Work Instruction

015 Research + Development

SmarTeam Basics

W_015_015

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1 Purpose

The purpose of this work instruction is to optimise the management and quality of the data generated by CATIA V5 / *NX* at GRAMMER. A uniform procedure and structure for data management is intended to increase efficiency in the application of SmarTeam in combination with CATIA V5 / *NX*. Moreover, data should be created using the new, uniform procedure and also structured and managed in such a way that its usability within the process chain is ensured. The subsequent processes shall draw the maximum benefit from the data provided.

2 Scope

This work instruction applies to all GRAMMER Group companies as well as to external partners that are part of the GRAMMER Group's design/development process and create data for the GRAMMER Group in this context or provide the GRAMMER Group with such data.

This work instruction contains specifications for data generation and management using the *Team Data Management* system SmarTeam in connection with the CAD system CATIA version 5 (CATIA V5) / *NX* in the GRAMMER Group's Design department. This work instruction also applies to all departments within the GRAMMER Group that are involved in the PLM process.

This work instruction as well as the methods and procedures described in it shall also apply to customer projects unless replaced completely or in part by separate agreements.

Upon entering into force, this work instruction shall replace all previously valid work instructions / directives on this subject. The following applies to the application of the new work instruction and the expiry of the previous work instruction:

- For "ongoing old projects", the rules specifically defined for that project shall apply. If such rules are missing completely or in part, this work instruction shall apply where necessary.
- If there are definition gaps, in case of doubt the rules applicable up to now as well as implemented methods shall apply in these situations.

3 Terminology

CATIA/*NX*/SmarTeam-specific descriptions of functions are written in italics.

CAx Data	Umbrella term for data/files created in a **c**omputer-**a**ided system, e.g. CATIA */ NX*.
CATPart	CATIA V5 part (single part)
NX Part	*NX part (single part)*
CATProduct	CATIA V5 product (assembly of CATParts and subassemblies)
NX Assembly	*NX assembly (assembly of NX parts und subassemblies)*
CATDrawing	CATIA V5 drawing (comprising one or more drawing sheets)
NX Drawing	*NX drawing (comprising one or more drawing sheets)*
Component	CATIA V5 structuring element in the specification tree (without own physical document, e.g. on hard disk), as a representation either of a part without a geometric counterpart or of a product comprising CATParts, CATProducts or Components
Root Product	CATProduct that includes several subassemblies (CATProducts)
SmarTeam	Software for managing CAx data
PDM	Product Data Management system
PLM	Product Lifecycle Management
Workflow	A predefined sequence of activities that can be recorded by, among other things, a PDM system (SmarTeam in this case) in digitised form to support the process.
Vault	A directory on the data server where the data is stored securely. Users do not have direct access to the vault.
Viewer	Software for viewing 2D images and 3D representations
Redlining	A functionality in image viewing software that allows annotations and markings to be added to existing illustrations without having to change the original illustration. This allows end users to furnish illustrations with notes. (Application example: SmarTeam Workflow)

4 Responsibilities

The implementation of and compliance with this work instruction are the responsibility of the vice president R&D; queries should be addressed to the R&D Engineering Services Team.

5 Description

5.1 CAD data management with SmarTeam

The GRAMMER Group uses the product data management system SmarTeam to manage documents from the CAD system CATIA V5 */ NX*. In principle, only CAx data originating from CATIA V5 */ NX* is managed with SmarTeam.
Sole exceptions: management of TIF files of CAD systems not integrated into SmarTeam (e.g. Euclid, AutoCAD) via MultiCADTif workflow, Cutter data. Other, non-approved applications and formats (e.g. Office data, neutral CAD data formats or also CATIA V4 data) must not be copied to or saved in the SmarTeam database.

Advantages of SmarTeam for data management:

- Secure, structured data storage
- Management of product structure
- Mapping of CATIA V5 */ NX* links (e.g. CCP link, 3D drawing link)
- Automated revision management
- Data security
- Document management
- Integration into CATIA V5 */ NX*
- Viewing of CAD documents (Viewer)

5.1.1 General information

CATIA V5 */ NX* data formats that may be managed in SmarTeam within the GRAMMER Group are:

- CATPart */ NX Part*
- CATProduct */ NX Assembly*
- CATDrawing */ NX Drawing*
- CATProcess (CATIA process data, e.g. for tool manufacture)

All other file formats, even if they can be generated by CATIA V5 */ NX*, are not currently compatible with the SmarTeam data model within the GRAMMER Group.

5.1.1.1 SmarTeam Editor

This Windows client presents the core functions of the SmarTeam product portfolio. The SmarTeam Editor is also the main front-end (user interface), which can be used to access other SmarTeam functional areas. With the SmarTeam Editor, data can be displayed and managed in GRAMMER Group companies throughout the entire lifecycle of a product. The SmarTeam Editor also has search functions for locating specific characteristics, for example.

5.1.1.2 SmarTeam CATIA / *NX* integration

In addition to the data management function with the SmarTeam Editor described above, this functionality has been partly integrated into CATIA V5 / *NX*. With the CATIA integration of SmarTeam, the main data management tasks can be undertaken in CATIA V5 directly.

Note:
Generally speaking, it is possible and easy to manage documents using both tools mentioned. Important: the CATIA/*NX* integration does not offer the full range of functions found in the SmarTeam Editor. However, the range of functions of the CATIA/*NX* SmarTeam integration is sufficient in most cases.
Advantage of the CATIA/*NX* integration: depending on the task, there is no need to open an additional SmarTeam window in which, for example, the designer manages his documents.

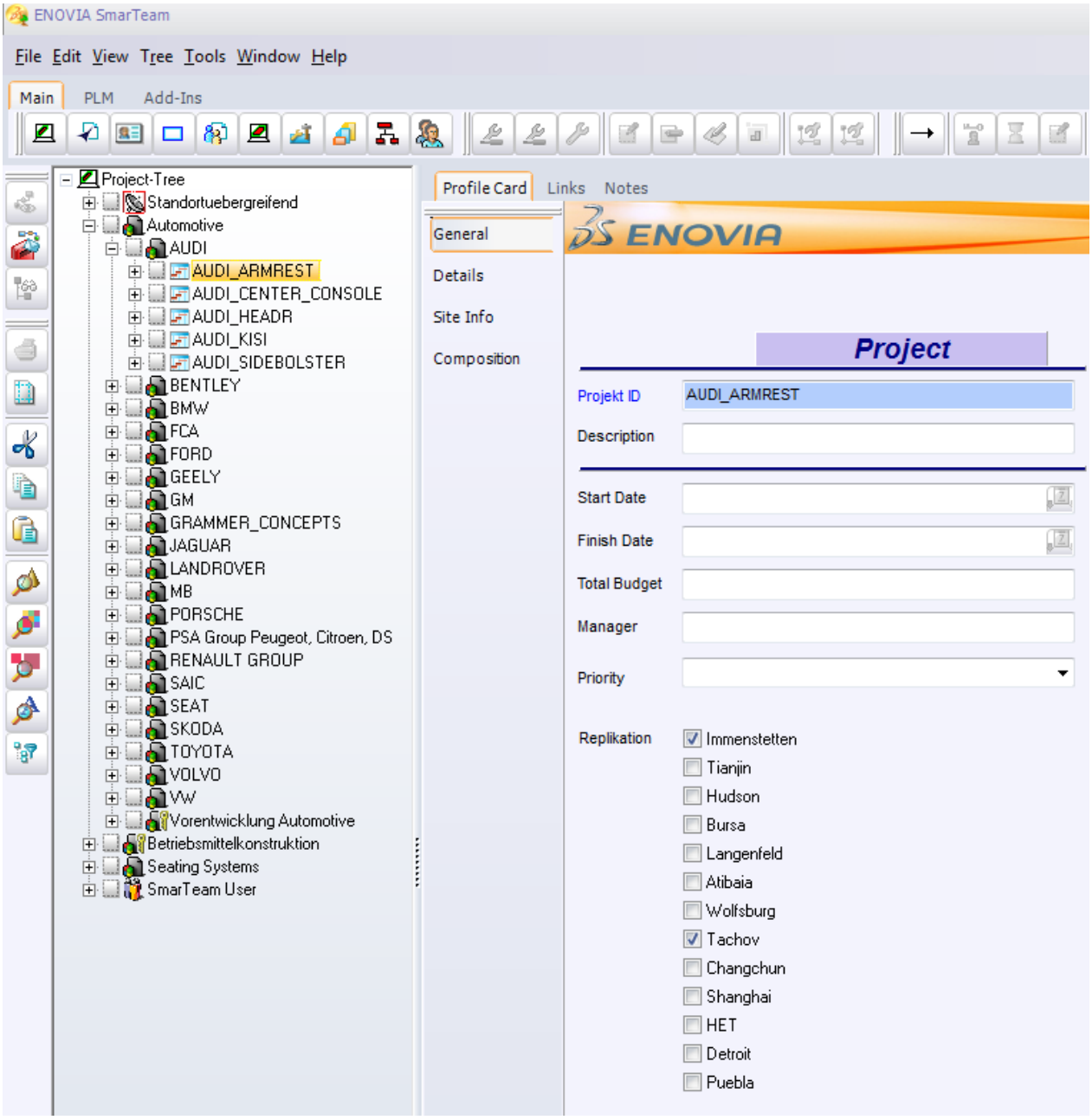
It is left to the designer or user to decide which procedure is preferable for document management (SmarTeam Editor or SmarTeam CATIA/*NX* integration). The software manufacturer makes no stipulation here, so both ways of using the software are essentially correct. It is only with the lifecycle operation *Check in* that preference should be given to the SmarTeam CATIA/*NX* integration. The checking-in of CATDrawings / *NX Drawings*, in particular, must be done using the CATIA/*NX* integration as this is the only way to generate viewing files *(*.tif for Catia, *.pdf for NX*), which are fundamental for viewing drawings in the SmarTeam Viewer.

5.1.2 Database replication (synchronisation of distributed databases = SmarTeam Multisite)

Simultaneous development across different sites is possible within the GRAMMER Group thanks to the SmarTeam Multisite architecture.
There is a database at every SmarTeam development site. All the information contained in the database is compared with all the other databases every 2 minutes. Changes to a document (e.g. update to the profile card, lifecycle such as check-out, check-in, release) are announced to all the sites directly in this manner. An integrated mechanism ensures that amendments cannot be made to the same object at more than one site at the same time. This guarantees data consistency between the distributed databases.

5.1.3 Data replication (vault replication)

Unlike the synchronisation of databases, which provides all the sites with all the information about the data (= metadata, such as entries in the profile card, status details, display of product structures, etc.), the distribution/provision of CAD files is not universal but dependent on the project.
As with the distributed databases, every SmarTeam development site also has its own vault. This contains the files created at this site as well as files approved for replication there.
The information as to which project data is available at which site can be found in the profile card of the respective SmarTeam project (see *screenshot* below).



All the CAD files linked to this project are synchronised with the site(s) marked. Synchronisation happens continuously and takes between 2 minutes and max. 60 minutes depending on the size of the file, the quality of the target site's network connection, and the available capacity of the job server and replication server.
Orders to replicate project data to a different site must be placed by the responsible R&D manager in writing (e-mail).

Note:
Both the CAD files (CATPart, CATProduct, CATDrawing*, NX Part, NX Assembly, NX Drawing*) and their viewing files (*.tif and *.cgr *for Catia, *.pdf and *.jt for NX*) are only replicated to the site selected in the project profile card; this means that none of the other sites has access to the geometries and drawings either in the SmarTeam Viewer.

Important: For data replication to work correctly, the CAD data must be stored in the right project from the outset. Moving data linked to the wrong project at a later stage is a laborious process. The project links have to be corrected, and then the PDM Administration also has to be notified so that the distribution of the data can be initiated.
For this reason, the right project should be selected in SmarTeam at the time when a new document is created.
Further information on this document or project linking process (with user problems as well) is provided on the Intranet in the *SmarTeam Links* document.

5.1.4 Document generation

All CATIA V5 */ NX* documents that are to be managed in SmarTeam must be generated using the script-based "*New Documents*" function in CATIA V5 */ NX*. This is the only way to guarantee that:

All NX Documents have to be generated with "File new", at next choose the tab "GRAMMER" and select your needed drawing size.

- a test number or SAP number is generated automatically.
- filenames receive the SAP-compliant designation / part number.
- the right starting model has been chosen.
- the correct drawing standard has been activated.
- a unique SmarTeam ID is allocated.

Alternatively, external development partners with access to SmarTeam can use the functionalities of the data exchange module built into SmarTeam for generating new documents (see W_015_006_SmarTeam_Data_Exchange_Methodology).

5.1.5 Lifecycle management

Nearly all design data, apart from standard parts, is subject to change management. To ensure the management of all relevant CAD data for describing a/the product throughout the entire PDS lifecycle, the data within the GRAMMER Group must remain available in SmarTeam.

- Product design (advanced development, design) P3/P2
- Product development (new development) P1
- Production, manufacture (application development) P0

} PDS lifecycle with SmarTeam

The next part of the lifecycle is controlled by SAP. The data in SmarTeam remains (in part) the master for CAD document management with SAP. However, in principle, the following stages of the lifecycle are managed via SAP:

- Production, manufacture (in part)
- Sales, dispatch
- Maintenance, repair
- Recycling, product disposal

} SAP lifecycle

Together, these two parts (PDS & SAP lifecycles) represent the *ProductLifecycle* of the GRAMMER Group.

PLM refers to the holistic management of <u>all</u> the data that must be edited and distributed when developing new products or upgrading existing ones, with the ability to manage and control the editing and distribution process.

5.1.5.1 Statuses

Documents, with or without a process reference, usually have different statuses. If, for example, a designer creates a CAD drawing of a part or of a complete seat, it must be possible to determine the status of this document or CAD drawing at any time. The drawing may be in progress at that moment, or it may have just been amended and saved or also released. All these different document statuses can be viewed continuously in SmarTeam. Possible statuses are:

New ➔ File has been saved in SmarTeam. The metadata were transferred to the database during saving. The file is located in the user's local work directory (it has not yet been stored in the vault!), so this file could be deleted inadvertently.
This status does not exist with NX.

Checked in ➔ File was already saved in SmarTeam and has been moved to the vault due to the change in the lifecycle.

Checked out ➔ File has been retrieved from the vault for editing. During the check-out process, the file is written to the user's work directory from the vault. This process triggers the system to create a new version of the file automatically.
This status does not exist with NX. The NX data can be locked for other users during editing. This locking should always take place when the file is opened in order to avoid overlapping during processing.

Release ➔ File has been released via the workflow process. The file can no longer be amended in any way using the lifecycle tool built into SmarTeam. The file is located in a special Release vault. Released objects are identified by an alphanumeric index, starting at "a".

Obsolete ➔ If a file has a new release (e.g. revision index b), then the previous revision (e.g. revision index a) is automatically labelled as "*obsolete*" via the workflow process. The obsolete file remains in the system and can therefore be versioned for other designs, for example. Data marked as obsolete is deposited in a special Obsolete vault.

Pre-Series release ➔ This status is a special pre-series status triggered exclusively by the pre-series workflow. As early as the development process, defined drawing, part and product statuses can be documented and assigned to different maturity levels/phases using the pre-series workflow. These objects are furnished with a numerical index, which is also mapped to the drawing title block, starting at "01". Stamping of the CATDrawing with "Preliminary Design". At the end of the workflow, transfer of basic data and documents (tif, jt) with the status "VL" (= pre release) to SAP. See also 5.3.1.3 Pre-Series workflow.

5.1.5.2 History

For traceability reasons, etc., it is important that all documents have an unbroken change history. This change history makes it possible to trace which documents have been opened or amended, etc. by whom, on what date and at what time. A file's change history can also be used, for example, to find out which test number a document had before the SAP number.

5.1.6 Document display / viewing

All documents managed in SmarTeam can be viewed using an internal viewer (image viewing software). As well as the viewer function, there is also the option to navigate in models/drawings and to make printouts of the respective views. In addition, with drawing elements in a test number, pre-series, release or modification process, redlining can be used for marking drawings. These marks are visible in the sites where the drawings are also synchronised (depending on the project).

Advantage of redlining: This enables the workflow user to append notes to the drawing without having to print it out and then scan it in again! The drawing can be viewed/printed in the viewer both with and without redlining.

5.2 Document management / product data management

Generally speaking, the definition for the term document management has been replaced by that for product data management (PDM) as this expands the definition. Product data management includes the management of product data and the control of product data flows in the phases of the product lifecycle.

5.2.1 Versioning

If documents that are regarded as *versioned parts or products* are created with
CATIA V5 / *NX* / SmarTeam, i.e. there are several states for the part or product, versioning is ensured by SmarTeam. The respective versioning state can be requested at any time via SmarTeam.
The unique identifier of the current state ensures that the user is always offered the latest version of the part or product. But there is also always the option of viewing earlier development states or even using them if necessary.

5.2.2 Creation of Variants (Only possible with CATIA V5)

A SmarTeam functionality developed for the GRAMMER Group offers the option of variant design. Copies can be made of a part/product and developed in parallel to parts with the same part number, adding the affix *-VAR1 to *-VAR9. Since they have different filenames, they can also be loaded in CATIA V5 in parallel. If a design variant is chosen, this can be restored to the history of the original part/product (see SmarTeam training document “SmarTeam Vertiefung” on the Intranet).

5.2.3 Revision

The respective revision state and the revision history can be requested at any time via SmarTeam. The SmarTeam functionality ensures that the user is always offered the latest revision state of the part or product, provided it is available in the system.

There are two revision identifiers in the SmarTeam environment at GRAMMER:

- PDM revision: Consecutive, system-based labelling of the versions - from .0 to .99, after the first release for series production a.0 to a.99, after the next production release b.0 to b.99, etc.

- Index: Is only filled in the case of data accorded pre-series or production status by a corresponding SmarTeam workflow. The index is numerical (pre-series status) 01 to 99 or alphabetical (production status) a to zz.

It is therefore obvious at a glance whether certain versions relate to a purely work state (no entry in the index field) or a state that requires documentation (entry in the index field).

5.2.4 Symbols in CATIA V5 and NX

In capter 5.2.4.x is an overview of symbols of the CATIA V5 and NX structure tree. The symbols are displayed when CATIA V5 or NX are combined with SmarTeam.
The symbols in the following table relate to the status of product or instance documents, but they are transferable to part and/or drawing documents.
The symbols in the following table are only valid for CATIA V5, in NX does these statueses not exist.

Lifecycle status	Symbol in CATIA
New	
Checked in	
Checked in, not latest version	
Checked in, modified in read-only	
Checked in, not latest version, modified in read-only	
Checked out	
Released	
Released, not latest version	
Obsolete	
Opened in read-only, modified. The status of these documents in the CATIA session does not match the content of the SMARTEAM database, e.g. an old version of a component was loaded into a product opened in read-only mode.	

The symbols in the following table are valid for both systems (CATIA V5 and NX).

NX components	Symbol in SmarTeam	Catia components	Symbol in SmarTeam
NX Assembly		*CATProduct*	
NX Part		*CATPart*	
NX Drawing		*CATDrawing*	

The content-related structure of documents generated by CATIA V5 / *NX* is described under *6. Applicable documents*. The following definitions also apply:

5.2.4.1 The CATProduct element

Fig.: Icon of a CATProduct in CATIA (without SmarTeam integration)

The CATProduct is, as the name implies, a carrier of product information. The term product information groups together an extensive range of different types of non-geometric information that describe a product in more detail. This information is closely related to and supplements the geometry data (CATPart) of the individual parts. The following types of data are product information:

- Structural information (references to installed CATProducts and CATParts)
- Positional matrixes
- Element identifier (part number, component, etc.)
- Positional conditions (reference to individual geometric elements)
- Tolerance information
- Kinematics
- Publications (at CATProduct level)

This list is exemplary and not exhaustive. It contains typical types of data that are allocated to the broad category of “product information”. This data is stored in the CATProduct files of the CATIA structure at the time of saving.

5.2.4.2 The Component element

Fig.: Icon of a Component in CATIA (without SmarTeam integration)

The Component is also a carrier of product information within product structures. In contrast to the CATProduct, the Component is not an independent structural element for archiving purposes because CATIA does not generate a file for storing a Component's data. A Component is to be understood as part of the higher-level CATProduct. During the CATIA session, the Component receives its own product information. This information is stored in the file of the higher-level CATProduct at the time of saving. The higher-level CATProduct of a Component is the first CATProduct reached when going up from the Component in the structure tree. Every Component has such a CATProduct (because the structure tree of a product always starts with a CATProduct). The product information of several Components can be stored in the file of a CATProduct.

5.2.4.3 The CATPart element

Fig.: Icon of a CATPart in CATIA (without SmarTeam integration)

The CATPart is the carrier of the geometric description of a part (3D geometry data). Both solids and wire-frame & surface geometry (surfaces, planes, etc.), along with their properties, design elements, parameters and publications, etc., are represented by the CATPart and managed in it. This data is stored in a CATPart file at the time of saving.

5.2.4.4 The Instance element

Fig.: Icon of an Instance in CATIA (without SmarTeam integration)

The Instance realises the use (installation) of a CATPart in a product structure. The Instance of a part can be inserted under a CATProduct or a Component. For example, a CATPart can be installed under a CATProduct multiple times, whereby the Instance manages and defines the various positions of the part used several times within the product. The information stored in an Instance is saved in the corresponding CATProduct at the time of saving. Therefore, for archiving/saving purposes, the Instance is not an independent structural element (just like the Component).

5.2.4.5 The element Part in NX

Fig.: Icon of a Part in NX

The NX Part is the carrier of the geometric description of a part (3D geometry data). Both solids and wire-frame & surface geometry (surfaces, planes, etc.), along with their properties, design elements, parameters and publications, etc., are represented by the Part and managed in it. This data is stored in a .prt- file at the time of saving.

5.2.4.6 The element Assembly in NX

Fig.: Icon of Assemblies in NX

The NX Assembly is, as the name implies, a carrier of product information. The term product information groups together an extensive range of different types of non-geometric information that describe a product in more detail. This information is closely related to and supplements the geometry data (Parts) of the individual parts. The following types of data are product information:

- *Structural information (references to installed NX Assemblies and NX Parts)*
- *Positional matrixes*
- *Element identifier (part number, component, etc.)*
- *Positional conditions (reference to individual geometric elements)*
- *Tolerance information*
- *Kinematics*

This list is exemplary and not exhaustive. It contains typical types of data that are allocated to the broad category of "product information". This data is stored in the NX-Assembly-files of the NX structure at the time of saving.

5.2.5 Project and folder structures

In principle, a distinction is made in SmarTeam between project and document structures.
The project structure is displayed when the SmarTeam Editor is opened. It

- helps with structuring,
- is the basis for project-related rights administration and
- is the basis for project-related distribution/synchronisation of CAD data to/with other SmarTeam development sites.

Each project has its own document tree. This contains the folder structure with the data.

The project and folder structures in SmarTeam must not be altered by users. If projects and/or folders have to be added, modified or deleted for the management of product data, this may only be done by the SmarTeam Administration in the case of projects and additionally by authorised key users (Dept. of Engineering Services) in the case of folders.

The existing folder structures for document storage are to be filled according to their names.

5.2.6 Data archiving

Data from external partners and suppliers that is not GRAMMER's own data or is incompatible (in the sense of, for example, supplier drawings), but which must be archived in SmarTeam, is archived according to the following procedure:

- Create a new drawing document using the script-based "*New Documents*" function in CATIA V5 */ NX*
- Configure the sheet size
- Insert the external partner's TIF document into the carrier document (in this case the CATDrawing */ NX Drawing*)
- Save the "carrier drawing" in SmarTeam

Any further archiving of data from external partners/customers is not currently provided for in SmarTeam. An extended functionality in terms of archiving Office documents, for example, is handled within the GRAMMER Group by the SAP DMS.

5.3 Workflow

Definition of workflow: This is to be understood as a solution to the automation and control of work processes (e.g release process for CAD documents). Business processes for the implementation of technical modifications and releases are streamlined and accelerated by the *Workflow application* (within the GRAMMER Group this is the *SmartFlow* tool). With the *SmartFlow* application, users can manage the project sequence (graphically), view and log the project status, and automatically route the work to the next stage of the project. Users can initiate a document workflow using a list of predefined modification and/or release processes.

Once the workflow has started, different stations for a document review can be triggered depending on the workflow class. At each station specialist checks are performed on the documents/designs. Specialist checks may only be executed by employees who have been assigned this role (in SmarTeam). A check must not be performed by the person who made the design or started the workflow process.

The following applies to all the stations of the workflow: only once a document has passed through all the stations without any errors will the actual document release be executed and the document published.

5.3.1 SmarTeam workflow - SmartFlow

Within the GRAMMER Group, the workflow sequence is controlled by *SmartFlow*. All documents generated with CATIA V5 */ NX* are released exclusively via the workflow integrated into SmarTeam. For this to work, all the relevant documents for a workflow must have been checked into or be present in SmarTeam.

One part of *SmartFlow* is the *WorkFlow Manager*. This entitles users with administrative rights:

- to make inquiries about ongoing and/or completed workflow processes.
- to request statistics about ongoing and/or completed workflow processes.
- to redirect workflows if necessary (e.g. checker not available).

All users have the opportunity to submit a status request concerning documents in an active workflow by pressing the *Display workflow* function button in the *SmartFlow* toolbar in the SmarTeam Editor.

Additional workflow processes & stations can only be defined by the Engineering Department.

Different actions are triggered by the workflow defined in SmarTeam (*SmartFlow*). The function is accessed from the toolbar within the SmarTeam Editor window. Which workflow must be started under the function selection *Initiate process* depends on the respective project and document status. The possible workflow processes are listed below and then explained briefly in subsections:

- Release Process *(with or without C-Part Check / TechnologyCheck)*
- Modification Process *(with or without C-Part Check / TechnologyCheck)*
- Bagatelle Process
- Pre-*Release* Process
- V-Number Process
- TIF Release Process

Detailed explanations of these workflow processes can be found in the work instruction W_015_003.

To avoid a situation where old/invalid versions are used and/or issued to suppliers and customers, it is essential to check whether the document is in a workflow process. Access the *Document view* in SmarTeam, click on the *Link* tab, then open the *Workflow* sub-tab. The workflow status can be viewed under this option.

5.4 SAP integration

SmarTeam is the PDM backbone system of the GRAMMER Group in which the design data is managed. All the relevant information that must be provided in the value chain across divisions (e.g. purchasing) is read from the CATIA*/NX* documents managed in SmarTeam and transferred to SAP. Meanwhile, SAP, for example, is the basis for data exchange with external partners via the *cFolder* system. The exchange of data is organised with this tool. Current data, both for customers and from customers, is kept available for further processing in CATIA V5 */ NX* and for archiving in SmarTeam. This achieves another aspect of the definition of integrated product data management through to PLM.

In terms of its application, SAP is a so-called ERP (Enterprise Resource Planning) system. ERP systems primarily support the logistics in industrial manufacturing companies, but they are also used in finance, controlling and human resources management. Logistics cover the entire value chain of procurement, production and sales. In general, ERP systems are intended to optimise and control the production process and resource planning in a company, so it is fundamentally important that the current version of data is always kept available in both systems (SmarTeam and SAP).

SAP and SmarTeam exchange data via an interface. This is partly necessary because SAP provides the number generator for part number generation. The SAP installation within GRAMMER also features a DMS extension. This allows document management for released drawings, for example.

5.4.1 Bills of material (BOM)

Once the pre-series, release and modification workflows have been concluded, the design bill of material (PPKO) is transferred from SmarTeam to SAP provided that products are involved in the workflow. These design BOMs serve as a template for the creation of production BOMs.

To enable the developer beforehand to perform a structural product check or to assess the accuracy of the design BOM, functions for issuing BOMs (e.g. in PDF format) are integrated into SmarTeam in the TPDM toolbar. See also training document “Guidance SmarTeam bill of material export”.

5.4.2 Master data maintenance

The information that is transferred to SAP is listed below:

- Part number
- Revision
- Part / product name
- Base unit of measure
- Weight
- Weight unit
- Size / dimensions
- Material
- DIN format of drawings
- Assigned design documents

Given that this information is passed on to the ERP system, special care is advised here when it comes to maintaining this data as carelessness may lead to major complications in some cases. An incorrect entry may, for example, have a negative impact in terms of material planning costs.

5.4.3 Document transfer

At the end of pre-series, release and modification workflows, the following documents are transferred to SAP:

- JT (converted 3D CAD geometries, such as CATParts, CATProducts, *NX Parts, NX Assemblies*)
- TIF (converted 2D CAD drawings – CATDrawings */ NX Drawings*)

5.5 General notes & rules

The following is a brief summary of what the user needs to bear in mind when designing or handling data in SmarTeam and CATIA V5 */ NX*:

- The SmarTeam's CATIA*/NX* integration function should be used primarily. Only if the range of functions here is not enough should the user turn to the functions of the SmarTeam Editor.

- The assignment of data to projects must be correct from the very first time the data records are saved in SmarTeam onwards. A subsequent data assignment to a different or an additional project is possible, but usually involves a lot of effort.

- Data must be stored in the structures/folders of the projects according to its type and status.

- CATIA V5 functions not allowed: *File / New from* and *File / Save As* with the option *Save as new document* must not be used.

- The construction and modification of product structures must be done using CATIA V5 */ NX*.

- Clean up data records/models before saving (remove unnecessary geometries!).

- Review the data regularly using the Q-Checker *in Caita V5.*
  In NX, the data should be checked with corresponding checkers according to customer specifications (HQM, CHECK-MATE).

- Checking data into SmarTeam: It is not enough just to announce data in the database using *SmarTeam - save*. The data record remains local (it is not transferred to the Vault) and is not included in the backup that takes place at regular intervals.

- Regular clean-up of work directories: Data is stored here temporarily for editing. This area is <u>not</u> a data archive! The work directory must be cleaned up using the SmarTeam File Explorer. This makes it possible to view the status of the files in their connection to SmarTeam. A weekly clean-up is recommended.

- Maintenance of the master data in SmarTeam: Fill in the document tabs carefully.

- The information from the SmarTeam tabs is only mapped once the relevant CATIA*/NX* file is opened again in CATIA V5 */ NX*. Alternatively, the majority of information can also be loaded using the SmarTeam function "Load from Database" built into Catia V5 */ NX*.

- All SmarTeam sessions must be terminated at the end of the working day.

6 Applicable documents

Sammlung CAD-/PDM Dokumente / Collection CAD/PDM Documents	A_015_012
Work Instruction SmarTeam_Workflow_EN	*W_015_003*

7 Changes

06/2017
Insert contents
Expanded for NX
New picture on page 6
Explanation of Workflow-Processes removed, reference to Work Instruction W_015_003