

Why is CATIA V5 Data Management different?

An introduction to the importance of managing CATIA V5 objects and links in a synchronized way using the CATIA Integration for SMARTTEAM

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Agenda

- Why is CATIA V5 data management different?
- Requirements for CATIA V5 PDM systems
- Solution alternatives
- Limitations of the CDI/SAP interface
- Advantages by using SMARTEAM

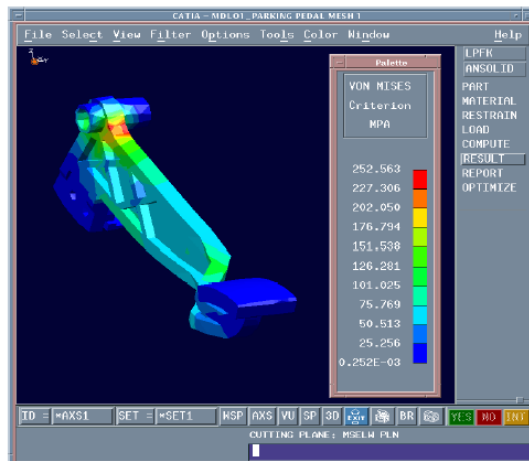
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Change in methodology

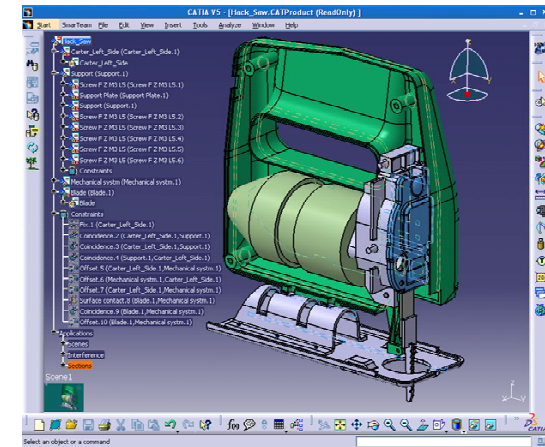
CATIA V4

- Engineers are working in single .model files, which contain all the information (3D, 2D etc.)
- There are no structural components to realize real assemblies within CATIA V4 – for this, a PDM-system is required



CATIA V5

- Working in context of assemblies
- Creating smart modules
- Assemblies are build up in CATIA V5, not in an external PDM-System
- Those Assemblies (CATProducts) define the common context for all its components



Press coverage of the EDM Report (translated)

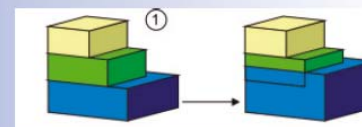
- „In the middle of 2000, the BMW Group initiated a program to handle the implementation of CATIA V5. After evaluating the software, it was recognized, that **CATIA V5 is not just a new version, but a new generation of CAD, which differs in Architecture, Engineering Philosophy, but also in complexity – compared to CATIA V4**“
- „The main innovation of CATIA V5 is the **parametric associative method** of engineering. It enables to **perform changes more easier** and with **time savings**. On the other side, by saving the **logic of the engineering process**, this **process remains understandable**. This offers chances to streamline engineering processes. Therefore, the BMW group decided to switch from CATIA V4 to V5 [..].“

Erschließung des Potentials durch neue Arbeitsweisen

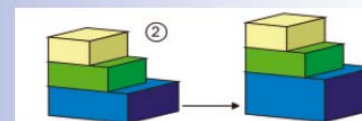
- „CATIA V5 ist nicht nur eine neue Version, sondern eine **neue CAD-Systemgeneration**, die sich in der Architektur, **Konstruktionsphilosophie** aber auch in der Komplexität deutlich von CATIA V4 unterscheidet.“
- „Die wesentliche Neuerung bei CATIA V5 ist die **parametrisch assoziative Konstruktionsmethodik**. Sie ermöglicht einerseits eine vereinfachte Durchführung von Änderungen mit **zeitlichem Einsparpotenzial**. Andererseits wird durch die Hinterlegung der Logik des Konstruktionsprozesses dieser jederzeit nachvollziehbar. Da sich hierdurch Chancen der **Prozessverbesserung** bieten, hat sich die BMW Group für einen Umstieg von CATIA V4 auf V5 entschlossen.“
- „Einer der ersten Standards [...], der parametrisch mit der Adaptertechnik aufgebaut wurde, [...] wird in die eigene Konstruktion eingesetzt und kann mit Parametern an unterschiedliche Situationen angepasst werden. Die reine Konstruktionszeit der Konsole in CATIA V5 kann damit gegenüber V4 um bis zu **40 Prozent reduziert** werden.“
- „In verschiedenen [...] durchgeführten Pilotprojekten [...], in denen erstmals CATIA V5 produktiv eingesetzt wird, kommt überwiegend die Adaptermethode zum Einsatz. Damit wird besonders die Übersichtlichkeit der Konstruktion gewährleistet.“
- „Im Bereich der Fertigungsmittelkonstruktion bestehen derzeit noch einige Hemmschwellen, die einem umfassenden Einsatz von CATIA V5 entgegenstehen. Dies betrifft primär die Verwaltung der CAD-Daten der Anlagen, [...] basierend auf komplexen Strukturen [...], durch ein PDM-System. Zwar liegt ein großer Vorteil von CATIA V5 in der Steuerungsmöglichkeit ganzer Baugruppen oder Produkte über Adapter. Dabei werden aber bauteilübergreifende Beziehungen aufgebaut. Diese Links in einem PDM-System unter Beachtung einer möglichen Versionierung und den Mechanismen des BMW Group eigenen Datenbanksystems PRISMA konsistent zu verwalten, ist heute noch nicht möglich.“
- „Im Fokus aktueller Bestrebungen steht daher die Schaffung einer Systemumgebung mit konsistenter Datenbasis, die eine übergreifende Verlinkung der Strukturen erlaubt und so zu geringeren Datenbeständen und durchgängiger Assoziativität führt.“

Parametrisierungsmethoden für die Konstruktion mit CATIA V5

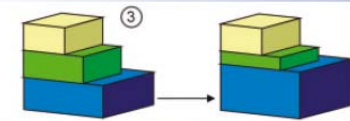
Die aus Sicht eines CATIA-V4-Konstrukteurs einfachste Möglichkeit zu CATIA V5 zu wechseln besteht darin, die V4-Konstruktions-Methode in V5 nachzubilden. Es wird keine explizite Parametrik erzeugt, lediglich die automatisch von V5 angelegte Parametrik wird verwendet. Aus einzelnen Teilen wird eine Baugruppe erzeugt, ohne Zusammenbau-Bedingungen zu verwenden. Ein Teil hat damit keinen Einfluss auf die Lage oder Geometrie anderer Teile. Wie im Bild gezeigt, führt eine Änderung des blauen Quaders zu einer nicht mehr stimmigen Baugruppe, da sich dann der blaue und der grüne Quader überschneiden.



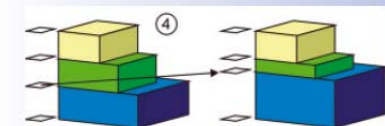
Die nächste Möglichkeit besteht darin, die erzeugten Teile mit Zusammenbau-Bedingungen zueinander zu positionieren. Damit kann ein Teil die Lage anderer Teile beeinflussen, nicht aber deren Geometrie. Eine Änderung des blauen Quaders führt zu einer anderen Lage des grünen und gelben Quaders.



Eine weiterer Ansatz basiert auf einer teileübergreifenden Parametrik. Damit kann jedes Teil die Lage und Geometrie weiterer Teile steuern. Eine Vergrößerung des blauen Quaders verursacht hier eine Verkleinerung des grünen, jedoch keine Lage- oder Geometrieänderung des gelben Quaders.



Weiterhin kann die teileübergreifende Parametrik strukturiert werden, indem Adaptermodelle eingesetzt werden. Damit lässt sich die Lage und die Geometrie jedes Teiles ausschließlich über den mit Drahtgeometrie aufgebauten Adapter, in diesem Fall Ebenen, steuern. Eine Verschiebung der zweiten Ebene von unten bewirkt eine Größenänderung des blauen und des grünen Quaders.

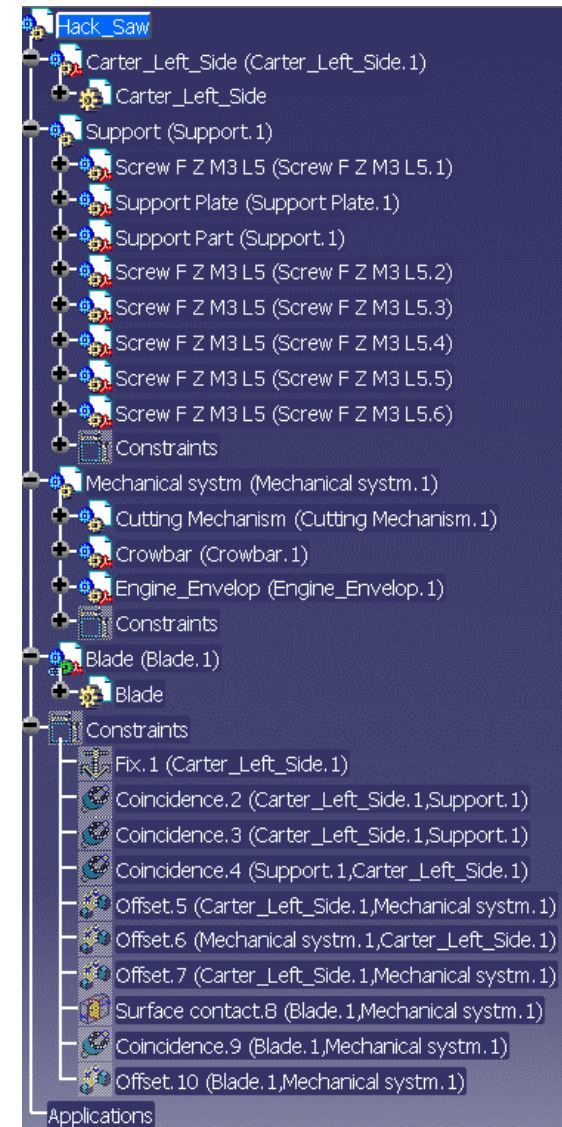


Structural information

Within CATIA V5's new structures, there is lots of new information that did not exist in CATIA V4 before:

- Part-References
- Part-Instances
- Product-Structures
- Documents
- Assembly Features
- Constraints
- Publications
- Application Data
- Context Information
- ...

This information is created by using all CATIA functional components. It represents – beside the geometry – the engineer's **Intellectual capital**.



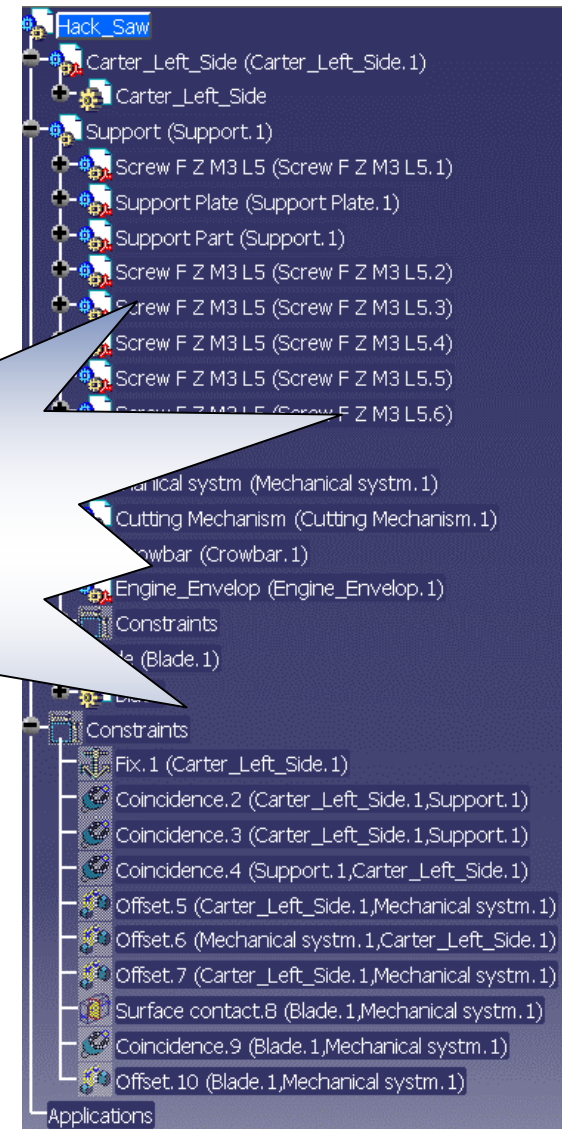
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Only by using those link information, the full capability of CATIA V5 can be used!

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Modular breakdown

Contrary to CATIA V4, V5 uses different file-types to save the information of all engineering applications. This offers a lot of advantages:

- **Improved File Handling**
 - smaller files causing better performance
 - work is being done in hierarchical structures
 - enables functional modules and re-use of geometry
- **Concurrent Engineering**
 - Concurrent work in same modules
 - Parallel work in different departments (NC / geometry / drawing)
- **Relational Design (associativity)**
 - Creation of smart geometry by using relational links
 - Enabling design templates
- **Dedicated Change Management**
 - Diversion of data enables dedicated changes of single components
 - Independent versioning of components (Drawing, Single Part, Assembly ...)
- **Improved Supplier integration**
 - dedicated exchange of single modules
 - only the required information can be exchanged

This data needs to be managed in context with the link information!

CATIA Type	File suffix	Description
Analysis	CATAnalysis	FEM Analysis
CatalogDocument	CATCatalog	CATIA Library
cgr	Cgr	DMU Viewing
DesignTable	xls/txt	Design Table
Drawing	CATDrawing	Drawing
FeatureDictionary	CATfct	Feature Library
FunctionalSystem	CATSystem	Functional System
Internal Component	-	Structural Componente
Material	CATMaterial	Material Library
Model	model	V4 Geometry
Part	CATPart	Single Part
Process	CATProcess	NC
ProcessLibrary	feat	NC Tools Library
Product	CATProduct	Assembly
Shape	CATShape	Simplified Visualization

CATIA V5 Links: **Assembly Links**

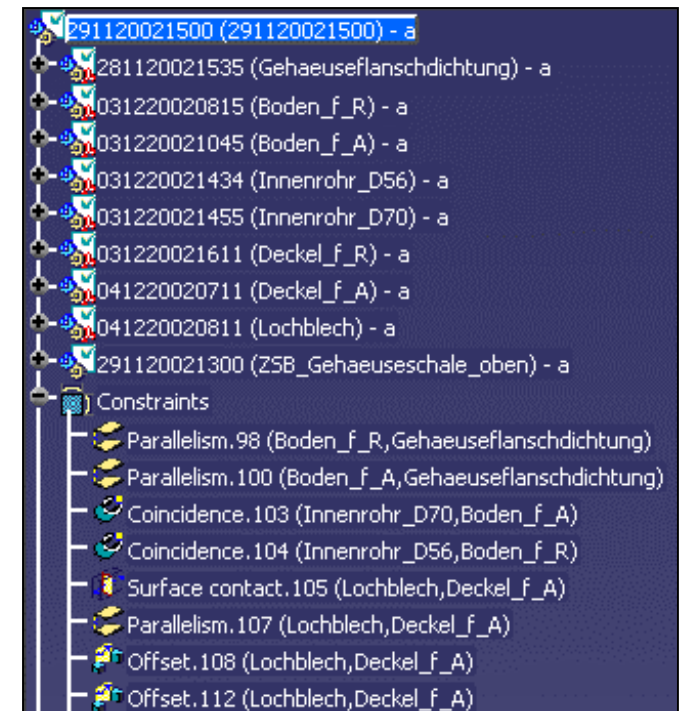
■ Instance Links

Are used when creating an assembly, representing structural information about the CATProduct to

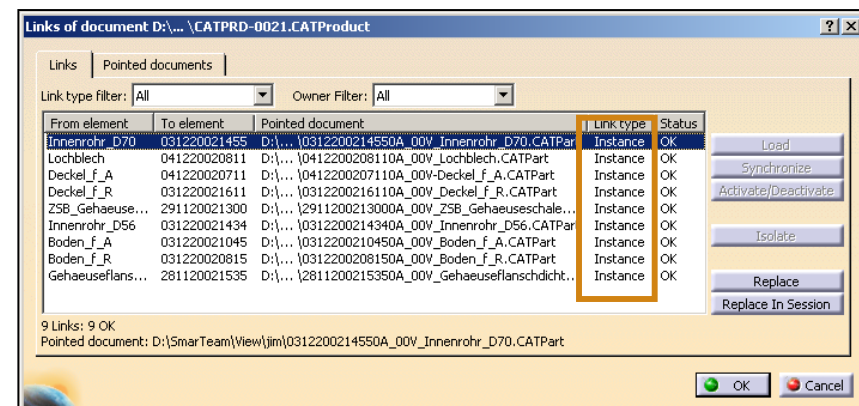
- CATProducts („Instance“)
- CATParts („Instance“)
- V4 Models („shape“)
- Pro/E Parts („Graphic“)
- CGR-files („Shape“)

■ Constraints

Positional information within the assembly



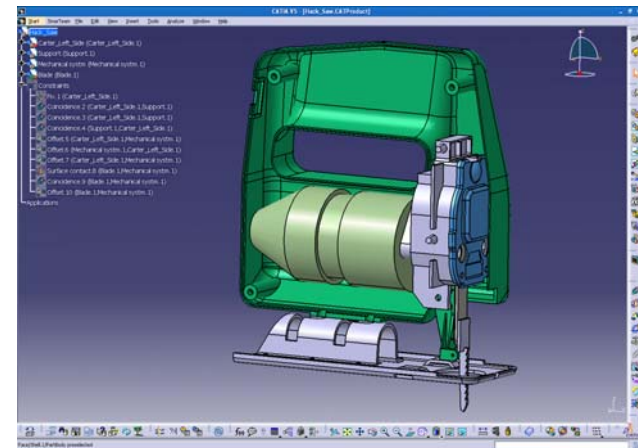
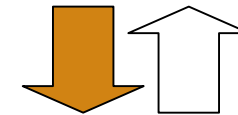
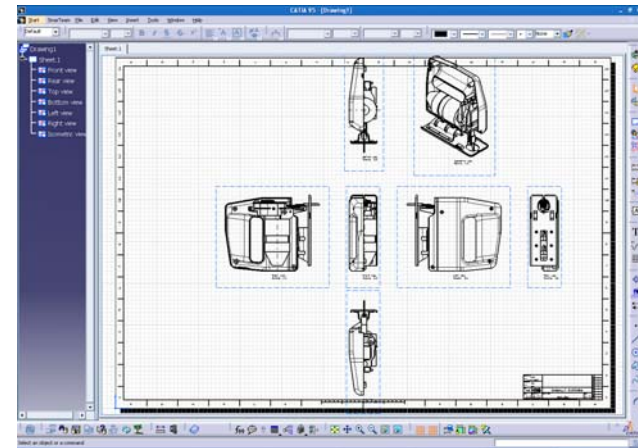
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CATIA V5 Links: Document Links

- They bring information from different departments into a common context
 - CATDrawing – CATPart
 - CATDrawing – CATProduct
 - CATProcess – CATProduct
 - CATAnalysis – CATPart
 - ...

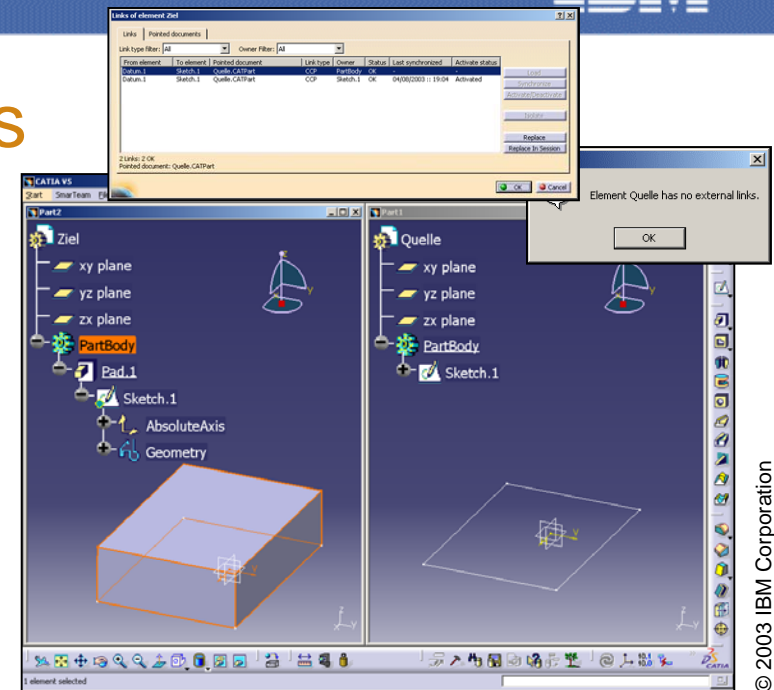
Important: Only the child (e.g. Drawing) knows about its Parent (Geometry) – but not the other way around!



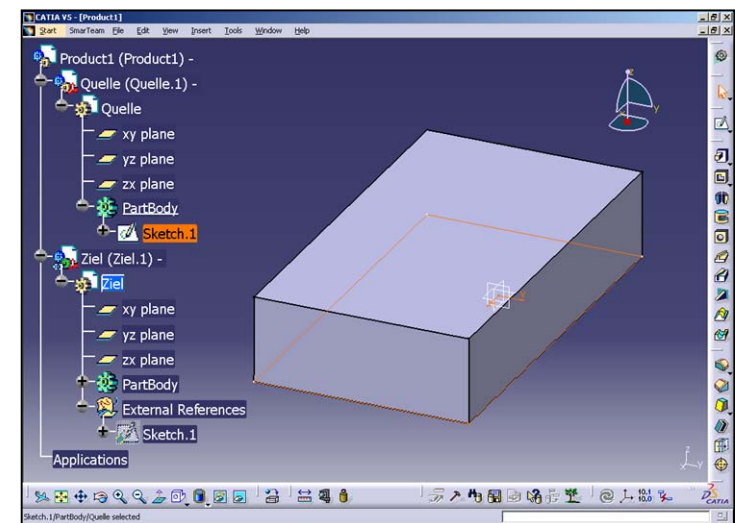
CATIA V5 Links: Reference Links

- **CCP- Links (CutCopyPaste)**
 - Similar to CATIA V4
 - Link between Parts
 - Without context to a certain usage in an assembly
- **Context/Import Link**
 - Created in the context of an assembly
 - Accessing the context information is vital for updates and changes on the links

Again: Reference Links can only be access in one direction (from child to parent)

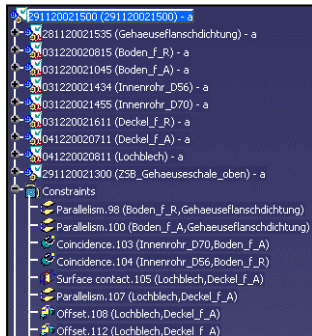


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Summary of CATIA V5 Links

By modularizing the objects in CATIA V5, they are brought into a common context by using diff. types of links

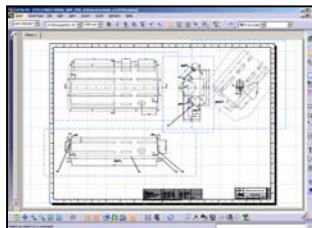


Assembly Links

= Information within the CATProduct

1. **Instance Links**
= Product-Structure-Information
2. **Constraints**
= Positioning information within the assembly

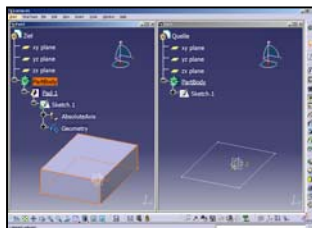
Simply integration on assembly level



Document Links („ViewLink“)

= Parent / Child-Relationship, where only the child knows about its parent – and not the parent about its children

Integration capable to manage different documents



External References / Technological Links

= Relations within the geometry, only known in one direction

1. **CCP (CutCopyPaste)**
Between CATPart and CATPart
2. **Import / Context Link**
Relation between CATPart and CATPart created in an assembly
The assembly (=context) is required for further changes

Deep integration required for Design in Context

Example scenario

Target: Create simple geometry

1. V4 Methodology

Overview

- 4 single isolated parts
- Diameter is defined multiple times
- ... As well as the height

Change efforts

2. External References

view

Intelligence

efforts

- Sketch is re-used by the edges
- The position of the hole is determined in one and re-used in another edge

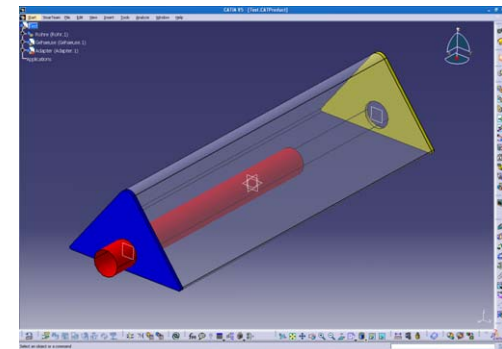
3. External References with Adapter

Overview

Intelligence

s

- New Part has to be created (=Adapter)
- It contains geometry to control the whole assembly
- And enables easy and flexible geometrical controls



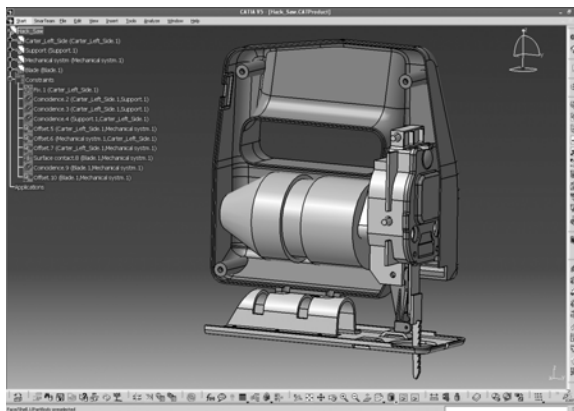
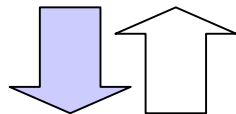
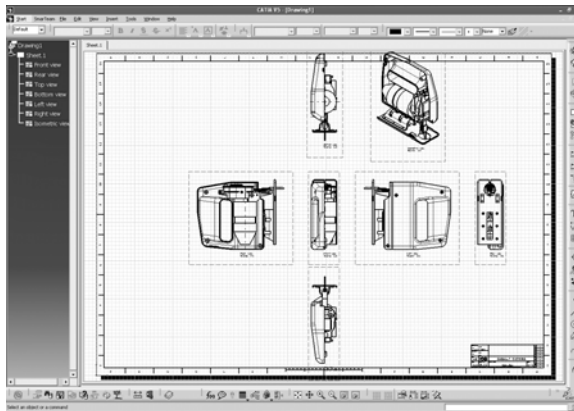
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Detailanforderungen: Arbeit mit Instance-Links



- Beim Öffnen der Baugruppe muß gezielt gewählt werden können, welche Komponenten bearbeitet werden sollen - Ggf. Muß die **check-out** Funktionalität aus CATIA erreichbar sein
- Ist eine Baugruppe in CATIA geöffnet, benötigt der Anwender Informationen aus dem PDM:
 - den **Status der Objekte** (In Bearbeitung, Viewing)
 - die **Aktualität** - wurden Objekte in der Zwischenzeit überarbeitet?
- Weitere Baugruppen-Funktionalität mit direktem Zugriff auf das PDM-System:
 - **Einfügen von neuen Objekten** aus der DB direkt in die aktive Sitzung
Wird eine Komponente mit externen Referenzen eingefügt, müssen die referenzierten Objekte dem CATIA ebenfalls bereitgestellt werden (Adapter)
 - Ersetzen von Objekten durch **geänderte Version** aus der DB
 - Ersetzen von Objekten durch **andere Komponenten** – inklusive Korrektur der Links
- Wird für eine Baugruppe eine **alternative Repräsentation** (Hüllgeometrie) erstellt, muß diese verlinkt werden und bei der Versionierung erhalten bleiben
- Aufgrund der Link-Struktur muß dem Anwender die Wahl offen stehen, wann ein **Versions-Wechsel** der referenzierten Objekte durchgeführt wird.
- ...

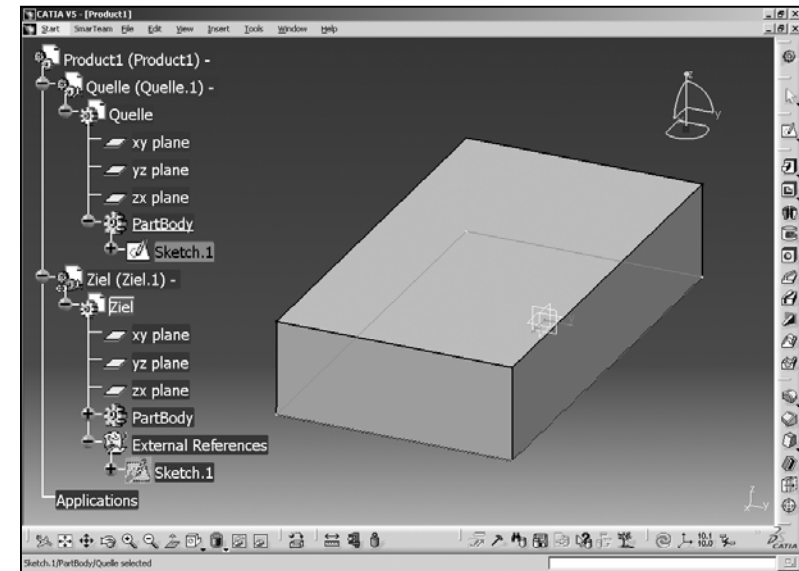
Detailanforderungen: Handling der Dokument Links



- Wird eine Zeichnung/NC-Datei/FEM-Analyse gespeichert, muß die Geometrie ebenfalls gespeichert und entsprechend **gelinkt werden**
- Bearbeitet ein Anwender die Geometrie, muß dieser die zugehörige **Zeichnung in der richtigen Version identifizieren** können – ebenso von der Zeichnung aus. Dies gilt auch für die Zeichnungsableitungen von Einzelteilen innerhalb von Baugruppen.
- Wenn das Drawing geöffnet wird, muß automatisch die **Geometrie mit aus dem Vault** bereitgestellt werden
- Wird die **Zeichnung oder die Geometrie versioniert**, muß dem Anwender die Möglichkeit bereitgestellt werden, das Gegenstück ebenfalls zu versionieren – bestenfalls gestützt durch einen Automatismus.
- Wenn eine Baugruppe versioniert wird, gilt dies auch für **Zeichnungs-Ableitungen der Unterbaugruppen und Einzelteile!**
- Im Sinne des concurrent engineering muß die **getrennte, parallele Bearbeitung der einzelnen Funktionen** (Geometrie, NC, FEM) ermöglicht werden

Detailanforderungen: Externe Referenzen

- Die **Aktualität** bei referenzierten Objekten muß sichtbar sein
- Wird ein Adapter versioniert, müssen die Auswirkungen dargestellt werden (**impact on / impacted by Graph**)
- Die Aktualisierung der Link-Ketten muß vom Anwender gesteuert werden können
- Der **Kontext** einer Externen Referenz muß zugänglich sein
- Korrekte Behandlung dieser Links für die Generierung der **Stückliste im ERP**

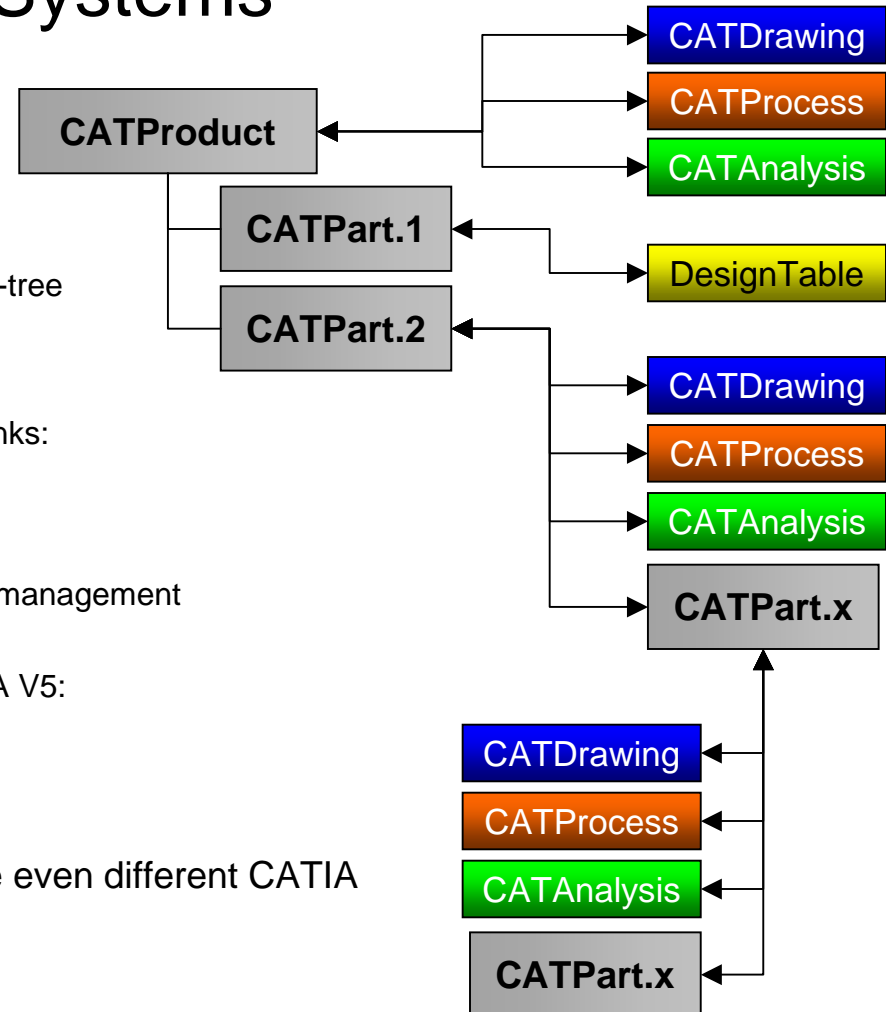


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Requirements for V5-PDM-Systems

- **Support all the file types**
 - Providing according classes
 - Enabling different attributes for each class
 - Capability to treat them in different ways when translating the document-structure into an item-tree for the integration into an ERP-system
- **Intelligent Management of Links**
 - Automatic generation and visualization of V5 links:
 - Assembly Links
 - Document Links
 - Reference Links
 - Taking those links into account during change management
 - Saving Context-Information
 - Providing information that are missing in CATIA V5:
 - Impact On / By (Drawings, References)
 - Access Permissions
 - Actuality
- ... In combination with the desired and maybe even different CATIA releases at the same time

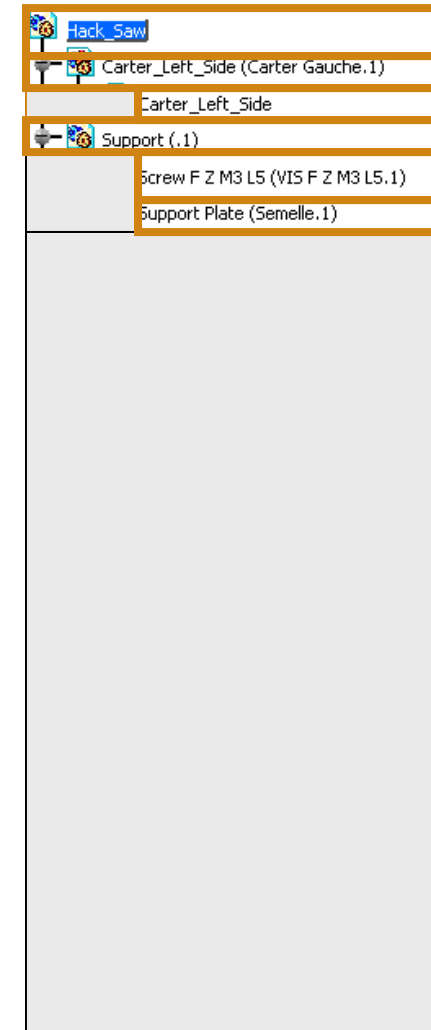


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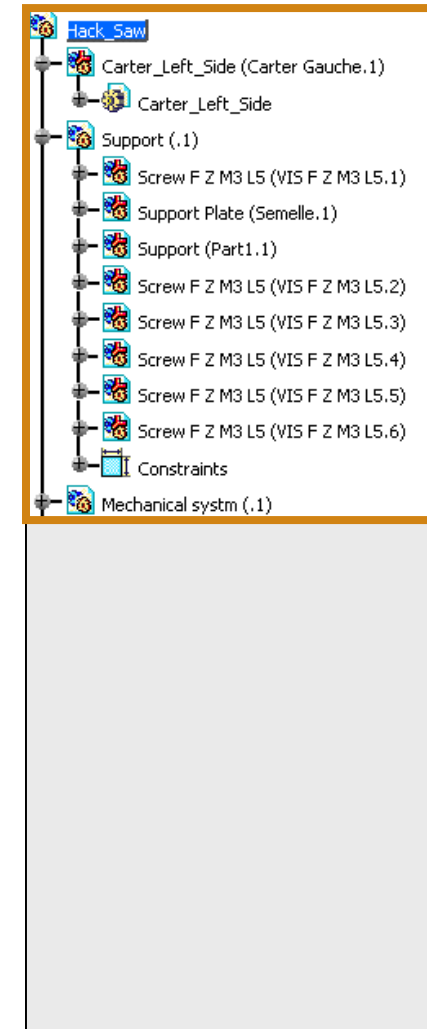
CATIA V5 PDM-Solution: Direct-Integration

- Based on xPDM or VB
- All Relation are kept in the CATProduct
- Only it's realtions to subassemblies and parts are maintained and visualized
- Not all CATIA V5 objects are visible
- Links will only be covered partially or not taken into consideration when a part is versioned (Example: generated drawings and release of an assembly)
- Since reference links are not being considered parametric design is almost impossible



CATIA V5 PDM-Solutions: SMARTTEAM

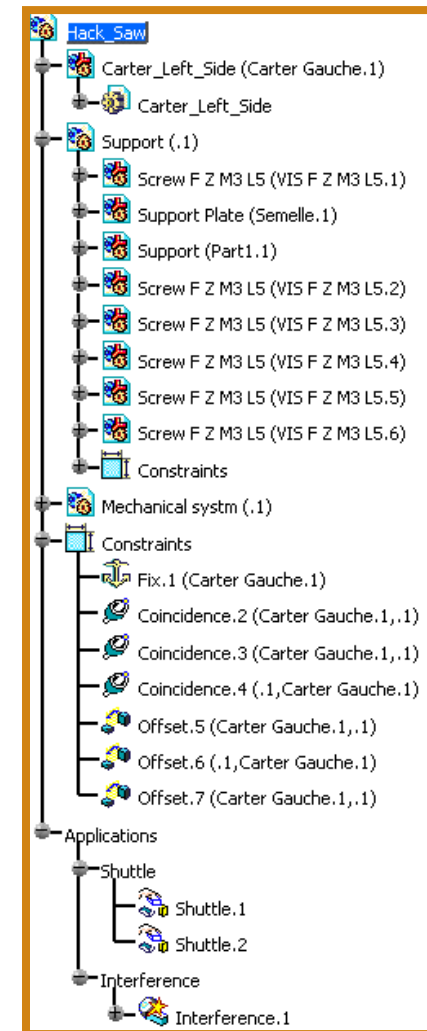
- All relations are kept in the CATProduct
- The internal links as well as all link between parts and other applications (FEM) are visualized and managed
- An optional rule mechanism allows to change the behaviour of the link management when adopted to customer specific processes



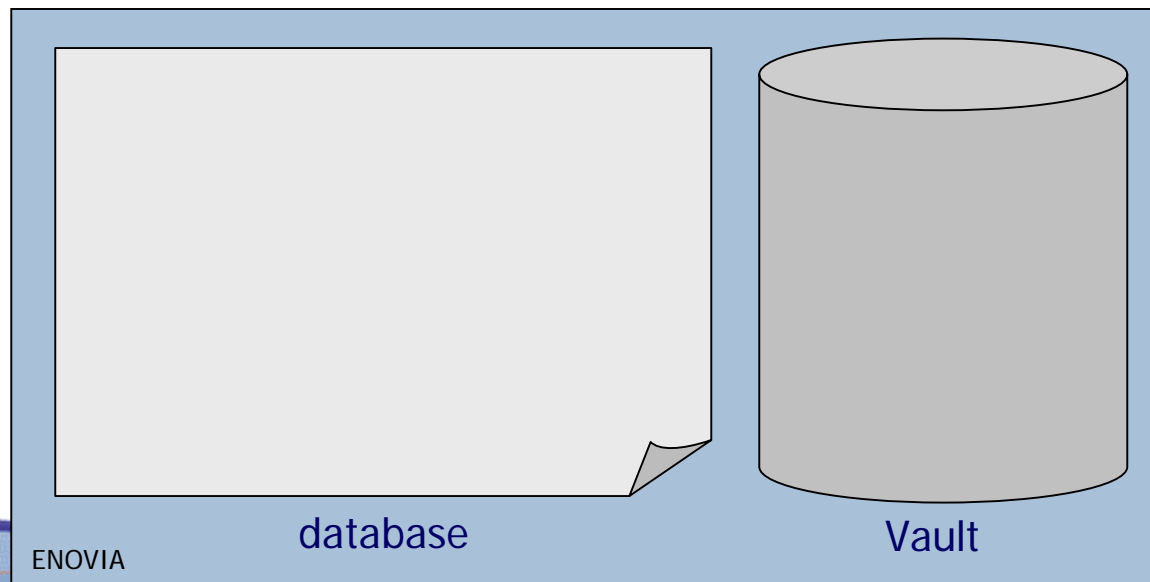
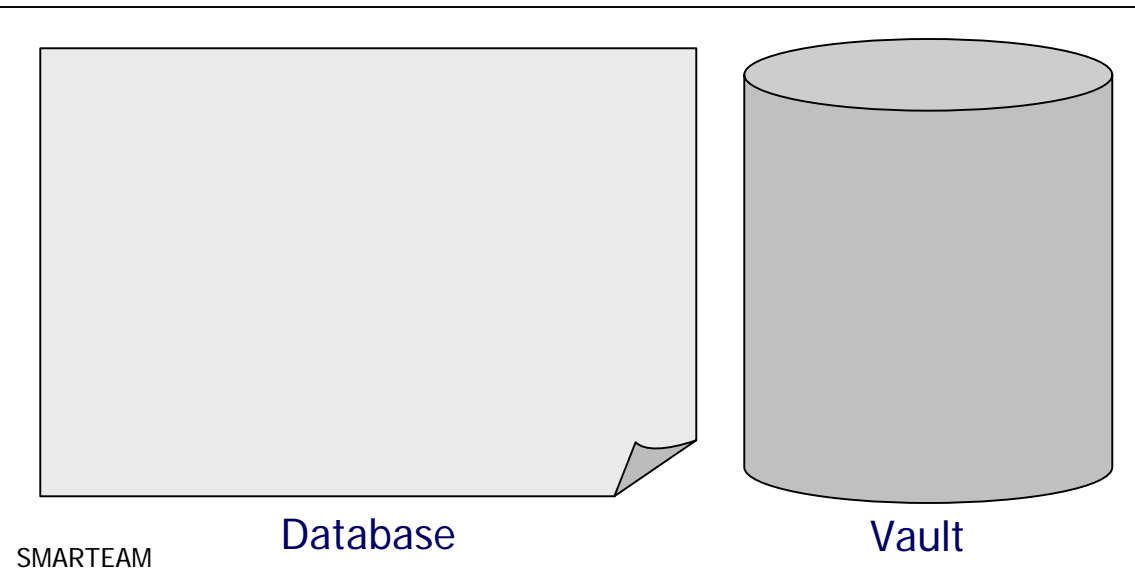
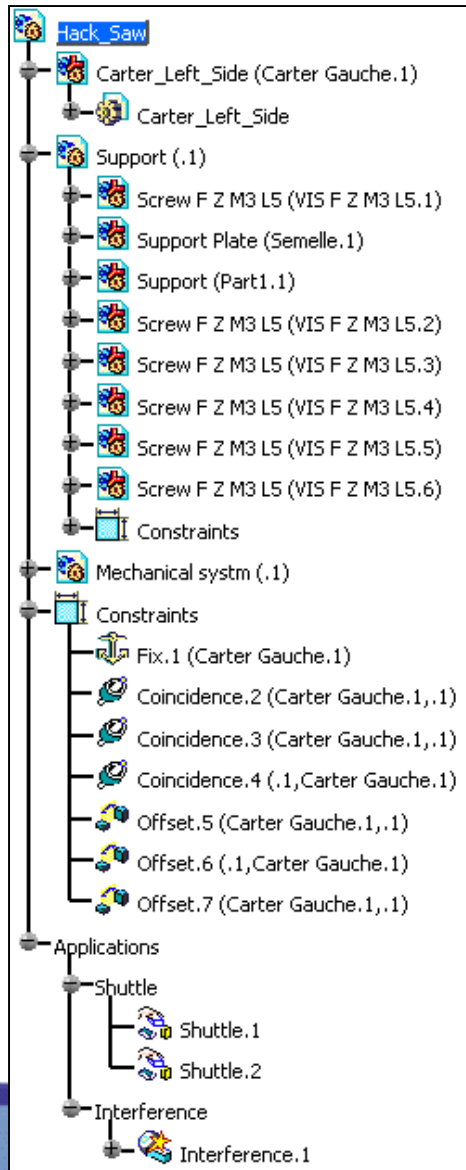
CATIA V5 PDM-Solutions: ENOVIA

■ Exploded Mode

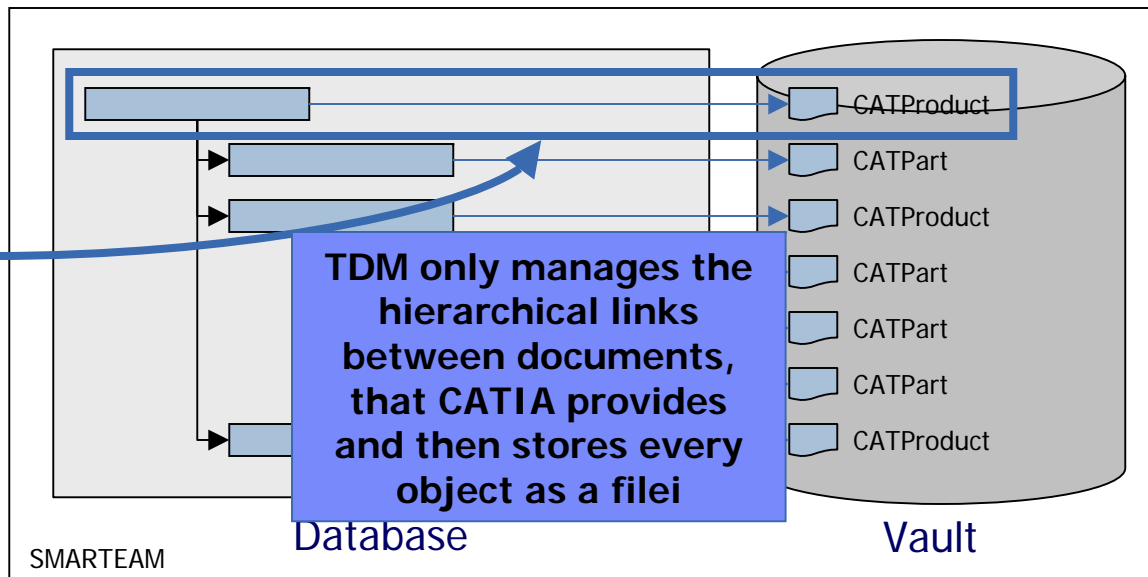
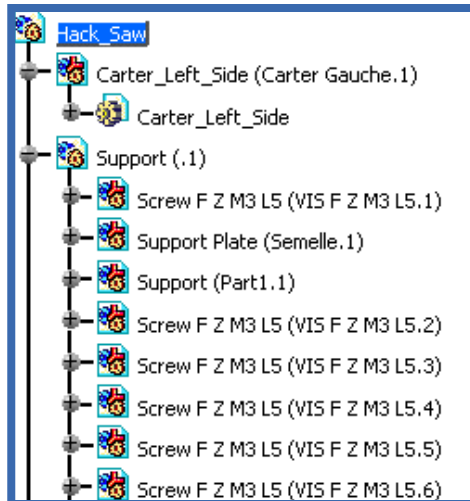
- All information contained in parts and assemblies (Relations, Position-Informationen, Clash-Informationen as well as Referenzen) are stored in the database and can be visualized.
- Consequently ALL information from within CATIA can be made available in the PDM System – including the positioning in the work space



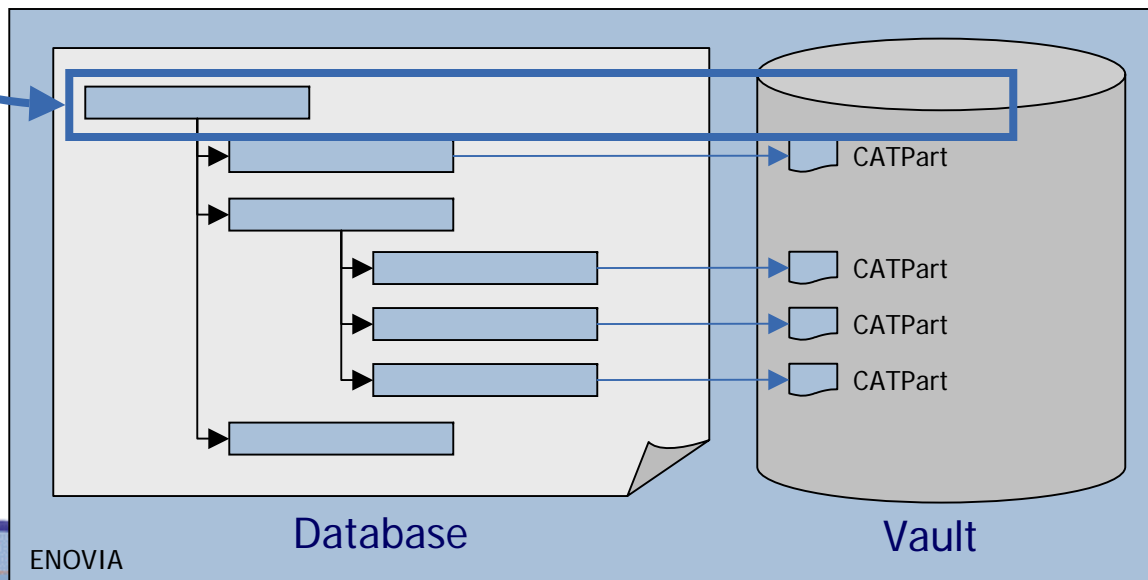
SMARTTEAM <> VPM/LCA: Model



SMARTTEAM <> VPM/LCA: DMS

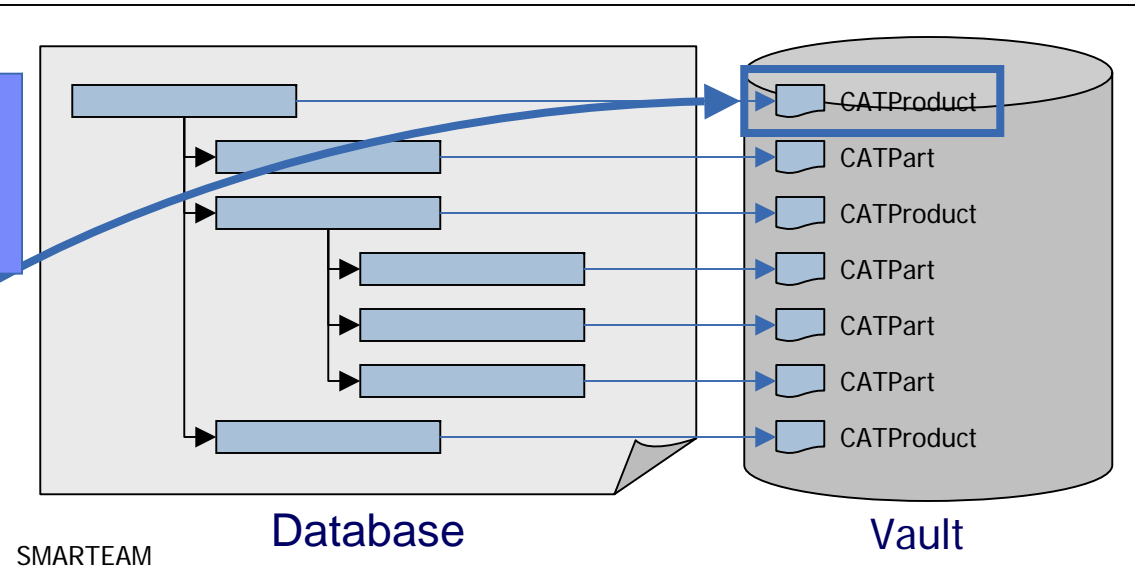


ENOVIA does not store the real assemblies, but keeps this information in the DB and rebuilds (dynamically) it



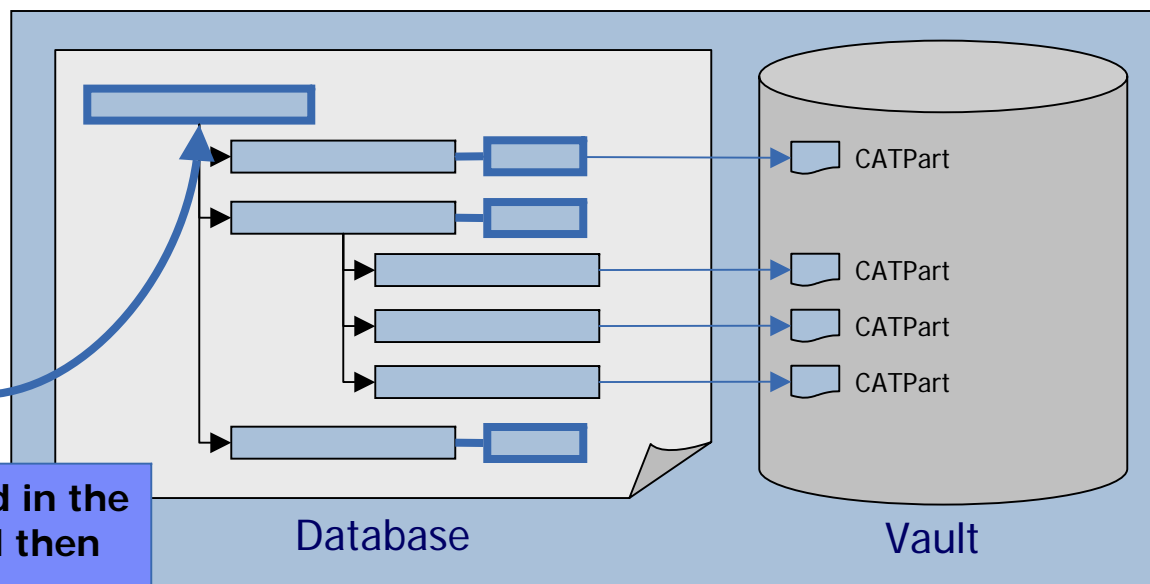
SMARTTEAM <> VPM/LCA: Constraints

The informations are kept in the CATProduct the DB does not have any access



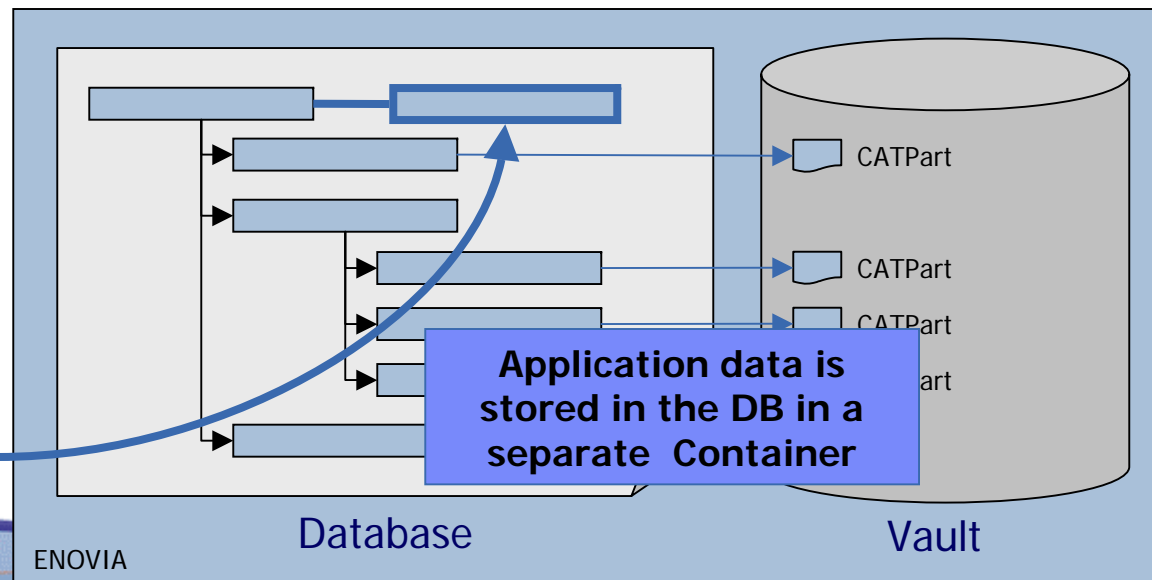
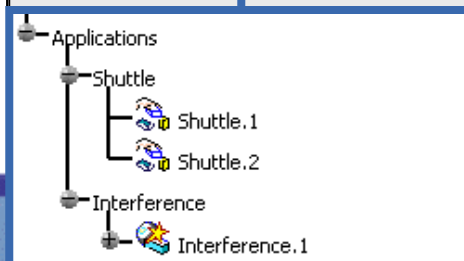
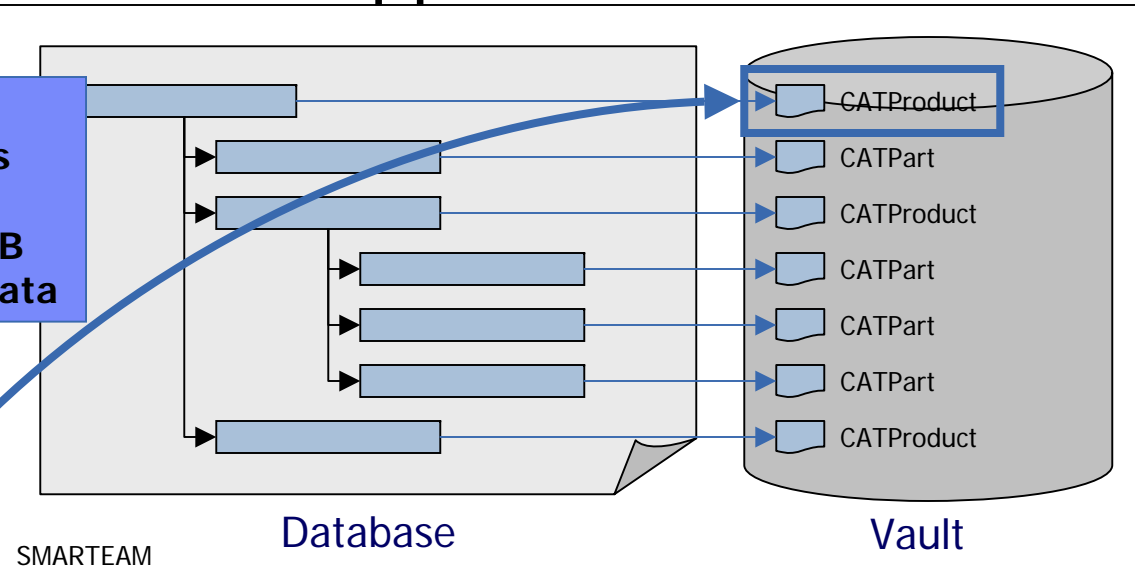
- Constraints
- Fix.1 (Carter Gauche.1)
 - Coincidence.2 (Carter Gauche.1,.1)
 - Coincidence.3 (Carter Gauche.1,.1)
 - Coincidence.4 (.1,Carter Gauche.1)
 - Offset.5 (Carter Gauche.1,.1)
 - Offset.6 (.1,Carter Gauche.1)
 - Offset.7 (Carter Gauche.1,.1)

Data are resolved in the ENOVIA DB and then visualized



SMARTTEAM <> VPM/LCA: ApplicationData

The information is stored in the CATProduct, the DB cannot access this data



Application data is stored in the DB in a separate Container

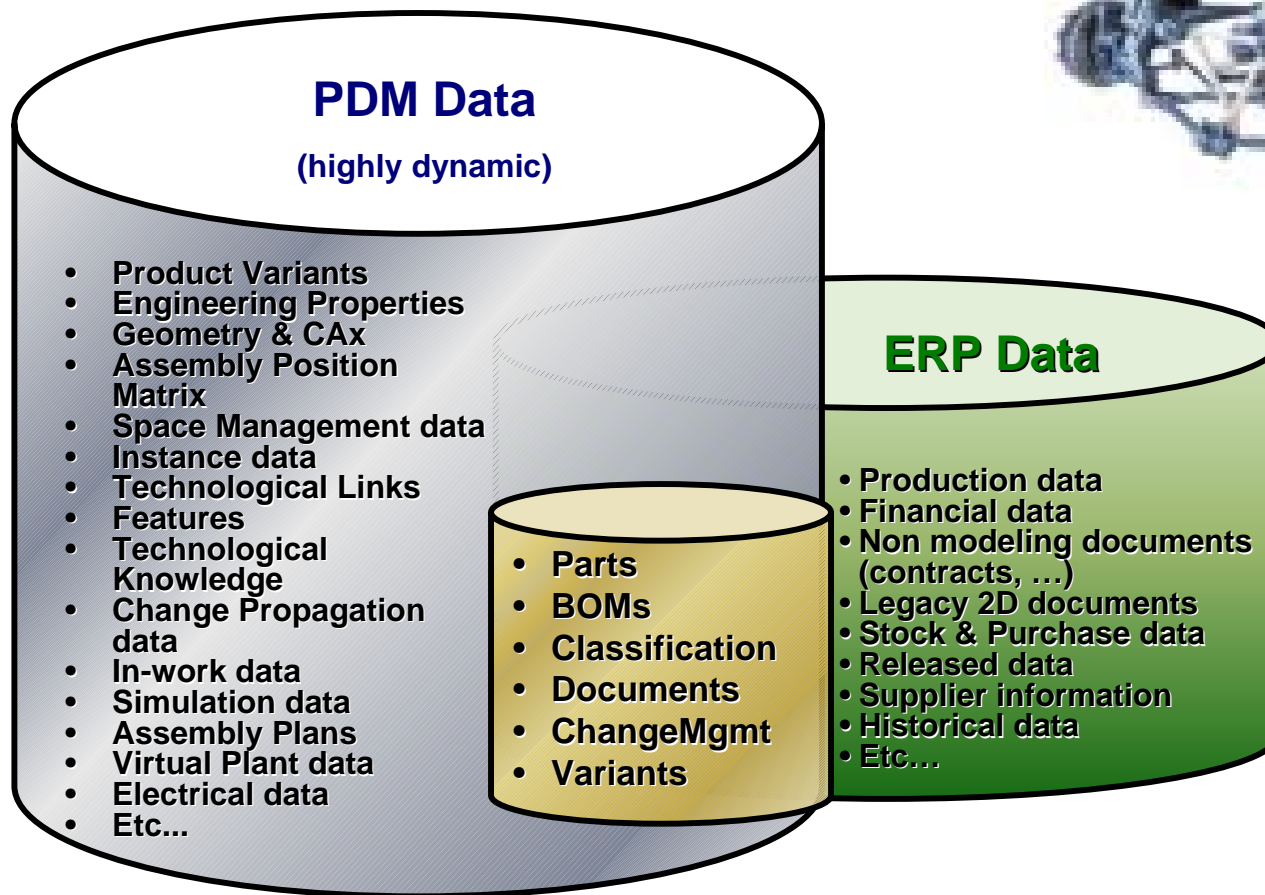
Summary

- **Direct interfaces** limit the usage of CATIA V5 considering ...
 - ... The effective product development by using parametric structures for
 - Variants
 - Engineering Changes
 - Object reuse
 - etc...
 - ... Concurrent Engineering
 - ... The management of ALL V5 data (FEM, DMU, etc)
 - ... The usage of DMU (Cache-Management)
- **SMARTTEAM**
 - requires product structure changes to be managed inside the CAD application
 - But enables management of all CATIA V5 data including their links
- **ENOVIA**
 - Allows changes to the prooduct structure without using CATIA. It can modify the internal CATIA Link structure
 - Can visualize geometric information such as constraints and application data inside PDM
 - Generates CATIA assemblies on the fly before loading them into CATIA
 - Only because of this true, configuration and clash is possible!

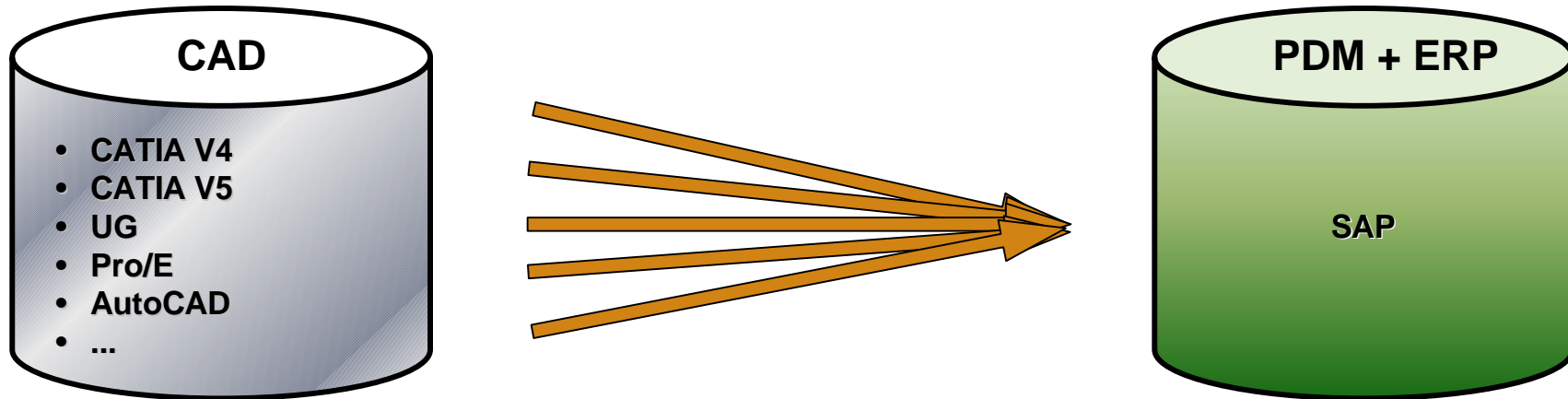
Agenda

- Why is CATIA V5 data management different?
- Requirements for CATIA V5 PDM systems
- Solution alternatives
- **Limitations of the CDI/SAP interface**
- Advantages by using SMARTEAM

Different Targets



Different approaches: Direct-Integration



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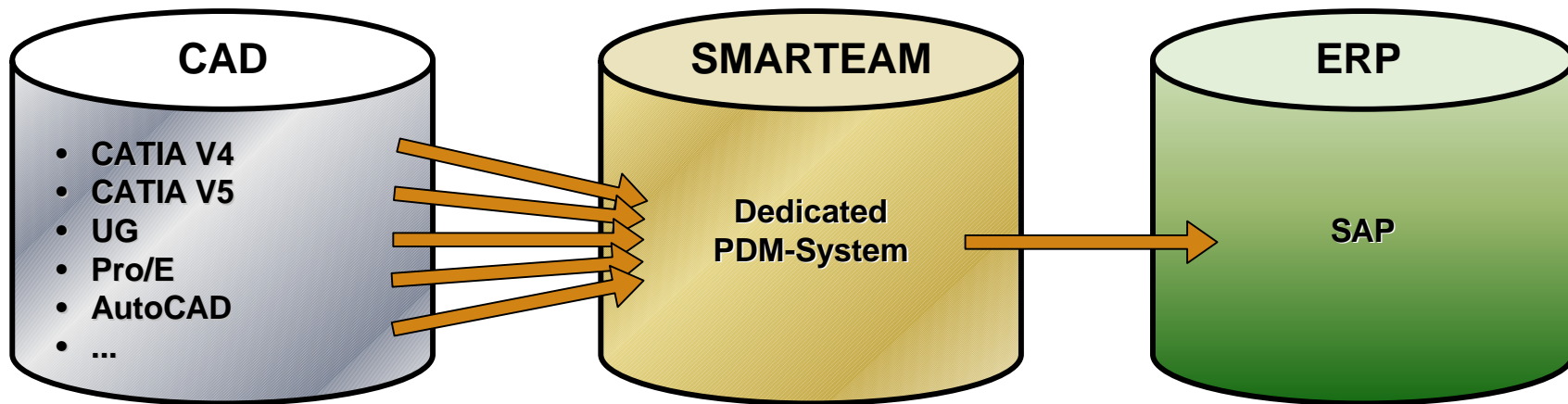
Advantages

- No additional interface between SAP and CAD necessary

Disadvantages

- Many interfaces directly into SAP increase customizing efforts
- Asynchronous Release-Management
- SAP has to catch up with CAD's dynamic evolution
- ALL data are saved into SAP (even prototypes, sketches etc.)
- Low influence, but high impact on SAP data model and CDI interface

Different approaches: TDM Layer



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Advantages

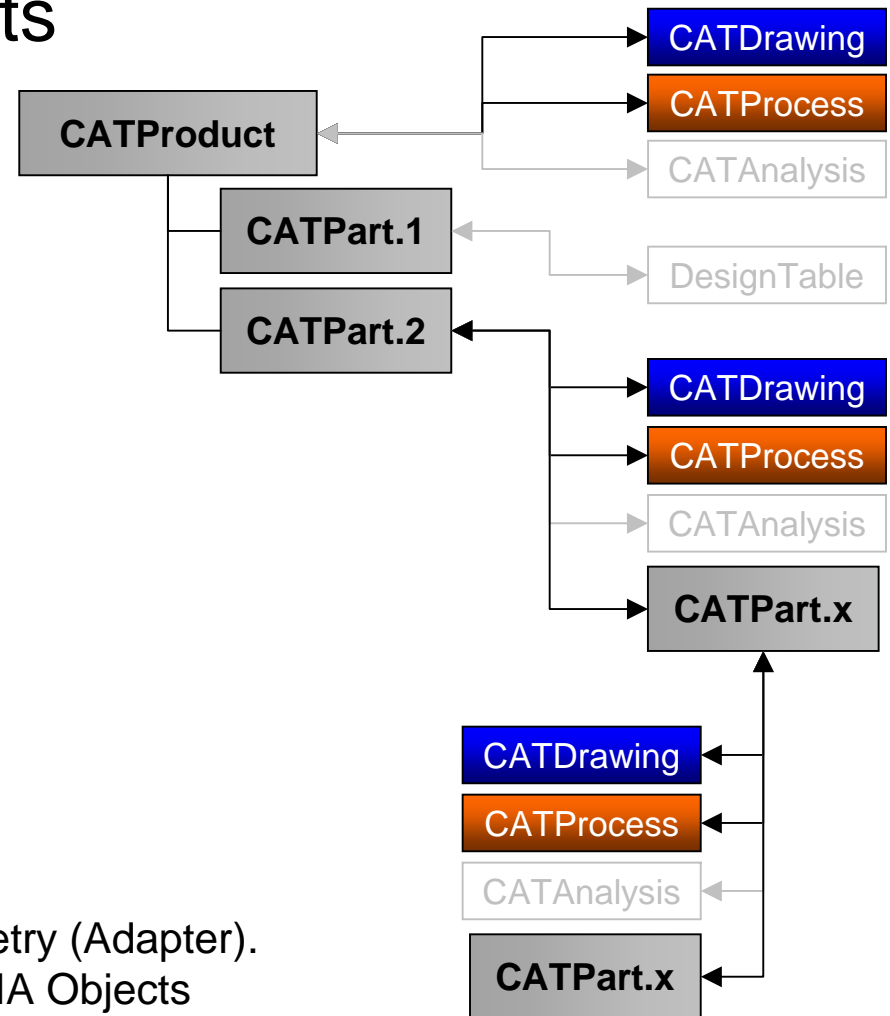
- Only one interface to SMARTTEAM decreases customizing in SAP
- Smart interfaces due to flexible data- and link model
- Only few efforts during release changes after the initial implementation
- PDM can be considered as a filter for the ERP System – only data relevant for production will be transferred

Disadvantages

- One additional interface – but which is more static. Dynamic CAD is caught by SMARTTEAM instead

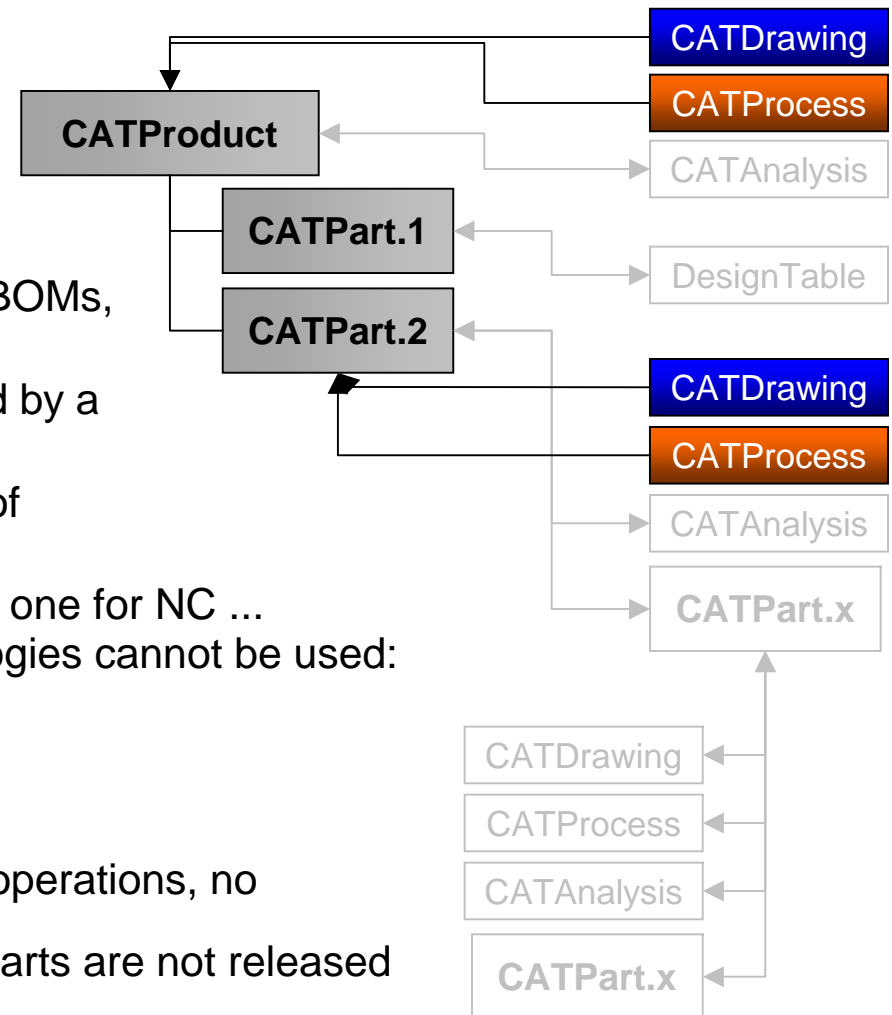
Support of CATIA V5 Objects

- Supported are
 - CATPart
 - CATProduct
 - CATDrawing
 - CATProcess
 - Components
- Not supported**
 - Design Tables
 - cgr
 - CATCatalog
 - CATAnalysis
- No differentiation for helping geometry (Adapter). Therefore, BOMs include ALL CATIA Objects



Support of CATIA V5 Links

- CDI and SAP only use one link type: Hierarchical BOMs, even for the connection between 2D and 3D:
 - Drawings and Process are heads of BOMs, 3D objects are their components
 - Drawings of 3D can only be accessed by a Where used query
 - This makes easy release processes of DRW including 3D hard to realize
 - Lots of parallel BOMs: One for DRW, one for NC ...
- Due to limited links, MML-based methodologies cannot be used:
 - Relational Design
 - Design in Context
 - ...
- Each object has to be marked for lifecycle operations, no „Propagate Operation“ – insecure!
- Assemblies can be released, even if Sub-Parts are not released
- No Design Copy



Abgrenzung der Integrationen im CAD-System

- CDI only interacts with the active window – examples:
 - Locate Active Document
 - Insertion of Parts and Products within CATIA is only possible by first loading an object in a new window
 - Creation of new objects requires the creation in a new window and an initial save in SAP
- No automatic synchronisation of the CAD structure – each document has to be saved manually
- Less information in the CATIA tree (never revision, access permissions)
- Releases of CATIA are less synchronized
- Changes without newer revisions are possible
- No Bottom-Up-View



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Arguments for SAP?

- „One single infrastructure“
 - Data model is not flexible! (No Adapters, Skeletons)
 - Lots of data and revisions within the engineering departments
 - ... Which are excessively linked together (on multiple levels)
- „Single user interface“
 - Not linked to engineers needs (Accessing the Viewer, Revision history, Where used list)
 - Representation is not flexible enough
 - SMARTTEAM as well is one single user interface for the engineers
 - SAP anyway requires a second application window to initiate PDM functionality
- „No PDM-interface necessary“
 - Any data (useful or not) will be saved in SAP (Adapters, Alternatives etc.)
 - Only ONE static and rather simple interface is necessary
 - But customizations on SAP are necessary in any case
- „All data is available for production planning at once“
 - Can be realized with SMARTTEAM as well (Reference: Siemens VDO)

Agenda

- Why is CATIA V5 data management different?
- Requirements for CATIA V5 PDM systems
- Solution alternatives
- Limitations of the CDI/SAP interface
- Advantages by using SMARTEAM

Ergonomic Issues

■ SMARTTEAM menus in CATIA V5 user interface

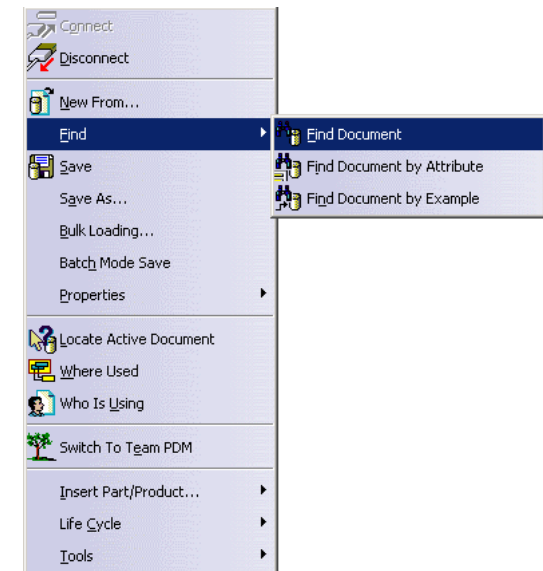
Dedicated Menu- and Icon-Bars within CATIA V5 provide most important functionality to work with SMARTTEAM:

- Query for Objects
- Save Object to DB
- Check In / **Check Out** / Release
- **Locate Object in SMARTTEAM**
- **Insert Object from SMARTTEAM**
- **Replace Revision**
- **Replace Componente**
- **Where Used**
- **Query owner of Object**

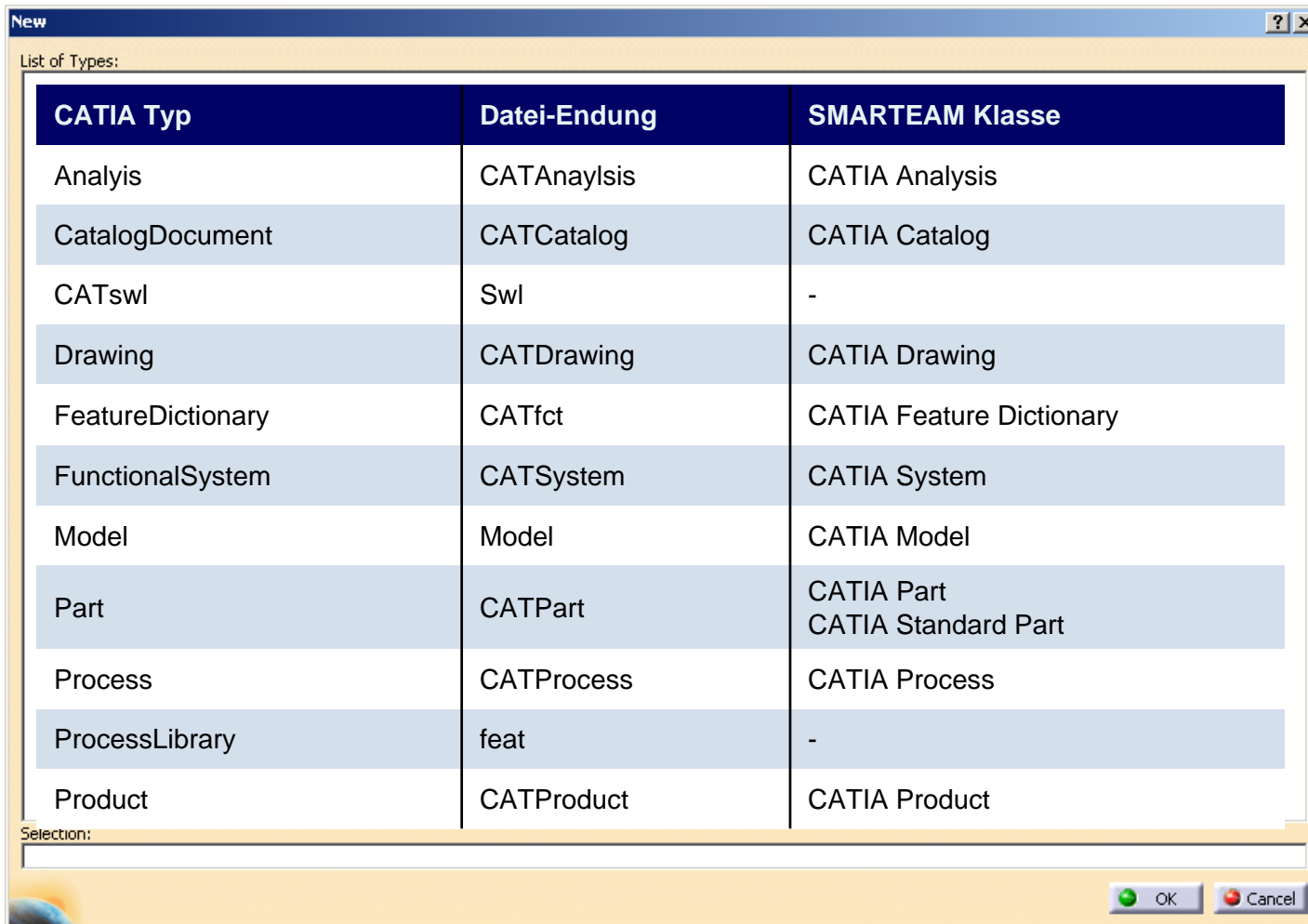
Engineers are using SMARTTEAM functionality directly from CATIA. SMARTTEAM UI itself is not necessary. That way, engineers can work most efficient, the PDM overhead is reduced to a minimum.

■ PDM-Status in CATIA V5 Structure

- Access Permissions
- Actuality
- ...



CATIA File-Types (default)











The screenshot shows the 'New' dialog box in CATIA. It has a title bar with 'New' and standard window controls. Below the title bar is a label 'List of Types:' followed by a table. The table has three columns: 'CATIA Typ', 'Datei-Endung', and 'SMARTEAM Klasse'. The rows list various file types including Analysis, CatalogDocument, CATswl, Drawing, FeatureDictionary, FunctionalSystem, Model, Part, Process, ProcessLibrary, and Product. At the bottom of the dialog is a 'Selection:' label and a text input field. The bottom right corner contains 'OK' and 'Cancel' buttons.

CATIA Typ	Datei-Endung	SMARTEAM Klasse
Analysis	CATAnalysis	CATIA Analysis
CatalogDocument	CATCatalog	CATIA Catalog
CATswl	Swl	-
Drawing	CATDrawing	CATIA Drawing
FeatureDictionary	CATfct	CATIA Feature Dictionary
FunctionalSystem	CATSystem	CATIA System
Model	Model	CATIA Model
Part	CATPart	CATIA Part CATIA Standard Part
Process	CATProcess	CATIA Process
ProcessLibrary	feat	-
Product	CATProduct	CATIA Product

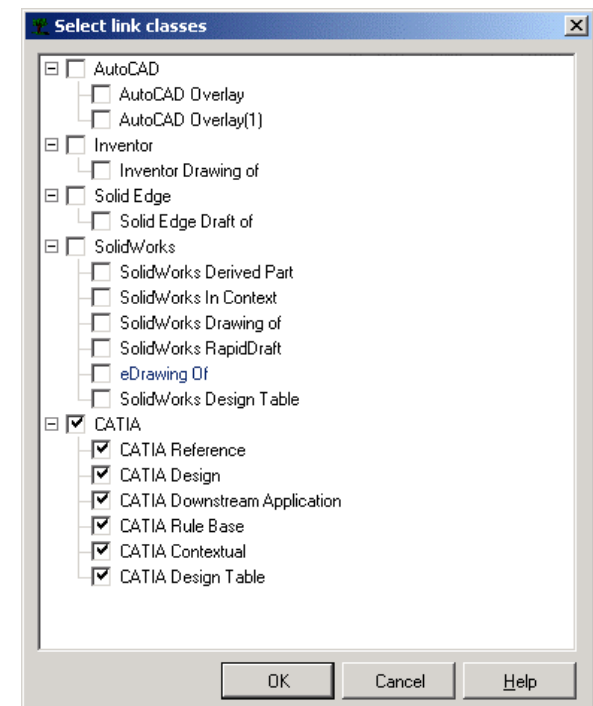
Verwaltung der V5-Links

SMARTTEAM

-  **Product Structure Link**
Assembly links representing the Product Structure
CATProduct > CATProduct, CATPart
-  **Design Link**
= Reference Links (CCP + Context): CATPart > CATPart
-  **Downstream Application Link**
Link between downstream application and geometrical data
CATDrawing <> CATProduct, CATPart
-  **Contextual Link**
Link between Part containing Reference Links and its context Product:
CATPart <> CATProduct
-  **Design Table Link**
Link between Geometry and Excel or text file containing parameters:
CATProduct, CATPart <> xls, txt
-  **Result Link**
CATIA output links: CATProcess <> aptsource, CATNCCode
-  **Rule Base Link**
Link between an instance of a rule base and its reference
-  **Reference Link**
All other links: CATMaterial <> CATProduct, CATPart

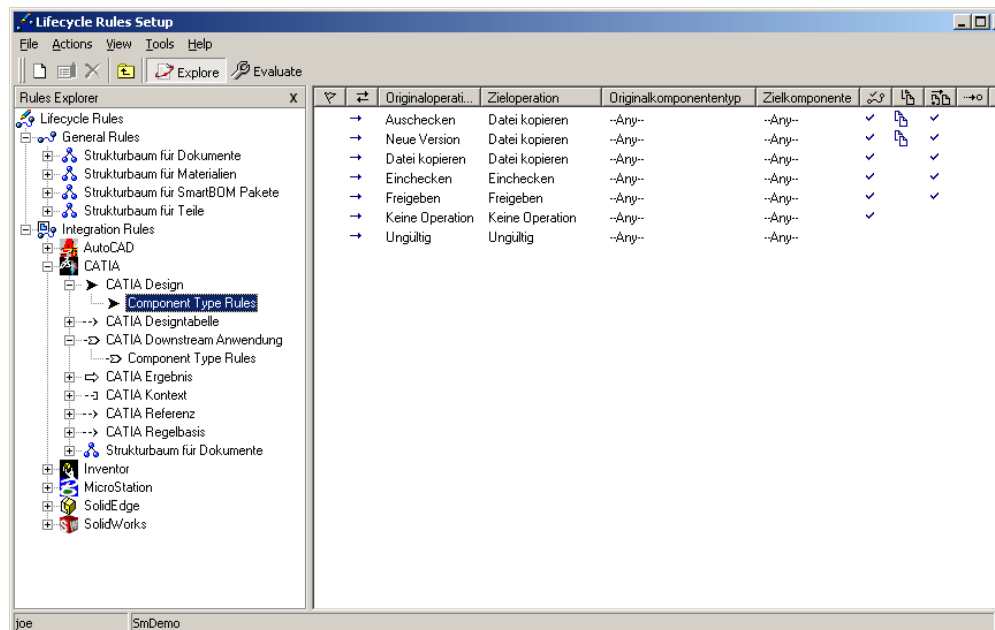
Represents information which is not available in CATIA and enables logical connectors (e.g. for Drawings and Geometry)

icon	V5R11 LINKS
	CATIA Product Link (P)
➤	CATIA Design Link (D)
-->	CATIA Rule Base Link (RUL)
-->	CATIA Design table Link (DT)
-D	CATIA Downstream Application Link (DA)
-->	CATIA Reference Link (REF)
-C	CATIA Contextual Link (C)
-->	CATIA Result Link (RES)
	CATIA Is Composed Of (IS)



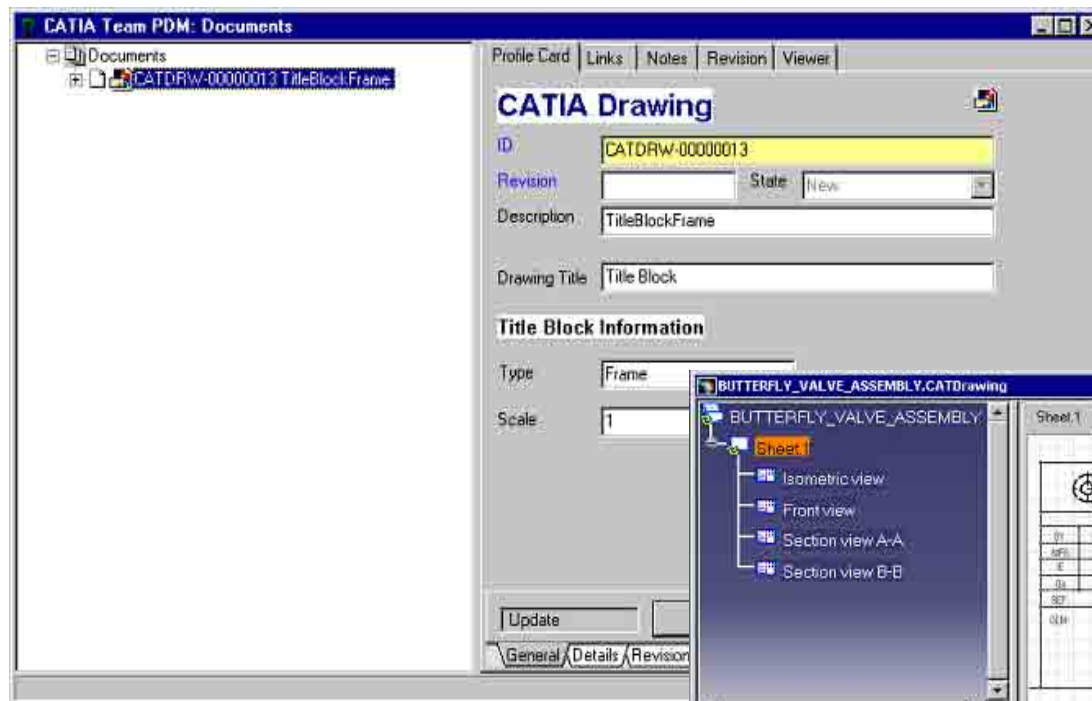
Intelligence of Links

- Objects can be considered as unit. For example, Drawings, Geometry, NC and FEM belonging together can be versioned in one step
- Dependencies are taken into account during lifecycle operations
- ... In both directions if necessary
- Such behaviour can easily be customized by using the **Lifecycle Rules Setup**

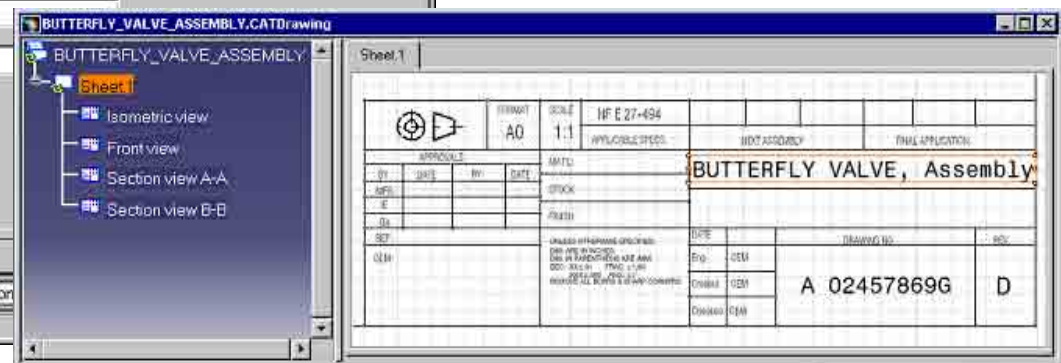


Flexible Property Mapping

- Exchange information between CATIA and SMARTEAM:
 - Save CATIA V5 Properties in SMARTEAM,
 - Fill CATIA V5 Properties by attributes coming from SMARTEAM
- Example: **Titleblock**, **Material**

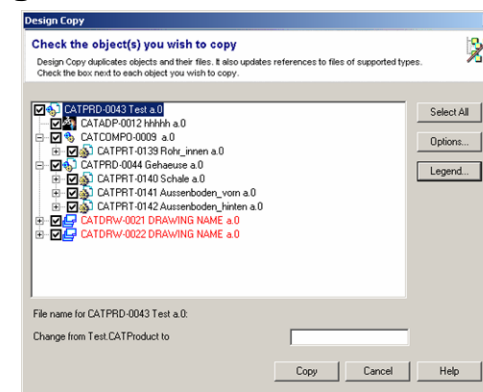


Dieses Mapping ist bspw.
auch bei Office
Dokumenten möglich!

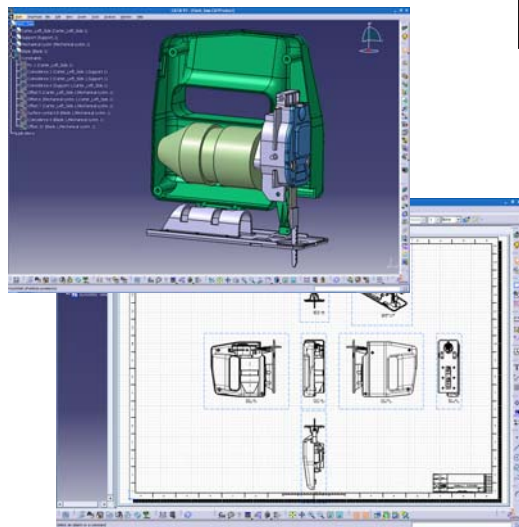


Other Advantages

- Own classes for Standard Parts, Built Parts and Skeletons can be realized easily due to flexible data model and customization within CATIA. This enables
 - **Special numbering schemas**
 - **Individual sets of attributes and icons**
 - **Dedicated access permissions**
 - **And dedicated treatment for the Integration in an ERP system**for those classes
- 1:1 Visualization of structures and their links in CATIA and SMARTEAM
 - **Icons for different object types (CATProduct, CATPart, CATDrawing)**
 - **Automatic synchronization of structures with a simple mouse click**
 - **Even for linked objects (when saving the Drawing, the CATProduct and its CATParts will be saved)**
- Easy support for Design Alternatives
 - **Design Copy functionality even considers the Links if necessary**
- **Cache Management**



The comparison of real scenarios show the difference



Scenario: Ergonomic aspects

Scenario: Handling of assemblies

Scenario: Store structures

Scenario: Concurrent engineering

Scenario: Adapter methodology

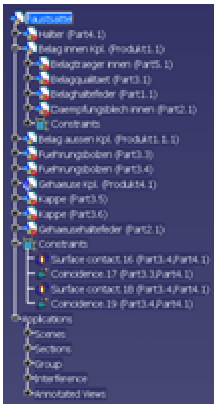
Scenario: Data type handling

Scenario: link type handling

Eberspächer Scenario: Concurrent Engineering

SMARTEAM

- The Adapter-Methodology can be used to support Concurrent Engineering
- Status-Icons in CATIA show the actuality of the objects (was a loaded object developed further in parallel?)
- Checkout on the fly, to quickly and fast put objects in work – directly from within CATIA for the actual, selected model
- Automatic Versioning of changes from within the CAD system help to synchronize teamwork
- Versions can be changed if needed but also automatically
- „Shared Workspaces“ allow working in a small project team outside of PDM



CDI

- No specific support for Concurrent Engineering within the CAD system
- Information about Status, access rights only from the CAD desktop, not from within CATOIA V5
- Versioning (change of an object by a newer version) is very difficult

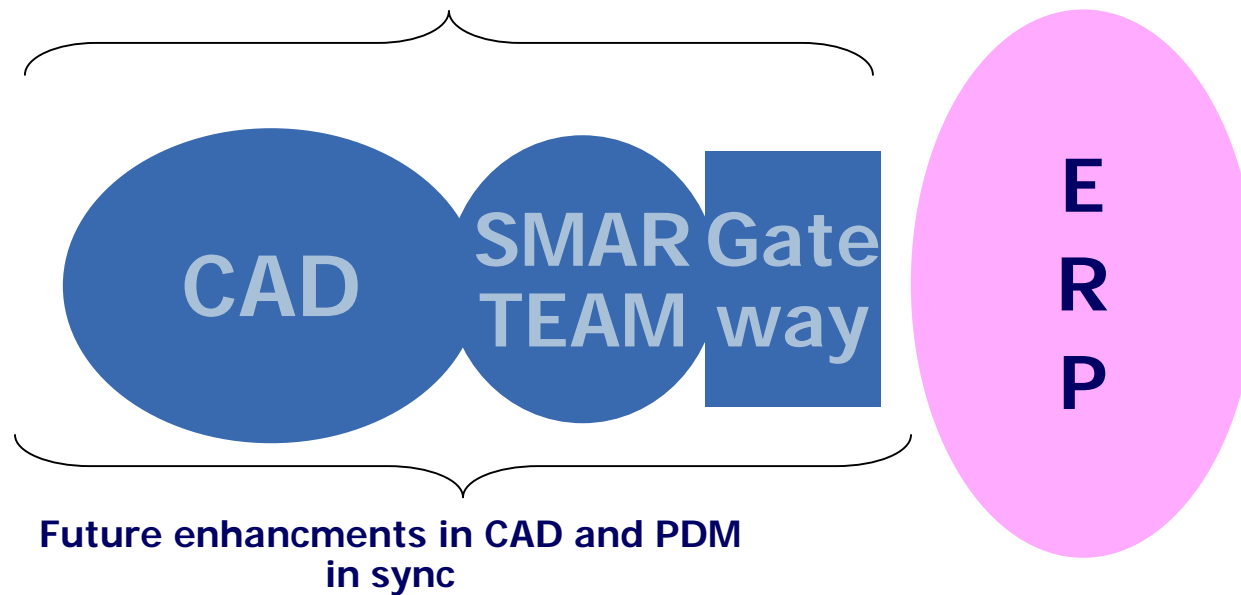
Team work requires a flexible PDM-System, since access rights and actuality of the objects is changing

The associative V5 methodology of concurrent engineering is supported by SMARTEAM.

Therefore development times can be reduced!

One ISV

Development from one partner with one technology: V5



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Development
dynamics