Guideline for the design of fixtures using CATIA V5

- BMW specific supplement to the OEM-overlapping „Basic guideline for the design of devices and fixtures with CATIA V5“

Created by: Working group „fixture design“
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<tr>
<td>1.0</td>
<td>02/2003</td>
<td>Initial version – merging the TI- and TK-2-guideline into one document</td>
</tr>
<tr>
<td>1.1</td>
<td>03/2004</td>
<td>General revision – Extension of the section “Construction &amp; design methods in assembly context”</td>
</tr>
<tr>
<td>1.2</td>
<td>05/2004</td>
<td>Documentation of new contents (new All.CATPart functionality, positioned sketch, new naming convention of the Body-in-White assembly dept.; position number assignment for bill of materials, assembly mirroring, flame-cutting template – new method, T-Basics LU1)</td>
</tr>
<tr>
<td>1.3</td>
<td>07/2004</td>
<td>Editorial revision, adding the introduction, basic models for “Parts” and “Products”</td>
</tr>
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<td>1.4</td>
<td>12/2004</td>
<td>General revision, new method “direct link flow“ (section 3.3), T-Basics LU2, integration ASP-creation (section 6), new editorial layout</td>
</tr>
<tr>
<td>2.0</td>
<td>03/2006</td>
<td>Fundamental revision by merging the document with the OEM-overlapping „Basic guideline for the construction &amp; design of plants and devices using CATIA V5“ This guideline documents the BMW specific contents only which represent a supplement to the basic guideline or describe deviations from the latter; adaptation and/or synchronization of the structuring systematics Adapting the document to T-Basics scope of supply 3 (FKF package)</td>
</tr>
<tr>
<td>2.1</td>
<td>10/2006</td>
<td>Adaptation to CARISMA (successor version of T-Basics) Extensions as to the belongings of the Assembly technology department. Renaming of the document.</td>
</tr>
<tr>
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<td>10/2007</td>
<td>Revision of chapter 6 Integration of chapter “Detailing the component-contacting geometry in the Body-in-White technology” to chapter 6 (formerly chapter 7.1.7)</td>
</tr>
<tr>
<td>2.3</td>
<td>06/2008</td>
<td>Modifications in consequence of synchronization with version AE02 of the OEM-overlapping Basic guideline, admittance of the hyphen (2.2), extension of chapter 3.5, general updates (contact persons, download paths, …)</td>
</tr>
<tr>
<td>Version</td>
<td>Date</td>
<td>Comments</td>
</tr>
<tr>
<td>---------</td>
<td>----------</td>
<td>----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>2.4</td>
<td>02/2009</td>
<td>Hint for possibility of using “static supplier title blocks” added (2.6.2), new method description for NuW parts not originating from BMW libraries (3.5.2), advice for erasing the working-products (3.6), supplementation for design of sandwich plates (shims) (7.1.5), drawing detail for frequently used standard parts (screws and parallel pins) (7.1.5), description of the new method for flame cutting templates (7.2)</td>
</tr>
<tr>
<td>2.5</td>
<td>10/2009</td>
<td>Actualization contact persons; 2.6.2 hint layer; 2.8 Actualization structure categories CAD PDMM; 3.1 Pictures CAD PDM actualized</td>
</tr>
<tr>
<td>2.6</td>
<td>02/2010</td>
<td>Section 1.3: update contact persons; Section 2.6.2: manually created title blocks; Section 2.8: update CARISMA working method for CAD PDM; Section 3.5.3: re-use of parts and modules in Assembly technology; Section 8.1: BOM in Assembly technology; Section 8.3: identification label</td>
</tr>
</tbody>
</table>
1 Generalities

1.1 Application area and other valid documents

This guideline must be used in conjunction with the current, valid “OEM Basic Guideline for the Design of Devices and Fixtures with CATIA V5”. It serves as a supplement to the aforementioned guide, adding BMW specific requirements. The specifications found in both documents are binding for designs and constructions used in the Body-in-White technology and the Assembly technology departments.

When designing standard, purchase and repetition parts, special rules apply as described in an own guideline.

All guidelines and supplemental documents are available in pdf-format on the Internet. To view, please logon at: https://b2b.bmw.com/ >> Departments >> Technologies >> Body-in-White >> Fixture-Design >> List of documents in fixture design
https://b2b.bmw.com/ >> Departments >> Technologies >> Assembly >> Production tooling and fixtures

(Access to the BMW Partner Portal from within the BMW Corporate Network is available via: https://b2b.bmwgroup.net/. Later in this document only the internet URL will be mentioned.)

1.2 Harmonization of methods

Under special circumstances, it is possible to deviate from this design method for individual projects based on prior consultation with the Project Leader and the responsible Technical Designer responsible for the fixture design. Every deviation requires prior, written consent from the Project Leader.

1.3 Contact persons

For any questions about the methods described in this guideline, please feel free to contact:

Plant Regensburg, Body-in-White and Assembly
Andreas Kempke, phone (0941) 770-4276
Andreas.Kempke@bmw.de

Plant Leipzig, Body-in-White and Assembly
Ronny Bartel, phone (0341) 445-32313
Ronny.Bartel@bmw.de
1.4 Qualification external engineering partners in the Body-in-White technology

As a prerequisite for an External Partner to provide services for the BMW Body-in-White and Assembly departments, the Partner must observe the basic principles described in this guideline. The Partner can prove their knowledge of these principles by either demonstrating the techniques through the collaborative implementation of a CATIA V5 project or submitting data for auditing.

Companies seeking to become an Engineering Partner with BMW will be admitted only after the respective Body-in-White and Assembly departments have tested and checked the Company’s qualifications and experience. Once verified, the new Engineering Partner will be added to the list of Qualified Suppliers within the respective technology department. Only Qualified Suppliers will be awarded projects using CATIA V5 per the BMW data quality requirements.

For any further information or queries please contact:

FIZ (R&D), Body-in-White  
Georg Potschka  
(089) 382-11336  
Georg.Potschka@bmw.de

FIZ (R&D), Assembly  
Florian Harnatt,  
(089) 382-58134  
Florian.Harnatt@bmw.de
2 Conventions and defaults

2.1 CATIA V5 Settings

In addition to section 2.1 of the basic guideline, BMW provides a document entitled “Technology specific V5 settings”. This document contains detailed settings-information on the current, valid version of CATIA V5. You will find this document on the Internet by logging on at: https://b2b.bmw.com/ >> Departments >> Technologies >> Body-in-White >> Fixture-Design >> List of documents in fixture design.

For engineering and design projects in the Assembly technology without using “control by adapters” (see also section 3.3.2) – an exception to the OEM basic guideline – the “Keep Link with selected Object” switch has to be disabled in CATIA Options.

2.2 Naming convention

File and directory names may only contain the OEM – standard characters as follows: [A - Z], [0 - 9], underscore [_] and hyphen [-]. For details on the systematic structure of file and directory names, please refer to section 2.8 of this document.

2.3 Colors

In general, the color convention for the drawing-free production process is not mandatory.

2.4 Drawing derivation

For details on the basic drawing model, please refer to section 2.7.2. General questions regarding the drawing format and execution are answered in section 7.1 of this document.

2.5 Data format

According to the OEM basic guideline, all data are to be created in native CATIA V5. Integration of V4 data in the “*.model” format into CATIA V5 structures is not permitted.

2.6 Data quality, model preparation and archival

To check data quality, BMW uses the Q-Checker software to verify the existence of a valid test seal. Since it is not possible to use a batch process to check an entire container structure, Q-Checker is not yet required for the Body-in-White and Assembly departments’ Engineering
Partners. Independent thereof verification of data quality has to fulfill the specifications made in this guideline.

Within the assembly directories, all files not used must be deleted / removed.

In the CATProduct, all reference geometry must be put to the “Hide” mode and visible data updated before final storage. Ghost links within a CATProduct must be removed (e.g. use the CATDUAV5 Application).

After verifying data quality and integrity, an “All.CATPart” must be created for each partno. or assembly (see also section 2.6.1). The Engineering Partner is responsible to ensure that each All.CATPart is up to date before delivery to BMW.

For data exchange and archiving in PRISMA, the CARISMA CAD PDM software (not CARISMA classic) has to be used (see section 2.10.2). The CARISMA working method has to be set to “Karosserie Montage FM”. For each partno. a unique CARISMA container is to be archived in PRISMA containing the following documents:

- Structure document for the container (ST)
- classified All.CATPart (5P)
- classified Drawing(s) (5D)
- Bill of materials document (SL)
- the real container contents (Parts and Products) packed with CARISMA (5Z)

Back-converted *.model files of the All.CATPart must not be archived together with the container structure anymore!

### 2.6.1 Creating an “ALL.CATPart”

To secure sequential processes, the CATIA V5 data for each partno. (i.e. per container), must be summarized into one single CATPart. This CATPart then contains the entire product geometry. The so-called “All.CATPart” is to be classified while packing it with the CARISMA software and consequently will be filed explicitly as a 5P document when importing it into PRISMA.

The “All.CATPart” should be created in CATIA using the Assembly Workbench function “Generate CATPart from Product” in the “Tools” menu. This opens a pop-up window on the screen in which the user must select the product to be converted. Be sure the option “Merge all bodies of each Part in one body” is deactivated. Each “Part” in the Assembly will become a “Body” in the resulting “All.CATPart”. Notice that the original “Part” names carry over to become the “Body” names and that each “Body” contains a single “Solid” representation of the original part.
With this function, all elements of the “Product” that are in the “Show” mode are converted into the All.CATPart. Elements that are not loaded or in the “Hide” mode, are not included in the All.CATPart.

2.6.2 Supply to the DZA (= Digitales Zeichnungsarchiv = digital drawing archive)

When using CARISMA, no substitute formats (such as *.CGM) have to be created for supply to DZA. Additionally, it is no longer necessary to insert and manually fill in the drawing title blocks.

The necessary metadata (title block contents or PRISMA attributes) are administered in CARISMA. After executing a “Sheet enumeration Dialog” in CARISMA, title blocks can be temporarily stamped on the drawings for printing and viewing purposes using the ZKUtil application. The current metadata are transferred to PRISMA during import. When the PRISMA maturity is set to FMFF, the metadata are automatically stamped onto the drawings and sent to DZA.

For the Body-in-White technology, it is possible for the Suppliers to use “static” title blocks (meaning hand-edited) for their internal use. Please be aware that static title blocks are irrelevant for BMW and that data administration in the sheet enumeration dialog is binding. Static title blocks must be put on layer 350. Prior to packing and archiving in PRISMA, a visualization filter has to be set to ensure that all layers > 300 are not visible.

Attention: Supplier-specific title blocks have to be saved in separate views, which are set to “Hide” before sending data to BMW and archiving in PRISMA. This is necessary to ensure that only the BMW title blocks are printed at BMW. All other drawing elements, which shall not be visible for BMW, have to be removed the same way.

For more detailed information on the general drawing process, please refer to the CARISMA user documentation “drawing management” by logging on at: https://b2b.bmw.com/ >> Departments >> Technologies >> Body-in-White >> Fixture-Design >> List of documents in fixture design.

2.7 BMW Start- and basic models

2.7.1 Start models for CATPart and CATProduct

The following partno. are used as start models: G000198 for CATParts and G000199 for CATProducts.
When working with the CARISMA Structure Manager – which is mandatory for the creation of fixture structures – these start models which are included in the CARISMA installation will be used automatically.

The start models already contain the bill of materials parameter (BOM parameter) for automatic generation of the FM Stüli BOM. These parameter fields appear as a parameter set in the tree and in the file properties (Figure 2-1). The BOM parameter fields can only be modified by direct input to the parameter set in the tree. The data is automatically copied to the file properties.

When the start model is used, data from the file name automatically fills the BOM parameter fields for the following: partno., designation and position number.

Furthermore, the start model features an analysis check function which generates a warning message when the fundamental naming convention has not been observed (see section 2.8).
2.7.2 Basic model for CATDrawing

See section 7.1.1

2.8 BMW-specific naming convention

With CARISMA, certain structure categories are assigned to the CATIA files depending on which of the different valid naming conventions is used. A substantial feature here is whether it is dealt with container-creating files or whether they have to be handled only within a container. The CARISMA container directory and the container-creating files should follow this generic naming convention:

Partno.\_Dwg.index\_DocumentPart\_Alternative\_Designation.Extension
(7 digits) (1 or 2 digits) (1 or 2 digits) (1 digit) (max. 40 digits)

e.g. ../5678901\_A\_1\_A\_EXAMPLE for the directory and
5678901\_A\_1\_A\_EXAMPLE.CATProduct

Single files within a container, according to the generic structure, will be as follows:

Partno. (of the container)\_Position no. of BOM\_Designation.Extension
(7 digits) (4 or 9 digits) (max. 40 digits)

e.g. 5678901\_0101\_PART\_EXAMPLE.CATPart

Refer to section 2.9 for ranges of position numbers and corresponding classifications.

The single parts of a name are separated by two underscores. In accordance to the OEM basic guideline and the PRISMA rules, upper case letters must be used. The CATIA PartNumber is identical to the file name except for the file extension.
**For the Body-in-White and Assembly technologies**, the following structure categories and associated file types with naming conventions are used:

<table>
<thead>
<tr>
<th>Strukturkategorie (Structure Category)</th>
<th>Naming convention</th>
</tr>
</thead>
<tbody>
<tr>
<td>Assembly with Partnumber (Baugruppe mit Sachnummer)</td>
<td>Assemblies (BGs) given a partno., e.g. 5432109_C_1_A_GRUNDRAHMEN.CATProduct. Hint: ASPs in the Body-in-White technology are created as “Assembly with Partnumber” in CARISMA CAD PDM.</td>
</tr>
<tr>
<td>Assembly without Partnumber (Baugruppe ohne Sachnummer)</td>
<td>Modules such as weld assemblies or similar, e.g. 5432109_0001_GRUNDGESTELL.CATProduct</td>
</tr>
<tr>
<td>Part without Partnumber (Einzelle ohne Sachnummer)</td>
<td>For manufactured parts, i.e. single parts to be designed for the assembly, e.g. 5432109_0101_GRUNDPLATTE.CATPart, or if this part is part of a module e.g. 5432109_0001.0105_RIPPE.CATPart</td>
</tr>
<tr>
<td>Catalog Part (Katalogteil)</td>
<td>Single parts of the NuW libraries available from the Body-in-White technology have a fixed partno., position number and designation; version control does not apply, for design-relevant changes a new partno. must be assigned. e.g. 3744785_1090_WINKEL_FUER_AUFAHMEBOLENZEN.CATPART. NuW parts from catalogs, part servers and so on (see section 3.5.2), e.g. __9501_PURCHASE_PART.CATPart</td>
</tr>
<tr>
<td>Working Product</td>
<td>Except for the prefix WP, the name corresponds to the naming convention of “Assemblies with Partnumber”, e.g. WP_5432109_C_1_A_GRUNDRAHMEN.CATProduct</td>
</tr>
<tr>
<td>Drawing (Zeichnung)</td>
<td>Corresponds to the naming convention of “Assemblies with Partnumber” except for the file extension, e.g. 5432109_C_1_A_GRUNDRAHMEN.CATDrawing</td>
</tr>
<tr>
<td>Drawing Flame Cutting Template (Brennschablone)</td>
<td>Corresponds to the naming convention of “Assemblies without Partnumber” without a position number and the file extension. The name will be assigned automatically, e.g. 5432109_-GRUNDRAHMEN.CATDrawing</td>
</tr>
</tbody>
</table>
In addition to the naming conventions described above, pay attention to the parameter called “Source (Quelle)”. This parameter has been integrated into the start models and influences the file name and analysis checks (see Figure 2-2).

The three possible values for this parameter are:

1. Production part (Fertigungsteil) — default setting for single parts belonging to the assembly
   Name: AssemblyPartNumber__PositionNumber__designation.CATPart

2. Standard part (Normteil) — for purchase and standard parts not contained in the NuW libraries. The geometry for these parts can be copied to the start model from Internet libraries.
   Name: free name can be selected without restrictions and check

3. Catalog part (Katalogteil), for single parts from NuW catalogs available by BMW.
   Name: own partno.__fix position number__designation.CATPart.

![Figure 2-2 Parameter Source (Quelle) in the start model for parts](image)

The following example depicts an assembly structure (Figure 2-3).

![Figure 2-3 Tree structure with naming convention](image)

When creating structures and the associated files with CARISMA Structure Manager, the naming convention will automatically be followed (see section 3.1).
2.9 Position number systems

2.9.1 Bill of materials position numbers

Position numbers for modules, single parts and NuW parts are always 4-digit numbers. The only exceptions are single parts in modules which have 9-digits (refer to Figure 2-3). For catalog parts (Katalogteile), the 4-digit position numbers have been already determined by BMW. The position numbers for all non-catalogue parts are assigned by the technical designer based on the ranges depicted in the following illustration (see Figure 2-4).

Overview of the material groups of manufactured and purchased parts

<table>
<thead>
<tr>
<th>Manufactured parts - user defined position numbers for</th>
<th>Reworked parts - user defined position numbers for</th>
<th>Catalog parts with predefined position numbers</th>
<th>Purchase and standard parts not from BMW catalogues - user defined position numbers</th>
</tr>
</thead>
<tbody>
<tr>
<td>Construction e.g. weldment 6001-0100</td>
<td>Part to be manufact. FM 6101-0600</td>
<td>Purchase parts ME 1061-3000</td>
<td>Standard parts NT 7001-0600</td>
</tr>
<tr>
<td>KER B7 Assembling parts 1001-1009</td>
<td>KER K7 Equipment technique 4401-4409</td>
<td>KER N7 Pneumatics 4251-4209</td>
<td>Bearing/guiding</td>
</tr>
<tr>
<td>KER B9 1501-1700</td>
<td>Pneumatics 4251-4209</td>
<td></td>
<td>Connectors</td>
</tr>
<tr>
<td>Projection welding 1001-2009</td>
<td>Hydraulics 4801-6000</td>
<td></td>
<td>Pneofitting</td>
</tr>
<tr>
<td>Weld guns 2501-3009</td>
<td>Electric 4701-4009</td>
<td></td>
<td>Locking devices</td>
</tr>
<tr>
<td>Modular console 3001-3009</td>
<td>Control technique 5501-6009</td>
<td></td>
<td>Miscellaneous</td>
</tr>
<tr>
<td>Euro gripper 3501-3500</td>
<td>Process engineering 6201-6309</td>
<td></td>
<td>Reserve</td>
</tr>
<tr>
<td></td>
<td>Miscellaneous 8001-6000</td>
<td></td>
<td>8501-7000</td>
</tr>
<tr>
<td></td>
<td>Reserve 8501-7000</td>
<td></td>
<td>Number range for parametric catalogue parts with predefined position number 9995</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>9001-9999</td>
</tr>
</tbody>
</table>

Figure 2-4 Position numbers

Please note: Adapters are always given the position number 0000!

2.9.2 Listing components in the bill of materials

To ensure that all components are displayed in the bill of materials from FM-Stüli, it is necessary to turn on the “Visualize in the Bill of Materials” switch. This switch is located on the “Properties” window of each component (see Figure 2-5). The default for this switch is on.
Conversely, components not required in the bill of materials should have the “Visualize in the Bill of Materials” switch deactivated. For example, if a purchased part is represented as a product, then the single parts of that product would not be relevant for the BOM.

For adapters, this check box always has to be disabled! As a result, the adapter will not be shown in the bill of materials since adapters are not relevant to the “bill of materials”.

2.10 Software requirements

External Partners are required to use the same software environment as BMW (especially with respect to identical versions) in order to ensure data compatibility.

2.10.1 CATIA V5

The licensed basic software with required modules can be obtained from the respective business partner. For information about BMW specific service packs, hot fixes and overlapping documentation, please consult the Internet by logging on at: https://b2b.bmw.com/ >> Departments >> Development >> Applications >> CATIA V5 >> “News” respective “BMW Standards and Applications”.

2.10.2 CARISMA Software

For data management and administration of CATIA V5 data, the CARISMA software (successor of T-Basics Software) was developed. CARISMA is a CAA-application integrated in CATIA V5 supporting the file-based creation and modification of fixture structures using the BMW start models. It offers additional functions such as version control and alternating as well as reusing fixture structures. Additionally the software supports scanning of manually created
CA-data, generating a “basic bill of materials” in *.XML format (which can be opened and edited in FM-Stüli) as well as packing and unpacking containers. CARISMA containers are necessary for archiving in PRISMA. Furthermore the software transfers the CARISMA Structure Manager metadata into the drawing title blocks and automatically exports the data to DZA (see section 2.6.2).

In the BMW environment, CARISMA is integrated with the “Immersive Client” program allowing direct access to PRISMA from CATIA V5. The external version of CARISMA for suppliers has all of the features of the internal version, but without the access to PRISMA.

The CARISMA program package with installation instructions for suppliers can be found on the BMW Partner Portal at: https://b2b.bmw.com/ >> Departments >> Development >> Applications >> CATIA V5 >> BMW Standards and Applications.

For advanced CARISMA user documentation, please log onto the BMW Partner Portal at: https://b2b.bmw.com/ >> Departments >> Technologies >> Body-in-White >> Fixture-Design >> List of documents in fixture design.

### 2.10.3 PRISMA bill of materials (FM-Stüli)

For the computer-aided creation/generation of the fixture BOMs, the program “FM-Stüli” (fixture BOM) is available in German and English. Internally at BMW, FM-Stüli is found in BMW IAP (IAP = Integrierter ArbeitsPlatz = integrated workstation). For the external use, the software can be downloaded via the BMW Partner Portal at: https://b2b.bmw.com/ >> Departments >> Development >> Applications >> FM-Stüli.

### 2.11 Making data available by BMW

BMW will supply the necessary CATIA V4 (*.model) and V5 (*.CATPart, *.CATProduct, *.CGR) CAD data to Suppliers for engineering and design via OFTP data exchange. Alternatively data can be exchanged via OpenDXM over a web-based connection for Suppliers without OFTP infrastructure. Suppliers must apply for authorization for data exchange on the BMW Partner Portal at: https://b2b.bmw.com/ >> Departments >> Development >> CA-Data Exchange.

CARISMA containers with existing components and sample designs that conform to the CARISMA standards are available for export from PRISMA or, in special cases, on CD-ROM/DVD. For further details, please refer to the user documentation regarding data exchange for CARISMA.
Libraries for standard and repetition parts are also available in CARISMA container format on CD-ROM, DVD and in PRISMA under the following PRISMA numbers:

catalog KER B7 (design elements Body-in-White, BMW-assembling parts): 5330801
catalog KER B8 (design elements Body-in-White, BMW-welding parts): 5330802
catalog KER K7 (design elements Body-in-White, purchase parts): 5330803
catalog Modular Console: 5330804
catalog Euro Handling Tooling: 5330805
catalog welding gun: 5330806
catalog electric: 5330808
catalog projection welding: 5330809

The containers must be unpacked using CARISMA. To unpack the catalogues, make sure the “unpack as catalog” switch is enabled in the “Unpack container(s)” dialog box.

It is the Engineering Partner’s responsibility to ensure any geometry defining surfaces and spaces are up to date.

Designs for the Assembly department require that vehicle components that contact the fixture or tooling are included in the CARISMA container as Reference Parts.
3 Structuring CATIA V5 assemblies

3.1 General information

The CARISMA Structure Manager is required for creating a new fixture design and for modifying or completing an existing design (see Figure 3-3-1).

![Figure 3-3-1 CARISMA Structure Manager (STM)](image)

The CARISMA Structure Manager generates a file-based CATIA product structure including directories where all of the files and data are stored (see Figure 3-3-2).
3.2 Input data and preparation of data

The ASP is the German abbreviation for Aufnahme- und Spannplan (locating and clamping plan in the Body-in-White technology). The ASP serves as the main adapter representing the central control element for all input data. For more information please refer to section 6.

3.3 Control by adapters

3.3.1 Body-in-White technology

No supplement to the OEM basic guideline.

3.3.2 Assembly technology

Besides control by adapters as described in the OEM basic guideline, the Assembly technology allows the use of Assembly Constraints for fixture design. In this method, the position of all CATParts are defined by these Assembly Constraints. However it is possible to use geometrical references from other CATParts without an active link.

Figure 3-3-2 Creating structures automatically (CATIA tree and file system)
The links to the external references are not kept. The reference geometries are copied automatically into the new CATPart’s “Hide” area and the external references are isolated. There is no concern for the flow of reference links.

The design method to be used requires prior consultation with the Technical Designer responsible for the fixture design.

Prerequisites and settings

In order to design using external references to other CATParts, the CATIA options must be set properly. Open the “Options window” from the “Tools” menu. Select the “General” tab in the “Part Structure” branch in the “Infrastructure” tree. Deactivate the “Keep link with selected object” setting. At this point the reference geometry can be selected without prior publication. (See: Tools - Options - Infrastructure/Part Infrastructure - Restrict external selection with link to published elements; Figure 3-3-3).

Figure 3-3-3 Settings under “Tools – Options”
Assembly constraints in assembly design

Once the new CATParts are created they must be positioned in the structure using the constraints in the Assembly Workbench. Only one CATPart that contacts the vehicle component shall be fixed relative to the vehicle data. Otherwise the part can be dimension relative to a position adapter fixed to the origin. The ultimate goal is to have the structure defined in the vehicle coordinate system relative to the reference vehicle components (see section 5.1). For all fixture design, the left side of the vehicle is to be used as the reference.

When the vehicle-contacting CATPart is part of an assembly, the superior CATProduct must be fixed in the zero position. For all other assemblies, the position-defining, relevant CATPart should be fixed within the CATProduct. All CATParts and CATProducts are then positioned relative to the vehicle-contacting CATPart using the Assembly Constraints, Coincidence, Contact, Off-set, etc. The “Fix Together” constraint, as an exception, should only be used in necessary, rare circumstances.

The constraints have to be structured in meaningful, named sets. For tracking purposes, ensure that the instance name coincides to the part no.

Assembly-overlapping parametrics

For CATProducts, make sure the parameters are added for assembly constraints that illustrate the kinematic movement of the fixture, e.g. the travel path of a clamping fixture. These parameters must be named to include the prefix “KIN”, the name of the movement function and indicate the possible path area or path position e.g.

KIN_Spanner_unten_rechts_(0°to45°)
KIN_Schieber_(-10mm,20mm,30mm)

In the “Edit parameter” dialog box, limit the value range to the constructive range of movement using the context menu and the option “Add Range”. When changing the driving constraints, no inconsistency is allowed. For large tools with sub-assemblies, be sure to include an explanatory overview of the driving Parameters.
3.4 Link flow with control by adapters

Figure 3-4 shows an example of the assembly structure and the link flow.

The clamping point elements of the ASPs/main adapters are copied to the assembly adapter and then published. Support and pressure pads reference these clamping point elements (see Figure 3-5). In the assembly adapter, a support/pressure pad plane must also be created and, when possible, based on even dimensions in the vehicle grid. For positioning the contour pieces, reference elements in the contour pieces (intersection planes, clamping points and connecting plane) are exchanged for identical clamping mark references of the assembly adapter. Please structure the design for a grid-oriented layout of the contour pieces.
Positioning has to be done by manually replacing the reference elements in the geometrical set “Control elements adapter”.

For the further assembly design, reference elements of the corresponding support and pressure pads or location bolts shall be used, e.g. connecting contours, hole axes, intersection plane supports, etc. Do not use clamp mark reference elements.

**Important note:**
The use of assembly adapter clamp mark references for the remaining assembly design must be avoided. Higher-ranking, superset references such as tool design planes are taken from the ASP/ main adapter.
3.4.1 Direct link flow in the Body-in-White technology

Within an assembly, parts are allowed to directly exchange their published references. In this case, no link flow via the assembly adapter is required.

Hole patterns / geometry contours used for referencing subsequent assemblies can be controlled through surface connecting contours or sketches containing center points. This way, it is possible to use points defined in an additional sketch for creating the hole patterns / drilling patterns. By publishing the sketch, references can be transferred, e.g. to a tool plate. The sketch offers the advantage that the holes can be made without manually generated constraints.

In an assembly, it is recommended to first position a clamp. The assembly structure has to be related to the clamp references, e.g. the clamp intersection plane shall be used as sketch plane. In any case, the clamp should be referenced.

Prior to generating the drawing, the clam, or another geometry-determining standard component, must be fixed as the last step despite the clamp's position being defined. This fixed component is also called the geometry node since it controls most of the other designed CATParts in the assembly. The additional fixing of the clamp has to be renamed to “GEOMETRIESICHERUNG” (= safeguarding geometry). Since the assembly design references relate to the geometry-defining standard component, the fixing prevents the assembly from accidental modification.

The hole pattern/drilling pattern of the assembly has to be transferred to assembly adapter of the tool plate/ screw-down surfaces through a published surface connecting contour or sketch with points.

Conclusion:
Modeling new CATParts is done in context or in connection with their respective CATProducts. The CATParts are designed based on the other CATParts, e.g. single fixture parts, vehicle components or based on the assembly adapter. Their position and shape is defined/determined via external references. Standard and repetition parts (NuWs) are an exception here, they are located with assembly constraints.

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

For the Body-in-White technology, the standard and repetition parts (NuWs) are taken from the BMW standard catalogues (NuW). Within the Assembly technology, the use of BMW catalogues is to be verified for each specific project.

When a purchased part is not available in the NuW catalogue, but it is released in the Body-in-White technology's purchased parts list and required multiple times, then the part must be
created with a project order as specified in the “Guideline for the design of standard- and repetition parts with CATIA V5”. Further details are to be coordinated with the BMW project leader and the CAD-library team (rohbau.standardkomponenten@bmw.de).

All standard and repetition parts used in a project (tooling designs or a single device) need to be included in the data exchange. The archived contents have to be complete, consistent and free of links to centrally saved libraries.

Clash relevant standard parts, such as nuts and bolts, must be shown in the design. For the Body-in-White technology, only the relevant part of the geometry like bolt heads, nuts and washers should be simplified and displayed.

For each device, a sub-folder called “NuW” has to be created under the “root-folder”. The NuW folder must contain all standard and repetition parts.

Advantage of this method: NuW data are only saved once per device. This reduces the data size for data exchange and stored in PRISMA.

### 3.5.1 Integration from BMW libraries

All standard and repetition parts (NuW = Norm- und Wiederholteile) used in the device have to be copied to the NuW folder from the PRISMA catalogues (B7, B8, K7, Modular Console etc.). Once in the NuW folder, they have to be inserted into the various assemblies as “Existing Component”.

Alternatively, it is possible to design with the option “with link to a centrally saved catalog”. Prior to completing the design, the standard and repetition parts have to be detached from the catalog and saved in the NuW folder.

**Utilization of parameterized standard- and repetition parts with preallocated position number 9999:**

For NuW parts, available in various dimensions, only one value per type or size is available as CATPart (profiles, axle support, guides etc.). The sample part is always the shortest length.

Systematics of partno. and filename is as follows:

3699413__9999__AXLE_SUPPORT_WITH_BOLT_L40

First, copy the NuW part to the NuW folder. Subsequently the position number 9999 needs to be replaced with a consecutive position number between 3601 and 4000 and the desired length entered in the part. Rename the file name, e.g.:

3699413__3601__AXLE_SUPPORT_WITH_BOLT_L45
Finally, adjust the value / length of the given parameter in the parameter-set “Geometriesteuerung” (geometry control) in the CAD model.

**Method for parts that have to be reworked:**

When using catalog parts to be reworked, the NuW part has to be copied to the folder of the assembly.

The default NuW position number needs to be replaced with a consecutive position number between 0601 and 1000 (see Figure 2-4). The SAP-Ident number and the US-Ident number must be deleted. The given partno. remains unchanged. These entries have to be checked in the bill of materials parameter set and, if applicable, adjusted manually.

### 3.5.2 Integration from other sources

The use of standard and repetition parts from other sources (e.g. purchase part libraries from the supplier, CADENAS, DIN- / ISO-parts from part servers, etc.) is permitted. The workflow is similar to BMW library parts. Save the part to the NuW folder. The position numbers should be selected from the following number range: 9501 – 9998. If these NuW parts have to be reworked, make sure to use the correct position numbers (see section 3.5.1).

To enable automatic generation of the bill of materials (SL document in PRISMA), the BMW start models and the catalog parts have a parameter-set “Stueckliste”. When NuW parts without these parameters are used in a design, perform one of the following:

- After adding the NuW part to the CATIA Structure with the function “insert existing component”, the filename and CATIA partno. must be changed as follows: __positionnumber__designation.CATPart (e.g. __9501__PURCHASE_PART.CATPart). By Use CATIA “Save Management” to save the part to the NuW folder for the fixture. A definition of a structure category and free attributes within the CARISMA Structure Manager is not necessary.

Afterwards the missing parameters are added by running the macro “Ergaenzen_BMW_CARISMA_Stuecklistenparameter.CATScript”. The macro can be downloaded from the BMW Partner Portal at [https://b2b.bmw.com/](https://b2b.bmw.com/) >> Departments >> Technologies >> Body-in-White >> Fixture-Design >> Downloads.

Please ensure that the parameter “Quelle” (source) is set to “Normteil” (standard part) (refer to Figure 2-2 ). Then fill in the necessary parameters in the bill of materials parameter-set. The parameter “SachNr” (partno.) remains empty. Enter the position number corresponding to the specified number range from 9501 to 9998 for NuW parts and from 0601 to 1000 for reworked NuW parts (see section 2.9.1).
The same macro may be used for existing CATParts and CATProducts that were created without the basics models. However, in CARISMA Structure Manager, the Structure Category must be edited and changed to “Part without Partnumber” or “Assembly without Partnumber” respectively.

- As an alternative, a new part may be added to the structure with the CARISMA Structure Manager function “insert using template” and selecting “Part without Partnumber”. Enter the free attributes “BOM Name” for the designation and “BOM Number” (number range from 9501 to 9998). The input box “Catalog Part Number” remains empty. Within the CARISMA Org-Data Manager the structure category “Catalog Part” has to be chosen. “Save and Update” automatically creates a new PartNumber and filename.

The geometry is copied and inserted in this new CATPart. The necessary bill of materials parameter-set has to be filled completely as described earlier. Use “Save Management” in CATIA to save the part to the NuW folder of the fixture.

### 3.5.3 Assembly-overlapping re-use of in-house production parts

For the **Body-in-white technology**, re-used, in-house production parts must be given a unique partno. and archived in PRISMA with a separate detail drawing. During tooling design, these parts are handled like NuW parts. As alternative, the part is designed as a normal, in-house production part in the context of the assembly where it is used for the first time. For all other, instances, the part has to be copied independently by “file – new from” to the other assemblies. For each use, a detail drawing is necessary.

For the **Assembly technology**, the re-use of parts is only allowed within the context of an upper-level tool. The drawing of each tool (with its sub-assembly partno. drawings) has to be complete. Re-used parts and modules are detailed in the drawing of the first-use partno. and the BOM-parameter "comment" has to be appended by the note "drawing see partno. ...".

![Diagram](image)

Re-use of parts and modules ("Assembly without Partnumber") in a different tool is only allowed as an independent copy using "File - new from". The structure category and free attributes have to be updated in CARISMA Structure Manager and the drawing has to be appended.
3.5.4 Integrating a NuW-CATPart from a BMW-catalogue with constraints into an assembly

This method represents the normal case scenario and must be employed if NuW parts do not change geometry. Make sure constraints are created only within an assembly. It is best to constrain the NuW part to the set points in the adapter or directly to the reference part.

To reduce the risk of errors, constraints must only be created using the published elements.

Using existing reference elements in standard parts for the further design is possible thanks to the admitted link flow. This means the part-overlapping use of “surface connecting contours”, “hole patterns / drilling patterns”, etc. is possible without the need of an adapter.

The standard parts B7 / B8 and the purchased parts are equipped with identical reference elements. Reference elements such as surface connecting contours, drilling axes, drilling template points, construction planes, intersection planes, etc., shall be used for designing the assemblies (see Figure 3-5 and Figure 3-6).
3.6 Working Products

The use of Working Products is suggested if several design engineers are working on one design or if the scope renders it obvious to split the scope to increase performance.

Working Products are no longer created automatically since the introduction of CARISMA CAD PDM. If a Working Product is needed, it has to be created using CARISMA STM. The Working Product becomes the name of the assembly product with the prefix "WP__", e.g. WP__1234567__A__1__A__Name.CATProduct.

It is saved to the same directory as the corresponding structure and archived in Prisma.

3.7 Mirroring Parts and Products

The macro “Mirror_Part.catvbs” can be obtained from: https://b2b.bmw.com/>>> Departments>> Technologies >> Body-in-White >> Fixture-Design >> Downloads.

Within BMW, the macro can be selected from the following path:
L:cat/v5/rohbau/v5_powercopies_makros/allgemein/ (Windows XP) or cat/v5/rohbau/v5_powercopies_makros/allgemein/ (UNIX)
4 Structuring CATIA-V5 in-house production parts

4.1 Basics

Basically, all parts designated on the bill of materials are to be designed as a separate CATPart. In doing so, various problems can be avoided when creating the drawing such as different hatching of the parts and parameters for the bill of materials.

In general, when designing a part, the result has to be a “Solid. Avoid creating surface-based parts whenever possible. In the event that a surface model is absolutely necessary, a “Solid” has to be derived from it. This process guarantees a faultless conversion with following systems.

4.2 Body structure

PartDesign in CATIA V5 requires a clearly structured specification tree for perfect construction, trouble-free updates and model clarity. Basically, the more features used and integrated into the “Part”, the greater the importance for structuring the specification tree with additional “Bodies”. This ensures that Technical Designers, not involved with the design of the part, are given a clear overview that allows them make necessary modifications.

For more complex parts, the body structure, as saved in the start model and as described in the OEM basic guideline in section 4, shall be used. The start model for CATParts already contains body structure with color coding for the different machining processes, such as grinding, rough machining, etc. (see section 2.3). Using these bodies is not mandatory at BMW, but desired because of the better clarity (Figure 4-1).

Many “Parts” in the fixture design are only composed of some features. Therefore, it is not necessary to choose a special tree structure. All features can be created in the PartBody.

Parts based on flame-cutting templates are to be designed in accordance with the specifications made in the OEM basic guideline.
4.3 Output elements

As synonym to “Output” or “Output elements”, BMW uses the term “transfer features” (=Weitergabeelemente).
5  3D-design

5.1 Axes and position in space

All CATParts that are designed by means of external references shall be in vehicle position.

5.2 Sketches

No supplement to the OEM basic guideline.

5.3 Illustration in open position

For illustrating the “open position” state, the Body-in-White technology department recommends variant 2 (Component) of the OEM basic guideline.

For the Assembly technology, all working positions are generated and made available as ALLCATParts. The ALLCATPart – representing the working position – is used as document part 1, e.g.

5678901__A__1__A__EXAMPLE_ALLCATPART.CATPart.

Further positions are created with consecutive document parts such as

5678901__A__2__A__EXAMPLE_ALLCATPART.CATPart.

When classifying the All.CATParts in CARISMA, enter the part description in the comment field for each part, e.g. working position or open position.

5.4 Accuracies

Unlike V4, model accuracy in V5 is no longer adjusted. Consequently, 3D always works with the V5 accuracy.

5.5 Layers

No supplement to the OEM basic guideline.

5.6 Design tables

No supplement to the OEM basic guideline.
6 Aufnahme- und Spannplan (ASP) (= support & clamping plan) in the Body-in-White technology

6.1 Generalities

The ASP, German abbreviation for Aufnahme- und Spannplan (support & clamping plan), forms the basis for each fixture of an assembly sequence. This plan shows the component-touching geometry, such as locating pins and support & pressure pads. Consequently, it has to be considered as a fixture-component. The following description relates to the CAD model required for manufacturing the clamping pieces.

For the preliminary states required during the planning process (Basic ASP), the degree of detailing (itemization) has to be defined with the project leader.

For the ASP of one device/station, a CATProduct is created. This CATProduct includes a CATPart defining the so-called clamping marks. The components referenced by the ASP are directly in the ASP-Product as well, if applicable/necessary, collected in a SubProduct (Figure 6-1). Additionally, there is another CATPart for each support and pressure pad or each locating pin.

Figure 6-1 CATIA specification tree of the ASP
6.2 Working method clamping point

The support and pressure pads (contour pads, clamping pads, clamping blocks, etc.) are intersected with the parameterized clamp marks (Spannmarken) in the ASP. The clamp marks represent a clipping / cutout of the component surface. To create a clamp mark, the corresponding vehicle component has to be loaded with the CATPart of the ASP as the active part. The clamp marks are generated using “Powercopy 2_Spannmarken_xxDatumxx.CATPart” made available by BMW.

The “PowerCopy” as well as the “Renaming Macro” (see below) can be obtained from the Internet by logging on at: https://b2b.bmw.com/ >> Departments >> Technologies >> Body-in-White >> Fixture-Design >> Downloads. Within BMW, these files can be selected from the following path: L:\cat\v5\rohbau\v5_powercopies_makros\Serien-ASP (Windows XP) or cat/v5/rohbau/v5_powercopies_makros/Serien-ASP (UNIX).

The PowerCopy requires the component, a reference plane, a clamping point (Spannpunkt) – determining the position of the clamp mark – and an intersection plane (Schnittebene).

The specification features for the PowerCopy are generated in the body “Control features adapter” (=Steuerelemente Adapter) and named as follows:

Clamping points: SPP_XXXX, where XXXX is numbered consecutively from 0001 to 9999.
Intersection planes: SPLN_XXXX, where XXXX is numbered consecutively from 0001 to 9999.

The elements generated with the PowerCopy have to be renamed and published according to the “Vorgabeelemente” (specification features). The generated parameter sets also have to be renamed according to the “Vorgabeelemente” (specification features) (see Figure 6-2).

Important: Each time clamp marks are created, parameter sets have to be renamed at once, because it is very difficult to later assign the parameter sets to the respective clamp marks. As an aide, the BMW macro “Umbenennenmakro_2_Spannmarken_xxDatumxx.catvbs” is available at the above mentioned paths.

In the “Body” “Weitergabeelemente” (transfer features), e.g. the tool mounting plane is defined.

Advice: instead of executing Powercopy and a separate renaming-macro, BMW offers, upon request, a start-script which combines these two steps and simplifies the execution. However, this requires the KT1 license for CATIA V5.
Figure 6-2 Clamp marks generated with the PowerCopy, parameter sets and published features of the clamp mark
6.3 Describing parameters in PowerCopy

The clamp mark PowerCopy contains some parameters and rules the properties of which will be explained in the following (see Figure 6-3).

```
Parameters

`Breite_Spannmarke.XXX` = 25 mm
`Länge_Spannmarke.XXX` = 25 mm
`Drehwinkel_Spannmarke.XXX` = 0°
`Projektionswinkel.XXX` = 0°
`Toleranz.XXX` = 0.2 mm
`Spannklotzfertigung_01.XXX` = CNC
`Spannklotzfertigung_02.XXX` = CNC
`Schiefe Ebene erlaubene_01.XXX` = Ja
`Schiefe Ebene erlaubene_02.XXX` = Ja
`Autom Fertigungswahl_01.XXX` = Nein
`Autom Fertigungswahl_02.XXX` = Nein
`Extrapolation_Spannmarke_01.XXX` = 10 mm
`Extrapolation_01.XXX` = Nein
`Extrapolation_Spannmarke_02.XXX` = 10 mm
`Extrapolation_02.XXX` = Nein
```

Figure 6-3 Parameter set of clamping mark PowerCopy
Width of clamp mark (Breite_Spannmarke) and length of clamp mark (Laenge_Spannmarke): Here, the user may change the size of the clamp mark (Figure 6-4).

![Figure 6-4 Adapting width- and length of the clamp mark](image)

Rotation angle clamp mark (Drehwinkel_Spannmarke): This option allows rotating the clamp mark around its axis (see Figure 6-5).

![Figure 6-5 Adapting the rotating angle of the clamp mark](image)

Projection angle (Projektionswinkel): Here, the projection angle can be tilted from the clamp point onto the component (Figure 6-6).

![Figure 6-6 Adapting the projection angle of the clamp mark](image)
Automatic selection of production type (Autom. Fertigungswahl):
The default parameter value is “No” and should be used as such. This way, the clamp mark
 corresponds exactly to the component surface. The information about the production method
 must be stated when creating the drawing. Under these boundary conditions, the parameters
 Tolerance (Toleranz), Clamping pad manufacture (Spannklotzfertigung) and Allow inclined
 plane (Schiefe Ebene erlauben) are irrelevant.

Extrapolation and extrapolation clamp mark
The “Extrapolation” parameter allows the user to select whether the clamp mark shall be
 extrapolated or not. An extrapolation may be necessary at component edges, if the clamp mark
 itself is too small for cutting the contour piece. Thanks to the parameter “Extrapolation_clamp
 mark”, the user may select how far the extrapolation should go. These parameters can be set
 separately for both clamp marks generated.

6.4 the locating pins, support and pressure pads
Two alternatives for the design of locating pins, support and pressure pads (clamping pieces)
 are permitted. The alternative to be used requires prior consultation with the Technical
 Designer responsible for the fixture design. The alternative must be kept throughout the
 project.

6.4.1 Alternative 1 – Detailing within the Assembly
With Alternative 1, generally all locating pins, support and pressure pads (clamping pieces) are
 designed and detailed/itemized (drawing derivation) in the respective assemblies. Only 3D
 geometries are pasted into the ASP as instances.

After copying all CATParts of the locating pins, support and pressure pads from the assemblies
to the ASP CATProduct, the user may now generate the drawing. Since only the “ASP
 CATProduct” summarizes the entire component-contacting geometry, an ASP overview
drawing has to be generated, representing an assembly overview drawing.
Drawing contents of the ASP in case of alternative 1 (sheet 1 and continuation sheets, if situation requires so):

- Overview of the support and pressure pads with component geometry. Depending on clarity of the component-join, the component geometry can be either derived from the ASP CATPart or from a geometrical set in the ASP CATPart with boundaries.
- Assembly partners of the locating pins or support and pressure pads
- Vehicle grid

![Figure 6-7 Clipping of an ASP overview drawing](image)

6.4.2 Alternative 2 – Detailing within the ASP

With Alternative 2 all locating pins, support and pressure pads (clamping pieces) are designed in 3D, either in the respective assemblies or directly in the ASP for a new design.

If all locating pins, support and pressure pads (clamping pieces) are designed in the respective assemblies, then all CATParts of the pins, support and pressure pads (clamping pieces) have to be copied from the assembly folder to the ASP folder. A (file based) renaming to the ASP partno. is necessary. Afterwards they are added to the CATIA structure of the ASP.CATProduct using “insert existing component”. The next time the CARISMA Structure Manager is opened, the CATParts will be scanned. Finally, the structure category “Part without Partnumber” has to be assigned. In the assembly, the pins, support and pressure pads (clamping pieces) are changed to the “Hide” mode.
If an assembly is taken over, new pins, support and pressure pads (clamping pieces) are always to be designed in 3D within the ASP.

Basically all locating pins, support and pressure pads (clamping pieces) are given the ASP partno. and are also detailed (drawing derivation) in the ASP.

**Drawing contents of the ASP in case of alternative 2**

**Sheet 1**

- Overview of the support and pressure pads with component geometry. Depending on clarity of the component-join, the component geometry can be either derived from the ASP CATPart or from a geometrical set in the ASP CATPart with boundaries.
- Corresponding assembly partnos. of the locating pins or support and pressure pads
- Vehicle grid
- Sectional views

![Figure 6-6-8 overview sheet](image-url)
Sheet 2 and/or further continuation sheets

- Cross sections of the support and clamping points
- Position numbers of the single parts
- Vehicle grid including position of the single parts
- If clarity of the drawing is not impaired, the cross section views can be dimensioned. Otherwise a detail drawing has to be created. In such case, a vehicle grid is not necessary.
- For NC-machining, the “DATENSATZFLÄCHE” detail has to be used. This detail has to be filled in with a leader arrow indicating the edge of the contacting contour piece.

Figure 6-6-9 continuation sheet with detailing
7 Fixture drawing execution

This section is a supplement to the currently valid drawing guideline BV-Documentation (follow-up document of BV-Z0). Log on to the partner portal at: https://b2b.bmw.com/ >> Departments >> Purchasing / indirect material >> Production Equipment Specification / Standards >> Overview of the existing production equipment specification.

7.1 Execution guidelines

For all 3D models created in CATIA V5, a drawing is also generated in CATIA V5.

7.1.1 Basic model

In general, the basic model G000060, is to be used for all drawings (CATDrawings and the basic model G000066, for flame cutting templates (see section 7.2). Using CARISMA Structure Manager (see section 2.10.2), required for creating new structures, guarantees the correct basic models are used.

The manual for using basic models from the BMW Partner Portal has to be applied. The basic models used for tool design have been derived from a standard BMW basic model. The manual is located at: https://b2b.bmw.com/ >> Departments >> Technologies >> Body in White >> Fixture- Design >> Downloads >> Applications

7.1.2 Storage state

The drawing has to be saved in the “Working Views” mode with sheet01 (=Blatt01) being active. All frames have to be created using the frame application made available by BMW. Drawings must not be provided with title blocks, they are generated by automation in PRISMA/DZA. To guarantee automated supply to the DZA, each CATDrawing has to be classified with the sheets numbered by using the “Sheet Enumeration Dialog” in the CARISMA Structure Manager (see section 2.6.2). All 2D details must be detached from the external catalogues (Expose 2D). Details originating from the basic model don’t have to be “exposed”.

7.1.3 Drawing derivation

Default sheet size is DIN A0. Each sheet should contain the details for several individual parts. A CATDrawing consists of one or more sheets named “Sheet01”, “Sheet02”, “Sheet03”, etc. (or “Blatt01”, “Blatt02”, “Blatt03”, etc.). As a rule, each partno. should contain only one CATDrawing. Splitting the drawing into several CATDrawings for performance reasons is only
permitted in well-founded, exceptional cases. Deriving an assembly drawing has to be done from the assembly-forming CATProduct. With detail drawings, the respective CATPart has to be selected from the product structure. Deriving a drawing from a Working Product is not permitted.

### 7.1.4 Left- / right- / mirrored execution

Components that appear on both the left and right sides of the vehicle, i.e. doors, side frames, etc., require separate partnos. Therefore, separate partnos. are required for corresponding fixtures. When the right-side fixture is the symmetrical opposite of the left-side fixture, only the left-side data is created. This procedure is to be clearly marked on sheet 1 of the drawing.

```
5 555 555 B7 left as drawn
5 555 556 B7 right, mirror-inverted
```

The standard convention is to use an odd partno. for the left side and the subsequent even partno. for the right side.

### 7.1.5 Standard parts

The customary DIN- or ISO-parts such as bolts, nuts, washers or parallel pins do not appear on the BOM (bill of materials). Entering a note into the drawing, e.g. at the centre line, will be sufficient. Details of frequently used cylinder head screws and cylinder bolts may be omitted, if the following drawing detail is placed on page 1 of the drawing:

```
Cylinder head screw ISO 4762-M8x35-8.8
and Cylinder bolt ISO 8734-6x35-A-St
without drawing notation
```

In the Body-in-White technology, Sandwich plates (Shims) no longer have to be added to the fixture designs (design with a 3mm offset). Assigning of position numbers in the 2D Drawing is obsolete. Instead, a drawing detail “sandwich plates according to KER B7 not shown” is added to page 1 of the drawing. To avoid additional manual effort, the sandwich plates are not relevant for the Bill of materials (no completion with the FM-Stüli editor).

### 7.1.6 Single part drawing

Each single part has to be provided with a single part title box. The single part title box can be found on the detail sheet of the basic model.
7.1.7 Assembly drawing

On the 2D-views, the mounting dimensions and the assembly numbers are entered.

7.1.8 Rework drawing

If purchase parts and standard parts are changed, the comment “Rework” has to be entered into the BOM (bill of materials) under Comment. In the detail drawing, only the dimensions necessary for the modification have to be shown.

7.1.9 Invalid drawing sheets

In the event of a sheet becoming invalid, it has to be treated as empty sheet and can be used with a later version. The existing sheets keep their original sheet number and the total number of sheets on the main title block remains correct. Furthermore, the entries in the BOM column E/Z (E = Einzelteil = single part, Z = Zusammenbau = assembly) do not need to be corrected. In the sheet overview history, you will be given the assignment of the sheets per version.

History sheet overview (sample):

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>Continuation sheet – single parts</td>
</tr>
<tr>
<td>4</td>
<td>Continuation sheet – single parts</td>
</tr>
<tr>
<td>3</td>
<td>Empty sheet</td>
</tr>
<tr>
<td>2</td>
<td>Continuation sheet – intersections</td>
</tr>
<tr>
<td>1</td>
<td>Main sheet</td>
</tr>
</tbody>
</table>

7.2 Drawing derivation of flame-cutting templates

According to the OEM basic guideline, flame-cutting template drawings are derived from the Body “function part” (Funktionsteil). In general, separate CATDrawings are created for flame-cutting templates which are not classified in CARISMA (no supply to DZA). Flame-cutting templates must be generated without a drawing frame, based on the basic model G000066. This basic model is automatically used when the flame-cutting template is created with the CARISMA Structure Manager function “Insert using template” > “flame cutting template”. Finally, using the “Save as” function in CATIA, save the required sheets as *.dxf files to the container folder. The *.dxf files are to be classified in CARISMA as PRIMSA documents with the document type “ANLDOK” and format “MO”.

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8 Documentation

8.1 Bill of Materials (BOM)

For **Body-in-White technology**, the BOM needs to be completed using the BOM-Editor (FM-Stüli) — if the situation requires so — after using CARISMA BOM-Generation for the first creation of the BOM (see section 2.10.3). The defined position numbers are adopted. For further information about the bill of materials, please consult the “Guideline for the application of PRISMA jig part list”. To do so, log on at: [https://b2b.bmw.com/](https://b2b.bmw.com/) >> Departments >> Technologies >> Body-in-White >> Fixture-Design >> List of documents in fixture design.

For **Assembly technology**, the BOM-parameters have to be filled completely within CATIA (see section 2.7.1 and 2.9.2). The BOM has to be created using CARISMA BOM-Generation and must not be modified or extended afterward in FM-Stüli (BOM must match 3D-model).

A unique bill of materials will be created for the right side. In that case the following rule shall be considered:
On the “right side” bill of materials, always enter a reference to the set of drawings. The BOM-no. and the partno. will not match when the right-side device is symmetrically opposite to the left-side device for which the drawings are created.

8.2 Shim documentation in the Body-in-White technology

Besides each dimension defining KER sandwich plates set, a label with the fixture partno. 3 491 999 B7 is positioned as 3D-detail for the shim documentation. The assigned assembly partno. and the associated KER sandwich plate set no. are engraved on this label. This information must be clearly stated in the drawing. The manufacturer has to enter the ACTUAL values of the shims in an associated EXCEL spreadsheet. For more information, please see the B7 production equipment specification.

8.3 Identification label

The identification label for **Body-in-White technology** is defined in the Building Specification for Body-in-White Fixtures.

For **Assembly technology**, the label with the partno. 3 741 452 B7 has to be used. The information for the label is taken from the title box of the drawing.

All coordinates of the existing calibration holes have to be identified on the label with the partno. 3 491 997 B7.