Work Instruction

015 Research + Development **CATIA V5 - Assembly Design** **W_015_011**

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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 / directives:

- 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
RPS	Reference Point System
CATDUA	Additional application for cleaning 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 products with CATIA V5

5.1.1 General information

This chapter specifies the different techniques and methods for constructing and controlling products.

There are essentially 2 different methods for constructing products:

5.1.1.1 Constraint method

The most common technique, and the one generally used at GRAMMER AG, for defining products in CATIA V5. The CATParts are modelled either in association with the product (design in context) or independently of any construction space definition (individually) in the work environments for part generation.

Single parts and/or subassemblies with no context link (external references) between them are positioned in a superordinate product in the AssemblyDesign module using *assembly constraints*.

In the case of context-oriented parts and/or subassemblies, no additional assembly constraints should be used. Only the Fix function may be applied to define the spatial position as well as the Fix Together function.

5.1.1.2 Adapter method

Adapters are controlling CATParts within products. Via these CATParts, geometry and position references are allocated for control within the root product and the subassemblies.

Newly defined single parts are created exclusively in the context of a product (CATProduct) to which the part (CATPart) belongs. The part is designed dependently on the basis of the prescribed data (some of which may already be present in the product). The adapter functions as a central "*hub*" for the management of design-relevant geometric references. It passes the geometry and position references for the detailed design on to parts that are newly created, for example. The new part geometry generated is controlled via dependencies on the adapter in position and geometry.

The *adapter* for a design can, in terms of its definition, be a self-created CATPart or also a data record supplied by a customer.

Note:

Which method should preferably be used depends on the changes expected as well as on the further process steps. If a product is not very complex and is subject to frequent change operations, the adapter method may prove beneficial.

If a product will require kinematic processing and analysis at a later point in time, the *constraint method* is preferable.

More often than not, however, a combination of the two methods is applied.

5.1.2 External references in products

All external references or links (which are used in one of the two aforementioned methods) which refer to other CATIA parts and products are to be generated on the basis of *Published Elements*. Geometries and parameters that should be used in other parts or products must be published in the corresponding (generator) CATPart. It must be guaranteed that a publication and the underlying geometric element have the same name. This is possible using the Publication function if the designation is adapted when publishing elements. If the publication name of a parameter or geometric element is amended subsequently, the underlying element is also renamed.

SeatingSystems	***Automotive***
Import links created in the context of products are not desirable as this impacts on parts that have been incorporated in other products, for example. Exception: ALLCATPart!	If projects are edited in a folder, links between CATParts are prohibited. There should be no dependencies between parts. This means that when making changes to a CATPart another part should not be amended automatically. Exception: mirrored parts & ALLCATPart!

5.1.2.1 Link management

To achieve maximum data transparency, no links of the type CCP ("*CutCopyPaste*" link) should be generated across parts, as a general rule. Instead, contextual links should be used (type: *import/context link*), i.e. designing is done in the context of the product (*design in context* method). Only published geometries and parameters should be linked. If an adapter part is used, all geometries and parameters for further use must be published in the adapter part in order to guarantee that external references can be generated and exchanged without any errors in the other departments, customer situations and the downstream process chain.

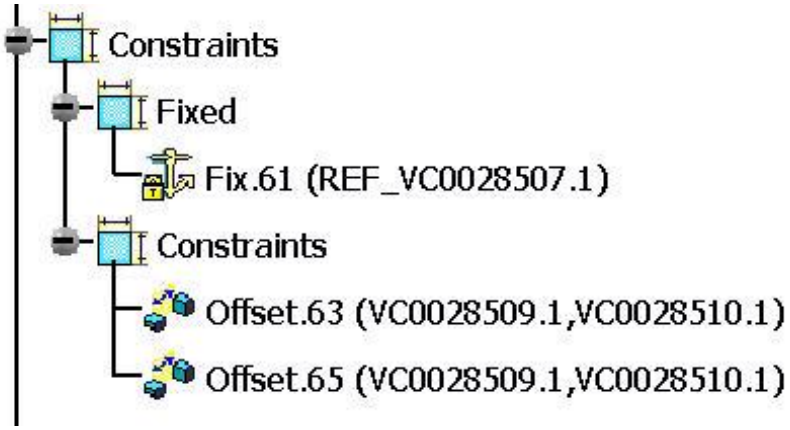
5.1.3 Assembly constraints

If the *constraint method* referred to under 5.1.1 is used to construct products, the following must be borne in mind:

When allocating *assembly constraints* it is essential to make sure of a consistent definition. The constraints between the elements must be defined unambiguously. It may be necessary, in some OEM environments, to generate assembly constraints exclusively with *Publications*.

5.1.3.1 Constraint sets

In the interests of achieving greater clarity it is necessary from a *"larger"* assembly to define additional *Set* structures for the structuring/organisation of constraints in the *Constraints* structure node. The *Sets* generated must be named after the connections contained (see the following figure).



The Set function is applied from the right-mouse button shortcut menu on the corresponding object/constraint.

The aim with the allocation of *assembly constraints* is to reduce the degree of freedom of the components (part/product) to degree=1 or degree=0. Whether 0 or 1 may be chosen for the degree of freedom depends on whether the product is to be animated at a later point in time via the *Manipulation* function with the option *With respect to constraints*.

The following constraints may be assigned among the single parts and/or products whose structure is not context-oriented:

- Fix component: Fixing a component prevents it from being moved away from its parents or current position during updating. The fixing should be applied as early as possible when constructing products as doing so may also prevent loss of the product origin. There are two options for fixing components:
  - by fixing their position in relation to the geometric origin of the product, i.e. by defining an absolute position. This operation is known as “Fix in space”.
  - by fixing their position in relation to other components, i.e. by defining a relative position. This operation is known as “Fix”.
- Coincidence constraints: Coincidence constraints are designed for the orientation of elements of different geometry types. Using coincidence constraints, these elements can be arranged so that they are coincident with each another.

The following table shows the elements that can be selected for a coincidence constraint.

	Point	Line	Plane	Sphere (centre)	Cylinder (axis)	Cone	Axis system	Curve	Tab. cylinder	Sur-face
Point						NA	NA			
Line						NA	NA	NA	NA	NA
Plane						NA	NA	NA	NA	NA
Sphere (centre)					NA	NA	NA	NA	NA	NA
Cylinder (axis)				NA		NA	NA	NA	NA	NA
Cone	NA	NA	NA	NA	NA		NA	NA	NA	NA
Axis system	NA	NA	NA	NA	NA	NA		NA	NA	NA
Curve		NA	NA	NA	NA	NA	NA	NA	NA	NA
Tab. cylinder		NA	NA	NA	NA	NA	NA	NA	NA	NA
Surface		NA	NA	NA	NA	NA	NA	NA	NA	NA

- NA: Not applicable

Note: As the table shows, this type of constraint can be used with almost all geometric references. However, it is also always important to note the *"real situation"* as well as the process orientation.

In other words: if, for example, a screw is positioned, the screw axis to hole axis is a coincidence constraint; where the screw contact surface sits, a surface contact must be defined as a constraint.

- Contact constraints: Constraints of the contact type can be generated between two controlled surfaces.
  - Controlled means that an inside and an outside can be defined on the basis of a geometric element. This definition excludes surface elements and wire-frame surfaces as these are not controlled.
  - Every surface has a surface normal. This is arranged contrarily in the orientation with the contact constraint type.
  - The common area between the two planar surfaces can be a plane (plane contact), a line (line contact) or a point (point contact).

The following table shows the elements that can be selected for a contact constraint.

	Planar surface	Sphere	Cylinder	Cone	Circle
Planar surface				NA	NA
Sphere		(1)	NA		
Cylinder		NA	(2)	NA	NA
Cone	NA		NA		
Circle	NA		NA		NA

- NA: Not applicable

- (1) A contact between spheres is possible if their radii are identical. The resulting contact constraint is equivalent to a coincidence constraint. The spheres appear merged.

- (2) A contact between cylinders is possible if their radii are identical. The resulting contact constraint is equivalent to a coincidence constraint. The cylinders appear merged.

- <u>Offset constraints:</u> When defining an offset constraint between planar elements, the orientation of the surfaces must be indicated.
    - The offset value is always displayed next to the offset constraint.
    - The unit used is indicated/controlled in *Tools* -> *Options*. It can be changed if necessary.

The following table shows the elements that can be selected to define an offset constraint.

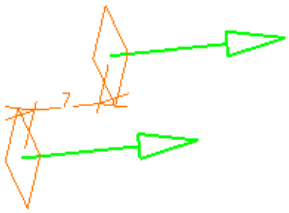
	Point	Line	Plane	Planar surface
Point				NA
Line				NA
Plane				
Planar surface	NA	NA		

- NA: Not applicable

Positive and negative offset definitions

Positive or negative offset values can be used when executing an offset constraint. The following must be borne in mind:

- At least one of the components to be constrained must be a planar element. Otherwise, it is not possible to set positive or negative offset values.
- The normal vector (vertical) to the planar element indicates the positive offset value.
- If the planar element is an oriented plane (e.g. a planar surface of a pad), the normal vector pointing to the side opposite the material indicates the positive value.
- If the planar element is a wire-frame plane, the application automatically deduces the positive or negative value. Green arrows show the positive value.



- <u>Angle constraints:</u> Angle constraints are divided into three categories. When defining an angle constraint between planar elements, the orientation of the surfaces must be indicated. The value is always displayed next to the angle constraint:
    - Angle
    - Parallelism (if the angle value equals zero): When setting the parallelism constraint green arrows appear on the selected surfaces to indicate the orientation.
    - Perpendicularity (angle value equals 90 degrees)

The following table shows the elements that can be selected for an angle constraint.

	Line	Plane	Planar surface	Cylinder (axis)	Cone (axis)
Line					
Plane					
Planar surface					
Cylinder (axis)					
Cone (axis)					

- <u>Grouping components (Fix Together):</u> This “constraint” defines the bond between components. It may make sense to apply this function for parts/products that are separably or inseparably linked (e.g. through screws/welding), for example. It can also be used to adapt parts/products to other components so no explicit constraints have to be assigned.
    - Advantage of this function: Parts/products have no explicit constraint and do not need one. If a grouped component is moved (e.g. with the compass), the system triggers an alert that a connection exists. This may be useful if components have been designed without external referencing to the product origin.

5.1.4 Construction & structure of CATIA V5 products

The structure of a product essentially comprises components of the format CATProduct and CATPart. The construction and sequence of the components in the products should be based on production-oriented criteria. If, when the product was created, it was not organised according to production-oriented criteria, the structure must be reorganised retroactively in the *AssemblyDesign* module using the *Graph tree reordering* function.

It is possible, in exceptional cases, to integrate CATIA V4 data (model and session files) or neutral formats (STEP, IGES, etc.) into the structure of a product temporarily in order to migrate existing geometry "*on the fly*" or to carry out construction space analyses. These documents must be removed from the structure before the product is saved.

If the adapter method is chosen for the construction of a product, the main adapter defines the dependency between the vehicle geometry and the products of the design.

The main adapter must be fixed immediately after installation.

5.1.4.1 General information

The structure tree of any CATProduct that has been generated using the "*New Documents*" function within GRAMMER AG appears as shown below.

It is prohibited to delete or rename objects in the prescribed basic structure!!!

Other structure nodes, parts and subassemblies must be added.

When adding objects to the structure make sure they are given a meaningful name.

If several positions are incorporated in a product multiple times as one unit, it is possible to incorporate these positions in a *component*. A *component* is identified as `COMP_´ upon its generation using the script-based "*New Documents*" function in CATIA V5 when creating new documents. A logical name affix can be added on to this name (e.g. standard parts) depending on what this structure node contains.

5.1.4.2 Materials

Products (e.g. welding assemblies) that are coated (painted/varnished) in a subsequent process step following their assembly receive a material assignment. For this purpose, a new “varnished” product is generated, with the welding assembly as its component. The coat/varnish is assigned to the *root structure* of the varnished product. The corresponding definitions for this are stored within GRAMMER AG in the CATIA V5 material library under the material group *Varnish & Powder*. It is prohibited to give the material assignment to the instance of the welding assembly.

<u>Tip on product structuring:</u>

Defining a good product requires careful consideration – so give some thought to your approach and your objective!

5.2 Views

In principle, “flexible” purchased parts (cables, springs, etc.) are managed in different views within products at GRAMMER AG. Both the installed state and the purchased state are mapped in CATIA V5. This special management technique produces the following rules:

5.2.1 Flexible parts - installed state

In this state, the components (parts/products) are presented in such a way that the representation shows the exact geometric position in the context of the product. This serves as a geometric safeguard (DMU & construction space analysis). Note the following:

- Standard or repetition parts that are available in different versions (e.g. varying lengths) are managed and controlled within GRAMMER AG using CATIA V5 product structures. With this approach, the variants of the part are managed within a product. If a new variant is generated, this must be included in the product.
- The designation of a new variant must be declared unambiguously in the name of the component. Example: *partnumber_geom.variance* (08154711_D25).
- All variants of the part that are not required are *hidden*. Make sure that it is the instance of the part and not the geometry that is hidden.
- The variants of a component should only be hidden from the active root product (superordinate main product).
- If standard or repetition parts (e.g. springs, cable ties, etc.) are used, the representation can be adopted from another project, provided that this is a 100 % match. Otherwise, the part must be re-designed in the exact position representation.

5.2.2 Flexible parts - purchased state

In this representation, the standard or repetition parts are captured in their purchased state. This means, for example, that a cable tie or cable (harness) is designed straight. In addition, springs, for example, are presented in their relaxed state. Note the following:

- If a part is designed as a “new” purchased part, it must be defined in a product structure in its original delivered state.
- The affix REF must be added to the name of the purchase variant. It is placed after the part number. Example: *partnumber_REF* (08154711_REF).
- The saved state of the product in which the purchased state and the variants have been generated must be defined in such a way that the purchase variant in the product structure is all that is visible.

5.2.3 Presentation modes - installed & purchased state

The variants that are not required as well as the purchased state of the component must be hidden using Hide/Show. The hiding/showing of variants may only be carried out from the main product.

Automotive-specific products only contain the CATPart which represents the installation situation. The reasons for this are as follows:

- to reduce loading times
- to prevent the delivery of “irrelevant” data from other projects to customers

Variants that are not required must not be removed (visually) from the context of the product using the *unload component* or *deactivate node* functions.

5.3 Creating an ALLCATPart from a product

Within GRAMMER AG, “*linked*” ALLCATParts are used to derive assembly/offer drawings from complex products. In addition, some OEMs from the automotive industry require these files for archiving design data in their system.

These ALLCATParts have the advantage that, if they are defined in the context of a product with corresponding dependency, any changes to geometries in the product are transferred directly to the ALLCATPart. Another advantage is the fact that the data volume of the files is usually small and therefore easily manageable.

5.3.1 Definition of an associative ALLCATPart

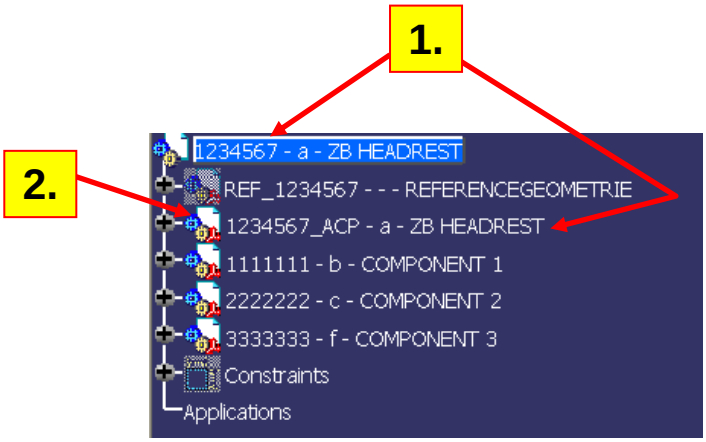
The starting point for generating an associative ALLCATPart is always a product environment in which this "*target document*", which will receive the geometry from the other single part areas, is entered.

Only the basic model specific to GRAMMER AG or to the OEM may be used to generate a "*linked*" ALLCATPart.

The following procedure and restrictions apply to the generation of the associative ALLCATPart in Automotive:

1. The part number of the product is identical to the part number of the ALLCATPart. The additional designation and analogies are shown in the following illustration.

2. The designation of the reference geometry part also contains the part number as shown.

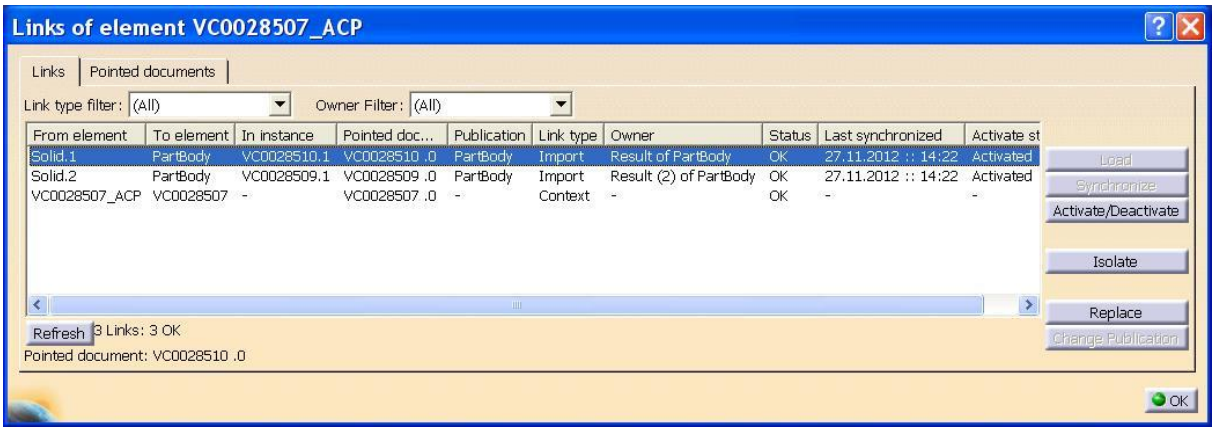


3. The required geometric bodies are inserted into the ALLCATPart using Copy & Paste (paste spec. with the option "*with link*"). A context link with reference to the product is created as a result. This defines the geometry and position of the part.

5.3.2 Definition of an ALLCATPart without product reference

There are two options available for this version of generating an ALLCATPart:

1. Isolate the links of an existing associative ALLCATPart. Use the *Edit -> Links* function to open the following window. Here you can select and isolate every link.



2. Generate an ALLCATPart from the AssemblyDesign environment using the given function. However, in contrast to the procedure described above, all the contents of the product are copied into a new standard part.

This function must **not** be used in *Automotive*.

5.4 Geometry & structure check

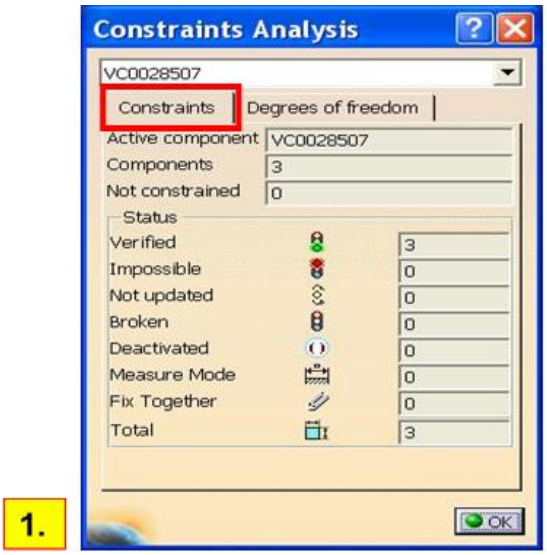
To ensure that CAD data generation is of a uniformly high standard of quality, it is absolutely vital that the design data is checked constantly. This involves subjecting the data to a continuous geometry and structure check and making sure that the CAD data for the respective release version is up-to-date.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

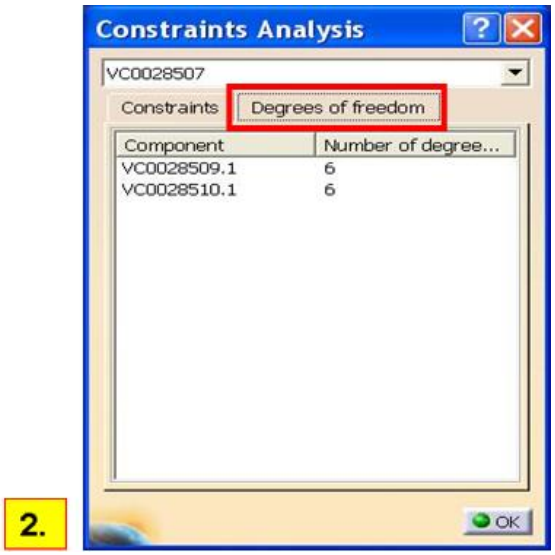
The procedures and application scenarios for the above tools are described under ***5.2 Data checking & identification*** in GRAMMER AG's ***CAD Data Generation and Exchange Work Instruction***. This regulation must be applied likewise to products and their components.

5.4.1 Reviewing constraints & degrees of freedom

Within the CATIA V5 *AssemblyDesign* module the user has access to the ***Constraints*** and ***Degrees of freedom*** functions under the *Analysis* menu item. Users/designers must use these functions to carry out the corresponding checks.

The application scenarios and functionality of the ***constraints analysis*** are as follows:

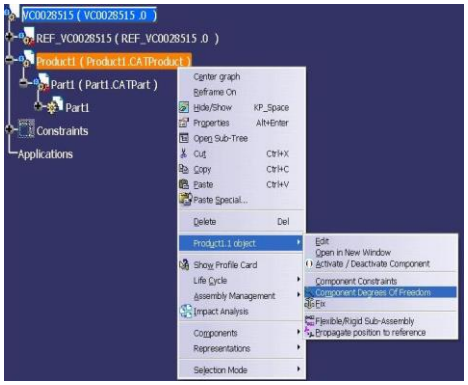


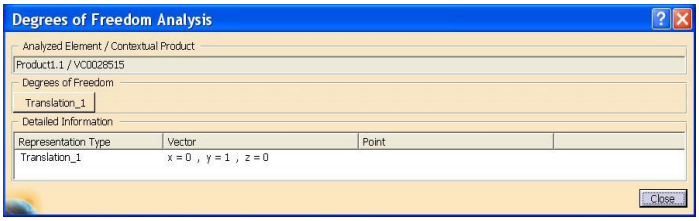


1. Check the status of all the defined constraints in the product and make sure they are up-to-date. The outcome of this analysis should not produce any constraints with the status *Impossible* or *Not updated*.

2. Analyse existing degrees of freedom for components of the product. The maximum permissible number of degrees of freedom per component should not exceed the value 1.

An extension to the degrees of freedom analysis is provided by the *Degree(s) of freedom…* function. This can be used to analyse and itemise any mechanical degrees of freedom that exist for components. So that this function can be executed correctly, the product instance of the respective component must be activated before the analysis function is executed.





5.4.2 Clash check

Checking for any clashes between parts and/or products in CATIA V5 is a necessary process. Every designer (internal/external) is obliged to check his own products for clashes. This measure is part of the product creation process and serves as a geometric safeguard for virtual prototypes.

There are essentially two types of clash check:

- static clash analysis -> the following tools are used for this:
  - Compute Clash...
  - Clash...
  - Sectioning…
  - Distance and Band Analysis

- dynamic clash analysis -> with the Manipulation function and with the "*Stop Manipulation on Clash*" option activated

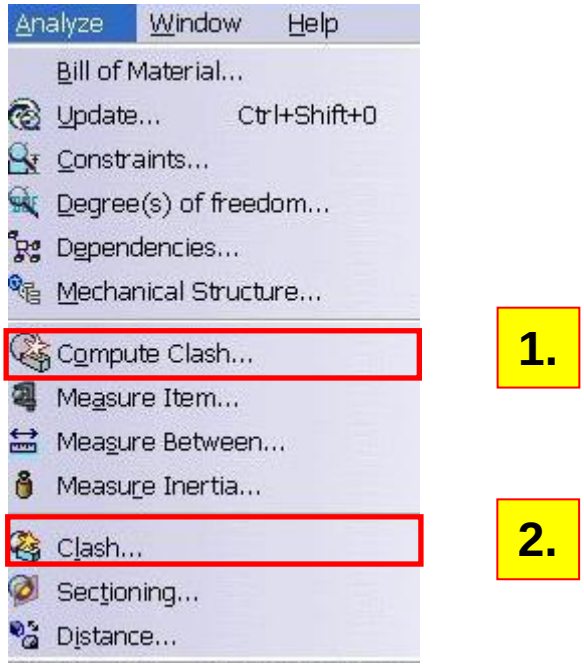
GRAMMER AG has a documentation obligation towards its clientele. The clash check with its integrated documentation tool is hugely important for this, as analysed product interferences can be exported in log form.

The data check is seen as an integrated operation in an innovative product creation process and is carried out continuously during data creation. This makes it easier to detect errors early on.

For the static clash analysis, a distinction is made between two tools:

1. *Compute Clash…*: This function allows the selection of up to two components which can be analysed against each other.

2. *Clash…*: Using this function, individual components as well as the contents of an entire product can be analysed in terms of clash/distance.



Static clash and distance analyses can also be performed using the tools "*Sectioning…*" and "*Distance and Band Analysis…*".

6 Applicable documents

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

7 Changes