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UP-DATE DIGITAL FORMAT OF THE CAD FILES IN ORDER TO FIT INTEGRATED DESIGN PROCESS

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Abstract: *Integrated design process in industry is becoming more dependent on advanced computer technologies. Whilst the fast, efficient sharing of product information is critical, poor data translation is the most publicised and costly problem in CAD data exchange. While the transfer function can be fully automatic with IGES, STEP or native formats, the transformation process is typically subjective, application specific and requires engineering judgement, and is therefore not readily automated.*

Key words: *data transfer, DXF, STEP, IGES, native CAD format, neutral format.*

1. INTRODUCTION

Using CAD data for analysis is becoming more common, but is still fraught with dirty geometry problems such as slivers, crossovers, minute edge lengths, stray points 'on the moon', wonderful clean-as-a-whistle models that are useless for meshing, patchworks of faces that attract unnecessary elements, birds nests of draughtting geometry stuck in a corner, etc. Even if binary, native CAD files are received, the engineering analyst is at the mercy of the idiosyncrasies of the CAD operator and the CAD software [1].

The data transfer can only provides the exchange of geometric entities, 'as-is'. A CAD model usually requires significant modification, or transformation to get it into a form suitable for engineering analysis [2].

Exchanging the data through standards or native formats is only part of the solution.

2. DATA TRANSFER PROBLEMS INTO CAD-CAM-CAE SYSTEMS

Whilst the fast, efficient sharing of product information is critical, poor data translation is the most publicised and costly problem in CAD data exchange. Translation problems can commonly be attributed to geometry, topology, model precision, feature recognition etc.

Depending upon the severity level of the errors encountered the only option available may be to re-create the data or the complete model.

Typical errors encountered:

- edges and faces having small gaps between them,
- sliver faces having very thin aspect ratios; difficult to detect,
- duplicate vertices and edges etc.

Inadequately trained CAD Designers/Engineers can compound the problem. There is no substitution for investing in data exchange training for CAD users [3].

There are four trends that influence the need for CAD data repair:

- CAD mixture models,
- improving CAD technology,
- legacy models,
- geometry based meshing, and increasing demand to re-use CAD geometry for CAE analysis [3].

3. 3D GEOMETRY CHARACTERISTICS

A solid geometrical model will consist of a hierarchy of volume(s), faces, edges and points. Non-manifoldness needs to be included but need not complicate the basic considerations.

Each volume will be defined by one or more shells of faces, where each shell is watertight.

Each face will be defined by one or more boundary loops of edges where each loop is complete together with some function representing the interior region (also called the embedding geometry).

Each edge will be defined by some path in space and two end points.

All the CAD systems use a tree of primitives which need to be evaluated for most purposes such as NC machining and FEM meshing process. Evaluation of these models produces a Boundary Representation.

Most solid modeling systems are based on a Boundary Representation, which follow strict rules to maintain integrity and sanity. Key to this sanity is the hierarchy. But in addition edges and points are determined by intersecting faces. This in turn enforces manifoldness; in each edge is the intersection of only two faces, each point is the intersection of only three faces or two edges.

Non-manifold properties would be: a cell wall representing the junction of two materials or two internal regions within a solid, or a ‘scratch’ edge on a surface that does not contribute to topology but is needed to specify where a load is to be applied, a ‘flap’ with no internal thickness or volume protruding out from a solid, etc [4].

While a solid CAD system only has to manage the entities it creates itself, it is not especially difficult to maintain sanity and integrity. Usually a Hierarchical tree of primitives is used to define the complex 3D model (Fig. 1).

Problems arise however as soon as one CAD system attempts to import entities from another system.

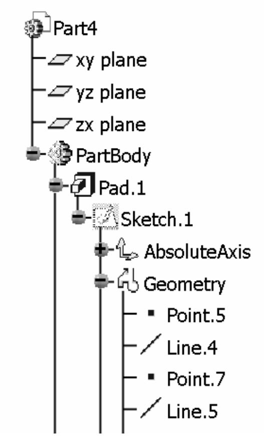


Fig. 1. Hierarchy tree of primitives into a CAD system.

Typically such modelers have a filter at the 'front door' which tidily rejects all intruders, in all entities with any form of insanity or lack of integrity is not allowed in.

Data exchange typically provides transfer of entities 'as is'.

Transfers between different CAD/CAM/CAE systems require conversions from one form of representation to another. Transfer alone is likely to be insufficient to achieve the required data exchange.

Data exchange between one solid CAD system and another, where each operates to a similar level of geometrical tolerance and use the same types of edge and face representation may well be effective with a straightforward transfer on an as-is basis. In practice, such as-is transfers produce geometrical insanity. And such insanity may be introduced by operational errors by the original CAD user, CAD modeling inadequacies or errors in data export, transfer or import.

4. NEUTRAL FILE FORMATS FOR CAD MODELS

The most used formats for data transfer between similar CAD-CAM-CAE system as CATIA, Solid Works, Unigraphics etc. are DXF for 2D representation, IGES for surfaces and STEP for 3D solids.

4.1. DXF file format

The Drawing Exchange Format (DXF) from AutoDesk is a verbose ASCII geometry format capable of representing lines, simply defined surfaces, and polygons (3D faces). [4]

The file structure of a DXF file is divided into four main sections:

- HEADER: identifying the system variables;
- TABLES: providing general information such as line style, user-defined coordinate systems, etc.;
- BLOCKS: defining instanced blocks in the model ENTITIES: describing the entities of the model.

Each section is preceded by a group code 0, followed by the SECTION string, a group code 2 and the type of section. A section is concluded with the 0 groupcode and the ENDSEC string. In each section, there are different items, each consisting of 2 lines: the first to declare the nature of the item, the second to define its value. Each item in a DXF file takes a new line; therefore even the description of a simple vertex takes up to 12 lines.

The group codes (GC) defines the nature of the value on the next line. There are general and section specific group codes

4.2. IGES file format

Initial Graphics Exchange Specification (IGES) is a neutral file format. This format enabled different systems to transfer data without the need of a specific translator on each system [4].

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 crossing two CAD programs. 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:

- Start (S) section;
- Global (G) section;
- Directory (D) section;
- Parameter (P) Data Section;
- Termination (T) Section.

In the Fig. 2 is presented a sequence from an IGES file of an ax. The shaft generated bases on the *iges* file is presented in the Fig. 3. For a better visualization some of the spindle surfaces were hidden.

The specific of the *iges* files is that generate only points, edges and surfaces.

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

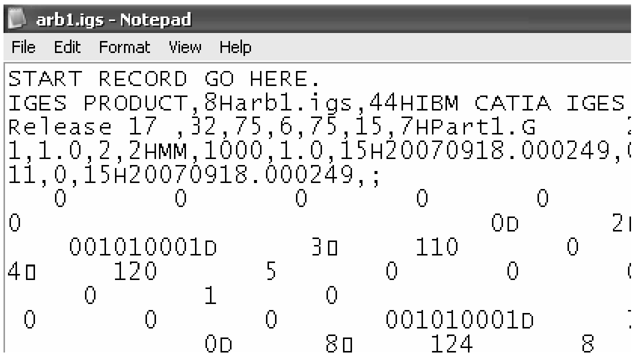


Fig. 2. IGES file sequence for a 3D shaft.

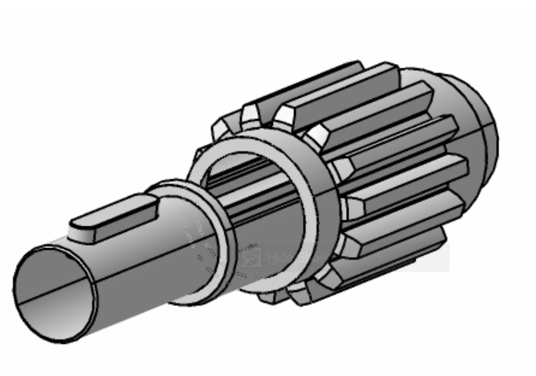


Fig. 3. A spindle generated by the IGES file.

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 [4].

4.3. STEP file format

Standard for the Exchange of Product Data (STEP) is an ISO standard industrial automation systems product data representation and exchange format [4].

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 as follows:

Description methods (11-->13). These classes describe the languages and methods that are used to create a STEP file.

Till now, only two description method classes exist:

- Part 11: The EXPRESS language reference manual;
- Part 12: The EXPRESS-I language reference manual.

The EXPRESS language is a data definition language that is used to represent the structure of data and any constraints that may apply to it. The information models contained in STEP Integrated Resources and Application Protocols are defined using EXPRESS.

Implementation methods (21-->26): describe the correspondences between STEP and other formal languages (text encoding, C++ binding, etc.).

Conformity tests (31-->35): used to check the degree of conformity of the software associated with the Application Protocol.

Integrated generic resources (41-->49): the conceptual building blocks for STEP.

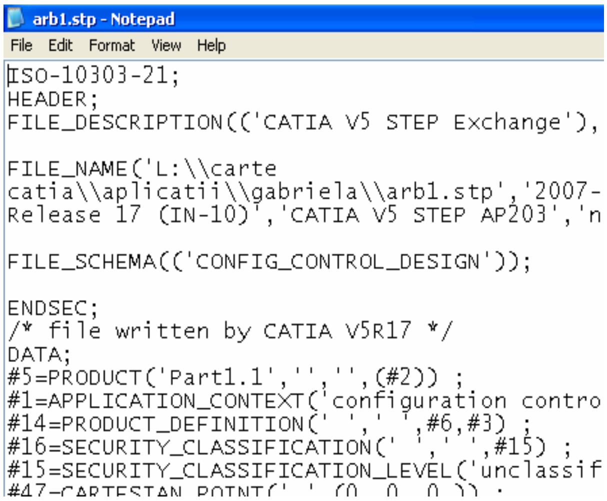
Integrated application resources (101-->106): contains the actual database, the building block of the file divided into Generic resources and Application resources.

Application protocols (201-->233): contain all the branch-specific classes.

Abstract test suites (301-->332).

Application interpreted constructs Descriptions methods (501-->518).

The STEP Application protocols are**:** +201: 2D explicit technical design; +202: 2D associative technical design; + 203: configuration of mechanical parts and assemblies; + 204: Brep 3D mechanical design; + 205: Surface 3D mechanical design; + 206: Wireframe 3D mechanical design; + 207: Sheet Metal Die Planning; + 208: Life Cycle Change Process; + 209: Composite Structures; + 210: PCA: Design & Manufacture; + 211: Elect, Test, Diagnostics & Remfg; + 212: Electrotechnical Plants; + 213 : NC Process Plans; + 214: Automotive design; + 215: Ship Arrangement; + 216: Ship Moulded Forms; + 217: Ship Piping; + 218: Ship Structures; + 219: Inspection Process Plans; + 220: PCA: Manufacturing Planning; + 221: Functional Data & Schematic Rep. for Process Plants; + 222: Design to Manufacturing for Composite Structures; + 223: Exchange of Design, and Manufacturing Product


```
arb1.stp - Notepad
File Edit Format View Help
ISO-10303-21;
HEADER;
FILE_DESCRIPTION(('CATIA V5 STEP Exchange'),

FILE_NAME('L:\\carte
catia\\aplicatii\\gabriela\\arb1.stp','2007-
Release 17 (IN-10)','CATIA V5 STEP AP203','n

FILE_SCHEMA(('CONFIG_CONTROL_DESIGN'));

ENDSEC;
/* file written by CATIA V5R17 */
DATA;
#5=PRODUCT('Part1.1','','',(#2)) ;
#1=APPLICATION_CONTEXT('configuration contro
#14=PRODUCT_DEFINITION(' ',' ',#6,#3) ;
#16=SECURITY_CLASSIFICATION(' ',' ',#15) ;
#15=SECURITY_CLASSIFICATION_LEVEL('unclassif
```

Fig. 4. STEP file sequence for a 3D spindle.

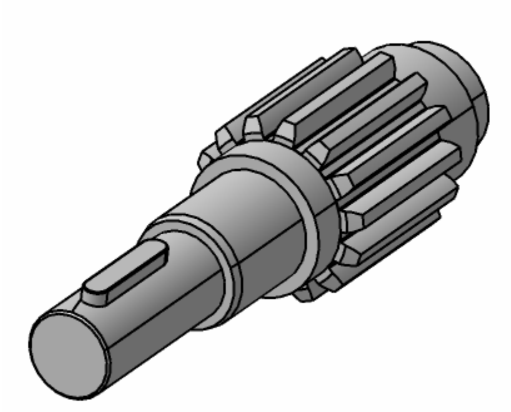


Fig. 5. Spindle case-study using STEP file.

Information for Cast Parts; + 224: Mechanical Products Definition for Process Planning Using Form Features; + 225: Strctrl. Blg. Elements Using Explicit Shape Representation; + 226: Ship Mechanical Systems/Moulded Forms; + 227: Plant Spatial Configuration; + 228: Building Services: Heating, Ventilation, and AC; + 229: Exchange of Design and Manufacturing Info for Forged Parts; + 230: Building Structural Frame: Steelwork; + 231: Process engineering Data: Proc Design & Proc Specs of Major Equipment; + 232: Technical Data Packaging.

In Fig. 4 a sequence from a STEP file exported from CATIA is presented. This file generates the shaft from Fig. 5.

Beside these two neutral formats, the designers can also use the native formats of the most known software's but with the risk of facing problems. For example, in many cases the native formats may not translate accurately into Solid Works 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 software, Catia V5 and Solid Works.

5. COMPATIBILITY OF CAD SYSTEMS

In Tables 1 and 2 the suggested 3D file formats and methods to be used when importing from a known 3D

Table 1

3D file formats saving options

	File format			
CAD System	**native**	**dxf**	**iges**	**step**
AutoCAD	1	2	-	-
Mechanical Desktop	1	2	-	3
Autodesk Inventor	1	4	3	2
CADKey	1	-	3	2
Catia V5	1	2	3	2
Pro/Engineer	1	-	2	1
Solid Edge	1	-	-	-
SolidWorks	1	3	2	1
Unigraphics	1	4	1	2

Table 2

Best format compatibility

To / From	**CATIA V5**	**Pro/ Engineer**	**SW**	**UG**
1	**2**	**3**	**4**	**5**
CATIA V5	Native	STEP, IGES	IGES / STEP, .cgr, Native (.model, .exp)	Parasolid, IGES, STEP
Pro/ Engineer	Native (.asm), STEP, IGES	Native (.prt, .xpr, .asm, .xas)	Native (.prt, .xpr, .asm, .xas), STEP/ IGES	.prt, IGES, STEP
Solid Works	IGES, STEP, .wrl	STEP	Native	Para-solid, native (.prt), IGES, STEP
Unigra-phics	IGES, STEP	.prt, IGES, STEP	Parasolid, native (.prt)	Native (.prt)

To / From	**CATIA V5**	**Pro/ Engi-neer**	**SW**
Mecha-nical Desktop	STEP, IGES	STEP, IGES	ACIS (.sat), STEP,IGES
CADKey	STEP IGES	.prt, STEP, IGES	Native (.prt, .ckd), IGES, STEP
Solid Edge	Native (.asm) Parasolid	.asm	Parasolid, Native (.par, .psm)
Inventor	STEP, IGES, STL	STEP, IGES	Native (.ipt), STEP, IGES

CAD program are listed. The numbers beside each checkmark specify the order of preference amongst the various file formats which can be exported from each CAD program [5].

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 reasons we must work in V5 R7, the best choice is to use the *step* format.

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 referred file types, in order of probability of success, which would be most helpful when import/export [5].

6. CONCLUSIONS

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.

DXF, IGES and STEP are the neutral file formats that enable different systems to transfer data without the need of a specific translator on each system.

Data exchange is widely available through standards or native formats, but is commonly only seen as a transfer task.

Data transfer only achieves the delivery of geometric entities, ‘as-is’, i.e. each entity in the source CAD model is transferred either into an identical entity in the target system or is converted to some equivalent form.

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