ProSTEP iViP
Documentation

Final Project Report

Parametrical 3D Data Exchange via STEP

Version 2.2, 18.12.2006

Status: Final

Abstract

Studies show that approximately 80% of design solutions in the automotive industry are variant designs, respectively adaptations of existing solutions. The full potential for cost and development reduction inherent in variant designs cannot be implemented yet due to restrictions relating to the data exchange. Fully modifiable designs including parametric and feature information can currently not be exchanged across different CAx systems neither in a standardized nor in a proprietary way.

Despite the availability of qualitative STEP processors, data exchange partnerships between automotive OEMs and their key suppliers are increasingly based on a policy to exchange native CAD data. This policy is not cost effective and produces isolated islands of system specific data. Moreover, many process chains at OEM and supplier side are characterized by the need to bridge the shortcomings of data transfer between different CAD systems, which is time consuming and cost intensive.

The document contains usage scenarios for parametric CAD data exchanges. These where used to get an overview of the possibility for a parametric CAD data exchange between several common CAD Systems. It gives also a mapping between the parametrical parameters of the different systems and problems in the parameter exchange.

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1 Introduction

Studies show that approximately 80% of design solutions in the automotive industry are variant designs, respectively adaptations of existing solutions. The full potential for cost and development reduction inherent in variant designs cannot be implemented yet due to restrictions relating to the data exchange. Fully modifiable designs including parametric and feature information can currently not be exchanged across different CAx systems neither in a standardized nor in a proprietary way.

It should not be left unmentioned that some CAD vendors have first solutions for the import of CAD models stemming from a different system while maintaining the feature information (e.g. feature recognition in CATIA V5). But these approaches are unidirectional and neither full functional nor yet standardized. In addition, there are some recently published parametric related international standards. These are the parts 55, 108 and 109 of ISO 10303 (*Product data representation and exchange - STEP*) of which an overview is presented in the appendix of this proposal.

The project “Parametrical three-dimensional data exchange via STEP” aims at overcoming this restriction and to enable the exchange of parametric CAD models in a standardized (STEP-based) manner.

The project is a coordinated effort of a consortium of users and selected key vendors of CAD systems to extend commercially available STEP technology, to support concurrent engineering scenarios, and to allow the migration of design work between organizations and related IT platforms. The selected key systems are CATIA V5 from Dassault Systèmes, Pro/ENGINEER Wildfire 2.0 by Parametric Technology Corporation (PTC) and Unigraphics NX 2.0 from Unigraphics Solutions (UGS). A future amplification about other CAD systems is possible.

The approach of the project is evolutionary, i.e. it is a strong requirement that the derived solution is upward compatible to the available and productive STEP geometry exchange existing today. The evolutionary approach is chosen to protect existing investments as well as to allow a flexible usage of the added parametric capability. The extent of exchanged parametric information should be configurable and its usage would depend on the bilateral agreements of the exchange partners.

2 Members of the consortium

The project group “Parametrical three dimensional data exchange via STEP” is an user driven working group. The members of the users consortium (in alphabetical order) are:

- Audi AG
- BorgWarner Turbo Systems
- CEFE CAD/CAM-Entwicklungsgesellschaft
- Continental Teves AG & Co Ohg
- DaimlerChrysler AG
- Delphi Automotive Systems
- FH Augsburg
- FH St. Gallen
- KUKA Schweissanlagen GmbH
- PROSTEP AG
- Robert Bosch GmbH
- Siemens AG
- Volkswagen AG
- ZF Friedrichshafen AG

From side of the vendors the Parametric Technology Corporation (PTC) and UGS Solutions GmbH participate in the working group.

3 Current Situation

Despite the availability of qualitative STEP processors, data exchange partnerships between automotive OEMs and their key suppliers are increasingly based on a policy to exchange native CAD data. This policy is not cost effective and produces isolated islands of system specific data. Moreover, many process chains at OEM and supplier side are characterized by the need to bridge the shortcomings of data transfer between different CAD systems, which is time consuming and cost intensive.

The reason for the current situation is that at the moment neutral geometry exchange can only deliver explicit representations of product shape. This product shape does neither include parametrical nor feature information nor construction history. Thus, such models can hardly be modified in the target system.

From a technical perspective the current situation in neutral geometry exchange is characterized by transferring the explicit current result obtained by a sequence of interactions with the CAD system. This explicit result has a corresponding parametrical information, construction history and is also including features that all currently are not exchanged.

4 Objectives

Neutral data exchange especially via STEP has proven successful in practice. Various STEP processor implementations exist that allow the exchange of product geometry between all key systems in the automotive industry on a high quality level. Nevertheless, the problem with this data exchange is that it is not possible to edit the models on the destination side. Both, construction history and design intent, which is to some extent captured by parametric, are lost.

The main objective of parametrical three-dimensional data exchange is to enable the transfer of hence intelligent CAD models into a target system via STEP keeping the ‘intelligence’ of a parameterized feature-based model, i.e. a model that maintains the original relationships and that can be modified or edited. The goal is that geometry received via neutral file exchange or native system geometry can be handled almost equally.

The underlying rationale of this objective is to overcome restrictions that produce non-interoperable system specific process chains. In technical terms these restrictions can be overcome if the construction history together with parametric feature information can be exchanged. Recently published international standards of ISO 10303 will support such an exchange. This are the parts 55 (*Procedural and hybrid representation*), part 108 (*Parameterization and constraints for explicit geometric product models*) and part 109 (*Kinematic and geometric constraints for assembly models*). Not yet available as international standard is part 111 that represents construction history features (such as rounds, holes or chamfers) which are also essential for a complete parametrical data exchange where parts 108 and 109 build the essential basis. The common idea of these norms is to capture the intrinsic parametric in a system-independent representation in analogy to the successful definition of data models for explicit geometry in STEP.

Yet it is clear, that systems always have specific features and individual characteristics in definition of parametrical information. Therefore, it cannot be the objective of the project to leverage all CAD systems and their capabilities. According to the defined use cases by the project members the analysed main systems are CATIA V5 and Pro/ENGINEER Wildfire 2.0. The focus is rather to provide general purpose modification capability along the complete process and exchange chain and to transfer specialized parametric information. During this project we will analyse in the main the possibilities of a data exchange of constraints which are defined in part 108 and 109 of ISO 10303. To realize a complete parametrical transfer also other parts of the standard will be considered.

5 Usage Scenarios

During a previous project addressing the STEP-based exchange of parametric in 2004 at ProSTEP iViP association, Darmstadt, the participants identified their major requirements to such a neutral parametrical data exchange. In the meantime those requirements have been further detailed and consolidated in cooperation with FH St.Gallen, Switzerland.

Analysing their process chains the users identified the missing possibility to exchange editable parametrical models between the different CAx systems as the major problem for an efficient handling of CAD data along the product development process. Not only between OEMs and suppliers, but also inside a company between the different tasks of probably concurrent process chains using different CAx systems.

The users consortium defined the following use cases present in the enterprises:

1. The first use case contains the parametrical data exchange of standard and carry-over parts. It arised from the necessity to work with these parts in two different CAD systems. The intention is to construct parametrical models of these parts in only one system e.g. to simulate the delivery and assembly conditions in assembly models. To use these - to the higher requirements of the constructers adapted - parts in both systems a parametrical data transfer is needed to avoid extra work. This is the reason for the system evaluation carried out comparing the parametric capabilities of CATIA V5 and Pro/ENGINEER Wildfire 2.0 during the beginning of the actual project. To assess the feasibility of a standard-based parametric model exchange from a users point of view was the focus in this stage of the project.
2. The second use case resulted of the necessity to exchange parametric models of production equipment.

Inside these use cases the following requirements to a parametrical data exchange were defined:

- construction elements keep the correct order and their name (construction history)
- sketches control the three-dimensional geometry in the target system (internal sketches)
- sketches keep the dimension schema and constraints
- transfer of three-dimensional tolerances (shape tolerances) and notes
- transfer of rules, inquiries and formulas
- data exchange of model parameters and material parameters
- transfer of cosmetic threads and design features (rounds, chamfers, holes)
- order and names of elements in assembly models
- data exchange of position constraints in assembly models

All users stress that in any scenario their final goal is to have a complete but configurable model means to exchange editable models. Complete exchange of parametric is required since concrete usage scenarios use most of the parametric capabilities provided by the different systems. Configurable exchange is required to allow the selective protection of engineering know-how expressed in terms of parametric relationships between geometric entities and model parameters. But it is understood by the user consortium that in order to obtain fast practical results and due to descriptive and functional dependencies the development will have to be performed in a sequence of implementation steps. Thereby every step will build on the functionality and the best practice developed in the previous step.

Finally, it is commonly agreed that the exchange of parametric geometry will extend the capabilities of STEP-based geometry exchange. In a given business process the following exchange settings could be possible:

- transfer complete shape as explicit geometry (current state);
- transfer complete shape as implicit geometry (full parametric);
- transfer complete shape as both explicit and implicit geometry (redundant combination, for instance, to check correctness of the implicit representation);
- transfer complete shape as a hybrid model containing both implicit and explicit geometry to include explicit geometric elements into a history-based parametric data exchange (define complete shape with its construction history without the need to describe every single feature parametrically);
- transfer explicit geometry with configurable exchange of implicit behaviour (extrinsic parametric relationships between features) to control the degree of engineering know-how transferred.

6 Project Organization

Due to their importance for the user consortium in the defined use cases, the CAD systems CATIA V5, Pro/ENGINEER and Unigraphics NX 2.0 have been selected for the analysis of a STEP-based transfer of parametric models. In order to be successful it is the declared intention of the project participants promoting and pushing parametrical thee-dimensional data exchange. The user consortium will intend to collaborate with Dassault Systèmes, PTC and UGS whose commitment for an engagement in the implementation of the stated objectives is mandatory for a successful realization of the project.

Four aspects are essential for the organization of this project:

1. It is understood that it is a cross-sectional task to define specifications for system independent representations of parametric information. Such a cross-sectional task needs a joint effort. The budget for this effort is provided by a user consortium. This consortium as well defines the requirements for the parametric exchange technology. This fact reflects the business interest of the users.
2. Vendor commitment is crucial. This project is set up for cooperation with Dassault Systèmes, PTC and UGS. These three will be involved in early evaluation of the specifications. It has to be understood that information how parametric is handled in a given system is sensitive to vendors. They can not be expected to publish all the details and internal structures of their approach to parametric. Thus, this project anticipates that an initial specification is defined by STEP and CAx experts. In iterations of review cycles in close cooperation with the vendors, the initial specification evolves to a neutral specification which conforms with the involved systems CATIA V5 as well as Pro/ENGINEER, Unigraphics and the ongoing STEP activities regarding parametric.
3. The data integration and modelling expertise of ProSTEP and the existing forums for communication between users and vendors provides an ideal basis for such a joint project. Thus ProSTEP leads this effort and intends to collaborate with PDES, Inc. ProSTEP will be supported by the FH St Gallen and the other members of the user consortium.
4. The results of the project will be reported to the joint ProSTEP/PDES CAx Implementor Forum and coordinated with the work on ISO 10303. This will guarantee that parametric three-dimensional data exchange results are incorporated into the international standardization approaches towards parametric and features. Furthermore, once the results have proved its feasibility, it makes sure that they can be picked up by other CAD vendors in order to extend the availability of parametric STEP processors.

The project will provide a framework for a joint effort towards the ultimate goal, the commercial availability of STEP processors for the neutral exchange of parametric geometry.

7 Parametric Data Exchange

Based on the capabilities of the CAD systems CATIA V5, Pro/ENGINEER Wildfire 2.0 and Unigraphics NX 2.0 a comparative analysis in accordance to the project objectives has been performed to evaluate the feasibility of a STEP-based parametric data exchange between these systems (see appendix C).

From the comparison the following short conclusion can be drawn: over the last years the CAD systems are getting more and more similar with respect to their parametric feature set. Meanwhile, the systems follow a feature-based approach promising a successful realization of the project aim to exchange parametric information in a standardized way.

Of course, there are differences, which have to be considered carefully when developing a standard-based parametric information exchange. Comparable to standard-based geometry exchange, the internal data has to be mapped to the standardized description back and forth when reading and writing a standard file respectively. In case of standard-based feature and parametric exchange, the mapping process needs more semantics and interpretation rules compared to pure geometric data exchange. To successfully realize the standard-based parametrical exchange based on defined use-cases the committee for standardization of ISO 10303 published some new parts.

It is ensured by the project participants and project organization which follows a users' requirements driven approach, involving CAD vendors to realize the processors from their internal representation to the standardized model to be developed by the involved institutes considering current developments within the ISO.

The dependencies between the different building blocks of feature-based parametric models, e.g. 2D sketches, constraints and features, demand a certain strategy in order to achieve the project goals (see chapter 7.1).

In addition the user consortium indicated that the project should be organized in some functional stages with reviews of the project progress and success. In consequence we propose the following road map for implementing parametrical three-dimensional data exchange. This road map is ordered according to the technical dependencies and the priorities extracted from potential user scenarios.

7.1 Technical Challenges

In order to exchange parametric models four technical challenges have to be proved:

1. the definition of a neutral model describing general purpose construction history and the operations (features) applied which is realized with the international standard ISO 10303 part 55,
2. the identification of topological entities such as vertices, edges, and faces to which parametric modeling operations (features) have to be applied (consistent naming/identification mechanisms) which is realized with the international standard ISO 10303 part 42,
3. the exchange of sketches, their parameters and constraints which is realized since February 2005 with the international standard ISO 10303 part 108,
4. the handling of mixed models which include both parametric modeling operations and explicitly described shape within a construction history which is also realized with the international standard ISO 10303 part 55.

In reference to the defined use-cases also the part 109 for position references (position constraints) in assembly models, part 47 for geometric tolerances and notes in three-dimensional models, part 45 for material definitions and part 50 for mathematical formulas have to be included. All these parts of ISO 10303 are published as international standard.

7.2 Basic Parametrics

The aim of “Basic Parametrics” is to address the challenges stated above in order to prove the feasibility of a neutral exchange of editable parametric models via STEP. To accomplish this task the first project phase worked on solutions for the following aspects of parametric data exchange:

1. Construction history, i.e. the basic mechanism to model a sequence of parametric operations with parameters that may reference implicit topological entities like edges and faces.
2. 2D sketches, their parameters and intrinsic dimensional and geometric constraints: Sketches are the basic means for creating solids build up by features and parametric. For this topic, Part 108 builds a basis to be considered. In appendix C a proposal of a mapping between the defined STEP entities and the concerning elements of the CAD systems CATIA V5, Pro/ENGINEER Wildfire 2.0 and Unigraphics NX 2.0 is released.
3. Linear extrusions (additive, subtractive): To extrude a 2D sketch is the basic operation for creating a parametric model Swept solids and other geometric models are defined in ISO 10303-42.
4. Position constraints in assembly models: for the parametrical exchange of assembly models the position constraints defined in part 109 of ISO 10303 have to be supported. In appendix C a proposal of a mapping between the defined STEP entities and the concerning elements of the CDA systems CATIA V5, Pro/ENGINEER Wildfire 2.0 and Unigraphics NX 2.0 is published.
5. Simple blendings (linear chamfers and circular roundings and fillets): Although in most CAD systems blendings should be applied at the end of a modelling process (this is due to the robustness of parametric modifications), they nevertheless represent an important class of features which often needs to be adopted/modified along a process chain. Therefore, the standardization of ISO 10303 part 111 has to be forced.
6. Explicit boolean operations (union, intersection, difference).
7. Hybrid parametric models: A complete exchange of parametric models is mandatory for a successful parametric model exchange and for the users’ acceptance. On this account the implementation of ISO 10303-55 into an application protocol (preferring the AP 214) has to be supported.

It was the goal to analyse the state of standardization of the defined use-cases in the parts of ISO 10303 (see appendix A). In reference to topic 2 and 4 a mapping table is published in (see appendix C) this document.

7.3 Configurable Parametric Exchange

For most users it is crucial to have a means of filtering the parametric information to be exchanged, because in some cases it is not intended to give the receiver of a model full access to the engineering know-how stored with the model.

To this end, we propose a classification scheme for the constraints to allow the user to toggle the exchange of constraint information on a class by class basis. For instance, constraints that describe inter-relationships between features shall not be exchanged at will. We classify the constraints as follows:

- Position of a feature: Cannot be filtered, since it is needed to rebuild the model.
- Dimensions of a feature: Cannot be filtered, since it is needed to rebuild the feature.
- Algebraic expressions, which determine the position of a feature: Can be filtered, if the related parameters are fixed for the data exchange.
- Algebraic expressions, which determine the shape of a feature: Cannot be filtered, since they are needed to define the shape.
- Geometrical constraints (parallelism, perpendicularity, etc.): Can be filtered, if the related entities are fixed within a sketch.

7.4 Occurred Problems

During the analysis of the standard and carry-over parts provided by Volkswagen AG some problems occurred and will be presented in the following chapters. The main problems result from the different representation forms in the CAD systems and also from special features and functions, which only exist in one of the analysed systems. Thus, the possibilities to reach the project aims in respect to these features have to be discussed and possible alternatives have to be mentioned.

7.4.1 Perimeter Dimension

In Pro/ENGINEER it exists the possibility to define the exact perimeter of a spline in dependence to a variational dimension. The spline is defined with two basis points and reaches its final geometry by definition of tangential constraints to auxiliary geometry. In the analyzed model (see Figure 7-1) this function is used to define a spline in dependency to a radius dimension for simulation of the delivery and assembly conditions of the model.

The problem of data transfer is not the representation in the neutral STEP format, but this type of definition is not existent in the sketcher of CATIA V5. There is no possibility to define a variable dimension in CATIA V5. Also the perimeter of the spline is not dimensionable. A detailed graphic description of the problem is shown in Figure 7-1.

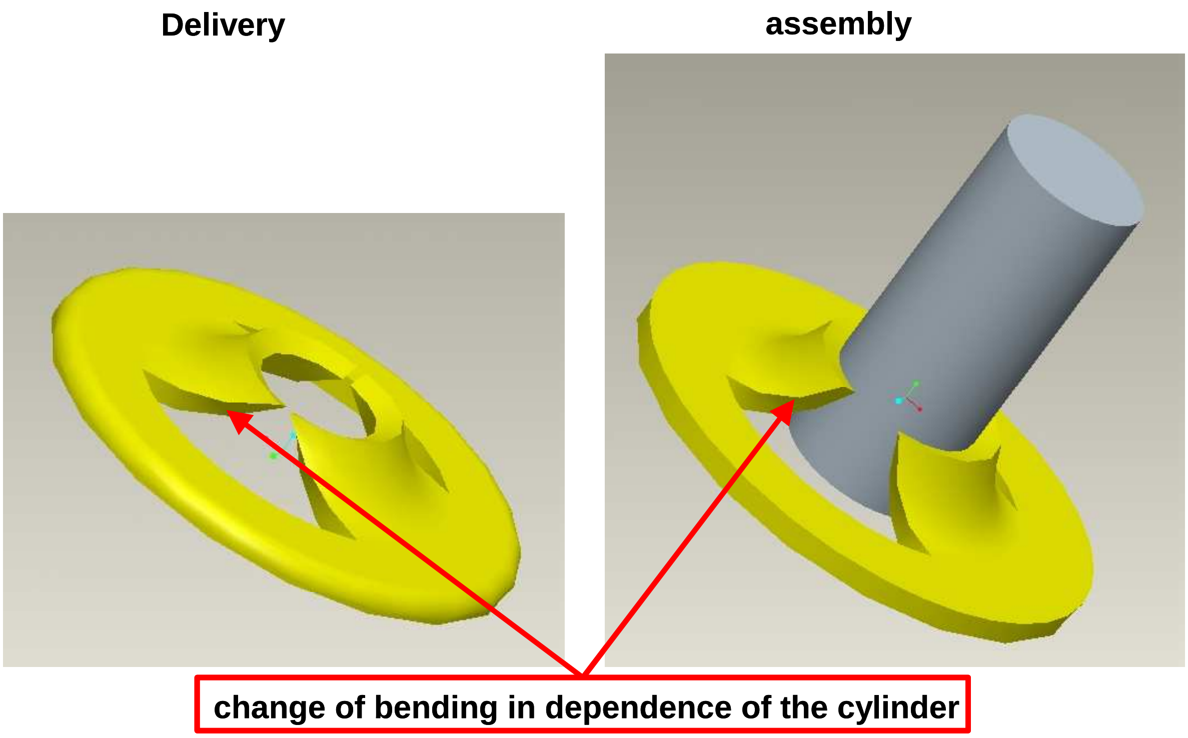

Figure 7-1: Visual presentation of the dependence of perimeter dimensions [Source: Volkswagen AG]

7.4.2 Predefined position constraints in parts

In Pro/ENGINEER the user has the possibility to predefine in the part the position constraints when adding these component to an assembly model. When positioning the part the constructers only needs to select the necessary reference elements in dependence of the in the assembly selectable models. In the shown plastic clip in Figure 7-2 also the parametric of the model is dependent from the selected reference element (the diameter of the cylinder shown in Figure 7-1). The problem in data transfer is that no adequate mapping to STEP entities and to CAD elements in CATIA V5 is possible.

7.4.3 Rules, formulas and inquiries

To support the data transfer of rules, formulas and inquiries the parts 50 (Mathematical constructs) and 51 (Mathematical description) of ISO 10303 have to be implemented to the pre and postprocessors of CAD systems. The only difference between CATIA V5 and Pro/ENGINEER Wildfire 2.0 is that in the workbench for knowledgebase engineering of the programs in CATIA different functions for definitions of rules, formulas and inquiries exist whereas in Pro/ENGINEER theses functions are abstracted in one function (constraints).

7.4.4 Construction history

Mentioned in chapter 7.2 part of the basic parameters to reach the project aim to realize a first parametrical three dimensional data exchange via STEP also the construction history has to be transferred. The Construction history defined in Part 55 of ISO 10303 has to be implemented in the same time with parts 108 and 109. Only with a common consideration of these standards the project aims can be realized successfully.

Also the exchange of construction history features such as blendings, chamfers, fillets or patterns have to be considered. Defined in part 111, which is published in CD ballot, in this context the standardization ambitions in respect to this part have to be supported. Therefore the mapping tables created in the first state of this project have to be expanded by the STEP entities defined in part 55 and 111 of ISO 10303.

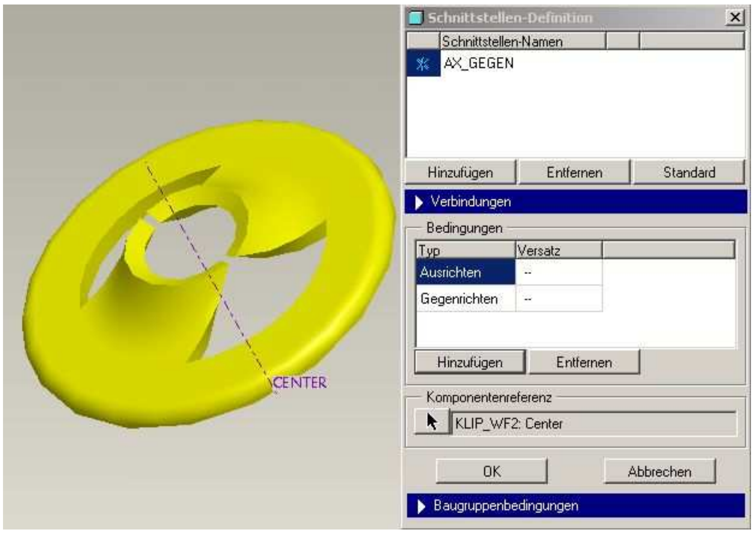

Figure 7-2: Predefinition of position constraints in Pro/ENGINEER [Source: Volkswagen AG]

7.5 Problems on Data Exchange

In this section known problems in the Data Exchange between UG and STEP are described. First the test case models are described and below the construction the occurred problems during the data exchange are listed.

7.5.1 Boolean Operation

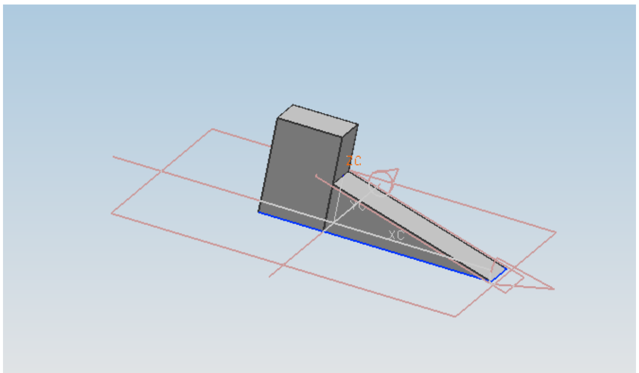

Figure 7-3: Boolean Operation

To model the above model as part (not as assembly) you need to do the following steps:

1. Extrusion with create parameter (possible flag(union, subtraction, intersection, create) => Solid 1
2. Extrusion with create parameter = > Solid 2
3. Trim (Split, Slice) of the second solid (Solid 2), please note, that the trim plane intersects the first solid.
4. Union of the first solid (Solid 1) with the second solid (Solid 2)

<u>Problem:</u>

Some CAD Systems can not create a boolean operation in the end, so it will be difficult to exchange the above model, without changing the tree. If the CAD system adds solid 2 direct to solid 1 the slice will trim both solids.

STEP, UG API can do this direct.

ProToolkit has to make a workarround with assembly.

7.5.2 Boolean Operation on Part References

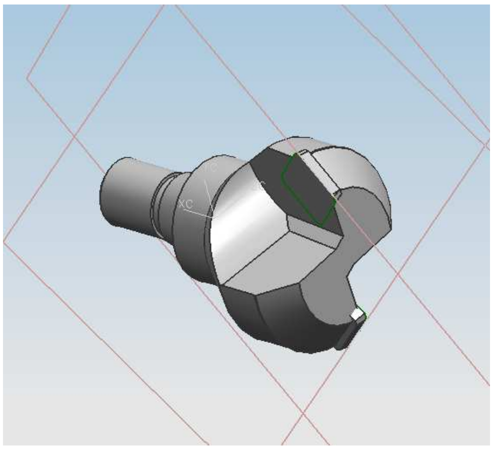

Figure 7-4: Boolean Operation on Part References

This model has been constructed with assembly. It consists of three parts.

Part 1: tool body
Part 2: flute
Part 3: insert

The flute is modelled positiv and is subtracted from the tool body. This can be done with UG API with WAVE_LINKED_BODY.

Problem:

Not all CAD Systems can do operations on an assembly. This problem is also in STEP.

7.5.3 Attribute (Property)

Every entity should have properties, this is important for defining for example the nose radius to be used in a CAM module. In UG this is possible, so the property NOSE_RADIUS can be assigned to an arc.

Problem:

STEP and others CAD Systems do not support this feature.

7.5.4 Twist

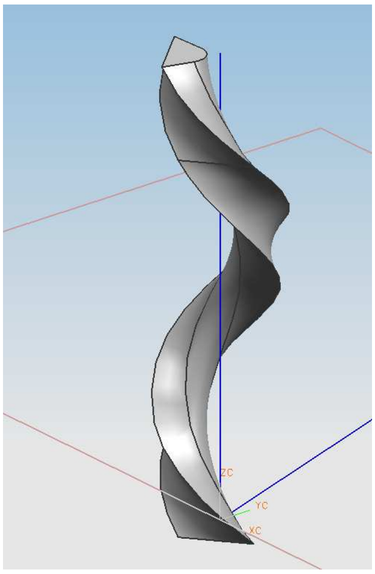

Figure 7-5: Twist

The twist feature can not be modelled on all CAD Systems. A Twist with an angle greater than 360° can not be modelled in UG, so separate solids have to be generated for every 360° degree using the twist featu re and then add them together. In this case the trees are not the same.

Problem:

In STEP there is no twist feature.

7.5.5 Create 3D Model without sketch

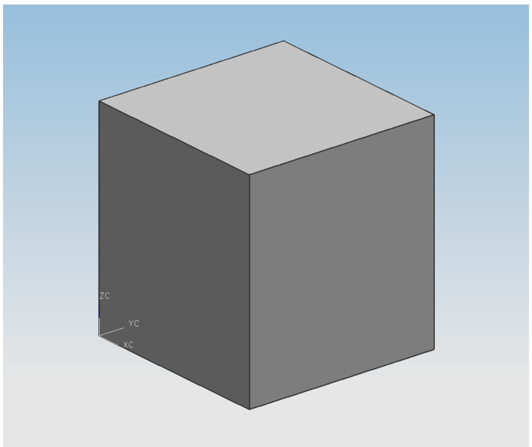

Figure 7-6: 3D Model without sketch

Models can be built direct with 3D model Feature, e.g. there is a feature box, which will generate a box.

Problem:

If the CAD System does not support this feature, a sketch and an extruion has to be generated.This will change the tree!

If the tree is changed, the parts will differ from each other. This problem can be fixed by forbidding the use of 3D direct modelling.

7.5.6 Symmetric Extrusion

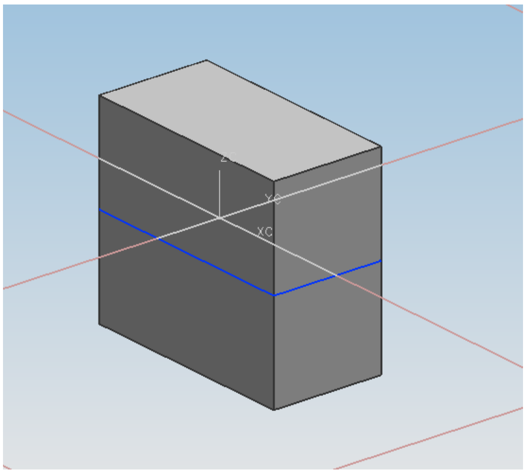

Figure 7-7: Symmetric Extrusions

Some CAD Systems support the symmetric extrusion. The extrusion is symmetric on the sketch, (10 mm in pos. and 10 mm in neg. direction).

<u>Problem:</u>

This feature can be reproduced by two extrusions, but it will change the tree. Also in STEP there is no symmetric extrusion.

7.5.7 Extrusion with LimitMin at -5 and LimitMax at 10

In some CAD System it is possible to make an extrusion, based on a sketch and a direction, with positive and/or negative values.

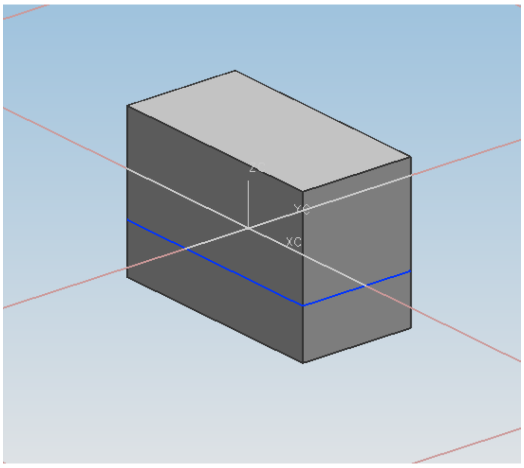

Figure 7-8: Extrusions with two limits

E.g. a sketch can be extruded 10 mm in pos. direction and 5 mm in neg. direction.

Problem:

This feature can be reproduced by two extrusions, but this will change the tree. Also STEP has no symmetric extrusion.

7.5.8 Slice Operation

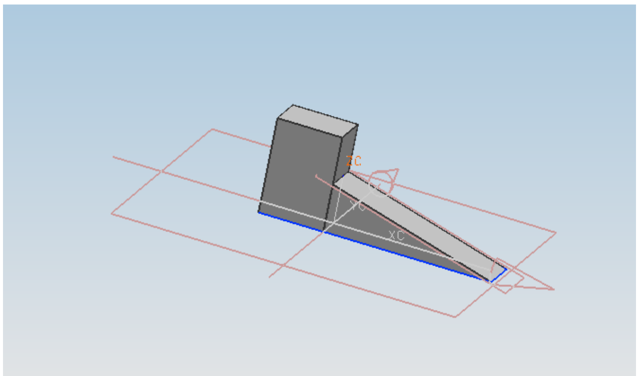

Figure 7-9: Slice Operations

Problem:

The slice (trim, split) operation can not be saved in STEP. A box has to be generated and then it is subtracted from the solid.

7.5.9 Extrusion with Taper Angle

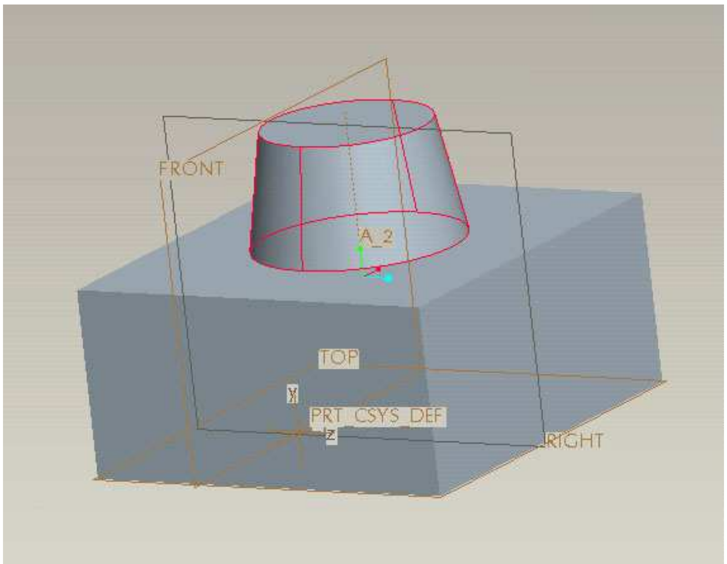

Figure 7-10: Extrusions with Taper Angle

The above model will be constructed in ProE with the following steps:

1. Create solid 1 (box)
2. Create solid 2 (cylinder)
3. Apply the taperAngle on solid 2

4. Load the two solids in the Assembly
5. Merge the parts
6. Create the masterpart

In other CAD System one has to do the following:

1. Create solid 1 (box)
2. Create solid 2 with extrusion with UNION parameter set and taper angle (cylinder)

Problem:

This means that the trees are different and that we can not exchange this part!

7.5.10 Constraints

In Pro/E one can not set Sketch Constraints directly with the API. The solver willl do this automatically and one has to delete all constraints which are not necessary. After that, there is no way to set new constraints.

7.5.11 Other problems

Not all CAD System have the full handling with API, that means that not all features, which can be done interactive can be done by programming.

The behaviour of a feature regarding assemblies is not the same in the different CAD Systems.

Mirror operation: In some system a copy does exist or can be defined. In other system the original will be referenced, so it is not possible to delete it.

The same problems occures with subtractions. In some CAD System the subtraction operation in the assembly causes also a subtraction in the part.

8 Use cases

1. Standard- and carry-over parts: This use case aroused from the necessity to work with standard and carry-over parts in different CAD systems that are in use during the design process. Parametrical models are necessary e.g. to simulate deliver and assembly conditions of springs in assembly models. The intention is to construct parametrical models only in one system, but have them available in all systems. To avoid extra work during data exchange it is necessary to transfer parametrical models. (see figure for an Normpart example Figure 8-1: Analyzed Normpart [Source: Volkswagen AG]).

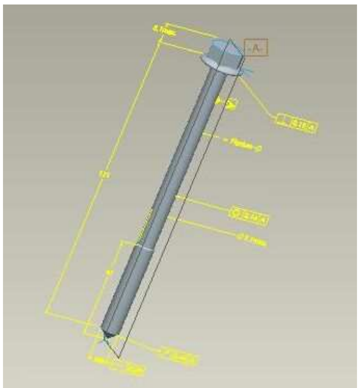

Figure 8-1: Analyzed Normpart [Source: Volkswagen AG]

2. Supplier-OEM partnerships: Suppliers need to satisfy the demands of different OEMs. Therefore it is necessary to have the designs available in different required data formats.

3. Migration Projects: When the main CAD system used in one design department is being changed to a different one, it is necessary to migrate the CAD data to the new format. In order to keep the models editable they need to be parametric in the target format. An example of a migration project is shown in Figure 8-1. After migrating the Turboloader into CATIA V5 and Pro/ENGINEER the Turboloarder had to be extended along the spriral defined in the original Turbolader.

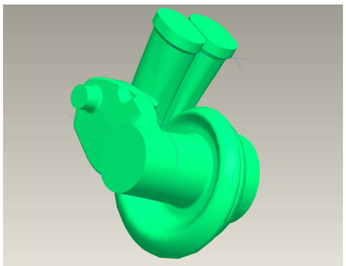

Figure 8-2: Turboloader original [Source: Brog Warner]

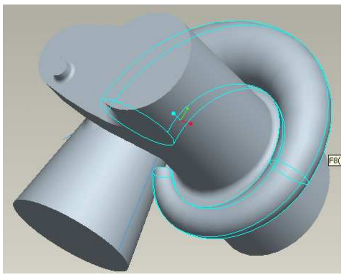

Figure 8-3: Turboloader after migration to Pro/E

Figure 8-4: Turboloader after migration to CATIA V5

9 Activities in the project group

In this project group some different activities were planned to force the realization of parametrical three-dimensional data exchange.

1. Activities around Part 108 and 109

Part 108 (parameterization and constraints for explicit geometric product models) is the basis for other parametric related activities. Since February 2005 part 108 is published as international standard. Tightly coupled to Part 108 is Part 50 (mathematical constructs) which is heavily influenced by the requirements of Part 108. Since December 2004 part 109 (kinematic and geometric constraints for assembly models) is released as international standard.

In the appendix of this document a mapping table of in to these parts concerning STEP schemes defined entities is published. A integration of the necessary entities in an product data model on base of the short form of AP 214 and the compilation of the independent long form are objectives of this project. On base of this mapping and generated long form of AP 214 an implementation in the processors of the CAD systems is possible.

2. Support of standardization of Part 111

The Construction History Working Group (led by Bill Anderson and Mike Pratt) is a group of people interested in that topic. They didn't have financial support; thus progress was rather slow. They had some test cases where they could show first results of parametric exchange between certain CAD systems. Our impression is that NIST and some processor developers are trying hard to push parametric exchange technology but vendor support seems to be little Nonetheless, this group – and their intermediate results - has to be considered seriously within this projects activities.

10 Summary and Forecast

Of course there are differences in the internal representation of the different CAD systems, which have to be handled carefully when mapping to the defined STEP entities. But because of the conclusion that these systems are very similar in respect to the parametrical representation and feature-based approach it is possible to realize an exchange of parametric information in a standardized way.

To reach the project aim for a parametrical data transfer the support of the project participants by the vendors in implementation of the new published international standards in the processors of the CAD systems is crucial. Also the encouragement of OEMs to realize first parametric data exchange and in future projects when expanding to more complex use-cases must be guaranteed.

In respect to this project a coordination with the ambitions in standardization of part 111 of ISO 10303 (Construction history features) is necessary because of the importance of this part for the future realization of a parametrical data exchange in an acceptable quality for the users.

As a basis for future implementation the mapping tables in respect to the defined STEP entities of part 55 and part 111 are provided for the CAD systems Unigraphics, Pro/Engineer and CATIA V5. Continuing the implementation of parametric three-dimensional data exchange a future scope for any implementer must be the implementation of construction history, internal sketches and constraints in one step. Also the implementation of the defined units of functionality (UoF) standardized in the application protocol 214 must be forced in implementation.

Annex A: Stand of standardization

In the following table the stand of standardization and implementation into a product data model on basis of the application protocol 214 or the CAD-Systems is documented.

STEP Part	ISO status	Implementation in			
		AP 214	CATIA V5 processors	Unigraphics NX 4.0 processors	Pro/ENGINEER processors
45	IS	Yes	Yes	Yes	Yes
47	IS	Yes	No	No	Partially
50	IS	Yes	No	No	No
51	IS	Yes	No	No	No
55	IS	No	- - -	- - -	- - -
101	IS	Yes	No	No	No
108	IS	No	- - -	- - -	- - -
109	IS	No	- - -	- - -	- - -
111	CD	- - -	- - -	- - -	- - -

Table 1: stand of standardization and implementation of ISO 10303

Annex B: Detailed descriptions of occurred problems

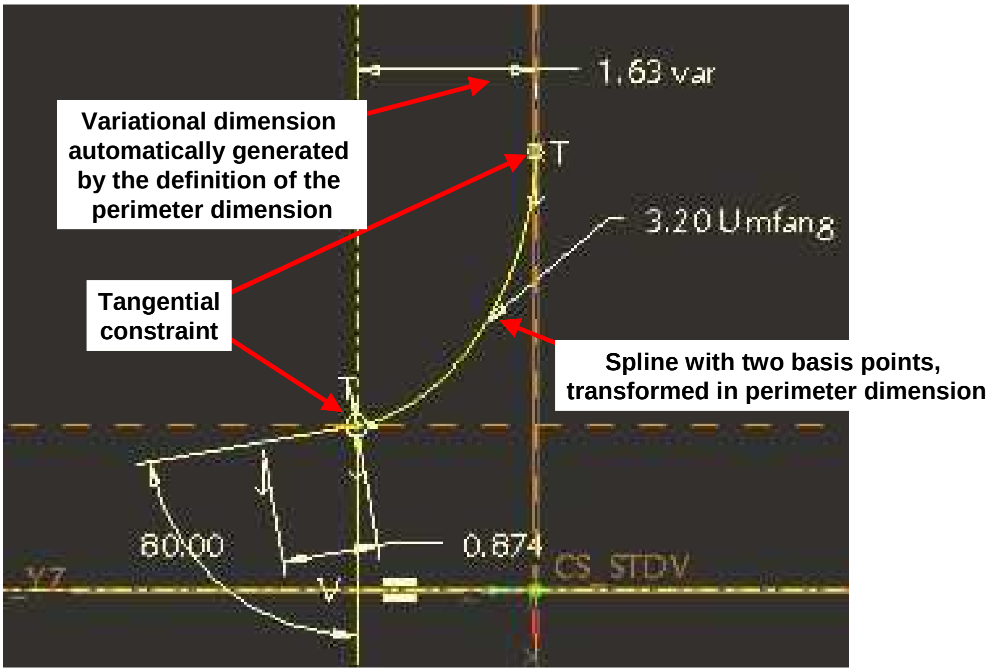

Figure 10-1: Detailed graphical description of the problem with perimeter dimensions

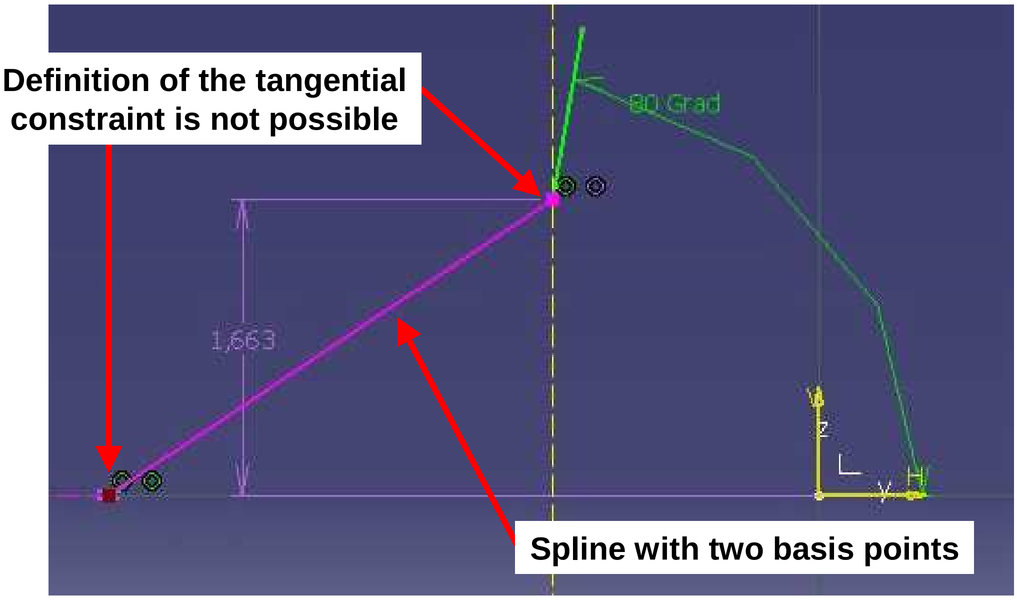

Figure 10-2: perimeter dimension in Pro/ENGINEER

Annex C: Mapping tables ISO 10303 part 108

The following tables were separated by the defined STEP schemata in part 108 and part 109 of ISO 10303. In these tables as basic element the STEP entities are assigned to the elements belonging to the CAD systems CATIA V5,Pro/ENGINEER Wildfire 2.0 and Unigraphics NX 4.0. If more elements are defined within the representations of the CAD systems these are assigned to STEP entities (or combinations of STEP entities) in separate tables. The descriptions of the STEP entities are withdrawn of the ISO definitions.

To create these tables the CAA (Component Application Architecture) documentation of CATIA V5 Release 13, the Pro/ENGINEER TOOLKIT documentation and the documentation of UG Open API were used. These documents support the user in user defined programming inside these systems.

C1: Parametrization_schema

In the subsequent Table 2, the coordinations of the model parameters of the parameterization_schema are cited. Principle components of this data model are the bound_model_parameter as well as the unbound_model_parameter.

ISO 10303 – Part 108: parameterization_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
model_parameter	This entity is divided in bound an unbound model parameters The fact that model parameter is a type of maths variable restricts its underlying domain of values to subsets of the real or integer numbers, Booleans or strings.	CatParameter	ProParameter	UF_PARAM oder UF_ATTR
bound_ model_parameter	The bound model parameter entity data type is a type of model parameter whose instances can be bound to (associated with) explicit attributes of entity instances.	CatAgregatedParameter	PRO_PARAM mit ProParamowner	UF_PARAM oder UF_ATTR
unbound_ model_parameter	This parameter entity data type is a type of model parameter representing a variable that is not bound to an attribute of any entity instance in the model.	CatFreeParameter	PRO_PARAM ohne ProParamowner	UF_PARAM oder UF_ATTR

Table 2: Mapping table of the parameterization_schema of ISO 10303-108

C2: explicit_constraint_schema

In the Table 3, the general explicit conditions are assigned the corresponding objects of the CAD-systems.

ISO 10303 - Part 108: explicit_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
explicit_constraint	The entity asserts a relationship between model elements that the receiving system is expected to maintain when the model is modified. This entity data type is the generic supertype of all explicit constraints.	CatConstraintType	PRO_CONSTRAINT	No OO
defined_constraint	Constraint with no specification of a precise mathematical relationship between parameters and entity data type instance attributes.	CatConstraintType	PRO_CONSTRAINT	
equal_parameter_ constraint	All constrained elements have the same value. It is possible to select a reference element or to share a common value.	CatParamIsEqualTo (Value Compare, Func IsEqualTo)	PRO_CONSTRAINT _ EQUAL_SEGMENT S, PRO_CONSTRAINT _ EQUAL_RADII	UF_SKET_equal_len gth UF_SKET_equal_rad ius
Free_form_constrain t	A special purpose constraint that cannot be modelled with the defined constraint entities available in any particular application context.	CatFreeParameter	ProParameter	UF_PARAM
free_form_assignme nt	Assignment of a value to one ore more reference elements.	IntParam RealParam StrParam	PRO_PARAM_ INTEGER PRO_PARAM_ STRING	UF_PARAM

ISO 10303 - Part 108: explicit_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
free_form_relation	Representing a boolean valued relation between reference elements.	BoolParam	PRO_PARAM_ BOOLEAN	UF_PARAM

Table 3: Mapping table of the explicit_constraint_schema of ISO 10303-108

C3: variational_representation_schema

In the mapping table for variant constructions, the head entity is the variational_representation entity. In Table 4, the entities defined in STEP are defined to variants constructions.

ISO 10303 - Part 108: variational_representation_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
Variational_ representation_item	It defines an element of a representation that does not affect the static characteristics of a transferred model at the time of transfer, but that has the potential to control its behaviour when the model is edited in a receiving system following a transfer.	DesignTable	PROUITable	UF_PARAM
auxiliary_gemetric_ representation_item	It provides a representation for geometric elements that exist in a variational representation for use as reference elements in constraints but are not part of the current representation.	DesignTable-Configuration	PROUITableCompon ent	UF_PARAM

ISO 10303 - Part 108: variational_representation_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
variational_ representation	This entity data type defines parameterization and constraint information that may be used to edit the model, following a transfer, in a manner consistent with the designer's original intent.	DesignTableColumn	PROUITableColumn PROUITableRow PROUITableCell	UF_PARAM
variational_current_ representation_ relationship	This entity defines the relationship between a variational representation and its embedded nonvariational 'current result' representation.	DesignTable-Configuration	PROUITableCompon ent	UF_PARAM

Table 4: Mapping table of the variational_representation_schema of ISO 10303-108

C4: explicit_geometric_constraint_schema

In Table 5 and Table 6the explicit geometric constraints are assigned the corresponding CAD-elements. These constraints serve as definition of dependencies in so-called internal sketches, the parametrical two-dimensional sketches for the construction of sweep models. In this schema surprised that in STEP no conditions for horizontality and verticality are defined. These in the CAD systems defined constraints were assigned to the parallel_geometric_constraint with reference to a horizontal or vertical element.

ISO 10303 - Part 108: explicit_geometric_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
explicit_ geometric_constrai nt	This entity asserts relationships between elements of a geometric model in descriptive terms. These constraints can be defined in direct form with or in indirect form without reference element.	CatConstraintType	PRO_SECTION_ CONSTRAINTS	No OO

ISO 10303 - Part 108: explicit_geometric_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
fixed_element_ geometric_constraint	This element is fixed.	catCstTypeReference	PRO_CONSTRAINT_ FIXTURE (PRO_ASM_COMP_ TYPE_FIXTURE)	UF_SKET_fixed
parallel_ geometric_constraint	This entity asserts that the members of a set of two or more linear geometry constraint element instances (lines, planes, directions or vectors) are mutually parallel. A reference element may be provided; the constraint is directed if this is the case, and undirected if not.	catCstTypeParallelism	PRO_CONSTRAINT_ PARALLEL_ENTS	UF_SKET_parallel
pgc_with_dimension	Subtype of the parallel_ geometric_constraint asserting a value for the distance between two parallel line or plane instances. If the elements concerned are a line and a plane, the distance between them shall be measured in the direction of their common normal.	catCstTypeParallelism in combination with CatDimDistance	PRO_CONSTRAINT_ PARALLEL_ENTS in combination with PRODIMTYPE_LINEAR	UF_SKET_parallel UF_ASSEM_parallel UF_ASSEM_distance
point_distance_ geometric_constraint	A constraint defining the distance of points from each other, or from one or more reference elements that may be points, curves or surfaces.	CatDimDistance	PRODIMTYPE_LINEAR	UF_ASSEM_distance

ISO 10303 - Part 108: explicit_geometric_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
pdgc_with_dimension	Subtype of the point_distance_geometric_constraint asserting a value for the distance between two points, or between multiple points and a set of one or more reference elements.	CatDimDistance	PRODIMTYPE_LINEAR	UF_ASSEM_distance
skew_line_distance_geometric_constraint	This entity is asserting a value for the distance between two skew (non-parallel) lines, measured along their common normal.	CatDimDistance	PRODIMTYPE_UNKNOWN	UF_ASSEM_distance
near_point_relationship	It is relating a curve or surface element to a point that lies on or close to it. This allows the specification of an approximate location where a constraint condition is satisfied on that curve or surface.	This allows the asserting of a point in a small tolerance field. There is unfortunately no possible relation to the elements of the CAD systems.		
curve_distance_geometric_constraint	This entity asserts a constraint on the distance between two curves in the undirected case, or between one curve and up to four reference elements in the directed case.	CatDimDistance	PRODIMTYPE_LINEAR	UF_ASSEM_distance

ISO 10303 - Part 108: explicit_geometric_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
cdgc_with_dimension	Subtype of the curve_distance_geometric_constraint defining the minimum distance between two curves or a curve and up to three reference elements.	CatDimDistance	PRODIMTYPE_LINEAR	UF_ASSEM_distance
surface_distance_geometric_constraint	This entity defines a constraint on the distance between two surfaces in the undirected case, or between one surface and up to three reference elements in the directed case.	CatDimDistance	PRODIMTYPE_LINEAR	UF_ASSEM_distance
sdgc_with_dimension	Subtype of the surface_distance_geometric_constraint defining a minimum distance between two surfaces or a surface and up to three reference elements.	CatDimDistance	PRODIMTYPE_LINEAR	UF_ASSEM_distance
radius_geometric_constraint	Definition of a radius.	catCstTypeRadius	PRODIMTYPE_RADIUS	UF_SKET_radius_dim
rgc_with_dimension	Subtype of the radius_geometric_constraint asserting a value for the radius in form of a dimension.	catDimRadius	PRODIMTYPE_RADIUS	UF_SKET_radius_dim

ISO 10303 - Part 108: explicit_geometric_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
curve_length_ geometric_constraint	This entity is asserting that the lengths of all members of a set of bounded curve instances have the same value. It is an undirected constraint, having no reference element.	catCstTypeLength	PRO_CURVE_LENGTH	UF_SKET_equal_length
clgc_with_dimension	Subtype of the curve_length_ geometric_constraint with a specified length.	catCstTypeLength	PRO_CURVE_LENGTH	UF_SKET_equal_length
parallel_offset_ geometric_constraint	This entity defines that the members of a set of curves or a set of surfaces are parallel offsets of each other.	catCstTypeParallelism	PRO_CONSTRAINT_ PARALLEL_ENTS	UF_ASSEM_parallel
pogc_with_dimensions	Subtype of the parallel_ offset_geometric_constraint, defining a specified distance between the selected elements.	catCstTypeParallelism in combination with CatDimDistance	PRO_CONSTRAINT_ PARALLEL_ENTS in combination with PRODIMTYPE_LINEAR	UF_ASSEM_parallel

ISO 10303 - Part 108: explicit_geometric_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
angle_ geometric_constraint	This entity is asserting constraints on angles between instances of linear geometry constraint element (lines, planes, directions and vectors) as described below. It shall not be instantiated in undirected form, with no reference element, except in the form of its dimensional subtypeagc with dimension.	catCstTypeAngle catCstTypePlanarAngle	PRODIMTYPE_ANGLE	UF_SKET_angular_dim UF_ASSEM_angle
agc_with_dimension	Angles wit a specified value..	catDimAngle	PRODIMTYPE_ANGLE	UF_SKET_angular_dim UF_ASSEM_angle
perpendicular_ geometric_constraint	asserting that instances of linear geometry constraint element are perpendicular to each other. The constraint may be directed or undirected, in the undirected case there can be up to three reference elements.	catCstType-Perpendicularity	PRO_CONSTRAINT_ORTHOG_ENTS	UF_SKET_perpendicular UF_ASSEM_perpendicular

ISO 10303 - Part 108: explicit_geometric_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
incidence_ geometric_constraint	This entity is asserting that one or more geometric constraint element instances lie on (or are incident on) one or more reference geometric constraint element instances. In the undirected case, where no reference element is present, the number of constrained elements is restricted to two, one of which is required to be incident on the other.	catCstTypeOn	PRO_CONSTRAINT_SAME_POINT, PRO_CONSTRAINT_PNT_ON_ENT	UF_SKET_coincident
coaxial_ geometric_constraint	It defines that a set of axial geometry constraint element instances share the same axis. The constrained set may contain a mixture of points, lines, circles and axially symmetric surfaces and solids. Points are constrained to lie on the axis, lines to be coincident with it and planes perpendicular to it.	catCstType-Concentricity	PRO_CONSTRAINT_COLLINEAR_LINES	UF_SKET_concentric UF_ASSEM_center
tangent_ geometric_constraint	Two ore more curves or surfaces are oriented tangential.	catCstTypeTangency	PRO_CONSTRAINT_TANGENT_ENTS	UF_SKET_tangent UF_ASSEM_tangent

ISO 10303 - Part 108: explicit_geometric_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
symmetry_ geometric_constraint	Two geometric elements are symmetrically disposed with respect to a specified mirror element.	catCstTypeSymmetry	PRO_CONSTRAINT_ SYMMETRY	UF_SKET_mirror
swept_point_curve_ geometric_constraint	Definition of constraints in sketches (swept_edge_curves) of sweep models. The constraint asserts that these edge curves are constrained.	SweepSketch-Constraints	PRO_E_SWEEP_ SECTION_ CONSTRAINTS	
swept_curve_surface_ geometric_constraint	It is asserting constraints on the swept surfaces of a computed explicit configuration corresponding to any sweep model. The constraint asserts that these surfaces are constrained.	SweepSketch-Constraints	PRO_E_SWEEP_ SURFACE_ CONSTRAINTS	
curve_segement_set	It defines a set of composite curve segment elements, for use in the curve_smoothness_ geometric_constraint.	HybridShapeBoundary	ProCurveCollection	
curve_smoothness_ geometric_constraint	Definition of specified degrees of smoothness at junctions between the individual composite curve segment instances involved in a composite curve instance.	HybridShape-CurveSmooth	ProCurveData (p_degree,p_params)	UF_CURVE_spline_s

ISO 10303 - Part 108: explicit_geometric_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
surface_path_set	Defines a set of surface patch elements, for use in the surface_smoothness_ geometric_constraint.	HybridShapeBlend	ProSrfCollection	UF_SURF_REG
surface_smoothness_ geometric_constraint	Definition of specified degrees of smoothness at boundaries between the individualsurface patch instances involved in a rectangular composite surface instance.	HybridShapeFactory	ProSurfaceData	UF_SURF_REG

Table 5: Mapping table of the explicit_geometric_constraint_schema of ISO 10303-108

ISO 10303 - Part 108: explicit_geometric_constraint_schema				
In addition in the CAD systems defined constraints				
CATIA V5 and Pro/ENGINEER				
STEP entity	**Description**	**CATIA V5 element**	**Pro/E element**	**UG element**
parallel_ geometric_constraint (to a horizontal reference element)	Defines a horizontal constraint between elements.	catCstTypeHorizontality	PRO_CONSTRAINT_ HORIZONTAL_ENT	UF_SKET_horizontal
parallel_ geometric_constraint (to a vertical reference element)	Defines a vertical constraint between elements.	catCstTypeVerticality	PRO_CONSTRAINT_ VERTIKAL_ENT	UF_SKET_vertical
parallel_ geometric_constraint (to a horizontal reference element)	Align an arc horizontally.	catCstTypeHorizontality	PRO_CONSTRAINT_ HORIZONTAL_ARC	UF_SKET_horizontal

ISO 10303 - Part 108: explicit_geometric_constraint_schema				
In addition in the CAD systems defined constraints				
CATIA V5 and Pro/ENGINEER				
parallel_ geometric_constraint (to a vertical reference element)	Align an arc vertically.	catCstTypeVerticality	PRO_CONSTRAINT_ VERTIKAL_ARC	UF_SKET_vertical
incidence_ geometric_constraint	Align arcs to a common midpoint.	catCstTypeOn	PRO_CONSTRAINT_ SAME_POINT	UF_SKET_coincide nt
CATIA V5				
STEP entity	**Description**	**CATIA V5 element**	**Pro/E element**	**UG element**
parallel_ geometric_constraint	The selected axis is parallell to one of the three room axes.	catCstType- AxisParallelism	PRO_CONSTRAINT_ PARALLEL_ENTS	UF_SKET_parallel
perpendicular_ geometric_constraint	The selected axis is perpendicular to one of the three room axes.	catCstType AxisPerpendicularity	PRO_CONSTRAINT_ ORTHOG_ENTS	UF_SKET_perpendi cular
equal_parameter_ constraint in combination with incidence_ geometric_constraint	A point is defined as the exact midpoint of a line.	catCstTypeMidPoint	PRO_CONSTRAINT_ EQUAL_SEGMENTS (with two segments) in combination with PRO_CONSTRAINT_ PNT_ON_ENT	UF_SKET_midpoint
equal_parameter_ constraint perhaps in combination with incidence_ geometric_constraint	A third point has the same distance to two other selected points These points con lie on the same line.	catCstType Equidistance	PRO_CONSTRAINT_ EQUAL_SEGMENTS perhaps in combination with PRO_CONSTRAINT_ PNT_ON_ENT	UF_SKET_equal_le ngth And UF_SKET_point_on _curve

ISO 10303 - Part 108: explicit_geometric_constraint_schema				
In addition in the CAD systems defined constraints				
Pro/Engineer				
STEP entity	**Description**	**CATIA V5 element**	**Pro/E element**	**UG element**
From part 42: circle (u = 90, u is corresponding to the angle area)	Restriction of an arc on 90 degrees.	HybridShapeCircle in combination with SubSetLimitation 0 (Angles; Value= 90)	PRO_CONSTRAINT_90_ARC	Line startPoint – Center and Line endpoint Center with UF_SKET_perpendicular
From part 42: circle (u = 180, u is corresponding to the angle area)	Restriction of an arc on 180 degrees.	HybridShapeCircle in combination with SubSetLimitation 0 (Angles; Value= 180)	PRO_CONSTRAINT_180_ARC	Line startPoint – Center and Line endpoint Center with UF_SKET_collinear

Table 6: Mapping table of the explicit_geometric_constraint_schema of ISO 10303-108 - additional in CAD systems defined entities

C5: sketch_schema

In the following Table 7 the STEP entities necessary for definition of internal sketches are assigned. These sketches as base for the parametric construction are designated in the CAD-systems CATIA V5 and Pro/ENGINEER as a Section.

ISO 10303 - Part 108: sketch_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
implicit_point_on_plane	It is the abstract supertype of a class of implicitly defined points lying in the plane of a positioned sketch for use as reference elements in constraints.	CATIAPoint2D	PRO_DPOINT_TYPE_SKETCHED	No OO
implicit_planar_intersection_point	It provides an implicit representation for a point generated by the intersection of a three-dimensional curve with the plane of a positioned sketch..	CATIAIntersection-Point2D (Func CreateIntersections)	PRO_DPOINT_TYPE_GENERAL	UF_SKET_point_on_curve And On UF_CURVE_SECTION

ISO 10303 - Part 108: sketch_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
implicit_planar_ projection_point	Defines an implicit representation for a point generated by the parallel projection of an external point in a specified direction onto the plane of a positioned sketch.	CATIAProjection-Point2D (Func CreateProjections)	PRO_DPOINT_ TYPE_OFFSET_CSYS	UF_CURVE_PROJ
implicit_planar_curve	It is the abstract supertype of a class of implicitly defined curves lying in the plane of a positioned sketch for use as reference elements in constraints.	CATIALine2D CATIASpline2D	PRO_CURVE_ TYPE_SKETCHED	No OO
implicit_ intersection_curve	It provides an implicit representation for an imported intersection curve in the plane of apositioned sketch.	CATIAIntersection-Line2D CATIAIntersection-Spline2D (Func CreateIntersections)	PRO_CURVE_ TYPE_INTSRF	UF_CURVE_SECTION…
implicit_projected_curve	It provides an implicit representation for an imported curve generated by parallel projection of an external curve onto the plane of a positioned sketch instance.	CATIAProjection-Line2D CATIAProjection-Spline2D (Func CreateProjections)	PRO_CURVE_ TYPE_OFFSET	UF_CURVE_PROJ
implicit_model_ intersection_curve	Defines an implicit representation for an imported curve defined by the intersection of the plane of a positioned sketch with a surface model or a solid model.	CATIAIntersection-Line2D CATIAIntersection Spline2D (Func CreateIntersections)	PRO_CURVE_ TYPE_INTSRF	UF_CURVE_SECTION

ISO 10303 - Part 108: sketch_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
implicit_ silhouette_curve	Defines an implicit representation for an imported curve defined by the intersection of the plane of a positioned sketch with a surface model or a solid model.	CATIAProjection-Line2D CATIAProjection-Spline2D (Func CreateProjections)	PRO_CURVE_ TYPE_OFFSET	UF_CURVE_PROJ
neutral_sketch_ representation	It represents a planar configuration of geometry defined in a neutral 2D coordinate space. Such sketches are often stored in a libraries for reuse in a variety of circumstances. A neutral sketch may be interpreted either as a set of general curves or as the area bounded by a set of non-intersecting closed curves. It can be repositioned in 3D model space by means of a transformation.	CATIASketch	ProSection (in 2D, the name is identical with this of a sketch in 3D)	UF_SKET

ISO 10303 - Part 108: sketch_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
positioned_sketch	It provides a representation for a planar geometric configuration that may be subjected to a sweep operation or used as a section curve in the construction of a skinned or lofted surface. In a CAD system, such a configuration may be created directly in three dimensional model space or may be derived from a neutral sketch by the application of a transformation.	CATIAStrSection	ProSection (in 3D, the name is identical with this of a sketch in 2D)	UF_SKET
repositioned_ neutral_sketch	It provides details of the transformation applied to a two-dimensional neutral sketch representation instance for its reposition in three-dimensional model space.	CATIASketch-Constraints	PRO_SECTION_ CONSTRAINTS	UF_ASSEM…
implicit_explicit_ positioned_sketch_ relationship	Defines the relationship between implicit and explicit representations of a positioned sketch.	CATIASketch-Constraints	PRO_SECTION_ CONSTRAINTS	UF_ASSEM…
subsketch	Defines a partial sketch in terms of a subset of the geometric elements composing a full sketch.	Not defined in CATIA, recommended is a conversion in CATIAStrSection	ProSectionSubsection	UF_SKET

ISO 10303 - Part 108: sketch_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
rigid_subsketch	The elements of an instance of rigid subsketch are required to remain invariant in shape and in their relationships with respect to each other when the sketch to which they belong is edited, while the location and orientation of the subsketch as a whole may change subject to constraint relationships with other elements not belonging to the subsketch. The intended effect is that the group of elements behaves as a rigid body.	CATIAStrSection in combination with catCstTypeReference	ProSectionSubsection in combination with PRO_CONSTRAINT_FIXTURE	UF_SKET In combination with constraints

Table 7: Mapping table of the sketch_schema of ISO 10303-108

Annex D: Mapping Tables ISO 10303 part 109

The position constraints within assembly models are assigned to the equivalent elements of the CAD-systems.

D1: assembly_feature_relationship_schema

ISO 10303 - Part 109: assembly_feature_relationship_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
Shape_aspect_relationship_representation_association	This entity is used to describe the relative position and orientation between two shape aspect instances.	ShapeFactory	PRO_E_FEATURE	
representative_shape_representation	Representation of a shape aspect.	ShapeFactoryObjects	PRO_E_FEATURE_TYPE	
free_kinematic_motion_representation	This entity represents a free kinematic motion in three dimensional space of an assembly feature with respect to another assembly feature.	CONSTRAINTS	PRO_ASM_CONSTRAINT_TYPE	UF_MOTION_...
constrained_kinematic_motion_representation	This entity represents a kinematic motion constrained by a kinematic pair between a pair of assembly feature.	CONSTRAINTS	PRO_ASM_CONSTRAINT_TYPE	UF_MOTION_...

Table 8: Mapping table of the assembly_feature_relationship_schema of ISO 10303-109

D2: assembly_constraint_schema

In the Table 9 and Table 10, the geometric constraints between the components of an assembly model are listed.

ISO 10303 - Part 109: assembly_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
assembly_geometric_constraint	Supertype of all geometric_assembly_constraints, sybtype of the explicit_constraints defined in ISO 10303-108	CONSTRAINTS	PRO_E_COMPONENT_CONSTR_TYPE	No OO

ISO 10303 - Part 109: assembly_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
Binary_assembly_constraint	Supertype of all binary_assembly_constraints, describing the relationship between two constituents of an assembly model.	CONSTRAINT	PRO_ASM_CONSTRAINT_TYPE	No OO
fixed_constituent_assembly_constraint	Definition of a fixed constituent inside an assembly model.	catCstTypeReference	PRO_ASM_FIX	First inserted Part is Fix in UG
parallel_assembly_constraint	It specifies that two linear geometry constraint element instances, each belonging to different constituents, are constrained to be parallel. The linear geometry constraint element shall be either a line or a plane.	catCstTypeParallelism	PRO_ASM_ORIENT	UF_ASSEM_orient
parallel_assembly_constraint_with_dimension	Subtype of the parallel_assembly_constraint that is used for constraining distances between two parallel Elements of different constituents.	catCstTypeParallelism in combination with catCstTypeDistance (between two elements)	PRO_ASM_MATE_OFF	UF_ASSEM_v16_mate
Surface_distance_assembly_constraint_with_dimension	Constrains the minimum distance between two surfaces belonging to different constituents.	catCstTypeDistance (between two elements)	PRO_ASM_ALIGN_OFF	UF_ASSEM_v16_align

ISO 10303 - Part 109: assembly_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
angle_assembly_ constraint_with_ dimension	It specifies between constrained elements a specified angle with a given instance specified by reference elements. The two elements concerned shall belong to different constituents.	catCstTypeAngle	PRO_ASM_ ALIGN_ANG_OFF, PRO_ASM_ MATE_ANG_OFF	UF_ASSEM_angle
perpendicular_ assembly_constraint	Instances belonging to different constituents be constrained to be perpendicular. The linear elements should be either a line or a plane.	catCstType- Perpendicularity	PRO_CONSTRAINT _ ORTHOG_ENTS	UF_ASSEM_perpen dicular
incidence_assembly _ constraint	This entity specifies that two elements belonging to different constituents have the relationship that one of them, regarded as a point set, is entirely included in the other.	catCstTypeOn catCstTypeSurfCont act (contact surfaces) catCstTypeLinConta ct (contact lines) catCstTypePonCont act (contact points)	PRO_ASM_MATE (surfaces, planes), PRO_ASM_ALIGN (points, surfaces), PRO_ASM_ PNT_ON_SRF (point on surface/plane), PRO_ASM_ EDGE_ON_SRF (edge on surface/plane) PRO_ASM_ PTN_ON_LINE (point on edge/axis)	UF_ASSEM_v16_ma te And UF_ASSEM_v16_ali gn
coaxial_assembly_ constraint	This entity specifies that the axes of two elements belonging to different constituents are constrained to be identical.	catCstType- Concentricity	PRO_ASM_ALIGN (axes), PRO_ASM_INSERT	UF_ASSEM_v16_ali gn

ISO 10303 - Part 109: assembly_constraint_schema				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element**
tangent_assembly_ constraint	It specifies that two geometric elements belonging to different constituents are constrained to be tangent to each other.	catCstTypeTangenc y	PRO_ASM_TANGE NT	UF_ASSEM_tangent

Table 9: Mapping table of the assembly_constraint_schema of ISO 10303-109

ISO 10303 - Part 109: assembly_constraint_schema				
In addition in the CAD systems defined constraints				
STEP entity	**STEP description**	**CATIA V5 element**	**Pro/E element**	**UG element / function**
incidence_assembly _ constraint	Direct positioning of an constituent by aligning the coordinate system of the constituent to the coordinate system of the assembly model.	catCstTypeOn (expansion in CATIA around selection possibility of coordinate systems necessarily)	PRO_ASM_CSYS	UF_ASSEM_repositi on_...
incidence_assembly _ constraint	Direct positioning of an constituent by aligning the standard coordinate system of the constituent to the standard coordinate system of the assembly model.	catCstTypeOn (expansion in CATIA around selection possibility of coordinate systems necessarily)	PRO_ASM_ DEF_PLACEMENT	UF_ASSEM_repositi on_...
It is no mapping possible in STEP.	For use in simplified representations when on constituent will be replaced by a simplified one	There are no simplified representations defined in CATIA V5.	PRO_ASM_ SUBSTITUTE	UF_ASSEM_substitu te_component

Table 10: Mapping table of the assembly_constraint_schema of ISO 10303-109 – additional in the CAD systems defined constraints