

Handbook for Design Methodology

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1. General

This document deals with the **general, methodical and constructive procedure** for creating both solid and shape geometry by observing the design method.

Adhering to these specifications assures the **data quality requested by Kärcher** and provides easy and expedient creating as well as fast changing.

1.1 Field of application

This document is applicable for:

Development

Design

All suppliers using CATIA V5

1.2 System requirements for CATIA V5

All Parts which are generated for Kärcher have to observe the Kärcher settings. Divergences from these settings may lead to massive errors and are not tolerated.

For CATIA drawings it is mandatory to use the Kärcher *Drawing Standard*.

All system requirements and settings are available at <https://ddx.kaercher.com/ddx>

2. Design Methodology

2.1 Structuring

The structuring is important for the success with parametric designs.

Well-structured designs permit easier modification of the component part and the re-use of partial designs.

2.1.1 Definition of the sub-structures

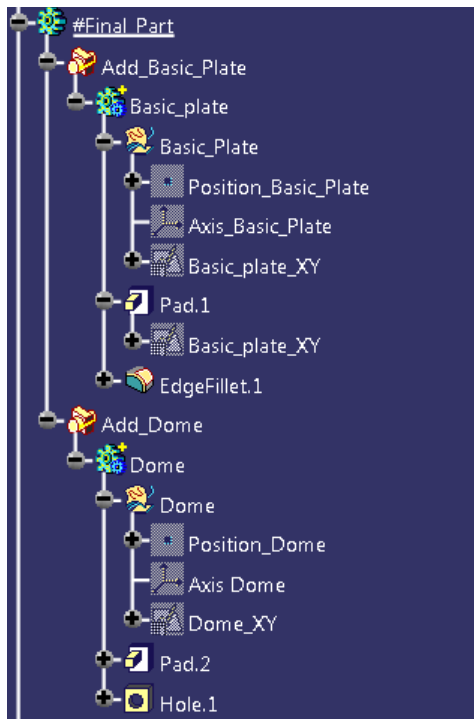
The entire component part is divided into individual sub-structures.

For each component part function an own sub-structure will be established. The sub-structures are to be generated as self-contained designs independent of the development of the other sub-structures. Sub-structures can also be used as own component part or in other component parts by means of copy and paste.

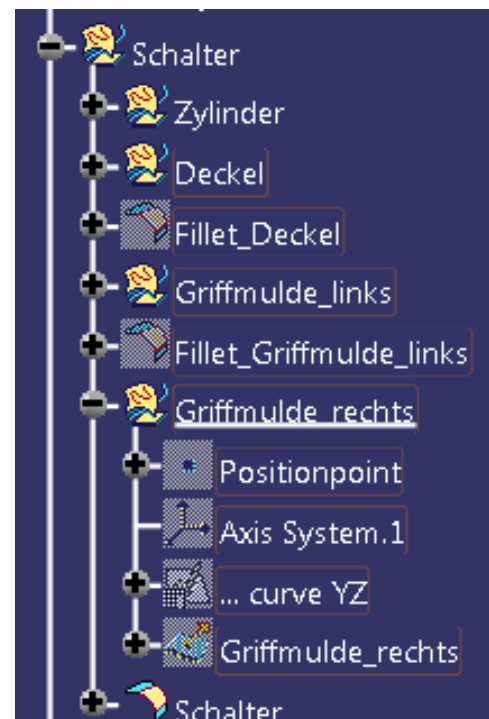
Sub-structures should fulfil the following requirements:

- Documented in an own "**Body**" or "**Geometrical Set**"
- Should be freely positionable using a position point
- Based on concept curves (positioned sketch)
- Each sub-structure should be independent from other ones
- Concept curves are always visible
- Should be given a useful name (in case of a surface result the feature is given the same name as the Geometrical Set)

Solid Geometry:



Surface Geometry:



2.2 Geometry generation in the start part

2.2.1 Basic procedure

“Bodies” are used for solid geometry and “Geometrical Sets” are used for surface geometry. The structuring methodology can be used in the same way for the surfaces and for solid elements.

Combination of “Bodies” and “Geometrical Sets” for solid-part design

For the integration of surface and solid design a dedicated “Geometrical Set” will always be created in each “Body”. All geometry elements needed for the solid design are thus to be stored under the solid body.

If a more extensive surface design is needed for a solid feature, then instead of at least one “Geometrical Set” being situated beneath a “Body”, a hierarchical “Geometrical Set” structure should be developed.

Firm instruction sequences and pre-defined structures

Each “Geometrical Set” begins with the same standardized instruction sequences (position point, plane, and sketch)

For the simplification of the design work, these consistently recurring design sequences are offered in the start model as template. The pre-defined structures can be copied for further sub-structures, both for surface and solid partial designs.

These firm instruction sequences and pre-defined structures thus save work steps while designing.

2.2.2 Concept geometry

Position points	Always hidden
	Never red, orange or black
Axis systems	Always hidden
	Never red, orange or black
	Not current
	Always right-handed
Planes	Always hidden
	Never red, orange or black
	Smaller than 1 mm
Sketches	Always hidden
	Never red, orange or black
	Smaller than 1 mm
	Have a descriptive name
	Include only active geometry
	Always positioned (sliding sketches are not allowed)
	Parallel to a main plane if possible
	show a relative and complete dimensioning ("ISO constraint") to the position point and/or to the prepared auxiliary axes in the Sketcher
	independent of other structures (no direct links)
	in this case the sketches must be named accordingly (e.g. curve xy Ref_for_Dome) and copied to the other structure by using Copy / Paste Special / As Result with Link

2.2.3 External geometry

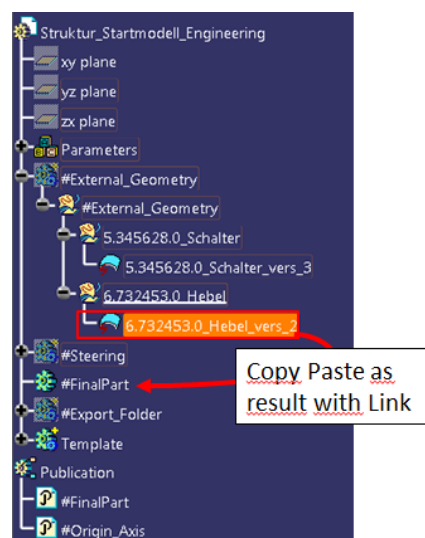
Note: This part of the methodology will change in the near future!

External reference elements such as design geometry or surfaces from the adjacent part will be filed in the “Geometrical Set” “#External_Geometry”. The geometry may only be filed in as explicit geometry without associative connections (no MMLs) to other CATParts (exceptions like e.g. right – left parts are permitted upon consultation). External geometry may not contain unused explicit elements at the release of the component part.

In order to be able to reconstruct the origin of external geometry, the following name convention must be observed.

- Part Number
- Designation
- Version (if possible)
- Name of the geometry elements in the original model

Generally, it is recommended to establish a “Geometrical Set” below “#External_Geometry” for each adjacent component. The explicit elements are filed under this “Geometrical Set” then. This particular “Geometrical Set” is likewise subject to the name convention. This way, a clear allocation is possible, even at a later date. The respective “Geometrical Set” must be created by the user and named accordingly.



2.2.4 Cross reference

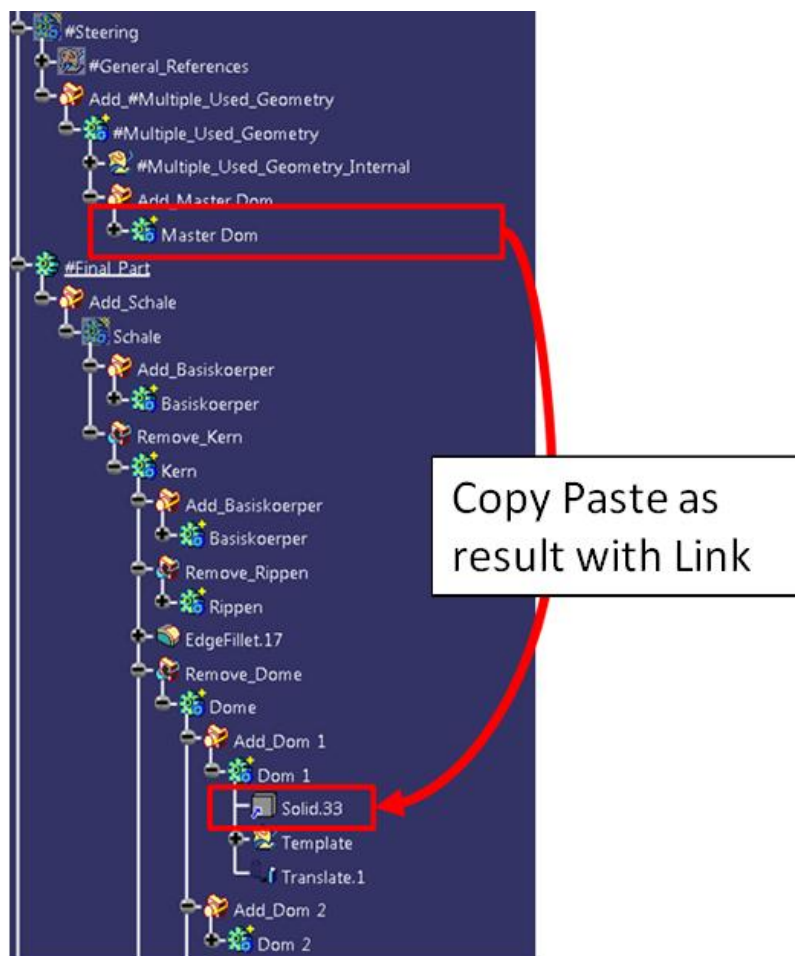
Cross reference means that one element is multiple used in the structure tree. Structures, which are developed thus, are so **highly intertwined** with one another that changing, restructuring, or reusing them will only be possible with an extremely **high expenditure of time**.

In order that the structure remains manageable, the sub-structures should be developed in parallel and independent from each other.

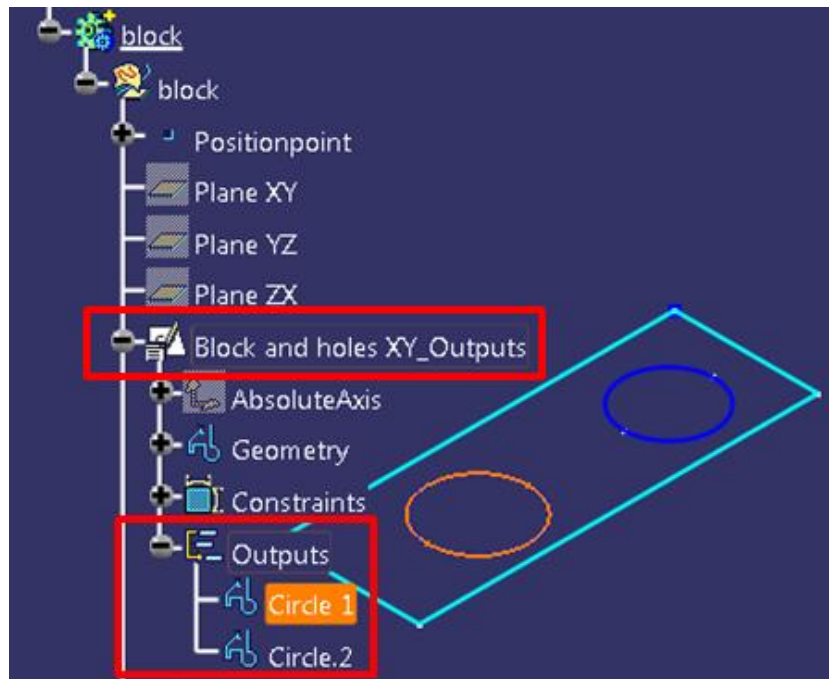
Should elements in a sub-structure be needed from outside of this sub-structure, this element must be filed in **"#Multiple_Used_Geometry_Internal"** and be copied with **"Copy / Paste Special / As Result with Link"** into the sub-structure.

Thus cross references are directly evident in **"#Multiple_Used_Geometry_Internal"** and in the structure tree.

Always put the original element in **#Multiple_Used_Geometry**. The copies with link should be linked directly to the original. (Do not create a copy of a copied element!)



If **outputs** are used, the sketches must be completed with the **addition ..._Outputs**.



3. Standards

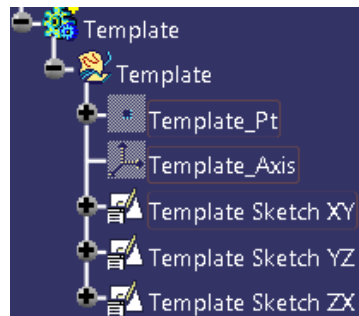
3.1 Methodology Standards

The body „Final_Part“ is used for all geometries which are part of the design.

The specification tree can be arbitrarily deeply structured.

The body “Template” (Add_Template Macro) serves for the definition of further sub-structures.

You can also just copy the „Geometrical Set“ if required. A „Geometrical Set“ can directly be inserted on every place in the specification tree.



Important „bodies“, operations or other important geometries should be given useful and unique names. (Context menu – properties), especially if referenced as external geometry.

All important interim results (e.g. „bodies“) should be published (Tools/Publications).

All information needed to design one sub-structure must be filed into the correspondent ”Body“ or “Geometrical Set“ (exception: superimposed elements like parting elements or references to external geometry)

Prevent selecting „BREP,, (Boundary Representation)” geometric elements.

Create dependencies only if really necessary for your design.

Drafts and fillets should be created as soon as possible in the structure tree

Do not create too much parameter. Limit them to the important ones.

3.2 Industrial Design

The results of the Industrial Design are documented in „#Export_Folder“.

Multi model links in this case are not allowed. Exceptions have to be verified by the CATIA Support Team.

4. *Final condition and Q-Checker*

The file name must not contain any special character. (Exception: - _)

Legal characters: A-Z, a-z, 0-9, - _

Display Accuracy:

In the end the part has to be saved with a Display Accuracy of 0,2 mm. (This setting can be found under: Tools – Options General – Display – Performance)

CATDUA Check and Clean

After finishing the design, the files have to be checked with CATDUA. This tool finds and repairs internal errors, which the user cannot see. These errors cause problems in downstream processes. The Q-Checker automatically starts a CATDUA check and clean procedure.

Q-Checker:

All CAT-files which are sent to Kärcher or saved to the Kärcher SAP-System have to be checked by Q-Checker with the Kärcher Profile. Ko-Errors are not tolerated.

The newest version of the Kärcher Q-Checker Profile can be downloaded on www.ddx.kaercher.com

Questions? Problems? Suggestions?

If you have any questions or suggestions, don't hesitate to contact the CATIA support.

Tel. +49 7195 14-2711, e-Mail: catia.support@de.kaercher.com