

SmarTeam CATIA Web Integration

User's Guide

Version 5 Release 20 SP2



Overview

Conventions

What's New?

User Tasks

Setting Up the CATIA Environment

Connecting to the SmarTeam Database

Saving Documents

Document Lifecycle States

Checking In Documents

Checking Out Documents

Releasing Documents

Creating a New Release

Undoing Check Out Operations

Creating a Document from an Existing One Using New From...

Refreshing your Display in CATIA

Showing Profile Cards

Managing Assemblies

Interface Description

SmarTeam Menu

SmarTeam Toolbar

Modifying a Catalog

Customizing

Document

SMARTEAM (CWi)

Administration Tasks

Defining Property Mapping

Defining Property Mapping for the Revision Block

File Naming

Methodology

Tools For Working in a Concurrent Engineering Environment

Enriched Decision Support with All V5 Links

The User Working Area

Handling Catalogs

Handling CATAnalysis Documents

Glossary

Index

Overview

This book is intended for the user who needs to become quickly familiar SmarTeam - CATIA Web Integration product.

This overview provides the following information:

- [SmarTeam-CATIA Web Integration Product in a Nutshell](#)
- [Before Reading this Guide](#)
- [Getting the Most out of This Guide](#)
- [Conventions Used in this Guide](#)

SmarTeam - CATIA Web Integration Product in a Nutshell



SmarTeam - CATIA Web Integration Product is a seamless, integrated, rapidly usable drawing and document management tool for CATIA users. It gives you the power to manage your parts, products and drawings easily, effectively and affordably, and provides powerful functions to assist you in building assemblies.

SmarTeam together with the CATIA Integration product data management solution (PDM) is designed to give you the tools to create, edit, view and control CATIA documents, in an intuitive and friendly way.

SmarTeam is a revolutionary PDM solution. This philosophy stands behind the product and enables users to install, set up and implement a full fledged Product Data Management solution very easily and rapidly, while maintaining a broad spectrum of functionality.

SmarTeam streamlines the flow of documents through an organization's business process, thereby promoting communication, cooperation and teamwork. SmarTeam together with the CATIA Integration provides a process-oriented approach which enables individuals to work together as a team throughout a product's life cycle.

What are the benefits of the web based SmarTeam - CATIA Web

Integration Product?

SmarTeam - CATIA Web Integration Product is a new architecture of the SmarTeam CATIA Integration based on web applications.

The most important benefit is that no more installation of SmarTeam is needed on the client. SmarTeam - Web Editor enables secure product lifecycle collaboration through the SmarTeam database from standard Internet browsers. SmarTeam -Web Editor exploits standard web technologies to provide remote individuals and teams working on multi-platforms, with comprehensive access to the central SmarTeam database and managed files, enabling users to manipulate product data easily in a dynamic and secure environment from anywhere.

Before Reading this Guide



Before reading this guide, you should be familiar with basic Version 5 concepts such as document windows, standard and view toolbars. Therefore, we recommend that you read the *Infrastructure User's Guide* that describes generic capabilities common to all Version 5 products. It also describes the general layout of V5 and the interoperability between workbenches.

Note that the name of the product referred to in this user's guide has been changed to *ENOVIA SmarTeam - CATIA Web Product* (ENOVIA added). Whenever *SmarTeam - CATIA Web Product* is mentioned in the documentation, read *ENOVIA SmarTeam - CATIA Web Product*.

Getting the Most out of this Guide



This book is intended for users who need to use the SmarTeam - CATIA Web Integration product. The user should already be familiar with basic SmarTeam - CATIA Integration TDM functionalities and with CATIA Version 5 concepts such as document windows, standard and view toolbars.

To get the most out of this guide, we suggest you start reading the [User Tasks](#).

For installation please refer to the installation guide of SmarTeam Web Editor as there is no installation on the client computer.

Conventions Used in this Guide



To learn more about the conventions used in this guide, see the [Conventions](#) section.

Conventions

Certain conventions are used in CATIA, ENOVIA and DELMIA documentation to help you recognize and understand important concepts and specifications. This page describes:

- [Graphics Conventions](#)
- [Text Conventions](#)
- [Mouse Usage](#)
- [Navigation compass](#)

Graphic Conventions

The three categories of graphic conventions used are as follows:

- [Graphic conventions used to structure the tasks](#)
- [Graphic conventions used to indicate the configuration required](#)
- [Graphic conventions used in the table of contents](#)

Graphic Conventions Used to Structure Tasks

Graphic conventions used to structure the tasks are as follows:

This icon

Identifies



estimated time to accomplish a task



a target of a task



the prerequisites



the start of the scenario



a tip



a warning



information



basic concepts



methodology



reference information



information regarding settings, customization, etc.



the end of a task



functionalities that are new or enhanced with this release



allows you to switch back to the full-window viewing mode

Graphic Conventions Used to Indicate Required Configuration

Graphic conventions used to indicate the configuration required are as follows:

This icon

Indicates functions that are



specific to the P1 configuration



specific to the P2 configuration



specific to the P3 configuration

Graphic Conventions Used in the Table of Contents

Graphic conventions used in the table of contents are as follows:

This icon

Gives access to



Site Map



Split View Mode



What's New?



Overview



Getting Started



Basic Tasks



User Tasks or Advanced Tasks



Interoperability



| |
|----------------------------|
| Workbench Description |
| Customizing |
| Administration Tasks |
| Reference |
| Methodology |
| Frequently Asked Questions |
| Glossary |
| Index |

Text Conventions

The following text conventions are used:

- The titles of CATIA, ENOVIA and DELMIA documents *appear in this manner* throughout the text.
- **File > New** identifies the commands to be used.
- Enhancements are identified by a blue-colored background on the text.

Mouse Usage

The mouse usage differs depending on the type of action you need to perform.

Use this mouse button Whenever you read



Select (menus, commands, geometry in graphics area, etc.)

Click (icons, dialog box buttons, tabs, location in the document window)

Double-click

Shift-click

Ctrl-click

Check (check boxes)

Drag

Drag and drop (icons onto objects, objects onto objects)



Drag

Move



Right-click (to select contextual menu)

Navigation Compass



The navigation compass is located in the guide banner next to the guide name. It lets you navigate through the information in the guide.

Use this button To go



Back to the last page visited



Forward through links previously visited



Up to the parent level



Down to the child level



To the previous page at the same level



To the next page at the same level

What's New?

The following enhancements have been added in this release:

- The new SmarTeam icons are displayed in the CATIA menus.

User Tasks

The links below lists the information you will find in this section.

[Setting Up the CATIA Environment](#)
[Connecting to the SmarTeam Database](#)
[Saving Documents](#)
[Document Lifecycle States](#)
[Creating a Document from an Existing One Using New From...](#)
[Refreshing your Display in CATIA](#)
[Showing Profile Cards](#)
[Managing Assemblies](#)

Setting Up the CATIA Environment



This first task of this guide shows you how to set up the options you need for working in the CATIA environment.



This task assumes that a CATIA V5 session is running and that SmarTeam client components are installed on your machine.



1. From your CATIA session, select **Tools > Options...** to access the options you need.

The **Options** dialog box appears.

2. Select the **General** category in the left-hand box, if not already selected.

Document Tab

3. Click the **Document** tab.

The **Document Environments** area defines the various ways of accessing your documents.

4. Select **SmarTeam (CWI)**.

5. Click the **Current** button.

This lets you access documents stored in the SmarTeam vault.

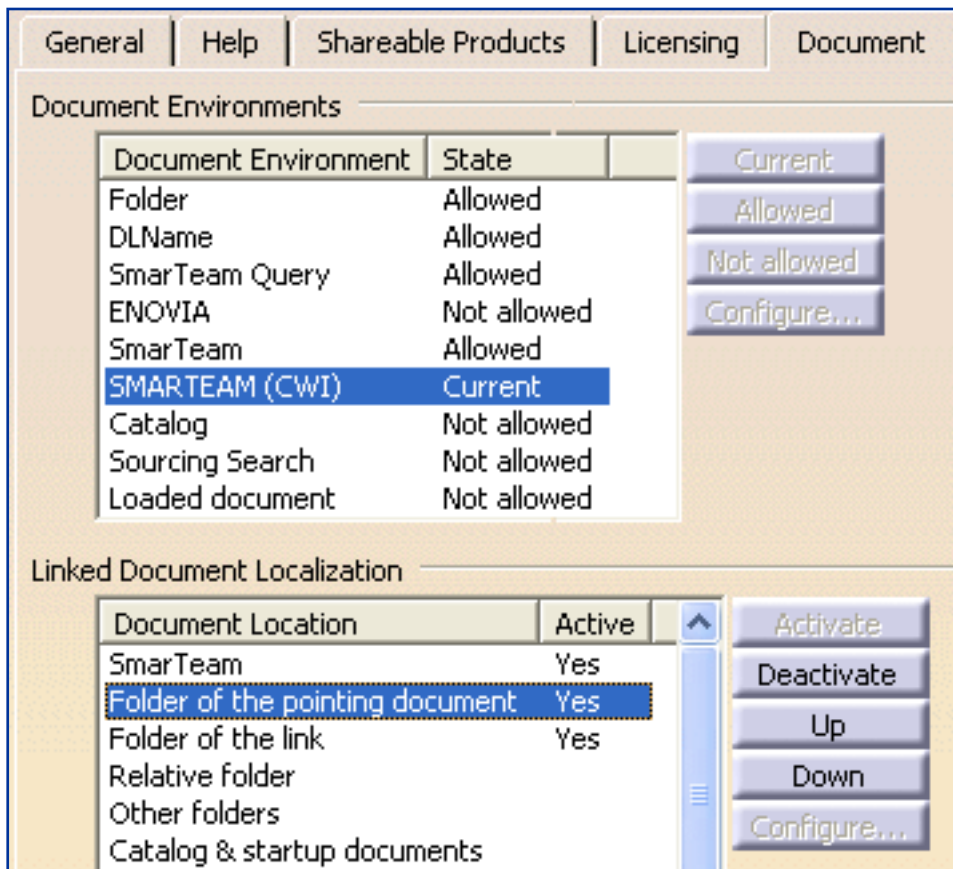
6. In the **Linked Document Localization** area select **SmarTeam**.

The **Linked Documents Localization** capability lets you define a personal strategy that will be used to locate your linked documents and therefore resolve document links.

7. Click the **Up** button.

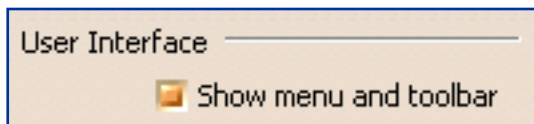
8. Repeat the operation for **Folder of the pointing document**, which provides the current folder of your document.

When done, you must get this:



SmarTeam Tab

9. Select the **Compatibility** category in the **Options** dialog box.
10. Click the **SmarTeam (CWI)** tab.
The **SmarTeam** tab lets you customize compatibility settings in CATIA V5 in order to work in a SmarTeam CWI - CATIA Interoperability context.
11. Select **Show menu and toolbar** in the **User Interface** area.
This option displays SmarTeam CWI menu and toolbar in your CATIA session.



12. In the **Expose Mode** area, clear **Internal components of CATProduct document** and **Sheets of CATDrawing document** in order to be compatible with the SmDemo database. This ensures you do not create SmarTeam objects corresponding to the components of product documents and sheets whenever you check in CATProduct and CATDrawing documents.

Expose Mode

Create SmarTeam objects for:

☐ Internal components of CATProduct document

☐ Sheets of CATDrawing document

13. Select **SmarTeam detailed status** (new, checked-out, checked-in, etc.). This ensures that icons identifying all possible statuses for documents (new, checked-out, checked-in, released etc.), are displayed in CATIA specification tree.

CATIA Tree Icons


☐ SmarTeam simplified status (editable, not editable)

☒ SmarTeam detailed status (new, checked-out, checked-in, etc.)

14. Take a look at the **SmarTeam Web Server** area to ensure that you are using the requested server. Otherwise, enter the server's address.

SMARTeam Web Server

Server URL

15. Click **OK**  to close the **Options** dialog box.
16. Restart a CATIA session to take your modifications into account.
You are ready to [connect to the SmarTeam Database](#).



For more information about the different settings, see [Customizing](#).



Connecting to the SmarTeam Database



This task shows you how to connect to the SmarTeam database.



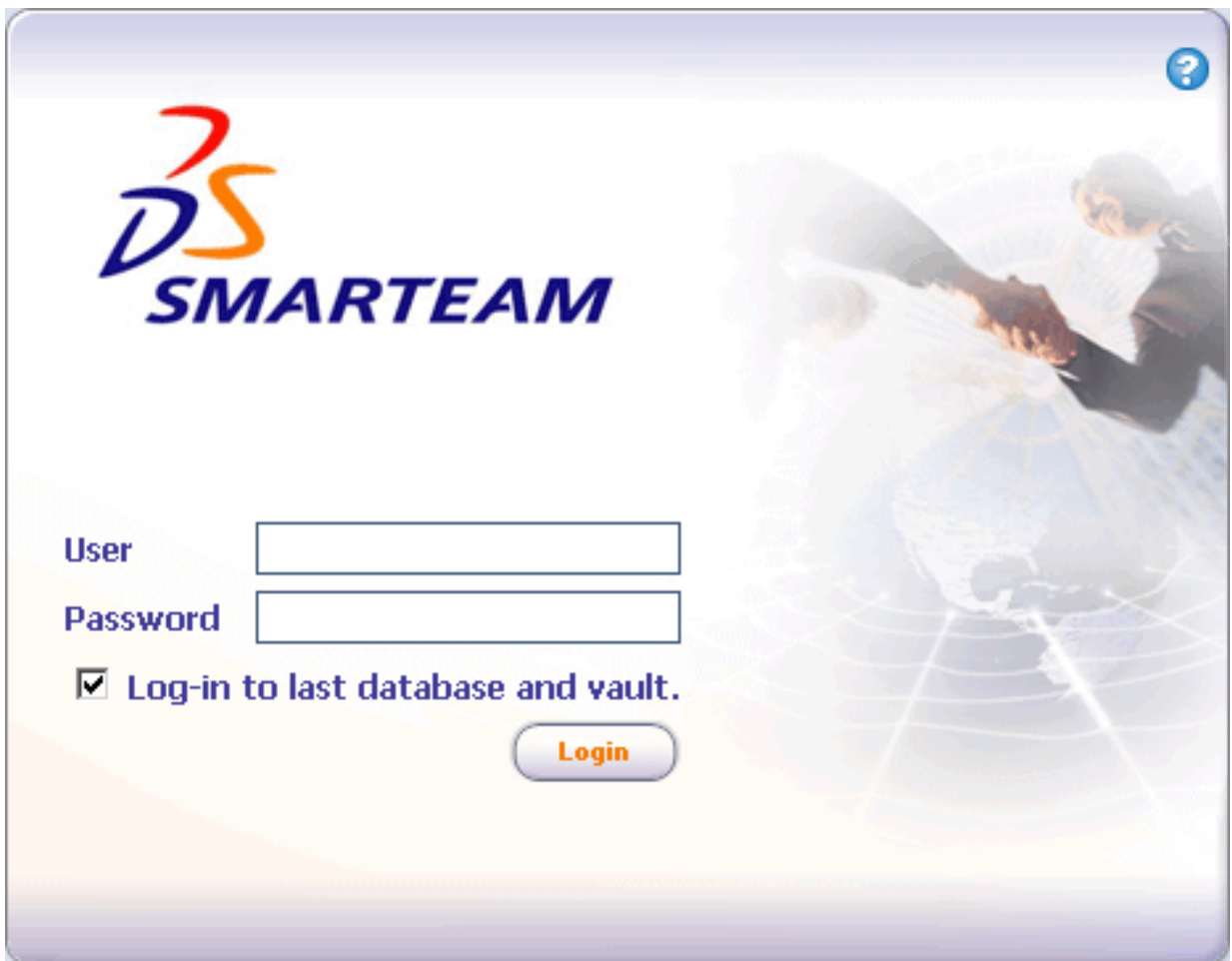
This task assumes that the web server's name was indicated in the **Options** dialog box as indicated in [Setting Up the CATIA Environment](#).



1. From the **SmarTeam** toolbar, select **Connect** .

Alternatively, select **SmarTeam** >  **Connect**.

The **SmarTeam Web Editor-Login** dialog box appears:

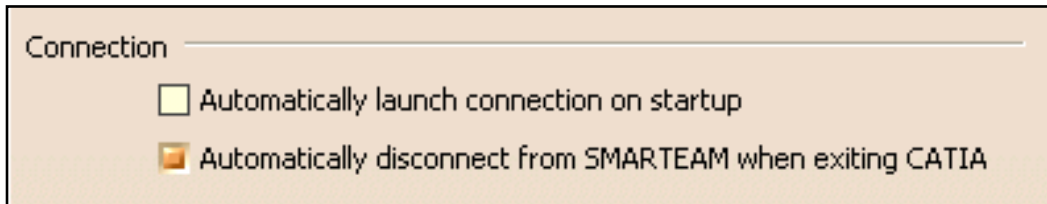



The dialog box features the SmarTeam logo (a stylized 'S' and 'M' in blue and orange) and the text 'SMARTTEAM' in blue. Below the logo, there are two input fields: 'User' and 'Password'. A checkbox labeled 'Log-in to last database and vault.' is checked. At the bottom right, there is a 'Login' button with orange text. The background of the dialog box shows a faint image of two people shaking hands over a globe.

2. Enter your user name and password.
'Joe' is the default user of the SmDemo database. Using this user, there is no need to enter a password to connect to the database.
3. Select **Connection**.
Now that you are connected, you are ready to work with SmarTeam.

Disconnecting

When exiting CATIA, by default you disconnect from the SmarTeam database. Because sometimes it may be useful not to disconnect from the Web Editor, you can modify this behavior by deactivating a dedicated setting. This setting is **Automatically disconnect from SmarTeam when exiting CATIA**. For more information, see [Connection](#) in the SmarTeam (CWI) page.



 When session expiration time is over, any SmarTeam operation may cause crash. It is recommended to increase the timeout to avoid crash.



Saving Documents



This task shows you how to save a CATProduct document you created in a CATIA workbench, in the SmarTeam database. The instructions provided here also apply to any CATIA documents you want to save.



This task assumes that you have just created a product 'MMLStructure.CATProduct', including five parts in a CATIA workbench.



1. Open the CATProduct document you want to save.

2. From the **SmarTeam** toolbar, click **Save** .

Alternatively, select **SmarTeam** > **Save**.

The **Save As** dialog box that appears. In case no leading project is defined, the dialog box lets you choose the project in which you wish to save the document. If, as shown below, a leading project exists, the dialog box displays the name of this leading project. For more about projects, see the SmarTeam documentation.

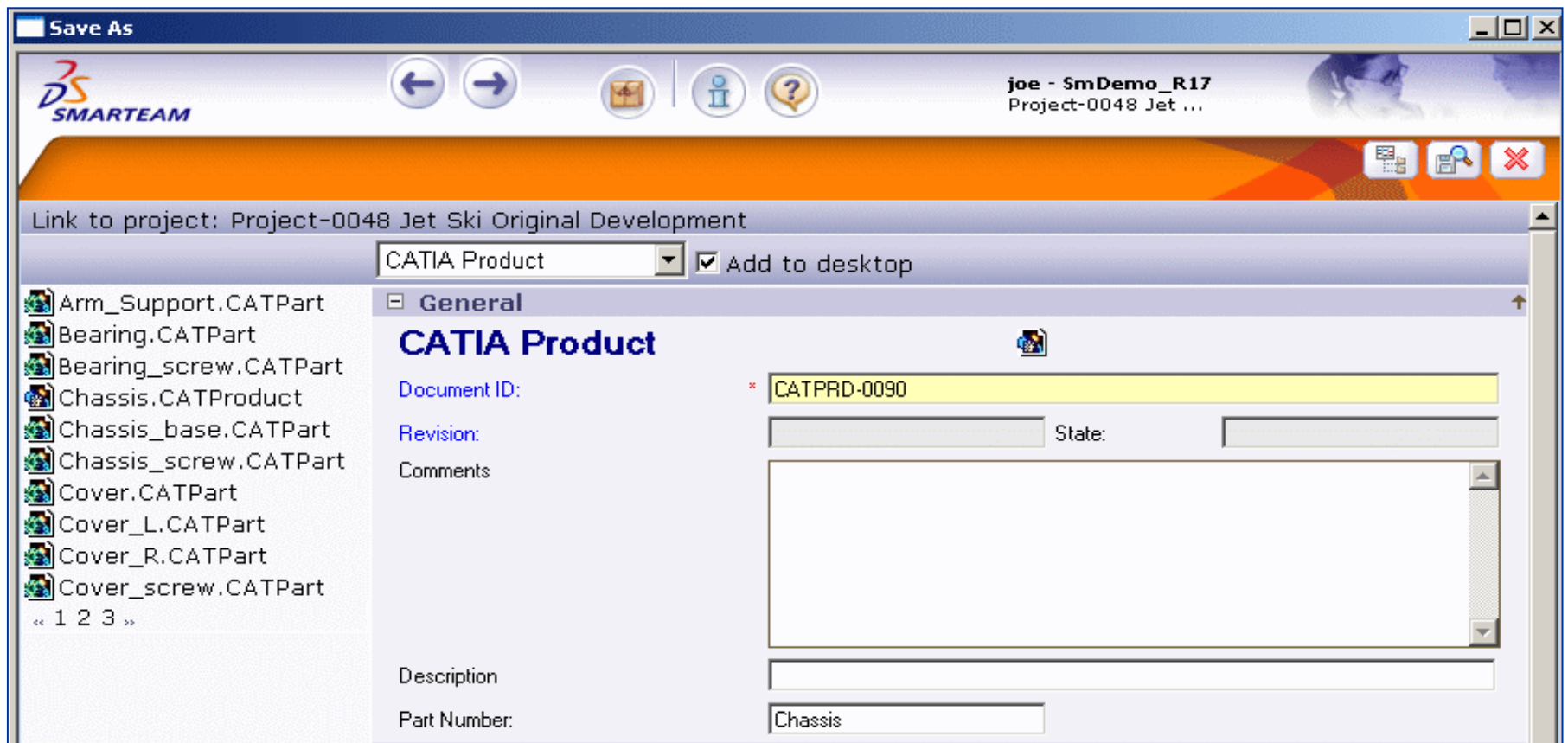
3. Select **Link to project** to save the document in the current project.

By default, this option is not selected.

4. Select **Add to desktop**, now available, to ensure that the document will be shown in the different SmarTeam windows. In case you do not select this option, the only way of finding this document will be by using the SmarTeam **Search** capability. For more information about searching for documents, see the SmarTeam documentation.

5. Click **Advanced** to access additional information.

This part of the dialog box shows all the different document linked to the CATProduct document you are saving.




6. Click on any of these CATPart documents.
The corresponding SmarTeam profile card is displayed. It provides an organized interface that displays the attributes of the selected part.



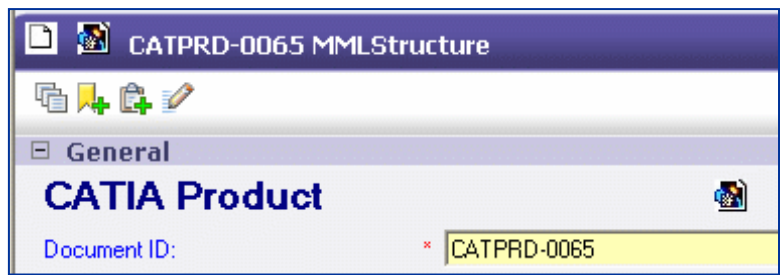
The profile card is divided into different groups of data, that change according to the type of object selected. As shown in our scenario, a CATPart profile card has three groups:


- General
- System
- Notes

You just need to click the + sign to the left of the group name to expand it and access the requested information.

7. Clicking **Project Manager**  allows you to return to the initial dialog box.
SmarTeam shows the profile card corresponding to the CATProduct document. In our scenario, five parts are to be saved. SmarTeam has

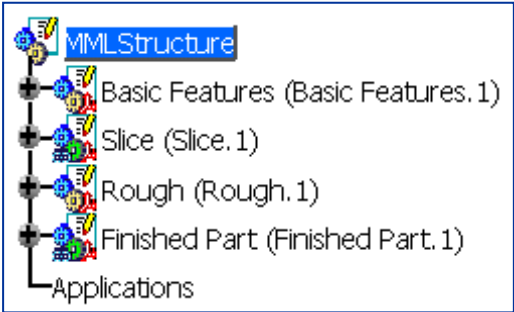
already identified the product as shown in the document ID field of the document profile card:



- 8. Enter the information you need in the different editable fields of the profile card. Fill in the fields as described below in [Save As Dialog Box](#), or accept the default attributes. Other attributes are filled in automatically with CATIA information.
- 9. Now that you have entered your own information, click **Save**  to save the document in SmarTeam.
- 10. Take a look at the icons for the documents in CATIA specification tree. New icons indicate the current status of the CATProduct and CATPart documents. The document you created is now saved in SmarTeam, and you still can edit and save it.

Note that the SmarTeam status is displayed on the top right side of each document icon. Here, the icons indicates that the CATProduct document as well as CATPart documents can be edited.

There are two ways of displaying icons. A dedicated setting lets you choose the display that best meets your needs. Here, you can note that the icons are of the simplified type. For more about CATIA tree icons, see [Customizing](#).



Editing Documents

If after saving a document in SmarTeam as shown above, you later on edit and save this document in different ways (graphic properties, geometry etc.) in CATIA, these modifications are reflected in SmarTeam. The document's profile card displayed reflects the changes you made to the document.

Save As Dialog Box

The following describes the options in the **Save As** dialog box:

| Name | Description |
|------|-------------|
| | |




| | |
|------------------------|---|
| Add to desktop | Ensures that the document you are saving will be shown in the different SmarTeam windows. In case you do not select this option, the only way of finding this document will be by using the SmarTeam Search capability. By default, this option is not selected. |
| Link to project | Directly links the document to the chosen project |
| Advanced | Displays all the different parts linked to the CATProduct document you are saving. |
| Project Manager | Displays the project in which you are saving the selected document. |
| Save | Performs the Save operation and you exit the dialog box. |
| Cancel | Cancels the Save operation |



Document Lifecycle States

When a new object is created, it has the status **New**. Its file is not yet placed in the vault. When you perform a **Check In** operation on the object its file is then placed in the vault and cannot be modified until the object is checked out of the vault. Lifecycle management operations can now be applied to this object; after each Lifecycle operation, the object's status is changed.

The table below lists all Lifecycle operations, the resulting state and a description of the state.

| Lifecycle Options | Resultant State | Description |
|--|---|--|
| Check in - Checks in a new object and places its file in the vault or places an object that is checked out (being modified) back in the vault. | Checked In  | The object's file is placed in the vault, and cannot be modified but may be accessed. You can copy it to view it but not modify it. In order to change a file, a SmarTeam - Web Editor user must check the object out of the vault. |
| Check Out - checks out a Checked In object from the vault. or New Release - makes a new copy of the Released object placed in the Released vault. The resultant object is a new revision of the source object. | Checked Out  | This state is assigned to an object that has been checked out. The object's file is undergoing changes and is in a user's work directory. No other user can currently modify the file but other users can view the file of the parent revision. After the object is checked back in or released, the state is changed to Checked In or Released. |
| Release - transfers an object to the Released state. | Released  | After a stage of development is completed, the object's file is saved in the vault for released files. |

[Checking In Documents](#)
[Checking Out Documents](#)

Releasing Documents
Creating a New Release
Undoing Check Out Operations

Checking In Documents



This page provides instructions for checking in documents.

You will check in documents:

- After creating a new document: to protect the document from modifications that cannot be tracked, your first step in ensuring the document protection is to perform a Check In Lifecycle operation. See [Checking In a Document for the First Time](#)
- After a document has been checked out and perhaps modified. You then need to place it back into the vault. See [Checking In a Document \(other than the first time\)](#).

In both cases, the instructions to follow and detailed in this page are the same.




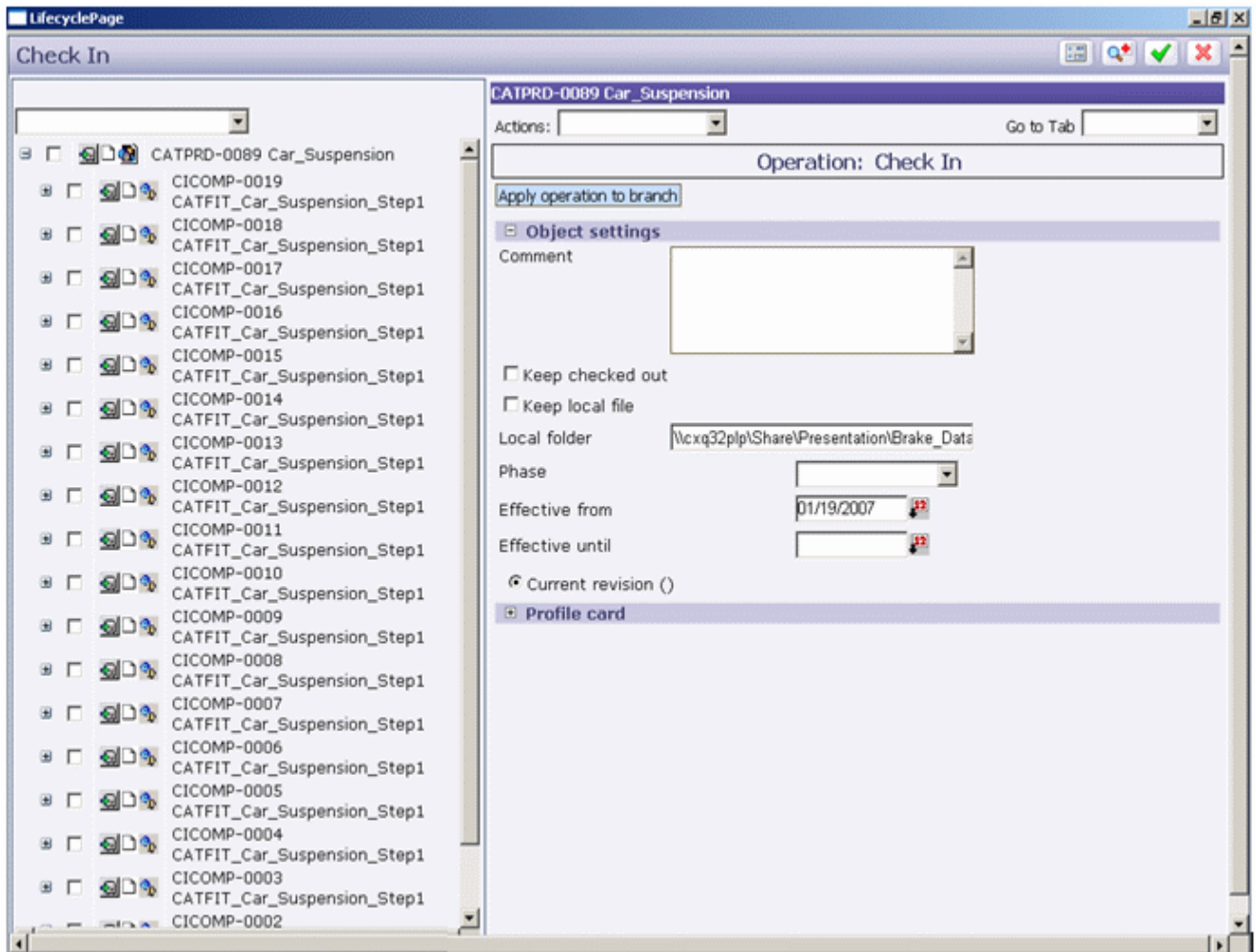
How to Check In Documents

Checking in a document means closing the document in your CATIA session and placing it in the electronic vault. Once placed in the vault, everybody can then access it to read it. however, the document is protected from unauthorized modifications.

The electronic vault ensures that only those persons with access permission may access a file, and that a file cannot be modified by more than one person at a time.


1. From the SmarTeam toolbar, click Check In .

Alternatively, select **SmarTeam > Life Cycle >  Check In**.
The Check In dialog box is displayed.



2. Fill in the fields as described in [Check In Dialog Box](#), or accept the default attributes.

3. Click **OK**  to confirm the **Check In** operation and exit the dialog box.

When done, the status of the document is changed to "Checked In" .

Note


While placing the document in the vault, the application also enables you to keep a copy of the document in your session (use [Keep Local File](#)), for instance to reuse it in another workbench.

Check In Dialog Box

The following describes the options in the **Check In** dialog box:

| Name | Description |
|---|--|
| Set to No Operation | Performs no operation on the selected object |
| Set to Check In | Checks the selected document into the vault |
| Apply Operation to Branch | Applies this operation to the entire document branch |
| Comment | Allows you to enter any comments relating to the Check In process |
| Keep Checked Out | Keeps the file checked out for modifications after performing the Check In operation |
| Keep Local File | Keeps the file located in the working directory in read-only mode |
| Released Parents of Previous Revision | Links the object to all the released parents of the previous revision |
| Checked in Parents of Previous Revision | Links the object to the checked in parents of the previous revision |
| Phase | Open the combo box to select an option to define the phase for this revision. This field is descriptive only and does not affect the revision status |
| Effective From/Until | Click the Date button to enter applicable dates for effectivity in these fields |
| Current Revision | Checks the document in as a new revision. |
| Replace Previous Revision | Checks the document in under the previous revision, overwriting the previous revision |
| Apply Changes | Performs the Check In operation and keeps the dialog box open. |
| OK | Performs the Check In operation and you exit the dialog box. |

Checking In a Document for the First Time

When a document is first created and saved into the SmarTeam database, it is automatically assigned the status "New". This means that the document has not yet been checked into a SmarTeam vault for safekeeping. To protect the document from modifications, place the document into the SmarTeam vault by checking it in. After the document has been checked in, its status is changed to "Checked In" .

Checking In a Document (other than the first time)

After a document has been checked out and perhaps modified, it must be placed back into the vault. At this point, you can check the document back into the vault in one of two ways:

- **Check In** - simply places the document back in the Check In vault. (You can later apply the **Check Out** operation on the document to make further changes).
- **Release** - moves the document to the Released vault.

What happens next?


At any time it is possible to reopen the document in your CATIA session. If you need to modify it, the part must be checked out. Otherwise, the part can be copied in your session in read-only mode.





To be able to make changes to a document checked in to the vault, check it out of the vault as explained in [Checking Out Documents](#).



Checking Out Documents

 When you need to make modifications to a checked-in document, you must check it out of the vault.

When the checked out document is launched into CATIA for modification changes, no other user can currently modify it, but other users can view it or copy the file to their working directory. After the document is checked back in or released, the status is replaced by Checked In  or Released . For more about the Released status, see [Releasing Documents](#).

Depending on whether the document you want to modify is in your CATIA session or not, the application behaves differently:

- If the document is in your session, you can check it out before modifying it, or if you start modifying it, the application detects it and then prompts you to check it out. This behavior is made possible if **Display Check-Out/New Release dialog box on first modification** is active. For more information about this option, see [SmarTeam \(CWI\)](#).
- If the document is not in your session and that you are opening it, you can use the **Check Out** option before confirming the open operation.



This task shows you how to check out a document from CATIA.


This section provides the following information:

- [How to Check Out Documents](#)
- [Check Out Dialog Box](#)

How to Check Out Documents


You can perform the Check Out operation if the part is currently displayed in your session (in read-only mode), or not.

1. Select the document you want to check out.
2. Select **SmarTeam > Life Cycle >**  **Check Out**.
The **Check Out** dialog box is displayed.
3. Fill in the fields as described below in [Check Out Dialog Box](#), or accept the default attributes.
4. Click **OK** .

When done, the status of the document is changed to "Checked Out"  .
You can then modify the document. A new revision number is assigned to it.

Using the SmarTeam Open For... Capability

It is also possible to check out a document as follows:

1. In the Web Editor, once the corresponding profile card of the document you want to check out is displayed, just click **OK**  .
This displays the **Open For...** window.
2. You then just need to click **Check Out** in that window.
This displays the **Check Out** dialog box.
3. Perform steps 3 and 4 as described in the instructions provided above.

Note

The application also enables you to copy a file (use [Set to Copy File](#)) to your working directory without checking it out of the vault. This is useful when another user is working with the document (and has checked it out of the vault), but you wish to view the document at your desktop.


Set to Copy File also to be used whenever you are handling a CATProduct document including several documents. In case you do not need to check out all of these documents, you can therefore restrict the number of documents to be checked out.

Check Out Dialog Box

The following describes the fields in the **Check Out** dialog box:


| Name | Description |
|------------------------------|--|
| Actions | |
| • Set to No Operation | Performs no operation on the selected object |
| • Set to New Release | Makes a new copy of the released document placed in the Released vault. The resulting document is a new revision of the source document. |

| | |
|---|---|
| <ul style="list-style-type: none"> • Set to Copy File | Copies the file of a document to your working directory without checking out the document from the vault. You can view the document in your working directory, but you cannot modify it. |
| <ul style="list-style-type: none"> • Set to Check Out | Checks the selected document out of the vault |
| Go to Tab | Selects the group to which to jump directly (optional) |
| Apply Operations to Branch | Applies this operation to the entire document branch |
| Switch Children to Latest Available | Switches the children of the selected object to the latest available objects |
| Global Settings Group | |
| <ul style="list-style-type: none"> • File Name | Shows the name of the physical name (name on the disk) of the file to check out |
| <ul style="list-style-type: none"> • Local folder | Contains the directory path of the vault in which the document's file will be located |
| <ul style="list-style-type: none"> • Comment | Allows you to enter any comments relating to the Check In process |
| <ul style="list-style-type: none"> • Do not get the file from the vault | Performs the check out operation without copying the file from the vault to the working directory |
| Leading Object Settings Group | |
| <ul style="list-style-type: none"> • Phase | Defines the phase for this revision: <ul style="list-style-type: none"> • Yes: replaces existing local files when checking out • No: leaves existing local files when checking out • Yes, for read only |
| <ul style="list-style-type: none"> • Current Revision | Displays the current revision. |
| <ul style="list-style-type: none"> • Next revision | Presents the next revision number to be applied to the document after completing this check out operation. |

 To check the document back into the vault for safekeeping, check the document in as explained in [Checking In Documents](#).



Releasing Documents


 After a document has been modified and when it is ready to be moved to the next development phase, you place it in the Released vault. The Release operation is generally performed when a supervisor releases a stage of development of the document.

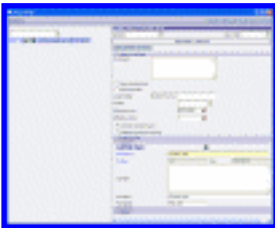
Once a document is placed in the Released vault, it can only be released as a new revision to ensure the safekeeping of this version of the document. The document can be checked out as a **New Release** with a new revision number.



You can perform a **Release** operation on a document assigned one of the following states:

- **New**
- **Checked Out**



1. Select **SmarTeam > Life Cycle >  Release**.
The **Lifecycle Page** dialog box is displayed.



2. Fill in the fields as described in [Release Dialog Box](#), or accept the default attributes.
3. Click **OK**  to confirm the **Release** operation and exit the dialog box.
When done, the document is then assigned the "Released" status .

Release Dialog Box


The following describes the options in the **Release** dialog box:

| Name | Description |
|---------------------|--|
| Set to No Operation | Performs no operation on the selected object |
| Set to Check In | Checks the selected document into the vault |

| | |
|--|--|
| Apply Operation to Branch | Applies this operation to the entire document branch |
| Comment | Allows you to enter any comments relating to the Check In process |
| Keep Checked Out | Keeps the file checked out for modifications after performing the Check In operation |
| Keep Local File | Keeps the file located in the working directory in read-only mode |
| Released Parents of Previous Revision | Links the object to all the released parents of the previous revision |
| Checked in Parents of Previous Revision | Links the object to the checked in parents of the previous revision |
| Phase | Open the combo box to select an option to define the phase for this revision. This field is descriptive only and does not affect the revision status |
| Effective From/Until | Click the Date button to enter applicable dates for effectivity in these fields |
| Current Revision | Checks the document in as a new revision. |
| Replace Previous Revision | Checks the document in under the previous revision, overwriting the previous revision |
| Apply Changes | Performs the Check In operation and keeps the dialog box open. |
| OK | Performs the Check In operation and you exit the dialog box. |







Creating a New Release

 After a document has been released to the next production stage, it may be required to carry out additional modifications to the document, even though it has been released. In this case, a document assigned a **Released** status can only be taken out of the vault by carrying out a New Release operation. This operation will automatically create a new revision for the document.

You can perform a Release lifecycle operation on a document that is assigned one of the following states:

- **New**
- **Checked In**
- **Checked Out**

-  1. Select **SmarTeam > Life Cycle >  New Release**.
The **New Release** dialog box is displayed.
2. Fill in the fields as described in [Check Out Dialog Box](#), or accept the default attributes.
3. Click **OK**  to confirm the **Release** operation and exit the dialog box. The document is released to the SmarTeam vault. The status of the document is changed to "Checked Out" . You can then modify the document.



Undoing Check Out Operations

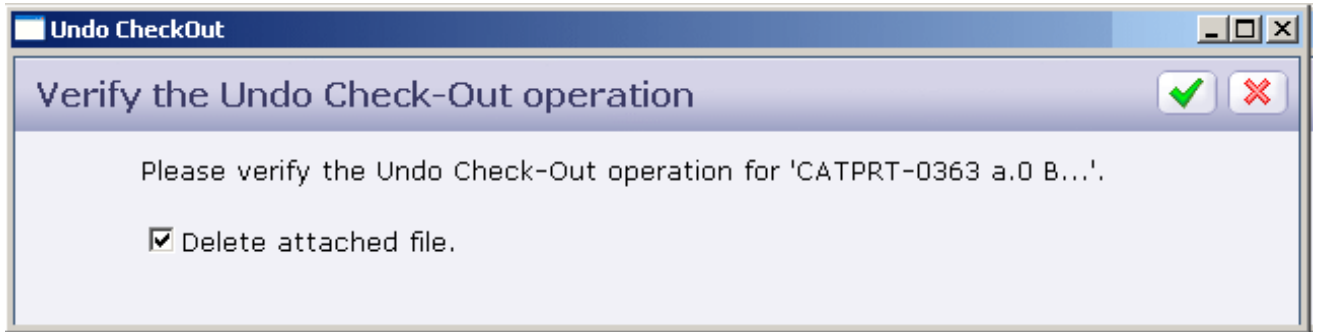



You can cancel the check-out of a document as explained in this task.




1. Select **SmarTeam > Life Cycle >  Undo Check Out**.

The **Undo Check Out** dialog box is displayed, in which you are prompted to verify the operation you are about to perform.



2. Click **OK**  to confirm the operation and exit the dialog box.

Any changes made following the check-out are lost. The document status reverts to "Checked In" .


Note that the **Undo Check Out** capability is not fully supported by modified documents. We recommend you save the documents before.



To know more about Check Out operations, see [Checking Out Documents](#).



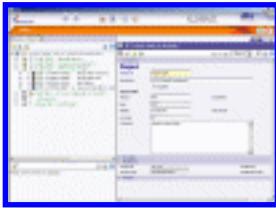
Creating a Document from an Existing One Using New From...

 In the CATIA session you can create a new document based on an existing SmarTeam database document. To do so, you need to use **New From....** This task shows you how to do so.




1. Select **SmarTeam** >  **New From...**

The **Open** dialog box is displayed, showing you your leading project.





If the document to be used as a template is located in another project, select the **All Projects** button on top of the window, and navigate to the project of interest. Another way of accessing the document is to perform a search operation. For more about searches, see the SmarTeam documentation.



2. Select the requested document.
3. Click **OK**  to confirm the creation operation.
A new document is created from the document you selected in SmarTeam. In our example, the original document was a CATProduct document.



Refreshing your Display in CATIA

 The **Refresh**  command updates the product structure and file desk icons with regard to SmarTeam information.


This is particularly useful for concurrent engineering. For instance, if you are using checked-in parts within their assembly, you can use **Refresh** to check whether its loaded parts correspond to the latest revision. Another user may have performed a check-out, modifications and a check-in while the first user is still using the old file revision.

-  1. Select **SmarTeam** >  **Refresh**.
The display of icons representing documents in the specification tree is updated. For more about the different icons, see [CATIA Tree Icons](#).



Showing Profile Cards




The **Show Profile Card**  command provides the database view of the document currently displayed in a CATIA session. If you wish to view the relationships between documents open in your CATIA session and other documents as saved in SmarTeam, you can also use this capability.

It is available for all documents and can be accessed:

- From CATIA workbench
- From CATIA Desk tree

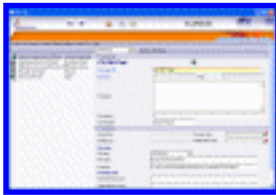


From CATIA Workbench

1. Select the document of interest.
2. From the **SmarTeam** toolbar, click **Show Profile Card** .

Alternatively, select **SmarTeam** >  **Show Profile Card**.

A SmarTeam document window opens, giving access to the profile card displaying information corresponding to the selection as saved in the SmarTeam database. If your selection contains a new entity not saved in SmarTeam, the application informs you.



Note that the capability does not apply to multi-selected documents.

From CATIA Desk Tree

The **CATIA Desk** window lets you view the relationships between different documents and obtain information about their properties.

1. In the **CATIA Desk** window, right-click the document of interest.

Once in the window, you can use **Show Profile Card**  for all documents.

2. Select **SmarTeam (CWI)** > **Show Profile Card** contextual command.

The profile card of the selected document is now displayed.


More About Profile Cards

The profile card is divided into different groups of data, that change according to the type of object selected.

Profile cards are custom-designed by your system administrator. For more information, see the SmarTeam documentation.



Managing Assemblies

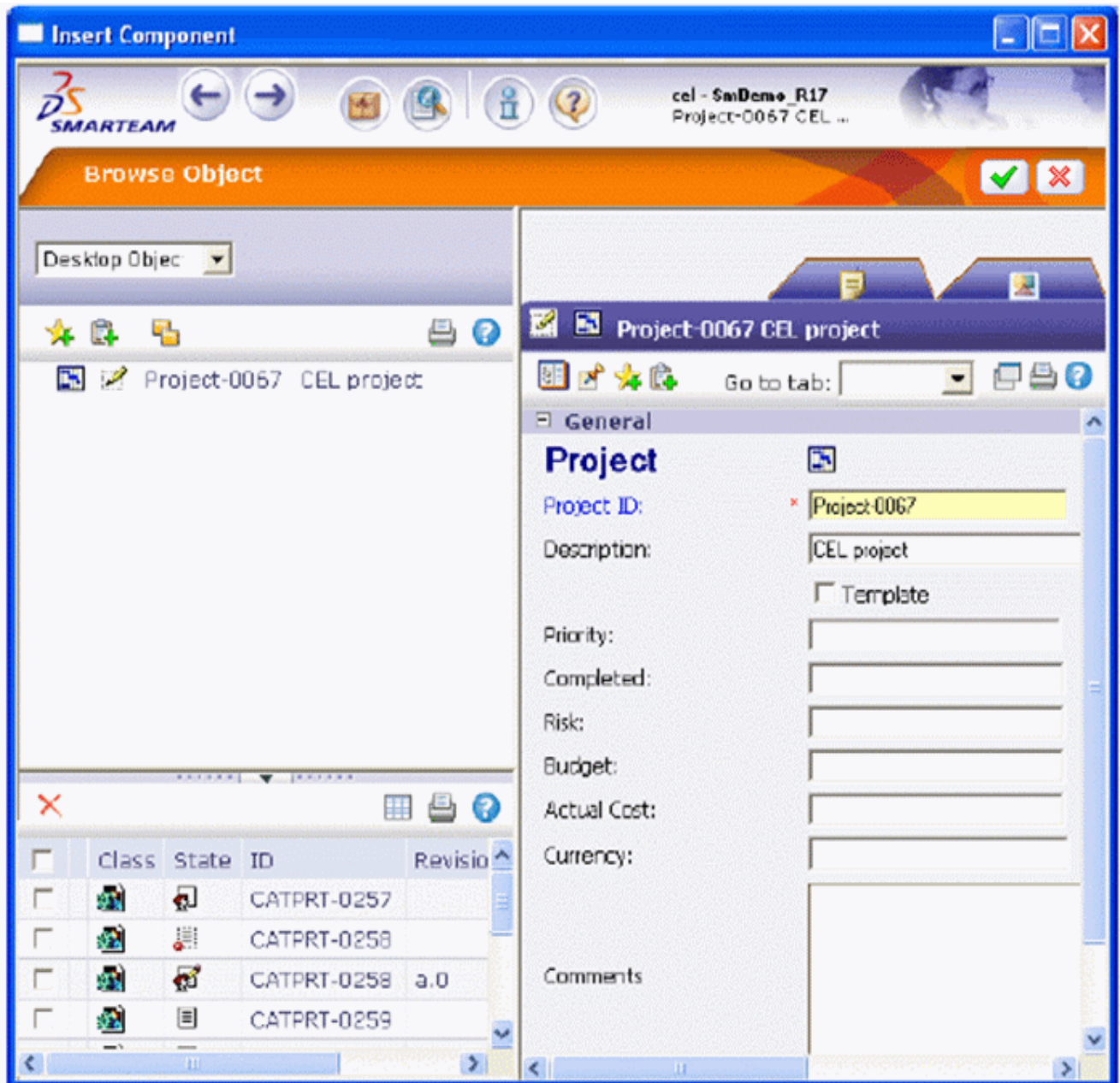
 This task shows you how to use the **Assembly Management** commands. These enable you to insert existing parts, assemblies as components of the current assembly and to replace an existing part or product or a component of the current product.

Inserting a Component

1. In CATIA specification tree, select the document you wish to replace from the assembly.

2. Select **SmarTeam > Assembly Management >  Insert Component**.

The **Insert Component** dialog box that appears, displays your leading project to let you access the required parts or products.



3. Select the document you want to insert.

This displays the corresponding profile card to the right of the dialog box. You can then check if it is the part you wish to insert in your CATProduct document.

4. Click **OK**  to confirm the insertion.



The CATProduct document now contains the inserted document. More precisely, that document is a read-

only copy of the original document still in the vault.

Replacing a Component

Perform the same steps as for component insertion described above but this time select **SmarTeam > Assembly Management > Replace Component**.

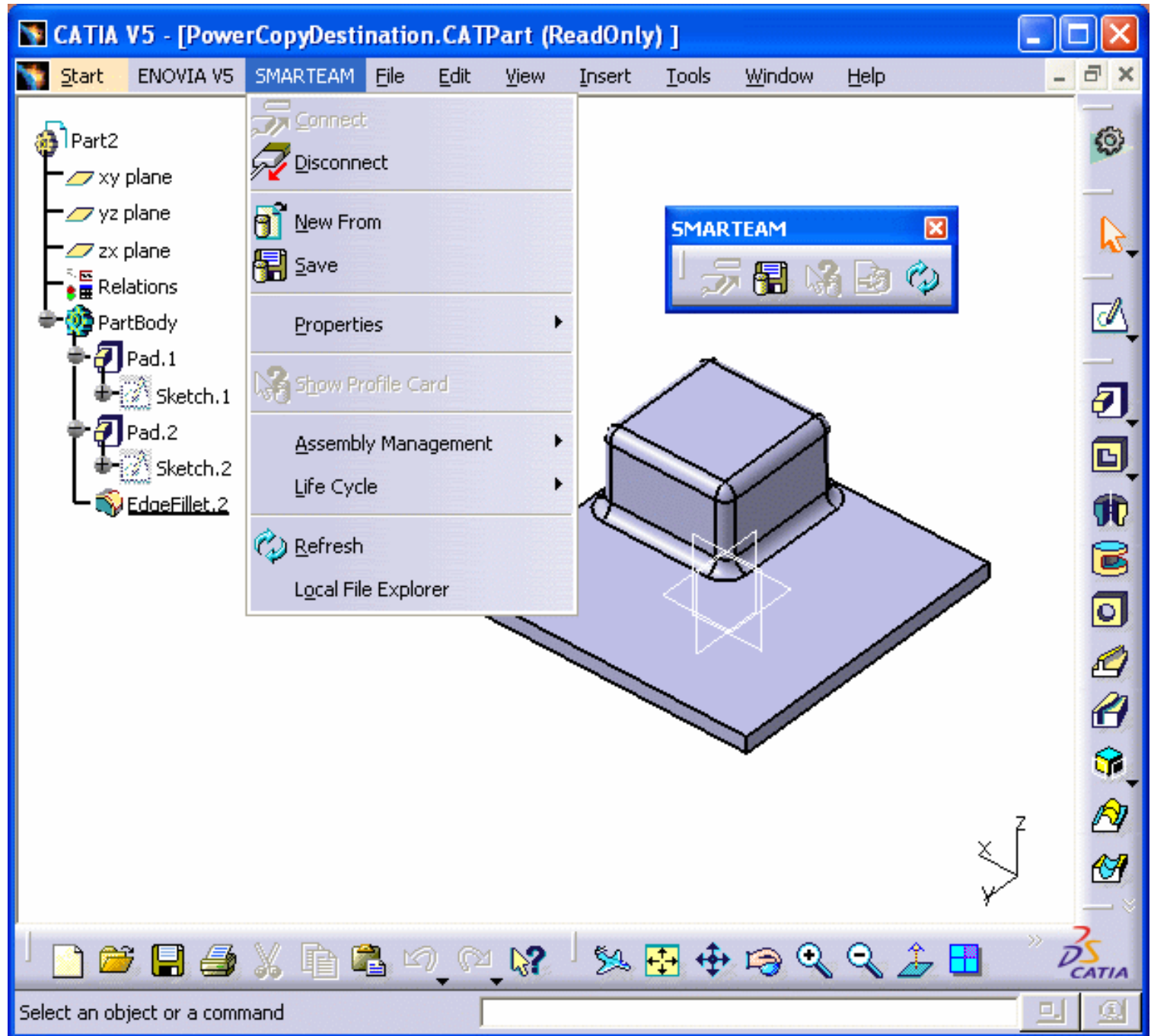
Replace with Selected Revision

1. In CATIA specification tree, from your assembly select the document you wish to replace.
2. Select **SmarTeam > Assembly Management >  Replace with Selected Revision**.
The dialog box that appears, displays all revisions related to the document to be replaced.
3. Select the revision you want to use.
The **Copy File** dialog box is displayed.
4. Click **OK**  to confirm the change.



Interface Description

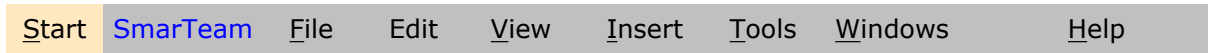
The SmarTeam - CATIA Web Integration Product interface looks like this:



For more information about the items of the **SmarTeam** menu bar or toolbar, either click on the item concerned in the image above or go into one of the two sections below.

[SmarTeam Menu](#)
[SmarTeam Toolbar](#)
[Modifying a Catalog](#)

SmarTeam Menu



SmarTeam

| SMARTeam | File | Edit | View | For | See |
|--|------|------|------|---------------------|--|
|  Connect | | | | Connect | Connecting to the SmarTeam Database |
| | | | | Disconnect | |
|  New From | | | | Disconnect | Connecting to the SmarTeam Database |
| | | | | Save | |
| Properties | | | | New From | Creating a Document from an Existing One Using New From... |
|  Show Profile Card | | | | Save | Saving Documents |
| Assembly Management | | | | Properties | <ul style="list-style-type: none">• Load from Database Defining Property Mapping |
| | | | | | |
|  Refresh | | | | Refresh | Refreshing your Display in CATIA |
| Local File Explorer | | | | Local File Explorer | Lists all files copied to the <i>view</i> and <i>work</i> directories. For further information on the SmarTeam File Explorer refer to the SmarTeam – Web Editor online help. |

SmarTeam Toolbar

This section describes the icons available in the SmarTeam (CWI) workbench. There is only one toolbar, which lets you access some of the capabilities available from the [SmarTeam Menu](#).



Connect

See [Connecting to the SmarTeam Database](#)



Save

See [Saving Documents](#)



Show Profile Card

See [Showing Profile Cards](#)



Switch to SmarTeam

Launches the SmarTeam - Editor (Web-based) application



Check In

See [Checking In Documents](#)



Refresh

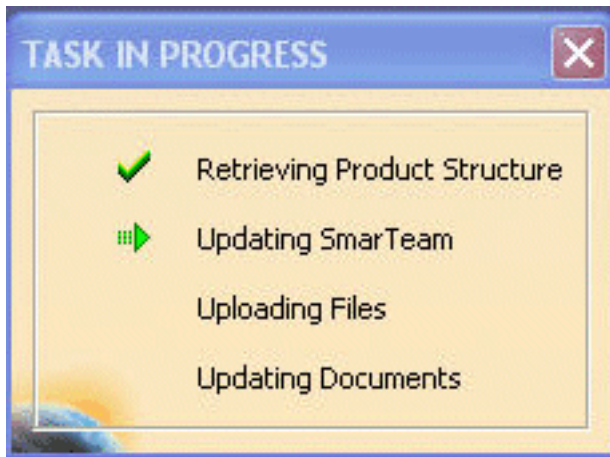
See [Refreshing your Display](#)

Progress Indicator

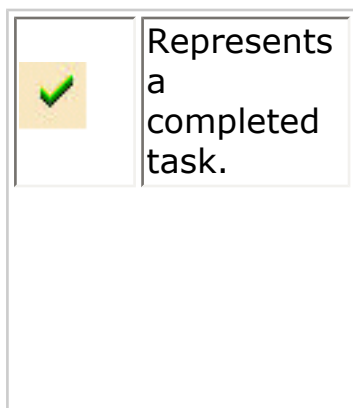
When performing certain operations in CATIA - Web Integration, a Progress Indicator window appears. These operations are:

- New From...
- Save to SmarTeam
- Lifecycle operations
 - Copy
 - Check out
 - Check in
 - New Release
 - Release
- Open
- Refresh

The Progress Indicator presents the tasks that are being performed. A sample Progress Indicator for a Check In operation is shown below:



The following symbols appear in the Progress Indicator window:





Represents
a task that
is
currently
in
progress.



Customizing

A certain number of settings is available to let you customize your session.

The customization you perform is stored in permanent setting files, meaning that these settings are not lost when you end your session.

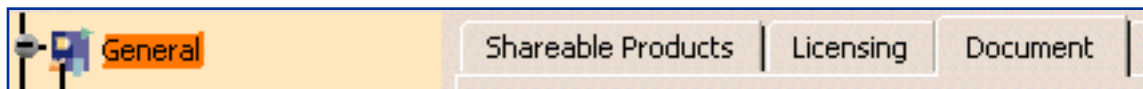
To access them, proceed as follows:



1. Select **Tools** > **Options**.

The **Options** dialog box displays.

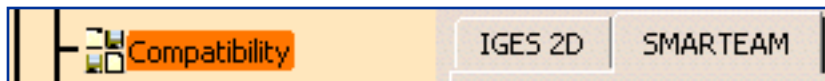
2. Select the **General** category in the left-hand box, if not already selected.
Various tabs are displayed.



The **Document** tab provides options dealing with:

- [Document Environments](#)
- [Linked Document Localization](#)

3. To access settings about compatibility, select the **Compatibility** category in the left-hand box.



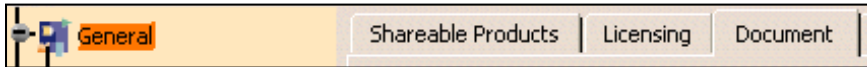
The **SmarTeam** tab provides options dealing with:

- [User Interface](#)
- [Connection](#)
- [Expose Mode](#)
- [CATIA Tree Icons](#)
- [Modifications to Read-Only Documents](#)
- [SmarTeam Web Server](#)

4. Change these options according to your needs.

5. Click **OK**  when done to validate your settings.

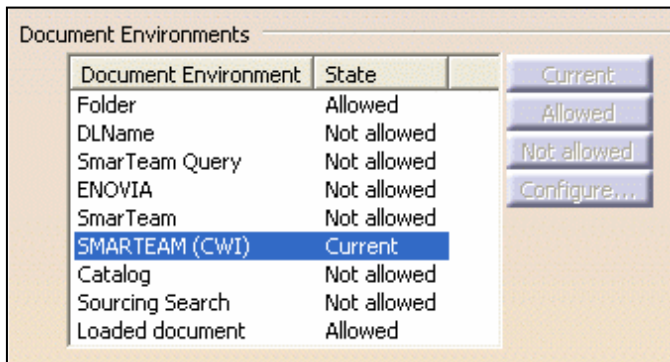
Document



The **Document** tab enables the display of the **SMARTeAM File > Open** user interface. It lets you define options related to the following categories:

- [Document Environments](#)
- [Linked Document Localization](#)

Document Environments



SMARTeAM (CWI)

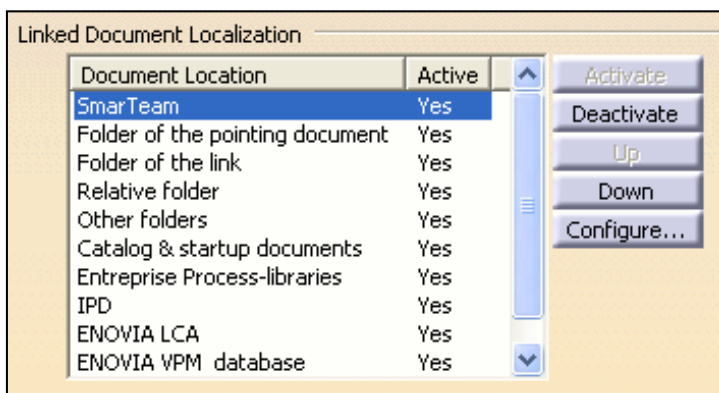
In **Document Environment**, there is a list of work environments you can choose. You can order these environment and activate or deactivate them.

In the context of SMARTeAM - CATIA Web Integration Product application, you need to make the **SMARTeAM (CWI)** environment current. This lets you access documents stored in SMARTeAM.

👉 By default, **SMARTeAM (CWI)** is not current.

All information related to customizing the session is also described in the *CATIA*, *ENOVIA DMU*, *DELMIA Infrastructure* guides.

Linked Document Localization



When working with documents from the SMARTeAM database, the documents are first extracted from the vault and copied to a local folder then loaded in the CATIA session.

There are two main folders:

- the **work** directory containing all files that are checked out or copied out of the vault
- the **view** directory containing all files that are just viewed.

The consequence of this behavior is that each time a document is linked to another document, the linked document should be searched for in the folder containing the pointing document. You can easily set this behavior in CATIA by following these instructions:

SmarTeam

Set **SmarTeam** as the active document location to define the personal strategy you want to be used to locate your linked documents and therefore resolve document links. For this, if necessary, select the **Activate** button to activate this strategy, and then the **Up** button to place this strategy at the top of the list.

Folder of the pointing document

Set **Folder of the pointing document** as the current folder of your documents. For this, if necessary, select the **Activate** button to activate this strategy, and then the **Up** button to place this strategy as the second item of the list.

Folder of the link

Set **Folder of the link** as the third document location in the list. For this, if necessary, select the **Activate** button to activate this strategy, and then the **Up** button to place this strategy as the third item of the list.

Note

When extracting documents from the vault, you can also specify different *Work* directories for each extracted file (by changing the default directory in the life-cycle window). In this specific case, the **Folder of the pointing document** strategy will not be able to recover the linked documents. Another strategy, called **SMARTEAM Database** has to be used. This strategy will ask the SMARTEAM database where the linked document has been copied.

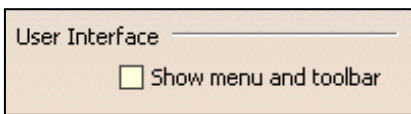
SMARTTEAM (CWI)



The **SMARTTEAM (CWI)** tab lets you define options related to the following categories:

- [User Interface](#)
- [Connection](#)
- [Expose Mode](#)
- [CATIA Tree Icons](#)
- [Modification to Read-Only Documents](#)
- [SMARTTEAM Web Server](#)

User Interface

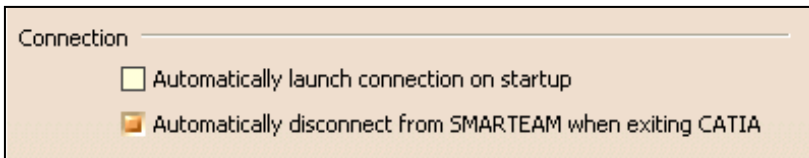


Show Menu and Toolbar

Select this option to display SMARTTEAM CWI menu and toolbar in your CATIA session. Once enabled, you need to restart CATIA to take this change into account.

🔴 By default, this option is not selected.

Connection



Automatically launch connection on startup

Select this option to ensure that the connection to SMARTTEAM CWI is enabled on startup.

🔴 By default, this option is not selected.

Automatically disconnect from SMARTTEAM when exiting CATIA

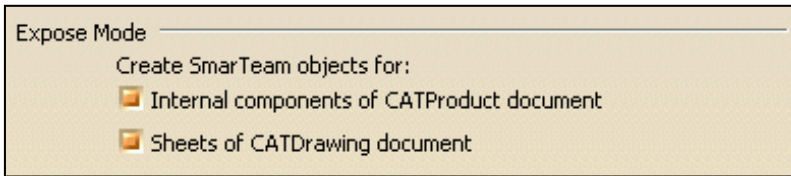
Select this option to ensure that you disconnect from SMARTTEAM CWI when you are closing CATIA.

🔴 By default, this option is selected.

Expose Mode

Create SMARTTEAM Objects for:





These two options are associated with save and check in operations.

Internal components of CATProduct documents

If activated, you create SMARTEAM objects corresponding not only to CATProduct documents but also to their different internal components.

For more about CATProduct components, refer to the *Product Structure User's Guide*.

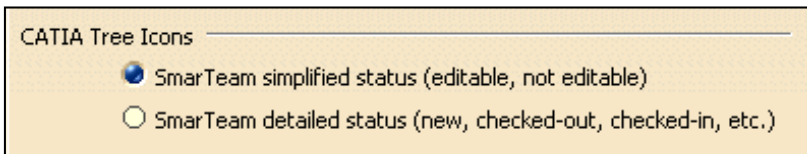
▶ By default, this option is selected.

Sheets of CATDrawing documents

If activated, you create SMARTEAM objects corresponding not only to CATDrawing documents but also to their sheets.

▶ By default, this option is selected.

CATIA Tree Icons



Icons in CATIA Specification tree can display additional information about the status of the documents in SMARTEAM. This additional information is always displayed on the top right side of the icon.

SMARTEAM simplified status (editable, non-editable)

Select this option to ensure icons identifying editable and non-editable documents are displayed in CATIA specification tree. Editable documents are either new or checked-out documents. Non-editable documents are either checked-in or released documents.

This display mode not only simplifies the display but also improves the application performances.

Example of a non-editable CATProduct document



Example of an editable CATProduct document



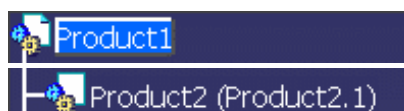
▶ By default, this option is selected.

SMARTEAM detailed status (new, checked-out, checked-in, etc.)

Select this option to ensure icons identifying all possible statuses for documents (new, checked-out, checked-in, released etc.), are displayed in CATIA specification tree. This display mode provides a comprehensive view of the documents.

Example of all possible statuses for a CATProduct document:

- Unknown to SMARTEAM
- New



- Checked In
- Checked In, Modified
- Checked In, Dirty
- Checked In, Not Latest
- Checked In, Not Latest, Modified
- Checked In, Not Latest, Dirty
- Checked Out
- Released
- Released, Modified
- Released, Dirty
- Released, Not Latest
- Released, Not Latest, Modified
- Released, Not Latest, Dirty
- Obsolete
- Obsolete, Dirty

| | |
|--|-------------------------|
| | Product3 (Product3.1) |
| | Product5 (Product5.1) |
| | Product5 (Product5.1) |
| | Product4 (Product4.1) |
| | Product14 (Product14.1) |
| | Product6 (Product6.1) |
| | Product7 (Product7.1) |
| | Product8 (Product8.1) |
| | Product5 (Product5.1) |
| | Product10 (Product10.1) |
| | Product9 (Product9.1) |
| | Product14 (Product14.1) |
| | Product11 (Product11.1) |
| | Product12 (Product12.1) |
| | Product14 (Product14.1) |

By default, this option is not selected.

Modification to Read-Only Documents



Modification To Read-Only Documents

☐ Display Check-Out/New-Release dialog box on first modification

Display Check-Out/New Release dialog box on first modification

This option controls the application's behavior whenever you modify read-only documents. If this option is on, the **Check out** or the **New Release** dialog box automatically displays as soon as you have modified documents. Otherwise, you need to explicitly invoke the **Check In** or **New Release** capability.

By default, this option is selected.

SMARTeam Web Server



SMARTeam Web Server

Server URL

Server URL

This field specifies the server's name. In case a SMARTeam database is already installed, the server's name is displayed in this field.

Administration Tasks

This section assists administrators in configuring their environments to fit their specific needs. The following topics are discussed:

[Defining Property Mapping](#)

[Defining Property Mapping for the Revision Block](#)

[File Naming](#)

Defining Property Mapping



Mapping properties between CATIA and SmarTeam is one of the key features of CATIA-SmarTeam integration. Properties can be mapped on both directions: from CATIA to SmarTeam and from SmarTeam to CATIA. Some CATIA actions perform mapping on only one direction, other on both directions. This mapping definition is usually performed by the administrator.

This page deals with the following issues:

- [When are Properties Automatically Mapped?](#)
- [Integration Tools Setup Utility](#)
- [Using the Integration Tools Setup Utility to Define Properties Mapping](#)
- [Using 'Load From Database'](#)

When are Properties Automatically Mapped?

Once the mapping definition has been done by the administrator, CATIA users can then take benefit of this during their daily works. To be more precise, properties are mapped automatically:

- during first **SmarTeam > Save**, properties are mapped on both directions (properties are first mapped from CATIA to SmarTeam, then profile-card are displayed, then properties are mapped from SmarTeam to CATIA) during:
- next **SmarTeam > Save**, properties are mapped only from CATIA to SmarTeam
- **Check In** or **Release**, properties are mapped from CATIA to SmarTeam
- **File > Open**, properties are mapped from SmarTeam to CATIA
- **Check Out** or **New Release**, properties are mapped from SmarTeam to CATIA
- **SmarTeam > Properties > Load From Database**, properties are mapped from SmarTeam to CATIA

Load From Database

In some scenarios, you may want to update manually CATIA properties from SmarTeam values. For instance, if you have modified some attributes in a SmarTeam profile-card, you may want to update the CATIA properties accordingly.

This is possible by using the **SmarTeam > Properties > Load From Database**. Selecting this capability automatically retrieves all SmarTeam properties of current document and updates corresponding CATIA attributes according to the mapping definition.

Integration Tools Setup Utility

The most commonly used tool to define properties mapping is the **Integration Tools Setup** utility. This utility defines the settings of the integration between SmarTeam and an external CAD system. One of these settings is the properties mapping.

Using **Integration Tools Setup** utility, the administrator can define mappings between CATIA information and SmarTeam properties. In addition to this mapping, the direction is also set (mapping only from CATIA to SmarTeam or from SmarTeam to CATIA or both).

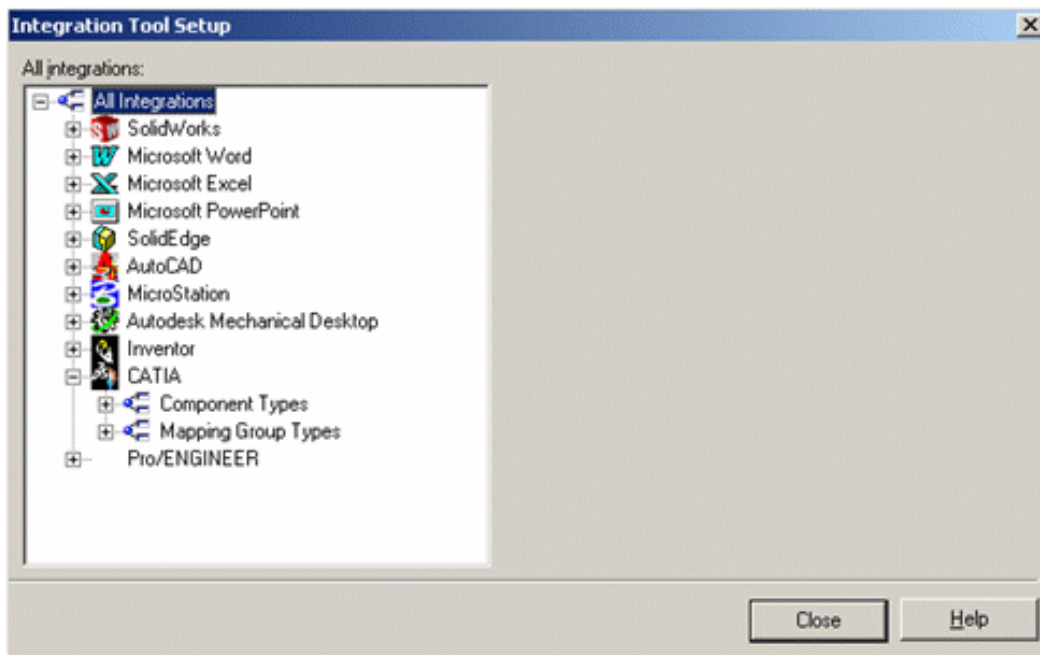
Using the Integration Tools Setup Utility to Define Properties Mapping

The **Integration Tool Setup** utility is available on the SmarTeam server (it may also be available on SmarTeam client machines). For reference information about the **Integration Tool Setup** utility, see the SmarTeam documentation.

1. To launch it, select **Start > All Programs > SmarTeam > Administrative Tools > Integration Tools Setup**.

After the database selection and the user login, the main window is displayed.

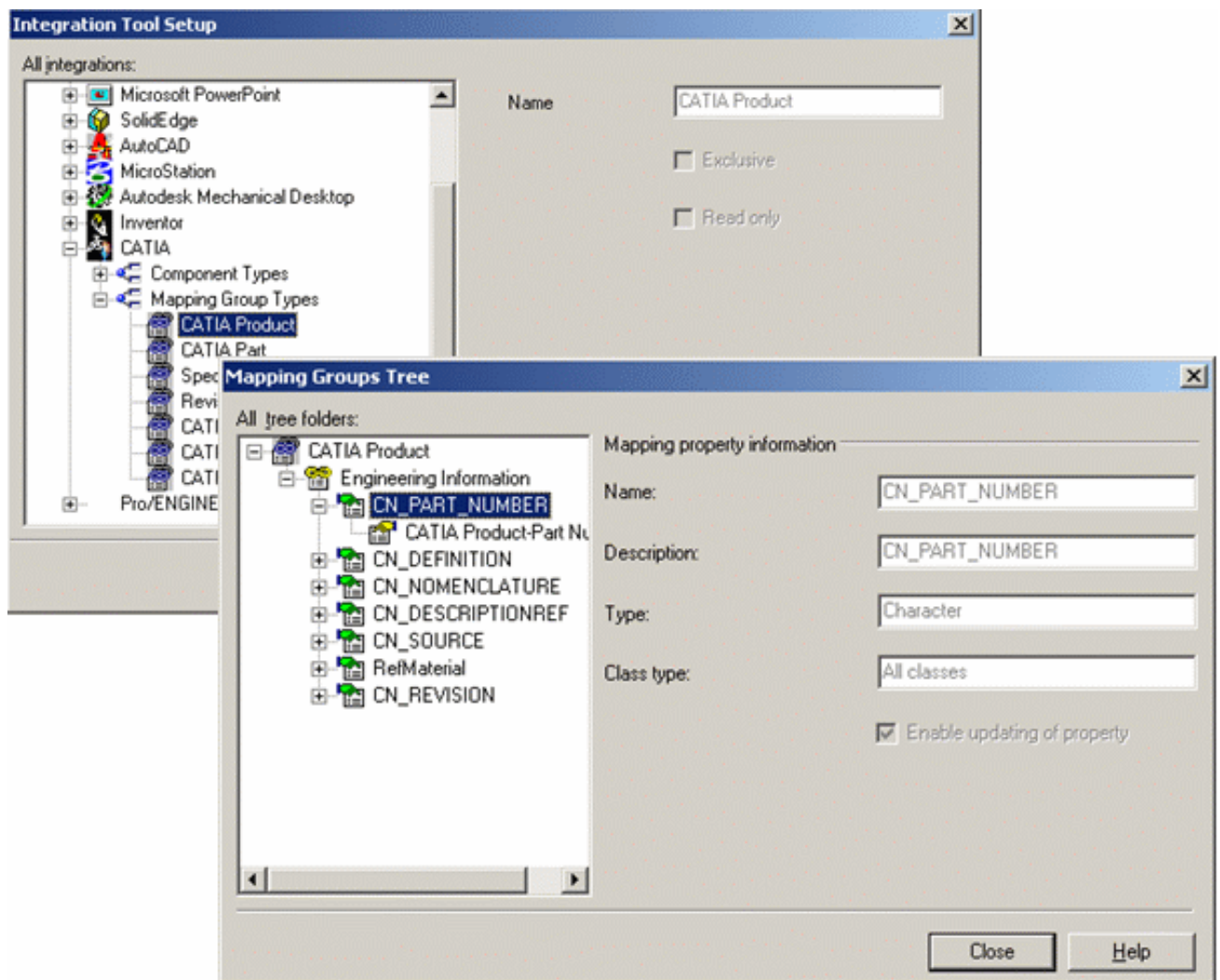
2. Locate and open the CATIA; section. You will see two categories:
 - **Component Types**
 - **Mapping Group Types**: dedicated to the definition of properties mapping.



Mapping Group Types and **Mapping Groups** classify all single mappings.

The **Mapping Group Types** category contains a list of Mapping Group Types;. Then, each Mapping Group Types groups "Mapping Groups". Each Mapping Groups groups "Mapping Properties". Finally a Mapping Property defines:

- the name of the CATIA information
- the corresponding SmarTeam attribute
- the mapping direction



Each CAD integration (including CATIA integration) need to define their rules for the classification of "Mapping Properties" under dedicated "Mapping Group Types" and "Mapping Groups".

Mapping Directions

| Directions | Definition |
|---|--|
| From SmarTeam to CATIA (unidirectional) | Attributes controlled by SmarTeam like revision, state or ID should be mapped in this direction only. |
| From CATIA to SmarTeam (unidirectional) | This is dedicated for CATIA attributes (like product properties) that you want to store in the database (for query purpose for instance) |
| Both (bidirectional) | This kind of mapping is possible but is not recommended because it may be difficult to the user to know where (in CATIA or in SmarTeam) is the official value of the property. |

Units Management

CATIA attributes may have units: for instance, a $f(x)$ parameter representing a length may be associated to the "Length" magnitude. Then, the user is able to decide which unit will be used to display the attribute in CATIA dialogs.

Note that changing the displayed unit does not change the value inside the CATIA document: CATIA attributes are always stored with a defined unit (for instance, weights are stored in kilogram)

Unfortunately, SmarTeam attributes cannot be associated to magnitudes or units.

It has been decided that the mapping between CATIA and SmarTeam will be always performed based on the CATIA stored unit.

Rules for Mapping Definitions

We consider that in CATIA there are many groups of attributes, all of them with the same semantics: product properties, drawing sheet properties, user parameters.

We will use **Mapping Group Types** to define the mapping of each of these groups of attributes. **Mapping Group Types** correspond to these groups of CATIA attributes. For instance, we will have a **Mapping Group Type** for the definition of the mapping of all product properties and another one for the definition of all user parameters, a third one for the future document properties;

Some of these **Mapping Group Type** will be available for all CATIA documents. This is the case for instance for the Mapping Group Type containing the mapping of the user parameters. Other Mapping Group Types will be available only for some specific CATIA documents. For instance, the Mapping Group Type defining the mapping of product properties will be only available for CATProduct and CATPart document.

Even if a **Mapping Group Type** is dedicated to all CATIA documents, the customer may want to define specific mappings for specific documents and other mapping for other documents. This can be done by defining specific **Mapping Groups** for specific documents:

- if mappings are defined in a Mapping Group named "All", they will be available for all documents
- if mappings are defined in a Mapping Group which name correspond to a CATIA document type (for instance "CATProduct; or CATPart), they will be available only for this specific type of document.

Following is a subset of known CATIA document types:

- CATProduct
- CATProduct_ASMPRODUCT (a specific type dedicated to product internal components)
- CATPart
- CATDrawing
- CATDrawing_Sheet (a specific type dedicated to drawing sheets)
- CATProcess

Notes:

- Every document will take benefit of the "All" mapping group and also of the dedicated mapping group corresponding to its document type
- If the user decides to arrange his mappings in various mapping groups, then a mapping property defined in "All" mapping group should not be declared anywhere else (because both mappings will be run and we cannot know in which order they will be called)

CWI Mapping Group Types

Following is the list of all Mapping Group Types recognized by CWI.

Intrinsic Properties

This Mapping Group Type is used to define the mapping of all intrinsic properties of documents or

features.

Document and features have some intrinsic properties (properties which are automatically available. Product and Parts have the following intrinsic properties:

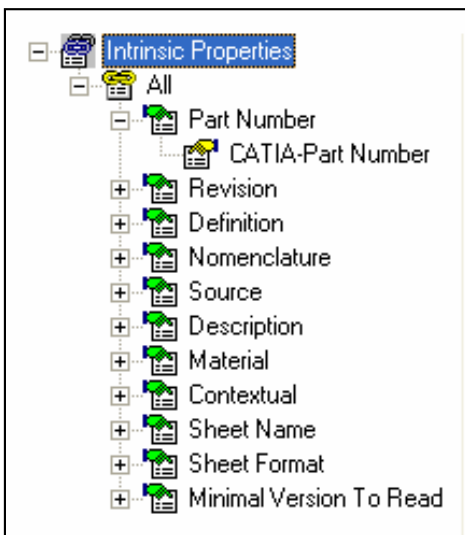
| Properties | Availability |
|----------------------------------|---|
| • Part Number | Available on both product and part |
| • Revision | Available on both product and part |
| • Definition | Available on both product and part |
| • Nomenclature | Available on both product and part |
| • Source | Available on both product and part |
| • Description | Available on both product and part |
| • Material | Material is stored on root-product reference. But in case of internal components, it is stored at the instance level |
| • Contextual | Available on CATPart document. In case it is designed in the context of a CATProduct document, the contextual information is the filename of the CATProduct (without the path) |
| • Sheet Name | Available on drawing sheets. |
| • Sheet Format | Available on drawing sheets. |
| • Minimal Version To read | Available on all CATIA V5 documents. Indicates the minimal CATIA version necessary for reading the document. The string looks like: CATIAV5R14 |

Even if some mapping types are only applicable to specific document types, they can be defined in a single Mapping Group named "All" or they can be defined in Mapping Groups dedicated to specific document types. But don't try to define a Mapping Type named "Sheet Name" in a Mapping Group named "CATProduct". It will be ignored because "Sheet Name" is not a recognized name in the context of a "CATProduct" document !

Example

In this example, all intrinsic properties are defined in a single Mapping Groups named "All". It means that these mappings will be applicable to all documents.

Of course, if one MT is not applicable to current in-work document, it will just be skipped for this document.



Product Properties

This Mapping Group Type is used to define the mapping of the Product other properties

These properties are created in CATIA with the **Define other properties;;** button that is displayed in the Product tab of the **Product** properties dialog. Once created, they are displayed on the ;Product; tab of the ;. There are also displayed in the Formulas; dialog with a Properties; prefix in front of their name.

This Mapping Group Types can be applied on CATProduct document, Internal Product of CATProduct and CATPart document.

As usual, mapping properties can be created inside a single Mapping Group named "All" or inside a Mapping Group which name correspond to a specific document type.

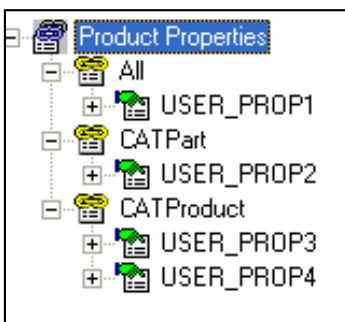
Mapping Property name correspond to the exact name of the CATIA property.

Example

In this example, we have one product property called 'USER_PROP1' which is applied on all documents.

Then, we have one product property called 'USER_PROP2' which is dedicated for CATPart documents.

The two last product properties are dedicated for CATProduct documents.



User Parameters

This Mapping Group Types is used to define the mapping of user parameters.

user parameters = correspond to f(x) parameters created by the user with "New Parameter of type" button of "Formulas" dialog. These properties may be created on various CATIA features (on products for

instance).

NOTE: In a CATDrawing document, user parameters are defined only at the level of the document itself.

Mapping Property name correspond to the exact name of the CATIA user parameter.

Note that only parameters created by the user will be allowed for the mapping. If you want to map an internal parameter (for example the pad's thickness) you will have to create a user parameter that drive this internal parameter and use the name of this user parameter as Mapping Property name.

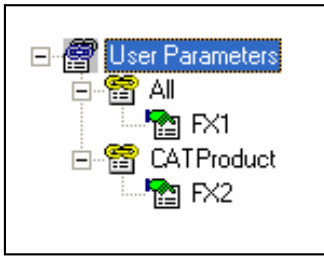
NOTE:

In CATProduct document, User parameters may be defined on the root-product or on internal components.

Example

In this example, we have one user parameter called FX1 which is dedicated to all documents.

Then we have another user parameter called FX2 which is dedicated to CATProduct documents.



NOTE:

If CWI ExposeMode is set to true, parameter will be read at the feature level, then if not found, at the document level. But parameter will always be written at the feature level.

For instance, if we have defined that FX1 is a user parameter which is mapped between CATIA and SmarTeam and if we are currently working on a document which has this user parameter set at the document level (ie: it is set on the root product) and if we are currently running the mapping on an internal component of this document and if this internal component does not have a FX1 user parameter, then, component profile-card will be filled with FX1 value read from the root-product.

If CWI ExposeMode is set to false, parameters will always be read and written at the document level.

A Reserved Mapping Group Type

In addition to the Mapping Group Types described above, "Revision Block" Mapping Group Type has been defined to drive the behavior of the integration code.

"Revision Block" Mapping Group Type is used to define which SmarTeam attributes will be displayed in drawing revision blocks.

Problematic Mapping Definition

As said previously, mapping can be either defined in Mapping Group named "All" or in a Mapping Groups which name correspond to a specific document type.

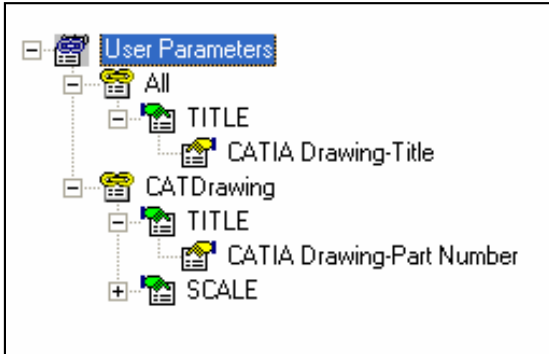
Definition of the same mapping on both locations is not recommended because it will produce unpredictable results.

Example

In this example, we see that 'TITLE' user parameter is defined twice but with different mappings.

At the 'All' level, it is mapped to CATIA Drawing 'Title' field.

At the 'CATDrawing' level, it is mapped to CATIA Drawing 'Part Number' field.




When running a mapping "from CATIA to SmarTeam", there are no particular problem: the CATIA TITLE value will be stored on both locations.

But the problem will occur when running the mapping "from SmarTeam to CATIA". During this process, there may be two SmarTeam values (one from 'Title' field, the other from 'Part Number' field) to be stored in one single CATIA user parameter.

Unfortunately, there is no way to define which Mapping Group will be run first. The 'All' one or the 'CATDrawing' one. This is the reason why the result will be unpredictable.

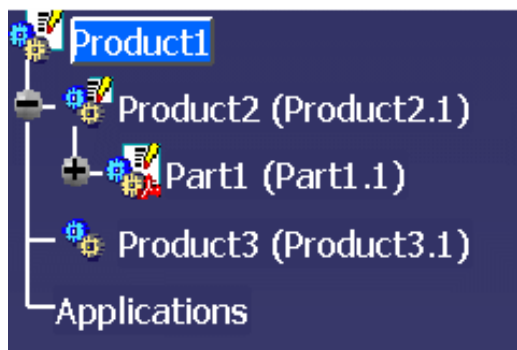
Using 'Load From Database'

Launching the **Load From Database** command performs the properties mapping on all objects belonging to the document of the selected object. For instance, if you have selected one component of a product stored in SmarTeam, all components of the document will be treated. You can select any component belonging to the document, even if selected component is not already stored in SmarTeam.


Only properties mapping from SmarTeam to CATIA will be run (use **SmarTeam** >  **Save** to perform mapping from CATIA to SmarTeam)

After properties mapping has been performed, there is no message indicating that the job has been done.

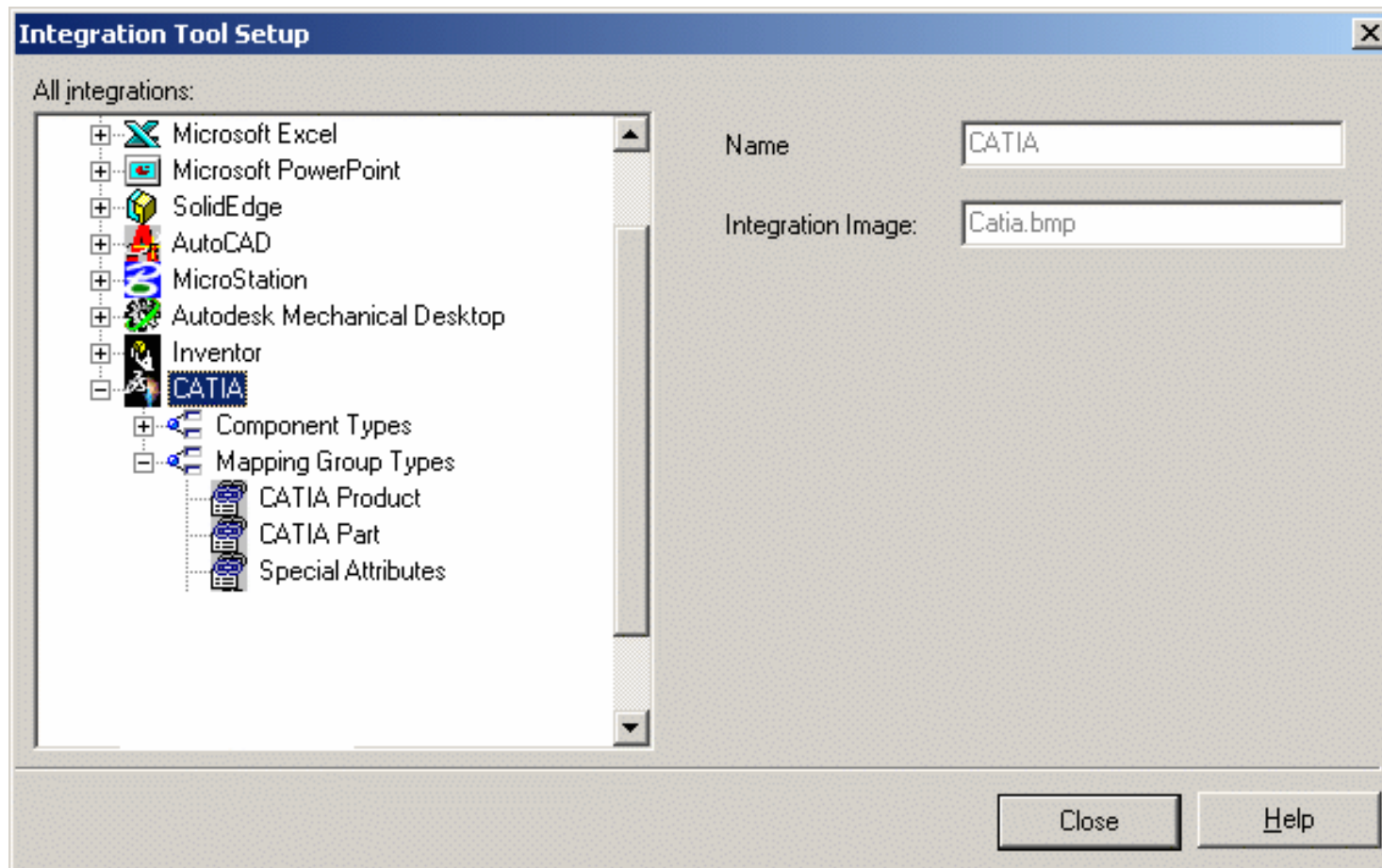
In the following example, if you want to apply **Properties** > **Load From Database** on the root CATProduct document, you need to activate either "Product1" or "Product2".



Defining Property Mapping For the Revision Block

 This section describes how to define property mapping for revision blocks.

-  1. Launch the **Integration Tool Setup** utility.
The Integration Tool Setup dialog box appears:

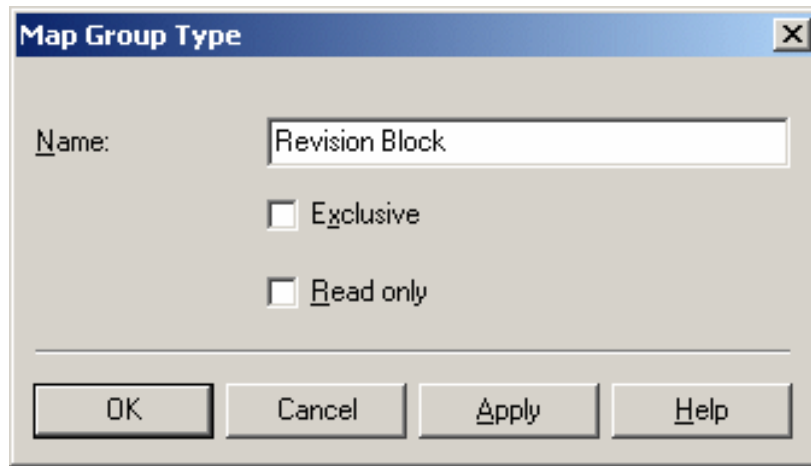


A **mapping group type** i.e. a set of mapping groups, applicable to a specific SmarTeam class must now be defined that corresponds to the CATIA Drawing class.

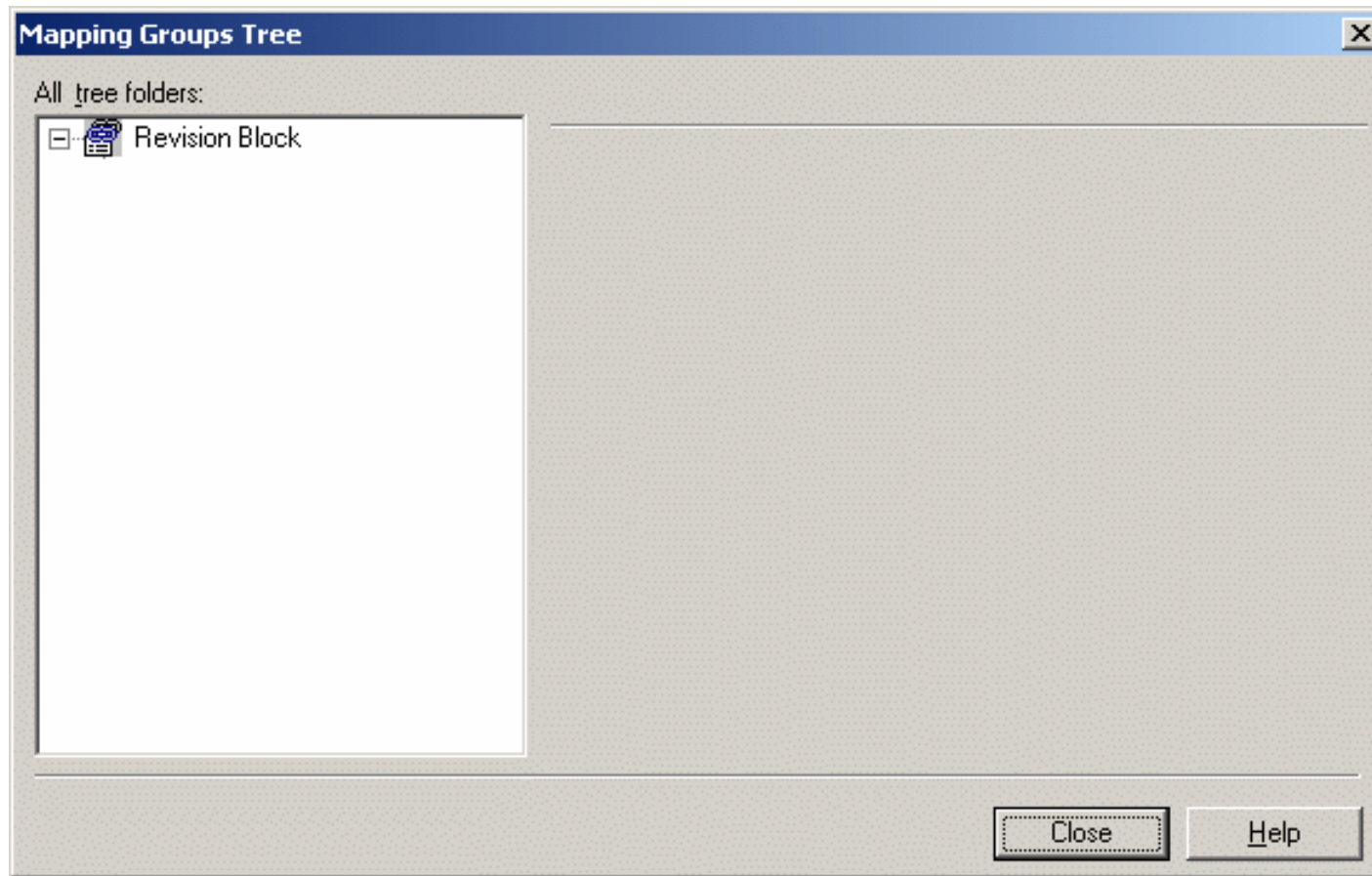
2. In the tree displayed, right-click on **Mapping Group Types** and select **Add Mapping group type**.

The Map Group Type dialog box appears.

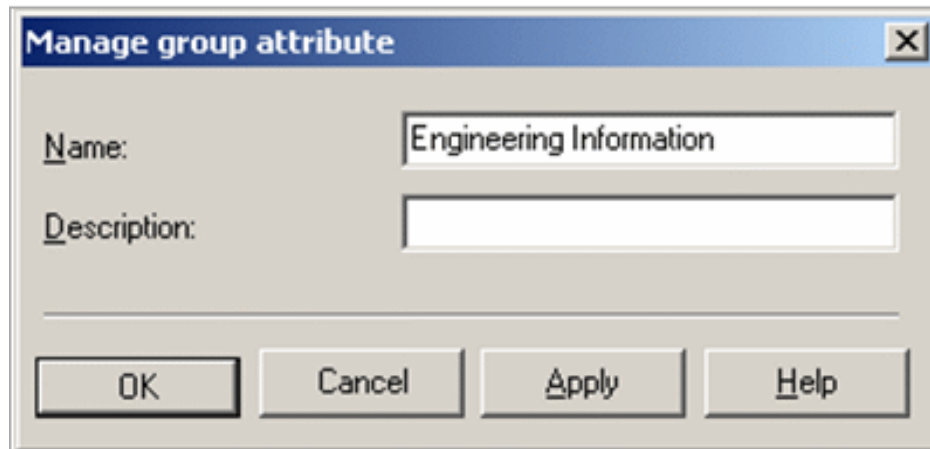
3. In the **Name:** field, enter the group type name *Revision Block* as shown:





4. Click **OK**.
5. In the tree, right-click on the **Revision Block** item and select **Open Groups Tree**.
The Mapping Groups Tree dialog box appears.

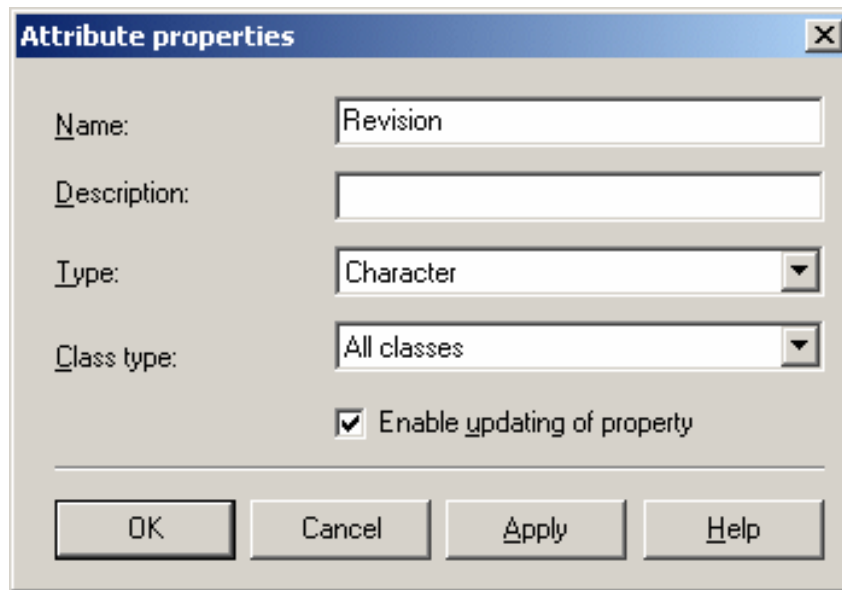


6. In the tree, right-click on **Revision Block groups** and select **Add**.
The Manage Group Attribute dialog box appears.
7. In the **Name:** field, enter *Engineering Information*:




8. Click OK.
9. To define the CATIA Formula property  as **Type**: Go to the tree and right-click *Engineering Information*
10. Select **Add**.
The Attribute property dialog box appears.
11. Enter **Revision** in the **Name**: field.
12. Check the box **Enable updating of property**.

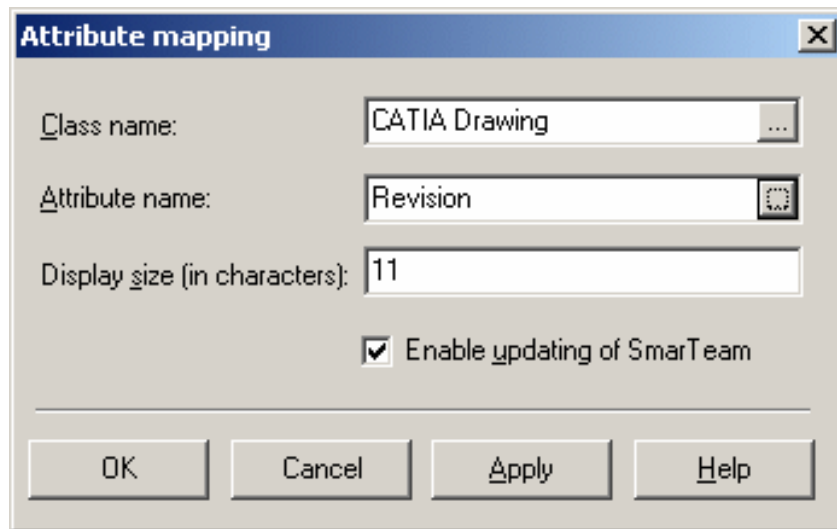
A CATIA Formula property  can thus be updated after a change to a SmarTeam attribute.

The image shows a 'Attribute properties' dialog box with a blue title bar and a close button. It contains four labeled text fields: 'Name' with 'Revision', 'Description' (empty), 'Type' with 'Character', and 'Class type' with 'All classes'. Below these is a checked checkbox labeled 'Enable updating of property'. At the bottom are four buttons: 'OK', 'Cancel', 'Apply', and 'Help'.

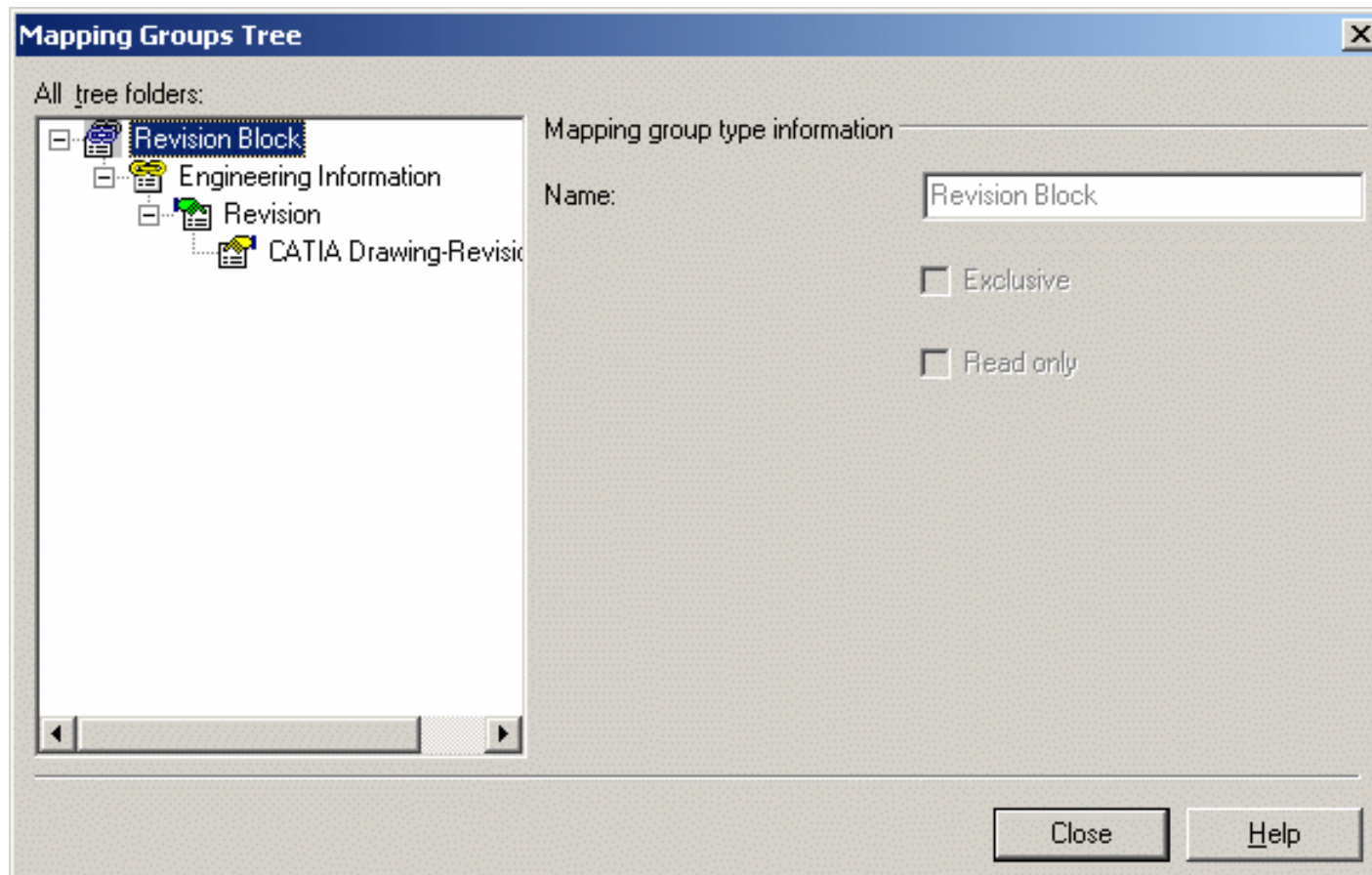
13. Click **OK**.
14. Go to the tree and right-click on **Revision** then select **Add**.

This defines as **Drawing Type** the SmarTeam attribute to be linked to the CATIA Formula property 
The Attribute Mapping dialog box appears.

15. Select the class name **CATIA Drawing**.
16. Select the attribute name **Revision**.
Note that the list of the possible attributes is specific to the selected class name.



17. Click **OK**.
The Revision Block Mapping Groups tree dialog box appears:




The following table is a summary of the steps that have just been performed.

| Mapping Property Name | Enable Property Update | Mapping Attribute Name | Enable SmarTeam Update |
|-----------------------|------------------------|------------------------|------------------------|
| Revision | yes | Revision | |

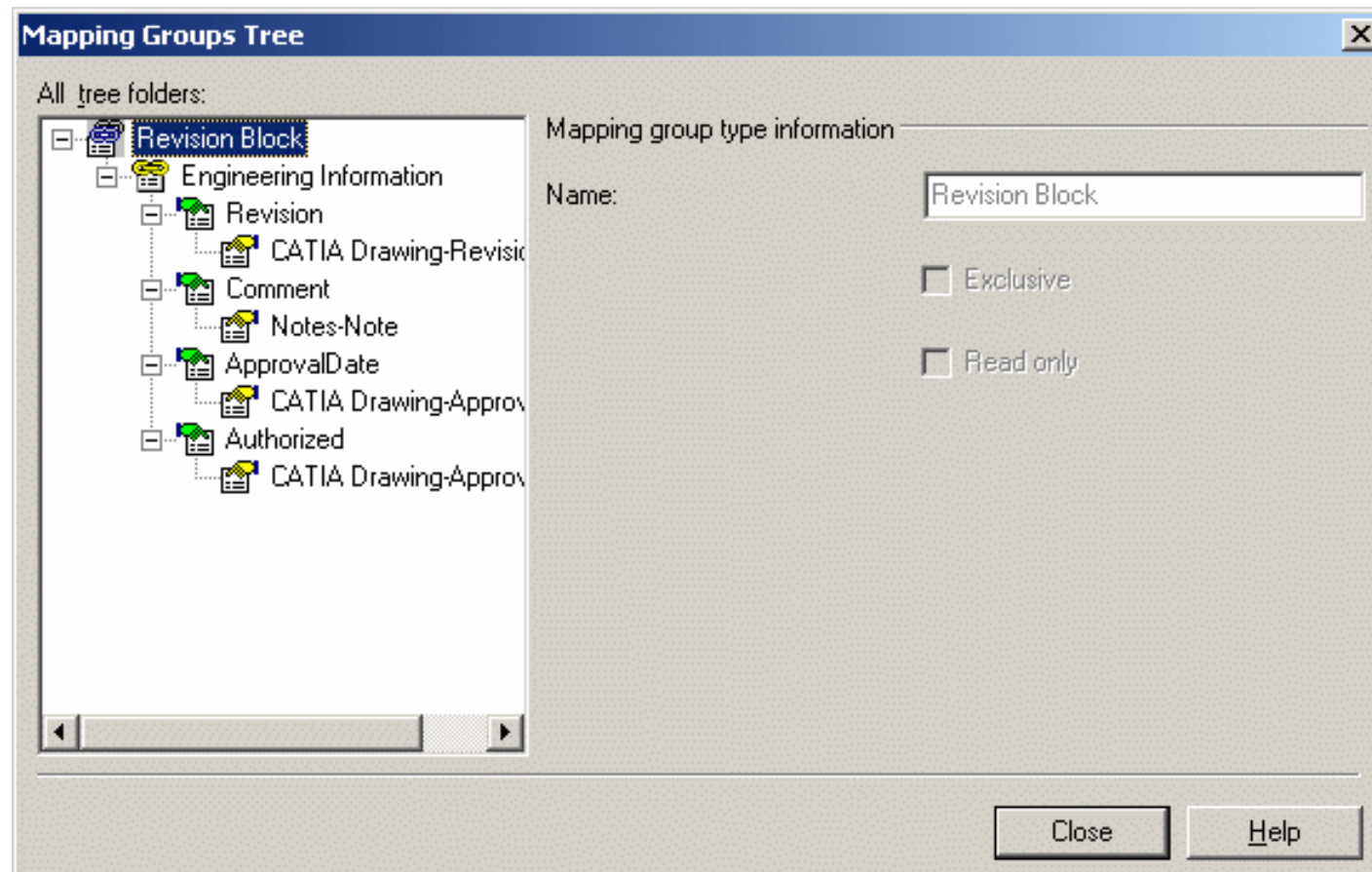
18. Repeat steps 9 through 16 for all other mapped properties.

Here is a list of all the other properties to be mapped:

| Property Name in CATIA  | Enable Property Update | Attribute Name in SmarTeam | Enable SmarTeam Update |
|---|------------------------|-------------------------------|------------------------|
| | | | |

| | | | |
|--------------|-----|---------------|--|
| Comment | yes | Note | |
| ApprovalDate | yes | Approval Date | |
| Authorized | yes | Approved By | |

Once this has been done the Revision Block Mapping Groups tree dialog box should look like this:



As you can see, mapping of the CATIA properties and the SmarTeam attributes has now been completed.



File Naming



CATIA documents often reference other CATIA documents. A simple example may be a part drawing, where the drawing document points to the part document. A more complex example is file relationships created from contextual design.

The fact that document references are based on file names puts a strong emphasis on proper naming of your CATIA files.

An example of a problematic scenario is a situation where one designer, Joe, creates a bearing and names its part file Bearing while another designer, Mark, creates a different bearing size, yet still names its file as Bearing. Everything may work well, until both bearings have to be placed in the same product. This causes a conflict at the CATIA level and the user will have to rename one of the files. As mentioned above, renaming files can cause many problems. Documents that are looking for the file name may lose the link to this renamed part.

Recommendation

Therefore, it is recommended that SmarTeam generates the file name. It is also recommended that the filename includes a sequential number that ensures uniqueness.



Methodology

Methodology and conceptual information about the following topics are provided in this section.

[Tools For Working in a Concurrent Engineering Environment](#)

[Enriched Decision Support with All V5 Links](#)

[The User Working Area](#)

[Handling Catalogs](#)

[Handling CATAnalysis Documents](#)

Tools For Working in a Concurrent Engineering Environment



SmarTeam - CATIA Integration facilitates working concurrently. This is performed by:

- **displayed information:**
 - **non-latest revision** status
 - **modified and read-only** status
- **operations:**
 - **refresh**
 - **check-out on the fly**

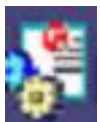
Thus, working concurrently is easier:

- by promoting:
 - the opening of documents without any check-out (View)
 - the check-out using the CATIA **SmarTeam > Life Cycle > Check-Out** command or by means of a check-out on the fly
 - the use of the **Refresh** command (to update the product structure and desk tree icons)
- thru easy access to the product structure tree and, in File Desk, to the *read-only and modified* status.

Displayed Information

Non-Latest Revision Status

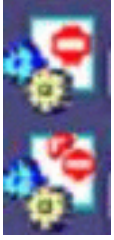
If a document is opened in CATIA with a revision that is not the latest, the SmarTeam information is displayed in the product structure and file desk trees.



Modified and Read-Only Status


If a checked-in or released document is opened in CATIA and is modified, a


specific icon mask is displayed in the CATIA product structure and file desk trees.



Operations

Tree Refresh

The CATIA **SmarTeam** >  **Refresh** command updates the product structure and file desk icons with regard to SmarTeam information.

This is particularly useful for concurrent engineering. For instance, if you are using checked-in parts within their assembly, you can use  **Refresh** to check whether its loaded parts correspond to the latest revision. Another user may have performed a check-out, modifications and a check-in while the first user is still using the same file revision.

Note :

The **Tree Refresh** capability updates the display of icons representing documents in the specification tree (but does not update your session).

Check-Out on the Fly

After an initial modification on a checked-in or released document, a window is displayed asking you if you want to perform a check-out on the fly operation. This functionality enables you to perform check-outs only when necessary and thus contributes to concurrent engineering.



Enriched Decision Support with All V5 Links



There is a large number of documents and relations between CATIA, DELMIA and ENOVIA documents. The native SmarTeam integration of CATIA link semantics enables dedicated behavior on each link type. It provides an easier navigation thanks to a dedicated link type display (icon), and makes it possible to filter document relations display.

The benefits are as follows:

- easier manipulation
- fewer interactions
- reduced data flow between vault & local disk
- reduction of the file brought onto the local disk
- facilitation of concurrent engineering.

For example, viewing a .catalog document does not bring the referenced Parts onto the local disk. Another example is the handling of CATAnalysis documents: editing a CATAnalysis file by default checks out and copies the results (CATAnalysisResult and CATAnalysisComputation files) to the local disk.

CATIA Links



Enriched decision support is provided by means of the native SmarTeam support of V5 links summarized as follows:

| Icons | Links |
|-------|--|
| | CATIA Product Link (P) |
| ➤ | CATIA Design Link (D) |
| --➤ | CATIA Rule Base Link (RUL) |
| --➤ | CATIA Design Table Link (DT) |
| -➤ | CATIA Downstream Application Link (DA) |
| --➤ | CATIA Reference Link (REF) |
| --➤ | CATIA Contextual link (C) |
| --➤ | CATIA Result Link (RES) |
| | CATIA Is Composed Of (IS) |

Here is the list of CATIA links natively supported in SmarTeam:

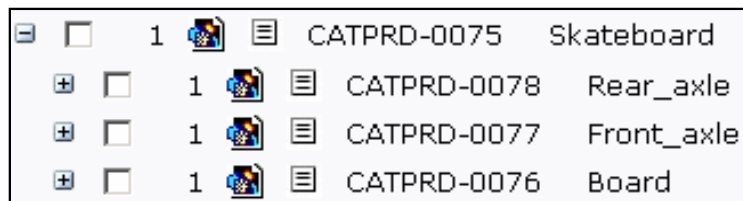
- [Product Structure](#)

- Is Composed of
- Design
- Downstream Application
- Contextual
- Design Table
- Result
- Rule Base
- Reference
- Product Structure link

The *Product Structure* link plays a part in the BOM structure. It is displayed as a standard hierarchical link.

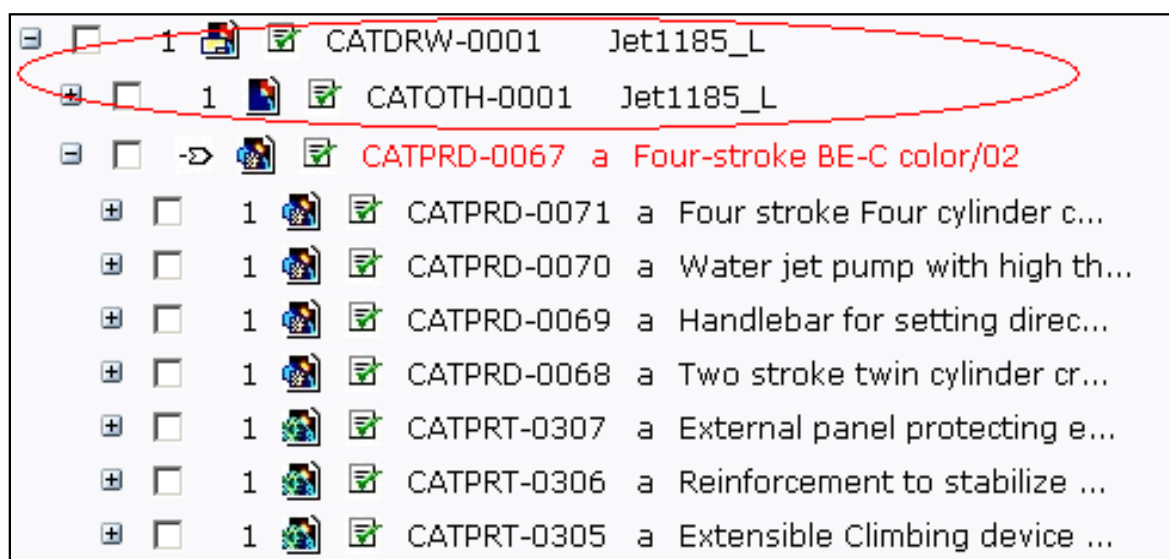
Example:

CATProduct->CATProduct, CATProduct->CATPart, CATProduct->Internal Component...



- Is Composed Of link

The *Is Composed Of* link is an aggregation link but which plays no part in the BOM structure. It is displayed as a standard hierarchical link.



Example:

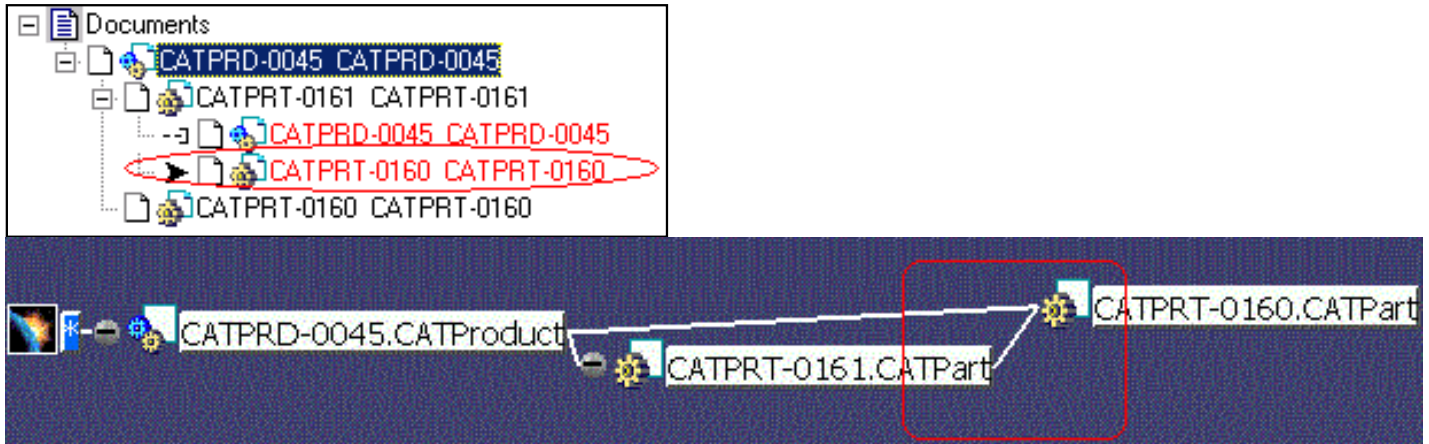
CATDrawing->Sheet, Catalog->catalog

- Design link

The *Design* link plays a direct part in the design process.

Example:

copy-paste with link & import link (*CATPart->CATPart*, *CATPart->model*), link to external parameters (formulas), MultiCAD links (*CATProduct/CATPart->MultiCAD document*) etc.

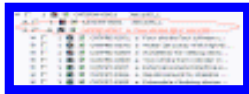


- Downstream Application link

The *Downstream Application* link is a link between a downstream application and the design data

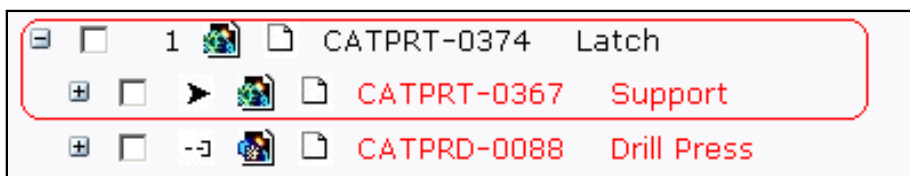
Example:

CATDrawing->CATProduct/CATPart, *CATProcess->CATProduct/CATPart*, *CATAnalysis->CATProduct/CATPart*



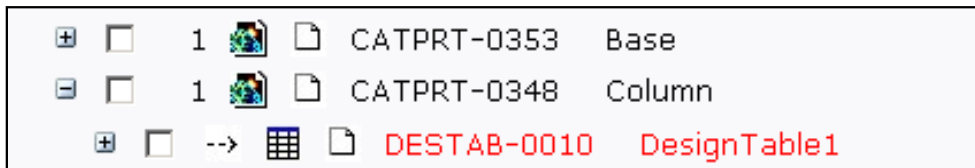
- Contextual link

The *Contextual* link is a link between the contextual Part and its product context (*CATPart->CATProduct*).



- **Design Table link**

The *Design Table* link is a link between a CATIA document and its design table (Excel or text file).



- **Result Link**

The *Result* link corresponds to CATIA output links:

Example:

CATAnalysis->CATAnalysisResult, CATAnalysisComputation...

CATProcess->NC (aptsources, CATNCCode, tlp)

CATProcess documentation link (html)

Knowledge optimization link (CATIADocument->excel/text)

- **Rule Base link**

The *Rule Base* link describes a link between an instance of a rule base and its reference.

- **Reference link**

The *Reference* link includes all the CATIA links which do not correspond to the other eight link types:

Example:

CATDrawing->Image/OLE

CATProduct/CATPart->CATMaterial

CATProcess->NC Manufacturing Part (CATPart)



The User Working Area



Single Directory vs. Multiple Directories

As mentioned earlier, CATIA document references are to the physical file name. When a document is opened by CATIA, it searches for its references. When CATIA identifies that it needs to load a reference into memory, it starts to search for the referenced document by name according to the search rules.

This fundamental file-referencing behavior requires clear and reliable methodology for the users working areas. In cases where multiple copies of documents are spread across the network, document references may be loaded from unexpected or inconsistent locations.

CATIA may not allow you to open the same filename from two different directories. Therefore, if a part or product is already loaded from a given directory, this loaded instance will be used for all referencing documents (other drawings, products or parts), even if these referencing documents would have attempted to load the reference from a different directory. Due to this behavior, we recommend that the user's working area is well defined. If you decided to enable more than one area, check your methodology to ensure that a single CATIA session will load documents from the correct location.

Recommendations

- Use a dedicated location as a working area.
- Do not allow access to the working directory of other users. Sharing files between users must be done using vault operations (check in / check out).

Location of Working Directories

Administrators sometimes need to choose between configuring the user's working area locally (on the user's local hard drive) or setting up the working areas on the network.

As far as CATIA is concerned, loading documents from the local hard drive may be faster than loading from the network. Configuring the working directories of the users on the network usually provides:

- Easier ways to backup the working areas (preventing the need to perform check in for backup purposes)
- Better performance copy files from the vault (the working directories are on the same server as the vault server, or the vault server to the file server)

channel is significantly robust)

SmarTeam supports both configurations. However, it is common to find that users who work with local directories perform check in operations on a daily basis in order to back up their work. Sometimes this also applies when the working directories are on the network.

We suggest discussing this issue with your system/network administrator and a SmarTeam consultant. It has been proven that correct setup can significantly improve performance.

Recommendations

- It is highly recommended to perform a backup of the user working directories, whether on the network or local drive. This can be done using one of the various tools available to enable remote backup.
- Check in operations for backup purposes only are problematic in terms of both performance and usability, due to the redundant revisions that are exposed to team members.



Handling Catalogs

Many engineering groups use catalogs of standard parts. This section describes the necessary steps a librarian should follow to enable engineers manipulate catalogs as necessary in CATIA and provides a few recommendations.

- [Authorization Setting for Standard CATIA Parts](#)
- [Setting Up a Shared Directory for CATIA Catalogs](#)
- [Product Resource Management \(PRM\)](#)
- [Creating and Saving CATIA Catalogs](#)
- [Accessing CATIA Catalogs](#)



For reference information about catalogs, see the *CATIA Component Catalog Editor User's Guide*.

SmarTeam helps you manage your [catalogs](#), however, instantiating parts from the catalog must be done while creating a reference and not thru creating a new instance (as multiple documents will be stored in the SmarTeam database for each instantiation).

When you perform lifecycle operations on the catalog, SmarTeam may attempt to retrieve all catalog components from the vault. In the Lifecycle dialog set only the relevant parts of the catalog to copy.

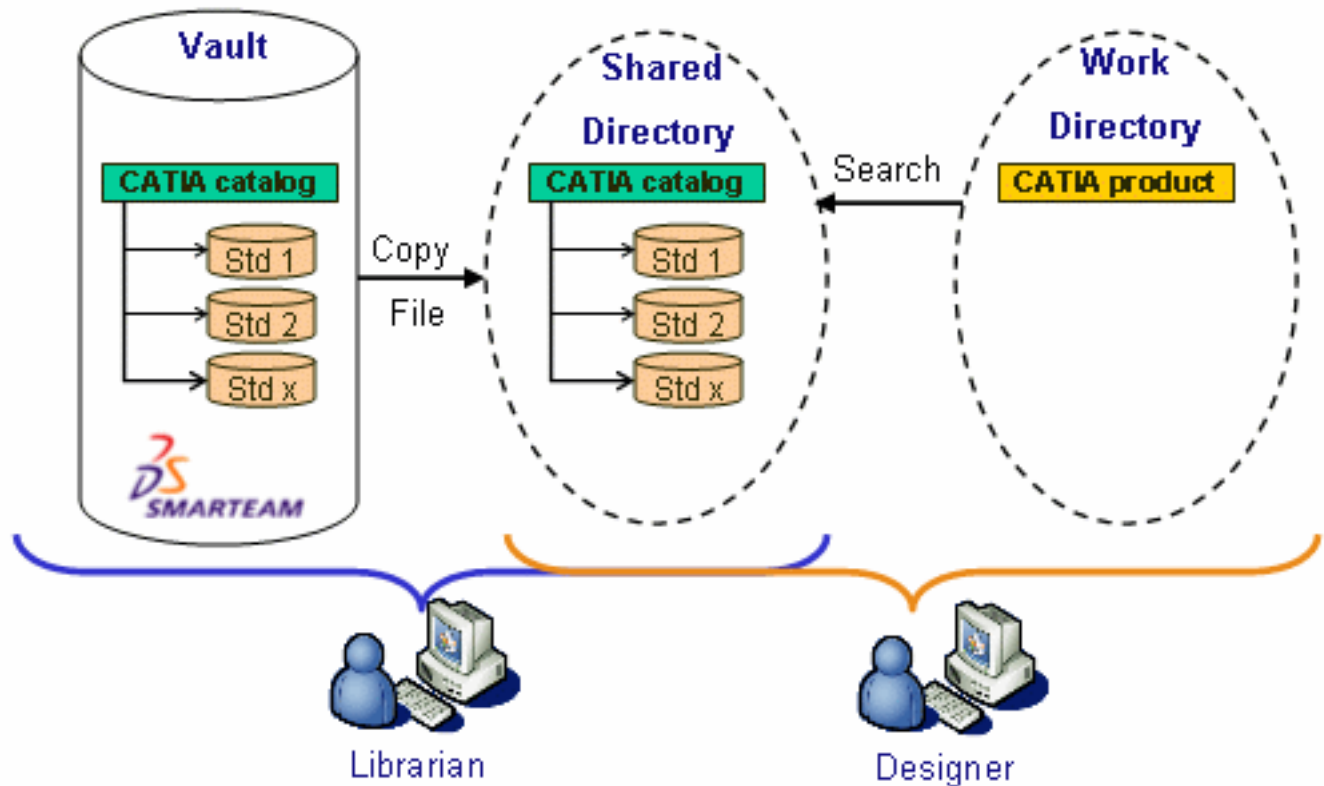
Authorization Setting for Standard CATIA Parts

Remove permissions for all lifecycle operations for design engineers. Only a librarian must have permissions for lifecycle operation on Standard CATIA Parts.

Setting Up a Shared Directory for CATIA Catalogs

Assign a shared directory for which all design engineers have read-only permission.

Only a librarian must have full permission for the shared directory. Users should not have permission to delete files from the shared directory.



Product Resource Management (PRM)

Catalogs allow a certain number of CATIA workbenches such as Piping Design, Tubing Design etc. to manage specific components. To integrate SmarTeam into these workbenches, librarians must specify the paths of the shared directories they create for their catalogs in their Product Resource Management (PRM) files (Project.xml files).

Creating and Saving Catalogs

When managing catalogs, as a librarian the first step you must perform is creating a catalog in CATIA. Then you need to save it in SmarTeam and make it available to users by copying it into a shared directory.

Here are the main instructions to follow for integrating CATIA catalogs into your SmarTeam database. This procedure can be divided in the following main steps:

- [Creating a Catalog](#)
- [Resolving Part Families and Part Family Components](#)
- [Saving a CATIA V5 catalog inside SmarTeam](#)

Creating a Catalog

1. From the CATIA **Start** menu, select **Infrastructure > Catalog Editor** to



open the Catalog Editor workbench .

2. Create a catalog from scratch using one of both methods available:
 - Using the Catalog Editor interactive commands: Activate the chapter under which you want to create a chapter or a subchapter.
 - In Batch mode: Open the BatchCatalog.csv file with an editor like Microsoft Excel. This file contains the information required to create a complete chapter referencing subchapters

Resolving Part Families and Part Family Components

Resolving a part family or a part family component means generating the .CATPart documents referred to by the part family or the part family component. These documents are generated in a specific folder (you specify in the catalog settings) and each generated document is a copy of the generative part configured with the matching row in the design table.

1. In the specification tree, select the part family to be resolved.
2. Right-click then select **xxx object > Resolve**.
The corresponding reference documents are generated in the folder you specified in the catalog settings. To access these settings, use **Tools > Options > Infrastructure > Catalog Editor > Catalogs** tab.

Note:

Prior to resolving catalogs, also make sure that **Allow family component dynamic resolution in catalog** available too from the **Catalogs** tab is on.

Saving CATIA V5 catalogs inside SmarTeam

Before saving catalogs into SmarTeam, we recommend you first save all CATPart documents. To do this, perform **File > Open, SmarTeam > Save, File > Close** operations for each of these documents. Once done, you can open the catalog file and save it into SmarTeam.

Accessing CATIA Catalogs

Design engineers who need to instantiate catalog components (parts, products, V4 model etc.) can access their catalogs using the shared directory.



Because most of the time catalogs include massive data, we recommend librarians to set create shared directories. Searching in the shared directory therefore is the method we recommend for accessing catalogs.

1. To search for your catalog in a shared directory, click **Catalog Browser**

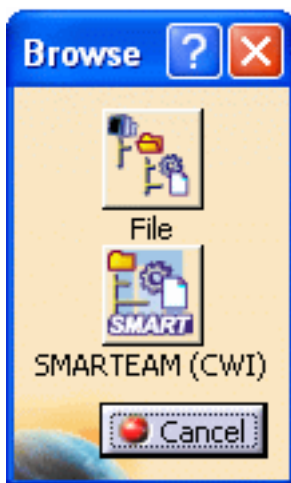


Available in several CATIA workbenches, the Catalog Browser provides interactive commands to browse catalogs and lets you instantiate components.

2. If the default catalog displayed in the Catalog Browser is not the one you are searching for, depending on how your administrator configured your CATIA session:

- click the **Browse another catalog icon**  to open the **File Selection** dialog box that enables you to navigate to the shared directory containing your catalog.

- or from the **Browse** dialog box, click **File**  to open the **File Selection** dialog box.




3. From the **File Selection** dialog box, navigate to access the shared directory.
4. Open the document of interest.
Using this method, the CATIA product creates a reference to standard CATIA parts in the shared workspace.



Handling CATAnalysis Documents



This scenario outlines the different operations you need to perform when handling CATAnalysis documents in the Generative Structural Analysis workbench .

As you know, these documents are linked to specific files. These require some customizations from your system administrator, but you also need to rename them as explained below.


The different steps described are the following:

- [Performing an Analysis in CATIA Generative Structural Analysis](#)
- [Renaming the Files Generated](#)
- [Saving the CATAnalysis Document in SmarTeam](#)
- [Checking Out the CATAnalysis Document](#)



Performing an Analysis in CATIA Generative Structural Analysis

1. Retrieve the part to which a material has already been assigned.
2. For example, from the New Analysis Case dialog box, select **Static Analysis** as the type of analysis you wish to perform.
3. Define restraints and distribute the forces you want.
Refer to the *CATIA Generative Structural Analysis Guide* for reference information about mechanical analyses for 3D systems.

4. Launch the computation .

Among other things, two files are generated and their paths are also displayed in the specification tree. You obtain a result similar as this:

E:\users\slg\AnalysisOutput\Analysis1_1.CATAnalysisResults

E:\users\slg\AnalysisOutput\Analysis1_1.CATAnalysisComputations

You can notice that the Results as well as the Computations files:

- are located in the appropriate directory defined by the system administrator, not in a temporary one.
- have a name which will not be unique in the SmarTeam database.

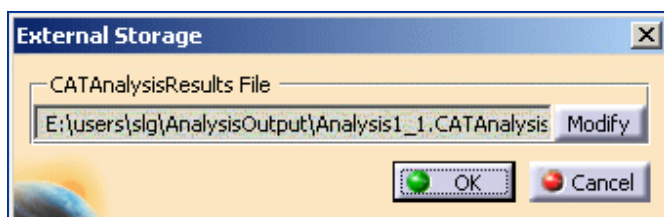


Renaming the Files Generated

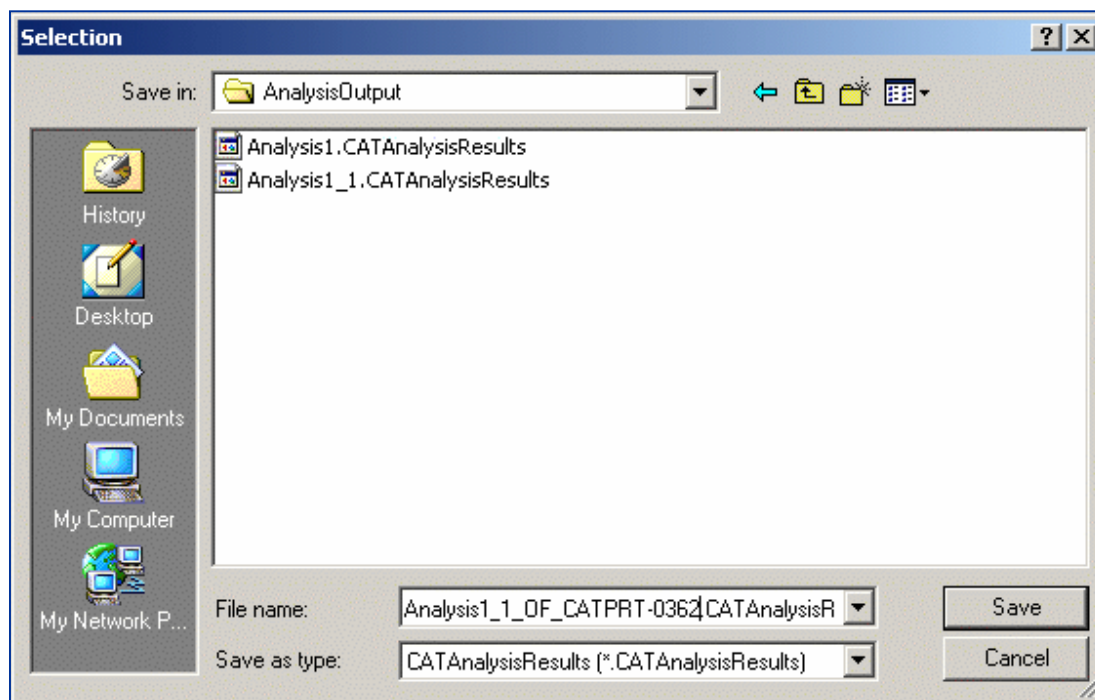
You may have noticed that the application provides names that cannot be easily reused in the SmarTeam database.

5. To ensure that both files will be identified with a unique name in the database, you need to rename them. To do so, right-click the file to be renamed and select **Results object->Definition...**

The External Storage dialog box is displayed. It is recommended to place the files in the user's workspace.




6. Press **Modify** and enter a unique name.
We recommend you use the names of the referenced Part or Product.

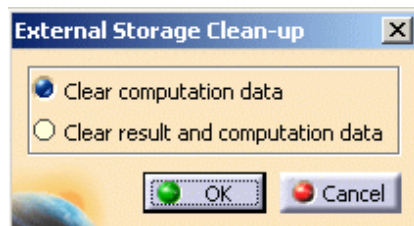


When done, the specification tree reflects the modifications. The name you have just modified are displayed.



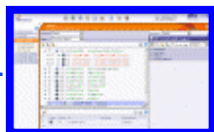
Saving the CATAnalysis Document in SmarTeam

7. Prior to saving CATAnalysis documents in SmarTeam, if you wish not to save the computation data in the vault, click  to clear this data.



8. Save the documents in the SmarTeam database.
As soon as you save your CATAnalysis file, SmarTeam creates a link between this document and:
 - o the CATPart document
 - o the two files generated after computation.

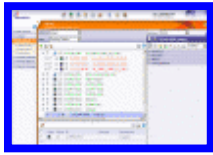
Notice the link icons to the CATAnalysisResults & CATAnalysisComputations. It is indeed stored as a CATIA Result Link as these two files are a result of the analysis.



9. To secure the documents in the electronic vault, select **SmarTeam->Life Cycle->Check-In**.

Checking Out the CATAnalysis Document

9. In case you wish to modify the CATAnalysis document later on, perform an Open for Edit operation.
You will note that a check-out of the analysis proposes by default the check-out of the result files.



Glossary



C

catalog A CATIA tool storing and classifying objects, each with its own specific characteristics (such as shape, color, size, diameter, length, standard, etc.) so that they can be retrieved fast and easily.

check in operation A lifecycle operation that checks in:

- a new object and places its file in the SmarTeam Checked In vault, or
- a checked out object (being modified) and places the file back in the SmarTeam Checked In vault.

check out operation A lifecycle operation that retrieves files from the SmarTeam Checked In vault. It is the first operation to do prior to making modifications to a file that is checked in to the SmarTeam vault.

D

default An operation or value that the system or application assumes, unless a user makes an explicit choice.

document A common unit of data (typically a file) used in user tasks and exchanged between users.

N

new release A lifecycle operation that retrieves files from the SmarTeam Release vault.

R

release A lifecycle operation that changes the revision designation of a part.

resolve operation A CATIA operation that generates the .CATPart documents referred to by part families or part family components.

S

specification tree Area of the document window reserved for viewing the design specifications of a part, presented in the form of a tree structure.

V

vault