A Review and Comparison of IGES and STEP

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ABSTRACT
Product data exchange among different CAD/CAM/CAE systems or even animation and documentation software are of great importance to computer-integrated manufacturing CIM. This sharing of product data is also a key to successful deployment of concurrent engineering. Furthermore, this practice mainly contributes in reducing lead-time of product development. Due to the large variety of CAD systems in the market, design data exchange among different CAD systems is indispensable. Currently, data exchange standards such as IGES and STEP provide a unique approach for interfacing among different CAD platforms. This paper reviews the development in IGES and STEP by categorizing their topics of researches, key players and years of publications ranging from 2000 to 2009 access from well known databases namely SCOPUS, Sciencedirect and Emerald. Comparisons between them in respects of structures, advantages and disadvantages are discussed together. Among all neutral files available in the market, IGES and STEP are chosen because of two reasons. First, they are standards developed by standard bodies, ANSI and ISO. Second, both are extensively applied and most popular in product data exchange among heterogeneous CAD systems.

General Terms
Design, Standardization.

Keywords
IGES, Neutral File, Product Data Exchange, STEP, Translator.

1. INTRODUCTION
Products may be designed as either a two-dimensional, three-view drawing layout or as a full three-dimensional model with associated drawing views and dimensions by using a CAD system. This model developed in CAD software can also be utilized for other specific applications. The model can be used for engineering analysis through finite element software, for machining simulation and NC code generation through CAM software, for animation development through animation software and even for documentation purpose [1].

Product data are usually imported from a CAD system either from a computer-aided drafting system or a solid modeling system. All CAD systems store design results in their own system-specific data structures [2]. These inconsistent with the required input format of the application program to be used. Thus, a data communication problem often arises when two or more of CAD/CAM/CAE systems are tied together to form an application that shares common data. To solve this communication problem for pairs of systems, it is necessary to translate the product data of one system into a form that the other system can use and vice versa.

In order to make this model transfer possible, a method named neutral file is used [3]. Although by theory only one neutral file is required, there are several neutral files available in market. They have been developed by various countries and organizations. By general, these neutral files can be in categories as follows:

i) Neutral files developed by standard institutions e.g. IGES and STEP;
ii) Neutral files developed by CAD software e.g. DXF and ACIS;
iii) Neutral files developed by specific applications e.g. lithography, VRML, VDAFS and graphics.

The IGES and STEP formats serve as neutral data formats to transfer the design to a dissimilar system. Translators developed to the IGES and STEP standards are used to export a design into an IGES and STEP files for exchange and for importing the IGES and STEP files into the destination system [4].

The transfer of original file highly depends on neutral file. Some neutral files transfer all information of geometry entities and non-geometry entities. In the CAD file, the fundamental unit of design data is the entity. Entities are categorized as geometry and non-geometry. Geometry entities represent the definition of the physical shape. These include vertices, edges, curves, surfaces, solids and relations which are collections of similarly structured entities. Non-geometry entities meanwhile typically serve to enrich the model by providing a drawing definition, annotation, dimensioning, attributes and groupings. These include view, drawing, general note, witness line, leader, property and associativity entities.

The necessity to the usage of one standard in transferring CAD design is the main course in forming a standards institution responsible to the development of neutral file. This was proved when the main user namely General Electric challenged CAD producers in one conference in September 1979. Initiating from this milestone, a project known as Interim Graphic Exchange Standard is established to generate an IGES standard. The profit gained from the development of IGES has next been extended to create a new standard named as STEP.

2. TRANSLATOR AND NEUTRAL FILE

Since the features of respective software are unique and varied, they should be able to communicate among themselves. This communication means the transfer of data and minimal data loss. The solution for this problem is to possess translators among software [5]. These translators between each specific pair of the systems are called direct translators. In case each and every software needs its own translator, in other words every communication between software involves two translators. Theoretically when the number of software increases, so do the translators. This can be understood by looking at the Figure 1 below.

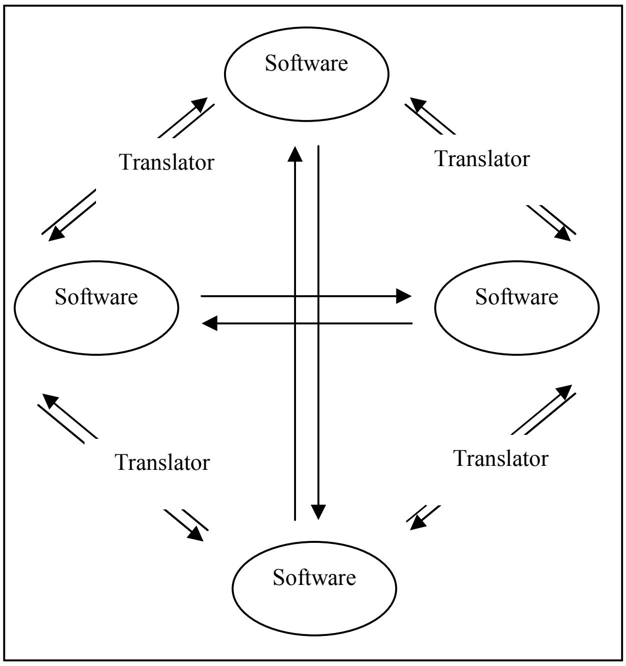

Figure 1. Data exchange using specific format

Figure 1 clearly describes that using a specialized translator is uneconomical and difficult to manage. The following equation shows the number N of two-way translators needed for n CAD systems to exchange data among them:

$$N = C\binom{n}{2} = \frac{n!}{2!(n-2)!} \qquad \text{Eq. 1}$$

Nevertheless, this problem of plenty number of translators can be reduced to only one translator by implementing a method given name neutral file [3]. This neutral file is independent of existing or future CAD/CAM/CAE systems. The latter can be differed from the previous through Figure 2 as illustrated.

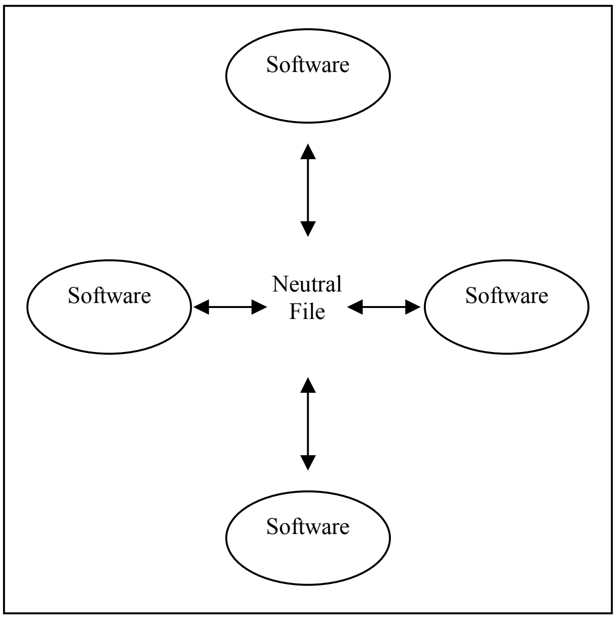

Figure 2. Data exchange using neutral file

When this neutral file is used, each CAD system needs only one preprocessor and one postprocessor which form one set of translators [5]. Figure 3 illustrates the way the pre-processor imports the data into the system and the post-processor outputs the file into standard.

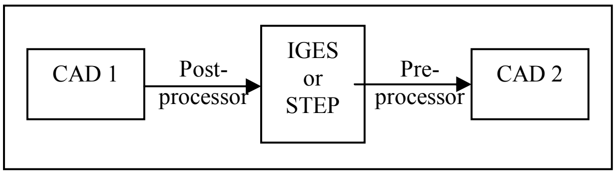

Figure 3. CAD data exchange

This indirect approach does not suffer from the disadvantage of requiring an increasing number of programs to be written as in the case of the direct approach. This is the main reason the indirect approach is accepted as a typical exchange method between different systems.

3. IGES

IGES stands for International (originally 'Initial') Graphics Exchange Standards and a neutral data format that allows the digital exchange of information among CAD systems. In 1979, a technical committee consisting of the Boeing Company, the General Electric Company and the then National Bureau of Standards (now National Institute of Standards and Technology) was assigned the task of developing a data exchange method under the U.S. Air Force integrated computer-aided manufacturing program [6]. As a result of this effort, IGES Version 1.0 was developed in January 1980 meant for two-dimensional and three-dimensional mechanical drawings. It then became an American National Standards Institute (ANSI) standard in September 1981 after been approved by ANSI Y14.26M which it is used for communicating 'Product Definition Data'.

IGES was the first standard exchange format developed to address the need to communicate product definition data among different CAD/CAM/CAE systems [7]. The early versions of IGES implicitly aimed at the CAD/CAM systems of the 1970s and early 1980s. Recent versions have extended the types of data to be exchanged. Version 2.0 of IGES in July 1982 was approved to support the exchange of finite-element data and printed circuit-board data. Along with Version 2.0 was a general committee towards ensuring that IGES can handle the graphic representations for all products. Version 3.0 later enhanced the capabilities of user-defined Macros that are essential to exchange the standard part libraries. Version 4.0 supports the CSG tree of a solid. For solid modeling with boundary representation, IGES version 5.0 was produced in 1991.

The IGES file has the format of .igs and is based on ASCII standard code. This is the reason the IGES neutral file is readable by all text editors. This file is written by sequence. At the end of each line, there is a code showing which section that line is and which line for every section. In entirety, the IGES neutral file consists of 5 sections including Flag (optional), Start, Global, Directory Entry, Parameter Data and Terminate [6][8]. These structures are discussed under Discussion and Findings topic.

4. STEP

STEP is adopted from official title of ISO 10303, an ISO standard for the computer-interpretable and exchange of industrial product data and referred to as 'Standard for the Exchange of Product Model Data' [9]. The development of STEP is an undertaking continued after the development of IGES. It originally was a major effort undertaken by European countries under ISO Technical Committee TC 184 (Industrial Automation Systems) and its subcommittee SC4 (External Representation of Product Model Data) in July 1984. This explains the reason the topology structures and geometries of STEP are same as IGES. Parallel to the STEP effort was the Project Description Exchange Standard (PDES) project in the United States in 1983. The general emphasis of PDES was to eliminate the human presence from the exchange of product data rather than the exchange of product definition data. In other words, the emphasis was to eliminate the need for engineering drawings and other paper documents in passing information about different product phases being performed on similar or dissimilar CAD/CAM/CAE systems. Both PDES and STEP had the identical goal. In June 1985 therefore, the IGES Steering Committee decided that PDES should represent U.S. interests in the STEP effort. The international standard that resulted is a combined STEP-PDES standard which is at present generally accepted as STEP.

STEP development is based on three principles. One, it has to target product data which encompass the data relevant to the entire lifecycle of a product including design, manufacturing, quality assurance, testing and support [10]. Thus, it has to consider as data the tolerance information, form feature information, the finite-element model, the kinematic analysis model and others as well as the product definition data relevant primarily to the product shape.

Two, in STEP's data structure application-specific, data should be stored in a module of the application layer separate from the generic shape information [10]. This approach ensures that the data structure is sufficient to support a wide range of applications while disallowing redundancy in the generic data structure.

Three, it has to utilize a formal language to define the data structure [10]. In STEP, the data are described in the EXPRESS language which then maps to the physical file. In this way, ambiguities can be avoided when product data extracted from the file are interpreted.

STEP has the extension of .step and is written using ASCII [2]. This implies this file is readable by most of text editors. Basically, STEP is used to exchange data between CAD, CAM, CAE, other CAx systems and Product/Engineering Data Management (P/EDM).

In ISO standard, information about digital product must be able to explain the complete product lifecycle which starts from design until the stages of analysis, production, quality control, inspection and product support. This means the information on STEP data must embrace the graphic representations as geometries and topologies inclusive of all engineering data of that product as product functions, a process plan and others. The product geometry includes drawing, features, CSG and B-rep are arranged into application areas as mechanical, electrical, electronics and others.

The STEP file consists of 2 sections including HEADER and DATA [2]. These structures are discussed under Findings and Discussions. STEP classifies standard based on its AP.

A development of an integration model and system for concurrent assembly, design and planning using STEP has been initiated by [7]. The model enables exchange of product model data in standardized format.

5. FINDINGS AND DISCUSSIONS

Along the way of this reviewing process, three kinds of findings have been successfully achieved. These findings are gathered and organized based upon the following categories:

i) Comparisons of advantages between IGES and STEP as presented in Table 1 and Table 2;
ii) Comparisons of disadvantages between IGES and STEP as presented in Table 3 and Table 4;
iii) Classification criteria of CAD data exchange as presented in Table 5.

Table 1. Advantages of IGES

Advantage	Reference (Author and Year)
Stores drawing information in an ASCII or binary neutral format which can then be exchanged between various users.	Bhandarkar et al. (2000) [8]
IGES originally supported drawings and wireframes, and was later extended to support surfaces and solids.	Rappoport (2003) [12]

Table 2. Advantages of STEP

Advantage	Reference (Author and Year)
Allows viewing and modification of geometry using any CAD tool capable of interpreting STEP geometry and breaks the dependency between CAD systems and product definition.	Bhandarkar et al. (2000) [8]
Very useful for grouping mechanical elements in certain view.	Zha and Du (2002) [7]
Included in the assembly schema containing all connector entities.	Zha and Du (2002) [7]
Easy for derivation of the implicit information contents.	Zha and Du (2002) [7]

Table 3. Disadvantages of IGES

Disadvantage	Reference (Author and Year)
Does not have a formal information model.	Bhandarkar et al. (2000) [8]
Lack of a formal information model, problems during file exchanges and manipulations, hard to understand file formats.	Bhandarkar et al. (2000) [8]
If there is an error in the IGES file, it is very difficult to find the mistake and correct it.	Bhandarkar et al. (2000) [8]
Problem of incomplete exchange due to various 'flavors' added by CAD vendors.	Bhandarkar et al. (2000) [8]
IGES does not support life-cycle information which may be relevant for engineering applications other than design.	Bhandarkar et al. (2000) [8]

Table 4. Disadvantages of STEP

Disadvantage	Reference(Author and Year)
High costing and skills are needed to standard exchange.	Riza and Yuwaldi (2002) [2]

Table 5. Classification criteria of CAD data exchange (IGES and STEP)

Classification Criteria	Reference (Author and Year)
Translator	Yanping et al. (2000) [13], Chao and Wang (2001) [14], Iyer et al. (2001) [15], Smith (2002) [16], Zhao et al. (2002) [17], J. Pratt and Junhwan (2006) [18]
Feature identification	Shin et al. (2000) [19], Bhandarkar and Nagi (2000) [11], Iyer et al. (2001) [15], Zhao et al. (2002) [17], Zha and Du (2002) [7], Fu et al. (2003) [20], Spitz and Rappoport (2004) [21], Pralay et al. (2005) [22], Dmitriy et al. (2005) [23], Xionghui et al. (2007) [24], Sunil and Pande (2008) [25], Rameshbabu and Shunmugam (2009) [35]
Network CAD/CAM	Shin et al. (2000) [19], Chao and Wang (2001) [14], Kim et al. (2002) [26], Suh et al. (2006) [27]
Visualization	Ristic and Brujic (2000) [28], Bhandarkar et al. (2000) [8], Iyer et al. (2001) [15], Zheng et al. (2001) [29]
Application protocol	Bhandarkar et el. (2000) [8], Yanping et el. (2000) [13], Shin et el. (2000) [19], Bhandarkar and Nagi (2000) [11], Iyer et al. (2001) [15], Pratt and Anderson (2001) [30], Smith (2002) [16], Kim et el. (2002) [26], Ball et al. (2008) [31]
Data Exchange	Bhandarkar et al. (2000) [8], Yanping et al. (2000) [13], Shin et al. (2000) [19], Chao and Wang (2001) [14], Iyer et al (2001) [15], Liangyu and Nagi (2002) [32], Zha and Du (2002) [7], Smith (2002) [16], Pratt et al. (2005) [33], Junhwan et al. (2008) [34]

6. CONCLUSIONS

Through this review, it is obvious that in current product development process, design data exchange among the different CAD systems is indispensable. It is also found that the STEP standard for product data exchange is popular over the legacy format such as IGES. STEP has a great influence on many engineering applications and it is widely accepted in the area of mechanical design. IGES whereas has been found to be restrictive because it does not capture data about the product through its life-cycle.

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