

DATA EXCHANGE COMPATIBILITY BETWEEN CAD/CAM  
SOFTWARE IN INTEGRATED DESIGN PROCESS FO THE  
TECHNICAL PRODUCT

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**Abstract:**

In order to have good compatibility between CAD/CAM software, actually are used formats like STEP or IGES. In this paper are explained the internal structure for neutral formats and the most significant recommendations about data transfer.

**Keywords:** 3D Models, Computer Aided Design, data transfer, native formats, neutral formats,

## 1. INTRODUCTION

Initial Graphics Exchange Specification (IGES) and Standard for the Exchange of Product Data (STEP) are two neutral file formats. These formats enable different systems to transfer data without the need of a specific translator on each system. IGES covers a wide range of application areas including electrical, plant design, as well as mechanical applications, and provides a standard format by which the user can transfer the data from one CAD software to another. A standard format like IGES requires two levels of processing. At the first level, the CAD data as described in the software native format is converted into the IGES format. At the second level the data is converted from IGES to a format that can be understood by the CAD software. The basic file structure of IGES contains five sections. The sixth section is optional (Flag section - B/G).

Table 1. IGES file structure

SECTION	CODE
Start	S
Global	G
Directory	D
Parameter Data	P
Termination	T

IGES does not provide data that may be relevant to applications other than drawing or 3D modeling. For example the designer may be interested only in the drawing or the geometry of the part to be manufactured but the process planner would be more interested in the form features and the manufacturability of the part and the inspection department may be interested in the tolerance information for the part. Each of the departments would share the same IGES file and it is left to the interpretation of the individual to derive the relevant information. All these reasons lead to the development of the STEP standard.

STEP provides representation of product information along with the necessary mechanisms and definitions to enable product data to be exchanged. The exchange is among different computer systems and environments associated with the complete product life cycle including design, manufacturing, inspection, maintenance and disposal. The information generated about a product during these processes is used for many purposes. STEP enables all people contributing to the design, manufacturing, marketing and supply of a product and its components to contribute to, to access, and to share information.

The file structure for a STEP file has a modular structure which makes it easier for developers to adapt the format to their own needs. The modules are called "classes" and are classified using numbers.

Beside these two neutral formats, the designers can also use the native formats of the most known softwares but with the risk of facing problems. For example, in many cases the native formats may not translate accurately into SolidWorks parts. We can find a large number of unwanted surfaces and other geometry problems.

In the following sections of this article we will develop a compatibility table for the native and neutral formats and recommend the best transfer formats for two of the most used CAD softwares, Catia V5 and SolidWorks.

## **2. FORMAT COMPATIBILITY BETWEEN CAD PROGRAMS**

The following table are listed the suggested 3D file formats and methods to be used when importing from a known 3D CAD program. The numbers beside each checkmark specify the order of preference amongst the various file formats which can be exported from each CAD program [1].

The numbers displayed in the table refers to the option regarding the file format to use. The number "1" represents the first saving option and "5" is the last. For example, in Catia V5 it is recommended to work in native formats. It is normal to do so, but if the part is

modeled in V5 R15 and for some reason we must work in V5 R7, the best choice is to use the .step format.

Table 2. 3D file formats saving options

Program	Formats							
	Native	IGES	STEP	PARAS OLID	ACIS	STL/SPL	XGL/ZGL	Other
Autodesk Auto CAD	1	-	-	-	2	3	3	.3ds
Mechanical Desktop	4	2	1	-	3	-	-	-
Autodesk Inventor	4	2	1	-	3	5	5	-
CADKey	3	2	1	2	2	4	4	-
Catia V5	4	2	1	-	-	3	-	.vda
Pro/Engineer	3	2	1	-	-	4	-	.obj
Solid Edge	1	-	-	2	-	3	3	-
SolidWorks	4	2	1	3	3	5	-	.vda, .vrml
Unigraphics	1	2	4	2	-	5	-	.vda

Each product has its own considerations that must be taken into account in order to make the most of imported files from other systems. The following is a listing of the preferred file types, in order of probability of success, which would be most beneficial when imported/exported [2].

Table 3. Best format compatibility (1/2)

	MDT	Inventor	CADKey	Catia V5
<b>Mechanical Desktop</b>	Native (.dwg)	STEP, IGES, ACIS (.sat)	ACIS (.sat) STEP, IGES	STEP, IGES
<b>Autodesk Inventor</b>	STEP IGES ACIS (.sat)	Native (.ipt)	STEP IGES ACIS (.sat)	STEP, IGES, STL, SPL
<b>CADKey</b>	ACIS (.sat) STEP, IGES	STEP, IGES, ACIS (.sat)	Native(.prt, .ckd)	STEP, IGES
<b>Catia V5</b>	STEP, IGES	STEP, IGES, STL	STEP IGES	Native
<b>Pro/Engineer</b>	STEP, IGES	STEP, IGES	.prt, STEP, IGES	Native (.asm), STEP, IGES
<b>Solid Edge</b>	-	-	-	Native (.asm) Parasolid
<b>Solid Works</b>	ACIS (.sat), STEP, IGES	STEP, IGES	.prt	IGES, STEP, .wrl
<b>Unigraphics</b>	IGES, STEP	IGES, STEP	.prt, IGES, STEP	IGES, STEP

Table 3. Best format compatibility (2/2)

	Pro/Engineer	Solid Edge	Solid Works	Unigraphics
<b>Mechanical Desktop</b>	STEP, IGES	-	ACIS (.sat), STEP,IGES	IGES, STEP
<b>Autodesk Inventor</b>	STEP, IGES	-	Native (.ipt), STEP, IGES	IGES, STEP
<b>CADKey</b>	.prt, STEP, IGES	-	Native (.prt, .ckd), IGES, STEP	.prt, IGES, STEP
<b>Catia V5</b>	STEP, IGES	-	IGES / STEP, .cgr, Native (.model, .exp, .session)*, .wrl	Parasolid, IGES, STEP
<b>Pro/Engineer</b>	Native (.prt, .xpr, .asm, .xas)	.asm	Native (.prt, .xpr, .asm, .xas), STEP/ IGES	.prt, IGES, STEP
<b>Solid Edge</b>	.asm	Native (.par, .psm, .asm)	Parasolid, Native (.par, .psm)	Parasolid
<b>SolidWorks</b>	STEP	Parasolid	Native	Parasolid, native (.prt), IGES, STEP
<b>Unigraphics</b>	.prt, IGES, STEP	Parasolid	Parasolid, native (.prt)	Native (.prt)

In order to help the conversion of native formats into neutral ones, the software developers have come up with the idea of creating “transition” programs. These tools convert native formats without the need of the native software (for example, Catia V5 to Step, Catia V5 to Solid Works or Inventor to IGES).

Catia and SolidWorks are the most used in education. For this reason we have chosen to explain the data transfer in/from these two softwares.

For data transfer from Auto CAD, Inventor and Mechanical Desktop to SolidWorks no conversion is needed. The layers, line types, fonts, colors, blocks, and attributes will be recognized by SolidWorks. If running MDT and SolidWorks on the same computer, you can import MDT files directly into SolidWorks with feature history.

If Pro/E is used to model a part, STEP is the best format to transfer it. STEP files generated by Pro/E are quite accurately translated into SolidWorks. The native formats are also used but in many cases they are not accurately translated.

Although SolidWorks has the ability to read Native Unigraphics files, Parasolid over UGII when given a choice is preferred. UG native files also contain all the construction geometry which is unnecessary when importing a part of assembly. The Parasolid file strips all the unnecessary data from the file and makes the import much faster. Furthermore, UG

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\* this is not a standard part of SolidWorks

Native assemblies (.prt) externally reference the part files which will also need to be included in any dataset. A Parasolid file of an assembly contains the assembly and ALL the part files in a single file.

For Solid Edge to SolidWorks data transfer no conversion is needed.

### 3. STUDY CASE: DATA TRANSFER BETWEEN CATIA V5 AND SOLIDWORKS

As we mentioned before, Catia V5 and SolidWorks are two of the most used softwares in education and industry. Moreover they offer the largest number of “open” and “save” options. Not all the file formats permits the best data transfer but we can easily identify the most common ones like STEP, IGES, PARASOLID and ACIS for both softwares [3].

For the data transfer we have chosen the assembly in the figure 1.

The model was saved as follows: for Catia V5 .stp, .igs, .cgr, .wrl and for SolidWorks .step, .igs, .wrl. Regarding the STEP, IGES and VRML formats, Catia V5 has special format compatibility settings: STEP exporting options refers to the application protocols (203 and 214 – both can be read by SolidWorks) and to the structure and geometry (for assemblies), IGES options refers to the curve and/or surface type and representation mode and VRML options refers to the version used (1.0 or 97).

Solid Works importing options includes: showing IGES levels, map configuration data for STEP and import options (graphics body, solid body and surface body) for VRML format.

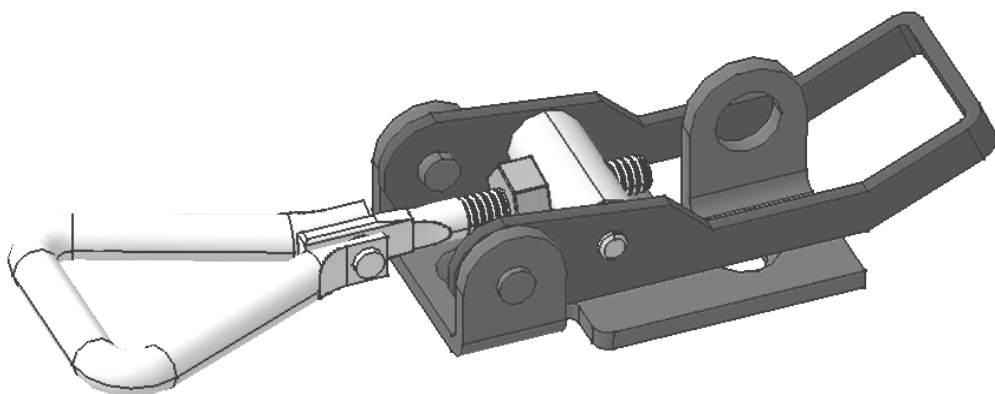


Fig.1. The model used for data transfer

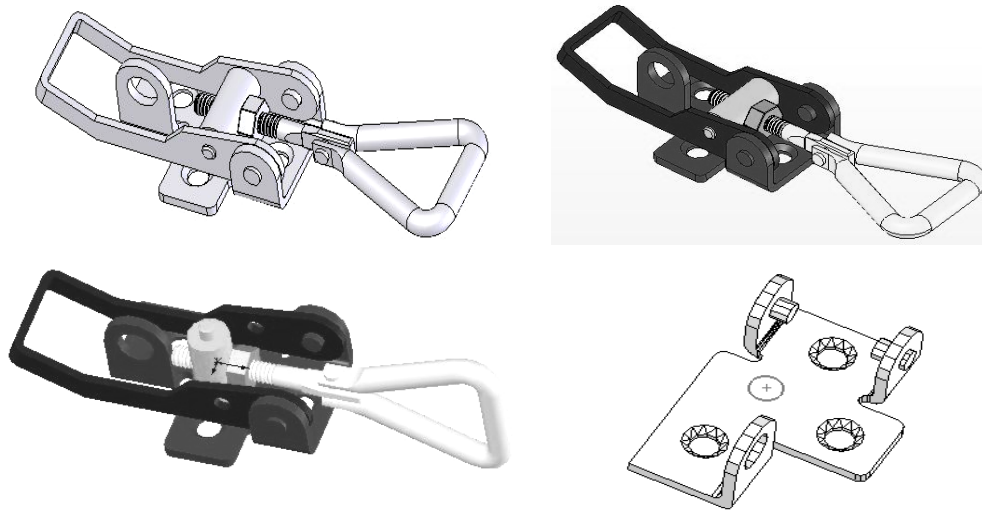


Fig.2. SolidWorks results

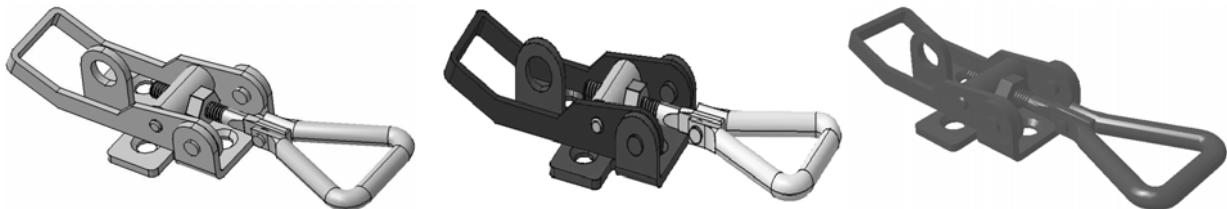


Fig.3. Catia V5 results

Catia V5 importing options for STEP and IGES files refers to the automatic or advanced optimization of curves and surfaces continuity and offers also the possibility of importing one or more CAT Parts for assemblies. VRML option refers only to the import unit (mm, cm and m) and crease angle.

#### 4. CONCLUSION

IGES and STEP are two neutral file formats that enable different systems to transfer data without the need of a specific translator on each system. In order to help the conversion of native formats into neutral ones, there were developed “transition” programs that convert native formats without the need of the native software.

For Catia V5 to SolidWorks, with the exception of STEP and IGES formats, SolidWorks can also import Catia native formats but this option is not a standard part of the program. Other formats recognized by SolidWorks are: VRML (.wrl) and Catia Graphics (.cgr). Catia V5 offers more saving compatibility options for data transfer to SolidWorks.

#### 5. REFERENCES

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