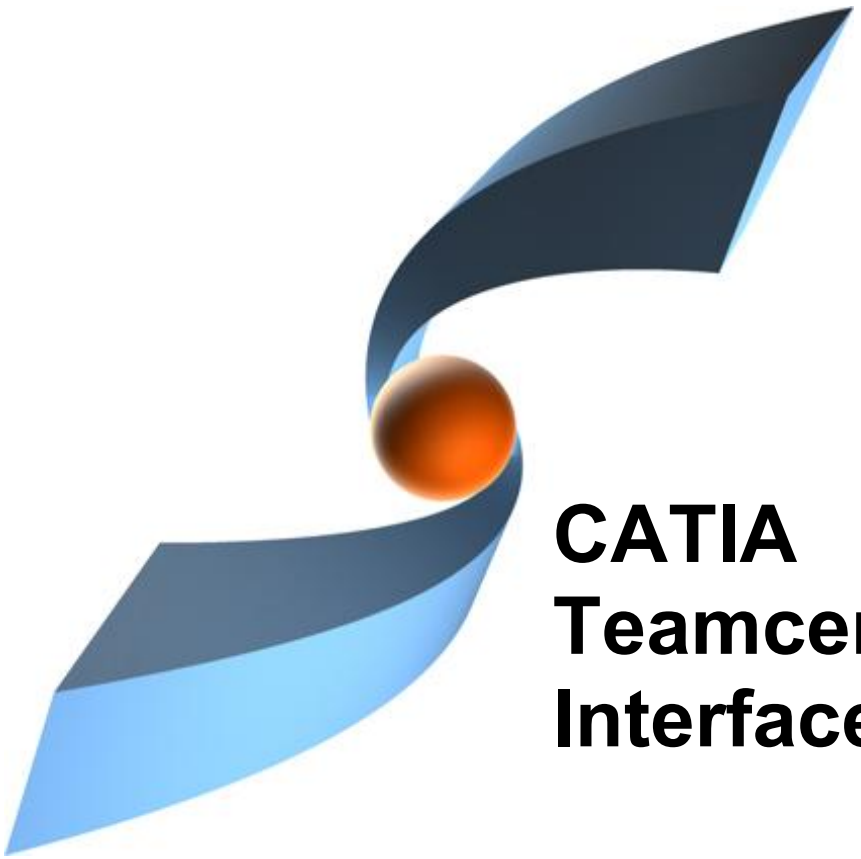

CATIA Teamcenter Interface

CMI Release 10.19

User Manual

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Preface

This manual describes the main functionality delivered by the CATIA Teamcenter Interface (CMI) and includes the creation, storage, modification and management of CATIA models and assembly structures in the Teamcenter Enterprise 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 Enterprise *Object Management Framework* (OMF) as described in the Object Management Framework User's Manual.

Related Documents

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

Manual Title	Version
CATIA Teamcenter Interface Installation & Administration Guide	10.19
CATIA Teamcenter Interface User Manual	10.19
CATIA Teamcenter Interface Customizing Guide	10.19

Organization

This manual contains the following chapters, some are specific to a particular version of CATIA, either V4 or V5, or are valid for both:

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 CATEDM Module in the CATIA Function *Palette*. Assumed is that the CATEDM module will be referenced under the function name 'METAPHSE' within the CATIA working session (default setting during the installation).

Chapter 3 introduces the CATIA Workbench and lists its functions. The goal of this chapter is to make the user familiar with this component of CMI. The CATIA Workbench functions will be explained in depth in subsequent chapters.

Chapter 4 describes how to create and search for CATIA models.

Chapter 5 describes how to load documents stored in Teamcenter into CATIA, how to update a model, how to save it under another name and how to create plot files.

Chapter 6 explains the concepts underlying the work with structures and describes how to create assemblies and components, how to search for parts, how to load parts into CATIA and how to modify the assembly structure and assembly positions.

Chapter 7 delves into the menu items of the METAPHSE function in CATIA and describes how to modify the structure and the position of sub-assemblies.

Chapter 8 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
METAPHSE	Is used to identify a CATIA V4 function.
	This icon is used to identify tips and attention advices.
	This *icon* is used to identify *vault* related tips.
	This icon is used to identify Teamcenter Enterprise related sections and actions.
	This icon is used to identify CATIA related sections and actions.
	This icon is used to identify CATIA V5 related sections and actions.
	This icon is used to identify actions that need confirmation.

Your Comments are Welcome

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

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

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

CMI 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 Enterprise. 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) 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;

Attachment of plot files to their corresponding CATIA models;

Distribution of CATIA data in a network;

Simultaneous management of CATIA data and structures;

Construction of part structures within Teamcenter Enterprise;

Individual Parts to contain more than one physical CATIA model;

Modification of the position of the structures;

Search for CATIA data by different attributes;

Automatic loading of attributes stored in Teamcenter Enterprise to fill the title blocks of CATIA drawings.

CMI Architecture

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

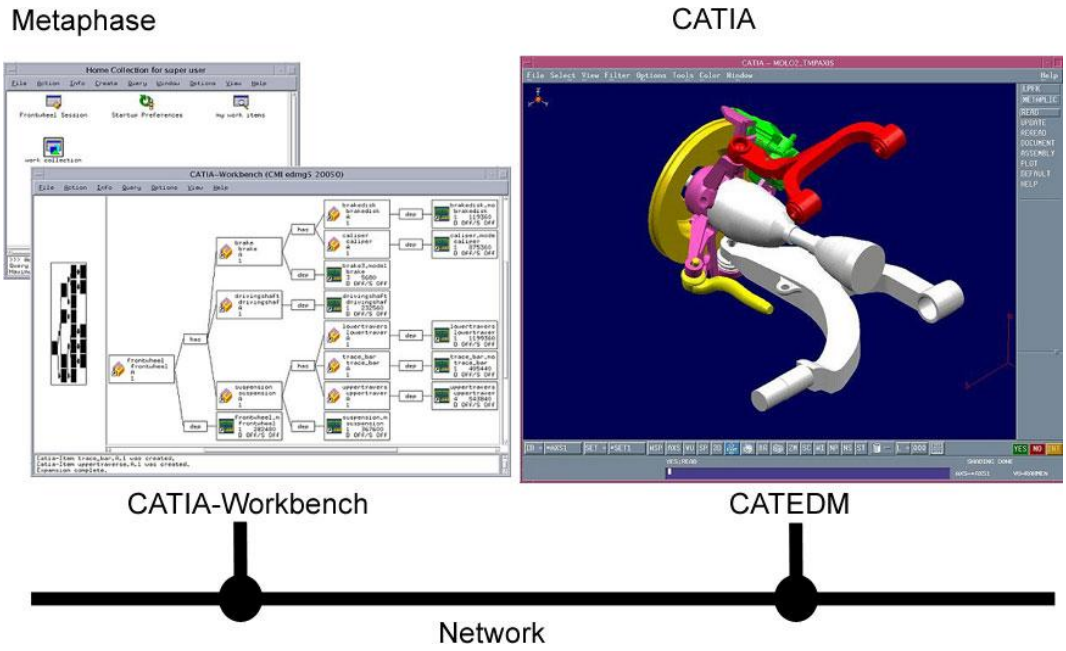


Figure 1: CATIA Teamcenter Interface architecture

The user can expand an assembly within the CATIA Workbench and send this assembly to CATIA V4 or CATIA V5. After changing geometries and / or positions in CATIA the user can update the assembly in Teamcenter Enterprise.

CHAPTER 2

Getting Started

This chapter describes how to start CATIA within the Teamcenter Enterprise environment and also how to enable the CMI functionality within CATIA V4 and CATIA V5.

Assumptions

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

The CATIA V4 functionality is installed with the menu item name ‘CMI’ in CATIA.

CATIA can be called both directly from the user’s desktop environment and from a UNIX shell via the command ‘catstart.sh’.

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 it is necessary to start the OMF, the classic Teamcenter Enterprise user interface, and the Motif CATIA User Interface or the Windows version of CATIA.

For information on starting OMF see the Object Management Framework User’s Manual – chapter 1 (Before You Begin), section Starting OMF provided by UGS PLM Solutions.

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

If the *CATIA Launching Tool* is installed, it is possible to start CATIA from the OMF *Browser* in Teamcenter Enterprise by double-clicking the *icon*:

To install the CATIA Launching Tool you should take a look at the *CATIA Teamcenter Installation Manual* – chapter 2 (Installing CATIA Launching Tool).

If CATIA is not started out of the OMF, it makes no difference if Teamcenter Enterprise or CATIA is started first.

Enabling the CMI toolbar in CATIA V5

In CATIA V5, select the menu **View>Toolbars>CMI , View>Toolbars>CMIGeometry Tools, View>Toolbars>Advanced Tools**

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.

Enabling CMI in the CATIA V4 Function Palette

In CATIA, select the KEY permanent function or the **FILE** > **KEYBOARD** item.

Select *METAPHSE* in the list of functions on the right.

Select a function key on the left (select a function that will probably not be used).

The *METAPHSE* function will be swapped with the function selected in step 3.

It is possible to store the *palette* so that it is not necessary to enable the METAPHSE function every time CATIA is restarted. For information about storing *palettes* see the *Motif CATIA Solutions User's Guide* – chapter 3 (*Presenting the Motif CATIA User Interface*), section *Storing Palettes: STORE Item*.

If the METAPHSE option is not in the list of functions on the right, 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 CATIA Workbench

This chapter introduces the *CATIA Workbench* 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 *CATIA Workbench* functions will be explained in subsequent chapters.

What is the *CATIA Workbench*?

The *CATIA Workbench* is a standard Teamcenter Enterprise structure *browser*, used to graphically 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 Enterprise and CATIA is achieved via the *CATIA Workbench*.

First of all you should register or create some CATIA model objects in Teamcenter Enterprise. CMI defines some new classes in Teamcenter that can be used to manage CATIA models plus plot files generated from them, and also new classes to manage the CATPart, CATProduct and CATDrawing files of CATIA V5. The management of CATIA models is described in *Chapter 5*, CATIA V5 files in *Chapter 8*

The CATIA Workbench in Teamcenter Enterprise has to be started before you can call the functions of CMI (CATIA Teamcenter Interface).

To open the *CATIA Workbench* either select the Teamcenter Enterprise function:

Window > ***CATIA Workbench***

Or select a CATIA model in Teamcenter Enterprise and double click on it

A window as the one seen below will appear:

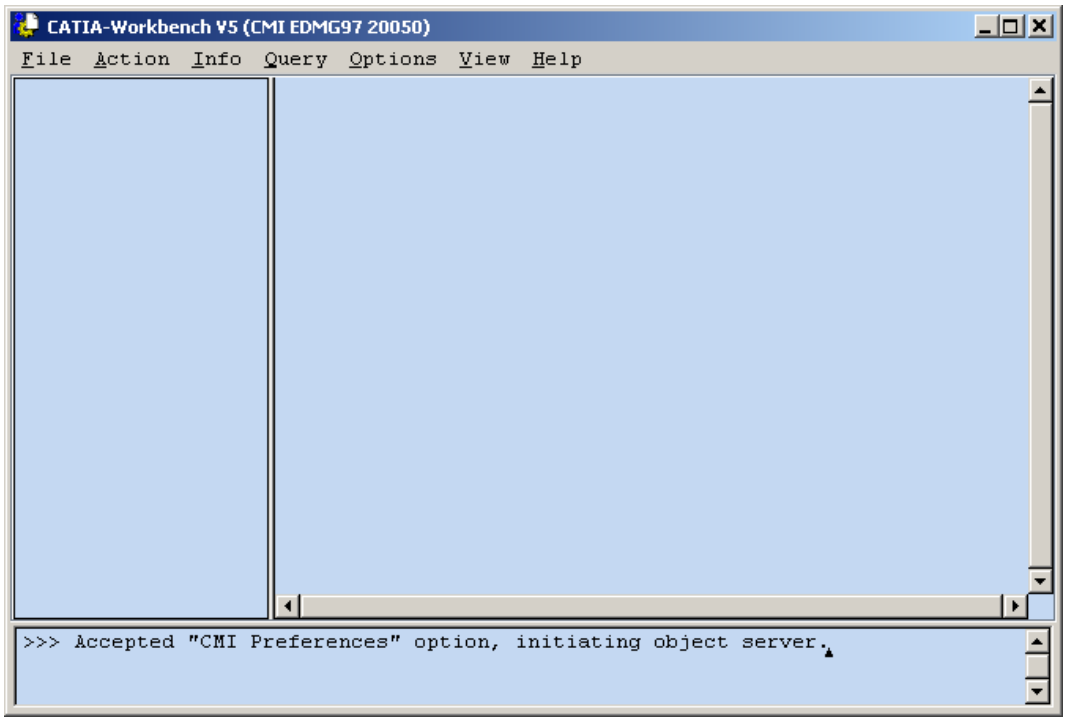


Figure 2: CATIA Workbench window.

The initialization of the communication between CATIA and Teamcenter Enterprise was successful if the title of the *CATIA Workbench* has the following format:

Figure 3: CATIA Workbench Title bar description

If the title bar of the CATIA Workbench doesn't show your workstation host name and a port number, please consult your administrator.

The CATIA Workbench Menu

Below the CATIA Workbench menu items will be described.

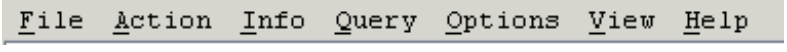


Figure 4: CATIA Workbench menu items.

Menu item File

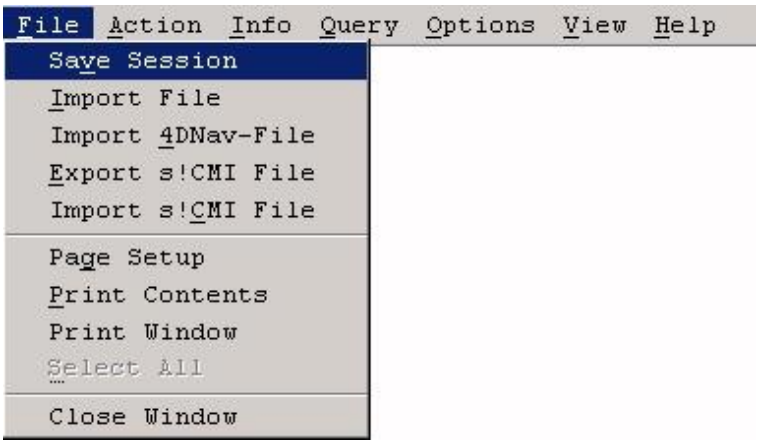


Figure 5: CATIA Workbench menu 'File'.

The menu item *File* > *Save Session* will save the contents of the CATIA Workbench in the database. To restore the contents of the CATIA Workbench use the standard Teamcenter Query for '*Saved CATIA Session*', please refer to chapter 5.

The menu item *File* > *Import 4D-Nav File* will load a scene-file that had been written and modified by the 4D-Navigator. All models below parts that are 'visible' in the 4D-Nav will become marked in the CATIA-Workbench.

The other points in the menu item File contain standard Teamcenter Enterprise File items. For more information about the entries, please refer to:

Help > Menus

Menu item Action

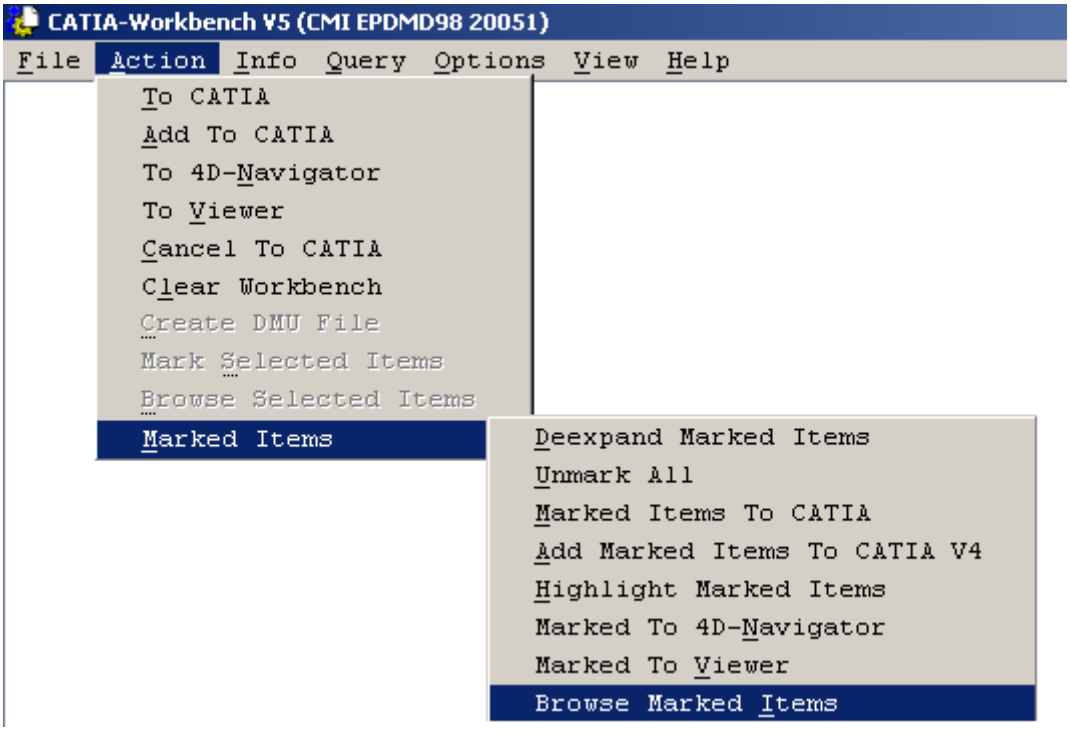


Figure 6: CATIA Workbench menu 'Action'.

The *Action* menu allows the sending of the CATIA Workbench contents to CATIA V4 and CATIA V5.

The menu item *Action* > *To CATIA* will send the contents of the CATIA Workbench to the CATIA session and the CATIA session window will appear.
This menu item is only selectable, if items are present in the CATIA Workbench.

The menu item *Action* > *Add To CATIA* will add additional contents or remove deexpanded contents of the CATIA Workbench to the CATIA session and the CATIA session window will appear. Only the newly added items will be loaded into the existing Catia session.
This menu item is only selectable, if items are present in the CATIA Workbench.

If working with **Catia V5**, *Add To Catia* will add the Workbench contents to a single product Window, below a synthetic Product tagged “CMI Workbench”.

The menu item *Action* > *To 4D*-Navigator will send the contents of the CATIA Workbench to the 4D-Navigator that will appear. If there is still a 4D-Navigator running, the content of the CATIA-Workbench is added to the former content of the 4D-Navigator.

With the “To viewer” command the content of the CMI Workbench can be sent to a specific viewer to obtain a quick preview out of TC Enterprise. This functionality is also available without a CATIA V5 installation!

The menu item *Action* > *Cancel To CATIA* cancels the communication between CATIA Workbench and CATIA session and the CATIA session window will appear.

The menu item *Action* > *Clear Workbench* will erase all data in the Workbench.

The menu item *Action* > *Mark Selected Items* will mark all items that are currently selected.

The menu item *Action* > *Browse Selected Items* will generate a Browser with all selected persistent objects. (The Workbench contains only dynamic Representation-objects of persistent Teamcenter-objects, to work on the origin-objects you should apply this command)

The menu item *Action* > *Marked Items* > *De-expand Marked Items* removes the marked parts from the *CATIA Workbench* including all subassemblies.

The menu item *Action* > *Marked Items* > *Unmark All* will unmark all items in the workbench.

The menu item *Action* > *Marked Items* > *Marked Items To CATIA* will send all marked items to CATIA.

The menu item *Action* > *Marked Items* > *Add Marked Items To CATIA V4* will send all marked items to CATIA V4.

The menu item *Action* > *Marked Items* > *Highlight Marked Items* will highlight all marked items in CATIA.

The menu item *Action* > *Marked Items* > *Marked to 4D-Navigator* will send all marked items to the 4D-Navigator.

The menu item *Action* > *Marked Items* > *Marked to Viewer* will send all marked items to a specific Viewer.

The menu item *Action* > *Marked Items* > *Browse Marked Items* will generate a Browser with all marked persistent objects

The menu item *Action* > *Create DMU File* creates a CMI-DMU-File for the clicked Part in the DMU-Vault Location. Every Revision of the Part has its own DMU-File stored in the Vault Location. If the DMU-File already exists in the Vault, the user has to confirm the overwrite. The Root-Part is expanded for all Catia-Models. For each Catia-Model an entry is stored in the file with the absolute postion and the assembly path from the Root-Part to the Model.

Menu item Info

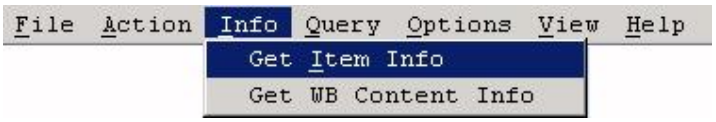


Figure 7: CATIA Workbench menu 'Info'.

The menu item *Info* > *Get Item Info* displays extended information about an item that the user has selected in the CATIA Workbench.

The menu item *Info* > *Get WB Content Info* displays the number of CATIA parts and models within the CATIA Workbench. Also the size of the contained CATIA models will be displayed. This information dialog helps to estimate the size of the CATIA-Session.

Menu item Query

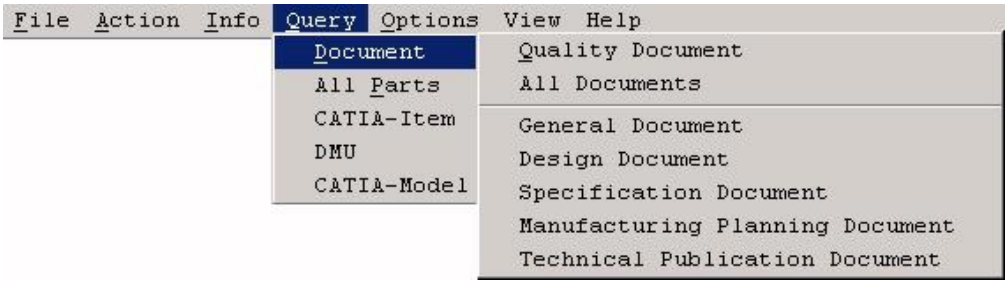


Figure 8: CATIA Workbench menu 'Query'.

The menu item *Query* searches for documents and parts (standard Teamcenter Enterprise).

The menu item *Query* > *CATIA-Item* searches for CATIA-Items within the Workbench. All items matching the search criteria are marked (for more information about marked items, please refer to '*Working with marked Parts*').

The menu item *Query* > *CATIA-Model* searches for CATIA-Models within the Workbench. All models matching the search criteria are marked.

Menu item Query > DMU

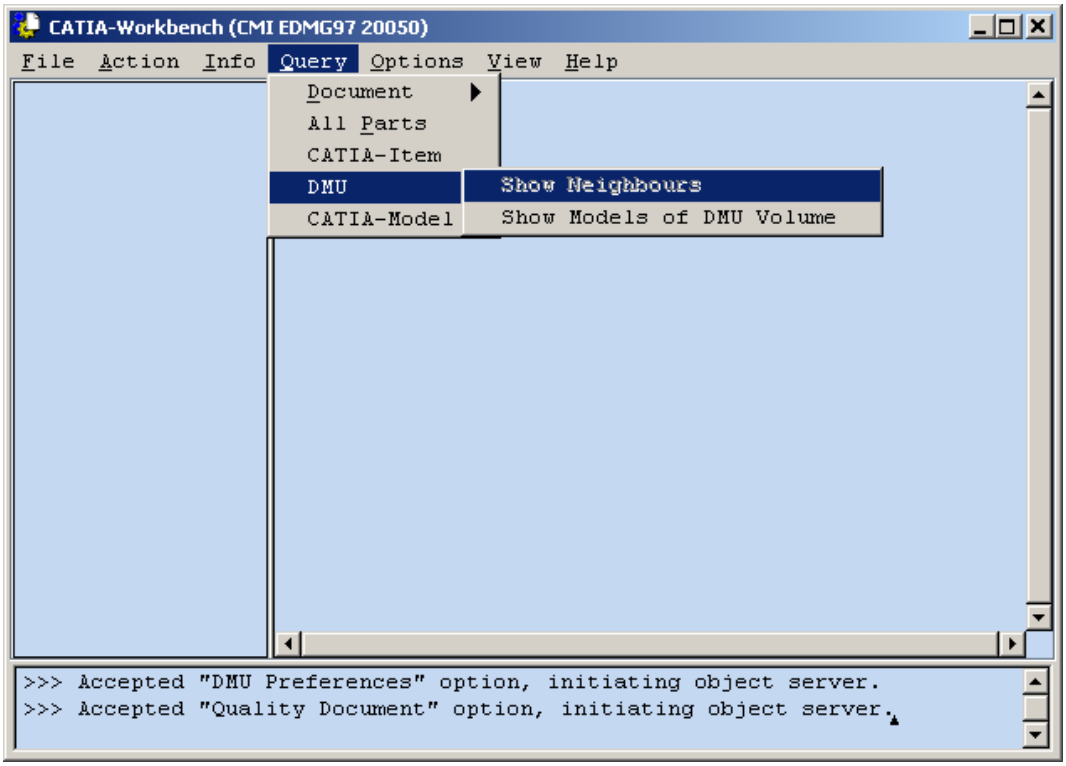


Figure 9: CATIA Workbench menu 'Query DMU'.

You should use this functionality only for frozen data. Therefore use the configuration context to query only for frozen data.

The menu item *Query* > DMU > Show Neighbours is active when at least one Catia-Model is selected (*selected* models have thick black frames). A dialog window is presented to ask for the root part of all selected Catia-Models:

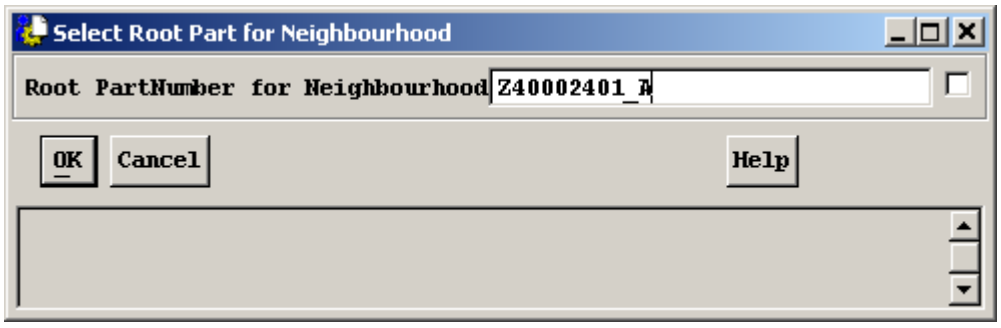


Figure 10: root Part for neighbourhood search.

After pressing the OK-Button, the neighbourhood search starts, and all neighbour models of all the selected Catia Models are presented in the Workbench under the same root Part. All clicked Catia Models are highlighted in the workbench.

Only Catia Models with Bounding-Boxes are searched.

Prior to the Neighbourhood-Search a CMI-DMU-File must be created for the root-Part via the Workbench-Action > "Create DMU-File"

You can modify the extent of the neighbourhood by changing the DMU-Tolerance in

the DMU-Preferences. That means the bounding box values of the selected Catia-Model are augmented with the DMU-Tolerance. (Default = 0.0)

The menu item *Query* > DMU > “Show Models of DMU-Volume” presents a Dialog-window to ask for the root Part and for the DMU-Volume:

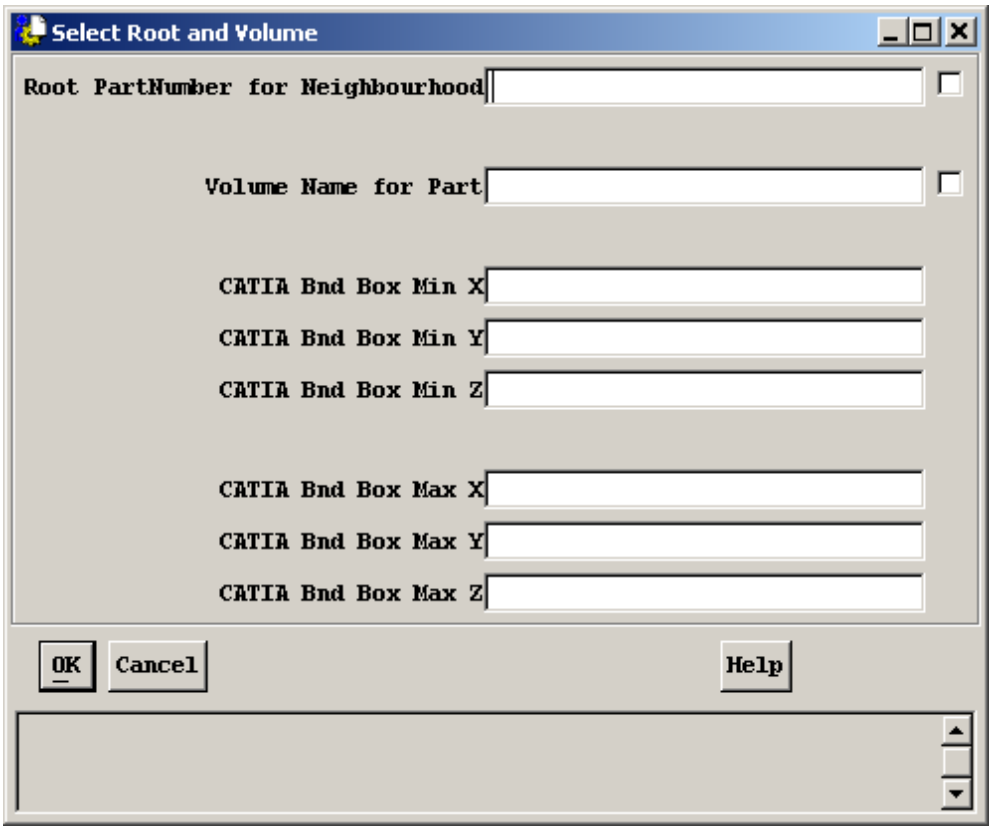


Figure 11: Dialog for selection of root and volume.

If you click the selection box for the root part, all Part Numbers and Revisions are shown for which a CMI-DMU-File exists. When the root part entry is set, a click on the volume name for Part selection box shows all Volumes for this specific root Part.

If there is no volume that meets your requirements for the root Part, you can perform a dynamic “Volume Search” by specifying all bounding box values in the dialog window.

When pressing the OK button, the Volume search begins, and all Catia models within the selected Volume will be shown in the Workbench under the same root Part.

Only Catia-Models with Bounding-Boxes are searched.

Prior to the Volume-Search a CMI-DMU-File must be created for the root Part via the Workbench-Action > “Create DMU-File”.

Prior to the Volume-Search a DMU-Volume must be created for the root Part via Action > “Create DMU-Volume”.

Menu item Options

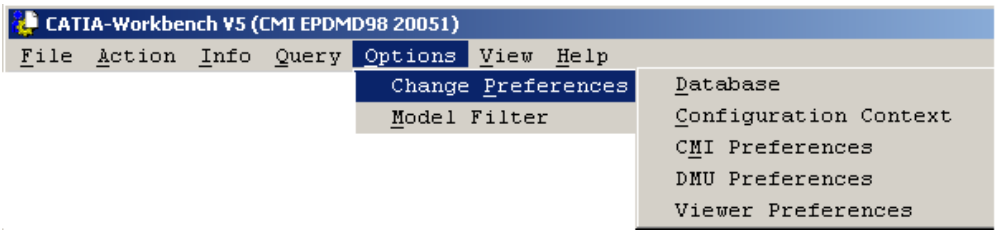


Figure 12: CATIA Workbench menu 'Options'.

The Options Menu has two Menu items the *Change Preferences* Submenu and the *Model Filter* Option.

The menu item *Configuration Context* sets/changes the user's *configuration context*.

Within the database preferences you can define the query scope for the databases to be searched in.

If the user wants to work with CATIA the settings at *Options* > *CMI Preferences* have to be set correctly.

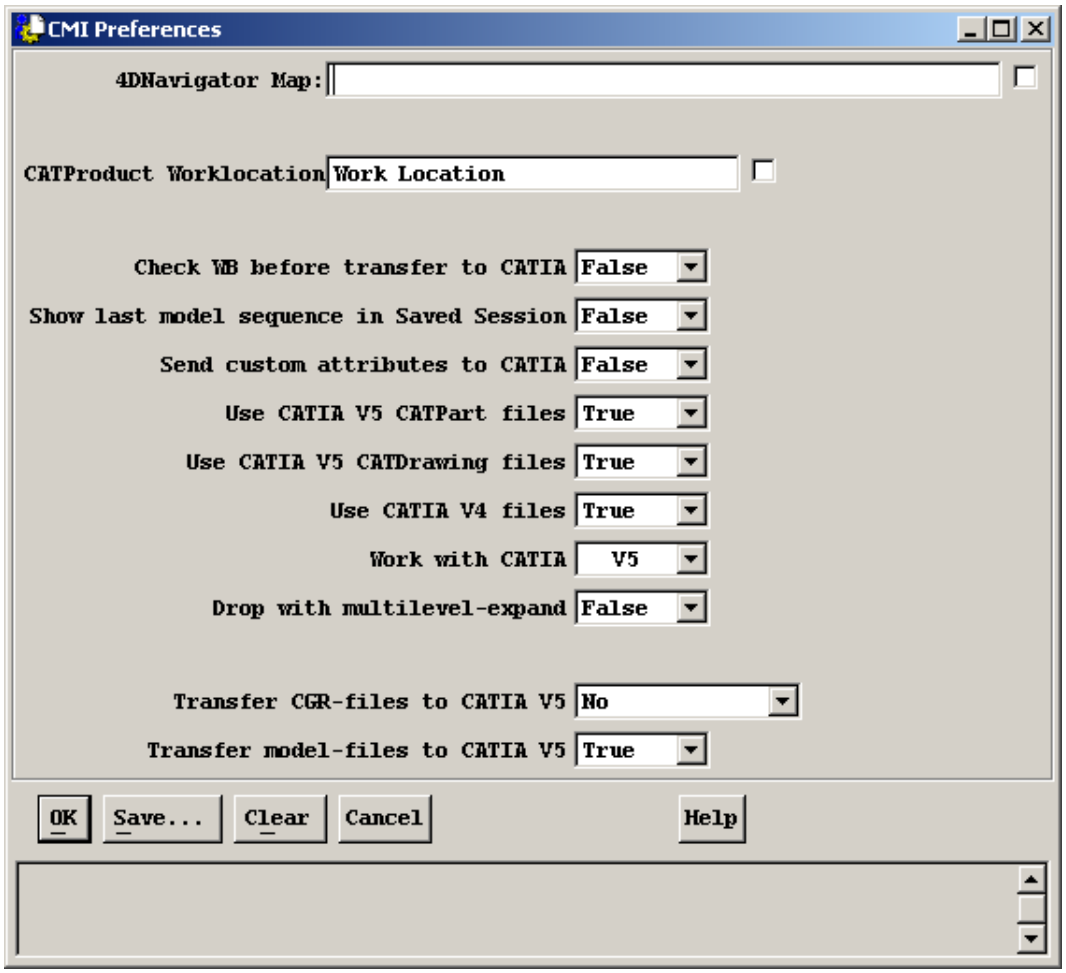


Figure 13: CMI Preferences menu

4D Navigator Map: If 4D Navigator integration is desired user has to set path for 4D Navigator exchange map.

CATProduct Worklocation: Location where CATProducts and other files created from within Catia V5 are saved.

Check WB before transfer to CATIA: If user tries to transfer a Workbench content it will be checked if it probably will not fit to the CATIA Environment Setup. This will avoid an unsuccessful attempt of transferring a lot of data that may cause an error message or even a CATIA system crash.

Show last model sequence in Saved Session: Is this button set, saved session is reloaded with the newest sequences of models in CATIA Workbench.

Use CATIA V5 files : CATDrawings and CATParts are sent to CATIA V4 / V5 This attribute was labelled **Enable CATIA V5 files** in former releases.

Use CATIA V4 files : CATIA V4 Models are sent to CATIA V4 / V5

Work with CATIA: User can choose between work with CATIA V4 or CATIA V5.

Drop with multilevel-expand: Automatic expand of the product structure when an Assembly is dropped onto the Workbench

Transfer CGR-Files to CATIA V5: Preference allows user to send CGR files to CATIA V5 (see *Working with CATIA V5 Released Cache*).

Transfer model-files to CATIA V5: Preference allows user to not send model files to CATIA V5. Usually this preference should remain *not set* or **true**. It is used with the optional *Get original geometry from Teamcenter* feature in Catia V5.

For further information about working with CATIA V5 please read the CMI Preferences chapter for CATIA V5.

Use the 'Check WB before transfer to CATIA' preference only when you are handling very large CATIA-Sessions, because it will reduce the interface performance.

DMU Preferences

If user wants to use DIGITAL MOCKUP (DMU) you can change the tolerance (extension) of the bounding box for the neighbourhood-search via the *DMU Preferences* menu option. This tolerance is added to all bounding-box-values of the selected Catia-Model.

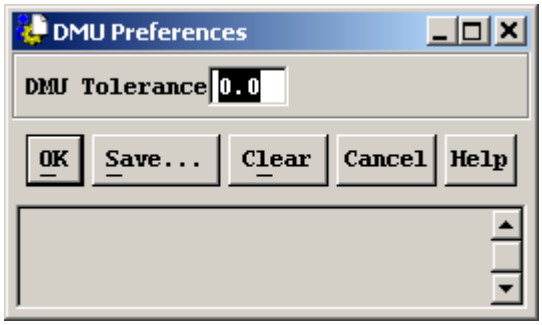


Figure 14: DMU Preferences menu

Model Filter

The Menu item Model filter will start the Model Filter preferences Dialog (Figure 15), which is used to set the actual Model Filter.

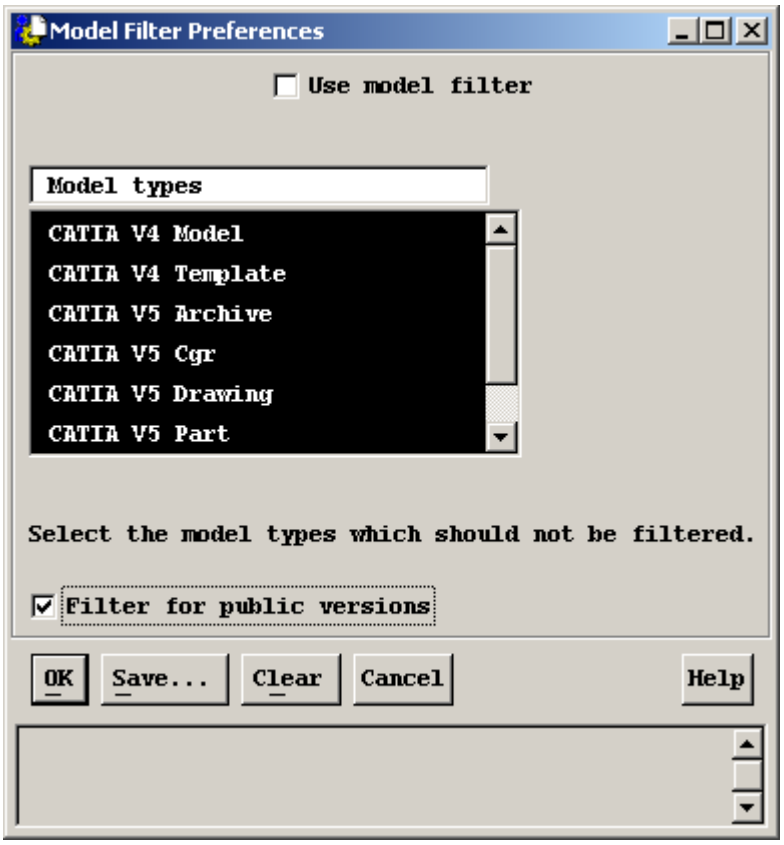


Figure 15: Model Filter Preferences dialog

The *Use model filter* option will define, if the Filter should be used. If the Filter is Used, the selected Model types will pass in the Expand Function of the CATIA Workbench, unselected Model types will be filtered.

Filter for public versions

If the Filter for public versions is activated, only private and public Catia-Files are transferred to Catia. Private means that the file is owned by the actual user, public means that the file is checked in to a vault. If there is a checked out file of another user during “To Catia”, the latest public predecessor file is transferred to Catia, if existent.

Menu item View

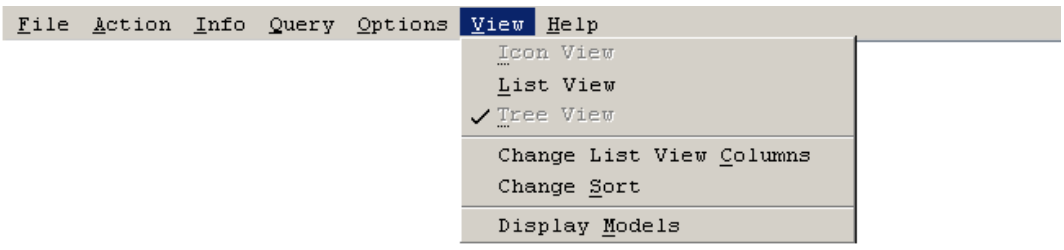


Figure 16: CATIA Workbench menu ‘View’.

The menu item *View* displays items in the *CATIA Workbench* in *list view* or in *tree view*. You can also modify the list view columns or sort the items. Finally you can choose whether models are to be displayed within the CATIA Workbench or not.

For more information about the entries, please refer to:

Help > Menu

Menu item Help

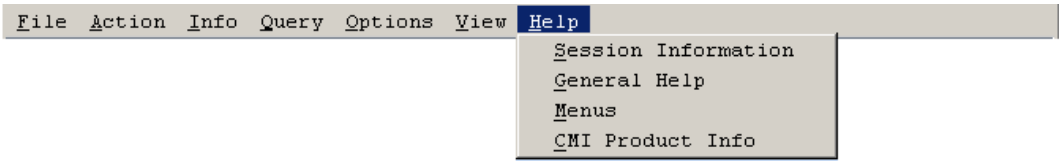


Figure 17: CATIA Workbench menu 'Help'.

The menu item *Help* displays information about your session, about the menus and their meaning. Additionally CMI product information can be displayed.

Pop-Up menus of Workbench Items

If you select a 'Part' or a 'Document' or a 'CATIA Model' and drop it into the CATIA Workbench, a new object will be created dynamically. The new created icon represents the CATIA relevant data of the dropped object. The icons in the Workbench are the same as in the Standard Teamcenter Enterprise browsers, with an additional arrow on the lower left corner indicating that this item is to represent an originating Teamcenter Enterprise object.

Parts in the Workbench

On the 'part representants' displayed in the Workbench browser there are the following functions available: (notice that assembly-parts and component-parts use the same icon.)

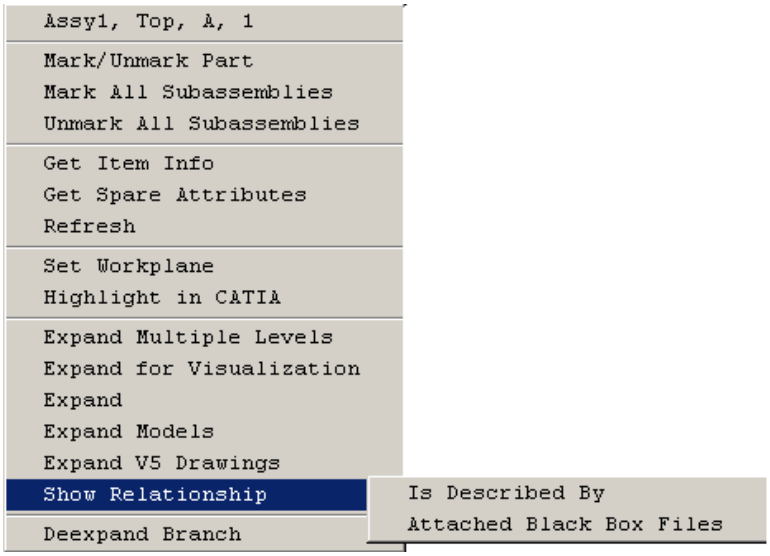


Figure 18: Context menu on Part in Workbench

Pop Up Menu Item		Description
Mark/Unmark Part	:	Marks the selected part representant
Mark All Subassemblies	:	Marks the part and all subassemblies
Unmark All Subassemblies.	:	Unmarks the part and all subassemblies
Get Item Info	:	Displays extended part representant information
Refresh	:	Refreshes the displayed attributes of the part representant
Set Work plane	:	Sets the working plane to the selected part representant
Highlight in CATIA		Marks the selected part in CATIA
Expand Multiple Levels	:	Expands the part relationship until the last level
Expand for Visualization	:	Expands the part relationship until the last level, if visualizations are found, the level expand is stopped and only the visualization is expanded.
Expand	:	Expands the part item relationship in one level
Expand Models	:	Expands only the models attached to the part
Expand V5 Drawings	:	Expands any CATDrawings attached to the Part
Show Relationship > Is Described By	:	Displays related Documents
Show Relationship > Attached Black Box Files	:	Displays any Black Box files attached to the Part
Deexpand Branch	:	Removes the selected part or assembly including all subassemblies

Documents in the Workbench

For the “document representative” displayed in the Workbench browser there are the following functions available:

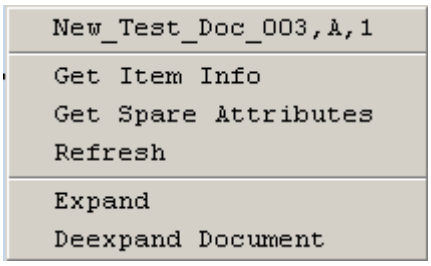


Figure 19: Context menu on Document in Workbench

Pop-Up Menu Item		Description
Get Item Info	:	Displays extended document representative information
Refresh	:	Refreshes the representation of the document
Expand	:	Expands the related models
Deexpand Document	:	Removes the selected document including all related models

Models in the Workbench

Context sensitive to the “model representatives” displayed in the Workbench browser there are the following functions available: (notice that you can decide to show or to hide the model representative with the toggle ‘View > Display Models’.)

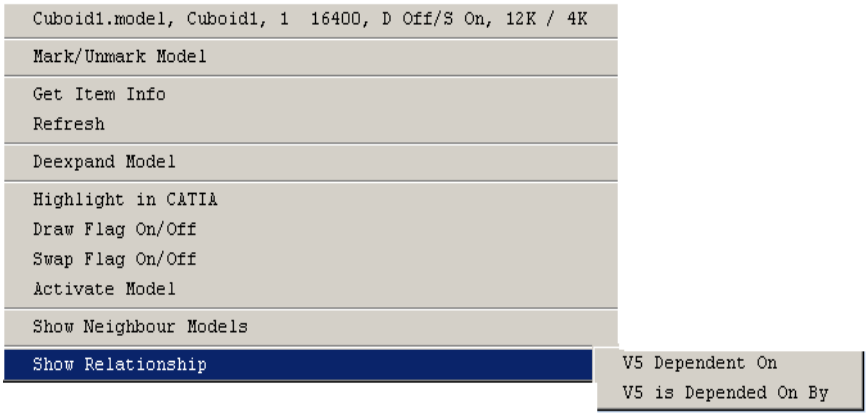
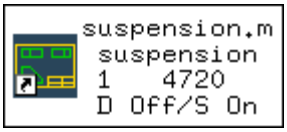


Figure 20: Context menu on Model in Workbench

Pop-Up Menu Item		Description
Mark/Unmark Model	:	Marks or unmarks the model representative
Get Item Info	:	Displays extended model representative information
Refresh	:	Refreshes the representation of the model
De-expand Model	:	Removes the selected model
Highlight In CATIA	:	Highlights the selected model in CATIA
Draw Flag On/Off	:	Toggles the Draw-Flag
Swap Flag On/Off	:	Toggles the Swap-Flag
Activate Model	:	Marks the model to be the active model in CATIA
Show Neighbour Models	:	Marks the neighbour models in Workbench (needs DMU Export File see Figure 11)

The model representative in the Workbench shows information about the model file and the flags you have set:



File name	
Description	
Sequence	File size (byte)
Draw Flag	Swap Flag

“Show Neighbour Models” performs the Neighbourhood-Search for the clicked CATIA-Model. If the complete path from the Root-Part to the clicked CATIA-Model is in the Workbench, and there exists a CMI-DMU-File for the Root-Part, the search is done immediately. Otherwise a Dialog-window is presented to ask for the Root part of the selected CATIA-Model:

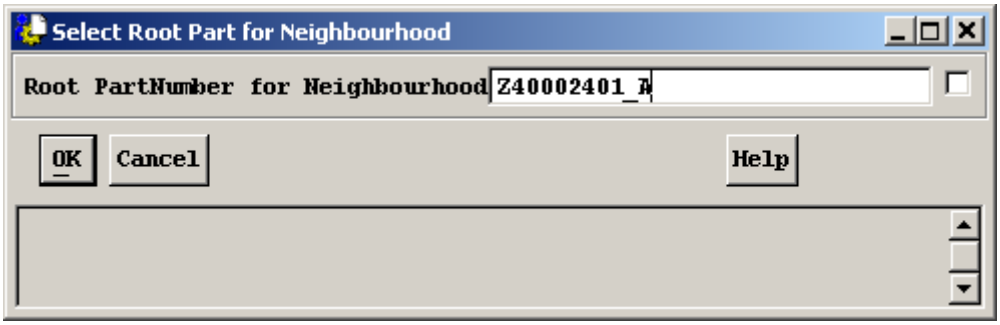


Figure 21: Select Root Part for Neighbourhood

By pressing the OK-button, the neighbourhood search starts and all neighbour models of the selected Catia models are presented in the Workbench under the root Part. The clicked Catia model is highlighted in the workbench.

You should use this functionality only for frozen data. Therefore use the configuration context to use only frozen data.

Only Catia models with bounding boxes will be searched.

Prior to the “Neighbourhood Search” a CMI-DMU-File has to be created for the root Part via the Workbench-Action > “Create DMU-File”

You can modify the extent/size of the neighbourhood by changing the DMU-Tolerance in the DMU-Preferences. That means the bounding box values of the clicked Catia model will be increased by the DMU-Tolerance. (Default = 0.0)

CHAPTER 4

General Functions

This chapter describes how to handle CATIA relevant data in Teamcenter Enterprise. It describes operations, such as creating CATIA models and assembly structures, as well as how to search for them.

Predefined Object Structure

The *CATIA Teamcenter Interface (CMI)* contains the following predefined object structure:

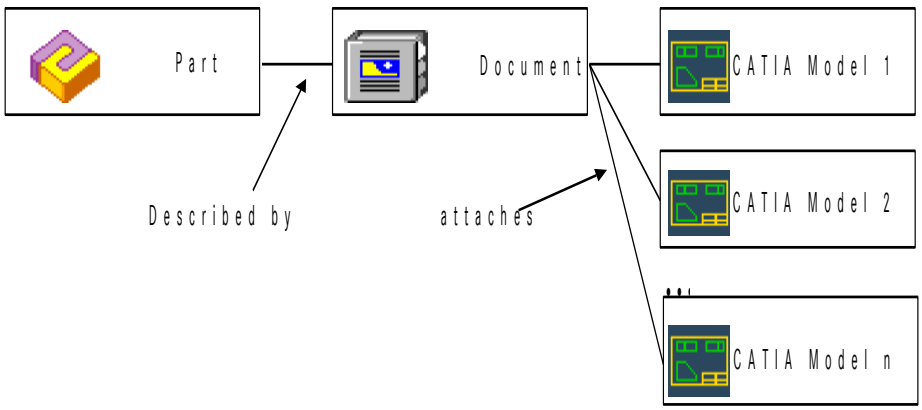


Figure 22: Predefined CATIA model structure.

For each part a relation ‘described by’ with a document element must exist. The relationship can be created by dragging any document object and by dropping it to the part object. Please refer to the Teamcenter user documentation for managing relationships between Teamcenter elements. The documents can contain at least one relation ‘attaches’ with CATIA model elements. You can attach more than one CATIA model to a single document, and more than one document can be associated with each part. Any CATIA model object can contain one relation ‘Model has plot file’ with any plot file, which can be created from CATIA.

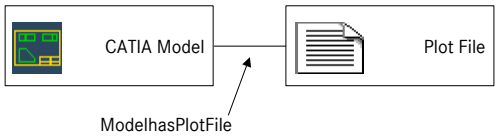


Figure 23: Plot file attached to a CATIA model.

All of the elements and the relations between them can be customized. For more information please refer to *CATIA Teamcenter Interface Customizing Documentation*.

CATIA V4 Templates

CATIA V4 templates are so-called predefined empty CATIA V4 models. They are useful if you want to use CATIA V4 models with the same settings and predefined views, such as project environment and title block settings for drawings. CMI prevents you from creating a new CATIA V4 model object without a CATIA V4 template object.

Templates are always generated by registering file item objects as CATIA Templates. This work is generally done by your administrator.

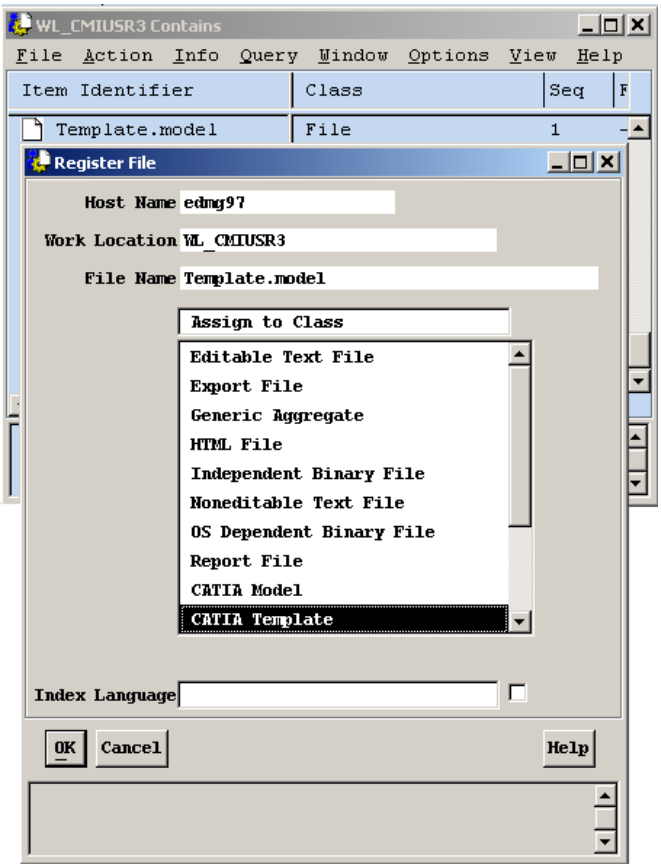


Figure 24: Registering an existing CATIA V4 model as CATIA V4 Template

You can copy an existing CATIA V4 model to your Template location and register it as a CATIA V4 Template object as shown above.

The registered CATIA Templates may then be used to create new CATIA model objects.

Creating a CATIA V4 Model

From the Menu Bar in *Home Collection* select:

Create > **CATIA Files** > **CATIA V4** Model

A *dialog window* as the one below appears.

Fill in the *field* 'File Name'

Fill in the *field* 'Work Location'.

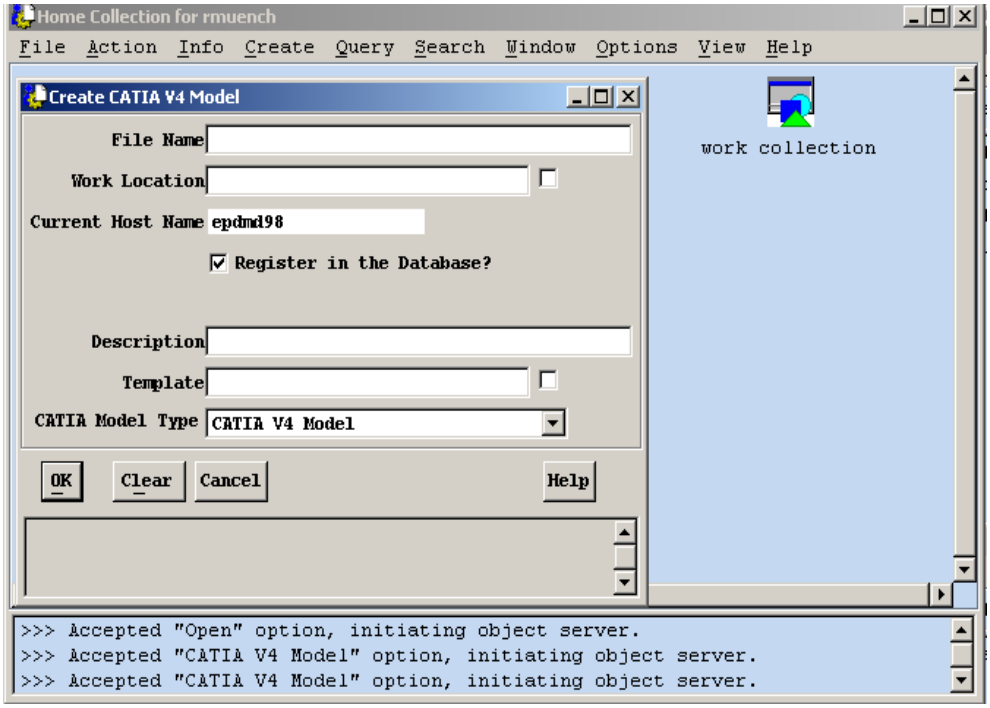


Figure 25: Dialog 'Create CATIA V4 Model'

Push the square button beside the *fields* 'Work Location' or 'Template' to choose from a list of possible values. If there is only one possible value, pushing the button will fill in the *field* automatically.

Fill in the *field* 'Template'. CATIA templates are CATIA models, predefined by the *administrator*, used as a basis to create other CATIA models.

If the 'Description' *field* has not been filled the Warning 'Description of CATIA-Model is missing!' appears on the status line.

It is useful and recommended to fill in the 'Description' *field*, because this value will appear in the Window menu of the Motif CATIA User Interface after the model has been loaded via the CMI interface.
Further on the dialog for i.e. METAPHSE>UPDATE>ALL menu option will not list CATIA models without this description. You will see '-' instead of the names.

Press Ok.

It is possible to open a new *browser* window containing the recently created 'CATIA Model' *object* by selecting it and choosing: **Window > Browse Selected Items.**

Creating a CATIA V5 CATPart / CATDrawing

From the Menu Bar in *Home Collection* select:

Create > CATIA Files > CATIA V5 Part File / CATIA V5 Drawing File

A *dialog window* as the one below appears.

Fill in the *field* 'File Name'

Fill in the *field* 'Work Location'.

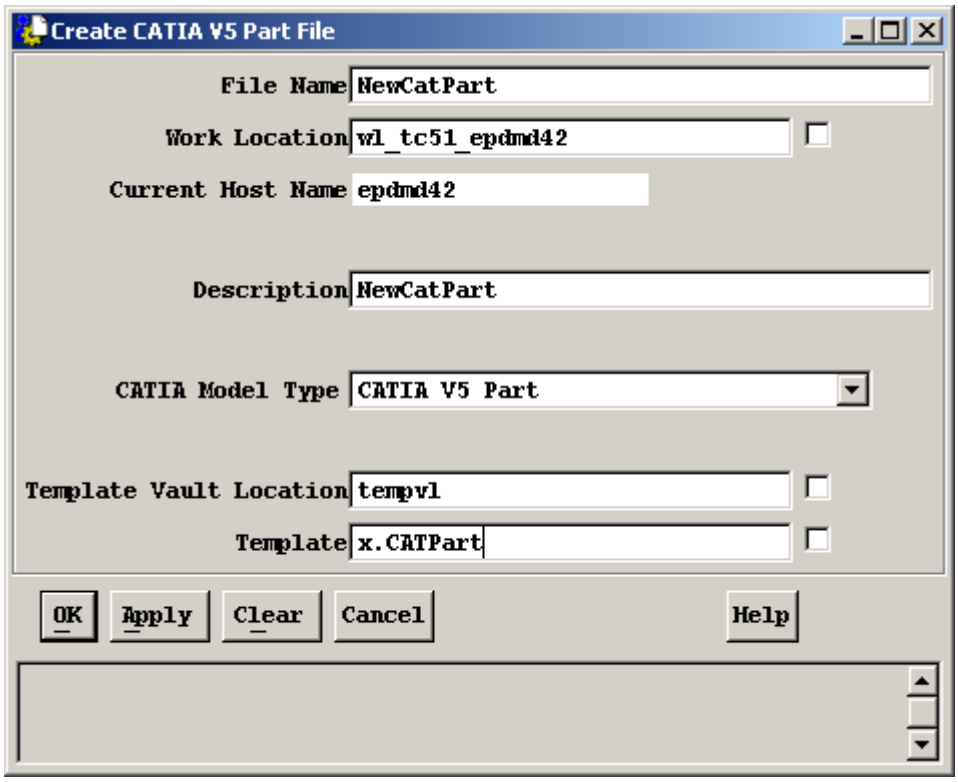


Figure 26: Dialog 'Create CATIA V5 Part File'

Push the square button beside the *fields* 'Work Location' or 'Template Vault Location' or 'Template' to choose from a list of possible values. If there is only one possible value, pushing the button will fill in the *field* automatically.

Fill in the *field* 'Template Vault Location' and 'Template'. CATIA V5 templates are CATIA V5 files, predefined by the *administrator* in specific Vault Locations, used as a basis to create other CATIA V5 files. *See also CMI Installation Manual*

If the 'Description' *field* has not been filled in, a warning appears in the status line.

It is useful and recommended to fill in the 'Description' *field*, because this value will be used as the part number property in Catia V5

Press Ok.

Creating a CATIA V4 Model from a CATIA V4 Template

From the Menu Bar in 'Home Collection', select:

Query > CATIA Files CATIA V4 Template

Fill in the *fields* with data to refine the search

Press Ok.

Select the CATIA Template you want to use

Use the right mouse button to get the context menu and select ‘Create CATIA model’

Fill in the fields as described under ‘Creating a CATIA model’

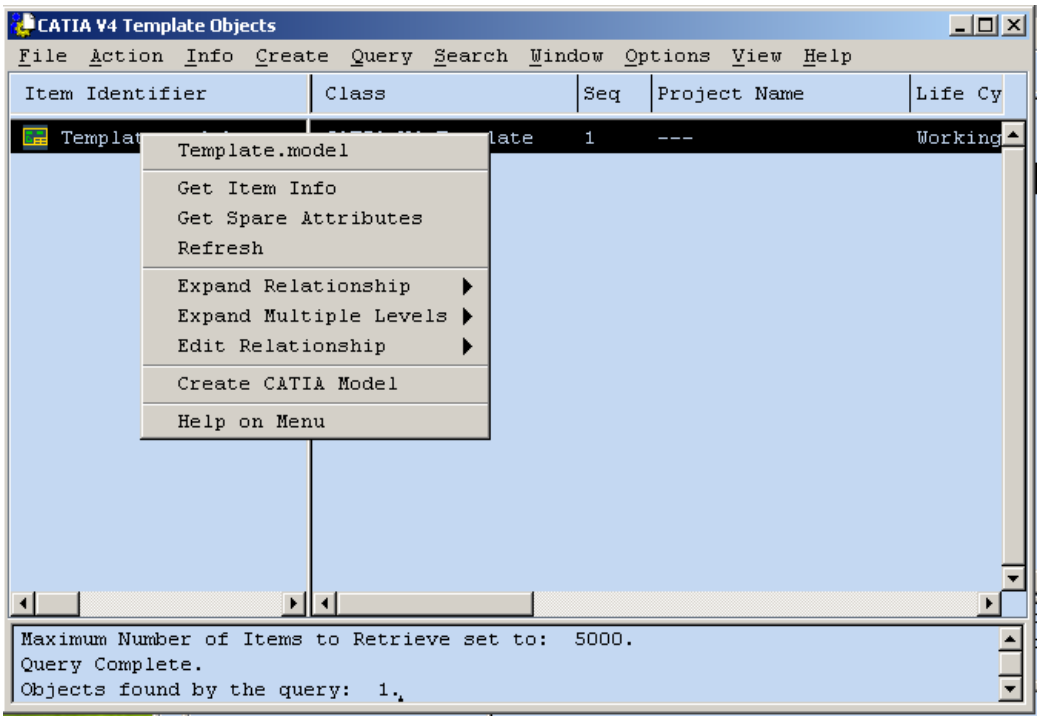


Figure 27: Teamcenter Browser Window ‘CATIA V4 Template Objects’ and popup

Register Existing CATIA V4 Models

Another way to register CATIA models is to copy your existing CATIA model files (with the suffix <.model>) to your work location. If you open this work location in Teamcenter, the CATIA model files will then be identified as ‘File’ with an empty icon.

Now you can select the CATIA models and register them as ‘CATIA-Model’.

Searching for CATIA V4 Models

In a *Teamcenter Browser* window, select:

Query > CATIA Files CATIA V4 Template

Fill in the *fields* with data to refine the search.

Press Ok.

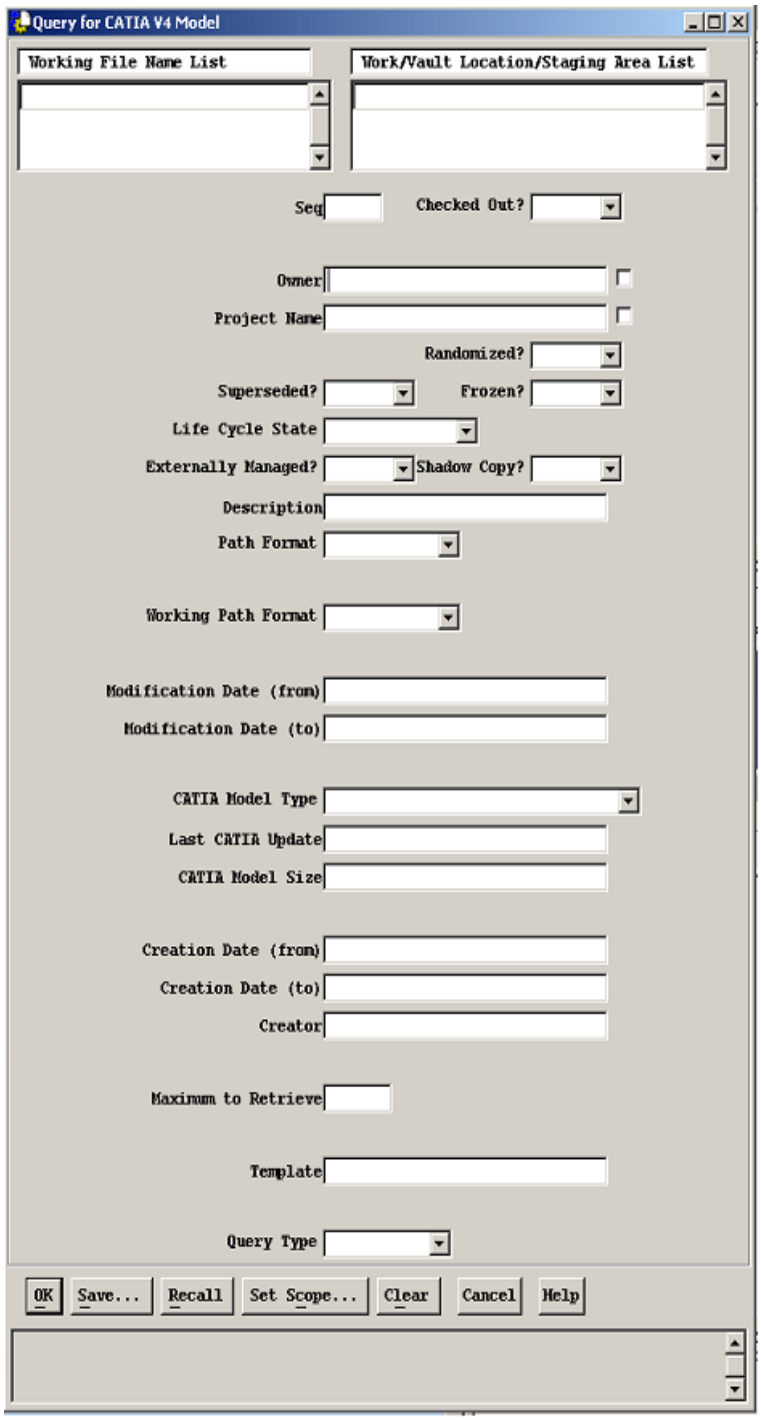


Figure 28: Searching a CATIA V4 Model in Teamcenter database.

Searching for CATIA V4 Templates

In a Teamcenter Enterprise *Browser* window, select:

Query > **CATIA V4 Template**

Fill in the *fields* with data to refine the search.

Press Ok.

Searching for Saved CMI Sessions

In a Teamcenter Enterprise *Browser* window, select:

Query > **Saved CATIA Session**

Fill in the *fields* with data to refine the search.

Press Ok.

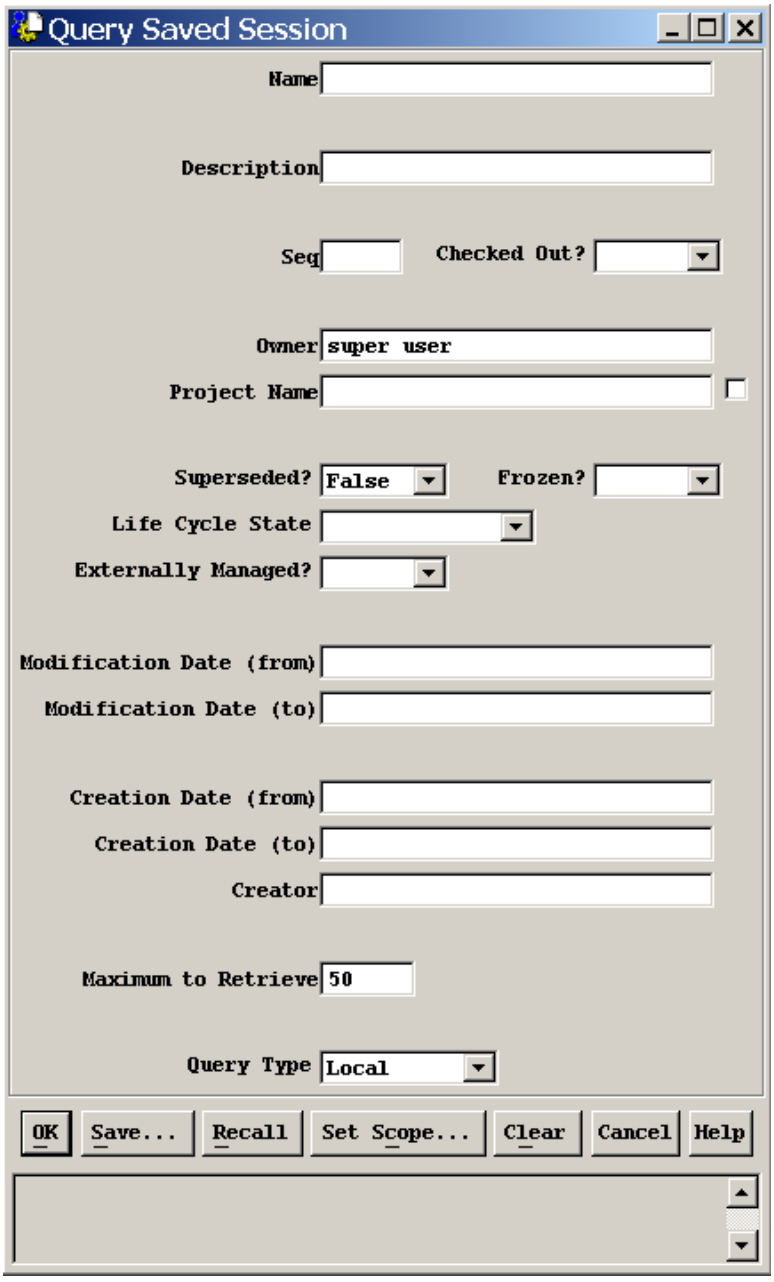


Figure 29: Query Saved Session.

Restore a saved CMI Session

To restore a saved CMI Session simply drag and drop the session icon into the CATIA Workbench. The contents of the Workbench will automatically be restored except for items that are no longer valid, e.g. those parts that have been revised or removed from the structure. In these cases an asterisk (*) is shown at the parent item, to indicate that you might want to re-expand this item

You can see the last sequence of the models in saved CMI Session when the button

Option→Change Preferences→CMI Preferences

Set “Show last model sequence in saved session” = TRUE.

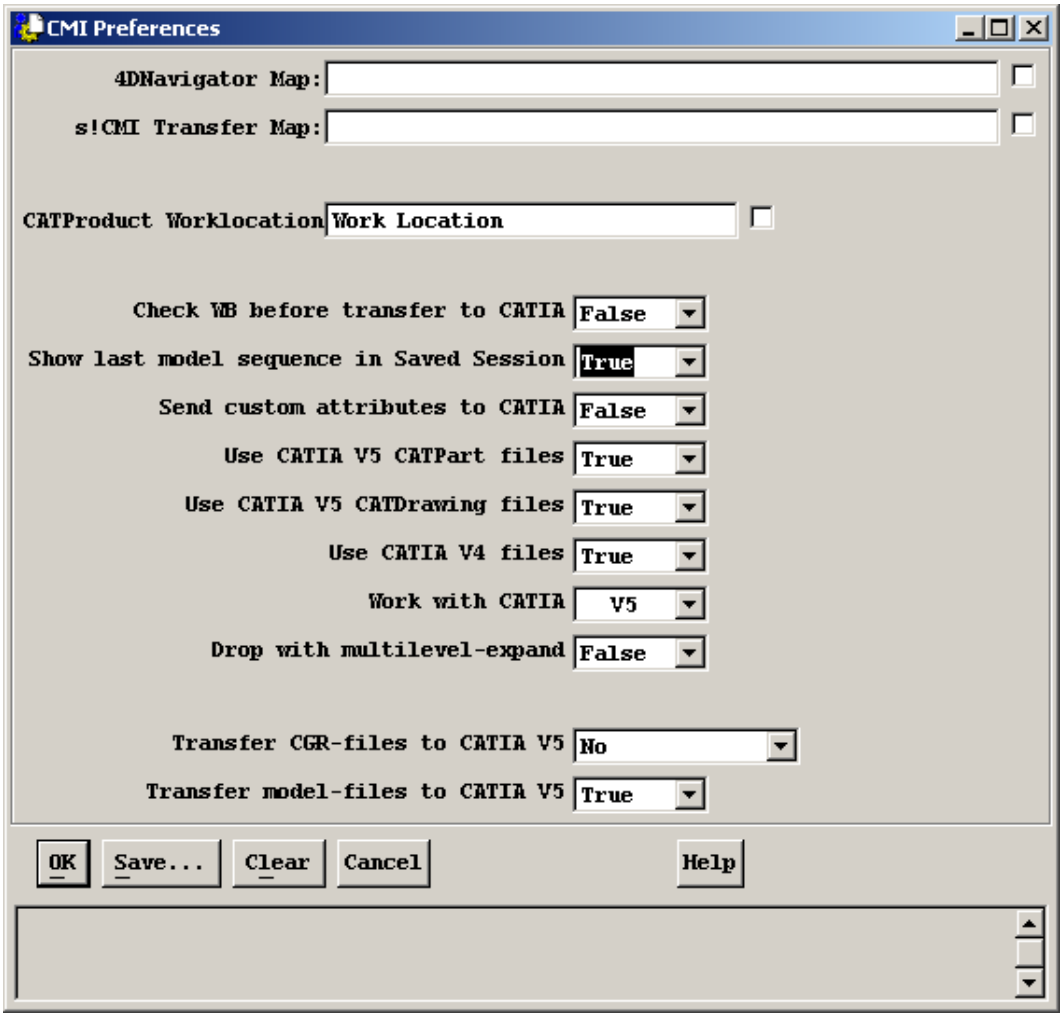


Figure 30: CMI Preferences

Loading Several Saved Sessions in CMI-Workbench.

You can drag/drop additional saved CMI–Sessions into the CMI-Workbench.
See **Figure 28**.

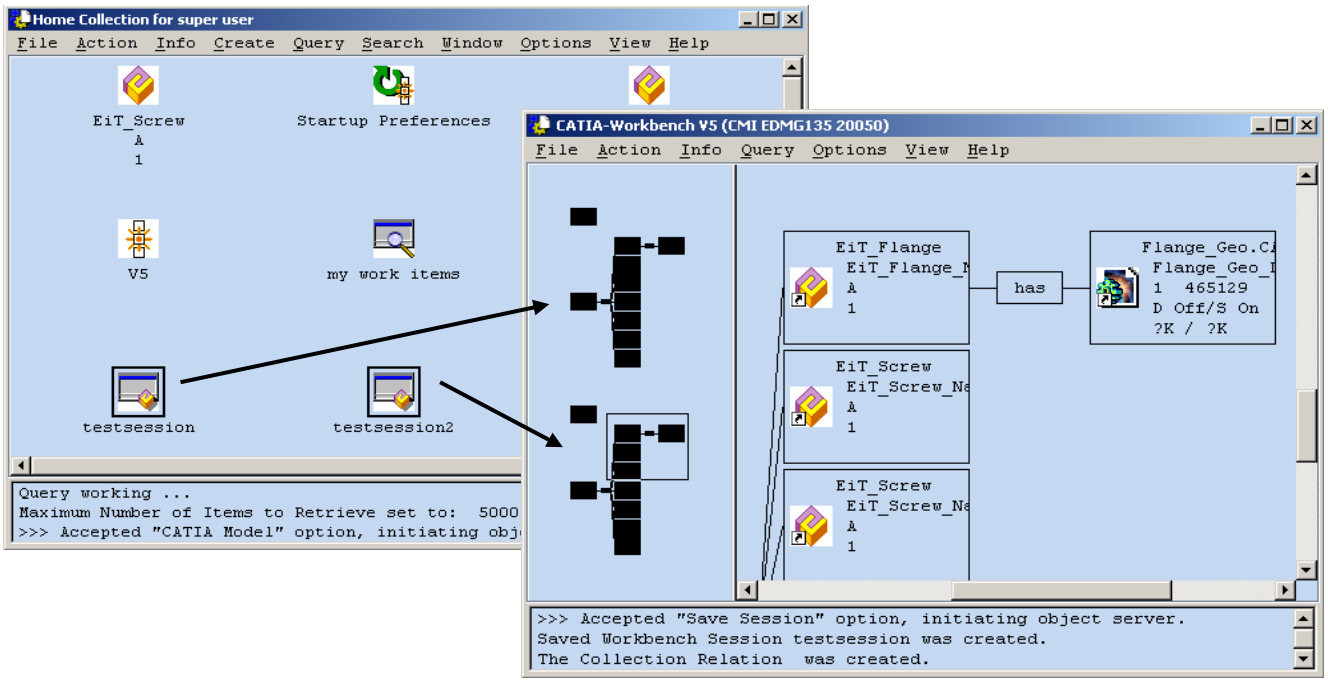


Figure 31 Several Saved Sessions in CMI-Workbench

Make sure that the *Configuration Context* and *Database Scope* are set correctly so that all items can be found.

Multilevel Auto Expand

Due to CMIs *Multilevel Auto Expand* you can expand parts selectively under an assembly in the CATIA Workbench. To expand selectively, first drop the top level Assembly onto the Workbench without expanding it. Now drop any subassembly onto the assembly already in the Workbench. CMI will expand all the way to all occurrences of this subassembly, without expanding unnecessary product structure. The function allows to load dedicated parts in context to each other, without unnecessary clutter.

You may drop several parts at once onto a part in the CATIA Workbench.

Drop with multilevel expand

The preference *Drop with multilevel expand* will automatically expand the assembly structure if you drop an Assembly onto the Catia Workbench.

In Catia-Workbench: Options>Change Preferences>CMI-Preferences

CHAPTER 5

Working with CATIA V4 models

This chapter describes some typical tasks such as loading documents stored in Teamcenter Enterprise to CATIA, updating a model, saving it under another name and creating *plot files*.

All CATIA model related actions can also be used for the CATIA Template objects (i.e. you can load a CATIA Template in CATIA and update the modified model if your access permissions allow this action.).

Reading a model from *Teamcenter*

To read a CATIA model stored in the Teamcenter Enterprise PDM System:

Start the CATIA Workbench as described in chapter 3.

The CATIA Workbench has to be started for CATIA relevant functions (i.e. read, update etc.). Once the CATIA Workbench is started, you can keep this window open during your Teamcenter Enterprise session.

In CATIA: **METAPHSE** > **READ**

Any operation in the CATIA - Teamcenter Enterprise function has to be confirmed by the permanent key YES in order to become effective in Teamcenter Enterprise.

In this manual all the CATIA functions selections shall be immediately followed by this confirmation, unless it is specified otherwise.

CATIA goes to the background. The *CATIA Workbench* comes to the foreground.

You can now retrieve either a document that contains one or more CATIA models or a single CATIA model.

Double-click the CATIA model or Drag&Drop it in the CATIA Workbench and select '**ACTION > To CATIA**'. The model is automatically loaded in CATIA.

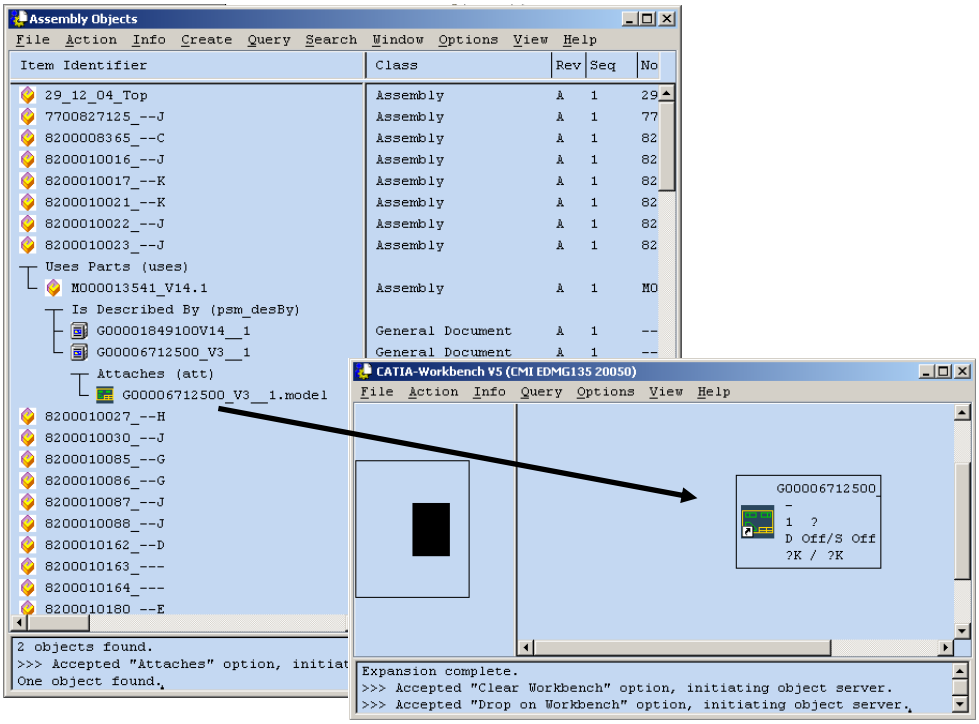


Figure 32: Drag & Drop a single CATIA Model into the CATIA Workbench.

You can also double click a document that contains one or more CATIA models or Drag&Drop it in the CATIA Workbench. Now you can expand / de-expand the models related to the document.

If Teamcenter Enterprise (the OMF server) and CATIA run on the same host, you can directly double click a model in Teamcenter to display it in CATIA. It is not necessary to initialize the action from within CATIA first.

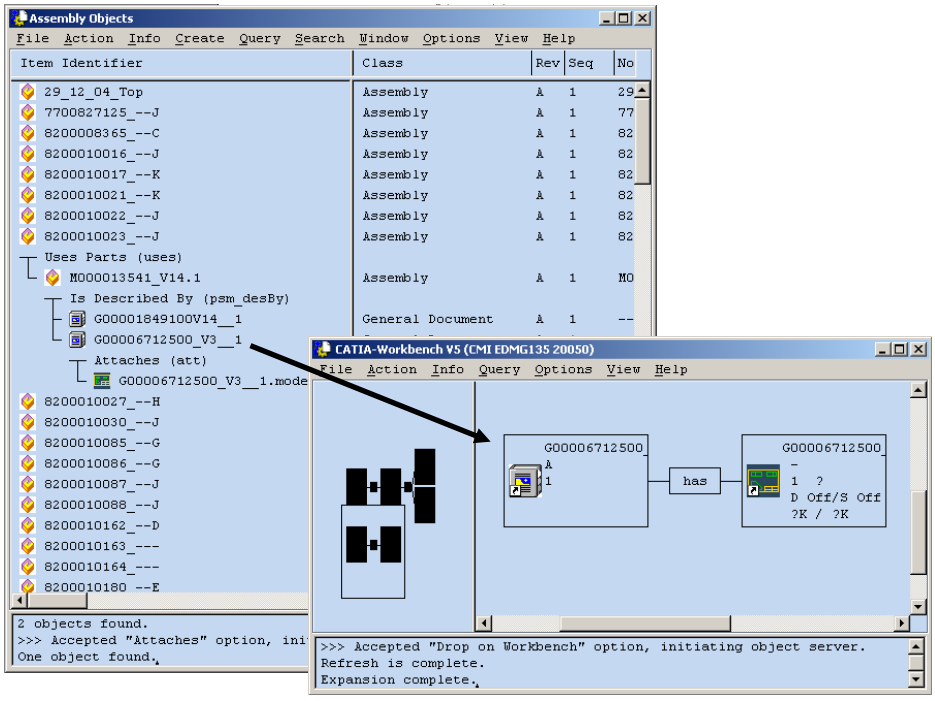


Figure 33: Drag&Drop a document into the CATIA Workbench

If the selected model is in a *vault*, it will be loaded in '**read only'** mode.

You can drop more than one CATIA model or CATIA document to the *CATIA Workbench*.

Updating a model

After having read the document from the Teamcenter Enterprise database, the designer might carry out his activities in CATIA and might probably want to save the changes. To update the model, he simply chooses:

In CATIA: **METAPHSE > UPDATE > ACTIVE**

It is not possible to update models that are already in a *vault*. To update these models, they must first be *checked out* to a Work Location.

Saving a model under another name

Create a Teamcenter Enterprise CATIA Model object with the desired name (see *Chapter 5* of this manual, section *Creating a CATIA Model*)

Load or create your CATIA Model in the CATIA session.

In CATIA: **METAPHSE > DOCUMENT > SAVE AS**
CATIA goes to the background.

Double click the created Teamcenter Enterprise model object or drag-and-drop it into the *CATIA Workbench*. The model that was in CATIA will overwrite the created document.

It is also possible to use **METAPHSE > DOCUMENT > SAVE AS** for overwriting old models. Thus, be sure that the model to be overwritten can be deleted.

It is not possible to overwrite models that are in a *vault*.

Creating a new model

Load or create your CATIA Model in the CATIA session.

In CATIA: **METAPHSE** > **DOCUMENT** > **CREATE**
CATIA goes to the background.

A Teamcenter Enterprise 'Create CATIA-Model' dialog window appears (Figure 26)

Enter a model name (the suffix <.model> will be created automatically) and a description and select the work location where the model should be saved.

Press Ok.

If you press 'Apply', the window will not close after creating a new model, and you can enter a new model name and generate another CATIA-model with the same content.

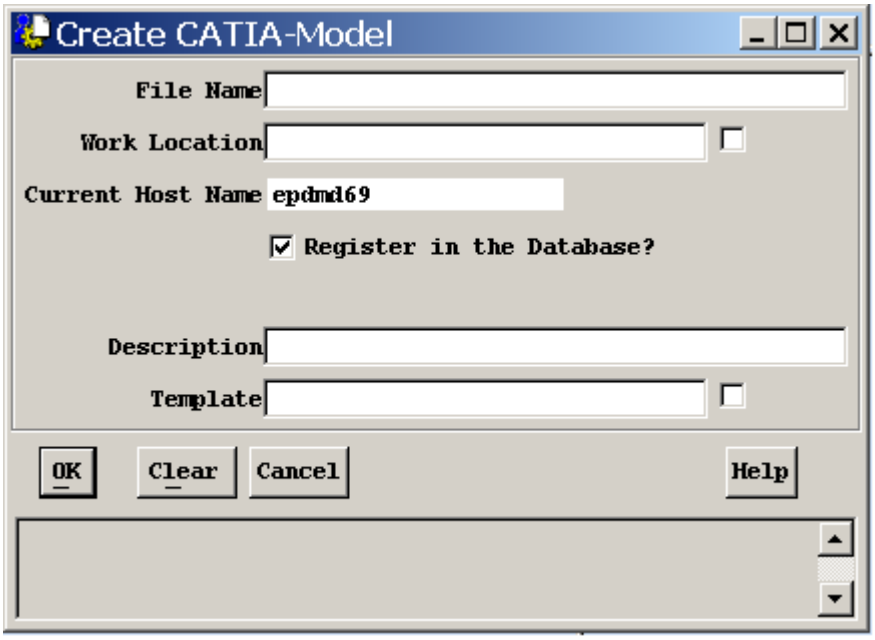


Figure 34: Dialog 'Create CATIA Model' opened from CATIA

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 function 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 dittos 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 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*:

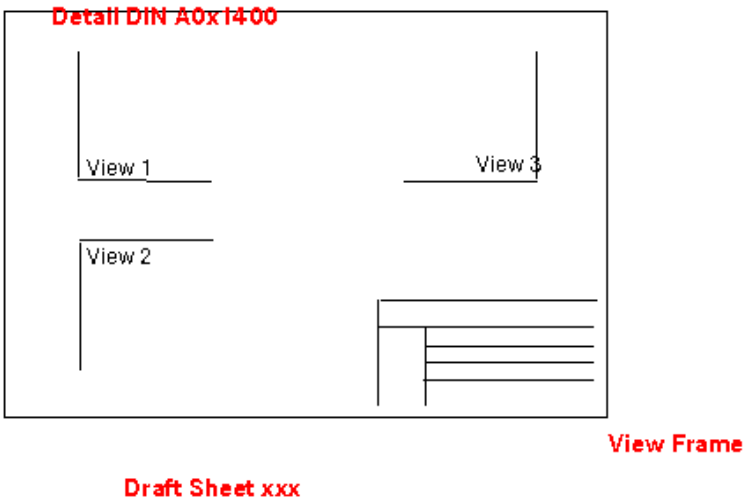


Figure 35: Naming conventions for drawing sheets

Example for a drawing sheet

In the following we describe a sample, which fits to the conventions that we need to define in the file appdefault.obj. This file must be located in a path that can be found in $PATH. You can find a sample for this file in the CATIA part of your CMI installation: .../catedm/data/appdefault.obj.

In this example (in appdefault.obj following the keyword “@frame-option”) we will define a

Draft named “BLATT” (in appdefault.obj the value following the keyword “indicatordraft”), a View named ”FRAME” (in appdefault.obj the value following the keyword “indicatorview”) and Details like “DIN A0” and ”DIN A1” (in appdefault.obj one of the values following the keyword “frames”) a.s.o. Each Detail represents a possible title-block which is to contain values out of the PDM-System.

Catedm/data/appdefault.obj

```
@general-options
{connectmethod} xtitle CMI
{exchangemap} /tmp
{exchange_dlname} CATIA.$USER.MP01.MAP
{logfile} stdout
{modelnametype} 1
{modrenallowed} 1
{geoposallowed} 0
{edbapphost} $HOST
{debug} 0
{project} UNKNOWN
{catiaversion} 422
{tmpaxis} 1
{tmpaxis-dlname.example} CATIA.STANDARD.MP01
{tmpaxis-name.example} CATIA_START_MODEL
{wpmodus} 0
{connectx-script} dshgui_find.sh
{normpart-support} 0
{assembly-symmetry} 1
{modellinks} 1
{license-startscript} licman12
{autorefresh} 0
{lld-autostart} 1
{load-warning} 1
{merge-support} 1
{remove-models} 1
{newupd} 1
{replace-model} 1
{cmi-custom-update} 3
{bbox-support} 0
{bbox-layer-list.example1} 0 1 2 3 4 5 6 7
{bbox-layer-list.example2} -1

@frame-option
{redraw-default}  yes
{indicatordraft}  BLATT
{indicatorview}   FRAME
{standard-format} ISO
{frames} {DIN A0} {DIN A1} {DIN A2} {DIN A3} {DIN A4}
{definition-script} dshdrawingframe.sh

@plot-option
{create-plotfile} yes
{plotformat} IBM-GL
{sizeX} 1189.0 841.0 594.0 420.0 210.0
{sizeY} 841.0 594.0 420.0 297.0 297.0
{orientation} V
{plottracefile} /tmp/$USER.plottrace
{plotinfile} /tmp/$USER.plotin
{plotconffile} $CATEDM_SCRIPTDIR/plotconf

@helper-object
{helper} xpdf -z 2 -g 890x700 -remote xpdf -raise -title "CMI User Manual"
{alias} /../../metaphse_4.x.x/pdfdoc/userman.pdf
```

The values for sizeX and sizeY in appdefault.obj following the keyword “plot-option” must be written in the order of above Detail-names. Thus each value pair corresponds to one of the title blocks (defined by its detail-name) appearing at the same sequential position after the already mentioned keyword “frames”. In our example the title block corresponding to the detail “DIN A0” has coordinate values x=1189.0 and y= 841.0, the title block corresponding to the detail “DIN A1” has coordinate values x=841.0 and y=594.0 a.s.o. Those coordinate values refer to the axis-system of the CATIA-view (the view in our example was named “FRAME”) and define the size of such a titleblock.

Now we create a CATIA model with the CATIA Teamcenter Interface and CATIA that fits to the previously explained naming conventions thus allowing to automatically update titleblock contents with their corresponding values out of the PDM-System (see appdefault.obj):

Load a CATIA model which should be used as root model
METAPHSE > READ

Double-click the CATIA model, or Drag&Drop it in the CATIA Workbench and select **ACTION > To CATIA**. The model will be loaded in CATIA.

Split CATIA screen into ‘DRAFT’ and ‘3D’ with
IMAGE > SCREEN > DEFINE

Create a ‘Draft’ with **DRAFT > CREATE**
SP (SPace (3D)) > DR (Drawing): Drawing mode has to be activated !!!
(has to be done in the ROOT model, name e.g. ‘Blatt’ !!!)

Create geometry (e.g. a ‘PLANE’)
DR> SP: 3D Space mode has to be activated !!!

Create a 2D point in your new created Draft **POINT> COORD**
SP> DR: Drawing mode has to be activated !!!

Create ‘View’ with
AUXVIEW> CREATE functionality.

SP> DR: Drawing mode has to be activated !!!
Describes the projection to the plane created before.

SELECT LINE/ PLANE (created in point 5)
SELECT POINT (created in point 6)
KEY VIEW ID (e.g. ‘FRAME’)

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 with **METAPHSE > UPDATE > ACTIVE**

How to define the contents to be shown in a title block?

Beneath the keyword “@frame-option” in appdefault.obj the keyword “definition-script” indicates the name of the script which defines the PDM-contents of the title-blocks (“dshdrawingframe.sh” in the example provided on delivery of CMI).

Catedm/bin/dshdrawingframe.sh

```
#!/bin/sh
# ------------------------------------------------------------------------
----
# this is an example shell-script
#
# usage: drawingframe [frame-name]
#
# It creates a set of entries which are used e.g. from CATIA
#
# The output must have a certain format
# {string {@text}} {xpos 0.0} {ypos 0.1} {font font} {orientation 0} {color
0}
# {size 5.0} {width 3.5} {slant 0.0} {space 1.5} {linetype 1} {thickness
0.35}
# {justification 9}
#
# description  :
#
# string       : mandatory
#                free text e.g. "this is a text"
# xpos, ypos   : mandatory
#                text position; real values e.g 10.0
#                belongs to view axis system
#                default: xpos = 0.0, ypos = 0.0
# font         : graphical representation e.g. CHR_DSFT
#                ATTENTION: some other attributes may depend on this
attribute
#                default: like defined in standard
# orientation  : 0 -> horizontal, 1 -> vertical
#                default: 0
# color        : = 0 -> no
#                = 1 -> white
#                = 2 -> red
#                = 3 -> green
#                = 4 -> blue
#                = 5 -> yellow
#                default: 1
#
# width        : mandatory
#                real value e.g. 3.0
#                default: 3.5
# size         : height of text, real value e.g. 5.0
#                if size is set to 0.0 the text height is linked to the
width
#                default: 5.0
# slant        : degrees, real value, must be between 0.0 and 45.0
#                default: 0.0
# space        : space between text characters, real value
#                default: 1.0
# linetype     : = 1 -> no scoring
#              : = 2 -> underscoring
#              : = 3 -> scoring
#              : = 4 -> overscoring
#                default: 1
# thickness    : real value, e.g. 3.0
#                default: 1.0
# justification: 0-9, text justification
#                = 0 -> default justification is middle left (2) else
#                [1][4][7]
#                [2][5][8]
#                [3][6][9]
#
# exit values must be
# 0  everything was fine
# 1  failed
# 2  unknown frame-name
# ------------------------------------------------------------------------
----
```

```
if [ $# -le 0 ] ; then
  exit 1
fi

TOKEN=$1

# echo "TOKEN=$TOKEN"

# case $TOKEN in

#   "DIN A0")
     echo "{string {Created:  @Date}}             {xpos 70} {ypos 15} {font
CHR_DSFT} {color 1} {orientation 0} {size 8.0} {width 0.0} {space 0.1}
{slant 0.0} {linetype 2} {thickness 4.0} {justification 1}"
     echo "{string {Modified: @ModificationDate}} {xpos 60} {ypos 15} {font
CHR_DSIS} {color 2} {orientation 0} {size 8.0} {width 0.0} {space 0.5}
{slant 0.0} {linetype 1} {thickness 5.0} {justification 2}"
     echo "{string {Name: @Name}}                 {xpos 40} {ypos 15} {font
EUDT_VCF} {color 3} {orientation 0} {size 2.0} {width 0.0} {space 0.5}
{slant 0.0} {linetype 1} {thickness 1.0} {justification 3}"
     echo "{string {User: $USER}}                 {xpos 20} {ypos 40} {font
CHR_DSIS} {color 4} {orientation 0} {size 2.0} {width 0.0} {space 0.5}
{slant 0.0} {linetype 1} {thickness 5.0} {justification 2}"
#     ;;

#   *)
#     exit 2
# esac

exit 0
```

In this shell script *drawingframe* (to be found in directory 'catedm/bin') the user is to define the entries (PDM-contents) for his previously defined views (like „**DIN A0**“, **“DIN A1**” a.s.o). In this example the view name (“DIN A0”) is put in comment “#”. Thus all subsequent entries would apply to just any titleblock defined by one of those views (“DIN A0”, “DIN A1” a.s.o.) not only to “DIN A0”.

Such a view might 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}
```

In general those entries represent constant values like “Company Name”, values delivered from the environment like “User” and values coming out of the PDM like ModificationDate.

Those entries representing values out of the PDM are preceeded by “@”. These values will be provided by means of the customizable *x3GetAttrsForCAD* CMI method which will be described soon.

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“.

How to get values from the PDM-System into one of the defined titleblocks (like DIN A0, DIN A1 a.s.o) automatically on reading the model via CMI?

The parameters that begin with "@" are those coming from PDM and are thus 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 all those titleblocks containing these entries with the most recent database information.

PdmItem: x3GetAttrsForCAD

```
DOCBEGIN
 ****************************************************************************
 *                                                                          *
 * Method/Function    x3GetAttrsForCAD                                      *
 *                                                                          *
 * Class              PdmItem                                               *
 *                                                                          *
 * Description        Handles the x3GetAttrsForCAD message for the PdmItem  *
 *                    Message type is an object message.                    *
 *                    This method delivers a set of attribute values found  *
 *                    for a set of demanded attribute types.                *
 *                    It is called immediately after a Model Read in case the *
 *                    Model read is structured in accordance to a drawing-  *
 *                    frame defined in the script "drawingframe".           *
 *                    The method  provided on deliverance of CMI serves as an *
 *                    example showing how "drawingframe" and                *
 *                    "x3GetAttrsForCAD" are supposed to cooperate. In      *
 *                    practical use this method will be overwritten in the  *
 *                    course of customization.                              *
 *                                                                          *
 * called within      x0WkBnch:x3GetAttrs                                   *
 *                                                                          *
 * Language           C                                                     *
 *                                                                          *
 * Input Parameters                                                         *
 *   Type          | Name           | Description                          *
 *   --------------+----------------+------------------------------------- *
 *   ObjectPtr     | this           | PdmItem object                       *
 *   SetOfStrings  | Attrs_demanded | Attributes demanded by CAD system    *
 *                                                                          *
 * Update Parameters                                                        *
 *   Type          | Name           | Description                          *
 *   --------------+----------------+------------------------------------- *
 *                                                                          *
 * Output Parameters                                                        *
 *   Type          | Name           | Description                          *
 *   --------------+----------------+------------------------------------- *
 *   SetOfStrings  | AttrVals_found | Values for demanded attributes       *
 *   integer       | mfail          | method success/ failure status       *
 *                 |                |   0 - (USC_OKAY) - successful        *
 *                 |                |   !0- unsuccessful                   *
 *                                                                          *
 * Return Value                                                             *
 *   --------------+----------------+------------------------------------- *
 *   status        | dstat          | OKAY                  - Success      *
 *                 |                | NOT_OKAY, BAD_PARAMETER- Failure     *
 ****************************************************************************
 DOCEND
 */
message PdmItem:x3GetAttrsForCAD (
   input : ObjectPtr    this           ::
   input : SetOfStrings Attrs_demanded ::
   output: SetOfStrings *AttrVals_found ::
   output: integer      *mfail) code
{
   /* ------------------------------------------------------------------------ */
   /* Declaration of variables                                                 */
   /* ------------------------------------------------------------------------ */
   string  attr_type= NULL;
   string  value    = NULL;

   integer numVal   = 0;
   integer i        = 0;

   /* ------------------------------------------------------------------------ */
   /* Begin                                                                    */
   /* ------------------------------------------------------------------------ */
   MPDBXFSTART ("PdmItem:x3GetAttrsForCAD")

   /* ------------------------------------------------------------------------ */
   /* Initialize return values                                                 */
   /* ------------------------------------------------------------------------ */
   *mfail= USC_OKAY;

   numVal= setSize (Attrs_demanded);
   if (!numVal) {
      goto CLEANUP;
```

```
   } /* end if (!numVal) .. */

   *AttrVals_found = setCreate(setSize(Attrs_demanded));

   /* ------------------------------------------------------------------------ */
   /* Get Attributes; before, inflate dynamic attributes                       */
   /* ------------------------------------------------------------------------ */
   dstat= InflateDisplayedName (this, mfail);
   CHECK ("InflateDisplayedName")

   for (i= 0; i< numVal; i++) {
      attr_type= setGetStr (Attrs_demanded, i);

      value="";

      if (STRCMP (attr_type, "Date")) {
         dstat= objGetAttribute (this, "CreationDate", &value);
         DSTAT ("objGetAttribute CreationDate")
      } /* end if (STRCMP (attr type, "Date")) .. */

      else if (STRCMP (attr_type, "ModificationDate")) {
         dstat= objGetAttribute (this, "LastUpdate", &value);
         DSTAT ("objGetAttribute LastUpdate")
      } /* end else if (STRCMP (attr_type, "ModificationDate")) .. */

      else if (STRCMP (attr_type, "Name")) {
         dstat= objGetAttribute (this, "TDisplayedName", &value);
         DSTAT ("objGetAttribute TDisplayedName")
      } /* end else if (STRCMP (attr_type, "Name")) .. */

      if (i<setSize(*AttrVals_found)) {
         setPutStr (*AttrVals_found, i, value);
      }
      else {
         setAddStr (*AttrVals_found, value);
      }

   } /* end for (i= 0;i< numVal; i++) .. */

CLEANUP:

   /* ------------------------------------------------------------------------ */
   /* Free allocated memory                                                    */
   /* ------------------------------------------------------------------------ */

EXIT:

   /* ------------------------------------------------------------------------ */
   /* End                                                                      */
   /* ------------------------------------------------------------------------ */
   MPDBXFEND
   if (dstat!= OKAY)
      dlow_return_trace (mod_name, dstat);
   return (dstat);
}
;
```

You will find the customizable method *x3GetAttrsForCAD* in custom/x3Custom.mth. This method is called within x0WkBnch:x3GetAttrs for Assemblies and CATIA-models. So you can overwrite it for your specific Assembly- or CATIA-Model-classes.

Creating a Plot File object

The *plot file object* is an *object* that contains the information to be filled in the draft of the model to which it is attached. To create a *plot file object*, perform the following steps:

In CATIA: select **METAPHSE** > **READ**

Retrieve or create a CATIA model, which is created with the conventions as described above (i.e. containing a title block).

Double-click the CATIA model, or Drag&Drop it in the CATIA Workbench and select **ACTION > To CATIA**. The model will be loaded in CATIA.

In CATIA, select **METAPHSE** > **PLOT**
The *plot file object* is created.

In CATIA: select **METAPHSE > UPDATE > ACTIVE**
The model will be updated.

The *plot file object* will be copied to Teamcenter Enterprise and *registered* as a *plot file*.

The *plot file object* will be related to the model *object* by the relation: 'Model has *plot files*'.

'Model has *plot files*' is a (1:1) relationship, what means that only one *plot file* can be attached to a CATIA model and that a *plot file* can be attached to only one CATIA model. Thus, the next time a *plot file* is created, the existing *object* in Teamcenter Enterprise will be overwritten.

CHAPTER 6

Working with Structures

This chapter explains the concepts underlying the work with structures and describes how to create assemblies and components, search for parts, load parts in CATIA and modify the structure position.

Understanding the Structure

A structure is made of one or more parts. Teamcenter Enterprise knows two kinds of parts,

Assemblies:

and Components:

An Assembly is a Part that can contain other Parts (i.e. Assemblies or Components). A Component cannot contain other Parts.

Please refer to the 'Figure 23: Predefined CATIA model structure.' on page 19 for more information about a default assembly data structure.

Creating a Part

Creating an assembly

In a Teamcenter Enterprise *browser* window select:
Create >**Product Structure Classes** > **Assembly**

Creating a component

In a Teamcenter Enterprise *browser* window select:
Create >**Product Structure Classes** > **Component**

Searching for Parts

Searching for Assemblies

In a Teamcenter Enterprise *browser* window select:
Query > **Product Structure Classes** > **Assembly**

Searching for Components

In a Teamcenter Enterprise *browser* window select:
Query > **Product Structure Classes** > **Component**

Creating a Structure

To create a structure in Teamcenter Enterprise, the parts of the Assembly need to have been created (see the section *Creating a Part* in this chapter).

A relation ‘uses part’ is generated by dragging and dropping the ‘used’ part onto the part that uses it.

For example: to create the relationship ‘Part 1 *uses part* Part 2’ just drag-and-drop Part 2 onto Part 1.

The relationships need not to be made in a specific order. For example, the two steps to create the structure of the figure below are interchangeable:

drag-and-drop Part 3 onto Part 2

drag-and-drop Part 2 onto Part 1

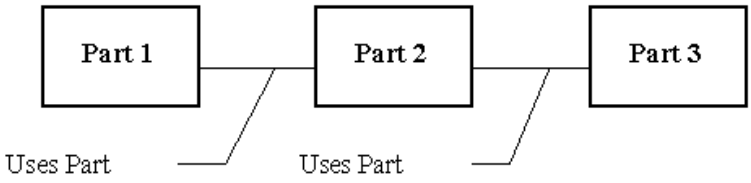


Figure 36: A simple assembly structure.

Please refer to the Figure 21: Predefined CATIA model structure on page 19 for more information about the default assembly data structure.

The figure below shows an example assembly in *tree view* where Document objects and model objects have also been created and related to the Part objects:

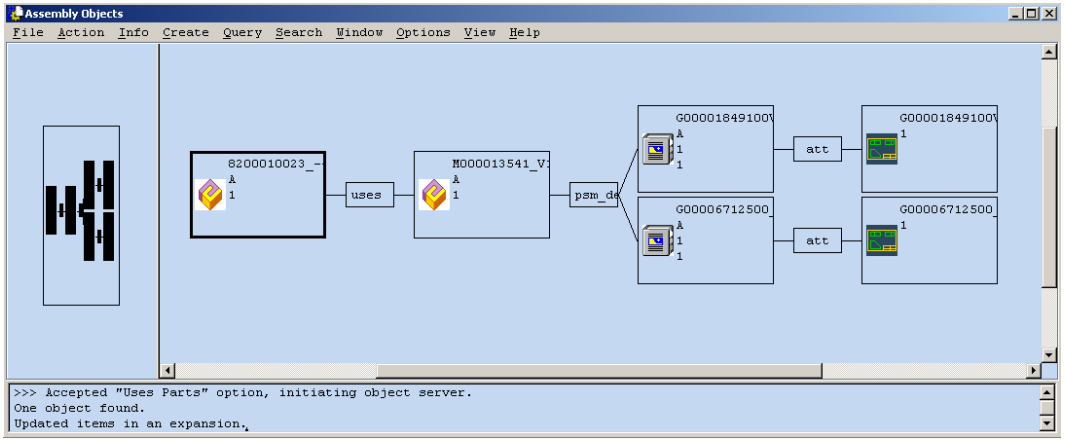


Figure 37: Example of an expanded Assembly in Teamcenter Enterprise browser.

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:

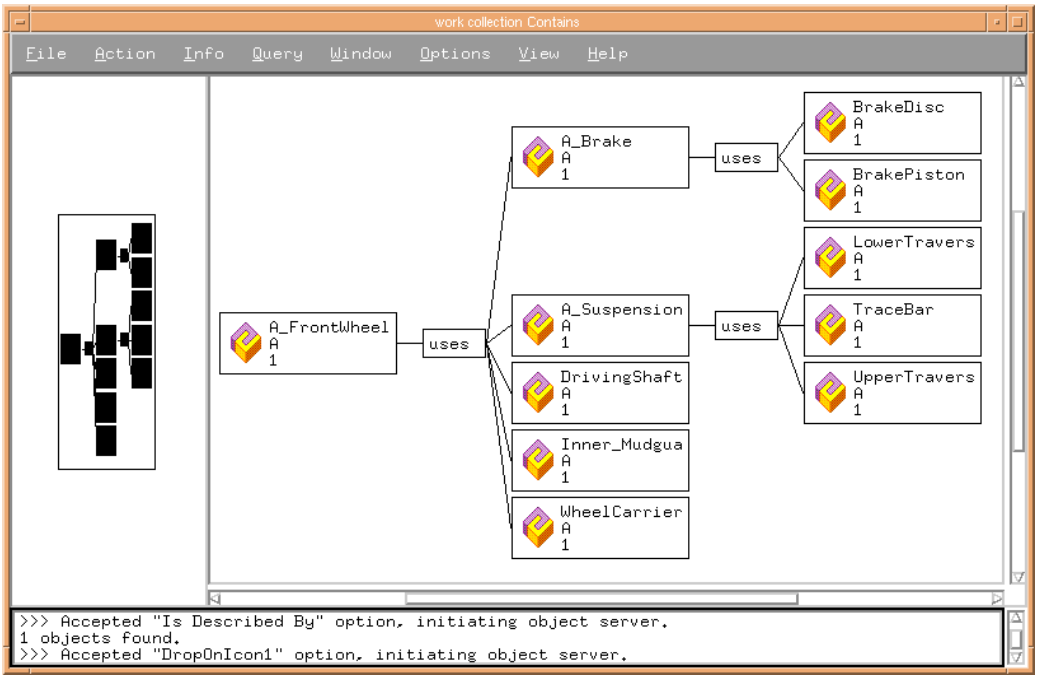


Figure 38: Example assembly structure in Teamcenter Enterprise browser.

Create the following Assembly objects:

A_FrontWheel
A_Brake
A_Suspension

DrivingShaft
Inner_Mudguard
WheelCarrier
BrakeDisc
BrakePiston
LowerTraverse
UpperTraverse
TraceBar

Create the relationships between the Assembly objects like you can see in the figure above.

Tip: After Drag Drop; Expand Relationship\Uses Parts

Create the following Document objects for the following Assembly objects:
DrivingShaft
Inner_Mudguard
WheelCarrier
BrakeDisc
BrakePiston
LowerTraverse
UpperTraverse
TraceBar

Create the relationships between the Assembly objects and the corresponding Document objects as listed above (i.e. Drag & Drop the Document ‘DrivingShaft’

onto the Assembly ‘DrivingShaft’)

Tip: After Drag Drop; Expand Relationship\Is Described By

Register the example native CATIA files included in the distribution (ask your administrator, if you do not have those files) as CATIA V4 Models.

Tip: The CATIA files have to be stored in the users Worklocation

Create the relationships between the Document objects and corresponding CATIA Models (i.e. Drag & Drop the CATIA Model ‘DrivingShaft’ onto the document ‘DrivingShaft’)

The so created product structure you can now put into the CMI Workbench and expand it (see figure below). Then you can send it to CATIA. See the following chapters.

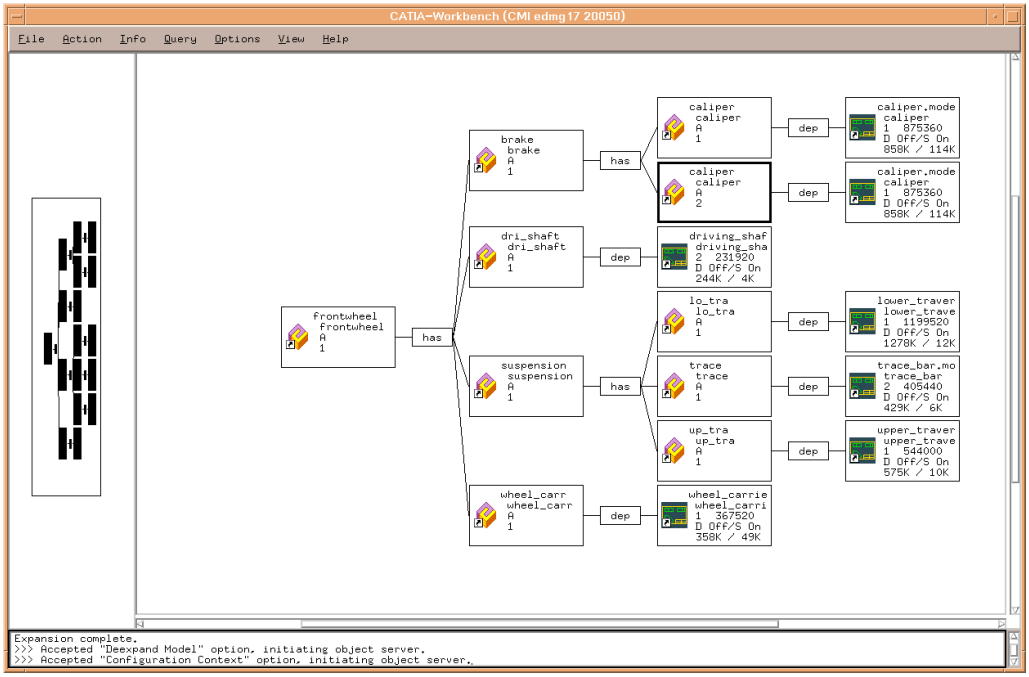


Figure 39: Example assembly structure within CMI CATIA Workbench.

Loading a Structure in CATIA

To load an assembly structure in CATIA:

Start the CATIA Workbench as described in chapter 3.

The CATIA Workbench has to be started for CATIA relevant functions (i.e. read, update etc.). Once the CATIA Workbench is started, you can leave this window open during your Teamcenter Enterprise session.

In CATIA: **METAPHSE** > **READ** (confirm with “yes”)
CATIA goes into the background.

The *CATIA Workbench* comes to the foreground.

Drag & Drop the Part that is the 'root' of the selected structure in the *CATIA Workbench* (or double-click it).

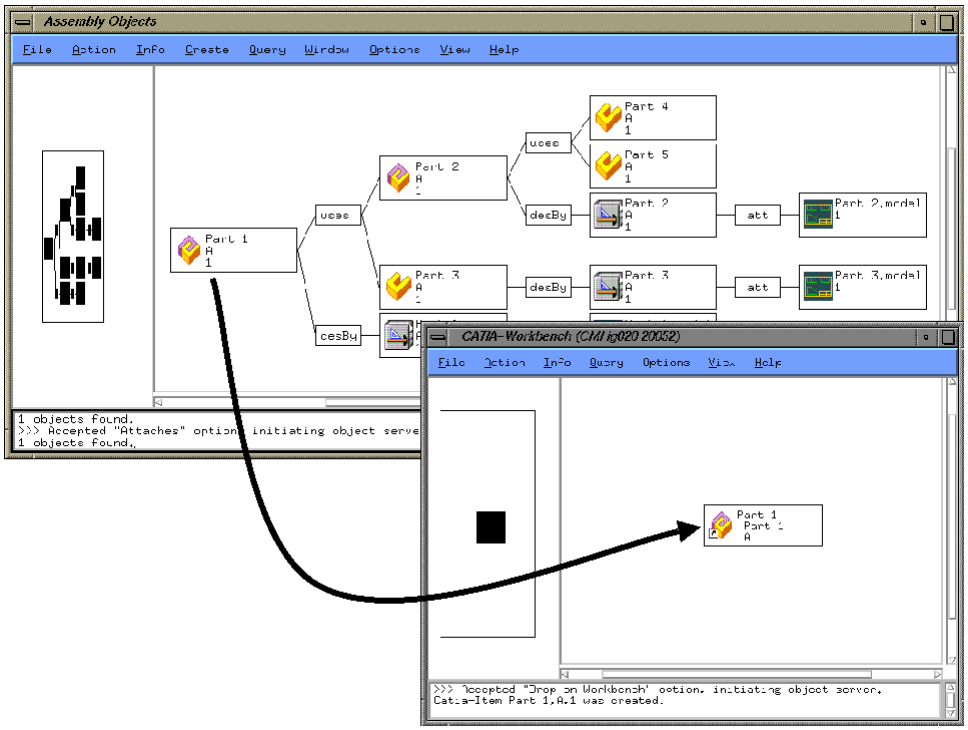


Figure 40: Drag & Drop an assembly into the CATIA Workbench.

Select 'Part 1' in the *CATIA Workbench*. Choose the '**Expand**' option in the *Pop-Up menu* of the part. The parts that are used by the selected assembly appear. You can expand one level or multiple levels (Caution: if you have created complex assembly structures, this action may need some time to expand all levels of the assembly).

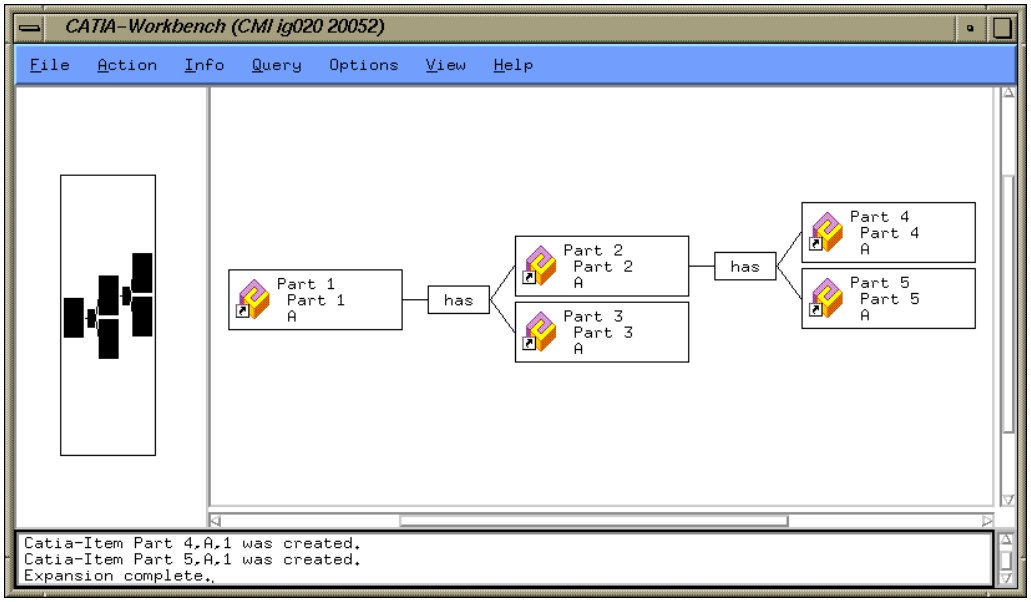


Figure 41: Expanded assembly structure in CATIA Workbench

It is also possible to choose the Expand Multiple Levels function with the '**View**'→'**Display Models**' option. In this case, the structure shown in the *CATIA Workbench* is the same as before, plus the models.

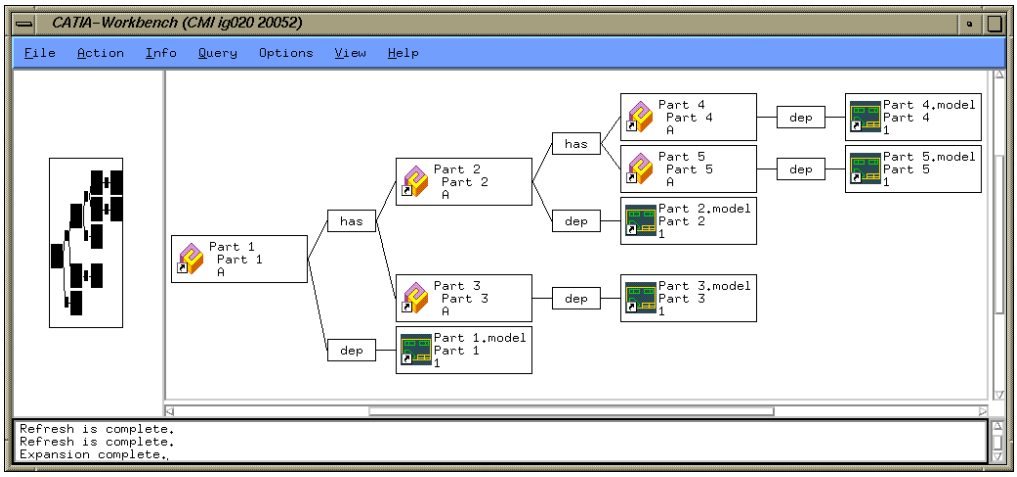


Figure 42: Expanded assembly structure with models in CATIA Workbench

You can now decide to either load the whole assembly, or to load the assembly with only a selection of sub-components/parts. To exclude a subpart from being loaded into CATIA, select the part and hide it with the **Deexpand** option in the popup menu

In the *CATIA Workbench*, select the option **Action** > **To CATIA** in the menu. The assembly is transferred to CATIA

After having transferred the data, the CATIA session appears with the desired assembly structure. You can modify the assembly and save it in Teamcenter.

You can drop more than one part or assembly to the *CATIA Workbench*. It is also possible to mix parts and assemblies with CATIA models or documents.

If you drop another part onto a part in the *CATIA Workbench* instead of onto the background, and the dropped part is a subassembly, it will be expanded.
You can use this to expand only selected subassemblies, to avoid expanding a complex structure only to de-expand most of it afterwards. For further information please refer to the section entitled "**Multilevel**".

Highlighting a Model 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 equivalent model in the CATIA session of a Part/model displayed within the *CATIA Workbench*.

For this purpose, it is possible to highlight CATIA models in CATIA using the currently selected Model in the *CATIA Workbench*.

Model highlighting is also available in CATIA V5 CMI.

In the *CATIA Workbench*, select the CATIA model you want to be highlighted in the CATIA session and choose the **Highlight in CATIA** option in the *Pop-Up menu*. CATIA comes to the foreground with the selected CATIA model highlighted.

Since this option is only available at CATIA models in the *CATIA Workbench*, you have to expand the assembly structure with the 'Show Models'-option before transferring the assembly to CATIA.

Updating a Structure

There are 3 ways to update a structure:

In CATIA: **METAPHSE** > **UPDATE** > **ACTIVE**

Updates the active model (geometry information)

In CATIA: **METAPHSE** > **UPDATE** > **GEOMETRY**

Updates the geometry information of all activated models.

In CATIA: **METAPHSE** > **UPDATE** > **POSITION**

Updates the position information of all changed positions.

There are two restrictions that shall be considered:
If a model related to a part of the structure is in a *vault*, this model cannot be modified (for example it is not possible to change its color). Despite of this it is possible to modify the relative position between this part and the others (the position data is stored in the part relation *object*).
If a part of the structure is in a *vault*, the model related to this part can be modified (if it is not *checked in*) but there is no way of changing the position of this part relative to its subparts.

Rereading a Structure

If you choose **METAPHSE > READ** in CATIA the modifications to models and positions that you currently have in CATIA will be lost. To send additional data to CATIA or to restore only part of your data (e.g. to undo changes) you can use **METAPHSE > REREAD**

There are 3 ways to reread a structure:

In CATIA: **METAPHSE > REREAD > GEOMETRY**

Reads the current content of the *CATIA Workbench.* Rereads all models but does not read positions that are already in CATIA.

Items that are no longer in the Workbench will be removed from CATIA. Items added to Workbench will be also added to the CATIA Session.

In CATIA: **METAPHSE > REREAD > POSITION**

Reads the current content of the *CATIA Workbench.* Rereads all positions but does not change models that are already in CATIA.

Items that are no longer in the Workbench will be removed from CATIA. Items added to Workbench will be added also to the CATIA Session.

In CATIA: **METAPHSE > REREAD > RESYNC**

Reads the current content of the *CATIA Workbench.* Reads only models that are not already loaded in CATIA, but does not change any existing positions and models. Items that are no longer in the Workbench will be removed from CATIA. Items added to Workbench will be also added to the CATIA Session.

If you want to add or remove few items from CATIA you can use **METAPHSE > REREAD > RESYNC**. This is faster than the other options as CATIA does not load items that are already loaded.

Working with marked Parts

To speed up your work with the *CATIA Workbench* it is possible to mark parts, assemblies or models. Marked items are shown with a blue icon.

It is possible to apply the following functions to the marked items:

In the *CATIA Workbench*: **Action > Marked Items > Deexpand Marked Items**
All marked items and sub-items are removed from the *CATIA Workbench.*

In the *CATIA Workbench*: **Action > Marked Items > Unmark All**
All items are set to the unmarked state.

In the *CATIA Workbench*: **Action > Marked Items > Marked Items to CATIA**
All marked items are sent to CATIA.

If Teamcenter (the OMF server) and CATIA run on the same host, you can use this command directly. Otherwise you have to initialize the communication from CATIA

with **METAPHSE > READ** or **METAPHSE > REREAD** first.

In the *CATIA Workbench*: **Action > Marked Items > Highlight Marked Items**
All marked items and sub-items are highlighted in CATIA.

Note that this function is only available if Teamcenter (the OMF server) and CATIA are running on the same host.

To Mark or Unmark Parts you can use the following actions:

In the *CATIA Workbench*: in the *Pop-Up menu* of parts and assemblies you can use the **Mark/Unmark Part** option.

Use the standard selection mechanisms (left click, left click and [Shift], left click and [Cntrl] to select several items, then choose **Action > Mark Selected Items**

In the *CATIA Workbench*: **Action > Marked Items > Unmark All**
All items are set to the unmarked state.

In the *CATIA Workbench*: **Query > CATIA-Item**

A Query-Dialog appears. Fill out the fields of the Query-Dialog to specify the parts and assemblies to find, as you are used to do in a standard Teamcenter Enterprise Query-Dialog.

Use the Clear-Button to empty all input fields. The Query will find and mark all parts and assemblies in the *CATIA Workbench*.

Mark Items functionality also supported with CATIA V5.

Set the active model

Predefine the model to be the ‘active model’ in the next CATIA Session in your *CATIA Workbench*.

Start the CATIA Workbench as described in chapter 3.

In CATIA: **METAPHSE** > **READ** (confirm with “yes”)

CATIA goes into the background. The *CATIA Workbench* is automatically emptied and comes into the foreground.

Drop one or more assemblies into your CATIA Workbench and expand the structure.

To see the models in the *CATIA Workbench* you have to activate ‘**View > Display Models**’ in your *CATIA Workbench*.

Use the right mouse button to get the context menu and select ‘Activate Model’. The model is now displayed red.

Notice:
It is only possible to activate models which are owned by you.
Only one model can be the active model.

Select '**Action > To CATIA**'. The content of the *CATIA Workbench* is loaded in CATIA and the model displayed red is set the active model.

If no model is activated in your CATIA Workbench, CATIA activates the first model in the CATIA model list.

Sending Parts to 4D-Navigator

It is possible to send the content of the CATIA Workbench to the 4D-Navigator.

In *CATIA Workbench*, select menu **Action** > **To 4D-Navigator**
The assembly is transferred to 4D-Navigator.

In the *CATIA Workbench*, it is also possible to select menu **Action** > **Marked Items > Marked To 4D-Navigator**. On using this function, only the marked items are transferred to 4D-Navigator.

Remember that the path to the 4D-Navigator exchange-map has to be set with:

In *CATIA Workbench*: **Options > Change Preferences > CMI Preferences**

DMU-Volume

For a volume based DMU-Search you must create DMU-Volumes for CMI.

In a Teamcenter Enterprise standard browser select **Create** > **DMU Volume**

Notice:

To evaluate the volume-search of Catia-models, bounding boxes must be generated for all Catia-models, and the administrator has to set up the DMU-facilities by configuration-variables. The generation of bounding boxes is specified with installation of Catia. To verify the bounding box values of a Catia model, select *Info > Get Item Info*

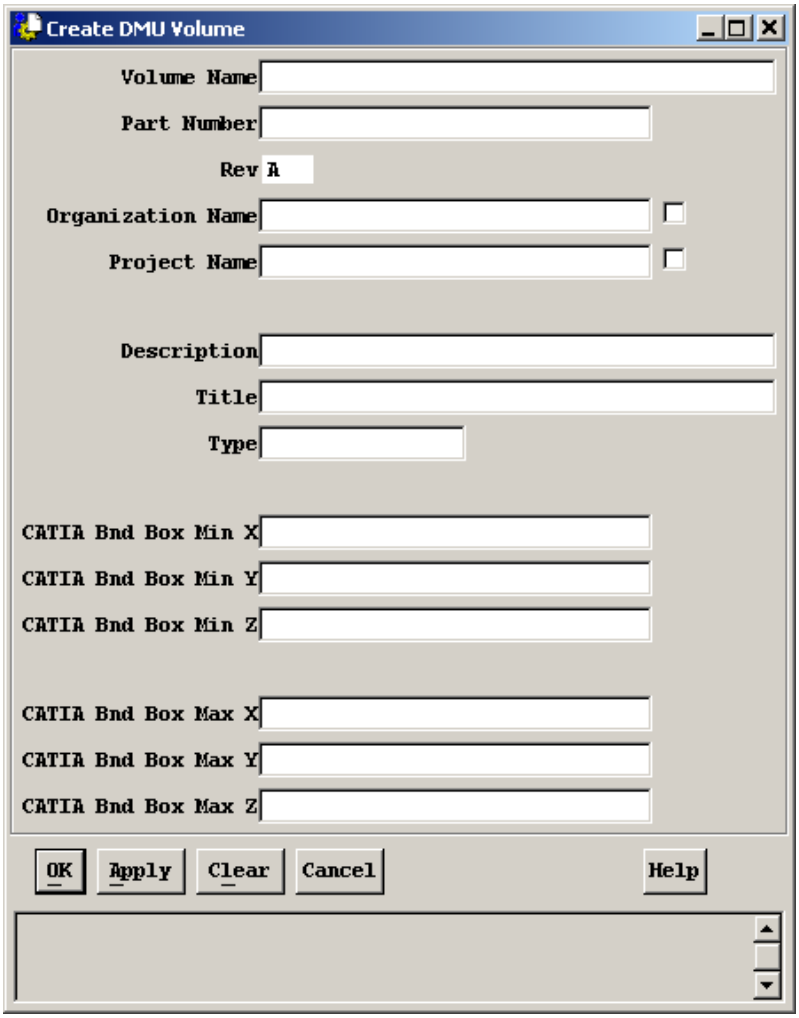


Figure 43: The Create DMU Volume dialog

A DMU-Volume is defined by two x/y/z coordinates like a bounding box. The volume relates to a Root-Part, that means the same volume for a Child-Part must differ from the Root-Part corresponding to the transformation-matrix.

Neighbourhood of models

Notice:

To evaluate the neigbourhood of Catia-models, bounding boxes must be generated for all Catia-models, and the administrator has to set up the DMU-facilities by configuration-variables. The generation of bounding boxes is specified with installation of Catia. To verify the bounding box values of a Catia model, select *Info > Get Item Info*

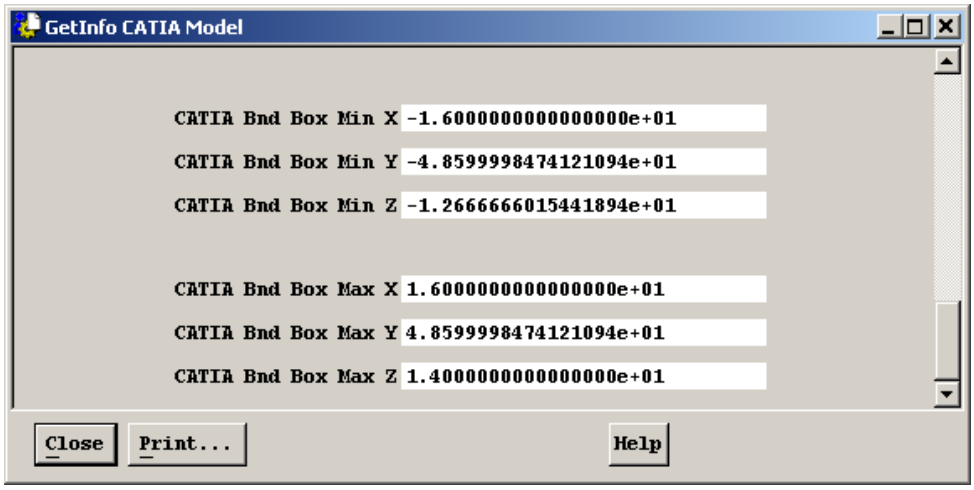


Figure 44: GetInfo CATIA Model dialog

You can display all “neighbour-models” to a model in the workbench.

In *CATIA Workbench*, select a root-assembly and select menu **Action** > **Create DMU-File.** This assembly will be expanded down to all leafs and a DMU-File is created with all absolute positions and the bounding-boxes of all models. This file is stored in a DMU-Vault.

Select the desired Catia model and select menu **Action** > **Show Neighbourhood.** The root-assembly is expanded down to the selected Catia model and to all “neighbour-models” by evaluating the DMU-file.

For several Catia models Select the desired Catia models and select menu **Query** > **DMU** > **Show Neighbours.** The root-assembly is expanded down to the selected Catia model and to all “neighbour-models” by evaluating the DMU-file.

Notice:

You can affect the computation of neighbourhood by increasing the DMU-Tolerance.

Increasing the DMU-Tolerance means increasing the bounding-box of the selected model.

In *CATIA Workbench*: **Options** > **Change Preferences** > **DMU Preferences**

Volume search for models

You can display all models of a predefined volume in the workbench.

In *CATIA Workbench* select menu **Query** > **DMU** > **Show models of DMU-Volume.** All models of a volume will be expanded from a Root Part down to the leaf-parts.

The menu item *Query* -> DMU -> “Show Models of DMU-Volume” presents a Dialog-window to ask for the Root part and for the DMU-Volume:

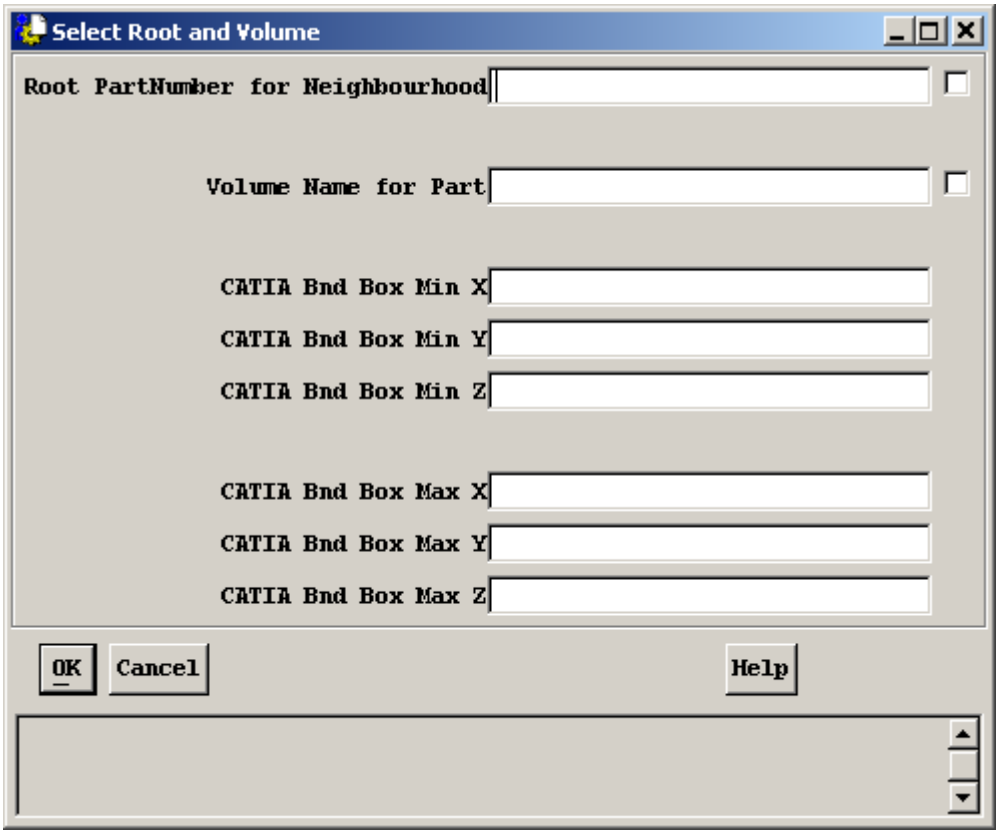


Figure 45: Select Root and Volume dialog

If you click the selection-box for the root Part, all Part Numbers and Revisions are shown for which a CMI-DMU-File exists. If the root Part is determined, the click of the Volume-Name-selection-box shows all Volumes of this specific root Part.

If there is no Volume of your choice for the root Part, you can perform a dynamic Volume-Search by specifying all volume-values in the dialog-window.

With the OK-button, the Volume search starts, and all Catia-Models of the selected Volume are presented in the Workbench under the same root Part.

Only Catia-Models with bounding boxes are searched.

Prior to the Volume-Search a CMI-DMU-File must be created for the root Part via the Workbench-Action -> “Create DMU-File”.

Prior to the Volume-Search a DMU-Volume must be created for the root Part via Action -> “Create DMU-Volume”.

Import a Scene File

It is possible to load a scene file that has been written with the 4D-Navigator. As result all models that are described in the scene file will be marked in the CATIA Workbench.

In *CATIA Workbench*, select menu **Action** > **To 4D-Navigator**
The assembly is transferred to 4D-Navigator.

In the 4D-Navigator: Modify the scenario by using only the Show / NoShow – toggle. After that, save the scenario in a so-called Scene-file

In the *CATIA Workbench*, select menu **File** > **Import 4Dnav-File.** Choose the file that has been written in the last step. All models that were set to Show in 4D-Navigator will be marked in the Workbench.

Notice: The assembly has to be loaded with the **Action** > **To 4D-Navigator** to the 4D-Navigator, and the contents of the Workbench must not be changed until this task.

Canceling Communication

If necessary, you can cancel the communication while the CATIA session is waiting for an assembly structure:

In *CATIA Workbench*, select menu **Action** > **Cancel To CATIA**
The communication is cancelled, the CATIA window becomes active.

Support of Quantity

In Teamcenter Enterprise the relation between part version and part master (*Uses Part Masters*) has an attribute *Quantity*. This attribute indicates how many instances of a component are used by an assembly.

E.g. if a certain screw is used 100 times in an assembly, you would add a component for the screw to the assembly only once, but set the quantity to 100 to indicate that there are 100 pieces of these screws needed for the assembly.

But you will not necessarily keep a CATIA model for the screw, and you don't want to position each screw individually. Therefore with the default parts CMI ignores the quantity. Each part is transferred to CATIA only once, matching the visible product structure.

To indicate that CMI should instead observe the quantity, you can create a special subassembly relation, the *Uses Part Masters with Qty relation*. Whenever this relation is used to build an assembly, the quantity will be observed in CMI. In the example each of the 100 screws would be represented in CATIA. Only one physical model exists, but you can position each screw individually using the CMI functionality.

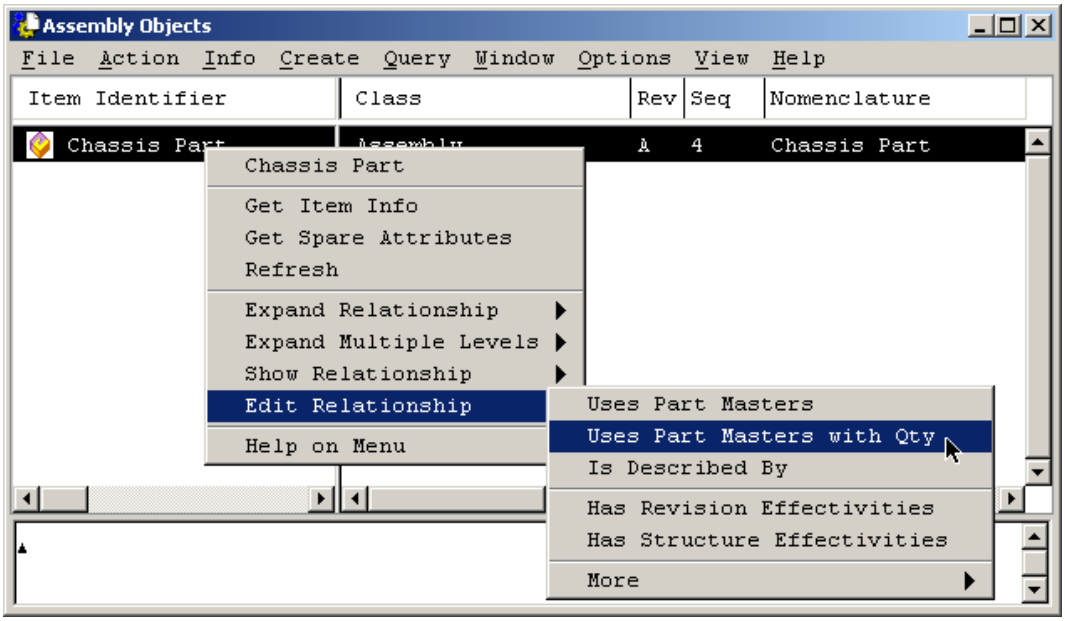


Figure 46: How to create subassembly relations that support Quantity.

The *Uses Part Masters with Qty* relation is created by dropping parts onto an *Edit Relationship* browser.

Using a Multi-Quantity Relation the Quantity is limited to 100 instances by default. An administrator can change this limit.

See *CATIA Teamcenter Interface Customizing Guide* for this and additional considerations regarding CMI Quantity support.

CHAPTER 7

The CATIA V4 CMI Function

This chapter describes how to use the CATIA function METAPHSE within the *CATIA* environment and its interaction with CMI and Teamcenter Enterprise.

Menu structure in CATIA

The following figure shows the menu structure of the CMI CATIA V4 module. In this structure it is possible to see which menu items can be reached through this CATIA function

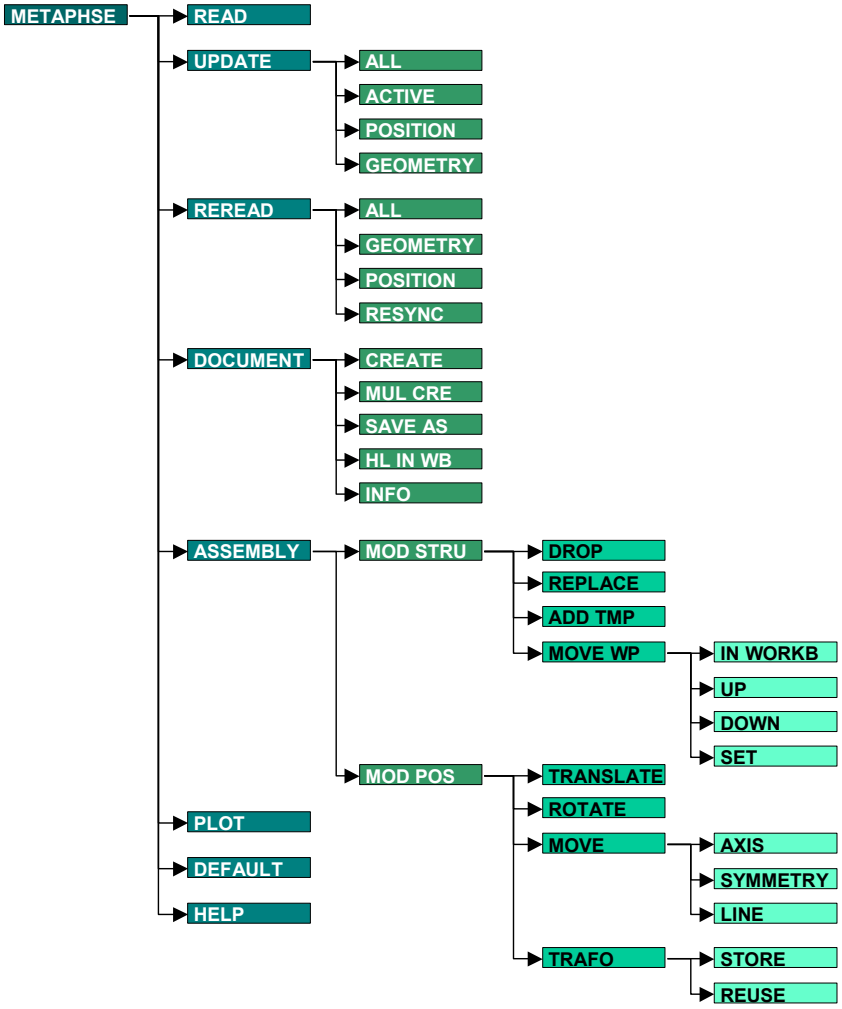


Figure 47: CMI CATIA function menu structure

Usage of the CATIA CMI function

The METAPHSE function is designed conformably to the look and feel of the CATIA Motif user interface. If you are familiar with the CATIA user interface, you will be able to work with the METAPHSE function just as well as usual.

If any message appears on the status line, you can type

/help

on the CATIA command line to get more information about this message.

To get help for the active menu point, you can type

/edmhelp.

METAPHSE->READ

This menu item reads CATIA models, CATIA documents and CATIA parts and assemblies placed in the *CATIA Workbench*.

If you want to send one or more assemblies or models use:

Read Teamcenter CATIA items with **METAPHSE > READ**

After your confirmation Teamcenter comes up.

Drag and drop parts, assemblies, documents or models into the Workbench.

Expand or Deexpand the assemblies or models in the workbench.

Send it to CATIA with **Action > To CATIA**

Modify your CATIA model

Write the modifications back with **METAPHSE > UPDATE > ALL**

After your confirmation the geometry and position information will be updated in Teamcenter Enterprise.

You've loaded a Teamcenter Enterprise 'document' with n CATIA models and want to modify one of these CATIA models?

You can switch to any CATIA model in memory and modify it. After you've finished, write the modifications back with **METAPHSE > UPDATE > ACTIVE** (see also chapter 'Switching the active model')

METAPHSE > UPDATE

The menu item **'UPDATE'** contains actions to update geometry and position information in Teamcenter Enterprise.

UPDATE > ALL

This menu item updates all geometry and position information in the Teamcenter Enterprise database. All items must have been loaded from your Teamcenter Enterprise system before.

Read Teamcenter Enterprise CATIA items with **METAPHSE > READ**

Modify the geometry or position information

Write the modifications back with **METAPHSE > UPDATE > ALL**

After your confirmation a list of models that have been modified will come up in CATIA. If some models were active, but the user must not change these models, a status panel will pop up. It is not necessary to confirm this panel.

Select the models you want to update.

The selected models will be updated in Teamcenter Enterprise.

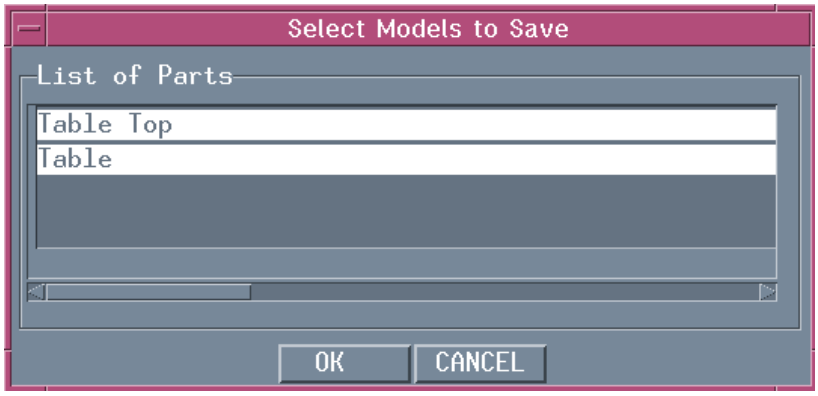


Figure 48: Dialog for Update CATIA Models

UPDATE > ACTIVE

This menu item updates the geometry information of the active model in CATIA in the Teamcenter Enterprise database. As a prerequisite the model must have been loaded from your Teamcenter Enterprise system before.

Read Teamcenter Enterprise CATIA items with **METAPHSE > READ**

Modify the geometry information

Write the modifications back with **METAPHSE > UPDATE > ACTIVE**

After your confirmation the geometry information of the active model in CATIA will be updated in Teamcenter Enterprise.

This menu item can also be used to update the following models:

- Models which have been loaded as a part of an assembly structure,

- Models which have been loaded as a part of a Teamcenter 'Document' that contains more than one CATIA model.

UPDATE > POSITION

This menu item updates all position information changed in CATIA in the Teamcenter Enterprise database. As a prerequisite the assembly must have been loaded from your Teamcenter Enterprise system before.

Read Teamcenter Enterprise CATIA items with **METAPHSE > READ**

Modify the position information

Write the modifications back with **METAPHSE > UPDATE > POSITION**

After your confirmation all position information changed will be updated in Teamcenter Enterprise.

To modify the position information between two parts you have to use the **METAPHSE > ASSEMBLY > MOD POS** command in CATIA. Please refer to chapter '*The Workplane*' to see how to use the workplane concept.

UPDATE > GEOMETRY/MODELS

This menu item updates the geometry information of all activated models in CATIA in the Teamcenter Enterprise database. As a prerequisite the models must have been loaded from your Teamcenter Enterprise system before.

Read Teamcenter Enterprise CATIA items with **METAPHSE > READ**

Modify the geometry information

Write the modifications back with **METAPHSE > UPDATE > GEOMETRY**

After your confirmation a list of models that have been modified will come up in CATIA. If some models were active, but the user must not change these models, a status panel will pop up. It is not necessary to confirm this panel.

Select the models you want to update.

The selected models will be updated in Teamcenter Enterprise.

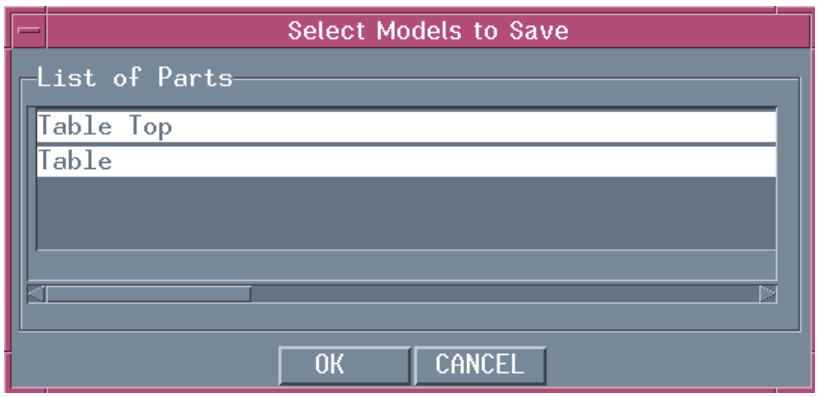


Figure 49: Dialog for Update CATIA Models

METAPHSE > DOCUMENT

The menu item ‘DOCUMENT’ contains actions to save the active model in CATIA in the Teamcenter Enterprise environment.

DOCUMENT > SAVE AS

This menu item creates a Teamcenter Enterprise ‘CATIA model’ object from the model that is currently active in CATIA. A CATIA model selected in Teamcenter Enterprise and dropped into the Workbench is then overwritten by the contents of the CATIA model in CATIA-memory. (Please refer to *Chapter 6* for more information how to create a ‘CATIA model’ object in Teamcenter Enterprise).

Design or load your local CATIA model

In CATIA select **METAPHSE > DOCUMENT > SAVE AS**

Double click or drag the Teamcenter Enterprise CATIA model object that you want to create into the workbench. Its contents will be overwritten with the model that is active in CATIA.

Notice:
If the model you save was opened with File->Open in CATIA, it will be replaced by the model you dropped.
If the model you save came from Teamcenter Enterprise the model you dropped will be added to the CATIA Session and made the active model.
So in both cases you can continue to work with the model and later update it in Teamcenter Enterprise.The content of *the CATIA Workbench* is not cleared.

DOCUMENT > CREATE

This menu item creates a Teamcenter Enterprise ‘CATIA model’ object from the model that is currently active in CATIA. A Teamcenter dialog window appears and you can enter additional attributes.

Design or load your local CATIA model

In CATIA select **METAPHSE > DOCUMENT > CREATE**

A Teamcenter Enterprise dialog window appears.

Fill out the fields and press ‘OK’

A new model will be created in the Teamcenter Enterprise environment.

Notice:
If the active file was opened with File->Open in CATIA, it will be replaced by the new model registered in Teamcenter Enterprise.
If the active model came from Teamcenter Enterprise the new model will be added to the CATIA Session and made the active model.
So in both cases you can continue to work with the model and later update it in Teamcenter Enterprise. The content of *the CATIA Workbench* is not cleared.

DOCUMENT > MUL CRE (multi create)

This menu item creates multiple Teamcenter Enterprise 'CATIA model' objects from the CATIA Session. The new CATIA models are assigned to one 'DOCUMENT'. For each model a Teamcenter Enterprise dialog window appears and you can enter additional attributes.

It is recommended to use this functionality to import supplier CATIA Sessions / models into the Teamcenter Enterprise database. So only models loaded by native CATIA FILE/SESSION OPEN, will be offered to be created.

Load several models from a native CATIA MAP

METAPHSE > DOCUMENT > MUL CRE

Select the models to be created in Teamcenter Enterprise

Notice:
If the active model was opened with File ->Open in CATIA, it will be pre selected

CREATE to confirm

Drop a DOCUMENT into the CATIA Workbench

For each model a Teamcenter Enterprise dialog window appears.

Fill out the fields and press 'OK'

A new model will be created in the Teamcenter Enterprise environment.

Notice:
The models registered in Teamcenter Enterprise will be replaced in CATIA by the new Teamcenter Enterprise models.

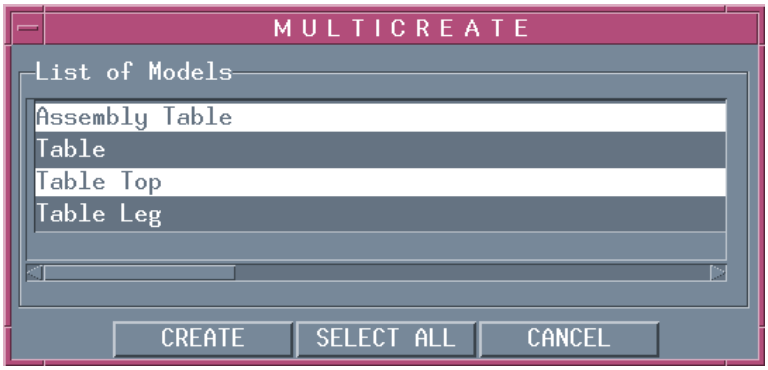


Figure 50: Dialog for Multi Create Models in CATIA

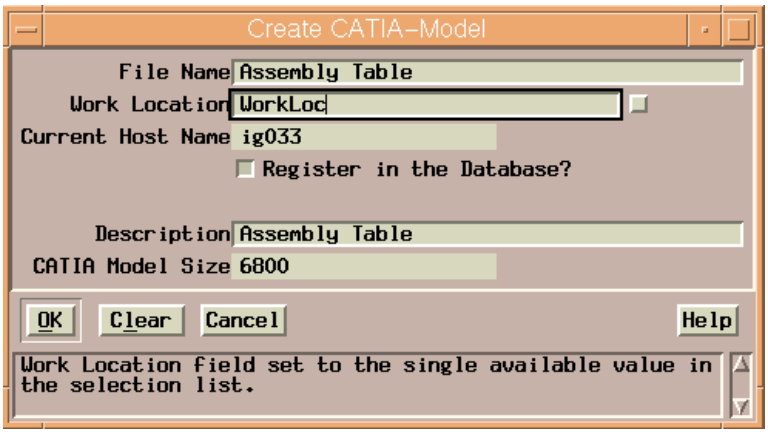


Figure 51: Dialog for Multi Create Models in Teamcenter.

DOCUMENT > HL IN WB

This menu item marks the selected model in the *CATIA Workbench*. This provides a link back from CATIA to Teamcenter Enterprise.

Models must be displayed in the Workbench to see the effects of this function.

Choose **DOCUMENT > HL IN WB**

Select the model you want to mark in Teamcenter Enterprise

After your confirmation the model will be marked in the CATIA Workbench. See Chapter 7: *Working with marked Parts*

DOCUMENT > INFO

This menu item gives you information, stored in Teamcenter Enterprise, about parts you will select.

Choose **DOCUMENT > INFO**

Select the model from which you want to have information.

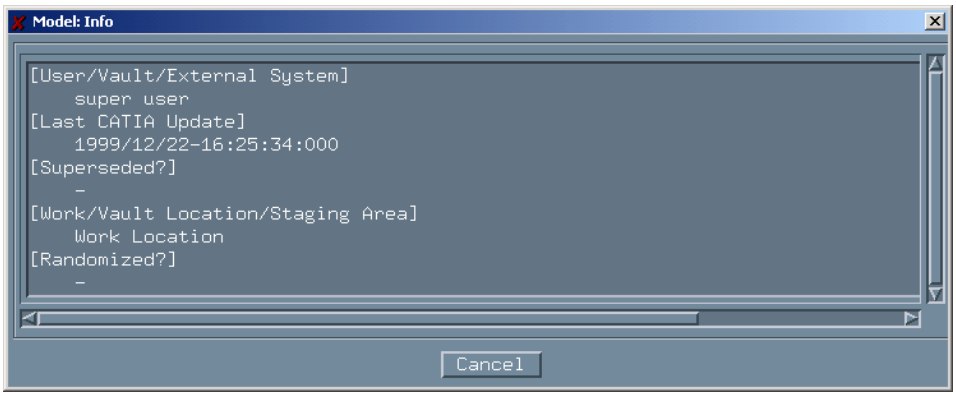


Figure 52: Model Info dialog

After pressing ‘Cancel’ you can select other parts to get their information.

METAPHSE > REREAD

This menu item reads CATIA models, CATIA documents and CATIA parts and assemblies placed in the *CATIA Workbench*.

It works the same way as the METAPHSE > READ function, but in difference to it all functions do not delete the current CATIA Session. All METAPHSE > REREAD functions leave non-CMI models unchanged. This means the designer is able to use models/assemblies from CMI in conjunction with other models, e.g. models loaded over the CATIA File Open… function.

This function is also valuable in cases where you want to expand the assembly already loaded into the CATIA Workbench in a different way (e.g. de-expand one sub tree and re-expand another one instead). It is also possible to add new CATIA models, CATIA documents and CATIA parts and assemblies to the Workbench. After having reread the modified contents of the CATIA Workbench, CATIA will reflect the new situation shown in the Workbench.

Generally all METAPHSE > REREAD functions are much faster than the METAPHSE > READ because only your current CATIA Session is only updated with the parts of the Workbench content not yet loaded to CATIA.

REREAD > ALL

All unsaved modifications (positions and geometries) will be updated with the original data from CMI. REREAD->ALL leaves non-CMI models untouched in the session.

REREAD > GEOMETRY

REREAD->GEOMETRY updates the current CATIA Session with the CMI Workbench content. In addition REREAD->GEOMETRY resets all changed and unsaved models to its original condition. REREAD->GEOMETRY leaves modified positions untouched. REREAD->GEOMETRY leaves non-CMI models untouched in the session.

REREAD > POSITION

REREAD->POSITION updates the current CATIA session with the CMI workbench content. REREAD->POSITION resets all changed and unsaved positions. REREAD->POSITION leaves modified geometries untouched.

REREAD->POSITION leaves non-CMI models untouched in the session.

REREAD > RESYNC

REREAD->RESYNC updates the current CATIA session with the CMI Workbench content. Only models which are not already loaded in CATIA will be added to the current CATIA session. In difference to REREAD->ALL REREAD->RESYNC leaves your modified models and positions unchanged in your session.

This is the fastest way to synchronize the contents of CATIA with the workbench. REREAD->RESYNC leaves non-CMI models untouched in the CATIA Session.

Choose the appropriate **METAPHSE > REREAD** > .... in CATIA.

After your confirmation Teamcenter Enterprise comes up.

Drag and drop parts, assemblies, documents or models into the Workbench if you want to add new ones, and/or Expand or Deexpand assemblies or models in the workbench. Send it to CATIA with **ACTION > To CATIA**

Introduction to the Assembly Structure

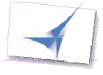
Switching the active model

By switching the active model all individual parts in an assembly can be worked on. Switching will only work on parts belonging to the user.

MODEL > SWAP :Swapping the active model (standard CATIA function)

Select a model

CONFIRM with YES



Edit model

Write back the modifications with:
METAPHSE > UPDATE > ACTIVE

CONFIRM with YES

You can find out the loaded and the active models by help of the standard CATIA function: SESSION > MANAGE

METAPHSE > ASSEMBLY

The menu item ‘ASSEMBLY’ contains assembly structure relevant actions.

ASSEMBLY > MOD STRU

This menu item (Modify Structure) with its sub items allows you to modify the assembly product structure in Teamcenter Enterprise from CATIA. The following functions suppose that an assembly is loaded into the CATIA memory:

ASSEMBLY > MOD STRU > DROP

This menu item removes the relationship between selected sub part and assembly.

Read assembly as described in chapter **‘METAPHSE->READ’**

Choose
ASSEMBLY > MOD STRU > DROP

Select a sub part (the selected part will be highlighted)

You can also pick the part to set the subassembly to drop. Click YES instead of selecting a model:

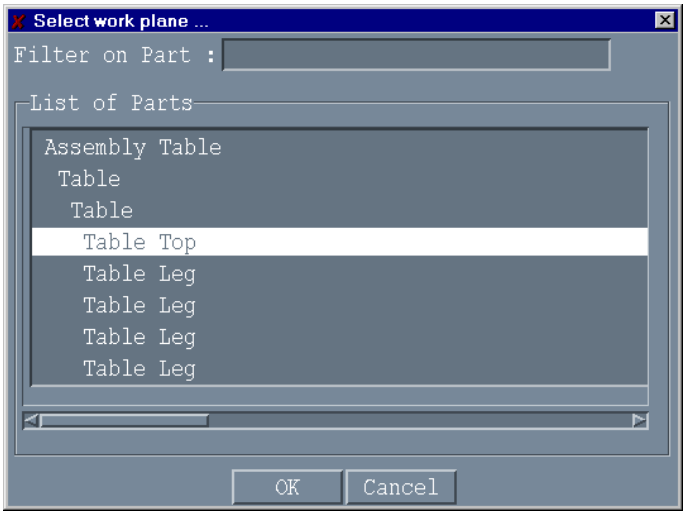


Figure 53: Panel for Select a Subassembly to Drop

After confirmation the sub part will disappear in CATIA memory.

A panel asks if you want to drop the subassembly in Teamcenter Enterprise also. If you click yes the relationship will be dropped in the actual Teamcenter product structure.

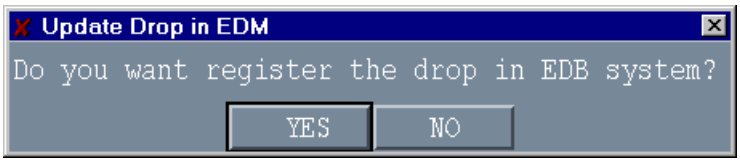


Figure 54: Confirm Drop in EDM dialog

CAUTION:
The relationship is also dropped in the Teamcenter product structure.

ASSEMBLY > MOD STRU > REPLACE

This menu item replaces the selected sub part by another one.

Read assembly as described in chapter '**METAPHSE->READ**'

Choose
MOD STRU > REPLACE

Select a subassembly (the selected part will be highlighted)

YES NO After confirmation the CATIA Workbench window comes to the foreground.

Drop the Part you want to replace the subassembly with into the CATIA Workbench. The Workbench will show the new part in place of the old one. Click Action > To CATIA

CATIA comes to the foreground and the selected part is replaced by the new one.

CAUTION:
The relationship between two parts is also replaced in the Teamcenter Enterprise product structure.

ASSEMBLY > MOD STRU > ADD TMP

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 compartment is big enough, he can load a suitcase as a temporary structure by doing the following (supposing that the car structure is already loaded):

Read the whole assembly as described in chapter '**METAPHSE->READ**'

Choose
MOD STRU > ADD TMP

YES NO After confirmation the CATIA Workbench window comes to the foreground.

Drop the assembly or model you want to add into the CATIA Workbench. Expand it as necessary. Click Action > To CATIA in the Workbench menu. Changes you make to other assembly structures in the Workbench will not be sent to CATIA at this time.

CATIA comes to the foreground with both the car and the suitcase structure. In this case, it is not possible to carry out any changes in the suitcase structure.
The temporary added part is removed from the CATIA Workbench after you send it to CATIA.

ASSEMBLY > MOD STRU > MOVE WP

The work plane defines the level in a sub tree of a hierarchical assembly relative to which modifications in the hierarchical assembly become effective. The relative location and rotation between a part and one of its sub-parts or sub-structures is defined by a relationship. If there is only one tree structure, then the first Part of the tree becomes the root. If there are several trees or other models in CATIA, then a virtual root joins the independent models and sub-trees to one tree.

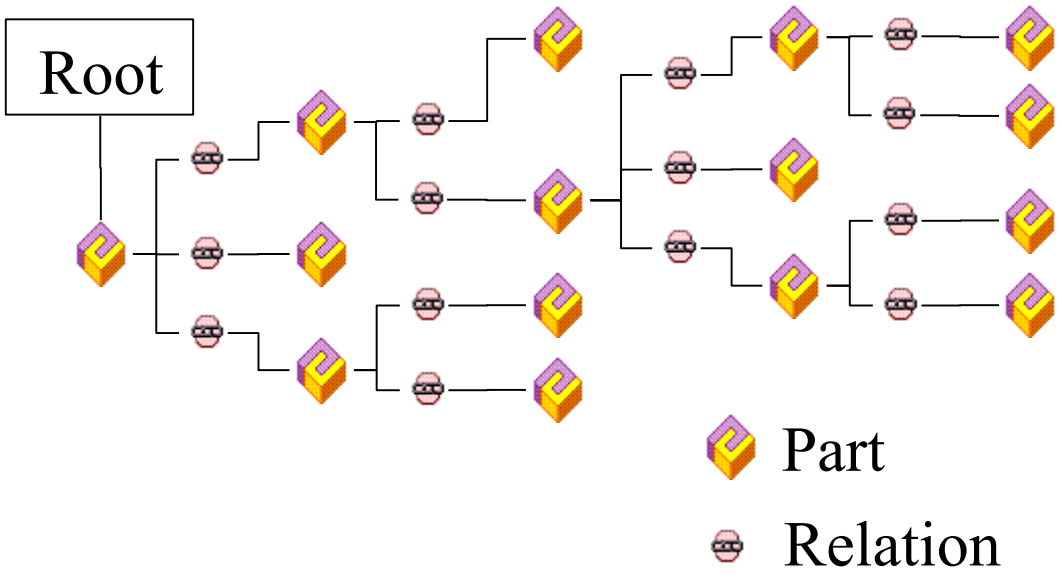


Figure 55: Tree-structure with one tree

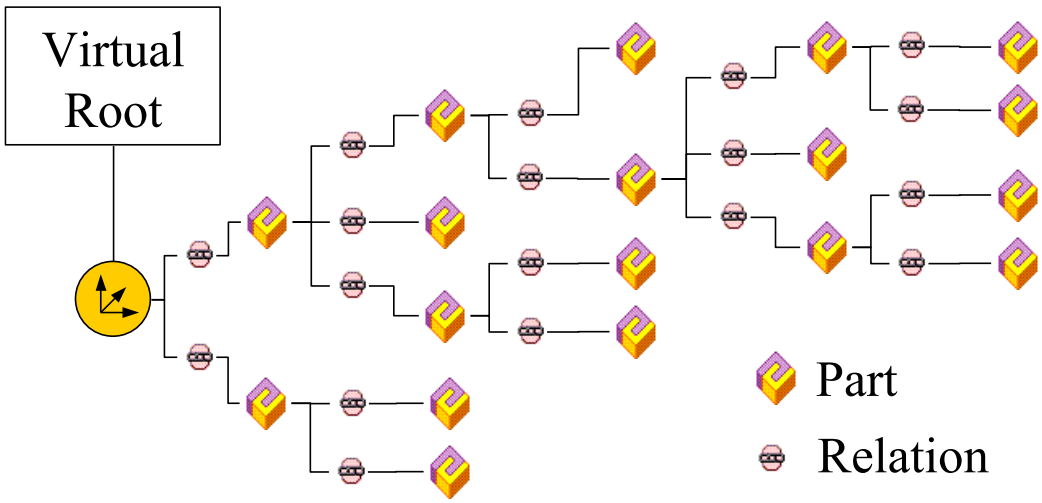


Figure 56: Tree-structure with several trees and a virtual root

It is only possible to transform the relationships between the active *work plane* and its direct sub-parts and its sub-structures.

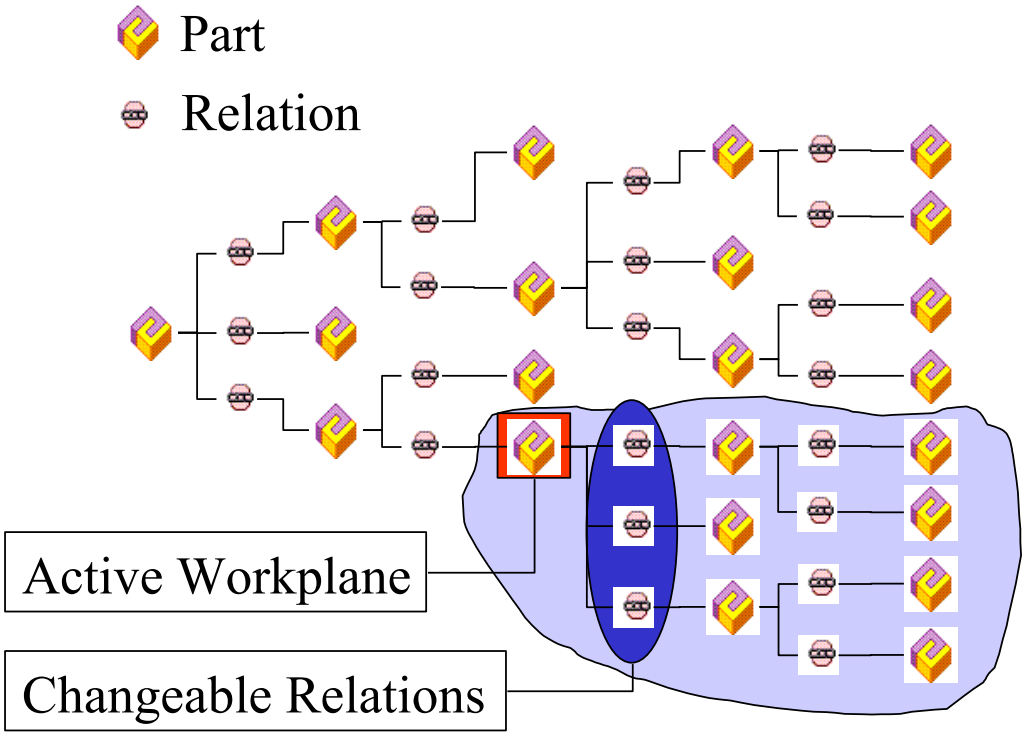


Figure 57: Active workplane in the tree structure

The active work plane and the active model are different. If you want to edit a model you have to make it active.

If you move or rotate a model in the structure by using the TRANSFOR Function then you do not change the relationship to the root structure element, but the geometry of the active model.

There are several possibilities to choose the active work plane:

ASSEMBLY>MODSTRU>MOVE WP > IN WORKB

Call the active work plane from the workbench

YES NO CONFIRM with YES

Choose the assembly in the CATIA-workbench with the right mouse button
Set work plane (see fig. Below)

Send the active work plane from the Workbench to CATIA

Choose the assembly in the CATIA-workbench with the right mouse button
Set work plane

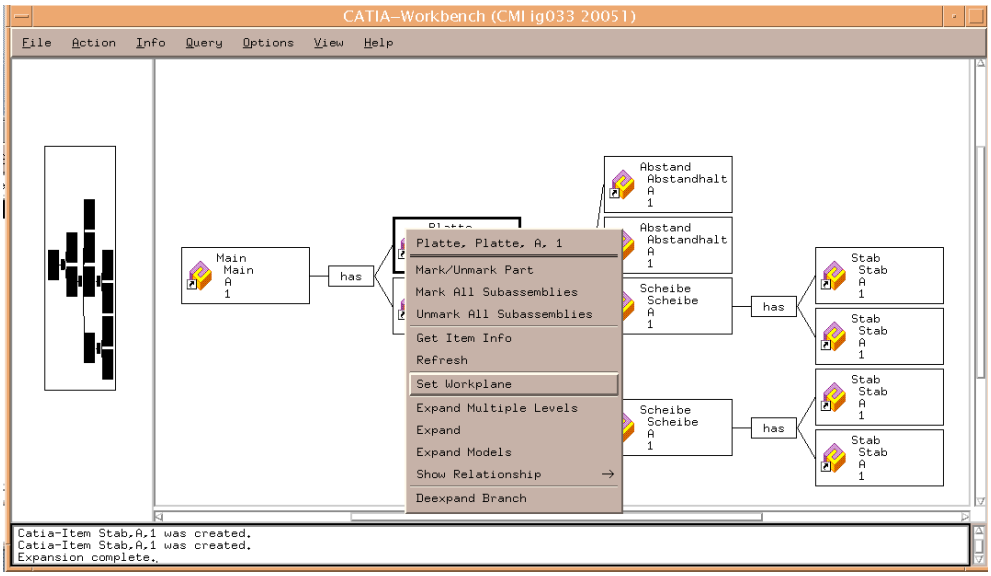
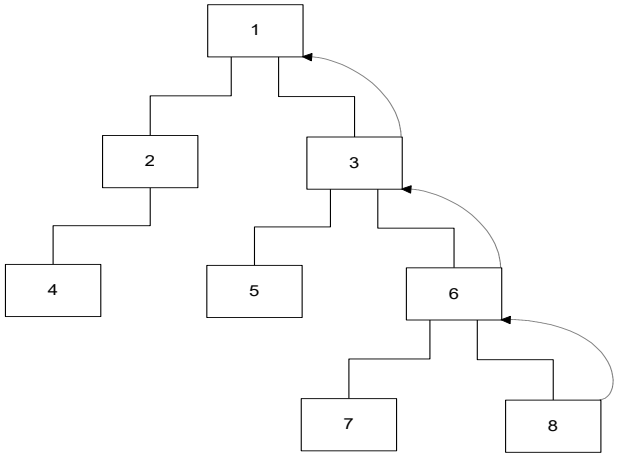


Figure 58: Choose the active workplane in the CATIA Workbench

ASSEMBLY>MODSTRU>MOVE WP > UP

YES NO CONFIRM with YES

Moving the *Workplane* up means that the *Workplane* will be a part in a higher layer of the structure, see in the figure below. The models below the active workplane are highlighted. If there is no higher layer a message will be displayed.



1.

Figure 59: Moving the Workplane ‘up’.

ASSEMBLY>MODSTRU>MOVE WP > DOWN

Moving the workplane down means that the active workplane crosses the tree-structure downwards, see in the figure below. The models below the active workplane are highlighted. If there is no lower workplane a message will be displayed.

YES NO CONFIRM with YES

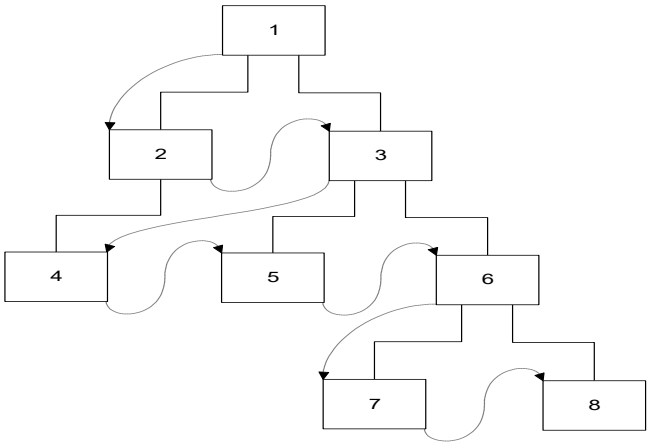


Figure 60: Moving the *Workplane* 'down'.

ASSEMBLY>MODSTRU>MOVE WP > SET

Select Element:

The active workplane and its sub-trees are highlighted.

Or

YES NO CONFIRM with YES

Choose the new workplane from the list (the chosen part and its sub-trees is highlighted) and confirm with OK.

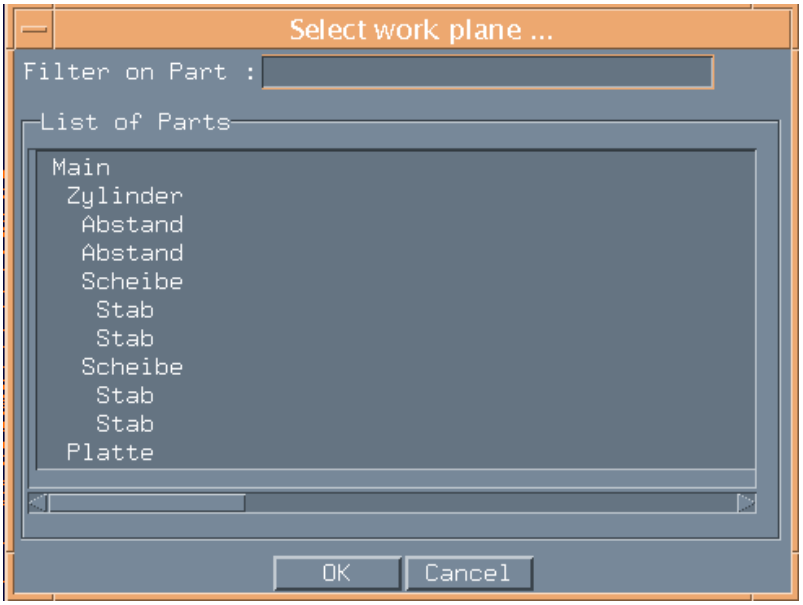


Figure 61: Selection panel for the new workplane

ASSEMBLY > MOD POS

This menu item (Modify Position) with its sub items allows you to modify the positioning of the assembly product structure in Teamcenter from CATIA. The following functions suppose that an assembly is loaded into the CATIA memory:

With all operations in this menu you can use the CATIA YES-button to get a list of all models below the current workplane. So it is easy to select the part you want to move.

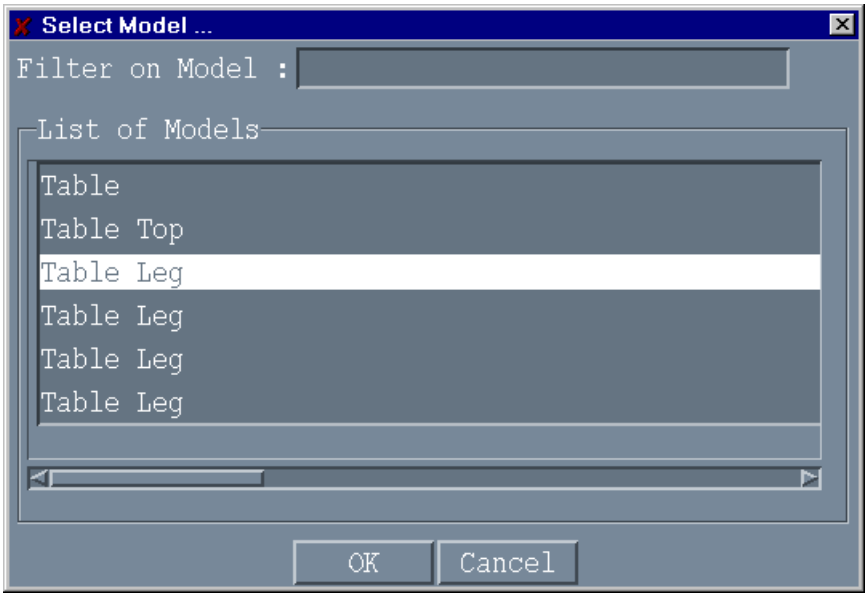


Figure 62: Model selection panel

Please use the 'Filter On Model' field to find only some specific models.

ASSEMBLY > MOD POS > TRANSLAT

This menu item (Translate) translates the selected part in the direction indicated by the selected line, axis or from point to point.

Read an assembly as described in chapter **'METAPHSE->READ'**

Select workplane as described in chapter '**ASSEMBLY > MOD STRU>MOVE WP**'

Select: **MOD POS > TRANSLAT**

Select a part in a sub-tree of the actual workplane (the selected tree will be highlighted red).

Select a vector or line (the selected line/vector will be highlighted red). You can also select a first and second point (a red highlighted vector between the 2 points will be created).

Enter a distance or select the second point.

Click at the vector or press the ENTER Button in order to change the transformation direction.

YES NO

Confirm with YES.
The selected part will be translated.

ASSEMBLY > MOD POS > ROTATE

This menu item rotates the selected part around an axis.

Read an assembly as described in **'METAPHSE->READ'**

Select workplane as described in chapter **ASSEMBLY > MOD STRU>MOVE WP**'

Select: **MOD POS > ROTATE**

Select a part in a sub-tree of the actual workplane (the selected tree will be

highlighted red).

Select an vector/line (the selected vector/line will be highlighted red). In addition a rotation angle occurs to visualize the rotation direction.

Enter an angle or
Select a point/line/plane and a 2nd point/line/plane.
A rotation angle will be calculated and displayed.

Confirm with YES.
The selected part will be rotated.

ASSEMBLY > MOD POS > MOVE > AXIS

This menu item moves the selected part from one coordinate system to another one.

Read an assembly as described in chapter **'METAPHSE->READ'**

Select workplane as described in chapter ' **ASSEMBLY > MOD STRU >MOVE WP'**

Select:
MOD POS > MOVE > AXIS

Select a part in a sub-tree of the actual workplane (the selected tree will be red highlighted)

Or

Confirm with YES

Select part in list

Select first axis

or

create your axis by:

Select a point to create the base

Select another point, line, plane to create the x-axis (change direction by ENTER)

Select another point, line or plane to create the y-axis (change direction by ENTER)

Confirm with YES to create the z-axis

Select second axis

or

create your axis by:

Select a point to create the base

Select another point, line or plane to create the x-axis (change direction by ENTER)

Select another point, line or plane to create the y-axis (change direction by ENTER)

Confirm with YES to create the z-axis

Click at the vector or press the ENTER Button in order to change the transformation direction.

Confirm with YES.
The selected part will be moved to the second coordinate system.

ASSEMBLY > MOD POS > MOVE > SYMMETRY

This menu item mirrors the selected part in relation to a 2D plane.

Read an assembly as described in **'METAPHSE->READ'**

Select workplane as described in chapter **' ASSEMBLY > MOD STRU >MOVE WP'**

Select:
MOD POS > MOVE > SYMMETRY

Select a part in a sub-tree of the actual workplane (the selected tree will be highlighted red).

Select a plane (the selected plane will be highlighted red).

Confirm with YES.
The selected part will be mirrored.

ASSEMBLY > MOD POS > MOVE > LINE

This menu item moves a part. The relative position between the first selected line and the selected part is identical to the relative position between the second selected line and the part after the action is completed.

Read an assembly as described in **'METAPHSE->READ'**

Select workplane as described in chapter **' ASSEMBLY > MOD STRU >MOVE WP'**

Select:
MOD POS > MOVE > LINE

Select a part in a sub-tree of the actual workplane (the selected tree will be highlighted red).

Select the first and second line (the selected lines will be highlighted red).

Confirm with YES.
The selected part will be moved.

ASSEMBLY > MOD POS > TRAFO

The TRAFO functionality is to facilitate the work with all transformation routines of the CMI CATIA module.
This function gives the user the ability to reuse a transformation once defined multiple times. To work with this function at least one transformation must have been performed before.

ASSEMBLY > MOD POS > TRAFO > STORE

This menu item stores the last transformation done (ASSEMBLY > MOD POS> TRANSLATE / ROTATE / MOVE). The stored transformation can be used on any other part (TRAFO > REUSE).

Change the position of an assembly: **METAPHSE > ASSEMBLY > MOD POS >** ...

Select:
MOD POS > TRAFO > STORE

Confirm with YES.
The last transformation part will be stored.

ASSEMBLY > MOD POS > TRAFO > REUSE

After you have stored a transformation, you can reuse it on any part you want. This menu item reuses the last stored transformation.

Select work plane as described in chapter **' ASSEMBLY > MOD STRU >MOVE WP'**

Select a part in a sub-tree of the actual workplane (the selected tree will be highlighted red).

Select:
MOD POS > TRAFO > REUSE

Confirm with YES.
The last transformation part will be reused on the highlighted part.

PLOT

The menu item 'PLOT' attaches the created plot file to the active document object in Teamcenter Enterprise.

Read a Teamcenter 'CATIA Model' object with:
METAPHSE > READ

Design your view which you want to plot

Select: **METAPHSE>PLOT**

After confirmation the plot file is generated.

Save the document back with:
METAPHSE > UPDATE > ACTIVE

After confirmation the generated plot file is registered within Teamcenter Enterprise and attached to the Teamcenter Enterprise CATIA Model object.

Menu structure in CATIA

With the menu item 'DEFAULT' you can switch the work plane functionality. The work plane concept is described in *ASSEMBLY > MOD STRU > MOVE WP*.

If you do geometry transformations with the **ASSEMBLY > MOD POS >** ... function then it is necessary to have the right work plane active

This panel allows you to select the way in which the work plane is activated.

The default selection of the panel is Assembly.

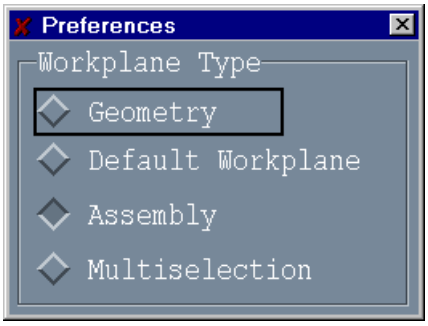


Figure 63: Default panel

Geometry

If you check 'Geometry', the ASSEMBLY > MOD POS functions will change the position of the selected geometry (Model) instead of the assembly. This option is only available if Geometry positions are supported in your customization of CMI.

Default Workplane

If 'Default Workplane' is checked the work plane is automatically set to the next higher assembly in the assembly tree when you select a model.

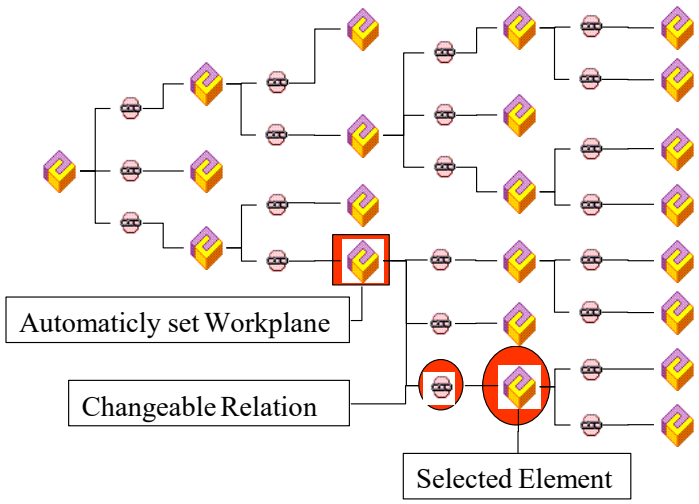


Figure 64: Automatically set default work plane

Assembly

If you check 'Assembly' you can set the work plane with the ASSEMBLY > MOD STRU > MOVE WP commands. If you switch ASSEMBLY > MOD STRU > MOVE WP or set the work plane from the CATIA - workbench, the selection is automatic reset to assembly.

Multiselection

Check Multiselection to apply ASSEMBLY > MOD POS positioning to multiple parts at once. The parts to be positioned can be selected in a panel.
Note that only the leaves of an assembly are displayed for selection. Also a part that occurs in the assembly both as a leaf and with subassemblies is not displayed for selection.

To-do:

Check **METAPHSE > DEFAULT**: Multiselection

METAPHSE > ASSEMBLY > MOD POS > ...

Confirm to get a list of the selectable elements

Select the elements to be transformed (The selected elements are highlighted red) and confirm with OK.

Do the transformation as described in **METAPHSE > ASSEMBLY > MOD POS >** ...

Menu structure in CATIA

To get help for CMI you have 2 possibilities:

Click on METAPHSE->HELP and the browser, which was specified in the customization, will appear in order to show the complete CMI user manual.

To get help for a specific menu point of the CMI CATIA module do the following:

Activate the menu point for which you want to get the help.

Select in the Help Drop Down menu ‘CMI Context Help’

The browser, which was specified in the customization, will appear in order to show the Help chapter for the active menu point.

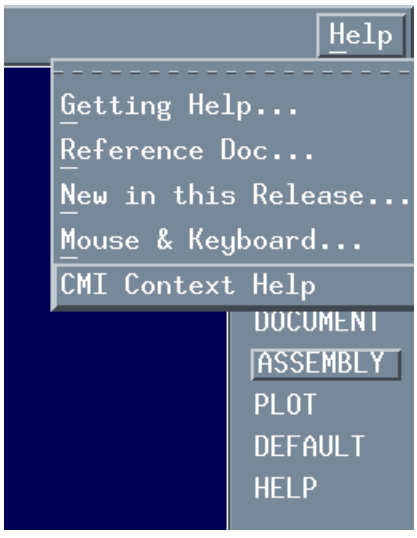


Figure 65: CMI Context Help

Please refer to the installation guide how to customize your default help viewer launching.

The Messages in CATIA

In addition to the online help, it is also possible to get more information to the messages inside the CATIA session on the CATIA command line as following:

/help

This is the list of message numbers and their meaning

Number	Panel output	Description
20501	MODEL LOAD OK	Your model has been loaded successfully.
20502	MODEL SAVE FINISHED	Your model has been saved successfully.
20503	MODEL UPDATE FINISHED	Your model has been updated successfully.
20504	LOAD OK	Your assembly is now loaded.
20505	TEMPORARY ASSEMBLY LOAD OK	Your assembly is now loaded temporary.
20506	ASSEMBLY UPDATE FINISHED	Your assembly has been successfully updated.
20507	ASSEMBLY DROP FINISHED	Your DROP action has been successfully finished.
20508	NO MODEL HAS BEEN CHANGED	No model or assembly has been changed since loading.
20509	MODEL IS ALREADY IN MAP	The model you want to create is already in exchange map. So a overwriting would be performed and could cause lost of model data.
20510	NO POSITION HAS BEEN CHANGED	No position has been changed since loading.
20600	INIT PDM MODULE	This is just a hint that the init process was started
20605	INIT OK	This is just a hint that the init process was ok
20606	REQUEST REFUSED	The request was refused because an EDM process is always pending. Please end the current action and try again.
20610	CAN NOT ALLOCATE PANELFILES	The panel files can not be found, please consult your CATIA system administrator
20615	ERROR IN PERSONAL ENVIRONMENT SCRIPT	Your private environment settings script is corrupt. Please contact your system administrator.
20616	ERROR IN GENERAL ENVIRONMENT SCRIPT	The general environment settings script is corrupt. Please contact your system administrator.
20620	FAILED TO OPEN THE LOGFILE	You have specified a log file in your setting. This log file cannot be opened.
20625	TEMPORARY PART	You have selected a temporary part, which cannot be transformed in its position.
20626	TEMPORARY MODEL	Your active model is a part of a temporary added document. Such models can't be created or updated. Please swap back to the model loaded active if you wish to save it.
20630	NO ASSEMBLY	You have selected a normal model that isn't a part of an assembly.
20631	DROP ERROR	An error occurred while you tried to drop a model. This may happen, if a drop to an active model was performed. To fix it, please change your work plane before DROP.
20632	NO ASSEMBLY LOADED	Only one model is loaded!
20633	NO MODEL LOADED	No model was loaded to perform this action.
20634	SYMMETRY NOT ALLOWED	Symmetry is allowed only for geometry position changing. Change the current working mode to geometry.
20635	ACTIVE MODEL WRONG	The active model is inside of the subassembly selected for modification, Please swap to a model outside of this subassembly.
20636	ALL MODELS IN READ ONLY ACCESS	Your active model is a temporary empty model; it is only used internally. Please swap to the model your selected action is related to.
20637	ACTIVE MODEL WRONG	You tried to replace an assembly outside of your work plane. Please change your work plane.
20638	NO MODEL SELECTED	No model was selected. Please change your work plane.

20639	ACTIVE MODEL HAS NO FRAME	Your currently selected active model is not your first model below the root. Only the first root model contains valid drawing frame attributes! No plot sheet will be generated.
20640	MODELNAME IS MISSING	You tried to save a model without a title. Please setup a title for this model.
20641	SOME MODEL(S) ARE NOT READABLE	CATIA was unable to read the following model(s):
20642	FAILED TO RENAME MODEL	The model RENAME function failed probably because of problems with your exchange map.
20643	MODEL IS NOT READABLE	CATIA was unable to read the model C01 located in C02
20644	FAILED TO SAVE MODEL	CATIA cannot save the active model in the PDM system. Check your access rights of this model in the PDM system.
20645	FAILED TO UPDATE MODEL	CATIA cannot update the active model in the PDM system.
20646	FAILED TO LOAD MODEL	CATIA cannot load the specified model.
20648	MODEL UPDATE NOT POSSIBLE	You can't update a model that isn't loaded from PDM system. Please use CREATE to insert the model in PDM.
20650	NO CORRECT MODEL FOUND	CATIA cannot handle this model in the PDM system.
20653	NO WRITE ACCESS TO MODEL	You tried to write a model in read only mode. You are not the owner of this model.
20654	NO ROOT MODEL	The assembly you have loaded has no root or an empty root.
20655	INCORRECT ASSEMBLY	Something is wrong with your assembly.
20656	MODIFICATION NOT ALOWED	It isn't allowed to modify the selected model; this model is a part of root.
20760	ACTION CANCELED	The action performed was canceled by EDM system.
20761	MODEL HAS WRONG PROJECT	Some Models have incompatible project environment settings to your CATIA project. Check your project environment.
20762	ASSEMBLY PARTIALLY UPDATED	Your assembly has been successfully updated, For more information refer your EDM system.
20763	MODEL PARTIALLY UPDATED	Some models of the splitted document couldn't be updated.
20764	COMMUNICATION FAILED	Is your workbench open and initialized?
20765	ACTION FAILED	The action performed was ended with an error.
20800	PANEL ERROR	The panel function failed, check if the desired panel definition exists in the panel install directory.
20801	NO INFO RETURNED	No info available for this model
20810	NO VALID PLOT FORMAT ERROR	The ditto in current view isn't holding a valid plot format. Please check the ditto.
20811	PLOT FINISHED	The plot is finished.
20812	PLOT IN PROGRESS	The plot is in progress.
20900	PLOT FORMAT TYPE ERROR	The selected plot format is invalid.
20910	CANNOT MOVE WORKPLANE	The work plane can't be moved in this direction. The deepest or highest level in assembly structure has been reached.
20911	FAILED TO CHANGE WORKPLANE	No valid document found to set the WP.
20920	MODEL IN EXCHANGE MAP ALWAYS EXISTS	A model with such a name always exists in your exchange map. Please rename your model or check in the model in your exchange map first and retry.
20921	NO MODEL HAS BEEN LOADED	Before something can be added temporary, a model must be loaded with the read function.

20922	WARNING: MAP NOT FOUND IN DECL SERIES	The map path searched for is not defined yet in your declaration series. A new map will be created temporary.
20923	NO MAP NAME FOR EXCHANGE MAP FOUND	There is no MAP entry in your environment. Please contact your system administrator.
20924	NO VALID MAP PATH OR DLNAME FOUND	Couldn't find a valid entry in declaration series For the given search criteria. Maybe the map you want to read from is invalid Please consult your CATIA system administrator.
20930	SAME POINT SELECTED	Please select 3 different points.
20931	WRONG VECTOR SELECTED	Please select the transformation vector to change the direction of current transformation.
20932	SAME AXIS SELECTED	Please select 2 different axes.
20940	SELECTED MODEL IS NOT IN ASSEMBLY	The selected model belongs not to an assembly loaded, so it can't be managed here. Please use the standard CATIA functions
20941	EXCHANNG MAP PATH DOES NOT EXIST	The specified exchange map does not exist Please contact your system administrator.
20942	CHANGES TO RO MODELS NOT ALLOWED	Update a read only model.
20943	MODEL DOES NOT EXIST	The model does not further exist in CATIA memory. The model was deleted.
20944	MODEL WRITE WAS NOT SUCCESSFULL	The model could not be correctly written to Map. Maybe disk storage is low or NFS is poor. Please contact your system administrator.
20950	WARNING: CHANGES IN SUBASS	There are changes in the subassembly selected for DROP. All model and relation changes have been made in this subassembly will be lost if you continue. If you are unsure save the assembly before DROP. No model or assembly to update. The structure you have loaded contains no model.
20960	NOTHING TO SAVE/UPDATE	No model or assembly to update. The structure you have loaded contains no model.
20965	NO TRANSFORMATION STORED	Before you are able to use this function you have to store the last transformation you have done.
20966	NO VALID TRANSFORMATION FOUND	Unable to found a valid transformation for reuse. First a position must be changed, to be able to store a transformation.
20967	NO TRANSFORMATION DONE YET	First a position must be changed, to be able to store a transformation.
20970	NO HELP SYSTEM DEFINED	A help system (Netscape or mosaic) must be defined. Please contact your system administrator.
20971	CAN NOT CONNECT TO HELP SYSTEM	Problems to connect to help system. Please contact your system administrator.
20972	NO HELP FILE FOUND	No or a wrong help file path was defined in your environment. Please contact your system administrator.
20973	MODEL IS NOT PART OF WORKPLANE	The model you've tried to change is not a part of a subassembly defined by the work plane. Please reset your work plane.
20974	FAILED TO READ STORAGE	Failed to read session data from application data. Please contact your system administrator.
20975	MODEL NEED TO BE WRITTEN	Before you can add a new model to an assembly you have to save this model on file system.
20976	CATIA SESSION HAS BEEN CHANGED	Current CATIA session has been changed. Please redo your selections.

20980	NO LICENSE NUMBER SET	No license number is set in environment. Please contact your system administrator.
20981	LICENSE INIT FAILED	Could not init license process. Please contact your system administrator.
20982	NO MORE FREE LICENSES	All available licenses are currently in use. Retry later or contact your system administrator.
20983	LICENSE EXPIRED	Your license has been expired. Please contact your system administrator.
20984	NO LICENSE FOR MODULE	No license installed for this module. Please contact your system administrator.
20985	DIFFERENT MACHINE TIMES	The LLD machine time differs from the GLD machine time. Please contact your system administrator.
20986	NO CONNECTION TO LLD	No connection to local license daemon. Please contact your system administrator.
20987	NO CONNECTION TO GLD	No connection to global license daemon. Please contact your system administrator.
20988	UNKNOWN LICENSE ERROR	Something is wrong with your license environment. Please contact your system administrator.
20989	UNABLE TO START LLD	Please check your license environment or contact your system administrator.
20990	NO MODEL SELECTED	No model was selected.
20991	NO MODEL FOUND	No model found.
20992	NO POINT SELECTED	No point was selected.
20993	SAME POINT SELECTED	Please select 3 different points.
20994	POINTS ARE LYING ON ONE LINE	Please select at least one different point.
20995	POINTS ARE IDENTICAL	Distance between two points is too small.
20996	INVALID ANGLE VALUE	Input angle is less than angular tolerance. Invalid value for defining rotation.

CHAPTER 8

Working with CATIA V5

Introduction

This chapter describes the CMI interface between CATIA V5 and Teamcenter Enterprise. It provides you with a short overview of the object structure in Teamcenter Enterprise and it's mapping to the CATIA V5 product structure. Typical tasks discussed are CMI Preference configuration, loading models/ parts/ structures/ drawings stored in Teamcenter Enterprise into *CATIA V5* and creating/ updating parts and drawings in Teamcenter Enterprise. One of the main subsections deals with creating and saving CATIA V4 models.

Predefined Object Structure

The CATIA Teamcenter Interface contains the following predefined object structure to support CATIA V5:

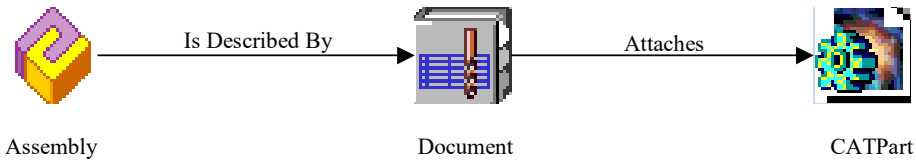


Figure 66: CMI Structure for CATIA V5

The relation 'Is Described By' exists between a part and a document. This relation can be created by dragging and dropping any document onto a part object. [Please refer to the Teamcenter Enterprise user documentation for managing relationships between Teamcenter Enterprise objects]. The document can contain at least one relation 'Attaches' for CATIA V5 model elements. It is possible to attach more than one CATIA V5 model to a single document, however, a particular model may only be attached once within a document. More than one document can be associated with each part.
A structure is made of one or more parts. Teamcenter Enterprise knows two kinds of Part objects:

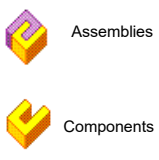


Figure 67: Assembly and Component

An assembly can contain other subparts (i.e. assemblies or components), whereas a component cannot contain other parts.

Object Mapping

Teamcenter Enterprise uses the product structure classes assembly and component, CATIA V5 uses CATProducts and CATParts.
Both assemblies and components are mapped to CATProducts in CATIA V5.
Models in Teamcenter Enterprise are represented by CATParts in CATIA V5.

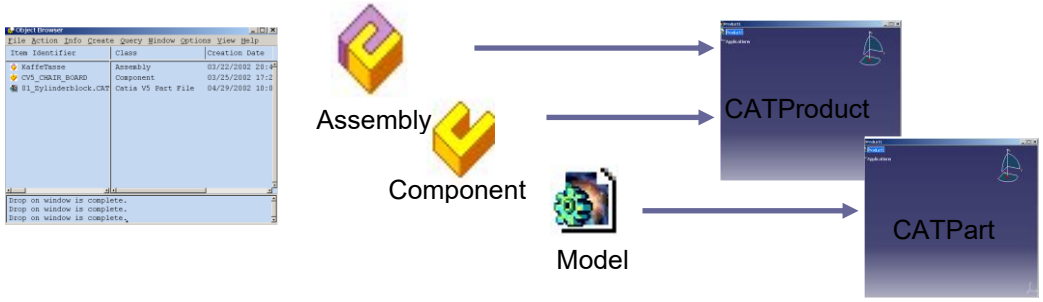


Figure 68: Assembly, Component, Model, CATProduct and CATPart

The CATProduct and the CATPart are stored under different documents of a part.

CMI Preferences

It is necessary to set the CMI preferences correctly in order to work with CATIA V5. The following actions open the CMI Preferences dialog window:

Window-> CATIA-Workbench (the Workbench will then be open and active)

Options-> Change Preferences-> CMI Preferences.

The main preference settings for working with CATIA V5 are as follows:

Work with CATIA	Toggles modes between working with CATIA V4 or working with CATIA V5.
Use CATIA V4 files	Enables the loading of CATIA V4 files to CATIA
Use CATIA V5 CATPart files	Enables the loading of CATParts
Use CATIA V5 CATDrawing files	Enables the loading of CATDrawings
CATProduct Worklocation	Storage location for saving CATProducts and other Catia V5 files
Transfer CGR-files to Catia V5	Allows the user to send CGR files to CATIA V5 (see *Working with CATIA V5 Released Cache*)
Transfer model-files to Catia V5	User can choose to not send model files to CATIA V5

Use CATIA V4 Files

The Use CATIA V4 Files option is evaluated during 'Expand' and 'To Catia' in the Workbench and also during a Read command in CATIA. If Use CATIA V4 Files = “False” then all V4 files are ignored by these functions.

Use CATIA V5 CATPart files

The Use CATIA V5 CATPart option is evaluated during 'Expand' and 'To Catia' in the Workbench and also during a Read command in CATIA. If Use CATIA V5 CATPart Files = “False” then all CATPart files are ignored by these functions.

Use CATIA V5 CATDrawing files

Use CATIA V5 CATDrawing files = “True” loads all CATDrawings during “To Catia”.

Transfer CGR files to Catia V5

Transfer CGR files to Catia V5 is used in the context of Catia V5. It means that CGR-files are transferred to the Catia V5 Release Cache instead of CATParts / V4–models to the exchange-map. In standard CMI, this applies only for checked-in models.

A prerequisite for this is that CGR files are provided by your PDM System. The CGR-files are expected in the same vault-location where the CatPart / V4-model are located, but you can also assign a specific CGR-location via the config-variables CMI_CGR_HOST, CMI_CGR_VAULTLOC_PATH.

There are three allowed values for “Transfer CGR files to Catia V5”

No	“No” suppresses transfer of cgr-files
Only CGR	Only CGR” transfers the cgr-files instead of CATParts / V4–models.
CGR + geometry	CGR + geometry” transfers the cgr-files to the “Release Cache” and the CatParts / V4–models to the exchange-map

Product Structure CATIA V5 vs. Teamcenter Enterprise

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

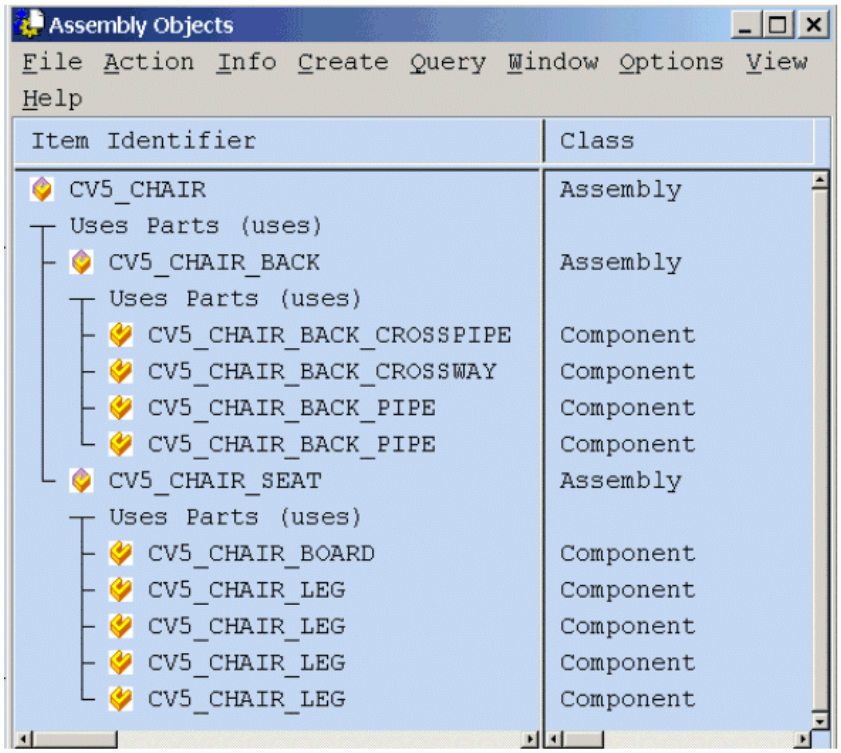


Figure 69: Product Structure in Teamcenter

Assembly CV5_CHAIR has subparts CV5_CHAIR_BACK and CV5_CHAIR_SEAT. Every subpart of the top assembly has further subparts. For example CV5_CHAIR_SEAT has component CV5_CHAIR_LEG attached four times and a single ‘CV5_CHAIR_BOARD’.

If we expand the CV5_CHAIR_SEAT Component we find the part’s CATProduct and CATPart files attached to their respective document folders. The CATProducts are attached within Catia V5 Product Document folders and the CATParts are stored within the Design Document folder of the Component.

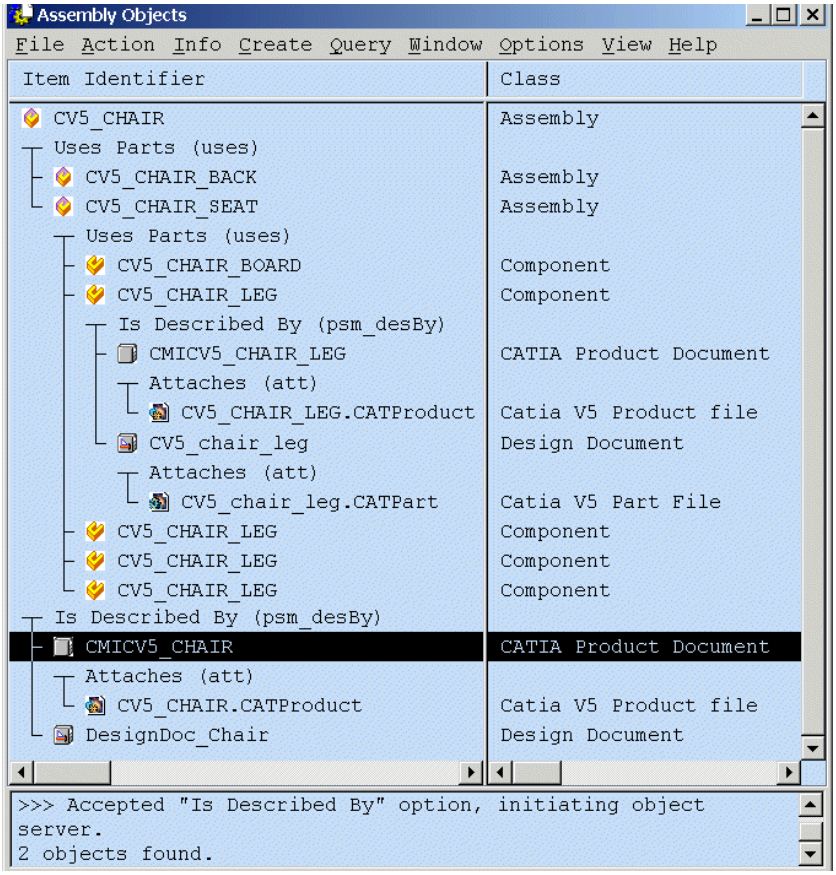


Figure 70: Expanded Product Structure in Teamcenter

This structure will be transferred via the CATIA Workbench to CATIA V5 and displayed in the Product Structure Navigator of CATIA V5 as shown in the following figure.

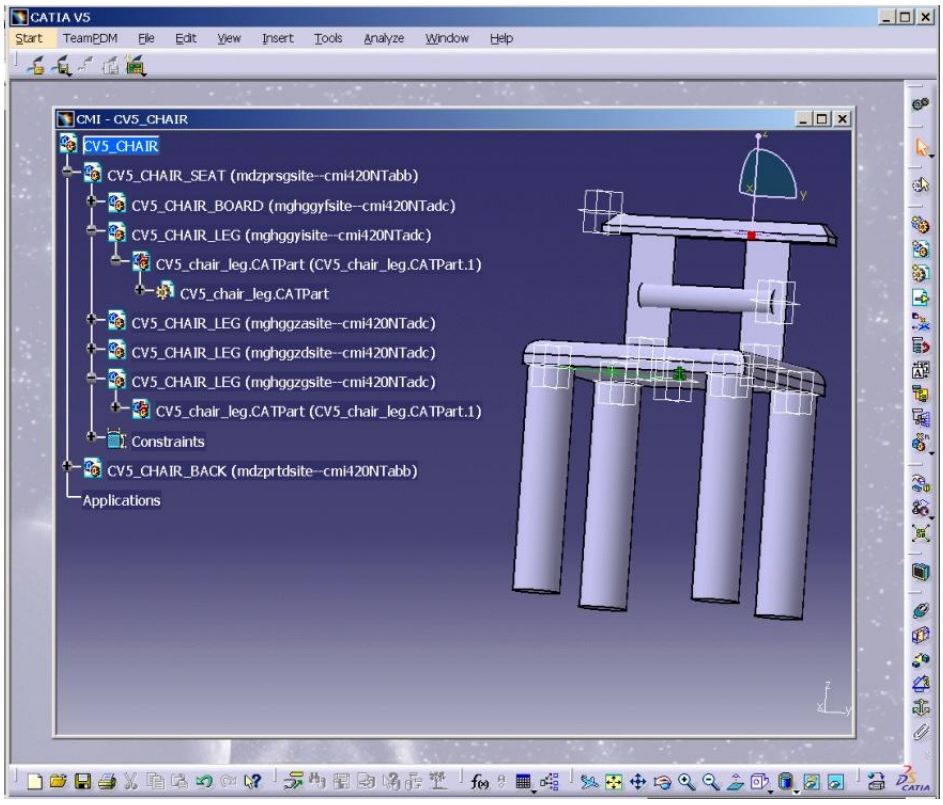


Figure 71: Product Structure Navigator in CATIA V5

In the diagram above the ‘CV5_CHAIR_BACK’ Teamcenter Enterprise Parts of object type “Component” are mapped to CATProducts within CATIA V5.

The part ‘CV5_CHAIR_LEG’ demonstrates the deeper structure of a part in CATIA V5. Immediately under the CATProduct ‘CV5_CHAIR_LEG’ you will find the icon for the same part ‘CV5_chair_leg.CATPart’. This node and all following levels picture the associated model of the CV5_CHAIR_LEG and its characteristics. This is named the CATPart.

You may also build completely new assemblies or extend current assemblies (including geometry) by creating your product structure in CATIA V5 and via the “Synchronization in Teamcenter” command which automatically creates the equivalent assembly in Teamcenter Enterprise

By default, CMI uses a CATProduct with each Component, as shown above. It is also possible to have Components that only use a CATPart. See *Use BOM -- NO BOM geometries*

CMI Toolbar

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

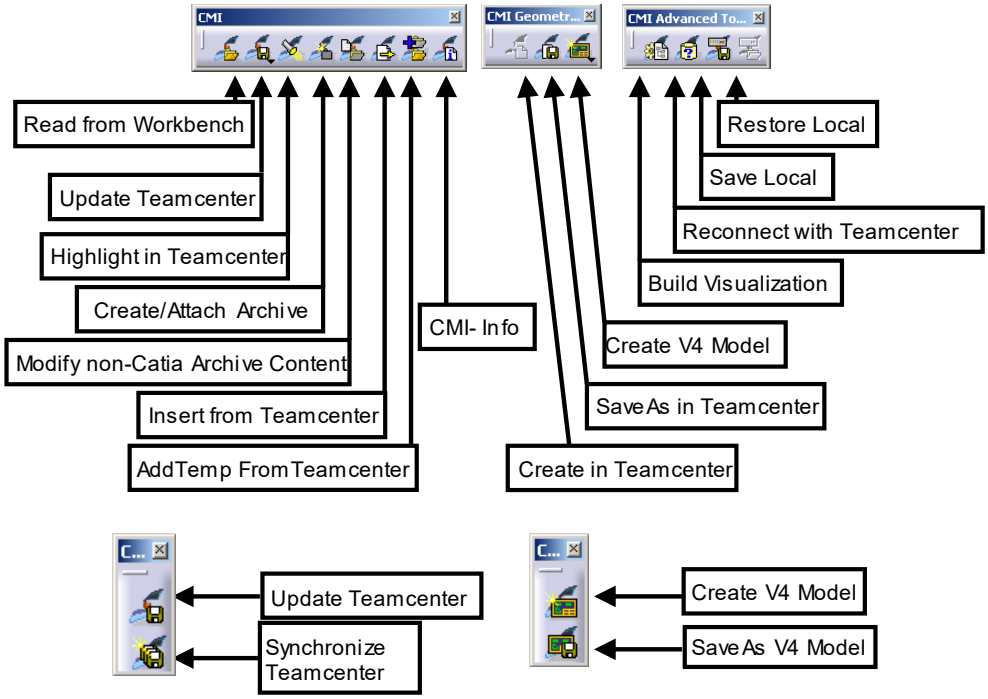


Figure 72: CMI Toolbar in CATIA V5

Using the *CMI toolbar* in the *CATIA Product Structure Workbench* it is possible to read files from the *CATIA Workbench* in Teamcenter Enterprise. You can then modify the geometry in your CATIA V5 session. The modifications can be updated in Teamcenter Enterprise via the ‘Update Teamcenter or the ‘Synchronization in Teamcenter’ button in CATIA V5.

The icons ‘Create in Teamcenter’ and ‘SaveAs in Teamcenter’ are only active if you are working with a CATPart in a CATPart-specific Workbench or with a CATDrawing in a CATDrawing-specific Workbench. The icon ‘Synchronization in Teamcenter’ is only active if you are working in the *Product Structure Navigator*.

CMI also supports storing CATParts as V4 Models. The icons ‘Create V4 Model’ and ‘SaveAs V4 in Teamcenter’ are only active if you are working in a CATPart-specific Workbench, but you can also create CATIA V4 model if you’ve selected a CATPart in the Product. 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.

An administrator can remove any CMI Command from the toolbars and menu, in order to keep the interface lean. Please see *Customizing Manual* for details.

On UNIX the commands on subordinate toolbars are part of the main toolbar.

CMI Options

The CMI options in CATIA can be found in the “Tools->Options” menu in the General->Compatibility section under the CMI tab.

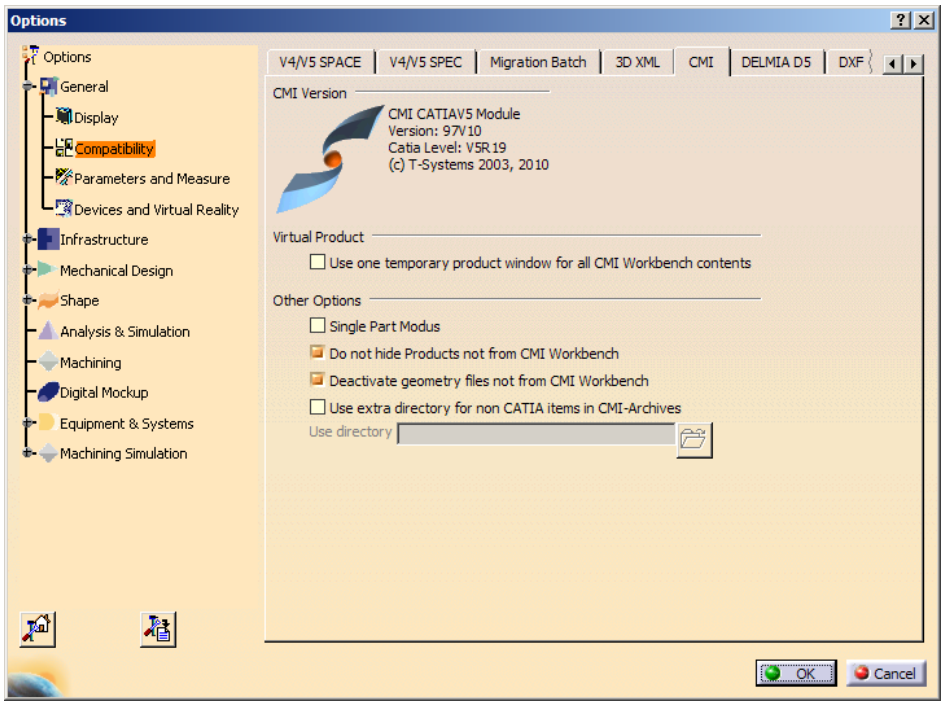


Figure 73: CMI Options dialog

Use one temporary product window for all CMI Workbench contents

When this option is checked all top level Assemblies in the CMI Workbench are loaded under a temporary CATProduct in CATIA.

Single Part Modus

When this option is checked all CATParts of a structure are loaded in separate windows during CMI Read. The rest of the structure will not be loaded.

Do not hide Products not from CMI Workbench

When this option is checked CMI does not hide the any node in the Product structure during CMI Read. By default CMI hides all nodes in the CATIA tree not loaded by CMI.

Deactivate geometry files not from CMI Workbench

When this option is checked CMI deactivates all geometry nodes not loaded by CMI during the current CMI Read action in the Product structure.

Use extra directory for non CATIA items in CMI-Archive

Select an additional directory where non CATIA files (like jpeg, doc, …) are handled for CMI Archives.

CATParts and CATDrawings in CATIA-Workbench in Teamcenter Enterprise

CATIA V5 will read all expanded assemblies, single CATParts, CATDrawings or CATIA V4 models from the Teamcenter Enterprise CATIA Workbench.

The following figure shows a CATDrawing item and a CATPart item in the CATIA Workbench:

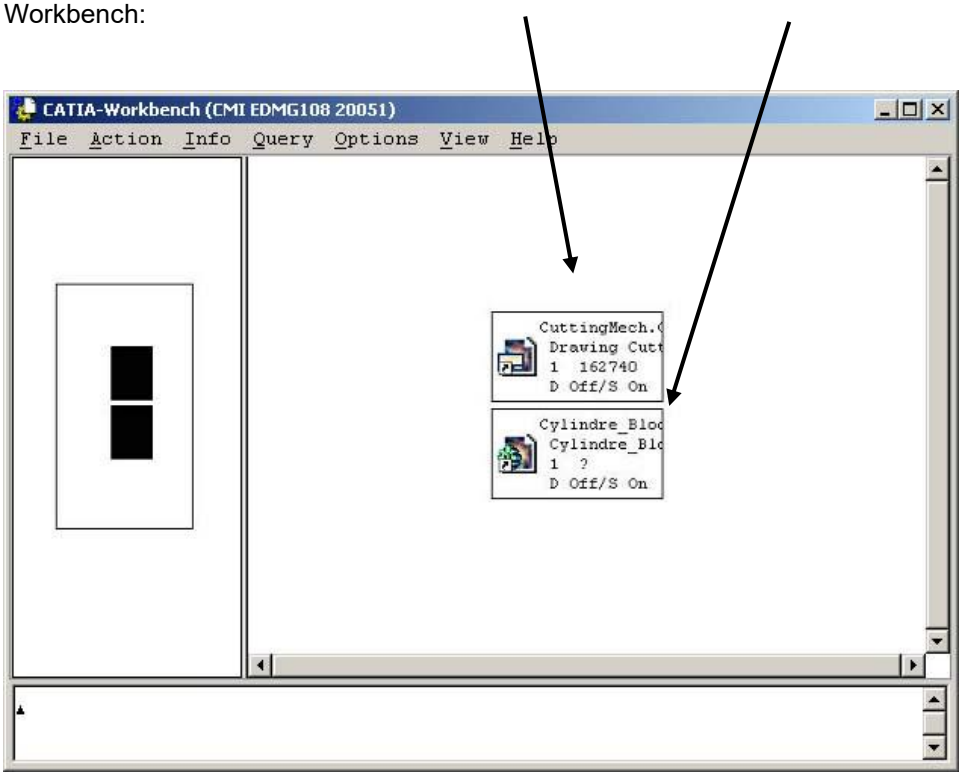


Figure 74: CATIA Workbench with a CATDrawing item and a CATPart item

When you have transferred a CATPart from Teamcenter Enterprise to CATIA V5 via the CATIA Workbench, the part 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 Enterprise, however in CATIA V5 they will be displayed in separate windows as shown in the figures below:

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

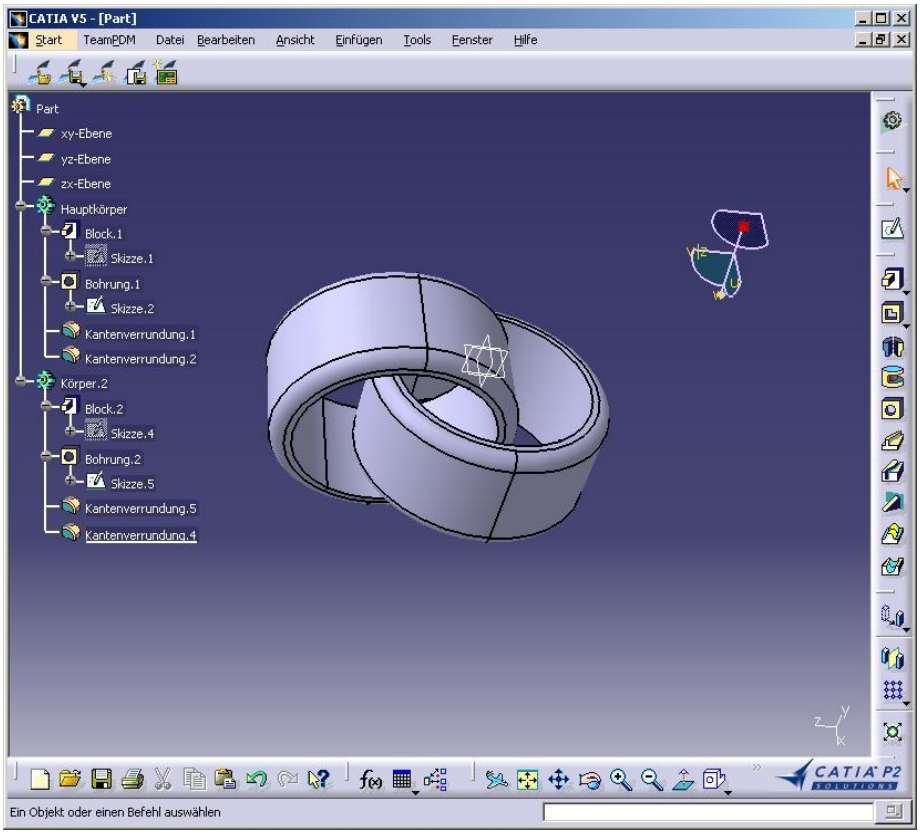


Figure 75: Part Design Workbench for Read, Update, Create or SaveAs CATPart

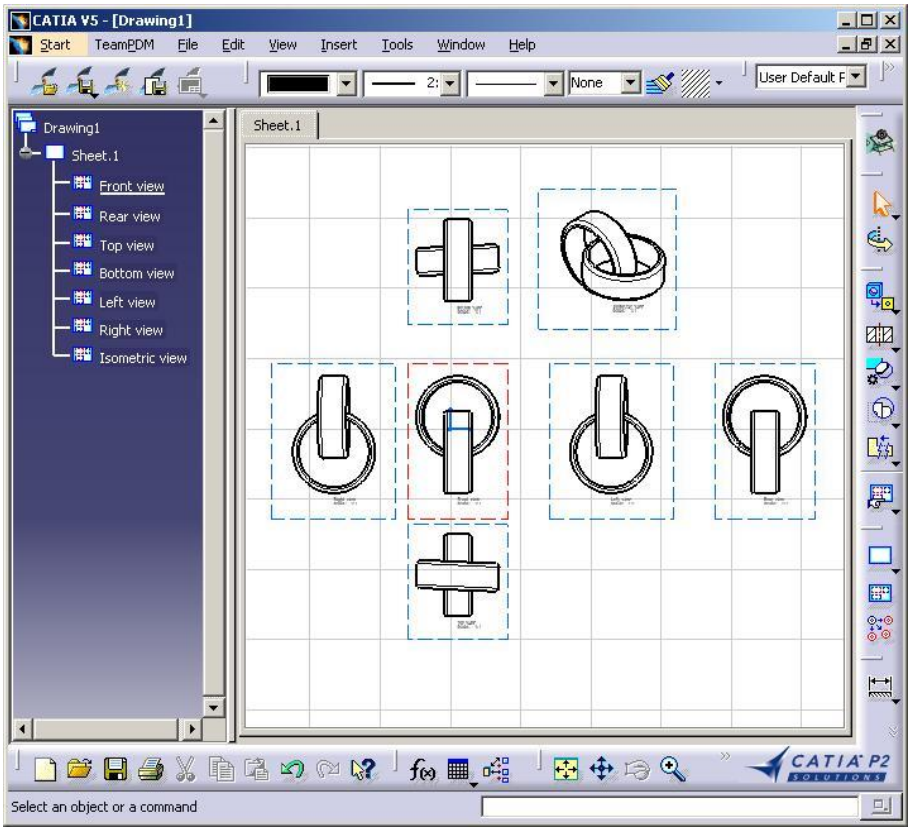


Figure 76: Drawing Workbench for Read, Update, Create or Save As CATDrawing

CMI toolbar: Read From Workbench

With the 'Read from Workbench'-button in the CMI Toolbar the contents of the CATIA Workbench in Teamcenter Enterprise may be read into CATIA V5. The CATIA Workbench may contain CATParts, CATDrawings, CATIA documents, CATIA V4 models and assembly structures.

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

Drag and drop parts, drawings, models, assemblies or documents into the Workbench.

Expand or Deexpand the assemblies in the Workbench.

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

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

CMI toolbar: Update Teamcenter

This command updates geometry and position information in Teamcenter Enterprise.

Modify the contents in CATIA.

Write the modifications back with **'Update Teamcenter'** icon .

Modified geometry, CATProducts and transformation matrices will be updated.

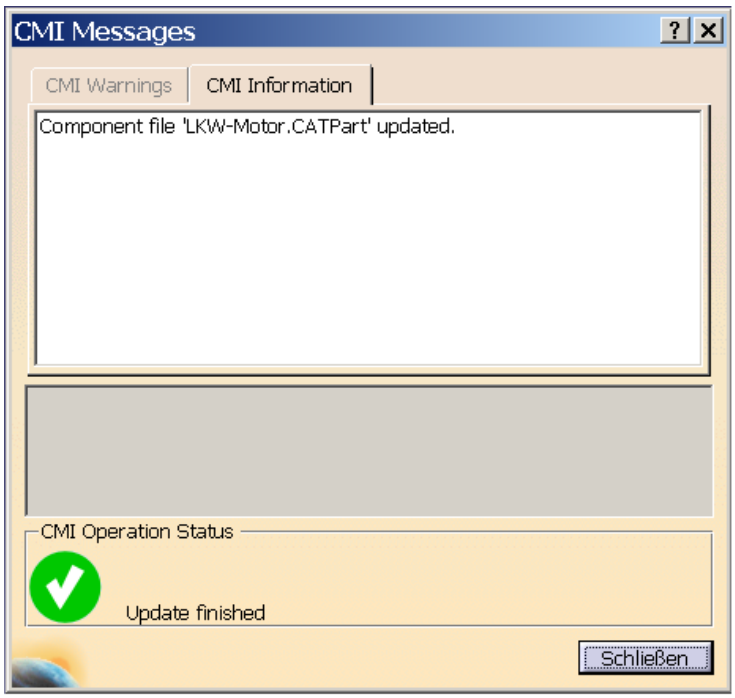


Figure 77: Dialog Window for Update CATIA Models

CMI toolbar: Synchronization in Teamcenter

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

- any geometry and position information changes
- the creation of any new CATParts/CGR and CATProducts that have been added to your Product Structure
- the creation of any new CATPart/CGR and CATProduct instances
- the removal of CATProducts from the Product Structure

The list of changes made to the Product Structure since the last Read/Update/Synchronize is presented in the form of list of operations to be performed in Teamcenter Enterprise.

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 Enterprise with the **'Synchronization in Teamcenter'** icon .

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

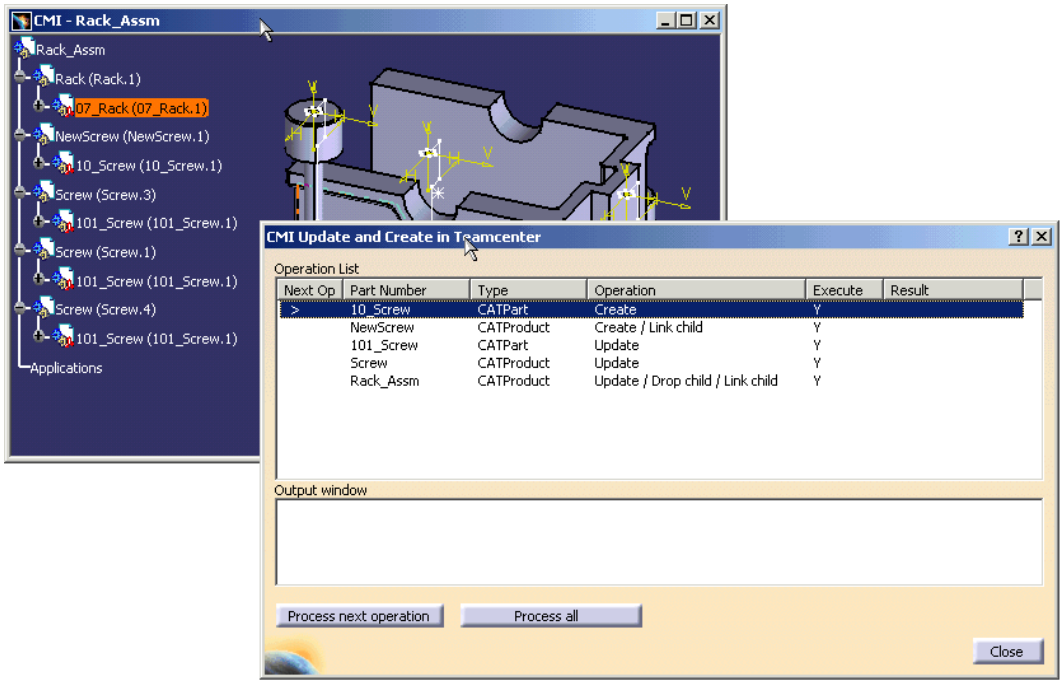


Figure 78: The Synchronization in Teamcenter Dialog

The dialog's *Operation* column shows which procedures need to be performed in Teamcenter Enterprise 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 Enterprise.
Create	The object will be created in Teamcenter Enterprise. The object is completely new and not already in Teamcenter Enterprise.
Link child	At least one new parent-child relation will be created in Teamcenter Enterprise. This reflects the addition of a child - product in CATIA.
Drop Child	At least one parent-child relation will be deleted in Teamcenter Enterprise. This reflects the deletion of a child-product of the object in CATIA.
Prepare	A CATPart has been brought into the product structure. It has been recognized as being from CMI/ Teamcenter Enterprise, but it requires preparation before being brought into the assembly in Teamcenter Enterprise. Preparation could, for example, be the creation of a General Document to attach the file to.

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 the *Process Next Operation* button is pressed, the operation marked in the Next Op column by the > symbol is executed.

If the *Process All button* is Pressed, the list of executable operations is worked through in the order shown on the screen.

In both cases the results from Teamcenter Enterprise are shown in the Output Window

below the list.

The **update** operation:

For *CATPart* objects this means geometric information will be updated in Teamcenter Enterprise. If the model in the assembly has been modified, a dialog window will appear in Teamcenter asking for confirmation of the model that needs to be updated in Teamcenter. (see figure 76). You may deselect the model if you do not want the model to be updated.

For *CATProduct* objects this means that the file will be updated in Teamcenter Enterprise. No user interaction is necessary.

The **create** operation:

For *CATProduct* objects, if the *nomenclature* property field has been filled in CATIA then no user interaction will be required and the following steps will be completed:

- a Teamcenter Enterprise Part will be created
- a document object will be created
- the CATProduct file will be registered
- the document will be attached to the Part and the CATProduct file to the document

If the *nomenclature* property field has not been filled in CATIA, then the Create Assembly Teamcenter Enterprise dialog box will appear requesting the missing nomenclature attribute. When the nomenclature attribute has been filled, the procedure completes as above.

For *CATPart/CGR* objects a dialog window for creating a CATPart/CGR will appear - similar to 'Action → Prepare' in Teamcenter Enterprise.

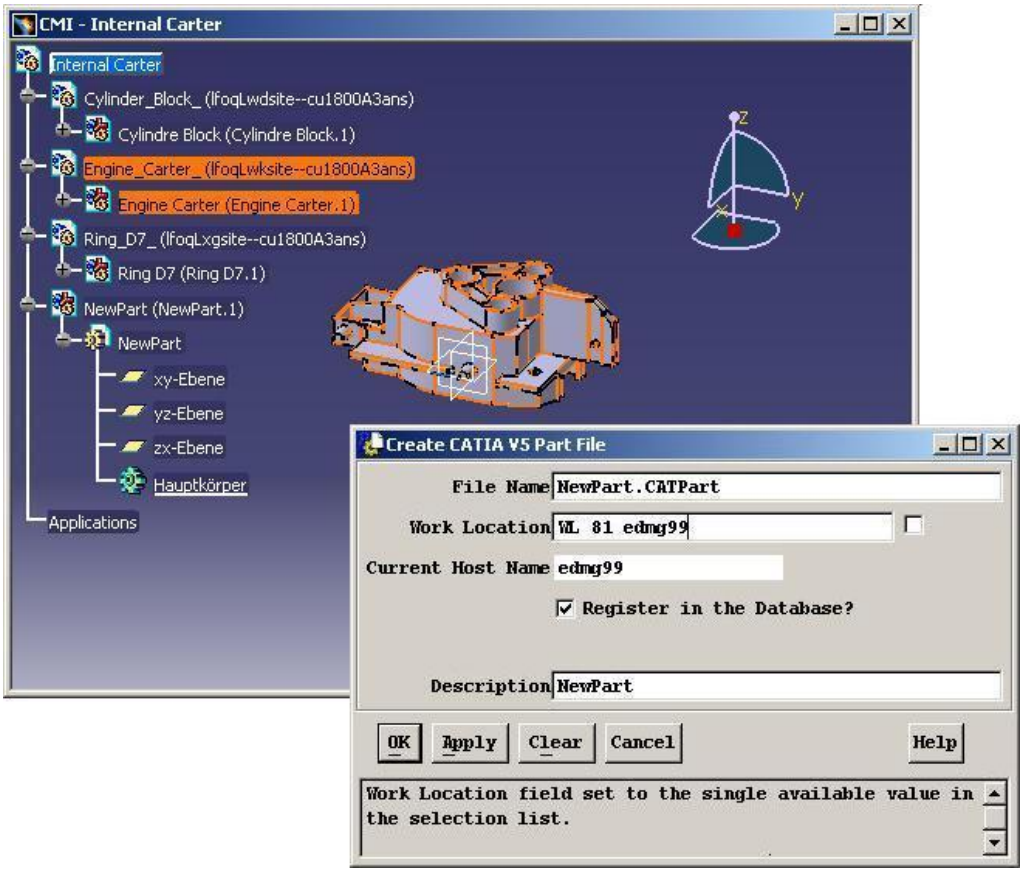


Figure 79: Dialog Window to Create a new CATPart within the Product Structure in Teamcenter Enterprise

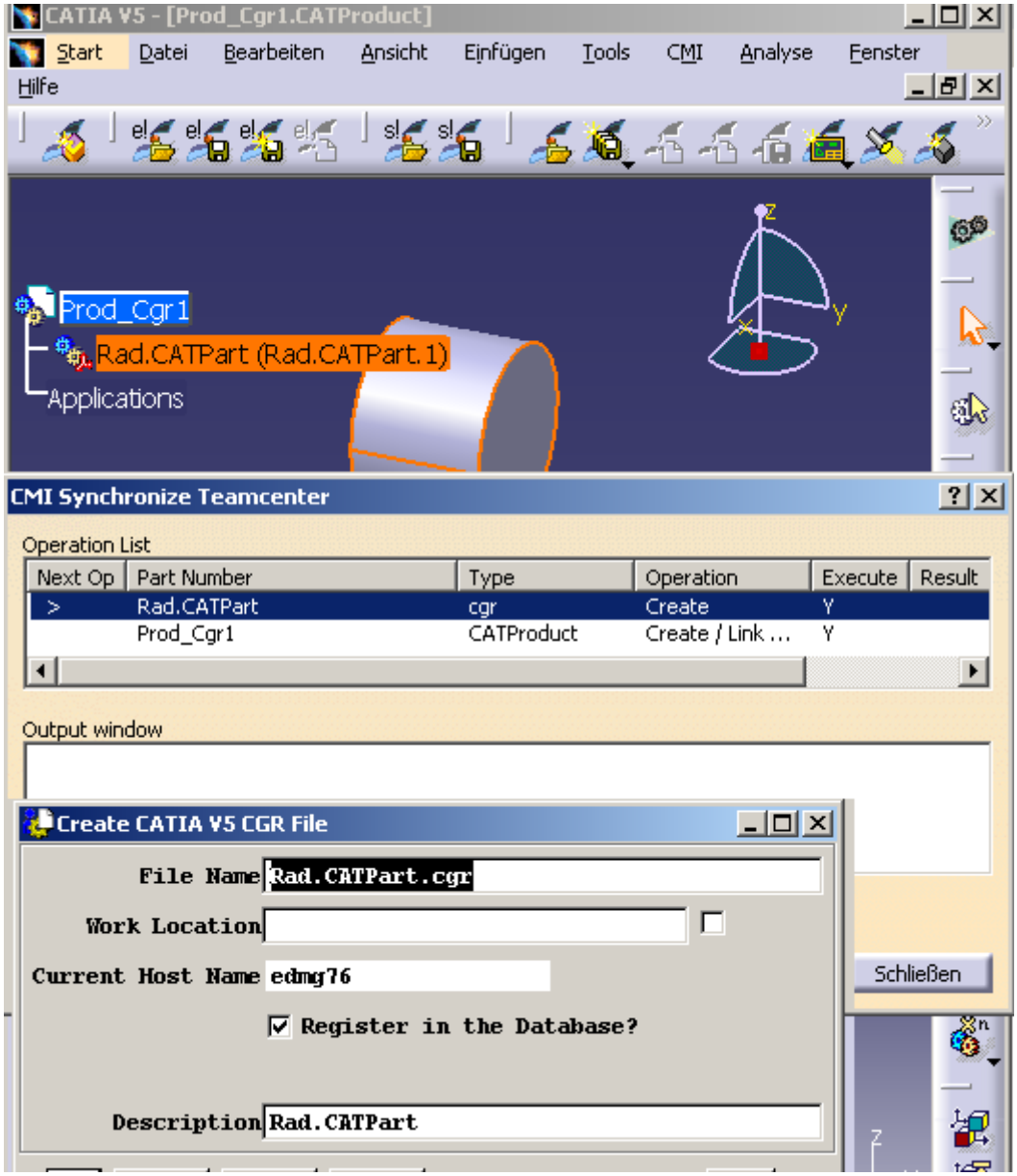


Figure 80: Dialog Window to Create a new CGR within the Product Structure in Teamcenter Enterprise

Fill in the fields and press ‘OK’

The CATPart/CGR will be created in Teamcenter Enterprise. After your confirmation a dialog window for creating a corresponding document will appear as follows.

Figure 81: Dialog Window to Prepare a Document in Teamcenter

If this document exists in Teamcenter Enterprise, the CATPart will be attached to this document, otherwise a new document will be created in Teamcenter Enterprise.

The **link child** operation:

This may only appear as an operation for *CATProduct* objects, as CATParts cannot contain sub-products. In Teamcenter Enterprise any new child CATProducts or CATParts will be attached to the Teamcenter Enterprise Part of the parent object using the following methods:

- if the child is a CATPart, then the document attached to the child CATPart is attached to the parent Part object
- if the child is a CATProduct, then its Teamcenter Enterprise Part is attached to the parent Part to create the Part-Part relation.

The **drop child** operation:

This may only appear as an operation for *CATProduct* objects as CATParts cannot contain sub-products. Also, CATParts cannot be dropped in Teamcenter Enterprise (without customization). In Teamcenter Enterprise any child CATProducts deleted from the CATIA product structure will be detached from the Teamcenter Enterprise Part of the parent object using the following method:

- if the child is a CATPart, then no drop will occur
  (see also *Deleting NON-BOM Parts during Synchronize*)
- if the child is a CATProduct, then its Teamcenter Enterprise Part is detached from the parent Part (the Part-Part relation is deleted)

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.

CMI Geometry toolbar: Create in Teamcenter

With this toolbar button a CATPart or CATDrawing file currently set active in CATIA V5 can be created in Teamcenter Enterprise. A Teamcenter *create* dialog window appears to enable the user to set additional attributes.

Design or load your local CATPart or CATDrawing.

In CATIA V5 select the **'Create in Teamcenter'** icon **.**

A Teamcenter Enterprise dialog window appears.

Fill in the fields and press 'OK'.

A new CATPart or CATDrawing will be created in the Teamcenter Enterprise environment.

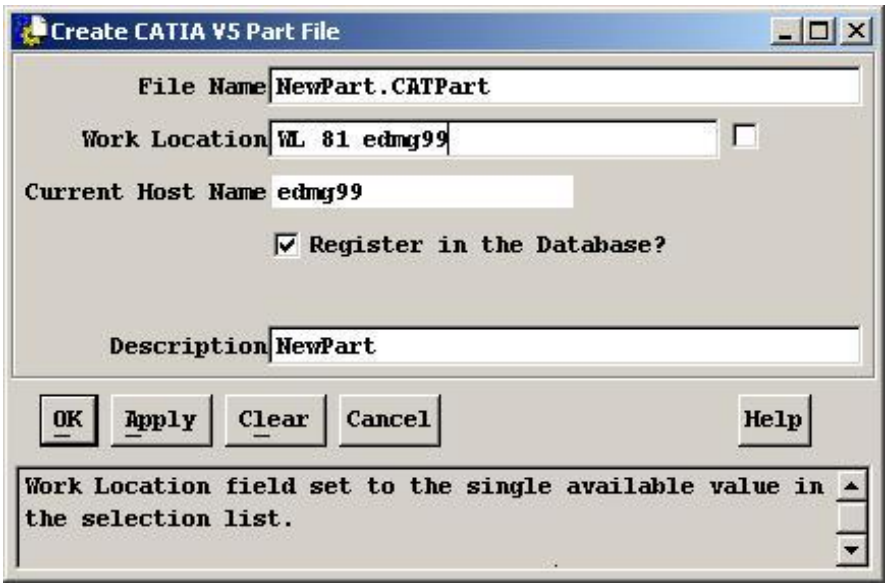


Figure 82: Dialog Window for Create CATIA V5 Parts or CATIA V5 Drawings

The Create in Teamcenter Enterprise command is available if the CATPart or CATDrawing has been opened in a separate CATPart or CATDrawing Workbench window in CATIA V5.

CMI Geometry toolbar: SaveAs in Teamcenter

With this command a CATPart file or CATDrawing file currently set active in CATIA V5 can replace another in Teamcenter Enterprise. A CATPart or CATDrawing selected in Teamcenter Enterprise and then dropped into the Workbench 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 .

Teamcenter Enterprise will request you to drag and drop an item into the CMI Workbench.

In order to 'Save As in Teamcenter' with a CATPart, double click or drag the CATPart (or the document with an attached CATPart) to be overwritten into the CATIA Workbench in Teamcenter Enterprise. The CATPart active in CATIA V5 will then overwrite its contents. The same process applies to CATDrawings.

'Save As in Teamcenter' is only allowed if the item (CATPart or CATDrawing) has been opened in a CATPart or CATDrawing Workbench window in CATIA V5.

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.

CMI Geometry toolbar: Create V4 Model

This command converts a CATPart to a CATIA V4 model and creates it in Teamcenter Enterprise.

Design or load your local CATPart.

In CATIA V5 select the **'Create V4 Model'** icon .

A Teamcenter dialog window appears.

Fill in the fields and press 'OK'

A new CATPart will be created in the Teamcenter Enterprise environment.

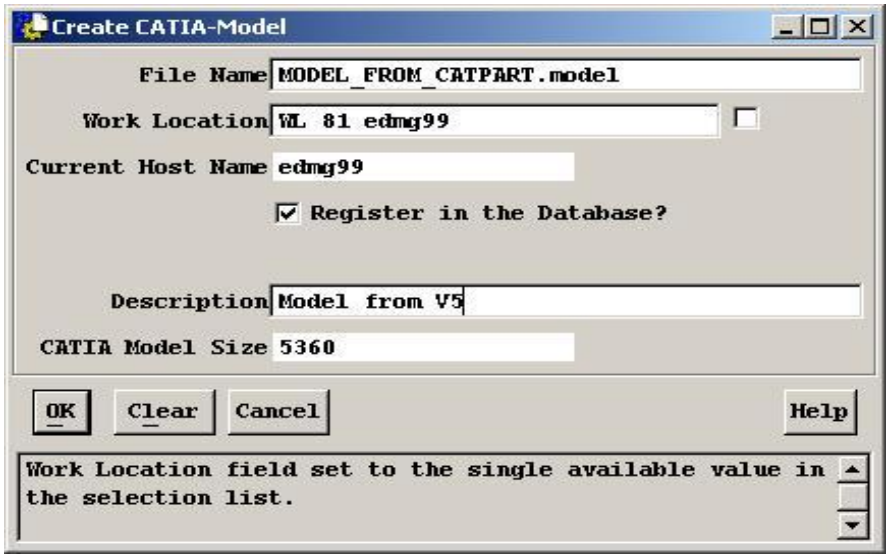


Figure 83: Dialog Window for Create CATIA Model in Teamcenter

You may only create CATIA V4 Models from CATParts. These parts must be opened in a CATPart Workbench window or be selected in the Product Structure Navigator.

CMI Geometry toolbar: SaveAs V4 Model in Teamcenter

With this toolbar button a CATPart file currently set active in CATIA V5 may be created in Teamcenter Enterprise as a CATIA V4 model. A CATIA V4 model selected in Teamcenter Enterprise and dropped into the Workbench is overwritten by the contents of the converted CATPart in CATIA-memory.

Design or load your local CATPart.

In CATIA V5 select the **'Save As V4 in Teamcenter'** icon .

Teamcenter Enterprise will request you to drag and drop an item in the Workbench.

In order to 'Save As V4 in Teamcenter' with a CATPart, double click or drag a CATIA V4 model (or document with an attached CATIA V4 model) to be overwritten into the Workbench in Teamcenter Enterprise. Its contents will be overwritten by the CATPart active in CATIA V5 in CATIA V4 model format.

The 'Save As CATIA V4 Model' toolbar button is only enabled if the CATPart item is open in a separate Workbench window in CATIA V5.

CMI toolbar: CMI Info

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

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

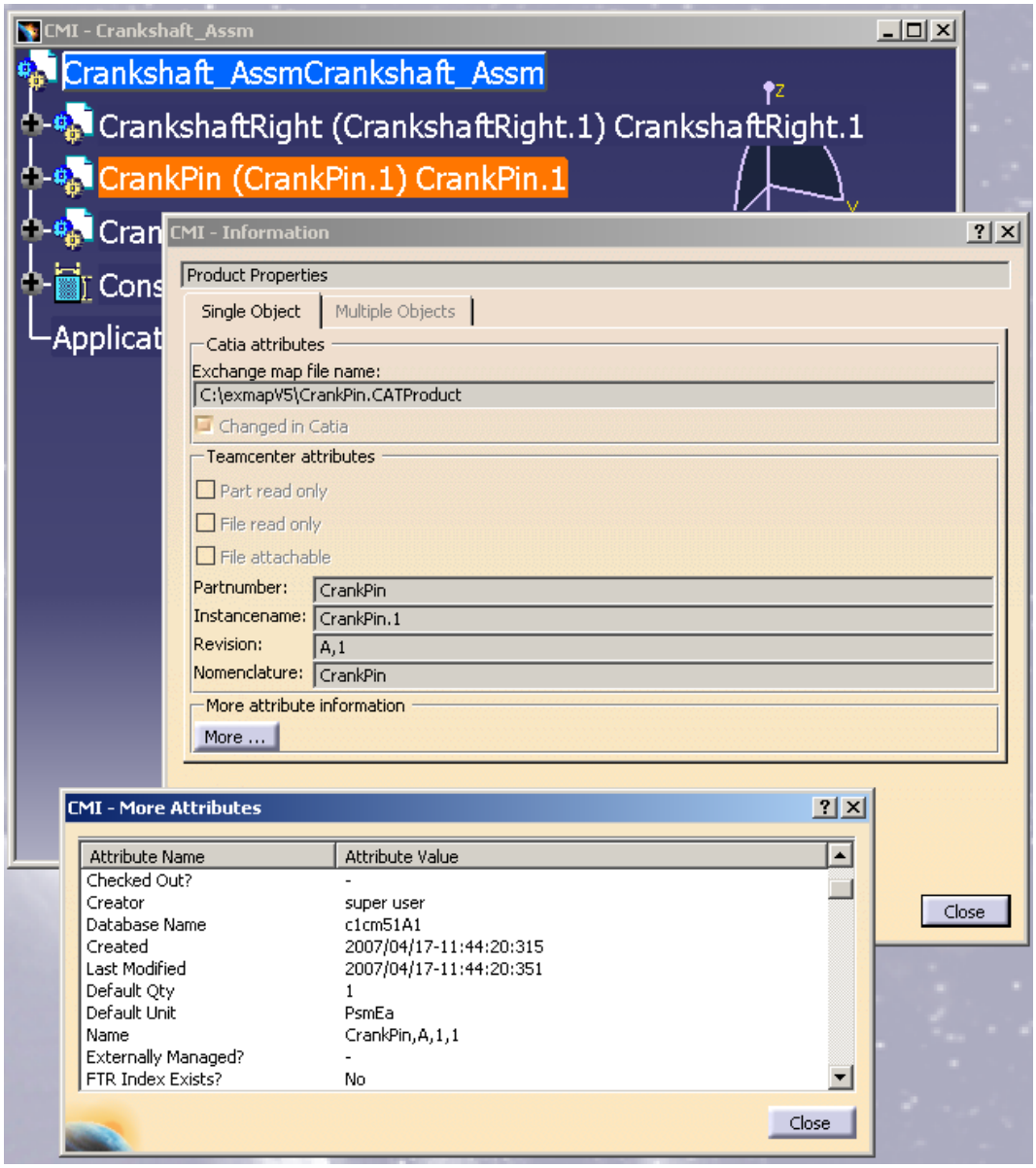


Figure 84: Information dialog window for CATProduct with "More Attributes"

Select the CATPart, CATProduct, CATDrawing or 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.

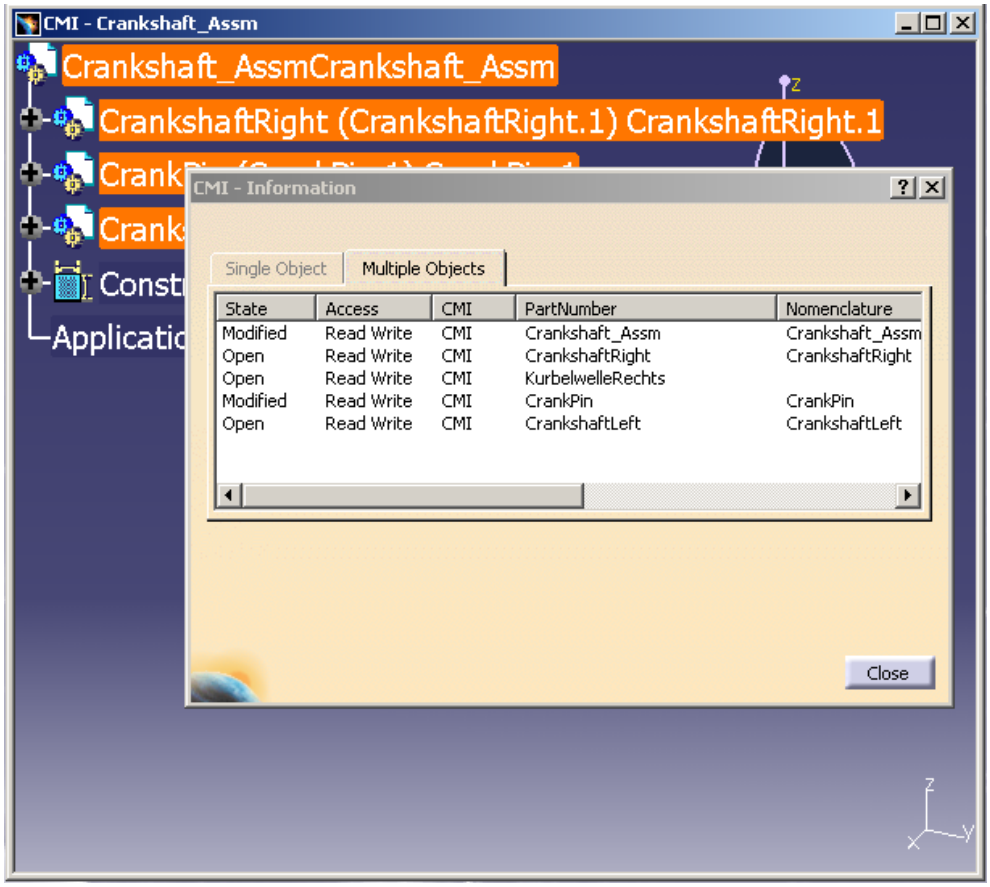


Figure 85: CMI Information for multiple items

CMI toolbar: Add Temp

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):

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

In CATIA V5 select the **'Add Temp'** icon .

Drop the assembly or model you want to add temporarily to CATIA (the suitcase) into the CATIA Workbench. Expand it as necessary. Click Action > To CATIA in the Workbench menu. Changes/expands you make to other assembly structures in the Workbench will not be sent to CATIA.

Instead of an Assembly, you can also drop a *Saved Workbench Session*

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 CATIA Workbench after you send it to CATIA.

All Partnumbers and Filenames 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 customisable by the customer.

Limitations:

- Product are not transferred to CATIA, the products will be represented by “CATIA Components”, only position information from Teamcenter is processed.
- Standard Parts are not supported.

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**

You will find that it is opened in Catia underneath a product tagged “CMI Workbench”

In CATIA V5 select the **‘Add Temp’** icon **.**

Find and expand Revision A of your assembly. Send it “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: Compare Version

Similarly to Add Temp, Compare Version is used to load different versions of the same assembly. However in this case, the version that is already open is read only and the version that is added can be modified.

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 A of your assembly as described in **CMI toolbar Read**

You will find that it is opened in Catia underneath a product tagged “CMI Workbench”

In CATIA V5 select the ‘Compare Version’ icon .

Notice that Revision A is renamed with the “TMP#_” prefix. Revision A can no longer be modified and saved in Teamcenter.

Find and expand Revision B of your assembly. Send it “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.

Use *Synchronize Teamcenter* to update Revision B of the assembly. Notice that Revision A and all it referenced files are ignored and will not be modified in Teamcenter.

CMI toolbar: Insert from Teamcenter

Allows the user to load a CMI-structure under a selected product.

Read an assembly into CATIA using the **CMI toolbar Read** 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 or a model you want to use in CATIA V5 into the CATIA Workbench. Expand it as necessary. Click Action > To CATIA in the Workbench menu. Changes/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 CATIA Workbench after you send it to CATIA.

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

CMI toolbar: Replace from Teamcenter

Replace the selected product in CATIA by a different Part from Teamcenter.

Read an assembly into CATIA using the **CMI toolbar Read** command

Select a product to be replaced.

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

Drop the Part that you want to use in CATIA V5 into the CATIA Workbench. Expand it as necessary. Click Action > To CATIA in the Workbench menu. Changes/expands you make to other assembly structures in the Workbench will not be sent to CATIA.

CATIA comes to the foreground, the selected product is replaced by the part or structure from Teamcenter. The instance name and positioning information is preserved.
The dropped part/structure is removed from the CATIA Workbench after you send it to CATIA.

Via Synchronize you can save the new Use-Relation in Teamcenter. The previous relation is deleted.

Highlighting of CATParts, CATProducts and models in the CMI Workbench

If you want to highlight CATParts, CATProducts and models in the CMI workbench triggered by CATIA V5.

In *CATIA V5* select the CATIA models/parts you want to highlight in the CATIA workbench and choose the **Highlight in Workbench** command from the CATIA CMI toolbar. Now the models/parts are highlighted in the workbench as “Marked Item”.

Highlight a part/model in the CATIA workbench.

In CATIA V5 select the **“Highlight in Workbench”** icon .

CMI Advanced Toolbar: Local Save and Restore

Using *Save Local* the contents 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*, ie. you can perform Updates.
The *Restore Local* command is only available when the Catia is empty – ie. 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. Ie. 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 Advanced Toolbar: Reconnect Teamcenter

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

Other files, that you load with eg. 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 Teamcenter* to make CMI recognize these files. It will recognize Products if the Partnumber is correct, and CATPart/model files based upon their filename. If the CATPart is a BOM CATPart (Component) in Teamcenter, the Partnumber must also match.

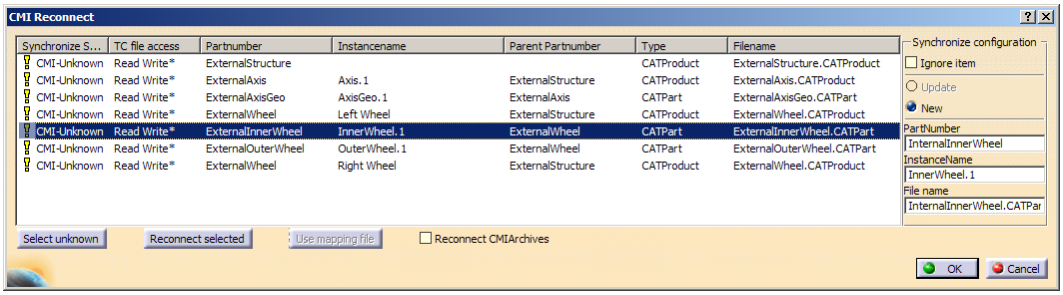


Figure 86: CMI Reconnect dialog

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.

`Can be reused:` CMI knows that 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. Synchronize would create a new instance of the existing Part in Teamcenter.

`Can be updated:` CMI knows that the object as well as this instance is in Teamcenter. It can be updated in Teamcenter.

`Can not be updated:` CMI knows that the object as well as this instance is in Teamcenter, but it is not writeable and cannot be updated in Teamcenter.

`Archive content:` The object is part of a CMIArchive.

`Broken Link:` A broken link cannot be reconnected. It will be ignored during synchronize.

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 do want to save them as new Teamcenter objects.

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

Load a CATProduct into CATIA using File→Open

In CATIA V5 select the **Reconnect 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*.

Use mapping file

If a mapping file was created when the product structure was first imported into Teamcenter, this file can be used to map the CATIA names to the names the products have in Teamcenter. Click *Use mapping file* to retrieve a mapping file that is attached to the top level Part from Teamcenter and map between external names and Teamcenter names. Import/Export using mapping files needs to be enabled by your CMI administrator. See *Optional CMI Functionality*.

Synchronize Configuration

Here you can control for individual files how they should be treated during Synchronize, as well as how they are going to be reconnected.

Update/New radio buttons

In order to be updateable, a file has to be connected with Teamcenter. If the file is connected, but you rather want to store it as a new file, click New to disconnect the file, so that it will be created as a new object in Teamcenter during Synchronize. The same allows to copy a part or data item that you have originally opened from Teamcenter.

Ignore checkbox

Ignore this Item in Synchronize (neither create a new item nor update)

Part number, Instance name, File name

If the file is archived in Teamcenter under a different name than it has in CATIA, you can suggest the name here. It will take the place of the name in CATIA when Teamcenter is searched for the item.

Reconnect CMIArchives checkbox

By default, Reconnect does not search Teamcenter for CMIArchives to reconnect a CATProduct. Check *Reconnect CMIArchives* to also search for CMIArchives. Keep in mind, however, that several different Archives can contain a root product with the same part number or file name. If this is the case, you will be asked to select one of these.

Handling of CATDrawings

CATDrawings can be included in your assembly structures in Teamcenter Enterprise. 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, dependant on the context of the CATDrawing.

Creating a CATDrawing From a Single Model

When creating a CATDrawing for a CATPart, always send the CATPart singly (without its assembly) from Teamcenter Enterprise 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 into the CMI Workbench.

Select: **Action > ToCatia**

Create the CATDrawing with the usual CATIA V5 functions.

Select the **'Create in Teamcenter'** icon

Creating a CATDrawing From a Product Structure

Drag and drop the assembly into the CMI Workbench.

Select: **Action > ToCatia**

Create the CATDrawing with the usual CATIA V5 functions.

Select the **'Create in Teamcenter'** icon

CMI can fill in Drawing title blocks with attributes from Teamcenter. This requires customization. See *Customizing Guide*.

Handling of cgr and model files

In CATIA V5 cgr and 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.

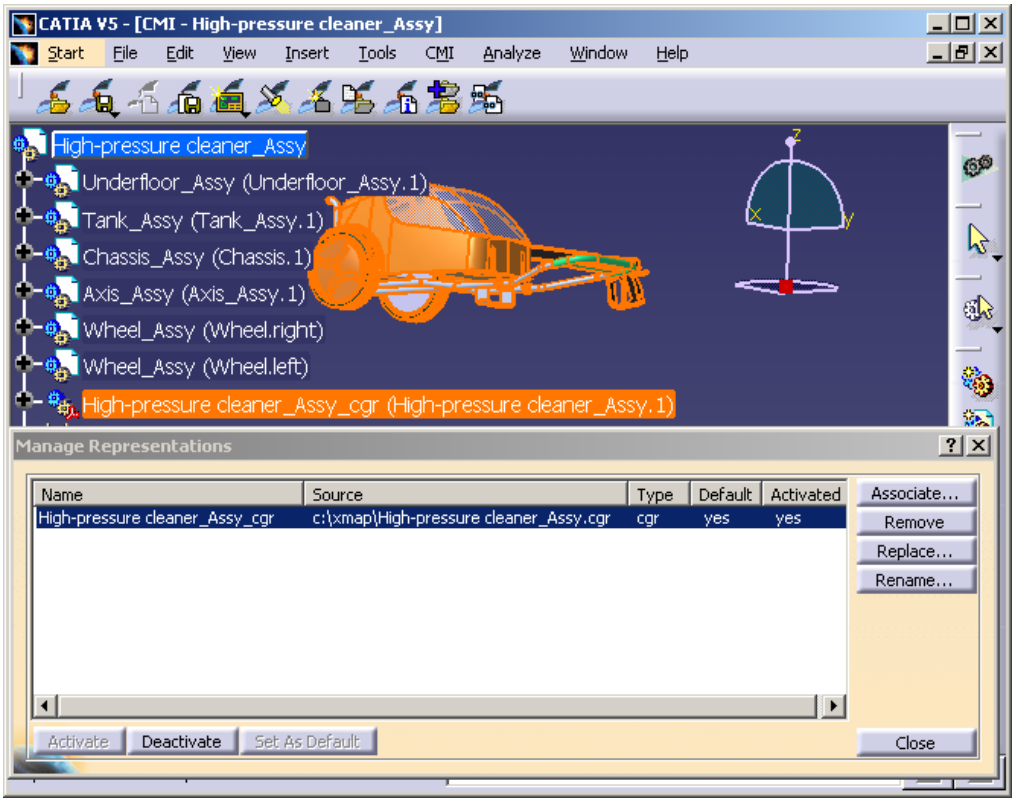


Figure 87: Manage representations dialog

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

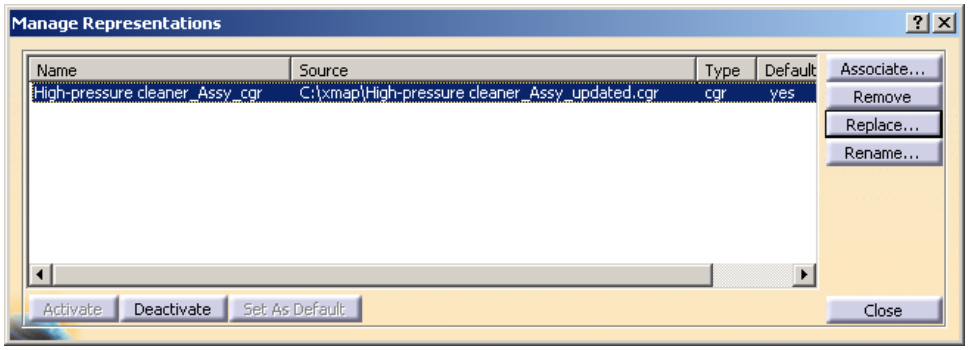


Figure 88: Manage representations dialog - Replace

Use 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.

Handling of CATProcess

CATProcess is a CATIA V5 file type. In Teamcenter Enterprise the CATProcess file can be registered with the standard Teamcenter function register (Figure 89).

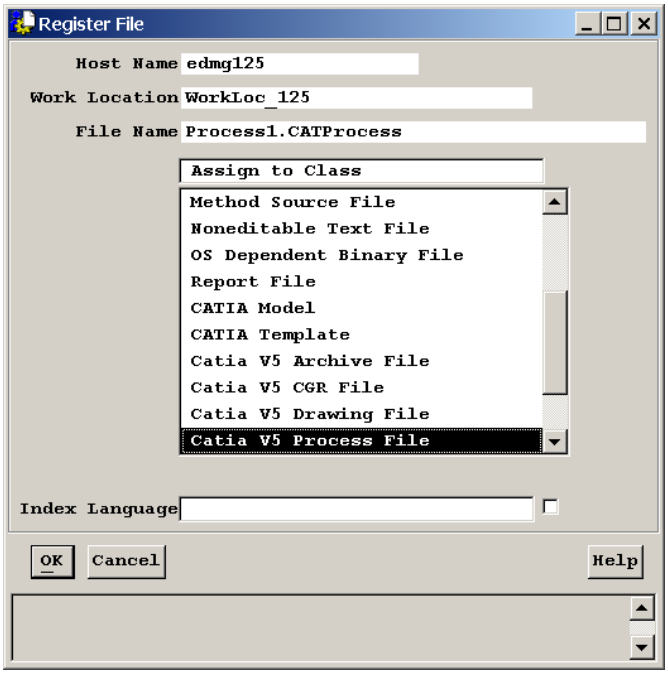


Figure 89: Teamcenter Register File dialog

After registering the CATProcess file the CATProcess data item can be attached to an Assembly. Figure 90 shows an example for a process element with an attached CATProcess data item.

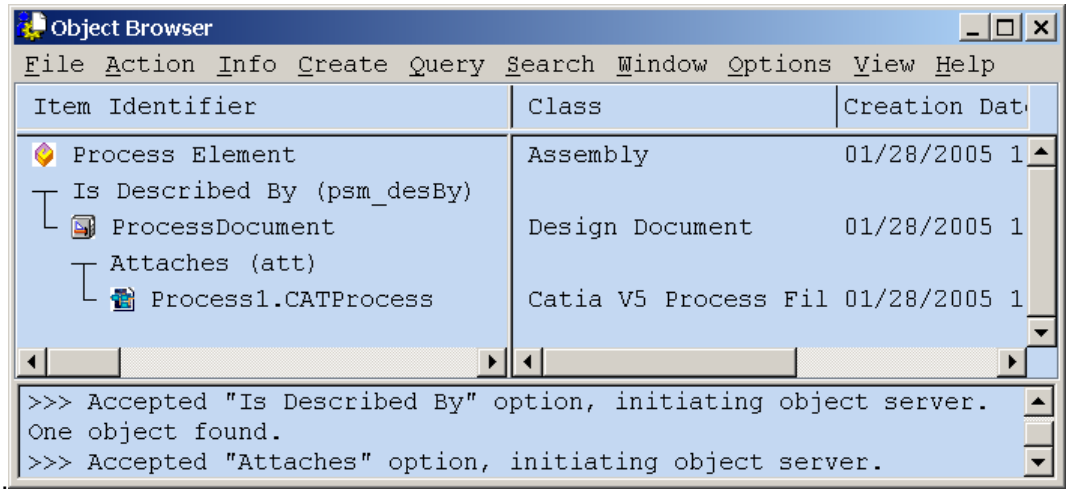


Figure 90: Part with attached CATProcess data item

After dropping the Process Element in the CATIA Workbench CMI will detect the attached CATProcess and changes the icon of the dropped Process Element. Figure 91 shows an example for a dropped Process Element.

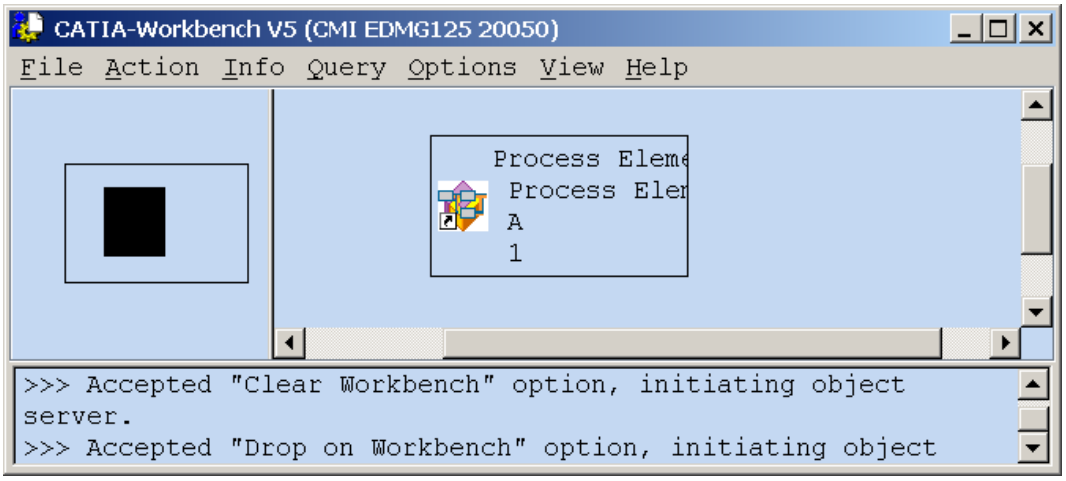


Figure 91: CATIA Workbench with dropped Process Element

After an expand of the Process Element in the CATIA Workbench the expanded Parts are by default marked as Product view elements. Figure 92 shows an expanded structure. By default all child items of the Process Element are classified as Product View. By customization this can be enhanced to Product View and Resource View. The views are later used in the CATIA V5 module to define the View where the Assembly should build in.

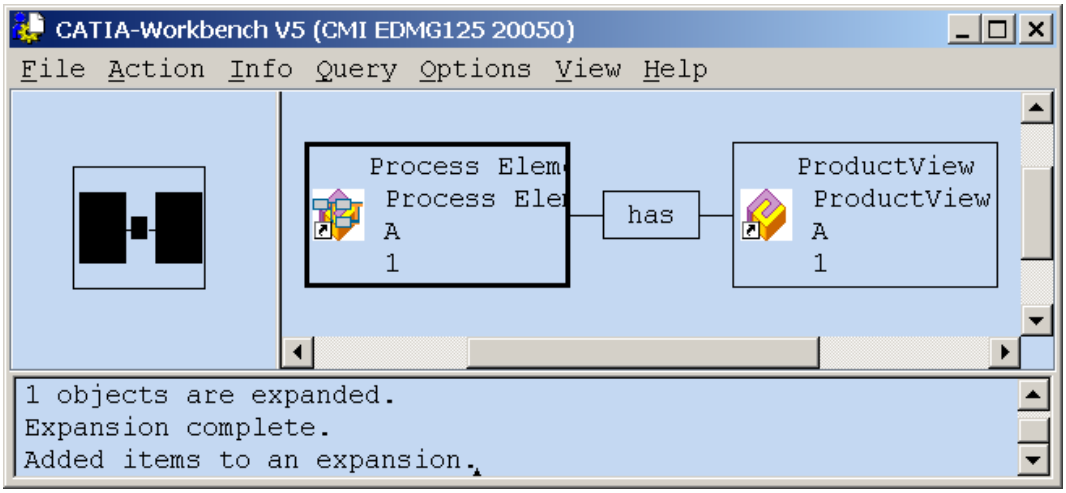


Figure 92: Expanded Process Element

After sending (Action->To CATIA or The Read Command in CATIA V5) the CMI Workbench content to CATIA V5 the CATProcess document is loaded in the CATProcess Workshop of the CATIA V5 application.

The ProductView Assembly is built into the Product List. As described above, the adding of Assemblies to the Resource list can be customized in Teamcenter. Figure 93 shows the opened CATProcess in CATIA V5.

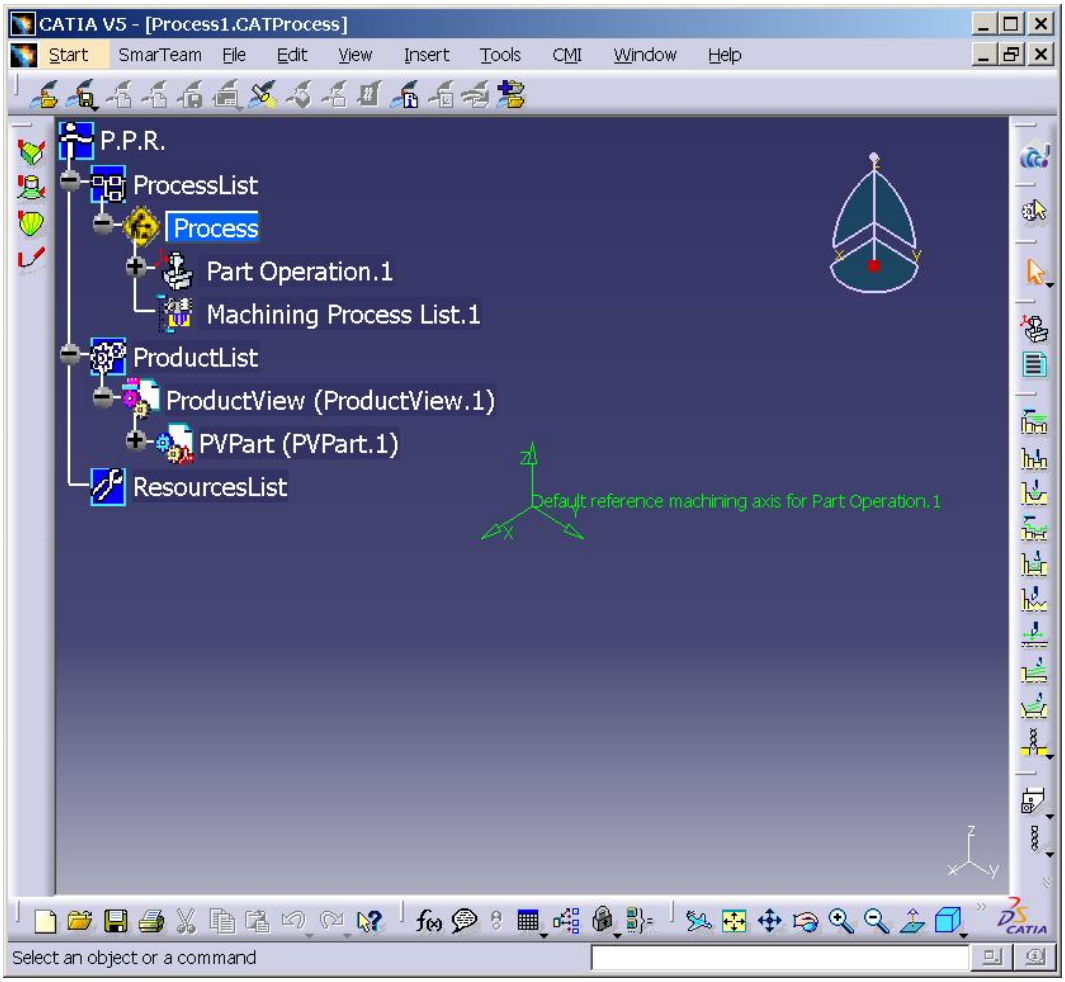


Figure 93: Loaded CATProcess in CATIA V5

Changes in the CATProcess document in CATIA can be updated with the Update Command of CMI in CATIA V5 (see CMI toolbar: Update Teamcenter). All Documents in the ProductList and ResourcesList, which are loaded from CMI will also be updated.

You can also Save a CATProcess that you have created in CATIA using *Synchronize Teamcenter*. CMI will create a new Assembly item for the Process, that references the products and resources. The CATProcess is handled much like a CATProduct.

Handling of CMIArchives

CMI supports the use of CMIArchives. A CMIArchive is a data item in TeamCenter which contains a CATIA V5 structure. The Structure has one root item which can be a CATProduct or a CATPart. If the root item is a CATProduct, it can contain more items of type CATIA V5 Component, CATIA V5 CATProduct or CATIA V5 CATPart.

In the following sections the handling of CMIArchives is described in detail.

First open the CMI Workbench in Teamcenter and drop an Assembly into the CMI Workbench (see Figure 94). After sending (Action->To CATIA or The Read Command in CATIA V5) the CMI Workbench content to CATIA V5 the product structure is loaded in the CATIA V5 application.

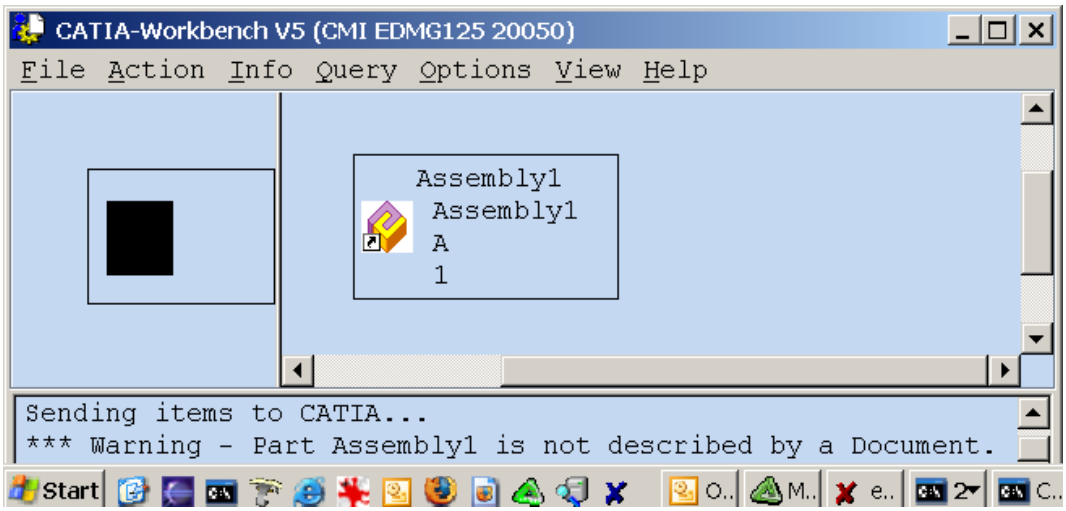


Figure 94: Assembly dropped in the CMI Workbench

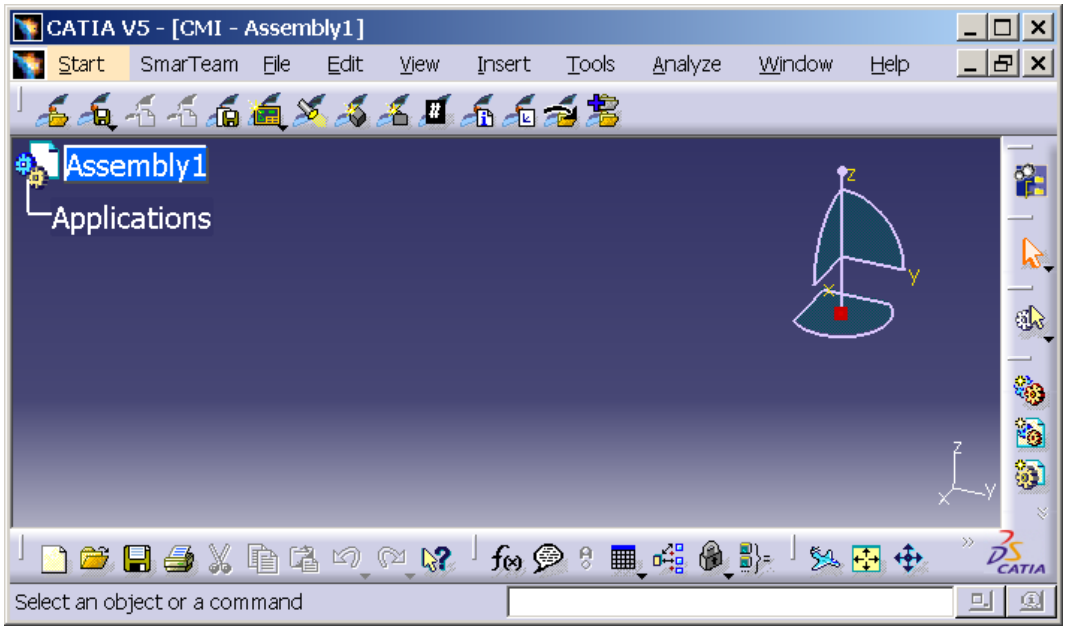


Figure 95: Loaded Assembly in the CATIA V5

Figure 95 shows the loaded Assembly Product in the CATIA V5. Now we add the Structure which should be attached as CMIArchive as child items to the Assembly1. In our example the three possible types are added under the Root item (Figure 96).

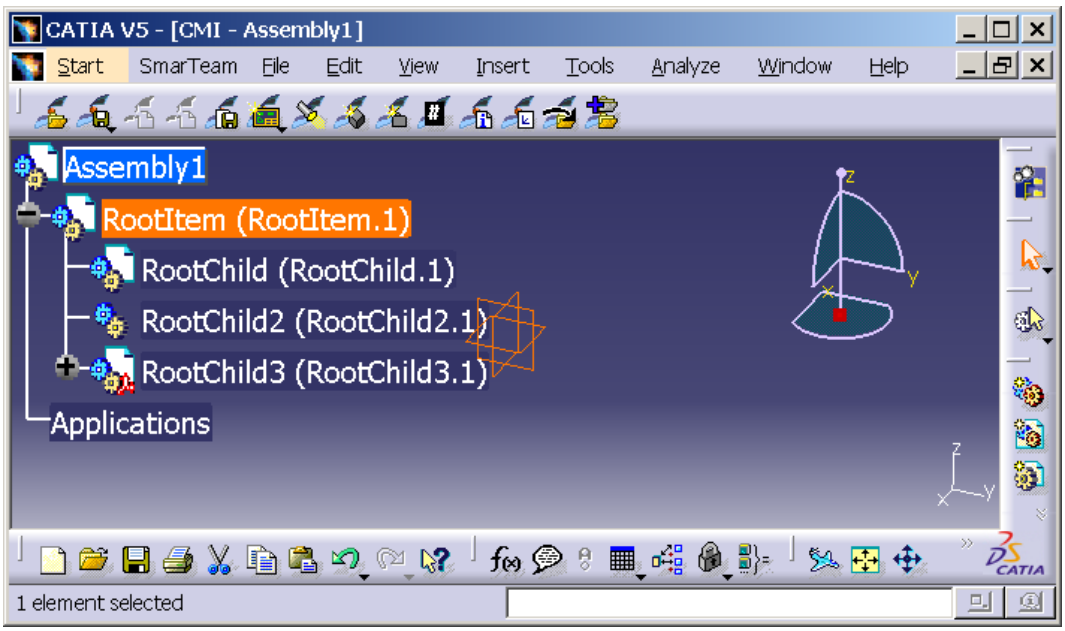


Figure 96: CMIArchive Root item with three children

Select the root Item of the Product which should be attached as CMI Archive (see “RootItem (RootItem.1)” in Figure 96). The parent object of the selected must be loaded from CMI. The CMIArchive will be attached to an existing CMI structure by using the” Attach Archive” Toolbar button.

In CATIA V5 select the **“Attach Archive”** icon.

In the TeamCenter client the Dialog shown in Figure 97 will appear.

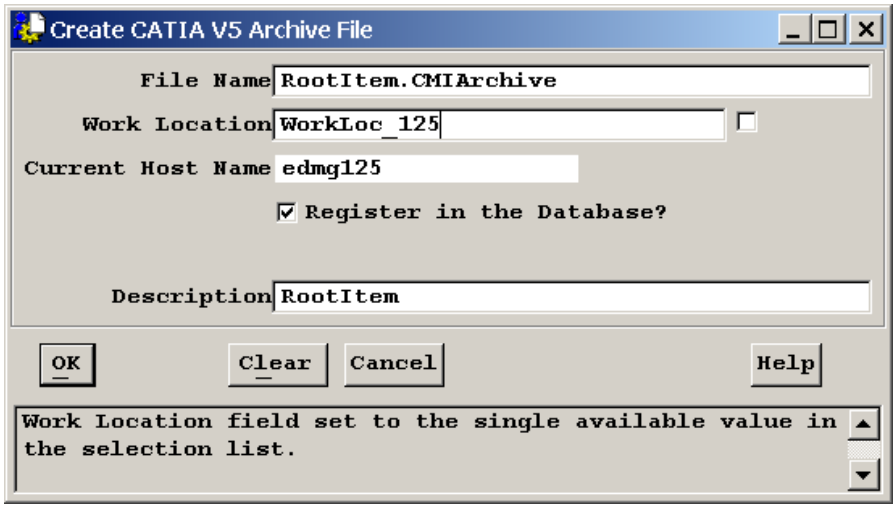


Figure 97: Create Archive Dialog in TeamCenter

After pressing the OK Button the *Prepare* dialog for the Teamcenter Document is presented to select an existing or to create a new Document for the Archive. In the

example (Figure 98) we create a new document in TeamCenter.

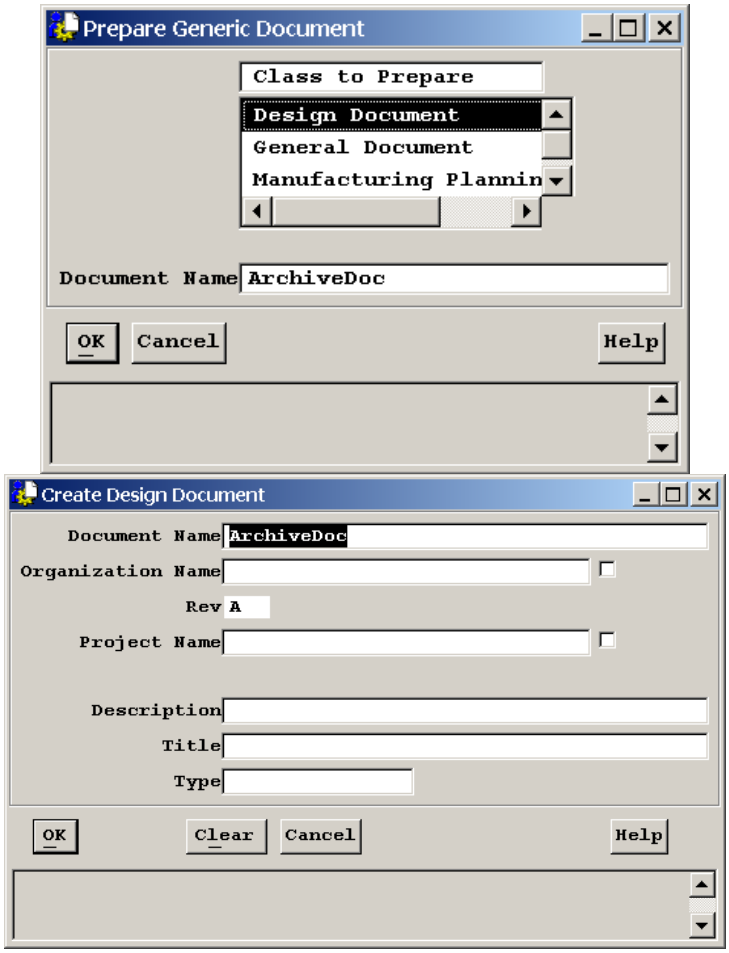


Figure 98: Document create dialogs in TeamCenter

After creating all Objects and relations in TeamCenter the User gets the Information from TeamCenter in a Message Window within CATIA V5 (see Figure 99).

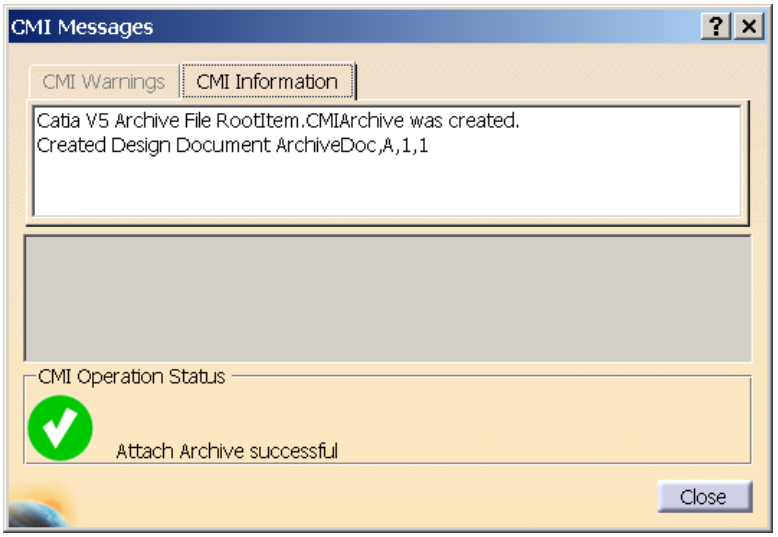


Figure 99: CMI Message Window

CMI has created a new CMIArchive data item and a document in TeamCenter. The Document is linked with the Assembly. Figure 100 shows the expanded CMIArchive after the “Expand Models” Option was used in the CMI Workbench.

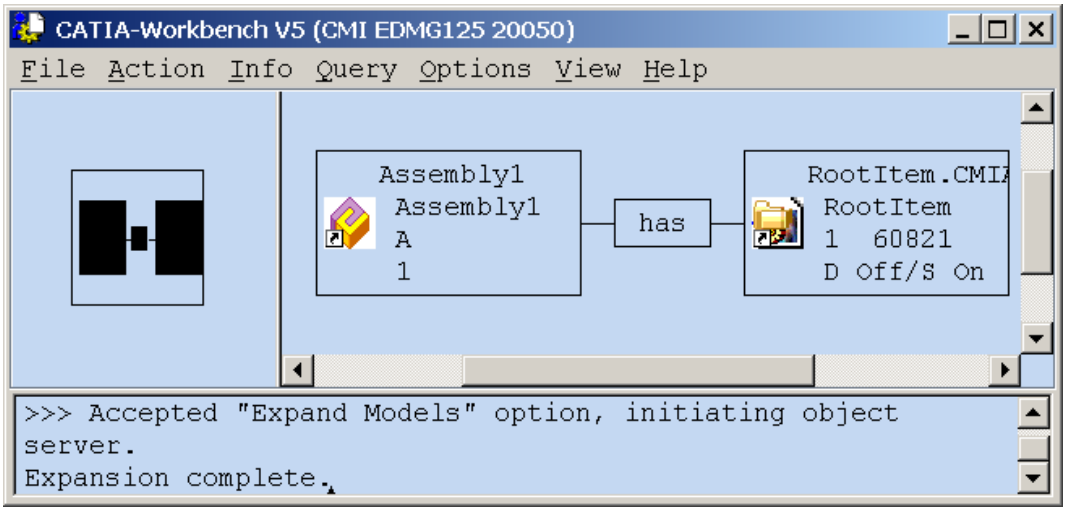


Figure 100: Expanded Archive Item in the Workbench

In CATIA V5 the TeamCenter information is attached for the CMIArchive. The CMI info dialog shows the Information for the CMIArchive object in CATIA V5.

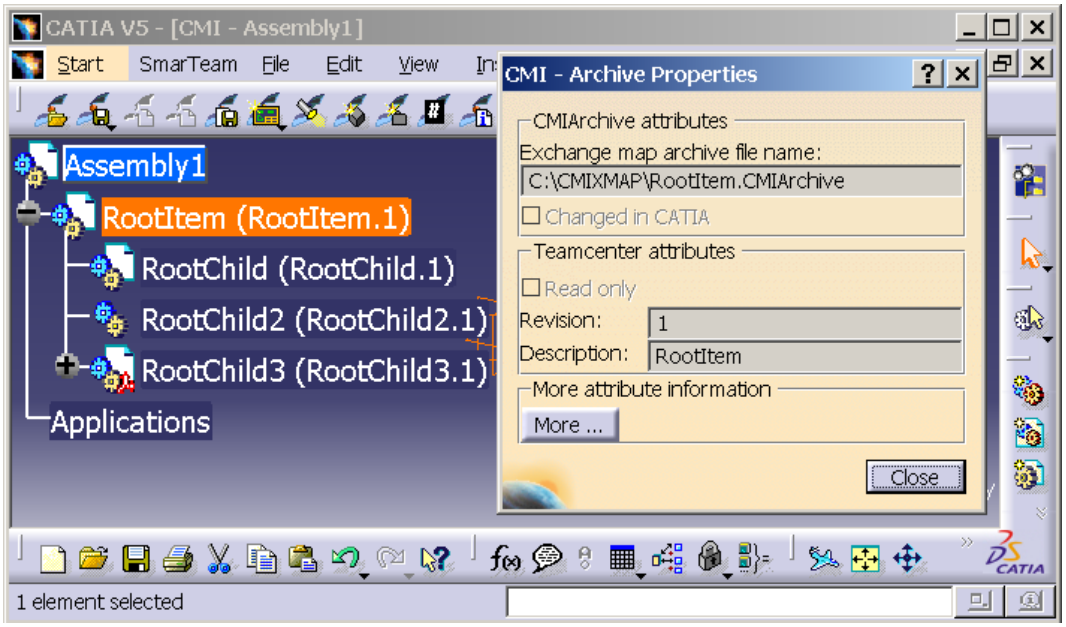


Figure 101: CMI Archive Info

The CMIArchive can be modified or extended, e.g. by adding new CATParts. The changed CMIArchive can be updated in TeamCenter with the Standard CMI Update Commands.

Use BOM Parts in CMI Archives

It is possible to use BOM Parts from Teamcenter within an Archive. To enable this functionality, the administrator should set the following variable in the CATIA V5

environment:

CMI_ARCHIVE_BOM_CHILDREN = ON

BOM Parts can only be inserted into already existing Archives.

Load an existing Assembly from Teamcenter.

Insert the Root product of the product structure you want to become a CMI Archive.

Use *Create/Attach Archive* to create the CMIArchive.

Load a Part or Assembly from Teamcenter and insert it into a product that is part of the CMI Archive; or use the *Insert from Teamcenter* command.

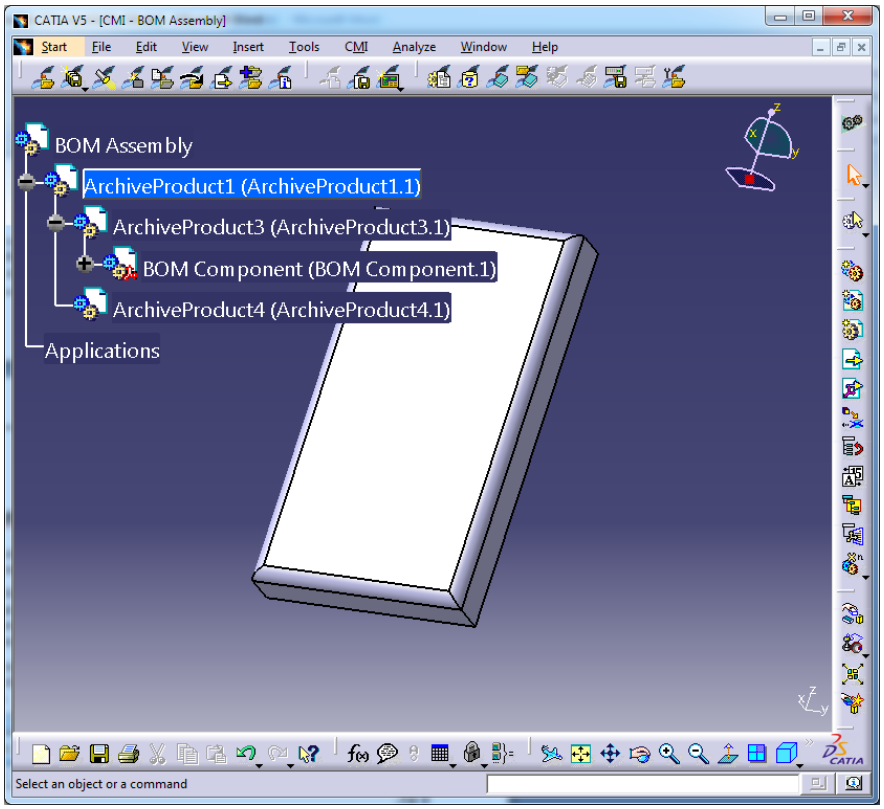


Figure 102: CMIArchive in BOM Assembly

An Archive with BOM Parts can only be updated with *Synchronize Teamcenter.* CMI will create an Assembly Structure relation between the BOM Part and the Assembly that attaches the Archive.

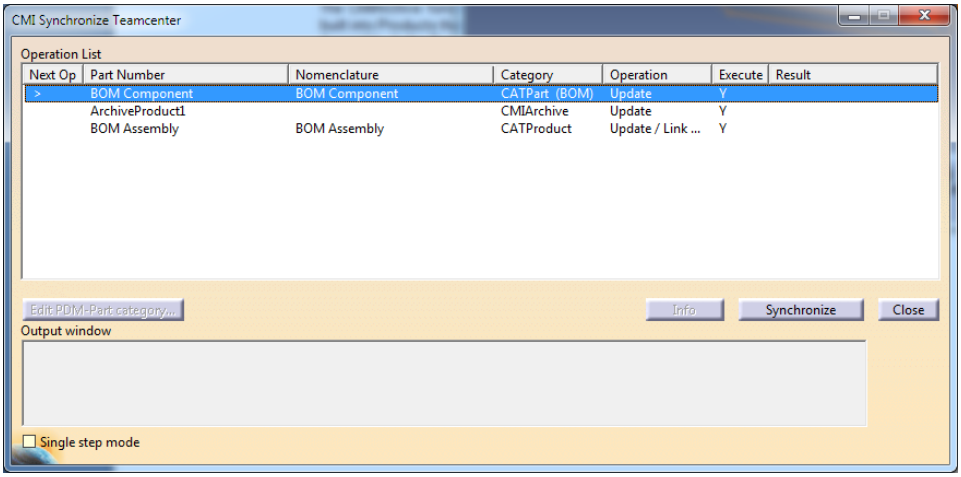


Figure 103: Synchronize dialog with BOM CMIArchive

For that reason you can only attach BOM Parts if the Archive was loaded together with its Teamcenter Assembly.

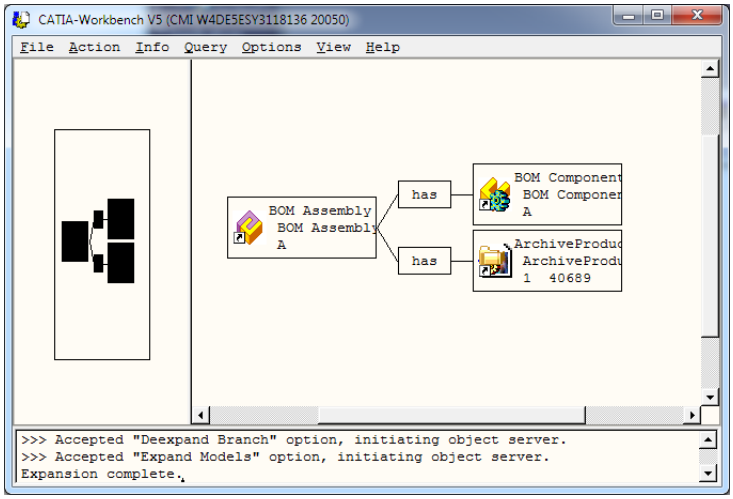


Figure 104: BOM Assembly with CMIArchive

The transformation (positioning information) stored in Teamcenter for this part instance is a summary of the transformations that are hidden in the CMIArchive; so the positioning eg. in a DMU is correct relative to the Assembly.

Since in CATIA it may not be obvious which parts of the product structure are stored in the CMIArchive and which are individual parts in Teamcenter, the ability to use a BOM Part under a CMIArchive should only be used for specific use cases, where this is implicitly clear to the user.

One use, for instance, is to build supplier Parts into an OEM structure, where differenciation may be possible by the different naming conventions.

Handling of CATAnalysis

Store a CATAnalysis file

CATAnalysis files can be stored in Teamcenter as a DataItem The Analysis is attached to the Teamcenter Part of the product or CATPart that it is referencing. It is handled in a similar way as a CATDrawing – normally the Analysis files are not loaded; it is loaded if it is explicitly expanded in the Workbench.

Unlike with a CATDrawing, the assemblies referenced by the Analysis are updated if you use the *Synchronize Teamcenter* function in the Analysis Workshop.

To manage CATAnalysis as a native file, your administrator should add the following setting to the CATIA environment:

CMI_STORE_ANALYSIS = ON

Load an Assembly or Part from Teamcenter.

In CATIA, generate the Analysis.

Use *Synchronize Teamcenter* to store the CATAnalysis file in Teamcenter.

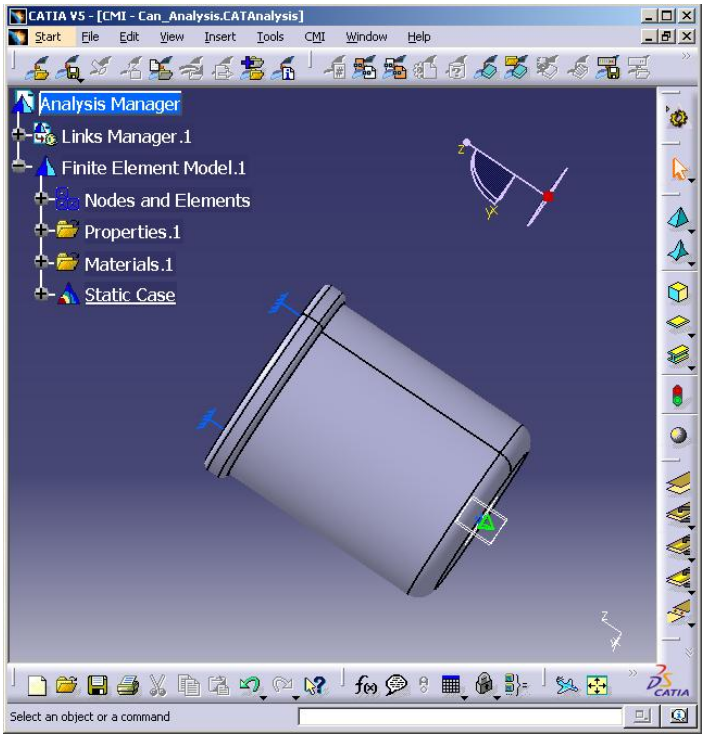


Figure 105: CATAnalysis

Synchronize will also update the Assembly the Analysis is referencing.

In Teamcenter, the Data Item for the Analysis will be created and attached to the Document of the Part or Assembly for which the Analysis was generated.

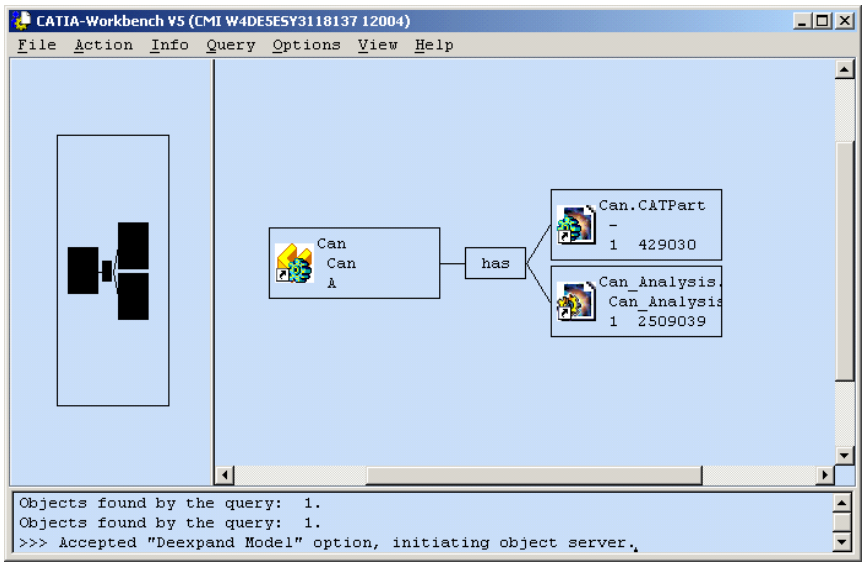


Figure 106: Analysis

To read the Analysis, Drop the Part into the CMI Workbench and expand the Analysis with Expand V5 Analysis.

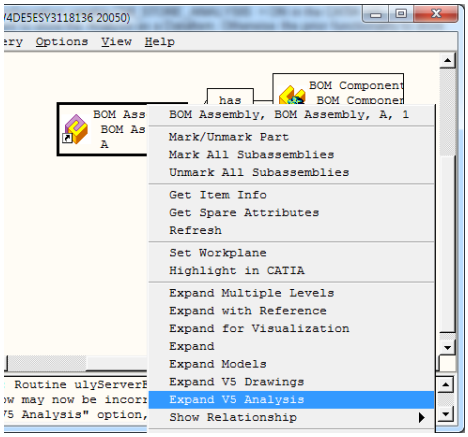


Figure 107: Expand V5 Analysis

You can also drop the Analysis file into the Workbench, but in order to update the Analysis results you should load and expand the Assembly along with its Analysis.

Analysis using a CMIArchive file

The CATIA V5 data type CATAnalysis can also be stored in a CMIArchive along with a snapshot of the product structure (see also Handling of CMIArchives).

Read an existing product structure document to CATIA V5 (Figure 108).

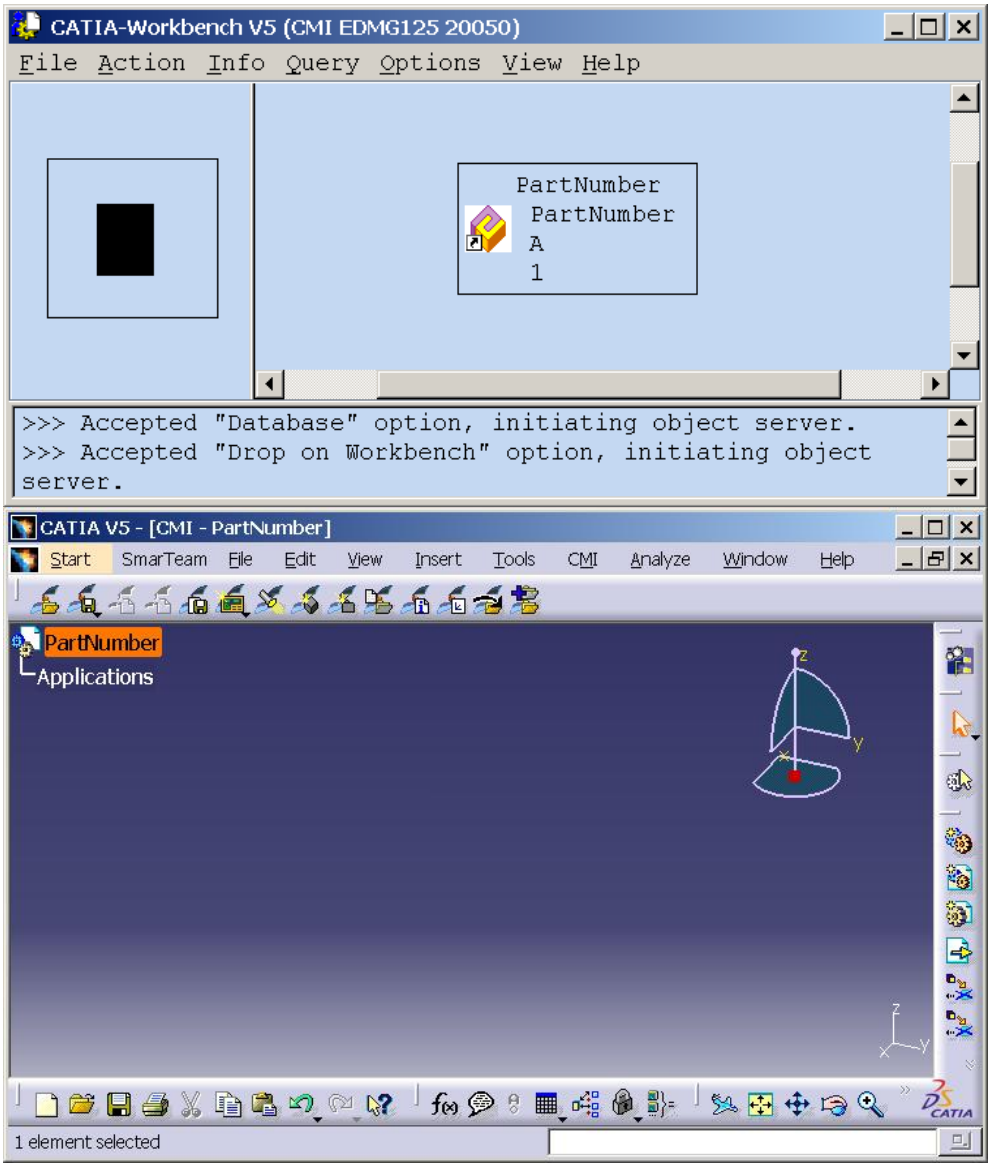


Figure 108: Existing product structure element loaded by CMI

Attach an existing CATAnalysis file with the CATIA V5 functionality *add existing component* (Figure 109).

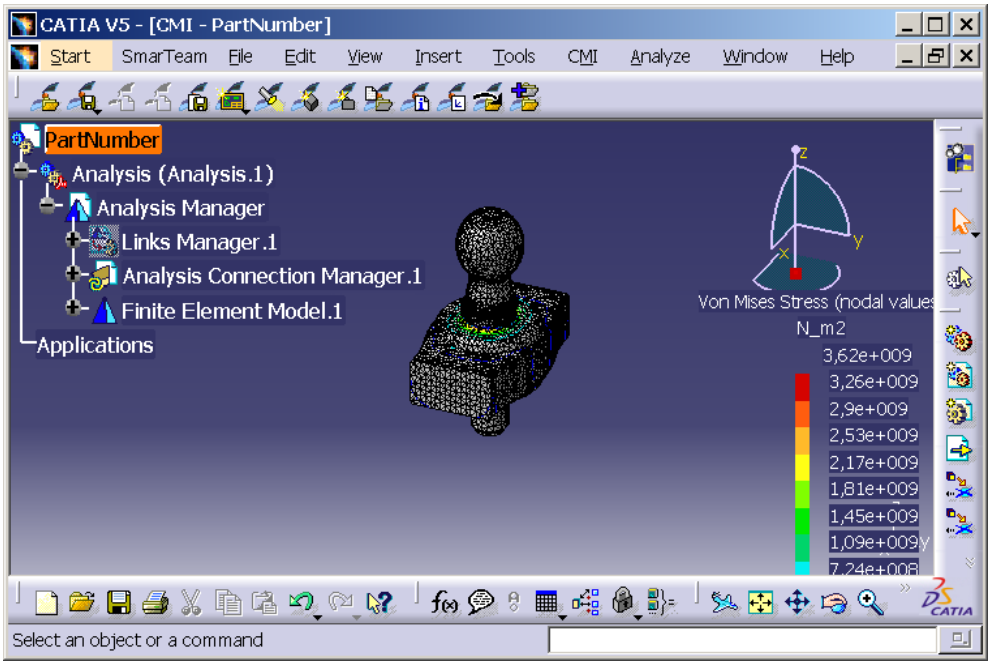


Figure 109: CMI Product with existing CATAnalysis.

Select the Analysis in the Product structure and use the *Attach Archive* Function (see Handling of CMIArchives).

The result is an new CMIArchive with the CATAnalysis file, CATProducts, CATParts, and the optional CATAnalysisResults file, which is attached to the existing PartNumber. Figure 110 shows the PartNumber with the new CMIArchive. attached to it.

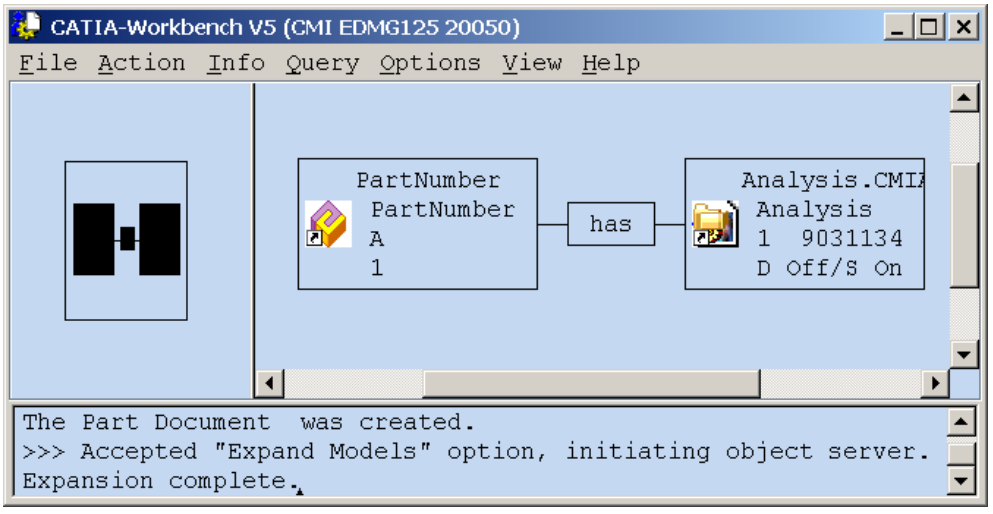


Figure 110: CATIA Workbench after Expand model on PartNumber

The newly created Archive can be dropped into the CATIA Workbench and loaded to CATIA with the *Read* command.

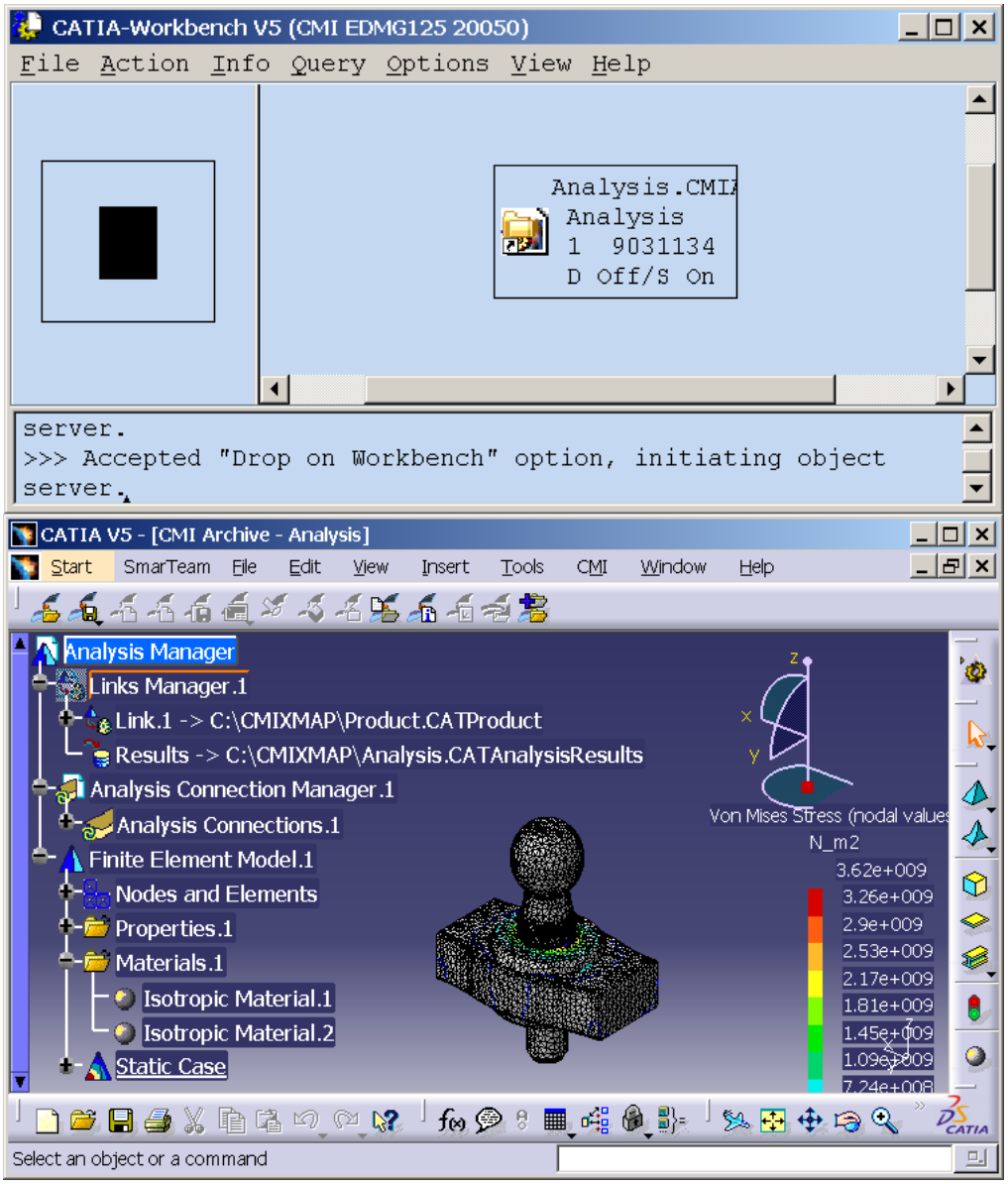


Figure 111: Existing CMIArchive with CATAnalysis loaded by CMI

The result is the CATAnalysis document in CATIA V5.

Changes made by the user can be updated with the standard CMI-Update commands.

If you only have the V5 GPS product but not a GAS license, you can still store a CATAnalysis in a CMIArchive.

Your administrator should add the following settings to the CATIA environment:

CMI_ENABLE_CRE_ANALYSIS_ARCHIVE=ON

CMI_ANALYSIS_ARCHIVE_OWN_WINDOW=ON

With these settings, you can create a CMIArchive out of the “Analysis Manager”. The second setting ensures that CATAnalysis opens in a separate window, even if the CMIArchive is attached to an Assembly.

Working with CATIA V5 Released Cache

CMI supports the use of CGR-files in the *release 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.

CATIA-Workbench: Options -> Change Preferences -> CMI Preference.

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

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

“Check timestamps” must be “On”.

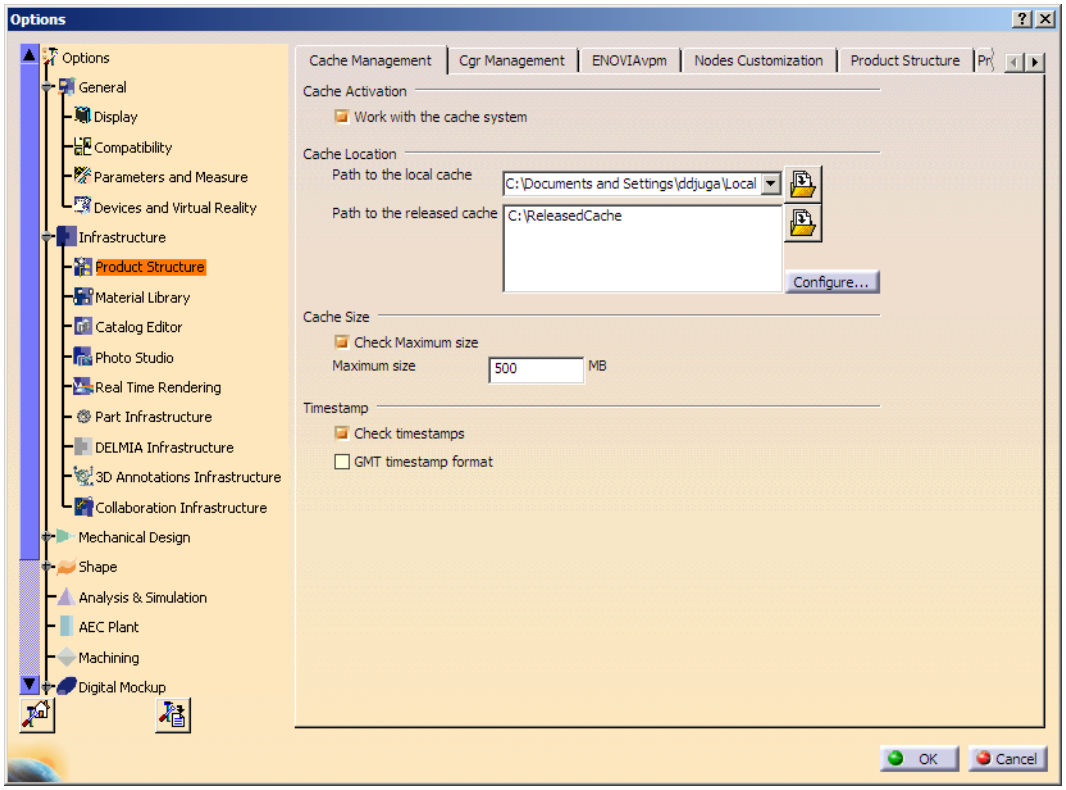


Figure 112: CATIA Options - Cache Management

For further information on the preconditions for the CGR-file-handling and for customization hints refer to the same chapter in the customization manual.

Support of external references

With the commands Update/Create/Save As/Synchronize CMI reads out the external references from CATParts and CATDrawings to other CATIA documents.
In TC Enterprise a customizing point is available to handle the CATIA V5 link information provided via CMI.

In TC Enterprise links between CATParts (CCP) can be managed. Each link of a CATPart to another CATPart is represented by a “depends on” relation.

In CATIA V5 the functionality “Get referenced documents” automatically loads the referenced documents from TC Enterprise into the current CATIA V5 session.

If you want to leverage this functionality, please contact us at cmi_support@t-systems.com

Viewer integration for JT (PLMXML) and CGR (3DXML)

With the “To viewer” command the content of the CMI Workbench can be sent to a specific viewer to obtain a quick preview out of TC Enterprise. This functionality is also available without a CATIA V5 installation! This functionality works OOTB with two file formats and protocols:

- JT format – based on the PLMXML protocol the UGS JT viewers (JT2GO *http://www.jt2go.com*; TC Visualization) are supported.
- CGR format – based on the 3DXML protocol the Dassault Systèmes viewer (*http://www.3ds.com/products-solutions/3d-for-all/3d-xml*) is supported.

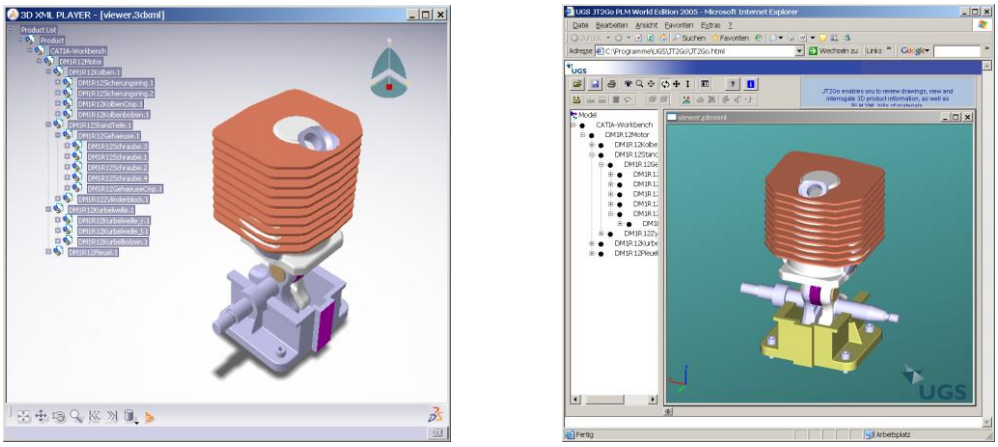


Figure 113: Integrated viewers

Prerequisites

CGR or JT files respectively must be provided by your Teamcenter PDM implementation. CMI may have to be customized to use them.
The viewer executable must be configured by the administrator.
See *CMI 8.8 Customization Manual* for the necessary steps.

Usage

Expand a product structure in CATIA Workbench. `Action-> To Viewer` will send the actual Workbench content to the Viewer.
`Action-> Marked Items->Marked To Viewer` will send the marked Items to the Viewer.

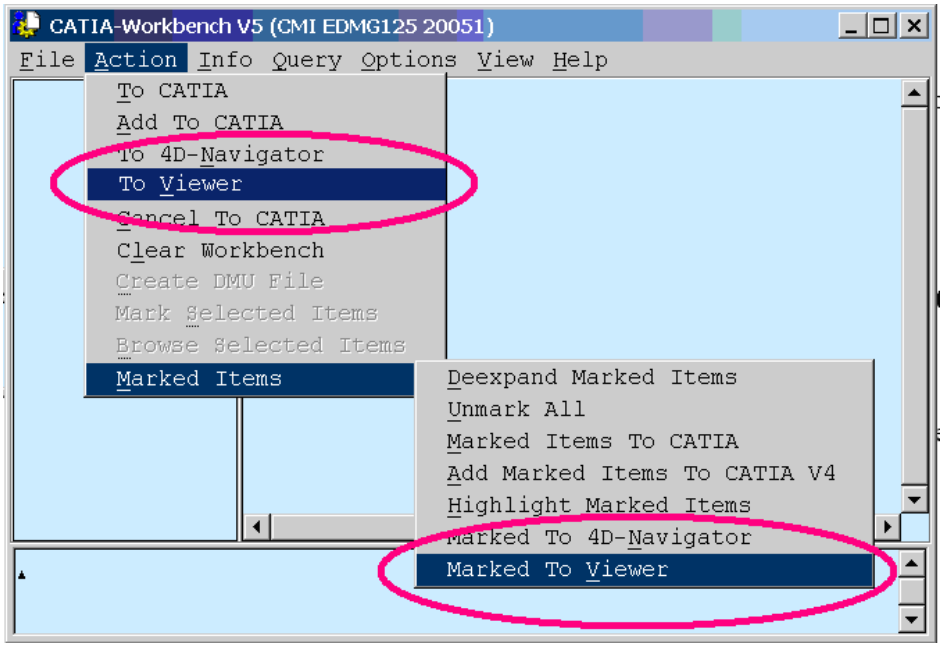


Figure 114: To Viewer action

In order to exclude the CGR that are provided exclusively for viewing, the Model Filter preference in the CATIA Workbench is extended by a *Viewing* model type (**`CATIA V5 Viewing`**) which will only be sent to CATIA if *Use model filter* is set.

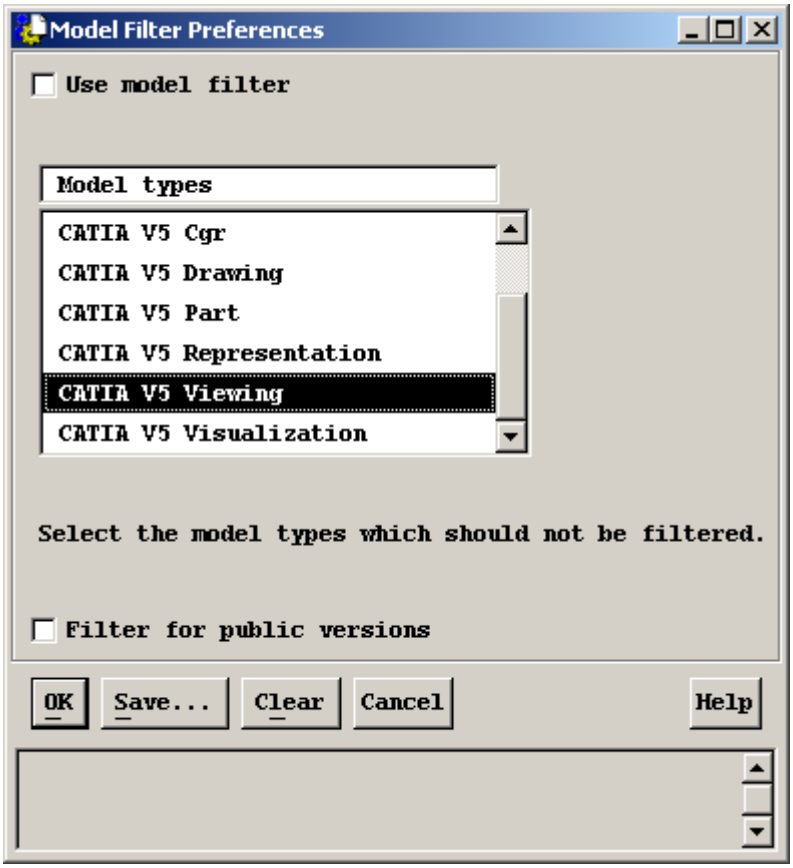


Figure 115: Model Filter Preferences dialog

Configuration

Directory and format (3dxml, pdmxml, etc.) are configurable in preferences.
New Viewer Configuration in options-> Change Preferences Menu option.

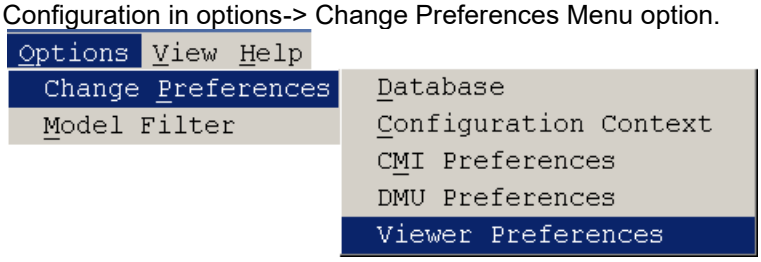


Figure 116: Change Viewer Preferences

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

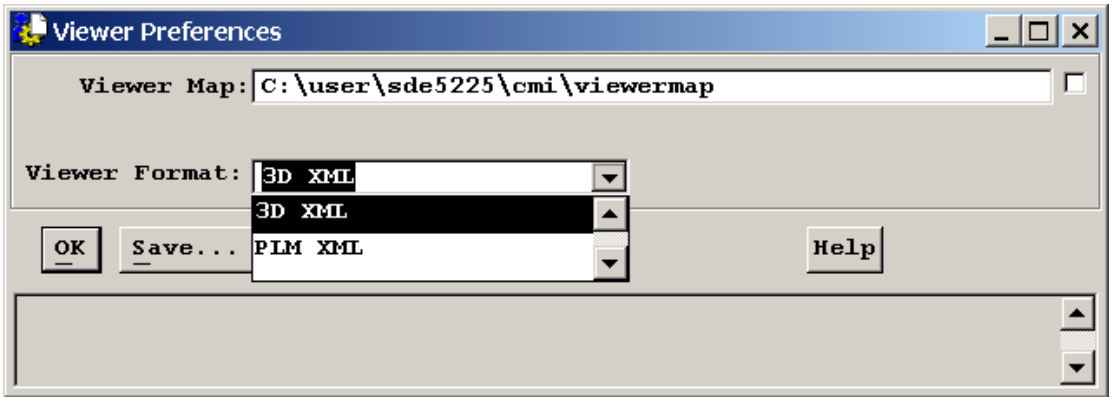


Figure 117: Viewer Preferences dialog

Interoperation with VisMockup

An assembly that was opened into VisMockup from the CMI Workbench with the “To Viewer” functionality can be imported back into CATIA V4/V5 based on configurations or selections made in VisMockup.

In order to send the configured product structure from VisMockup to CATIA (V5 or V4), you need to export it from VisMockup as plmxml. You can keep the file name.

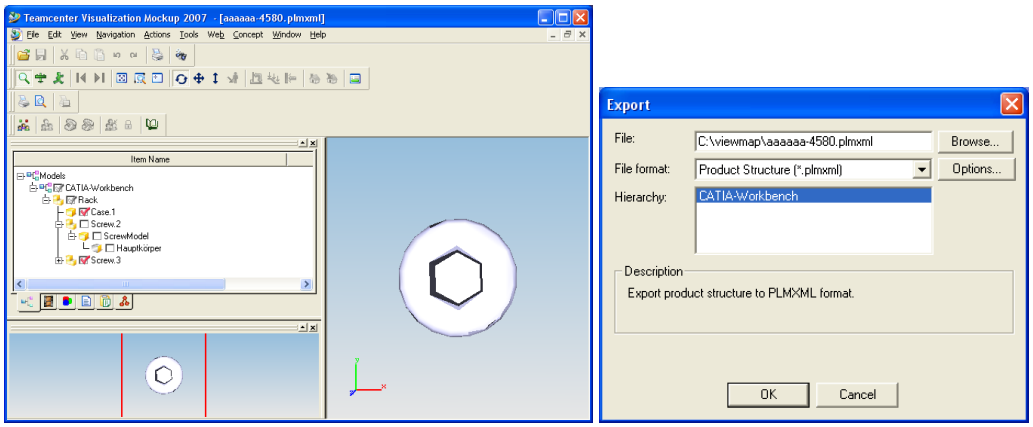


Figure 118: Export from VisMockup

In the Workbench click *File->Viewer File To Catia* to have CMI mark-up the product structure and automatically send it to Catia via the *Marked Items To Catia* functionality.

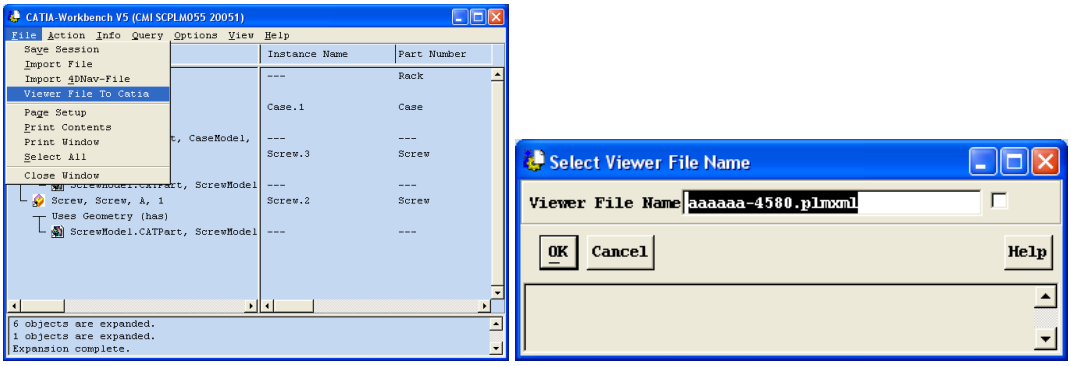


Figure 119: Select Viewer File Name dialog

Only those geometries that were selected as visible in VisMockup will be opened in CATIA.

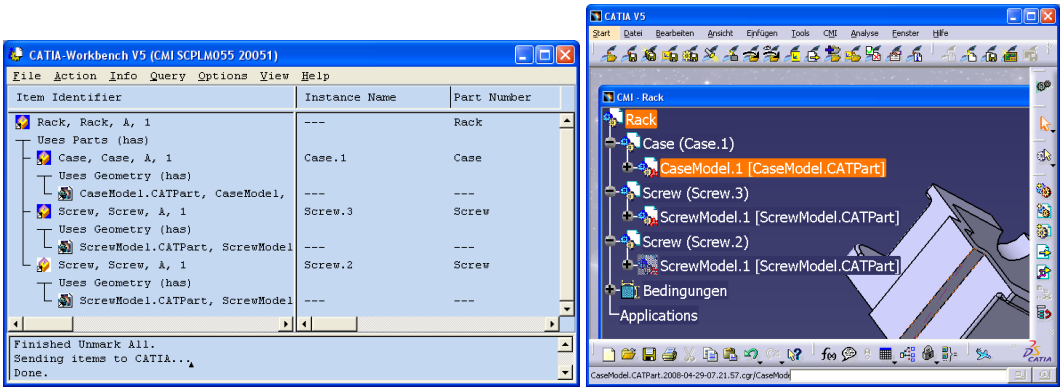


Figure 120: Opened in CATIA

Expand for Visualization

The context menu of the Catia Item representations in the CATIA Workbench contains a new option “Expand for Visualisation.

The option expands the selected Branch over multiple levels with models displayed. If a part with an attached Visualization-model is found the expand is stopped at this node and only the Visualization-model is expanded.

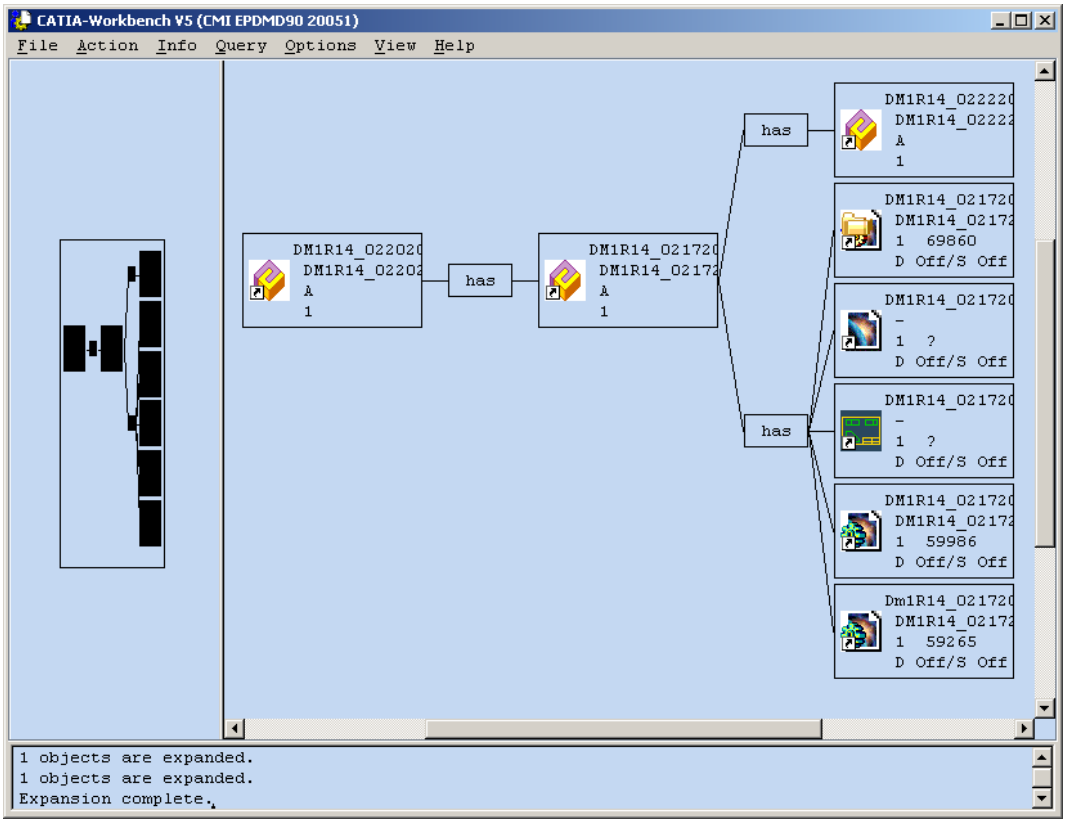


Figure 121: Result of an “Expand Multiple Levels”

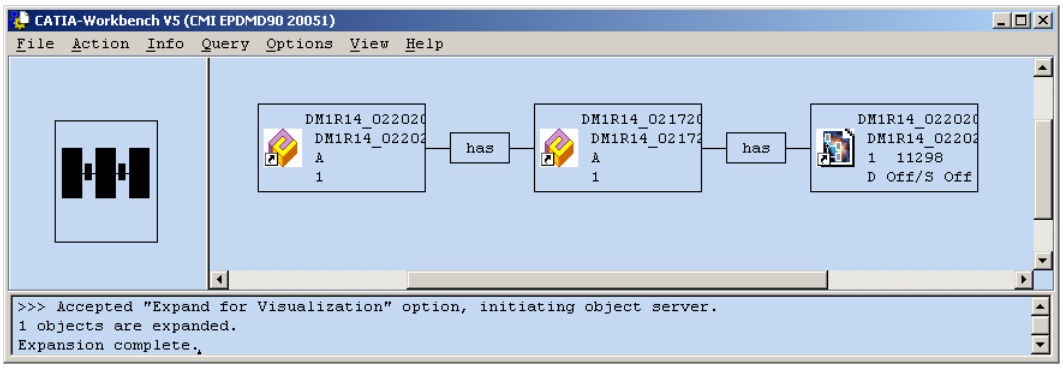


Figure 122: Result of an "Expand for Visualization"

Figure 121 shows a normal expand multiple level with all structures and models. Figure 122 shows that the second part contains a Visualisation file which represents the whole part with models and substructures.

In CATIA V5 a new menu item *Build Visualization* is available that builds up or updates a visualization file from a selected product and stores the visualization under the selected item. These cgr can then be created/updated with the CMI Synchronize command.

You can also use CATParts as a visualization file by setting the CATIA Model Type Attribute to *CATIA V5 Visualization*. CMI does not control the content or topicality of the visualization file.

Catalog Management

CMI can also manage CATCatalog files in Teamcenter. PLM compatible Catalogs reference files and Parts that are actually stored in the PDM system. If the user opens or inserts a file from the Catalog, CMI retrieves the corresponding data from Teamcenter.

By default CMI retrieves the exact revision of a Part that was inserted in the Catalog, but a mode where the Part Master is referenced in the catalog is also supported. In order to create Master-based catalogs, set

CMI_DYNAMIC_CATALOG=ON

in your CATIA environment.

Manage Catalogs

The administrator needs to set `CMI_ENABLE_CMICATALOGCMD=ON` in the Catia Environment, to enable the *Manage Catalogs* function.

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

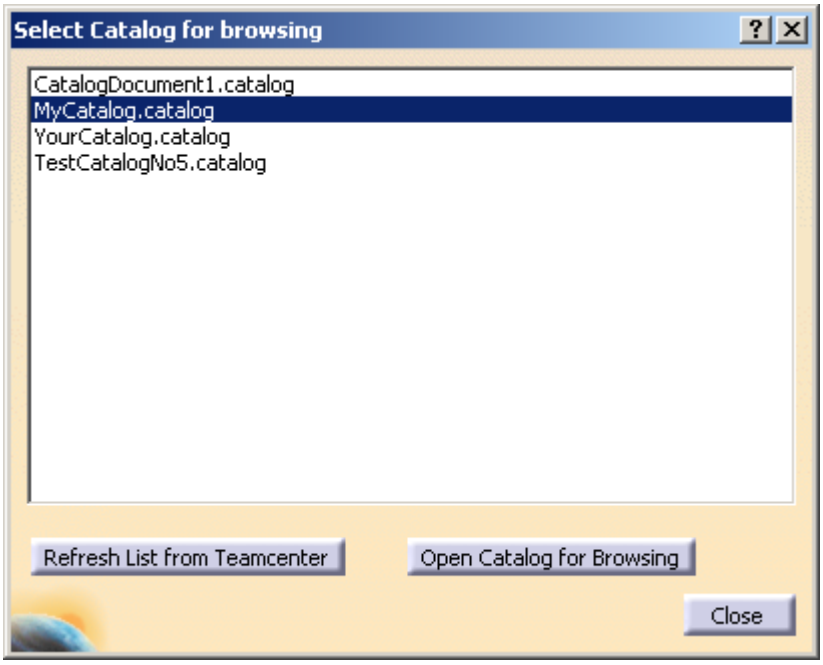


Figure 123: Select Catalog for Browsing dialog

Pick from the list of catalogs available in Teamcenter.

The catalog will be opened in the Catalog Browser.

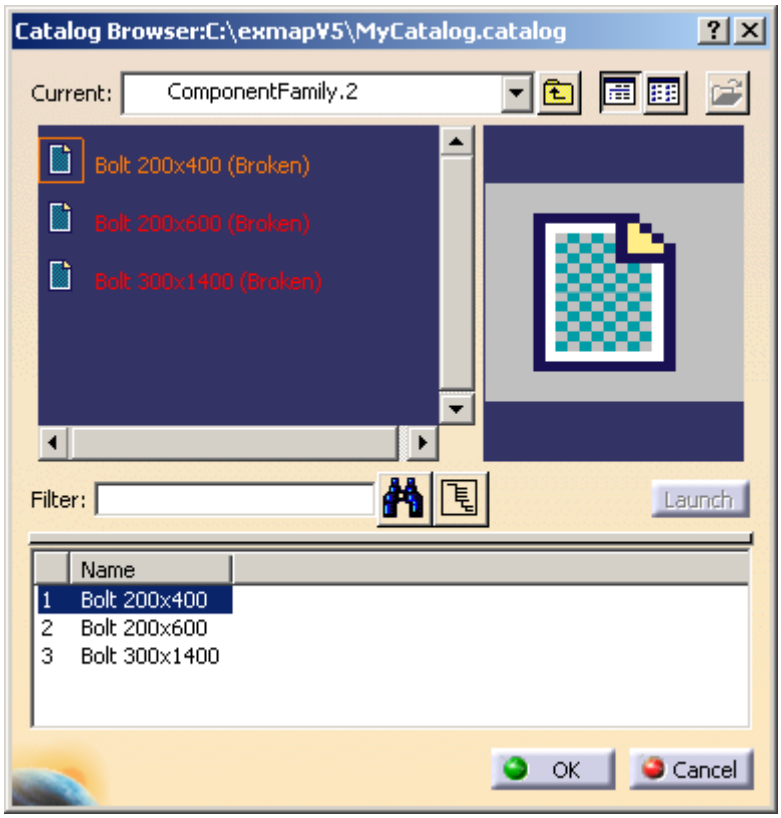


Figure 124: Catalog Browser

Double click an end chapter to open the file or Part from the PDM system. 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.
Read Catalog allows to retrieve CATCatalogs from Teamcenter for editing.

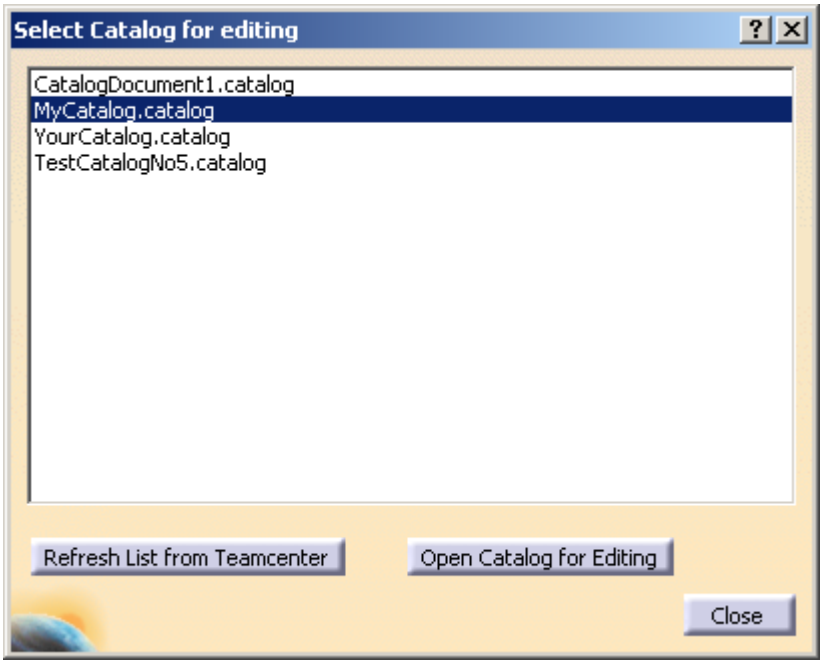


Figure 125: Select Catalog for Editing

Pick from the list of catalogs available in Teamcenter.

The catalog will be opened in the Catalog Editor.

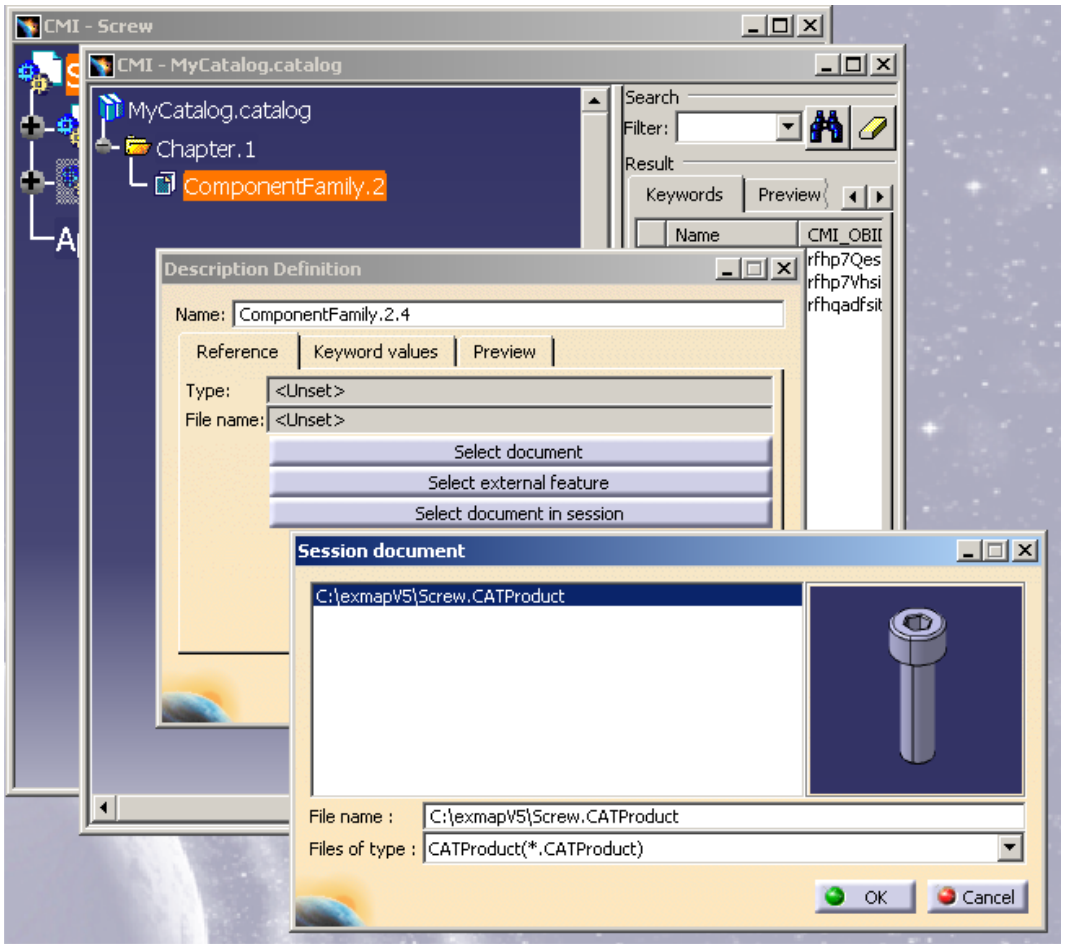


Figure 126: Select Document in Session

Use *Select document in session* 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 and Create function for CATCatalogs.

Update or Create Catalog allows to save new or updated CATCatalogs in Teamcenter. It will manage the necessary keyword attributes so that the Parts 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:

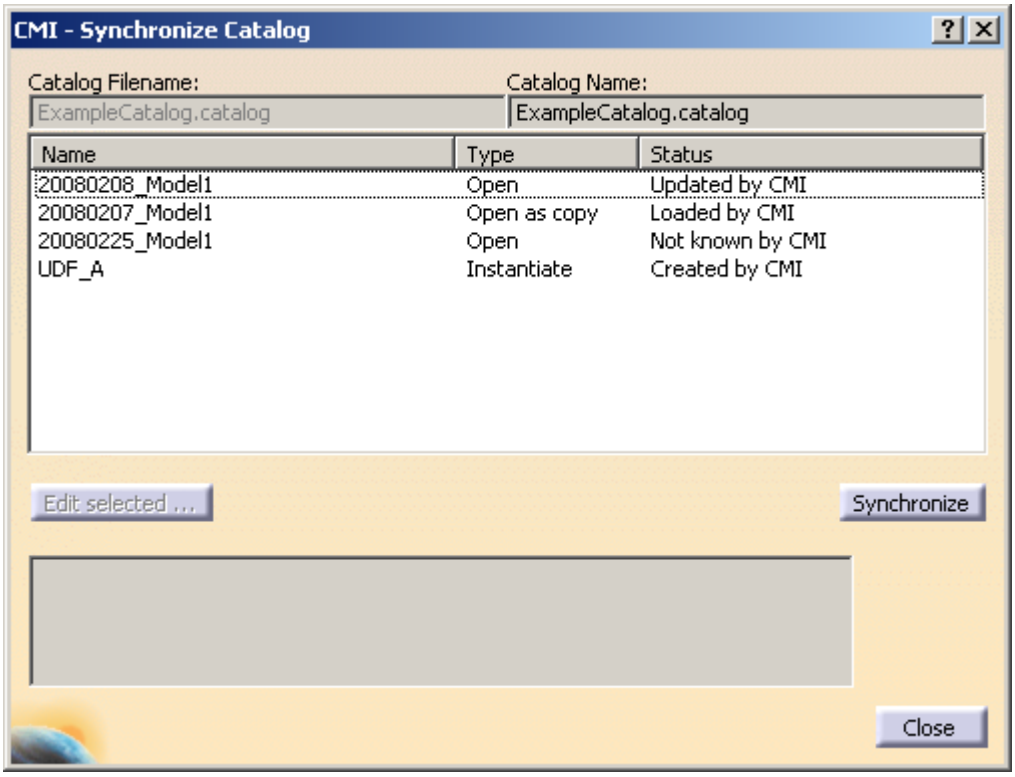


Figure 127: Catalog synchronize dialog

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 parts will be handled when the catalog is used

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

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

- Loaded by CMI: The part is from PDM and already was in the Catalog at load time.
- Updated by CMI: The Part is from PDM but the Type was changed.
- Not Known by CMI: The Part is not from PDM
- Created by CMI: The referenced File was opened from PDM and is new in the Catalog.

After Pressing the Synchronize button CMI will check all entries and if there are unknown entries, the user must confirm, that he will continue:

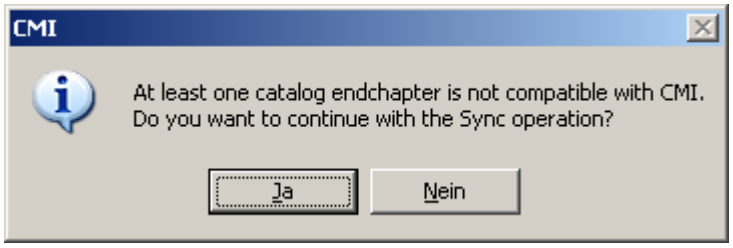


Figure 128: Confirm to continue Sync operation

This indicates that there is a Part that is not managed by PDM.

Next the Catalog is registered or updated in Teamcenter:

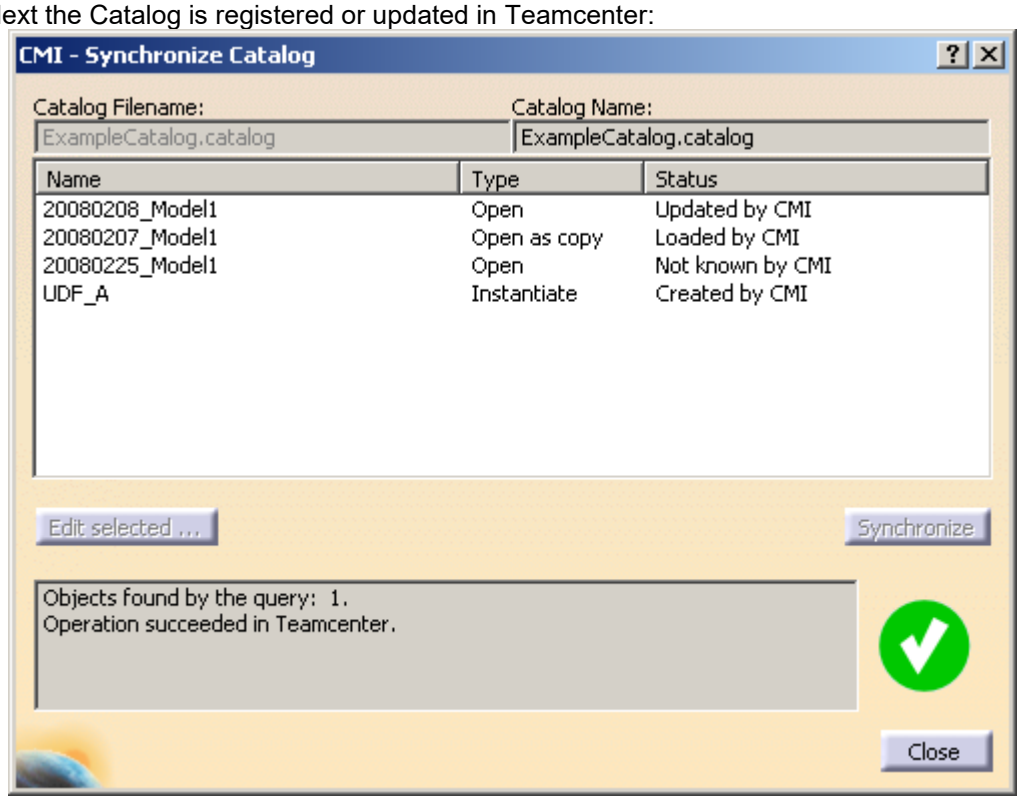


Figure 129: Successfully finished Synchronize Catalog

Using Macros / CATScripts in Catalogs

Set `CMI_ENABLE_CMICATALOGINSERTSCRIPTCMD=ON` to enable the *Insert CATScript from Teamcenter* command. This allows to reference CATScripts that are held in Teamcenter in existing or new catalogs.
To use a CATScript in a Teamcenter managed catalog the CATScript must already be registered in Teamcenter, as a *CATIA Script* class object. Copy your CATScripts to your Worklocation and register them in Teamcenter.

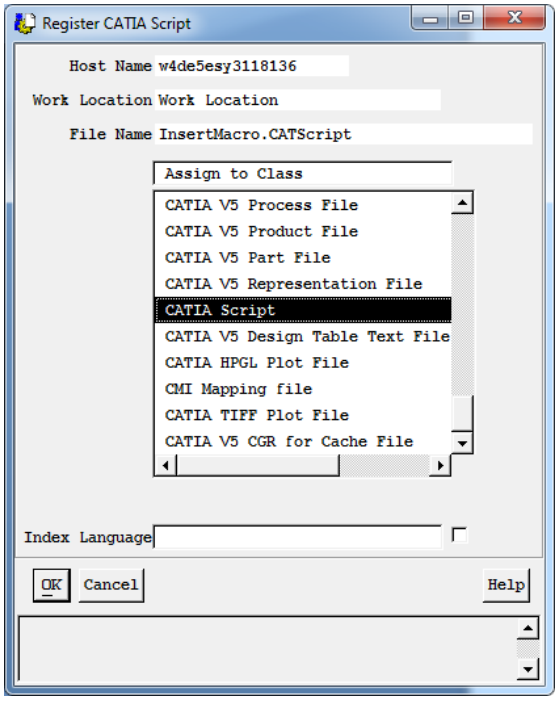


Figure 130: Register CATIA Script dialog

To insert CATScript macros into a Catalog, open the Catalog for editing. To open a Catalog that is already managed in Teamcenter for editing, use the *Read Catalog* command.
Select an end chapter. Click the *Insert CATScript from Teamcenter* command. A Teamcenter query for CATIA Scripts will appear.

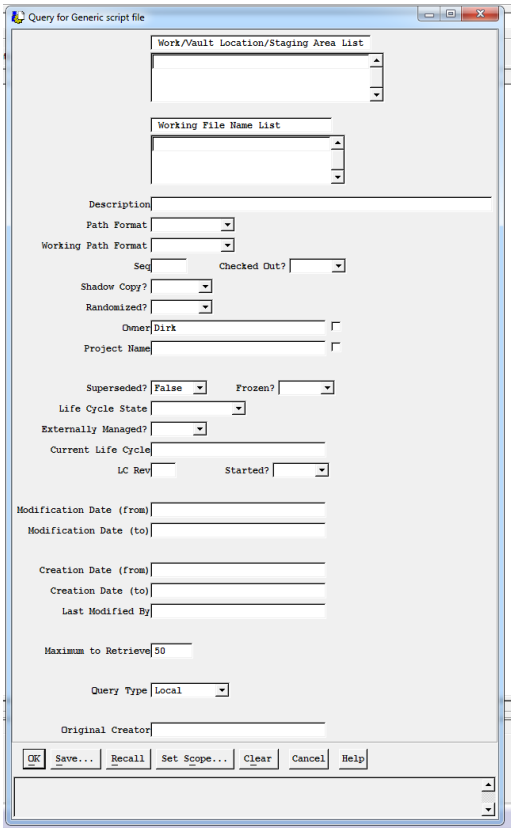


Figure 131: Query CATIA Script dialog

Pick a CATScript from the list. The CATScript will be inserted under the selected end chapter in CATIA. The keyword attributes for Teamcenter will be filled in automatically.

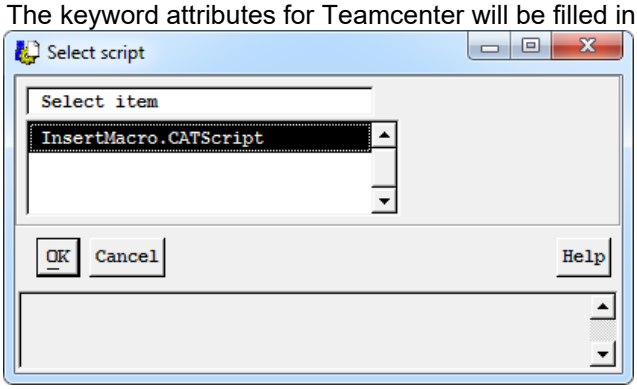


Figure 132: Select CATScript

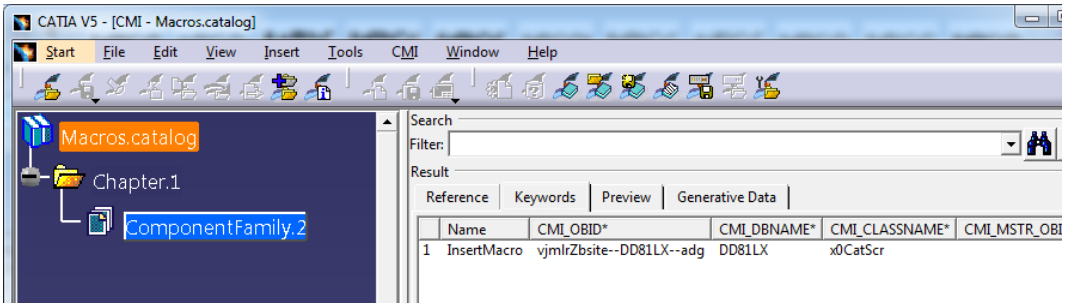


Figure 133: Inserted CATScript

If the catalog already references a CATScript and you want to connect it with Teamcenter, Select the existing CATScript reference instead of the end chapter.

Finally, to store the modified catalog in Teamcenter, use the *Update or Create Catalog* command.

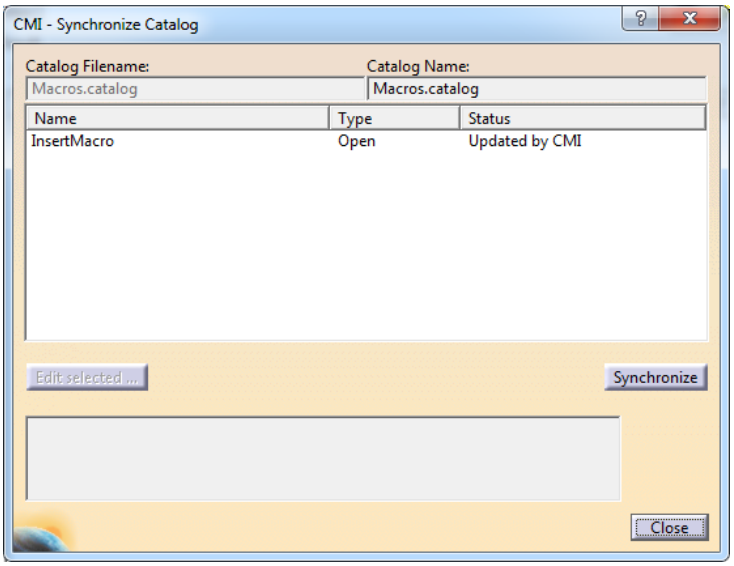


Figure 134: Synchronize Catalog dialog

Now you can use the catalog to execute the CATScript that is stored in Teamcenter, with *Manage Catalogs*.

Optional CMI CATIA V5 functionalities

There are numerous CMI features that are not enabled in OOTB CMI. Often these support specific methodologies used by individual CMI customers. These features are usually enabled by an administrator, by setting environment variables.

Automatic CATDUA

To clean CATIA files that are stored in Teamcenter with the file based CATDUA utility you can use the *Read with CATDUA* command.
In order to enable this command the following setting must be present in the CATIA environment:
`CMI_ENABLE_CMICATDUAREAD=ON`

The CMI CATDUAV5 integration needs a template configuration file to run CATDUAV5. The configuration file can be generated like described in the CATIA documentation: *"Using CATDUA V5 outside an interactive Session"*
Use the procedure described in *"Method 2: Accessing the Batch Interface from the CATUTIL / CATDUA V5 Batch Tool"* to define the batch parameters. Make sure the "Document Selection" is empty. Save the configuration file to CMI_CATDUAV5.xml:

The configuration file CMI_CATDUAV5.xml must be available on every client.
The following variable must be set to point to the CATDUAV5 configuration file:
`CMI_CATDUAV5_CONFIG=<path>\CMI_CATDUAV5.xml`

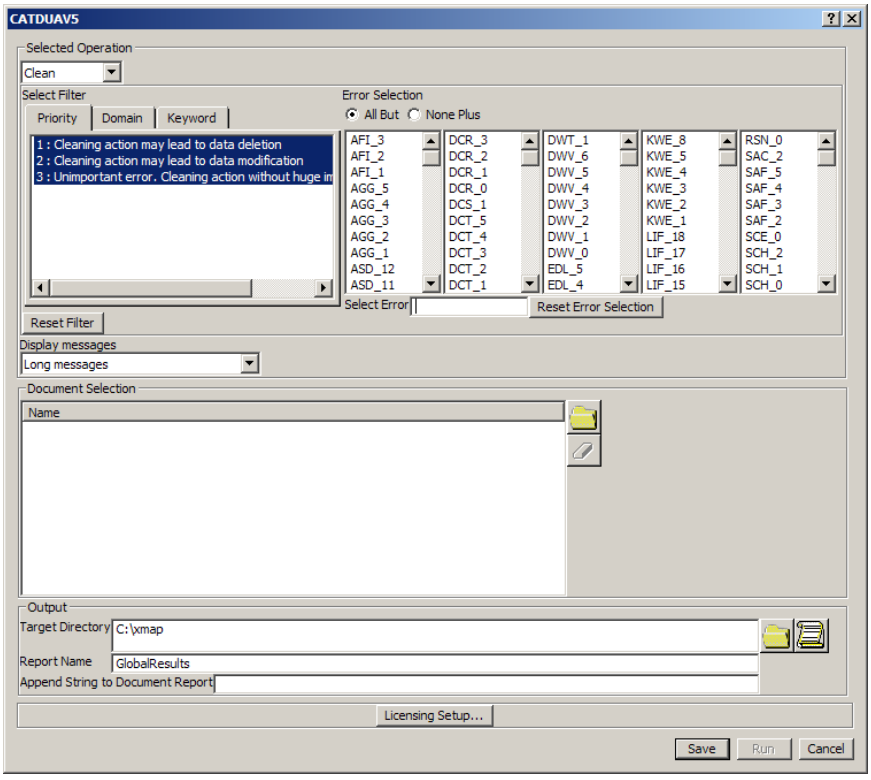


Figure 135: CATDUAV5 dialog

Use from the CMI toolbar to read the data that you want to clean.
Select the cleaning method you want to use:

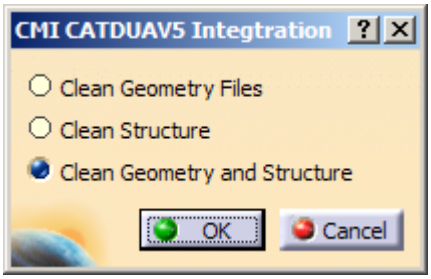


Figure 136: CMI CATDUAV5 Integration dialog

“Clean Geometry Files” uses CATDUAV5 for all CATParts and CATDrawings to be loaded from the CMI Workbench.
“Clean Structure” uses CATDUAV5 for all CATProducts to be loaded from the CMI Workbench.

“**OK**” starts the “Read from Workbench” and launches the CATDUAV5 utility before loading the files.
“**Cancel**” aborts the Command.
After cleaning the files the result is shown in the web browser, it can be found in `%CMIXMAP%\CMI_CATDUA_Result_<YYYY><MM><dd><hh><mm><ss>.htm`

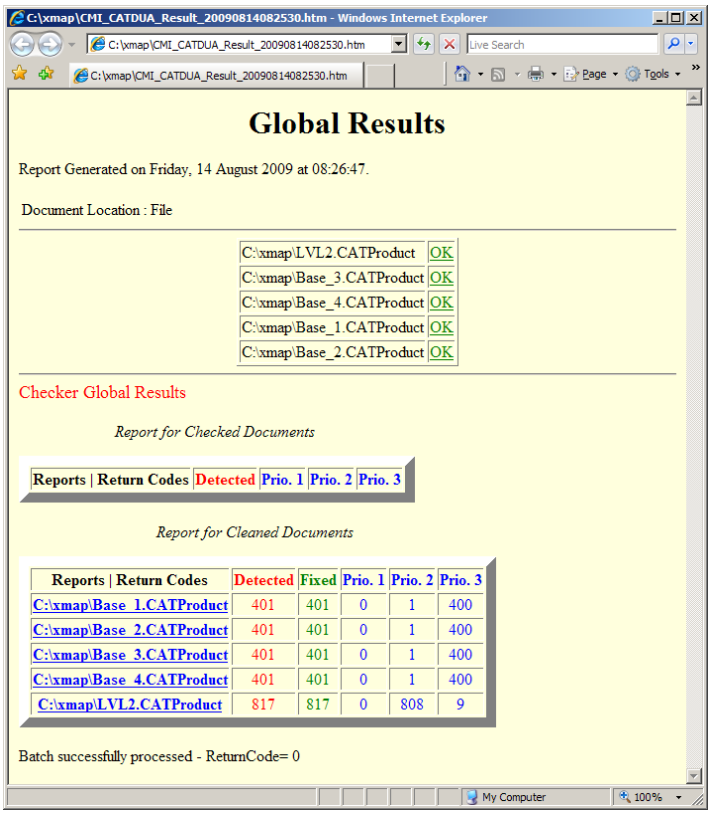


Figure 137: CATDUAV5 Global Results

After inspection, you can update the cleaned files in Teamcenter. If you do not update, the changes made by CATDUAV5 are not stored.

Note:
Cleaning files may take a long time.

Attention:
For files that are already open, CMI cannot load the version that was cleaned by CATDUA. Make sure that you close the files that you want to clean.

Verification of CATIA Release

CMI can check the CATIA version of files against the current CATIA session during Read. This can be used to avoid loading a document from a newer CATIA release to an older CATIA version.

The CATIA Release/SP/HF that was used to save a file is stored in Teamcenter.

To enable the check, set the following in the Teamcenter Configuration:

```
set CMI_VALIDATE_CATIA_VERSION "ON";
```

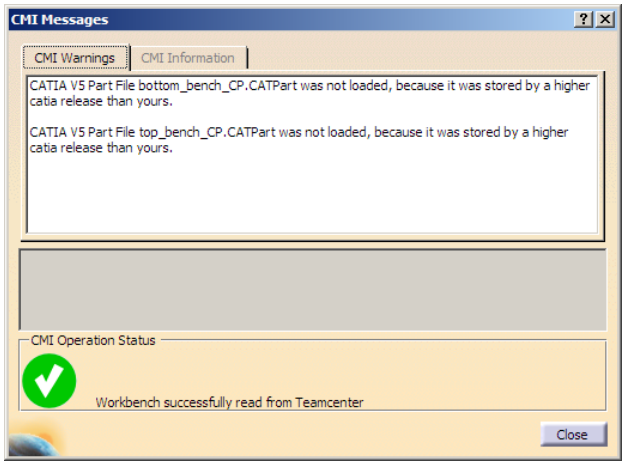


Figure 138: CATIA version verification

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}
```

In this example the `3dmap` and `CATShape` Representation formats will be handled by the CMI Catia Module. (Figure 139 and Figure 140)

Other Formats that may be relevant are `stl` and `jt`.

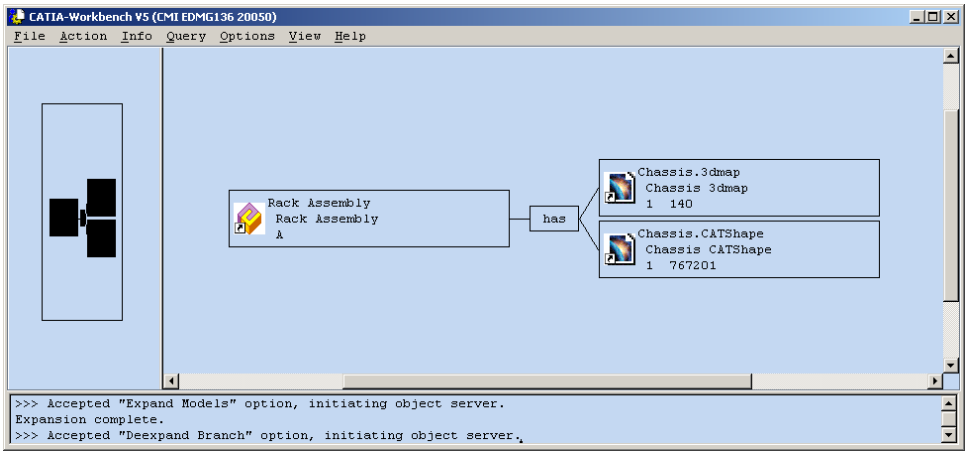


Figure 139: Representation formats in CATIA V5 (CATIA Workbench)

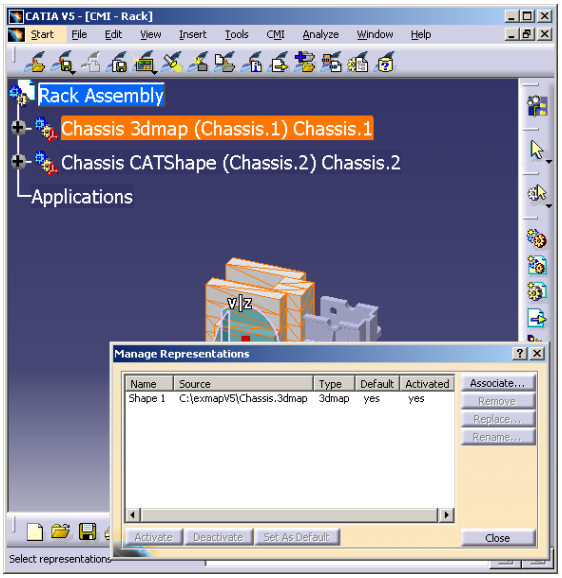


Figure 140: Representation formats in CATIA V5 (CATIA V5)

Use BOM – NO BOM geometries

The CATIA V5 module can handle two different types of CATParts.

- **BOM** relevant and
- **NON BOM** relevant.

NON BOM Relevant CATParts

NON BOM relevant CATParts are CATIA V5 geometry files which are stored as a model under a Part in Teamcenter. In CATIA V5 the Part and model is build up as CATProduct and CatPart (see Figure).

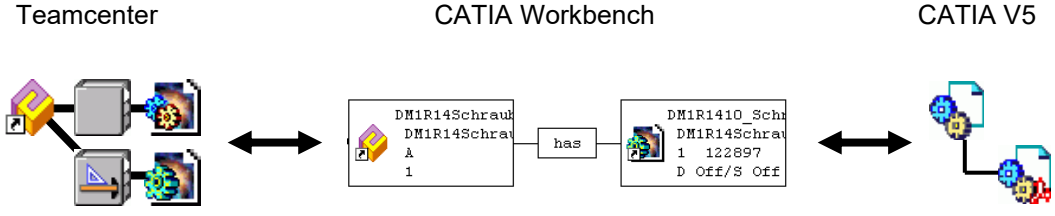


Figure 141: Mapping for NON BOM parts in Teamcenter and CATIA V5

BOM Relevant CATParts

BOM relevant CATParts are CATIA V5 geometry files which are stored as the only model under a Part in Teamcenter. In CATIA V5 the Teamcenter Part is represented by a CATPart (see Figure).

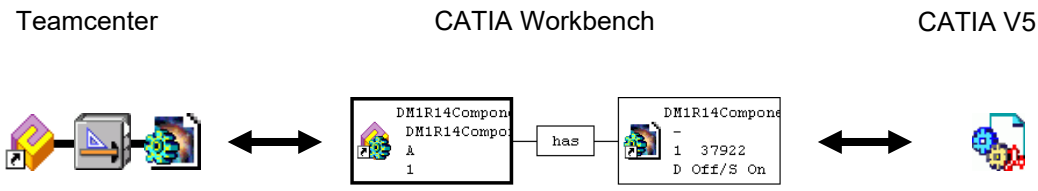


Figure 142: Mapping for BOM parts in Teamcenter and CATIA V5

The option to use the BOM and NON BOM parts must be configured by the Administrator.

The user is able to decide which type of geometry mapping should be used in *Syncronize* command, *Create* command and *SaveAs* command.

To configure this behaviour, an administrator can set the following variable in the Catia V5 environment:

`CMI_ENABLE_ASK_FOR_BOM_PART=ON`

The default can be set with

`CMI_BOM_PART_DEFAULT_FOR_SYNC=` one of {`BOM` | `NOT_SET` | `NOBOM`}

`NOT_SET`: forces the user to make a choice

`BOM`: by default, creates a BOM CATPart

`NOBOM` or not defined: Creates NO BOM CATParts (CMI OOTB behaviour)

Synchronize command

The *Synchronize* command has a new function button *Edit PDM-Part category* in the Synchronize dialog.

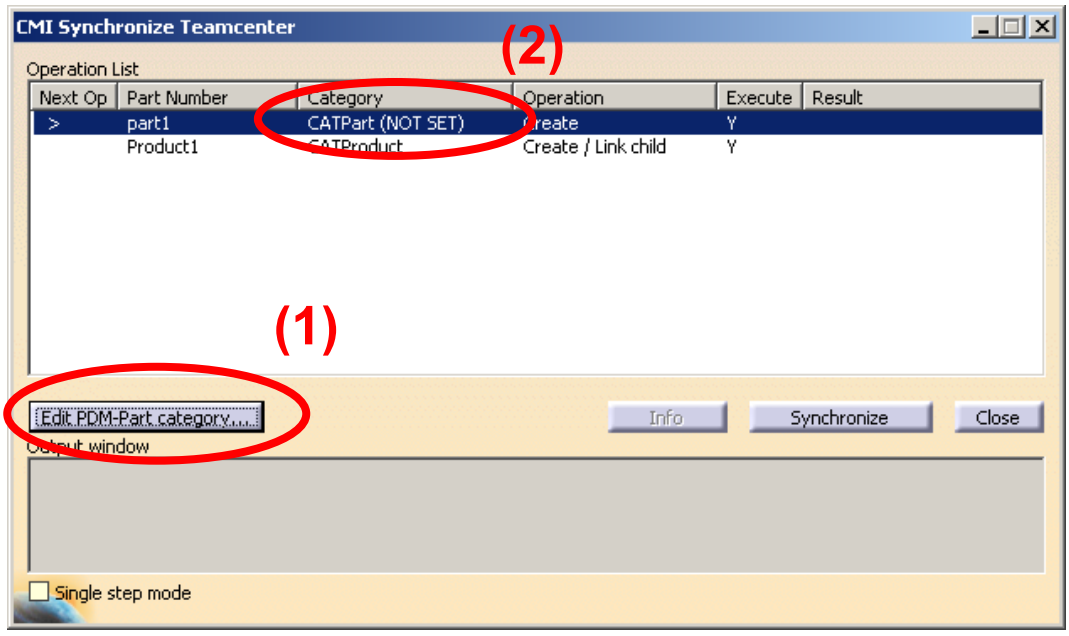


Figure 143: Synchronize dialog with enabled Edit PDM-Part category button

This button is enabled when a line with a CATPart, model or CGR is to be created. Then the user can select the BOM or No BOM type in the Edit new PDM-Part category (see Figure)

For the BOM type Teamcenter will create a new Part, document and data item.

For the No BOM type only the data item will be created.

If NOT_SET is selected, the Synchronize command will not start.

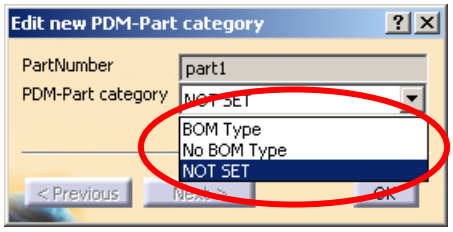


Figure 144: Edit new PDM-Part category

Create command

After starting the Create command the user is asked for the type of the part which should be created in Teamcenter. Figure shows the part type ask dialog and the user has the possibility to choose a BOM type, a No BOM type or to cancel the command.

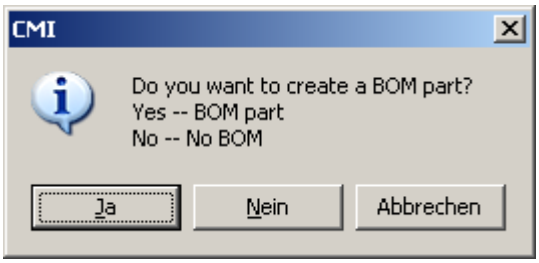


Figure 145: Part type ask dialog with Create command

For the BOM type Teamcenter will create a new Part, document and data item.

For the No BOM type only the data item will be created.

SaveAs command

After starting the SaveAs command the user is asked for the type of the part which should be used in Teamcenter. Figure shows the part type ask dialog and the user has the possibility to choose a BOM type, a No BOM type or to cancel the command.

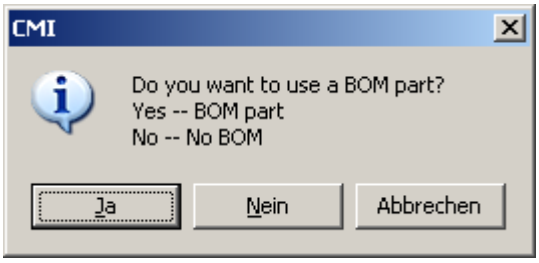


Figure 146: Part type ask dialog for SaveAs command

For the BOM type Teamcenter will create a new Part, document and data item.

For the No BOM type only the data item will be created.

Design Tables

CATIA V5 Design Tables can be stored in Teamcenter. Both CSV and Excel™ files are supported. A Catia V5 specific Dependency relation is created between the CATPart and its Design Table. Design tables referenced by CATParts and CATProducts are supported.

This functionality needs to be enabled in the Teamcenter configuration:

set CMI_DESIGN_TABLES “ON”;

To use Design Tables for CATProducts also, CMI_USE_DTFORPRODUCT = ON needs to be set in the Catia V5 configuration.

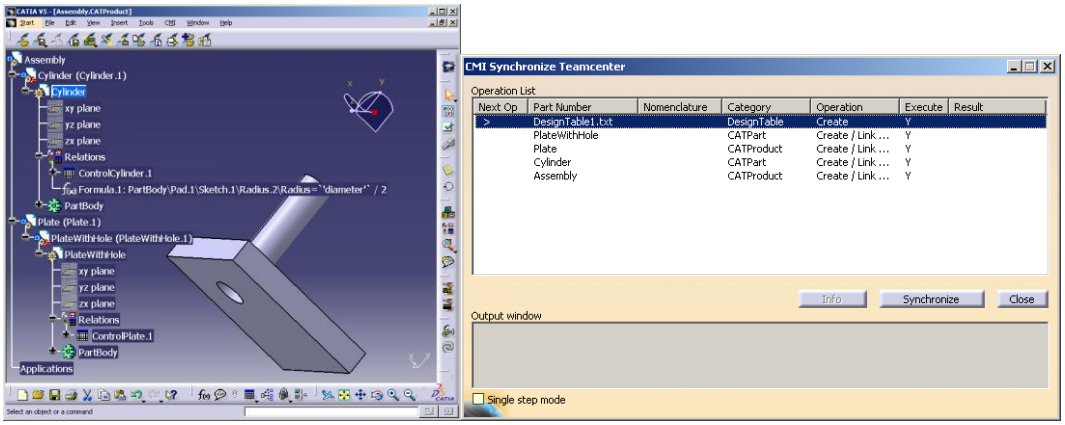


Figure 147: Synchronize with Design Table

Design tables are not stored by default because of performance considerations; also, you need a sound methodology if you allow the use of design tables in your methodology, since design tables have the power to change your product structures.

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 CATProducts and CATParts 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 CATParts in Teamcenter. Transformations and CATProducts 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.

Reset Invalid Positions

An optional CATIA V5 menu item that allows to reset invalid Model-positions to the initial state (identity-matrix).

`CMI_ENABLE_RESETINVALIDPOSCMD=ON`

Get original geometry from Teamcenter

To enable the *Get original geometry from Teamcenter* command the administrator has to set the CATIA V5 environment
`CMI_ENABLE_CMIGETORIGGEOCMD=ON`

A Part or V4 Model has to be selected in the product structure. The command retrieves the geometry file (CATPart or V4 model) from Teamcenter if it is not yet loaded in CATIA V5. The function is used in context of the Release Cache (Working with CATIA V5 Released Cache), to get the original geometry file from Teamcenter if only CGR's are transferred by the configuration.

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"

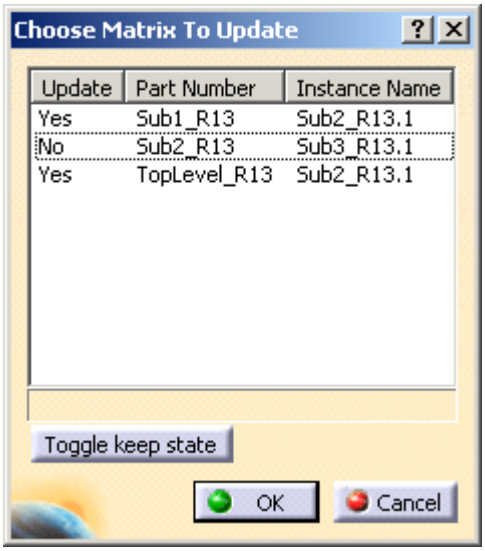


Figure 148: Choose Matrix to Update

If you have changed some positions in the CATIA session this function allows you to select the relations 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.

Version check during Read

If you read a file from Teamcenter, and another version of the file is already open in CATIA, the new version of the file cannot be opened, until you close the file in CATIA. However, CMI will set the metadata, so the open file can eg. be saved as the new version.

You can enable a more strict control of version mismatches. If the variable
CMI_PREVENT_DIFFERENT_VERSIONS=ON
is set in your CATIA environment, CMI will abort the Read if there is a version ambiguity either in the Workbench product structure or between the Workbench product structure and the current contents of the CATIA session.

Check for Educational Flag

CMI can make sure that no CATIA documents that were created with an educational license of CATIA are entered into your PDM system.
Set CMI_ENABLE_EDUFLAG_CHECK=ON to check for educational flag during Update / Synchronize / Create / SaveAs. If any file was created with an educational license, the Save action will be declined.

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.

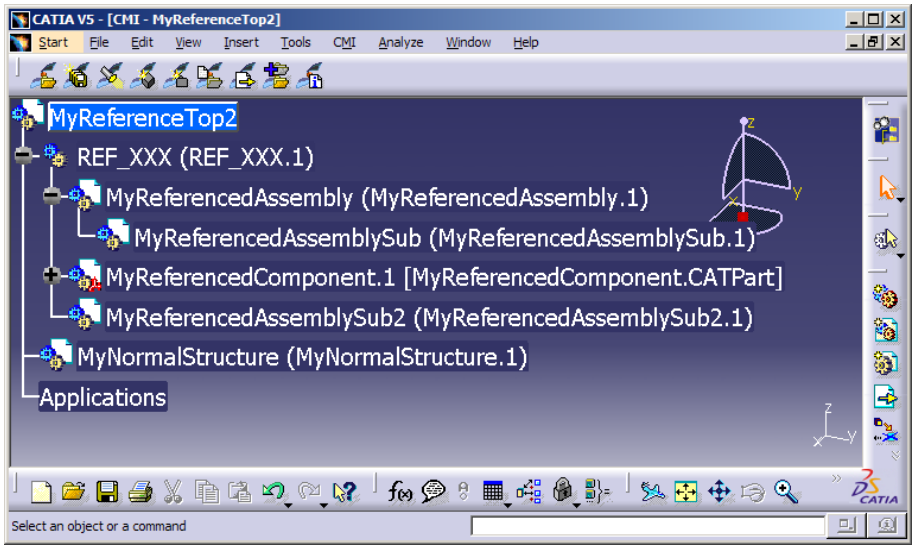


Figure 149: Reference Geometry

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.

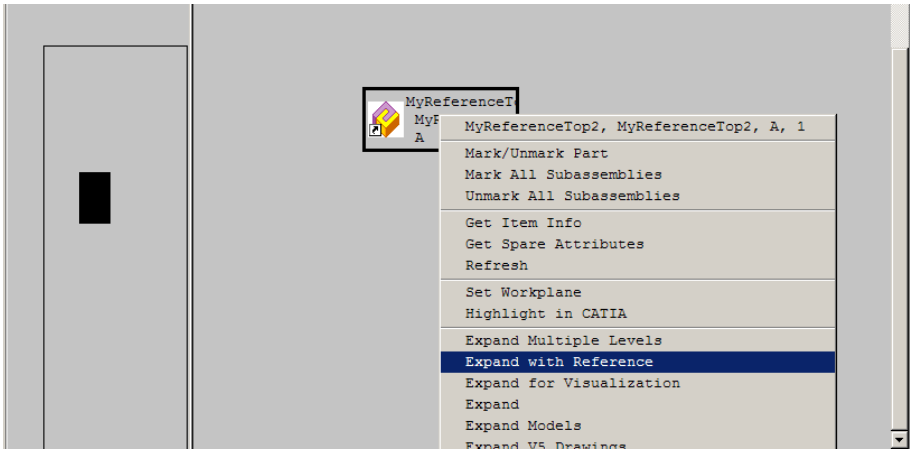


Figure 150: Expand with Reference

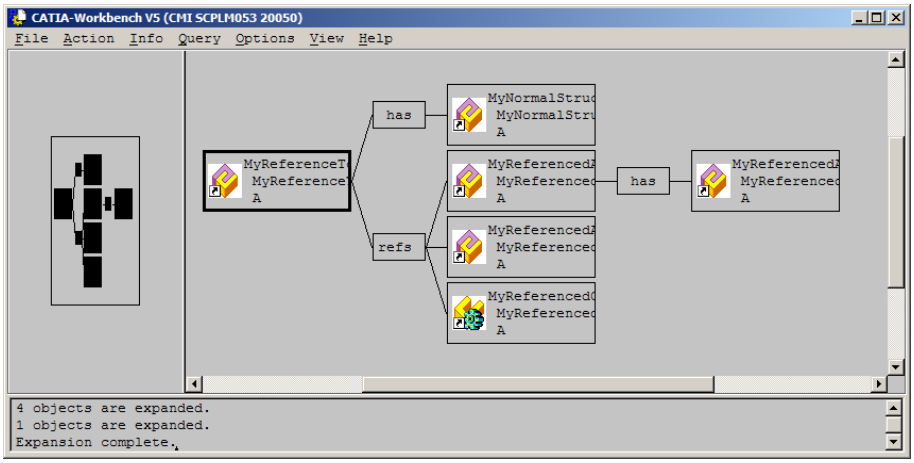


Figure 151: Expanded Reference

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

In order to use this functionality, your administrator has to configure the prefix for the Reference component in CATIA V5. See *CMI Customizing Guide*.

Import / Export using a mapping file

Set CMI_ENABLE_CREATE_IMPORT_FILE=ON to enable options in CMI Reconnect and Synchronize Teamcenter commands to map CATIA data that exists both in your PDM system and at a partner site, but under different naming conventions.

Synchonize

Use the *Write Import/Export mapping file* option in the Synchronize dialog to record or map original names for CATIA data that is stored in Teamcenter.

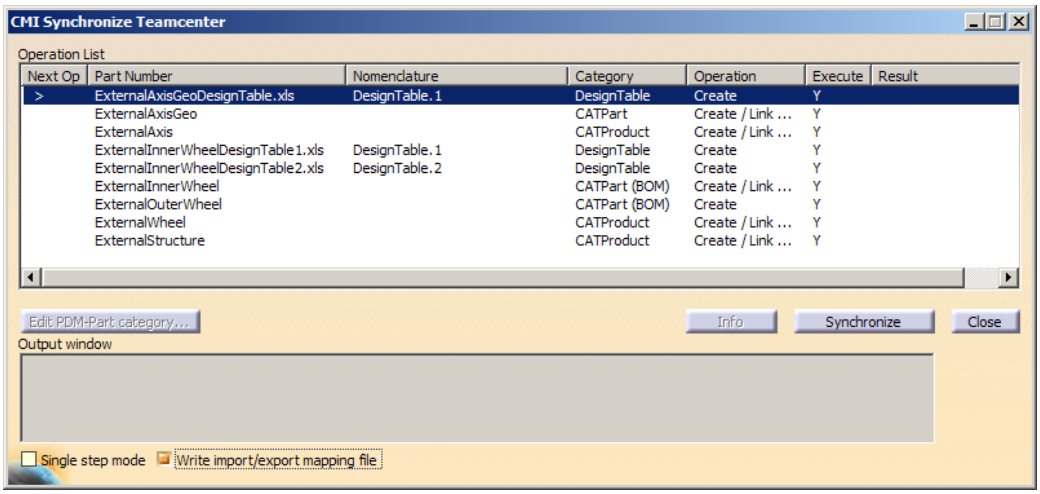


Figure 152: Synchronize - Write import/export mapping file

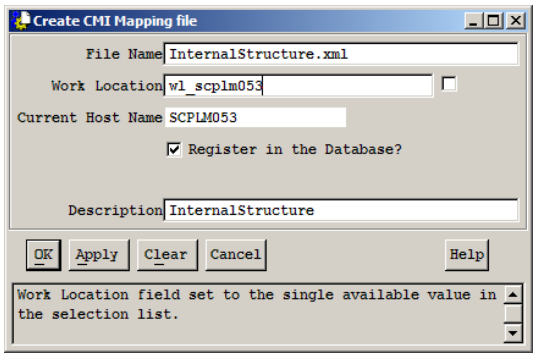


Figure 153: Create CMI Mapping file

As a result of the Synchronize, an XML mapping file is created and attached to the Top level assembly in Teamcenter of the new structure. This file maps original names to Teamcenter names and will be used by the Reconnect and Export commands.

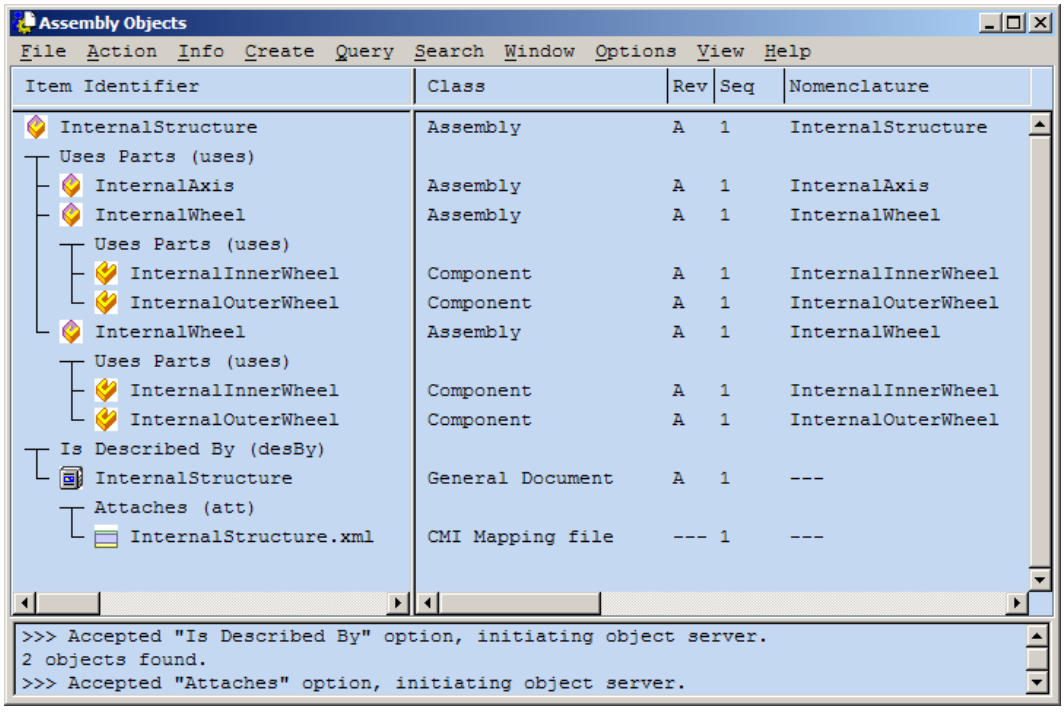


Figure 154: CMI Mapping file in Teamcenter

You should also check this option if you receive a changed version of a structure that was imported, to update the mapping file with new subassemblies.

Reconnect

If a mapping file was created when the product structure was first imported into Teamcenter, this file can be used to map the CATIA names to the names the products have in Teamcenter. Click *Use mapping file* to retrieve a mapping file that is attached to the top level Part from Teamcenter. If a part was imported from outside, the internal name will be supplied by the mapping file.

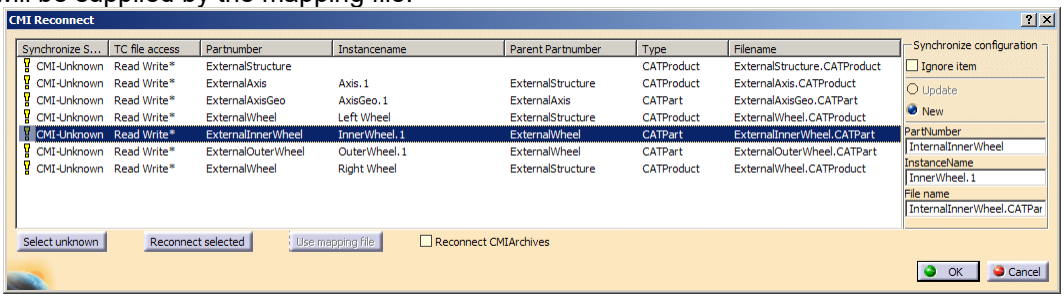


Figure 155: CMI Reconnect

Export to Folder Command

Set CMI_ENABLE_EXPORTCMD=ON to enable the Export command in the CATIA toolbar.

With the export command you can save data that you have opened from Teamcenter to a folder of your choice. If a mapping file is available the products in the structure can be renamed to their original part numbers and file names.

This allows you to return files that you have originally received from an external partner, in their original state.

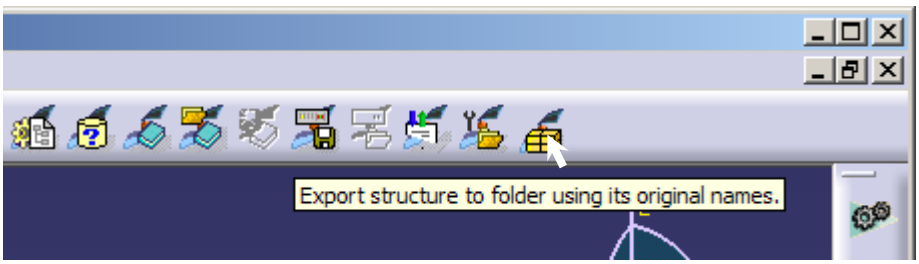


Figure 156: Create CMI Mapping file

CMI asks if you want to use the mapping file:

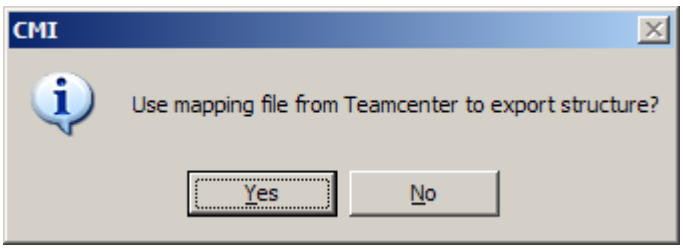


Figure 157: Use mapping file for export dialog

If “Yes” the structure is stored in the export directory using the original names of the CMI Mapping file.

If “No” the structure is stored into the export directory without renaming.

CMI opens a folder selection dialog to select a directory where CMI should store the exported files:

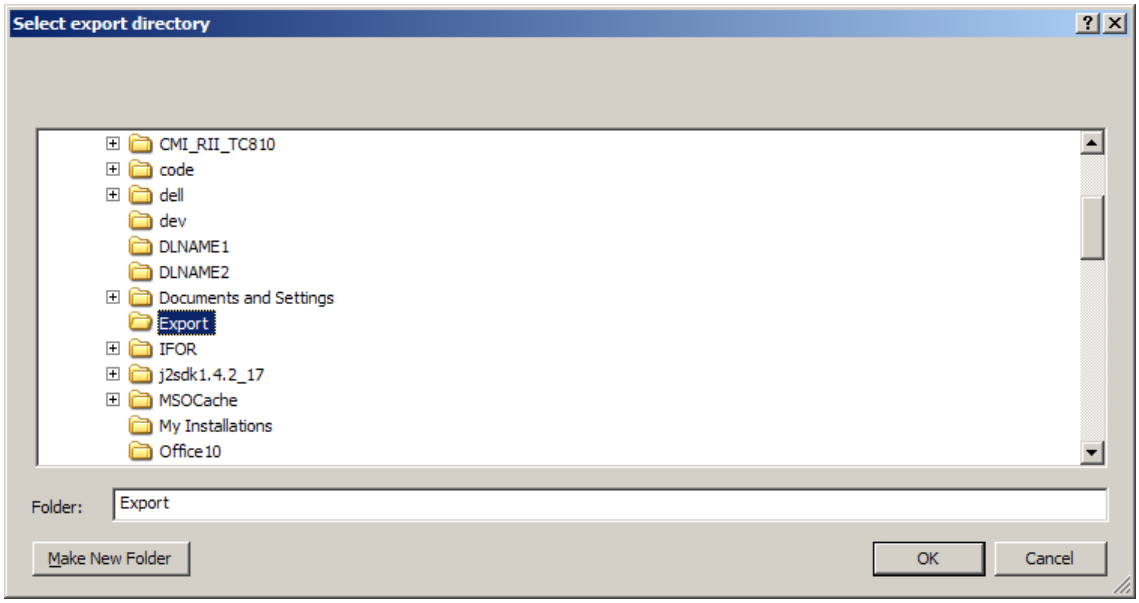


Figure 158: Select export directory dialog

With “OK” CMI performs the export.

After the structure is stored in the export directory CMI shows a summary about the renamed files.

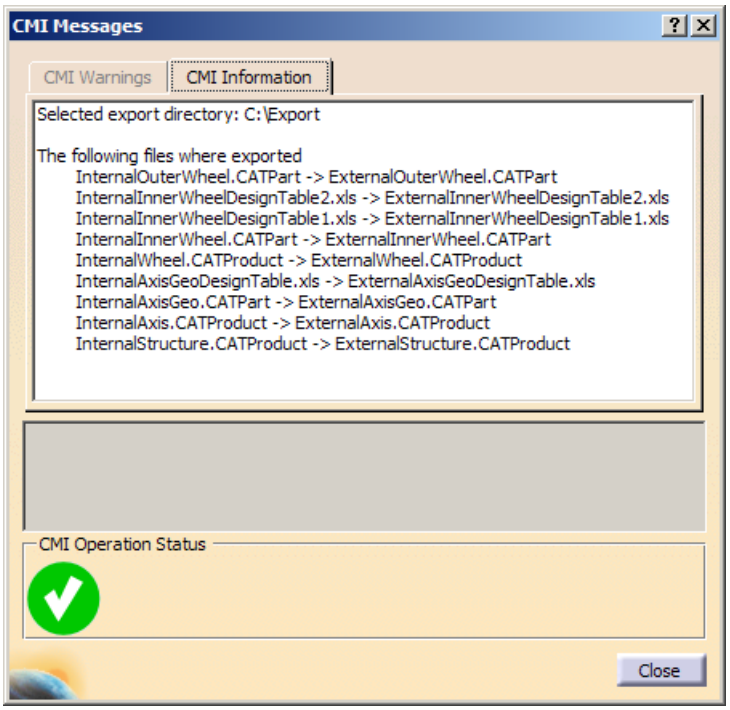


Figure 159: Export Information

The export directory now holds the CATIA files of the exported structure and an XML file which holds the actual mapping of names between Teamcenter and the external structure.

The files in the CATIA session are now “external” and can no longer be updated with CMI. The CATProducts / CATParts point to the external files in the export directory. The part numbers are set to the original part numbers stored in the CMI Mapping file. If CMI_EXPORT_STDATTRIBUTES was set the configured attributes Nomenclature, Definition, Revision or Description are also set back to their original values.

CGM File support

In order to use CGM files with CATIA, set CMI_ENABLE_CGM=ON in the CATIA environment. CMI will support the Teamcenter OOTB class *CGM:Computer Graphics Metafile* from the Industry Standard Classes for basic Read and Update operations with CATIA V5. Please observe the following:
A CGM is always updated in Update or Synchronize regardless of modifications made.
A CGM in a product structure is by default not sent to Catia or expanded in the Workbench. To expand and send it, go to *Options → Model Filter* in the Workbench and check *Use model filter*. In order to view CGM files, the Model Type CATIA V5 Visualization must be selected.

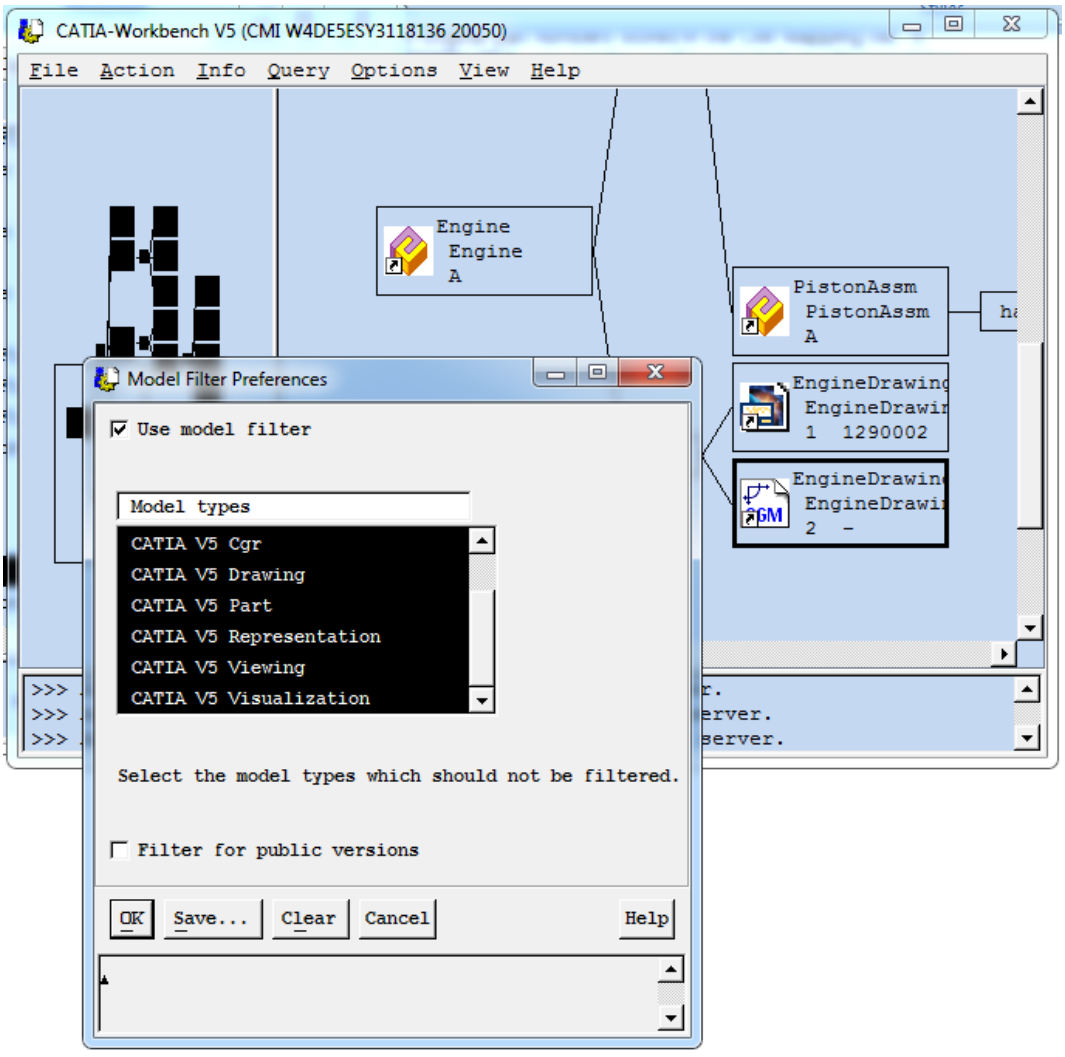


Figure 160: Model Filter Preferences

Deleting NON-BOM Parts during Synchronize

By default, when you remove an instance of a geometry (Non-BOM CATPart) in CATIA, *Synchronize Teamcenter* does not perform any action in Teamcenter. This is because there is no clearly equivalent action in Teamcenter that CMI can provide – for instance, it

cannot tell if you would remove the document from the Part or the CATPart from its Document.

If a user would regularly remove the Document from its Part, an administrator can set the following variable in the Teamcenter configuration:

set CMI_REMOVE_GEOMETRY “DOCUMENT”;

this would remove all versions of the document from the Part that attach the CATPart which was removed in CATIA. This is appropriate if your business logic is that each CATPart has its own document (the document attaches only one model)

If you would remove the Model from its document an administrator can set the following variable in the Teamcenter configuration:

set CMI_REMOVE_GEOMETRY “MODEL”;

This method is suitable if your Document attaches more than one model, and no Document is attached to more than one Part.

If none of these definitions match your business logic, you should instead override the customizing method for the dropping of geometries (see *CMI Customizing Manual*) or let the user handle it.

Glossary

Administrator

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

Browser

Window that displays *icons* representing *objects*.

CATEDM

CATIA Engineering Data Management System Integration. The CATIA add on (GII module) which enables several PDM systems to communicate with the CATIA system in terms of managing CATIA models and structures. The name of the module can differ from installation to installation (i.e. METAPHSE).

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 Part versions 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.

GII

Graphics Interactive Interface. The GII is a powerful programming tool, which completes the Open System Access to the CATIA environment.

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.

Object Management Framework (OMF)

Teamcenter module where it is possible to find the tools and the information needed to the common users activities. Within OMF, each piece of information is an *object* represented by an *icon* that users can manipulate.

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.