Work Instruction

015 Research + Development **CATIA V5 - Drafting Advanced** **W_015_022**

created	15.10.2012	Wunschel Rosemarie
proved	13.12.2012	Holzner Martin
released	08.01.2013	Kammerer Georg
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1 Purpose

The purpose of this work instruction is to support CATIA V5 users at GRAMMER AG and to present a standardised procedure and definition for the generation of drawing derivations of complex products (assemblies).

2 Scope

This work instruction shall be binding for GRAMMER AG as well as for external partners that are part of GRAMMER AG's design/development process and provide GRAMMER AG with data in this context.

New knowledge regarding the handling of design tasks and design orders with CATIA V5 and concerning associated data processing with SmarTeam has been incorporated into this guide. Alongside this work instruction, the attachment "A_015_012_Sammlung_CAD_PDM_Dokumente_ Collection_CAD_PDM_Documents" shall also be binding.

3 Terminology

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

RMB	Abbrev.: **R**ight **M**ouse **B**utton -> to access shortcut menus, for example
Instance	Objects that may be used (several times as well) in the structure of products
Instance node	Structure object used to manage the instance.
SmarTeam	Software for managing CAx data documents from CATIA V5
120% model	Master product containing all the parts and functional products for the derivation of all existing variants of a product

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 General information

5.1.1 Scope and information on this document

This document describes and explains the procedure for generating drawing derivations from complex products. This document should provide users with a work of reference for everyday use.

The following points present the basic procedure in sequence and are addressed in this document:

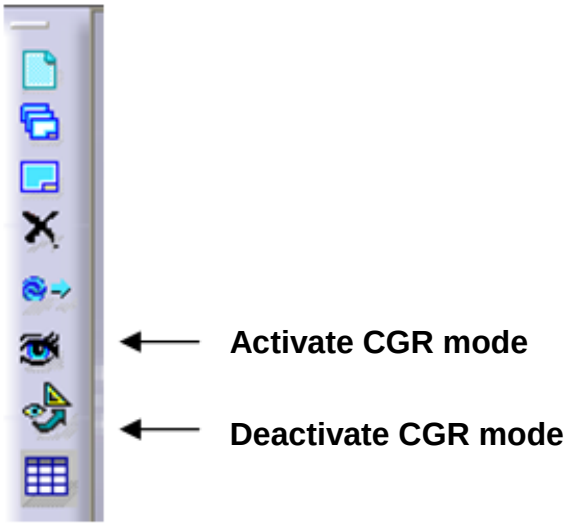
- Starting the CATIA V5 session
  - Basic concepts of the *CGR* work mode
- Link management of views
  - Connection query
  - Manipulating connections
- Deriving other views
  - Adding projection components
- "*Overload control*" of views
  - Settings and general concepts

5.2 Starting the session for the derivation of drawings of complex products

To simplify the handling of complex products in CATIA V5 and to improve editing performance, work in CATIA V5's so-called CGR mode.

This means that not all the model information for the parts contained in the product is loaded, just the visualisation or surface data, putting less strain on the memory.

To access the CGR mode, press the following button in the CatiaV5 interface: "activate CGR mode"



5.2.1 Basic concepts of the CGR work mode

Setting the CGR mode makes the following program definitions effective:

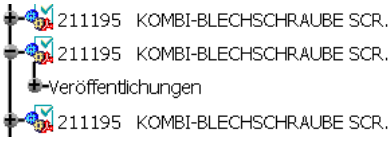
1. Definition of a cache directory in which the tessellated data for the surface representation of the parts is stored.

2. The tessellated data of the product components is generated when a product file is opened. If a CGR file of components already exists in the cache directory, these are checked to see whether they are up-to-date.

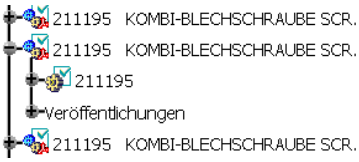
The program properties make sure that only the graphical representation of the data sets within the product is loaded and not the historical geometric structure (development history). There is therefore not so much strain on the computer's working memory. However, this means that there are also a few special features when working with CATIA V5:

a) The product might not have a resolved status. As a result, the symbol or is displayed with regard to product updating. The reason for this is that there may be no exact geometries and possibly erroneous references between context objects. This also means that the geometry was not loaded or was possibly broken (for example, an exclamation mark is displayed on every published element, a question mark is shown on every comment, etc.). Do **<u>not</u>** run the update as this would load the entire volume of data again.

b) The structure of the product can no longer be extended to the geometry objects in display mode.



Structure in *display mode*



Structure in *draft mode*

c) With the award of assembly constraints, the cursor changes when pointing to a geometry located in display mode. An "*eye*" symbol is also displayed next to the arrow.

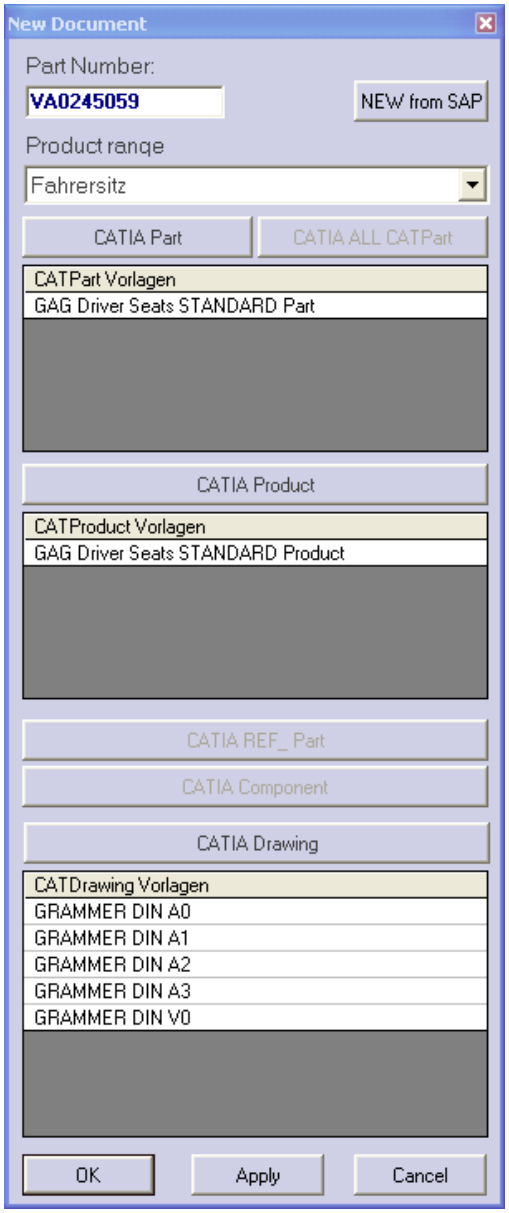
Selecting the geometry automatically places this instance in draft mode.

d) If individual data sets of the product are to be edited, these can be "reloaded" specifically. This is done either by opening the individual documents in a separate window or, for example, by right-clicking on the corresponding structure/object node or by clicking on the geometry area with the *Displays -> Draft mode* function.

5.3 Switching to the Generative Drafting area

The loaded product documents are loaded in the AssemblyDesign workbench as standard. To derive the drawing, a file of the type *.CATDrawing must be loaded and active.

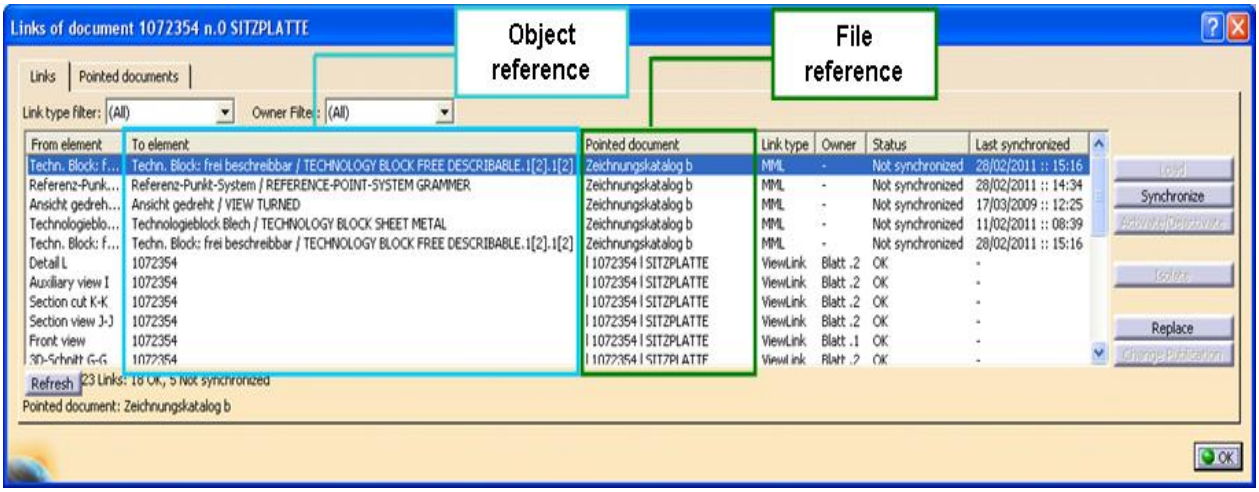
To achieve this, either open an existing empty file (so-called starting model) or



create a new, registered file.

5.4 Link management of views

The main view produced has a direct reference to the components selected as well as the higher-level product. This can be traced using the Edit -> Links function.



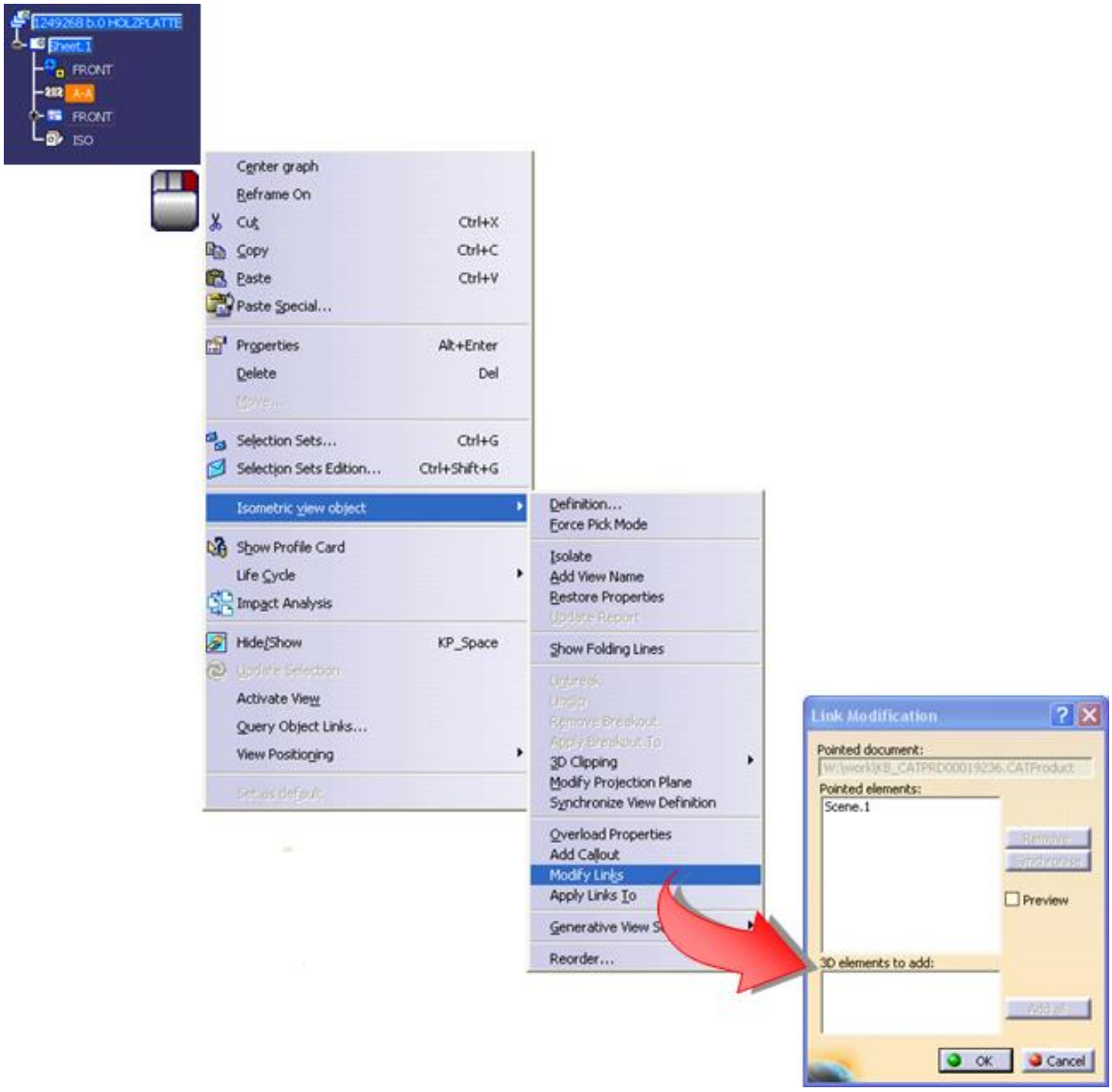
Object reference: Object that is derived/projected onto the drawing.

File reference: This is the product (highest node) in which the objects that are projected onto the drawing are managed.

5.4.1 Connection query and link manipulation of the view

The connection query of the view is best executed via the RMB shortcut menu. To do this, point the mouse at the view in the structure and perform the following actions:

1. In the Assembly Design environment, configure the product selection using the button
2. Switch to the GenerativeDrafting workbench.



3. Select the objects to be projected that are also relevant to the view.
   a) Note: The objects can also now be selected on the graphical user interface
   b) Note: To select more than one, press the Ctrl key

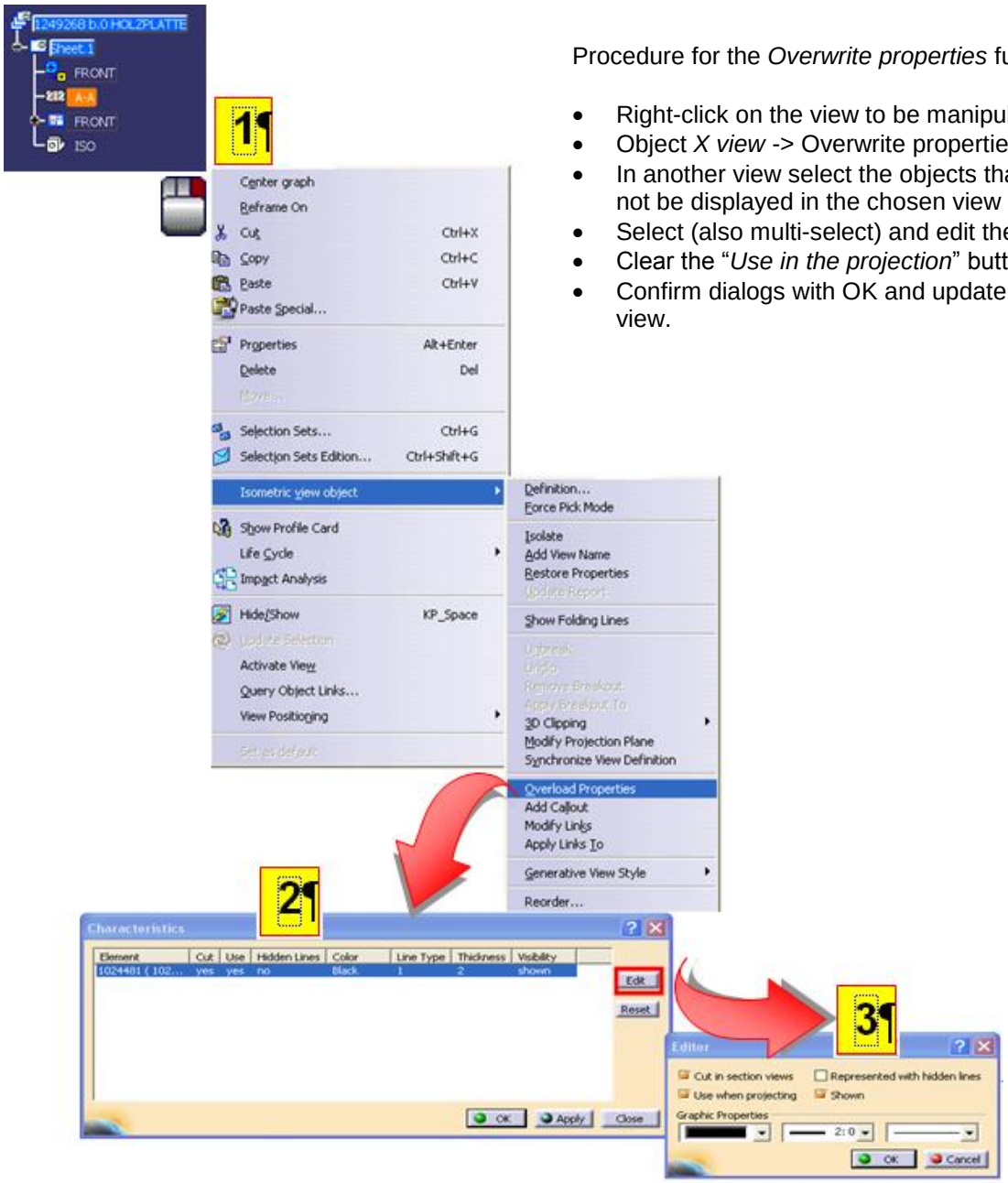
If further objects from the 3D window of the product are now selected by holding down the Ctrl key, these are added to the field "*3D elements that …*" and can be added, via the "Add all" button, to the view for projection. Once this has been done, the view must be updated after confirming the dialog with OK.

5.4.2 “Overload control” of views

The overload control can be used with individual views to configure the following specifically:

- Whether objects are derived in the view.
- How objects are exported in the view.

This control can therefore be used on an individual basis to define afterwards how the elements should be managed in the view.



Procedure for the *Overwrite properties* function:

- Right-click on the view to be manipulated
- Object *X view* -> Overwrite properties
- In another view select the objects that should not be displayed in the chosen view
- Select (also multi-select) and edit the objects
- Clear the “*Use in the projection*” button
- Confirm dialogs with OK and update the view.

5.5 Summary, notes and rules

The general procedure for deriving drawings of **complex products** should be as follows:

1. Start CATIA V5 in CGR mode
2. Load the product from SmarTeam in read access
3. Define the CATDrawing into which the product should be exported
4. Start the function for generating the front view
5. Select the visible components for the main view (front view) The best way to select the components to be projected is via the structure.
   To select, hold down the Ctrl key
6. Switch to the window of the drawing derivation -> view position and create
7. Create the projected view
8. Call up the so-called overload properties via the RMB shortcut menu with the *Overwrite properties* function
9. Select the components/objects that should be manipulated
10. Mark and edit the selected components/objects
11. Deselect the button for projecting the components/objects
12. Confirm dialogs with OK and update the view

6 Applicable documents

Attachment “Sammlung CAD-/PDM-Dokumente / Collection CAD / PDM Documents”	A_015_012

7 Changes

Replacement for Work Instruction “AA-13-04.2008-FE Basics & Methodology CATIA V5 - Drawing Derivation”
Full updating / revision
New draft