

CATIA Interface (CATV[®])

On-line Help

Version 5.0 (October 18, 2000)

CGTech on-line Help works best with Internet *Explorer 5.0* or higher, or *Netscape 4.75* or higher.

CATV is a software program that provides a fully integrated CATIA to VERICUT interface. It is a Graphics Interactive Interface (GII) function that uses the CATIA Application Architecture (CAA) to give the user interaction the look and feel of CATIA. CATV with VERICUT provides immediate simulation and verification of CATIA generated geometry and tool path data; in short- taking the guesswork out of NC programming with CATIA.

<= Use Contents/Index/Search tabs, or click on a topic below:



What's **NEW** in VERICUT 5.0

Introduction to CATV

CATV Functions Reference

CGTech On-line Help Library

Tip: Once in VERICUT, press "F1" in any VERICUT window to receive help on that window.

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Chapter 1 Introduction

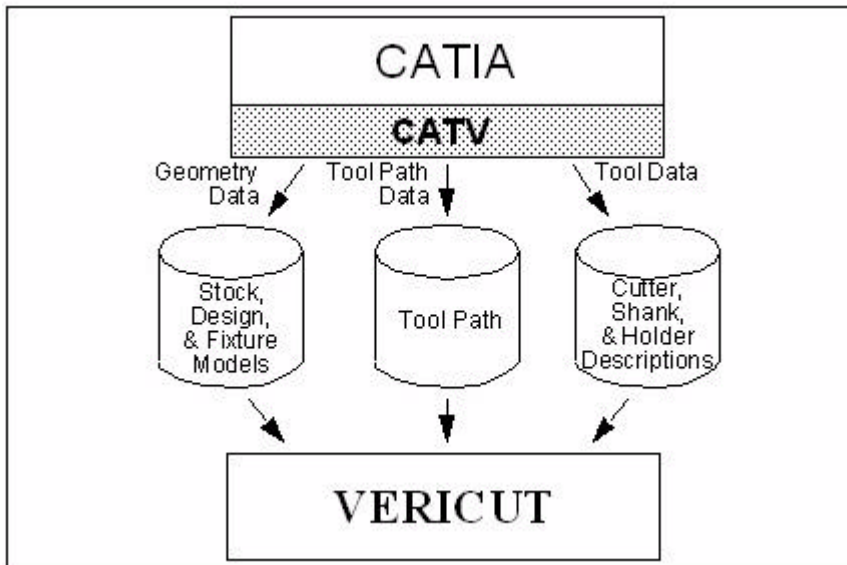
Introduction to CATV

CATV is a software program that provides a fully integrated CATIA to VERICUT interface. It is a Graphics Interactive Interface (GII) function that uses the CATIA Application Architecture (CAA) to give the user interaction the look and feel of CATIA.

CATV menus provide the means of transferring CATIA geometry, tools, tool paths, and display view data to VERICUT. With CATV, VERICUT is run directly from CATIA either interactively or in batch mode, using the transferred data to provide immediate verification of CATIA generated geometry and tool path data. While VERICUT is running, the CATIA window can still be accessed.

VERICUT is a software program by CGTech that has the ability to display and analyze geometric models as well as simulate cutter motion and the material removal of an NC tool path. It also verifies the quality and correctness of the tool path. Programming errors or inefficient motion are visually detected during the simulation process, which aids the NC programmer in correcting the tool path. Measurement tools are used at any time during the simulation process to verify or compute the volume, distances and locations, as well as to compare various stages of the "cut" part with other stages or with the "design" part.

Flow of information through CATV:



Starting CATIA with CATV

CATIA with CATV can be started using the "catv" command file, for example in an operating system window type "catv". If the command is not found, include the full path, for example: "/usr/cgtech5x/computertype/commands/catv" where "/usr/cgtech5x" is where CGTech products are installed and "computertype" is a computer type capable of running CATIA and CATV.

The "catv" command files contains the UNIX export commands required to allow VERICUT to be operated from CATIA. Site specific CATIA execution environments should be modified to incorporate CATV by adding these commands to the CATIA execution environment. When CATIA is executed via a command other than the "catv" command file created during installation, then CATV may not be available

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during that CATIA session.

After executing CATIA with CATV, select **COLD ST** and the CATIA main menu appears with the “**CATV**” function included in the function list (right column). Use the left mouse button to select **CATV** from the function list and the CATV menus are displayed.



About CGTech documentation

CGTech provides documentation in the following forms:

HTML books

PDF books

All books are available in HTML and PDF formats on the CGTech CD-ROM. You can view HTML books with your Internet browser. To view and print PDF books, you must have the Acrobat Reader installed.

Note: Printed documentation is available from CGTech at extra cost. Please contact your CGTech representative for more information.

CGTech welcomes your suggestions and comments on its documentation. You can send feedback in the following ways:

Send comments electronically to support@cgtech.com, or via the CGTech web site at www.cgtech.com.

Fax comments to CGTech, attention: Documentation Department at (949) 753-1053.

Conventions

CGTech documentation uses the conventions described in this section.

Fonts

The table below describes text fonts used to help orient the user to the CGTech product user interfaces, and actions.

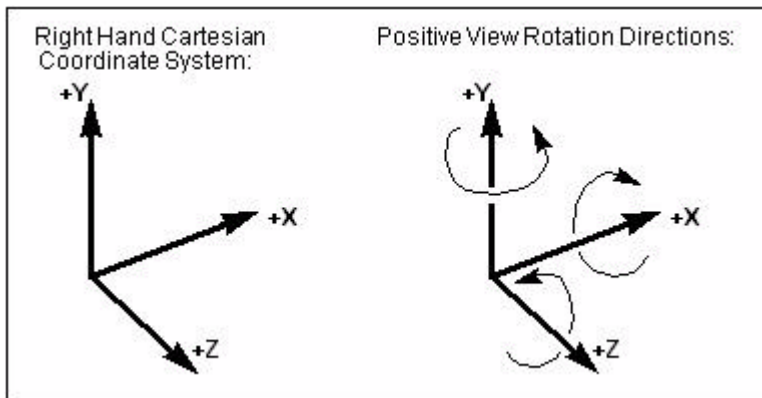
Font type	Where used	Examples
Bold	Menus paths, dialog box options, buttons, and other user interface elements.	Choose File menu > Save . Choose Model Type=Fixture .
<i>Italics</i>	Names of other CGTech documents, system messages, file names.	For more information, consult the <i>VERICUT Installation</i> documentation. When done, you will see a message similar to <i>Processing complete</i> .
Less-than and greater-than symbols (< >)	Variables for which the user substitutes an appropriate value.	Enter File=<yourname.usr> PPRINT/VERICUT-CUTCOLOR <n>

Coordinate systems and angles

CGTech products use a right-hand Cartesian coordinate system to describe location and orientation.

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Rotation angle values are specified in decimal degrees. A positive angle goes counter-clockwise about an axis when viewed from the positive end of the axis, looking towards the origin. A negative angle specifies clockwise rotation as shown in the next figure.



Chapter 2 **CATV installation**

Installing CATV

Follow the general procedure below to install CATV and VERICUT for use with CATIA.

1. Verify system requirements for CATIA and CATV/VERICUT.
2. Install CATIA per the instructions provided with the CATIA software.
3. After CATIA has been installed, install VERICUT with CATV per the instructions in the CD booklet and the VERICUT Installation documentation.

IMPORTANT! *CATIA requires that the directory path to CGTech product licensing be a maximum of 24 characters.* Choose an installation directory for CGTech products that does not violate this requirement, or CATV may not operate correctly. For example, installing VERICUT on an RS computer in the "/usr/cgtech" directory provides a 22 character licensing directory path: "/usr/cgtech/rs/license". The same restriction applies to CATIA sample files. Using the recommended installation path "/cgtech", the sample files would be stored in "/usr/cgtech/samples/catv" (24 characters).

Note: CATV should be incorporated into the user's normal CATIA environment. One way to do this is to append the Declarations file ".../catv/dcls/CATIA.dcls" to the "USRENV.dcls" file in each user's local directory.

After VERICUT and CATV have been installed, the CATV function should be available when CATIA is run via the "catv" command file located in the "commands" directory. However, the CATV function can not be accessed in CATIA until it has been added to the CATIA keyboard.

To add CATV to the CATIA keyboard:

1. From the CATIA **FILE menu**, select the **KEYBOARD** function.
2. Choose **CATV** from the list of available functions.
3. Choose a location on the CATIA keyboard where the **CATV** function is to reside.

Once configured in this manner, CATV should remain accessible for subsequent CATIA sessions.

System requirements

This release of CATV can be operated on the UNIX computers with the CATIA systems listed below. All systems require a CD-ROM drive, mouse, graphics device supporting 256 or more colors, and ethernet network interface card. Token Ring network interface card is supported.

Supported computer types and CATIA requirements:

HP

CATIA 4.1.4-4.1.5 (no patches required) or

CATIA 4.1.6 or higher with "Best Available" PTF or latest "Refresh" package

IBM

CATIA 4.1.2 with PTF UB14570, UB14571 or

CATIA 4.1.3 with PTF UB14543 or UB16141 or

CATIA 4.1.4-4.1.5 (no patches required) or

CATIA 4.1.6 or higher with "Best Available" PTF or latest "Refresh" package

SGI

CATIA 4.1.8 or higher with latest "Refresh" package

Solaris

CATIA 4.1.8 or higher with latest "Refresh" package

Note: For information about VERICUT system requirements, see the *VERICUT Installation* documentation.

CATIA load modules

The following programs or load modules are required to support the CATV product within CATIA, and are listed in the suggested order of installation:

1. BASE
2. 3D_DESIGN
3. Exact Solids (CATIA 4.1.2 only)

Declaring directories CATV can access

CATV must be able to write files during export and transfer operations. The “CATIA.dcls” Declarations file located in the “catv/dec” directory contains the declarations for directories and files accessed by CATV. By default, the user's home and present working directories are declared for CATV access.

Sample “CATIA.dcls” Declarations file:

```
/* Model files for CATV */
catia.MODEL = $CATVSAMP,CATV_SAMPLES;
/* APT source files */
catia.APTSOURCE = $CATVSAMP,CATV_SAMPLES;
/* CATIA CLF files */
catia.CATCLFILE = $CATVSAMP,CATV_SAMPLES;
alias VCNFILES = catia.VCNFILES : SET OF STRING;
/* DO NOT MODIFY VCN00000; reserved for CATV */
alias VCN00000 = catia.VCNFILES = $VCCOMMANDS;
/* Output files for CATV */
alias VCN00001 = catia.VCNFILES = $PWD;
alias VCN00002 = catia.VCNFILES = $HOME;
/* DO NOT MODIFY BELOW THIS LINE; reserved for CATV */
/* ERROR MESSAGE FILE FOR CATV */
catia.MESSAGE = $CATMSG;
/* NLS (National Language Support) */
catia.NLS = TRUE;
```

Edit the Declarations file by adding entries after VCN00002 to enable CATV to write to additional directories. An example is shown below.

Sample edited “CATIA.dcls” Declarations file:

```
/* Model files for CATV */
catia.MODEL = $CATVSAMP,CATV_SAMPLES;
/* APT source files */
catia.APTSOURCE = $CATVSAMP,CATV_SAMPLES;
/* CATIA CLF files */
catia.CATCLFILE = $CATVSAMP,CATV_SAMPLES;
alias VCNFILES = catia.VCNFILES : SET OF STRING;
/* DO NOT MODIFY VCN00000; reserved for CATV */
alias VCN00000 = catia.VCNFILES = $VCCOMMANDS;
/* Output files for CATV */
alias VCN00001 = catia.VCNFILES = $PWD;
alias VCN00002 = catia.VCNFILES = $HOME;
alias VCN00003 = catia.VCNFILES = `~/users/user2/CATIA`;
/* DO NOT MODIFY BELOW THIS LINE; reserved for CATV */
/* ERROR MESSAGE FILE FOR CATV */
catia.MESSAGE = $CATMSG;
/* NLS (National Language Support) */
catia.NLS = TRUE;
```

Translating CATV text

The VCNCATIN.dcls Declarations file located in the “catv/dec” directory contains declarations for text appearing in CATIA resulting from CATV usage. This file provides NLS (National Language Support) capability for CATV. By default, the VCNCATIN.dcls file provides English language text. However, the file can be edited similar to editing CATIA NLS files to change the characters displayed in CATIA when CATV is used.

Configuring to transfer non-CATIA binary tool paths

By default, CATV will process and transfer CATIA binary tool paths via the **CAT CLF** function. Environment variables in the “catv” command file control the type of binary tool path file that can be processed, as well as several other aspects of binary tool path conversion. The command files are located in the “computertype/commands” where “computertype” represents the computer type on which CATV is to be run (e.g. “rs/commands”).

CATV environment variables

The environment variables listed below provide control over transferring binary tool paths with CATV.

BINCL2V_CLTYPE={1 | 2 | 3 | 4}—Sets the binary tool path file type for interpreting binary CL-file data, where “n” specifies the following tool path file types:

- 1= IBM Mainframe APT
- 2= Workstation IBM APT
- 3= Workstation CATIA APT (default)
- 4= VAX APT

When the default binary tool path type is specified (BINCL2V_CLTYPE=3), the remaining binary conversion environment variables are ignored. When BINCL2V_CLTYPE=1, 2 or 4, the variables below are applied.

BINCL2V_TABLE=<tablefile>—Specifies the APT Table file used to interpret the binary CL data. The full directory path can be included with the file name. (Default: BINCL2V_TABLE=\$CGTECH_PRODUCTS/bincl2v/aptwords.tbl where the CGTECH_PRODUCTS environment variable has been previously set to specify where CGTech products have been installed)

BINCL2V_CIRCLES={YES | NO}—Controls if circular motions are to be represented by CIRCLE records or chordal GOTO records. YES (default) uses CIRCLE records. No uses chordal GOTO records.

BINCL2V_HEIGHT=<minimumcutterheight>—Specifies a minimum cutter height to be used for cutter descriptions having height less than this value. (Default=0)

For additional information about the binary tool path file conversion process, consult the *Binary CL-file Converter* documentation.

Chapter 3 Using CATV

Overview of using CATV

The following is an overview of how CATV is typically used to simulate the cutting action of CATIA-generated tool paths.

1. **Prepare CATIA data**—Create the necessary CATIA geometry and tool paths to be transferred to VERICUT.
2. **Select CATIA geometry to use in VERICUT**—Use **GEOMETRY menu** functions to select the CATIA geometry to use in VERICUT as stock, fixture, and design models. (See how) Solids, volumes, faces, skins, and surfaces are selectable for these models. You can also select 2D lines and arcs that represent cutting tool profiles, including non-cutting shanks and holders so VERICUT can check for interference. (See how)
3. **Select a tool path to simulate**—Use **TOOLPATH menu** functions to select one or more CATIA binary CL-files or ASCII APT source tool path files for VERICUT to simulate. (See how)
4. **Process (export) the CATIA data and run VERICUT**—Use **PROCESS menu** functions to process (export) selected CATIA data. (See how) After processing the CATIA data, run VERICUT interactively or in batch mode for immediate simulation and verification of your CATIA CAM process. (See how) All that's left to do is press the **Play** icon in VERICUT (green arrow pointing right)!

When VERICUT is run from CATIA, the view orientation is the same as it was in CATIA. Both application windows are available for you to work in at the same time.

What do you want to do?

Learn about working with part geometry

Learn about working with tool geometry

Learn about selecting CATIA tool paths for VERICUT to simulate

Learn about exporting or transferring CATIA data to VERICUT

Learn about running VERICUT from CATIA

Part geometry

Working with part geometry

Part geometry includes models of the stock workpiece, holding fixture, and design model of the finished part. Using the **PART** function in the **GEOMETRY** menu, you can select part geometry for export as Stereolithography rapid-prototyping models, or transfer to VERICUT. (See how) **PART** can also be used to import VERICUT cut parts back into CATIA. (More about importing VERICUT models into CATIA)

CATIA geometry selectable for part models:

solids

volumes

surfaces

faces

skins

entity labels corresponding to any of the above

Notes:

Solids: CATIA Exact solids (SOLIDE, or “*SOLE”) are recommended over Mockup solids (SOLIDM, or “*SOLM”) because they are much more reliable. You should convert Mockup solids to Exact solids before exporting or transferring to VERICUT.

Surfaces, faces and skins: To use models composed of surfaces, faces and skins as solid models, ensure that the selected elements form a watertight skin. If gaps or overlaps exist in the selected part geometry, the resulting solid may have holes, or portions (possible all) of the model may fail.

What do you want to do?

See how to select part geometry for export or transfer to VERICUT

Export or transfer selected part geometry to VERICUT

Import a VERICUT model into CATIA

Selecting part geometry

Part geometry can be selected in any order. Selected geometry is highlighted by CATIA. Selections for each type of geometry can be reviewed by re-selecting the corresponding geometry type in the **PART** window. Geometry is unselected by selecting it again. Unselected geometry returns to the original CATIA color.

Note: It is not recommended to use the CATIA “MULTI-SELECT” feature because hidden (invisible) geometry may also be selected.

To select CATIA geometry to represent the stock workpiece, fixture, or theoretical designed part:

1. Select **GEOMETRY** menu > **PART**.
2. Choose **STOCK**, then select the CATIA stock model geometry. (Must be solid or composed of watertight surfaces/faces.)

You can select solids, faces, skins, surfaces, volumes, or entity labels corresponding to any of these geometry types as part geometry.

3. (Optional) Choose **FIXTURE**, then select the fixture model geometry. (Must be solid or composed of watertight surfaces/faces.)
4. (Optional) Choose **DESIGN**, then select the design model geometry. (Does not have to be solid or composed of watertight surfaces/faces.)

After selecting part geometry, use the CATV **TRANSFER** function to transfer selected data to VERICUT, or export the data to external files.

Importing VERICUT models into CATIA

VERICUT FastMill or machine simulation turned part models can be imported into CATIA using the CATV "**IMPORT**" option in the **GEOMETRY menu > PART** function. Prior to importing, the VERICUT model must first be exported from VERICUT. The imported VERICUT model can be a CATIA exact solid, or net surface.

Note: If "*Vector length too small*" is reported during an import operation, try reducing the CATIA "Intersection Projection" geometric standard value (e.g. .001 -> .0000001), then retry.

To import a VERICUT FastMill model:

1. Ensure the VERICUT FastMill cut model has been exported from VERICUT, for example: **File menu > Export > FastMill Model, File Type=Surface.**
2. Select **GEOMETRY menu > PART.**
3. Choose **IMPORT.**
4. Enter the file name of the VERICUT FastMill model to import.
5. Choose the appropriate **Creation Type (Surface- NET surface, Solid- exact solid), then Apply.**

To import a Machine Simulation turned model:

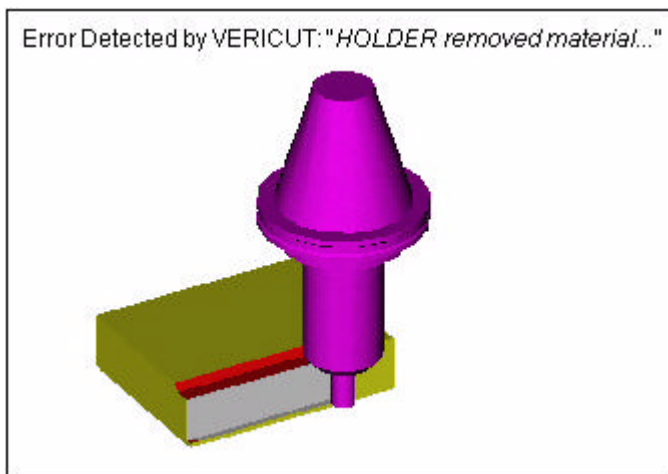
1. Ensure the Machine Simulation turned model has been exported from VERICUT Machine Simulation, for example: **File menu > Export Model, File Type=IGES.**
2. Select **GEOMETRY menu > PART.**
3. Choose **IMPORT.**
4. Enter the file name of the Machine Simulation turned model to import.
5. Choose **Creation Type=Solid, then Apply.**

Tool geometry

Working with tool geometry

Tool geometry can describe the tool axis, profiles representing standard CATIA cutter shapes, odd cutter shapes such as form tools, and non-cutting portions of the tool assembly such as shanks, extensions, holders, collets, chucks, etc. Using the **TOOL** function in the **GEOMETRY** menu, you can select tool geometry to transfer complete tool assemblies to VERICUT. (See how)

Non-cutting portions of the tool that remove material in VERICUT, such as shanks, extensions, holders, collets, chucks, etc., cause errors and the resulting cut surfaces are shaded using the red "error" color. (See below)



CATIA geometry selectable for tool assemblies:

2-D lines

2-D arcs

Notes:

Composite curves are NOT selectable as tool geometry, even those composed of lines and circles.

Tool geometry lines and arcs must be defined in in "SPACE" mode (not "2D-DRAW" mode), and drawn in a plane parallel to either the XY, ZX, or YZ plane of the current axis system (e.g. AXS1). When multiple axis systems are defined, verify the current axis system contains the tool geometry to select.

VERICUT must be configured to access the tools in the Tool Library. See the *VERICUT* documentation for more information.

What do you want to do?

See how to select tool geometry for transfer to VERICUT

Export or transfer selected part geometry to VERICUT

Selecting tool geometry

In general, selection of tool geometry begins by choosing a line to represent the tool **AXIS** (centerline), followed by selecting the profiles of up to three components in a tool assembly: **CUTTER**, **SHANK**, and **HOLDER**.

To select CATIA geometry to represent a tool assembly (cutter, shank, holder):

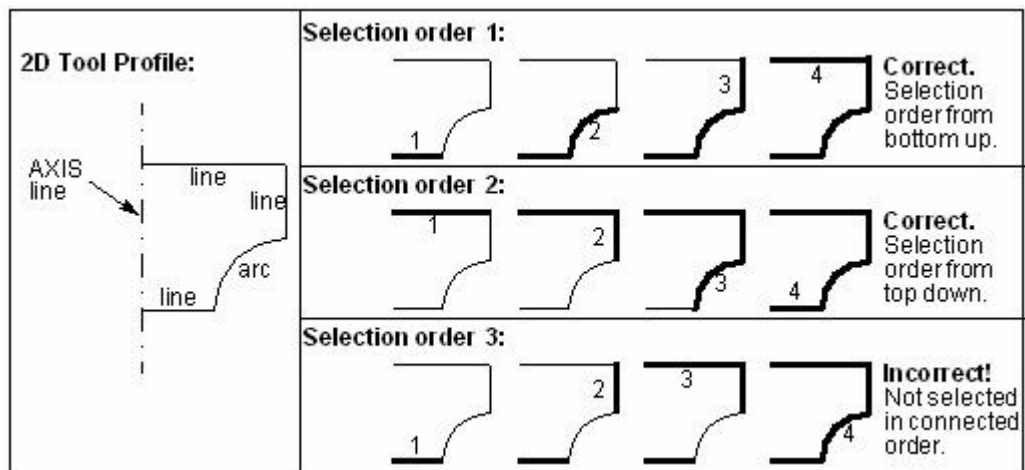
1. Verify the current axis system contains the tool geometry to select.
2. Select **GEOMETRY menu > TOOL**.
3. Choose **AXIS**, then select a line to represent the tool axis.

The **AXIS** must be selected before **CUTTER**, **SHANK** or **HOLDER** geometry. When selecting geometry in the **XY** plane, the axis line must be parallel to the **Y**-axis. When tool geometry is in the **ZX** or **YZ** planes, the axis line must be parallel to the **Z**-axis.

4. Choose **CUTTER**, then select 2-D geometry to represent the cutting portion of the tool. (Must select geometry in connected order, beginning at the tool axis. If you make a mistake, clear your selections and try again.)

You can select lines and circles as tool geometry. CATIA composite curves are **NOT** selectable for this type of geometry.

Selection of each profile begins at the tool axis. Select lines and arcs in connected order. (See below) Selection can be from the bottom of the profile towards the top, or from the top downwards. If a line or arc is selected out of order, unselect all and start over.



5. (Optional) Choose **SHANK**, then select the shank profile geometry.
6. (Optional) Choose **HOLDER**, then select the holder profile geometry.

After selecting tool geometry, use the CATV **TRANSFER** function to transfer selected data to VERICUT, or export the data to external files.

Tool paths

Selecting tool paths

Using the functions in the **TOOLPATH menu**, you can select one or more CATIA tool paths to transfer to VERICUT for simulation. Multiple tool paths are concatenated during transfer in the order of selection into one tool path file.

To select CATIA tool paths for VERICUT to simulate:

1. Ensure tool paths to be transferred have been exported via the CATUTIL “CATNC” utility.
2. Select **TOOLPATH menu** > **APT CLF** or **CAT CLF**, depending on which tool path type you want to simulate: APT source or CATIA binary APT, respectively.
3. In the **TOOL PATH** window, select directory where the desired tool path resides.

If the desired directory can not be accessed by CATV, consult the section titled "Declaring directories CATV can access".
4. In the **TOOL PATH FILES** window, select the tool path(s) to simulate.

After selecting the tool path(s), use the CATV **TRANSFER** function to transfer selected data to VERICUT, or export the data to external files.

Exporting or transferring CATIA data to VERICUT

About exporting or transferring CATIA data to VERICUT

CATIA data selected using other CATV functions can be exported/transferred to VERICUT using the CATV **TRANSFER** function in the **PROCESS** menu. (See how) There is no difference between exporting or transferring- in either case, the CATIA data is processed as a "transfer" action and exported to a series of files external to CATIA (see below). When VERICUT is run from CATIA (see how) after transferring selected data, all transferred data is available in the VERICUT session.

Files created

During a transfer, files patterned after the project **FILE NAME** are created in the directory specified by the **OUTPATH** function. For example, assuming the project name is "PROJ", the table below show the files that could be created, depending on what was selected for transfer.

CATIA data selected	File name (file type)
Stock part geometry	PROJ.stk (VERICUT model file) or PROJ_stk.stl (STL model file)
Fixture part geometry	PROJ.fxt (VERICUT model file) or PROJ_fxt.stl (STL model file)
Design part geometry	PROJ.dsn (VERICUT model file) or PROJ_dsn.stl (STL model file)
Tool geometry	PROJ.tls (Tool Library file)
APT source tool path (.aptsource)	PROJ.tp (ASCII APT toolpath file)
CATIA binary CL-file tool path (.clfile)	PROJ.tp (converted ASCII APT toolpath file)
	PROJ.usr (Add-user file referencing above files)

Subsequent transfer operations with the same file name causes project files to be overwritten. An exception is made for transferring tool geometry, where the new tools are added to the existing VERICUT Tool Library file (see below).

Tool geometry

When selected tool geometry is transferred to VERICUT, a Tool Library file is created containing the tool assemblies. CATV automatically assigns tool IDs (pocket numbers) in sequential order of transfer. For example, the first tool assembly transferred receives Tool Library ID "1". The next tool transferred receives ID "2", and so on.

VERICUT must be configured to access the tools in the Tool Library. See the *VERICUT* documentation for more information.

Tool paths

Binary tool paths (CATIA CL-files) are converted into ASCII APT equivalent during the transfer process. Multiple tool paths are concatenated during transfer in the order of selection into one tool path file for simulation by VERICUT.

Help on CATIA Interface (CATV)

What do you want to do?

See how to export or transfer CATIA data

Set the outpath for CATV-generated files

Select part geometry or tool geometry

Select tool paths

Exporting or transferring selected CATIA data to VERICUT

Follow the procedure below to export or transfer selected CATIA data to VERICUT. There is no difference between exporting or transferring- in either case, the CATIA data is processed as a "transfer" action and exported to a series of files external to CATIA. (See how files are named)

To export or transfer selected CATIA data to VERICUT:

1. Use **GEOMETRY menu** and **TOOLPATH menu** functions to select CATIA geometry and tool path(s) to transfer/export.
2. Select **PROCESS menu > TRANSFER**.
3. If a new project **FILE NAME** is desired, edit this field and press <Enter>.

Actions applicable to transferring part geometry:

4. Set the **MAXIMUM SAG** deviation tolerance to triangulate part geometry, press <Enter>.
5. Choose the desired **FILE TYPE** for part geometry model files.
6. Choose **FILE FORMAT** for part geometry model files. **BINARY** is recommended for fastest processing, and smallest file sizes.
7. When **FILE TYPE=VERICUT**, choose the **NORMAL DIRECTION** which describes the CATIA defined surface normal vector directions for the models.
8. Choose the **AXES SYSTEM** in which to transfer geometry. **REFERENCE** is recommended for VERICUT to automatically align transferred CATIA tool paths to models.

Actions applicable to all transfer processes:

9. Choose a **TRANSFER** option that specifies the desired data to transfer.
10. Verify the **VERICUT OUTPATH** directory where files will be stored is correct. To change, use the **OUTPATH** function.
11. Verify the **CLFILE INPUT PATH** and **TOOLPATH FILE** shown in window below the **TRANSFER** window are correct. To change, use functions in the **TOOLPATH menu**.
12. Select "Yes" to approve, and begin the transfer process. CATIA notifies you when the transfer process is finished with a message similar to "**TRANSFER COMPLETE**".

After transferring CATIA data, use the **VERICUT** or **BATCH** functions to run VERICUT interactively or in batch mode.

Setting the outpath for CATV-generated files

During a transfer operation, files are generated by CATV containing the selected CATIA part, tool, and/or tool path data. Using the **OUTPATH** function in the **PROCESS menu**, you can set the directory to receive these files. You must have appropriate permissions to write files in the directory designated as the outpath, or CATV will not be able to process selected CATIA data.

To set the outpath for CATV-generated files:

1. Select **PROCESS menu > OUTPATH**.
2. In the **VERICUT DIRECTORIES** window, select the directory to receive files generated by CATV.

As installed with no alterations, CATV only shows the present working and home directories. To add directories to this list, consult the section titled "Declaring directories CATV can access".

Running VERICUT

Running VERICUT from CATIA

CATV menus provide the means of transferring CATIA geometry, tools, tool paths, and display view data to VERICUT. With CATV, VERICUT is run directly from CATIA either interactively or in batch mode, using the transferred data to provide immediate verification of CATIA generated geometry and tool path data.

Running interactively: opens the VERICUT window to provide access to all VERICUT functions and capabilities. While VERICUT is running, the CATIA window can still be accessed.

Running in batch mode: performs an automated simulation in the background without immediately opening the VERICUT window. When the simulation is complete, the VERICUT window “pops open” the VERICUT window when done. Batch allows the user to continue working in CATIA while VERICUT processes the tool path.

To run VERICUT from CATIA:

1. Ensure all desired CATIA data has been transferred via the **TRANSFER** function.
2. Click the **PROCESS** menu.
3. To run VERICUT interactively, click **VERICUT**.

The VERICUT window opens. Press the **Play** icon in VERICUT (green arrow pointing right) to start the simulation.

4. To run a VERICUT batch session, click **BATCH**.

The VERICUT window will not open until processing has been completed.

Chapter 4 **CATV Functions Reference**

CATV functions, listed by menu

CATV functions are listed below by menu, as they appear in the CATV user interface. Refer to each function for details.

GEOMETRY menu functions

PART

TOOL

TOOLPATH menu functions

APT CLF

CAT CLF

PROCESS menu functions

TRANSFER

OUTPATH

VERICUT

BATCH

GEOMETRY menu functions

PART

Location: **GEOMETRY menu > PART**

The **PART** function is used to select CATIA part geometry for export as Stereolithography rapid-prototyping models, or to transfer to VERICUT. Part geometry includes the stock workpiece, holding fixture, and design model of the finished part. **PART** can also import VERICUT cut models into CATIA.

STOCK—select geometry representing the raw stock workpiece to be machined. (See how)

FIXTURE—select geometry representing the fixture used to hold the stock for machining. (See how)

DESIGN—select geometry representing the design model, or theoretical finished part. Design models transferred to VERICUT are referenced by the **AUTO-DIFF** model comparison function. (See how)

IMPORT—import a VERICUT simulated cut part back into CATIA. (See how)

VERICUT cut model must have been exported from VERICUT prior to importing into CATIA. Also, ensure CATIA has adequate table space to receive imported model data, otherwise an error similar to “*TABLES FULL*” may be issued by CATIA.

TOOL

Location: **GEOMETRY menu > TOOL**

The **TOOL** function is used to select CATIA 2-D profile geometry for transfer to VERICUT as cutting tool assemblies. Tool geometry can describe the tool axis, profiles representing standard CATIA cutter shapes, odd cutter shapes such as form tools, and non-cutting portions of the tool assembly such as shanks, extensions, holders, collets, chucks, etc.

AXIS—select a line representing the tool axis. (See how)

CUTTER—select 2-D geometry representing the profile of the cutter (cutting portion of the tool). (See how)

SHANK—select 2-D geometry representing the profile of the non-cutting shank. (See how)

HOLDER—select 2-D geometry representing the profile of the tool holder; also considered to be non-cutting. (See how)

TOOLPATH menu functions

APT CLF

Location: **TOOLPATH menu > APT CLF**

The **APT CLF** function is used to select one or more ASCII APT SOURCE tool paths (".aptsource" extension) to be simulated by VERICUT. (See how) Multiple tool paths are concatenated during transfer in the order of selection into one tool path file.

Note: Tool paths must first be processed from the CATIA model using the CATUTIL "CATNC" utility.

CAT CLF

Location: **TOOLPATH menu > CAT CLF**

The **CAT CLF** function is used to select one or more binary CATIA CLFILE tool paths (“.clfile” extension) to be simulated by VERICUT. (See how) Multiple tool paths are concatenated during transfer in the order of selection into one tool path file. Binary tool paths are converted into ASCII APT equivalent during the transfer process.

Note: Tool paths must first be processed from the CATIA model using the CATUTIL “CATNC” utility.

PROCESS menu functions

TRANSFER

Location: **PROCESS menu > TRANSFER**

The **TRANSFER** function exports or transfers selected CATIA data to VERICUT. (See how) Tolerances, file formats used during part geometry transfer, and the base filename for files created during the transfer process are controlled by this function. Be sure to press <Enter> after editing any of the data fields.

FILE NAME—specifies the base filename of files created during the transfer process. (See an example)

MAXIMUM SAG—specifies the chordal deviation tolerance for outputting part geometry. The default value corresponds to the CATIA “INDENTICAL CURVES” model standards tolerance. Small values result in more accurate, but larger geometry files that require more system resources.

MAXIMUM DISTANCE—specifies the maximum distance in which part geometry selected for transfer will be processed. The default value corresponds to the CATIA “INFINITY” model standards tolerance. This value rarely affects the quality of transferred models and does not typically require modification. If modified, it is recommended to specify a value that is as large as possible, e.g. 99999999.

FILE TYPE—specifies the type (**VERICUT** or **STL**) of part geometry model files output from a transfer process. Descriptions of both file types can be found in the *VERICUT* documentation.

Note: Binary VERICUT Model files are platform specific, and are usable only on the type of computer on which they were created.

FILE FORMAT—specifies the format (**ASCII** or **BINARY**) in which part geometry model files are output. Binary files are recommended because they process faster and more efficiently than an ASCII equivalent.

NORMAL DIRECTION—identifies the CATIA-defined surface normal direction relative to the surface of part geometry. Options are **INWARD**, **OUTWARD**, or **INCONSISTENT**. This feature is applicable only to part geometry transferred via **FILE TYPE=VERICUT** (see above).

Note that models with inconsistent normals require significantly more processing time and computer resources than those with consistent inward or outward-pointing normals. For best performance, it is recommended to orient inconsistent normals in CATIA to be outward pointing.

AXES SYSTEM—specifies the coordinate system, or “AXS” system, in which part geometry is transferred.

REFERENCE—transfers part geometry with respect to the CATIA reference coordinate system “AXS1”. When used, CATV automatically sets VERICUT **Process Matrix=Yes** to align the CATIA tool paths and the models.

CURRENT—transfers part geometry with respect to the current work coordinate system. When used, CATIA models and tool paths are not automatically aligned, and may require orientation in VERICUT.

TRANSFER—controls what selected CATIA data is transferred. You can transfer all selected

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data at once, or any single data type.

VERICUT OUTPATH—uneditable data field showing the directory in which files created during the transfer process will be saved. To change this directory, use the **OUTPATH** function.

CLFILE INPUT PATH—uneditable data field showing the directory in which selected tool path files reside. If the directory shown is not as expected, use the **APT CLF** or **CAT CLF** function to reselect tool paths from the desired directory.

OUTPATH

Location: **PROCESS menu > OUTPATH**

The **OUTPATH** function is used to set the directory where VERICUT files generated by CATV will be stored. (See how) As installed with no alterations, CATV only shows the present working and home directories. To add directories to this list, edit the CATIA.dcls file as described in the section titled "Declaring directories CATV can access".

VERICUT

Location: **PROCESS menu > VERICUT**

The **VERICUT** function runs VERICUT from CATIA for interactive use. (See how) Assuming you previously transferred the necessary CATIA data (part/tool geometry, tool paths, etc.), VERICUT is will be preconfigured using the transferred data to provide immediate verification of CATIA generated geometry and tool paths. All that is left to do is press the **Play** icon in VERICUT (green arrow pointing right) to start the simulation!

Although CATIA transfers control to VERICUT, however, CATIA can still be accessed by iconifying or minimizing the VERICUT window. When VERICUT is exited, control is returned to CATIA.

For a complete description of VERICUT capabilities and options, consult the *VERICUT* documentation.

Hint: Once in VERICUT, press "F1" in any VERICUT window to receive help on that window.

BATCH

Location: **PROCESS menu > BATCH**

The **BATCH** function runs VERICUT in batch mode. (See how) VERICUT automatically simulates the tool path in the background without user interaction, then “pops open” the VERICUT window when done. Batch allows the user to continue working in CATIA while VERICUT processes the tool path.

The **BATCH** function of CATV uses the VERICUT “batchp” command line option for this method of batch processing. For a more detailed description, consult the "Command Line Options" section of the *VERICUT* documentation.

For a complete description of VERICUT capabilities and options, consult the *VERICUT* documentation.

Chapter 5 Training Sessions

Session 1- CATV Sample Session

This session demonstrates how to execute CATIA with CATV and interact with CATV via standard CATIA user actions. Follow the steps in this sample session to transfer CATIA data and use VERICUT to simulate the cutting action of a CATIA tool path.

This session assumes CATV sample files are installed and their location is known to the user. If you do not know if these files have been installed or where they reside, contact your system administrator prior. For reference purposes, the CATV sample files location is depicted as “samples/catv”.

Licenses you will need:

CATV

Part Verification

Multi-axis (if CATVSMP5.aptsource or CATVSMP5.clfile are simulated)

AUTO-DIFF (if the AUTO-DIFF operation is performed)

Session Steps:

1. Run CATIA with CATV

For example, in an operating system window type “catv”. After a few seconds, the CATIA window appears.

2. Select COLD ST (cold start)

3. Open the “CATVSAMP” sample CATIA model supplied by CGTech

By default “**FILE**” is selected from the CATIA menu.

- Press <Enter> to expose the list of available CATIA model directories.

- Select the “CATV_SAMPLES” model directory, press <Enter> to expose the list of available CATIA model files.

- Select the “CATVSAMP” model file.

The sample CATIA model is loaded.

4. Access the CATV menu

From the CATIA menu, select **CATV**.

5. Select CATIA geometry to transfer to VERICUT for use as stock, fixture, design models

PART is the active option in the **GEOMETRY** menu.

- Select **STOCK**, then select the “STOCK VOLUME” label. The stock part geometry is selected.

- Select **FIXTURE**, then select the “FIXTURE BASE VOLUME” and the 4 “CLAMP” labels.

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- Select **DESIGN**, then select then “DESIGN VOLUME” label.
- 6. Select the CATIA tool path to simulate
 - 3-axis toolpath simulation: select “CATVSAMP3” APT source toolpath to be transferred
 - 5-axis toolpath simulation: select “CATVSAMP5” (requires “Multi-axis Mill/Drill” license)
- Select the **TOOLPATH** menu.

APT CLF is the active option.

- Select the “CATV_SAMPLES” tool path directory.
- Select “CATVSAMP3” (3-axis tool path file) or “CATVSAMP5” (5-axis tool path file).
- Select “**Yes**” to approve your selection.
- 7. Transfer the selected CATIA data to VERICUT

If desired, first orient the CATIA model to provide a good overall view of machining. CATV will automatically transfer the CATIA view orientation to VERICUT, along with the CATIA data.

- Select the **PROCESS** menu.

TRANSFER is the active option. Note that **TRANSFER** is set to transfer all selected CATIA data to VERICUT.

- Select “Yes” to begin the transfer process.

A message similar to “*TRANSFER COMPLETE*” indicates the data has been transferred.

- 8. Run VERICUT and perform the tool path simulation

- Select **VERICUT**.

After a few seconds, the VERICUT window opens.

- In VERICUT, press **Play** (solid green arrow pointing right) to start the simulation.

An error message similar to “*SET ACCESS NOT ALLOWED*” indicates the directory from which CATV has been executed does not have write permission for you. Exit the CATIA/CATV session, change directories to one that files can be written to, then re-execute CATIA/CATV and redo the steps in this session.

- 9. Use VERICUT's "AUTO-DIFF" function to check for gouges and excess material

Check for gouges:

- From the **Applications** menu, select **AUTO-DIFF > Compare**. By default, AUTO-DIFF will identify gouges deeper than .002".
- Press **Apply**. VERICUT displays material gouged in error by the tool path. To see the design model, press **Display**.

Check for excess material:

- Press **Restore**.
- Press **Settings**.
- Choose **Show=Excess Material, OK**.

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- Press **Apply**. VERICUT displays excess material greater than .002" that was left by the tool path.
- ### 10. Exit VERICUT

From the VERICUT **File menu**, select **Exit**, then **Yes** to confirm exiting.

Congratulations on completing your first CATV session! This session provided experience using the CATIA/CATV/VERICUT system and interacting with CATV using standard CATIA user actions. VERICUT was used to display CATIA geometry and simulate the cutting action of an APT source tool path. VERICUT's **AUTO-DIFF** function identified errors in the tool path responsible for gouging the expected finished part, and leaving too much excess material. If left uncorrected, this tool path will probably produce a scrap part.

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