Training document

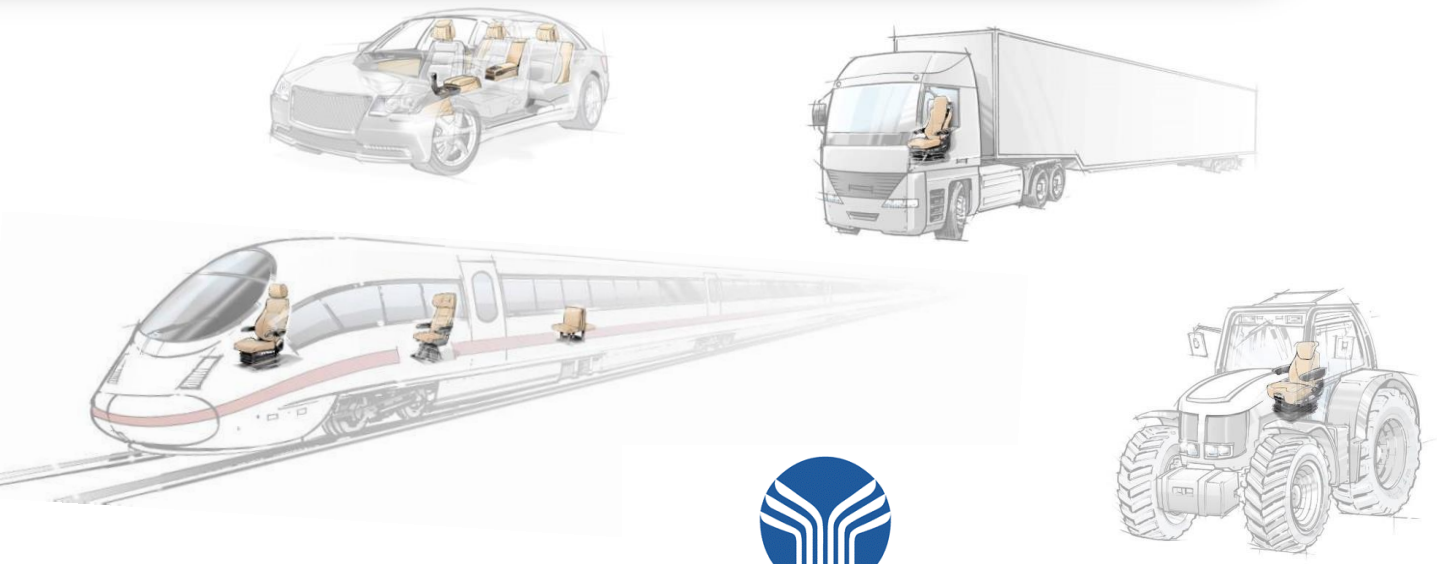
015 Research + development

SmarTeam Training Document

T_015_004

Training document

SmarTeam



GRAMMER

Table of Contents

1 General information ... 6
1.1 What is SmarTeam? ... 6
1.2 What is a PDM system? ... 6
2 SmarTeam at GRAMMER ... 7
2.1 Sites ... 7
2.2 SmarTeam Start ... 7
2.2.1 Requirements ... 7
2.2.2 Launch SmarTeam ... 8
2.2.3 SMARTBOX ... 8
3 Project tree and folder structure ... 9
3.1 Automotive project tree ... 10
3.2 Seating Systems project tree -> Driver Seats ... 11
3.3 Seating Systems project tree -> Passenger Seats ... 11
3.4 Global data replication ... 12
3.5 General folder structure ... 13
3.6 Automotive folder structure ... 14
3.7 Seating folder structure ... 14
4 Search ... 15
4.1 Standard search ... 15
4.2 Save own searches ... 17
4.3 More search icons on the search bar ... 19
5 Icons, Admin Console, language, windows ... 20
5.1 Icons ... 20
5.1.1 SmarTeam status: ... 20
5.1.2 SmarTeam documents: ... 20
5.1.3 Catia V5 ... 21
5.1.4 NX ... 21
5.2 Admin Console ... 22
5.2.1 Change password ... 22
5.3 Language ... 24
5.4 Window ... 25
5.4.1 Arrange windows ... 26
5.4.2 Navigate between windows ... 26
6 Profile Card ... 27
6.1 General ... 27
6.2 Additional Info ... 28
6.3 Details ... 29
6.4 Revision ... 30
6.5 Site Info ... 30
6.6 OEM ... 31
6.7 Populating a Profile Card ... 32
6.7.1 Title GR/Title EN ... 32
6.7.2 Additional GR ... 34
6.7.3 Material ... 34

6.7.4 Standardization ... 36
7 Viewer in SmartTeam for CAD data ... 37
7.1 Catia 3D Viewer ... 37
7.1.1 Functions of the Catia 3D Viewer ... 37
7.1.2 Commands using mouse buttons ... 38
7.1.3 Toolbar with commands ... 38
7.2 Catia 2D Viewer ... 41
7.2.1 Functions of the Catia 2D Viewer ... 41
7.2.2 Catia 2D Viewer Toolbar with commands ... 42
7.3 NX 3D Viewer ... 44
7.3.1 Functions of the NX 3D Viewer ... 44
7.3.2 Commands using mouse buttons ... 45
7.3.3 Toolbar with commands ... 45
7.4 NX 2D Viewer ... 47
7.4.1 Difference between 2D NX Viewer and 2D Catia Viewer ... 47
8 Links ... 48
8.1 Business Units ... 48
8.2 DEX ... 48
8.3 CATIA links ... 48
8.4 Workflow ... 49
8.5 Where Used ... 49
8.6 Composed Of ... 50
8.7 General Info ... 50
9 File Operation ... 51
9.1 Open For ... 51
9.1.1 Open as read-only ... 52
9.1.2 Open for edit ... 53
10 Open Views ... 56
10.1 Top-Down Tree ... 57
10.2 Bottom-Up Tree ... 58
10.3 Versions View ... 58
10.4 Navigate between views ... 59
10.5 Customising views ... 59
11 Bills of Materials (BOM) ... 60
11.1 EBOM Engineering Bill of Materials ... 60
11.2 BOM comparison ... 62
12 Design Copy ... 63
13 Workflows ... 66
13.1 Smartbox ... 66
13.2 Initiating a workflow ... 67
13.3 Pre-production workflow ... 68
13.4 Release workflow ... 69
13.5 Modification workflow ... 70
13.6 Bagatell process ... 70
13.7 V-number (trial numbers) workflow ... 71
13.8 TIF release ... 71

13.9 SAP interface 72
13.10 Workflows with particular release points (welding, bonding) 73
13.11 History for frame colour during a workflow 74
14 DEX 75
14.1 What is DEX? 75
14.2 DEX in SmarTeam 75
14.3 DEX Export 77
14.4 DEX Import 79
14.5 Creating start models 81
15 Other applicable documents 82
16 Amendments 82

1 General information

1.1 What is SmarTeam?

SmarTeam is a PDM/PLM system developed by Dassault Systèmes. It can be integrated into many CAD systems supplied by leading CAD providers such as Catia V5. Native integration is available for integration into CATIA and SolidWorks, as these systems are also produced by Dassault Systèmes . In addition to classic drawing management and bills of material structures, SmarTeam offers comprehensive document management with flexible workflow management. Among other things, it can also be integrated into MS Office. SmarTeam can also automate and standardise data exchange with Production, Sales or Service. SmarTeam provides interfaces with ERP systems such as SAP for this purpose. The user interface was revised for release 18.

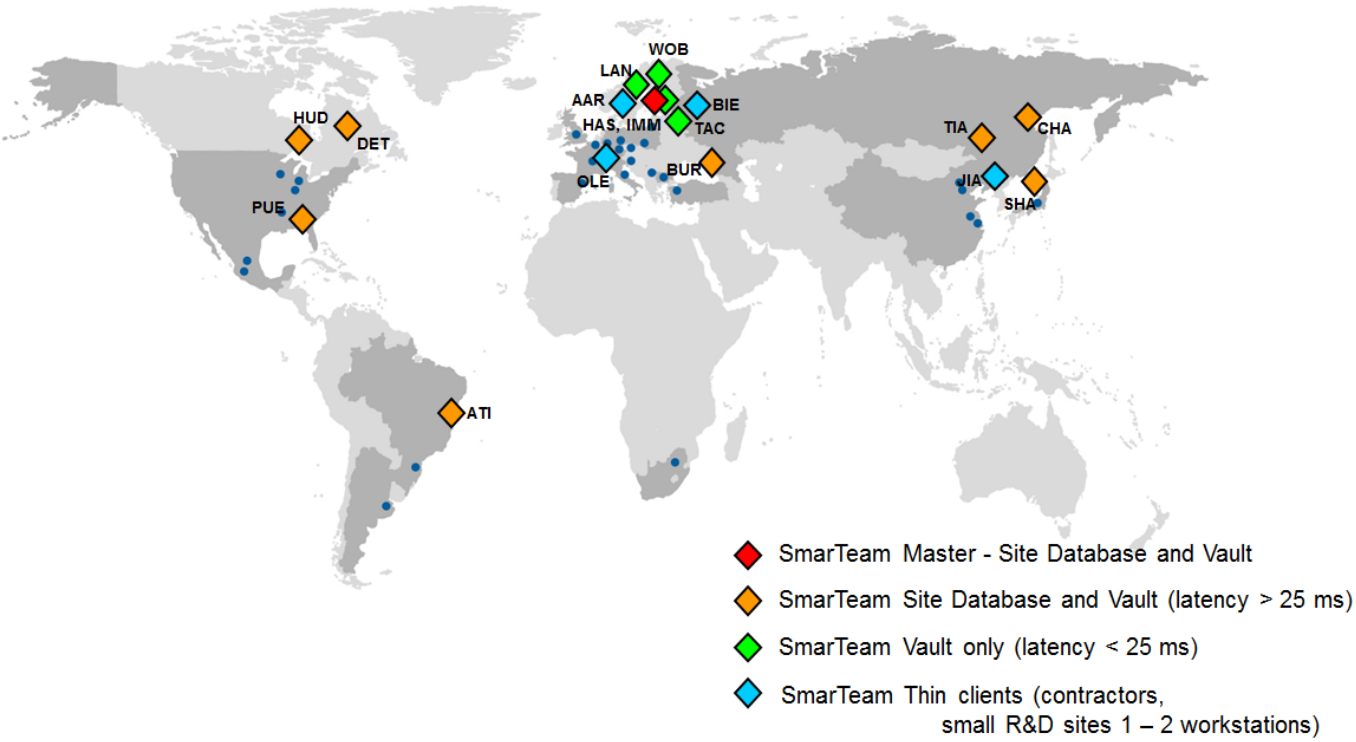
1.2 What is a PDM system?

Product Data Management (PDM) stores and manages data resulting from product development, which defines, represents and presents products, and makes it available in downstream phases of the product life cycle.
The introduction of a PDM solution serves to improve the quality of product development and to reduce product development time and costs. With the goal of ensuring end-to-end information flow, these benefits are passed on downstream to organisational units that are involved in the product life cycle. This method of working ensures seamless reproducibility of the configurations of a product.

2 SmarTeam at GRAMMER

2.1 Sites

All GRAMMER AG's development sites have access to the SmarTeam PDM system. It is also possible for development partners and sales offices to access the system via a thin client connection.
The master database is located in the Haselmühl Data Centre. All data created in SmarTeam is copied to the sites via n:n communication and is backed up in the central database in Haselmühl.



2.2 SmarTeam Start

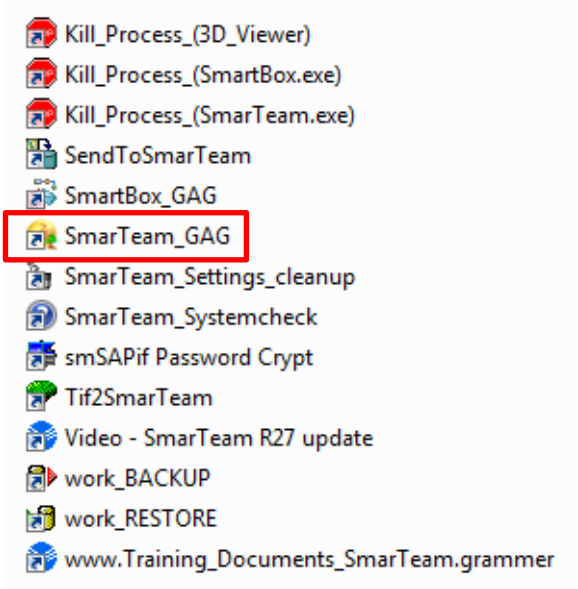
2.2.1 Requirements

To launch SmarTeam, it must be installed on the user's computer. The client must also be in the GRAMMER network at a Vault location. If you are not in the GRAMMER network or at a Vault location, you will require a thin client connection using Citrix technology.

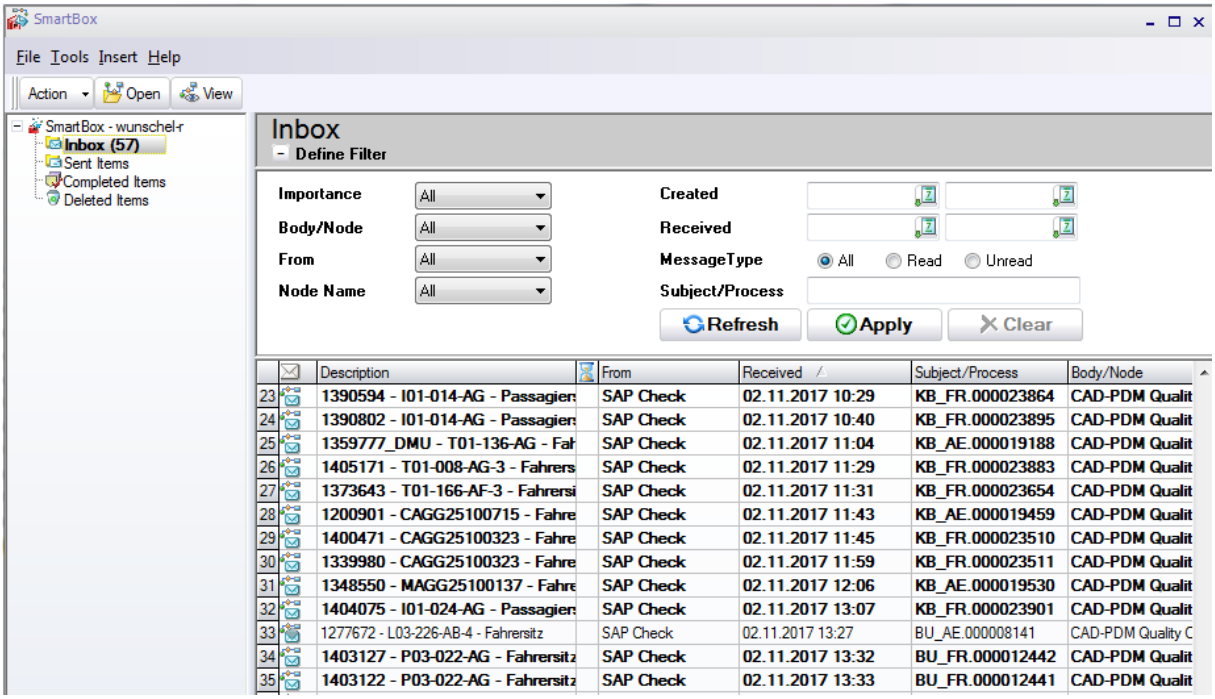
If SmarTeam installation is required or if you have any questions, please contact CAD Service:
GR-CAD@grammer.com

2.2.2 Launch SmarTeam

SmarTeam is launched from the CAD Start menu. This menu can be found on the desktop, in the Windows Start menu or on the taskbar. Open the CAD Start menu and, in the **PDM_SmarTeam** folder, double-click the **SmarTeam_GAG** start command to launch SmarTeam. A script then runs that copies the required latest software, sets various user- and location-specific settings and launches SmarTeam automatically.

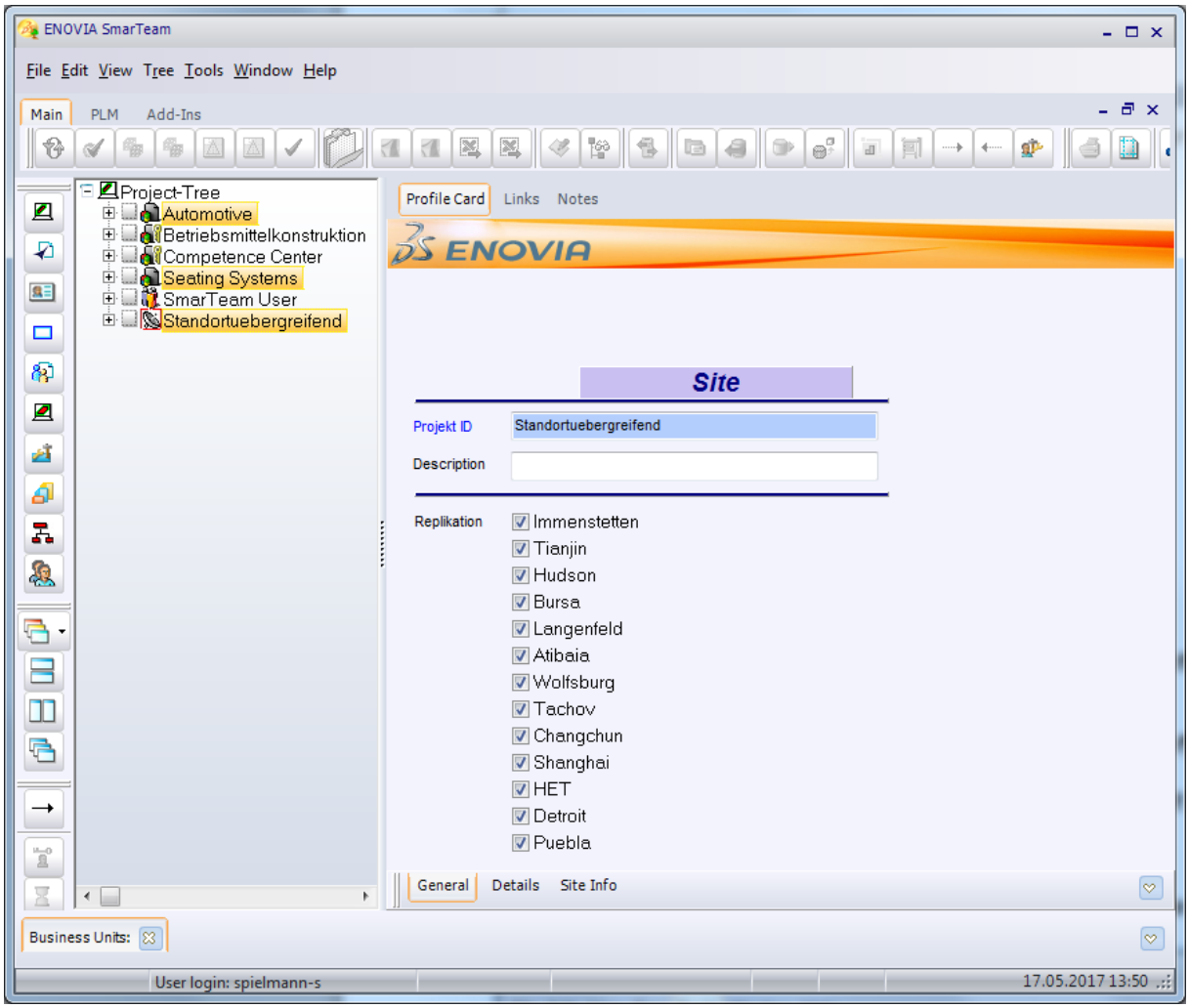


2.2.3 SMARTBOX



3 Project tree and folder structure

Once SmarTeam has been launched, the main window and the project tree appear immediately.



There are three main work areas:

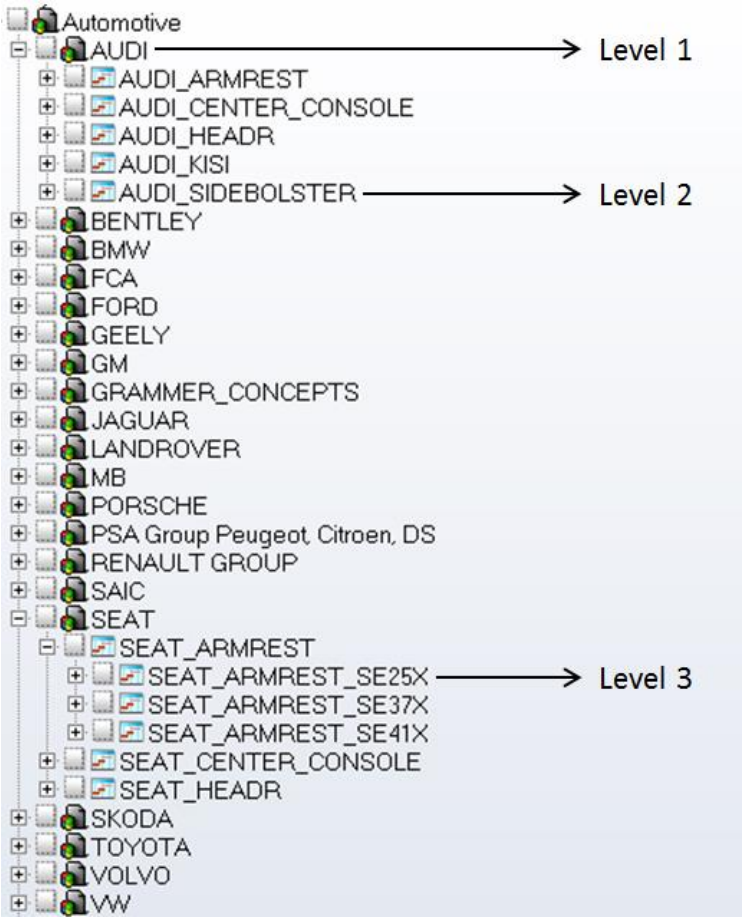
- **Automotive** → all automotive projects
- **Seating Systems** → all seating projects
- **Company-wide** → all standard parts, accessible worldwide, catalogues, etc.

Other areas:

- Equipment design → for toolmaking in Haselmühl
- Competence Center → currently under construction
- SmarTeam User → for testing purposes only, there must be <u>no</u> production data here.

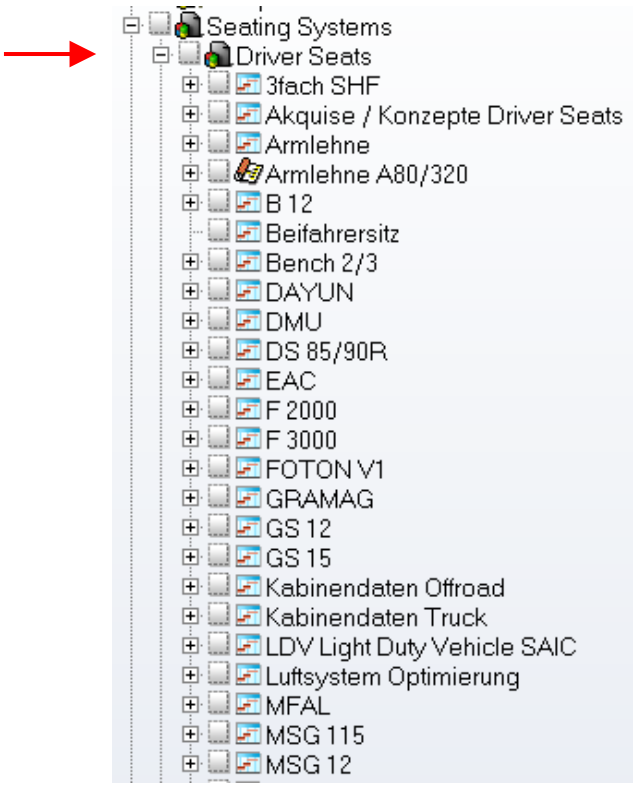
3.1 Automotive project tree

The tree is structured according to customer (Level 1), then product area (Level 2) and then relevant vehicle (Level 3).
If there are several vehicle derivatives, they are identified by an “X”.



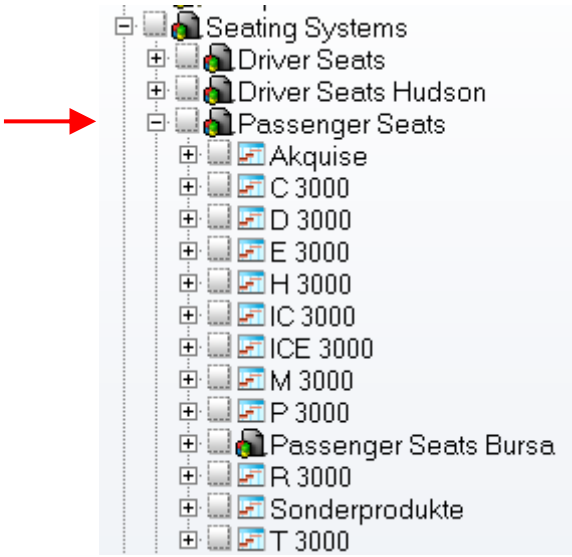
3.2 Seating Systems project tree -> Driver Seats

Projects in the “Driver Seats” section are structured according to seat type and project.



3.3 Seating Systems project tree -> Passenger Seats

Projects in the “Passenger Seats” section are structured according to seat version and project.

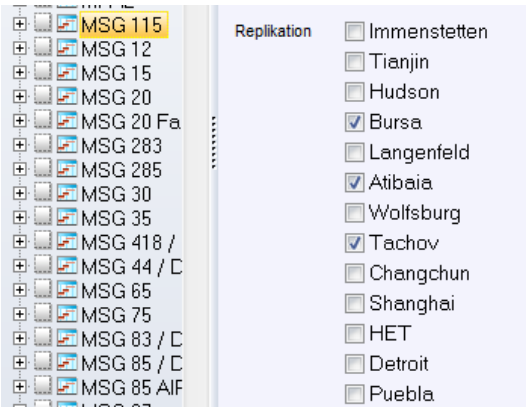


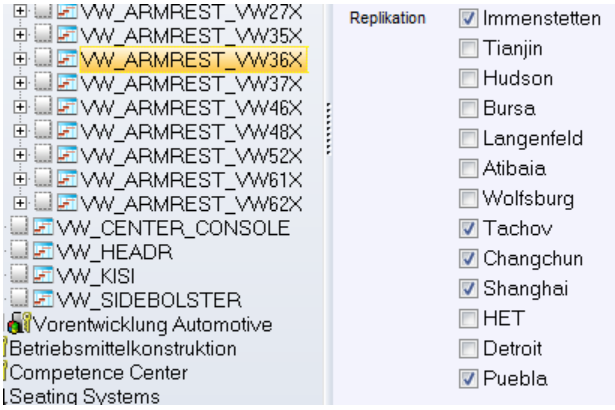
3.4 Global data replication

Global data synchronisation (replication) takes place exclusively via project links. These links must be set correctly when data is created.

See example for

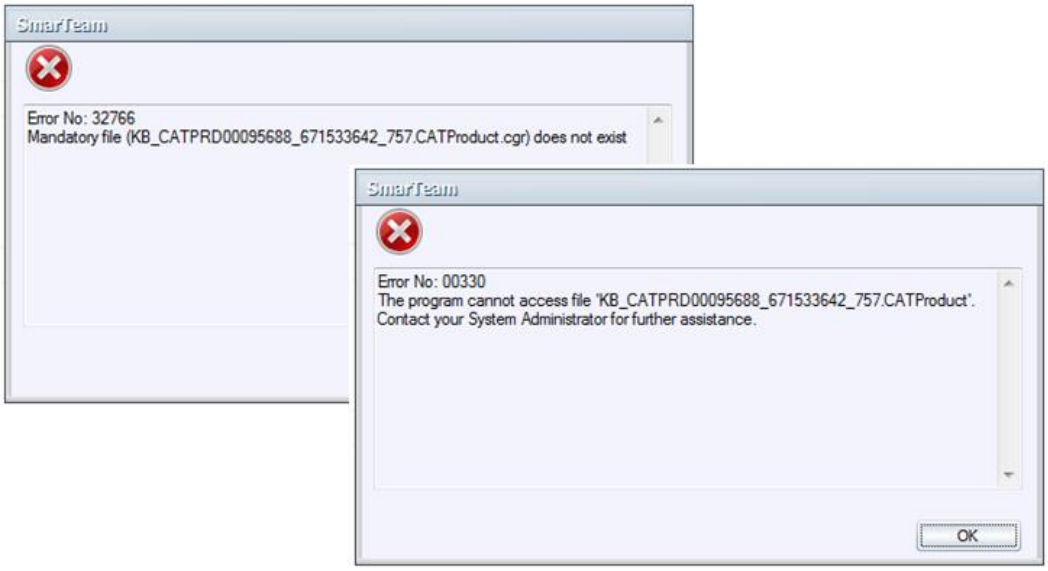
MSG115 Seat and VW Jetta armrest





Note:

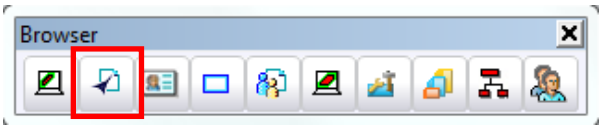
- Project links must be correct when data is initially created
- In the case of a design copy, data must also be individually and correctly linked
- In the case of Carry Over Part (COP) projects, data must also be correctly re-linked
- In addition to the project, all standards parts must also be linked to “Standard parts”
- More information about this can be found under the heading “Links”
- If data is not available due to missing project and site links, the following error message appears in the viewer:



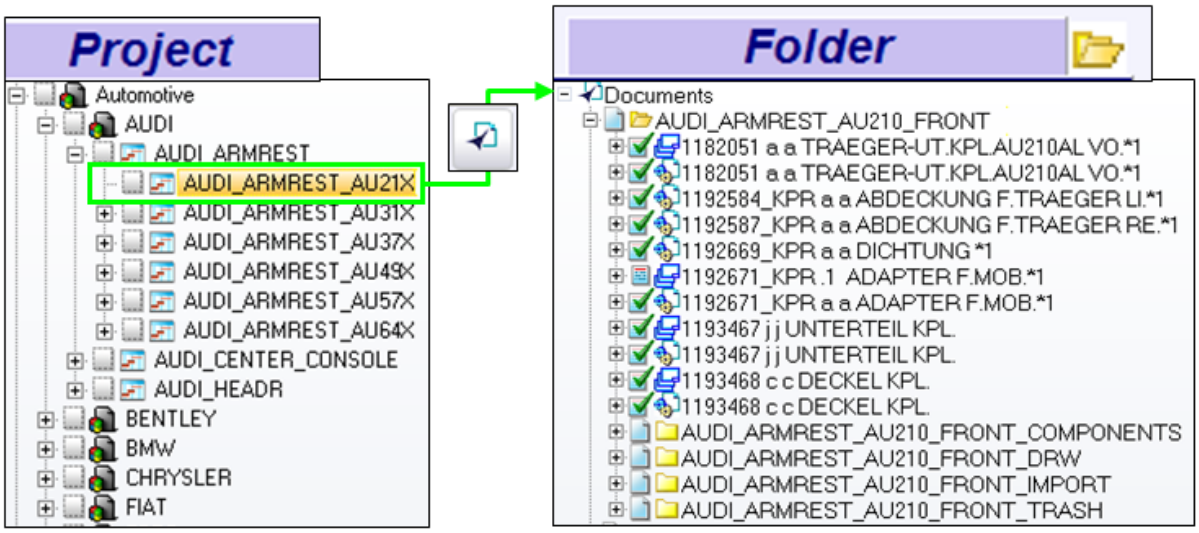
3.5 General folder structure

Switching from the Project structure to the Folder structure is only possible via the “Documents” command. This command is located on the “Browser” toolbar.

On the left side of the ST tree, select the project (must be highlighted in orange) and execute the “Documents” command.



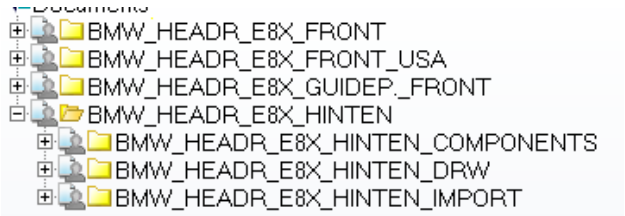
SmarTeam then switches from the Project structure to the Folder structure.



3.6 Automotive folder structure

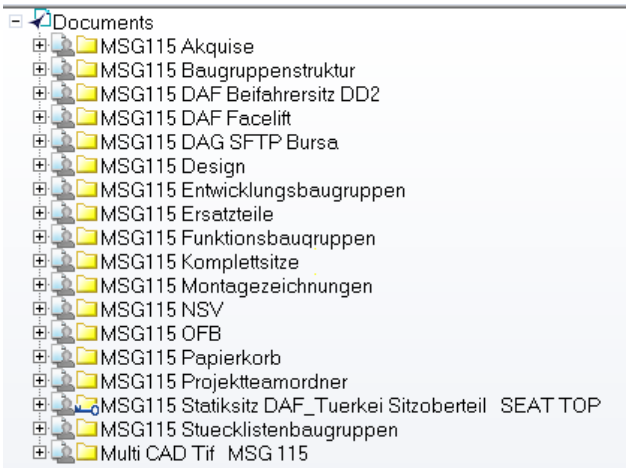
- Top level → all 2D & 3D customer scope
- Parts → all 3D data (apart from customer scope)
- DRW → all drawings (apart from customer scope)
- Other folders → e.g. import, trash, FEA etc.

If there are several assemblies for one vehicle, these are created individually.



3.7 Seating folder structure

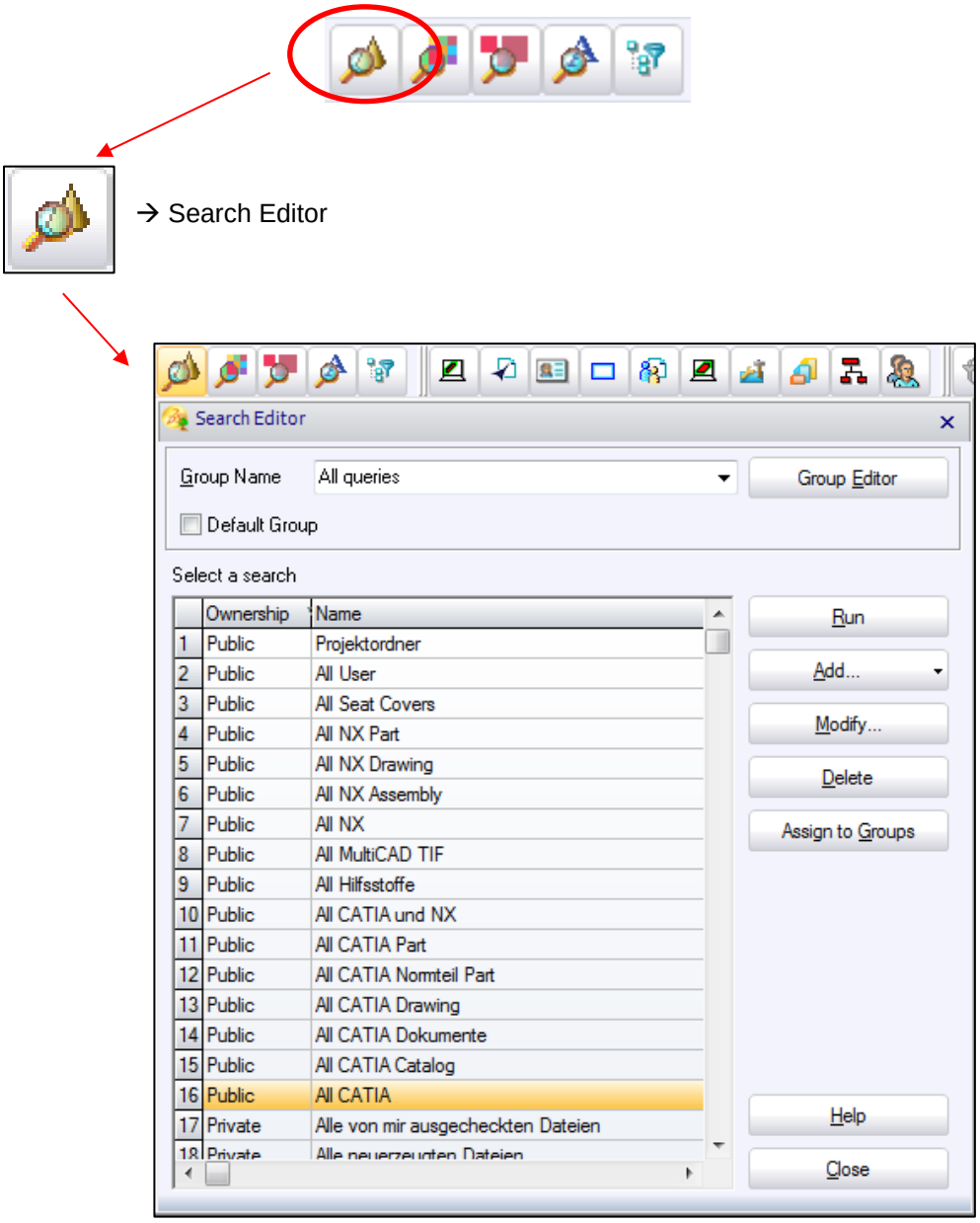
There is no uniform structure in the Seating area.



4 Search

4.1 Standard search

The Search Editor must be called up in order to search for files in SmarTeam. This command can be found on the search bar.



Predefined searches or personal searches are shown in the Search Editor.

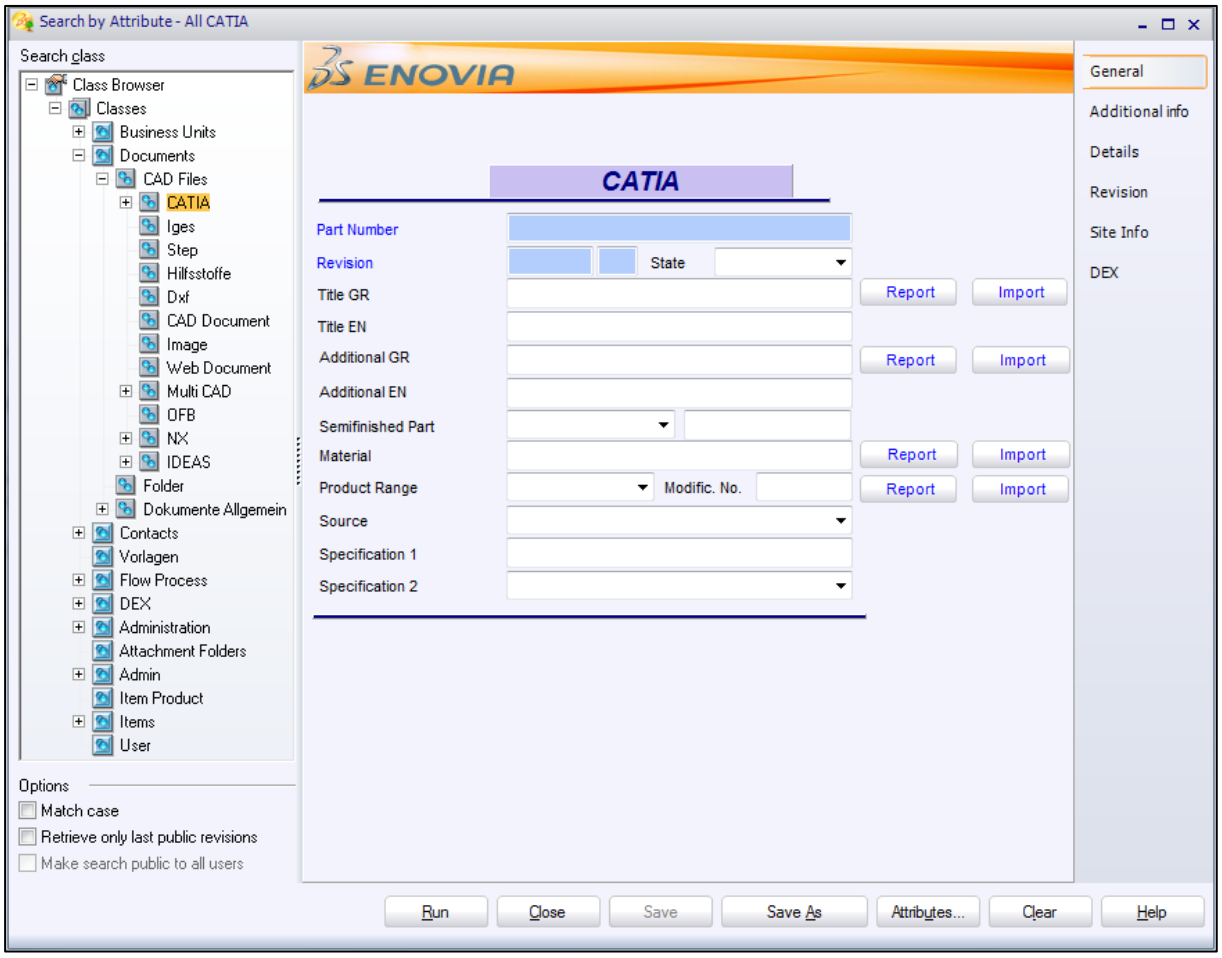
These searches can be executed by using the “Run” command. The “Run” command starts a search. For example, a search is performed for all CATIA documents (ALL CATIA) as shown above.

The “Modify” command can be used to edit or amend these searches.

Predefined searches can only be saved, modified or deleted by a system administrator.
They are available to all users.

Personal searches are only available to the user who saved them.

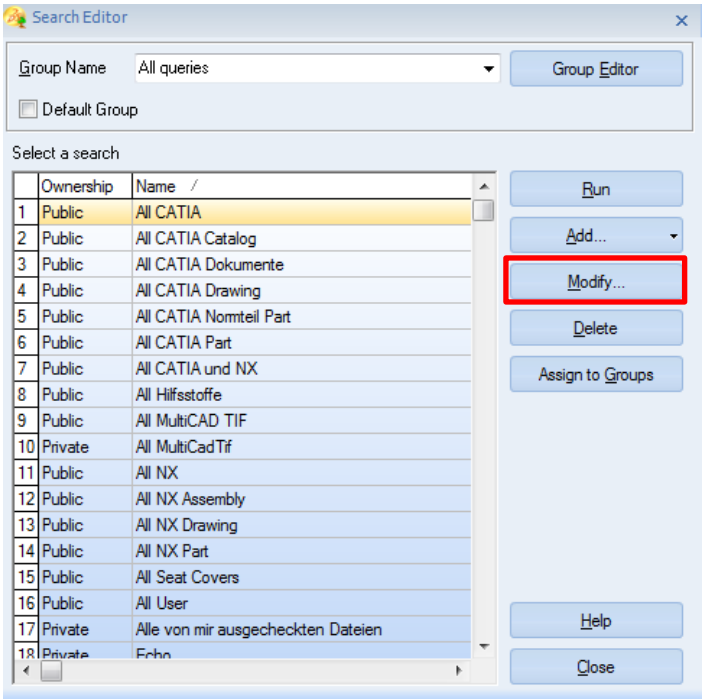
If a search is modified by the “Modify” command, this window appears. Details are entered in the given fields in order to narrow down the search for a document.



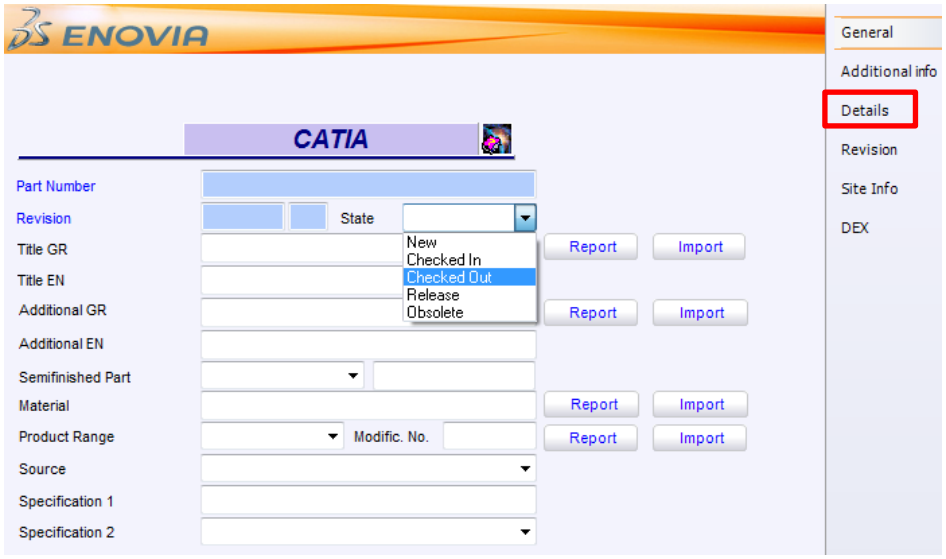
4.2 Save own searches

The Search Editor can be used to save a personal search.

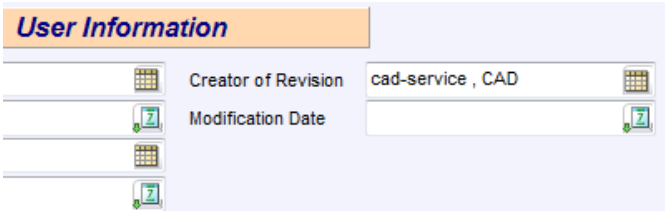
Example: Search for own checked-out documents:



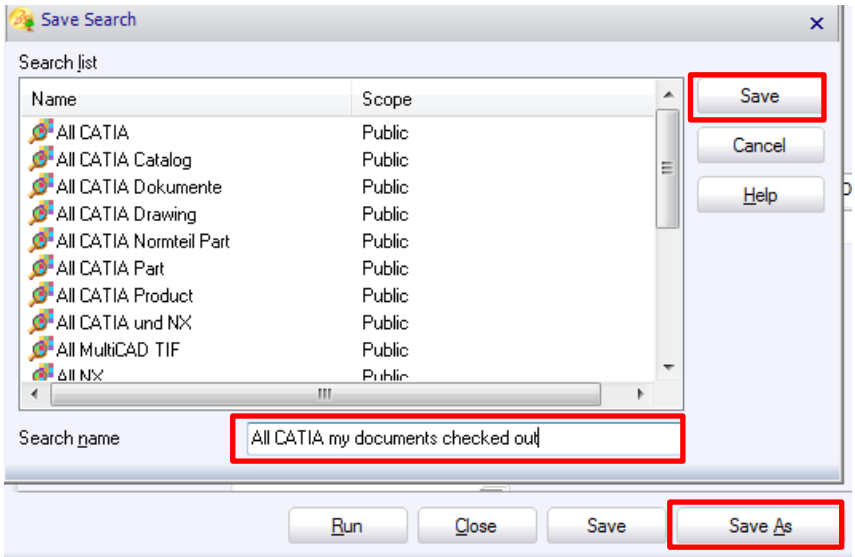
To do this, select an existing search, e.g. “ALL CATIA” and modify it using the “Modify” command. Select “Checked Out” in the “State” pull-down menu on the Profile Card.



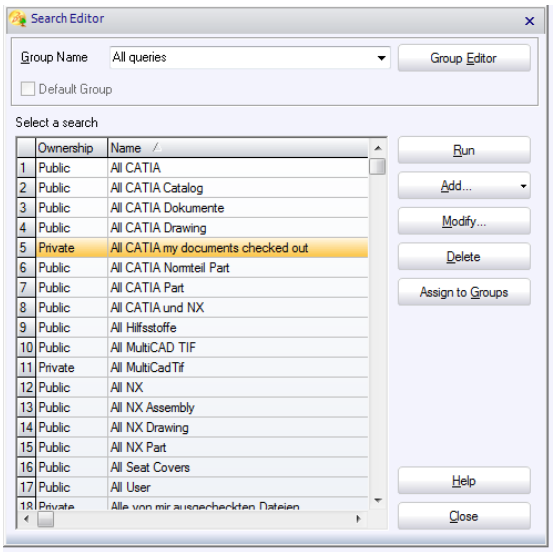
Then, on the sub-tab of the search window, select the creator of the version by clicking “Details” and “Creator of Revision”.



If “Save As” is selected, the search can be saved with a new search name (e.g. All CATIA my documents checked out).



The newly saved search can then be found in the Search Editor as a private search together with all specified or previously saved searches.



These searches can be found in SmarTeam and CATIA accordingly and are used in these programs to search for data or open data in CATIA.

4.3 More search icons on the search bar

The “Find by attributes” command can be used to duplicate a search that has previously been created and queried. This is not possible on the Business Units tab.

An extended search in one or two classes can be executed using this search option. Searching is based on the attributes of these classes and on the attributes of the relationship.

The “Find Text” command makes it easier to search for part numbers or terms in folders. To do this, the folder must be expanded by clicking +.

The “Tree Filter” command can be used in a project to choose whether the last public version, all versions or the last version of a part, an assembly or a drawing is displayed.

5 Icons, Admin Console, language, windows

5.1 Icons

5.1.1 SmarTeam status:

New part

Checked in

Checked out processing in progress

Not up-to-date status

Released (series a, b, …)

Released, not up-to-date (ongoing series modification)

Obsolete series status

5.1.2 SmarTeam documents:

Project

Folder

5.1.3 Catia V5

Part

Assembly

Drawing

Standard part

TIF file

5.1.4 NX

Part

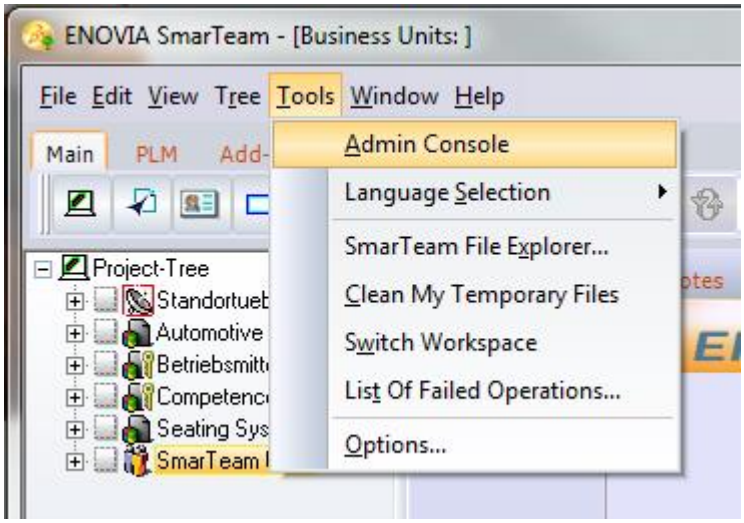
Assembly

Drawing

5.2 Admin Console

5.2.1 Change password

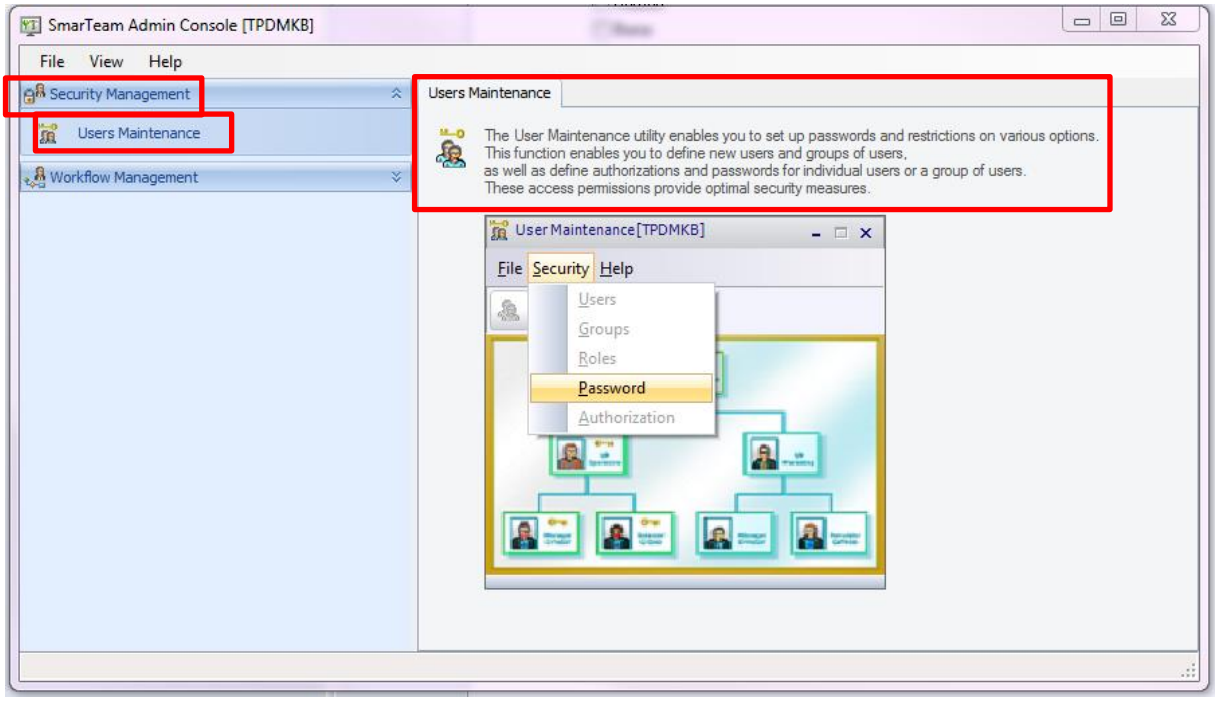
Every user can change their own password as follows, using the Admin Console:



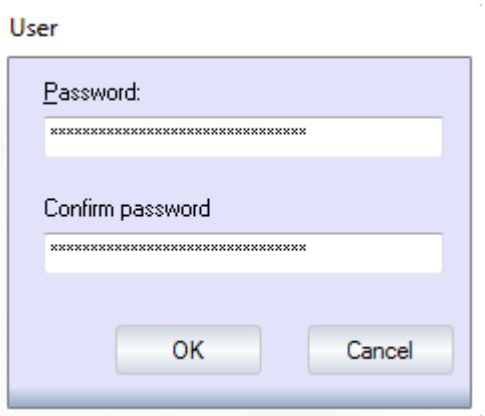
The new login window appears when the “Tools” function is called up in the Admin Console.

Enter the existing username and password here.

The following Admin Console is opened:

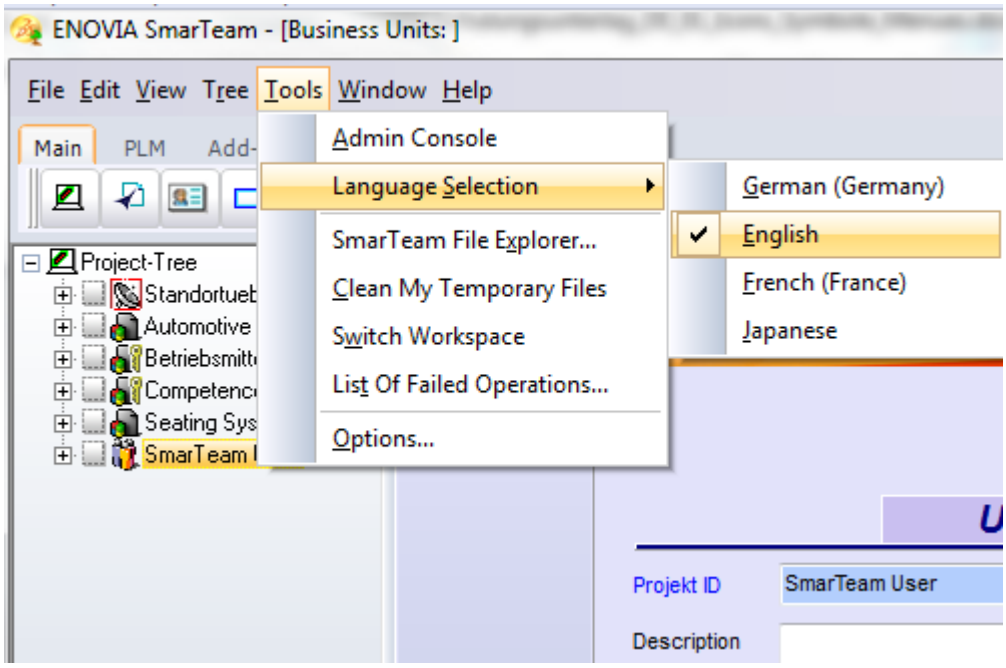


Click “Security Management” to expand it.
In "Users Maintenance", click on the link in field on the right.
Your password can then be modified under “Security” and “Password”.



5.3 Language

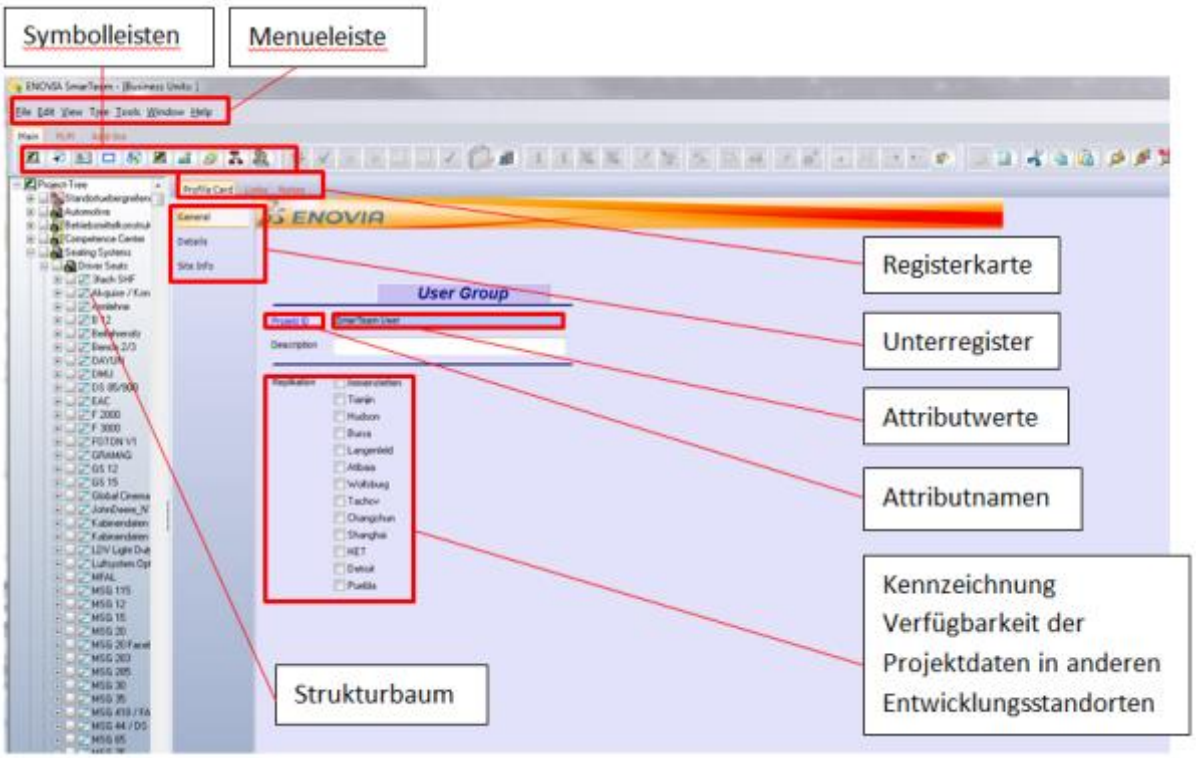
Every user can change the language in SmarTeam as follows:



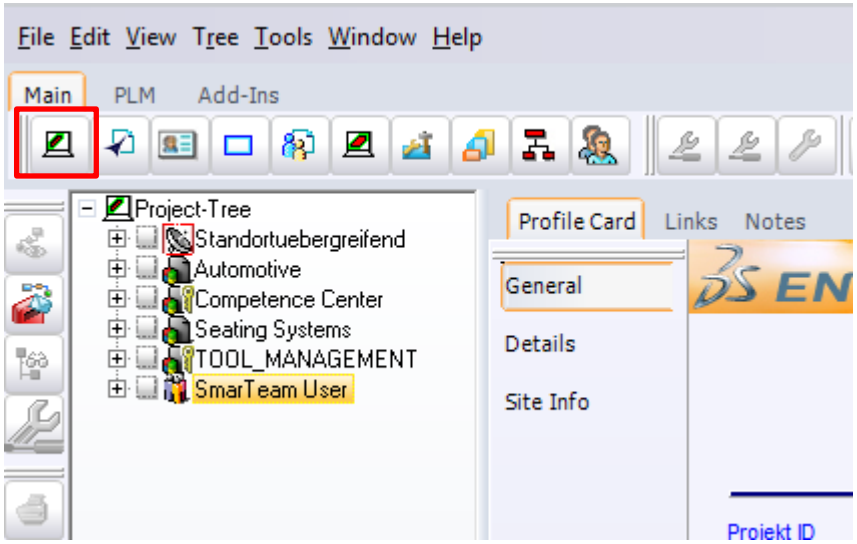
Under “Tools”, select “Language Selection” and select one of the preset languages.

5.4 Window

Designations in the SmarTeam user interface:



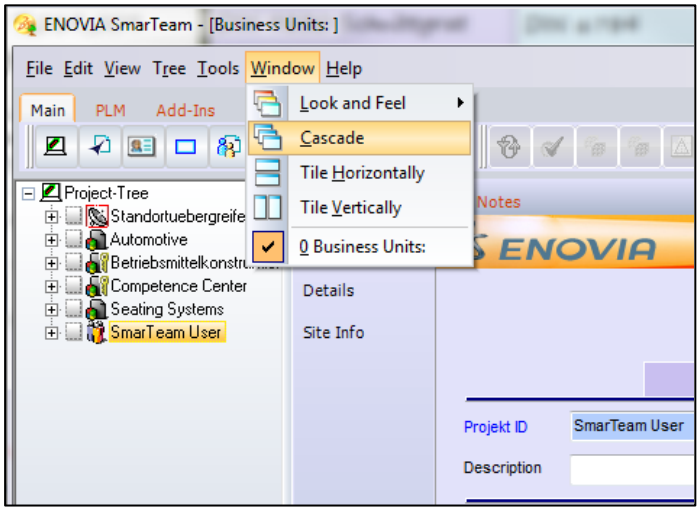
Toolbars can be moved freely as required (up, down, right or left to the edge of the screen or directly into the window).
Click and hold the double bar (red frame) to move the actual bar as required.



5.4.1 Arrange windows

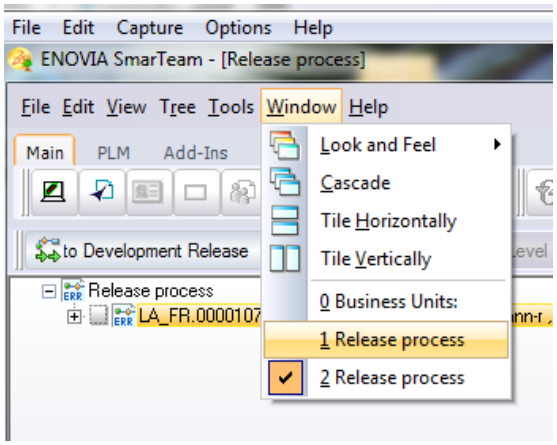
The appearance of the SmarTeam Editor window on the desktop can be modified like all the other default windows, i.e. zoom in, zoom out, move or close.
The “Business Units” window cannot be closed, it can only be minimised!

All windows open on the desktop can be optionally cascaded or tiled from the main menu via “Window”.



5.4.2 Navigate between windows

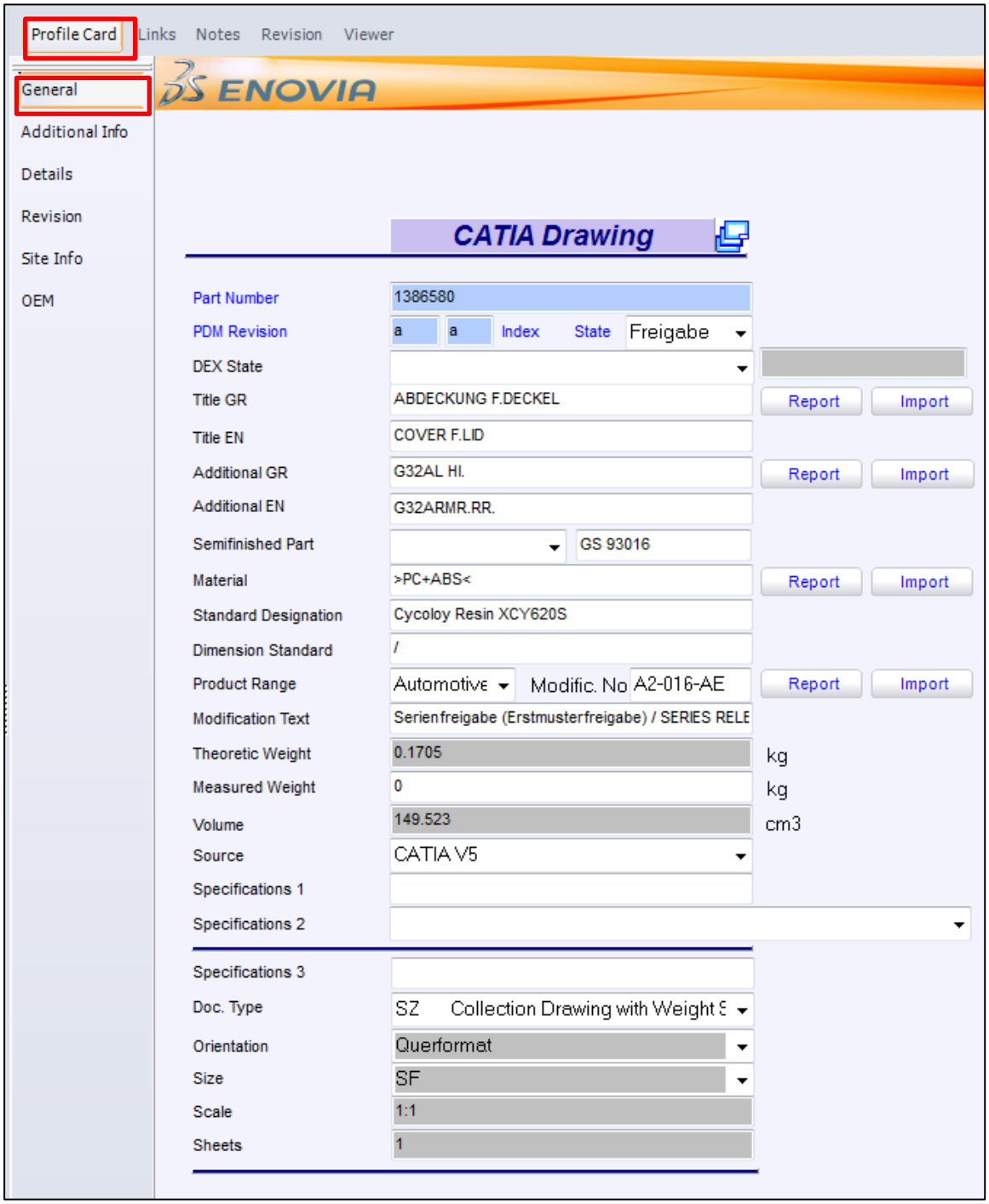
If there are several open windows, you can navigate between them as required from the main menu via “Window”.



6 Profile Card

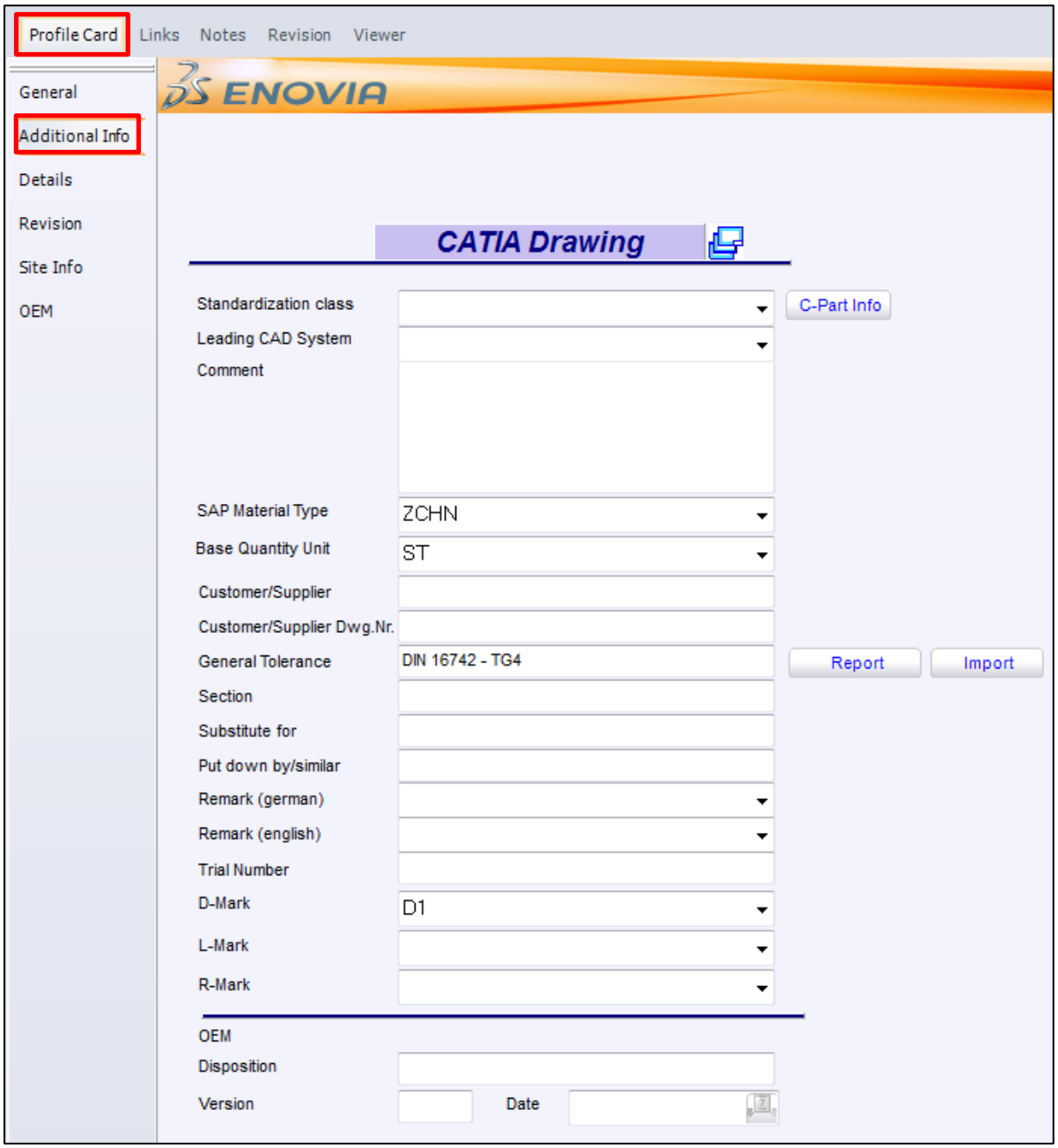
6.1 General

There is a Profile Card for each file (Part, Assembly, Drawing, etc.) that is stored in SmarTeam. It contains general information such as description, material, origin, volume, weight, etc.



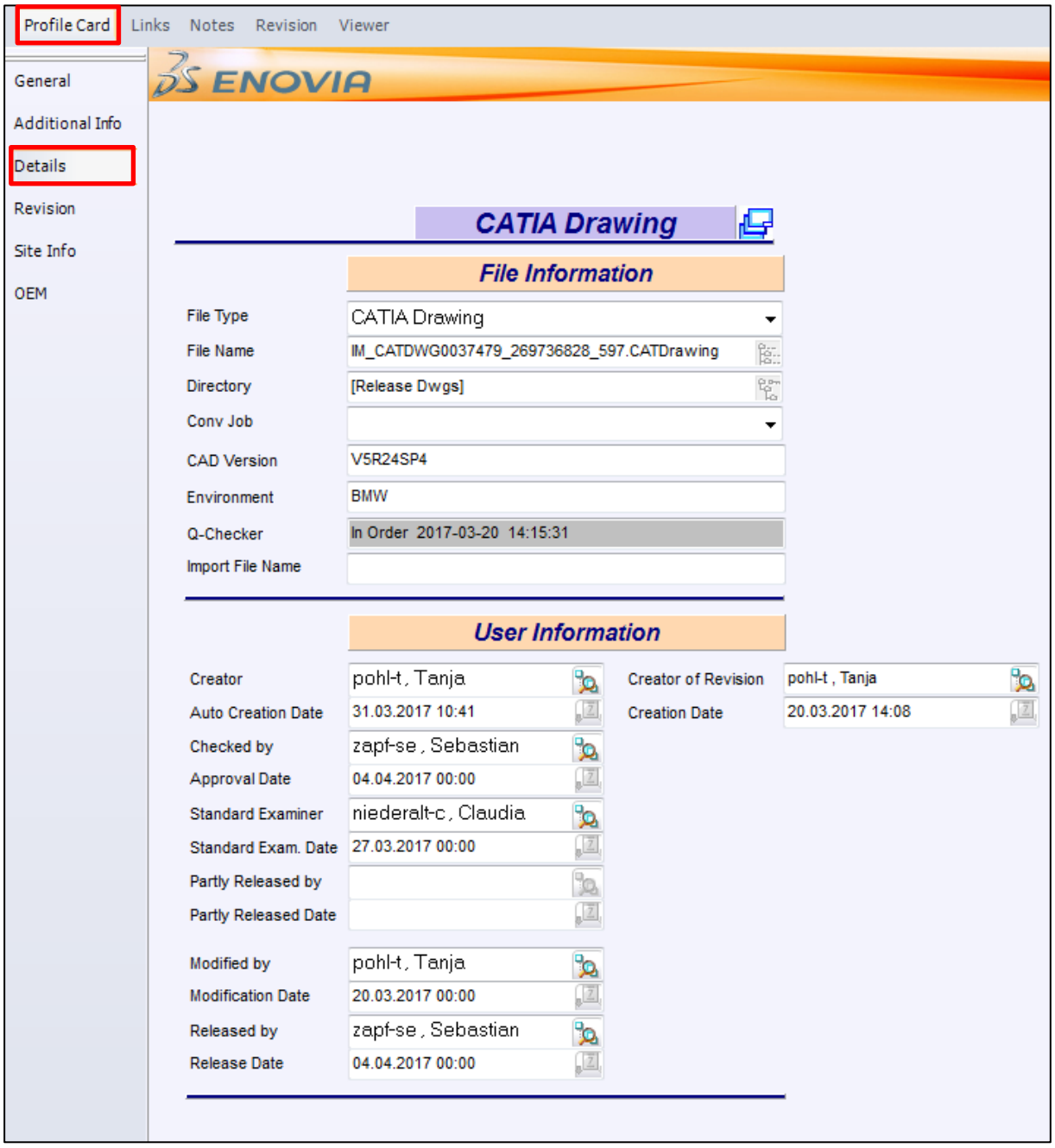
6.2 Additional Info

More general data such as SAP Material Type, Base Quantity Unit, General Tolerance, etc. can be found on the Additional Info tab.



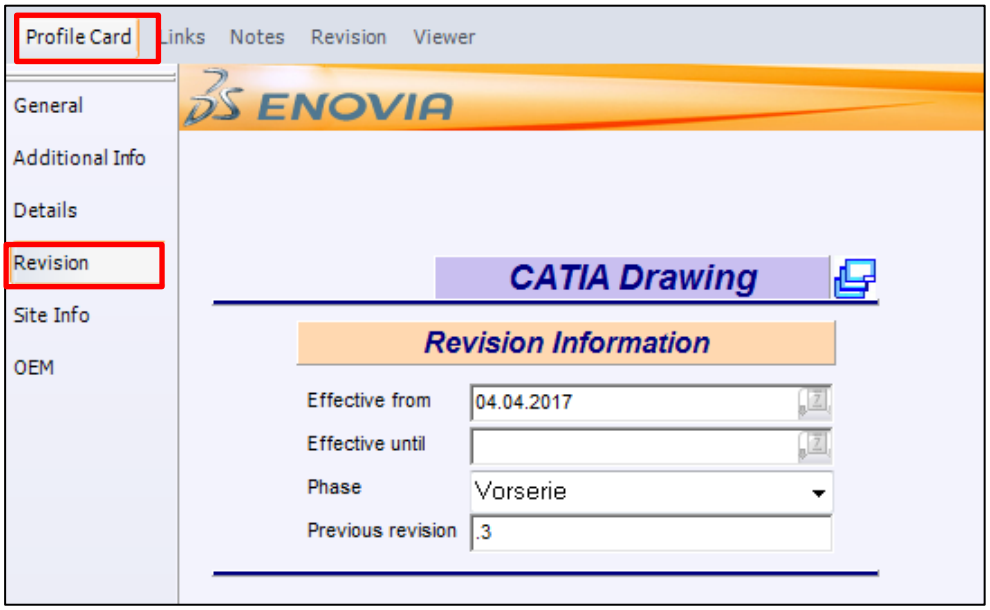
6.3 Details

The Details tab contains information about the File Type, File Name, Creator, Modified by, Modification Date, etc.



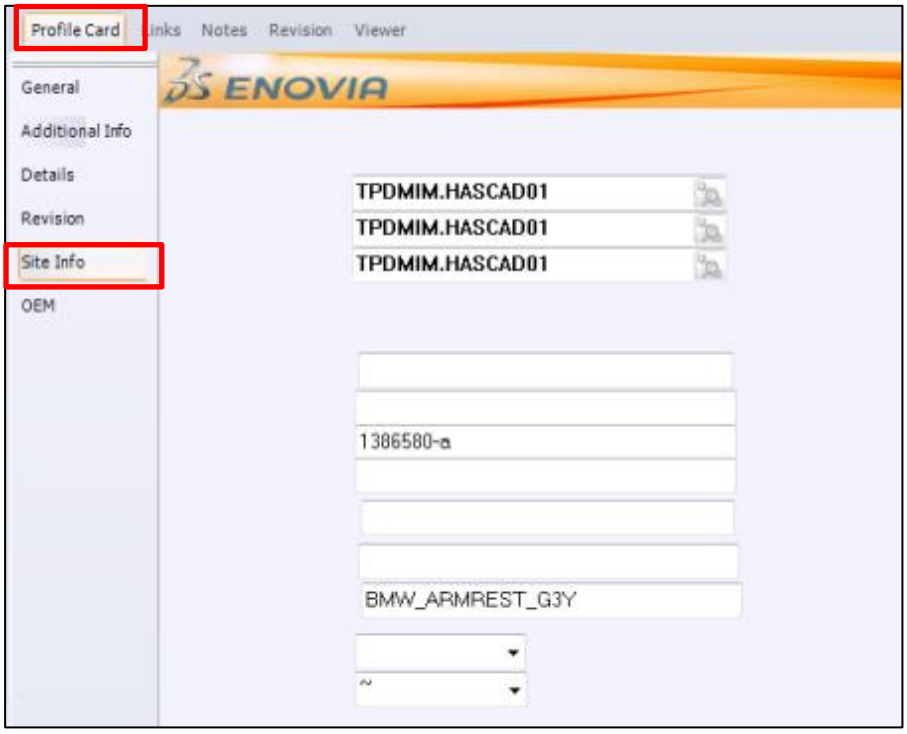
6.4 Revision

The information on the “Revision” tab is not important for the user.



6.5 Site Info

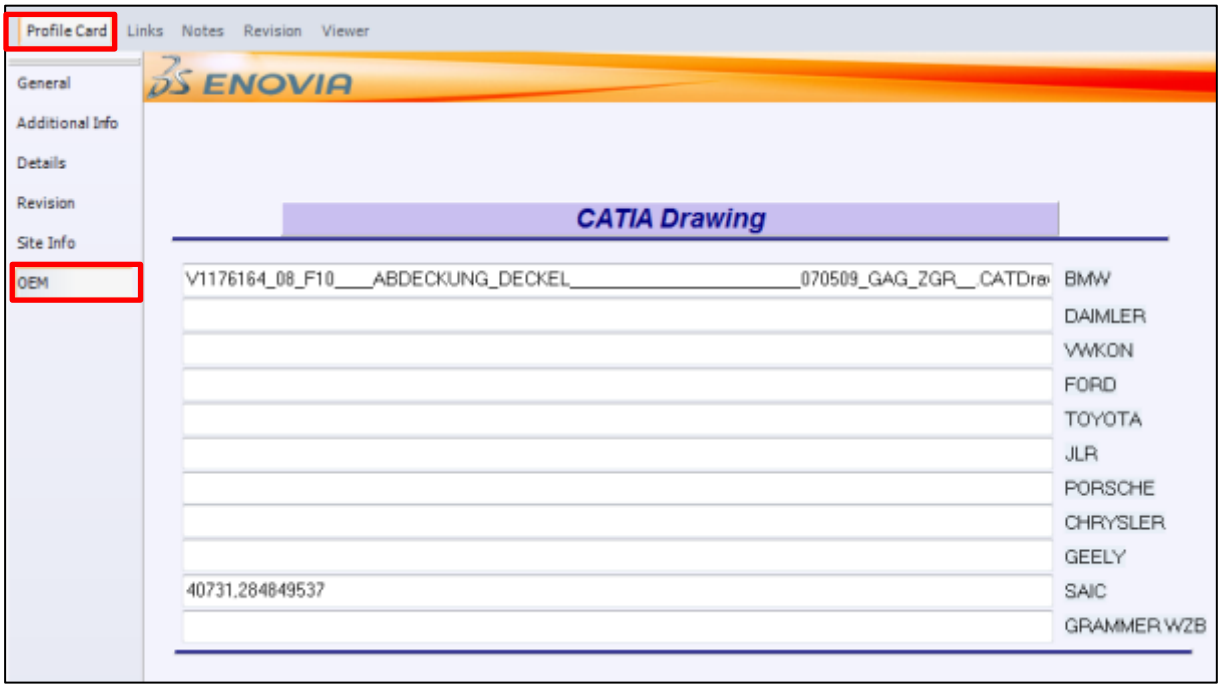
The information on the “Site Info” tab it is not important for a “regular” user.



6.6 OEM

OEM-specific numbers or descriptions can be entered here.
Example:
If a drawing is read out via DEX and the BMW profile, it is renamed as:

V1176164_08_F10____ABDECKUNG_DECKEL_____070509_GAG_ZGR__.CATDrawing

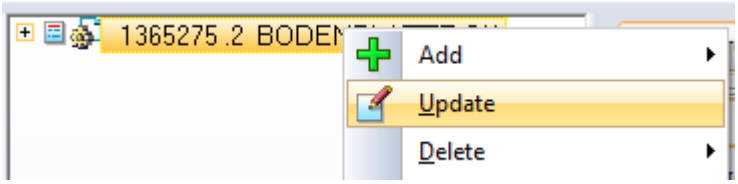


At the end of a process (release/modification process, pre-production release, etc.) all the contents of the Profile Card are transferred to SAP.

6.7 Populating a Profile Card

There are two ways of populating a Profile Card:

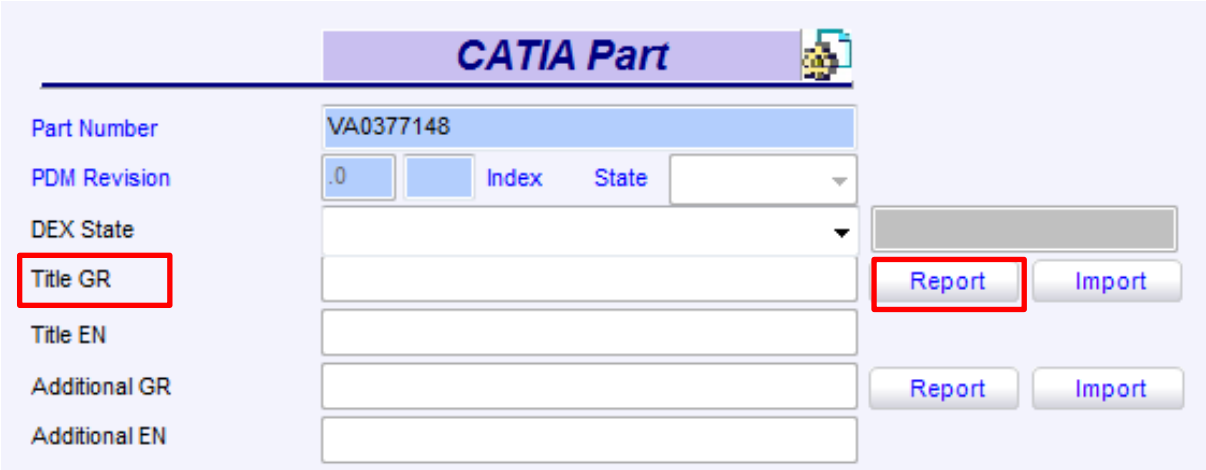
1. Immediately after saving a file in SmarTeam
2. In SmarTeam, after a file has been saved, by using the command “Update” Profile Card



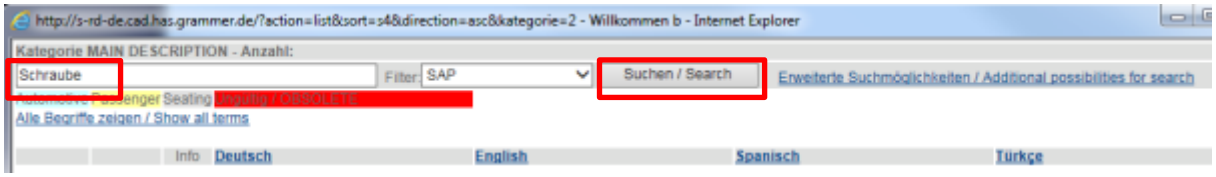
The procedure is the same, however option 2 is preferable.

6.7.1 Title GR/Title EN

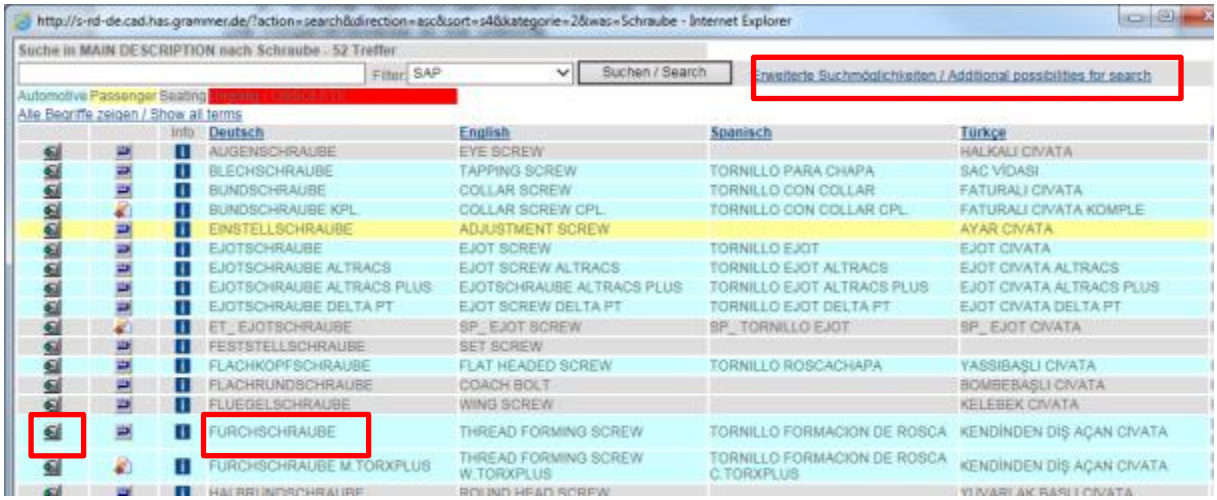
After clicking on “Report”, the description database opens.



You can search for the desired description in the “Kategorie MAIN DESCRIPTION – Anzahl:” field, e.g. “Screw”.

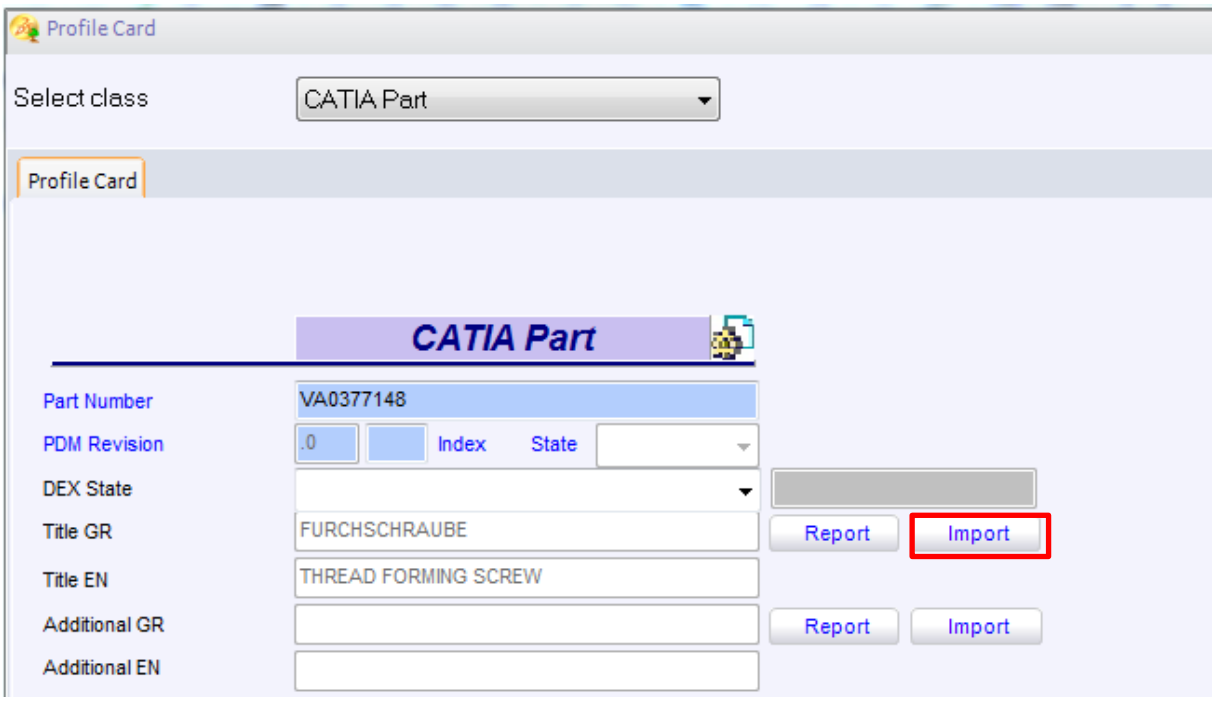


Select the desired result with “Export for SmarTeam”, e.g., “FURCHSCHRAUBE”.



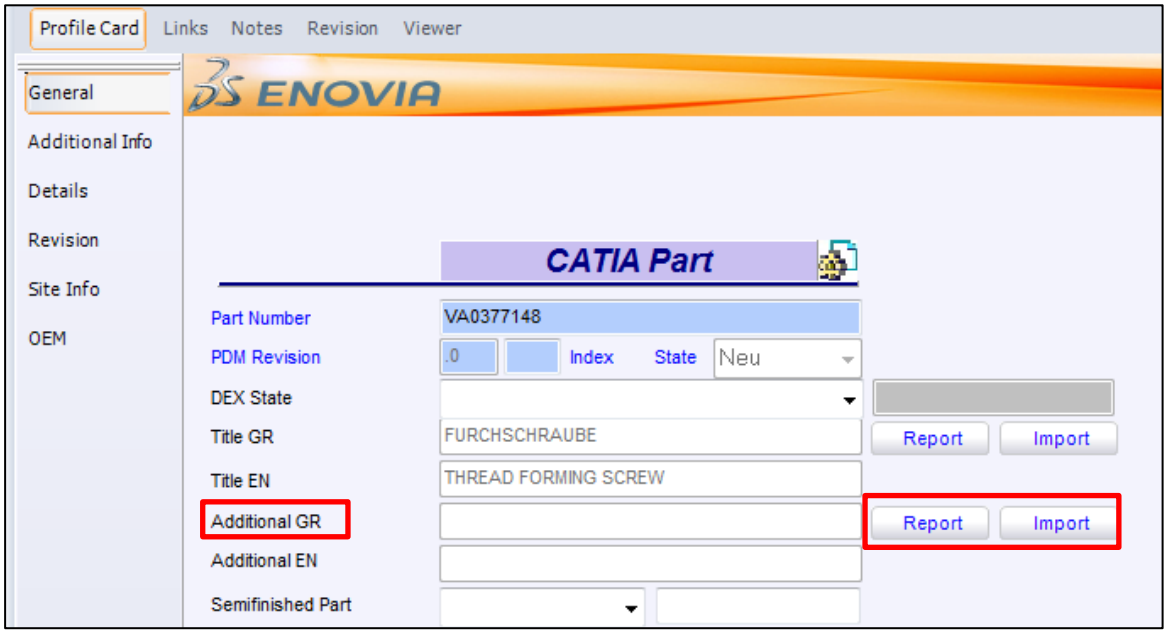
Under “Additional possibilities for search” you will find options for restricting a search.

After clicking on “Import”, the description is mapped to the SmarTeam Profile Card.



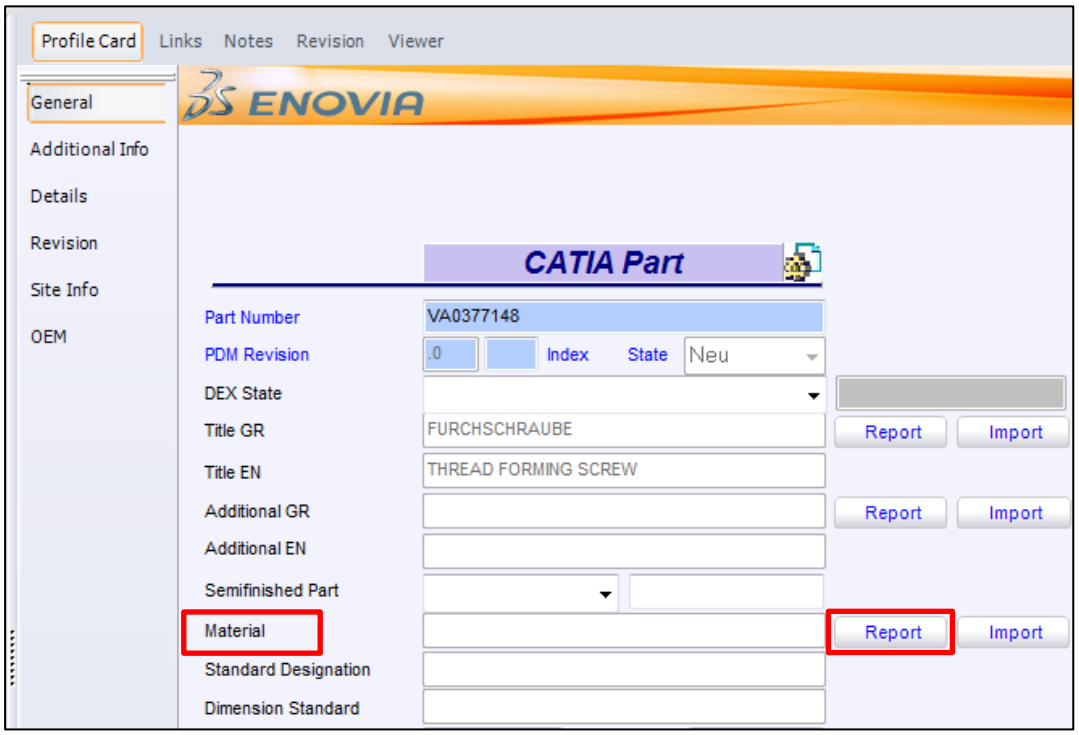
6.7.2 Additional GR

The procedure for an additional description is the same (“Additional GR”)

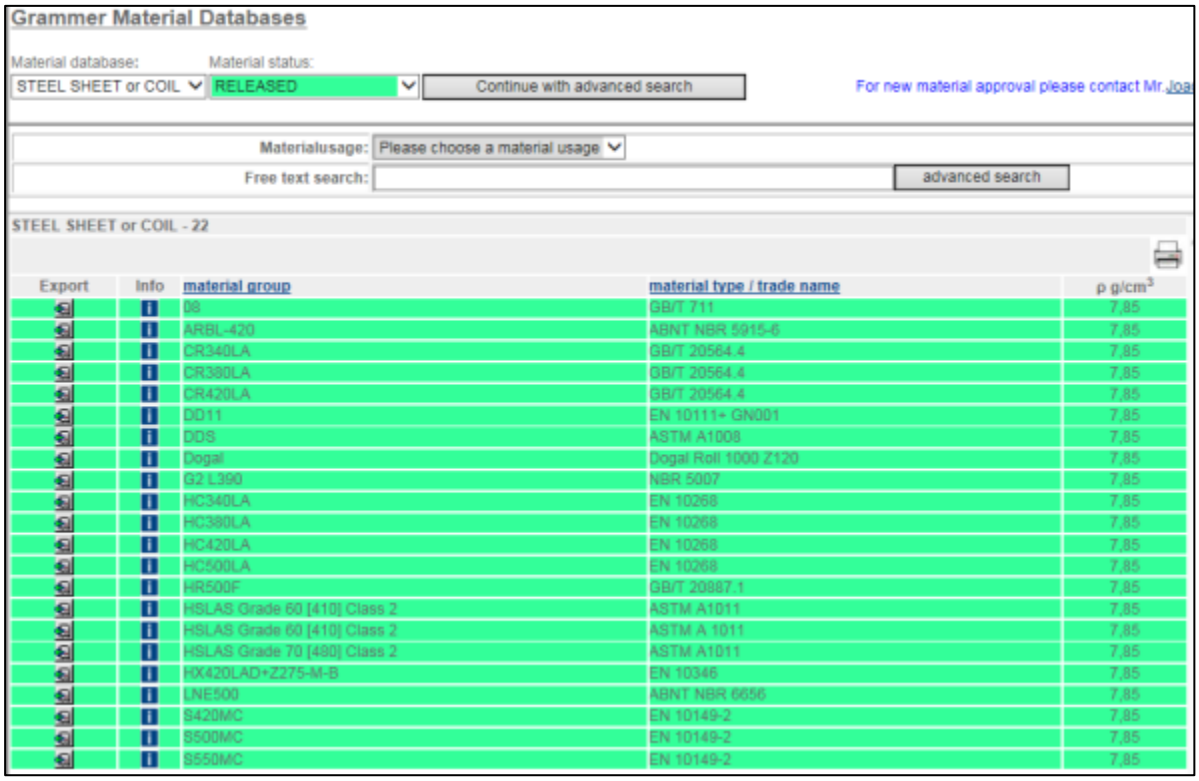


6.7.3 Material

After clicking on “Report”, the material database opens.



Clicking on “Continue with advanced search” lists all the materials.

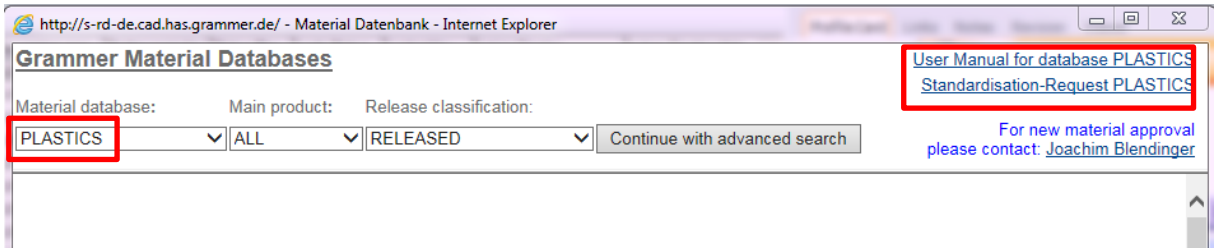


Export	Info	material group	material type / trade name	ρ g/cm³
		08	GB/T 711	7,85
		ARBL-420	ABNT NBR 5915-6	7,85
		CR340LA	GB/T 20564.4	7,85
		CR380LA	GB/T 20564.4	7,85
		CR420LA	GB/T 20564.4	7,85
		DD11	EN 10111+ GN001	7,85
		DDS	ASTM A1008	7,85
		Dogal	Dogal Roll 1000 Z120	7,85
		G2 L390	NBR 5007	7,85
		HC340LA	EN 10268	7,85
		HC380LA	EN 10268	7,85
		HC420LA	EN 10268	7,85
		HC500LA	EN 10268	7,85
		HR500F	GB/T 20887.1	7,85
		HSLAS Grade 60 [410] Class 2	ASTM A1011	7,85
		HSLAS Grade 60 [410] Class 2	ASTM A 1011	7,85
		HSLAS Grade 70 [480] Class 2	ASTM A1011	7,85
		HX420LAD+Z275-M-B	EN 10346	7,85
		LNE500	ABNT NBR 6656	7,85
		S420MC	EN 10149-2	7,85
		S500MC	EN 10149-2	7,85
		S550MC	EN 10149-2	7,85

The filters provided can be used to restrict material hits.
If the desired material is listed, it can also be mapped to the Profile Card by clicking “Export for SmarTeam” and then “Import”.

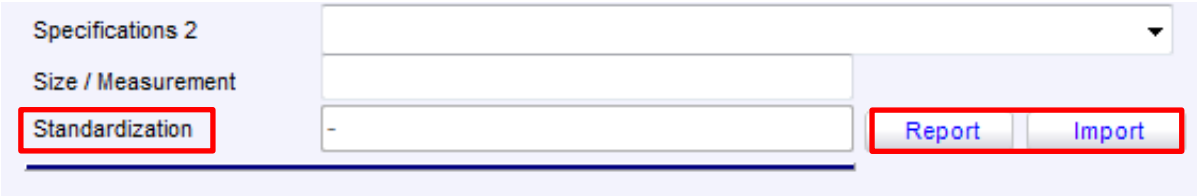
A guide for plastics is available under the following link:

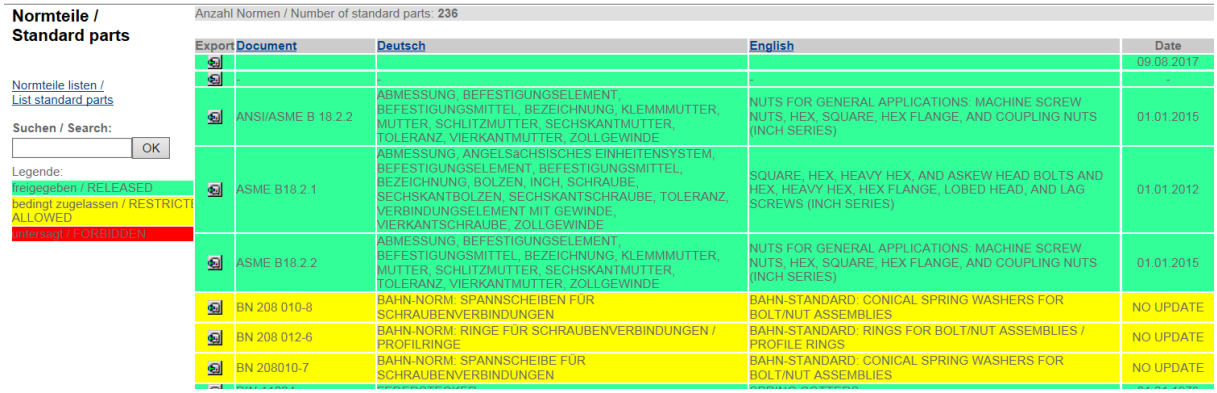
“User Manual for database PLASTICS”



6.7.4 Standardization

Export and import are performed as for the other lists.

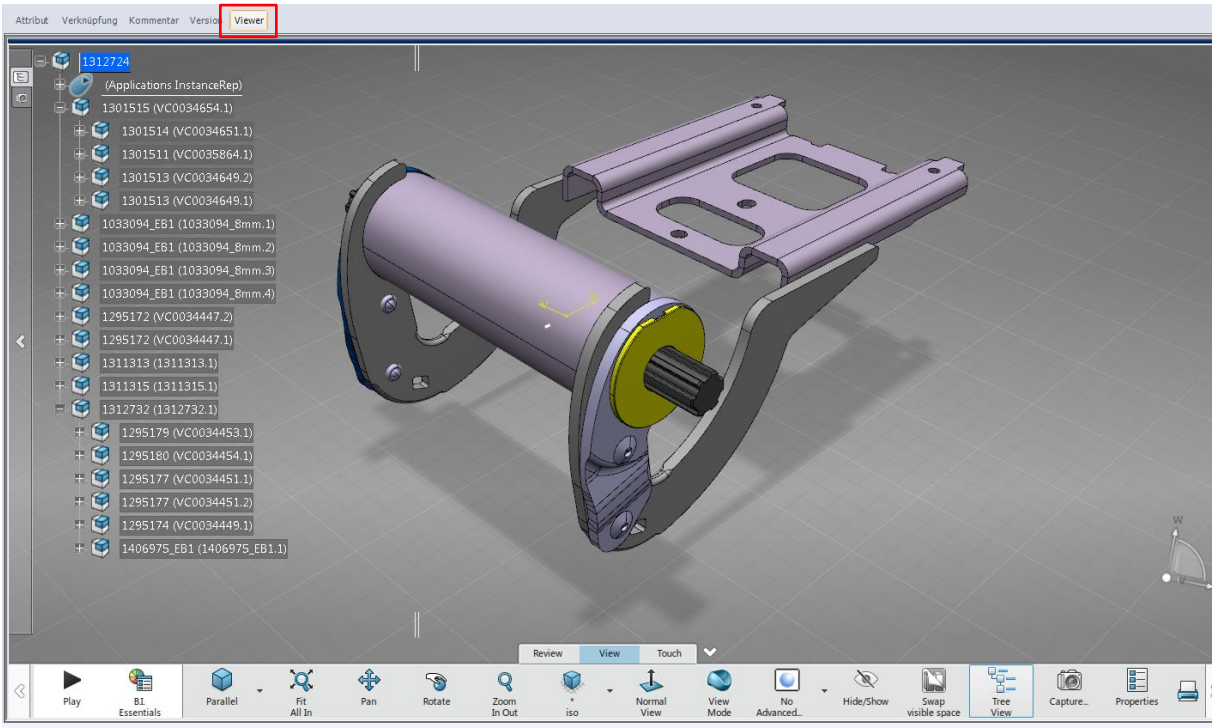




Export	Document	Deutsch	English	Date
				09.08.2017
	-			-
	ANSI/ASME B 18.2.2	ABMESSUNG, BEFESTIGUNGSELEMENT, BEFESTIGUNGSMITTEL, BEZEICHNUNG, KLEMMMUTTER, MUTTER, SCHLITZMUTTER, SECHSKANTMUTTER, TOLERANZ, VIERKANTMUTTER, ZOLLGEWINDE	NUTS FOR GENERAL APPLICATIONS: MACHINE SCREW NUTS, HEX, SQUARE, HEX FLANGE, AND COUPLING NUTS (INCH SERIES)	01.01.2015
	ASME B18.2.1	ABMESSUNG, ANGELSäCHSISCHES EINHEITENSYSTEM, BEFESTIGUNGSELEMENT, BEFESTIGUNGSMITTEL, BEZEICHNUNG, BOLZEN, INCH, SCHRAUBE, SECHSKANTBOLZEN, SECHSKANTSCHRAUBE, TOLERANZ, VERBINDUNGSELEMENT MIT GEWINDE, VIERKANTSCHRAUBE, ZOLLGEWINDE	SQUARE, HEX, HEAVY HEX, AND ASKEW HEAD BOLTS AND HEX, HEAVY HEX, HEX FLANGE, LOBED HEAD, AND LAG SCREWS (INCH SERIES)	01.01.2012
	ASME B18.2.2	ABMESSUNG, BEFESTIGUNGSELEMENT, BEFESTIGUNGSMITTEL, BEZEICHNUNG, KLEMMMUTTER, MUTTER, SCHLITZMUTTER, SECHSKANTMUTTER, TOLERANZ, VIERKANTMUTTER, ZOLLGEWINDE	NUTS FOR GENERAL APPLICATIONS: MACHINE SCREW NUTS, HEX, SQUARE, HEX FLANGE, AND COUPLING NUTS (INCH SERIES)	01.01.2015
	BN 208 010-8	BAHN-NORM: SPANNSCHEIBEN FÜR SCHRAUBENVERBINDUNGEN	BAHN-STANDARD: CONICAL SPRING WASHERS FOR BOLT/NUT ASSEMBLIES	NO UPDATE
	BN 208 012-6	BAHN-NORM: RINGE FÜR SCHRAUBENVERBINDUNGEN / PROFILRINGE	BAHN-STANDARD: RINGS FOR BOLT/NUT ASSEMBLIES / PROFILE RINGS	NO UPDATE
	BN 208010-7	BAHN-NORM: SPANNSCHEIBE FÜR SCHRAUBENVERBINDUNGEN	BAHN-STANDARD: CONICAL SPRING WASHERS FOR BOLT/NUT ASSEMBLIES	NO UPDATE

7 Viewer in SmartTeam for CAD data

7.1 Catia 3D Viewer



7.1.1 Functions of the Catia 3D Viewer

- Shows the tree structure as in CatiaV5 with the possibility of showing and hiding parts
- Generate additional annotations (only temporary)
- Generate 3D sections
- View scenes generated in CatiaV5
- Display No Show area
- Show annotations, FTA and scenes
- Show and hide single parts and assemblies in the tree structure
- Switch to different views (solid, wireframe, material, etc.)
- Drag compass to parts in order to move and rotate them
- Fit, move, rotate, zoom, select new pivot point
- Generate images

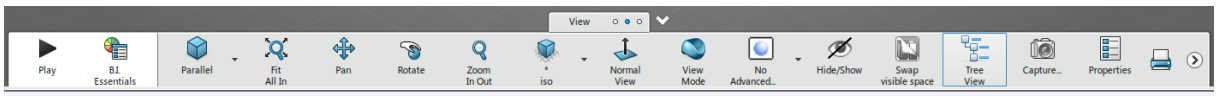
7.1.2 Commands using mouse buttons

Move → press middle mouse button (MMB)

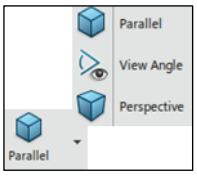
Rotate → press MMB and right mouse button (RMB) simultaneously → select new pivot point → use MMB to select element

Zoom → press MMB + right RMB → release RMB

7.1.3 Toolbar with commands



Shows front view, top view, side view, etc. successively



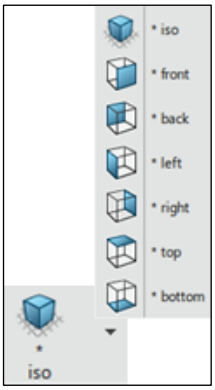
View options: Parallel, View Angle and Perspective

Zooms the geometry in or out

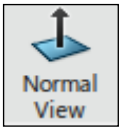
Pan across the screen

Rotate the part

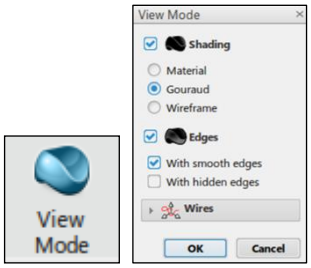
Zoom in or out



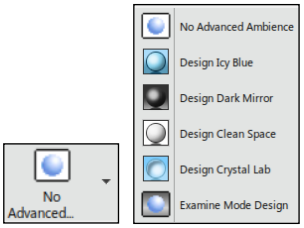
View options: iso, front, back, left, right, top, bottom



View objects vertically relative to a selected plane



Set the View Mode

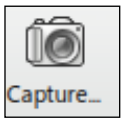


Change the background colour

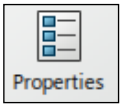
Hide/show objects

Switch to no show area

Show/hide tree structure



Capture image of current view



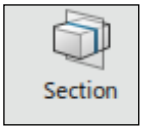
Settings

Print

Generate temporary images

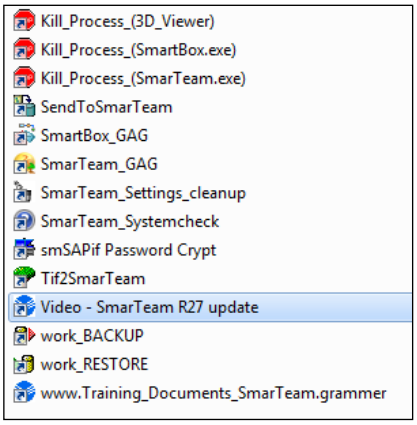
Generate temporary annotation

Measure individual elements and measure between 2 elements

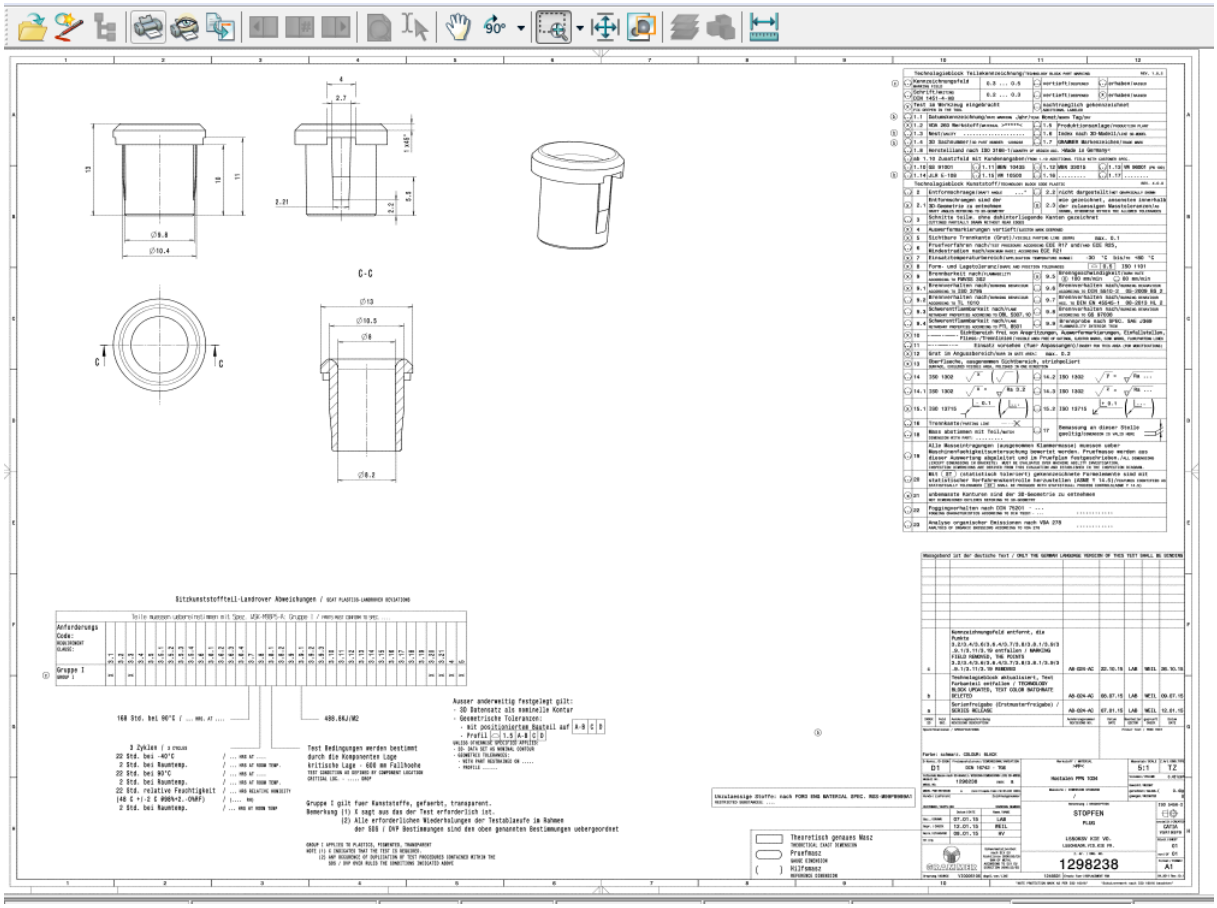


Take a section of parts and assemblies

Videos providing more detailed explanations can be found in the “CAD Start menu”:
Q:\tools\CAxStartMenu\PDM_SmarTeam



7.2 Catia 2D Viewer



7.2.1 Functions of the Catia 2D Viewer

- Fit, move, zoom
- Measure
- Markup
- Print
- Convert (generate PDF)
- Open two drawings for comparison

7.2.2 Catia 2D Viewer Toolbar with commands

The most important functions are listed below

Open a file

Create and edit markups

Print options

Convert (generate PDF)

Move

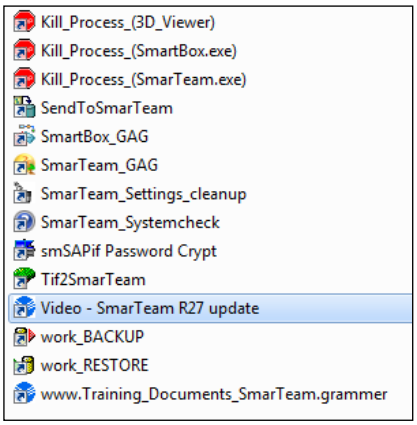
Turn sheet

Zoom functions

Fit all

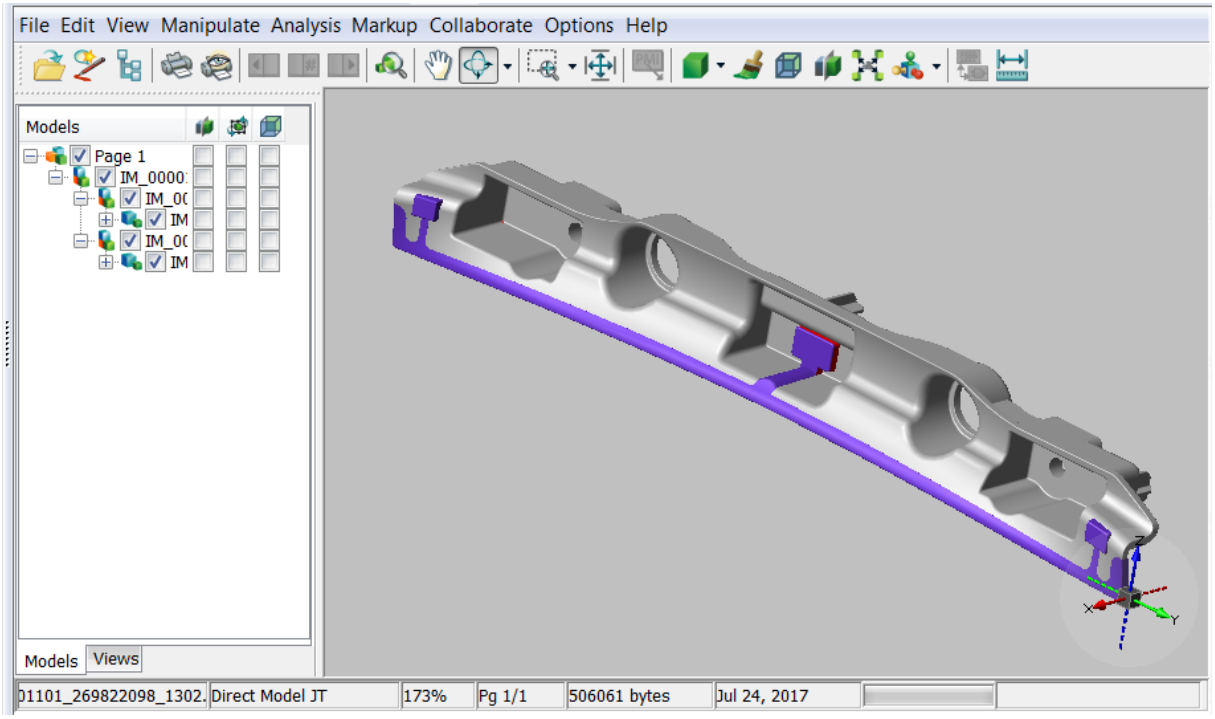
Measure

Videos providing more detailed explanations can be found in the “CAD Start menu”:
Q:\tools\CAxStartMenu\PDM_SmarTeam



Detailed documentation can be found under Q:\catiaV5docu\GRAMMER\AutoVue .

7.3 NX 3D Viewer



7.3.1 Functions of the NX 3D Viewer

- Shows the tree structure with the possibility of showing and hiding parts
- Generate 3D sections
- Switch to different views (solid, wireframe, material, etc.)
- Fit, move, rotate, zoom, select new pivot point

7.3.2 Commands using mouse buttons

Move → press middle mouse button (MMB)

Rotate → left mouse button (LMB)

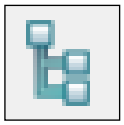
Zoom → scroll wheel on MMB

7.3.3 Toolbar with commands

The most important functions are listed below.

Open a file

Create and edit markups



Hide/show tree structure

Print options

Move

Rotate

Zoom functions

Fit all

Set the View Mode

Transparent settings

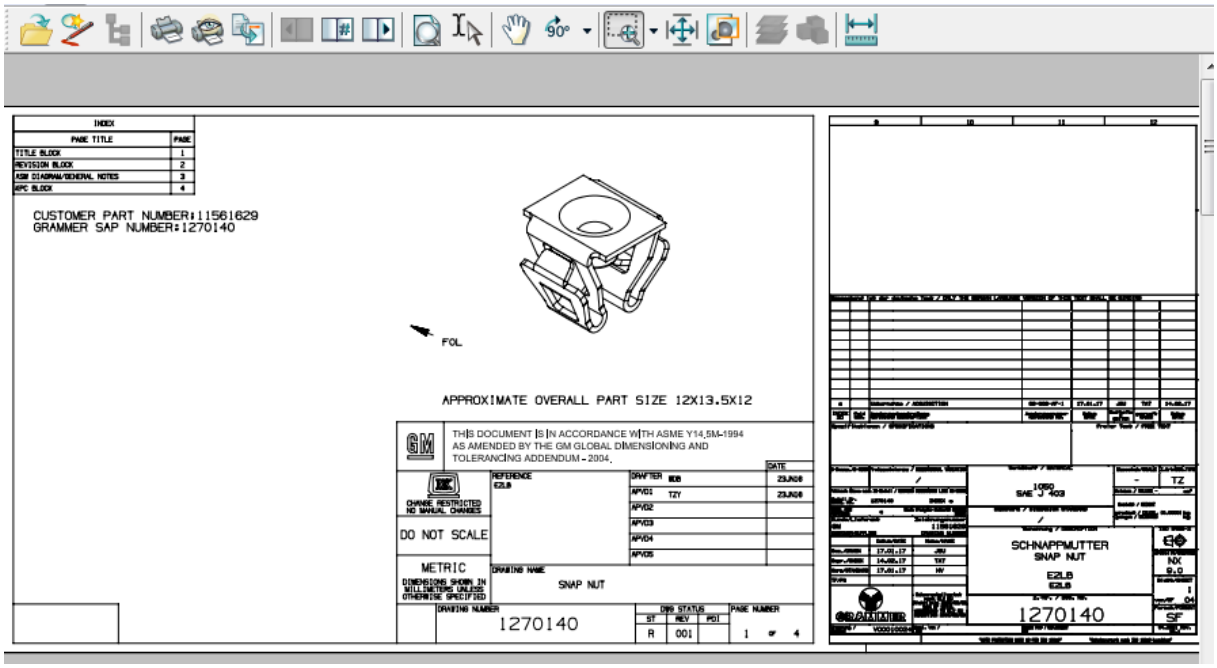
Generate 3D sections

Explode

Measure

Detailed documentation can be found under Q:\catiaV5docu\GRAMMER\AutoVue .

7.4 NX 2D Viewer



7.4.1 Difference between 2D NX Viewer and 2D Catia Viewer

The 2D NX Viewer does not differ from the 2D Catia Viewer, they therefore both provide the same functions.

The 2D NX also provides the following function::

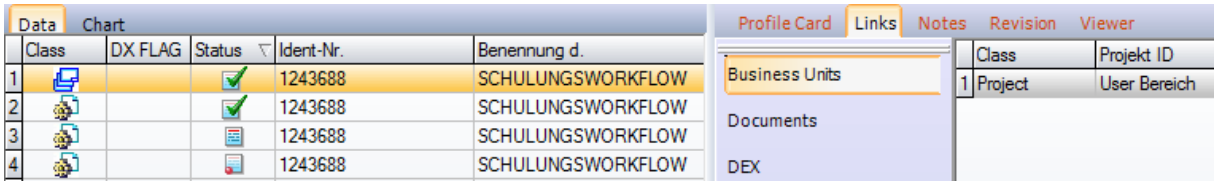
Scroll between pages

Detailed information can be found under Q:\catiaV5docu\GRAMMER\AutoVue .

8 Links

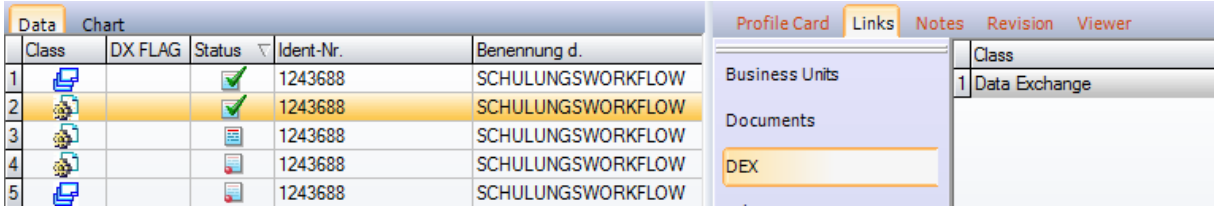
After searching for documents or opening a project, the user can view the various links in SmarTeam via the “Links” tab.

8.1 Business Units



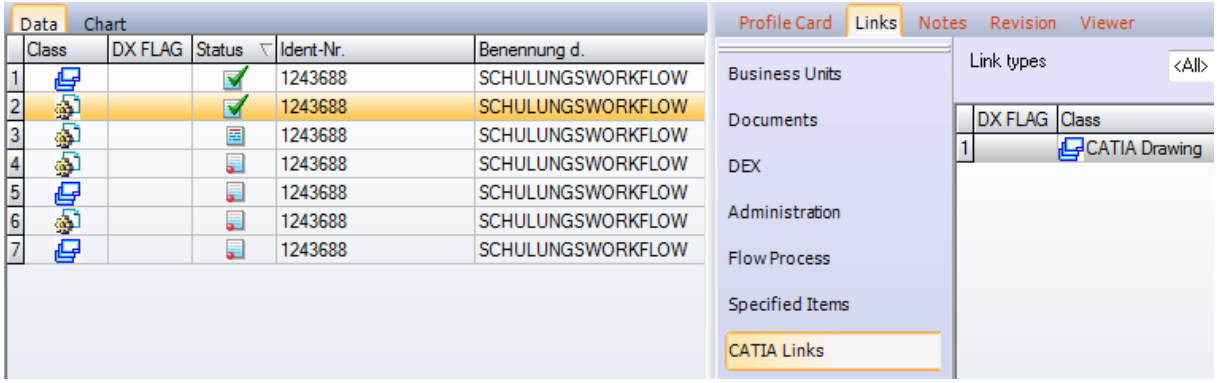
The “Business Units” heading shows the links to projects in SmarTeam.

8.2 DEX



The “DEX” heading shows the links to data exchange processes.

8.3 CATIA links



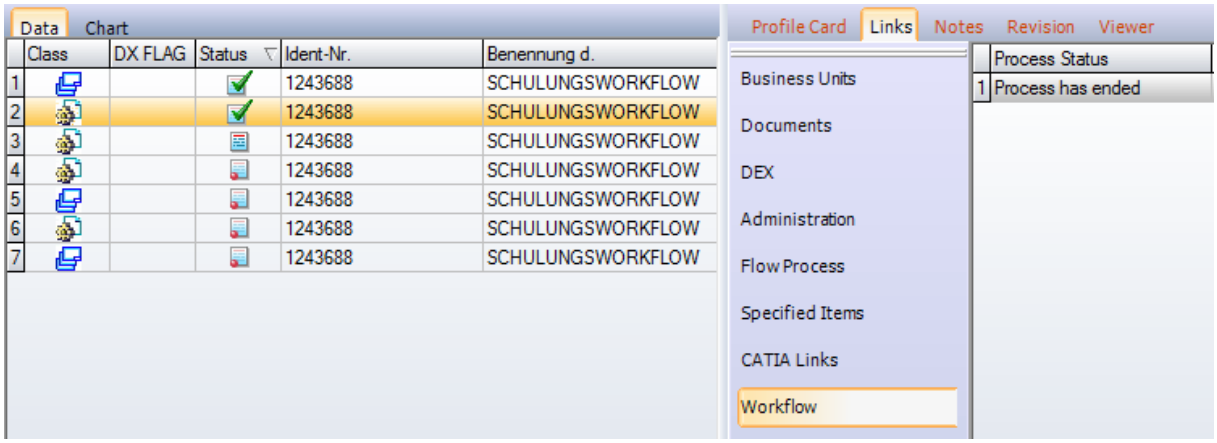
The heading “CATIA Links” shows all links that have been generated internally in the CATIA V5 CAD program.

Examples:

- Derivation of views in a drawing
- Part-in-part links (in both directions)

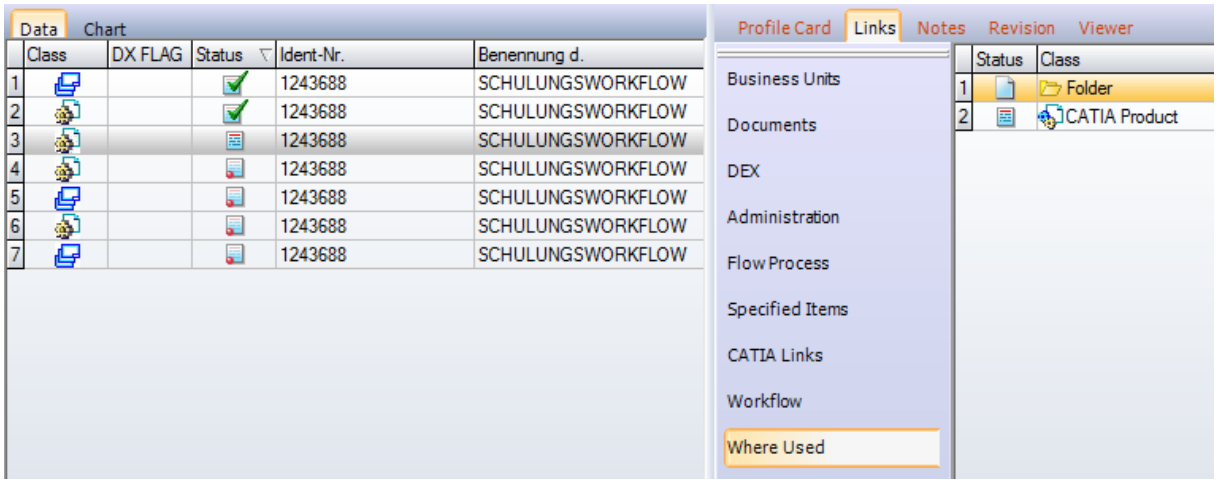
Links can be filtered using the “Links type” selection box.

8.4 Workflow



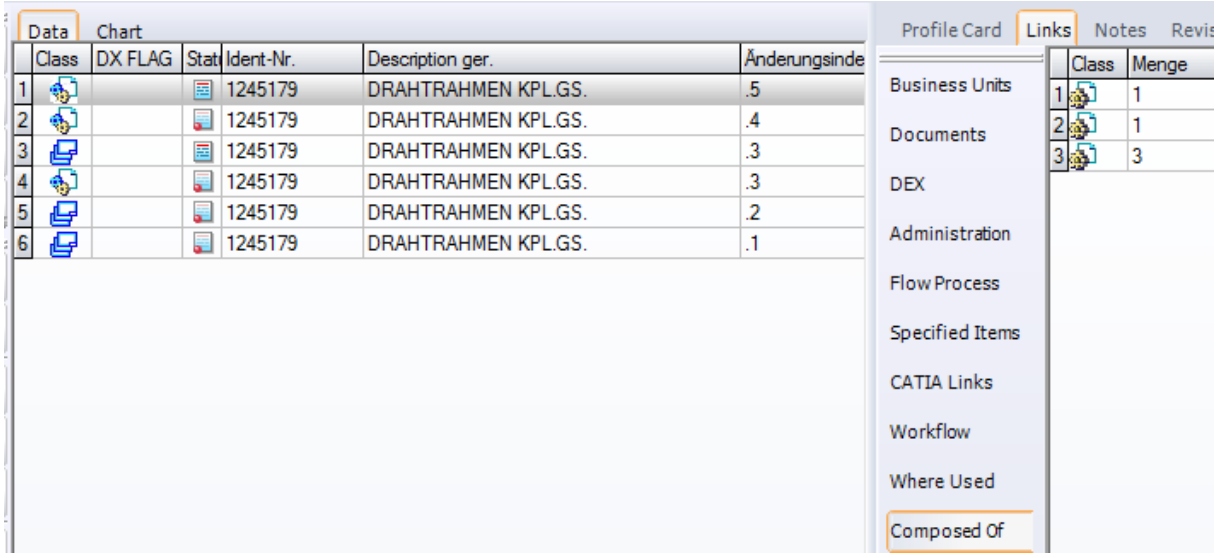
Links to ongoing or completed SmarTeam workflows are displayed here.

8.5 Where Used



All bottom-up links are displayed here.
Folder shortcuts in SmarTeam and use in assemblies are also shown.

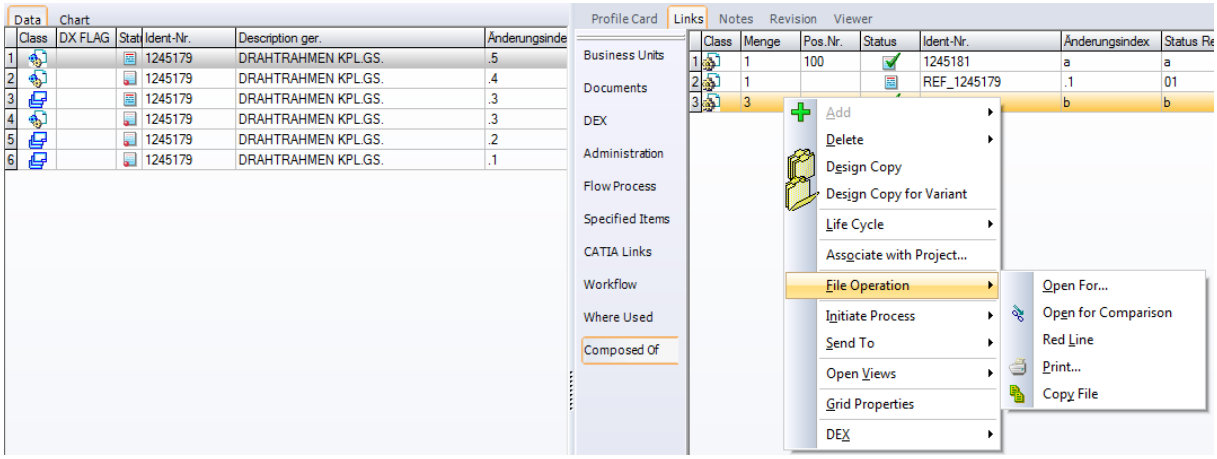
8.6 Composed Of



This heading shows the content of assemblies.
“Composed Of” shows the top-down tree of an assembly.

8.7 General Info

All the links displayed in the individual headings are fully functional, just as they are in the result of a SmarTeam search. Right-click on links to start workflows and select top-down trees and views, for example.

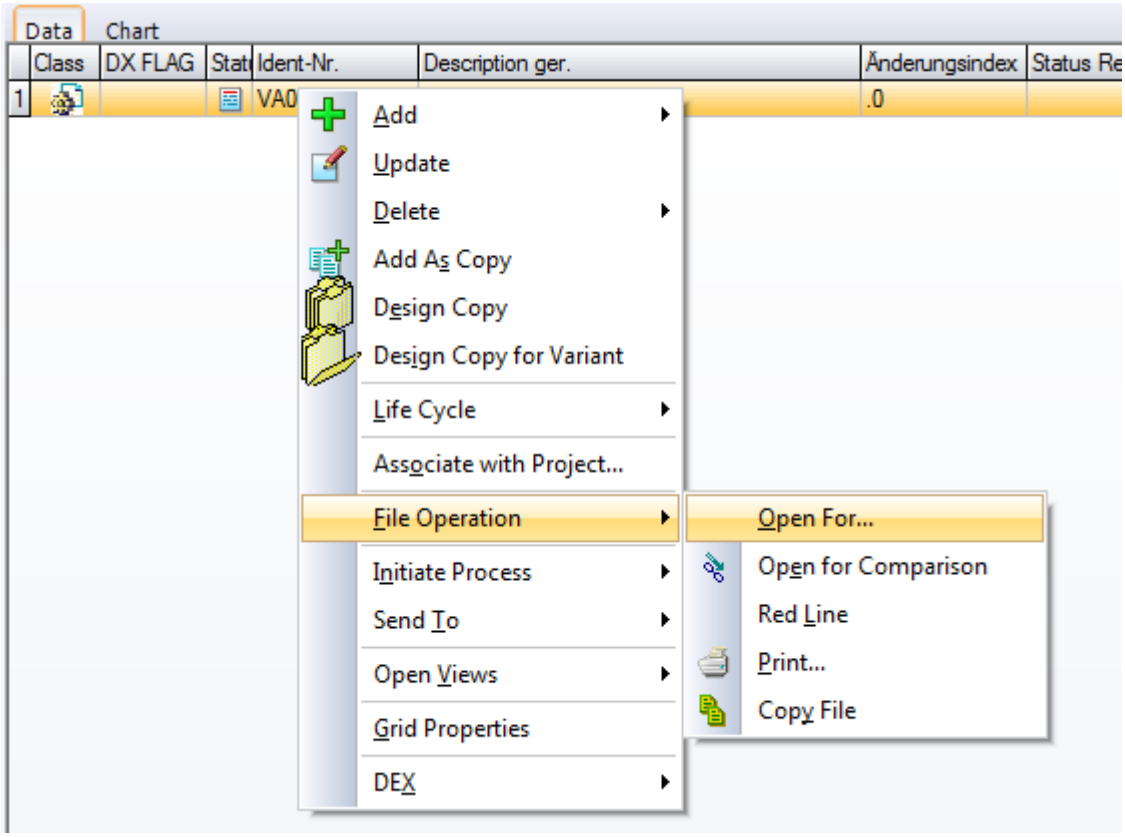


9 File Operation

9.1 Open For

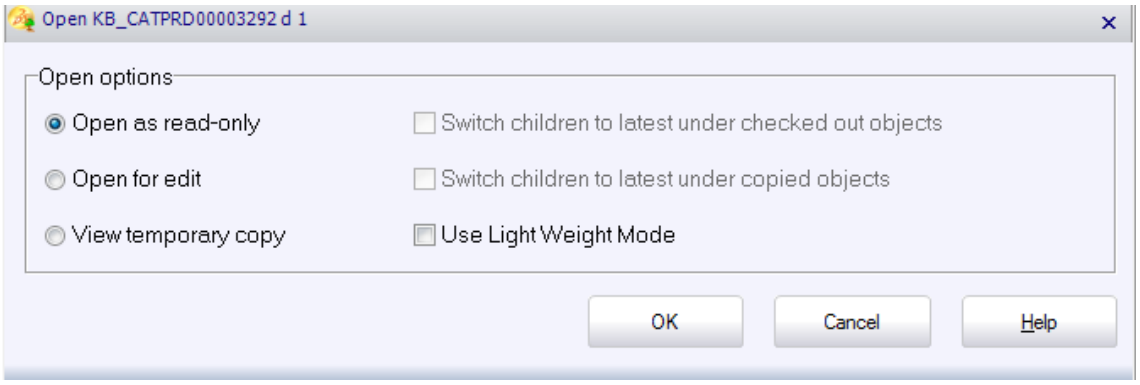
The command “File Operation → Open For…“ is used to open CatiaV5 data from SmarTeam.

Select the part number and right-click on → “File Operation → Open For…“

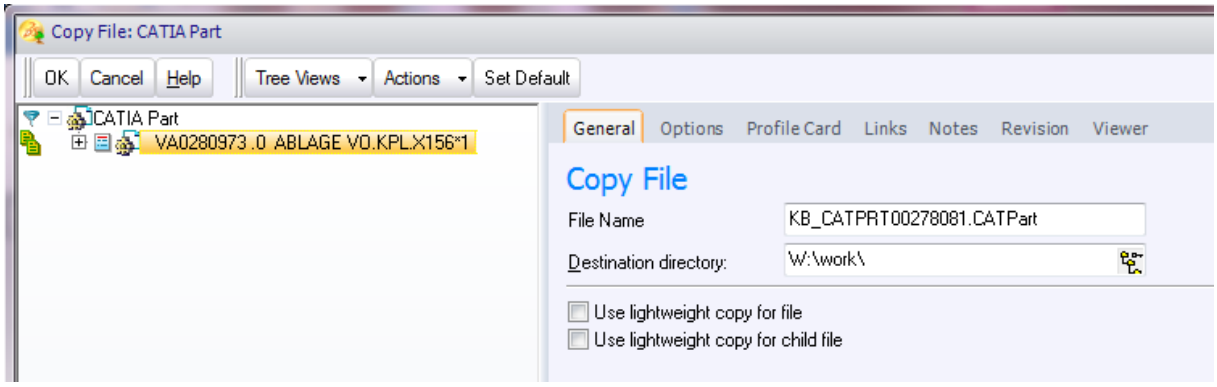


9.1.1 Open as read-only

→ If the file is not to be edited.

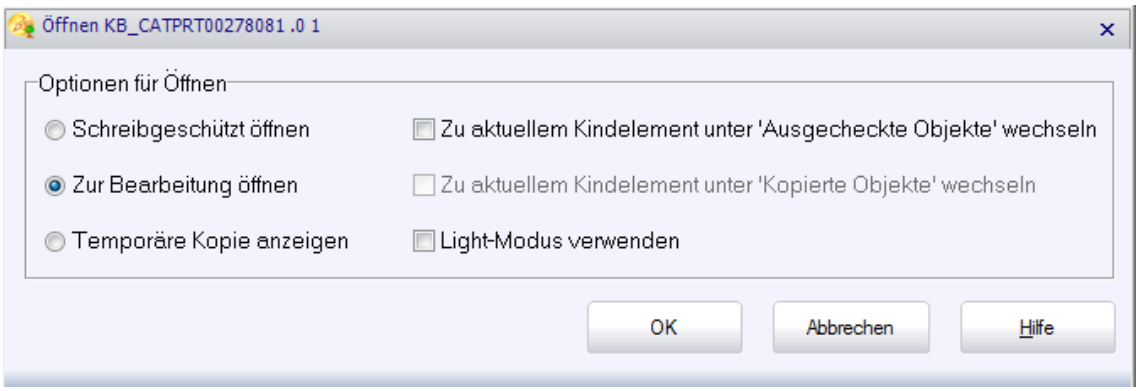


After clicking “OK”, the file is opened in CatiaV5.

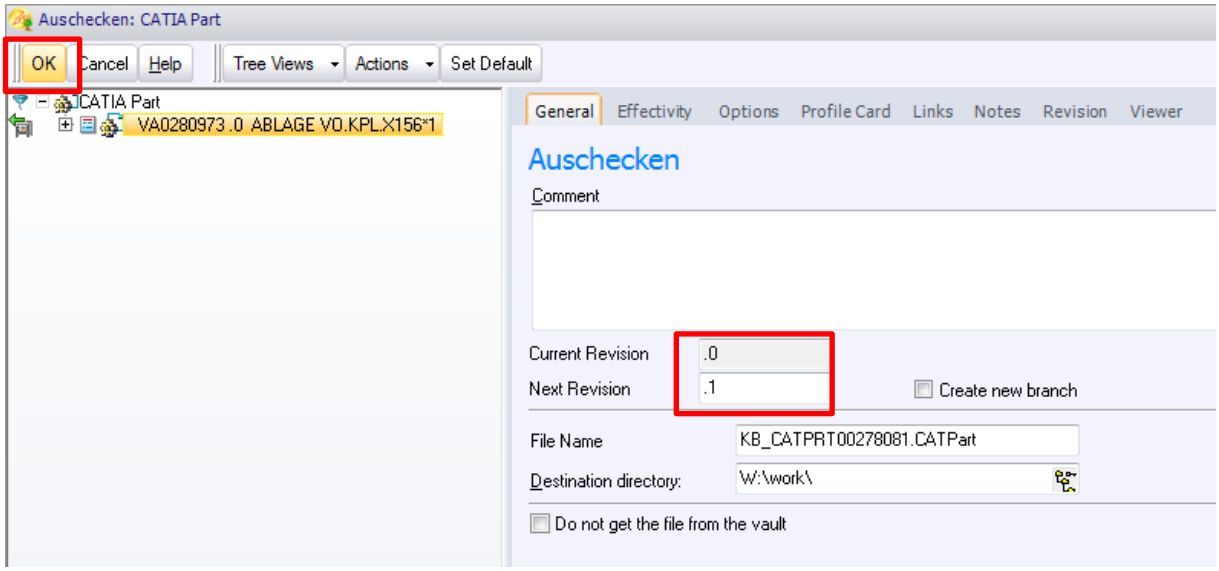


9.1.2 Open for edit

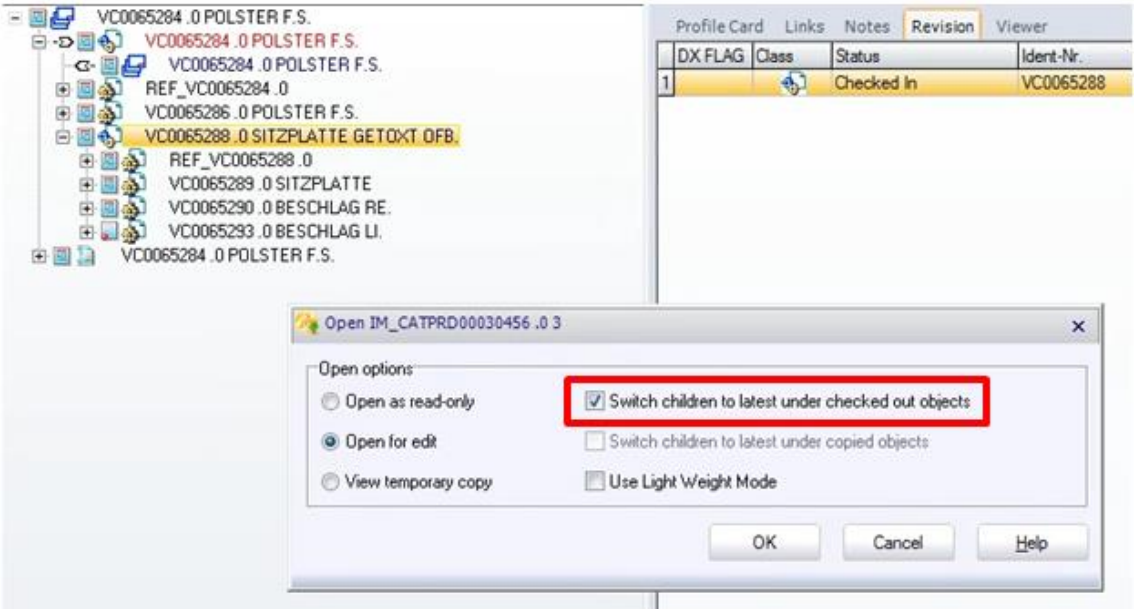
→ If the file is to be edited.



After clicking “OK”, a new version is generated and opened in Catia.

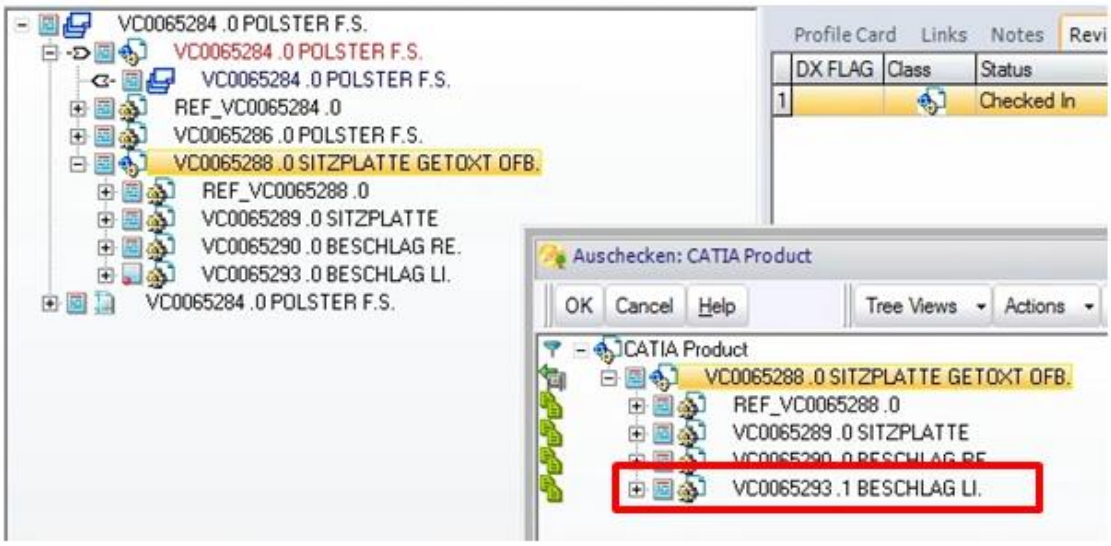


In order to link to the latest version of a part or assembly in the tree structure, the assembly must have been “checked out” and re-assigned in SmarTeam.



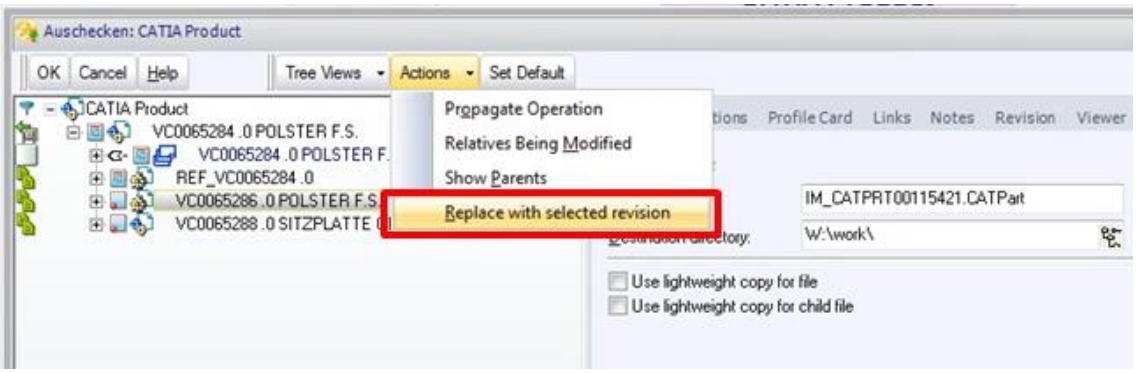
In the case of the “Open for edit” file operation, the following operation can be selected:

“Switch children to latest under checked out objects”

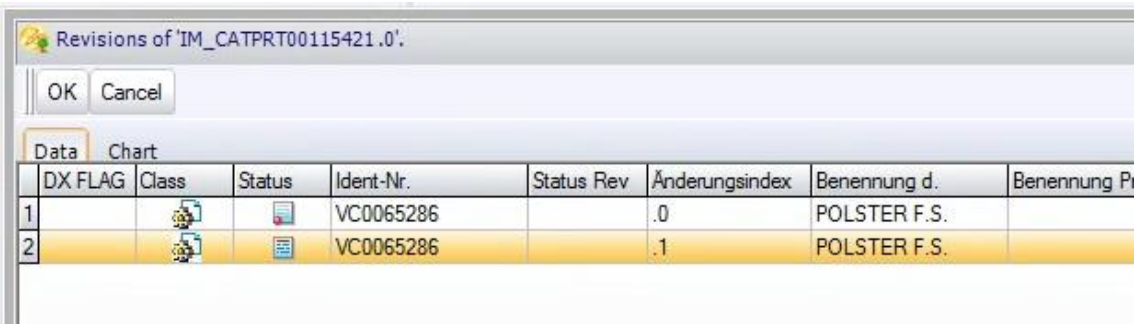


This makes sure that the last version to be checked in is used.

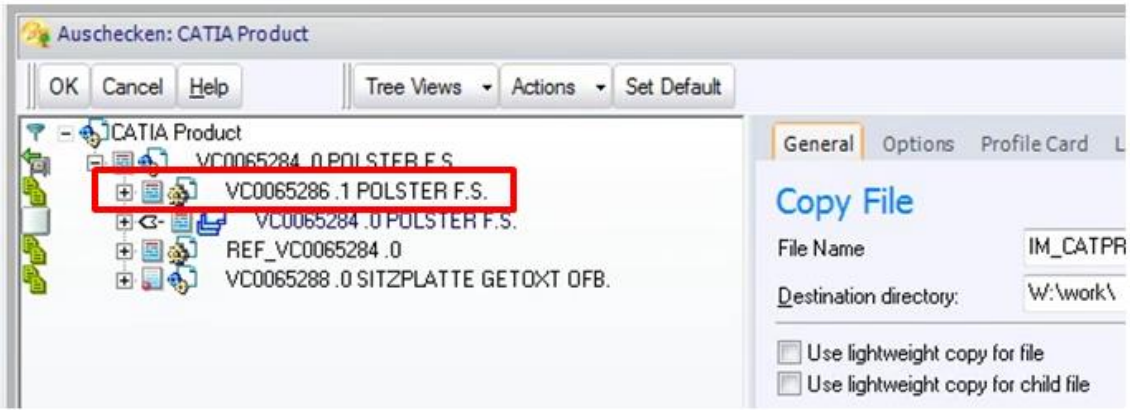
The user can also take advantage of additional functions:



If you select the part or assembly and click “Actions”, in the subsequent window you can choose which version should be linked to.



After clicking OK, the tree structure then looks as follows:

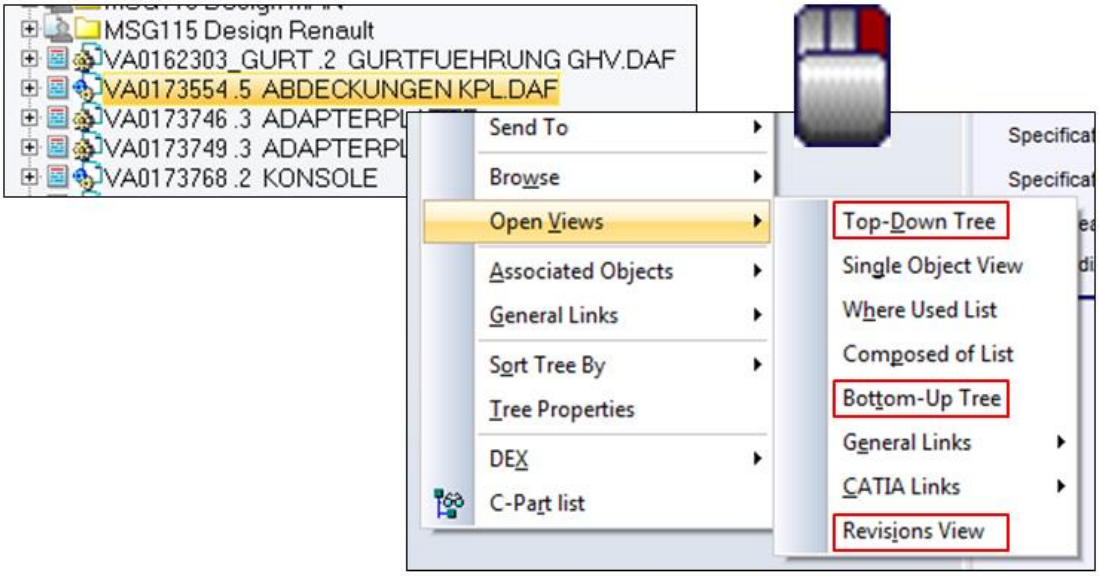


The same procedure is necessary in order to update the associated drawing.

10 Open Views

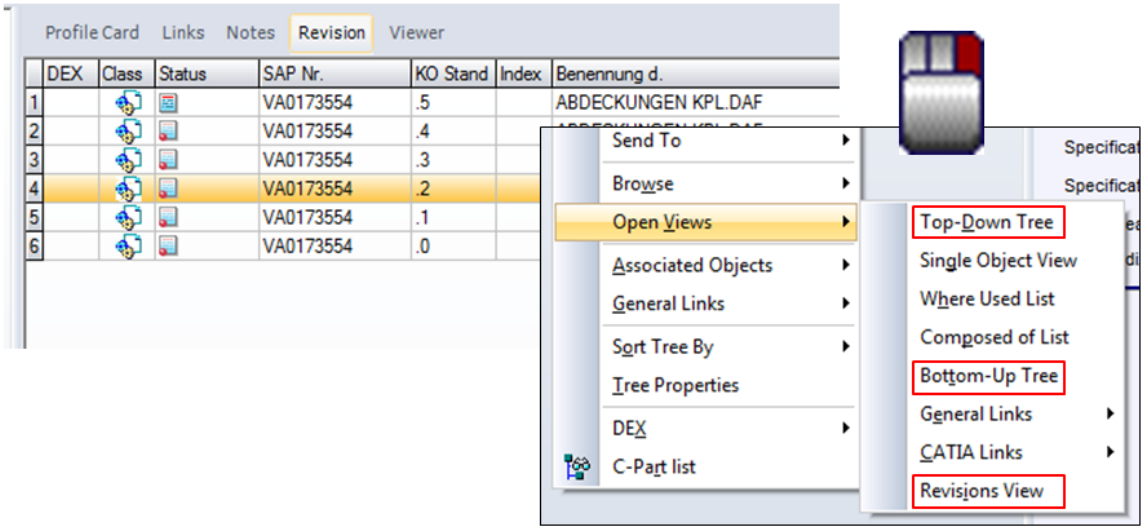
In SmarTeam, select an element with a left click and open the “Open Views” menu with a right click.

This command works from any SmarTeam view.



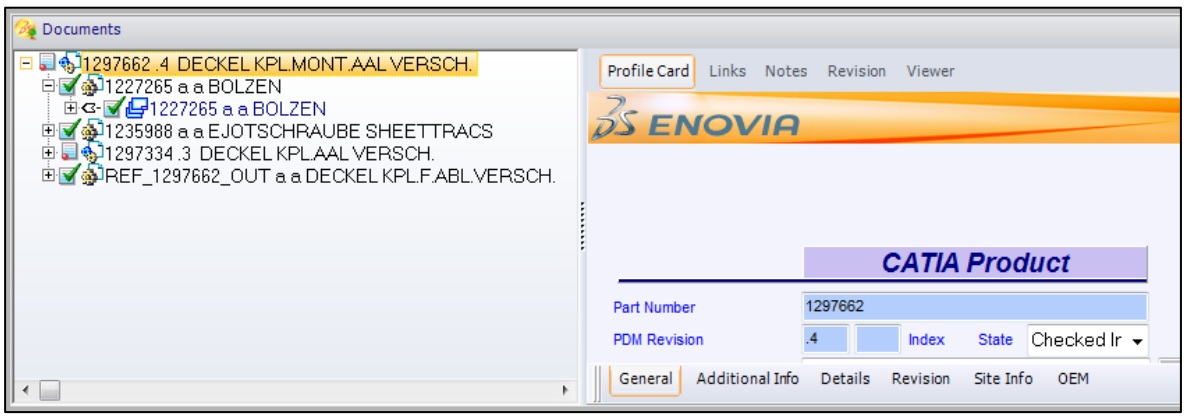
The three most important views are:

- The **“Top-Down Tree”** shows all the links from top to bottom.
- The **“Bottom-Up Tree”** shows the function in higher-level elements.
- The **“Revisions View”** shows all saved versions (statuses).

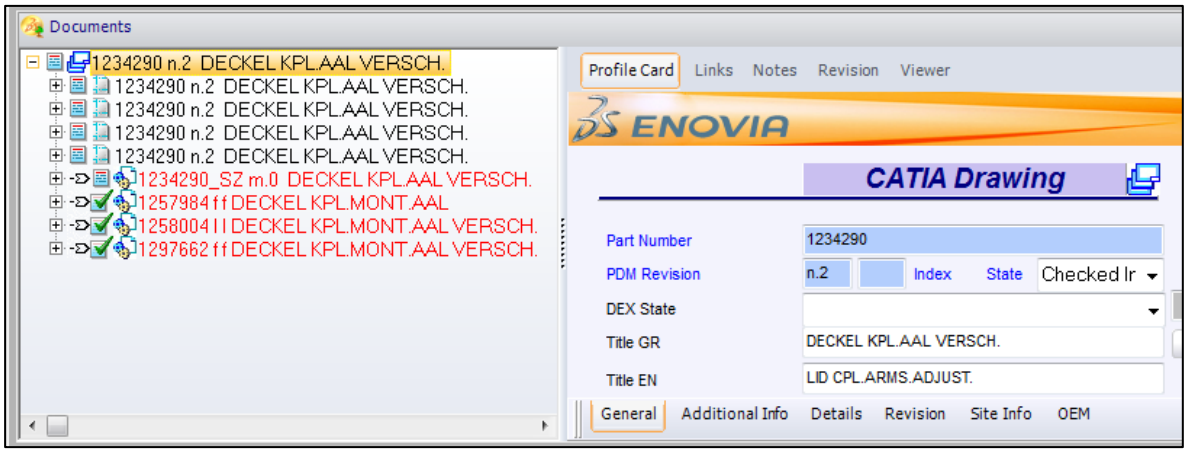


10.1 Top-Down Tree

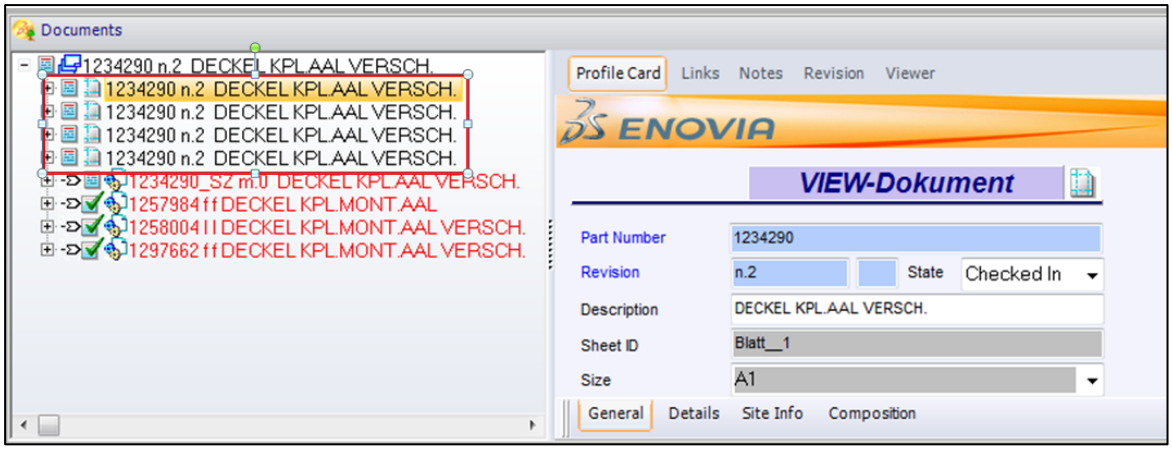
The entire construction tree of the assembly, down to drawing links, is shown on the left hand side.



In the case of a drawing, the links to the parts are shown on the left hand side.

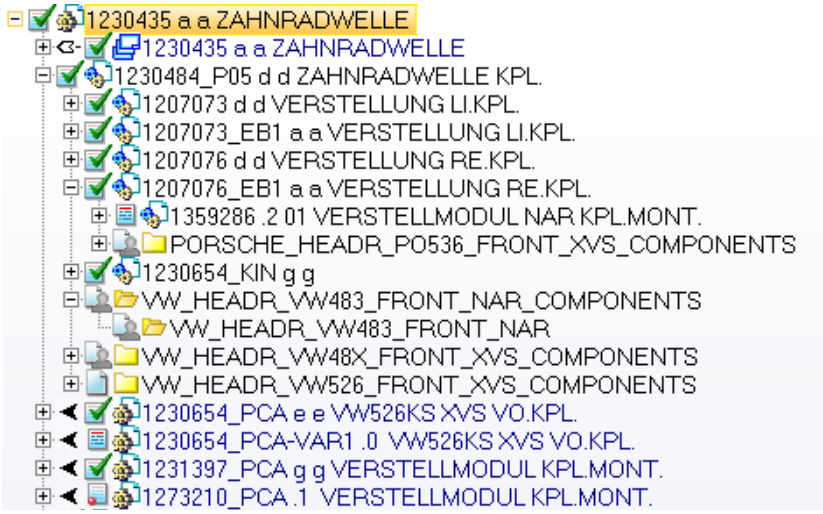


In the case of a multi-sheet drawing, the individual sheets are also displayed.



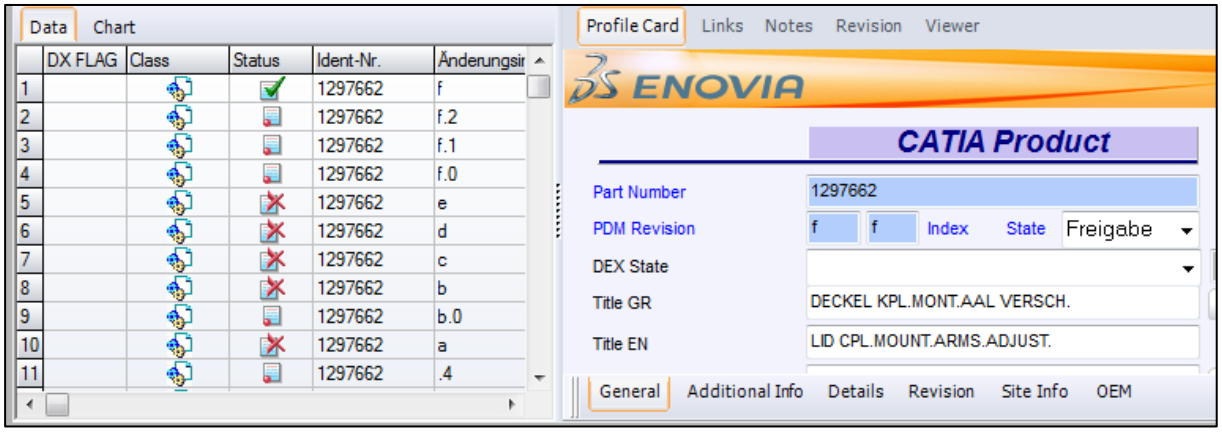
10.2 Bottom-Up Tree

All the assembly links to all higher-level assemblies, parts, drawings and folders are shown.

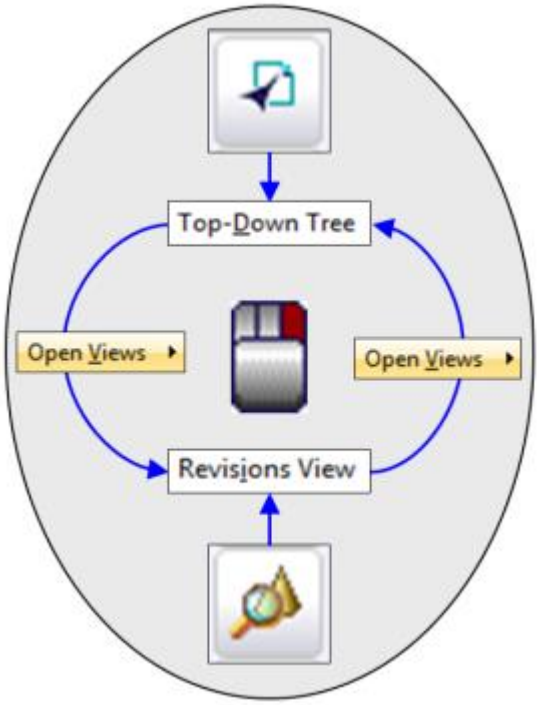


10.3 Versions View

All the versions stored in SmarTeam are shown on the left-hand side.



10.4 Navigate between views



Expanding the folder structure always results in a top-down tree.

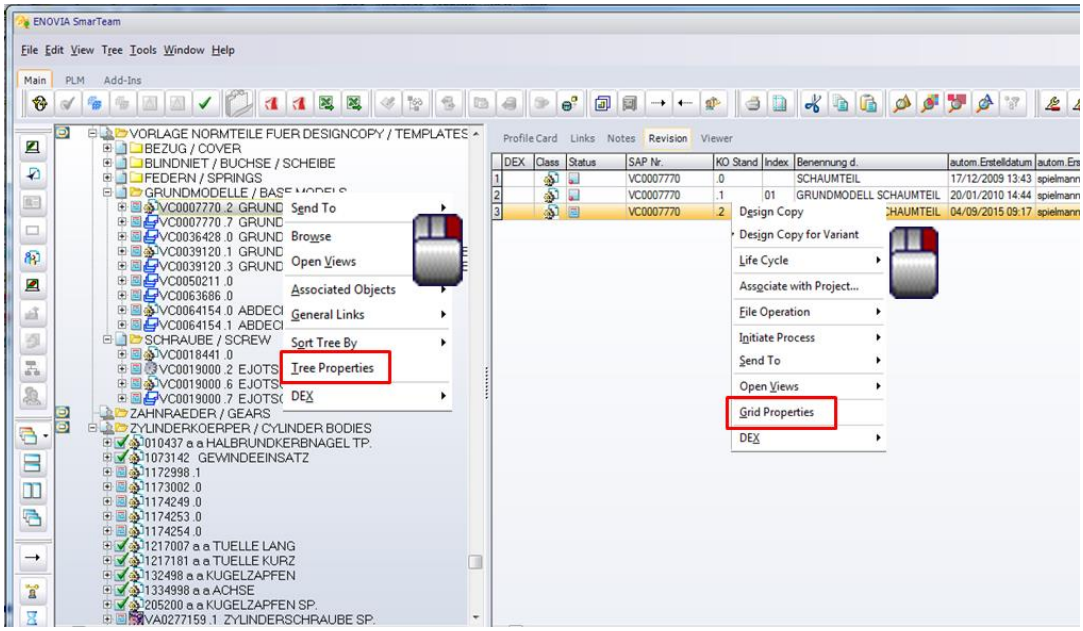
“Right-click → Open Views → Versions View” takes you to the versions.

A search always results in the Versions View.

“Right-click → Open Views → Top-Down Tree” takes you to the assembly structure view.

10.5 Customising views

The information displayed in the different views can be customised by right-clicking.

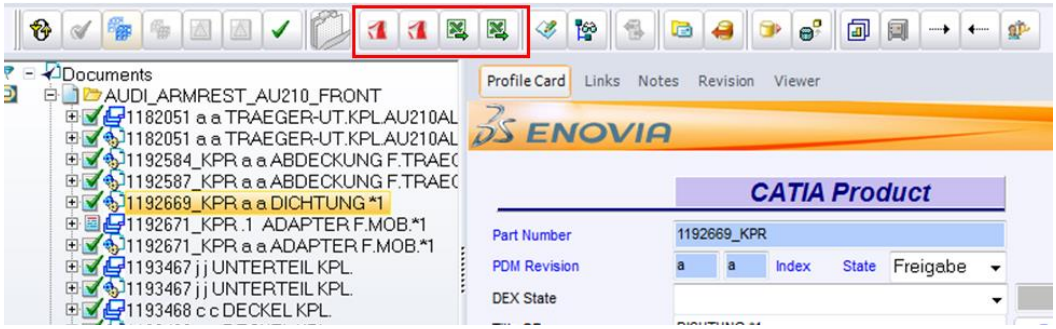


The commands for doing this are “Tree Properties” in the left-hand window and “Grid Properties” in the right-hand window.

11 Bills of Materials (BOM)

11.1 EBOM Engineering Bill of Materials

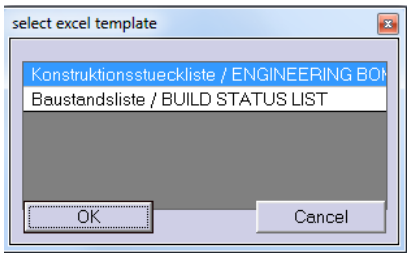
The “Output a Bill of Materials” function, referred to as “BOM” for short, can be found on the TPDM Tools toolbar, it **only** works for assemblies.



The EBOM engineering bill of materials does **not** correspond to the later manufacturing bill of materials.

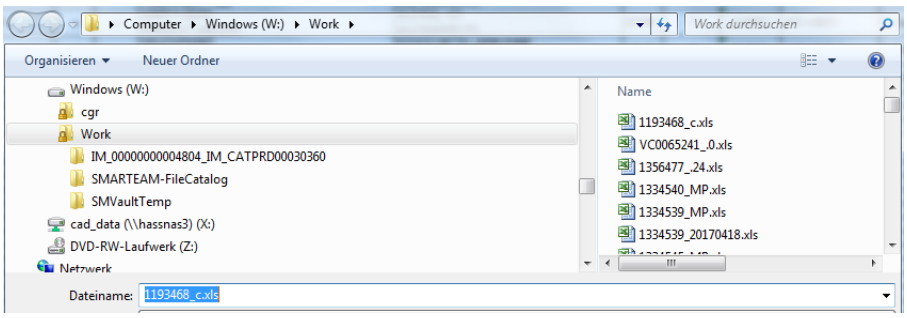
It can be output in the following variants:

Bill of materials PDF single-level	→ only the first level is displayed
Bill of materials PDF multi-level	→ all levels are displayed
Bill of materials XLS single-level	→ only the first level is displayed
Bill of materials XLS multi-level	→ all levels are displayed



Both bill of materials templates (Build Status List and Engineering BOM) are identical apart from three additional columns that are shown in Excel.

The file is opened and automatically saved in w:\work under the SmarTeam part number.



In the SmarTeam BOM, only parts with genuine SAP numbers are output, all other parts (with part number suffixes) are not listed but skipped and the structure below is drawn up.
e.g.: _P01, _TZ, _SZ, _ACP, *EB_*, etc.

Please note the document
"T_015_007_Training_document_Transfer_Engineering_BOM_SmarTeam_SAP" in GBS!

Konstruktionsstueckliste / ENGINEERING BOM
Baustandsliste / BUILD STATUS LIST

X

	Sachnummer	Index	PDM Rev
Sachnummer / PART NUMBER	1193468	c	c
gezeichnet auf / DRAWN ON	1193468	c	c
Benennung / DESCRIPTION	DECKEL KPL.		

Program / CAR NAME
Tier1 Nr. / TIER1 NO.
OEM Nr. / OEM NO.

Aenderungstext / CHANGE DESCRIPTION
Gewicht in kg / WEIGHT: 0,4130
erstellt von / ISSUED BY
Erstelldatum / ISSUE DATE

Stufe DEPTH	Pos	Sachnummer PART NUMBER			gezeichnet auf DRAWN ON			Benennung DESCRIPTION	Zusatzbenennung ADDITIONAL DESCRIPTION	SAP Data Typ	Menge QUANTITY	Gewicht/ kg WEIGHT	Werkstoff MATERIAL	
		Sachnummer PART NUMBER	Index	PDM Rev	Zeichnungsnr. DRAWING NUMBER	Index	PDM Rev						Werkstoff MATERIAL	Normbez. STANDARD DE
1	100	1192578	a	a	1192578	a	a	EINLEGER F.DECKEL INSERT F.LID	AU210AL VO. AU210ARMR.FR.	HALB	1	0,1234	>PC+ABS<	Pulse A35-1(
1	110	1192580	c	c	1192580	c	c	ABDECKUNG DECKEL COVER LID	AU210AL VO. AU210ARMR.FR.	ZCHN	1	0,1307	>PC+ABS<	Pulse A35-1(
1	120	1192582	a	a	1192582	a	a	SCHAUMTEIL F.DECKEL FOAM PART F.LID	AU210AL VO. AU210ARMR.FR.	HALB	1	0,0521	>PUR<	Schaum / FO
1	130	1192588	a	a	1192588	a	a	GRIFF F.ENTR. HANDLE F.UNLOCK.	AU210AL VO. AU210ARMR.FR.	ZCHN	1	0,0053	>PC+ABS<	Pulse A35-1(
1	140	807691			807691	a	a	DRUCKFEDER COMPRESSION SPRING	F01F02 MITTELARMLEHNE F01F02 CENTER ARM REST	HALB	1	0,0000	X10CrNi18-8	1.4310 EN 102
1	170	1052644						PUFFER SCHWARZ BUMPER BLACK		HALB	4	0,0001	>EPDM< 62±5 Shore A	ISO 1629

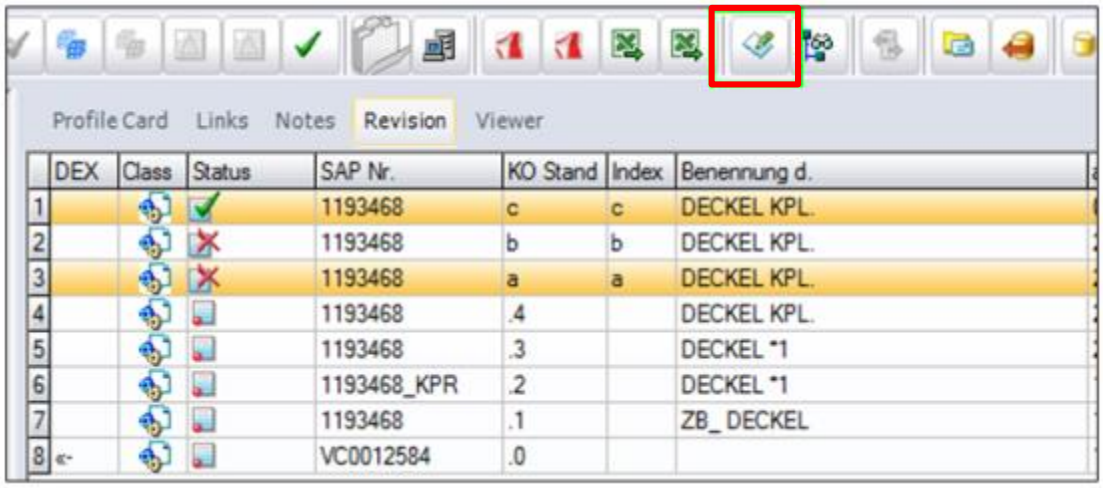
Explanation of the individual columns:

A	Depth
B	Position
C	3D SmarTeam and SAP number
D	3D SmarTeam and SAP index
E	3D SmarTeam PDM revision
F	2D SmarTeam and SAP number
G	2D SmarTeam and SAP index
H	2D SmarTeam PDM revision
I	SmarTeam and SAP description
J	Additional description
K	Customer description (if assigned, this is hidden by default)
L	SAP Material Type from the 3D
M	Quantity
N	Weight
O	Material
P	Standard designation
Q	Dimensional standard
R	Comments
S	Comments that can be added manually
T	Comments that can be added manually
U	Comments that can be added manually

11.2 BOM comparison

SmarTeam also offers the option of comparing two assemblies. This function is called “BOM comparison”.

Select two assemblies in the Versions View and click the “BOM comparison” icon.



	DEX	Class	Status	SAP Nr.	KO Stand	Index	Benennung d.
1				1193468	c	c	DECKEL KPL.
2				1193468	b	b	DECKEL KPL.
3				1193468	a	a	DECKEL KPL.
4				1193468	.4		DECKEL KPL.
5				1193468	.3		DECKEL *1
6				1193468_KPR	.2		DECKEL *1
7				1193468	.1		ZB_ DECKEL
8				VC0012584	.0		

SmarTeam outputs a comparison of both statuses.

STÜCKLISTENVERGLEICH

Benennung / description			DECKEL KPL. LID CPL.			Benennung / description			DECKEL KPL. LID CPL.		
Teilenummer / part number			1193468			Teilenummer / part number			1193468		
Version / release			a			Version / release			c		
Stufe depth	Pos pos	Teilenummer part number	Benennung description	Menge quantity	Version release	Stufe depth	Pos pos	Teilenummer part number	Benennung description	Menge quantity	Version release
1	100	1192578-A	EINLEGER F.DECKEL	1	a	1	100	1192578-A	EINLEGER F.DECKEL	1	a
1	110	1192580-A	ABDECKUNG DECKEL	1	a	1	110	1192580-C	ABDECKUNG DECKEL	1	c
1	120	1192582-A	SCHAUMTEIL F.DECKEL	1	a	1	120	1192582-A	SCHAUMTEIL F.DECKEL	1	a
1	130	1192588-A	GRIFF F.ENTR.	1	a	1	130	1192588-A	GRIFF F.ENTR.	1	a
1	140	807691	DRUCKFEDER	1		1	140	807691	DRUCKFEDER	1	
1	170	1052644	PUFFER SCHWARZ	2		1	170	1052644	PUFFER SCHWARZ	4	

12 Design Copy

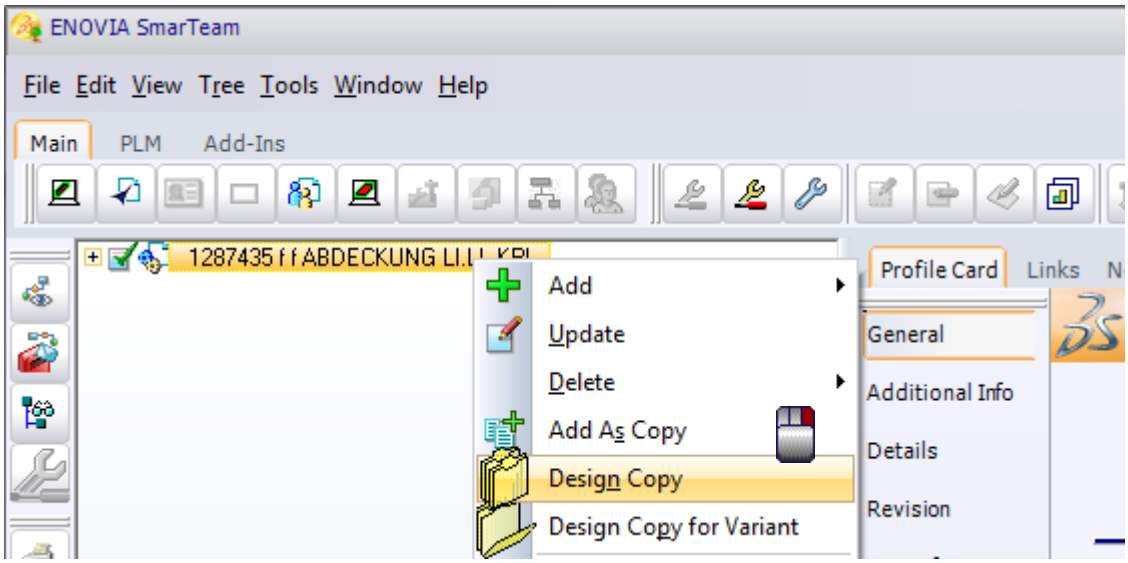
The “Design Copy” function can be used to generate a new drawing, assembly or part, with the aid of SmarTeam, based on a template.

IMPORTANT NOTE: The newly generated object is independent of the template!

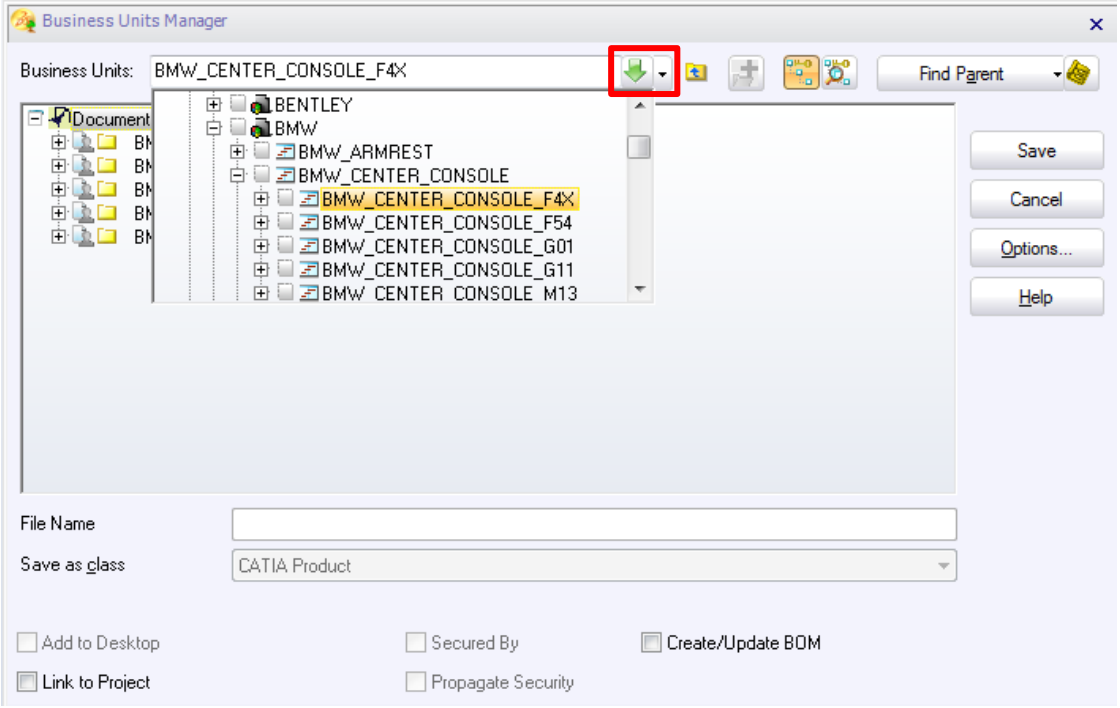
Right-click to highlight the assembly structure to be copied and select “Copy Design” in the context menu.

Design Copy can also be launched from the assembly, part or drawing.
It is preferable to launch it from the assembly where there are links right down to the drawings of individual parts.

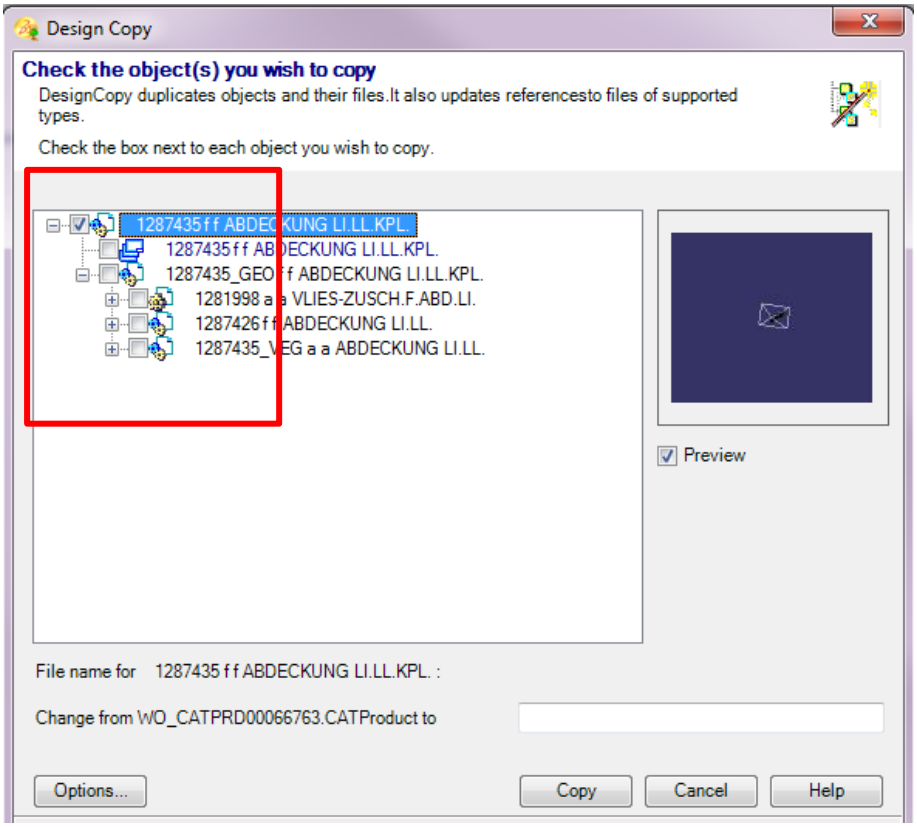
Select the part number and right-click to select “Design Copy”.



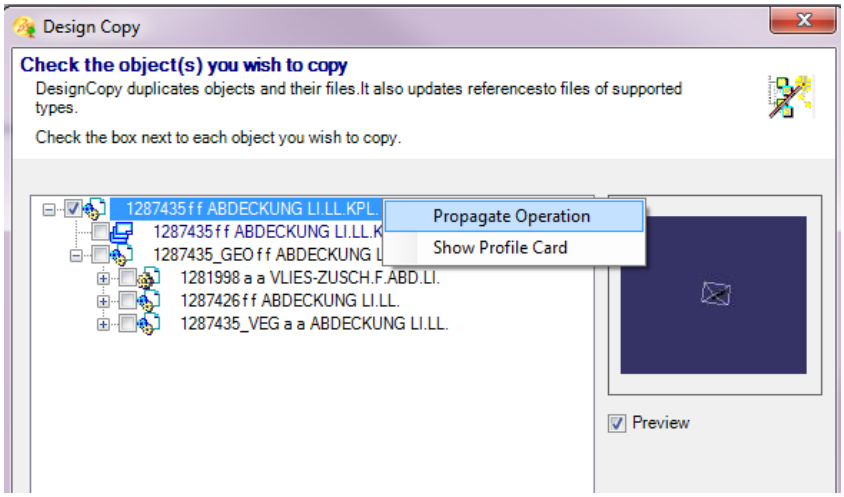
Select the project and folder where the file(s) is/are to be saved.



In the subsequent window, highlight the objects that are to be copied.



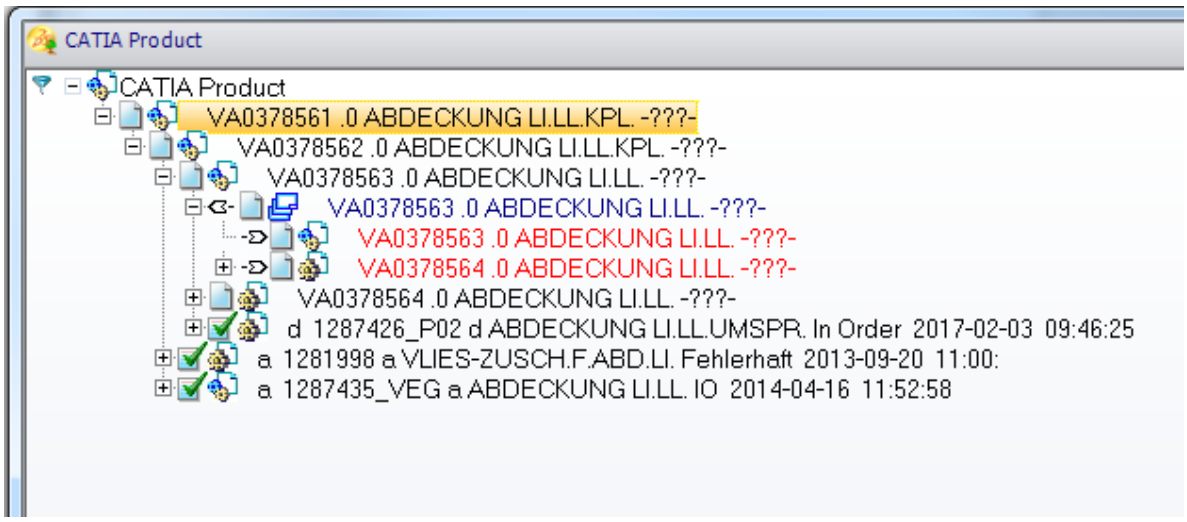
You can select all the parts of an assembly with a single mouse click by right-clicking→ Propagate Operation.



This function performs the following actions:

- Duplicate the meta data
- Duplicate the files
- Duplicate the links
- New trial numbers for copied objects
- Change the file references

The following assembly structure is created, the highlighted part numbers are given a new V-number (trial number) shown in SmarTeam.

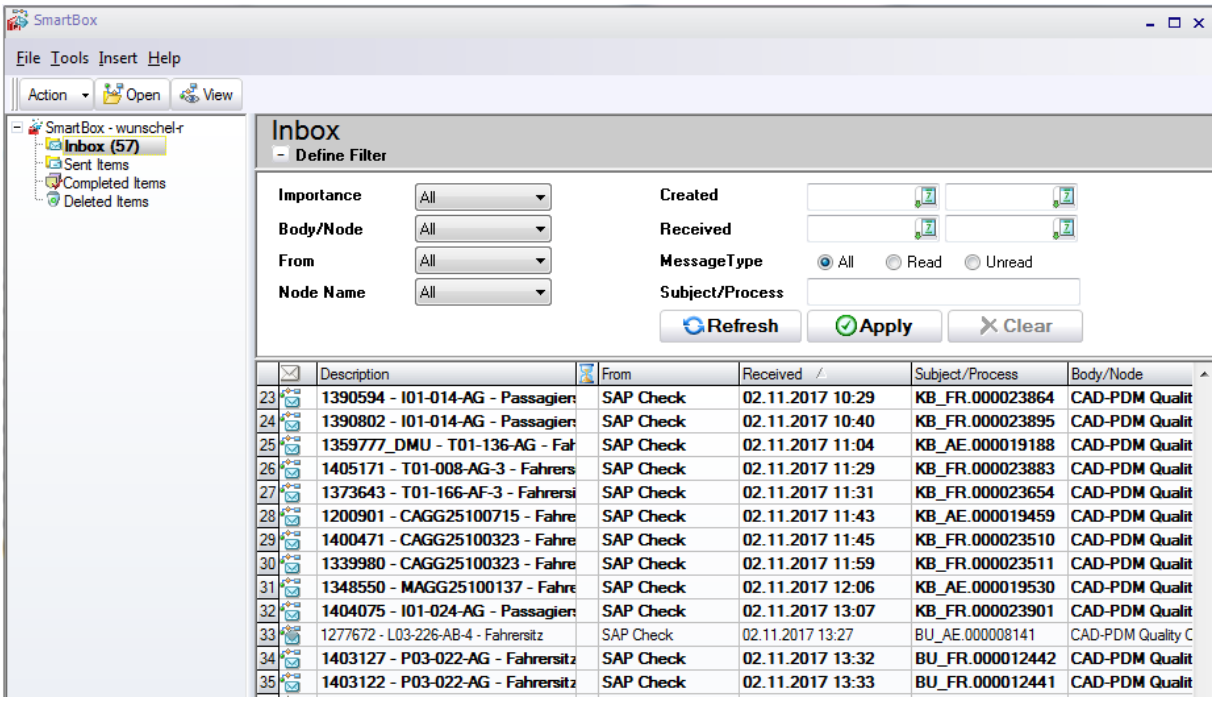


13 Workflows

13.1 Smartbox

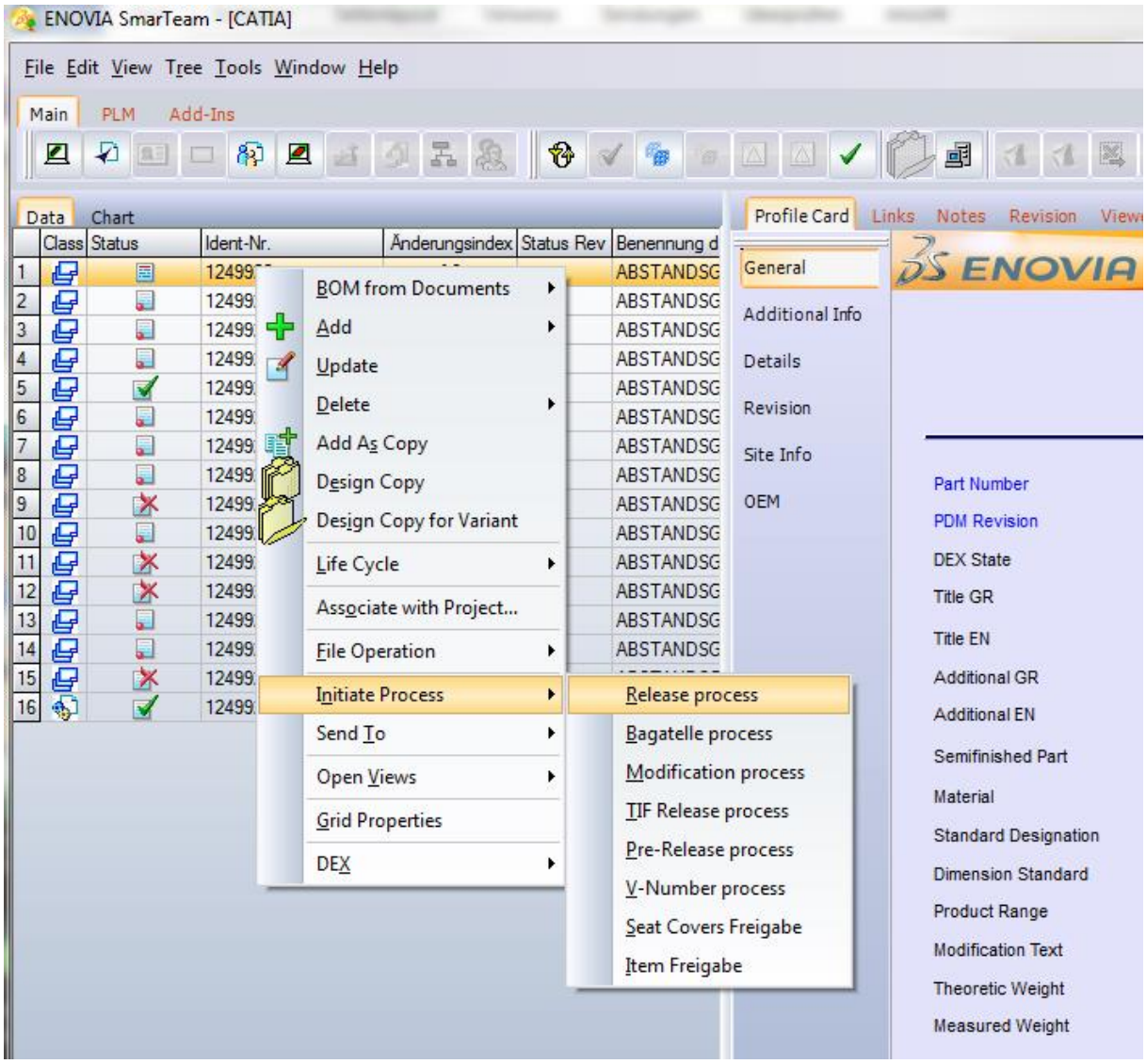
When a workflow is initiated, it automatically appears in the initiator's Smartbox. Double-clicking the relevant number opens the workflow.

- All part numbers needing attention are listed in the Smartbox.
- The respective part number can be opened, accepted/rejected or marked as read by right-clicking.
- Press key F5 on the keyboard to update/refresh the Smartbox.



13.2 Initiating a workflow

Search for previously generated part numbers, highlight, right-click, initiate process, then select the process e.g. Modification process or Release process, etc.



13.3 Pre-production workflow

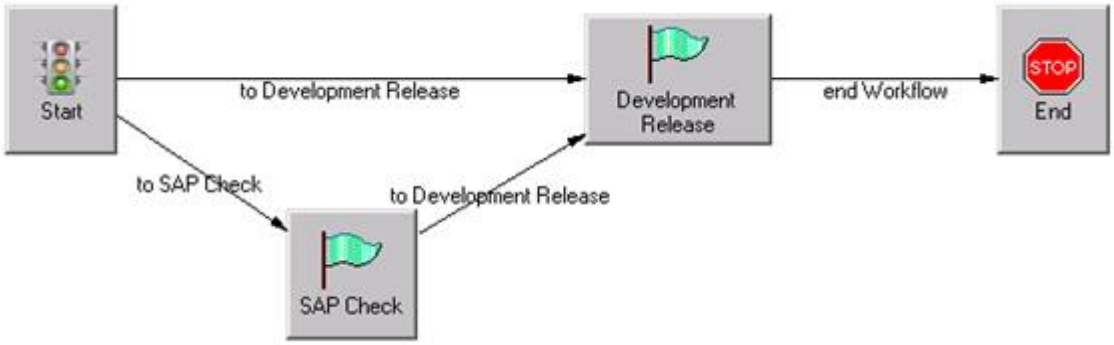
During the development process, selected statuses of a drawing, part or assembly can be generated and “frozen” using the pre-production workflow.

Objects that are linked by the pre-production workflow are given a numeric index that is also mapped to the drawing title block, e.g. numeric index 01 for the first run of the pre-production workflow, index 02 for the second run, and so on.

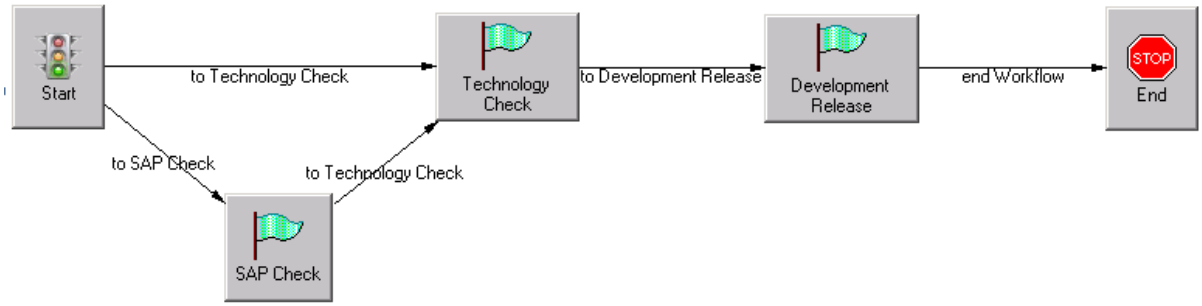
The CAD drawing is stamped with the words “Preliminary Design”

When the workflow is completed, basic data and documents (TIF, JT) are transferred to SAP with the status “VL” (Pre-release).

- Structure of the pre-production workflow **without** Technology Check:



- Structure of the pre-production workflow **with** Technology Check:



13.4 Release workflow

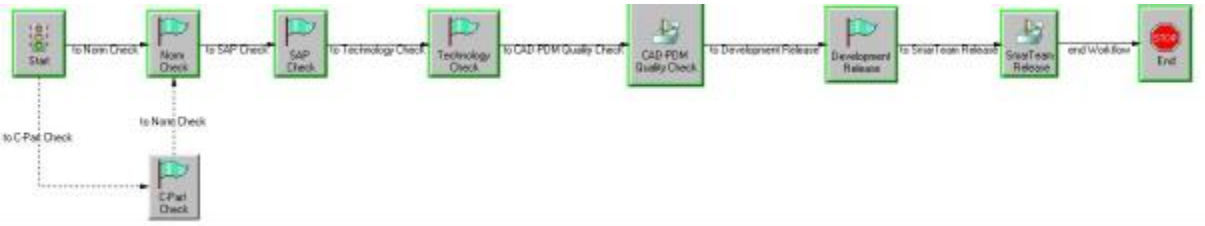
The release workflow is a multi-stage document workflow dealing with new documents and/or the versioning of carry over documents (e.g. Euclid etc.).

In order to feed a new (not yet released) document into the release workflow, a user (e.g. technical draughtsman) must open the release process in SmarTeam.

The individual release steps (stations) are automatically controlled by forwarding to "Norm Check”.

- Release workflow structure and sequence with C-Part Check **without** Technology Check:

- Release workflow structure and sequence with C-Part Check and **with** Technology Check:



After release of the workflow, transfer to SAP-DVS (TIF, JT) takes place.

Basic data 1 and 2 are populated with the information from the Profile Card and a CAD indicator is set, the transferred data can then only be modified in SmarTeam.

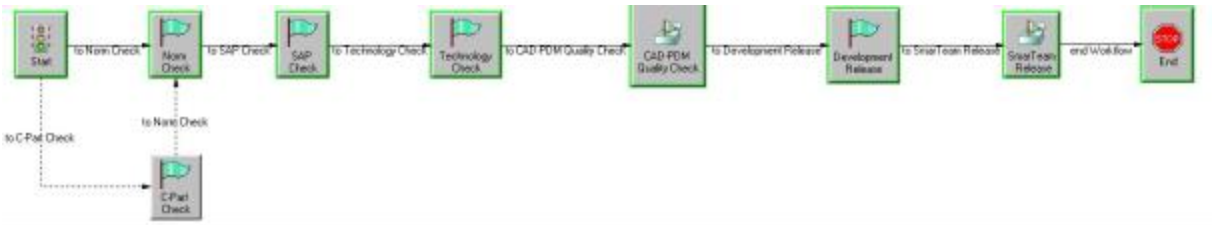
13.5 Modification workflow

In order to modify a document (drawing and/or 3D model) that has already been released, a user must initiate a SmarTeam modification process.

The modification workflow process is patterned on the release workflow.

- Modification workflow structure and sequence with C-Part Check **without** Technology Check:

- Modification workflow structure and sequence with C-Part Check and **with** Technology Check:



After release, the previous status is set as invalid (UN) in SAP-DVS (TIF, JT) and the new index is transferred or released.

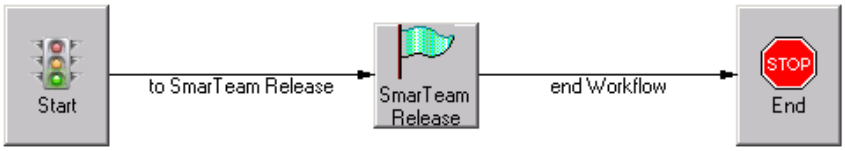
13.6 Bagatell process

This simplified workflow is used to modify/adapt documents that have already been released.
This shortened modification process is used to make only **“minor changes”** to documents.

The term **“minor change”** can refer to, for example:
a forgotten or additional publication in an individual part of a CAD model or similar.

Such a change **must not have any impact** on the **meta data** (mapped to SAP) or change the **volume in 3D**!

- Structure of the bagatell process workflow:

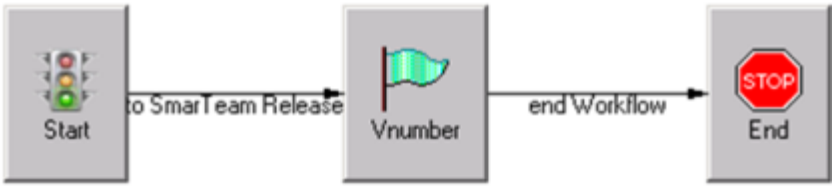


After release of the workflow, the most up-to-date data (TIF, JT) is saved to SAP-DVS.

A bagatell process is not an alternative to an ECM process (modification workflow)!

13.7 V-number (trial numbers) workflow

Structure of the V-number workflow (**ONLY** V-numbers are permitted)



It is not possible to mix an assembly with 7-digit part numbers and V-numbers.

The V-number workflow is only possible for complete V-number parts, assemblies and drawings.

The V-numbers in question are given a numeric release index.

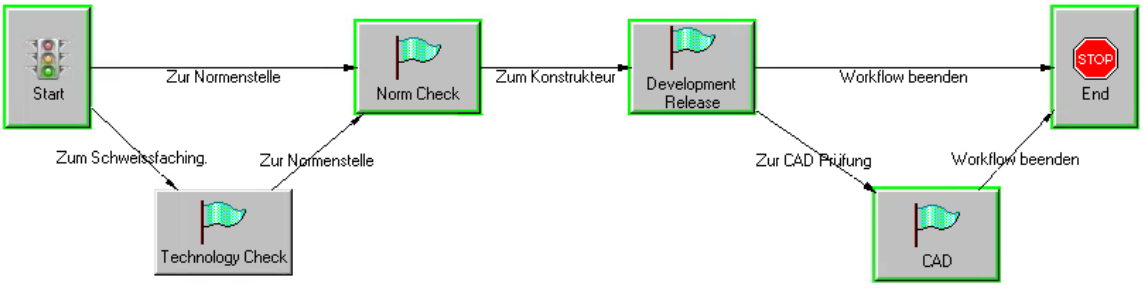
13.8 TIF release

- Release and modification of drawings from Euclid, CatiaV4 and AutoCad and/or manually generated drawings.
- Release and modification of prototype approvals in TIF format (creation in Microsoft Office).
- Release and modification of BOMs in TIF format (creation in MS Word or MS Excel), minor modification of BOMs (without index increase).

See also the “Tif2SmarTeam” instructions on the GRAMMER intranet:

http://intranet.grammer.com/fileadmin/user_upload/intranet/de/Organisationseinheiten/FuE/Produktentwicklung/Schulungsdoku_Feb_2014/Smarteam_Schulungsut/Tiff2Smarteam_Beschreibung_de_rev04_01_lesezeichen.pdf

- Structure and sequence of TIF workflow with/without Technology Check:



13.9 SAP interface

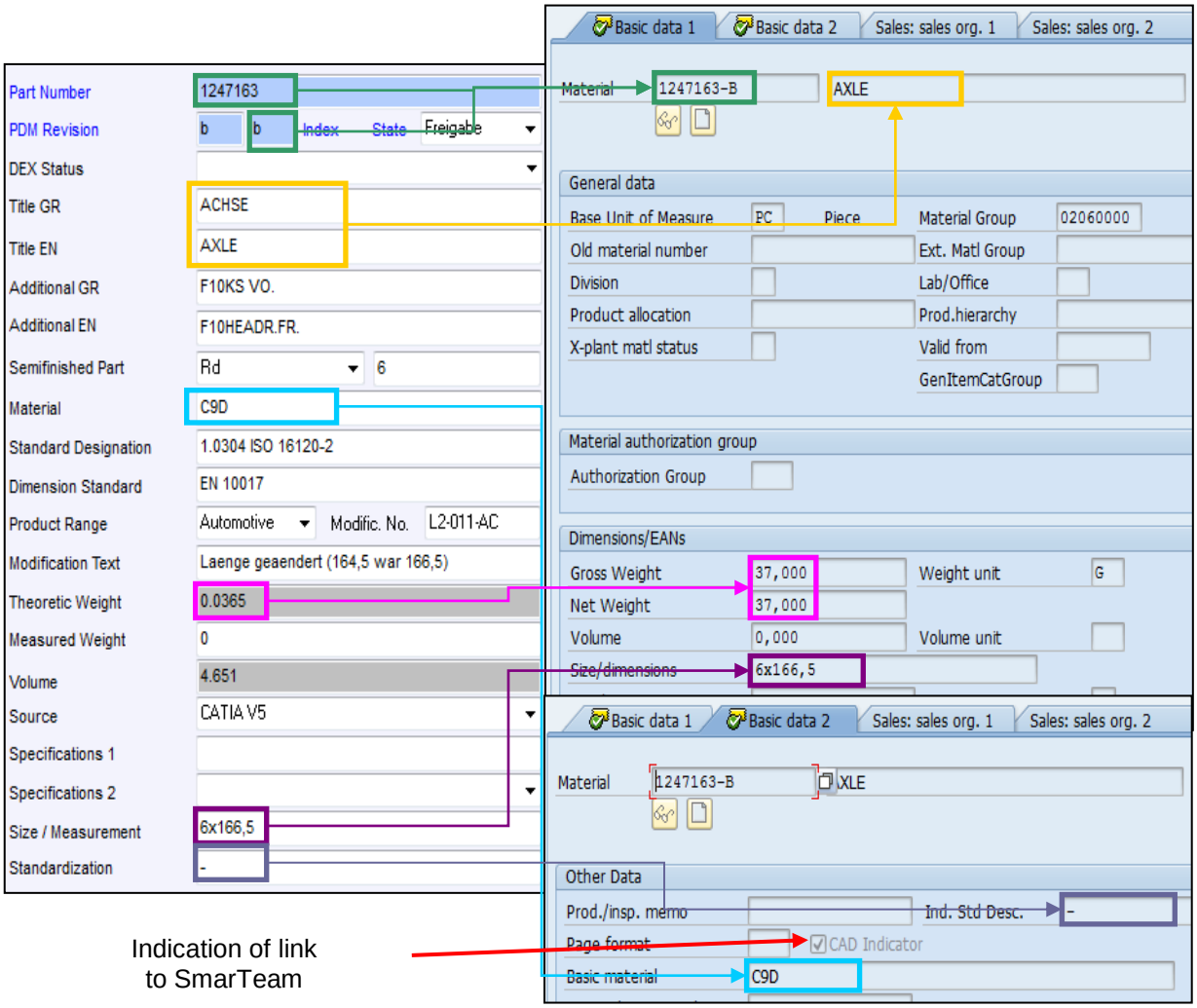
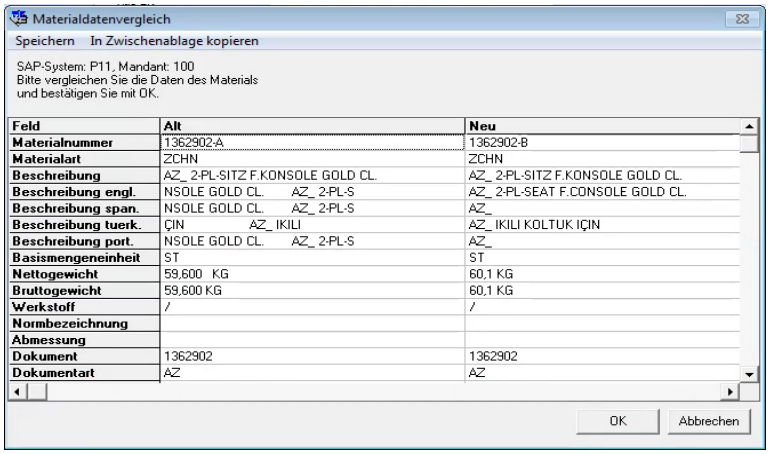


Diagram showing which data from the Profile Card is transferred to SAP. This takes place for all pre-production, release and modification processes. When workflows are edited, information is displayed in the form of a basic data comparison.



Materialdatenvergleich

Speichern In Zwischenablage kopieren

SAP-System: P11, Mandant: 100
Bitte vergleichen Sie die Daten des Materials
und bestätigen Sie mit OK.

Feld	Alt	Neu
Materialnummer	1362902-A	1362902-B
Materialart	ZCHN	ZCHN
Beschreibung	AZ_ 2-PL-SITZ F.KONSOLE GOLD CL.	AZ_ 2-PL-SITZ F.KONSOLE GOLD CL.
Beschreibung engl.	NSOLE GOLD CL. AZ_ 2-PL-S	AZ_ 2-PL-SEAT F.CONSOLE GOLD CL.
Beschreibung span.	NSOLE GOLD CL. AZ_ 2-PL-S	AZ_
Beschreibung tuerk.	ÇIN AZ_ IKILI	AZ_ IKILI KOLTUK ÇIN
Beschreibung port.	NSOLE GOLD CL. AZ_ 2-PL-S	AZ_
Basismengeneinheit	ST	ST
Nettogewicht	59,600 KG	60,1 KG
Bruttogewicht	59,600 KG	60,1 KG
Werkstoff	/	/
Normbezeichnung		
Abmessung		
Dokument	1362902	1362902
Dokumentart	AZ	AZ

OK Abbrechen

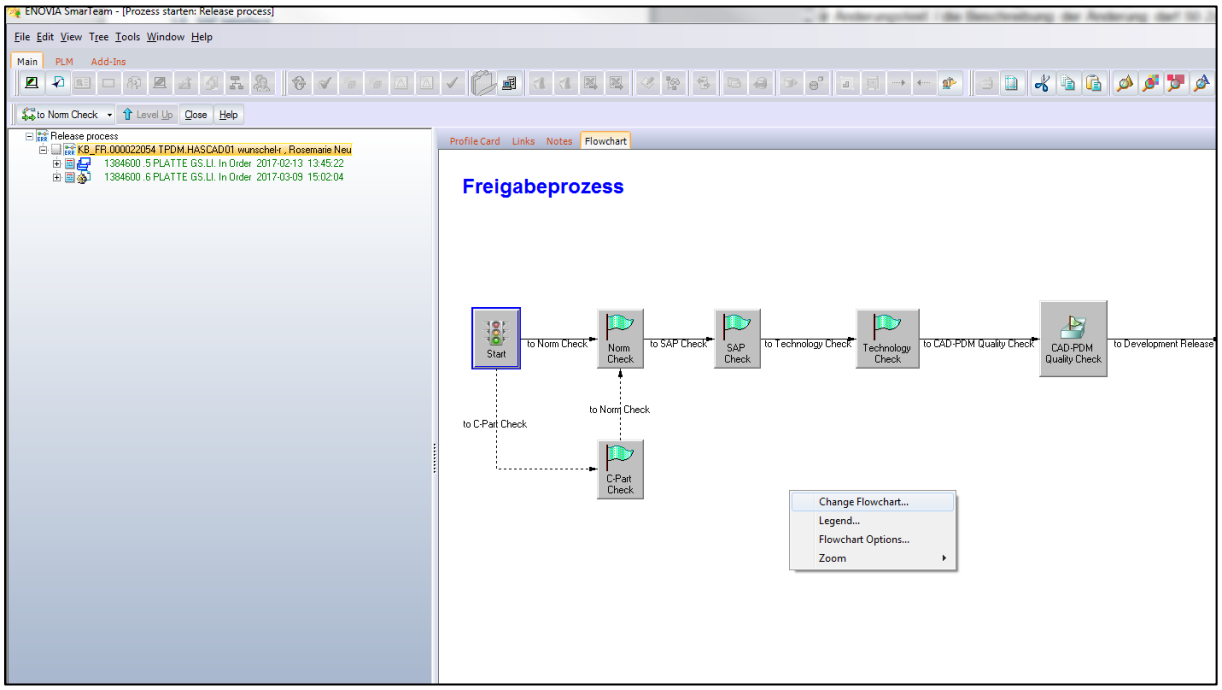
13.10 Workflows with particular release points (welding, bonding)

For welded parts (description must include “GS” or “GK”), the relevant specialist welding engineer must be included in the workflow as a technology check (name must be entered by the relevant workflow initiator).

If a workflow began with a non-welded part/assembly and a welded part/assembly is subsequently added to the workflow, the workflow can be renamed as a welding workflow at the start node.

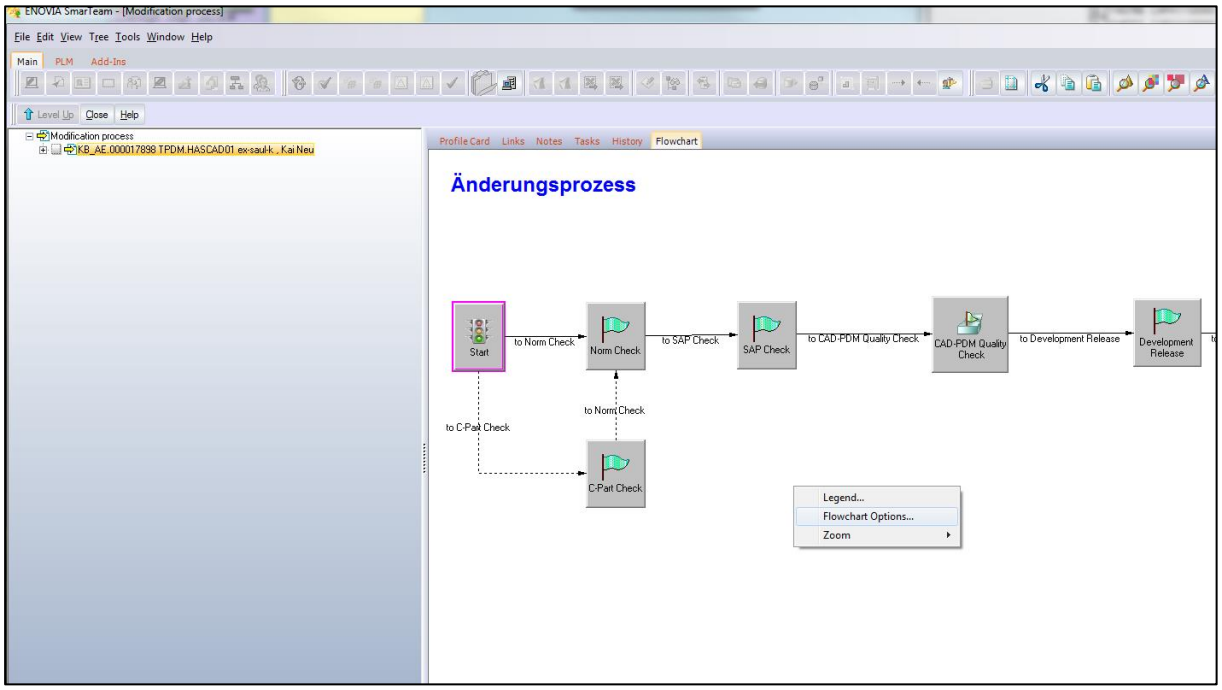
This is only possible at the start note as long as the workflow has not been sent.

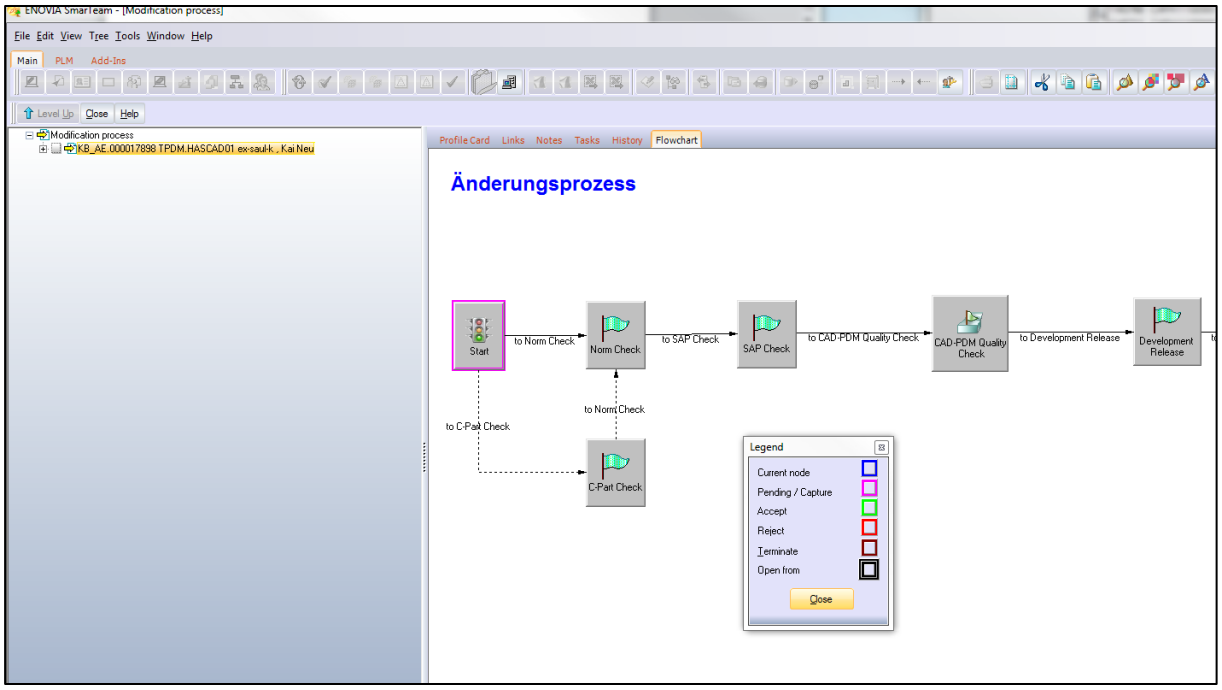
To do this, right-click on the white background area under the Flowchart tab. Then click on Change Flowchart:



13.11 History for frame colour during a workflow

A legend is shown to explain the respective frame colour during the workflow.
To do this, right-click on the white background area under the Flowchart tab. Then click on Legend:





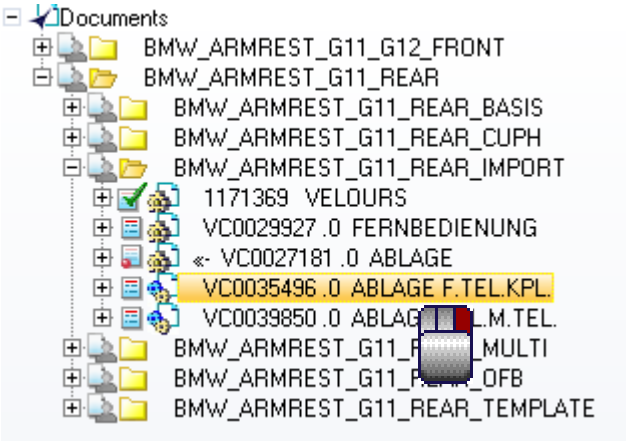
14 DEX

14.1 What is DEX?

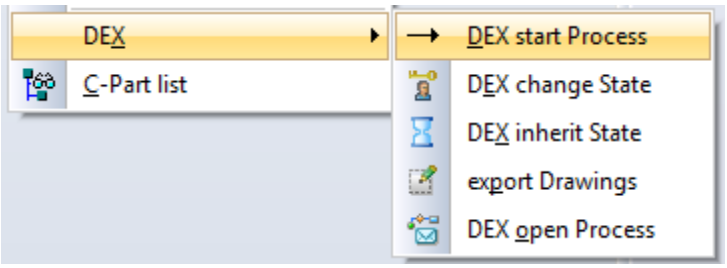
DEX is a tool that is fully integrated into SmarTeam. This tool can export data in various formats and import native (Catia/NX/JT) data.

14.2 DEX in SmarTeam

The function buttons are only activated when the document view is called up and a document selected.



DEX can also be launched by using the right-click menu.

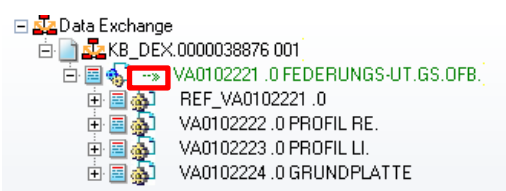
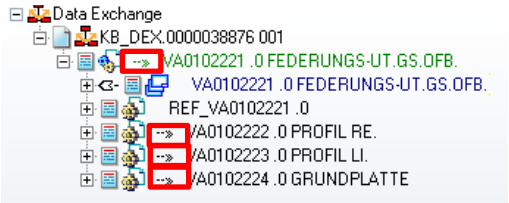


Launch DEX process		Launches the data exchange process
Change DEX status (DEX export only)		The data exchange status of a file can be changed here. The following status definitions are possible, among others: „ „ (no sign in front of the part number) data is sent for information only. This data cannot be reimported! -- For drawings only, so that they can also be read out. This icon is automatically deleted after export (only if 3D and 2D have the same part number). → Data is read out for versioning and locked for internal processing. Data is versioned when imported.
Inherit DEX status		Transfers (inherits) DEX status from the highest-level assembly to subordinate documents (parts and assemblies only) !!! In the case of released data, a modification process must previously have been initiated!!!
Export drawings		All drawings of single parts and subassemblies are also exported for information only (are identified by “--”). !! Only works if the assembly is highlighted!!!
Open the DEX process		Opens the view for the last data exchange process that was executed with this document. If no data exchange has yet been performed via DEX with the selected document, no window/dialogue opens when the button is pressed.

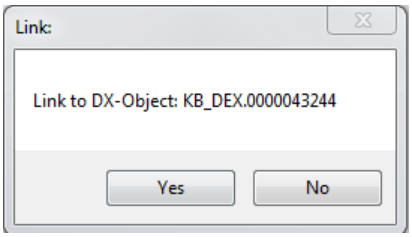
14.3 DEX Export

1. Search for document in SmarTeam.
2. Highlight the data record set to be exported by left-clicking.
3. Generate the export process using the *Arrow* exchange workbench or by right-clicking. function button from the TPDM data
4. Use the **Change DEX status** function to change the status of the individual documents.

(The DEX status must only be changed if the data is read out for editing)

 Status: Only the assembly document is exported for processing, the dependent documents (included in the assembly) must not be changed.	 Status: The assembly and all the selected parts included in the assembly may be modified (red frame). In this example, the drawing is not read out. Via Change DEX status, this drawing can also be read out.

If the DEX status is set to “Exported for processing”, the “Link” box appears. This must be confirmed with “Yes”, otherwise the status is not changed.

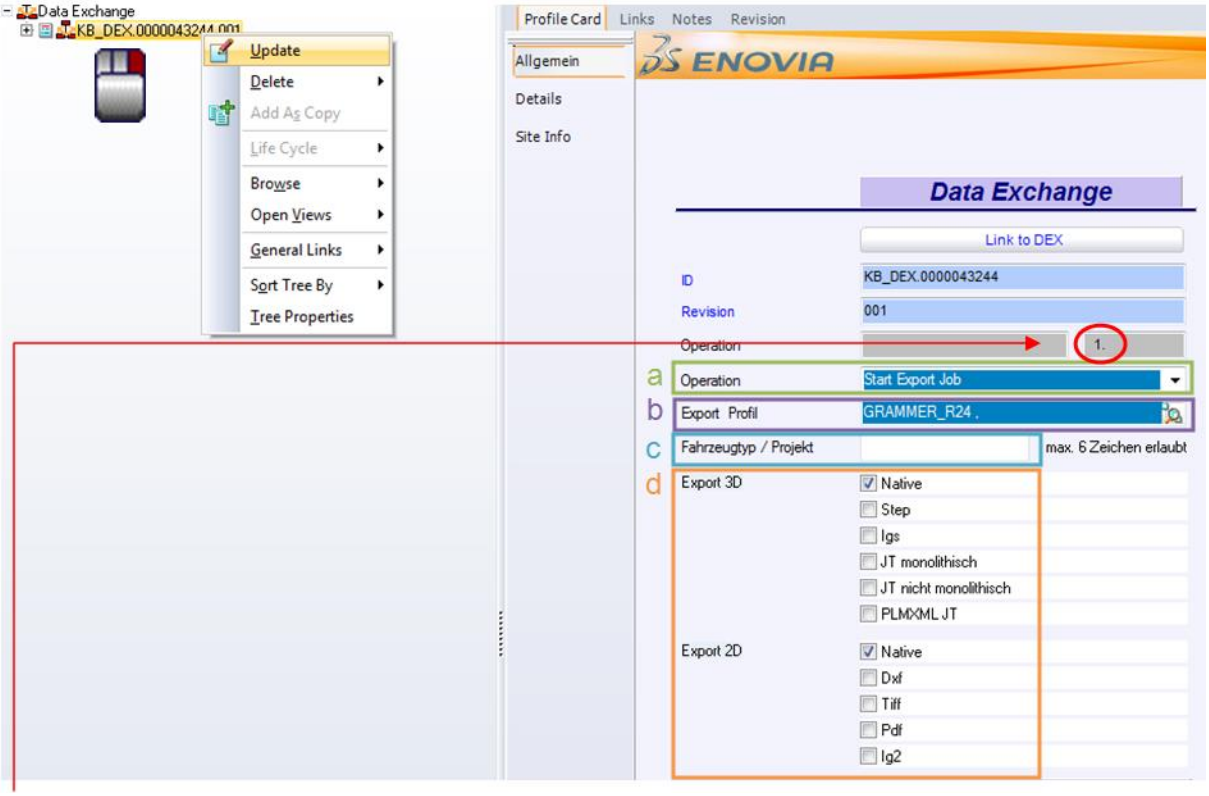


Note:

If DEX is started from 3D, the drawing is only automatically exported at the same time if the 3D and 2D part numbers are identical and the status of the drawing is set to information “--”.
If DEX is started from the drawing, the 3D data is automatically exported at the same time (information only).

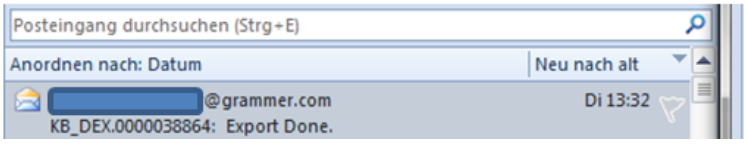
5. Activate export process: In the DEX process, right-click to select the *Update* function.

   a) Select operation (*Start Export Job*)
   b) Select DEX profile (e.g. VWKON_X, BMW_X, GRAMMER_X, ...)
   c) Enter project if necessary
   d) Select export formats for 3D / 2D
   e) Confirm by clicking OK (bottom right) → this starts the process



INFO:
Here you can see the processing position of the DEX job on the server!
1. = In process, 2. = in 2nd position, ...

Note:
Once the data has been exported, the DEX initiator receives an e-mail notification containing a link to the export data!

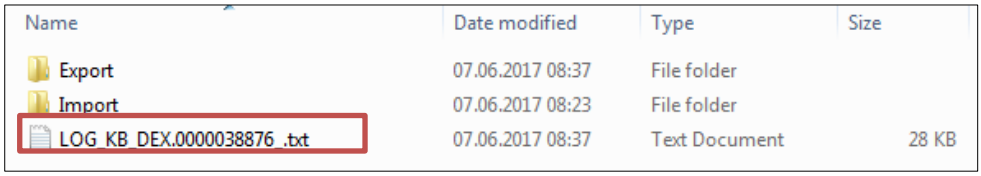


\\HASSNAS2\DEX_KB$\KB_DEX.0000038864

Completion is also displayed in the “Status” field of the DEX job (update with F5 key if necessary).

Executing this process may take some time.
The greater the volume of data and the more jobs there are to export and import, the longer execution will take.

A log file is saved under “Link to DEX” with every DEX job.



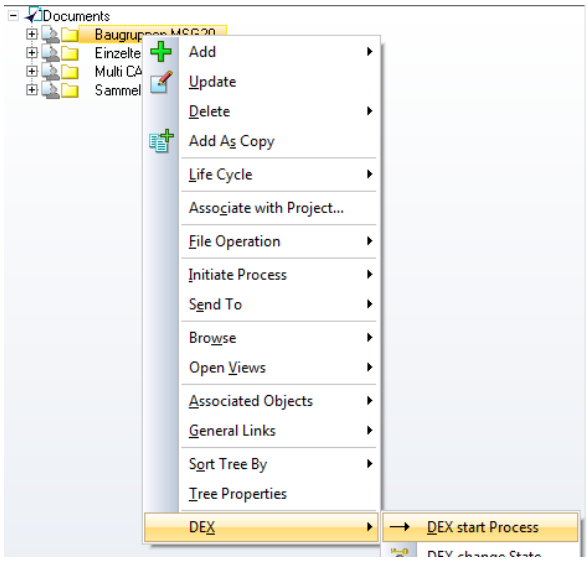
The data exchange process is logged in the “log file”. Any errors are therefore also entered in this log file. The DEX initiator also receives e-mail notification in the event of an error,.

When Catia, NX and JT data is read out, a parameter is automatically written to the models. This is important for the import process. It must therefore not be copied, deleted or changed!

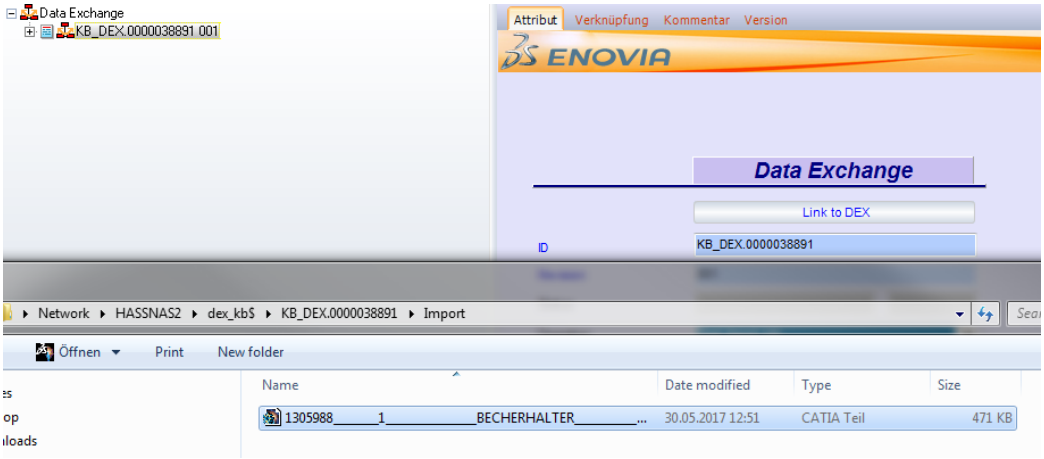
14.4 DEX Import

Importing in order to version data only works for data that was previously exported using an export process with the DEX status “-->” (exported for processing).
All import data that does not have this DEX status in SmarTeam is recreated.

1. Start empty DEX process. In the project in SmarTeam, highlight the folder to which the imported data is to be automatically linked.

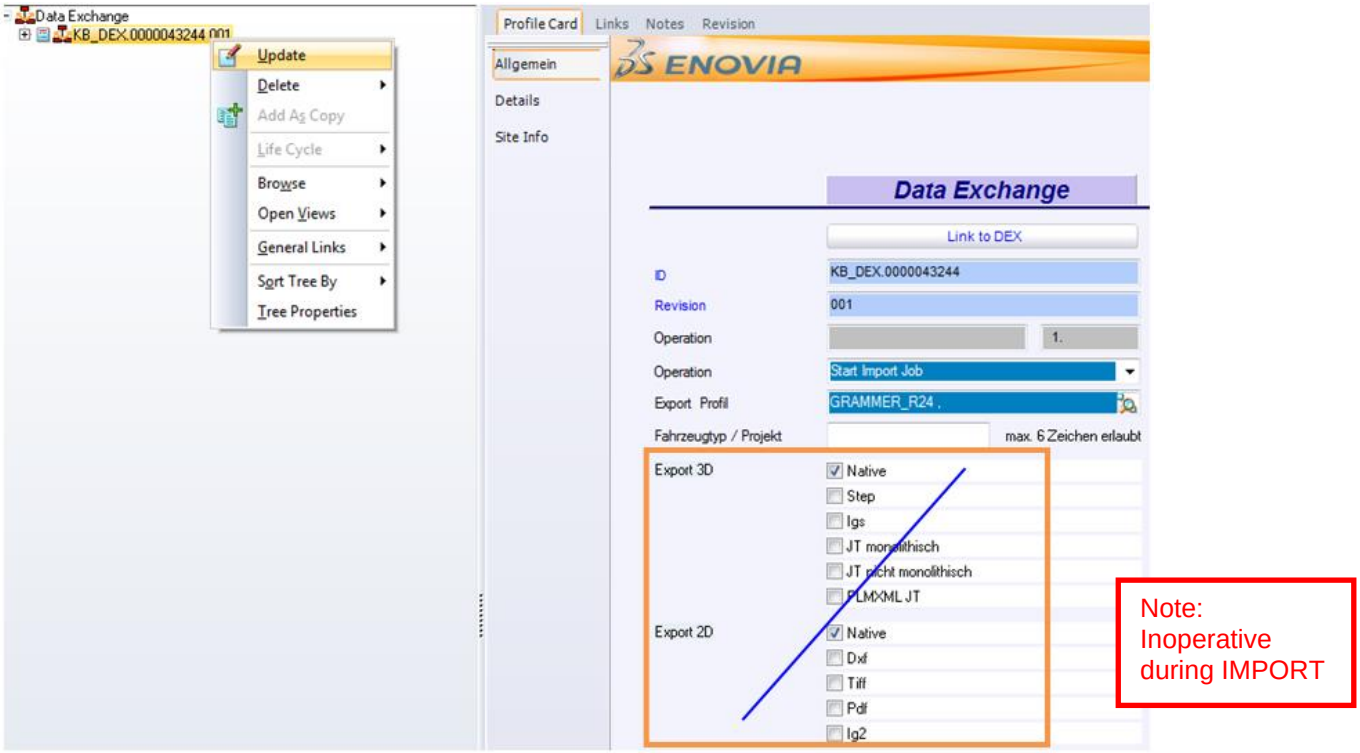


2. Click “Link to DEX” and save/store the data to be imported in the import folder.



3. Click “Update” in the DEX process to start the import job.

- Select operation (Start Import Job)
- Select DEX profile (e.g. VWKON, BMW, GRAMMER_XX_Standard, ...)
- Confirm by clicking OK → this starts the process



Note:
Completed import is confirmed with a notification e-mail!
The DEX status is now switched to “Import Done” (update with F5 key if necessary)

14.5 Creating start models

Empty start model with DEX parameter (GRAMMER environment ONLY)

1. Start empty DEX process. In the project in SmarTeam, highlight the folder to which the imported data is to be automatically linked.

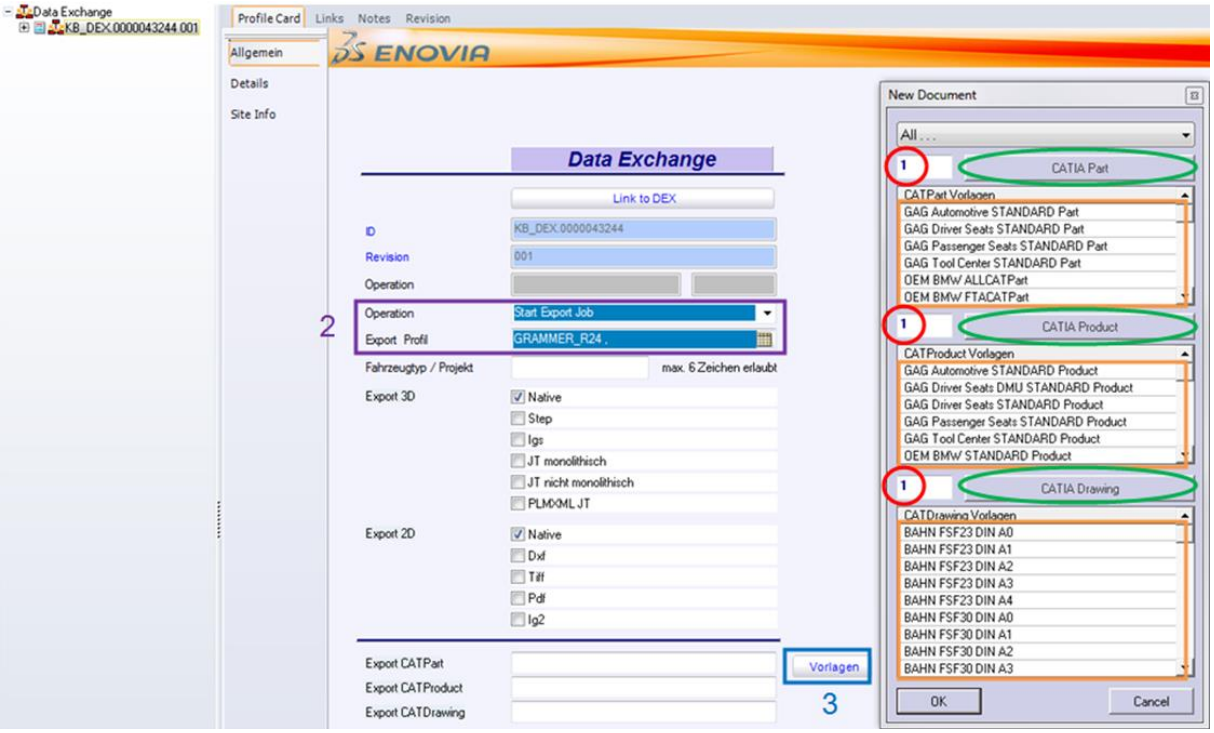
2. "Update" the DEX process

   - Select operation (Start Export Job)
   - Select DEX profile (GRAMMER_XX_Standard)

3. Click on “Vorlagen” (Templates)

4. In the “Neue Dokumente” (New Documents) window, select/highlight items accordingly and confirm with “OK”.

   f) Number of documents
   g) Document type
   h) Template



5. After selection, launch the DEX process by clicking “OK”, bottom right.

When “Export Done” is shown, the empty start models are highlighted with the status “Exported for processing” in SmarTeam and can be copied from the export folder to another folder.
The DEX parameter has now been entered in the empty models.

The DEX parameter is important for the import process. It must therefore not be copied, deleted or changed! The start models must not be copied!

15 Other applicable documents

Training_document_Transfer_Engineering_BOM_SmarTeam_SAP	*T_015_007*

16 Amendments

06/2019
Chapter 11.1