

**Guideline (not under Configuration Control)**

## **CAD Manual 04-2 CAD Data Structuring**

This document describes the CAD data structuring processes in the IO DO.

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<i>Read Access</i>	<b>GG: MAC Members and Experts, GG: STAC Members &amp; Experts, AD: ITER, AD: External Collaborators, AD: IO_Director-General, AD: EMAB, AD: Auditors, AD: ITER Management Assessor, project administrator, RO, LG: DO Management, AD: OBS - Design Office Division (DO), AD: OBS - Design Office Division (DO) - E...</b>		

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## 4.2 CAD Data Structuring

### 4.2.1 Purpose

The purpose of this section of the CAD Manual is to describe the CAD data structuring processes and methodologies that are common for mechanical and plant design and for ENOVIA based or file based design.

### 4.2.2 Scope

This section of the CAD manual describes the general rules for CAD data structuring to be applied by the designers in the IO-DO, DA-DO and Suppliers DO using the ITER CAD platform.

Complementary to this CAD manual are the PBS specific CAD Design Handbooks.

Refer to chapter 4.2.5 PBS CAD Design Handbooks for more information.

The handbooks can be found here:

[CAD Design Handbooks \(3URXHL\)](#)



**This manual contains hyperlinks to IDM documents that need to be launched in order to get the full picture of the topics covered.**

### 4.2.3 Definitions

For a complete list of ITER abbreviations see:



[ITER Abbreviations \(2MU6W5\)](#)

and [DO Abbreviations \(24844F\)](#)


Abbreviations used in this document:


CAD	=	Computer Aided Design
CCP	=	Copy/Cut/Paste
CGR	=	CATIA Graphical Representation
CM	=	Configuration Model
CMD	=	Configuration Model Detailed
CMM	=	Configuration Model Management
CMS	=	Configuration Model Skeleton
CSKE	=	Component SKEleton
CV5	=	CATIA V5
DM	=	Detailed Model
DMU	=	Digital Mock-Up
EV5	=	Enovia V5
FB	=	File Based
IDM	=	ITER Document Management system
IO	=	ITER Organization
LCA	=	Life-Cycle Application (ENOVIA)
PBS	=	Plant Breakdown Structure
PRC	=	Product Root Class
RDS	=	Rule Data Structuring

RE = Responsible Engineer  
RO = Responsible Officer  
RSKE = Reference SKEleton  
SKE = SKEleton  
WP = Work Package


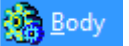
#### 4.2.4 Quick Reference Guide to rules


The following table provides an overview of the rules contained in this section of the CAD Manual.


Categories: M = Mandatory (Shall comply) R = Recommended (Should comply)						
Topic	Rule No. 	Rule Description	Category	Q-Checker Rule	EV5/FB	CAD Manual Section
General	<b><u>RDS01</u></b>	Consult the complementary <u>CAD Design Handbooks (3URXHL)</u> for PBS specificities.	M		EV5/FB	<b>4.2.5</b>
Writing and naming conventions	<b><u>RDS02</u></b>	The language used by ITER is English.	M		EV5/FB	<b>4.2.6.1</b>
	<b><u>RDS03</u></b>	The part name used should clearly and concisely describe the part.	M		EV5/FB	<b>4.2.6.1</b>
	<b><u>RDS04</u></b>	Part names are limited to 35 characters.	M	IO_PART_02, IO_Pro_401	EV5/FB	<b>4.2.6.1</b>
	<b><u>RDS05</u></b>	The first preference for the naming of objects in CATIA and ENOVIA is to use the full names – full words and no abbreviations. GRAVITY SUPPORT instead of GS.	M		EV5/FB	<b>4.2.6.1</b>
	<b><u>RDS06</u></b>	The second preference for the naming of objects in CATIA and ENOVIA is to use a combination of full words and abbreviations.	M		EV5/FB	<b>4.2.6.1</b>
	<b><u>RDS07</u></b>	Only abbreviations included in the ITER abbreviations list are permitted.	M		EV5/FB	<b>4.2.6.1</b>
	<b><u>RDS08</u></b>	All part design bodies and open bodies must have speaking names. FLANGE instead of Body.21.	M		EV5/FB	<b>4.2.6.1</b>
	<b><u>RDS09</u></b>	Important geometrical elements, features and sets should have speaking names.	M		EV5/FB	<b>4.2.6.1</b>
	<b><u>RDS10</u></b>	Published element names shall start with 3D, 2D or letters. Length shall be between 4 and 60 characters. Letters, underscore, space, parenthesis, number, dot, minus, + or = symbols are allowed.	M	IO_PUB_1	EV5/FB	<b>4.2.6.1</b>
	<b><u>RDS11</u></b>	Forbidden publication name: “Plane.X”, etc. Publication names generated by E&S modules of CATIA are kept unchanged.	M	IO_PUB_1	EV5/FB	<b>4.2.6.1</b>

Categories: M = Mandatory (Shall comply) R = Recommended (Should comply)						
Topic	Rule No. 	Rule Description	Category	Q-Checker Rule	EV5/FB	CAD Manual Section
Writing and naming conventions	<b><u>RDS12</u></b>	The following characters ONLY are permitted to be used: Upper A to Z, a – z. Numbers 0 to 9, Dot ., Equal =, Minus sign –, Plus sign +, Underscore _, Blank. It is not permitted to use national accented characters or characters other than the Roman alphabet. (E.g. Cyrillic, Kanji, Hangul etc.) Except for supplier manufacturing drawings in specific cases.	M	IO_PART_02,03, IO_Pro_401	EV5/FB	<b>4.2.6.2</b>
	<b><u>RDS13</u></b>	Do not use forbidden colors (Red, orange, khaki green)	M	IO_COL		<b>4.2.7</b>
Forbidden functions	<b><u>RDS14</u></b>	Inside a Work package (WP) the following functions are forbidden in ASSEMBLY DESIGN: ASSEMBLY FEATURES (except for STEEL STRUCTURE DESIGN and REUSE PATTERN) SYMMETRY, COMPONENTS/NEW COMPONENT (except for CGR files) WELD DESIGN Workbench.	M		EV5/FB	<b>4.2.8</b>
	<b><u>RDS15</u></b>	For structure exposed assemblies all kind of applications and assembly features are forbidden.	M	IO_Pro_404	EV5	<b>4.2.8</b>
	<b><u>RDS16</u></b>	CATPart, CATProduct, and CATDrawing must be updated.	M	IO_Pro_403	EV5/FB	<b>Table 4.2-2</b>
General modelling rules	<b><u>RDS17</u></b>	No broken links inside assemblies, to skeleton or from drawing views to 3D.	M	IO_Pro_405	EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS18</u></b>	No context/import links (except Secondary Structure, Flexibles in Conduit design and Electrical Harness).	M		EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS19</u></b>	For a 3D EV5 document, the maximum file size should be 80Mb.	R	IO_MAX_Maximum Document File Size	EV5/FB	<b>4.2.9</b>
	<b><u>RDS20</u></b>	Publication name shall match published element name.	M	IO_PUB_2	EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS21</u></b>	Publication must be synchronized and resolved.	M	IO_PUB_4	EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS22</u></b>	Publications shall follow naming & writing conventions.	M	IO_PUB_1	EV5/FB	<b>4.2.6</b>
	<b><u>RDS23</u></b>	Part version & document revision only 3 characters: dash or letter.	M	IO_REV	EV5/FB	
	<b><u>RDS24</u></b>	Elements shall follow naming & writing conventions.	M	IO_PART_003	EV5/FB	<b>4.2.6.2</b>



Categories: M = Mandatory (Shall comply) R = Recommended (Should comply)						
Topic	Rule No. 	Rule Description	Category	Q-Checker Rule	EV5/FB	CAD Manual Section
General modelling rules	<b><u>RDS25</u></b>	If an alternative design in parallel to the reference design, a variant (slightly different part), or a part based on a parametric part from the ITER catalogue have to be made, the specific methodology described in <u>ITER_D_33CPWB - How To Create New Part Filebase Using Existing Ones Extracted From ENOVIA</u> shall be used.	M		FB	<b>4.2.20</b>
Sketcher	<b><u>RDS26</u></b>	Sketch should be fully constrained for normal part.	R		EV5/FB	<b>4.2.11</b>
	<b><u>RDS27</u></b>	For skeleton part sketch shall be fully constrained.	M	IO_SKE_201	EV5/FB	<b>4.2.11</b>
	<b><u>RDS28</u></b>	No contour directly (implicit) linked to sketch origin, use constraints instead.	R		EV5/FB	<b>4.2.11</b>
	<b><u>RDS29</u></b>	Minimize the number of entities inside a sketch.	M			<b>4.2.11</b>
	<b><u>RDS30</u></b>	Sketches must be created without superposition & discontinuity (use Sketch analysis).	M		EV5/FB	<b>4.2.11</b>
	<b><u>RDS31</u></b>	Do not apply colours on contour inside a sketch, if needed, apply colours on 3D.	M		EV5/FB	<b>4.2.11</b>
	<b><u>RDS32</u></b>	No empty sketch should remain in the CATPart.	M	IO_PART_008	EV5/FB	<b>Table 4.2-2</b>
CATIA Specification tree	<b><u>RDS33</u></b>	Do not use Hybrid Bodies for better management & sharing the same working standard.	R	IO_PART_014	EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS34</u></b>	All Bodies  have to be in Show.	M	IO_PART_005_ter	EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS35</u></b>	All Solid Features have to be in Show.	M	IO_PART_005_ter	EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS36</u></b>	All Sketches under Solid Feature shall be hidden.	M	IO_PART_005_ter	EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS37</u></b>	A maximum of 50 bodies inside a CATPart is recommended.	R	IO_PART_005_ter	EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS38</u></b>	The current active object has to be the Part Body before promotion.	M	IO_PART_004	EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS39</u></b>	No empty Body.	M	IO_PART_009	EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS40</u></b>	No unused elements remaining.	M	IO_PART_010	EV5/FB	<b>Table 4.2-2</b>

Categories: M = Mandatory (Shall comply) R = Recommended (Should comply)						
Topic	Rule No. 	Rule Description	Category	Q-Checker Rule	EV5/FB	CAD Manual Section
CATIA Specification tree	<b><u>RDS41</u></b>	No inactive or disconnected features or operations in specifications tree for 3D modelling.	M	IO_PART_007	EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS42</u></b>	No inactive features or operations in specifications tree for skeleton.	R	IO_Ske_203	EV5/FB	
	<b><u>RDS43</u></b>	No children on Dress-up features (no geometry base on Filleted, Drafted or Chamfered faces and edges).	R	IO_PART_006	EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS44</u></b>	Surface only allowed in Geometrical Set.	M	IO_PART_012	EV5/FB	<b>Table 4.2-2</b>
Assemblies management	<b><u>RDS45</u></b>	No unresolved assembly constraints.	M	IO_CST	EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS46</u></b>	Use the File+Desk function to check assembly links.	R		EV5/FB	<b>Table 4.2-2</b>
	<b><u>RDS47</u></b>	Structure the assemblies from global to detail.	R		EV5/FB	<b>4.2.10.1</b>
	<b><u>RDS48</u></b>	Part positioning from detail to global.	R		EV5/FB	<b>4.2.10.2</b>
	<b><u>RDS49</u></b>	Each part or assembly has to be ISO constrained with 0 degree of freedom <b>OR</b> the minimum of constraints has to be "Fix".	M		EV5/FB	<b>0</b>
	<b><u>RDS50</u></b>	Mechanical standard parts shall be placed with CADENAS application.	M		EV5/FB	<b>4.2.12</b>
Drawings	<b><u>RDS51</u></b>	DRW name shall follow naming convention.	M	IO_DRW_607	EV5/FB	<b>4.2.6</b>
	<b><u>RDS52</u></b>	Check title block is well filled & updated: attributes, sheet scale.	M	IO_DRW_612	EV5/FB	<b>4.2.13.8</b>
	<b><u>RDS53</u></b>	Do not isolate views.	M	IO_DRW_605	EV5/FB	
	<b><u>RDS54</u></b>	Do not lock views.	M	IO_DRW_606	EV5/FB	
	<b><u>RDS55</u></b>	Do not create or use fake dimensions.	M	IO_DRW_608	EV5/FB	<b>4.2.13.12</b>
	<b><u>RDS56</u></b>	MODIFICATION_SHEET must exist.	M	IO_DRW_611	EV5/FB	<b>4.2.13.7</b>
	<b><u>RDS57</u></b>	Sheet format must be respected.	M	IO_DRW_610	EV5/FB	<b>4.2.13.6</b>
	<b><u>RDS58</u></b>	Sheet name shall follow naming convention	M	IO_DRW_601	EV5/FB	<b>4.2.13.7</b>
	<b><u>RDS59</u></b>	Keep the best associativity between DRW & 3D with "Overload properties" or "Scene" function.	M		EV5/FB	
	<b><u>RDS60</u></b>	Generate as few manual 2D entities with interactive drafting (line etc.) as possible.	R		EV5/FB	
	<b><u>RDS61</u></b>	The Drawing must be linked to the correct 3D document, 1 CATProduct or 1 CATPart (Edit/link).	M		EV5/FB	<b>4.2.13.9</b>

Categories: M = Mandatory (Shall comply) R = Recommended (Should comply)						
Topic	Rule No. 	Rule Description	Category	Q-Checker Rule	EV5/FB	CAD Manual Section
Drawings	<b><u>RDS62</u></b>	View name must Start with letter It should use letters, number, +, -, =, parenthesis, bracket, dot, underscore & space.	M	IO_DRW_602	EV5/FB	
	<b><u>RDS63</u></b>	Only one modification sheet allowed.	M	IO_DRW_603	EV5/FB	4.2.13.7
	<b><u>RDS64</u></b>	Views without link are allowed for modification sheet or comment.		IO_DRW_604	EV5/FB	4.2.13.10
	<b><u>RDS65</u></b>	Views must be updated.	M		EV5/FB	
	<b><u>RDS66</u></b>	10 drawing sheets maximum (max size: 60Mb).	R	IO_MAX_Maximum CATDRW File Size	EV5/FB	
Properties / Attributes	<b><u>RDS67</u></b>	Fill all relevant properties/attributes.	M	IO_PRO_2	EV5/FB	4.2.14
	<b><u>RDS68</u></b>	In CATIA use only the ITER properties panel to fill the properties/attributes.	M		EV5/FB	4.2.14
	<b><u>RDS69</u></b>	Description of modification must be filled.	M		EV5/FB	4.2.144.2.14
	<b><u>RDS70</u></b>	Link to “Description Document”.	M		EV5/FB	4.2.14
	<b><u>RDS71</u></b>	Link to “Bill of Material”.	M		EV5/FB	4.2.14
Quality	<b><u>RDS72</u></b>	The default representation of design data is non-isolated elements with full feature history.	M	IO_PART_15	EV5/FB	4.2.154.2.16
	<b><u>RDS73</u></b>	Isolated solid or CGR representation can be used for special purpose like IP-protection, simplification and simulation result.	M		EV5/FB	4.2.15.2
	<b><u>RDS74</u></b>	Perform regularly light check.	M	IO_PART_005_bis	EV5/FB	4.2.16
	<b><u>RDS75</u></b>	Perform clash analysis.	M		EV5/FB	4.2.16.1
	<b><u>RDS76</u></b>	Use check list and/or Q-Checker.	M		EV5/FB	4.2.16.3
	<b><u>RDS77</u></b>	The CATDUA V5 utility must be performed without errors.	M	IO_DUA_CATDUA V5	EV5/FB	4.2.16.5
	<b><u>RDS78</u></b>	The CATIA Part number and the filename of new CV5 files must be identical.	M		FB	4.2.20
	<b><u>RDS79</u></b>	If the CATIA Part number and the filename of <b>new</b> CV5 files are changed all kind of CV5 links must be maintained.	M		FB	4.2.20
	<b><u>RDS80</u></b>	For the same component (for example a standard part) on different location and usage (instances) only one reference including CATIA unique identifier shall be used.	M		FB	4.2.20


Categories: M = Mandatory (Shall comply) R = Recommended (Should comply)						
Topic	Rule No. 	Rule Description	Category	Q-Checker Rule	EV5/FB	CAD Manual Section
	<b><u>RDS81</u></b>	If standard parts are received from the IO and an additional instance is needed the same part must be used.	M		FB	<b>4.2.20</b>
Quality	<b><u>RDS82</u></b>	Concerning plant design using Equipment and systems workbenches, data delivered shall only use catalogue items and settings from the Equipment and systems supplier package.	M		FB	<b>4.2.20</b>
	<b><u>RDS83</u></b>	In case of missing Equipment and systems standard items, supplier will have to make a catalogue request ticket to get in contact with ITER catalogue team.	M		FB	<b>4.2.20</b>
	<b><u>RDS84</u></b>	All the data file-based should be located in the same folder, check this data management before sending.	M		FB	<b>4.2.20</b>
Design work performed by Suppliers using CV5 for File based work	<b><u>RDS85</u></b>	Use always the relevant (normally the most recent) set of data received from ITER.	M		FB	<b>4.2.20</b>
	<b><u>RDS86</u></b>	The filename and the CV5 part number of a document CATPart, CATProduct, CATDrawing <b>received from the IO must not be modified.</b>	M		FB	<b>4.2.20</b>
	<b><u>RDS87</u></b>	If you have sent new data to ITER, do not continue to work on this set of data in parallel. You have to wait for a new dataset from ITER containing ITER ID etc. Further changes have to be made with this new set of data received from ITER.	M		FB	<b>4.2.20</b>
	<b><u>RDS88</u></b>	<p><i>Only the following links are allowed between CAD documents:</i></p> <ul style="list-style-type: none"> <li>- CATProduct &amp; CATpart</li> <li>- CATProduct &amp; CATProduct</li> <li>- CCP links &amp; CATPart to SKE and symmetrical part</li> <li>- CATDrawing &amp; CATPart/CATProduct</li> </ul> <p><i>ITER IO needs to synchronize links between:</i></p> <ul style="list-style-type: none"> <li>- CATIA documents and Catalogues</li> <li>- CATIA documents and material libraries.</li> </ul>	M		FB	<b>4.2.20</b>
	<b><u>RDS89</u></b>	For drawings, The ITER tool FTB manager has to be used to generate for each sheet a drawing frame and an ITER title block with default values. The proper values will be filled in after saving the drawings in Enovia. For new drawings the CV5 properties shall be modified using the ITER properties panel. For manufacturing drawings the supplier title block can be used.	M		FB	<b>4.2.22</b>

Table 4.2-1 Quick reference guide to rules

#### 4.2.5 *PBS CAD Design Handbooks*

The CAD Design Handbooks contain processes and methodologies specific to a PBS. **RDS01**  
They are useful as guidance to new designers and include the information on the following:

- Global design process, design status, design goal
- Structure PRC, assembly structure, instances
- Organizations involved in the design
- Positioning
- Skeleton
- Interfaces
- ITER methodologies
- Other specific methodologies
- Drawings

Here is the link to the IDM folder for the CAD Design Handbooks:

[CAD Design Handbooks \(3URXH\)](#)

#### 4.2.6 *Naming and Writing conventions*

##### **RDS22, RDS24, RDS51**

##### ***4.2.6.1 Naming conventions***

Care must be used in the correct spelling of a part name, as a spelling mistake will cause difficulties in searching for the part by its name

- The language used by ITER is English. The naming and spelling used should comply with this rule. **RDS02**
- The part name used should clearly and concisely describe the part. **RDS03**
- Part names are limited to 35 characters. **RDS04**
- The first preference for the naming of objects in CATIA and ENOVIA is to use the full names – full words and no abbreviations. **RDS05**
- The second preference for the naming of objects in CATIA and ENOVIA is to use a combination of full words and abbreviations. **RDS06**
- Only abbreviations included in the ITER abbreviations list are permitted. **RDS07**. For the current list see [DO Abbreviations \(24844F\)](#).
- For a new abbreviation request, create a ticket “CAD-Manual” category.
- All part design bodies must have speaking names. **RDS08**
- Important geometrical elements, features and sets should have speaking names. **RDS09**
- Published element names shall start with: 3D, 2D or letters. **RDS10**
  - ✓ Length shall be between: 4 and 60 characters.
  - ✓ Allowed: Letters, underscore, space, ( ), number, point, minus, + or =.
  - ✗ Forbidden: “Plane.X”, “Face.X”, “Line.X”, “Extract.X”, “Sketch.X”. **RDS11**

##### ***4.2.6.2 Writing conventions***

The characters used must be compatible with all computer operating systems e.g. Windows. With this in mind the following rules shall be followed: **RDS12**

- Permitted characters:
  - ✓ Upper and lower case letters A to Z, a to z

- ✓ Numbers 0 to 9
- ✓ Dot .
- ✓ Equal =
- ✓ Minus sign –
- ✓ Plus sign +
- ✓ Underscore \_
- ✓ Blank space
- ✓ The part or object name must start with a letter.

- Forbidden characters:
  - ✗ It is not permitted to use national accented characters
  - ✗ Characters different to the Roman alphabet. (e.g. Cyrillic, Kanji, Hangul etc.).  
Exception for Supplier manufacturing drawings
  - ✗ Forward slash / and back slash \ for element names inside the CATIA files.

#### 4.2.7 *Forbidden colours*

Following colours are forbidden: **RDS13**

- ✗ Red (220,0,0 – 255,0,0)  
It is the default colour of an update request
- ✗ Orange (220,120,0 – 255,170,0)  
It is the default colour of selected or highlighted elements
- ✗ Khaki (190,180,110 – 220,210,130)  
It is used for other CATIA functionalities as dimmed 3D geometry

#### 4.2.8 *Forbidden functions*

The following CATIA functions are forbidden. ITER uses different methodologies and/or ENOVIA cannot save the entities created by these functions.

- A. Inside a Work package (WP) the following functions are forbidden **RDS14**:
  - ✗ ASSEMBLY DESIGN + ASSEMBLY FEATURES details (except for STEEL STRUCTURE DESIGN and REUSE PATTERN)
  - ✗ ASSEMBLY DESIGN + SYMMETRY
  - ✗ COMPONENTS+NEW COMPONENT (except for CGR files)
  - ✗ WELD DESIGN
- B. For structure exposed assemblies:
  - ✗ All kinds of application and assembly features are forbidden. **RDS15**

#### 4.2.9 *Rationale for general modelling rules*


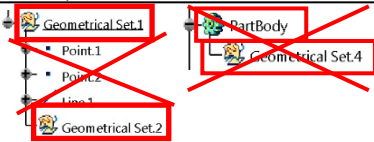
##### **Rationale:**

**Robustness:** CATIA is a parametric CAD system which keeps the history of the design. Therefore link chain and feature tree should be made robust to avoid update errors, software errors and minimize the effect of changes to other than the changed geometry.


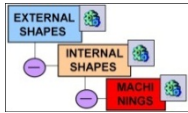
**Usability:** To design and change the geometry in an efficient way, the designer must be able to easily understand the design intent, the link chain and identify relevant elements.

Flexibility: During the design process many changes have to be implemented. High flexibility allows easier and more efficient changes and reduces the impact on other geometry.

Performance: File size shall be reasonable (maximum 80Mb) to avoid long loading, update time and slowing down database application. **RDS19**

Topic	Description	Rationale			
		Robustness	Usability	Flexibility	Performance
General	Use wireframe elements like point, line, plane instead of solid edges, vertices, faces (to avoid unstable links between geometries).	X			
	Delete unused elements, elements without child elements. Particularity for SKE: unselect all publications in the delete useless elements panel.		X		X
	Hide auxiliary geometry		X		X
	Links between CATParts shall only be built with CCP = CATIA Copy Paste methodology not as CIP = Context Import Links. <b><u>RDS18</u></b> This type of Link is forbidden in mechanical and they are also forbidden in Equipment and Systems except in three specific cases and under certain conditions. <ul style="list-style-type: none"> <li>No CIP links between WP allowed.</li> <li>CIP links allowed <b>inside</b> a Work Package (WP) only for three exceptions : <ul style="list-style-type: none"> <li>Secondary structure.</li> <li>Flexibles made in Conduit design workbench.</li> <li>Harness design.</li> </ul> </li> </ul> In these three cases CIP are allowed due to software usage. These links are automatically created when inserting a secondary structure or routing a flexible cable in conduit design or electrical cable in harness.	X	X		
	Links between CATParts shall be based only on published elements.	X	X		
	Avoid “create datum” function (isolated elements) 	X	X	X	
	Do not use ordered geometrical sets → 		X	X	
	CATPart, CATProduct and CATDrawing must be updated. <b><u>RDS16</u></b>	X	X	X	X
	No broken link inside assemblies, to skeleton or from drawing views to 3D. <b><u>RDS17</u></b>	X	X	X	
	Description instance attribute correctly filled.		X	X	
General	For an EV5 document, the maximum file size should be 80Mb. <b><u>RDS19</u></b>		X		X
	Publication name must match published element name. <b><u>RDS20</u></b>		X	X	
	Publication must be synchronized and resolved. <b><u>RDS21</u></b>	X	X	X	X
	Respect writing convention for publications.		X	X	



Topic	Description	Rationale			
		Robustness	Usability	Flexibility	Performance
Sketch	Respect writing convention for element under CATPart.		X	X	
	No contour directly linked to sketch origin, use constraints instead.		X	X	
	No colour on sketch contour, apply colour in 3D.		X	X	
	No superposition or discontinuity inside one sketch (Sketch analysis).		X	X	
	Sketches have to be fully constrained for Skeleton.	X	X		
	Sketches should be fully constrained for normal part.	X		X	X
	No empty sketch should remain in the CATPart. <b>RDS32</b>		X		X
Part design	No dead solid (except for IP reason or catalogue parts).		X	X	
	For changes use original sketch or feature definition, not for example thickness or fill up with other feature.	X	X	X	X
	No empty Partbody shall remain in CATPart. <b>RDS39</b>		X		
	Result Partbody shall be the in work object. <b>RDS38</b>		X		
	Do not use Multipad feature 	X	X		X
	Do not use Hybrid bodies in part design. <b>RDS33</b> Hybrid bodies are prevented by ITER specific CATSettings, which are part of the ITER environment or supplier package, because as a result the tree structure is not clear and more difficult to handle. If a DA supplier has created parts with hybrid bodies, the DA takes responsibility to support. IO will not provide support on this data.		X	X	
	Do not deactivate branches, features or operation in specification tree. <b>RDS41</b>		X		X
	Draft, shell, fillet & chamfer shall not be integrated in sketches.		X	X	X
	Create dress-up operations like draft, fillet & chamfer as soon as possible on the primitive feature in the specification tree.	X	X	X	X
	Create draft first then fillet.	X	X	X	X
	Make fillets from largest radius values to smallest.	X	X		
	Maximum number of edges per fillet 10.	X	X	X	
	Mirror or pattern operations on specific feature in a specific body together with the affected feature.	X	X	X	
	Use hole with thread instead of groove for thread representation.	X	X		X
	For pattern of holes use pattern feature to be able to reuse in assembly design for instantiation.		X	X	
	For complex Mechanical Parts, use Boolean operation to structure geometry. 	X	X	X	X
	All Bodies have to be in Show. <b>RDS34</b>		X	X	






Topic	Description	Rationale			
		Robustness	Usability	Flexibility	Performance
	All Solid Features have to be in Show. <b>RDS35</b>		X	X	
Part design	All Sketches under Solid Feature shall be hidden. <b>RDS36</b>		X	X	
	A maximum of 50 Bodies inside a CATPart is allowed. <b>RDS37</b>		X	X	X
	No unused elements remaining. <b>RDS40</b>		X	X	X
	No children on dress-up features (no geometry based on faces which are result of fillet, draft or chamfer features). <b>RDS43</b>	X			
	No Geometrical Set  Geometrical Set... inside Bodies  Body		X	X	
	Surface only allowed in Geometrical Set  Geometrical Set... <b>RDS44</b>		X	X	
	Use only points, lines and planes instead of edges, faces and vertices for assembly constraints.	X	X	X	
	All parts must be positioned or protected in position by assembly constraints. At least a fix constraint must exist for each part.	X	X		
	No unresolved assembly constraints. <b>RDS45</b>	X	X	X	X
	Use the Desk function to check assembly links. <b>RDS46</b>		X		
	Structure the assemblies from global to detail.		X		
	Part positioning from detail to global.		X	X	

Table 4.2-2 Rationale for modelling rules

#### 4.2.10 Assemblies management

##### 4.2.10.1 Structure the assemblies from global to detail

To allow a better organization of the product structure (see Figure 4.2-1), we start from the PRC (Product Root Class), and develop from the PBS first level (Product Breakdown Structure) down to the leaf instance. This method allows a better visibility and understanding of the system structure. The part positioning starts from detail to global (see next paragraph). In addition the good level of assembly has to be defined, to inherit the position matrix to subassemblies. For example in Vacuum Vessel system one complete sector is rotated around the z-axis of the tokamak. All subassemblies and components inside the sector will inherit this position as long as no other position matrix is defined on lower level. **RDS47**.

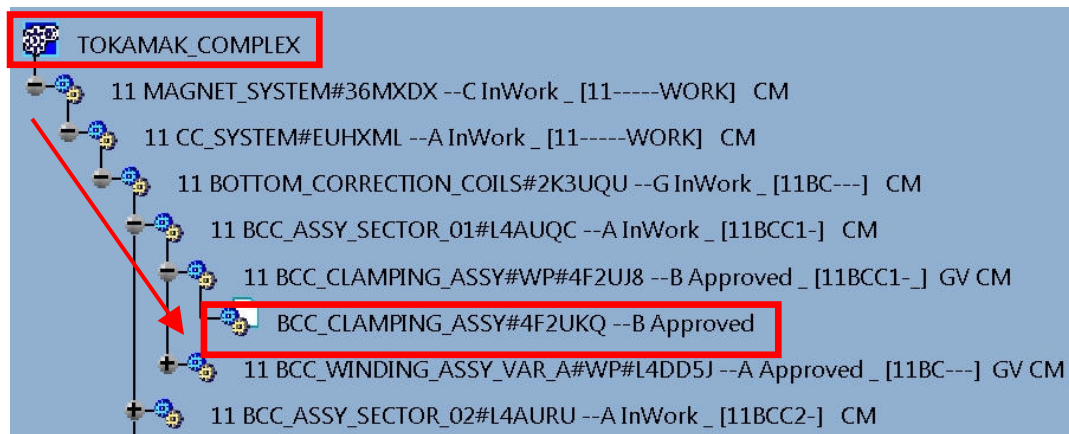


Figure 4.2-1 Product structure created from global to detail

#### 4.2.10.2 Part positioning from detail to global

For a better control of part positions, it is strongly recommended to drive the positioning from the lowest level to the main nodes, in the aim to drive all intermediate matrix of positioning (case for relative positions between parts/assemblies). **RDS48**

##### Example:

- 1- We start by positioning the bolt, washer and nuts, relative to each other.
- 2- Then this sub-assembly is positioned relative to the gravity support.
- 3- And the gravity support sub-assembly is positioned with cryostat main base.

This method ensures that all matrix of position are properly defined, and keeps the control for any future modifications.

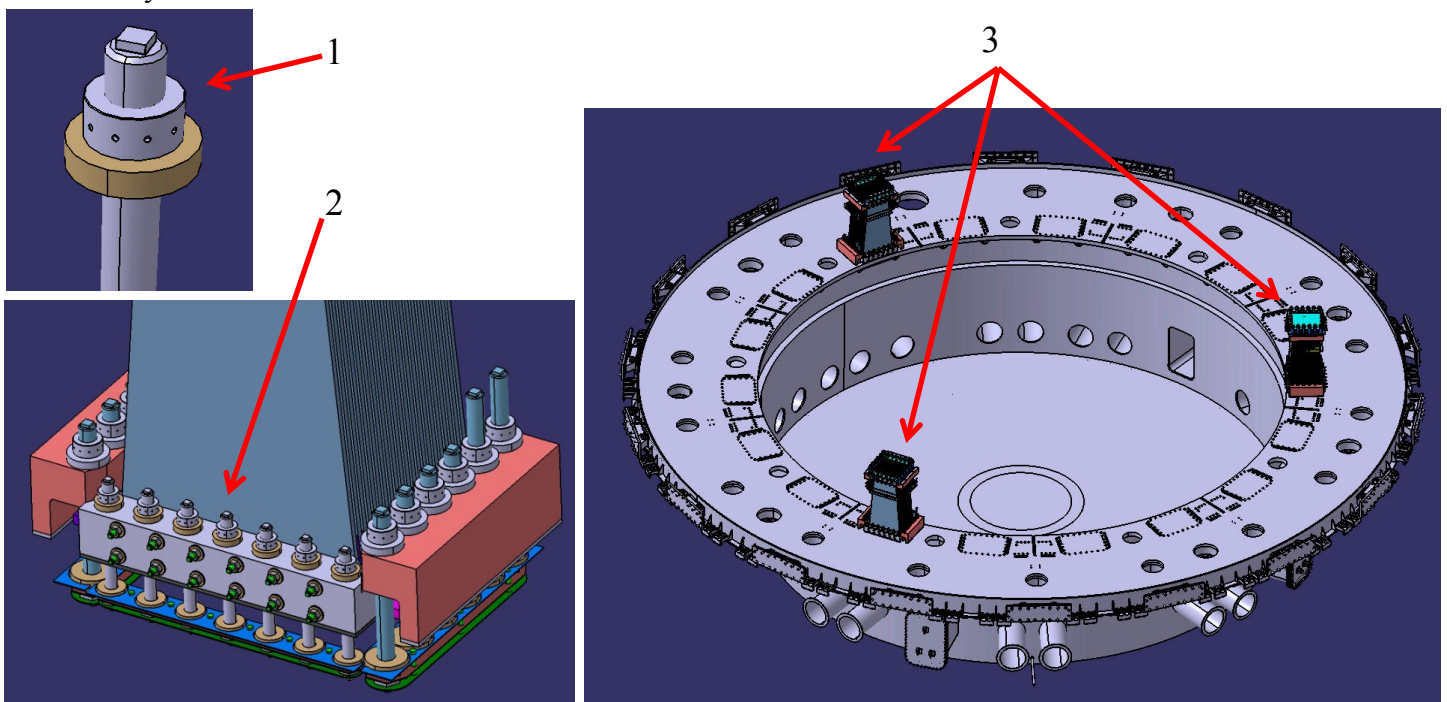
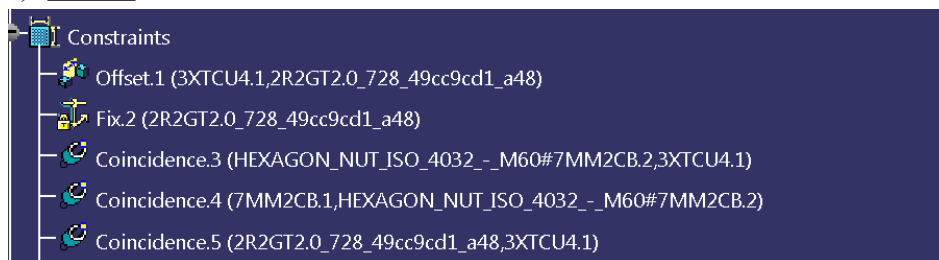


Figure 4.2-2 Example for definition of positioning matrix, from detail to global

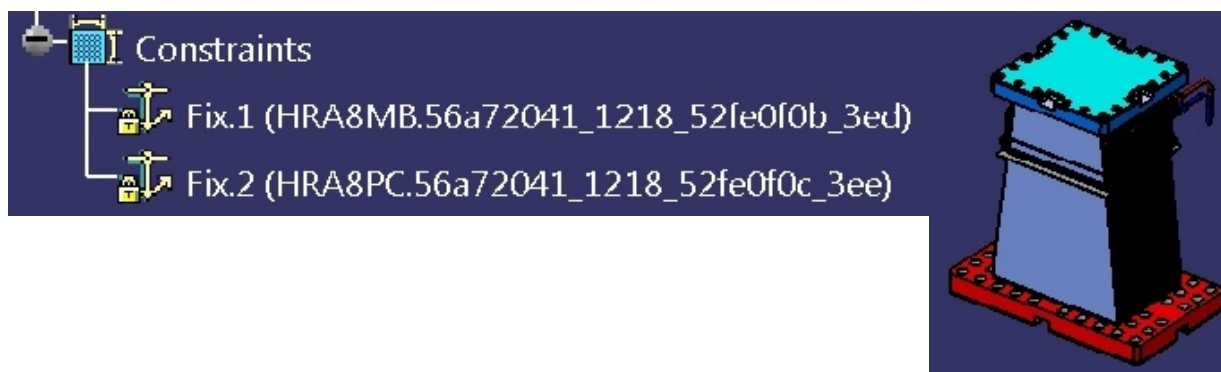
#### 4.2.10.3 Each part or assembly has to be ISO constrained with 0 degree of freedom

Each 3D part should be properly defined in position with assembly constraints, to remove all degrees of freedom, and to be able to simulate physical behaviour of components (kinematic for example). **RDS49**



**Figure 4.2-3 Example for definition of assembly constraints**

But, if for any reason assembly constraints are not appropriate, these parts must at least be fixed to avoid risk of wrong manipulation.



**Figure 4.2-4 Example of Fix assembly constraints**

#### 4.2.11 The Sketcher

For normal use during the design phase, sketch should be fully constrained because it allows to take into account more quickly modifications and ensure at the same time stability of the basic feature related to this sketch. **RDS26**

For skeleton sketch shall be fully constrained to allow an automatic update in the case of modifications made in the driving part. **RDS27**

#### **No contour directly linked to sketch origin, use constraints instead: RDS28**

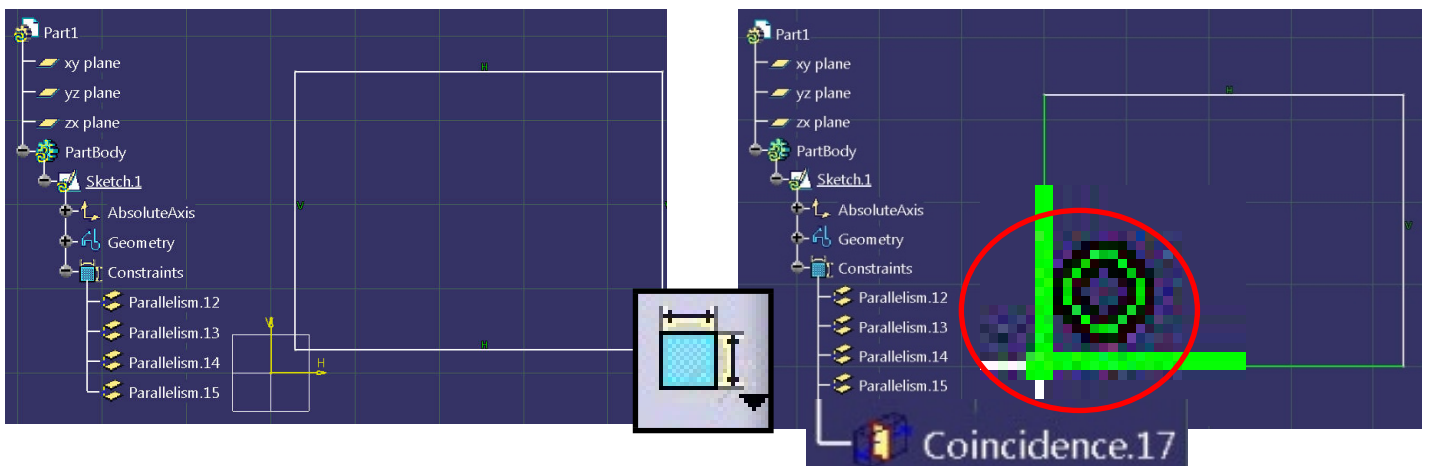
For a better control of the contour during creation or modification phases, it is strongly recommended **not to** create contour directly from the origin/attached to the origin.

In this case CATIA V5 (Release 23) will create an implicit coincidence (see picture 1 below) between the origin and the contour that we cannot remove. This implicit coincidence doesn't appear in the specification tree. **Consequence** → **The contour cannot be modified in position.**



**Figure 4.2-5 Contour in coincidence with origin with implicit constraint**

Instead of starting the contour from the sketch origin, it is better to create the contour with an offset from the sketch origin and create in a second step an explicit coincidence constraint.



**Figure 4.2-6 Contour in coincidence with origin by explicit constraint**

Consequence → After removal of the coincidence constraint, the contour can be modified in position.

### **Minimize the number of entities inside a sketch: RDS29**

In most of the case, the content of a sketch is supposed to be basic because a simple contour is easier to create, to constraint, to understand and modify if necessary.

From this basic contour we create basic 3D features (pad, pocket, shaft, groove, rib, etc ...) and this is the combination of all these features (with or without Boolean operations) which generate a complex 3D solid if necessary.

### **Sketches must be created without superposition & discontinuity (Sketch analysis): RDS30**

To ensure the best data quality and allow to CATIA to create quickly 3D basic feature, the contour inside a sketch must be created without superposition and discontinuity. The tool sketch/Analysis ensures this need.

### **Do not apply colours on contour inside a sketch, if needed, apply colours on 3D: RDS31**

The sketcher is using its own colour code to inform user during creation/modification phase (white, green, purple, grey, etc ...). It is important to not apply colours manually inside a sketch for a better understanding. If needed colours shall be applied on resulting 3D basic feature.

#### 4.2.12 *Standard Parts*

To place mechanical standard parts in an assembly “CADENAS Part Solution” shall be used.  
**RDS50**

Mechanical standard parts from CADENAS have a CADENAS ID, are in Approved status, PBS 76, with a CADENAS user and CA representation.

For more details refer to documents in the [CADENAS](#) how to folder.

Mechanical standard parts not from Part Solutions shall be replaced by Part Solution parts when anyway changes on the assembly have to be performed.

For Plant design use catalogues as described in [CAD Manual 04-5 Plant Design Processes \(33PE8P\)](#).

#### 4.2.13 *General Rules for Drawings*

##### 4.2.13.1 *Drawing Types*

We have 3 types of drawings:

Target	Modelling approach	View generation mode	Link in edit links	Collaboration	Methodology for...
Single part drawing	Multi part	Exact view	CATPart	Easy, no limitation	manufacturing drawings
Single part and assembly drawings	Multi part, (multi body for some legacy)	Exact view	CATProduct or CATPart	Easy, no limitation	Multi body: past solution to save time during conceptual design
Assembly drawing (Multi PBS, general arrangement, assembly study etc.)	Dedicated persistent CATProduct (WP) for assembly drawing with representation .DRW	Approximate view	CATProduct	Easy, no limitation	Assembly drawings

**Table 4.2-3 Summary of drawing types**

The position of drawings in the product structure of Enovia is described in:

[CAD Manual 05 - Design Data Management \(249WSM\)](#).

#### **4.2.13.2 Sheet Scale**

The sheet scale must be used for the general scale the views in a sheet instead of individual view scale values.

#### **4.2.13.3 Drafting Standard**

For drawings the ITER.xml for mechanical components or ITER\_PLANT.xml for plant design drafting standard must be used.

#### **4.2.13.4 Multiple Drawing Sheets**

A CATDrawing document contains only one drawing but can contain multiple drawing sheets. Sheet.1 is reserved for the modification sheet; no 3D views shall be created in Sheet.1.

#### **4.2.13.5 Assembly and Single Part Drawings**

Assembly and single part drawings are saved as separate CATDrawing documents to allow proper change management.

#### **4.2.13.6 Sheet format**

Sheet format must be either A0 (1189-841) or A1 (594-841) or A2 (420-594) or A3 (297-420) or A4 (210-297). **RDS57**

#### **4.2.13.7 Sheet naming convention**

Sheet name must start with letter.

Sheet name must contain between 2 and 35 characters. It should use letters, number, +, -, =, parenthesis, underscore & space. **RDS58**

Modification sheet shall be called MODIFICATION\_SHEET or Sheet.1. **RDS56**

Only one modification sheet is allowed. **RDS63**

#### **4.2.13.8 Title block content for ITER title block**

##### **RDS52**

##### **Checking:**

- DRAWING DATE must be Day (2 number) Month (3 letters starting with capital letters) Year (2 number).
- DRAWING STATUS has to be 1 Capital letter.
- DRAWING PART VERSION must be 3 characters (capital letters or dash).
- DRAWING DES has to be between 4 and seven characters.

Default values & values have to be correct for:

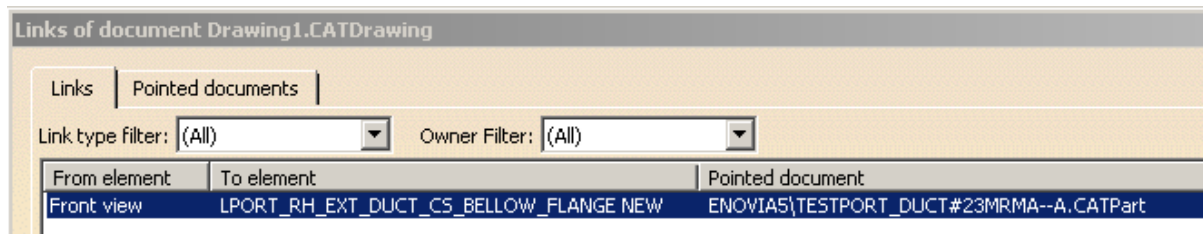
- DRAWING\_DATE
- DRAWING\_MATURITY
- DRAWING\_PART\_VERSION
- DRAWING\_DES, DRAWING\_DOS
- DRAWING\_DH, DRAWING\_RE
- DRAWING\_DIM, DRAWING\_REVISION
- DRAWING\_PART\_ID, DRAWING\_ID
- DRAWING\_PART\_PBS1\_FULL
- DRAWING\_PART\_PBS2\_FULL
- DRAWING\_PART\_MATURITY
- DRAWING\_NUMBER,
- DRAWING\_PART\_STATUS.



#### 4.2.13.9 Links of Drawings

CATIA links between CATDrawing files are not allowed. The CATIA link for single part drawing shall point to one CATPart. **RDS61**

Use the edit+link function to check.



**Figure 4.2-7 Edit links result of a CATDrawing**

The CATIA link for an assembly drawing shall always point to a persistent CATProduct. Persistent means it is or will be a WP (Work Package) in ENOVIA.

If a persistent CATProduct does not yet exist, for example for a multi PBS drawing, it shall be created for the purpose of drawing generation.

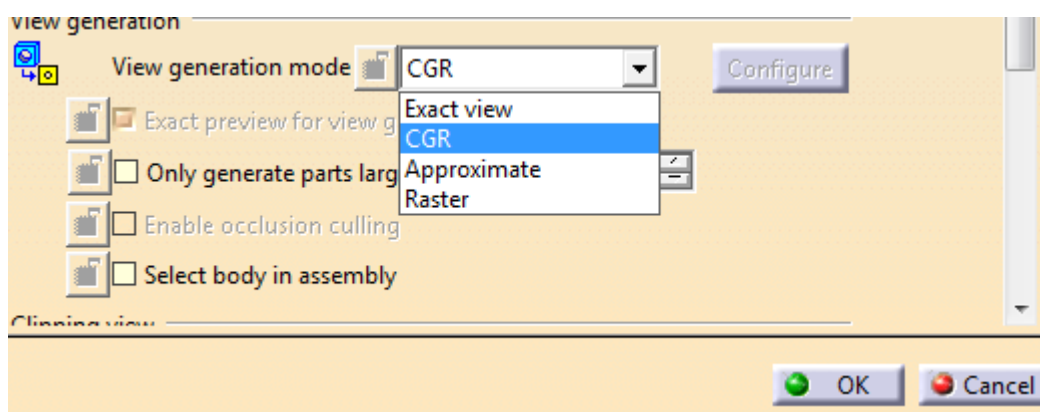
It has to be in absolute position and the representation attribute shall be DM- or CM- or AM DRW.

#### 4.2.13.10 Views without link

Views without link to model called “MODIFICATION\_SHEET” or “Sheet.1” is allowed in order to fill the Modifications Sheet & add some comment. **RDS64**

#### 4.2.13.11 View generation mode

Drawings for single components and assemblies that require a lot of dimensions shall be generated using Exact View mode.



**Figure 4.2-8 View generation mode for drawing view**

(Tools / option / Mechanical Design / Drafting/ View)

General arrangement drawings, assembly simulation drawings and other drawings with low number of dimensions should preferably be made with ‘Approximate’ mode to save time for loading of the 3D data and computation of the views.

#### **4.2.13.12      *Associativity of dimensions***

Dimensions in views made with exact mode shall be made associative.


Dimensions in views made with other modes cannot be associative.

Do not create or use fake dimensions. **RDS55**

#### **4.2.14    *Properties and attributes***

Properties are used to manage the data, for search or filtering of parts in the Enovia V5 database, for lifecycle, configuration & change management and to fill the title block of drawings. **RDS67**

- A. **It is forbidden to use standard CATIA properties** (right click menu +  ).  
**RDS68**

- B. Modify the properties in the ITER panel  . The ITER properties panel is part of the CAD supplier package and therefore can be used for ENOVIA and file-based work. **RDS69 RDS70 RDS71**



ITER Product Properties ev909 (Oct 18 2011 - 17:09:09) - (WERNERW/LCAMAN)

**Part** | Document

Part Master

Name	CGVS_ACB_CP#WP#2K25AZ	1	
Description Reference	CGVS_ACB_CP	2	
Assembly Type	WP	3	
Representation	CM	4	
Option	-	5	
External Part ID	-	6	

Item Instance

Instance ID	2K25AZ.0_2804_492193da_d62	7	
Description Instance	31GV00	8	
PBS 1	--	9	
PBS 2	--	10	
PBS 3	--	11	

Part Version

Description of modification	-	12	
Preferred	Y	13	
Description Long	-	14	
PBS 1	31	15	
PBS 2	GV	16	
PBS 3	00	17	
DEPT	-		
DIVN	-		
GROUP	-		
UNIT	-		
PARTY	-		
RO	PEARCER	18	
RE	BERSIEJ	19	
PRO	---		
Change Type	Minor	20	
Maturity	CD	21	
Export Number	-	22	

Logbook Documents Links

Description documents	-	23	
Bill of Material	-	24	

**PART section:**

Need to be checked

1. Automatic: "Description reference"#"ID"
2. Described the content & if it's part or WP
3. SE or WP (indicate in Name) or PD (part detail) Choose SE in properties panel ( )
4. **DM** (Detail model), **DMDRW** (DM Drawing), **CM** (Configuration model)
5. Inform if alternative model, DWO number or reasonable text
6. Identifier for DA or supplier
7. Identify generated by the system
8. Functional reference or identifier
9. PBS 1 for the instance (pull-down)
10. PBS 2 for the instance (pull-down)
11. PBS 3 for the instance (pull-down)
12. Description of modification performed for the current version ex. PCR number
13. Choose whether this alternative is preferred or not – allows loading of only preferred data. (pull-down). Not relevant if no alternative
14. More detailed identification of the part (optional)
15. Product breakdown structure level 1
16. Product breakdown structure level 2
17. Product breakdown structure level 3
18. Responsible Officer
19. Responsible Engineer
20. Type of change set in case of modification (minor or major change).
21. Design phase example: CD for Conceptual Design
22. Number for collaboration purpose
23. IDM ID to document with further information about modification or history
24. IDM ID of Bill of material

**Figure 4.2-9 ITER Properties panel for part and instance**

**Document Master**

Description: CGVS\_ACB\_CP  
External Document ID: \_

**Document Revision**

Description of Modification: \_  
Description Long: CGVS\_ACB\_CP

DIM: CHIOCCS  
DOM: MARTINE  
DOS: ---  
DH: ---  
GL: ---  
UL: ---  
RO: PEARCER  
RE: BERSIEJ  
PBSA: GLISSC  
DES: ABEDS

**Logbook Documents Links**

Description documents: \_  
Bill or Material: \_

Figure 4.2-10 Properties panel for 3D document

**Document Master**

Description: ITF\_COIL\_PRE-COMPRESSION\_RINGS 1 ☒  
Drawing Number: 006836 2 ☒  
External Document ID: \_ 3 ☒

**Document Revision**

Description of Modification: \_ 4 ☒  
Description Long: TF\_COIL\_PRE-COMPRESSION\_RINGS 5  
DIM: --- 6  
DOM: MARTINE 7  
DOS: VOLLMAT 8  
DH: SAVARYF 9  
GL: --- 10  
UL: --- 11  
RO: SAVARYF 12 ☒  
RE: KNASTEJ 13 ☒  
PBSA: VOLLMAT 14 ☒  
DES: ANDRADC 15 ☒

**Logbook Document Links**

Description documents: \_ 16 ☒  
Bill of Material: \_ 17 ☒

OK Apply Close

☒ Need to be checked

1. The description is automatically identical with the Part. Reference Description 3D, to be manually filled for drawing
2. Sequential number generated by the system
3. Identifier for DA or supplier
4. Description of modification performed for the current revision
5. More detailed identification of the part (optional)
6. Design Integration Manager
7. Design Office Manager
8. Design Office Manager Section? Supervisor?
9. Division Head
10. Group Leader
11. Unit Leader
12. Responsible Officer
13. Responsible Engineer
14. Design coordinator or key designer
15. Designer responsible for data. To be linked to the logon name & the P&O structure
16. IDM ID of document with further information about modification or history
17. IDM ID of bill of material

Figure 4.2-11 Properties panel for CATIA drawing

#### 4.2.15 Geometrical representation

The default CATIA geometrical representation is either based on part design feature tree with non-isolated elements for mechanical design or catalogue components and Equipment and system entities like run etc. for plant design. **RDS72**

For technical or contractual reason, other geometrical representation can be used like:

- Isolated solid
- CGR (CATIA Graphics Representation)

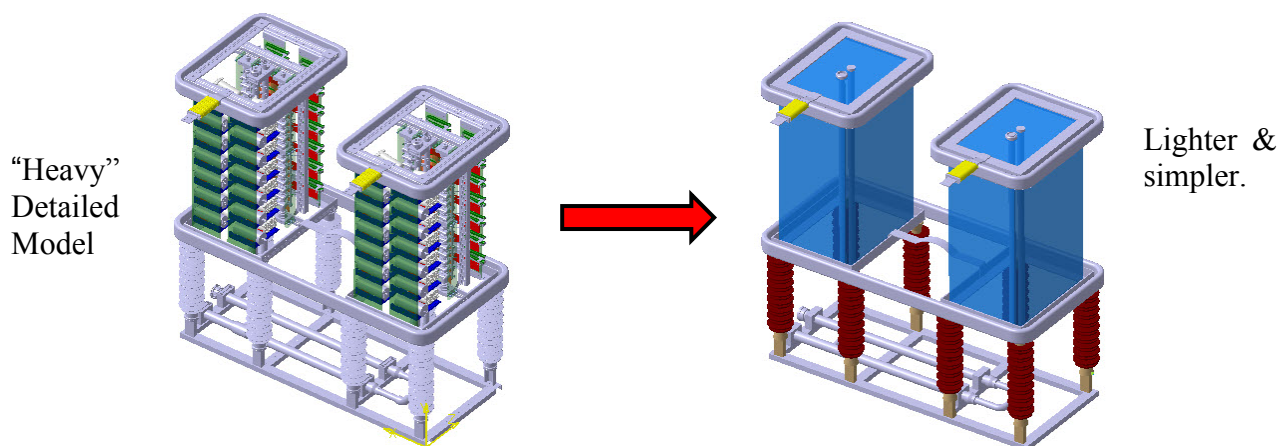
##### 4.2.15.1 Product to part function

Product to part function leads to:

- Replacement of assemblies and/or subassemblies (CATProducts) by CATParts and therefore to a simplification of the product structure.
- Bodies with isolated solid.

This approach can be used:

- To have a lighter and simpler representation as long the full CATIA feature information is not requested. An example is described in [How to integrate detailed electrical design \(4FG78D\)](#)
- To enable IP (Intellectual Properties) protection on the geometrical level because the supplier will not deliver all details of the CAD design but the at least the envelope and interfaces to allow the integration.
- For catalogue components if the possibility to implement changes is not required.



**Figure 4.2-12 Example from detailed design to lighter 3D representation**

##### 4.2.15.2 CGR

After the use of specific functionalities (ex: Kinematic, maintenance simulation, etc ...), CATIA V5 or DELMIA V5 could generate some CGR representation inside the workpackage and/or CATPart. This 3D “silhouette” could be relevant for analysis, clash detection, minimum and maximum space needed, etc... **RDS73**

Following the needs of designer this silhouette can be kept. To use CGR inside a CATProduct exceptionally CATIA component is allowed to be used.

This approach might be also used to enable IP (Intellectual Properties) protection on the geometrical level because the supplier will not deliver all details of the CAD design but the at least the envelope and interfaces to allow the integration. But for IP protection the Product to part approach is the preferred methodology.

#### 4.2.16 Design quality

Design quality means the CAD data

- Meets functional and engineering requirements
- Can be integrated in the context of all systems – clash/clearance/interfaces
- Meets CAD quality requirements

To ensure the design quality, the light check process before promotion to Draft or DA\_Draft and the full check during approval process have to be applied. **RDS74**

##### 4.2.16.1 Clash detection

During all the design phase, the designer must take care of:

- Using the valid and traceable configuration of the valid context for his design.
- Perform clash detection if necessary with clearance to ensure the nitration of his design in the context of all (static & dynamic condition, maintenance, mounting & dismounting, etc ...). **RDS75**

##### 4.2.16.2 CAD quality

It is important to maintain and improve the quality control in the Virtual Product Development by applying standards. The errors should be identified as early as possible in order to

- Improve the processes.
- Allow faster error recovery.
- Ensure that error data is not transferred in downstream processing stages.

The benefits are:

- Monitor and control the quality of the CATIA V5 data throughout the design process.
- Ensure that Methodologies and Best Practices are being followed.
- Ensure that Suppliers are receiving good quality data.
- Ensure that Suppliers are delivering good quality data.
- Promote Design Reuse.
- Enable Downstream Processes like drawing generation, simulation, calculation and manufacturing.

##### 4.2.16.3 Q-Checker

Q-Checker is an application inside CATIA to perform analysis and/or correction of CATIA CAD data with a sunset of rules defined in the CAD manual. Q-Checker: **RDS76**

- Should be used from time to time during the design work to perform corrections in an early stage.
- Shall be used before promotion during light check or before data sending in the collaboration process.
- Only the official, current IO check profiles for mechanical and plant design shall be used. [Q-Checker profile for mechanical design \(96M8CP\)](#) or [Q-Checker profile for plant design \(96NL8C\)](#).
- Check result shall be OK or ACCEPTED for final delivery.

The link below refers to a document describing the detail of Q-Checker rules checked.

[Q-Checker\\_Rules-Table-of-Profile \(3U9GFF\)](#)

##### 4.2.16.4 Check list

Some rules defined in the CAD manual cannot be checked by Q-Checker because of technical limitations. These rules have to be checked “manually”.

For this reason a complete Excel table is available (link below) and shall be used promotion or data exchange to ensure, in addition to Q-Checker the required minimum level of quality.

The check list shall also be used as a guideline if Q-Checker tool is not available.

[CAD Data Promotion Checklist \(2FBLRJ\)](#)

#### **4.2.16.5 CATDUA**

The CATIA internal cleaning tool CATDUA shall be used regularly to clean the CAD data.

#### **RDS77**

#### **4.2.17 Logbook**

The purpose of the design logbook is to trace all the actions that are performed on a reference that can be a part, an assembly and drawing to be able to retrieve the history of modifications on the CAD data.

The logbook will show actions which are automatically tracked if the data is inside ENOVIA like:

- The creation of a reference: (Part Version and Document Revision)
- Its promotion status and promotion date
- Its versioning (Version/Revision index)
- The justification of change type (Minor/Major).
- The modification of other attributes like representation, option.....

The design logbook will also show logbook specific information filled by the user. Therefore it is also relevant for file-based work:

- Description of modification
- Description document – typically IDM number with more details
- BOM – IDM number to link bill of material

The link below refers to a document describing how to use the Logbook functionality under Enovia V5, and outside with IDM for example.

[E17\\_ITER\\_EV5\\_CV5\\_DESIGN\\_LOGBOOK \(767Z87\)](#)

#### **4.2.18 Design work performed by Suppliers using CATIA V5 file-based**

For partners and subcontractors who cannot have a direct connection to an ENOVIA database (or local replication) have to work file-based.

#### **4.2.19 Introduction**

All design activity performed with CATIA V5 file-based must follow a set of mandatory rules to enable effective change management:

- To be able to identify components and drawings
- To allow integration and reintegration of data in the ITER design database ENOVIA V5
- To enable downstream applications and processes like drawing generation and analysis.
- To enable easy understanding of the design intent.

#### **4.2.20 Summary**

All rules defined in other chapters have to be applied if they are applicable.



Before starting any work the following actions shall be performed:

- ↳ The appropriate release and service pack of CATIA shall be installed.

The language environment used for CATIA shall be set to English.

- ↳ The most recent [02 ITER CAD Supplier Package \(24KWJF\)](#) and if necessary the Equipment & System related extensions shall be down loaded and installed including:
  - ✓ ITER CATSettings for mechanical and/or Equipment & Systems for plant design
  - ✓ ITER drafting standard ITER and/or ITER\_Plant
  - ✓ The ITER properties panel
  - ✓ FTB manager for drawing frame and title block
- The filename and the CV5 part number of a document CATPart, CATProduct, CATDrawing received from the IO must not be modified. **RDS86**
- If an alternative design in parallel to the reference design, a variant (slightly different part), or a part based on a parametric part from the ITER catalog have to be made, the specific methodology described in [How To Create New Part Filebase Using Existing Ones Extracted From ENOVIA \(33CPWB\)](#) shall be used. **RDS25**
- The CATIA Part number and the filename of new CV5 files must be identical. **RDS78**
- If the CATIA Part number and the filename of new CV5 files are changed all kind of CV5 links must be maintained. **RDS79**
- For the same component (for example a standard part) on different location and usage (instances) only one reference including CATIA UUID shall be used. **RDS80**
- If standard parts are received from the IO and an additional instance is needed the same part must be used. **RDS81**
- Concerning plant design using Equipment and systems workbenches, data delivered shall only use catalogue items and settings from the Equipment and systems supplier package. **RDS82**
- In case of missing Equipment and systems standard items, supplier will have to make a catalogue request ticket to get in contact with ITER catalogue team **RDS83**. [3D Catalog Request tickets \(2FA2WP\)](#). Depending on context, new items wished will be produced by supplier or IO but will in any case be publish by IO through revision of the Equipment and systems supplier package.
- For the modification of CATIA properties the ITER properties panel have to be used.
- Modification of CATIA properties with CATIA standard function is not allowed.
- The following tasks must be carried out before saving the CATIA data:
  - ↳ Unnecessary elements should be deleted especially environment data which is copied for temporary usage.
  - ↳ All the data file-based should be located in the same folder, check this data management before sending. **RDS84**
  - ↳ The following links between CATIA documents or CATIA documents and other documents are allowed **RDS88**:
    - ✓ Links between CATProduct and other CATProduct (subassemblies) or CATPart.
    - ✓ CCP links and links to parameters between CATPart - skeleton parts and driven parts, symmetrical parts.
    - ✓ Links between CATDrawing views and CATPart or CATProduct.
    - ✓ Links between CATIA documents and Catalogues synchronized with ITER IO.
    - ✓ Links between CATIA documents and material libraries synchronized with IO.
  - ↳ The following links between CATIA documents or CATIA documents and other documents are not allowed and must be isolated:

- ✗ Links between CATDrawing documents.
- ✗ Links between CATPart without skeleton approach or non-symmetrical parts.
- ✗ Links between CATIA documents and design table, local material libraries, local catalogues, knowledge ware rules etc.
- Use always the relevant (normally the most recent) set of data received from ITER.  
**RDS85**
- If you have sent new data to ITER, do not continue to work on this set of data in parallel. You have to wait for a new dataset from ITER containing ITER ID etc. Further changes have to be made with this new set of data received from ITER.  
**RDS87**

#### 4.2.21 *CATIA Properties*

For this new data the CV5 properties shall be modified using the ITER properties panel.

For more details please see chapter 4.2.14 Properties and attributes.

The ITER properties panel tool is part of the PLM package and/or the [02 ITER CAD Supplier Package \(24KWJF\)](#).

#### 4.2.22 *Drawings*

The ITER tool FTB manager has to be used to generate for each sheet a drawing frame and an ITER title block with default values. The proper values will be filled in after saving the drawings in Enovia. For new drawings the CV5 properties shall be modified using the ITER properties panel. **RDS89**

For manufacturing drawings the manufacturer's title block can be used.

For more details please see chapter 4.2.14 Properties and attributes.

The ITER properties panel tool and the FTB manager tool are part of the PLM package and/or the [02 ITER CAD Supplier Package \(24KWJF\)](#).