Vericut Model Interfaces

What is a "Model" Interface?

Model Interfaces enable Vericut to read the designated model file formats and use them as stock, fixture, design, tool holder and machine models. The modules do not require a CAD/CAM system be available for Vericut to read or write any of the formats.

Vericut includes the ability to use several industry-standard model file formats: STEP, STL, IGES, VDA-FS, and DXF. Additionally, the following optional model interface modules allow Vericut to use formats: ACIS, CREO, SolidWorks, CATIA V4, and CATIA V5. Also, NX .prt part files can be opened directly in Vericut.

CATIA V5 Model Interface

The CATIA V5 Model Interface reads CATIA V5 part files (.CATPart). CATIA topological entities (such as CATBody, CATNurbsSurface, CATplane, etc..) are converted to triangular facets and written out as STL or Vericut polygon files. When importing a CATIA V5 part file, the interface includes a check-box to optionally separate STEP assembly files into individual components. A CATIA part file can optionally be referenced directly in Vericut's modeling interface-no intermediate file is written.

When used in conjunction with a new output option in the existing CATIA V5 to Vericut Interface (CATV5), the CATIA V5 Model Interface is used to read .CATPart files directly referenced in the Vericut project file written by CATV5.

The CATIA V5 Model Interface allows 3D CAD models of cutting tools to be imported directly into Vericut's Tool Manager. The Tool Manager displays the CAD Geometry window that allows you to identify which parts of the CAD model file correspond with holders, cutters, or inserts. Many 3D CAD models of tooling are available from leading cutting tool manufacturers.

CATIA V4 Model Interface

The CATIA V4 Model Interface reads CATIA V4 model files (.model), version 4.1.9 and 4.2.4. CATIA topological entities (such as SOLIDE, SKIN, VOLUME, FACE, surface, plane, etc..) are converted to triangular facets and written out as STL or Vericut polygon files. Optionally a CATIA model file can be referenced directly in Vericut's modeling interface. No intermediate file is written.

ACIS Model Interface

The CATIA V4 Model Interface reads CATIA V4 model files (.model), version 4.1.9 and 4.2.4. CATIA topological entities (such as SOLIDE, SKIN, VOLUME, FACE, surface, plane, etc..) are converted to triangular facets and written out as STL or Vericut polygon files. Optionally a CATIA model file can be referenced directly in Vericut's modeling interface. No intermediate file is written.

CREO Model Interface

The CREO Model Interface reads CREO part and assembly files (.prt and .asm). Topological entities (such as surface, solid body, shell, face, etc.) are converted to triangular facets and written out as STL or Vericut polygon files. Optionally a CREO file can be referenced directly in Vericut's modeling interface. No intermediate file is written.

When used in conjunction with VCP the CREO Model Interface reads surfaces, curves, points and coordinate systems for use in composite layup.

SolidWorks Model Interface

The SolidWorks Model Interface reads SolidWorks part and assembly files (.prt and .asm). Topological entities (such as surface, solid body, shell, face, etc.) are converted to triangular facets and written out as STL or Vericut polygon files. Optionally a SolidWorks file can be referenced directly in Vericut's modeling interface. No intermediate file is written.

When used in conjunction with VCP the SolidWorks Model Interface reads surfaces, curves, points and coordinate systems for use in composite layup.

Example Module Combinations:

You currently own Vericut and want to read IGES, STL or STEP files as output from various CAD systems.

> Required Module:
> None

You currently own Vericut and want to read ACIS ".sat" files as output from various CAD systems.

   Required Module:
   ACIS Model Interface

You currently own Vericut and want to write ACIS ".sat" files from Vericut's cut stock.

   Required Modules:
   ACIS Model Interface
   Model Export

You currently own Vericut and want to read CATIA V5 .CATPart files, and currently uses the CATIA V5 to Vericut Interface (CATV5)

   Required Module:
   CATIA V5 Model Interface

You need to add a new Vericut seat to simulate CATIA V5 CATProcesses on your machines, and want to use direct reference of CATPart files in Vericut. Additionally, you want to write CATPart files of Vericut's cut stock for use in CATIA V5.

   Required Modules:
   Verification
   Machine Simulation
   CATIA V5 to Vericut Interface
   CATIA V5 Model Interface
   Model Export

NX Model Files NX .prt part files can be opened directly in Vericut. This feature is included with the NX-to-Vericut interface. Since a NX part file can contain multiple parts, a pop-up dialog displays with a list of parts, by name, that can be opened. If the part does not have a name, then "Unnamed" followed by a number displays in the list. Opening an NX part file requires NX 4, 5, or 6 installed on the computer that is running ericut, and an NX license must be available.