THE DEVELOPMENT OF NEW STEP-NC MAPPING SYSTEM FOR STEP-NC INTEGRATION

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ABSTRACT

This thesis developed STEP-CNC Mapping System, with an aim to solve the problem faced by the current CNC systems, which is the disconnection between CAD and CAM developments for the manufacturing industry activities. This research is based on the concept that a STEP-NC program is machine-independent and has an advantage over the conventional G-code based NC program that is always generated for a specific CNC machine. The central part of the STEP-CNC Mapping System is the mapping mechanism which accepts STEP AP and translates it into the type of G-code that a specific controller can understand. The STEP-CNC Mapping System has two principal elements, a STEP Converter and a STEP-NC Translator. The STEP Converter takes a generic STEP AP 214 program as its input and produces a native STEP-NC file with a more specific version of the STEP-NC program, by "adapting" it to a local manufacturing environment and capabilities. This native STEP-NC file is a more structured populated version of the generic STEP AP file. It may be re-adapted to another manufacturing environment. The STEP-NC Translator takes a native STEP-NC file and converts it to the required machine control data, for example G-codes, that are ready to be loaded to the targeted CNC controller. The conversion process is also called a mapping or translating process that utilizes the manufacturing environment databases developed for specific machine tools, or rather CNC controllers. The system uses Visual Basic as its development tools and Microsoft Access and text file as its primary database. This system realizes the future of a common interface for CNC machine tools. With this developed STEP-CNC Mapping System in place, a generic STEP AP 214 program can be made to drive different CNC machine tools and a STEP-NC data file based on ISO 14649 Part 11 is being generated and machined.

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CHAPTER 1

INTRODUCTION

1.1 Research background

Machine tools have evolved from simple machines with controllers that had no memory, driven by punched tape to today's highly sophisticated CNC multi-process workstations, over the last 5 decades (Xu *et al.*, 2004a). Process planning has become an essential factor for enhancing the adaptability and flexibility of manufacturing systems. It deals with the selection of manufacturing processes and parameters to transform a part from its initial form to final shape according to design specifications. Computer aided process planning (CAPP) is generally acknowledged as a significant activity to achieve Computer Integrated Manufacturing (CIM). Many new generations CAPP systems have been developed recently. These systems have the advantages of implementing advanced artificial intelligence techniques like neural networks and fuzzy logic as well as an integrated product data models.

Computer Numerical Control (CNC) controllers worked as a brain for manufacturing automation and are a high value added product. CNC technology is generally considered as a measure of the level of manufacturing technology of a nation. The purpose of the Numerical Control (Newsletter) machines was to machine parts with complex shape in a precise manner. NC is the system that enables machine tools to machine parts with various shapes rapidly and precisely. NC means a control device that machines a target part by activating that servo motor according to commands

The pursuit for automated manufacturing process control started in 1950s with the introduction of numerical control machines (Kumar *et al.*, 2007). The first numerically controlled machine tool was a three-axis milling machine developed in 1952 at the Servomechanisms laboratory at the Massachusetts Institute of Technology, USA (Kumar *et al.*, 2007). The continuous development of computer technologies proved beneficiary and the first CNC machine was developed in late 1960s where the hardwired logic of previous controllers got replaced by an onboard computer (Kumar *et al.*, 2007).

The early NC machines and today's CNC utilize the same standard of programming, namely G & M Code formalized as the ISO 6893 standard (ISO6983-1, 1982). The background of the CNC equipment at that time is much like that of the computer systems in that era where punched cards and tapes were the main information medium and the computing power was only a tiny fraction of what the current computers are offering.

The next generation of NC system should be more intelligent and should be able to carry out complex tasks in a distributed environment with the current advancement of microprocessors and communication systems.

ISO 14649, a data model for computerized numerical controllers or normally referred to as STEP-NC, is the result of an international effort to achieve full interoperability and bi-directional information exchange throughout the manufacturing network. STEP-NC data model represents a common standard specifically aimed at the intelligent CNC manufacturing workstations, making the goal of a standardized CNC controller and NC code generation facility a reality. STEP-NC was targeted to replace the current G & M code standard by aiming to overcome the problems of ISO 6983 (G & M code) and take advantage of current advances in computing and controllers. STEP-NC has been designed in such a way that information remains in its context and not reduced to primitive instructions such as G & M code.

With STEP-NC, a CNC system is given a full description of the part of the manufacturing process. This description allows the CNC to perform complete safety checking before it begins to make the part, and it allows the CNC to optimize the manufacturing process for its current operating condition.

1.1.1 G Codes

ISO 6983 or better known as G & M code is a list of instructions that has been used for decades to communicate with the CNC systems. In actuality, G-codes are only a part of the NC programming language that controls NC and CNC machine tools.

ISO 6983 limits program portability (Zhang *et al.*, 2011b) for three reasons; First, the language focuses on programming the tool center path with respect to machine axes, rather than the machining process with respect to the part. Second, the standard defines the syntax of program statements, but in most cases leaves the semantics ambiguous. Third, vendors usually supplement the language with extensions that are not covered in the limited scope of ISO 6983 (ISO14649, 2002).

The ISO 6983 standard focuses on programming the path of the cutter centre location (CL) with respect to the machine axes, rather than the machining tasks with respect to the part. Thus, ISO 6983 defines the syntax of program statements, together with low-level limited control over program execution. These programs, when processed in a CAM system by a machine-specific post-processor, become machine-dependent (Xu *et al.*, 2006). Part of the issue for manufacturing programming is that industry has been using the same vector-based languages for manufacturing control for more than forty years as shown in Figure 1.1. The discipline applied to the use of these languages is superb with Product Data Management (PDM), Computer-Aided Design (CAD), Computer-Aided Manufacturing (CAM) and Computer-Numerical Control (CNC) systems each managing their part of the problem, but like an assembly language, a vector language is very difficult to manage (Hardwick *et al.*, 2012). (Hushim *et al.*, 2009) also did a comparative study between physical file Part-21 STEP-NC and G&M Code ISO 6983 to prove this point.

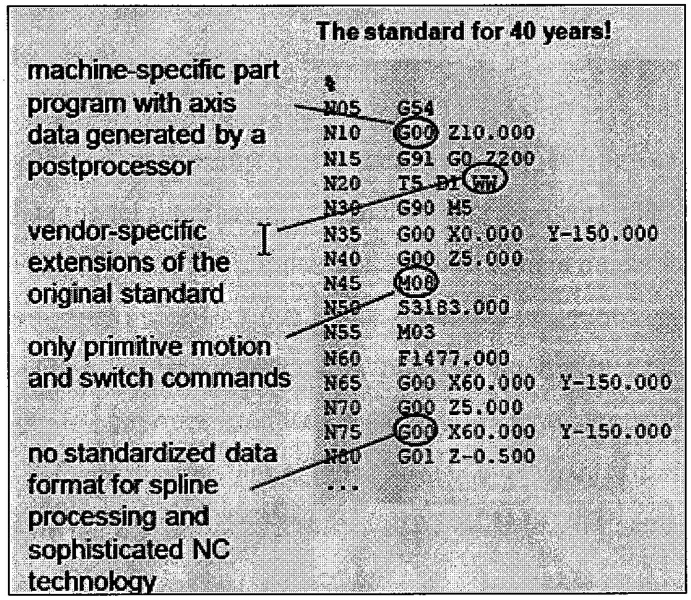

Figure 1.1 Vector Based Control Codes (Hardwick *et al.*, 2012).

The above-mentioned ISO 6983 is widely used as a standard of data interface for a Numerical Control apparatus in a manufacturing process. However, the ISO 6983 is a low-level international standard that just defines an axis movement command and switching commands. Several CNC manufacturers attempt to solve this problem by adding standards high-level command code of their own to the existing ISO 2983 standards. Moreover, this pattern gives freedom to the CNC machine manufacturer to create its own programming language (a dialect), which consequently prevents interoperability between design and manufacture (Cesar *et al.*, 2012).

However, since each manufacturer has their own standard, compatibility of the part programs that differs is widened, but despite its drawback, Mazak, Fanuc and many others CNC manufactures have developed the most advanced manufacturing system based on ISO 6983.

In addition, since the shop floor programming in manufacturing is using the basic G & M Code as what was in the ISO 6983, various production information such as machining features, machining process, machining technology, cutting tools, machining knowledge and feature information may not be included in the part program or worst become loss.

1.1.2 STEP-NC

ISO 14649 standard or commonly recognized as STEP-NC is not a programming language, but a data model interface for computer numerical controllers. STEP-NC goes much further that G codes and has a more comprehensive data model that overcomes the lack of process information in ISO 6983 files (G codes programming). STEP-NC was developed to provide a data model for a new breed of intelligent CNC controllers. The STEP-NC data model provides standard data requirements for machining processes associated with CNC machining. The STEP-NC standard is an extension of STEP and allows for integrations between STEP-based CAD/CAM/CAPP and CNC (Ridhwan, 2011).

The ISO 14649 is a set of standards where models are written in EXPRESS language and are in the form of Application Reference Model (ARM) type models, where they use domain terminology to describe machining. The ARM of STEP-NC, that is, ISO 14649, is made up of several Parts. In general, STEP-NC is the application of STEP methods to numerically controlled machines, representing a common standard specifically aimed at part programming, making the goal of a standardized CNC controller and a part program code-generation facility a reality.

With conventional process planning workflow employing CAD/CAM/CNC technology, the digital numerical data chain is broken and information feedback from machining (CNC) is problematic because of only an unidirectional data flow (Rauch *et al.*, 2012). The data flow chain is broken apart by post processing, which translates either drawing coordinates or technical plans into G-code (ISO 6983). This was the underlying reasons for The STEP-NC standard. STEP-NC (ISO 14649) is a high level language that allows a seamless bidirectional flow of data between CAD/CAM and CNC.

A STEP program contains both geometric and technological information, where different parts of the standard specify different machining processes (Zhao *et al.*, 2008). The STEP-NC data format is independent of the type of machine and control system, which implies that postprocessors are not required (Krzic *et al.*, 2009). The bidirectional transfer of information is particularly useful for closed-loop machining, where touch probes are used for on-line inspection, which feeds measured values back to the CNC controller (Zhao *et al.*, 2008).

STEP-NC allows more information about the machining process to be sent to the machine control and adds new information about the product being machined (Xu, 2006). This "Smart Data for Smart Machining" (Hardwick *et al.*, 2007), enables many applications to be done as it contains information about toolpaths, visual process, tolerances, feed and speed information and most of all the machining processes. Details on the STEP and STEP-NC standards is being further discuss and explained in chapter 2.

The STEP-NC compliant approach brings the possibility to integrate the machine controller to the overall enterprise information network, with the intention of having bi-directional data flow. This will provide the novel capability for upstream feedback of information from the numerical controller to the design and process areas in the process planning and manufacturing. Figure 1.2 shows the bi-directional information exchange of STEP-NC.

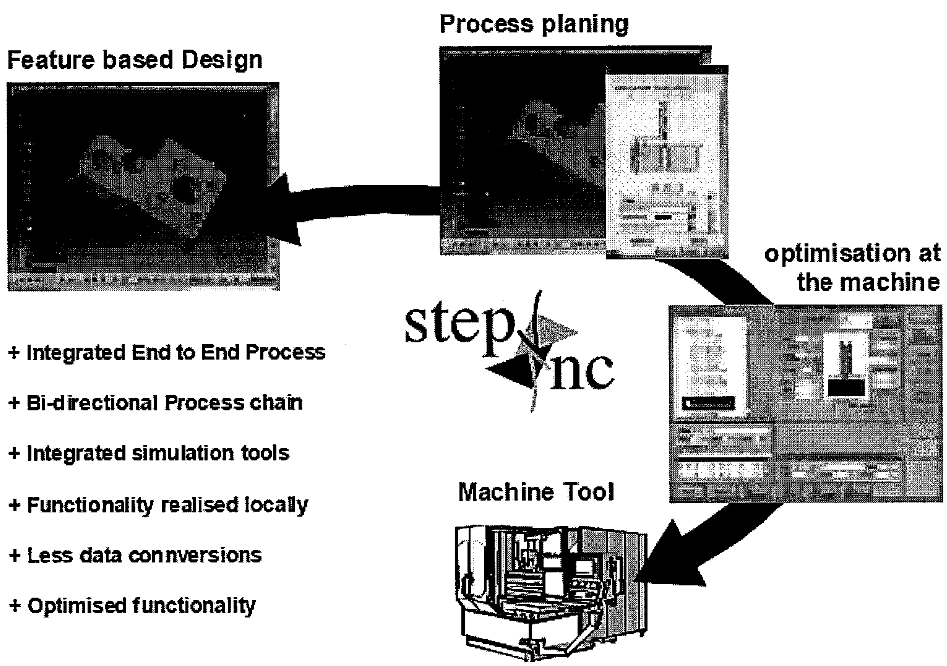

Figure 1.2 STEP-NC enables a bi-directional information exchange (Weck *et al.*, 2001).

1.2 Problem statement

The purpose of an intelligent machining process is to provide a means for representing and exchanging information about a product gathered during, and used in, the design and manufacture of that product. Therefore, the contents of a data model, systems or framework must be able to support the information needs of a large variety of computerized manufacturing applications. The popularity of using CAD systems as a means for creating, representing and exchanging product designs has created various standard product data exchange formats and have shown a success in transferring data between CAD systems, but they have failed to transfer product data from CAD to CAM applications. This is because current CAD systems are not able to support all the information concerning a part that is needed to support the CAM activities (Yang *et al.*, 1993).

One solution that offers the most promising solution is the creation of a standard method for representing product data that would serve the needs of all applications. The Standard for the Exchange of Product model data (STEP) is such a standard, offering the most promising answer to these problems (Amaitik *et al.*, 2002). STEP-NC is intended to be a replacement for G codes that provides much richer data and allows feedback of process information from the machine tool back upstream to design and process planning (Newman, 2002).

STEP-NC is currently being validated in research projects, and is showing promise as the next generation data interface to machine tools. The current manufacturing situation needed a system structures and effective algorithms that help bringing out a successful implementation of STEP-NC, which also is the main objective of this research that is to develop a STEP-CNC Mapping System that will convert generic STEP data to native STEP-NC and then translate the native file to manufacturing machine. This research work, will developed a STEP-NC compliant systems that enables machining to be done based on G-code, but will make full use of the richness of the STEP-NC data format as its interface. This new interface will make visible the richness of the STEP-NC data to users, where users can monitor the data flow of a product cycle from design to manufacturing.

1.3 Research Scope and Objectives

The aim of this research work is to develop an integrated, STEP-based process planning system that help designers and process planners to improve their design and planning in the early stages of the product life cycle. In order to achieve this aim, the following specific objectives have to be accomplished:

(i) To develop a system that can integrate design with manufacturing for a STEP-NC based system for different machine tool.

(ii) To validate the data flow of STEP-NC through a case study on developed system.

1.5 Importance of the research

This research provides some insight into a future generation of CNCs with open flexible architecture. This research is also important in helping industries and academician to understand the importance of making the change from using G & M code with the new STEP-NC. This research will give industries and academician more in-depth understanding and confidence to switch to STEP-NC by providing them with software that enables to generate STEP-NC codes and having the data transition that happens between CAM to CNC control being visualized.

1.6 Organization of the Thesis

In this thesis, the chapter is organized into seven chapters. Introduction to the project and the explanation of objective, scope of work and project background are in the first chapter.

Chapter II gives detailed information of STEP and STEP-NC standards. It starts with an overview, history and objectives of STEP, which is followed by a description of various parts in STEP. Next, the technical details of STEP and STEP-NC are given to explain the fundamentals of this research.

Chapter III presents some of the research that was done by other researchers on the subject of STEP-NC as well as implementation idea that has been performed on the subject and challenges that are being faced by it. Most of the research work surveyed herein is from teams in Europe, USA, Asia as well as New Zealand.

Chapter IV presents the conceptualization of the system, with all the process involved in the development of the system.

Chapter V presents the developed system framework as a whole which is the STEP-CNC Mapping System that consists of a STEP Convertor and a STEP-NC Translator.

Chapter VI presents the implementation of STEP-CNC Mapping System to the actual machining of parts and case study accomplished by the research thesis.

Chapter VII presents the conclusion for this thesis with recommendations for future works for this project are also suggested.

The summary for the organization of the thesis can be viewed in figure 1.2 where it were divided into 4 main parts, with part 1 for introduction, part 2 for background and literature review related to STEP and STEP-NC, part 3 for theoretical and design of STEP-CNC Mapping System and part 4 for research conclusion.

The Appendices at the end of the thesis contains sample of a generic STEP file generated from a CAD drawing by the solidwork software, a STEP-NC file populated with machining information mapped from generic STEP file and G-Code translated from a STEP-NC file that is being used for CNC machining in the end of the research process.

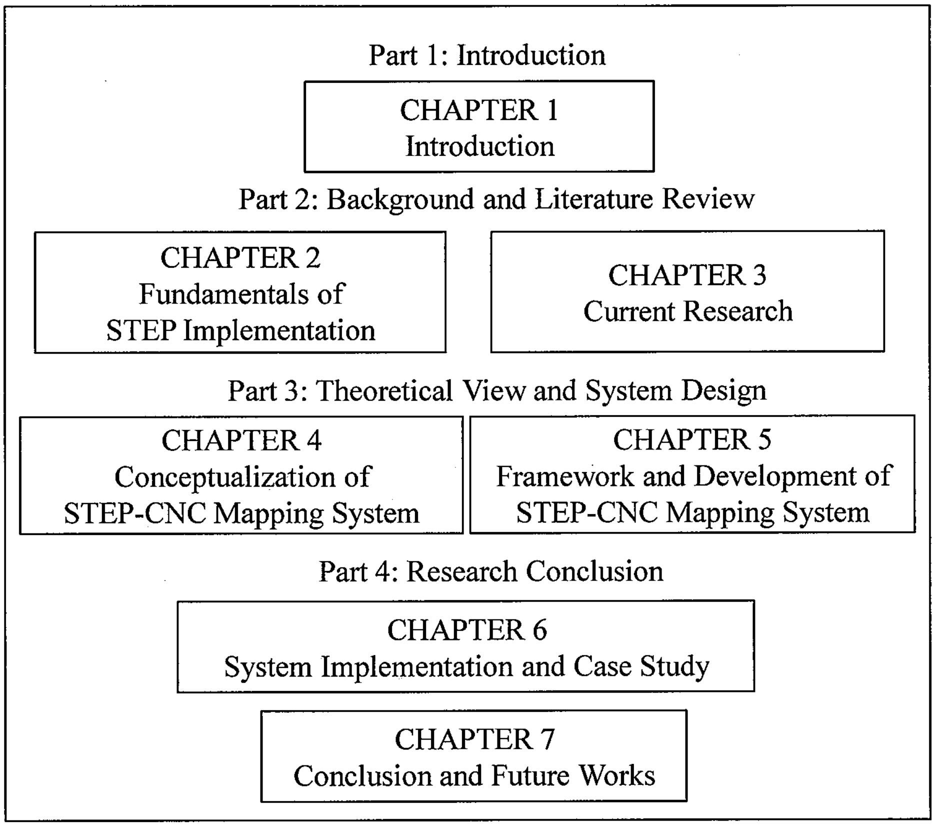

Figure 1.3 Description of Structure of Thesis

1.7 Summary

The aim of chapter one is to explain briefly about the flow of the research. It will later be explained in detail in other chapters. This research was aim at building a platform to industries and academician in changing their perception about making a switch from G & M code to STEP-NC and turn it as a priority in future advancement in the manufacturing sector. But the most important thing about this research hopefully is to provide a guideline on STEP-NC implementation in industries and its benefits.

In summary, STEP-NC is more adaptable and compliant with the current and future advancement of computers and technology. Information available in STEP-NC will make it more useful and present a more constructive environment to the manufacturing field.

CHAPTER 2

FUNDAMENTALS OF STEP IMPLEMENTATION

2.1 Introduction

Process planning is a time consuming and highly skilled job and is traditionally performed by a human expert with several years' experience (Yamada *et al.*, 2006). Process planning can be defined as the systematic determination of the detailed methods by which workpieces or parts can be manufactured from initial stages to finished stages. A process plan is an important document for production management. According to Newman (Newman, 2008), starting in the 1970s, significant development has been made towards automatic and reliable CNC machines with new processes such as punching and nibbling, laser cutting, and water jet cutting, which are now common place. The invention of minicomputers, and later microcomputers, has brought a massive improvement in the capabilities of CNC machines with the ability of multi-axis, multi-tool, and multi-process manufacturing. Various studies have been conducted to enhance the capabilities and expanding the applications of product machining. Till now, a variety of approaches and methods have been developed and utilized to simulate manufacturing processes (Abdul Kadir *et al.*, 2011a).

In many CNC manufacturing environments, CAD/CAM systems are used to define the component design and machining operation processes. The CNC programmer defines the part design by using a CAD system and determines the machining operation through the use of a CAM system. The end result is a cutting tool path written in cutter location data format, which is subsequently imported to the post-processor generating the NC part program for a vendor-specific machine controller. However, the NC part program is associated with G&M codes with simple 'go to point' instructions, which no longer satisfy the functional requirements of the modern-day CNC manufacturing. Furthermore, the CNC has the exclusive use of G&M codes, which isolate the CNC from the other CAD/CAM functions, in terms of information sharing and exchange. Such constraints are at the basis of a need to explore a high-level process planning description and interface, which can integrate the CAD to CNC process chain.

In the current CAD/CAM/CAPP machining chain, in order to machine a part, a CAD system is used to generate a file containing geometry, dimensions and tolerances information. This information - often contained in a STEP file - is imported into a CAM system where machining information is added. Manufacturing process selection forms the first step for inserting machining information; the operator chooses the process, e.g. milling or turning and then the information for that operation is linked to the geometrical features (Safaieh *et al.*, 2012). Finally a post processer is used by the CAM system to generate CNC machine codes. This system is shown in figure 2.1.

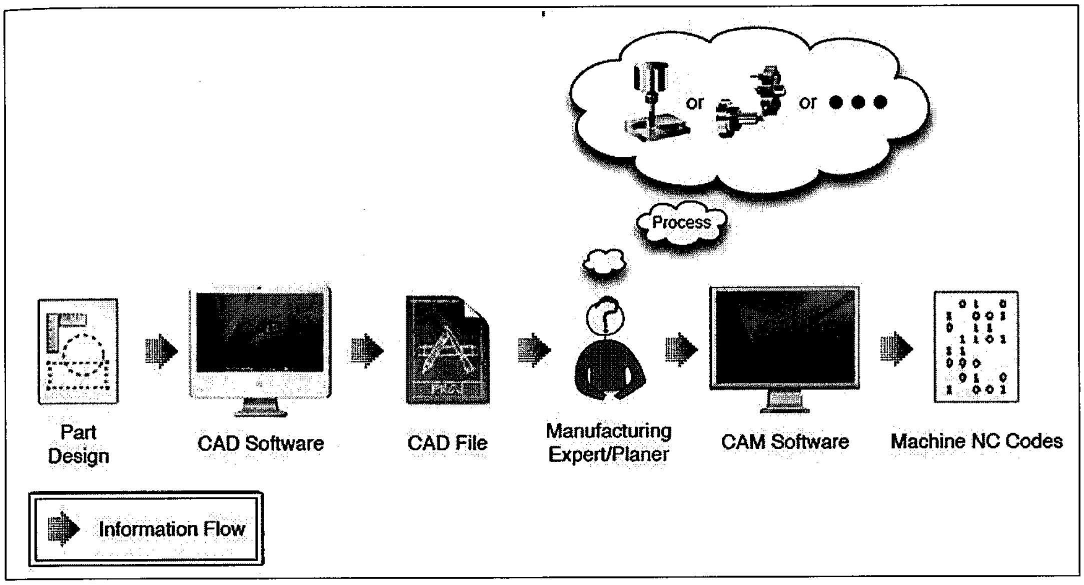

Figure 2.1 Current flow of information in the CNC machining chain (Safaieh *et al.*, 2012).

2.1.1 Computer Aided Process Planning

Machine-tool motion accuracy has a significant influence on the quality of the machining operations and, therefore, the development of a current measurement and evaluation methods has become a significant research subject (Du *et al.*, 2010). Following integration of CAD/CAM systems arises the problem of realization of the actual manufacturing of part, that is, CAPP system. The base of the whole problem is that output files of one part of suggestion are not input files of next part. This problem is solved by duplication of saved data in form of documentation, plan of tabs (Yang *et al.*, 2009).

Computer Aided Process Planning (CAPP) can be defined as the functions which use computers to assist the work of process planners. The level of assistance depends on the different strategies employed to implement the system. Lower level strategies only use computers for storage and retrieval of the data for the process plans which will be constructed manually by process planners, as well as for supplying the data which will be used in the planner's new work. In comparison with lower level strategies, higher level strategies use computers to automatically generate process plans for some workpieces of simple geometrical shapes. The highest level strategy, which is the ultimate goal of CAPP, generates process plans by computer, which may replace process planners, when the knowledge and expertise of process planning and working experience have been incorporated into the computer programs (Amaitik, 2005).

2.2 Early standards for design and manufacture

The efforts to create a neutral data exchange format begin in the 1970's with the contribution of X3/SPARC Committee of the American National Standard Institute (ANSI) in the United States. It proposed a three layer methodology that gives different views of the same information entitled conceptual, internal and external views. This methodology was used by the Integrated Computer Manufacturing (ICAM) program to develop the formal methods for information modeling named IDEF0 (ICAM Definition Level 0) later proposed as a Standard by NIST (NIST, 1993a), IDEF1X (NIST, 1993b) later named IDEF1X and IDEF2 (Jekemmerer, 1999). The ICAM program and its subsequent contracts such as the Product Definition Data Interface (PDDI) and Geometric Modelling Application (GMAP) (Goldstein *et al.*, 1998) program influenced the future of STEP.

The ICAM program also influenced the birth of the Initial Graphics Exchange Specification (IGES) at the end of the 1970's. This effort to create IGES was an initiative of the US government, CAD/CAM users and vendors (Rosso Jr, 2005). They worked in a committee coordinated by the National Bureau of Standards (now the National Institute of Standards Technology – NIST) which generated the first release of IGES in 1980 (Jekemmerer, 1999).

2.3 Overview of STEP

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 (ISO10303-1, 1994) 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 Representatiom 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 (Brecher *et al.*, 2006a).

2.3.1 History of STEP

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 (Amaitik, 2005). 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 (Jekemmerer, 1999).

The following factors contribute to the perceived deficiencies of standards such as IGES (Jekemmerer, 1999):

- 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 (Amaitik, 2005). 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 (Gallaher *et al.*, 2002).

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). Within ISO, there is a Technical 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 (Amaitik, 2005).

2.3.2 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(ISO10303, 1994). STEP permits different implementation methods to be used for storing, accessing, transferring, and archiving product data (ISO10303, 1994). The STEP project was initiated with the following objectives (Amaitik, 2005):

- 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 (Fowler, 1995). 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 (Newman *et al.*, 2009).

2.3.3 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 2.2 illustrates the structure of the STEP standard.

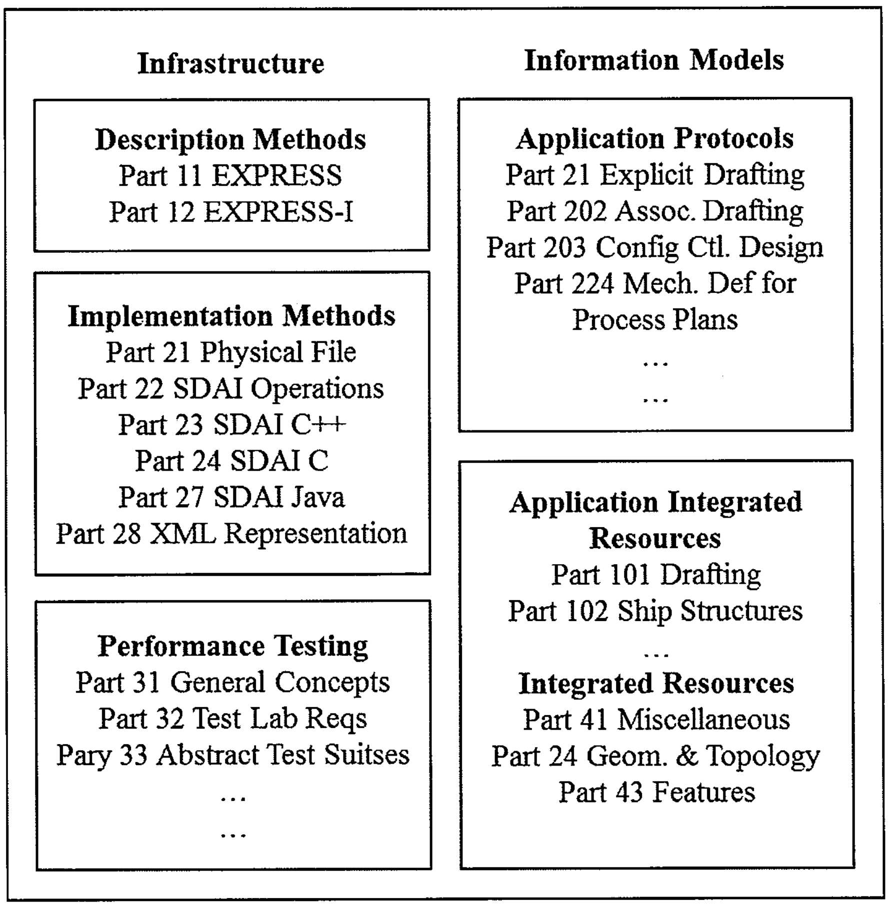

Figure 2.2 Structure of the STEP Standard (ISO10303, 1994).

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. The total number of Parts of the other types listed below is about 120.

- Overview and fundamental principles (1)
  This is a single document giving an overview of STEP and an exposition of its fundamental principles.
- Description methods (11-19)
  These cover the information modelling language EXPRESS and its graphical form, EXPRESS-G.
- Implementation methods (21-29)
  These cover methods of representing data that has been modelled in EXPRESS.
- Conformance testing methodology and framework (31-39)
  These give the general concepts of conformance testing as well as actual test methods and requirements on testing labs and clients.
- Application protocols (201-299)
  These are the Parts intended for implementation in industry. As described in more detail below, each application protocol includes several documents.
- Integrated generic resources (41-59)
  These are EXPRESS information models of widely useful specific subject domains, such as geometry, topology, and tolerances.
- Integrated application resources (101-199)
  These are EXPRESS information models of more narrowly focused specific subject domains.

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.

2.3.4 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(Wang, 2009).

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 (Zhao, 2009):

- A product model specific file format called Part 21 physical file (ISO10303-21, 2002);
- 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 (ISO10303-22, 1998; ISO10303-23, 1998; ISO10303-24, 1998; ISO10303-27, 1998);
- The three methods for mapping the EXPRESS defined data into XML described by Part 28 Edition 1 (ISO10303-28, 2002); and
- The XML Schema-governed representation of EXPRESS described by Part 28 Edition 2 (ISO10303-28, 2004).

2.4 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 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 (ISO10303-201, 1994).
- AP 202: Associative Draughting. 2D/3D drawing with association, but no product structure (ISO10303-202, 1996).
- AP 203: Configuration controlled 3D designs of mechanical parts and assemblies (ISO10303-203, 2011).
- AP 204: Mechanical design using boundary representation (ISO10303-204, 2002) (ISO10303-207, 1999).
- AP 207: Sheet metal die planning and design (ISO10303-207, 1999).
- AP 209: Composite and metallic structural analysis and related design.
- AP 214: Core data for automotive mechanical design processes (ISO10303-214, 2010).
- AP 235: Materials information for the design and verification of products (ISO10303-235, 2009).
- AP 236: Furniture product data and project data (ISO10303-236, 2006).
- AP 242: Managed model based 3D engineering (under development) (ISO10303-242).

2.5 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. In particular it will contain major updates in the area of Geometric dimensioning and tolerancing, kinematics, and tessellation. Future editions of AP 242 will extend the scope further into areas such as electrical harnesses.

2.6 EXPRESS language

EXPRESS is a standard data modeling language for product data. EXPRESS is formalized in the ISO Standard for the Exchange of Product model STEP (ISO 10303), and standardized as ISO 10303-11 (ISO10303-11, 2004). Data models formally define data objects and relationships among data objects for a domain of interest. Some typical applications of data models include supporting the development of databases and enabling the exchange of data for a particular area of interest. Data models are specified in a data modeling language (McCaleb, 1999). EXPRESS is a data modeling language defined in ISO 10303-11, the EXPRESS Language Reference Manual (McCaleb, 1999).

An EXPRESS data model can be defined in two ways, textually and graphically. For formal verification and as input for tools such as SDAI the textual representation within an ASCII file is the most important one. The graphical representation on the other hand is often more suitable for human use such as explanation and tutorials. The graphical representation, called EXPRESS-G, is not able to represent all details that can be formulated in the textual form.

EXPRESS is similar to programming languages such as Pascal. Within a SCHEMA various datatypes can be defined together with structural constraints and algorithmic rules. A main feature of EXPRESS is the possibility to formally validate a population of datatypes - this is to check for all the structural and algorithmic rules.

EXPRESS-G is a standard graphical notation for information models. It is a useful companion to the EXPRESS language for displaying entity and type definitions, relationships and cardinality. (ISO10303-11, 2004) graphical notation supports a subset of the EXPRESS language. One of the advantages of using EXPRESS-G over EXPRESS is that the structure of a data model can be presented in a more understandable manner. A disadvantage of EXPRESS-G is that complex constraints cannot be formally specified. The data model presented in figure could be used to specify the requirements of a database for an audio compact disc (CD) collection (McCaleb, 1999).

2.7 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 (Jekemmerer, 1999).

2.8 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 (Newsletter, 2000). 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 2.3 portrays the relationship between ISO 14649 and some APs of STEP ISO 10303. Based on the Figure 2.3, 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.

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