Compatibility of Data Transfer between CAD Applications

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Abstract: *Compatibility of data transfers between individual CAD programs is topical when in another program we need to use a drawing or a 3D model created in a program that uses a different modeling kernel. The paper characterizes features and structures of existing CAD formats, problems in their transfer, it notes the export/import conversion through the IGES, SAT, STEP and STL formats. It takes a closer look at using the STL format for visualization and animation.*

Keywords

Data transfer, compatibility, data format, Boolean functions, optimization, faces, objects, animation.

1. Introduction

As part of our grant projects we created an animation of an alternator and a starter. Individual parts of these machines were modeled in PRO/Engineer and the scene and movements were created in 3ds max. The problem lies in transfer of 3D models, that is, in compatibility of data formats. When using commonly advised formats it was possible to create visualization, but Boolean operations could not be used because the models consisted only of areas, and in some formats surfaces were even missing and had to be remodeled in a complicated way. Our task was, besides animation, to find the best way - an appropriate format - of data transfer. There is no valid standard in this field that all users could conform to. The basic problem of data exchange between CAD systems is not the transfer itself but the question of what type of data should be used so that the program can process it. STL format does not belong in a category of currently recommended formats for a data transfer between 3D CAD packages. Either STEP or SAT formats are used for the data exchange. Selection and verification of the STL format for 3D solids exchange is a novel approach in the design practice. It has a great benefit for communication between software packages possessing different modeling engines.

2. Data Transfer

One of the key problems of computer science is data input. When transferring data we need to know what data the CAD systems actually transfer. [1]

Geometry - When transferring an entity such as line, circle or point, there is no problem. It applies even to general curves. It is different for solids or surfaces – it does not concern just geometric definitions, but about special data operations working with multilayer management of many definitions of mentioned lines, circles or curves. This management has a specific “language” for each particular system that is part of its know-how.

Drawings - Transfer of drawings even in one workplace is not necessarily all right – export and then import of the same data format even in the same system is not necessarily all right.

Bindings - To transform all “hidden” information between systems – e.g. project parameters, structure of entities, etc. – is really very hard. One of the ways would be a totally new data format or public data format of all data structures.

Conclusion - When we transfer not just geometry or drawings, but a whole project, the scenario of transfer is a very complicated task. (To get information about a version of data format printed drawing, etc.)

3. 3D Curves

Objects in computer graphics are usually sets of points limited by surfaces, eventually by curves in a plane. Curves and surfaces are best represented by functions, and functions are difficult to input. The task of a CAD program user is usually to input only several control points and the mathematical apparatus (from which the user is shielded) creates the curve itself. Curves are usually represented in a computer as sets of parameters of certain equations that are then generatively displayed. This quantification can be

explicit – $y=f(x)$, ,
implicit - $F(x,y)=0$,
parametric – $P(t)=[x(t),y(t)]$ - the most frequent case.

The basic element of theory of curves in computer graphics are polynomial curves ($P_n(t) = a_0 + a_1t + ... + a_nt^n$). They constitute partially polynomial curves, e.g. curves whose parts are polynomial curves (*Uniform non-rational cubic B-spline, non-uniform rational B-spline curves - NURBS).*

Most frequently used are the third order curves – cubic, which offer a wide enough scale of shapes. Modeling is usually carried out by defining several control points (control polygon) and the mathematical apparatus determines the shape of the curve from their positions. There are two basic types of interpretation of control points. These are *interpolation* (Hermit and Catmul-Rom spline) and *approximation (Bezier curves, Bezier cubic)*. In the case of interpolation, the generated curve intersects the given points, whereas in the case of approximation, the control points determine the shape of the curve. The curve, however, does not have to intersect them. An important condition for choosing individual curve options is the rate of smoothness and continuity. In the field of CAD/CAM we most often use the third order curves, where continuity of the order C^2 [2] can be ensured. Curves of more complicated shapes are obtained by connecting parts (segments) of the curves together. The connecting points are called *nodes*. What is important, however, is the way in which they are connected, the so-called *parametric continuity* at a node.

The above-described curves are frequently used in 3D due to their characteristics, especially for generating surfaces and for constructing 3D objects. An important modification of 3D modeling was the use of rational Bezier curves (and areas) and in particular their special instance - rational Bezier curves with non-uniform parameter adjustment, so-called NURBS. These methods allow the generation of classic geometrical shapes (such as spheres, cylinders, conic sections etc.) using approximation methods. By means of known operations (extrusion, revolution, ruled) NURBS curves are most frequently used in this process of creation.

4. Formats for CAD Programs

Data exchange formats used in industry so far, IGES, VDAIS, VDAFS, SET, are usable for changing product information as a technical drawing or simple geometric models. For exchanging complex information containing complex data about properties of used materials, tolerance, bills of materials, used technologies and other information for production planning, they are not suitable. These drawbacks have to be eliminated by using a complex information product format for the whole lifetime, called STEP [3]. This format is standardized in the ISO 10303 certificate and is still being developed and improved. Parts of the STEP format are besides model descriptions also methods of implementation and verification.

Product information files are stored in the data reports according to predefined rules and these reports are called application protocols (AP). For CAD AP 203, for example, is used.

A very old and suitable system is IGES, which is no secret to CAD systems designers. It allows selective reading, but it does not include methods of solid and surface description. There is an unofficial standard for surfaces, but the situation is complicated for solids, because rendering kernels of applications are continuously developing.

In recent times, there is a special format for storing solids called SAT (vector data format of ACIS CAD application) added to CAD applications. Other vector format used for data transformations between CAD products is stereolithographic format *.STL.

Transformation between *.DWG, *.IGS, *.STP and *.VDA is bidirectional and in some cases it allows users to process files made by one CAD in another CAD system. It is often used for code generation for NC machines (CAM).

4.1 Formats Used for CAD Drawings

Graphic editors and designers have their own output data format, e.g. AutoCAD has *.DWG. The structure of this format is still developing together with the graphic editor and keeps compatibility with lower versions (the new product can read drawing from lower versions without problems).

At the beginning, drawing conversion from one product to another was generally solved by using AutoCAD's (Autodesk) text format *.DXF. It is possible to generate drawings in *.DXF format for drawing 2D objects in most CAD products. Possible changes are usually made automatically or it is possible to change the text file manually or with a simple text editor. Conversion using *.DXF is also possible for drawings with 3D objects, but due to the use of different objects in different products it is not necessarily successful. During this process some objects can be lost, e.g. those which AutoCAD does not use.

The situation is more complicated for drawing volume 3D objects [4]. Nowadays, software companies mostly use two different modeling kernels – ACIS and Parasolid. Volume objects – solids – are stored in the form of called boundary models or in forms, where some (or up to all) information about steps of object creation – so-called history, is stored. Storing all volume object creation history places enormous demands on the storage scale and speed of display processing, while storing just boundary models has a smaller scale and is displayed fast. However, a boundary model does not allow the use of a vector record of the original model part for additional changes. Because Autodesk uses the ACIS modeler for AutoCAD, *.DXF format is not suitable for transfer to and from products with the Parasolid modeler.

Storing the history of a solid creation allows another type of change to the final solid – so-called parametric modeling. Single parts of the final solid are modeled as protuberant, pulled out or rotated draft, where they are (or can later be) dimensioned and equipped with bindings between single elements – lines. The same is true with trajectories, pulled-out, pull etc. For solid editing, even various surfaces can be used – a plane, surfaces made by definition, surfaces made by approximation etc. Dimensional and binding changes can be made whenever and lead to a change to the modeled solid.

For exchange of drawings made in various graphic editors and modelers, IGES, STEP a VDA/FS formats have become popular. Because transformations of drawings containing volume solids bring various complications, there are other formats, for example SAT modeler formats, ACIS or formats for stereolitography STL (stereolithography, rapid prototyping equipment, a 3D plotter makes a 3D model, for example, from fast hardening plastic, laying areas cut from stickers, contour lines etc.).

Format transfer is usually implemented directly into the graphic editor and usually allows import or export of some bitmap formats and input and output transformation text formats, usually *.DXF and *.IGS formats. Transfer into other CAD systems is done through these formats.

Export and import from CAD products to common exchange formats (DXF, IGES, STEP, VDAFS, SAT, STL) is steadily expanding. CAD products are equipped with transformation modules that have been improved, so that transformation is exhaustive and flawless. Solid parametric models are usually only exported to some parametric modeled products. While exporting to other formats, parametric models are substituted by boundary models.

4.2 Data Exchange Formats Used for 3D Data Transfer

IGES - (*Initial Graphics Exchange Specification*) is probably the most common standard for transfer from 2D drawings to 3D models between different CAD systems today. Unfortunately, the transfer of 3D models has some restrictions. Possibilities of IGES end in the spline curves and surfaces based on NURBS (*Non Uniformity Rational Bspline*). Generally, it is not possible to transfer a whole 3D solid model, but only the surface of this model made from trimmed surfaces. For transformation with IGES format the IGES solid model pays by losing its own internal logic (model topology) and besides, there is a problem with different precision of CAD systems.

STEP - (*Standard for the Exchange of Product Model Data*) is a relatively new format for CAD data exchange. Besides, clear 2D and 3D data can also read further information about the product. Transformation of 3D models is easier and the results are usually better than with IGES. Thanks to its ability to transform closed solid consisting surfaces *(volume)*, there is no problem with non-continuity of a transformed 3D solid. The internal structure (topology) of a volume model *(solid)* is also lost even in this case. STEP format is standardized in ISO 10303 and exists in two versions, STEP AP214 and STEP AP203. STEP is probably the most progressive format for transforming CAD data.

VDA-FS - (*Verband der Deutsche Automobil-hersteller – Flächenschnittstelle*) has its origin in the German car industry. VDA-FS was developed for transformation of 3D free form surfaces. This format is rare in our environment.

STL - (*Stereolithography*) was developed by producers of stereolithography machines. While transforming by this format, the volume model is substituted by small surfaces of its surface. Outside of stereolithographic machines, it has not expanded. (For our conversion STL format is the best, evidently).

For 3D transformation, you should use formats able to carry out transformation of full volume model. Exchange formats such as STEP or IGES should be used just in case there is no other solution (Catia-Mechanical Desktop).

4.3 Data Exchange Formats Used for 2D Data Transfer

DXF - (*Drawing Exchange Format*) was originally made by Autodesk, but thanks to its public definition was taken over by most producers of CAD systems. This format is also supported in many drawing and office applications, such as Corel Draw, Paint Shop Pro etc. DXF format can store 2D drawings and 3D wire solid models.

DWG - (*Drawing*) is not actually a data exchange format. It is a closed format from Autodesk. Considering its expansion, other companies started to support this format and their effort for transformation of DWG format to an open platform - the Open-DWG Alliance. This format could be a standard in the area of 2D data transfer.

4.4 Data Exchange Formats for CAD systems Based on the Same Model Kernel

If the used CAD systems are based on the same modeling kernel, there is the chance to use the advantages of one of these formats. This way of data exchanges gives the best results qualitatively. It is possible to transfer directly volume models without internal logic lost in the area of 3D models. It is the most perfect way of 3D model transfer.

SAT - is a format allowing data exchange for CAD systems based on the ACIS modeler (e.g. AutoCAD).

Parasolid - (extension *.X_T) is a format allowing data exchange for CAD systems based on the Parasolid modeler (Pro/E, Solid Edge, Unigraphics).

Printing formats - These are not exactly exchange formats. If you get a drawing in one of the printing formats, you can only view it or print it directly on a printer or plotter. It is usually not possible to edit it in a CAD system.

PDF - Another format often used for exchange of 2D drawings is the PDF format. It is again a closed printing format. This format is very common on Internet. It is one of the most used formats ever.

HPGL - HPGL and HPGL2 (*Hewlett Packard Graphics Language*) are printing formats developed by Hewlett Packard for their plotters.

5. Results of CAD Data Transfer Using STL from PRO/Engineer to 3DS max

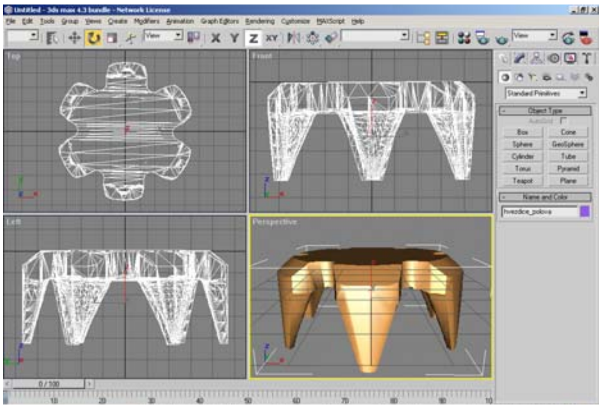

Fig. 1. Format STL loading into 3ds max – the alternator pole spider

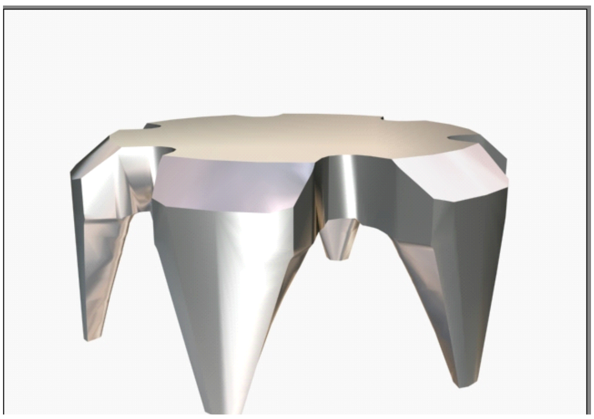

Fig 2. The pole spider – after rendering the spider is displayed as a full 3D object (from STL format).

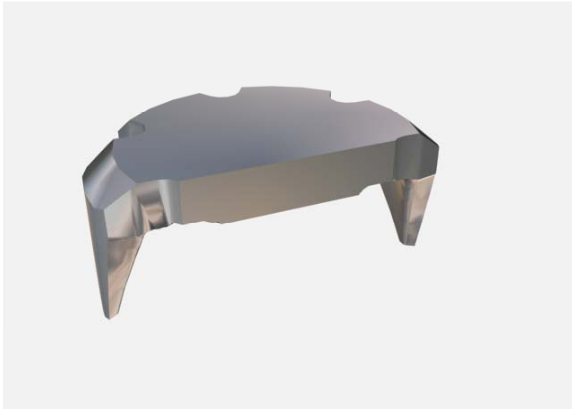

Fig. 3. The spider after using a Boolean operation.

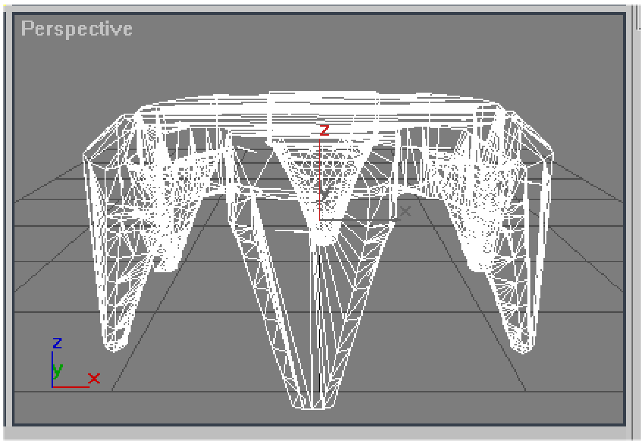

Fig. 4. Wireframe presentation in 3ds max.

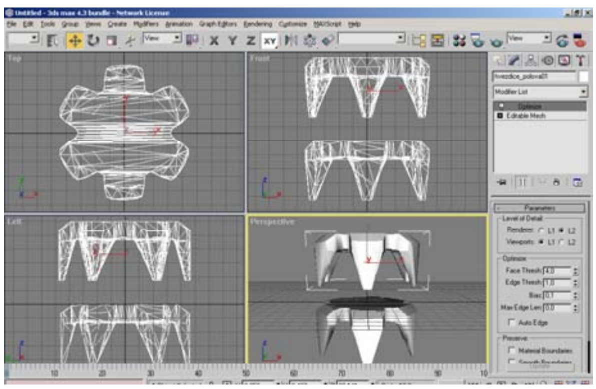

Fig. 5. The spider after optimization (change in the *FACE THRESH* parameter) – fewer small faces, faster display, but still sufficient both for visualization and for creating *COMPOUND OBJECT*.

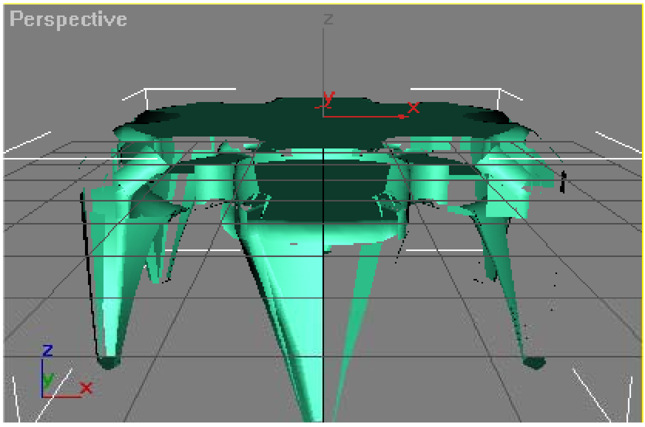

Fig. 6. IGS format loading – some surfaces are missing, unusable Boolean operation – improper for additional editing.

6. Conclusion

Conversion of the STEP, IGES and STL [5] formats obtained by Pro/Engineer into the AutoCAD 2002 and 3ds max 4 formats was tested on individual parts of a compact alternator provided by Magneton Kroměříž (a pole spider was chosen for the purpose of demonstration). Particular parts of the alternator were designed using Pro/Engineer software. To prevent repetitive modeling of the objects in 3ds max there was a necessity to find out a way of well qualitative data transfer. It concerns not only the data exchange between mechanical CAD systems, data quality is important for all CAD programs (Autodesk Inventor, Catia, OrCAD, Rhino, EasyCAD, …).

When using the STEP format from Pro/Engineer to Mechanical Desktop (and eventually using conversion through STL or IGS into 3ds max) it was possible to render the object and hide or add material, but Boolean operations cannot be used nor can REGION be created (again usable for further 3D operations). After exploding these entities are surfaces. For further work in 3ds max this method is not advisable. When using the IGS format directly to 3ds max some areas are missing in the display. Their additional drawing for components with such a complex geometry would be very complicated and time-consuming. PRO/Engineer does not allow export into the SAT format.

The best format for our purpose seems to be STL format – after import to 3ds max it is possible to apply Boolean operations to this entity (union, subtract, intersect), to apply new materials or textures. For further creation, it is also possible to use the *EDIT MESH* function, which allows you to edit surfaces or vertexes, without which it would not be possible to create other modifications. The resulting file is relatively small.

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References

[1] TOMIS, M. Datové přenosy mezi CAD/CAM aplikacemi. *itCAD Journal,* 2001, vol. 10. no.3, p.3-5.

[2] STEINBERG, B., RZDAN, A., FARIN, G. *Reverse Engineering Trimmed NURBS Surfaces from Laser Scanned Data.* Available from http://link.springer.de/link/service/journals/00170/papers/ /0016011/00160784.pdf

[3] BEČKA,J. *Použití STEP pro výměnu formátů CAD*. Available from http://www.czstep.cz/step/aktivity/cvut_99.html

[4] NĚMEC, L. *Konverze formátů*. http://forum.3dscena.cz/

[5] ČEVELA L. *Výměna dat mezi různými systémy CAD.* [HTML document], http://www.automa.cz/automa/2001/

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