Basic guideline for the design of devices and jigs with CATIA V5

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CONTENTS

Preface

This guideline serves to support the introduction of CATIA V5 in the jig design for systems, devices and test equipment and to standardize them across all OEMs.
CAD-documents shall be created at the single OEM and the common suppliers by using CATIA V5 according to the same design methods and shall be structured such to achieve a shared understanding of the data.

Terms

CATIA specific descriptions of functions are in *italics*

Abbreviations:

CATPart	CATIA V5 component (component part)
CATProduct	CATIA V5 assembly (ASSY of CATParts and sub-assemblies)
CATDrawing	CATIA V5 drawing (comprising one or several drawing sheets)
Catalog	CATIA V5 catalogue (collection of pre-defined components)
Component	CATIA V5 Structuring element in the specification tree (without own document on hard discs, either as representation of a component without any geometric equivalent or of an assembly of CATParts, CATProducts or, again, components
DL-Names	When using Dynamic Link Names, the absolute path information on a document are replaced by a virtual name (DL-NAME). Subsequently, only this virtual name will be saved within a reference. A substitution table is provided as CATIA V5 configuration file (*DLNames.CATSettings*); possibly it has to be adjusted prior to using CATIA V5 documents.
CATDUA	Additional application to clear CATIA V5 documents
ZSB or ZB	Assembly (German: **Z**u**s**ammen**b**au)
OEM	Original Equipment Manufacturer (here: the vehicle manufacturers involved)
Adapter	The adapter, also referred to as skeleton or control part, contains both original geometry and references of leading CATParts. An adapter may either contain the input information (input data) for a design or output data for the subsequent designs and processes. It serves as interface in the design process and may have different characteristics to be defined by the design engineer.
SPM	Clamping mark in the main adapter
WP	Working Product
Root-Product	CATProduct including several subassemblies, such as station, working sequence, operation, jig, gripper

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1 Generalities

1.1 Field of application and applicable documents

This standard contains specifications as to working with the CAD-System CATIA, version 5 in the design of systems, devices and test equipment and serves as a basis for the guidelines of the single OEMs.

Further OEM-specific information are available via the customer and can be found on the OEMs' supplier websites. All these specifications have to be kept.

Audi:	www.vwgroupsupply.com ; https://extranet.audi.de/
Volkswagen:	www.vwgroupsupply.com
BMW Group:	www.b2b.bmw.com
DaimlerChrysler:	https://engineering.supplier.daimler.com

2 Conventions and defaults

2.1 CATIA V5 Settings

By all means, publications will result in a more stable design and they shall be continuously used wherever this is useful. To that end, the below mentioned settings shall be made and must not be changed during the design process. In **Figure 1** all mandatory settings are marked in red; **Figure 2** shows the additionally recommended settings.

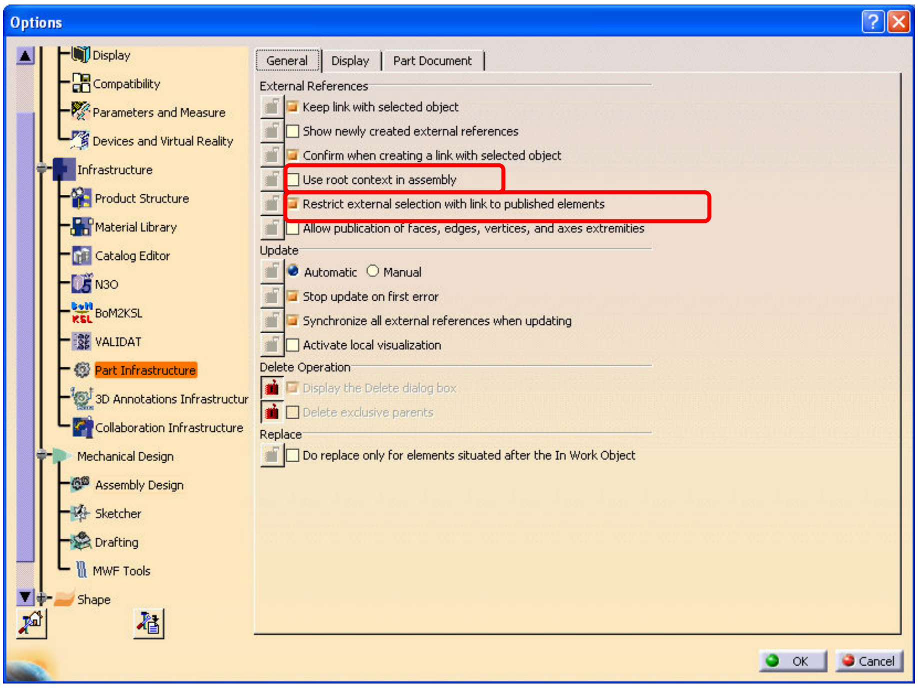


Figure 1 - Tools/Options-dialogue "Part Infrastructure" concerning links

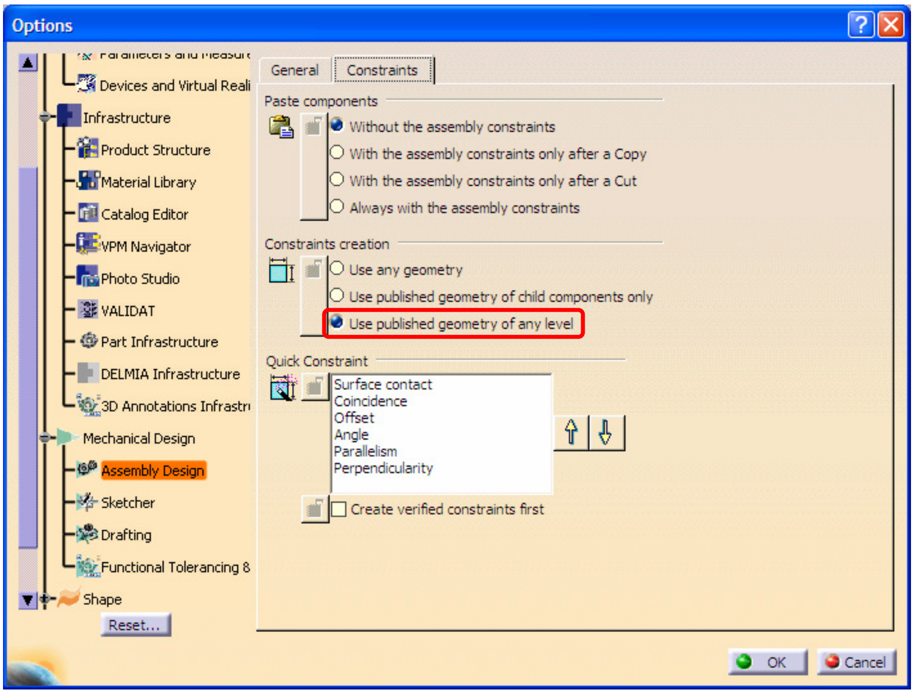
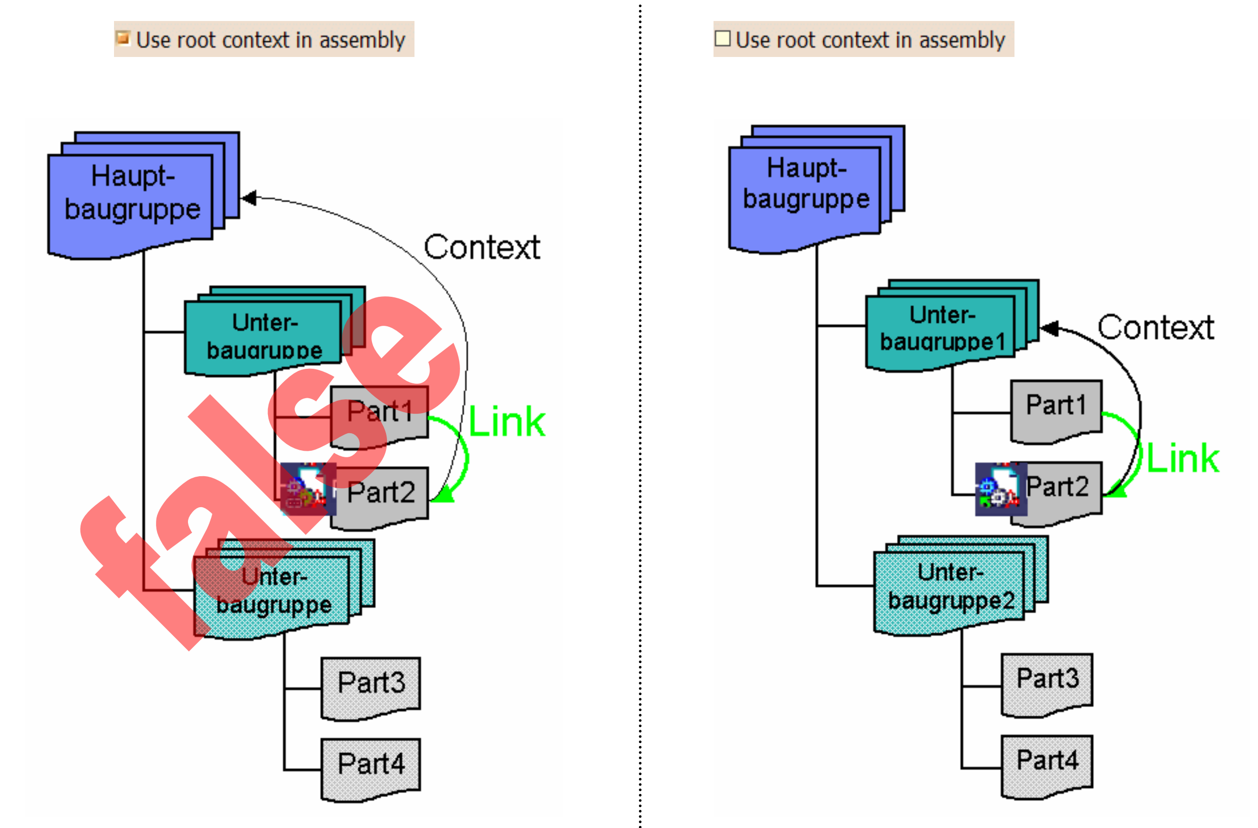


Figure 2 - Tools/Options-dialogue "Assembly Design" concerning constraints and publications

The option “Use root context in assembly” has to be disabled. This option defines that when creating across-the-component links within an assembly with several sub-assemblies, each time the uppermost CATProduct (root-product) will be used. When this option has been disabled, the next higher CATProduct will be used as context. This is also shown in the below comparison (**Figure 3**).



Main assembly opened:
→ Part2 „**in context**“
(green toothed wheel + chain symbol)

Sub-assembly 1 alone opened:
→ Part2 „**out of context**“
(brown toothed wheel + red flash)

Main assembly opened:
→ Part2 „**in sub context**“ to subassembly
(white toothed wheel + green arrow)

Subassembly 1 opened alone:
→ Part2 „**in context**“
(green toothed wheel + chain symbol)

Figure 3 – Comparison of context behavior of CATParts in CATProducts

2.2 Naming conventions

File names may only contain capital letters, numerals and underscores [A to Z, 0 to 9, _].
OEM-specifically, also the hyphen is permitted [-].
Space characters, umlauts and special signs are not allowed. Space characters have to be replaced by underscores. OEM-specific regulations exist as to document name lengths. Part Number and Part name (File name) have to be the same (see **Figure 4**). Also an OEM-specific regulation exists as to instance name equalities. File names must not be renamed on operating system level.

The same specifications apply for object names in CATIA V5 documents (e.g. Features, Bodies, Geometrical Sets, Parameter, Relations). Here, only lower case letters, dots and space characters [a-z, .,] may be additionally used.

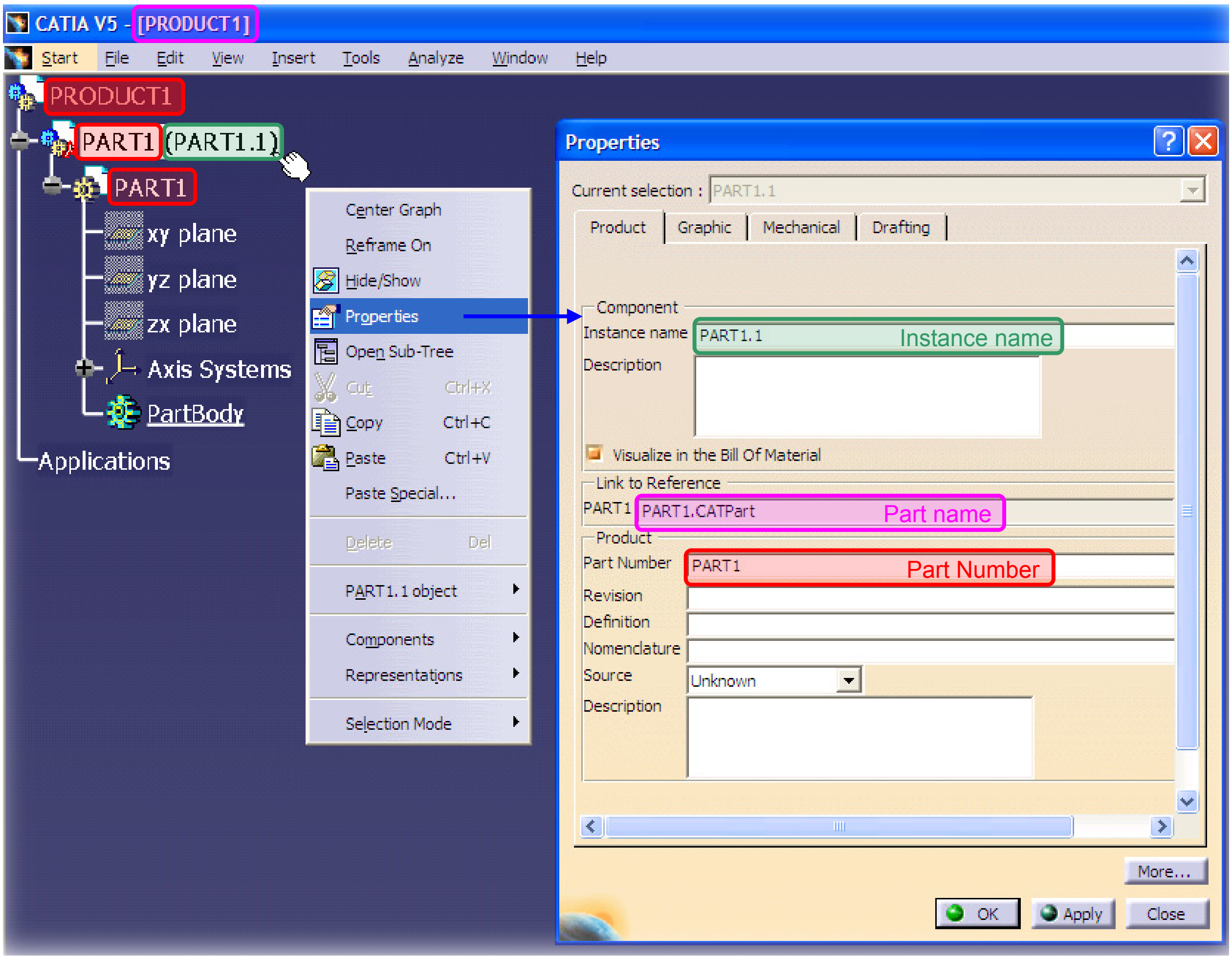


Figure 4 - Part Number, Part name and Instance name of V5 documents

If geometrical elements or features are renamed subsequently, the name should be structured such that the element type is contained in the name. The element name has to begin with a type abbreviation followed by an underscore. The below list shows the most commonly used:

Type abbreviation	Element type
AXS	Axis system
PT	Point
LN	Line
CRV	Curve
PLN	Plane
SUR	Face / Surface
PB*	Part Body
BRP*	BRep-elements (without own representation in the tree)

* designation of element types only applies to Publications being created form a PartBody or BRep-element respectively.

Table 1 – Assignment of element types and element type abbreviations

Thus, the element type is immediately obvious also with Publications. Except PartBodies and published BRep-elements it has to be strictly observed that the Publication-Name and the element name are the same. If elements are subsequently changed via the Tools / Publication dialogue equality of names can be ensured through a corresponding option (see **Figure 5**).

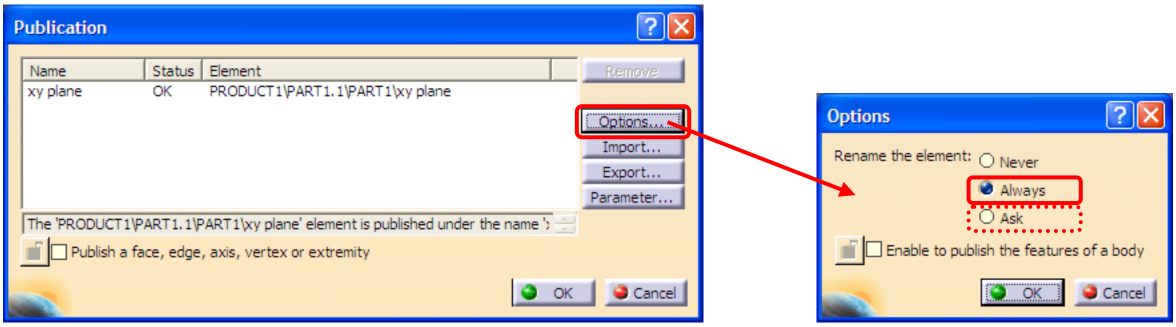


Figure 5 - Tools / Publication Dialogue with options

2.3 Colors

To ensure a simple, visual evaluation of the models for production of parts, the colors listed in **Table 2** and the RGB values have to be defined and used as custom colors (see **Figure 6**). In addition to the unchanged 48 Basic Colors, exactly 16 Custom Colors are available. The colors serve to designate different machining types. Parameter-details, however, are not differentiated by color (e.g. data concerning position tolerance, thread pitch etc.). Machining attributes are saved via parameters (own parameters of CATIA V5 hole features or specifically defined parameters with special bores).
NC processing sequences or NC work piece types are not differentiated by color.
Unprocessed surfaces to be left in the standard color of the Solid. They must not be colored.

Machining type	Comments	CATIA V5	RGB values
Areas depending on the method / *Contour-machining in the area of devices and systems construction*	All machining operations which are not performed acc. to Solid model, but using special data record / *Machining component-related surfaces acc. to Solid or surface data record*	Olive green	175,255,175
Precision machining	*Grinding etc. or specials*	*Ivory*	*255,255,175*
Smoothing	Fine machining (additional information provided in operating instructions)	Rosa	255,175,175
Rough working	Rough machining	Reddish brown	095,000,000
Fitted bore H11	Also special milling processes	Blue	095,095,175
Fitted bore H8	Also special milling processes	Lila	095,000,095
Fitted bore H7	Also special milling processes	Blue	000,000,255
Fitted bore H6	Also special milling processes	Dark blue	000,000,095
Threads	Metric right-hand thread acc. to DIN / ISO	Yellow	
Fine threads	Metrical fine thread (right-hand) acc. to DIN / ISO	Light orange	255,175,000
Spiralock / special thread	All other special threads	Orange	255,095,000
Special bore / stepped bore	Complex bores, bore combinations	Magenta	255,000,255
Bores / simple bores	Through holes etc., other counter bores	Cyan	000,175,175
Areas of change	Sphere or Solid with transparency 192	Light blue	000,127,255
ASSY-bores	*Treatments/processing in assembly*	White	255,255,255

Table 2 - Coloring in CATIA V5

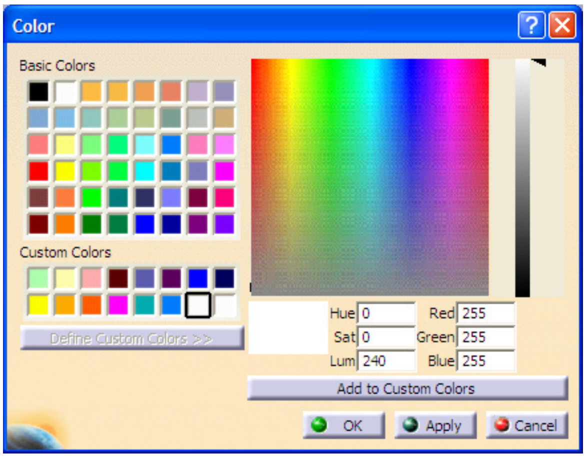


Figure 6 – Defining colors with Custom Colors in CATIA V5

Movable parts are shown in transparent mode in their corresponding opened position. To that end, transparency has to be set in the instance level ensuring that the CATPart itself remains unchanged.

2.4 Drawing derivation

OEM-dependently, for the drawing derivation, standard CEG_1 or respectively CEG_2 has to be used. This way, cross-factory settings such as line thickness, arrow types, fonts, dimension style etc. are ensured.

The ceg_1.xml and ceg_2.xml files with these settings may be also downloaded by partnership companies from www.ceg.de Aktuelles (News), CATIA V5 Standards.

A Start- or original CATDrawing or Catalog document with own drawing frames and drawing stamps has to be requested by the corresponding OEM.

2.5 Data format

CATIA V4 data or other CAD foreign formats must not be directly integrated into the tree or implemented as instance.

Using data in foreign formats requires complete or partial migration or respectively conversion. Only CATIA-V5 native data are accepted as data format for the design.

2.6 Data quality, model preparation and archival

Generally, in final storage state, all relevant parts have to be in „Show“ mode and the device / tooling has to be fully visible.

Ghost links within CATProducts have to be removed using „*CATDUA*“ (callable via Tools / Utilities, Desk or as Batch). Ghost links are analyzed via the command „*Send to*“. Documents not having been used up in the design or linked in the CATProducts must not exist here.
When comparing two version states of a CATPart, CATProduct or respectively CATDrawings, one status has to be created with New From.
Given the different archival systems, the rules of the corresponding OEM have to be considered.

3 Structuring of CATIA-V5 assembles

3.1 Generalities

Basically, the structure of the device (and the bill of materials) has to be taken as a basis for the product structure. All models and references belonging to a CATProduct have to be saved to the same directory of the CATProduct or to a sub-directory contained therein.
With a directory-based method of designing, a directory structure which follows the product structure should be preferred (see **Figure 7**).

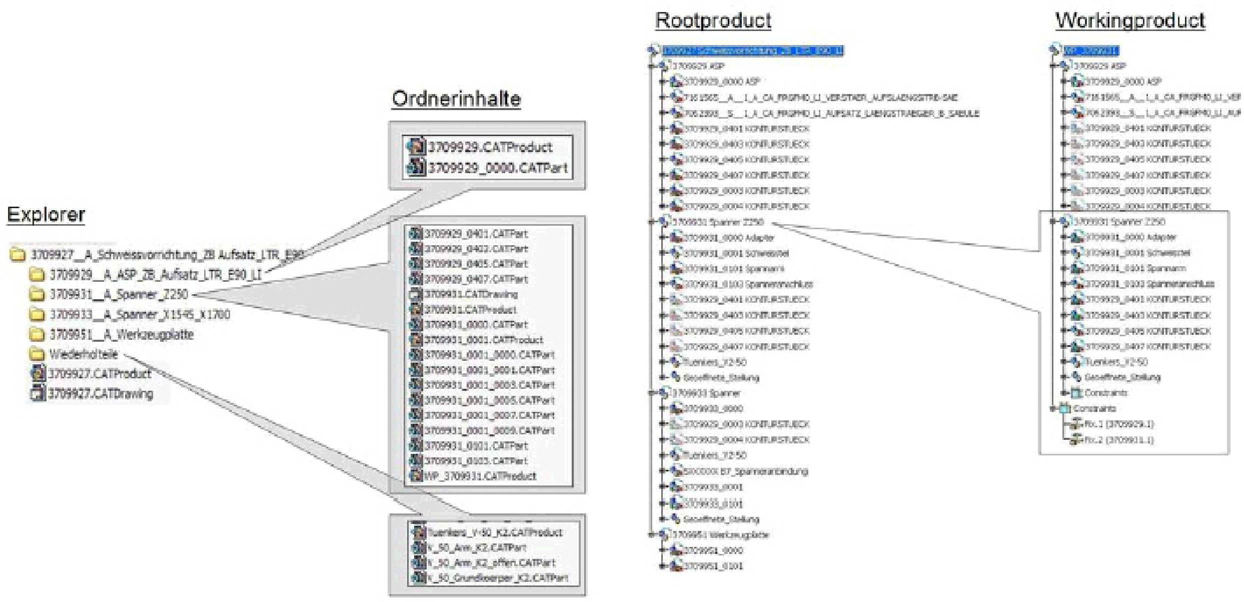


Figure 7 - Example

With a procedure being based on DL-names, please decide how to proceed in each individual case by considering the expenditures.

3.2 Input data and data preparation

A main adapter has to be constructed for each device. The following elements have to be created in this adapter:

- Copy of the component geometry
- Clamping points
- Clamp marks
- Control geometry of clamping points

Here, one may work with "PowerCopy" creating the clamp mark and the control geometry of the clamping points. The Open Bodies or respectively Geometrical Sets containing the control geometry of the clamping points are designated „SPM_", followed by a four digit number (example: SPM_0003), see **Figure 8**. Please refer to the corresponding OEM-specific guidelines as to the procedure to be applied when creating the main adapter.

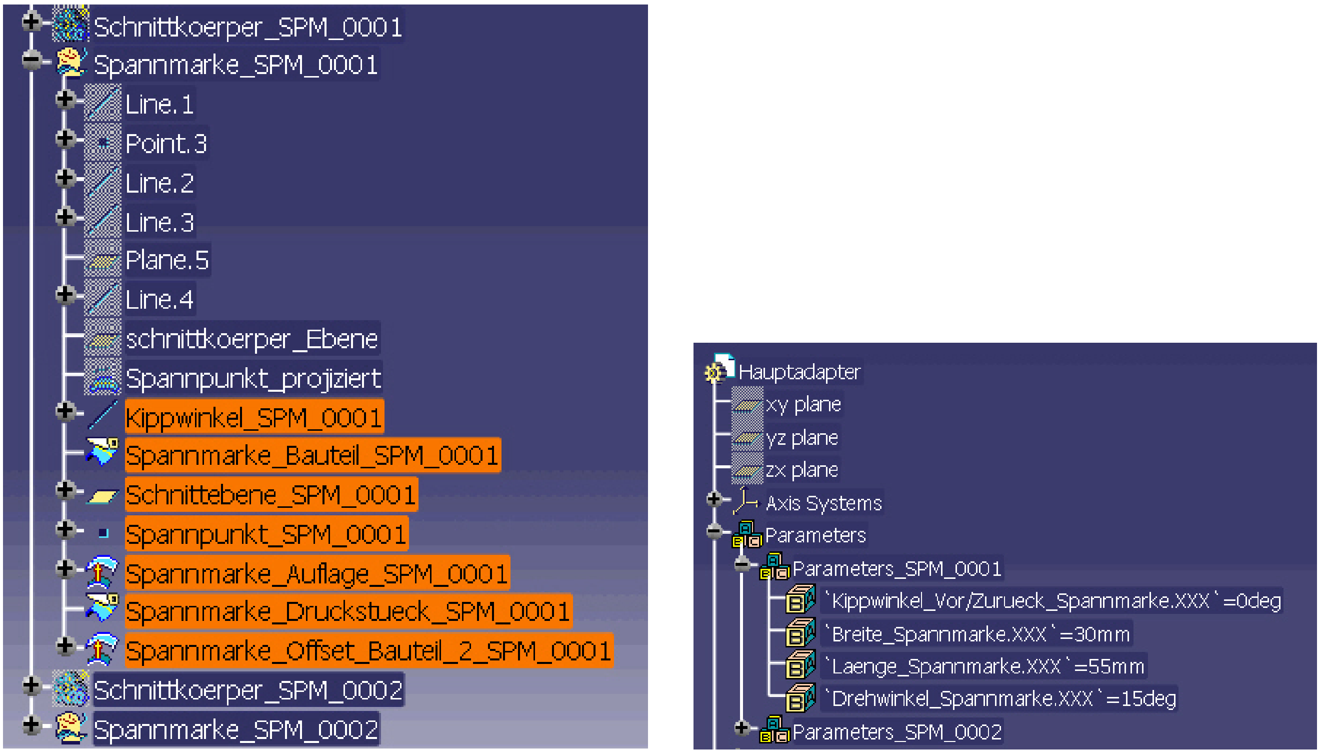


Figure 8 – Result elements of PowerCopy

3.3 Control by adapters

Adapters are used with each CATProduct, where at least one component is associative to another part outside this assembly.

Compared with a free assembly design (via assembly constraints), the control by adapters (also called skeletons or skeleton models) offers the following advantages:

- Documented, hierarchical reference- or link flow (only from the main adapter to the assembly adapters and from the assembly adapters to the parts). This guarantees a clearly structured, hierarchical structure.
- Avoiding so-called loops (reference loops)
- Easy handling when exchanging/replacing and inserting controlling elements (e.g. vehicle components)
- Modifications are simplified, because control elements will only have to be searched in the adapter

Adapters are controlling CATParts within assemblies. Via these CATParts, geometry- and position references are allocated for control within the root product and the sub-assemblies. The major part of these CATParts consists of “Wireframe and Surface” elements (points, planes and lines) which – if the situation requires so – are backed up by a body of rules and regulations (rules, checks and formulas).
Differentiation has to be made between a main adapter (e.g. ASP) defining how the vehicle geometry and components of the device depend upon each other, and the component adapters which – on the component level – define the constraints between the main adapter and the single parts of the assembly.
With a main adapter, the following elements are generated in the Geometrical Sets:

- The geometrical set “External References“ contains geometries that are derived form the required vehicle geometry. This body has to include all elements used by the device for positioning, cutting, etc. The number of these elements, however, has to be kept at the lowest possible level. This reference geometry serves to decouple the device design from the vehicle geometry. The elements of the vehicle geometry are copied to the main adapter.
- „Konstruktionselemente“ (= design features) includes all auxiliary geometries required within an adapter to create the „Output“-elements. Via „Konstruktionselemente“ (= design features) assemblies are controlled within stations.

- „Output" ("Weitergabeelemente" = transfer features) includes those elements being required by subordinate/minor assemblies for defining their geometry and position. All elements of the gemometrical set "Output" ("Weitergabeelemente" = transfer features) have to be published.

Assembly adapters and main adapters are identical in structure. In the assembly adapter, the generation of the link to the main adapter is of significant importance to guarantee the reference flow according to chapter 3.4 of this guideline.

Result: the main adapter controls the assembly position to the component via the assembly adapter.
The normal start model for parts also serves to create an adapter (see **Figure 22**).

Important note:
All products (assemblies) in the root directory the position of which is determined by adapters have to be fixed in the vehicle grid by means of the "Fix" constraint to prevent them from being displaced /shifted by mistake, e.g. by using the compass. This applies to all kinds of adapters. Any errors caused by an erroneous displacement/shifting of non-fixed Parts or Products can be only found out and eliminated by an extremely high expenditure in time.

Exceptions from control by adapters are only possible after prior consultation with the technical designer in charge and in case of standard parts. They are attached to the adapter or to other parts by means of constraints (free assembly design).

3.4 Link flow

3.4.1 Assembly-overlapping link flow

Figure 9 again shows in a schematic view the reference flow in a jig that is controlled by means of an adapter. Published elements are taken out of the main adapter that are relevant to the entire construction (e.g. clamping points, clamp marks, mounting plane tool, rotating axes etc.)
Exchanging reference elements between two or several assemblies has to be effected exclusively via the corresponding assembly adapters. To avoid so-called "loops", strict adherence to the reference flow has to be guaranteed (e.g. hole pattern/drilling pattern of an assembly to the tool plate).

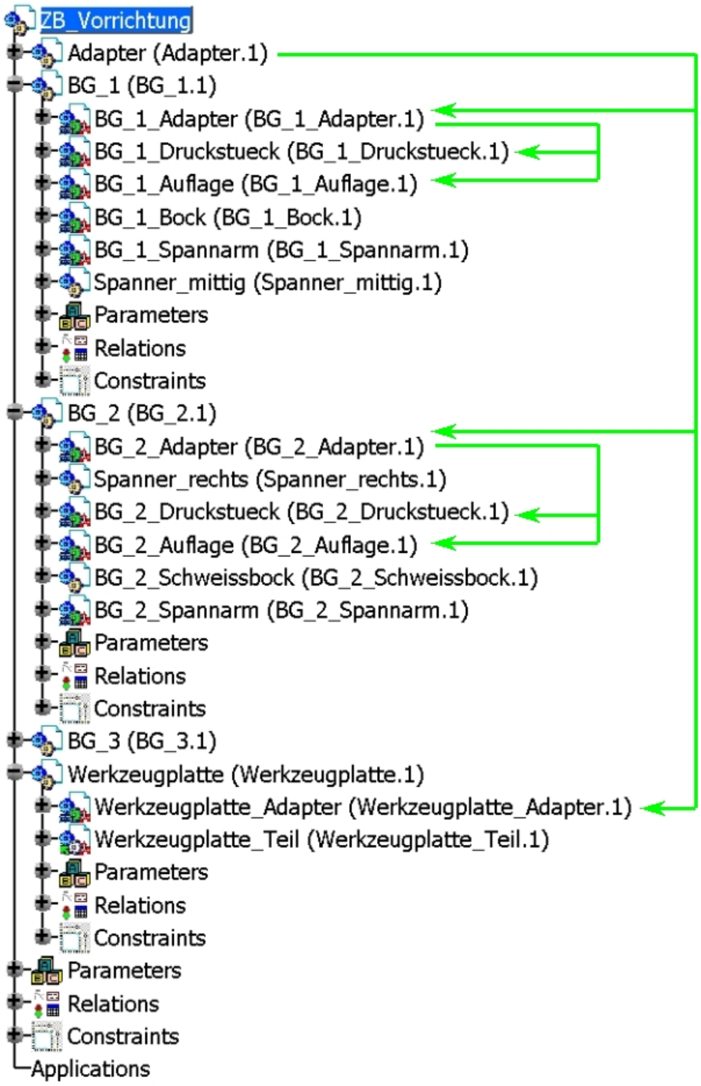


Figure 9 - Reference flow in schematic view only between adapters

3.4.2 Link flow within an assembly

Within an assembly, Parts are allowed to directly exchange their published references without implementation of the assembly adapter (see **Figure 10**). E.g. surface connecting contour of a pad & pressure pad to the connection part (here clamping arm)

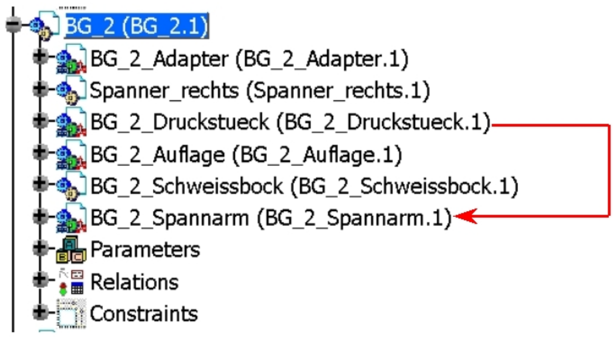


Figure 10 – Reference flow in schematic view without adapter

3.4.3

Referencing tool plates

Basically, tool plates have to be positioned to the vehicle grid. In case the tool plates are defined in the sketcher with regard to their shape and position, thus, their contour must not refer to assembly references (surface connecting contours, hole patterns/drilling patterns). In this case, their position has to be determined with reference to the coordinate system of the origin.
The hole pattern/drilling pattern of the assembly has to be transferred to the tool plate/ screw-down surfaces by means of a published surface connecting contour or sketch with points. Controlling via the assembly adapter is absolutely imperative (see **Figure 11**).

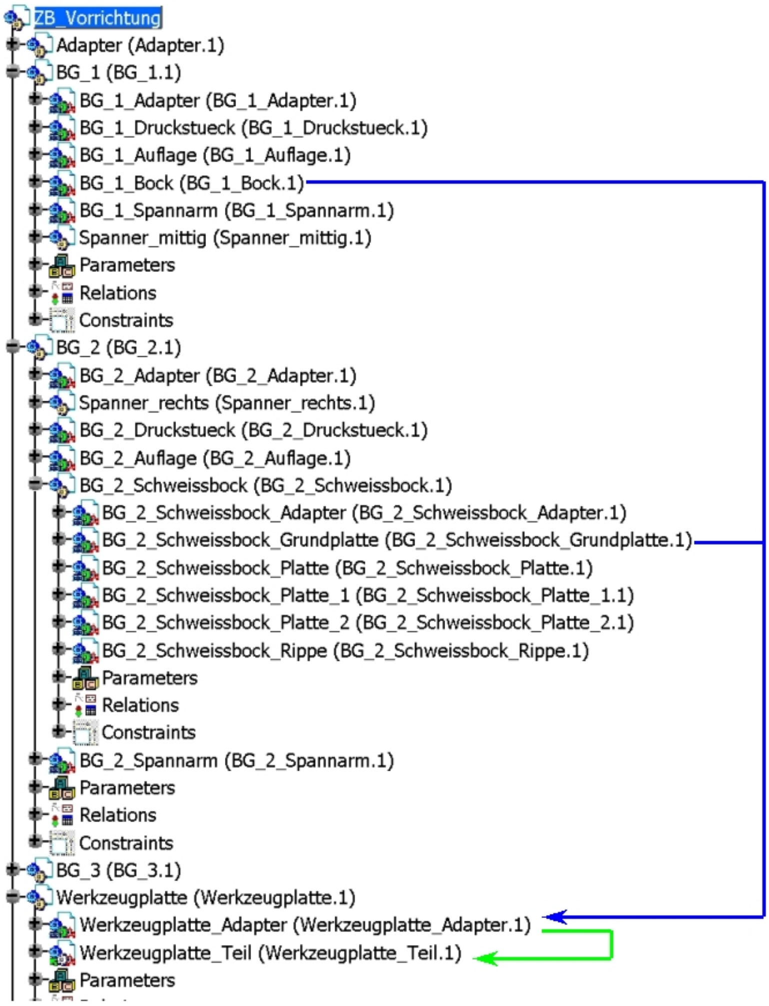


Figure 11 – Reference flow in schematic view, tool plate

3.4.4 Reference flow when designing a gripper

Since the handling design often starts with defining the robot connection and the installation directions or flanges, the gripper frame represents the first partly controlling CATProduct. In this case, the reference flow from the clamp units to the gripper frame is impeded (creation of loops). The surface connecting contours and hole patterns/drilling patterns thus have to be transferred from the frame to the clamp unit products.

Recommendation:
All controlling geometry elements for frames and clamp unit products should be defined in a higher-ranking, superset adapter (gripper adapter in analogy to the main adapter).

3.5 Integration of NuW (NuW = standard- and repetition parts)

This topic is regulated specific by each OEM, refer to supplementary guidelines.

3.6 Working Products

If a new assembly should not be created in context of a root product (overall device), it is recommended to create a WorkingProduct. This method allows designing a new assembly outside the overall device without being in their context. Thanks to using a WorkingProduct, two technical designers can work on one device (root product) at the same time and without any risk.
First, a new Working Product is created whose name has to contain the abbreviation „WP". Using the option "Insert Existing Component", the reference models important for design (e.g. main adapter, disturbing edges, guns etc.) are loaded into the WorkingProduct.
Now it is possible to create the assembly as such in context to the WorkingProduct. A controlling adapter model is created for this assembly. The single reference elements of the main adapter are copied to the adapter model of the new assembly by means of a link.
After completion of the assembly, the latter can be inserted into the Product of the overall assembly by means of the " Insert Existing Component" option. Now, there exists a direct link between the overall device and the assembly. In the overall device, the adapter model of the assembly is marked by a brown gearwheel (not in context to the overall device). In the WorkingProduct, it is marked by a green gearwheel (in context to the WorkingProduct). (see **Figure 13**). Once having inserted the assembly into the CATProduct of the overall device, the Context of the assembly adapter has to be modified such to correspond to match the CATProduct of the overall device. To that end, the function *„Define contextual links"* on the Edit/Components menu has to be used (see **Figure 12**).

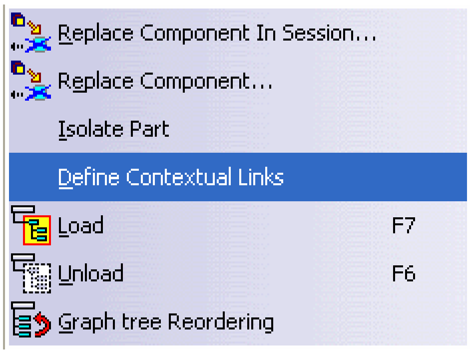


Figure 12 - Modifying Context-Links

Modifications (inserting or deleting Parts) to the assembly are only allowed to be made in the WorkingProduct, but not in the root product. Otherwise there is the risk of generating ghost links in the WorkingProduct. This means for example that deleted Parts will be still searched for in the WorkingProduct
Furthermore, publications must not be created in the Working Product and it is not permitted to derive a drawing from the Working Product.

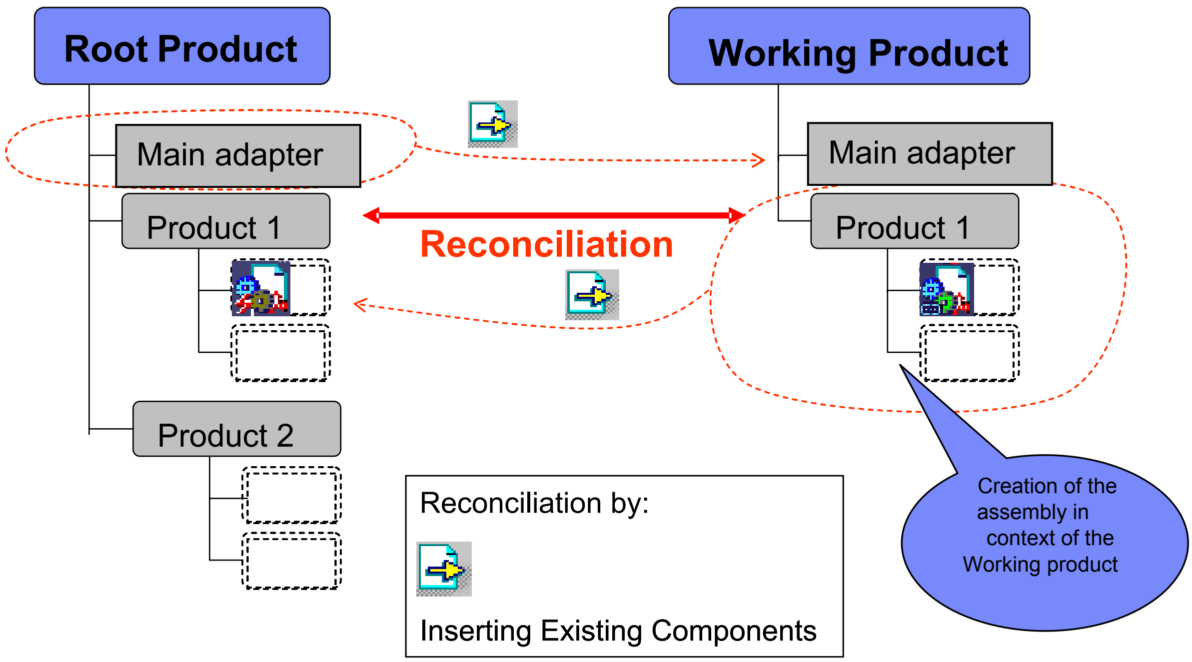


Figure 13 - Reconciliation Working Product / Root-Product

3.7 Mirroring of Parts and Products

3.7.1 Mirroring assemblies

In general, there are two possibilities to mirror assemblies (CATProducts):

- The *„Symmetry"* function of the Workbench Assembly Design in CATIA
  Will not be considered in detail and not to be used either!
- Mirroring using a macro

Mirroring of assemblies has to be used with scopes of devices being symmetrical beyond the centre of the vehicle and with devices for the right side partly being different from the left side.

Application:

First, the assembly to be mirrored has to be duplicated (2. Instance). Select this instance (see **Figure 14**) and, under Tools/Macro/Macros execute the mirror macro „*OEM_Mirror_V2.catvbs* " (see **Figure 15**). After selection of any mirror plane the assembly or the single part to be mirrored has to be chosen. Afterwards the mirroring is directly performed by modification of the transformation matrix. (If the usage of the catvbs macro is prevented for any reason, the previous version „*Mirror_an_ZX_all.CATScript*" (Copyright 2003) may be used anymore without restrictions. In the input mask, one of the 3 main planes XY, XZ, YZ may be entered as mirror plane furthermore.) Once having acknowledged, the mirrored image of the instance of the assembly is provided (see **Figure 16**), which, subsequently, has to be fixed. Associativity to the original assembly is ensured.

With the „right" assembly having any portions not being symmetric to the „left" one, instead of instancing, this assembly has to be saved under a new name. Subsequently, the macro will be executed in the new assembly with the scopes to be mirrored. Thereafter, those portions that deviate from the symmetry may be created.

The macro can be found at the OEMs' supplier websites (see chapter 1.1). It is also available from the customer.

Having used the mirror macro, modified methods have to be applied with subsequent drawing derivations; for example, detailed drawings of the assemblies mirrored with the macro cannot be created with the correct, mirrored geometry presentation. ***Therefore, using the macro in any case requires prior agreement from the customer.***

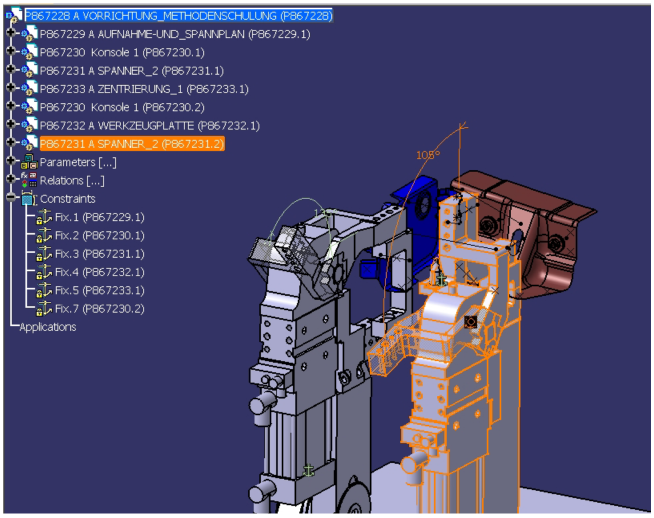
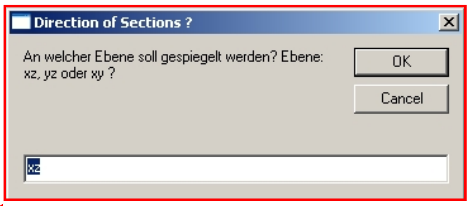


Figure 14 – Creating second instance



Hint:
The new macro does not use an input mask anymore, but provides the following messages in the CATIA V5 status bar (in the lower left corner):

Select symmetry plane
Select Part or Product to be transformed

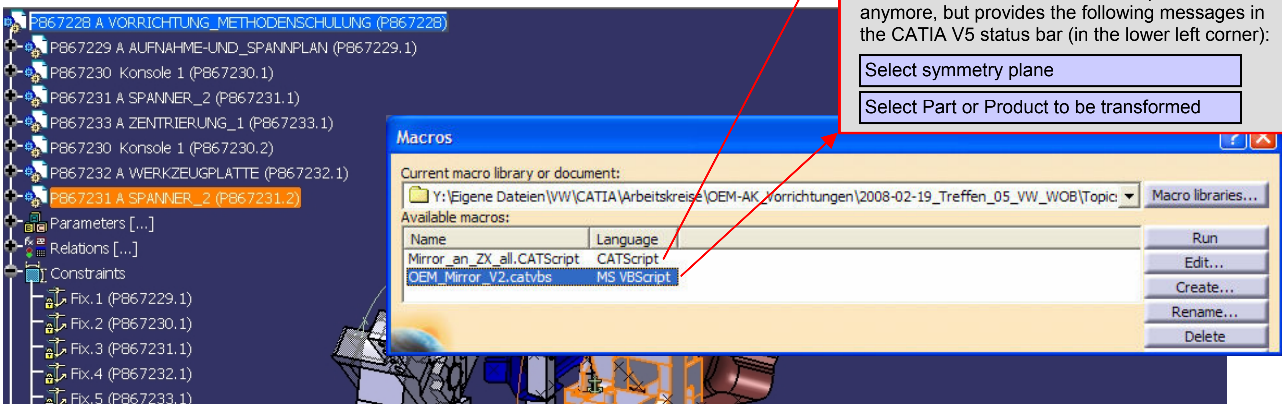


Figure 15 – Executing mirror macro

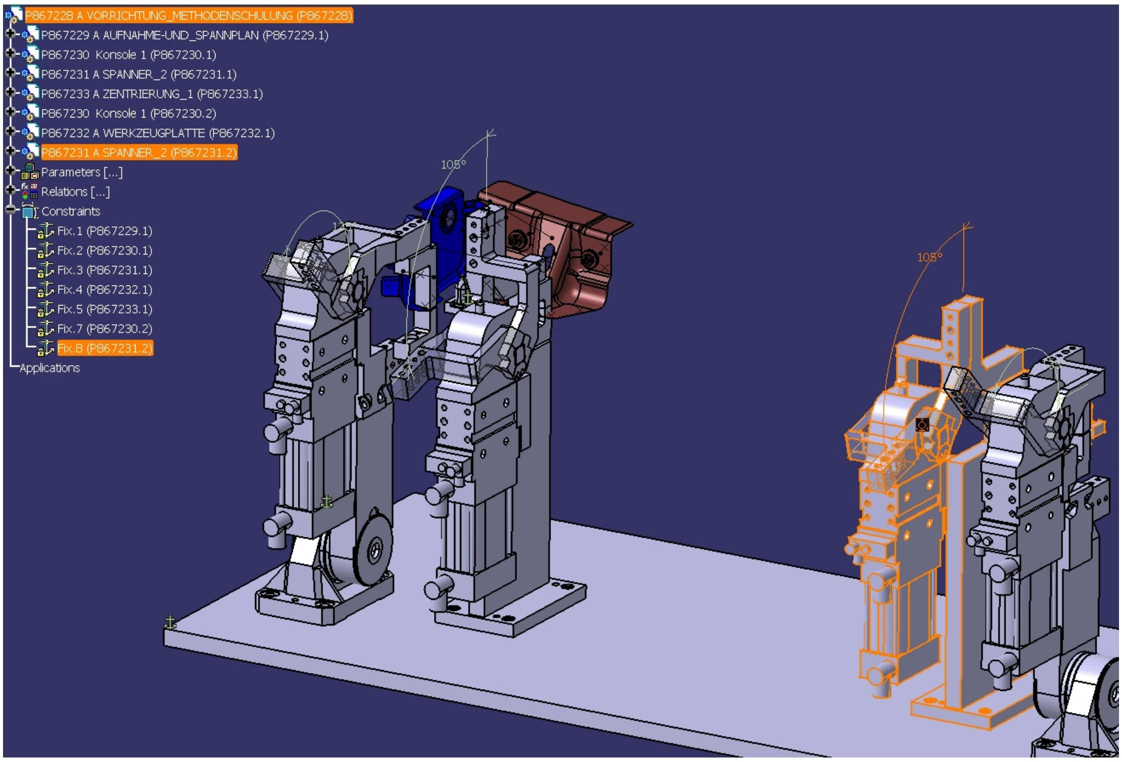


Figure 16 – Result with manually added FIX-Constraint

3.7.2 Mirroring single parts within a CATProduct

Single parts may be mirrored as follows:

- „*Symmetry*" function of Workbench Assembly Design in CATIA
  Will not be considered in detail and not to be used either!
- Mirroring using „*Symmetry*" function in Part Design

These mirroring processes should be applied only with exactly symmetrical components.
First, the CATPart for the mirrored side has to be created. In the „original part", the PartBody has to be published and copied (see **Figure 17**). It can be pasted by means of the function *Paste Special/As Result With Link* (see **Figure 18** and **Figure 19**). Subsequently, the pasted body may be mirrored to any plane by means of the function *Insert/Transformation Features/Symmetry* (see **Figure 20** and **Figure 21**). The part is associative to the original.

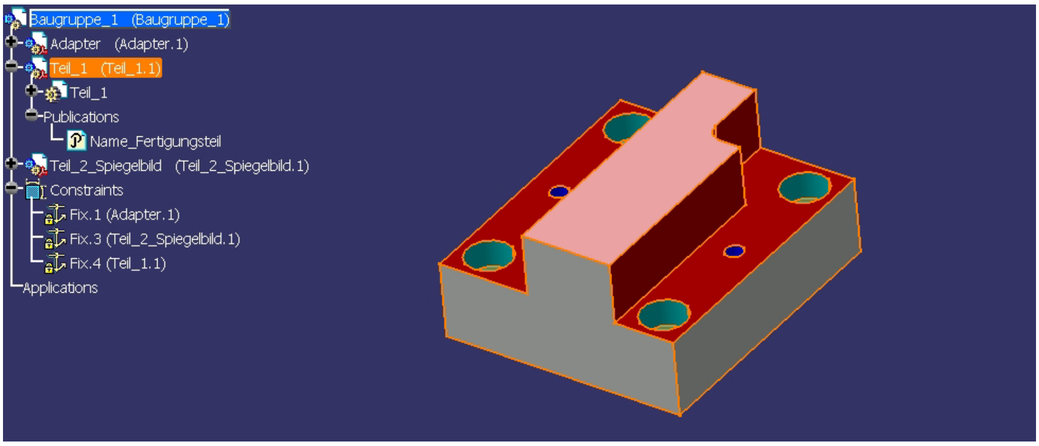


Figure 17 – Initial situation: Part with published PartBody

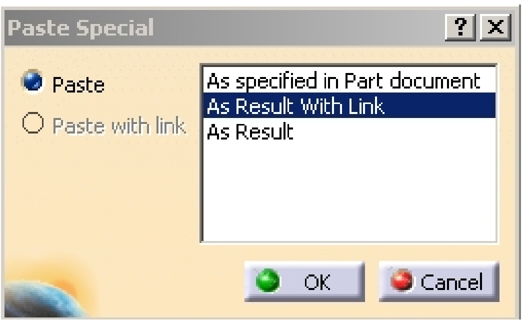


Figure 18 – Option to be selected under `Paste Special`

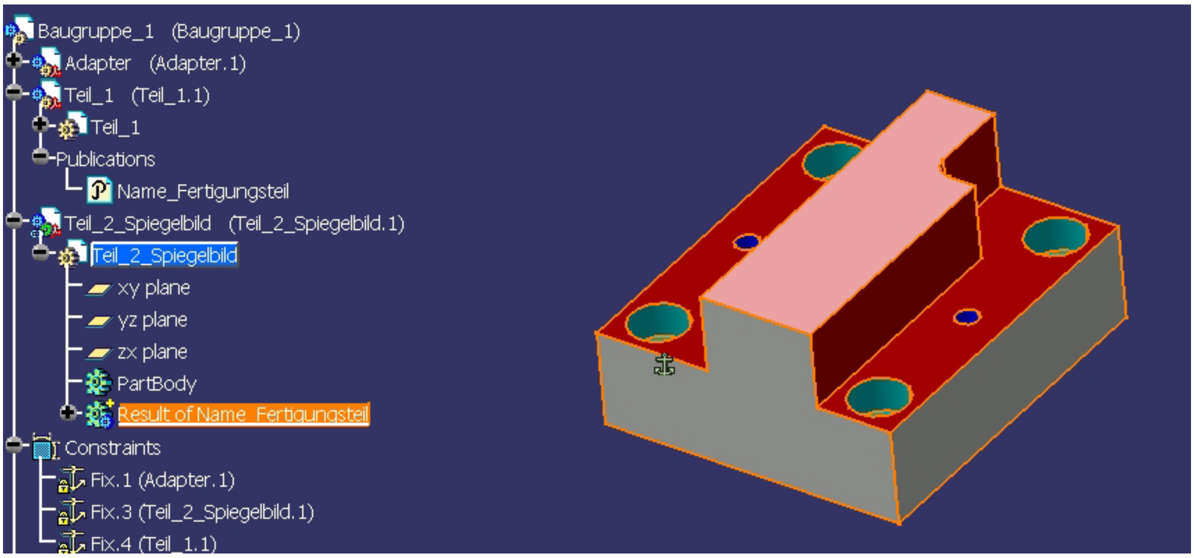


Figure 19 – Result in mirrored part after pasting


Figure 20 - Button for mirroring in the part

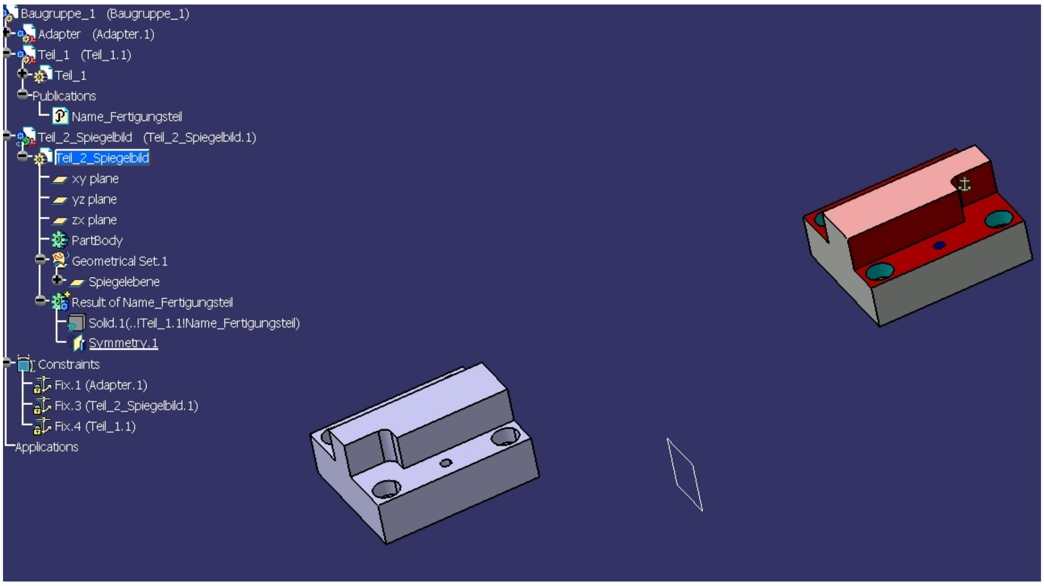


Figure 21 – Result after mirroring

4 Structuring CATIA-V5 production parts

4.1 Fundamental information

Unless described otherwise in the OEM-specific directives, the tree of the Parts generated from the start model looks like illustrated in **Figure 22** .

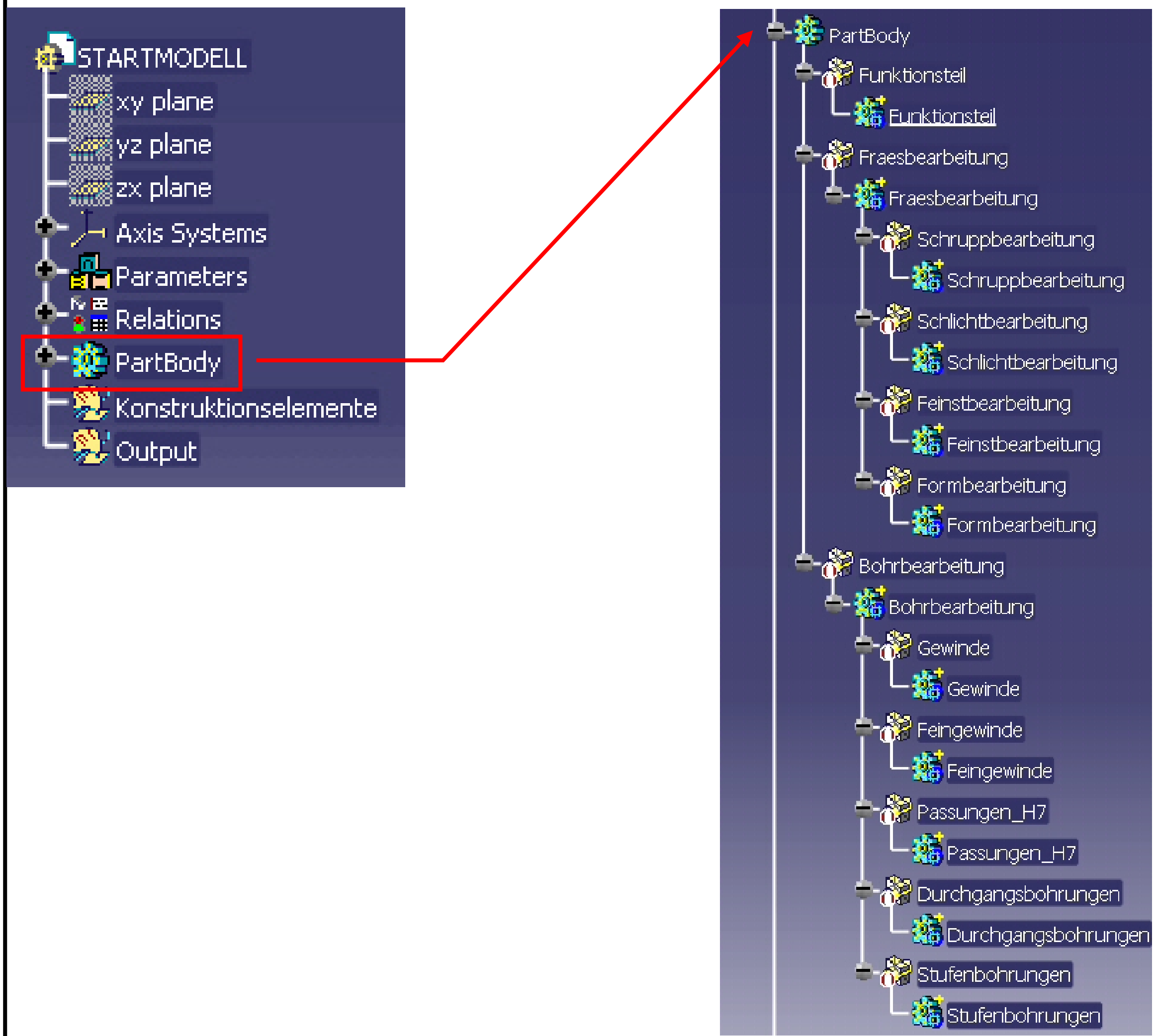


Figure 22 - Tree from the start model

The „Assemble Operations“ in the tree of the start model are deactivated/disabled because if the „Assemble Operations“ have no volume, this would lead to an incomplete result.

4.2 Body structure

When talking about the body structure, a differentiation has to be made between production parts which are based on a flame-cutting template as unprocessed blank part and such parts being produced/manufactured out of semi-finished parts.

If the unprocessed blank part is a flame-cutting template, hence, it has to be represented in the body function part (=Funktionsteil) (see **Figure 23**).

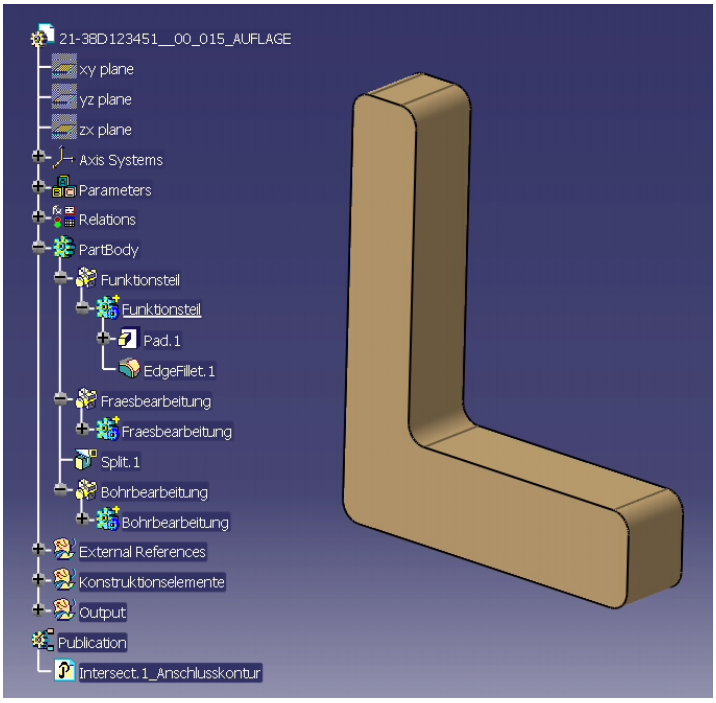


Figure 23 - Body function part contains unprocessed blank part (here: flame-cutting template)

Subsequently, the machining steps and drilling processes are added to the corresponding bodies (smoothing, rough machining, threads, fits, etc.) (please refer also to **Figure 24**).

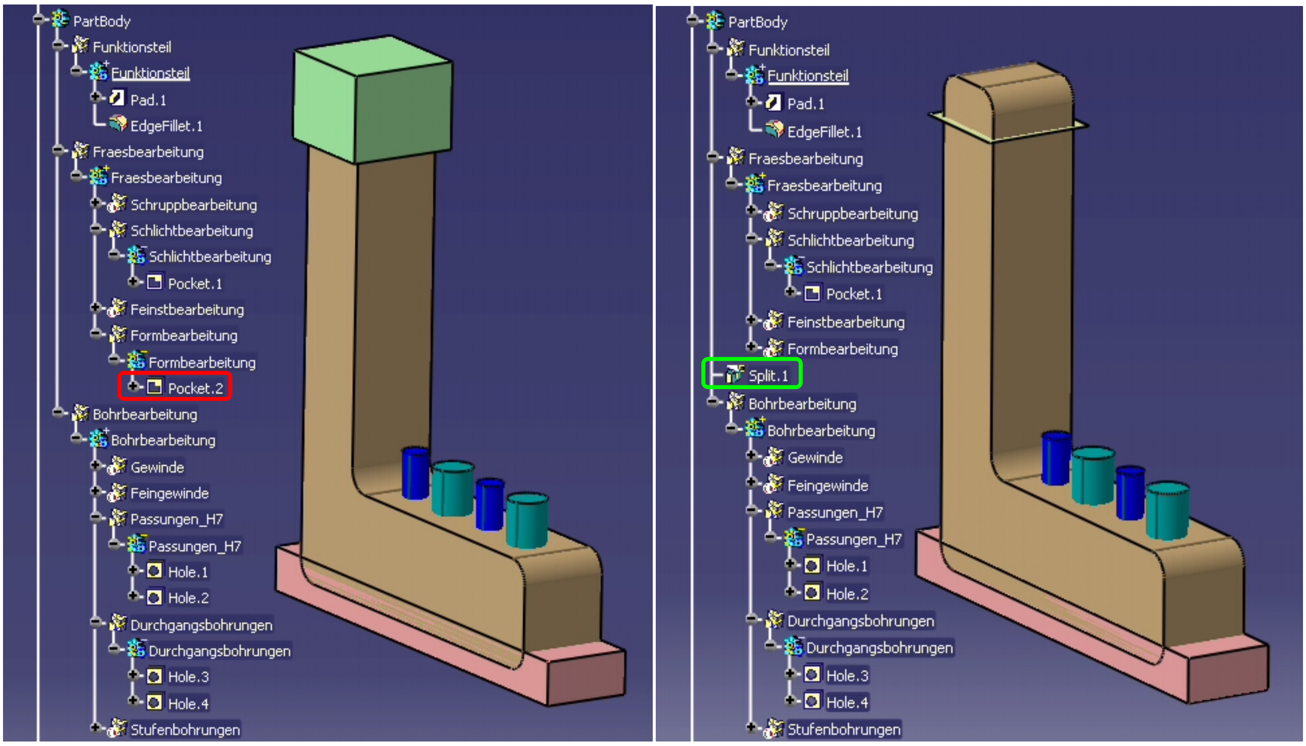


Figure 24 – Machining by milling and drilling

When intersecting a volume with a surface, it is possible to proceed in two ways. On the one hand, intersecting can be carried out by means of a pocket (see **Figure 24**, red marking). On the other hand, it is possible to generate the intersection by means of using the split function (see **Figure 24**, green marking) while the latter should be preferred. This operation, however, cannot be carried out in the corresponding body (in this example: shape processing). It has to be employed directly in the PartBody. Here, however, the respective element (here: Split.1) then has to be colored correspondingly.

For parts where no flame-cutting template is needed, there may be an already treated body in the body function part where the treated surfaces only have to be colored correspondingly (see **Figure 25**). To ensure a more stable, reliable design – however – the procedure using „subtraction"-bodies is to be preferred, since color coding of the treated surfaces remains unaffected.

Machining through drilling, however, has to be added for each part to be produced under the respective body.

If required, new bodies can be added to this structure by means of the „Assemble" operation.

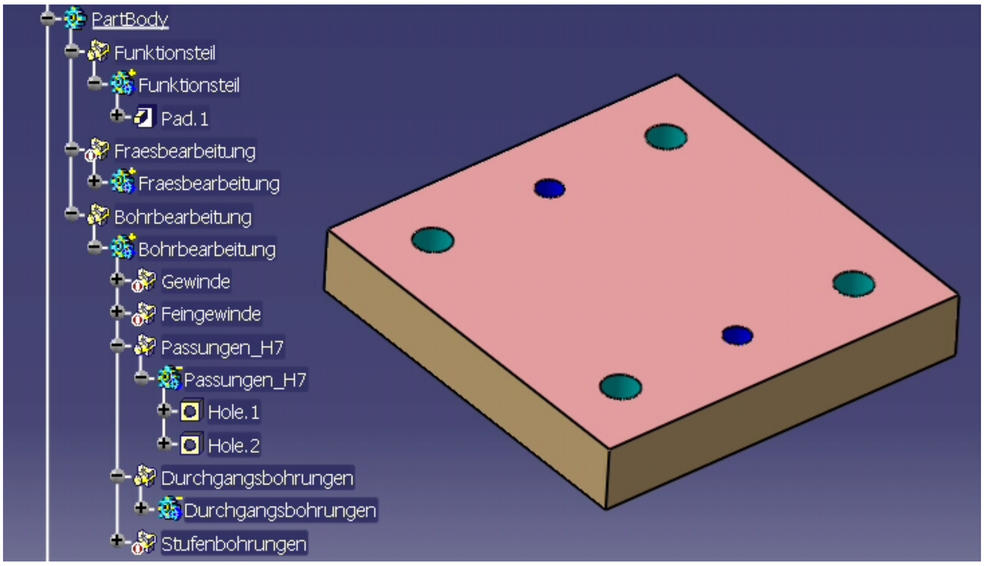


Figure 25 - Finished body in the body function part

In case of more than two drillings of similar type, working with the „Pattern" function is possible according to the corresponding OEM-specific guideline. In any case, the pattern must be based on one geometrical feature only (e.g. hole).

4.3 Output elements

As output elements, preferably explicit geometry elements should be used. BRep-elements must not be used, because they do not guarantee a stable design.
Output elements can be also surface connecting contours (e.g. sketcher outlines/contours and drilling templates) and can be very useful for transfer to neighboring components. Thanks to the parametric dependence, a consistent transfer process to the corresponding element is guaranteed with modifications.
The surface connecting contour is generated by means of the *„Boundary"* function. In a final step, the surface connecting contour has to be published (see also **Figure 26**).
Use of the surface connecting contour:

- By means of projection, lines can be used for sketcher dimensioning.
- Constraints for positioning the drilling (e.g. concentricity)
- Contour and position of drilling for tool plate or clamping frame
- etc.

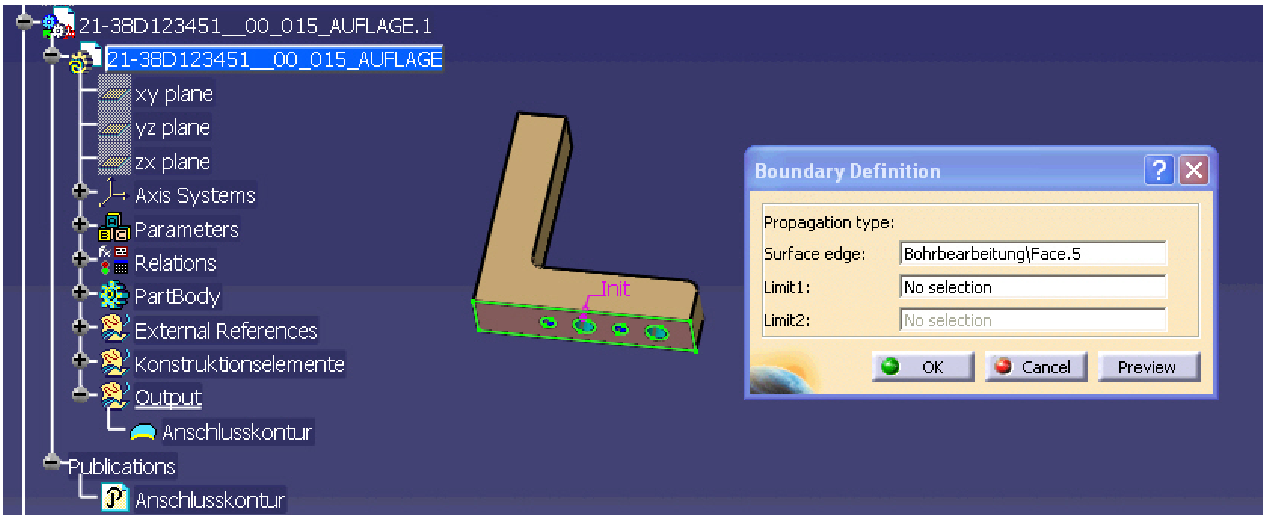


Figure 26 – Generating a surface connecting contour

5 3D-Design

5.1 Axes and position in space

Designing all component-contacting parts has to be accomplished in vehicle position.

Locating- or centering pins are not considered as component-contacting parts (in sense of designing in vehicle position), because they rather represent standard parts designed in production position. They are positioned in the CATProduct (in the assembly) by means of suitable constraints (offset, coincidence, etc.).
Tool plates are handled in the same way as component-contacting parts, i.e. their construction/design has to be accomplished in vehicle position.
The position of the construction in space (reference to the vehicle axis) is described in the OEM specific additional guidelines.

5.2 Sketches

5.2.1 Sketches in general

Since the sketcher is to a 100% integrated into V5 and used for almost all features, here as well, various rules have to be observed to simplify later modifications.
In the sketcher, all elements have to be given dimensions (in default view, the geometric elements of the sketcher are completely shown in green). This way, it is also ensured that the modification of a geometry-controlling adapter does not lead to an unintentional change of the sketch. The position of the geometric elements of the first sketch can be determined by fixing some of the elements. All further elements in sketches that were subsequently created are then dimensioned with reference to higher-level features.
When CATParts are driven associatively by external parameters, it is not permitted to reference dimensions to „H“ or „V“.
Dimensions or constraints of a sketch should only refer to planes/surface connecting contours/axes of already existing features, because with modifications of the CATParts the dimension or constraint will be lost when choosing the edge of a body as dimensioning reference (especially fillet, rounded corner). Consequently, fillets in the sketch shall be avoided by all means. We recommend you to generate the fillets/rounded corners of body edges at the pad (depending on the complexity of the profile), since changing the support geometry of the sketcher may lead to changes in direction.
Wherever it seems useful, formulas are to be employed. The decision largely depends on whether this is going to simplify later modifications and perhaps justifies the expenditure in time which is slightly higher when creating the sketch.

5.2.2 Positioned Sketch

It is often useful/helpful to determine the position of the origin and the direction of the h/v axes. This way, the sketch is defined even more precisely than a normal sketch. Each time you wish to dimension with reference to the origin of the sketch, a “positioned sketch” has to be used.
With a normal sketch, one chooses solely the plane on which the sketch should be located. Here, however, the position of the origin of the sketch as well as the direction of the h/v-axes cannot be defined, but is determined by Catia itself.
This is of special importance for the definition of “Powercopy”-components which are usually implemented in always changing positions. By means of positioning the sketch, such flexibly usable components are defined uniquely, thus being more stable.
But also for other variable components which are either subject to frequent changes or where the sketch depends on 3D-elements, using a “positioned sketch” is often advantageous.

All sketches should be basically defined as positioned sketch to allow a trouble-free refreshing/update of the construction in case of changes.

Icon for „Positioned Sketch“

5.3 Representing open positions

Movable parts are represented in its respective open position transparently. For this reason, transparency has to be adjusted on instance level to make sure that the CATPart itself remains unchanged.
Devices and systems must be designed in working position (closed position).
There are several possibilities of generating the open position:

Variant 1:
In assemblies where an open position needs to be represented, a corresponding part must be generated from the start model where the open position is represented.
All bodies that need to be represented in open position are published in the respective parts and copied to the part for the open position (see also **Figure 27**, example: red arrow identifies this for the pressure piece). Here, the open position now can be generated by means of the transformation commands.

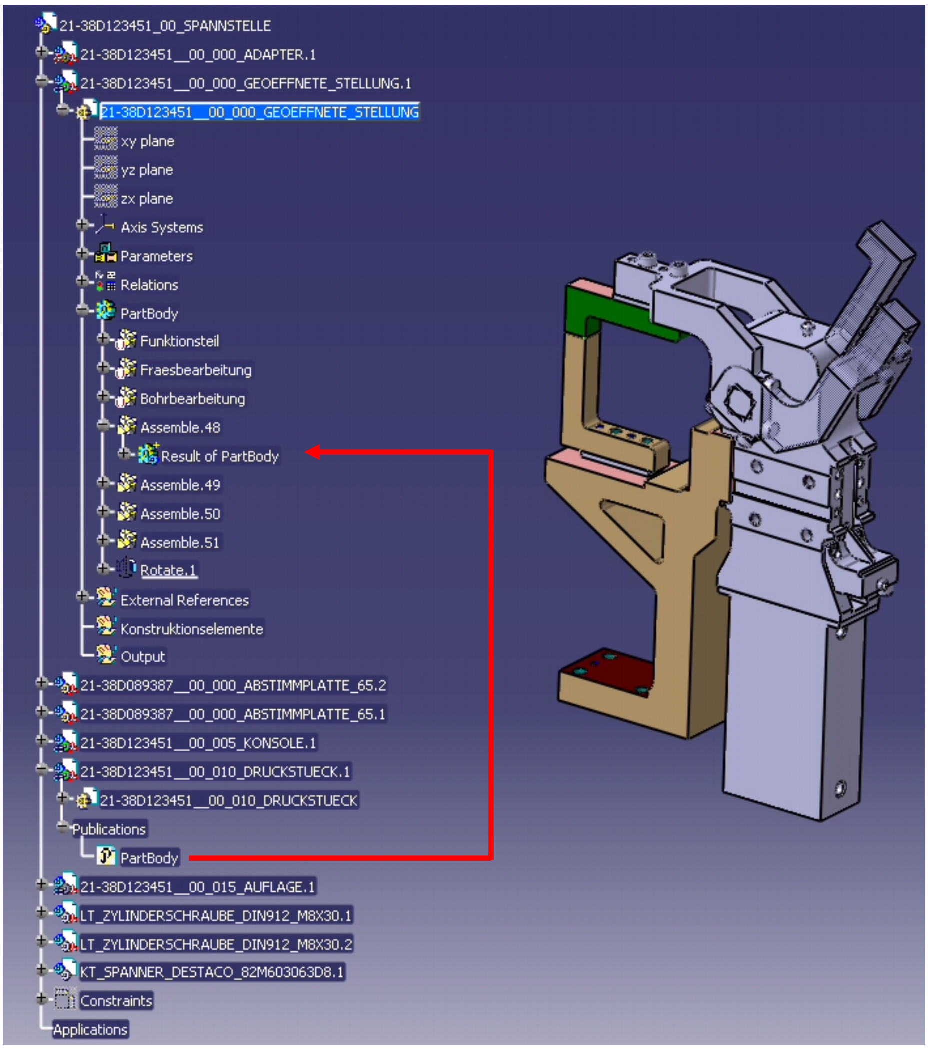


Figure 27 – Open position (Variant 1)

Variant 2:
For representing an open position, a second or higher instance of each movable part - positioned correspondingly via constraints – is generated using the option "not in BOM" (Properties/Visualize in the Bill Of Material) (see **Figure 28**).

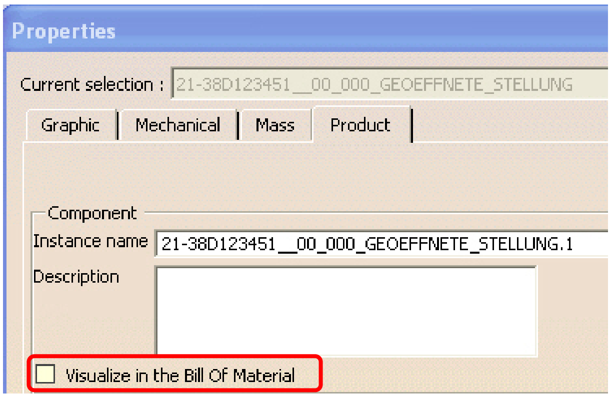


Figure 28 - Visualize in the Bill Of Material

To reduce the constraints to be generated, the instances of the movable parts can be added/pasted into a separate component. Parts concerned are to be pasted as existing components and to be positioned by means of constraints.

In case the open position is only composed of a few parts, variant 1 is to be preferred. For a larger amount of parts (e.g. travel units, pivotable units, carriages, complete devices) variant 2 is to be selected due to the smaller amount of data.

5.4 Accuracies

Here, the specifications made by each OEM have to be considered.

5.5 Layers

In CATIA V5-models the use of layers/filters is not permitted. Thanks to the hierarchical context of the model structure, the single constituents can be separated clearly.

5.6 Design tables

Design tables serve the purpose to control versions or parameters and basically need to be adopted in the CATIA model using the option *„Duplicate data in CATIA Model".*

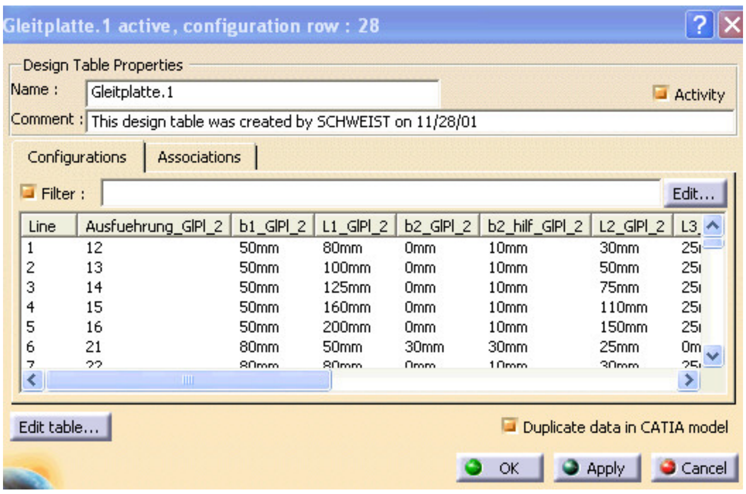


Figure 29 - Definition of a design table

Hence, as shown in **Figure 29,** the following settings have to be made:
Activation of option *„Duplicate data in CATIA model"* is obligatory.

5.7 Useful remarks for 3D-construction using CATIA V5

The following remarks are meant to facilitate working with CATIA V5 and shall standardize its use:

- For the product structure, the structural architecture of the devices and systems has to be taken as orientation basis.
- If external geometries are used, referencing should not be made to Faces/corners/edges (BRep-(Boundary)-Access) of these elements, but to explicitly generated elements.
  (Explanation: BRep-Elements may be omitted in the further progress and thus the model could become instable).
- The philosophy of a position in the BOM = apply CATPart/CATProduct; in case of multiple use within an assembly, instancing can be used.
- Ensure that dependence between components/assemblies has always the same direction (otherwise risk of Update-Cycle).
- The position of „In context“ generated parts is always determined by their context. For this reason, no assembly constraints should be generated for the part (except for: Fix), since this in most cases leads to loops. Hence the following applies: a part is either generated „in context“ or implemented into the Product by means of assembly constraints.
- Only use „Publish“ – elements (published = published geometry or parameters) for expressing dependencies.
- In bodies – to which further bodies should be added via Boolean operations – the bodies should be designed such to be overlapping. This way, error messages and long update times can be avoided.
- Looking after the specification tree should be done in parallel to geometry creation. Bodies, sketches, features, axis systems and other operations are to be designated uniquely or to be sorted logically and grouped.
- If you wish to compare to version of a CATParts, CATProducts or CATDrawings, hence, one state/version has to be generated using the option „New From“.
- The use of assembly features is not permitted, because this way the information flow towards production cannot be guaranteed.

6 Modifying this guideline

6.1 Modification index

Init. version (modif. No. 0) of 09.09.2005

- First draft of the basic guideline by the working group

1st modif. (modif. No. 1) of 18.09.2006

- Restrictive supplements for mirroring assemblies using macros

2nd modif. (modif. No. 2) of 29.02.2008.

- Preface, update of the contact persons
- 1.1 … update of the respective OEM supplier-website links
- 2.1 … Figure 1 updated according to V5 Release 16
- 2.1 … Figure 3 (obvious designation of the wrong method added)
- 2.2 … Table 1 (axis system supplemented in listing)
- 3.2 … The following elements have to be created in this adapter: Copy of the component geometry *(instead of component geometry)*
- 3.3 … discard of Figure 9 - Basic structure with corresponding body structure also valid for adapter, replaced by a pointer to Figure 22 (numbering of all following Figures shiftet by one)
- 3.5 … Integration of standard- and repetition parts is OEM-specific regulated in future
- 3.7.1 … pointer to new version of the mirroring macro, Figure 15 supplemented
- 5.2.1 … detailed definition of the restrictions concerning the interdiction of H-and V-References in sketches
- 5.2.2 … formal restructuring of the text
- 5.3 … extension of the method for any intermediate positions
- 5.6 … usage of "duplicate data in CATIA model" is a must
- 6.1 … modification index added (all following chapters shifted by one)
- 6.2 … New description of modification process
- 6.3 … Modification form changed

6.2 How to proceed with modifications

Modification requirements are to be written down in the following form and should be sent to the contact persons named in the preface. Then these requests/requirements are brought into agreement by the „Working group for system- and device design using CATIA V5 of the German automotive industry". In case of approval, the modifications are implemented by the person in charge of the documentation.

6.3 Modification form

See annex

Dated: 20.02.2008	**Proposal for Basic Guideline Modification**

Written by:	Company:
Date:	Phone:
E-Mail:	

State:

Modification proposed:

Remarks/comments: