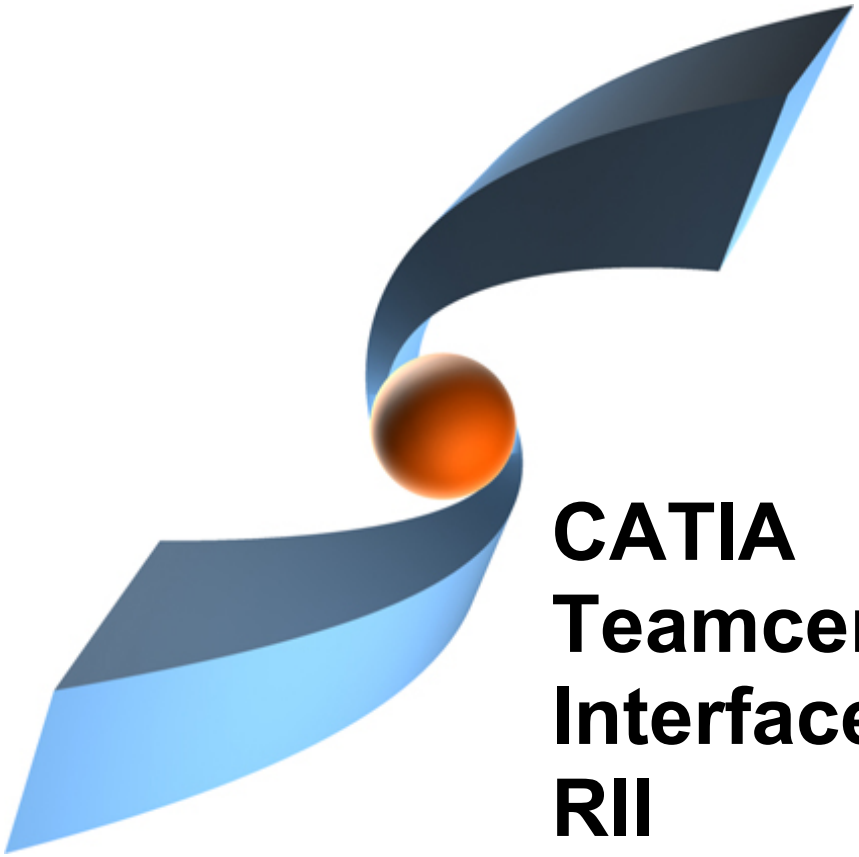

CATIA Teamcenter Interface RII

CMI RII Release 2.0

User Manual

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Contact

T-Systems Enterprise Services GmbH
Solution Center PLM
Fasanenweg 5
70771 Leinfelden-Echterdingen
Germany

http://www.cmi-support.com

☎ +49-711-972-40 304
✉ +49-1805-3344910453
mail: *cmi_support@t-systems.com*

Manual History

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1.0	December 2008
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This edition obsoletes all previous editions.

Trademarks

Preface

This manual describes the main functionality delivered by the CATIA Teamcenter Interface (CMI RII) and includes the creation, storage, modification, and management of CATIA models and assembly structures in the Teamcenter PDM system.

About this Manual

This manual is intended for end users of the CATIA Teamcenter Interface. It assumes that the reader is familiar with the CATIA application and with Teamcenter Rich Client.

Related Documents

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

Manual Title	Version
CATIA Teamcenter Interface RII Installation Manual	2.0
CATIA Teamcenter Interface RII User Manual	2.0

Organization

This manual contains the following chapters:

Chapter 1 provides basic information about the CATIA Teamcenter Interface and describes some features of this application interface.

Chapter 2 describes how to start CATIA within the Teamcenter environment and also how to enable the CMI functionality within CATIA V5.

Chapter 3 introduces the *CMI RII Application* with a general explanation and then by listing its functions. The goal of this chapter is to make the user familiar with this component. The *CMI RII Application* functions will be explained in subsequent chapters.

Chapter 4 describes how to handle CATIA relevant data in Teamcenter.

Chapter 5 explains how to work with CATIA V5.

Glossary contains the CATIA Teamcenter Interface terminology.

Conventions Used in this Manual

This font	Is used for document titles and emphasis.
Item 1→Item 2	Is used to identify a menu path to reach a specific function.
	This icon is used to identify tips and attention advises.
	This icon is used to identify Teamcenter related sections and actions.
	This icon is used to identify CATIA V5 related sections and actions.

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 Enterprise Services GmbH
Solution Center PLM
Fasanenweg 5
70771 Leinfelden-Echterdingen
Germany

E-mail: *cmi_support@t-systems.com*

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CHAPTER 1

Overview

This chapter provides basic information about the *CATIA Teamcenter Interface* and lists some features of this application interface.

Introducing CMI (CATIA Teamcenter Interface)

The *CATIA Teamcenter Interface* (CMI RII) was developed by T-Systems as a high-end integration between the CAD system *CATIA* V5 and the PDM system Teamcenter. With this interface it is possible to manage *CATIA V4* and *CATIA V5* models and assemblies in *Teamcenter* and *CATIA V5*.

CATIA V5 uses assemblies similar to *Teamcenter*. CMI makes a bidirectional mapping between the *Teamcenter* structure and the *CATIA V5* structure. So users have the full functionality of *Teamcenter* and *CATIA V5*.

CMI RII joins the advantages of CAD with PDM

The *CATIA Teamcenter Interface* combines the CAD Excellency of CATIA with the power of the PDM system Teamcenter. It provides the user with a more sophisticated way of working with CATIA by allowing the management of product structures and multiple level assembly structures within the PDM system.

The CATIA Teamcenter Interface (CMI RII) permits:

- Integration of CATIA data in workflow (e.g. release control);
- Management of CATIA data in vaults, without knowledge about the underlying file system;
- Updating concurrent engineering processes by different users;
- Distribution of CATIA data in a network;
- Simultaneous management of CATIA data and structures;
- Construction of part structures within Teamcenter;
- Modification of the position of the structures;
- Search for CATIA data by different attributes.

CMI Architecture

The following figure explains the architectural basics of the CATIA Teamcenter Interface (CMI RII).

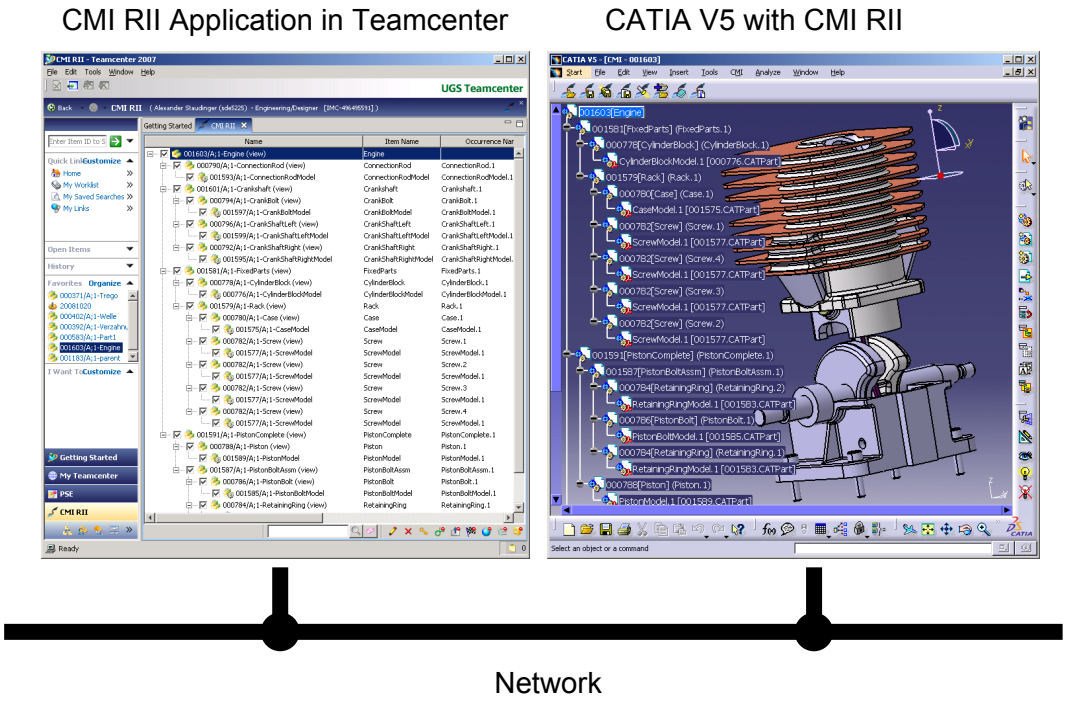

Figure 1: CATIA Teamcenter Interface architecture

The user can expand an assembly within the CMI RII Application and send this assembly to CATIA V5. After changing geometries and/or positions in CATIA the user can update the assembly in Teamcenter.

CHAPTER 2

Getting Started

This chapter describes how to start CATIA within the Teamcenter environment and also how to enable the CMI functionality within CATIA V5.

Assumptions

Some assumptions, for a better understanding of the descriptions and examples:

CATIA V5 should display the CMI toolbar:

UNIX:
Start CATIA V5 with the command cmicatstart.sh

Windows:
Make sure you have set up the right environment (refer to Installation and Administration Guide). Launch CATIA V5 with the command:
CNEXT.exe -env CatiaCMIEnv -direnv "C:\<path to the CMIEnv>"

Starting Applications

Before working with CMI RII it is necessary to start the Teamcenter Rich Client, the Teamcenter user interface, and the CATIA V5.

For information on starting Teamcenter Rich Client see the Teamcenter Help Library – Getting Started with Teamcenter provided by Siemens PLM.

For information on starting the CATIA User Interface see the CATIA Solutions User's Guide.

It makes no difference if Teamcenter or CATIA is started first.

Enabling the CMI toolbar in CATIA V5

In CATIA V5, select the menu **View→Toolbars→CMI**.

If the CMI toolbar is not in the list of toolbars, a problem with the installation may have occurred. In this case, contact the *administrator*. Information about this kind of problem can be found in the Installation Manual.

CHAPTER 3

The CMI RII Application

This chapter introduces the *CMI RII Application* with a general explanation and then by listing its functions. The goal of this chapter is to make the user familiar with this component. The *CMI RII Application* functions will be explained in subsequent chapters.

What is the CMI RII Application?

The *CMI RII Application* is a Teamcenter application used to interactively prepare an assembly structure for sending to CATIA and to provide several manipulation facilities for the CMI commands found in CATIA. The entire data interchange between Teamcenter and CATIA is achieved via the *CMI RII Application*.

CMI RII defines some new dataset types in Teamcenter that can be used to manage CATIA V5 files (CATPart, model, cgr, CATProduct, and CATDrawing). The management of CATIA V5 files is described in *Chapter 5*.

The CMI RII application has to be started in Teamcenter before you can call the functions of CMI RII.

To start the *CMI RII application* use the standard Teamcenter start mechanism as shown in Figure 2.

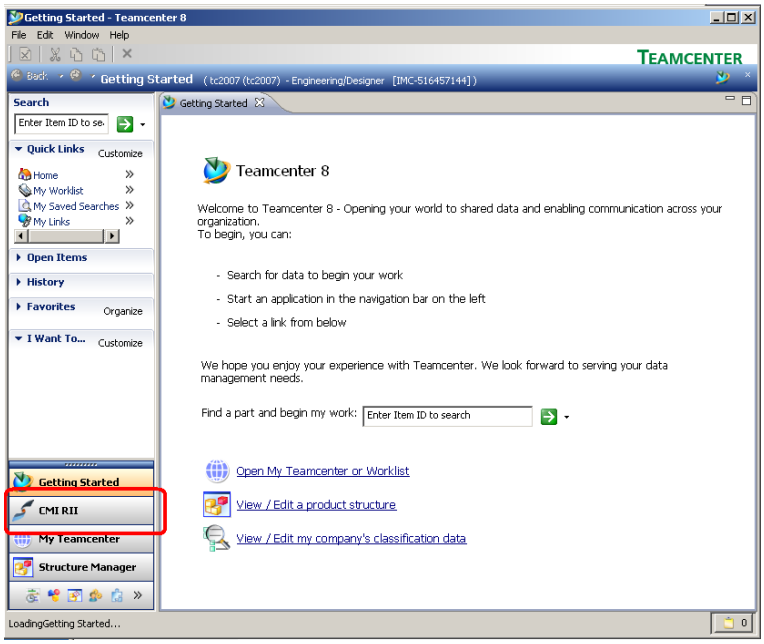

Figure 2: Teamcenter Rich Client start window

A window with the CMI RII application will appear after logging in in the PDM system Teamcenter (see Figure 3).

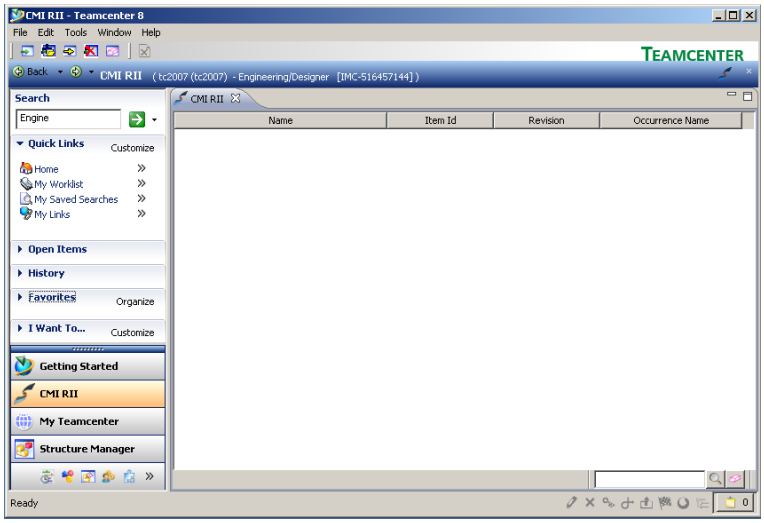

Figure 3: CMI RII application window

The CMI RII application Menu

Below the CMI RII application menu items will be described.

Figure 4: CMI RII application menu items

Menu item File

Figure 5: CMI RII application menu 'File'

The menu item *File→Close* will close the CMI RII application.

The menu item *File→Exit* will exit the Rich Client application.

Menu item Edit

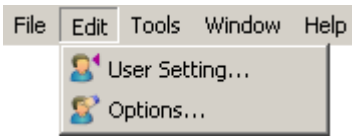

Figure 6: CMI RII application menu 'Edit'

The menu item *Edit→User Setting …* will open the User Setting dialog of the Rich Client application.

The menu item *Edit→Options …* will open the Preferences dialog of the Rich Client application.

The CMI RII preferences show the CMI RII options (see Figure 7).

- Send Drawings To CATIA
  Checked if drawings (CATDrawing) should be sent to CATIA.
- Transfer CGR files To CATIA
  - No — No CGR are sent to CATIA
  - Only.CGR — Only CGR are sent to CATIA
  - CGR + geometry — CGR and geometry files are sent to CATIA
- Work with CATIA Version
  - V5 — work with CATIA V5
  - V4 — work with CATIA V4

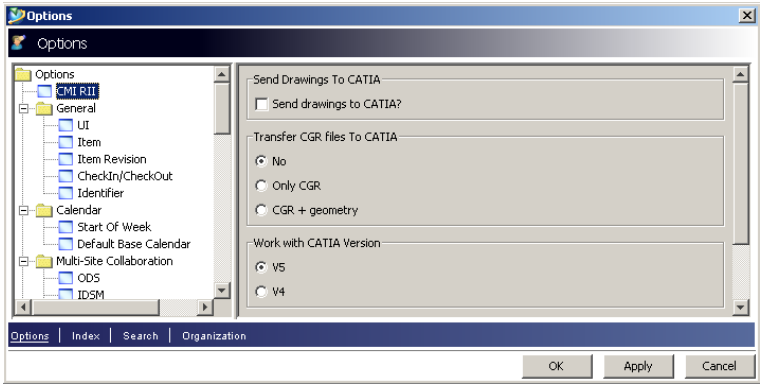

Figure 7: The CMI RII application options dialog

Menu item Tools

Figure 8: CMI RII application menu 'Tools'

The menu item *Tools→ Send To Catia* starts the Read To Catia Action and will read the CMI RII application content to the CATIA.

The menu item *Tools→ Temp To Catia* starts the Add Temp To CATIA Action and will read the temporary added items to the CATIA. This menu item is only usable if the Add Temp action is started from the CATIA.

The menu item *Tools→Insert To Catia* starts the Insert To CATIA Action and will read the queried items to the CATIA. This menu item is only usable if the Insert from Teamcenter action is started from the CATIA.

The menu item *Tools→Cancel Catia Action* cancels actions which are started from CATIA. (e.g. *SaveAs* and *AddTemp*). This menu item is only usable if an action is started from the CATIA (e.g. *SaveAs* and *AddTemp*).

The menu item *Tools→Clear Workbench Action* removes all Items from the CMI RII Application window.

Menu item Window

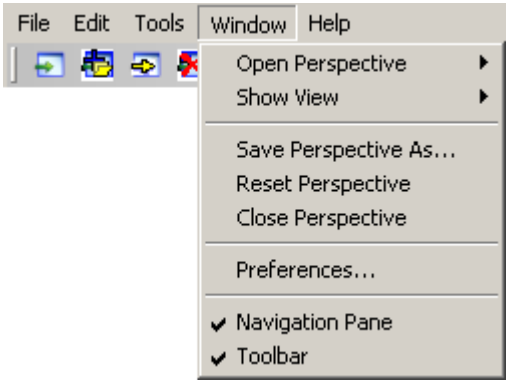

Figure 9: CMI RII application menu 'Window'

The menu item *Window* includes the standard menu items provided by the Teamcenter Rich Client application.

Menu item Help

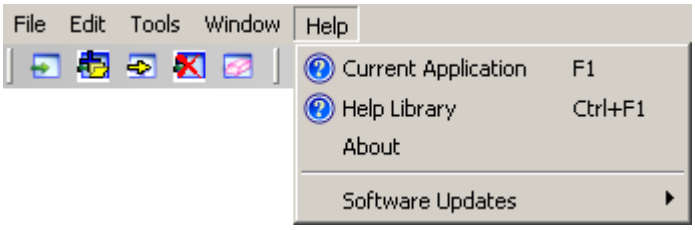

Figure 10: CMI RII application menu 'Help'

The menu item *Help* points to the standard application Help provided by the Teamcenter Rich Client Application.

The menu item *Help→About* opens the About dialog.

The About dialog informs about the actual CMI RII application (see Figure 11).

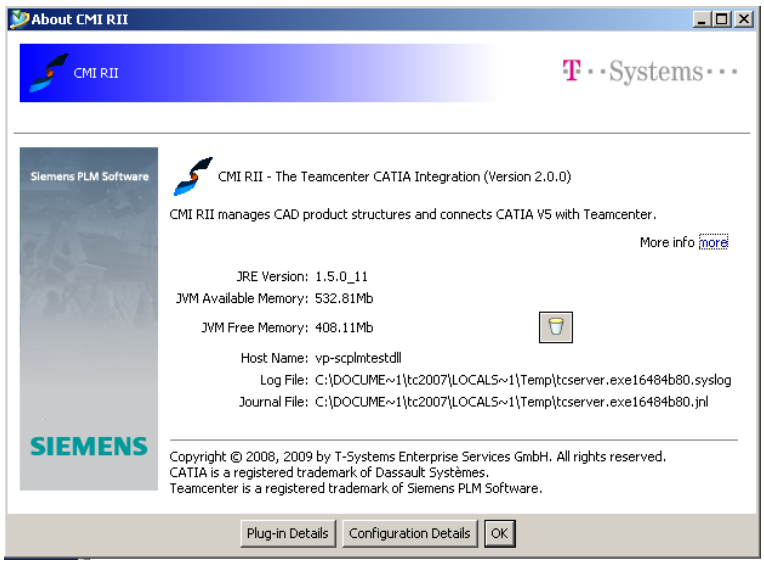

Figure 11: CMI RII application About dialog.

Pop-Up menus of CMI RII Items

If you select an Item and *drop it into*, or *Send it to* the CMI RII application, a BOMLine will be created dynamically. Items which will be handled as geometry items are represented by , structure elements are represented by . If you select a Drawing dataset and *drop it into*, or *Send it to* the CMI RII application a dataset is shown in the CMI RII application. The Drawing dataset is represented by .

Items in the CMI RII application

On the Item objects displayed in the CMI RII application window there are the following functions available:

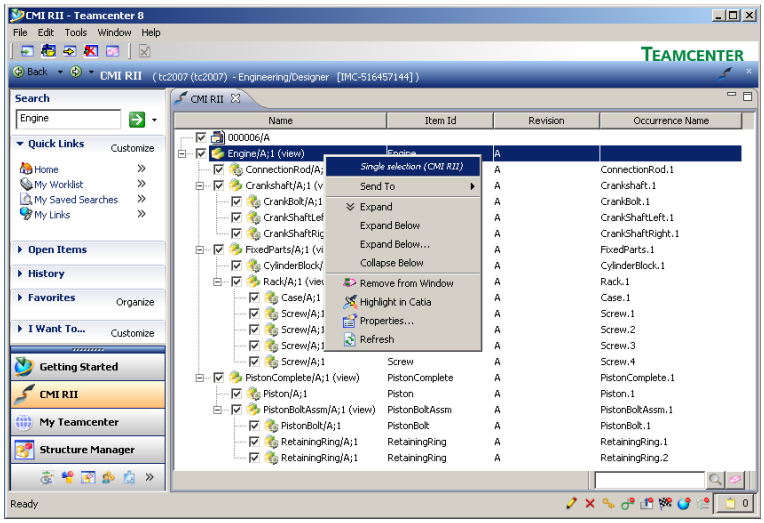

Figure 12: Context menu on a BOMLine in the CMI RII application

The *Remove from Window* command is only available on top level elements.

Single selection (CMI RII)	Description
Send To	Sends the selected object to another Teamcenter application.
Expand	Expands the object one level.
Expand Below	Expands the object multiple levels.
Expand Below…	Expands the object multiple levels, a dialog will be displayed where the level number can be defined.
Collapse Below	Collapses all children.
Remove from Window	Removes the selected object from the CMI RII application window.
Highlight in Catia	Highlights the selected objects in CATIA.
Properties…	Displays the Teamcenter properties dialog.
Refresh	Refreshes the selected object from the database.

Models in the CMI RII application

On the Dataset (CATDrawing) objects displayed in the CMI RII application window there are the following functions available:

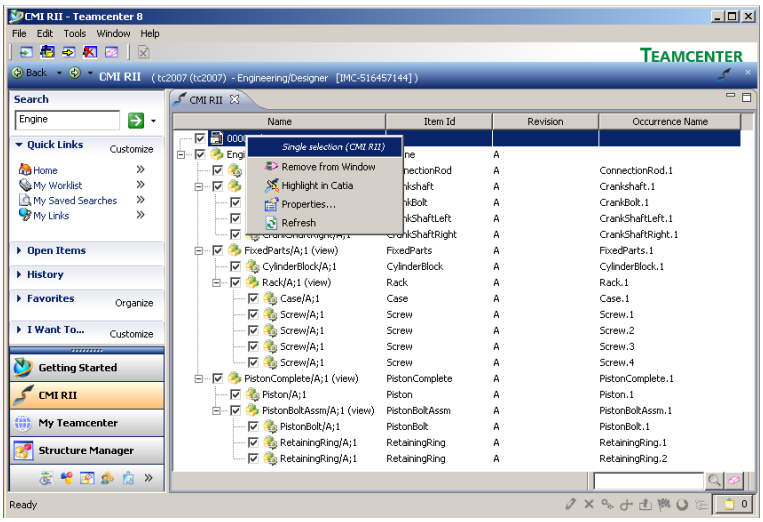

Figure 13: Context menu on a Dataset in the CMI RII application

Single selection (CMI RII)	Description
Remove from Window	Removes the selected object from the CMI RII application window.
Highlight in Catia	Highlights the selected objects in CATIA.
Properties…	Displays the Teamcenter properties dialog.
Refresh	Refreshes the selected object from the database.

Enhancements in the PSE Application

This section introduces the enhancements in the PSE Application by listing its functions.

CMI RII Menu bar in the PSE Application

The PSE application menu is enhanced by a new CMI – RII Menu Item (Figure 14).

Figure 14: PSE application with CMI RII menu

Menu item CMI – RII

The Menu item CMI – RII Menu (Figure 15) contains two commands to send data to CATIA.

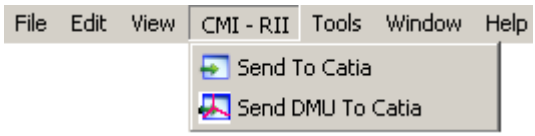

Figure 15: PSE menu 'CMI - RII'

CMI – RII → Send To Catia

The menu item *CMI – RII → Send To Catia* will send the expanded PSE content to CATIA.

The *Send To Catia* command will send the content of the current active PSE application window to CATIA. Only expanded structures will be sent to CATIA. Unexpanded structures will not be sent to CATIA.

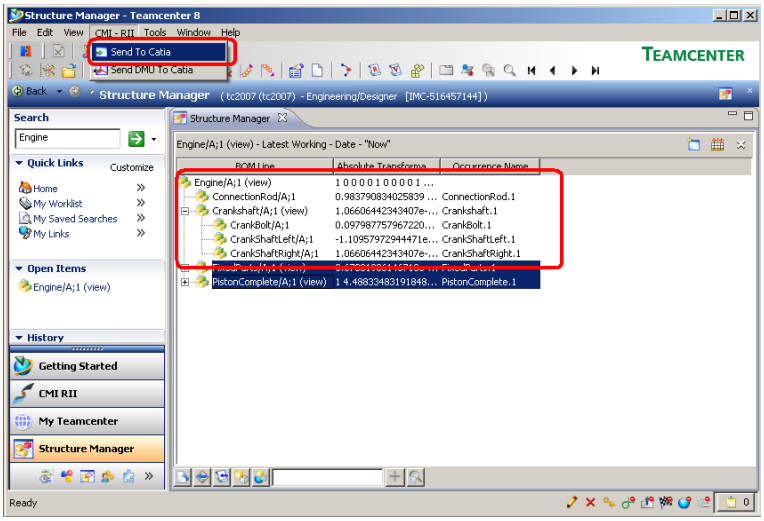

Figure 16: PSE Application with Send To Catia command

Only the expanded structures from the example structure in Figure 16 will be sent to CATIA. The selected items *FixedParts* and *PistonComplete* are not expanded and the substructures of these items are not sent to CATIA.

CMI – RII→Send DMU To Catia

The menu item *CMI – RII→Send DMU To Catia* will send the DMU 'marked' PSE content to CATIA.

The *Send DMU To Catia* command will send the checked content (checkboxes are presented by the viewer functionality) of the current active PSE application window to CATIA. Only checked structures will be sent to CATIA. Unchecked structures will not be sent to CATIA.

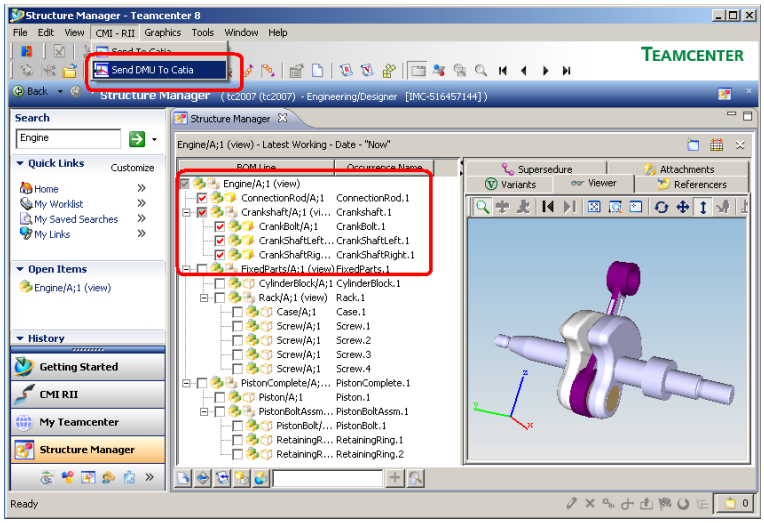

Figure 17: PSE Application with Send DMU To Catia command

Only the checked items from the example structure in Figure 17 will be sent to CATIA. The unchecked structures *FixedParts* and *PistonComplete* will not be sent to CATIA.

Enhancements in the My Teamcenter Application

This section introduces the enhancements in the My Teamcenter Application by listing its functions.

CMI RII Menu bar in the My Teamcenter Application

The My Teamcenter application menu is enhanced by a new CMI – RII Menu Item (Figure 18).

Figure 18: My Teamcenter application with CMI RII menu

Menu item CMI – RII

The Menu item CMI – RII Menu (Figure 19) contains a command to send data to CATIA.

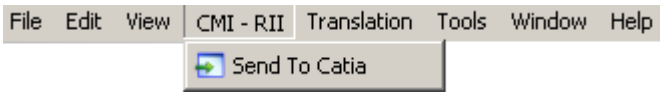

Figure 19: My Teamcenter menu 'CMI - RII'.

CMI – RII→Send To Catia

The menu item *CMI – RII→Send To Catia* will send the selected My Teamcenter objects to CATIA.

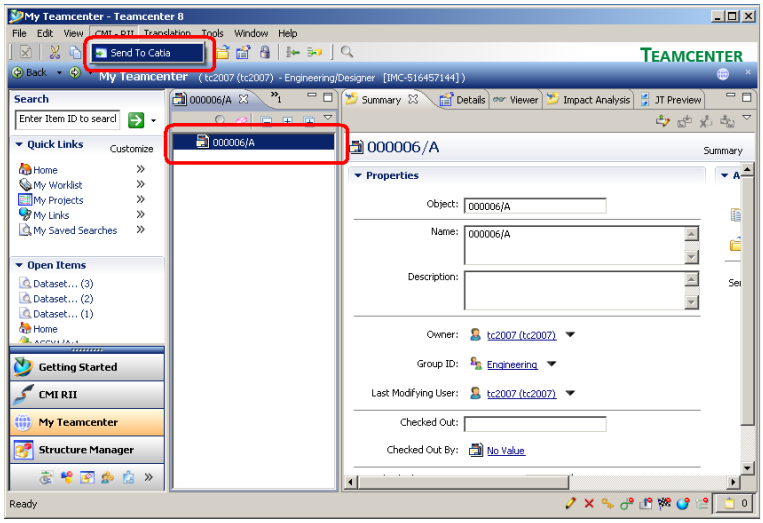

Figure 20: My Teamcenter application

The *Send To Catia* command will send the selected objects (see Figure 20) of the current active My Teamcenter application window to CATIA. Only CATDrawings are supported now.

CHAPTER 4

CATIA relevant Objects in Teamcenter

This chapter describes how to handle CATIA relevant data in Teamcenter.

Predefined Object Structure

The *CATIA Teamcenter Interface (CMI RII)* uses the following predefined object structure for CATProducts.

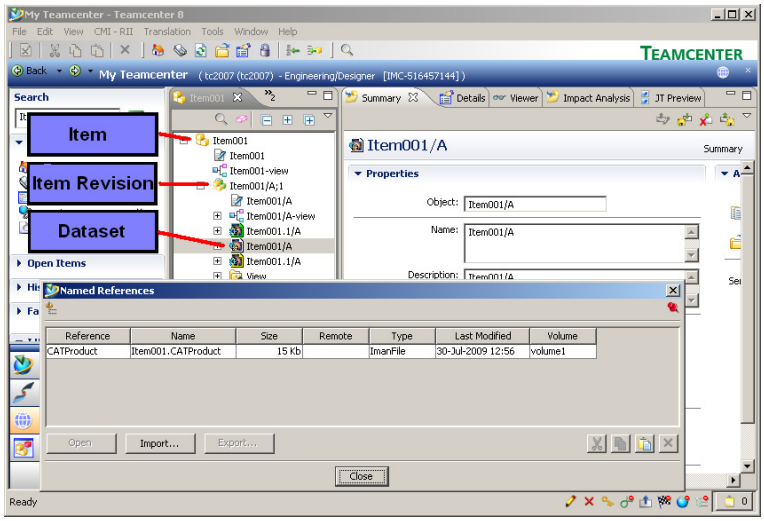

Figure 21: Predefined CATProduct object structure

An Item Revision has a Dataset attached of the type CMIStructure. The Dataset has exactly one Named Reference of the type CATProduct. This will be loaded to CATIA as a CATProduct.

If no Dataset is attached to the Item Revision, a new Product is created in CATIA with the Read operation.

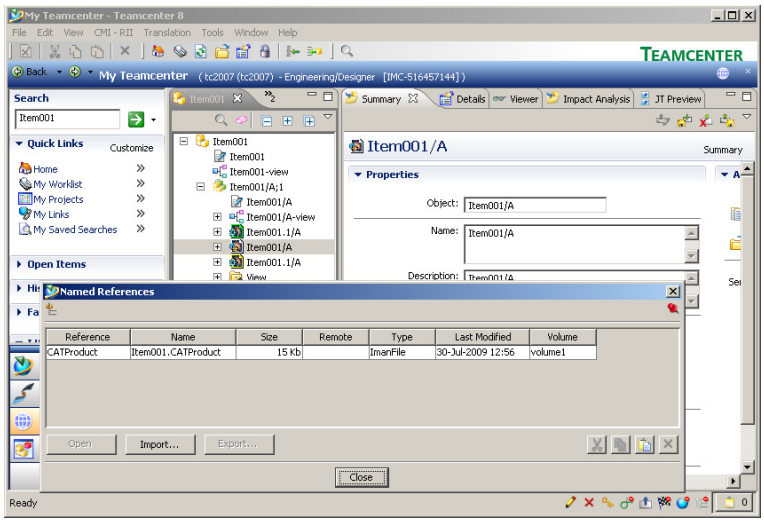

Figure 22: Predefined CATPart object structure

An Item Revision has a Dataset attached of the type CMI3DGeo. The Dataset has exactly one Named Reference of the type CATPart. This will be loaded to CATIA as a CATPart.

If an empty Dataset of the type CMI3DGeo is attached to the Item Revision, a new CATPart is created in CATIA with the Read operation.

Named References of the type CMI3DGeo can handle references of the type *model* and *cgr* which will be loaded in CATIA with the Read operation.

'model' is used for V4 CATIA model files.

'cgr' is used for light geometry representation files.

Working with Structures

A structure is made of one or more items.

An item can contain other items.

Creating an Item

In a Teamcenter Rich Client (e.g. *My Teamcenter* or *PSE* application) select: **File→New→Item...**

The Item creation wizard is started and guides you through the Item Creation Process.

Searching for Items

In a Teamcenter Rich Client (e.g. *My Teamcenter* application) press the Search button

In the Search window use the Predefined Search for Items. The result is displayed in the My Teamcenter application.

Creating a Structure

To create or modify a structure in Teamcenter use the PSE application.

The figure below shows an example assembly with expanded items in the PSE application browser.

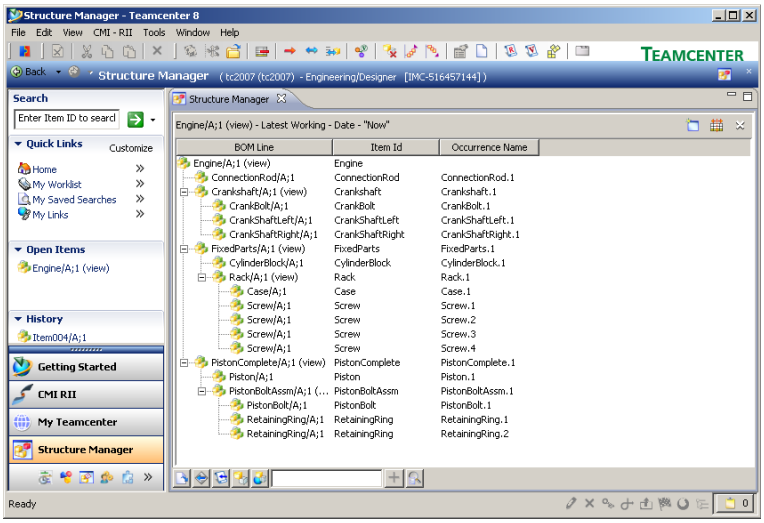

Figure 23: Example of an expanded assembly in Teamcenter PSE application

Example Structure for CMI

The following instructions describe step by step how to create an example assembly structure which can then be loaded into the CATIA session:

Start the PSE application in Teamcenter Rich Client (Figure 24).

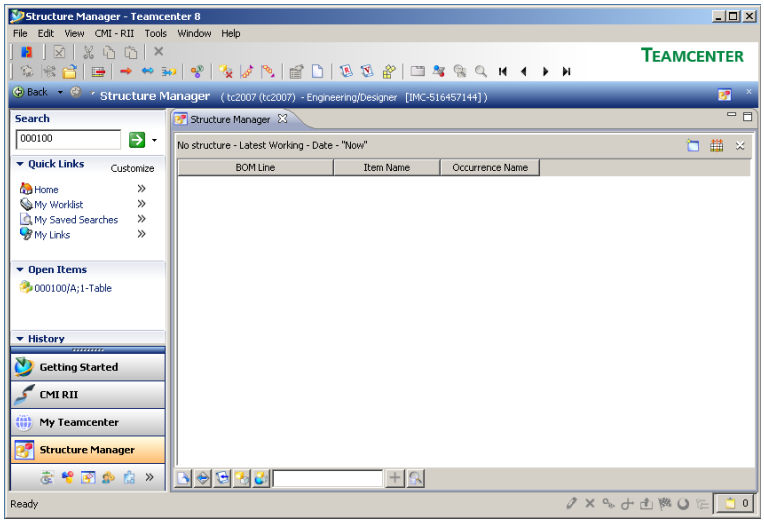

Figure 24: Empty Teamcenter PSE application

Create a new item named “Table” with **File→New→Item…**

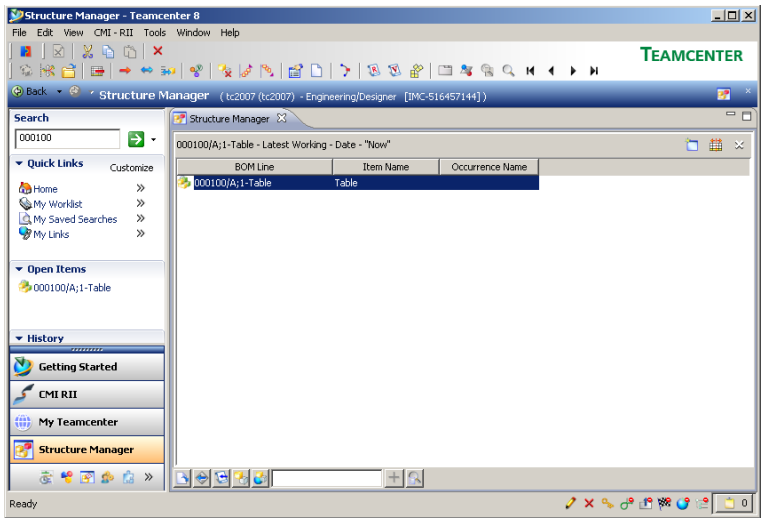

Figure 25: PSE application with "Table"

Select the "Table" and create a new item named "Plate" with **File→New→Item…**

Select the "Table" and create a new item named "Leg" with **File→New→Item…**

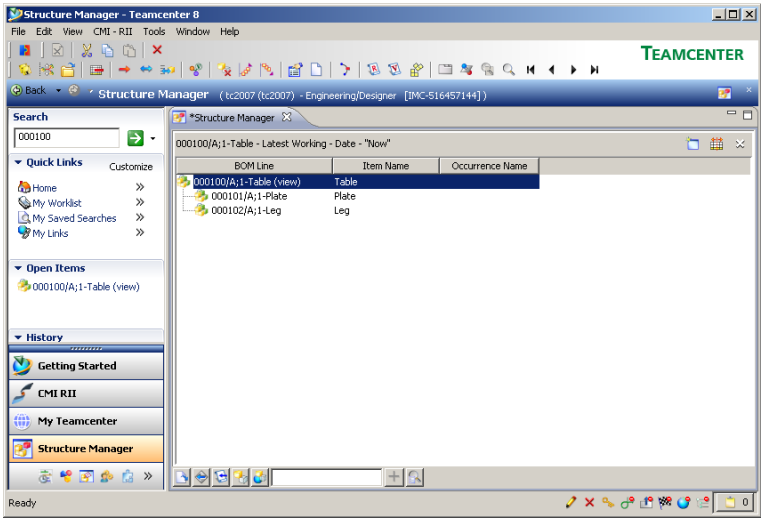

Figure 26: PSE application with "Table", "Plate", and "Leg"

Select the "Plate", right Click→**Send To→My Teamcenter**.

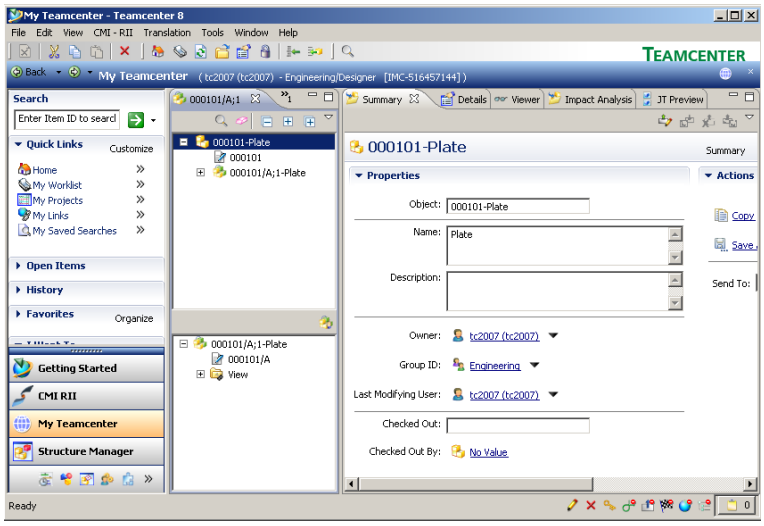

Figure 27: My Teamcenter application with "Plate"

Select the Item Revision "xxxxx/A;1 – Plate" and use the command **File→New→Dataset** to create a new dataset of type CMI3DGeo to define a geometry.
Use the More… button on the left side to select the type CMI3DGeo.
The CMICatiaV5 Tool is set automatically.

Press the OK button to continue.

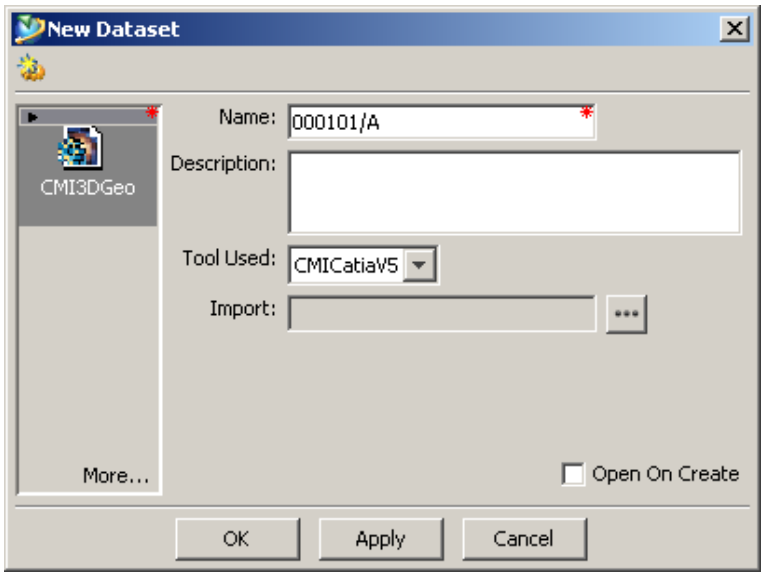

Figure 28: New Dataset dialog

Repeat the same procedure for the "Leg" item and switch to the PSE application.

Select the "Leg" of the "Table" in the PSE application and copy the item with **Edit→Copy**.

Select the "Table" item and paste the "Leg" item three times with **Edit→Paste**.

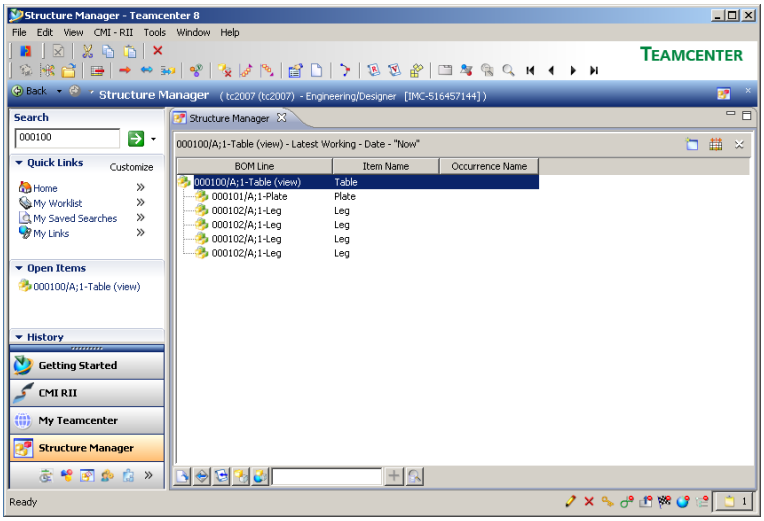

Figure 29: My Teamcenter application with complete "Table"

Check that CATIA is started with installed CMI RII.

Select the "Table" in the PSE application and right Click→**Send To→CMI RII**.

Select the "Table" in the CMI RII application and right Click→**Expand Below**.

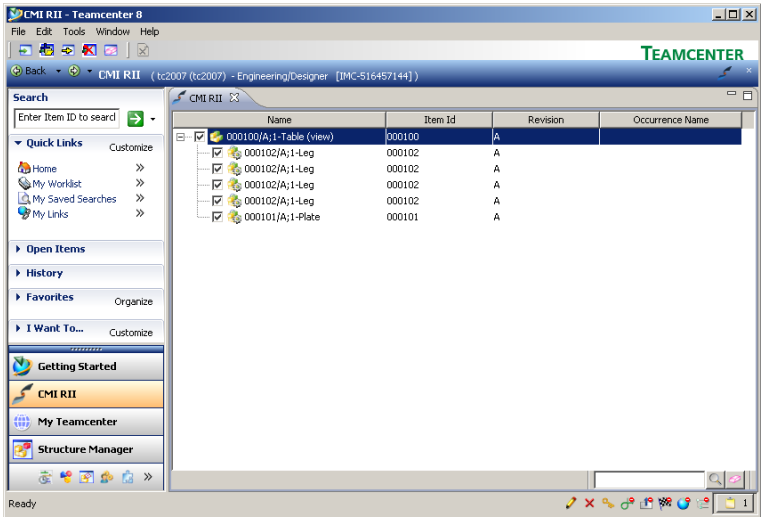

Figure 30: CMI RII application with complete "Table"

Verify that all items are checked. Only checked and expanded items will be sent to CATIA.

Use the **Tools→Send To Catia** command to send the content to CATIA.

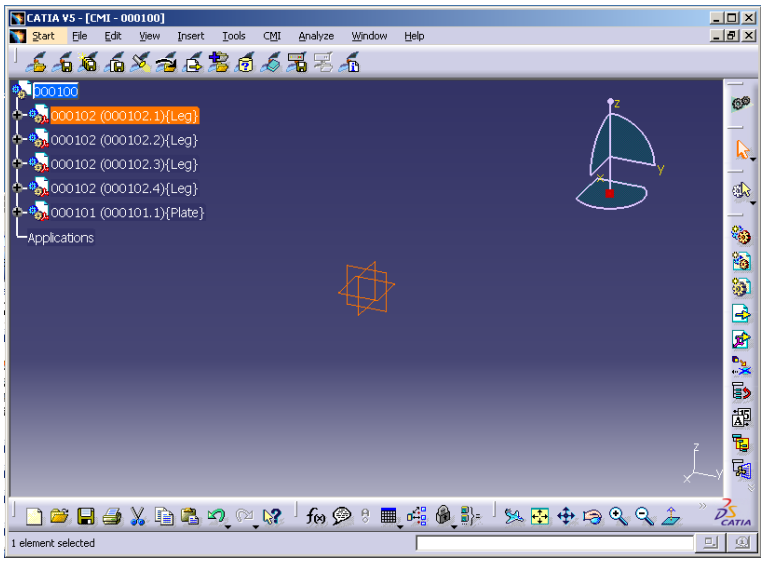

Figure 31: CATIA V5 with loaded "Table"

Updating a Structure

Complete construction in CATIA and press the Update button in CATIA V5.

The files in CATIA and the metadata are updated to Teamcenter.

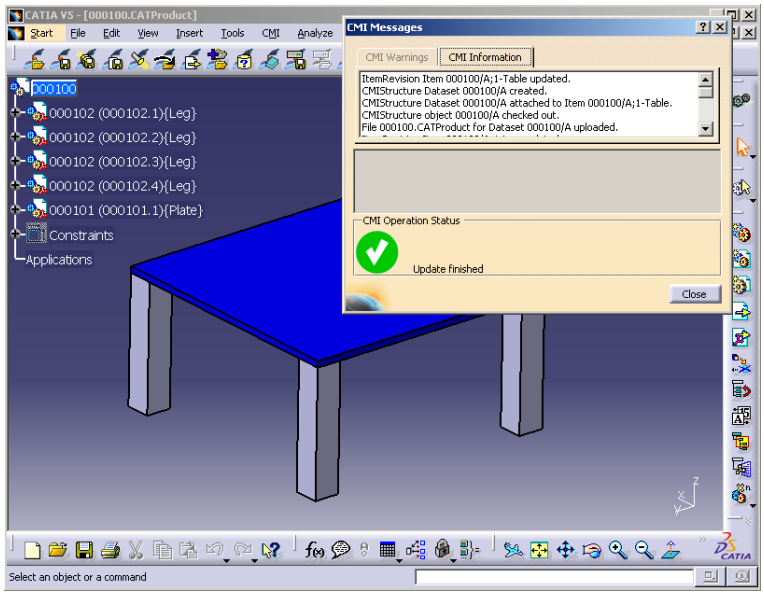

Figure 32: CATIA V5 with updated "Table"

Change the visible attributes in the CMI RII application and check the values for Occurrence name and Matrices. Both attributes are filled with the current CATIA data.

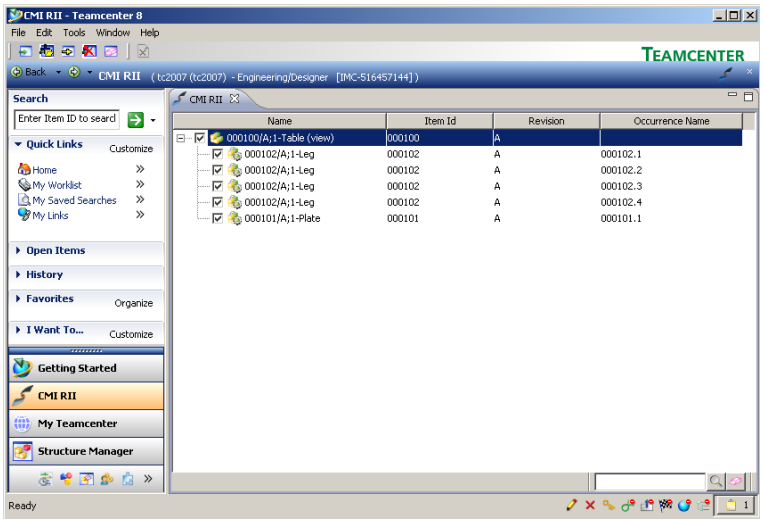

Figure 33: CMI RII application with complete "Table"

Select the "Plate" and right Click→**Send To→My Teamcenter**
Verify that the dataset and a named reference are added.

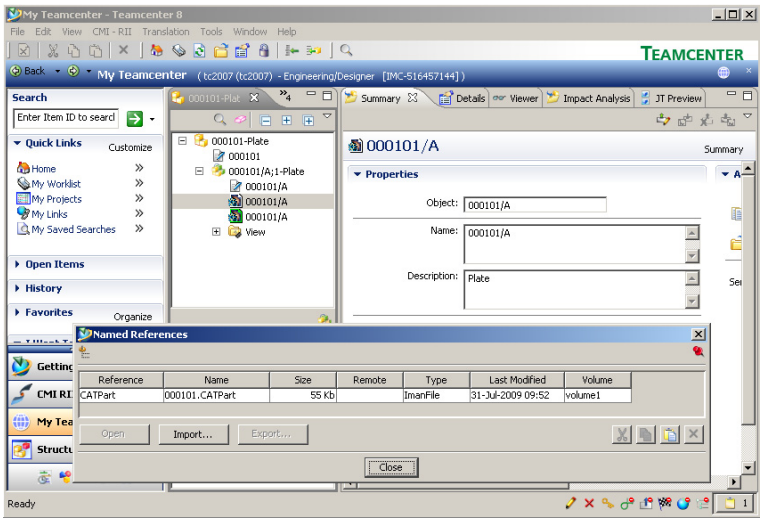

Figure 34: "Table" item in My Teamcenter

Highlighting an Item in CATIA

In cases where you have loaded an assembly structure to CATIA as described in the steps above, it may be useful to find the corresponding model in the CATIA session of an item displayed within the CMI RII application.

For this purpose, it is possible to highlight CATIA models in CATIA using the currently selected item in the CMI RII application.

In the CMI RII application, select the item you want to be highlighted in the CATIA session and choose the *Highlight in Catia* option in the *Pop-Up menu*.

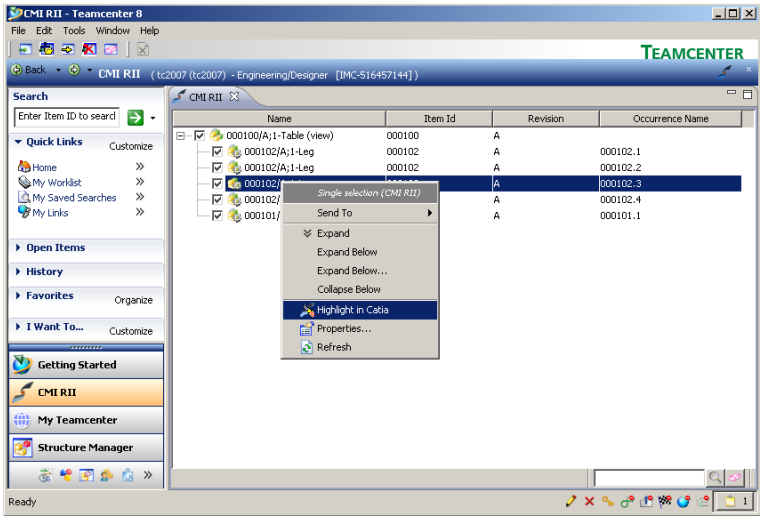

Figure 35: Highlight Pop-Up menu in CMI RII application

CATIA comes to the foreground with the selected CATIA model highlighted.

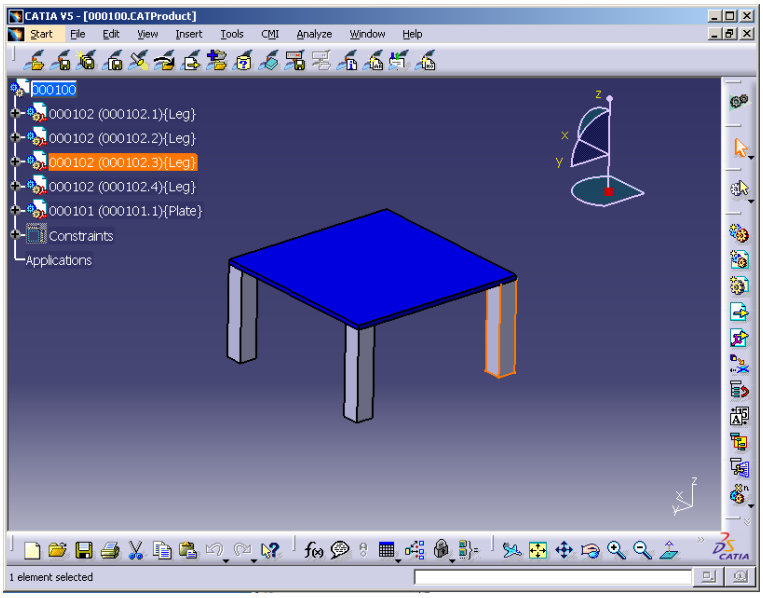

Figure 36: Highlighted item in CATIA V5

CHAPTER 5

Working with CATIA V5

Introduction

This chapter describes the CMI interface between CATIA V5 and Teamcenter. Typical tasks discussed are loading items/structures/drawings stored in Teamcenter into *CATIA V5* and creating/updating parts and drawings in Teamcenter. One of the main subsections deals with creating and saving CATIA V4 models.

Product Structure CATIA V5 vs. Teamcenter

Product structures in Teamcenter correspond well to product structures found in CATIA V5. The following graphic illustrates the product structure in Teamcenter and CATIA V5.

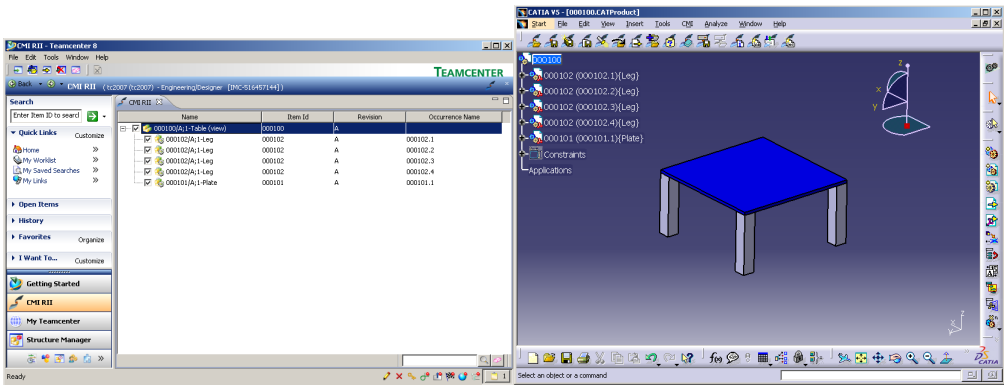

Figure 37: Product Structure in Teamcenter and CATIA V5

CMI Toolbar

The following figure shows the toolbars of the CATIA V5 CMI module. Descriptions of the toolbar and the command icons follow below.

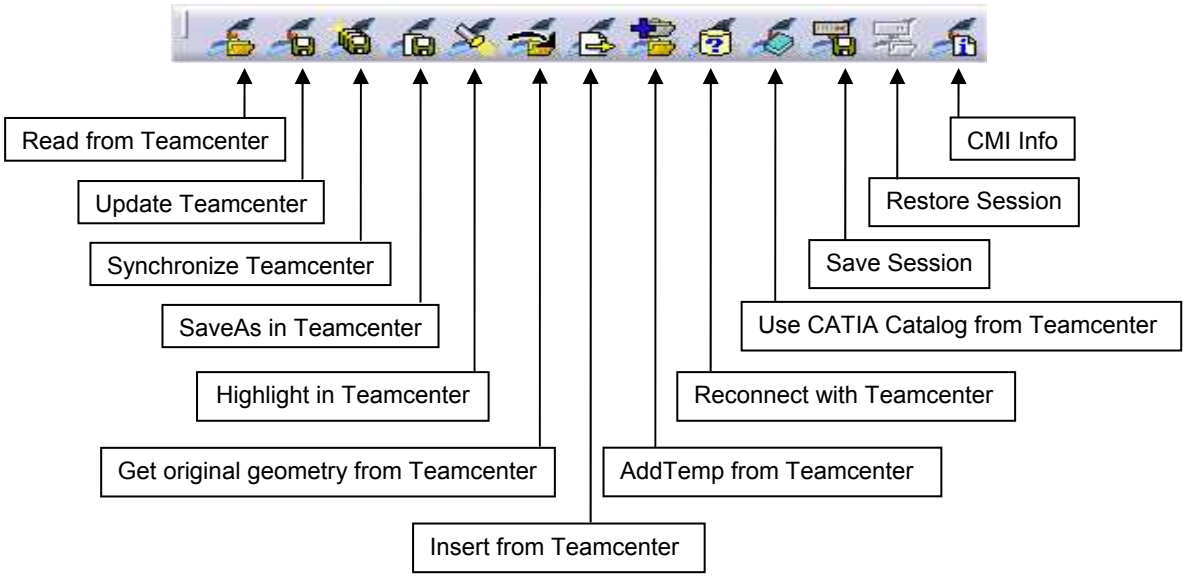

Figure 38: CMI Toolbar in CATIA V5

With the *CMI toolbar* it is possible to read files from the *CMI RII application* in Teamcenter. You can then modify the geometry in your CATIA V5 session. The modifications can be updated in Teamcenter via the 'Update Teamcenter' or the 'Synchronize Teamcenter' button in CATIA V5.

The icon 'Highlight in Teamcenter' is only active if you are working with a product structure.

You can get the Teamcenter information of the CATParts, CATProducts, CATDrawings, and CATIA V4 Models with the CMI Info button if you loaded the product structure with CMI to CATIA V5.

The other commands will be described in more detail later in this chapter.

CATDrawings in CMI RII application in Teamcenter

CATIA V5 will read all expanded assemblies, single CATDrawings from the Teamcenter CMI RII application.

The following figure shows a CATDrawing in the CMI RII application:

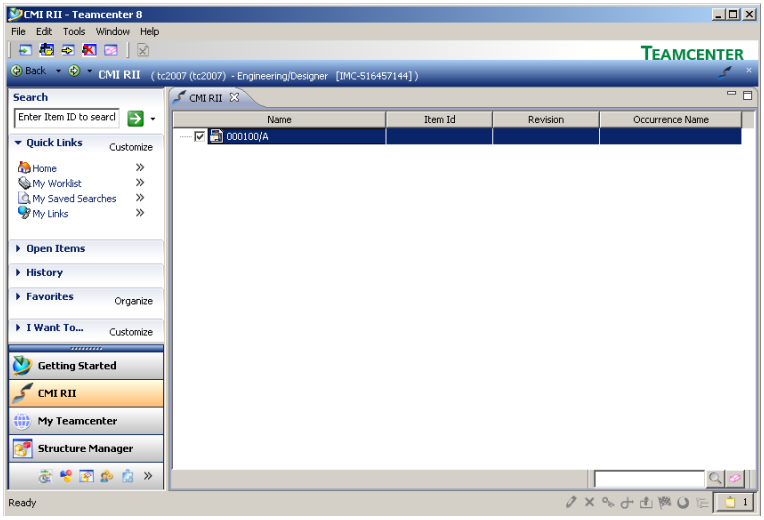

Figure 39: CMI RII application with a CATDrawing dataset

When you have transferred a CATDrawing from Teamcenter to CATIA V5 via the CMI RII application, the drawing will be displayed in CATIA V5 in a new window.

CMI for CATIA V5 also supports CATIA V4 models and CATIA V5 Drawing files (CATDrawings). CATDrawings may be included in assembly structures in Teamcenter, however in CATIA V5 they will be displayed in separate windows as shown in the figure below.

All normal CMI functions (Read, Update, Create and Save As) may be used with CATDrawings and CATParts within CATIA V5.

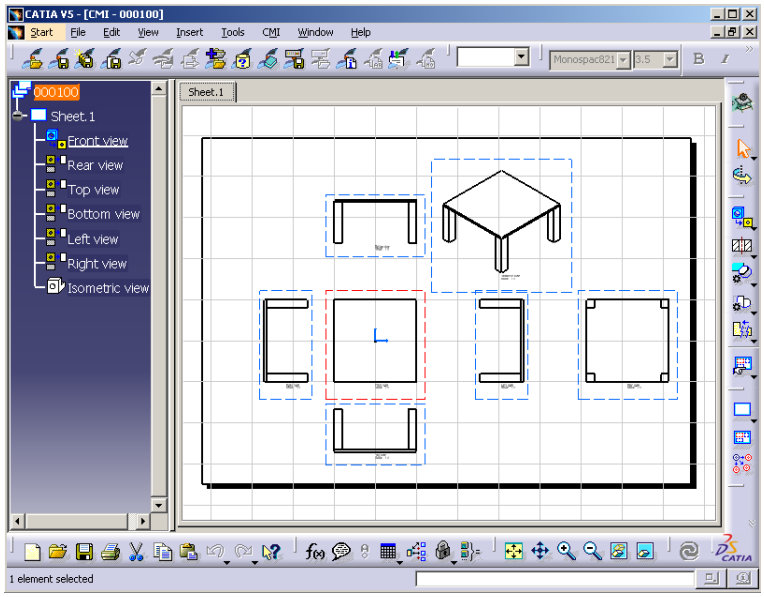

Figure 40: Drawing Workbench for Read, Update, Create or Save As CATDrawing

CMI toolbar: Read From Workbench

With the ‘Read from Workbench’ button in the CMI Toolbar the content of the CMI RII application in Teamcenter may be read into CATIA V5. The CMI RII application may contain CATDrawings, and assembly structures.

If you want to send one or more assemblies, CATParts or CATDrawings to CATIA follow these instructions:

Drag and drop, Send To drawings (CMI2DGeo datasets) or assemblies (items) into CMI RII.

Expand or Deexpand the assemblies in the CMI RII application. Only those items that are shown in the CMI RII application will be opened in CATIA.

Read Teamcenter CATIA items with *Read From Workbench* icon  .

Every top-level item in the CMI RII application will be opened in a separate browser in CATIA V5.

If the PSE application is the current active application in the Teamcenter Rich Client the PSE content will be read to CATIA instead of the content of the CMI RII application.

CMI toolbar: Update Teamcenter

This command updates geometry and position information in Teamcenter.

Modify the content in CATIA.

Write the modifications back with *Update Teamcenter* icon  .

All files in the assembly and the position information (transformation matrix) which have been modified in CATIA will be updated in Teamcenter.

The success (green icon) and more detailed information (Figure 41) are then displayed to the user.

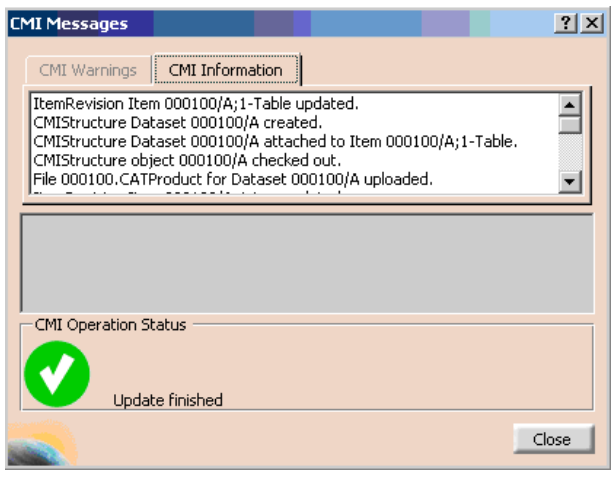

Figure 41: Dialog Window for Update Teamcenter

CMI toolbar: Synchronization in Teamcenter

This command updates Teamcenter with any changes made to your CATIA V5 product structure. The changes that will be reflected in Teamcenter include:

- any geometry and position information changes,
- the creation of any new CATParts/cgr/models and CATProducts that have been added to your product structure,
- the creation of any new CATPart/cgr/model and CATProduct instances,
- the removal of CATProducts and CATPart/cgr/model instances from the product structure.

The changes made to the product structure since the last Read/Update/Synchronize are presented in the form of a list of operations to be performed in Teamcenter.

Modify your product structure in CATIA by adding a new CATPart/CGR, adding a new CATProduct, removing a product or by changing the geometry of a CATPart.

Write the modifications back to Teamcenter with the *Synchronization in Teamcenter* icon  .

The following dialog will appear, showing which changes need to be written to Teamcenter:

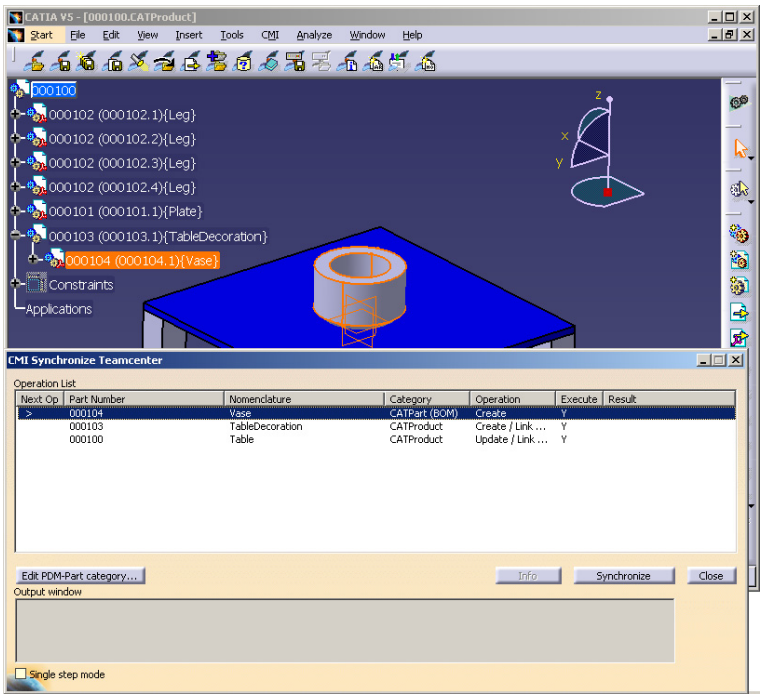

Figure 42: The Synchronize Teamcenter dialog

The dialog's *Operation* column shows which procedures need to be performed in Teamcenter for each object and may be a combination of the following:

Update	The file and positional changes of any children will be updated in

	Teamcenter.
Create	The object will be created in Teamcenter. The object is completely new and not already in Teamcenter.
Link Child	At least one new parent-child relation will be created in Teamcenter. This reflects the addition of a child product in CATIA.
Drop Child	At least one parent-child relation will be deleted in Teamcenter. This reflects the deletion of a child product of the object in CATIA.

The column *Execute* shows whether the operation will be executed or not. If there is any reason why the operation cannot be executed, then the reasons will be shown in the *Result* column.

If new geometry files are to be created in Teamcenter the default is BOM Type. This will become an item and a dataset in Teamcenter.

With the *Edit PDM-Part category* button the category can be changed to *No BOM Type* in the Edit new PDM-Part category dialog (see Figure 43). This file will become a dataset in Teamcenter and will be attached to the parent item.

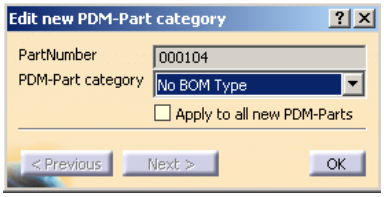

Figure 43: Edit PDM-Part category dialog

If the *Synchronize* button is pressed, the list of executable operations is worked through in the order shown on the screen.

The results from Teamcenter are shown in the Output Window below the list.

The *Update* operation:

CATIA File objects will be updated in Teamcenter. No user interaction is necessary.

The *Create* operation:

CATIA File objects will be created in Teamcenter and the following steps will be completed:

- a Teamcenter item will be created.
  The created item is also linked to the users Newstuff Folder.
- a dataset will be created under the item
- the file will be imported in the dataset as Named Reference with the correct type (CMI3DGeo or CMIStructure).

The *Link Child* operation:

- This may only appear as an operation for *CATProduct* objects, as CATParts cannot contain sub-products. In Teamcenter any new children will be added to the Teamcenter item of the parent object by creating a new BOM Line.

The *Drop Child* operation:

- This may only appear as an operation for *CATProduct* objects as CATParts cannot contain sub-products. In Teamcenter any child CATProducts deleted from the CATIA product structure will be detached from the Teamcenter item of the parent object by removing the BOM Line.

After every operation the success or failure will be registered in the *result* column of the dialog. If an operation fails the output window will provide you with the cause of

the failure.

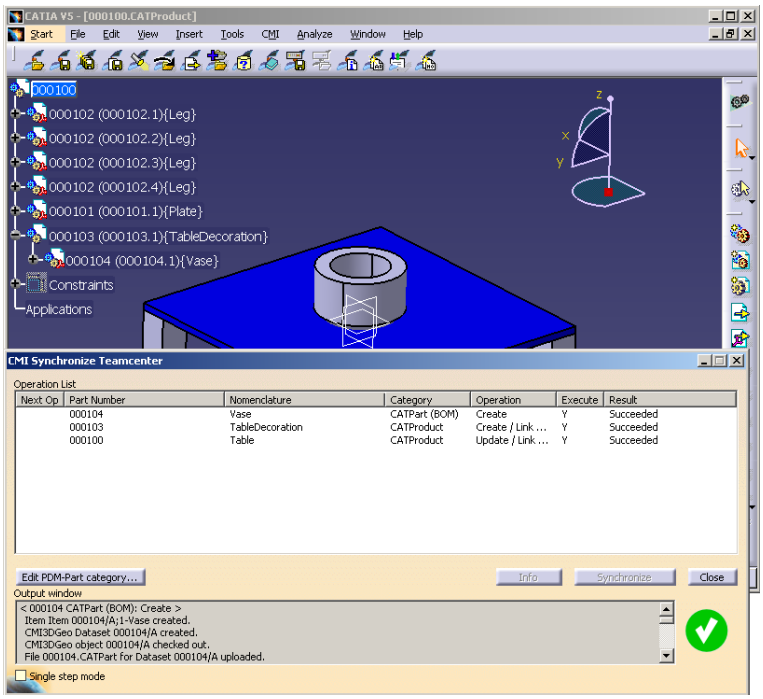

Figure 44: The Synchronization in Teamcenter dialog with success

CMI toolbar: SaveAs in Teamcenter

With this command a CATPart or CATDrawing currently set active in CATIA V5 can replace another file in Teamcenter. A CATPart or CATDrawing selected in Teamcenter and then dropped into CMI RII will be overwritten by the contents of the CATPart or CATDrawing in CATIA memory.

Design or load your CATPart or CATDrawing in a separate window in CATIA.

In CATIA V5 select the *Save As in Teamcenter* icon  .

Drag and drop or Send an item into the CMI RII application in Teamcenter.

In order to ‘Save As in Teamcenter’ with a CATDrawing, drag the ‘new’ CATDrawing dataset to be overwritten into the CMI RII application in Teamcenter. The CATDrawing active in CATIA V5 will then overwrite its content.

In Order to ‘Save As in Teamcenter’ with a CATPart, drag the ‘new’ item to be overwritten into the CMIRII application in Teamcenter. The CATPart active in CATIA V5 will then overwrite its content.

Please take care when you ‘Save As’ a CATPart which is used in a CATProduct - a reference to the CATPart is stored within the parent CATProduct. Make sure that the new CATPart is compatible.

CMI toolbar Highlight in Workbench

If you want to highlight CATParts, CATProducts, and CATIA V4 models in the CMI RII application triggered by CATIA V5 perform the following steps.

In *CATIA V5* select the CATIA objects you want to highlight in the CMI RII application and choose the *Highlight in Workbench* icon  from the CATIA CMI toolbar.

Now the items are selected in the CMI RII application.

CMI toolbar: Get original geometry from Teamcenter

The 'Get original geometry from Teamcenter' command retrieves geometry files (CATPart or CATIA V4 model) from Teamcenter if the geometry files are not yet loaded in CATIA V5.

A CATProduct, CATPart or CATIA V4 Model has to be selected in the product structure.

The function is used in context of huge product structures in CATIA to get the original geometry file from Teamcenter if the *Do not activate default shapes on open* option is set in the CATIA Product visualization settings (see Figure 45).

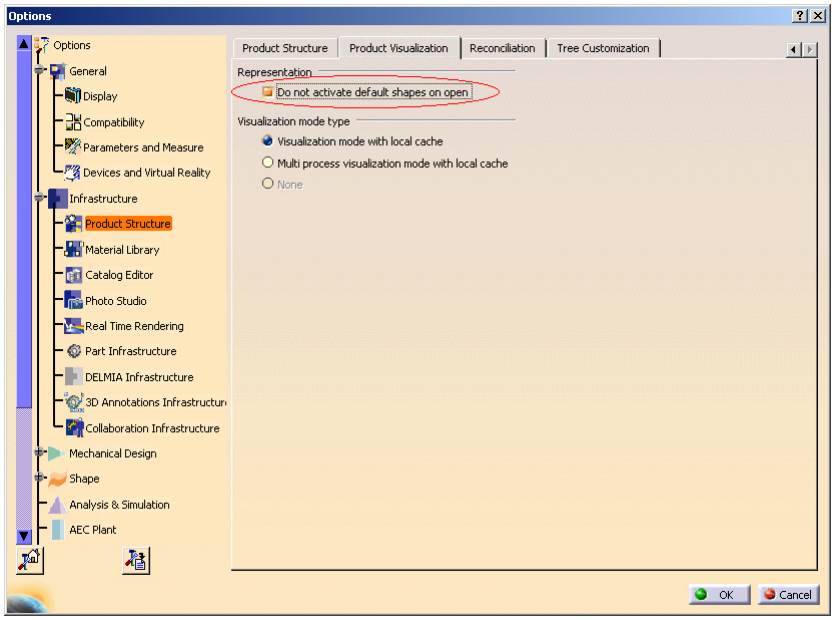

Figure 45: *Do not activate default shapes on open* option in CATIA V5

CMI toolbar: Add Temp

The 'AddTemp' command allows the user to visualize a temporary structure together with the working one.

For example, if the user is making a design of a car and wants to see if the luggage boot is big enough, he can load a suitcase as a temporary structure by doing the following (supposing that the car structure is already loaded):

In CATIA V5, go to **Tools→Options→Compatibility** (Tab: "CMI") and make sure that the option "Use one temporary product window…" is checked.

Read an assembly (here the car assembly) into CATIA using the *CMI toolbar Read* command.

In CATIA V5 select the *AddTemp from Teamcenter* icon  .

Drop the assembly or model you want to add temporarily to CATIA (the suitcase) into the CMI RII application. Expand it as necessary. Click **Tools→Temp To Catia** in the menu or press the *Temp to Catia* icon  in the CMI RII toolbar. Changes or expands you make to other assembly structures in the CMI RII application will not be sent to CATIA.

CATIA comes to the foreground with both the original and temporarily added structures loaded in the same window – here the car and the suitcase structure.

It will not be possible to make any changes to the temporarily added (suitcase) structure.

The temporarily added part/structure is removed from the CMI RII application after you sent it to CATIA.

All Part Numbers and File Names in the temporarily added structure are prefixed with "TMP#_", where "#" is a counter in CATIA V5, beginning with 1. Every AddTemp

will increase the counter. This prefix is customizable by the customer.

Limitation:
CATProducts are not transferred to CATIA, the products will be represented by "CATIA Components". Only position information from Teamcenter is processed.

Visualize multiple revisions of an Assembly using Add Temp

Add Temp in CATIA V5 can be used to load different versions (revisions) of an assembly or model at the same time. In order to make this possible, the Part Numbers and File Names of the temporarily added assembly are prefixed with "TMP#_".

In CATIA V5, go to **Tools→Options→Compatibility** (Tab: "CMI") and make sure that the option "Use one temporary product window…" is checked.

Read revision B of your assembly as described in *CMI toolbar Read from Workbench*.

You will find that it is opened in CATIA underneath a product tagged "CMI Workbench".

In CATIA V5 select the *Add Temp from Teamcenter* icon  .

Find and expand Revision A of your assembly (drop it in the CMI RII application window and use the Expand Below command in Teamcenter). Click **Tools→Temp To Catia** in the menu or press the *Temp to Catia* icon  in the CMI RII toolbar. Changes or expands you make to other assembly structures in the Workbench will not be sent to CATIA.

Revision A and B of your assembly are opened in the same Window in CATIA V5. You can overlay and position the assemblies relative to each other for comparison.

CMI toolbar: Insert from Teamcenter

The 'Insert from Teamcenter' command allows the user to load a CMI structure under a selected product.

Read an assembly into CATIA V5 using the *CMI toolbar Read from Workbench* command.

Select a product to be the root of the inserted structure.

In CATIA V5 select the *Insert from Teamcenter* icon  .

Drop the assembly-structure you want to use in CATIA V5 in the CMI RII application window. Expand it as necessary. Click **Tools→Insert To Catia** in the menu or press the *Insert To Catia* icon  in the CMI RII toolbar. Changes or expands you make to other assembly structures in the Workbench will not be sent to CATIA.

CATIA comes to the foreground with the selected product and the used structure.

The dropped part/structure is removed from the CMI RII application window after you sent it to CATIA.

Via Synchronize you can save the new Use-Relation in Teamcenter.

CMI toolbar: Local Save and Restore

 Using *Save Local* the content of the CATIA V5 session can be stored locally.

This snapshot of the session will persist even when CATIA V5 is closed.

 When the session is restored with *Restore Local*, you can continue your work as if you had just used *Read from Teamcenter*, i.e. you can perform Updates.

The *Restore Local* command is only available when the CATIA is empty – i.e. no documents are open – to avoid conflicts.

With Save/Restore Local you can avoid reading large assemblies from Teamcenter repeatedly. However, the status of the data in Teamcenter may change if you keep your local snapshot for a long time. It may get “stale”.

The following restrictions apply:

- Only one saved session is maintained at a time.
- The session can only be saved when the data is up to date. I.e. there are no files in a modified state, and no new files.
- The save/restore session does not restore files that were not loaded from the PDM system.

CMI toolbar: Reconnect with Teamcenter

CMI recognizes files that have been loaded into CATIA V5 through the CMI RII application window. Since it knows that they are from Teamcenter, it can update them there.

Other files, that you load with e g. **File→Open**, it will not know. They will be regarded by Synchronize as new files, and it will create new Teamcenter objects for them.

If you open a product structure from disk that contains files that are already in Teamcenter, you can use *Reconnect with Teamcenter* to make CMI RII recognize these files. It will recognize products if the part number is correct, and CATPart/model files based upon their filename. If the CATPart is a BOM CATPart (Component) in Teamcenter, the part number must also match.

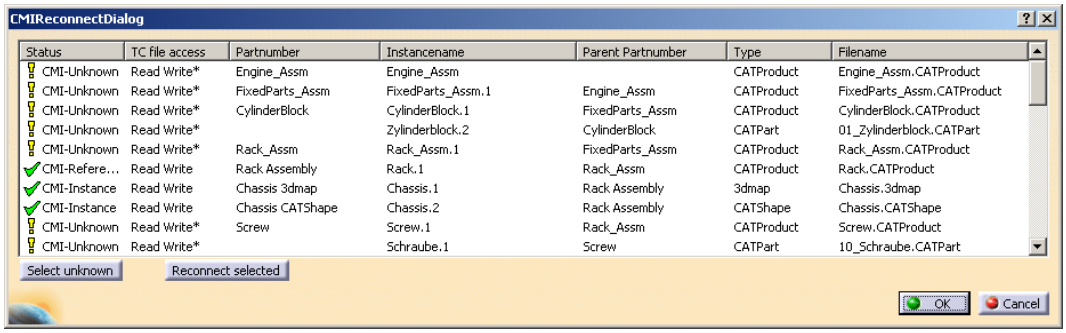

Figure 46: Reconnect dialog

There are three possible states:

`CMI-Unknown`: CMI does not know this item at all. It would be created as a new part in Teamcenter.

`CMI-Reference`: CMI knows this item is in Teamcenter. But it appears that in Teamcenter there is no instance known by the same instance name under the same parent item. Maybe it was deleted or is new. It would be created as a new part instance in Teamcenter.

`CMI-Instance`: CMI knows this instance is in Teamcenter. It can be updated in Teamcenter.

An asterisk (*) signifies that the Reconnect Dialog recognized the item, but it is not yet reconnected. Items are actually reconnected when you close the dialog with *OK*. Click *Cancel* to keep them as unknown, e.g. if you want to save them as new Teamcenter objects.

Reconnect does not create anything in Teamcenter or updates any files. Use *Synchronize* to update data.

Load a CATProduct into CATIA V5 using **File→Open**.

In CATIA V5 select the *Reconnect with Teamcenter* icon  .

In the CMI Reconnect Dialog, click *Select unknown*. In this case, all lines will be selected. Click *Reconnect selected*. If any items are recognized in Teamcenter, they will change their status. To reconnect these items, close the dialog with *OK*. Click *Cancel* to leave the dialog without reconnecting the items.

Reconnected items that are writeable in Teamcenter will be in the “modified” state, to be able to update them in Teamcenter.

Use Synchronize to create the unknown/new data and to update the reconnected files.

If you want to keep individual items as new, unselect them before you click *Reconnect selected*.

CMI toolbar: CMI Manage Catalogs

This command is described in section *Manage Catalogs* on page 40.

CMI toolbar: CMI Info

This command displays the information from Teamcenter for CATParts, CGRs, CATProducts, CATDrawings and CATIA V4 Models.

The *More …* button displays attributes of the correspondent Teamcenter-objects. The range of attributes can be customized.

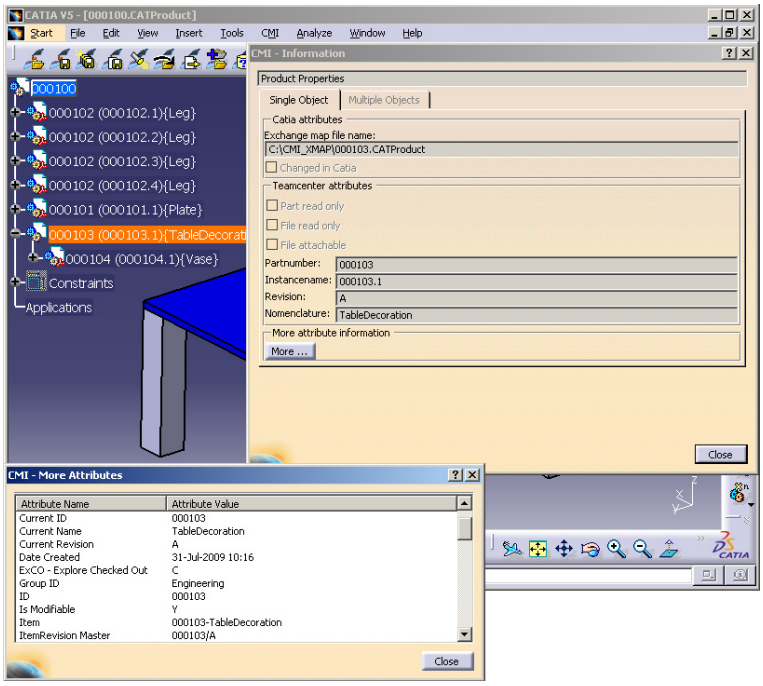

Figure 47: Information dialog window for CATProduct with “More Attributes”

Select the CATPart, CATProduct, CATDrawing or CATIA V4 Model.

In CATIA V5 select the *CMI Info* icon  . The appropriate dialog info window appears.

You can also select multiple CATIA items. Information will be displayed in a table view.

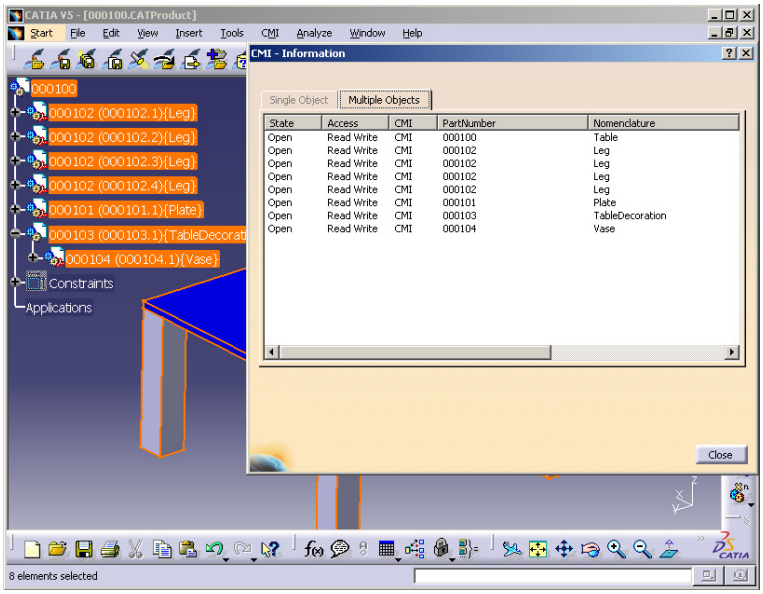

Figure 48: CMI Information for multiple items

Handling of CATDrawings

CATDrawings can be included in your assembly structures in Teamcenter. All the usual functions (Read, Update, Create and Save As) are supported for CATDrawings in CATIA V5 via the CMI Toolbar.

CATDrawings may be created for single models or for assemblies. There are however several caveats when working with CATDrawings and CMI, depending on the context of the CATDrawing.

Creating a CATDrawing From a Single Model

When creating a CATDrawing for a CATPart, send the CATPart item from Teamcenter to CATIA V5, and then create the CATDrawing. This method will preserve the context of the CATDrawing - irrespective of whether the CATPart is later loaded into CATIA singly or as part of an assembly. If you later make changes to the CATPart, and you wish the CATDrawing to update and reflect the changes, you must load both the CATPart and the CATDrawing in CATIA V5 using CMI before you update the CATDrawing.

Drag and drop the CATPart item on the CMI RII application or use **SendTo→CMI RII** with the CATPart Item.

Select **Tools→Send To Catia** or use the *Send To Catia* icon  in the CMI RII toolbar.

Create the CATDrawing with the usual CATIA V5 functions.

Select the *Synchronize in Teamcenter* icon  .

Creating a CATDrawing From a Product Structure

Drag and drop the assembly into the CMI RII application.

Select **Tools→Send To Catia** or use the *Send To Catia* icon in the CMI RII toolbar.

Create the CATDrawing with the usual CATIA V5 functions.

Select the *Synchronize in Teamcenter* icon .

Handling of cgr and model files

CATIA V5 cgr files and CATIA V4 Model files are managed as representation of an embedded component in the product structure. It is not possible to modify such a representation directly. It is only possible to use the Save As function of CATIA V5 to create a new cgr or model file of a loaded CATPart or CATProduct. To update the representation you have to replace the original representation by the new representation.

Load an assembly which uses a cgr or model in the product structure. And use the right Click→**Representations→Manage Representations …** command (Figure 49).

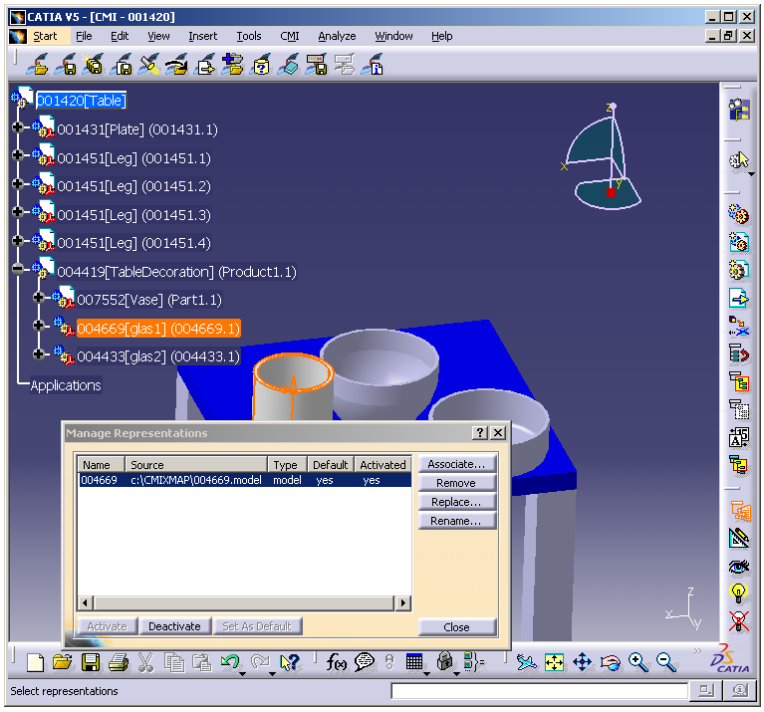

Figure 49: Manage Representation dialog of CATIA V5

Use the “Replace” function to use the cgr or model file with a new representation.

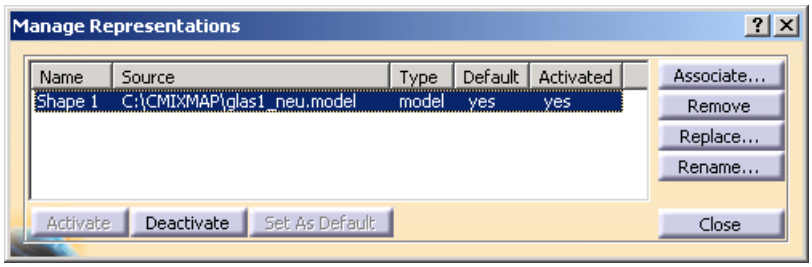

Figure 50: Manage Representation dialog of CATIA V5 with new model file

Use the CMI Update or Synchronize functionality to update the new cgr or model file in Teamcenter.

During the CMI Update/Synchronize process CMI replaces the original representation file in the exchange directory by the new representation file and changes the “Source” path of the representation back to its original value. Then the new representation file (which has the name of the old file) is updated in Teamcenter.

To use this function you need write access to the cgr or model data item in Teamcenter. Although the cgr or model is represented in an embedded component of a higher CATProduct you need no write access to the CATProduct.

Restrictions:
- There must be only one representation in one component.
- The “Name” of the representation must not change.

Catalog Management

CMI can also manage catalog files in Teamcenter. Teamcenter compatible catalogs reference files and items that are actually stored in Teamcenter. If the user opens or inserts a file from the catalog, CMI retrieves the corresponding data from Teamcenter and starts the defined catalog function (see section *Update or Create* Catalog for function details) with the retrieved file(s).

Manage Catalogs

The *Manage Catalogs* icon is enabled when you have a CATProduct active. It allows to retrieve catalogs from Teamcenter for browsing.

After starting the *Manage Catalogs* command the select catalog dialog will be opened in CATIA.

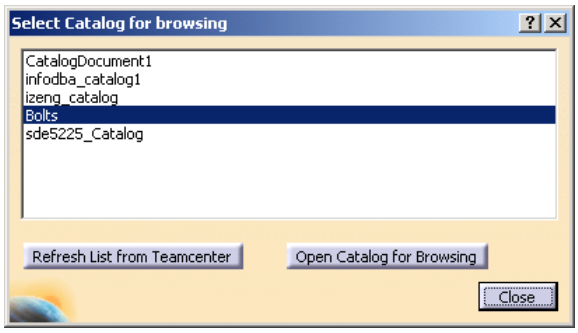

Figure 51: Select catalog for browsing dialog

Pick from the list of catalogs available in Teamcenter and select the *Open Catalog for Browsing* button. The catalog will be opened in a CATIA Catalog browser.

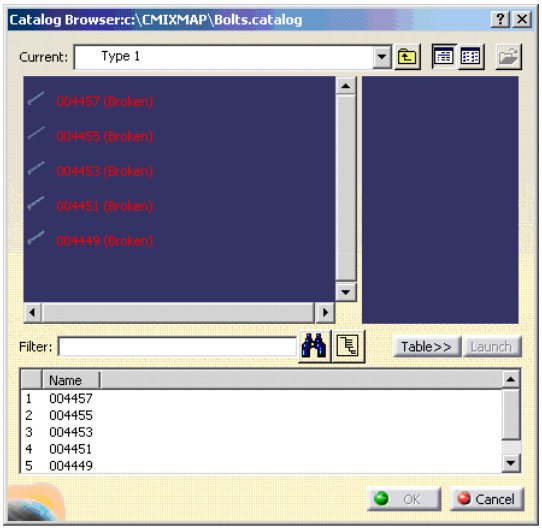

Figure 52: Bolt Catalog opened in CATIA

Double click an end chapter to open the item from Teamcenter. CMI RII will load the required files from Teamcenter if needed.

You cannot use Open Document or Open As New Document in this context.

Read Catalog

The administrator needs to set `CMI_ENABLE_CMICATALOGREADCMD=ON` in the CATIA environment in order to enable the *Read Catalog* function.

The *Read Catalog* icon is enabled when you have a CATProduct active. It allows you to retrieve catalogs from Teamcenter for editing.

After starting the *Read Catalog* command the Select Catalog dialog will be opened in CATIA. *Read Catalog* allows to retrieve CATCatalogs from Teamcenter for editing.

Only catalogs with write access in Teamcenter will be retrieved from Teamcenter.

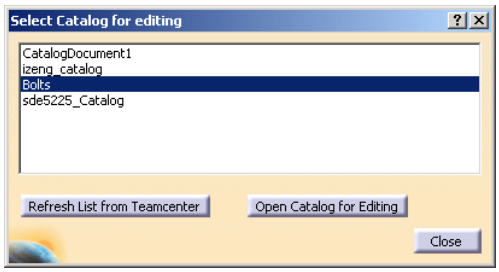

Figure 53: Select Catalog for editing dialog

Pick from the list of catalogs available in Teamcenter and select the *Open Catalog for Editing* button. The catalog document will be opened in a CATIA catalog editor.

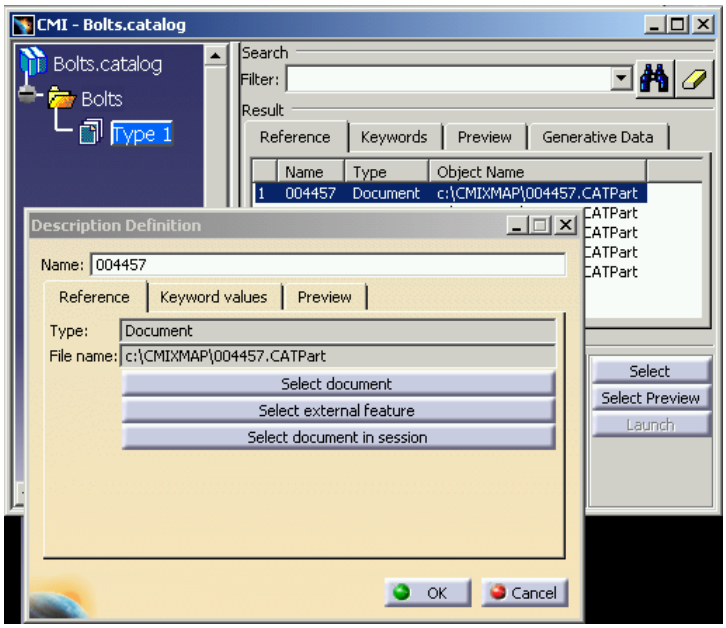

Figure 54: Edit catalog in CATIA

Use the *Select document in session* button to build the catalog from parts or files that were opened from Teamcenter via CMI.

Update or Create Catalog

The administrator needs to set `CMI_ENABLE_CMICATALOGUPDCRECMD=ON` in the CATIA environment in order to enable the *Update or Create Catalog* function. The *Update or Create Catalog* icon is enabled when you have a catalog active.

Update or Create Catalog allows saving new or updated catalogs in Teamcenter. It will manage the necessary keyword attributes so that the items contained in the catalog can be retrieved from the Teamcenter database instead of the local file

system.

After starting the *Update or Create Catalog* command the catalog document is analyzed and the following dialog is shown.

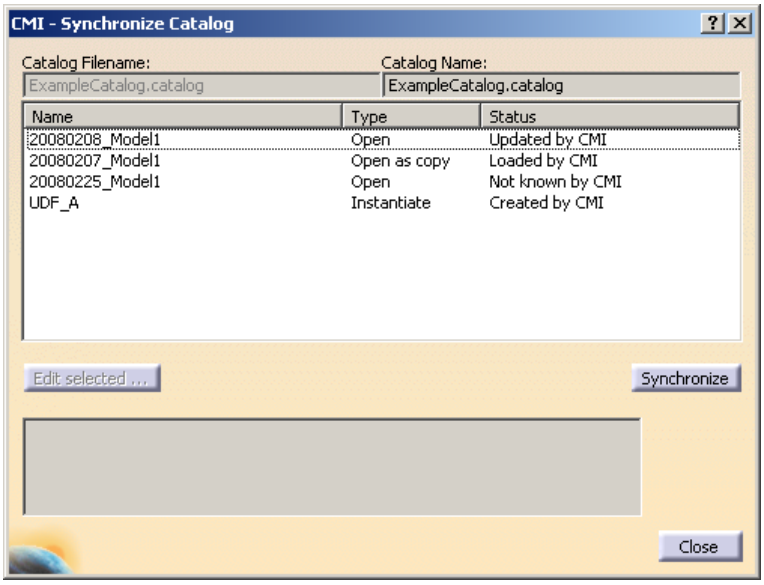

Figure 55: Synchronize Catalog dialog

For new catalogs that were not already opened from Teamcenter, the user can change the file name which should be used in Teamcenter.

The type can be **Open**, **Open as copy**, and **Instantiate**. These types describe the way in which the items will be handled when the catalog is used with the **Manage Catalogs** command.

- **Open**
  Open the referenced object in CATIA so that the existing Teamcenter item is used (e.g. standard part).
- **Open as copy**
  Open the referenced object in CATIA so that it will be recognized as a new item in Teamcenter (e.g. templates).
- **Instantiate**
  Instantiate is used for Features and Power copies which will be instantiated in the active document.

The Status Field can be **Loaded by CMI**, **Updated by CMI**, **Not known by CMI**, and **Created by CMI**

- **Loaded by CMI**
  The item is from Teamcenter and already was in the catalog at load time.
- **Updated by CMI**
  The item is from Teamcenter but the type was changed.
- **Not Known by CMI**
  The item is not from Teamcenter.
- **Created by CMI**
  The referenced file was opened from Teamcenter and is new in the catalog.

After pressing the *Synchronize* icon CMI will check all entries and if there are unknown entries, the user must confirm, that he will continue.

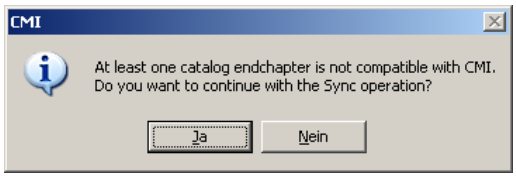

Figure 56: Confirmation dialog for unknown entries in the synchronize catalog command

This indicates that there is an item referenced in the catalog, that is not managed by Teamcenter.

Next the catalog is registered or updated in Teamcenter.
Figure 57 is shown when the Synchronize command ends successfully.

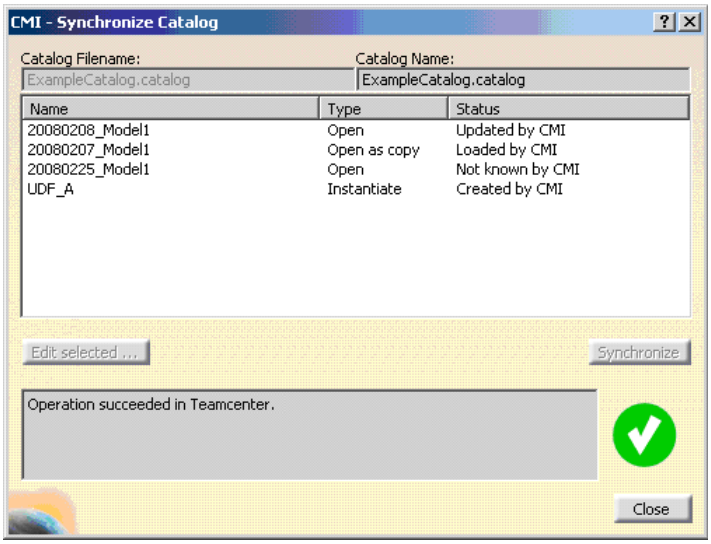

Figure 57: Synchronize Catalog with success

Support of CATIA V5 Released Cache

CMI supports the use of cgr files in the *released cache* of CATIA V5. For this purpose the cgr files of CATIA models are stored in Teamcenter. 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.

Use **Edit→Options** to set the CMI RII preferences.

Set “Transfer CGR-File to CATIA V5” to “Only CGR” or “CGR + geometry”

Use **Tools→Options** to set the preferences in CATIA V5.

Set “Work with the Cache System” to “On” and set the path to the released cache.

“Check timestamps” must be “On”.

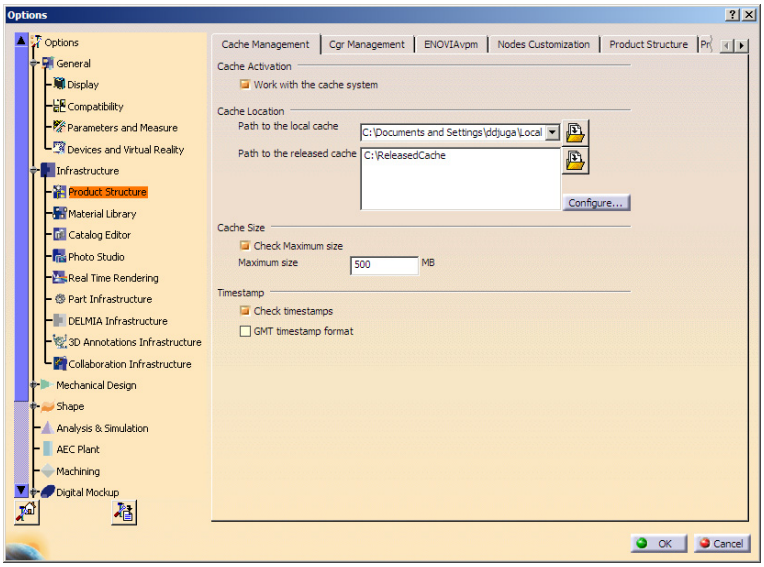

Figure 58: Tools→ Options dialog in CATIA

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 CATIA V5 configuration-variable CMI_USERELEASEDCACHE is set to “ON”.
- The CMI RII preference “Transfer CGR-File to CATIA V5” is set to “Only CGR” or “CGR + geometry”.
- A valid cgr file exists for the CATIA model in Teamcenter.

If the CATIA V5 configuration variable `CMI_SAVEADDITIONALCGR=ON` is set, CMI RII stores a cgr file in Teamcenter whenever a CATPart is created/updated.

Optional commands

In the following section, optional commands are described which must be enabled by the administrator.

Update Position

To enable this function the administrator has to set the CATIA V5 environment `CMI_ENABLE_CMIUPDATEPOSITIONCMD=ON`.
This function updates only the transformation matrices in Teamcenter. The CATProduct and CATPart files are not updated.

Update Parts

To get this function the administrator has to set the CATIA V5 environment `CMI_ENABLE_CMIUPDATEPARTCMD=ON`.
This function updates all modified CATPart files in Teamcenter. Transformations and CATProduct files are not updated.

Restore Positions

To enable this function the administrator has to set the CATIA V5 environment `CMI_RESTORE_POSITION=ON`.
This function restores the positions to their original state loaded from Teamcenter.

Choose Update Position

To get this function the administrator has to set the CATIA V5 environment `CMI_ENABLE_UPDATEPOSITIONDIALOG=ON`.
This function is available for normal “Update” and the “Update Position” command.

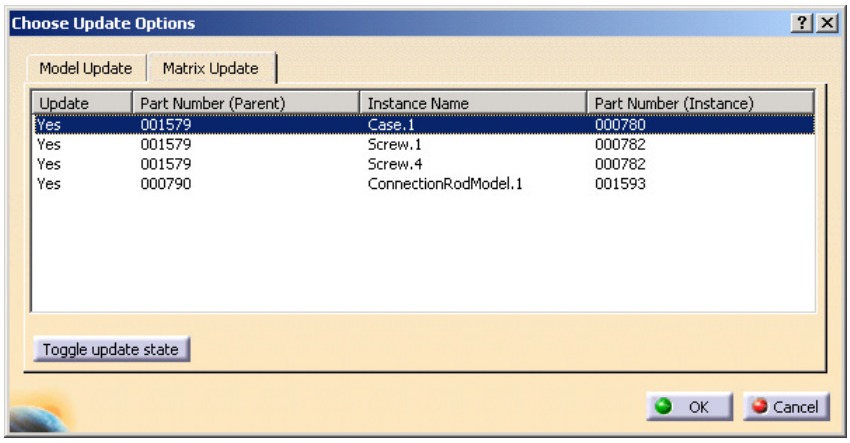

Figure 59: Choose Matrix To Update dialog

If you have changed some positions in the CATIA session this function allows you to select the matrices to be updated. If you select a line in the dialog the corresponding instance will be highlighted in the CATIA structure tree.

- Yes → Matrix will be updated in Teamcenter
- No → Matrix will stay unchanged in Teamcenter

To change the state you can double click a line or select the line and press the *Toggle keep state* button.

Choose Update Geometry

To get this function the administrator has to set the CATIA V5 environment `CMI_ENABLE_UPD_MODELSELECT_DIALOG=ON`.
This function is available for normal *Update* and the *Update Parts* command.

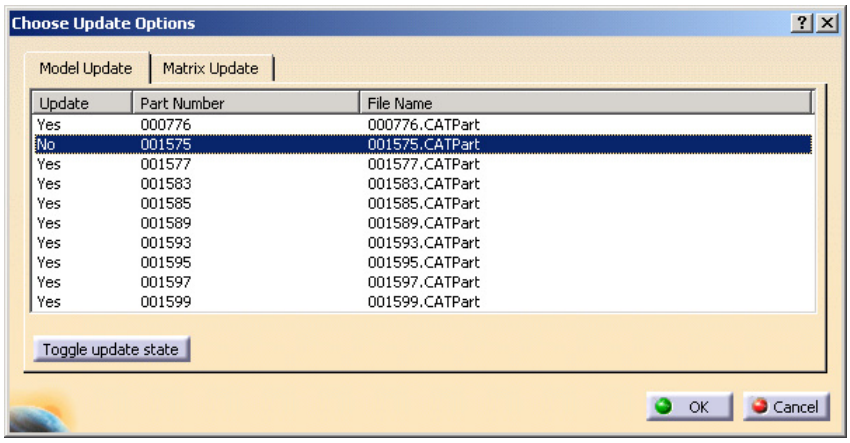

Figure 60: Choose Model To Update dialog

If you have changed some geometries in the CATIA session this function allows you to select the models to be updated.

- Yes → Model will be updated in Teamcenter
- No → Model will stay unchanged in Teamcenter

To change the state you can double click a line or select the line and press the *Toggle keep state* button.

Glossary

Administrator

Person, who configures the system, inserts users, manages user permissions and maintains the database.

Browser

Window that displays *icons* representing *objects*.

Check In

Action that takes from the users the right to update a work item.

Check Out

Action that gives the user the exclusive right to update a work item. Checking out an item creates a copy of the work item in the specified work location.

Configuration Context

An attribute that determines which Item Revision should be displayed (for example, only the last revisions).

Dialog Window

Window in which the user enters information.

Field

Component of a *dialog window* in which the user can enter text.

Generic Workbench

Set of components such as windows, dynamic items and information items, used in *Teamcenter* integration products.

Icon

Graphical representation of an *object*.

List View

Style of *browser* window in which *objects* are viewed as small *icons* in a list with attribute values in columns.

Object

An item or relationship.

Palette

Any group of CATIA functions created for specific requirements.

Plot File

File in a specified format that can be plotted or opened in a viewing tool and represents a 2D view of a model.

Pop-Up Menu

The menu that appears when the user points an *icon* and holds the right mouse button pressed.

Query

To search the database for *objects* that match specific criteria.

Register

To make a file or directory known as an *object* maintained in the Teamcenter database.

Tree View

Style of *browser* window in which *objects* are displayed as relationships in horizontal branches.

Vault

Secure place for storing information. To access the data confined in a vault it is necessary to *check out* or revise the data.