

# **CATIA Teamcenter Interface**

**CMI Release 9.7**

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**Release Notes**

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## Manual History

Version	Date
9.7	April 2010

This edition obsoletes all previous editions.

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# Preface

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## About this Guide

This guide provides general release information for the CATIA Teamcenter Interface.

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## Related Documents

The following manuals contain information about installation and administration of the CATIA Teamcenter Interface:

Manual Title	Release
<i>CATIA Teamcenter Interface Installation &amp; Administration Guide</i>	9.7
<i>CATIA Teamcenter Customizing Guide</i>	9.7
<i>CATIA Teamcenter User's Guide</i>	9.7

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## Your Comments are Welcome

Your comments on our publications are welcome. Please write us at:

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# CHAPTER 1

## Platform Support

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### Supported platforms

HP-UX 11  
Sun Solaris 8 (10<sup>1</sup>)  
IBM AIX 5.3 32 bit / 64 bit  
SGI Irix 6.5  
Windows XP 32 bit / 64 bit, Vista  
SuSE Enterprise 9.0<sup>2</sup>

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### Supported Teamcenter Releases

Teamcenter 5.0  
Teamcenter 5.1  
Teamcenter 2007<sup>1</sup>

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### Supported Catia V4 Releases

CATIA V4.2.0 – V4.2.4  
CATIA V4.2.5<sup>1</sup>

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### Supported Catia V5 Releases

CATIA V5R17  
CATIA V5R18<sup>1</sup>  
CATIA V5R19<sup>1</sup>  
CATIA V5R20<sup>13</sup>

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<sup>1</sup> Solaris 10 replaces Solaris 8

<sup>2</sup> TC 2007 server side only

<sup>3</sup> AIX6.1 replaces AIX 5.2



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# CHAPTER 2

## Functional Changes

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### PDM Centric Synchronize

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.

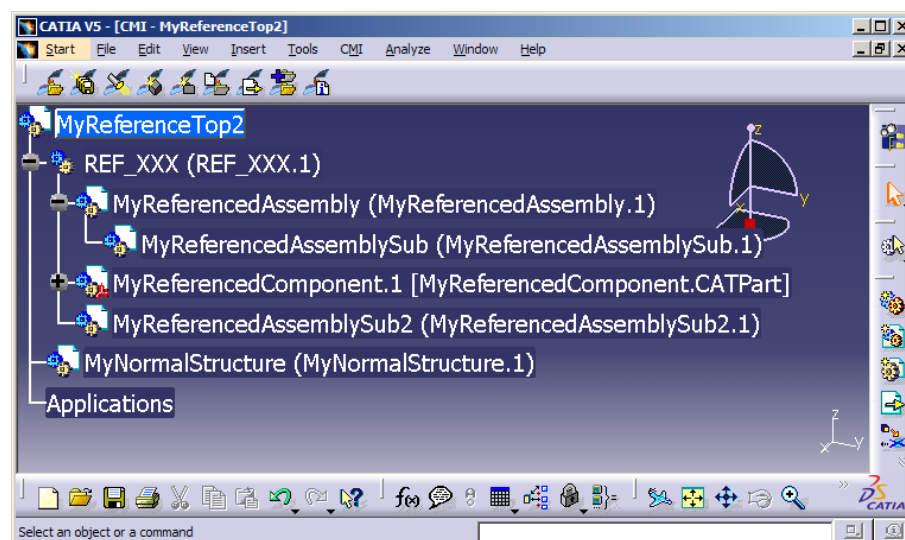
The new process does not support custom validation as of yet.

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.

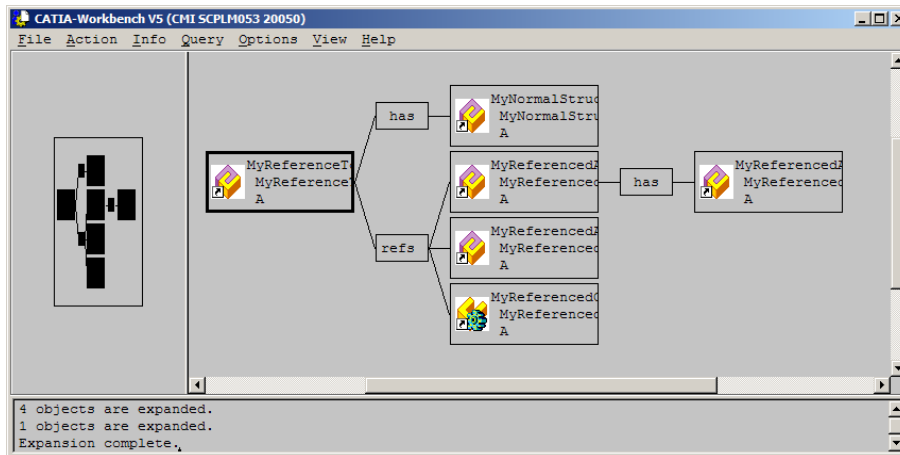
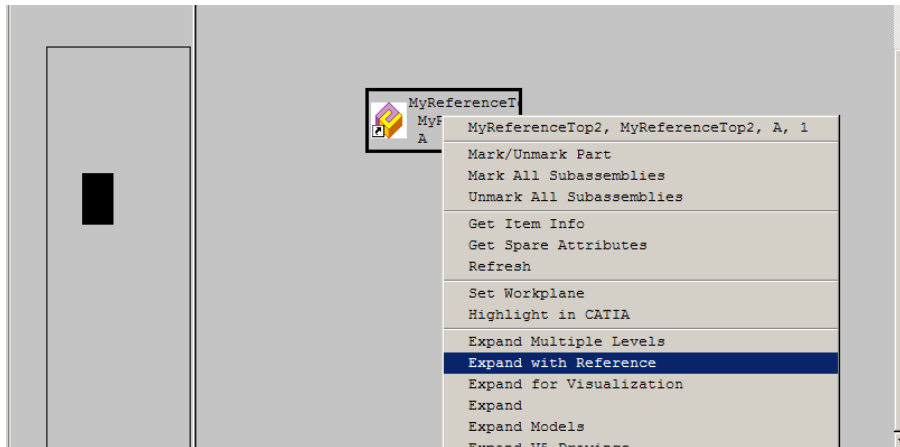
---

### 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.



Reference geometry can be inserted below a CATIA component that adheres to a special, configurable naming convention. Synchronize will create a CMI specific relation between these Parts and the Teamcenter assembly structure. From then on, you can load these reference parts along with the regular assembly structure whenever required.



The reference geometry relation is a product element structure relation. Configuration context applies.

## Multi-use Catalog Browser

With the CMI Manage Catalogs functionality, the Catalog Browser will now stay open so that multiple Parts can be opened in succession.

## CATAnalysis use case with GPS license

CMI supports storing CATAnalysis by way of a CMIArchive. Previously, the CATAnalysis had to be inserted into a CATProduct. A top level CATAnalysis can now be stored, eg. from a single CATPart.

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## **Inertia Handling**

Some configuration options have been introduced to allow for finer control of the transfer of Inertia properties. It is now possible to transfer Inertia only for CATParts. Also it can be limited to Inertia only of main bodies.

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# CHAPTER 3

## New Environment Variables

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### CATIA V5 Environment

CMI\_NEW\_SYNCHRONIZE

If set to "ON" will enable the new Synchronize Command and disable the old Synchronize, Update and Create functions

### *Configuration for Reference Geometries*

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

<?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>
```

### *Handling of Top Level CATAnalysis*

CMI\_ENABLE\_CRE\_ANALYSIS\_ARCHIVE

Set to ON to allow saving a top level CATAnalysis document as a CMIArchive

CMI\_ANALYSIS\_ARCHIVE\_OWN\_WINDOW

Set to ON to make CMIArchives containing a CATAnalysis open up in a separate window

### *Inertia*

CMI\_READ\_INERTIA

If set to ONLY\_CATPART, Inertia will only be calculated for CATParts, not for CATProducts.

CMI\_CONFINE\_INERTIA\_TO\_MAINBODIES

If set to ON, Inertia will only be calculated from main bodies.

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## *Templates*

### 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.

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## **Teamcenter Configuration**

n/a

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# CHAPTER 4

## New Customizing Points

The following customization methods have been introduced in order to handle the “Has Reference Geometries” product structure relation:

```
g0GenWB:g3ExpandRefsCus  
g2Relatn:x3DelAssmStrcRel  
g0GenWB:GetPartsInAssSetInt
```

The following customization methods have been changed in order to accommodate for Reference Geometries:

```
x0WkBnch:x3GetRelevRelViaObid  
x0CTItem:x3GetSubAssRelsForObid  
Part:x3GetRelToBeUpdated  
g0GenWB:g3GetPartsInAssSet
```

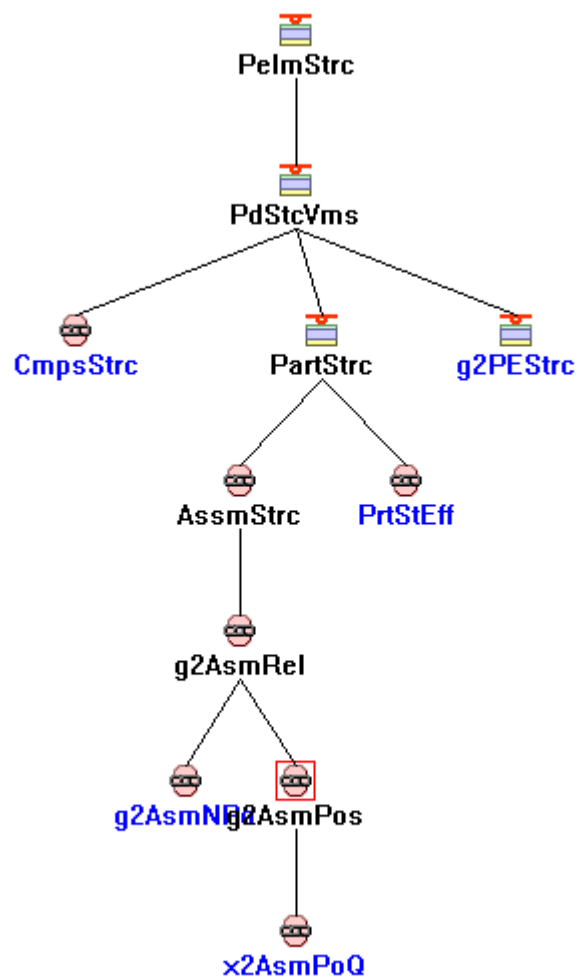
If you have overridden any of these methods please consult the files `g3Custom.mth/x3Custom.mth` for necessary changes.

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# CHAPTER 5

## Data Model Changes

A new product structure class *g2PEStrc* has been introduced for Reference Geometries.



The dynamic revision-revision counterpart for *g2PEStrc* is *g2PERvRv*. The relation is created via CATAI V5 and can be expanded in the CMI Workbench by the *Expand with Reference* menu item. It is not meant to be created in OMF.



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# CHAPTER 6

## Bug Fixes

If you encounter any problems please contact us at [cmi\\_support@t-systems.com](mailto:cmi_support@t-systems.com)