Strategies for Using STEP in Parametric Design

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Abstract-**The STEP stands for the standard for exchange of product model data. It represents ISO 10303. In the STEP, the parametric design methodology is supported by various Parts. The first Part is ISO 10303-55: integrated generic resource: procedural and hybrid representation. The second Part is 10303-108: integrated application resource: parameterization and constraint for explicit geometric product models. The third Part is ISO 10303-112: integrated application resource: 2D standard modeling commands for the procedural parametric exchange. And the last Part is ISO 10303-111: integrated application resource: construction history features. In this paper, these Parts are summarized in terms of parametric design supports and the relationships among these parts are analyzed. Based on the analysis, strategies for using STEP in parametric design are proposed.**

Keywords-STEP(ISO 10303); parametric design; product data exchange; neutral format.

I. INTRODUCTION

The main goal of the parametric design group in STEP (STandard for Exchange of Product model data, ISO 10303) is to accommodate procedural models in STEP and models constructed through explicit geometric constraints. [9, 10] To achieve these goals, the parametric group develops a framework for constructing and exchanging hybrid product models including explicit design and parametric design. Furthermore it extends the framework to accommodate evolutionary information throughout the life-cycle of the product. Under the framework, several Parts are developed in STEP. The Parts related to parametric shape representation in STEP are as follows: [5]

- ISO 10303-55: Industrial automation systems and integration - Product data representation and exchange: Integrated generic resource: Procedural and hybrid representation
- ISO 10303-108: Industrial automation systems and integration - Product data representation and exchange: Integrated application resource: Parameterization and constraint for explicit geometric product models
- ISO 10303-112: Industrial automation systems and integration - Product data representation and exchange: Integrated application resource: Modeling commands for the exchange of procedurally represented 2D (two dimensional) CAD(Computer Aided Design) models
- ISO 10303-111 Industrial automation systems and integration - Product data representation and exchange: Integrated application resource: Construction history features

In this paper, we first summarize the major contents of the parts. Then, the relationships among parts are analyzed. Finally, based on the analysis, we propose a guideline how to use the parts in parametric design to increase the productivity of design activities

II. PROCEDURAL AND HYBRID REPRESENTATION PARTS IN STEP

STEP (ISO 10303) Part 55 is provided for procedural and hybrid representation. The major content of the Part is as follows:

A. Scope

STEP Part 55 represents product models using operations in construction instead of composing elements. The basic element of procedural model is operation_sequence. Individual operations are represented by entity data types in STEP.

CAD models are usually represented by hybrid shapes including procedural elements and explicit elements. For example, explicitly defined sketch or profile can be related to a procedural sweep operation to constructed 3D (three dimensional) shape with volume. STEP part 55 provides necessary rules to represent such hybrid models.

The mechanism 'screen pick' is widely used by commercial CAD system operators to identify basic modeling elements for additional operations. The modeling element operation_ sequence provides a method how to represent 'screen pick.' In addition to the construction operations, deletion, modification,

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and query operations are also provided. The reason to offer such operations is that they are frequently used in construction and optimization of CAD models.

One of the goals of STEP Part 55 is to relate the 'design rationale' information to operation_sequence. In other words, it tries to transmit the 'design rationale' why the specific construction method is used.

STEP Part 55 also provides a method to connect procedural or hybrid model to explicit representation (for example, STEP Application Protocol 203). This is a typical example of parametric classes defined by procedural representation. The explicit models are used to verify reconstituted procedural models in the receiving systems. Basic procedural representation generally implemented to represent all kinds of models even if it provides independently special procedure for shape models.

STEP Part 55 specifies the resource constructs for the representation of models of the procedural or construction history types, defined in terms of the sequence of constructional operations used to build them. Representations of the operations themselves are not provided in this Part. The mechanisms provided in this Part allow the use of entity data types specified in other Parts of STEP for this purpose.

The following are within the scope of ISO 10303 Part 55. [1]

- The specification of sequences of constructional operations for the generation of any kind of explicit representations or models,
- Hierarchical structuring of constructional sequence,
- The embedding of explicitly defined elements in constructional sequences for the representation of hybrid models,
- Use of representation_item definition from other STEP parts to represent constructional operations for instances of those representation_item,
- The definition of a dual representation by association of a procedural model with an explicit 'current result' model, the latter acting as a representative example of the parametric family of models defined by the former,
- The association of design rationale information with procedural models,
- The identification, in a procedural model, of explicit elements selected by interactive picking from the visual display of the model in the sending system,
- Specialization of the foregoing capabilities for the procedural representation of shape models.

B. Goals and appropriateness

The goal of STEP Part 55 is to minimize information loss while exchanging models using STEP by transmit construction procedures instead of composite elements. Since such method is already widely used in modern CAD systems, procedural modeling capability in STEP is essential. Procedural models can accommodate design intents. The design intents inherently have parametric characteristics. We can easily reconstruct new models by changing parameter values of the existing models.

Fundamentally, if the receiving system has the sequence of constructional operations, we know how the model will be changed when modification occurs. In this case, we can use operators of the receiving system to understand the designed models conceptually. It also leads us to easy modification of models.

This procedural modeling is compatible to already existing STEP Parts. It also backs up the initial STEP objectives - 'STEP has to maintain complete and compatible information throughout the life-cycle of product while it represent product information in computer perceptible neutral.' Especially, this Part is inevitable to maintain the completeness.

C. Position in STEP

Related STEP documents to STEP Part 55 are Parts 41, 42, 43, 44, and 108. The capability of Part 55 can be extended to modular concepts of all STEP AP (Application Protocol). Part 55 is included in the integrated generic resources because of the following reasons:

- Procedural models in STEP are rather new concepts. Therefore it should be treated equally importantly as geometric and topological modeling methods.
- STEP Part 55 offers completely new concepts instead of specialization of the existing resources of STEP.
- The procedural representation can be applied to whole applications of STEP.

Relationship between STEP part 55 and standard for part library (PLIB, ISO 13584) should be clearly established. Even if PLIB Part 31 provides the capability of procedural modeling, it limited to represent parts family. The capability was installed to apply to very specific parts library, not for the compatibility with functions of modern CAD systems. In other words, it cannot be applied to represent hybrid models with procedural and explicit models constructed in CAD systems. It also cannot be applied in recognition of 'screen pick' information, which is essential in CAD models.

PLIB modeling capability is limited to 2D and 3D wireframe models and CSG(Constructive Solid Geometry) models. It also can accept very limited geometry provided by STEP Part 42. On the other hand, STEP Part 55 can be naturally compatible with the existing STEP Parts including STEP Part 108. Therefore, to utilize the originally intended modeling capability of PLIB Part 31, the procedural representation method of STEP Part 55 must be applied to entities defined by the existing STEP documents.

D. Major Schemas Included

STEP Part 55 includes the following two schemas: [1]

- procedural_model_schema: schema to represent design intents and procedural and hybrid models.
- procedural_shape_model_schema: specialized schema to specify geometric models.

D-1. Procedural_model_schema

The procedural_model_schema provides representation methods specified in the scope of this Part. The resource constructs provided in this Part generally used in representation, sharing, and exchange of procedural and hybrid models.

Originally, the major goal of STEP Part 55 is to provide a method to represent geometric shapes in procedural models as they were constructed in the initial CAD system. Therefore many examples given for this schema consider CAD modeling aspects.

D-2. Procedural_shape_model_schema

The procedural_shape_model_schema provides the following representation methods:

- Specifications of the sequence of construction operations for creating shape models,
- Representation of construction operations for instances of entity data formats used in shape modeling in other STEP Parts.

This resource constructs are defined in procedural manner. This general tool can be used in representation, exchange, and sharing hybrid shape models. The entity data formats defined in procedural_shape_model_schema are specialization of corresponding entity data formats in procedural_model_schema. The general description of procedural_model_schema can be applied to this schema.

III. PARAMETERIZATION AND CONSTRAINTS REPRESENTATION PARTS FOR EXPLICIT PRODUCT MODELS

STEP Part 108 is about parameterization and constraints for explicit geometric models. This Part specifies parameterization and constraints for rigid geometric product models as a part of integrated application resource class. This Part includes the following schemas:

- Parameterization schema,
- Explicit constraint schema,
- Explicit geometric constraint schema,
- Assembly constraint schema,
- Sketch schema.

A. Needs for Development and Concepts

Modern CAD systems construct product models with parameterization and geometric constraints. Parameterization, i.e. the relationship between variables and size or other values in the model, informs what values can be changed. Constraints, i.e. enforced relationships among geometric or topological elements, provide unchangeable properties in the model. Constraints are usually provided to maintain the product functionality during the exchanges in product models. These two capabilities enable the designer to generate alternatives and to cope with feedbacks from the downstream activities, such as manufacturing, maintenance, etc. It eventually provides flexibility in cooperative concurrent engineering.

Parameterization and constraints together compose an important element known as design intents. [7, 8] Relationships between parameters and constraints for a specific model make it easy to modify models when the model is transmitted to a different CAD system. Designers can modify the model in the same manner as it was constructed. If a model is exchanged without such information, modification in the receiving system is very difficult or impossible because details of construction processes are missing. On the other hand, if design intents information is exchanged, modification in the receiving system is very easy. STEP Part 108 is developed to provide capability of neutral format exchange of design intents information.

One of the parameterization based design and constraints based design can be implemented in a CAD system. The first method is to relate parameters to explicit elements in the model and to define constraints by relationships among sets of modeling elements. For this purpose, the basic representation can be defined by geometric and topological elements specified in STEP Part 42.

The second method is based on construction history. In such procedural or history-based approach, sequential operations are stored when a product model is constructed. In this case, parameters in the constructed model are input as variables of construction operations. The model can be modified by changing the input variables and re-executing the original sequence of construction operations. Constraints can be implicit in the construction process. For example, if a sequence of operations is given to construct a parallel plane to a certain plane, purely history-based defined model does not have explicit faces, edges, or vertices. Such explicit elements are not created until the operations are executed.

Most of the commercial CAD systems use the combination of above mentioned two methods. Only for parameterization and constraints, STEP Part 108 provides 2D explicit sketches which are often used as a basic element of 3D shapes defined by procedural operations.

In the most of CAD systems, explicit models are constructed with specific property values for parameters. Such model is referred as 'current result' in STEP Part 108. The current result is used when shape of the modeled product is displayed during design process. The displayed model supports user to select elements during construction and modification. The selected element interacts with the CAD system. The current results also used in engineering analysis such as volume or mass calculation and other internal geometric calculations by the CAD system.

In the first method mentioned above (explicit types using parameters and constraints), constructed current result model includes solution derived by constraints relationship in the CAD system, The current result model corresponds to effective geometric configuration of the model and it provides a specific indicator to the solution selected by designer.

In the CAD system using the second method (history-based approach), the current result is created by performing series of construction operations with some basic values for parameters. Even if a single theory can be applied to any CAD models, many CAD systems use their own solid modelers with different procedural representations. When a specific model is modified in a CAD system, the current result of the model is usually deleted and a new model is constructed by re-executing the procedures in the original construction history. Transmitting the current result without procedural information means loss of design intents. Designers who use the receiving CAD system cannot have original design intents and relationships among geometric elements in the model. Consequently, once the model is modified in the receiving system, the model does not contain the original design intents.

As mentioned before, STEP Part 108 partly avoids the loss of design intents by providing a mechanism to relate parameters and constraints to elements of the explicit model including 2D sketch and 3D current result. Integrated generic resources of STEP only allow static representation of a model. On the other hand, Part 108 provides some mechanisms to transmit modeling activities to the receiving system.

B. Scope

STEP Part 108 defines resource constructs along with necessary mechanisms to relate parameters and constraints to their explicit elements of the CAD model. Provided capability can be used to enrich the constructed static model using STEP integrated generic resources. The capability can accommodate some of the dynamic natures of design activities in the CAD systems. The following are within the scope of STEP Part 108: [2]

- Parameterization of explicit models expressed by the association of variables with quantities occurring in them,
- Constraints on explicit models expressed as relationships between their constituent elements or their attributes,
- The representation of constraints defining mathematical relationships between parameters,
- The application of parameterization and constraints to explicit models in general, to create variational models of products, plans, processes or organizations,
- Specific representations for geometric constraints commonly used in product shape modeling,
- 2D and 3D variational models of shapes and assembled products;
- The representation of 2D geometric sketches with parameterization and constraints,
- The representation of incompletely defined models in the sense that certain values in the model may be regarded as not fully constrained.

IV. MACRO PARAMETRIC CONCEPT

Macro Parametric (MP) includes design intents and parametric information in exchange of 3D CAD shape models. STEP Part 108 cannot completely transmit the design intents and design history during the exchange of 3D CAD models. To solve the problem, MP concept is proposed and implemented partially in STEP Part 112 for 2D models. In this paper, we fully investigate the MP capability.

Macro files in the most of CAD systems are created with modeling information when the model is constructed. MP technology utilizes the macro files to fully transmit the designers' intents to other CAD systems. The benefits of the MP technology are as follows:

- Shape information is transmitted in terms of series of commands user used instead of explicit boundary representation.
- Parametric information is automatically transmit through commands since the modeling commands themselves include parametric information and the command is a basic element in data exchange
- Design intents are usually included in feature shape and design history. Since MP uses design history based method and the basic element of data exchange is modeling command, design intents are completely transmitted.
- The most prominent benefit of MP is the ease of the translator development. Since the basic element of MP is publicly available modeling commands, a translator can be easily developed by anybody without CAD system development experience.

The side effects of the MP technology for Korean Industry are as follows:

- Leading the international standard: Once the set of standard modeling commands is completely specified, we can lead the ISO international standardization activities.

- Development of MP Translator: By leading the standardization of commands, we can develop MP translator for the first time in the world.
- Dissemination of 3D CAD models: By allowing Korean manufacturing companies to use 3D models with low cost, we can utilize 3D CAD models in design and manufacturing.

Such useful MP technology is developed in 3 steps as follows:

A. *The 1st Step*

To exchange CAD models using MP, standard modeling commands should be defined. Modeling operations performed by a user are included in the macro file of a CAD system. To extract standard commands, first we collect modeling commands of various CAD systems. Then we group them based on the similarity of the function. The command groups are the initial solution for the classification of the modeling commands. The groups are tested through actual modeling processes. By evaluating the test results, we can finalize the grouping for the standard commands. The standard commands are mapped to commands of each CAD system. Using CAD models constructed by the commands, we perform experiments for model data exchange. The results of the experiments are used for refining the standard commands.

The group of standard modeling commands is used in developing MP translators. To develop MP translators, first we define mapping relationships between the group of standard modeling commands and the group of modeling commands in commercial CAD systems. The relationships are used in verifying the MP translators. The MP translators are tested by selected experimental CAD models. The conceptual structure of a MP translator between SolidWorks and CATIA is given in Figure 1. Both SolidWorks and CATIA are well-known commercial CAD systems.

MP method can also be used as a standard to represent parametric information in 2D drawings for architectural area. For 2D application, STEP Part 112 is developed. Part 112 defines sketch based 2D parametric modeling commands. The baseline concept is the same as the MP method. The scope of the Part 112 is as follows: [4]

- Entities representing creation commands for geometric elements such as lines, arcs, chamfers and fillets;
- Entities representing transformation operations such as rotations and translations.

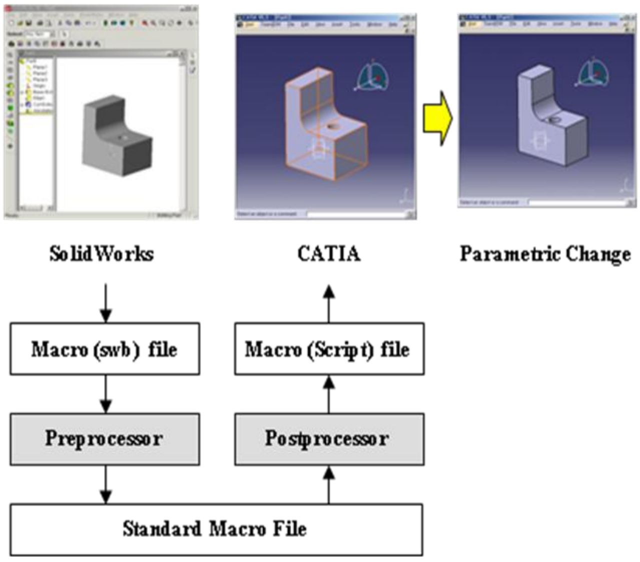

Figure 1. Conceptual structure of macro-parametric translator

B. *The 2nd Step*

In the most commercial CAD systems, API(Application Program Interface) is provided for language based program interface. However possible entities to be translated are limited to the capabilities of API. Therefore the capability of translation based on the MP method depends on the parametric information provided by macro files of a specific CAD system. If the macro file does not include parametric information, design intents cannot be exchanged. In this case, API should be used for exchanging parametric information. We call it a hybrid method. The hybrid method uses MP methods as a basis and uses API as a supporting tool. The hybrid method can be used in exchange of product model data as given in Figure 2.

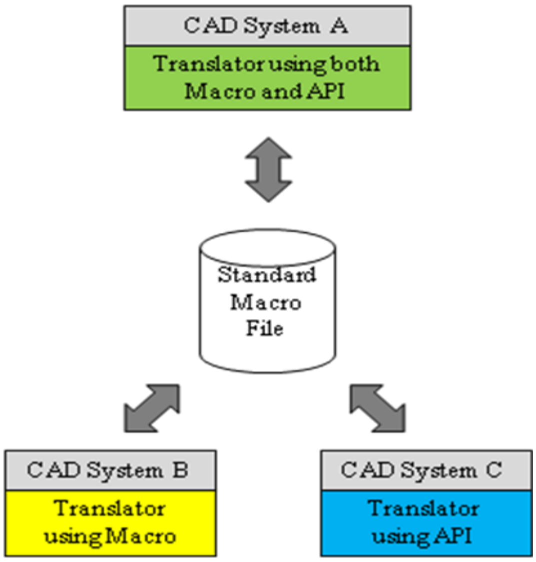

Figure 2. Product data sharing environment using hybrid translator

C. The 3rd Step

The parametric design system is developed using standard modeling commands and MP translators. MP transformation system is based on MP data transformation system as given in Figure 3. The MP transformation system also includes STEP and VRML(Virtual Reality Mark-up Language) model translators and parametric CAD model translators. To complete the system, additional modules are also included such as macro file clean-up module, geometry healing module, parametric data verification module etc.

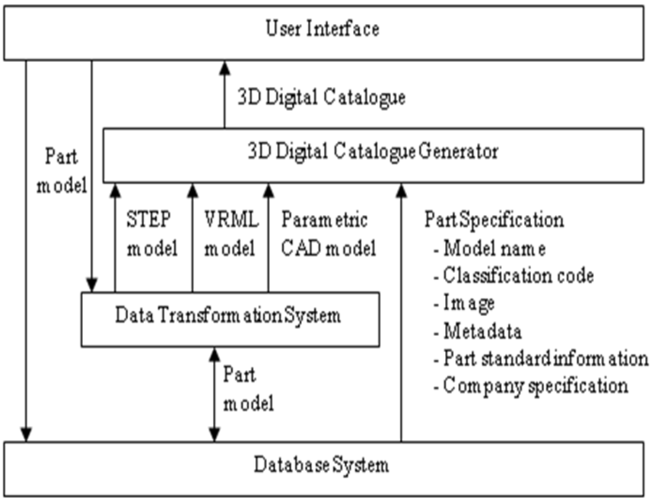

Figure 3. Macro-parametric transformation system

V. PARAMETRIC FEATURES FOR CONSTRUCTION HISTORY

STEP Part 111 provides a group of integrated solid modeling design features that provide the capability to exchange a set of feature-based CAD modeling operations. A key fact about the definition of a modeling or design feature is that it operates on or with geometric definitions to create or modify geometric definitions, so that a solid modeling feature would create another solid entity definition. [3, 6]

The entities in Part 111 are associated directly with operations in a solid modeling CAD environment. For example, an entity that represents a type of fillet is related to a CAD operation involving filleting, and which results in new solid model. [3]

The sequence of operations used to create a design is referred to as construction history or procedural modeling. Part 111 should to be used in conjunction with STEP Part 55: Procedural and hybrid modeling, which provides the mechanisms to capture the sequence of operations. The Scope of Part 111 is as follows: [3]

- Features that represent operations that capture filleting and chamfering;
- Representation of features that enable the results of offsetting, thickening, and shelling operations;
- Features that model operations associated with several types of holes, pockets, and slots;
- Representation of patterns of features, such as circular and rectangular arrangements.

A. Background for Part 111

Part 111 provides a method to identify construction history in the level of database definition. The data is retrieved in the form of being initially stored by the CAD systems. The reason is to focus on the necessity of archiving and exchange of models.

To represent construction history, different CAD systems use different methods such as parametric modeling, feature based modeling, etc. Even if many methods are used in representing construction history, we assume that the design intents model can accommodate all the methods in terms of archiving and exchange of models.

B. Needs

Construction history is functionally similar to the traditional CSG method. However recent solid modeling systems use different terminologies. Some terminologies are used to describe operations on features. For example, constructing fillet on a solid object is recognized as an addition of fillet feature to the solid. A manufacturing feature 'hole' is constructed by subtracting 'hole' primitive from the original shape. The hole primitive is described by setting necessary parametric values.

In the solid modeling, a fillet feature is constructed by modifying the original shape using one or more new faces or surfaces and several modified faces. On the other hand, fillet feature in manufacturing is provided as construction information with the original shape. The construction information enables the automatic manufacturing of the fillet.

This Part distinguishes concave blend surface and convex blend surface used in STEP AP214 and AP224. The latter is a fillet and the former is round. However, there is no difference in solid modeling. A fillet surface is just a bending of two different surfaces. The angle between two surfaces does not affect the feature naming. Solid model itself has information about inside and outside. In fact, Dassault Systems described the possibility of constructing a fillet surface that is concave in the one end and the other end is convex.

The discussion point here is difference of model construction methods between solid modeling and modeling for manufacturing. In solid modeling, new solid model is constructed for fillet feature. The fillet feature for manufacturing is just a property assigned to the solid body. Therefore, fillet data for AP214 and AP224 should be defined by a subtype of shape_aspect. This data is an entity that provides partial product definition information about specific part of a product shape. The entity shape_aspect relates faces or surfaces that have fillet property and provides the property data for defining radius of the fillet surface.

In STEP Part 111, construction history module is designed not to conflict with existing usages of feature information. In simple case, this strategy works well. However, this method generally provides biased data definition which cannot lead to effective data recognition. Also, it is hard to extend the method to general solid modeling.

Using fillet example, we can explain the reason. In the existing methods, feature definition describes the method specifying the radius of the fillet. In the design history module, similar method is used to transmit the radius. Edges to be filleted are connected using subtype in STEP Part 55. The subtype makes it possible to represent many properties related to the entity as a simplified on real number representing the radius.

In solid modeling, more complex mechanism is needed. The definition of a fillet with one radius value can be associated to an operation with edges to be filleted. However, existing STEP Parts cannot accommodate such case. Even if variable radius filleting is possible in solid modeling, no clear and simple method to represent such case is given in existing Parts. Even if the limitation can be possibly fixed, the interpretation confusion cannot be completely removed within the existing Parts. This leads to differentiation of requirement for property centric manufacturing information and requirement for construction sequence. Current STEP Parts does not accommodate the differentiation.

This is the basis for STEP Part 112. The Part 112 accommodates the capability to represent construction history model. The construction history model relies on the procedural and hybrid representation schema (Part 55) which represents the construction history data. The Part 112 also uses explicit geometric data defined in Part 42 in representing the construction history.

VI. RELATIONSHIP AMONG PARTS

The parametric related Parts in STEP add functions of representing procedural design history to the existing explicit shape representation capability of STEP. STEP Part 108 enables parametric design using variational geometry method by adding constraint relationships among shape elements. However this method requires strong solver to solve the relationship equations simultaneously. It becomes more difficult to get a solution as the number of the equations grows.

To solve such problem, STEP Part 55 is provided, which enables procedural and hybrid representation model. The Part 55 provides a method to identify the sequence of design operations. The method enables construction history based design. The receiving CAD system simply re-executes the received design sequence from the sending system to construct the exchanged product model. Therefore this method does not need strong solvers. During the reconstruction, the receiving CAD system can generate explicit models so that the new model is compatible with the existing shapes.

The commands for procedural operations based on STEP Part 55 are defined in Part 112 and Part 111. The Part 112 defined 2D sketch commands based on the MP concept and the Part 111 defines construction history features. The Part 112 and Part 111 provide commands to be used in practical operations for procedural models. The relationships among Parts related to parametric shape representation are given in Figure 4.

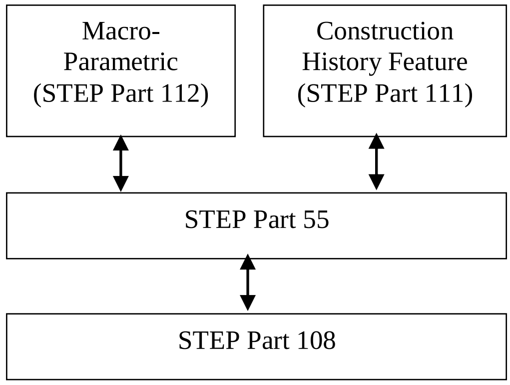

Figure 4. Relationships among STEP Parts related to parametric shape representation

A. *Comparison of 2D Modeling Commands Between STEP Part 108 and Part 112*

Both the sketch_schema of STEP Part 108 and the procedural_sketch_schema of Part 112 define 2D resources in the sketch plane for 3D model construction in the CAD systems. The sketch_schema in Part 108 defines elements and operations during sketch using other Parts of STEP. On the other hand, the procedural_sketch_schema in Part 112 provides a method to represent 2D modeling commands for exchange of parametric information of 2D models. That means the purposes of two Parts are different even if both schemas define resources in the plane. The entities from two Parts producing similar but different results are examined and analyzed in the following sections.

B. *Entity sketch_operate_intersect*

The entity sketch_operate_intersect defined in procedural_ sketch_schema of Part 112 provides construction commands to find the intersection points of two objects. This entity produces similar results to the implicit_planar_intersection_point or implicit_model_intersection_curve defined in sketch_schema of Part 108. However the entity sketch_operate_intersect is a construction command to produce all the points between two objects with lines or curves. On the other hand, the entity implicit_planar_intersection_point produces intersection points between 3D curves and the sketch plane and the entity implicit_model_intersection_curve defines the intersection curve between external reference surface and the sketch plane. Since the two entities are related to 3D model elements, two entities for intersection in Part 108 are out of the scope of Part 112.

C. *Entity sketch_reference_curve*

The entity sketch_reference_curve defined in procedural_ sketch_schema of STEP Part 112 constructs the reference curves as a conversion command. The entity converts an existing curve to a reference curve. The entity planar_curve defined in sketch_schema of Part 108 also selects a reference curve in the sketch plane. However the entity planar_curve can selects projected curve, which is originally 3D, in the sketch plane using the entity implicit_projected_curve. Therefore this entity is also out of the scope of Part 112.

D. *Entity sketch_reference_point*

The entity sketch_reference_point defined in procedural_ sketch_schema of STEP Part 112 selects the same type of planar point as the entity implicit_planar_projection_point defined in Part 108. However the entity sketch_reference_point refers a simple point in the plane. On the other hand, the entity implicit_planar_projection_point can selects projected point in the sketch plane.

E. *Comparison Conclusion*

The entities about elements constructed in the sketch plane defined by STEP Part 108 and Part 112 are different due to the discrepancy of scopes of the schemas and the objectives of the Parts. The scopes and construction methods are different between entities from the two Parts even if the similar modeling results are represented. As a result, the sketch_ schema of Part 108 and the procedural_sketch_schema of Part 112 are defining completely different geometric meanings.

VI. CONCLUSION

The parametric related Parts in STEP are investigated in this paper. Based on the investigation, we propose some strategies how to use the Parts in parametric design in producing CAD models. The major focus is given to the Macro Parametric(MP) concepts, which enables the exchange of design intents by transmitting the procedural design operations. Even if the MP is implemented in STEP only for 2D sketches, the concept can be applied to 3D CAD models. In the future, we expect the 3D MP concept is implemented in STEP.

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