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INTEROPERABILITY ISSUES AMONG CAD SYSTEMS: A BENCHMARKING STUDY OF 7 COMMERCIAL MCAD SOFTWARE

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1. Introduction

In the current design contest, where several very different types of expertise need to be integrated, and where engineers, designers and manufacturers need to share all or part of the same product data and work together on it throughout the design process, interoperability among the computer-aided tools used is a requirement. For many design applications, the CAD model of the product is the basis on which any virtual analysis into downstream software environments is performed. This is the case for FE analysis, Kinematics or Dynamics, CAM, Virtual Reality and so on. It is also common for companies to move to a new modeling system to improve product quality and to reduce production costs, while maitaining the original legacy data archives. Unfortunately, it quite often happens that the translation task fails resulting in geometry faults and information loss. As described in a previous work [Gerbino 2003], interoperability issues come from several factors, among which the different geometric modeling kernel of the applications and the accuracy adopted to perform geometric calculations have a dominant and critical role. See also [Brondi 2002]. Many problems in CAD file translation result from human error in modeling practices because the software leaves ambiguous geometry behind after the CAD operator has performed various modeling operations involving especially Boolean operators. These details are hidden and none of the CAD packages are forgiving of imported geometry which is ambiguous. Thus, common good modeling guidelines should be established/adopted to ensure the quality of all CAD models [Searle 2002, Smigel 2002]. Shammaa has pointed out the need to have robust kernel technology that allows the transfer of data based on known rules and tolerances of each specific CAD system [Shammaa, 2002]. These criteria have been included in the translator software by Elysium Inc., which uses automatic technology that adapts using known rules based on the sending and receiving CAD system.
In the recent years the interest in interoperability issues is growing, as demonstrated by the increased attention of many CAx vendors to the offering of more reliable and proven translators. Some big vendors are joined together in consortiums to implement and improve data exchange among their software and releasing documents by pursuing best practices in the preparation and exchange of CAD data. Maybe the most important is ProSTEP, a German automotive consortium, dedicated to the development of STEP, the International standard ISO 10303, which is becoming the current neutral standard in data exchange. Based on the periodic improvements made by ProSTEP, some reports are available related to the benchmark tests in the automotive industries by using STEP processors. A comparison with other standards, like IGES (ANSI standard) and VDAFS (DIN standard), is available

as well. All of these benchmarks are related to the best use of pre- and post-processors of some CAD systems.
However, the data exchange by neutral translators is not the only way to move data among CAx systems. Direct translators (costly, and available only for some high-level CAD modelers) and geometric modeling kernel-level data exchange (see especially ACIS and Parasolid), which uses their basic neutral file representation (respectively *.sat* and *.x_t*), are also adopted in exchanging data, even though their use is, at present, very limited. The neutral standard exchange still remains the most widely adopted method.
With respect to 7 mechanical CAD (MCAD) modeling software (CATIA, Pro/E, Rhinoceros, SolidEdge, SolidWorks, ThinkDesign, and Unigraphics have been tested), ranging from high-level to low-level systems, a quite complex model has been created in each environment and translated to the other systems via IGES and STEP neutral files. Best practices have been adopted to improve translations, both by using the internal repair tools or the manual ones. Thus, the authors analyze what healing tools are available and important to repair a CAD model, and how to use them, as well as how to prepare a model to ensure CAD interoperability and prevent failure in data exchange via neutral standard formats.
This paper does not replace the best practice manuals or technical white papers, if any, released by some software companies to do geometric translations. Where available these documents should be read before starting any data exchange operation, especially where many files and complex geometric models are involved. Typically, these documents provide a resumée of all settings and procedures necessary to perform the translation, sometimes customized for the receiving/sending system. Here, the authors want to suggest some useful hints for using the basic tools available in the tested CAD programs, and guide general users to investigate the geometry and to look for tools able to give useful information about the model being translated.

2. Main features of the tested CAD systems

Analyzing the CAD systems tools dedicated to realize a well-done data exchange is the aim of this work. Dealing with the choice of the CAD systems to test, those that cover different marketplaces and with complementary features have been considered. High-level CAD environments, like CATIA, Pro/E and Unigraphics are oriented to very advanced modeling both with parametric solid and surface features, while middle-level CAD systems, like SolidEdge, SolidWorks and ThinkDesign are more oriented to parametric solid modeling even though the implementation of surface features is growing and improving with every new release. ThinkDesign, in particular, offers many geometric tools for surface modeling and editing, and the same is true for Rhinoceros, more and more appreciated by CAD users for its high ability to handle NURBS curves and surfaces which may be used in other CAD environments. The releases of the CAD systems tested in this benchmark are illustrated in the Table 1.

Table 1. MCAD systems tested

CAD SYSTEM	RELEASE
CATIA	5.8
Pro/ENGINEER	2000 i2
Rhinoceros	2
SolidEdge	14
SolidWorks	2001
ThinkDesign	8
Unigraphics	NX 1.0

The study has been carried out using as a benchmark model an exhaust manifold for a four cylinder engine. The geometric features of this model were considered suitable to emphasize problems usually related with data exchange, because it has several crucial zones, despite its relative lack of complexity. Recent studies have highlighted that the hardest objects to transfer are trimmed surfaces with a conic or toroidal shape, multiple fillets with small radius, sliver faces, small dimension faces and curves, and so on (see ProSTEP). The benchmark model contains all of these features, as shown in figure 1.

The experimental work has been developed in several steps. At first, the collector has been created in every CAD environment using approximately the same modeling approach and checking it with internal check tools, where available. Next, data exchange between many combinations of the seven CAD systems has been performed through IGES and STEP (AP203) neutral formats.
Finally, where problems have occurred, an attempt to repair bad data using model check and healing tools has been made. Before translating data, a preliminary analysis has been carried out to look for the best conversion parameter settings, leading to an efficient transfer. To do this, all of the suggested settings for each environment available in the help guides and those recommended by international organizations, like ProSTEP for STEP translations [ProSTEP], have been adopted.

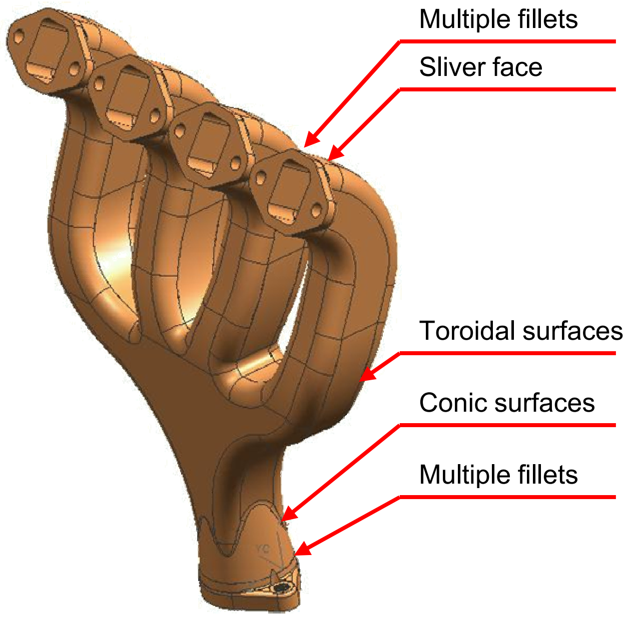


Figure 1. Critical zones in the tested exhaust manifold model

3. Healing tools

Every CAD system offers, besides data exchange processors, many tools that help users to obtain a correct model. Those tools can be used before and after translation and can be divided into three categories:

I) Check tools
II) Diagnostic tools
III) Healing tools.

Check tools are used before conversion to verify that the original model has no problems that could lead to a partial data loss. Instead, diagnostic tools show the problems which occurred during conversion and propose methods to resolve them. At last, healing tools are useful to repair those models that produce alterations (loss of surfaces, untrimmed or no-sewed surfaces, etc.). Those tools can be included in a post-processor in order to automatically repair the corrupt models or can be a separate module used after the conversion process. Later on in the paper, the main tools available for each CAD system tested will be presented.

3.1 CATIA

In CATIA there is not a specific check tool that is able to analyze the geometry at the end of the modeling phase and verify the topology correctness. By setting TOPOLOGY_HEALING “on” in the post-processor, the system is able to repair the topology of the imported solid model during the translation phase. However, this system points out free edges that may indicate surfaces loss or the presence of gaps and overlaps. If free edges are found, the connection check command is able to give information about the distances between them. Healing tools enable the zipping of gaps by locally deforming surfaces. When some faces are not correctly exchanged, a solid body is not generated, but their edge curves and underlying surfaces usually will do, thus allowing reconstruction in the receiving system. A user could rebuild the missing faces by using edge curves and underlying surfaces or by creating ex-novo the connection surfaces, and constraining them in tangency and curvature to neighboring faces. A solid model can be then reconstructed by sewing all surfaces and filling the closed surface that comes out from this operation.

3.2 Pro/ENGINEER

The Pro/ENGINEER's model check module is useful after and before conversion. At the end of modeling it allows emphasizing of critical entities on the screen, showing why they could be dangerous and, in addition, it gives information useful for optimal regeneration of the model. For example, it warns if tiny edges are found and it proposes the reconstruction of one or more features. After conversion, it points out altered geometric entities, indicates which problems have occured and recommends actions to correct these alterations. For example, it shows gaps between surfaces and suggests using the zip gaps command to close them. Some healing operations can even be activated in the Pro/ENGINEER (Pro/E) post-processor, i.e. during the reading phase. These indications and tools permit the repair of surface boundaries and close open boundaries. After importing geometry in Pro/E environment, it is possible to act either with manual healing tools or with the automatic ones. Manual healing performs several actions such as: zipping gaps between surfaces, fixing boundaries, tangency and vertices. Automatic healing redefines the imported feature in order to correct errors automatically. In both cases, the operations cause local geometry alterations. It is possible to visualize the deviation from the original geometry before accepting the changes.

3.3 Rhinoceros

Rhinoceros offers several check and diagnostic commands for curves and surfaces. These commands can be used to determine whether entities that may cause problems are present, such as a surface with a self-intersecting curve or tiny trim curve edges, or corrupted entities, such as naked edges or broken edges. The user guide gives a detailed list of the operations to do to solve those problems by using healing commands. For example, when naked edges (free unmatched surface edges between two adiacent surfaces) are present, it usually suggests joining them by automatically adjusting the local tolerance, thus allowing inaccuracies to be managed separately from the rest of the model and considering the two edges coincident. Instead, when broken edges (edges that do not pass some internal checking functions) are found, it advises detaching, trimming and re-trimming the surface. The solid model (closed polysurface) may be created only if the model topology is tested successfully and no non-manifold edges (edges shared by more than two faces) are found. Regarding to surface modeling and rebuilding, this modeler offers very powerful commands. In fact, during the generation of a curve-network surface, it permits the management of position, tangency and curvature continuity along its boundaries, and, by using the rebuild command, it allows the separate setting of surface degrees in U and V directions.

3.4 SolidEdge

SolidEdge has a geometry inspector modulus that analyses models and provides a list of problematic areas giving, at the same time, useful information to repair geometry, where possible. Warning messages indicate whether a corrupt data structure or a lack of surfaces or a degenerate geometry is present. However, they often suggest the solution for geometry reconstruction. There are also commands that show free edges and sew unjoined surfaces. In SolidEdge the automatic healing command is only available as an option in its post-processor interface. In repairing automatically solid models, imported as IGES file format, it tries to apply a wide range of tolerances, from smaller to larger, to sewing free edges. Few commands are available to repair surfaces in the modeling environment and to build new ones.

3.5 SolidWorks

SolidWorks also provides useful geometry check, diagnosis and healing tools. In particular, the check tool is used at the end of the modeling phase to emphasize which elements could cause problems, such as invalid faces, tiny edges and invalid edges. The diagnostic tool identifies gaps or invalid faces and corrects them automatically. A healing tool is also available that tries to improve geometry. It simplifies parametric geometry by converting it into analytic form, removes redundant geometry and repairs it.

3.6 ThinkDesign

The ThinkDesign user guide provides interesting modeling rules and advice in order to avoid the presence of errors or critical areas before data exchange. Nevertheless, it does not offer specific check and diagnosis tools. In fact, the topology check tool only points out whether solid is closed, open or non-manifold, but it does not indicate the location of the bad entities. The Tolerance Check toolbar provides a tool to check the tolerance of each entity, and the maximum tolerance in the model. In the event of problems, the user should use this information to manually adjust the tolerance of the entities that do not match each other. The Global Shape modeling commands are the best tool for modifying surfaces seen up to now, since it allows easy variation of edges with a shape control.

3.7 Unigraphics

The Unigraphics' Examine Geometry command is perhaps the most complete tool among those seen in CAD systems. It analyses models trying to find tiny or misaligned objects, self-intersecting faces, inconsistencies, bad data structure, gaps between surface edges, spikes and cuts, and edges whose adjoining faces do not join smoothly. The heal geometry command attempts to repair these bad entities modifying and simplifying geometry. For example, it tries to automatically replace tiny edges and faces with vertices, if within a specified tolerance. A wide and documented list of the problems occurring and repaired is also released. The model check command is also available in the import/export menu, suggesting to the user to run it before translating any geometry.

4. Modeling approach

As described in the first paragraph, one of the hidden problems which may cause interoperability issues is the lack of proven modeling guidelines. Many different methods can be adopted to create a solid model using several combinations of union, intersection, addition and subtraction Boolean operations. But standard rules specifying preferred modeling practices are important and manufacturing companies should train CAD users in the preferred methods for modeling parts and assemblies. It is proven that, sometimes, simply using a different modeling sequence will result in the optimization of the final model.

For the manifold CAD model of this benchmark study the adopted modeling rules are outlined in the following (Figure 2). Generally speaking, the same basic modeling procedure has been followed in every CAD environment, thus avoiding possible differences in the models being tested.

At first, the main body of the exhaust manifold is generated by extruding two profiles, sketched on two perpendicular working planes, then intersecting the extruded solids and adding fillets on the corners (see Figure 2a, b and c). Then, the conic feed pipe (solid of revolution) is added and the intersection edges are filleted (Figure 2d). The whole solid model is then hollowed out with a shell-like command (Figure 2e). One extruded flange with draft is added at one end (Figure 2f). A similar geometry is then added and copied at the four end pipes (Figure 2g). A repeated cut operation opens all the ends in the five flanges (Figure 2h). The final body comes out by adding fillets between flanges and pipes and cutting the conic feed end around the two holes highlighted in Figure 2i. The same value (0.001) for the modeling tolerance (relative accuracy) has been set in every environment.

The approach described above has been easily followed within CATIA and Unigraphics models by using simple-to-use and full-of-option commands.

In the other environments, more ingenuity has been necessary. In ThinkDesign, for example, the fillet between the conic feed end and the main body with a specified value failed. Thus, a smaller fillet has been chosen and applied also to the models created in the other CAD systems. In Rhinoceros a command to hollow a solid does not exist, so the use of offset surfaces has been necessary.

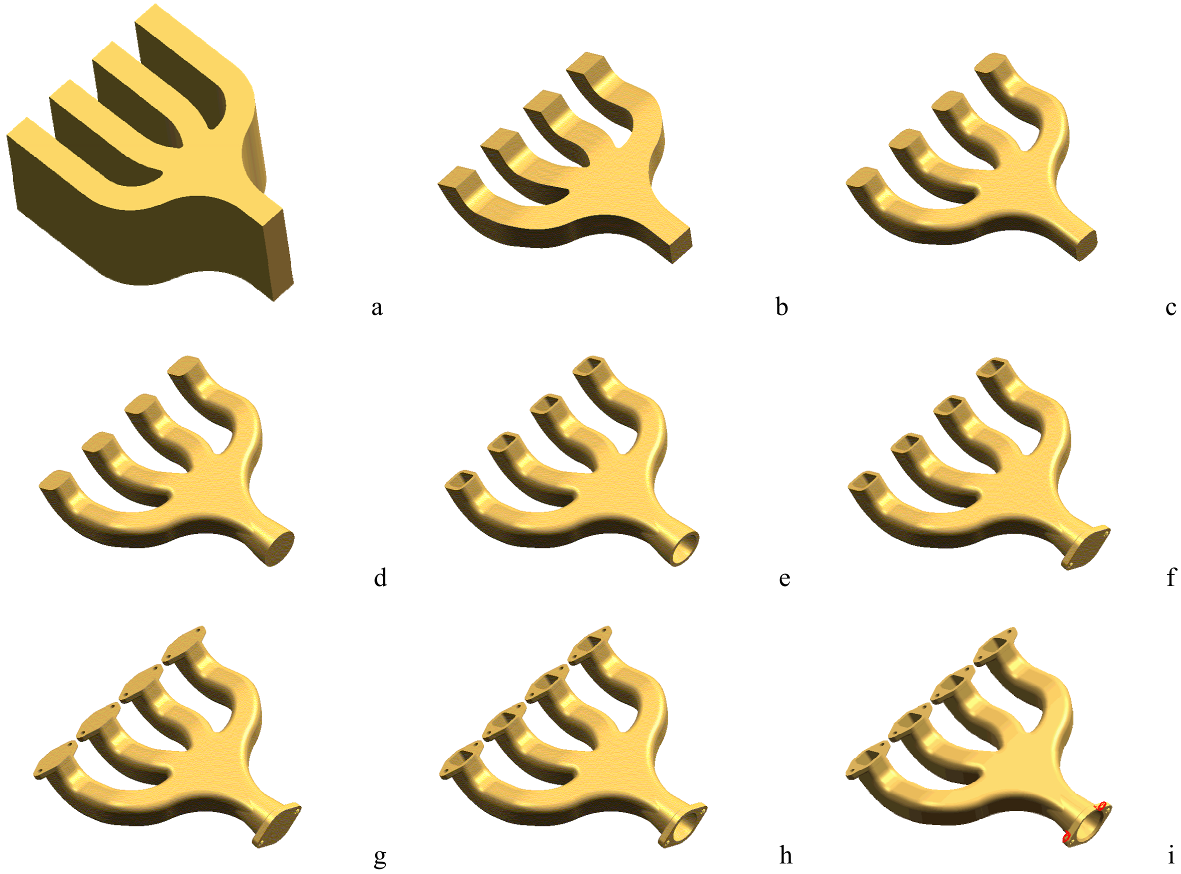


Figure 2. Main CAD modeling steps of the exhaust manifold

Moreover, with solids some problems occur when using Boolean operations involving geometries with some edges almost coincident. In Pro/E, in order to respect the fillets, the use of surface features has been necessary. Solid Edges and Solid Works do not have a direct method for performing Boolean operations by creating geometry all within the same file.
Table 2 reports some final data of the native CAD model in each environment.

Table 2. Select final data of the native manifold model in each CAD system

	CATIA	Pro/E	Rhinoceros	SolidEdge	SolidWorks	ThinkDesign	Ugs
Volume (in^3)	67.895	67.899	67.902	67.902	67.568	67.512	67.905
Area (in^2)	550.413	550.417	550.415	550.415	548.334	547.742	550.441
# of patches	737	719	422	745	737	735	700

5. Setting of conversion parameters

Before starting data transfer, a careful choice of the conversion parameters in each environment should be done. There are many setting parameters that control data exchange between CAD systems. According to their purpose, they can be grouped into the following three typologies:

I) parameters that control entity geometric mapping
II) accuracy parameters

III) parameters for repairing and sewing geometric entities.

The first typology is relative to the correspondence between entities of the native CAD format file and those of a neutral format file. The second typology concerns the tolerance of some conversion operations, like entities approximation or surface sewing. The last typology comprises parameters that make active healing operations to repair geometries and rebuild the solid model successfully.

Generally speaking, after choosing the appropriate Application Protocol (AP) – AP203 or AP214 – in the sending and receiving system, STEP processor interfaces do not require to setting mapping parameters because of the constraints imposed by the ISO 10303 International standard. Among the CAD systems tested, only the Unigraphics' STEP pre-processor allows setting a tolerance for curve/surface b-spline approximation.

IGES processors permit the setting of a wider range of parameters, depending on the system, because IGES has many more geometric entities and CAD vendors usually adopt several mapping strategies to convert native data into IGES data. Most IGES parameters regard curve/surface mapping and their approximation tolerance. In this work, these entities have been converted preferably into b-spline form (IGES #126 and #128), and trimmed surfaces into IGES #144 entity. The b-spline approximation tolerance has been set equal to the modeling tolerance adopted in each CAD system (0.001).

Instead, IGES post-processor parameters are primarily concerned with surface healing and sewing. Before importing any geometry, every option that permits those operations has been activated. Most of the CAD systems analyzed allow joining surfaces if the distance between edge curves is within a specified tolerance. In the environments where it could be set (CATIA, SolidEdge and Unigraphics), it was set equal to the common modeling tolerance. This value has been gradually increased (to 0.01) when the sewing operation initially did not turn out well. In the SolidEdge post-processor interface the option activate the simplifying and healing operation is available. In Pro/E it is possible to drastically decrease many errors in the imported model by setting on the options *fix_boundaries_on_import* and *intf3d_in_close_open_boundaries*. Lastly, in converting the trimmed surfaces with ThinkDesign post-processor the "cut surfaces with UV curves" option permits the solving of most of the problems that may occur with the "cut surfaces with 3D curves" option.

6. Model check and healing tools employment

In this paragraph the experimental work will be exposed describing, for each CAD, which problems occurred when importing models; which healing operations have been performed and whether they turned out well.

In CATIA many IGES files were imported as no-sewed surfaces. By using its check and healing tools the presence of gaps has been highlighted and their closure has been done. The unconverted surfaces have been easily reconstructed from the neighboring edge curves (always converted) by setting tangency conditions with the boundary surfaces.

Even in Pro/E environment many imported IGES files have not been reconstructed as solid model. The Check Geometry module has succeeded in describing problems and advising the appropriate healing methods. When there were gaps, it suggested acting with manual healing to zip gaps, whereas in the case of degenerated surfaces it advised entities regeneration. Often, by coupling manual and automatic healing, a correct model comes out.

Regarding Rhinoceros, it has already been noted that the model check module has a strong capability for pointing out errors within the model being exported. The presence of those errors could negatively influence the data transfer. An imported IGES file may often contain naked edges as alerted by the geometric inspector. The user guide suggests several methods to eliminate those free unmatched edges and, when it does not work, for example when naked edges appear as a point or as a loop, the guide proposes rebuilding edges by just using the rebuild command. The flexibility both of these healing tools and the surface modeling commands has permitted the generation of surfaces very similar to the original ones.

SolidEdge's Geometry Inspector is able to point out errors related to the imported models in a precise and detailed way. Nevertheless, when the model does not successfully pass the check and repairing actions made during the importing phase, it becomes quite arduous to heal the model due to the lack of automatic repairing tools, and to the difficulty inusing rebuild commands. When these problems occur,

it may be preferable to import the models again with the no-sewing option activated. Moreover, the sewing operation may require building some guide curves and this may complicate the regeneration of the model. Lastly, a sewing tolerance value which is too high could lead to a degenerate geometry.
In SolidWorks the diagnostic tools are very easy to handle. In a single window it is possible both to visualize errors (gaps and degenerated surfaces) and to act in order to correct them. The results have always been successful, except when geometry was altered significantly after conversion.
As mentioned in a previous paragraph, ThinkDesign does not have an efficient check and diagnostic tool for imported models. When tiny errors have been found, time-consuming actions have been necessary to identify areas requiring repair. Where the link between surfaces and edge curves was lost, it has been possible to act with global modeling commands to reestablish this link. However, these tools are not able to manage continuity conditions with neighboring surfaces. This lack could lead to deviation between volumes of original and imported models greater than 0.1%, which is the volume limit imposed to consider the conversion task valid. Other surface rebuilding commands require skilled users and they could lead to undesired effects.
Unigraphics' Geometry Inspector has never found errors both within the native model being exported and within the imported models. Nevertheless, it has not been able to point out some critical problems, related especially to bad trimmed surfaces imported via IGES and corresponding to original toroidal or conic shape surfaces. It has been necessary to modify boundary edges and use them to regenerate the surfaces, but these actions sometimes have led to bad results, since percent deviation was at the end greater than 0.1%. Some problems with trimmed surfaces have occurred also with models imported via STEP. In this case a degenerate solid with wrong faces has been generated. Repairing solids has not been always possible, due especially to edge curves lost.

7. Final results

Table 3 resumes the results of the data exchange via IGES and STEP neutral formats. The matrix element Aij expresses the judgment about the conversion from the "i" CAD system to the "j" CAD system. The legend below the table explains the symbols used. The quality of a data transfer is valued with the number of surfaces correctly converted and with the volume deviation [(Vj - Vi) * 100 / Vi].
Looking at the table 3, related to data transfer without any application of healing tools, it is clear that the interoperability issue is still open with neutral exchange translators, especially with IGES. Using of healing tools is essential to achieve higher quality solid models.
Table 4 shows the results of the data translations with healing tools. As it can be seen, those instruments often succeed in repairing geometry. The cases, where rebuilding the exact solid model has not been possible, were due to their inefficiency or to excessive alterations of the imported model.
Finally, some considerations about the file size. With STEP usually the file size decreases 10-30% relative to the original one, and it is lower than IGES file size, depending on the processor and the CAD systems.

8. Brief final considerations

Many improvements have being made by CAD vendors in the last years as demonstrated by the better results obtained by the authors who made a few data exchange tests with new the releases of some CAD systems (CATIA V10 and Rhinoceros 3). Several CAE and CAM manufactures are also implementing new and more efficient check and repairing tools for CAD models coming from a wider and wider range of CAD systems.
As mentioned in the introduction, neutral data exchange is not the only way to do translations. Geometric modeling kernel-level data exchange is becoming the most interesting approach to follow as many CAx software are based on ACIS (by Spatial Technology Corporation) or Parasolid (by EDS) modeling kernel. Very recently Parasolid has been considered the world's leading modeling kernel in the global CAM/CAE software industry. Eight of the top ten CAM and CAE software vendors has released Parasolid-based software products and this enhances interoperability between manufacturing, analysis and product design.

Table 3. Results of the benchmark without using healing tools

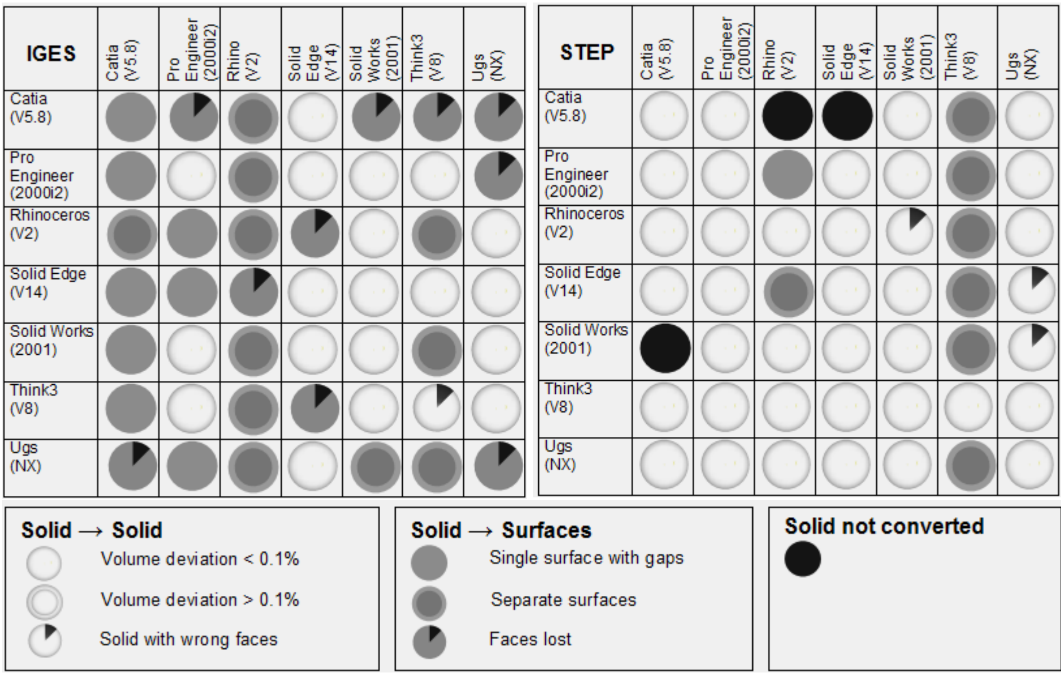
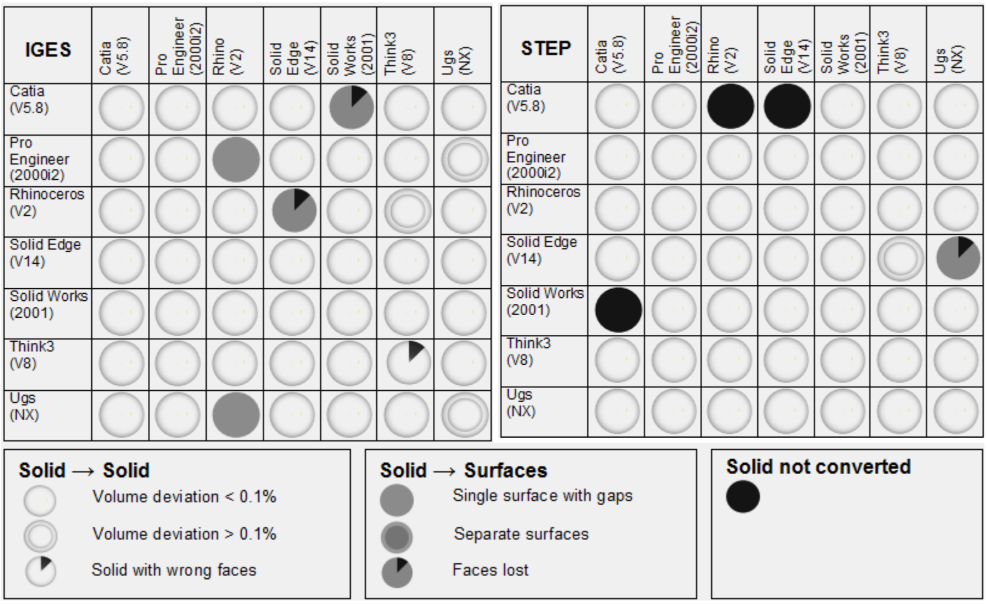


Table 4. Results of the benchmark by using healing tools



EDS offers also Parasolid Bodyshop, a toolkit used to automatically heal inconsistencies in models that have been imported from non-Parasolid-based systems. On the other hand, to guarantee an efficient data exchange between native and non-native formats, ACIS by Spatial Corp., a Dassault Systèmes S.A. company, has created 3D InterOp translators which offer advanced 3D data interoperability capabilities among several CAD/CAE/CAM software via ACIS format.
STEP tools Inc. (www.steptools.com) offers a free-of-charge translation service from/to STEP AP203/214 to/from ACIS or Parasolid format, thereby confirming the importance of the translation from/to the legacy format of the two most common CAD geometric modeling kernels available on the market. A validation tool for STEP AP203 and AP214 files is also available.

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