# Parametrical 3D Data Exchange via STEP



WHITE PAPER

Parametrical 3D Data Exchange via STEP

ProSTEP iViP

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## 1 Introduction

Studies show that approximately 80% of design solutions in the automotive industry are variant designs or 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 data exchange. Fully modifiable designs, including parametric and feature information, cannot currently be exchanged across different CAx systems – neither in a standardized nor in a proprietary manner.

Some CAD vendors offer initial solutions for importing CAD models from systems of other vendors while at the same time retaining feature information (e.g. feature recognition in CATIA V5). But these solutions are unidirectional, and they are neither fully functional nor have they yet been standardized.

Several international standards relating to parametrics have recently been published. These are Part 55, 108, and 109 of the ISO 10303 (STEP) standard. The objective of the ProSTEP iViP project "Parametrical 3D Data Exchange via STEP" was therefore to overcome this limitation and enable the exchange of parametric CAD models in a standardized (STEP-based) manner. Members of this group were Audi, BorgWarner Turbo Systems, DaimlerChrysler, Delphi, University of Applied Sciences Augsburg, University of Applied Sciences St. Gallen, PTC, PROSTEP, Robert Bosch, Siemens VDO, UGS, Volkswagen, ZF Friedrichshafen et al.

The main results achieved in the "Parametrical 3D Data Exchange via STEP" project are discussed in the following. This includes a description of the relevant use cases, as well as the experience gained within the context of a comparative analysis. The aim of the analysis was to evaluate the feasibility of a STEP-based parametric data exchange between leading CAD systems.

More detailed information can be found in the final report of the "Parametrical 3D Data Exchange via STEP" project group, which is available in the download area of the ProSTEP iViP website (www.prostep.org).

## 2 Need for action

Despite the availability of STEP processors, data exchange scenarios between automotive OEM and their suppliers are increasingly based on a policy involving the exchange of native CAD data. This procedure is not efficient and produces isolated islands of system-specific data. Moreover, many process chains on the OEM and supplier sides 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 supply explicit representations of product shape. This product shape includes neither parametrical nor feature information nor construction history. Therefore, such models can hardly be modified in the target system.

Neutral data exchange, especially via STEP, has proven successful in practice. Various STEP processor implementations exist. They allow the exchange of product geometry between all key systems in the automotive industry on a high-quality level. Nevertheless, the problem within these data exchange processes is that it is difficult to process the models on the destination side. Both construction history and design intent, which is to some extent captured by parametrics, are lost.

The main objective of parametrical 3D data exchange is to enable the transfer of intelligent CAD models to 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 the geometry received via neutral file exchange or native system geometry can be handled in almost the same way.

The underlying rationale of this objective is to overcome restrictions that produce non-interoperable, system-specific process chains. In technical terms, to overcome these restrictions, the construction history together with parametric feature information has to be exchanged. Recently published parts of ISO 10303 support such an exchange. The common idea of these standards is to capture the inherent parametrics in a system-independent representation similar to the successful definition of data models for explicit geometry in STEP.

# 3 Practice-oriented use cases

The members of the project group "Parametrical 3D Data Exchange via STEP" defined two major use cases:

Parametrical data exchange of standard and carry-over parts:

This use case reflects the necessity of working with standard and carry-over 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 parts (adapted to the higher requirements of the constructers) in both systems, a parametrical data transfer is needed to avoid extra work.

This was the reason for evaluating the parametric capabilities of CATIA V5 and Pro/ENGINEER Wildfire 2.0 within the context of the comparative analysis.

Exchange parametric models of production equipment:

This use case deals with the transfer of parametric information to different systems for NC programming, simulation or set up of production equipment. In order to be able to work efficiently within this process chain, the generation of redundant parametric data must be avoided. Other possibilities arise in complex systems used to plan virtual factories or assembly lines.

The following requirements for parametrical data exchange were defined for each of these use cases:

- 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

An additional requirement is the ability to exchange complete and configurable models. 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.

In a given business process, the following exchange settings are possible:

- transfer complete shape as explicit geometry (current state)
- transfer complete shape as implicit geometry (full parametrics)
- 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 behavior (parametric relationships between features) to control the degree of engineering know-how transferred

# 4 Results of the comparative analysis

Based on the capabilities of the CAD systems CATIA V5, Pro/ENGINEER Wildfire 2.0 and Unigraphics NX 2.0, a comparative analysis was performed to evaluate the feasibility of STEP-based parametric data exchange between these systems.

Over the last few years, it could be observed that CAD systems are becoming increasingly similar with regard to their parametric feature set. Meanwhile, the systems follow a feature-based approach promising successful realization of the project objective of exchanging parametric information in a standardized manner.

There are, of course, differences that have to be considered carefully when developing 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-based file. In the case of standard-based feature and parametric data 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 the standardization of ISO 10303 published some new parts.

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 objectives (cf. 4.1).

## 4.1 Relevant standards

The following standards have to be taken into consideration with regard to parametrical 3D data exchange via STEP:

1. ISO 10303 Part 55:
   - Definition of a neutral model describing the general purpose, construction history and the operations (features)
   - Handling of mixed models which include both parametric modeling operations and explicitly described shape within a construction history
2. ISO 10303 Part 42: Identification of topological entities such as vertices, edges and faces to which parametric modeling operations refer (features)
3. ISO 10303 Part 108: Exchange of sketches, their parameters and constraints

In addition, to realize the defined use cases (cf. 3), consideration must also be given to 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. All these parts of ISO 10303 have been published as an international standard.

## 4.2 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 following aspects of parametric data exchange are of interest:

1. Construction history, i.e. the basic mechanism for modeling a sequence of parametric operations with parameters that may reference implicit topological entities like edges and faces.
2. 2D sketches, their parameters and inherent dimensional and geometric constraints: Sketches are the basic means of creating solids comprising features and parametrics. Part 108 provides a basis to be considered with regard to this topic.
3. Linear extrusions (additive, subtractive): Extruding 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: With regard to the parametrical exchange of assembly models, the position constraints defined in part 109 of ISO 10303 have to be supported.
5. Simple blendings (linear chamfers and circular roundings and fillets): Although in most CAD systems blendings should be applied at the end of a modeling 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 promoted.
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. Therefore, the implementation of ISO 10303-55 in an application protocol (e.g. AP 214) has to be promoted.

Proposals for mappings between the defined STEP entities and the related elements of the CAD systems CATIA V5, Pro/ENGINEER Wildfire 2.0 and Unigraphics NX 2.0 are provided in the final report of the project group "Parametrical 3D Data Exchange via STEP".

## 4.3 Configurable parametric exchange

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

The following information has to be considered with regard to the exchange of constraints:

- 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

## 4.4 Problems encountered

Some problems were encountered during analysis of the standard and carry-over parts, and these problems will be presented in the following sections. The main problems result from the different forms of representation in the CAD systems and from the special features and functions that exist in only one of the analyzed systems.

### 4.4.1 Perimeter dimension

Pro/ENGINEER offers the option of defining the exact perimeter of a spline depending on a variable dimension. The spline is defined with two basis points and achieves its final geometry by defining tangential constraints to auxiliary geometry. In the analyzed model (see Figure 1), this function is used to define a spline depending on a radius dimension for simulation of the delivery and assembly conditions of the model.

The problem of data transfer is not representation in the neutral STEP format but the fact that this type of definition does not exist in the CATIA V5 sketcher. There is no possibility of defining a variable dimension in CATIA V5. In addition, the perimeter of the spline cannot be dimensioned. A detailed graphic description of the problem is shown in Figure 1.

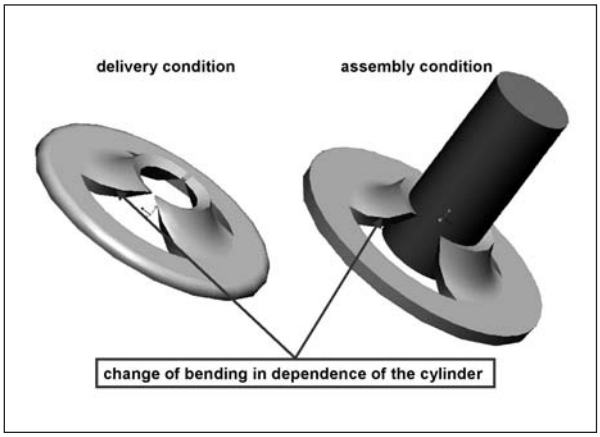



Figure 1: Dependence of perimeter dimensions [Source: Volkswagen AG]

### 4.4.2 Predefined position constraints in parts

In Pro/ENGINEER, the user has the possibility of predefining – in the part – the position constraints when adding these component to an assembly model. When positioning the part, the constructers only need to select the necessary reference elements depending on the models that can be selected in the assembly. In the plastic clip shown in Figure 2, the parametrics of the model also depend on the selected reference element (the diameter of the cylinder shown in Figure 1). The problem with regard to data transfer is that no adequate mapping to STEP entities and to CAD elements in CATIA V5 is possible.

### 4.4.3 Rules, formulas and inquiries

To support the data transfer of rules, formulas and inquiries, Part 50 (Mathematical constructs) and Part 51 (Mathematical description) of ISO 10303 have to be implemented for 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 knowledge-based engineering of the programs in CATIA, different functions for the definition of rules, formulas and inquiries exist whereas in Pro/ENGINEER, these functions are abstracted in one function (constraints).

### 4.4.4 Construction history

As mentioned in section 4.2, the construction history also has to be transferred for initial parametrical 3D data exchange via STEP. The construction history defined in Part 55 of ISO 10303 has to be implemented at the same time as Part 108 and Part 109. Data exchange can only be realized successfully if consideration is given to these standards.

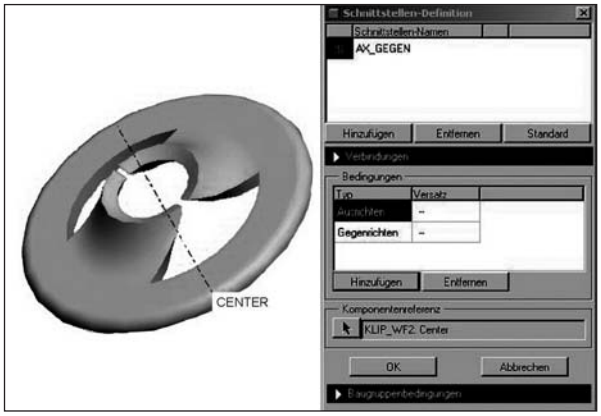



Figure 2: Predefinition of position constraints in Pro/ENGINEER [Source: Volkswagen AG]

Consideration must also be given to the exchange of construction history features such as blendings, chamfers, fillets or patterns. In this context, the standardization ambitions with regard to Part 111 have to be supported. Therefore, mapping tables were created for the STEP entities defined in Part 55 and Part 111 of ISO 10303.

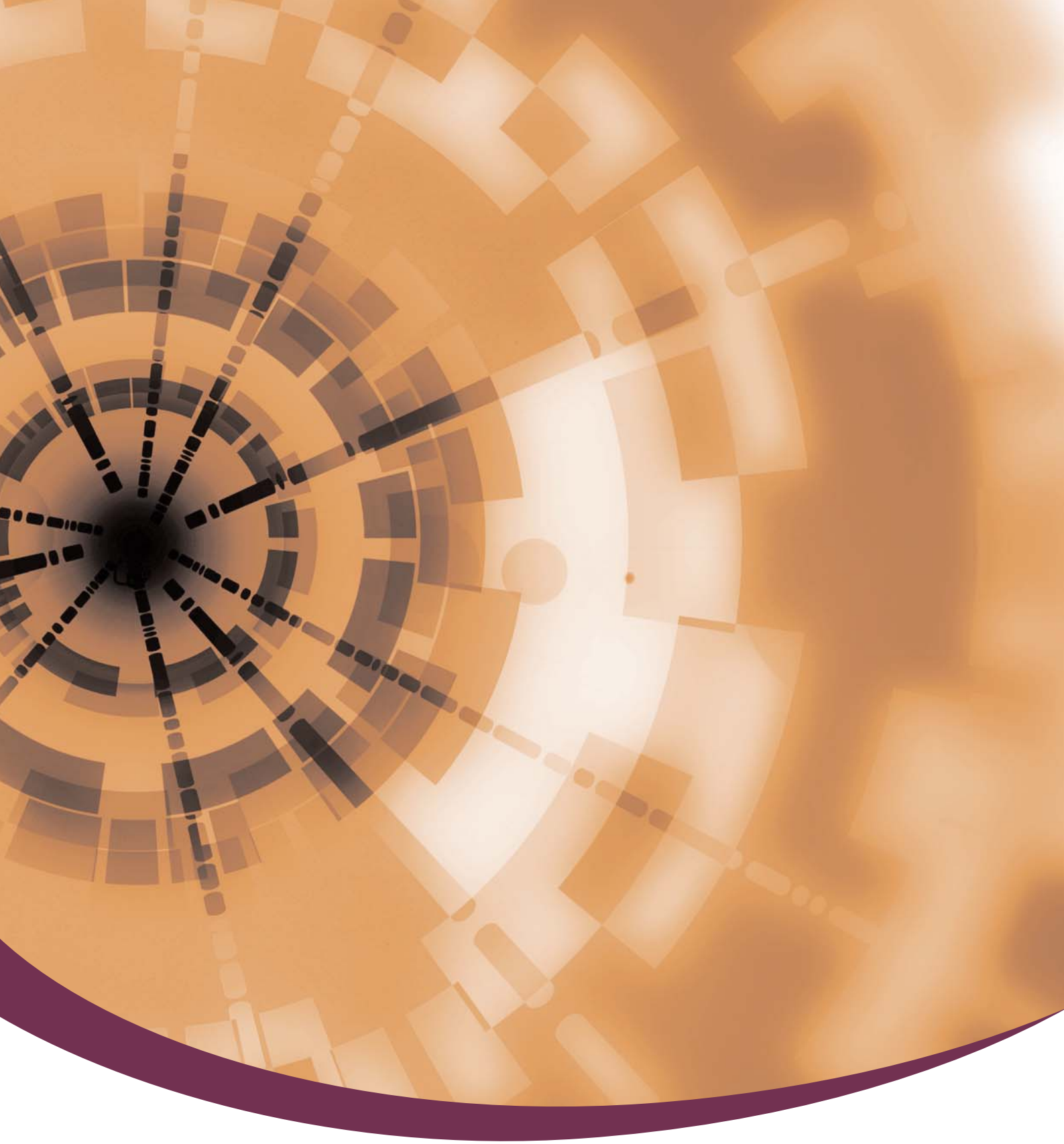
## 5 Summary

There are, of course, differences with regard to the internal representation of the different CAD systems, and these have to be handled carefully when mapping to the defined STEP entities. However, because the conclusion was reached that these systems are very similar with regard to parametrical representation and the feature-based approach, it is possible to realize an exchange of parametric information in a standardized manner.

Coordination of efforts regarding the standardization of Part 111 of ISO 10303 (construction history features) is necessary because of the importance of this part for the future realization of parametrical data exchange with a level of quality acceptable to users.

As a basis for future implementation, the mapping tables relating to the defined STEP entities of Part 55 and Part 111 for the CAD systems Unigraphics, Pro/Engineer and CATIA V5 have been included in the final report of the project group "Parametrical 3D Data Exchange via STEP". With regard to the further implementation of parametric 3D data exchange, future scope for any implementer have to include the implementation of construction history, internal sketches and constraints in one step. The implementation of the defined units of functionality (UoF) standardized in the STEP AP 214 must also be promoted.

## NOTES



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