

# **CATIA Teamcenter Interface**

**CMI Release 10.0**

---

**Customizing Guide**

---

## Copyright

© 2000, 2011 T-Systems International GmbH.  
All rights reserved. Printed in Germany.

---

## Contact

T-Systems International GmbH  
PDC Engineering  
Fasanenweg 5  
70771 Leinfelden-Echterdingen  
Germany

<http://www.cmi-support.com>

☎ +49 (0711) 972 - 40 304

✉ +49 (0711) 972 - 95 975

mail: [cmi\\_support@t-systems.com](mailto:cmi_support@t-systems.com)

---

## Manual History

Version	Date	Version	Date
1.0	November 1996	8.5	March 2004
2.0	February 1997	8.6	September 2004
3.0	July 1997	8.7	April 2005
4.1	March 1998	8.8	September 2005
4.2	December 1998	8.9	March 2006
4.3	May 1999	9.0	October 2006
7.0	September 1999	9.1	March 2007
7.1	April 2000	9.2	October 2007
7.2	July 2000	9.3	March 2008
7.3	September 2000	9.4	October 2008
7.4	December 2000	9.5	March 2009
8.0	August 2001	9.6	October 2009
8.1	December 2001	9.7	April 2010
8.2	July 2002	9.8	October 2010
8.3	January 2003	9.9	April 2011
8.4	July 2003	10.0	October 2011

This edition obsoletes all previous editions.

---

## Trademarks

CATIA is a registered trademark of Dassault Systèmes.

Metaphase is a registered trademark of Metaphase Technology, Inc.

Teamcenter Enterprise is a registered trademark of Siemens PLM Corporation.

Names of other products mentioned in this manual are used only for identification purpose and may be trademarks of their companies.





---

# Preface

---

## About this Guide

This guide describes customizing and advanced configuration information for the CATIA Teamcenter Interface with its two Teamcenter Enterprise servers generic Workbench and CATIA Workbench. Before using this guide, be sure you understand:

- the UNIX-based operating system
- the administration of the CATIA system
- the administration of the Teamcenter Enterprise system

---

## Related Documents

The following manuals contain information about installation, usage and customizing of CATIA Teamcenter Interface:

Manual Title	Version
<i>CATIA Teamcenter Interface Installation &amp; Administration Guide</i>	10.0
<i>CATIA Teamcenter Interface User's Manual</i>	10.0

---

## Your Comments are Welcome

Please feel free to tell us your opinion; we are always interested in improving our publications. Mail your comments to:

T-Systems International GmbH  
PDC Engineering  
Fasanenweg 5  
70771 Leinfelden-Echterdingen  
Germany

mail: [cmi\\_support@t-systems.com](mailto:cmi_support@t-systems.com)



---

# Table of Contents

---

<b>COPYRIGHT</b> .....	<b>II</b>
<b>CONTACT</b> .....	<b>II</b>
<b>MANUAL HISTORY</b> .....	<b>II</b>
<b>TRADEMARKS</b> .....	<b>II</b>
<b>ABOUT THIS GUIDE</b> .....	<b>V</b>
<b>RELATED DOCUMENTS</b> .....	<b>V</b>
<b>YOUR COMMENTS ARE WELCOME</b> .....	<b>V</b>
<b>CHAPTER 1</b> .....	<b>1</b>
<b>OVERVIEW</b> .....	<b>1</b>
<b>CHAPTER 2</b> .....	<b>3</b>
<b>SYSTEM ARCHITECTURE</b> .....	<b>3</b>
<b>CHAPTER 3</b> .....	<b>5</b>
<b>ASSEMBLY STRUCTURE</b> .....	<b>5</b>
<b>PREDEFINED OBJECT STRUCTURE</b> .....	<b>5</b>
<b>ENHANCED ASSEMBLY STRUCTURE RELATION OBJECTS</b> .....	<b>6</b>
<b>CLASS DESCRIPTION OF G2ASMPOS</b> .....	<b>7</b>
<b>CLASS DESCRIPTION OF G2ASMNPO</b> .....	<b>8</b>
<b>CLASS DESCRIPTION OF X2ASMPOQ</b> .....	<b>9</b>
<b>CHAPTER 4</b> .....	<b>11</b>
<b>CUSTOMIZATION OPTIONS</b> .....	<b>11</b>
<b>MESSAGE ACCESS RULES</b> .....	<b>11</b>
<i>Restrict permission to load a model into CATIA</i> .....	<b>11</b>
<i>Restrict permission to modify a model from CATIA</i> .....	<b>11</b>
<i>Restrict permission to modify Assembly positions from inside CATIA</i> .....	<b>11</b>
<b>CLASS CONSTANTS</b> .....	<b>11</b>
<b>COMMON CUSTOMIZATION TASKS</b> .....	<b>12</b>
<i>Use of Structured Documents (eg. StDocmnt)</i> .....	<b>12</b>
<i>Use a single Document per Part, for multiple CATParts</i> .....	<b>13</b>
<i>Getting rid of the CATProduct Document</i> .....	<b>14</b>
<i>Exclude objects or classes from the CMI Workbench</i> .....	<b>15</b>
<i>Fill in custom attributes at a CATIA-Item in the Workbench</i> .....	<b>16</b>
<i>Fill in custom attributes at a Document Representant in the Workbench</i> .....	<b>16</b>
<i>Fill in custom attributes at a Model Representant in the Workbench</i> .....	<b>16</b>
<i>Perform additional actions after a model was updated from CATIA</i> .....	<b>16</b>
<i>Customize the name of parts in CATIA</i> .....	<b>17</b>
<i>Customize for sending custom attributes to CATIA for Parts, Relations ,Models</i> .....	<b>17</b>
<i>Show Teamcenter meta data in CATIA</i> .....	<b>17</b>
<i>Customize CATProduct Worklocation</i> .....	<b>18</b>
<i>Customize the CATDrawing filename</i> .....	<b>18</b>
<i>Customize the CATPart filename</i> .....	<b>19</b>
<i>Customize support CATProduct in CATIA V5</i> .....	<b>19</b>
<i>Customize the CATProduct document (x0PrdDoc)</i> .....	<b>19</b>
<i>Customize the CATPart document (GenDoc)</i> .....	<b>19</b>
<i>Customize the Creation of the Part-Part relation during a Synchronize "Link Child" operation</i> .....	<b>20</b>
<i>Customize the Black Box functionality (deprecated)</i> .....	<b>20</b>
<i>Customize the Component CATPart Data Model functionality</i> .....	<b>20</b>
<i>Customize the Deletion of CATPart and CATProduct instances during Synchronize21</i>	

<i>Define your own model types</i> .....	21
<i>Filter Parts that are sent to Catia or VisMockup</i> .....	21
<b>CHAPTER 5</b> .....	<b>22</b>
<b>OPTIONAL CMI-FEATURES</b> .....	<b>22</b>
PLOTTING A MODEL .....	22
<i>Conventions</i> .....	22
<i>Sample for the definition of some naming conventions</i> .....	23
<i>Creation of a CATIA-Model</i> .....	24
<i>Definition of the relevant attributes</i> .....	24
LINX CUSTOMIZING METHODS .....	27
<i>From LINX to CATIA</i> .....	27
STANDARD PROPERTIES IN CATIA V5 .....	27
<i>Customization: Sending standard attributes to CATIA V5</i> .....	27
<i>Customization: Receiving user defined attributes from CATIA V5</i> .....	27
USER DEFINED PROPERTIES IN CATIA V5 .....	28
<i>Customization: Sending user defined attributes to CATIA V5</i> .....	29
<i>Customization: Receiving user defined attributes from CATIA V5</i> .....	30
<i>Customization: using CATIA V5 Properties during Part creation or model registration</i> .....	30
CONFIGURABLE BEHAVIORS IN CATIA V5 .....	31
<i>Descriptions of the behaviors:</i> .....	33
USAGE OF DATABASE NAME OF OBJECTS .....	33
<i>Customization messages</i> .....	33
READ REFERENCE/SHEET INFORMATION FROM CATIA V5 DRAWINGS .....	35
<i>Reference Documents for Drawings</i> .....	35
<i>Sheet-Information for Drawings</i> .....	35
ENHANCED 4D-NAVIGATOR INTEGRATION .....	35
CATIA PROJECT ENVIRONMENT SUPPORT .....	36
SUPPORT OF TEAMCENTER "QUANTITY" .....	36
WORKING WITH CATIA V5 RELEASED CACHE .....	37
<i>Configuration:</i> .....	38
CATPROCESS CUSTOMIZATION .....	38
<i>customization methods in TeamCenter</i> .....	38
VIEWER SUPPORT .....	40
<i>Prerequisites</i> .....	40
<i>Features</i> .....	40
<i>Configuration</i> .....	41
<i>Customization</i> .....	41
<i>Display User Data</i> .....	43
DESIGN TABLE SUPPORT .....	43
<i>Features</i> .....	43
<i>Configuration</i> .....	43
<i>Customizable Methods</i> .....	43
MML SUPPORT .....	44
<i>Features</i> .....	44
<i>Installation</i> .....	44
<i>Configuration</i> .....	44
<i>Customizable Methods</i> .....	45
REPRESENTATION FORMATS IN CATIA V5 .....	46
AUTOMATIC UPDATE OF CATDRAWING TITLE BLOCKS WITH TEAMCENTER DATA .....	47
TRANSFER OF WEIGHT PROPERTIES (INERTIA) FROM CATIA V5 TO TEAMCENTER .....	47
SET BOM-TYPE OF NEW CATIA-FILES BY TEAMCENTER-CUSTOMIZATION .....	49
CUSTOM EXPAND IN THE CMI CATIA WORKBENCH .....	50
<i>Customization example</i> .....	51
SUPPLY ATTRIBUTES FROM CATIA TO TEAMCENTER AND BACK .....	52
<i>Supply Attributes from Catia to Teamcenter</i> .....	52
<i>Supply Attributes from Teamcenter to Catia</i> .....	53
SUPPLY FILE FROM CATIA TO TEAMCENTER .....	53
<i>Supply File from Catia to Teamcenter</i> .....	53
POST -PROCESS CATPRODUCTS FOR SYNCHRONIZE / UPDATE .....	54

---

ENABLE PDM-CENTRIC SYNCHRONIZE .....	54
<i>Validate actions before PDM- Centric Synchronize</i> .....	55
<i>Often used slots in the NVSET's given in the messages above</i> .....	55
REFERENCE GEOMETRIES .....	56
VALIDATE CATIA VERSION .....	58
EXPORT TO FOLDER .....	58
<i>Name customization for Export</i> .....	59
HANDLING OF MAPPING FILES .....	61
CATSCRIPT SUPPORT .....	62
<b>CHAPTER 6.....</b>	<b>64</b>
<b>WORKBENCH ARCHITECTURE .....</b>	<b>64</b>
GENERIC WORKBENCH .....	64
CATIA WORKBENCH.....	64
<b>CHAPTER 7.....</b>	<b>65</b>
<b>TEAMCENTER ENTERPRISE CONFIGURATION VARIABLES .....</b>	<b>65</b>
CMI SERVER OPTIONS .....	65
<b>CHAPTER 8.....</b>	<b>68</b>
<b>CATIA V4 DIRECTORY STRUCTURE .....</b>	<b>68</b>
DIRECTORIES .....	68
FILES .....	69
MODIFY CATIA V4 ENVIRONMENT .....	70
USER DEPENDENT CONFIGURATIONS .....	71
USER DEPENDENT API IN CATIA .....	76
<b>CHAPTER 9.....</b>	<b>77</b>
<b>CATIA V5 DIRECTORY STRUCTURE .....</b>	<b>77</b>
DIRECTORIES .....	77
FILES .....	78
CUSTOMER DEPENDENT CONFIGURATIONS FOR CATIA V5 .....	80
<i>Environment settings</i> .....	80
<i>Display CATIA Node Name in Synchronize</i> .....	88
<b>CHAPTER 10.....</b>	<b>89</b>
<b>DATA MODELS.....</b>	<b>89</b>
DATA STRUCTURE CATIA-WORKBENCH.....	89
DATA STRUCTURE OF CMI-CLASSES .....	90
<i>g0GenBin Class Hierarchy</i> .....	91
<i>g0Repltm Class Hierarchy</i> .....	92

---

# Table of Figures

---

FIGURE 1: CATIA WORKBENCH.....	1
FIGURE 2: SYSTEM ARCHITECTURE OF CMI .....	3
FIGURE 3: AN EXAMPLE ASSEMBLY STRUCTURE .....	6
FIGURE 4: DATA MODEL EXTENSIONS FOR POSITIONING.....	7
FIGURE 5: SIMPLE DRAFT WITH DRAWING FRAME CONVENTIONS.....	22
FIGURE 6: USER DEFINED PROPERTIES.....	28
FIGURE 7: DATA MODEL EXTENSIONS FOR SUPPORT OF 'QUANTITY' .....	36
FIGURE 8: CATEDM INSTALLATION PATH STRUCTURE .....	68
FIGURE 9: INITIALIZATION FILES WITH THEIR ORDER.....	71
FIGURE 10: DIRECTORY STRUCTURE OF THE CMICATV5 MODULE .....	77
FIGURE 11: DIRECTORY STRUCTURE OF THE CMICATV5 INSTALLATION DIRECTORY .....	78
FIGURE 12: EXAMPLE OF DIRECTORY STRUCTURE OF THE CMICATV5 INSTALLATION SUBDIRECTORY MSGCATALOG.....	80
FIGURE 13: DATA STRUCTURE OF CATIA-WORKBENCH.....	89
FIGURE 14: DATA STRUCTURE OF CMI-CLASSES.....	90
FIGURE 15: CLASS HIERARCHY UNDER G0GENBIN.....	91
FIGURE 16: CLASS HIERARCHY UNDER G0REPITM .....	92

---

# CHAPTER 1

## Overview

Individual parts and multiple-layer assemblies can now be processed directly in CATIA thanks to the integration of CATIA into Teamcenter Enterprise; for this the CATIA data is under the control of Teamcenter Enterprise. The CATIA-Workbench in Teamcenter Enterprise serves as the communication medium between the PDM-System Teamcenter Enterprise and the CAD-System CATIA. This is a special structure browser which displays the current storage contents of the CATIA session graphically and provides manipulation facilities.

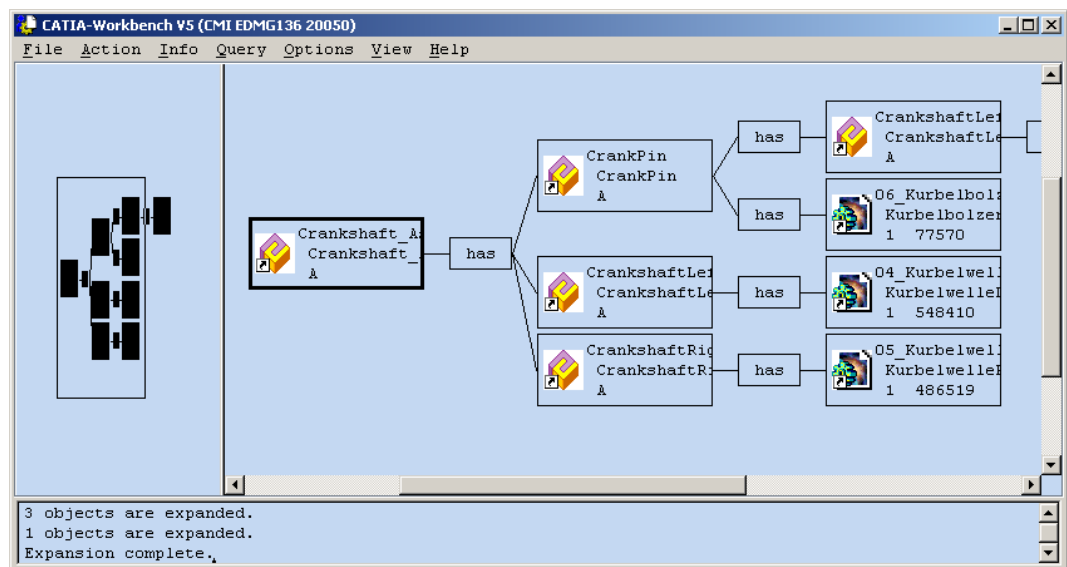


Figure 1: CATIA Workbench

The entire data interchange between Teamcenter Enterprise and CATIA is effected via the Workbench.

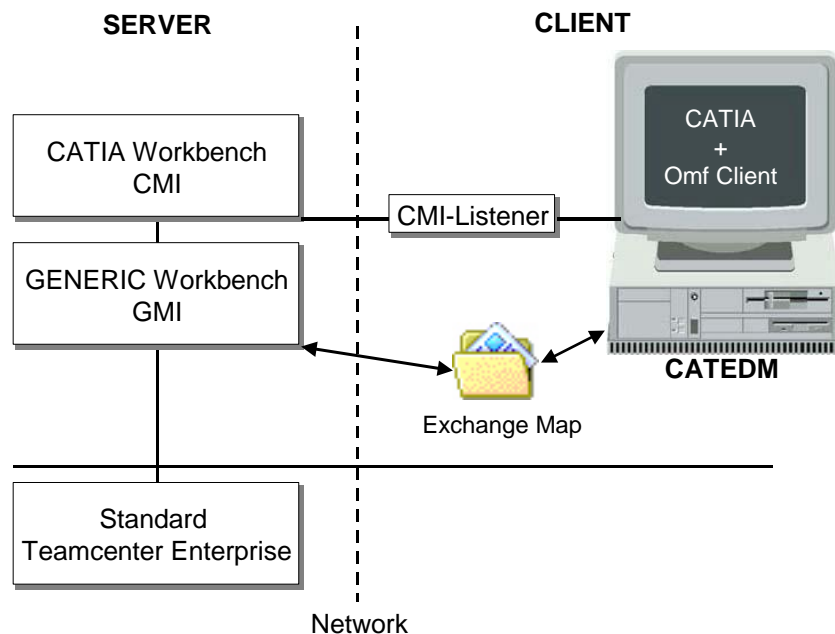


# CHAPTER 2

## System Architecture

The components of CMI are:

GMI	Generic Workbench defines some base classes (see „Data Structure of Catia-Workbench“ on page 896)
CMI	CATIA Workbench defines CATIA specific classes and allows the interaction between the user and CATIA (see „Data Structure of “ on page 907).
Listener	Listener manages the communication between CATIA and Teamcenter Enterprise(Collaboration Foundation, Metaphase).
CATEDM	The CATIA V4 GII module provided by T-Systems Enterprise Services GmbH allows CATIA V4 to manage multiple-level assemblies.
CMICATV5	The CATIA V5 module provided by T-Systems Enterprise Services GmbH allows CATIA V5 to be integrated into Teamcenter Enterprise.
Exchange Map	A dedicated user directory on the client workstation. The CATIA extension expects the model files to be only within this directory.



**Figure 2: System architecture of CMI**

The interaction starts either on the CATIA side or in Teamcenter Enterprise. The listener allows the communication between the Teamcenter Enterprise client and CATIA session. They communicate through RPC and ToolTalk protocol.



---

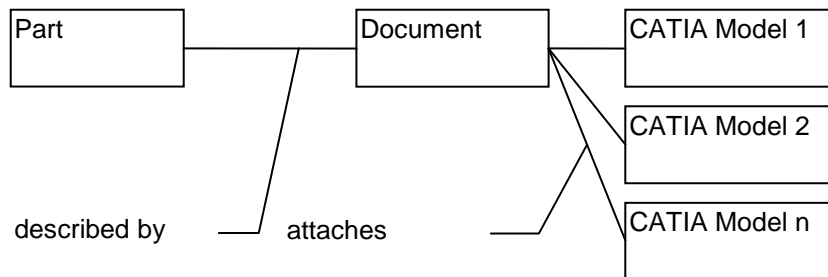
# CHAPTER 3

## Assembly Structure

---

### Predefined Object Structure

The *CATIA Metaphase Interface* contains the following pre-defined object structure:



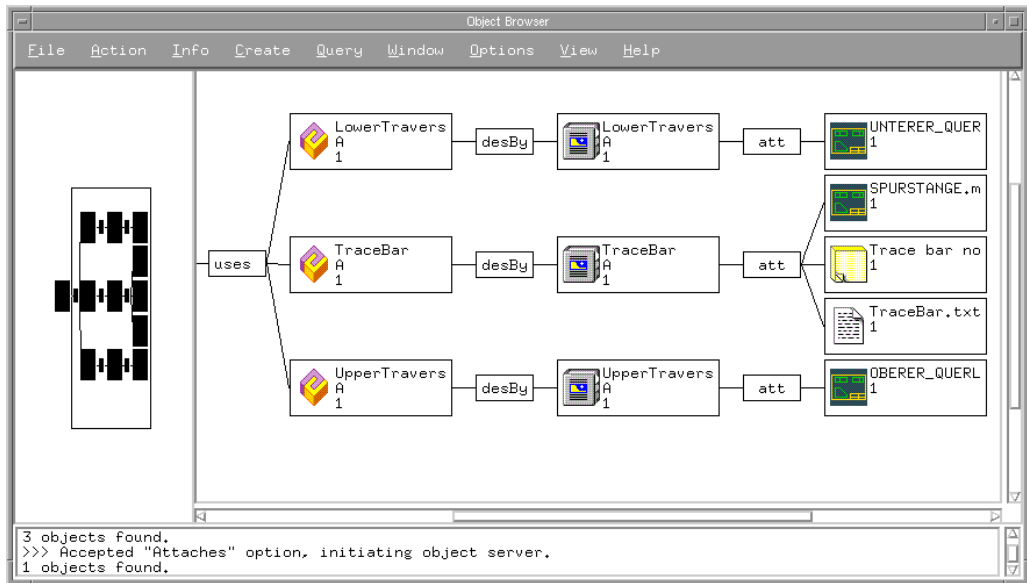
For each part only one relation „**described by**“ with a document element has to exist. The relationship can be created by dragging any document object and dropping it to the part object. Please refer to the Teamcenter Enterprise user documentation for managing relationships between Teamcenter Enterprise objects.

The documents can contain at least one relation „**attaches**“ with CATIA model elements. You can attach more than one CATIA model to a certain document.



Each certain CATIA model object can contain one relation „**Model has plot file**“ with any plot file, which can be created from CATIA.

An example structure is shown in the following figure:



**Figure 3: An example assembly structure**

If you drag a part object and drop it into the CATIA Workbench, only the CATIA relevant data will be extracted and shown in the CATIA Workbench window (see Figure 1: CATIA Workbench on page 1).

The following chapters describe the customizing possibilities to influence the behavior of the CATIA Workbench.



**It is recommended to consult the custom/README file in order to ensure the actuality of source code and customizing tasks.**

## Enhanced Assembly Structure Relation Objects

In order to manage assembly structures with and without position information, CMI extends the standard Teamcenter Enterprise data model with three new classes. "g2AsmRel" is used in order to define the general behavior of the CMI assembly structure relation objects which can be found directly below the standard Teamcenter Enterprise class "AssmStrc". "g2AsmNPo" defines the assembly structure relation object without position information and "g2AsmPos" for assembly structures with position information. Both classes are derived from "g2AsmRel".

1. g2AsmRel - defines the general behavior of CMI assembly structure relation objects
2. g2AsmNPo - for assembly structures without position information
3. g2AsmPos - for assembly structures with position information

The Teamcenter Enterprise data model with CMI looks like this :

- |     |                 |   |
|-----|-----------------|---|
| 1 a | Structur        | "Structure"                               |
| 2 p | AssmStrc        | "Assembly Structure"                      |
| 3 a | <b>g2AsmRel</b> | "CMI Assembly Structure Relation"         |
| 4 p | <b>g2AsmNPo</b> | "CMI Assembly Structure without position" |
| 4 p | <b>g2AsmPos</b> | "CMI Assembly Structure with position"    |

If your specific assembly structure relation is derived from g2AsmPos (assembly structure relation with position information) some additional methods are called for the validation of this information. Due to the fact that validation routines always need some time to perform their actions, it is necessary to decide if the new relation object needs position information or not. If yes, you need to derive this class from g2AsmPos, if not you should use g2AsmNPo.

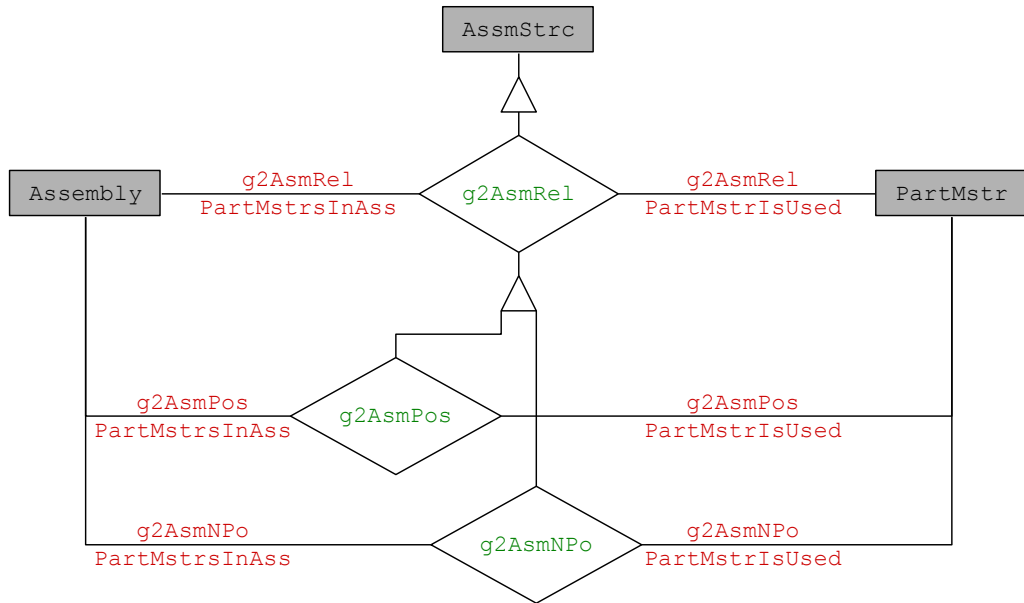


Figure 4: Data model extensions for positioning

## Class Description of g2AsmPos

**Class :** g2AsmPos  
**Parent :** g2AsmRel  
**Prompt :** "g2AsmPos"  
**Left Side Class :** Assembly  
**Right Side Class :** PartMstr  
**Forward Relationship Name :** g2AsmPosPartMstrsInAss  
 (droppable)  
**Backward Relationship Name :** g2AsmPosPartMstrIsUsed  
 (droppable)

Attributes	Opts	Type	Prompt
TMatrix11		string(30)	"TMatrix11"
TMatrix12		string(30)	"TMatrix12"
TMatrix13		string(30)	"TMatrix13"
TMatrix14		string(30)	"TMatrix14"
TMatrix21		string(30)	"TMatrix21"
TMatrix22		string(30)	"TMatrix22"
TMatrix23		string(30)	"TMatrix23"
TMatrix24		string(30)	"TMatrix24"
TMatrix31		string(30)	"TMatrix31"
TMatrix32		string(30)	"TMatrix32"
TMatrix33		string(30)	"TMatrix33"

TMatrix34	string(30)	"TMatrix34"
TMatrix41	string(30)	"TMatrix41"
TMatrix42	string(30)	"TMatrix42"
TMatrix43	string(30)	"TMatrix43"
TMatrix44	string(30)	"TMatrix44"

Opts: i = inherited, d = dynamic, c = cached, r = required

Constants	Inherit	Value
GetInfoDialogC	no	x1AtcGet
g4TrafoTypeC	no	4x4

Messages	Opts	Type
g3CreateOBID	i	ObjectMessage
g3CreateRelationObj	i	ClassMessage
g3CreateWidgetId	i	ObjectMessage
g3DeleteObject	i	ObjectMessage
g3DeleteRelationObject	i	ObjectMessage
g3GetTrafo	i	ObjectMessage
g3UpdateTrafo	i	ObjectMessage
g3ValForUpdTrafo	i	ObjectMessage

## Class Description of g2AsmNPo

**Class :** g2AsmNPo

**Parent :** g2AsmRel

**Prompt :** "g2AsmNPo"

**Left Side Class :** Assembly

**Right Side Class :** PartMstr

**Forward Relationship Name :** g2AsmNPoPartMstrsInAss  
(droppable)

**Backward Relationship Name :** g2AsmNPoPartMstrIsUsed  
(droppable)

Constants	Inherit	Value
g4TrafoTypeC	yes	NO

Messages	Opts	Type
g3CreateOBID	i	ObjectMessage
g3CreateRelationObj	i	ClassMessage
g3CreateWidgetId	i	ObjectMessage
g3DeleteObject	i	ObjectMessage
g3DeleteRelationObject	i	ObjectMessage
g3GetTrafo	i	ObjectMessage
g3UpdateTrafo	i	ObjectMessage
g3ValForUpdTrafo	i	ObjectMessage

## Class Description of x2AsmPoQ

**Class :** x2AsmPoQ

**Parent :** g2AsmPos

**Prompt :** "x2AsmPoQ"

**Left Side Class :** Assembly

**Right Side Class :** x0AssmMr

**Forward Relationship Name :** x2AsmPoQPartMstrsInAss  
(droppable)

**Backward Relationship Name :** x2AsmPoQPartMstrIsUsed  
(droppable)

All Attributes	Opts	Type	Prompt
x0TrafoList		Table	"Transformation List"

Opts: i = inherited, d = dynamic, c = cached, r = required

All Constants	Inherit	Value
GetInfoDialogC	no	DAtcGetI
PsmAssocStRevRevClassC	no	g2ARvPos
UpdateDialogC	no	x1APQUpd

All Messages	Opts	Type
DoCreateRelPost	o	ObjectMessage
DoUpdatePre	o	ObjectMessage
g3CreateOBID	i	ObjectMessage
g3CreateRelationObj	i	ClassMessage
g3CreateWidgetId	i	ObjectMessage
g3DeleteObject	i	ObjectMessage
g3DeleteRelationObject	i	ObjectMessage
g3GetOccName	i	ObjectMessage
g3GetTrafo	i	ObjectMessage
g3UpdateTrafo	i	ObjectMessage
g3ValForUpdTrafo	i	ObjectMessage
x3AddTrafo		ObjectMessage
x3CheckForMultiOcc	i	ObjectMessage
x3DelAssmStrcRel	o	ObjectMessage
x3DeleteTrafo		ObjectMessage
x3GetTrafos	o	ObjectMessage
x3GetTrafosOfTable		ObjectMessage
x3UpdateTrafo	o	ObjectMessage



---

# CHAPTER 4

## Customization Options

---

### Message Access Rules

With message access rules you can control read and write access to CMI data based on properties of the user or the data itself.

#### *Restrict permission to load a model into CATIA*

A user needs access to the “**View**” message for *xOCTFile* objects in order to load model files under Teamcenter Enterprise control into CATIA. The default rules allow any user to view any model.

#### *Restrict permission to modify a model from CATIA*

A user needs access to the “**Edit**” message for *xOCTFile* objects in order to modify model files under Teamcenter Enterprise control from CATIA. The default rules don’t allow a user to update a model that doesn’t belong to him

#### *Restrict permission to modify Assembly positions from inside CATIA*

A user needs access to the “**g3UpdateTrafo**” message for *Part* objects in order to modify Assembly positions. The default rules don’t allow a user to update a position of a part if the parent assembly doesn’t belong to him.

---

### Class Constants

There are some class constants that you can override to meet the specific needs of your customization.

```
define value set g0RelsOfPart ["PartDoc",  
"DocumentsDescribingPart"];
```

```
Part.met:Part.g4RelsC = "g0RelsOfPart";
```

This determines which Part-Document relationships are searched for documents. If your Parts are described by a large variety of documents but few attach CATIA models, you may want to use a dedicated relation class for “CATIA documents”. By overriding this class constant you can make this relation class known to CMI, so it will only search for documents with this relation to the Part.

```
define value set  
g0RelsOfDoc["Attach", "DataItemsAttachedToBusItem"];  
GenDoc.g4RelsC= "g0RelsOfDoc";
```

---

This determines which Document-DatItem relations/relationships will be searched for CATIA models. If your documents attach a large number of files other than CATIA models, you may want to use a dedicated relation class for the CATIA models. By overriding this class constant you can make this relation class known to CMI, so it will only search for files with this relation to the document.

```
define class constant x4CreateCatPartC;  
x0WkBnch.met: x0WkBnch.x4CreateCatPartC = x0CatPrt;
```

This determines which class is used to create a CATPart in Teamcenter Enterprise from CATIA V5. If you want to create a CATPart in Teamcenter Enterprise with another class than class **x0CatPrt**, you should override this class constant by the name of the other class. This new class must be a child class of **x0CatPrt**.

```
define class constant x4CreateCatDrawC;  
x0WkBnch.met: x0WkBnch.x4CreateCatDrawC = x0CatDrw;
```

This determines which class is used to create a CATDrawing in Teamcenter Enterprise from CATIA V5. If you want to create a CATDrawing in Teamcenter Enterprise with another class than class **x0CatDrw**, you should override this class constant by the name of the other class. This new class must be a child class of **x0CatDrw**.

---

## Common Customization Tasks

### *Use of Structured Documents (eg. StDocmnt)*

Since CMI 8.9 you can use structured documents classes without writing any method code. It is sufficient to set the corresponding class constants in the data model. You can use any other document class in the same way.

The following example shows use of `StDocmnt` / `PrtSDocR` (Part to Document Master relation). Also, in this example the same class is used for the Product Document that is used for “regular” documents.

Example:

```
O:x0WkBnch.x4CreatePrdDocC      = StDocmnt;  
O:x0WkBnch.x4PartToPrdDocC     = PrtSDocR;  
O:x0WkBnch.g4PrdDocClassC      = StDocmnt;  
O:x0WkBnch.x4CmpDocClassC      = StDocmnt;  
O:x0WkBnch.x4CmpPartDocRelClassC = PrtSDocR;  
O:x0WkBnch.x4CmpDocPartRelShipC = AddedInfoForParts;  
O:x0WkBnch.g4BlackBoxDocClassC  = StDocmnt;  
O:g0GenWB.g4DocClassC          = StDocmnt;  
O:g0GenWB.g4PrdDocClassC       = StDocmnt;  
O:g0GenWB.g4PartPrdDocRelC     = PrtSDocR;  
O:g0GenWB.g4PartPrdDocRSC      = AddedInfoDocuments;  
O:g0GenWB.g4BlackBoxDocClassC  = StDocmnt;
```

---

```
O:g0GenWB.g4PartBlackBoxDocRelC = PrtSDocR;  
O:g0GenWB.g4PartBlackBoxDocRSC = AddedInfoDocuments;
```

```
//Base Class of documents relevant for Catia V4/V5
```

```
define value set StrcRelsOfPart ["PrtSDocR",  
"AddedInfoDocuments"];
```

```
O:Part.g4RelsC = "StrcRelsOfPart";
```

```
//Teamcenter "Prepare" Dialog for Catia Files should offer  
//Structured Document for Prepare
```

```
O:g0GenBin.StartClassToPrepareC = "StGenDoc";
```

```
O:g0GenBin.DefaultClassToPrepareC = "StDocmnt";
```

If you want to use the PartSDoc-relation(Part to Structured Document), the same example must be changed to:

```
O:x0WkBnch.x4CreatePrdDocC = StDocmnt;
```

```
O:x0WkBnch.x4PartToPrdDocC = PartSDoc;
```

```
O:x0WkBnch.g4PrdDocClassC = StDocmnt;
```

```
O:x0WkBnch.x4CmpDocClassC = StDocmnt;
```

```
O:x0WkBnch.x4CmpPartDocRelClassC = PartSDoc;
```

```
O:x0WkBnch.x4CmpDocPartRelShipC = PartsRepresentedbyDocument;
```

```
O:x0WkBnch.g4BlackBoxDocClassC = StDocmnt;
```

```
O:g0GenWB.g4DocClassC = StDocmnt;
```

```
O:g0GenWB.g4PrdDocClassC = StDocmnt;
```

```
O:g0GenWB.g4PartPrdDocRelC = PartSDoc;
```

```
O:g0GenWB.g4PartPrdDocRSC = DocumentRepresentingPart;
```

```
O:g0GenWB.g4BlackBoxDocClassC = StDocmnt;
```

```
O:g0GenWB.g4PartBlackBoxDocRelC = PartSDoc;
```

```
O:g0GenWB.g4PartBlackBoxDocRSC = DocumentRepresentingPart;
```

```
define value set StrcRelsOfPart ["PartSDoc",  
"DocumentRepresentingPart"];
```

```
O:Part.g4RelsC = "StrcRelsOfPart";
```

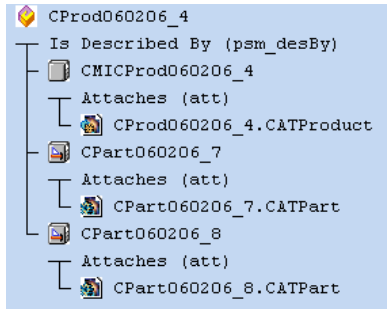
```
O:g0GenBin.StartClassToPrepareC = "StGenDoc";
```

```
O:g0GenBin.DefaultClassToPrepareC = "StDocmnt";
```

### ***Use a single Document per Part, for multiple CATParts***

In the OOTB Synchronize function in CATIA V5, where CMI creates new Parts and Documents, the user can create a fresh Document for each CATPart, or he may use an existing document at the Part for all CATPart files that are linked to the Part. This is

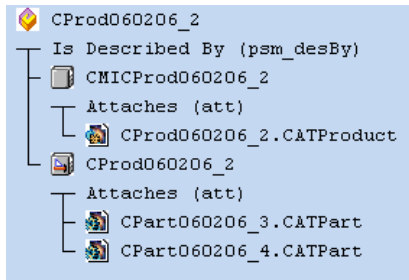
effected by using the TC *Prepare* functionality, where either an existing or a new document name can be given by the user.



In an organization where there is always only one Document describing a Part, the following Teamcenter configuration variable can greatly enhance usability and consistency:

```
set CMI_SINGLE_PART_DOCUMENT "ON";
```

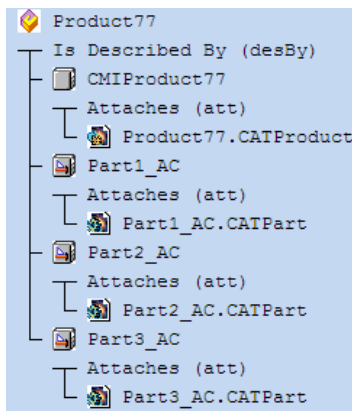
If this variable is set, only one document is created for each Part during Synchronize; all CATParts will be attached to this document. Also, if a Part already has a describing Document of the right class, it will be used.



The complementary data model is obtained with the following setting

```
set CMI_SINGLE_PART_DOCUMENT "SINGLE_FILE";
```

This will create a new document for each CATPart, which will be created automatically and named after the CATPart.



### ***Getting rid of the CATProduct Document***

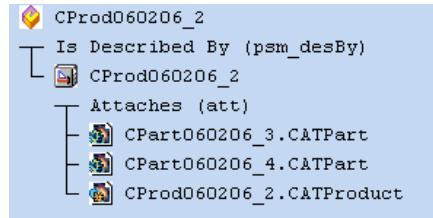
In order to improve consistency CMI uses a special class and relation for the Document that attaches a CATProduct. In an OOTB CMI installation you will typically see two

---

Documents: one that attaches the CATProduct and one that attaches the CATParts. This is because the CATProduct should not be exposed to direct user actions.

Using the TC configuration  
set CMI\_SINGLE\_PART\_DOCUMENT "ON"; (see above)

You can get rid of this special document. Just set the class constants pertaining to the Product Document to point to your regular Document class.



The following example shows how to set the Product Document to the same document class used to attach CATParts (*DesDoc*)

Example:

```
O:x0WkBnch.x4CreatePrdDocC      = DesDoc;
O:x0WkBnch.x4PartToPrdDocC      = PartDoc;
O:x0WkBnch.g4PrdDocClassC       = GenDoc;
O:x0WkBnch.x4CmpDocClassC        = DesDoc;
O:x0WkBnch.x4CmpPartDocRelClassC = PartDoc;
O:x0WkBnch.x4CmpDocPartRelShipC = DocumentsDescribingPart;
O:g0GenWB.g4PrdDocClassC        = GenDoc;
O:g0GenWB.g4PartPrdDocRelC      = PartDoc;
O:g0GenWB.g4PartPrdDocRSC       = DocumentsDescribingPart;
//make sure that the same Document class is used by the
//"Prepare" function
O:g0GenBin.StartClassToPrepareC = DesDoc;
O:g0GenBin.DefaultClassToPrepareC = DesDoc;
```

in config.cfg:

```
set CMI_SINGLE_PART_DOCUMENT "ON";
```

### ***Exclude objects or classes from the CMI Workbench***

CMI calls the message `g0PdmItm:g3CheckIfItemInScope` to check whether it must handle a document or model object or ignore it. You can exclude an entire class or individual objects based on their attributes.

For Documents you can also override `GenDoc:g3CheckIfDocInScope` if you want to filter documents based on attributes of the Part they describe.

See also

`g0PdmItm:g3CheckIfItemInScope`

`DataItem:g3CheckIfItemInScope`

`g0GenMod:g3CheckIfItemInScope`

---

**GenDoc : g3CheckIfDocInScope**

in g3Custom.mth

### ***Fill in custom attributes at a CATIA-Item in the Workbench***

By default, all attributes of the Part are copied to the CATIA Item. So to add attributes from your Part class you only need to attach those attributes to **x0CTItem**.

To do additional work after a CATIA-Item is created, override **g0GenItm : g3CreateGIPost**.

To set attributes based on the **AsmStrc** relation between the part and its parent Assembly, override **g0GenItm : g3SetAttrsFromRelation**. Note that the passed relation may be NULL, if the CATIA Item is a root level assembly.

See also

**g0GenItm : g3SetGIAttrs**

**g0GenItm : g3CreateGIPost**

**g0GenItm : g3SetAttrsFromRelation**

in g3Custom.mth

### ***Fill in custom attributes at a Document Representant in the Workbench***

To fill in additional attributes based on the original Document after a Document Representant was created, override **g0DocRep : g3SetSpecificAttrs**

See

**g0DocRep : g3SetSpecificAttrs**

in g3Custom.mth

### ***Fill in custom attributes at a Model Representant in the Workbench***

To fill in additional attributes based on the original CATIA model after a Model Representant was created, override **x0ModRep : g3SetSpecificAttrs**.

Note that the default implementation is not empty, so you must call the parent method. This requires to derive your own class from **x0ModRep**.

See

**x0ModRep : g3SetSpecificAttrs**

in x3Custom.mth

### ***Perform additional actions after a model was updated from CATIA***

To perform additional actions after a CATIA model was modified from inside CATIA, override **x0CTFile : x3SaveModelPost**

See

**x0CTFile : x3SaveModelPost**

in x3Custom.mth

---

### ***Customize the name of parts in CATIA***

To customize the name of parts displayed in CATIA override `x0CTItem:x3GetDescriptionInCAD`. By default the part number is shown.

See

`x0CTItem:x3GetDescriptionInCAD`  
in `x3Custom.mth`

### ***Customize for sending custom attributes to CATIA for Parts, Relations, Models***

Override:

`x0CTFile:x3GetCustomDataForCAD`  
`Part: x3GetCustomDataForCAD`  
`g2AsmPos: x3GetCustomDataForCAD`  
`x0WkBnch:x3SendCustomAttrPref`  
in `x3Custom.mth`

Following you will find a detailed description which method has to be overwritten in which case:

Activate LINX interface

→ `x0WkBnch:x3SendCustomAttrPref` in `x3Custom.mth`

Send custom attributes from Model to CATIA

→ `x0CTFile:x3GetCustomDataForCAD` in `x3Custom.mth`

Send custom attributes from Part to CATIA

→ `Part:x3GetCustomDataForCAD` in `x3Custom.mth`

Send custom attributes from Relation between Assemblies to CATIA

→ `g2AsmPos:x3GetCustomDataForCAD` in `x3Custom.mth`

### ***Show Teamcenter meta data in CATIA***

The *More* Button in the CMI Info command allows to retrieve realtime information about the selected CATIA V5 item from Teamcenter. By default the *Get Item Info* dialogs define the information shown.

In CATIA V4 this information is displayed by the MODEL INFO panel.

You can customize the information that is displayed by overriding the following methods:

```
message g0GenBin:x3GetItemInfoForCAD (  
    input : ObjectPtr    this          ::  
    output: SetOfStrings *LabelSet     ::  
    output: SetOfStrings *ValueSet     ::  
    output: integer      *mfail) code
```

---

To retrieve information from the data item

```
message Part:x3GetPartInfoForCAD (  
    input : ObjectPtr      this      ::  
    input : NULL ObjectPtr PartRel   ::  
    output: SetOfStrings   *LabelSet ::  
    output: SetOfStrings   *ValueSet ::  
    output: integer        *mfail) code
```

To retrieve information from the TC Part (V5 only)

```
message Relation:x3GetRelInfoForCAD (  
    input : ObjectPtr      this      ::  
    output: SetOfStrings   *LabelSet ::  
    output: SetOfStrings   *ValueSet ::  
    output: integer        *mfail) code
```

To retrieve information from the Assembly Structure Relation (V5 only)

Original implementation available in x3Custom.mth

### ***Customize CATProduct Worklocation***

Return the `x1PrdDataWorkloc` Attribute from the Workbench Object. The returned string must be freed with `nlsStrFree()`.

```
x0WkBnch:g3CatPrdV5Workloc (  
    input : string      classname    ::  
    output: string      *sCatPrdV5WL ::  
    output: integer     *mfail)
```

Original implementation available in x3Custom.mth

### ***Customize the CATDrawing filename***

Customize the default filename of a new CATDrawing

```
x0CatDrw:x3CreateModelName (  
    input : ObjectPtr this      ::  
    input : string   pHost      ::  
    input : string   pUserName   ::  
    input : string   pDirectory  ::  
    input : NULL ObjectPtr DocObj ::  
    output: string   *filename   ::  
    output: integer  *mfail)
```

---

Original implementation available in x3Custom.mth

### ***Customize the CATPart filename***

Customize the default filename of a new CATPart

```
x0CatPrt:x3CreateModelName (  
    input  :      ObjectPtr this      ::  
    input  :      string   pHost      ::  
    input  :      string   pUserName  ::  
    input  :      string   pDirectory ::  
    input  : NULL ObjectPtr DocObj    ::  
    output :      string   *filename  ::  
    output :      integer  *mfail)
```

in x3Custom.mth

### ***Customize support CATProduct in CATIA V5***

Supress CATProducts to CATIA V5

```
x0WkBnch:g3CatPrdV5Support (  
    input : string   classname      ::  
    output: boolean  *bCatPrdV5Sup  ::  
    output: integer  *mfail)
```

Original implementation available in x3Custom.mth

### ***Customize the CATProduct document (x0PrdDoc)***

Customize the creation of CATProduct document

```
x0PrdDoc:x3CreatePrdDoc (  
    input  : string      className      ::  
    input  : string      partNumber     ::  
    input  : ObjectPtr   partObj        ::  
    output : ObjectPtr   *newPrdDocObj  ::  
    output : integer     *mfail)
```

Original implementation available in x3Custom.mth

### ***Customize the CATPart document (GenDoc)***

Customize the creation of Model or CATPart document

```
Part:x3LinkWithDocument (  
    input  : ObjectPtr thisObject      ::  
    input  : ObjectPtr docObject       ::  
    output : integer *mfail)
```

```

xOCTFile:x3CreateDocForModel (
    input      : ObjectPtr  thisObject      ::
    input NULL : ObjectPtr  partObject      ::
    output     : ObjectPtr  *newDocObject   ::
    output     : integer *mfail)

```

Original implementation available in x3Custom.mth

If partObject is NULL, then the document is being created at file-registration time (during a "Create" operation in Synchronize) and the parent Part is unknown. If you would prefer to rather create the document later (or to be able to select a document already attached to a Part) at the time when the File is to be attached to the Part, then you should change this method to block the creation of the document when the partObject is NULL.

### ***Customize the Creation of the Part-Part relation during a Synchronize "Link Child" operation.***

Customize the following message to your needs:

```

class message AssmStrc:x3CreateUsesRel (
    input : string      className      ::
    input : ObjectPtr   leftPrtObj     ::
    input : ObjectPtr   rightPrtObj    ::
    input : NvSet       usesInfos      ::
    output: ObjectPtr   *newRelObj     ::
    output: string      *trafoIndex    ::
    output: integer     *mfail)
in x3Custom.mth

```

### ***Customize the Black Box functionality (deprecated)***

The CATIA V5 Black Box creation/add/remove functionality uses the following API's:

<code>x0PrdDoc:x3AttachBlackBoxFiles ()</code>	to attach the black box files to the document
<code>Part:x3CreBlackBoxFileObj ()</code>	to create a black box part file object
<code>Part:x3RemoveBlackBoxFiles ()</code>	to delete black box files
<code>Part:x3AddBlackBoxFiles ()</code>	to copy black box files to the user's work location and register them

in x3Custom.mth.

### ***Customize the Component CATPart Data Model functionality***

The CATIA V5 Component CATPart Data Model functionality uses the following API's:

<code>x0CatPrt:x3CreateCmpDataPost ()</code>	Operations after a x0CatPrt has been created in Teamcenter Enterprise (new Data Model)
<code>ProdBI:x3CreateCmpDocPost ()</code>	Operations after a Document has been created in Teamcenter Enterprise (new Data Model)

---

**x0CatPrt:x3SaveComponentPost ()** Operations after a Part has been updated in Teamcenter Enterprise (new Data Model)  
in x3Custom.mth.

### ***Customize the Deletion of CATPart and CATProduct instances during Synchronize***

This is a part of CATIA V5 Synchronize Command functionality and uses the following API's:

**x0CTFile:x3ProcessCATIADeletion ()** Called during the Interactive "Update and Create" procedure for each CATPart or V4 Model deleted from the product structure by the user in CATIA V5. In OOTB CMI, this is not implemented. Override to remove file instances during Synchronize.

**Part:x3ProcessCATIADeletion ()** Called during the Interactive "Update and Create" procedure for each CATProduct removed from the product structure in CATIA V5.

in x3Custom.mth.

### ***Define your own model types***

If you'd like to have your own values for filtering models in the Catia-Workbench you can change the attribute **g0ModelType**. This attribute is also contained in the Query- and Create dialogs for Catia-Files. Then you define your own value set for **g0ModelType**. After that you have to set the class constant **g4ModelTypesC** to the name of your value set at the class **g0GenMod**. It is also necessary to attach this value set to the attribute **g0ModelType** with a condition.

Example:

```
o:g0GenMod.g4ModelTypesC="[your_value_set]";
```

*It is necessary to have a condition for changing the value set at the attribute g0ModelType.*

```
define condition ALWAYS_TRUE(obj) := "1=1";
```

```
attach value set [your_value_set] to g0ModelType if (ALWAYS_TRUE);
```

### ***Filter Parts that are sent to Catia or VisMockup***

You can filter parts, which should not be sent to Catia or to the viewer. If a part is filtered, the part itself and all its subtrees are not sent to Catia / to the viewer. The following method can be overwritten to filter parts:

```
class message x0WkBnch:x3FilterPartsToSend (  
    input:      string          classname      ::  
    input:      string          sTargetSystem ::  
    update:     SetOfObjects     *parts        ::  
    output:     integer          *mfail)
```

The parameter **sTargetSystem** can contain the values "V4", "V5" or "Viewer". This is the system where the parts are currently sent to. If objects are removed from the SetOfObjects parameter **parts**, they and their subtrees are not sent. This method does not impact the display in the CMI Workbench.

---

# CHAPTER 5

## Optional CMI-Features

---

### Plotting a model

#### *Conventions*

The *CATIA Teamcenter Interface* permits an automatic update of title blocks in CATIA-models with title block values that are retrieved from Teamcenter Enterprise. The CMI functionality identifies title blocks in a CATIA model by means of a naming convention. This naming convention is threefold in the way that it affects three levels of CATIA data in the model whose identifiers have to start with strings qualifying them to be constituents of a title block.

Those qualifying identifier sub strings are to be defined for

...**drafts**: if any draft identifier **starts** with the qualifying string the CMI function continues to search for

...**views**: if any view identifier **starts** with the qualifying string the CMI function continues to search for ditto referring to ...

...**details**: if any detail identifier **starts** with the qualifying string defining a title block the CMI function retrieves all values defined for this title block from Teamcenter Enterprise and displays them in the corresponding view positions.

There is only one (configurable) start string for drafts and one other for views but there may be a multitude of strings for details (as many as different title blocks) defining their corresponding title blocks. Thus the need for probably many different title blocks in a company is covered.

The following figure shows a simple draft with the name conventions used by the *CATIA Teamcenter Interface*.

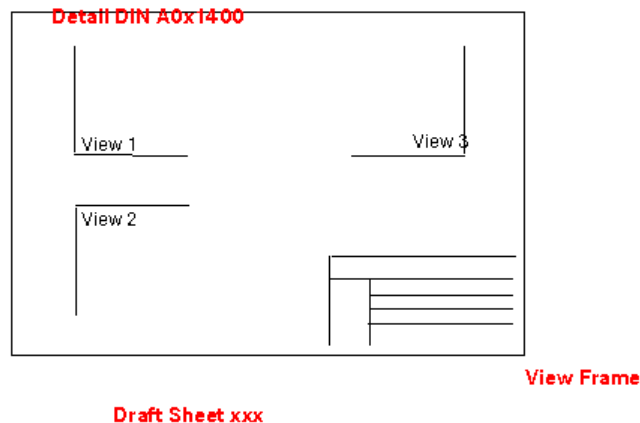


Figure 5: Simple draft with drawing frame conventions

---

### *Sample for the definition of some naming conventions*

In the following we describe a sample which fits to the conventions that we need to define in the file `dshdrawingframe.sh`. This file must be located in a path that can be found in `$PATH`. You can find a sample for this file in your `catedm` installation path `catedm/data/dshdrawingframe.sh`.

In this example we will define

Draft           : BLATT  
View            : RAHMEN  
Detail          : 'DIN A3' and 'DIN A5'

```
Catedm/data/dshdrawingframe.sh
..
CATIAFRAME)
    Case $TOKEN in
        ALL)
            echo "redraw-default yes"
            echo "indicatordraft BLATT"
            echo "indicatorview RAHMEN"
            echo "standard_format DIN"
            echo "frames {\\"DIN A3\\" \\"DIN A5\\"}"
            ;;
        redraw-default)
            echo "yes"
            ;;
        indicatordraft)
            echo "BLATT"
            ;;
        indicatorview)
            echo "RAHMEN"
            ;;
        standard_format)
            echo "DIN"
            ;;
        frames)
            echo "{DIN A3} {DIN A0}"
            ;;
        *) ;;
    esac
;;
...
```

---

### Creation of a CATIA-Model

Now we create a CATIA-Model with CMI and CATIA that fits to the previously declared naming conventions (see appdefault.sh) :

- Load a CATIA-Model with the CATEDM functionality  
**METAPHSE: READ**
- Split CATIA screen into "DRAFT" and "3D" with  
**"IMAGE → DEFINE"**
- Create a "Draft" with  
**"DRAFT → CREATE"**  
  
SP (SPace (3D)) -> DR (DRawing) : activate **DR**awing mode  
(name e.g. "Blatt")
- Create geometry (e.g. a "PLANE")  
DR → SP : activate 3D **SP**ace mode
- Create a 2D point in your new created Draft "POINT -> COORD"  
SP → DR : activate **DR**awing mode
- Create "View" with  
**"AUXVIEW → CREATE"** functionality.  
  
SP → DR : activate **DR**awing mode  
Describes the projection to the plane created before.  
  
SELECT LINE/ PLANE (created in point 3)  
SELECT POINT (created in point 4)  
KEY VIEW ID (e.g. "Rahmen")
- Create a Detail with **"DETAIL → CREATE"**  
KEY DETAIL ID (name e.g. "DIN A3")  
Select in your 2D View (created with AUXVIEW) a Line/Point to create a Ditto onto this view.
- Save the model  
**METAPHSE: UPDATE → ACTIVE**

### Definition of the relevant attributes

The customizing of the company specific drawing frames now can be made in the shell script *drawingframe* that can be found in directory 'catedm/bin'. In this script file the user can define several views like „**DIN A3**“ whereat a view can contain several entries. An entry for an example looks like the following:

```
{string { "Company Name" }}           {xpos -380} {ypos 0} {size 10}
{string {Created: @Date}}             {xpos 2}    {ypos 62} {size 1}
{string {Modified: @ModificationDate}} {xpos 2}    {ypos 42} {size 1}
{string {Name: @Name}}                {xpos -0}   {ypos 5}  {size 1.5}
                                       {color 3}
{string {User: $USER}}                {xpos -20}  {ypos 2}  {size 3}
{orientation 1}
```

---

Each view section in the drawingframe script file can contain the following entries within a single line:

Statement	Syntax	Description
Xpos	{xpos x}	x position of text entry
Ypos	{ypos y}	y position of text entry
Size	{size s}	size of text entry
Color	{color c}	color of text entry
Orientation	{orientation o}	orientation of text entry
String	{string {sub string} }	text entry.

The *sub string* can contain following statements:

Statement	Example	Description
"string"	"T-Systems"	The string will be written into the drawing
String	T-Systems	The string will be written into the drawing
\$<var>	\$USER	The UNIX user name will be written into the drawing
@<var>	@Date	The alias name for the x3GetAttrsForCAD message. In our example listing it is the alias name for the Teamcenter attribute "CreationDate".

The parameter that begin with "@" are relevant for the customization. In this case, the parameter list:

Date  
ModificationDate  
Name

will be transferred to the *x3GetAttrsForCAD* CMI method as *Attrs\_demanded* string list. Now this string list can be scanned; for each custom parameter the variable *AttrVals\_found* is extended and given back to the CMI server. The CMI server can now actualize the drawing frame with the most recent database information.

Please consult the files

custom/g3Custom.mth (GMI related methods) and custom/x3Custom.mth (CMI related methods) for more information.



---

## LINX customizing methods

### *From LINX to CATIA*

Rebuild data structure from LINX in CMI-WB with calling OMF client and send it automatically to CATIA.

Override:

**x0WkBnch:x3FindCatModelByCusKey**

**x0WkBnch:x3FindCatPartByCusKey**

**x0WkBnch:x3FindCatRelByCusKey**

in x3Custom.mth

Find with custom attributes the OBID from Model

→ x0WkBnch:x3FindCatModelByCusKey in x3Custom.mth

Find with custom attributes the OBID from Part

→ x0WkBnch:x3FindCatPartByCusKey in x3Custom.mth

Find with custom attributes the OBID from Relation

→ x0WkBnch:x3FindCatRelByCusKey in x3Custom.mth

---

## Standard Properties in CATIA V5

In CATIA V5 standard properties (Revision, Definition, Nomenclature and Description) can be set from CMI. The property values could be changed by the user and all changed properties are sent back to Teamcenter Enterprise during update. Standard properties may be set / stored for the Part-, x0CatPrd- or x0CtFile-Class.

### *Customization: Sending standard attributes to CATIA V5*

Overwrite:

x3GetCATIARevision

x3GetCATIANomenclature

x3GetCATIADefinition

x3GetCATIADescription

### *Customization: Receiving user defined attributes from CATIA V5*

It is possible to receive changed standard properties from CATIA V5. These attributes may be saved back in Teamcenter Enterprise.

Overwrite:

x3SetCATIARevision

x3SetCATIANomenclature

x3SetCATIADefinition  
x3SetCATIADescription

## User Defined Properties in CATIA V5

In CATIA V5 you can add user-defined properties to the standard CATIA V5 properties form (Added Properties).

CMI provides two new messages to work with such user defined properties.

It is possible to send user defined properties from Teamcenter Enterprise to CATIA V5 and display these properties within the standard properties dialog. The property values could be changed by the user and all changed properties are sent back to Teamcenter Enterprise during update.

It is not possible to define new properties in CATIA V5 dialog and save them back to Teamcenter Enterprise.

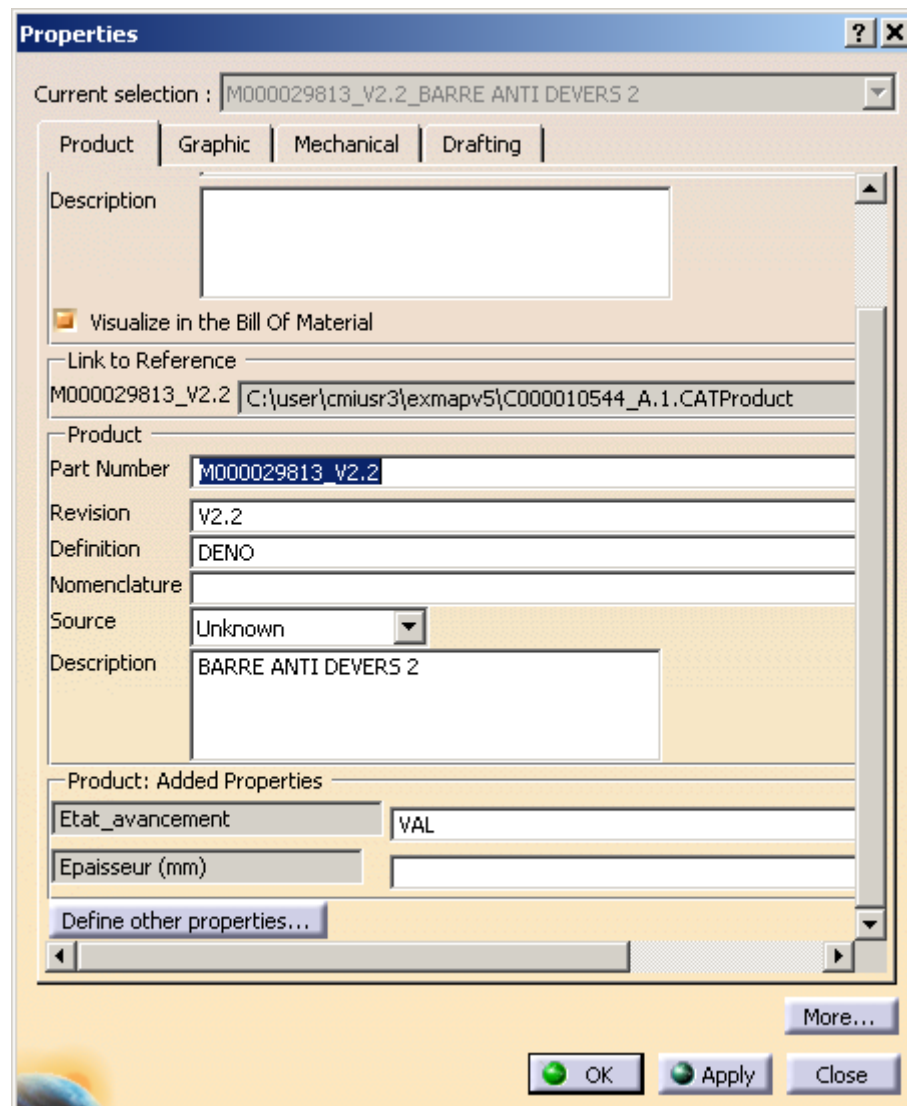


Figure 6: User defined properties

---

### ***Customization: Sending user defined attributes to CATIA V5***

It is possible to send user defined attributes to CATIA V5 and display them within the standard CATIA V5 properties dialog. Therefore you have to customize the CMI message “**x3SendUserDefProps**”. This message is attached to several classes → it is possible to add properties for CATProducts, CATParts, V4 models and Black Box objects.

```
x0CTFile/Part/x0CatPrd: x3SendUserDefProps (  
    input:      ObjectPtr    thisObj      ::  
    output:     SetOfStrings *vPropDisplaySet ::  
    output:     SetOfStrings *vPropNameSet  ::  
    output:     SetOfStrings *vPropValueSet  ::  
    output:     integer *    mfail) code
```

**vPropDisplaySet** ... the display names of attributes within properties dialog

**vPropNameSet** ... the database name of the attributes to be able to save changed values back during update

**vPropValueSet** ... the attribute values

If you want to add user defined properties to the CATProduct properties dialog you have to customize “**Part:x3SendUserDefProps**”

If you want to add user defined properties to the CATPart, V4 Model or Black Box CATPart properties dialog you have to customize “**x0CTFile:x3SendUserDefProps**”

If you want to add user defined properties to the Black Box CATProduct object properties dialog you have to customize “**x0CatPrd:x3SendUserDefProps**”

If you want to delete user defined properties you have to customize the method **x3SendUserDefPropsExt** (Wrapper of **x3SendUserDefPropsExt**) :

```
x3SendUserDefPropsExt(  
    input:      ObjectPtr    thisObj      ::  
    output:     SetOfStrings *vPropDisplaySet ::  
    output:     SetOfStrings *vPropNameSet  ::  
    output:     SetOfStrings *vPropValueSet  ::  
    output:     SetOfStrings *vPropDelSet   ::  
    output:     integer *    mfail)
```

where **vPropDelSet** contains the attribute-names to delete in Catia V5

---

### **Customization: Receiving user defined attributes from CATIA V5**

It is possible to receive changed user defined properties from CATIA V5; These attributes may be saved back in Teamcenter Enterprise. Therefore you have to customize the CMI message "x3ReceiveUserDefProps". If newly created user defined properties within CATIA V5 standard dialog should be stored in Teamcenter Enterprise, they have to be declared in the CMI\_CONFIGURATION\_FILE.

```
x0CTFile/Part/x0CatPrd: x3ReceiveUserDefProps (  
    update:      ObjectPtr    thisObj      ::  
    input:       SetOfStrings vPropDisplaySet ::  
    input:       SetOfStrings vPropNameSet  ::  
    input:       SetOfStrings vPropValueSet ::  
    output:      integer *    mfail) code
```

The declaration in the CMI\_CONFIGURATION\_FILE of newly created properties in catia, which should be stored in Teamcenter Enterprise looks as follows:

```
<UserDefinedProperties>  
    <UserDefinedProperty Name="CustomerProp1" />  
    <UserDefinedProperty Name="CustomerProp2" />  
</UserDefinedProperties>
```

### **Customization: using CATIA V5 Properties during Part creation or model registration**

The customizing messages x3CreatePartExt and x3CreateFileForUpdtExt provide property attributes from Catia V5, in order to have this information available before a Part, Document or Model is created in Teamcenter.

```
class message x0WkBnch:x3CreatePartExt (  
    input  :      string      strClassname  ::  
    input  :      string      newPartClass  ::  
    output :      ObjectPtr *partObject     ::  
    input  :      NvSet       pAttributeSet  ::  
    input  : NULL string      strPartCategory ::  
    input  :      boolean     bUseDialog     ::  
    update :      SetOfStrings *vStats      ::  
    output :      integer     *mfail) code  
  
message g0GenMod:x3CreateFileForUpdtExt (  
    update:      ObjectPtr    thisObj      ::  
    input  :      string      file         ::  
    input  :      string      partnumber   ::  
    input  : NULL NvSet       ModelInfos   ::  
    input  : NULL NvSet       PartAttributes ::  
    input  : NULL SetOfStrings matrix      ::  
    update: NULL ObjectPtr    existingPart ::
```

```

output:      ObjectPtr *   createdPart  ::
output:      ObjectPtr *   document      ::
output:      ObjectPtr *   relObject     ::
update:     NULL SetOfStrings vStats      ::
output:      integer      *mfail) code

```

The NvSet arguments pAttributeSet and Part Attributes respectively contain the following CATIA Standard Properties:

```

CATIADefinition
CATIANomenclature
CATIARevision
CATIADescription
Nomenclature
PartNumber

```

Use nvsGet to retrieve the values of these properties.

The default implementation of these methods can be found in the file x3Custom.mth.

You can retrieve User Defined Properties that were present in CatiaV5 via the function x0WkBnch:x3ConvertUserDefProps

```

dstat= x3ConvertUserDefProps (x0WkBnchClass,
                               ModelInfos,
                               &vDisplayNameSet,
                               &vPropNameSet,
                               &vPropValueSet,
                               mfail);

```

for Model-Properties or

```

dstat= x3ConvertUserDefProps (x0WkBnchClass,
                               PartAttributes,
                               &vDisplayNameSet,
                               &vPropNameSet,
                               &vPropValueSet,
                               mfail);

```

for Part-Properties.

## Configurable Behaviors in CATIA V5

It is possible to configure the behavior of catia while update & synchronize. Dependend on a given prefix of the partnumber in a CATProduct/Component it is possible to force the update to:

- Ignore a Component.
- Ignore the CATProduct/Component and it's subtree.
- Create a special relation in Teamcenter

This has to be configured in the CMI\_CONFIGURATION\_FILE. The scheme looks as follows:

```

<CMIConfigTopics>
.
.
<ConfigurableBehaviors>
    <ConfigurableBehavior UniqueID = "[Unique ID]">
        <BehaviorType>[Behavior Type]</BehaviorType>
        <PartNumberPrefix>[Prefix]</PartNumberPrefix>
        <Behavior>[Behavior]</Behavior>
    </ConfigurableBehavior>
</ConfigurableBehaviors>
.
.
</CMIConfigTopics>

```

There can be multiple tags <ConfigurableBehavior> in the tag <ConfigurableBehaviors>

The tag <UniqueID> has to contain a value which have to be unique in this file.

The following options exist to define which CATIA components a configurable behavior shall apply to:

**Based on its part number -**

```
<PartNumberPrefix>Spec_</PartNumberPrefix>
```

The behavior applies to components whose part number begins with "Spec\_"

An empty PartNumberPrefix makes the behavior apply to any component.

**Based on its product type -**

```
<ProductType>ElecWireGroup</ProductType>
```

The behavior applies to components of the type ElecWireGroup. To help with configuration, the Product type is shown in the CMI Info dialog.

**Based on its instance name -**

```
<InstanceNamePrefix>XY_</InstanceNamePrefix>
```

The behavior applies to components where the instance name begins with "XY\_".

When these tags are combined, a component must match the requirement of either tag.

The following combinations of values are valid:

No.	BehaviorType	Behavior
1	EmbeddedNodeBehavior	SkipNode
2	EmbeddedNodeBehavior	DeepSkipNode
3	EmbeddedNodeBehavior	IgnoreNode

4	EmbeddedNodeBehavior	ReferenceGeometry
5	ProductNodeBehavior	IgnoreNode

***Descriptions of the behaviors:***

**1. SkipNode:**

The component is skipped and its children are instantiated as a direct child of the CATProduct/Assembly that contains the component.

**2. DeepSkipNode:**

The component and all subsequent components beneath it are skipped, up to the next regular CATProduct/CATPart.

**3. IgnoreNode:**

The Component and its substructure is ignored in Teamcenter.

**4. ReferenceGeometry:**

The Component is skipped and its children are instantiated in Teamcenter with a special Reference relation instead of the standard Assembly relation. So the substructure of this component will not be part of the BOM.

**5. IgnoreNode (Product):**

CATProduct is ignored in Teamcenter. This may result in broken links as the Product is not provided by Teamcenter during a load.

---

**Usage of database name of objects**

There is the possibility to query for objects by OBID and DB name during a CMI update action instead of only querying by OBID in all databases in scope or session.

To enable this feature you have to set the following configuration variable within PDM\_CONFIG (config.cfg):

**CMI\_USE\_DB\_NAME = "ON"**

During the CMI Read action the database names of all objects will be read and written to the Object Manager file. During update it is now possible to query for objects exactly within the database they are saved in.

Additionally you may customize some or all of the following messages to your own needs:

**x3GetRelevRelViaObidDb**

**x3CusGetRlvRlViaObidDb**

**x3GetPartByAttrsAndDb**

**x3GetModPosViaObidDb**

**x3GetPrdPartViaObidDb**

You can find these messages within x3Custom.mth

***Customization messages***

Messages contain input parameter database name which is the Teamcenter Enterprise attribute "CurDbName" of the object.

---

Rest of code should be the same as in already existing messages "Get...ViaObid". But instead of "QueryWhere" or "QueryDBObject" the database specific message "QueryWhereByDbName" is used.

```
class message x0WkBnch:x3GetRelevRelViaObidDb (  
    input :      string      ClassName      ::  
    input :      string      RelOBID       ::  
    input : NULL string      DataBaseName  ::  
    output:      ObjectPtr *Relation      ::  
    output:      integer     *mfail) code
```

```
class message x0WkBnch:x3CusGetRlvRlViaObidDb (  
    input :      string      ClassName      ::  
    input :      string      RelOBID       ::  
    input : NULL string      RelDbName     ::  
    input : NULL ObjectPtr  ctItemObj     ::  
    output:      ObjectPtr *Relation      ::  
    output:      integer     *mfail) code
```

```
class message x0WkBnch:x3GetPartByAttrsAndDb (  
    input :      string      ClassName      ::  
    input : NULL string      databaseName  ::  
    input : NULL string      partOBID     ::  
    input : NULL string      partNumber   ::  
    input : NULL string      partRevision ::  
    input :      string      lastSeq      ::  
    output:      ObjectPtr  *subassObj    ::  
    output:      integer     *mfail) code
```

```
class message x0WkBnch:x3GetModPosViaObidDb (  
    input :      string      className     ::  
    input :      string      ModPosClass  ::  
    input :      string      Obid         ::  
    input : NULL string      DataBaseName  ::  
    output:      ObjectPtr  *ModPosObj    ::  
    output:      integer     *mfail) code
```

```
message ProdBI:x3GetPrdPartViaObidDb(  
    input :      string      className     ::  
    input :      string      partOBID     ::  
    input : NULL string      partDbName   ::  
    input : NULL NvSet      customData    ::
```

---

```
output:      ObjectPtr      *partObj      ::
output:      integer        *mfail) code
```

---

## Read Reference/Sheet Information from CATIA V5 Drawings

### *Reference Documents for Drawings*

Via the CATIA V5 config-variable `CMI_REFERENCE_OF_DRAWING = "ON"` all "Reference documents" of a CATDrawing are stored in the CatDrawing-model-info of the exchange-file. You can use these links to attach the drawing to a relevant folder. Customization can get this information with the method `x3SetCusAttrModInf`. Within this method you have to call:

```
"dstat= nvsGet(ModelInfos, " SOURCE_PARTNUMBERS ", &ModelData);"
to get the reference part numbers.
```

```
"dstat= nvsGet(ModelInfos, " SOURCE_DOCS ", &ModelData);"
to get the reference documents.
```

```
"dstat= nvsGet(ModelInfos, " SOURCE_OBIDS ", &ModelData);"
to get the reference part OBIDS.
```

### *Sheet-Information for Drawings*

Set `"CMI_CALC_SHEETS=ON"` in the CATIA V5 Environment.

Use `"dstat= nvsGet(ModelInfos, "SHEETS", &ModelData);"` to get all information on the sheets.

---

## Enhanced 4D-Navigator Integration

New custom-point: `x3Inflate4DNavName`

Handles the `x3InflateExMapName` message for the `xOCTFile` class. Message type is an Object Message. File system name within exchange map is identical to the `RelativePath` of the `xOCTFile` object.

As standard the `RelativePathAttr` is used.

New custom-point: `x3GetSceneFile`

This method interacts with the user to get the Scene file describing the situation in the 4D-Navigator. In the parameter `fileContents` the content of the file is handled back to the calling method.

Notice: You can also load the Scene file from a pre-defined directory (like the Scene file creation)

As standard the 4D-Nav-Exchangemap is used as relative path, with an attached `"/NEW_SCENE.wrl"`.

---

## CATIA project environment support

New custom point `x3SetCatiaProject`.

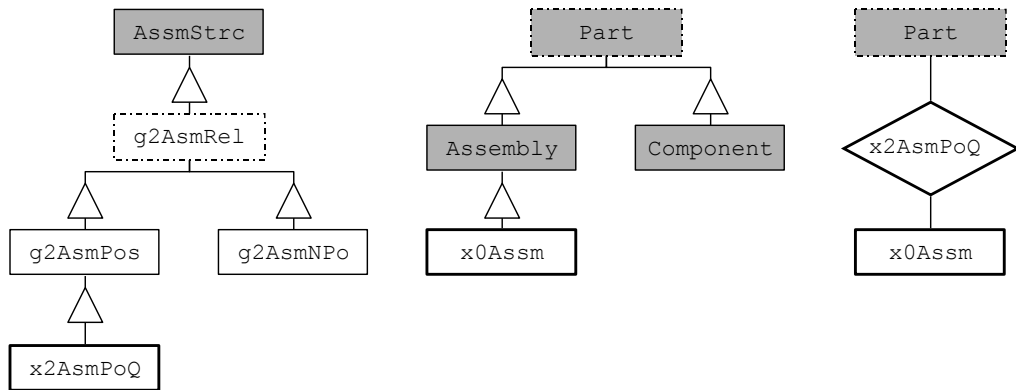
Handles the `x3SetCatiaProject` message for the `xOCTFile` class.

Set a new attribute to a Catia model, when it's updated, created and saved in Catia.

---

## Support of Teamcenter “Quantity”

In standard Teamcenter Enterprise, the “AssmStrc” relation class carries the attribute “Quantity”. A part can be used multiple times in an assembly, according to the value set for the “Quantity”. In order to support the multiple usage of parts within an assembly, the “x2AsmPoQ” relationship carries a number of transformation matrixes. The picture below shows the extensions within the data model:



**Figure 7: Data model extensions for support of ‘Quantity’**

Customizations may utilize the `x0Assm` class in order to create `x2AsmPoQ` relationships. The `x2AsmPoQ` relationship stores the amount of ‘Quantity’ transformation matrixes within a ‘FullTable’ attribute `x0TrafoList`. Each single transformation matrix can be identified by an index passed to the CMI APIs `g3GetTrafo` and `g3UpdateTrafo` (see description below).

In order to ensure the consistency between the value of the ‘Quantity’ attribute and the number of transformation matrixes, the ‘Quantity’ mustn’t be edited manually! This means that all functionality like ‘Update Relationship’ e.g. has to be altered in a way which prevents the user from editing the ‘Quantity’ attribute. If the ‘Quantity’ is raised, a new unity matrix is added by default.

For each item of Quantity an instance of the Model geometry will be shown in CATIA, and a transformation matrix has to be kept in the database. Therefore you should limit the Quantity to a reasonable number, eg. in user dialog validation. In the standard implementation of CMI this is controlled by setting the variable `GCVMI_MAX_QUANTITY` in your `config.cfg` file.

Example:

**SET GCVMI\_MAX\_QUANTITY “20”;**

## Working with CATIA V5 Released Cache

CMI supports the use of CGR-files in the released cache of CATIA V5. For this purpose the Teamcenter customization has to store the CGR-files of Catia-models in a specific CGR-Vault. During "To Catia" these CGR-files are copied to the Released Cache instead of the Catia-models to the exchange-map. In CATIA V5 the CGR-files are loaded in visualization mode. For each file in the workbench CMI decides via customizable methods whether to copy the standard file or the CGR-file.



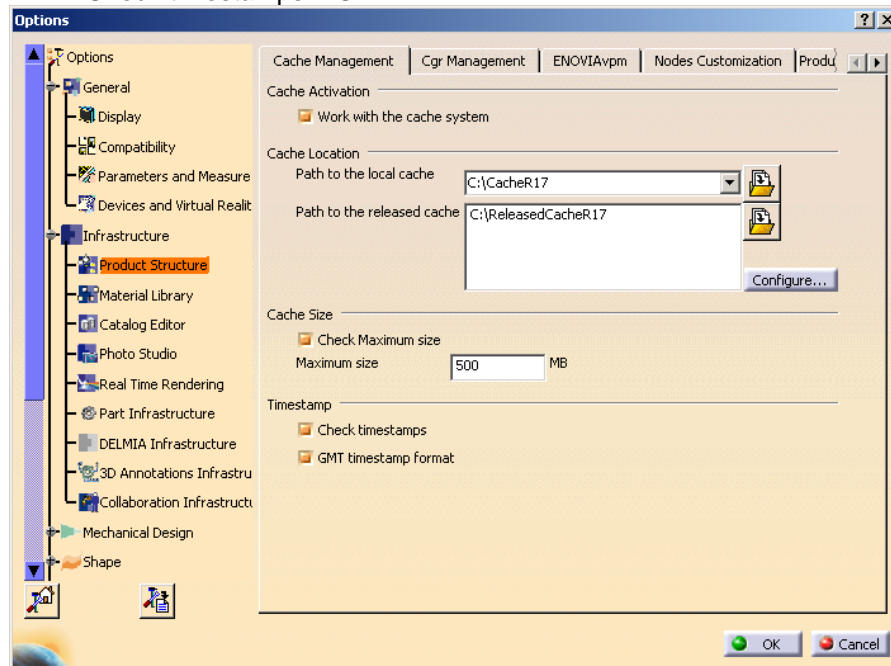
Options -> Change Preferences -> CMI Preference.

Set "Transfer CGR-File to CATIA V5" to "Only CGR"/ "CGR + geometry"



In CATIA you have to use the following settings

- Work with the cache system = ON
- Path to the local cache = <path to local cache>
- Path to the released cache = <at least one path to released cache>
- Check timestamps = ON



These are the necessary preconditions to copy a CGR-file:

Work with the cache system in CATIA V5 is enabled.

The Released cache is set and exists.

The CMI-preference "Transfer CGR-File to CATIA V5" is set to "Only CGR" or "CGR + geometry".

The Catia-model is checked in.

In the CGR-Vault exists the CGR-file with following name: "<model-name>.cgr" where <model-name> is the filename of the Catia-model.

Customization may change the behavior of the "CGR-search". Then you should customize the methods x3UseCgrFile and x3GetCgrFilePaths. x3UseCgrFile decides whether to use the CGR-file; x3GetCgrFilePaths delivers the relevant file-path of the CGR-file.

---

If the customization needs a Teamcenter-class to register the CGR-file, it is recommended to use a subclass of the CMI-class x0V5CGR. ("CATIA V5 CGR for Cache File").

**Configuration:**

1) Settings in \$PDM\_CONFIG:

```
# host where the CGR files are located (optional)
# if not set CGR files has to be located in the same vault location like the
# corresponding CATPart
set CMI_CGR_HOST "<CGR_HOST>";
```

```
# path to the vault location on the CMI_CGR_HOST (optional)
# if not set CGR files has to be located in the same vault location like the
# corresponding CATPart
set CMI_CGR_VAULTLOC_PATH "<Path to the CGR Vault>";
```

2) Settings during startup of CATIA V5

```
# enable CMI released cache functionality
set CMI_USERRELEASEDCACHE=ON
```

```
# use a special released cache directory out of the list in CATIA
# this setting is optional; if not set use the first released cache in list
set CMI_RELEASEDCACHEDIR=<Path to released cache>
```

```
# enable the "Get original Geometry" button
set CMI_ENABLE_CMIGETORIGGEOCMD=ON
```

```
# Optional setting
# use temporary CATIA components which contain the related CGR as shape
# representation,
# instead of the original CATPart. (see load structure to CATIA)
CMI_CREATETEMPCGRCOMP=ON
```

3) Settings in CATIA no longer needed to work with released Cache

Earlier releases of CMI didn't support the CATIA setting "Check Timestamp ON". So it was necessary to remove the CGR files of the released Cache dir to make sure that the correct version was loaded. Since CMI 8.9 the setting CMI\_CLEANRELEASEDCACHE=ON is no longer needed and should not be used.

---

## CATProcess customization

### *customization methods in TeamCenter*

The message g3IsProcessElement is used to detect a process element. The method is called after a drop of a Part in the Workbench. The default method expands to an x0CatPrc data item which is the storage class for a CATProcess file.

```
class message Part:g3IsProcessElement(
    input : ObjectPtr    thisObj      ::
    input : ObjectPtr    WorkBnchObj  ::
    output: boolean      *bIsProcessElement ::
    output: integer      *mfail) code
```

---

The message `g3IsProductView` is called if a process element expands its children. The Catia V5 structure under the product view is added in the product view of the CATProcess in CATIA V5.

```
class message Part:g3IsProductView(  
    input : ObjectPtr    thisObj        ::  
    input : ObjectPtr    WorkBnchObj    ::  
    output: boolean      *bIsProductView ::  
    output: integer      *mfail) code
```

The message `g3IsResourceView` is called if a process element expands its children. The Catia V5 structure under the resource view is added in the resource view of the CATProcess in CATIA V5.

```
class message Part:g3IsResourceView(  
    input : ObjectPtr    thisObj        ::  
    input : ObjectPtr    WorkBnchObj    ::  
    output: boolean      *bIsResourceView ::  
    output: integer      *mfail) code
```

The message `g3GetProcessFiles` expands the Document and returns the Process data items, attached to the Document.

```
message ProdBI:g3GetProcessFiles (  
    input :      ObjectPtr    thisPtr        ::  
    input :      ObjectPtr    WorkBnchObj    ::  
    update:      SetOfObjects *processesOfPart ::  
    output:      integer      *mfail) code
```

---

## Viewer support

### *Prerequisites*

A custom service has to provide viewing files like JT or CGR. The viewing files are needed for the following geometry types:

CATPart

model

CGR (if you use JT)

CMIArchive (You need one viewing file for the content of the CMIArchive)

To create the viewing files you can use the following utilities:

**CGR:** DassaultSystems - CATDMUUtil (Part of CATIA V5)

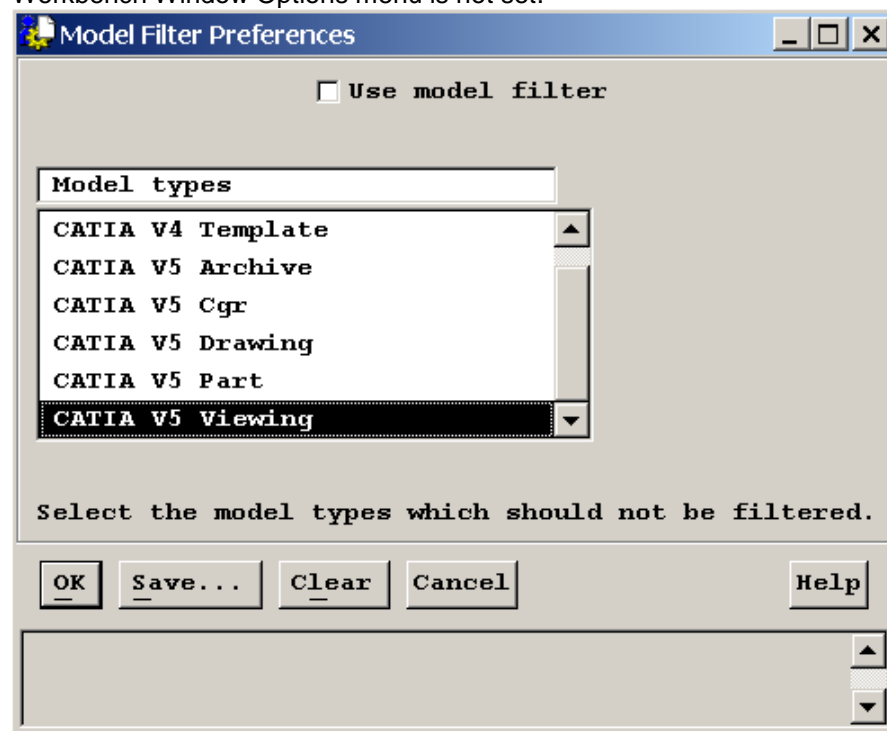
**JT:** T-Systems - ComFox Translator

CMI doesn't use viewing files for the CATProducts in the Assembly structure. The structure is created on the fly.

### *Features*

This functionality reads the CMI Workbench content, gets the viewing files (from customisation) and builds up a format file to open viewer.

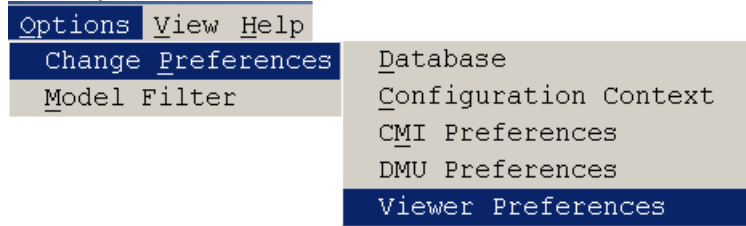
The CMI Model Filter is extended with a Viewing File type (**CmiCatiaV5ViewType**) which will be filtered out by default even if the "Model filter preferences" in the CMI Workbench Window Options menu is not set.



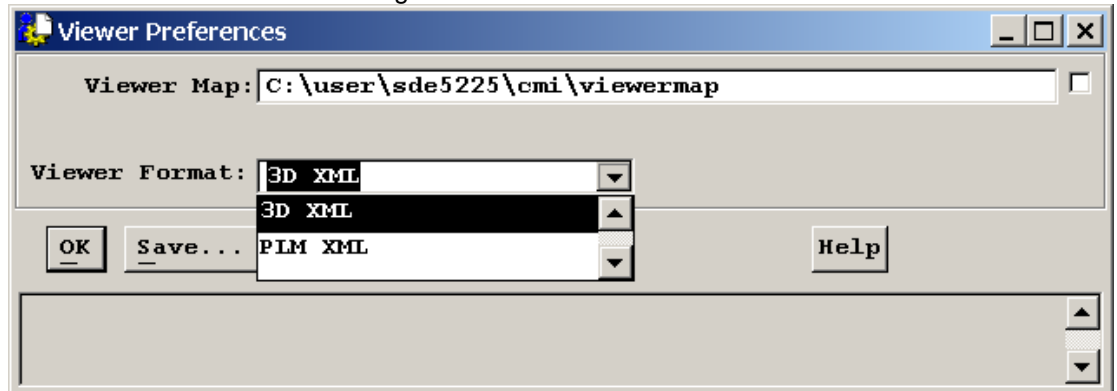
## Configuration

The directory and the viewer format (3dxml, pdmxml, etc.) are configurable in CMI Viewer Preferences.

New Viewer Configuration in CMI Workbench window Options->Change Preferences Menu option.



The Viewer Map directory defines the local Directory which is used for the storage of the viewer file and the related viewing files.



In \$PDM\_CONFIG file there are the following configuration vars:

If a start script is used to launch the viewer on Windows, the variable CMI\_3DXML\_STARTUP\_SYNC\_WIN must set to ON. This forces the synchronous start of the viewer batch command in Synchronous mode because in the asynchronous mode the execution of .BAT files fail on Windows.

For 3DXML with IE and plug-in:

```
set CMI_3DXML_STARTUP "<path>\StartIE.bat";
set CMI_3DXML_STARTUP_SYNC_WIN "ON";
or for the direct call of the executable:
set CMI_3DXML_STARTUP "<path>\code\bin\3DXMLPlayer.exe";
```

For PLMXML:

```
set CMI_PLMXML_STARTUP "<path>JT2 GoLauncher.exe";
set CMI_PLMXML_STARTUP_SYNC_WIN "ON";
for batch startup on Windows
set CMI_VIEWER_ENCODING to the charset used in th plmxml – file,
if it is different from "ISO-8859-1".
```

Class Constants in x0WkBnch

x4ViewDocToJtRelClassC	= Attach;
x4ViewDocToJtRelShipC	= DataItemsAttachedToBusItem;
x4ViewModToJtRelClassC	= GenDeriv;
x4ViewModToJtRelShipC	= GenDerivSourceOfItem;
x4ViewJtModelClassC	= IndepBin;

## Customization

You can define the way JTs or CGRs are retrieved by overriding the following customisation messages.

For JTs attached to the Parts Document, this is:

```

object message x0CTItem:x3GetViewerFileSet (
    input :      ObjectPtr      partRep      ::
    input :      ObjectPtr      partObj      ::
    input :      string         viewFormat   ::
    input :      ObjectPtr      workBench    ::
    output :     SetOfStrings    *pHosts     ::
    output :     SetOfStrings    *pUsers     ::
    output :     SetOfStrings    *pFullPaths ::
    output :     SetOfStrings    *pNewFileNames ::
    output :     integer         *iStatus    ::
    output :     integer         *mfail      ::
) code

```

More than one JTs can be attached to a document.

If the JTs are linked to the CATParts (Has Visualization), it is:

```

message x0ModRep:x3GetViewerFile(
    input :      ObjectPtr      modelRep     ::
    input :      ObjectPtr      modelObj     ::
    input :      string         viewFormat   ::
    input :      ObjectPtr      workBench    ::
    output:      string         *sHost       ::
    output:      string         *sUser       ::
    output:      string         *sFullPath   ::
    output:      string         *sNewFileName ::
    output:      integer        *iStatus     ::
    output:      integer        *mfail) code

```

Only one JT is expected for any one CATPart.

PartRep/modelRep is the Representation Object for the Object message.

PartObj/modelObj is the related Persistent object.

viewFormat is the viewing format from the Preference, by default **x13DXml (3D XML)** and **x1PlmXml (PLM XML)** are supported by CMI.

workBench is the Workbench Object.

the Return values are sHost, sUser, sFullPath, which defines the Source location of the viewer file.

sNewFileName is the File name to be used in the viewer map.

Return of iStatus == 1 in **x0CTItem:x3GetViewerFile** defines, not to search the models for Representation.

Standard behaviour in CMI for **x0CTItem:x3GetViewerFile**:

3DXML:

Get all documents, then get all models, the first model with the type

**CmiCatiaV5ViewType** (Should be CGR) will be returned. iStatus will return 1, if cgr is found, →no models will be searched.

PLMXML:

Get all documents then expand through **x4ViewDocToJtRelClassC and**

**x4ViewDocToJtRelShipC to x4ViewJtModelClassC.**

Standard behaviour in CMI for **x0ModRep:x3GetViewerFile**:

3DXML:

Get the Cgr in the same manner as the Released Cache Concept from

**CMI\_CGR\_VAULTLOC\_PATH and CMI\_CGR\_HOST.**

PLMXML:

Expand through **x4ViewModToJtRelClassC and x4ViewModToJtRelShipC to x4ViewJtModelClassC.**

---

## ***Display User Data***

The following methods can be overridden to display custom Teamcenter properties in VisView/VisMockup:

For a Representation / JT:

```
g0RepItm:x3GetInfoForViewFile (
    input :      ObjectPtr    thisRep          ::
    input :      ObjectPtr    thisObj          ::
    input :      string       FileName         ::
    output:      NVSET        *nvAdditionalAttributes  ::
    output:      integer      *mfail)
```

For a Part:

```
x0CTItem:x3GetInfoForViewPart (
    input :      ObjectPtr    partRep          ::
    input :      ObjectPtr    partObj          ::
    output:      NVSET        *nvAdditionalAttributes  ::
    output:      integer      *mfail)
```

The properties added to `nvsAdditionalAttributes` will be shown as `usr data/user` values in VisView.

## **Design Table Support**

### ***Features***

Design Tables are managed in Teamcenter by Synchronize of Catia V5.

### ***Configuration***

In `$PDM_CONFIG` file there is the following configuration var:

```
CMI_DESIGN_TABLES
```

If this variable is set to "ON", during *To Catia* all Catia Model/Products are expanded for Design Tables and the relevant Design Tables are transferred to Catia V5. There is a performance impact.

Class Constants

**x0WkBnch.x4CreateCatTxtC = x0CatTxt; Class for csv design tables**

**x0WkBnch.x4CreateCatExcC = x0CatExc; Class for Excel™ design tables**

**x0WkBnch.x4DesTabRelClassC = x2DepDes; Dependency relation**

**x0WkBnch.x4DesTabRelShipClassC = DesTabDependentOnItems;  
Relationship**

### ***Customizable Methods***

```
define message x3GetDesignTables (
    input :      ObjectPtr    catiaObj          ::
    input :      NULL ObjectPtr    partObj          ::
    output :      SetOfObjects *desTabObjs       ::
    output :      SetOfStrings *dtFileNames     ::
    output :      integer      *mfail);
```

```

define message x3IsLinkedWithDesTab (
  input  :      ObjectPtr      modelObj      ::
  input  :      ObjectPtr      desTabObj     ::
  output :      integer        *bLinked     ::
  output :      integer        *mfail);

define message x3LinkWithDesTab (
  input  :      ObjectPtr      catiaObj      ::
  input  : NULL NvSet          ModelInfos    ::
  input  :      ObjectPtr      desTabObj     ::
  input  : NULL string         sPartClassName ::
  input  : NULL string         sPartOBID     ::
  input  : NULL string         sPartDbName   ::
  output :      ObjectPtr      *relObject    ::
  update :      SetOfStrings   *vStats      ::
  output :      integer        *mfail);

define message x3RemoveDesTab (
  input  :      ObjectPtr      modelObj      ::
  input  : NULL NvSet          ModelInfos    ::
  input  :      ObjectPtr      desTabObj     ::
  input  : NULL string         sPartClassName ::
  input  : NULL string         sPartOBID     ::
  input  : NULL string         sPartDbName   ::
  update :      SetOfStrings   *vStats      ::
  output :      integer        *mfail);

```

These methods are used to get related Design Tables for a Catia Model/Product or to relate a Design Table to a Catia Model/Product.

---

## MML Support

### *Features*

MMLs can be managed in Teamcenter.

### *Installation*

This use case requires additional CMI CATIA library:

```

Windows:  CMIExt.dll
Solaris:  libCMIExt.so
HP-UX:    libCMIExt.sl
AIX:     libCMIExt.a
IRIX:    libCMIExt.so

```

This library is not part of the CMI product. Please contact [cmi\\_support@t-systems.com](mailto:cmi_support@t-systems.com) to get more information.

Copy this library to the binary path of the CMI CATIA installation, e.g.  
 <CMI\_Installation\_Dir>\CMICATV5\_R16\_91V10\intel\_a\code\bin\

### *Configuration*

Teamcenter setting in \$PDM\_CONFIG:

Optional:

CMIDEPENDENT\_TYPES

Default: CCP

Set this environment if you want to use other link types than CCP.

Settings during startup of CATIA V5

CMI\_GETPOINTEDDOCUMENTS=ON

Default: OFF

Set this environment to "ON" to provide the information about referenced CATIA files to Teamcenter.

CMI\_ENABLE\_CMIEXTERNALDOCCMD=ON

Default: OFF

Set this environment to "ON" to enable the "Get Referenced Geometries" button in CATIA V5.

CMI\_ENABLE\_CMIGETDEPBYPYCMD=ON

Default=OFF

Set this environment to "ON" to enable the "Get Depended By Geometries" button in CATIA V5 (Teamcenter Customizing is required due to version ambiguities.)



### **Customizable Methods**

Manage dependencies during Update / Synchronize / Create / SaveAs:

```
message xOCTFile:x3UpdateDepV5 (  
    input :      ObjectPtr      this      ::  
    input : NULL NvSet          ModelInfos ::  
    update:      SetOfStrings   *vStats  ::  
    output:      integer        *mfail);
```

Get Referenced Geometries:

```
message xOCTFile:x3DependsOnFiles (  
    input :      ObjectPtr      this      ::  
    output :      SetOfObjects   *depFiles ::  
    output :      integer        *mfail);
```

Get Depended By Geometries:

```
message xOCTFile:x3IsDependedOnByFiles (  
    input :      ObjectPtr      this      ::  
    output :      SetOfObjects   *depOnByFiles ::  
    output :      integer        *mfail);
```

When you are using "Get Depended By Geometries", you have to make sure your customizing of x3IsDependedOnByFiles filters the depended on by files. It is not possible to load multiple versions of one file into CATIA V5.

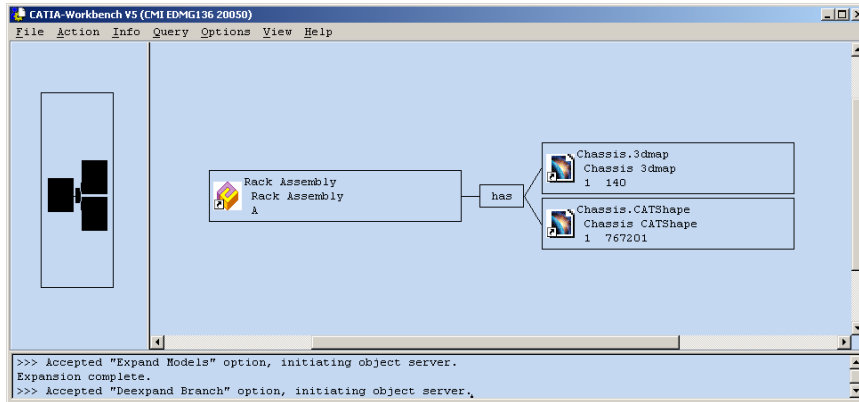
## Representation Formats in CATIA V5

The CATIA V5 module can be configured to support CATIA V5 shape representation files in a generic way. CATIA V5 Representation files can be used in a product structure and are handled in a similar fashion to V4 Models.

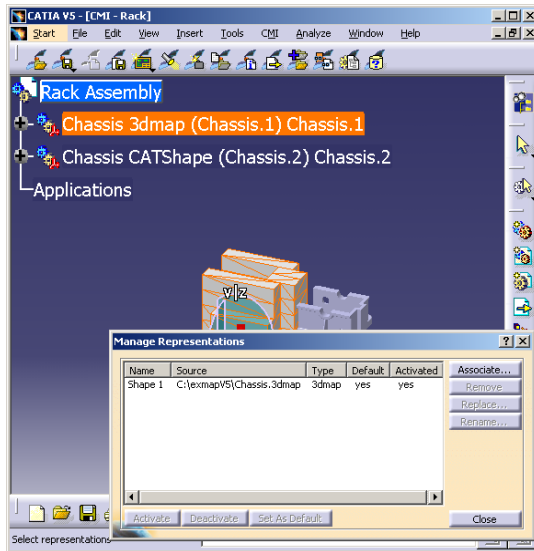
To enable support for specific Shape Representations, an administrator can set the following variable in the Catia V5 environment:

```
CMI_REP_FORMATS={ 3dmap } { CATShape }
```

The legitimate types can be found in the Manage Representations dialog in CATIAV5 (in this example 3dmap and CATShape formats will be handled by the CMI Catia Module.)



**Representation formats in CATIA V5 (CATIA Workbench)**



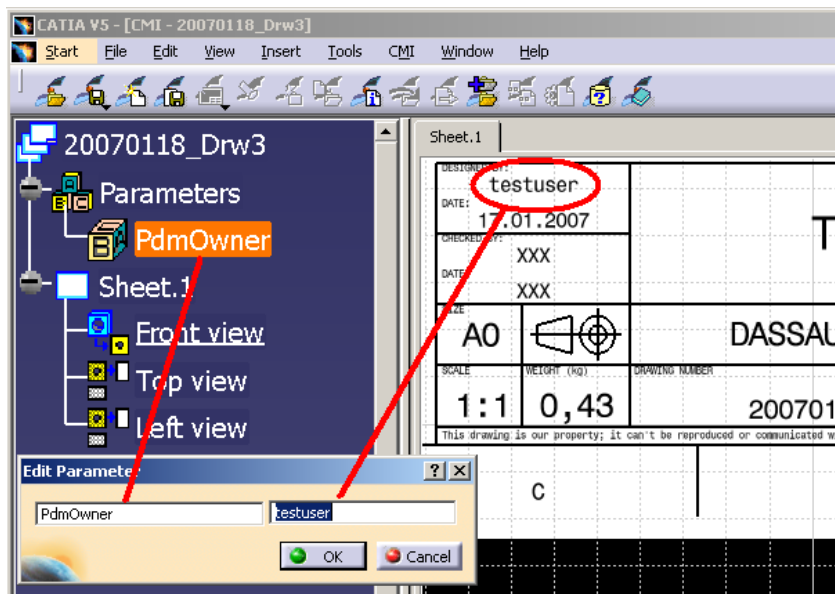
**Representation formats in CATIA V5 (CATIA V5)**

```
class message x0WkBnch:x3GetClassForCatRep (
    input :      string      sClassName      ::
    input :      string      sOrigCatRepClassName  ::
    input : NULL  string      sOrigFileName     ::
    input : NULL  string      sOrigPartNumber   ::
    output:      string      *sNewCatRepClassName  ::
    output:      integer     *mfail) code
```

Override this message to provide dedicated classes for individual Catia V5 Shape representations. The default Class name returned is x0CatRep. If you introduce your own Representation classes, they should derive from this base class.

## Automatic update of CATDrawing title blocks with Teamcenter data

Customizing option to fill in Drawing title block from Teamcenter data. Implement the customizing message `x3GetParamForFileV5` to define Teamcenter data that should be transferred to Knowledgeware Parameters in Catia V5.



Override the Customizing message `x3GetParamForFileV5` in Teamcenter to define the attributes for Catia:

```
class message g0GenBin:x3GetParamForFileV5 (  
    input : string          sclassname      ::  
    input : ObjectPtr      thisObj        ::  
    output: SetOfStrings   *vAttributeList ::  
    output: SetOfStrings   *vValueList    ::  
    output: integer        *mfail);
```

`sclassname` is the class name of the Drawing object.

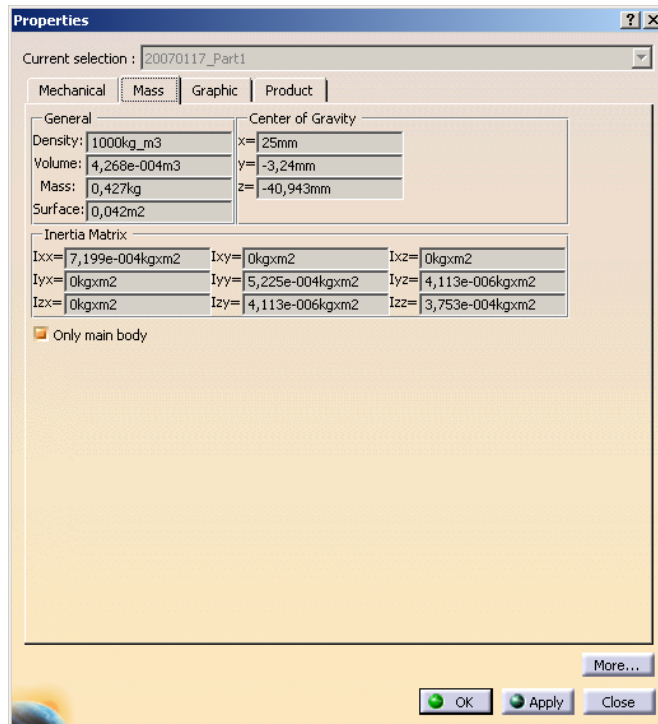
`thisObj` is the object itself

`vAttributeList` is the list of parameter names

`vValueList` is the list of related values for the attributes. Both lists must be the same length.

## Transfer of weight properties (inertia) from Catia V5 to Teamcenter

Inertia properties – eg. Mass – can be read from Catia V5 and stored in Teamcenter.



In order to read these properties from Catia, the following variable has to be set in the Catia V5 environment:

**set CMI\_READ\_INERTIA=ON** (Inertias from CATProducts and CATParts are sent to Teamcenter.)

**set CMI\_READ\_INERTIA=ONLY\_CATPART** (Only the inertias from the CATParts will sent to Catia. This may improve the performance especially in case of large assemblies.)

The inertias can be confined to the main bodies of the CATParts by a further configuration variable:

**set CMI\_CONFINE\_INERTIA\_TO\_MAINBODIES=ON**

The properties can be used in Teamcenter by overriding the customizing method `g0GenBin:x3SetCusAttrModInf` for the CATProduct and/or the CATPart. The customizing method is called in the course of Update, Create and similar actions. CMI itself does not use this information, but you can store it in custom attributes.

```
message g0GenBin:x3SetCusAttrModInf (
    update: ObjectPtr  this      ::
    input  : NvSet     ModelInfos ::
    output: integer    *mfail) code
```

The following named values are provided to the API in the NvSet ModelInfos:

mass  
INERTIA\_MASS

position of the center of gravity  
INERTIA\_POSITION\_0  
INERTIA\_POSITION\_1  
INERTIA\_POSITION\_2

inertia matrix  
INERTIA\_MATRIX\_0  
INERTIA\_MATRIX\_1  
INERTIA\_MATRIX\_2

```

INERTIA_MATRIX_3
INERTIA_MATRIX_4
INERTIA_MATRIX_5
INERTIA_MATRIX_6
INERTIA_MATRIX_7
INERTIA_MATRIX_8

components of principal axes
INERTIA_COMPONENTS_0
INERTIA_COMPONENTS_1
INERTIA_COMPONENTS_2

INERTIA_COMPONENTS_3
INERTIA_COMPONENTS_4
INERTIA_COMPONENTS_5
INERTIA_COMPONENTS_6
INERTIA_COMPONENTS_7
INERTIA_COMPONENTS_8

principal moments values
{INERTIA_VALUES_0} 1.0341075818066402e-004
{INERTIA_VALUES_1} 1.0341075818066402e-004
{INERTIA_VALUES_2} 1.9634954084936208e-004

{INERTIA_VOLUME} 1.5707963267948968e-004
{INERTIA_DENSITY} 1.0000000000000000e+003
{INERTIA_AREA} 2.1991148575128551e-002

```

---

## Set Bom-Type of new Catia-files by Teamcenter-customization

In the Synchronize-dialog all new Catia-files are presented with a Bom-Type(Bom/Non-Bom/Not Set). This Bom-Type is initialized by a default and can be changed by the user. Depending on the Catia-V5-config-variable CMI\_GET\_BOMTYPE\_FROM\_TC = ON the values for the Bom-Type are fetched from Teamcenter. With Standard-CMI the bomType is set to "", i.e. the Bom-Type in Catia remains unchanged.

If you want to customize the setting of the Bom-Type you have to override:

```

message x0WkBnch:x3GetBomType(
  input:      string      strClassname      ::
  input:      NVSET       nvsObjInfos       ::
  input:  NULL NVSET      nvsUserDefInfos   ::
  input:  NULL NVSET      nvsParentObjInfos  ::
  input:  NULL NVSET      nvsParentUserDefInfos::
  output:     string      *bomType          ::
  output:     integer *   mfail) code

```

Allowed values for the return-value bomType:

```

"NOT_SET"
"BOM"
"NOT_BOM"

```

For all other values the Bom-Type in Catia remains unchanged.

The NVSET nvsObjInfos contains the following Catia attributes for the new Catia-File, if available:

```

PARTNUMBER
FILENAME

```

---

## NOMENCLATURE

The NVSET `nvsUserDefInfos` contains the User-Defined-attributes for the new Catia-File, if existent.

The NVSET `nvsParentObjInfos` contains the following attributes for the Father-Object, ie. the Assembly, if existent:

```
PARTNUMBER
NOMENCLATURE
DB_NAME
OBID
CLASSNAME
```

The NVSET `nvsParentUserDefInfos` contains the User-Defined-attributes for the Father-Object, if existent.

---

## Custom Expand in the CMI CATIA Workbench

The method for the multi-level expand in the CMI CATIA workbench was extended by an optional expand information string. If no special expand information is given, then the default expansion is being performed. Giving an expand information string to the new multi-level expand method will pass this string to the lower-level custom methods that define which sub-parts should be returned for a given assembly.

With this CMI extension customization can define one or more new "custom expand" context menu options in the CMI CATIA workbench context menu. The custom methods that are called by these menu options can call the extended CMI multi-level expand method, passing it a specific expand information string.

The expand information string is stored in the CMI-Filter-preference. At the end of the command this attribute is reset in the filter-preference.

If you want to customize the Expand of an Assembly-structure with the catia-models in the WorkBench you have to customize the following methods:

```
define trusted external nondisplaying nonprompting class message
```

```
g3GetAllModelsOfDocCus (
    input :      string      classname  ::
    input : NULL string      expandInfo  ::
    input :      ObjectPtr   this       ::
    input :      ObjectPtr   WorkBnchObj::
    output:      SetOfObjects *modelSet  ::
    output:      SetOfObjects *posSet    ::
    output:      integer     *mfail);
```

`g3GetAllModelsOfDocCus` delivers all models and trafo-relations(if existent) of a given folder-object. Depending on the `expandInfo`, you can call your own method.

```
define trusted external class message g3GetPartsInAssSetCus (
    input :      string      classname  ::
    input : NULL string      expandInfo  ::
    input :      SetOfObjects assObjSet  ::
    output:      SetOfObjects *partsInAss::
    output:      SetOfObjects *partRels  ::
    output:      integer     *mfail);
```

---

`g3GetPartsInAssSetCus` delivers all child-parts and `AssmStrc`-relations for a given part-object. Depending on the `expandInfo`, you can call your own method. This method is called for the setbased expand (standard CMI-behaviour, `g3SetBasedAllowed = TRUE`)

```
define trusted external nondisplaying nonprompting class message
g3GetPartsInAssemblyCu (
    input :      string      classname  ::
    input : NULL string      expandInfo  ::
    input :      ObjectPtr   assObj     ::
    output:      SetOfObjects *partsInAss::
    output:      SetOfObjects *partRels ::
    output:      integer     *mfail);
```

`g3GetPartsInAssemblyCu` delivers all child-parts and `AssmStrc`-relations for a given part-object. Depending on the `expandInfo`, you can call your own method. This method is called for the non-setbased expand(`g3SetBasedAllowed = FALSE`).

### **Customization example**

You have a new menu item *MyMultiLevelExpand* and you want to filter the assembly-structure by your own behaviour:

Define the method and attach it.

```
define message MyMultiLevelExpand like SelectedItemMsg;

attach object message MyMultiLevelExpand to g0GenItm in server
cussvr;
```

Attach your custom expand to the popup menu of the Assembly in the Workbench, below the default Expand

```
define option MyMultiLevelExpandOpt
    using message MyMultiLevelExpand
    with relation g2GnItDp
    with relationship useslowerGI;
display MyMultiLevelExpandOpt as "My Expand Multiple Levels";

attach option MyMultiLevelExpandOpt to GITreePopupEiwOptL
after GIuseslowermultiGIEiwOpt;
```

write the following method code:

```
message g0GenItm: MyMultiLevelExpand (
    update: ObjectPtr   this  ::
    output: integer     *mfail) code
{
    string expandInfo = NULL;
    *mfail= USC_OKAY;
    dstat= uiDeactivateWindow ();
    /* ----- */
    /* Call internal part of method */
    /* ----- */
    string expandInfo = "myPartExpand";
    dstat= g3ExpandBranchMultiInt (this, expandInfo, mfail);
CLEANUP:
EXIT:
    uiActivateBrowserWindow ();
    return (dstat);
```

```
}
```

Now override g3GetPartsInAssSetCus to make your custom expand return only a special type of assemblies:

```
O:attach class message g3GetPartsInAssSetCus to g0GenWB in server  
cussvr;
```

```
class message g0GenWB:g3GetPartsInAssSetCus (  
    input :      string          classname  ::  
    input : NULL string          expandInfo ::  
    input :      SetOfObjects    assObjSet  ::  
    output:      SetOfObjects    *partsInAss::  
    output:      SetOfObjects    *partRels  ::  
    output:      integer         *mfail) code  
{  
    *mfail= USC_OKAY;  
  
    dstat= g3GetPartsInAssSet (classname, assObjSet,  
                               partsInAss, partRels, mfail);  
  
    if (expandInfo && nlsStrCmp(expandInfo, "myPartExpand") == 0))  
    {  
        /* ----- */  
        /* TODO: Filter partsInAss/partRels */  
        /* according to the special requirements of your */  
        /* custom Expand */  
        /* ----- */  
  
        ...  
    }  
  
    ...  
}
```

---

## Supply Attributes from Catia to Teamcenter and back

CMI provides a set of APIs which allows the transfer of attributes from CATIA to Teamcenter and back. This functions can only be used within a customer specific CATIA application (CAA or Automation / VBA). Customizing on Teamcenter is required.

### *Supply Attributes from Catia to Teamcenter*

CAA-Function:

```
HRESULT CMIGetPartAttributeFromTC(const CATIProduct_var spProduct,  
                                  const CATString &sAttribute,  
                                  CATString &sValue,  
                                  CATString &sReturnCode);
```

Get Attribute from Teamcenter.

This Function calls x3GetAttValueForObject in Teamcenter.

In:	spProduct	CATProduct to examine in Teamcenter
	sAttribute	sAttributeName
Out:	sValue	sAttributeValue
	sReturnCode	Message from Teamcenter

CAA-Function:

```
HRESULT CMIGetDrawingAttributeFromTC(const CATDocument *pDocument,
```

---

```
const CATString &sAttribute,  
CATString &sValue,  
CATString &sReturnCode);
```

Get Attribute from Teamcenter.

This Function calls *x3GetAttValueForObject* in Teamcenter.

In:	pDocument	CATDrawing to examine in Teamcenter
	sAttribute	sAttributeName
Out:	sValue	sAttributeValue
	sReturnCode	Message from Teamcenter

### ***Supply Attributes from Teamcenter to Catia***

CAA-Function:

```
HRESULT CMISupplyPartAttributeToTC (const CATIPProduct_var spProduct,  
const CATString &sAttribute,  
CATString &sValue,  
CATString &sReturnCode);
```

Supply Attribute to Teamcenter:

This Function calls *x3SupplyAttValueForObject* in Teamcenter.

In:	spProduct	CATProduct to examine in Teamcenter
	sAttribute	sAttributeName
	sValue	sAttributeValue
Out	sReturnCode	Message from Teamcenter

CAA-Function:

```
HRESULT CMISupplyDrawingAttributeToTC (const CATDocument *pDocument,  
const CATString &sAttribute,  
CATString &sValue,  
CATString &sReturnCode);
```

Get Attribute from Teamcenter:

This Function calls *x3SupplyAttValueForObject* in Teamcenter.

In:	pDocument	CATDrawing to examine in Teamcenter
	sAttribute	sAttributeName
Out:	sValue	sAttributeValue
	sReturnCode	Message from Teamcenter

---

## **Supply File from Catia to Teamcenter**

CMI provides a set of APIs which transfer a file from CATIA to Teamcenter. This functions can only be used within a customer specific CATIA application (CAA or Automation). Customizing on Teamcenter is required. (Standard Teamcenter copies the file to the Work-Location of the correspondent product/drawing)

### ***Supply File from Catia to Teamcenter***

CAA-Function:

```
HRESULT CMISupplyAdditionalFileForPrdToTC ( const CATIPProduct_var spProduct,  
const CATString &sFullPath,  
const CATString &sInfoForTC,  
CATBoolean &bSuccess,  
CATString &sReturnCode);
```

This Function calls *x3GetSupplyFilePath* and *x3StoreSupplyFileCus* in Teamcenter.

In: spProduct	CATProduct to examine in Teamcenter
sFullPath	Full Path of file
sInfoForTC	additional Info for TC
Out: bSuccess	Success status of Teamcenter
sReturnCode	Message from Teamcenter

**CAA-Function:**

```
HRESULT CMISupplyAdditionalFileForDrwToTC ( const CATDocument *pDocument,
                                           const CATString &sFullPath,
                                           const CATString &sInfoForTC,
                                           CATBoolean &bSuccess,
                                           CATString &sReturnCode);
```

This Funktion calls *x3GetSupplyFilePath* and *x3StoreSupplyFileCus* in Teamcenter.

In: spProduct	CATDrawing to examine in Teamcenter
sFullPath	Full Path of file
sInfoForTC	additional Info for TC
Out: bSuccess	Success status of Teamcenter
sReturnCode	Message from Teamcenter

---

## Post-process CATProducts for Synchronize / Update

An option is provided to perform post-processing on CATProducts that are created during Update or Synchronize. An example of this is to transfer the ownership of these files according to the parts they represent. The Post-processing step needs to be enabled in Catia by setting

CMI\_PRODUCT\_CREATE\_POST=ON

in the CATIA environment.

When Synchronize or Update is finished, CMI will call the Teamcenter method *x3PostProcCreatePrd*. Information is provided about each CATProduct and the Teamcenter Part that it represents.

```
message x0WkBnch:x3PostProcCreatePrd(
  input :      string      className      ::
  input :      string      partNumber     ::
  input :      string      sPartClassName ::
  input :      string      sPartOBID      ::
  input : NULL  string      sPartDbName    ::
  input :      string      sPrdClassName  ::
  input :      string      sPrdOBID       ::
  input : NULL  string      sPrdDbName     ::
  update: SetOfStrings *vStats           ::
  output: integer          *mfail) code
```

---

## Enable PDM-Centric Synchronize

In CMI 9.7, a new concept for Synchronize Teamcenter has been devised. The goal of this effort was to minimize Teamcenter-CATIA roundtrips. The classic Synchronize function would perform each operation individually, with the benefit of immediate user feedback in CATIA, eg. for each Part created. However, this had the downside of slow progress where complex product structures or remote sites were involved.

The new Synchronize processes the product structure entirely in Teamcenter. Only few roundtrips are necessary, regardless of the size of the product structure.

---

Since the entire process has been renovated, we cannot guarantee that the new Synchronize is compatible with every existing customization, though this was naturally our goal. Therefore the new process is optional in CMI 9.7.

Setting the environment variable CMI\_NEW\_SYNCHRONIZE=ON in the CATIA environment will enable the new Synchronize. At the same time, it will disable the old Synchronize, and the individual Update and Create functions which are obsoleted.

Users must have message access to the PdmSessn:x3SynchFromFile message.

### ***Validate actions before PDM-Centric Synchronize***

If the Catia Configurion variable CMI\_ENABLE\_VALIDATE\_BEFORE\_UPD=ON then the actions of synchronize are validated. The default behavior is, that all validations are successful. If an operation on an item has to be done, the kind of the operation ("CREATE" (for creating an item), "UPDATE" (for updating an item) and "DROP" (for delete a relation)) is given in a parameter.

For each object one of the following methods are called:

<b>x3ValV5Part</b>	for Parts
<b>x3ValV5Model</b>	for CATParts, model, cgr, CMIArchive
<b>x3ValV5Product</b>	for CATProducts
<b>x3ValV5DesTbl</b>	for design tables

For each relation the following methods are called:

<b>x3ValV5PartInst</b>	for the relation between Parts
<b>x3ValV5ModelInst</b>	for the relation between Parts and Files (whitout CATProducts and design tables)
<b>x3ValV5DesTblInst</b>	for the relation between Parts or CATParts and design tables

### ***Often used slots in the NVSET's given in the messages above***

In these methods information about the objects and related objects is provided. For details see the method's headers in the file x3Custom.mth. If one validation fails, the process will continue until all operations are validated, but after this the program returns to the synchronize-dialog in Catia without doing any change in Teamcenter.

OBID	The obid of the object in Teamcenter. It doesn't exist, if the operation is CREATE
DB_NAME	The name of the database in Teamcenter. It doesn't exist if the operation is CREATE.
CLASSNAME	The classname in Teamcenter. If the operation is CREATE, the standard CMI-Class for the object is given.
PARTNUMBER	The partnumber in Teamcenter
EXTERNAL_LOCATION	The full path of the object. Exists, if the object is a FSItem in Teamcenter
INSTANCENAME	The instance name of a relation
NOMENCLATURE	The nomenclature of a CATProduct in Catia

## Reference Geometries

The concept of reference geometries lets you include Parts and Assemblies in the CATIA product structure that are not Part of the general Teamcenter Assembly structure / Bill of Material (BOM). This allows to reference context geometry in the CATIA structure, eg. for drawing creation.

The following environment variables have to be set before startup of CATIA V5:

```
set CMI_CONFIGURABLE_NODE_BEHAVIOR=ON
set CMI_CONFIGURATION_FILE=<path>\CMICatiaV5Config.xml
```

In the CMI Configuration File (CMICatiaV5Config.xml) you need to define a Prefix for the ReferenceGeometry behavior. The following example defines a prefix of "REF\_":

```
<?xml version="1.0" encoding="UTF-8"?>
<!DOCTYPE CMIconfigTopics SYSTEM "CMICatiaV5Config.dtd">

<CMIconfigTopics>

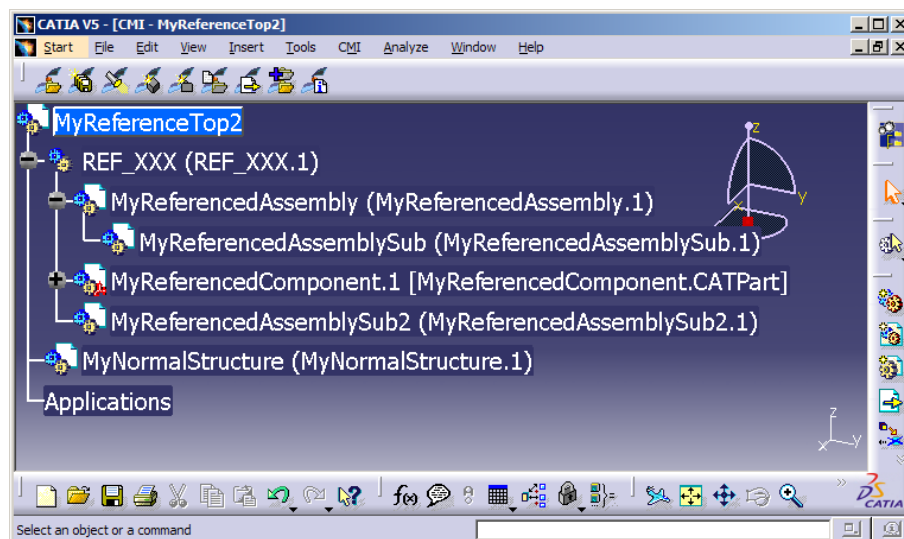
    <ConfigurableBehaviors>

        <ConfigurableBehavior UniqueID =
"EmbeddedNode_REF_Node">

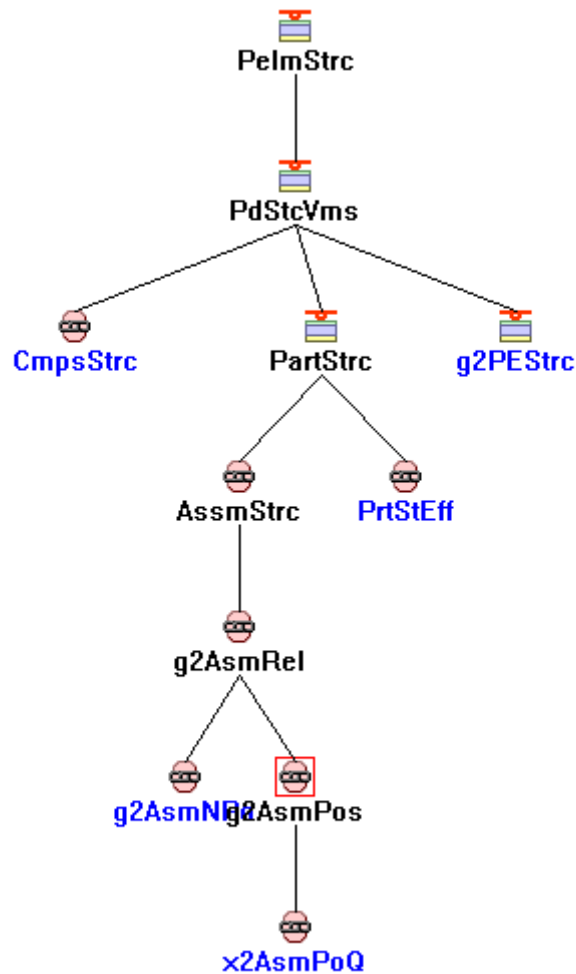
            <BehaviorType>EmbeddedNodeBehavior</BehaviorType>
                <PartNumberPrefix>REF_</PartNumberPrefix>
                <Behavior>ReferenceGeometry</Behavior>
            </ConfigurableBehavior>

        </ConfigurableBehaviors>

    </CMIconfigTopics>
```

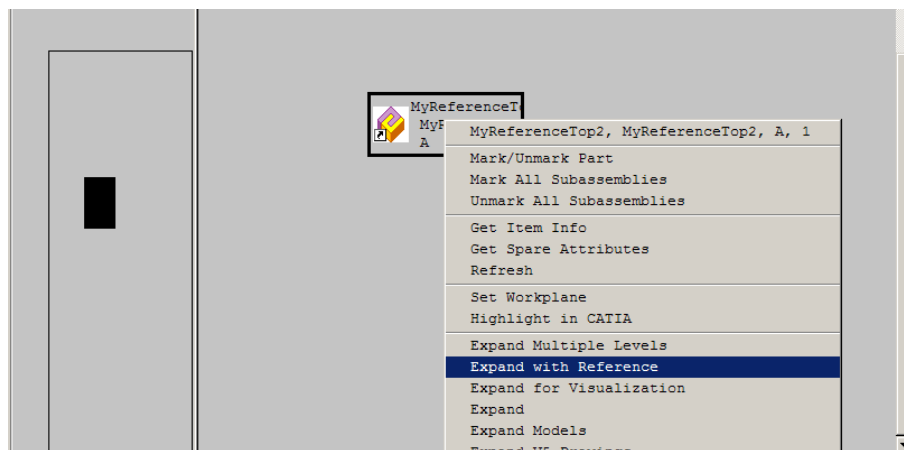


Parts inserted below a component prefixed with "REF\_" will create a special "Has Reference Geometries" Relationship in Teamcenter (during Synchronize), instead of the standard product structure relation *g2AsmPos*. The new product structure relation is called *g2PEStrc* and is derived from *PelmStrc*. Hence it is not an *AssmStrc* relation.



These Relations will be expanded in the CMI Workbench, but not by the *Uses Parts* expand in the standard browsers.

Observe the message access rule for the Teamcenter method *g0Genltn:g3ExpBranchMulRef* in order to remove or enable the Workbench expand relating to Reference Geometries.



If you need to define your own reference geometries product structure relation, set the class constant **x0WkBnch.x4PartToRefPartC**.

---

## Validate CATIA Version

CMI stores the CATIA Release, Service Pack and Hotfix that were used to save a CATIA file. This information can be used during Read from Teamcenter to avoid opening a file that was stored with a newer version of CATIA.

set CMI\_VALIDATE\_CATIA\_VERSION "ON"; in the Teamcenter configuration to enable this validation. OOTB, only the Release Level is validated. You can adapt the validation scheme by overriding the following message:

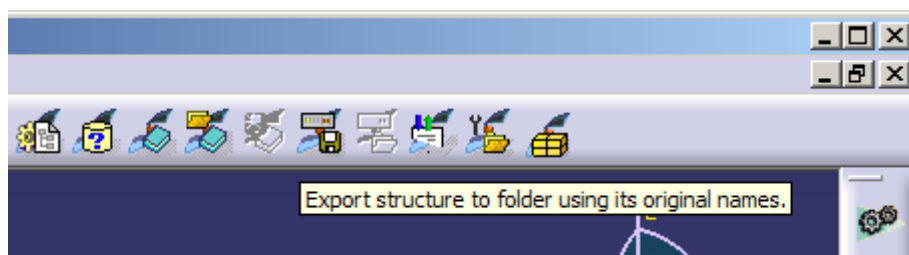
```
object message g0GenBin:x3ValFileVerForRead (
  input:   ObjectPtr   CatiaFile      ::
  input:   integer     FileRelease    ::
  input:   integer     FileHotfix     ::
  input:   integer     FileServicePack ::
  input:   integer     CurRelease     ::
  input:   integer     CurHotfix      ::
  input:   integer     CurServicePack ::
  output:  boolean     *ReadOk        ::
  output:  string      *Message       ::
  output:  integer     *mfail         ::
) code
```

Return `ReadOk = FALSE` to prohibit loading the file. You should also return an instructive message text in `Message`, which will be displayed in CATIA.

---

## Export to Folder

Set `CMI_ENABLE_EXPORTCMD=ON` to enable the *Export Structure to folder* function. The functionality allows to export CATIA data that was loaded from Teamcenter to a specific folder. Original file names can be restored thanks to the mapping file.



It is also possible to reset the CATIA standard attributes to their initial value before first import.

The following attributes are supported: "Nomenclature", "Definition", "Revision" and "Description".

To enable the mapping of the attributes the following setting has to be added to the CATIA environment:

```
SET CMI_EXPORT_STDATTRIBUTES=<attribute1> [<attribute2> [<attribute3>
<attribute4>]]]
```

---

## **Name customization for Export**

To enable the functionality you have to set the following environment before start up of CATIA V5:

```
SET CMI_EXPORT_CUSTOMIZE_NAMING=ON
```

In order to be able to use different naming schemes for the same data, an initialization method `x0WkBnch:x3GetExpNamingSchema` is called. The single purpose of this method is to optionally return an arbitrary string that distinguishes one naming scheme from another in subsequent method calls. For example, you may show a dialog where the user chooses a naming scheme.

```
class message x0WkBnch:x3GetExpNamingSchema (
    input :      string    classname           ::
    input :      string    rootPartNumber     ::
    input :      string    rootPartObid       ::
    input :      string    rootPartClass      ::
    input :      string    rootPartDbName     ::
    input : NULL string    mappingFileObid    ::
    input : NULL string    mappingFileClass   ::
    input : NULL string    mappingFileDbName  ::
    output:      string    *expNamingSchema   ::
    output:      SetOfStrings *pStats         ::
    output:      integer   *mfail);
```

with:

classname	x0Wkbnch
rootPartNumber	Partnumber of the top level CATProduct in CATIA
rootPartObid	Obid of the top level TC Part in CATIA
rootPartClass	ClassName of the top level TC Part in CATIA
rootPartDbName	DBName of the top level TC Part in CATIA
mappingFileObid	Obid of the mapping file
mappingFileClass	ClassName of the mapping file
mappingFileDbName	DbName of the mapping file
expNamingSchema	Identifier for schema to be used (can be NULL)
pStats	Messages displayed in CATIA

For each file in turn the following method is called, where custom values for The reference attributes can be returned. The `expNamingSchema` string from the previous method is given as an argument:

```
class message x0WkBnch:x3GetExportRefNames (
    input :      string    classname           ::
    input :      string    expNamingSchema     ::
    input :      string    tcPartNumber       ::
    input :      string    tcObid             ::
    input :      string    tcClassname        ::
    input :      string    tcDbName           ::
    input :      string    catFileName         ::
    input : NULL string    catNomenclature     ::
    input : NULL string    catRevision         ::
    input : NULL string    catDefinition       ::
    input : NULL string    catDescription      ::
    input : NULL string    extPartNumber       ::
    input : NULL string    extFileName         ::
    input : NULL string    extNomenclature     ::
    input : NULL string    extRevision         ::
    input : NULL string    extDefinition       ::
    input : NULL string    extDescription      ::
    output:      string    *partNumber         ::
    output:      string    *fileName           ::
    output:      string    *nomenclature       ::
    output:      string    *revision           ::
```

```

output:      string  *definition      ::
output:      string  *description     ::
output:      SetOfStrings *pStats     ::
output:      integer *mfail)

```

**with:**

```

classname          x0Wkbnch
expNamingSchema    Identifier for schema to be used
tcPartNumber       Catia Part Number from Teamcenter
tcObid             TC Obid
tcClassName        TC Classname
tcDbName           TC Db name
catFileName        Filename in CATIA
catNomenclature    Nomenclature in CATIA
catRevision        Revision in CATIA
catDefinition       Definition in CATIA
catDescription     Description in CATIA
extPartNumber      External Part Number from mapping file
extFileName        External File ame from mapping file
extNomenclature    External Nomenclature from mapping file
extRevision        External Revision from mapping file
extDefinition      External Definition from mapping file
extDescription     External Description from mapping file
partNumber         CATIA Part Number to set
fileName           CATIA File Name to set
nomenclature       CATIA Nomenclature to set
revision           CATIA Revision to set
definition         CATIA Definition to set
description        CATIA Description to set
pStats            Messages displayed in CATIA

```

As an input, you receive the identifiers in Teamcenter (tc...); the current values of attributes in CATIA (cat...); and the original attributes from the mapping file (ext...).

The ext... attributes are NULL if no mapping file exists / is used for the exported structure. It is possible to return NULL for partNumber, fileName, nomenclature, revision, definition, description.

For each instance, the following method is called where you can return a new instance name:

```

class message x0Wkbnch:x3GetExportInstName (
input :      string  classname          ::
input :      string  expNamingSchema    ::
input :      string  relObid            ::
input :      string  matrixIndex        ::
input :      string  catInstanceName    ::
input : NULL string  extInstanceName    ::
output:      string  *instanceName      ::
output:      SetOfStrings *pStats       ::
output:      integer *mfail)

```

**with:**

```

classname          x0Wkbnch
expNamingSchema    Identifier for schema to be used
relObid           Relation Id
matrixIndex        Index for Multy-Quantity
catInstanceName    Instance Name in CATIA
extInstanceName    External Instance Name from mapping file

```

---

instanceName	CATIA Instance Name to set
pStats	Messages displayed in CATIA

The new names in the exported structure are built using the following rule:

- Any CATIA attribute (Part Number, File Name, Nomenclature Revision, Definition Description) from `x3GetExportRefNames` or Instance Name from `x3GetExportInstName` that is returned (not NULL) is used in the exported structure.
- If CATIA attributes are not returned in `x3GetExportRefNames` or `x3GetExportInstName`, the values from the mapping file are used. Standard properties are only set if configured with `CMI_EXPORT_STDATTRIBUTES` and available in the mapping file.
- other the attributes currently available in CATIA are kept.

---

## Handling of Mapping Files

Set `CMI_ENABLE_CREATE_IMPORT_FILE=ON` to enable the use of a name mapping file in Synchronize, Reconnect and the Export function.

To set the CATIA standard attributes Nomenclature, Revision, Description or Definition back to the original value of the initial import set the value of `CMI_EXPORT_STDATTRIBUTES` to the attributes you want to set back. Only used with the CMI Export Cmd.

Example: `CMI_EXPORT_STDATTRIBUTES=<Nomenclature> <Definition> <Revision> <Description>`

The following class constants have been introduced for the CMI Mapping File:

**g0GenWb.g4MapDocClassC** - Class for the Document attaching Mapping File (GenDoc)

**g0GenWb.g4MapPrtDocRelClassC** - Class of relation between part and Document for mapping files

**x0WkBnch.g4MapFileClassC** - Class for mapping files

**g0GenWb.g4MapDocFileRelClassC** - Class of relation between Document and mapping file

The following customizing methods have been introduced for the CMI Mapping File

**Part:g3GetMappingDocs** – return Documents where mapping files are attached

**GenDoc:g3GetMappingFiles** – return mapping files attached to Documents

**x0WkBnch:x3FindMappingFile** – find mapping file for a given part number

**x0WkBnch:x3GetArchiveFiles** - find Archives for a list of root part numbers

see `x3Custom.mth` for details

---

## Syntax of the Mapping file

To open the CMI mapping file in a browser or editor with syntax check, place a file CMIMapping.dtd in the folder of the xml file.

CMIMapping.dtd:

```
<!ELEMENT mapping (references*, instances*) >
<!ELEMENT references (reference)+ >
<!ELEMENT instances (instance)+ >
<!ELEMENT reference (tcPartNumber, tcFileName, tcNomenclature,
tcRevision, tcDefinition, tcDescription, tcClassName?, extPartNumber,
extFileName, extNomenclature, extRevision, extDefinition,
extDescription) >
<!ELEMENT instance (tcInstanceName, extInstanceName) >
<!ELEMENT tcClassName (#PCDATA) >
<!ELEMENT extInstanceName (#PCDATA) >
<!ELEMENT tcInstanceName (#PCDATA) >
<!ELEMENT tcPartNumber (#PCDATA) >
<!ELEMENT extPartNumber (#PCDATA) >
<!ELEMENT extFileName (#PCDATA) >
<!ELEMENT tcFileName (#PCDATA) >
<!ELEMENT extNomenclature (#PCDATA)>
<!ELEMENT tcNomenclature (#PCDATA)>
<!ELEMENT extRevision (#PCDATA)>
<!ELEMENT tcRevision (#PCDATA)>
<!ELEMENT extDefinition (#PCDATA)>
<!ELEMENT tcDefinition (#PCDATA)>
<!ELEMENT extDescription (#PCDATA)>
<!ELEMENT tcDescription (#PCDATA)>
<!ATTLIST reference
      id ID #REQUIRED
>
<!ATTLIST instance
      id ID #REQUIRED
      parentId IDREF #REQUIRED
      childId IDREF #REQUIRED
>
```

---

## CATScript support

CATScript can be registered in Teamcenter and used as Part of a CMI-managed Catalog.

To insert CATScripts into a Catalog enable the *Insert CATScript from Teamcenter*:

Set CMI\_ENABLE\_CMICATALOGINSERTSCRIPTCMD=ON in the CATIA environment.

Two messages have been introduced for CATScript support, that can optionally be customized.

Override x3GetScriptForInsClg to change the way the Script is queried for insertion into the catalog:

```
class message g0GenBin:x3GetScriptForInsClg (
  input:      string      classname      ::
  input:  NULL string      sOldFilename  ::
  input:  NULL string      sOldDescription ::
  input:  NULL string      sOldObid      ::
  input:  NULL string      sOldClassname  ::
  input:  NULL string      sOldDbName     ::
  input:  NULL string      sOldMstrObid   ::
  input:  NULL string      sOldMstrClassname::
  input:  NULL string      sOldMstrDbName  ::
```

```

output:      ObjectPtr    *scriptObj      ::
output:      string       *sMstrObid       ::
output:      string       *sMstrClassname  ::
output:      string       *sMstrDName      ::
output:      string       *sErrorMsg       ::
output:      integer      *mfail          ::
);

```

The default implementation of this method displays a Teamcenter query for Generic Script, with the supplied attributes pre-filled. Return the script object to insert into the Catalog as `scriptObj`.

Override `x3GetScriptForUseClg` to change the way the script is retrieved when the catalog is used:

```

class message g0GenBin:x3GetScriptForUseClg (
input:      string       classname      ::
input:      string       sObid         ::
input:      string       sClassname    ::
input:      string       sDbName       ::
input: NULL  string       sMstrObid    ::
input: NULL  string       sMstrClassname ::
input: NULL  string       sMstrDbName  ::
output:     ObjectPtr    *scriptObj    ::
output:     string       *sErrorMsg    ::
output:     integer      *mfail        ::
);

```

The default implementation of this method retrieves the CATScript by OBID.

---

# CHAPTER 6

## Workbench Architecture

---

### Generic Workbench

The Generic Workbench (GMI) is the base server of the CATIA Teamcenter Integration and other products of T-Systems International GmbH such as „Voxel Metaphase Integration“. The Generic Workbench contains basic classes for the following functionalities:

- creating the window,
- creating some dynamic items,
- creating some information items about the exchange map,
- creating the relations,
- set/reset the communication between CATIA and Teamcenter Enterprise,
- controlling of permissions,
- controlling of visibility.
- ...

There are some public methods in the Generic Workbench which allow a customer to control the permissions and visibility. Please consult the files

- custom/g3Custom.mth (GMI related methods) and
- custom/x3Custom.mth (CMI related methods)

for more information.

---

### Catia Workbench

The CATIA Workbench (CMI) is the main server of the CATIA Teamcenter Integration and bases on the Generic Workbench.

---

# CHAPTER 7

## Teamcenter Enterprise configuration variables

---

### CMI Server Options

These are the possible settings for CMI within the PDM\_CONFIG file (config.cfg):

Variable	Comment
CMILISPATH	Path to listener (cmilis will be searched in \$PATH if not given)
CMICITRIX	Set to "Yes" if CMI is used with Citrix
CMILISINFO	Path to communication file for CITRIX
CMIEXMAPHOST	Host for Exchangemap with Citrix
GCVMI_SERVER_DEBUG	Set to "ON" to receive CMI debug output from the dispatcher
GCVMI_ENABLE_LINK	If set to "YES", CMI will try to create links to model files instead of copying them to the exchangemap. Recommended. Links are only available on UNIX OS.
GCVMI_MAX_QUANTITY	Maximum Quantity for CMI Multiquantity relations
CMI_CATN4D_STARTUP	Path to 4D Navigator startup script
GCVMI_SERVERPORT	Port range used on the server for CMI V4 communication
GCVMI_CLIENTPORT	Port range used on the client for CMI V4 communication
CMIDATAMODEL	DEPRECATED: Set to COMPONENTDATAMODEL to run the optional CatiaV5 centric data model (Map CATParts to Cmpnent)  Use CMI_BOM_PART_DEFAULT_FOR_SYNC instead.
CMI_DMU_VAULT_LOC	Vault location for DMU preprocessed file
CMI_DMU_EXP_PATH	Path of DMU_Exp utility
CMI_DMU_HOST	Host where DMU preprocessed file will be generated
CMI_DMU_STARTUP_PREF	Set to "ON" to use Startup-preferences of "super user" with DMU_Exp

CMI_CT_V5_SUBDIR	Subdirectory in Work Location for Catia V5 files (supports both Windows and UNIX format)
CMI_CT_V4_SUBDIR	Subdirectory in Work Location for Catia V4 files (supports both Windows and UNIX format)
CMI_V5_PRD_SUPPORT	Set to "OFF" to disable CATProduct management in Catia V5 (use temporary CATProducts). Doing so is not recommended.
CMI_STDPART	Set to "YES" to enable Standard Part behaviour in CatiaV5, based on StdPartIndicatorAttr in Teamcenter
CMI_CGR_HOST	Host where the vault location for CGR files resides
CMI_CGR_VAULTLOC_PATH	Path of the vault location where CGRs are searched
CMI_SHOW_BB_INCHES	Set to YES to display / enter Bounding box coordinates in inches
CMI_USE_CCS	Set to "True" for client machines or workgroups to use a CCS service for Catia V4 communication.
CMI_CREATE_DOC_INTERACTIVE	set to "ON" to create the Product Document and the CATPart Document and resp. Part objects of a BOM CATPart interactively
CMI_STORE_MULTI_INSTANCENAME	set to "ON" to store the Instance names in the x2AsmPoq Relation for the multi quantity relation (by default they are numbered (#))
CMI_USE_DB_NAME	Set to "ON" to query for objects via OBID and database name during CMI update. That means objects will be only searched within the database they are really saved in.
CMI_ITEM_INFO_ORIG	Set to "ON" to show Get Item Info of the original object (eg. Part) instead of the Catia Workbench Item
CMI_USE_BLACKBOX	Set to "ON" if you use the deprecated "BlackBox" functionality. This has a performance impact.
CMI_SINGLE_PART_DOCUMENT	Set to "ON" to reuse a single Document per Part, for multiple models
CMI_DO_NO_LINK_DRW_TO_PART	Set to "ON" to suppress the link of the Drawing to the part
CMI_USE_ORIG_NAME_FOR_PRODUCT	If set to "ON" the original filename from catia is used in Synchronize for Products and BOM Part
CMI_DESIGN_TABLES	If set to "ON", design tables are managed by CMI. This has a performance impact.
CMI_VIEW_NETWORK_EXPAND	Set to "ON" if you require to use View Networks in the context of CAD Design. This has a performance impact during update.
CMI_TEMPLATE_VAULTLOCS	Names of vault locations for Create CATPart/CATDrawing in Teamcenter
CMI_DMU_USE_WORKLOC	Set to "ON" to create the DMU_File in the product-worklocation
GCVMI_DISABLE_CGR_LINK_V5	Set to "ON" not to Link V5-cgr-cache-files during "to Catia"
CMI_TRANSLATE_IP	If Set to 'OFF' no net_host_ipaddress_get

	(Host) is done with g3SetPortAndHost. Set if there are name resolution issues.
CMI_DMU_FILENAME_SEP	default is "_"; separator to build DMU-filename from <PartNumber>_<Revision>[_<Context>].dmu; this separator must not occur in PartNumber, Revision or Context
CMI_SINGLE_PART_DOCUMENT	(changed): Set to "SINGLE_FILE" to create one document for each CATPart Set to "ON" to reuse a single Document per Part, for multiple models
CMI_RESTORE_WB_WITH_ACT_CONTEXT	If set to "ON", the current configuration context is preserved while restoring a saved session. By default, the configuration context is temporarily disabled.
CMI_VIEWER_ENCODING	Encoding for the plmxml-file for the viewer. If not set, the default value is "ISO-8859-1"
CMI_ONLY_PUBLIC_TO_VIEWER	If set to "ON" the option 'Filter for public versions' in the Model Filter preference is temporarily enabled when sending to viewer.
CMI_VALIDATE_CATIA_VERSION	If set to "ON" the Catia release of the files is validated against the current sessions CATIA release. If the file is from a newer release, it is not opened.

---

# CHAPTER 8

## CATIA V4 Directory Structure

---

### Directories

The following figure shows the standard directory tree of the CATEDM installation.

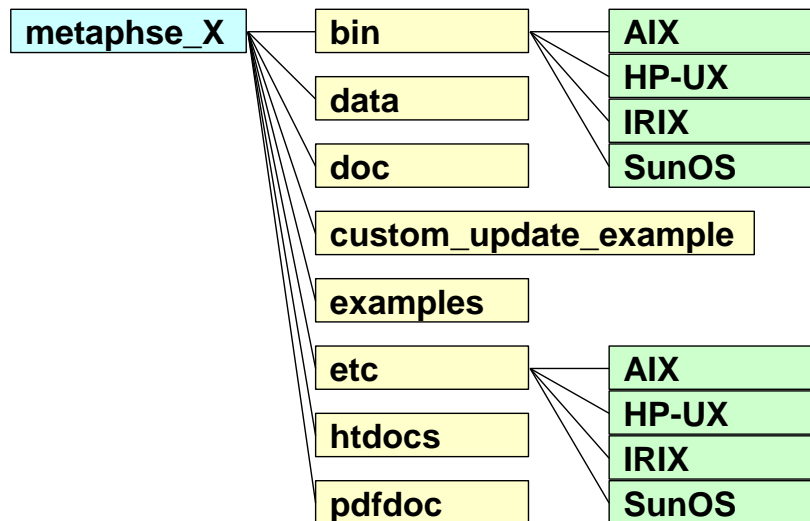


Figure 8: CATEDM installation path structure

- `bin/<your os>:` Some executables of the integration. All CATIA V4 user should have access privilege to this directory.
- `custom_update_example/` : This is an example how you can edit the code and generate your own shared library.
- `data/` : Declaration include files, initialization scripts, error definition file, menu definition file, message definition file.
- `doc/` : The latest information about the product, script file "xt0request.sh" and a simple CATIA V4 start script "catstart.sh".
- `examples/` : Some example CATIA V4 models.

---

`Etc/<your OS>`: CMI CATIA V4 GII module in binary format (METAPHSE). All CATIA V4 user should have access privilege to this directory.

`htdoc/` : Help files in HTML-Format (start page is index.htm).

`pdfdoc/` : Help file in the PDF-Format (XPDF uses this file to get the context sensitive help)

---

## Files

The Following section describes some important files and their meaning.

`doc/README_FIRST.txt`:

This file contains some necessary tips for the CATIA V4 integration. You should read this file.

`doc/WHATSNEW.txt`:

This file contains the changes of the CATIA V4 module. There are new features, changes and bugfixis.

`doc/README.environment`:

This file contains the possible environment settings of the CATIA V4 module. The File `README.env.xls` is the same in the Micosoft Excel format.

`data/xt0request.sh`:

This file is needed for the communication between CATIA V4 client and Teamcenter Enterprise server. You should modify this file and copy it into any directory in search path of CATIA V4 clients.

`data/ini.env`:

This initialization script contains some necessary environment settings for the integration. The meaning of the certain environment variables is described in the `ini.env` file. This file should be adjusted and each user must run this script before starting CATIA V4. Please refer the `README.environment` file for more information.

`data/METAPHSE.include`:

CATIA V4 declaration file for the integration. This file should be included to the users `USRENV.dcls` or to any other local or global CATIA V4 declaration file.

`data/appdefault.obj` :

This file contains some environment settings of the CATIA V4 module. Please refer the `README.environment` file for more information. The used syntax is "`{<parameter>} <value>`". Some parameters are named `<parameter>.example`. If you want to set this parameter you have to delete the extension ".example".

`data/ERREDB` :

---

This file contains the error messages in CATIA V4.

`data/dshdrawingframe.sh` :

This file contains some information how to fill a drawing title block. The customizing of title block filling is described in "**Error! Reference source not found.**".

`data/cleanbox` :

All text inside the boxes described in this file will be deleted during filling the drawing frame.

`data/plotconf`:

This file contains the options of the CATIA V4 PLOT UTILITY.

`data/catiaedb.msg` :

This file contains a list of all used messages appearing in CATIA V4 when you are using the CMI Module. You can customize this message file to your own needs.

`data/catiaedb.menu` :

This file contains a list of all menu points of the CMI Module. You can change each menu point to your own needs. A menu name consists of 8 characters at maximum. An empty entry means that the menu point is disabled.

`data/edmhelf.conf` :

If xpdf is used as help tool, this file contains the bindings between the menu and the pages of the help file. If you write an own help file you must edit this file.

`$HOME/exchangemap` :

This directory is a local UNIX directory in user's home directory. On each CATIA V4 client workstation an exchange map must exist. The task of this UNIX directory is to exchange data between CATIA V4 and Teamcenter Enterprise. Each user should have an own exchange map. For more information about customizing tasks please refer to „**Error! Reference source not found.**“.

`$HOME/.dshcatiaarc.obj` :

This optional script file overwrites the default settings for the user. Please refer the `README.environment` file for more information.

---

## Modify CATIA V4 Environment

extend STEPLIB and CATDEC environment settings as following (example):

```

STEPLIB= . . . . :/usr/lpp/catia/v4r1_code/gii/steplib
STEPLIB=$STEPLIB:/catia/gii/metaphse_catedm/etc/<your OS>
export STEPLIB
CATDEC=$CATDEC: /catia/gii/metaphse_catedm/data
export CATDEC

```

Include the file `data/METAPHSE.include` to each user's `USRENV.dcls` file as following (example):

```

/*-----*/
/* USRENV.dcls      DECLARATION FILE          */
/*-----*/
INCLUDE ('/catia/v4r1/prod/USRENV.include');

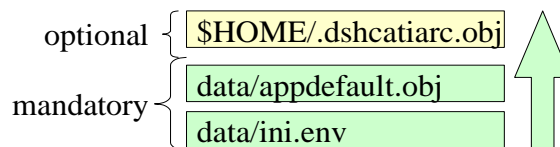
/*-----*/
/* INCLUDE all other Configuration - files    */
/*-----*/
INCLUDE ('/catia/gii/metaphse_catedm/data/METAPHSE.include');
/*-----*/

```

## User dependent configurations

Normally you don't need any modifications for a certain user (except user's personal `USRENV.dcls` file). Therefore, each user can overwrite the default settings. You may copy the file `data/appdefault.obj` to the user's home directory and rename it to `.dshcatiarc.obj`. Now you can edit this file and overwrite the existing settings.

Following the order of running the setting files:



**Figure 9: Initialization files with their order.**

The following Environment settings are possible (see next tables)

Name of Environment Variable (ini.env)	Name of CONFIGURATION FLAG (APPDEFAULT.OBJ .DSHCATIARC.OBJ)	Short Description	Possible Values
CATEDM_DEBUG	Debug	set a debug level	0 - no debug 66 - full debug

CATEDM_BINDIR	-	path to the executables of CATEDM installation	
CATEDM_SCRIPTDIR	-	path to the scripts and configuration files of CATEDM installation	
CATEDM_CONN	Connectmethod	method to connect with CMI workbench	
CATEDM_EXCHANGEMAP	Exchangemap	directory where CMI and CATIA V4 exchange model files	
CATEDM_EXMAPDLNAME	exchange_dlname	logical map name which is related to the exchange map	
CATEDM_MLINKS	Modellinks	create a symbolic link (description) to each model in exchange map	
CATEDM_TMPAXIS	Tmpaxis	use a start model	0 - no start model 1 - use a start model
CATEDM_TMPAXISDLNAME	tmpaxis-dlname	logical CATIA V4 map for the TEMPAXIS model	If there is no
CATEDM_TMPAXISDSNAME	tmpaxis-dsname	CATIA V4 map for the TEMPAXIS model	TEMPAXIS MODEL
CATEDM_TMPAXISNAME	tmpaxis-name	template for the TEMPAXIS model: the CATIA V4 model without ".model" extension	(MAP) defined the INITIAL_MODEL of CATIA V4 is used
CATEDM_NAMETYP	modelnametype	position from which a CATIA V4 model name is displayed in CATIA V4 status	
		field. useful for names longer than 32 types.	
CATEDM_LOGF	Logfile	name of the file where the log should go.	
CATEDM_HELPERTOOL	Helper	name of the installed helper application (xpdf, netscape, acroread)	
CATEDM_HELPFILE	Alias	name of the online help file	
CATEDM_CATIAVER	Catiaversion	to set the current CATIA V4 version	
CATEDM_PROJECT	Project	defines the name of the current CATIA V4 project	
CATEDM_WPMODUS	wpmodus	default workplane modus	0 - assembly 1 - default workplane 2 - geometry 3 - multiselection
CATEDM_GEOPOS	geoposallowed	geometry position support	0 - off 1 - on
CATEDM_APPDEF	-	name of the CATEDM configuration file (default is appdefault.obj)	
CATEDM_RCFILE	-	path and name of the user specific configuration file (default is \$HOME/.dshcatiaarc.obj)	
CATEDM_CONNECTXFILE	connectx-script	alternative way to connect to the	

		workbench – not used yet	
CATEDM_NORMPART	normpart-support	support for norm part integration NIS	0 - off 1 - on
CATEDM_REFRESH	autorefresh	refresh the current CATIA V4 session after a CATEDM READ action	0 - off 1 - on
CATEDM_RM_MODELS	remove-models	remove models from exchange map at first CATEDM startup time	0 - off 1 - on
CATEDM_ASSEMBLY_SYMMETRY	assembly-symmetry	allow MOD POS->MOVE->SYMMETRY for assembly positions	0 - off 1 - on
CATEDM_LOADWARNING	load-warning	show a warning panel when a read action is started from workbench	0 - no warning 1 - at READ 2 - at REREAD 3 - at READ and REREAD
CATEDM_MERGE	merge-support	CATEDM supports CATIA V4 MERGE	0 - off 1 - on
DebisLICDIR	-	location of debis licman license file	
DebisLICBIN	-	location of debis licman executables	
CATEDM_LICMAN_START	license-startscript	license manager start script - default is licman12	Licman12
CATEDM_LLD_AUTOSTART	lld-autostart	start local license daemon at first CATEDM startup time	
CATEDM_SETUPSTAT	setupstatus	reads configuration flags from appdefault.obj at any module entry only used for debug	0 – off 1 - on
CATEDM_DMPF	stdumpfile	information file - only needed for VMI	
CATEDM_EDBHOST	edbapphost	CATIA V4 client host name	
CATEDM_SMD_WEIGHT	smd-weight-support	CATEDM supports to read the weight and the position of SMARAGD models	0 – off 1 - on
CATEDM_DESCINFOAPP	desc-info-applications	read the user defined blocks and write the data to the info object. To	application list
CATEDM_DESCINFOELE	desc-info-elements	access to the blocks you need three things:	pt, ln, ...
CATEDM_DESCINFODESC	desc-info-descriptions	1. The application string: before any description, modification or read routine is used, the user must declare the application string 2. Element type 3. Types of the description: (1-16000) This routine restricts the size of the data block to 32 elements of each type.	types of the descriptions

	example:	<p>CATEDM_DESCINFOAPPEDBCATIA  CATIAEDB  CATEDM_DESCINFOELEPT LN TXTN  CATEDM_DESCINFODESC{12345 2456} 3457  {1111}  "==&gt; PT: search for application string  EDBCATIA and CATIAEDB and types 12345  and 2456</p>	
CATEDM_COMMENT	comment-support	<p>Read the comment lines of a  model file and write them into a  file into  the exchangemap directory. The  full filename stands into the info  obj</p>	<p>0 – off  1 - on</p>
CATEDM_NEWUPD	Newupd	<p>After your selecting <b>METAPHSE</b>  &gt; <b>UPDATE</b> &gt; <b>ALL</b> a list of models  that have been modified will come  up in CATIA V4. If some models  were active, but the user mustn't  change these models, a status  panel will pop up. It is not  necessary to confirm this panel.  After your confirmation the  selected models will be updated in  Teamcenter.</p>	<p>1 - in CATIA  V4 (default)  0 -no  selection of  the models  in CATIA V4</p>
CATEDM_REPLACEMODEL	replace-model	<p>Save As / Create can replace the  original CATIA V4 model in the  current SESSION by the new  registered Teamcenter CATIA V4  model. (This is only possible if the  model was loaded by CATIA  FILE-&gt;open) If the replace  functionality is turedned off:  The new model is loaded  additional into the CATIA  SESSION</p>	<p>1 - replace  (default)  0 - do not  replace</p>
CATEDM_CUSUPD	cmi-custom-update	<p>calls the shared lib  libcmi_custom_update.&lt;a,sl,so&gt;  The customer can use its own libs  to perform some CATIA action at  update / create</p>	<p>1 - use the  shared lib  0 - do not use  the shared lib</p>
CATEDM_BBOX	bbox-support	<p>bounding box generation for  CATIA models (2 points for each  model: D3D_{X,Y,Z}{1,2})  bounding box points will be sent  for each model at UPDATE  ALL/MODELS, CREATE/SAVE  AS and MULT CRE.</p>	<p>0 - no bbox  support at all  1 - standard  bbox support  (without  checkbox at  update and  muticreate)  2 - bbox  support with  checkbox  (default  support off)  3 - bbox  support with  checkbox  (default  support on)</p>

CATEDM_BBOXLAYER	bbox-layer-list	Only the elements of these layers are used to generate the bounding box. If no layer is set the current layer is used.	0 1 2 ... 254 - creates a bounding box using these layers -1 - uses the actual layer filter -2 - creates the box using all layers
------------------	-----------------	---	---

This is a short documentation of the configuration environment of CUSTOM ATTRIBUTS

ATTRIBUTE	VALUE	DESCRIPTION
VolumeMass	Type: REAL CATIA V4 uses the units of the specific model	1. Sets the model standards 2. Calculates the weight and the Center of Gravity (COG) of the Volume (VOL) and the Polyhedral and exact solid (SOL) The results are written into the info object: {SOLWEIGHT} value {SOLCOG} {x-value} {y-value} {z-value} {VOLWEIGHT} value {VOLCOG} {x-value} {y-value} {z-value}
SurfaceMass	Type: REAL CATIA V4 uses the units of the specific model	1. Sets the model standards 2. Calculates the weight and the Center of Gravity (COG) of the Surface (SUR), the Face (FAC), the Skin (SKI) and the SPACE polyhedral surface (POL) The results are written into the info object: {SURWEIGHT} value {SURCOG} {x-value} {y-value} {z-value} {FACWEIGHT} value {FACCOG} {x-value} {y-value} {z-value} {SKIWEIGHT} value {SKICOG} {x-value} {y-value} {z-value} {POLWEIGHT} value {POLCOG} {x-value} {y-value} {z-value}
cmi-custom-update	Type: INTEGER	if [VAL] > 0 then call the functions of the libcmi_custom_update.<a,sl,so>. If the configuration flag {cmi-custom-update} is set to 1 or 2 this attribute will have no effect.
CMI_DRAWING	Type: INTEGER	possible values: 1 --> the drawingframe of this model will be updated 0 --> the drawingframe of this model will be unchanged -1 or no attribute --> the behavior belongs to the environment

## User dependent API in CATIA

There are different ways to call the CATEDM custom functionality:

- Edit the file metaphse\_4.x.x/data/appdefault.obj

{cmi-custom-update} 1	Calls the functions void cmi_custom_update(int *ERROR); at UPDATE ALL / UPDATE GEOMETRY / UPDATE ACTIVE  and void cmi_custom_create(int *ERROR); at DOCUMENT CREATE / DOCUMENT SAVE_AS / DOCUMENT MUL_CRE
{cmi-custom-update} 2 (catedm version 4.0.3 and later)	Calls the function void cmi_custom_update(int *ERROR); at UPDATE ALL / UPDATE GEOMETRY / UPDATE ACTIVE
{cmi-custom-update} 3 (catedm version 4.0.3 and later)	Calls the function void cmi_custom_create(int *ERROR); at DOCUMENT CREATE / DOCUMENT SAVE_AS / DOCUMENT MUL_CRE

- overwrite the message `x0CTFile:x3GetCustomDataForCAD` in the `x3Custom.mth` file of the CMI installation. (catedm version 4.0.3 and later) It is possible to sent the FLAGS

```
attributelist = [{attr_1} [{attr_n}]] {cmi-custom-update}
valuelist     = [{value_1} [{value_n}]] {1}
```

with READ / REREAD to CATIA (needs customisation in MP)

At UPDATE ALL / UPDATE GEOMETRY / UPDATE ACTIVE of the related model the function "`void cmi_custom_update(int *ERROR);`" is called.

---

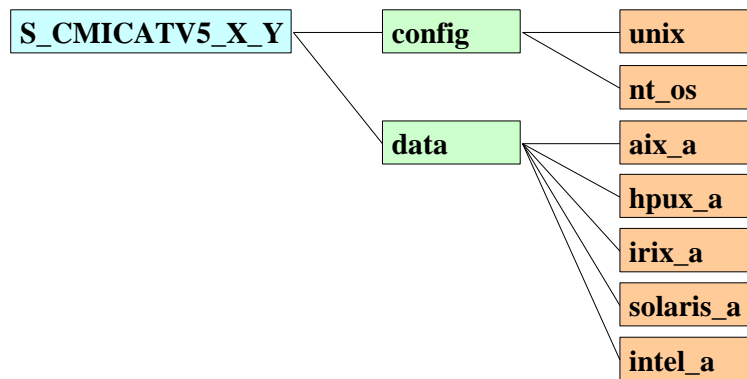
# CHAPTER 9

## CATIA V5 Directory Structure

---

### Directories

Following figure shows the standard directory tree of the CMI CATIA V5 (CMICATV5).



**Figure 10: Directory structure of the CMICATV5 module**

The **config** directory contains sample CATIA V5 Configuration files for unix and Windows 2000. The **unix** configuration contains two sample environment settings for the shells **sh** and **csh**. The **nt\_os** configuration contains a sample CATIA V5 Environment file for the CATIA V5 Environment editor.

The data directory contains the binary distributions for the CMICATV5 module for the supported operating system mnemonics.

The supported operation systems and their mnemonics are:

AIX5.1	aix_a
HPUX 11	hpux_a
IRIX 6.5	irix_a
Solaris 7	solaris_a
Windows XP	intel_a

The mnemonic "aix\_a" will be chosen as an example of a CMICATV5 installation directory on Unix.

Following figure shows the directory tree of "aix\_a".

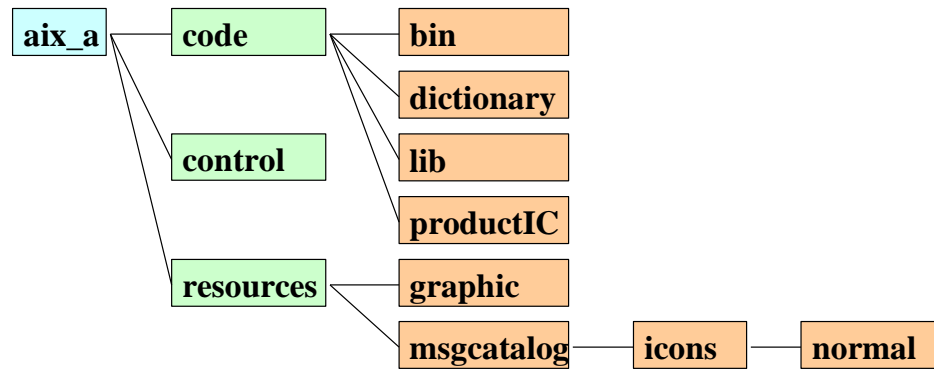


Figure 11: Directory structure of the CMICATV5 installation directory

`msgcatalog/` : If you want to change Text Messages in CATIA V5 the files with the extension CATNIs in the `msgcatalog` directory can be changed. This message files contain string variables. After adapting this strings changes will be shown in CATIA V5 (see examples below!).

All other files in the CMICATV5 installation directory should not be touched.

## Files

This section describes some important files and their meaning. The Files have a Text Message catalog with messages displayed in CATIA V5.

The following example shows the content of the file `CMIUpdateCommandHeader.CATNIs`:


```

// (c) T-Systems 2001

//=====
// Message catalog for the CMIAddinHeader command headers of the
// CMIAddin addin
//=====

CMIUpdateCommandHeader.CMIUpdateCommandHeader.Category = "CMI";
CMIUpdateCommandHeader.CMIUpdateCommandHeader.Title     = "Update";
CMIUpdateCommandHeader.CMIUpdateCommandHeader.ShortHelp= "Update Teamcenter";
CMIUpdateCommandHeader.CMIUpdateCommandHeader.Help      = "Update Teamcenter";
CMIUpdateCommandHeader.CMIUpdateCommandHeader.LongHelp  = "Update
This command updates positions and files in Teamcenter.";
  
```

---

If the mouse pointer is over the tool icon (in this example: the “**Update Teamcenter**” icon ) the **Title** you will see in the status line before the command line. The **ShortHelp** messages will appear in the tooltip and the **Help** message appears in the status line left.

After moving the “What’s This?” icon  to the toolbar icon the text in **Long Help** will be shown.

The following changeable files have the same structure as this example file.

`resources/msgcatalog/CMIReadCommandHeader.CATNls :`

This file contains the Text Messages catalog for the CMI Command

“**Read from Workbench**” .

`resources/msgcatalog/CMIUpdateCommandHeader.CATNls :`

This file contains the Text Messages catalog for the CMI Command

“**Update Teamcenter**” .

`resources/msgcatalog/CMIUpdateCreateInteractiveCommandHeader.CATNls :`

This file contains the Text Messages catalog for the CMI Command

“**Synchronize Teamcenter**” .

`resources/msgcatalog/CMICreateCommandHeader.CATNls :`

This file contains the Text Messages catalog for the CMI Command

“**Create in Teamcenter**” .

`resources/msgcatalog/CMISaveAsCommandHeader.CATNls :`

This file contains the Text Messages catalog for the CMI Command

“**SaveAs in Teamcenter**” .

`resources/msgcatalog/CMICreateV4CommandHeader.CATNls :`

This file contains the Text Messages catalog for the CMI Command

“**Create V4 Model**” .

`resources/msgcatalog/CMISaveAsV4CommandHeader.CATNls :`

This file contains the Text Messages catalog for the CMI Command

“**SaveAs V4 in Teamcenter**” .

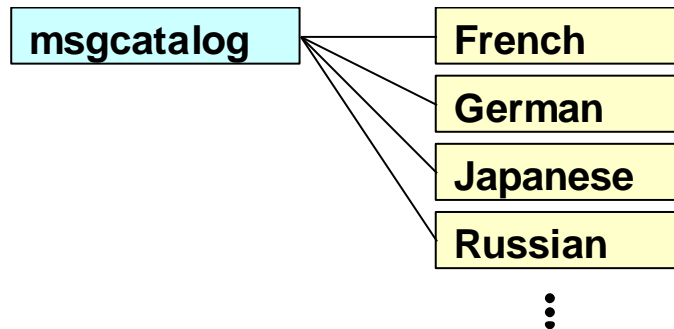
`resources/msgcatalog/CMIInfoCommandHeader.CATNls` :

This file contains the Text Messages catalog for the CMI Command

“CMI-Info” .

For working with Text Messages in different languages the files in the directory `resources/msgcatalog` should be copied in different subdirectories and adapted there.

Following figure shows an example of possible subdirectories under `msgcatalog`.



**Figure 12: Example of directory structure of the CMICATV5 installation subdirectory `msgcatalog`**

The English message files are located directly under the directory `msgcatalog`.

## Customer dependent configurations for CATIA V5

### *Environment settings*

Environment Variable	Comment
<code>CMI_CREATE_V4_WITH_PARENT</code>	If set the CreateV4 Command only works in the Product Structure with a selected CatPart
<code>CMIXMAP</code>	The location of the Catia V5 Exchange Map
<code>CMI_DEBUG</code>	If Set to 'ON' the Debug output is written to stdout, else no output is created
<code>CMI_REMOVE_CMIREADCMD</code>	If Set the Read Command is not available
<code>CMI_REMOVE_CMIUPDATECMD</code>	If Set the Update Command is not available
<code>CMI_REMOVE_CMICREATECMD</code>	If Set the Create Command is not available
<code>CMI_REMOVE_CMISAVEASCMD</code>	If Set the SaveAs Command is not available
<code>CMI_REMOVE_CMIUPDATECREATECMD</code>	If Set the Synchronize Command is not available
<code>CMI_REMOVE_CMICREATEV4CMD</code>	If Set the CreateV4 Command is not available
<code>CMI_REMOVE_CMISAVEASV4CMD</code>	If Set the SaveAsV4 Command is not available
<code>CMI_REMOVE_CMIINFOCMD</code>	If set the Info command is not available

CMI_REMOVE_CMIHLINWBCMD	If set, the Highlight in Workbench command is not available
CMI_REMOVE_CMIUPDATECREATEINTERACTIVECMD	If set, the Update and Create Interactive command is not available
CMI_REMOVE_CMIINFOCMD	If set, the CMI Info command is not available
CMI_SAVEAS_V4_WITH_DIRTYCHECK	If Set the SaveAsV4 Command only works with saved CatParts
CMI_SAVEAS_V4_WITH_CMICHECK	If Set the SaveAsV4 Command only works with CatParts from CMI
CMI_USE_COMMAND_SUBMENU	if Set to "OFF", button similar-command grouping is turned off in the toolbar.
CMI_CALC_BBOX	Calculate and save bounding box info at CATParts on update (for DMU)
CMI_DISABLE	Set to 'ON' to get NO CMI toolbar. No check for CMI license is done.
CMI_ENABLE_VP_SAVE	Set to '1' to get the Functionality Save Virtual product
CMI_CALC_SHEET	When a drawing is saved, make the list of sheets available in Teamcenter
CMI_REFERENCE_OF_DRAWING	When a drawing is saved, make the referenced Products/Parts available in Teamcenter
CMI_BOUNDING_BOX_EXCLUDE_HIDDEN	Set to "ON" to exclude hidden objects from the bounding box calculation
CMI_DISABLE_HIDESHOW	Set to "ON" to load the hide /show status as in the product (products not in the CMI Workbench are not hidden)
CMI_DISABLE_SET_TIMESTAMP	Set to "ON" to disable to set the last modification date to the files in the exchangemap.
CMI_REMOVE_CMIADDTEMPCMD	Set to "ON" to disable the AddTemp Command
CMI_ADDTEMP_PRAEFIX1	Default is "TMP" the first praefix for the rename of the Partnumbers and Filename for the AddTemp command
CMI_ADDTEMP_PRAEFIX2	Default is "_" the second praefix for the rename of the Partnumbers and Filename for the AddTemp command The Default Praefix is set to TMP#_ # is a counter in catia
CMI_ENABLE_CHECKMODTIMESTAMP	Default is "ON". Set to "OFF" to disable the fuctionality to check for "Saved" Files, which are saved by the native CATIA Save command; also, if Set to "OFF" out of sync cgr files in the local cache are not treated by CMI.
CMI_ENABLE_CACHEMODE_RESETDOCLINKS	Default is "OFF". Set to "ON" if it should be attempted to reset(refresh) document links in Cache Mode. Default is OFF as R12 sp3 can't refresh geometric document links properly
CMI_ENABLE_CMIUPDATEPOSITIONCMD	Default is NULL. Set in "ON" to enable the Update Position Command
CMI_ENABLE_CMICREATEFORDOCCMD	Default is NULL. Set to "ON" to enable the Create For Doc Command
CMI_ENABLE_CMIGETORIGGEOCMD	Default is NULL. Set to "ON" to enable the Get orig. Geometry Command

CMI_USERRELEASEDCACHE	Default is NULL. Set to "ON" to transfer CGR-files to Released Cache
CMI_RELEASEDCACHEDIR	Only used if CMI_USERRELEASEDCACHE=ON.Sets the Released Cache dir used by CMI to a specific member of the list of Released Cache directories in CATIA. Default: not set -> CMI uses the first member of the list. If set to a member, this member must be part of the list.
CMI_CLEANRELEASEDCACHE	Only used if CMI_USERRELEASEDCACHE=ON.Default is "OFF" Set to "ON" if you do not use version independent filenames
CMI_CREATETEMPCGRCOMP	Only used if CMI_USERRELEASEDCACHE=ON.Default is "OFF" Set to "ON" to use temporary CATIA components which contain the related CGR as shape representation, instead of the original CATPart.
CMI_CHECK_LINKED_DRAWING	If set to "ON" then with Create / Create & Link / Save As the CatPart is checked for a related opened Drawing
CMI_CHECK_LINKED_PRODUCT	If set to "ON" then with Create / Create & Link / Save As the CatPart is checked if it is opened in another product
CMI_DEFAULT_UNIT	If no unit of measurement is passed from the PDM system to CATIA, then the unit may be set here.
CMI_CONFIGURATION_FILE	Full filename and path to the CMI XML Configuration file (alternative/complement to system environment variable declaration)
CMI_ENABLE_SINGLEPARTMODUS_READ	If set to "ON" , the "Single Part Modus" option in the CMI Options CATIA V5 property page is enabled
CMI_CONNECTPDM	String to use to override the default command-line omfcl call
CMI_ENABLE_NATIVEPRODUCTRAFO	Set "ON" to enable the option to suppress CMI Transformation
CMI_ENABLE_APPLY_VISUMODE	If set to "ON", CMI will switch CATParts that it loads in Design mode, back to Visualization mode if possible (supported by CATIA beginning R13)
CMI_RESTORE_POSITION	"ON" / "OFF" --> if "ON" user can reset Matrix position to original CMI Matrix position within CATIA V5
CMI_ENABLE_REMOVED_DOCCOM	Default is "ON", Set to "OFF" will disable support for removed Models in TC, if "ON" the removed models will be highlighted at Read Command
CMI_DRAWING_CHECKUPDATEOFASSEMBLY	If Set to "ON" CMI will warn if you update a CATDrawing when at least one related CATPart / CATProduct is not saved yet
CMI_ENABLE_UPDATEPOSITIONDIALOG	If set to "ON" then show a dialog of modified positions at update
CMI_DISABLE_LOAD_STDCATPARTS	If set to "ON" then standard CATParts will not be loaded into design mode automatically. They are identified not by their part number (which is not available in cache mode) but by their instance name, which must be "Part Number + .(dot) + some string".
CMI_IGNORE_NONCMI_ROOT_CHILDREN	If set to "ON" and "Use Virtual Root" is turned on in the CMI Settings in CATIA, then any children attached to the virtual root that are not from CMI will be ignored during Update/Sync commands
CMI_DISABLE_SAVETOXMAP	Set to "ON" if files located outside of the exchange-map should NOT be moved into the exchange map before they are Created/updated in the PDM System

CMI_PACK_ARCHIVE_CMD	String to use to override the default command-line for packing/zipping archive files. Default is "zip -0 -q -j", where -0 is "store only", -q is "quiet operation", -j is "junk (don't record) directory names"
CMI_UNPACK_ARCHIVE_CMD	String to use to override the default command-line for unpacking/unzipping archive files. Default is "unzip -o -j -q -d", where -o is "Override without prompting", -j is "do not use Directory names", -q is "quiet mode", -d <Dir> "extract to dir"
CMI_REMOVE_CMIATTARCCMD	If set, the Attach Archive command is unavailable
CMI_GLOBAL_DISABLE	If set to "ON", the CMI General Update Addin commands are disabled
CMI_DISABLE_REPLACE_WRONG_PRD	If set to "ON", the following functionality is disabled: During a Read, if CMI recognizes that a file with a different UUID has been received from the PDM System instead of the file UUID named in the parent CATProduct, then the new file is attached in place of the old.
CMI_ENABLE_VALIDATE_BEFORE_UPD	Set to "ON" to enable customer specific validation of all operations in Synchronize Command before a sync may be executed.
CMI_DISABLE_STEP_SYNC	Set to "ON" to remove the "Synchronize operations singly" check box from the Synchronize dialog
CMI_DISABLE_NEW_CGR	If set to "ON", this disallows the addition of new CGR files to the product structure
CMI_DISABLE_NEW_CGR_INSTANCE	If set to "ON", this disallows the addition of new instances of CGR files to the product structure
CMI_DISABLE_NEW_V4MODEL	If set to "ON", this disallows the addition of new V4 Model files to the product structure
CMI_DISABLE_NEW_V4MODEL_INSTANCE	If set to "ON", this disallows the addition of new instances of V4 Model files to the product structure
CMI_DISABLE_ANALYSIS_IN_ARCHIVE	If set to "ON" the support for CATAnalysis in CMIArchives is disabled.
CMI_ENABLE_ANALYSIS_COMPUTATION_IGNORE	If set to "ON" the the computations in CATAnalysis is ignored, else the computations must be deleted by the user.
CMI_DISABLE_MODEL_IN_ARCHIVE	If set to "ON" the support for V4 models in CMIArchives is disabled.
CMI_DISABLE_CGR_IN_ARCHIVE	If set to "ON" the support for cgr in CMIArchives is disabled.
CMI_ENABLE_ARCHIVE_ROOT_PRODUCT_ONLY	If set to "ON" the Root in CMIArchives must be a CATProduct.
CMI_REMOVE_CMIMODNCARCCMD	If set to "ON" the Modify non CATIA command is not available
CMI_ENABLE_UPD_MODELSELECT_DIALOG	If set to "ON" a dialog for model update selection is shown in update command
CMI_ENABLE_CMIMULTIQTYSYCMD	If set to "ON", the Build as Multi Quantity command is available
CMI_ENABLE_CHECKISUPTODATE	If set to "ON" the user can cancel the Update /Create / Save As action if a CATPart / CATProduct is not synchronized in the CATIA Session
CMI_ENABLE_CMIEXTERNALDOCCMD	If set to "ON" the command to load referenced documents is available. Additional software is required for this.
CMI_DISABLE_LOADOK_MESSAGE	If set to "ON", no message is shown after a successful Read from Teamcenter
CMI_DISABLE_UPDATE_WB	If set to "ON", the Workbench is not updated with new items created during Synchronize
CMI_ENABLE_CHECKISUPTODATE	If set to "ON", a check is performed during Update if any geometry needs to be updated in Catia

CMI_ENABLE_CHECKMULTIEMBARC	If set to "ON", ambiguous (same partnumber) local components are disallowed in an archive
CMI_GETPOINTEDDOCUMENTS	if set to "ON", dependencies based on referenced documents are created in Teamcenter. This requires additional software.
CMI_ENABLE_CHECK_PRD_VAL_IGNORE	If set to "ON" the validate function of Synchronize succeeds if a product with no structural changes is modified and has to be updated
CMI_ENABLE_CMIOPTIONSCMD	If set to "ON" the CMI Options dialog command button is available
CMI_ENABLE_STDPARTINFO	If set to "ON" the standard Part infos are requested from TC, needed for the Component standard part integration
CMI_REMOVE_CMIUSEPDMSTRUCTURECMD	If set to "ON" the "Insert from Teamcenter"-Command is not available
CMI_ENABLE_RESETINVALIDPOSCMD	If set to "ON" the "Reset Invalid Position"-Command is available
CMI_ENABLE_CHECK_INVALID_POS	If set to "ON" during Update all models are checked for invalid positions
CMI_BOM_PART_DEFAULT_FOR_SYNC	Set the default value for new Models in the sync dialog: NOT_SET, BOM, all other settings are normal models
CMI_ENABLE_ASK_FOR_BOM_PART	If set to "ON" the user is asked which kind of Part should be created in TC
CMI_ENABLE_CMIARCHIVE_CREATE	If set to "ON" it is possible to create (Attach) archives without parent and without CMI Parent
CMI_CAT_ENV_SCRIPT	points to the cat start file (full path with extension) which is used to start the CMISender executable(Used in Omf and CNEXT environment), CNEXT startup will create the file for the actual CATIA and the Omf will use this for starting CMISender
CMI_DISABLE_CMIBUILDVISUCMD	If set to "ON" the CMI BuildVisu Command is not available
CMI_DISABLE_CREATE_CATDRAWING	If set to "ON" the creation of new CATDrawings is disabled
CMI_DISABLE_CREATE_CATPART	If set to "ON" the creation of new CATParts without Parents is disabled
CMI_DISABLE_CMIRECONNECTCMD	If set to "ON" the CMIREconnect Command is not Available
CMI_ENABLE_ARCHIVE_CACHE	If set to "ON" the archive file can handle Release cache files
CMI_PDM_MANAGED_STDCATPARTS	If set to "ON" then CMI treats standard part geometry (CATPart files) like regular component part geometry, that is, the geometry is expected to be attached to a document describing the Component in PDM and it is transferred to the exchange map at "To Catia".
CMI_PACK_ADD_ARCHIVE_CMD	String to use to override the default command-line for Adding files to archives. Default is "cml_zip -0 -q -j", where -0 is "store only", -q is "quiet operation", -j is "junk (don't record) directory names"
CMI_PACK_ARCHIVE_MAX_FILES	Define the maximal number of files which will be packed with the Zip command. If not set, Default is 100. Used to reduce the needed length of the command line for the system call.
CMI_NO_PUBLIC_POS_UPDATE	Do not try to update transformations under products that are read only
CMI_PACK_ARCHIVE_MAX_COMMANDLINE	Define the maximum length (in Bytes) of the system call which will pack a CMI Archive. If not set use system limits of the OS. If the limit is reached, the pack Archive Command will be split (see CMI_PACK_ADD_ARCHIVE_CMD)
CMI_ARCHIVE_AUTOUPDATE	Automatic update after "Attach an Archive"

CMI_REPLACE_WRONG_PRD_AUTO	If set to "ON" The confirmation dialog for the following functionality is disabled: During a Read, if CMI recognizes that a file with a different UUID has been received from the PDM System instead of the file UUID named in the parent CATProduct, then the new file is attached in place of the old
CMI_ENABLE_DEACTIVATED_CHECK	If set to "ON" the active window is searched for deactivated products, if deactivated are found, the update operation is not allowed
CMI_ENABLE_PARTINFO_FOR_MODEL	If set to "ON" the more info button will provide information about the Part in TC (instead of only the data item)
CMI_DISABLE_DT_IN_ARCHIVE	If set to "ON" CMI does not store DesignTables in Archive even if CMI_DESIGN_TABLES is set in Teamcenter
CMI_USE_DTFORPRODUCT	If set to "ON" CMI also handles DesignTables for CATProducts if CMI_DESIGN_TABLES is set in Teamcenter
CMI_REP_FORMATS	Sets the Representation formats, which should be handled. Example: "{wrl} {stl}"
CMI_DISABLE_REP_IN_ARCHIVE	If set to "ON" the support for Representations in CMIArchives is disabled.
CMI_REMOVE_UNKNOWN_FILES_IN_XMAP	If set to "ON" unknown files in CMIXMAP will be subject to cache size management
CMI_XMAP_CACHE_SIZE	Maximum size in MB of cached files in CMIXMAP after closing CATIA
CMI_RELMAP_CACHE_SIZE	Maximum size in MB of cached cgr files in CMI_RELEASEDCACHEDIR
CMI_DISABLE_CMISAVELOCALCMD	If set To "ON" the CMI SaveLocal Command is not available
CMI_DISABLE_CMIRESTORELOCALCMD	If set To "ON" the CMI RestoreLocal Command is not available
CMI_USE_91_TOOLBAR	If set to "ON" the 9.1 Toolbar style (only one toolbar) is used
CMI_GET_BOMTYPE_FROM_TC	Set to "ON" to get the Bom-Type from Teamcenter for new Catia-Files
CMI_ENABLE_CMICATALOGREADCMD	Set to ON will enable the Catalog Read Command in CATIA V5.
CMI_ENABLE_CMICATALOGUPDCRECMD	Set to ON will enable the Catalog Create/Update Command in CATIA V5
CMI_DELETE_STALE_DESIGNTABLES	If set to "ON" DesignTables in CMIXMAP that are no longer referenced by a CATPart/CATProduct will be deleted.
CMI_USE_FILENAME_WINDOWTITLE	If set to "ON" the title of windows loaded by CMI shows "CMI - <Filename>" instead of the default: "CMI - PartNumber>"
CMI_ENABLE_CMICATDUAREAD	If set to "ON" Enable the CMICatDuaRead Command in the toolbar
CMI_CATDUAV5_CONFIG	Full path to the CATDUAV5 configuration file
CMI_CATDUAV5_RESULTVIEWER	Viewer to show the CATDUA result (Windows optional / unix required). If set to "OFF" no result is shown
CMI_CATDUAV5_COMMAND	Command to start CATDUAV5 (optional). Default: catstart -run "CATBatchStarter -input @CONFIG@ -output @OUTDIR@"
CMI_ARCHIVE_ALLOW_BROKEN_LINKS	If set to "ON", broken links in CMI Archive are ignored. Default is "OFF" - broken links are not allowed in a CMI Archive.
CMI_PREVENT_DIFFERENT_VERSIONS	If set to "ON", "To Catia" is checked against different versions of same file in Catia and in WB

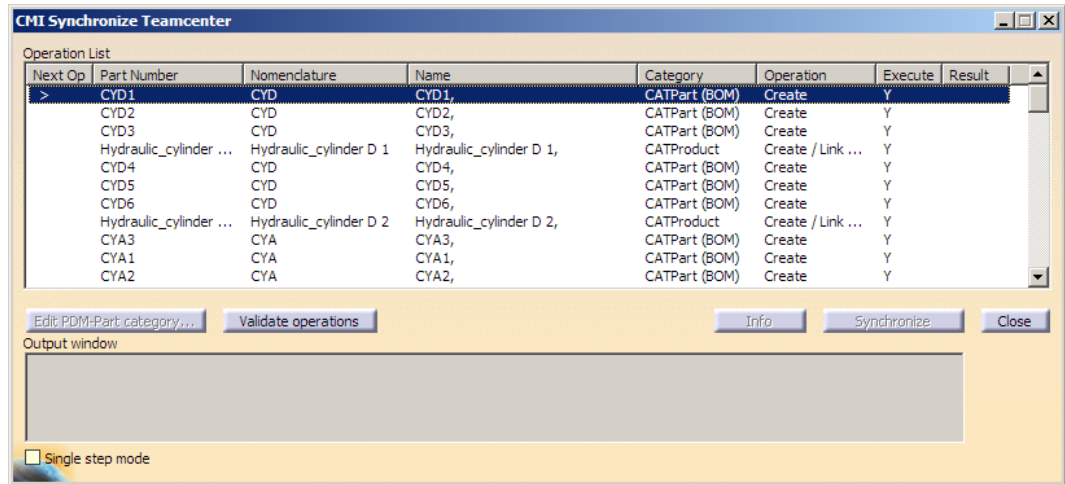
CMI_DYNAMIC_CATALOG	If set to "ON", CMI stores the Part Master ID in CATCatalogs. Manage Catalogs opens always the latest version of the Part, and not a static revision.
CMI_READ_INERTIA	If set to "ON", inertias from CATParts and CATProducts will sent to Teamcenter, if set to "ONLY_CATPART", only the inertias of CATParts will sent to Teamcenter
CMI_CONFINE_INERTIA_TO_MAINBODIES	If set to "ON", the inertias sent to Teamcenter are confined to those of main part bodies.
CMI_ENABLE_CRE_ANALYSIS_ARCHIVE	If set to "ON", the Button CreateArchive is enabled, if the top-level-node is a CATAnalysis.
CMI_ANALYSIS_ARCHIVE_OWN_WINDOW	If set to "ON", CMIArchives with CATAnalysis as top-level-node will load into an own window in Catia.
CMI_NEW_SYNCHRONIZE	If set to "ON" enable the new SynchronizeCmd and disable the old Synchronize, update and Create functions
CMI_SKIP_RO_TEMPLATE	If set to ON, Template CATProducts that were read as Read-only will be skipped during synchronize, ie. they will not cause a warning
CMI_GETORIGGEO_DESIGNMODE	If set to "ON", GetOrigGeoCmd: If all files of the selected structures are located in the exchange directory ask to load the selected assemblies into DesignMode.
CMI_ENABLE_EDUFLAG_CHECK	If Set to "ON" check for educational flag during Update /Synchronize / Create / SaveAs. If Edu flag is set for a file to be saved, the Save action will be declined
CMI_ENABLE_ACTIVATEDDEACTIVATE	If set to "ON" the default value of the user option "Deactivate geometry files not from CMI Workbench" is set to checked. To disable Hide / Show also set CMI_DISABLE_HIDESHOW=ON
CMI_ENABLE_CHECKFOREMBEDDEDCHANGE	If set to "OFF" do not check if there are embedded changes in the structure (Performance) This is permissible if you have not customized the message <b>x3UseEmbeddedProduct</b> in Teamcenter.
CMI_GEOPOS_NOLINK	Set to "OFF" if you use geometry positions but do not link the document to the part when a CATPart is created (default ON)
CMI_DISABLE_SYNC_PROCESS	If set to "ON" the synchronize button is disabled if the current window contains a CATProcess
CMI_HIDESHOW_ADDTOSESSION	If set to "ON" HideShow / ActivateDeactivate only affects CATIA Windows where the Top-Level Items are available in the current CMI Workbench.
CMI_ENABLE_CREATE_IMPORT_FILE	If set to "ON" the check box "Write Import/Export Mapping File" is available in the Synchronize dialog and a Mapping file can be used in Reconnect Teamcenter
CMI_ENABLE_EXPORTCMD	If set to "ON" the "Export to Folder" Command is available in the CMI Toolbar
CMI_EXPORT_STDATTRIBUTES	To set the CATIA standard attributes Nomenclature, Revision, Description or Definition back to the original value of the initial import set the value of CMI_EXPORT_STDATTRIBUTES to the attributes you want to set back. Only used with the CMI Export Cmd.
CMI_REMOVE_CONTEXTMENU	Set to "ON" to disable the contextual CMI menu in the CATIA Product Structure
CMI_IGNORE_EMBEDDED_LEAFNODE	Set to "ON" to ignore embedded ccomponents with no children
CMI_ENABLE_CMICATALOGINSERTSCRIPTCMD	Set to "ON" to enable the "Insert CATScript from Teamcenter" command

---

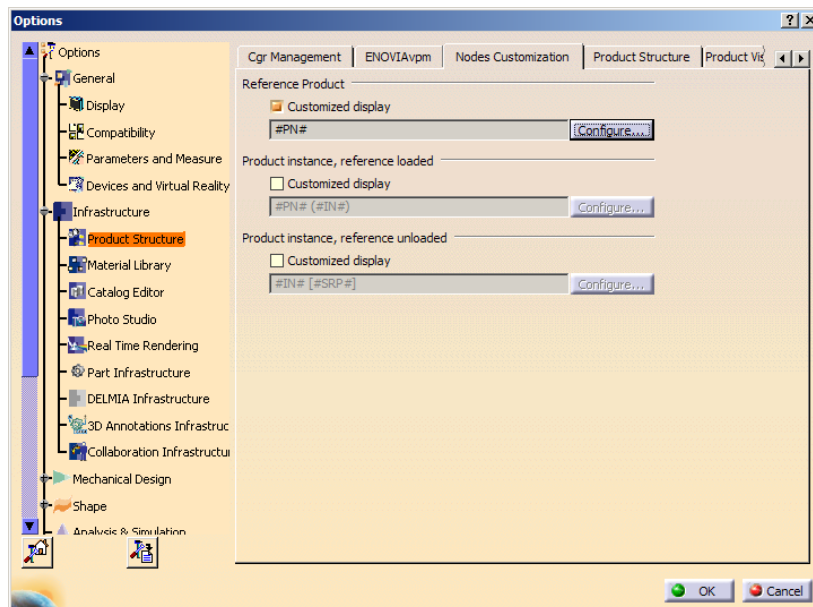
CMI_EXPORT_CUSTOMIZE_NAMING	Set to "ON" to invoke the custom TC methods x3GetExpNamingSchema, x3GetExportRefNames and x3GetExportInstNames. This option must not be set if x3GetExportRefNames and x3GetExportInstNames are not customized.
CMI_ENABLE_CGM	Set to "ON" to enable the Update, Synchronize, Save As and Save for Doc commands for CGM documents in CATIA

## Display CATIA Node Name in Synchronize

The width and visibility of columns in the Synchronize dialog can be configured by editing the file `intel_a\resources\msgcatalog\CMIUpdateCreateDialog.CATRsc`



E.g. the column **Name** is not shown by default, but it contains the text configured in the customized display for Reference Product:



To make the **Name** Column visible by default, the following setting has to be changed in the file `CMIUpdateCreateDialog.CATRsc`:

```
// Column width of Name  
JobMultiList.ColumnWidth4 = "0";
```

To hide **Nomenclature** and show **Name** instead you have to set:

```
// Column width of Nomenclature  
JobMultiList.ColumnWidth3 = "0";  
// Column width of Name  
JobMultiList.ColumnWidth4 = "16";
```

# CHAPTER 10

## Data Models

The following figures show the entire data model of the CATIA Workbench (CMI) and of the structure of the persistent CMI-classes.

### Data structure Catia-Workbench

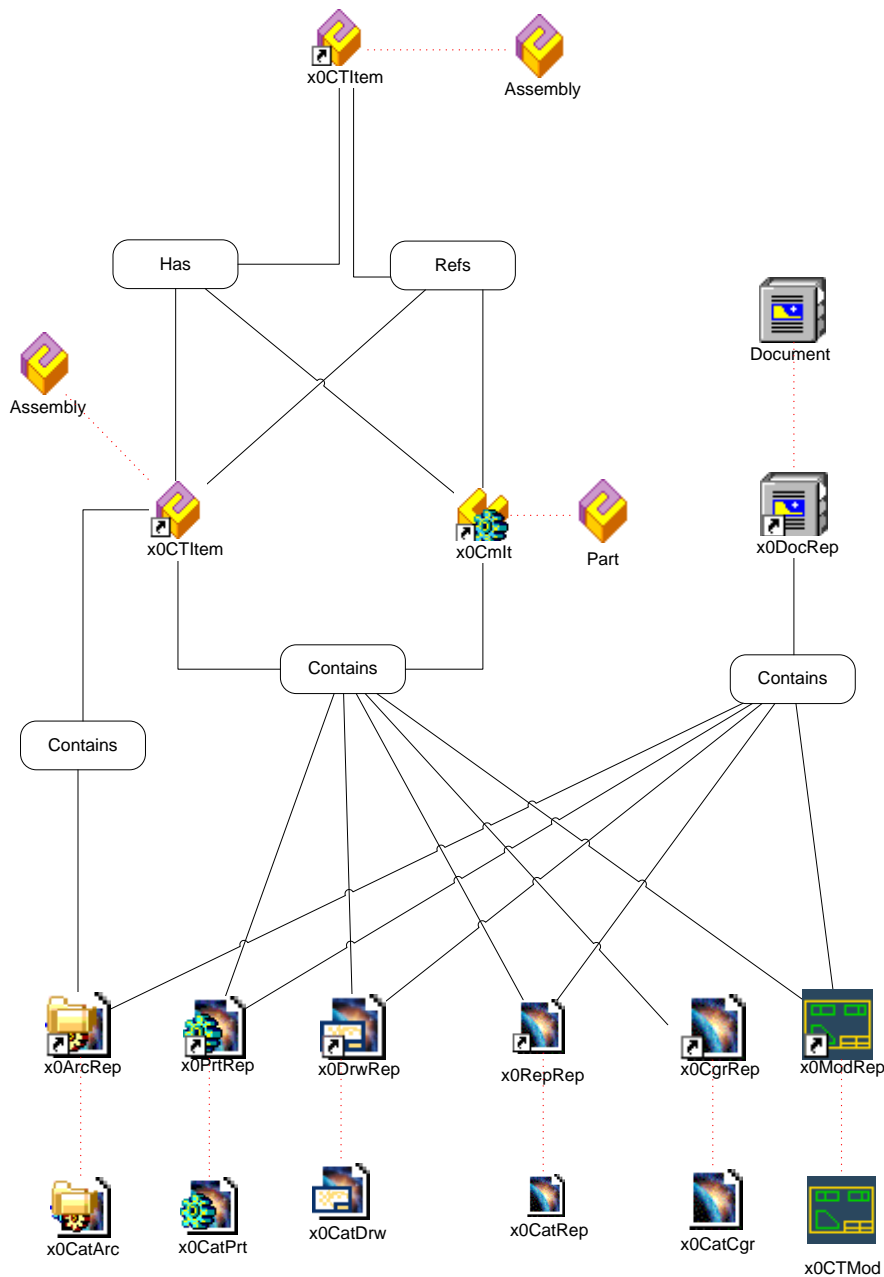


Figure 13: Data Structure of Catia-Workbench

## Data structure of CMI-Classes

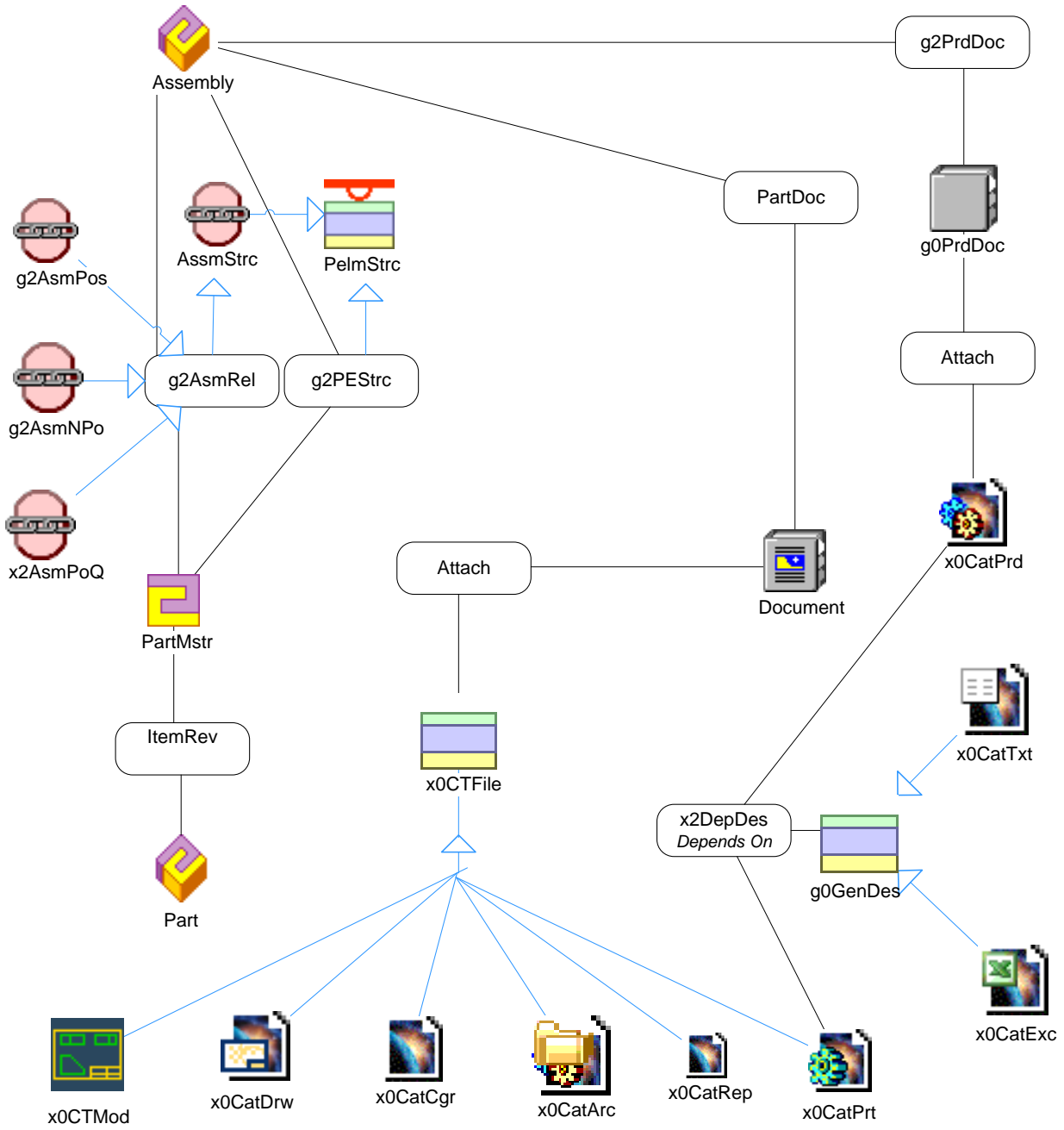
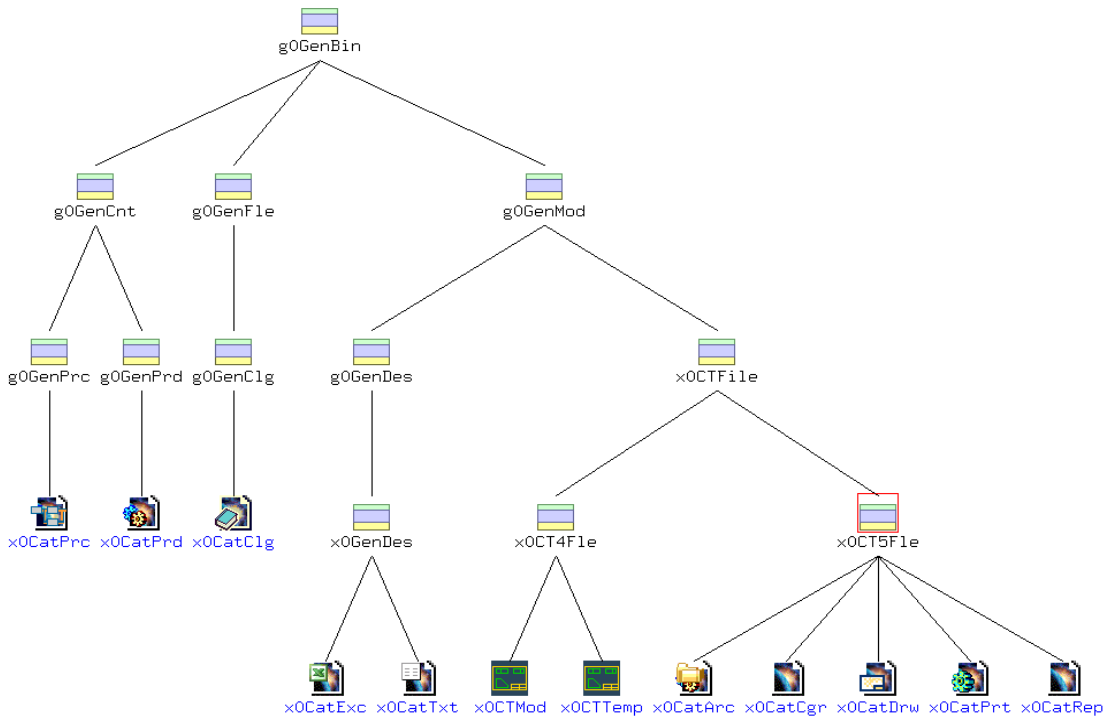


Figure 14: Data Structure of CMI-classes

### ***g0GenBin Class Hierarchy***

Figure 15 shows the class hierarchy under the CMI class g0GenBin.

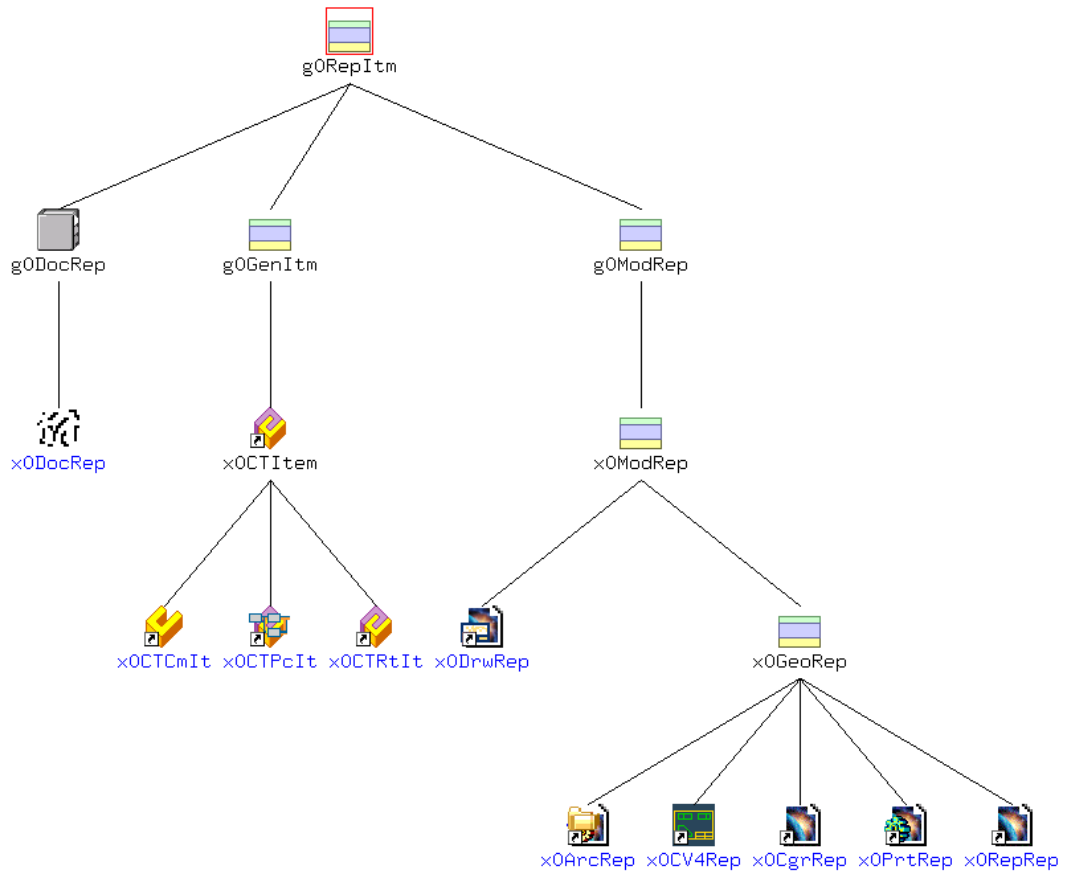


**Figure 15: Class hierarchy under g0GenBin**

- g0GenBin has the parent class IndepBin and is the container for all CATIA data classes
- g0GenCnt is a child class of g0GenBin and is the container class for the CATIA V5 documents, which are not visible in the CATIA workbench.
- g0GenPrc/x0CatPrc is the data class for the CATIA V5 CATProcess file.
- g0GenPrd/x0CatPrd is the data class for the CATIA V5 CATProduct file.
- g0GenFile/g0GenClg/x0CatClg is the data class for the CATIA V5 Catalog file.
- g0GenMod/g0GenDes/x0GenDes/x0CatExc is the data class for a design table in the MS-Excel-Format
- g0GenMod/g0GenDes/x0GenDes/ is the data class for a design table in the Textformat.
- g0GenMod/x0CTFile is a child class of g0GenBin and is the container class for the CATIA documents, which can be visible in the CATIA workbench.
- x0Ct4File/x0CTMod is the data class for the CATIA V4 model file.
- x0CT4File/x0CTTemp is the data class for the CATIA V4 model template file.
- x0CT5File/x0CatPrt is the data class for the CATIA V5 CATPart file.
- x0CT5File/x0CatDrw is the data class for the CATIA V5 CATDrawing file.
- x0CT5File/x0CatCgr is the data class for the CATIA V5 cgr file.
- x0CT5File/x0CatArc is the data class for the CMI Archive file.
- x0CT5File/x0CatRep is the generic data class for the configurable shape representations.

## ***g0Repltm Class Hierarchy***

Figure 15 shows the class hierarchy under the CMI class g0Repltm.



**Figure 16: Class hierarchy under g0Repltm**

- g0Repltm has the parent class g0Pdmltm and is the container for all CATIA representation classes which are visible in the CATIA Workbench.
- g0DocRep/x0DocRep represents a document.
- g0GenItm is the container class for the Part representations.
- x0CTItem is the main Representation class for the Assembly class, x0CTCmIt is the representation class for the Component class. x0CTPcIt is the representation class for a CATProcess root item.
- g0ModRep/x0ModRep is the main container for the Model representations.
- x0DrwRep represents the CATIA V5 Drawing data item class. (CATDrawing).
- x0GeoRep is the container for the geometrical representations.
- x0ArcRep represents the CMIArchive data item class (CMIArchive).
- x0CV4Rep represents the CATIA V4 model data item class (model).
- x0CgrRep represents the CATIA V5 cgr data item class (cgr).
- x0PrtRep represents the CATIA V5 Part data item class (CATPart).
- x0RepRep represents the data item class x0PrtRep.