

**ZEISS CAD guideline for creating 3D models
and drawings with the CAD system PTC Creo
Parametric**

A specification for engineers and design service provider

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CARL ZEISS AG



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(I) Creo models and drawings as documents in the PLM system

At CARL ZEISS AG, Creo files are numbered according to strict rules and stored as documents in the PLM system (SAP PLM) to ensure quality assurance and product liability.

(II) Model parameters and names of Creo files

If a Creo workstation of the CARL ZEISS AG with an online connection to SAP is available, the model parameters and names of the Creo files are automatically maintained by the SAP PLM integration.

If there is no online connection to SAP the model parameters and Creo file names must be maintained manually:

- the PLM metadata (i. e. administrative data such as number, name, laboratory, creator,...) must be maintained in the Creo model parameters
- the rules for the creation of file names must be followed ([see chapter 4](#))

(III) General guidelines for working with Creo

Creo models and drawings must be created according to given rules which guarantee robustness, easy changeability and reusability as well as the use of the data in the manufacturing process ([see chapter 1](#)).

(IV) Creo-Konfiguration

Creo data creation according to the specifications of the CARL ZEISS AG is supported by Creo configuration files (config.pro, zeiss.dtl ...) and templates (start models, drawing frames ...). The files can be downloaded under <https://www.zeiss.com/cad>.

(V) Creo Version

The Creo version to be used is specified by the CARL ZEISS AG.

(VI) Design requirements

When selecting base materials, surface treatments, standard and catalogue parts, the specifications such as RoHS conformity, lists of prohibited substances in medical technology and preferred ranges must be considered ([see chapter 2](#)).

(VII) Control of data quality

The data quality and adherence to the rules described in this document is checked (ModelCHECK, as well as checks by the PLM system, [see chapter 3](#)).

(VIII) Additional agreements

The specifications described in this document are minimum requirements and can be extended and modified by additional agreements if required.

1 Quality requirements for Creo Objects

1.1 Principles of 3D modelling

- The starting models of the CARL ZEISS AG are to be used
- External references are only permitted in justified cases after prior consultation with the customer
- Unwanted external references must be eliminated
- In particular, it is important to avoid external references of design elements that were created in assemblies on part level
- Required references within assemblies should preferably be transferred via skeleton models
- Set the Scopus in the Reference control settings to "Skeleton"
- External references may only be created with reference type "External"
- Family tables may only be used in justified exceptional cases, after prior consultation with the client, on a one-step level and only for instances of the same model (Please note that release, versioning and change management must always be carried out together for all family members)
- Multibodys (new functionality Creo 7.0) are only permitted in exceptional cases after consultation with the customer
- Only standard part models from the Zeiss standard part library are allowed
- The "Restructure" assembly function must not be used. Instead, components must be reassembled in the corresponding assembly
- Model units are to be set for the **length** in **mm** and for the **mass** in **kg** (permissible also g or t)
- In relation to the units of length and mass, the **density** shall be given in kg/mm³ or in g/mm³ or t/mm³. (Standard density value of the start model is 2.7e-06 kg/mm³)

1.2 Minimum requirements for the 3D model

- Models must be robust, easily modifiable and error-free
- The insert mode must not be active
- Sketches are as simple as possible and the number of 10-15 elements per sketch should not be exceeded
- Circular references are not allowed
- Avoid geometric errors. Geometry errors are only permitted in very complex models, e. g. castings in exceptional cases after consultation with the customer (Set suitable accuracy at the beginning of creation!)
- UDFs may only be placed independently
- Suppressed features must be deleted
- Incomplete features are not allowed
- Hidden features must be deleted (features only cuts air or is completely hidden within solid body)
- Same features should be patterned as far as possible
- No suppressed or frozen components are allowed in assemblies
Packaged components (components without placement specifications) are only allowed in mechanisms
- No hidden components (before transfer unhide and save layer status!)

1.3 **Layers**

- Threads must be placed on the layer "GEWINDE"
- The layer status "isolated" is prohibited!
- Further layer regulations and layer names must be agreed with the customer

1.4 **Combination State**

The following combination states must be created in the models:

- Z_ALL
- Z_NOTES
- Z_SURFACES
- Z_TOL_DATUM

The following annotation elements must be added to the combination state **Z_ALL**:

- All notes except drilling notes
- Controlled dimensions of all kinds
- Standard surface finishes
- shape and position tolerances
- Symbols of all kinds
- tolerance references

The following annotation elements must be added to the combination state **Z_NOTES**:

- All notes except drilling notes

The following annotation elements must be added to the combination state **Z_SURFACES**:

- surface finishes

The following annotation elements must be added to the combination state **Z_TOL_DATUM**:

- Controlled dimensions
- General 2D Symbols
- shape and position tolerances
- tolerance references

1.5 **Design of base parts and finished parts**

The design of base parts and finished parts must be carried out with external merge functionality.

Other procedures must be agreed with the customer.

1.6 **Large assemblies, simplified representation/replacement models**

For large assemblies, the delivery and maintenance of simplified representations or replacement models can be agreed.

1.7 Requirements for drawings

The file zeiss_DE.dtl must be used as the default drawing setup file.

The use of different drawing setup files, such as zeiss_US.dtl or zeiss_CN.dtl, must be agreed with the customer.

1.7.1 Line styles and text fonts

- Self-defined line styles, xhatching, text fonts or layer are not allowed.
- The standard font is Isofont.
- For glass hatching, use the glass hatching files from the Zeiss Library.
- Line styles for glass hatching must be copied to the Creo working directory.

1.7.2 BOM balloons

- Components in group drawings are marked parametrically via Pro/REPORT using tables with repeating ranges and symbols.
- The parameters CZ_POS_TXT and CZ_POS_TXT2 must be maintained by agreement.
- The Pro/REPORT tables must be placed inside or above the drawing frame. They must not be cut through the extended sheet boundaries.

1.7.3 Symbols / Symbol library

- New drawing symbols may be created but must be compliant with the current ISO standards.
- The symbols from the ZEISS symbol library are preferred.

1.7.4 ZEISS drawing frame

- The ZEISS drawing frames must be used.
(special drawing frames must be coordinated with the ordering business area)
- For multi-sheet drawings, use the same frame size for all sheets.
- The drawing text field is not to be filled in. Instead, all necessary information must be entered in the corresponding model parameters in the 3D model.
The ZEISS drawing frames are only used for orientation to divide up the space on the drawing.
- The drawing text field is automatically filled in when entering the data into the data management system.

1.7.5 Dimensions and tolerances

- Preference is given to displaying dimensions defined in the model
- Geometric Dimensioning and Tolerancing must be generated in the solid model (no 2D symbols!)
- Dimensional tolerances must be set in the model using the dimensioning properties

1.7.6 Views

- 2D elements have to be created view related
- The view representation is to be defined for all views.
Standard: "Visible edges", "Tang dimmed".
Deviations are to be consciously switched.

1.7.7 Not permitted are

- Incorrectly regenerated drawings and non-associative (magenta colored) dimensions
- Views outside the drawing frame
- Coloured drawings, shaded views and OLE objects must be coordinated separately!

2 Design specifications

2.1 *Use of standard parts and purchased parts*

- Standard and purchased parts are added at ZEISS.
In case of a corresponding order volume, the ZEISS standard and catalogue parts will be handed over according to a separate contractual agreement.
The transferred standard and catalogue parts are not subject to the automatic modification service!
- Only standard parts from the ZEISS library are to be used. ZEISS standard part models can be requested from your personal support staff member as required.
No foreign family table instance models may be used.
- The use of the standard part models may only take place in accordance with the contractual regulations.
- Before using the ZEISS standard parts library, an instruction is given by the responsible CAD department.
- The models and part families from the ZEISS standard parts library must not be changed.

2.2 *Use of materials (RoHS conformity, list of prohibited substances in medical technology)*

- To ensure RoHS conformity and, if necessary, taking into account the "Medical Technology Prohibited Substance List", only materials listed in the ZEISS Material List must be used when selecting materials for products.
- The base materials to be used must be agreed with the customer.

Attention: When using data from ZEISS material lists, always pay attention to the correct spelling of the entries (case sensitive, spaces ...)!

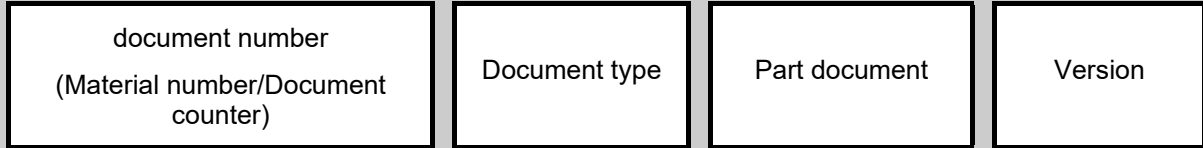
3 Inspection of Creo objects

- Every incoming model and every drawing is checked with the ModelCHECK program integrated in Creo.
- After a successful check, the objects are entered in the data management system.
- The ModelCHECK configuration files can be downloaded from <https://www.zeiss.com/cad>

4 Numbering, file names and metadata

4.1 Numbering of Creo Objects

In the PLM system, a document is identified by:



4.1.1 Document number

The **document number** is created by adding a two-digit continuous document number (document counter) to the material number. This allows several alternative models per part or assembly to be managed. The continuous number is separated from the material number by "/", resulting in a document number of the following format:

123456-1234-123/01

4.1.2 Document type

CAD relevant document types are:

Name	Meaning	Assigned to	Corresponds to Creo
FUM	Material-dependent production document	Drawings	DRW, DGM
FU3	Material-dependent production document (3D)	3D Models	PRT, ASM, LAY

4.1.3 Part document

The three-digit number of the **part document** distinguishes between documents that belong together in terms of content and have the same document number, which are always distributed together but have their own document info record.

The number of the part document is incremented as soon as a document consists of several part documents. (The counting starts with "001", the incrementing is carried out according to the order of generation or according to given rules ([see section 4.4](#)).

4.1.4 Version

The **version** identifies the sequential change states of an object.

4.2 Rule for the creation of Creo file names

For all Creo objects managed with the **SAP PLM** data management system, the file name is formed from:

- the document number (SAP), where "/" is replaced by "_".
- the number of the part document.

The version and document type are not part of the file name.

4.2.1 Example for the creation of a file name

From the SAP document number **000000-1234-123/01**
and the part document number **001**
the Creo file name will be **000000-1234-123_01-001.[Extension]**

4.3 Metadata

- The required metadata to manage the Creo files in the PLM system are maintained in the PLM system.
- If the external partner does not have access to the PLM system, the metadata must be maintained in the Creo model parameters.

Note: You can only maintain metadata in the model parameters if the Creo files have not yet been imported into the PLM system (that means, if no material master records and document info records exist in the PLM system).

4.3.1 Maintenance of PLM metadata in Creo model parameters

Table 1 on the following pages provides an overview of the model parameters.

- Only the model parameters marked with X in the first column must be maintained.
- The grayed out model parameters (not marked with an X in the first column) are not relevant for the import into the PLM system.
- The second column contains the maximum number of characters allowed for each parameter.

4.3.2 Creo Model Parameters

Table 1

		model parameters	example value	intended use	comments
		SAP_DOC	000000-1234-123/01,FU3,001,03	key (Number) of SAP Document	Complete document key (complete document number)
X	18	ZEICHNUNGSNUMMER ¹⁾	000000-1234-123/01		Document number
		DOCUMENTART	FU3		Document type
X	3	TEILDOKUMENT ¹⁾	001		Part document
X	2	VERSION ¹⁾	01		Version
		SAP_DOC_DE_BESCHREIBUNG	Teil 1	Description of SAP Document	Description german
		SAP_DOC_EN_DESCRIPTION	Part 1		Description english
X	40	BENENNUNG	Teil 1		Description in logon language when checking in
		ERSTELLUNGSDATUM	30.08.2010	creation date of the SAP document, responsible employee, Lab/Office	Date on which the document info record version was created
X	10	BEARBEITER ²⁾	Zxxxx		Responsible employee in the additional data of the document
X	3	KONSTRUKTIONSGRUPPE ²⁾	K01		Responsible Lab/Office
X	35	MATERIAL ⁴⁾	EN 10088	Base material (from linking the model with material)	Do not confuse with material master.
X	35	MATERIAL2 ⁴⁾	X14CrMoS17		
X	15	MATERIALNUMMER ¹⁾	000000-1234-123	Details of the SAP material master (from linking the model with material)	material master
X	14	PROJ_KLASS ²⁾	P12345678		project class
X	1	KOMP_CHAR ³⁾	M		Component character of the project class
X	1	KOMP_STUFE ³⁾	T		component level of the project class
X	3	KOMP_ZAEHLER ³⁾			Component counter of the project class

Table 1 (continued)

		model parameters	example value	intended use	comments
X	2	KENNWORT		Specifications for marking the model on assembly drawings. KENNWORT controls relevance for material BOM and marking on assembly drawings	Controls which data is synchronized in CZ_POS_TXT*.
		CZ_POS_TXT	1842-568		MATERIALNUMMER without leading zeros or for KT Description in logon language
		CZ_POS_TXT2	P12345678-T M		Project classification or for KT MATERIALNUMMER without leading zeros
		RFC	CI	Auxiliary parameters for the data administration in SAP	Indicator whether DIS has been created or the original has been checked in
		SAP_LAST_PARAM_UPDATE	zxxxx-201008310750		Last saving in SAP on 31.09.2010 by Zxxx
		FNH000000123412301001	000000-1234-123_01-001		File name at the last saving
		CAD_TYP	PRT		File type, type of Creo object (ASM, PRT, ...)

- 1) ZEICHNUNGSNUMMER, TEILDOKUMENT, VERSION, MATERIALNUMMER
Only maintain if the document number or material number is known.
- 2) BEARBEITER, KONSTRUKTIONSGRUPPE, PROJ_KLASS
Maintenance is to be agreed with the customer.
(Values can alternatively be assigned during import via PLM tools.)
Only permanently defined values from the PLM system are permitted.
The list of values must be agreed with the customer.
- 3) KOMP_CHAR, KOMP_STUFE, KOMP_ZAEHLER
If there are any deviations from the default values, you must maintain them.
Only permanently defined values from the PLM system are permitted.
The list of values must be agreed with the customer.
- 4) MATERIAL, MATERIAL2
Only firmly defined material designations according to the material list from the PLM system are permitted.
(At all times pay attention to the correct, predefined spelling of the entries such as upper/lower case, spaces, ...)

4.4 Numbering and naming of special models

Object type	Numbering, value for parameter „BENENNUNG“	specifics
auxiliary models		No link to a material master record
part framework for assemblies	BENENNUNG: „Framework“ part document 00x of assembly document number	Count up part document number
Skeleton Model	BENENNUNG: „Skeleton“ part document 00x of assembly document number	parameter KENNWORT = „HT“
Merge Group (Usage not necessary for external merge)	BENENNUNG: „MERGE“ part document 00x of finished part (part document number to be agreed)	
Familytable	For instances count up part document number.	Instancename according to ZEISS conventions for filename creation, Example: 123456-1234-123_01-001
Flattened sheetmetal model (familytable instance)	part document 002 of bended sheetmetal model; Additional word „flattened“ enter in BENENNUNG KENNWORT = „HT“	Instances of part families are given their Instance names by the PLM system according to the rules for normal single part models.
Model of purchased parts	KENNWORT = „KT“ BENENNUNG = Order description, order text from SAP material master	
Purchase parts with fixed Geometrie	Derive document number from the material number of the SAP material master	
Purchase parts without I-Number (not yet released)	Part document 00x of assembly in which the model is used for the first time. The material number is added when the purchased part is released; the document number can be retained.	
Model geometry adapted to assembly installation conditions (flexible model) If possible, use flexible components instead!	Part document 00x of assembly in which this adapted model is installed (Own copy of the catalog part model for each assembly)	

Object type	Numbering, value for parameter „BENENNUNG“	specifics
<p>standard part with a model geometry adapted to the assembly installation conditions (flexible model) If possible, use flexible components instead!</p>	<p>Copy of the standard part as part document 00x to the assembly in which the model is installed. MATERIALNUMMER = material number of standard part KENNWORT: replace „NT“ with „KT“!</p>	
<p>Envelope that replaces a component in a simplified representation, or replacement model for an assembly</p>	<p>Document /02 for the assembly to be replaced. BENENNUNG: „Envelope“ or additional word „replacement model“</p>	<p>MATERIALNUMMER = material number of the represented component or assembly.</p>
<p>Envelope that replaces several components in a simplified representation.</p>	<p>Partial document 00x for the assembly in which the components are replaced. BENENNUNG: „Envelope“</p>	<p>Without material number and without material link (cannot be labelled automatically with item texts on the assembly drawing)</p>
<p>Models for NC Programming</p> <p>Model copy for definition NC machining</p> <p>Work piece for NC machining</p> <p>Fixtures</p> <p>NC machining</p> <p>NC Machining assembly</p>	<p>Part document 001 BENENNUNG: NC Model / ...</p> <p>Part document 002 BENENNUNG: Work piece / NC machining</p> <p>Part document 003 ... 005</p> <p>Part document 006 BENENNUNG: NC machining</p> <p>Part document 007 BENENNUNG: NC machining</p>	<p>Document type NC3</p> <p>Counting up the document number for the material; 123456-1234-123/0x</p>
<p>Accessory parts, which are to be marked separately on the assembly drawing</p>	<p>Material number as in SAP Set CAX subtype to "KT" Enter "Accessory" in the Description field If a model with a different description already exists for the accessory (for other uses), then create a new document for the material: Counting up the document number for the material; 123456-1234-123/0x</p>	<p>A special Pro/Report table for accessory parts is available for assembly drawings.</p>