STEP for E-Manufacturing: Concepts and Applications

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ABSTRACT

The ISO standard, ***ST****andard for the* ***E****xchange* ***P****roduct Data Model or STEP is known as a standard data exchange format for design and manufacturing. STEP is intended to provide a complete representation of a product throughout the product lifecycle. More recently, with the idea of e-manufacturing is being put into great emphasis; this brings the consideration of implementing STEP into that new idea. This paper presents the concepts, applications, and perspectives of STEP in the e-manufacturing environment. The objective is to provide guidelines and references for future research and implementation related to this area.*

Keywords

Design and Manufacturing, Data Exchange Format, STEP and E--Manufacturing, Concurrent Engineering, CAD/CAM

1.0 INTRODUCTION

The trend of manufacturing industry today is toward *concurrent engineering* work environment in which extensive knowledge of design and manufacturing are crucial. It is obvious that the product/process designs should have predominantly challenging characteristics in this working trend. These important characteristics are: (i) efficient communication and exchange of design and manufacturing data (ii) real time with minimal errors during this communication and data exchange (Lau et al, 2002).

Meanwhile, through the use of computer-aided design system (CAD) as well as other computer aided systems leads to automation of various design and manufacturing functions. However, these computer systems such as computer-aided design, computer-aided manufacturing, computer-aided process planning is still classified as 'island of automation' (Lau and Jiang, 1998). This is because of the fact that dissimilar formats for data representation are being used by these various systems.

For several years now, there have been extensive efforts in developing product data exchange method and standard for these different computer aided systems. The choice of data exchange method includes the translating of the system data into a form of a neutral file. Currently, they are different types of neutral file formats available, but the particular interest of this paper is the STEP standard. The subsequent section reviews concepts of neutral file formats/ standards.

2.0 OVERVIEW OF NEUTRAL FILE FORMATS/ STANDARDS

A neutral file format is a type of file with a format that is independent of any specific computer-aided system format or standard. It is intended as an 'agent' to connect dissimilar computer application systems that cannot communicate with each other due to format incompatibility (Lau and Jiang, 1998). Concepts of neutral file format can be visualized in the figure below:

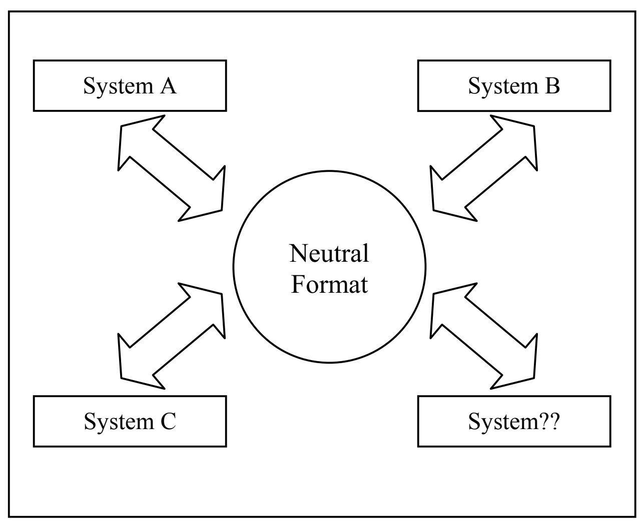

Figure 1: Using neutral format for data exchange (McMahon and Browne, 1998)

Currently, reviews show that manufacturing enterprises have been using different types of neutral file for design and manufacturing system integration. These neutral file standards are:

IGES = Initial Graphic Exchange Specification
SET = Standard d Exchange et de Transport
VDAFS = VDA-Flachenshnitt
DXF = Data Exchange File
STEP = Standard for the Exchange of Product Data Model

Method of sharing the CAD file can be visualized as a 3D CAD model being translated into these neutral files. This 3D design model is created in the CAD/ CAM system by a designer. Figure 2 provides the illustration.

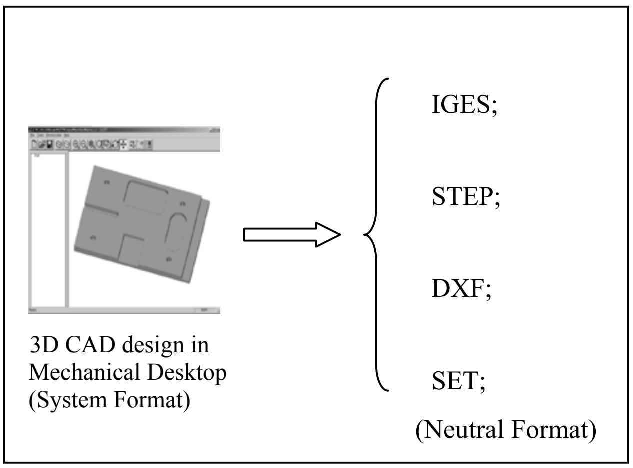

Figure 2: Method of sharing 3D CAD model into neutral file format

The first major work in data exchange was in the establishment of Initial Graphics Exchange Specification (IGES) (McMahon and Browne, 1998). However, this standard was unreliable and limited to only several applications. Meanwhile, AutoDesk Inc owner AutoCAD software developed its own neutral file format, which is known as DXF. Unfortunately, this neutral file format is limited for two dimensional draughting geometry only. Limitations of the former neutral file formats leads to a new set of standard, the ISO 10303 also known as STEP, which is intended to replace all existing standard for data exchanges. Next section provides the concepts of STEP and several works on STEP in the new e- manufacturing environment.

3.0 INTRODUCTION TO STEP

STandard for **E**xchange of **P**roduct data (STEP) is a set of international standards, designated as ISO 10303 (ISO is the International Organization for Standardization) (Shah, JJ, 2000). STEP aims to provide a common method of defining product data that can be easily interpreted and used by any application system throughout the life cycle of the product. Since the STEP application is extremely broad, it is issued in sections, known as Parts.

There are several types or classifications of Parts. The Parts formally known as Application Protocols (APs) define the context, scope and information requirements of a designated application area. The scopes of an AP are defined by: the type of manufactured product, the supported stages in the life cycle of the product, the required types of product data, the uses of the product data and the disciplines that create and use the product data (ISO, 1994). Table 1 shows some of the APs that are already achieved International Standard (IS) status and being implemented into CAD/ CAM system.

Currently there are a few of APs that received IS, examples of them are STEP AP203 and AP214. These particular APs have already been implemented into several CAD/ CAM system such as Unigraphics, Pro/ Engineer and Mechanical Desktop. Next section describes the method of generating this STEP file.

Table 1: Some of available APs for STEP

Part	Abstract Test Suite
AP201	Explicit Draughting
AP202	Associative Draughting
AP203	Configuration Controlled 3D Designs of Mechanical Parts and Assemblies
AP207	Sheet Metal Die Planning and Design
AP210	Electronic Assembly, Interconnection and Exchange
AP212	Electro technical Design and Installation
AP213	Numerical Control Process Plans for Machined Parts
AP214	Core Data for Automotive Mechanical Design Processes
AP224	Mechanical Product Definition for Process Planning Using Machining Features
AP238	STEP-NC Manufacturing

In order to start translation of 3D CAD into STEP using CAD/CAM system, a STEP exporter available in the system is usually used. For instance in Mechanical Desktop, a 3D CAD can be exported into STEP using STEPout translation function. Two different types of APs can be generated from this function.. In this CAD/CAM system, by using export function. Figure 3 shows how a design bracket can be exported into STEP using the translation function. In the case of Mechanical Desktop, user can select either AP203 or AP214 translator to generate the file from a 3D CAD design as shown in Figure 4.

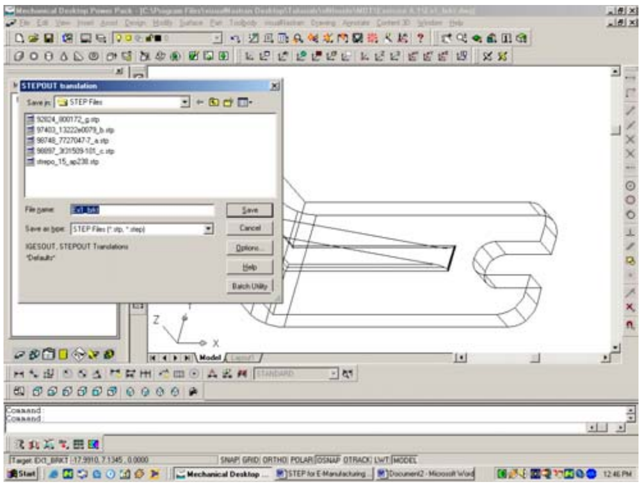

Figure 3: Exporting 3D CAD into STEP

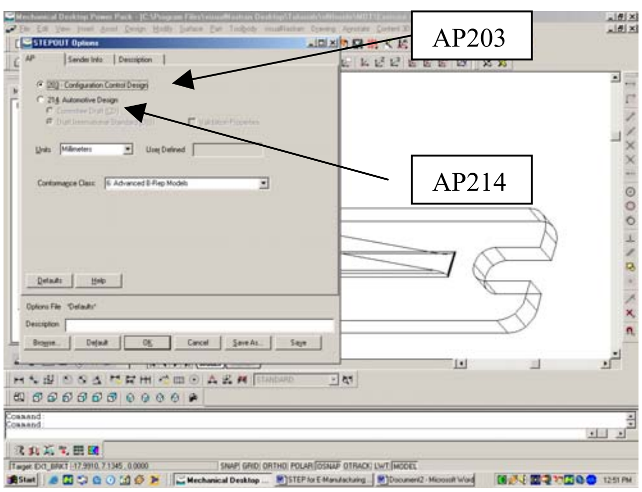

Figure 4: Choosing either AP203 or AP214 from the STEPout options in Mechanical Desktop

4.0 STEP FILE DATA STRUCTURE

This section explains the STEP file data structure of a 3D CAD model generated by CAD/ CAM system. As being explained previously, there are a few formats of AP for a STEP file. To explain the STEP data structure, an AP203 format is chosen.

The STEP file structure is a language based with unambiguous context free of grammar. The file begins with the keyword ISO-10303-21 and ends with the keyword END-ISO-10303-21. In between both of the keywords, contains entity instances which describe the object of interest or the data. The data format of entity instances is explained as follows. Each entity instance, with its own individual name, is addressed with an identifier in the form of #N, where N is a unique integer. The important data for the entity instance follows the individual name by enclosed parentheses (Liang et al., 1996). An entity instance can be a reference to another entity instance within the file, whether it is defined earlier or later.

Figure 5 shows the portion of STEP file, that illustrates the previous explanation. In this case, an identifier #44 addressed the Cartesian Point entity instance. The important data for this Cartesian point is given by the enclosed parentheses. In this example, it gives the value of x, y and z coordinates which are -0.031, -0.183 and 0.0 respectively. The same is true for other entity instances which are the vertex point, direction, vector and line.

```
#44=CARTESIAN_POINT('',(-0.031,-0.183,0.0));
#45=CARTESIAN_POINT('',(-0.031,0.183,5.84E-34));
#46=VERTEX_POINT('',#44);
#47=VERTEX_POINT('',#45);
#48=DIRECTION('',(0.0,1.0,0.0));
#49=VECTOR('',#48,0.366);
#50=LINE('',#44,#49);
```

Figure 5: Examples of entity instances and the respective addresses

In the STEP AP203 for instance, there are three types of data namely; Descriptive, Geometrical and Topological. These types of data are arranged into two major sections; Header section and Data section. In the Header section, it contains the Descriptive data. This descriptive data are providing information about the STEP translator version, CAD software used in modeling, name of the file, type of file schema or AP type and date of stamp (Bhandarkar and Nagi, 2000). Here is the header section of the STEP AP203 file.

```
ISO-10303-21;
HEADER;
FILE_DESCRIPTION(('AP203 File'),'2;1');
FILE_NAME('Ex1_brkt.stp','2003-06-
02T08:49:42',(''),(''),'AutoCAD STEP
2000i','AutoCAD 2000i',', , ');
FILE_SCHEMA(('CONFIG_CONTROL_DESIGN'));
ENDSEC;
```

Figure 6: The Header section in STEP AP203 file

Meanwhile, the Data section contains of geometrical and topological data. It consists of geometrical entity definitions and topological elements such as faces, loops and bounds as shown in Figure 7 for a simple rectangular block. Geometrical entities and topological elements are the core data of the 3D CAD model and they are nested within the Data section. For an AP203 file, the data entities and elements are for the configuration control design of a product model. This paper will not provide detailed definition of geometrical entity definitions and topological elements. The respective definitions along with their attributes are available in the ISO/TC/184/SC4 (ISO, 1994).

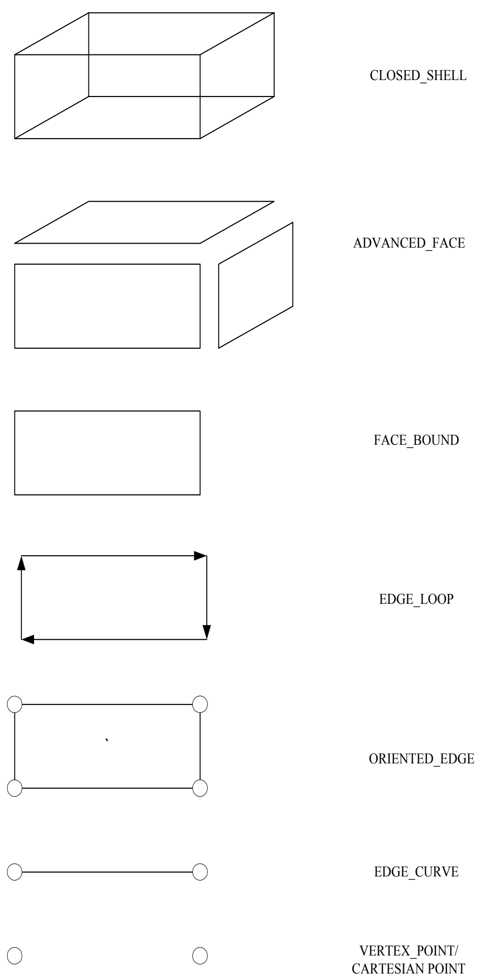

Figure 7: Entities and elements of a rectangular block for illustration of the hierarchy tree of the STEP AP203 file (Liang et al., 1996)

The entire data in this STEP AP203 is stored in a hierarchical tree structure in a “bottom-up” style (Liang et al., 1996). Each of the entities is arranged at bottom while the elements are arranged top of the entity definition. A rectangular block is used to illustrate the hierarchical tree structure. In Figure 8, it shows that the CLOSED_SHELL is represented by pointer #150, in which it is the rectangular block. Tracing down the tree, pointer #17, #19, #69 and so on shows that it represents the CARTESIAN_POINT or coordinate point of the vertices of the rectangular block. This tree is the representation of a simple rectangular block in a STEP AP203 format. For a more complex and irregular shape, the representation of the product can be done on a different STEP AP format. For instance STEP AP224 and AP238, in which they represent the manufacturing and machining aspects of the product.

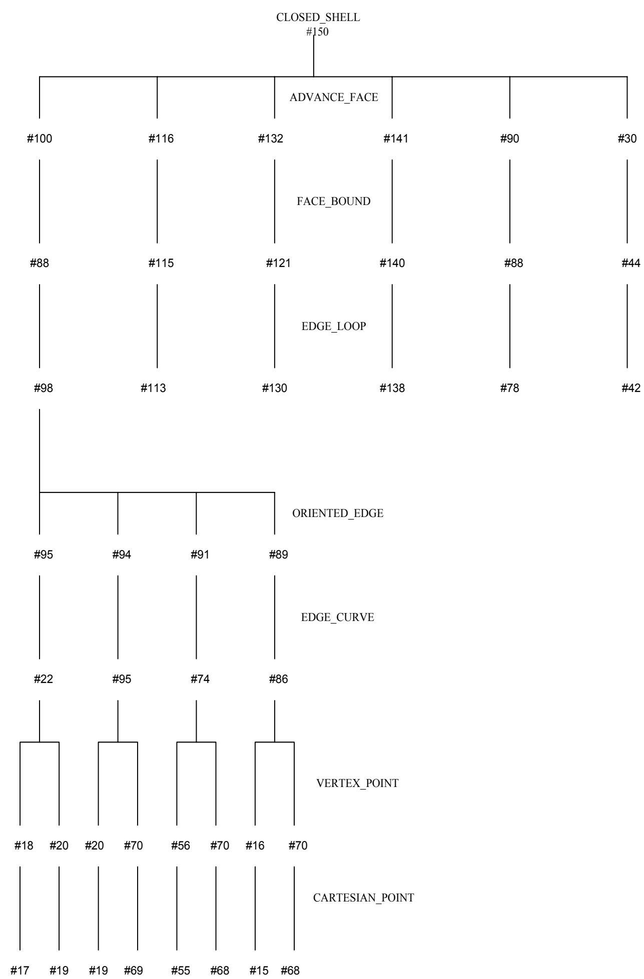

Figure 8: Hierarchical tree of the geometrical entities and topological element for a STEP AP203 file (Liang, et al., 1996)

5.0 STEP FOR E-MANUFACTURING

STEP is becoming a major force in the accurate exchange of product data and now become increasingly important for rapid, cost-effective manufacturing of quality parts. There are three STEP APs that are especially of interest to support the future need of the e-manufacturing environment (SCRA, 2001).

In this paper, the two STEP APs that are of interest for discussions in the e-manufacturing area are:

(i) ISO/IS10303-224, "Mechanical product definition for process planning using machining features," which is an IS.
(ii) ISO/WD 10303-238, .Application interpreted model for computerized numerical control (CNC)

Both of the standards are discussed in section 5.1 and 5.2 respectively.

5.1 STEP AP224 for Manufacturing Suite

STEP AP224 is a standard for "Mechanical product definition for process planning using machining features" (IS0, 2000). Traditionally, whenever a manufacturer of machined parts receives technical data from either a CAD activity or from a purchasing organization it would contain inaccuracies, most likely incomplete, may be in a e-format but is not fully compatible with the manufacturer's own computer automated tools, or is not computer interpretable which means it does nothing to expedite the accomplishment of process planning activities.

STEP AP224 resolves these problems with machine part information that ensures the design information being provided is 100% complete, accurate, computer interpretable and reusable (SCRA, 2001). AP224 contains all of the information necessary to manufacture the required through the definition of manufacturing and machining feature in it data structure. The scopes of areas for AP224 are the manufacturing and machining features definitions, features definition items, part administration data, manufacturing part properties, tolerances and shape representation (SCRA, 2001). Figure 9 illustrates these scopes of areas in STEP AP224 Manufacturing Suite.

AP224 data structure contains higher level representation of the geometrical and topological information of a product by way of these machining features. The sample of 3D CAD design as shown in Figure 10 is used as an example to generate the text form of STEP AP224 file for this part.

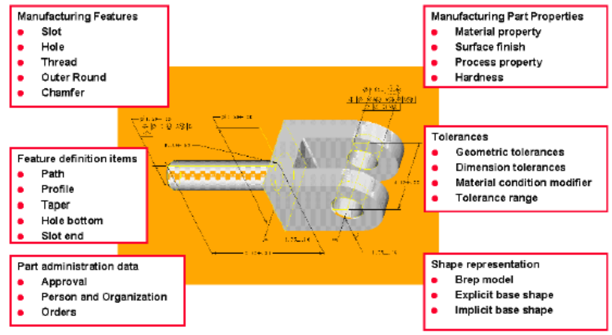

Figure 9: STEP AP224 for Manufacturing Suite Definition (SCRA, 2001)

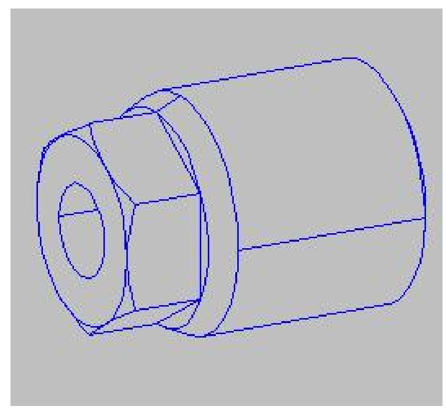

Figure 10: Part design for STEP AP224 illustration

Figure 11 is illustrating the arrangement of machining feature entities and elements in the STEP AP224 file for a product as shown in Figure 10.

```
#1054=CYLINDRICAL_SHAPE_REPRESENTATION('base
shape',(#1035,#1044,#1053),#34);
.
.
#1157=(CHARACTERIZED_OBJECT('','')FEATURE_DEFINIT
ION()INSTANCED_FEATURE()MACHINING_FEATURE()ROUND_
HOLE()SHAPE_ASPECT('','',#43,.T.));
#1158=PROPERTY_DEFINITION('','',#1157);
#1159=PRODUCT_DEFINITION_SHAPE('','',#1157);
#1160=CARTESIAN_POINT('',(0.186,1.139E-
17,0.250));
.
.#1061=(CHARACTERIZED_OBJECT('','')FEATURE_DEFINIT
ION()INSTANCED_FEATURE()MACHINING_FEATURE()SHAPE_
ASPECT('','',#43,.T.)SLOT());
```

Figure 11: Part of a STEP AP224 file for product in Figure 10

In Figure 11, it shows several definitions for machining features. These include round hole and slot features as well as base shape feature which is a cylindrical shape. The data of AP224 is also arranged in a similar way as in STEP AP203 which is in an inverted tree structure. Although STEP AP224 is used as an input for process planning, currently it is implemented along with STEP AP238 in the e- manufacturing environment. Next section discusses the future of manufacturing through the application of STEP AP224 and STEP AP238 in the e-manufacturing environment.

5.2 STEP AP238 or STEP NC

For several decades data for NC machine controllers has been defined by using ISO 6983 (SCRA, 2001). This standard defined a series of 'G and M' codes for the machine controller functions and motion. ISO 6983 only has the capability of specifying basic motion and switch commands. Figure 12 shows the current method of controlling CNC machines using 'G and M' codes. The problem with this ISO 6983 is that the language focuses on programming the path of the cutter or the cutter location with respect to the machine axes, rather than the machining tasks or what to be machine (Xun, 2003).

Besides that, this method does not use directly the high level geometric data from CAD system on the machine. Instead, a machine-specific post processor has to process the CAD data into a set of low level geometric data (Xun, 2003). Adding to this is that, CNC controller method only supports one-way information flow from design to manufacturing. Thus, any alteration in the shop floor cannot be directly fed back to the designer.

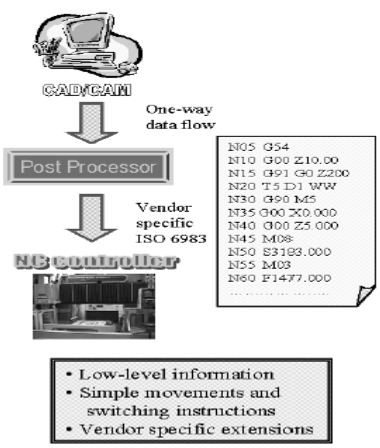

Figure 12: Current method of CNC controlling based on ISO 6983 (Xun, 2003)

Instead of trying to modernize this ISO 6983 standard, a new standard, ISO 14649, is being developed to handle the next generation NC machine controllers in the e-manufacturing environment. ISO 14649 is referred to as 'STEP NC' because of it's interaction with ISO 10303. ISO 14649 is defining product data input to a CNC machine controller using ISO 10303 geometry and AP224 features (Suh et al., 2002). As mentioned previously, the AP224 features are used to define the shape for machine operations like boring, milling, or drilling. STEP NC is being developed under the Model-Driven Intelligent Control of Manufacturing (MDICM) project awarded to STEP Tools Inc by National Institute of Standards and Technology or NIST (STEP Tools, 2003).

Obviously, STEP NC is claimed to have direct control of manufacturing tasks that have to be performed. The data of STEP NC describes "what to do" instead of "how to do it" in performing manufacturing tasks. Information contents of STEP NC are composed of: (1) *task description*, (2) *technology description*, (3) *tool description* and (4) *geometry description* (Suh et al., 2002). Since the data model of STEP NC contains tasks, it will bring a high quality of information to the shop floor. In addition, shop floor modification can be fed back to the designers enabling the two-way information flow. Figure 13 illustrates the future of the method of STEP NC controller for the next generation of CNC machines.

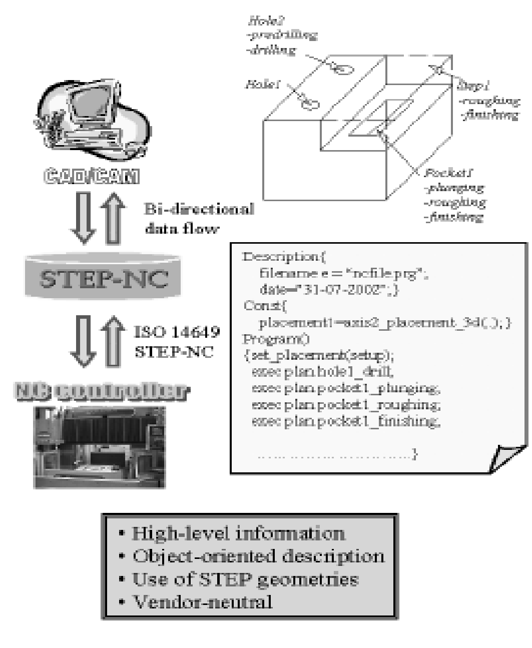

Figure 13: Future method of CNC controlling based on STEP NC (Xun, 2003)

5.3 STEP NC IMPLEMENTATION METHODS

Commercial implementation methods and prototype of STEP NC have being developed. The testing of the method is an important stage toward claiming that the method conforms to specification and achieves interoperability. As an overall process, STEP NC provides input data for a CNC machines, those are tooling specification, setup specification and processing specification. However, manual processes such as tool setting, part fixturing and machine setup are still required. Thus, the machining using the STEP NC method is a more automated process as compared to the current method. Meanwhile, the output of this new CNC activity is the machined part. Overall process is a shown in Figure 14 below:

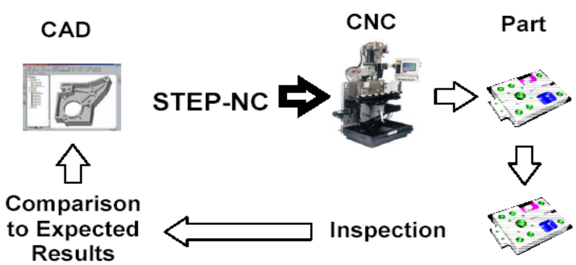

Figure 14: STEP NC Implementation Method (STEP Tools, 2003)

As a final process, the output of STEP NC machining has to be verified. Verification is evaluated against the input (the CAD model). Evaluations are based on whether the features are machined as intended, as well as whether geometry and surface finish within the specified tolerances.

6.0 CONCLUSIONS

This paper has put forward the major concepts of STEP as standard data exchange in the e-manufacturing environment. STEP AP224 is a standard for mechanical product definition using feature definition, while AP238 concerns on standard for computerized numerical control machining. Currently, research efforts have been put into integration of both STEP AP224 and AP238 for the future CNC machines that will run without NC codes. Although numerous prototypes and implementations of STEP NC (AP238 with AP2224) are available, more research is required toward demonstrating the applicability of the prototypes in the real manufacturing industry.

7.0 REFERENCES

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