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Abstract

This work aims to provide an overview of standard 3d data formats. Furthermore it aims on evaluating current CAD conversion pipelines. First of all this text describes the most popular data formats. Afterwards important properties that need to be taken into account for examining the conversion pipeline will be identified and described. As a result this paper suggests an adequate data format that satisfies CADPIPEs demands.

Contents

List of symbols

IGES	Initial Graphics Exchange Specification
STEP	Standard for the Exchange of Product model data
VRML	Virtual Reality Modelling Language
X3D	extensible 3d Graphics
OpenFlight	
DXF/DWG	Drawing Exchange File
3dS	3d-Studio File Format
STL	Stereolithography format
JT	Jupiter Tesselation
IFC	Industry Foundation Classes
GDL	Geometric Description Language
VDA	Verband der Automobilindustrie
OBJ	Object File
CDF	CAD Distillation Format
SAT	Standard ACIS Text Save File
VR	Virtual Reality
CAD	Computer Aided Design

1 Introduction

There are a number of several CAD systems on the market that all provide different data formats. The large number of available data formats makes it nearly impossible to define a standard. Such standard is necessary for avoiding laborious manual work. CADPIPE project is targeting to an, as automatic as possible, production chain that starts at developing a CAD model and ends up at the inclusion of that model to real-time rendering.
Therefore existing data formats have to be analysed regarding to their use and their advantages and disadvantages for use in real-time rendering. Furthermore it is necessary to compare these formats regarding to their use in different CAD systems. This paper will prototypical present the conversion pipelines of CATIA V5 and Pro /Engineer Wildfire. Result of an efficient conversion pipeline is the reduction of manual effort for converting CAD data to data that can be used in real-time environment. This allows an efficient use in CAD/CAM using companies across Europe that are not able to handle all the available formats on the market. Manual work will be reduced what will lead to an enormous reduction of costs while quality is guaranteed.

2 Overview of some CAD-systems

CAD-systems are more and more used for the development of new products. Many companies are using computer aided design software to provide more effectiveness in the development of their new products. In this case we'll take a focus only on CAD-systems for the mechanical design, which are working primarily with parametric 3d models. Pro /Engineer and CATIA are typically used and have many possibilities for creating high quality 3d models. Pro /Engineer was developed by Parametric Technology Corporation in the eighties. From begin Pro /Engineer was based on a full parametric 3d model kernel. In the eighties, most of others systems have been working either in 2d or with simple 3d versions. Nowadays the most CAD-systems are based on parametric 3d kernels. CATIA was developed by Dassault Systems. The parametric kernel came up first time with the version V5. Both systems are able to create very complicated and complex 3d geometry models. These 3d models will usually be used for tasks in the product development (generation of drawings, parts lists, using for the analyzing and simulation process, visualizations …).

Pro /Engineer
The creation of the 3d models with Pro /Engineer is based on working with so called "features", which describe the models geometry and additional properties of the part or assembly. Using the main volume features the material can be added (protrusion feature) or removed (cat feature). The creation of volume features is supported through the "datum features" (datum plane, datum axis, datum point, etc.). Thus a 3d model of an item can be created, which is called a part. The part represents a physical item (an object). One of several parameters for a part can be its material e.g. steel, aluminium, etc. The parts can be "assembled" together in an assembly. Any assembly can be assembled with other parts and assemblies together in the main assembly. The parts in an assembly or assemblies in the main assembly are called "components". The position of components is controlled through the boundary conditions. Pro /Engineer is mostly used for the development of engines, transmissions, gearboxes, sheet metal components. It is rarely used for high quality surfaces for automobile or aircraft styling.

CATIA V5
Working with CATIA V5 seems to be very close to working with Pro/ENGINEER. Features define the geometry of a part. Parts and assemblies take again the role of components in parent's assemblies. Besides working with volume-features a 3d model in CATIA can excellently be created through the definition and manipulation of surfaces. In CATIA V5 an assembly is called a "product", because this digital product can contain many several properties and data in the whole product life cycle. In CATIA V5 the 3d models can voluntary be non-parametric. The permanently developed and improved new modules for CATIA V5 from Dassault Systems are more and more user friendly and focused on specific technical problems. This makes it possible solving complicated tasks for most users. CATIA is very often used for the styling of aircrafts, cars and other products with very high surface quality requirements.

3 Overview of standard 3d data formats

This paragraph is meant to give an overview of the most popular and known data-formats used in CAD systems.

a. IGES (Initial Graphics Exchange Specification)

IGES was first developed by the National Bureau of Standards, Boeing Corporation, and General Electric Corporation and published by the National Bureau of Standards in 1980. Initially it was only possible to exchange basic entities like points, lines, arcs and circles. In September 1981 IGES became ANSI standard Y14.26M for CAD/CAM communications.
The latest versions of IGES are able to transfer constructive solid geometry and boundary representations of solid models.
The IGES can be in either binary or ASCII format. In ASCII the file is easy to read and modify by hand but it tends to be larger.

b. STEP (Standard for the Exchange of Product model data)

The STEP-format is defined in a number of ISO 10303-xxx Standards (where xxx is the part number). It consists of Integrated Resources, Description Methods, Implementation Methods and Conformance Testing Methodology and Framework STEP can be understood as a construction kit for describing application protocols using integrated resources with defined rules and standardised methods. It integrates product data, that means all data that is often stored in different systems is in STEP combined in one document.
STEP data integration eliminates redundancy and the problems caused by redundant information.

c. VRML (Virtual Reality Modelling Language)

VRML is a Modelling Language for describing 3d-worlds. Besides the geometry it is possible to define interactivity and behaviour of objects. The benefit of VRML, compared to other formats, is the ability to use it via internet and hyperlink it with the World Wide Web. Therefore special browsers and plug-ins are available.

In VRML, a 3d-scene is described by a hierarchical structure of objects, called nodes. Every node in the scene graph represents functionality. It can be distinguished between shape-nodes and grouping-nodes. While shape-nodes represent the geometric representation of an object, group nodes can hold child-nodes that may describe the scene. Each node contains a list of fields that hold values defining parameters for its function. There are nodes for defining geometry and transformation of an object, materials, light sources and sensors for detecting interaction.

d. X3D (extensible 3d Graphics)

X3D is an Open Standards XML-enabled 3d file format to enable realtime communication of 3d data across all applications and network applications. It is the next revision of the VRML97 ISO specification.

In contrast to VRML97 that is defined in a monolithic manner, X3D has a component-based architecture. This makes it possible to define only a "subset" of functionality. Such subsets are specified as "Profiles". A profile is composed of modular blocks of functionality, so called "Components". Users can extend profiles by changing or adding levels of components. This architecture allows adapting the application to the users demands. So, high flexibility and efficiency are guaranteed.

X3D supports:

- **3d graphics** - Polygonal geometry, parametric geometry, hierarchical transformations, lighting, materials, multi-pass/multi-stage texture mapping, pixel and vertex shaders, hardware acceleration
- **2D graphics** - Spatialized text; 2D vector graphics; 2D/3d compositing
- **CAD data -** Translation of CAD data to an open format for publishing and interactive media
- **Animation -** Timers and interpolators to drive continous animations; humanoid animation and morphing
- **Spatialized audio and video -** Audiovisual sources mapped onto geometry in the scene
- **User interaction -** Mouse-based picking and dragging; keyboard input
- **Navigation -** Cameras; user movement within the 3d scene; collision, proximity and visibility detection
- **User-defined objects -** Ability to extend built-in browser functionality by creating user-defined data types
- **Scripting -** Ability to dynamically change the scene via programming and scripting languages
- **Networking -** Ability to compose a single X3D scene out of assets located on a network; hyperlinking of objects to other scenes or assets located on the World Wide Web
- **Physical simulation -** Humanoid animation; geospatial datasets; integration with Distributed Interactive Simulation (DIS) protocols

e. OpenFlight

OpenFlight, MultiGen-Paradigm's native 3d content is the leading visual database standard in the world and has become the defacto standard format in the visual simulation industry.
OpenFlight contains a hierarchical scene description. It includes:

- Levels of detail (LOD)

- Culling volumes
- Switch Nodes
- Drawing priority
- Binary separating planes

f. DXF/DWG (Drawing Exchange File)

DXF is available as ASCII or binary encoded file format. It is used for transferring vector information between different applications, first of all CAD systems.
The format was developed by AutoDesk and is important for transferring data from and to AutoCAD. DXF is rich in features, including: support for 3d objects, curves, text, associative dimensioning, and is an easy format to parse.
The DXB format is a binary representation of a DXF file and they are usually smaller and faster to load than the equivalent DXF file.

g. 3DS (3d-Studio File Format)

It is a proprietary format of 3d-Studio Max, one of the most widely used 3d-modelers.

h. STL (Stereolithography format)

STL files are the native file format of the SLA CAD software created by 3d Systems of Valencia, CA, USA.
The stereolithography format is available as ACSII or binary encoded file, although binary is far more common due to the resulting size that is much larger for ASCII-encoded files. It contains a list of triangular surfaces that describe a computer generated solid model.
Although STL is a very simple format it is very common because of its use in Rapid Prototyping processes.

i. JT (= Jupiter Tesselation; Direct Model)

The DirectModel format.JT was developed by EAI and Hewlett-Packard as an efficient format for the visualization of large 3d models.
JT is widely used in the automobile and aircraft industry for visualization of 3d geometry as well as for analysis and optimization. A number of CAD systems are able to export JT format. Otherwise it is possible to convert existing data to the JT format.
It can store directly renderable geometry, analytical geometry, geometric attributes, facet information, lighting models, texture maps, user metadata, hierarchical product structures, product manufacturing information (PMI), geometric and functional dimension and tolerancing data, and attributes (such as color, layer and font).
JT is a closed propriety format. Therefore it is not adequate for CADPIPE projects demands.

j. IFC (Industry Foundation Classes)

IFC is a format that is used for exchanging architectural data. Compared to non-architectural data it is important to keep the intelligence of the objects. That means a door shall not only be transferred as the geometry of a door, but also with the information that this object is a door. So it is possible to apply “door-specific” operations to that object.
Missing information can cause lots of costs because data can’t be processed the right way or aren’t processed at all.
Data elements represent the parts of buildings, or elements of the process, and contain the relevant information about those parts.
It is defined with the EXPRESS language as specified by STEP und SC4 of ISO (Specifically for construction industry).

k. GDL (Geometric Description Language)

It’s a language for describing assemblies and subassemblies. GDL has been continuously developed since 1982. GDL files contain 3d geometry, 2D symbols such as text-based and numerical parameters and properties for describing material.
An advantage of GDL is the ability of changing parameters very quickly via dialogboxes.

l. VDA (Verband der Automobilindustrie)

VDA-FS is a data format described by the German Association for Automotive Engineering. “FS” stands for “**F**lächen**s**chnittstelle” (surface interface). It is used for exchanging CAD data between different CAD systems. “FS” describes the kind of interface that is reduced to surfaces. That means you can only transfer 3d information and no drawing information as it is possible with formats like STEP or IGES.
There are subsets of this interface that restrict the kind of exchangeable information, e.g. only curves and lines.
VDA-FS is primarily used for converting 3d information from complex and expensive CAD systems like CATIA and Pro/ENGINEER to cheaper systems like Tebis.

m. OBJ (Object File)

Wavefront OBJ (object) files are used by Wavefront's Advanced Visualizer application to store geometric objects composed of lines, polygons, and free-form curves and surfaces.
Object files can be in ASCII format (.obj) or binary format (.mod).

n. CDF (CAD Distillation Format)

As a descriptive language for three-dimensional worlds, VRML97 is used in the widest variety of fields of application. A frequent disadvantage is that extensions and adaptations for the respective requirements are only possible individually and the advantages of a standard get lost as a result. In the case of X3D, however, the option of various characteristics was already incorporated in its development. Separate working groups are often formed for the individual fields of application. They are intended to push ahead the development of X3D from specific viewpoints. The Web3D Consortium also established the CAD Working Group, which is intended to enable and promote the application of X3D in the environment of computer aided design. The objective is to make complex CAD data accessible to applications for visualization, communication and technical documentation in the fields of customer service, marketing and sales. This is to be achieved by creating an open data format based on X3D as well as by defining the related data transfer processes. The CAD Distillation Format (CDF) developed by the working group is intended not only to facilitate the transmission of CAD data in an open format but also to reduce the complexity of the data. To this end, the information necessary for the particular application has to be precisely extracted from the CAD data, hence the term *distillation*. The quality of the data has to be adapted to its intended use. A rigid approach may not be used to resolve this problem. Rather, a flexible concept has to be created. The CDF approach is to define a pipeline framework composed of the following modules:

- CDF Exporter,
- CDF Filter and
- CDF Importer.

In the process, a CDF Exporter very generally describes a component that writes CDF data and a CDF Importer very generally describes a component that reads CDF data. A CDF Filter reads CDF data, transforms it in various ways and writes it again afterward. Thus, for example, a filter can reduce the number of polygons of a tessellated geometry or even delete unnecessary metadata (e.g. comments) from the CDF data.
Making statements about the scope and the concrete implementation of the CDF concepts is difficult to the extent that development is still ongoing and results have only been partially published so far. Thus, for example, a so-called "Proposed Draft Amendment (PDAM)" for ISO/IEC 19775-1:2004 exists, which already contains the description of the CAD geometry component and the CAD interchange profile. The CAD geometry component contains the nodes:

- IndexedQuadSet,
- QuadSet,
- CADAssembly,
- CADPart,
- CADFace und
- CADLayer.

The first two nodes support the description of the geometry and define Level 1 of the component. The nodes CADAssembly, CADPart and CADFace are needed to describe the product structure and together with the node CADLayer make up Level 2 of the component CADGeometry. The profile CADInterchange is based on this level. It contains another series of other components however such as Grouping (Level 1), Shape (Level 2), Navigation (Level 2), etc.

o. SAT (Standard ACIS Text Save File)

SAT is an external file format of ACIS and stores modelling information created by ACIS. It has an open format so that external applications, even those not based on ACIS, can have access to the ACIS geometric model.

4 Properties of 3d data formats and their relevance for CADPIPE

For evaluating the conversion pipeline between different CAD systems it is necessary to look at a number of properties of 3d data formats. This paragraph will give an overview of the properties taken into account. They will be examined regarding to their relevance for the CADPIPE project.

a. triangulated vs. parametric format

In triangulated data format objects geometry is represented by many triangles (faces). To define one face three points (vertexes) are needed. Any vertex is represented by three coordinates in space. The precision of the surface description depends strongly on the number of triangles. For high precision the number of triangles increases, what yields to large file size. Triangulated formats are easier to handle than parametric formats.

On the other hand the parametric format describes the surface mathematically exactly. This reduces the effort for changing geometry. The precision of the surface description does not depend on the file size. For handling parametric data formats you need to have special mathematically knowledge.

Also for interactive real-time visualizations, parametric formats can be too difficult to handle efficiently.

b. ASCII vs. binary encoding

ASCII encoding convinces because of its readability. This makes it easy to create and edit CAD data with help of an editor. Binary encoding is not readable and therefore not easy to handle. Encoding CAD data in binary format provides a better performance.

c. One vs. multi file representation of geometry

An assembly can be represented by one or more CAD files. Both representations have their pros and cons. A one file representation is more compact. For exchanging data only one file needs to be transferred. On the other hand this file can be very large and complex.

The advantage of a multi file representation is the reusability of some files. If there is for example an assembly with a number of identical screws, the file describing that screw needs to exist only one time. For the description of the assembly it can be referenced. So the assembly is described by a number of files where each file represents one part. For exchanging data the large number of files might be a disadvantage as you have to pay attention not loosing any information.

d. Hierarchy

According to their functionality assemblies are hierarchically structured. The relationships between objects are mapped to a transformation hierarchy. As a property of 3d data formats it can be examined if objects are stored in that hierarchy when exporting them to a special data format.

Loosing hierarchal information by exporting objects to a data format is also loss of information as for example animations are based on that hierarchy.

e. Object names

Naming objects makes it possible to clearly identify each object. Therefore it is import to keep object names by exporting objects to a 3d data format.

f. Save material inside the CAD file vs. extern material representation

3d objects can be mapped with material information. Using materials makes the objects appearance more realistic and is therefore a big support for the acceptance of models in CAD- and VR-systems. A material is defined by several parameters such as ambient, diffuse and specular light, shininess and transparency.

A 3d data format needs to encode this material information for every object that is mapped. The material information can be saved inside the CAD file or in an extern file. In this case material description will be referenced from the CAD file. If you save the material information inside the CAD file only one file needs to be transferred when exchanging data between different working groups. On the other hand, saving material description in an extern file makes it possible to reuse that material for several objects.

g. Materials per object

One object can be mapped with different materials. As a property of 3d data formats it can be examined whether it is possible to keep these materials when exporting the object to a specific 3d data format.

h. Texture representation

Besides material information it is possible to map textures to 3d objects. Textures are used for improving performance because modeling precision can be reduced. Textures can replace a very precise modeling that needs lots of triangles. Textures make the appearance of an object more realistic than materials can do, as it is possible to use photos for texturing.

Texture representation has no priority in CAD systems. Therefore that function is not that good supported. Mostly you can just choose between using texture or not using texture, that means there are no further options.

i. Viewpoints

Viewpoints can be created for storing characteristic positions from which a model can be viewed. For keeping these positions it is important to export the viewpoints to the 3d data format.

j. Level of Detail (LOD)

Level of Detail describes the reduction of the calculation complexity by providing one object in different levels of details. The Level of Detail of an object results in different objects that differ in quality. The show of a Level of Detail object mostly depends on the distance of the viewer to the object. The object that is shown from far-away position needs to be less detailed. Therefore the number of triangles (polygons) can be reduced or the objects shape can be approximated by a simpler object. Another way of reduction is the change of texture.

If Level of Detail objects are defined it is important to keep them when exporting data to specified 3d data format. Therefore Level of Detail is chosen as an important property of 3d data formats.

k. Animation

One application area for virtual reality systems is the spatial representation of motion sequences of CAD-models. They allow a better description of complex technical behaviors. For visualizing the parts movements of one assembly in virtual environments you need information that transcend the pure geometry.
Most of that information is contained in CAD-data, e.g. as animations or as the result of an analysis.
As it is an aim of VR to present animations for describing procedures and processes one should analyze the ability to keep information on motion when converting CAD-data to a 3d data format appropriate to real-time rendering.

5 Description of the conversion pipeline in CATIA V5 and Pro / Engineer Wildfire

As current conversion pipelines the CAD systems CATIA V5 and Pro / Engineer Wildfire were evaluated. It was checked what formats can be imported and exported by these systems. Furthermore the exported 3d data formats were examined with respect to the above mentioned properties.

5.1 CATIA V5

From the set of declared 3d standard data formats (see point 3) CATIA V5 can import the following ones:

- DXF 3d
- STEP

- IGES

Besides CATIA can import a number of CATIA intern formats and such that are not that well known and often used.

The following 3d standard data formats can be exported:

- VRML 1.0
- VRML 2.0
- STL / DMU
- DXF 3d
- STEP (203 iso, 203+ext, 214 iso)
- IGES

Now these formats will be examined in consideration of the properties described above.

5.1.1 VRML 1.0

VRML 1.0 is a triangulated data format that is encoded in ASCII. Exporting VRML 1.0 data format with CATIA V5 yields to one file that stores all information. This file contains the object hierarchy, material information and viewpoints. Object names are lost when exporting VRML 1.0 with CATIA V5. Furthermore only one material per object is stored in the vrml-file. Textures, material in terms of physical properties and Level of Detail aren't exported.

5.1.2 VRML 2.0

VRML 2.0 supports the same properties as VRML 1.0. In addition it stores texture-information.

5.1.3 STL / DMU

STL is a triangulated format where all information is included in one file. It is binary encoded and results in a one file representation. User can choose precision of the geometries triangulation.

5.1.4 DXF 3d

DXF 3d is a triangulated format that is encoded in ASCII. Exporting DXF 3d format with CATIA results in one file.

Properties like hierarchy, objectnames, materials, textures, LODs and viewpoints aren't supported.

5.1.5 STEP

The STEP format can be exported in several versions: STEP 203 ISO STEP 203+ext, STEP 214 ISO. All versions are parametric formats that are encoded in ASCII. STEP 203

ISO and STEP 203+ext result in a one file representation while STEP 214 ISO can also provide a multi file solution. All versions keep hierarchy and object names when they are exported with CATIA V5. STEP 203 ISO isn't able to store material information. The other versions store material information inside the CAD file. Only one material per object is exported. Information on texture, physical material, viewpoints and LODs aren't exported to any STEP format.

5.1.6 IGES

The IGES format is a parametric format that is encoded in ASCII. Exporting IGES with CATIA V5 results in one file. Information about hierarchy, object names, material, texture, viewpoints and LODs are not exported to that file.

CATIAs export options of all mentioned formats are summarized in Figure 1.

5.2 Pro / Engineer Wildfire

From the set of declared 3d standard data formats (see point 3) Pro /Engineer Wildfire can import and export the following ones:

- VRML 1.0
- VRML 2.0
- STL
- DXF 3d
- STEP
- SAT
- IGES
- SET
- VDA

5.2.1 VRML 1.0

Pro /Engineer exports VRML 1.0 as a triangulated and ASCII encoded file. It results in a multi file representation. VRML 2.0 stores information on hierarchy, object names, material, viewpoints and Level of Detail. Material information is saved inside the vrml-file and it is possible to attach more than one material to one object. It is not possible to export texture and material in terms of physical properties.

5.2.2 VRML 2.0

VRML 2.0 differs from VRML 1.0, as it is not able to export LODs. Besides it is encoded binary.

5.2.3 STL

The exported STL format is triangulated, binary encoded and monolithic. User can choose precision of the geometries triangulation.

5.2.4 DXF 3d

Exporting DXF 3d with Pro /Engineer results in a triangulated representation that is either ASCII or binary encoded. Exportation provides one file.

5.2.5 STEP

STEP is generally a parametric exchange format with ASCII encoding. Pro /Engineer creates just one STEP file, which contains the geometry and information about model structure (hierarchy) and objects names. It is possible to export a STEP file in several standards (ap203_is, ap214_cd, ap202_is, 203_is_ext, ap_214_dis, ap209_dis, ap214_is). Appearances, textures, physical properties, viewpoints and LOD parameters aren't supported. STEP based data exchange between several CAD systems is mostly excellently and precisely working.

5.2.6 SAT

This data format is also a parametric and ASCII encoded format. Disadvantage of that format is the loss of hierarchy and object names. Features concerning the objects appearance are supported (material in term of colors). Textures, physical properties, viewpoints and LOD are not supported.

5.2.7 IGES

IGES is supported by most CAD systems. Anyhow data exchange is accompanied with several precision problems. Mainly the IGES based geometry imported in Pro /Engineer costs many time for repairing or reconstruction of the geometry. IGES is ASCII encoded and the geometry is parametrically defined. Conditioned on an optionally configuration file it is possible to export either only one file or one file per assembled component. Thus hierarchy and object names can be optionally exported. Textures, physical properties, viewpoints and LOD properties are not supported.

5.2.8 SET

It is a parametric, ASCII encoded data format. SET is based on surfaces and provides no solid geometry. Conditioned on an optionally configuration file it is possible to export either only one file or one file per assembled component. Hierarchy is supported, but the object names will be lost. That's all, what SET supports.

5.2.9 VDA

VDA is a parametric, ASCII encoded data format. It is based on surfaces and provides no solid geometry. Conditioned on an optionally configuration file it is possible to export either only one file or one file per assembled component. No additional options are supported.

Pro /Engineers export options of all mentioned formats are summarized in Figure 2.

6 Interfaces for transferring kinematic data

In Virtual Reality the description of processes and procedures can be supported by using animations. They allow the description of complex and complicate behavior as it is impossible with words.

Animations are also used for the examination of space in the construction process. There mustn't be any collision for example. One way in production could be that you construct the models geometry in CAD. Then you convert your model to a VR-system and define kinematics inside the virtual environment. Now it can happen that you detect a lack in construction (e.g. a part has not enough space for moving). Then you have to go back to the CAD and change the construction of the part.

Therefore it is more comfortable to define the geometry and all properties, including kinematics, inside the CAD-system and convert all the information to a VR-system afterwards. Problems can already be detected earlier in the production chain.

For keeping the information on kinematics data when converting CAD-data to VR-data it is necessary to analyze the interfaces provided by the CAD-systems.

This chapter shall give a short introduction to the possibilities Pro / Engineer and CATIA V5 provide for the export of kinematic data.

6.1 Pro / Engineer Wildfire

In CAD systems kinematic data are particularly produced when modeling the assemblies. As only jointed parts can move relatively to each other, the kind of joint is very important. In virtual development it is a basic principle to map the real degrees of freedom to the model in virtual environment. For being able to correct the design and function of a model in an early step of the production process, properties should already be defined in the CAD systems.

For describing steep connections Pro / Engineer provides the opportunity of defining constraints. They restrict the joints degrees of freedom. As a rule there are defined as many constraints as possible for determining the connection exactly.

For modeling loose connections two different kinds of joints are provided that define the constraints and keep the essential degrees of freedom footloose. Unlike spatial relationships a joint describes a loose connection logically complete. A model created in that manner contains information on geometry as well as information on possible moving sequences and is therefore a base for the analysis of motion and often for an animation inside the CAD-system.

For a motion study inside Pro / Engineer a ground body has to be specified. Furthermore it is necessary to define a motor on at least one of the remaining parts. It describes the kind and the starting point of the prefaced motion. After the definition of parameters like duration and frame rate of the analysis it is possible to identify the motion sequences the whole assembly has to fulfill as a result of the prefaced motion. This identification depends on the geometry and the joints defined before.

The result of such an analysis can be viewed as an animation and can be saved. In doing so a so called Playback-file (PBK-file) is produced. That file is only for usage inside Pro / Engineer and serves for loading the result of the motion analysis in a later session.

Besides the common possibility of creating a video, a so called frame-file (FRA-file) can be exported. It is used for creating a motion envelope. The frame-file contains a mathematical description for the position and orientation of all parts of every frame of the simulation. A motion envelope, often called as translation or rotation volume, serves for the examination of needed installation space.

Pro / Engineer also allows the free creation of animations. This can for example be a so called snapshot-animation where any arbitrary view and any state of the CAD-model are displayed in one frame of the animation. These animations can also be saved and exported as described above.

6.2 CATIA V5

For the analysis of motion sequences CATIA provides the *DMU-Kinematics-Simulator*. Together with the *DMU-Space-Analyser*, the *DMU-Fitting-Simulator* and the *DMU-Optimizer* it forms the *DMU-Navigator*.

With the *DMU-Kinematics-Simulator* it is possible to define and simulate very complicate mechanisms such as limbs, joints and strengths. The simulation also results in an animation that can be converted to a video sequence (AVI-, MPEG- or JPEG).

The STEP interface does not support the transfer of kinematic data.

7 Results and discussion

The examination of several 3d data formats and the conversion pipelines of CATIA V5 and Pro /Engineer Wildfire have shown big differences between the exported properties that are coded inside a 3d file.
Most important properties that were analysed are the preservation of the object hierarchy and the object names when exporting a CAD file to another format. Formats that don't keep names and hierarchy are difficult to handle because the identification of objects is not clear and the assemblies' structure is lost.
For keeping the realistic appearance of a CAD model it is necessary to use data formats that can encode material information. For increasing efficiency and speeding up performance the use of level of detail objects is very profitable.
Some of the suggested 3d standard formats don't support all of these properties. VRML 1.0 and VRML 2.0 provide most of the mentioned features and therefore satisfy CADPIPEs demands best.

8 Conclusions and recommendations

It is recommended to use a 3d file that keeps hierarchy, object names, material information and level of detail objects.

While analyzing the declared standard formats and the conversion pipelines of CATIA and Pro /Engineer we came to the conclusion that VRML 1.0 and VRML 2.0 provide most features and therefore fit best for CADPIPE project. Both formats are triangulated and well supported in the native export filters of the CAD systems.

Furthermore we suggest the use of X3D internal CADPIPE format as it ensures upward compatibility. It is the successor of VRML 2.0 and satisfies because of its modular design what guarantees a future extensibility. It seems to be a trend-setting format in CAD applications and should therefore be taken into account in CADPIPE project as well.

References

X3D	-	www.web3d.org
CATIA V5	-	http://www-306.ibm.com/software/applications/plm/catiav5/
Pro/Engineer	-	http://www.ptc.com/appserver/mkt/products/home.jsp?k=403
IGES	-	http://www.nist.gov/iges/
STEP	-	http://www.steptools.com/
		http://www.alias.com/eng/support/studiotools/documentation/DataTransfer/appendix13.html
VRML	-	http://www.mathworks.com/access/helpdesk/help/toolbox/vr/ch_introduction16.html
General	-	http://www.cad.de/de.shtml
		http://www.okino.com

Appendix A:

3D format	version	CATIA V5												
		import	export											
			yes/ no	**t**riangulated / **p**arametric	**A**SCII / **B**inary	geometry files **Mo**no / **Mu**lti	hierarchy	object names	material (optical) **i**ntern / **e**xtern	materials per object	texture	material (physical)	viewpoints	LOD
VRML	1.0	-	x	t	A	Mo	x	-	i	1	-	-	x	-
VRML	2.0	-	x	t	A	Mo	x	-	i	1	int/ext	-	x	-
X3D		-	-											
STL /DMU		-	x	t	B	Mo	-	-	-	-	-	-	-	-
DXF 3D		x	x	t	A	Mo	-	-	-	-	-	-	-	-
JT		-	-											
STEP	203 iso	x	x	p	A	Mo	x	x	-	-	-	-	-	-
	203 + ext		x	p	A	Mo	x	x	i	1	-	-	-	-
	214 iso		x	p	A	Mo/Mu	x	x	i	1	-	-	-	-
SAT		-	-											
IGES		x	x	p	A	Mo	-	-	-	-	-	-	-	-
SET		-	-											
VDA		-	-											

Figure 1 Conversion pipeline CATIA V5

3D format	version	Pro/E Wildfire												
		import	export											
			yes/ no	**t**riangulated / **p**arametric	**A**SCII / **B**inary	geometry files **Mo**no / **Mu**lti	hierarchy	object names	material (optical) **i**ntern / **e**xtern	materials per object	texture	material (physical)	viewpoints	LOD
VRML	1.0	x	x	t	A	Mu	x	x	i	n>=1	-	-	x	x
VRML	2.0	x	x	t	B	Mu	x	x	i	n>=1	-	-	x	-
X3D		-	-											
STL		x	x	t	B	Mo	-	-	-	-	-	-	-	-
DXF 3D		x	x	t	A/B	Mo	-	-	-	-	-	-	-	-
JT		-	-											
STEP	ap203_is, ap214_cd, ap202_is, 203_is_ext, ap_214_dis, ap209_dis, ap214_is	x	x	p	A	Mo	x	x	-	-	-	-	-	-
SAT		x	x	p	A	Mo	-	-	i	n>=1	-	-	-	-
IGES		x	x	p	A	Mo/Mu	-/x	-/x	i	n>=1	-	-	-	-
SET		x	x	p (no solids)	A	Mo/Mu	x	-	-	-	-	-	-	-
VDA		x	x	p (no solids)	A	Mo/Mu	-	-	-	-	-	-	-	-

Figure 2 Conversion pipeline Pro /Engineer Wildfire