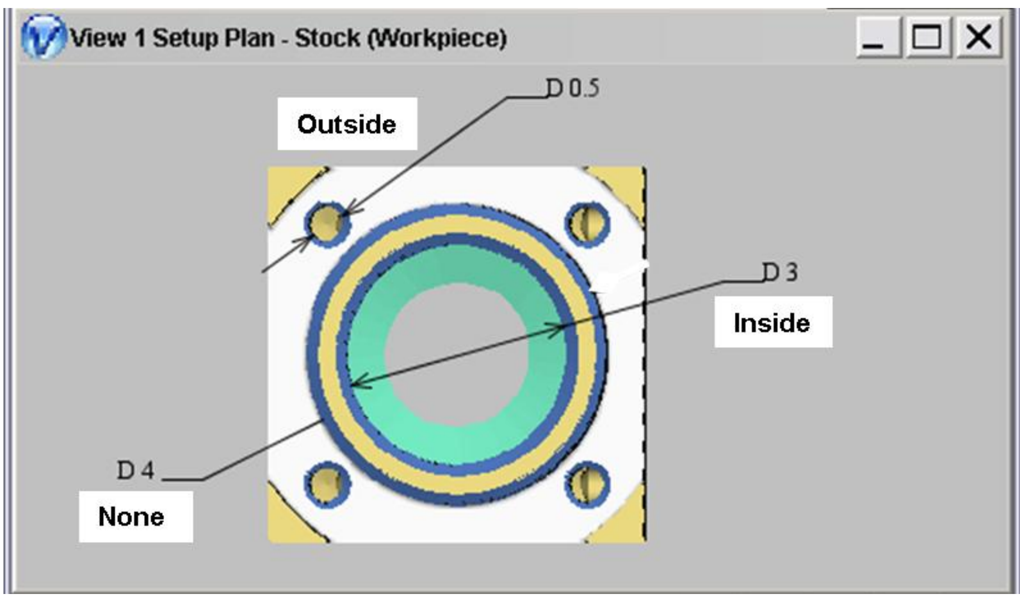
Outside — Places the arrow outside of the radial entity being dimensioned.

None — Do not use arrows for the dimension. Only display a line outside of the radial entity being dimensioned.

The following picture illustrates the difference between **Inside**, **Outside**, and **None** when creating **Radius** dimensions.



The following picture illustrates the difference between **Inside**, **Outside**, and **None** when creating **Diameter** dimensions.



Center Point — Choose one of the construction methods from the pull-down list, then click the arrow  to enter "pick" mode and follow the prompts below the message area (logger) to define the point representing the center of the circle/arc being dimensioned. The following construction methods are available:

Point — Use to select a point.

Vector/Plane — Define a point represented by the intersection of a vector with a plane.

3 Planes — Define a point represented by the intersection of three planes.

Circle — Define a point represented by the center of a circle.

CSYS Origin— Define a point represented by the origin of a coordinate system (CSYS).

Once you have finished picking the geometry required to satisfy the chosen construction method, VERICUT will display the point in View 1 and display a red check next to **Center Point** on the Radial tab, as shown in the Radial tab picture above, to indicate that the center point has been selected. VERICUT will also automatically activate "pick" mode for **Point on Radius**.

If you want to re-select Point 1, repeat the above steps.

Point on Radius — Choose one of the construction methods from the pull-down list, then click the arrow  to enter "pick" mode and follow the prompts below the message area (logger) to define a point on the circle/arc being dimensioned. The following construction methods are available:

Point — Use to select a point.

Vector/Plane — Define a point represented by the intersection of a vector with a plane.

3 Planes — Define a point represented by the intersection of three planes.

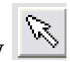
Circle — Define a point represented by the center of a circle.

CSYS Origin — Define a point represented by the origin of a coordinate system (CSYS).

Once you have finished picking the geometry required to satisfy the chosen construction method, VERICUT will display the point in View 1 and display a red check next to **Point on Radius** on the Radial tab, as shown in the Radial tab picture above, to indicate that point on the circle/arc being dimensioned has been selected. VERICUT will also automatically activate "pick" mode for **Text**.

If you want to re-select **Point 2**, repeat the above steps.

Text — VERICUT will automatically fill in the dimension in the **Text** text field after you select **Center Point** and **Point on Radius**. Pick the position in View 1 where you want the dimension text placed.

If you want to move the text position, click the arrow  to enter "pick" mode, then pick the new location in View 1.

You also have the option of adding to the text, for example adding a tolerance value. If the text that you add is typical of all radial dimensions, use the **Number Label** feature on the Settings tab to have VERICUT automatically add the additional text to all radial dimensions.

You can also override the value that VERICUT enters, for example to create a "not to scale" dimension.

In either case, when you finish editing the text, click the arrow  to enter "pick" mode, then pick the position in View 1 where you want the dimension text placed.

Add — Use to add the dimension to the setup plan record. The dimension will display in View 1 in the **Dimension** color specified on the Settings tab.

Modify — The **Add** button changes to **Modify** when you are in "edit" mode. Use **Modify** to add your changes to the setup plan record.

To enter "edit" mode, simply select an existing dimension or note in View 1 so that it becomes highlighted in the **Edit** color specified on the Settings tab. VERICUT will automatically display the correct tab for editing the selected dimension or note. Use the tab features to change the dimension as desired. When finished with your changes, use the **Modify** button to add the changes to the setup plan record and display the modified dimension in View 1 in the **Dimension** color specified on the Settings tab.

See the Tips on Editing Setup Plan Dimensions topic below for additional information.

Delete — Use when in "edit" mode to delete the selected (highlighted) dimension from the setup plan record.

Cancel — Use to cancel the current selections for the dimension, and return it to its original condition. In "add" mode the dimension returns to the un-defined condition. In "edit" mode the dimension returns to its un-edited condition.

Steps for creating a radial dimension

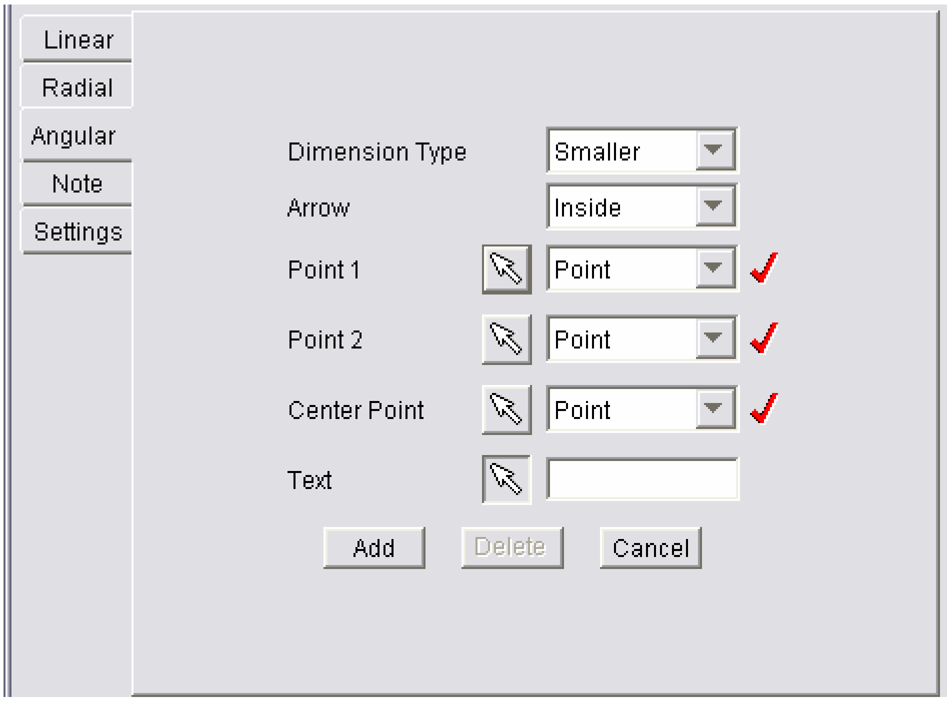
1. Choose the dimension type.
2. Choose the arrow type.
3. Pick the center point of the circle/arc being dimensioned (**CenterPoint**).
4. Pick a point on the circle/arc being dimensioned (**Point on Radius**).
5. Pick the location for the dimension text.
6. Select **Add** to accept the current selections and add the radial dimension to the setup plan record.

To Setup Plan window

Setup Plan window, Angular tab

Location: **Project menu > Report > Setup Plan**

The features on this tab enable you to add new angular dimensions to, or edit existing angular dimensions in, a setup plan record.

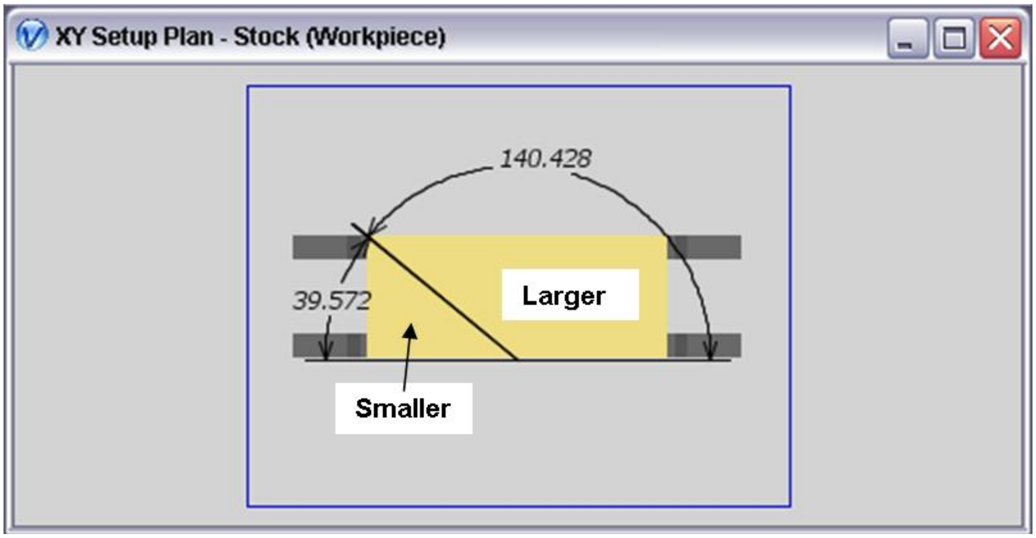


Dimension Type — Use to specify the type of angular dimension that you want to create.

Smaller — Displays the smaller of the two angles defined using the three points specified below. For example the same three points could define both a 30 degree and a 150 degree angle. Use **Smaller** to dimension the 30 degree angle.

Larger — Displays the larger of the two angles defined using the three points specified below. For example the same three points could define both a 30 degree and a 150 degree angle. Use **Larger** to dimension the 150 degree angle.

The following picture illustrates the difference between **Smaller** and **Larger**.



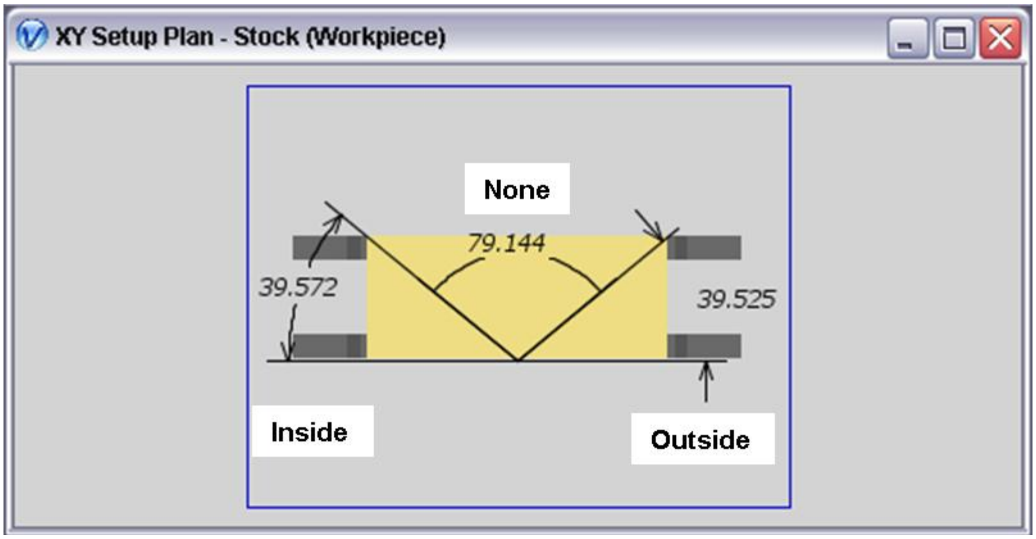
Arrow — Use to specify the type of arrow to be used for the dimension.

Inside — Place the arrows between the lines defining the angle.

Outside — Place the arrows outside of the lines defining the angle.

None — Do not use arrows for the dimension. Only an arc displays between the lines defining the angle.

The following picture illustrates the difference between **Inside**, **Outside**, and **None**.



Point 1 — Choose one of the construction methods from the pull-down list, then click the arrow  to enter "pick" mode and follow the prompts below the message area (logger) to define a point on the line representing one side of the angle. The following construction methods are available:

Point — Use to select a point.

Vector/Plane — Define a point represented by the intersection of a vector with a plane.

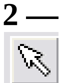
3 Planes — Define a point represented by the intersection of three planes.

Circle — Define a point represented by the center of a circle.

CSYS Origin — Define a point represented by the origin of a coordinate system (CSYS).

Once you have finished picking the geometry required to satisfy the chosen construction method, VERICUT will display the point in View 1 and display a red check next to **Point 1** on the Angular tab, as shown in the Angular tab picture above, to indicate that a point on a line representing the first side of the angle being dimensioned has been selected. VERICUT will also automatically activate "pick" mode for **Point 2**.

If you want to re-select **Point 1**, repeat the above steps.

Point 2 — Choose one of the construction methods from the pull-down list, then click the arrow  to enter "pick" mode and follow the prompts below the message area (logger) to define a point on the line representing the other side of the angle. The following construction methods are available:

Point — Use to select a point.

Vector/Plane — Define a point represented by the intersection of a vector with a plane.

3 Planes — Define a point represented by the intersection of three planes.

Circle — Define a point represented by the center of a circle.

CSYS Origin — Define a point represented by the origin of a coordinate system (CSYS).

Once you have finished picking the geometry required to satisfy the chosen construction method, VERICUT will display the point in View 1 and display a red check next to **Point 2** on the Angular tab, as shown in the Angular tab picture above, to indicate that a point on a line representing the second side of the angle has been selected. VERICUT will also automatically activate "pick" mode for **Center Point**.

If you want to re-select **Point 2**, repeat the above steps.

Center Point — Choose one of the construction methods from the pull-down list, then click the arrow  to enter "pick" mode and follow the prompts below the message area (logger) to define the point representing the vertex of the angle. The following construction methods are available:

> **Point —** Use to select a point.
>
> **Vector/Plane —** Define a point represented by the intersection of a vector with a plane.
>
> **3 Planes —** Define a point represented by the intersection of three planes.
>
> **Circle —** Define a point represented by the center of a circle.
>
> **CSYS Origin —** Define a point represented by the origin of a coordinate system (CSYS).

Once you have finished picking the geometry required to satisfy the chosen construction method, VERICUT will display the point in View 1 and display a red check next to **Center Point** on the Angular tab, as shown in the Angular tab picture above, to indicate that point representing the vertex of the angle being dimensioned has been selected. VERICUT will also automatically activate "pick" mode for **Text**.

If you want to re-select **Center Point**, repeat the above steps.

Text — VERICUT will automatically fill in the dimension in the **Text** text field after you select **Point 1**, **Point 2**, and **Center Point**. Pick the position in View 1 where you want the dimension text placed.

If you want to move the text position, click the arrow  to enter "pick" mode, then pick the new location in View 1.

You also have the option of adding to the text, for example adding a tolerance value. If the text that you add is typical of all angular dimensions, use the **Number Label** feature on the Settings tab to have VERICUT automatically add the additional text to all angular dimensions.

You can also override the value that VERICUT enters, for example to create a "not to scale" dimension.

In either case, when you finish editing the text, click the arrow  to enter "pick" mode, then pick the position in View 1 where you want the dimension text placed.

Add — Use to add the dimension to the setup plan record. The dimension will display in View 1 in the **Dimension** color specified on the Settings tab.

Modify — The **Add** button changes to **Modify** when you are in "edit" mode. Use **Modify** to add your changes to the setup plan record.

To enter "edit" mode, simply select an existing dimension or note in View 1 so that it becomes highlighted in the **Edit** color specified on the Settings tab. VERICUT will automatically display the correct tab for editing the selected dimension or note. Use the tab features to change the dimension as desired. When finished with your changes, use the **Modify** button to add the changes to the setup plan record and display the modified dimension in View 1 in the **Dimension** color specified on the Settings tab.

See the Tips on Editing Setup Plan Dimensions topic below for additional information.

Delete — Use when in "edit" mode to delete the selected (highlighted) dimension from the setup plan record.

Cancel — Use to cancel the current selections for the dimension, and return it to its original condition. In "add" mode the dimension returns to the un-defined condition. In "edit" mode the dimension returns to its un-edited condition.

Steps for creating a radial dimension

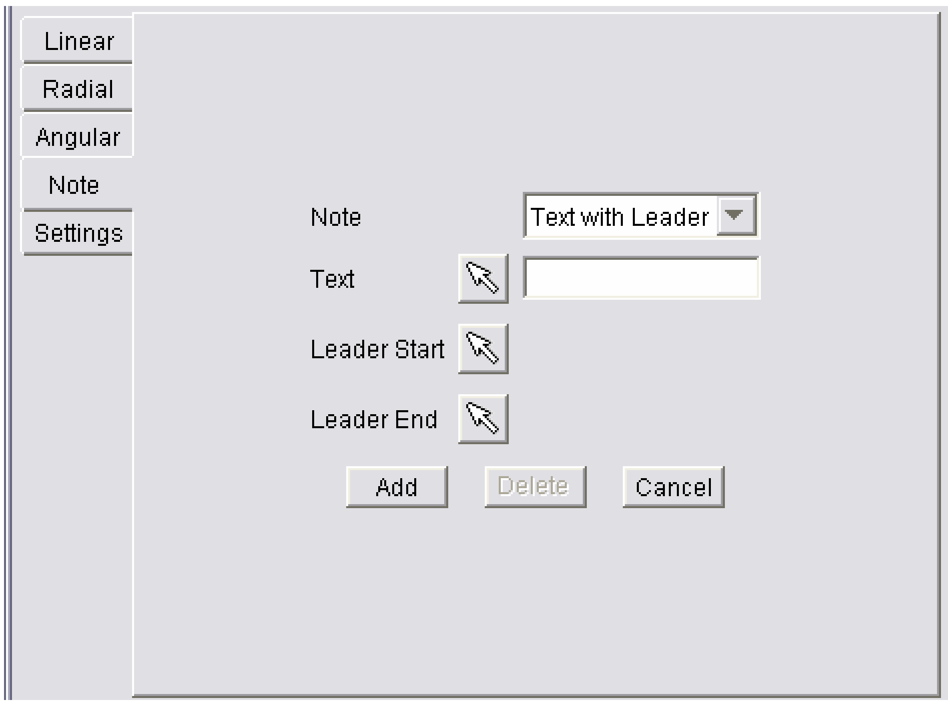
1. Choose the dimension type.
2. Choose the arrow type.
3. Pick a point on a line representing one side of the angle (**Point 1**).
4. Pick a point on a line representing the second side of the angle (**Point 2**).
5. Pick the point representing the vertex of the angle being dimensioned (**CenterPoint**).
6. Pick the location for the dimension text.
7. Select **Add** to accept the current selections and add the radial dimension to the setup plan record.

To Setup Plan window

Setup Plan window, Note tab

Location: **Project menu > Report > Setup Plan**

The features on this tab enable you to add new notes to, or edit existing notes in, a setup plan record.



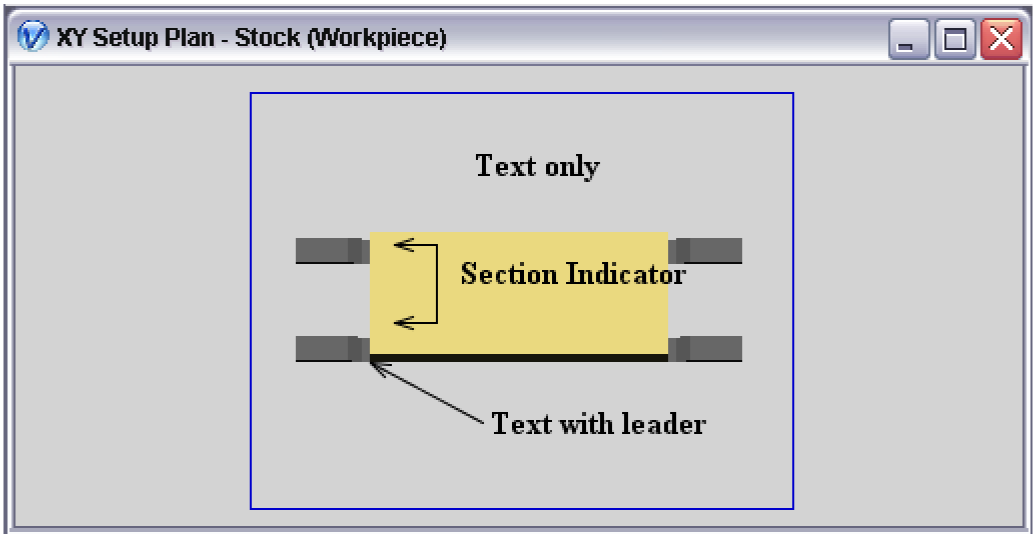
Note — Select the style of note that you want to add from the pull-down list. Select one of the following styles:

Text — Use to add a text only note to the setup plan.

Text with Leader — Use to add a text note with a leader to the setup plan.

Section Indicator — Use to add a section indicator with label to the setup plan.

The following picture illustrates the difference between **Text**, **Text with Leader**, and **Section Indicator** when creating notes.



Text — Use the text field to specify the text associated with the note, or section indicator label.

Then click the arrow , to enter "pick" mode, and pick the location in View 1 where you want the note/section indicator text placed.

Leader Start — Click the arrow , to enter "pick" mode, then pick the location in View 1 where you want the note leader to start. This is the end of the leader with the arrow.

When creating Section Indicators, this feature is used to specify the start location of the section indicator. The relationship between this location and the one specified using Leader End (described below) are used to indicate the length and orientation if the linear portion of the section indicator.

The relationship between the linear portion of the section indicator and the location that you picked to position the section leader text determines the direction of the section indicator arrows. The arrows will always point away from the text pick location.

Leader End — Click the arrow , to enter "pick" mode then pick the location in View 1 where you want the note leader to end. This is the straight end of the leader.

When creating Section Indicators, this feature is used to specify the end location of the section indicator. The relationship between this location and the one specified using Leader Start (described above) are used to indicate the length and orientation if the section indicator.

The relationship between the linear portion of the section indicator and the location that you picked to position the section leader text determines the direction of the section indicator arrows. The arrows will always point away from the text pick location.

Add — Use to add the note/section indicator to the setup plan record. The notes will display in View 1 in the **Text** color specified on the Settings tab. The note leader , or section indicator will display in the **Dimension** color specified on the Settings tab.

Modify — The **Add** button changes to **Modify** when you are in "edit" mode. Use **Modify** to add your changes to the setup plan record.

To enter "edit" mode, simply select an existing note, or section indicator, in View 1 so that it becomes highlighted in the **Edit** color specified on the Settings tab. VERICUT will automatically display the correct tab for editing the selected note. Use the tab features to change the note/section indicator as desired. When finished with your changes, use the **Modify** button to add the changes to the setup plan record and display the modified dimension in View 1 in the colors specified on the Settings tab.

See the Tips on Editing Setup Plan Dimensions topic below for additional information.

Delete — Use when in "edit" mode to delete the selected (highlighted) dimension from the setup plan record.

Cancel — Use to cancel the current selections for the dimension, and return it to its original condition. In "add" mode the dimension returns to the un-defined condition. In "edit" mode the dimension returns to its un-edited condition.

Steps for creating a note:

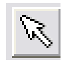
1. Choose the **Note** type (**Text** or **Text with Leader**).
2. Enter the note text in the text field.
3. Click on  , then pick the position in View 1 where you want the note placed.

If you are creating a **Text with Leader** note, continue with the following:

4. Click on the **Leader Start**  , then pick the start point of the Note leader.
5. Click on the **Leader End**  , then pick the end point of the Note leader.
6. Select **Add** to accept the current selections and add the note to the setup plan record.

Steps for creating a section indicator:

1. Set **Note** type to **Section Indicator**.
2. Enter the text associated with the section indicator in the text field.

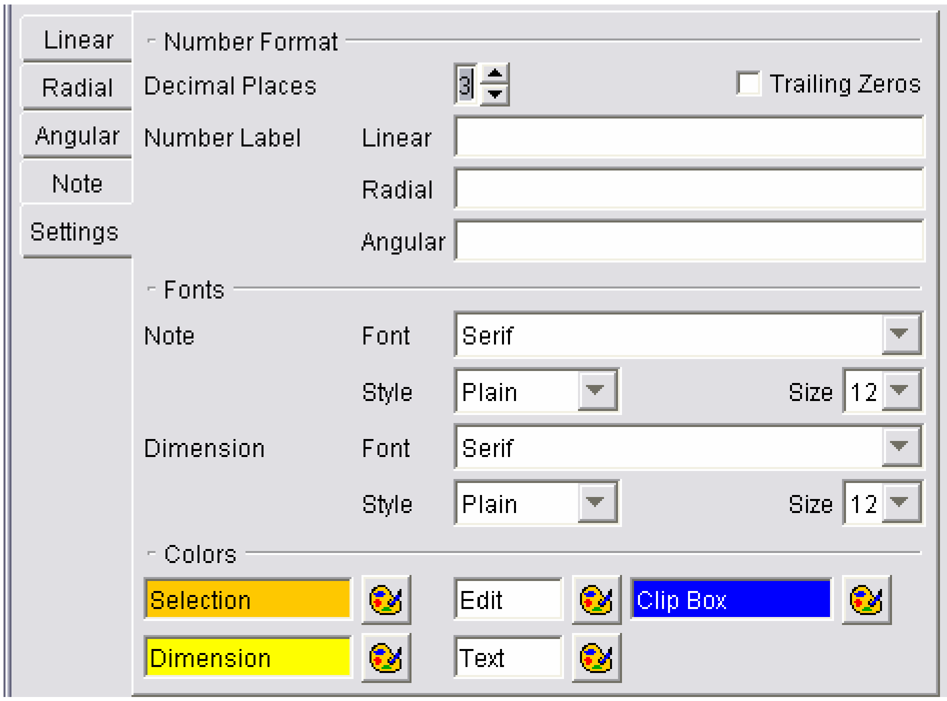
3. Click on  , then pick the position in View 1 where you want the section indicator text to be placed.
4. Click on the **Leader Start**  , then pick the location in View 1 where you want the section indicator to start.
5. Click on the **Leader End**  , then pick the location in View 1 where you want the section indicator to end.
6. Select **Add** to accept the current selections and add the note to the setup plan record.

To Setup Plan window

Setup Plan window, Settings tab

Location: **Project menu > Report > Setup Plan**

The features on this tab enable you to specify, or edit existing, text characteristics (format, font, color, etc.), to be used in the setup plan.



Number Format — Use the Number Format features to specify how numbers are to be displayed in the Setup Plan.

Decimal Places — Use to specify the number of decimal places to be displayed.

Trailing Zeros — When toggled "On" trailing zeros will be displayed.

Number Label — Enables you to append the specified characters to the end of a number (i.e., in, mm. TYP, etc.). This feature only appends to VERICUT calculated values. They will not append to user entered values (for example, when creating "not to scale" dimensions).

Linear — Use to specify the characters to be added at the end of linear dimensions.

Radial — Use to specify the characters to be added at the end of radial dimensions.

Angular — Use to specify the characters to be added at the end of angular dimensions.

Fonts — Use the Fonts features to specify the font characteristics (font, style, and size) to be displayed in the Setup Plan.

Note — Use the following to specify font characteristics to be used for Notes.

Font — Select the desired font from the pull-down list.

Style — Select **Plain**, **Bold**, or **Italic** from the pull-down list.

Size — Select the desired font size from the pull-down list.

Dimension — Use the following to specify font characteristics to be used for Dimensions.

Font — Select the desired font from the pull-down list.

Style — Select **Plain**, **Bold**, or **Italic** from the pull-down list.

Size — Select the desired font size from the pull-down list.

Colors — Use to specify colors for specific Setup Plan entities. Click on the color pallet icon,  associated with each entity type (**Selection**, **Edit**, etc.) to display a color selection pallet. Use it to select the desired color for the entity type. The entity type label on the Settings tab will display in the currently selected color so that you can tell at a glance what the current setting is.

Selection — Use to specify the selection (highlight) color to display dimensions and notes when you hold the cursor over them, to assist in selecting existing entities editing.

Edit — Use to specify the color to display dimensions and notes when they are in edit mode.

Clip Box — Use to specify the color to display the clip box (ref. "Clip" in the Setup Plan window).

Dimension — Use to specify the color to display dimension lines, note leaders, and section markers.

Text — Use to specify the color to display the text in dimensions, notes and section indicators.

To Setup Plan window

Things to Remember When Working With Setup Plan

The following things should be kept in mind when working with Setup Plan.

- Setup Plan always displays in the first viewport.
- A setup plan is always controlled by a stored view. You either select one that exists, or one is automatically created for you.
- If you choose Active View, or one of the internal views (XY, YX, etc.), from the first dialog, then a stored view is automatically created. The content of the currently highlighted view determines the setup plan's view type and attributes for automatically created stored view.
- Do not use an OpenGL machine view for Setup Plan.
- All dimensions are 2D, projected into a dimension plane. They are either projected into the stored view's plane, or they are projected into the XY place of a selected CSYS. Depends on your choice in the CSYS column of the Setup Plan table.
- All dimensions are point-to-point. They do not dimension "features". They are all constructed from points projected into the dimension plane. Each dimension type has point-constructor choices.
- The dimension text field is text, not numeric. You can enter any text string you want to into a dimension text field. When the dimension is initially created the text field is filled by a calculated number. The text field is re-calculated during modification if:
    a. the text field is blanked
    b. an end-point is moved
    c. the **Recalculate** button is pressed.
- Setup plan "settings" are saved in the preferences file (cgtech_*xx*_user.prefs, where *xx* is the VERICUT version number).

Tips on Editing Setup Plan Dimensions

The following is intended to get you started with using Setup Plan window "Edit" mode.

To enter edit mode:
Select an existing dimension or note in the view so that it becomes highlighted.
VERICUT will automatically display the correct tab for editing the selected dimension.

Use the tab features to change the dimension as desired. Most of the features just required selecting a different option from the pull-down list. The ones shown below may be slightly less obvious.

Editing a point:
In "edit" mode, select the dimension in the view so that it becomes highlighted.
Changing a point in a dimension is just like creating it initially. For the point that you want to edit:
Choose one of the construction methods from the pull-down list, then click the arrow  and follow the prompts in the message area to define the new point.

Changing the position of the dimension text:
In "edit" mode, select the dimension in the view so that it becomes highlighted.
Click the arrow  next to **Text**, then pick the location in the view where you want the dimension moved to. You can move the location of the dimension closer, or further away, than its current position depending on the new location that you pick. You can also move just the text associated with the dimension to another position along the current dimension line.

Add changes to the Setup Plan record:
When finished with your changes for a dimension, use the **Modify** button to add the changes to the setup plan record.
Use **OK** or **Apply** in the Setup Plan window to save the setup plan to the project file.

Adding a Setup Plan Image to a Report.

The following provides an overview of the steps needed to add a setup plan to a report.

1. Use **View menu** > **Select/Store** to display the Select/Store View window and use it to create the views that you want in the setup plan.

2. Use **Project menu** > **Report** > **Setup Plan**, or click on  (**Setup Plan**) in the toolbar, to display the Setup Plan window and use it's features to create the setup plan record(s) (images) that you want in the Setup Plan report.

3. Use **Project menu** > **Report** > **Report Template** > **Edit** to display the Report Template window and use it's features to create a new, or modify an existing report template, to include the Setup Plan image, or images.

   Single Image Setup Plan

   - On the **Report Template window: Page Layout tab** select the record that you want the Setup Plan added after so that it becomes highlighted.
   - In the Report Template window menubar select **Edit** > **Add** > **Picture** to display the Picture window.
   - In the Picture window, select **Setup Plan** from the pull-down list, then **OK**. The Setup Plan record will be added to the report template.
   - Save the report template.

   NOTE: If you use this approach for a multiple image setup plan, only the first record in the Setup Plan table will be included in the report.

   Multiple Image Setup Plan

   - On the Report Template window: Page Layout tab select the record that you want the Setup Plan added after so that it become highlighted.
   - In the Report Template window menubar select **Edit** > **Add** > **Start View Loop** to add the Start View Loop record to the report template.
   - In the Report Template window menubar select **Edit** > **Add** > **Picture** to display the Picture window.
   - In the Picture window, select **Setup Plan** from the pull-down list, then **OK**. The Setup Plan record will be added to the report template.
   - In the Report Template window menubar select **Edit** > **Add** > **End View Loop** to add the End View Loop record to the report template.
   - Save the report template.

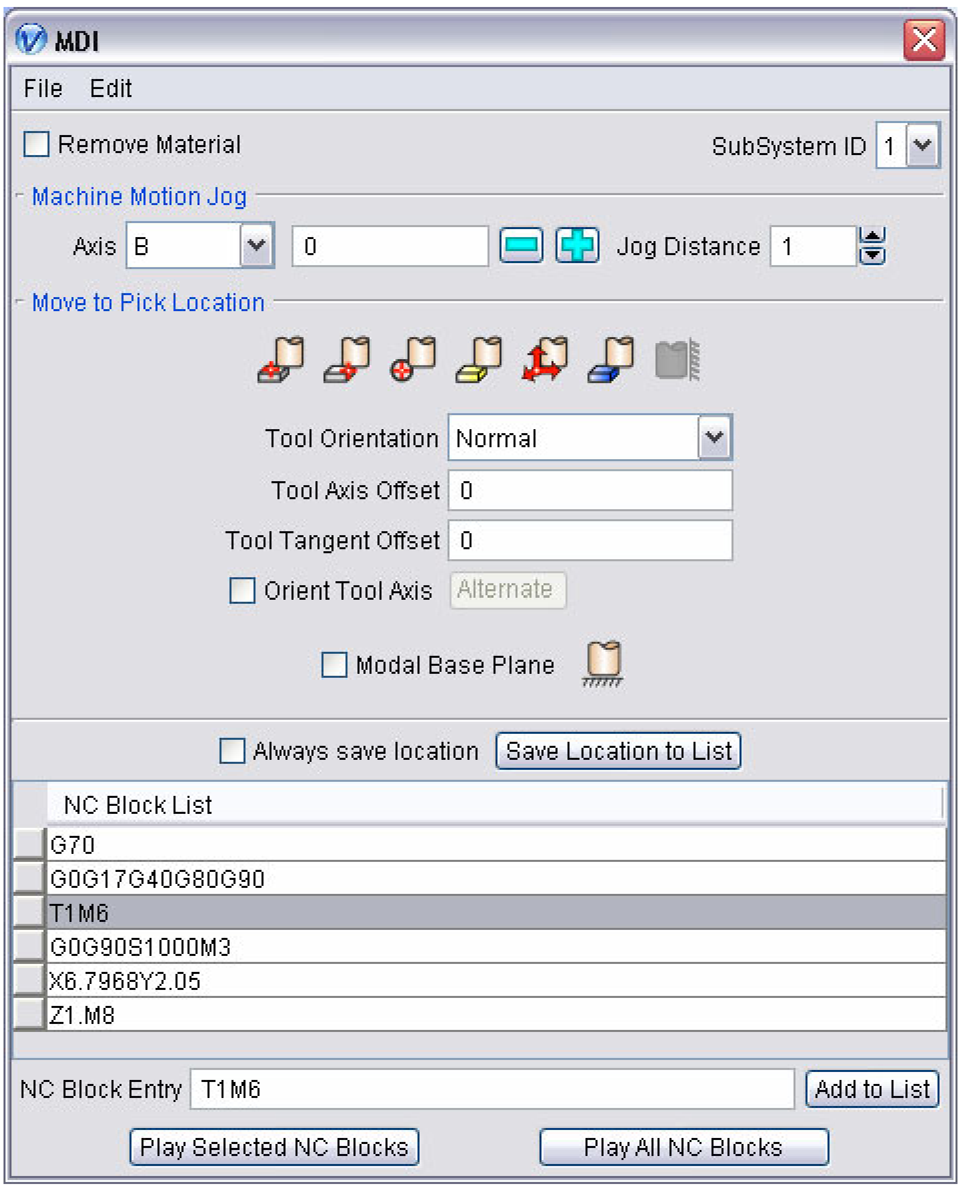
4. Use **File menu** > **Create Report** > **(*format*)** to generate the report.

MDI window

Location: **Project menu > MDI**

VERICUT toolbar short cut: 

Opens a window enabling you to manually enter and process blocks of G-Code data. The MDI, or "Manual Data Input" function provides a quick and easy way of verifying that the machine/control combination responds to G-Code data commands as expected.



Menu Bar

File Menu

The features in the File menu enable you to import, export or print NC blocks created in the NC Blocks table.

Options:

Open — Opens a file selection window enabling you to specify a text, or .mcd, file to populate the NC Blocks table.

Save — Saves (updates) an existing text (.mcd) file with the current NC blocks.

Save As — Opens a window enabling you to save the contents of the NC Blocks table to a new file.

Print — Opens a window enabling you to print the NC Blocks table.

Close — Closes the MDI window.

Edit Menu

The features in the Edit menu enable you to manipulate blocks in the NC Block table.

Options:

Copy — Use to copy the highlighted block(s) in the NC Block table to the Windows clipboard.

Paste — Use to paste a copied block(s) after the highlighted block in the NC Block table.

Delete — Use to delete the highlighted block(s) from the NC Block table

Clear All — Use to clear all blocks from the NC Block table.

Main Window

Remove Material — Use to activate/de-activate VERICUT material removal.

When toggled "**On**" (check), material removal as well as machine motion will be displayed.

When toggled "**Off**" (no check), only machine components are positioned to specified location and no cutting is performed. This is useful for positioning the tool at specific location before cutting, or to verify the actual tool position before executing new blocks.

SubSystem ID — Use this feature to specify which machine subsystem the G-Code data command is to be applied to.

Machine Motion Jog

The **Machine Motion Jog** features enable you to incrementally jog the position of the VERICUT machine in the same way that you can jog the real machine.

Axis — Indicates the current axis that the "Jog" feature will apply to. To change the current axis, simply select it from the pull-down list. The Axis list includes all machine moving axes as well as the Tool Axis.

The current axis position (linear or rotary), with respect to the **Axis** "zero" position, is displayed in text field next to **Axis**.

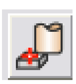
Use the  (increase) and  (decrease) buttons to jog the machine the specified "**Jog Distance**" each time you press them. You can also press and hold the button down to repeatedly move the specified distance.

Jog Distance — Use to set the distance the axis position will move by a single click on increase/decrease buttons.

NOTE: Moving the machine axes with **Machine Motion Jog** will not produce a block in the NC Blocks table until the **Save Location as NC Block** button is pressed.

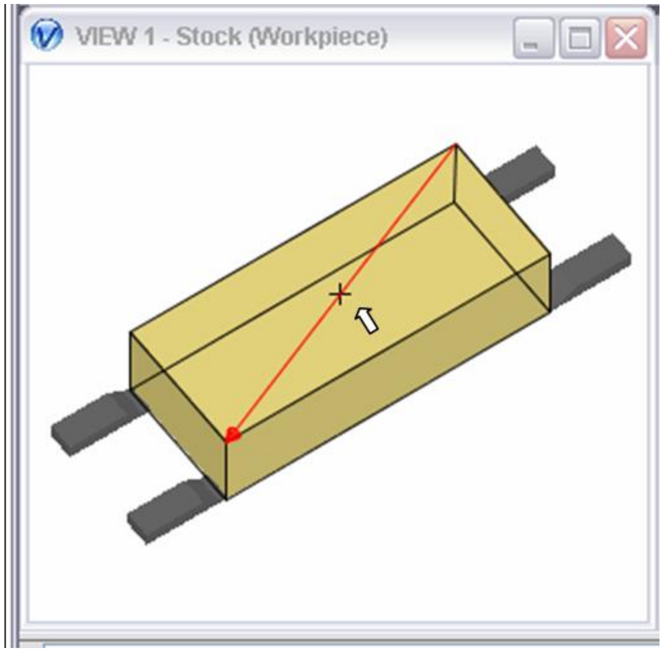
Move to Pick Location

The **Move to Pick Location** features enable you to position a tool component at a specific point by picking on stock, or machine models and components, displayed in the VERICUT Graphics Area.

 **Pick Point —** enables you to position the tool at any point in a Workpiece or Machine view. VERICUT will use the selected point and a vector normal to the surface of model or stock at the selected point. Moving the cursor in the view over model will display the point and vector associated with the current position. Click the left mouse button to select the point.

 **Pick Vertex —** enables you to easily position a tool on one of the six key points associated with the triangles representing the faces of an uncut model (machine, fixture components or stock model before cutting). For each triangle, the points consist of the three vertices and the midpoint of each of the triangle's three sides. VERICUT selects the point closest to your mouse pick. Moving the cursor in the view over model will display the nearest triangle side and point. Click the left mouse button to select the point. VERICUT will use the selected point and a vector normal to the surface of model or stock at the selected point. If the mouse pick is over the cut stock, **Pick Vertex** works in the same way as the **Pick Point** feature described above.

Pick Vertex Example



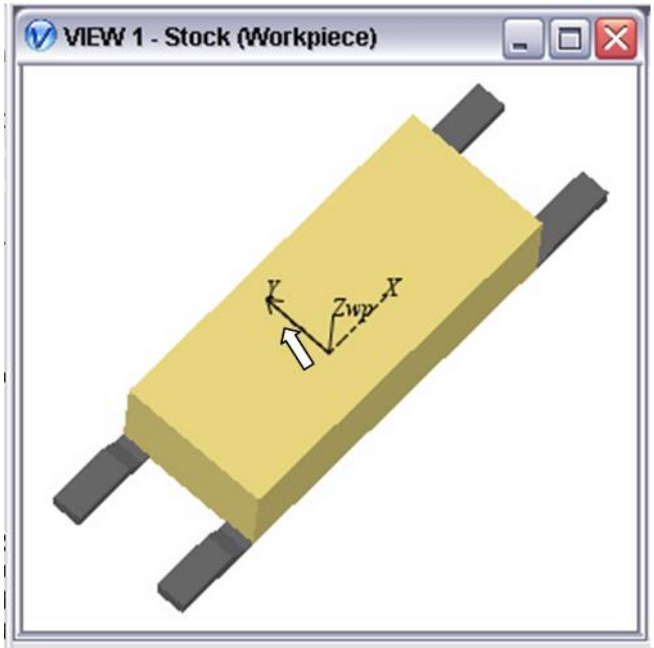
 **Pick Circle —** enables you to position a tool at a point along the axis of a cylinder. This feature requires two picks. The first pick designates the plane that the point is to be in. The second pick is the cylindrical feature whose axis the point lies along. VERICUT will use the point where the axis of the cylindrical feature intersects the plane. The associated vector is defined by the cylindrical axis.

 **Pick Model Origin —** enables you to position the tool at a point represented by the origin of a model's coordinate system. The vector is defined as the model coordinate system's Z-axis. Placing the cursor over any model will display the model's coordinate system. Select the model using the left mouse button.

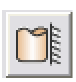
 **Pick CSYS Origin —** enables you to position a tool at a point represented by the origin of any displayed coordinate system. VERICUT uses the coordinate system axis closest to the pick point as the vector. Move the cursor near one of the coordinate system's axis and an arrow representing the vector will display along the axis. Click with the left mouse button select the origin point and vector.

Pick CSYS Origin Example

Notice the cursor positioned near the Y-axis of the CSYS and the arrow displayed on the axis indicating that it will be used as the vector.



 **Pick Component Origin —** enables you to position the tool at a point represented by the origin of a component's coordinate system. The vector is defined as the component coordinate system's Z-axis. Placing the cursor over any component will display the component's coordinate system. Select the component using the left mouse button.

 **Pick Next Side Plane —** when active, this feature enables you to select a plane to position the side of the tool to. Activate "**Pick Next Side Plane**" mode by:

1. Use the **Pick Base Plane** feature, described below, to select a base plane.
2. Toggle **Modal Base Plane**, described below, "**On**".
3. Set **Tool Orientation**, described below, to **Tangent**.
4. Position the tool using any of the above methods. At this point the **Pick Next Side Plane** icon should be active.

Note that the order in which these steps are carried out is important.

After completing the above steps, the tool is positioned tangent to the surface at the pick point. This tangent plane can be driven along for the next motion if the next pick is in "**Pick Next Side Plane**" mode. After the motion to the "next side plane" is completed, the previous "side plane" is replaced by current "next side plane" and tool is ready to be driven to the next "**Pick Next Side Plane**" location. You can continue driving the tool along in this manner as long as you continue selecting "**Pick Next Side Plane**" locations.

Tool Orientation — Use to specify the tool vector orientation in relation to the picked vector (using any of 6 methods specified above). Choose either **Normal**, or **Tangent**.

Normal specifies that tool vector is set to the pick vector value. **Tangent** specifies that tool vector will be aligned tangent to the plane created by crossing last tool vector and pick vector. The tool position will be offset by maximum tool radius from the plane. Using a **Tangent** orientation with the **Pick Point** method will position the tool tangent to a wall by using one mouse click on the wall. **Tool Tangent Offset** can be applied in addition to tool radius offset.

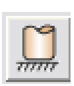
Tool Axis Offset — Use to specify a value representing a distance offset the tool along the tool axis vector. A positive value moves up the tool axis from the tool pick point. A negative value moves down the tool axis from the pick point.

Tool Tangent Offset — Use to specify a value representing an offset of the tool in radial direction (perpendicular to the tool axis), in addition to tool radius, when **Tangent** tool orientation is active. This offset can be used to prevent scratching a part when positioning tool tangent to wall (positive value), or for cutting a wall (negative value).

Orient Tool Axis — Use to change tool axis orientation. When the box is checked and machine has 5 axis capabilities, the tool axis will be set based on pick vector value and **Tool Orientation** method. If the box is not checked, the tool axis vector remains unchanged while changing to the position to pick point.

Alternate — Use this button to orient the tool using alternative rotary axes solution. When tool axis vector is resolved on a pick, the shortest route for rotary axes is applied. To find what the alternative solution is, use this button. This button is active only after a change to the tool axis vector. Subsequent use of this button toggles from one possible solution, to another.

Modal Base Plane — Use this check box to specify a modal base plane is to be used as the reference for **Tool Axis Offset**. If box is checked, tool positions for all pick generated motions are offset relative to the specified modal base plane. Use **Pick Base Plane**, described below, to select the plane. If the box is unchecked, tool positions for all pick generated motions are offset relative to current pick point.

 **Pick Base Plane —** Use to select a modal base plane (see above) to establish a common base for **Tool Axis Offset**. This icon is in same group with other Move to Pick Location icons (deactivates the others when pressed) but it is a one shot pick (becomes inactive after each successful pick).

Always Save Location — Use this feature to simplify NC block generation. When the check box is checked, an NC block is created in **NC Block Entry** text field for every resolved pick location and is saved in NC Blocks List. If the box is not checked, you must use the **Save Location to List** button, at each pick location, to create an NC block and store it in NC Block List. This feature is not applicable when **Machine Motion Jog** is being used since a single jog does not create any NC Block information to save.

Save Location to List — Use to create an NC block from the current pick location and save it in NC Block List.

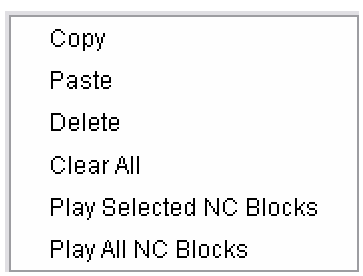
NOTE: The **Save Location to List** button does not execute the block in **NC Block Entry** text field. It simply creates the NC block and stores it in the NC Block List.

In **Machine Motion Jog** mode, it is important to create NC blocks for each desired machine position ***before*** jogging the machine to the next location. In this case, after using **Save Location to List** to create the block and store it in the NC Block List, make sure that the newly added block is highlighted, and then press **Play Selected NC Blocks** to process the NC block and update the control with the machine component's current location.

In **Move to Pick Location** mode, the machine moves to the pick location (at the mouse click) and this location is processed through the control, and all other possible translations, before actual machine axes are defined and positioned.

NC Blocks List — the list contains a sequence of NC blocks that have been created. The list enables you to select a block, or sequence of blocks, in order to delete, copy and paste, or execute the selected block(s). A right mouse click displays a pop-up menu with features that enable you to edit, or execute, blocks in the list. Edit the content of a block by double clicking on the block to put it in "edit" mode, then make the necessary changes.

Shortcut: Right-click in the NC Block List to display the following menu:



NOTE: **Copy**, **Paste**, **Delete** and **Clear All** are described in the Edit Menu section above. **Play Selected NC Blocks** and **Play All NC Blocks** are described below.

Manipulating blocks in the in the NC Block List:

3. Use the "**Shift**" key to select a consecutive range of blocks. Click on the first block in the range so that it becomes highlighted, and then while holding down the "**Shift**" key, click on the last block in the range. All of the blocks between the first and last should now be highlighted.
4. Use the "**Ctrl**" key to select multiple individual blocks. Click on the first block so that it becomes highlighted, and then while holding down the "**Ctrl**" key, select additional blocks. Each block will become highlighted when it is selected.
5. Re-order blocks in the NC Block List by left-clicking on the square button at the beginning of the row and drag the row (while keeping the left mouse button depressed) to the desired position.

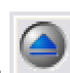
NC Block Entry — is a text buffer where an internally generated NC block is displayed, or an NC block can manually be entered. Use the **Add to List** button to add the block displayed in the **NC Block Entry** text field to the NC Block List.

Add to List — Use to add the contents of the **NC Block Entry** text field to the NC Block List.

Play Selected NC Blocks — Use this button to execute a single NC block, or a selected sequence of blocks, in NC Block List.

Play All NC Blocks — Use this button to execute all blocks in the NC Block List.

NOTES:

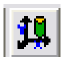
1. Use of **Machine Motion Jog** and **Play All NC Blocks** require a Machine Simulation license.
2. Do not attempt to use MDI before machine construction is complete. If the Stock, Tool, and all motion components are not present in the machine definition, the machine may not respond to motion commands as expected.
3. After adding motion type components to a machine, always press  (**Reset Model**) before using MDI to test machine movements.
4. Calls to subroutines and NC macros ***are not*** supported while in **Move to Pick Location** mode. Use a short test tool path file instead. Calls to subroutines and NC macros ***are*** supported while in **Machine Motion Jog** mode.
5. See **Manually Moving the Machine via MDI**, in the *Using VERICUT* section, in the *CGTech Help Library* for more information on using the MDI window features.

Also see "**Manually Moving the Machine via MDI**", in the *Using VERICUT* section, in the *CGTech Help Library*.

Coordinate Systems

Define (Coordinate System window)

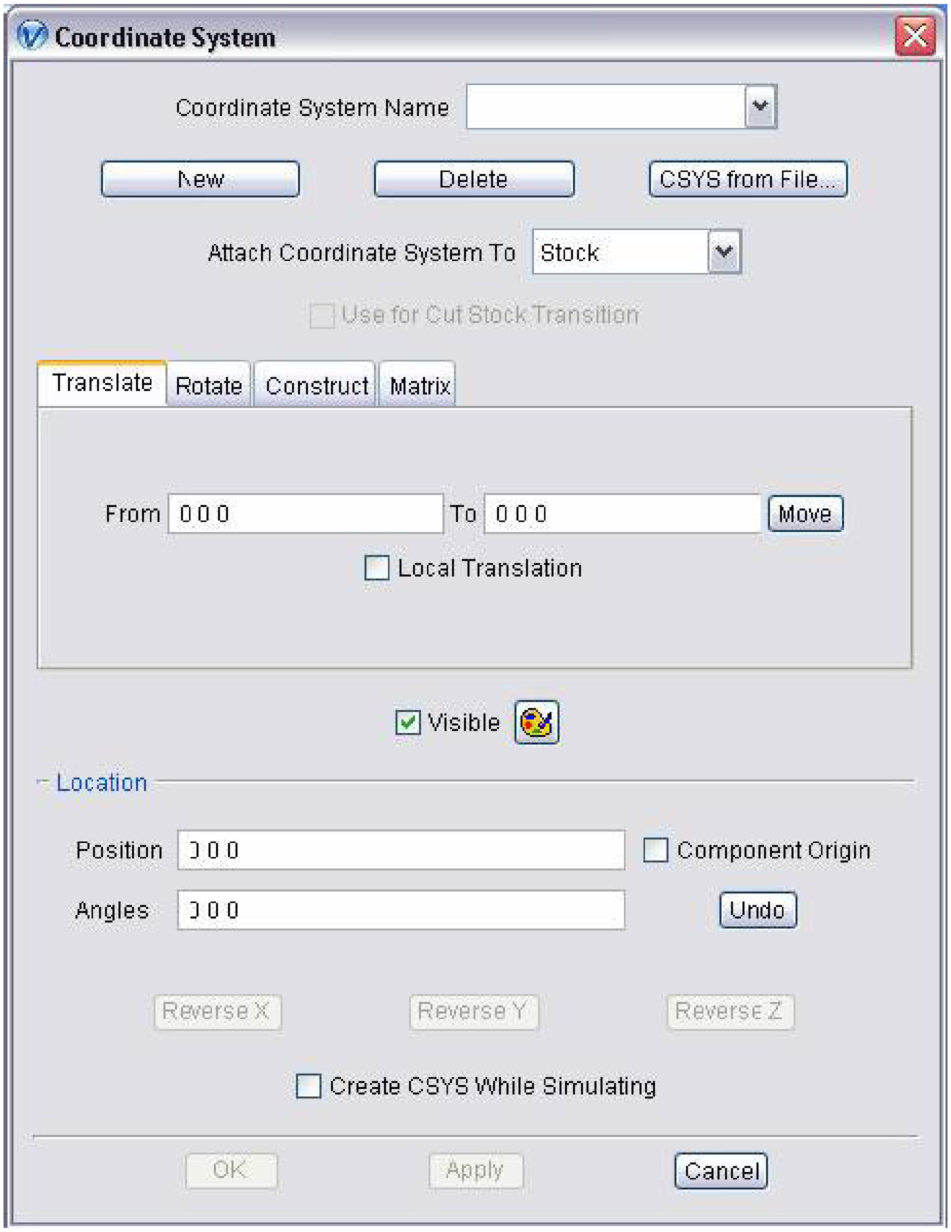
Location: **Project menu > Coordinate Systems > Define**

Toolbar shortcut: 

Opens a window enabling you to define a coordinate system, also known as a "CSYS". The active CSYS applies to X-caliper measurements, Section plane values, and tool path motions (except when orientations are defined in a tool path list). To see axes representing the active CSYS (XcsYcsZcs), select **View menu > View Axes: Coordinate System**. A user-defined CSYS is typically used to locate an APT or CLS tool path for proper relationship to the workpiece. You can also use a user-defined CSYS for defining section planes, or for gathering measurement data. Use **Project menu > Coordinate Systems > Active Coordinate System** to activate the desired coordinate system.

NOTES:

1. Defining a CSYS is not recommended to locate G-Code tool paths simulated on a 3-D machine, as this can cause the workpiece to move in the machine view. Instead, use an **Input Program Zero table**, or **Work Offsets table** if work offsets are present in the tool path file.
2. Coordinate systems specified under **NC Program Origin** on the **NC Program window**, override the active user-defined CSYS.
3. VERICUT can also be configured to process the following APT-CLS tool path records to set the tool path CSYS: **CATIA0**, **MSYS**, **VERICUT-MATRIX**.



Coordinate System Name— Enables you to add a new CSYS or select an existing CSYS from the list for modification or deletion. Enter a name in the top position of the **Coordinate System Name** list to create a new CSYS, or use **New** (described below) to create a new CSYS and let VERICUT automatically supply a unique name.

New — Adds a new CSYS to the **Coordinate System Name** list. VERICUT automatically provides a unique name to the CSYS. The following user actions also automatically add a new CSYS:

- **Translate tab**: pressing **Move**
- **Rotate tab**: pressing one of the rotation direction buttons (**X+/X-**, **Y+/Y-**, **Z+/Z-**)

- **Construct tab**: pressing **Update**
- When VERICUT processes one of the following APT-CLS tool path records to set the tool path CSYS: **CATIA0**, **MSYS**, **VERICUT-MATRIX**. The coordinate system is relative to the first stock's machine component parent. The coordinate system is named "*filename_nn*" where *filename* is the name of the toolpath file and **nn** is the file's line number containing the matrix definition. Additional coordinate systems are not created for matrices with identical values.

Delete — Deletes the selected CSYS from the **Coordinate System Name** list.

CSYS from File — Displays a file selection box enabling you to specify the CSYS file to be used. VERICUT will read the file and creates CSYS's attached to the selected component (typically the Stock). The **Create CSYS While Simulating feature** (described below) must be toggled **Off** when using **CSYS from File**. This functionality is also available via the csys_file command line option.

Attach Coordinate System To — Enables you to attach a CSYS to a particular component so that if the component gets repositioned, the CSYS also gets repositioned, maintaining the defined relationship. Select the component from the pull-down list.

Use for Cut Stock Transition — Use to designate that the coordinate system is used for transitioning the cut stock(s) from one setup position to another. Stock transition coordinate systems can be created for each **Stock** component.

CSYS Construct/Modify features:

The features on the following tabs enable you construct a new, or modify an existing CSYS.

Translate tab **—** Features on this tab enable you to translate the selected CSYS via indicating "from" and "to" points to move the CSYS. Movement occurs each time you press the **Move** button. If the applied rotation is incorrect, press **Undo** to return the object to its previous location.

Rotate tab **—** Features on this tab enable you to rotate the selected CSYS about a rotation center point. Movement occurs each time you press one of the rotation direction buttons: **X+/X-**, **Y+/Y-**, **Z+/Z-**. If the applied rotation is incorrect, press **Undo** to return the object to its previous location.

Construct tab **—** Features on this tab enable you to construct a CSYS via defining 3 points. Each point can be defined using points, planes, vectors, or the center of a hole or boss.

Matrix tab **—** Features on this tab move the selected CSYS via a twelve parameter transformation matrix.

Display features:

These features enable you to override the "global" visibility and color characteristics, as set with **View menu > Axes: Coordinate System**, for the individual coordinate system currently displayed in the **Coordinate System Name** list.

Visible — When coordinate systems are displayed using **View menu > View Axes: Coordinate System, Visible** enables you to toggle the display of an individual coordinate system "Off/On".

 — Enables you to override the "global" coordinate system display color set using **View menu > View Axes: Coordinate System**, for an individual coordinate system. Click on the **Color Pallet** icon, then select the desired color from the chart.

Location features:

These features show the selected CSYS's position and angle, and can be used to move the CSYS or verify its current location. Values shown are relative to the workpiece origin.

Position — Specifies the absolute XYZ position of the CSYS, separated by spaces.

Component Origin — Enables you to position the origin of the coordinate system currently displayed in the **Coordinate System Name** list at the origin of the selected component. Toggle **Component Origin** "**On**", click in the **Position** field so that it becomes highlighted yellow, then select the desired component in the graphics area. Press **Apply** to move the coordinate system.

Angles — Specifies the absolute XYZ rotation of the CSYS, separated by spaces.

Undo — Returns the CSYS to its previous location, or as it was when the Coordinate System window was opened.

Reverse X — Reverses the X axis of the "active" coordinate system. (See note below)

Reverse Y — Reverses the Y axis of the "active" coordinate system. (See note below)

Reverse Z — Reverses the Z axis of the "active" coordinate system. (See note below)

NOTE: In order to maintain a right handed csys, the following will occur when an axis is reversed:

If the X axis is reversed, then the Z axis is also reversed.

If the Y axis is reversed, then the X axis is also reversed.

If the Z axis is reversed, then the Y axis is also reversed.

Create CSYS While Simulating — When toggled On (the default), coordinate systems are unconditionally created when a matrix statement is encountered in the NC program being simulated. The coordinate systems are attached to the stock's parent (the "attach" component). This feature must be toggled Off when creating them on your own or using the CSYS from File feature (described above).

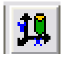
OK — Applies the changes and closes the Coordinate System window.

Apply — Applies the changes and leaves the Coordinate System window open.

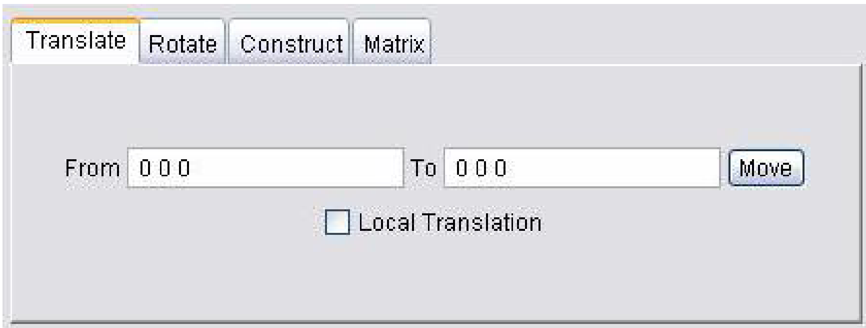
Cancel — Closes the Coordinate System window without applying changes.

Coordinate System window, Translate tab

Location: **Project menu > Coord. System > Define**

Toolbar shortcut: 

Features on this tab enable you to translate the selected CSYS via indicating "from" and "to" points to move the CSYS. Movement occurs each time you press the **Move** button.



From / To — Specifies the locations to move the CSYS from and to, relative to the workpiece origin. XYZ values can be entered (separated by spaces), or selected by clicking in the field then clicking on a model. As you move the mouse over the VERICUT model, a crosshair and vector show you the pending pick-point location. Graphical selection supports picking corner points and midpoints of uncut model geometry, or virtually any point on machined features.

Move — Moves the selected CSYS by the incremental distance, as calculated from the "**From**" point to the "**To**" point location.

Local Translation — When toggled "on", From/To values are relative to the current CSYS.

NOTE: This feature is useful when trying to align a CSYS to the center of a machined hole.

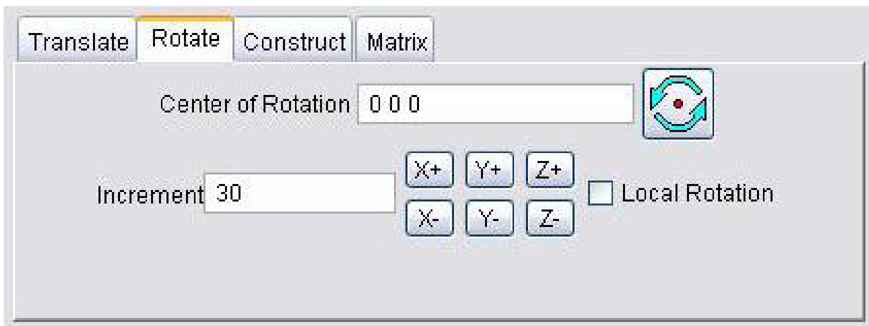
To Coordinate System window

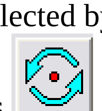
Coordinate System window, Rotate tab

Location: **Project menu > Coord. System > Define**

Toolbar shortcut: 

Features on this tab rotate the selected CSYS about a rotation center point. Movement occurs each time you press one of the rotation direction buttons: **X+/X-**, **Y+/Y-**, **Z+/Z-**.



Center of Rotation — Specifies XYZ point location about which to rotate the CSYS. XYZ values can be entered (separated by spaces), or selected by clicking in the field then clicking on a model. To see the center of rotation, press . To remove the center of rotation symbol press the button again, or close the Coordinate System window.

Increment — Specifies incremental degrees of rotation to apply when one of the rotation direction buttons are pressed.

Rotation direction buttons — (**X+/X-**, **Y+/Y-**, **Z+/Z-**) When pressed, applies the incremental rotation specified in the **Increment** field. Rotation occurs about the **Center of Rotation**, relative to the workpiece origin.

Local Rotation — When toggled "on", the incremental rotation values are relative to the current CSYS.

NOTE: This feature is useful when trying to align a CSYS to the center of a machined hole.

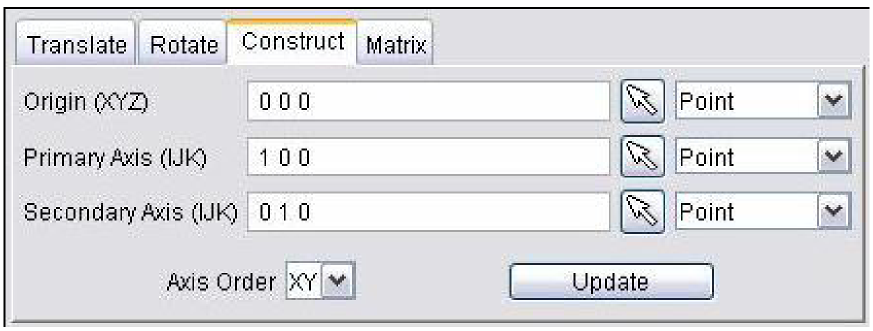
[To Coordinate System window](#)

Coordinate System window, Construct tab

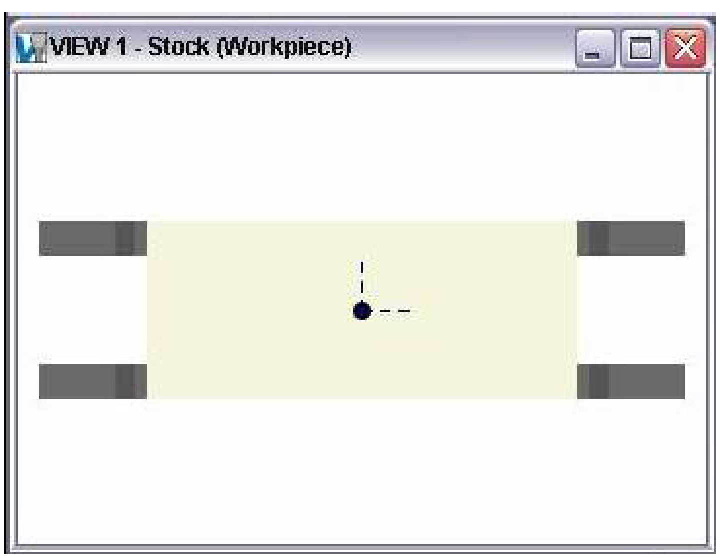
Location: **Project menu > Coord. System > Define**

Toolbar shortcut: 

The features on this tab enable you to construct a CSYS via a point representing the CSYS origin, and two direction vectors representing the axes of the CSYS. Each can be defined using points, planes and vectors.



While the Construct tab is active, a marker is displayed in the graphics area (see the picture below) showing the current position/orientation of the axis being constructed based on the Construct tab settings. The display is updated each time you change a value on the Construct tab. The marker is automatically clears when you leave the Construct tab.



Origin (XYZ) — Use these features to define a point representing the origin of the CSYS.

Primary Axis (IJK) — Use these features to define a point or vector representing the direction of the CSYS primary axis.

Secondary Axis (IJK) — Use these features to define a point or vector representing the direction of the CSYS secondary axis.

For each of the items above, you can either enter the coordinates of the point, separated by spaces, in the text field or select one of the construction methods and pick geometry in the graphics area to define the item. Choose one of the construction methods from the pull-down list, then click the arrow  and follow the prompts in the message area to define the item. The following construction methods are available:

Point — Use to select a point.

Vector/Plane — Define a point represented by the intersection of a vector with a plane.

3 Planes — Define a point represented by the intersection of three planes.

Circle — Define a point represented by the center of a circle.

CSYS Origin — Define a point represented by the origin of an existing coordinate system.

NOTE: When using one of the point construction methods to define the Primary or Secondary Axis, the origin is subtracted from the generated point to define the direction vector representing the axis.

Vector — Define the direction of a CSYS axis by selecting a vector. (not available for Origin)

Plane/Plane — Define the direction of a CSYS axis by a vector represented by the intersection of two planes. (***not*** available for **Origin**)

Axis Order — Choose from the pull-down list to assign X, Y or Z to the Primary and Secondary axes of the CSYS.

XY — assigns the Primary Axis to be X and the Secondary Axis to be Y.

YZ — assigns the Primary Axis to be Y and the Secondary Axis to be Z.

ZX — assigns the Primary Axis to be Z and the Secondary Axis to be X.

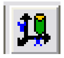
Update — Use the values selected above to update an existing CSYS, or create a new CSYS. When the **Update** button is pressed the following occurs:

1. The Primary Axis and the Secondary Axis are crossed to determine the third axis.
2. The third axis is then crossed with the Primary Axis to ensure that the Secondary Axis is orthogonal.
3. The CSYS currently active in the Coordinate System window is updated with the resultant orthogonal coordinate system and the origin. If no CSYS is currently active, VERICUT creates a new CSYS and automatically assigns it a unique name.

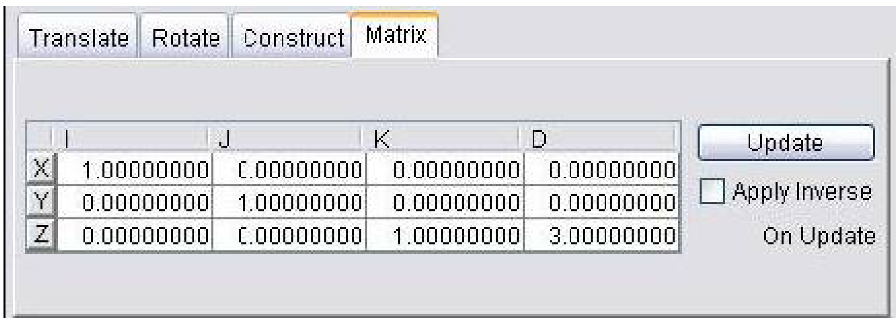
To Coordinate System window

Coordinate System window, Matrix tab

Location: **Project menu > Coord. System > Define**

Toolbar shortcut: 

Features on this tab enable you to move the selected CSYS via a twelve parameter transformation matrix.



Matrix Table — The transformation matrix table is similar to the matrix used in programming APT tool paths. Its twelve parameters reveal the geometrical attributes of the local (transformed) coordinate system (CSYS) in terms of the workpiece origin.

The format of the matrix table is as follows:

	I	**J**	**K**	**D**
X	I1	J1	K1	D1
Y	I2	J2	K2	D2
Z	I3	J3	K3	D3

Each row represents an axis of the local CSYS. The first three columns represent the vector associated with each axis: I1, J1, K1 as the positive X-axis vector; I2, J2,K2 as the positive Y-axis vector; and I3, J3, K3 as the positive Z-axis vector. The fourth column values D1, D2, D3 represent the coordinates of the origin point of the local CSYS.

NOTE: If you prefer to see the Matrix Table displayed with the I, J, K along the vertical axis and the X, Y, Z along the horizontal axis, set the environment variable, **CGTECH_MATRIX_FORMAT**=**VERTICAL**.

Update — Updates the CSYS location to reflect the matrix table transformation. After updating, press OK or Apply to move the object.

Apply Inverse On Update — When selected, inverts the matrix so that its twelve parameters reveal the geometrical attributes of the workpiece origin in terms of the local (transformed) coordinate system.

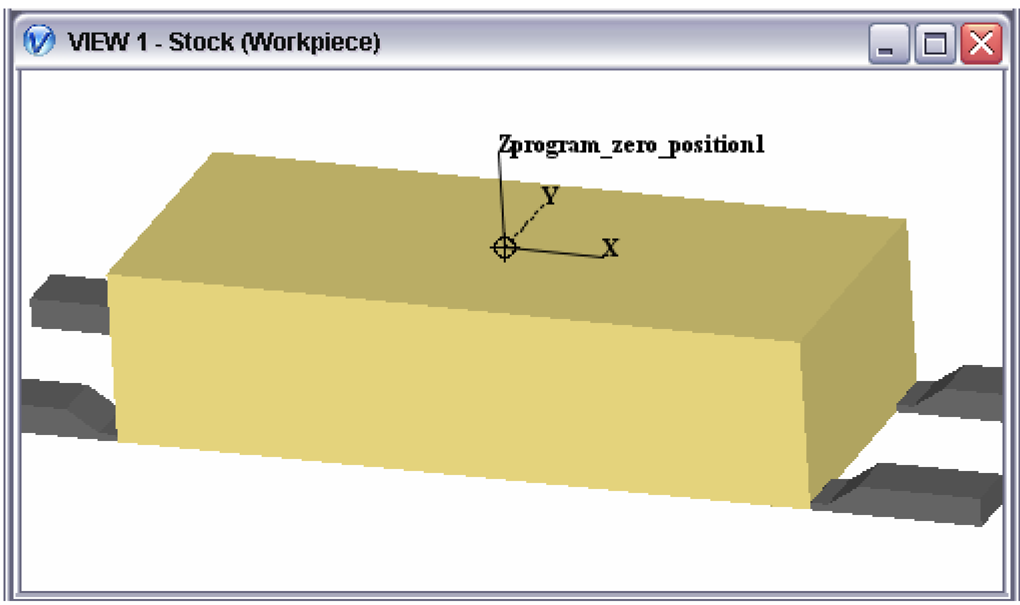
To Coordinate System window

Set Active Coordinate System

Location: **Project menu > Coordinate Systems > Set Active Coord. Sys.**

You control the "active" coordinate system, or "CSYS", that VERICUT uses for simulating tool path motions, X-Caliper measurements, and sectioning models. A user-defined coordinate system is typically used to locate an APT or CLS tool path in the proper orientation to the workpiece.

To make an existing CSYS the "active" CSYS, use **Project menu > Coordinate Systems > Set Active Coord. Sys.** and then select the desired CSYS from the list of available coordinate systems. If **Display Active Coord. Sys.**, in the View Axes window, is toggled "On", a coordinate system with the name of the active coordinate system will display in the VERICUT Graphics area as shown below. Only the "active" coordinate system will display with the marker at its origin as shown.

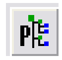


Shortcut: You can also change the active coordinate system by right-clicking in the view, and selecting **Set Active Coord. Sys.** and selecting the desired coordinate system from the displayed list. See **Graphics Area Right Mouse Button Shortcut Menus**, in the Getting Started with VERICUT section of VERICUT Help for additional information.

Use **Project menu > Coordinate Systems > Define...** to create additional coordinate systems.

Project Tree

Location: **Project menu > Project Tree**

VERICUT toolbar short cut: 

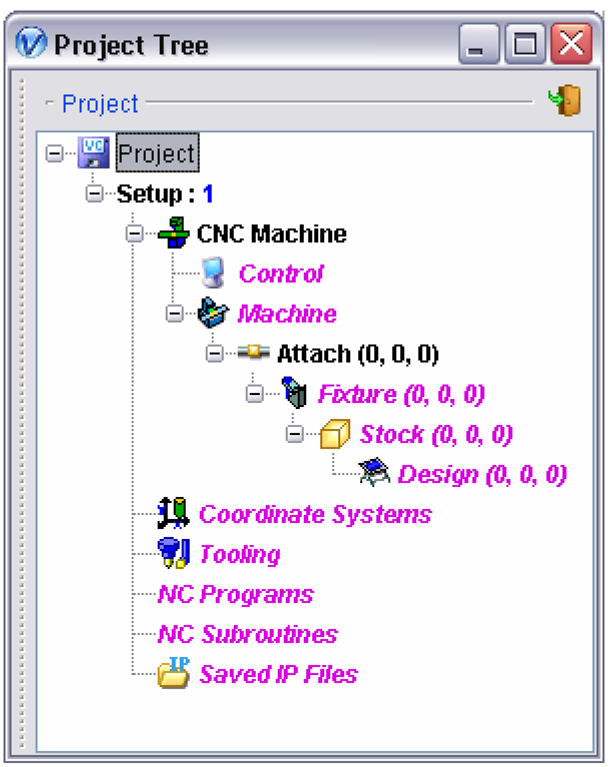
The Project Tree is one of the dockable windows enabling you to dock it inside the VERICUT main window if you choose. See **Dockable Windows** in the *Getting Started with VERICUT* section of *VERICUT Help* for additional information.

NOTE: When the **Project Tree** window is docked, make sure that you click in the window so that it becomes the "active" window before using F1 to get help specific to the window. Otherwise F1 will go to the CGTech Help Library.

The Project Tree shows all Setups and each Setup's configuration in a tree hierarchy. A project can consist of one or more setups. VERICUT processes each "active" setup sequentially, starting at the top of the tree. The functionality available in the Project Tree duplicates the functionality available in the **Project menu**, but in a more visual tree format.

The following examples describe some of the information available to you from the way that the items in the Project Tree are displayed.

Project Tree Example 1: "New" project with a single setup.

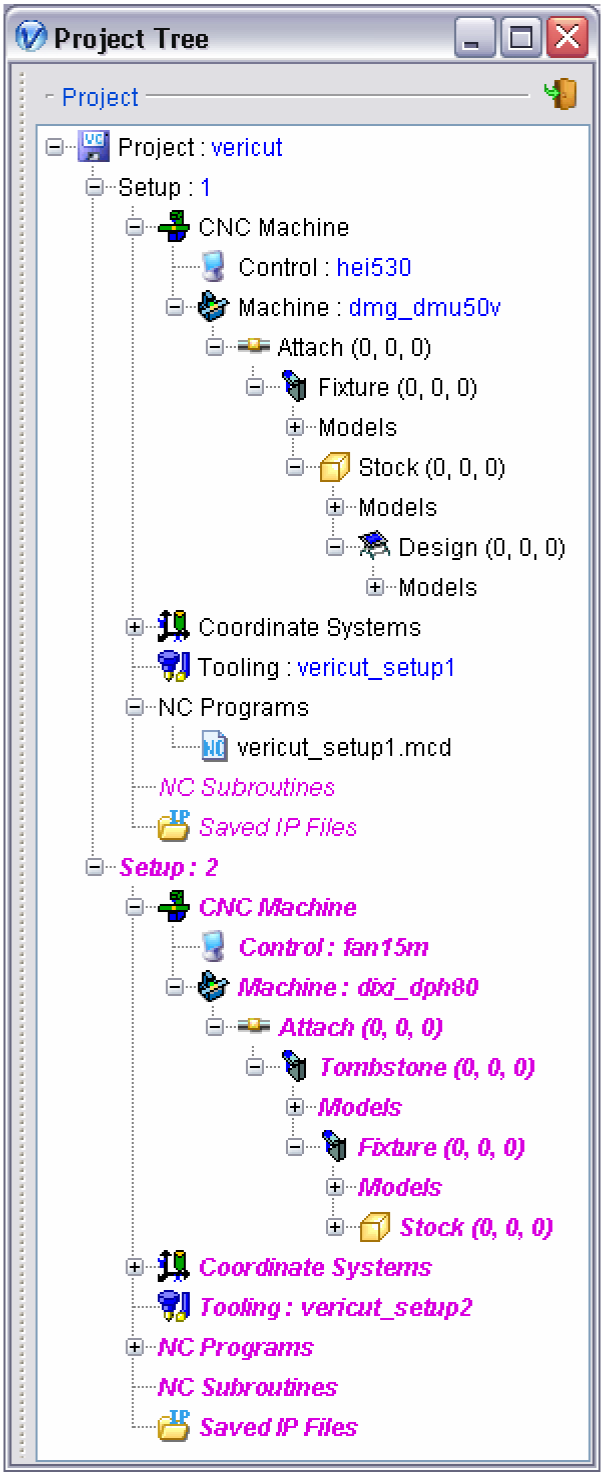


The picture at the left shows the Project Tree for a "new" project containing a single setup. Since it is the only setup, it is by default the "current" setup, indicated by the bold text.

The red text here indicates that no data has been assigned to any of these items.

The initial machine is defined with Stock, Design and Fixture components but no models have been assigned

Project Tree Example 2: A project with multiple setups.



The picture at the left shows the Project Tree for a two setup job.

Both setups are fully defined.

Setup: 1 is displayed in black text indicating that it is in an "active" state. The bold text indicated that Setup 1 is the "current" setup.

The NC Subroutines label in Setup: 1 is displayed in red text indicating that no subroutines are used for Setup 1.

All of the items in Setup: 2 are displayed in red text indicating that Setup 2 is in an "inactive" state.

Each item in the Project Tree has a Right Mouse Button Shortcut menu containing features specific to the particular item.

- Project Branch
- Setup Branch
- CNC Machine Branch
- Control Branch
- Machine Branch
- Attach Component
- Component Branch
- Model
- Coordinate Systems Branch
- Coordinate System
- Tooling
- NC Program branch
- NC Program file
- NC Subroutines branch
- NC Subroutine file
- Saved IP Files branch
- IP file

For more information on these, and all, VERICUT Right Mouse Button Shortcut menus, see **Right Mouse Button Shortcut menus** in the **Getting Started with VERICUT** section of *VERICUT Help*.

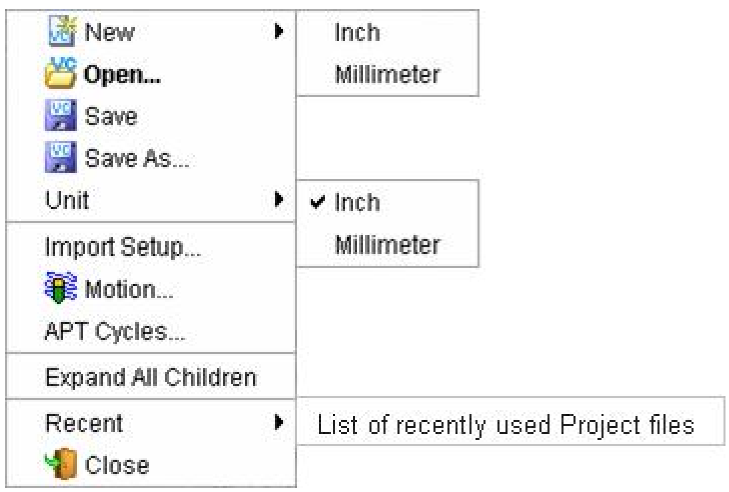
Project Tree Right Mouse Button Shortcut Menus

Each item in the Project Tree has a right mouse button shortcut menu containing features specific to the particular item.

NOTE: Any item shown in bold a Project Tree right mouse button shortcut menu can also be reached directly by double clicking on the item (branch, component, model, etc.) in the Project Tree that you right -clicked on to display the shortcut menu. For example, double-clicking on the Project branch in the Project Tree displays the **Open Project window**, just like clicking on **Open** in the shortcut menu shown in below.

Project Branch

Right-click on the **Project branch** in the Project Tree window to display the following menu:



New — Use to create a new project file. (ref. **New Project**, in the File Menu section of *VERICUT Help*)

Open — Displays the Open Project file selection window enabling you to open an existing project file.

Save — Use to save an existing project file with the current VERICUT session settings.

Save As — Opens the Save Project as window, enabling you to save the current project file under a different name, or in a different location.

Unit — Enables you to change the units of the current project file. A check indicates the "current" units.

Import Setup — Opens the Setup Import window, enabling you to copy a setup from either a pre-V6.0 user file, or from a V6.x project file, and append it after the current setup. (ref. **Import Setup**, in the Project Menu section of *VERICUT Help*.

Motion — Opens the Motion window. (ref. **Motion window**, in the Project Menu section of *VERICUT Help*.

APT Cycles — Opens the Cycles window enabling you to maintain cycle and modal definitions. (ref. **Cycle Definitions**, in the APT Settings window: Cycle tab section of *VERICUT Help*)

Expand all Children — Expands all branches of the Project Tree.

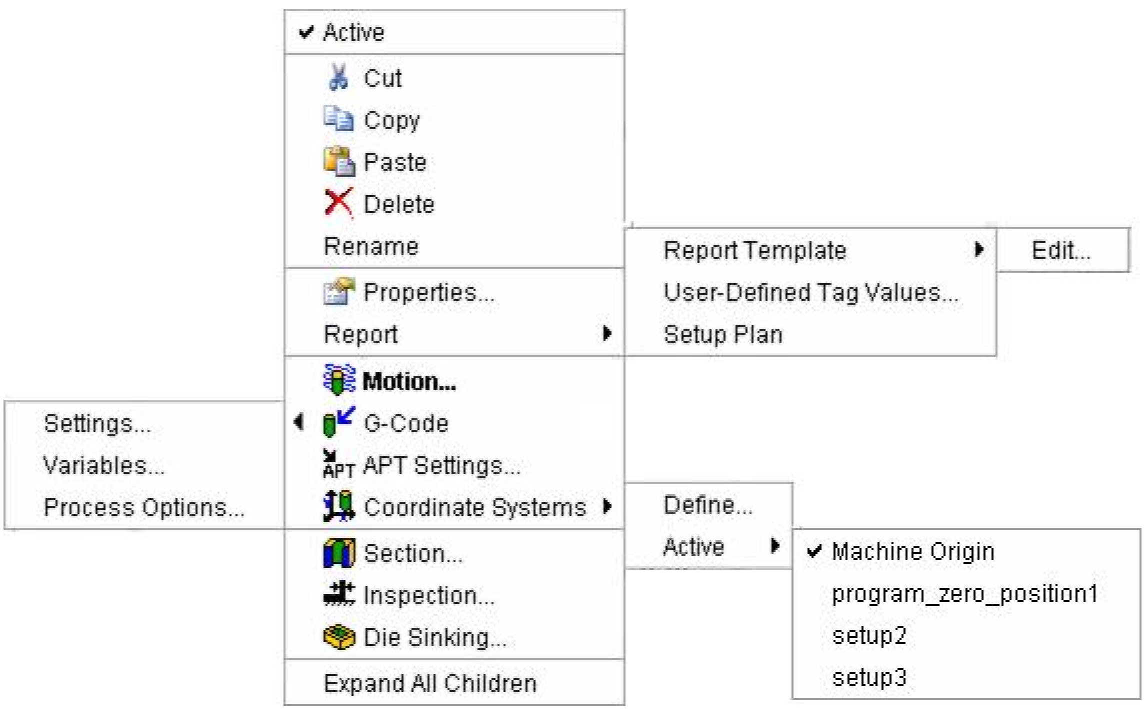
Recent — Displays a list of recently used project files that you can select from.

Close — Closes the Project Tree window.

Shortcut: Double clicking on the Project Branch will open the Open Project file selection window.

Setup Branch

Right-click on a **Setup branch** in the Project Tree window to display the following menu:



Active — Use to designate whether the setup is "active", or "inactive". (ref. **Introduction to the Project Setup menu**, in the Project Menu section of *VERICUT Help*.

Cut — Cuts the highlighted setup in the Project Tree and puts it in the paste buffer.

Copy — Copies the highlighted setup in the Project Tree to the paste buffer.

Paste — Puts the contents of the paste buffer after the highlighted setup in the Project Tree.

Delete — Deletes the highlighted setup from the Project Tree.

Rename — Use to rename the highlighted setup.

Properties — Opens the Properties window. (ref. **Properties window**, in the File Menu section of *VERICUT Help*)

Report >

> **Report Template > Edit —** Opens the Report Template window. (ref. **Report Template window**, in the Project Menu section of *VERICUT Help*.
>
> **User-Defined Tag Values —** Opens the User-Defined Tag Values window. (ref. **User-Defined Tag Values window**, in the Project Menu section of *VERICUT Help*.
>
> **Setup Plan —** Opens the Setup Plan window. (ref. **Setup Plan window**, in the Project Menu section of *VERICUT Help*.

Motion — Opens the Motion window. (ref. **Motion window**, in the Project Menu section of *VERICUT Help*.

G-Code >

> **Settings —** Opens the G-Code Settings window. (ref. **G-Code Settings window**, in the Project Menu section of *VERICUT Help*.
>
> **Variables —** Opens the Variables window. (ref. **Variables window**, in the Project Menu section of *VERICUT Help*.
>
> **Process Options —** Opens the Process Options window. (ref. **Process Options window**, in the Project Menu section of *VERICUT Help*.

APT Settings — Opens the APT Settings window. (ref. **APT Settings window**, in the Project Menu section of *VERICUT Help*)

Coordinate Systems >

> **Define —** Opens the Coordinate System window. (ref. **Coordinate Systems window**, in the Project Menu section of *VERICUT Help*)
>
> **Active —** Displays a list of all available coordinate systems that you can choose from. A check indicates the active coordinate system.

Section — Opens the Section window. (ref. **View Section window**, in the View Menu section of *VERICUT Help*)

Inspection — Opens the Inspection window. (ref. **VERICUT Inspection**, in the Analysis Menu section of *VERICUT Help*)

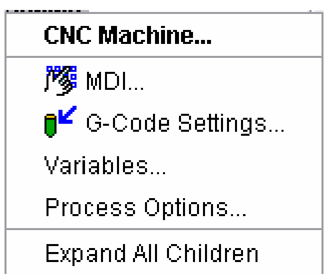
Die Sinking — Opens the Die Sinking Simulation window. (ref. **Die Sinking Simulation window**, in the Analysis Menu section of *VERICUT Help*)

Expand all Children — Expands all branches of a Setup branch.

Shortcut: Double clicking on the Project Branch will open the Motion window.

CNC Machine Branch

Right-click on a **CNC Machine branch** in the Project Tree window to display the following menu:



CNC Machine — Opens the CNC Machine window. (ref. **CNC Machine window**, in the Project menu section of *VERICUT Help*)

MDI — Opens the MDI window. (ref. to **MDI window**, in the Project menu section of *VERICUT Help*)

G-Code Settings — Opens the G-Code Settings window. (ref. **G-Code Settings window**, in the Project Menu section of *VERICUT Help*.

Variables — Opens the Variables window. (ref. **Variables window**, in the Project Menu section of *VERICUT Help*.

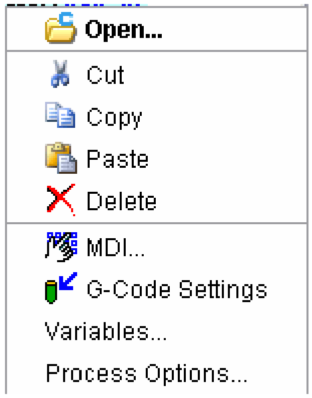
Process Options — Opens the Process Options window. (ref. **Process Options window**, in the Project Menu section of *VERICUT Help*.

Expand All Children — Expands all branches of a CNC Machine branch.

Shortcut: Double clicking on a CNC Machine Branch will open CNC Machine window.

Control Branch

Right-click on a **Control branch** in the Project Tree window to display the following menu:



Open — Opens the Open Control file selection window enabling you to open a control file. (ref. **Open (Control file)**, in the Configuration Menu section of *VERICUT Help*.

Cut — Cuts the highlighted control file from the Project Tree and puts it in the paste buffer.

Copy — Copies the highlighted control file from the Project Tree to the paste buffer.

Paste — Uses the contents of the paste buffer to replace the highlighted control file in the Project Tree.

Delete — Deletes the highlighted control file from the Project Tree.

MDI — Opens the MDI window. (ref. to **MDI window**, in the Project menu section of *VERICUT Help*)

G-Code Settings — Opens the G-Code Settings window. (ref. **G-Code Settings window**, in the Project Menu section of *VERICUT Help*.

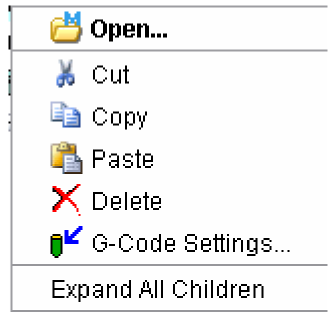
Variables — Opens the Variables window. (ref. **Variables window**, in the Project Menu section of *VERICUT Help*.

Process Options — Opens the Process Options window. (ref. **Process Options window**, in the Project Menu section of *VERICUT Help*.

Shortcut: Double clicking on a Control Branch will open the Open Control file selection window enabling you to open a control file.

Machine Branch

Right-click on a **Machine branch** in the Project Tree window to display the following menu:



Open — Opens the Open Machine file selection window enabling you to open a machine file. (ref. **Open (Machine file)**, in the Configuration Menu section of *VERICUT Help*.

Cut — Cuts the highlighted machine file from the Project Tree and puts it in the paste buffer.

Copy — Copies the highlighted machine file from the Project Tree to the paste buffer.

Paste — Uses the contents of the paste buffer to replace the highlighted machine file in the Project Tree.

Delete — Deletes the highlighted machine file from the Project Tree.

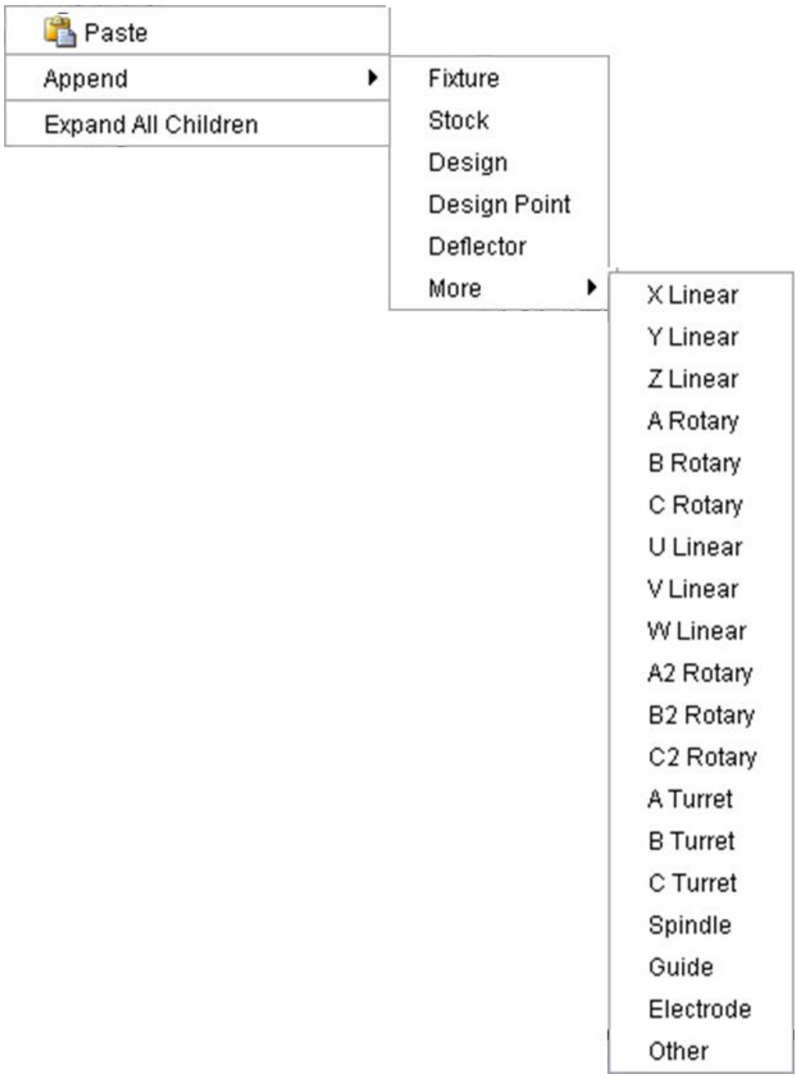
G-Code Settings — Opens the G-Code Settings window. (ref. **G-Code Settings window**, in the Project Menu section of *VERICUT Help*.

Expand All Children — Expands all branches of a Machine branch.

Shortcut: Double clicking on a Machine Branch will open the Open Machine file selection window enabling you to open a machine file.

Attach Component Branch

Right-click on an **Attach Component** branch in the Project Tree window to display the following menu:



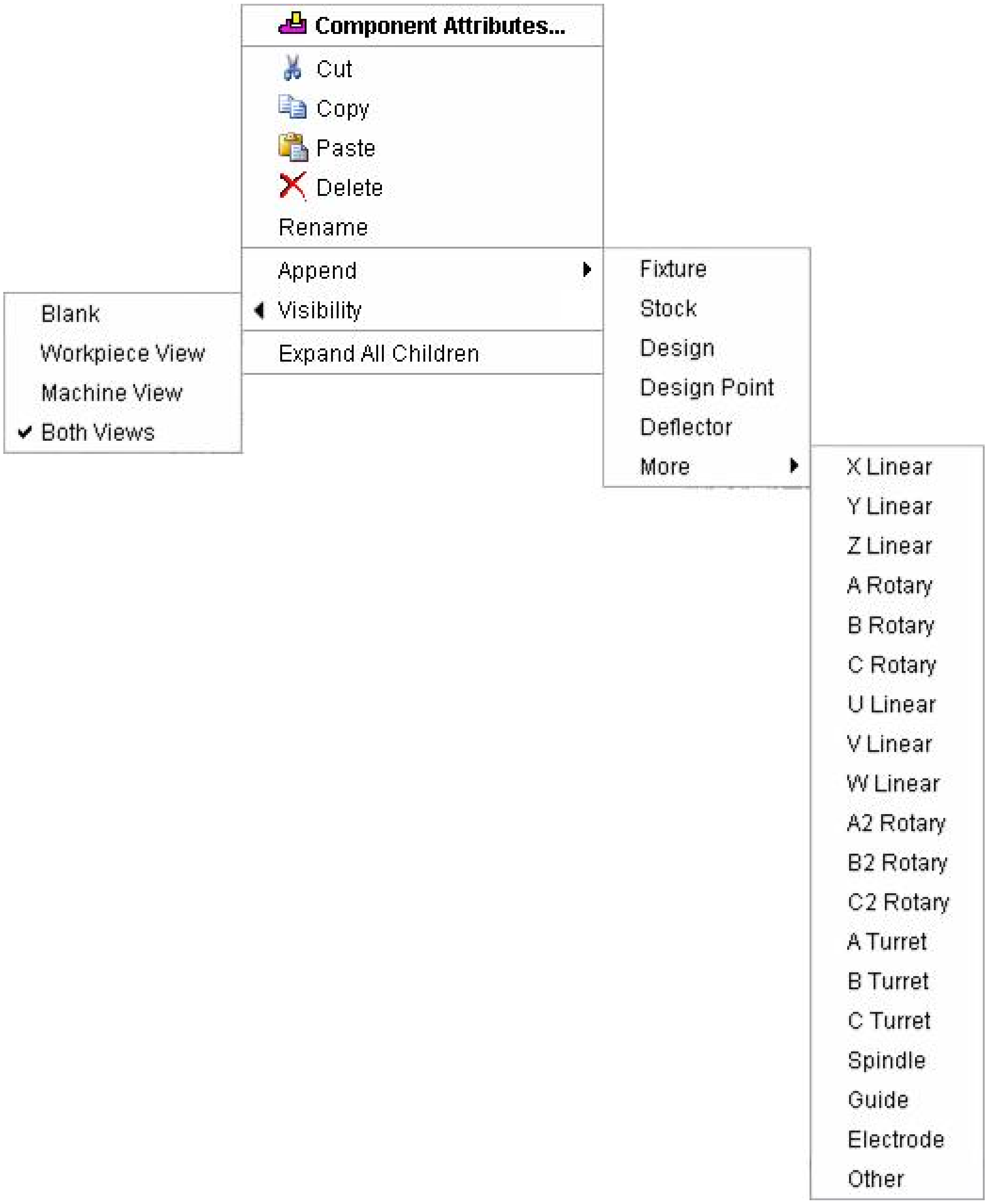
Paste — Use to paste the contents of the paste buffer to the Attach Component.

Append —Use to connect the selected component to the Attach Component. (ref. **Component Tree window: Component Menu**, in the Configuration Menu section of *VERICUT* Help)

Expand All Children — Expands all branches below the Attach Component.

Component Branch

Right-click on a **Component** branch (Fixture, Stock, Design) in the Project Tree window to display the following menu:



Component Attributes — Opens the Modeling window: Component Attributes tab. (ref. **Modeling window: Component Attributes tab** in the Project Menu section of *VERICUT Help*)

Cut — Cuts the highlighted component branch from the Project Tree and puts it in the paste buffer.

Copy — Copies the highlighted component branch from the Project Tree to the paste buffer.

Paste — Uses the contents of the paste buffer to replace the highlighted component branch in the Project Tree.

Delete — Deletes the highlighted component branch from the Project Tree.

Rename — Use to rename the highlighted component branch.

Append —Use to connect the selected component to the component branch. (ref. **Component Tree window: Component Menu**, in the Configuration Menu section of *VERICUT* Help)

Visibility — Use the Visibility features to specify whether or not a component branch is visible, and in what views. The check indicates the "current" visibility status.

Blank — The highlighted component branch is not visible in any view. When Visibility is set to "Blank" for a component, the icon representing the component in the Project Tree will be displayed in gray instead if a color.

Workpiece View — The highlighted component branch is only visible in a Workpiece View.

Machine View — The highlighted component branch is only visible in a Machine View (Machine, or Machine/Cut Stock).

Both — The highlighted component branch is visible in any view.

Expand All Children — Expands all branches of a Component branch.

Shortcut: Double clicking on a Component Branch will open the Modeling window: Component Attributes tab.

Models Branch

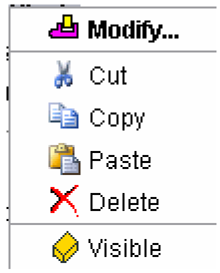
Right-click on a **Models Branch** in the Project Tree window to display the following feature:

Expand All Children — Use to paste the contents of the paste buffer to the Attach Component.

Shortcut: Double clicking on a Models Branch will open the Modeling window: Model tab. (ref. **Modeling window: Model tab** in the Project Menu section of *VERICUT Help*)

Model

Right-click on a **Model** in the Project Tree window to display the following menu:



Modify — Opens the Modeling window: Model tab. (ref. **Modeling window: Model tab** in the Project Menu section of *VERICUT Help*)

Cut — Cuts the highlighted model from the Project Tree and puts it in the paste buffer.

Copy — Copies the highlighted model from the Project Tree to the paste buffer.

Paste — Uses the contents of the paste buffer to replace the highlighted model in the project tree.

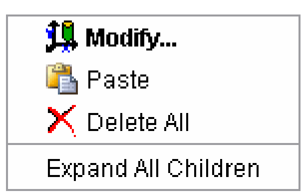
Delete — Deletes the highlighted model from the Project Tree.

Visible — Use to make the highlighted model visible, or not visible in the VERICUT graphics area. The Visible icon will indicate the visibility status of the model. indicates "visible", indicates "not visible". Click on Visible to toggle between the two modes. In the Project Tree, any model in the "not visible" state will be displayed in gray instead if a color.

Shortcut: Double clicking on a Model in the Project Tree window will open the Modeling window: Model tab.

Coordinate Systems Branch

Right-click on a **Coordinate Systems** branch in the Project Tree window to display the following menu:



Modify — Opens the Coordinate System window. (ref. **Coordinate System window** in the Project Menu section of *VERICUT* Help.

Paste — Use to paste the contents of the paste buffer to the Coordinate Systems branch.
Delete All — Deletes all coordinate systems from the Coordinate System branch.
Expand All Children — Expands the Coordinate System branch showing all coordinate systems available in the setup.

Shortcut: Double clicking on a Coordinate Systems Branch will open the Coordinate System window.

Coordinate System

Right-click on a **Coordinate System** in the Project Tree window to display the following menu:

Modify — Opens the Coordinate System window. (ref. **Coordinate System window** in the Project Menu section of *VERICUT* Help.

Cut — Cuts the highlighted coordinate system from the Project Tree and puts it in the paste buffer.

Copy — Copies the highlighted coordinate system from the Project Tree to the paste buffer.

Paste — Adds the contents of the paste buffer after the highlighted coordinate system in the Project Tree.

Delete — Deletes the highlighted model from the Project Tree.

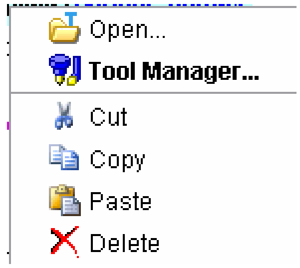
Active — Use to designate the "active"coordinate system. (ref. **Set Active Coordinate System**, in the Project Menu section of *VERICUT Help*.

Visible — Use to make the highlighted coordinate system visible, or not visible in the VERICUT graphics area. The Visible icon will indicate the visibility status of the coordinate system. indicates "visible", indicates "not visible". Click on Visible to toggle between the two modes. In the Project Tree, any coordinate system in the "not visible" state will be displayed in gray instead if a color.

Shortcut: Double clicking on a Coordinate System in the Project Tree window will open the Coordinate System window.

Tooling

Right-click on **Tooling** in the Project Tree window to display the following menu:



Open — Opens a file selection window enabling you to open a tool library file.

Tool Manager — Opens the Tool Manager window. (ref. **Tool Manager window**, in the Project Menu section of *VERICUT Help*)

Cut — Cuts the highlighted tool library from the Project Tree and puts it in the paste buffer.

Copy — Copies the highlighted tool library from the Project Tree to the paste buffer.

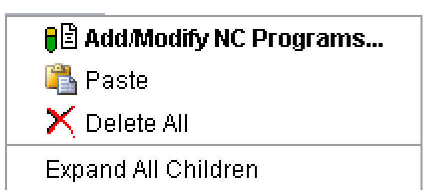
Paste — Replaces the highlighted tool library, with the tool library contained in the paste buffer.

Delete — Deletes the highlighted tool library from the Project Tree.

Shortcut: Double clicking on Tooling in the Project Tree window will open the Tool Manager window.

NC Program Branch

Right-click on an **NC Program branch** in the Project Tree window to display the following menu:



Add/Modify NC Programs — Opens the NC Program window. (ref. **NC Program window**, in the Project Menu section of *VERICUT Help*)

Paste — Adds the contents of the paste buffer to the highlighted NC Program branch in the Project Tree.

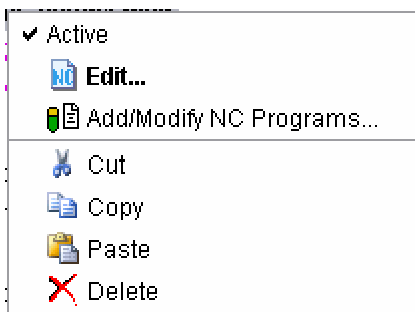
Delete All — Deletes all NC Programs from the NC Program branch.

Expand All Children — Expands the NC Program branch so that all of the NC Program files are shown.

Shortcut: Double clicking on an NC Program Branch in the Project Tree window will open the NC Program window.

NC Program File

Right-click on an **NC Program file** in the Project Tree window to display the following menu:



Active — Use to make the highlighted NC active, or inactive. A check in the menu indicates that the current status is "active". When an NC Program is "inactive" it is not processed. An "inactive" NC Program is indicated by red text in the Project Tree window. The **Active** feature provides the same functionality as the **Use** feature in the NC Program window. (ref. **NC Program window**, in the Project Menu section of *VERICUT Help*)

Edit — Opens the highlighted NC Program in the NC Program (edit) window. (ref. **NC Program (edit)**, in the Edit Menu section of *VERICUT Help*)

Add/Modify NC Programs — Opens the NC Program window. (ref. **NC Program window**, in the Project Menu section of *VERICUT Help*)

Cut — Cuts the highlighted NC Program file from the Project Tree and puts it in the paste buffer.

Copy — Copies the highlighted NC Program file from the Project Tree to the paste buffer.

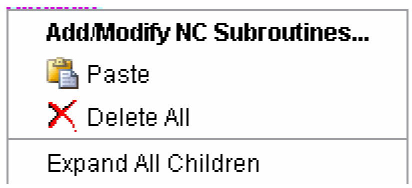
Paste — Replaces the highlighted NC Program file, with the NC Program file contained in the paste buffer.

Delete — Deletes the highlighted NC Program file from the Project Tree.

Shortcut: Double clicking on an NC Program file in the Project Tree window will opens the NC program in the NC Program Edit window.

NC Subroutine Branch

Right-click on an **NC Subroutines branch** in the Project Tree window to display the following menu:



Add/Modify NC Subroutines — Open the G-Code Settings window: Subroutines tab. (ref. **G-Code Settings window: Subroutines tab**, in the Project Menu section of *VERICUT Help*)

Paste — Adds the contents of the paste buffer to the highlighted NC Subroutines branch in the Project Tree.

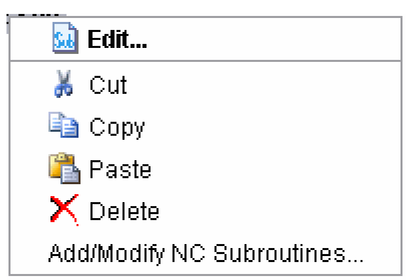
Delete All — Deletes all NC Subroutines from the NC Subroutines branch.

Expand All Children — Expands the NC Subroutines branch so that all of the NC Subroutine files are shown.

Shortcut: Double clicking on an NC Subroutines Branch in the Project Tree window will open the G-Code Settings window: Subroutines tab.

NC Subroutine File

Right-click on an **NC Subroutine file** in the Project Tree window to display the following menu:



Edit — Opens the highlighted NC Subroutine in the NC Program (edit) window. (ref. **NC Program (edit)**, in the Edit Menu section of *VERICUT Help*)

Cut — Cuts the highlighted NC Subroutine file from the Project Tree and puts it in the paste buffer.

Copy — Copies the highlighted NC Subroutines file from the Project Tree to the paste buffer.

Paste — Replaces the highlighted NC Subroutines file, with the NC Subroutines file contained in the paste buffer.

Delete — Deletes the highlighted NC Program file from the Project Tree.

Add/Modify NC Subroutines — Opens the G-Code Settings window: Subroutines tab. (ref. **G-Code Settings window: Subroutines tab**, in the Project Menu section of *VERICUT Help*)

Shortcut: Double clicking on an NC Subroutine file in the Project Tree window will open the highlighted NC Subroutine in the NC Program (edit) window.

Saved IP Files Branch

Right-click on a **Saved IP Files** branch in the Project Tree window to display the following menu:

Expand All Children — Expands the Saved IP Files branch so that all of the IP files are shown.

Refresh — Updates the status of IP files in the list. Enables you to see the status of IP files that have been manipulated (deleted, moved, renamed, etc.) outside of VERICUT. IP files in the list that cannot be found are displayed with red text.

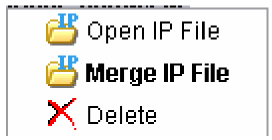
Delete All — Deletes all IP files from the Saved IP Files branch.

Save As — Opens the Save In-Process As window enabling you to save an IP file at the "current" state of processing.

Shortcut: Double clicking on a Saved IP Files branch in the Project Tree window to expand the Saved IP Files branch so that all of the IP files are shown.

IP File

Right-click on an **IP file** in the Project Tree window to display the following menu:



Open IP File — Opens (loads) the highlighted IP file.

Merge IP File — Merges the highlighted IP file into the current session. (ref. **Merge (In Process file)**, in the File Menu section of *VERICUT Help*)

Delete — Deletes the highlighted IP file from the Project Tree.

Shortcut: Double clicking on an IP file in the Project Tree window will open

Configuration Menu

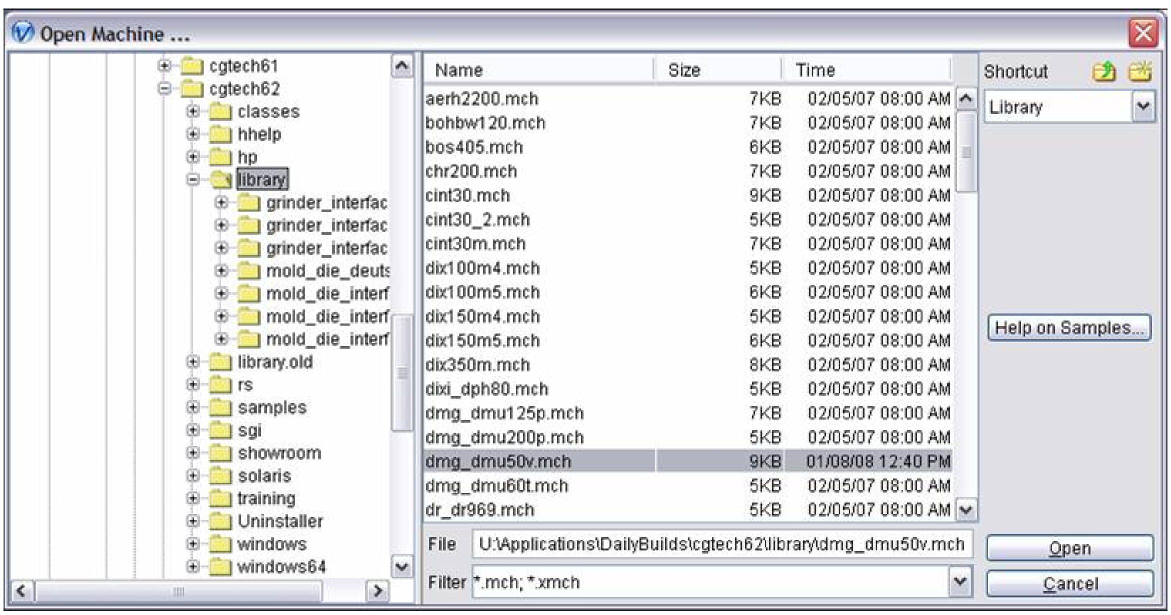
Machine

Open (Machine file)

Location: **Configuration menu > Machine > Open**

Toolbar short cut for opening Machine files: 

Opens the Open Machine window enabling you to open (load) a Machine file (ref. **Machine File** in the Getting Started with VERICUT section of *VERICUT Help*). Machine files contain data that describes NC machine construction, kinematics, and other properties.



Most features on this window are standard file selection window features that enable you to navigate through directories, filter files, and type, or select, /path/filenames. A description of features specific to VERICUT can be found in the **Introduction to VERICUT File Selection Windows** in the Getting Started section of *VERICUT Help*.

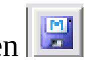
To save space, the features described below are unique to this window.

Help on Samples — Opens the *VERICUT Help* section on "**Machine files**". Both library and sample machines are listed. Find the file you want in the on-line Help, then cut & paste the file name into the file selection window. For machines located in the library, set **Shortcut=**CGTECH_LIBRARY and press **Open** to open the file. For "sample" machines, set **Shortcut=**CGTECH_SAMPLES and press **Open**.

Save (Machine file)

Location: **Configuration menu > Machine > Save**

Toolbar short cut for saving Machine files: 

TIP: Right-click on the icon to toggle between  (Save Machine) and  (Save Machine As) modes.

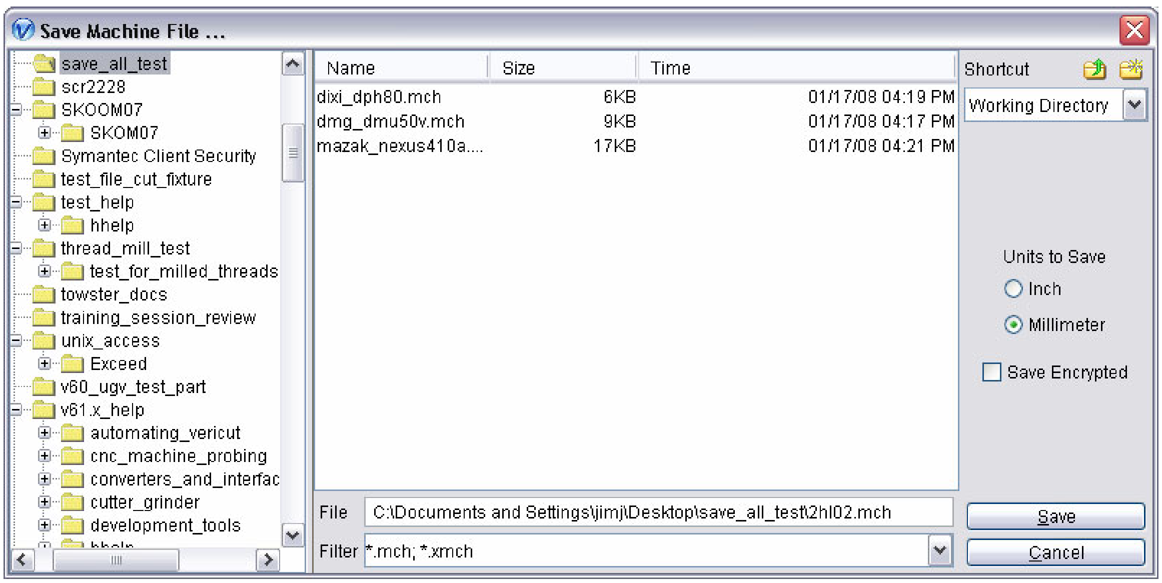
Saves (updates) an existing Machine file (ref. **Machine File** in the Getting Started with VERICUT section of *VERICUT Help*) with the current NC machine settings. VERICUT will save the machine file if you have sufficient permissions to save the file in its present directory. Otherwise, the Save Machine File window will display enabling you to specify a location to save where you have write permissions.

Save As (Machine file)

Location: **Configuration menu > Machine > Save As**

Toolbar short cut for saving Machine files:

TIP: Right-click on the icon to toggle between (Save Machine) and (Save Machine As) modes.



Opens the **Save Machine File** window enabling you to save a Machine file (ref. **Machine File** in the Getting Started with VERICUT section of *VERICUT Help*). Machine files contain data that describes NC machine construction, kinematics, and other properties.

Most features on this window are standard file selection window features that enable you to navigate through directories, filter files, and type, or select, /path/filenames. A description of features specific to VERICUT can be found in the **Introduction to VERICUT File Selection Windows** in the Getting Started section of *VERICUT Help*.

To save space, the features described below are unique to this window.

Units to Save — Controls the units (**Inch** or **Millimeter**) in which to store machine configuration data. If the session units are different than the units to save, then values are converted accordingly when stored in the new file.

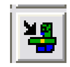
Save Encrypted — Saves the Machine file as a standard encrypted (X-File) with an .xmch extension.

A "standard" encrypted machine file:

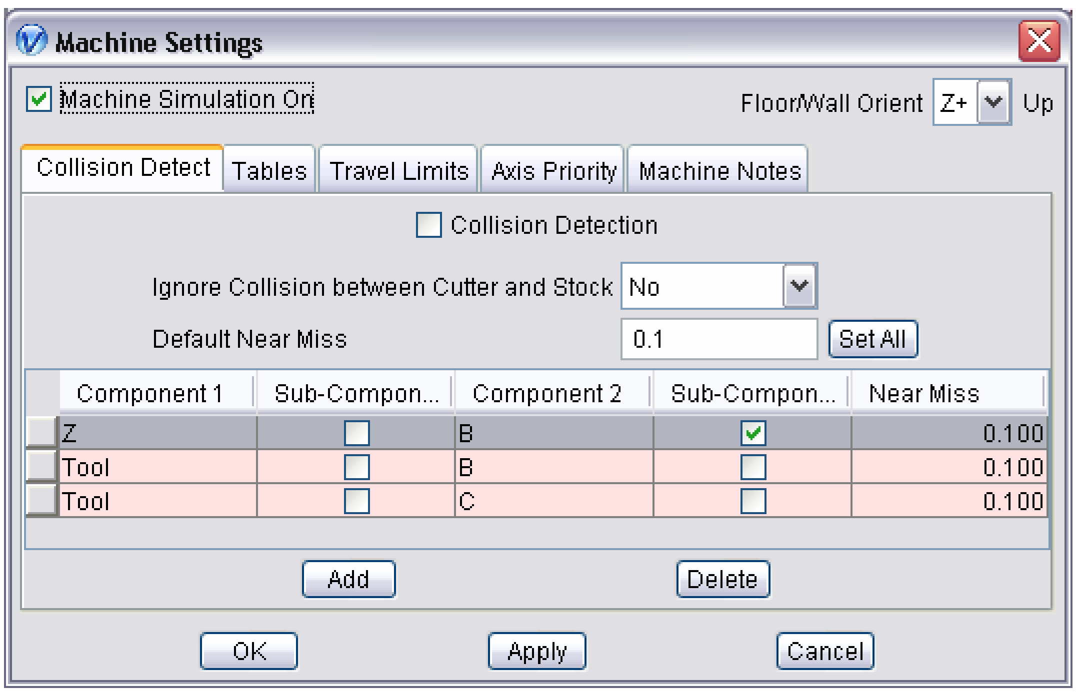
- works with a standard VERICUT license.
- is a compressed binary file.
- contains all files referenced by the Machine file (no external models).
- simulates exactly the same as a standard machine file (.mch)
- disables the **Configuration menu** features when used in a setup.
- ***can not*** be decrypted/expanded except by CGTech so retain your original machine (.mch) file

Machine Settings (Machine Settings window)

Location: **Configuration menu** > **Machine Settings**

Toolbar short cut for Machine Settings window: 

Opens a window to configure settings for an NC machine, such as: collision checking, travel limits, axis priority (for rapid motion), location tables, and when machine motions are simulated.



Machine Simulation On — When toggled "On", simulates machine tool motions when a 3-D machine is displayed in a machine view.

Shortcut: You can quickly toggle **Machine Simulation On**, "On" or "Off" by clicking on the **No Machine Simulation** icon  in the toolbar.

Floor/Wall Orient Up — Use to specify which way is "up" (towards the ceiling) for a machine when a machine view displays walls in the background. (Ref. **View menu** > **Attributes: Background**). The up direction is relative to the machine origin. Select the appropriate axis direction from the pull-down list.

Collision Detect tab **—** Features on this tab control when collisions between machine components are detected, which components are protected, and tolerances used for detecting collisions.

Tables tab **—** Features on this tab are used to specify important machine locations and detail tool change motion.

Travel Limits tab **—** Features on this tab define how far each machine axis can go, and control when travel limit errors are detected.

Axis Priority tab **—** Features on this tab control how machine axes move in rapid positioning mode (e.g. G0).

Machine Notes tab **—** Features on this tab are used to enter "message notes" and "comment notes" in the current machine file.

OK — Applies the changes and closes the Machine Settings window.

Apply — Applies the changes and leaves the Machine Settings window open.

Cancel — Closes the Machine Settings window without applying changes.

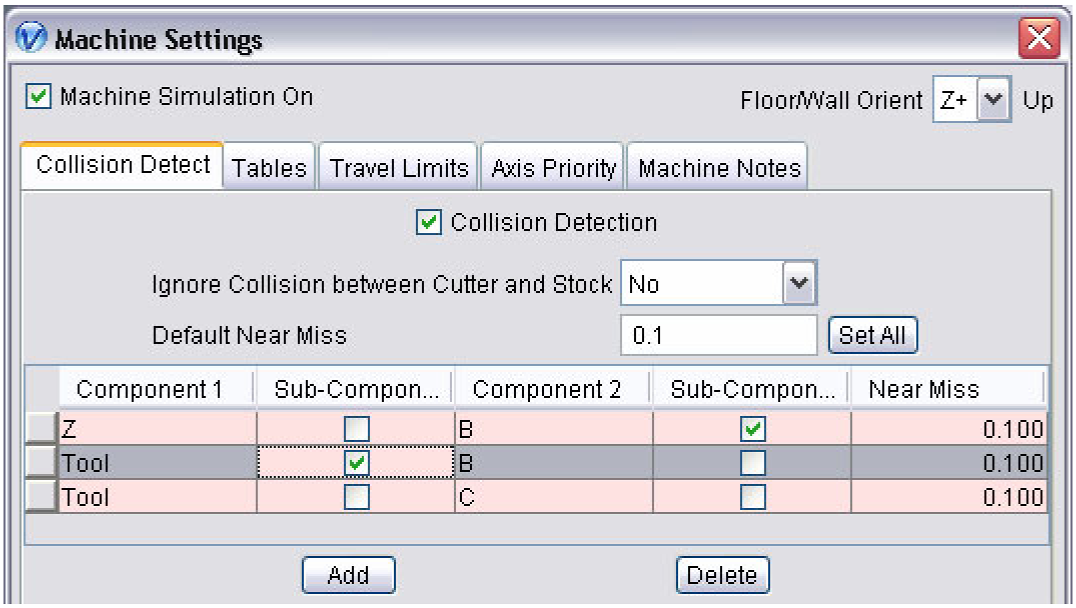
Also see "**Building NC Machines**", in the *Using VERICUT* section, in the *CGTech Help Library*.

Machine Settings window, Collision Detect tab

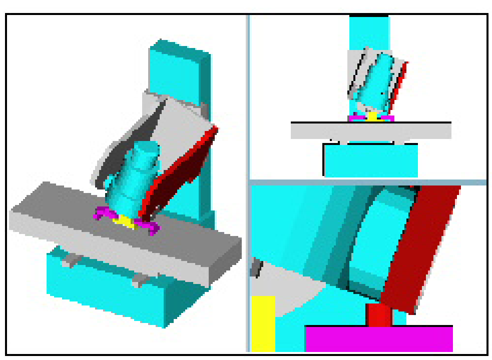
Location: **Configuration menu > Machine Settings**

Toolbar short cut for Machine Settings window: 

Features on this tab control when collisions between machine components are detected, which components are protected, and tolerances used for detecting collisions.



Colliding components are highlighted using the red Error color, and errors are issued to the Log file identifying collision causing block(s) and machine components.



Collision Detection — When toggled "On", detects collisions between specified components.

Ignore Collision between Cutter and Stock — Controls when collisions between the cutter and the Stock component are ignored. This feature is useful when collision detection is desired between the stock and shank or holder portions of the tool assembly in the machine view, but not with the cutter.

Options are:

No — (default) ***Does not*** ignore cutter-stock collisions. All collisions are reported.

All Tools — Ignores cutter-stock collisions for all tools, even inactive tools in multi-tool machines.

Active Tool — Ignores cutter-stock collisions for the active tool. However, collisions between stock and inactive tools are detected.

Default Near Miss — Specifies the default collision tolerance applied to all collision cases when **Set All** is pressed (see below).

Set All — Sets the default collision tolerance for all collision cases to the **Default Near Miss** value. You can edit the supplied tolerance for individual cases.

Component/Component collision list

Lists the component-to-component collision cases that are checked when collision detection is turned on.

Component1/Component 2 — These features are used to specify the component-to-component collision cases to check. Clicking on a component field in a record displays a pull-down list of component to choose from.

NOTE: Do not configure for collision detection between components that move (slide or rotate) against each other, such as connected motion axes. In these cases, errors may occur each time the components move.

Sub-Components — Toggle On/Off to include Sub-Components of **Component1/Component2** during collision checking.

Near Miss — Use to specify a value that controls how close the components are permitted to be before reporting a collision. Enter a positive value to be alerted if components come near each other within the specified clearance, zero to indicate components may not touch, or a negative value if components are expected to collide by the specified value.

NOTE: "Near Miss" tolerances are not supported for collision checking against the cut model. The accuracy of collisions with the cut stock is dependent on the "Cutting Tolerance".

Add — Adds a new collision case record to the list.

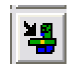
Delete — Deletes the selected collision case record from the list.

Shortcut: You can right-click in the Component/Component collision list to display a menu containing **Add** and **Delete**. These provide the same functionality described above.

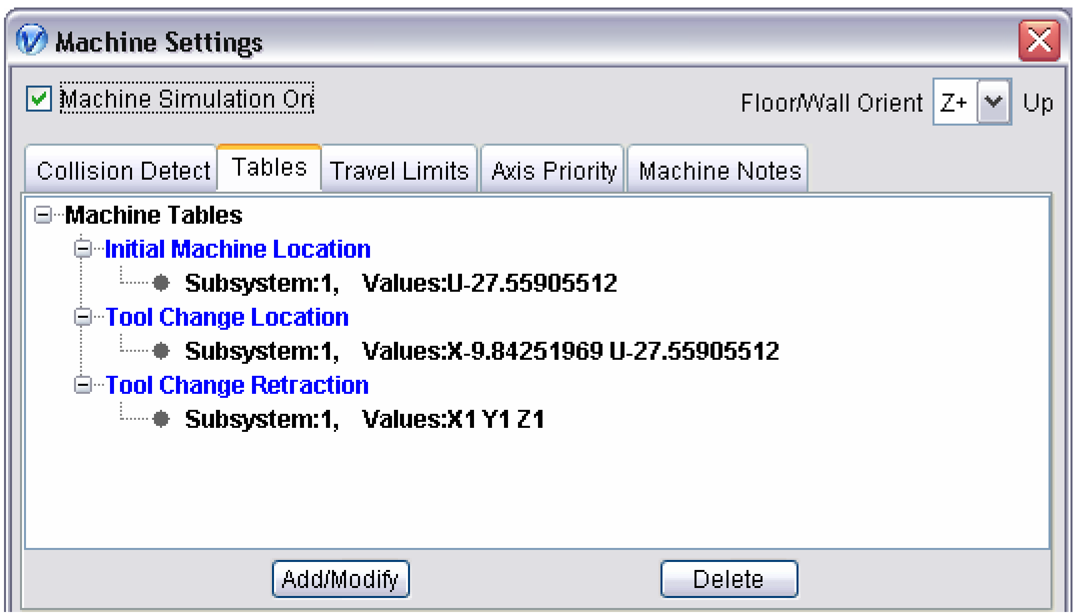
To Machine Settings window

Machine Settings window, Tables tab

Location: **Configuration menu > Machine Settings**

Toolbar short cut for Machine Settings window: 

Features on the Tables tab are used to specify Machine Tables to store important machine locations and detail tool change motion. Machine Tables are used to specify "standard" data values specific to a particular machine.



Tables list — A list of tables that are currently defined. The Machine Settings tables list will contain only "machine" tables. The G-Code Settings tables list will contain both "job" tables and "tool" tables.

Add/Modify — Use to access the **Add/Modify Tables window** enabling you to add additional tables or modify existing tables. To add a table, simply click on **Add/Modify**. To modify an existing table, select the table to be modified from the Tables list and click on **Add/Modify**. (You can also "double-click" on the table in the Tables list.)

Delete — Deletes the selected table entry from the Tables list. If it is the last entry for a particular table, then the table is also deleted.

Use the **G-Code Settings window: Tables tab** to create Job Tables to specify values to override "standard" values set in a Machine Table for particular NC program file(s). Machine Tables values are stored in the Machine file.

The following Machine tables can be added or modified with the **Machine Settings, Tables tab**:

Base Work Offset
Initial Machine Location
Input Machine Zero (obsolete)
Machine Reference Location
Tool Change Location
Tool Change Retraction
Work Offsets
Input Program Zero

See the **Tables for Processing G-Codes** section, in the *CGTech Help Library*, for additional information on VERICUT tables.

Also see "**Building NC Machines**", in the *Using VERICUT* section, in the *CGTech Help Library*.

To Machine Settings window

Add/Modify Tables window

The following discussion applies to both the **Add/Modify Machine Table window**, accessed from the **Machine Settings window: Tables tab** and to the **Add/Modify G-Code Table window**, accessed from the **G-Code Settings window: Tables tab**.

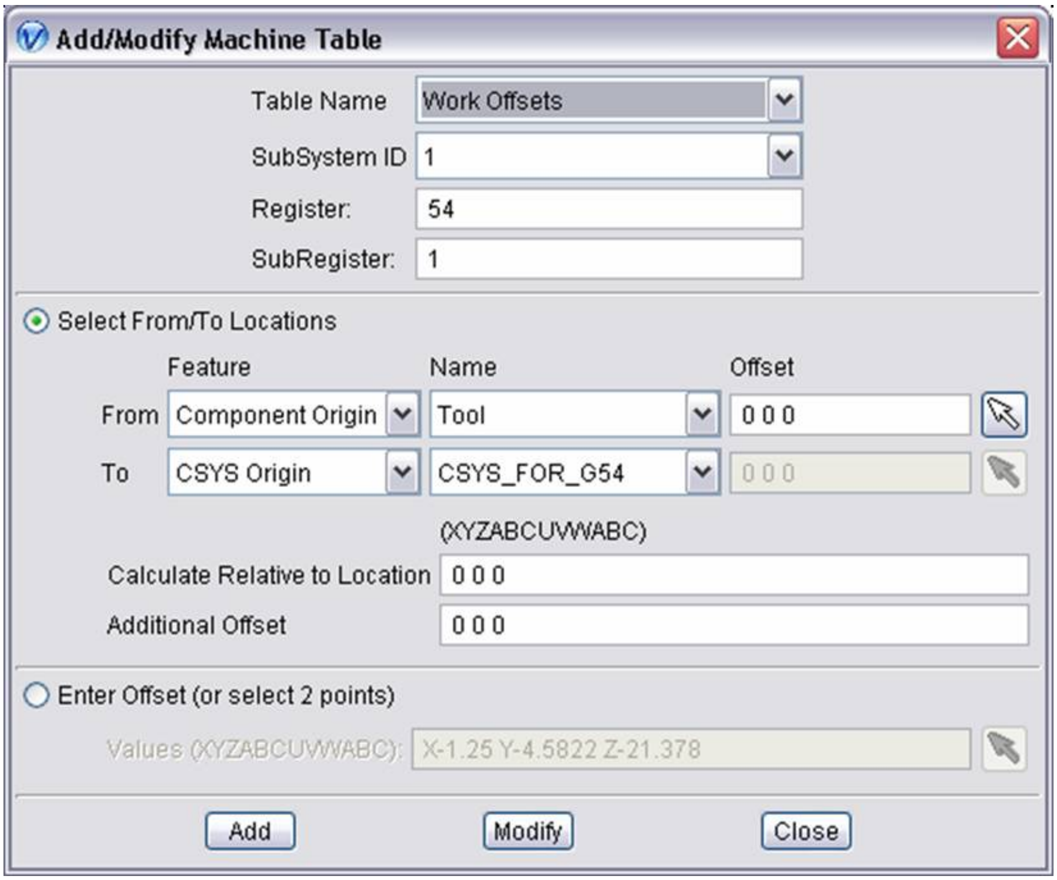


Table Name — Use to identify the table to be added, or modified, if it already exists. The **Table Name** list in the **Add/Modify Machine Table** window will only contain "machine" tables. The **Table Name** list in the **Add/Modify G-Code Table** window will contain both "job" tables and "tool" tables.

SubSystem ID — Use to specify ID of the machine subsystem for which the table is being defined.

Register — The Register number that will be used by VERICUT to access corresponding table data. For "tool" tables, this number typically corresponds to a tool or offset register number. For "job" and "machine" tables, the Register number may correspond to an offset register number, or an integer value, as required by a particular table.

SubRegister — Use of this feature enables you to access multiple sub-values for the same tool from tool related tables. For example, groove tools often have multiple "driven" points, which then correspond to multiple gage offsets, and possibly multiple Cutter Comp values, Tool Nose Comp values, and Tool Length Comp values.

This feature can also be used for Work Offset Tables for controls, like Yasnac, that support the sub-register feature.

SubRegister is only active when the selected "Table Name" identifies a "tool" related table or for Work Offset tables.

Select From/To and **Enter Values**:

These features are only active for the following tables:

Input Program Zero
Input Machine Zero
Base Work Offset
Work Offsets
RTCP Pivot Offset
RPCP Pivot Offset

For the above tables, VERICUT provides two ways to enter table values.

Select From/To Locations — Enables you to specify table values based on a "relational" offset between a "from" point and a "to" point. The "from" and "to" points are designated by specifying a particular component or CSYS. VERICUT will use the origin of the component/CSYS as the point. You can also specify an offset from the specified component/CSYS origin point. Once this relationship has been established, you can change the machine configuration or the location of the CSYS origin and the table values will updated automatically during the initial **Single Step** or during **Play to End**.

For example: After Input Program Zero has been defined to be the offset from the Tool Component to the "Program Zero" CSYS, you can change the machine configuration, or change the location of "Program Zero" CSYS, and the Input Program Zero offset values will be updated automatically during the initial **Single Step** or during **Play to End**.

NOTE: The offsets are calculated based on where the corresponding origin points are located when all linear axes are driven to machine zero (zero with no offsets in place).

When **Select From/To Locations** is toggled "On", the **From/To Feature/Name** lists and the **Offset** text fields become activated. Use the **From/To Feature** lists to specify whether the point is associated with a **Component** or a **CSYS** (coordinate system). Use the **From/To Name** lists to specify a specific component or coordinate system.

NOTE: Only the coordinate systems that have been defined with reference to a machine component (i.e., visible in a Machine or Machine/Cut Stock view when **Coordinate System** axis is toggled "on") will appear in the **CSYS/Names** lists. See the discussion for

Project menu > Coordinate Systems for more information on VERICUT Coordinate Systems.

Use the **Offset** text fields to enter 3 values, separated by spaces, representing the X, Y, and Z offset from the specified origin point. You can also click on the "select" button , then pick a point in the graphics display area. You can pick the point in either a workpiece or machine view. VERICUT will calculate the **Offset** value between the selected point and the designated **Feature**/**Name** origin point. **Offset** is only active when **Feature** is set to **Component**.

Calculate Relative to Location — This feature enables you to have a relational offset recalculated in the machine position where the offset will be used. The new position is immediately calculated and stored and therefore is not dependent on the machine position when the offset is activated.

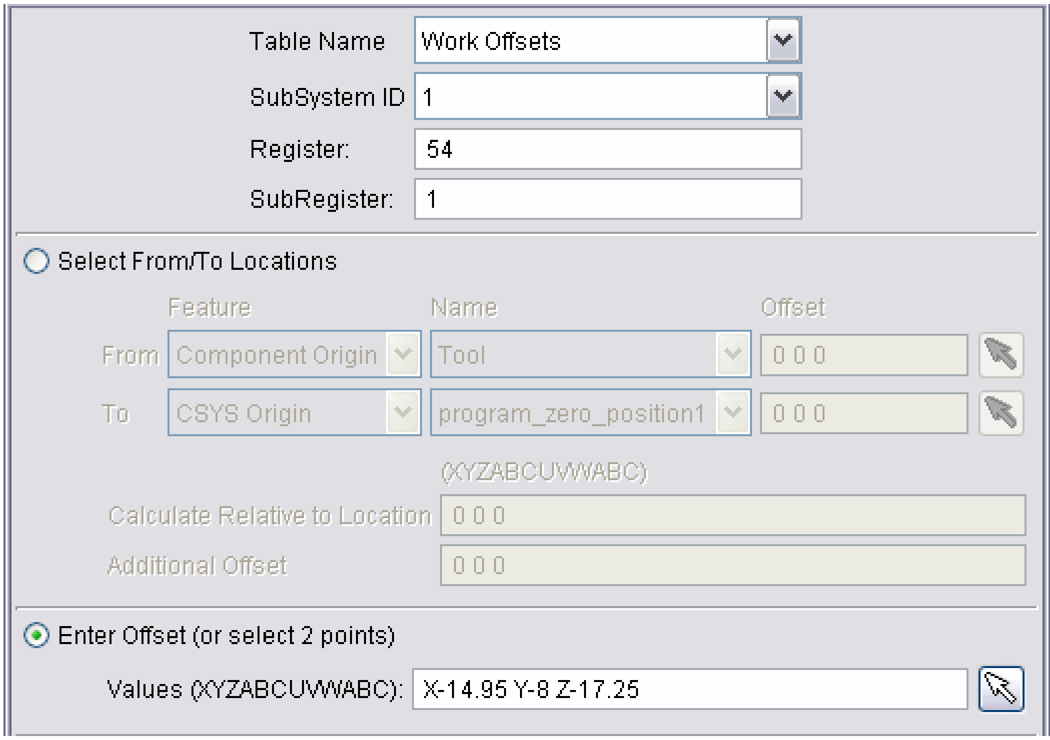
TIP: See **tombstone_work_offsets_single_part.vcproject** and **tombstone_work_offsets_multiple_parts.vcproject** in the *Sample-Demo Files* section, in the *CGTech Help Library* for examples of using this feature.

Additional Offset — Use this text field to specify an additional offset value.

Enter Offset (or select 2 points) — when toggled "On", the **Values (XYZABCUVWABC)** text field is activated enabling you to enter one to twelve numeric values as required by the specific table. Multiple value entries require spaces between the values.

You can also click on the "select" button , then pick two points in the graphics display area. You can pick the points in either a workpiece or machine view. VERICUT will calculate the **Offset** value between the selected points.

For all other tables, **Select From/To** and **Enter Offset (or select 2 points)** are both inactive. Enter the values in the **Values (XYZABCUVWABC)** text field as described for **Enter Offset (or select 2 points)** above.



In the sample window shown above, the current table is the **Work Offsets** table. The current table entry is register "54", and contains the values "-14.95 -8 -17.25". The table is applicable to subsystem "1" of the machine.

The following tables support "axis values" (up to 12 values – XYZABCUVWABC):

Base Work Offset
Initial Machine Location
Machine Zero
Machine Reference Location
Tool Change Location
Tool Change Retraction
Work Offsets
Program Zero
Input Program Zero

VERICUT supports two ways to input values into these tables from the **Add/Modify Table window**. Both use the **Values (XYZABCUVWABC)** text field, but use a different syntax):

Traditional format: values are entered in the order specified (XYZABCUVWABC) and are separated by spaces.

WORD/VALUE format (X10 C45)

In either case, the resulting display of the axis values will be in the WORD/VALUE format. Only non-zero values will be displayed. In the case of all zero's, X0 Y0 Z0 will be displayed.

The WORD/VALUE format has the following rules:

1. Valid Words are: X, Y, Z, A, B, C, U, V, W, AA, BB, CC
2. Mixed formats are not allowed. You can use ***either*** the Traditional format, ***or*** the WORD/VALUE format.
3. The Words can be entered in any order.
4. You only need to specify words with non-zero values.
5. If a word is specified twice, the last value specified will be used.
6. If all values are zero, you can leave the field blank. In this case, the display will be X0 Y0 Z0.

See the **Tables for Processing G-Codes** section, in the *CGTech Help Library*, for additional information on VERICUT tables.

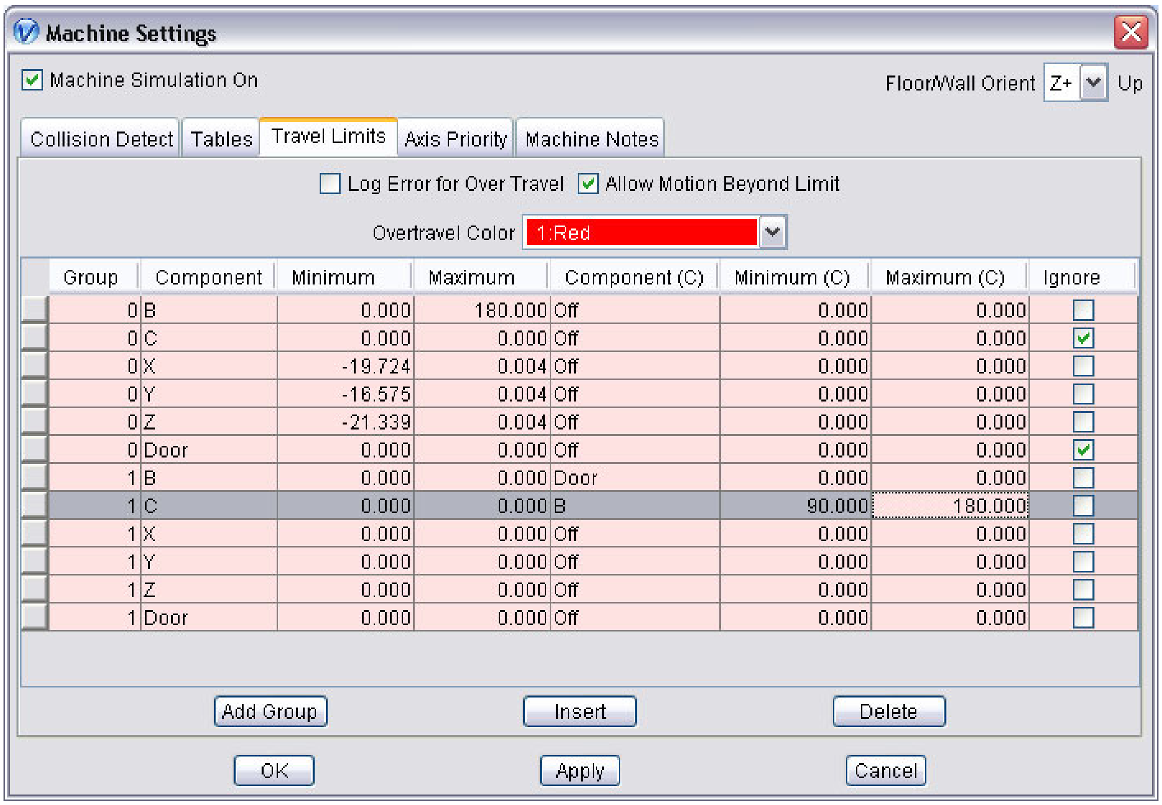
Also see "**Building NC Machines**", in the *Using VERICUT* section, in the *CGTech Help Library*.

Machine Settings window, Travel Limits tab

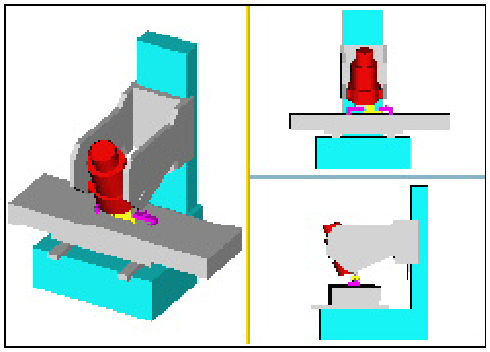
Location: **Configuration menu > Machine Settings**

Toolbar short cut for Machine Settings window: 

Features on this tab enable you to define travel limits for each machine axis , and control when, and under what conditions, travel limit checking is done and errors reported.



Overtravel errors cause the violating machine component to "light up" in the Overtravel Color and errors are output to the Log file identifying the problem component.



Log Error for Over Travel — When active, turns on overtravel detection (travel limit checking). Components that move beyond specified limits are highlighted in the **Overtravel Color** and errors are issued to the Log file identifying error causing block(s) and machine components.

Allow Motion Beyond Limit — When toggled "On", axes are allowed to freely travel beyond defined limits. When toggled "Off", an axis is never allowed to travel beyond the defined limit.

Overtravel Color — Use to specify the color in which components that move past their specified limits are shaded. Available shade colors are defined on the **Edit menu > Colors: Define tab**.

Travel Limit Record Table

List of travel limit records in which each record consist of a group number, motion component (and its axis limits), travel limit condition and an ignore switch.

Group — Enables travel limit records to be grouped together. The default group ID is 0. You can change the group ID by clicking on the Group field of the highlighted record to put it in edit mode, then enter the desired group number. A specific group of travel limit records is activated by using the **TravelLimitsGroup** macro. For information on the **TravelLimitsGroup** macro, and all VERICUT macros, see *VERICUT Macros* in the *CGTech Help Library*.

Component — Use to specify a motion component for travel limit checking. Click on the Component field of the highlighted record to put it in edit mode. Select the desired motion component from drop down list. The list contains of all motion components in the machine; linear, rotary or turret.

Minimum / Maximum — Use to specify the minimum, and maximum, travel limits for the motion axis selected in the list. Click on the **Minimum**/**Maximum** field of a highlighted record to put it in edit mode. Enter the desired axis limit.

The next three columns in the **Travel Limit Record Table** are used to specify conditions when the travel limit record is used. The condition can be set to "OFF" indicating no special condition for this record or set to a motion component on the machine.

When a motion component is specified, then the travel limit record is used if the axis of the conditional component is within the conditional minimum and maximum limit values.

Component (C) — Use to specify a condition when the travel limit record is used. Set to "OFF" indicating no special condition for this record, or specify a conditional motion component on the machine. Click on the Component (C) field of the highlighted record to put it in edit mode. Select the conditional motion component from drop down list. The list contains of all motion components in the machine; linear, rotary or turret.

Minimum (C) / Maximum (C) — Use to specify the minimum, and maximum, travel limits for the conditional motion component (Component (C)). Click on the Minimum (C) / Maximum (C) field of a highlighted record to put it in edit mode. Enter the desired axis limit.

Ignore — When toggled "On", over travel checking is ignored for the corresponding axis.

Add Group — Use this feature to add a group of travel limit records. The number of travel limit records added will be equal to the number of motion components in the machine. The group number assigned will be the next available ascending group number in the list of travel limits.

Insert — Use this feature to add a travel limit record.

Delete — Use this feature to delete the selected travel limit record(s).

OK or **Apply** must be used to apply changes. If you use **Cancel** and have made changes but have not used **OK** or **Apply**, you will be queried whether or not you want to apply the changes before leaving.

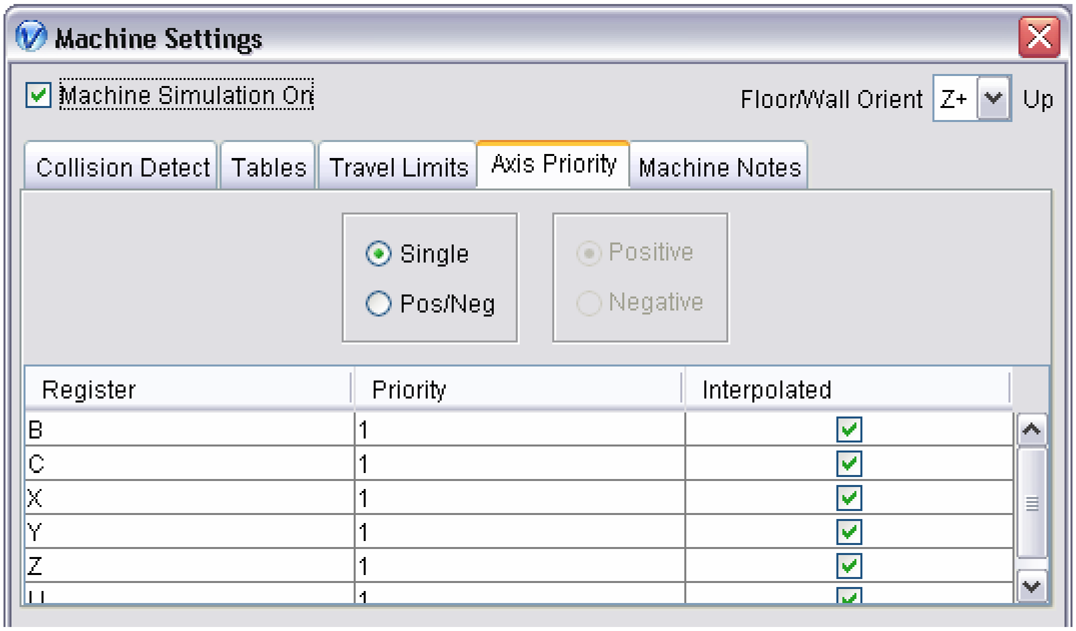
To Machine Settings window

Machine Settings window, Axis Priority tab

Location: **Configuration menu > Machine Settings**

Toolbar short cut for Machine Settings window: 

Features on this tab control how machine axes move in rapid positioning mode (e.g. G0). By default, machine axes are interpolated with equal priority to arrive at their destinations at the same time. You can change the priority to simulate non-interpolated rapid motion such as "dog leg" or "squared off" movement.



Single — This option provides a single set of rapid priorities are used. The rapid priorities applied to the tool moving in the negative Z direction are the same as those applied to the tool moving in the positive Z direction.

Pos/Neg — This option provides different rapid priorities depending on whether the tool is moving in a positive or negative Z direction. With this option, two rapid priority lists are provided. Use the **Pos/ Neg** options to control which list is seen.

Positive / Negative options — When Pos/Neg priority is selected, these options control which rapid priority list is seen: the list applied when the tool is moving in the Positive Z direction, or the list for Negative Z tool motion.

Axis Priority list — List of all machine motion axes and corresponding priorities.

Priority — Priority for moving motion axes move in rapid. A value of "1" moves that axis first. A "2" moves the axis with secondary priority (after those with priority "1"), and so on. Axes having the same priority move together.

Interpolated — When active, interpolates axes that move together such that they arrive at their destination at the same time. Clear this checkbox to have axes move based on their defined Rapid Rate- ref. **Modeling window: Component Attributes tab**.

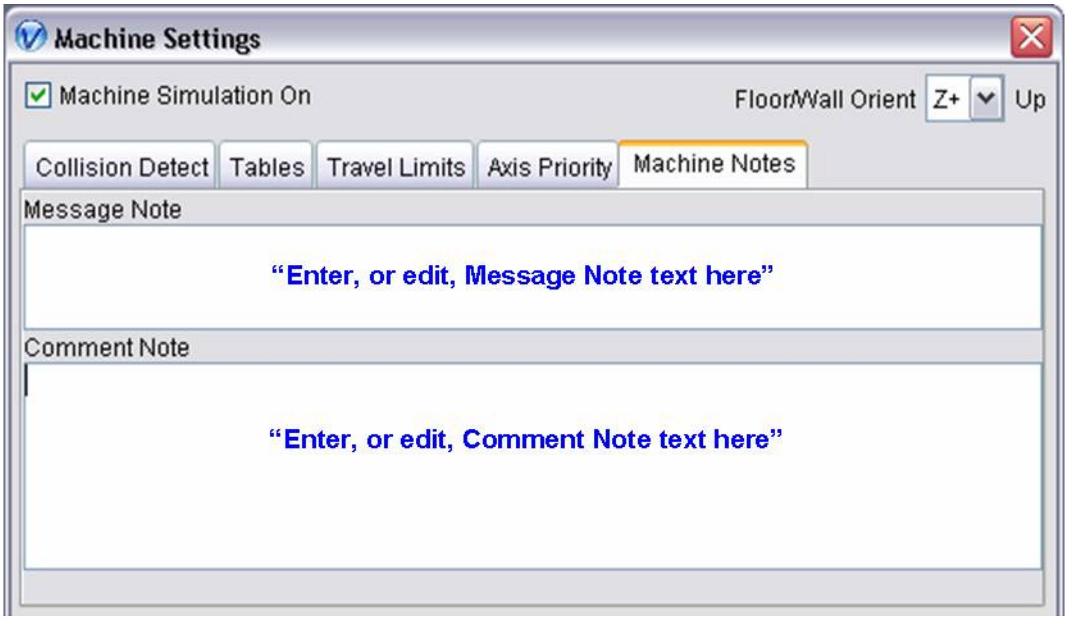
To Machine Settings window

Machine Settings window, Machine Notes tab

Location: **Configuration menu > Machine Settings**

Toolbar short cut for Machine Settings window: 

Features on this tab enable you to add **Message Notes** and **Comment Notes** to the machine file.

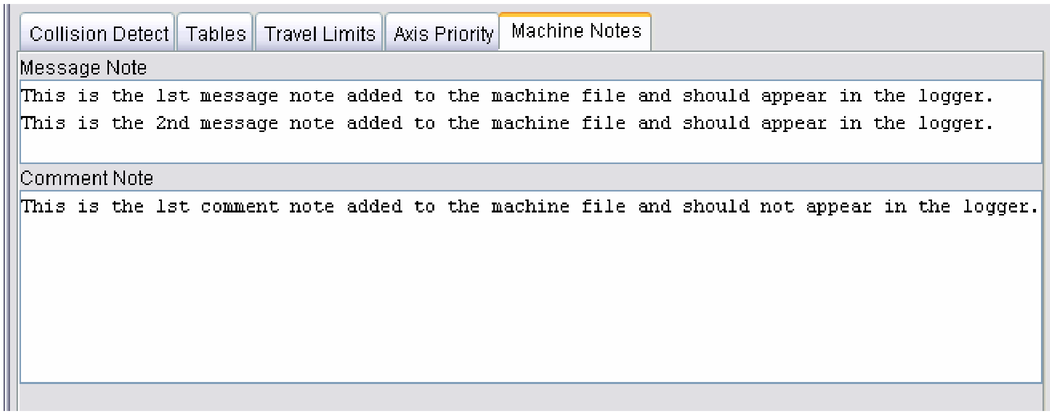


Message Notes are saved in the header of the machine file and are displayed in the VERICUT message area (Logger) when the machine file is loaded. **Comment Notes** are saved in the header of the machine file but do not display in the VERICUT Logger.

There is a limit of 255 characters per line, but there is no limit to the number of lines that can be added.

Any Message/Comment Notes that currently exist in the machine file are displayed in the appropriate text field as shown below. Any note can be edited and re-saved in the machine file.

Existing Message/Comment Notes in the machine file:



In the machine file, each line of note is added to the header in quotes, and is preceded by either the keyword MESSAGE (for message notes), or COMMENT (for comment notes) as shown below.

How the above messages look in the machine file:

```
CGTech Machine
Version-6.2
MESSAGE "This is the 1st message note added to the machine file and
should appear in the logger."
MESSAGE "This is the 2nd message note added to the machine file and
should appear in the logger."
COMMENT "This is the 1st comment note added to the machine file and
should not appear in the logger."

UNITS MILLIMETER

MACHINE "" {
   PRIORITY_TYPE SINGLE
```

Adding /editing notes:
Add a note by typing the desired text, or edit the text of existing notes in the appropriate text field. When finished, press **OK**, or **Apply**, depending on whether or not you want to leave the Machine Settings window open. Then save the machine file using the **Configuration menu** > **Machine** > **Save (or Save As)** feature in the menu bar, or use  (**Save Machine**) in the tool bar.

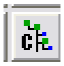
NOTE: Machine files containing notes can not be used in pre-V6.2 versions of VERICUT.

To Machine Settings window

Component Tree (Component Tree window)

VERICUT Users:

VERICUT Location: **Configuration menu > Component Tree**

VERICUT toolbar short cut: 

Mold and Die Users:

Mold and Die Location: **Advanced Options page > Modeling**

Notebook Feature: 

Cutter Grinder Users:

Cutter Grinder Location: **Advanced Options page > Modeling**

Notebook Feature: 

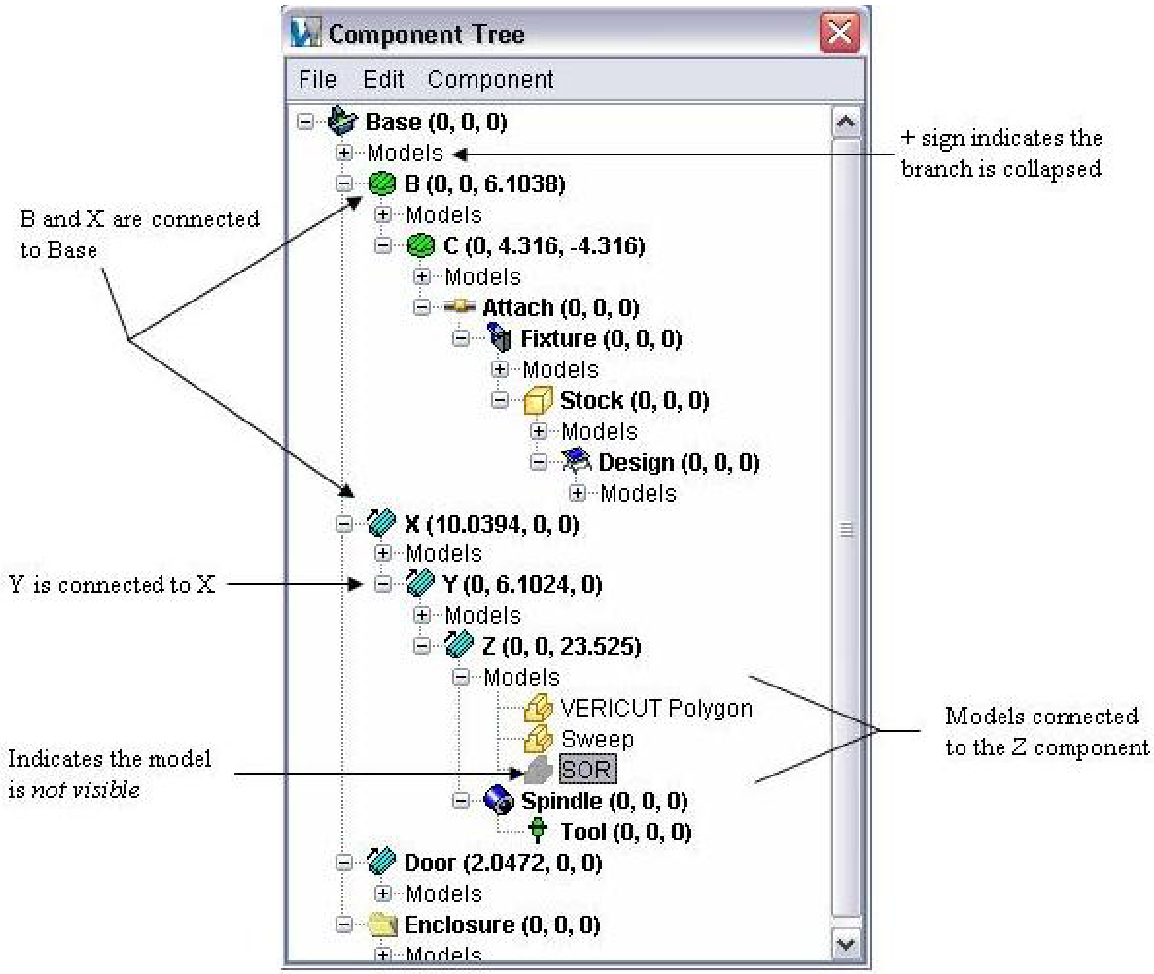
Opens a window that shows the "tree" of components and how they are connected, as well as provides a fast and easy way of manipulating components and their models. The component tree shows defined components by name and their connect positions. Indentation shows how components are connected in the tree. Select a component by picking its name in the tree. Actions performed using Components Tree window features are performed on the active (selected) component.

To learn more about modeling and using the component tree, see "**About modeling in VERICUT**" in the *Using VERICUT* section, in the *CGTech Help Library*.

Always press  (**Reset Model**) after changing components in the component tree.

Component Tree — Tree of defined components and models. Indentation indicates that a component is connected to a parent component. Component names are listed in bold text with connect position values. Models added to a component are listed beneath the component name. Click the + / - symbols to expand or collapse branches of items in the tree.

Sample Component Tree window:



File menu —

Open Machine, **Save Machine** and **Save As Machine** functions enable you to open or save a Machine file containing data that describes NC machine construction, kinematics, and other properties.

Save Cut Stock allows you to save the highlighted "Cut Stock" model as a VERICUT Solid file (.vct). This option is only active when the Cut Stock model is highlighted in the tree. The default file extension is .vct.

A cut stock model can be saved as a VERICUT Solid file "with features" or "without features". Toggle the **Save with Features** option, in the Save Cut Stock window, **On** (the default) or **Off**.

When the cut stock model is saved "with features", the cut database, the full history database, cut and stock colors are stored in the VERICUT Solid file in addition to the geometry data. When the cut stock model is saved "without features", only the geometry data is saved in the VERICUT Solid file.

VERICUT Solid (.vct) files are by default compressed when they are saved. VERICUT can read these compressed files directly.

To disable writing compressed VERICUT Solid files, set the environment variable CGTECH_COMPRESS=VCT (ref. **Environment Variables** in the *VERICUT Help* section, in the *CGTech Help Library* for additional information).

See **VERICUT Solid File**, in the *Getting Started with VERICUT* section, of *VERICUT Help* for more information.

This functionality can also be accessed using **File menu > Save Cut Stock > *stock name***.

Edit menu — Functions in this menu manipulate components in the tree, as well as specify the component name, type, and attributes.

Cut / **Copy** / **Paste** / **Delete —** Cuts, copies, pastes, or deletes a selected component in the component tree. All connected components and models are also affected.

Shortcuts:

1. Right-click on the component in the component tree, then choose the desired option from the menu that appears. See Component Tree Right Mouse Button Shortcut Menus below.
2. You can also move components via dragging them from one parent component to another, and copy components via holding down the <**Ctrl**> key and dragging.
3. Delete components or models by selecting the object in the graphics area, then use the <**Delete**> key on your keyboard.

Rename — Allows you to rename the selected component. Type the new component name.

Component Attributes — Accesses the features on the **Modeling window: Component Attributes tab**-see that section for details.

Component Type — Enables you to edit the component's "Type" setting which determines the component's function in the simulation. Select the desired "Type" from the pull-down list.

Component menu — The features in this menu enable new components to be added to the tree.

Insert — adds the selected component *before*, and at the same level as the highlighted component.

Append — adds the selected component *after*, and as a child of the highlighted component.

The component type describes the component's function in the simulation. Choose the appropriate component type from the option list, based on the component's function in the simulation. Use **Edit menu > Component Attributes** to supply any other attributes that are needed.

Component types:

X Linear, **Y Linear**, **Z Linear —** Linear motion axes on an NC machine. Motion axes are parallel to X,Y,Z axes, respectively-but this can be changed via the Motion Axis component attribute.

A Rotary, **B Rotary**, **C Rotary —** Rotary motion axes on an NC machine. The component origin is the pivot point of rotation. By default, rotation occurs about the X, Y, Z axes, respectively-but this can be changed via the **Motion Axis** component attribute.

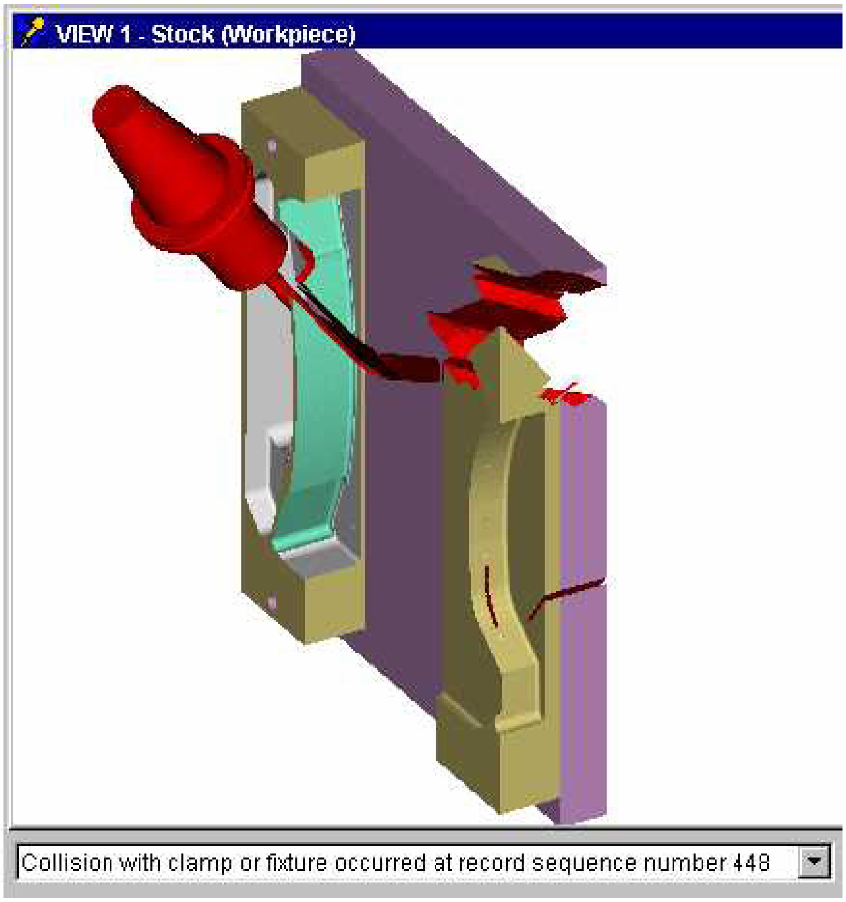
U Linear, **V Linear**, **W Linear —** Co-linear motion axes on an NC machine. Motion axes are parallel to X,Y,Z axes, respectively-but this can be changed via the **Motion Axis** component attribute.

A2 Rotary, **B2 Rotary**, **C2 Rotary —** Secondary rotary motion axes on an NC machine. The component origin is the pivot point of rotation. By default, rotation occurs about the X, Y, Z axes, respectively-but this can be changed via the **Motion Axis** component attribute.

A Turret, **B Turret**, **C Turret —** Indexing tool turret that rotates about X, Y, or Z axes. Like rotary components, the turret component origin is the pivot point for rotary indexing. The presence of these components cause all connected tools to be displayed making it possible to detect collisions resulting from turret indexing. A Turret component is commanded to move using a tool change command (e.g. Tn). The turret indexes based upon activating a Tool component connected to the turret. Tool component orientation determines the tool index position in the turret. Rotary control settings determine how turrets rotate, e.g.: rotation direction, etc.

Base — A non-moving component to which other components are connected. Every machine definition begins with this component.

Fixture — Hardware used to hold the workpiece for machining, such as: base plates, clamps, bolts, vises, pallets, etc. VERICUT assumes fixtures are not to be machined. Cuts in a fixture are shaded using the red Error color and an error is issued to the Log file.



Stock — Workpiece to be machined. This component type must be present in all simulations.

Design — Solid or surface representation of the theoretical designed part.

Design Point — Point representation of the theoretical designed part.

Deflector — Deflectors used for water jet machining.

Spindle — Spindle on an NC machine. The function of this component is different, depending on the machine type: milling or turning. Milling machine spindle- portion of the machine where cutting tools are loaded. Turning machine spindle - rotating component that turns the workpiece about the machine Z-axis.

Tool — The location where cutting tools are loaded in an NC machine. Every NC machine definition must include one or more of these components.

Guide — Wire guides on a wire EDM machine.

Electrode — Electrodes for Die Sinking Simulation (EDM machining).

Tool Chain — The Tool Chain component is used to define and model large tool handling mechanisms associated with machining centers. See **Toolchain Operation** in the *Using VERICUT* section, in the *CGTech Help Library* for more information.

NOTE: The ToolChain component's location, in the component tree, ***must not*** precede any tool component(s). An error message will be displayed during loading, and during resets, if the ToolChain component precedes tool component(s).

Attach — The Attach component is used to connect "setup" components in the project file to the machine.

- Setup components are saved in the project file, not the machine file.
- Stock/fixture/design components are "setup" components
- "Setup" components can be axes and have kinematics relationships
- Auxiliary devices can be connected to the Attach component

How does the "Attach" component work?

- A machine must have at least one Attach component, but multiple Attach components are allowed. If none is found, one is automatically created at the end of the first non-tool branch of the machine kinematics tree.
- Components are contained in both in the machine, and in the setup.
- A parent component in a "setup" references an Attach component in a machine.

Other — Any non-moving component not represented by one of the preceding types. This component type is often used to define housings, gear boxes, shields, etc.

NOTES:

1. **Right-click shortcut menus:** Features used to manipulate components are displayed in a shortcut menu when you right-click on a component in the Component Tree window. Similarly, features used to manipulate models are displayed when you right-click on a model in the tree. See Component Tree Right Mouse Button Shortcut Menus below for more information.

2. Double-clicking on a model or component in the Component Tree window opens the Model Definition window.

3. You can move objects in the Component Tree window via dragging them from one parent component to another, or copy them via holding down the <**Ctrl**> key and dragging.

4. Click repeatedly on a model in the graphics area to toggle selection between the model and its component. Refer to the Component Tree window to review what is selected.

5. Using the center mouse button of a three button mouse, click repeatedly on a model in the graphics area to toggle selection between different models along the line of sight from the pick point. Use this technique to select models hidden from view by an object in the foreground.

6. After cutting, a "Cut Stock" model appears as 

in the Component Tree window. This object can be manipulated using modeling features just like any other component or model, allowing you to easily relocate the workpiece for additional machining. You can also save the "Cut Stock" model as a VERICUT Solid file (.vct).

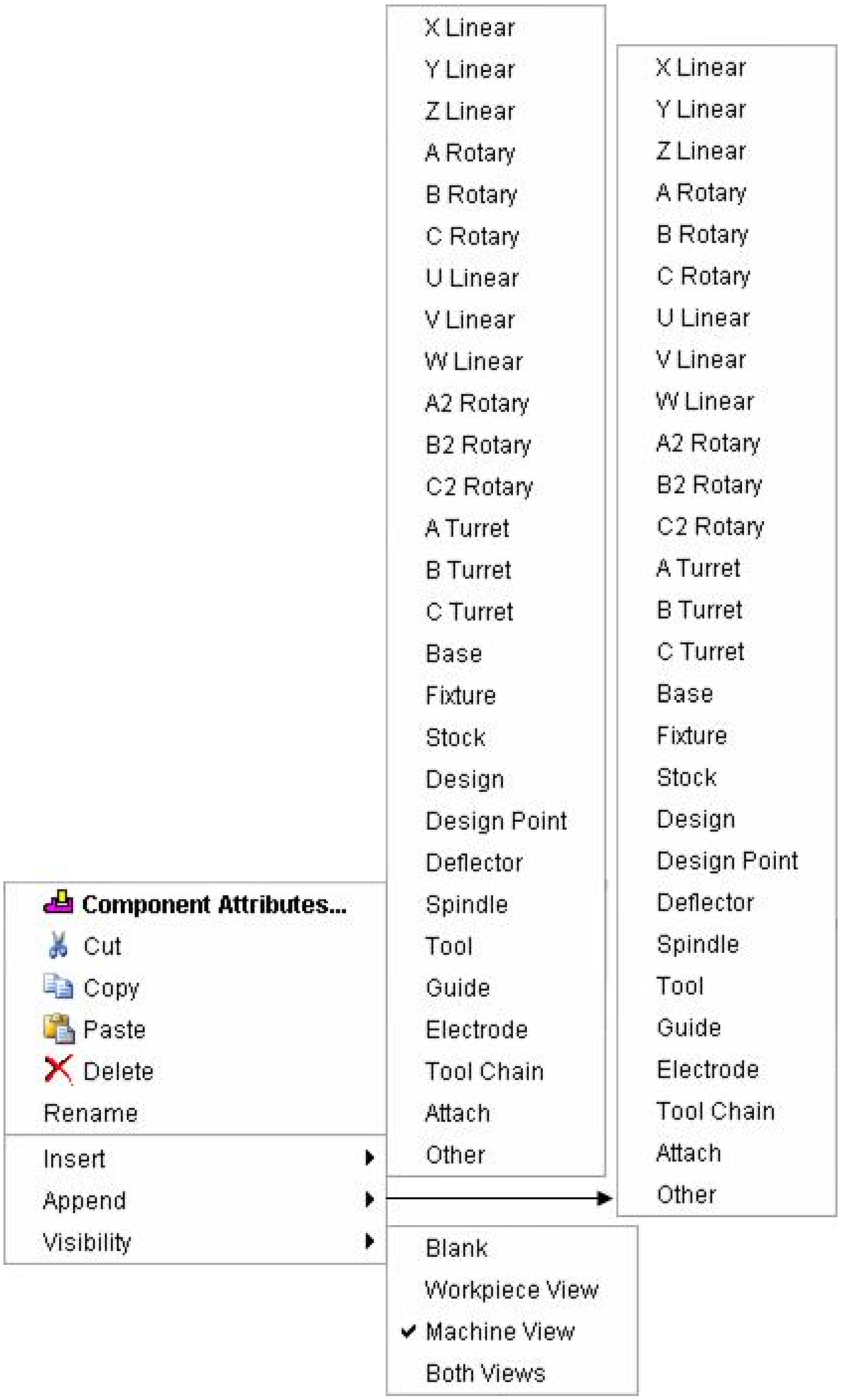
Component Tree Right Mouse Button Shortcut Menus

Each item in the Component Tree has a right mouse button shortcut menu containing features specific to the particular item.

NOTE: Any item shown in bold a Component Tree right mouse button shortcut menu can also be reached directly by double clicking on the item (component, model) in the Component Tree, that you right-clicked on to display the shortcut menu. For example, double-clicking on the Component branch in the Component Tree displays the **Modeling window: Component Attributes tab**, just like clicking on Component Attributes in the right mouse button shortcut menu shown below.

Component

Right-click on a **component** in the Component Tree window to display the following menu:



Component Attributes — Opens the Modeling window: Component Attributes tab. (ref. **Modeling window: Component Attributes tab** in the Project Menu section of *VERICUT Help*)

Cut — Cuts the highlighted component from the Component Tree and puts it in the paste buffer.

Copy — Copies the highlighted component from the Component Tree to the paste buffer.

Paste — Uses the contents of the paste buffer to replace the highlighted component in the Component Tree.

Delete — Deletes the highlighted component from the Component Tree.

Rename — Use to rename the highlighted component.

Insert — Use to add the selected component above and at the same level as the highlighted component. (ref. Component Tree window: Component Menu, in the Configuration Menu section of *VERICUT* Help)

Append —Use to add the selected component below, and as a child of the highlighted component. (ref. Component Tree window: Component Menu, in the Configuration Menu section of *VERICUT* Help)

Visibility — Use the Visibility features to specify whether or not a component is visible, and in what views. The check indicates the "current" visibility status.

Blank — The highlighted component branch is not visible in any view. When Visibility is set to "Blank" for a component, the icon representing the component in the Component Tree will be displayed in gray instead if a color.

Workpiece View — The highlighted component branch is only visible in a Workpiece View.

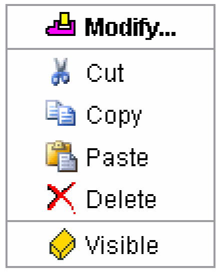
Machine View — The highlighted component branch is only visible in a Machine View (Machine, or Machine/Cut Stock).

Both — The highlighted component branch is not visible in any view.

Shortcut: Double clicking on a Component will open the Modeling window: Component Attributes tab.

Model

Right-click on a **model** in the Component Tree window to display the following menu:



Modify — Opens the Modeling window: Model tab. (ref. **Modeling window: Model tab** in the Project Menu section of *VERICUT Help*)

Cut — Cuts the highlighted model from the Component Tree and puts it in the paste buffer.

Copy — Copies the highlighted model from the Component Tree to the paste buffer.

Paste — Uses the contents of the paste buffer to replace the highlighted model in the Component Tree.

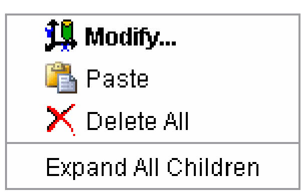
Delete — Deletes the highlighted model from the Component Tree.

Visible — Use to make the highlighted model visible, or not visible in the VERICUT graphics area. The Visible icon will indicate the visibility status of the model. indicates "visible", indicates "not visible". Click on Visible to toggle between the two modes. In the Component Tree, any model in the "not visible" state will be displayed in gray instead if a color.

Shortcut: Double clicking on a Model in the Component Tree window will open the Modeling window: Model tab.

Coordinate Systems Branch

Right-click on a **Coordinate Systems** branch in the Component Tree window to display the following menu:

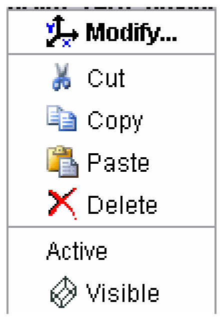


Modify — Opens the Coordinate System window. (ref. **Coordinate System window** in the Project Menu section of *VERICUT* Help.
Paste — Use to paste the contents of the paste buffer to the Coordinate Systems branch.
Delete All — Deletes all coordinate systems from the Coordinate System branch.
Expand All Children — Expands the Coordinate System branch showing all coordinate systems available in the setup.

Shortcut: Double clicking on a Coordinate Systems Branch will open the Coordinate System window.

Coordinate System

Right-click on a **Coordinate System** in the Component Tree window to display the following menu:



Modify — Opens the Coordinate System window. (ref. **Coordinate System window** in the Project Menu section of *VERICUT* Help.

Cut — Cuts the highlighted coordinate system from the Component Tree and puts it in the paste buffer.

Copy — Copies the highlighted coordinate system from the Component Tree to the paste buffer.

Paste — Adds the contents of the paste buffer after the highlighted coordinate system in the Component Tree.

Delete — Deletes the highlighted model from the Component Tree.

Active — Use to designate the "active"coordinate system. (ref. **Set Active Coordinate System**, in the Project Menu section of *VERICUT Help*.

Visible — Use to make the highlighted coordinate system visible, or not visible in the VERICUT graphics area. The Visible icon will indicate the visibility status of the coordinate system. indicates "visible", indicates "not visible". Click on Visible to toggle between the two modes. In the Component Tree, any coordinate system in the "not visible" state will be displayed in gray instead if a color.

Shortcut: Double clicking on a Coordinate System in the Component Tree window will open the Coordinate System window.

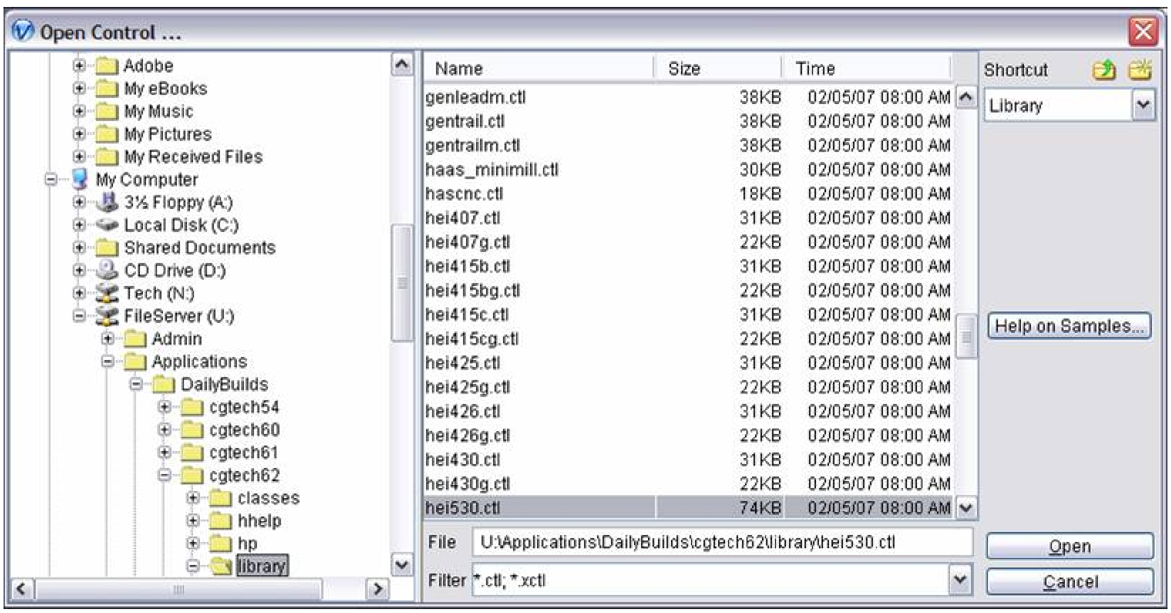
Control

Open (Control file)

Location: **Configuration menu > Control > Open**

Toolbar short cut for opening Control files: 

Opens the Open Control window enabling you to open (load) a Control file (ref. **Control File** in the Getting Started with VERICUT section of *VERICUT Help*). Control files contain data that describes how the NC control processes machine code data.



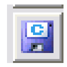
Most features on this window are standard file selection window features that enable you to navigate through directories, filter files, and type, or select, /path/filenames. A description of features specific to VERICUT can be found in the **Introduction to VERICUT File Selection Windows** in the Getting Started section of *VERICUT Help*.

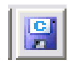
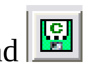
To save space, the features described below are unique to this window.

Help on Samples — Opens the *VERICUT Help* section on "**Library Control Files**". Assuming **Shortcut=**CGTECH_LIBRARY, you can find the file you want in the on-line Help, then cut & paste the file name into the file selection window and press **Open** to open the file.

Save (Control file)

Location: **Configuration menu > Control > Save**

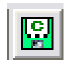
Toolbar short cut for saving Control files: 

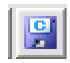
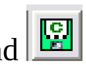
TIP: Right-click on the icon to toggle between  (SaveControl) and  (Save Control As) modes.

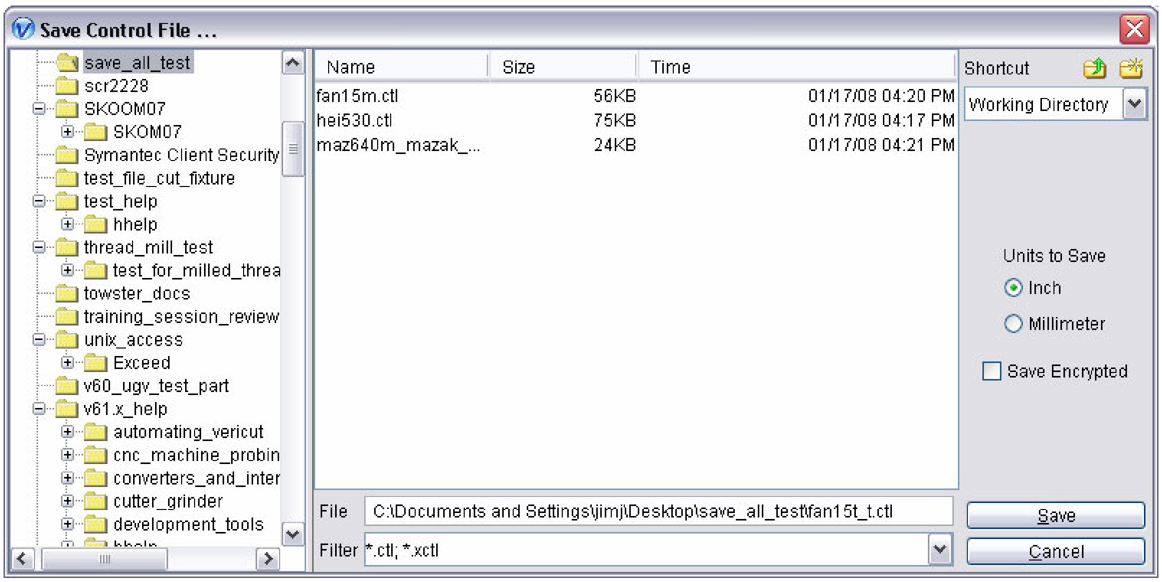
Saves (updates) an existing Control file with the current NC control settings. VERICUT will save the control file if you have sufficient permissions to save the file in its present directory. Otherwise, the Save Control File window will display enabling you to specify a location to save where you have write permissions.

Save As (Control file)

Location: **Configuration menu > Control > Save As**

Toolbar short cut for saving Control files: 

TIP: Right-click on the icon to toggle between  (SaveControl) and  (Save Control As) modes.



Opens the **Save Control File** window enabling you to save a Control file (ref. **Control File** in the Getting Started with VERICUT section of *VERICUT Help*). Control files contain data that describes how the NC control processes machine code data.

Most features on this window are standard file selection window features that enable you to navigate through directories, filter files, and type, or select, /path/filenames. A description of features specific to VERICUT can be found in the **Introduction to VERICUT File Selection Windows** in the Getting Started section of *VERICUT Help*.

To save space, the features described below are unique to this window.

Units to Save — Controls the units (Inch or Millimeter) in which to store control configuration data. If the session units are different than the units to save, then values are converted accordingly when stored in the new file.

Save Encrypted — Saves the control file as a standard encrypted (X-File) with an .xctl extension.

A "standard" encrypted control file:

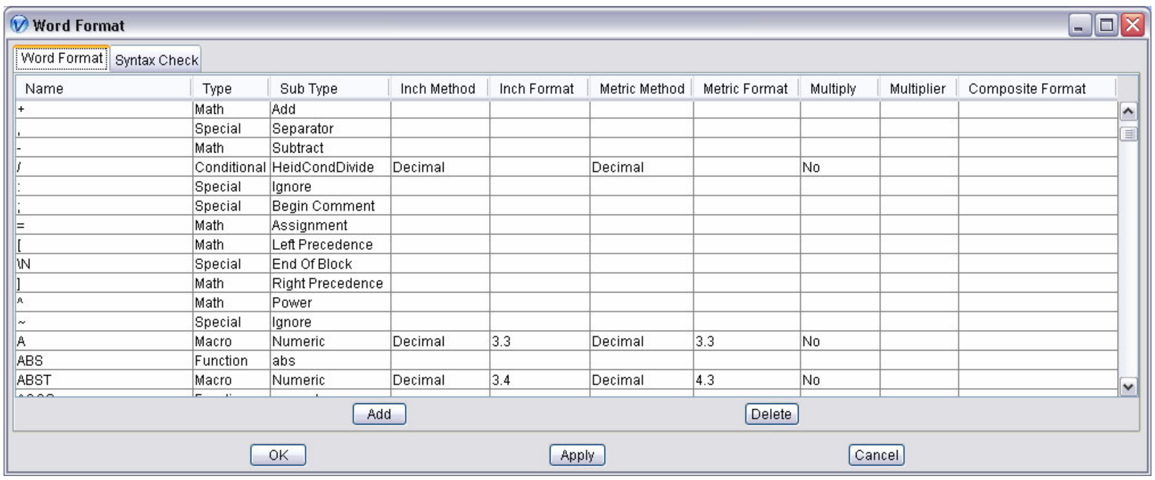
- works with a standard VERICUT license.
- is a compressed binary file.
- contains all files referenced by the Control file (no external subroutines).
- simulates exactly the same as a standard control file (.ctl)
- disables the **Configuration menu** features when used in a setup.
- ***can not*** be decrypted/expanded except by CGTech so retain your original control (.ctl) file.

Word Format (Word Format window)

Location: **Configuration menu > Word Format**

VERICUT toolbar short cut: 

Opens a window enabling you to specify how the NC control interprets G-Code words and special characters, to specify syntax checking rules (error conditions) for checking your NC program file for valid syntax.



NOTE: Always reset VERICUT (press  (**Reset Model**) on the VERICUT main window) after making changes to the control configuration.

Word Format tab **—** List of machine code, or "G-Code" data words and characters that are interpreted by the control, arranged in alphabetical order. The list provides a summary of word types, sub-types, inch and metric formats, and multipliers and composite formats.

Syntax Check tab **—** The features on this tab enable you to turn On/Off VERICUT defined syntax checking rules (error conditions) and to define custom syntax checking error conditions for checking the statements in your NC program file for valid syntax.

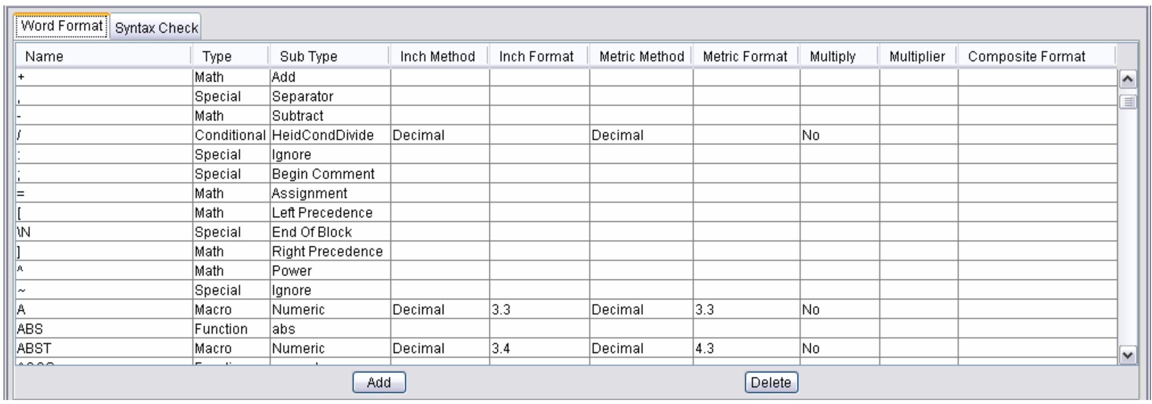
OK — Saves additions/modifications and dismisses the Word Format window.

Apply — Saves additions/modifications and leaves the Word Format window open.

Cancel — Closes the Word Format window without saving any additions/modifications.

Word Format window, Word Format tab

Opens a window enabling you to specify how the NC control interprets G-Code words and special characters. Once words are defined, they can be "grouped" with address values via the **Configuration menu > Word/Address** function to perform specific actions by calling CGTech supplied or custom macros. Undefined words cause errors when processed, and are not acted on by VERICUT.



Name	Type	Sub Type	Inch Method	Inch Format	Metric Method	Metric Format	Multiply	Multiplier	Composite Format
+	Math	Add							
,	Special	Separator							
-	Math	Subtract							
/	Conditional	HeidCondDivide	Decimal		Decimal		No		
:	Special	Ignore							
;	Special	Begin Comment							
=	Math	Assignment							
[	Math	Left Precedence							
\N	Special	End Of Block							
]	Math	Right Precedence							
^	Math	Power							
~	Special	Ignore							
A	Macro	Numeric	Decimal	3.3	Decimal	3.3	No		
ABS	Function	abs							
ABST	Macro	Numeric	Decimal	3.4	Decimal	4.3	No		

Each column in the Word Table represents a specific feature that enables you to define how the NC control interprets specific G-Code words or special characters.

Name — Identifies the word or special character. Each word name must be unique.

Type — Specifies the type of control function performed by the word. Options:

> **Logical —** Evaluates a logical expression. See "Logical Sub-type" below for more detailed classifications.
>
> **Special —** Accesses special NC control functions. See "Special Sub-type" below for more detailed classifications.
>
> **Math —** Performs a mathematical operation. See "Math Sub-type" below for more detailed classifications.
>
> **Function —** Calls the control function specified in the Function Name field.
>
> **Type II —** Specifies that the word is a three character mnemonic representing a Type II data command, e.g. "AXO", "CLS", etc.
>
> **Macro —** Word will be paired with a value, then configured via the **Configuration menu> Word/Address** function to call one or more action macros.
>
> **Conditional —** Similar to the Functions word type, except performs multiple functions, depending on the condition of word use. Conditional words are typically

supported by special routines written by CGTech or other developers to evaluate word use.

Sub Type — Sub-types further differentiate word functions. Available sub-types are dependent on the word Type.

Logical Sub Type — Options for evaluating logical expressions:

Equal
Not Equal
Greater than
Greater Than or Equal
Less Than
Less Than or Equal
AND
OR

VERICUT also supports the following bitwise operators:

Bitwise AND: If both of the corresponding bits are on, then the resulting bit is on.
Bitwise OR: If either of the corresponding bits are on, then the resulting bit is on.
Bitwise XOR: If either, but not both, of the corresponding bits are on, then the resulting bit is on.

Simple examples:

9 AND 8 = 8
9 OR 8 = 9
9 XOR 8 = 1
7 AND 8 = 0
7 OR 8 = 15
7 XOR 8 = 15

These operators follow the standard rules of precedence.

NOTE: For a compound equation, it is always best to use parenthesis, or brackets, to insure the order in which the equation will processed. Not all controls follow the standard rules of precedence.

Special Sub Type — Options for performing special control functions. Click on the record "Sub Type" field and select from the list of available options in the pull-down menu.

Skip — Skip remaining data in block.

Begin Data / End Data — Denotes the beginning (or ending) of G-Code data to be processed by the control. If a Begin Data special word is defined with out specifying an End Data word, the Begin Data word is used for both.

Begin Comment / End Comment — Denotes the beginning (or ending) of a comment record.

Begin Type II / End Type II — Denotes the beginning (or ending) of a Type II format record.

Separator — Character used to separate data that is acted on differently by the control, for example a list of arguments or values.

Variable Tag — Character which identifies the number that follows as a variable register number, e.g. "#". Example- "#100" indicates variable register number "100" is being referenced.

Variable Name — Alpha-numeric word that identifies a variable which does not have an associated variable register number, e.g. "PPX" (type II variable format).

End of Block — Denotes the end of the G-Code data block.

Console Message — Identifies the block as a message displayed on the machine operator's console.

Quoted Text — Identifies start/end delimiter for a quoted text string. The default character for delimiting a quoted text string is the double quote ("). If a different character(s) is used as a delimiter, it must be defined as a SPECIAL type word with a sub-type of Quoted Text. Quoted text is valid in Alpha-Numeric and List-Alpha-Numeric Word Value types.

Ignore — Ignores specific words in the tool path file.

Sin840D CASE — When this word is found, a special 840D parser will be used to process the remainder of the block.

See "**Notes about the Siemens 840D CASE and REPEAT commands**" in the *Notes about Special Topics* section, in the *CGTech Help Library*.

Sin840D DEF — When this word is found, a special 840D parser will be used to process the remainder of the block.

See "**Notes about the Siemens 840D DEF command**" in the *Notes about Special Topics* section, in the *CGTech Help Library*.

Sin840D REPEAT — When this word is found, a special 840D parser will be used to process the remainder of the block.

See "**Notes about the Siemens 840D CASE and REPEAT commands**" in the *Notes about Special Topics* section, in the *CGTech Help Library*.

Sin840D DEFINE — When this word is found, a special 840D parser will be used to process the remainder of the block. It will parse the remainder of the line, and establish a temporary substitution entry.

The format is: DEFINE str AS … The "str" will be used as the "Input Text", and everything that follows the AS will be used as the "Output Text".

This substitution will be removed with the next reset, and will not be saved in the control file. Similar to variables, entries which are created in this manner will be marked with an "*" in the table on the Substitute tab.

Sin840D SET — When this word is found, a special 840D parser will be used to process the remainder of the block.

See "**Notes about the Siemens 840D DEF command**" in the *Notes about Special Topics* section, in the *CGTech Help Library*.

Sin840D REP — When this word is found, a special 840D parser will be used to process the remainder of the block.

See "**Notes about the Siemens 840D DEF command**" in the *Notes about Special Topics* section, in the *CGTech Help Library*.

Text Argument — Used when defining Siemens 840D Frame access modifiers TR (translate value), FI (fine value), RT (rotate value), SC (scale factor) and MI (mirror flag).

See "**Notes about the Siemens 840D DEF command**" in the *Notes about Special Topics* section, in the *CGTech Help Library*.

NUM VAR Define — When this word is found, a special parser will be used to process remainder of the statement up to the closing word ENDV. This includes the creation and initialization of the numerical variables, and the creation of the corresponding words (of type "Special" and sub-type "Variable Name"). Word ENDV is part of the NUM VAR Command but it doesn't have to be specified in Word Format table. The VAR command can contain several blocks but is processed as a single block.

The following defines the basic format of the VAR Command:

```
VAR
  [name] = value [name]….
  [name] [name]….
  …
ENDV
```

where:

name – Variable name enclosed in brackets according to NUM syntax.

value – assigned initial value (or expression) to the variable when present after "=" character. If the initial value is in the form of an expression, it only can use previously defined and initialized variables.

Rules for using:

1. Brackets are part of the variable name
2. No spaces or other delimiters are required between name and/or value (if used they are ignored).

3. Spaces can be used as a part of name (example [NBR AUBE]) inside brackets.
4. Variable can be initialized immediately when defined or later (variable [INCR] in example below).

Example:

```
VAR
[ORI]
[NBR_AUBE]=19
[RET_BLOC][NB_AU_US][INCR][POSIT_D][NBR_FOIS]
[INCR]=360/[NBR_AUBE]
[AUBE_NUM] [TOOL_VIE][NB_AA_US][CHG_1][CHG_2]
ENDV
```

NOTE: Numerical arrays are not supported.

Math Sub Type — Options for performing math operations. Click on the record "Sub Type" field and select from the list of available options in the pull-down menu.

Add

Subtract

Multiply

Divide

Power

Left Precedence

Right Precedence

Assignment

Mod

The Mod math operation is calculated as follows:

N - D * INT(N/D) ==> -382 - 180 * INT(-2.12222) ==> -382 - 180 * -2 ==> -22 <-- TRUNC Returns Integer part of number)

The Mod operator will produce the following results:

382 Mod 180 = 22

-382 Mod 180 = -22

10 Mod 3 = 1

10.1 Mod 3 = 1.1

Concatenate String

This operator is used to concatenate two string variables.

Example:

VarA, VarB, and VarC are all defined in the Variables window as "Text" variables

VarA, VarB, and VarC are all defined in the Word Format window as Type=Special, Sub Type=Variable Name

<< is defined in the Word Format window as Type=Math,
Sub Type=Concatenate String

```
VarA = "ABC"
VarB = "DEF"
VarC = VarA << VarB
VarC = ABCDEF
```

Link Frame

Use to configure the control to use the Siemens Frame link operator "**:**". The Frame link operator is used to combine the settings of 2, or more, functions or Frames in one Frame setup.

See **Notes about Siemens 840D DEF Command**, in the *Notes about Special Topics* section in the **CGTech Help Library** for additional information.

Function Sub Type — The control function that is called when the word is processed. Click on  in the record "Sub Type" field and select from the list of available options in the pull-down menu. See Functions – Listed Alphabetically below.

You can also enter the name of a "custom" mathematical function, created with the **CGTech Macro Language (CML)**, in the **Sub Type** text field as these will not appear in the pull-down list.

NOTE: The only time a value should ever be typed in is if it is a CMS function.

Type II Sub Type — Describes syntax expected in a "Type II" command. Enter one or more of the descriptors listed below in the "Sub Type" text field to describe the expected syntax. Descriptors used in combinations are separated by commas.

WV — word/value pair, e.g. "X1"
T — text string
V — numeric value

EXAMPLES:

1. Configuration for set axis offsets command "(AXO,X1,Y2,Z3)":

   Name=AXO
   Type= Type II
   Sub Type = WV,WV,WV

2. Configuration for call subroutine command "(CLS,SUB1)":

   Name=CLSType= Type II

   Sub Type= T,V,V,V,V,V

NOTE: The "V" descriptors shown in the example above, allow passing variables to the subroutine, although none are being passed in the above call example.

Macro Sub Type — Specifies how the control interprets values, or "addresses" that follow the word. Click on the record "Sub Type" field and select from the list of available options in the pull-down menu.

Numeric — Interprets word values as numbers. The system reads the numeric characters following the macro word until a non-numeric character is found, for example: alphabet, symbol, or separator. The non-numeric character indicates the beginning of a new word.

Alphabet — Interprets word values as alphabet characters. The system reads the alphabet characters following the macro word until a non-alphabetic character is found, for example: number, symbol, or separator.

Alpha-Numeric — Interprets word values as an alpha-numeric text string. The system reads the characters following the macro word until a symbol or separator character is found.

If a "Word" is defined to have an alpha-numeric value, then the text that follows is assumed to be the corresponding value.

1. If the text value corresponds to an existing variable, then the value passed will be the contents of the corresponding variable.
2. If the text value corresponds to a variable tag followed by a numeric value, then the value passed will be the contents of the corresponding variable.
3. Array variables are not supported.
4. Expressions other then variables are not supported.
5. Alpha numeric characters include: A-Z, 0-9, '_" and sometimes '-".
6. The above enhancement does not apply to block like the following, since an alpha-numeric string does not follow the word
   a. Word = NAME
   b. Word [NAME]

See details on what is passed to a macro given various situations.

Also see: **Notes about String Capabilities in VERICUT**, in the *Notes about Special Topics* section, in the *CGTech Help Library*.

Composite-Numeric — Separates word values into pieces which can be individually acted on. The system uses the Composite Format to "parse" the numeric characters following the word until a non-numeric character is found, for example: alphabet, symbol, or separator. See **Composite Format** for examples. Each separated piece of a composite-numeric word/value is available for calling macros via the **Configuration menu > Word/Address** function, and can be graphically selected from a word pick list.

List-Numeric — Separates an argument list following the word into pieces which can be individually acted on. This option is useful in a situation like acting on "n" values differently in the sample codes: TEXT(n,n,n,n,...) or TEXT n,n,n,n,... Parens may be present but are not required, and commas separate the arguments. Up to 32 arguments can be listed. Each argument must be numeric, or a math expression which equates to a numeric value. Similar to Composite-Numeric value types, each separated piece of a list-numeric word/value is available for calling macros. However, these pieces do not appear in the word pick list. Specify the pieces via listing the word followed by a space and the sequential number of the argument in the list. Following the above example, you could use the **Configuration menu > Word/Address** function to associate macros with "TEXT 1" (corresponds to the first "n" parameter value, "TEXT 2" (second "n" parameter), and so on.

List-Alpha-Numeric — Same as above description for List-Numeric with one addition. The value may be a quoted-text string. In this case the quoted-text is not analyzed, and is just passed as text to an associated Word/Address macro.

Alpha-Numeric + Arguments — This subtype is used to support subroutines with arguments for the Siemens 840D PROC command. The text argument associated with this subtype value will include an alpha-numeric string, and optionally, a string beginning with a "(" and ending with a ")".

See "**Notes about the Siemens 840D PROC command**" in the *Notes about Special Topics* section, in the *CGTech Help Library*.

None — The word does not have an associated value. The first character after the word indicates the beginning of a new word.

EXAMPLES:

1. Assume the word "PGM" is defined as a Macro word type. The table below shows how **Sub Type** can affect the interpretation of the values paired with "PGM".

Sample data block	Sub Type= Numeric	Sub Type= Alphabet	Sub Type= Alpha-Numeric	Sub Type= None
PGM123ABC	123	none	123ABC	none
PGMABC123	none	ABC	ABC123	none

2. Consider interpreting the following block of data: "TEST(10,20,30)"

   In **Word Format**, add word "TEST" as follows:

   Name=TEST
   Type=Macro
   Sub Type=List-Numeric

   In **Word/Address**, add group to act on first argument in TEST list "10":

   Word=TEST 1
   Range=*
   Macroname=<desired macro> (group is passed a value of "10")
   and then add groups to act on second & third arguments in TEST list "20" and "30":
   Word=TEST 2
   Range=*
   Macroname=<desired macro> (passed value: "20")
   Word=TEST 3
   Range=*
   Macroname=<desired macro> (passed value: "30")

Conditional Sub Type — The conditional control function that is called when the word is processed. Click on in the record "**Sub Type**" field and select from the list of available options in the pull-down menu. See Conditionals – Listed Alphabetically below. You can also enter the name of a "custom" conditional function, created with the **CGTech Macro Language (CML)**, in the **Sub Type** text field as these will not appear in the pull-down list.

NOTE: The only time a value should ever be typed in is if it is a CMS conditional.

Inch Method — Specifies how to interpret inch address values. This feature is significant only for values that do not contain a decimal point. Values with a decimal are always interpreted via the "Decimal" method. See NOTE below.

Options:

Decimal — Interprets values as floating point decimals.

Leading or Decimal — Interprets values without a decimal as having leading zeros. Depending on the active unit system (inch or metric), values are interpreted as described by the corresponding "Format" (see below).

Trailing or Decimal — Similar to Leading or Decimal, except interprets values as having trailing zeros.

Inch Format — Specifies the number of digits before and after the decimal point when interpreting **Inch Method**, "leading", or "trailing" zero values. Data entry format is: a.b where "a" specifies the number of digits before, and "b" specifies the number of digits after the decimal point. See NOTE below.

Metric Method — Specifies how to interpret metric address values. This feature is significant only for values that do not contain a decimal point. Values with a decimal are always interpreted via the "**Decimal**" method. See **Inch Method**, above, for list of options. See NOTE below.

Metric Format — Specifies the number of digits before and after the decimal point when interpreting **Metric Method**, "leading", or "trailing" zero values. Data entry format is: a.b where "a" specifies the number of digits before, and "b" specifies the number of digits after the decimal point. See NOTE below.

Multiply / Multiplier — When Multiply is set to "Yes", the word value is multiplied by the amount specified in the Multiplier field. (Note that a multiplier of "1" has no affect on the word value.)

NOTE: Only available when **Type = Macro** or **Type = Conditional**)

Composite Format — Specifies how to break up, or "parse" **Composite-Numeric** value types. Enter one or more numbers separated by spaces to specify the quantity of parsed number values and the significant places of each. An asterisk "*" can be used as a wild card entry. Each parsed value appears in the word pick list of the **Configuration menu > Word/Address** function to perform individual actions.

EXAMPLE:

Parsing "T" word/values:

Sample data block	Composite Format	Parsed values
T0203	2 2	T=203, T1=02, T2=03
T102	1 2	T=102, T1=1, T2=02

T0304 or T304	* 2	T=304, T1=03, T2=04
T10001	* 2	T=10001, T1=100, T2=01
T102	1 1 1	T=102, T1=1, T2=0, T3=2
T12345678	2 * 2	T=12345678, T1=12, T2=3456, T3=78
T102.3	* 2.1	T=102.3, T1=1, T2=02, T3=3
T102.3	2.1	T=102.3, T1=10, T2=3
T12345678.321	2 2.2	T=12345678.321, T1=12, T2=34, T3=32

Add — Adds a record to the Word Table enabling you to add words or special characters to the control configuration.

Delete — Deletes the highlighted word record from the control configuration.

NOTE: Always reset VERICUT (press  (**Reset Model**) on the VERICUT main window) after making changes to the control configuration.

Shortcut: You can right-click in the Word Table to display a menu containing **Add** and **Delete**. These provide the same functionality described above.

Also see "**Building NC Controls**", in the *Using VERICUT* section, in the *CGTech Help Library*.

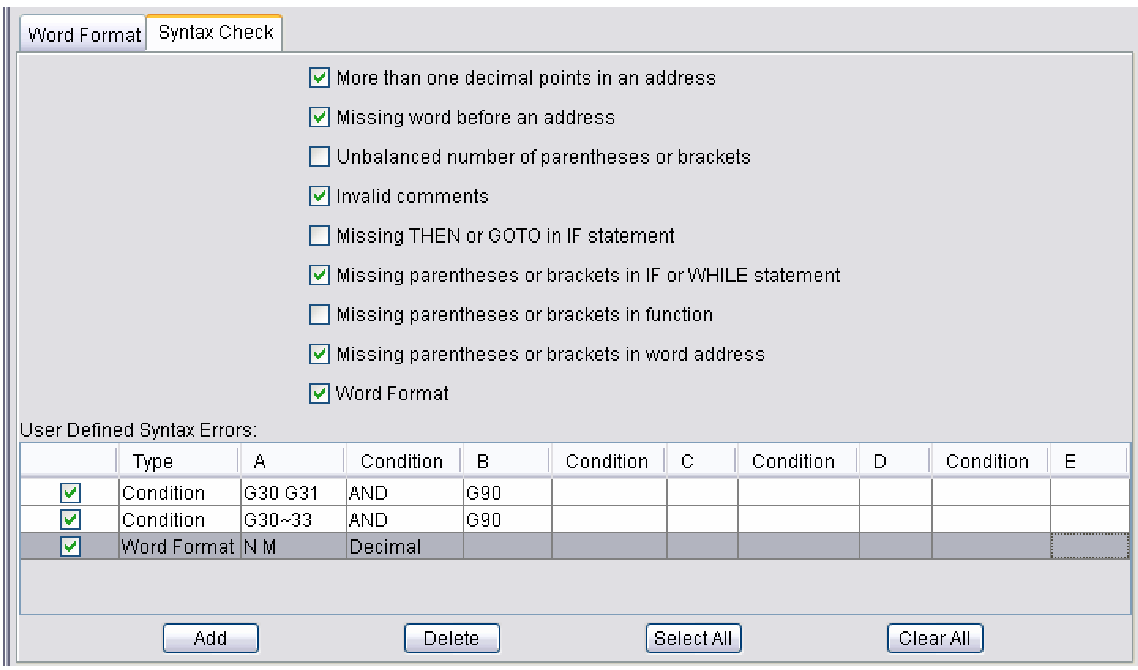
To Word Format window

Word Format window, Syntax Check tab

The features on this tab enable you to turn On/Off VERICUT defined syntax checking rules (error conditions), and to define custom syntax checking error conditions for checking the statements in your G-Code NC program file for valid syntax. The syntax rules and settings are stored in the control file.

NOTE: This feature is not applicable to APT NC programs.

The actual syntax checking of an NC program file is started in the NC Program window. See **NC Program (Info)**, in the Info menu section of *VERICUT Help* for more information.



VERICUT Defined Syntax Errors

The following error conditions are provided with VERICUT. Simply toggle **On** (checkmark displayed) the syntax error conditions that you want checked.

More than one decimal point in an address — Outputs an error when more than one decimal point is found in an address.

Missing word before an address — Outputs an error when a necessary word is missing before an address.

Unbalanced number of Parentheses or brackets — Outputs an error when the statement contains more "opening" characters (i.e. **(**, or **[**) than "closing" characters (i.e. **)**, or **]**).

Invalid comments — Outputs an error when invalid comment statements are found, for example, a comment in the middle of a block.

Missing THEN or GOTO in IF statement — Outputs an error when an IF statement is missing a THEN, or GOTO, statement.

Missing parentheses or brackets in If or While statement — Outputs an error when an IF, or While, statement is missing necessary parentheses, or bracket, characters.

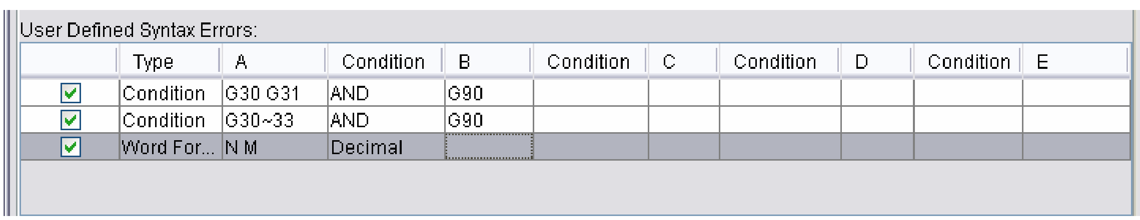
Missing parentheses or brackets in function — Outputs an error when necessary parentheses, or bracket, characters are missing in a Function (or Conditional).

Missing parentheses or brackets in word address — Outputs an error when necessary parentheses, or brackets, characters are missing in a word address.

Word Format — Outputs an error when the word format is incorrect.

User Defined Syntax Errors

User Defined Syntax Error Record Table



User Defined Syntax Errors:

	Type	A	Condition	B	Condition	C	Condition	D	Condition	E
☑	Condition	G30 G31	AND	G90						
☑	Condition	G30~33	AND	G90						
☑	Word For...	N M	Decimal							

Each record in the table contains the following features:

Check box — Each record has a checkbox enabling you to toggle the record **On**/**Off**.

Type — You can define the following two types of User Defined syntax errors

Condition — Enables you to specific conditions that will be considered an error.

A, B, C, D, E — Each of these fields represent a word, or group of words, used with the **Condition** fields to define the error condition. Each field is a text field that uses a space to separate words. The "space" is interpret as an "**OR**" condition.

For example, "G30 G31" is interpret as "G30 or G31"

The tilda character, "**~**", is used to define a range of numbers/values.

For example, "G30~33" is interpret as "G30 or G31 or G32 or G33".

It can be used to check for mutual exclusive groups.

For example: "G30~39" AND "G30~39" means you can only have a G3* code in the same line, otherwise it's an error.

It can also be used to check the limits.

For example "X-6~5" will find all X values between -6 to 5.

Condition — Use each of the **Condition** fields to specify the condition between the words, or groups of words, in the adjacent columns (i.e, "A" NOT "B"), that together define the error condition. Select the desired condition from the pull-down list. Choose "**blank**", (interpret as an "**OR**" condition), **AND**, or **NOT**.

For example, "G30 G31" AND "G90" means if there is a "G30" or a "G31" and there is a "G90" on the same line, it's an error.

Word Format — Enables you to check the numeric format of a word's address.

A — Use this column to specify the word, or words, used with the **Condition** field to define the error condition. Use a space to separate the words in this field.

Condition — Use this field to specify the error condition for the word, or words specified in column **A** that define the error condition. Select the desired condition from the pull-down list. Choose **Decimal** or **Integer**.

For example, "M N" Decimal means Word M and Word N must followed by an integer, otherwise it's an error.

Add — Adds a User Defined Syntax Error record, after the highlighted record, to the table.

Delete — Deletes the highlighted User Defined Syntax Error record from the table.

Select All — Toggles On ***all***, both VERICUT defined and User Defined syntax error conditions.

Clear All — Toggles Off ***all***, both VERICUT defined and User Defined, syntax error conditions.

To Word Format window

Conditionals - listed alphabetically

The conditionals which can be referenced via **Configuration menu > Word Format** are listed below in alphabetical order. Conditionals are passed the text string of the current block, up to and including the current word, and returns a word type specifying how the current word is to be interpreted (typically identified by the characters "Cond" in the function name)

Conditionals

AbCondEqual

This function determines if the "=" should be interpreted as an assignment or as a logical operator. If the "=" is part of an IF or GTO Type II comand, it will be defined as a logical operator. Otherwise, it will be determined as an assignment operator.

AbCondLeftParen

Conditional function that looks at the next two characters to determine if it is a known "TYPE II" command. If the following 2 characters are: AB, AP, AT, CC, CL, CN, CS, CY, FA, KB, MD, MI, MT, NS, QU, RN, TA, TB, TC, TS, TT, or ZT, then the left paren is interpreted as the beginning of a type II command. Otherwise it is interpreted as a comment.

AbCondRightParen

If a comment is currently being processed, then the right paren is interpreted as an "END COMMENT" special control word. Otherwise it is interpreted as an "END TYPEII" word.

AtanCondDivide

AtanCondDivide(block_str)
If the "/" symbol is part of an ATAN argument and is not imbedded within brackets ([ ]), then this function returns the type SEPARATOR. Otherwise this function returns the type DIVIDE.

CinciCondEqual

If G10 exists in the block, ignore the current word. If G11 exists in the block, interpret the current word as a logical EQUAL. Otherwise, interpret the current word as an assignment.

CinciCondLeftBracket
If the previous character was "T", ignore the current word. Otherwise, interpret the current word as a LEFT PRECEDENCE. Requires values to be initialized by the **CinciBlockInit** macro.

CinciCondLeftParen

Returns "Begin Type II" if a Type II command follows. Otherwise returns "Left Precedence". Requires values to be initialized by the **CinciBlockInit** macro.

CinciCondRightBracket

The current word will be either ignored or interpreted as a RIGHT PRECEDENCE depending on how the matching **CinciCondLeftBracket** was interpreted. Requires values to be initialized by the **CinciBlockInit** macro.

CinciCondRightParen

Returns "End Type II" if currently processing a Type II command. Otherwise returns "Right Precedence". Requires values to be initialized by the **CinciBlockInit** macro.

CinciCondTWord

If the current word was preceded by a "[", interpret the current word as a VARIABLE TAG. Otherwise, interpret the current word as a MACRO.

CinciCondVWord

Returns "Assignment" if G10 is in the block. Otherwise returns "Command".

CycleCondPWord

Sets P_1 (default) or P_2 after checking the context of the block containing the P-word. P_2 is set when the block contains a G76 and either X, Y, U or W words.

CycleCondQWord

Sets Q_1 (default) or Q_2 after checking the context of the block containing the Q-word. Q_2 is set when the block contains a G76 and either X, Y, U or W words.

CycleCondRWord

Sets R_1 (default) or R_2 after checking the context of the block containing the R-word. R_2 is set when the block contains a G74, G75, G76 and either X, Y, U or W words.

CycleCondUWord

Sets U_1 (default) or U_2 after checking the context of the block containing the U-word. U_2 is set when the block contains a G71, G72, G73 and either P or Q words.

CycleCondWWord

Sets W_1 (default) or W_2 after checking the context of the block containing the W-word. W_2 is set when the block contains a G71, G72, G73 and either P or Q words.

FadalCondAsteriskWord

FadalCondAsteriskWord(block_str)
If the Asterisk Word comes after a "#" character, then this function returns the type MULTIPLY. Otherwise this function returns the type COMMENT.

FadalCondEqualWord

FadalCondEqualWord(block_str)
If the Equal Word comes after the string "IF", then this function returns the type EQ . Otherwise, this function returns the type ASSIGNMENT.

FadalCondLParenWord

FadalCondLParenWord(block_str)

If the Left Parenthesis Word comes after a "#" character, then this function returns the type LPAREN. Otherwise this function returns the type COMMENT.

FadalCondNWord

FadalCondNWord(block_str)
If the N Word comes after a "#" character, then this function returns the type IGNORE. Otherwise this function returns the type COMMAND.

FadalCondRParenWord

FadalCondRParenWord(block_str)
If the Right Parenthesis Word comes after a "#" character, then this function returns the type RPAREN. Otherwise this function returns the type END_COMMENT.

FadalCondRWord

FadalCondRWord(block_str)
If the R Word comes after a "G" character, then this function returns the type COMMAND. Otherwise this function returns the type VARIABLE_TAG.

GeminiCondEqual
This conditional determines if the "=" should be interpreted as an assignment or as a logical operator. If the "=" is part of an IFT, it will be defined as a logical operator. Otherwise, it will be determined as an assignment operator.

GLCondPWord

GLCondPWord(block_str)
If the P word is part of a PDO command, in CYCLE mode, or if the next non-space character on the block is a "=", then this functions returns the type COMMAND. Otherwise this function returns the type VARIABLE_TAG.

GLCondPWord2

GLCondPWord2(block_str)
The P word can either be part of a COMMAND, or a VARIABLE_TAG. If P is part of a VALUE argument of a TYPE II command, or is preceded by =, +, -, *, /, [, or (, then the P will be interpreted as a VARIABLE TAG. If an integer value does not follow the P, then the P will be interpreted as a COMMAND. If the integer number is followed by a =, +, -, *, /,], or), then the P will be interpreted as a VARIABLE TAG. Otherwise the P will be interpreted as a COMMAND. This conditional function is an alternative to **GLCondPWord**. You can use which ever function produces the correct results for you.

HeidCondDivide

- Conditional function which determines how a "/" character is to be interpreted.
- If a "REP" precedes the "/" on the line, then it will be interpreted BEGIN COMMENT.
- If a "CYCL DEF 14.1" precedes the "/" on the line, it is recognized as a list separator used between contour subroutines. See "**Notes about simulating Heidenhain SL pocket cycles**" in the *Using VERICUT* section for additional information. *Using VERICUT* can be found in the *VERICUT Help Library*.
- Otherwise it will be interpreted as DIVIDE.

HeidCondLeftParen

This conditional is designed for Heidenhain MillPlus controls, where standard comments are preceded by a semicolon, but it is also permissible to use (*comment*) at the end of a block. It is used to determine when the "(" word is used as a left precedence for mathematical expression, and when it is used as a start of comment character.

HeidCondMultiply

This conditional is designed for the Heidenhain 530i control where the End Of Block (EOB) character is "*". It is used to determine when the "*" word is used as a mathematical multiply operator, and when it is used as an EOB character. If the "*" is only followed by white space characters (blank space, tab, etc.), it is interpret as an EOB character.

HeidCondRightParen

This conditional is designed for Heidenhain MillPlus controls, where standard comments are preceded by a semicolon, but it is also permissible to use (*comment*) at the end of a block. It is used to determine when the ")" word is used as a right precedence for mathematical expressions, and when it is used as an end of comment character.

HeidCondQWord

If the Q word comes after a D0 or a D1 string and prior to a P0 string, then this conditional function returns the type COMMAND. Otherwise this conditional function returns the type VARIABLE TAG

KtCondLeftParen

KtCondRightParen

KtCondLeftParen(), KtCondRightParen()
These functions support the "(" and ")" words when used as mathematical precedence and as Begin Type 2 data.

NumCondAWord

NumCondAWord(block_str)
If the non-white space character immediately preceding the A Word is a "=", "<" , ">", "+", "-", "*", "/", "&", or "!", it is interpret as type FUNCTION, and sets the word to mathematical function ATAN. Otherwise the word is interpreted as a COMMAND (or MACRO).

NumCondCWord

NumCondCWord(block_str)
If the non-white space character immediately preceding the C Word is a "=", "<" , ">", "+", "-", "*", "/", "&", or "!", it is interpret as type FUNCTION, and sets the word to mathematical function COS. Otherwise the word is interpreted as a COMMAND (or MACRO).

NumCondDollarSignWord

NumCondDollarSignWord(block_str)
If the Dollar Sign Word comes after an "=", and the previous non-space character is a '+', '-', '*', '/', '=', '<', '>, or a '[', then this function returns the type FUNCTION, and sets the word to NUM_INPUT. Otherwise this function returns the type CONSOLE_MSG.

NumCondEqualWord

NumCondEqualWord(block_str)
If the Equal Word comes after the string "G79", or if the Equal Word comes after the string "IF" and before the word "THEN", then this function returns the type EQ. Otherwise, this function returns the type ASSIGNMENT.

NumCondEWord

NumCondEWord(block_str)
If the E Word comes after a number, or if it begins the block , then this function returns the type VARIABLE_TAG. Otherwise this function returns the type FUNCTIONS, and sets the word to E_FUNC.

NumCondRWord

NumCondRWord(block_str)
If the non-white space character immediately preceding the R Word is a "=", "<" , ">", "+", "-", "*", "/", "&", or "!", it is interpret as type FUNCTION, and sets the word to mathematical function SQRT. Otherwise the word is interpreted as a COMMAND (or MACRO).

NumCondSWord

NumCondSWord(block_str)
If the non-white space character immediately preceding the S Word is a "=", "<" , ">", "+", "-", "*", "/", "&", or "!", it is interpret as type FUNCTION, and sets the word to mathematical function SIN. Otherwise the word is interpreted as a COMMAND (or MACRO).

NumCondTWord

NumCondTWord(block_str)
If the non-white space character immediately preceding the T Word is a "=", "<" , ">", "+", "-", "*", "/", "&", or "!", it is interpret as type FUNCTION, and sets the word to FIX. Otherwise the word is interpreted as a COMMAND (or MACRO).

OkumaCondNWord

OkumaCondNWord(block_str)
This function internally redefines the "N" word during parsing depending upon how it is being used:

- If the next non-space character is numeric and the string "IF" or "GOTO" existed previously on the line, then this routine returns the type CONDITIONAL, and sets the word to N_GOTO_LABEL.

- If the next non-space character is numeric and the string "IF" or "GOTO" does not previously exist on the line, then this routine returns the type CONDITIONAL, and sets the word to N_LABEL.
- If the next non-space character is not numeric, and the string "IF" or "GOTO" existed previously on the line, then this routine returns the type CONDITIONAL, and sets the word to N_GOTO_SEQ.

For this functionality to work, the following words must be defined in the **Configuration menu > Word Format window**:

- Name=***N***, Type=Conditional, Function Name=**OkumaCondNWord**
- Name=***N_GOTO_LABEL***, Type=Macro, Value Type=Alpha-Numeric
- Name=***N_GOTO_SEQ***, Type=Macro, Value Type=Numeric, Inch Method/Metric Method=Decimal
- Name=***N_LABEL***, Type=Macro, Value Type=Alpha-Numeric

Then, the **Configuration menu > Word/Address** function must also be configured to process the above-listed words, for example in the "Specials" class:

- Word=***N*** Range=*, Condition=* *, Macroname=**Sequence**, Scan=Yes, After=No
- Word=***N_GOTO_LABEL*** Range=*, Condition=* *, Macroname=**LabelName** & **GotoLabel**, Scan=No, After=No
- Word=***N_GOTO_SEQ*** Range=*, Condition=* *, Macroname=**GotoJump**, Scan=No, After=No
- Word=***N_LABEL*** Range=*, Condition=* *, Macroname=**LabelMacro**, Scan=Yes, After=No

OliCondLeftParent

OliCondLeftParent(block_str)
If the Left Parenthesis Word comes after a "=" character, then this function returns the type LPAREN. Otherwise this function returns the type BEGIN_TYPEII.

OliCondRightParent

OliCondRightParent(block_str)
If the Right Parenthesis Word comes after a BEGIN_TYPEII word, then this function returns the type END_TYPEII. Otherwise this function returns the type RPAREN.

Siemens3RCond

Siemens3RCond(block_str)
Same as SiemensRCond(block_str), except it also looks to see if the @ command had finish prior to this point in the block. It does this by looking backwards from the current point, and checking to see if a non-operator, non-R, non-digit character exists between the current point and the @ command. When the R word is used as a command, it typically calls the VariableArgument macro. This sets up the addition arguments that are then processed by the specific Siemens macro.

SiemensCondLParenWord

SiemensCondLParenWord()
If MCALL is on the line, the word type is set to Left Precedence, otherwise it is set to Begin Comment.

SiemensCondRParenWord

SiemensCondRParenWord()
If MCALL is on the line, the word type is set to Right Precedence, otherwise it is set to End Comment.

SiemensRCond

SiemensRCond(block_str)
If the R Word comes after a "@" character, then this function returns the type COMMAND. Otherwise this function returns the type VARIABLE_TAG.

TosnucCondVWord

If the current word is within square brackets ("[" and "]"), interpret the current word as a VARIABLE TAG. Otherwise, interpret the current word as a MACRO.

Functions - listed alphabetically

The functions which can be referenced via **Configuration menu > Word Format** are listed below in alphabetical order. Functions are typically passed a single numerical value, and return a numerical value.

Functions

abs

abs(value)
Returns the absolute value of the specified value.

AbsCeil

Rounds the specified value to the next higher absolute integer value. For example: 2.3 would be rounded to 3, and -2.3 would be rounded to -3.

AbsoluteDimension

This new functions will handle formats like: X=AC[2] Y=IC[3] Z=AC[2] I=AC[1] J=AC[2] where "[" and "]" are the left and right precedence words. In the above format, AC and IC should be defined as words which then call the function.

If the primary word is X,Y,Z,A,B,C,U,V,or W then:
AbsoluteDimension will cause the specified value to be interpreted as absolute, and if the control is currently in incremental mode, it will convert it.
If the primary word is I,J, or K then:
AbsoluteDimension will use the absolute mode, the "Circle Center Mode" settings, and the "Incremental Circle Center Method" settings to determine how to convert the value.
If the primary word is anything other then the above, no conversion will be executed, and the incoming value will be returned.

Also see: **IncrementalDimension**

acos_d

acos_d(value)
Returns the arc cosine 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 0.0 to 180.0.

acos_r

acos_r(value)
Returns the arc cosine of the specified value. The specified value must be in the range of -1.0 to 1.0. The return value is in radians, and is in the range of 0 to pi.

asin_d

asin_d(value)
Returns the arc sine 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 -90.0 to 90.0.

asin_r

asin_r(value)
Returns the arc sine of the specified value. The specified value must be in the range of -1.0 to 1.0. The return value is in radians, and is in the range of -pi/2 to pi/2.

atan2_d

atan2_d(yvalue, xvalue)
Returns the arc tangent of yvalue/xvalue. The return value is in degrees, and is in the range of -180.0 to 180.0. If yvalue and xvalue are both zero, the return value will be zero.

atan2_r

atan2_r(yvalue, xvalue)
Returns the arc tangent of yvalue/xvalue. The return value is in radians, and is in the range of -pi to pi. If yvalue and xvalue are both zero, the return value will be zero.

atan_d

atan_d(value)
Returns the arc tangent 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 -90.0 to 90.0.

atan_r

atan_r(value)
Returns the arc tangent of the specified value. The specified value must be in the range of -1.0 to 1.0. The return value is in radians, and is in the range of -pi/2 to pi/2.

ceil

ceil(value)
Returns the least integral value greater than or equal to the specified value.

cosh_d

cosh_d(angle)
Returns the hyperbolic cosine of the given angle specified in degrees.

cosh_r

cosh_r(angle)
Returns the hyperbolic cosine of the given angle specified in radians.

cos_d

cos_d(angle)
Returns the cosine of the given angle specified in degrees.

cos_r

cos_r(angle)
Returns the cosine of the given angle specified in radians.

exp

exp(power)
Returns the natural logarithm "e" raised to the specified power.

floor

floor(value)
Returns the greatest integral value less than or equal to the specified value.

Fractional

Fractional(value)
Returns the fractional portion of the specified value.

get_current_N_seqno

get_current_N_seqno()
Returns value of the current N word. If the N word was not specified on the current line, the value of the last specified N word is returned.

IncrementalDimension

This new function will handle formats like: X=AC[2] Y=IC[3] Z=AC[2] I=AC[1] J=AC[2] where "[" and "]" are the left and right precedence words. In the above format, AC and IC should be defined as words which then call the function.

If the primary word is X,Y,Z,A,B,C,U,V,or W then:

IncrementalDimension will cause the specified value to be interpreted as incremental, and if the control is currently in absolute mode, it will convert it.

If the primary word is I,J, or K then:

IncrementalDimension will use the incremental mode, the "Circle Center Mode" settings, and the "Incremental Circle Center Method" settings to determine how to convert the value.

If the primary word is anything other then the above, no conversion will be executed, and the incoming value will be returned.

Also see: **AbsoluteDimension**

IntTrunc

IntTrunc(value)
Returns the integer portion of the specified value.

IntTruncAdj

This function is identical to **IntTrunc**, except that it automatically adjusts for computer round off errors. The following are the results you can expect:

#1 = TRUNC[4.1]	(ANSWER: 4)
#2 = TRUNC[-4.1]	(ANSWER: -4)
#3 = TRUNC[4.1 - 0.1]	(ANSWER: 3)
#4 = TRUNC[-4.1 + 0.1]	(ANSWER: -3)
#5 = TRUNC_ADJ[4.1]	(ANSWER: 4)
#6 = TRUNC_ADJ[-4.1]	(ANSWER: -4)
#7 = TRUNC_ADJ[4.1 - 0.1]	(ANSWER: 4)
#8 = TRUNC_ADJ[-4.1 + 0.1]	(ANSWER: -4)

ISNUMBER

ISNUMBER(string)

Checks if string parameter is the valid representation of a real number. Returns 1 if the string represents a number, or 0 if not.

See also function NUMBER.

ln

ln(value)

Returns the natural logarithm of the specified value. The specified value must be positive otherwise zero is returned.

log

log(value)
Returns the base 10 logarithm of the specified value. The specified value must be positive otherwise zero is returned.

NumAtan

NumAtan(value)
Similar to atan2_d(yvalue, xvalue), except the return value is in thousandths of a degree.

NUMBER

NUMBER(string)

Converts the input string into a real number and returns its value. "String" must represent a valid real number. Scientific notification with exponent is supported. Returns 0 if "string" does not represent a valid real number.

See also function ISNUMBER.

NumEFunc

NumEFunc(value)
If the variable is being assigned to a COMMAND word, a "NUM" specific factor is applied to the value of the variable.

NumInput

NumInput()
Causes the program to prompt the user for input. If the first character of input is a-z or A-Z, a value of 1-26 is returned. If the user enters a numeric value, the value entered will be returned.

Round

Round(value)
Returns the integral value closest to the specified value.

SiemensAxName

Passed the name of an AXIS, and returns the AXIS (an integer value).

SiemensAxString

Passed an AXIS value, and returns the AXIS name (a string value).

SiemensCFINE

Use to configure the control to use the Siemens Frame setting function CFINE. Function CFINE is used to set fine values of translation for specified axes in Frame.

See **Notes about Siemens 840D DEF Command**, in the *Notes about Special Topics* section in the **CGTech Help Library** for additional information.

SiemensCMIRROR

Use to configure the control to use the Siemens Frame setting function CMIRROR. Function CMIRROR is used to set a mirrored (reverse) orientation for specified axes in Frame.

See **Notes about Siemens 840D DEF Command**, in the *Notes about Special Topics* section in the **CGTech Help Library** for additional information.

Siemens CROT

Use to configure the control to use the Siemens Frame setting function CROT. Function CROT is used to set rotation angle values around specified axes in Frame.

See **Notes about Siemens 840D DEF Command**, in the *Notes about Special Topics* section in the **CGTech Help Library** for additional information.

SiemensCSALE

Use to configure the control to use the Siemens Frame setting function CSALE. Function **CSCALE** is used to set scale values of specified axes in Frame.

See **Notes about Siemens 840D DEF Command**, in the *Notes about Special Topics* section in the **CGTech Help Library** for additional information.

SiemensCTRANS

Use to configure the control to use the Siemens Frame setting function CTRANS. Function **CTRANS** is used to set translation values for specified axes in Frame.

See **Notes about Siemens 840D DEF Command**, in the *Notes about Special Topics* section in the **CGTech Help Library** for additional information.

SiemensMEAFRAME

Use to configure the control to use the Siemens Frame setting function MEAFRAME. Function **MEAFRAME** is used to create Frame based on difference between the actual measured location of 3 points on the machine and the assumed theoretical location of the three points.

See **Notes about Siemens 840D DEF Command**, in the *Notes about Special Topics* section in the **CGTech Help Library** for additional information.

sinh_d

sinh_d(value)
Returns the hyperbolic cosine of the given angle specified in degrees.

sinh_r

sinh_r(value)
Returns the hyperbolic cosine of the given angle specified in radians.

sin_d

sin_d(value)
Returns the cosine of the given angle specified in degrees.

sin_r

sin_r(value)
Returns the cosine of the given angle specified in radians.

Sqr

sqr(value)
Returns the square of the specified value. This function can be used with the word list to process statements like: #1 = SQR(#2 + #3)

sqrt

sqrt(value)
Returns the square root of the specified value.

tanh_d

tanh_d(angle)
Returns the hyperbolic tangent of the given angle specified in degrees.

tanh_r

tanh_r(angle)
Returns the hyperbolic tangent of the given angle specified in radians.

tan_d

tan_d(angle)
Returns the tangent of the given angle specified in degrees.

tan_r

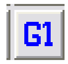
tan_r(angle)
Returns the tangent of the given angle specified in radians.

ToolnumToPocket

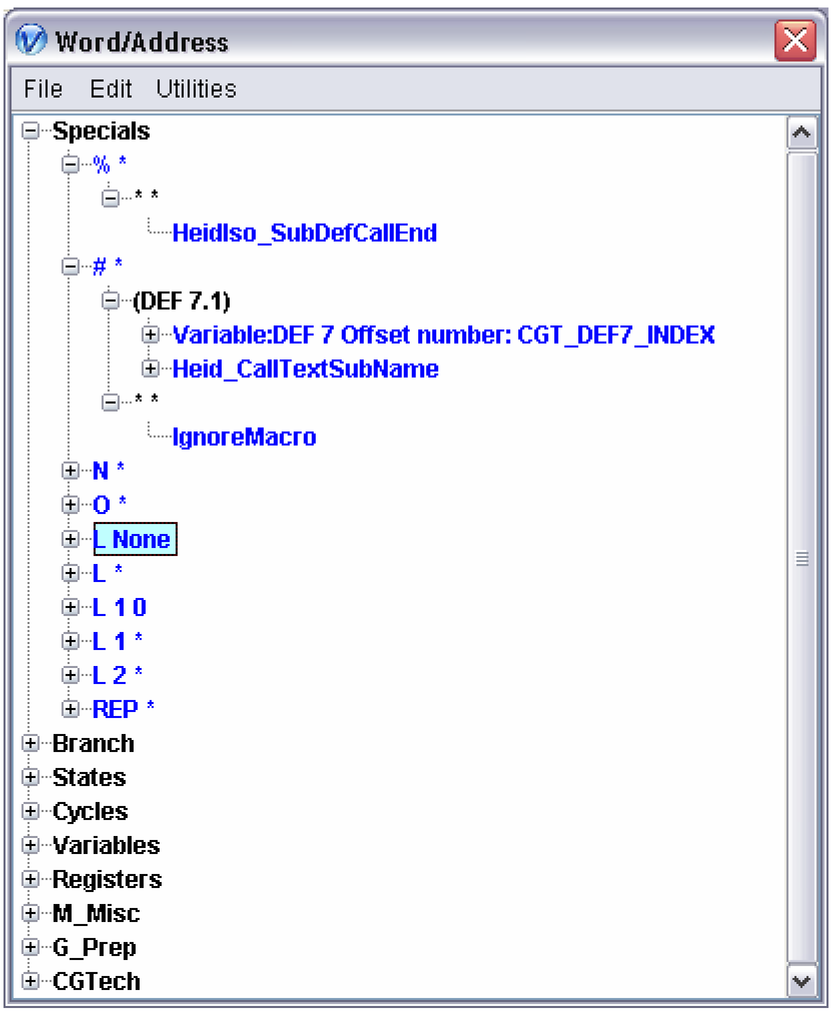
Returns the pocket number given the tool number. Specifically, it searches the tool list for the first matching cutter tool id, and returns the corresponding pocket number. If no list exist, or if the list is not based on pocket numbers, or if no matching entry can be found, the function will return the tool number.

Word Address (Word/Address window)

Location: **Configuration menu > Word/Address**

VERICUT toolbar short cut: 

Opens a window to "group" G-Code words previously defined via **Configuration menu > Word Format** with a range of address values, and associate them with action macros to simulate an action in the control, or on the machine. For simplicity, we'll call each of these address/value associations a "group". Each group can be configured to call one or more action macros. This functionality allows you to virtually "wire" a custom NC control based on machine code information present in the tool path file. Groups with similar functions are arranged into a "class". The listed order of classes in the Word/Address window helps determine the timing of when corresponding actions occur.



The Word/Address window consists of a "menu bar" and a "tree structure" representing the configuration of a particular NC control.

The Menu Bar

File

Open — Opens a window enabling you to open (load) a control file. Provides the same function as **Configuration menu > Control > Open**.

Save — Saves (updates) an existing Control file with the current NC control settings. Provides the same function as **Configuration menu > Control > Save**.

Save As — Opens a window enabling you to save the current control file under a different name or path. Provides the same function as **Configuration menu > Control > Save As**.

Close — Closes the Word Address window without saving the control file.

Edit

Add/Modify — Opens the **Add/Modify Word/Address window** enabling you to add or modify groups within the control configuration.

Cut — Enables you to "cut" the object highlighted in the control configuration.

Copy — Enables you to "copy" the object highlighted in the control configuration.

Paste — Enables you to "paste" a previously "cut" or "copied" object after the object highlighted in the control configuration.

NOTE: When you "**Cut**", "**Copy**", or "**Paste**" an object in the "tree", all objects below it in that particular "branch" (children) are cut, copied, or pasted with it.

Utilities

Find — Opens the **Find Word window** enabling you to find the groups in a control configuration that are associated with a specific word/address value, or call specific macros.

Validate — Opens the **Word/Address Validate window** enabling you to validate the control configuration.

The Control Configuration

Object Hierarchy

The illustration above shows an expanded "group" from the configuration "tree". The top level object, "Specials", represents a "class". The second level object (or branch), "**Variable: 3000 ***", represents "Word/Variable and Range", in this case Variable and Range. The third level object (branch), "******", represents a "conditional Word/State/Variable and conditional Range/Value ". The lowest level objects (branches), "**AlarmSignal**" and "**EndProgram**", represent Macro/Variable Names with associated data. Each lower level (or branch) is considered to be "a child or children" of the next higher level.

Configuration "tree"

The control configuration of a particular NC control is displayed as a "tree" structure. When you expand a "class" object, the "Word/Range" branch shows the word/address of the groups within the class. Expand a "Word/Range" object and the "Condition" branch shows conditions defined for the selected group. Finally, expanding a "Condition" object, displays the "Macro name" branch which shows the macros associated with the selected condition in the group.

To summarize, expand a "class" to display the "groups" associated with it. Expand a "group" to display the different "conditions" associated with it. Similarly, expand a "condition" to display the macros/variables associated with the condition.

Change the order of class, group, condition and macro objects in the control configuration by "left clicking" on an object and "dragging" it to the desired position. When you reposition an object, all of the objects below it in the branch (children) are also repositioned.

Shortcut: "Right click" in the Control Configuration window to display a shortcut menu with the following features:

Add/Modify, Cut, Copy, Paste, and Find — These features provide the same functionality described above under Edit in the Menu Bar.

Contract All Children — Use to contract (remove the display) all "children" of the highlighted object.

Expand All Children — Use to expand (display) all "children" of the highlighted object.

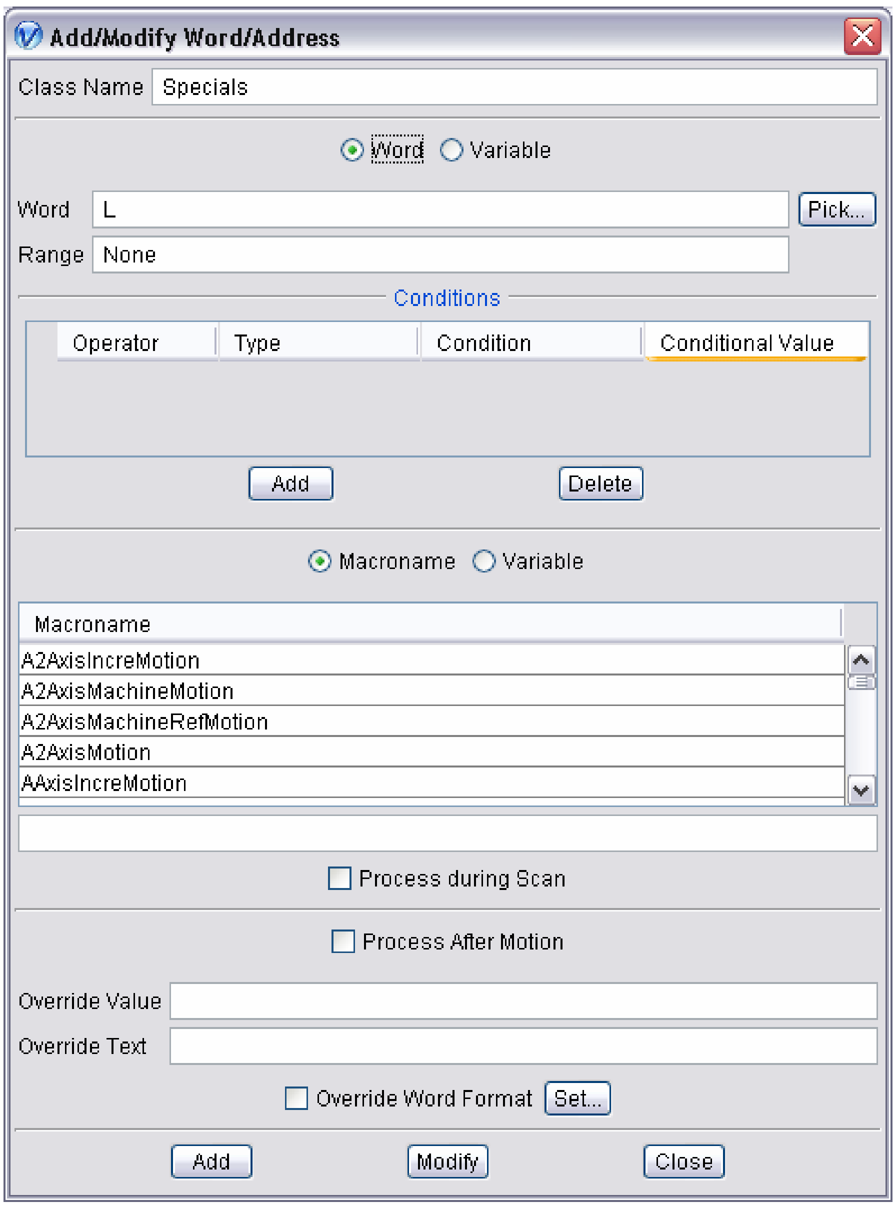
NOTES:

1. Always reset VERICUT (press  (**Reset Model**) on the VERICUT main window) after making changes to the control configuration.
2. Use the **Debug Macro Arguments** feature, on the **Project menu > Processing Options > G-Code > Process Options: Debug tab**, to output the macro name, the word, the text string value, and the numeric value for each macro as it is called.

Also see "**Building NC Controls**", in the *Using VERICUT* section, in the *CGTech Help Library*.

Add/Modify Word/Address window

Opened by pressing **Add/Modify** on the **Word/Address window**, the features on this window enable you to maintain groups/conditions in the control configuration. VERICUT will attempt to add a new group after the group selected in the list. A new group condition is added after the selected condition, and so on. However, if the word and range of an added group matches that of a previously defined group, the new group is automatically added as a condition to the previously defined group. This occurs even when the previously defined group resides in a different class.



Class Name — Enables you to add a new "class" or rename an existing "class".

Word / Variable — Controls the type of group being defined.

Word Options:

Word — These features define a group whose actions are performed when a G-Code having a corresponding word and address value is processed. Enter the group word in the text field. The word ***must*** be previously defined via the **Word Format** function. You can type the word character, or click on **Pick,** and then select it from the list of defined words that displays.

Range — Use the **Range** text field to specify a value, or range of values, that will cause the specified action to be processed. The Range value commonly specifies a single address. However, you can also specify multiple range values (separated by spaces or commas), inclusive ranges, or use "*****" as a wildcard to denote "all values". The following shows all valid Range value entries and their meanings:

Range Entry	Meaning
*****	Any value
value	A specific value like 7, or 9
7,9	A series of numbers
5-9	Integer values between 5 and 9 inclusive (5, 6, 7, 8, and 9)
5.0-9.0	All numbers between 5.0 and 9.0
#2	A variable. The variable ***must*** be prefixed with a **#**
$	The current value associated with the word. NOTE: This really only make sense with a "Conditional Value"
<n	Less than n, where n is a value, a variable (designated with the **#**), or a $
<=n	Less than or equal to n, where n is a value, a variable (designated with the **#**), or a $
>n	Greater than n, where n is a value, a variable (designated with the **#**), or a $

>=n	Greater than or equal to n, where n is a value, a variable (designated with the #), or a $
=n	Equal to n, where n is a value, a variable (designated with the #), or a $
#3-$	A range of values specified using a variable and a $
NONE	Supports processing words without values differently from the same words with values. For example, you can configure N010**X** to be handled differently from N010**X0**.

NOTE: Typically, Range values are integers. If you need to specify a group of NC codes, some of which contain decimal values (for example, G45, G45.1 G46) define the group Range as a series (45 45.1 46), rather than as an "inclusive" Range (45-46). Inclusive Ranges of real values may be defined by including a decimal point with the value. For example, the Range (45.0-46.0) will select all values between 45 and 46.

The following are typical examples of how **Word** and **Range** can be used to specify G-Codes acted on for a specific group "L".

Group Word	Range	G-Codes acted on by the group
L	2	L2
L	1 5	L1, L5
L	1 5-10	L1, L5, L6, L7, L8, L9, L10
L	NONE	L ("L" without a numeric value)
L	*	Ln (where "n" is any value)

Variable Options:

Variable — Use to define a group whose actions are performed when a specified variable is set to a corresponding value, or range of values. Enter the group variable name in the Variable text field.

Range — Use the **Range** text field to specify a value, or range of values, that will cause the specified action to be processed. Variable ranges are specified in the same manner as described above for Word ranges.

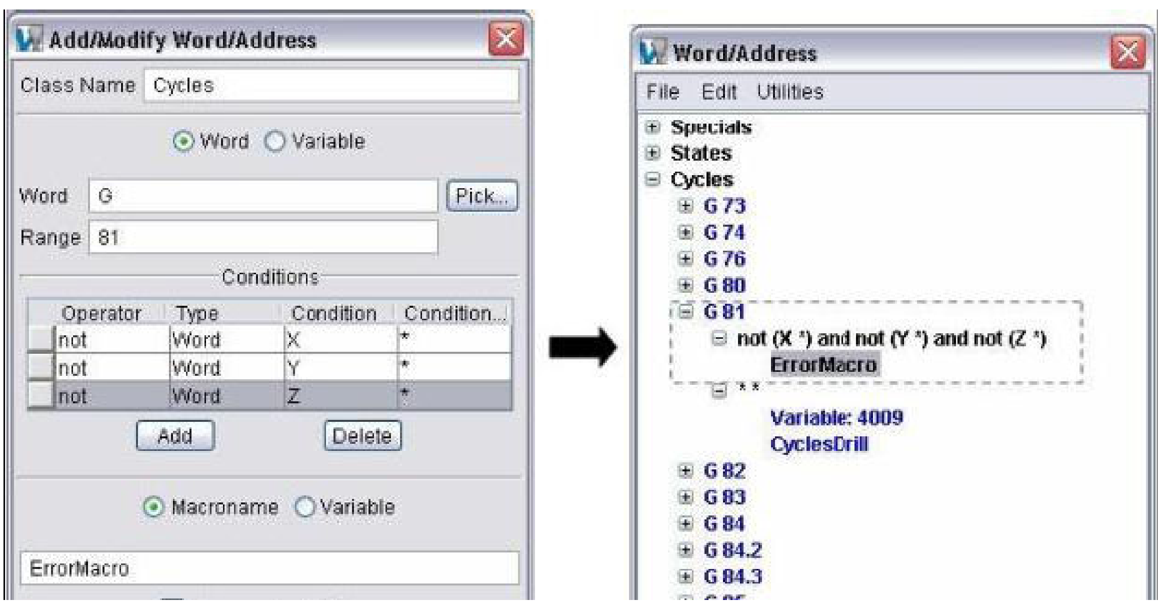
Conditions — Use to specify one or more conditions, which if met, causes the group to perform different action(s).

Operator — Choose **and** or **not** from the pull-down list.

Type — Use to specify the type of condition.

Word — When **Type** is set to **Word**, the **Condition** is based on another word and address value range appearing in the G-Code data block. This is the most common condition type. Select the desired **Condition** from the pull-down list, and then enter the **Conditional Value**(s). Conditional Values are specified in the same manner as described above for Word ranges.

For example: G 81 **not** (X *) and **not** (Y *) and **not** (Z *) calls **ErrorMacro**



State — When **Type** is set to **State**, the **Condition** is based on a machine state. Choose the desired **Condition** from the pull-down list, and then choose a value from the **Conditional Value** pull-down list. Notice that only values that are valid for the selected **Condition** appear in the **Conditional Value** list.

Variable — When **Type** is set to **Variable**, the **Condition** is based on when a variable is set to a corresponding **Conditional Value**. In the **Condition** column enter the variable name, and then enter the **Conditional Value**(s). Conditional Values are specified in the same manner as described above for Word ranges.

Add — Use to add a condition to the conditions list. The new condition will be added after the highlighted condition in the list.

NOTE: It is never necessary to have the list in any specific order since all items in the list must be TRUE for the corresponding macro to be executed. As a personal preference, you might want to read the list in a specific order. To this end, the new condition is always added after the highlighted condition in the list. You can also change the position of a condition in the list by clicking on the button, on the left side of each row, and drag the condition to the desired position in the list.

Delete — Use to delete the highlighted condition from the conditions list.

Macroname / Variable — Use to specify whether a group calls a macro, or sets a variable.

Macroname — Specify the macro to be called by selecting from the list or by typing the macro name in the text field. Text entered is not case sensitive. Use the automatic filtering capability to help you find macros. As you enter text characters in the **Macroname** text field, the list of macros is automatically filtered to show those that match the entered text.

Process during Scan — When active, calls the macro during a "scan pass" prior to NC program processing (ref. "**Scan NC Program Files**" on the **Project menu > Processing Options > G-Code > Settings: Settings tab**). The default condition is to call the macro during NC program processing.

Variable — Use the **Variable Name**, and **Variable Description** text fields to specify, and describe, the variable to be set. Use the **Override Value** text field, described below, to specify the variable value.

Process After Motion — When active, performs the group action after motion commands in the data block have been processed. The default condition is to perform the group action according to the rules of normal G-Code data processing. (Ref. "**About building NC controls**" in the *Using VERICUT* section. *Using VERICUT* can be found in the *VERICUT Help Library*.)

Override Value — This feature acts differently, depending on if **Macroname**, or **Variable**, is selected:

With **Macroname** *selected* **—** Used to specify a value to be passed to the specified macro. If blank, the address value accompanying the word is passed.

With **Variable** *selected* **—** Used to specify a value to be assigned to the specified variable.

Math expressions and variables, understood by the control, are supported. Enter "$" to specify using the address value accompanying the word. Enter "#", followed immediately by the variable name/number, followed by a space, to specify a variable value. Multiple variables can be added, separated by a blank space. If the override expression does not contain either of these characters, it is evaluated immediately and only the value is retained. If the expression can not be evaluated immediately, the full expression is retained in the control configuration and evaluated separately for each block that activates the group.

Also see "**Using Equations/Expression In VERICUT**" in the *Using VERICUT* section, in the *CGTech Help Library*, for more information.

Examples of using Override Value: (given that the variable, #2 = 3)

Text entered in Override Value field	What is retained in control configuration	Sample G-Code data block	Resulting value when processed
SIN(30)	.5	X5	.5 (constant value- not affected by G-Code data)
SIN(30)	.5	X10	.5 (see note above)
$*10	$*10	X5	50
($+#2)*10	($+#2)*10	X5	80

Override Text — Similar to **Override Value**, except that it is used to specify text to be passed to the macro. Only certain macros are designed to accept text values. Consult the "**Macros - listed alphabetically**" in the *VERICUT Macros* section, in the *CGTech Help Library*, for more information.

Expression(s) can be used in macro override text. If the override text contains the sequence "{*expression*}" , the expression text inside the curly-braces is evaluated, and then replaced with the calculated value.

The syntax for the expression(s) is the same as defined above for **Override Values**.

Examples:

1. If the **MessageMacro** is used with the following override text:

    Override Text = "The spindle speed is {#speed} rpm"

    The variable "#speed" is evaluated and if set to 500, the resulting message would be:

    "The spindle speed is 500 rpm"

2. $$ within {} enables you to include the incoming text string. For example:

    Override Text = Operation Name: {$$}

3. Text variables are also supported. For example: if the variable "NAME" was set to the operation name, you could have:

    Override Text = Operation Name: {#NAME}

If the contents within {} has either a "$$" or a text variable, then the expression will be evaluated as a text string. When dealing with text, it is recommended that you have only have 1 item within the {}. For example, if the variable "DATE" contained "May 2007", you could have:

Override Text = Operation Name: {$$}, {#DATE}. This syntax is preferable to:
Override Text = Operation Name: {$$: #DATE}

The output would be: Operation Name: 1-TOP0-FOAM-LICKA, May 2007

Also see "**Using Equations/Expression In VERICUT**" in the *Using VERICUT* section, in the *CGTech Help Library*, for more information.

Override Word Format — When toggled "on", the word format specified with the macro will be used rather than the default word format. Use the **Set** button, described below, to display the Override Word Format window, and then use its features to define the word format for the specific macro. For example, this feature would enable you to define a P with a G71 to be decimal, and a P with a G76 to be Trailing 3.4.

NOTE: This feature ***does not*** support expressions. For example, you could not use this feature with: X=54000 + 30000

Set — Used in conjunction with **Override Word Format**, described above. Displays the Override Word Format window enabling you to specify a word format for a specific macro.

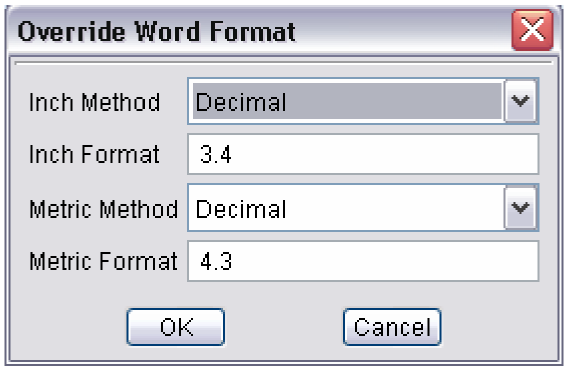
Add — Select when adding a new group or condition.

Modify — Select when modifying an existing group or condition.

Close — Close the Add/Modify Word/Address window.

Override Word Format window

Used in conjunction with the **Override Word Format** feature, on the Add/Modify Word/Address window, to specify a word format to be used with a specific macro rather than use the default word format. Use the **Set** button, on the Add/Modify Word/Address window, to display the Override Word Format window.



Inch Method / Metric Method — Specifies how to interpret inch (or metric) address values. This feature is significant only for values that do not contain a decimal point. Values with a decimal are always interpreted via the "Decimal" method.

Options:

Decimal — Interprets values as floating point decimals.

Leading or Decimal — Interprets values without a decimal as having leading zeros. Depending on the active unit system (inch or metric), values are interpreted as described by the corresponding "**Format**" (see below).

Trailing or Decimal — Similar to **Leading or Decimal**, except that values are interpreted as having trailing zeros.

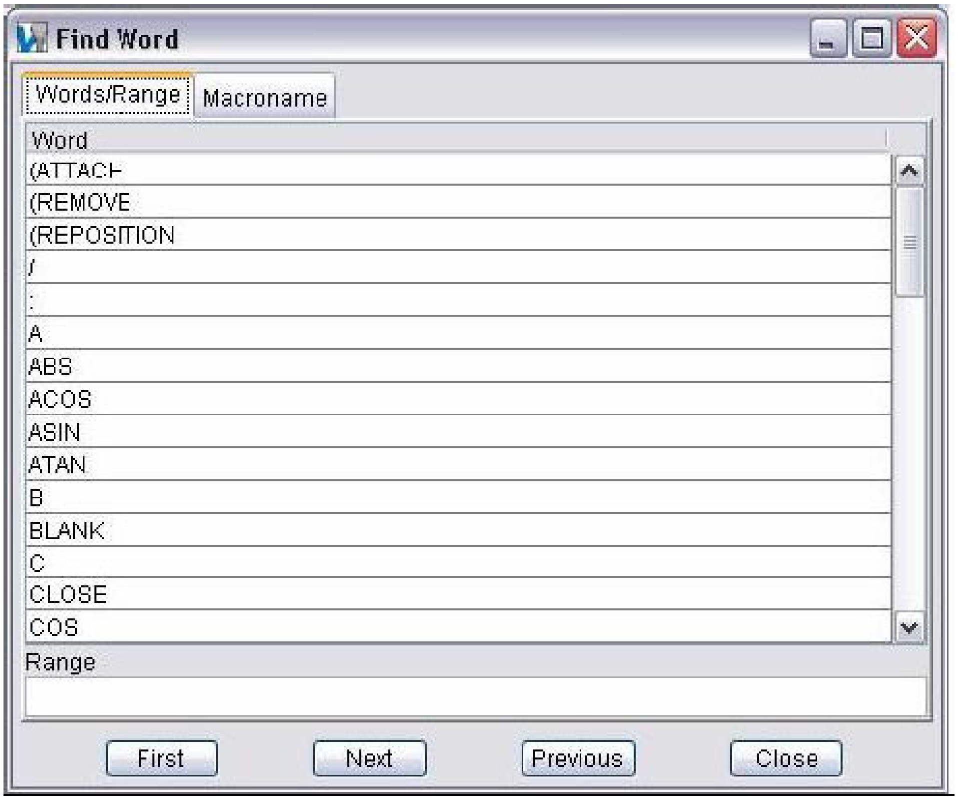
Inch Format / Metric Format — Specifies the number of digits before and after the decimal point when interpreting leading or trailing zero values. Data entry format is: a.b where "a" specifies the number of digits before, and "b" specifies the number of digits after the decimal point.

OK — Applies the settings and closes the Override Word Format window.

Cancel — Closes the Override Word Format window without accepting the settings.

Find Word window

Opened using **Utilities menu >Find** on the **Word/Address window**, the features on this window help you find the groups in a control configuration that are associated with a specific word/address value, or call specific macros.



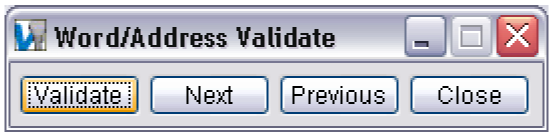
Words/Range tab — Enables you to find groups associated based on a specific word/address value. Pick the desired word from the **Word** list, and then enter the address value in the **Range** field.

Macroname tab — Enables you to find groups that call a specific macro. Type or pick the name of the desired macro from the **Macroname** list.

After configuring the window to find the desired group, use the buttons at the bottom of the window to find the **First**, **Next**, or **Previous** occurrence in the control configuration.

Word/Address Validate window

Opens the **Word/Address Validate window** enabling you to validate the control configuration.



Validate — Starts the validation process. VERICUT searches through the control checking for errors or conflicts within the Word/Address groups. If an error\conflict is detected, the validation process stops and the entity that has the problem is highlighted and a message is output to the message log area. If no errors/conflicts are found, the validation process continues until the end of the control configuration is reached.

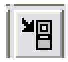
Next — Use to continue the validation process after an error/conflict is identified.

Previous — Use to go back to the previous error/conflict.

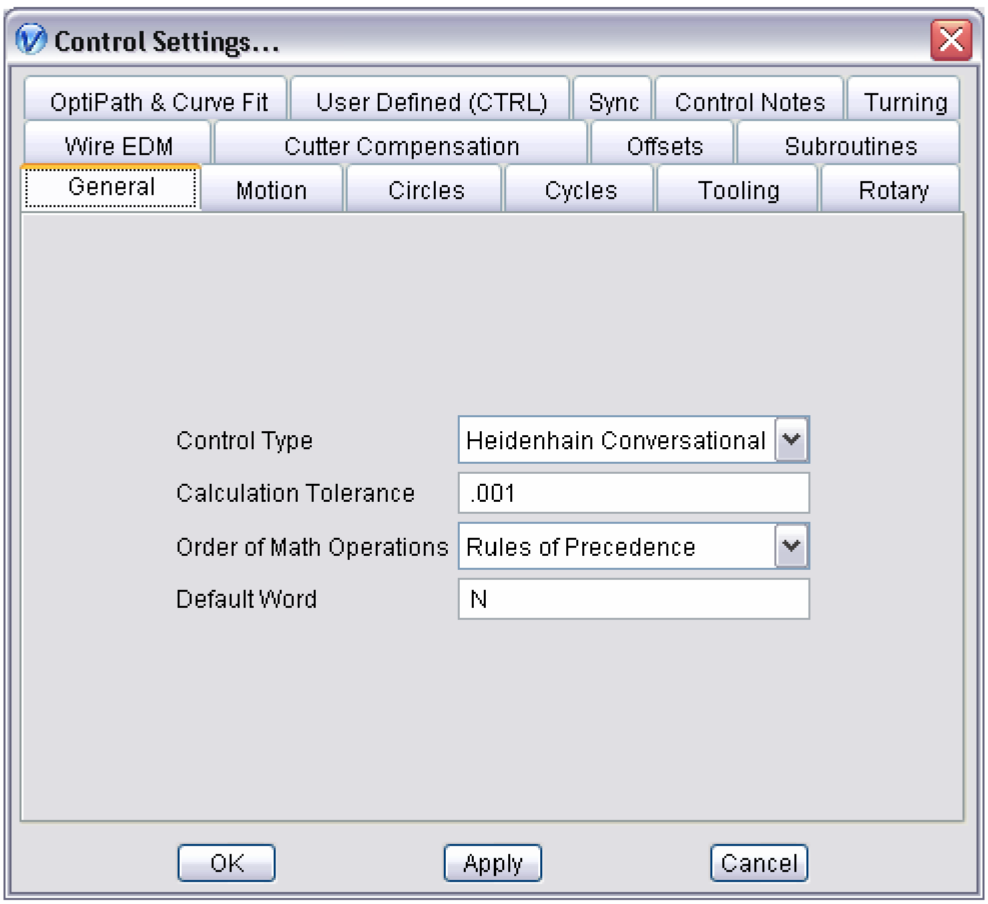
Close — Stops the validation process and closes the Word/Address Validate window.

Control Settings (Control Settings window)

Location: **Configuration menu > Control Settings**

VERICUT toolbar short cut: 

Opens a window enabling you to configure how the machine control will process machine code data.



General tab — Features on this tab are used to configure general guidelines for how the control processes machine code data and number equations.

Motion tab — Features on this tab are used to configure default NC control motion states, and the precision used when outputting calculated values.

Circles tab — Features on this tab are used to configure how the circle center data (e.g. I J K) is interpreted.

Cycles tab — Features on this tab are used to configure how fixed tool axis, or "canned" cycle motion blocks are interpreted, e.g. G8n.

Tooling tab — The features on this tab are used to configure default tooling conditions and tool change activity.

Rotary tab — Features on this tab are used to configure how rotary motion commands are interpreted, e.g. A, B, C.

Wire EDM tab — Features on this tab are used to configure wire EDM machining settings.

Cutter Compensation tab — Features on this tab are used to configure how the programmed tool path is compensated when cutter diameter compensation, or "CDC" is used, e.g. G41-42.

Offsets tab — Features on this tab are used to configure an initial work offset, or “fixture offset”, to be in effect at the start of tool path processing.

Subroutines tab — Features on this tab are used to configure how subroutine names are referenced by the NC control.

OptiPath & Curve Fit tab — Features on this tab are used to configure settings referenced when G-Code tool path files are optimized by OptiPath.

User Defined (CTL) tab — Features on this tab are used to configure user defined control settings supplied via a custom CME file.

Sync tab — Features on this tab configure VERICUT for simulating machining with synchronized subsystems.

Control Notes tab — Features on this tab are used to enter "message notes" and "comment notes" in the current control file.

Turning tab — Features on this tab configure VERICUT for lathe turning simulations.

OK — Applies the changes and closes the Control Settings window.

Apply — Applies the changes and leaves the Control Settings window open.

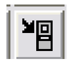
Cancel — Closes the Control Settings window without applying changes.

Also see "**Building NC Controls**", in the *Using VERICUT* section, in the *CGTech Help Library*.

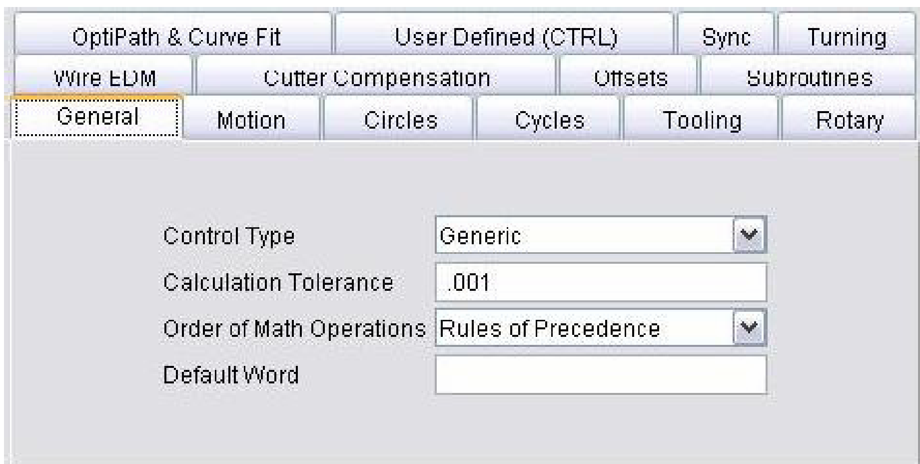
Control Settings window, General tab

Location: **Configuration menu > Control Settings**

OptiPath menu > Control > G-Code Output Options tab

VERICUT toolbar short cut: 

Opens a window to configure general guidelines for how the control processes machine code data and number equations.



Control Type — Sets the type of NC control being used. Choosing the proper option establishes a mode of operation consistent with how the control processes NC data. **Options are:**

Generic — Fanuc and most other controls using standard controller functions.

NUM — French-made control. This option affects how G-Code data is interpreted, for example: how variables are initialized and processed, and how to interpret parameters to macros that shift the location of the tool path.

Heidenhain Conversational — Heidenhain Conversational control. This option causes VERICUT to recognize the "L" word as an optimizable word. Also, the "L" word is included with any cuts added during optimization.

Siemens — Siemens 840D control. Special logic is added in the "filter" logic for the processing of NURBS.

Toshiba — Toshiba control. Special logic is added in the "filter" logic for the processing of NURBS.

Heidenhain ISO — Heidenhain ISO control. If a block is marked as non-optimizable, and the block contains a I, J, or K (circle record), then the Feedrate is not restored (similar to going into RAPID mode). A series of OptiPath output formatting decisions are based on this control type.

K&T — Kearney & Trecker control. Special handling is added for TYPE II processing. If a TYPE II argument is defined to be a V (Value), but an "=" character is found within this argument, then the argument is internally processed as a WV argument.

Calculation Tolerance — Tolerance for rounding mathematical evaluations, such as determining if two calculated values are equal to, greater than, or lesser than each other.

NOTE: The **Calculation Tolerance** should be adjusted to eliminate "invalid circle" errors caused by circle calculations that produce differences larger than the tolerance value when processing G-Code data.

Order of Math Operations — Order in which math operations are performed. Options are:

Rules of Precedence — Follow the basic rules of math precedence:

1. Perform exponential (power).
2. Perform multiplication and division operations.
3. Perform addition and subtraction operations, example: 5 + 5.3 * 3 * sin(30)= 12.95

Enclosing math operations in parenthesis causes them to be performed before those not enclosed in parenthesis. The same rules of precedence are applied to math operations enclosed in parenthesis.

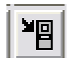
Left to Right — Use left to right sequence, example: 5 + 5.3 * 3 * sin(30)= 15.45

Default Word — Word assumed by VERICUT for NC data blocks beginning with numbers, such as NC data for some Heidenhain controls. For example, for VERICUT to interpret blocks like "50G01X5", set **Default Word** to "N". VERICUT then interprets the block without error as "N50G01X5".

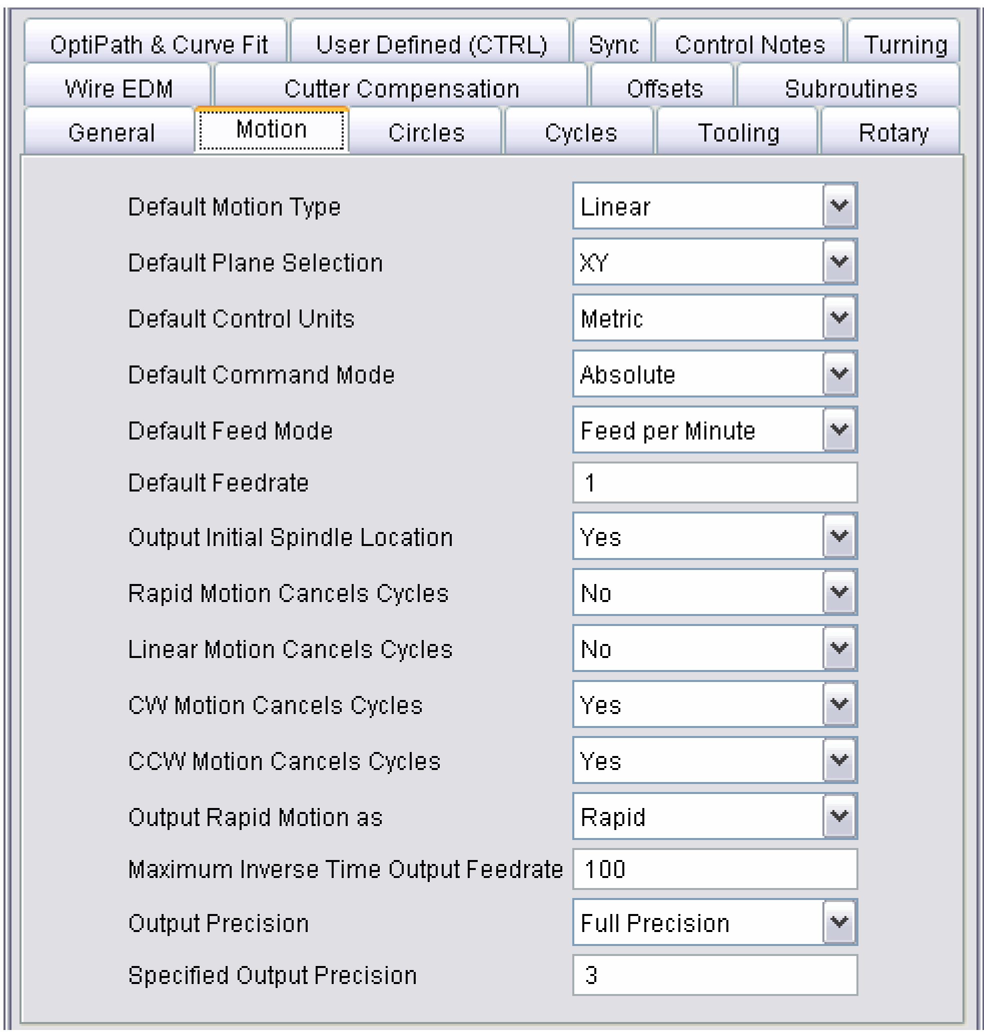
To Control Settings window

Control Settings window, Motion tab

Location: **Configuration menu > Control Settings**

VERICUT toolbar short cut: 

Opens a window enabling you to configure default NC control motion states, and specify the precision used when outputting calculated values. These settings are in affect when the control and machine are initially powered on.



Default Motion Type — Default motion type for the control.

Options are:

Rapid
Linear

Default Plane Selection — Default motion plane, or "cutting plane" for the control.
Options are:

XY
ZX
YZ

Default Control Units — Default measurement units for the control.

Options are:

Inch
Metric

Default Command Mode — Default command mode, or "input dimension mode" for the control.

Options are:

Absolute
Incremental

Default Feed Mode / Default Feedrate — Default feed rate mode and value for the control.

Feed mode options are:

Feed per Minute
Feed per Revolution

Output Initial Spindle Location — When active, causes the machine to move to its initial spindle location at the beginning of tool path processing. Enter initial machine location values in an **Initial Machine Location table**.

Rapid Motion Cancels Cycles — When active, rapid motions (e.g. G0) cancel canned cycles (e.g. G81-89).

Linear Motion Cancels Cycles — When active, linear motions (e.g. G01) cancel canned cycles (e.g. G81-89).

CW Motion Cancels Cycles / CCW Motion Cancels Cycles — When active, clockwise (or counterclockwise) circular interpolation motions (e.g. G02) cancel canned cycles (e.g. G81-89).

Output Rapid Motions as — Controls the format in which rapid motions are written to the APT Output file during reverse post-processing.

Options are:

> **Rapid —** Output the word "**RAPID**" followed by a single tool tip location.
>
> **Feedrate —** Output "**FEDRAT/n**" where the feed rate is calculated based on the rapid priority of moving axes, and then output the tool tip location(s). When rapid priority causes axes to move independently, multiple tool tip positions are output.

Maximum Inverse Time Output Feedrate — Highest feedrate value which can result from interpreting inverse time feed rate values (e.g. G93). The output feed rate is set to this rate when in inverse time feed mode and the calculated feed rate is zero or greater then the specified maximum.

Output Precision — Controls the accuracy used to interpret calculated values, such as: CDC offset positions, mathematical operations, etc.

Options are:

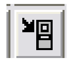
> **Full Precision —** Highest possible accuracy. Note that full precision output values can result in higher accuracy than the NC control uses, which may result in differences between simulated motion and the actual motion performed by the NC control.
>
> **Input Precision —** Use the accuracy used in the input tool path file. For example, when tool path file data is in 3.4 format, the precision used is also 3.4.
>
> **Specified Precision —** Use the accuracy specified via **Specified Output Precision** (see below)

Specified Output Precision — When **Output Precision**= **Specified Precision**, this value reflects the accuracy used by the NC control to perform the operations described above for **Output Precision**.

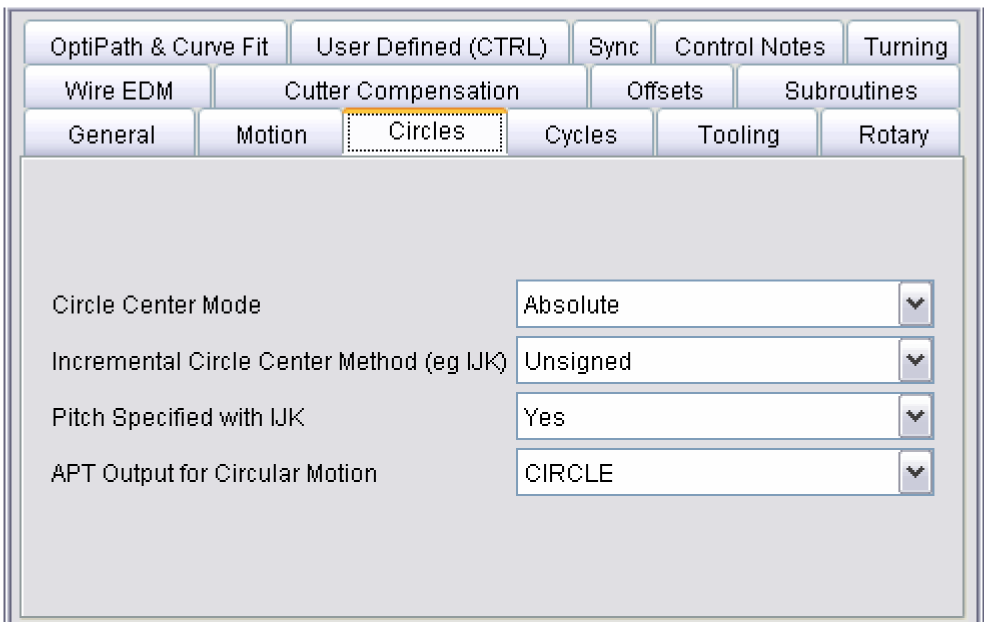
To Control Settings window

Control Settings window, Circles tab

Location: **Configuration menu > Control Settings**

VERICUT toolbar short cut: 

Opens a window to configure how the circle center data (e.g. IJK) is interpreted. The **Interpolation Tolerance** determines the quantity of machine positions which simulate arc and helical motions.



Circle Center Mode — Mode in which to interpret circle center data.

Options:

Absolute — Absolute center coordinates.

Incremental — Incremental distances. (See "**Incremental Circle Center Method**" below).

G Code Dependent — As described above, except depends on the active input dimension mode: absolute or incremental.

Incremental Circle Center Method — When interpreting incremental circle center values, controls how incremental circle center data is interpreted.

Options:

From Start to Center Point — Distance from circle start point to circle center.

From Center to Start Point — Distance from circle center to circle start point.

Unsigned — Unsigned, positive incremental distance between circle start point and circle center). Unsigned incremental circle motion blocks are always positive in value, and do not cut circles greater than 90 degrees.

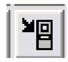
Pitch Specified with IJK — This option determines if the **CircleCenterX**, and CircleCenterY, and **CircleCenterZ** macros can be used to determine the pitch. Only the macro which corresponds to the direction perpendicular to the current motion plane will be used for the pitch value. The pitch is defined as depth per revolution. The default is "**No**". If set to "**Yes**", the delta depth distance divided by the pitch will determine the number of full loops. If set to "**No**", the **HelicalFullLoops** macro should be called, and the pitch will be calculated based on the number of full loops, the delta depth distance, and the starting and ending angles.

APT Output for Circular Motion — Controls the format in which circular motions are output during conversion to an APT output file. An APT Output file is created when **Create APT Output File** is active. (Ref. **Process Options window: G-Code Output Files tab**).

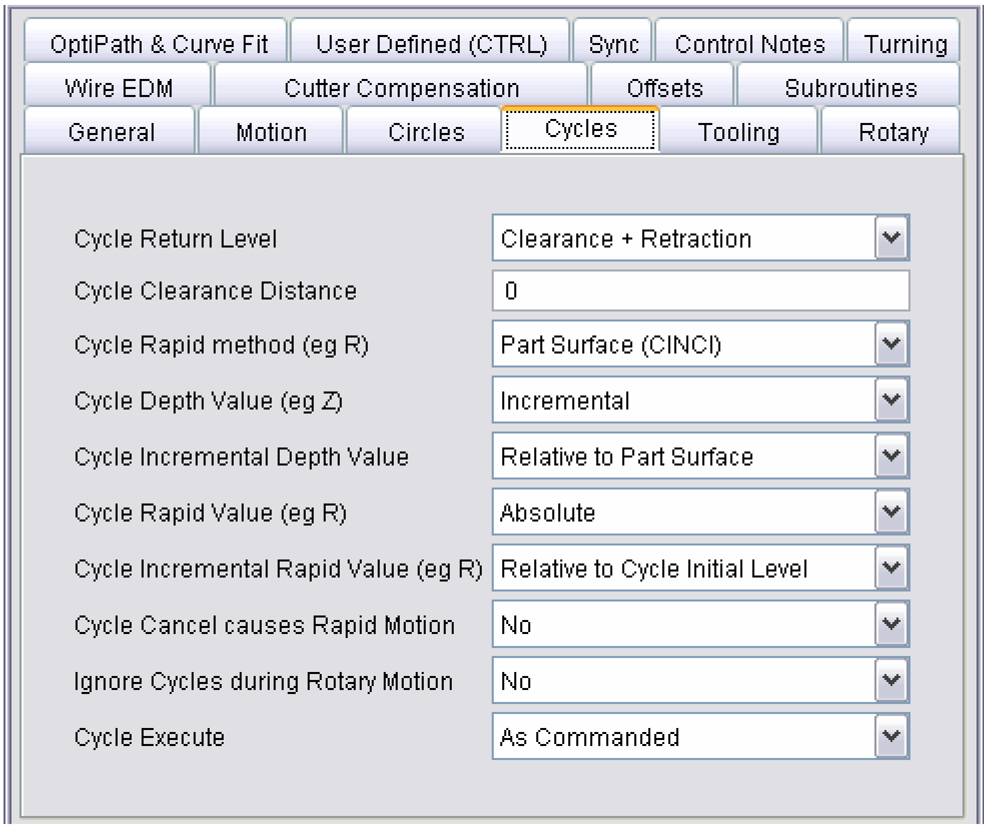
To Control Settings window

Control Settings window, Cycles tab

Location: **Configuration menu > Control Settings**

VERICUT toolbar short cut: 

Opens a window to configure how fixed tool axis, or "canned" cycle motion blocks are interpreted, e.g. G8n.



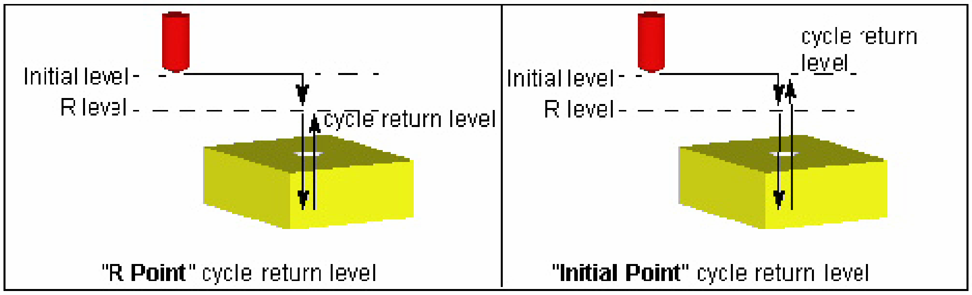
Cycle Return Level — Controls where the tool tip is returned (retracted) after processing a cycle command.

Options:

R Point — Rapid positioning or "R level".

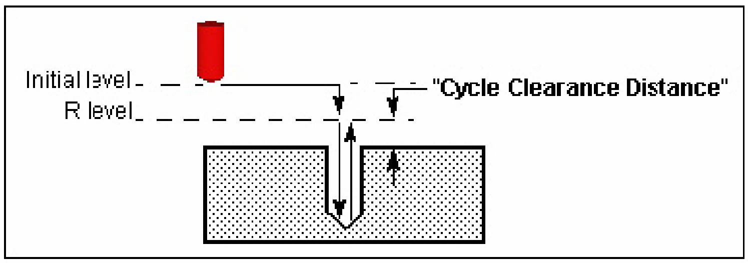
Initial Point — Initial level from where the cycle motion began.

Specified Point — Clearance distance from the workpiece as specified by a word/address, such as G71Zn with some Okuma controls, or **Clearance + Retraction** (similar to **Specified Point**, except a retraction clearance can be specified).



"**R Point**" cycle return level "**Initial Point**" cycle return level

Cycle Clearance Distance — Distance from the workpiece to the R level.

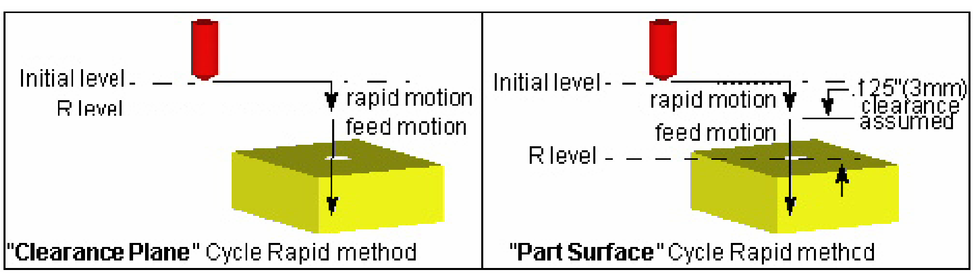


Cycle Rapid method — Controls how rapid positioning is performed between cycle executions.

Options:

Clearance Plane — FANUC method-R level located at some clearance above the workpiece, tool tip placed at the R level).

Part Surface — Cincinnati Acramatic method-R level located on the workpiece surface, tool tip placed .125", or 3 mm above the R level.



"**Clearance Plane**" Cycle Rapid method "**Part Surface**" Cycle Rapid method

Cycle Depth Value — Controls how the cycle depth values are interpreted.

Options:

Absolute — Absolute cycle depth value.

Incremental — Incremental distance as described by the **Cycle Incremental Depth Value** below).

G Code Dependent — As described above, except depends on the active input dimension mode: absolute or incremental.

Cycle Incremental Depth Value — When interpreting incremental cycle depth values, controls how incremental distances are interpreted.

Options:

Relative to Cycle Rapid Value — Incremental distance from R level.

Relative to Cycle Initial Level — Incremental distance from initial level.

Relative to Part Surface — Incremental distance from the part surface.

Cycle Rapid Value — Controls how cycle rapid values are interpreted to determine the R level.

Options:

Absolute — Absolute R level position.

Incremental — Incremental distance to the R level, as specified by the **Cycle Incremental Rapid Value**.

G Code Dependent — As described above, except depends on the active input dimension mode: absolute or incremental.

Cycle Incremental Rapid Value — When interpreting incremental cycle rapid values, controls how incremental distances are interpreted to determine the R level.

Options:

Relative to Cycle Initial Level — Incremental distance from the initial level.

Relative to Part Surface — Incremental distance from the part surface.

Cycle Causes Rapid Motion — When active, sets the rapid motion mode with a cancel cycle command (e.g. G80).

Ignore Cycles during Rotary Motion — When active, ignores canned cycles during rotary motions. Clear this checkbox to allow cycles during rotary motions. The use of this modal becomes more important when the **Output Intermediate Points** control setting is in use.

Cycle Execute — Controls when cycle motions are executed.

Options:

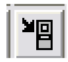
On Motion — Execute cycle on cycle block and each following motion until cancelled. (Default)

As Commanded — Cycle definition block is for setup only; execute cycle when commanded by a specific word/address, such as G79 with some Phillips NC controls. Requires using the **CyclesExecute** macro in the NC control configuration for the codes that command cycles to be executed.

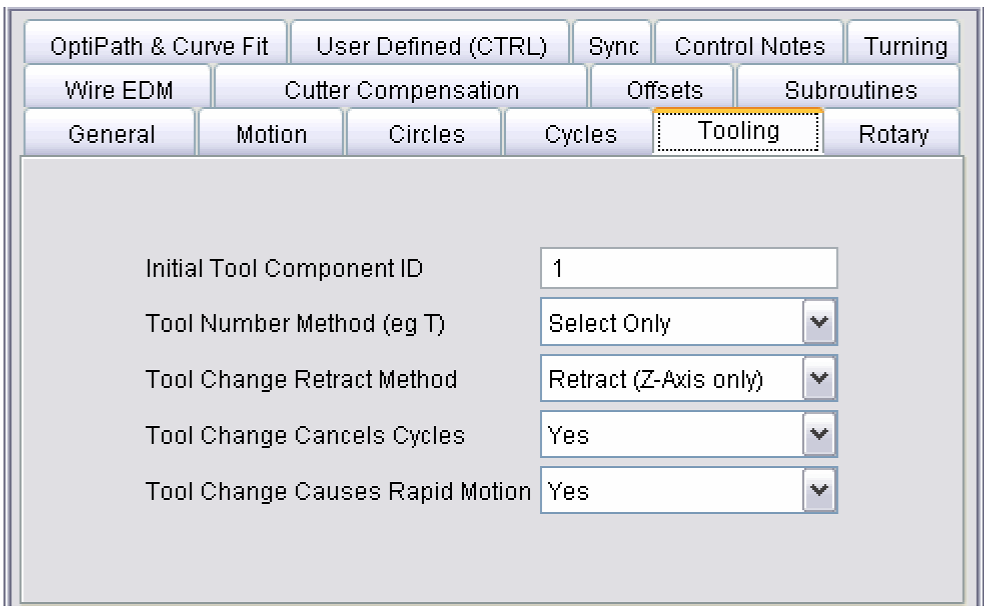
To Control Settings window

Control Settings window, Tooling tab

Location: **Configuration menu > Control Settings**

VERICUT toolbar short cut: 

Opens a window enabling you to configure default tooling conditions, and define tool change activity.



Initial Tool Component ID — Specifies the Tool Index Number of the tool component initially active for tool changes.

Tool Number Method — Controls how to interpret tool number word/addresses (e.g. Tn).

Options are:

> **Select Only —** Tool number only selects the tool. The tool is changed by a separate command, e.g. M6)
>
> **Select & Change —** Tool number selects and changes the tool. A separate tool change command is not used.

Tool Change Retract Method — Method of retracting for a tool change. Methods which retract machine axes reference location values stored in a **Tool Change Location table**.

Options are:

No Retract — Change the tool at its current location.

Retract (Z-Axis only) — Retract the tool only along the Z-axis.

Retract All Axes — Retract all machine motion axes to their respective tool change locations.

Retract Tool Side Axes — Similar to Retract All Axes, except applies only to motion axes connected between the machine Base and Tool components.

Use Retraction Table — Refer to the **Tool Change Retraction table** to determine which machine axes to retract.

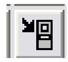
Tool Change Cancels Cycles — When active, cancels canned cycles with a tool change.

Tool Change Causes Rapid Motion — When active, sets the rapid motion mode with a tool change.

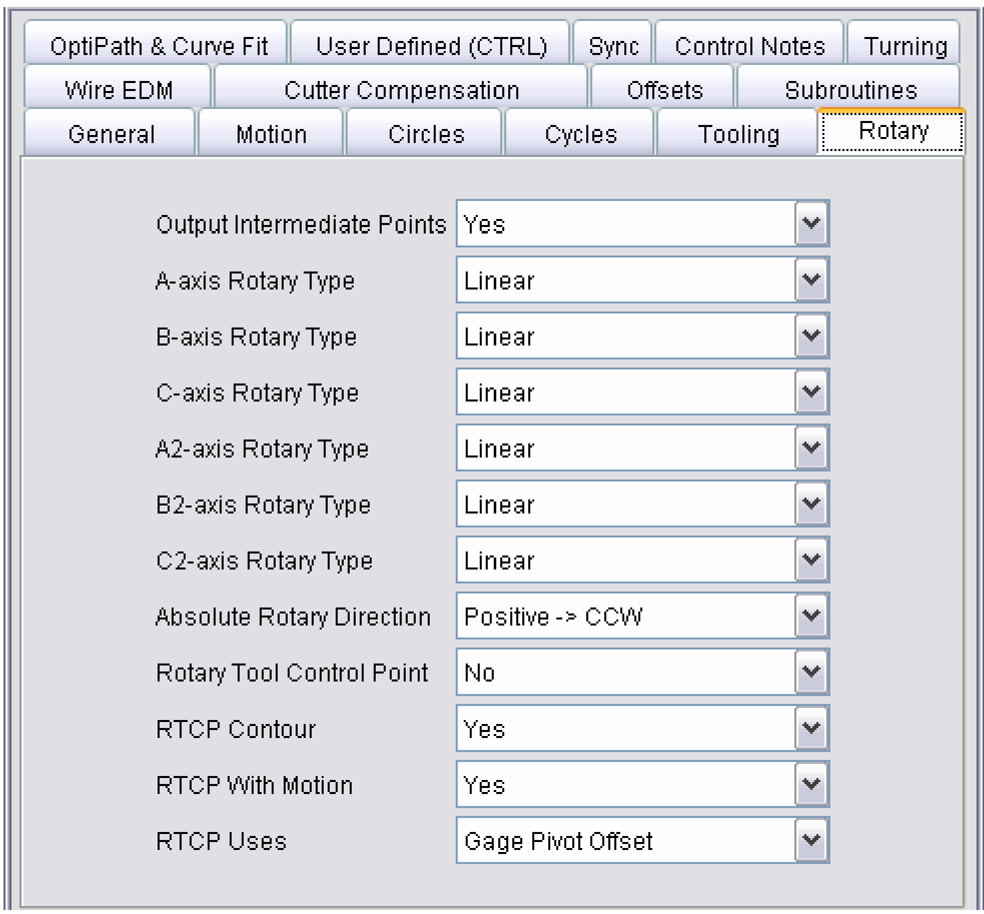
To Control Settings window

Control Settings window, Rotary tab

Location: **Configuration menu** > **Control Settings**

VERICUT toolbar short cut: 

Opens a window to configure how rotary motion commands are interpreted, e.g. A, B, C. A "moving tool philosophy" is assumed when describing rotations.

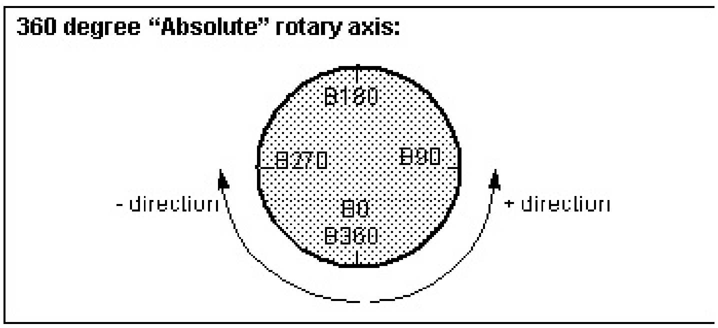


Output Intermediate Points — When active, generates intermediate tool positions during rotary motions. If turned off, only the final tool position of a rotary motion is shown. Rotary intermediate points help you see the tool-to-workpiece relationship during rotary motions, however, the simulation will be slower. The **Interpolation Tolerance** determines the quantity of machine positions which simulate rotary motions. (Ref. **File menu > Properties: Tolerance tab**)

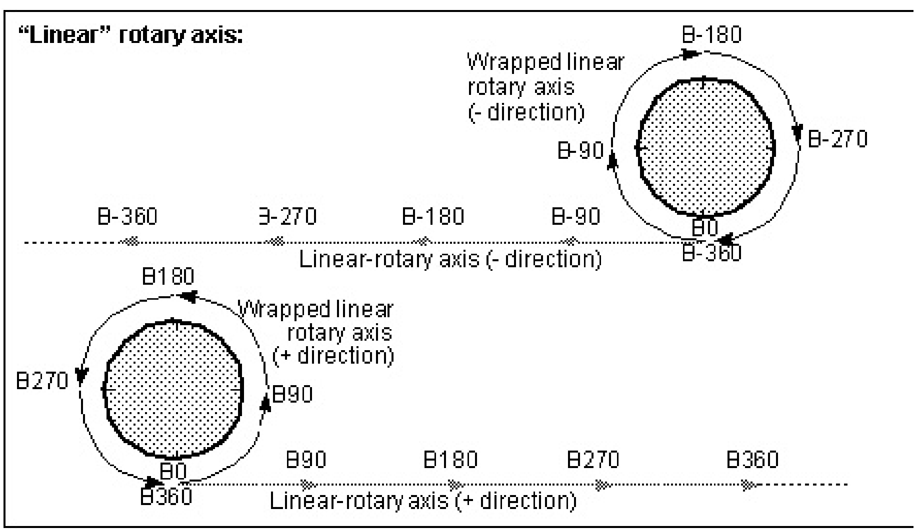
OPTIPATH NOTE: Multi-axis motions are not optimized when Output Intermediate Points is active.

A-axis Rotary Type, B-axis Rotary Type, C-axis Rotary Type — The settings control how rotary commands are interpreted (e.g. ABC). Each axis is individually controlled.
Options:

EIA (360 Absolute) — Refers to absolute angle positions. **Absolute Rotary Direction** controls how rotation values determine the direction of rotation.



Linear — Refers to angles on a linear axis "wrapped" around the rotary component. In absolute input dimension mode (e.g. G90), rotary values specify absolute locations along the linear-rotary axis, while the sign (+ / -) controls which end of the linear axis is used. In incremental input dimension mode (e.g. G91), rotary values specify degrees to rotate from the current position, and the sign controls the direction of rotation: plus = CCW, minus = CW.



Example rotations of a B-axis linear-rotary component:

Current axis position	Command	Rotary motion (moving tool philosophy)	Rotary destination
B0	G90B75	75 deg CCW	B75
B0	G90B-75	75 deg CW	B285 (B-75)
B35	G90B75	40 deg CCW	B75
B35	G90B-75	110 deg CW	B285 (B-75)
B150	G90B120	30 deg CW	B120
B150	G90B-120	270 deg CW	B240 (B-120)
N/A	G91B75	75 deg CCW	Current +75
N/A	G91B-75	75 deg CW	Current -75

A2-axis Rotary Type, B2-axis Rotary Type, C2-axis Rotary Type — The settings control how rotary commands for secondary rotary axes are interpreted (e.g. A2, B2, C2). Each axis is individually controlled. Options are same as described above for ABC rotary types.

Absolute Rotary Direction — Controls how rotation values determine the direction of rotation for **EIA (360 Absolute)** rotary type components. Options and examples follow.

Positive -> CCW — Sign controls the direction of rotation: plus = CCW, minus = CW, and absolute value of the rotation value specifies the rotary destination.

NOTE: You can configure the same behavior using **Configuration menu > Word/Address** and associating the **RotaryDirPosCCW** macro.

Examples of rotary behavior with Absolute Rotary Direction=Positive -> CCW:

Current axis position	Command	Rotary motion (moving tool philosophy)	Rotary destination
NA	G90B0	NA	B0
B0	G90B380	20 deg CCW	B20
B20	G90B-90	290 deg CW	B90
B90	G90B60	330 deg CCW	B60

Positive -> CW — Similar to **Positive -> CCW**, except the rotation directions are reversed.

NOTE: You can configure the same behavior using **Configuration menu** > **Word/Address** and associating the **RotaryDirPosCW** macro.

Examples of rotary behavior with Absolute Rotary Direction=Positive -> CW:

Current axis position	Command	Rotary motion (moving tool philosophy)	Rotary destination
NA	G90B0	NA	B0
B0	G90B380	340 deg CW	B20
B20	G90B-90	70 deg CCW	B90
B90	G90B60	30 deg CW	B60

Always CCW — Rounds the rotation value to increments of 360, then subtracts 360 until the subtracted value is between 0-360. The rounded value specifies the rotary destination. The tool always rotates in a CCW direction about the rotary center point.

NOTE: You can configure the same behavior using **Configuration menu** > **Word/Address** and associating the **RotaryDirCCW** macro.

Examples of rotary behavior with Absolute Rotary Direction=Always CCW:

Current axis position	Command	Rotary motion (moving tool philosophy)	Rotary destination
NA	G90B0	NA	B0
B0	G90B380	20 deg CCW	B20
B20	G90B-90	250 deg CCW	B270
B270	G90B60	150 deg CCW	B60

Always CW — Similar to Always CCW, except the rotation directions are reversed.

NOTE: You can configure the same behavior using **Configuration menu > Word/Address** and associating the **RotaryDirCW** macro.

Examples of rotary behavior with Absolute Rotary Direction=Always CW:

Current axis position	Command	Rotary motion (moving tool philosophy)	Rotary destination
NA	G90B0	NA	B0
B0	G90B380	340 deg CW	B20
B20	G90B-90	110 deg CW	B270
B270	G90B60	210 deg CW	B60

Shortest Distance — Uses the rotation value and its sign to specify the absolute rotary position. For each rotary motion, the tool rotates the shortest distance about the rotary center point to the rotary destination.

NOTE: You can configure the same behavior using **Configuration menu** > **Word/Address** and associating the **RotaryDirShortestDist** macro.

Examples of rotary behavior with Absolute Rotary Direction=Shortest Distance:

Current axis position	Command	Rotary motion (moving tool philosophy)	Rotary destination
NA	G90B0	NA	B0
B0	G90B380	20 deg CCW	B20
B20	G90B-90	110 deg CW	B270
B270	G90B60	150 deg CCW	B60

Linear — Rounds the rotation value to increments of 360, then subtracts 360 until the subtracted value is between 0-360. The difference between the rounded value and the current position is the rotary destination while the sign of the difference specifies the rotation direction. A positive difference value rotates the tool in a CCW direction about the rotary center point, a negative value rotates the tool in a CW direction.

NOTE: You can configure the same behavior using **Configuration menu** > **Word/Address** and associating the **RotaryDirLinear** macro.

Examples of rotary behavior with Absolute Rotary Direction=Linear:

Current axis position	Command	Rotary motion (moving tool philosophy)	Rotary destination
NA	G90B0	NA	B0
B0	G90B380	20 deg CCW	B20
B20	G90B-90	250 deg CCW	B270
B270	G90B60	210 deg CW	B60

Shortest Distance – 180 CCW — Uses the rotation value and its sign to specify the absolute rotary position. For each rotary motion, the tool rotates the shortest distance about the rotary center point to the rotary destination, except when the angle is exactly 180 degrees. When the angle is exactly 180 degrees, the direction will always be set to counterclockwise.

NOTE: You can configure the same behavior using **Configuration menu > Word/Address** and associating the **RotaryDirShortestDist180CCW** macro.

Shortest Distance – 180 CW — Uses the rotation value and its sign to specify the absolute rotary position. For each rotary motion, the tool rotates the shortest distance about the rotary center point to the rotary destination, except when the angle is exactly 180 degrees. When the angle is exactly 180 degrees, the direction will always be set to clockwise.

NOTE: You can configure the same behavior using **Configuration menu > Word/Address** and associating the **RotaryDirShortestDist180CW** macro.

Positive -> CCW ABSOLUTE — Sets the direction for an EIA (360 Absolute) rotary table to be counterclockwise when the angle is positive and clockwise when the angle is negative.

NOTE: You can configure the same behavior using **Configuration menu > Word/Address** and associating the **RotaryDirPosCCWAbsolute** macro.

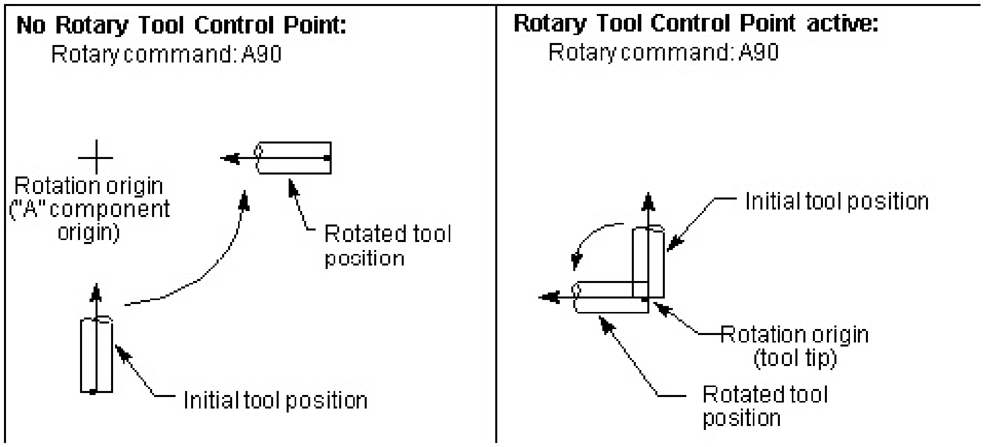
Positive -> CW ABSOLUTE — Similar to **Positive -> CCW ABSOLUTE**, but rotary directions are reversed. Sets the direction for an EIA (360 Absolute) rotary table to be clockwise when the angle is positive and counterclockwise when the angle is negative.

NOTE: You can configure the same behavior using **Configuration menu > Word/Address** and associating the **RotaryDirPosCWAbsolute** macro.

Rotary Tool Control Point — When active, with default settings, Rotary Tool Control Point, or "RTCP" causes rotary motion about a specific control point-typically the tool tip. When RTCP is not active, rotation occurs about the rotary component origins. This feature is intended for use only with tilting head machines.

RTCP compensates for the gage offset (XYZ distance from tool tip to Tool component) and pivot offset (XYZ distance from spindle face/centerline to tool side rotary components rotation/pivot point) as tool side rotary components rotate.

Affect of using RTCP with an A-axis rotary component:



RTCP only applies to rotary components between a Tool component and the Base component (tool side). Rotary components between a Stock component and the Base component (part side) do not effect the orientation of the tool, and therefore do not affect the tool side offsets.

RPCP, or Rotary Part Control Point, is used with part side rotary components, where the part is moved relative to the tool tip. RPCP actually rotates the workpiece coordinate system as the part side rotary components rotate. RPCP should only be used when programming in the part coordinate system. See reference information on the "**RpcpOn**, **RpcpOff**" macros, or the "**RPCP Pivot Offset table**" in the Tables for Processing G-Codes section for more information.

NOTES:

1. The default state for RTCP is set on the Rotary Settings panel. The RTCP state can be modified via the **RotaryControlPointOnOff** or **RtcpOn**, **RtcpOff** macros.
2. See also: "**RTCP Pivot Offset table**" or "**RPCP Pivot Offset table**" for possible additional configuration for correct RTCP simulation.

RTCP Contour — Use this feature to set the mode for RTCP contouring. "**Yes**" puts RTCP in contouring mode (XYZ offsets are updated and applied continuously during rotary motion). "**No**" puts RTCP in non-contouring mode (XYZ offsets are not updated and applied during rotary motion). Use the **RtcpContour** macro to over-ride this setting. RTCP must be active for this setting to have an effect.

RTCP With Motion — Use this feature to set the mode for RTCP with motion. "**Yes**" causes the RTCP updated XYZ offsets to only take effect when the corresponding X, Y, or Z axes are programmed in a rotary motion block. "**No**" causes the XYZ offsets to take effect immediately, meaning X, Y, or Z axes do not have to be programmed in a rotary motion block. Typically, this feature is only applicable when in non-contouring mode (**RTCP Contour** = "No"). Use the **RtcpWith Motion** macro to over-ride this setting. RTCP must be active for this setting to have an effect.

RTCP Uses — Use this feature to specify which type of pivot offset compensation to use. There are two Pivot Offset compensation types: **RTCP** and **Gage**.

RTCP specifies the programmed/driven point is assumed to be the spindle face/centerline with no tool loaded. The **RTCP Pivot Offset table** can be used with this type to override the automatic calculation. The centerline of rotation for multiple tool side rotary components must intersect.

Gage specifies the programmed/driven point is assumed to be the rotation/pivot point of the tool side rotary component, closest to the Base component. The **PivotOffsetCompName** macro can be used with this type to override where the XYZ pivot offset distance will be measured to. The centerline of rotation, for multiple tool side rotary components, does not have to intersect. The macros **TurnOnOffGageOffset** and **TurnOnOffGagePivotOffset** can be used to turn on and off the gage offset and the pivot offset, thereby driving either the spindle face or the tool tip. If Programming Method is set to Tool Tip, the gage offset is always active. RTCP must be active for this setting to have an affect.

Choose one of the following:

Default Pivot Offset — Use the RTCP Pivot Offset compensation type. The **ApplyGagePivotOffset** macro can be used to switch to the Gage Pivot Offset compensation type.

Gage Pivot Offset — Use the Gage Pivot Offset compensation type. This option should be used for all new jobs.

NOTE: The Gage Pivot Offset might currently be turned off, or may be set to (0,0,0).

RTCP Pivot Offset — Use the RTCP Pivot Offset compensation type.

NOTE: The RTCP Pivot Offset might currently be turned off, or may be set to (0,0,0).

Only Gage Offset — Do not compensate for XYZ pivot offset. Compensate only for the XYZ gage offset.

TECH NOTES:

Traditionally, Rotary Tool Control Point (RTCP) referred to a state in which the tool would rotate about the tool tip when a tool-side rotary was rotated.

Today this feature has a more generic definition. Rotary Tool Control Point refers to a state in which the tool side offsets are automatically updated when a tool-side rotary is rotated. This means that even though Rotary Tool Control Point is active, the results might not be as described above for the following reasons:

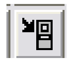
1. The traditional definition assumed that the offsets are adjusted continually as the rotary axis is rotated. This is not always the case. In the above diagram, an "A90" command is being executed. If the offsets are not updated continuously, the tool tip position will not stay fixed, but will "wander", and then end up in its original position. The **RTCP Contour** flag allows for configuration of either type.
2. The traditional definition assumed that the offsets would be applied immediately to all axes even though the corresponding linear axes were not specified on the block. This is not always the case. If the offsets are only applied when the corresponding linear axes are specified, then in the above case (A90), even though RTCP was active, the machine would behave as illustrated on the left. Then when XYZ is specified (with the original coordinates), then the updated offsets would be applied, and the tool tip would return to its original position. The **RTCP With Motion** flag allows for configuration of either type.

Some controls only adjust some of the 4 tool side offsets. If this is true, then the tool tip will not return to the original position.

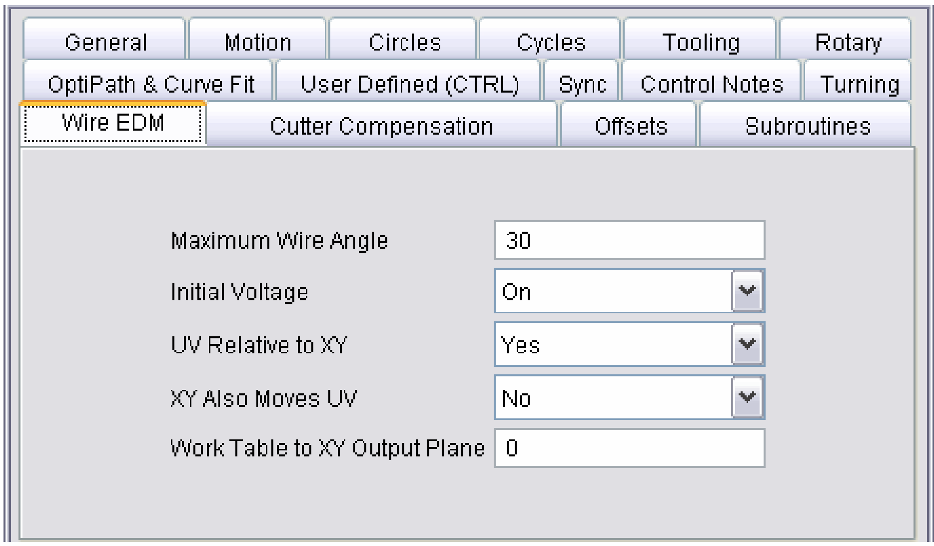
To Control Settings window

Control Settings window, Wire EDM tab

Location: **Configuration menu > Control Settings**

VERICUT toolbar short cut: 

Opens a window to configure wire EDM machining settings.



Maximum Wire Angle — Maximum angle the wire can tilt in any direction from vertical alignment. If the wire is tilted beyond this angle, an error message is issued.

Initial Voltage — When active, indicates voltage to the wire is on at the start of tool path processing.

UV Relative to XY — When active, U and V tool path values are relative to the current X and Y axis positions.

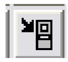
XY Also Moves UV — When active, X and Y tool path values also cause the corresponding U and V axis to move. In typical wire EDM machine construction, the U and V axes are connected such that they are slaved off the X and Y axes. In other words, when the X and Y axes move, the corresponding U and V axes also move by default. The **XY Also Moves UV** setting is intended to work with machines built such that the U and V axes are separate from the X and Y axes.

Work Table to XY Output Plane — Distance from the top of the work table (machine Z zero) to the XY output plane (output Z values). The XY output plane is the Z-axis level at which X and Y values are assumed to be driven. This value is measured along the machine Z-axis, and is usually zero.

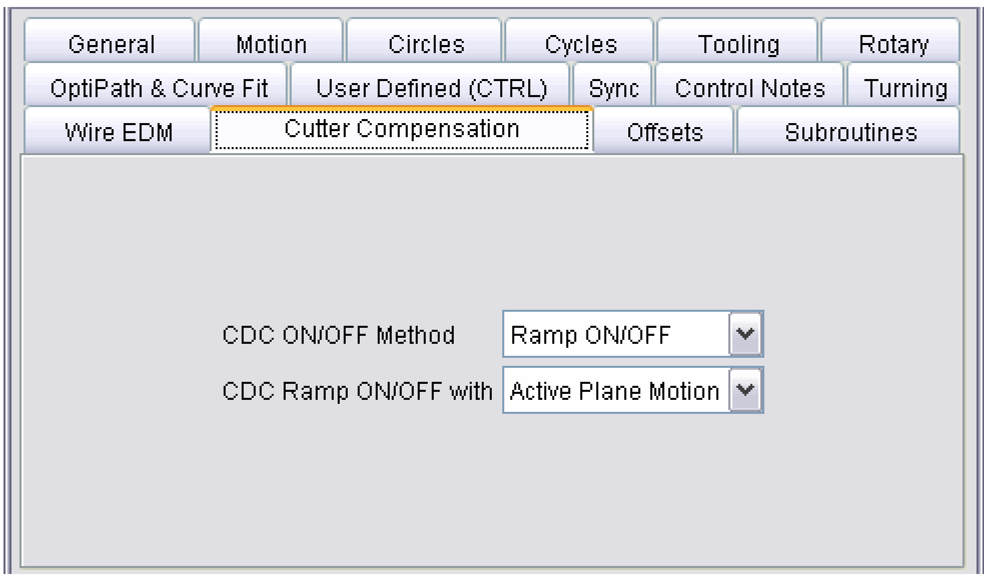
To Control Settings window

Control Settings window, Cutter Compensation tab

Location: **Configuration menu > Control Settings**

VERICUT toolbar short cut: 

Opens a window to configure how the programmed tool path is compensated when cutter diameter compensation, or "CDC" is used, e.g. G41-42. Enter CDC offset values into a **Cutter Compensation table**. To apply CDC offsets to the programmed tool path, ensure **Process Cutter Comp.** is active.



CDC ON/OFF Method — Controls how the offset condition is established when CDC is turned on, and how the offset is cancelled when CDC is turned off. Options are:

Ramp ON/OFF — Establish the offset condition with a "ramp-on" motion after CDC is turned on. When CDC is turned off, the offset is cancelled with a "ramp-off" motion on the next tool move.

Immediate — Immediately establishes the offset condition when CDC is turned on, and assumes the offset has been established by a prior data block (e.g. G43-44 as interpreted by the Phillips CNC controls). When CDC is turned off, the offset is cancelled without a tool move.

CDC Ramp ON/OFF with — Specifies the motion types which establish/cancel the CDC offset. Options:

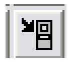
Active Plane Motion — Turn CDC on with a motion in the active cutting plane.

Any Motion — Turn CDC on with any motion, including motion along the tool axis.

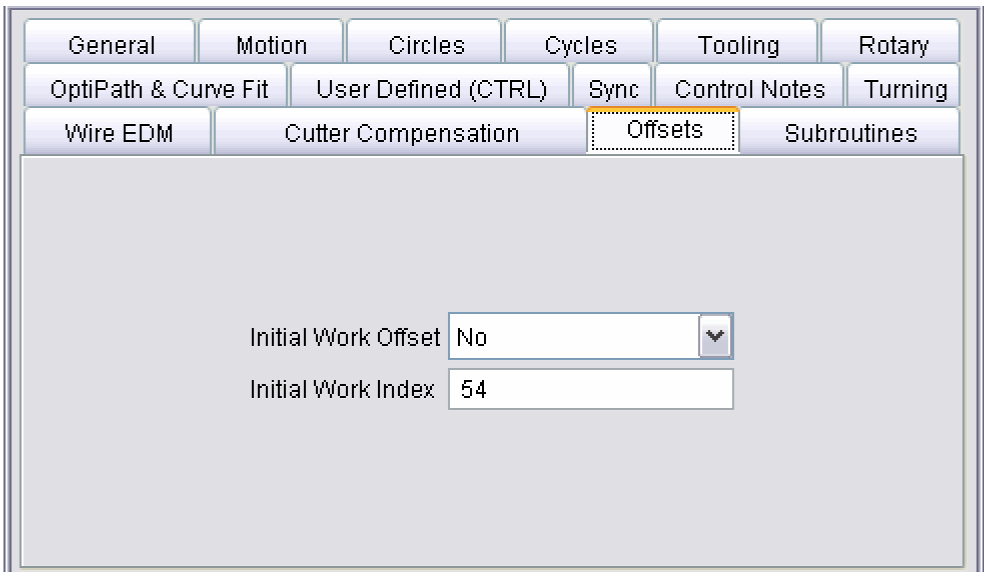
To Control Settings window

Control Settings window, Offsets tab

Location: **Configuration menu > Control Settings**

VERICUT toolbar short cut: 

Opens a window to configure an initial work offset, or "fixture offset", to be in effect at the start of tool path processing.



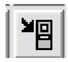
Initial Work Offset — When active, sets the initial work offset specified by the **Initial Work Index** (see below).

Initial Work Index — Index, or offset register value, containing the initial work offset. The offset remains effective until a different work offset (e.g. G54-59) is processed, or until cancelled. Enter the corresponding offset values into a **Work Offsets table**.

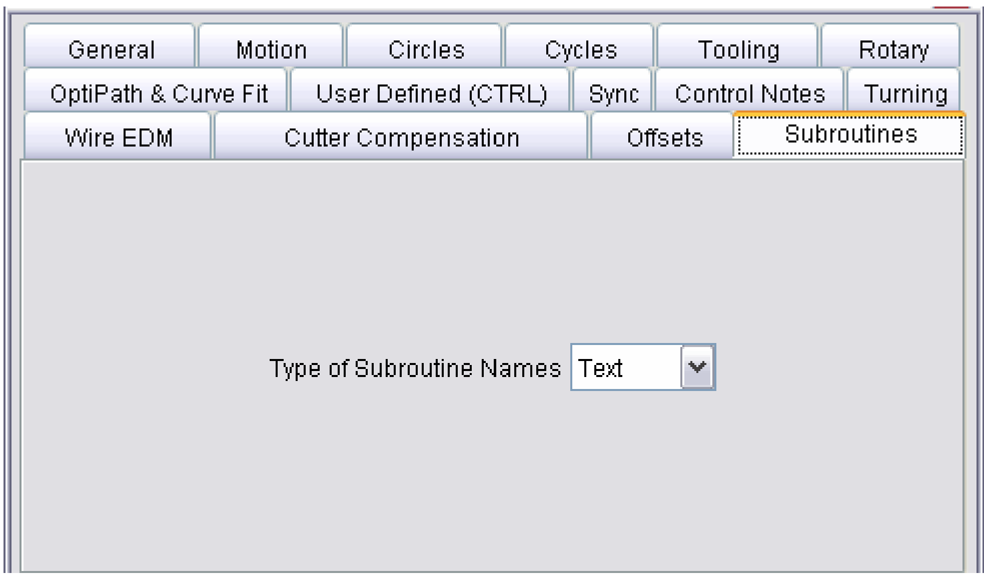
To Control Settings window

Control Settings window, Subroutines tab

Location: **Configuration menu > Control Settings**

VERICUT toolbar short cut: 

Opens a window to configure how subroutine names are referenced by the NC control.



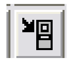
Type of Subroutine Names — Controls how names of subroutines are referenced.
Options are:

> **Numeric —** Use the numeric value; leading zeros are ignored. For example, "O0010" is interpreted the same as "O10".
>
> **Text —** Use the text string value. For example, "O0010" is not interpreted the same as the name "O10".

To Control Settings window

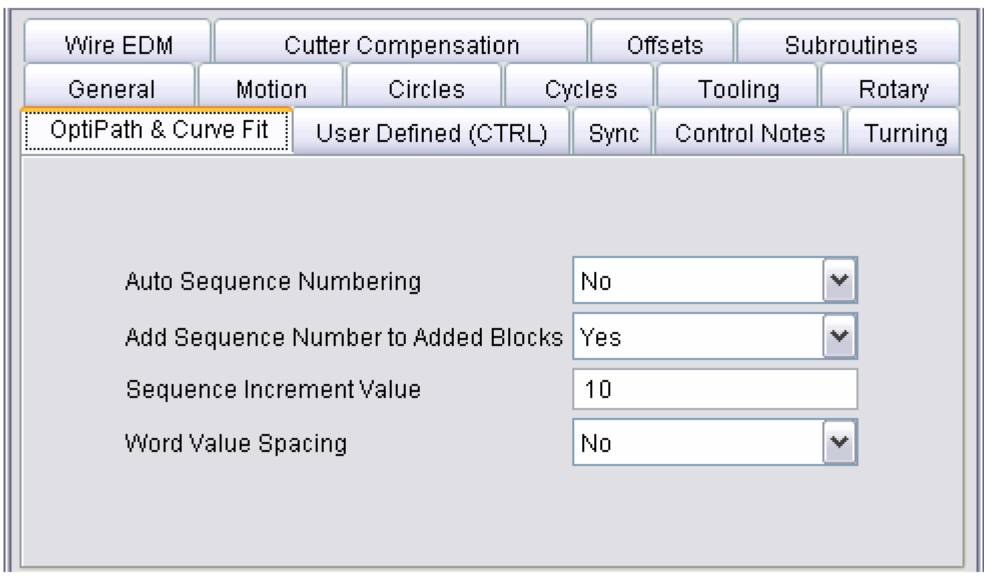
Control Settings window, OptiPath & Curve Fit tab

Location: **Configuration menu > Control Settings**

VERICUT toolbar short cut: 

Opens a window to configure settings referenced when G-Code tool path files are optimized by OptiPath.

The settings are used to control block sequencing and word/value spacing in the optimized output tool path file. Note that by default, optimized blocks receive the same sequence number, if any, that appears on the original block processed. This typically results in multiple blocks having the same sequence number.

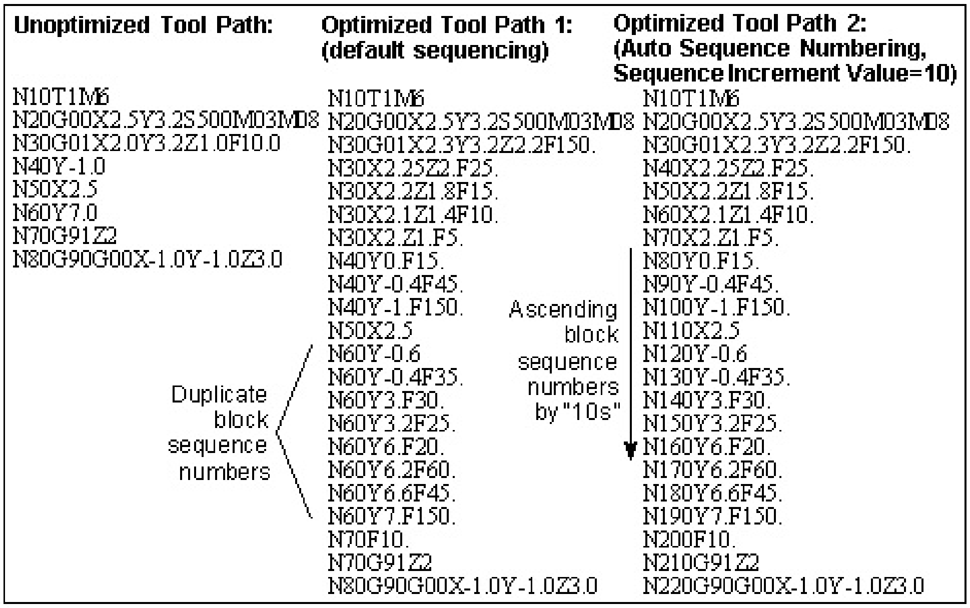


Auto Sequence Numbering — When active, re-sequences optimized tool path records in ascending order. The **Sequence Increment Value** (see below) is used for the beginning sequence number, and as the increment value for subsequent sequence numbers. Both optimized and non-optimized blocks with sequence numbers are re-sequenced. Sequence numbers are not added to un-optimized blocks without a sequence number.

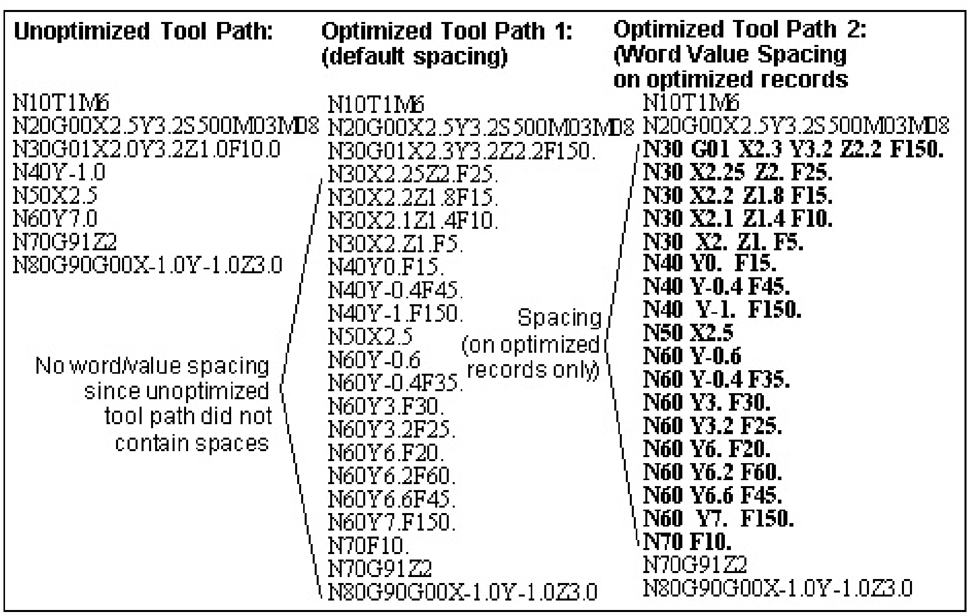
NOTE: If the general control setting **Default Word**="N", then "N" does not appear in the sequence number. The result is a sequence number without a word, for example: "50G01X5", such as required by Heidenhain controls.

Add Sequence Number to Added Blocks — When active, adds sequence numbers to blocks added during optimization via the **Add More Cuts** OptiPath setting.

Sequence Increment Value — Initial value and increment value for re-sequencing optimized G-Code tool path records.



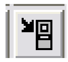
Word Value Spacing — When active, includes a space between word/value pairs in the optimized tool path file. Un-optimized records do not receive spacing. When this feature is not used, optimized tool path records are not spaced.



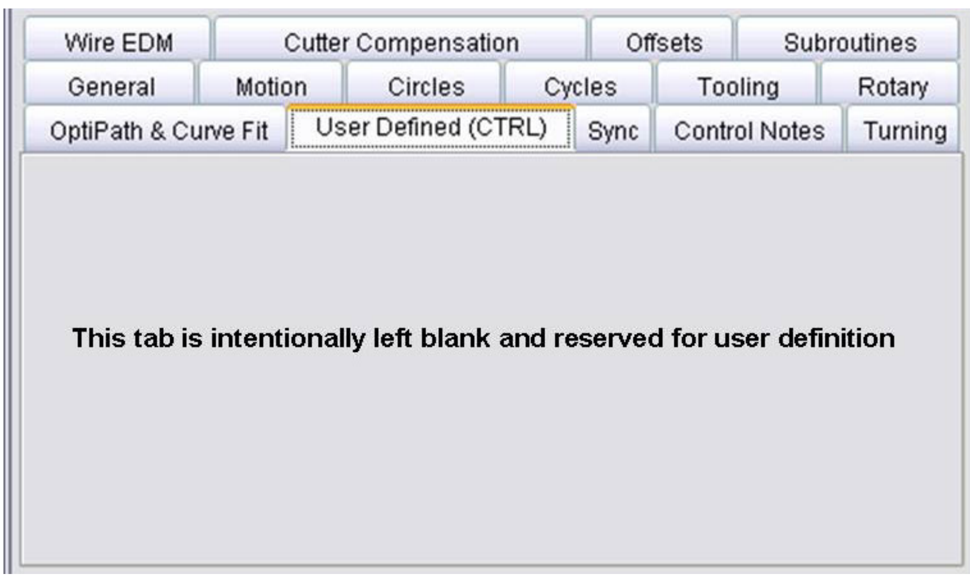
[To Control Settings window](#)

Control Settings window, User Defined tab

Location: **Configuration menu > Control Settings**

VERICUT toolbar short cut: 

Opens a window to configure user defined control settings supplied via a custom CME file. This window will appear differently, depending on the CME file in use. CME files are created using the **C Macro Extension – Application Programming Interface**, or **CME–API** (ref. **CME-API** in the VERICUT Development Tools section, in the *CGTech Help Library*), and specified for use in VERICUT via **Dev Kit CME tab** of the **Advanced Control Options window** (ref. **Advanced Control Option window: Dev Kit CME tab**, in the Configuration Menu section of *VERICUT Help*).

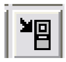


NOTE: CME-API defined settings can also appear under **Project menu > Processing Options > G-Code > Process Options: User Defined tab**, as determined by the developer.

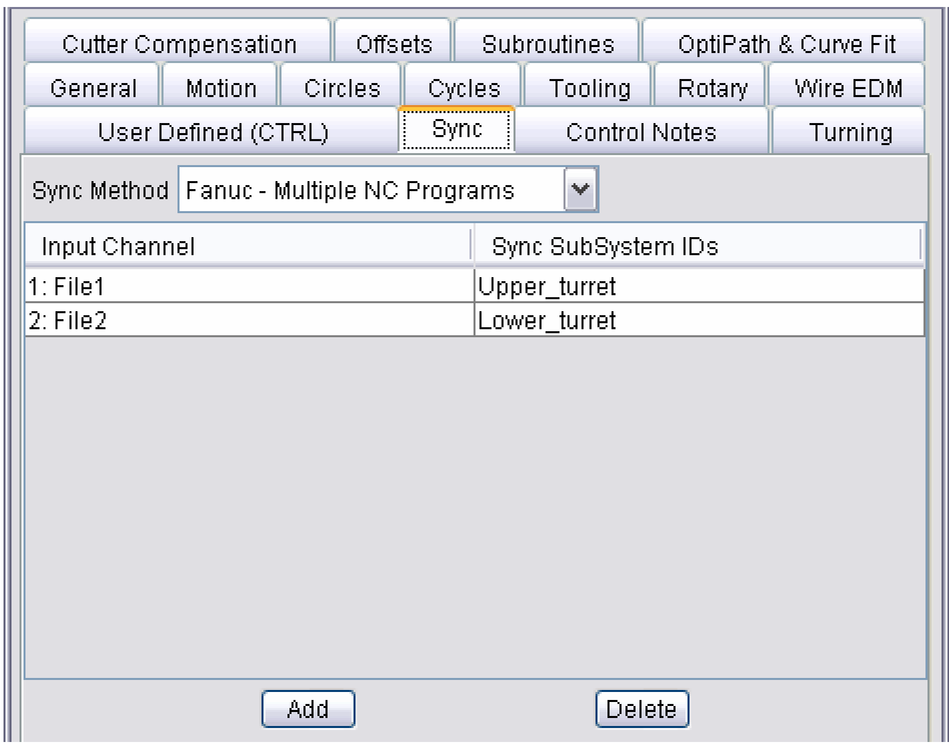
To Control Settings window

Control Settings window, Sync tab

Location: **Configuration menu > Control Settings**

VERICUT toolbar short cut: 

Features on this tab are used to set up VERICUT for simulating machining with synchronized subsystems.



The term "sync" refers to the idea of synchronizing multiple programs (or channels) at once. A program could be defined by a separate input file, or by sections of a single input file. Each of these programs drives a subsystem. The sync logic will obtain motions for all subsystems being synced, and then execute the motion. For example: If the upper system has a motion that will take 3 seconds, and the lower subsystem has a motion that will take .5 seconds. The lower subsystem motion and 1/6th of the upper subsystem motion will be executed. Then, the next motion will then be retrieved for the lower subsystem, and the process continues. The **Info menu > NC Program window** shows which blocks are being processed by the programs.

NOTE: The actual synchronization of the programs based on WAIT or SYNC codes are controlled by the Word/Address table.

Sync Method — The **Sync Method** turns on/off the Sync logic, and specifies the method that will be used to determine which data applies to which subsystem.

Options include:

None — (Default) Turns off Sync logic. Normal processing occurs.

G&L - N/O Block — An "N" sequence numbers mark commands for the first subsystem, "O" sequence numbers marks commands for the second subsystem, and "B" sequence numbers marks commands for both subsystems.

Okuma - G13/G14 — The input program is divided into sections. Each section begins with either a G13 or a G14. The G13 sections will drive the first subsystem, and the G14 sections will drive the second subsystem.

Fanuc - Multiple NC Programs — Independent tool path files are specified for each synced subsystem. The order of the subsystems specified must match the order of the NC program files in the NC Program list.

INDEX - Channel 1/Channel 2 — A single NC program file contains two subroutines, one for each head. A word/value, for example "M30", toggles between "channels" that control each head. These channels will then drive the corresponding subsystems. See the **IndexCallMainSub** macro in the "**Macros - listed alphabetically**" section for more information.

Gildemeister - $1/$2 — Blocks starting with $1 will be processed by the first subsystem, and blocks starting with $2 will be processed by the second subsystem. Blocks not beginning with either a $1 or a $2 will be processed by both subsystems.

Mazak - G109L1/G109L2 — The input program is divided into sections. Each section begins with either a G109L1 or a G109L2. The G109L1 sections will drive the first subsystem, and the G109L2 sections will drive the second subsystem.

Allen-Bradley - Primary/Secondary — This sync type has the following attributes:

1. A block can contain commands for both subsystems (controls).
2. Each control is processed independently.
3. The next block is not processed until both controls are done with the previous block.
4. Although the split up of the Info Toolpath is not necessary (or possibly desirable) for this sync type, it is still done for consistency with the way we handle sync motion.
5. When "stepping", both the upper and lower Toolpath windows (controls) will process 1 block.
6. The MDI window will ignore the subsystem specified, and will process the line as stated by sending it to both controls.

Citizen - $1/$2/$3 — The input program is divided into sections. Each section begins with a $1, a $2, or a $3. The $1 sections will drive the first subsystem, the $2 sections will drive the second subsystem, and the $3 sections will drive the third subsystem.

Sync Table

Each record in the Sync Table represent the information related to a particular "sync'd" subsystem.

Input Channel — This field defines what is driving each channel. The information displayed here will be different depending on the **Sync Method** being used.

Sync SubSystem IDs — This field defines the SubSystem that will be driven by the specified Input Channel. Select the subsystem ID from the pull-down list.

The number of records in the Sync Table must match the **Sync Method** defined above. For example the Citizen must have 3 Sync records, and the Okuma must have 2. The Fanuc style could have any number of Sync records.

NOTE: A machine might define a separate subsystem for a head changer, a pallet changer, … These are not Sync SubSystems. The Sync SubSystems corresponds to the primary SubSystem this is being driven by each of the Sync'd channels.

Add — Use to add a record to the Sync Table after the highlighted SubSystem ID.

Delete — Use to delete the highlighted record from the Sync Table.

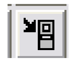
Shortcut: You can right-click in the Sync Subsystems IDs Table to display a menu containing **Add** and **Delete**. These provide the same functionality described above.

Tip: Once you have set up the sync environment using the features in this tab, you can use the features on the **G-Code Settings window: Sync tab** (**Project menu > Processing Options > G-Code > Settings**) to turn off one or more subsystems to assist in debugging Sync related problems.

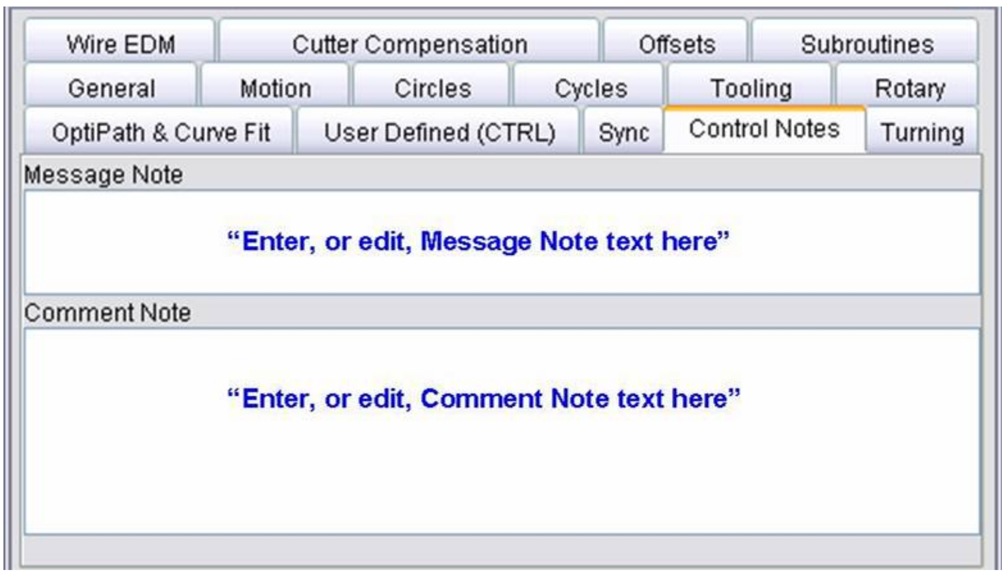
To Control Settings window

Control Settings window, Control Notes tab

Location: **Configuration menu > Control Settings**

Toolbar short cut for Control Settings window: 

Features on this tab enable you to add **Message Notes** and **Comment Notes** to the control file.

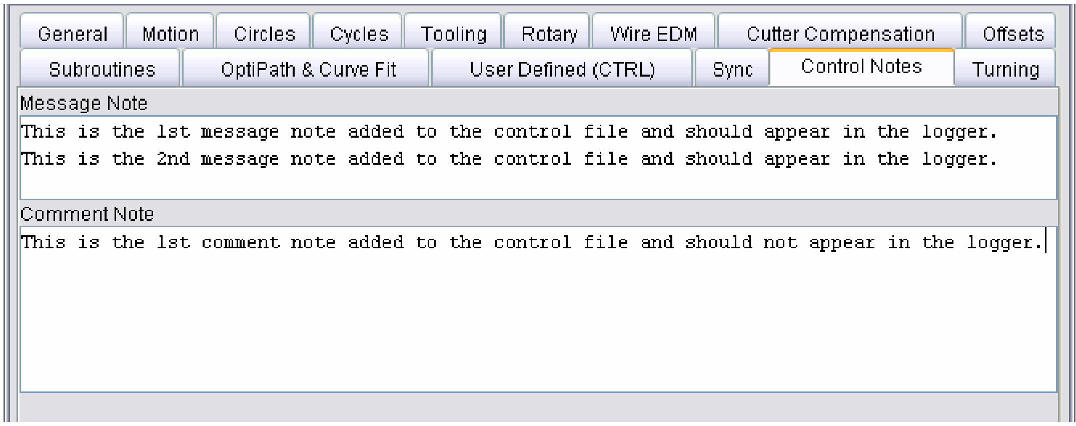


Message Notes are saved in the header of the control file and are displayed in the VERICUT message area (Logger) when the control file is loaded. **Comment Notes** are saved in the header of the control file but do not display in the VERICUT Logger.

There is a limit of 255 characters per line, but there is no limit to the number of lines that can be added.

Any Message/Comment Notes that currently exist in the control file are displayed in the appropriate text field as shown below. Any note can be edited and re-saved in the control file.

Existing Message/Comment Notes in the control file:



In the control file, each line of note is added to the header in quotes, and is preceded by either the keyword MESSAGE (for message notes), or COMMENT (for comment notes) as shown below.

How the above messages look in the control file:

```
CGTech Control
Version-6.2
MESSAGE "This is the 1st message note added to the control file and
should appear in the logger."
MESSAGE "This is the 2nd message note added to the control file and
should appear in the logger."
COMMENT "This is the 1st comment note added to the control file and
should not appear in the logger."

UNITS INCH

CSS_MAX_RPM 1000

SYNC_METHOD G13G14
```

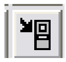
Adding /editing notes:
Add a note by typing the desired text, or edit the text of existing notes in the appropriate text field. When finished, press **OK**, or **Apply**, depending on whether or not want to leave the Control Settings window open. Then save the control file using the **Configuration menu** > **Control** > **Save (or Save As)** feature in the menu bar, or use  (**Save Control**) in the tool bar.

NOTE: Control files containing notes can not be used in pre-V6.2 versions of VERICUT.

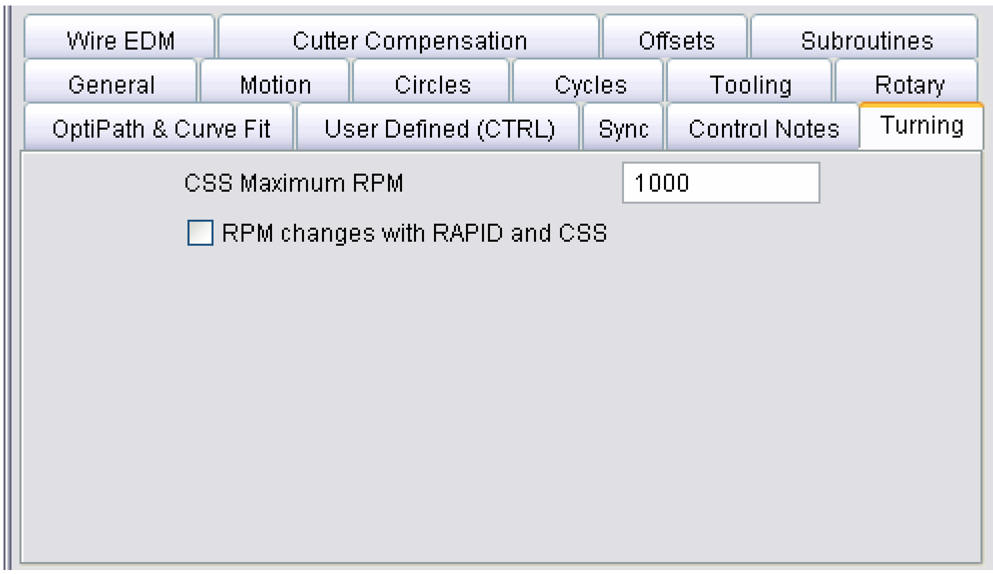
To Control Settings window

Control Settings window, Turning tab

Location: **Configuration menu > Control Settings**

VERICUT toolbar short cut: 

Features on this tab configure VERICUT for lathe turning simulations.



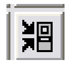
CSS Maximum RPM — When constant surface speed, or "CSS" is controlling spindle speed, this value specifies the fastest RPM to turn the workpiece.

RPM Changes with RAPID and CSS — When selected and CSS is controlling spindle speed, rapid motions (e.g. G0) can affect spindle speed, as well as linear and circular motions (e.g. G1-3). Clear this checkbox to have only linear and circular motions affect RPM.

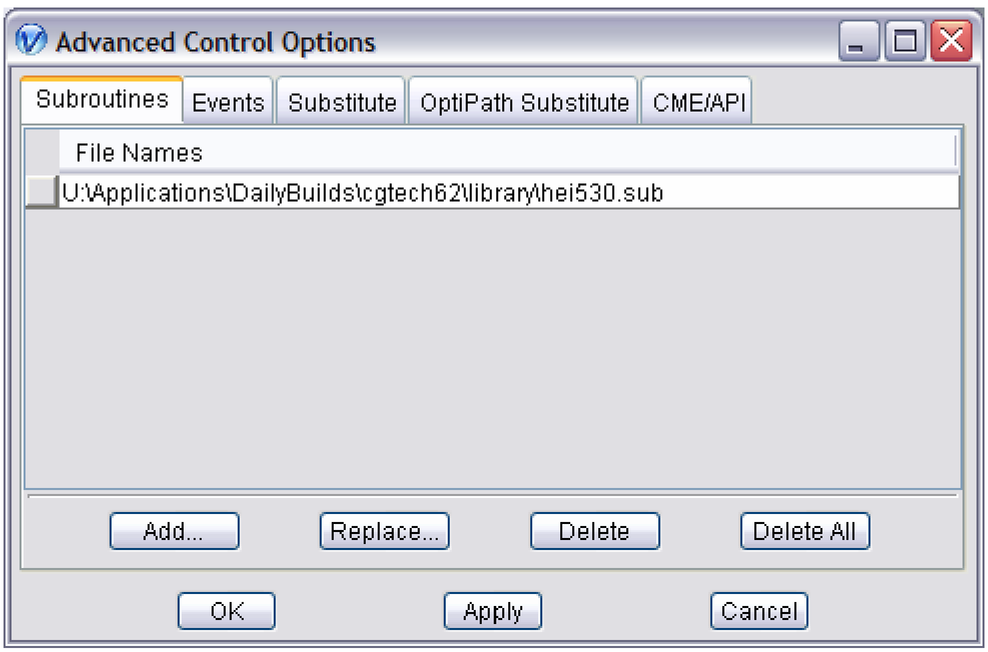
To Control Settings window

Adv. Options (Advanced Control Options window)

Location: **Configuration menu > Adv. Options**

VERICUT toolbar short cut: 

Opens a window to configure advanced NC control options, such as: specify subroutines available in the NC control, perform actions at key processing events (e.g. start of tool path processing, start processing a block, etc.), substitute text strings, and more.



Subroutines tab — The features on this tab specify names of external files containing subroutines accessible by the NC control.

Events tab — The features on this tab maintain macros called and variables set based on key events that occur during tool path file processing.

Substitute tab — The features on this tab maintain text strings that are substituted when specific text in the tool path file is encountered.

OptiPath Substitute tab — The features on this tab maintain text strings that are substituted when tool path files are optimized by OptiPath and written to the Optimized file.

CME/API tab — Features on this tab enable you to specify a custom CME file containing custom macros that assist the NC control with processing machine code data.

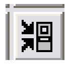
OK — Saves additions/modifications and dismisses the **Advanced Control Options window**.

Apply — Saves additions/modifications and leaves the **Advanced Control Options window** open.

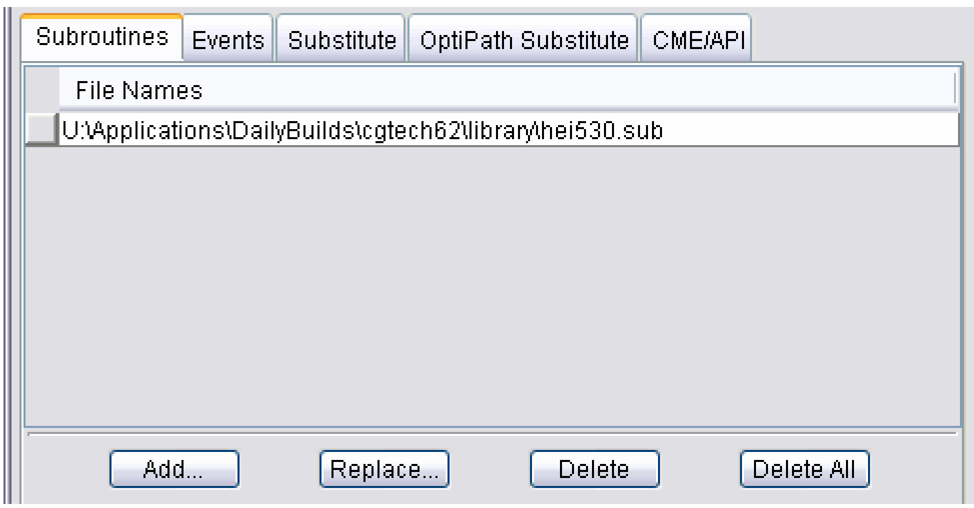
Cancel — Closes the **Advanced Control Options window** without saving any additions/modifications.

Advanced Control Options, Subroutines tab

Location: **Configuration menu > Adv. Options**

VERICUT toolbar short cut: 

The features on this tab specify names of external files containing subroutines accessible by the NC control.



File Names list — List of external files containing subs accessible to VERICUT.

Shortcut: Right-click in the **File Names list** area to display a pop-up window with the following features:

Add — Opens the Job Subroutine Files selection box enabling you to add external subroutine files to the File Names list.

Replace — Opens the Job Subroutine Files selection box enabling you to replace the highlighted subroutine file in the File Names list with another.

Delete — Deletes the highlighted subroutine file from the File Names list.

Edit — Displays the of the highlighted subroutine file in a text editing window. Standard text editing features are provided, such as: copy/cut, paste, search, etc. For more information on using the editing features see **Edit menu > Text File**.

Add — Opens the Job Subroutine Files selection box enabling you to add external subroutine files to the File Names list. (Same as **Add** in the pop-up window described above)

Replace — Opens the Job Subroutine Files selection box enabling you to replace the highlighted subroutine file in the File Names list with another. (Same as **Replace** in the pop-up window described above)

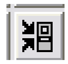
Delete — Deletes the highlighted subroutine file from the File Names list. (same as **Delete** in the pop-up window described above)

Delete All — Deletes all subroutines files from the File Names list.

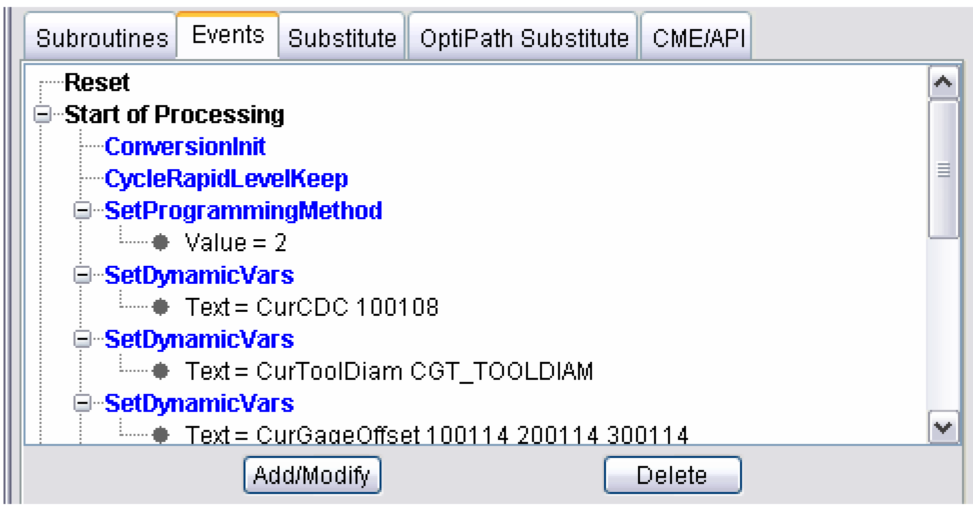
To Advanced Control Options window

Advanced Control Options, Events tab

Location: **Configuration menu > Adv. Options**

VERICUT toolbar short cut: 

The features on this tab enable you to specify the macros called and/or variables set based on key events that occur during tool path file processing. The **Events tab** displays the event configurations of a particular NC control as a "tree" structure. When you expand an "event" object, the macro calls/variable settings associated with that particular event are displayed.



The following Events are available:

Reset — Event occurs before the tool path file is scanned.

Start of Processing — Event occurs after the tool path file has been scanned, but before the tool path file is processed.

> NOTE: The **ConversionInit** macro must always be the first macro listed for this event.

End of Processing — Event occurs after the tool path file has been processed.

> NOTE: The **ConversionFinish** macro must always be the last macro listed for this event.

Start of Block Processing — Event occurs at the start of processing each G-Code data block.

> NOTE: The **BlockInit** macro must always be the first macro listed for this event.

End of Block Processing — Event occurs at the end of processing each G-Code data block.

NOTE: The **BlockFinish** macro must always be the last macro listed for this event.

Start of Type II Command / End of Type II Command — Event occurs at the start (or end) of processing a Type II data command.

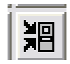
Add / Modify — Opens the **EventsAddMod window** enabling you to add (or modify) event-based macro calls/variable assignments.

Delete — Deletes the selected event-based macro call/variable assignment.

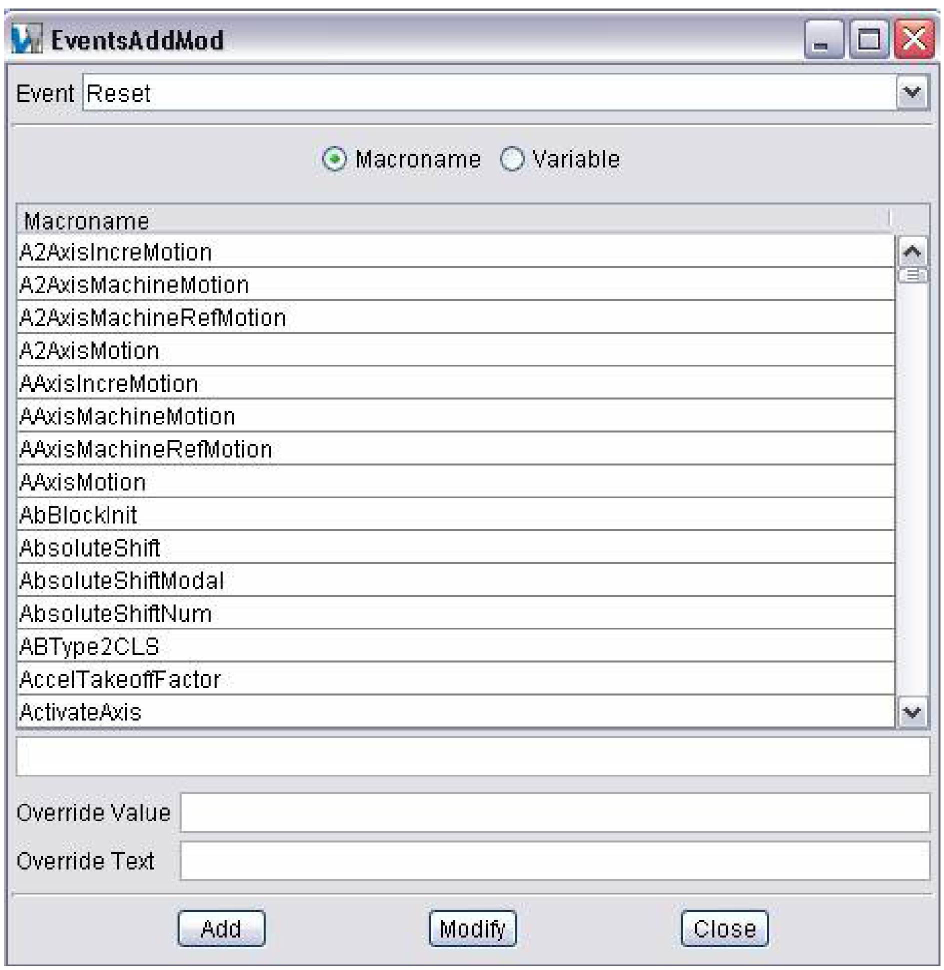
To Advanced Control Options window

EventsAddMod window

Location: **Configuration menu > Adv. Options: Events tab**

VERICUT toolbar short cut: 

Opened by pressing **Add/Modify** on the **Advanced Control Options window, Events tab**. The features on this window enable you to add (or modify) event-based macro calls/variable assignments. Features in this window behave similar to those described for word/address groups on the **Add/Modify Word/Address window**.



Event — Enables you to select the event that you want to add to or modify. Select from the pull-down list.

Macroname/Variable — Controls calls a macro, or sets a variable. Depending on the active choice, features are provided for specifying the macro called (select or type in field), or the Variable Name and Variable Description of the variable to be set (enter the variable value in the Override Value field). You can select macro names, or type them in. Text entered is not case sensitive. Use the automatic filtering capability to help you find macros. When you enter text characters in the Macroname field, the list of macros is automatically filtered to show only those that match the specified text.

Override Value — This feature acts differently, depending on whether **Macroname** or **Variable** is selected:

With **Macroname** *selected* **—** Specifies a value to pass to the specified macro. If blank, the address value accompanying the word is passed.

With **Variable** *selected* **—** Specifies a value to assign to the specified **Variable Number**. Use the **Variable Description** field to describe what **Variable Number** represents.

Math expressions and variables understood by the control are supported. Enter "$" to specify using the address value accompanying the word. Enter "#", followed immediately by the variable name/number, followed by a space, to specify a variable value. Multiple variables can be added, separated by a blank space. If the override expression does not contain either of these characters, it is evaluated immediately and only the value is retained. If the expression can not be evaluated immediately, the full expression is retained in the control configuration and evaluated separately for each block that activates the group.

Examples of using Override Value: (given that the variable, #2 = 3)

Text entered in Override Value field	What is retained in control configuration	Sample G-Code data block	Resulting value when processed
SIN(30)	.5	X5	.5 (constant value- not affected by G-Code data)
SIN(30)	.5	X10	.5 (see note above)
$*10	$*10	X5	50
($+#2)*10	($+#2)*10	X5	80

Also see "**Using Equations/Expression in VERICUT**" in the *Using VERICUT* section for more information. *Using VERICUT* can be found in the *CGTech Help Library*.

Override Text — Similar to **Override Value**, except specifies text to pass to the macro. Only certain macros are designed to accept override text values. Consult the "**Macros - listed alphabetically**" section for more information.

Expression(s) can be used in macro override text. If the override text contains the sequence "{*expression*}" the expression text inside the curly-braces is evaluated and replaced with the calculated value.

For example, if the **MessageMacro** is used with the following override text:

"The spindle speed is {#speed} rpm"

The variable "#speed" is evaluated and if set to 500, the resulting message would be:

"The spindle speed is 500 rpm"

The syntax for the expression(s) is the same as defined for override values.

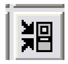
Add — Select when adding a new macro or variable to the event.

Modify — Select when modifying an existing macro or variable for the event.

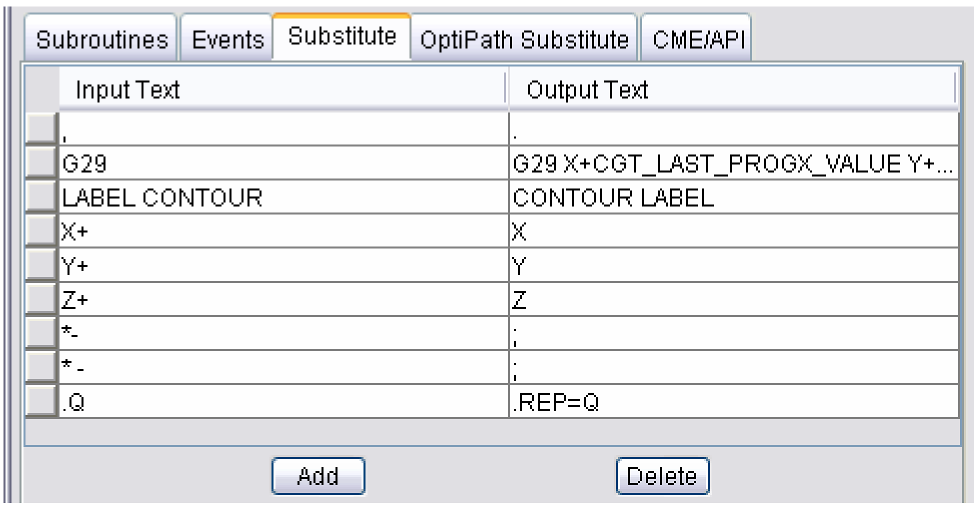
Close — Closes the **EventsAddMod window**.

Advanced Control Options, Substitute tab

Location: **Configuration menu > Adv. Options**

VERICUT toolbar short cut: 

The features on this tab maintain text strings that are substituted when specific text in the tool path file is encountered. Using substitutions lets you to alter how VERICUT "sees" data in a block without editing the G-Code tool path file. Substitutions occur at the beginning of processing each G-Code data block, prior to parsing block words and values into "tokens". Substitutions in the list are applied in listed order.



See "**About building NC controls**" in the *Using VERICUT* section for more information about how G-Code blocks are processed by VERICUT. *Using VERICUT* can be found in the *CGTech Help Library*.

Substitution (Input/Output Text) list — Lists text string substitutions that will occur when specified input text is encountered.

Shortcut: You can right-click in the Substitution (Input/Output Text) list to display a menu containing **Add** and **Delete**. These provide the same functionality described below.

Input Text / Output Text — Specifies an input text string to substitute, and the output text that VERICUT will use in its place.

NOTE: Substitution input/output text is case sensitive.

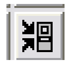
Add — Adds a new substitution record, after the highlighted record, to the list.

Delete — Deletes the selected substitution record from the list.

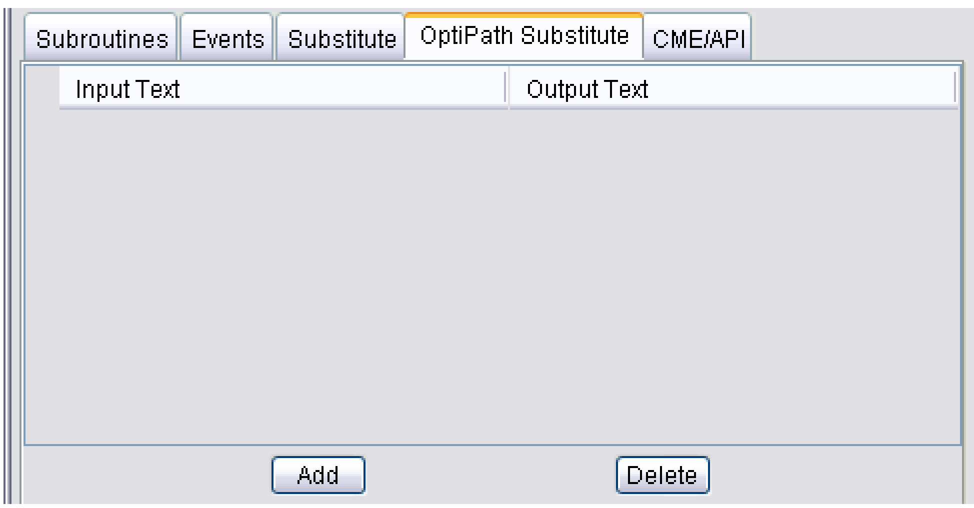
To Advanced Control Options window

Advanced Control Options, OptiPath Substitute tab

Location: **Configuration menu > Adv. Options**

VERICUT toolbar short cut: 

The features on this tab maintain text strings that are substituted when tool path files are optimized by OptiPath and written to the Optimized file. Using substitutions lets you to alter what VERICUT writes to optimized G-Code tool path file(s). Substitutions occur at the end of processing each G-Code data block, prior to outputting to the optimized tool path file.



OptiPath Substitution (Input/Output Text) list — Lists text string substitutions that will occur when specified input text is encountered.

Shortcut: You can right-click in the OptiPath Substitution (Input/Output Text) list to display a menu containing **Add** and **Delete**. These provide the same functionality described below.

Input Text / Output Text — Specifies an input text string to substitute, and the output text that VERICUT will use in its place.

NOTE: Substitution input/output text is case sensitive.

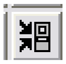
Add — Adds a new OptiPath substitution record, after the highlighted record, to the list.

Delete — Deletes the selected OptiPath substitution record from the list.

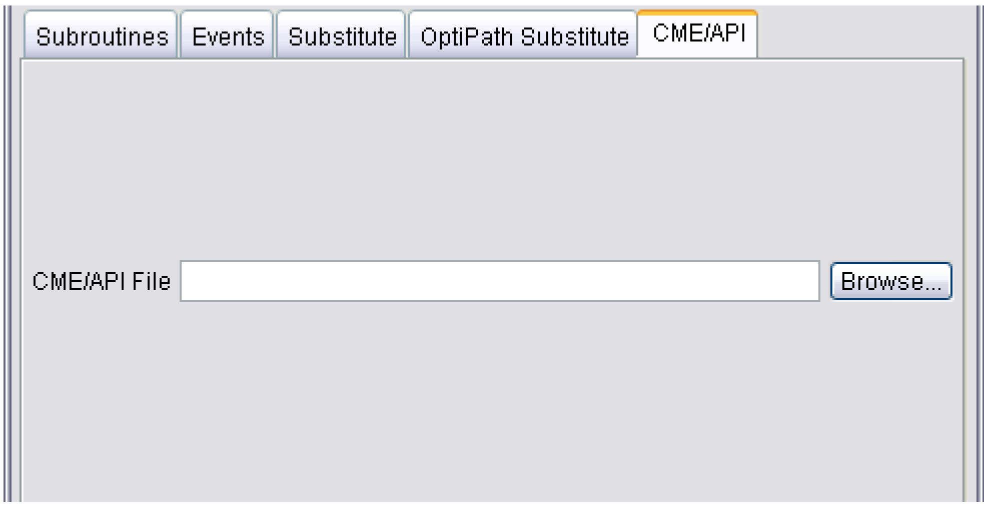
To Advanced Control Options window

Advanced Control Options, CME/API tab

Location: **Configuration menu > Adv. Options**

VERICUT toolbar short cut: 

Features on this tab enable you to specify a custom CME file, or CGTech Macro Executable file containing custom macros that assist the NC control with processing machine code data. This file can also supply "control settings" that allow users to further customize how machine code data is interpreted, and provide access to custom data tables for storing job, tool, and machine related data. A custom CME file is typically created by CGTech or other services organization using the **C Macro Extension – Application Programming Interface**, or **CME–API** (ref. **CME-API** in the *VERICUT Development Tools* section, in the *CGTech Help Library*). In this manner, end users can customize VERICUT's ability to simulate machine code data according to site-specific needs.



CME/API File — Opens a window to specify a custom CME file. Features on this window are standard file selection window features that enable you to navigate through directories, filter files, and type or select file names.

To Advanced Control Options window

Analysis Menu

X-Caliper (X-Caliper window)

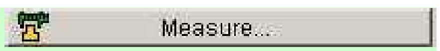
VERICUT Users:

VERICUT Location: **Analysis menu > X-Caliper**

VERICUT toolbar short cut: 

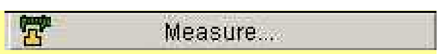
Mold and Die Users:

Mold and Die Location: **Analyze, View Files, Print page > Measure**

Notebook Feature: 

Cutter Grinder Users:

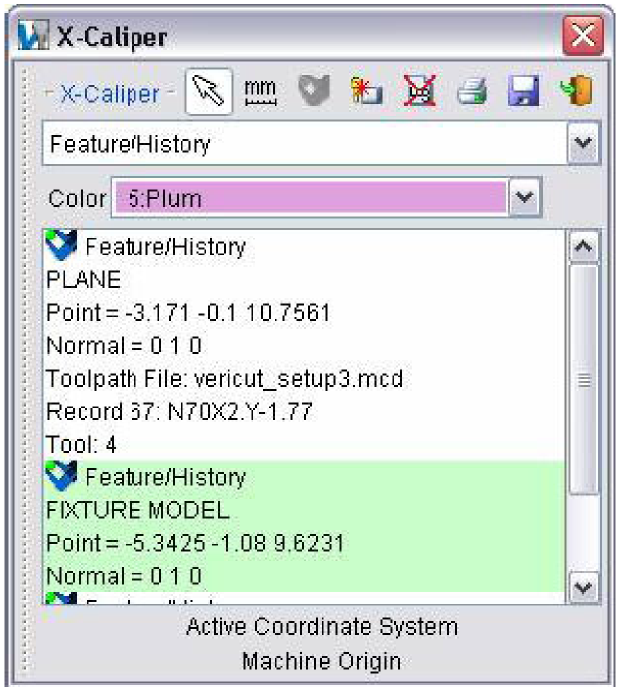
Cutter Grinder Location: **Analyze, View Files, Print page > Measure**

Notebook Feature: 

Opens a window that displays measurements, historical, and mathematical information about the VERICUT model. The appearance of the window will vary depending on the X-Caliper option being used. X-Caliper shows what is being measured on the model area by highlighting the "measure from" point (+) and the "measure to" point (o). Results from X-Caliper operations are displayed in the X-Caliper window. Measurements are described relative to the active coordinate system.

The X-Caliper window is also one of the **dockable windows** enabling you to dock it inside the VERICUT main window if you choose. See **Dockable Windows** in the *Getting Started with VERICUT* section of *VERICUT Help* for additional information.

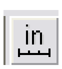
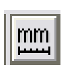
NOTE: When the **X-Caliper** window is docked, make sure that you click in the window so that it becomes the "active" window before using F1 to get help specific to the window. Otherwise F1 will go to the CGTech Help Library.



NOTE: Machined features can only be measured in a workpiece view.

Toolbar — The features in the X-Caliper Toolbar are common to all options except as noted below.

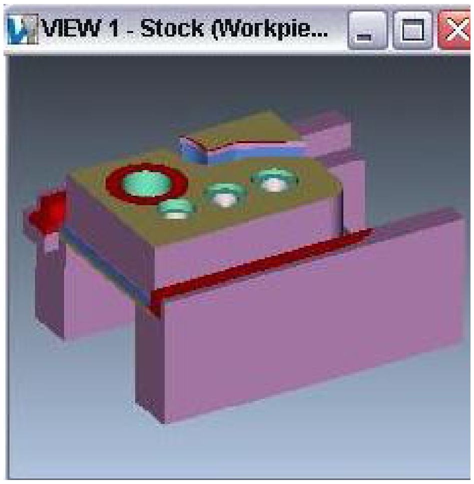
 **Enable/Disable Mouse Picks —** Toggles to enable/disable the ability pick features in a VERICUT workpiece view. Shown here in "enable" mode.

 or  **Set X-Caliper Units —** Toggle the icon between inch, and metric, to specify the units that you want X-Caliper data displayed in.

 **Measure —** Computes measurements based on the current X-Caliper window configuration. This feature is useful to have X-Caliper calculate measurements based on data keyed in, or edited data. This feature is only active for Distance/Angle, Stock Thickness, Air Distance and Closest Point options.

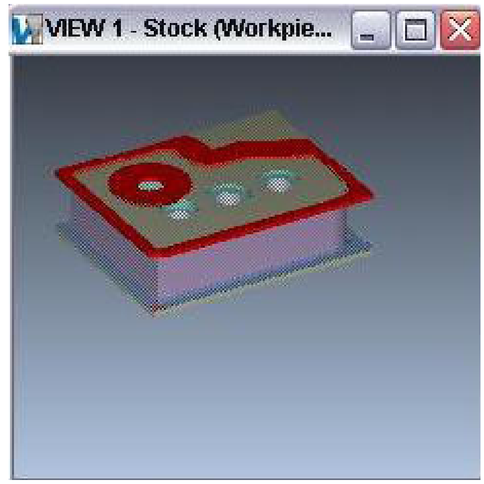
 **Display Tool/Stock Collisions —** Use to specifically highlight collisions between the tool holder and the stock model in the graphics area. Use the Feature/History option to obtain information about the collision (NC Program file, Record number, Tool ID, etc.). This feature is only active when holder/stock collisions occurred while cutting the part.

Disable Holder/Stock Collisions toggled "Off"



Notice that when toggled "Off", all collisions (holder/stock, cutter/fixture, etc.) are displayed.

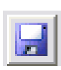
Disable Holder/Stock Collisions toggled "On"



Notice that when toggled "On", only holder/stock collisions are displayed. Including the collision areas that were cut away by subsequent cuts.

 **Clear X-Caliper Report —** Clears the results from the X-Caliper window.

 **Print —** Displays a Page Setup window enabling you to format the page, specify the printer, and print.

 **Save X-Caliper Report —** Saves the results from the X-Caliper window to a file. Use the file selection window that displays to specify the /path/filename for the file to be created.

 **Close Window —** Closes the X-Caliper window.

Options — Select the desired X-Caliper option from the pull-down list. The following options are available:

Feature/History — When a cut on the model is selected, this option displays the geometric description of the cut feature, and corresponding tool path history.

Distance Angle — Use to measure the distance and/or angle between any combination of two points, planes, conic axes, model or component origins, or edges formed by intersecting planes or cylinders.

Stock Thickness **—** Use to measure solid stock material.

Air Distance **—** Use to measure the air gap between two solid surfaces.

Closest Point **—** Use to measure the shortest distance from an XYZ point location to the closest point on the model surface.

Scallop **—** Use to measure the height of a scallop formed by parallel intersecting cylinders, typically cut by cutter corner radius.

Volume **—** Use to display the following volume values calculated for a selected stock workpiece: cut stock volume (cubic units), original stock model volume (cubic units), and volume of material removed from the original stock model.

Stock/Design Distance **—** Use to measure the distance between the cut stock and design models.

Highlight Same Planes **—** Use to display all "cut" features that lie in the same plane.

Hole Depth **—** Use to measure the depth of a hole, or the depth of a chamfer.

X-Caliper window — This is where the results from X-Caliper operations are displayed.

Active Coordinate System — Displays the name of the active coordinate system.

Also see "**X-Caliper**", in the *Using VERICUT* section, in the *CGTech Help Library*.

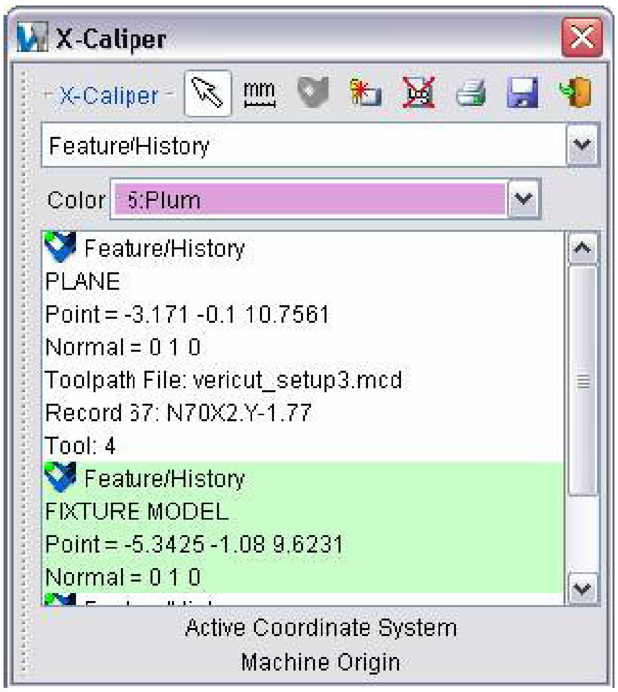
X-Caliper window, Feature/History

Location: **Analysis menu > X-Caliper: Feature/History option**

Toolbar shortcut: 

When a cut on the model is selected, the Feature/History option is used to display related cut feature (mathematical description) and tool path history. The historical data includes: NC program file name, record number, and tool path record text responsible for the cut. This function is useful to spot-check machined features, and determine the source of an error cut identified in red by VERICUT.

Sample X-Caliper window, Feature/History option:



Following are geometric features identified by VERICUT, listed alphabetically:

Auto-diff model — (X, Y, Z value and surface normal) an uncut surface on a model displayed after an auto-difference Solid method comparison.

Circle — (X, Y, Z center, axis vector and radius) a surface cut by a circular motion.

Cone — (X, Y, Z center, axis vector and side angle) a hole bottom cut by an angle tip drill, corner cut by a tapered endmill, or chamfer cut by a turning operation.

Cylinder — (X, Y, Z center, axis vector and radius) an inside corner cut by 2 linear motions having an interior angle less then 180 degree, fillet cut by a bull or ball endmill moving perpendicular to the tool axis, or diameter cut by a turning operation.

Ellipse — (X, Y, Z center, axis vector, major and minor focal distances) a ramping or plunging surface cut by the bottom of an endmill with a flat, or partially flat bottom.

Fixture model — (X, Y, Z value and surface normal) an uncut surface on a fixture model.

Holder Collision Area — (X, Y, Z value and surface normal) a holder/stock collision area cut stock model. Data also includes NC program file, record number and tool ID.

Plane — (X,Y,Z value and surface normal) a flat machined surface. Cylinders or circles cut using linear motions can also return this identity.

Ruled surface — (X,Y,Z value and surface normal) a surface cut by a multi-axis motion.

Sphere — (X, Y, Z center and radius) a surface cut by the end of a ball endmill.

Stock model — (X, Y, Z value and surface normal) an uncut surface on a stock model.

Torus — (X, Y, Z center, axis vector, major and minor radius) a fillet in a corner cut by a bull endmill, or fillet left on a diameter by a radiused cutter during a turning operation.

Torus sweep — (X, Y, Z center, axis vector and major radius) This feature is typical of an angled (ramping or plunging) surface cut by a bull endmill.

To X-Caliper window

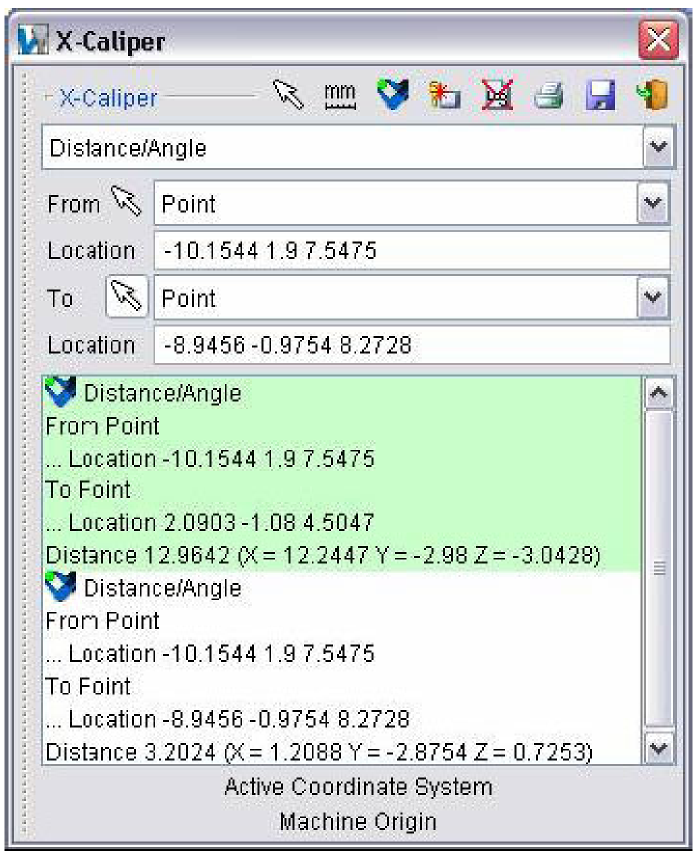
X-Caliper window, Distance/Angle

Location: **Analysis menu > X-Caliper: Distance/Angle option**

Toolbar shortcut: 

The Distance/Angle option is used to measure the distance and/or angle between any combination of two points, planes, conic axes, model or component origins, or edges formed by intersecting planes or cylinders. In general, select the "From" feature, then select the "To" feature. X-Caliper shows what is being measured on the model by highlighting the "measure from" point (+) and the "measure to" point (o) as well as a line representing the distance (or vectors representing the angle) between them.

Sample X-Caliper window, Distance/Angle option:



NOTE: Machined features can only be measured in a workpiece view.

From / To groups — Each group includes the **Feature**, **Location** and **Direction** used to specify what to measure from and what to measure to. Use the **From / To** arrows , to control which fields are filled in by screen pick data. You can select features on the VERICUT model, or enter number values (separated by blank spaces) directly into the fields. After selecting the **From** feature, VERICUT automatically transfers control to the **To** group. After selecting the **To** feature the X-Caliper window displays the measurement data.

Feature — Controls the types of features to measure between. Select the Feature type from the pull-down list.

Feature descriptions:

Point — The X, Y, Z of the pick point or entered point coordinates.

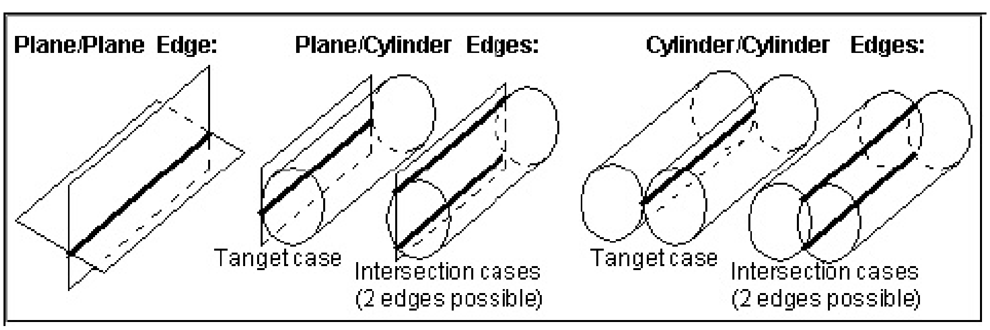
Plane — A flat, unbounded surface. If a curved surface is selected, data retrieved represents a plane tangent to the selected model surface at the pick point.

Axis — Centerline of a cylinder or conic.

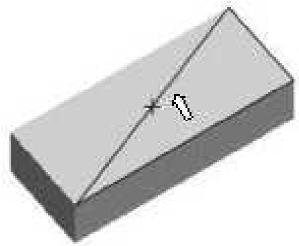
Edge — Edge formed by two intersecting planes or parallel cylinders. Special cases follow.

Plane/Cylinder edge — Only a cylinder with an axis parallel to the plane can produce an edge. Tangent conditions produce 1 edge. Intersecting conditions produce 2 possible edges, thus the one closest to the cylinder pick-point is used.

Cylinder/Cylinder edge — Only cylinders with parallel axes can produce an edge. Tangent conditions produce 1 edge. Intersecting conditions produce 2 edges, thus the one closest to the last cylinder pick-point is used.



Vertex — Enables you to easily select one of the six key points associated with the triangles representing the faces of a model. For each triangle, the points consist of the three vertices and the midpoint of each of the triangle's three sides. VERICUT selects the point closest to your mouse pick. If the mouse pick is over the cut stock, Vertex works in the same way as the Point feature described above.



Vertex is a useful tool for measuring the trace of tool/fixture collisions on uncut fixtures.

Model Origin — Origin of a selected model. Use the **View menu > View Axes** function to see model origin axes.

Component Origin — Origin of a selected component. Use the **View menu > View Axes** function to see component origin axes.

CSYS Origin — Origin of the selected coordinate system axis. Use the **View menu > View Axes** function to display the coordinate systems that are available.

CSYS Axis — The X, Y, or Z axis of the selected coordinate system. Use the **View menu > View Axes** function to display the coordinate systems that are available.

Location — Represents the XYZ coordinates of the screen pick.

Direction — Represents IJK of the normal vector at the location of the screen pick.

Data from the screen pick fills the **Location** and **Direction** data fields in the active **From** or **To** group. If needed, you can edit data in these fields, and then press **Measure** to obtain a measurement

Measurement results are displayed in the X-Caliper window. Data output includes the **From / To** values input and the measured distance and/or angle.

To X-Caliper window

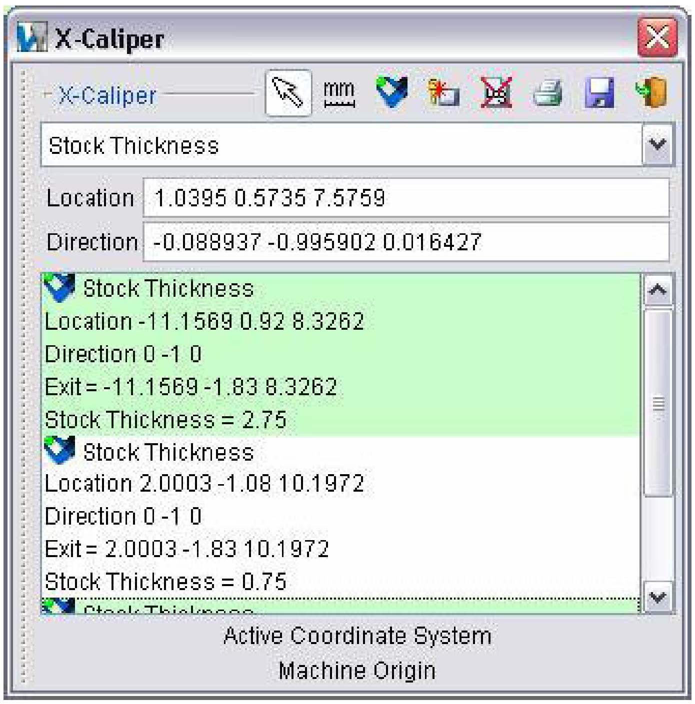
X-Caliper window, Stock Thickness

Location: **Analysis menu > X-Caliper: Stock Thickness option**

Toolbar shortcut: 

The Stock Thickness option is used to measure solid stock material.

Sample X-Caliper window, Stock Thickness option:



Location — Represents the XYZ coordinates of the point to measure from.

Direction — Represents IJK of the measurement direction vector into material.

Data from the screen pick fills the **Location** and **Direction** data fields. If needed, you can edit data in these fields, and then press **Measure** to obtain a measurement.

Measurement results are displayed in the X-Caliper window. Data output includes the **Location** and **Direction** values input, the XYZ point where the direction vector exits the material and the measured stock thickness between the **Location** and **Exit** points.

To X-Caliper window

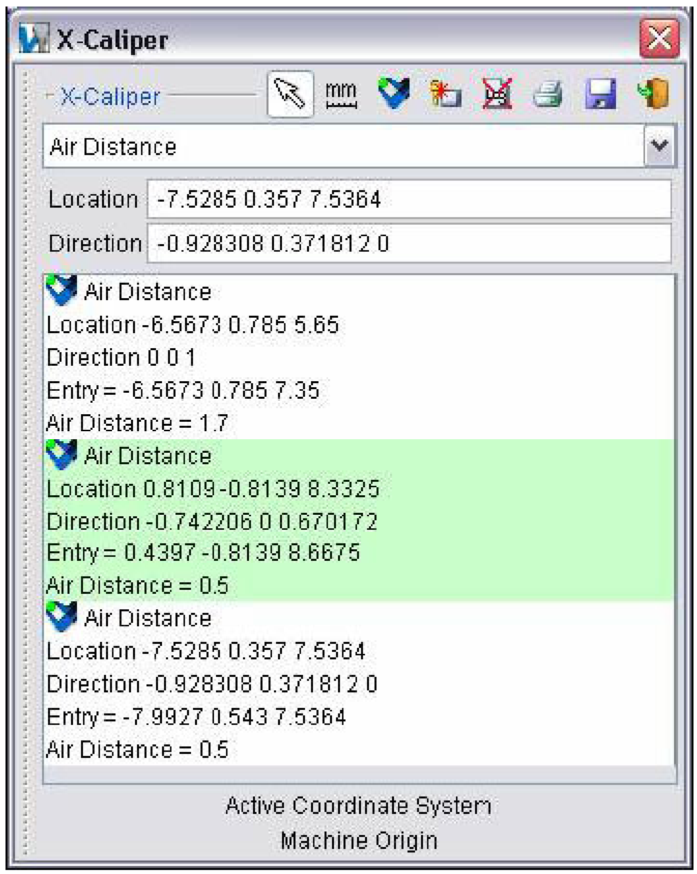
X-Caliper window, Air Distance

Location: **Analysis menu > X-Caliper: Air Distance option**

Toolbar shortcut: 

The Air Distance option is used to measure air gap between two solid surfaces.

Sample X-Caliper window, Air Distance option:



Location — Represents the XYZ coordinates of the point to measure from.

Direction — Represents IJK of the measurement direction vector away from solid material.

Data from the screen pick fills the **Location** and **Direction** data fields. If needed, you can edit data in these fields, and then press **Measure** to obtain a measurement.

Measurement results are displayed in the X-Caliper window. Data output includes the **Location** and **Direction** vector values input, the XYZ point where the direction vector re-enters solid material and the measured air distance between the **Location** and **Entry** points.

To X-Caliper window

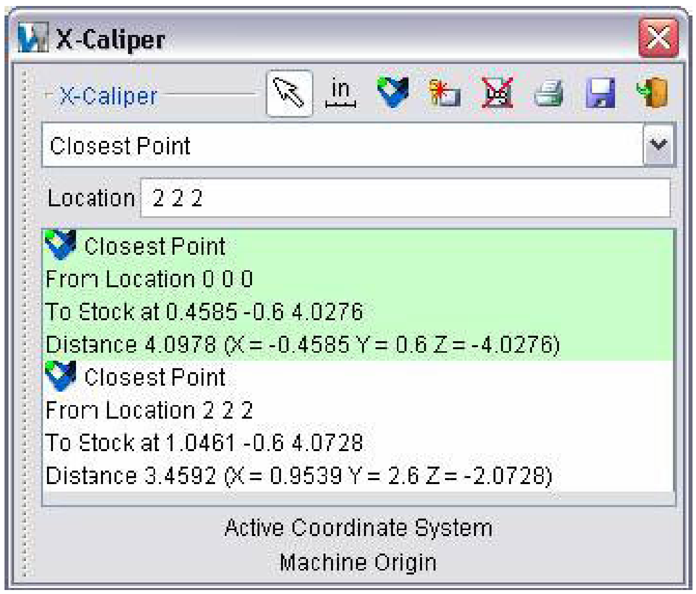
X-Caliper window, Closest Point

Location: **Analysis menu > X-Caliper: Closest Point option**

Toolbar shortcut: 

The Closest Point option is used to measure the shortest distance from an XYZ point location to the closest point on the model surface.

Sample X-Caliper window, Closest Point option:



Location — Represents the XYZ coordinates of the point to measure from. XYZ values must be manually entered. The Closest Point, "From" Location cannot be specified by selecting the location graphically.

Measurement results are displayed in the X-Caliper window. Data output includes the **From Location**, the XYZ coordinates of the closest point on the model, and the measured distance between the "**from**" point and the "**closest**" point on the model.

[To X-Caliper window]

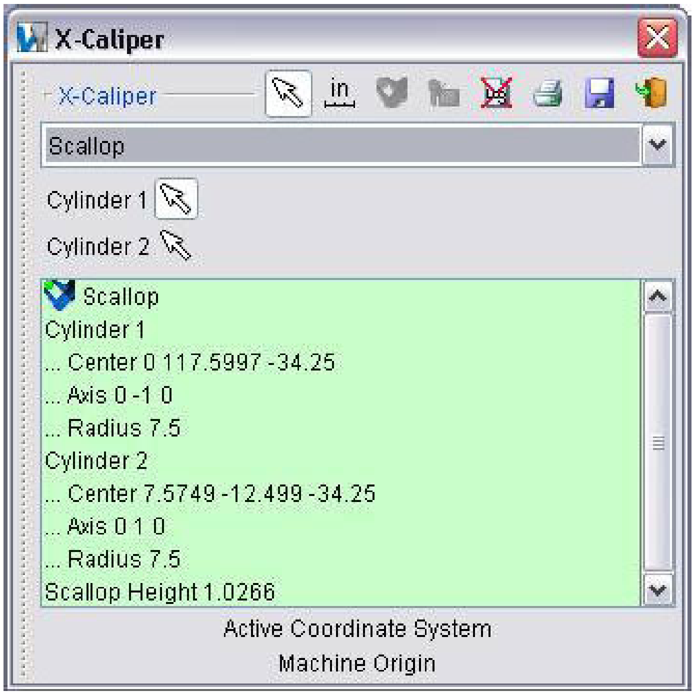
X-Caliper window, Scallop

Location: **Analysis menu > X-Caliper: Scallop option**

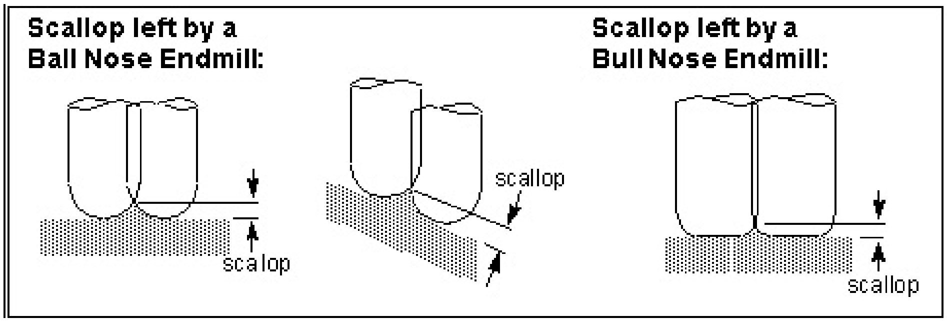
Toolbar shortcut: 

The Scallop option is used to measure the height of a scallop formed by two parallel intersecting cylinders. Select the two parallel and intersecting cylinders on the VERICUT model that form the scallop. X-Caliper identifies the cylinders being measured by displaying intersecting circles on the model.

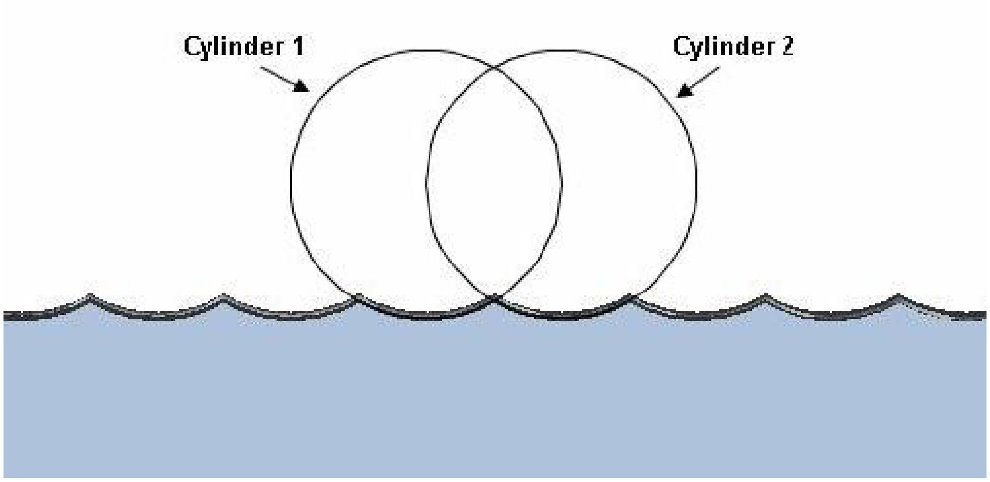
Sample X-Caliper window, Scallop option:



NOTE: Machined features can only be measured in a workpiece view.



Cylinder 1 / Cylinder 2 — Used to define the cylinders to measure from and to. Use the choice arrow  to control which cylinder is being picked. Select parallel cylinders on the VERICUT model. After selecting **Cylinder 1**, VERICUT displays a circle representing the cylinder in the graphics area and automatically transfers control to **Cylinder 2**. After selecting **Cylinder 2**, measurement results are displayed in the X-Caliper window.



Data output includes **Center** point, **Axis** vector and **Radius** values for each cylinder and the **Scallop Height** measurement.

To X-Caliper window

X-Caliper window, Volume

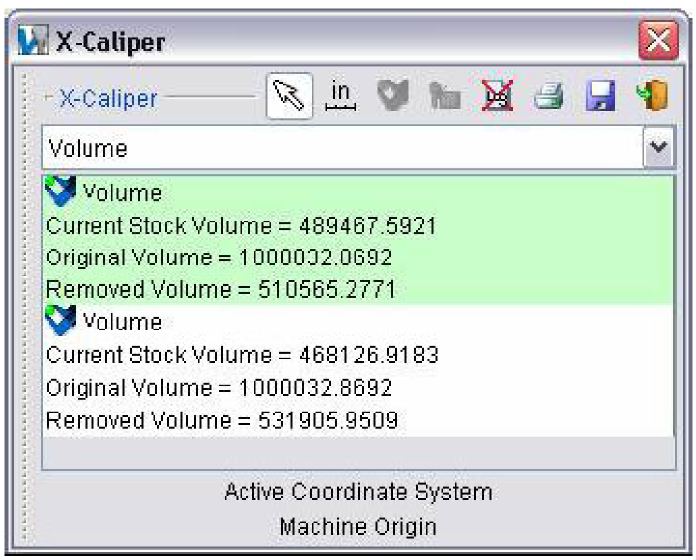
Location: **Analysis menu > X-Caliper: Volume option**

Toolbar shortcut: 

The Volume option calculates and displays the following volume values for a selected stock workpiece: current cut stock volume, original stock model volume, and volume of material removed from the original stock model. All volumes are in cubic units.

In addition to the total volume of the selected model, if model consists of several disjoint pieces, the volume of a selected contiguous piece of material can also be analyzed. In this case, a bounding box is drawn around the selected piece to indicate what is being calculated. If the model is only one piece, no bounding box is drawn.

Sample X-Caliper window, Volume option:



Calculated volumes are displayed in the X-Caliper window. Data output includes the **Current Stock Volume**, **Original Volume** and **Removed Volume**.

To X-Caliper window

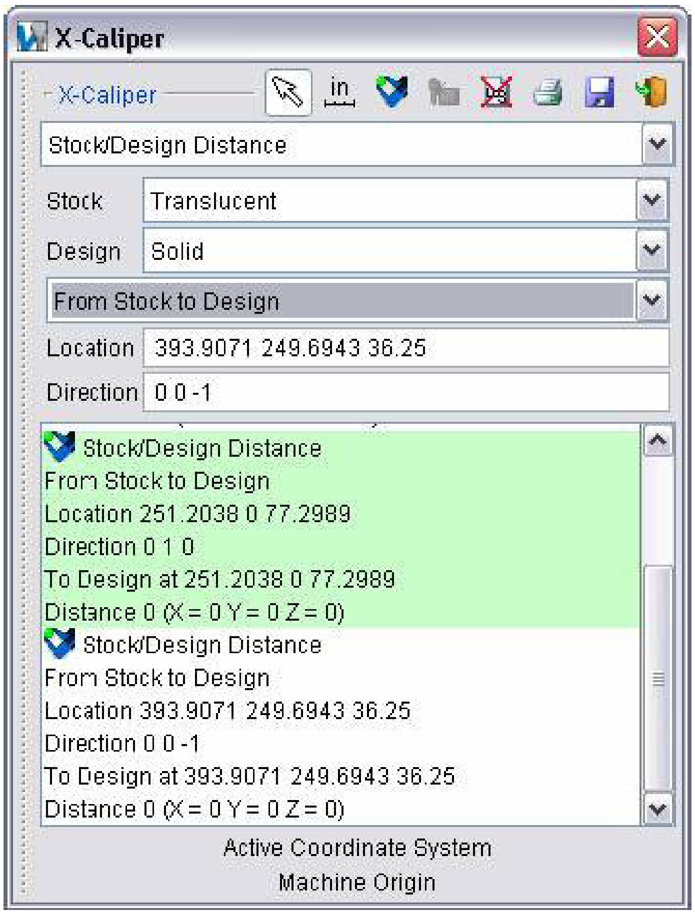
X-Caliper window, Stock/Design Distance

Location: **Analysis menu > X-Caliper: Stock/Design Distance option**

Toolbar shortcut: 

The Stock/Design Distance option enables you to measure the distance between the **cut stock** and the **design models**.

Sample X-Caliper window, Stock/Design Distance option:



Stock — Use to display the stock model in the workpiece view. Options are provided to display the models as **Solid** or **Translucent**, or turn the display **Off**.

Design — Similar to **Stock**, above, except that it applies to design model(s).

From Stock to Design / From Design to Stock — Use to specify the "**From**" and "**To**" components for the measurement.

Direction — Use to specify the I, J, and K components, separated by blank spaces, of a vector representing the direction of measurement from **Location** to the "**To**" component.

Location — Represents the XYZ coordinates of the point to measure from on the "**From**" component.

Direction — Represents IJK of the measurement direction vector toward the "**To**" component.

Pick a location on the "**From**" component in the workpiece view. X-caliper then measures the distance, normal to the pick location, to the nearest point on the "**To**" component. You can also enter the X, Y, and Z components, separated by blank spaces, of the desired measurement point on the "**From**" component.

Data from the screen pick fills the **Location** and **Direction** data fields. If needed, you can edit data in these fields, and then press **Measure** to obtain a measurement.

Measurement results are displayed in the X-Caliper window. Data output includes the measurement method, **Location** and **Direction** values related to the "**From**" component, the **XYZ point of contact** on the "**To**" component, and the measured **Distance** between the **Location** and the **point of contact** on the "**To**" component.

To X-Caliper window

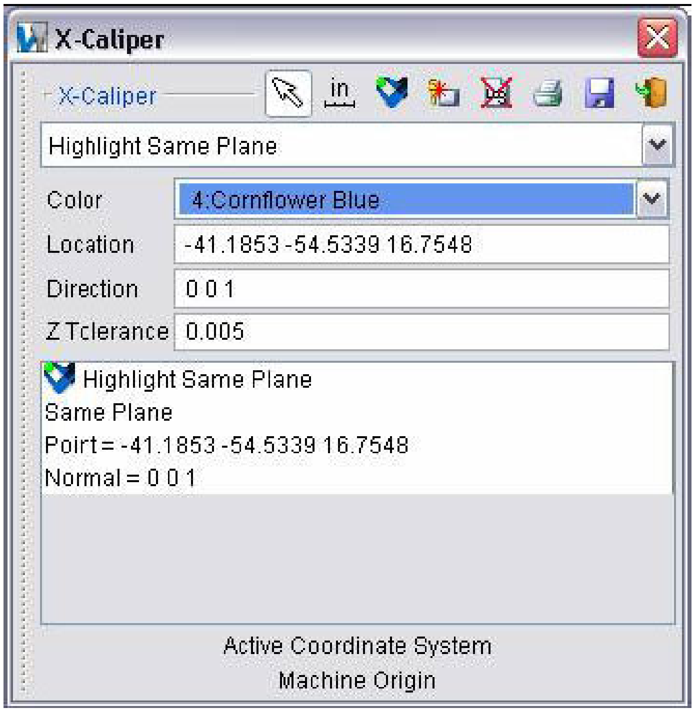
X-Caliper window, Highlight Same Plane

Location: **Analysis menu > X-Caliper: Highlight Same Plane option**

Toolbar shortcut: 

The Highlight Same Plane option enables you to see all "cut" features that lie in the same plane.

Sample X-Caliper window, Highlight Same Plane option:



Color — Use to select the color to highlight the planes. Select the desired color from the pull-down color list.

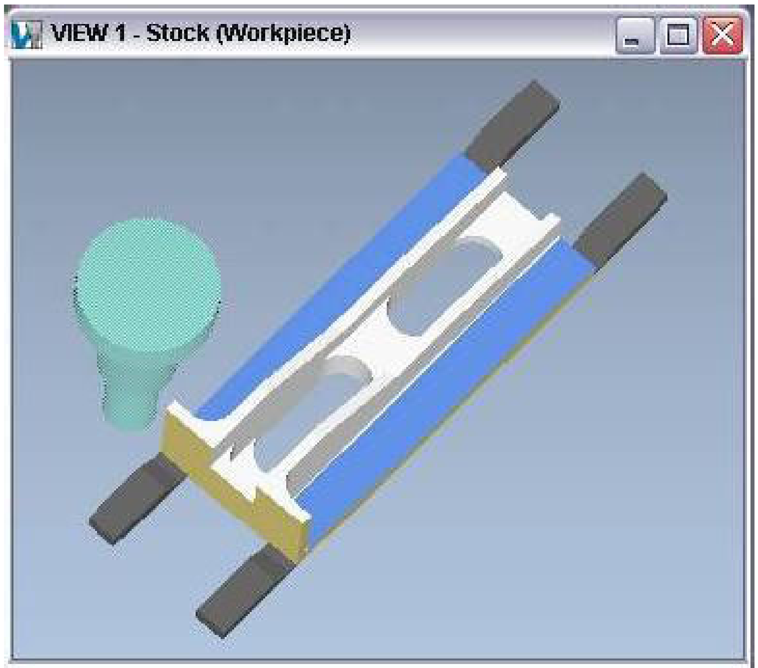
Location — Represents the XYZ coordinates of the screen pick.

Direction — Represents IJK of the plane's normal vector at the location of the screen pick.

Data from the screen pick fills the **Location** and **Direction** data fields.

Z Tolerance — Use to specify a tolerance value for determining "same" planes. All planes, within +/- the tolerance value, along the normal vector will be highlighted using the specified color.

Sample Graphics display after using the X-Caliper window, Highlight Same Plane option:



To X-Caliper window

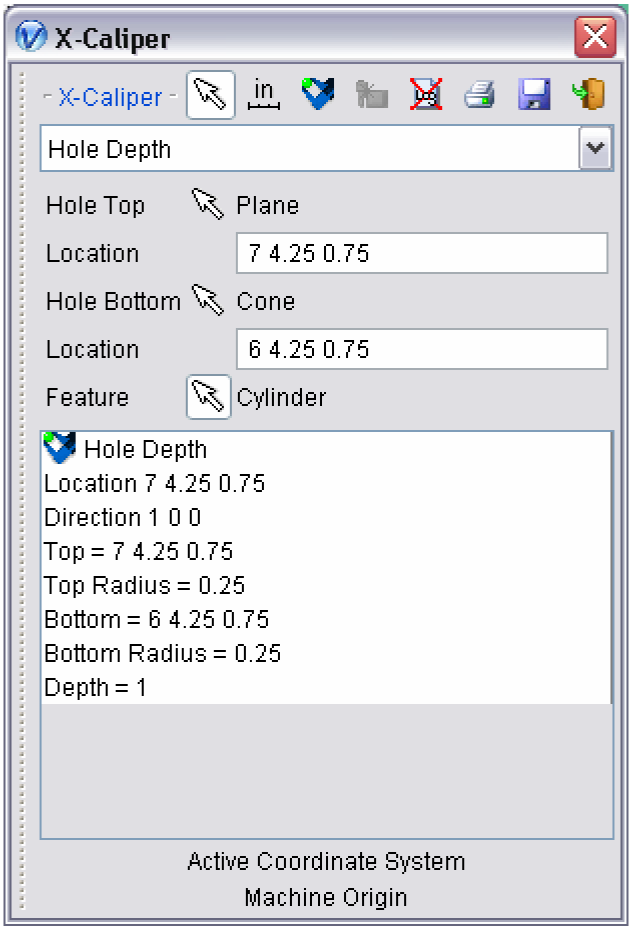
X-Caliper window, Hole Depth

Location: **Analysis menu > X-Caliper: Hole Depth option**

Toolbar shortcut: 

The Hole Depth option enables you to measure the depth of a hole, or the depth of a chamfer.

Sample X-Caliper window, Hole Depth option:



Hole Top — This is the feature that defines the top of the hole (generally a plane).

Location — Represents the XYZ coordinates of the screen pick.

VERICUT uses data from the screen pick to fill in the "feature type" and the **Location** data field.

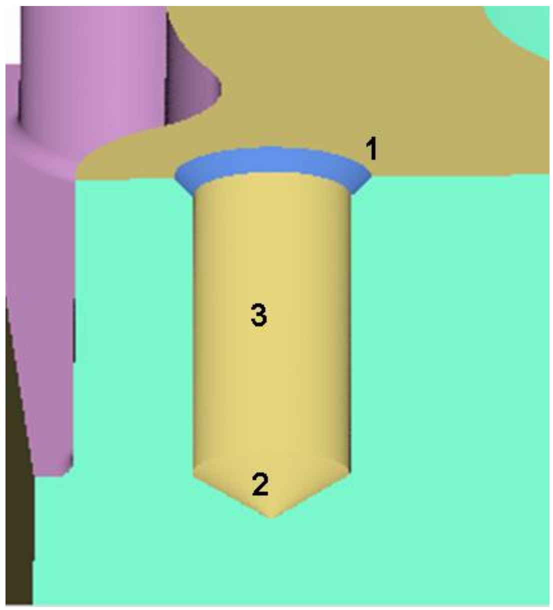
Hole Bottom — This is the feature that defines the bottom of the hole.

Location — Represents the XYZ coordinates of the screen pick.

VERICUT uses data from the screen pick to fill in the "feature type" and the **Location** data field.

Feature — Use to specify the feature that you want to measure.

Using Hole Depth



Refer to the picture of the hole cross section above for the following example.

To measure the depth of the hole:

1. In the X-Caliper window, click on the next to **Hole Top**, and then pick the feature representing the top of the hole. In this case it is a plane represented by **1** in the picture.
2. In the X-Caliper window, click on the next to **Hole Bottom**, and then pick the feature representing the bottom of the hole. In this case it is a cone represented by **2** in the picture.
3. In the X-Caliper window, click on the next to **Feature**, and then pick the feature that you want to measure. In this case it is a cylinder represented by **3** in the picture.
   X-Caliper will return the information shown in the X-Caliper window shown above. X-Caliper will calculate the depth from the intersection of the plane and the cylinder to the intersection of the cone and the cylinder.

To measure the depth of the top chamfer:

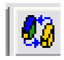
1. Pick the top plane for the Hole Top.
2. Pick the cylinder for the Hole Bottom.
3. Pick the cone representing the chamfer as the Feature.

To X-Caliper window

AUTO-DIFF (AUTO-DIFF window)

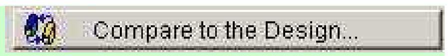
VERICUT Users:

VERICUT Location: **Analysis menu > AUTO-DIFF**

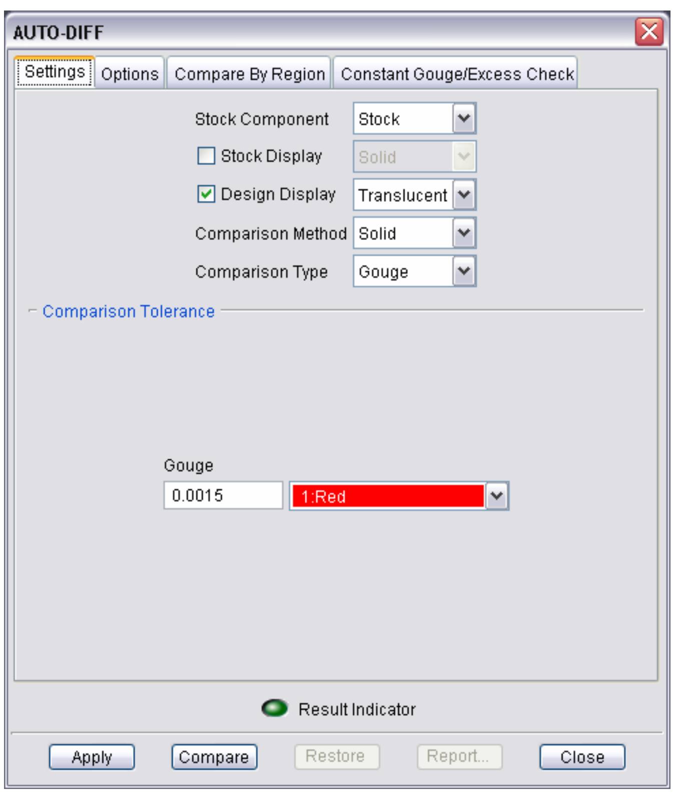
VERICUT toolbar short cut: 

Mold and Die Users:

Mold and Die Location: **Analysis and Viewing page > Compare to the Design**

Notebook Feature: 

Opens a window that enables you to compare design model data against the VERICUT simulated cut model. Using AUTO-DIFF, a licensed option, clearly identifies discrepancies between the cut model the intended design model. Errors are identified before actual machine time is wasted machining an incorrect part. You can use AUTO-DIFF interactively, in batch mode, or automatically via "**VERICUT-COMMAND**" records in the tool path file. After using AUTO-DIFF to identify gouges and excess material, use other VERICUT functions such as X-Caliper and Zoom (ref. **View menu > Orient**) to measure and help determine the source of the errors.



Settings tab **—** Features on this tab control the AUTO-DIFF comparison, and what you see as a result.

Options tab **—** Features on this tab are used to set processing and reporting options.

Compare By Region tab **—** Features on this tab enable you to select a region to perform AUTO-DIFF on.

Constant Gouge/Excess Check tab **—** Features on this tab enable you to specify tolerance values for gouge and excess checking, and turn Constant Gouge Check On/Off.

Compare Status Light — The compare status light enables you to determine at a glance the status of your AUTO-DIFF Compare. The color of the light, and label displayed, will change depending on the status as described below.

Result Indicator — displays when the AUTO-DIFF "compare" has not finished running, or that the "compare" has been "restored".

No Differences — displays when the AUTO-DIFF "compare" has finished running and no gouges/excesses were detected.

Error Detected, Check Report — displays when the AUTO-DIFF "compare" has finished running and gouges/excesses were detected.

Apply — Applies the current AUTO-DIFF window settings.

Compare — Compares the design model(s) specified by the AUTO-DIFF window settings. AUTO-DIFF automatically builds the report after the comparison has been done. If you are only interested in seeing the graphical results (no report), you can abort report building by pressing (**Stop**) after you see the message "**Creating AUTO-DIFF report**".

TIP: If you are only interested in the graphical AUTO-DIFF results, toggle "on" the **Disable Report** feature, on the Options tab, to prevent AUTO-DIFF from automatically building the report.

Restore — Restores the VERICUT model as it was before performing the AUTO-DIFF operation.

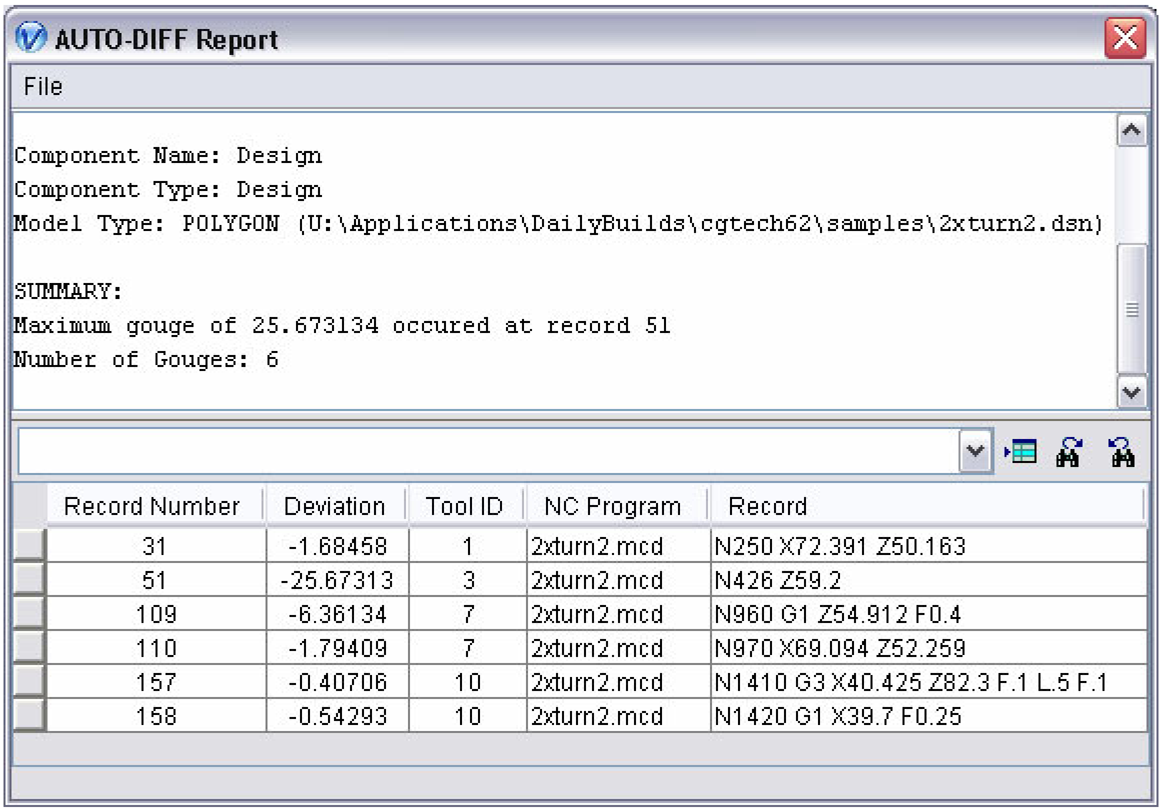
Report — Opens the AUTO-DIFF Report window containing an AUTO-DIFF Report file that identifies information related to errors detected by AUTO-DIFF.

Close — Closes the AUTO-DIFF window without applying window changes.

Also see "**Optimizing your NC Program**", in the *Using VERICUT* section, in the *CGTech Help Library*.

AUTO-DIFF Report window

Accessed by selecting **Report** in the AUTO-DIFF window, the AUTO-DIFF Report window provides information related to errors detected by AUTO-DIFF.



File menu

Save As —

Text File — Save the AUTO-DIFF report as a text file.

CSV File— Save the AUTO-DIFF report in CSV (**C**omma **S**eparated **V**alue) file format that can be opened as a spreadsheet. When saving the report in CSV format, only the lower "table" portion of the report is saved. The information in the top of the AUTO-DIFF Report window is not saved.

Print — Opens a window enabling you so print the AUTO-DIFF report.
Exit — Closes the AUTO-DIFF report window.

Each record in the AUTO-DIFF Report table provides the following information:

- The record number in the NC program listing responsible for the error
- The amount of the deviation from the design model.
- The ID of the tool being used at the time of the error.
- The name if the NC program being used.
- The NC record that was being processed at the time the error occurred.

Selecting an error record in the table highlights the location of the error on the model displayed in the VERICUT graphics area. Select any column heading in the table to sort the report by the entries in that column.

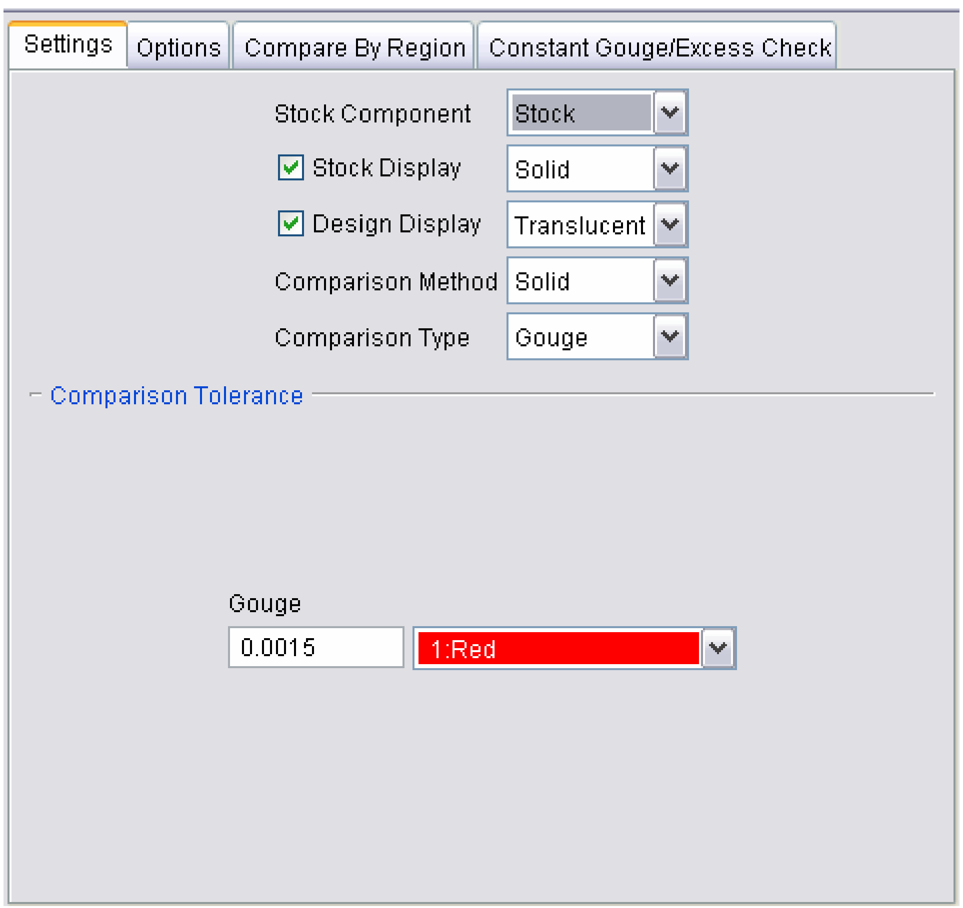
To AUTO-DIFF window

AUTO-DIFF window, Settings tab

Location: **Analysis menu** > **AUTO-DIFF**

VERICUT toolbar short cut: 

Features on this tab control the AUTO-DIFF comparison, and what you see as a result. You can compare four types of design data with AUTO-DIFF: solid models, surface data, inspection points and profiles. The **Comparison Method** controls which type of design data is compared, while **Comparison Type** determines the error condition checked for. **Comparison Tolerance** features set the inspection tolerances and the display colors used for identified gouge/excess errors. The appearance of the **Comparison Tolerance** portion of the **Settings tab** will vary depending on the **Comparison Method Setting**.



Stock Component — When multiple Stock components are defined, this option controls which one is used in the comparison. The Design component used in the comparison is that which is connected to, or connected to the same component as the chosen Stock component.

Stock Display — When selected, displays the stock model in the AUTO-DIFF view. Options are provided to display these models as Solid, Translucent, or Lines. The available options will vary depending on the Comparison Method that is selected.

Design Display — Similar to Stock Display above, except applies to design model(s).

Comparison Method — Controls the method of comparing design model data against the VERICUT model. The comparison method also re-configures the Settings tab to reflect the tolerances and other data required to compare the data as requested. All methods generate an AUTO-DIFF model that can be analyzed or saved as an In Process file. Regardless of the method chosen, the design model must first be loaded prior to comparison (ref. **Modeling window: Model tab**). Design models can be loaded at any time.

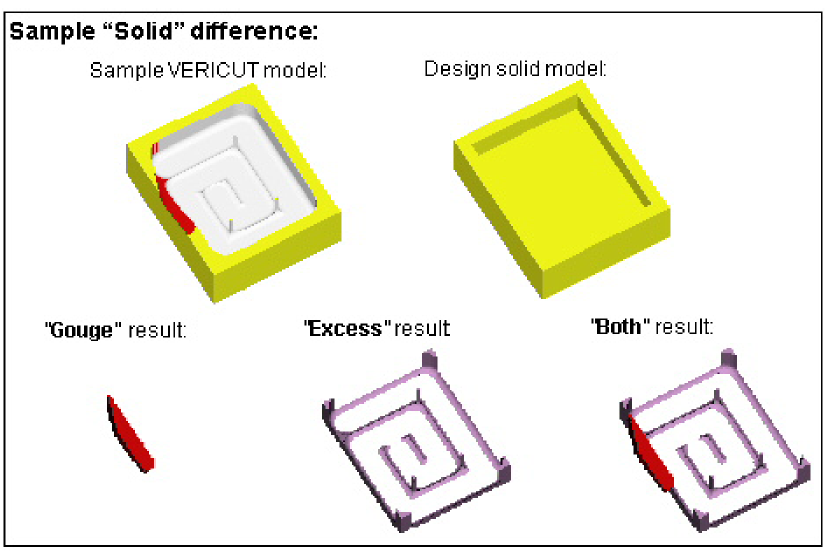
Options:

Solid — Compare a solid design model by subtracting solid models from each other. Solid models must be "watertight"-in other words, completely enclosed surfaces.

You can check solid design model integrity by loading it as a stock model, then select **Analysis menu > X-Caliper window: Volume tab** to analyze the volume of the stock model. After checking the volume, the following problems indicate the design model needs repair, or is suitable only for the **Surface** comparison method:

- small holes or gaps appear in the model
- model falls apart-is partially displayed
- model disappears completely

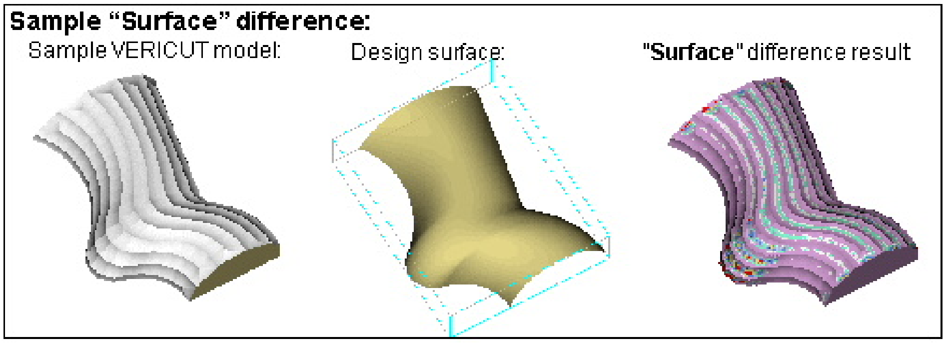
AUTO-DIFF "Solid" method example



Surface — Compare a surface, set of surfaces, or skin of a solid design model or VERICUT Solid by overlaying the surface on the cut model. VERICUT only compares where the design surface and cut model overlap. Surface comparison provides multiple colors and tolerances that produce a "rainbow" affect to show the extent of errors.

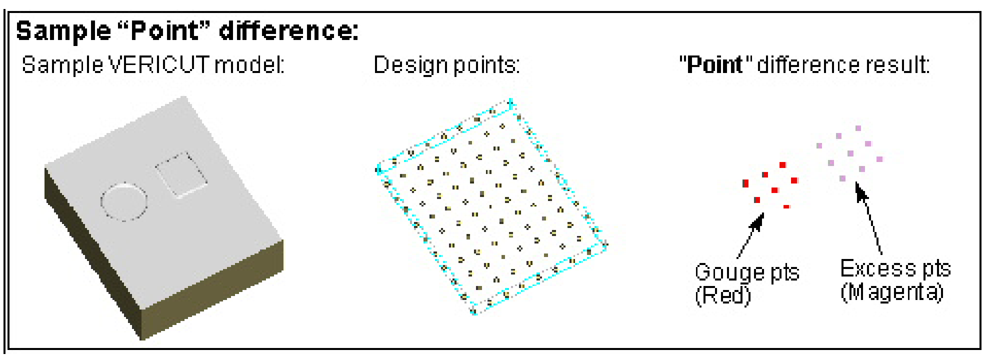
NOTE: VERICUT Solids can only be used for Standard (not FastMill) cutting mode.

AUTO-DIFF "Surface" method example



Point — Compare a set of design inspection points by overlaying the points on the cut model. VERICUT searches from each point for an intersection with the model surface. Surface normal vectors control the direction in which to search. Only points associated with errors are seen in the results. Points associated with gouges have the Error color-typically Red, while points associated with excess material have the 5th defined color in the Shade Color list (typically Magenta).

AUTO-DIFF "Point" method example



Profile— Enables you to compare the turning profile of the cut stock with profile of a design model.

When **Comparison Method** is set to **Profile**, up to four profiles (Gouge, Excess, Stock, and Design) may be displayed in the "Profile" view. Each can be displayed in one of two modes: **Solid** or **Lines**. The display mode for Stock and Design profiles is specified using the **Design Display / Stock Display** features. The display mode for Gouge and Excess is specified using the **Draw Mode** option in the **View Attributes window** for the currently active "Profile" view.

When display mode is set to "Solid", Stock and Design profiles are displayed the "component" color unless the "model" color is different than the "component" color. In this case, Stock and Design profiles are displayed the color of the 1st model. Gouge and Excess profiles are displayed in the color selected with the **Comparison Tolerance** "color" feature.

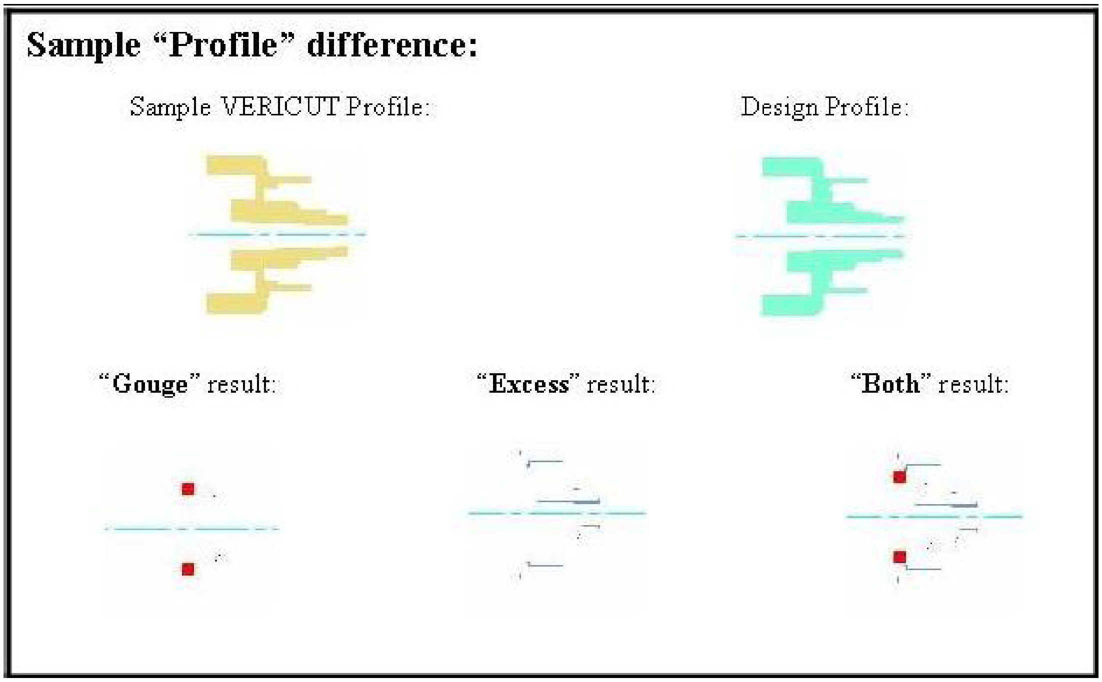
When display mode is set to "Lines", the same color scheme described above is used for "uncut" sections of the profile. "Cut" sections of the profile are displayed in the "cut" color.

X-Caliper can be used to analyze any of the profiles (Gouge, Excess, Design, and Stock profiles) currently displayed in a "Profile" view. Saving an In-Process file (IP file) stores all four profile types.

Any valid VERICUT model type can be used as a design model for an AUTO-DIFF Profile comparison. The design profile which is compared to stock profile is a Boolean union of the individual profiles of all Design Component models associated with specified Stock.

If a design model is a Cone, Cylinder, SOR, or IGES Profile, and its Z-axis is aligned with the turning axis, its profile is built directly from its definition profile and is "exact". In all other cases, the profile is built by spinning the model about the turning axis (as is common with other turning applications) and is an approximation. The smaller the Cut Tolerance that is used, the better the approximation. If the "design component" consists of multiple models, the individual model profiles are merged into one design profile.

AUTO-DIFF "Profile" method example



Comparison Type — Controls the error type(s) checked for by the AUTO-DIFF operation.

Options:

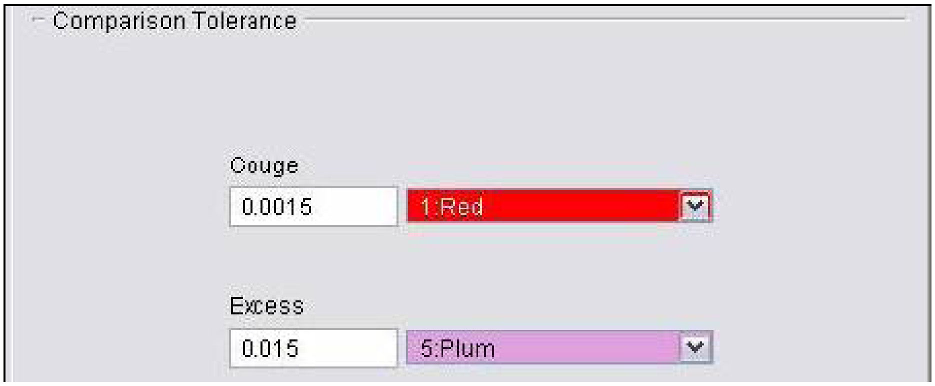
Gouge
Excess
Both

Comparison Tolerance Features

Comparison Tolerance features are used to specify the tolerances and colors used by AUTO-DIFF. The degree to which VERICUT can detect discrepancies is also affected by **Cutting Tolerance**. For best results set cutting tolerance smaller than the smallest tolerances used by AUTO-DIFF.

Comparison Tolerance features displayed will vary depending the on the **Comparison Method** and **Comparison Type** settings.

Solid comparison tolerances

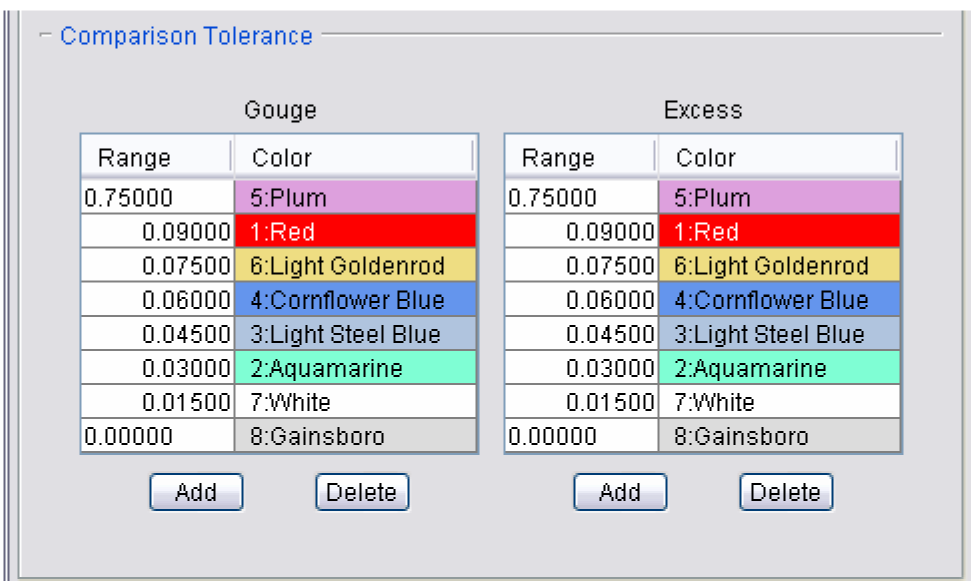


Gouge — Tolerance used when checking for gouges during a **Solid** comparison. Gouges detected by AUTO-DIFF are shaded in the associated gouge color.

Excess — Similar to **Gouge** above, except applies when checking for excess material.

NOTE: "Solid" Gouge/Excess tolerance/color values are independent of "Profile" Gouge/Excess tolerance/color values and are stored as separate values in .VcProject file.

Surface comparison tolerances



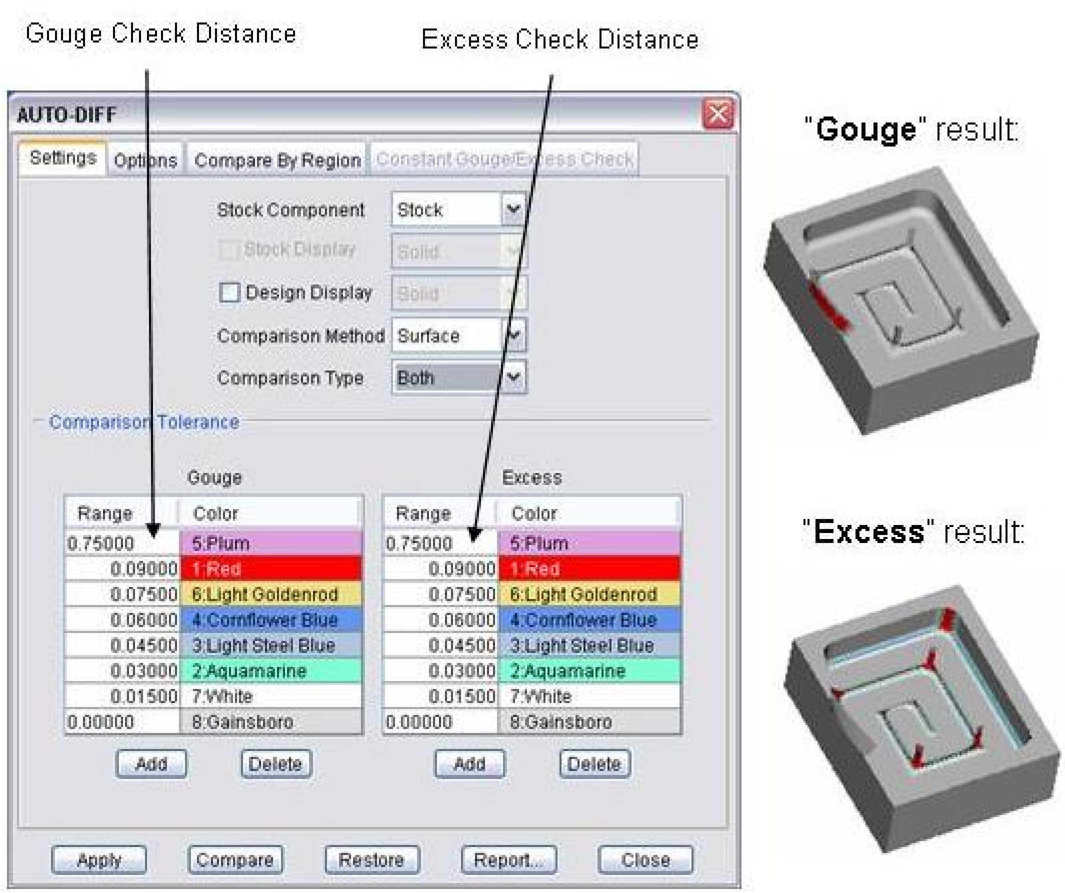
Gouge Range/Color list — Range of tolerances used when checking for gouges during a **Surface** comparison. Multiple tolerance values can be specified, in addition to the "Gouge Check Distance" (in this case, 0.75000) used to limit checking, and the Range value of 0.00000 which represents surfaces cut exact with no deviation from the design model. If more than six tolerance values are specified, a scroll bar will be added to the table. Use the **Add** and **Delete** buttons to add/remove tolerance values. The table row containing the Range value of 0.00000 cannot be deleted, and the 0.00000 value cannot be modified.

Gouges detected by AUTO-DIFF are shaded in the gouge color(s) associated with the depth of the gouge. Each pair of tolerance values represents a range of gouge depths to which a gouge color is associated. The gouge color used for each range is the color assigned to the lower tolerance value in the range. For example, in the gouge table shown above, all gouges with depths in the range between 0.04500 and 0.06000, will be displayed Light Steel Blue.

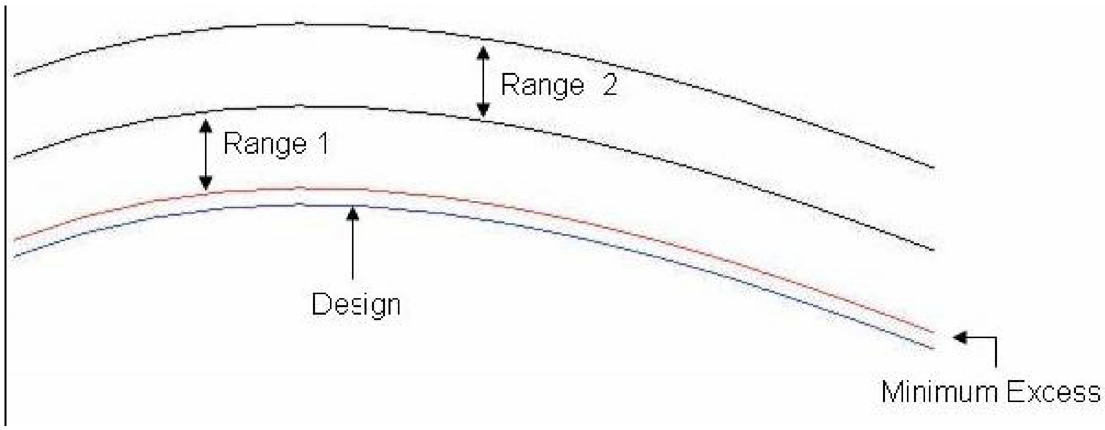
The color associated with the "Gouge Check Distance" can be different for the Gouge and Excess tables, but the color associated with the Range value of 0.00000 is forced to be the same for both tables.

Excess Range/Color list — Similar to **Gouge Range/Color list** above, except applies when checking for excess material.

Sample "Surface" comparison tolerances/colors:



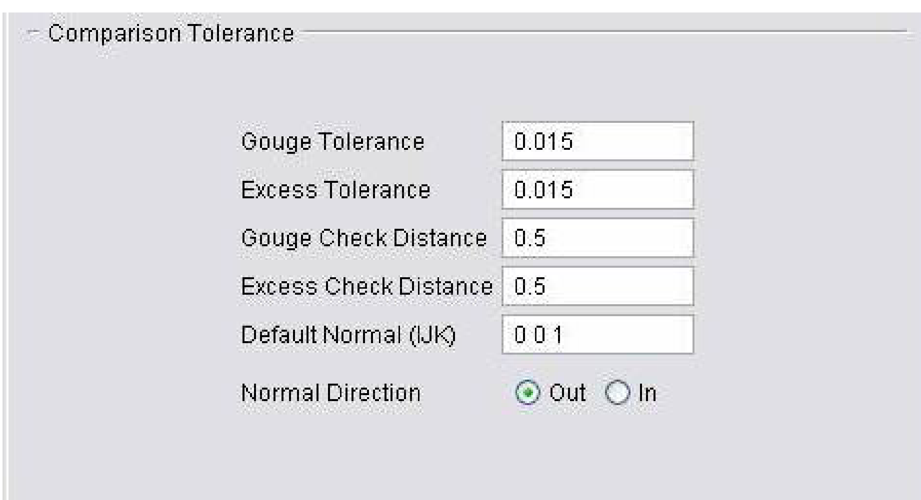
Minimum Excess — This feature enables you to specify a value and color to be used to represent an offset from the design surface. The **Minimum Excess** value is applied to the design surface in all directions. During the compare, any Excess value less than the **Minimum Excess** value is considered to be a gouge and displayed the **Minimum Excess** color. Excess material is measured from the **Minimum Excess** to the **Range** value (see below)



NOTE: The amount of gouge and excess reported is with respect to the offset design, not the original design. In other words, zero deviation is where the cut stock excess with respect to the original design exactly equals to the given "**Minimum Excess**" value. This also is where the Excess color table range starts from.

While in this mode, AUTO_DIFF > Report shows both gouges and excesses with respect to the condition described above. If a cut produces both a gouge and an excess, AUTO-DIFF reports the gouge (no matter what the value of the excess). This is different from the Report behavior when "Comparison Type" = "Both" where the biggest deviation would be reported.

Point comparison tolerances



Gouge Tolerance — Tolerance used when checking for gouges during a **Point** comparison. Gouges detected by AUTO-DIFF are shaded in the gouge color specified for a **Solid** comparison.

Excess Tolerance — Similar to **Gouge** above, except applies when checking for excess material conditions.

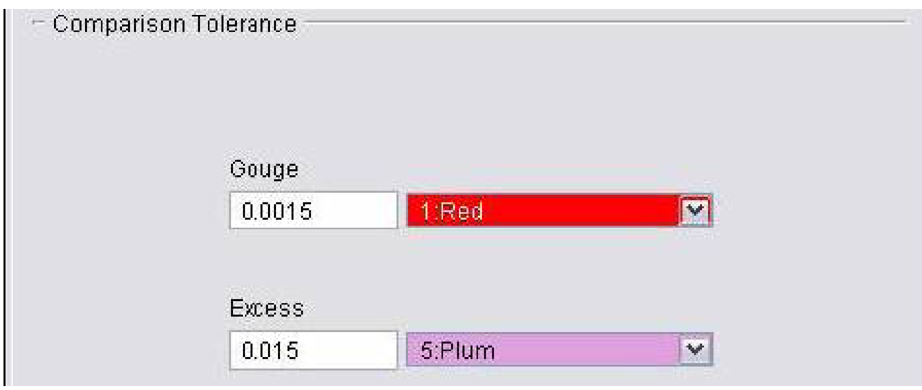
Gouge Check Distance — Maximum distance to check for the cut model surface in the gouge direction. Distance is measured from the inspection point along direction of the normal vector used with the inspection point.

Excess Check Distance — Similar to **Gouge Check Distance** above, except applies when checking for excess material.

Default Normal (IJK) — Default normal vector in which to check during a Point comparison when vector data is not present in the Design Points file. Enter three values separated by spaces, for example "0 0 1".

Normal Direction — Indicates if surface normal vectors associated with design points are pointing outward (**Out**) or inward (**In**) relative to the design model.

Profile comparison tolerances



The Profile comparison tolerances are applied "exactly", without tessellating the stock and/or design profiles. An arc offset by the tolerance value, will also be an arc and will be recognized by X-Caliper as such. The **Profile Gouge/Excess** tolerance values are independent of the **Solid Gouge/Excess** tolerance values described above, and are stored as separate values in .VcProject file.

Gouge — Tolerance used when checking for gouges during a **Profile** comparison. Gouges detected by AUTO-DIFF are shaded in the associated gouge color.

Excess — Similar to **Gouge** above, except applies when checking for excess material.

NOTE: "Profile" Gouge/Excess tolerance/color values are independent of "Solid" Gouge/Excess tolerance/color values and are stored as separate values in .VcProject file.

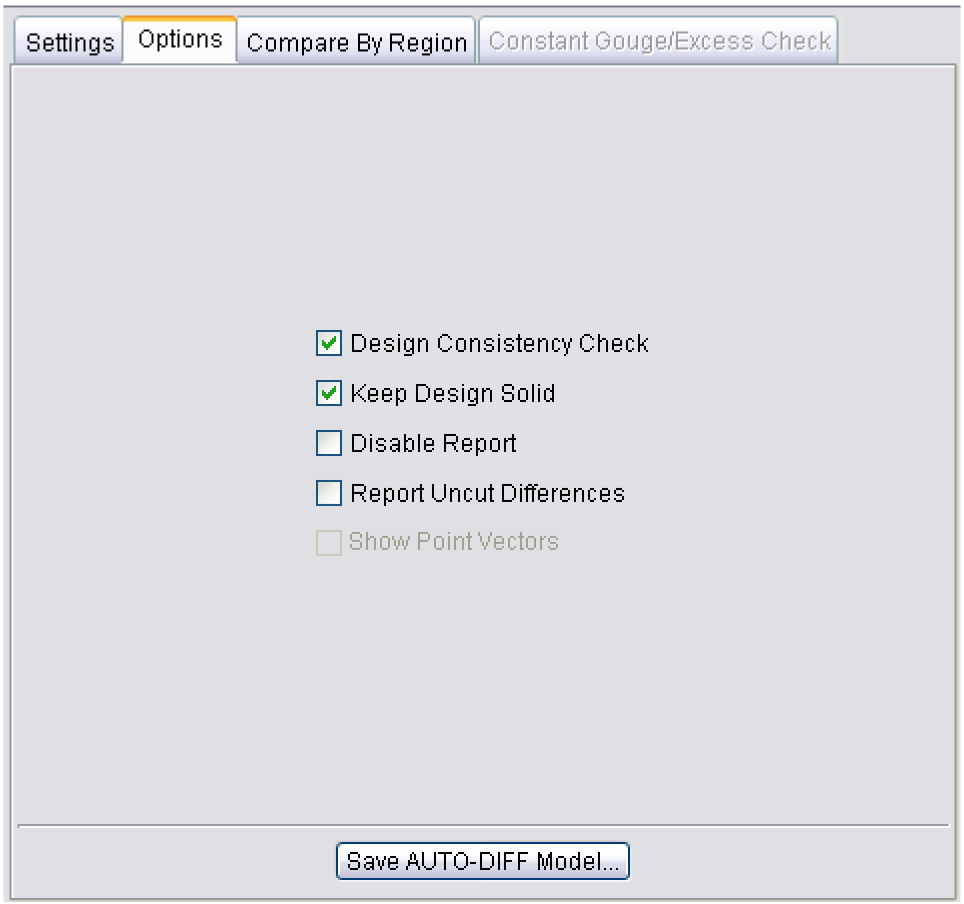
To AUTO-DIFF window

AUTO-DIFF window, Options tab

Location: **Analysis menu** > **AUTO-DIFF**

VERICUT toolbar short cut: 

The features on this tab are used to set processing and reporting options.



Design Consistency Check—When selected during a "**Solid**" comparison, checks consistency of solid design models, including: check for "watertight" solid- repair improperly trimmed surfaces (overlaps and gaps), and reconstruct insignificant missing surfaces. This option is highly recommended for design models creating from importing IGES data, and other model files which produce unsatisfactory results from the comparison.

When selected during a "**Profile**" comparison, **Design Consistency Check** is applicable when building a profile from an STL, VERICUT Polygon, or VERICUT Solid design model.

While checking design model consistency can take a while, it provides the best insurance for reliable AUTO-DIFF results. If you've used the model in VERICUT before and know the quality is good enough, you can save time by clearing this checkbox. AUTO-DIFF will take less time, but results may not be reliable if the solid is not water-tight.

Keep Design Solid — When selected, keeps the solid design model created by AUTO-DIFF so it can be used in additional comparisons. This action can reduce AUTO-DIFF time if multiple comparisons are made, but will require additional computer resources to keep the design solid. The design solid is kept until the VERICUT model is reset, a design model is removed or deleted, or the checkbox is cleared and **Apply** is pressed. Saving an IP file also contains the kept design solid. Clear this checkbox to have AUTO-DIFF create a new design solid for each comparison.

Disable Report — Enables you to turn off the AUTO-DIFF Report feature to reduce AUTO-DIFF processing time. To generate the report, the processor must search through all cut records and lists the maximum error for each bad cut record. This can take a significant amount of time for large NC program files. When is toggled "on", you will still see the AUTO-DIFF results in the graphic area. **Disable Report** is only active when **Comparison Method** is set to **Solid**, **Surface** or **Profile**.

Report Uncut Differences — When selected, differences between the design and uncut stock are added to the AUTO-DIFF report.

Show Point Vectors — Displays the measurement vector associated with each point for the "Point" Comparison Method. The Vector is drawn whether AUTO-DIFF compare is active or not.

Save AUTO-DIFF Model — Use to save gouge or excess data as a VERICUT solid model file (.vct). Selecting **Save AUTO-DIFF Model** displays a dialog enabling you to specify a directory path and filename for the model file. If both gouge and excess models are present, two files will be created (gouge_filename.vct and excess_filename.vct, where filename is the filename you entered in the **Save AUTO-DIFF Model** dialog.

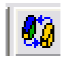
This option is only available for use if all of the following are true (otherwise it is grayed out):

1. **Standard Cut Mode** is active (not FastMill).
2. **Constant Gouge Check** is turned off.
3. AUTO_DIFF **Comparison Method** is set to **Solid** or **Profile**.
4. AUTO-DIFF **Compare** has been processed.

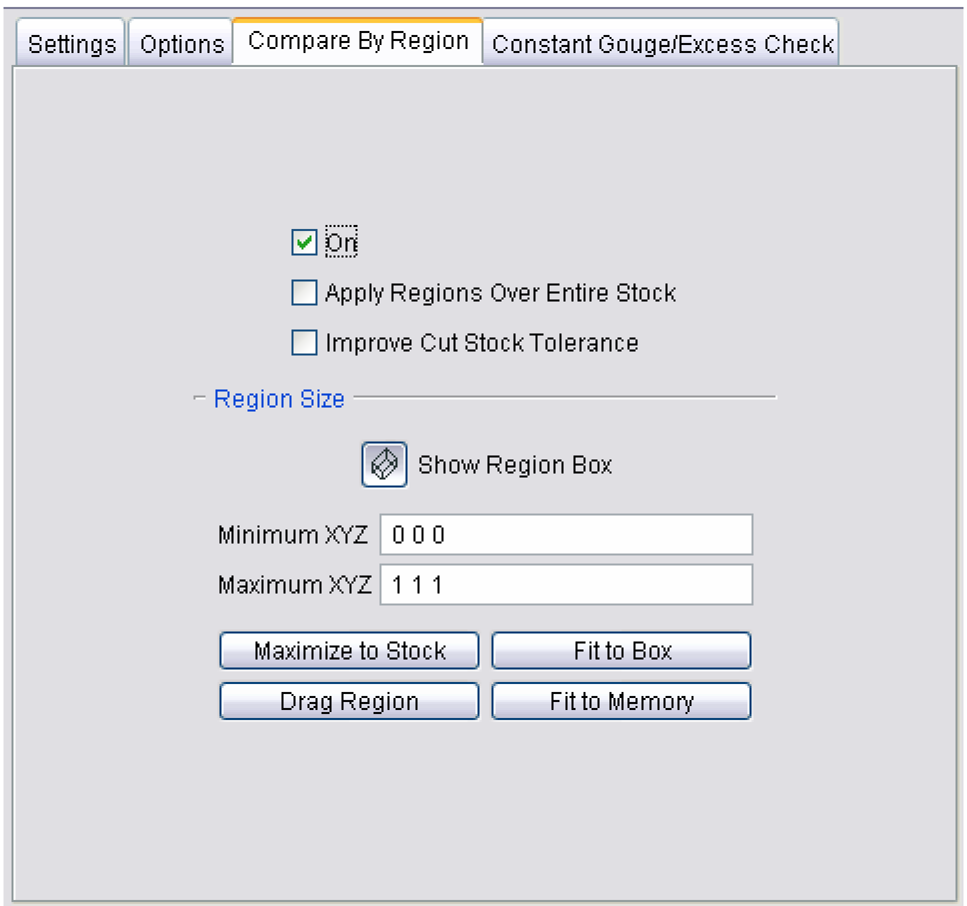
To AUTO-DIFF window

AUTO_DIFF window, Compare By Region tab

Location: **Analysis menu** > **AUTO-DIFF**

VERICUT toolbar short cut: 

The features on this tab enable you to select a region to perform AUTO-DIFF on. **Compare By Region** is created for very large parts that would require too much memory for AUTO-DIFF to process the entire part. Optionally, cut stock can be internally re-cut within the region with a better Cut Tolerance before doing AUTO-DIFF. The **Compare By Region tab** is not available when **Comparison Method** is set to "**Profile**".



On — When selected, activates the Compare by Region feature. When toggled "on", AUTO-DIFF comparison only processes the cut stock and design volumes within the specified "region".

Apply Regions Over Entire Stock — The check box, **Apply Regions Over Entire Stock**, does the following:

- Calculates a region that can achieve the specified Comparison Tolerance within the computer's physical memory.
- Re-processes that region of the cut stock at a tolerance that ensures the Comparison Tolerance can be achieved.
- Does the specified AUTO-DIFF comparison for the region.
- Repositions the region and does AUTO-DIFF multiple times over the entire cut stock.

During AUTO-DIFF processing, the Region Box is displayed at each position over the cut stock.

When Compare is pressed and this box is checked, a notice dialogue is displayed with the text, "Processing may take several minutes depending on tolerance, size of the cut stock, memory and processor speed. Press the stop button in the main window to cancel AUTO-DIFF.", OK.

When the **Apply Regions Over Entire Stock** feature is active, the **Region Size** features are disabled.

Improve Cut Stock Tolerance — When selected, the Cut Stock is re-evaluated at a new tolerance based on the specified minimum AUTO-DIFF tolerance. Having this box checked affects the results from both the **Fit to Memory** and the **Apply Regions Over Entire Stock** features. Typically the **Comparison Tolerance** is smaller than the **Cut Tolerance**, so processing will take longer when **Improve Cut Stock Tolerance** is active.

Region Size features

Show Region Box — Click on the  icon to show / not show the "region box", representing the AUTO-DIFF comparison region, in the Workpiece View.

Minimum XYZ — Use to enter the minimum X, Y, and Z values (relative to the workpiece coordinate system), separated by spaces. These values represent the X, Y, Z coordinates of first corner point of the "region box".

Maximum XYZ — Use to enter the maximum X, Y, and Z values (relative to the workpiece coordinate system), separated by spaces. These values represent the X, Y, Z coordinates of the diagonal corner point of the "region box".

Maximize to Stock — Use to size the "region box" to the maximum extent of the union of the stock and design volumes.

Fit to Box — Use to drag a 2 dimensional rectangle (similar to a "zoom" box) over the cut stock. The rectangle is projected into 3 dimensional space to define the "region box".

Drag Region — Enables you grab a corner of the "region box" and drag it to re-size the "region box". The new corner location is automatically projected onto the cut stock in 3 dimensional space. Grab the mid-point of one of the "region box" edges to re-size the "region box" by dragging a face. Click with the mouse pointer in the point representing the center of the "region box" and drag around the Cut Stock to reposition the "region box" without resizing it.

Fit to Memory — Use to automatically size the "region box" based on the available memory for AUTO-DIFF. A short delay occurs while memory is analyzed.

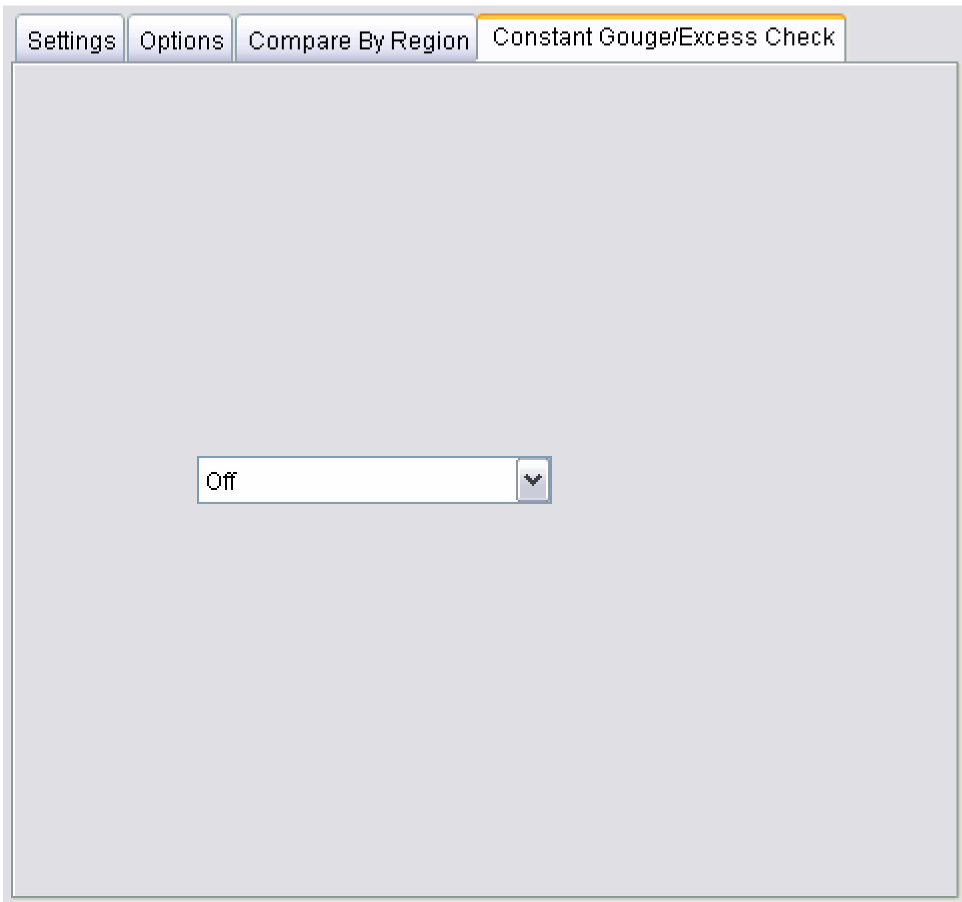
To AUTO-DIFF window

AUTO-DIFF window, Constant Gouge/Excess Check tab

Location: **Analysis menu** > **AUTO-DIFF**

VERICUT toolbar short cut: 

Enables you to specify a maximum gouge value, and a minimum excess value, and turn Constant Gouge Check on and off. Gouge areas found by Constant Gouge Check during NC Program processing are displayed in the assigned Error color.



Off — Turns **Constant Gouge Check** off.

Maximum Allowable Gouge — Select **Maximum Allowable Gouge** from the pull-down list and enter the value in the text field. Then select **Apply** to turn Constant Gouge Check on.

Minimum Allowable Excess — Select **Minimum Allowable Excess** from the pull-down list and enter the value in the text field. Then select **Apply** to turn Constant Gouge Check on.

NOTES:

1. A design model must be present in the component tree for Constant Gouge Check to detect potential gouges.
2. Constant Gouge Check is not compatible with FastMill. When cutting in FastMill mode, use one of the AUTO-DIFF comparison methods instead.
3. AUTO-DIFF and Constant Gouge Check are mutually exclusive. To use the AUTO-DIFF features Constant Gouge Check must be turned "off". If you are using AUTO-DIFF, you must click the **Restore** button before you will be able to turn Constant Gouge Check "on".

Tolerance — Specify a tolerance value for **Constant Gouge Check**. Only gouges, greater than the specified tolerance, will be displayed. You can turn **Constant Gouge Check** "off", change the tolerance, and reprocess the NC program at any time.

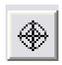
NOTE: A VERICUT Solid (.vct) file can be used for a design model when using **Constant Gouge Check** to find gouges during NC program processing.

To AUTO-DIFF window

Comparator (Comparator window)

VERICUT Users:

VERICUT Location: **Analysis menu > Comparator**

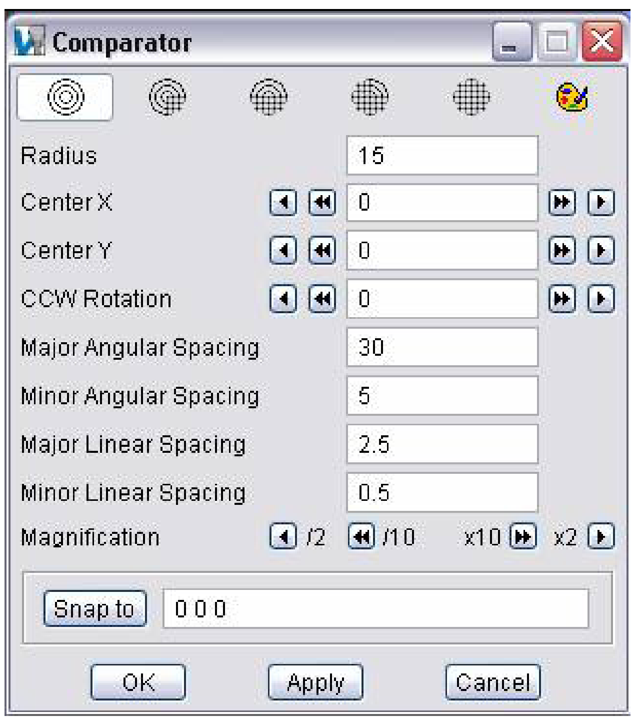
VERICUT toolbar short cut: 

Cutter Grinder Users:

Cutter Grinder Location: **Analyze, View Files, Print page > Comparator Grid**

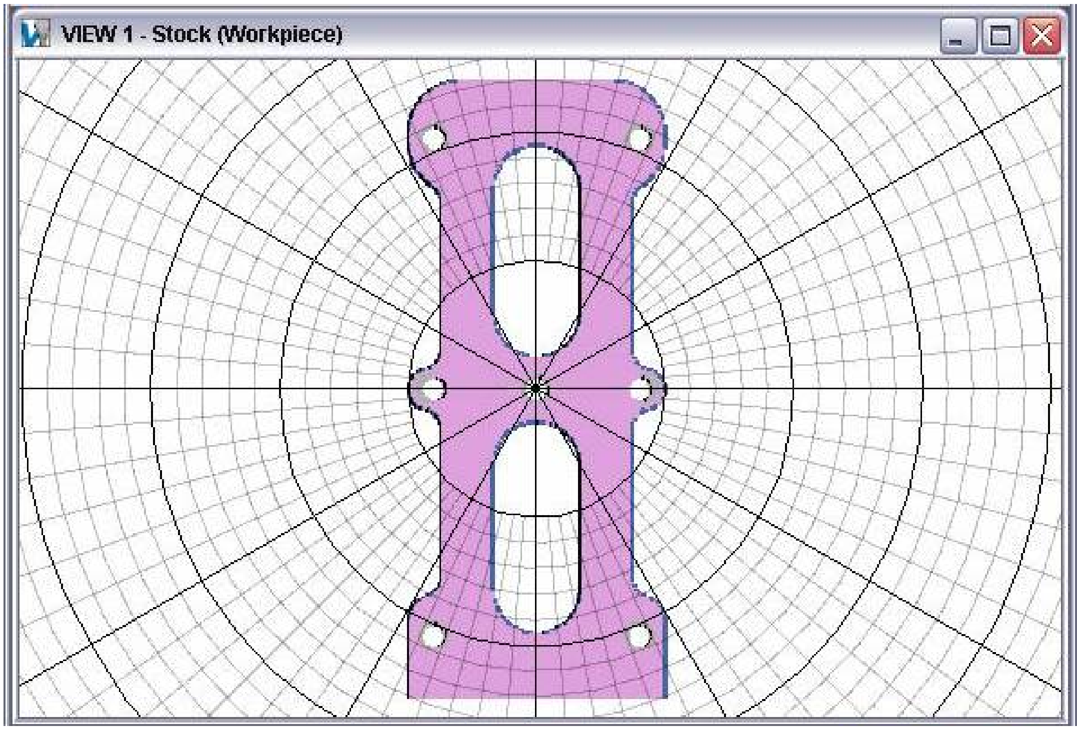
Notebook Feature: 

Opens the **Optical Comparator window** enabling you to overlay a polar or rectangular grid over a workpiece or machine view to emulate an optical comparator chart.



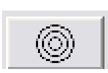
Use this feature to measure distance and angular relationships of features on your part like you would with optical comparator.

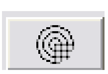
Comparator Grid Example

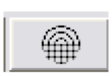


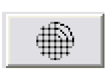
The grid can be displayed in any view but only in one view at a time. The grid is always displayed in the "active" view. The grid is always displayed parallel to the screen and its center (0,0) position is related to the origin of either the workpiece coordinate system or the machine coordinate system depending on the type of view that is active.

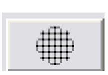
The grid will be displayed as long as the Optical Comparator window is open. Closing the Optical Comparator window removes the grid from the display.

 **Full Polar —** Displays a full polar grid .

 **Polar Grid With Linear Quadrant —** Displays a polar grid with one linear grid quadrant.

 **Half Polar / Half Linear —** Displays a grid that is half polar and half linear.

 **Linear Grid With Polar Quadrant —** Displays a linear grid with one polar quadrant.

 **Full Linear —** Displays a full linear grid.

 **Color Pallet —** Select a color from the pallet for the optical comparator grid. The default color is VERICUT's Foreground color. Click on the "**X**" in the upper right corner of the color pallet to close it.

Use the following options to specify the size and position of the grid as well as define the spacing between major and minor grid lines. Units are the same as the workpiece or machine depending on the type of view that is "active".

Radius — Specify the radius of the grid that you want to display.

Center X — Specify the horizontal position of the center of the grid. Enter a value in the text field. A negative number moves the center of the grid the specified number of units to the left of the coordinate system origin. A positive number moves the grid to the right.

You can also move the horizontal position of the grid center by selecting one of the arrow buttons. Selecting either the **<** or **>** button moves the center of the grid to the left or right of its current position by the value specified for **Minor Linear Spacing**. Selecting either the **< <** or **>>** button moves the center of the grid to the left, or right, of its current position by the value specified for **Major Linear Spacing**.

Center Y — Specify the vertical position of the center of the grid. Enter a value in the text field. A negative number moves the center of the grid the specified number of units down from the coordinate system origin. A positive number moves the grid up.

You can also move the vertical position of the grid center by selecting one of the arrow buttons. Selecting either the **<** or **>** button moves the center of the grid down, or up, from its current position by the value specified for **Minor Linear Spacing**. Selecting either the **< <** or **>>** button moves the center of the grid down, or up, from its current position by the value specified for **Major Linear Spacing**.

CCW Rotation — Use to rotate the grid about its center point. Enter an angle value in the text field. A positive number rotates the grid the specified number of degrees in a counterclockwise direction from its default (zero) position. A negative number rotates the grid the specified number of degrees in a clockwise direction from its default (zero) position.

You can also rotate the grid about its center by selecting one of the arrow buttons. Selecting either the **<** or **>** button rotates the grid clockwise or counterclockwise from its current position by the value specified for **Minor Angular Spacing**. Selecting either the **<<** or **>>** button rotates the grid clockwise, or counterclockwise, from its current position by the value specified for **Major Angular Spacing**.

Major Angular Spacing — Specify the number of degrees between the "major" grid lines of a polar grid.

Minor Angular Spacing — Specify the number of degrees between the "minor" grid lines of a polar grid.

Major Linear Spacing — Specify the distance between "major" grid lines of a linear grid.

Minor Linear Spacing — Specify the distance between "minor." grid lines of a linear grid.

Magnification — Use this option to quickly change the size of the grid by either a factor of 2, or a factor of 10. Selecting either the **<** or **>** button reduces or enlarges the size of the grid by a factor of 2. Selecting either the **< <** or **>>** button reduces or enlarges the size of the grid by a factor of 10.

Snap to — Use this option to center the grid on a specific feature of the workpiece or machine. Click on the text field so that it is highlighted. Select a feature in the graphics area and the grid center will move to that location. You can also enter the X, Y, and Z coordinates of the workpiece/machine coordinate system in the text field. Selecting the **Snap to** button (or <**Enter**> on the keyboard) moves the grid to the specified location.

OK — Saves the current settings and dismisses the **Optical Comparator window**.

Apply — Applies all of the current settings to the displayed grid without dismissing the **Optical Comparator window**. You can also use the <**Enter**> key after changing an individual setting to immediately apply the change to the current grid display.

Cancel — Dismisses the **Optical Comparator window** without saving the settings.

Also see "**Optical Comparator**", in the *Using VERICUT* section, in the *CGTech Help Library*.

NC Program Review

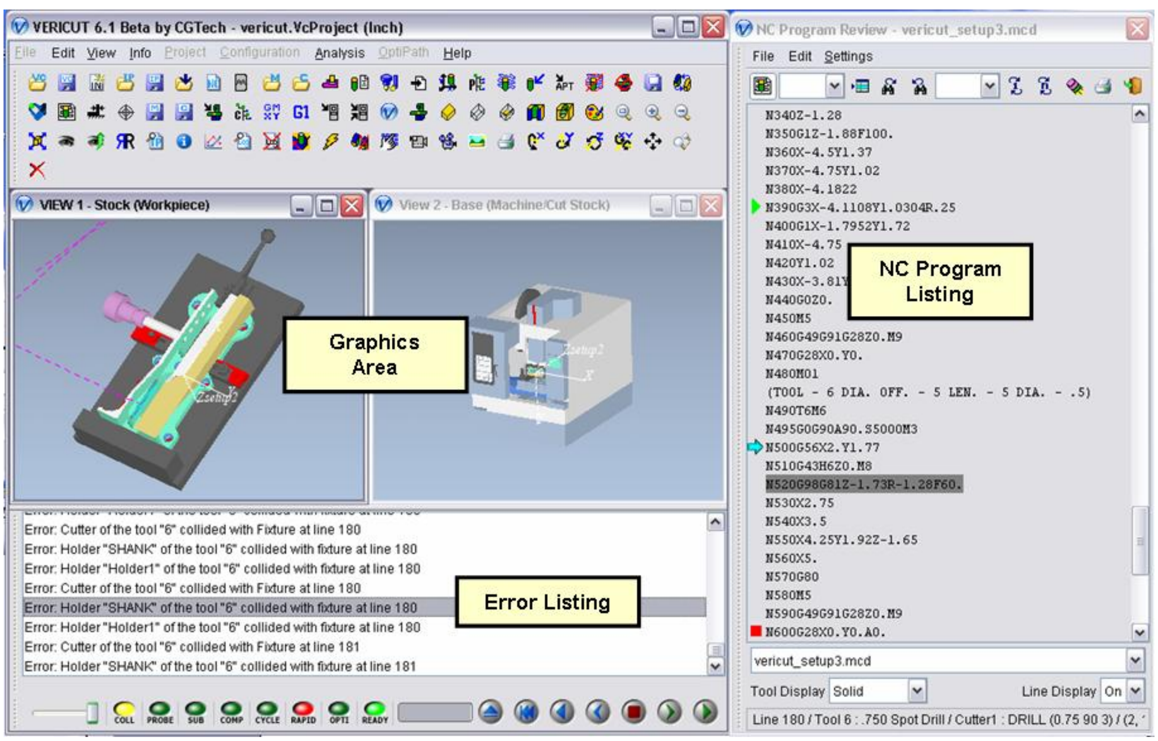
Location: **Analysis > NC Program Review**

VERICUT toolbar short cut: 

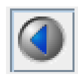
The **NC Program Review** feature enables you to quickly and easily view, replay and edit a previously processed NC program (toolpath). NC Program Review can be accessed at any point in the verification process, however, only information related to the "current" setup is available for review. It does not interrupt the current verification status, so reset or rewind of the NC program is not necessary.

NOTE: The following conditions must be met before a NC program can be viewed with NC Program Review.

1. The NC program must be processed far enough in VERICUT so that a tool has been loaded and at least one tool motion has been processed after the tool is loaded.
2. There must be a model attached to the first "Stock" type component in the Component Tree.
3. The first "Stock" type component must be visible in a workpiece view.



When you enter **NC Program Review** mode the VERICUT main window changes in the following ways:

1. Some of the menus in the menu bar become grayed out and are no longer available. These include the **File**, **Project**, **Configuration**, and **OptiPath** menus.
2. The icons in the **Toolbar** associated with the features in the menus listed above are also not available. If selected the message "Action not allowed in the NC Program Review window" will display at the bottom of the **Error Listing** area. These icons include the File Options, Simulation Settings, Project, Configuration, and Cut Options groups, AUTO-DIFF and Inspection in the Analysis Group, and Clear Log File and Logger in the Info group. For specific information on which icons are included in these groups, see the **View Toolbar window** section of *VERICUT Help*.
3. The graphics display will change and lines representing the actual tool paths will be displayed.
4. The NC Program Review, Error Listing area, will replace the VERICUT Logger.
5. Two additional icons,  (**Step Back**) and  (**Play Backward**) are added to the **Simulation (VCR) Controls**.
6. The **NC Program Review** window will display. If the **NC Program** window is already displayed, it will be replaced by the **NC Program Review** window.

Each of these changes is described in detail in the sections that follow.

The **NC Program Review** main window is composed of three distinct areas, each with different user interaction. The window header displays the current NC program file. This window can be moved and resized like most other windows, via dragging the window header, sides or corners. In addition, the size of the **NC Program Listing**, **Error Listing**, and **Graphics areas** can be changed within the main window by dragging the adjoining sides of each area.

NC Program Review enables you to quickly and easily associate the NC program record, the motion resulting from it, and any errors that may be associated with it. The three areas of the window are linked together so that if you select a NC program record in the listing, the tool moves to the corresponding location in the graphics area. If you select a location in the graphics area, the corresponding NC program record is highlighted in the NC program listing. If you select an error in the error listing, the NC program record that caused the error becomes highlighted in the NC program listing and the tool moves to the position where the error occurred in the graphics area.

NC Program Listing **—** This area displays a listing of the NC program records. In this area you can manually edit, add, or delete NC program records. Save the modified NC program file or open another NC program file associated with the current project file. You can also set **Start** and **End** markers to define the range of NC program records that you want to review. For more information about this portion of the NC Program Review main window, see the **NC Program Listing Area** section below.

Error Listing **—** This area displays a list of the errors that VERICUT found while processing the NC program. For more information about this portion of the NC Program Review main window, see the **Error Listing Area** section below.

Graphics Area **—** This area displays a visual representation of the motion associated with the records in the NC Program Listing. Here you can control how the tool is displayed, the method and direction of the replayed NC program as well as the speed of the replay. For more information about this portion of the NC Program Review main window, see the **Graphics Area** section below.

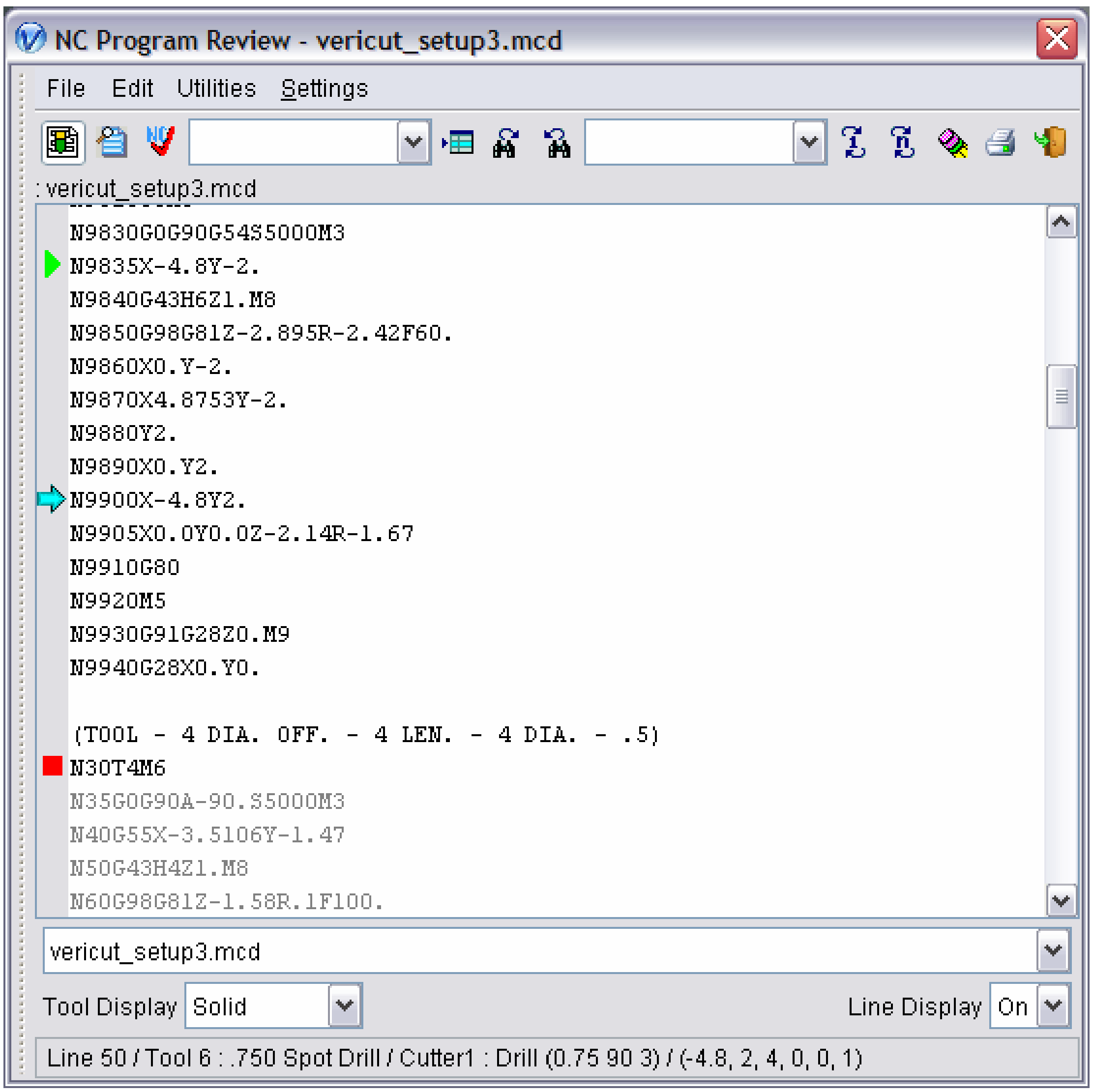
Also see "**NC Program Review**", in the *Using VERICUT* section, in the *CGTech Help Library*.

NC Program Review, NC Program Listing Area

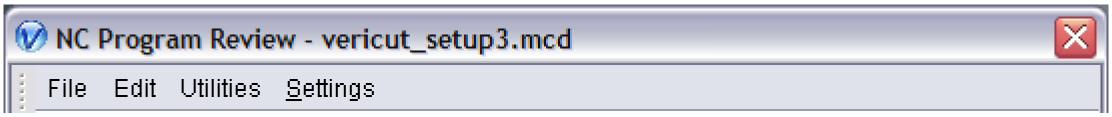
The **NC Program Review** window, or **NC Program Listing** area, contains a listing of the NC program records. Features in this window enable you to edit, save, or print the NC program, and define display characteristics, and ranges for replay of the NC program..

The **NC Program Review** window is one of the dockable windows enabling you to dock it inside the VERICUT main window if you choose. See **Dockable Windows** in the *Getting Started with VERICUT* section of *VERICUT Help* for additional information.

NOTE: When the **NC Program Review** window is docked, make sure that you click in the window so that it becomes the "active" window before using F1 to get help specific to the window. Otherwise F1 will go to the CGTech Help Library.



Main Menu



The menu located across the top of the **NC Program Review** window provides easy access to major **NC Program Review** functions. Each menu contains groups of related functions. Left-click on any of the menu names to display the list of functions available in that menu. Click on the function in the pull-down menu that you want to use. The name of the current NC program file is also displayed in this area.

File

Save As — Save the current NC program file under a different name.

Print — Displays a window enabling you to specify print characteristics and print the NC program.

Exit — Closes the **NC Program Review** window and returns to standard VERICUT mode.

Edit

Cut — Cuts the highlighted text in the NC program listing and puts it in the paste buffer.

Copy — Copies the highlighted text in the NC program listing to the paste buffer.

Paste — Puts the contents of the paste buffer at the location of the cursor in the NC program listing.

Restore — use to remove all edits from the NC program listing restoring it to its pre-edited state.

Shortcut: You can also access the **Edit** menu features by clicking the right mouse button in the **NC Program listing** area.

Utilities

Calculator — Opens the Calculator window enabling you to do mathematical calculations and conversions inside VERICUT.

NC Program Statistics — S

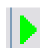
Colors — Opens the NC Program Color window enabling you specify colors for specific features (comments, variable, macros, etc.) in the NC Program listing.

Block Renumber — Opens the Block Renumber window enabling you to renumber one, or more blocks, in the NC program and specify format characteristics to be used for block numbering. The formats specified must conform to the characteristics defined in the current control file.

Remove Block Numbers — Removes all block numbers from the NC program file. Block numbers are identified by the characteristics defined in the current control file.

Check Syntax — Displays the Check Syntax window enabling you to check the displayed NC program for syntax errors.

Settings

Set Start — Sets the "**Start**" marker, , to the position specified by the cursor in the NC program listing or picking a location in the **Graphics Area**.

Set Current — Sets the "**Current**" marker, , to the position specified either by the cursor in the NC program listing or picking a location in the **Graphics Area**.

Set End — Sets the "**End**" marker, , to the position specified by the cursor in the NC program listing or picking a location in the **Graphics Area**.

Display to Box — Enables you to set "**Start**", "**Current**" and "**End**" markers by creating a box around the area of interest in the graphics area. Click on **Display to Box**, then left click in the **Graphics Area** and drag the cursor to create a box around the area of interest. The "**Start**" marker will be set to the lowest record number contained within the boxed area and the "**Current**" and "**End**" markers will be set to the highest record number contained within the "boxed" area.

Cut Line — Displays a color pallet enabling you to specify a color for "cutting" portions of the toolpath display in the **Graphics Area**.

Non-Cut Line — Displays a color pallet enabling you to specify a color for "non-cutting" portions of the toolpath display in the **Graphics Area**.

Current Line — Displays a color pallet enabling you to specify a color for "current" motion of the toolpath display in the **Graphics Area**.

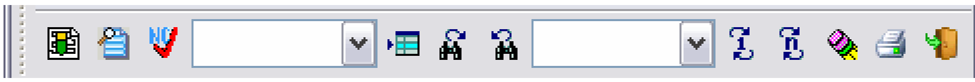
Goto Start — Moves the cursor in the NC program listing and positions the cutter in the **Graphics Area** to the position indicated by the "**Start**" marker.

Goto Current — Moves the cursor in the NC program listing and positions the cutter in the **Graphics Area** to the position indicated by the "**Current**" marker.

Goto End — Moves the cursor in the NC program listing and positions the cutter in the **Graphics Area** to the position indicated by the "**End**" marker.

Shortcut: You can also access the **Settings** menu features by clicking the right mouse button in either the **NC Program listing** area, or in the **Graphics Area**.

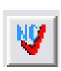
Icon Bar



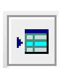
The items in the Icon Bar enable you to search for specific items in the NC program listing, print the NC program listing, or exit **NC Program Review**. Moving the cursor over the icon will display name of the option.

 **NC Program Review —** Use this icon to switch between standard VERICUT mode and NC Program Review mode (ref .**NC Program Review**, in the Analysis menu section , in *VERICUT Help* for more information).

 **NC Program Preview —** Use this icon to put VERICUT in NC Program Preview mode and display a tool trace represented by the NC program and the design model. This feature can be used without processing the NC Program first, as required by NC Program Review. See **NC Program Preview**, also in the Analysis menu Section of VERICUT *Help* for more information.

 **NC Program Syntax Check —** Displays the Check Syntax window enabling you to check the displayed NC program for syntax errors.

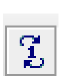
Line Number or Search Text — Use this text field to enter a line number or a string of text to search for.

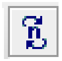
 **Goto Line Number —** Moves the cursor in the NC program listing to the line number specified in the **Line Number or Search Text** field.

 **Search Forward —** Searches forward in the NC program listing for the text string specified in the **Line Number or Search Text** field.

 **Search Backward —** Searches backward in the NC program listing for the text string specified in the **Line Number or Search Text** field.

Replacement Text — Use this text field to enter a "replacement text" string.

 **Replace One —** Replace one occurrence of text string in the **Line Number or Search Text** field (or the highlighted text in the NC program listing) with the text string in the **Replacement Text** field.

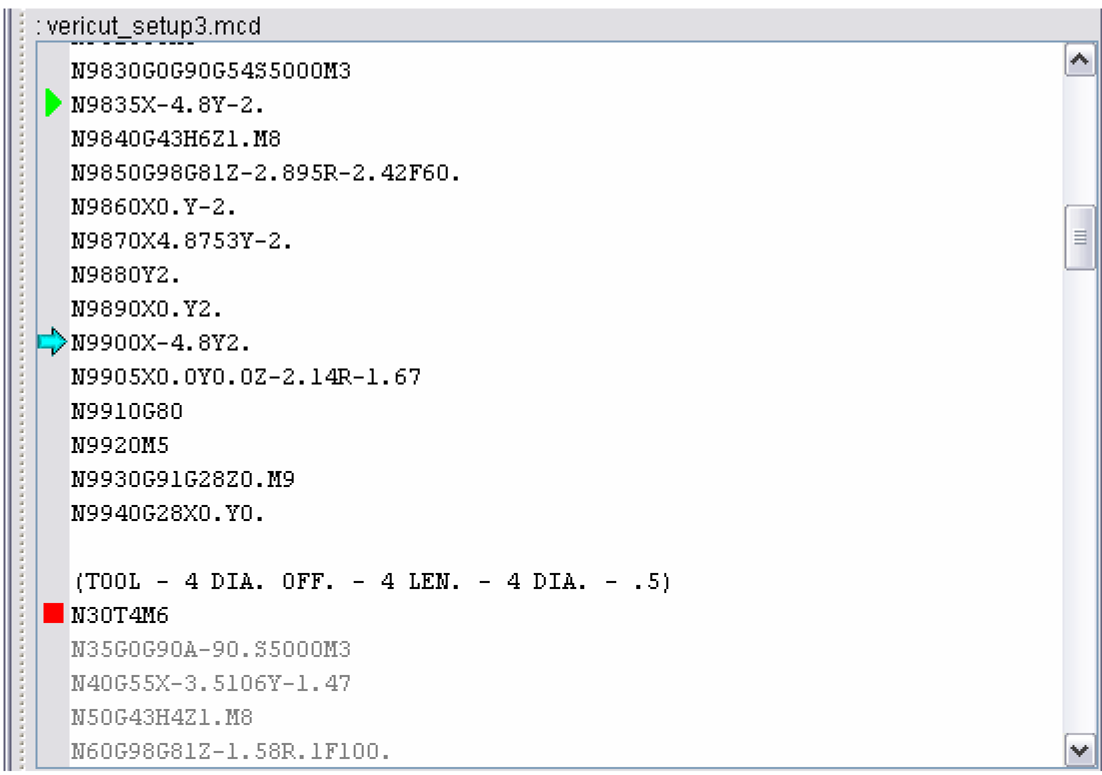
 **Replace All —** Replace all occurrences of the text string in the **Line Number or Search Text** field with the text string in the **Replacement Text** field.

 **Undo Highlighted Changes —** Use this feature to undo changes made to the NC program listing. Highlight the change and then click on the icon to "undo" the change.

 **Print —** Prints the NC program listing.

 **Close Window —** Closes the **NC Program Review** window and returns to standard VERICUT mode.

NC Program Listing Area



NC Program Editing

The NC program listing area allows you to edit the NC program with a full function text editor.

You can use the keyboard to add, change, or delete portions of the NC program listing. You can also use the **Cut**, **Copy** and **Paste** functions found under **Edit** in the main menu.

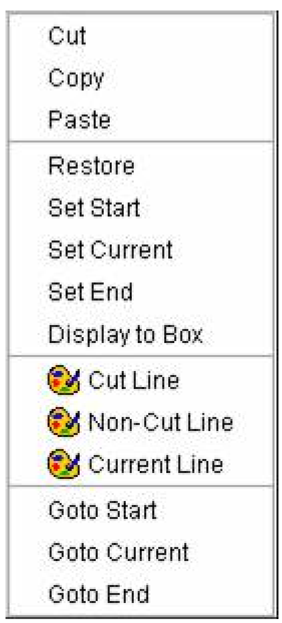
You can easily see the modifications that you have made to the NC program listing. Added, changed, and pasted text is displayed in blue and underlined. Cut and deleted text is displayed in red with a "strike through" font. Once you save the modified NC program, the red and deleted text is removed and the blue added, changed, and pasted text is returned to the original font.

Start/Current/End Markers

The **Start**/**Current**/**End** markers can be found along the left side of the **NC Program Listing** area. The **Start** marker is represented by the green triangle, the **Current** marker is represented by the aquamarine arrow, and the **End** marker is represented by the red square. You can easily identify each marker by moving the cursor over it.

The **Start** and **End** markers are used to define the range of NC program records that will be replayed when using **Play to End**. The **Current** marker moves to the NC program record that is currently being replayed. Set these markers by either selecting an NC program record in the NC program listing or the location in the graphics area where you want the marker placed. Then select the appropriate option (**Set Start**, **Set Current**, **Set End**) from the menu displayed when using the right mouse button in either the graphics area or the NC program listing. The menu can also be found under **Settings** in the **NC Program Review** window. You can also click on the marker in the NC program listing and drag it to the desired NC program record.

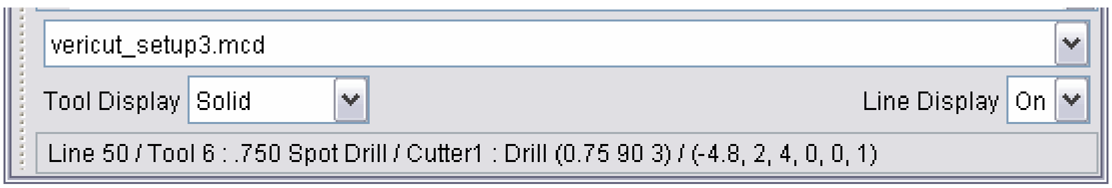
Shortcut: Click the right mouse button in the NC Program Listing Area to display a menu with the following features:



NOTES:

1. See the Edit section above for specific information on features **Cut** through **Restore**.
2. See the Settings section above for specific information on features **Set Start** through **Goto End**.

Other Features



The name of the displayed NC program file is displayed near the bottom of the **NC Program Listing** area. If the current setup contains more than one NC program file, clicking on the button to the right of the NC program file name displays a list of the NC program files contained in the current setup. Click on a file name in the list to change the NC program displayed in the **NC Program Listing** area.

Tool Display — enables you to specify how the tool is to be displayed in the graphics area while in NC Program Review. Choose **Off** (do not display tool), **Solid**, or **Translucent** from the pull-down list.

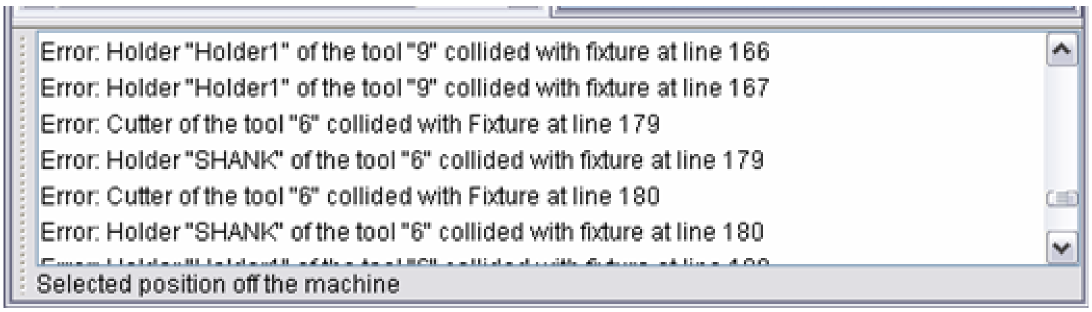
Line Display — enables you to specify whether or not to display the lines representing the tool's path in the graphics area while in NC Program Review. Choose **Off** (do not display), or **On** (display) from the pull-down list.

The line number of the "current" record and information related to the current tool (tool id or APT description) and the X, Y, Z, I, J, K of the current tool tip position is displayed at the bottom of the NC Program Listing area.

To NC Program Review

NC Program Review, Error Listing Area

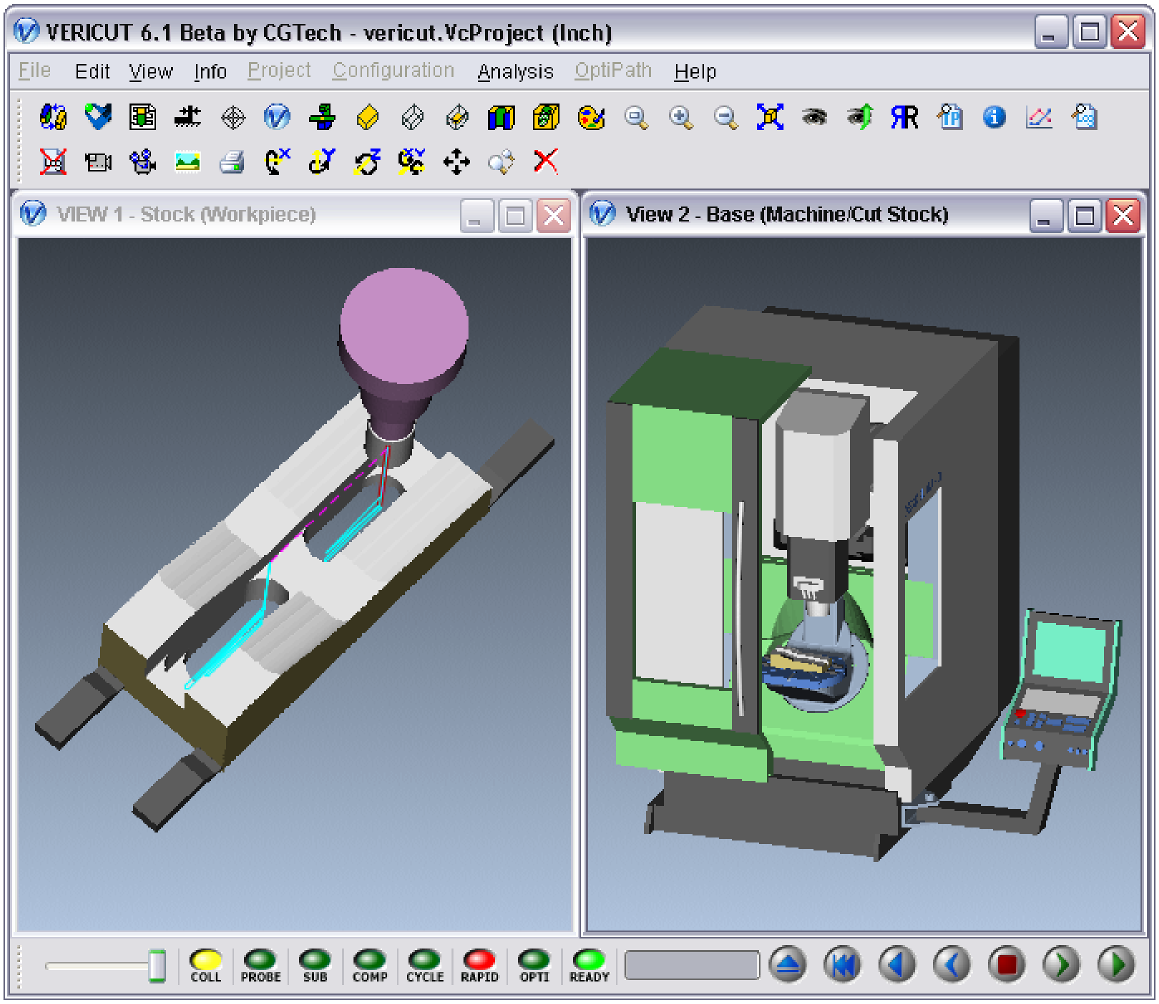
The **Error Listing** area replaces the VERICUT Logger while in NC Program Review. It contains a list of the errors that were found while processing the NC program in VERICUT.



Selecting an error in the **Error Listing** causes the line in the **NC Program Listing** associated with the error to become highlighted. At the same, the tool display in the **Graphics Area** moves to the position where the error occurred. This enables you to quickly and easily associate the error with the position in the NC program and in the toolpath display.

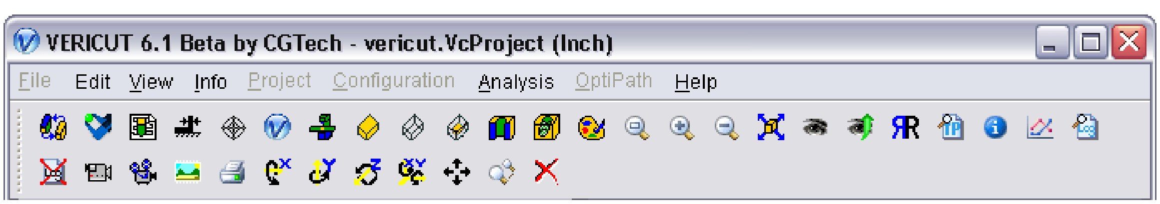
To NC Program Review

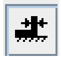
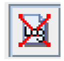
NC Program Review, Graphics Area



The graphics area displays the active views from the VERICUT main graphics area. Also, many of the analysis, display and view manipulation tools in the VERICUT main menu are available for use while you are in the **NC Program Review** graphics area.

Main menu and Toolbar icons



The icons shown in the picture above are all available in NC Program Review except  (AUTO-DIFF**)**,  (Inspection**)**, and  (Clear Log File and Logger).

The available Menu bar items and Toolbar icons perform the same function as those found in VERICUT.

Some of the menus in the Menu bar become grayed out and are no longer available while in NC Program Review. These include the **File**, **Project**, **Configuration**, and **OptiPath** menus.

The icons in the **Toolbar** associated with the features in the menus listed above are also not available. If selected the message "Action not allowed in the NC Program Review window" will display at the bottom of the **Error Listing** area. These icons include the File Options, Simulation Settings, Project, Configuration, Cut Options groups, AUTO-DIFF and Inspection in the Analysis Group, and Clear Log File and Logger in the Info group. For specific information on which icons are included in these groups, see the **View Toolbar window** section of *VERICUT Help*.

The icons shown in the picture above are all available in NC Program Review except **AUTO-DIFF**, **Inspection** and **Clear Log File and Logger**.

The Toolpath Display

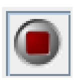
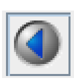
As the toolpath is replayed, a line representing the motion of the tool control point is displayed in the graphics area. Motions that remove material are displayed as solid lines with a default color of the foreground color. Motions that do not remove material are displayed as dashed lines with a default color of magenta. The line representing the motion of the current NC program record is highlighted with a default color of red. The tool image is displayed at the location of the current NC program record.

The color of the lines can be changed using the **Cut Line**, **Non-Cut Line** and **Current Line** options found in the menu that is displayed when using the right mouse button in the graphics area or the NC Program listing area, or found under **Settings** in the NC Program Review main window.

Graphics Area Controls

Animation Speed Slider — Controls the speed of the toolpath replay. Move to the left to slow down the replay of the toolpath. Move to the right to speed it up.

Simulation (VCR) Controls — The simulation controls, also known as VCR buttons, located at the bottom-right corner of the graphics area control interactive tool path replay. Use these controls to start and stop the replay of the toolpath. To see what action is associated with a Simulation control icon, simply position the cursor over the icon and a tip appears.

Icon	Name	Function
	**Play to End**	Replays from the **Current** position to the **End** position.
	**Step Forward**	Replays the next NC program record ("single block").
	**Stop**	**Stop** the replay during "Play" (after the current record displays)
	**Step Back**	Replays the previous NC program record ("single block"). See notes below.
	**Play Backward**	Replays from the **Current** position to the **Start** position. See notes below.
	**Rewind**	Sets the **Current** marker to the record with the **Start** marker and clears the toolpath display from the Graphics area.
	**Reset**	Sets the **Start** and **Current** markers to the first motion record in the NC program, sets the **End** marker to the last NC program motion record, and clears the toolpath display from the Graphics area.

NOTES:

1. You can see removed material replaced as you step back (**Step Back** or **Play Backward**) through the toolpath by turning on **Replace Material When Stepping Back** on the **File menu > Properties window: General tab** *before* simulating the toolpath in VERICUT.

   Replace Material When Stepping Back (ref. **Replace Material When Stepping Back**, on the Properties window: General tab, in the File menu section, also in the *VERICUT Help* section) and **FastMill** mode **(**ref. **FastMill**, on the Motion window, in the Project menu section, also in the *VERICUT Help* section) are mutually exclusive.

   When **Replace Material When Stepping Back** is toggled "On", VERICUT will turn **FastMill** "Off" Conversely, if FastMill is toggled "On", VERICUT will turn **Replace Material When Stepping Back** "Off. VERICUT will display a pop-up notification when these situations occur.

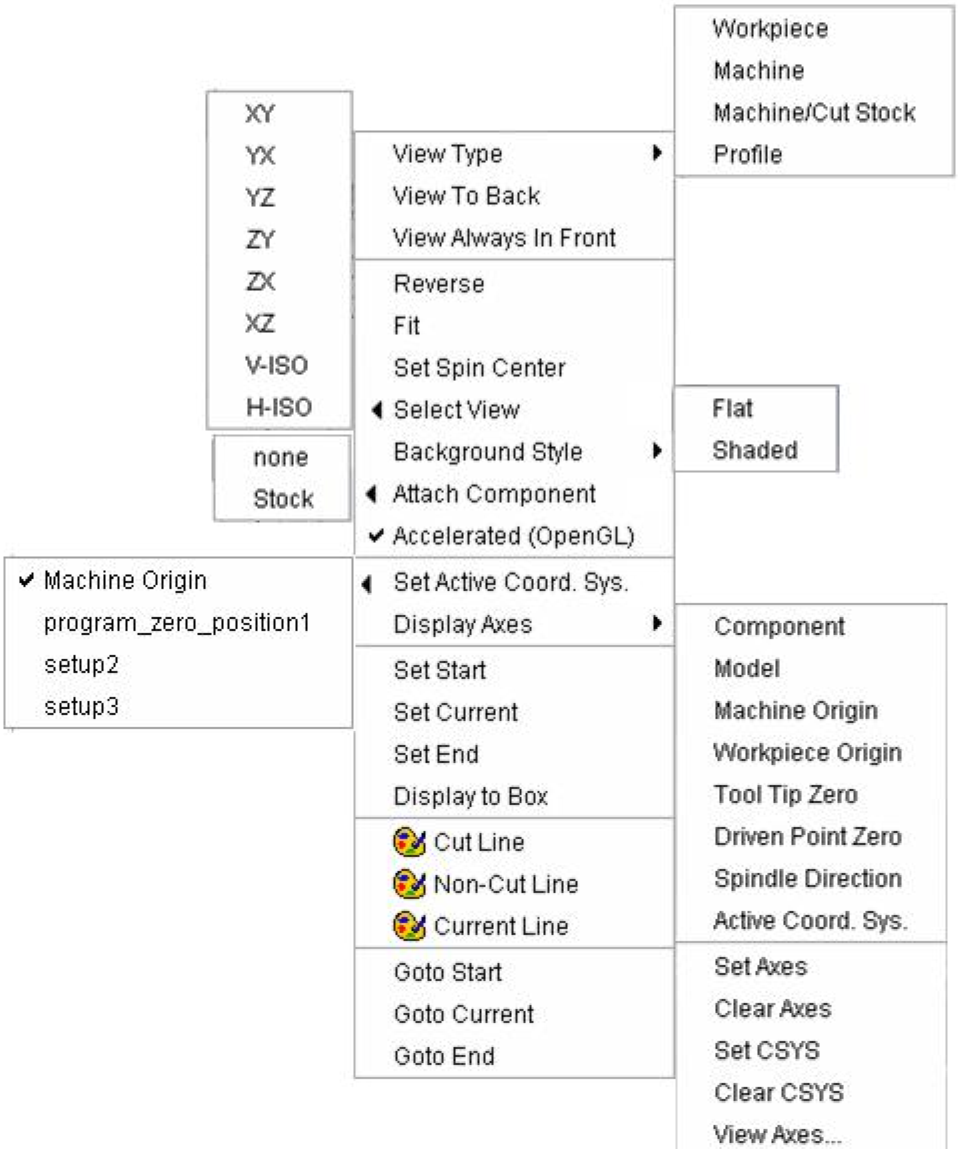
2. You can see machine animation as you step back (**Step Back** or **Play Backward**) through the toolpath by turning on **Animate Machine When Stepping Back** on the **File menu > Properties window: General tab** *before* simulating the toolpath in VERICUT.

To NC Program Review

Graphics Area Right Mouse Button Shortcut Menus

Workpiece View

Right-click in the **NC Program Review Main Window, Workpiece View** to display a menu with the following features:



NOTES:

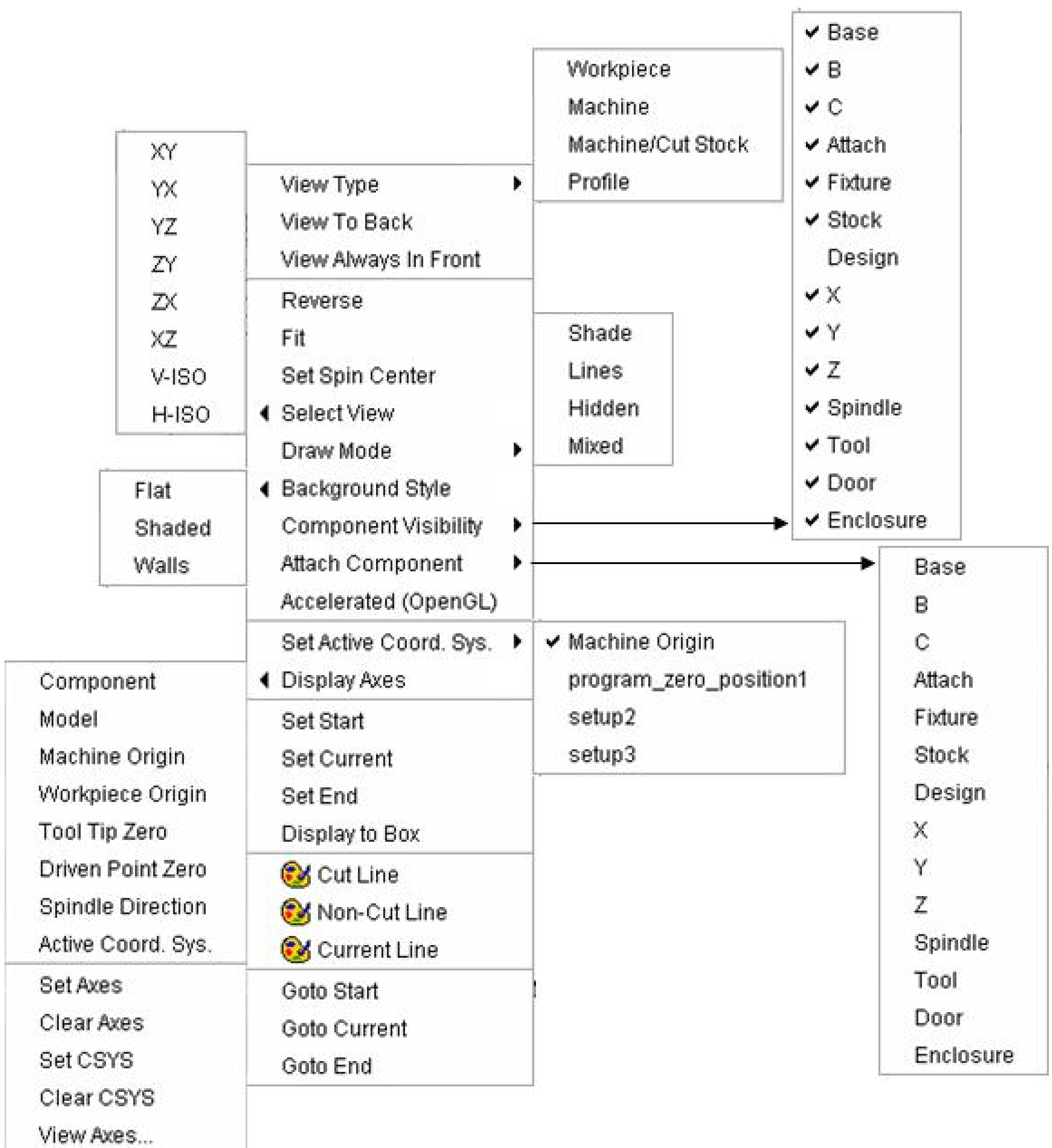
1. See **Workpiece View** in the Graphics Area Right Mouse Button Shortcut Menus, in the Getting Started with VERICUT section of *VERICUT Help*, for specific information on features **View Type** through **Display Axes**.

See Settings, in the NC Program Review, NC Program Listing area above, for specific information on features **Set Start** through **Goto End**.

2. A check next to **Accelerated (OpenGL)** indicates that the feature is toggled "On".

Machine View

Right-click in the **NC Program Review Main Window, Machine View** to display a menu with the following features:



NOTES:

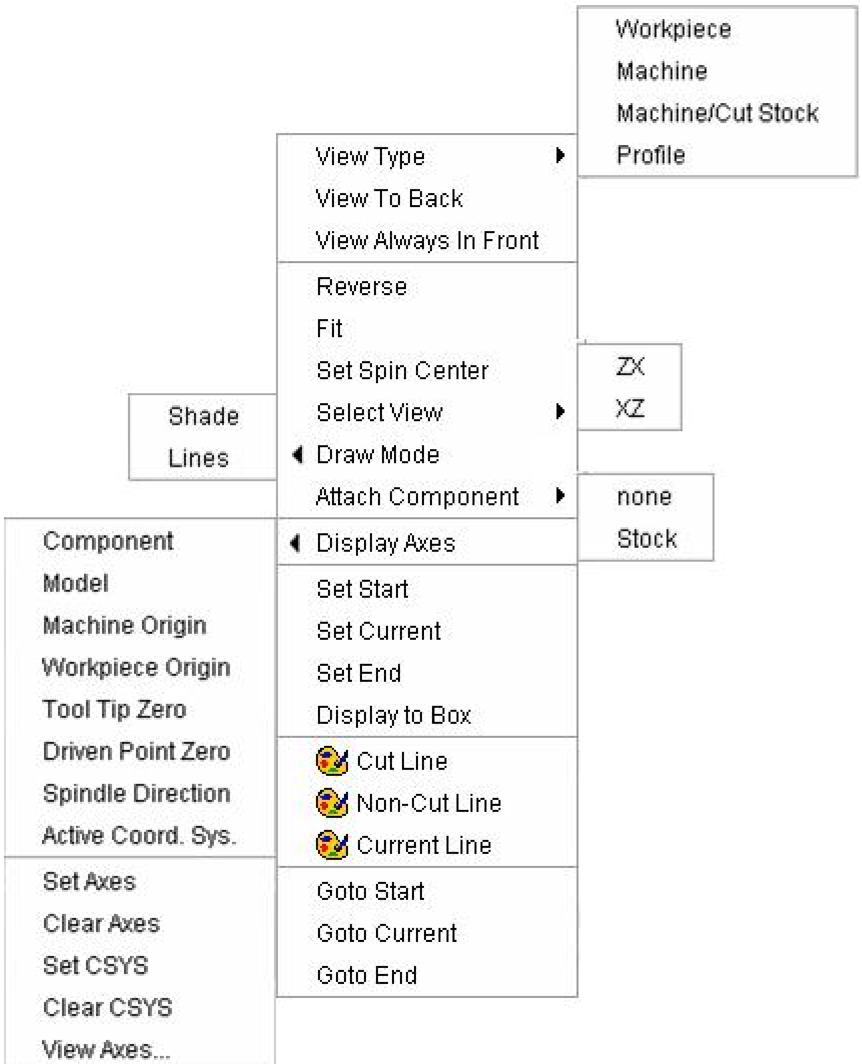
1. See **Machine View** in the Graphics Area Right Mouse Button Shortcut Menus, in the Getting Started with VERICUT section of *VERICUT Help*, for specific information on features **View Type** through **Display Axes**.

   See Settings, in the NC Program Review, NC Program Listing area above, for specific information on features **Set Start** through **Goto End**.

2. A check next to **Accelerated (OpenGL)**, or any of the **Display Axes** features, indicates that the feature is toggled "On".
3. A check next to any of the **Component Visibility** features, indicates that the feature is "visible".
4. The **Component Visibility**, and **Attach Component**, feature lists will contain all components in the current machine. This is especially useful when using an encrypted machine as the Component Tree is not available. If the list of components gets too long to display on your screen, VERICUT will automatically break up **Component Visibility**, and **Attach Component**, into multiple menu item/feature lists.
5. The **Select View** feature list will contain all available standard and custom views.

Profile View

Right-click in the **NC Program Review Main Window, Profile View** to display a menu with the following features:



NOTES:

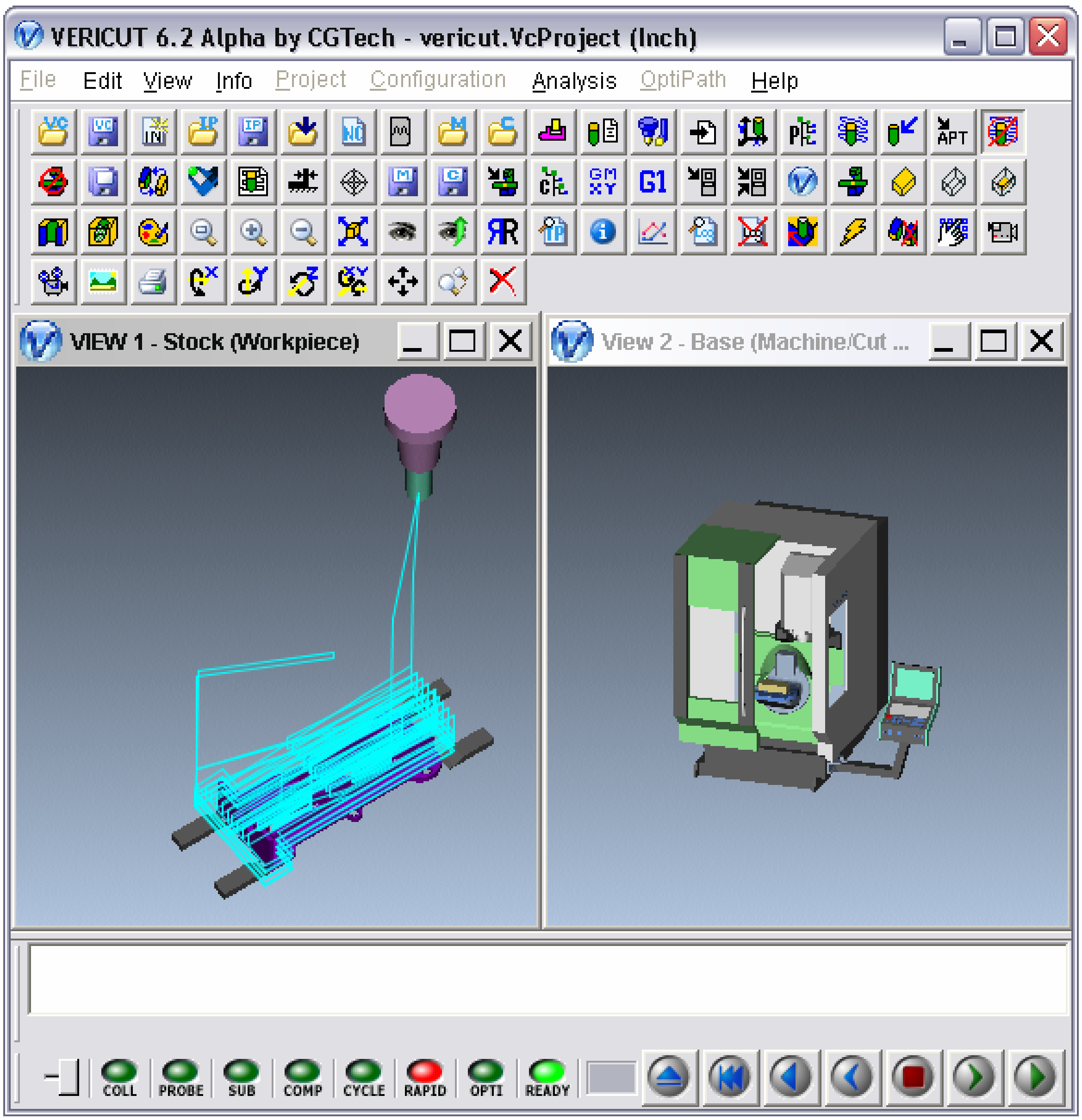
1. See **Profile View** in the Graphics Area Right Mouse Button Shortcut Menus, in the Getting Started with VERICUT section of *VERICUT Help*, for specific information on features **View Type** through **Display Axes**.

   See Settings, in the NC Program Review, NC Program Listing area above, for specific information on features **Set Start** through **Goto End**.

2. A check next to any of the **Display Axes** features, indicates that the feature is toggled "On".

NC Program Preview

The NC Program Preview is accessed from the NC Program (Info) window, or from the NC Program Review window: NC Program listing area, by using  (**NC Program Preview**) in the icon bar in either of the windows. NC Program Preview puts VERICUT in NC Program Review mode except that it displays a tool trace representing the NC program and the design model. This feature can be used without processing the NC program first, as required by NC Program Review. All of the VERICUT features behave in the same way as described above for NC Program Review.



NOTES:

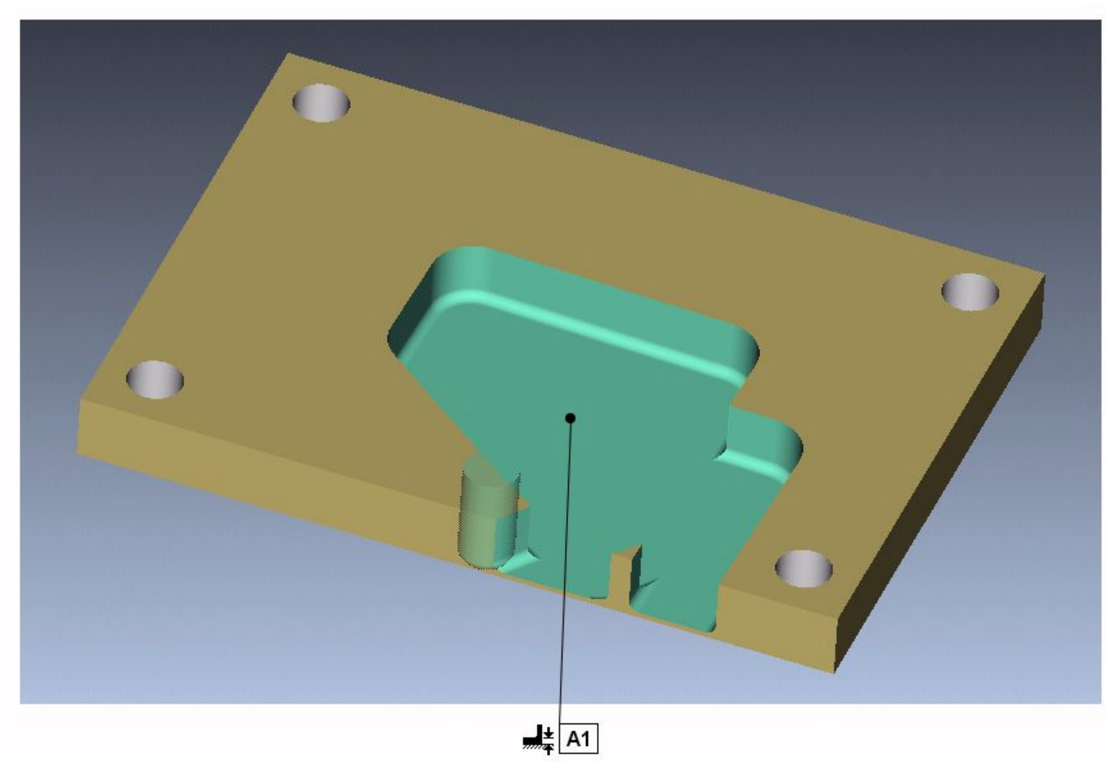
1. The tool trace representing the NC program and design model only display in a Workpiece view.
2. The NC Program Preview feature responds to the "Stop at ..." settings in the Motion window. See **Motion window**, in the Project menu section of *VERICUT Help*, for additional information on these settings.

Shortcut: NC Program Preview has the same right mouse button shortcut menus in the NC Program Listing Area, and in the Graphics Area, that are described above for NC Program Review.

Inspection

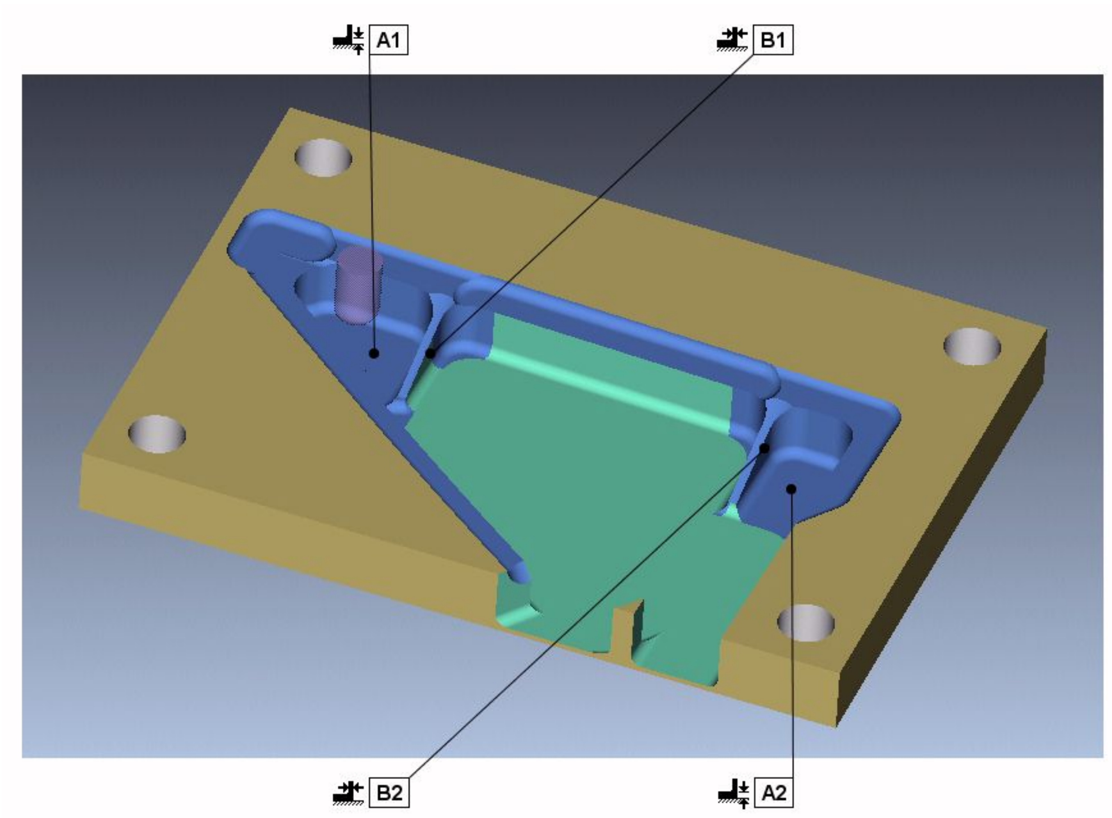
VERICUT Inspection

The "end product" of **VERICUT**'s Inspection module is a report, or set of reports, which define the inspection operations to be performed on a part. Typically these check measurements would be made while the part is still on the machine, perhaps after each tool's activity, to ensure that the job is proceeding as intended. If a discrepancy is found, no additional time will be wasted on subsequent machining. The format of the reports can be extensively customized, but there are two essential elements for each inspection report; a picture showing the features to be inspected, and a table giving the required dimensions and tolerances for each. The following four pairs of pictures and tables illustrate the inspection requirements after cutting a simple part with each of four tools. After the first tool, we simply check the remaining stock thickness at the "floor" of the part.



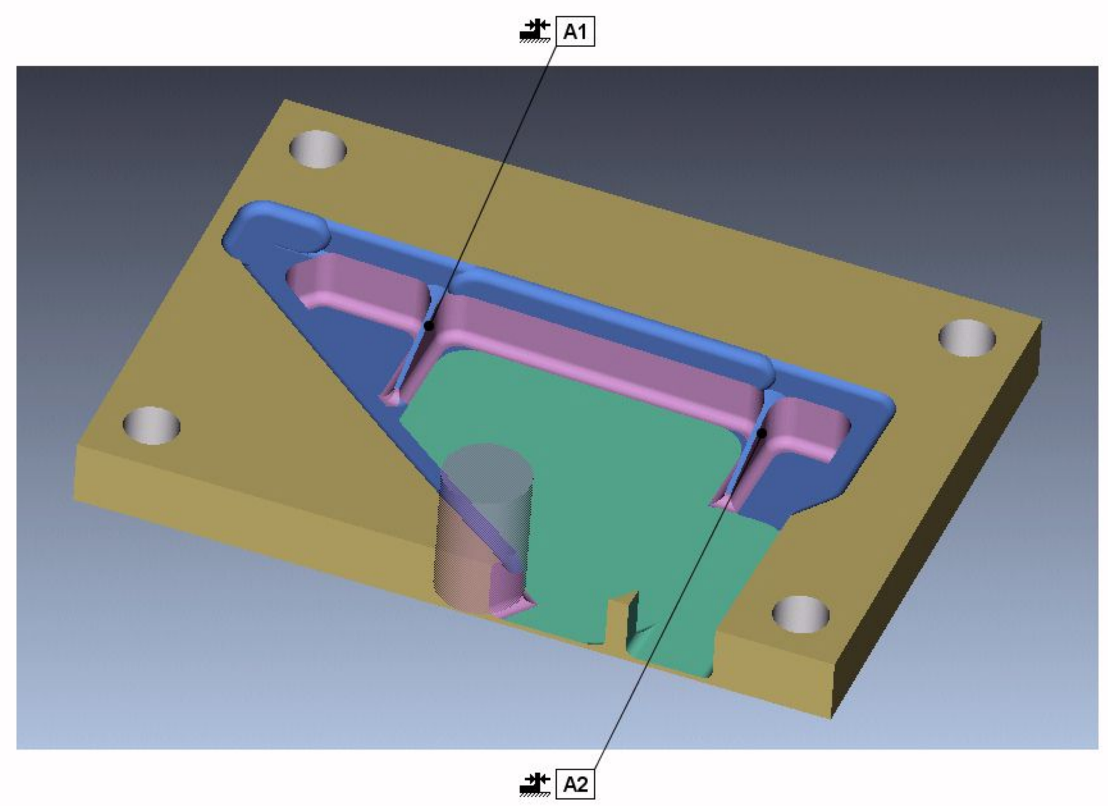
Symbol	Feature	Identifier	Instrument	Dimension	Tolerance	Geo.Tolerance	Measurement	Tool ID
	Floor Thickness	A1	Ultrasonic	2.000	±0.20			2

After the second tool, we check the floor thickness at each end of the part, and the "wall" thickness of the two webs.



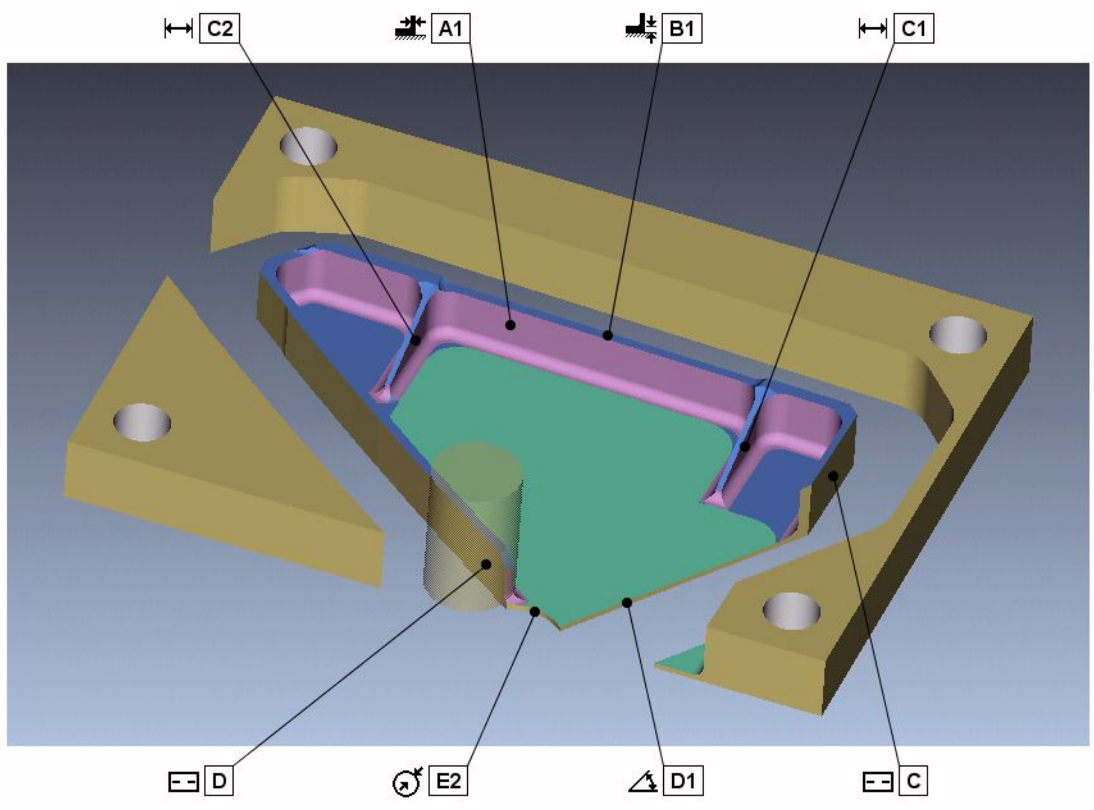
Symbol	Feature	Identifier	Instrument	Dimension	Tolerance	Geo.Tolerance	Measurement	Tool ID
	Floor Thickness	A1	Ultrasonic	2.020	±0.20			3
	Floor Thickness	A2	Ultrasonic	2.020	±0.20			3
	Wall Thickness	B1	Snap Caliper	3.000	±0.20			3
	Wall Thickness	B2	Snap Caliper	3.000	±0.20			3

After the third tool, we re-check the web thicknesses.



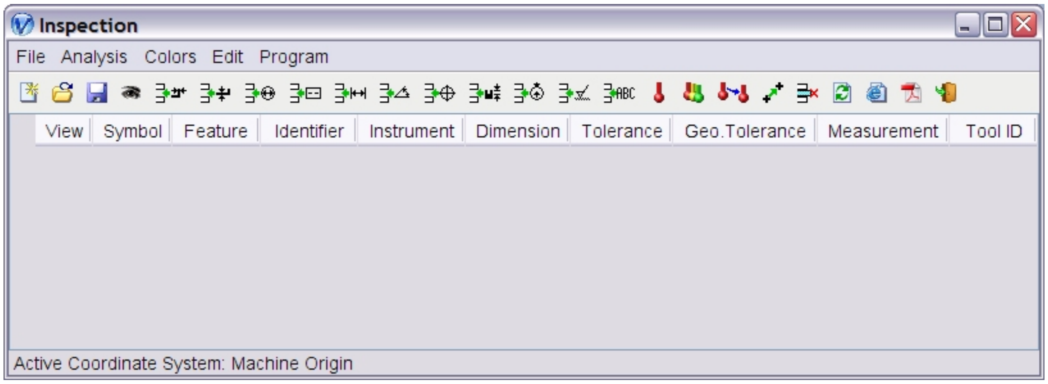
Symbol	Feature	Identifier	Instrument	Dimension	Tolerance	Geo.Tolerance	Measurement	Tool ID
	Wall Thickness	A1	Snap Caliper	2.000	±0.20			4
	Wall Thickness	A2	Snap Caliper	2.000	±0.20			4

And after the fourth and final tool, we specify many more measurements to be made. Note the use of datum planes for defining distance and angle dimensions, and how the feature identifiers tie the dimensions to the appropriate datum planes.

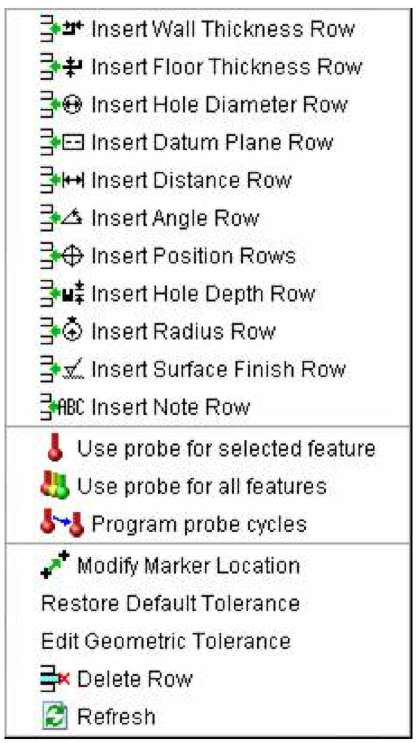


Symbol	Feature	Identifier	Instrument	Dimension	Tolerance	Geo.Tolerance	Measurement	Tool ID
	Wall Thickness	A1	Snap Caliper	3.700	±0.20			4
	Floor Thickness	B1	Ultrasonic	21.000	±0.20			3
	Datum Plane	C						
	Distance	C1	Caliper	33.500	±0.30			4
	Distance	C2	Caliper	157.500	±0.50			4
	Datum Plane	D						
	Angle	D1	Comparator	89.070	±0.04			5
	Radius	E2	Comparator	16.000	±0.20			5

The reports can be generated as HTML files that reference GIF and JPEG images, or as PDF files that have all the images embedded. But let's take a look at how you would start to use the inspection module. You will find it in **VERICUT**'s menus at "Analysis" > "Inspection Sequence ...". When you trigger it from a project file that has not previously exercised the module, a dialog similar to the following should appear.



You can re-size this window if you wish, re-arrange the columns in the table and set their widths. Your preferences will be remembered next time you display the dialog. If you let the cursor hover over any of the icons in the toolbar, a short tip will be displayed to explain its function. While all the functions are also accessible in the dialog's menus, it is likely that you will find the toolbar to be more convenient. In addition, each of these features is also available by clicking with the right mouse button in the inspection window to display the following right mouse button shortcut menu:



Following are the icons and a brief description of their intended use.

Clears the table, ready to start a new inspection sequence.

Prompts for and opens an existing inspection sequence file. Such files typically have the extension ".VcInspect". Note that you can also open the ten most recently accessed files more directly from the "File" menu.

Saves the current inspection sequence to a file. You may be prompted to select a file name if the current sequence was not derived from an existing file. There is also a "Save As" option in the "File" menu.

Opens another dialog to assist in identifying the features on the stock that were cut by each tool.

Inserts a row in the table for a wall thickness measurement. You will be prompted to pick the wall in a workpiece view.

Inserts a row in the table for a floor thickness measurement. You will be prompted to pick the floor in a workpiece view.

Inserts a row in the table for a hole diameter measurement. You will be prompted to pick the hole in a workpiece view.

Inserts a row in the table for a datum plane. You will be prompted to pick the plane in a workpiece view.

Inserts a row in the table for a distance from a datum plane. You will be prompted to pick the relevant feature in a workpiece view. It does not make sense to use this type of measurement without a datum plane.

Inserts a row in the table for an angle between a selected plane and a datum plane. You could use this type of feature without a datum plane if it is obvious which angle is to be measured.

Inserts two or three rows in the table for the coordinates of a position. You could use this type of feature to specify the location of the center of a hole, relative to the currently active coordinate system. You will be prompted to pick the hole in a workpiece view.

Inserts a row in the table for a hole depth measurement. You will be prompted to pick the hole in a workpiece view.

Inserts a row in the table for a radius measurement. You will be prompted to pick the hole or cylinder in a workpiece view.

Inserts a row in the table for a surface finish requirement. You will be prompted to pick the surface in a workpiece view.

Inserts a row in the table for a tabulated note. You will be prompted to pick a reference location in a workpiece view.

Changes the measuring instrument in the currently selected row to "Probe".

Changes the measuring instrument in all rows to "Probe". Typically this would be used just before the next icon.

Causes the Inspection Programming dialog to be displayed, and transfers any features that have "Probe" as their measuring instrument, to the extra dialog.

Allows you to re-position the selected row's symbol in a workpiece view. The row's dimension will be recalculated, and if the tolerance value was automatically generated, it will also be updated.

Deletes the selected row from the table.

Refreshes all the rows. This is equivalent to using the re-positioning icon, , for each row and picking all the same screen positions. It causes each dimension and non-edited tolerance to be recalculated, and is explained in more detail at the end of this document.

Opens the Save Inspection Sequence window enabling you to specify the /*path*/*filename* for the inspection report file to be created. Selecting **Save** generates and displays an HTML inspection report.

Opens the Save Inspection Sequence window enabling you to specify the /*path*/*filename* for the inspection report file to be created. Selecting **Save** generates and displays a PDF inspection report.

Exits the dialog. If you have made any changes to the table and not saved the file, you will be prompted to do so before the window closes.

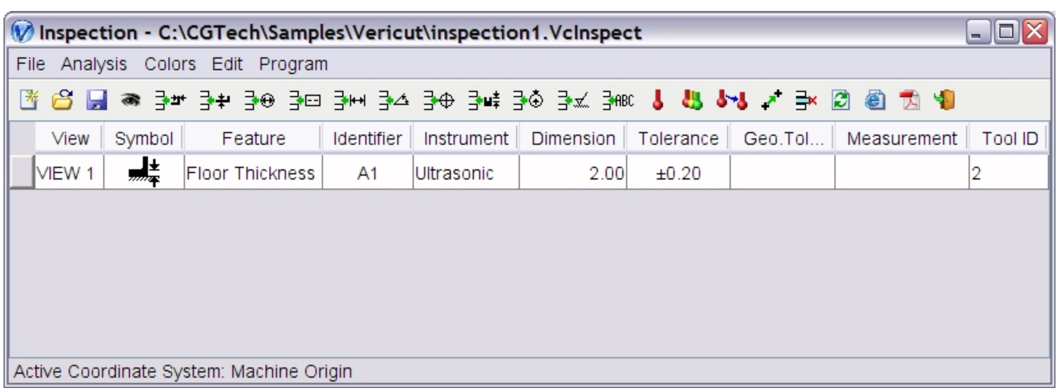
File Organization

An inspection sequence file retains all the information needed to reproduce the dialog's table and the corresponding symbols on the workpiece for a single report. Thus in the example at the start of this document there are four inspection files involved. A file is not the report itself. A report is generated on demand and will typically have the extension ".htm" or ".pdf". Unless you explicitly specify a file name when you first save an inspection sequence, the dialog will offer the project file name, with its extension changed to ".VcInspect". If there will be several sequences for a part, it is suggested that you adjust the file names to include suffices "1", "2", ... or "A", "B", ... or perhaps tool identifiers.

When you request that a report be generated, the report file name will be the same as the inspection sequence file name, with the extension changed to ".htm" or ".pdf". For a PDF file, any pictures required in the report will be embedded, which makes it trivial to pass the report around electronically. For an HTML file, the pictures are generated as separate GIF and JPEG files, making it more difficult to keep all the pieces together, unless you place all inspection files and reports for a part in their own folder, perhaps named using the part's identifier.

Inserting Rows

For measurements that do not require a datum plane, such as a wall or floor thickness, adding the required information to the table is very simple. Pick the relevant icon from the toolbar, for example for a floor thickness, then pick the location where the measurement is to be made on a workpiece view. A row will be appended to, or inserted in, the table and a feature symbol will be placed on the view. Next to the symbol will be an identifier that is repeated in the table. You can adjust the view orientation while it contains numerous feature symbols, but the orientation is going to be used for a report's picture eventually, so it makes sense to find one, sooner rather than later, that exposes all the features to be checked.



Let's look at each entry in the feature's row. Assuming you haven't already re-arranged the columns, the first one on the left contains the name of the workpiece view. An inspection report can have a page for each of several views if you need more than one to illustrate each required feature. The second column contains the same symbol as appears in a view. The third contains a very brief description of the type of feature, "Floor

Thickness" in this case. If you click on this entry, you will find that you can adjust the feature type, perhaps making it a "Wall Thickness" instead. When you do so, the symbol will be updated to reflect the new choice. You can also enter the name of a different feature type that we have not thought to offer. If you do so, there will be no corresponding symbol in the second column.

In the fourth column is an automatically generated feature identifier. If the first feature inserted in a table does not require a datum plane, its identifier will be "A1". The next will be "A2", then "A3", etcetera. If you were to edit the last one in this sequence, making it "B1", then subsequently appended rows would continue the new sequence, with "B2", "B3", and etcetera. If you then inserted a row after the last "A" feature, its identifier would continue the "A" sequence.

The fifth column indicates the type of instrument to be used to check the dimension. It will initially show a suitable instrument type, such as "Ultrasonic" for a floor thickness and "Snap Caliper" for a wall thickness. As with the feature types, you can edit the entry by picking from the drop-down list of instrument types, or by entering your own description. Also you can use the  or  buttons if probe cycles are to be used to measure the features on the machine.

The sixth column is used to specify the expected dimension. For many feature types, but not all, this column will be populated by the system. For our floor thickness example, behind the scenes we use X-Caliper's abilities to determine the distance through the stock from the position you picked to the next "air". The same calculation is performed for wall thickness. And if you pick a cylinder when locating a diameter or radius feature, we can obviously fill in the column easily. Things are a little more complicated when a datum plane is involved, which we will get to soon. Whether the column is populated automatically or not, you can always adjust the value.

If the dimension value is generated by the system when you insert the row, the seventh column is filled too. There are hard-coded default tolerances values, or you can supply your own. A sample file contains an explanation of the flexibility available and the format to be used. If the tolerances generated do not reflect the values you want, at least they give you initial entries to edit, with the advantage that you don't need to know how to keyboard enter a plus/minus symbol. If you need non-symmetric tolerances you could enter "+0.010/-0.008" for example, or you could employ DIN standards by typing an "H" number. Incidentally, if you do want to type a plus/minus symbol, on a Windows system try holding down the "Alt" key while you hit the numeric pad keys "0", "1", "7" and "7", then release the "Alt" key. Cut and paste is easier to remember.

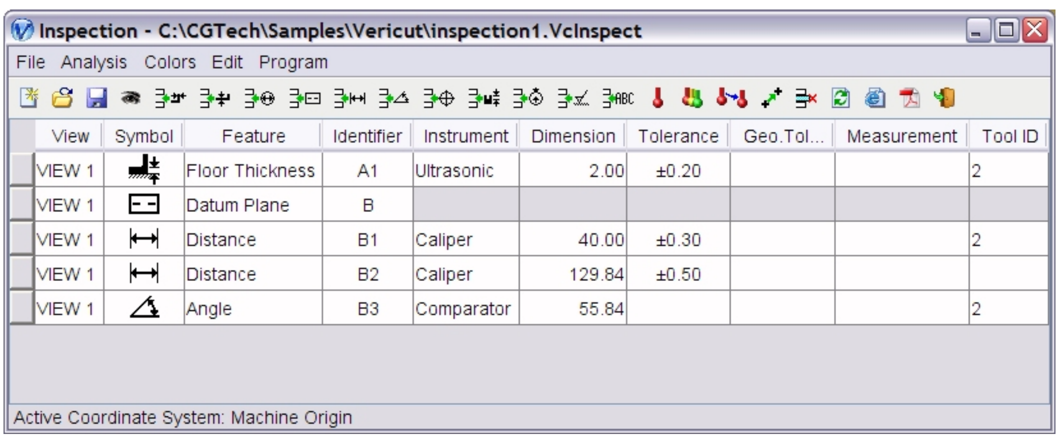
We'll cover the eighth, geometric tolerance, column towards the end of this document. The ninth column is where an inspector will enter the measurements he makes to generate a record of the part's compliance. Probably this will occur on paper copies of the reports generated by this module. But if your company is on the bleeding edge of digital record keeping and you make **VERICUT** available to your inspectors (which we whole-heartedly encourage) they could enter the values directly into the table.

The tenth and last column is filled automatically, if possible, with the identifier of the tool that cut the feature. You can edit this entry if you wish. Note that you can re-order the columns and your arrangement will be remembered.

Datum Planes

For measurements that require a datum plane, such as distances and angles, it makes sense to add a row for the plane first, followed by rows for one or more measurements to be taken from the plane. Pick the relevant icon, , and a position in a workpiece view. Note that in the fourth column, the datum plane's identifier is the next available single letter, and that columns 5 through 10 are shaded to show that they are unused. You can alter the identifier if you wish, but it is strongly suggested that you stay with the single letter convention. You can also edit the third column's entry if you wish to use "Right Face" or something more descriptive.

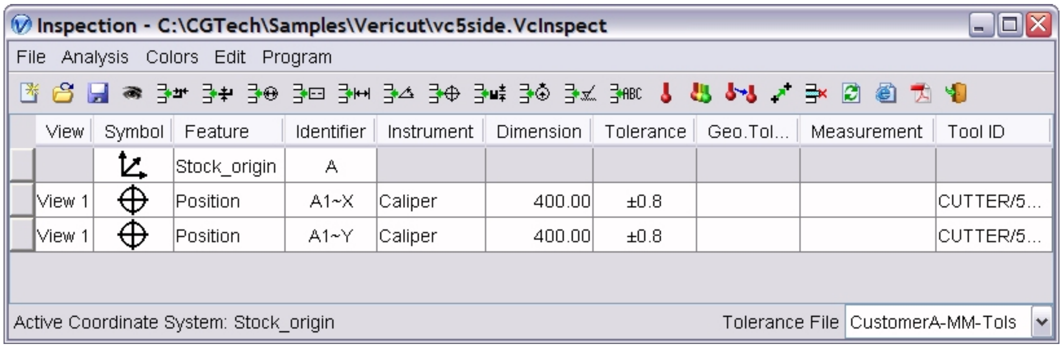
Now pick the icon for appending a distance row, , and pick another feature in a workpiece view. If you pick a plane which is parallel to the datum plane, then there is no doubt about the intention and we can automatically populate the sixth, dimension column. If you pick some other type of feature, such as the closest edge of a cylindrical boss, then the value we insert may not be exactly the one you need, and you would need to edit it. Repeat the steps in this paragraph to add other measurements from the datum plane. You will see that the identifiers all start with the plane's letter and are numbered consecutively.



For angle dimensions, , the value will be in degrees, is always acute, and is the angle between the surface normal at the picked position and the datum plane's normal. If the feature selected is not a plane, the angle may well be approximate and will need to be edited. To type a degree symbol on a Windows system, hold down the "Alt" key while you hit the numeric pad keys "0", "1", "7" and "6", then release the "Alt" key. No default tolerances are provided for angle measurements, unless you supply the "rules" in an external file.

Coordinate Systems

When you insert position rows, , their dimension values are in the active coordinate system. No other type of measurement is dependent on a coordinate system. When the first position is selected with a particular coordinate system active, the 2 or 3 rows for the position's coordinates are preceded by a row containing the name of the active coordinate system, and an identifying letter.



You can use more than one coordinate system, and each will be allocated a unique letter, which will be used in the coordinate system's row and all rows with positions to be measured in that system. If you adjust the name of a coordinate system elsewhere in **VERICUT** it would make sense to change it in the sequencing table too. If you adjust the location or orientation of a coordinate system, you can update the dimensions and tolerance on the associated position rows by refreshing, , the table. If you want to switch some position measurements from one coordinate system to another, you could can edit the system's name in the table and then use the refresh, , capability.

Coordinate system axes are reproduced in the inspection report pictures, provided they are visible in the workpiece views on the screen. The coordinate system rows in the report's tables are the only connection between system names and feature identifiers.

Modifying a Symbol Position

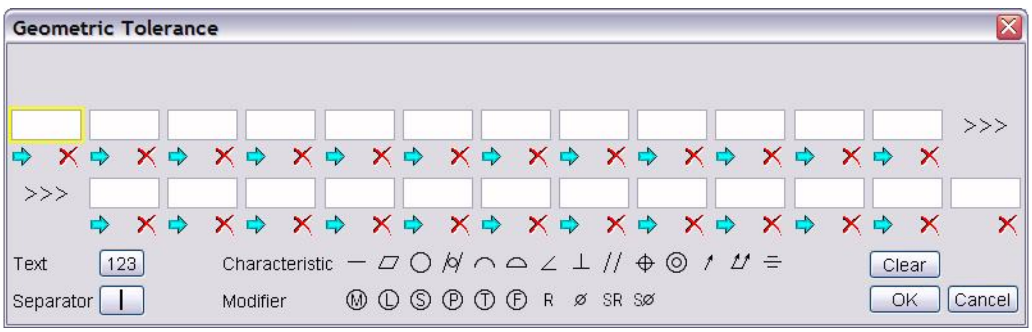
When you generate a report for an inspection sequence, with the or icon, the pictures are based on the workpiece views, but the feature identifiers are moved to the top and bottom edges. Primarily this is to reduce the chance of an inspector missing one in a cluttered picture. But the algorithm involved is not foolproof when it comes to ensuring that the lines from the features to their identifiers do not cross. When you see the pictures you may decide to re-orient their views and try again, or to move some of the symbol positions.

To move a row's symbol, select the row then pick the modification icon, . You will be prompted to select a new position in a workview. If the system is able to re-calculate the dimension at the new position, it will do so and update the value in the sixth column of the table. So if, for example, you move a wall thickness symbol from one wall to another, or if you move it along a wall of non-uniform thickness, you should expect the value to change. The tolerance value will be altered if it was generated by the system. A tolerance value which you entered, either by editing a supplied value or by typing into any empty cell, will not be adjusted. Such tolerance values are shown in italics.

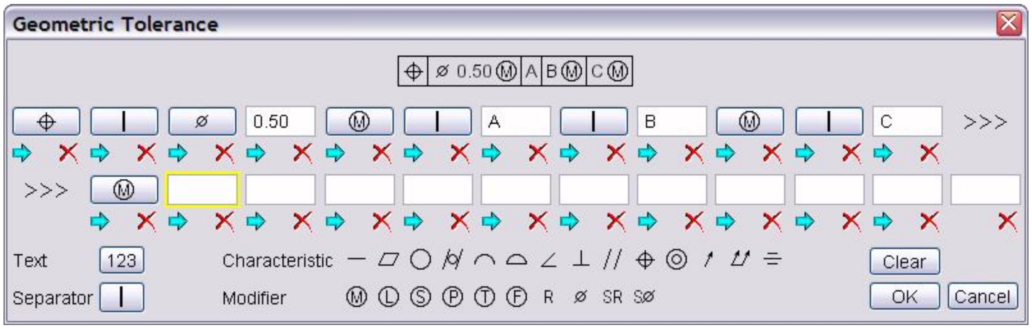
If you move a datum plane's symbol, dimensions that depend on that plane will **not** be updated. However if you move a distance or angle symbol, provided there is a datum plane with an identifier which is the first letter of the symbol's identifier, the system will try to update the individual dimension value.

Geometric Tolerance

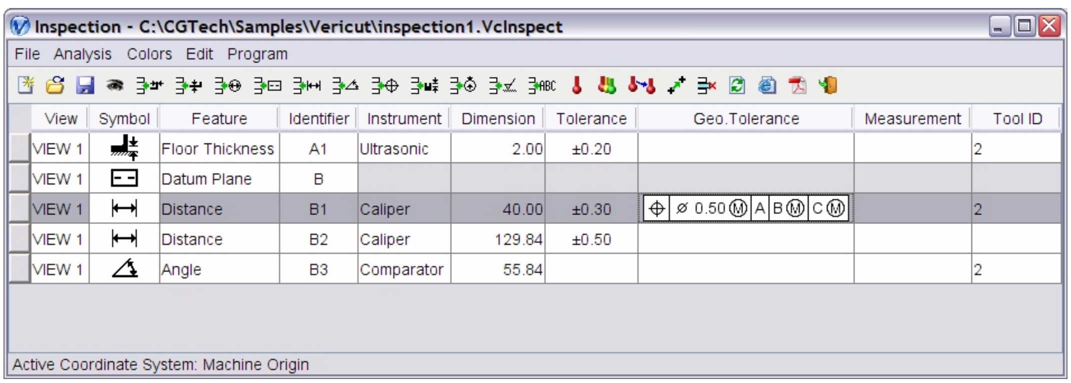
In the eighth column of an inspection sequence table you can place geometric tolerance specifications. If you do not use the widely misinterpreted GD&T standard, you can shrink this column's width to zero in the table, and omit it completely from your report template. But if your company uses geometric tolerancing, select a row that requires it and either select "Edit Geometric Tolerance" from the "Edit" menu, or right-click on the table and pick the same option from the pop-up menu. The following dialog will be presented.



In the twenty-four white boxes you can assemble the elements of a geometric tolerance specification. For a symbol, such as  for "Position", simply click on the required icon amongst the characteristics and modifiers. The symbol will be placed in the box that currently has the yellow border, which will automatically step to the next box. The same is true for a separator, or vertical bar. To enter text in a box, either a tolerance value or a datum plane reference, just type into an empty box, then hit the "Tab" or "Enter" key to step to the next one. The "Text" button can be used to convert an existing symbol or separator box to text. As you define the elements of the geometric tolerance specification, a picture is continually updated at the top of the dialog.



You can use one of the icons to shift elements to the right and make space for another one. The icons will delete an element and shift the remaining ones to the left. Once you are satisfied with the assembly of elements, click on the "OK" button. The dialog will disappear and the picture of the geometric tolerance box will be placed in the inspection table.



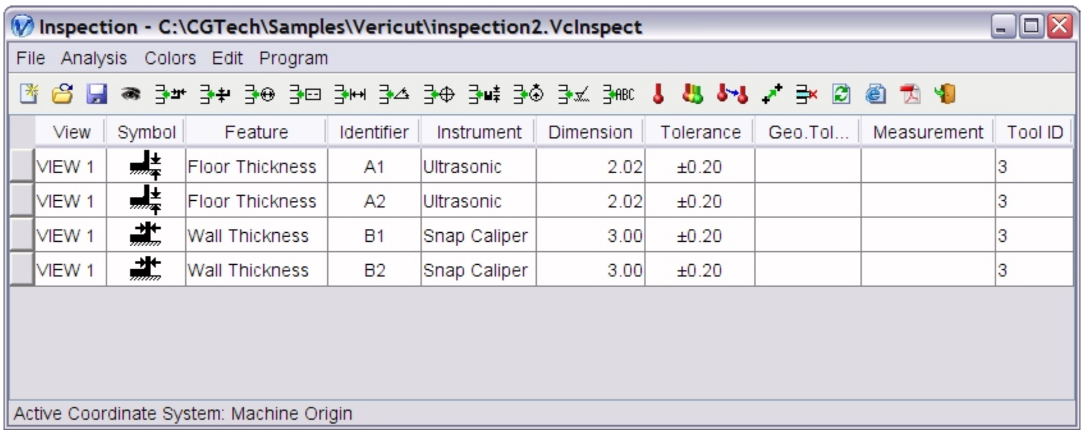
If the combination of dimension and geometric tolerance specification in this illustration doesn't make sense, we apologize. We prefaced this section by saying that the standard was widely misinterpreted, and count ourselves among the confused. The software does not attempt to enforce compliance.

Refreshing All Rows

When a design is changed slightly, it is possible that some of the features previously selected for inspection will be altered. The icon is intended to help in this situation by identifying which features have changed. It is equivalent to selecting the repositioning icon, , for each row of the table, and picking the same screen location for each feature. Let's take a wall thickness measurement as an example. The row's icon was on the side of the wall in the prior design. When we trigger the refresh capability, the software will determine the thickness of the current workpiece directly under the icon's screen position.

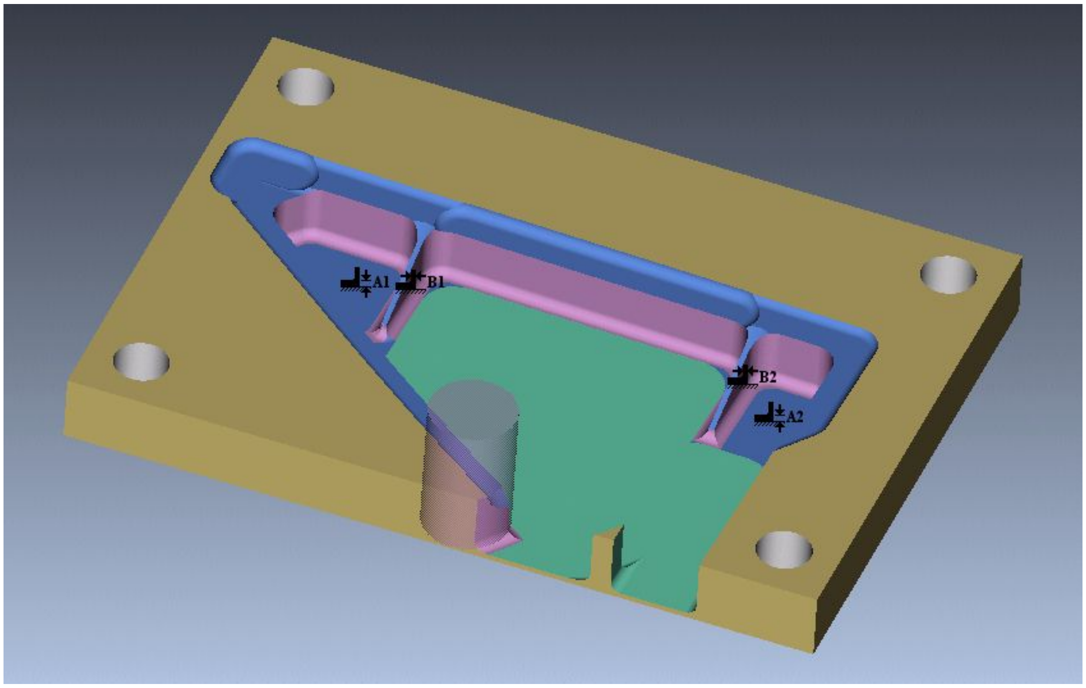
This thickness will be inserted in the "Dimension" column of the feature's row, and the tolerance value will be updated too, provided it was not previously edited by the user.

To highlight features that have changed significantly, the color of the dimension cells in the table may be changed to one of the following shades, . The lightest shade indicates a change greater than 10%, the next 20%, 30% and 40%, up to the darkest shade, which indicates a change of 50% or more. You can clear a colored cell simply by selecting its row. To illustrate the refresh function, we will again use the example at the start of this document. After cutting the part with the second tool, we inspected two wall and two floor thicknesses.

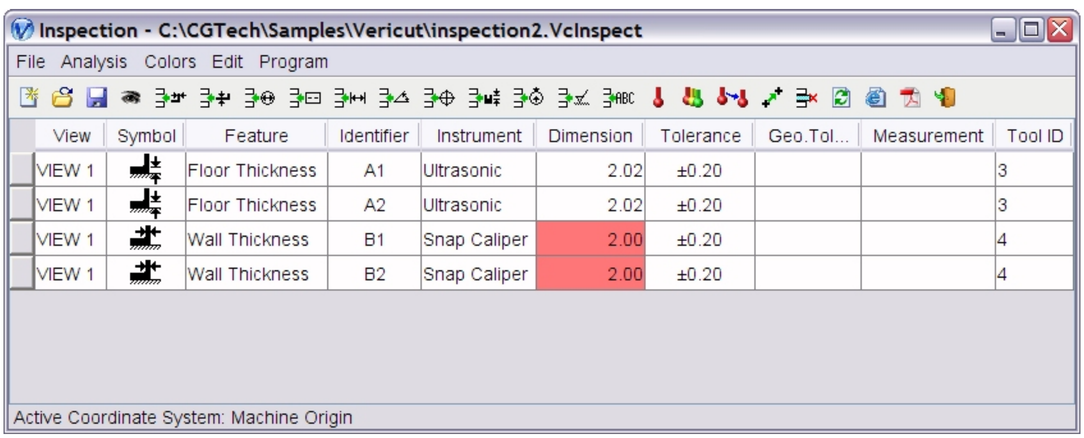


View	Symbol	Feature	Identifier	Instrument	Dimension	Tolerance	Geo.Tol...	Measurement	Tool ID
VIEW 1		Floor Thickness	A1	Ultrasonic	2.02	±0.20			3
VIEW 1		Floor Thickness	A2	Ultrasonic	2.02	±0.20			3
VIEW 1		Wall Thickness	B1	Snap Caliper	3.00	±0.20			3
VIEW 1		Wall Thickness	B2	Snap Caliper	3.00	±0.20			3

If we now proceed to cut with the third tool, the sides of the walls are finish machined.



We can now trigger the refresh function with the icon. Because the wall thickness icons B1 and B2 are still on the sides of the walls, their dimensions will be adjusted.

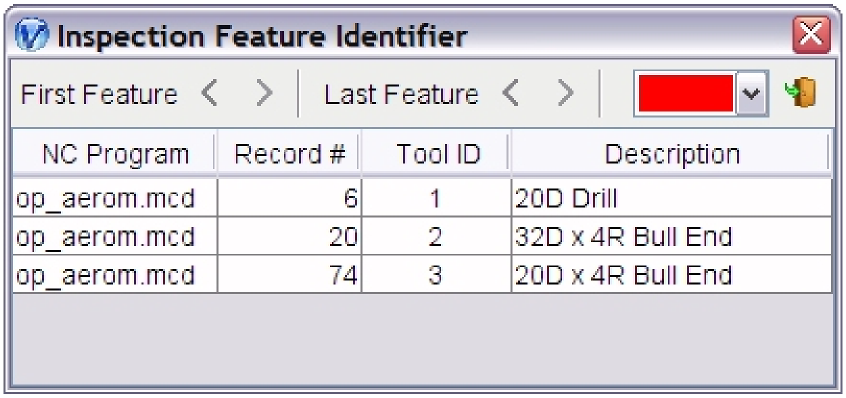


View	Symbol	Feature	Identifier	Instrument	Dimension	Tolerance	Geo.Tol...	Measurement	Tool ID
VIEW 1		Floor Thickness	A1	Ultrasonic	2.02	±0.20			3
VIEW 1		Floor Thickness	A2	Ultrasonic	2.02	±0.20			3
VIEW 1		Wall Thickness	B1	Snap Caliper	2.00	±0.20			4
VIEW 1		Wall Thickness	B2	Snap Caliper	2.00	±0.20			4

Note that the color of the changed dimension cells has been set to the third shade, because the values have been reduced by 33%.

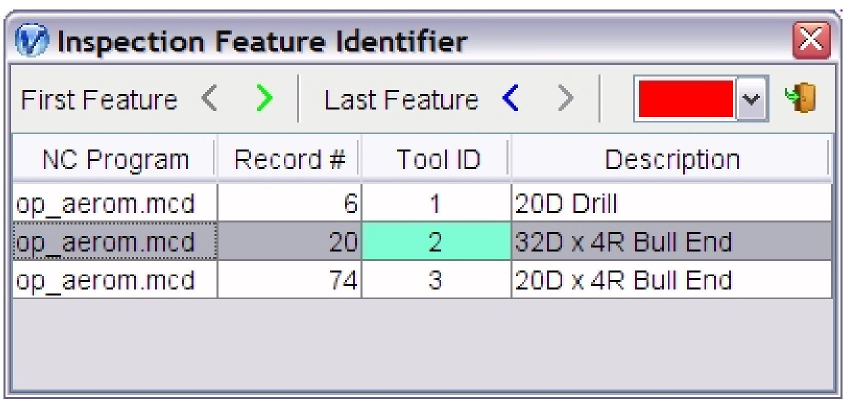
Analysis, Feature Identifier

It is common practice to inspect the first and last feature cut by each tool. Measurement of the first feature will confirm that the correct tool is loaded, and measurement of the last will check for excessive tool wear. However, on a complex part it may not be obvious where these two features are located. The dialog accessed from this menu item is designed to help. It displays a table with a row for each tool that currently has an "imprint" on the cut stock.



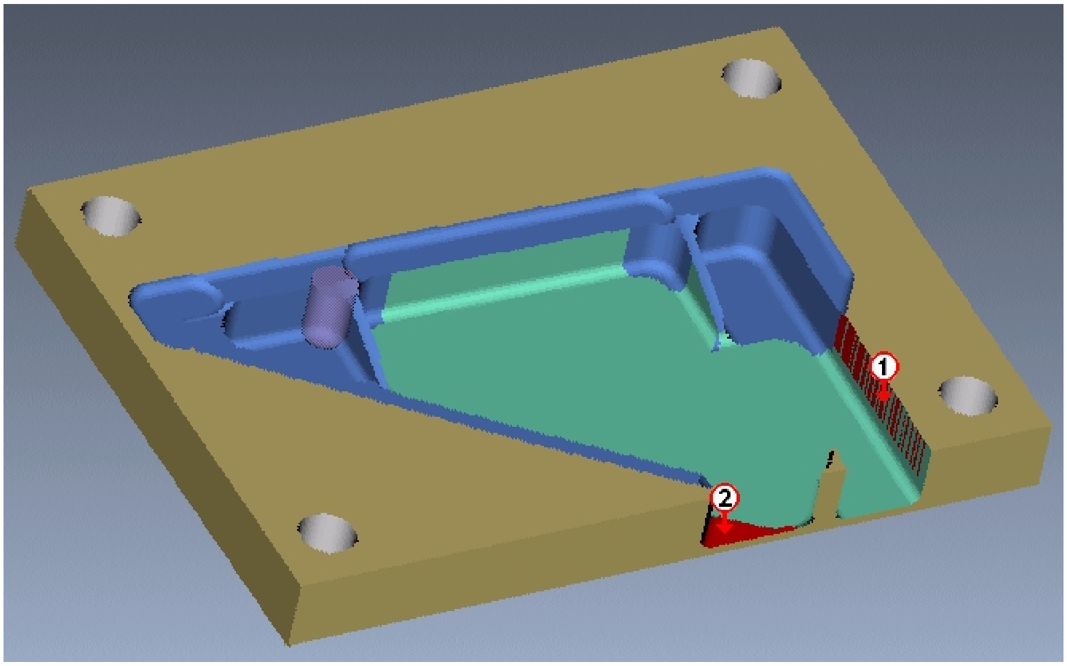
NC Program	Record #	Tool ID	Description
op_aerom.mcd	6	1	20D Drill
op_aerom.mcd	20	2	32D x 4R Bull End
op_aerom.mcd	74	3	20D x 4R Bull End

When you select a row, the background of the cell containing the tool's identifier is changed to match its "cut color",



NC Program	Record #	Tool ID	Description
op_aerom.mcd	6	1	20D Drill
op_aerom.mcd	20	2	32D x 4R Bull End
op_aerom.mcd	74	3	20D x 4R Bull End

and two markers will appear in the workpiece view to indicate the first (1) and last (2) features cut by the tool. The colors of these two features will also be adjusted to match the color in the dialogs drop-down palette.

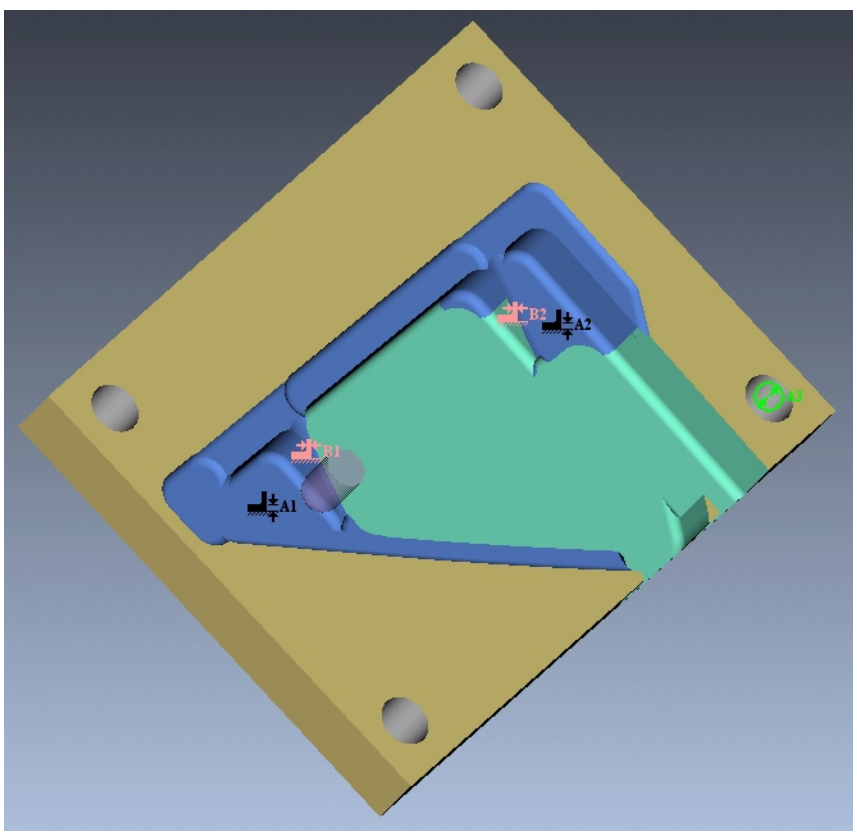


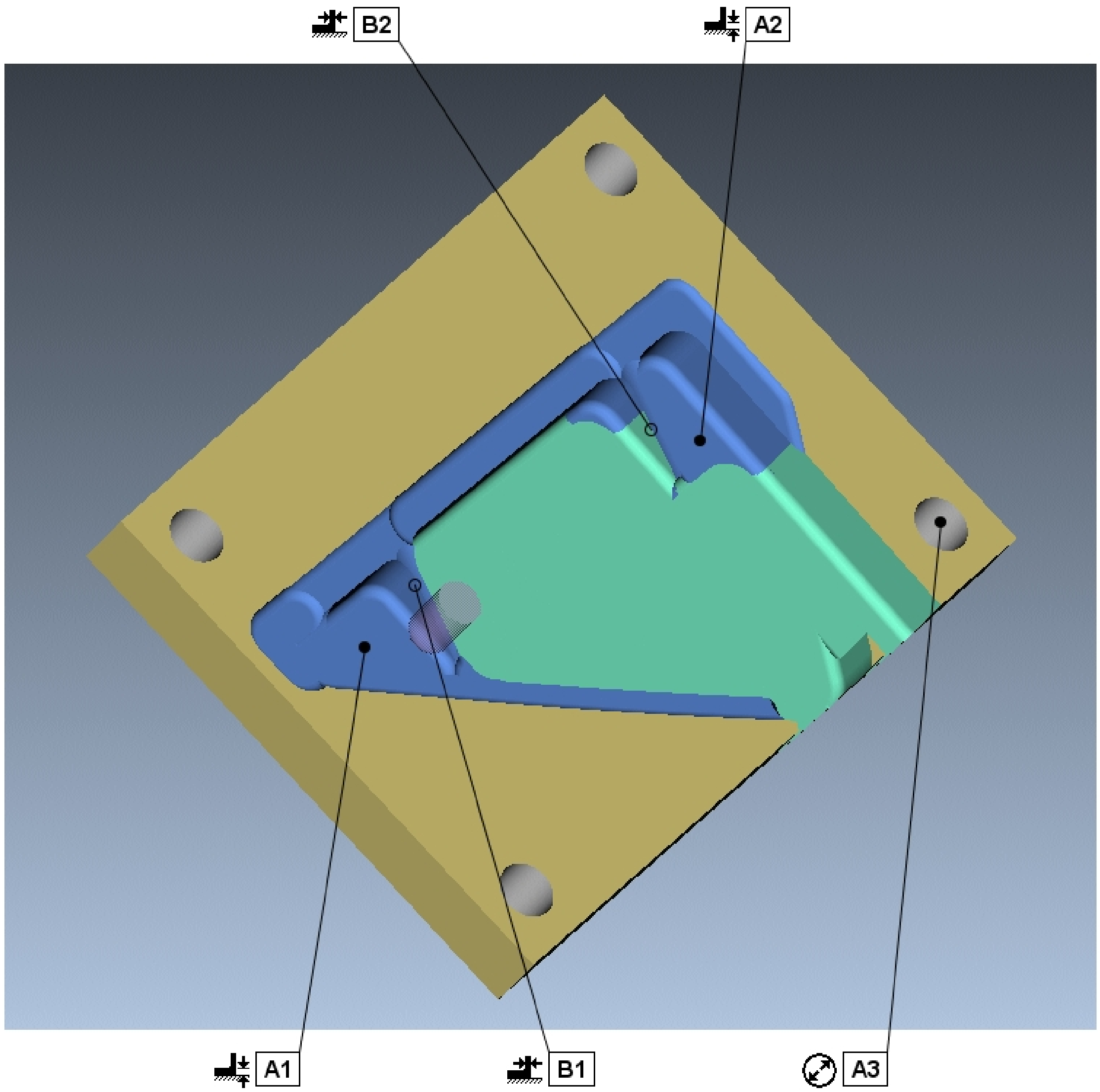
You can use the  and  buttons to walk through the features cut by the tool until you find ones that are suitable for inspection. It is not necessary to close the feature identifying dialog while you add rows and features to the inspection sequence table.

Colors

By default, feature icons are black, both in the workpiece views and the reports. And the icon corresponding to the selected row of the table will be orange in its workpiece view, to help you locate it quickly. These on-screen colors can be adjusted, without affecting the appearance of the reports. Under the "Colors" menu there are three choices;

-  In front
  The color you choose here will be used for any feature icon on a face of the cut stock whose outward normal points out of the screen.
-  Behind
  The color you choose here will be used for any feature icon on a face of the cut stock whose outward normal points into the screen. Having such icons in the view when you generate a report is not good practice, as it may not be clear to the reader where the feature's measurement is to be taken. If you do produce a report with features on the "back side" of the stock, they are marked with an open circle instead of the usual filled circle. The following two pictures illustrate this with a workpiece view and report. Icons "in front" are black, those "behind" are pink, and the selected one is green.





-  Selected

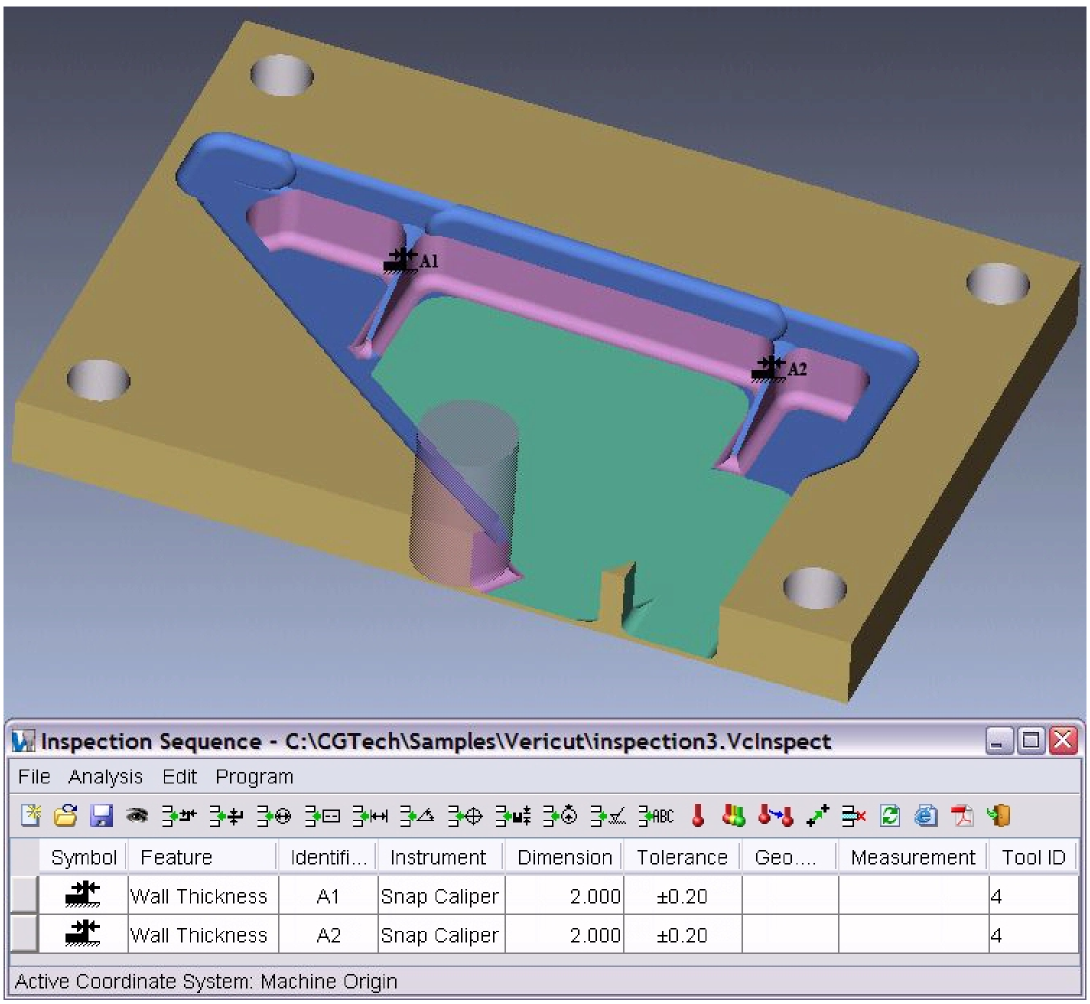
  The color you choose here will be used for the single feature icon corresponding to the selected row in the sequencing table.

Also see "**VERICUT Inspection**", in the *Using VERICUT* section, in the *CGTech Help Library*.

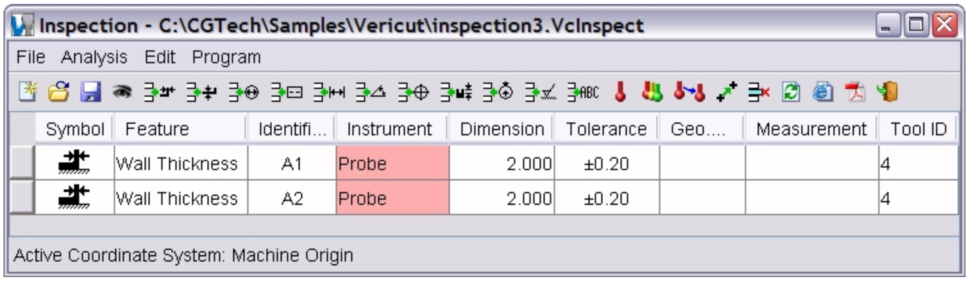
VERICUT Inspection Programming

The "end product" of **VERICUT**'s Inspection module is a report, or set of reports, which define the inspection operations to be performed on a part. Typically these check measurements would be made while the part is still on the machine, perhaps after each tool's activity, to ensure that the job is proceeding as intended. If a discrepancy is found, no additional time will be wasted on subsequent machining. This goal can be reached quickly if the inspection is performed on the machine and by the machine, using one or more probes.

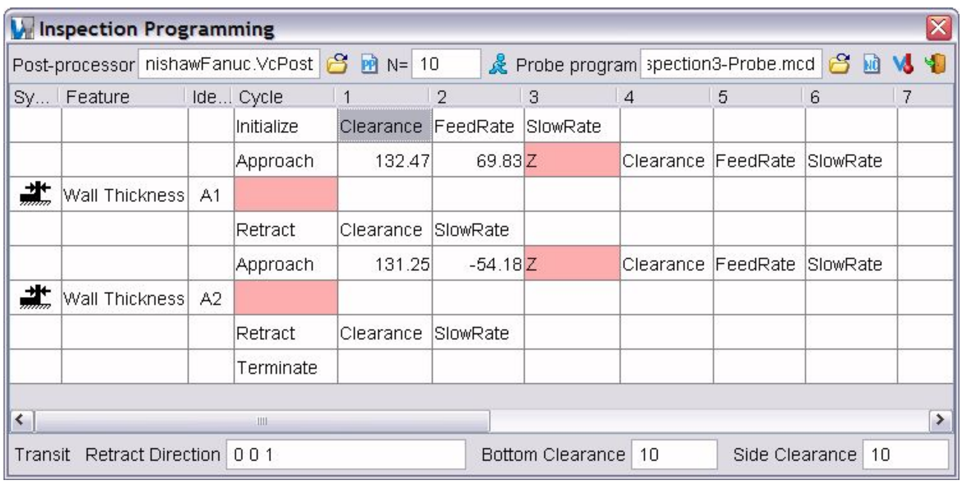
To explain the Inspection Programming module we will start with the third set of checks from the Inspection Sequencing document. This consisted of two wall thickness measurements after the third tool had cut the part. Note that the active coordinate system is "Machine Origin".



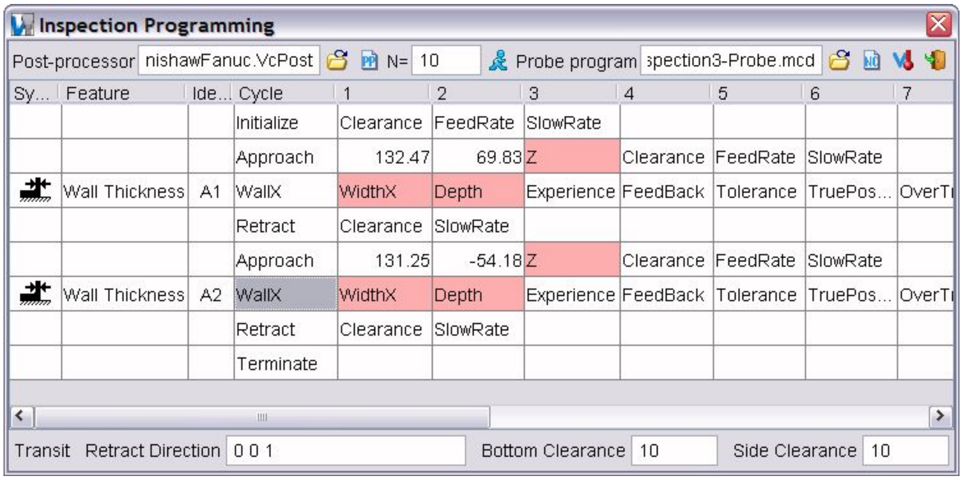
We use the button to override the measuring instrument for both features, so that they will be probed.



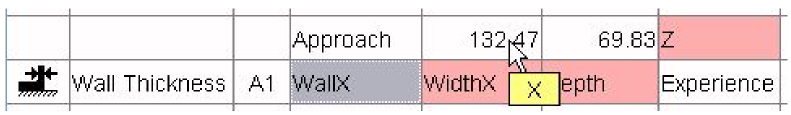
Next we use the button to request the Inspection Programming dialog. Before we see it, we will be prompted to select a post-processor file, which typically has the extension ".VcPost". For this explanation we shall use a post-processor developed for Renishaw probe cycles on a Fanuc controller. The name of the file is "RenishawFanuc.VcPost".



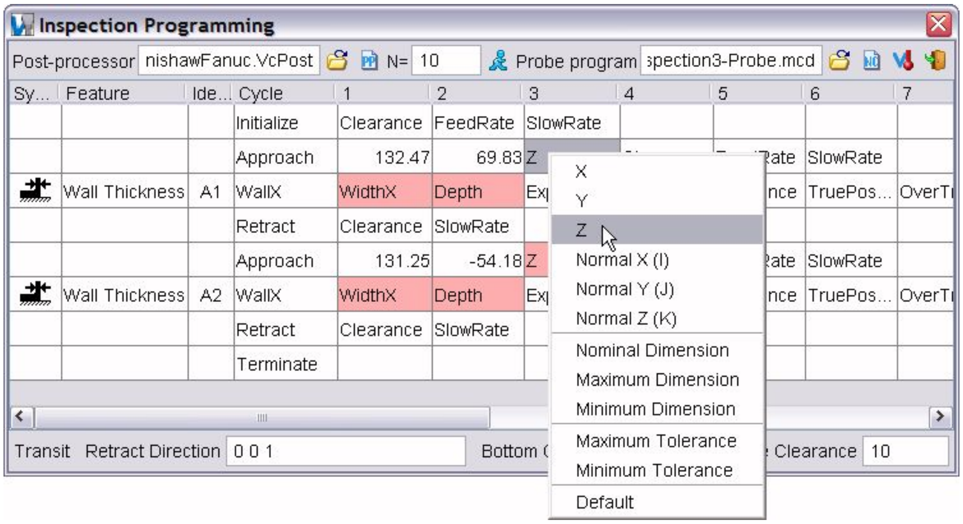
You can view the post-processor content, and edit it if you wish, by clicking the button. Development of a post-processor for probe cycle programming is documented elsewhere (refer to CGTech PostProcessor Help, found in postprocessor.pdf, located in the hhelp directory of your VERICUT installation). Note that a default name is supplied for the G-Code program that will be generated, using the current inspection file name with the ".VcInspect" extension replaced by "-Probe.mcd". The two blank pink cells in the table, in the "Cycle" column, need to be filled with the name of the cycles we wish to use. Because the two webs are in the machine's YZ plane, we can use a "WallX" cycle to measure each of them. Double-click on one of the pink cells and select the cycle name from the drop-down list. Then repeat for the second feature. The cycle names presented in the drop-down lists are defined in the post-processor, and will vary from one target machine to another.



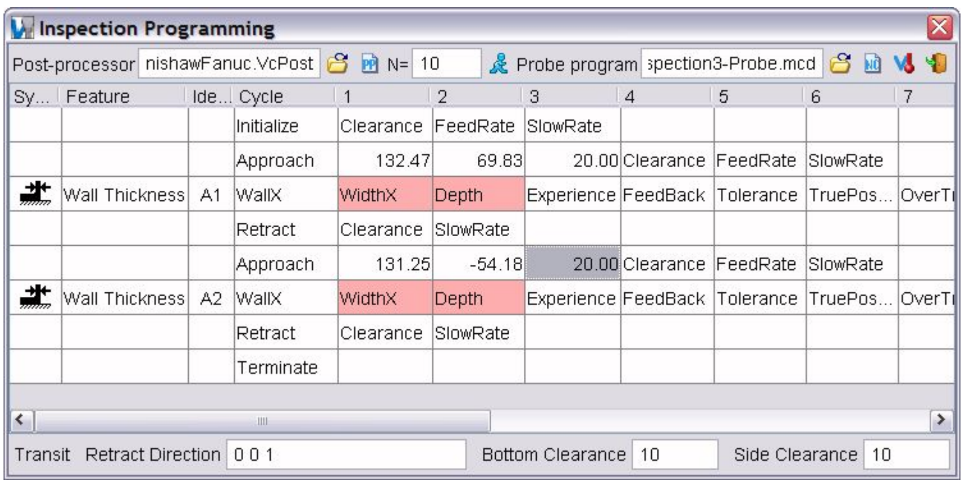
The numbered columns in the table contain the names of parameters for the cycles. Each probe cycle row is preceded by an "Approach" row, and followed by a "Retract" row. The table starts with an "Initialization" row and ends with a "Termination" row. All of these rows can have associated parameters in the numbered columns. Pink cells represent mandatory parameters. Once they are edited to contain numeric values, the color disappears. Optional parameters are not colored initially. You can hover over any parameter cell to see the parameter name displayed as a tool-tip, even after it contains a numeric value.



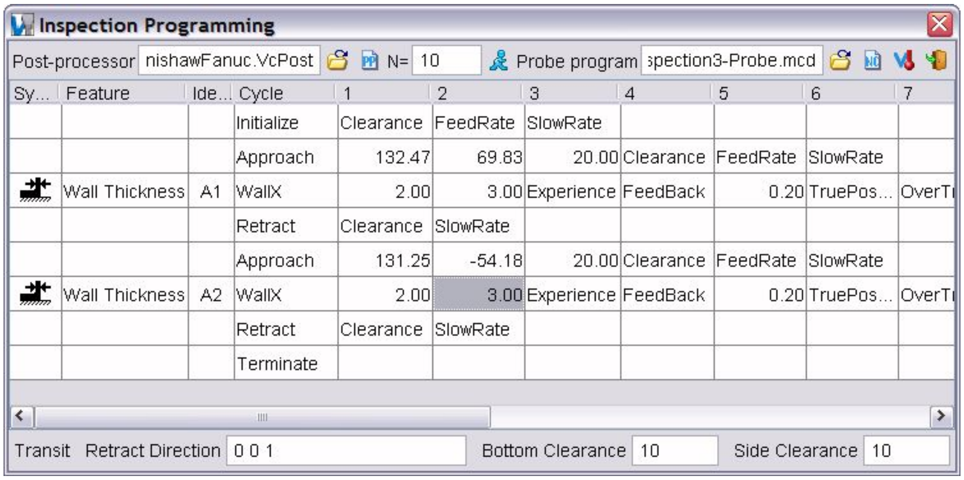
You will see that the X and Y parameters in the two "Approach" rows have been filled in automatically. These points will be in the center of the selected webs, as required by the probe cycle, but the Z parameter, while mandatory, has not been supplied. This is because the approach should place the probe near to, but not on, the workpiece. To see the value of Z on top of the first web, right-click on the pink Z cell in the first "Approach" row, and pick "Z" from the pop-up menu.



Then edit the value to increase it by a couple of millimeters. This should leave the probe above the web at the end of the approach. Repeat for the second web's approach.

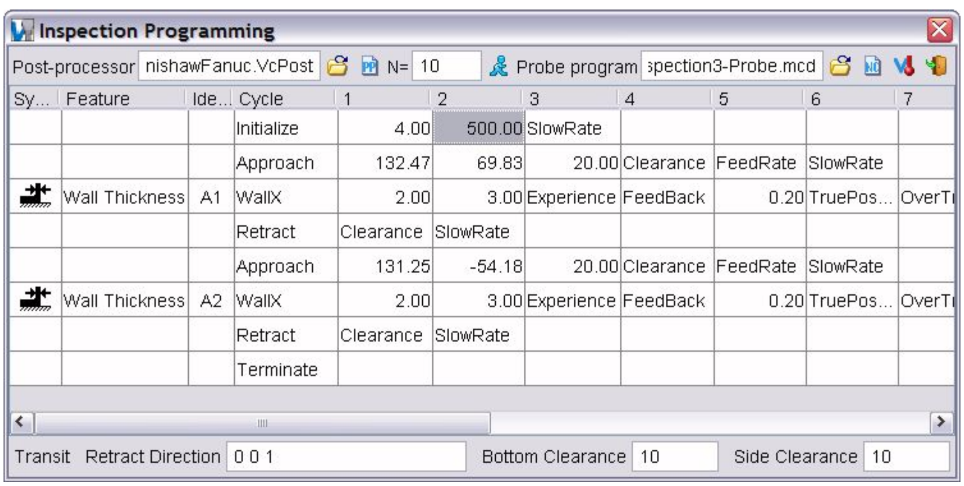


Before we can run the post-processor to turn the table content into a G-Code program, we need to eliminate all the pink by supplying numeric values for the mandatory parameters, and perhaps supply some of the optional parameters too. Parameter names and their interpretation are a function of the post-processor and the probe cycles available on the target machine. So the following is only representative of the process. To fill the "WidthX" cells we can right-click on each and select "Nominal Dimension" from the drop-down menu. The value is derived from the feature's "Dimension" column in the original Inspection Sequencing table. We should fill the "Tolerance" cells for each feature in the same way, even though these parameters are optional. Right-click on each such cell and pick "Maximum Tolerance" from the drop-down menu. The value comes from the "Tolerance" column of the original table. The remaining two pink cells are "Depth", which is the distance down the web, from the top, that the probe should move before stepping inward to make the measurement. There is no suitable value available in the drop-down menu, so we can just edit these cells to supply one.



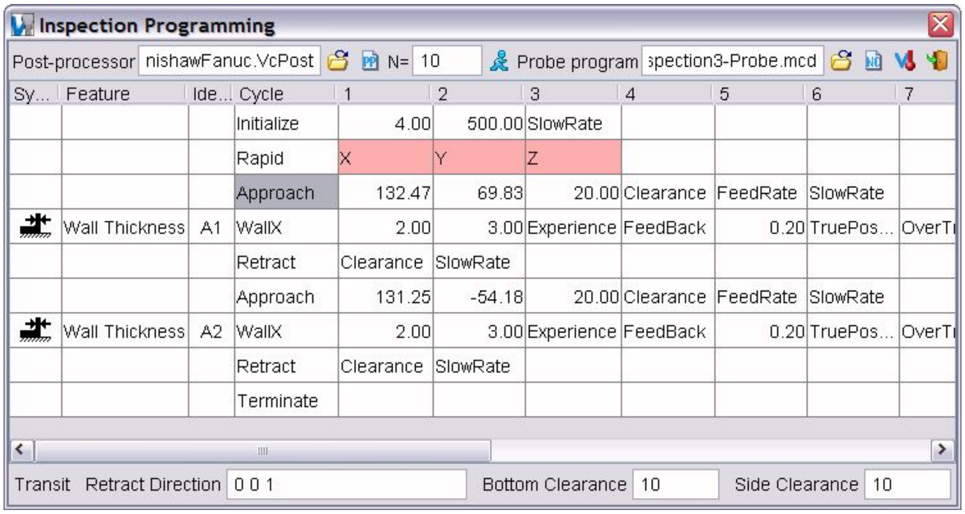
Sy...	Feature	Ide...	Cycle	1	2	3	4	5	6	7
			Initialize	Clearance	FeedRate	SlowRate				
			Approach	132.47	69.83	20.00	Clearance	FeedRate	SlowRate	
	Wall Thickness	A1	WallX	2.00	3.00	Experience	FeedBack	0.20	TruePos...	OverTi
			Retract	Clearance	SlowRate					
			Approach	131.25	-54.18	20.00	Clearance	FeedRate	SlowRate	
	Wall Thickness	A2	WallX	2.00	3.00	Experience	FeedBack	0.20	TruePos...	OverTi
			Retract	Clearance	SlowRate					
			Terminate							

Now that all the pink is gone, what else may we need before generating the probe cycle program? The "Initialization" row contains three parameters, "Clearance", "FeedRate" and "SlowRate". These parameters appear in "Approach" and "Retract" rows too, so they can be adjusted for each feature if necessary. "Clearance" is the height above the approach point where the probe should be armed, and is also used after each cycle for the retract distance. "FeedRate" is the normal feed rate which will be used for the first part of each approach. "SlowRate" is the much slower speed of the probe once it is armed, for which the probe cycles supply a default if necessary. So let's provide blanket "Clearance" and "FeedRate" values in the "Initialization" row.

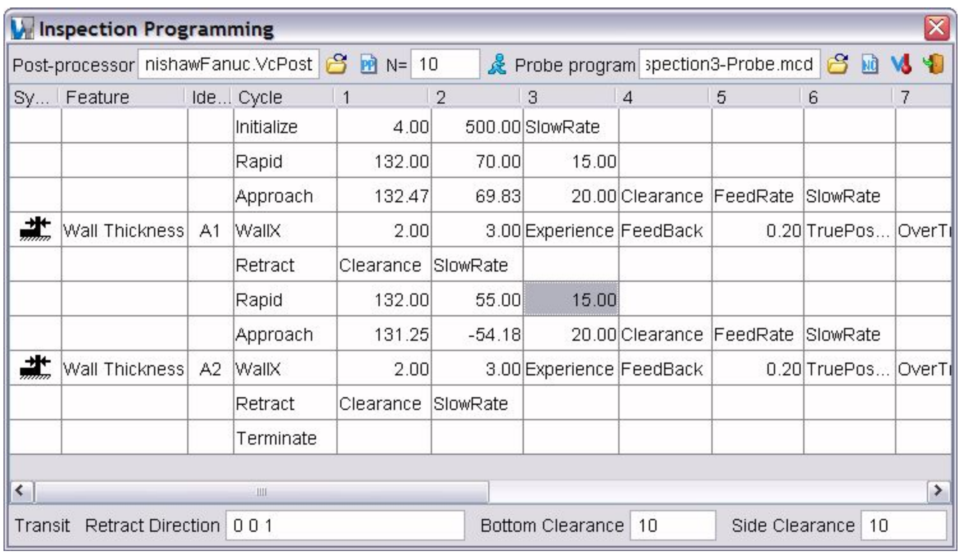


Sy...	Feature	Ide...	Cycle	1	2	3	4	5	6	7
			Initialize	4.00	500.00	SlowRate				
			Approach	132.47	69.83	20.00	Clearance	FeedRate	SlowRate	
	Wall Thickness	A1	WallX	2.00	3.00	Experience	FeedBack	0.20	TruePos...	OverTi
			Retract	Clearance	SlowRate					
			Approach	131.25	-54.18	20.00	Clearance	FeedRate	SlowRate	
	Wall Thickness	A2	WallX	2.00	3.00	Experience	FeedBack	0.20	TruePos...	OverTi
			Retract	Clearance	SlowRate					
			Terminate							

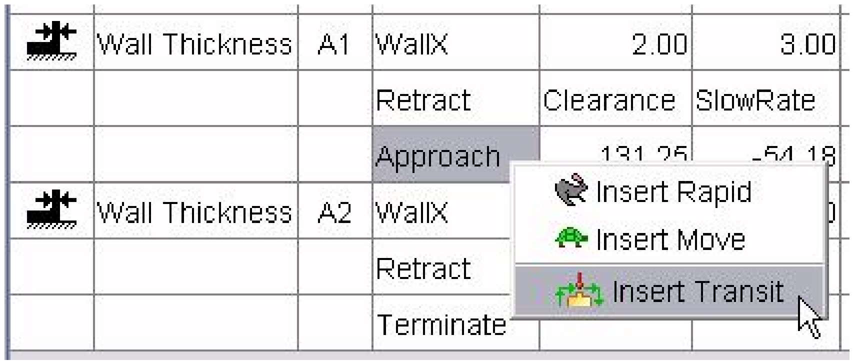
What's missing? It would be faster to use rapid motion before each approach. So right-click on the first "Approach" cell and pick "Insert Rapid" from the drop-down menu. Note that you can also insert moves at the normal feedrate if you wish to avoid obstacles.



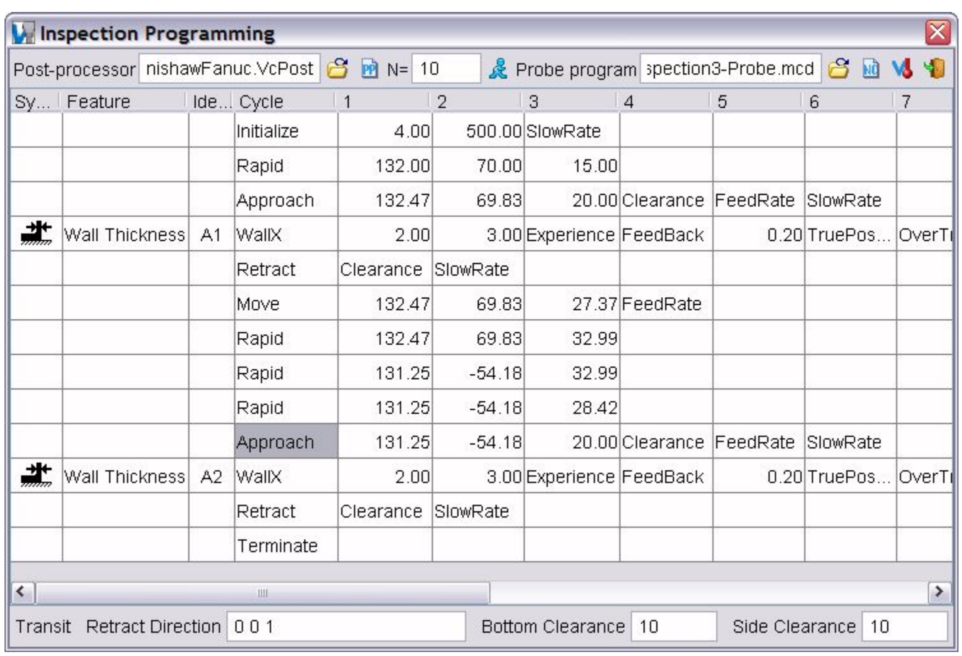
Replace the pink cell contents with suitable coordinates and repeat for the second web if you think a rapid motion over this short distance would make sense.



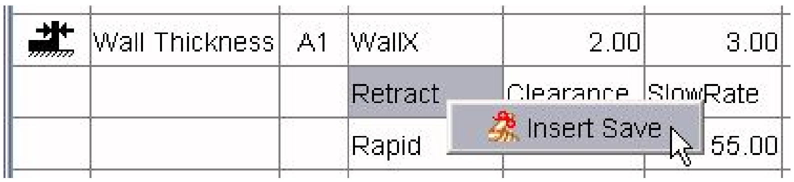
There is an alternative more automated way to insert motion between the retract from one feature and the approach to the next. If you wish to explore this, right-click on the second "Rapid" cell you have just inserted and pick "Delete" from the drop-down menu. When you right-click on an "Approach" cell that has a "Retract" cell immediately above, the drop-down menu contains three choices.



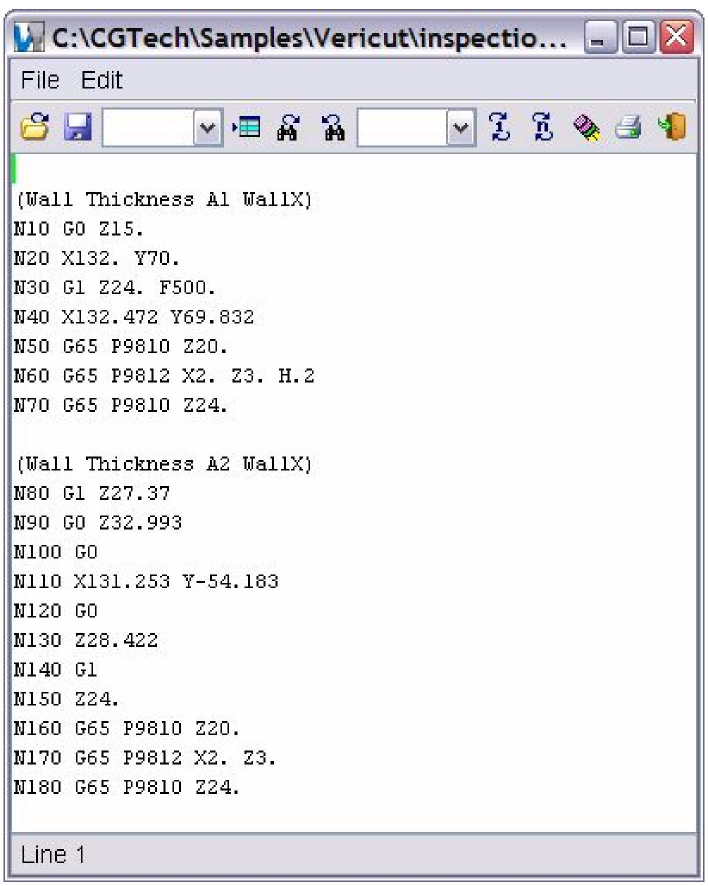
Picking "Insert Transit" will cause insertion of from one to four "Move" and "Rapid" rows. The "Transit" parameters along the bottom edge of the dialog are relevant. The "Retract Direction" is an IJK vector which determines how the probe is moved away from the workpiece, In this sample the probe is moved parallel to the Z axis. The "Bottom Clearance" defines how close the tip of the probe can come to any obstacle between the features. The workpiece itself, and any fixtures are considered potential obstacles. The "Side Clearance" defines how close the side of the probe is allowed to get to the obstacles. If you have a long probe and not much variation in workpiece height, you could use a side clearance just a little bigger than the probe's radius. With a short probe or a workpiece surrounded by large fixtures, it would be wiser to use a value generously bigger than the radius of the probe's holder. The first row inserted will be a "Move" to raise the probe above the prior feature by at least the bottom clearance. Then a "Rapid" row will raise it further to clear any obstacles between the feature, another "Rapid" will transit the probe, and a final "Rapid" will drop the probe back to within the bottom clearance of the next feature.



Some suites of probe cycles have the ability to save the result of one measurement and compare it with other measurements in a later cycle. For example, if you wanted to check that the Z dimension of one surface matched that of another, you could save the result of probing the first surface, then on the second one use a cycle that compared the new probed Z with the stored value. While it doesn't make sense for this example, as we won't be comparing the webs with each other, if the post-processor is written to handle the capability, you could insert a "Save" after any cycle by right-clicking on the corresponding "Retract" cell and picking "Insert Save" from the pop-up menu.



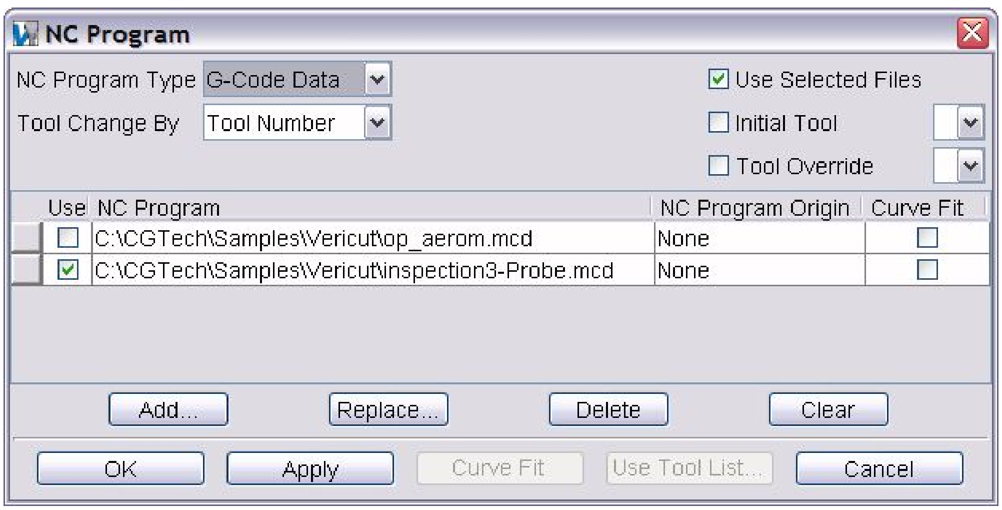
We are ready to click the button to run the post-processor. Hopefully this will be an anti-climax, with no error messages in additional debugging dialogs. To see the resulting G-Code program, and perhaps edit it, click o the button.




```
(Wall Thickness A1 WallX)
N10 G0 Z15.
N20 X132. Y70.
N30 G1 Z24. F500.
N40 X132.472 Y69.832
N50 G65 P9810 Z20.
N60 G65 P9812 X2. Z3. H.2
N70 G65 P9810 Z24.

(Wall Thickness A2 WallX)
N80 G1 Z27.37
N90 G0 Z32.993
N100 G0
N110 X131.253 Y-54.183
N120 G0
N130 Z28.422
N140 G1
N150 Z24.
N160 G65 P9810 Z20.
N170 G65 P9812 X2. Z3.
N180 G65 P9810 Z24.
```

The comments are derived from the second, third and fourth columns of the table for each feature. Note that the first block number in the program comes from the N= 10 entry box at the top of the Inspection Programming dialog. So you could use a much larger value if you intend to cut and paste the program into the one that cuts the part.

You can simulate the probe cycle program by clicking on the button. This will trigger display of the "NC Program" dialog, append the new program's file name to the end of the list, tick the "Use Selected Files" box, untick any other files in the list, and force an "Apply". You should then be able to click on **VERICUT**'s or button.



Also see "**VERICUT Inspection**", in the *Using VERICUT* section, in the *CGTech Help Library*.

Customizing VERICUT Inspection

The behavior of the Inspection Sequencing dialog can be customized in a number of ways by defining one or more text files and having an environment variable, CGTECH_INSPECTION_TOLERANCES, which points to one of them. You can define the default tolerance values that are placed in the dialog's table for each type of measurement. You can define the default instruments to be used for each feature type, and specify additional instrument types. And you can supply your own icons for the feature types, which will be used in the dialog and the generated reports.

Default Tolerances

There are hard-coded default tolerances values that conform to DAN 11304. When you add or modify a row in the sequencing dialog's table, and the nominal dimension is supplied by the system, a default tolerance will be generated too. If DAN 11304 isn't what you need, you can supply your own set of defaults, with different strategies for each feature type if necessary. A sample file contains an explanation of the flexibility available and the format to be used.

If you only need one set of default tolerances for all the workpiece types that you deal with, then the environment variable, CGTECH_INSPECTION_TOLERANCES should specify the fully qualified path and name of the sole text file. If you need more than one set of default tolerances, then the environment variable should reference a file that contains a list of the files that each contain a set of default tolerances. For example, if you machine parts for two customers, and one of them is still using a mix of units, you may have three default tolerance files;

```
CustomerA-Inch-Tols.txt
CustomerA-MM-Tols.txt
CustomerB-MM-Tols.txt
```

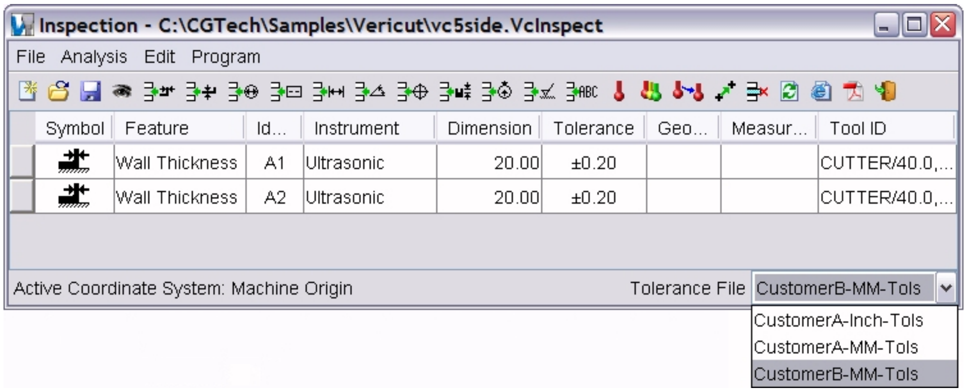
In this case, you would create one more file containing a list of these files. You might call it "Inspection-Tols.list" and it would contain something like the following;

```
C:\CGTech\Inspection\Defaults\CustomerA-Inch-Tols.txt
C:\CGTech\Inspection\Defaults\CustomerA-MM-Tols.txt
C:\CGTech\Inspection\Defaults\CustomerB-MM-Tols.txt
```

The environment might be defined in the batch file you use to invoke **VERICUT**, and could be as follows;

set CGTECH_INSPECTION_TOLERANCES=C:\CGTech\Inspection\Defaults\Inspection-Tols.list

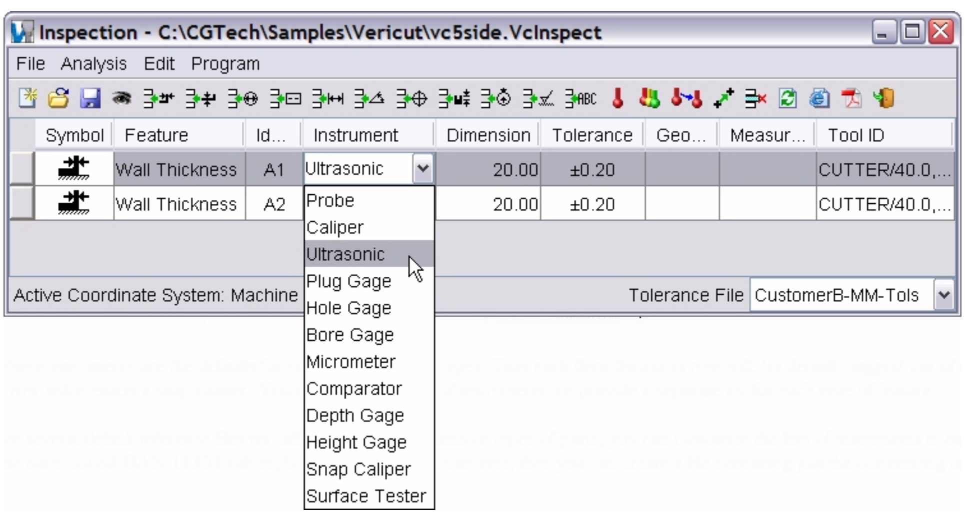
If you have more than one default tolerance file set up this way, then there will be a drop-down choice list in the bottom-right corner of the Inspection Sequencing dialog.



If you pick a different file of default tolerances while there are some rows in the table, then any tolerance values that have not been edited will be recalculated on the basis of the selected file's content.

Instruments

Each cell in the Instrument column of the Inspection Sequencing dialog's table contains a drop-down list of instrument types.

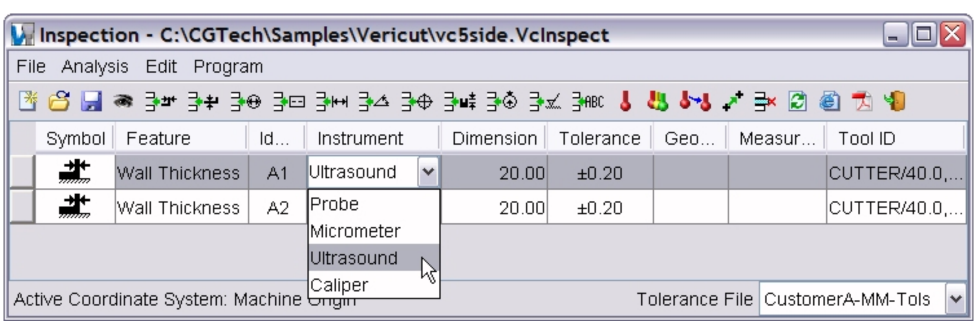


Some of these instruments are the defaults for some of the feature types. Thus each floor thickness row will, by default, suggest use of an ultrasonic depth gage, and each wall thickness row will mention a snap caliper. You can extend this list of instruments, or provide a separate list for each type of feature.

If you have several default tolerance files for different customers, units or types of parts, you can customize the lists of instruments in each of them. If you have no default tolerance file and use the hard-coded DAN 11304 values, but need different instruments, then you can create a file containing just the customizing text for the instruments. Following is a sample of such text;

```
Instrument Wall  Micrometer
Instrument Wall  Ultrasound
Instrument Wall  Caliper
Instrument Floor Ultrasound
Instrument Plumb Line
Instrument Ruler
```

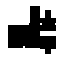
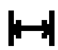
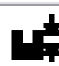
Each line starts with the keyword "Instrument". If the next word in a line is one of the feature types; Wall, Floor, Diameter, Length, Angle, Position, Depth, Radius or Finish, then you must provide similar lines for all the instruments that can be used to measure that feature type. The instrument name that follows the type in the first such line becomes the default for that feature type. The list of instruments for a feature type will replace the entire "un-customized" drop-down list. If no customizing lines are provided for any feature type, the "standard" list will still be used. If the second word on a line is not one of the feature types, then everything after the "Instrument" keyword is just the name of an additional instrument type which will be appended to the "standard" list. So with the sample customizing text shown above, "Plumb Line" and "Ruler" will be appended to the list for all feature types except "Wall" and "Floor". On the other hand, the drop-down list for a "Wall" feature will be much shorter.



Note that "Probe" is offered as an instrument type regardless of whether the list has been customized. This ensures compatibility with the Programming capability.

Icons

By default there are several 24x24 pixel, monochrome icons used in the left-most column of the Inspection Sequencing and Inspection Programming dialogs' tables and the Inspection Reports, which denote the type of feature to be measured.

Feature Type	Icon	File Name
Wall Thickness		InspectW.gif
Floor Thickness		InspectF.gif
Hole Diameter		InspectD.gif
Datum Plane		InspectS.gif
Distance		InspectL.gif
Angle		InspectA.gif
Position		InspectP.gif
Hole Depth		InspectH.gif
Radius		InspectR.gif
Surface Finish		InspectT.gif
Note		InspectN.gif

If these icons are not to your taste, you can redesign them, but making them larger is not recommended. The icon files are distributed as part of the file, CGTech.jar, which you will find in the "classes" folder under your CGTech installation directory. A "jar" file is a variation of a "zip" file which commonly available file compression programs can manipulate.

We suggest that you make a copy of CGTech.jar, calling it "CGTech.jar.zip" so that a program such as WinZip can recognize it. Then you can use WinZip to extract the icon files that you wish to adjust. Note that in each icon file, the white background color is actually transparent. You will need to use a paint program that supports GIF Version 89a, to handle the transparency. When you have adjusted, or entirely replaced the image, save it with the same file name and use WinZip again to update the compressed version in

CGTech.jar.zip. When you have changed all icons necessary, delete the original CGTech.jar and rename CGTech.jar.zip to take its place.

Die Sinking Simulation (Die Sinking Simulation window)

Location: **Analysis menu > Die Sinking Simulation**

Opens the **Die Sinking Simulation window** enabling you to select electrodes and simulate the die sinking process.

Simulation Display Features

Stock — Enables you to select the **Stock** component that is to be used for the simulation. Select the name of the Stock component from the pull-down list.

Display Stock — use to specify whether or not to display the **Stock** model during the simulation. The stock model can be displayed as **Solid** or **Translucent**. The stock model remains displayed during the Burn process unless **Display Burned Material** is toggled on. In this case the Stock model will be turned off when you select **Burn** and only the material removed by the simulated EDM burn will be displayed at the end of the burn process.

Display Electrodes — use to specify whether or not to display the **Electrode** model(s). Electrode models can be displayed as **Solid** or **Translucent**. Electrode models are always toggled off after **Burn** is selected.

Display Burned Material — use to specify whether or not to display the material that is removed by each electrode during the simulation. The "burned material" can be displayed as **Solid** or **Translucent**.

Animate — use to specify whether or not to display electrode animation during the "burn" process. Use the slider bar to control the speed of the animation. If toggled off, displays only the result of the burn process.

Electrode Table

Table of electrodes selected for use during Die Sinking Simulation.

On — Toggles on and off to activate/deactivate individual electrodes in the table. Only the active electrodes will be used during the "burn" process.

Electrode — The name of the electrode component.

Overburn — this column enables you to specify an "overburn" value for each electrode. The "overburn" value is applied as an offset to each electrode during "burn" processing.

Volume Removed — this column displays the volume of material that was removed by each active electrode during the burn process. The "volume removed" value is updated in the table as each electrode completes the burn process.

Click and drag the button in the first column of each row to change position of an electrode in the table. "Active" electrodes will be processed sequentially starting at the top of the table.

Add — use to add additional electrodes to the table. Select the electrode to be added from the pull-down list then select Add. The pull-down list will contain all electrode components found in the Component Tree that are not currently in the table.

Delete — use to remove the highlighted electrode from the table.

Shortcut: You can right-click in the Electrode table to display a menu containing **Add** and **Delete**. These provide the same functionality described above. Clicking on **Add** will display the pull-down list of available electrodes that you can choose to add to the table.

Set All — use to activate "all" electrodes in the table.

Clear All — use to deactivate "all" electrodes in the table.

Die Sinking Simulation features

Compare With Design — displays the AUTO-DIFF window enabling you to compare the stock model at the end of the burn process with the design model.

Save Model — enables you to save the stock model that resulted from the burn process as a VERICUT solid (.vct) file.

Apply — Use to save the current Die Sinking Simulation settings.

Burn — Use to start the "burn" process.

Restore — Restores the stock model to the state it was in prior to the "burn" process.

Report — Use to create a **Die Sinking Simulation Report**.

Close — Closes the **Die Sinking Simulation window**.

Also see "**Die Sinking Simulation**", in the *Using VERICUT* section, in the *CGTech Help Library*.

OptiPath Menu

Control (OptiPath Control window)

VERICUT Users:

VERICUT Location: **OptiPath menu > Control**

Project menu > Output: Logs tab, Settings button

VERICUT toolbar short cut: 

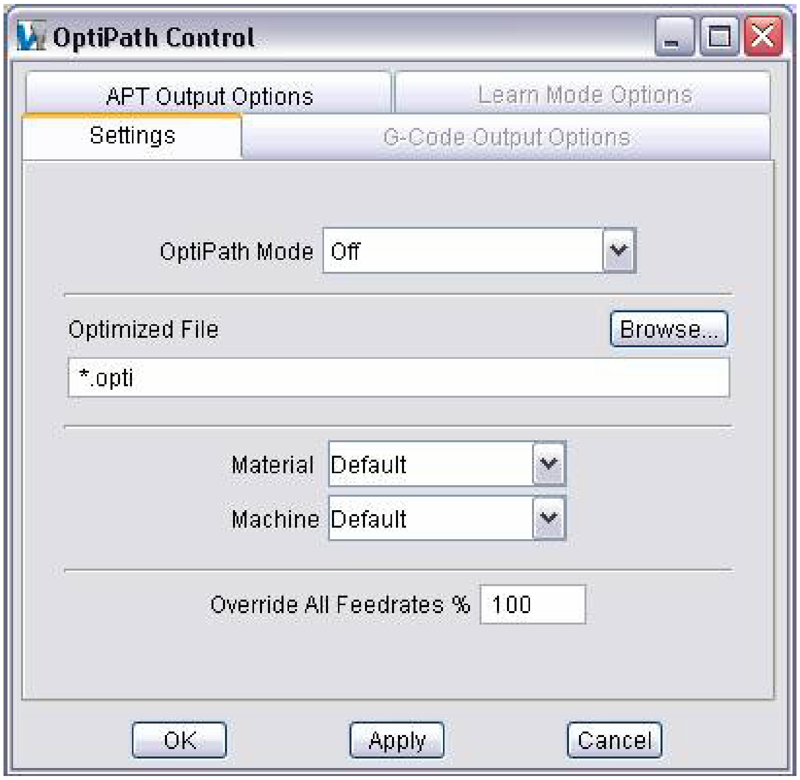
Mold and Die Users:

Mold and Die Location: **Preview and Optimize page > Optimize Tool Path**

Notebook Feature: 

Opens a window to control when tool paths are optimized by OptiPath, specify the file to receive the optimized data, choose the type of stock material and NC machine involved in the cutting the workpiece. Feed rates and spindle speeds can be monitored via the **Status window**. A summary of the cutting conditions encountered during optimization and min/max optimized feed rates is written to the Log and available for use when creating a report file showing an overview of the results of the current VERICUT session.

See "**Overview of Optimizing NC Programs**" in the *Using VERICUT* section, in the *CGTech Help Library* for information about using OptiPath to optimize your NC programs.



Settings tab — Features on this tab enable you to control when and how NC programs are optimized.

G-Code Output Options tab — Features on this tab enable you to configure how the machine control will process machine code data, and configure settings referenced when G-Code NC program files are optimized.

Learn Mode Options tab — Features on this tab enable you to control how and when OptiPath records are written to the Tool Learn Library file during **Learn From Toolpath** processing.

APT Output Options tab — Features on this tab enable you to configure settings referenced when APT NC programs are optimized.

OK — Applies the changes and closes the **OptiPath Control window**.

Apply — Applies the changes and leaves the **OptiPath Control window** open.

Cancel — Closes the **OptiPath Control window** without applying changes.

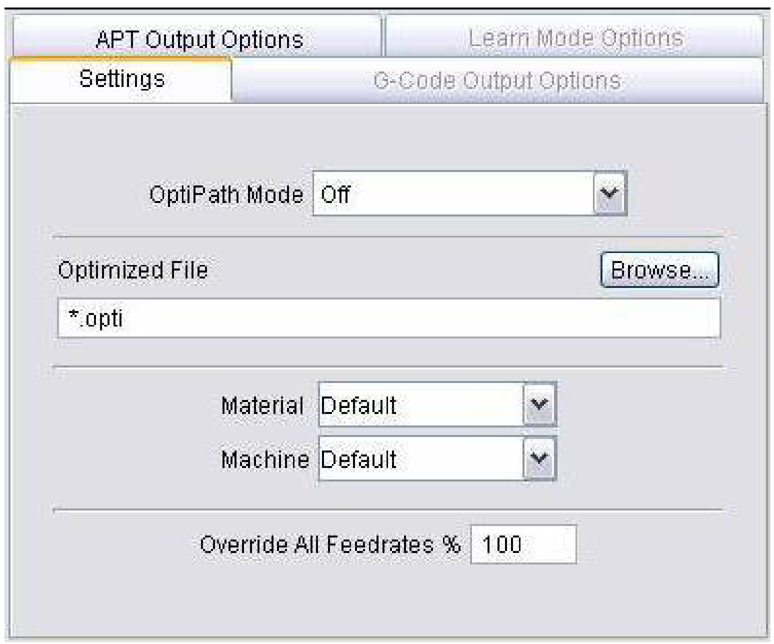
OptiPath Control window, Settings tab

VERICUT Location: **OptiPath menu > Control**

Project menu > Output: Logs tab, Settings button

VERICUT toolbar short cut: 

Features on this tab enable you to control when and how tool paths are optimized.



OptiPath Mode — Controls when and how tool paths are optimized. All OptiPath modes can be used with both Standard and FastMill cut modes.

Options:

Off — Turns optimization off. Optimization is automatically turned off when the end of the NC program file has been reached, or when the NC program is reset, or rewound.

On — Turns OptiPath on and writes optimized NC program data to the file specified with the **Optimized File** feature. For optimization to occur, OptiPath records must be "associated" with the cutting tools in the NC program file. There are two methods of doing this:

1. **Tool Library association method -** associate the OptiPath tool **Description** and number of **Teeth** with a tool in a Tool Library file, then choose the stock **Material** and **Machine** in the OptiPath Control window.

2. **Tool List association method -** choose the stock **Material** and **Machine** in the OptiPath Control window, then build a tool list that associates OptiPath records with cutting tools to be optimized in the NC program file.

With either method, tools without "cutting tool-to-OptiPath record" associations are not optimized. If both methods are used, associations in the tool list override those stored with tools in the Tool Library.

Prompt While Cutting — Turns OptiPath "**On**" and interactively prompts for optimization data at ***each*** tool change by displaying the Optimization Settings window. Use the Optimization Settings window features to enter data for optimizing the cuts performed by the current tool. When you finish, VERICUT automatically stores the information in the Tool Library file, and sets tool properties to point to the new OptiPath record. VERICUT restarts the simulation, turns OptiPath on and uses this information to write optimized NC program records to the specified Optimized File until the next tool change is encountered.

See "**Using OptiPath – Prompt While Cutting**" in the *Using VERICUT* section, in the *CGTech Help Library* for more information.

Learn From NC Program — Turns OptiPath "**On**" and automatically creates a new Tool Library file for the "current" project file. The new Tool Library file is created in the file specified by **Tool Learn Library** on the Learn Mode Options tab. The OptiPath records in the Tool Library file that is created can then be manually fine-tuned using features in the OptiPath window, and then used to optimize the NC program.

Learn From NC Program can be turned **On** / **Off** at any time during cut processing. Each time **Learn From NC Program** is turned on, it creates a Tool Library file. The file contains OptiPath settings for every tool loaded since cutting was started with **Learn From NC Program** turned "**On**", including the tool that was cutting at time that **Learn From NC Program** was turned "**Off**". These settings are based only on cutting conditions that occurred between the time cutting was started with **Learn From NC Program** was turned "**On**" and the time when cutting stops and **Learn From NC Program** turned "**Off**". To get complete and accurate information for each tool, process the whole job through **Learn From NC Program**. Turning **Learn From NC Program On** / **Off** during cutting can be used to analyze cutting conditions for a specific group of cuts.

When **Learn From NC Program** mode is turned on, any tools with matching OP records are skipped, not "learned". Any tools without OP records are "learned" and a new OP record is created. The old and new records are combined in a new OP library file.

For each tool, OptiPath finds the maximum volume removal rate and chip thickness that occurred during cutting and uses them for corresponding OptiPath settings for the tool. The optimization mode is set to a combination of "**Chip Thickness**" and "**Volume Removal**" for each tool. **Axial Depth** and **Radial Width** values are determined by the cut that produces the highest volume removal rate. Default values are used for other settings unless they have specifically been changed using **Learn Mode Options tab**.

Learn From NC Program only automatically creates an updated Tool Library file (.tls) containing the "**OP Description**" and "**# Teeth**" values associated with the newly created OptiPath records. It does not automatically optimize the tool path or produce the Optimized File. Once **Learn From NC Program** creates the updated Tool Library file, you are prompted whether or not you want to optimize the NC program using the newly created Tool Library file.

See "**Using OptiPath – Learn From NC Program**" in the *Using VERICUT* section, in the *CGTech Help Library* for more information.

Optimized File — Specifies the name of the modified (optimized) tool path file to receive optimized NC program data. The format of this file, APT vs. G-Code, is the same as the tool path file being optimized. When an APT NC program is optimized the programmed "FEDRAT" records are commented out using "$$" and written to the modified path for reference. The Optimized File name must be different from that of the original NC program file. OptiPath will not permit the original NC program file to be overwritten.

When optimizing multiple NC program files an optimized NC program file is output for each NC program file simulated. Each output file has an ascending sequence number appended to the end of the base file name. For example: assume **Optimized File**="part.mco" is entered (without the quotes). Optimizing 2 NC programs outputs the following files:

part001.mco => optimized NC program file created from NC program 1

part002.mco => optimized NC program file created from NC program 2

If a dot "**.**" is the last character of the optimized tool path file name, then three digits are appended to the file name as the extension. Continuing with the above example: assume **Optimized File**="part." is entered. Optimizing the same 2 NC programs outputs the following files:

part.001 => optimized NC program file created from NC program 1

part.002 => optimized NC program file created from NC program 2

If an asterisk "*" is used in the optimized tool path file name, the original tool path file name is used as the output file name in place of the asterisk. If characters follow the asterisk, the input tool path file name extension is replaced with the characters following the asterisk. Examples follow.

Original tool path file	Optimized File name	Output file name
dir1/filename.tp	dir2/*	dir2/filename.tp
dir1/filename.tp	dir2/*.op	dir2/filename.op
dir1/filename.tp	dir2/*op	dir2/filenameop
dir1/filename.tp	dir2/*op.tp	dir2/filenameop.tp
filename.tp	*.op	filename.op

Only a single asterisk is acted on in the optimized NC program file name, with one exception: asterisk dot asterisk ("*.*"). This case behaves exactly like the single asterisk file name described above. This is ***NOT*** a regular expression replacement; therefore any characters before the asterisk are ignored and lost. When there is more than one asterisk, only the first one is replaced. Examples follow.

Original tool path file	Optimized File name	Output file name
dir1/filename.tp	dir2/*.*	dir2/filename.tp
dir1/filename.tp	dir2/op*	dir2/filename.tp
dir1/filename.tp	dir2/op*.ext	dir2/filename.ext
dir1/filename.tp	dir2/*op.*	dir2/filenameop.*

Material — Identifies the stock material being cut. The material name is changed via clicking the arrow and selecting from a list of available materials, or clicking in the data field and typing a new material name.

Machine — Identifies the NC machine being used to cut the workpiece. This feature functions similar to **Material** described above.

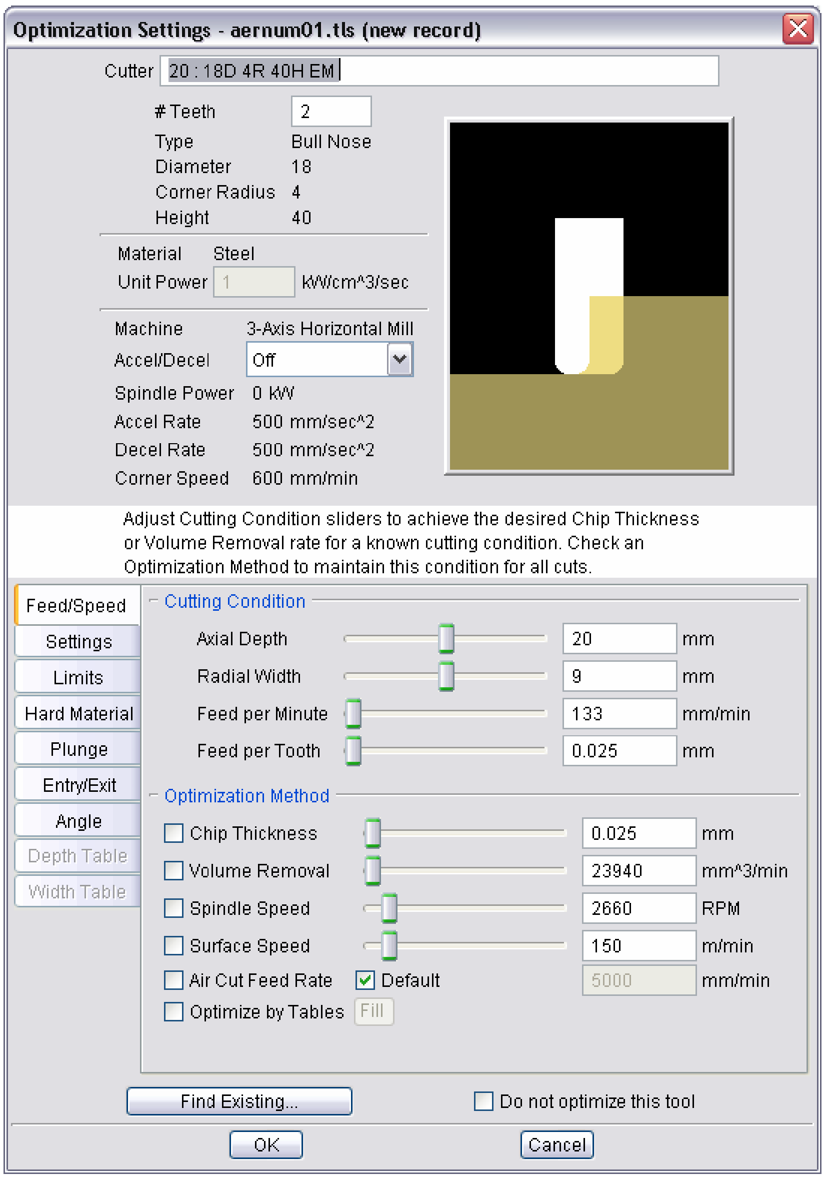
NOTE: For optimization to occur the **Material** and **Machine** names must match those of the desired OptiPath records in the Tool Library file.

Override All Feedrates — Overrides optimized feed rates calculated for all tools optimized. 100% uses optimized feed rates as is, 50% cuts feed rates in half, 200% doubles the feed rates, and so on.

To OptiPath Control window

Optimization Settings window

The Optimization Settings window is displayed by VERICUT each time that a tool change is encountered during simulation when **OptiPath Mode**, in the OptiPath Control window: Settings tab, is set to **Prompt While Cutting**.



The upper part of the Optimization Settings window provides information about the current tool, material, and machine that VERICUT has extracted from Tool Manager, the machine file, the OptiPath Control window, etc.

The tabs on the lower part of the Optimization Settings window are identical to those found on the OptiPath window. See **OptiPath window**, also in the *VERICUT Help* section for additional information.

Find Existing — Displays the Search OptiPath Record window enabling you to search Tool Library files and Optipath files for existing OptiPath records. See **Search OptiPath Record window**, also in the *VERICUT Help* section for additional information.

Do not optimize this tool — Turns optimization off for the current tool.

OK — Accepts the current OptiPath settings. VERICUT automatically stores the information in the Tool Library file, sets tool properties to point to the new OptiPath record, and restarts the simulation with OptiPath turned on. VERICUT uses this information to write optimized NC program records to the specified Optimized File, as the simulation continues, until another tool change is encountered.

Cancel — Causes VERICUT to stop prompting for remaining tools but continues optimizing where possible.

Also see "**Using OptiPath – Prompt While Cutting**" in the *Using VERICUT* section, in the *CGTech Help Library*.

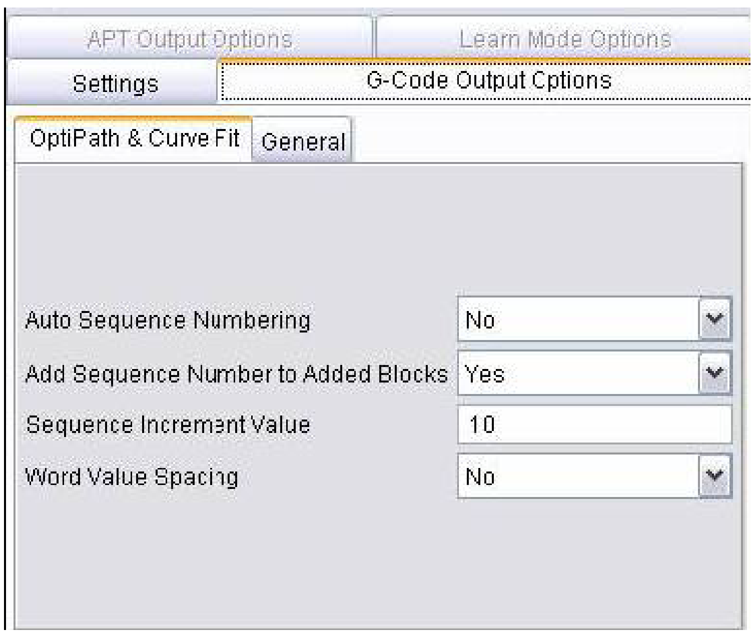
OptiPath Control window, G-Code Output Options tab

VERICUT Location: **OptiPath menu > Control**

Project menu > Output: Logs tab, Settings button

VERICUT toolbar short cut: 

Enables you to configure how the machine control will process machine code data and configure settings referenced when G-Code NC program files are optimized by OptiPath. The G-Code Output Options tab is only available when **NC Program Type** is set to **G-Code Data**.



OptiPath & Curve Fit tab — Features on this tab enable you to configure settings referenced when G-Code NC program files are optimized by OptiPath.

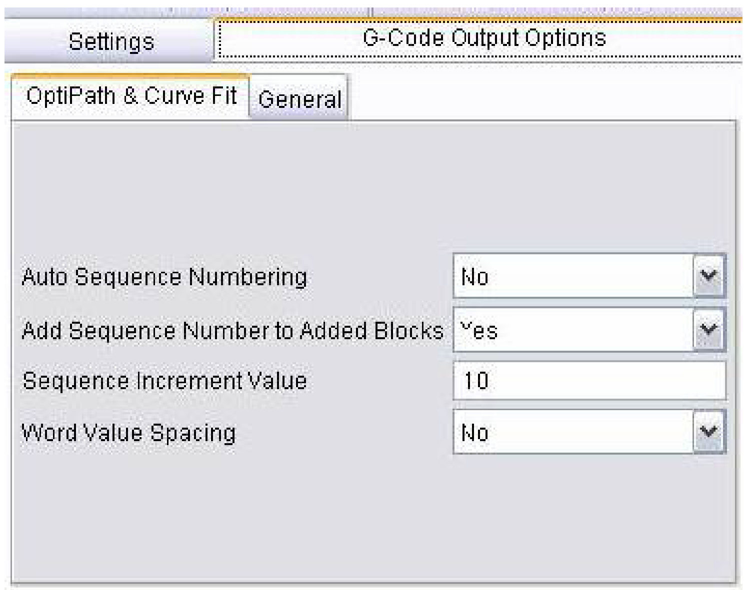
General tab — Features on this tab enable you to configure general guidelines for how the control processes machine code data and number equations.

To OptiPath Control window

OptiPath Control window, G-Code Output Options tab, OptiPath & Curve Fit tab

VERICUT Location: **OptiPath menu > Control > G-Code Output Options tab**

Features on this tab enable you to configure settings referenced when G-Code tool path files are optimized by OptiPath.



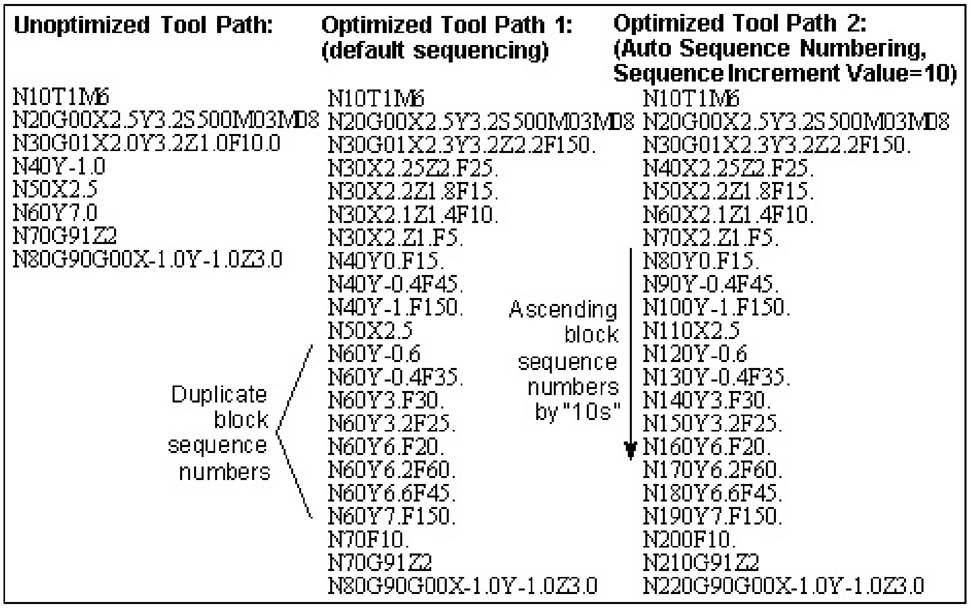
The settings are used to control block sequencing and word/value spacing in the optimized output tool path file. Note that by default, optimized blocks receive the same sequence number, if any, that appears on the original block processed. This typically results in multiple blocks having the same sequence number. The features on this tab are the same as those found on the **Control Settings window: OptiPath & Curve Fit tab**.

Auto Sequence Numbering — When active, OptiPath re-sequences optimized NC program records in ascending order. The **Sequence Increment Value** (see below) is used for the beginning sequence number, and as the increment value for subsequent sequence numbers. Both optimized and non-optimized blocks with sequence numbers are re-sequenced. Sequence numbers are not added to un-optimized blocks without a sequence number.

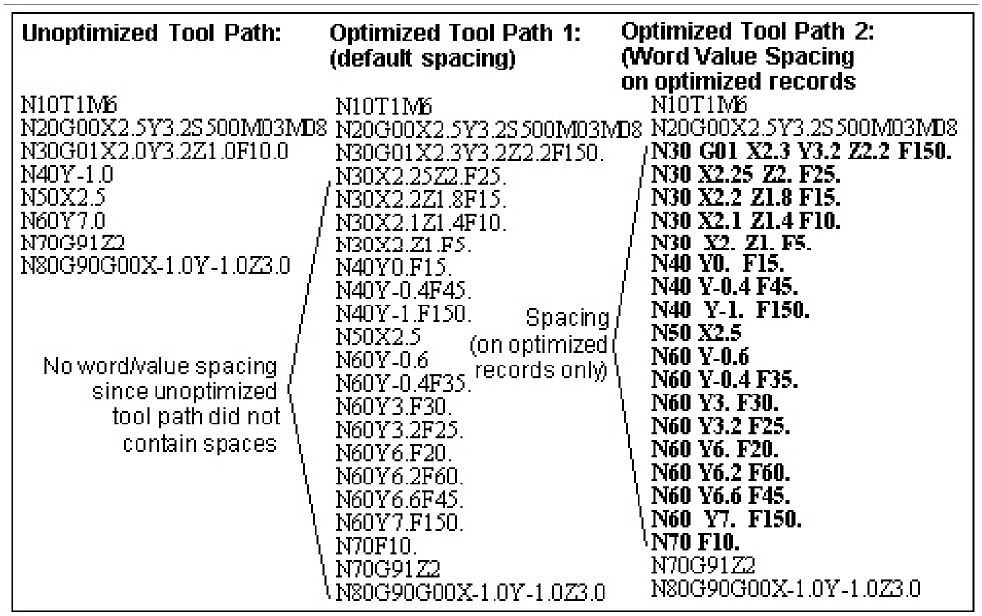
NOTE: If the general control setting **Default Word**="N", then "N" does not appear in the sequence number. The result is a sequence number without a word, for example: "50G01X5", such as required by Heidenhain controls.

Add Sequence Number to Added Blocks — When active, adds sequence numbers to blocks added during optimization via the **Add More Cuts** OptiPath setting.

Sequence Increment Value — Initial value and increment value for re-sequencing optimized G-Code NC program records.



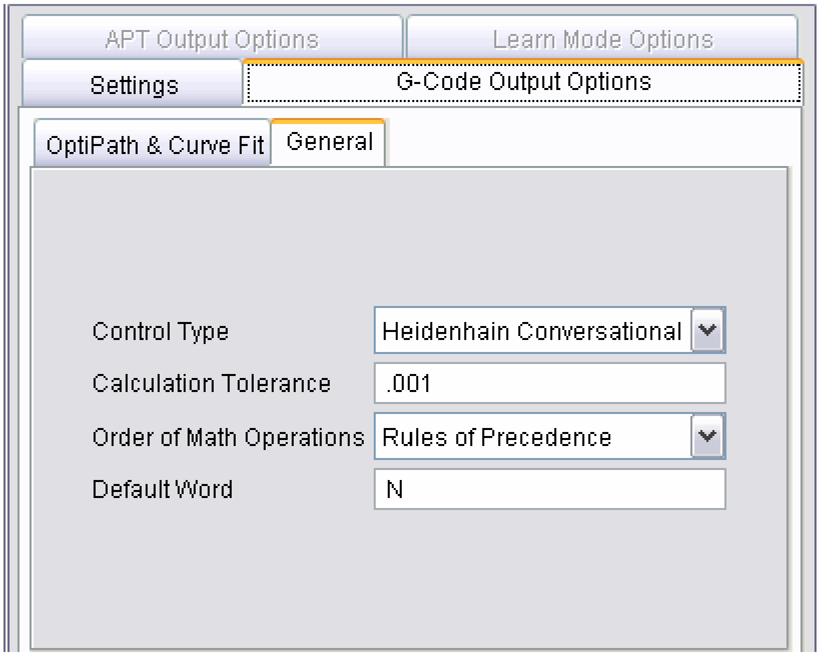
Word Value Spacing — When active, includes a space between word/value pairs in the optimized NC program file. Un-optimized records do not receive spacing. When this feature is not used, optimized NC program records are not spaced.



To G-Code Output Options tab

OptiPath Control window, G-Code Output Options tab, General tab

Features on this tab enable you to configure general guidelines for how the control processes machine code data and number equations.
The features on this tab are the same as those found on the **Control Settings window: General tab**.



Control Type — Sets the type of NC control being used. Choosing the proper option establishes a mode of operation consistent with how the control processes NC data.
Options are:

> **Generic —** Fanuc and most other controls using standard controller functions.
>
> **NUM —** French-made control. This option affects how G-Code data is interpreted, for example: how variables are initialized and processed, and how to interpret parameters to macros that shift the location of the tool path.
>
> **Heidenhain Conversational —** Heidenhain Conversational control. This option causes VERICUT to recognize the "L" word as an optimizable word. Also, the "L" word is included with any cuts added during optimization.

Calculation Tolerance — Tolerance for rounding mathematical evaluations, such as determining if two calculated values are equal to, greater than, or lesser than each other.

NOTE: The **Calculation Tolerance** should be adjusted to eliminate "invalid circle" errors caused by circle calculations that produce differences larger than the tolerance value when processing G-Code data.

Order of Math Operations — Order in which math operations are performed.

Options are:

Rules of Precedence — Follow the basic rules of math precedence:

1. Perform exponential (power).
2. Perform multiplication and division operations.
3. Perform addition and subtraction operations, example: 5 + 5.3 * 3 * sin(30)= 12.95

Enclosing math operations in parenthesis causes them to be performed before those not enclosed in parenthesis. The same rules of precedence are applied to math operations enclosed in parenthesis.

Left to Right — Use left to right sequence, example: 5 + 5.3 * 3 * sin(30)= 15.45

Default Word — Word assumed by VERICUT for NC data blocks beginning with numbers, such as NC data for some Heidenhain controls. For example, for VERICUT to interpret blocks like "50G01X5", set **Default Word** to "N". VERICUT then interprets the block without error as "N50G01X5".

To G-Code Output Options tab

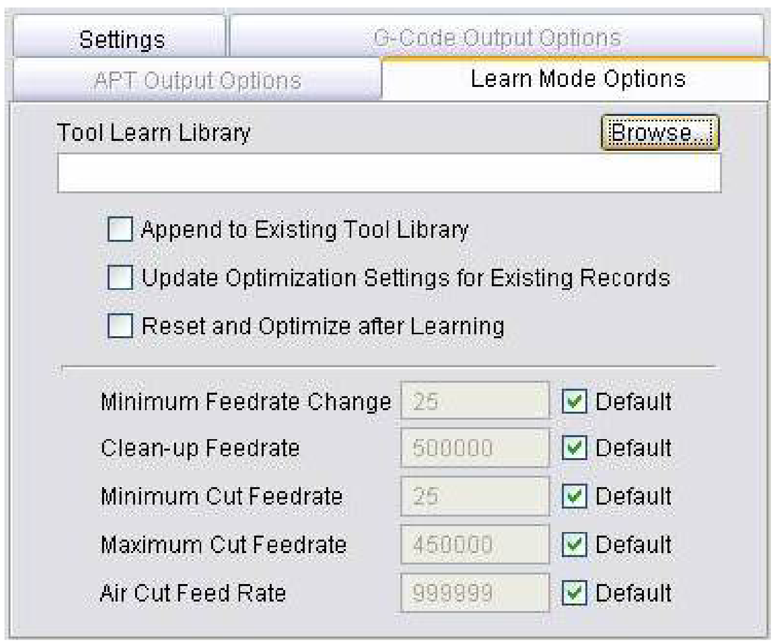
OptiPath Control window, Learn Mode Options tab

VERICUT Location: **OptiPath menu > Control**

Project menu > Output: Logs tab, Settings button

VERICUT toolbar short cut: 

Features on this tab enable you to control how and when OptiPath records are written to the ToolLearn Library file during **Learn From Toolpath** processing.



Enables you to override the default values for **Minimum Feedrate Change**, **Clean-up Feedrate**, **Minimum Cut Feedrate** and **Maximum Cut Feedrate** without going to the **OptiPath Control window: Settings tab**.

The **Learn Mode Options tab** is only active when **OptiPath Mode**, on the **OptiPath Control window: Settings tab**, is set to **Learn From Toolpath**.

Append to Existing Tool Library — If toggled "**On**", **Learn From NC Program** appends OptiPath records to the existing Tool Library file for the current project file. Otherwise the "Tool Library to be created" window will display enabling you to specify the */path/filename* for the new Tool Library file to be created.

Update Optimization Settings for Existing Records — If toggled "On", existing OptiPath records are updated with new volume, chip thickness, and spindle speed values.

NOTE: If **Update Optimization Settings for Existing Records** is toggled "**On**", and the same tool is loaded multiple times in the NC program, the Tool Library file, produced by **Learn From Toolpath**, will have only one record for the tool. The settings for the tool are based on the "worst case" cut condition of all cuts made with the tool during the current NC program. If toggled "**Off**", a record will be added to the Tool Library file each time the tool is loaded. The settings will be based on the "worst case" cut for the current "load tool" cut sequence.

Reset and Optimize after Learning — If toggled "**On**", VERICUT automatically resets and optimizes the NC program after the updated Tool Library file is created. Otherwise, you will be prompted whether or not you want to optimize the toolpath using the newly created Tool Library file.

Minimum Feedrate Change — Specifies the minimum change from the current optimized feed rate that will cause a new optimized feed rate to be output. This feature controls the quantity of feed rates output to the optimized NC program file. A small value causes more optimized feed rates to be output than when a larger value is entered.

Default — When selected, sets the default Minimum Feedrate Change = 1 IPM or 25 MMPM, whichever applies.

Clean-up Feedrate — Feed rate used when the tool is adjacent to, but not removing material. This condition is commonly referred to as a "spring pass".

Default — When selected, sets the default clean-up feed rate as follows:

G-Code NC program = 50% of the **Max Feed Velocity** of the machine's X-axis.

APT NC program = 50% of the **Fast Feed** value.

Tip: Set the **Clean-up Feedrate** to be less than the **Air Cut Feedrate**, since actual cutting leaves small amounts of material from tool deflection.

Minimum Cut Feedrate — Specifies the minimum optimized feed rates that can be output when removing material.

Default option — When selected, sets the default minimum feed rate = 1 IPM or 25 MMPM, whichever applies.

Maximum Cut Feedrate — Specifies the maximum optimized feed rates that can be output when removing material.

Default — When selected, sets the default maximum feed rate as follows:

G-Code NC program = 45% of the **Max Feed Velocity** of the machine's X-axis.

APT NC program = 45% of the **Fast Feed** value.

Air Cut Feed Rate — This option is useful for reducing time of proven NC programs, without affecting cutting feed rates and resulting surface finishes. When the checkbox is cleared, this feature controls the feed rate used by all other optimization methods to optimize air cuts.

Default — when selected, the air cut feed rate is determined as follows:

G-Code NC program = **Max Feed Velocity** of the machine's X-axis.

APT NC program = **Fast Feed** value.

To specify an air cut feed rate value, clear the **Default** checkbox and enter a value in the data field. The feed rate entered should reflect the maximum speed at which machine axes can move simultaneously in feed mode (e.g. G01).

To OptiPath Control window

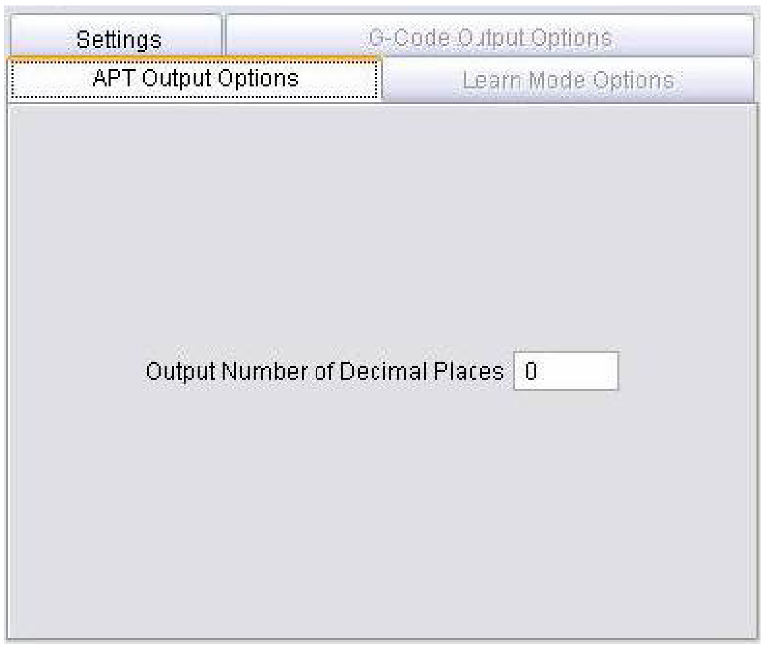
OptiPath Control window, APT Output Options tab

VERICUT Location: **OptiPath menu > Control**

Project menu > Output: Logs tab, Settings button

VERICUT toolbar short cut: 

Features on this tab enable you to configure settings referenced when APT NC programs are optimized.



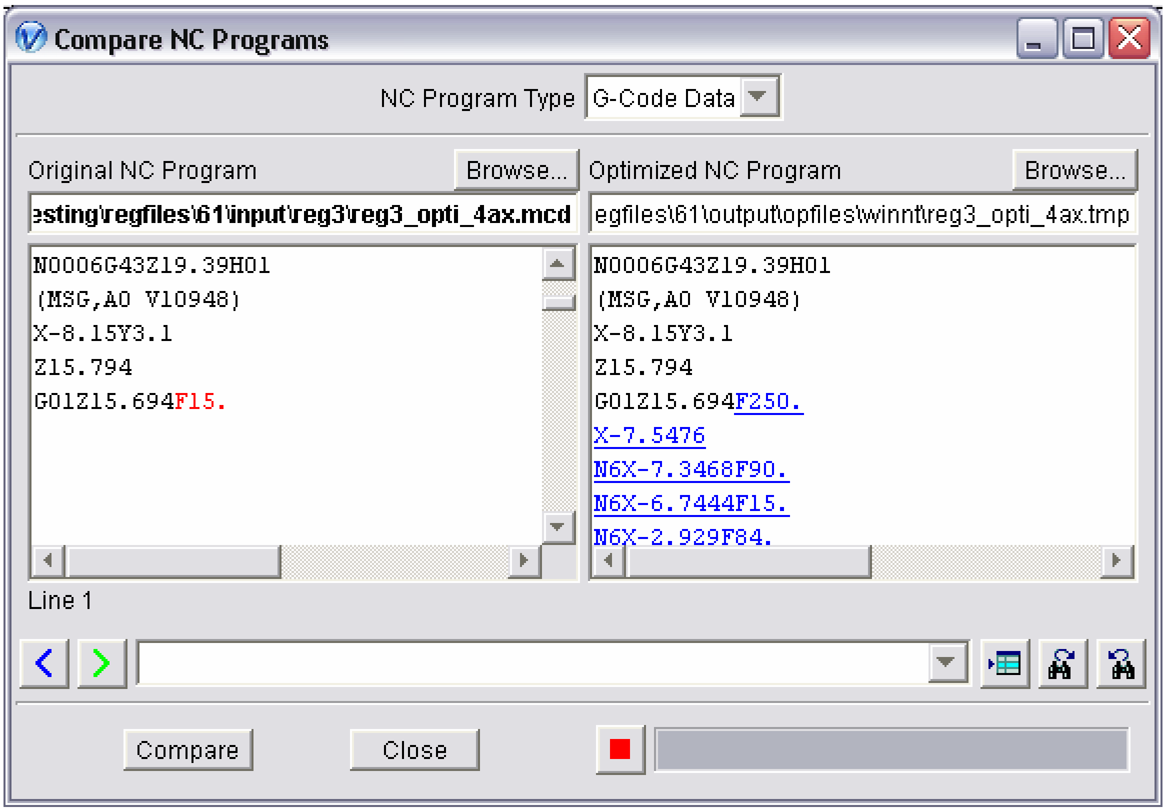
Output Number of Decimal Places — Use the text field to specify the number of decimal places that are to be output when OptiPath adds records. The default value is 10.

To OptiPath Control window

Compare Files (Compare NC Program Files window)

VERICUT Location: **OptiPath menu > Compare Files**

Opens a window to enabling you to select and compare the original NC program file and the optimized NC program file.



Toolpath Type — Select **APT** or **G-Code Data.**

Original Toolpath — Enter the *\path\file name* in the text field of the file to receive exported model data or click on **Browse** and use the Original NC program file selection window to specify the file.

Optimized Toolpath — Enter the *\path\file name* in the text field of the file to receive exported model data or click on Browse and use the Optimized NC program file selection window to specify the file.

Previous Difference — Use the  icon to step back through the file to the previous difference.

Next Difference — Use the icon to step forward through the file to the next difference.

Line Number or Search Text — Use to enter a line number or a string of text to search for.

Goto Line Number — Use the icon to move the cursor in the file listing to the line number specified in the **Line Number** or **Search Text** field.

Search Forward — Use the icon to search forward in the file listing for the text string specified in the **Line Number** or **Search Text** field.

Search Backward — Use the icon to search backward in the file listing for the text string specified in the **Line Number** or **Search Text** field.

Compare — Starts the compare of the two specified NC program files.

Close — Closes the **Compare NC Programs window**.

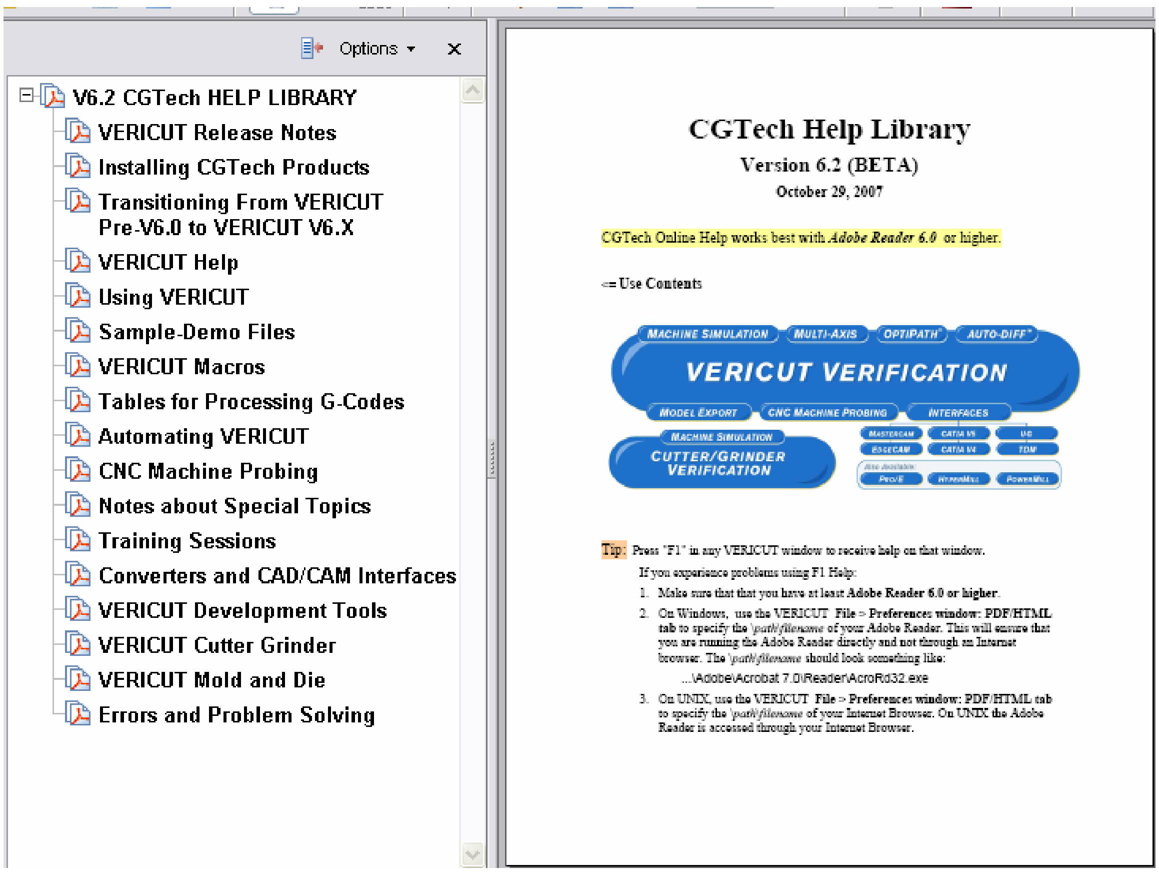
Stop — Use the icon to stop the compare.

Help Menu

Help On VERICUT

VERICUT Location: **Help menu > Help on VERICUT**

Opens the CGTech Help Library.



License

VERICUT Location: **Help menu > License**

Opens a window showing the status of VERICUT licenses on your license server.

License...

License Server Hostname:

Feature	Status	Total Available	Total In Use	Current Session
Verification	expires 12/31/2007	50	9	Yes
Multi-Axis	expires 12/31/2007	50	2	No
OptiPath	expires 12/31/2007	50	0	No
AUTO-DIFF	expires 12/31/2007	50	0	No
Model Export	expires 12/31/2007	50	0	No
Machine Simulation	expires 12/31/2007	50	4	No
Probing	expires 12/31/2007	50	0	No
Inspection	expires 12/31/2007	50	0	No
ACIS Model Interface	expires 12/31/2007	50	0	No
STEP Model Interface	expires 12/31/2007	50	0	No
CATIA V4 Model Interface	expires 12/31/2007	50	0	No
CATIA V5 Model Interface	expires 12/31/2007	50	0	No
VNCK Interface	expires 12/31/2007	50	0	No
TNC Interface	expires 12/31/2007	50	0	No
TDM Interface	expires 12/31/2007	50	0	No
Mold and Die Interface	expires 12/31/2007	50	0	No
Cutter Grinder Interface	expires 12/31/2007	50	0	No
Cutter Grinder/Machine Simulation	expires 12/31/2007	50	0	No
VERICUT Composite Programming	expires 12/31/2007	10	0	No
VERICUT Composite Simulation	expires 12/31/2007	10	0	No
VERICUT Limited Verification	expires 12/31/2007	10	0	No
VERICUT Limited Machine Simulation	expires 12/31/2007	10	0	No
VERICUT Limited OptiPath	expires 12/31/2007	10	0	No
VERICUT Limited Multi-Axis	expires 12/31/2007	10	0	No
VERICUT Limited AUTO-DIFF	expires 12/31/2007	10	0	No
Mulder and Scully Controller	expires 12/31/2007	10	0	
CATV Interface	expires 12/31/2007	50	0	
CATIA V5 Interface	expires 12/31/2007	50	0	
Unigraphics Interface	expires 12/31/2007	50	0	
Mastercam Interface	expires 12/31/2007	50	0	
EdgeCAM Interface	expires 12/31/2007	50	0	
GibbsCAM Interface	expires 12/31/2007	50	0	
Die Sinking	expires 12/31/2007	50	0	Obsolete
Advanced Machine Features	expires 12/31/2007	50	0	Obsolete
Machine Developer's Kit	expires 12/31/2007	50	0	Obsolete
WorkNC Interface	expires 12/31/2007	50	0	Obsolete
Customizer	expires 12/31/2007	50	0	Obsolete

Save Close

Feature — This column shows the VERICUT feature licenses that are available on your license server.

Status — This column shows the expiration dated of the VERICUT feature license.

Total Available — This column shows the total number of VERICUT feature licenses that are available on your license server.

Total in Use — This column shows the total number of VERICUT feature licenses that are currently in use.

Current Session — This column shows the total number of VERICUT feature licenses that you are using in your session.

Help about VERICUT

VERICUT Location: **Help menu > Help about VERICUT**

Opens a window showing version related information.