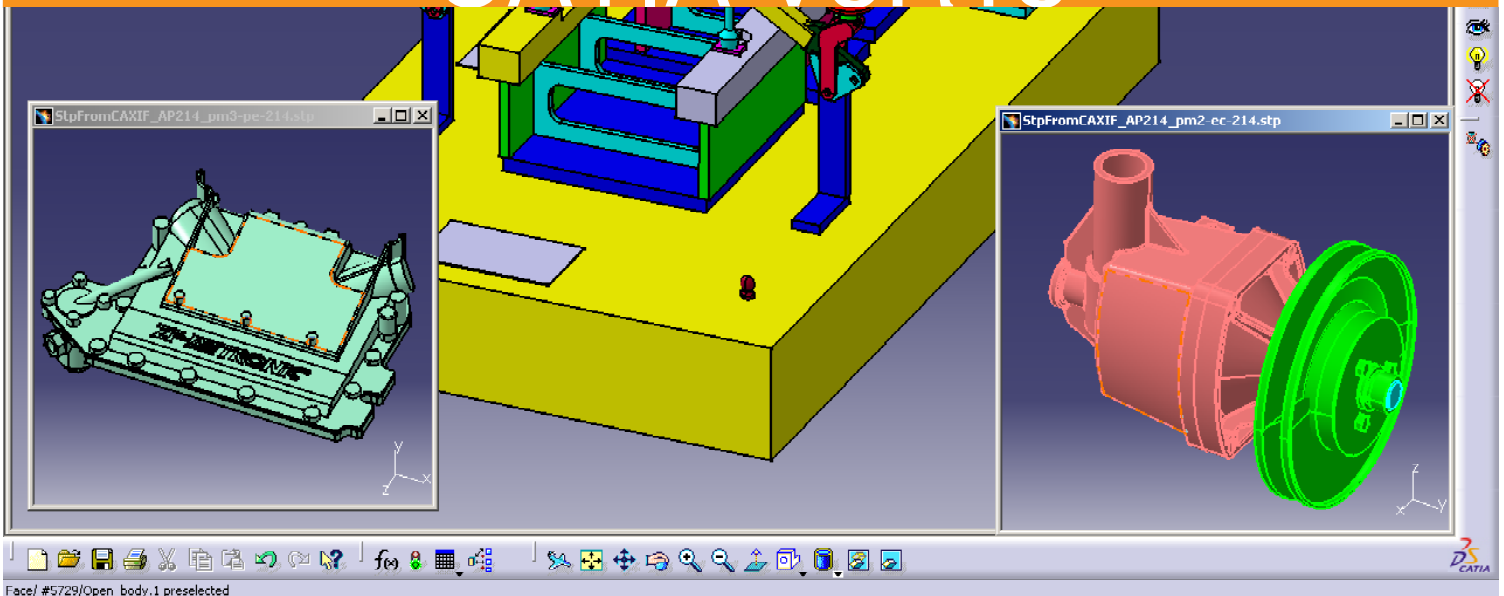


Infrastructure

CATIA STEP Core Interface 1 (ST1)

CATIA V5R18





Infrastructure

CATIA - STEP Core Interface

Allow users to read and write data in STEP AP214 and STEP AP203 data formats.

Product overview

CATIA - STEP Core Interface 1 (ST1) helps users working in a heterogeneous CAD/CAM environment to exchange data through a neutral format. This utility allows users to interactively read and write data in STEP AP214 and STEP AP203 data formats allowing reliable bi-directional data exchange between dissimilar systems. To facilitate access to data, CATIA Version 5 offers a homogeneous user interface for all supported formats, using Windows-compliant user interface controls (such as File > Open, File > Save as) and automatic recognition of the STEP file type.

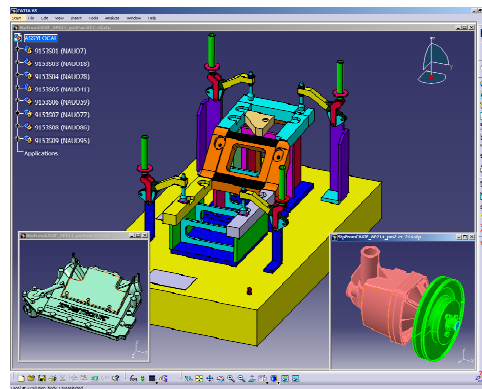
Product Highlights

- >Supports for AP214 and for AP203.
- >Windows-compliant access to STEP files.
- >Supports of geometry and assembly structures.
- >Ensures the transfer quality and reliability.
- >High data transfer performance.

Product Key Customers Benefits

Supports for Application Protocol...

The CATIA - STEP Core Interface 1 (ST1) supports both the AP214, and the AP203. Regarding the AP214, the Class 1 and 2 are supported: the Class 1 and 2 elements are curves, surfaces, shells and solids. Concerning the AP203, the Class 2, 3, 4, and 6 are supported: the Class 2 elements are



curves and surfaces, the Class 3 elements include curves and associated topology (edges), the Class 4 elements include curves, surfaces, and associated topology (faces) and the Class 6 elements are exact solids. By using the CATIA - STEP Core Interface 1 (ST1), users have access to a high level of data for an efficient work with dissimilar systems.

Access to STEP files...

The CATIA - STEP Core Interface 1 (ST1) provides integrated interoperability with STEP data formats through File Open, File Save As and Insert Component. For instance, users can read a STEP file, edit its contents in CATIA V5 workbenches, and save the results directly as a STEP file. The common and intuitive V5 interface make easier the use of STEP files, allowing simple and sure data transfers between systems.

Supports of geometry and assembly structure...

The geometry and the structure of the original file are preserved:

The geometry is preserved at a high level (solids, shells). And the geometry is adjusted (without or with controlled deformation) so that it can be used in the best conditions in the CATIA V5 modeler. If the STEP file contains an assembly structure (NAUO entities), then the structure is preserved: a CATProduct document is generated.

The AP214 mechanism of external references to STEP files is supported, in order to handle large and complex assemblies.

Transfer quality...

The ability to import and export Geometric Validation Properties (GVP) embedded in the STEP files ensures users of the quality and the reliability of the data transferred. Indeed, in addition of existing OK/KO transfer information, some specific properties on solids, products and instance of products, such as center of gravity, volume, or wet surface, can be included in the STEP file. During the import, according to the validation of these properties, a user knows if the data are reliable or if he has to perform some corrections to start with a consistent design (For AP214 only).

High data transfer performance...

The ability to handle very large STEP files, up to 150 Mo, allows the users to exchange large designs.

The translator does quick conversions. Additional data interface formats are supported, including DXF, STL, VRML and IGES, as part of other CATIA Version 5 Products.

ABOUT CATIA V5R18

CATIA is Dassault Systemes' PLM solution for digital product definition and simulation.

plm.3ds.com/CATIA

