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STEP-Based Product Data Exchange for Web-Enabled Collaborative Design and Manufacture

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STEP-Based Product Data Exchange of Web-Enabled Collaborative Design and Manufacture

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1 Introduction

1.1 Aims and objectives of the project

This research is directly related to the technique of Web-Enabled Collaborative Design and Manufacture, aiming to improve CAD/CAM files transfer quality and high data transfer performance between Europe and China. The main activities include: (1) Development of data exchange framework for Web-Enabled collaborative design and manufacture, (2) Research and development into product data real time exchange system among different sectors and different CAD/CAM systems, (3) Case studies to apply the methods, procedures and data exchange tools developed.

1.2 Background

1.2.1 The background of the research

The overall aim of the project " Web-enabled Collaboration in Intelligent Design and Manufacture " is to improve co-operation between China and Europe by increasing the mutual understanding of the complexity of transferring European information and communications technology to China, thus improving technology cross-flow and the quality of Europe-China partnerships.

The topics of data exchange of CAD-CAM files and sharing of such files in real time between distant users will be addressed in the project. A major emphasis of the project is to ensure that these exchange and sharing activities can take place regardless of the computer hardware at the different sites. It is important that each site can maintain flexibility in its local IT hardware, and not have to purchase additional hardware just so that each site has identical computers. The proposed research in this category also addresses the topic of ‘to remote control and pilot machinery’ which is interrelated with the CAM mentioned above regarding the Information Systems in Manufacturing.

Commercial CAD/CAM systems are easily available for different users. Most of them have been developed for many years and have proven to be excellent tools for design and manufacturing engineers. These systems are usually running on different environments and require different levels of supports. As product development requires shorter lead-time and faster response to the market demand, more communication between system is required. Concurrent engineering initiates the communication among different sectors and brings engineers together and thus, shortens the product development cycle time and enhances product quality. As the network communication expands rapidly, concurrent engineering can be carried out more effective and efficient through the use of the network facility. Under these situations, CAD/CAM systems will inevitably be communicated with each other and thus, bring out many problems due to lack of this capability.

On the other hand, different CAD/CAM systems are very often found using different formats to display their drawings, and a drawing developed by a system can hardly be viewed by another system. The problem of the format mismatch causes isolation of these systems, and companies that require drawing files from different sources may be forced to invest on many CAD/CAM systems. This situation brings up many problems in China, where small and medium sized enterprises are the major manufacturing source and the Original Equipment Manufacturer (OEM) is their major business pattern. For large companies, it also builds up an obstacle in integrating computer aided tools as well as in implementing concurrent engineering.

STEP (STandard for the Exchange of Product model data) is an evolving international standard for the representation and exchange of product data. It offers a system-independent mechanism to describe the product information in the WEE throughout its lifetime. STEP is a key international product data technology that effectively enables interoperability, supply chain integration, web-based collaboration and life cycle management. STEP enables manufacturers to achieve new and higher levels of quality and productivity while reducing costs and time-to-market. Today STEP use is growing through the world within many industries, including aerospace, automotive, shipbuilding, electronics, process plants, and construction.

This facility can reduce the lead-time for product development, reduce the overhead cost for personnel and facility, enhance the flexibility of individual engineers as well as the company. However, due to the problem of data formats and lack of the communication capability, we are not able to achieve this goal at the present time. The key issues to accomplish this goal are: a good infrastructure, a proper communication tool, and standard product data representation that can be accepted by most CAD/CAM software systems.

In this research, we propose a framework for a networked CAD/CAM environment. In the framework, STEP is used as the standard to represent product information. With proper infrastructure and communication tool, we can build up an environment in which CAD/CAM data files can be shared by different users and thus, concurrent engineering can be carried out more easily.

1.2.2 The framework of STEP-Based Product Data Exchange of Web-Enabled Collaborative Design and Manufacture

Fig. 1-1 presents a framework for STEP-Based Product Data Exchange of Web-Enabled Collaborative Design and Manufacture. Various services are installed in different enterprises. The services can be viewed or executed by web browsers.

The service description information can be accessed through internet services server, and Translator Server. Each enterprise has its own native database. One enterprise can access another company’s native database with permission. The native data may be either STEP or non-STEP format. To implement the data communication between different companies, non-STEP data will be translated into STEP data through Translator Components. The remote information access or remote service execution can be implemented with Web.

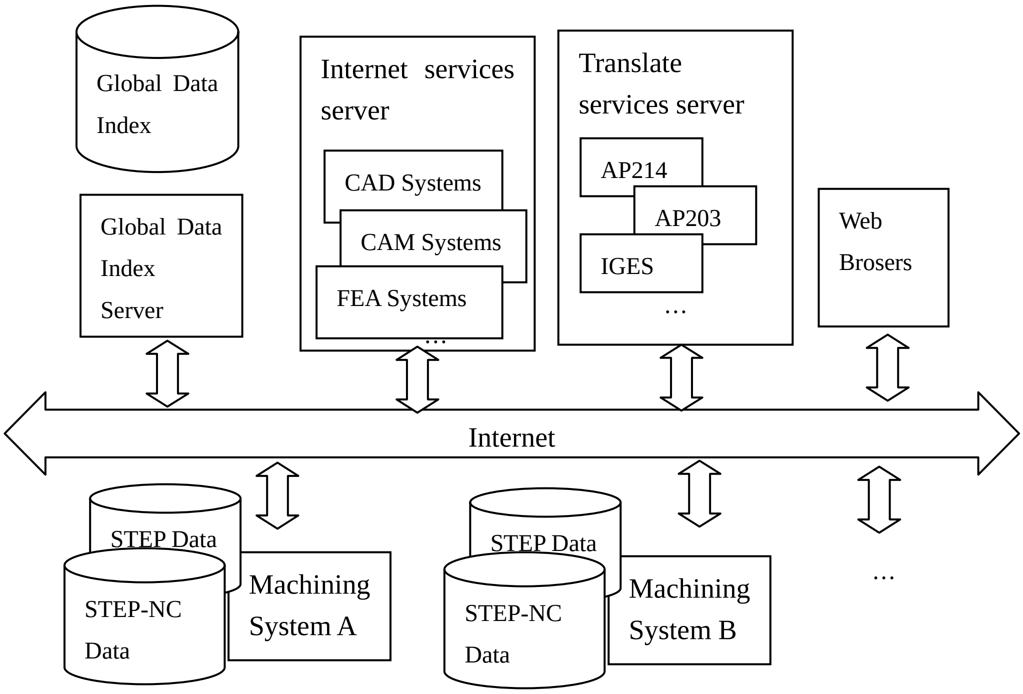


Fig 1-1 A framework for for STEP-Based Product Data Exchange of Web-Enabled Collaborative Design and Manufacture

2 Review of Product Data Exchange

The development of product data exchange can be divided into three phases. It could be called three generations of product data exchange: the direct translator, the neutral format file, and STEP-based product database.

2.1 The first generation - the direct translator

A direct translator is a piece of software which takes input data from one system and convert it, so that it can be used in a second system to present the same information.

The advantages of dedicated translators are as follows: (1) they may cope where other method are not available; (2) there is usually less processing time required; (3) if there is only one pair of systems, there is less codes to be written than using the neutral format described latterly.

The disadvantages of ad hoc translators are as follows: (1) the quantity of translators that need to be written increases quadratically with the number of CAD systems. The formula is N= n(n-1), as shown in Fig.2-1. (2) the programmer needs to know every system's representations of relevant information in detail. (3) it is not suitable for SMEs, considering the cost , time, experiences, and so forth.

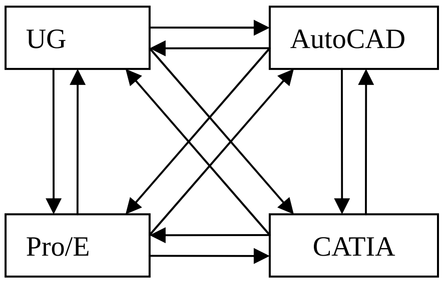


Fig.2-1 Direct translators (12 translators)

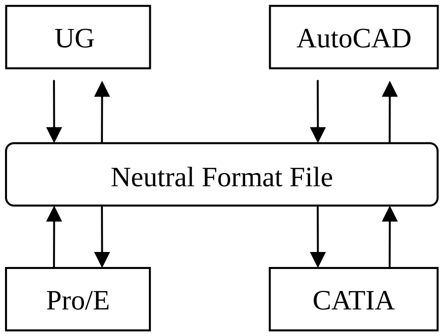


Fig.2-2 Neutral format file (8 translators)

2. 2 The second generation - the neutral format file

The neutral format can be thought of an “Esperanto” for CAD systems. All the information to be represented is system independent form. The vendor of a CAD system can write two pieces of software, named preprocessor and postprocessor. The preprocessor is used to translate its internal data form into the neutral format, and the postprocessor translate the neutral format file into its internal data form. The famous neutral formats are IGES (USA,1980), VDA-FS (Germen,1984, mainly deals with surface model in car industry), SET (France,1985, mainly overcomes the demonstrable deficiencies in aerospace and automotive industries), DXF (Data Exchange File or Format, Autodesk Inc.)

The advantages of neutral format data exchange are as follows: (1) the number of piece of software needed is less than that before, The formula is N= 2n, as shown in Fig.2-2. (2) the responsibility for writing the processors lies clearly with the vendors. (3) testing can be undertaken by an independent organization. (4) there are some conformance testing.

The disadvantages of neutral format data exchange are as follows: (1) it is difficult to find one which has adequate capability to achieve international standardization, and the technologies are still developing. (2) it is not the vendors interest to enable easy information flow out of their system. (3) the data are mainly about CAD/CAM system.

2. 3 The third generation - STEP-based product database

STEP, the STandard for the Exchange of Product model data, is a comprehensive ISO standard that describes how to represent and exchange digital product information. STEP is a key international product data technology that effectively enables interoperability, supply chain integration, web-based collaboration and life cycle management. STEP enables manufacturers to achieve new and higher levels of quality and productivity while reducing costs and time-to-market. Today STEP use is growing through the world within many industries, including aerospace, automotive, shipbuilding, electronics, process plants, construction.

2. 3.1 The history of STEP

The STEP project was initiated in 1984. The objectives are as follows: (1) The creation of single international standard, covering all CAD/CAM data exchange. (2) The implementation and acceptance of this standard by industry, superseding various national and de facto standards and specification. (3) The standardization of a mechanism for describing product data, throughout the entire life of a product, and independent of any particular system. (4) 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 database, and for long-term archiving. The development is supported by the leading nations of industrialized world[2].

In 1988, the Integrated Product Information Model (IPIM) and the describing language (EXPRESS) were published as a Draft Proposal for balloting process. And it was not approved for publication as a standard, because further technical work to be done. On September 1994, after years of development, 12 parts of STEP were approved as international Standards, ISO10303 "Industrial automation system – Product data representation and exchange".

It takes a long time for STEP to develop, since its objectives are setting standards for the future rather than capturing the common aspects of past or recent ways of working. The pilot projects played an important role in the development of STEP. Several famous projects are PDES (Product Data Exchange using STEP), CAD*I (ESPRIT CAD Interfaces), PDDI (US Air Force Product Definition Data Interface project).

2. 3.2 The importance of STEP

Some key reasons why STEP is important:

STEP is international, and was developed by users, not vendors. User-driven standards are results-oriented, while vendor-driven standards are technology-oriented. STEP has, and will continue to, survive changes in technology and can be used for long-term archiving of product data.

STEP has been constructed as a multi-part ISO standard. These parts cover general areas, such as testing procedures, file formats and programming interfaces, as well as industry-specific information. The product data contain enough information to cover a product's entire life cycle, from design to analysis, manufacture, quality control testing, and product support functions. Thus STEP covers geometry, topology, tolerances, relationships, attributes, assemblies, configuration and more.

STEP is a standard that can grow. It is based on a language (EXPRESS) and can be extended to any industry. A standard that grows will not be outdated as soon as it is published. The EXPRESS language describes constraints as well as data structure. Formal correctness rules will prevent conflicting interpretations. STEP CASE tools such as ST-Developer use these descriptions to create more robust, maintainable systems.

2. 3.3 The main contents of STEP

The STEP is composed of two main categories, those are infrastructure and information models.
Infrastructure:

- Description Methods (Part 11-19)
- Implementation Forms (Part21-29)
- Conformance Testing (Part31-39)
- Abstract Test Suite (Parts 301…)

Information models:

- General Resources (Part41…)
- Application Resource models (Part101…)
- Application Protocols (Part201…)
- Application Interpreted Constructs (Part501…)

3 Analysis of The STEP AP214 protocol

International Standard ISO 10303–214 was prepared by Technical Committee ISO/TC 184/SC4, Industrial automation systems and integration, Subcommittee SC4, Industrial data.
This International Standard is organized as a series of parts, each published separately. The parts of ISO 10303 fall into one of the following series: description methods, integrated resources, application interpreted constructs, application protocols, abstract test suites, implementation methods, and conformance testing.

3.1 The STEP Standard

As discussed before, the STEP standard is divided into a number of parts. The first class of parts is Part 1: “Overview and fundamental principles”, which describes the overall structure of the standard, defines the basic principles and the relationship between its various elements. The STEP document structure is depicted in Figure 3-1.

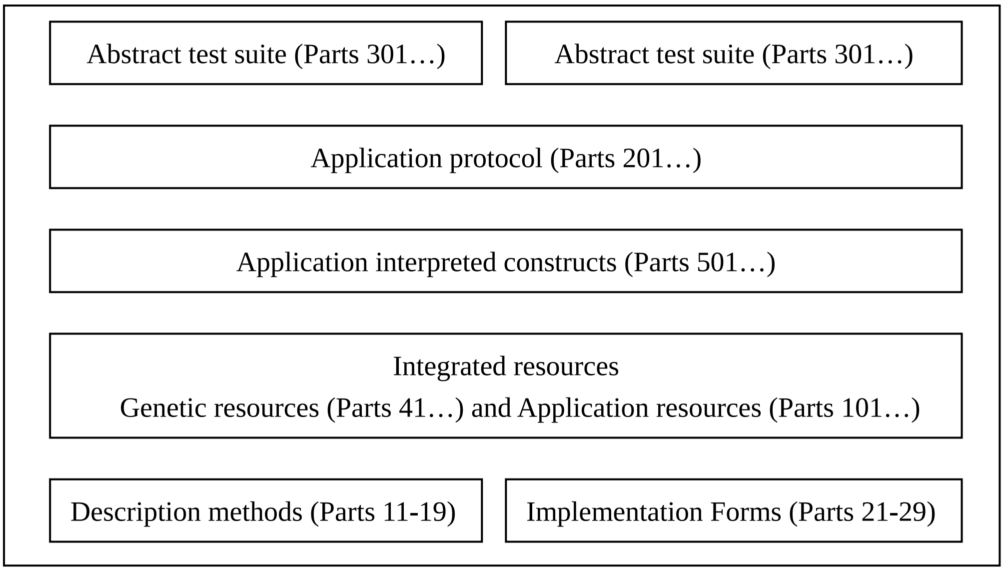


Fig. 3-1 The STEP document structure

3.1.1 Description Methods

Description Methods define the languages and methods used to create standard representations of product data. The description methods are listed in Table3-1.

Table 3-1 ISO10303 Description Methods

Part 11	EXPRESS Language Reference Manual
Part 12	EXPRESS-I Language Reference Manual
Part 13	Architecture and Methodology Reference Manual
Part 14	EXPRESS-X Language Reference Manual

3.1.1.1 Part 11 EXPRESS Language Reference Manual

(1) EXPRESS Language

EXPRESS is a specification language for the logical description of information models. EXPRESS has object-oriented characteristics as well as such defined through the entity relationship method. It enables the formal, unambiguous and complete description of a product model through objects, relations and condition.

Within the ISO 10303 (STEP) standard, all data models, ranging from the Integrated Resources to Application protocols, are described in EXPRESS.

The elements of ESPRESS are:

- **Schema:** A schema is a collection of information elements, which can be - as whole or separately - used by other schemas. Each information element has to be specified in a schema.
- **Entity:** An entity defines an object - consisting of properties and underlying definable rules.

Each entity has a unique name within the schema. An inheritance concept is supported by sub-super-relations.

- **Attributes:** Attributes are properties of objects, which are defined by a name and a type, and additional characteristics can also be specified.
- **Type:** EXPRESS supports basic types (Integer, Real, String, Boolean, Logical) as well as complex data types (Enumeration, Select, Complex).
- **Rules:** The description of conditions for the regularity of an instance, i.e. the occurrence of an entity, can be related locally to a single entity within its definition as well as globally to the totality of entities in the schema definition.

(2) EXPRESS-G

For the description language EXPRESS, there is also a graphical notation for information modeling----EXPRESS-G. EXPRESS-G supports a subset of EXPRESS and, by its visual representation in compact form, furthers the understanding of the data model.

2.1.2 Part 12 EXPRESS-I Language Reference Manual

EXPRESS-I is a standard data instance definition language. It may be used to specify actual values within an EXPRESS schema. It is particular useful in the specification of test data.

2.1.3 Part 13 Architecture and Methodology Reference Manual

Part 13 provides a definitive statement of the STEP architecture and methods, to enable the application of the methodology, and to serve as a basis for training.

2.1.4 Part 14 EXPRESS-X Language Reference Manual

EXPRESS-X is a method for the description of object mappings between defined EXPRESS schemas, which contain common semantic elements, but are defined in different ways.

3.1.2 Implementation Forms (Parts 21-29)

The implementation forms define standard format for data instance and values, and the mappings between these formats and the EXPRESS language. One of the key differences between the previous data exchange standards is the separation of data definition from implementation. Implementation forms are shown in Table3-2.

Table 3-2 ISO10303 Implementation Forms

Part 21	Clear text encoding of the exchange structure
Part 22	Standard data access interface (ASDI)
Part 23	C++ language binding to SDAI
Part 24	C Language Binding to SDAI
Part 25	EXPRESS to OMG XMI
Part 26	CORBA IDL Language Binding to SDAI (canceled)
Part 27	Java Language Binding to SDAI
Part 28	XML representation for EXPRESSS-driven

Part 21 defines the standard format for encoding data in a file, and supports the exchange of data between applications. It is commonly referred as physical file format.

Part 22 (ASDI) enable access to product data within an application independently of the internal form of data storage within the application.

Other parts define the bindings between the SDAI and the programming languages including C, C++ and COBRA. For the purpose of shared database access and integrated knowledge-base, Part 28 defines the XML (Extensible Markup Language) representation for EXPRESSS-driven.

3.1.3 Integrated resources

Within STEP, a single conceptual data model has been developed that reflects and supports the common requirements of many different product data application areas. The integrated resources are used to constitute this conceptual data model. They are divided into two series of parts: integrated generic resources and integrated application resources. The former includes information models independent from a certain application. The latter extend the former to support the need of specific groups of applications. The contents of them are listed in Table 3-3 and Table 3-4.

Table 3-3 Integrated generic Resources

Part 41	Fundamentals of Product Description and Support
Part 42	Geometric and Topological Representation
Part 43	Representation Structures
Part 44	Product Structure Configuration
Part 45	Materials
Part 46	Visual Presentation
Part 47	Tolerances
Part 49	Process Structure and Properties
Part 50	Mathematical Constructs
Part 51	Mathematical Description
Part 52	Mesh-based topology
Part 53	Numerical Analysis

Table 3-4 Application Resources

Part 101	Draughting
Part 103	Electrical and Electronic Connectivity
Part 104	Finite Element Analysis
Part 105	Kinematics
Part 106	Building and Construction
Part 107	Finite Element Analysis Definition relationship
Part 108	Parametrization and Constrains for Explicit Product Models
Part 109	Assembly Model for Products
Part 110	Mesh-based Computation Fluid Dynamics

3.1.4 Application Interpreted Constructs (Parts 501…)

Application interpreted constructs (AICs) mostly deal with the description of geometric models, which use the geometric and topologic basic objects defined in ISO 10303-42, and specialize the generic shape_representation object by specifying a number of conditions. By observing these, one instance represents exactly one representation of the according geometric model. ACIs contents are listed in Table 3-5.

Table 3-5 Application Interpreted Constructs

Part 501	Edge-based wireframe
Part 502	Shell-based wireframe
Part 503	Geometrically-bounded 2d wireframe
Part 504	Draughting annotation
Part 505	Drawing structure & administration
Part 506	Draughting elements
Part 507	Geometrically-bounded surface
Part 508	Non-manifold surface
Part 509	Manifold surface
Part 510	Geometrically-bounded wireframe
Part 511	Topological-bounded surface
Part 512	Faceted B-representation
Part 513	Elementary B-rep
Part 514	Advanced B-rep
Part 515	Constructive solid geometry
Part 516	Mechanical Design Context
Part 517	Mechanical design geometric presentation
Part 518	Mechanical design shaded presentation
Part 519	Geometric tolerances
Part 520	Associative draughting elements
Part 521	Manifold subsurfaces

3.1.5 Application protocol (Parts 201…)

An Application Protocol (AP) defines the use of integrated resources satisfying the scope and information requirements for in a specific application context. APs use the STEP integrated resources only, but do not extend the data model defined in integrated resources. Each AP aims at a specific discipline or industry sector. The role of an AP is to provide the basis for Implementations of STEP, and enable the assessment of conformance of implementations.

An AP is divided into a number of separate and interrelated sections. This structure not only encourages the consistent development but also eases and helps the processes of review, implementation, and use. There are four major elements of an AP, the scope, the information requirements, the application integrated model and the conformance requirements.

(1) **The scope (AAM)**

The scope specifies the industry processes and data that are designed to support. The scope is related to an Application Activity Model (**AAM**), a graphical model (such as IDEF0) of the related activities that the AP supports. In the AAM, the data classes that identify the input, output, method or parameters of the functions in focus are identified.

(2) **The information requirements(ARM)**

The definition of information requirements is supported by an Application Reference Model (ARM). The ARM is dependent on the AAM. It is a detailed description of data objects (entities and attributes), and the relationship between them within the scope of the industrial application. ARM is usually described by domain experts using the terminology of the application.

(3) **The application integrated model (AIM)**

AIM is an ESPRESS data model that defines how the information requirements are satisfied using the STEP Integrated Resources. The AIM is supported by a mapping table, which specifies how each identified information requirement is satisfied. And, a mapping is specified between the data objects of the ARM and those of the AIM.

The relationship between models is shown in Figure 3-2

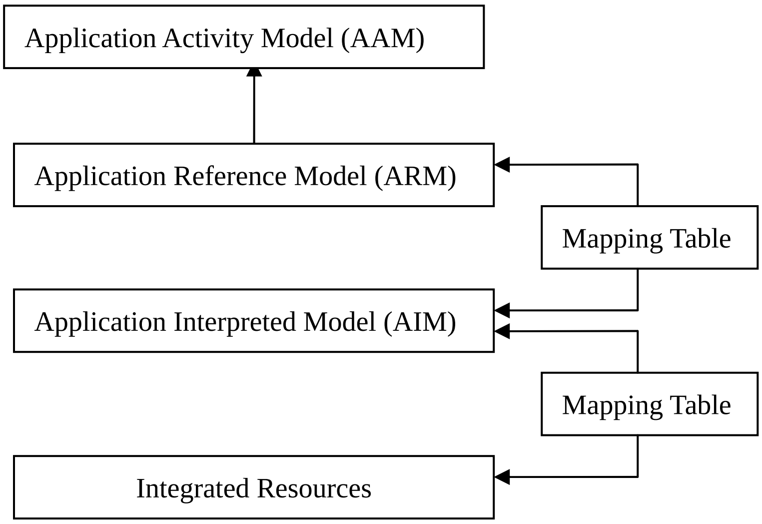


Fig. 3-2 Relationship between models

(4) **The conformance requirements**

The STEP APs need the completeness of implementation, that is, an interface conforming to the AP should support every entities, attribute and constraint specified. However, in recognition of the differences between the computer systems within a given application area, many APs identify a number of conformance clauses that specify the AP for which conformance may be claimed and assessed.

Table 3-6 Application Protocols

Part 201	Explicit Draughting
Part 202	Associative Draughting
Part 203	Configuration Controlled 3D Designs of Mechanical Parts and Assemblies
Part 204	Mechanical Design Using Boundary Representation (withdrawn)
Part 205	Mechanical Design Using Surface Representation (withdrawn)
Part 206	Mechanical Design Using Wireframe Representation (withdrawn)
Part 207	Sheet metal die planning and design
Part 208	Life-cycle product change process (canceled)
Part 209	Composite and metal structural analysis and related design
Part 210	Electronic Assembly, Interconnect and Packaging Design
Part 211	Electrical Printed Circuit Assembly, Test and Diagnosis (withdrawn)
Part 212	Electrotechnical Design and Installation
Part 213	Numerical control (NC) process plans for machined parts (canceled)
Part 214	Core Data for Automotive Mechanical Design Processes

Part 215	Ship Arrangement
Part 216	Ship Moulded Forms
Part 217	Ship Piping (withdrawn)
Part 218	Ship Structures
Part 220	Process planing, manufacturing assembly of layered electrical products
Part 221	Functional data and their schema representation for process plants
Part 223	Exchange of information for cast parts (canceled)
Part 224	Mechanical parts definition for process planning using machining features
Part 225	Building elements using explicit shape representation
Part 226	Ship Mechanical Systems
Part 227	Plant spatial configuration
Part 228	Building service: heating, ventilation and air conditioning (withdrawn)
Part 230	Building structure frame: steel work (withdrawn)
Part 231	Process engineering data (withdrawn)
Part 232	Technical data packaging: core information and exchange
Part 233	Systems engineering data representation (canceled)
Part 234	Ship operational logistics, records and messages
Part 235	Materials information for design and verification of products (cancelled)
Part 236	Furniture product and project data
Part 237	Computational fluid dynamics
Part 238	AIM for computer numeric controllers
Part 239	Product life cycle core

3.1.6 Conformance testing (Parts31-39)

The Conformance Testing Methodology and Framework class of parts provides test laboratories, implementers, and end users with consistent, comprehensive conformance testing of implementation of STEP.

Table 3-7 Conformance Testing Methodology and Framework

Part 31	General concepts
Part 32	Requirements on testing laboratories and clients
Part 33	Abstract test suites
Part 34	Abstract methods for Part 21 implementations
Part 35	Abstract methods for Part 22 implementations

3.1.7 Abstract test suite (Parts 301…)

Experience in testing Open Systems products has shown that without standardized test suites the same product can give different results in different test laboratories. An Abstract Test Suite defines the detailed requirements for the assessment of conformance, and includes a number of test cases that are to be used by all test laboratories.

3.1.8 Relevant Standards: Part Library

The use of standard parts within a design is commonly supported by the use of Part Library (ISO13584). P-LIB defines standard mechanisms for the creation and use of such part library. The content is listed in Table 3-8 below.

Table 3-8 Part Library (ISO13584).

ISO13584-1	Overview and fundamental principles
ISO13584-10	Conceptual model of a part library
ISO13584-20	Logical model of expressions
ISO13584-24	Logical model of a supplier library
ISO13584-26	Supplier identification
ISO13584-31	Geometric programming interface
ISO13584-42	Methodology for structuring art families
ISO13584-101	Geometric exchange protocol by parametric program
ISO13584-102	View exchange protocol by ISO10303 conforming specification
ISO13584-501	Standard dictionary on measuring instruments

3.2 STEP AP214 Protocol Analyzing

This part (214) of ISO 10303 specifies an application protocol (AP) for automotive mechanical design processes. This application protocol defines the context, scope, and information requirements for various development stages during the design of a vehicle and specifies the integrated resources necessary to satisfy these requirements. This application protocol addresses the requirements of the automotive industry covering cars, trucks, busses, and motorcycles. It may also be applicable to other forms of vehicles, such as rail, agricultural, and construction vehicles.

Figure 3-3 shows the planning model for this AP. It provides a high level overview of the information supported by the AP on the basis of the units of functionality (UoF).

3.2.1 Scope of AP214

Application protocols provide the basis for developing implementations of ISO 10303 and abstract test suites for the conformance testing of AP implementations.

This part of ISO 10303 specifies the use of the integrated resources necessary for the scope and information requirements for the exchange of information between the applications that support the development process of the mechanical aspects of automotive vehicles.

The following are within the scope of this part of ISO 10303:

— products of automotive manufacturers and of their suppliers. These products include parts, assemblies of parts, tools, assemblies of tools, and raw materials. The considered parts include the constituents of the car body, of the power train, of the chassis, and of the interior. The tools include those specific to the products produced and used by various manufacturing technologies.

EXAMPLE Typical manufacturing technologies are:

a) for primary shaping: moulding or casting;

b) for transforming: bending or stamping;

c) for separating: milling or turning;

d) for coating: painting or surface coating;

e) for fitting: welding or riveting.

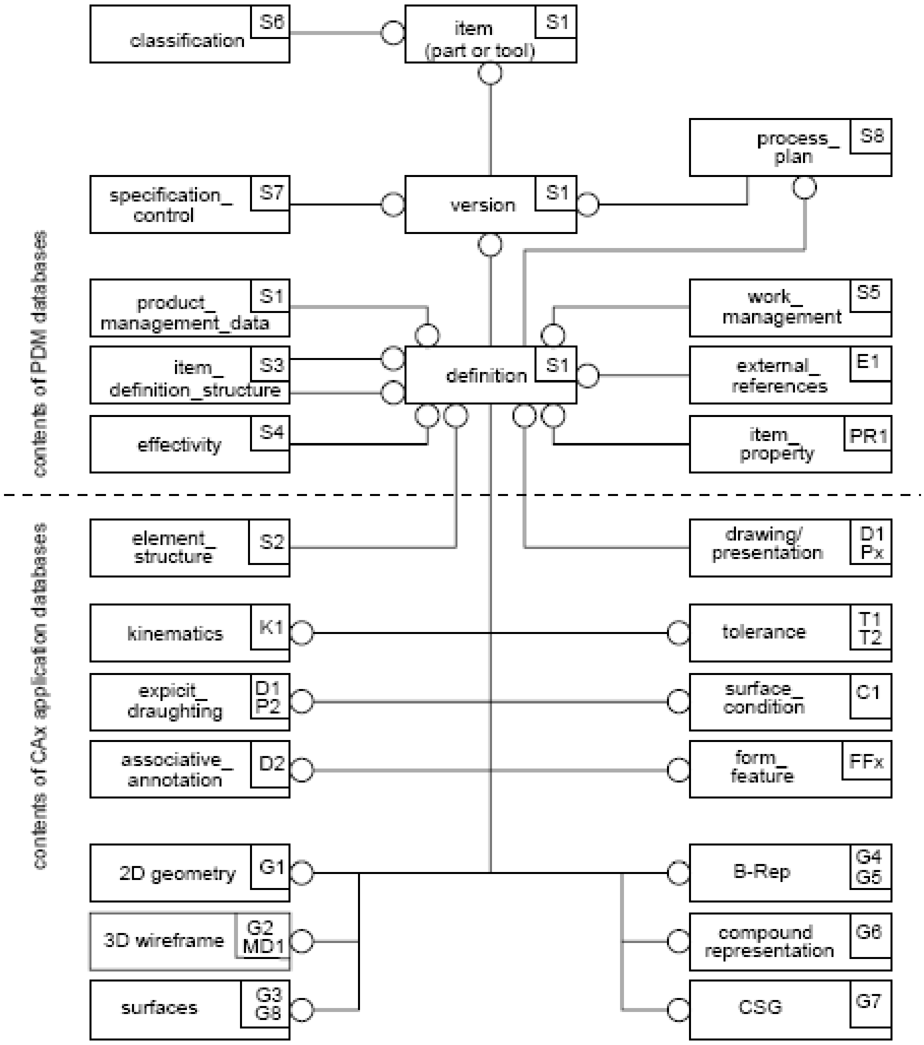


Fig. 3-3 Data planning model

The raw materials include those used to produce the parts or the tools;

— process plan information describing the relationships between parts and the tools used to manufacture them and to manage the relationships between intermediate stages of part or tool development;

— product definition data and configuration control data for managing large numbers of variants of automotive products during the design phase;

— data describing the changes that have occurred during the design phase, including tracking of the versions of a product and of the data related to the documentation of the change process;

— identification of physically realized parts or of tools, e.g., for prototype building;

— identification of standard parts, based on international, national, or industrial standards;

— release and approval data for product data;

— data that identify the supplier of a product and related contract information;

— the following eight types of representation of the shape of a part or of tool:

a) 2D – wireframe representation;

b) 3D – wireframe representation;

c) geometrically bounded surface representation;

d) topologically bounded surface representation;

e) faceted – boundary representation;

f) boundary representation;

g) compound shape representation;

h) constructive solid geometry representation.

— representation of the shape of parts or tools that is a combination of two or more of these eight types of shape representation;

— data that pertains to the presentation of the shape of the product;

— representation of portions of the shape of a part or a tool by form features;

— product documentation represented on drawings;

— references to product documentation represented in a format other than those specified by ISO 10303;

— simulation data for the description of kinematic structures;

— properties of parts or of tools;

3.2.2 Information requirements- Units of functionality(UoF)

This clause specifies the information required for core data for automotive mechanical design processes.

The information requirements are specified as a set of units of functionality, application objects, and application assertions. These assertions pertain to individual application objects and to relationships between application objects. The information requirements are defined using the terminology of the subject area of this application protocol.

The follows specifies the units of functionality for the Core Data for Automotive Mechanical Design Processes application protocol.

NOTE 1 - One or more UoFs, grouped together, provide a solution to identified data flows derived from the AAM (see 3.2.3).

This part of ISO 10303 specifies the following units of functionality:

NOTE 2 - Following the name of each UoF, its abbreviation is shown in brackets. The abbreviations consistof one or two letters and a number. The letters indicate a logical group to which the UoF belongs.

Figure 3-4 gives an overview for the UoFs, their logical groups, and the interdependencies among the UoFs and the logical groups.

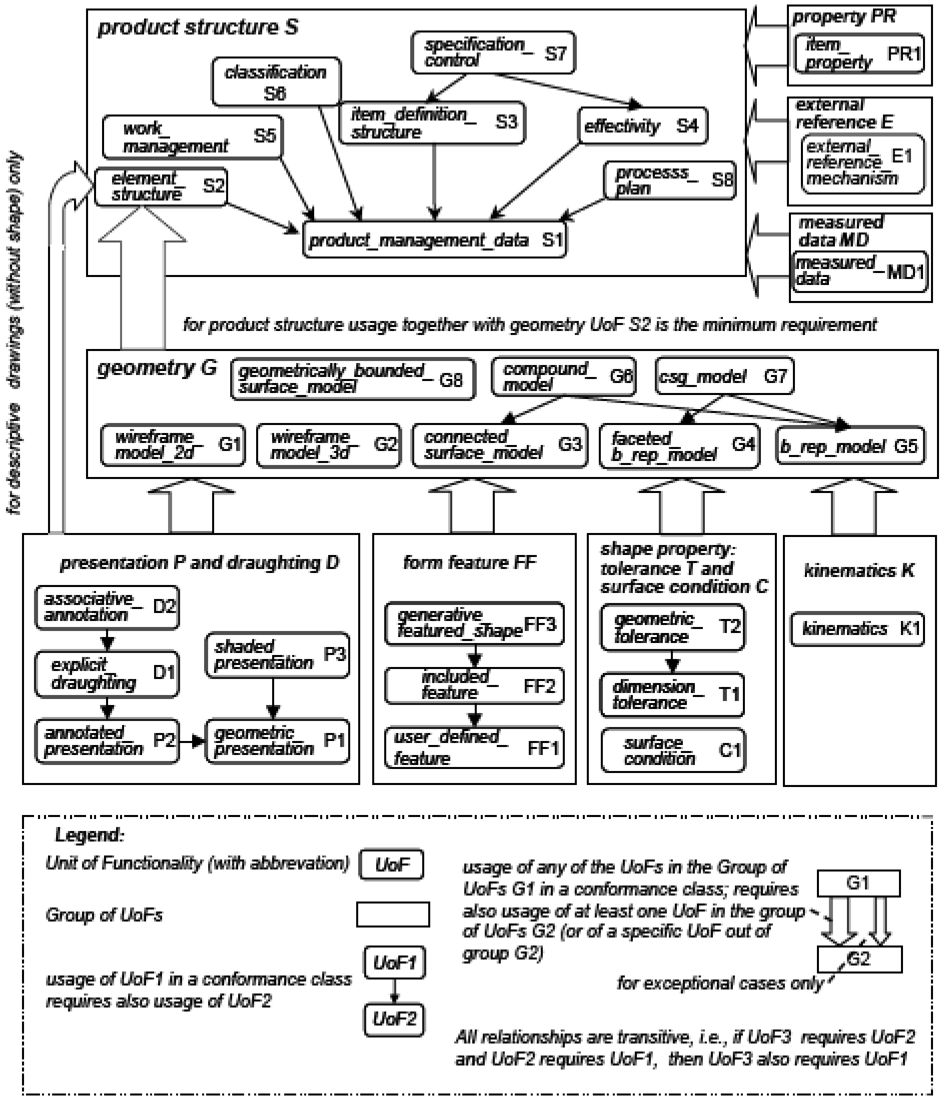


Fig. 3-5 Interdependencies among UoFs

3.3 Application interpreted model

3.3.1 Mapping Table

This clause contains the mapping table that shows how each UoF (see 3.2.2) and application object of this part of ISO 10303 maps to one or more AIM constructs (see 3.2.3). The mapping table is organized in five columns.

Column 1) Application element: Name of an application element as it appears in the application object. Application object names are written in uppercase. Attribute names and assertions are listed after the application object to which they belong and are written in lower case.

Column 2) AIM element: Name of an AIM element as it appears in the AIM (see 3.2.3), the term ‘IDENTICAL MAPPING’, or the term ‘PATH’. AIM entities are written in lower case. Attribute names of AIM entities are referred to as <entity name> : <attribute name>. The mapping of an application element may result in several related AIM elements. Each of these AIM elements requires a line of its own in the table. The term ‘IDENTICAL MAPPING’ indicates that both application objects of an application assertion map to the same AIM element. The term ‘PATH’ indicates that the application assertion maps to the entire reference path.

Column 3) Source: For those AIM elements that are interpreted from the integrated resources or the application interpreted constructs, this is the number of the corresponding part of ISO 10303. For those AIM elements that are created for the purpose of this part of ISO 10303, this is the number of this part.

Entities or types that are defined within the integrated resources have an AIC as the source reference if the use of the entity or type for the mapping is within the scope of the AIC.

Column 4) Rules: One or more numbers may be given that refer to rules that apply to the current AIM element or reference path. For rules that are derived from relationships between application objects, the same rule is referred to by the mapping entries of all the involved AIM elements. The expanded names of the rules are listed after the table.

Column 5) Reference path: To describe fully the mapping of an application object, it may be necessary to specify a reference path through several related AIM elements. The reference path column documents the role of an AIM element relative to the AIM element in the row succeeding it. Two or more such related AIM elements define the interpretation of the integrated resources that satisfies the requirement specified by the application object. For each AIM element that has been created for use within this part of ISO 10303, a reference path up to its supertype from an integrated resource is specified.

Table 3-8 Sample table – Mapping table surface_condition UoF (C1)

Application element	**AIM element**	**Source**	**Rules**	**Reference path**
COATING_LAYER	shape_aspect	41		{shape_aspect.name = ’coating layer’}
coating_layer to thickness_- dimensional_size (as thickness)	PATH			dimension shape_aspect<- dimensional_size.applies_to
coating_layer to material (as used_- aterial)	PATH			shape_aspect shape_definition = shape_aspect shape_definition characterized_definition <= shape_definition characterized_definition - material_designation.definitions[i] material_designation
CONTACT_RATIO #1: If the value_with_- unit is a	contact_ratio_- representation	214		contact_ratio_representation<= representation

numerical_value or a value_limit and theunit is not assigned globally. #2: If the value_with_-unit is a value_range				
…	…	…	…	…

3.3.2 AIM EXPRESS short listing

This clause specifies the EXPRESS schema that uses elements from the integrated resources (and the AICs) and contains the types, entity specializations, rules, and functions that are specific to this part of ISO 10303. This clause also specifies modifications to the text for constructs that are imported from the integrated resources (and the AICs). The definitions and EXPRESS provided in the integrated resources for constructs used in the AIM may include select list items and subtypes that are not imported into the AIM. Requirements stated in the integrated resources that refer to such items and subtypes apply exclusively to those items that are imported into the AIM.

EXPRESS specification:

```
*)
SCHEMA automotive_design;
USE FROM ISO13584_expressions_schema -- ISO 13584-20
(abs_function,
acos_function,
and_expression,
asin_function,
atan_function,
binary_boolean_expression,
binary_function_call,
binary_numeric_expression,
......
unary_numeric_expression,
value_function,
variable,
xor_expression);
REFERENCE FROM ISO13584_expressions_schema -- ISO 13584-20
(is_SQL_mappable,
is_int_expr,
used_functions);
USE FROM action_schema -- ISO 10303-41
(action,
```

```
action_directive,
action_method,
action_method_relationship,
action_relationship,
action_request_solution,
action_request_status,
action_resource,
action_resource_type,
action_status,
directed_action,
executed_action,
versioned_action_request);
```

…...

```
USE FROM aic_non_manifold_surface; -- ISO 10303-508
USE FROM aic_topologically_bounded_surface; -- ISO 10303-511
```

This follows describes fundamental concepts and assumptions related to the data structure defined within this part of ISO 10303.
ISO 10303-214 is designed to be used for data exchange, sharing, modeling, and storage in the design phase.
The intention is to gather information on a technical **product** related to various applications of product data.

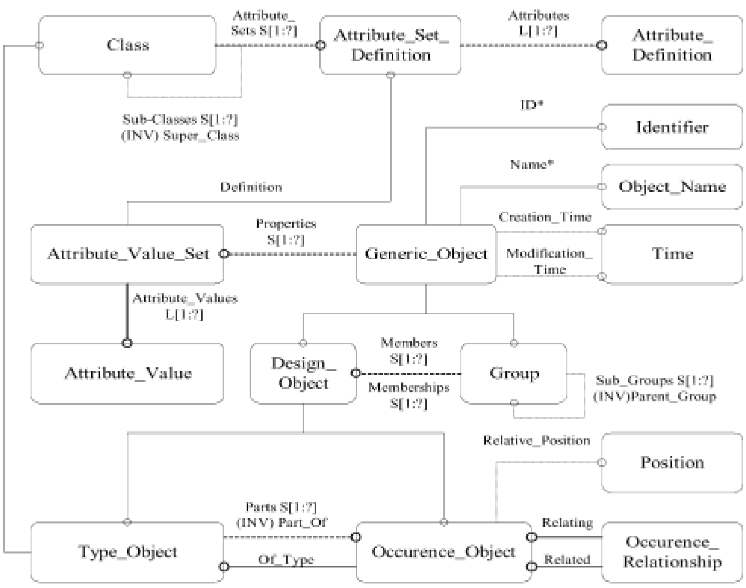


Fig. 3-6 EXPRESS-G of Product model

Example: Simple EXPRESS Example

EXPRESS specification:

```
*)
RULE restrict_process_product_association FOR
```

```
(process_product_association);
WHERE
WR1: SIZEOF ( QUERY ( ppa <* process_product_association | NOT
( ppa. name IN ['input' , 'output' , 'produced output' ] )
) ) =0;
WR2: SIZEOF ( QUERY ( ppa <* process_product_association | NOT
( ppa. name IN ['input' , 'output'] ) OR ( SIZEOF ( QUERY (
ar <* USEDIN ( ppa. process ,
'AUTOMOTIVE_DESIGN.'+'ACTION_RELATIONSHIP.'+
'RELATED_ACTION' ) | ar. name =
'process operation occurrence' ) ) =0 ) ) ) =0;
WR3: SIZEOF ( QUERY ( ppa <* process_product_association | (
ppa. name<> 'produced output' ) OR ( (
'AUTOMOTIVE_DESIGN.'+ 'PRODUCT_DEFINITION' IN TYPEOF ( ppa.
defined_product ) ) AND ( SIZEOF ( QUERY ( prpc <* USEDIN (
ppa. defined_product\product_definition.
formation.of_product ,
'AUTOMOTIVE_DESIGN.PRODUCT_RELATED_PRODUCT_CATEGORY.PRODUCTS'
) | prpc. name IN ['part' , 'tool' , 'raw material' ] ) )
=0 ) ) ) ) =0;
END_RULE;
(*
```

4 Implement of data exchange between CATIA & Unigraphic

4.1 STEP Compatibility of CATIA & Unigraphic

4.1.1 STEP Compatibility of CATIA

CATIA STEP Core Interface helps users exchange data through a neutral format in a heterogeneous CAD/CAM environment. You can interactively read and write data in STEP AP214 and STEP AP203 data formats allowing reliable bi-directional data exchange between dissimilar systems. To access to the data, CATIA Version 5 offers a homogeneous user interface for all supported formats, using Microsoft Windows-compliant user interface controls (such as File > Open, File > Save as) and automatic recognition of the STEP file type.

Features

- Supports for AP214 and for AP203.
- Windows-compliant access to STEP files.
- Supports of geometry and assembly structures.
- Ensures the transfer quality and reliability.

- High data transfer performance.

Key customer benefits

- **Supports for Application Protocol**

  The CATIA STEP Core Interface supports both the AP214 and the AP203. For AP214, the Class 1 and 2 are supported: the Class 1 and 2 elements are curves, surfaces, shells and solids. As for the AP203, the Class 2, 3, 4, and 6 are supported: the Class 2 elements are curves and surfaces, the Class 3 elements include curves and associated topology (edges), the Class 4 elements include curves, surfaces, and associated topology (faces) and the Class 6 elements are exact solids.
  By using the CATIA STEP Core Interface 1 (ST1), you have access to a high level of data for an efficient work with dissimilar systems.

- **Access to STEP files**

  The CATIA STEP Core Interfaceprovides integrated interoperability with STEP data formats through File Open, File Save As and Insert Component. For instance, users can read a STEP file, edit its contents in CATIA V5 workbenches, and save the results directly as a STEP file. The common and intuitive V5 interface make easier the use of STEP files, allowing simple and sure data transfers between systems.

- **Supports of geometry and assembly structure**
  The geometry and the structure of the original file are preserved:
  - The geometry is preserved at a high level (solids, shells). And the geometry is adjusted (without or with controlled deformation) so that it can be used in the best conditions in the CATIA V5 modeler.
  - If the STEP file contains an assembly structure (NAUO entities), then the structure is preserved: a CATProduct document is generated.
  - The AP214 mechanism of external references to STEP files is supported, in order to handle large and complex assemblies.
- **Transfer quality**

  The ability to import and export Geometric Validation Properties (GVP) embedded in the STEP files ensures the quality and the reliability of the transferred data. Indeed, in addition of existing OK/KO transfer information, some specifics properties on solids, products and instance of products, such as center of gravity, volume, or wet surface, can be included in the STEP file. During the import, according to the validation of these properties, a user knows if the data are reliable or if he or she has to perform some corrections to start with a consistent design (For AP214 only).

- **High data transfer performance**

The ability to handle very large STEP files, up to 150 MB, allows the users to exchange large designs, and the translator to do quick conversions.

4.1.2 STEP Compatibility of Unigraphic

Parasolid STEP in Unigraphics is a bi-directional translator between Parasolid and STEP-AP203. It can create STEP-AP203 files that represent Parasolid models and it can read STEP-AP203 files and can produce Parasolid models from them. Parasolid STEP supports STEP AP203 and AP214 Class II, class III, class IV, class V entities and class VI entities and assemblies. Parasolid STEP is a component that works with Parasolid and has a well documented programmable interface.

Features

- Designed to provide high quality exchange of geometric and topological data.
- Based on Parasolid geometry kernel.
- Parasolid STEP uses the body-checking and tolerant modeling capability of Parasolid kernel during the translation process.
- Satisfies strict conformance with the AP203 protocol. Output files have been checked by STEPCheck®
- Supports import/export of class 6 entities and assemblies in STEP AP203 files. (All Solid models)
- Parasolid STEP is architected as a component making integration within application straight forward.
- Facility to repair imprecise solids using Parasolid kernel features.

4.2 Data Exchange Using STEP between CATIA & Unigraphics

4.2.1 Problems

There are some problems in data exchange between using common files CATIA & Unigraphics:

4.2.1.1 When Using IGES File

First, it started out as an American standard, but standards organizations in other countries have created their own IGES-like product data exchange standards, such as SET in France and VDAFS in Germany. Second, all IGES translators seem to deviate from the IGES standard in some way. None seem to work the same and they require a lot of manual intervention to accurately exchange data between different CAD and computer-aided manufacturing (CAM) systems. Third, IGES was initially designed for exchanging strictly the two-dimensional geometric representation of a part design. Eventually, IGES could handle 3D data.

Fig. 4-1 shows the Original part

Fig. 4-2 shows the problems when translating form IGES to Unigraphics.

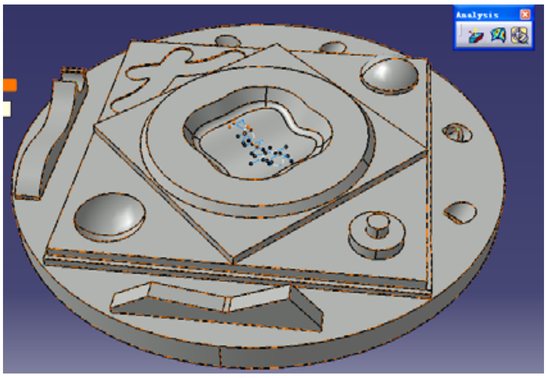


Fig. 4-1 Original part

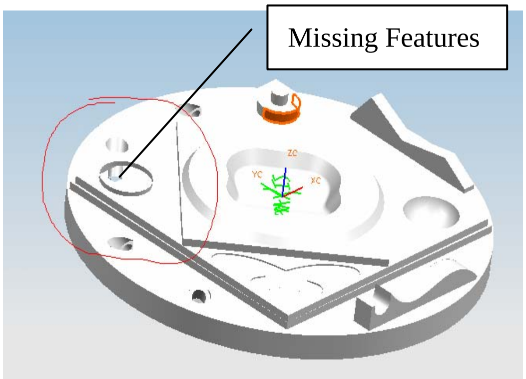


Fig. 4-2 the problems when translate form IGES to Unigraphics

4.2.1.2 When Using STEP File

Because the STEP translator in Unigraphics is developed by Steptools company, and the Dassault Co.'s CATIA is using themselves STEP translator, there are some differences in EXPRESS between CATIA & Unigraphics. Sometimes, there will be some troubles during translation between these two softwares on a individual condition.

4.2.2 Solution

When processing the STEP files that are difference between CATIA & Unigraphics, we use follow steps : (show in Fig. 4-3)

- According to the Data dictionary, which is abstracted after EXPRESS Compiling, we build a "STEP <->ROSE" **Data Exchange model**
- In **Data Exchange model**, we create agency ROSE database, and prepare the option in ROSE database
- In Data abstract model, we abstract data form STEP and abstract attribute by using user Expand Function.
- In order to generate legitimate STEP files, we must verify STEP files.

4.3 Instance of Data Exchange Using STEP between CATIA & Unigraphics

In demonstrating the Data Exchange Using STEP between CATIA & Unigraphics, two individual computers work on the CATIA system and Unigraphics respectively. We create a CATIA part and translate it into STEP AP214 file, and then verify the STEP file using the dataflow of Accidence Analyzer. At last the verified STEP file was imported into Unigraphic. (Fig. 4-4 shows instance of data exchange using STEP between CATIA & Unigraphics). We do the same from Unigraphics files into CATIA files. The result has proved that the translation accidence analyzer provides an effective and efficient technic and methods to translate file between CATIA & Unigraphics.

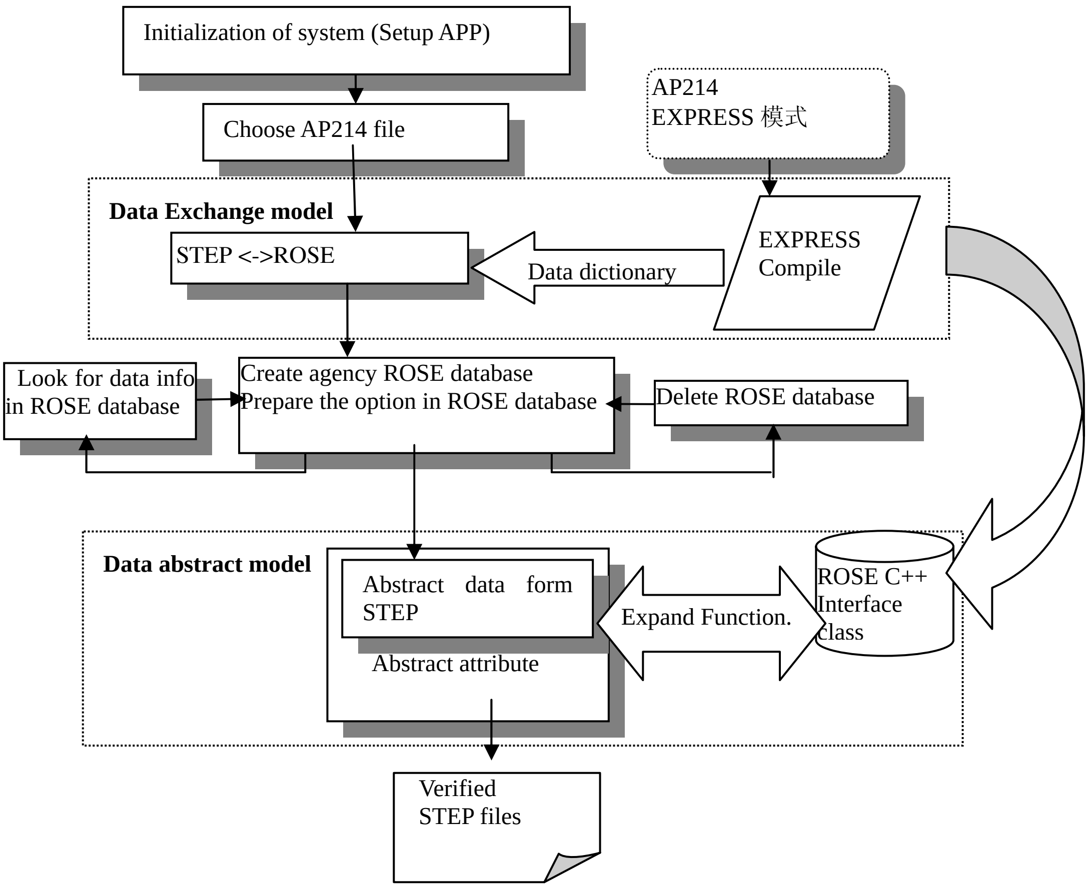
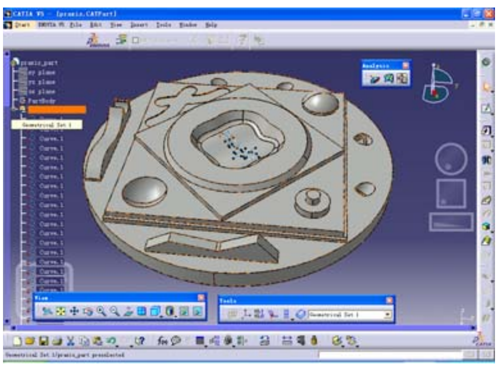
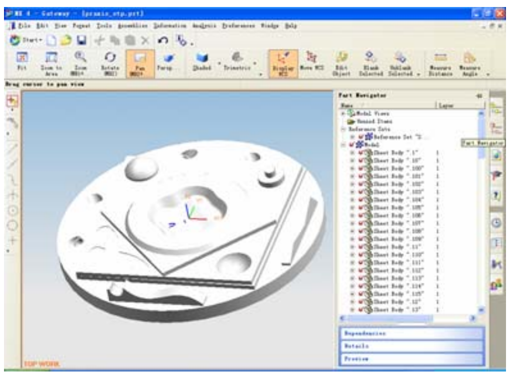


Fig.4-3 The Dataflow of Accidence Analyzer



a. a part in CATIA



b. Translate STEP file in to Unigraphics

Fig. 4-4 Instance of Data Exchange Using STEP between CATIA & Unigraphics

5 Following works

In the early development of STEP, web technology is not so popular as today. With the development of web technology, new requirements on product data have been satisfied. The main feature is collaborative in entire life cycle. Terms usually used are interoperation, heterogeneous and geographical disturbed system, life cycle support process.

Follow works are below:

1) Establish service of data exchange interface definitions among local & different construct CAD/CAM systems.
2) Development of data exchange framework for Web-Enabled collaborative design and manufacture.
3) Research and development into product data real time exchange system among different sectors and different construct CAD/CAM
4) Embody the implement technique and methods of service in Web-Enabled collaborative design and manufacture.
5) Research on STEP-NC (ISO10303 AP238, ISO 14649). (STEP-NC see Annex A &B)

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Annex A:The STEP-NC AP238 Standard

(From http://www.steptools.com/)

The STEP-NC AP238 standard is the result of a ten year international effort to replace the RS274D (ISO 6983) M and G code standard with a modern associative language that connects the CAD design data used to determine the machining requirements for an operation with the CAM process data that solves those requirements.

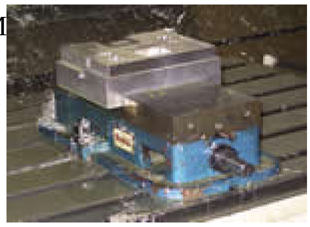
STEP-NC builds on the previous ten year effort to develop the STEP neutral data standard for CAD data, and uses the modern geometric constructs in that standard to define device independent tool paths, and CAM independent volume removal features.

Enabling e-Manufacturing for Mechanical Parts

Manufacturing organizations need to be able to seamlessly share information over the Internet. Using STEP-NC, machine shops can reduce "set up" times by up to 35% if they can seamlessly read the 3D product geometry and manufacturing instructions of their customers, and original equipment manufacturers can reduce the time they spend preparing data for their suppliers by up to 75% if they can seamlessly share the design and manufacturing data in their databases.

Working Steps

STEP-NC changes the way that manufacturing is done by defining data as "working steps": a library of specific operations that might be performed on a CNC machine tool. In other words, it breaks down every machining operation into the steps required to perform the operation. In the past, CNC machine tools had to be programmed using G and M codes (from ISO 6983), which were instructions that only told the machine what moves to make, without any semantic content that referred to the part being processed. STEP-NC will make G and M codes obsolete.



Advantages

With the concept of "working steps" in place, the manufacturing process becomes streamlined. Now, a machine tool can receive a file with STEP-NC data, "know" what it means, and proceed milling the piece without any more instructions. There will be no more programming the machine tool for each individual piece. Moreover, the benefit of the standard goes further. With a set of standard "working steps" in place, all manufacturers will be able to share information reliably and instantaneously. A STEP-NC converted CAD file that is completed on the east coast can be sent over the internet to a machine shop on the west coast and they can immediately start milling the part.

Annex B:Study on the Architecture of CNC Controller Based on STEP-NC

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Mechanics & Electronics Engineering college, Harbin Engineering University

Abstract **-STEP-NC is a new data model about CNC, currently under development by ISO TC184 SC1 and 4. Distinguished from the present CNC language (ISO 6983), the new data model includes rich information enabling feature-based programming and control for manufacturing operation. This paper aims to present a conceptual framework for a STEP-NC based CNC controler. The framework is derived from the analysis of information contents of ISO 14649, and the method to study the CNC system based on STEP-NC.**

Index Terms - STEP-NC, CNC, Architecture.

I. INTRODUCTION

The updating of standard for numerical control is a trend in manufacturing. In the past fifty years, the exchange of information based on ISO6983, namely use “G and M codes” to depict how to machining, facing the process of machining. But with the development of the system of CAD/CAM and the improvement of the capability of CNC control system, the shortcoming of the G-code is appeared: Programming language of it only offers the tool path of NC，not the processing of the work piece; ISO6983 has few the geometry information of the part, few process information; The problem within machining that appeared can not be solved in workshop, and the machines is too limited to control that procedure; Programming that standard define this exist ambiguously under the semanteme most situation; CNC supplier who carry on ISO 6983 define supplement outside to language, the same processing procedure can exchange at different controller; ISO 6983 ISO 6983 is difficult to support five axle mill sharpen and at a high speed cutting that curve process; Present NC just course of programming by CAD-CAM-CNC one-way information flow, any change done in NC code directly to mean this can be fed back to their early course automatically (such as the formulation course of craft planning of NC or part design process) [1]. This kind face sport and numerical control procedure limit CNC opening and intelligent development of this system, make CNC and CAX technology form the bottleneck too at the same time , hamper manufacturing industry's of machinery progress seriously [5].

STEP-NC (ISO 14649) developed by International Standards Organization (ISO) , is used for equipment to define STEP standard that the data expand for numerical control (NC). It define a kind of new interface in CAM between the craft plan system and NC control system, contain all message from products concept to finished product (part), including three-dimensional geometry information (STEP AP203, AP214), craft information (for instance mill, process, electric spark etc.), measure information (AP-219),and so on. STEP-NC has eliminated the back processors needed in the course of processing, can support more fast、 safe and intelligent processing equipment [3].

At present, the research and development of STEP-NC standard is still at the stage of studying, it studies to be world each numerical control equipment task of top priority of manufacturing enterprise even more to based on STEP-NC CNC system. This is that our country narrow the gap, develops domestic numerical control and even promotes a good opportunity of the automatic manufacture level of our country in an all-round way too. Research of this project based on ISO14649 standard , study on the basis of numerical control system of STEP-NC, and obtain the support of the important tackling key problem project of Heilongjiang Province.

Overview of STEP-NC Standard

STEP-NC is a data standard redefined for CNC system, it target -oriented form make design information of products message link up on the basis of STEP. STEP-NC define one new STEP use agreement AP-238 , as a standard of the data exchange between CAM and CNC .STEP-NC standard draft (ISO-14694) take shape basically already, and in revise among perfecting constantly.

STEP-NC (ISO14649) is a STEP expansion of standard, STEP allow in a different one and share and exchange this kind of data at the compatible computer platform, for how STEP-NC process CNC information increase to set up the standard with STEP products part file that mode describe. Whether in STEP-NC and STEP standard unanimous to geometry description method of entity, it made up of following part:

Part 1 Overview and fundamental principles

Part 2 Language bindings, Fundamentals

Part 3: Language binding in Java

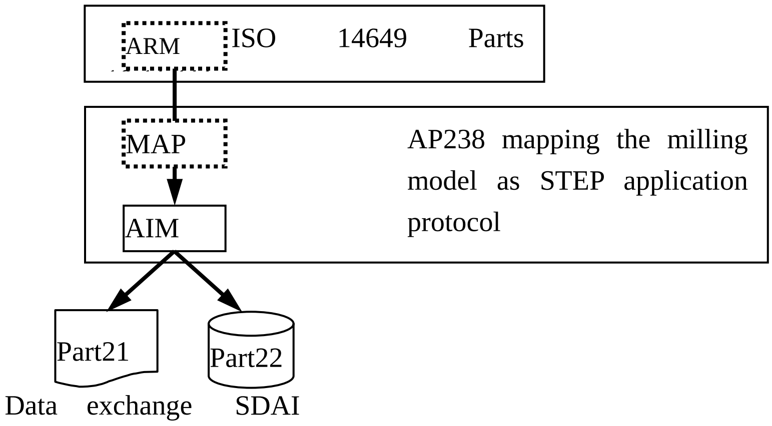

Fig. 1 CNC model is integrated in STEP-NC

Part 9: Glossary
Part 10: General Process Data
Part 11: Process Data for Milling
Part 12: Process Data for Turning
Part 13: Process Data for EDM，
Part 50: AIM of General Process Data,
Part 51: AIM of Process Data for Milling,
Part 52: AIM of Process Data for Turning,
Part 53: AIM of Process Data for EDM,
Part 111: Tools for Milling,

PART2, 3 is tied definitely according to the language of ISO 10303 method, PART10 whether 10303 ISO consult models to independent course application of datum, 10303 ISO consult to special application of technology model add after PART10.

AP238 describe method to realize STEP-NC, enable it to use the agreement to be compatible with existing STEP. It make the mill information of ISO 14649 map into other agreement definition, as Fig. 1. Integrated model defined to integrate is it is it sharpen message and STEP use the agreement to mill to mean (AP203, AP214 ,etc.), products that describe other part unanimous, can use CAD system and browser ,etc. that exist already

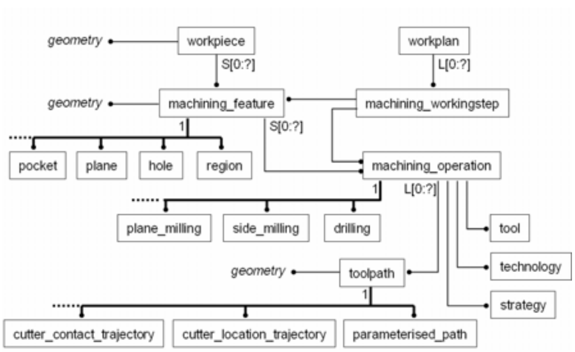

Fig. 2 EXPRESS-G of STEP-NC

Application Reference Model of STEP-NC (ARM) is ISO 14649 , and Application Interpret

Model is AP238. Application Interpret Model AP238 stipulates CAM system and CNC interface of system. AP238 is it Interpret model file namely use the model of consulting ISO 14649 difference information demand that part define to use, the application that changes into STEP-NC Interprets models, with ISO 14649 Parts10, 11,111 information based on the demand, is it consult model describe use agreement have anything to use, use and Interpret that the model describes how these contents on implementation method were expressed and exchanged, Fig. 2 is EXPRESS-G picture of STEP-NC.

The benefit of AP238 to use ISO 10303 in machining, and some agreements of STEP are set up on the same basis, synthesize two standard content, describe such as geometry of AP238 is the same as AP203 , AP214, feature description is the same as AP224, the public errand is defined the samely as AP219. So great to promote the development and application of STEP-NC, on one hand utilize the data of STEP to express models, reduce STEP-NC complexity of standard, can compatible each other with STEP on the other hand, in practical application , make to STEP interface some modification can discern STEP-NC. In the practical application , is it can use STEP datum model of agreement feedback among CAD directly to pinpoint the problems among CAM, realize the two-way flow of the data.

STEP-NC is it face target technology to use, set up new data model, new data cardinal principle of model according to processing characteristic programming, but not made the code of the movement order of the cutter directly originally. Emphasize and do something, but not how to do. "work step" (WORKSTEP) concept defines the processing course, has remedied the deficiency of ISO 6983, "work step" is the high-level processing concept , it face target and characteristic, describe processing of the work piece operate it, include the design data, processing characteristic , processing method and cutter information, etc.. It does not depend on the sport of the axle of the lathe , therefore it can be used in different lathe or the controller [3].

AP238 has offered the following function unit:

Project : STEP-NC main symbol of data, shine upon for different from it other product of classification define, have its edition controlling and relation with other projects instance ,etc. that are described too as the products.

Work piece: The work piece is expressed, describe all edition history of the products of a piece of STEP products, author's information, sanction, date and number of times, material , performance and form of work piece of work piece, products form express with senior B-REP form as AP224, any support CAD system of STEP can turn into.

Features: 2.5D milling features (the cavity, groove ,etc.), transition features(whether round border pour horn), repetition features, plane features and complicated feature, etc.

Executable : All execution is it for activity method subdivision , name processing course carry out to shine upon. Move and not approximate to it processes to be pieces of exception fast , is it operate to carry out. Besides subdivision, also including the cloth is expressed, the route , cut the package, carry out the course , middle geometry form , the procedure of processing and NC function.

Operation: Have four kinds of operations, Milling, Turning, EDM and Grinding . Describe state of machine, tool path tabulate, cutter direction wait for.

Milling Process Operations: Including mill subdivision of sharpenning each, adaptability controlled, mill the technology of sharpenning, course model, is it sharpen and meddle in tactics to mill, public errand.

Milling Cutting Tools: Cutting tools is expressed to come as one single complicated attribute , mapped as Representation. Cutter's every attribute is mapped. it is divided into embody and cutter size mapping.

The project, work piece , the characteristic , execution, operation, the tool path, size and operator function unit be used on the basis of Part 10 , the milling operation is to be based on Part 11 to mill, milling tools is based on Part 111- 1, turning is based on Part 12 for course to operate it to sharpen to mill, EDM course is operated and based on Part 13.

II. RESEARCH APPROACH TO TAKE

Research of this text purpose to STEP standard, study on the basis of numerical control structure of system of STEP-NC, one that is for STEP-NC general to use foundation of laying, explore relevant technology and method.

The research approach to adopt is: Start with studying STEP-NC standard, analyse STEP-NC application protocol , learn to set up the model based on process planning of STEP-NC according to STEP principle and method of standard especially, plan to craft AP238 file based on STEP-NC that model emerges carry on consistency measuring; Study through an analysis of SOFT type open numerical control system , confirm rational numerical control system develop overall scheme, set up STEP-NC controller model; Is it assess and test to go on to STEP-NC controller , prove controller and open numerical control compatibility of system this; Set up numerical control systematic STEP-NC database , make numerical control system set up the parameter of the lathe according to this database, analyse and carry out AP238 file ; Carry on STEP-NC controller , database and open numerical control system integration tactics and research of method.

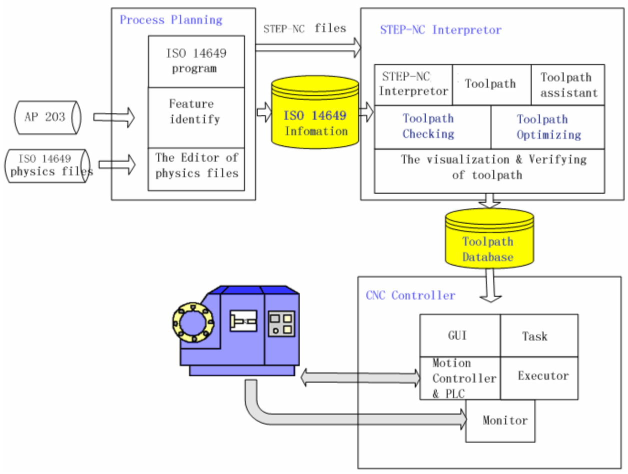

Fig.3 Process the procedure based on STEP-NC

1) To analyse STEP-NC Application Activity Model, Application Reference Model, Application Interpreting Model, use ST-Developer, ST-Plan tool of STEPTOOLS set up STEP-NC process plan models.

2) To adopt tools which offered by STEPTOOLS , and carry on consistency verify the AP238 file based on STEP-NC in order to guarantee that AP238 file produced accords with STEP-NC standard.

3) To study through an analysis of SOFT type open numerical control system , confirm rational numerical control system develop overall scheme, use open systematic API of numerical control, set up STEP-NC controller model. Including data research of interface, target of STEP,

concern database research of interface, EXPRESS-G picture describe STEP-NC controller model study , assess and test to STEP- NC controller, prove controller and open numerical control compatibility of system this.

4) To confirm EXPRESS describe database method that mode should be had , use SQLSERVER to set up systematic STEP-NC database of numerical control. Make numerical control system set up the parameter of the lathe according to this database, analyse and carry out AP238 file.

5) Carry on STEP-NC controller , database and open numerical control system integration tactics and research of method, carry on the system integration of numerical control on this basis, form the prototype of a machine of principle.

III. THE ARCHITECTURE OF STEP-NC-BASED NUMERICAL CONTROL SYSTEM

STEP-NC-Based Process the procedure as Fig. 3 shows, Plan to produce STEP-NC physics file in model through the craft at first, transform STEP-NC physics file into a cutter route that controls sports of the lathe through STEP-NC Interpreting device , control its control CNC to process through the lathe .

STEP-NC-Based numerical control system can read STEP file (AP238) and output in CAD/CAM system directly, so processor after the processing programming in traditional CAD/CAM has already disappeared in the STEP-NC-Based of system of CAD->CAM->CNC course of CNC, the substitute is put into the plug-in package of STEP-NC the direct output AP238 STEP file in CAD/CAM system. There will be a transition cycle from ISO 6983 in any case to ISO 14649, so based on STEP-NC numerical control system besides executable AP238 STEP file, executable traditional numerical control system executable G code of ISO 6983 also. So in the design based on numerical control system of STEP-NC, on one hand should consider that Interpret STEP AP238 file set up STEP-NC controller model promptly , keep ISO 6983 controller at the same time, And this controller is a subsystem which is regarded as CNC system, but not a part of STEP-NC controller model.

CNC system should have function that can revise AP238 file besides having corresponding STEP-NC controller , namely realize processing the data model and flowing two-wayly. Besides having information of processing because of AP238 file , also have information of geometry , tolerance information ,etc., So can't only depend on the characters editor to revise it , but need the intact one to have CAM system integration that AP238 file outputs the function in CNC system , require to opening , numerical control of system like this. So will choose OMAC open numerical control system, in order to carry on the secondary development to the numerical control system.

Only need to have three user function buttons in CNC system according to long-range objective of STEP-NC: Check-up , test run , running , Namely CNC system can carry out some crafts correlated with one's own function automatically while carrying out AP238 file, realize intellectuality is processed. But look from present angle, the degree of difficulty realized is relatively great.

We divide the system of numerical control based on STEP-NC into system of four layers structure: Grammar consistency check-up layer , Interpreting storey , processing the route check-up (can determine intelligence as to analyze storey from the long-range objective) , carrying out storey (NC-Kernel). See Fig. 4.

Grammar consistency check-up one layer of AP238 files based on STEP-NC planning to emerge in model to the craft carries on consistency measuring , guarantee AP238 file produced

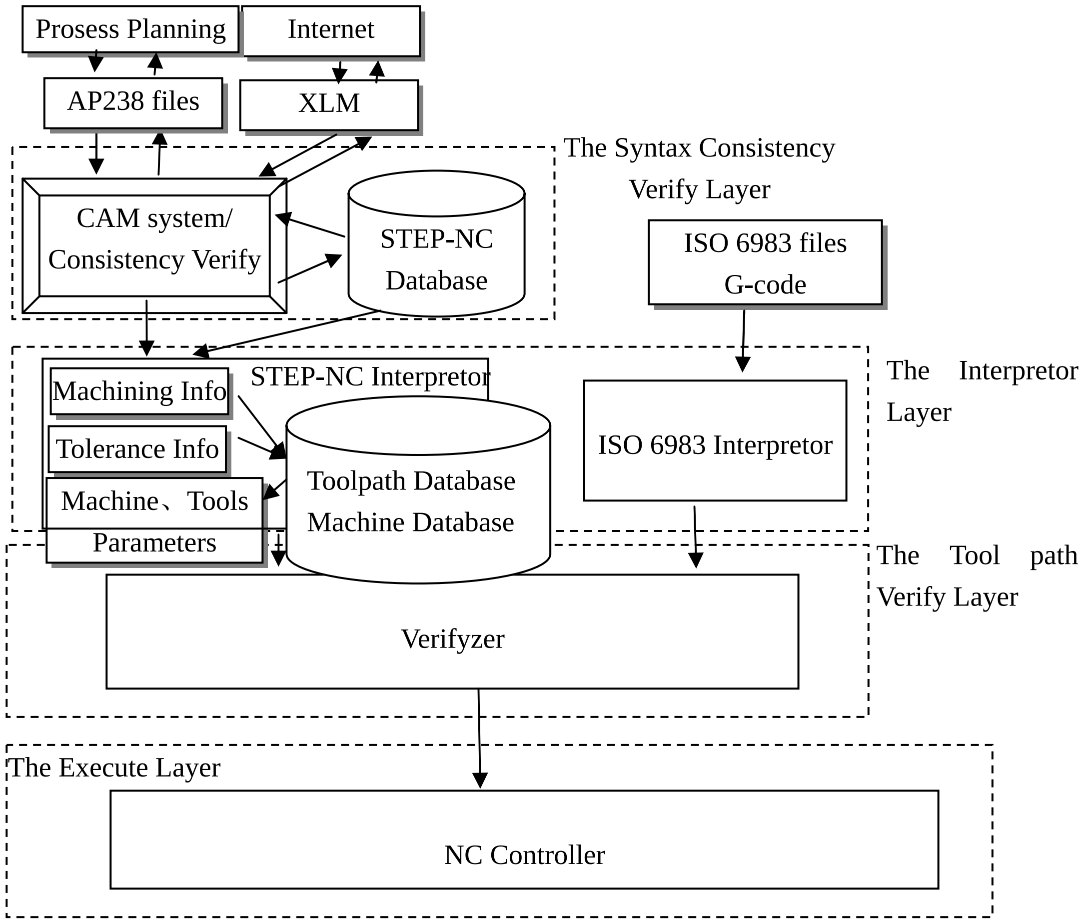

Fig. 4 System of four layers structure of the numerical control system based on STEP-NC

accords with STEP-NC standard.

The Interpret layer is to read AP238 file to responsible for , understand geometry information , public errand define , process message ,etc. or read ISO 6983 G codes directly, and turn it into order of the lathe .

Processing route check-up carry on analysis , check-up , test run to AP238 processing message that file produce.

The carry out layer and carry out and process the order , the processing work piece .

IV. Some Common Mistakes

The word “data” is plural, not singular. In American English, periods and commas are within quotation marks, like “this period.” A parenthetical statement at the end of a sentence is punctuated outside of the closing parenthesis (like this). (A parenthetical sentence is punctuated within the parentheses.) A graph within a graph is an “inset,” not an “insert.” The word alternatively is preferred to the word “alternately” (unless you mean something that alternates). Do not use the word “essentially” to mean “approximately” or “effectively.” Be aware of the different meanings of the homophones “affect” and “effect,” “complement” and “compliment,” “discreet” and “discrete,” “principal” and “principle.” Do not confuse “imply” and “infer.” The prefix “non” is not a word; it should be joined to the word it modifies, usually without a hyphen. There is no period after the “et” in the Latin abbreviation “et al.” The abbreviation “i.e.” means “that is,” and the abbreviation “e.g.” means “for example.” An excellent style manual for science writers is [7].

CONCLUSION

At present, it is infancy that it is studied as to the thing that based on numerical control system of STEP-NC in the world, only a few CNC controller manufacturers have participated in this research.

Our country has already joined “international World Trade Organization (WTO) ”, the manufacturing industry of our country, especially processing industry of numerical control face the keen competition of the overseas enterprise and severe challenge, if the manufacturing industry of our country can not catch up forthwith , will bring inestimable losses to the economy , technology of our country.

What this text studies proceeds from technological angle based of STEP-NC controller , study the scheme with feasible reality and develop the system of numerical control based on STEP-NC which suits our country's national conditions, fill the gap that our country is in this respect, narrow the gap with developed countries of foreign countries.

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