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REVIEWING ISO 14649 THROUGH ISO10303

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ABSTRACT

STEP-NC was designed to replace ISO 6983 G-codes with a modern, associative communications protocol that connects computer numerical controlled (CNC) process data to a product description of the part being machined. STEP-NC is a machine tool control language that extends the ISO 10303 STEP standards with the machining model in ISO 14649 or also known as STEP-NC. The combinations of implementation methods and application protocols defined in STEP and STEP-NC standards provide standardized physical file formats and data access interfaces for those physical file formats that can cover all the aspects of design and manufacturing activities. The data exchanges between different systems will no longer have barriers through the implementation of STEP-NC in the computerized manufacturing environment. It will provide the missing link between CAD to CAM applications. This paper will review ISO 14649 which is STEP-NC through an understanding of STEP which is ISO 10303 standard for the computer-interpretable representation and exchange of product manufacturing information.

Keywords: STEP, STEP-NC, ISO14649, ISO10303, CNC.

INTRODUCTION

The set of documents that forms the ISO 10303-series standard is commonly called STEP is an acronym for the Standard for the Exchange of Product data model. STEP is the result of the experience gained by the community of users and developers of standards through the initial efforts for standardizing data exchange. The purpose of STEP, as stated in the document giving the fundamental principles of STEP [1] is "to specify a form for the representation and unambiguous exchange of computer-interpretable product information throughout the life of a product".

The actual designation of the STEP standard is ISO 10303 Industrial Automation System – Product Data Representation and Exchange. STEP provides a computer interpretable representation and exchange of product data. Product data represents information about the product in formal manner suitable for communication, interpretation, or processing by human beings or by computers.

The objective of STEP is to provide a neutral mechanism capable of describing product data throughout the product life cycle independent of any particular system. STEP is a proactive effort, the focus being placed on developing a standard that caters for various user groups. Recently, STEP has been extended to cater for manufacturing data modeling and execution with an aim to filling the information gap between CAD/CAM/CAPP chain and CNCs [2].

The need for data exchange standards was originally recognized in the late 1970s and led to the development of specifications such as the Initial Graphics Exchange Specification (IGES) in the USA, Standard D'Exchange et de Transfert (SET) in France, and Verband der Automobilindustrie-Flachen-Schnittstelle (VDA-FS) in Germany. These standards all prescribe the use of standard file formats for the exchange of data [3]. While the use of such national standards represented best current practice, all required considerable effort to achieve effective results.

By the middle 1980s, it had become apparent that industry's needs would only be properly addressed by a more comprehensive international effort that would not only improve on the existing specifications, but also fulfill requirements for life-cycle product data support [4].

The following factors contribute to the perceived deficiencies of standards such as IGES [4]:

- The specification is open to ambiguous interpretation, and there are therefore variations in the quality of translator software.
- Every CAD system vendor supports a subset of the standard applicable to their own product.
- The standards are limited to the exchange of geometric information (the two- or three-dimensional shapes of objects), engineering drawings, and some non-graphical data such as connectivity.

The STEP project was initiated in 1984 with the intention to create a single international standard, covering all aspects of CAD/CAM exchange. The standard should be able to describe product data from its implementation such that it would be suitable for neutral file exchange but also provide the basis for shared product databases, for long-term archiving [3]. Major aerospace and automotive companies have proven the value of STEP through production implementations, resulting in saving of 150 million per year in the U.S [5].

STEP development is a multinational effort. At present, nineteen countries participate in the development of ISO10303 (STEP). The coordination of these efforts is accomplished through the International Standards Organization (ISO) [6]. Within ISO, there is a Technical

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Committee known as TC 184, which deals with Industrial Automation Systems and Integration. Further, within this technical committee there exists a sub-committee known as Subcommittee 4 (SC4) which coordinates and controls the development of the STEP standard throughout the world. The Sub Committee's title is Industrial Data [3].

STEP - ISO 10303

Objectives of STEP

The purpose of STEP is to specify a form for the representation and unambiguous exchange of computer-interpretable product information throughout the life of a product[7]. STEP permits different implementation methods to be used for storing, accessing, transferring, and archiving product data [7]. The STEP project was initiated with the following objectives [3]:

- The creation of a single international standard, covering all aspects of CAD/CAM data exchange.
- The implementation and acceptance of this standard in industry, superseding various national standards and specifications.
- The standardization of a mechanism for describing product data, throughout the life-cycle of a product, and independent of any particular system.
- The separation of the description of product data from its implementation, such that the standard would not only be suitable for neutral file exchange, but also provide the basis for shared product databases, and for long-term archiving.

Additional objectives of the original developers of STEP included the creation of a single international standard covering all aspects of CAD/CAM data exchange and the implementation and acceptance of this standard by industry in lieu of other methods [8]. Automatically generated process plans, often require further input from the engineers on the shop floor to ensure their appropriateness to the actual status of the resources and production policies as dictated by the manufacturing enterprise. In order to enable the process planning system to determine the most effective plan with respect to the actual available resources, it is necessary to provide resource information that reflects the status of the physical devices at the time that they will be utilized for manufacturing the part. STEP-compliant methodology will represents resources that is extended to allow the representation of machine tool capability profiles in a time-based manner while enabling users to define custom rules and policies [9].

Structure of STEP

The STEP standard is separated into many parts. These parts cover from presenting the standard, implementation architectures, conformance testing, resource information models and application protocols. The parts are called Description Methods, Information Models, Application Protocols, Implementation Methods and Conformance Tools. Figure-1 illustrates the structure of the STEP standard.

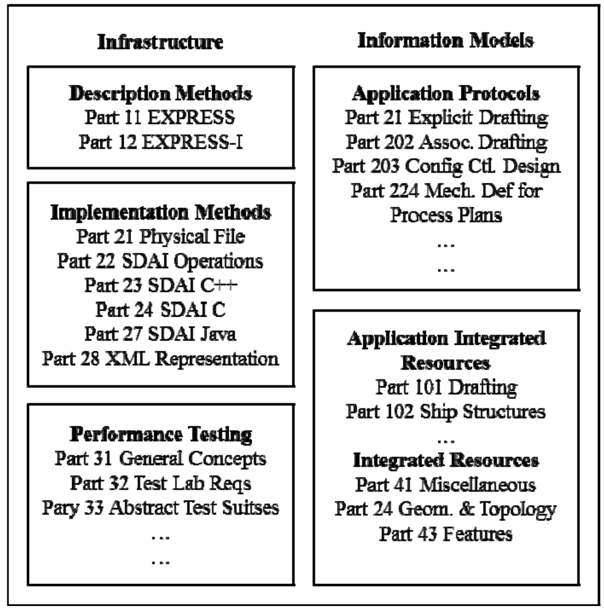

Figure-1. Structure of the STEP standard [7].

The Parts of STEP may be grouped by type as follows. The Parts are numbered so that all Parts of the same type fall in the same number range. The range is given below after the type. Not all numbers are used. There are several hundred application modules.

An application protocol can be built by including a (usually large) number of application modules. Using application modules is a more recent architectural approach than using application interpreted constructs and may replace application interpreted constructs.

STEP description method

The description methods are defined via the data modeling language, EXPRESS. EXPRESS is a completely generic modeling language and can therefore be used to model data objects of any type. It is a formal language for the definition of entity-attribute data models. Its original use was for the definition of standard data models describing 3D graphical representations of physical objects, i.e., CAD drawings.

The EXPRESS language is completely declarative and implementation independent, making it well suited for the standardized data models. On the other hand, EXPRESS is a data modeling language, which means it only defines entities and their properties, and does not define methods that might be applied to those entities in an application context[10].

The EXPRESS language does not define any implementation methods. Therefore, additional implementation methods are defined to describe STEP instances for building product exchange models. There are several implementation technologies available [11]:

- A product model specific file format called Part 21 physical file;
- A variety of programming language bindings that allow an application programmer to open a data set and access values in its entity instances. Bindings have been developed for C, C++ and Java
- The three methods for mapping the EXPRESS defined data into XML described by Part 28 Edition 1 and
- The XML Schema-governed representation of EXPRESS described by Part 28 Edition 2.

STEP Application protocols (AP)

Application protocols (APs) are the implementable data specifications of STEP. APs include an EXPRESS information model that satisfies the specific product data needs of a given application context. APs may be implemented using one or more of the implementation methods. They are the central component of the STEP architecture, and the STEP architecture is designed primarily to support and facilitate developing APs. Many of the components of an application protocol are intended to document the application domain in application-specific terminology.
This facilitates the review of the application protocol by domain experts. Below are some of the 200 series Application Protocols (APs) that are currently available for mechanical.

- AP 201: Explicit Draughting. Simple 2D drawing geometry related to a product. No association, no assembly hierarchy.
- AP 203: Configuration controlled 3D designs of mechanical parts and assemblies.
- AP 204: Mechanical design using boundary representation.
- AP 209: Composite and metallic structural analysis and related design.
- AP 214: Core data for automotive mechanical design processes.
- AP 235: Materials information for the design and verification of products.
- AP 236: Furniture product data and project data.
- AP 242: Managed model based 3D engineering (under development).

STEP AP 203 and STEP AP 214

Most of the previous researchers used generic STEP AP 203 as their input file. This research will used generic STEP AP 214 as its input file. Generally, AP 203 is a "general" STEP format. AP 203 defines the geometry, topology, and configuration management data of solid models for mechanical parts and assemblies. This file type does not manage Colors and Layers. AP 214 has everything an AP 203 file includes, but adds colors, layers, and geometric dimensioning and tolerance, and design intent. AP 214 is considered an extension of AP 203.

The original intent of STEP was to publish one integrated data-model for all life cycle aspects. But due to the complexity, different groups of developers and different speed in the development processes, the splitting into several APs was needed. But this splitting made it difficult to ensure that APs are interoperable in overlapping areas. Main areas of harmonization are: AP 214 and AP 203 in the area of 3D mechanical design. AP 214 took over all of the functionality of the earlier AP 203 edition and then extending the capabilities significantly. The second edition of AP 203 (published in 2011) took over bigger parts of AP 214 by adding again new functionality. The upcoming AP 242 will formally replace AP 203, AP 214 and other mechanical design APs. In middle 2010 the development of the new major AP 242 managed model based 3D engineering was initiated. The first edition of AP 242 is expected to be dedicated to replace the most successful STEP APs 203, 214 and other APs in the mechanical design area in an upward compatible way.

STEP implementation

STEP implementations fall into several categories. Translators take data from preexisting systems and convert it into STEP AP defined data. The tool converts non STEP data into STEP data. Other applications might take STEP data as input, and then perform some function on it, generating more STEP output. An example of such a thing would be an application that takes partial AP information from several sources, like geometry from a CAD system and configuration information from a CM system, and then merges them into a complete AP exchange file. Another category might be an application that takes specific AP data and performs some analysis on it, such as a finite element package or a geometry visualizer. These applications should all work from STEP exchange files and possibly a shared database as well, so it is important to consider how the application will be tied to the EXPRESS information models of the various APs that they will work with [4].

ISO 14649

STEP-NC

The emergence of STEP-NC is essential when talking about an information model for better CAD/CAM information sharing. STEP-NC, the manufacturing extension of STEP, annotates the design information with manufacturing data[12]. It is being developed through international effort, following the success of the international standard STEP, to provide a data model for CNC machines. A data model for basic milling, drilling and turning operations has already been developed.

ISO 14649 or STEP-NC is free of the STEP design data, or strictly speaking it was developed with an intention not to include the complete design data. The absence of design information makes the bi-directional data flow between design and manufacturing impossible. As a part of the STEP standard, STEP AP 238 inherits all the information from STEP AP 203 and AP 224 plus an interpreted model mapped from ISO 14649. Hence, bi-directional data exchange can be supported. Figure-2 portrays the relationship between ISO 14649 and some APs of STEP ISO 10303. Based on the Figure-2, the nominal and tolerance of geometrical information would be the foundation of the AP 203 and the entities for machining feature and tolerances capabilities are being included in STEP AP 224. AP 238 is an integrated resource file that covers all the relevant information regarding machining operations.

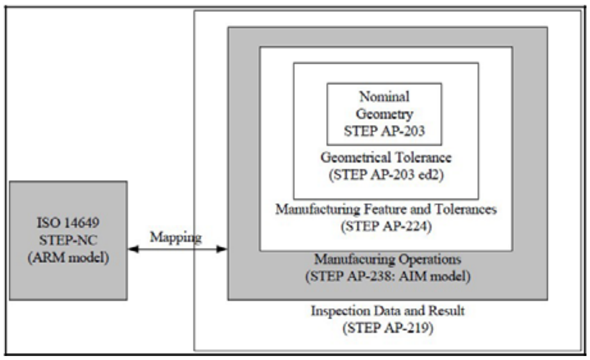

Figure-2. Relationships between ISO14649 and STEP APs [13].

ISO 14649 and ISO 10303-238 can be viewed as two different implementation methods of the STEP-NC standard. The ISO 14649 standard is more likely to be used in an environment in which CAM systems have exact information from the shop floor; whereas STEP AP 238, as a part of the STEP standard, is more suitable for a complete design and manufacturing integration.

STEP-NC provides direct input for CNC machine tools, consisting of product information such as geometry, features, machining steps and tool path [14-15]. Currently, CNC machines have to be programmed using G-codes, which only describe the exact tool movements, without any information of the part being processed. Unlike G-codes, STEP-NC tells machine what to do instead of how to do it.

Challenges of STEP-NC

Although STEP-NC is appealing and gathering more and more interest from researchers, there are some challenges to overcome before it can be realized [16]. Certain capabilities of CAM systems are moved to NC controllers, such as tools rotation information and feedrate by adopting STEP-NC. It results in much more complicated controllers and requires more knowledge from the machine operators [17]. Before STEP-NC can be practically used in manufacturing, new STEP-NC compliant controllers for NC machines have to be developed first. NC controller manufacturers then have to redesign the structures and strategies of their controllers to make it happen. There are two (2) versions of STEP-NC being developed concurrently by ISO, ARM (ISO 14649) and AIM (ISO 10303 – AP 238). The main difference between these two (2) models is the degree to which they use the STEP representation methods and technical architecture. Both versions can be viewed as different implementation methods of STEP-NC. Industry is still debating on whether to use AIM or ARM. Furthermore, many inconsistencies among standards remain to be resolved to standardize the STEP-NC data model. Therefore, a large amount of systems-building work is still needed to make STEP-NC related technology commercially viable.

CONCLUSIONS

The primary goal of STEP is to facilitate interoperability among implementations of different STEP application protocols. This is accomplished through consistent mapping of similar requirements to the generic building data structures in the integrated resources.

With the use of full fidelity STEP-NC data as direct input to a CNC machine, many bottlenecks to manufacturing productivity will be eliminated, such as cumbersome post processors, antiquated G-Codes, data redundancies and more [18]. The nature of STEP and STEP-NC data format makes it suitable not only as neutral data, but also as a platform for sharing, distributing and archiving data. By adopting a universal data standard, the issue of incompatibility among the system structure can be averted [19].

By using Workingstep as the basic entity, STEP-NC expands a set of related entity such as machining features, cutting tools and tolerance according to STEP definition. The high-level information provided by STEP-NC makes CNC programs more readable and modifiable; it also enables bi-directional data flow between CNC machines and CAM systems (or even CAD systems). Because of these technical advantages and benefits in STEP and STEP-NC, the research work reported in this thesis utilizes both STEP and STEP-NC as the underpinning product and manufacturing models.

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