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PROBLEMS WITH 3D DATA EXCHANGE BETWEEN CAD SYSTEMS USING NEUTRAL FORMATS

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Abstract: *This paper is focused on the results of 3D data exchange between CAD systems using neutral data formats such as STEP and IGES which have addressed the issue of the data transfer, but these are incapable of handling design intent information generated by contemporary CAD systems. Accentuate are problems of loss and change of the graphic information, topological data, naming and coordinates after the transformation. This project is called "CMS in CATIA" and is executed by the Engineering and Integration Office on Compact Muon Solenoid (CMS) Experiment, which is part of the Large Hadron Collider (LHC) at the European Counsel for Nuclear Research – CERN, Switzerland. The 3D models have been created at Institutes and Universities which are collaborators in the CMS experiment in different CAD environments and different version of the systems.*

Key words: *3D models, CAD systems, graphic information, neutral formats.*

1. INTRODUCTION

The mechanical engineering translation of 3D CAD data between different applications is a main issue. In an ideal design world, every CAD package would use a single perfect data format, and part designs could be shared easily between different packages without loss of information, features or design process. However, in the real design world, every CAD package has its own native format, as well as different abilities to import and export neutral formats and possibilities for other CAD system native formats.

There are two basic approaches to handling the problem of CAD data exchange: direct translation and the use of neutral formats. Direct translations attempt to preserve feature hierarchies and parent/child relationships. These are preferred when multiple parties may be required to perform significant editing of a part, especially in the nature of modifying existing features and dimensions. Due to the complexities of the feature hierarchies and the differences in implementations across CAD packages, neutral formats only contain the exact geometric definition of a part. Parts exchanged in neutral formats are generally imported into a CAD application as "dumb solids" without individual feature data [1].

In this paper the results and the faults of 3D data exchange from SolidDesigner, AutoCAD, Euclid, SolidWorks and I-Deas into CATIA using neutral file formats such as STEP and IGES are presented. This research is part of the data exchange project called "CMS in CATIA" which is executed by the Engineering and Integration Office on Compact Muon Solenoid (CMS) Experiment which is part of the Large Hadron Collider (LHC) at the European Counsel for Nuclear Research – CERN, Switzerland.

The models are created by collaborator Institutes and Universities from all over the world in various CAD systems. The project's main challenge is to convert all these models into CATIA and integrate them in one database.

IGES was the first attempt to create an industry-wide translation standard. (IGES is short for "Initial Graphic Exchange Specification.")

IGES was created by committee, and was used commonly among larger CAD systems, such as those used by automotive and aerospace corporations. Today, IGES is replaced by STEP [5].

Like IGES, STEP has been created for a translation standard for the entire CAD industry (PDES is abbreviator for "Standard for The Exchange of Product model data"). It has been under development since the late 1980s, and like IGES is being created by a committee.

STEP is meant to be powerful enough to handle all aspects of a project, from the initial design through to construction and decommissioning [4].

A protocol is a set of conventions or rules that govern the operation of functional units to achieve communication:

- IGES application protocols provide a formal procedure for specifying neutral IGES-based application specific formats [2]. This procedure involves identifying the information requirements of an application area and documenting them in a conceptual information model. The conceptual information model is then used to select the IGES constructs for representing the required information.
- The STEP application protocol AP203 [3] allows the transfer of boundary representation (B-rep) and close-

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Table 1

Numbers of 3D models for converting

CAD systems	Quantity
Euclid	21 236 pcs.
SolidWorks	6865 pcs.
AutoCAD	2048 pcs.
SolidDesigner	28 pcs.
I-Deas	127 pcs.

ly related types of models, including assemblies of such models. ISO 10303-111 ("Elements for the procedural representation of solid shapes") [4], provides representations of operations for the construction of feature based solid models. As far as shape is concerned, Edition 1 of ISO 10303-203 (AP203 of STEP) [3] does not support the exchange of features, parameters, constraints or construction history information, but only geometrical and topological information relating to the final shape of a CAD model.

In Table 1, three dimensional models which must be converted via direct translators and neutral formats are shown. The best method for translating CAD data depends on the specific application.

2. CONVERTED DATA FROM DIFFERENT CAD SYSTEMS TO CATIA

2.1. Data from SolidDesigner

The SolidDesigner system was used by engineers from Institute for Systems Engineering and Robotics where they have created 28 assemblies.

In Fig. 1,a model of a part of the Compact Muon Solenoid Cooling System created in SolidDesigner and converted into CATIA via STEP file is shown.

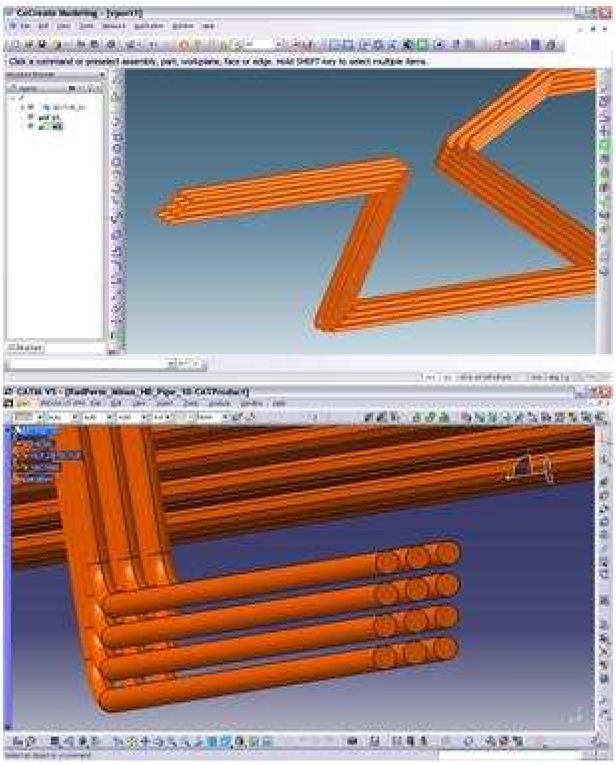

Fig. 1. Native SolidDesigner and converted by STEP into CATIA models.

After converting, the transformed graphical information is different from the original one. The pipe section in some areas is converted from circle into octagon and also all pipes in the model are converted into solid bodies. CATIA recognizes all these as assembly files and transfers correctly the assembly tree structures.

However, in lower level in the part tree structure there is no history of the creation and possibilities for alteration.

2.2. Data from AutoCAD

In Fig. 2 the model of EndCap Muon Chambers part of the Compact Muon Solenoid created by International Engineer team from Italy, China, Russia and Bulgaria is shown. The assembly is converted into CATIA by STEP file from AutoCAD. On this CAD system 2048 assemblies are created.

After conversion, in EndCap Muon Chambers models constructive lines appeared which are invisible in the original model. In CATIA, these lines can be hided only by hand but in some case when the line is hidden, the attached part to the line also hides. In a model like this one with a large number of parts, this operation is very complicated and time consuming.

The STEP file saves the tree structure with names of the parts but inside the parts the names and the creation history are lost. After translation, the parts lost the colors from the original assembly.

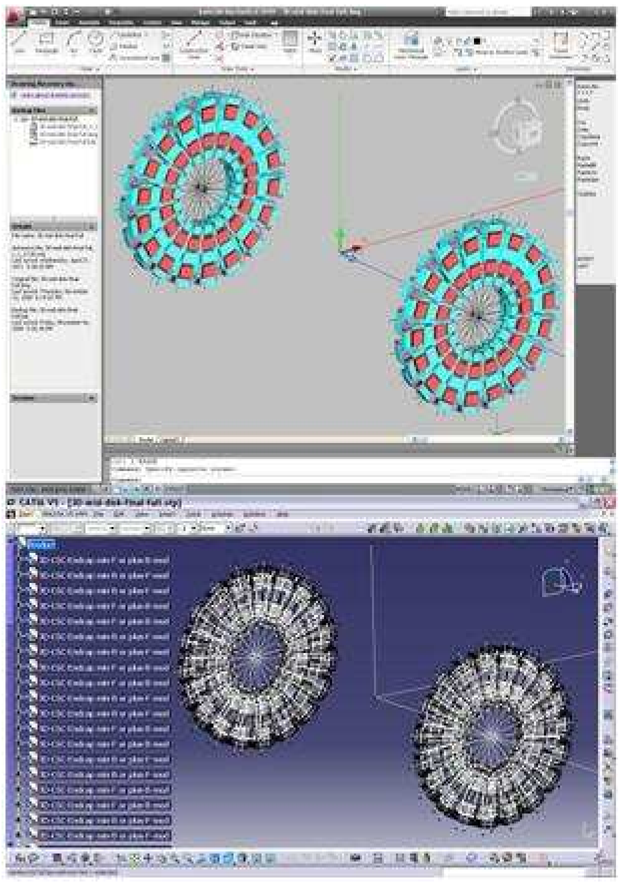

Fig. 2. Native AutoCAD and converted by STEP into CATIA models.

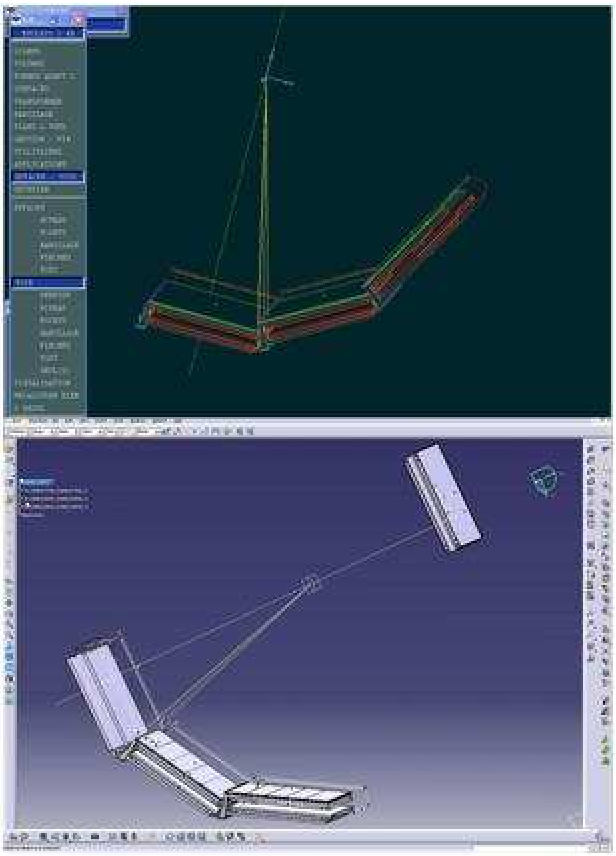

Fig. 3. Native Euclid and converted by STEP into CATIA models.

2.3. Data from Euclid

On the Euclid System, 21 236 assemblies has been created. The results of data exchange between Euclid and CATIA via STEP are shown on Fig. 3. The presented model is part of Barrel Muon Chambers, which is part of the Compact Muon Solenoid.

The converted model is with different orientation from the native model. Some parts are converted as mirror of the original parts. Tree structures with the correct names of the parts are preserved after the transfer as well as the graphical information and the topology. This result might be sufficient from the geometric point of view but the lost of the design intent that was present in the original system is unsatisfied and not practicable. There is also a loss of parametric data during the exchange. The modification of the model in receiving system is not possible. With the increasing of the parts in the assemblies, the errors highlighted above grow up. This leads to loss of graphical information and constraints, dislocation of the details and errors in tree structure.

2.4. Data from SolidWorks

SolidWorks is the second program that offer high number of three dimensional models used for creation of Compact Muon Solenoid Detector after Euclid. On this software, 6 865 assemblies are created as part of them are designed at CERN, the rest are built in External Institutes and Universities. It is still in use for upgrading of existing and development of new detectors.

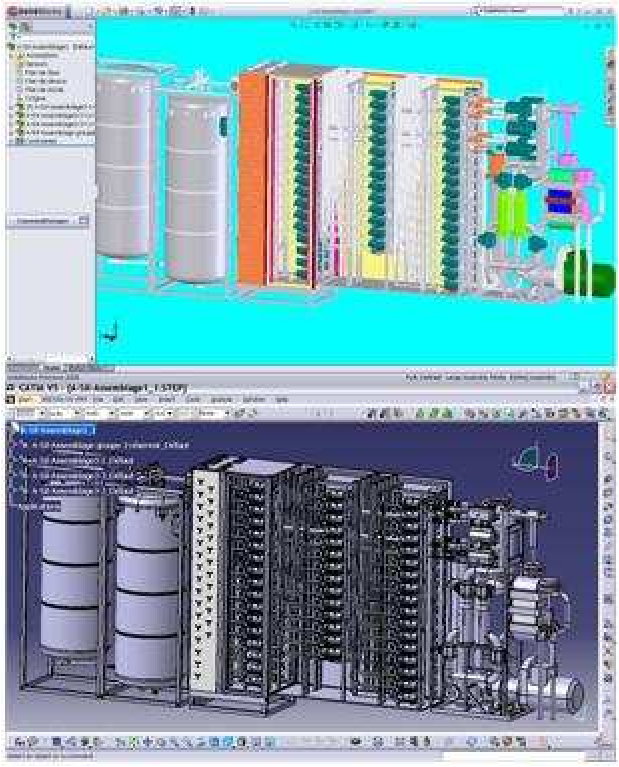

Fig. 4. Converted model from SolidWorks to CATIA by STEP format.

The model shown in Fig. 4 is part of the Cooling System of Pixel Detector part of CMS Project. The SolidWorks model is transferred by STEP format into CATIA. The graphical information and the topology are preserved after the transfer as well as a tree structure with the correct names of the parts. This may be enough from the geometric point of view, but the loss the design intent that was present in the original system is not acceptable for the proper correction and development of the models. There is also a loss of parametric data during the exchange. That is why the modification of the model in receiving system is not possible. In some cases of export from SolidWorks into STEP the size of the file becomes too large, which gives import errors in CATIA. This is caused by the method of creation of the assemblies and the used constraints.

2.5. Data from I-Deas

The I-Deas System is used for creation of Compact Muon Solenoid Detector and it has been used for the creation of approximately 127 assemblies. All of these models must be stored into CATIA and used later for upgrading the Detector.

In Fig. 5, a part of the Muon Chambers Barrel Detector in native format – above is shown and imported into CATIA model by IGES – below. For CMS barrel more than 500 numbers of this chamber with six different variations are designed and produced.

After the conversion of the Muon Chamber model constructive lines appeared which has been invisible in the original model. The converted model lines can be hided only by hand, in some case when we hide the line, the attached part to the line it also hides. The more we increase the number of the parts in an assembly the more

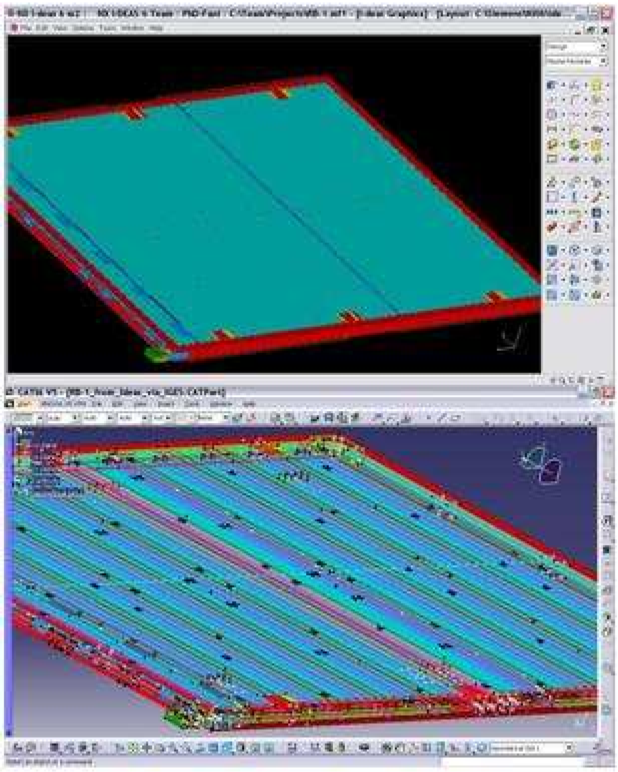

Fig. 5. Converted model from I-Deas to CATIA by IGES format.

complex the task of correction becomes. This also increases the time needed to perform the job.

After the conversion a loss of parts occurs (graphical information). For example, in Fig. 5 it is clearly visible where the missing top plates of the chamber are.

Another big problem is that some of the extruded parts are visualized as a wireframe. They cannot be seen like solid bodies, meaning that they should be crated again.

The IGES file does not save the tree structure with names of the parts. The assembly file is saved as CATPart file and assigned the name of the first part from the tree structure. The Part name is "Any" and inside the creation history is empty. There exists only geometrical sets which are broken and the model cannot be modified, hidden, etc. After translation, the bodies remain with the colors from the native file.

3. CONCLUSIONS

Native formats offer simple translation of dumb solids, but even so there are few pitfalls to watch out for. If two CAD packages use different representations for one type of geometry at some point the representation must be converted or even discarded (though this is thankfully uncommon), regardless of the type of translation. STEP and IGES were designed partly to solve this problem, but no format can completely eliminate all translation issues.

The best method for translating CAD data depends on the specific application. Neutral formats are the simplest and most widely portable solution for data translation and generally succeed to fulfill their job supporting solid models for mechanical design.

The most common problems in transferring the model via STEP and IGES are:

- loss of the architectural structure;
- change the names of the parts with numbers or names assigned to the directories where they are stored;
- loss of bodies from the assemblies;
- displace of details of their correct position relative to the original model;
- loss of original color of the parts;
- visualization of details such as wireframe;
- displaying the construction lines that are hidden in the original product;
- modification in the graphic information;
- modification on hollow bodies into solid bodies.

The final results of the converted models posses corrupted data due to the conversion. Some of the received errors cannot be corrected and those parts which can be rectified in CATIA environment are only by hand, which requires very good knowledge in at least two CAD systems and it is very time and cost consuming.

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