Data Exchange Interfaces

Preface

CATIA Version 5 is an open system, capable of interoperating with data in all of the mostly used data format standards in the CAD/CAM/CAE Industry.

For Importing and Exporting external files there are miscellaneous formats : STEP, IGES, DXF/DWG, CGM, STL, VRML, and STRIM/STYLER.

These formats are used to transfer geometric data (surfaces and wireframe) between different CAD-CAM systems in following situations :

- concurrent engineering with using several CAD-CAM systems
- migration of databases when changing system (example: for new CATIA customers)
- exchanges of geometric data with clients or suppliers

Data Exchange Interfaces are :

- **STEP AP203 / AP214** format (Standard for the Exchange of Product model data) : the CATIA - STEP AP203 Interface and the CATIA - STEP AP214 Interface : allow to interactively read and write data in STEP AP203 / AP214 data formats. Its supports geometry and assembly structures and handles topology (shells, solids) on export and import.
  For instance, you can read a STEP file, edit its content in CATIA V5 workbenches, and save the results directly as a STEP file.
- **IGES** format is supported by the CATIA - IGES Interface (IG1) product. CATIA - IGES Interface (IG1) helps users working in a heterogeneous CAD/CAM environment to exchange data through a neutral format. The Initial Graphic Exchange Specification (IGES) format, is the most used neutral format to transfer data between heterogeneous CAD systems. Users can perform bi-directional data exchange between dissimilar systems with direct and automated access to IGES files.
  IGES files containing 3D geometry are imported into CATPart documents. Their type should be "igs".
  IGES files containing 2D geometry and annotations are imported as CATDrawing documents. Their type should be "ig2".
- **DXF/DWG** : DXF formats are supported by the CATIA - Generative Drafting Products. After creating drawings, the designers can export data in DXF/DWG formatted files and import the 2D geometric data contained in a DXF/DWG file into a CATDrawing document.
- **CGM** format is supported by the CATIA - Object Manager Products.
- **STL** format is supported by the CATIA - Object Manager Products. STL concerns stereolithography document (.stl).
- **VRML** format is supported by the CATIA - Object Manager Products. Moreover CATIA DMO (DMU Optimizer) results can be exported as VRML files.

- **STRIM/STYLER :** CATIA - STRIM/STYLER To CATIA Interface 2 (STC) allows to process in CATIA V5 the Geometry from Strim and Styler Applications. It provides a unique direct Interface from Strim and Styler to CATIA, which operates on Strim and Styler Native Format Files in CATIA Environment. The product features a direct access to Styler or Strim data files to convert and store them into CATIA V5 format. The product enables to retrieve an existing Styler or Strim design into CATIA, and proceed to further transformations in CATIA Mechanical Solutions, Potentially NC Manufacturing Solutions and Shape Design & Styling solutions.
  STRIM and STYLER files (with extension ".tdg") can be selected in File Open to Create and Display a CATIA part document enclosing the geometry of the files in a CATIA Format. Files can be selected in the CATIA - DIGITAL MOCK-UP NAVIGATOR to be inserted as existing components in a Product.

The ***CATIA - Data Exchange Interfaces User's Guide*** has been designed to show you how to Import and Export external files in/from CATIA Version 5.

Prior to reading this book, we recommend that you read the Infrastructure User's Guide that describes generic capabilities common to all products. It also describes the general layout and interoperability between workbenches.

Conventions