Work Instruction

015 Research + Development **CATIA V5 - Part Design** **W_015_010**

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1 Purpose

The purpose of this work instruction is to optimise the quality of data generated by CATIA V5 at GRAMMER AG and to standardise the structure of CAD models. A further objective is to increase efficiency in the application of CATIA V5 through an identical approach to design. The aim is for models to be created using the new, uniform procedure and structured in such a way that their usability within the process chain is ensured. The subsequent processes shall draw the maximum benefit from the data provided.

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 create data for GRAMMER AG in this context or provide GRAMMER AG with such data.

This work instruction contains specifications for working with the CAD system CATIA version 5 (CATIA V5) in GRAMMER AG's Design department as well as in cooperation with external design and engineering partners that generate data for GRAMMER AG.

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.

New knowledge regarding the handling of design tasks and design orders with CATIA V5 has been incorporated into this work instruction. Alongside this work instruction, the attachment “A_015_012_Sammlung_CAD_PDM_Dokumente_Collection_CAD_PDM_Documents” shall also be binding.

The following applies to the application of the new work instruction and the expiry of the previous work instructions:

- 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-specific descriptions of functions are written in italics.

CATPart	CATIA V5 part (single part)
CATProduct	CATIA V5 product (assembly of CATParts and subassemblies)
CATDrawing	CATIA V5 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
ALLCATPart	Merger of a product to create a CATPart
RPS	Reference Point System
CATDUA	Additional application for cleaning and reviewing CATIA V5 documents
Assembly	Assembly of single parts, products and/or subassemblies to create one product
Root Product	CATProduct that includes several subassemblies (CATProducts)

4 Responsibilities

The implementation of and compliance with this work instruction are the responsibility of the vice president R&D.

5 Description

5.1 Designing parts with CATIA V5

This section sets out the basic principles, constraints and procedures for volume-based parts design (solid design) at GRAMMER AG using the PartDesign workbench with CATIA V5.

5.1.1 General principles

As a general rule:

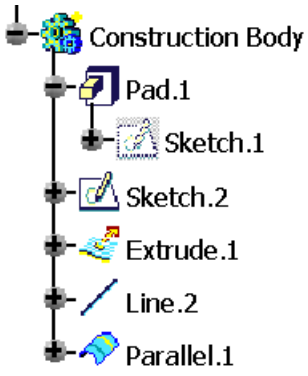
- All parts that receive their own entry in the bill of materials should be designed as separate CATParts.
- Deactivated features in the structure of single parts should be avoided. Deviation from this requirement is only permitted if it is controlled using tables or *KnowledgeWare*.
- Simple features like pad, pocket, shaft and groove are to be preferred to complex features like multi-pad, multi-pocket or filleted pad/pocket with draft angles.
- Unused elements that have been copied into a file for, say, installation space analyses, should be deleted (to reduce the amount of data).

5.1.1.1 Hybrid design

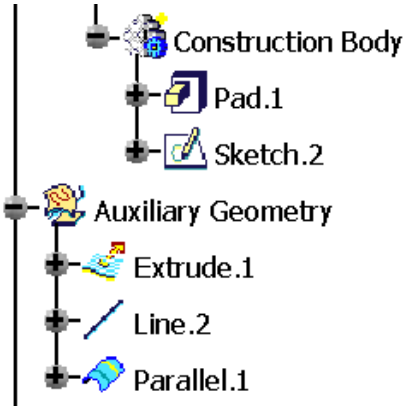
SeatingSystems	***Automotive***
The hybrid design mode is not available for universal application throughout GRAMMER AG. The reason for this, in most cases, is that the customer definition or customer specifications preclude the use of this mode in part.	The design method is prescribed by the respective customer/OEM in Automotive.

Since CATIA V5 release 14 it is possible at GRAMMER AG to work with so called hybrid design. This model structuring option is not to be confused with the "Hybrid-Design" (part definition consisting of a combination of solid and surface elements). In CATIA V5, hybrid design defines the orientation of the system in terms of the use and organisation of objects in the structuring elements. Selecting the hybrid design mode allows activation of the use of *hybrids* in which solid geometry as well as wire-frame and surface geometry can be managed together. This strategy for managing elements is not permissible in the traditional application.

The differences in the structure are clarified by the following screenshots.



Hybrid structure

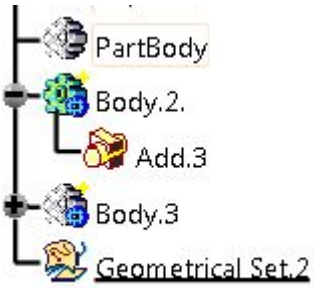


NON-hybrid structure

The system identifies data in active hybrid design mode as follows:

- Bodies that have been generated in the active environment are furnished with a green gear symbol
  - or
- Bodies that have been generated in the non-hybrid design environment are furnished with a grey gear symbol when loaded in the hybrid design environment
  - or
- Bodies that have been generated in the hybrid design environment are furnished with a yellow gear symbol when loaded in the non-hybrid design environment
  - or

<u>Note:</u> In principle, it is possible to edit data that has not been created in the active mode (hybrid / non-hybrid) and also to supplement it with further bodies. This would result in a mix of body objects (see graphic).



As things stand, it is not possible to convert data created in hybrid mode into a data record that has the attribute non-hybrid, and vice versa.

5.1.1.2 Layers

In CATIA V5 models, layers/filters are not allowed. It is possible to separate the individual components clearly through use of the multi-body structure and due to the hierarchical context of the model structure.

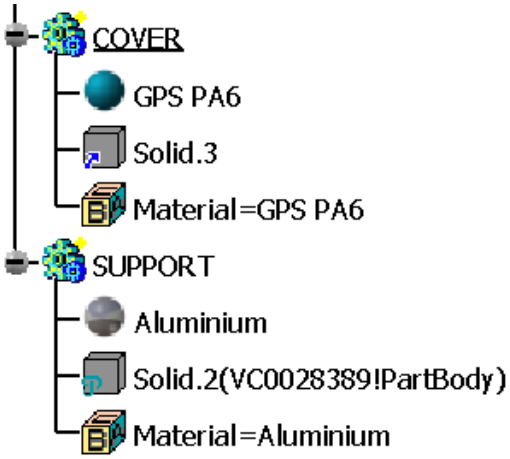
5.1.1.3 Design tables

Design tables serve to control versions and parameters and should, in principle, be adopted in the respective CATIA model with the option *"Duplicate data in CATIA Model"*.

5.1.1.4 Materials

Materials must be assigned from the Library for every single part (CATPart). If a material is assigned, note that materials in the single part must be stored in the first hierarchical level. This is the only way to guarantee the process reliability of the parts. It is prohibited to assign (different) materials to individual objects.

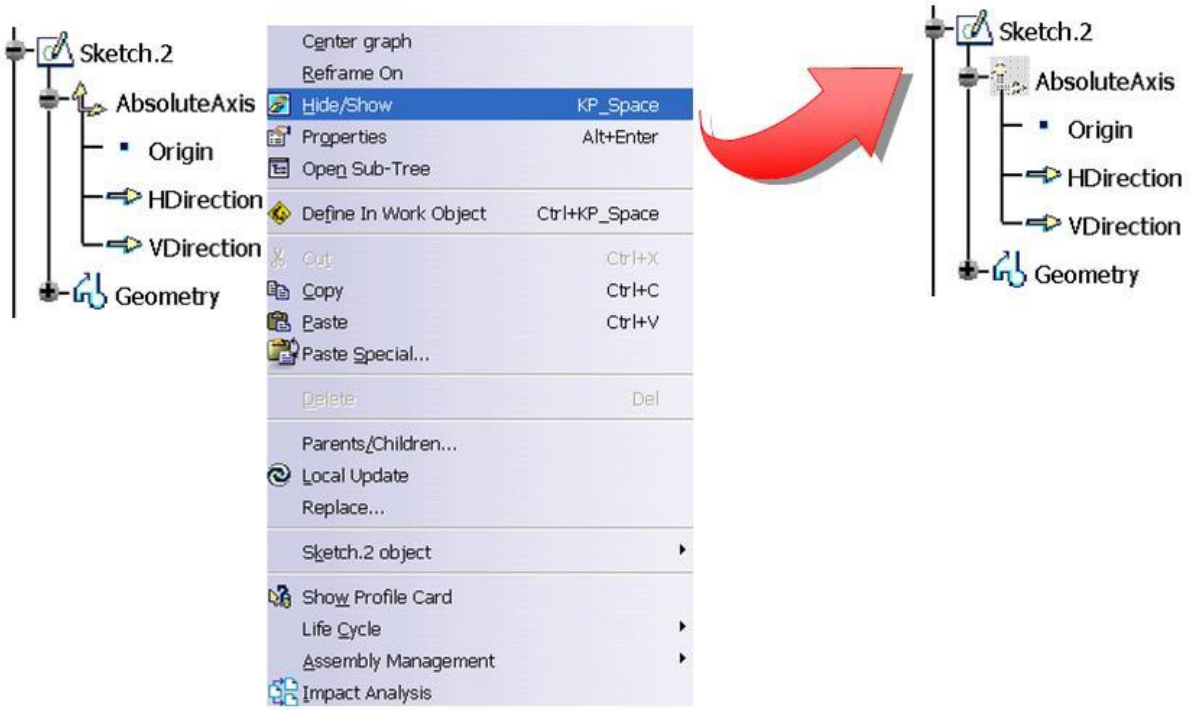
An exception to this rule is the ALLCATPart. Here, a body-related material assignment is sometimes necessary for correct drawing derivation of the 3D geometry. This procedure is shown below.



5.1.2 Sketcher basics

When working with the Sketcher (creating & editing), the origin and orientation of the sketch-internal axis system (the yellow “H” and “V”) must be defined appropriately (case-by-case). It makes sense, with this form of sketch, to reference dimensions on “H” and “V”.

The Sketcher’s yellow axis system must be hidden in the 3D display for sketches made visible. The procedure to be used to hide the sketch-internal axes involves the node in the structure tree. The advantage of this method is that the axis system is still visible when editing/opening the sketch.



5.1.2.1 Sketcher - Positioned sketches

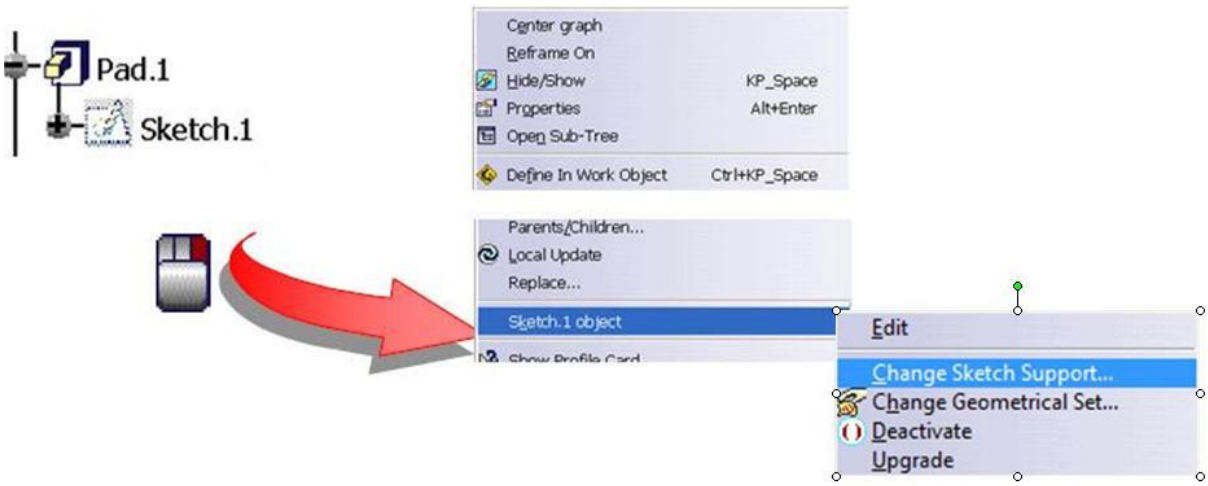
If new sketches are defined within the single part design, the “*Positioned sketch*” function must be used exclusively. This function guarantees a flexible adjustment of the position and orientation of the sketch if any changes are made.

Note the following points when defining positioned sketches:

- Sketches may only be generated on elements of the plane type or on the levels of axis systems and not “*on the fly*” on faces, edges and corner points of solids or faces.
- Both the reference point (origin) and the orientation of the H / V axis must be indicated.
- The origin and orientation must be defined using reference geometries (axis system, point, line, etc.).

5.1.2.2 Sketcher - Amending "*simple*" sketches

Every type of sketch can be amended subsequently. This is done using the command "*Change Sketch Support …*", which can be accessed from the shortcut menu that appears when right-clicking on the structure element of a sketch. With this command you can change the type, position and orientation of a sketch.



5.1.2.3 Sketcher - General information

In principle, all sketches should be defined fully (diagnostic colour: green or ISO-constrained), i.e. there should be no degrees of freedom for individual lines and points. This must be verified and assured using the "*Skizzenauflösung*" (Sketch resolution) or "*Skizzieranalyse*" (Sketch analysis) function.

The following points must also be observed when generating sketches:

- Sketches should be kept as simple as possible. This means:
  - there should only be one profile per sketch.
  - no repetition geometries should be defined in sketches, where possible (symmetry included)
  - holes, chamfers and fillets are realised in 3D using feature functions, where possible.
- No referencing from a sketch to feature elements (e.g. draft, fillet, etc.), if at all possible
- If 3D features, 3D wire-frame geometry or sketches are referenced from a sketch, projections or section views must be created for these.
- Define sketches fully using the fewest possible number of 3D references.
- Spread sketches over the fewest possible number of levels. Re-use existing levels, if at all possible.
- The rotational axis of the shaft and groove is defined within the sketch.

5.1.3 Axes and spatial position

SeatingSystems	***Automotive***
All parts in contact with the component should be designed relative to the point of origin of the vehicle or seat. Locating or centring pins, or (factory) standard parts, are not regarded as parts in contact with the component (for design relative to the point of origin of the seat) as these are usually standard parts designed in production position.	The design method is prescribed by the respective customer/OEM in Automotive. If no stipulations have been made by the OEM, the project manager must be consulted.
	Parts installed for a specific project can be positioned relative to the vehicle origin.
Standard parts should be created at the origin axis system (parts used multiple times!). These are positioned in the *CATProduct* using suitable *constraints* (*offset, coincidence*, etc.).	

In terms of the design of tools and/or tool plates, these are handled in the same way as (factory) standard parts. An individually defined origin can be used that should be based on the tool or part geometry.

In order to guarantee that the definition of assembly constraints in the following steps is as error-free as possible and to make sure that products can be updated, axes should be inserted in holes using the axis function. These elements must be published following their generation.

Part orientation is based on the absolute coordinate system:

- X direction of travel
- Y to the left
+ Z upwards

5.1.3.1 Reference Point System (RPS)

RPS and function points are managed in the designated structures (e.g. in the GRAMMER AG starting model under the *Applications* container). RPS and function points (e.g. welding points) are described and managed as reference geometry with the corresponding functions in CATIA V5. They must be defined in terms of RPS type, position and orientation.

5.1.4 Structuring of CATIA V5 parts

5.1.4.1 General information

The structure tree is described within GRAMMER AG in the CATIA V5 training documentation. For data to be created on behalf of an OEM, this is defined in the standard of the respective OEM.

When naming added bodies, make sure that the designation is meaningful and relevant to the part. Names must not be assigned to the features within the bodies; the designations assigned by CATIA V5 can be retained.

The basic structure distinguishes in its breakdown between finished part, bent sheet metal parts and auxiliary geometry. These areas must be filled accordingly.

Components can be present in different production states that relate to one another and should be archived, e.g. off-tool position or installation position. There are therefore different classes of geometry, such as unmachined part geometry vs. finished part geometry. Many of these production states are the intermediate states of the actual parametric part design and must be saved in individual CATPart files so that these states are documented clearly.

All the geometry elements of these part states (so-called views) that are necessary for the process chain must be managed in the body structure and can be made visible or invisible as required.

The elements of the structure tree are described below:

- In the "*PartBody*", the only sub-element allowed is "*Construction Body*"! The part volume for calculating the weight is only supplied by the main body (=*PartBody*) to the PDM system.

- The "*Construction Body*" contains the actual design of the part. Here, all the elements needed for the design are stored in a hierarchical structure tree.

- The "*Auxiliary Body*" can be used for basic sheet metal part geometry or for unmachined part geometry or to store volume geometry from other models locally in this CATPart.

- "*Auxiliary Geometry*": auxiliary elements or geometry are stored here.

- With the publication of elements, geometric components for a context-oriented definition of a part, for example, are made available (see → Publications).
  - **Important:** Publish part references and name them clearly.
  - Geometry publication is required by some OEMs -> otherwise *assembly constraints* cannot be assigned in some OEM environments, for example.

- Prior to the final CHECK-IN (checking of data into the PDM system for approval), all other structure nodes except for the "*PartBody*" must be hidden!

- Parameters should reflect the design intent and support dependencies of different variables.

- Parameters (e.g. linear dimensions, radii, …) can be generated with mutual dependence using values or formulae.

- "*Applications*" should be used by the designer to store additional information on the part (see → Greasing Lubricating Sealant Screw Locking). This container is also where the elements for the RPS (Reference Point System) method are managed.

5.1.4.2 Structure setup & definition

A good and logical specification tree is important for working with CATIA V5 in a comfortable and change-friendly manner. A well-executed specification tree ensures an error-free design setup, smooth updates and improved clarity in the models.

Many parts usually only comprise a few features. In such cases, there is no need for a special tree setup. All the features can be generated in the structures of the *PartBody* or *Auxiliary Body*. However, if it is likely that a part will be formed from more than 10 or so feature elements, the model must be given a structured setup. The individual features must then be generated in bodies and organised logically. The individual bodies are later added to the *Construction Body* or *Basic Sheet Metal Body* via Boolean operations.

<u>The following principles apply:</u>

- The more features are incorporated in a structure, the more important it is to structure a part with additional bodies.

- When it comes to structuring the design, the production method and assembly activity must be taken into account (part separations, draft angles, draft directions, jointing aids, connection technique, etc.).

As a general rule for the design of a part, the result must be a solid. If the part has been structured fully or partly with the help of faces, it is essential that a solid is derived at the end.

If a part is designed with drafts (cast parts, injection moulded parts, etc.), drafting should be done close to the feature when using the *Draft* functions. The following applies: every feature in the structure receives its drafting operand as early as possible. In principle, complex part drafting at the end of the structure is prohibited.

When filleting with the *Fillet* functions, this should be done close to the feature. Every feature in the structure receives its filleting operand as early as possible. In principle, complex part filleting at the end of the structure is prohibited.

Models must be cleaned up regularly to remove any data not needed. Unused elements must be deleted.

<u>Information on structuring parts:</u>

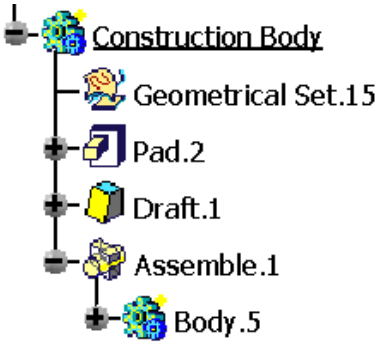
If objects are inserted into the structure, make sure that the following points are taken into consideration when structuring the part:

- Organisation in logical sub-areas: make sure, in particular, that part areas and part machining operations are created via the Multi-Body structure.

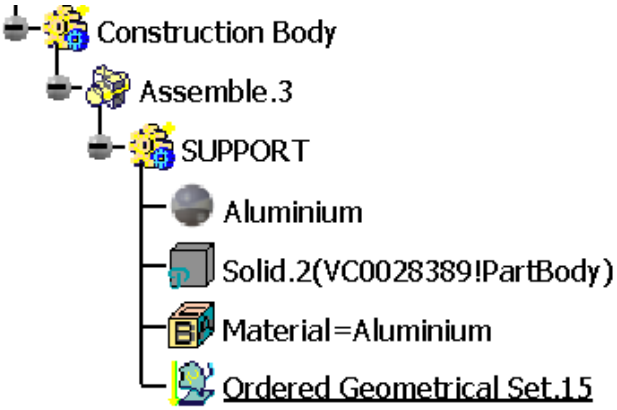
- Structuring via *Body, Ordered Geometrical Set* or *Geometrical Set*

- Auxiliary geometries are managed in sub-structures of the *Geometrical Set* type if they are not stored in the "*Auxiliary Body*" (see figure below). Note whether hybrid design is enabled or not. This depends on whether:

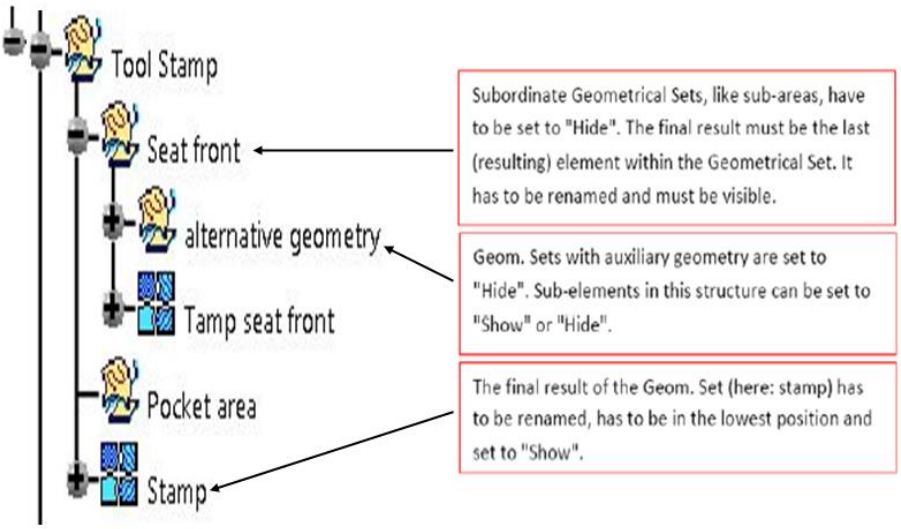
  - a *Geometrical Set* (with non-hybrid design) can be inserted into the body structure to manage auxiliary geometry containers.



  - an *Ordered Geometrical Set* (with hybrid design) can be inserted into the body structure to manage auxiliary geometry containers.



  - a *Geometrical Set* or *Ordered Geometrical Set* (with surface design) must be inserted into the Open Body structure to manage auxiliary geometry containers.



- Result geometries are at the highest level of the structure

- Size and content of a Geometrical Set:
  - The number of features should be limited for clarity's sake. There is little sense in defining an exact upper limit here. Breaking the geometry down into several sets makes it easier to manage the model.

- *Groups* are allowed at GRAMMER AG.
  - Grouping entails a reduced representation of the structure, which can be applied to a Geometrical Set. Grouping enables the *"suppression"* of individual elements in the structure of the Geometrical Set if these are too complex or too long.

5.2 Geometry & structure check

To ensure that CAD data generation is of a uniformly high standard of quality, it is absolutely vital that the design data is checked constantly. This involves subjecting the data to a continuous geometry and structure check and making sure that the CAD data for the respective release version is up-to-date. The tools used for these processes are CATDUA and, among others, the Q-Checker.

Further information can be found in the documents listed under "6 Applicable documents".

6 Applicable documents

Attachment "Sammlung CAD-/PDM-Dokumente / Collection CAD / PDM Documents"	A_015_012

7 Changes