

# SmarTeam CATIA Integration

## User's Guide

Version 5 Release 20 SP2



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# Overview

This overview provides the following information:

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

## SmarTeam - CATIA Integration in a Nutshell



SmarTeam - CATIA Integration is a seamless, integrated, rapidly implementable drawing and document management tool for users of CATIA . 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, rapidly implementable 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.

## Data Management Tasks

During the Data Management Process, the designer is required to perform activities such as searching, copying, linking, or viewing documents. These sub-tasks are supported in SmarTeam - CATIA Integration.

The example below illustrates SmarTeam - CATIA Integration's convenient support for the various data management sub-tasks. A designer can view and check revisions, and also perform many other activities by clicking one of the tabs provided. He can then evoke the dropdown menu to begin the editing process.

Using SmarTeam - CATIA Integration, the engineer functions within an open

environment, i.e. being able to manage all engineering data within the CATIA environment.

The SmarTeam - CATIA Integration functionality is deeply integrated and eliminates the time-consuming and bureaucratic process of opening and closing external software in order to perform design and data management tasks.

## Features and Benefits

The seamless integration between CATIA and SmarTeam enables the CATIA users to streamline their workflow in the following manner:

- Work in concurrent engineering through SmarTeam
- Browse through any SmarTeam window to view a hierarchical listing of SmarTeam documents.
- View the Profile Card of any document. The Profile Card displays file information, revision information, linked documents as well as thumbnail images of the part, product or drawing.
- Perform searches to locate any document saved in the SmarTeam database.
- Create and save Assemblies. Links are automatically created between the components of an Assembly to reflect their composition in CATIA . During revision management, the integrity of the Assembly and its components is easily maintained. You can view and access these links in the **Links** page of the Profile card.
  - Where Used links list all the parents of a document. These links are extremely useful in locating all the Assemblies in which a specific part is used.
  - Composed of links list all the children of a document. For example, it lists all the subassemblies (parts and drawings) of an Assembly.
- Create and save drawings. Links are automatically created between the drawing and the part/product on which it is based. During revision management, the integrity of the revisions is easily maintained.

## Getting the Most out of this Guide



This book is intended for the user who needs to become quickly familiar with the SmarTeam - CATIA Integration product. The user should be familiar with basic CATIA Version 5 concepts such as document windows, standard and view toolbars. Prior to reading this book, we recommend that you read the various CATIA V5 installation tasks in the CATIA - Infrastructure User's Guide Version 5.

To get the most out of this guide, we suggest you start reading [Getting Started](#). Then we suggest you read the [Basic Tasks](#).

## Logging In

In order to use the SmarTeam - CATIA Integration features, don't forget to log in using your login name and password as registered by the system administrator. Demo users should log in as **joe** (case sensitive) without any password.

## 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



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## 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?

## Enhancements

The following enhancements have been added in this release:

- **Large Assembly & Concurrent Design:**
  - **Global Refresh:** Enables you to update a complete CATIA product, applying the latest available revisions to all of the product's representation files (CGR).
  - **Design Compare:** Allows the opening of several versions of the same CATIA Part or CATIA Product within a CATIA session, helping the designer to integrate the preferred version into the final project.
- **Releasing a checked-in document using Job Server:**

If Job Server is implemented on your system, CATIA documents can now be released directly from SmarTeam - Editor without the need for a CATIA installation on your machine.
- **SmarTeam Document Status in the CATIA Document Node Tooltip:**

The SmarTeam Document status (Check In, Check Out, Release) has been added to the tooltip of the CATIA document Node.
- **Save as Type option:**

The new **Save as Type** option saves the current document into the SmarTeam database in a different format and optionally links it to the master CATIA document used to create this document. This safely saves the created document to SmarTeam, eliminating the possibility of mistakes when manually defining the file type and linking it to a specific revision.
- **Tools options:**

Some environment variables have been removed and their purpose is now exposed in the SmarTeam (CAI) Tools options tab:

  - Display the SmarTeam Project Manager window the first time a Save to SmarTeam operation is performed even if the root object was previously saved.
  - Prevent the SmarTeam Save operation from loading the context during the update of design links, in order to improve performance and memory consumption.
  - Allow the Check-Out/New-Release operation on non-latest revision.
  - Send the activated/deactivated status of each instance to SmarTeam to compute the Quantity attribute based on this information.
- **Non-latest available icon:**

The non-latest available icons have been updated.
- **Batch for instance exposure:**

The 3DLive functionality requires instance positioning information. This functionality provides a tool to save the instances information of CATIA products selected in SmarTeam. The tool may be automated to run on the entire database.

- **Quantity:**

When the Save process updates the links between each component of the assembly, the activated/deactivated status of each instance is exposed to SmarTeam. After that, the Quantity attribute of a component under its parent can be computed as the sum of all its active instances. The Quantity attribute value is recomputed and updated in SmarTeam each time a SmarTeam/Save or SmarTeam/CheckIn is done on a checked-out/new/released assembly.

- **Drawing Attribute Link:**

When the user creates a Drawing of a Product, generates balloons and saves the drawing to SmarTeam, the CATIA integration creates links of each Part of the Product that is also linked to the Drawing. SDE normally uses the balloon feature for drafting interactivity. Therefore, when a SmarTeam Save is performed, many links are generated between the actual Part and the Drawing that is saved in SmarTeam. However, according to the SDE LifeCycle Rule Setup, when the Part is checked out, the drawing is also Checked Out, which may not be necessary. The documentation explains how to avoid saving this link in SmarTeam.

- **Open for Edit:**

The Open for Edit check box allows you to open a document for editing. When this check box is marked, the Check Out or New Release lifecycle operation is invoked and the corresponding lifecycle window appears. After checking out/releasing the document, it is opened in CATIA. Note that this check box is only available for Checked In or Released documents. If the Check Out/New Release operation fails or is cancelled, the document will not be opened in CATIA.

- **Save/Project Manager**

The Project Manager dialog box is displayed for the first new document to be saved during a CATIA/SmarTeam Save transaction.

# Getting Started

Before getting into the detailed instructions for using SmarTeam - CATIA Integration capabilities, the following tutorial aims at giving you a feel as to what you can do with the product. It provides a step-by-step scenario showing you how to use key functionalities.

Suppose you are a designer and you have to modify a screw included in an assembly. Your work will be made up of several tasks letting you work alternatively both in SmarTeam and CATIA. The different tasks proposed in this section are:

Prior to starting this tutorial, you should read the information section [Before Performing the Getting Started Scenario](#).



All together, the tasks should take about 30 minutes to complete.

# Before Performing the Getting Started Scenario

## Displaying SmarTeam Status

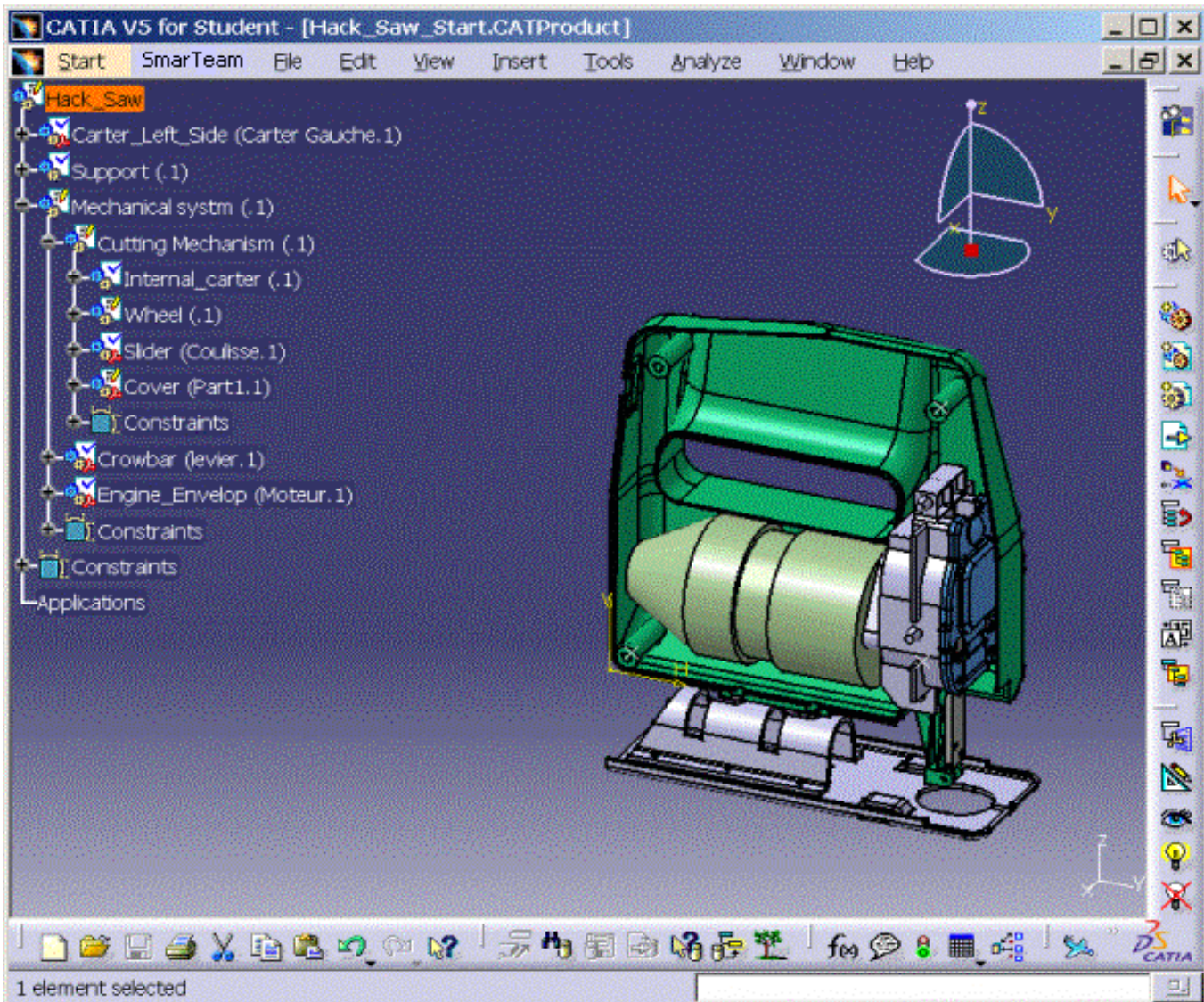
The product structure tree, the drafting workbenches and the desk tree show the SmarTeam status for each document. The status is displayed on the icons of the documents.

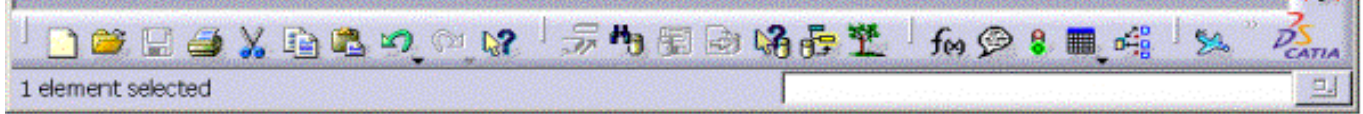
This section deals with the following information:

- [Product Structure Tree](#)
- [Drafting Workbench](#)
- [Desk Tree](#)
- [Displayed Statuses](#)

## Product Structure Tree

In the Product Structure tree, the SmarTeam status information is displayed thru icons on nodes corresponding to products and subproducts. Part status is displayed at the node corresponding to the Part instance inside the assembly.





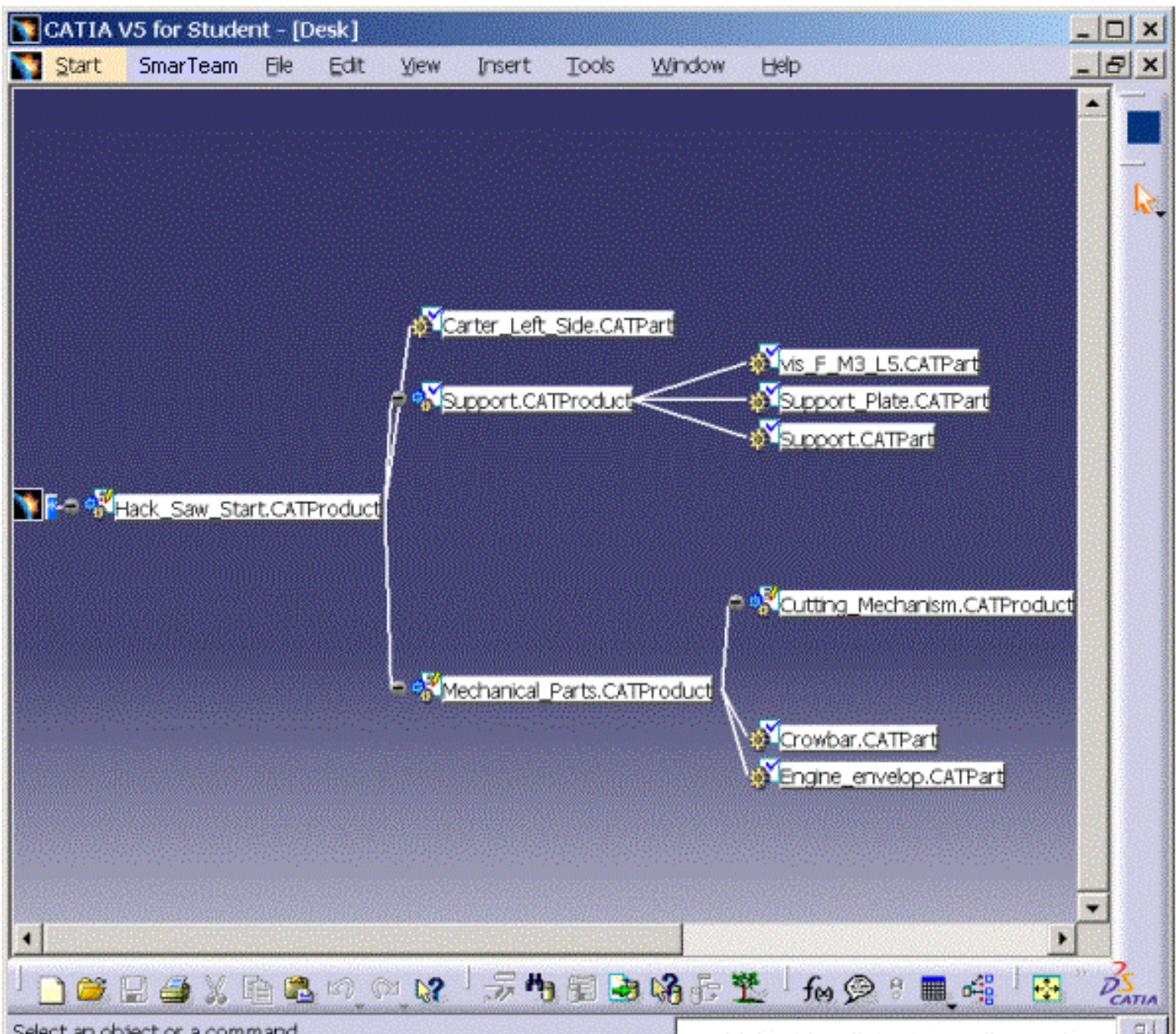
## Drafting Workbench

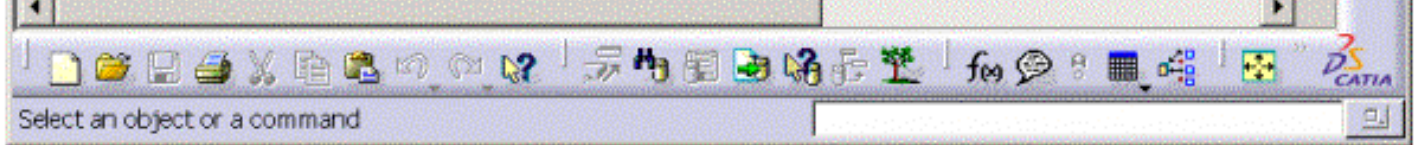
SmarTeam statuses are displayed for each drafting document as illustrated below:



## Desk Tree

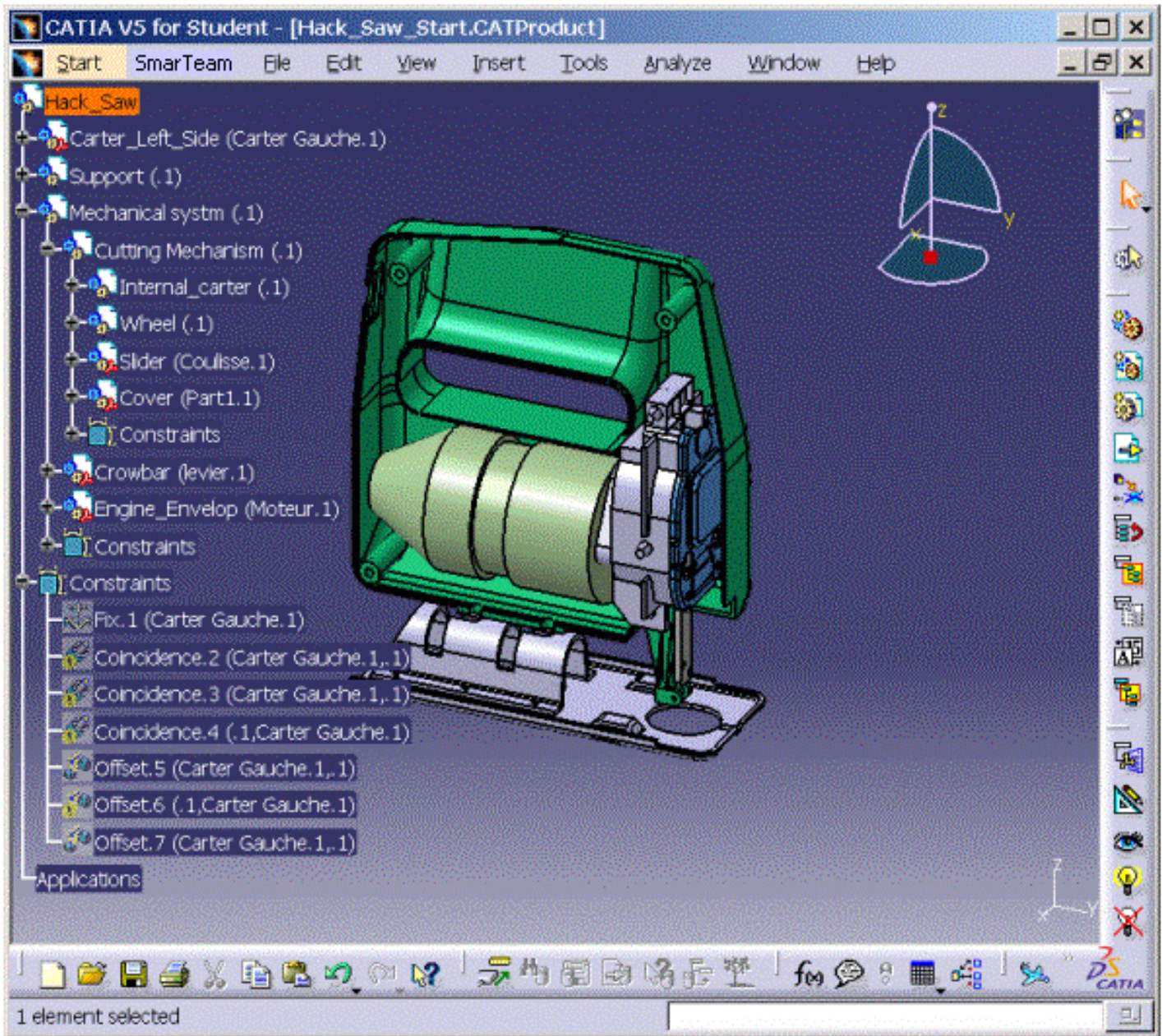
In the desk tree, information is displayed on all nodes corresponding to the documents (loaded or not). No information is displayed if the document is not found.





Bear in mind the following:

- No information is displayed if the user is not connected to the database:



- If you are connected, the top right corner of the icon indicates the SmarTeam status of the document.

## Displayed Statuses

















The statuses displayed by CATIA are identified by the following icons and symbols:

- Unknown



- New



• Checked In	 Product3 (Product3.1)
• Checked In, Modified	 Product5 (Product5.1)
• Checked In, Dirty	 Product5 (Product5.1)
• Checked In, Not Latest	 Product4 (Product4.1)
• Checked In, Not Latest, Modified	 Product14 (Product14.1)
• Checked In, Not Latest, Dirty	 Product6 (Product6.1)
• Checked In, Not Latest, Latest Checked Out	 Product15 (Product15.1)
• Checked Out	 Product7 (Product7.1)
• Released	 Product8 (Product8.1)
• Released, Modified	 Product5 (Product5.1)
• Released, Dirty	 Product10 (Product10.1)
• Released, Not Latest	 Product9 (Product9.1)
• Released, Not Latest, Modified	 Product14 (Product14.1)
• Released, Not Latest, Dirty	 Product11 (Product11.1)
• Released, Not Latest, Latest Checked Out	 Product15 (Product15.1)
• Obsolete	 Product12 (Product12.1)
• Obsolete, Dirty	 Product14 (Product14.1)

The SmarTeam Document status is also displayed in the tooltip of the CATIA document Node.




# Connecting to the SmarTeam Database



Your first task consists in connecting to the SmarTeam Database.



1. Launch a CATIA session.
2. Ensure that the settings dealing with linked [document localization strategies](#) as well as the [display of the SmarTeam File Open User Interface](#) are set to define the application's behavior we want.
3. From the SmarTeam toolbar, click **Connect** .



or select **SmarTeam > Connect**.

The SmarTeam User Login window appears:



The SmarTeam User Login window features the ENOVIA logo (a stylized '3S' in blue and orange) in the top left corner. Below the logo, there are three input fields: 'User name:' with an empty text box, 'Password:' with an empty text box, and 'Database:' with a dropdown menu showing 'PLMDB\_R20'. At the bottom of the window, there are two orange buttons labeled 'OK' and 'Cancel'.

4. For the purpose of our scenario, enter **joe** as the user name, without any password.
5. Click **OK**.

Now you are connected, go on to the next task to access the part to be edited.



# Copying a Part Stored in the SmarTeam Database



This task shows you how to access the original screw, then copy it so as to create a new screw for the assembly you are supposed to modify.

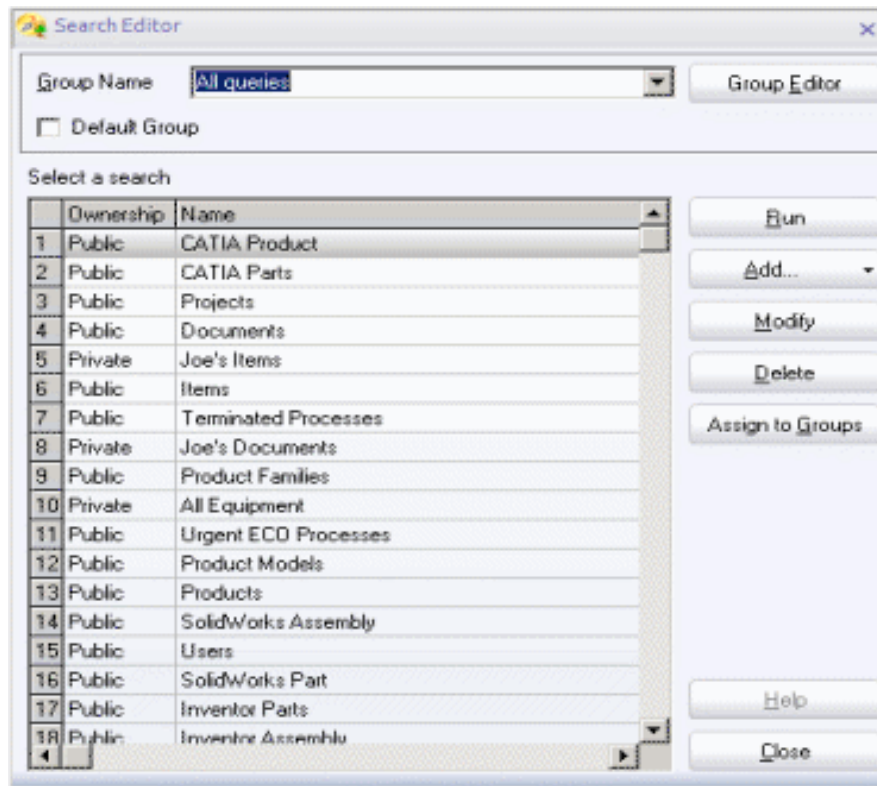
This task is made up of the following stages:

- [Searching for a CATPart Document from a CATIA Products List](#)
- [Creating a Copy from the CATPart Document](#)

## Searching For a CATPart Document From a CATIA Products List



1. Select SmarTeam > Find > Find Document.  
The Search Editor dialog box is displayed.



2. Select CATIA Products from the list.

3. Click **Modify** to perform the type of search that best meets your needs.  
The list of all CATProduct documents contained in the base is displayed in a new window.
4. In the **Description** field, enter `hack_saw`.

**Search by Attribute - CATIA Products**

Search\_class:

- + Inventor
- CATIA
  - CATIA Analysis
  - CATIA Catalog
  - CATIA Drawing
  - CATIA Material
  - CATIA Model
  - CATIA Part
  - CATIA Product**
  - CATIA Process
  - CATIA System
  - CATIA Library
  - CATIA Process Library
  - CATIA Feature Dictionary

**CATIA Product**

Document ID: \*

Revision:

Description: `hack_saw`

**Engineering Information:**

Part Number:

Definition:

Nomenclature:

Product description:

**Options**

- ☐ Match case
- ☐ Retrieve only last public revisions
- ☒ Make search public to all users

General Details Revision

Run Close Save Save As... Attributes... Clear Help

5. Click **Run** to confirm the search.  
A list of CATIA Products referencing the part we want is displayed.

Profile Card Links Notes Revision Viewer

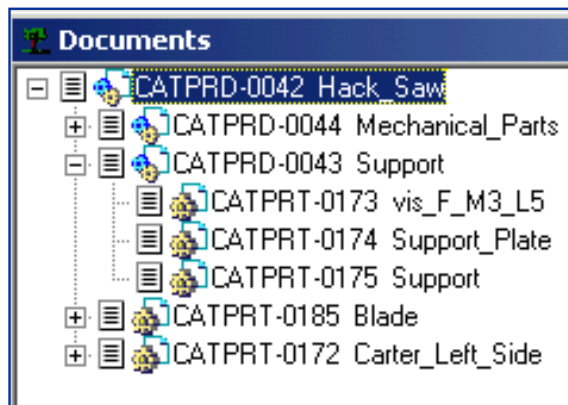
**ENOVIA**

### CATIA Product

Document ID:	CATPRD-0072	*
Revision:		State: New
Item number	Product1	Definition
Source	unknown	
Description	CATPRD-0072	
Nomenclature		

### Comments

- Right-click PN CATPRD-0042 and select **Open Views > Top down tree**.  
This displays in a new window all documents linked to the product you have just chosen.



7. Select the + sign to the left of CATPRD-0043 Support to see which documents are associated with it.
8. Select the CATPRT-0173 vis\_F\_M3\_L5 CATPart document.  
Selecting it displays on the right-hand side the corresponding profile card that contains information about it.


Profile Card

Links

Notes

Revision

Viewer



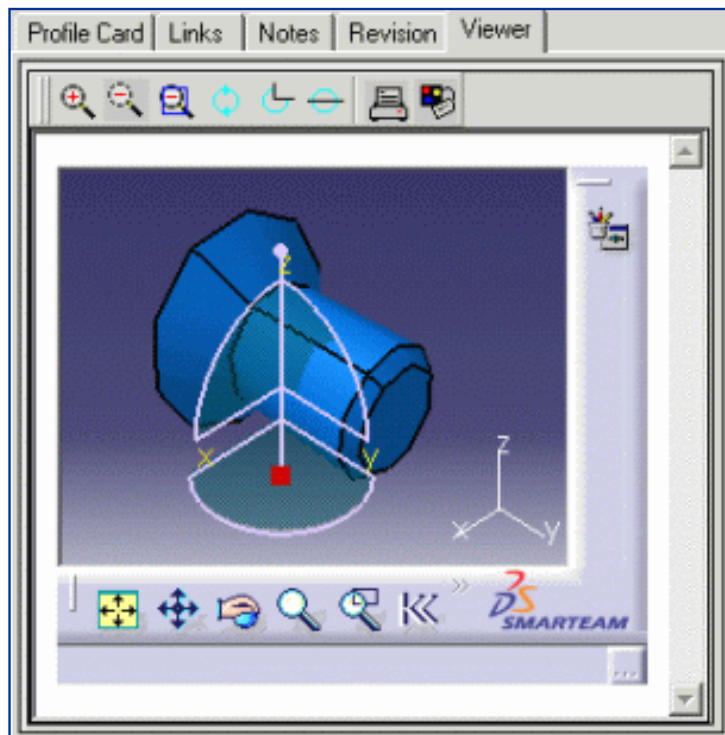
## CATIA Part

Document ID:	CATPRT-0326	*
Revision:		State: Checked In
Item number	Part2	Definition
Source	unknown	
Description	CATPRT-0326	
Nomenclature		

### Comments

CAd\_VERSION

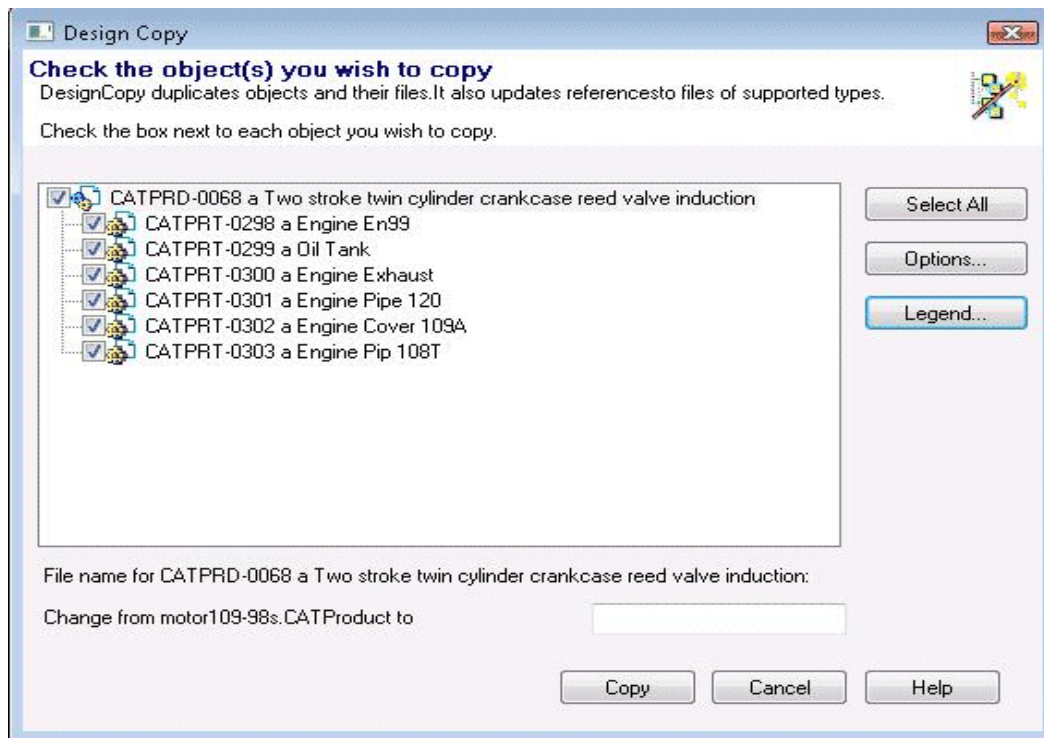
Optionally, click the **Viewer** tab to check that it corresponds to the part you want.



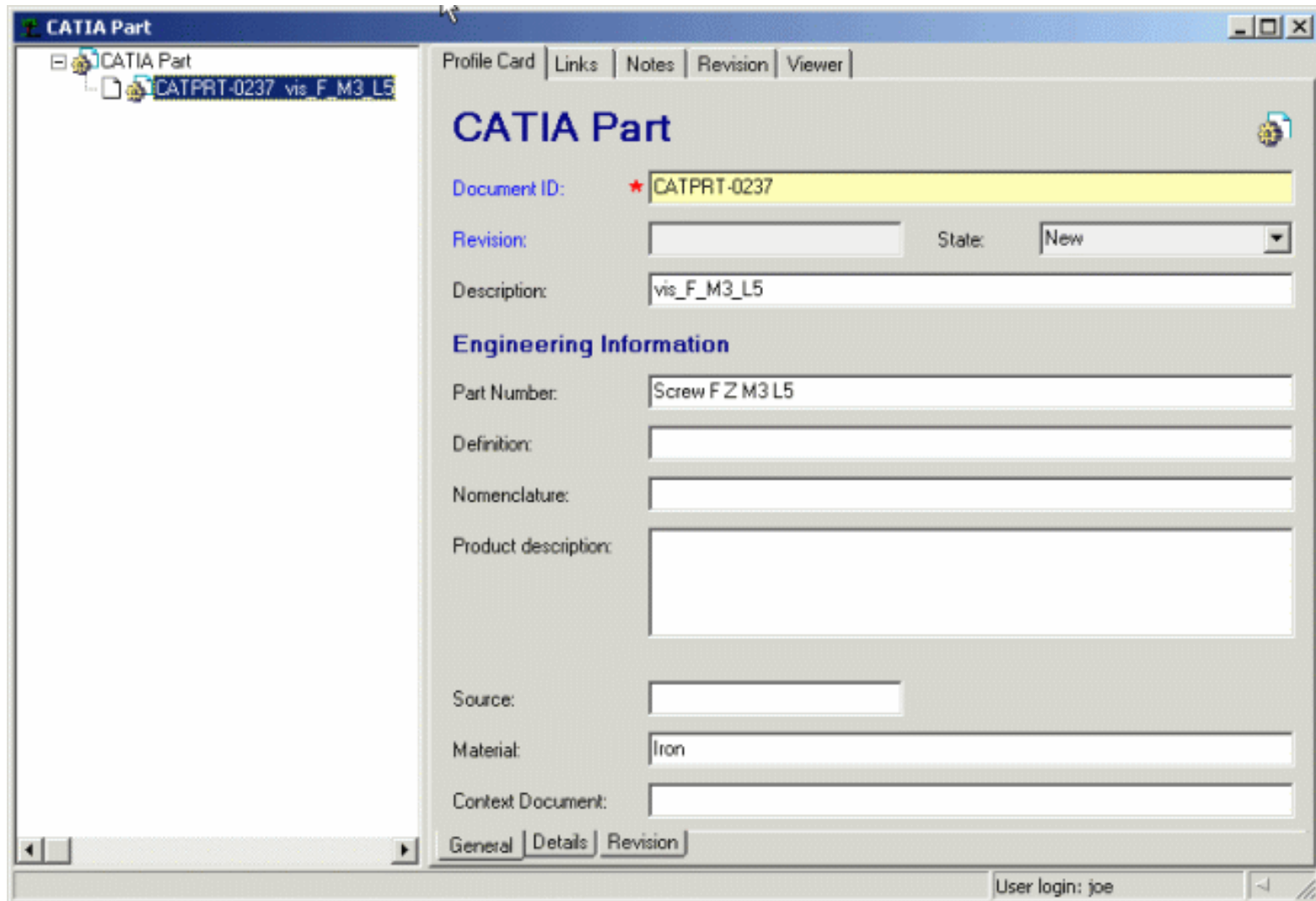
## Creating a Copy from the CATPart Document



9. Right-click on CATPRT-0173 vis\_F\_M3\_L5. and select **Design Copy**.  
The **Design Copy** dialog box appears.



10. Click the **Copy** button.  
Note that the CATIA Part window that appears indicates that the part is a new feature:



11. Right-click and select **File Operation** > **Open** to open the copy in the Part Design workbench. The CATPart document is displayed in the Part Design workbench, ready for being modified.

For more information about **Design Copy**, see [Copying Documents](#).



# Editing the Copy in the Part Design Workbench



Now that you have loaded the copy in the Part Design workbench, you can modify the part's geometry. This task proposes you to perform three modifications, then it shows you how to store this new part in the SmarTeam database.

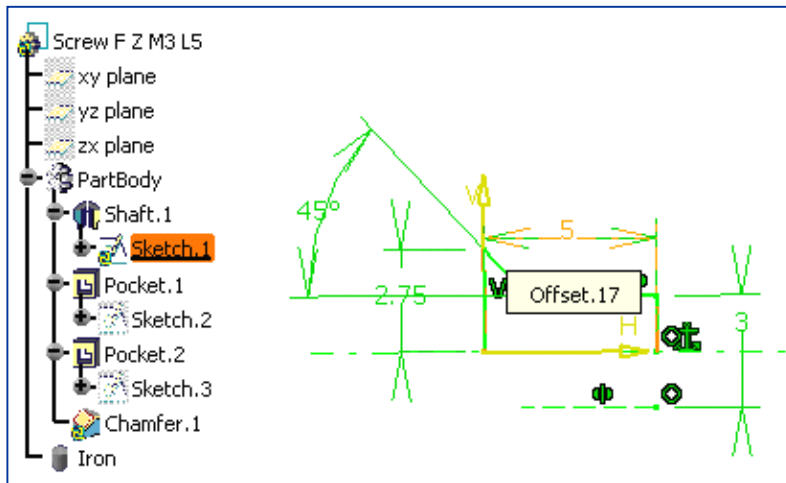
This task is made up of the following stages:

- [Editing a Sketch](#)
- [Applying a New Color to the Part](#)
- [Applying a Material to the Part](#)
- [Saving CATIA Properties in SmarTeam](#)

## Editing a Sketch



1. Double-click Sketch 1 to edit Offset.17.

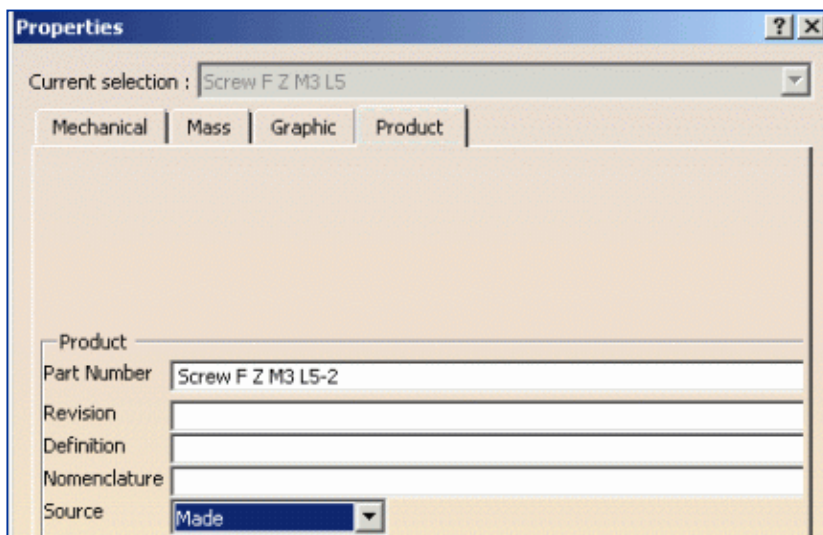


2. For example, enter 8 as the new offset value.
3. Exit the Sketcher and if required, update the part.

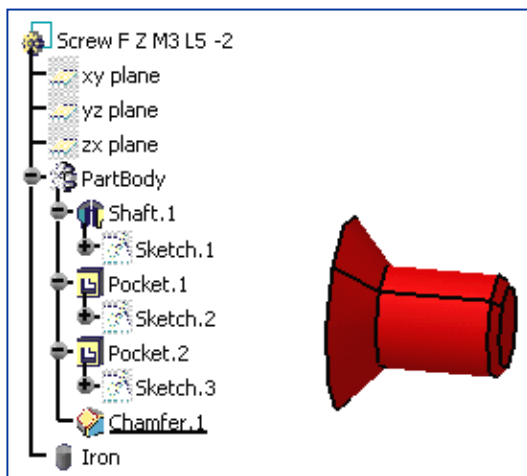
## Applying a New Color to the Part




4. Right-click PartBody and select **Properties**.
5. Click the **Graphic** tab and set the red color for this new screw.
6. Right-click Screw F Z M3 L5 and select **Properties**.
7. Click the **Product** tab.
8. Add -2 to the **Part Number** displayed, and set the **Source** field to **Made**.



9. Click **OK** to confirm these operations.  
The new part now looks like this:



10. Click the **Save** icon  or select **SmarTeam > Save**.

This updates the database information related to the document: take a look at the part number as well as to the **Source** field.

Profile Card | Links | Notes | Revision | Viewer

## CATIA Part

Document ID: ★ CATPRT-0237

Revision:  State: New

Description: vis\_F\_M3\_L5

### Engineering Information

Part Number: Screw F Z M3 L5-2

Definition:

Nomenclature:

Product description:

Source: made


Material: Iron

Context Document:

General | Details | Revision


## Applying a Material to the Part



11. Back in CATIA, select the part in the specification tree.
12. Click on **Apply Material** .
13. Select the **Metal** tab from the dialog box that appears, then select **Steel** as the new material to be applied to the copy.
14. Select the **OK** button.  
The material is now applied to the part.

## Saving CATIA Properties in SmarTeam



15. Click the **Save** icon  or select **SmarTeam > Save**.  
In the displayed SmarTeam: Documents window, you can see that the **Material** field now shows the name of the selected material (**Steel**).

Material: Steel

For more information on how to work in the Part Design workbench, see the *CATIA - Part Design User's Guide Version 5*.




# Storing the New CATPart Document in the SmarTeam Vault

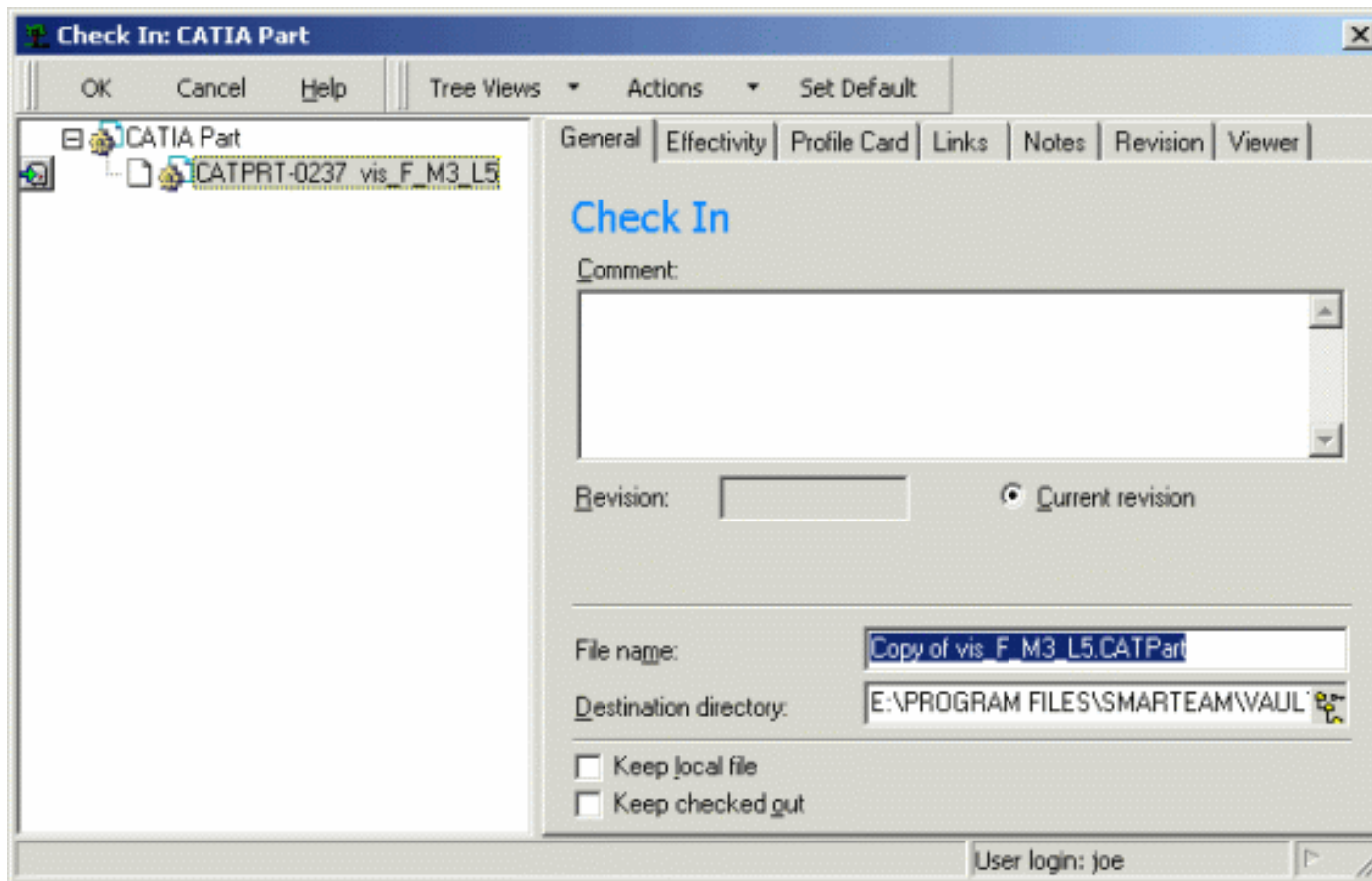


Now that you have finished the design of your part, you will store it in the SmarTeam vault so that:

- the document is stored in a safe place
- it can be used by all other users.



1. Click the **Check In** icon  or select **SmarTeam > Life Cycle > Check In**. The Check In: CATIA Part dialog box is now displayed.



The image shows the 'Check In: CATIA Part' dialog box. It has a title bar with 'Check In: CATIA Part' and a close button. Below the title bar are buttons for 'OK', 'Cancel', and 'Help'. There are also dropdown menus for 'Tree Views', 'Actions', and 'Set Default'. The main area is divided into two panes. The left pane shows a tree view with 'CATIA Part' and 'CATPRT-0237 vis\_F\_M3\_L5'. The right pane has tabs for 'General', 'Effectivity', 'Profile Card', 'Links', 'Notes', 'Revision', and 'Viewer'. The 'General' tab is active, showing a 'Check In' section with a 'Comment:' text area, a 'Revision:' field, and a radio button for 'Current revision'. Below these are 'File name:' and 'Destination directory:' fields. The 'File name' field contains 'Copy of vis\_F\_M3\_L5.CATPart' and the 'Destination directory' field contains 'E:\PROGRAM FILES\SMARTTEAM\VAULT'. At the bottom, there are two checkboxes: 'Keep local file' and 'Keep checked out'. The 'User login:' field at the bottom right shows 'joe'.

2. Optionally, enter some notes in the **Comment:** area.
  3. Click **OK** to confirm.
- When checking in a document, the document is saved in the SmarTeam vault and then moved from your local disk

to the vault. Once in the vault, all authorized users can access it.

The SmarTeam: Revisions of ... window is now displayed. You can check that the status of the document is now **Checked In**.

**SmarTeam: Revisions of 'CATPRT-0237'**

Data Chart

	Class	Revision	State	Creation Date	Phase
1				08/20/2004 09:3	Default

Profile Card Links Notes Revision Viewer

### CATIA Part

Document ID: \* CATPRT-0237

Revision:  State:

Description:

#### Engineering Information

Part Number:

Definition:

Nomenclature:

Product description:

Source:

Material:

Context Document:

General Details Revision

User login: joe

4. In the same way, by selecting the **Details** tab, you can check that the file has been moved to the "CHECKED IN" vault.

Profile Card | Links | Notes | Revision | Viewer

CATIA Part

File Information

File Type: CATIA Part

File Name: Copy of vis\_F\_M3\_L5\_1136\_687.CATPart

Directory: E:\PROGRAM FILES\SMARTTEAM\VAULTS\CHECKED IN

User Information

Created by: joe , Joe

Creation Date: 08/20/2004 09:39

Modified by: joe , Joe

Modification date: 08/20/2004 10:18

General | Details | Revision



# Modifying a Checked In Assembly




This task shows you how to access the assembly to be modified, open it in the Assembly Design workbench and then replace the original screw it contains with the new screw you have just designed.

This task is made up of the following stages:

- [Looking for the Assembly to Be Modified](#)
- [Opening the Assembly](#)
- [Replacing the Original Screw](#)



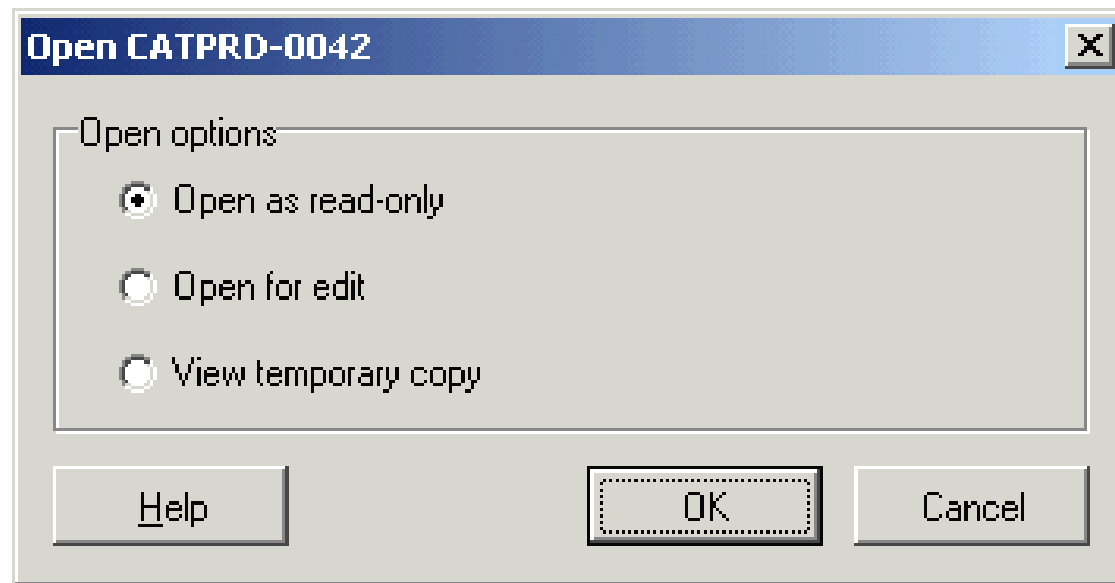
## Looking for the Assembly to Be Modified

1. Click **Find Document**  or select **SmarTeam > Find > Find Document**.
2. From the **Search Editor** dialog box displayed select CATIA Products as the search class.
3. Click **Run** to confirm the search.  
The list of all CATProduct documents contained in the base is displayed.
4. Select CATPRD-0042 Hack\_Saw.
5. Right-click CATPRD-0042 Hack\_Saw and select **Open Views > Top down tree**.  
A Documents window is now displayed showing the whole CATPRD-0042 Hack\_Saw assembly.

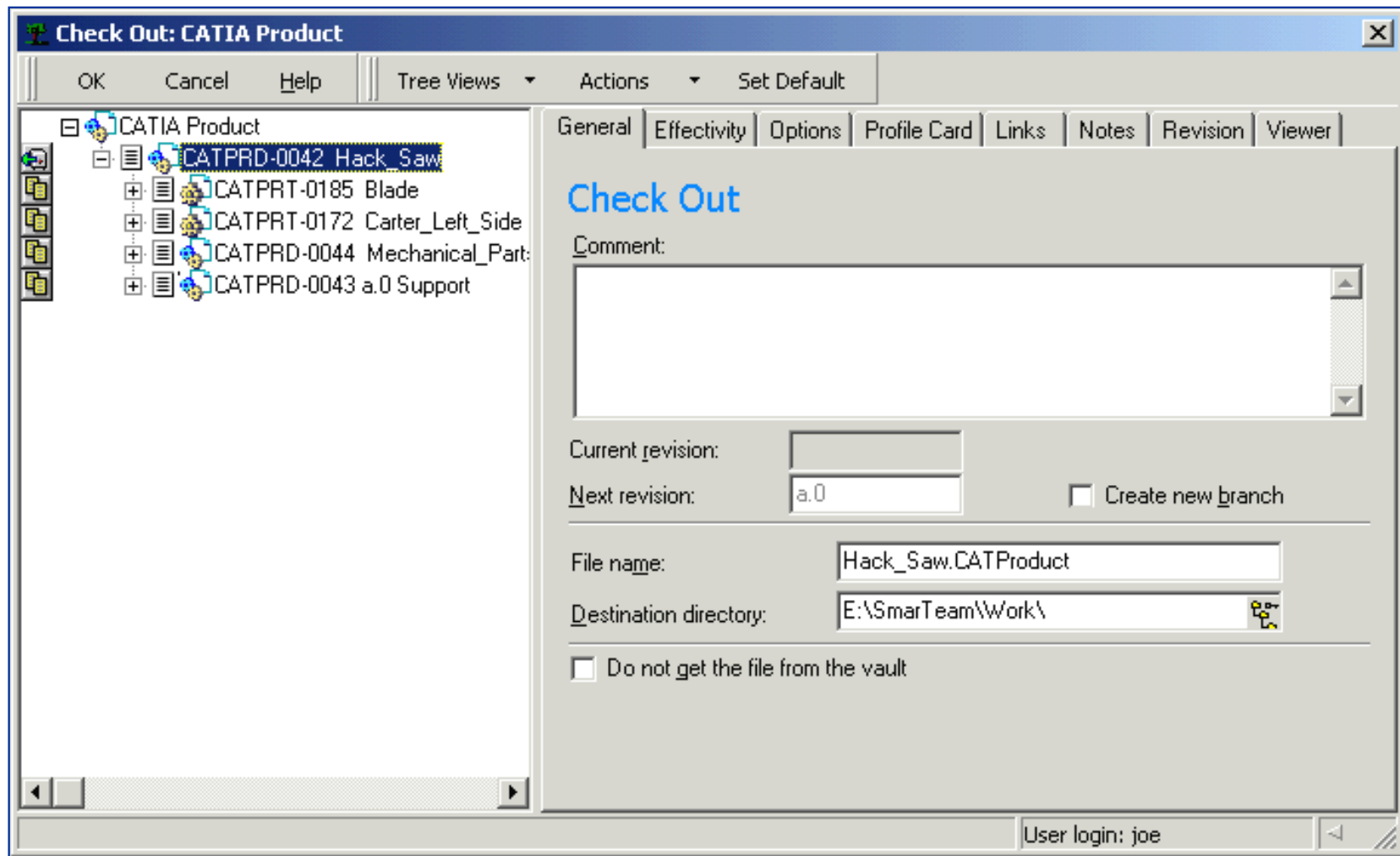
## Opening the Assembly




1. Right-click the assembly and select **File Operation > Open For...** .  
The **Open CATPRD-0042** dialog box is displayed. By default, the Open as read-only option is selected.

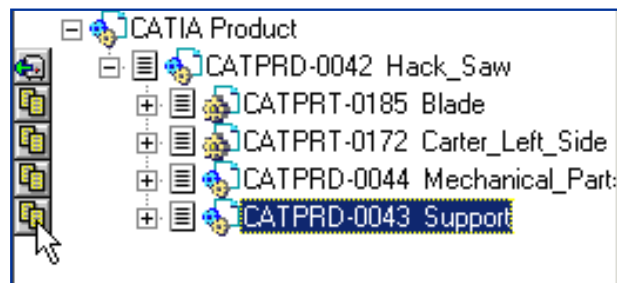



2. Because you want to modify the assembly, check the **Open for edit** option.
3. Click **OK**.  
The **Check Out: CATIA Product** dialog box appears to let you check out the documents of interest.



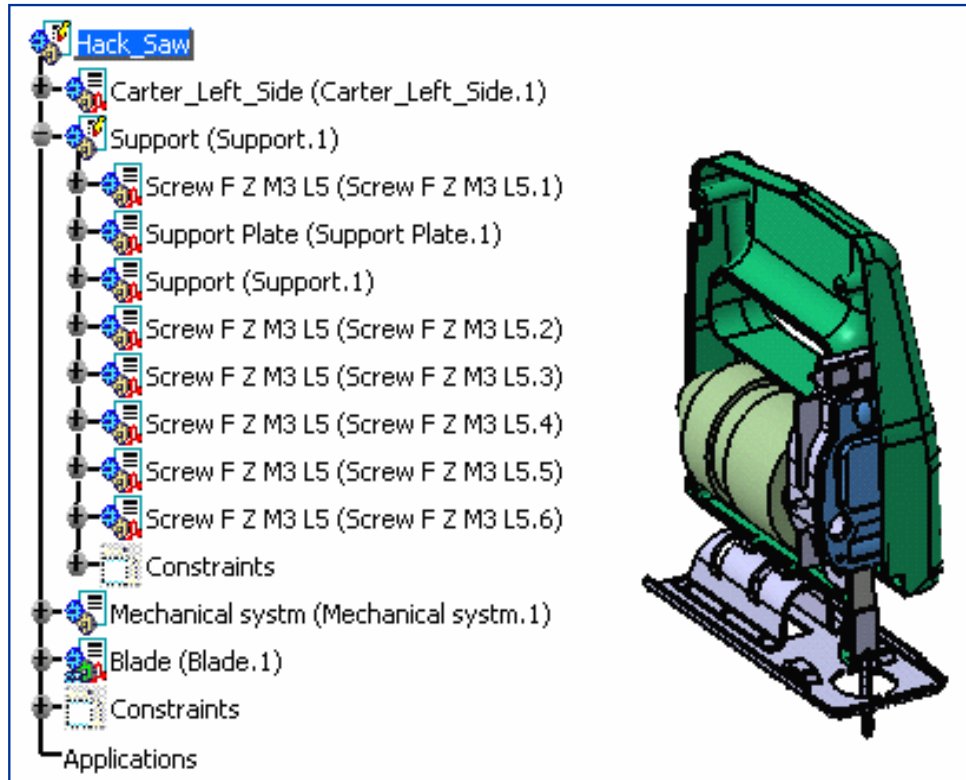
Check Out windows display SmarTeam information as set in the **Tree Properties** dialog box. For more information, see [Customizing SmarTeam Document Display Information](#).

4. To check out not only CATPRD-0042 Hack\_Saw but also one of its components, you need to click on the icon close to that component. For the purpose of our scenario, click the icon  in front of CATPRD-0043 a.0 Support.



Note that a new icon  indicates that the document is selected for being checked out.

5. Click **OK** to check out the assembly and its support component at the same time.  
The assembly is now open in the CATIA Assembly Design workbench.

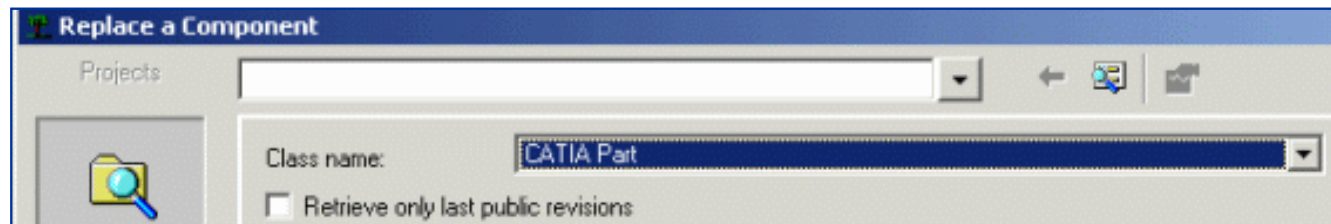


Note that the icons close to Hack\_Saw and Support (Support.1) indicate that you can edit these documents, which is not the possible for the other documents.

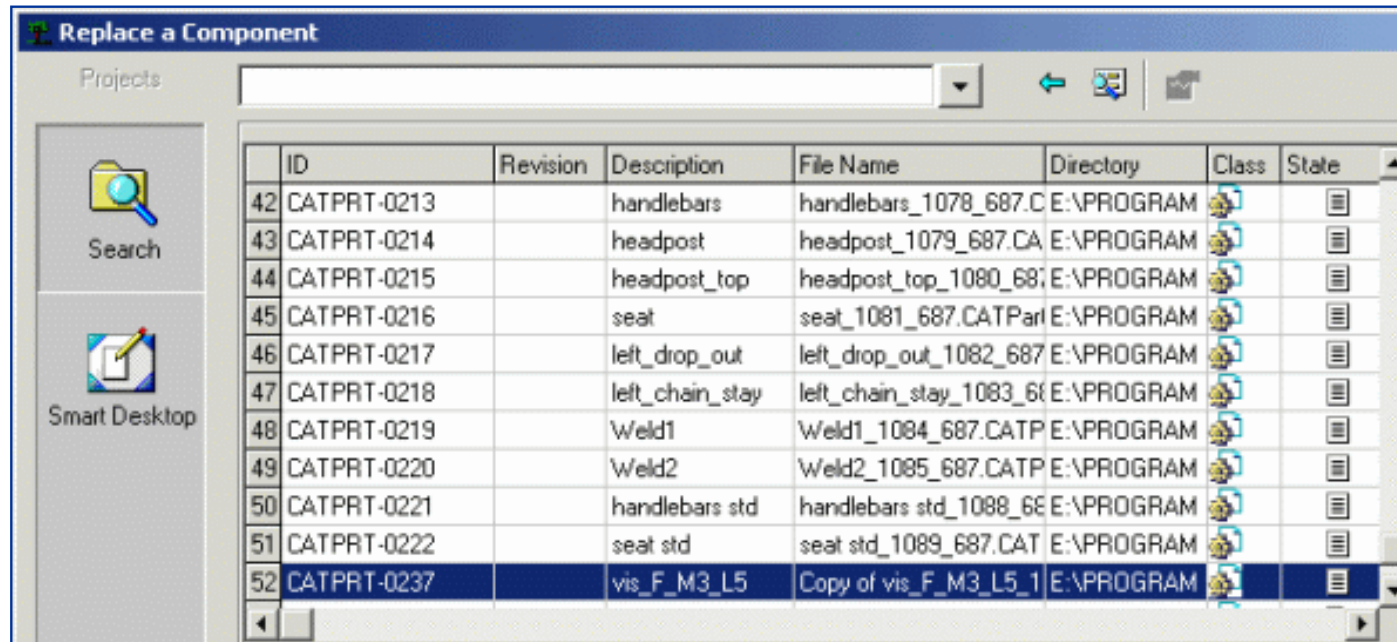
## Replacing the Original Screw



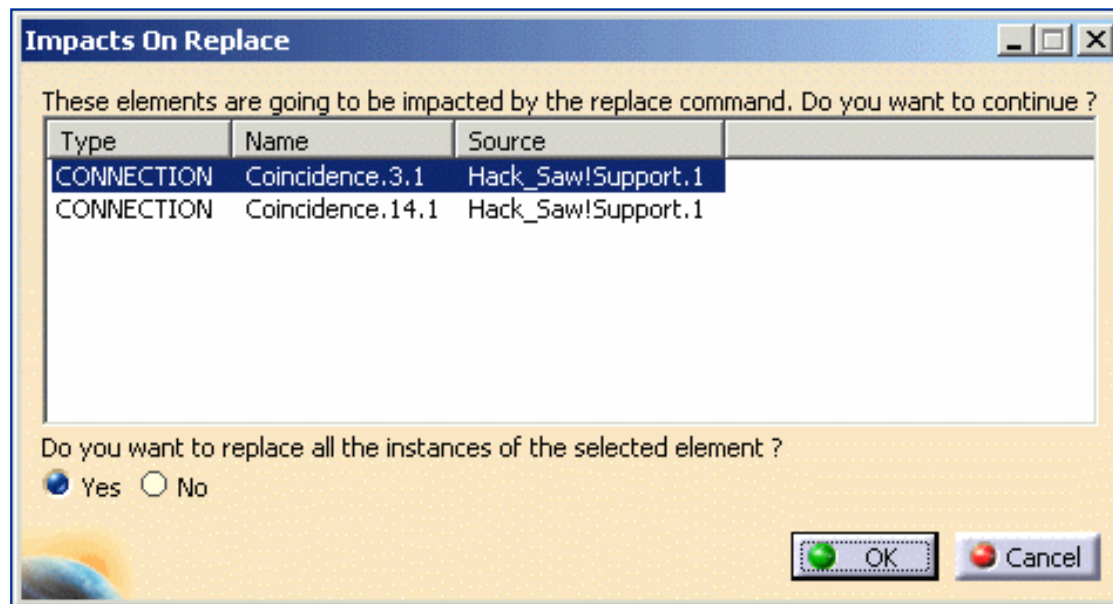
1. Select Screw F Z M3 L5 (Screw F Z M3 L5.1) as the part to be replaced.
2. Select **SmarTeam > Assembly Management > Replace Component...**  
The **Replace a Component** window displays.
3. Set the **Class name** combo list to CATIA Part.



4. Click Run to launch the query.
5. Select the part you created.



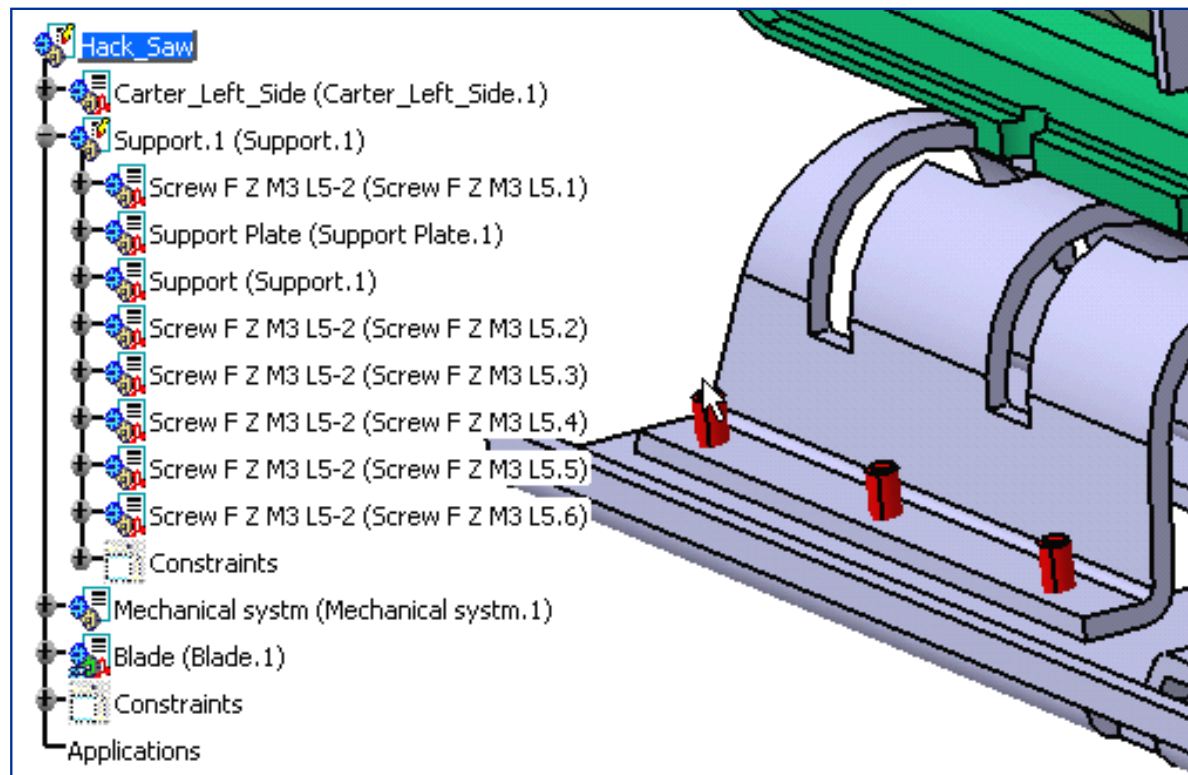
6. Click OK.  
The **Impacts On Replace** dialog box appears, showing the elements that will be affected by the change.



7. Click **OK** to confirm the operation.

Note that you can choose between replacing all instances or not. Because the option **Yes** is selected, all instances are going to be replaced.


The original screw has been replaced as shown here:




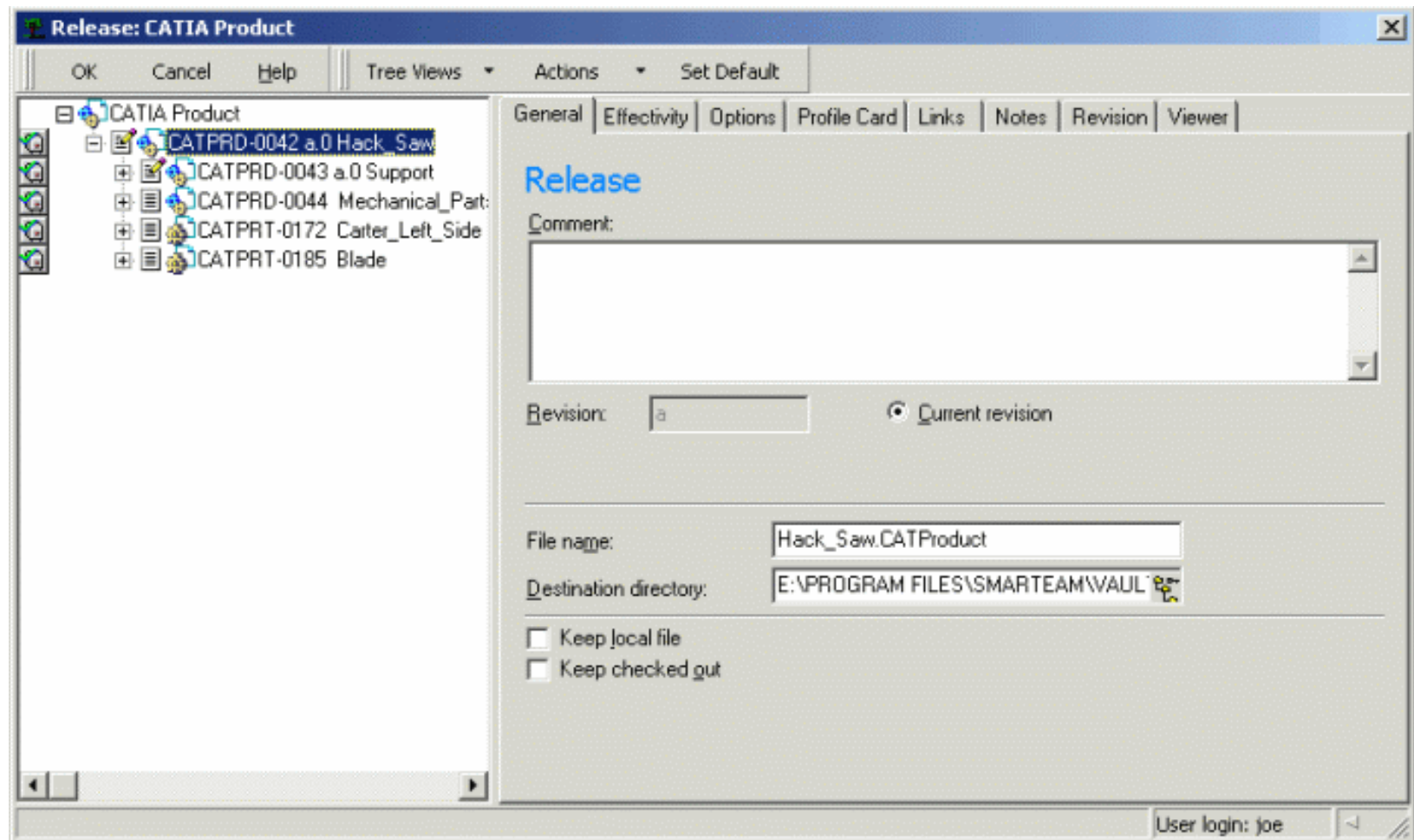
Now let's go on to [Releasing the Modified Assembly](#), which is our last task.




# Releasing the Modified Assembly

 In this last task, your job consists in creating a new release of the assembly you have just modified. Releasing an assembly means placing it in the **Released** vault. This operation is generally done when a supervisor releases a stage of development of the document.

-  1. Select **SmarTeam > Life Cycle > Release**.
2. Select the **OK** button in the **Release** dialog box.
- The *Release: CATIA Product* window is now displayed:



Note that the Release operation automatically releases the documents that you did not check out for replacing the original screw, as indicated by this icon  displayed close to each document.

3. Click **OK** to validate the Release operation.

The *SmarTeam: Revisions of CATPRD-0042-a* window appears, confirming that the operation is done. In the State column, you can notice that:

- the icon for the first document is the one identifying a Checked In, Not Latest document
- the icon for the second document is the one identifying a Released document

The screenshot shows the 'SmarTeam: Revisions of 'CATPRD-0042 a'' window. It features a table of revisions and a detailed profile card for the selected revision.

Class	Revision	State	Creation Date	Phase
1			03/25/2003 16:33	Default
2	a		08/20/2004 10:29	Default

**CATIA Product**

Document ID: ★ CATPRD-0042

Revision:  State:

Description:

**Engineering Information:**

Part Number:

Definition:

Nomenclature:

Product description:

Source:

Material:

General Details Revision

User login: joe



# User Tasks

This section presents the tasks you usually perform in the SmarTeam CATIA Integration product. These tasks are the following ones:

# Finding and Working with Documents


Locating parts, products and drawings is an essential task, but it can be time-consuming when creating complex assemblies. SmarTeam provides a number of powerful functions which enable you to locate a document in the SmarTeam data structure.

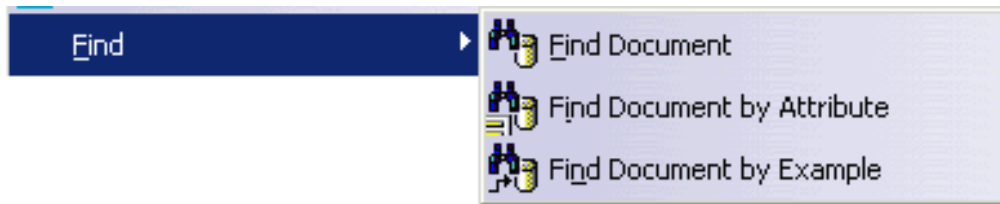
The following SmarTeam tools work together to help you find and modify any CATIA V5 document:

<b>Find</b>	Use the <b>Find</b> options to run a search and locate specific documents that match the search criteria. These documents are listed in a search results list.
<b>Browse</b>	Browses through each document in the list. Each time you select a document, its Profile Card is displayed. You can view general attributes of the document as well as its revision history. In addition, you can view a thumbnail image of the document in the <b>Viewer</b> tab.
<b>Find Out Where a Document Is Used</b>	Keeps track of all the assemblies that use a particular document as a component before you begin to modify the Part/Product.
<b>Open as read- only</b>	When you locate the exact document that you were searching for, you can launch it directly into CATIA V5, for viewing and inspection.
<b>Open for edit</b>	When you locate the exact document that you were searching for, you can launch it directly into CATIA V5, and modify it accordingly. See <a href="#">Editing</a> .
<b>View temporary copy</b>	Use <b>View Temporary Copy</b> to copy your design with all references to a temporary user location. Each revision will be copied to a different location, so one revision will not override its previous revision. See <a href="#">Viewing "exactly as released"</a> .
<b>Save</b>	When you have finished your modifications, save the document. The Profile Card (and appropriate revision history) is updated accordingly. See <a href="#">Saving a Part</a> , <a href="#">Saving an Assembly</a> or <a href="#">Saving a Drawing</a> .

# Finding/Browsing/Editing


## Finding

-  Locating Parts, products and drawings is an essential task, but it can be time-consuming when creating complex Assemblies. SmarTeam provides a number of powerful functions which enable you to locate and retrieve a document from the SmarTeam data structure:



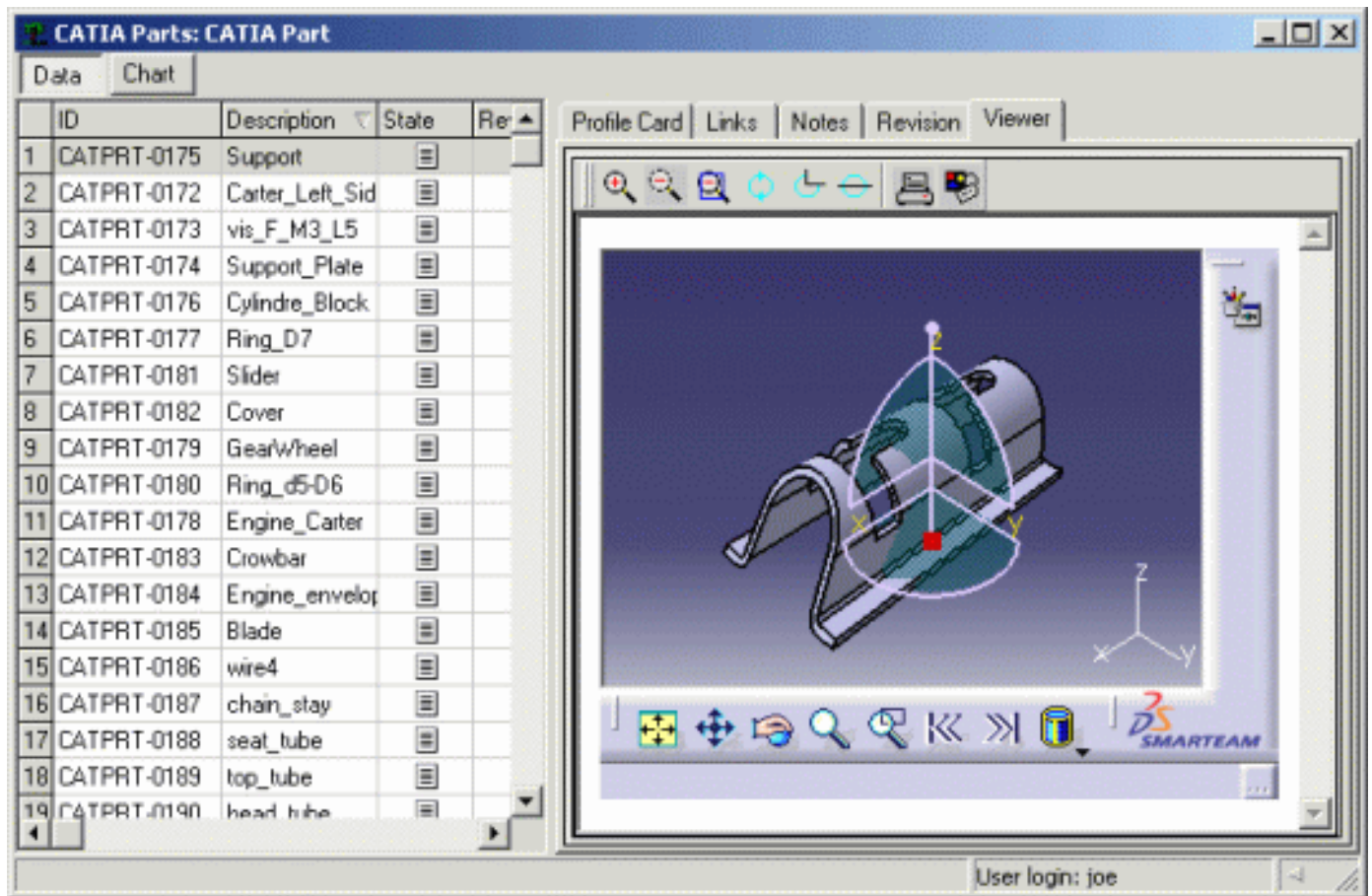
- **Find Document** : This option enables you to view the previously defined searches. From the Search Editor window, you can:
  - run a previously defined search (see [Running a Predefined Search](#))
  - modify a search.
  - create a new search.
- Each search may contain numerous search criteria. The results of the search are listed in a search results list. You can browse through the displayed list and view the Profile Card for each one. You can also select a document and launch it into CATIA V5.

## Browsing

-  After you run a search, the search results are listed in a search results list. You can then browse through the list to identify a specific document.

SmarTeam enables you to view CATIA parts, drawings and products in the **Viewer** page. This enables you to browse through the database and view the most recent image of a document, as displayed in CATIA V5.

Browsing through the **Viewer** page provides a means of searching for and identifying a specific document. For example, after running a search, the search results are displayed in a list. You can view the image of each of these documents and launch one into CATIA V5, if you wish.

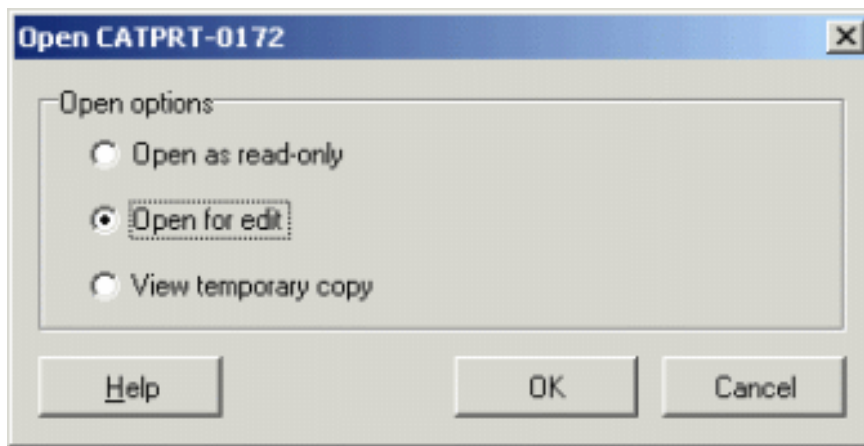


## Editing

When you have located a document, you can quickly launch it and edit it in CATIA V5 using the **Open for edit** command as described below.



1. From any SmarTeam window, select your document.
2. Right-click to select **File Operation > Open For...** .  
The **Open** dialog box that appears, provide three options.
  - **Open as read only** (default option)
  - **Open for edit**
  - **View temporary copy**
3. Check the **Open for edit** option.



4. Click **OK**.  
The **Check Out** dialog box appears to let you check out the documents of interest.
5. Select the documents you wish.
6. Click **OK**.  
The document is now open in CATIA ready for being modified.



# Viewing "exactly as released"



This section describes how to open a drawing or product exactly as it was released with all the references revisions at the time of the release.

## Recommendations

- If you are trying to open the document just for printing or viewing, i.e., not for referencing in other documents or for editing, using the Viewer is recommended rather than retrieving all the referenced documents.
- If you need to open the document in a full CATIA session, use [View Temporary Copy](#).

Using View Temporary Copy will copy your design with all references to a temporary user location. Each revision will be copied to a different location, so one revision will not override its previous revision.

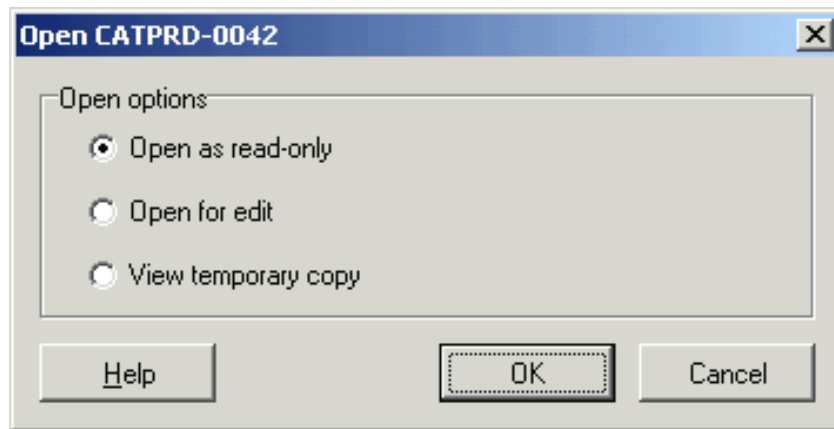
## Recommendations

- When working in the user working directory, do not use files from the temporary directory. If you do so, you will not be able to perform SmarTeam operations such SmarTeam save or check in.
- Use the SmarTeam File Explorer to clean up temporary folders.



## How to View a Temporary Copy

1. From any SmarTeam window, select your document.
2. Right-click to select **File Operation > Open For...** .  
The **Open** dialog box is displayed.



3. Check the **View temporary copy** option.
4. Click **OK**.  
The document is now open in CATIA.



# File - Open Integration



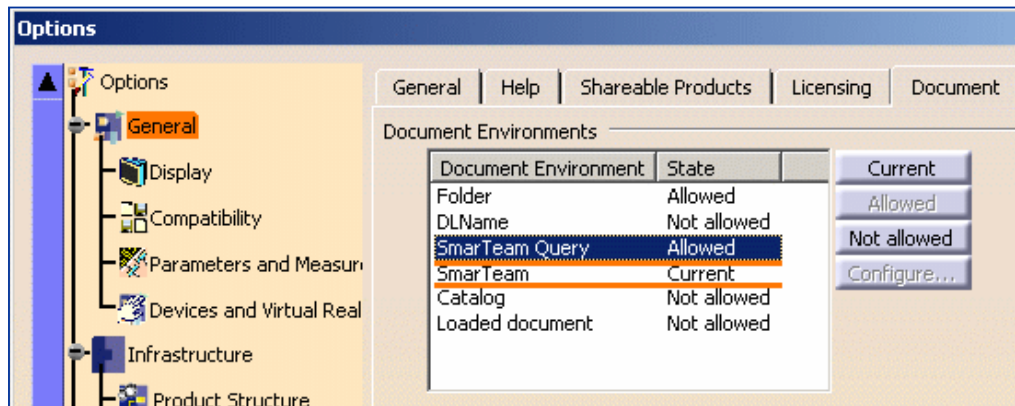
Many CATIA commands such as **File > Open...**, **Replace Component**, etc., require the selection of a document. The SmarTeam integration allows you to select the document from SmarTeam rather than from the local disk.


A SmarTeam dialog box can be displayed after running the following commands:

- **File > Open**,
- **Assembly Replace Component**,
- **User Defined feature instantiation**,
- **Change Source in Edit Links** etc.



1. Set the appropriate settings as explained in [Enabling the Display of the SmarTeam File Open User Interface](#).



2. From the SmarTeam toolbar, click **Connect** .
3. Select **File > Open...**



## SmarTeam Set as Current

If SmarTeam is set as current in the Document Environments list, the **Open document** dialog box is displayed. It enables you to run a quick find according to different fields: ID, Revision, Description, File Name, Directory, etc.  
For reference information, see the SmarTeam documentation.

For information on defining various settings for the Open dialog, see [Customizing the Open Dialog](#).

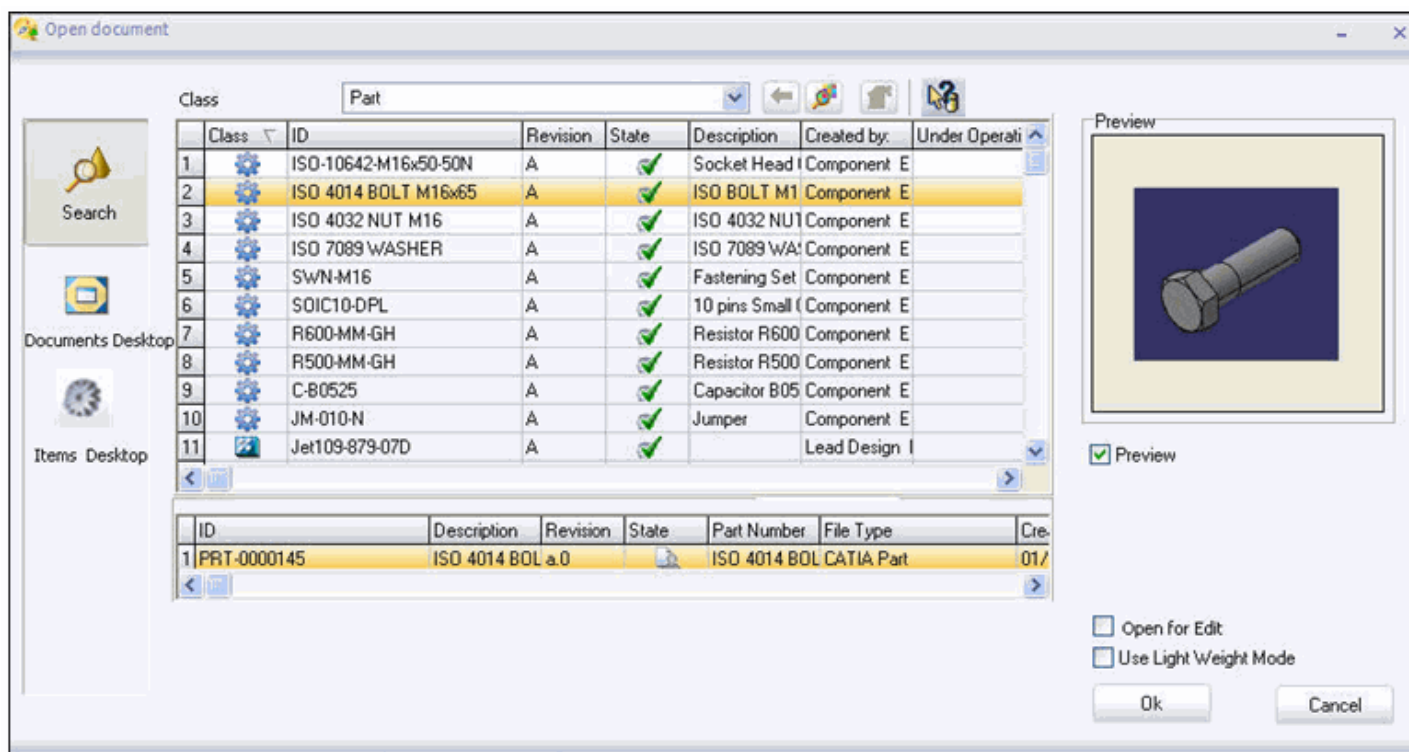
## Opening CATIA Documents Using the SmarTeam Open Dialog

Instead of requiring you to browse for files in local directories, SmarTeam provides the Open dialog, which lets you browse and search documents in the SmarTeam database using any of the following options, located in the options pane on the left side:

- **Search** - enables predefined Quick Search queries to search for CATIA documents or items
- **Documents Desktop** - enables browsing of SmarTeam projects to retrieve CATIA documents
- **Items Desktop** - enables browsing of the Items desktop to retrieve a specified CATIA document

**Open for Edit:** The **Open for Edit** check box allows you open a document for editing. When this check box is marked, the Check Out or New Release lifecycle operation is invoked and the corresponding lifecycle window appears. After checking out/releasing the document, it is opened in CATIA. Note that this check box is

only available for Checked In or Released documents. If the Check Out/New Release operation fails or is cancelled, the document will not be opened in CATIA.



**Note:** When you select a new or checked out document that exists in the user's working directory, the Open operation for the document and all its dependencies will not be performed. The document will be opened in CATIA as it appears in the working directory.

**Note:** When working in Light Weight Mode, the document must be opened using the SmarTeam Open Dialog or by using the **Open for Edit...** option from SmarTeam - Editor. Opening from the file system will result in broken links in CATIA, and when saving a CATIA product to SmarTeam, an incorrect structure may be produced.

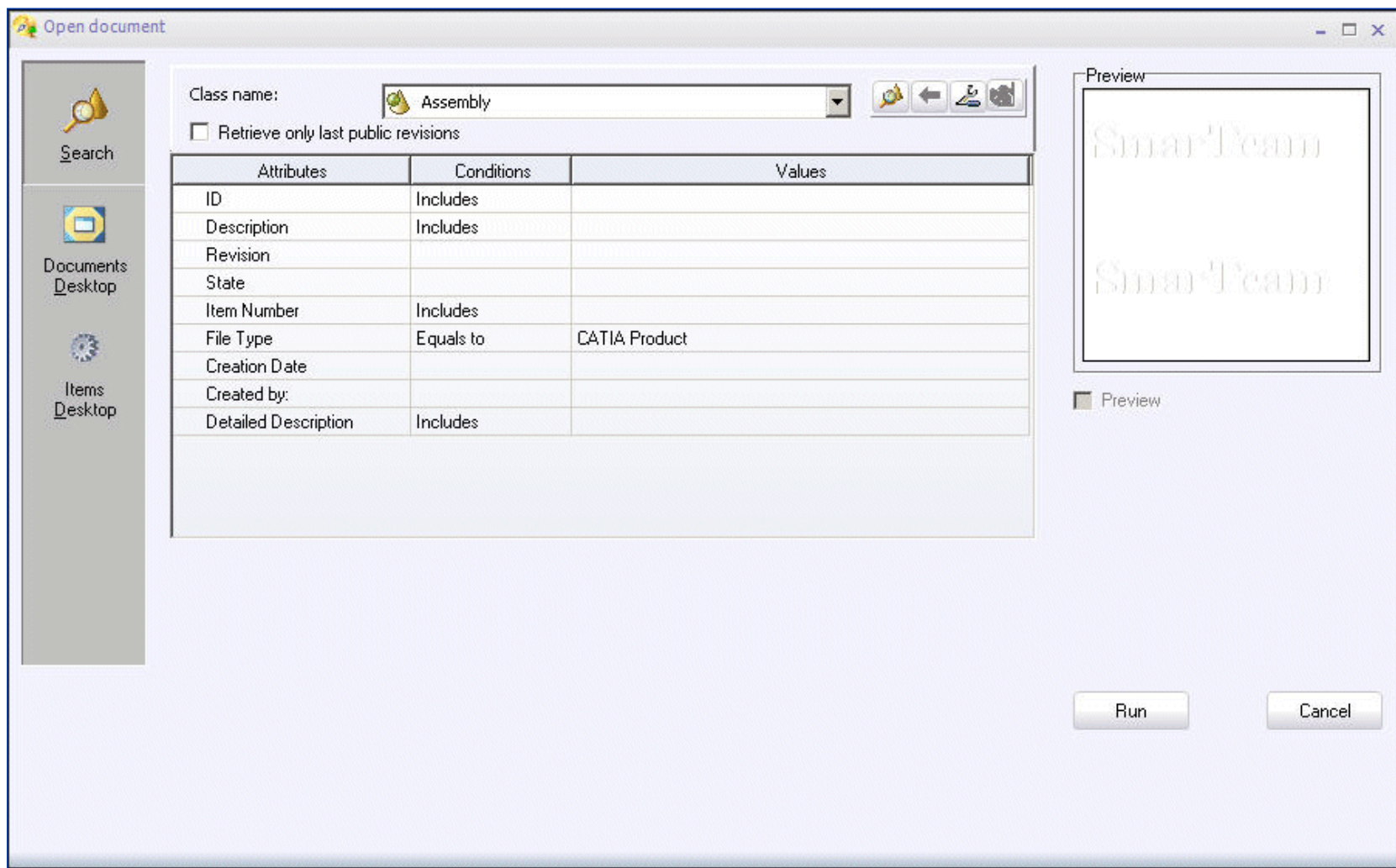
If Part Behavior is set, all states of parts and documents are appear in the State field. You must select the relevant state.



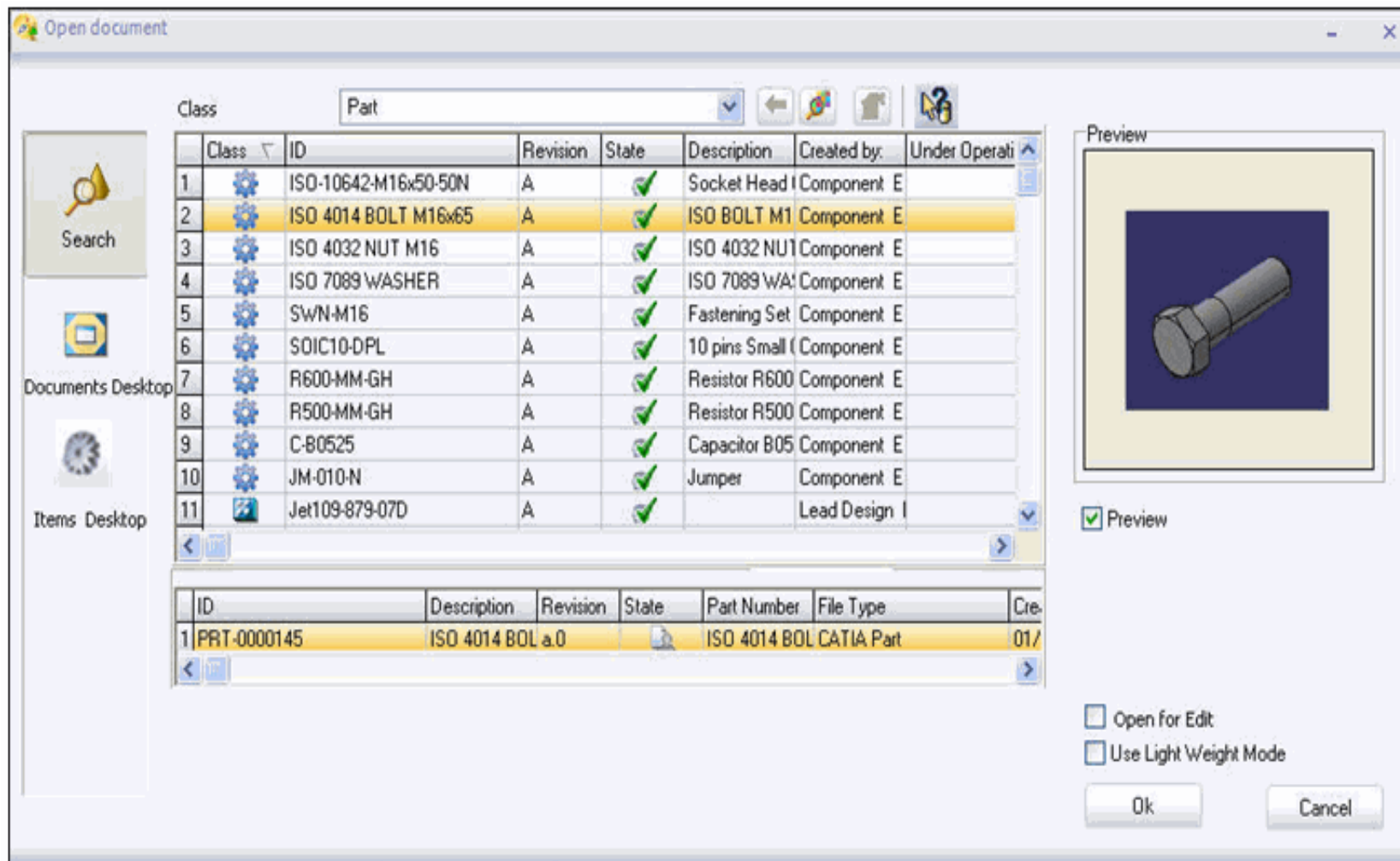
## Quick Search Option

The Quick Search enables the user to select a specific class from the **Class name** field and perform a search based on predefined attributes and conditions.

1. From the **Class name** field, select a class from the dropdown list.



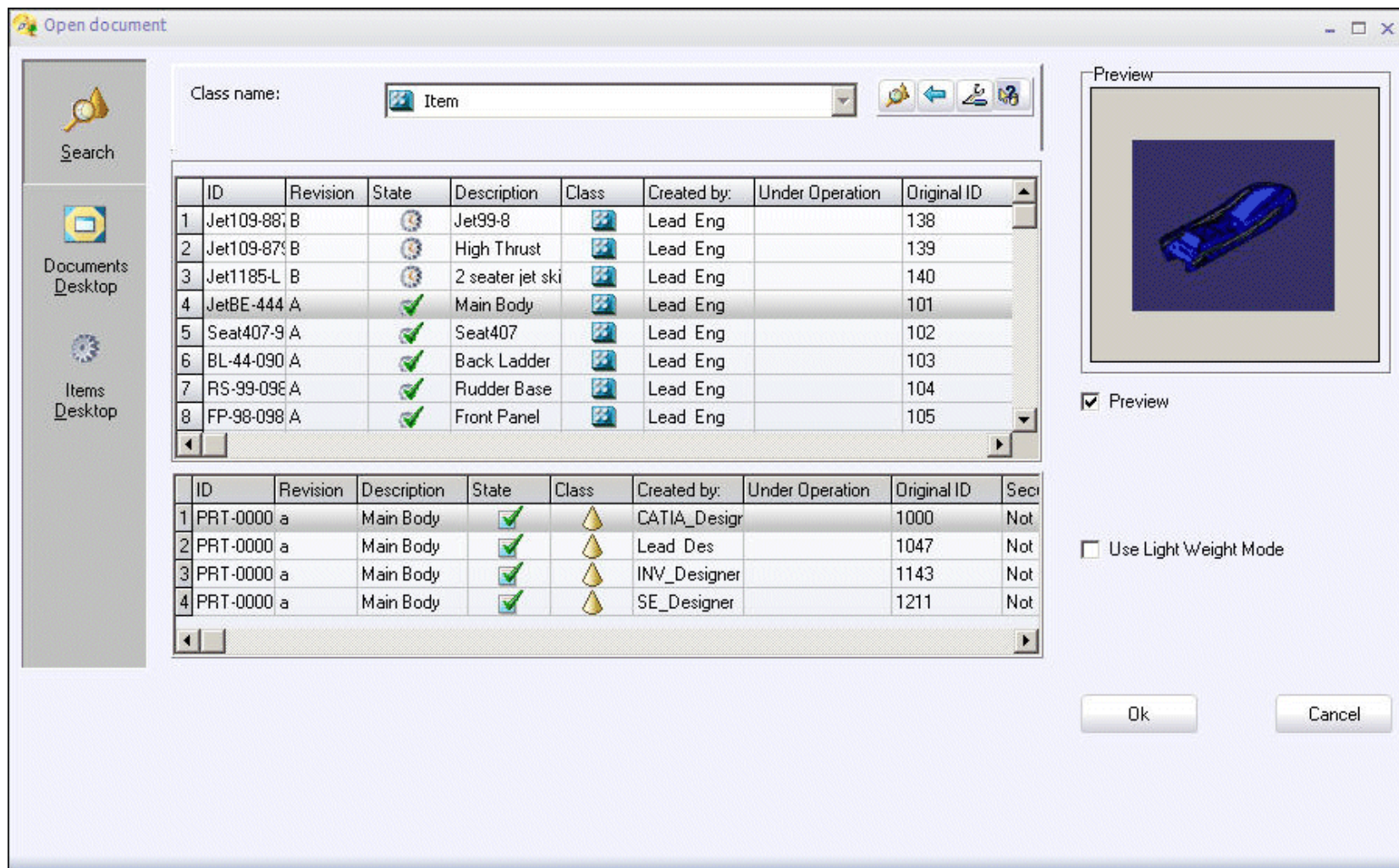
2. Enter conditions and values for the search.  
**Note:** Mark the **Retrieve only last public revisions** check box to limit the search to last public revision objects.
3. Click **Run**. The search results are displayed:



**Note:** The Preview option cannot be used in CATIA 64-bit.

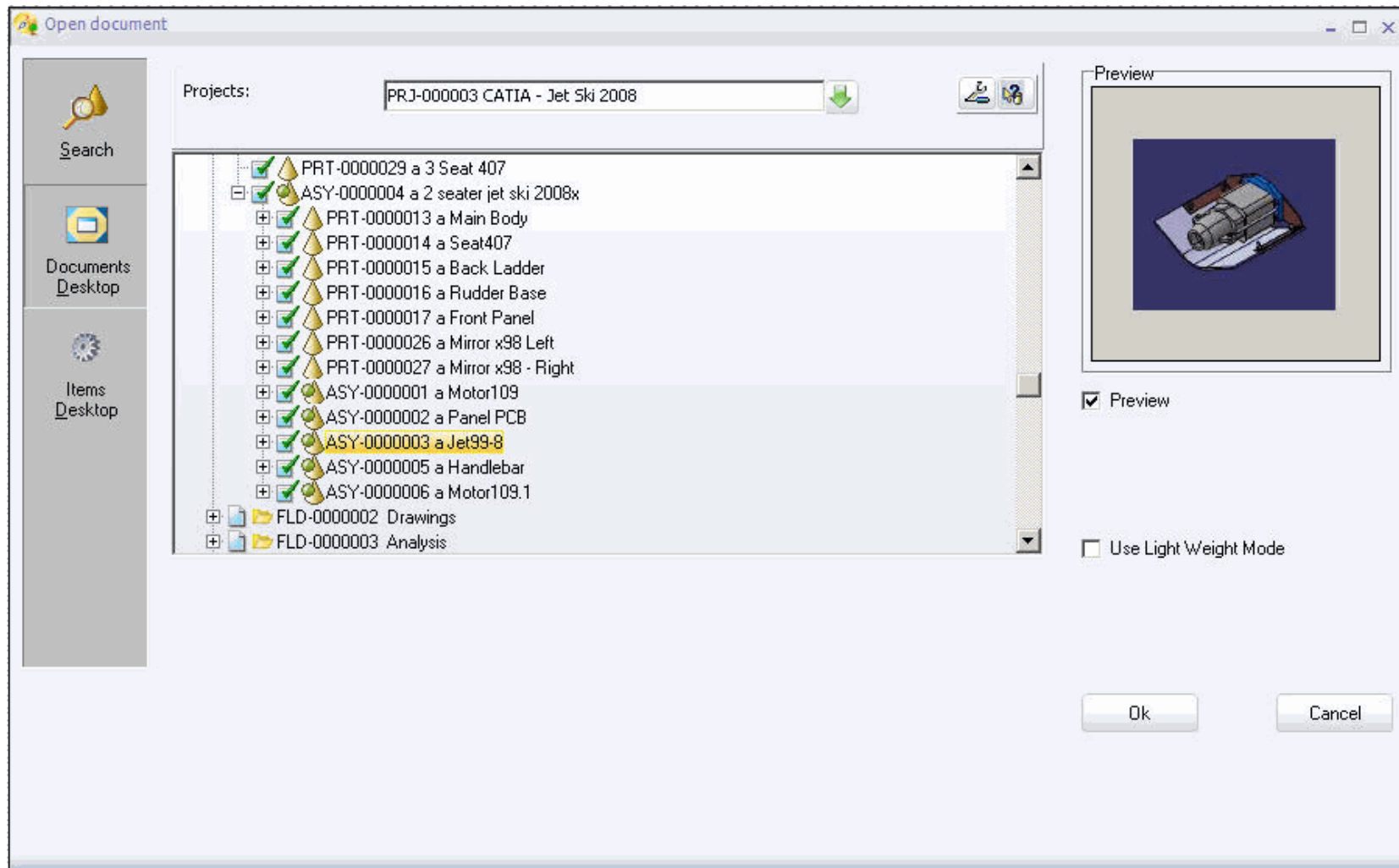
### Searching the Items Class

When you perform a search on the Items class, the Related Items pane appears in the lower section of the screen, displaying documents related to the Item selected from the Search Results list. Select a document from the Related Items pane and the related document opens.



## Documents Desktop option

Browse SmarTeam projects to retrieve CATIA documents.

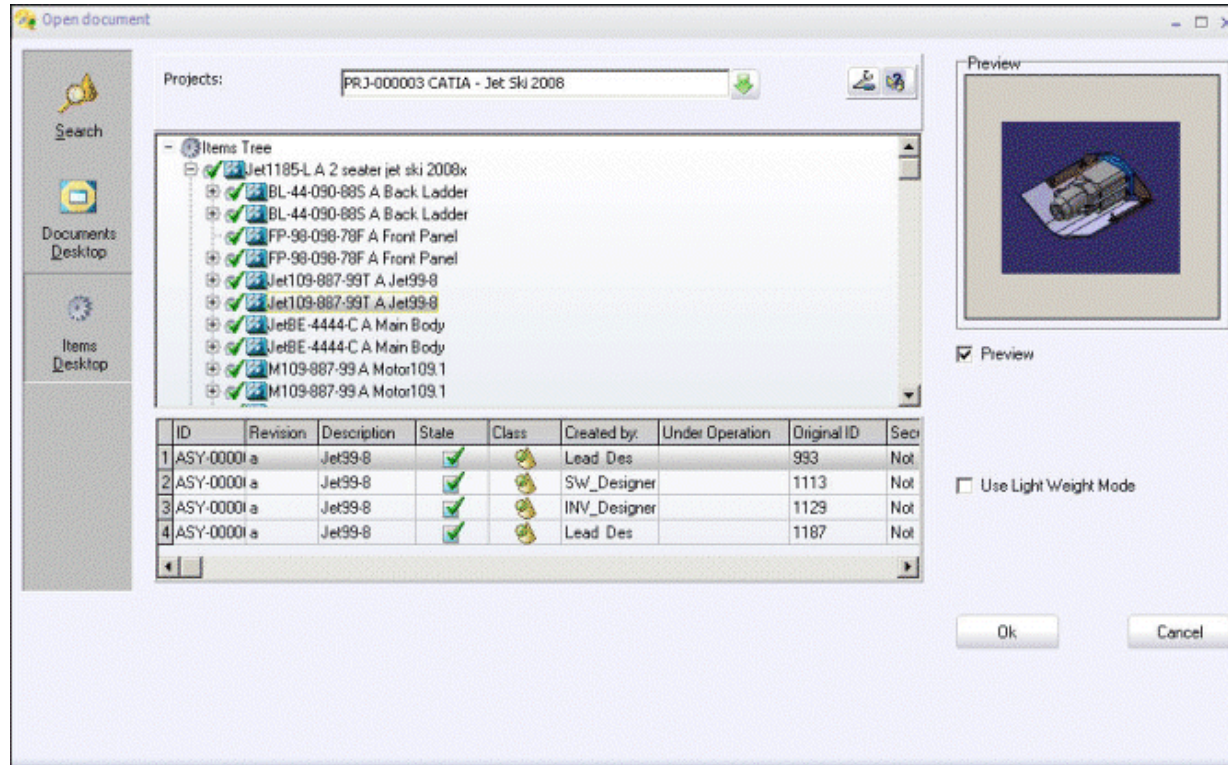


1. From the **Projects** field, select a project from the dropdown list.
2. Click **OK** to open the project.



## Items Desktop option


Browse the Item desktop and retrieve a specified CATIA document.

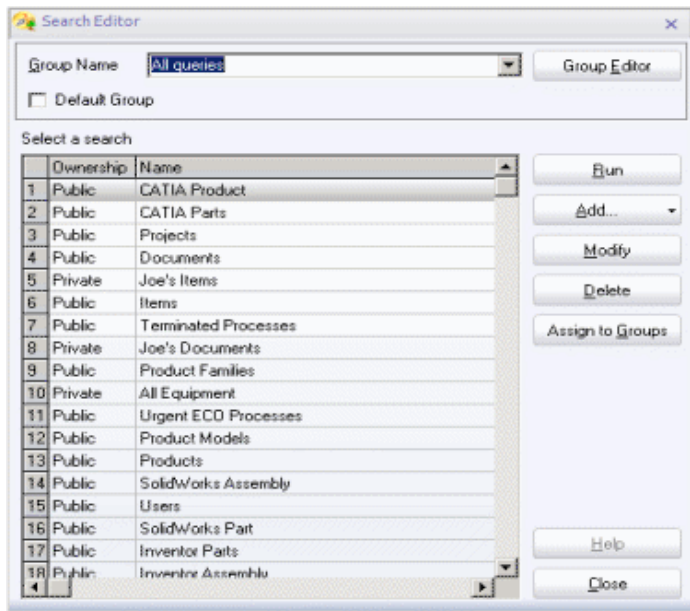


1. From the **Projects** field, select a project from the dropdown list.  
When you select a project, the Related Items pane appears in the lower section of the screen, displaying related items.
2. Select a document from the Related Items pane and click **OK** to open the document.



## SmarTeam Search Editor icon

In addition to the search options described above, you may also click the SmarTeam Search Editor icon  to perform a standard SmarTeam search.



For information on defining various settings for the Open dialog, see [Customizing the Open Dialog](#).



# Running a Predefined Search



This option enables you to view the previously defined searches. From the **Search Editor** window, you can:

- run a previously defined search.
- modify a search.
- create a new search.

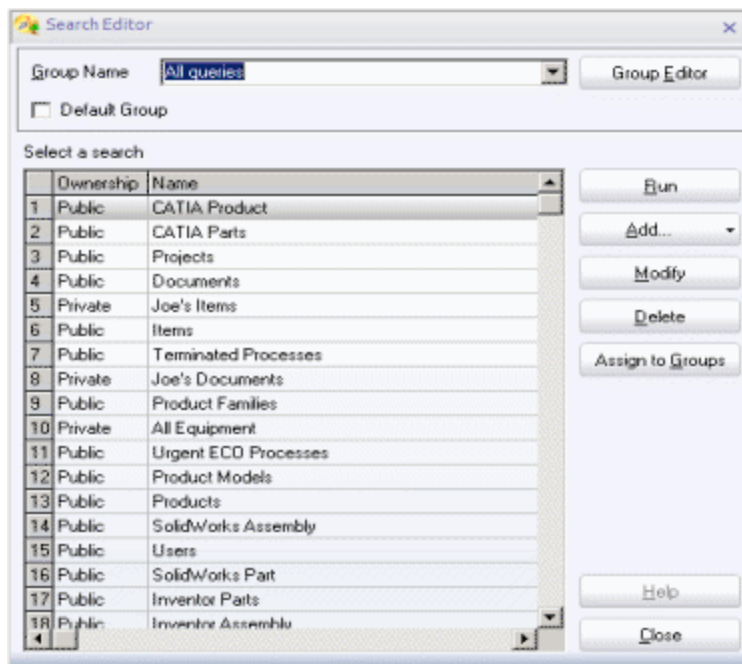
Once a search is defined and saved, you can run it over and over again. This powerful search tool can help you find your documents quickly and efficiently.

For example, you may have a search called New Parts whose search criteria is based on a specific creation date. Each time you run the search, you can locate the newest CATIA Parts. A search may contain several search criteria. The results of the search are displayed in a search results list. You can browse through the displayed list and view the Profile Card for each one. You can also launch a document straight into CATIA (by right-clicking on the document).



1. Click the **Find Document** icon  or select **SmarTeam > Find > Find Document**.

The **Search Editor** window is displayed, as shown below:



Note that the **Folder Document Environment** must be set to **Allowed** to be able to use the **Find** capabilities. For more information, see [Documents Environments](#). From the **Search Editor** window, you can:

- Click **Run** to run the selected search and display the search results, as described in step 2.
- Click **Add** to create a new search, and point to **By Attribute** or **By Example** to define a new search.
- Click **Modify** to modify the attributes of a previously defined search.
- Click **Delete** to delete a previously created search.

2. Choose a search and click **Run**.

The results are displayed in a search results list, as shown below:

CATIA Products: CATIA Product

Data Chart

ID	Description	SI
1	CATPRD-0042 Hack_Saw	
2	CATPRD-0044 Mechanical_Parts	
3	CATPRD-0043 Support	
4	CATPRD-0046 Internal_Carter	
5	CATPRD-0047 Wheel	
6	CATPRD-0045 Cutting_Mechanism	
7	CATPRD-0048 Trek_bike_R10SP2	
8	CATPRD-0049 Pedal_assy	
9	CATPRD-0050 Rear_wheel	
10	CATPRD-0051 handlebar	
11	CATPRD-0043 Support	
12	CATPRD-0042 Hack_Saw	
13	CATPRD-0054 CATPRD-0054	
14	CATPRD-0055 CATPRD-0055	

Profile Card Links Notes Revision Viewer

## CATIA Product

Document ID: ★ CATPRD-0042

Revision:  State:

Description:

**Engineering Information:**

Part Number:

Definition:

Nomenclature:

Product description:

Source:

Material:

General Details Revision

User login: joe

You can browse through the document displayed in the list. Each time you select a document, its Profile Card is shown on the right.



# Finding Out Where a Document Is Used

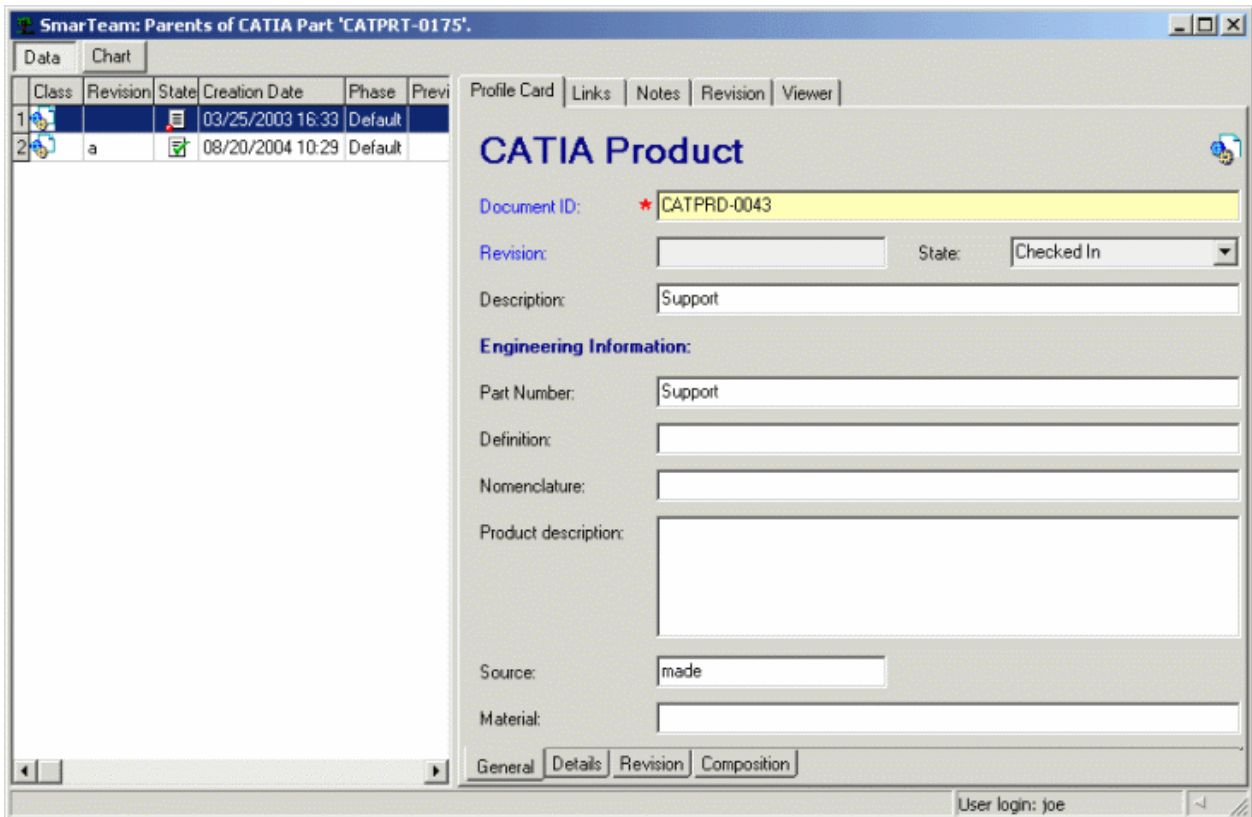
## Hierarchical Links (mainly Product Structure relationships)

When you open a Part or Product in CATIA, it is essential to keep track of all the Assemblies that use this particular document as a component before you begin to modify the Part/Product. SmarTeam enables you to locate all the parents of any document (using the **Where Used** option). This is particularly helpful when working with large Assemblies with many sub-Products and Parts as components.

The **Where Used** command determines the list of the documents which have a hierarchical link to the selected document. For example, it lets you define the list of assembly files using a given part.



1. Display a Part (or a Product) in CATIA.
2. Select **SmarTeam > Where Used**.  
A window is displayed listing all the parents of the Part.



Class	Revision	State	Creation Date	Phase	Preview
1	a	03/25/2003 16:33	Default		
2	a	08/20/2004 10:29	Default		

**CATIA Product**

Document ID: ★ CATPRD-0043

Revision:  State:

Description:

**Engineering Information:**

Part Number:

Definition:

Nomenclature:

Product description:

Source:

Material:

General Details Revision Composition

User login: joe

You can browse through the list to view the Profile Card of each document. You can also double-click on a document to launch the document into CATIA, but double-clicking it just opens it: this does not check it out.



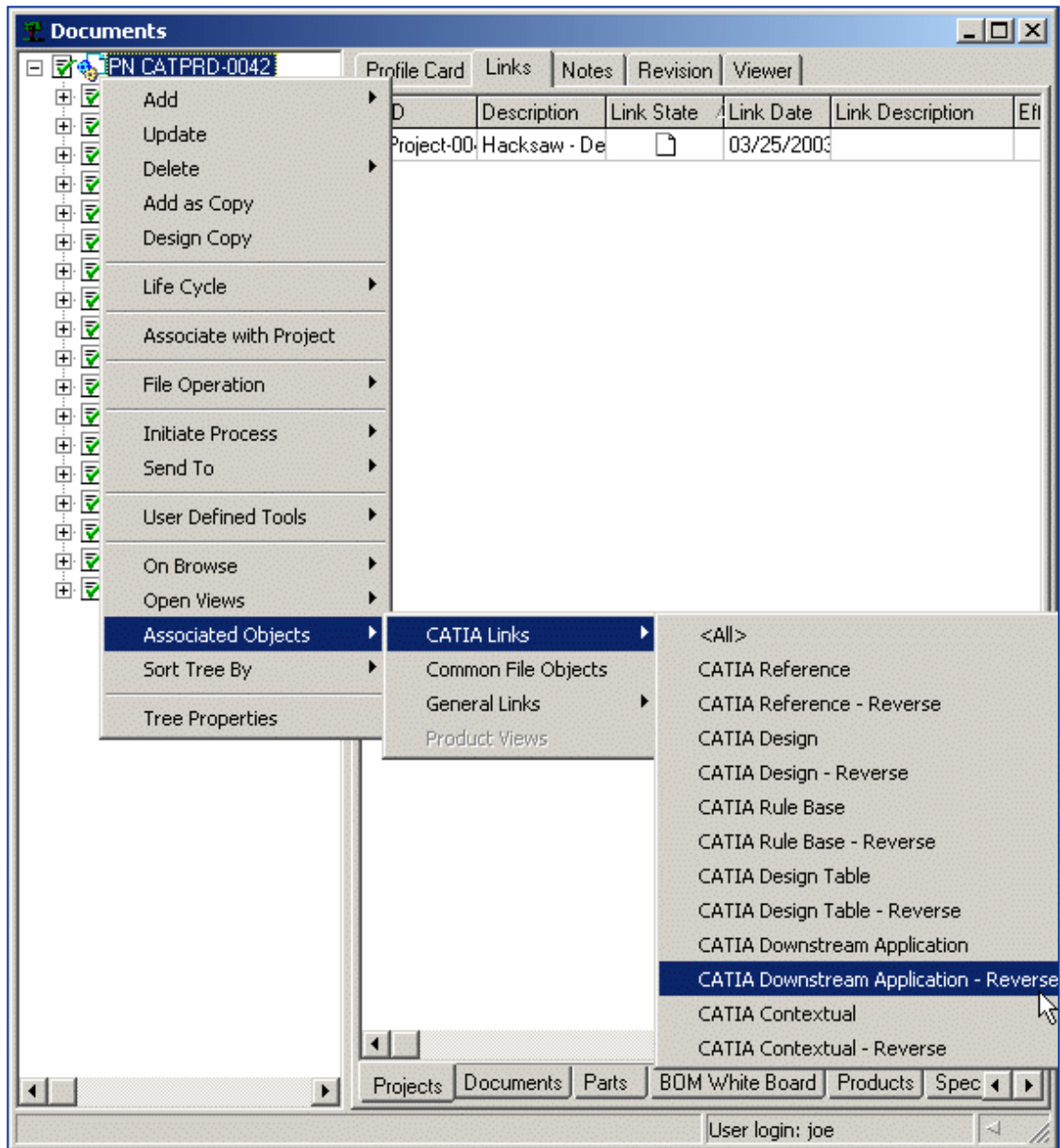
## CATIA Technological Links

To retrieve the same information on other types of links, you should look for the CATIA Reverse link information. you need to determine the appropriate link type used. For more information about the different types of links, see [Enriched Decision Support with All V5 Links](#).

Let's take the example of a CATDrawing referencing a CATProduct document. In that case, the link is a **CATIA Downstream Application** link.



1. Store a CATDrawing document referencing a CATProduct document inside SmarTeam.
2. Locate the CATProduct document inside SmarTeam.
3. To determine the documents referencing to the CATPart document, you have two ways to do so:
  - Thru the contextual menu: **Associated Objects>CATIA Links>CATIA Downstream Application Reverse**.



or

- From the Profile Card, click the **Links** tab. Select **CATIA Links > CATIA Downstream Application** in the **Link** dialog box.



# Showing Profile Cards




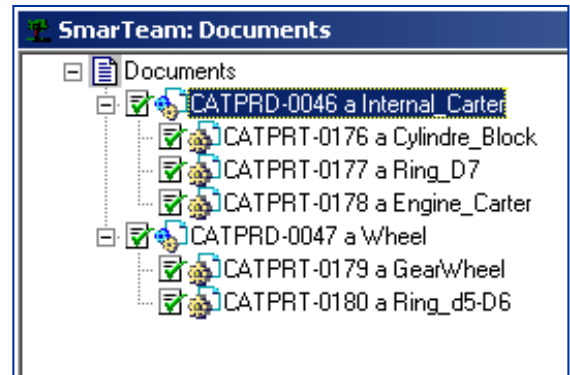
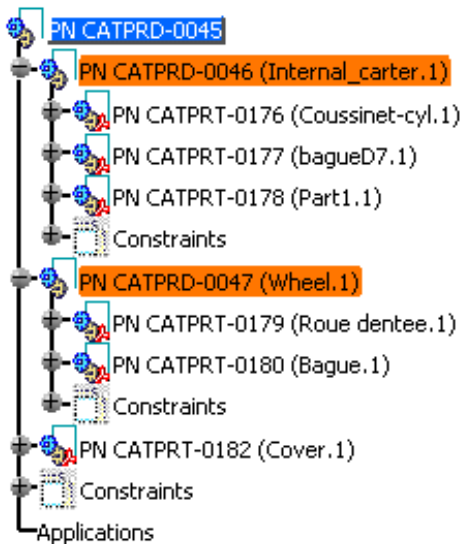
If you wish to view the relationships between documents open in your CATIA session and other documents as saved in SmarTeam, you can use **Show Profile Card**. This capability provides the database view of the document currently displayed in a CATIA session. It is available for all documents and can be accessed:

- From CATIA specification tree
- From the CATIA Desk tree
- From the 3D area



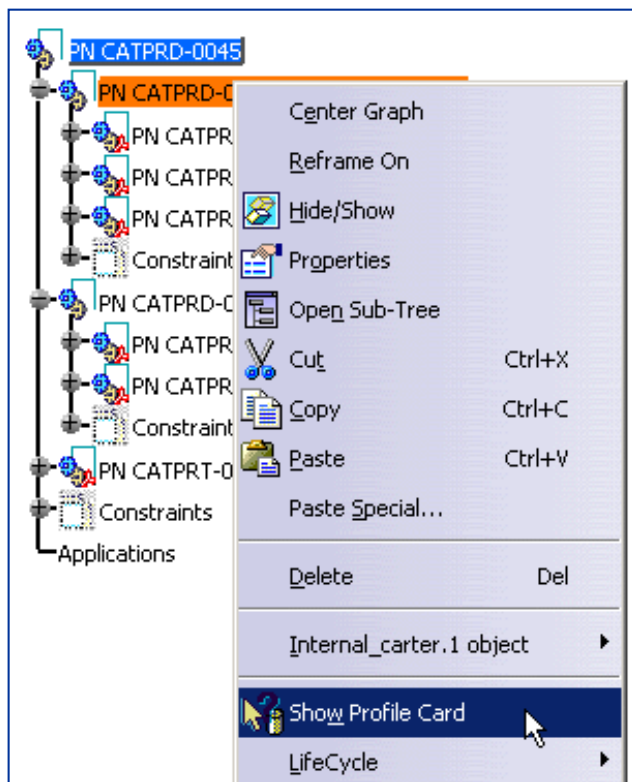
## From CATIA Specification Tree

1. Select the document of interest.
2. Click the **Show Profile Card**  icon or select **SmarTeam > Show Profile Card**.  
Note that the command applies to multi-selected documents:



A SmarTeam document window opens, giving access to the Profile Card, Links, Viewer etc (information as saved in the ST database), corresponding to the selection. If your selection contains a new entity not saved in SmarTeam, the application informs you.

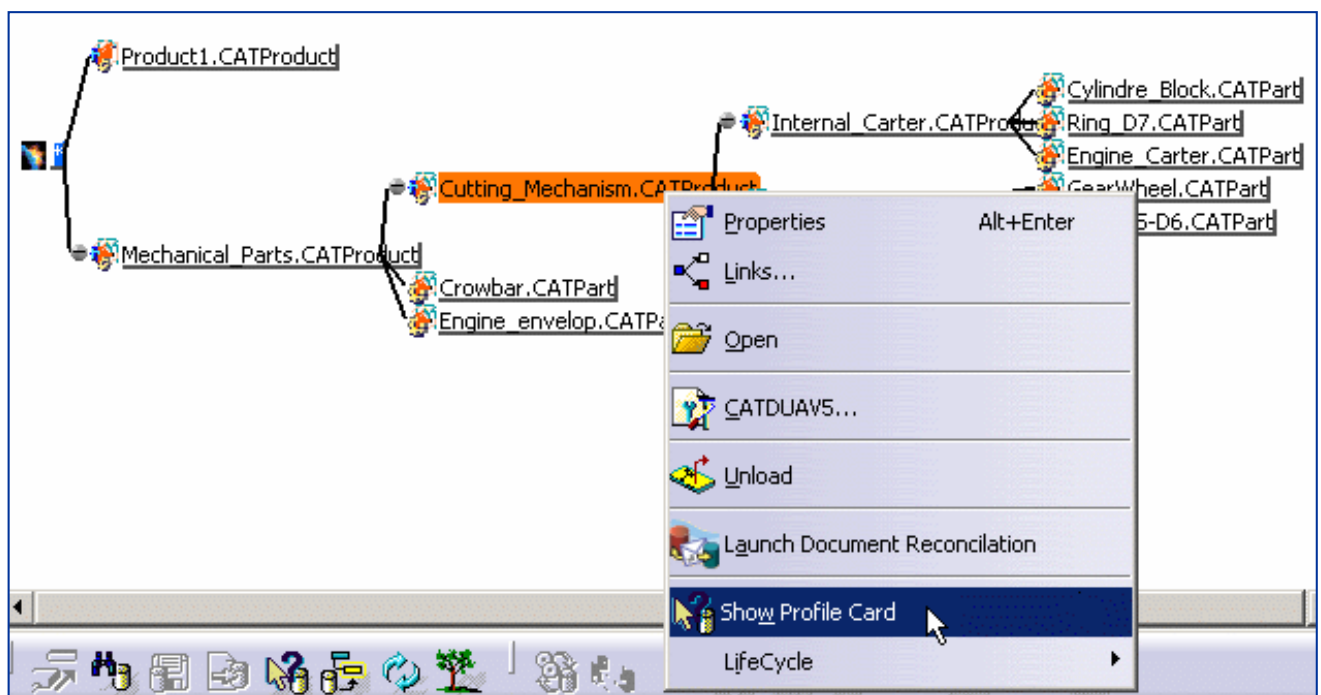
- You can run **Show Profile Card** via contextual menus available from the documents you select. The command applies to the documents you select, not to the current document. Just as a reminder, the application distinguishes **selected documents** from **active documents**.
  - Selected documents: To select a document, you just need to click it. Once selected, it appears as highlighted.
  - Active documents: double-clicking a document, activates that document, which sets up a working context. For example, if you double-click a Part, CATIA opens the Part Design workbench for you to access all different capabilities for editing that part.



## From CATIA Desk Tree

The CATIA Desk window lets you view the relationships between different documents and obtain information about their properties. Once in the window, you can use the **Show Profile Card** functionality on all documents.

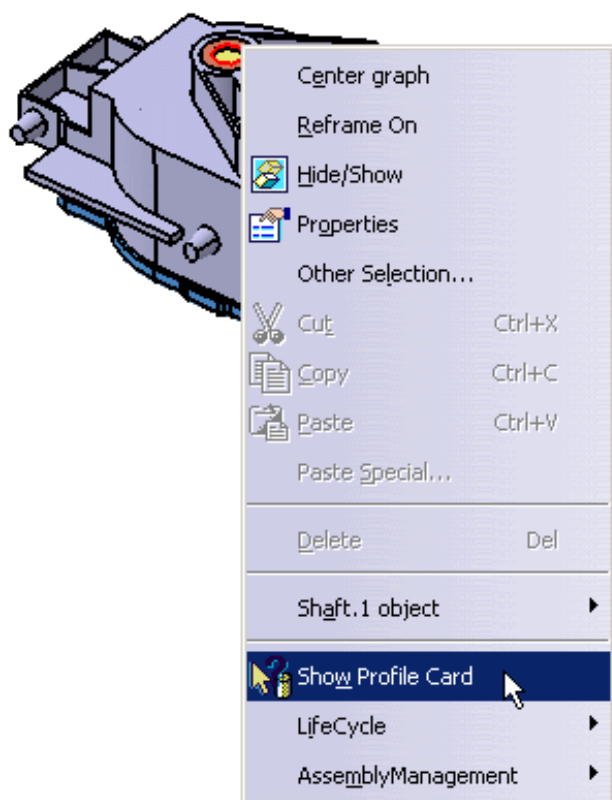
1. Right-click the document of interest.
2. Select **Show Profile Card** contextual command:



The profile card of the selected document is now displayed.

## From the 3D Area

You can also run **Show Profile Card** via contextual menus available from the geometry area.



# Duplicating an Existing Document



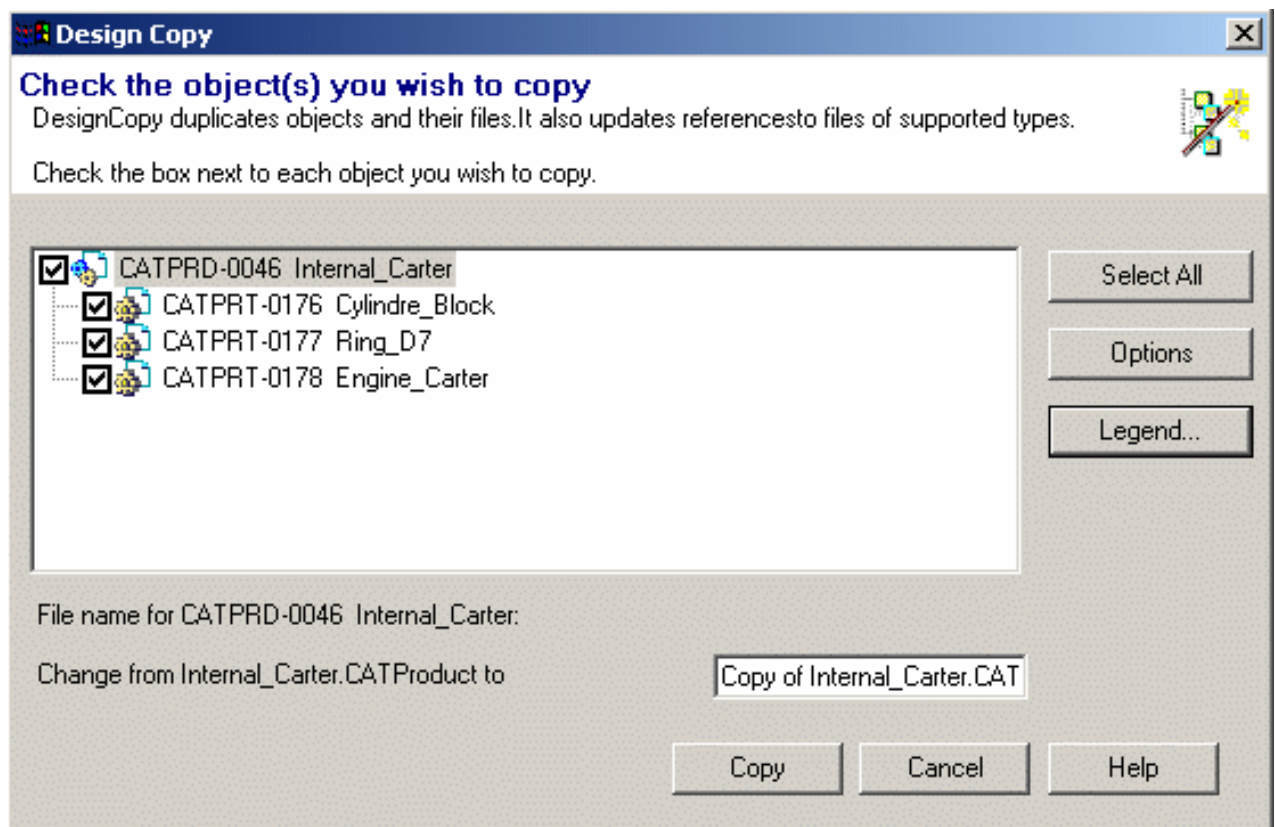
The **Design Copy** command in SmarTeam enables you to duplicate a design of an existing product, to be used as the design base for a new product. It also enables you to duplicate an existing document and its direct or reverse dependencies. It is a key function for re-using components and creating a new design which results in significant design phase productivity.



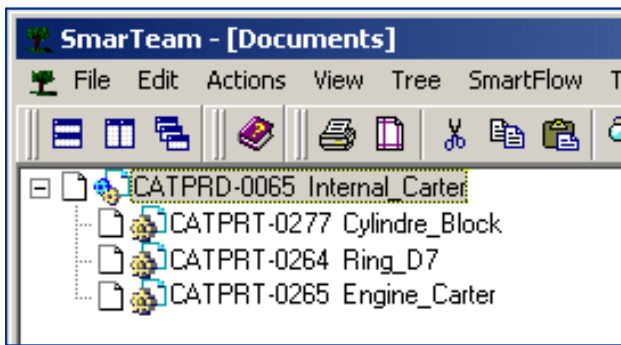
1. Right-click on the document from which you want to create a copy and select **Design Copy**.

The Design Copy dialog box appears, displaying all documents linked to the document you selected.

When there are contextual links, if an object is linked to the parent object with a tree link and also with one of CATIA links, these two objects will have the same status of checked/not checked. If the status of one of them is changed then the other one will be automatically changed. For example, if the CATIA Product contains a CATIA Part, which is also designed in the context of the CATIA Product (contextual link to product exists), then checking/not checking the CATIA Part will automatically check/not check the CATIA Product. This ensures the integrity of the design and prevents cross-references between CATIA Parts and CATIA Products.



2. Check the documents you wish to copy, if not already done.  
In our example, the user wants to copy a CATProduct document as well as the three CATPart documents it references.
3. To override the default setting for file names, you can type a new file name for the document selected for copying: just highlight the selected object and enter the new name in the lower text field.
4. After selecting and checking all files and options required to copy to the new assembly structure, click the **Copy** button to perform the copy operation.  
A new SmarTeam - Editor window appears, displaying the new copied document.



## Select All

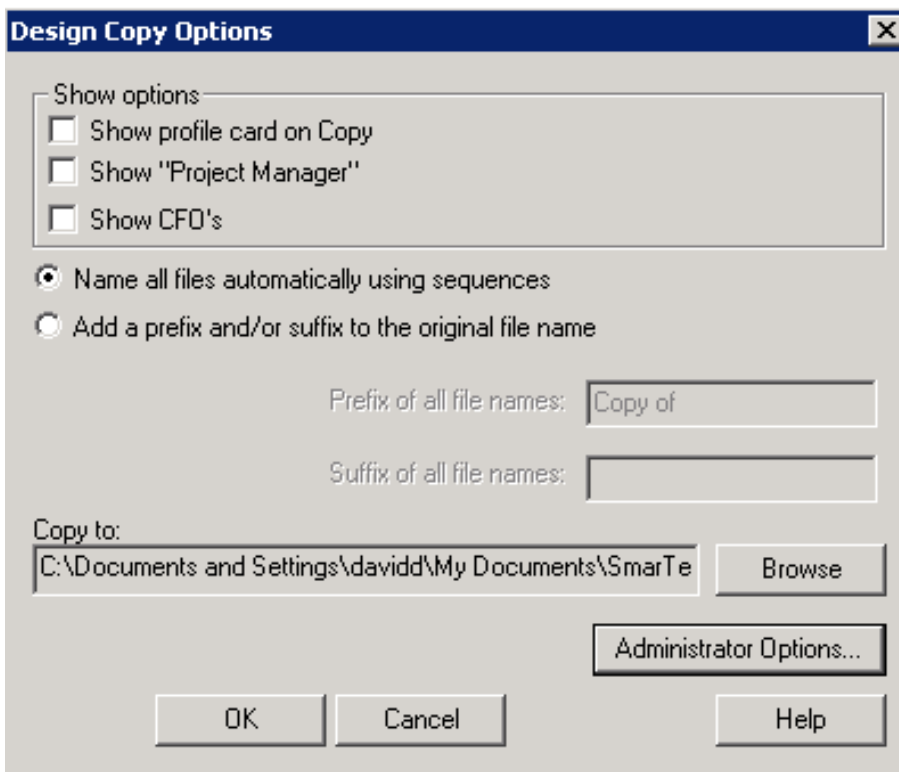
Clicking the **Select All** button selects all objects to copy to the new structure. When several designs share the same components, the **Design Copy** feature will copy the shared components only one time. The newly copied component will be the same in all designs.

**Note:** When a class is defined in the **Exclude** class list, the **Select All** button will not set it as selected. The user must select each object individually.

## Options

- Click **Options**.

The **Design Copy Options** window appears.



- In the **Show options** area check the relevant options:

- Check **Show profile card on Copy** to get a profile card for each new object. This profile card can be modified before creating new objects.
- Check **Show "Project Manager"** to link a new design to the project and set as a desk top object (Note: Design copy does not support Project Security and the user will not be able to set Project Security during the design copy).
- Check **Show CFO's** to display Common File Objects (CFO) in the Design Copy. When a member of a CFO is checked/not checked then all the other members of the CFO will also be checked/not checked. In the Design Copy window you can not set a different status for different members of a CFO. For example, if you check the CATIA Drawing, then all CATIA Sheets of the drawing will be checked automatically.

**7.** Select one of the following radio buttons:

- **Name all files automatically using sequences** in order to generate all the files names automatically.
- **Add a prefix and/or suffix to the original file name** in order to add a prefix and/or suffix to all original file names. If you select this option add the required prefix and/or suffix to the **Prefix of all file names** and/or **Suffix of all file names** fields.

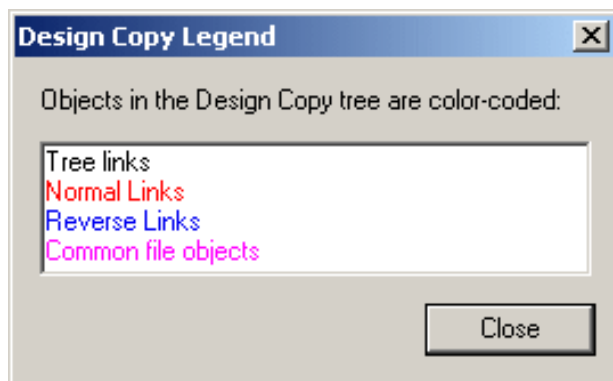
The file is copied to the destination entered in the **Copy to** field.

## Administrator Options...

The **Administrator Options** is disabled. It is only enabled if you are an administrator. If so, see [Customizing Design Copy](#) in the Administration Task section for further details.

## Legend

Clicking the **Legend...** button displays the Design Copy Legend window. This informative window identifies dependent object types according to colors set in the Tree Properties, Visual Setting dialog box.



**Design Copy** is described in greater detail in the *SmarTeam Editor* documentation.



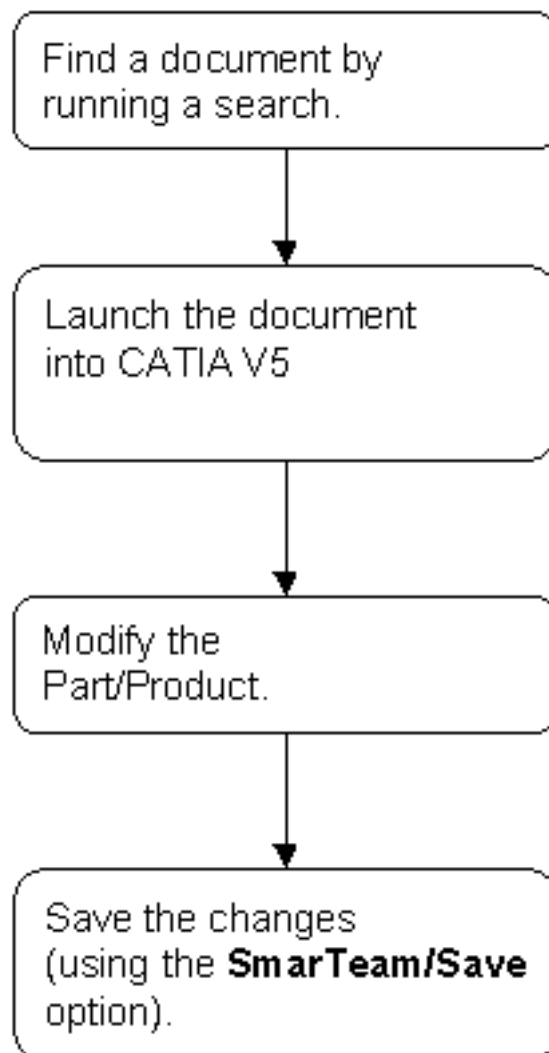
# Managing CATIA Parts

After you create a part in CATIA V5, save it in the SmarTeam database by choosing one of the SmarTeam **Save** options. This enables you to manage all your parts using the SmarTeam tools.

After you save the part, check the part into the SmarTeam vault by choosing the **Life Cycle > Check In** option. The part is placed into the Checked In vault.

As you design parts and products in CATIA, you often need to modify an object again and again. It is essential to locate the right document, often a time-consuming task. SmarTeam enables you to locate a document and launch it into CATIA quickly and easily.

The following steps illustrate how SmarTeam can assist you in locating and launching documents.



In addition, SmarTeam enables you to locate all the parents of any document (using **Where Used**). See "[Finding Out Where a Document Is Used](#)" for details.

In order to modify this part in CATIA you must launch the part into CATIA by choosing **Open For > Open for edit**. You are prompted to check the part out of the vault, thereby creating a new version of the part. When you wish to place the part into the vault for safekeeping, you can check it back into the vault (using **Life Cycle > Check In**).

In this manner, SmarTeam manages and protects all revisions of a part. The [Part and Product Comparison](#) functionality enables the opening of several versions of the same CATIA Part within a CATIA session, helping the designer to integrate the preferred CATPart version into the final project

**All life cycle operations on CATIA documents must be performed in the CATIA session.**

This section contains the following tasks:

# Checking Out a Part



In order to modify a part in CATIA, the part must be checked out of the vault.

SmarTeam also enables you to copy a file to your desktop without checking it out of the vault. This is useful when another user is working with the part (and has checked it out of the vault), but you wish to view the document at your desktop.

SmarTeam provides two methods for checking a part out of the vault, as summarized below.

- [Check out a Part \(from CATIA\)](#): If the part is currently displayed in your session (in read-only mode), you can perform the speedy Check Out operation.
- [Check out a Part \(from a SmarTeam window\)](#): If the part is not currently displayed at your desktop, you must find the part by running a search. From the displayed search results list, select the part and choose **File Operation > Open For...** then the **Open for edit** option.

Since the part is currently checked in, a message is displayed prompting you to check out the document. Click **Yes** to display the **Check Out** window and then check out the document. The part is immediately launched into CATIA.

## Checking Out a Part (from CATIA)

If a part is displayed at your desktop in read-only mode, you can check it out in order to modify the part.

When is a part displayed at your desktop in read-only mode?

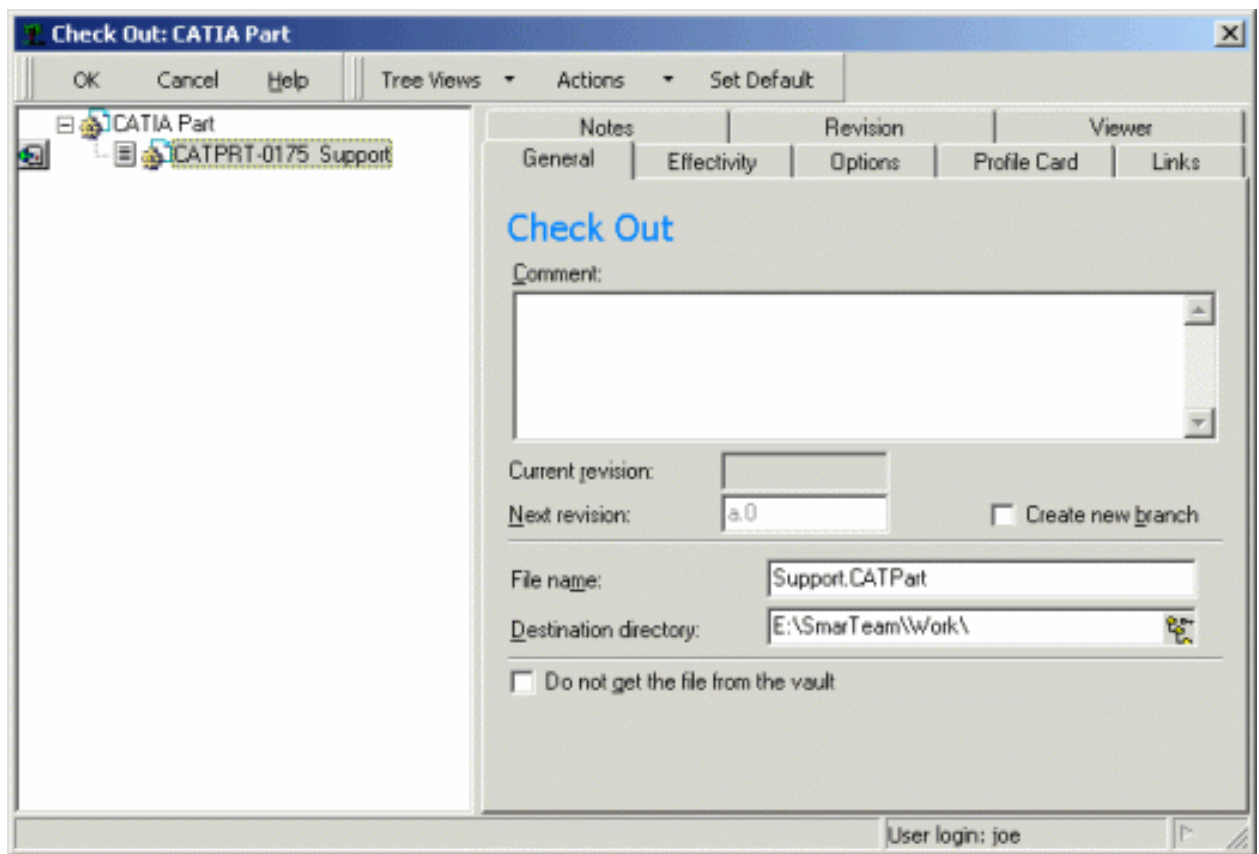
- You previously checked in the part, and a copy of the part remained at your desktop in read-only mode, as described in [Checking In a Part](#).
- You previously copied a file to your desktop (using the **Copy File** contextual command).

To check out a Part from the vault (when the Part is displayed at your desktop in read-only mode):

1. Activate the document containing the part.
2. From the **SmarTeam** menu, select **Life Cycle>Check Out**.

The Check Out window appears.

Check Out windows display SmarTeam information as set in the **Tree Properties** dialog box. For more information, see [Customizing SmarTeam Document Display Information](#).




3. Fill in the attributes, as described below in [Check Out Window](#), or accept the default attributes.
4. Click **OK**.  
The part is re-displayed at your desktop and it can now be modified.

## Checking Out a Part from a SmarTeam Window

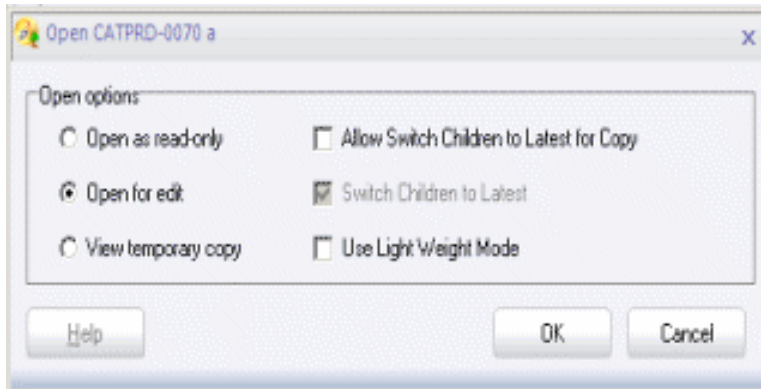
If a part is checked in to a vault, and it is not currently displayed at your project desktop, you must locate the part and then check it out in order to modify it in CATIA.

### To check out a Part:

1. Run a search to locate the document that you wish to check out:
  - Click the **Find Document** icon  or select **SmarTeam > Find > Find Document**.
  - Select a search and click **Run**. The search results are displayed in a search results list.
  - Browse through the list to locate the document you wish to modify.
2. Right-click the document and select **File Operation > Open For...**

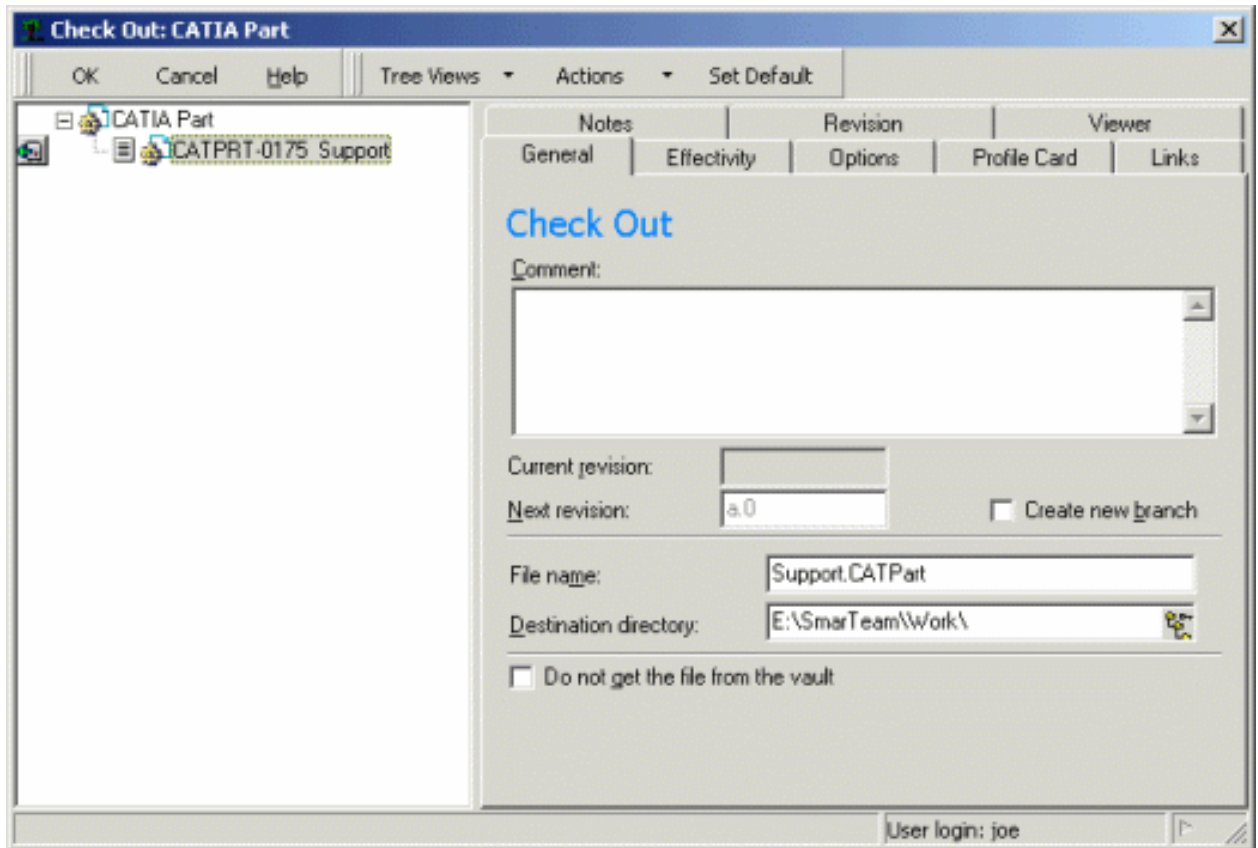
**A dialog box appears, providing three options for opening the document.**

3. Check the **Open for edit** option.



4. Click **OK** to confirm and close the dialog box.  
The **Check Out** window appears. Note that this window is also displayed when you choose the **Life Cycle > Check Out** option from the **SmarTeam** menu.

On the left side of the window, the **Check Out** icon appears to the left of the selected document. On the right side of the window, the revision number is automatically proposed by SmarTeam although you can assign a new revision to the document.




5. Fill in the fields in the **Check Out** window (optional).
6. Click **OK**.  
The status of the document is automatically changed to **Being Modified**.  
You may now work with the part in CATIA V5 and modify it. Remember to choose the SmarTeam **Save** option to save these changes and update the Profile Card.

## Check Out Dialog Box

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

Attribute	Description
<b>General tab</b>	
<b>Current revision</b> <b>Next revision</b>	The current revision field is the source revision identifier and the right field is the new revision. You may enter a different revision identifier in this field although it is not mandatory.
<b>Create new branch</b>	<p>Click this option to enable you to create parallel branches of a revision based on the same file. Each branch will be assigned a different revision number consisting of 3 characters (a.0.1 and a.0.2 for example) instead of the standard revision number consisting of 2 character (a.0 for example).</p> <p>For example: You may have a document named Pump*55. If the Branching value is Yes, two separate users may check out the file and each can create a new revision of this file. Two different revision numbers will be assigned to these files respectively (a.0.1 and a.0.2) and they can both be modified simultaneously.</p> <p><b>Note:</b> This field may be disabled according to the life cycle preferences defined by your administrator.</p>
<b>Destination directory</b>	The directory path of the vault in which the document will be located is automatically displayed. If you are using the vault server, the name of the generic server is displayed and the destination vault dropdown menu enables you to specify the generic destination vault in which the document will be located.
<b>Effectivity tab</b>	
<b>Phase</b>	Click an option from the dropdown list to define the phase of this revision. This field is descriptive only and does not affect revision status.
<b>Comments</b>	You may enter a comment in this field.
<b>Options tab</b>	
<b>Copy General links on Check Out/New Release</b>	Click on this option to copy general links from the previous to the newly created revision.
<b>Copy links to children on Check out</b>	Click on this option to copy all the links to children in the current revision to the new revision.

<b>Replace local files on Check Out from vault</b>	<p>Choose an option from the dropdown list:</p> <ul style="list-style-type: none"> <li>• <b>Yes</b> - to replace all local files that were copied during the Check Out operation.</li> <li>• <b>Yes for copied files</b> - to replace the file if a copy was made of this file during an earlier operation.</li> <li>• <b>Ask</b> - to prompt the user before making a copy of the file.</li> </ul>
<b>Set Default</b>	Click this option to assign the same check out information for all document revisions such as phase, effective dates, and notes.

 To check the document back into the vault for safekeeping, follow the instructions provided in [Checking In a Part](#).



# Saving a Part



Every CATIA part should be saved into the SmarTeam database. After you create a new part or modify an existing part, you need to choose one of the **SmarTeam Save** options:

- **Save:** Saves the document into the SmarTeam database.
- **Save As:** Saves the document into the SmarTeam database and defines the project and the parent folder of the document. For example, you can save the new part as a child of the **Beta Parts** Folder in the **Drive Shaft** project.
- **Save As Type:** Saves the document into the SmarTeam database in a different format and optionally links it to the master CATIA document used to create this document.  
**Note:** This option appears only for CATIA Part, CATIA Product, CATIA Shape and CATIA Drawing documents already saved in SmarTeam.

After you choose a SmarTeam **Save** option, a CATIA part **Profile Card** is displayed. After you fill in the attribute fields and click **OK**, the part is saved in the SmarTeam database.



**Although it is possible to save different document types in the SmarTeam database using SmarTeam Save Editor or Web Editor capabilities, you must use CATIA SmarTeam > Save to save the different CATIA documents in the database.**

This section shows you different tasks involved when saving a part:

- [Saving a Part for the First Time](#)
- [Saving a Part After Modifications](#)
- [Saving a Part as a Different Type](#)
- [Saving a Part with its Associated Design Table](#)
- [Viewing CATIA Links in the Design Table](#)

## Profile Card

Your administrator may customize the Profile Card (see Chapter 3, "Modifying a Profile Card" of the *SmarTeam Administrator's Guide*). This window will reflect the appearance of the Profile Card used in your SmarTeam application.

If you choose to work in **Batch Mode Save** mode (by checking the **Batch Mode Save** option from the SmarTeam menu), a new Profile Card is not displayed. Instead, the part is saved in the SmarTeam database with the default attributes. You can update the Profile Card attributes at any time.

In the Profile Card, the following information is displayed automatically:

- **Document ID:** This number is assigned by SmarTeam and it must be unique.
- Values may be entered in some attributes, if your administrator defined default values for this class of Profile Cards. These attributes may be changed.
- If a file name already exists in CATIA V5, a thumbnail image is displayed.

**Profile Card**

Select class: CATIA Part

Profile Card

## CATIA Part

Document ID: ★ CATPRT-0229

Revision: State:

Description:

### Engineering Information

Part Number: Part1

Definition:

Nomenclature:

Product description:

Source: unknown

Material:

Context Document:

General Details Revision

OK Cancel Help

- In the Details tab, CATIA file information is displayed.

**Profile Card**

Select class: CATIA Part

Profile Card

## CATIA Part

### File Information

File Type: CATIA Part

File Name: new-part

Directory: E:\SmarTeam\Work

### User Information

Created by:

Creation Date: 12

Modified by:

Modification date: 12

General Details Revision

OK Cancel Help

SmarTeam enables you to define the following information in the Profile Card:

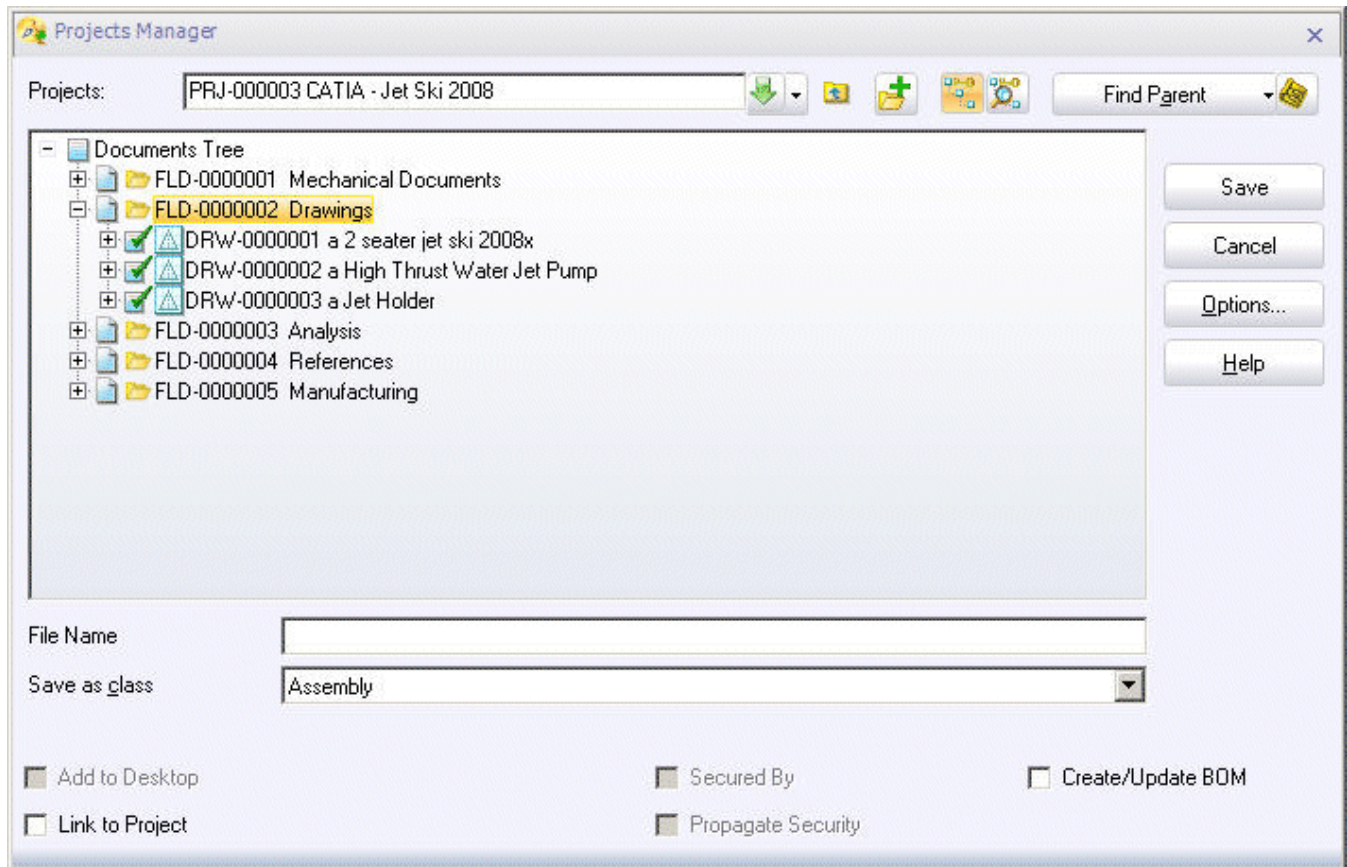
- You can save the part to a different SmarTeam class. A new Profile Card for the selected class is displayed.
- You can link the part directly to the SmarTeam project desktop.
- You can define attributes for the part.



## Saving a Part for the First Time




1. In CATIA V5, create a new part.
2. From the **SmarTeam** menu, choose **Save**.  
The Project Manager dialog box appears. In this window you can define the project and the parent folder to which the document belongs.

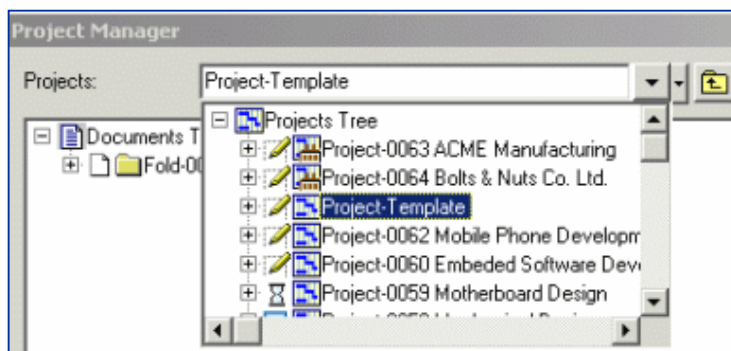


If the document is already saved, the CATIA file name is displayed.

You can define the level of sub-branches displayed in the project selection tree and/or the document selection tree.

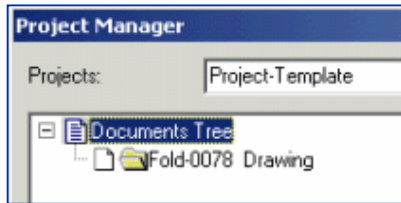
If you leave the file name field empty, when clicking **Save** an automatic file name will be generated.

3. Click the  icon and choose a project from the Projects Tree dropdown list that appears.  
The part will be saved as a document in the selected project.



4. Choose a parent folder from the Document Tree. The part will be saved as a child of the selected parent folder.

The classes displayed in the dropdown list are defined by your administrator.

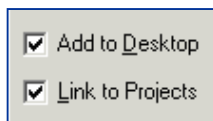


5. Select the **Add to Desktop** checkbox to ensure that the document will be shown in the different SmarTeam windows.

Select the **Secured By** checkbox to ensure that the object you select is linked to the project and receives security settings according to the Link to Project settings defined for the entire project.

**Note:** When the database contains Project-based security and you assign "Secured by Project" for a leaf class (but not for a super-class), the Secured By checkbox in the Project Manager window is disabled.

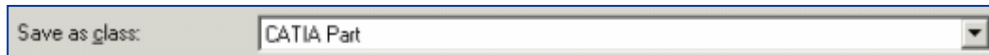
6. Select the **Link to Projects** checkbox to link the part directly to a project.



7. Click **Save**.


The **Profile Card** window is displayed with a CATIA part Profile Card, as shown above. It is recommended to enter a name for the part in the **Description** attribute.

8. If you wish to save the part to a different class, click the arrow to the right of the **Save as class** field and choose a class.




9. Select the tabs to review default information about the file.
10. Select the **Details** tab. If the document is new, the **File Name** field is empty. There are two ways to save: Either you give your part a file name or you let the **Save** function define a name for your part. If you leave the field blank, the name is built using the CN\_ID attribute.
11. Specify the attributes for the part.
12. Click **OK**. SmarTeam - CATIA Integration automatically saves your file to the database, giving it a unique identity, where it can be easily found for later use. By default, the text entered in the **Description** field is displayed in the tree browser next to the ID number. It is useful to assign a meaningful name to the document in the **Description** field. If you now want to check in the document, see [Checking In a Part](#).

## Creating a Folder in a Project or in an Existing Folder

When saving a part, you may create a new folder and save the part to this folder. The Create Folder  button is enabled when you select either a project or a folder and it is not available when you select a document.

### Creating a Folder in a Project

1. Select a project from the Projects dropdown list.
2. Select the Documents Tree node and then click . A profile card for the new folder is displayed.
3. Fill in the attributes in the folder's profile card. If the folder is created as the child of a project, the folder is automatically set as a desktop object. If the folder is created as a child of another folder, the folder will not be set as a desktop object. Note: This default setting can be overridden by the user.

### Creating a Folder under an Existing Folder

Perform the procedure described above, selecting a folder rather than a project in Step 1.

## Add to desktop

- **Add to desktop:** The **Add to desktop** check box will be available only when the **Link to Project** option is checked.
- **Secured by:** Select the **Secured by** check box when using Project-based security.



When you mark the **Secured by** check box, the **Link to Project** check box is marked and will be disabled for changes.

## Saving a Part After Modifications



After a part is added to the SmarTeam database, you are likely to modify the part in CATIA many times. Each time you modify the part, you must save it using one of the **SmarTeam Save** options.

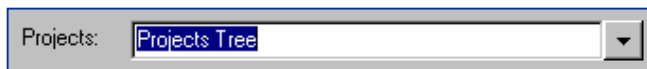
- **Save:** The part is saved in the SmarTeam database, with all its modifications. Its Profile Card is not displayed at this time.
- **Save As:** The SmarTeam Save As window is displayed. In this window, you can define the project and parent to which the part belongs. The part together with its new links is saved in the SmarTeam database.



1. Open an existing part in CATIA V5.  
You can run a search to locate the part and then launch it directly into CATIA V5.
2. From the **SmarTeam** menu, choose **Save**.  
The part is saved in SmarTeam and its Profile Card is updated accordingly.
3. You can define the level of sub-branches displayed in the project selection tree and/or the object selection tree.  
Click **Options** to display the Save Options window and click the **Tree Setting** tab.
4. Check the appropriate checkboxes.

Alternatively, you can do as follows:

1. Choose **Save As**.  
The SmarTeam Save As window is displayed.
2. Choose a project from the Project Tree. The part will be saved as a document in the selected project.



3. Choose a parent folder from the Object Tree.  
The part will be saved as a child of the selected parent.
4. Click **Save**.  
The part (together with its hierarchical links) is saved in SmarTeam and its Profile Card is updated accordingly.

## Saving a Part as a Different Type



The **Save as Type** option saves the current document into the SmarTeam database in a different format and optionally links it to the master CATIA document used to create this document. This safely saves the created document to SmarTeam, eliminating the possibility of mistakes when manually defining the file type and linking it to a specific revision.

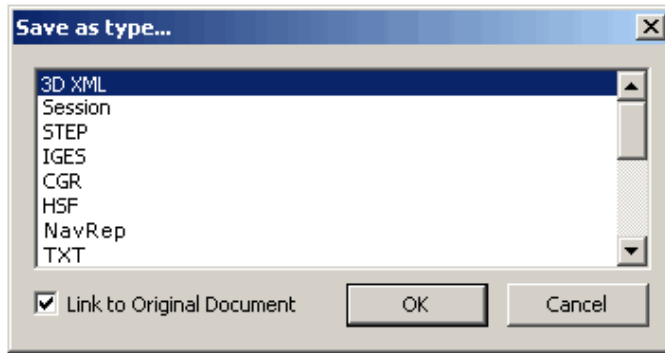
As a result, a new object is created in SmarTeam that represents the created format, with the correct behavior (according to the selected format), and is linked to the original format (if this option is selected).

All this is done from within CATIA, without the need for additional operations in SmarTeam - Editor.

**Note:** This option appears only for CATIA Part, CATIA Product, CATIA Shape and CATIA Drawing documents already saved in SmarTeam. This functionality is disabled when the user overrides the Save As dialog with the SmarTeam Save As dialog.

1. From the **SmarTeam** menu, choose **Save as Type....**

The **Save as Type...** window appears.



2. Select the file type.
3. Click **OK**.
4. Select the **Link to Original Document** check box to link the current file to its original document.
5. The Project Manager window appears. The name of the file shown in the File Name field is in the following format:  
<Original File Name>.<Original File Extension>.<Destination File Extension>
6. Type a new file name if required. The **.cgr** extension is added automatically.  
The Object and CATIA Exchange link are created. The direction of the link is from the original object to the exported object.

For example: In the Data Model Designer, the STEP mechanism appears. If the administrator does not assign the STEP mechanism to one of the classes (e.g., Exchange), STEP will not appear in Save as Type list.

For a complete list of the available file types, see [Available File Types](#).

To specify that the **Link to Original Document** check box will appear checked by default, set the **CATIA.LinkToOriginalDoc** key to Yes.

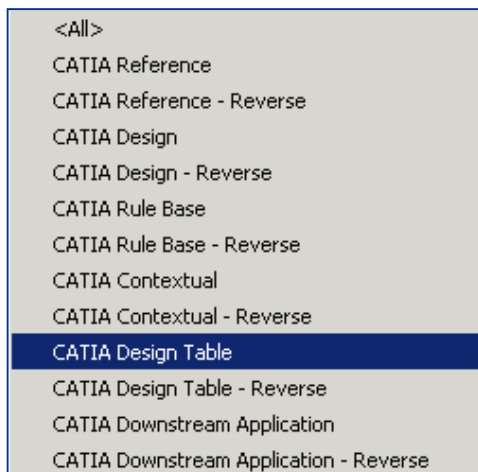
## Saving a Part with its Associated Design Table



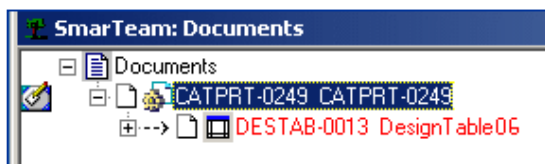
1. Create a basic part.
2. Click the **Design Table** icon  to create a design table for the part.  
To know how to create a design table, see the *Knowledge User's Guide*.
3. Once created, save the Excel file as well as the design table you generated.
4. Now, to save these documents in SmarTeam, click the **Save** icon  or **SmarTeam>Save**.  
This declares the two documents i.e. the part and the Design Table, and saves them in the database. The [Project Manager](#) dialog box appears.
5. Click **Save**.  
The Profile Card dialog box appears for the part.
6. Click **OK**.  
The Profile Card dialog box now shows information specific to the Design Table:



7. Click OK.
8. By default, you cannot see the link between the part and its design table. To visualize it, right-click the part and select **Associated Objects > CATIA Links**.
9. In the window that appears, choose **CATIA Design Table**.



You can now see the link between the CATIA part and the Design Table:



## Viewing CATIA Links in the Design Table

When you save a CATIA design table (Microsoft Excel) to SmarTeam, the table is assigned CATIA Design Table behavior. When you browse to this CATIA design table, CATIA links are visible in the profile card Link tabs.

When you create a table and save it to SmarTeam, perform a Check In or Release operation using the SmarTeam – Excel integration and then connect it to a CATIA document, a link is created when the CATIA document is saved to SmarTeam.

Note: When viewing the table object in SmarTeam, the CATIA links are not visible. This occurs because a checked in or released design table cannot be updated with CATIA behavior in SmarTeam. Therefore, the Check Out/New Release

operation should be performed on the table object before saving the CATIA document to SmarTeam.



# Available File Types

The following table contains the file types and behaviors for CATIA Product, Part, Shape and Drawing.

Original File Type	Available File Type	SmarTeam CLB	File Extension
CATIA Product	STEP	STEP	stp
	Session	CATIA V4 Session	session
	IGES	IGES	igs
	3DXML	3D XML	3dxml
	CGR	CATIA Representation	cgr
	HSF	HOOPS HSF	hsf
	NavRep	NavRep	NavRep
	TXT	CAD Document	txt
	WRL	VRML	wrl
CATIA Part	Stl	STL	stl
	IGES	IGES	igs
	CATIA V4 Model	CATIA Model	model
	STEP	STEP	stp
	3DMAP	3DMAP	3dmap
	3DXML	3D XML	3dxml
	CGR	CATIA Representation	cgr
	HSG	HSG	hcg
	NavRep	NavRep	NavRep
	VPS	VPS	vps
	WRL	VRML	wrl
CATIA Drawing	DXF	DXF	dxg
	DWG	AutoCAD Document	dwg
	CGM	CGM	cgm
	SVG	SVG	svg
	GL2	GL2	gl2
	PS	PS	ps
	PDF	PDF	pdf
	TIF	Image Document	tif

	JPEG	Image Document	jpg
	IG2	IGES	Ig2
	3DXML	3D XML	3dxml
CATIA Shape	stl	STL	stl
	IGES	IGES	igs
	CATIA V4 Model	CATIA Model	model
	STEP	STEP	stp
	3DMAP	3DMAP	3dmap
	CGR	CATIA Representation	cgr
	WRL	VRML	wrl



# Checking In a Part



This page discusses the following subjects:

- [Checking In a Part for the First Time](#)
- [Checking In a Part](#)
- [Check In Dialog Box](#)

## Checking In a Part for the First Time

When a Part is first saved into the SmarTeam database, it is automatically assigned the **New** status. This means that the Part has not yet been checked into a SmarTeam vault.

To protect the Part from modifications, place the Part into the SmarTeam vault by checking it in. After the Part is checked in, its status is changed to **Checked In**.

What happens next?

- To launch the Part into CATIA V5 and modify it, the Part must be checked out. When it is checked out, a new revision number is assigned to it. For more information, see [Managing Parts](#).
- The Part can be copied in your CATIA session in read-only mode.

## Checking In a Part

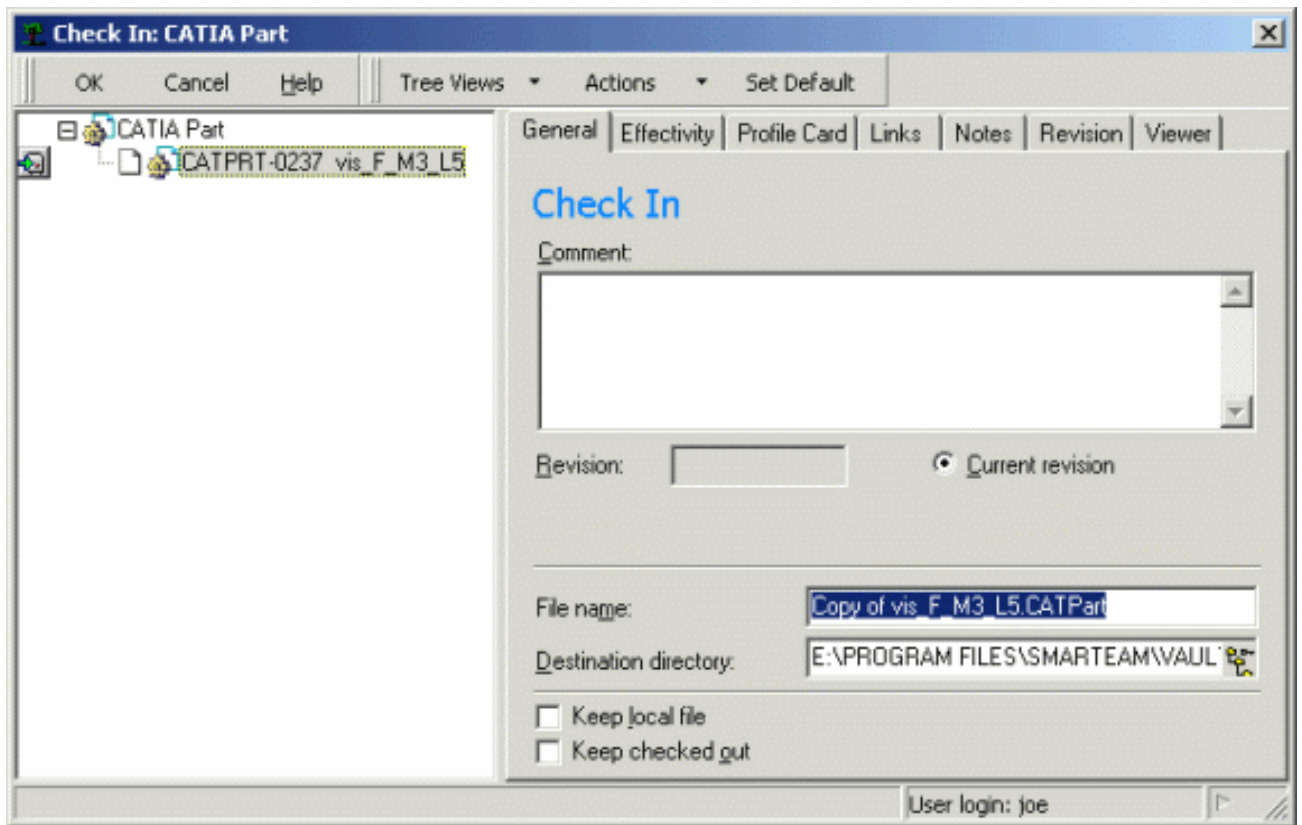
After a Part has been checked out and modified, it must be placed back into the vault. At this point, you can check the document back into the vault in one of two ways:


- **Check In**, as described below, 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**, as described in [Releasing a Part](#), moves the document to the **Released** vault. Thereafter, you can apply the **New Release** operation on the document to carry out additional changes.



1. Activate the document containing the Part you want to check in.
2. In CATIA, select **SmarTeam >Lifecycle>Check In**.

**The Check In window is then displayed.**



3. On the left-hand side of the window, the **Checked In** icon  appears to the left of the selected document. On the right-hand side of the window, the Check In dialog box is displayed.
4. Fill in the fields in the Check In dialog box, as described in [Check In Dialog Box](#) below. These fields are optional, and you may keep the default.
5. Click **OK** to check in the Part and exit the Check In view, or click **Apply** to perform the operation and remain in the view.  
The Part is now placed in the Check In vault for safekeeping. Note that the status of the document is now changed to **Checked In**. A copy of the file remains in your CATIA session in read-only mode.  
You can check it out again in order to edit the Part, as described in [Checking Out a Part](#).

## Check In Dialog Box

The following describes the attributes in the Check In dialog box:

Attribute	Description
Revision	The left revision field is the source revision identifier and the right field is the new revision. You may enter a different revision identifier in this field although it is not mandatory.
Directory	The directory path of the vault in which the document will be located is automatically displayed. If you are using the vault server, the name of the generic server is displayed and the destination vault dropdown menu enables you to specify the generic destination vault in which the document will be located.

<b>Comments</b>	You may enter a comment in this field.
<b>Set Default</b>	Click this option to assign the same registration information for all document revisions such as phase, effective dates, and notes.
<b>Effectivity</b> tab	
<b>Phase</b>	Click an option from the dropdown list to define the phase of this revision. This field is descriptive only and does not affect revision status.
<b>Effective From / Effective Until</b>	Click the <b>Date</b> button to enter dates in these fields.



# Releasing a Part



When a Part is ready to be moved to production, choose the **Release** option in order to place it in the **Released** vault. This is generally done when a supervisor releases a stage of development of the document. The Part is then assigned the **Released** status. A Part can be released from the **Checked In** status or the **Being Modified** status.

When performing a **Release** operation from Checked in state, only revision-related attributes are updated. To update non-revision attributes, you must check out, update and then release the document.

Note: When working with PLM DB, the Revision Property is not mapped by default in Integration Tools Setup and you must map the Revision Property manually.

Once a Part 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 Part can be checked out as a **New Release** with a new revision number.



1. Display the Part in CATIA.
2. In CATIA, select **SmarTeam >Lifecycle > Release**.  
The Release window is then displayed.  
The fields in the Release window are the same as those in the Check In window.
3. Fill in the fields.
4. Click **OK**.  
The status of the document is automatically changed to **Released** in the **State** field.

## Checking a Part Out of the Released Vault

To check a Part out of the Released vault, you must use the **New Release** option, as described in [Checking Out a New Release](#). A new revision of the document is created and the previously released revision remains in the vault.



# Creating a New Release



When a document has the **Released** status it can only be taken out of the vault by clicking the **New Release** command. This command automatically creates a new revision of the document, thereby saving the previously Released revision.

SmarTeam also enables you to copy a file to your desktop without checking it out of the vault. This is useful when another user is working with the part (and has checked it out of the vault), but you wish to view the Part at your desktop.



SmarTeam provides two methods for creating a New Release for a released document.

## Creating a New Release from CATIA

If the part is currently displayed at your desktop (in read-only mode), you can perform the **New Release** operation from the **SmarTeam** menu.

## Creating a New Release from a SmarTeam menu

1. If the part is not currently displayed in your CATIA session, you must find the part by running a search.
2. From the displayed search results window, double-click the Part.
3. Since the part is currently checked in, a message is displayed prompting you to check out the document. Click **Yes** to display the **New Release** window.
4. Check out the document.  
The part is immediately launched into CATIA.



# Using Mapped Product Properties



This section discusses the following topics:



- [Saving Properties in SmarTeam](#)
- [Updating a CATIA Property Following a Change in SmarTeam](#)

Prior to performing the scenarios, some properties will already have been mapped in CATIA and SmarTeam by the administrator. This is described in [Defining Property Mapping](#).

## Saving Properties in SmarTeam

This task shows you how to define materials and **Part Number** properties on a CATPart document and to transfer this information from CATIA to SmarTeam.



1. Select the part in your CATIA session.
2. Click on the **Apply Materials**  icon to display the **Library** dialog box.
3. For example, click on the **Plaster** item  and click **OK**.  
Plaster is added to both the specification tree and the geometry area.
4. Select **Edit > Properties**.
5. Select the **Product** tab from the Properties dialog box that appears.
6. In the **Part Number** field, enter the following text PN-0000053-A1:

**Properties** [?] [X]

Current selection : Part1

Mechanical | Mass | Graphic | **Product**

Product

Part Number PN-0000053-A1

Revision


Definition

Nomenclature

Source Unknown

Description

7. Click **OK**.

8. Click the **Save** icon  or select the **SmarTeam > Save** command.

9. From the dialog box that is displayed, select a project and click **OK**.

The document is now declared and its properties are saved in the database. In the SmarTeam profile card corresponding to the document, you can see that the fields **Material** and **Part Number** have been completed:

Profile Card | Links | Notes | Revision | Viewer

## CATIA Part

Document ID: ★ CATPRT-00000053

Revision:  State:

Description:

### Engineering Information

Part Number:

Definition:

Nomenclature:

Product description:

Source:

Material:

Content Document:

General | Details | Revision

Should you subsequently modify any CATIA properties you can update the database using **SmarTeam>Properties>Save in Database**.

## Updating a CATIA Property Following a Change in SmarTeam



You must already have performed the task Saving Properties in SmarTeam described above.  
Open the SmarTeam: Documents dialog box displaying the profile card of the CATPart document.



1. In the SmarTeam: Documents dialog box right-click, in the tree, on the document.
2. Select **Update**.  
The Profile Card is now ready to be changed.
3. Change the material. For example, enter PVC in the **Material** field as shown:

The image shows a screenshot of the 'Profile Card' for a 'CATIA Part' document in SmarTeam. The window has a title bar with tabs: 'Profile Card', 'Links', 'Notes', 'Revision', and 'Viewer'. The 'Profile Card' tab is active. The main area is titled 'CATIA Part' in large blue font. Below the title, there are several fields: 'Document ID' with a red star icon and the value 'CATPRT-00000053' (highlighted in yellow); 'Revision' with an empty text box and a 'State' label; 'Description' with an empty text box; 'Engineering Information' section with 'Part Number' (PN-0000053-A1), 'Definition', 'Nomenclature', and 'Product description' (all empty); 'Source' (unknown); 'Material' (PVC); and 'Content Document' (empty). At the bottom, there are three tabs: 'General', 'Details', and 'Revision', with 'General' being the active tab.

Profile Card	Links	Notes	Revision	Viewer
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## CATIA Part

Document ID: ★ CATPRT-00000053

Revision:  State:

Description:

### Engineering Information

Part Number:

Definition:

Nomenclature:

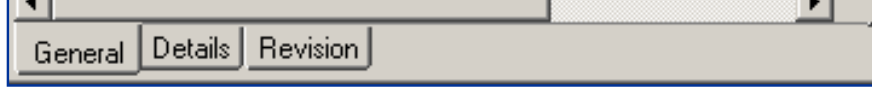
Product description:

Source:

Material:

Content Document:

General	Details	Revision
---------	---------	----------




The property has been modified in the SmarTeam database.

4. Back in CATIA, select **SmarTeam > Properties > Load from Database**.

In the CATIA session, you can now see the new Material attached to the document.

Note that when you edit a CATIA Document from SmarTeam, its properties are automatically updated to reflect the latest changes in the database.

## Reminder

To update a property value	Command to be used
from CATIA to SmarTeam	<ul style="list-style-type: none"><li>• <b>SmarTeam &gt; Save</b> (Save )</li></ul> or <ul style="list-style-type: none"><li>• <b>SmarTeam &gt; Properties &gt; Save in Database</b></li></ul>
from SmarTeam to CATIA	<b>SmarTeam &gt; Properties &gt; Load from Database</b>



# Managing Assemblies

Building assemblies can be very time-consuming, since much of your time is spent in searching for existing components and placing them in assemblies. SmarTeam - CATIA Integration provides powerful tools to assist you in building your assemblies as explained in [Using Tools for Building Assemblies](#).

This section contains the following tasks:

# Using Tools for Building Assemblies



Building assemblies can be very time-consuming, since much of your time is spent in searching for existing components and placing them in Assemblies. *SmarTeam - CATIA Integration* provides the following tools to assist you in building your assemblies:

- **Find:** Use **SmarTeam Find** to locate parts and products.
- **Copy File:** As you build your assembly, you can copy parts/products to your desktop to view how they fit together with the current assembly.
- **Light Weight Mode:** From V5R17 onwards, when loading a CATProduct document into your CATIA session, it is possible to restrict the number of associated CATPart documents to be extracted from the vault. You can extract only the CATPart documents you need, which optimizes performance and ensures that there are no broken links in the CATIA session: pointed documents are known even if they are not in the working directory. This new behavior is made possible thanks to the SmarTeam **Light Weight Mode** option that allows you to extract CATPart documents in **CGR** (CATIA Graphical Representation) format from the vault. This extraction type is to be used in visualization mode. Documents you need to load in the CATIA session, i.e. when switching to design mode, will be automatically extracted from the vault. To know more about the **Light Weight Mode**, see [Defining the Light Weight Mode for CATPart Documents](#).
- **Assembly Management options:**
  - When you wish to insert an existing part or product as a component of the current product, use [Insert Component](#).  
SmarTeam inserts the component into the current product. When you save the assembly, the components are saved as children of the assembly. In this way, SmarTeam accurately reflects the structure of your assembly.
  - When you wish to replace an existing part or product or a component of the current product, use [Replace Component](#).  
SmarTeam replaces the selected component with a SmarTeam part or product. When you save the new assembly, the previously selected components are saved as children of the assembly. In this way, SmarTeam accurately reflects the structure of your assembly.
  - Click **Replace with Selected Revision** to replace the current revision of a document with another, of your choosing.



Depending on the context, **Replace with Selected Revision** can be available for modified documents.

Be aware that the revision is replaced at **all** tree locations.

**Replace with Selected Revision** is available from contextual menus.

**Note:** The **Replace with Selected Revision** option is not available on UI Activated Documents, as these documents cannot be unloaded.

- **Save:** If you created new parts as components of the assembly, SmarTeam will save each of these parts/products into the SmarTeam database when you save the assembly. A Profile Card for each document is displayed. After these components are saved, the Profile Card for the product is displayed. In this manner, you can save the product and its components.
- **Batch Mode Save:** SmarTeam provides you with a **batch** method for saving assemblies. The **Batch Mode Save** does not display a Profile Card during a SmarTeam Save operation. Instead, each component is saved in the SmarTeam database with a unique ID number. At a later time, you can open a Profile Card for a component and enter information in the attribute fields.
- In addition, SmarTeam enables you to locate all the parents of any document using **Where Used**.



# Building an Assembly



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

In both cases, you can choose between two ways of accessing the documents of interest. A default behavior just need to have been set before using the commands.

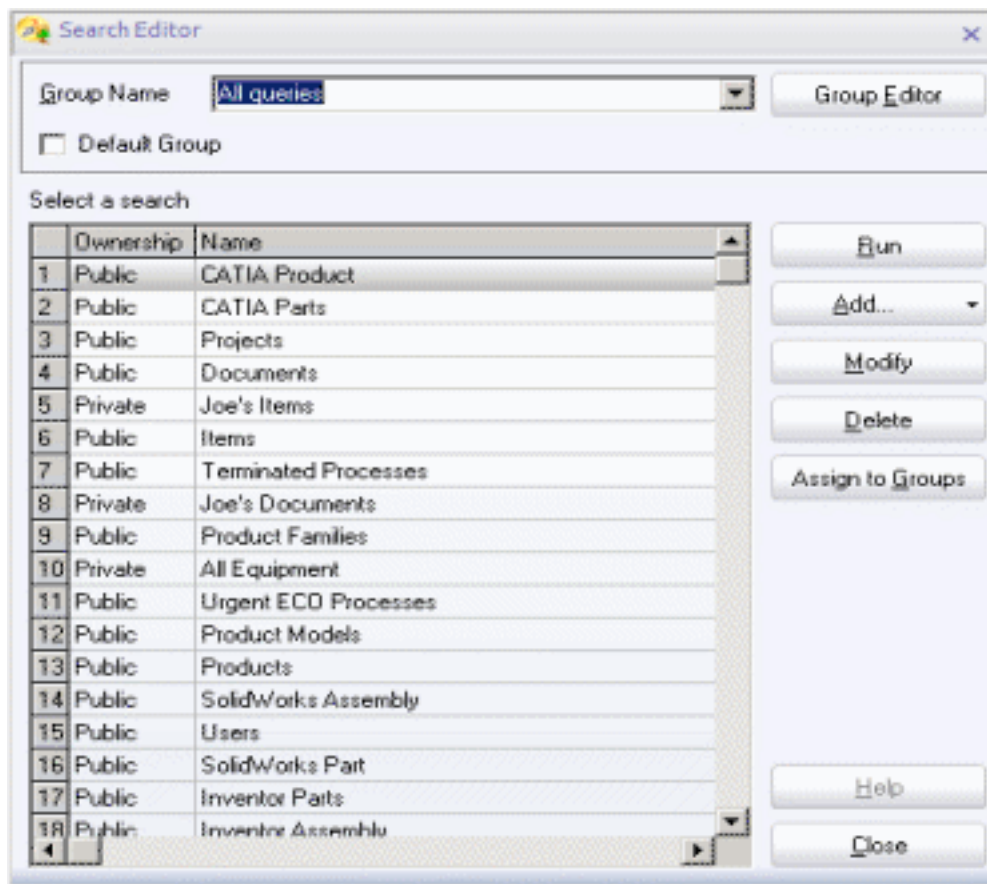


## Inserting a Component

1. Display an assembly structure in CATIA.

2. Click the **Insert Component** icon . Alternatively, right-click and select **Assembly Management > Insert Component**.


The **Search Editor** window is displayed, as shown below:



3. Choose a search.

4. Click **Modify**.
5. In the Profile Card enter relevant criteria.
6. Click **Run** to confirm the search.
7. In the displayed dialog box, choose a document.
8. Click **OK**.  
The document is inserted into the assembly structure.

## Alternate Method

From V5R17 onwards, it is possible to choose between opening the **Search Editor** dialog box as described above, or the **Insert Components** dialog box when using **Insert Component** .

Using the **Insert Components** dialog box enables you to run a quick search according to different fields: ID, Revision, Description, File Name, Directory etc. This dialog box is similar to **Open document** dialog box.

Whatever chosen method, the default behavior is customized thru the **Documents Environments** options. For more information, see [Enabling the Display of the SmarTeam File>Open User Interface](#).

## Replacing a Component

1. Perform the same steps as for component insertion described above but this time select **SmarTeam>Assembly Management>Replace Component...**  
As for inserting components, you can choose between using the **Search Editor** or **Replace Component** dialog box.



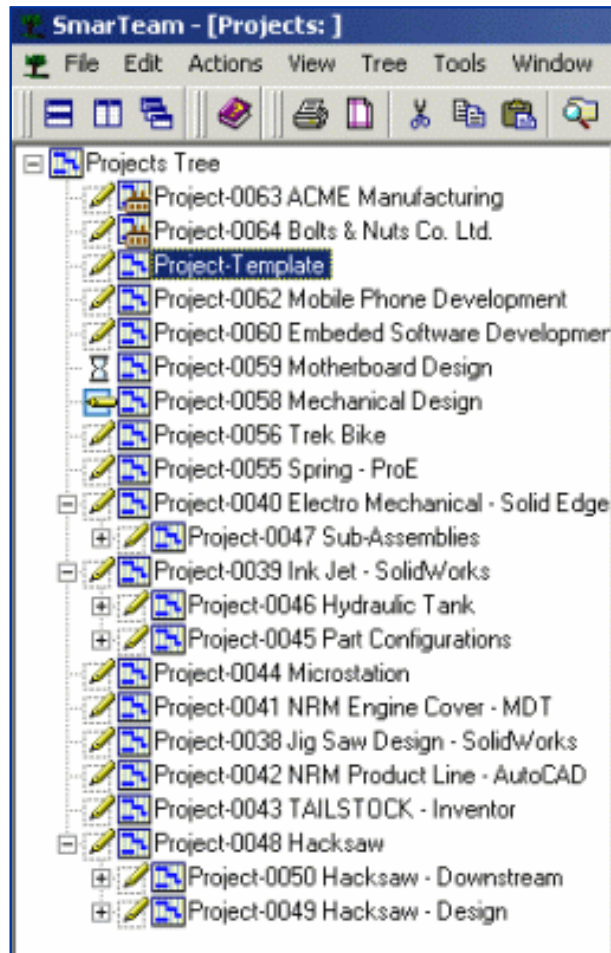
# Adding a New Assembly to SmarTeam



This task shows you how to add an assembly to the SmarTeam database. The procedure is similar to adding a part.



1. Display the assembly in CATIA.
2. Select **SmarTeam** > **Save**.
3. In the CATIA **Save As** dialog box that appears, name the file and click **Save**.
4. A SmarTeam dialog box appears.  
In this window you can define the project and the parent to which the document belongs.
5. Choose a project.  
The assembly will be saved as a document in the selected project.



6. Choose a parent folder.  
The assembly will be saved as a child of the selected folder. The CATIA file name is displayed.
7. Click **Save**.
8. A product Profile Card is displayed in which you can add relevant information. Enter a name for the Product in the **Description** attribute. By default, this attribute is displayed in the Data Manager tree along with the ID number.
9. Select the tabs to review default information about the file.
10. Click **OK**.  
SmarTeam - CATIA Integration automatically saves your product in the database, giving it a unique identity where it can be easily found for later use.  
In the SmarTeam tree hierarchy, all the components of the assembly are linked as components (children) of the product. This hierarchical link reflects the structure of the assembly as designed in CATIA.
11. If the components of the assembly have not yet been saved in SmarTeam, a Profile Card for each component is

displayed, one by one.

12. In each Profile Card, name the component (in the **Description** field).
13. Click **Add** to save the new parts/products in the SmarTeam database.  
For example, if a product has three parts as its components, and these components have not yet been saved in SmarTeam, a Profile Card for each part is displayed.



# Saving an Assembly



Every CATIA assembly should be saved into the SmarTeam database. After you create a new assembly or modify an existing one, choose one of the **Save** options. Then, check in the assembly by choosing **Lifecycle > Check In**. The assembly is then placed into the Checked In vault.

The SmarTeam integrated menu provides two methods for saving Assemblies:

- **Save:** Saves the assembly in the SmarTeam database. See [Saving a Part](#) for details.
- **Save As:** Saves the assembly into the SmarTeam database and defines the project and the parent folder of the document.  
For example, you can save the new assembly as a child of the Technical Assemblies folder in the Drive Shaft project.  
After you choose a **SmarTeam Save** option, a CATIA Product Profile Card is displayed in the Object Attributes window. After you fill in the attribute fields, click **OK**. The assembly is saved to the SmarTeam database.

**Note:** When a CATIA document is saved for the first time to SmarTeam, the Project Manager window appears, allowing you to choose the SmarTeam project and parent folder to which the document will belong. The window is displayed once only during the Save transaction for the first saved document. This functionality is controlled by the option [Save Documents > Choose SmarTeam project at first save](#).



**Although it is possible to save different document types in the SmarTeam database using the Save capabilities in the SmarTeam - Editor or SmarTeam - Web Editor , you must use CATIA SmarTeam > Save to save the different CATIA documents in the database.**

Since the process of saving a part is identical to that of saving an assembly, see the following pages for detailed instructions: To save a document for the first time, see [Saving a Part for the First Time](#). To save a document for subsequent saves, see [Saving a Part After Modifications](#).

This section discusses the following topics:

- [Saving the Components of an Assembly](#)
- [Using the Batch Mode](#)
- [Document Content Exposure](#)

# Saving the Components of an Assembly

When you save an assembly, SmarTeam - CATIA Integration automatically saves the components of the assembly as well:

- If the components of the assembly are already saved into the SmarTeam database, then the **Save** option updates the Profile Cards for all the components to reflect any modifications that were made to the components as well as the hierarchical link between the assembly and its components.
- If the components of the assembly are new and have not yet been saved to the SmarTeam database, then each of these components will be saved to SmarTeam one by one.



1. A Profile Card for each component is displayed. Fill in the Profile Cards, and click **OK**.  
After each component is saved, a Profile Card for the assembly is displayed.
2. Fill in the Profile Card.
3. Click **OK**.

**Note:** The SmarTeam - CATIA Integration saves all open documents in the CATIA session, even if the documents are not a part of the active assembly.

## Using the Batch Mode

SmarTeam enables you to save the components of an assembly **without** displaying each Profile Card. This can save you a great deal of time. There are two methods for batch saving components:



1. Select **SmarTeam > Batch Mode Save**.
2. Save the assembly using the **Save** or **Save As** option.  
SmarTeam saves all the components but does not display a Profile Card for each one.

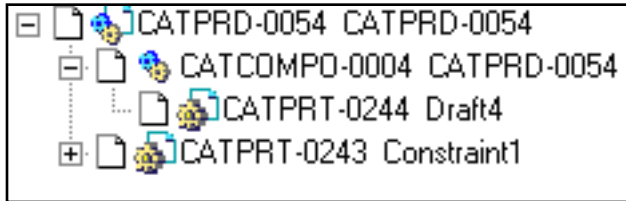
Or

1. Select **SmarTeam > Save As**.
2. In the Save As window, click the **Options** tab to display the Save Option dialog box.
3. In the **Save** tab, check the checkboxes.

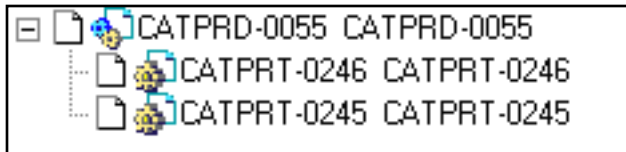
4. Click **OK**.

## Document Content Exposure

When you save CATProduct documents, the associated internal components are by default exposed in the database:




However, you can expose only the documents and their links as shown below:



Only the administrator can make the necessary modifications using the **System Configuration** tool. For more information, see the SmarTeam documentation.



# Managing the Revisions of a CATProduct Document

 As you work with an assembly, you can make continuous revisions to it. SmarTeam protects and manages these revisions in the same manner as a part.

However, an assembly is more complex than a part since it consists of many components. It is essential to maintain the integrity of an assembly by performing life cycle operations on the assembly and its children, in coordination with each other. SmarTeam ensures that the integrity of the assembly and its children is not jeopardized by disabling life cycle operations.

In addition, SmarTeam provides several lifecycle options which can help you manage your assembly. For example, you can choose:

- **Propagate Operation**  
to check in or check out the assembly and its components at the same time
- **Relatives Being Modified**  
to view the parents or children of the document that are currently being modified
- **Show Parents**  
to display the parents of a document. You can then select parents and check them out as well.

 These life cycle commands are only available when you choose **Check Out** or **Check In** from the **SmarTeam** menu. They are not available when you check out/in a document from the **CATIA** menu.

## Protecting the Integrity of an Assembly

The following examples show how SmarTeam protects the integrity of an assembly.

- When you release an assembly, its children must be Released as well. For example, if an assembly has ten subassemblies, each one must be assigned the **Released** status.
- You can only move a subassembly to the Obsolete vault if its parent assembly is also **Obsolete**.
- To perform a revision operation on an assembly and all its children simultaneously, SmarTeam provides the **Propagate Operation** option. For example, if you wish to check out a Stump Preacher Guitar and all its ten subassemblies, you can check them all out simultaneously.
- You have the option to perform a revision operation on an assembly and not


on its children, or perform a revision operation on a child and not on the parent assembly.

For example, you can check out a Stump Preacher Guitar assembly from the vault and leave the children in the vault.

- You can copy the children of an assembly to your desktop so that you can view them but not modify them. The status of the children remains **Checked In**, while the status of the assembly is **Checked Out**.
- You can check a subassembly out of the vault independently and leave the parent assembly in the vault.




# Checking In a Product for the First Time

 When a product is first saved into the SmarTeam database, it is automatically assigned the **New** status. This means that the product has not yet been checked into a SmarTeam vault.

To protect the product from modifications, place the product into the SmarTeam vault by checking it in. After the product is checked in, its status is changed to **Checked In**.

When you check in the assembly, you can check in the product and all its components simultaneously using the [Propagate Operation](#).

-  **1. Select **SmarTeam** > **Life Cycle** > **Check In**.**  
The **Check In** dialog box is then displayed.
- On the left-hand side of the window, the assembly and its components are displayed with the Checked In icon to the left of the selected document. On the right-hand side of the window, the **Check In** window is displayed.
- 2. To check in the assembly and all its components together, right-click the assembly and select **Propagate Operation**.**
- 3. Fill in the fields in the **Check In** window (optional).**  
Refer to [Checking In Dialog Box](#) for a description of these fields.
- 4. Click **OK**.**  
The assembly is now placed in the **Checked In** vault. Note that the status of the assembly is now **Checked In**.



# Checking In/Checking Out/Releasing a Product



Since the process of checking in and checking out an assembly is quite similar to the life cycle operations on a part, we will only provide a brief description of each life cycle operation. Instead, we will describe the unique features provided by SmarTeam which enable you to manage the assembly, together with its components, as you create revisions.

For each life cycle operation, you can:

- Check out/in an assembly and all its components together. For example, you can check out the Ski Draft assembly and its ten components.
- You can check out/in an assembly and handle each component individually. For example, you can check out an assembly and copy all its components to your desktop.

This section provides the following information:

- [Handling Components](#)
- [Checking Out an Assembly \(from CATIA\)](#)
- [Checking Out an Assembly from a SmarTeam Window](#)
  - [Checking out All the Components of a Product](#)
  - [Checking out Components Individually](#)
- [Checking In/ Releasing an Assembly](#)
  - [Checking in All the Components of a Product](#)
  - [Checking in Components Individually](#)


## Handling Components



SmarTeam enables you to manage an assembly and its components.

You can:

- Check out an assembly and copy all its components to the desktop.
- Perform the same life cycle operation (**Check Out**, **Check In**, **Release**) on the assembly and all its children, by choosing **Propagate Operation**, as described in [Propagate Operation](#).
- Handle each component individually. For each component, you can:
  - Check in or check out the component together with its parent assembly.
  - Copy the component to the desktop. The child remains in its current state.
  - Choose **No Operation** for the component. The **No Operation** capability enables you to maintain a subassembly in its present state while performing a revision operation on the assembly (or vice versa).

For example: You wish to insert a new part into an assembly to replace an existing part. You can check out the assembly, and copy all the parts to your desktop, except the part you wish to replace. For that part, choose the **No Operation** icon .

When you click **OK**, the assembly will be checked out of the vault, and all the children except one will be copied to your desktop. In SmarTeam, you can replace the old part with a new part at your desktop to see how it affects the assembly as a whole.

## Checking Out an Assembly (from CATIA)



If an assembly is displayed at your desktop in read-only mode, you can choose to check it out in order to modify it. SmarTeam enables you to do so.

1. Display the assembly in CATIA (in read-only mode).
2. To check out a document: Select a document (assembly or part), in **SmarTeam** menu, select **Life-Cycle >**

#### Check Out .

3. In the displayed **Check Out** window, fill in the fields.  
**Check Out** windows display SmarTeam information as set in the **Tree Properties** dialog box. For more information, see [Customizing SmarTeam Document Display Information](#).
4. Click **OK**.

At your desktop, you can view the assembly and its components. Those documents that were checked out may be modified. After you modify the document, remember to apply the **SmarTeam > Save** command.

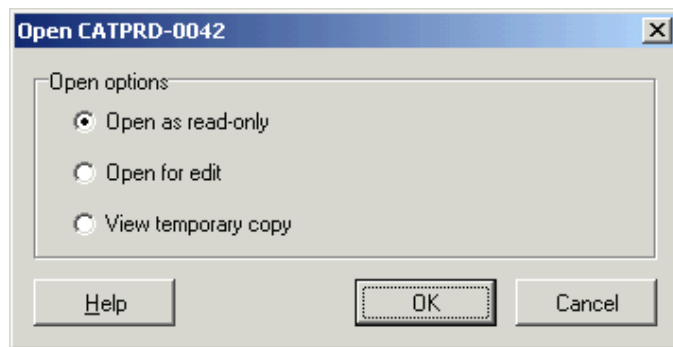
## Checking Out an Assembly from a SmarTeam Dialog Box



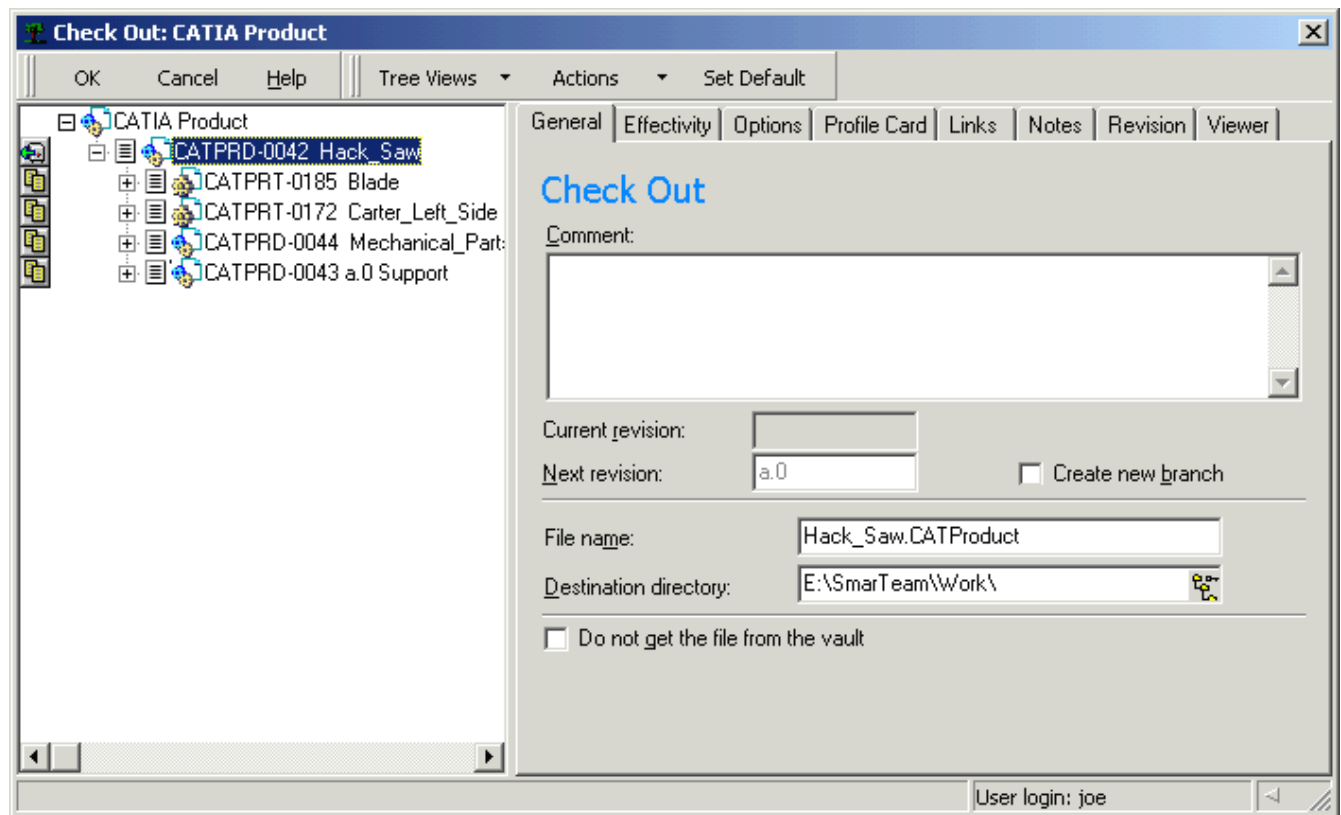
If an assembly is checked in to a vault, and it is not currently displayed at your desktop (in read-only mode), you must locate the assembly and then select **File Operation > Open For...** contextual command to launch it into CATIA and modify it.

In the displayed **Check Out** window, you can right-click to display a dropdown menu which provides several life cycle options. These life cycle options can assist you in managing your assembly. See [Life Cycle Options](#) for details.

1. Run a search to locate the assembly that you wish to check out.
2. Right-click the assembly and select **File Operation > Open For...**  
The **Open** dialog box is displayed. By default, the **Open as read-only** option is selected.



3. Check the **Open for edit** option.
4. Click **OK**.  
The **Check Out: CATIA Product** dialog box appears.  
On the left-hand side of the window, the **Check Out** icon appears next to the assembly and the **Copy File** icon appears next to the components.



 The default operation for the components (**Check Out** or **Copy File**) is determined by the administrator.

## Checking out All the Components of a Product

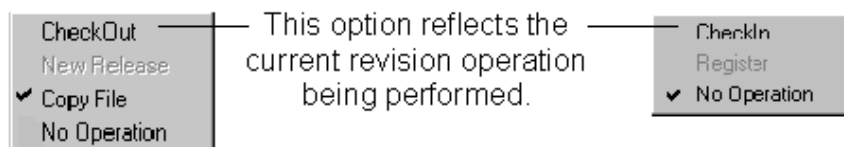
1. To check out not only an assembly (here CATPRD-0042 Hack\_Saw) but also all its components, you need to use the **Propagate Operation** contextual command. This checks out all the documents simultaneously.
2. For more information, see [Life Cycle Options](#).

## Checking out Components Individually

1. To check out an assembly and some of its components, you can define an operation for each component as follows:

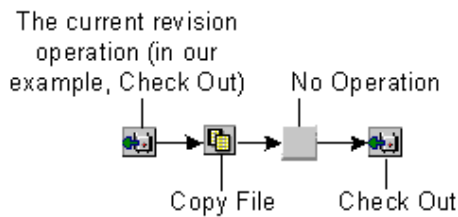
Either:

- Right-click on the icon of a document to display a list of options. This list differs for each life cycle operation, as shown below:



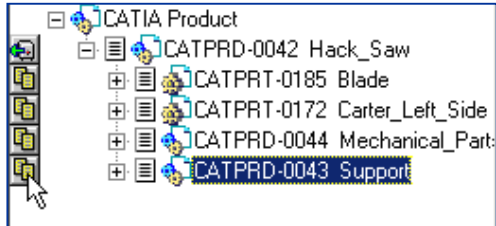
Or, more simply,


- Click on the icon of the document to choose an operation. The icon toggles between three states:



## Example

In the following example, double-click the icon  in front of CATPRD-0043 a.0 Support if you wish to check it out:



Once done, the Check out icon  indicates that the document is selected for being checked out.

2. Fill in the fields in the displayed revision window, and click **OK**.

## Checking In/Releasing an Assembly



If it is the first time you are checking in the part see [Checking In a Product for the First Time](#).

SmarTeam provides two methods for checking in or releasing an assembly:

- Check in the assembly and its components from any SmarTeam window. You must select each document one by one and check it in (or Release it).  
This method is identical to the checking in a part as described in [Checking In a Part](#). Below is a quick summary.
  1. Select the part or assembly and choose **SmarTeam > Life Cycle > Check In or Release**.  
The Check In (Release) window is displayed, as shown in [Checking In a Part](#).
  2. Fill in the fields and click **OK**.  
The status of the document is automatically changed to **Checked In**. The assembly is now checked into the vault for safekeeping. A copy of the file remains at your desktop in read-only mode.
- Check in the assembly using the **Life Cycle > Check In** (or **Life Cycle > Release**) capability (from the **SmarTeam** menu). Instructions are provided on the following page.  
When you choose the **Life Cycle > Check In** (or **Release**) option from the **SmarTeam** menu, a Check In (Release) window is displayed. In this window you can check in the assembly and its components at the same time. In addition, you can view life cycle options which help you manage your assembly, as described in [Life Cycle Options](#).

**Note:** The Check In operation requires saving all components that are open in CATIA. If there is a component you do not want to save, you must close it before the Check In operation.



For information on the **Release** operation, refer to [Releasing CATIA Parts, Drawings and Assemblies](#).



## To check in an Assembly

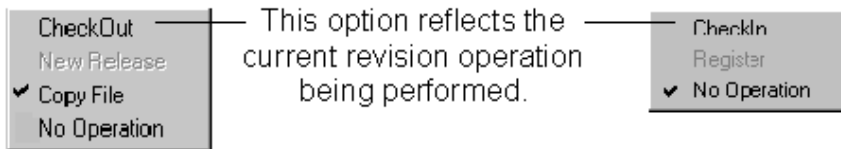
1. From the **SmarTeam** menu, point to **Life Cycle** and choose **Check In**.  
The Check In window is displayed. On the left side of the window, the assembly and its components are displayed.
2. You can either:
  - check in all the documents together:
  - handle each component individually:

## Checking in All the Components of a Product

3. To check in all the documents together, right-click and choose **Propagate Operation**.  
The **Check In** icon is displayed next to each component in the tree.
4. Fill in the fields in the **Check In** window (optional).
5. Click **OK**.  
The assembly and its components are checked in to the vault. A copy of these documents does not remain at your desktop.

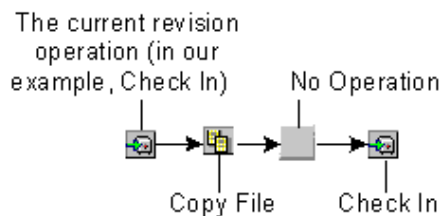
## Checking in Components Individually

1. To handle each component individually:  
Either:
  - Click on the icon of a document to display a list of options.  
This list differs for each life cycle operation, as shown below:



Or, more simply,

- Click on the icon of the document to choose an operation.  
The icon toggles between three states:



2. Complete the fields in the displayed window.
3. Click **OK**.



# Lifecycle Options



Your Assemblies are often complex with numerous subassemblies. As you manage the revisions of the Assembly, you must keep track of the status of the revisions of the parents and children of each Assembly. SmarTeam provides several life cycle options that can assist you in this task.



1. Select **SmarTeam>Life Cycle** and choose a life cycle operation (**Check Out** for example).
2. In the displayed dialog box, right-click on a document to display a dropdown menu.  
A list of lifecycle options is displayed. This list differs according to the lifecycle window being displayed.

Lifecycle menus include the following lifecycle options:

Command	Description	Example
<b>Associated Objects</b>	Enables you to view those documents that are linked to the selected document. You can then perform lifecycle operations on these Associated Objects. See <a href="#">Managing the Revisions of a Drawing</a> for a complete description.	In SmarTeam, a Drawing ( <b>Stump Preacher Guitar Drawing</b> ) was created based on a <b>Stump Preacher Guitar Drawing</b> . When you check out the <b>Stump Preacher Guitar Drawing</b> you can view and manage its link (the <b>Stump Preacher Guitar Drawing</b> ).
<a href="#">Propagate Operation</a>	Performs the same life cycle operation on the Assembly and all its children.	When you check in an Assembly, you can check in all the children simultaneously.
<a href="#">Relatives Checked Out</a>	Displays the parents or children of a document that are currently Checked Out.	When you manage an Assembly, you can view a list of its parents or children that are currently being modified (checked out) in the <i>Relatives Being Modified</i> window. This is particularly helpful when you wish to release an Assembly, since all its documents must first be checked in.

Show Parents	Displays the parents of a document in a separate window. You can then select the parents that you wish to check out as well.	When you check out a Part that is a child of a few different Assemblies, the <i>Show Parent</i> window displays the parents of the selected document and their revision status. You can then choose the Assemblies that you wish to check out together with the Part.
Switch to Latest Revision	Enables you to check out the latest revision of a document.	When you check out a Part that has numerous revisions, the <b>Switch to Latest Revision</b> option automatically checks out the most recent revision.
Replace Revision	Enables you to choose a different revision of a document and perform a life cycle operation on it. This option also enables you to link the latest revision of the subassemblies to the Assembly being checked out.	When you check out a Part that has three revisions, you can replace the displayed revision with a different one.  Also, when you check out an Assembly, you can link it to the latest public revision of each of its subassemblies.



## Propagate Operation

Performs the same life cycle operation on the Assembly and all its children.

1. In any lifecycle window, right-click an Assembly and select **Propagate Operation**.  
The icons of all the children change to reflect the revision operation to be performed.
2. Complete the fields in the revision window (optional).
3. Click **Apply** to perform the operation and remain in the view, or click **OK** to perform the operation and exit the view.

## Relatives Being Modified

Displays the parents or children of a document that are currently Checked Out.

1. From the Check Out/New Release dialog box, right-click a document (not its icon) and select **Relatives Being Modified**.  
The relatives (parents or children) that are currently in the Checked Out state are displayed:



The  icon represents a parent.

The  icon represents a child.

## Show Parents

Displays the parents of a document in a separate window. You can then select the parents that you wish to check out as well.

1. From any Check Out/New Release dialog box, right-click on a document (not its icon) and select **Show Parents**.

Note that using the Shift Key lets you select multiple parents.

The Select Parents dialog box displays the parents of the selected document.

2. To check out the parents, select the parents in the Select Parents dialog box.
3. Click **OK**.

The parents are added to the list in the Check Out dialog box and will be checked out together with the source document.

## Switch to Latest Revision

Checks out the latest revision of a document.

1. In the Check Out/New Release dialog box, select **Switch to the Latest Revision**.

A dialog box opens displaying the newly created revision.





## Replace Revision

Enables you to choose a different revision of a document and perform a life cycle operation on it. Enables you to link the latest revision of the subassemblies to the Assembly being checked out.

A document can have several revisions as it is checked in and checked out of the vault. If you are currently performing a life cycle operation, such as **Check Out**, on one of the revisions, you can check out a different revision of the document in its place.

During the development of an Assembly, its children may undergo several revisions. When you check out the Assembly, you can check out the latest revision of its children.

The **Replace Revision** option enables you to replace the following:



- Replace the document selected for the **Check Out/New Release** operation, using the **Replace Revision/Select** command. For example, if you selected to check out the Part phone b.1, and then choose **Replace Revision/Select**, you can check out a different revision of the Part, such as b.2 or b.3.
  - Replace the children of the Assembly being checked out, using the **Replace Revision/Revert to Last** option. For example, if you selected to check out the Stump Preacher Guitar Assembly, and then choose **Replace Revision/Revert to Last**, all the children of the Assembly switch to the last public revision. The newly created revision of the Assembly will be linked to the last public revision of the children. This option is particularly helpful when you wish to work with the latest revision of all the children of an Assembly.
1. From the Check Out/New Release dialog box, right-click on a document (not its icon) to display a dropdown menu and choose **Replace Revision**.
  2. To replace the document selected for the current **Check Out/New Release** operation: Choose **Select** to display a list of revisions.
  3. Choose a revision and click **OK**.  
In the Check Out window, the selected revision will replace the original revision.  
or  
Choose **Revert to Last** to check out the last public revision of the children of the Assembly currently being checked out. The newly created revision of the Assembly will be linked to the latest public revision of all its

children.




# Managing Drawings

SMARTEAM provides powerful tools to assist you in creating, saving and managing your drawings.

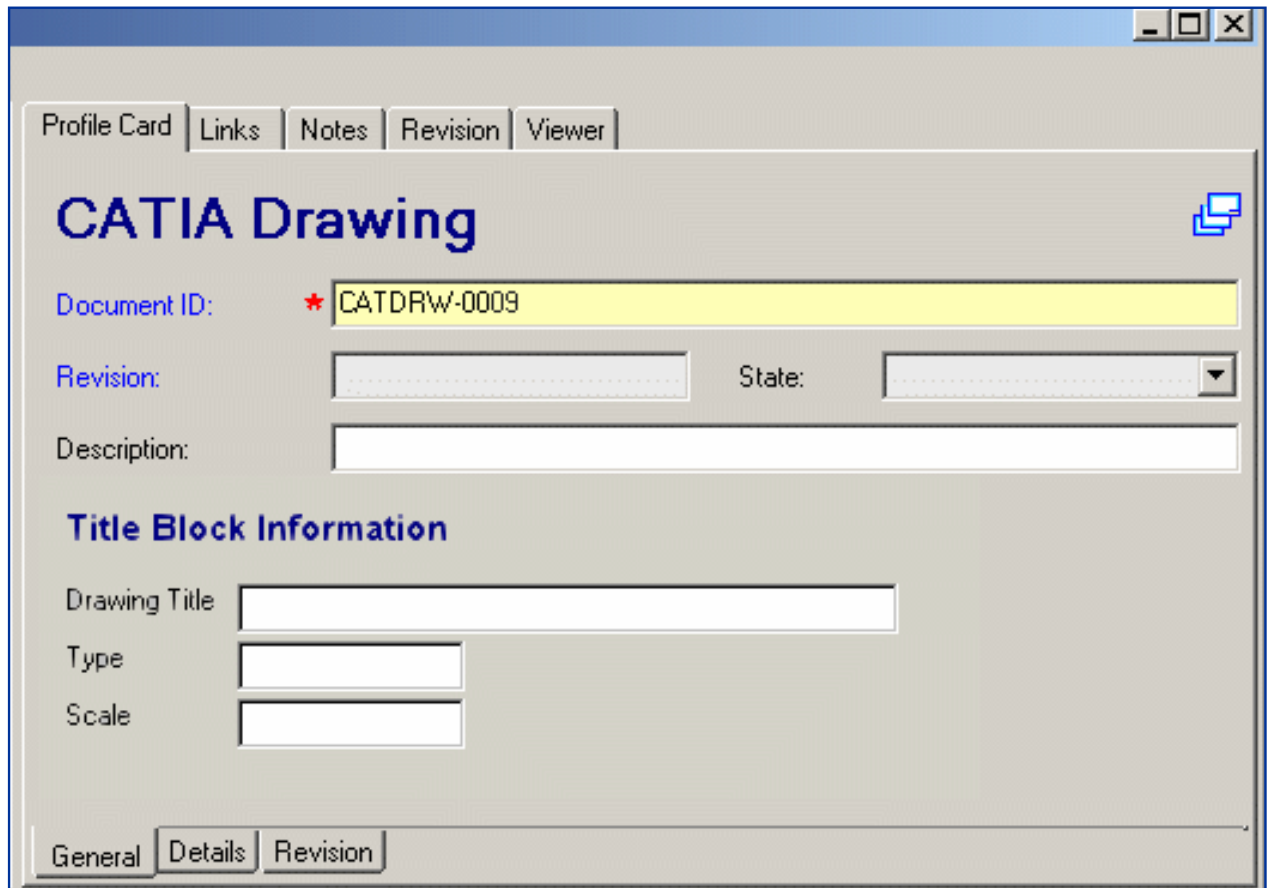
When you save the drawing into the SMARTEAM database, a general link is automatically created between the drawing and the part/product. This enables you to manage the drawing together with the part/product as you create revisions. In this manner, the status of the drawing remains parallel with the status of the document on which it was based and SMARTEAM protects the integrity of the drawing.

This section contains the following tasks:

# Saving a Drawing

 When you save the Drawing, a general link is created between the drawing and the part/assembly on which it was based:

1. From the **SmarTeam** menu, choose **Save** (or **Save As** ).  
A CATIA Drawing Profile Card is displayed.



The image shows a 'CATIA Drawing' profile card window. It has a title bar with standard window controls. Below the title bar is a tabbed interface with tabs for 'Profile Card', 'Links', 'Notes', 'Revision', and 'Viewer'. The 'Profile Card' tab is active. The main area is titled 'CATIA Drawing' in large blue font. Below this, there are several input fields: 'Document ID' with a red star icon and the value 'CATDRW-0009' highlighted in yellow; 'Revision' and 'State' fields; and a 'Description' field. Below these is a section titled 'Title Block Information' containing 'Drawing Title', 'Type', and 'Scale' fields. At the bottom, there are three tabs: 'General', 'Details', and 'Revision'.

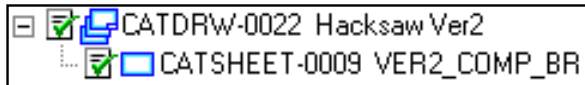
2. Fill in the fields in the Profile Card.
3. Click **OK** .  
The drawing is saved to the SmarTeam database, and a general link is created linking the Drawing to the part/assembly on which it was based.  
In this way, SmarTeam reflects the nature of the Drawing. As you manage the revisions of the part/assembly, you can manage the revisions of the Drawing in parallel.  
It is now possible to see the different sheets of a CATDrawing document in the SmarTeam viewer. For more information, see [Enabling Multi-Sheet CATDrawing Documents Preview](#).



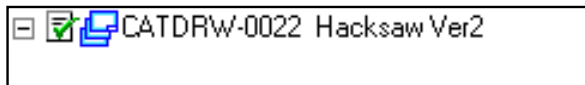
See [Saving a Part for the First Time](#) for detailed instructions on using **Save** or [Saving a Part After Modifications](#) for detailed instructions on using **Save As** .

## Document Content Exposure

When you save CATDrawing documents, the associated sheets are by default exposed in the database:



However, you can expose only the documents and their links as shown below:



Only the administrator can make the necessary modifications via the smarteam.std.legacyPreferences.config.xml file. For more information, see the SmarTeam documentation.



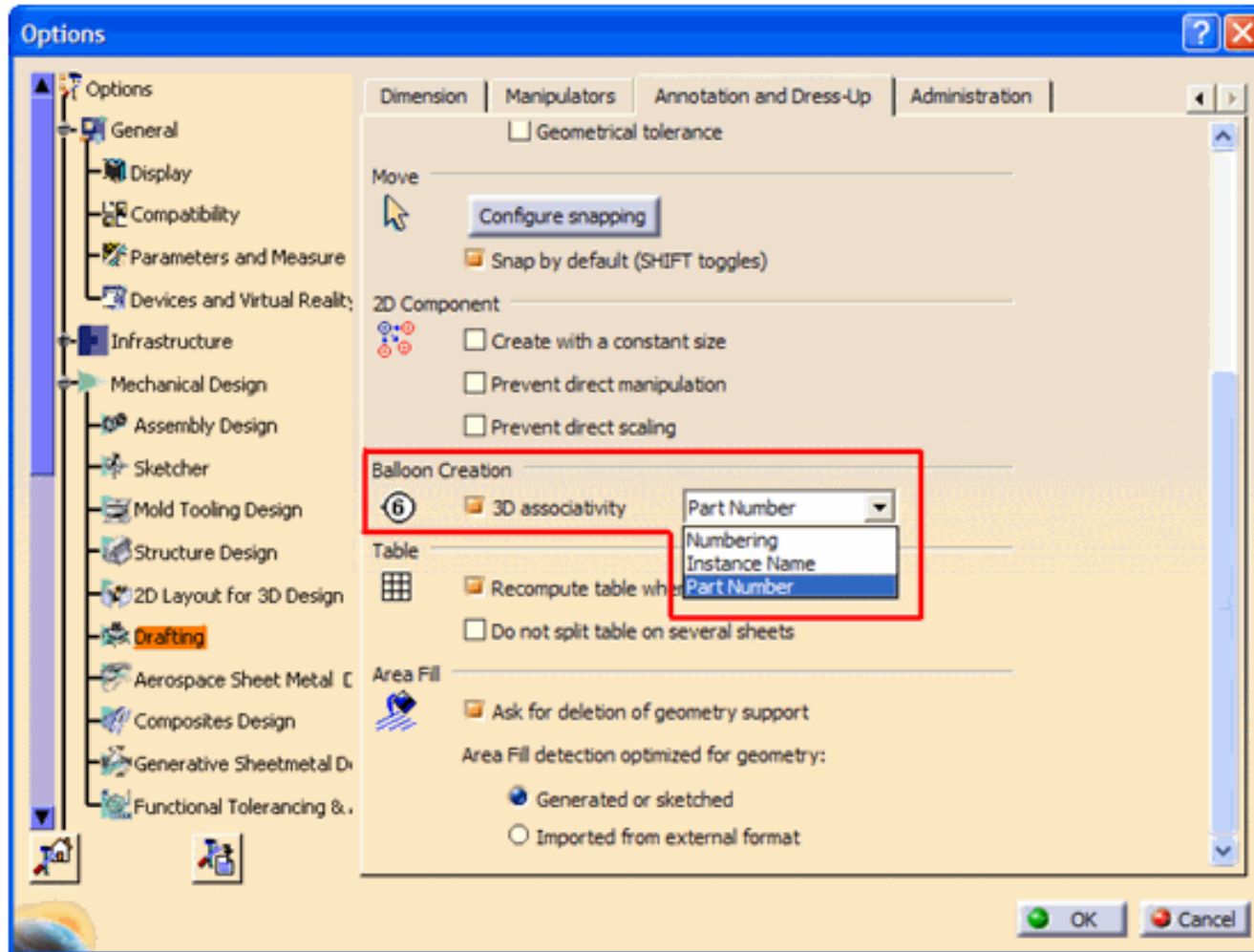
# Preventing the Saving of External Links



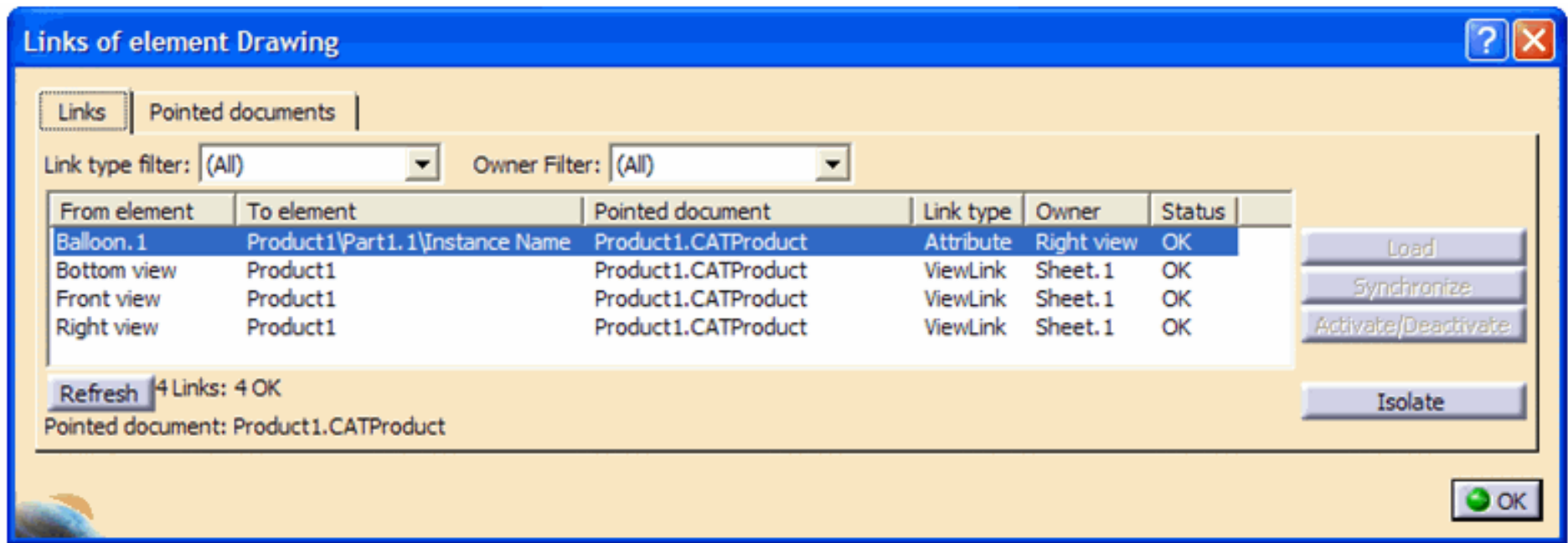
When the user creates a Drawing of a Product, generates balloons and saves the drawing to SmarTeam, the CATIA integration creates links of each Part of the Product that is also linked to the Drawing.

SDE normally uses the balloon feature for drafting interactivity. Therefore, when a SmarTeam Save is performed, many links are generated between the actual Part and the Drawing that is saved in SmarTeam. However, according to the SDE LifeCycle Rule Setup, when the Part is checked out, the drawing is also Checked Out, which may not be necessary. The functionality described below explains how to avoid saving this link in SmarTeam.

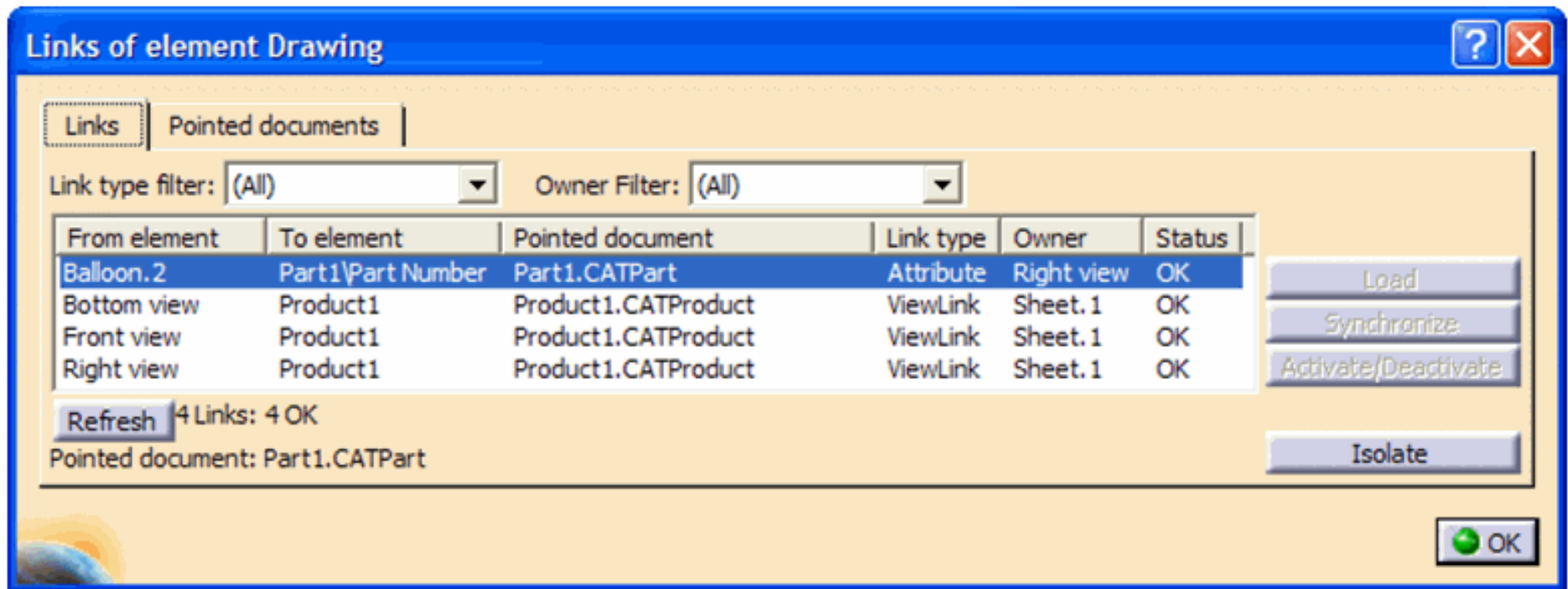
Balloons can be created on the drawing using the Balloon command in the Annotations Toolbar in Drafting.



When 3D associativity is set to Instance Name or Part Number, an external link of type **Attribute** is generated between the Part and Drawing as shown in the example below:



When **3D associativity** is set to **Numbering** no additional link is created between the Part and the Drawing as shown in the example below:



To implement this functionality, the system configuration key **CATIA.CreateDrawingAttributeLinks** should be set accordingly:

Name of Key: **CATIA.CreateDrawingAttributeLinks**

Description: Control the saving of an Attribute link on a drawing document to be saved in SmarTeam.


Possible Values: **True/False**

**True** = Save Attribute Links on a Drawing document to SmarTeam (default)

**False** = Do not save Attribute Links on Drawing document to SmarTeam



# Managing the Revisions of a Drawing

 SmarTeam manages the special relationship between a drawing and the part/assembly on which it was based. When you create a drawing by inserting a part/assembly and you then save it using **SmarTeam > Save**, a link is created between these two documents in SmarTeam. These linked documents are called **associated objects**.

When managing drawings, SmarTeam lets you expose two types of links:

- **CATIA Downstream Application**
- **CATIA Downstream Application-Reverse**

As you create revisions, you can view and manage these associated objects using **Associated Objects > CATIA Links**. This command provides options corresponding to the different possible links.

In addition, SmarTeam color indicates each of the associated objects, so that you can clearly identify a document's links and reverse links.


- Links are displayed in **red (default color)**.
- Reverse links are displayed in **blue (default color)**.

The default color settings can be changed, as explained in [Customizing color settings for associated objects](#).

## Working with Links and Reverse Links

Each time you perform a life cycle operation on a document, you can view its associated objects, meaning its links and reverse links. You can then manage the revisions of these associated objects by checking the associated objects in or out of the vault or copying their files to your desktop.

When you check out (or check in) a Drawing, its associated document (the part/assembly) is automatically checked out together with the Drawing.

 If the administrator sets the default option to **Copy File**, then the links will be copied to the desktop, not checked out.

## Viewing Associated Objects

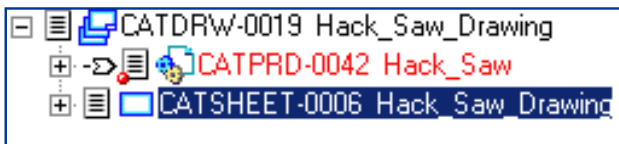
You can view the links of a document and choose a lifecycle operation for each associated document. If you do not view the links of a drawing, the associated documents will automatically be checked out/in together with the drawing (if that is the default setting).

You can view the reverse links of a document and choose a lifecycle operation for them. If you do not view the reverse links of a document, then no lifecycle operation is performed on them.

For example: A Stump Preacher Explode A4 Drawing was created based on a Stump Preacher Guitar assembly . When you check out the Stump Preacher Explode A4 Drawing , its associated document, the Stump Preacher Guitar Assembly will be checked out with it (unless you display the linked document and choose a different lifecycle operation). When you check out the Stump Preacher Guitar Assembly , you may view and check out its reverse link, the Stump Preacher Explode A4 Drawing .

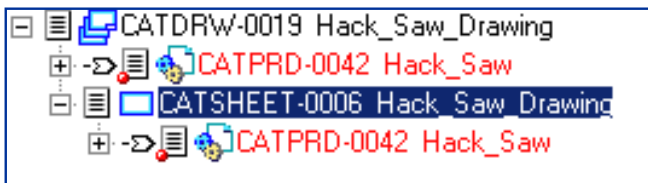


1. From any life cycle window (such as **Check In** or **Check Out**), right-click a drawing document and select **Associated Objects > CATIA Links** to display the list of view options which are as follows:



<All>  
CATIA Reference  
CATIA Reference - Reverse  
CATIA Design  
CATIA Design - Reverse  
CATIA Downstream Application  
CATIA Downstream Application - Reverse  
CATIA Design Table  
CATIA Design Table - Reverse

2. For example, select **CATIA Downstream Application**.  
The product to which our drawing is linked is now displayed in red:



## To manage associated objects:

When the associated objects are displayed in the tree browser, you can manage them in the same manner as any assembly and its children. You can:

- copy the associated objects to the desktop, as shown above.
- Check Out / Check In all the associated objects by choosing the **Propagate Operation** option.
- handle each associated object individually. For each associated object, you can:
  - Check out or check in the document.
  - Copy its file to the desktop. The state of the document is not changed.
  - Choose **No Operation** for the document.

Life cycle operations are always performed on linked documents, even if you do not display them. This is not the case for reverse linked documents: You must choose to display reverse links in order to perform lifecycle operations on them.



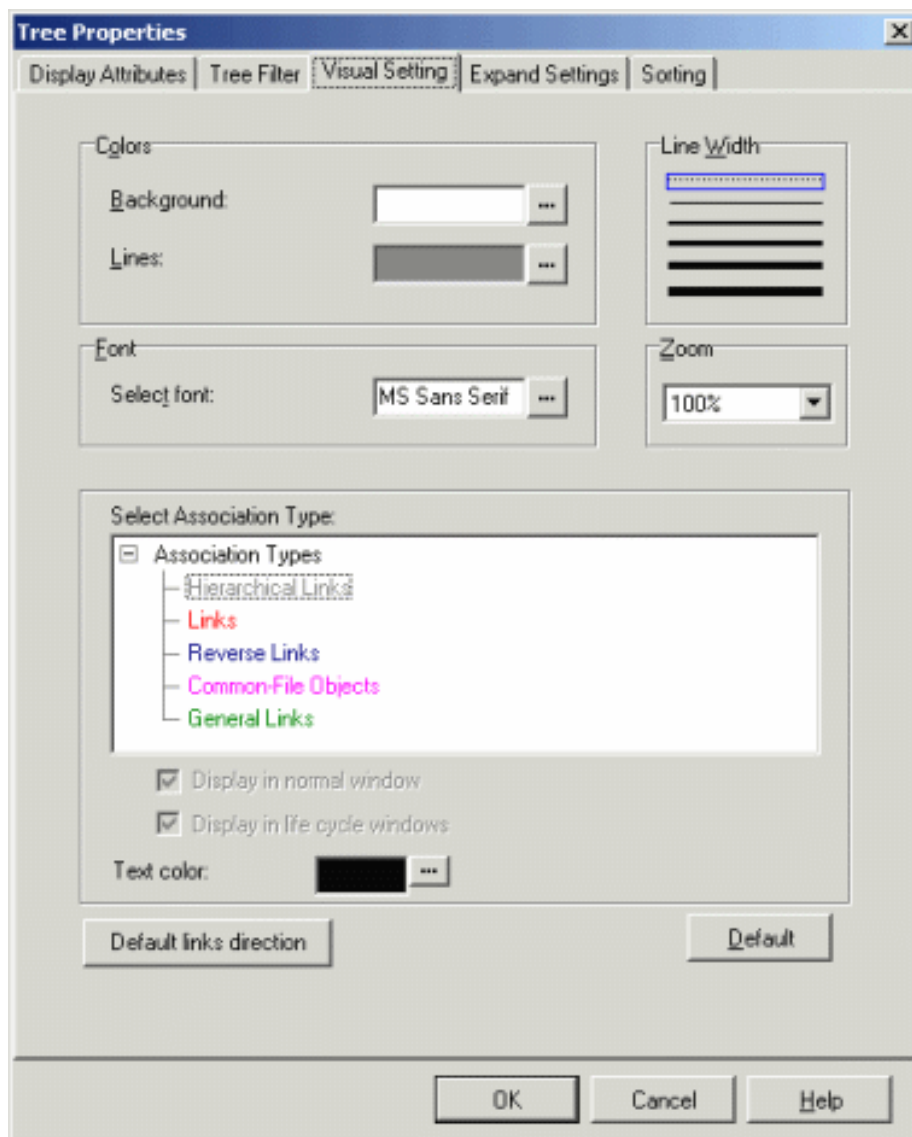
## Customizing Color Settings for Associated Objects

SmarTeam color indicates each of the associated objects, so that you can clearly view a document's links and reverse links.

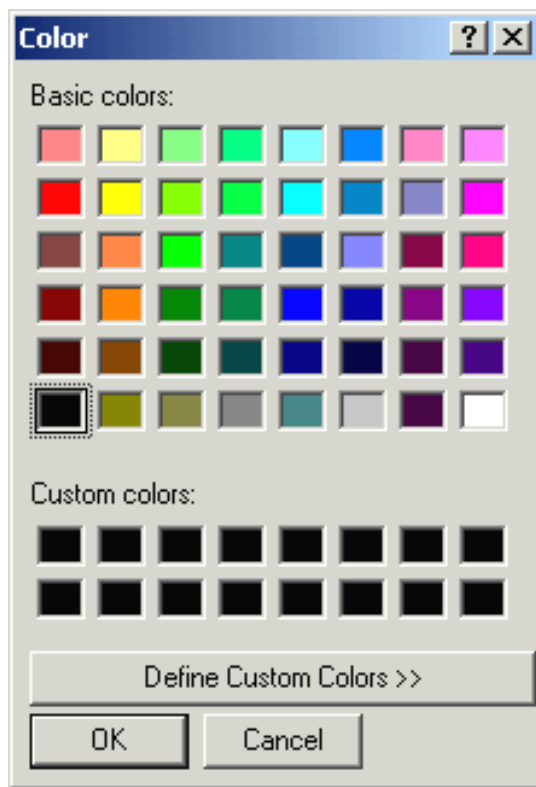
- Links are displayed in **red**.
- Reverse links are displayed in **blue**.

To change these default settings:

1. In any life cycle window, right-click to display a dropdown menu and choose **Tree Properties**.
2. Click the **Visual Setting** tab to display the following dialog box:



3. Click the button to the right of the color that you wish to change. A color selection dialog box is displayed, as shown below:



4. Choose a new color.
5. Click **OK** to confirm and close the dialog box.
6. In the Tree Properties dialog box, click **OK**.



# Document Associations and Links



This page deals with the following subjects:

- [Links/Reverse Links](#)
- [Revising Associated Objects](#)
- [Drawing & Link Exposure in Expose Mode](#)

## Links/Reverse Links

CATIA V5 enables you to create a drawing based on a part or a product. When you save the drawing into the SmarTeam database, a general link is automatically created between the drawing and the part/product on which it was based. This enables you to manage the drawing together with the part/product as you create revisions.

The drawing and its associated part/product are called *Associated Objects*. SmarTeam creates the following relationship between the two documents:

- **Link:** The part/product is linked to the drawing.
- **Reverse Link:** The drawing is a reverse link of the part/product.

For example: You create a drawing named Circuit drawing based on a part named Metallic Circuit. SmarTeam creates a link between these two documents as follows:

- The Metallic Circuit part is a link of the Circuit drawing(since the drawing is dependent on the part).
- The Circuit drawing is a Reverse link of the Metallic Circuit part.

For more information about CATIA links, see [Enriched Decision Support with All V5 Links](#).

## Revising Associated Objects



As you revise your documents, SmarTeam protects the relationship between *associated objects*:  
A drawing and its associated part/product.

When you perform a lifecycle operation on a document (such as **Check In** or **Check Out**), you can display and manage the document's associated objects. Each associated object is color-coded for easy recognition.

### To display associated objects:

1. From any life cycle window, right-click to display a dropdown menu.
2. Point to **Associated Objects** and choose the type of object you wish to display.

Each associated object is color-coded as follows:

- Links are displayed in **red**.
- Reverse links are displayed in **blue**.

SmarTeam protects the relationship between these associated objects as you make revisions:

- Each time you check out a drawing from a vault in order to revise it, its dependency (part/product) is automatically checked out together with it.
- When you check out a part/product from a vault in order to revise it, you have the option of viewing and checking out its associated drawing.

Before you can work with associated documents (dependencies, reverse dependencies), an administrator must enable them. The administrator can define dependencies based on a customized class structure.

## Drawing & Link Exposure in Expose Mode



You can customize the CATIA-SmarTeam integration product to expose the drawing sheets ("expose mode").

In the case of the expose mode, you may choose to expose only the links between the sheets to the external documents and not to expose the links of the drawing document to the external documents.

Here are the options required to customize the drawing link exposition.

- **Links exposed from both Drawing and sheet to the referenced documents**

In this case, the sheet will expose its direct dependency links and the drawing will expose all the sheet dependency links. This is the default behavior.

In case of error, the user should check that the variable `CATIA_TEAM_PDM_DRLINK` is not set.

- **Links exposed only from the Drawing to the referenced documents**

In this case, the sheet does not expose any dependency links.

The variable `CATIA_TEAM_PDM_DRLINK` should be set to *DrOnly*

- **Links exposed only from the Sheet to the referenced documents**

In this case, the drawing does not expose any dependency links. The variable `CATIA_TEAM_PDM_DRLINK` should be set to *ShOnly*



# Designing a Title Block



The scenario described below shows you how to design a title block using properties defined in the database.

This task is divided into the following stages:

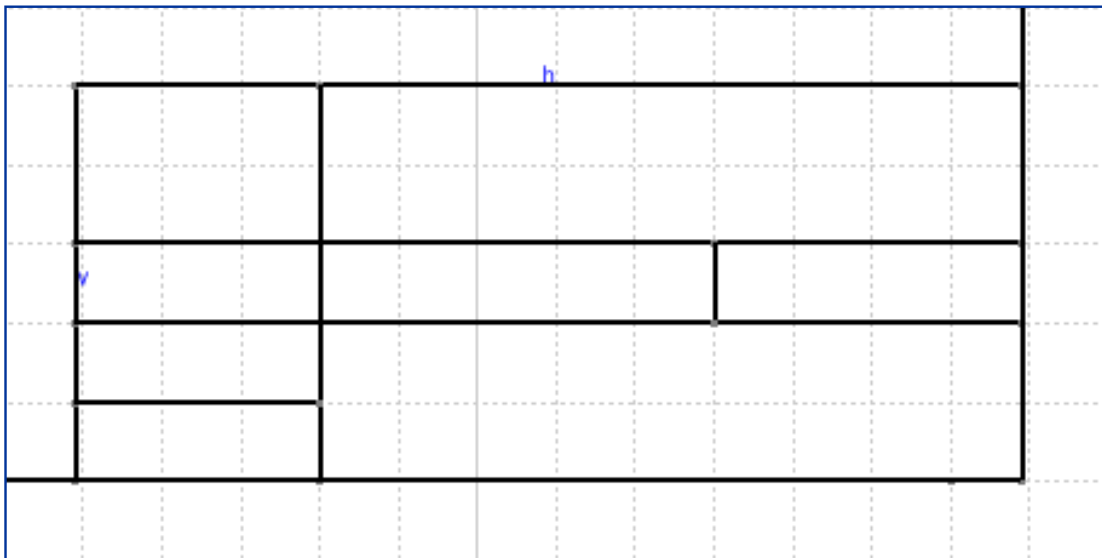
- [Creating the Title Block on a New Drawing](#)
- [Defining the CATIA Formula Property for Each Attribute from the Database](#)
- [Creating the Text Attributes in the Title Block](#)
- [Saving the Document in SmarTeam](#)
- [Updating a CATIA Property in SmarTeam](#)

The properties used for these tasks have already been mapped by the administrator between CATIA and SmarTeam (see [Defining Property Mapping for the Title Block](#)).

## Creating the Title Block on a New Drawing



1. Create a new CATDrawing document.
2. In the New Drawing dialog box that appears, make sure the default values (ISO in the *Standard* field and A0ISO in the **Format** field) are specified and click **OK**.
3. Design your title block as shown below:




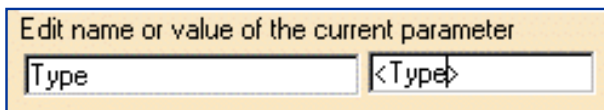
For more information about the title block, see "Sheet" in the *CATIA - Generative Drafting User's Guide*.

## Defining the CATIA Formula Property for Each Attribute from the Database






1. In the CATIA session, click on the **Formula**  icon.  
The **Formulas: Drawing** dialog box appears.
2. Select **String** in the **New Parameter of type** list.
3. Click the **New parameter of type** button.
4. In the field on the left specifying the name of the current parameter, change the name to **Type**.  
Remember that the mapping property name has already been defined in [Defining Property Mapping](#).
5. In the field on the right specifying the value of the current parameter, enter **<Type>**.  
Remember that this reflects the value of the attribute in the database.

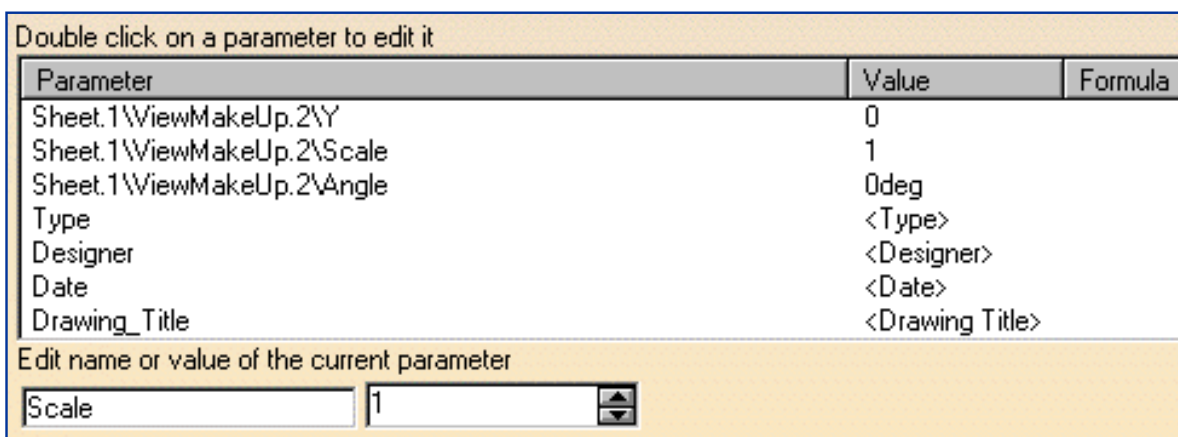


6. Click on the **Apply** button.
7. Repeat steps 2 through 6 for all mapped properties.

Here is a list of the properties defined in the title block and that are still to be mapped:

Property Name in CATIA 	Property Value	Property Type
Type	<Type>	String
Designer	<Designer>	String
Date	<Date>	String
Drawing_Title	<Drawing Title>	String

8. Locate the Scale property i.e. *Sheet1\ViewMakeUp.1\Scale* and rename it to **Scale** as shown below:




9. Click **OK**.



## Creating the Text Attributes in the Title Block



Here is what you will obtain at the end of this task:

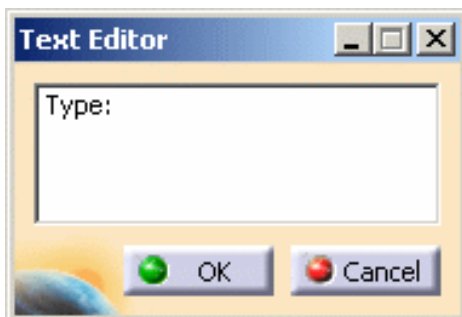
		Dassault Systemes 9, Quai Marcel Dassault BP 310 92156 Suresnes cedex FRANCE
Echelle : 1	Design by : <Designer>	Date : <Date>
A0	Drawing Title <Drawing Title>	
Type : <Type>		



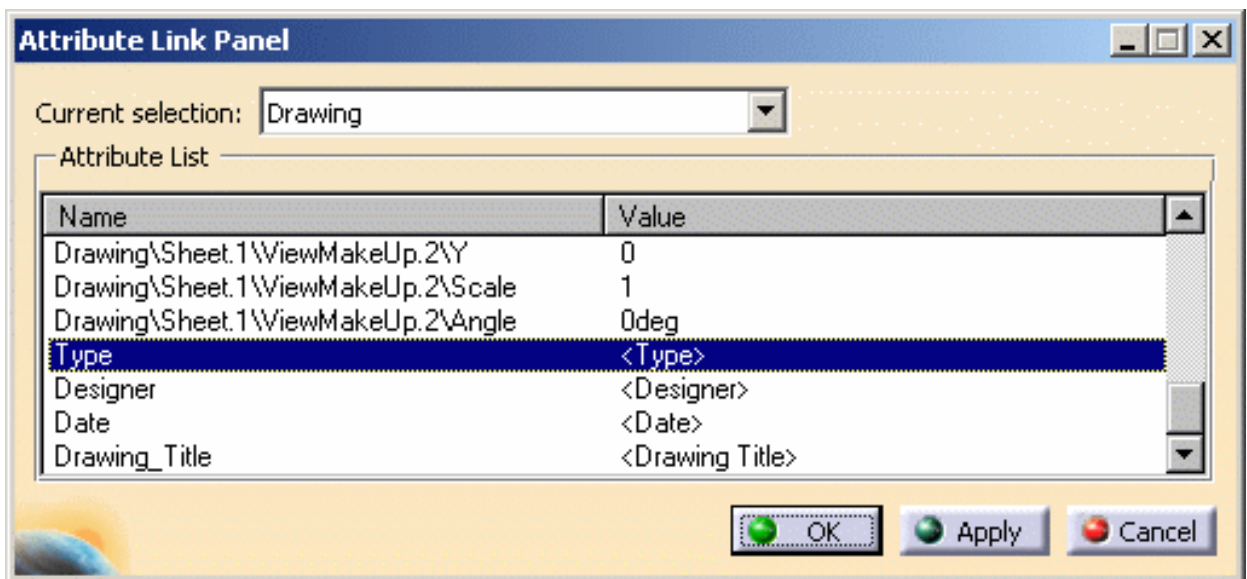
For more information see "Adding Attribute Link to Text" in the *CATIA - Interactive Drafting User's Guide*.



1. Select **Insert > Annotations > Texts > Text** .
2. Click inside the drawing to indicate where you want to display the drawing type. (If necessary, refer to the title block design.)  
A text editor window appears.
3. Enter **Type** in this window:

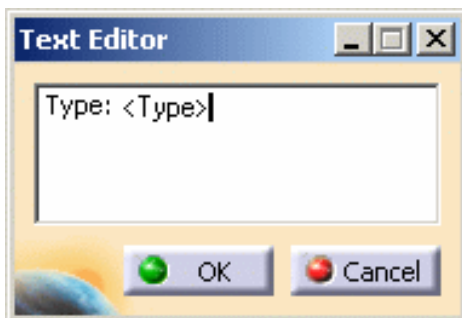


4. Right-click on the appropriate location in the title block and select **Attribute Links**.
5. In the specification tree of the drawing, click on the **Drawing**.  
The Attribute Link dialog box appears.
6. Select **Type** in the attribute list:



7. Click **OK**.

The value of the property i.e. **<Type>** is now displayed in the Text Editor dialog box:




8. Click **OK**.
9. Repeat steps 1 through 8 for all texts.

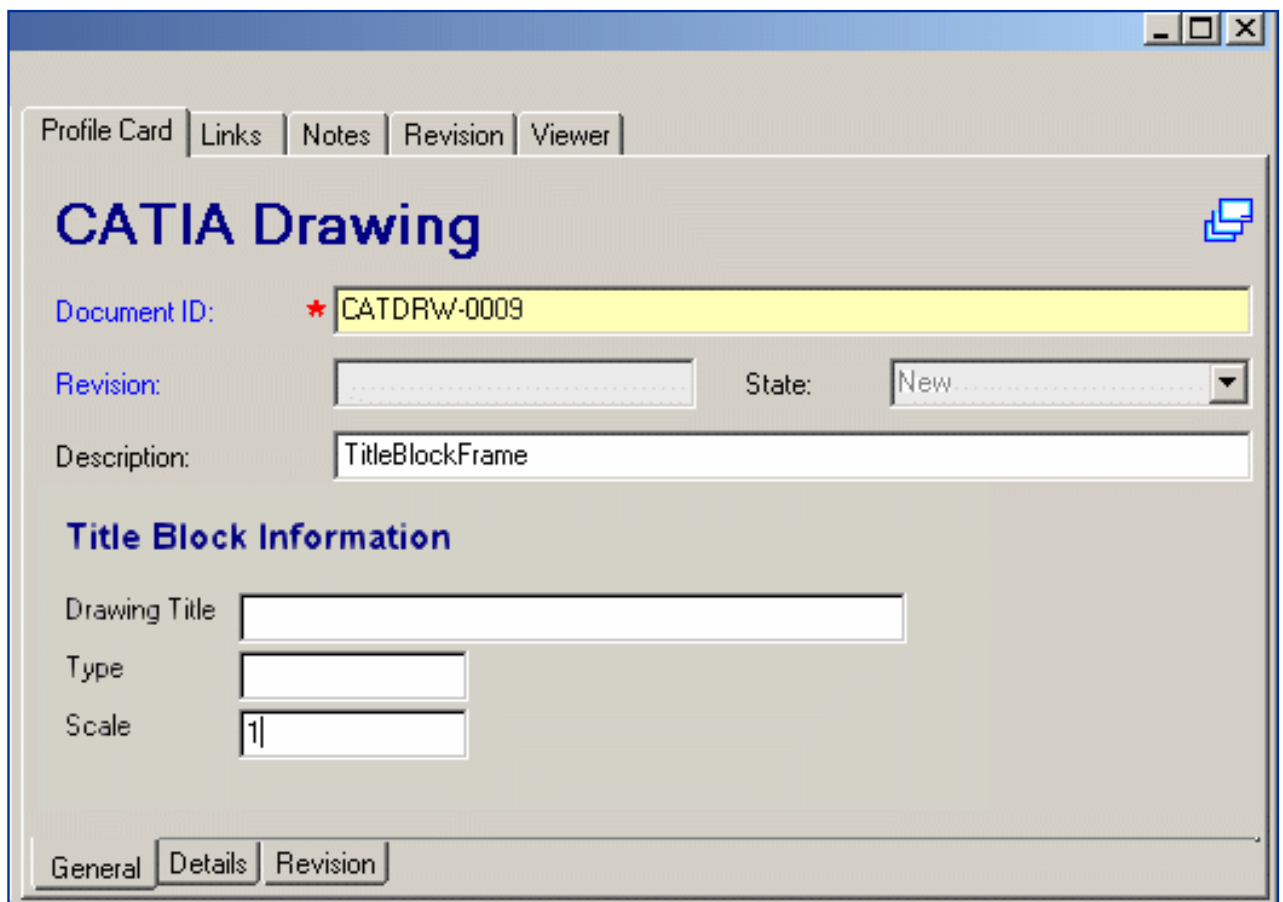


## Saving the Document in SmarTeam



1. Click the **Save** icon  or select **SmarTeam > Save**.

This declares the document and saves its properties in the database. In the SmarTeam profile card corresponding to the document, you can see that the fields are completed:



The screenshot shows a web-based interface for managing a CATIA Drawing. At the top, there are tabs for 'Profile Card', 'Links', 'Notes', 'Revision', and 'Viewer'. The 'Profile Card' tab is active. Below the tabs, the title 'CATIA Drawing' is displayed in a large blue font. To the right of the title is a small icon of two overlapping documents. Below the title, there are several input fields and a dropdown menu. The 'Document ID' field is highlighted in yellow and contains the text 'CATDRW-0009'. The 'Revision' field is empty. The 'State' dropdown menu is set to 'New'. The 'Description' field contains the text 'TitleBlockFrame'. Below these fields, there is a section titled 'Title Block Information' in blue. This section contains three input fields: 'Drawing Title' (empty), 'Type' (empty), and 'Scale' (containing the number '1'). At the bottom of the form, there are three tabs: 'General', 'Details', and 'Revision'. The 'General' tab is active.

Profile Card Links Notes Revision Viewer

## CATIA Drawing

Document ID: ★ CATDRW-0009

Revision: State: New

Description: TitleBlockFrame

### Title Block Information

Drawing Title

Type

Scale 1

General Details Revision

## Updating a CATIA Property in SmarTeam



1. Open the SmarTeam: Documents dialog box.
2. Right-click on the document and select **Update**.  
The profile card is now ready to be changed.
3. Modify the values of Drawing Title and Type.

Profile Card | Links | Notes | Revision | Viewer

## CATIA Drawing

Document ID: ★ CATDRW-0009

Revision:  State: New

Description: TitleBlockFrame

### Title Block Information


Drawing Title:

Type:

Scale:

General | Details | Revision

- Click OK.  
The property has been modified in the SmarTeam database.
- Back in CATIA, select **SmarTeam > Properties > Load from Database**.  
Still in CATIA, you can now see the new values attached to the Drawing title block.

		Dassault Systemes 9, Quai Marcel Dassault BP 310 92156 Suresnes cedex FRANCE	
Echelle : 1	Design by : joe	Date : 6/27/00 10:19:00 AM	
A0	Drawing Title		
Type : Frame	Title Block		



# Displaying a CATIA Part SmarTeam Attribute in a Title Block



The task described below shows how to ensure that the title block is automatically updated to display the part number of the associated Part.

A drawing title block contains information from various sources. It could look something like this, for example:

SCALE 1:1	NF E 27-494					
	APPLICABLE SPECS.	NEXT ASSEMBLY			FINAL APPLICATION	
Material	DRAWING NAME					
Part Number     PN-990256						
Designer						
UNLESS OTHERWISE SPECIFIED: DIM. ARE IN INCHES. DIM. IN PARENTHESIS ARE (MM) DEC: .XX±.01     FRAC: ±1/64 .XXX±.005    ANG: ±1° REMOVE ALL BURRS & SHARP CORNERS.	DATE		DRAWING NO.			ALT.
	CHK.	AT	DRAWING NO			A
	ENG.	JF				
	APP.	JF				

- The contents of the *Designer* field is a SmarTeam property. Its source is a field in the CATIA Product profile card. It corresponds to the *Created by...* field.
- The contents of the *Part Number* field is defined in CATIA. Property mapping enables this information to be available in the CATIA Part profile card as well.
- The contents of the *Material* field is also defined in CATIA.



1. Create a Part in a CATIA session.
2. Use **Edit>Properties** to define its part number.
3. Save the Part in the database.  
Default property mapping ensures that the **Part Number** field is automatically completed in the database.
4. Create a drawing of the same Part.
5. Design the title block creating a text with the Part Number information (see [Designing the Title Block](#) for more information).
6. Save the Drawing document in the database.
7. Use **SmarTeam>Properties>Map a Text Value...** to link this text to the Part Number property of the Drawing document (see [Mapping a Text Value](#) for more information).
8. Launch the *AttributesOfLinked.bs* script (see [Using SmarTeam Scripts in a CATIA Session](#) for more information).  
The title block is updated and now displays the Part Number of the part.

## Launching *AttributesOfLinked.bs*

When you launch the *AttributesOfLinked.bs* script, it:

- analyzes the logical links to other associated documents (i.e. dependencies)
- retrieves the value of the Part Number attribute for each associated document
- generates a character string containing a list of all values retrieved (one per line)
- stores the list of Part Numbers in the current document
- repeats this procedure on all the children i.e. sheets of the Drawing document.

## Customizing the *AttributesOfLinked.bs* Script

The *AttributesOfLinked.bs* script is designed to retrieve *Part Number* information from the linked documents. However, if you wish, you can:

- modify the property to be retrieved by editing the script to change the value of the *LinkedAttributeName* variable
- modify the property where the retrieved information is to be stored by editing the script to change the value of the *MainAttributeName* variable
- retrieve more than one property by editing the script to duplicate the section starting with the following comment line:
- Display information about the children.



# Designing the Revision Block



The scenario described below shows you how to design a revision block using properties defined in the database.

This task is divided into the following stages:

- [Creating the Revision Block on a New Drawing](#)
- [Defining the CATIA Formula Property for Each Attribute from the Database](#)
- [Creating the Text Attributes in the Revision Block](#)

The properties used for these tasks have already been mapped by the administrator between CATIA and SmarTeam (see Defining Property Mapping for the Revision Block).



## Creating the Revision Block on a New Drawing

1. Create a new CATDrawing document.
2. In the **New Drawing** dialog box that appears, make sure the default values (ISO in the **Standard** field and A0ISO in the **Format** field) are specified and click **OK**.
3. Design your revision block as shown below:

REVISIONS			
Revision	Comment	Approval Date	Authorized


For more information about the revision block, see the *CATIA - Generative Drafting User's Guide*.



## Defining the CATIA Formula Property for Each Attribute from the Database





1. In the CATIA session, click on the Formula  icon.  
The Formulas: Drawing dialog box appears.
2. Select **String** in the **New Parameter of type** list.
3. Select the **New parameter of type** button.  
You must now use the names defined in the mapping property (Revision, ApprovalDate, Comment & Authorized).
4. For example, change the name to *Revision0* in the field on the bottom left specifying the name of the current parameter. The numbers used must reflect the order in which the revisions were made e.g.: Revision0 = e, Revision1 = d, Revision1 = c, and so on.

Information are ordered, the index 0 will always contain the last revision information.

5. In the field on the right specifying the value of the current parameter, enter the "-" symbol.


Edit name or value of the current parameter

Revision0


-

6. Click on the **Apply** button.
7. Repeat steps 2 through 6 as often as required for all mapped properties.

Here is a list of the properties defined in the revision block and that are still to be mapped:

Property Name in CATIA 	Property Value	Property Type
Revision0, Revision1, Revision2, Revision3	-	String
Comment0, Comment1, Comment2, Comment3	-	String
ApprovalDate0, ApprovalDate1, ApprovalDate2, ApprovalDate3,	-	String
Authorized0, Authorized1, Authorized2, Authorized3	-	String

**Formulas: Drawing** ? X

 Import...

Filter On Drawing

Filter Name :

Filter Type :

Double click on a parameter to edit it

Parameter	Value	Formula	Active
Sheet.1\ViewMakeUp.2\Angle	0deg		
Revision0	-		
Comment0	-		
ApprovalDate0	-		
Authorized0	-		
Revision1	-		
Comment1	-		

Edit name or value of the current parameter

New Parameter of type  With

Add Formula

Delete Parameter Delete Formula

OK Apply Cancel

8. Click OK.



## Creating the Text Attributes in the Revision Block



Here is what you will obtain at the end of this task:

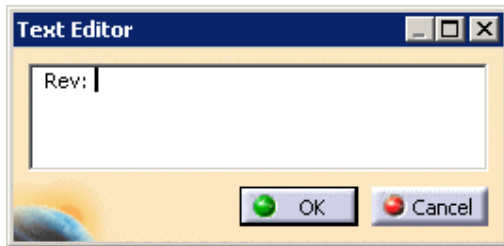
REVISIONS			
Revision	Comment	Approval Date	Authorized
Rev: -	Com: -	Date: -	Auth: -
Rev: -	Com: -	Date: -	Auth: -
Rev: -	Com: -	Date: -	Auth: -
Rev: -	Com: -	Date: -	Auth: -



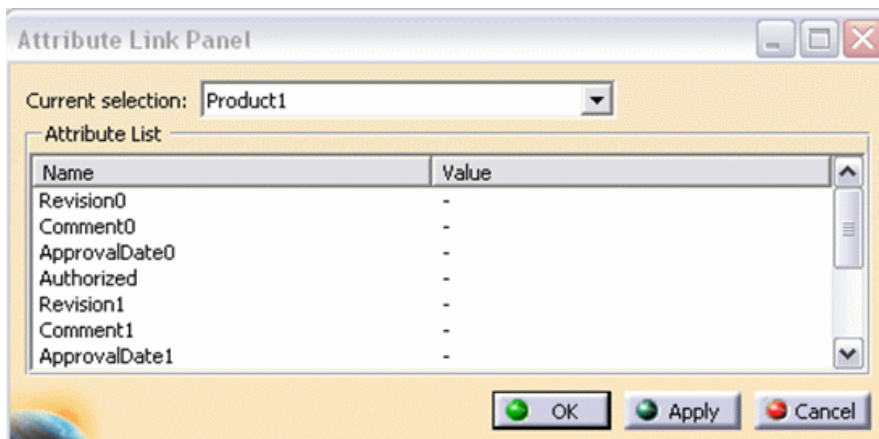
For more information see "Adding Attribute Links to Text" in the *CATIA - Interactive Drafting User's Guide*.



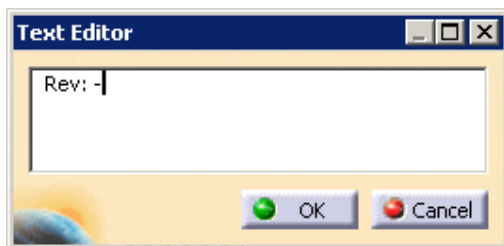
1. Select **Insert>Annotations>Texts>Text**.
2. Click inside the drawing to indicate where you want to display the drawing type. (If necessary, refer to the revision block design.)  
A text editor window appears.
3. Enter **Rev:** in this window:



4. Right-click on the appropriate location in the revision block and select **Attribute Links** in the contextual menu.
5. In the specification tree of the drawing, click on the **Drawing**.  
The Attribute Link dialog box appears.
6. Select **Revision0** in the attribute list:



7. Click **OK**.  
The value of the property, i.e., the "-" symbol, is now displayed in the Text Editor dialog box:



8. Click **OK**.
9. Repeat steps 1 through 8 for all of the cells.



# Displaying a CATIA Drawing SmarTeam Attribute in a Title Block



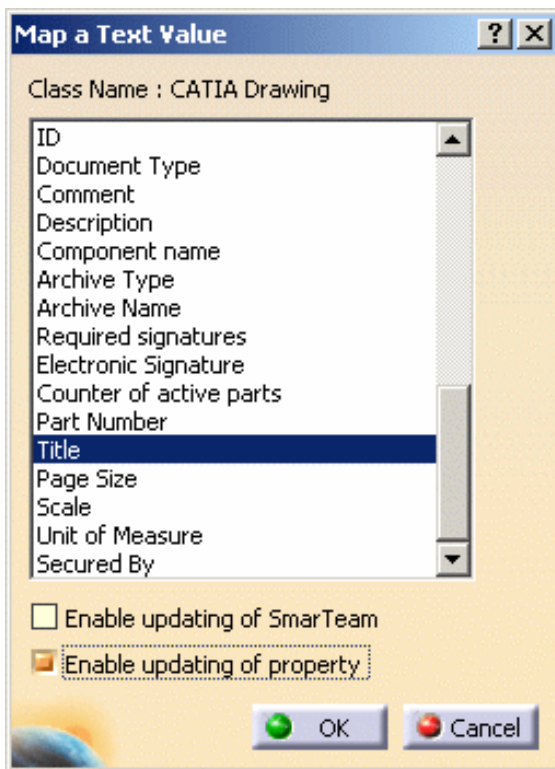
This task enables the administrator to quickly and easily map a drawing text with any of the CATIA Drawing attributes in the SmarTeam database.

There is no need to perform the two manual operations that consist in:

- creating a property  $f(x)$  in the drawing then linking it with the text
- going either to the Property Management or [Integration Tool Setup](#) utilities.



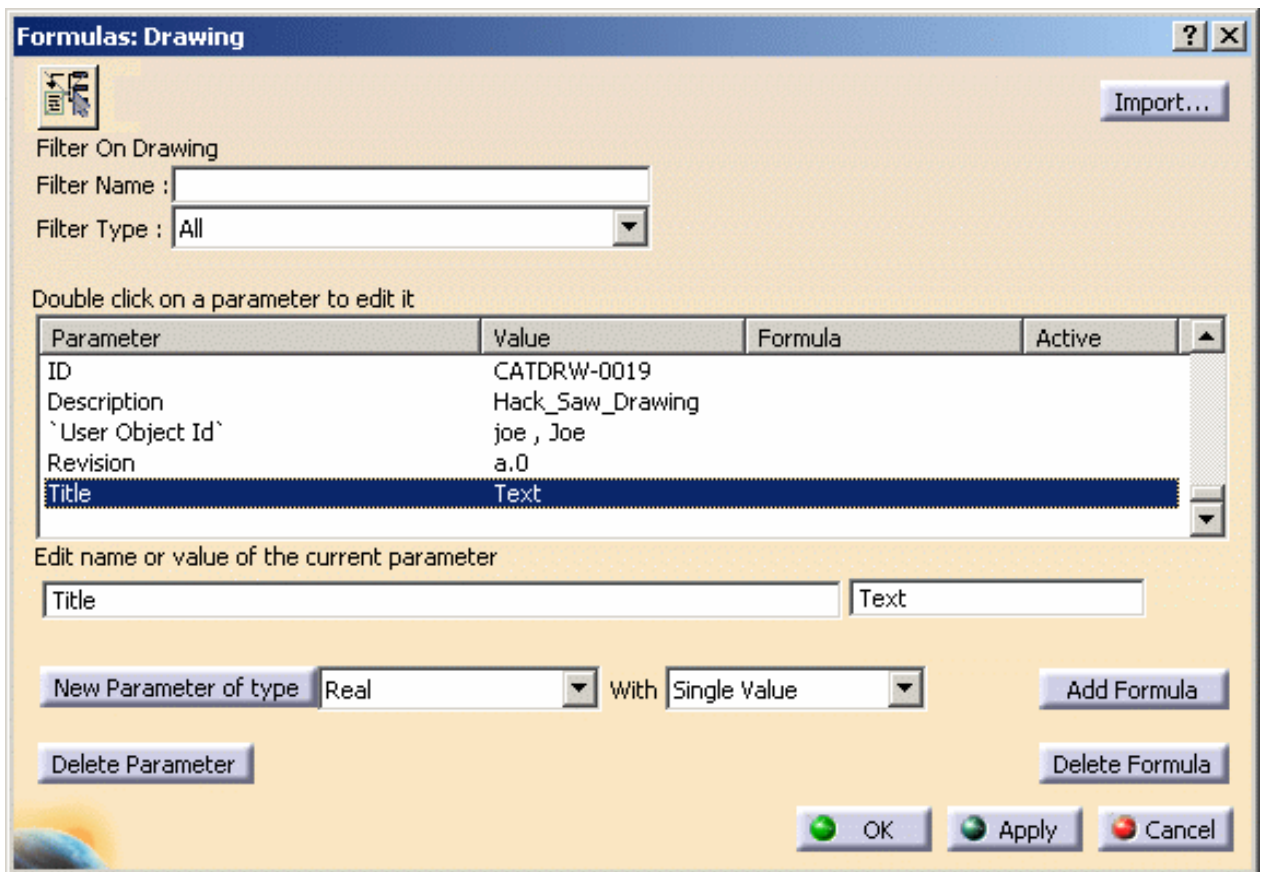
1. In the CATIA session, open a drawing document that has already been saved in the SmarTeam database.
2. Select a text of the drawing.
3. Select **SmarTeam > Properties > Map a Text Value...**  
The **Map a Text Value** dialog box appears.
4. In this dialog box, select an attribute of the document's class.
5. At the bottom of the dialog box, define the update direction by selecting the **Enable updating of property** option:



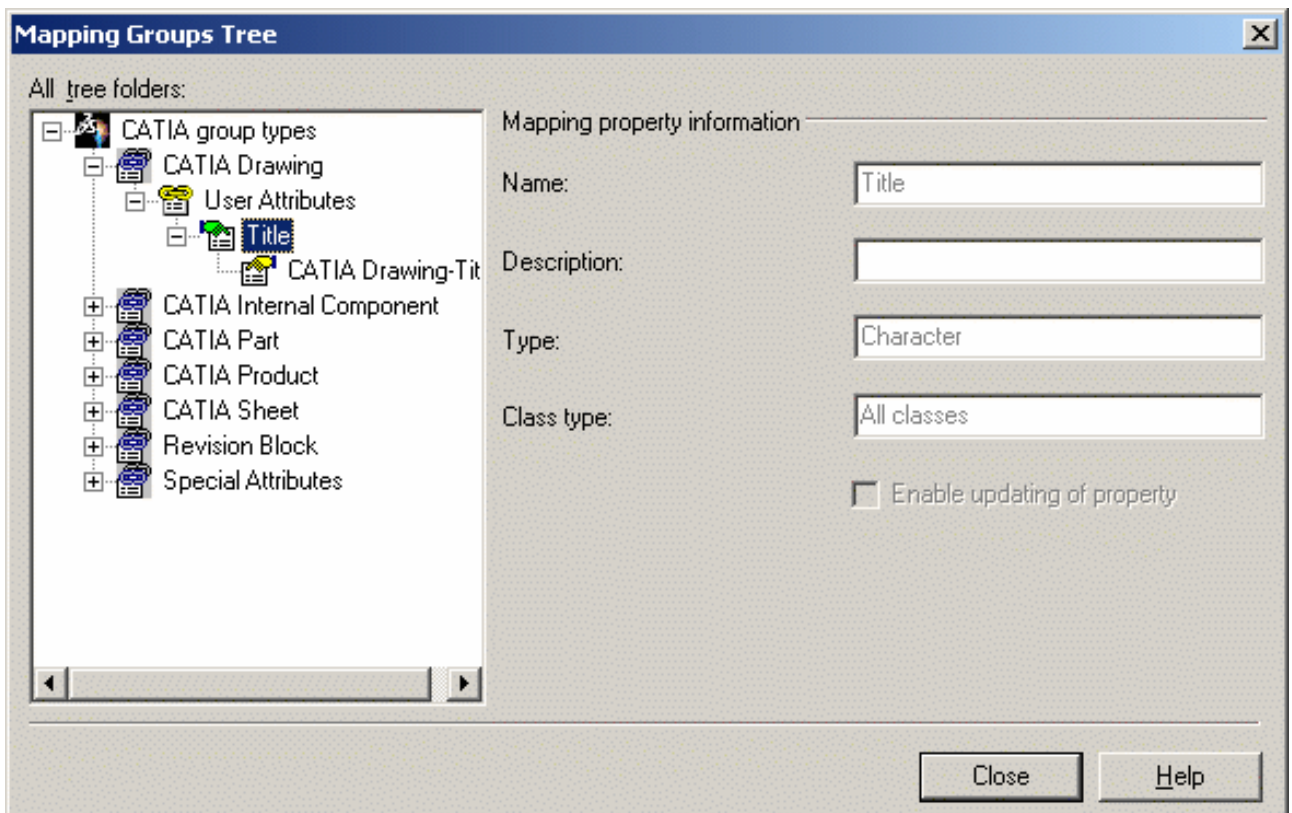
The **Map a Text Value** dialog box shows the current mapping for the selected text. In other words, if you have already used the **Map a Text Value...** command for the same text, the attribute name and the update direction(s) you originally selected are kept.

6. Click **OK**.

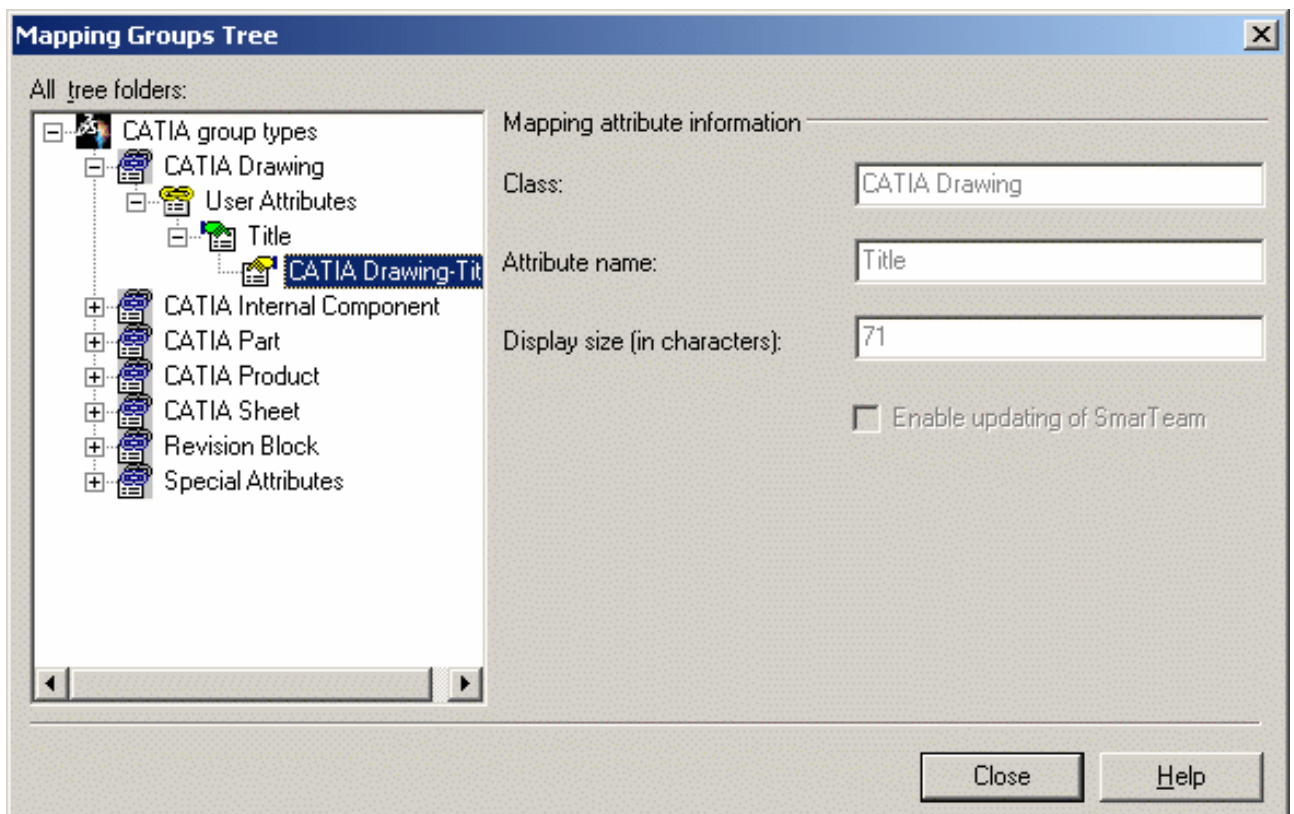
A CATIA property is created inside the drawing as shown in the **Formulas** dialog box, for example:



7. Selecting **SmarTeam > Tools > Property Management**, you can see that the newly mapped property looks like this:



and the newly mapped attribute like this:



From now on, whenever you reopen the document the mapped attribute will appear in the title block. You can also force the update by selecting **SmarTeam > Properties > Load from Database**.



# Creating a Drawing Document from a Template



This task shows you how to create a drawing document from a template.



1. Run a query to access your CATPart document.
2. Select **SmarTeam > New From...**  
The **New From...Documents** dialog box is displayed.
3. In the displayed tree, double-click on the folder of interest or select the + sign to open the folder.
4. Select an existing drawing.  
A new drawing is created starting from the selected template.  
Each drawing document is represented in the database as a tree. The root object of the tree represents the drawing itself then each sheet of the drawing is represented as a child of the drawing.  
If you want to verify the document before opening it, select the **Viewer** tab to preview it.  
It is now possible to see the different sheets of a CATDrawing document in the SmarTeam viewer. For more information, see [Enabling Multi-Sheet CATDrawing Documents Preview](#).
5. Click **OK**.
6. Insert in your drawing the projection views of the CATPart document and insert the dimensions.  
For more information about projections and dimensions, see the *CATIA - Generative Drafting User's Guide*.

## No Template Project is Defined

In case no templates project is defined, selecting **File > New From...** launches the standard SmarTeam open dialog allowing you to select any document from the database.



# Text ID Mapping



## Concept

Title Block text ID mapping allows users to map text and legacy data more easily. Texts in the Title Block can now be mapped directly from/to CATIA.

Until now, the Title Block mapping was based either on the CATIA property mapping mechanism or on a user-written CATIA macro. In V5R19, the only constraint is that the text id must be defined according to SmarTeam rules; the mapping is performed automatically.

In a CATIA drawing, texts are mapped to SmarTeam attributes that can be displayed in the SmarTeam profile card. The user can update the text that is mapped to SmarTeam in the drawing by modifying the text directly or by updating the drawing based on SmarTeam attribute values.



## Procedure

The user uses the Title Block defined by the Administrator. The user can change the text in the Title Block and, after a Save operation is performed, the Title Block is updated.

Note: The Text ID mapping only takes place for texts located in the background sheet.

## Mapping Order

If the same attribute has been mapped in different ways from/to CATIA the mappings take place in the following order:

1. File properties mapped from CATIA to SmarTeam
2. CATIA Text name mapped from CATIA to SmarTeam
3. CATIA knowledge parameters mapped from CATIA to SmarTeam
4. CATIA knowledge parameters mapped from SmarTeam to CATIA
5. CATIA Text name mapped from SmarTeam to CATIA
6. File properties mapped from SmarTeam to CATIA

For information on defining Title Block text ID mapping, refer to [Configuring Title Block Text ID Mapping](#).



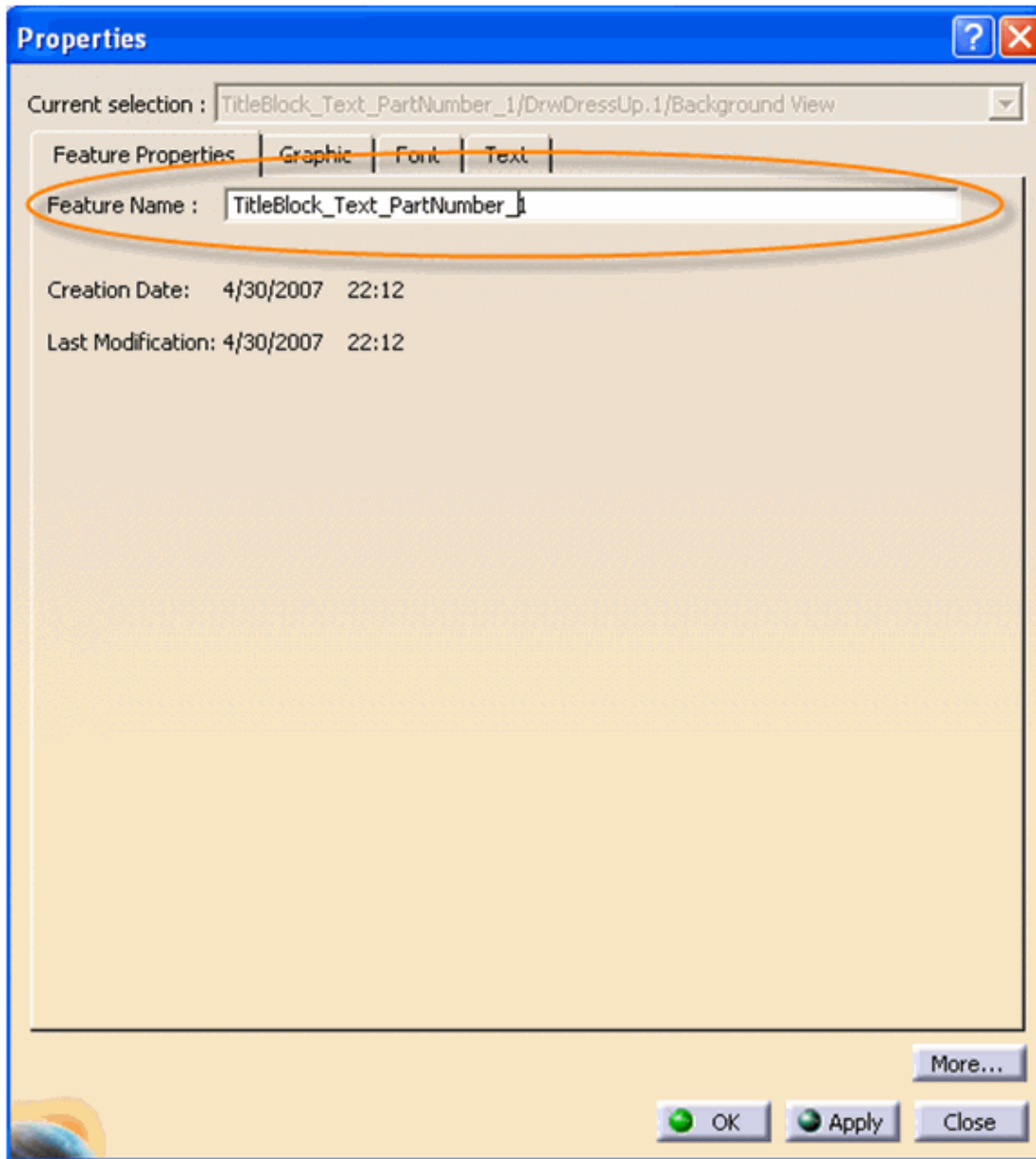
# Configuring Title Block Text ID Mapping

Configuration of the property mapping for the Title Block Text IDs is performed by the Administrator during the setup and involves two steps:

- Defining the Feature Name
- Defining Mapping in SmarTeam Integration Tool Setup

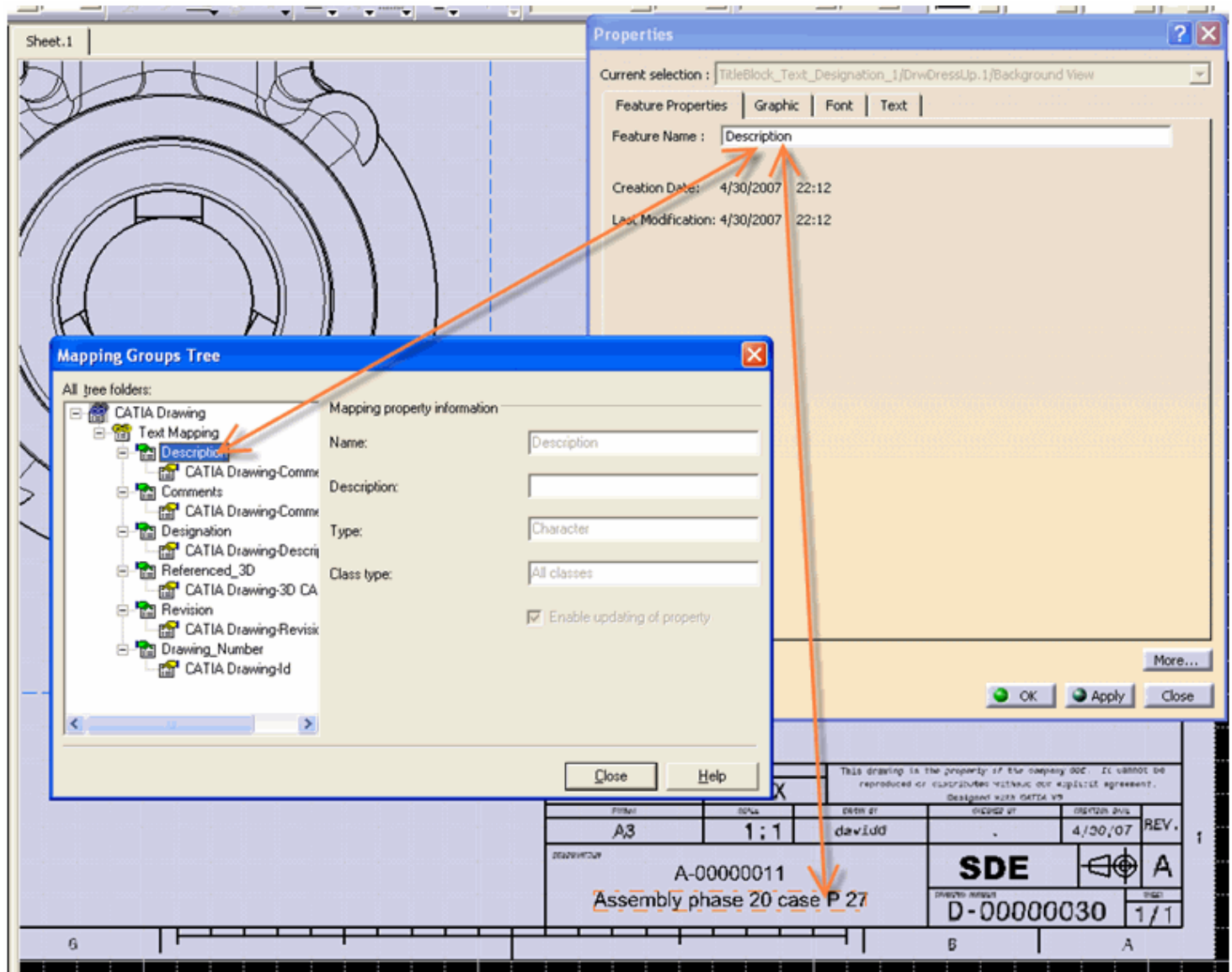
## Defining the Feature Name

In the **Title Block Properties** window, the administrator types in a specific Feature Name in the **Feature Name** field.



## Defining Mapping in SmarTeam Integration Tool Setup

The feature name is mapped to SmarTeam using the Integration Tool Setup. In the Integration Tool Setup, the administrator defines mappings between the CATIA Drawing text and a SmarTeam attribute, as shown in the example below:



The administrator can define the mapping direction by enabling/disabling the updating of properties. For details, see the Integration Tool Setup documentation in the SmarTeam – Editor Online Help.

- When **Exposed Mode** is set to No, the mapping will be defined in the mapping group with the name of the class to which the CATIA Drawing is mapped.
- When **Exposed Mode** is set to Yes, the mapping will be defined in the mapping group with the name of the class to which the CATIA Sheet is mapped.



# Managing the Document Lifecycle



SmarTeam enables you to maintain and manage any information related to a document throughout its life cycle. By mirroring the physical process of document management, SmarTeam uses vaults, check-in, check-out, and approval functions to manage the life cycle of your Product, Part or Drawing. It creates new versions of a document and protects it from unauthorized modifications.

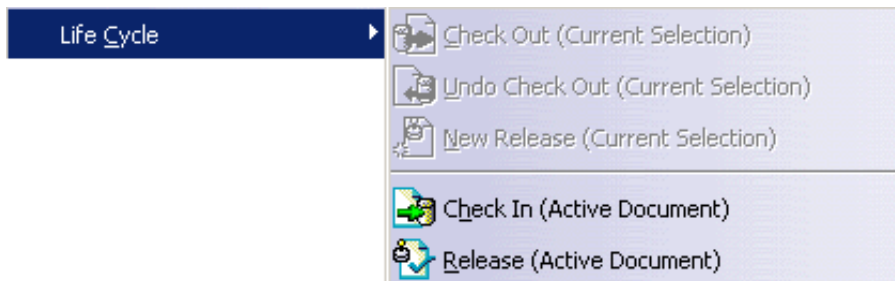
**All life cycle operations on CATIA documents must be performed in the CATIA session. Operations such as Check-Out, Check-In etc. must be explicitly done in CATIA to guarantee data consistency.**



## Lifecycle Menu Commands

The **Lifecycle** menu contains the life cycle commands for managing a Part, Product and Drawing as new revisions are created.

1. From the **Life Cycle** menu, choose the appropriate operation in accordance with its status in the product life cycle.



All lifecycle operations are controlled and managed by SmarTeam, by enabling and disabling different life cycle options in the **Life Cycle** menu. This provides for a logical flow of a Part, Product or Drawing through its life cycle. When a new document is first saved in the SmarTeam database, it has the **New** status. The document is not placed in a vault.

When you perform the **Check In** operation on the document, it is placed in a vault and cannot be launched into CATIA V5 until it is checked out of the vault.

## Contextual Commands

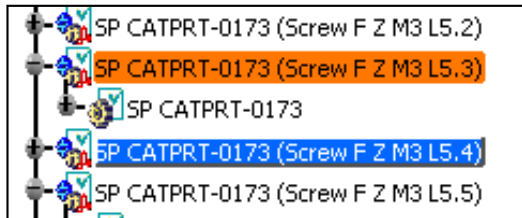
From V5R15 onward, you can access the following Lifecycle commands from contextual menus:

- **Check Out**
- **New Release**
- **Undo Check Out**

Whatever the operation you choose, it applies to the document you have selected from the specification tree (or from the geometry area).

Just as a reminder, the application distinguishes **selected documents** from **active documents**.

- **Selected documents:** To select a document, you just need to click it. Once selected, it appears as highlighted.
- **Active documents:** double-clicking a document, activates that document. In concrete terms, this sets up a working context. For example, if you double-click a Part, CATIA opens the Part Design workbench for you to access all different capabilities for editing that part.



> Selected document


> Active document

To use **Save**, **Check In** and **Release** via contextual commands, you still need to activate the documents on which you want to perform these operations.

## Life Cycle Operations



The table below lists the different life cycle operations, the status of the document resulting from each operation, and a description of each status.

Life Cycle Command Selected	Resulting Status	Description
Click <b>Check In</b> to to check a <i>new</i> document into the vault or to place a document that is <b>being modified</b> back in the vault.	<b>Checked In</b>	The document is placed in the vault, and it cannot be launched into CATIA V5. In order to launch it into CATIA V5, the document must be checked out of the vault. You can copy the file to your desktop in order to view (but not modify) the document in CATIA V5.
Click <b>Check Out</b> to check out a document from the vault.  or  Click <b>New Release</b> to make a new copy of a <b>Released</b> document that was placed in the <b>Released</b> vault. The resulting document is a new revision of the source document.	<b>Being Modified</b>	This is a temporary state assigned to a document that has been checked out. The document can be launched into CATIA V5 in order to modify it. No other user can currently modify it, but other users can view it or copy the file to their desktop. After the document is checked back in or released, the status is replaced by <b>Checked In</b> or <b>Released</b> .
Click <b>Release</b> to transfer a document to the <b>Released</b> status.	<b>Released</b>	The document is saved in the vault of released documents.
Click <b>Undo Check Out</b> to cancel the check-out of a document from the vault.	<b>Checked In</b>	Any changes made following the check-out are lost. The new revision is deleted. The document status reverts to <b>Checked In</b> status.   Depending on the context, <b>Undo Check Out</b> can be available for modified documents.



# Analyzing the Impacts of a Change



The **Impact Analysis** function displays all the documents pointing onto a selected object. This tool allows you to evaluate the impacts brought by the modifications made to the current object and therefore helps you choose the solution to implement with the least modification cost.

We recommend you use this tool prior to checking out and modifying documents.

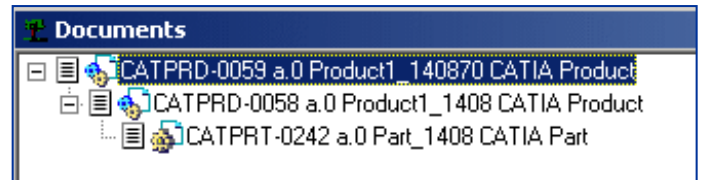
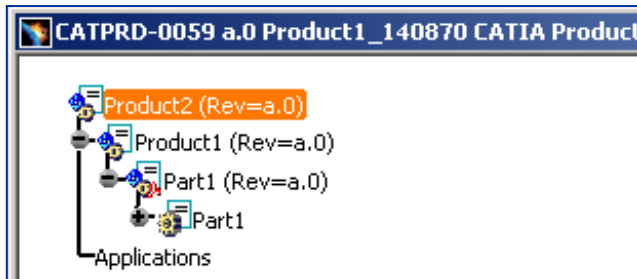
**Note:** Impact Analysis is not supported in 64-bit environment.



This scenario assumes your CATIA session contains a CATProduct document that comprises a CATProduct document including a CATPart document. A CATDrawing document was created from the assembly, it is thus also linked to that CATProduct document.



1. Select **Product2** as the document which related documents you want to see.



2. Click

**Impact Analysis** is also available from the **SmarTeam** menu, contextual menus and in the **Desk** window.

The **Impact Analysis** dialog box that appears displays all documents related to the selected product.

Note that you can resize it to improve the view.

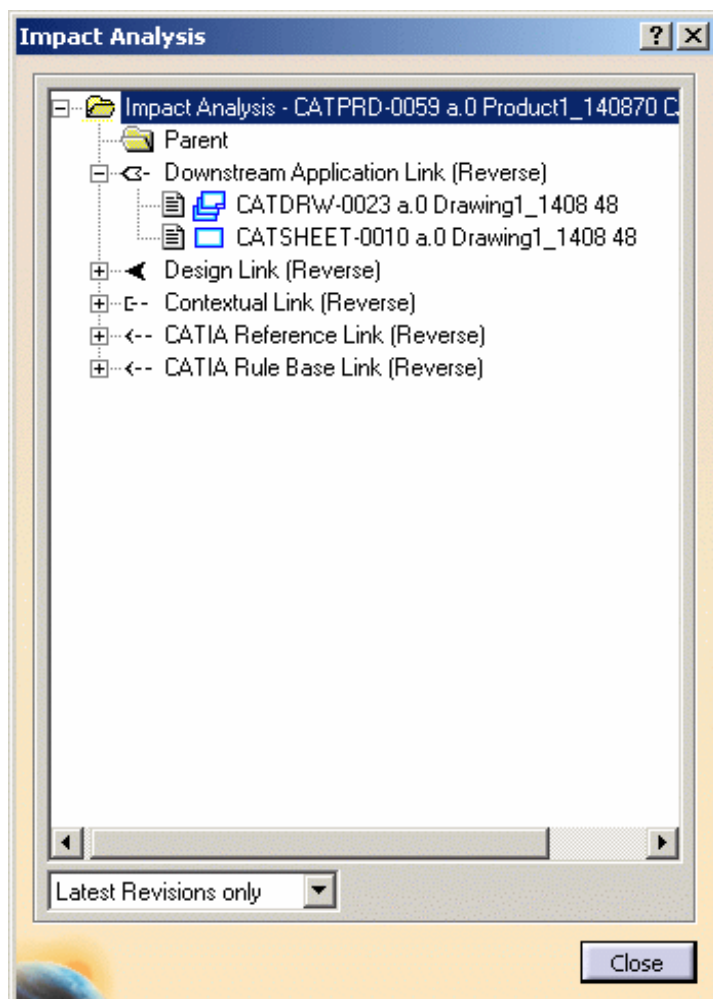
Three options let you choose the way in which the information is displayed.

- Latest Revisions only
- Grouped Cross Revisions
- Cross Revisions

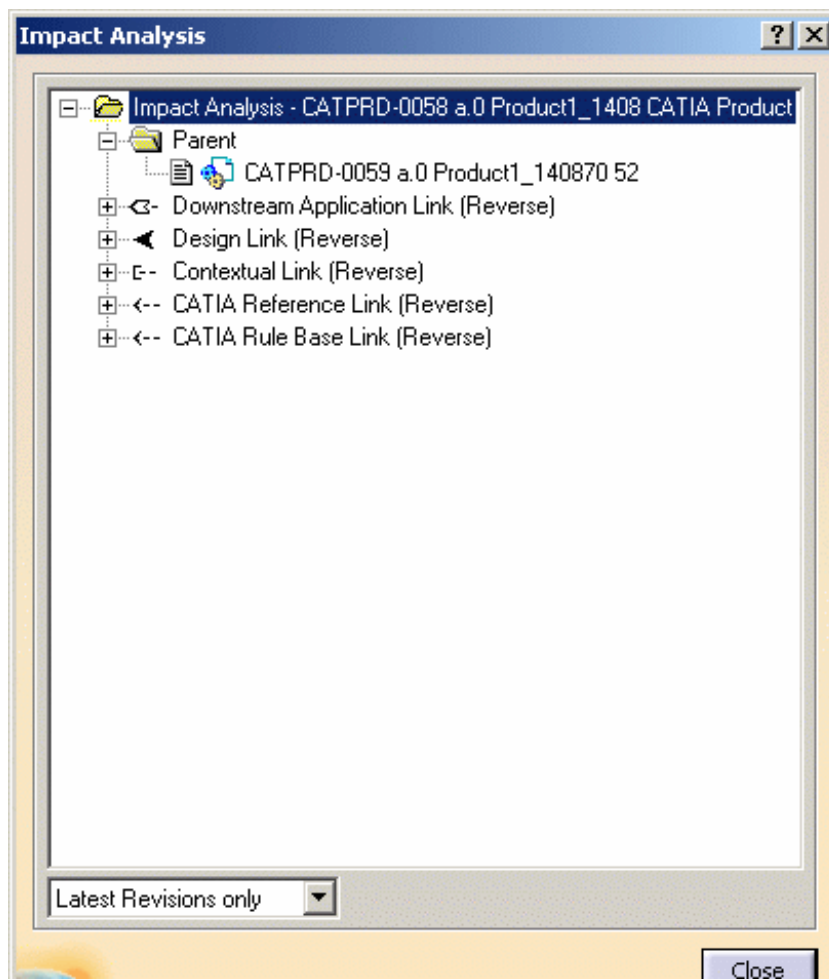
Whatever option you select, the way objects are displayed in the **Impact Analysis** dialog box depends on the way the SmarTeam tree properties has been customized. For more information, see [Customizing SmarTeam Document Display Information](#).

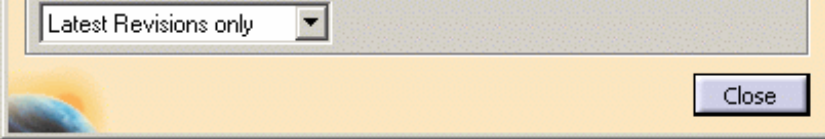
## Latest Revisions only

3. From the drop-down menu in the lower part of the dialog box, set **Latest Revisions only**. This option displays only the latest revisions of the documents that will be impacted by the current revision of the selected object.
4. Click the + sign in front of the **Downstream Application Link (Reverse)** category. This category lists the latest revisions of the drawing documents pointing onto the assembly you have selected. The **Parent** category is empty since CATPRD-0058 a.0 Product1\_1408 CATIA Product is referenced by no document.



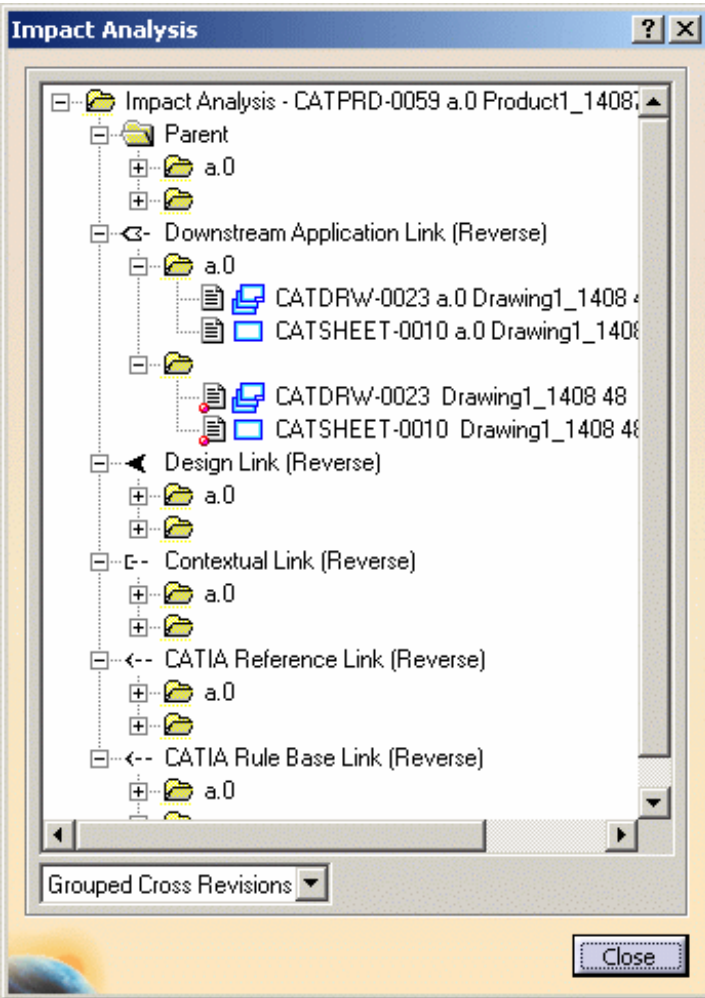
By applying **Impact Analysis** onto the sub-assembly, you could see in the **Parent** category that CATPRD-0058 a.0 Product1\_1408 CATIA Product is its parent. You would obtain this view:



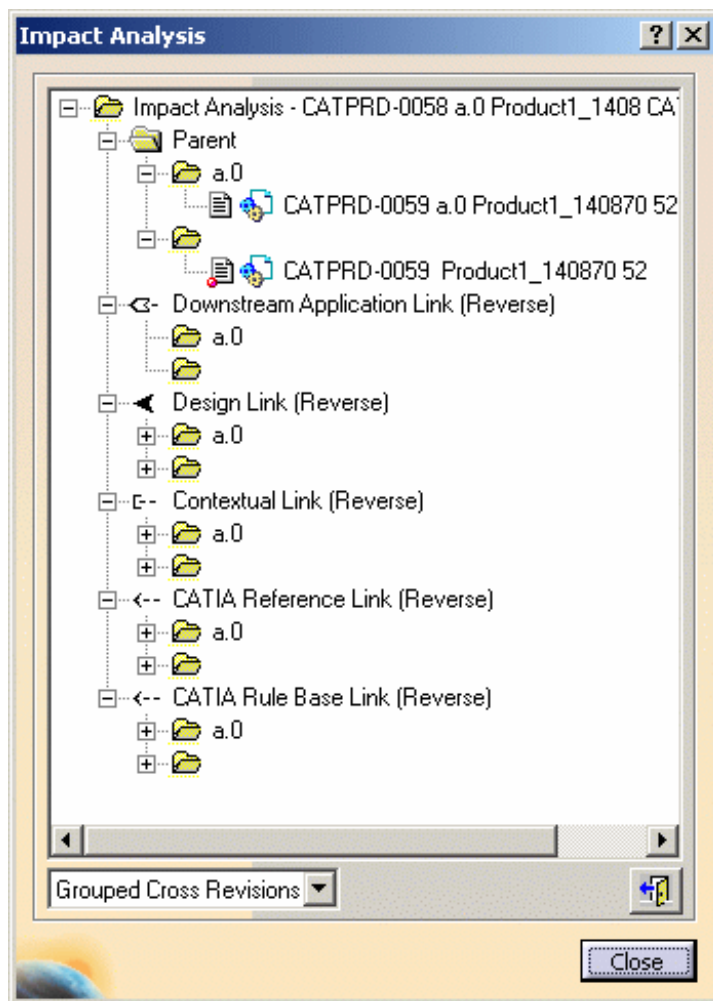


## Grouped Cross Revisions

3. Still using **Product2** as the selected document you are analyzing, from the drop-down menu, set **Grouped Cross Revisions**.
4. Click the + sign in front of the **Downstream Application Link (Reverse)** category. **Grouped Cross Revisions** displays also the impacted documents as well as all their revisions. You obtain this view:

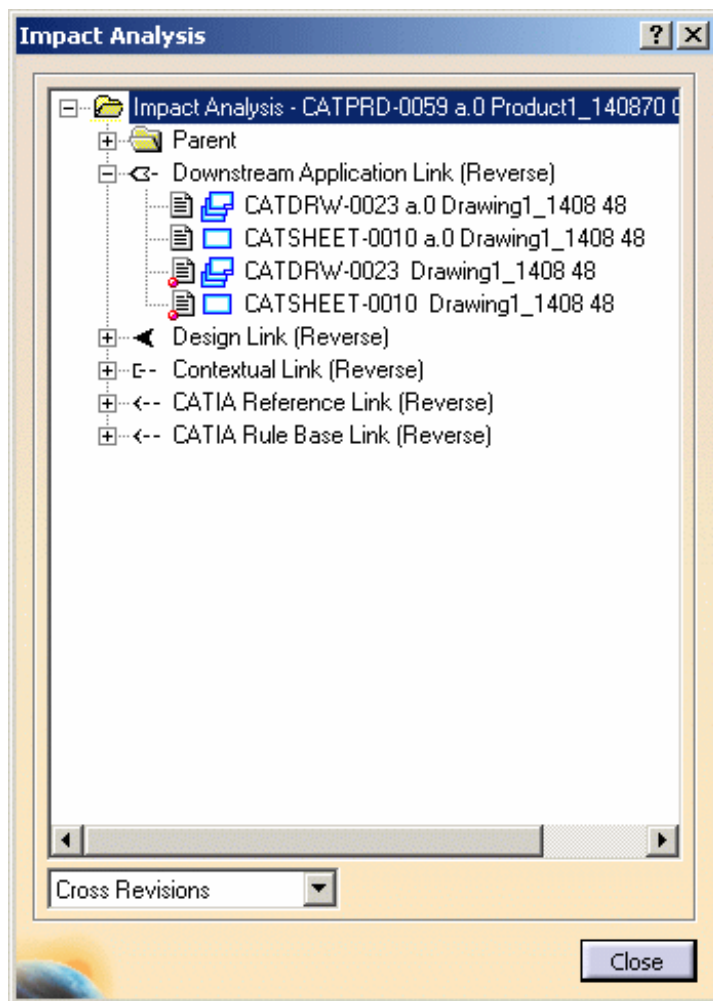


By applying **Impact Analysis** onto the sub-assembly, you could see in the **Downstream Application Link (Reverse)** category that an earlier version exists for its parent CATPRD-0058 a.0 Product1\_1408 CATIA Product. You would obtain this view:



## Cross Revisions

3. Still using **Product2** as the selected document you are analyzing, set **Cross Revisions**.  
This option displays the same contents as when using **Grouped Cross Revisions**, but the presentation of impacted documents is slightly different. There is no sub-category within each category. In our example, the display shows that the latest revision of the drawing as well as the previous one would be affected by changes to the assembly.



4. Click Close when done.

## Locate Active Document

In case you want to know more information about a document displayed in the **Impact Analysis** tree, you can:

- right-click the document and select **Locate Active Document**

Or

- directly double-click the document

This opens a SmarTeam dialog box with detailed information on the document.




# Administration Tasks

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

# Mandatory Settings

This section provides information about major operations a system administrator should perform prior to using SmarTeam CATIA Integration. These operations deal with the way of accessing documents in CATIA.


# Document Localization Strategies

 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.

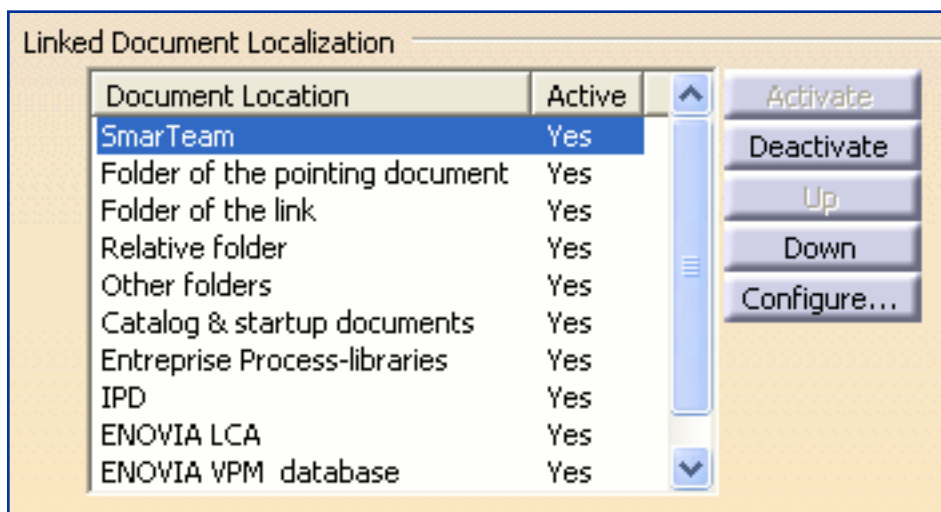
There are two main folders:

- 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.

 This behavior can easily be set in CATIA:

1. Select **Tools > Options....**
2. In the **General** category, click the **Document** tab.
3. In the **Linked Document Localization** section, locate and select **SmarTeam**. Ensure that **SmarTeam** is the first item in the list.
4. Use the **Activate** button to activate this strategy.
5. Use the **Up** button to place this strategy at the top of the list.
6. Repeat these operations for **Folder of the pointing document** strategy and **Folder of the link** so that they become the second and third items in the list respectively.
7. Ensure that the SmarTeam Document environment is allowed and is in the current state in the **Document Environment** selection list.



## SmarTeam Strategy

The application manages the links of documents saved in the database in this hierarchical order:

- Priority 1: folder of the pointing document (to find documents in the user's work directory)
- Priority 2: folder of the link (to find shared non-revision managed documents)

For further information about linked document localization strategies, see "Document" in the *CATIA - Infrastructure User's Guide Version 5*.

## 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). This is strongly not recommended: the multi-work strategy is not supported by the SmarTeam CATIA integration. In this specific case, no document localization strategy will be able to recover the linked documents. The pointing document will be opened with broken links towards the linked documents.



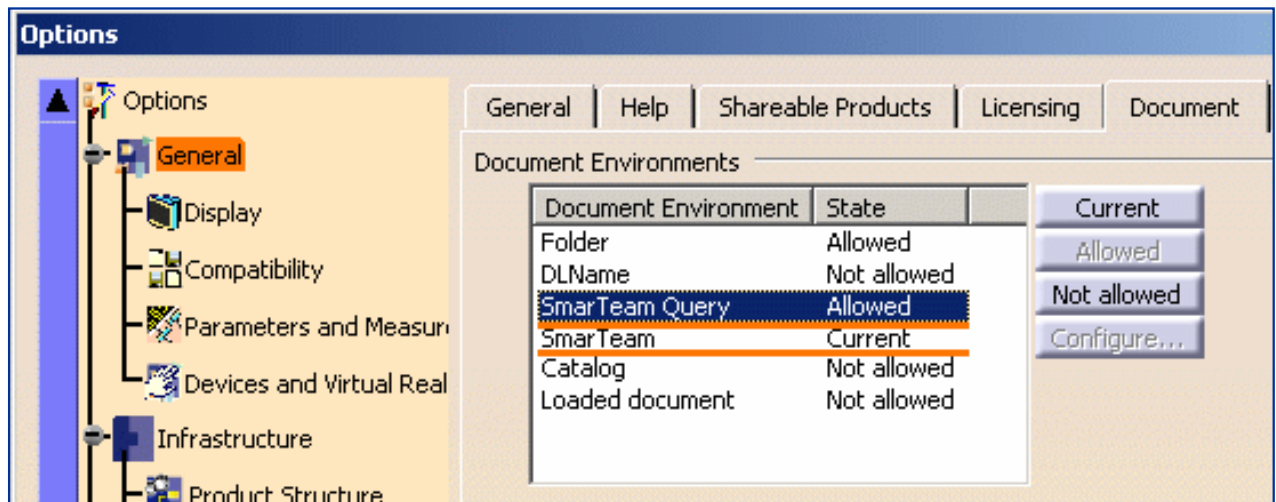
# Enabling the Display of the SmarTeam File > Open User Interface



This task shows you how to enable the display of the [SmarTeam File > Open](#) user interface.



1. Launch **Tools > Options**.
2. Select the **General > Document** tab.
3. From the **Document environments** display list, set **SmarTeam** as the **allowed** and **current** environment



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


## Document Environments

There are two possible document environments that allow you to retrieve the document from SmarTeam during a CATIA operation (**File > Open**, **Replace Component** etc.).

- **SmarTeam** Document Environment allows you to perform a simple query, or navigate thru a project structure or define a precise query to retrieve the document.
- **SmarTeam Query** Document Environment allows you to use the Find user interface to retrieve the document (through predefined queries or precise queries). Additionally, ensure that to be able to use the Find capabilities, the **Folder** Document Environment is set to **Allowed**.



# Lifecycle Rules for Copy File and Check Out Operations

 This task shows you how to ensure that each time you will perform Check out and Copy File operations, all the files and their linked objects will be copied to your working directory. What you need to do is define two lifecycle rules as explained below.

## "Copy File" as the Destination Operation

Using the Lifecycle Rules Setup utility, you must set **Copy File** as the destination operation for **Copy File** and **Check Out** original operations.

### Warnings

We strongly recommend you follow this methodology. If you do not set **Copy File** as the destination operation for:

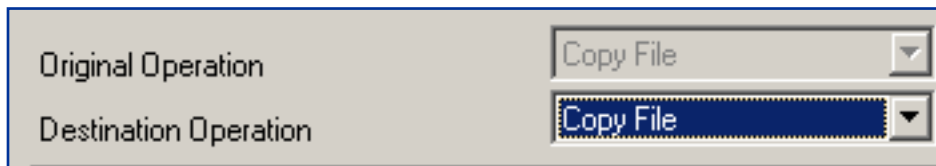
- the original "**Copy File operation**", this may:
  - induce broken links when opening a file in CATIA
  - prevent instantiations when using catalogs.
- the original "**Check Out operation**", this may induce the loss of links in SmarTeam which results in broken links in CATIA sessions, and incorrect impact analysis information.

## How to Change the Rules

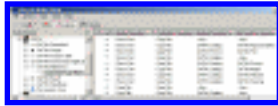
1. Select **Start > Programs > SmarTeam > Administrative Tools > Life Cycle Rule Setup**.  
This opens the Lifecycle Rules Setup utility.
2. Expand the tree until you reach the node representing the **CATIA Reference** category.
3. In this category, select **Component Type Rules**.
4. Set **Copy File** as the destination operation for **Copy File** and **Check Out** original operations:

Original Operation	Destination Operation
Copy File	Copy File
Check Out	Copy File

For this, right-click the original operation item of interest and use **Edit** contextual command to access the **Lifecycle Rules Properties** dialog box for the selected item.



When done, you must obtain this:



## Examples

You must make sure that:


- "**Copy File**" operation is set for original "**Copy File**" operation on **CATIA Reference Link** (this was not the default option delivered in SmDemo). Otherwise, you may have difficulties in instantiating objects from catalog browsers.
- "**Copy File**" operation is set for original "**Copy File**" operation on **CATIA Contextual Link** (this is the default option delivered in SmDemo). Not setting this option as recommended may lead to the loss of CATPart>CATPart link information in SmarTeam. This is very dangerous specially because impact analysis may not be guaranteed.




# Customizing

This section discusses different ways of customizing your SmarTeam CATIA Integration session.

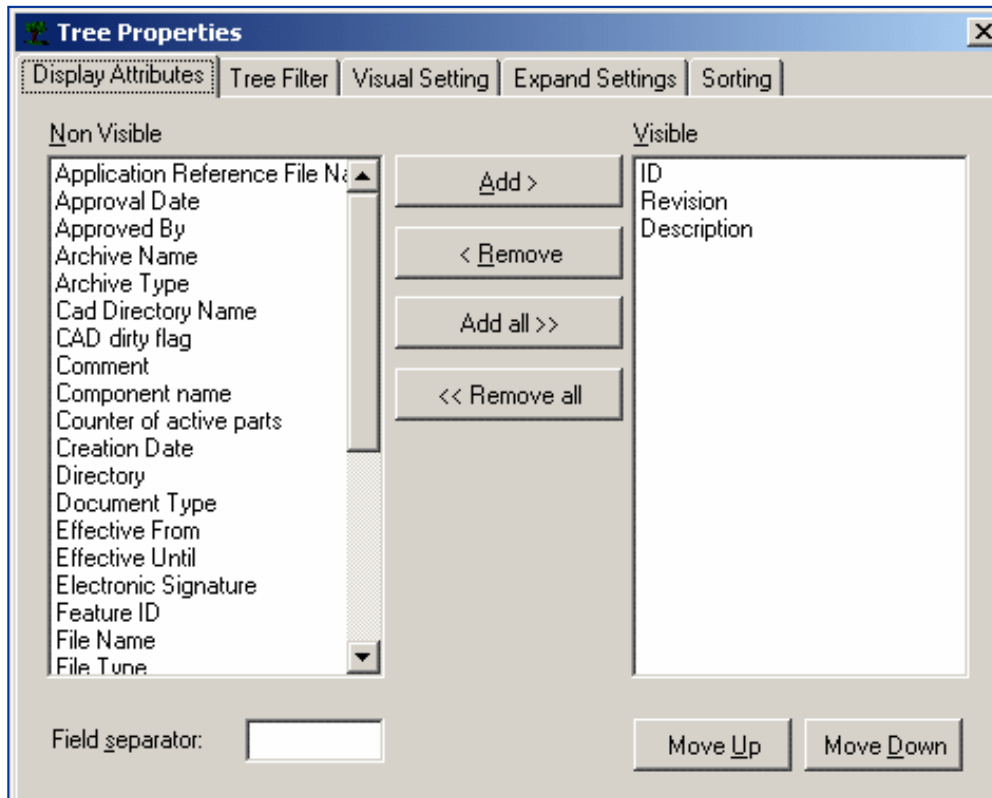
# Customizing SmarTeam Document Display Information

 The SmarTeam document display information can be customized in SmarTeam by means of the **Tree Properties** contextual command.

 1. Select your document in SmarTeam.

2. Right-click **Tree Properties**.

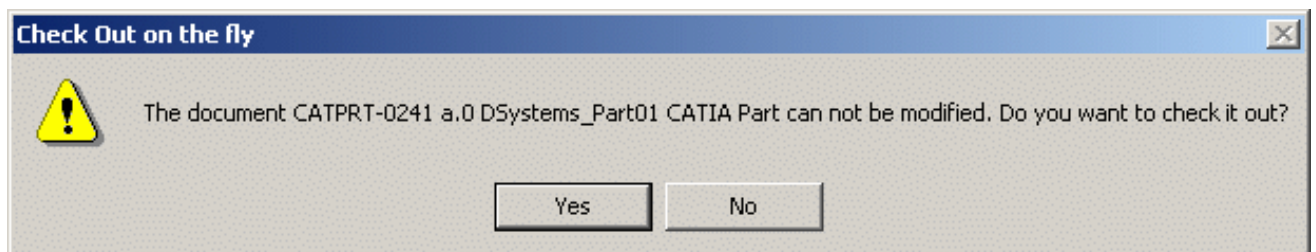
The **Tree Properties** dialog box that appears contains various tab pages to customize the appearance and tree content. The **Display Attributes** tab enables you to specify which attributes will comprise the name of the objects as they are displayed in the tree.




**Check out, check out on the fly** operation and **Not Saved** warning windows now display SmarTeam information as set in the **Tree Properties** dialog box.


In the following example, the following attributes have been defined:

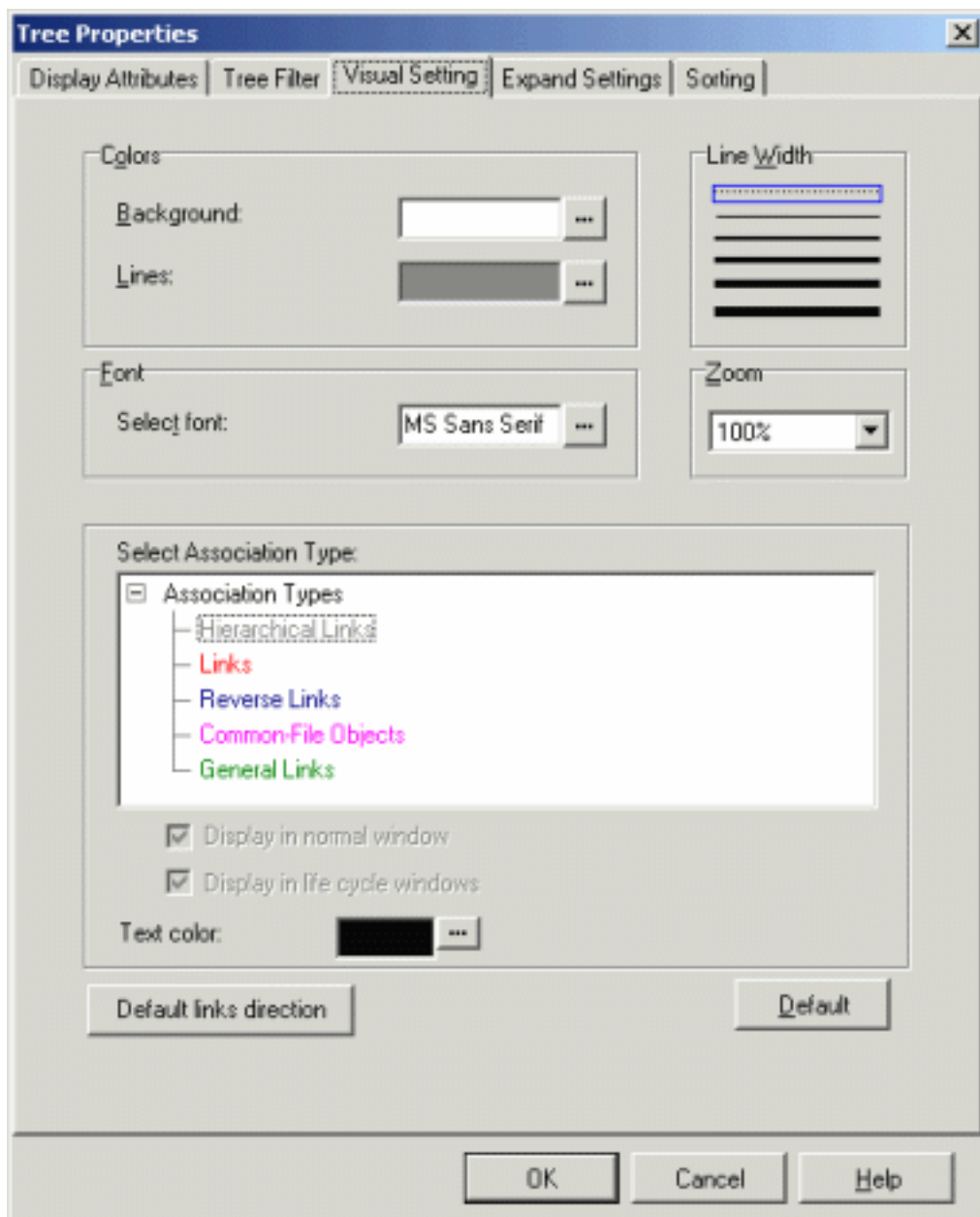
- ID
- Revision
- Description
- File Type



# Configuring the Links Display

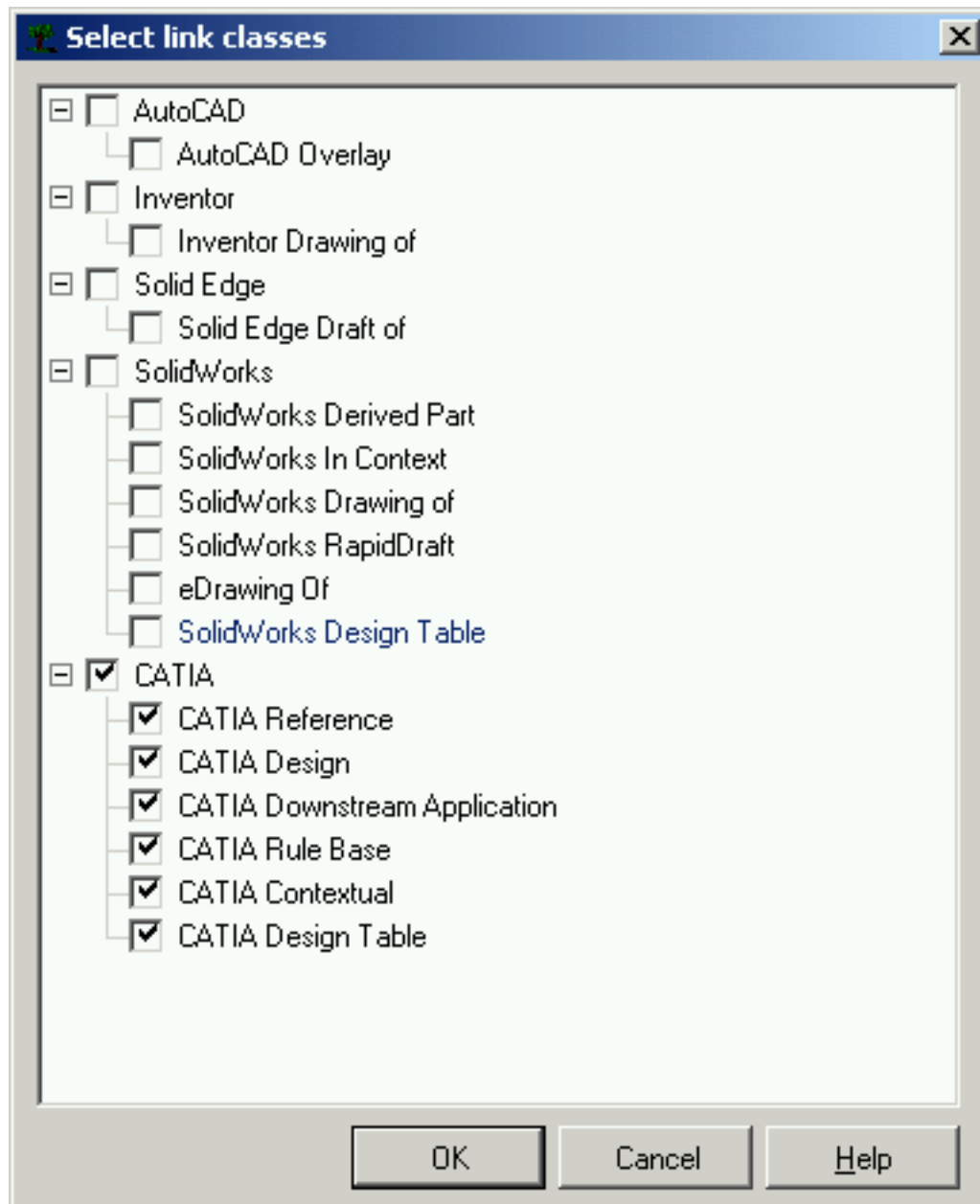
 This page shows you how to configure the links of the documents in SmarTeam by means of the **Tree Properties** contextual command.

-  1. Select your document in SmarTeam.
2. Right-click **Tree Properties**.
3. Click the **Visual Setting** tab.
- This tab enables you to adjust the background color, line color and size, font type and size as well as the zoom factor of the displayed tree.



4. Click on **Links**.

This dialog box appears.



5. You just need to check the classes you wish to display.
6. Click **OK** to validate and close the dialog box.

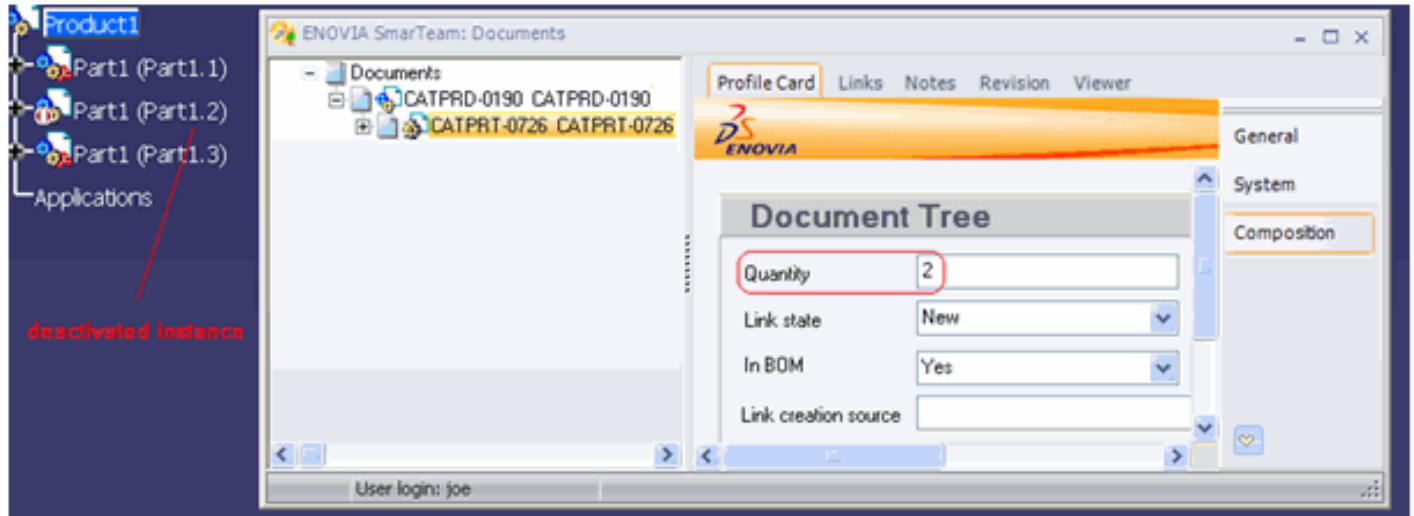


# Computing Quantity According to Instance Activity



When an assembly is saved in SmarTeam, it is useful to be able to obtain the quantity of a given component under its parent. This functionality provides SmarTeam with the activity information of each instance of a component under its parent. In this way, the quantity can be computed as the sum of all active instances of this component.

In the following example, the instance of Part1.2 is deactivated. The quantity of Part1 under Product1 is 2.



For details of how to implement this functionality, refer to [Save Documents](#)

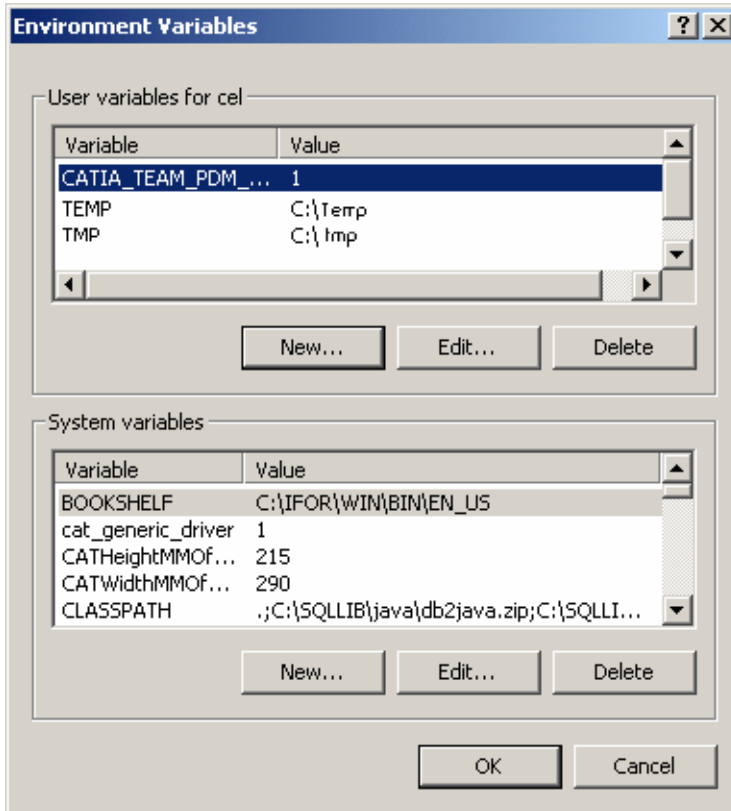


# System Variables for SmarTeam CATIA Integration



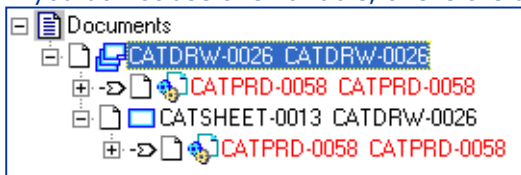
This page shows you how to set the appropriate variables:

1. Select **Start > Settings > Control Panel**.
2. From the **Control Panel** dialog box , double-click **System**.
3. From the **System Properties** dialog box that appears, double-click **System**.
4. Click the **Advanced** tab.
5. Click the **Environment Variables...** button.  
The Environment Variables dialog box is displayed.

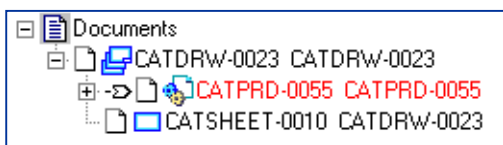


The following variable can be set in the Environment Variables of a user's workstation

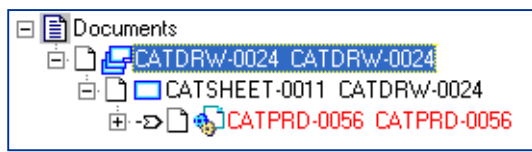
- **CATIA\_TEAM\_PDM\_DRLINK** = DrOnly/ShOnly  
Specifies how links are exposed in SmarTeam.  
If you do not use this variable, this is the default display:



- **DrOnly**: The links are exposed only at the document (CATIA Drawing) level.



- **ShOnly**: The links are only at the sheet level



For more information, see [Document Associations and Links](#).



# SmarTeam CATIA Integration Settings



This page provides the different settings available for working with SmarTeam CATIA Integration product.

- *Out Of Process*  
Sets CATIA in a process separate from SmarTeam. Must be set to YES.  
Values: YES/NO
- *Refresh Screen*  
If set to FALSE, the profile card of the document will not be displayed after:
  - the first SmarTeam Save operation performed from CATIA
  - Check in of Release operation  
Values: TRUE/FALSE
- *Add To Desktop*  
Specifies whether CATIA documents are placed on the Project Desktop after a Save operation.  
Values: YES/NO
- *Link To Main Class*  
Specifies whether CATIA documents are linked to the Project after a Save operation.  
Values: YES/NO
- *Batch Mode*  
Saves the components of an assembly without displaying each profile card.  
Values: TRUE/FALSE
- *Expose Mode*  
Exposes CATIA internal objects when saving an assembly or drawing inside the system. If set to YES, internal components and drawing sheets are saved in SmarTeam as distinct objects.  
Values: YES/NO  
  
**Note:** To work with CATIA.ExposeMode=YES, the user must select the *Enable CFO for Lifecycle operations* checkbox and select the *Force Checkout* option in Application Tools Setup. This is relevant when working with CATIA Product, Drawing and Sheet file types.
- *CATIA.SaveInstanceData*  
Exposes instance information (name + position matrix) when saving an assembly inside the system.  
Values: TRUE/FALSE



# Customizing the Open Dialog Window



The SmarTeam System Administrator can customize search grid attributes that appear in the SmarTeam Open Dialog Window.

**Important:** Customizing the search grid in the Open Dialog involves manual editing of the **smarteam.std.legacyPreferences.config.xml** file. To avoid damaging the original format of the file and rendering the file useless, it is strongly recommended that you make a backup copy of this file prior to editing it.



## Customizing the search grid

To customize the search grid:

- In the **smarteam.std.legacyPreferences.config.xml** file at the end of the **<runTimeKey>** group, under ... \SmarTeam\ConfigurationSetting\Data\Domain folder, add the following:

```
<Item xmlns="http://www.smarteam.com/dev/ns/config/legacyPreferences/1.0">
  <runTimeKey>QUERY_<integration>_<class>.<attribute name></runTimeKey>
  <value><condition>,<value>[;SHOW]</value>
</Item>
```

The name should be in the form **[QUERY\_<integration>\_<class>]** where **<class>** is one of the classes that the integration supports, as defined in the SmarTeam Integration Tools Setup utility.

where:

**<attribute name>** is the name of the SmarTeam attribute of the relevant class

**<integration>** is the name of the current integration, in this case CATIA

**<condition>** is one of the following:

- Starts with
- Ends with
- Includes

- Equals
- Greater than
- Less than
- Not equal to
- Less than or equal to
- Greater than or equal to

<value> is the value.

;SHOW is an optional parameter. If added, the attribute will be shown in the Open dialog as part of the query.

When a [QUERY\_<class>] key does not exist for a particular class, primary identifier attributes are displayed in the SmarTeam Open Dialog search grid.

**Note:** The fields in the query, as well as in the query results, will appear in alphabetical order. This default setting cannot be modified.

### Example

The following code can be used to define criteria for the Open dialog window:

```
<Item xmlns="http://www.smarteam.com/dev/ns/config/legacyPreferences/1.0">
  <runTimeKey>QUERY_CATIA_Part.0_TDM_ID</runTimeKey>
  <value>includes,;;show</value>
</Item>
```

```
<Item xmlns="http://www.smarteam.com/dev/ns/config/legacyPreferences/1.0">
  <runTimeKey>QUERY_CATIA_Part.1_TDM_DESCRIPTION</runTimeKey>
  <value>includes,;;show</value>
</Item>
```

```
<Item xmlns="http://www.smarteam.com/dev/ns/config/legacyPreferences/1.0">
  <runTimeKey>QUERY_CATIA_Part.2_REVISION</runTimeKey>
  <value>;SHOW</value>
</Item>
```

```
<Item xmlns="http://www.smarteam.com/dev/ns/config/legacyPreferences/1.0">  
  <runTimeKey>QUERY_CATIA_Part.3_STATE</runTimeKey>  
  <value>;SHOW</value>  
</Item>
```

```
<Item xmlns="http://www.smarteam.com/dev/ns/config/legacyPreferences/1.0">  
  <runTimeKey>QUERY_CATIA_Part.4_TDMX_RELATED_ITEM_ID</runTimeKey>  
  <value>includes,,;show</value>  
</Item>
```

```
<Item xmlns="http://www.smarteam.com/dev/ns/config/legacyPreferences/1.0">  
  <runTimeKey>QUERY_CATIA_Part.5_FILE_TYPE</runTimeKey>  
  <value>Equals,CATPart,;SHOW</value>  
</Item>
```

```
<Item xmlns="http://www.smarteam.com/dev/ns/config/legacyPreferences/1.0">  
  <runTimeKey>QUERY_CATIA_Part.6_CREATION_DATE</runTimeKey>  
  <value>;SHOW</value>  
</Item>
```

```
<Item xmlns="http://www.smarteam.com/dev/ns/config/legacyPreferences/1.0">  
  <runTimeKey>QUERY_CATIA_Part.7_USER_OBJECT_ID</runTimeKey>  
  <value>;SHOW</value>  
</Item>
```

```
<Item xmlns="http://www.smarteam.com/dev/ns/config/legacyPreferences/1.0">  
  <runTimeKey>QUERY_CATIA_Part.8_TDMX_DETAILED_DESCRIPTION</runTimeKey> <value>includes,,;  
  show</value>  
</Item>
```

Two of the resulting screens are shown below:

Open document



Search



Documents  
Desktop



Items  
Desktop

Class name:

CATIA Part

☐ Retrieve only last public revisions



Attributes	Conditions	Values
ID	Includes	
Description	Includes	
Revision		
State		
Related Item Number	Includes	
File Type	Equals to	CATPart
Creation Date		
User Object Id		

Preview

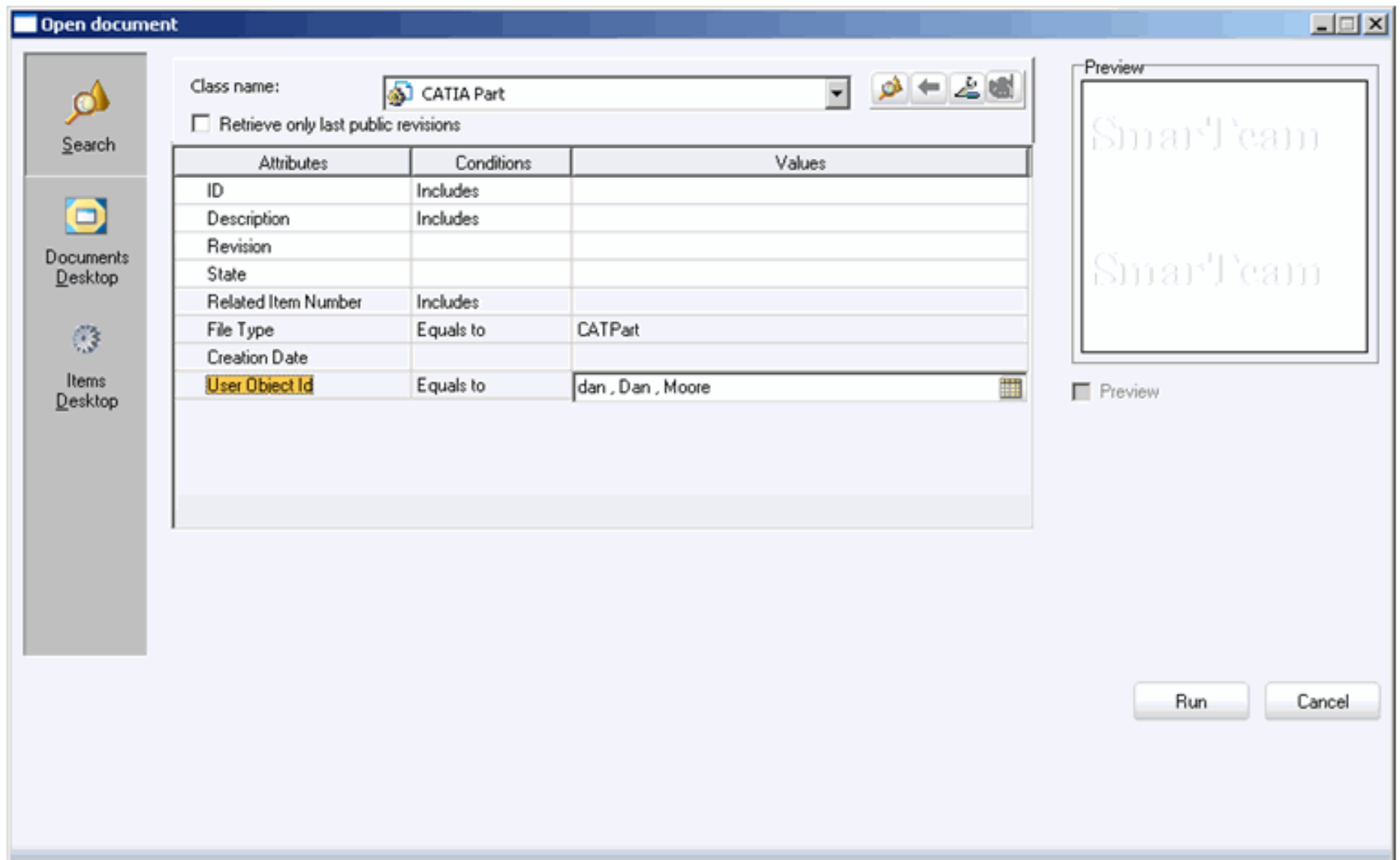
SmartTeam

SmartTeam

☐ Preview

Run

Cancel



## Example 2:

The following is an example of how to customize the order in which attributes will appear in the search grid of the SmarTeam Open Dialog.

In this example, the format `<runTimeKey>QUERY_<integration>_<class>.<number>_<attribute name></runTimeKey>` is used, where:

**<integration>** refers to the integration

**<class>** refers to a class supported by the integration

**<number>** refers to the sequential order of the attribute in the query

**<attribute name>** refers to the name of the SmarTeam attribute for the relevant class

```
<Item xmlns="http://www.smarteam.com/dev/ns/config/legacyPreferences/1.0">  
<runTimeKey>QUERY_SolidWorks_Drawing.0_TDM_ID</runTimeKey>
```

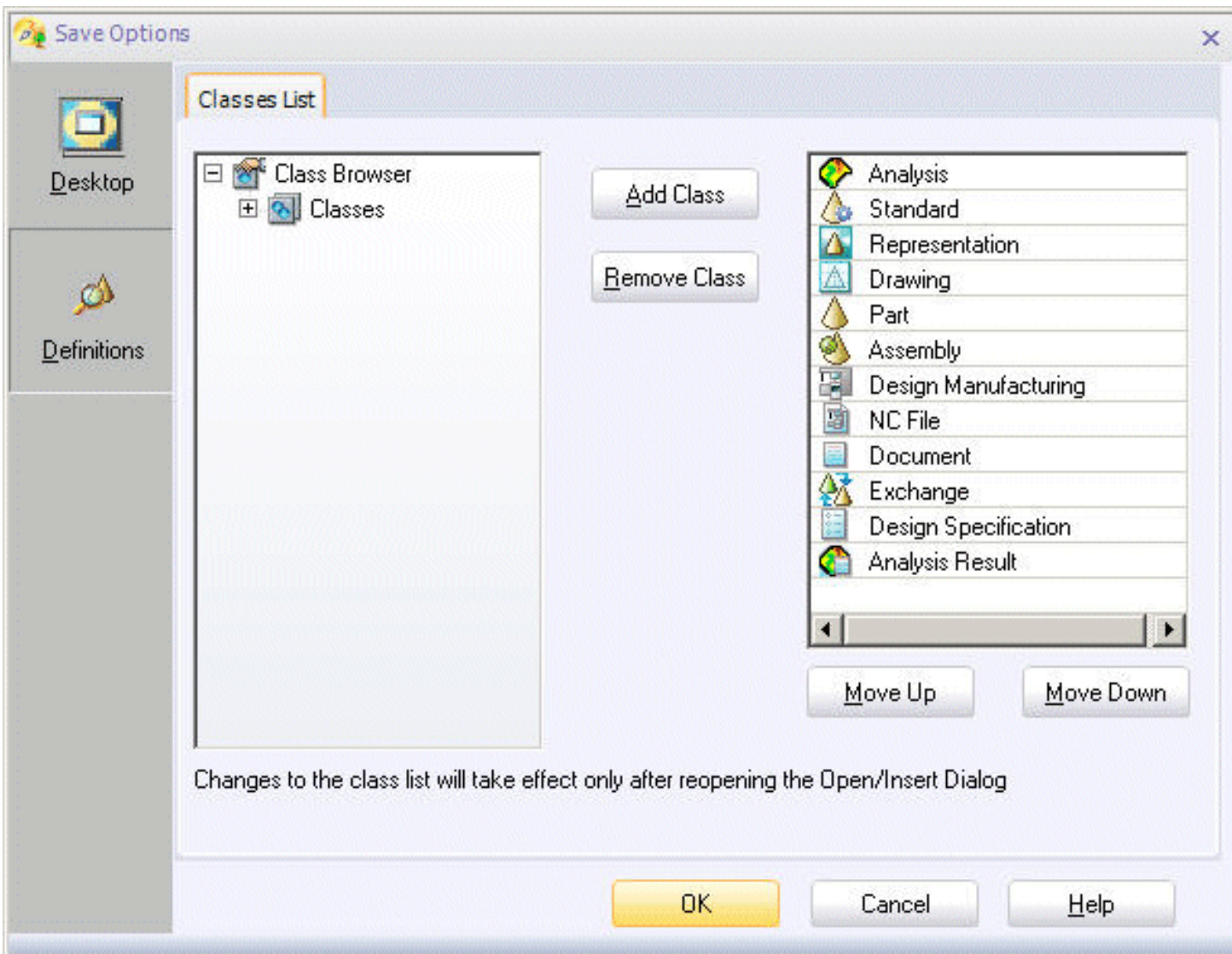
```
<value>;SHOW </value> </Item>
```

```
<Item xmlns="http://www.smarteam.com/dev/ns/config/legacyPreferences/1.0">  
<runTimeKey>QUERY_SolidWorks_Drawing.1_REVISION</runTimeKey> <value>Equals,123</value> </  
Item> <Item xmlns="http://www.smarteam.com/dev/ns/config/legacyPreferences/1.0">  
<runTimeKey>QUERY_SolidWorks_Drawing.2_CN_TITLE</runTimeKey> <value>;SHOW</value> </Item>
```



## Customizing the class name list

The administrator may customize the class name list to define which classes will be displayed in the list. In addition, the administrator defines which attributes appear for each class.

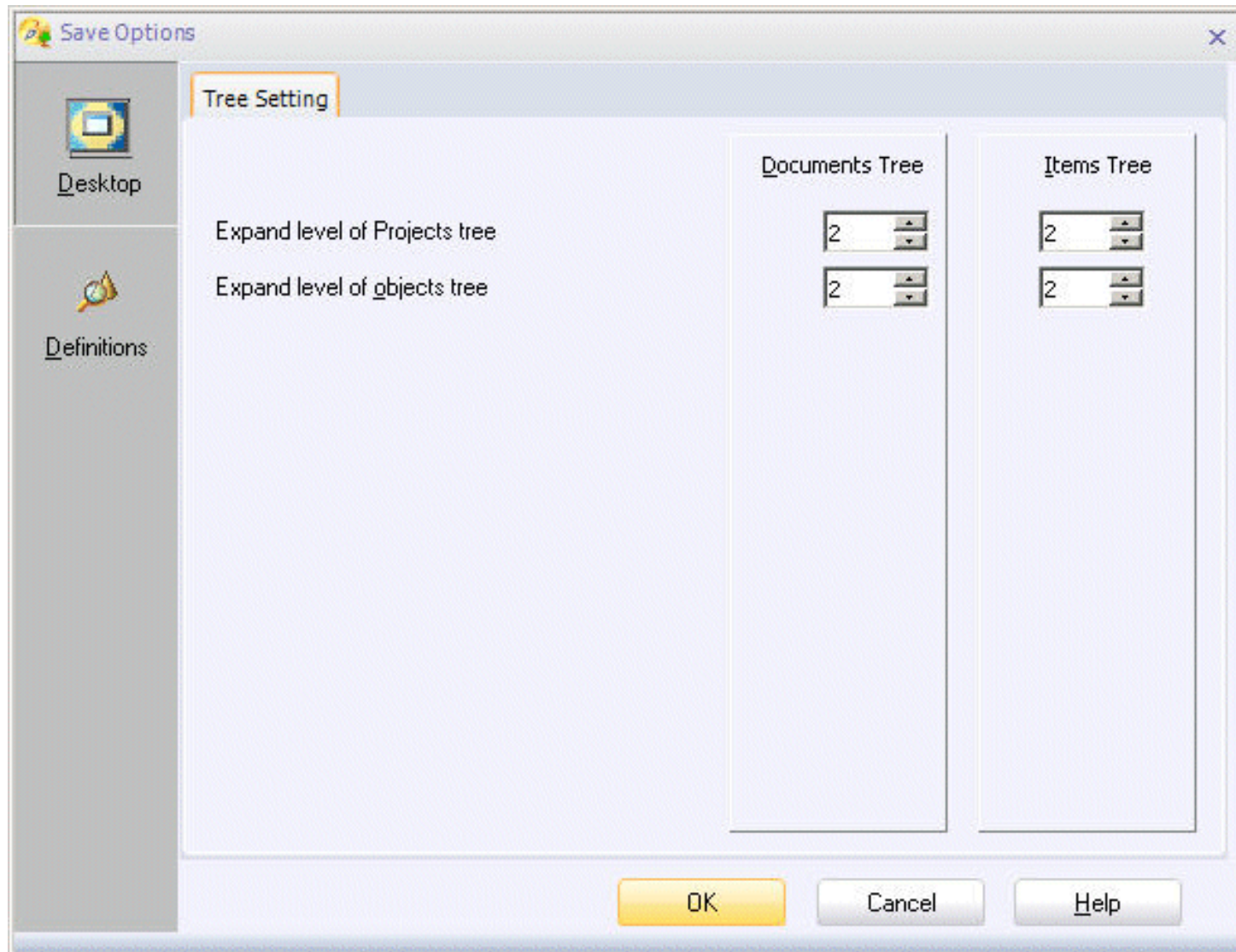




## Customizing tree settings

Define the expand levels of the Projects tree and objects tree for Documents and for Items.

From the **Save Options** dialog, select the **Desktop** option and then the **Tree Setting** tab. Using the arrows in the spin boxes, select the Expand level for each tree and then click OK to save the setting.



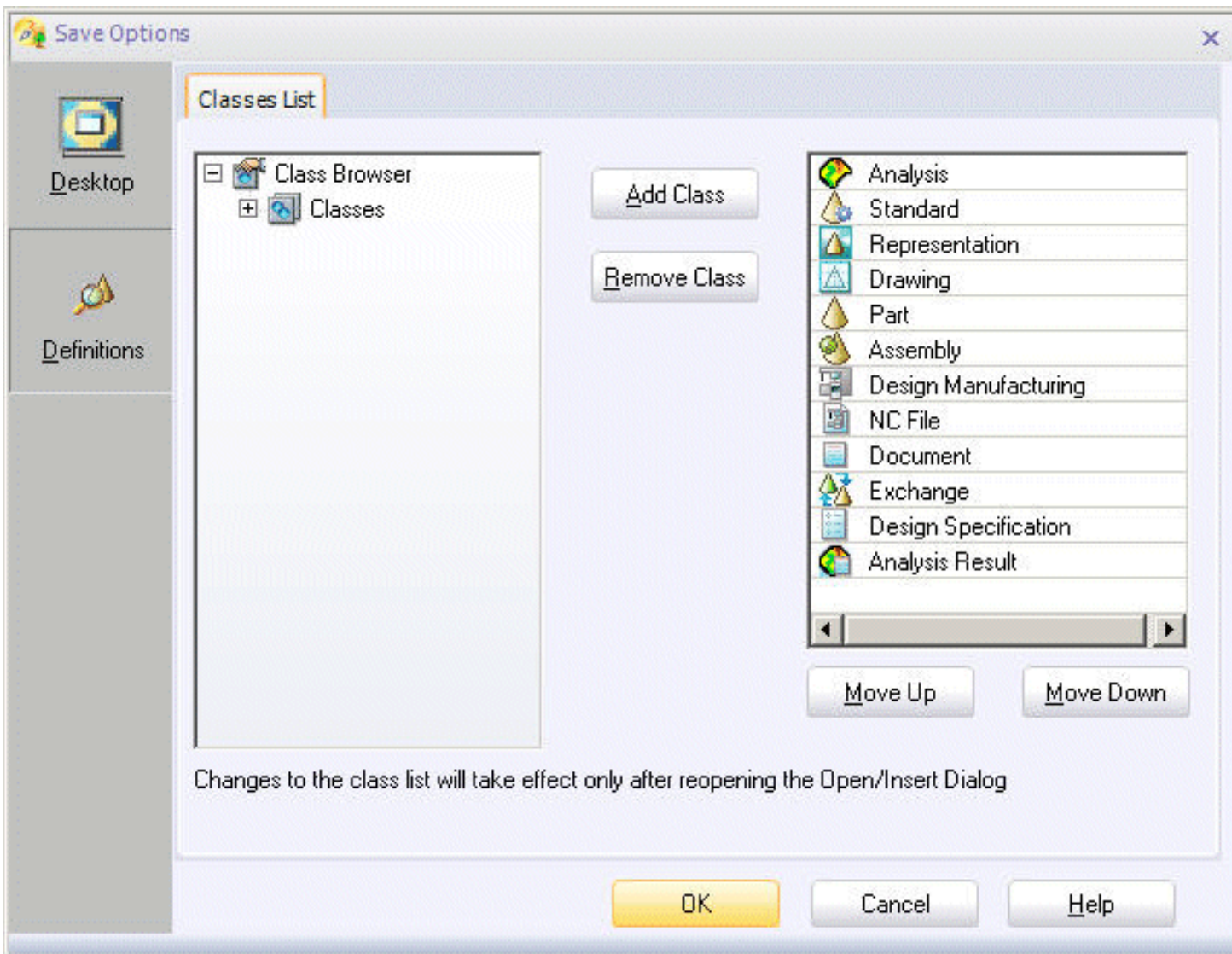


## Customizing the Classes List

Select the classes which will appear in the Classes listbox.

From the **Save Options** dialog, select the **Definitions** option and then the **Classes List** tab. From the right pane, select a class or classes and then click the **Add Class** button. The selected classes appear in the left pane under the Classes category.

To remove a class, select it in the left pane and click **Remove Class**.





## SmarTeam Query Set as Current

If **SmarTeam Query** is set as **Current** in the **Document Environment** list, a SmarTeam query user interface is displayed:

Document Environment	State	
Folder	Allowed	
DLName	Allowed	
SmarTeam Query	Current	
SmarTeam	Allowed	



# Customizing Bulk Loading



The **Bulk Loading** command enables massive imports of documents into SmarTeam. When the **Batch Mode Save** option is activated, no profile card is displayed to the user and the default class is used to save the data.

In order to set the classes to be used during the Bulk Loading operation, you should add/modify dedicated SmarTeam options using the [SmarTeam System Configurator](#) tool.

The syntax of the key is `Bulk_Loading_CATIAFileType`. The value of the key is the class to use for Bulk Loading in batch mode save. Ex: `Bulk_Loading_CATPart="Standard CATIA Part"` entry in SmTeam32 will set the default class for CATPart documents to Standard CATIA Part class.

To activate this development, the following environment variable:  
`CATIA_TEAM_PDM_FORCED_CLASS_NAMES_BULK_LOADING`  
should be set to 1.

This functionality is useful when saving a large amount of documents of one given file type. When saving lots of documents of various CATIA file types, the rule to save the objects will follow the SmTeam32 information. It is up to the user to check that the targetted class is consistent. When saving Drawing and a Part (Drawing>Part), the part may be saved to Standard CATIA Part. Thus, errors may appear in some cases.



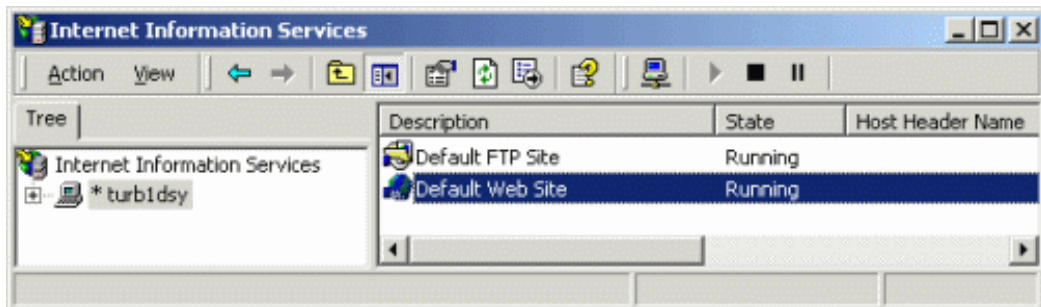
For more information about **Bulk Loading**, see [Importing CATIA Data Inside SmarTeam](#).



## SmarTeam System Configurator

In order to set the classes to be used during the Bulk Loading operation, you should add/modify dedicated SmarTeam options using the SmarTeam System Configurator tool and proceeding as follows:

1. Prior to launching SmarTeam System Configurator, ensure that Default Web Site is running.



2. Now, launch SmarTeam System Configurator.
3. Enter 'Bulk' in the **Search** field.
4. Click **Search**.  
Many items are found as displayed below:

bulk Search ? Logout

Search Text: "bulk"

Configuration Set Name:	Key Name
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_CATPart
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_CATProduct
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_CATDrawing
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_CATShape
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_CATProcess
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_CATAnalysisResults
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_CATAnalysisComputations
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_CATAnalysisInput
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_CATAnalysis
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_CATMaterial
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_CATSystem
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_Sheet
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_catalog
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_NCGM
smarteam.std.legacypreferences.config	CATIA.Bulk_Loading_cgr

< 12 >

5. Select one item from the list. For instance, CATIA\_Bulk\_Loading\_CATPart.
6. Set the appropriate value: for example, choose Domain from the Override Level combo box and enter Standard CATIA Part in the Value field.

bulk Search ? Logout

Save Changes Cancel

Configuration Set Name:  
smarteam.std.legacypreferences.config

Key Name:  
CATIA.Bulk\_Loading\_CATPart

Override Level:  
Domain

Value:  
Standard CATIA Part

7. Click the Save Changes button to confirm the changes.  
Once saved, the dialog box looks like this:

bulk Search ? Logout

Configuration Set Name:  
smarteam.std.legacypreferences.config


Key Name:  
CATIA.Bulk\_Loading\_CATPart

Add Value Delete Value

	Value	Override Level	Override Form
<input type="checkbox"/>	Standard CATIA Part	Domain	

< 1 >

**Note:** the current Bulk Loading functionality does not allow you to select projects.

 For more information about the SmarTeam System Configurator, see the *SmarTeam documentation*.



# Customizing Design Copy

This section describes the different ways of customizing the Design Copy command. The topics discussed are:

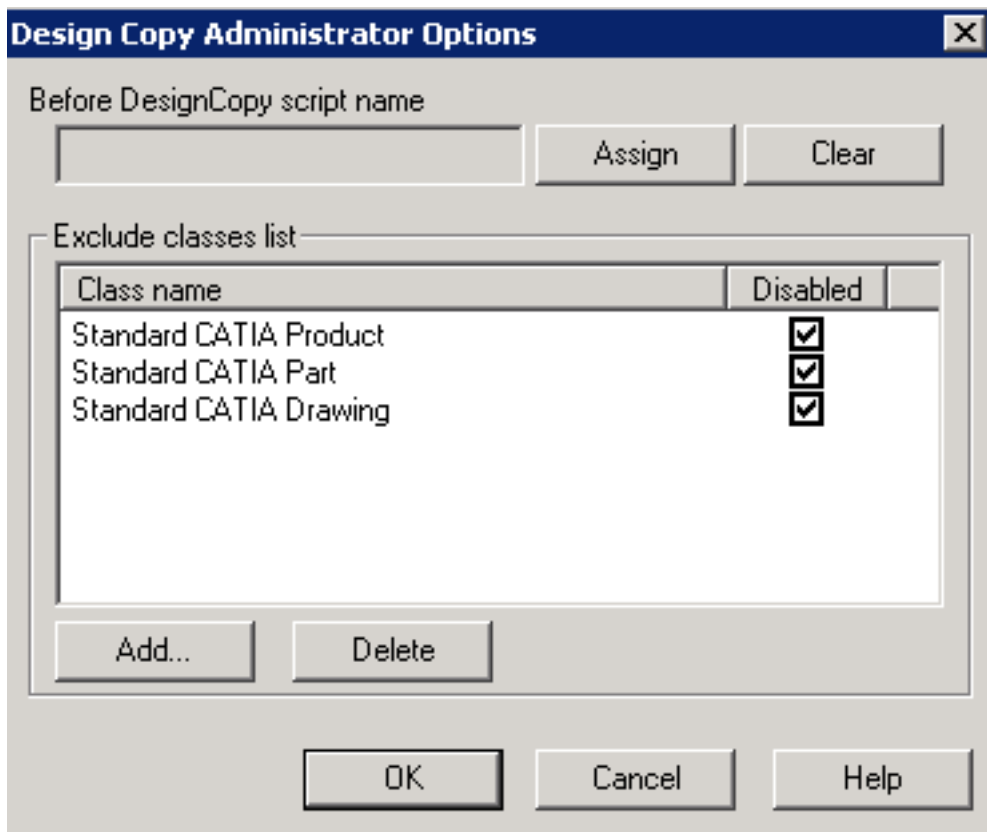
- [Design Copy Administrator Options](#)
- [Displaying CATIA links](#)
- [Reverse Links](#)



## Design Copy Administrator Options

The Design Copy Administrator Options enables you to customize the Design Copy window for the user. In addition, you can assign a script to run before the **Design Copy** window appears.

1. Customization of Design Copy is done through the **Design Copy Administrator Options** window. To display this window, click **Administrator Options** in the **Design Copy Options** window (for further details see [Duplicating an Existing Document](#)).



2. In the **Before DesignCopy script name** field click **Assign** to enter a script.  
This script will run before the Design Copy window appears.

3. Click **Add...** to include a class to the Exclude classes list.  
By default the added class is not checked. When checked the system disables the facility to allow the user to copy objects of a class. For example, if Standard CATIA Part is disabled (checked) then the user cannot create a new document based on Standard CATIA Part but only as is.
  - If it is checked in the **Design Copy Administrator Option Exclude Classes List**, then it is protected and not checked in the **Design Copy** window. You will not be able to change the status in the **Design Copy** window.
  - If it is not checked in the **Design Copy Administrator Option Exclude Classes List**, then it is not checked in the **Design Copy** window. You can change the status to checked in the **Design Copy** window.
4. Restart the Design Copy tool to apply the changes made to the Design Copy Administrator Options.

## Displaying CATIA links

When you launch the Design Copy, the **Design Copy** window displays all tree links, except for CATIA links. With CATIA links (both normal and reverse direction) only the CATIA links, which appear in the check out operation and where the **Propagation Allowed** check box is checked in the lifecycle rules setup, are displayed.

For example, every product needs to have a drawing in the Design Copy. To do this a rule needs to be defined for the Check Out Operation in the Lifecycle Rules Setup for CATIA Downstream Application link, reverse direction.

Note: If you have a rule but do not want it to be applied to the actual Check Out operation, then you must select **No operation** in the **Destination Operation** field of the SmarTeam **Lifecycle Rule Properties** window.

**Lifecycle Rule Properties**

Properties

Link Class: CATIA Downstream Applic...

Direction: Reverse

Original Class: CATIA Part

Destination Class: CATIA Drawing

Original Operation: Check Out

Destination Operation: No Operation

Link Operation: No operation

Switch to Latest: Allowed

☐ Propagation Allowed

☐ Check Destination Object Status

OK Cancel Help

## Reverse Links

For reverse links the structure of the object, which appears as a reverse link, is not displayed in the Design Copy.

It is highly recommended not to display the structure as it can bring additional objects and an unclear structure to the **Design Copy** window. If it is necessary to display the structure, then set the **DesignCopy.ShowHierarchicalLinksOfReverseDependencies** key to **Yes** in SmarTeam's System Configuration Editor.



# Enabling the Design Release Operation from SmarTeam



The SmarTeam – CATIA Integration can be customized to enable a design Release operation from SmarTeam.

.




Perform the customization by performing the following procedure:

- Install SmarTeam – Regulatory Compliance.
- Using the Data Model Designer (DMD), add the Electronic Signature and Job Server mechanisms to the database.
- Using Script Maintenance, add the function **CreateJobBasedOnFileType** to the Release operation *After* event.  
Script name: **RegulatoryCompliancy.ebs** .
- Click **Tools**.
- In the Admin Settings profile card, add the following values:
- In the **Section** field, enter Job Server
- In the **Subject** field, enter CATIA
- In the **First Value** field, enter INTERNAL
- In the **Second Value** field, enter LEAVE.
- In the **Fourth Value** field, enter the path of the Log folder.

All other fields remain empty.



# Using SmarTeam Scripts in a CATIA Session

 You can already adapt SmarTeam to your particular requirements by writing and delivering scripts that modify or replace existing functions or even create new ones. Typical uses would be to check data produced or generate data (for a BOM or an export operation for example).

Such scripts can be:


- attached to events in SmarTeam and be triggered before, after or instead of the action concerned
- placed in the Graphic User Interface (GUI):
  - by default, in the **Tools > User Defined Tools** menu item
  - anywhere in the GUI by means of the Menu Editor.

You can also launch as many as ten SmarTeam scripts from a **CATIA** session using the ten associated commands. There are already six sample scripts delivered in the form of ready-to-run samples (see the List of Sample Scripts below). These scripts as well as any others you decide to write must be assigned to one of the ten associated commands. You then only have to place the corresponding commands where it suits you best in the GUI.

This task is made up of the following stages:

- [Defining the SmarTeam Scripts to be Accessed from the CATIA Session](#)
- [Placing the Commands in the CATIA Graphic User Interface](#)
- [List of Sample Scripts](#)

## Defining the SmarTeam Scripts to be Accessed from the CATIA Session

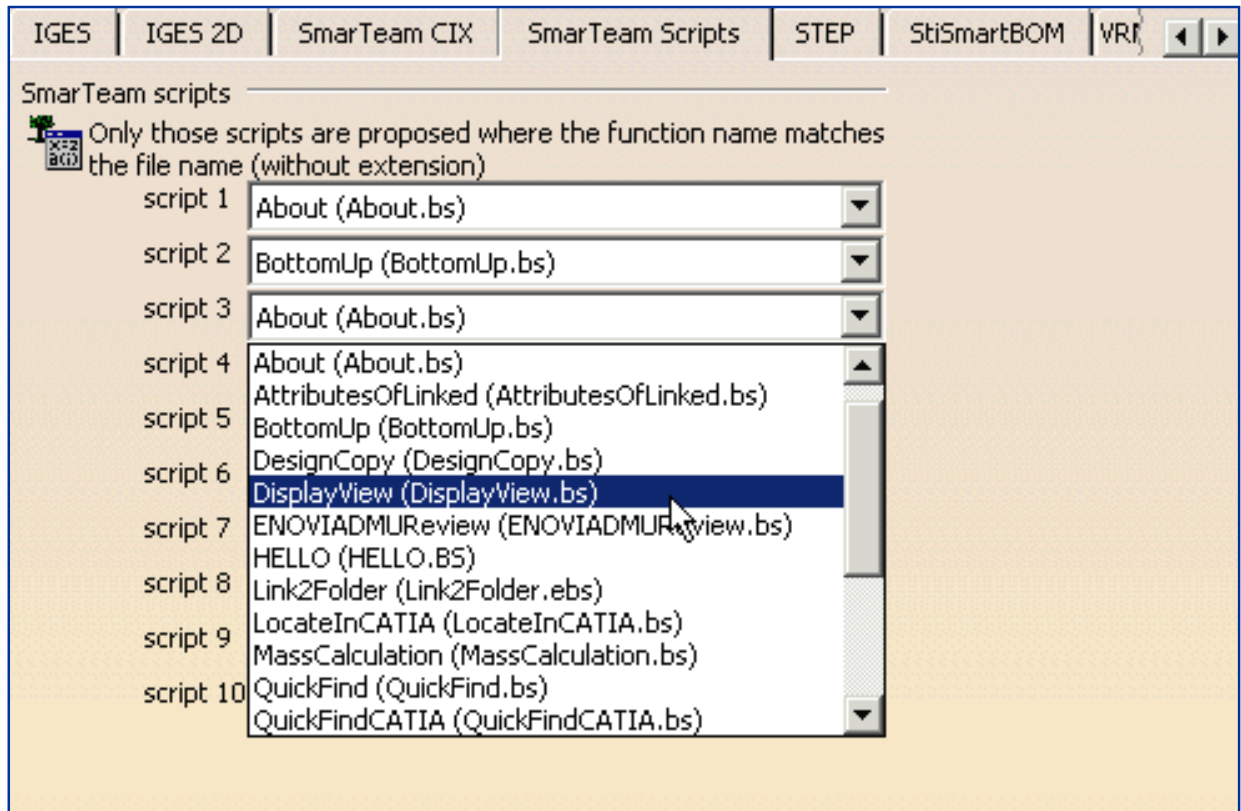
- 
1. Run a CATIA session.
  2. Select **Tools > Options....**  
The Options dialog box appears with the category tree in the left-hand column.
  3. In the **General** category of the Options tree, select **Compatibility**.
  4. Click on the **SmarTeam Scripts** tab.

Make sure you are connected to SmarTeam before accessing the Options dialog box.

Otherwise, an error message will appear prompting you to do so.


5. In the fields containing the command you want to use, open the combo box to display the list of scripts and select one of the scripts in the list to assign it to the command. In this way, you can associate one of ten scripts with each of the ten

commands available:

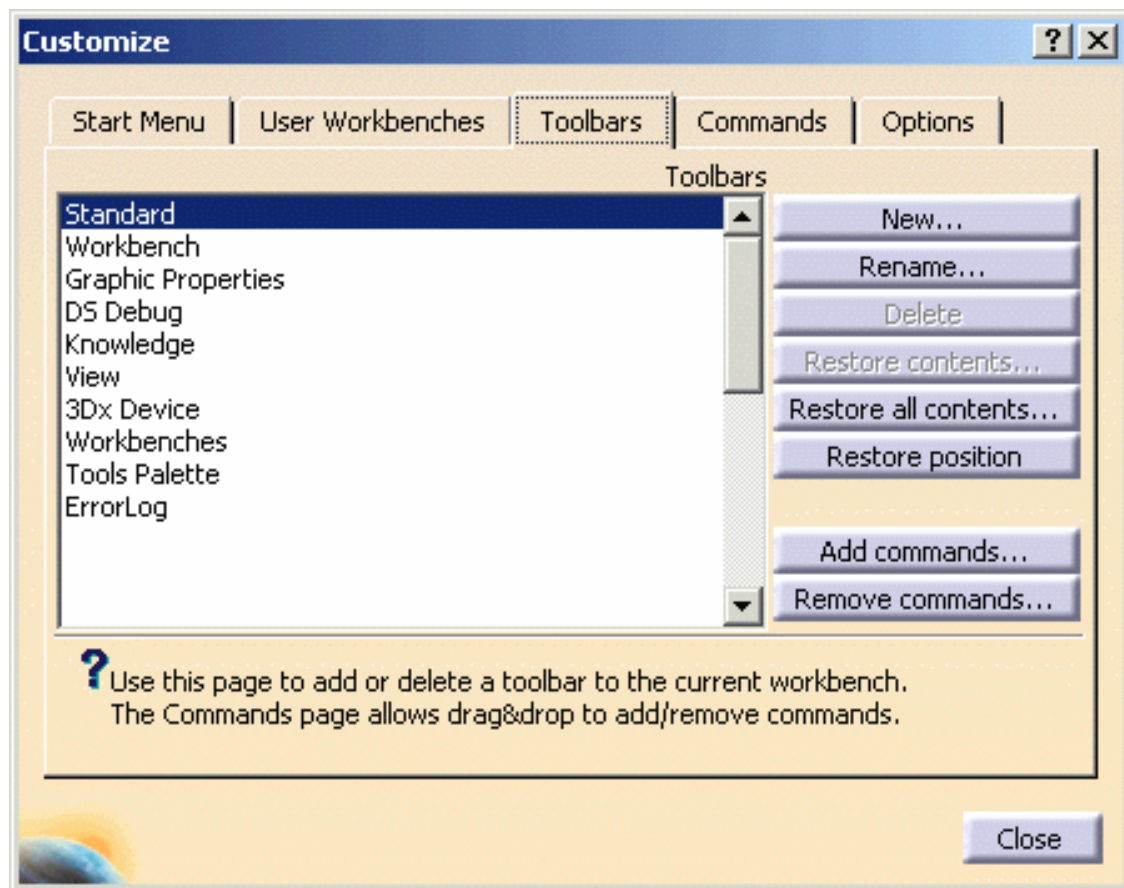


6. Click **OK** to confirm.

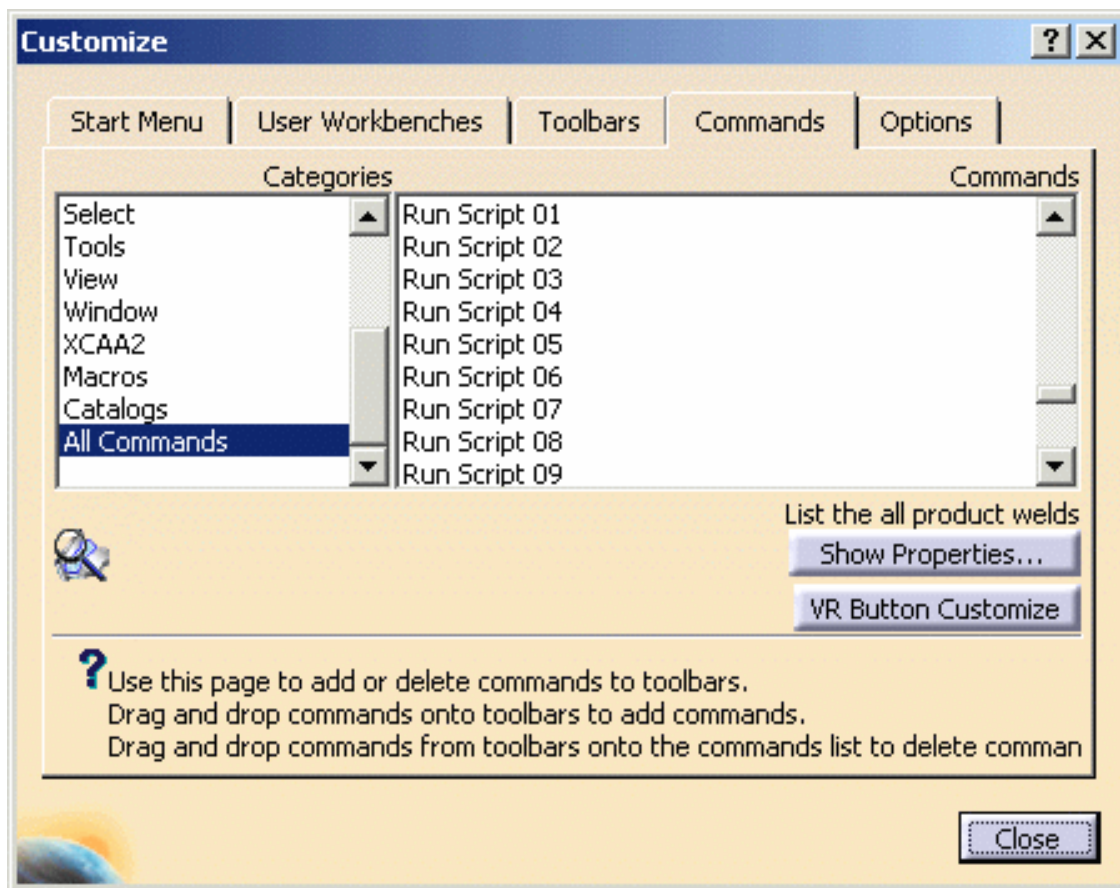
## Placing the Commands in the CATIA Graphic User Interface

 You must now decide where you want to put the SmarTeam commands in the CATIA GUI. You can, for example, create a SmarTeam toolbar containing the commands. To do this, do as follows:

1. Select **Tools > Customize...** or **View > Toolbars > Customize...**  
The **Customize** dialog box appears.
2. Select the **Toolbars** tab:



3. Click on the **New...** button to create a new toolbar (see "Managing User-Defined Toolbars" in the *CATIA - Infrastructure User's Guide*).
4. Click on the **Commands** tab and select the **All Commands** category on the left-hand side of the dialog box.
5. From the Commands list, find the ten SmarTeam commands:



6. Drag and drop the command(s) onto the toolbar you just created.

For more information on toolbar customization, see "Customizing a Toolbar by Dragging and Dropping" in the *CATIA - Infrastructure User's Guide*.

## List of Sample Scripts


- *About.bs*  
Provides information about the current database and current user.
- *AttributesOfLinked.bs*  
Retrieves attributes from a linked object (e.g. retrieves the part number of the documents displayed in the drawing sheets and updates the sheet profile card)
- *BottomUp.bs*  
Displays a bottom-up view of the selected object.
- *DisplayView.bs*  
Displays a stored view.
- *LocateInCATIA.bs*  
Highlights the document in the CATIA session, assuming that the object selected in the SmartTeams window is open in the CATIA session.
- *QuickFindCATIA.bs*  
Presents a quick find interface.
- *RevisionBlock.bs*  
Inserts required information in a revision block in the drawing.
- *SmartBox.bs*  
Opens the SmartBox window.

- *StartProcess.bs*  
Initiates a new process and attaches the current item to it.

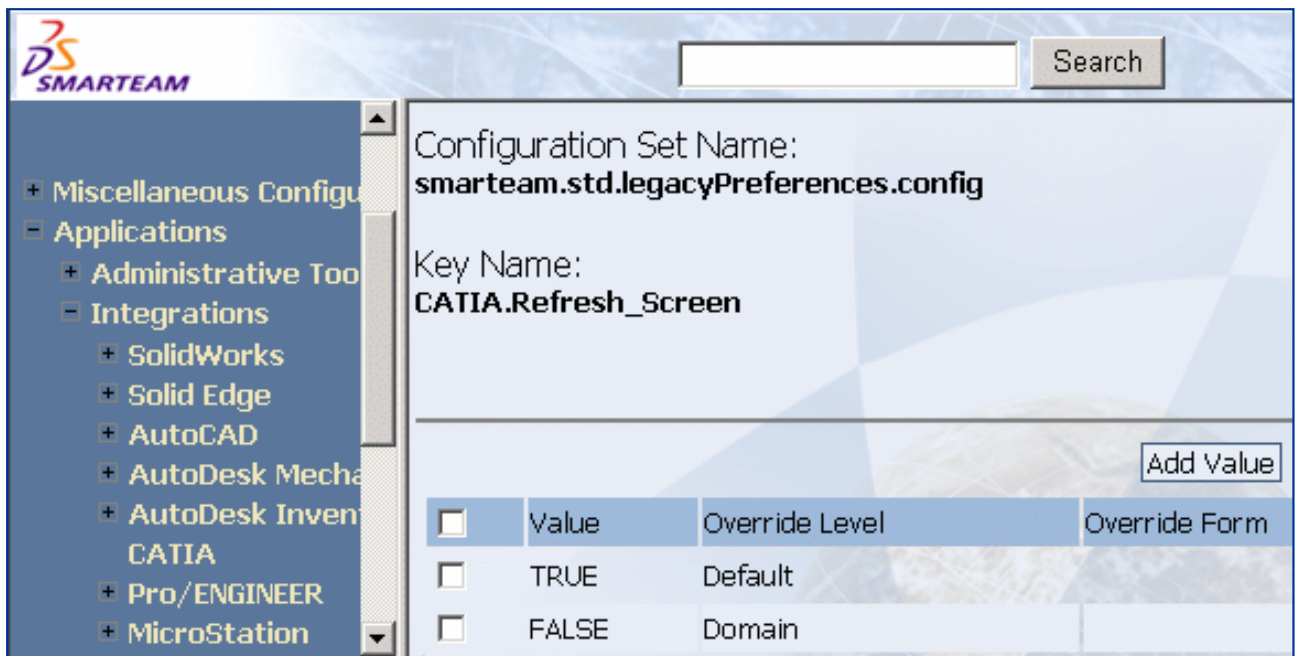


# Removing Profile Cards Display

The main function of Profile Cards being to describe the general properties of documents, most of the time end users do not need to display them when saving their documents for the first time. To make end users gain time, system administrators therefore can now prevent the application from displaying profile cards that appear during first save operations.

 To ensure that profile cards are not displayed during save operations, proceed as follows:

1. Launch **SmarTeam System Configurator**.  
To know how to use this tool, see the SmarTeam documentation.
2. Search for CATIA application.
3. Set **CATIA.Refresh\_Screen** to **FALSE**.
4. Click the **Save Changes** button to confirm the changes.  
Once saved, the dialog box looks like this:



The screenshot shows the SmarTeam System Configurator window. On the left is a tree view with the following structure:

- ± Miscellaneous Configuration
- ± Applications
  - ± Administrative Tools
  - ± Integrations
    - ± SolidWorks
    - ± Solid Edge
    - ± AutoCAD
    - ± Autodesk Mechanical Desktop
    - ± Autodesk Inventor
    - ± CATIA
    - ± Pro/ENGINEER
    - ± MicroStation

The main area displays the configuration for the selected key:

Configuration Set Name: **smarteam.std.legacyPreferences.config**

Key Name: **CATIA.Refresh\_Screen**


Below this is a table with columns: Value, Override Level, and Override Form. The first row shows the current value is TRUE with a Default override level. The second row shows the value FALSE with a Domain override level. An "Add Value" button is located to the right of the table.

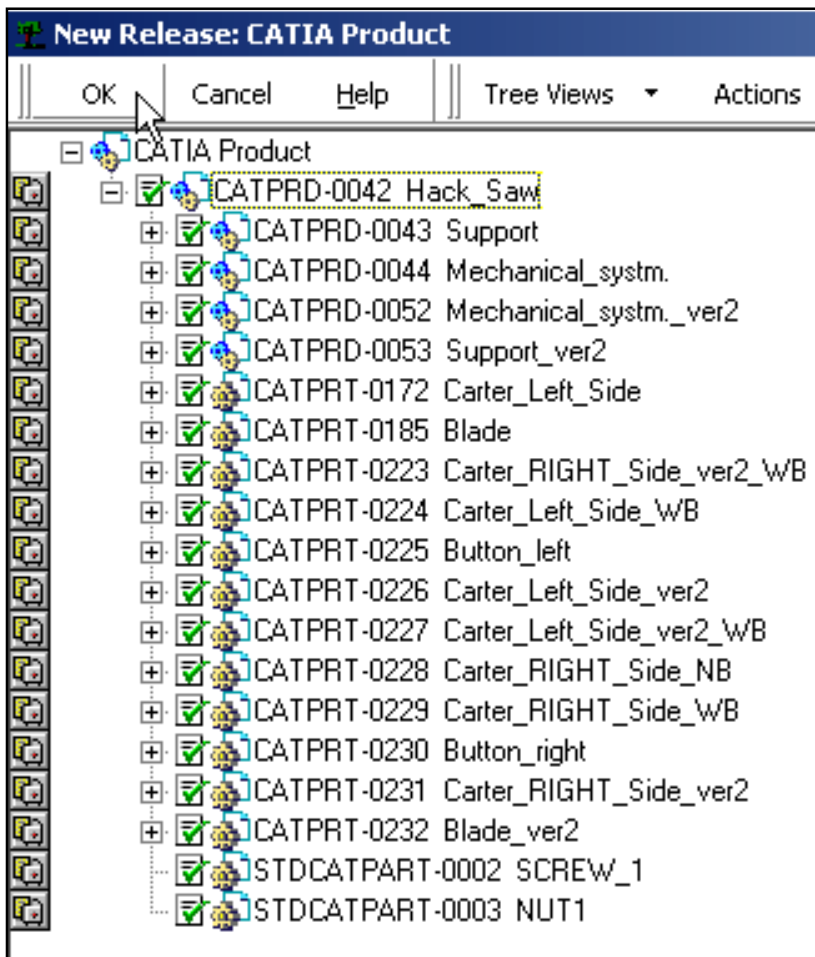
	Value	Override Level	Override Form
<input type="checkbox"/>	TRUE	Default	
<input type="checkbox"/>	FALSE	Domain	




# Simplifying Lifecycle Operations

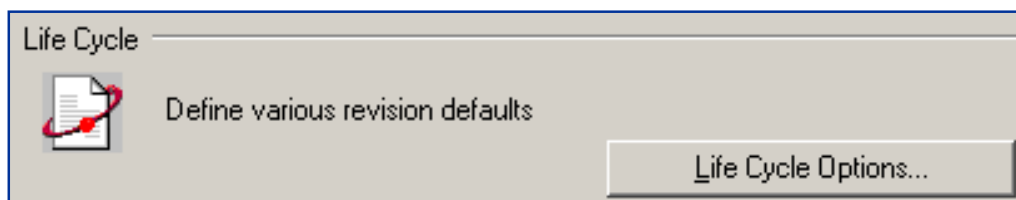
To make end users gain time, system administrators can now prevent the application from displaying all windows that appear during lifecycle operations (Check In, Check Out, Release, New Release, Undo Check Out) which needs to be performed in CATIA.

 For example, suppose you need to create a new release for CATPRD-0042. As you can see, this implies a certain number of interactions that would result in a tedious operation:



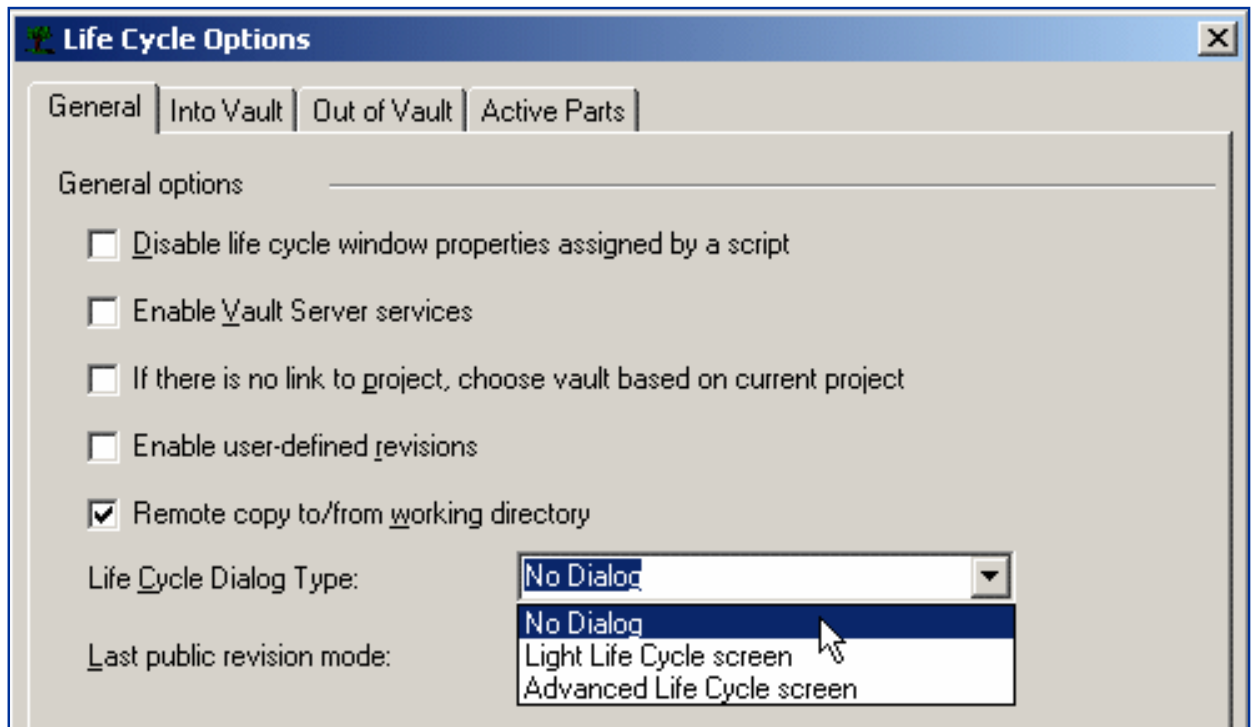
To eliminate the display of the different lifecycle windows, proceed as follows:

1. Click the  icon to switch to SmarTeam if not already done.
2. Select **Tools > Administrator Options...**
3. From the Administrator Options dialog box that appears, click **Life Cycle Options...**



4. From the Life Cycle Options dialog box now displayed, select **No Dialog** from the

drop-down list as the **Life Cycle Dialog Type** option.



5. Click **OK** to validate.

It is not necessary to restart SmarTeam services, nor to reconnect to SmarTeam to take this change into consideration.



# Enabling 2D Preview for 2D Products



This task shows you how to enable the preview of 2D elements making up CATProduct documents.

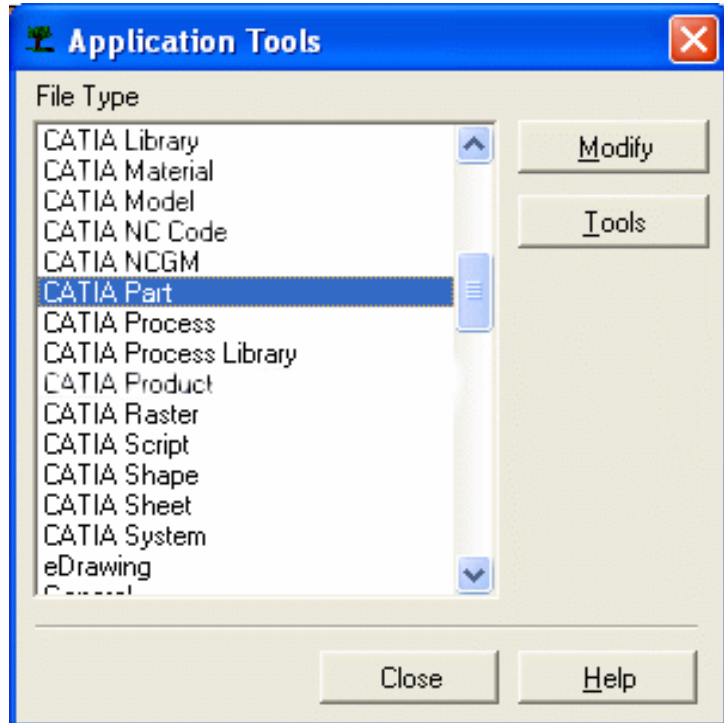
This operation is divided into three main steps:

1. Creating a dedicated SmarTeam file-type for 2D Products
2. Updating SetDesc.bs
3. Configuring CATIA so that 2D Products are saved using this file-type.

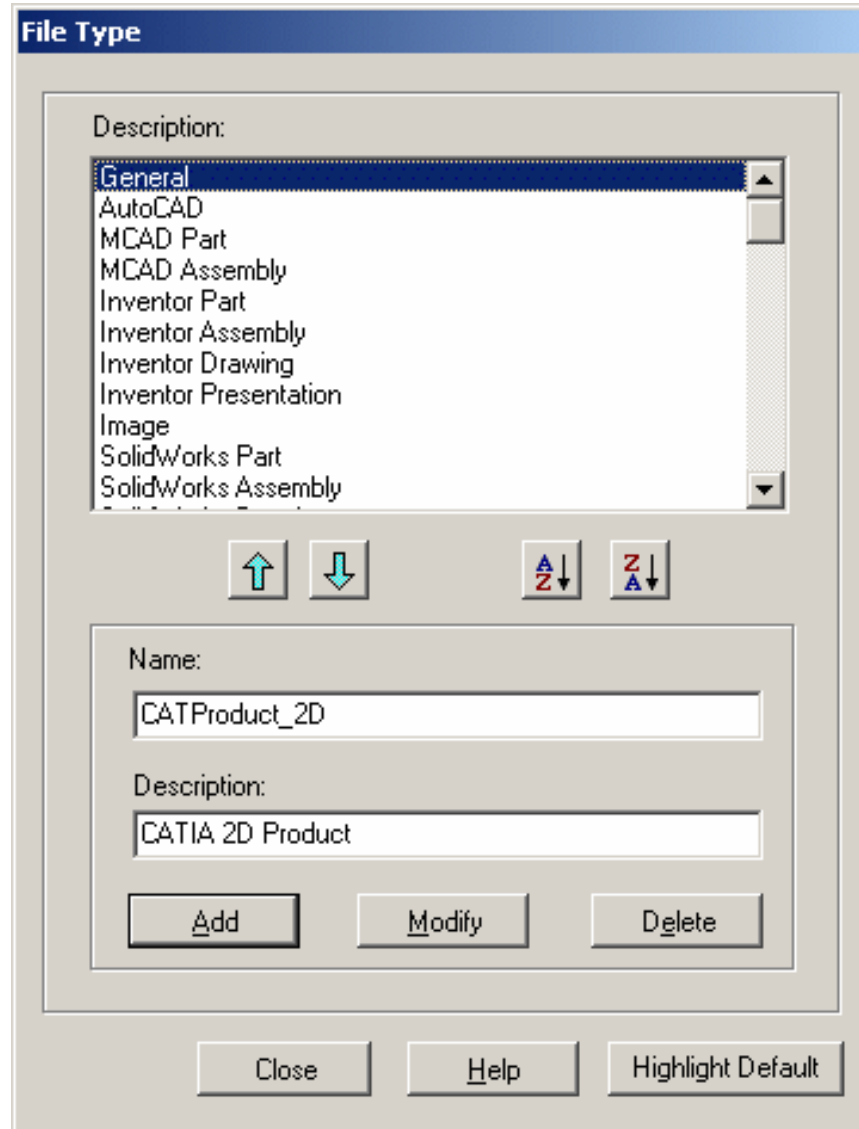


## Creating a Dedicated SmarTeam File-type for 2D Products

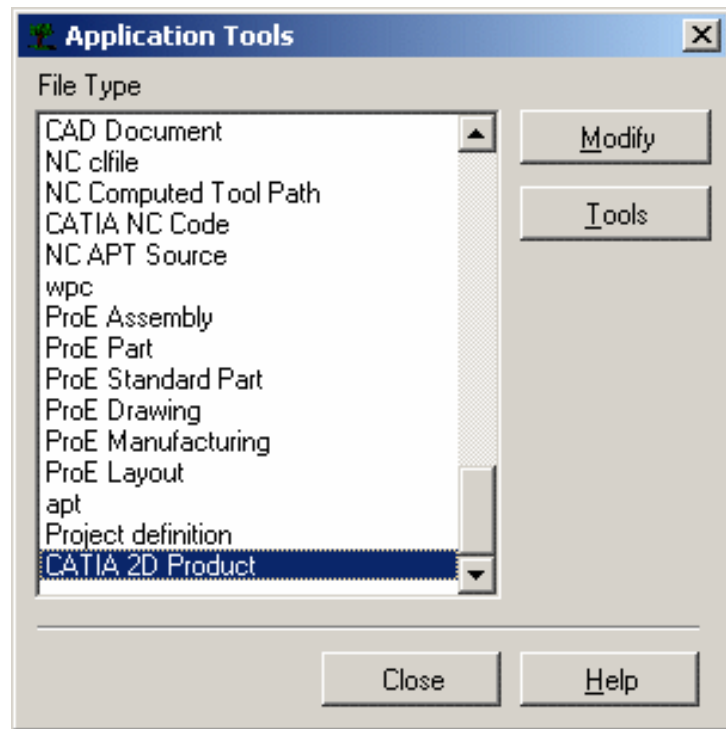
1. Select **SmarTeam > Tools > Applications Setup**.  
The **Applications Tools** window is displayed.



2. Click **Modify**.  
The **File Type** dialog box is displayed.
3. Enter **CATProduct\_2D** and **CATIA 2D Product** in the **Name** and **Description** fields respectively.

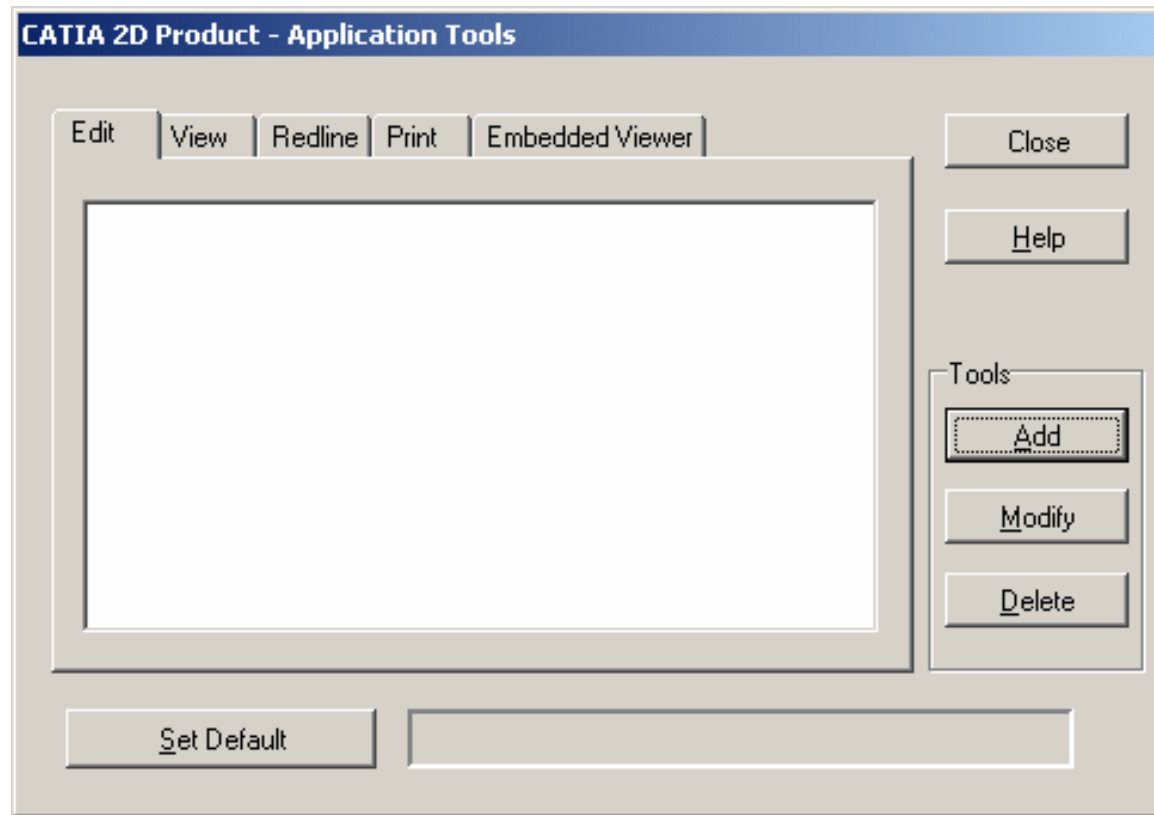


4. Click **Add** to confirm the operation.  
**CATProduct\_2D** is now included in the **File Type** field.
5. Click **Close** to close the **File Type** dialog box.
6. From the **Application Tools** dialog box, select **CATIA 2D Product**.



**7. Click Tools.**

The **CATIA 2D Product - Application Tools** dialog box is displayed. Each tab correspond to a capability. You need to provide information to ensure that the capability concerned works for the new file type that you have created.



8. Click **Add** to display the parameters for **Edit**.

The dialog box that appears includes two tabs. All the fields must be filled in using the information as shown in the following images.

## EDIT

General Setup tab

Advanced Setup tab

General Setup | Advanced Setup

Application tool:

Tool name

Original tool name

Command line

DDE command when not running

General Setup | Advanced Setup

☒ Search files in current folder

☒ Disable change of file name

☐ Place current object in the Global Data area

File extensions

Common-file objects (CFO)  
☒ Enable CFO for life cycle operations

Upon Check Out, do the following for other objects in CFO:

9. Click **OK** to confirm and close the dialog box.
10. Repeat this operation for **View**, **Print** and **Embedded View**.

## VIEW

General Setup tab

Advanced Setup tab

General Setup | Advanced Setup

Application tool:

Tool name:

Original tool name:

Command line:

Command line parameters:

Default directory:

☐ Switch to directory:

DDE application:  DDE topic:

DDE command when running:

DDE command when not running:

PRINT

General Setup tab

General Setup | Advanced Setup

☒ Copy CAD assembly files

☒ Disable change of file name

☐ Place current object in the Global Data area

File extensions:


Advanced Setup tab

General Setup | Advanced Setup


Application tool:


Tool name

Original tool name

Command line  
 

Command line parameters

Default directory  
 

☐ Switch to directory  
 

DDE application

DDE topic

DDE command when running

DDE command when not running

General Setup | Advanced Setup

☒ Copy CAD assembly files

☒ Disable change of file name

☐ Place current object in the Global Data area

File extensions

EMBEDDED VIEWER

The screenshot shows a configuration dialog box with two tabs: "Built-In Viewer" and "Plug-In Viewer". The "Plug-In Viewer" tab is selected. The dialog contains the following fields and options:

- Built-In viewers:** A section with a "Viewer type:" dropdown menu.
- Plug-In viewer:** A section with the following fields:
  - Name:** Smart IE Viewer
  - Class ID:** 156B79D3-C69C-11D4-B595-00104B3E8C2C
  - Event ID:** 156B79D1-C69C-11D4-B595-00104B3E8C2C
  - Event count:** 8 (with a spinner control)
- Viewer file extension:** .CATProduct.cgm
- Redline file extension:** .CATProduct.xml
- Options:**
  - ☒ Out of process
  - ☐ Copy CAD assembly files
  - ☒ Hide Toolbars

Buttons for "OK", "Cancel", and "Help" are located on the right side of the dialog.

## Updating SetDesc.bs

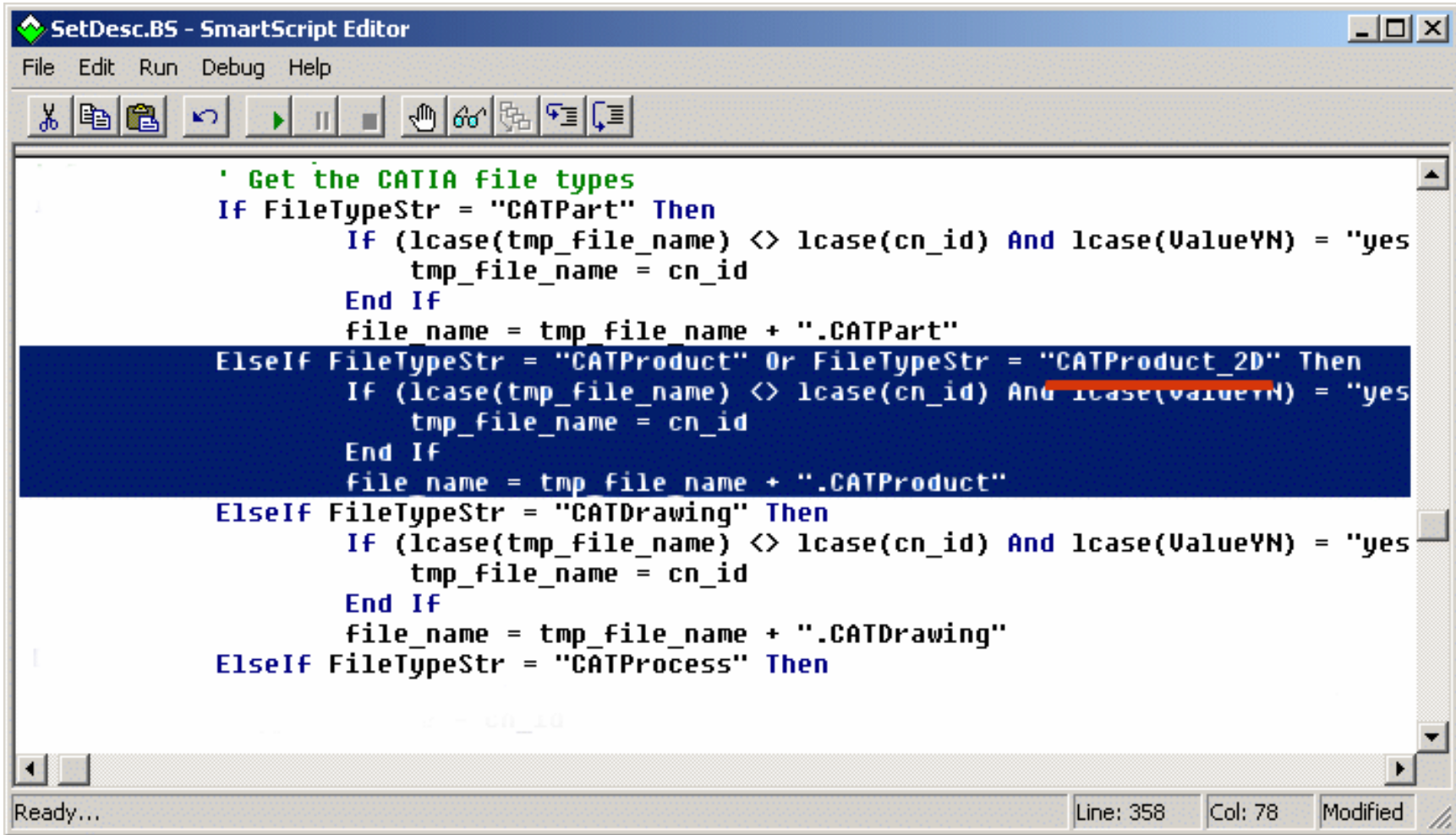
Update SetDesc.bs script to take this new file-type into account.

1. Open the SetDesc.BS. file using SmartScript Editor.  
As a reminder, by default this file is located in SmarTeam HOME/Script.
2. Search for "ElseIf FileTypeStr = "CATProduct" Then".
3. Replace:  

```
ElseIf FileTypeStr = "CATProduct" Then
```

 With:  

```
ElseIf FileTypeStr = "CATProduct" Or FileTypeStr = "CATProduct_2D" Then
```



4. Save the modified file.

# Configuring CATIA

Configure CATIA so that 2D Products are saved using this file-type. You must set this CATIA configuration for all CATIA seats.

1. Open the ClassBehaviors.txt file using the editor of your choice. By default this file is located in: CATIA HOME/OS\_name/reffiles/smarteam
2. Search for "catproduct\_2d".
3. Modify this line as follows: (take care when modifying the file: it is case sensitive).

Original line:

```
catproduct_2d                                ; CATProduct                                ; TDM_CATIA_PRODUCT
```

Modified line:

```
catproduct_2d                                ; CATProduct_2D                                ; TDM_CATIA_PRODUCT
```

4. Save the file.


## Notes

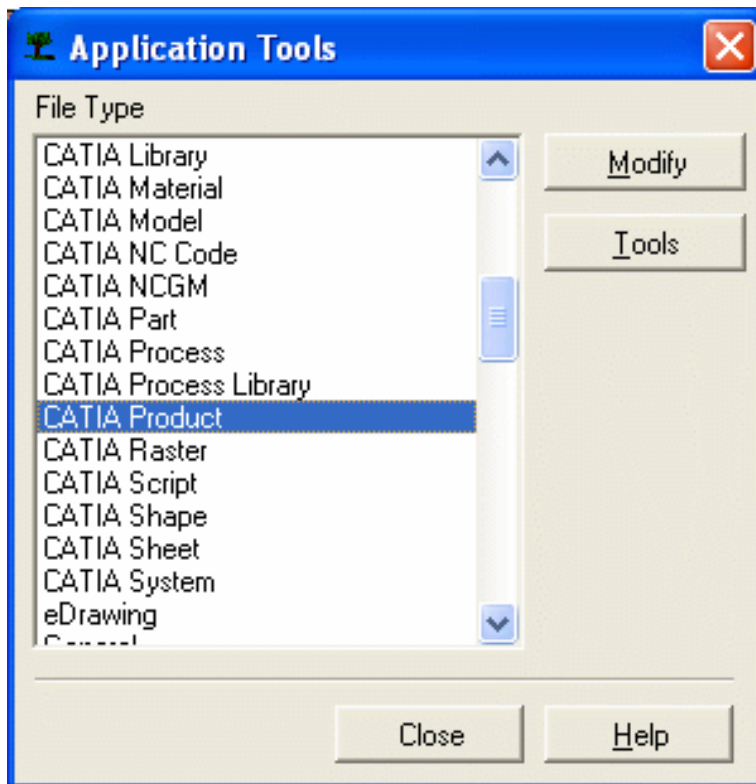
- To make this preview enhancement valid for 2D Products saved using releases prior to V5R16, you need to:
  - update manually their profile-cards so that their file-type is now 'CATIA 2D Product'.
  - save them again in SmarTeam so that a 2D preview is generated and replaces the regular 3D preview.
- This new preview type does not work if 2D Products (saved in SmarTeam) are not loaded in the CATIA session. The application then considers the CATProduct as a 3D Product.



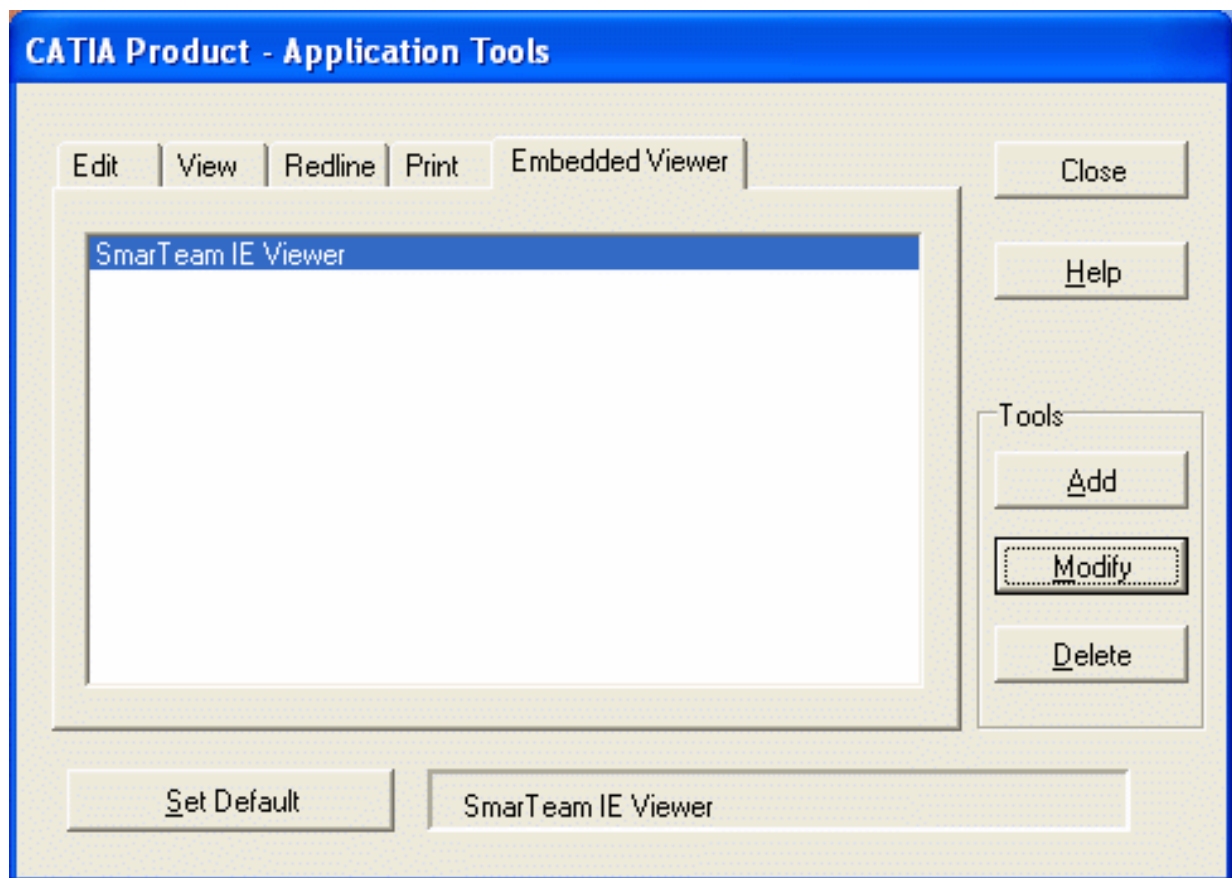
# Enabling 3DXML Format for CATProduct Documents

 This task shows you how to enable **3DXML** (EXtensible Markup Language) to preview CATProduct documents.

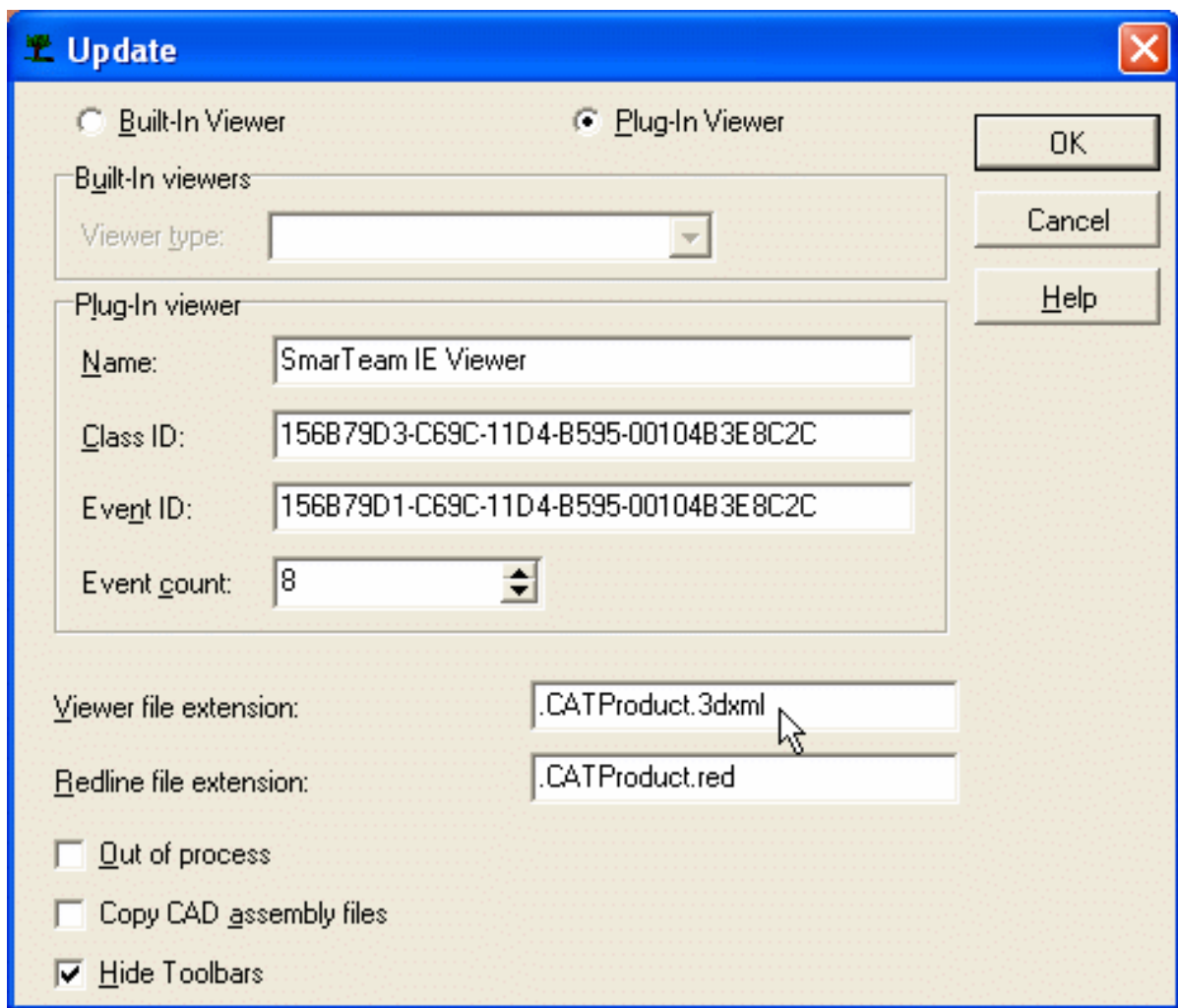
-  1. From SmarTeam, select **Tools > Applications Setup**.  
The **Application Tools** dialog box is displayed.
2. Select **CATIA Product**.



3. Click **Tools**.  
The **CATIA Product - Application Tools** dialog box is displayed.
4. Click the **Embedded Viewer** tab.



5. Click **Modify**.  
The **Update** dialog box is displayed.
6. Enter ".CATProduct.3dxml" in the **Viewer file extension** field:

The image shows a Windows-style dialog box titled "Update" with a blue header bar. It contains two radio buttons: "Built-In Viewer" (unselected) and "Plug-In Viewer" (selected). To the right of these are three buttons: "OK", "Cancel", and "Help". The "Built-In viewers" section has a "Viewer type:" dropdown menu. The "Plug-In viewer" section contains four fields: "Name:" (SmarTeam IE Viewer), "Class ID:" (156B79D3-C69C-11D4-B595-00104B3E8C2C), "Event ID:" (156B79D1-C69C-11D4-B595-00104B3E8C2C), and "Event count:" (8). Below these are two text boxes for file extensions: "Viewer file extension:" (.CATProduct.3dxml) and "Redline file extension:" (.CATProduct.red). At the bottom are three checkboxes: "Out of process" (unchecked), "Copy CAD assembly files" (unchecked), and "Hide Toolbars" (checked).

**Update**

☐ Built-In Viewer ☒ Plug-In Viewer

OK

Cancel

Help

Built-In viewers

Viewer type: [dropdown]

Plug-In viewer

Name: SmarTeam IE Viewer

Class ID: 156B79D3-C69C-11D4-B595-00104B3E8C2C

Event ID: 156B79D1-C69C-11D4-B595-00104B3E8C2C

Event count: 8

Viewer file extension: .CATProduct.3dxml

Redline file extension: .CATProduct.red

☐ Out of process

☐ Copy CAD assembly files

☒ Hide Toolbars

7. Click **OK** to validate.


The next time you save products in SmarTeam, the 3DXML file is generated.

## Pre-Version 5 Release 17 Files


It is not possible to convert the files stored in SmarTeam with versions before V5R17 into 3D XML files.

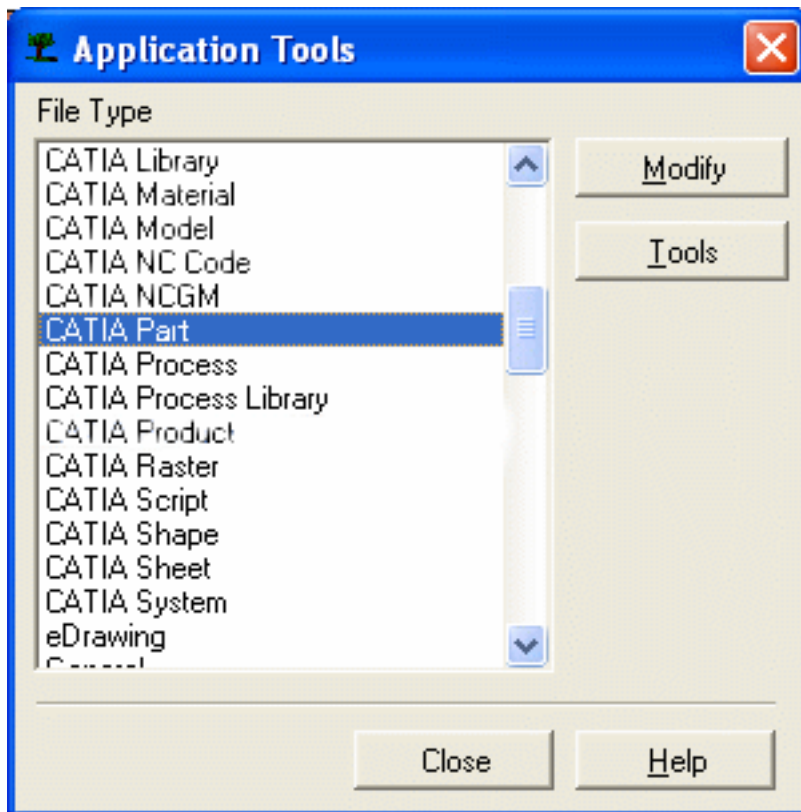


# Defining the Light Weight Mode for CATPart Documents

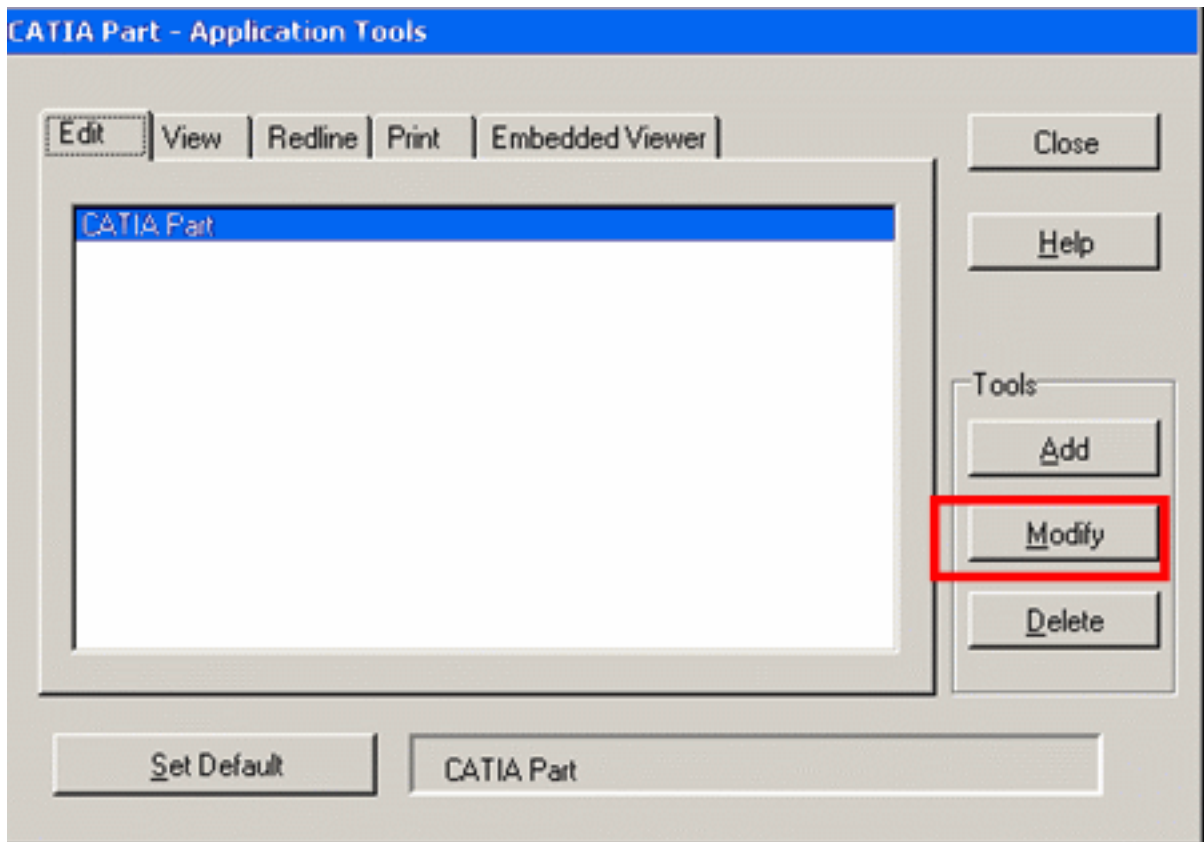
 The SmarTeam **Light Weight Mode** capability allows you to extract CATPart documents in CGR format from the vault. This capability proves useful especially when you are loading CATProduct documents and you do not want to load all related CATPart documents. As a reminder, this extraction type is to be used in visualization mode. For more information, see [Using Tools for Building Assemblies](#).

This task shows you how to set the SmarTeam **Light Weight Mode** for CATPart documents. For reference information on this capability, see the SmarTeam reference documentation.

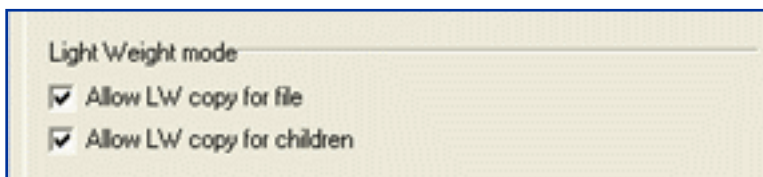
-  1. In SmarTeam, select **SmarTeam > Tools > Applications Setup**. The **Applications Tools** window is displayed.
2. Select **CATIA Part** as the file type to which you want to assign the **Light Weight Mode**.



3. Click **Tools**.
4. In the **Edit** tab of the dialog box that appears, click **Modify**.



5. The Update dialog box is displayed. Click the **Advanced Setup** tab.
6. In the **Light Weight mode** frame:
  - for CATPart documents, select **Allow LW (Light Weight) copy for file**,
  - for CATProduct documents, select **Allow LW (Light Weight) copy for children**.



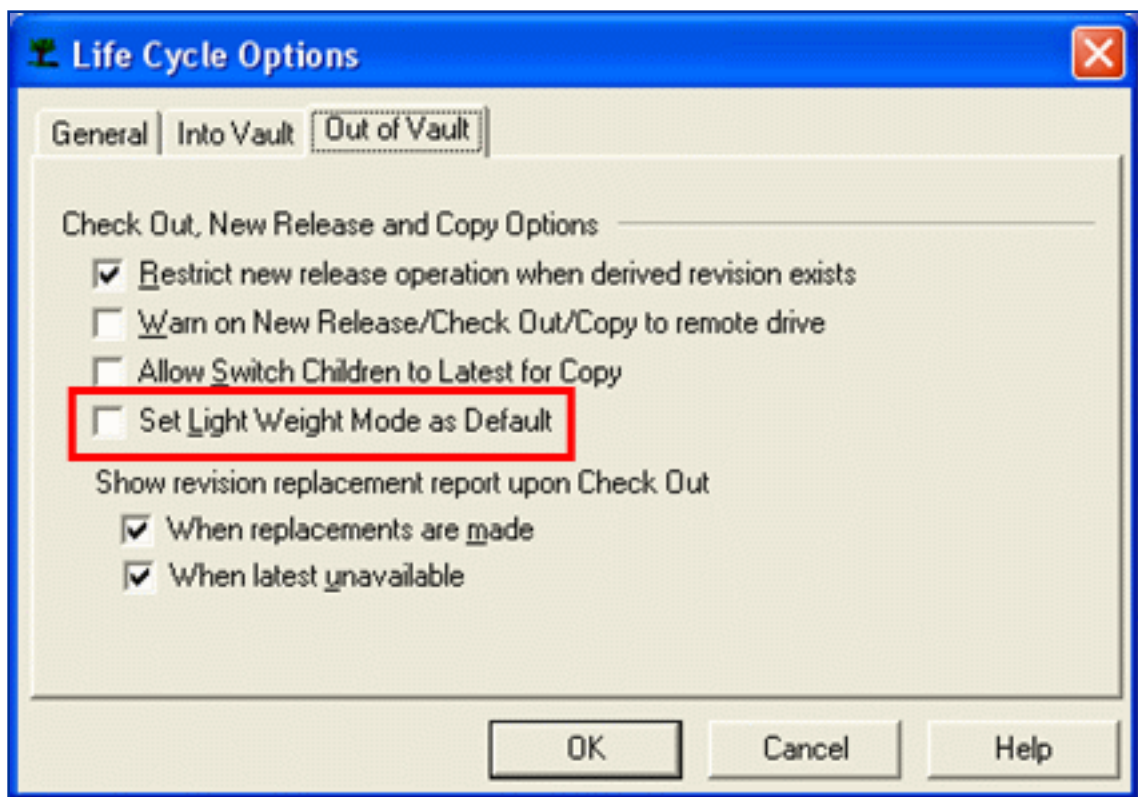
7. Click **OK** when done and close the **Update** dialog box.  
Next time any user perform an open operation, the **Use Light Weight Mode** option will be available in the **Open** dialog boxes.



## Defining Light Weight Mode as the Default Mode

In case you want to ensure that the **Use Light Weight Mode** option is the default option, you need to perform the following additional steps:

1. In SmarTeam, select **Administrator Options > Lifecycle Options**. The **Lifecycle Options** window is displayed.
2. Select the **Out of Vault** tab.
3. Select **Set Light Weight Mode as Default**.



4. Click **OK** when done and close the **Lifecycle Options** dialog box. During Copy operations, the light weight mode will be used by default for all CATPart documents marked for copy.



# Enabling Multi-Sheet CATDrawing Documents Preview



You can now customize the **CGM** (Computer Graphics Metafile) format used by the SmarTeam viewer to view multi-sheet drawings.

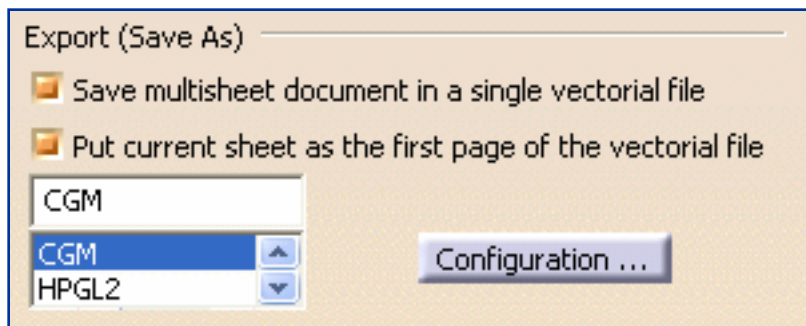
This task shows you how to enable the preview of the different sheets of a CATDrawing document in the SmarTeam viewer.



This capability requires that SmarTeam Release 17 is installed.



1. In CATIA select **Tools > Options**.  
The **Options** dialog box is displayed.
2. Choose the **General > Compatibility** category in the left-hand box.
3. Click the **Graphic Formats** tab.
4. Keep all default options.
5. Select **Save multisheet document in a single vectorial file**.



6. Click **OK** to validate.  
The next time you save drawings in SmarTeam, the customized CGM format is taken into account.

Once in the viewer, the new behavior is as follows:

- The default sheet displayed is the last saved sheet.
- Contextual commands (**Sheets > Sheet.x**) let you access the other sheets.
- The **Hide** or **Delete** additional commands allow you to hide a sheet.



# Data Model Modification for Assembly Variants



In order to work with the Assembly Variants functionality, changes must be made to the SmarTeam database as described in this section.

## Defining New Attributes for CATIA Product in the Data Model Designer

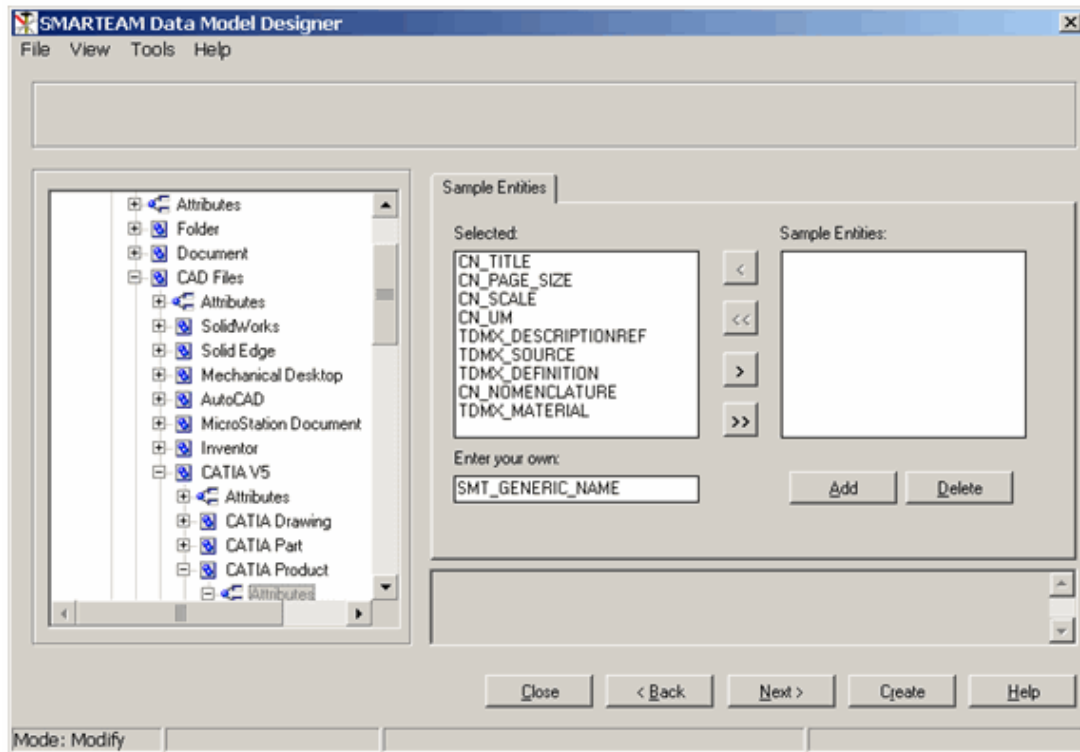
The following three new attributes must be defined:

- [SMT\\_GENERIC\\_NAME \(CHAR 128\)](#)
- [SMT\\_VARIANT\\_NAME \(memo\)](#)
- [SMT\\_VARIANT\\_COMMENT \(memo\)](#)

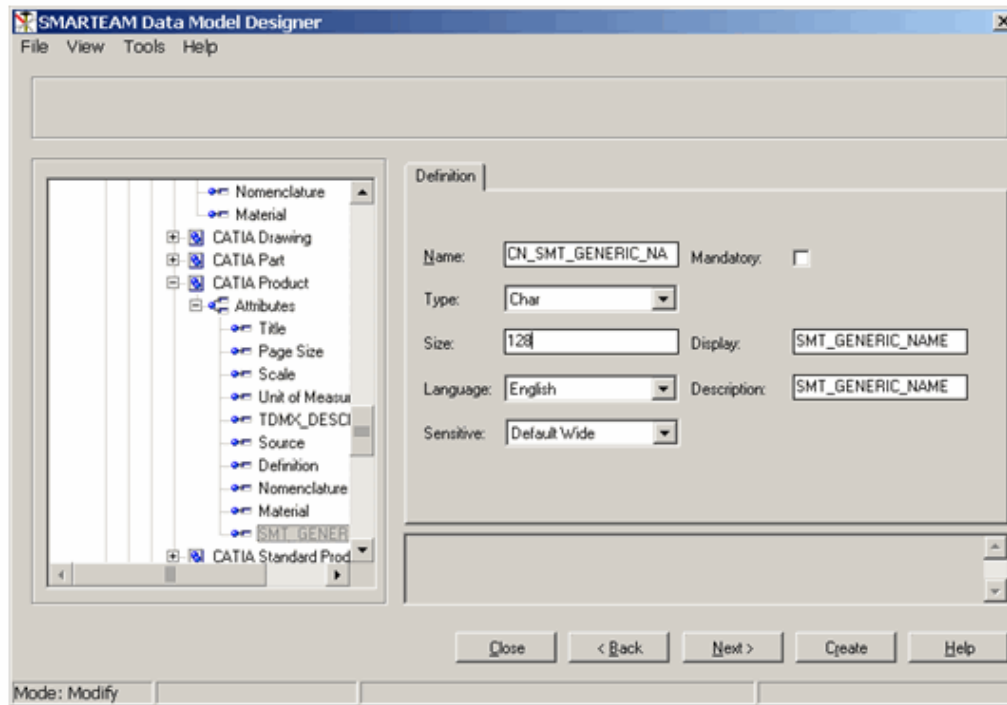


### SMT\_GENERIC\_NAME (CHAR 128)

1. Launch the SmarTeam Data Model Designer utility.
2. From the File menu, select **Modify Database Structure**
3. Select your database.
4. Create the **SMT\_GENERIC\_NAME** new attribute for the "CATIA Product" class.  
Note that **CN\_** is automatically added.

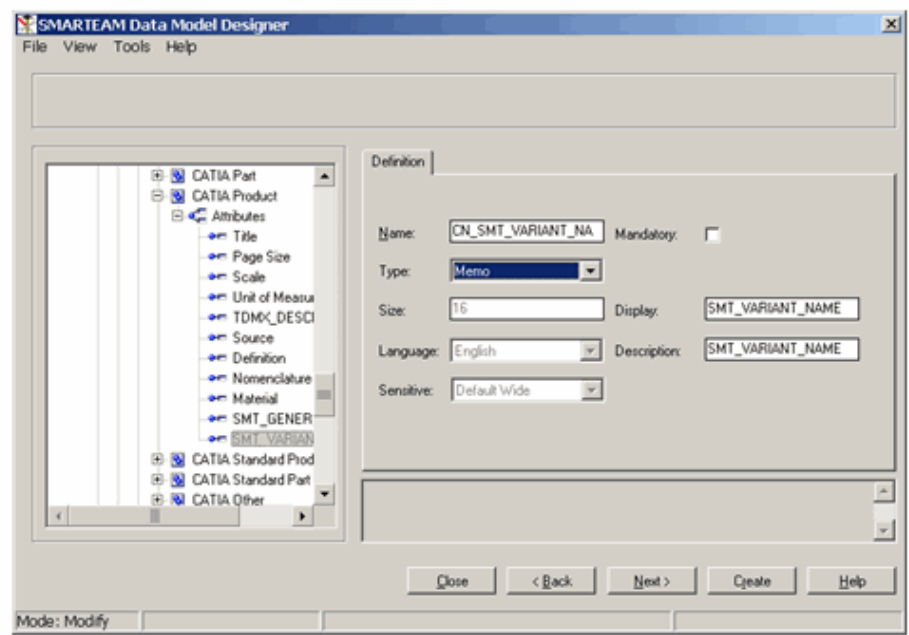
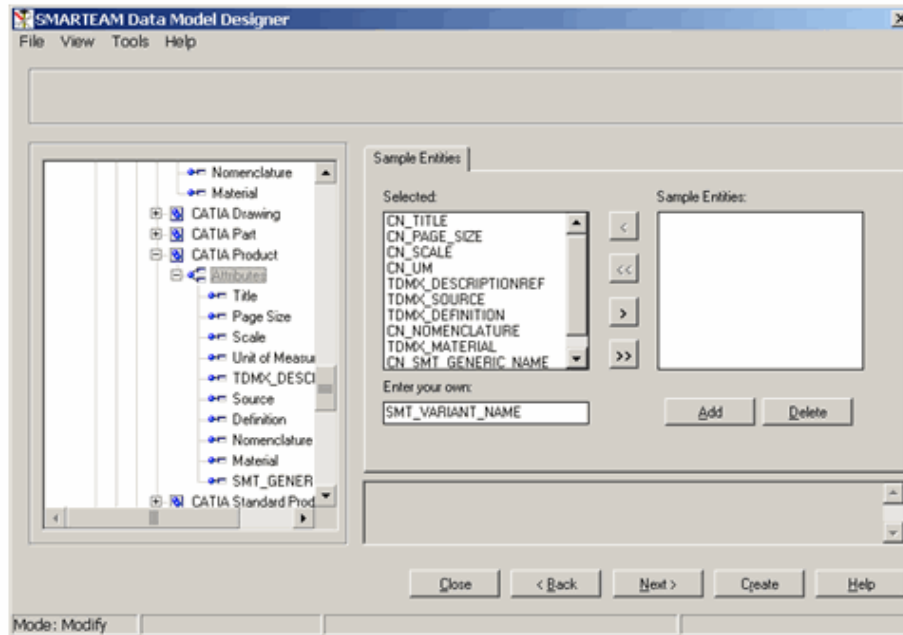


5. Modify the attribute definition as follows: set **Type** as Char and **size** as 128.



## SMT\_VARIANT\_NAME (memo)

1. Launch the SmarTeam Data Model Designer tool.
2. From the File menu, select **Modify Database Structure**.
3. Select your database.
4. Create the **SMT\_VARIANT\_NAME** new attribute for the "CATIA Product" class.  
Note that CN\_ is automatically added.



5. Modify the attribute definition as follows: set type as Memo.

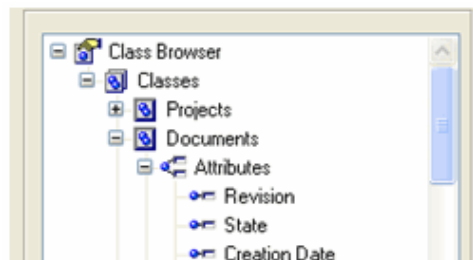


## SMT\_VARIANT\_COMMENT(memo)

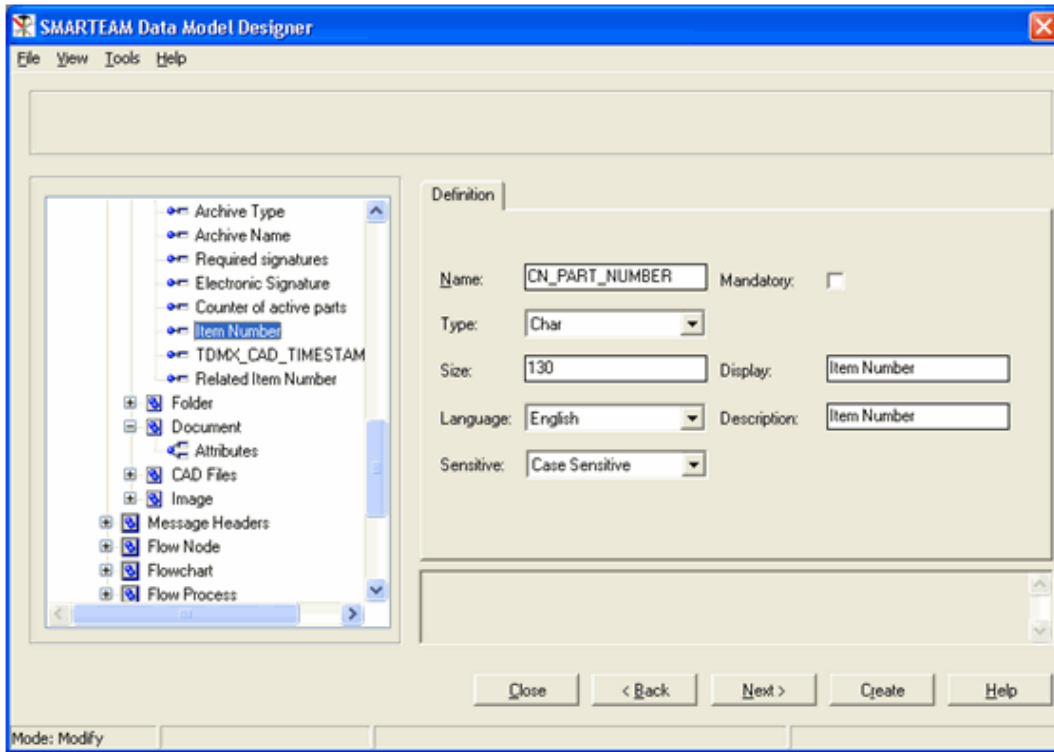
1. Same as SMT\_VARIANT\_NAME.
2. Save the modification in the database.

## Increasing the size of CATIA Product CN\_PartNumber

1. Select <Classes><Documents><Attributes> <Item Number>.



2. Change the size to 130.

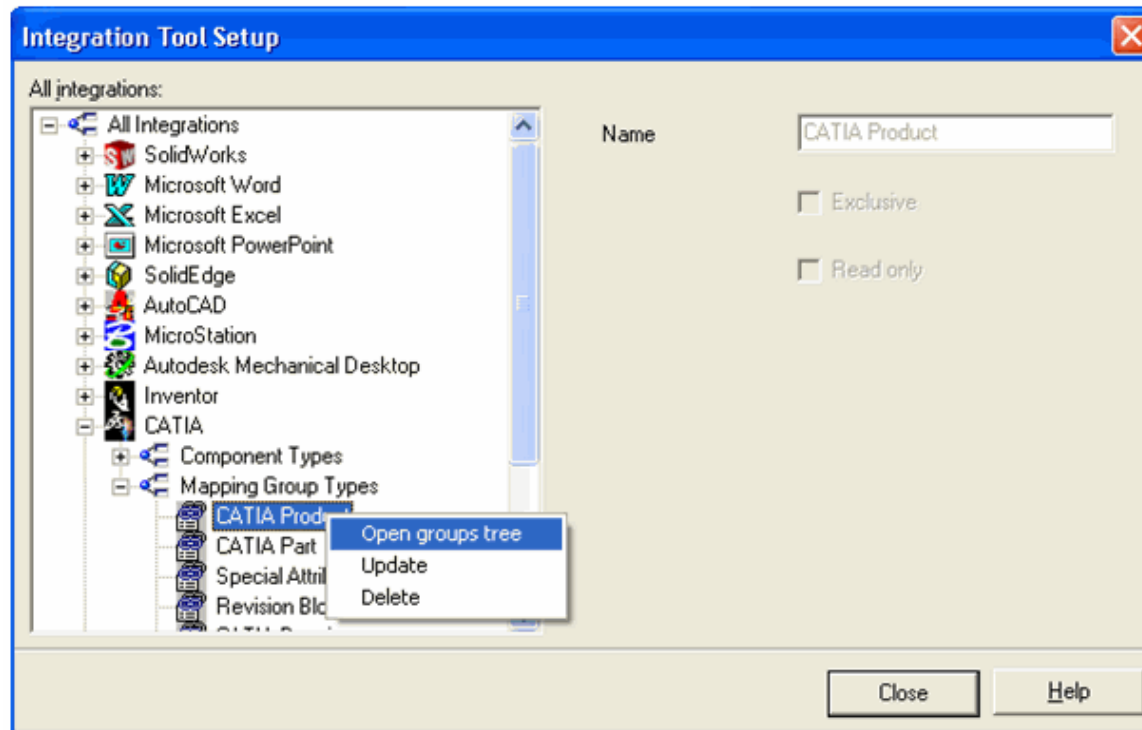


## Mapping between CATIA and SMARTeam in “Smarteam Tools Setup”

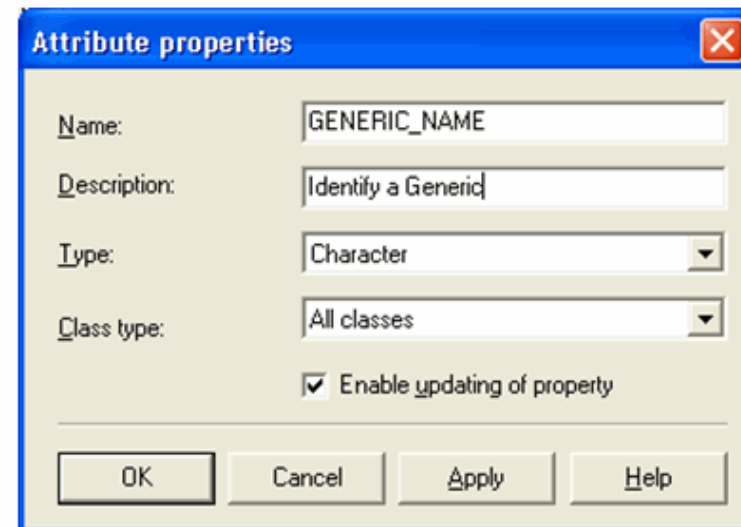
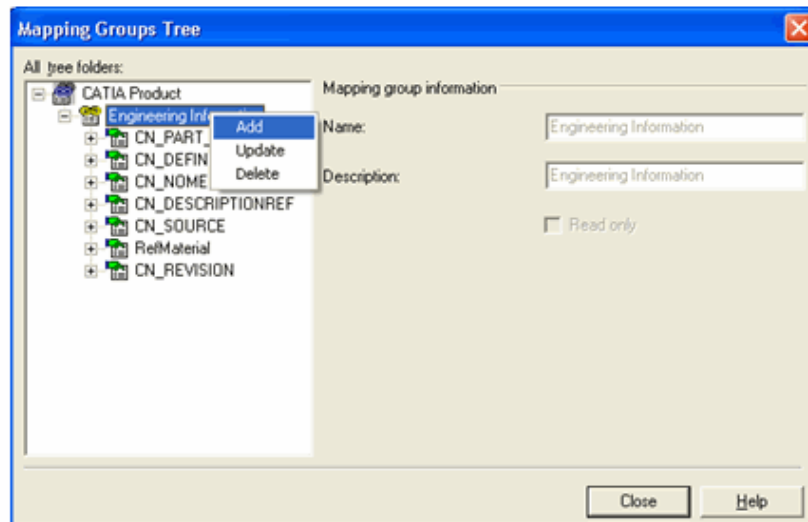
Launch the Integration Tools Setup utility.

Map **SMT\_GENERIC\_NAME** to **GENERIC\_NAME**

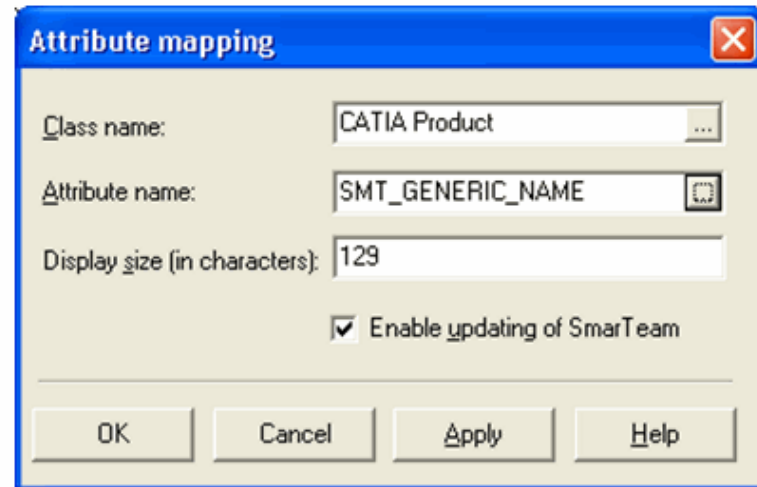
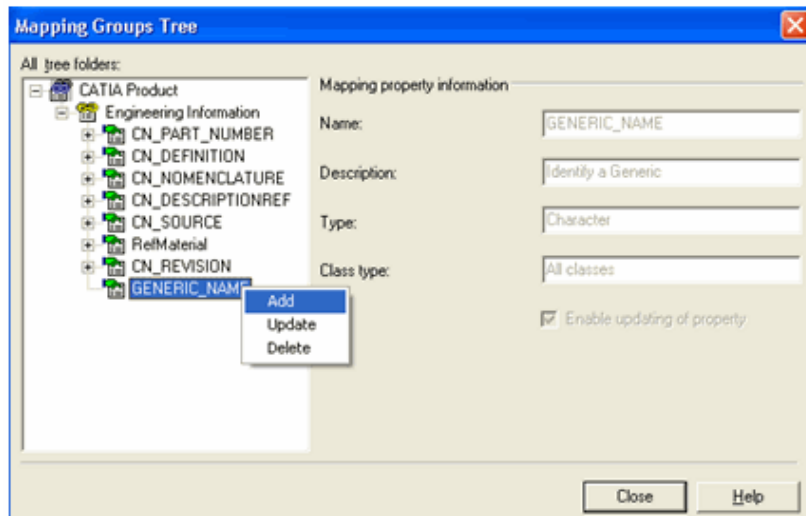
1. Open the CATIA Product group tree.



2. Create a new attribute **GENERIC\_NAME**



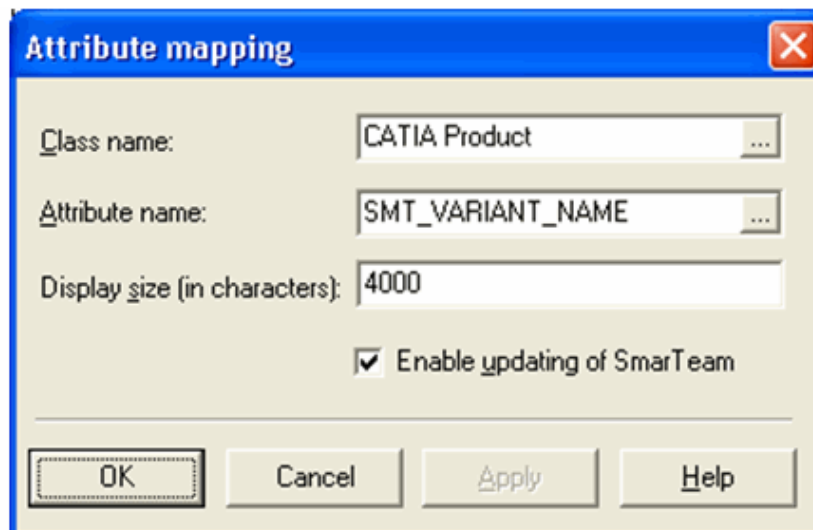
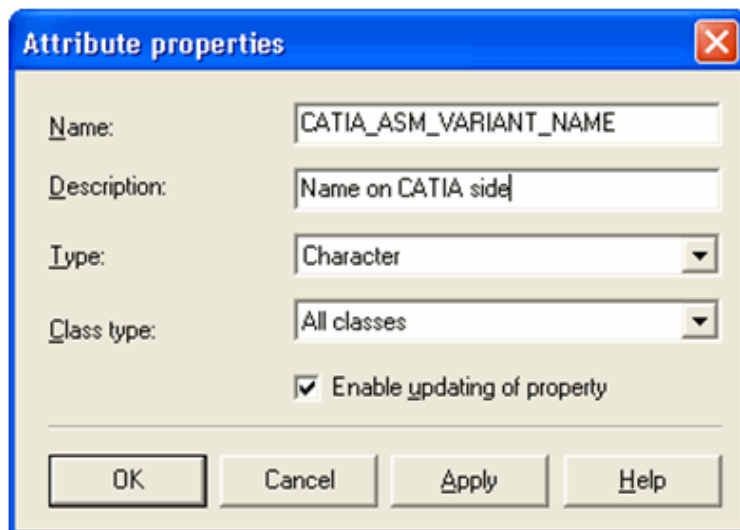
3. Map this attribute with a SmarTeam Attribute:



### SMT\_VARIANT\_NAME mapped to CATIA\_ASM\_VARIANT\_NAME

Add a new CATIA Attribute called CATIA\_ASM\_VARIANT\_NAME and map it to SMT\_VARIANT\_NAME.

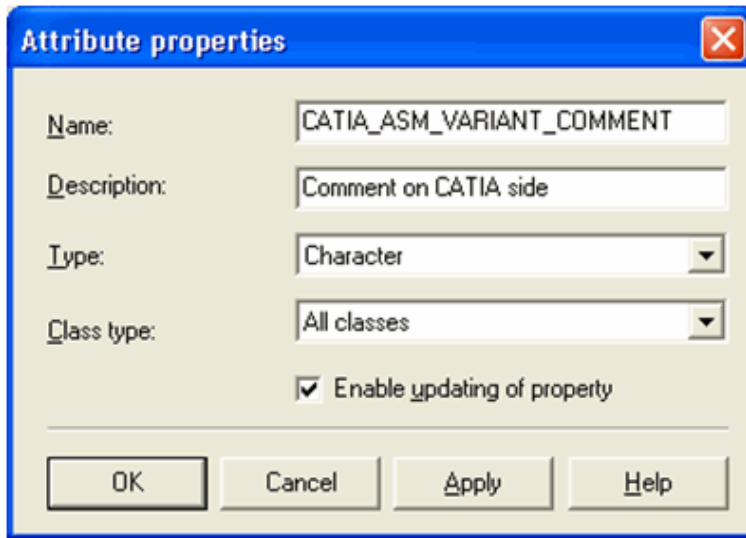
Set Display size to 4000.



## SMT\_VARIANT\_COMMENT mapped to CATIA\_ASM\_VARIANT\_COMMENT

Add a new CATIA Attribute called CATIA\_ASM\_VARIANT\_COMMENT and map it to SMT\_VARIANT\_COMMENT.

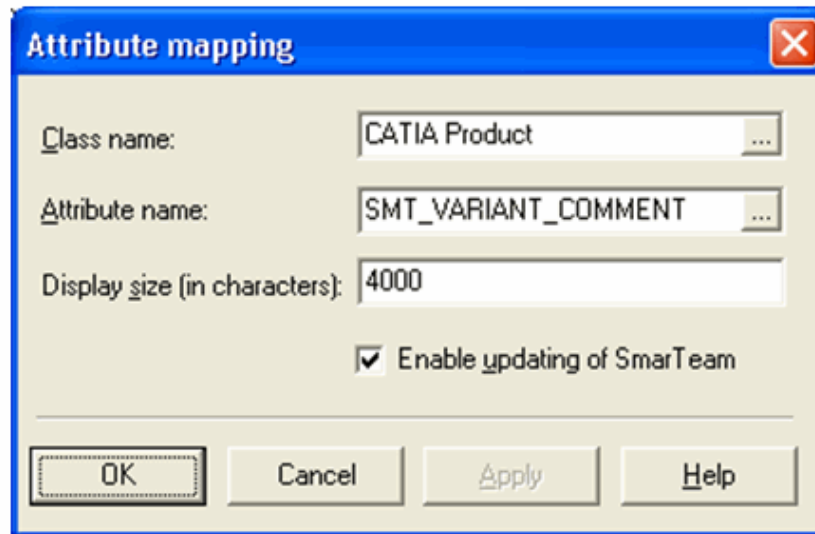
Set Display size to 4000.



The 'Attribute properties' dialog box is used to define the new attribute. It contains the following fields and options:

- Name:** CATIA\_ASM\_VARIANT\_COMMENT
- Description:** Comment on CATIA side
- Type:** Character (selected from a dropdown menu)
- Class type:** All classes (selected from a dropdown menu)
- ☒ Enable updating of property

Buttons at the bottom: OK, Cancel, Apply, Help.



The 'Attribute mapping' dialog box is used to map the new attribute to an existing one. It contains the following fields and options:

- Class name:** CATIA Product (selected from a dropdown menu)
- Attribute name:** SMT\_VARIANT\_COMMENT (selected from a dropdown menu)
- Display size (in characters):** 4000
- ☒ Enable updating of SmarTeam

Buttons at the bottom: OK, Cancel, Apply, Help.

 For more information about Assembly Variant functionality, see "Assembly Variant" chapter " " in the *CATIA Assembly Design User's Guide*.

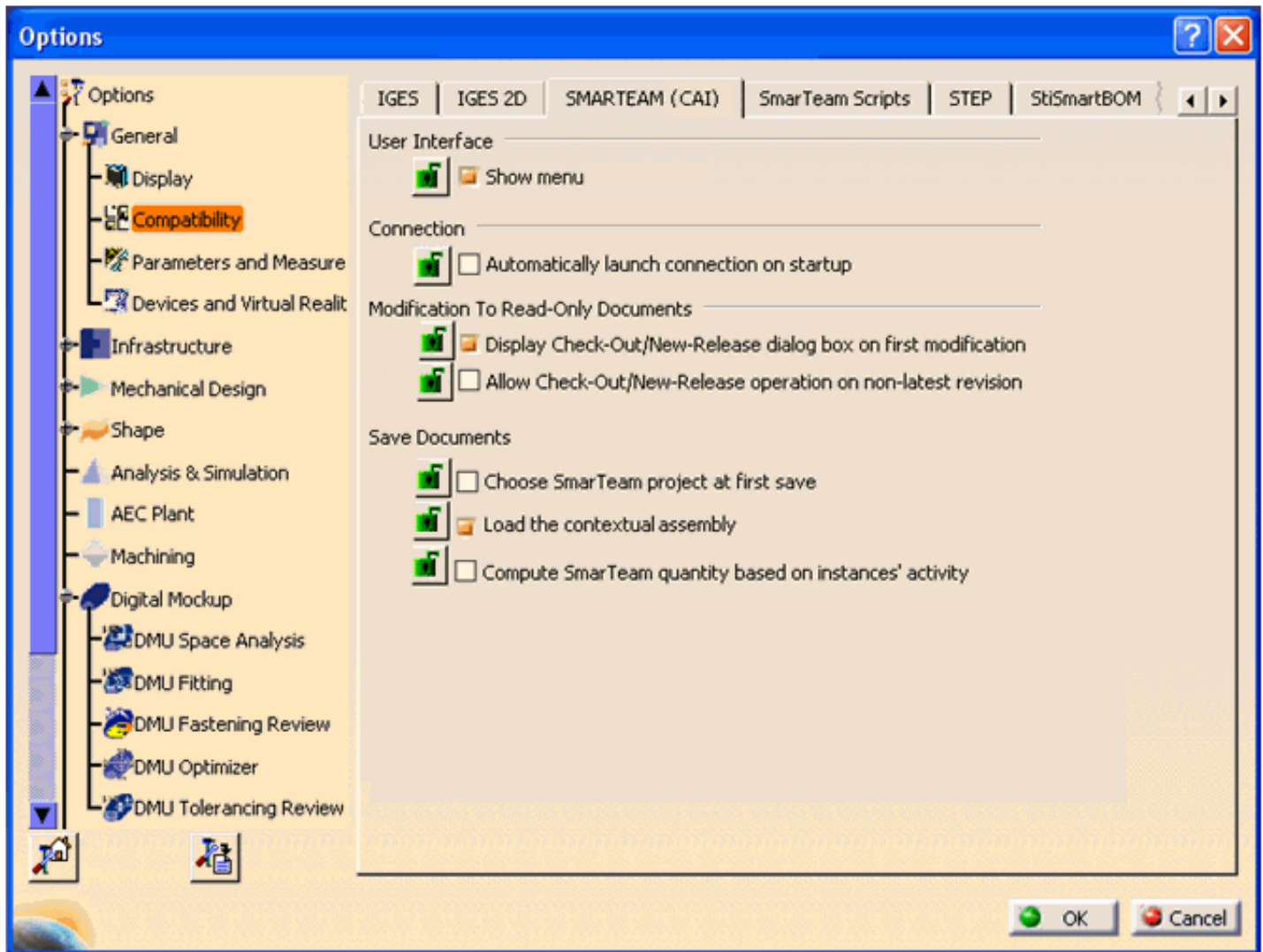


# Customizing Modification of Settings



When CATIA is launched with the Administrator switch (**CNEXT.EXE –admin**) the administrator will be able to lock or allow modification of settings.

After this switch has been set, the **SMARTTEAM (CAI)** tab is updated as follows:



# Global Refresh Administrator Tasks

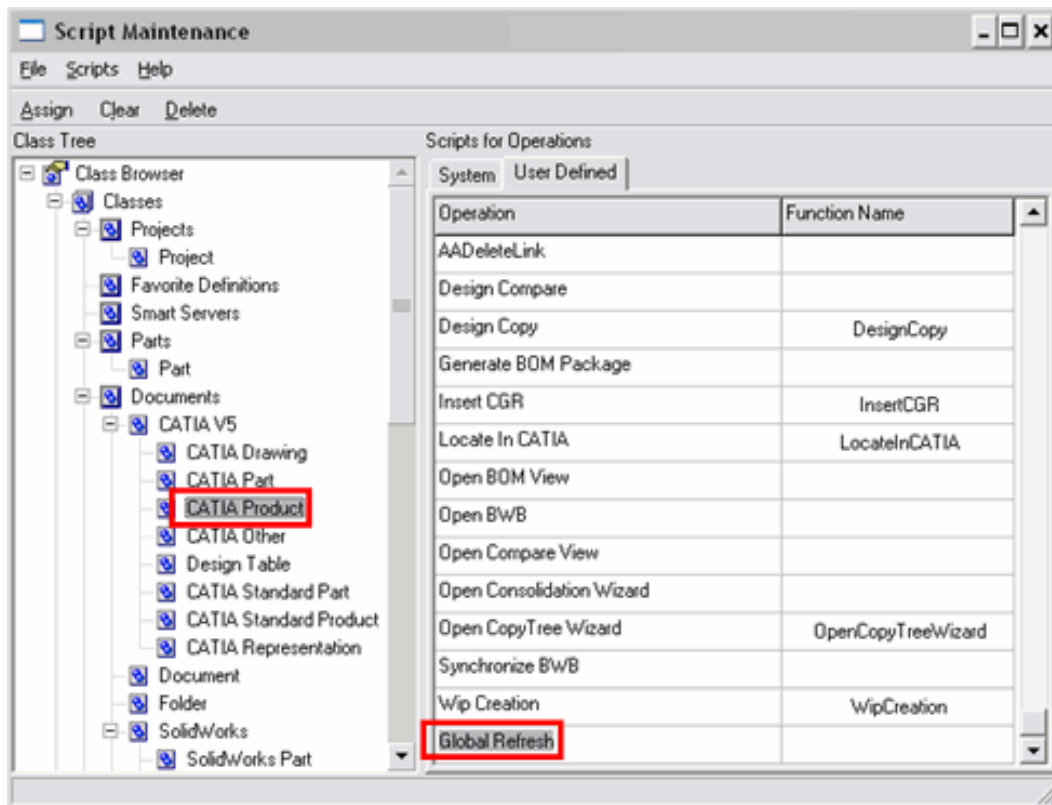


This section describes the procedure which the administrator must perform in order to configure the use of the **Global Refresh** operation with the SmarTeam database.

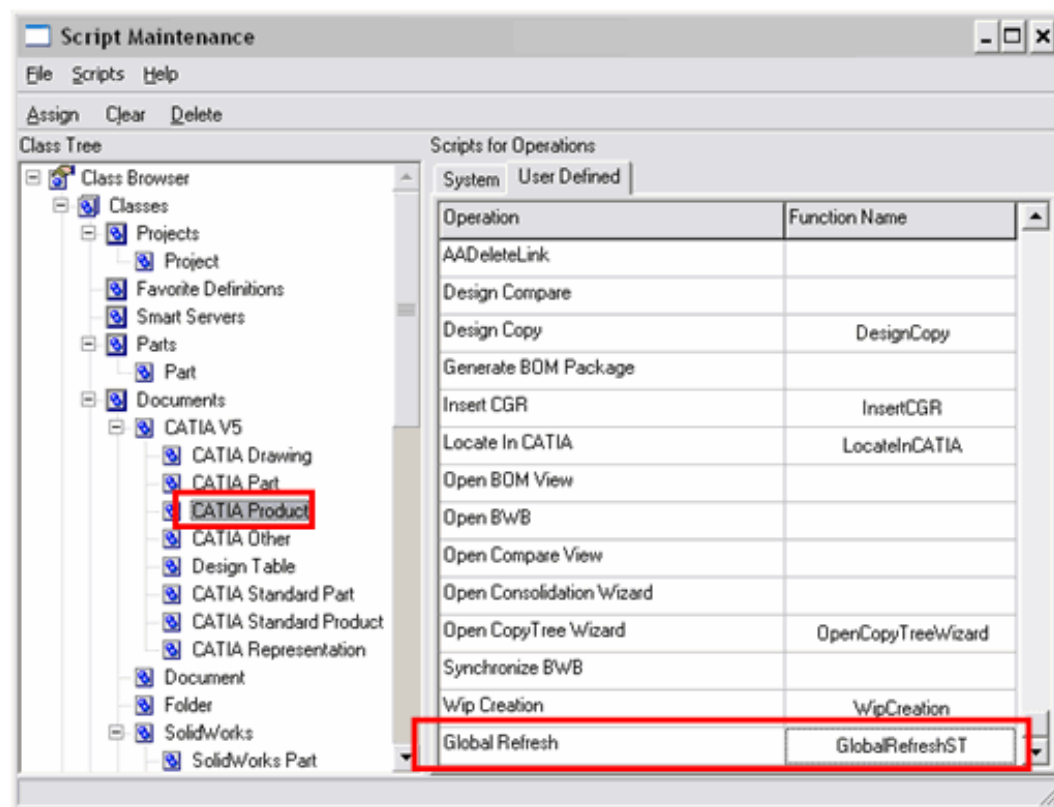


To enable the use of **Global Refresh**:

1. In Script Maintenance, select **CATIA Product** from the Documents class. In the right pane, open the **User Defined** tab.
2. In the Operation column, add a key: **Global Refresh**



3. Double-click in the **Function Name** cell of the **Global Refresh** operation. The Script Browser window opens.
4. In the **File Name** column, select **GlobalRefreshST.bs**.  
In the **Functions** column, select **GlobalRefreshST**.
5. Click **Compile**.
6. After clicking **OK**, the **Global Refresh** script is hooked to the database.




## Note

Following this procedure, you may update all CATIA products stored with the CATIA Product class in the database.

To update CATIA products stored with other classes, e.g., **CATIA Standard Product** class, repeat the same steps for other CATIA Product files classes.

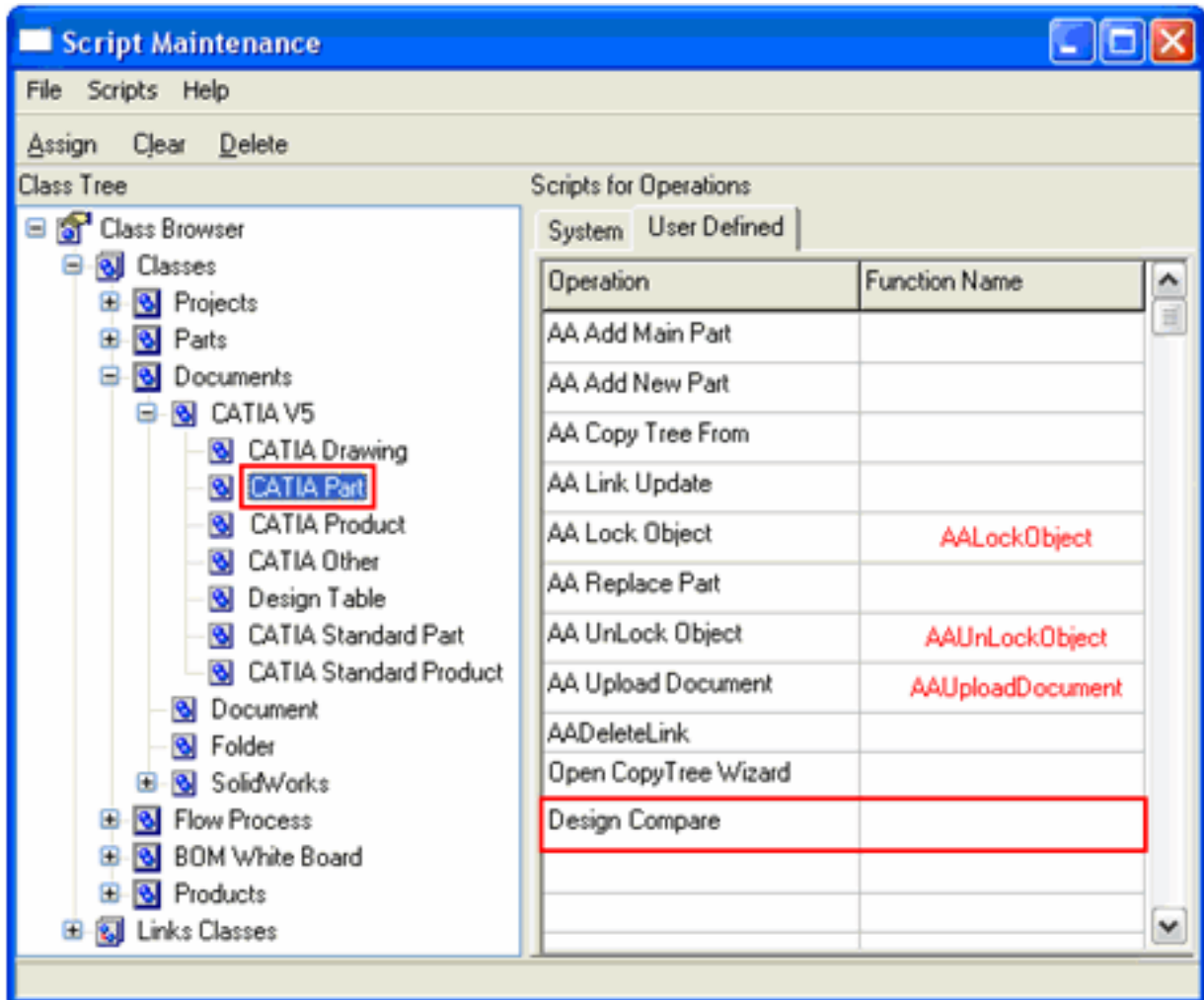


# Enabling Design Compare

 This section explains the steps necessary for enabling the Design Compare functionality and describes the script that must be attached in order to enable the loading of different revisions of the same CATIA Part or CATIA Product at the same time.

## Enabling Design Compare for CATIA Parts

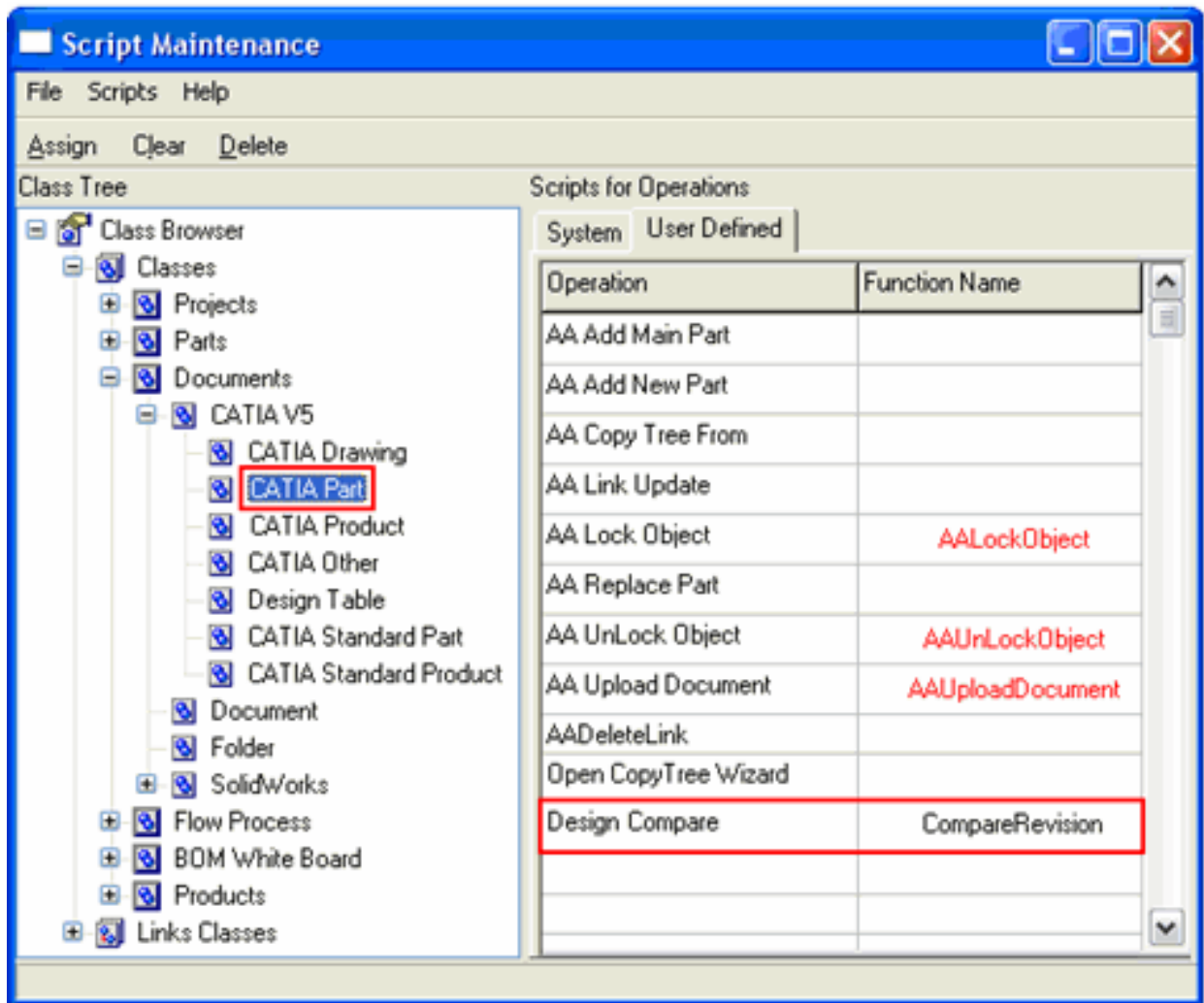
1. In Script Maintenance, select **CATIA Part** from the Documents class. In the right pane, open the **User Defined** tab.
2. In the Operation column, add a key: **Design Compare**.



3. Double-click in the **Function Name** cell of the Design Compare operation. The *Script Browser* window opens.
4. In the **File Name** column, select **DesignCompare.bs**. In the **Functions** column, select **DesignCompare**.

5. Click **Compile**.

6. After clicking **OK**, the DesignCompare script is hooked to the database:



**Note:** Following this procedure, you may compare several CATIA parts stored with CATIA Part class in the database. To compare several CATIA parts stored with other classes, e.g., CATIA Standard Part class, repeat the same steps for other CATIA Part files classes.



## Enabling Design Compare for CATIA Products


The Design Compare for CATIA Product function is performed in the same way as the Design Compare for CATIA Part function.



# CATIA Workbenches Integration

This section provides the methodology for the management of data specific to a few CATIA workbenches. The steps described are necessary to manage documents and to manipulate them in the most appropriate way.

# Managing 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 Documents Extraction Type](#) (recommended method) or [Setting Up a Shared Directory for CATIA Catalogs](#)
- [Creating and Saving CATIA Catalogs](#)
- [Product Resource Management \(PRM\)](#)

Instructions for design engineers who need to access catalog components are described in [Accessing CATIA Catalogs](#).

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

SmarTeam may help you manage your [catalogs](#), however, instantiating parts from the catalog must be done while creating a reference and not through 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

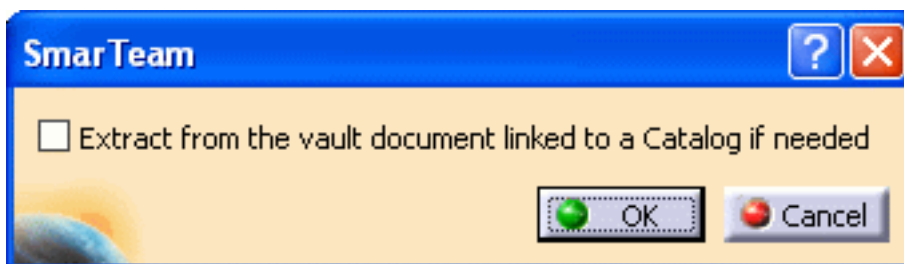
Remove permissions for all lifecycle operations for design engineers.

Only a librarian must have permissions for lifecycle operation on Standard CATIA Parts.

# Setting Documents Extraction Type

The option "**Extract from the vault document linked to a Catalog if needed**" makes it possible to restrict the number of CATPart documents to be extracted from the vault. You can load only the CATPart documents you need, which optimizes performance.

1. Select **Tools > Options....**
2. In the **General** category, click the **Document** tab.
3. In the **Linked Document Localization** section, locate and select **SmartTeam**.
4. Select the **Configure...** button.  
The **SmartTeam** dialog box is displayed.



5. Select **Extract from the vault document linked to a Catalog if needed**.
6. Click **OK** to confirm and close the dialog box.

## Note

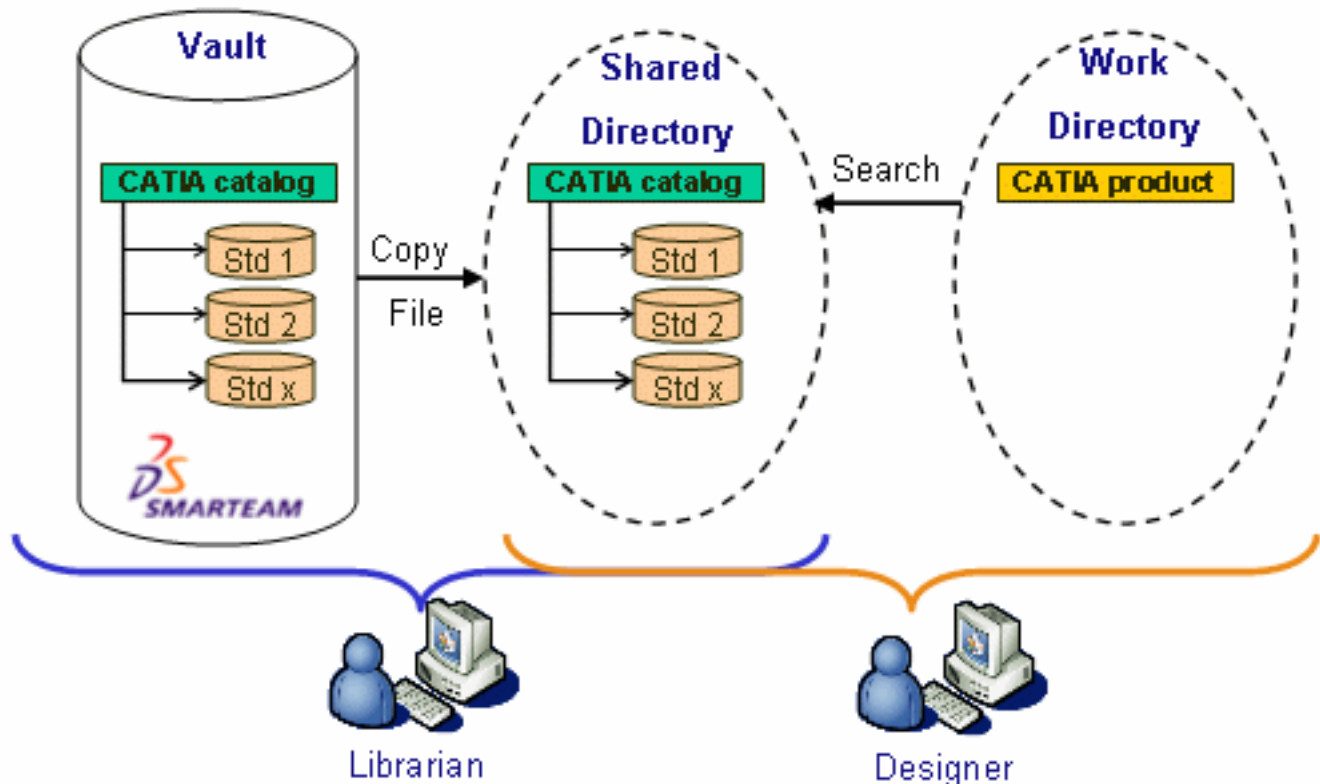
Make sure to use the "Local preview" option when you create catalogs. In case you do not create catalogs using the "Local preview" option (stored in the catalog file), CATIA always extracts the CATPart documents to generate the preview, resulting in poor performances. For more information about Local preview, see the CATIA Component Catalog Editor User's Guide.

# Setting Up a Shared Directory for CATIA Catalogs

Up to V5R17, we recommended the shared directory methodology for extracting catalogs parts. The explanations below remain valid, but for performance purposes, you now should work using **Extract from the vault document linked to a Catalog if needed** as described in the above paragraph.

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.



## Creating and Saving CATIA Catalogs

The procedure described below is a simplified version of the process. For detailed instructions, see to [Creating and Saving a Catalog](#).

1. Create CATIA catalogs.
2. **Resolve** the catalogs
3. Save the catalogs into SmarTeam using **Bulk Loading**
4. **Release** all CATIA Parts related to your catalog
5. **Copy** the catalogs and standard parts to the shared directory

## CATIA Standard Parts Class


6. Save catalog parts to CATIA Standard Parts class.

## 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). For more information, see [Integrating CATIA Equipment and Systems Engineering Workbenches](#).



# Creating and Saving a Catalog

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

This task gives you 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](#)
- [Releasing a CATIA V5 catalog inside SmarTeam](#)
- [Setting Up a Shared Directory for the Catalog](#)
- [Copying the Catalog in the Shared Directory](#)



## Steps in CATIA

### 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

For more about the Catalog Workbench, see the *CATIA Component Catalog Editor User's Guide*.

### 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.

**Note:**

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

3. In the specification tree, select the part family to be resolved.
4. 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.

## Steps in SmarTeam



### Saving a CATIA V5 catalog inside SmarTeam

5. Define classes in which you will save your documents.  
For detailed instructions, see [Customizing the Bulk Loading Command](#)
6. Connect to SmarTeam.
7. Select **SmarTeam > Bulk Loading...**
8. Choose the path to find all standard parts.
9. Multi-select all documents you wish to save as Standard Part documents.
10. Still in the catalog, select **SmarTeam > Save**.  
SmarTeam prompts you to save the generic part. At this stage, you need to switch the class to Standard Parts. SmarTeam will promote to save the design table.  
SmarTeam will automatically generate reference links between all saved parts and the catalog.

### Releasing a CATIA V5 catalog inside SmarTeam

11. Release the catalog.  
The Release operation automatically releases all documents. The catalog and all its children are now saved and released in SMARTEAM.

## Setting Up a Shared Directory for the Catalog

- 12.** Set up a shared directory folder as default.  
To keep this shared directory clean and secured, it is strongly recommended to assign the Windows permissions as below:
  - Keep administration rights
  - Assign engineer designers read-only rights only



## Copying the Catalog in the Shared Directory

- 13.** Search for your catalog.
- 14.** Right-click and select **Copy File**.
- 15.** Propagate this operation on all standard parts to perform the operation on the catalog and all its children.

The librarian's work is now complete: the catalog is in the electronic vault and the shared directory contains a copy of it. Design engineers can then [access CATIA Catalogs](#).



# Modifying a Catalog

-  This task explains the necessary steps a librarian should follow whenever he wishes to modify CATIA catalogs saved in SmarTeam.
-  Whatever modifications required (deletion or modification of a pointed document for instance), proceed as follows:
  - [Modify the Documents](#)
  - [Release the CATIA V5 catalog inside SmarTeam](#)
  - [Copy the Catalog in a Shared Directory](#)

## Modifying the Documents

1. Go to **Tools/Option/Infrastructure/Catalog Editor**.
2. Select the "Instantiate only resolved family component" option under Resolve Family Component.
3. Set the folder path to the default working folder (SmarTeam work directory).
4. Check out the catalog containing the documents to be modified.
5. Check out the documents you wish to modify.
6. Modify the documents in your CATIA session.
7. Save the documents in the SmarTeam database.

## Releasing the CATIA V5 catalog inside SmarTeam


1. Release the catalog.  
The Release operation automatically releases all documents. The catalog and all its children are now saved and released in SmarTeam.


## Copying the Catalog in a Shared Directory

1. Set the shared directory folder as the default shared directory folder.
2. Search for your catalog using the **Find** command.
3. Right-click and select **Copy File**.
4. Propagate operation on all standard parts.



# Managing CATIA Sheet Metal Tables

 This section aims at giving you a general overview of the necessary information for managing CATIA sheet metal bend tables. For detailed instructions, see [Integrating CATIA Sheet Metal Tables](#). Design engineers who need to access CATIA Sheet Metal Tables will find operating instructions in [Accessing Sheet Metal Bend Tables](#).

 The instructions provided below also apply to thread tables used in the Part Design workbench.

For reference information about sheet metal design, see the *CATIA Sheet Metal Design User's Guide*.

## Sheet Metal Bend Tables

SmarTeam supports sheet metal bend tables. These are made of two kinds of files:

- A thickness table includes material, thickness, bend radius and the path to the bend deduction file

	A	B	C	D
1	SheetMetalStandard	Thickness(mm)	DefaultBendRadius(mm)	BendTable
2	Steel (BDT)	1	1	S1-1.xls
3	Steel (BDT)	1	2	S1-2.xls
4	Steel (BDT)	1	4	S1-4.xls
5	Steel (BDT)	1.5	1.5	S1_5-1_5.xls
6	Steel (BDT)	1.5	3	S1_5-3.xls
7	Steel (BDT)	1.5	5	S1_5-5.xls
8	Steel (K-Factor = 0.4)	2	2	

- A deduction table includes the angle and the deduction value.

	A	B
1	OpenAngle(deg)	Deduction(mm)
2	35	0.01
3	45	-0.03
4	60	-0.04
5	75	-0.07
6	90	-0.12
7	105	-0.1
8	120	-0.07

Bend tables can be:

- .txt files
- .xls files

In a company, there is usually a unique thickness table but several deduction tables. These tables seldom change but are widely used by design engineers.

# Link between CATPart Documents and Bend Tables

Bend tables are recognized in CATPart documents as [design tables](#). Each sheet metal CATPart document points towards two bend tables:

- One thickness table
- One deduction table
  - The thickness table is pointed by all Sheet Metal CATPart documents
  - One deduction table is pointed by some Sheet Metal CATPart documents only

## Assigning Authorizations for Bend Tables

To avoid unnecessary lifecycle operation for tables, assign a class for these tables and remove permissions to perform check out and new release operations for this class. Only an authorized person, such as a librarian must have permissions for all lifecycle operations.

Rights to update or check-out documents can be managed by:

- User, role and group
- document class
- projects

## Setting Up a Shared Directory for Bend Tables

To allow design engineers to use (view and copy) the same tables, tables can reside in a shared directory.



# Integrating CATIA Sheet Metal Tables



This task provides the main instructions for integrating CATIA sheet metal bends into your SmarTeam database. This procedure can be divided into the following main steps, all of them to be performed in SmarTeam.

- [Creating a role](#)
- [Creating a project](#)
- [Assigning authorizations](#)
- [Saving bend tables inside SmarTeam](#)
- [Bend tables Inside SmarTeam](#)
- [Associating the design tables to the project](#)
- [Setting up a shared directory](#)
- [Copying the bend tables into the shared directory](#)



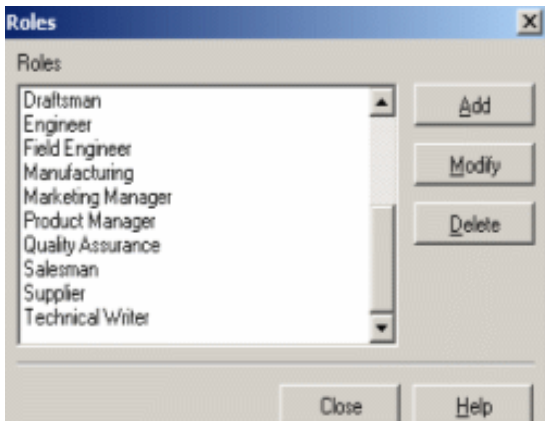
## Creating a Role

You need to create a role for which you will then assign some [authorizations](#).

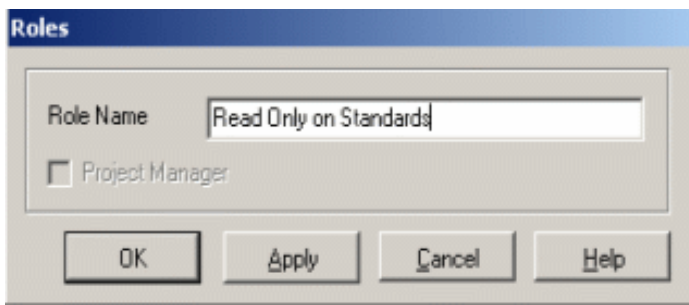
1. From the SmarTeam User Maintenance tool, display the Administration dialog box to access the command for creating roles.



2. Click the role icon to display the Roles dialog box:



3. Create the role of interest. For example, name it as "Read Only on Standards".



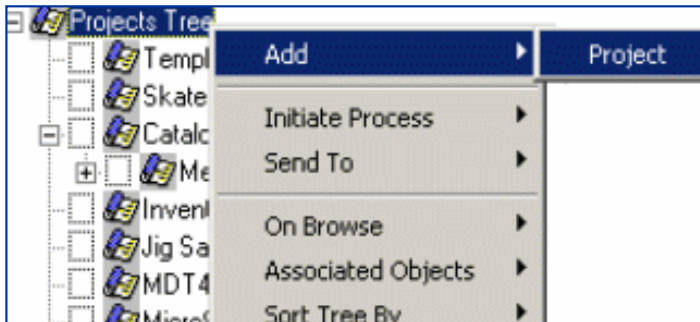
For more information on defining roles, see the *SmarTeam Editor Administration Guide*.

## Creating a Project



Create a project on which you will restrict design engineers authorizations. Attaching documents to projects is a good way of sharing and controlling these documents.

4. Add a project to which standard sheet metal tables will be linked.

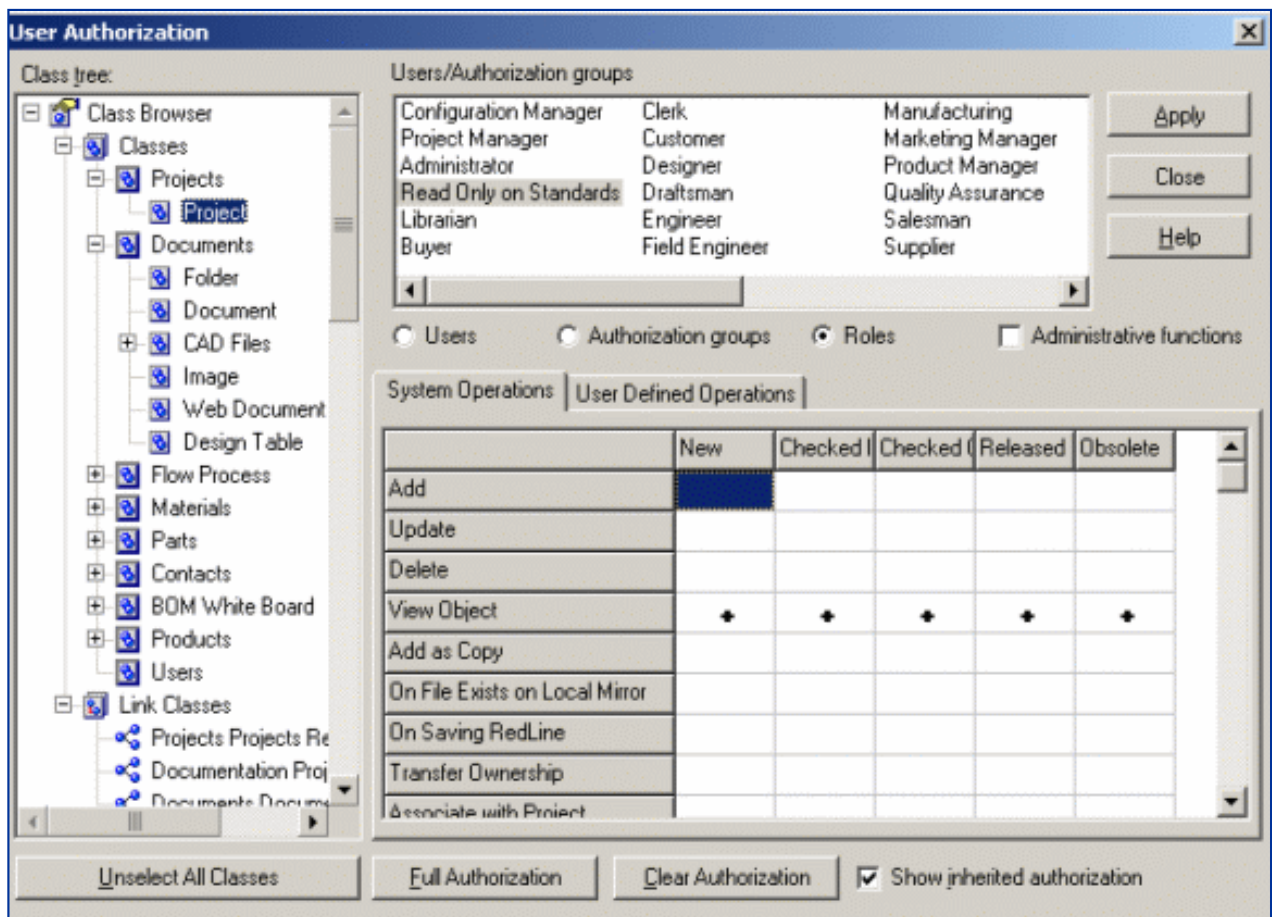


## Assigning Authorizations

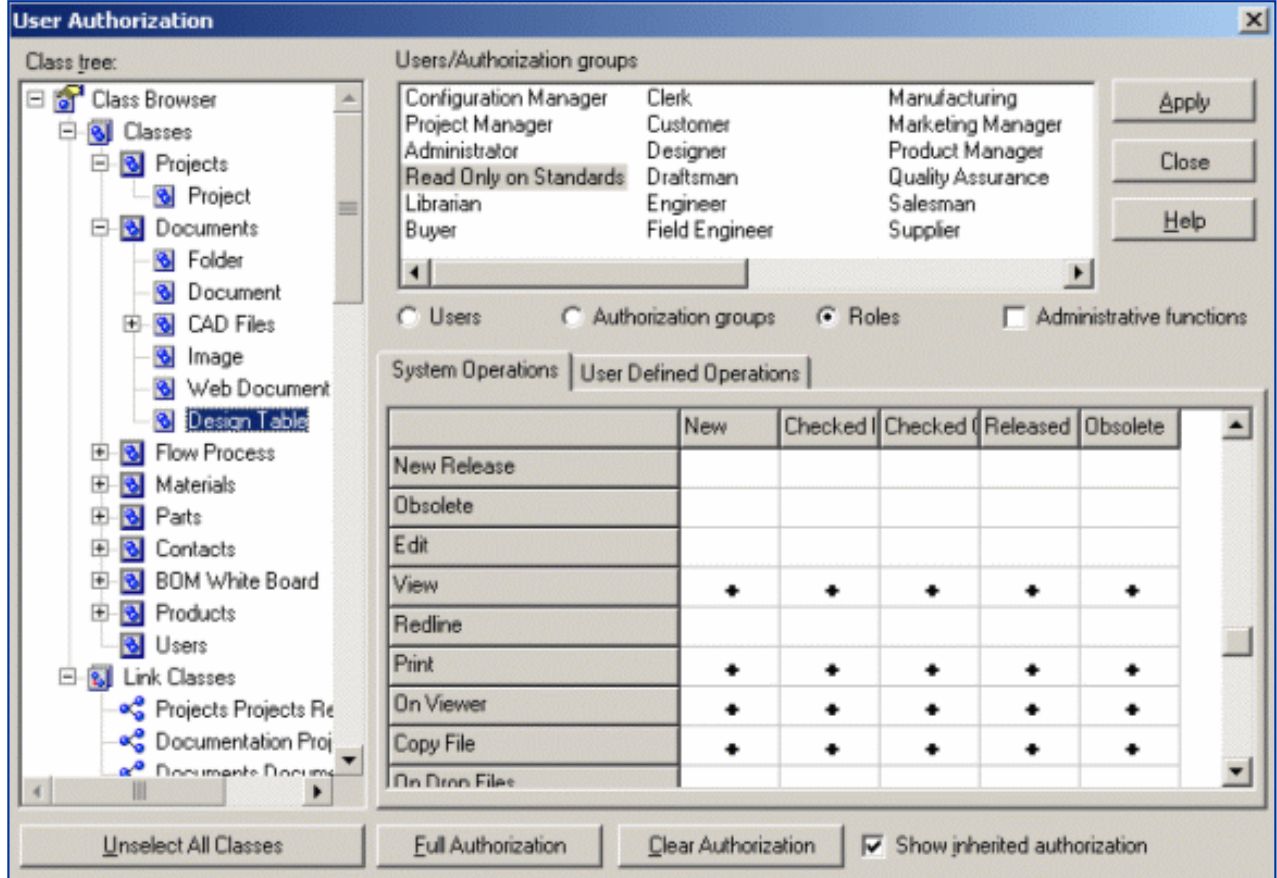


You need to assign authorizations for design engineers.

5. Define appropriate authorizations for the Read Only on Standards role, including:
  - o **View Object on project**



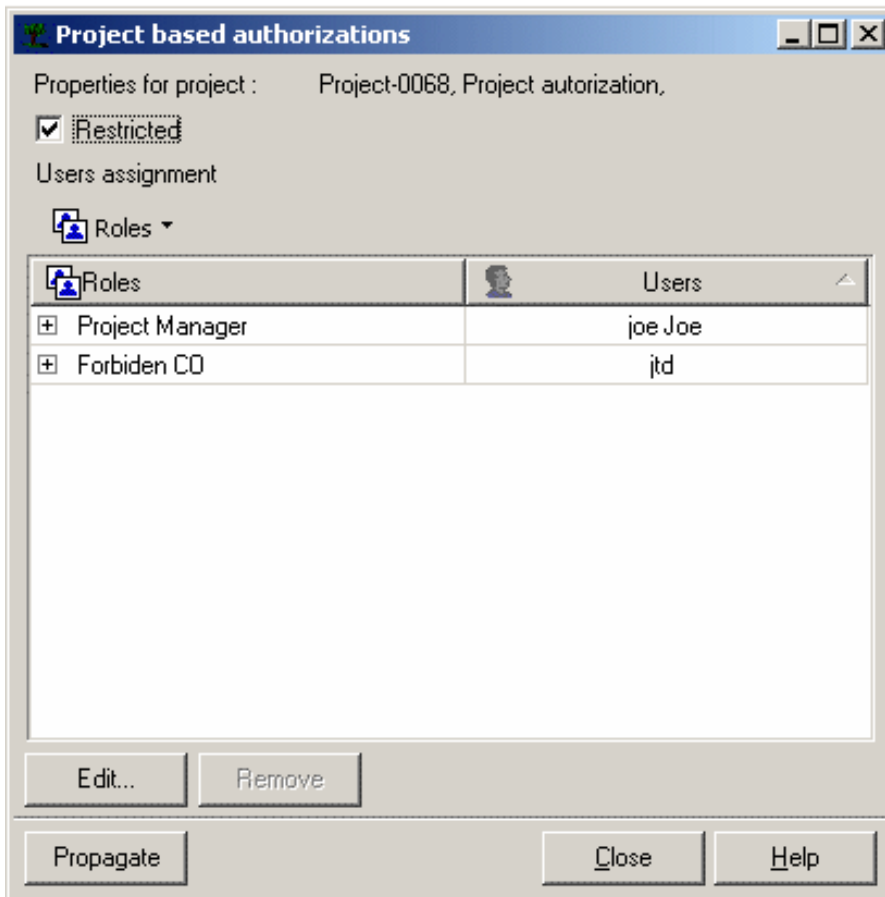
- View Object, View, Print, On Viewer and Copy File only for Design Tables:



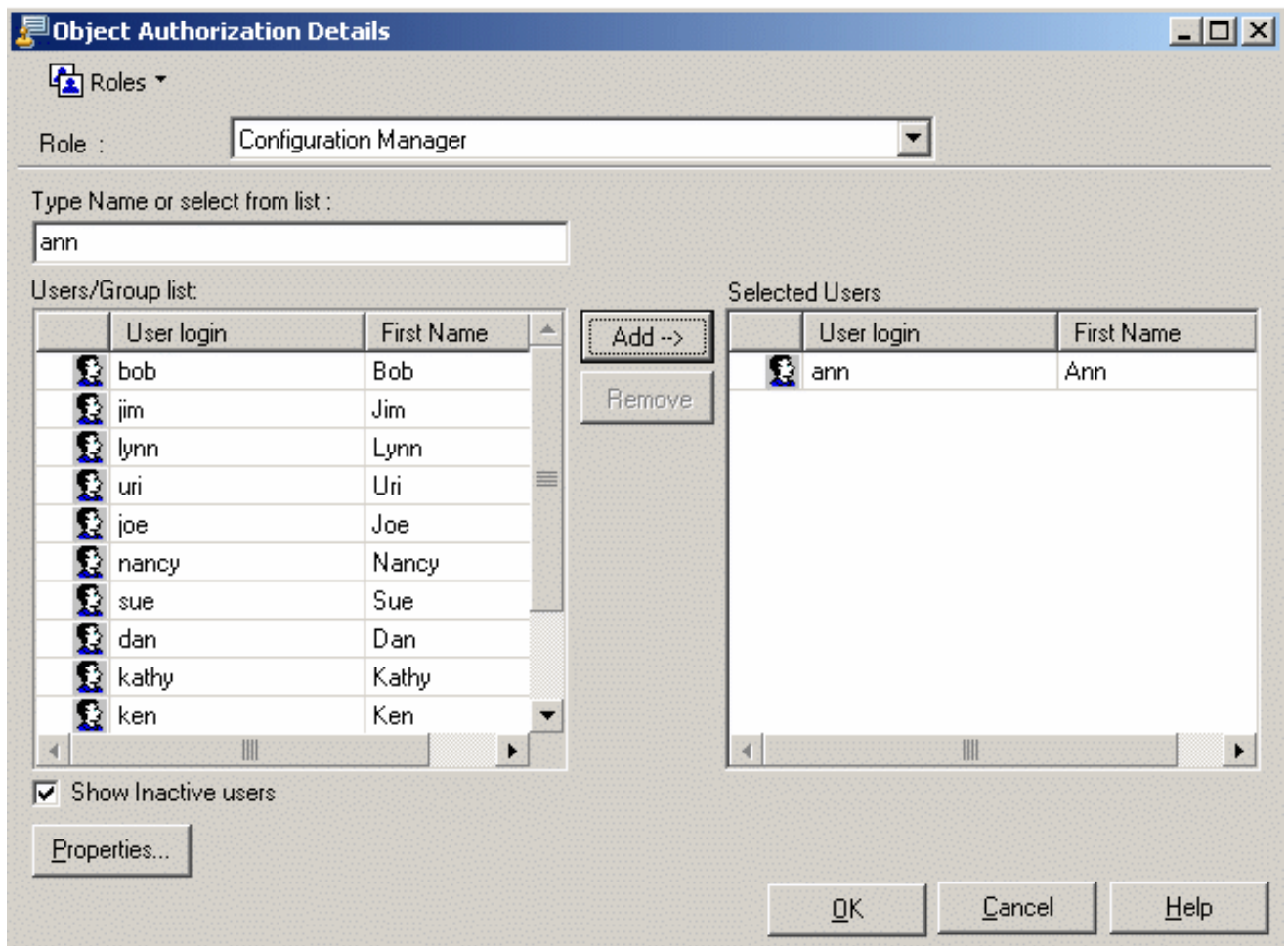
- View Object, on Projects Projects, Documentation Projects and Documents Documents relations

## Assigning a Specific Role to Design Engineers for the Project

6. Use the **Project Security** command to set the project specific for bend tables as **Restricted**.



7. Click **Edit...** to apply the **Read Only on Standards** local role to all designers who need to have read access to the bend tables.



See the *SmarTeam Editor Administration Guide* for further information.

## Saving Bend Tables inside SmarTeam



8. Save the bend tables in the "Design Table" class.
9. Link the bend tables to the "Sheet metal Bend Tables" project.

## Releasing Bend Tables Inside SmarTeam

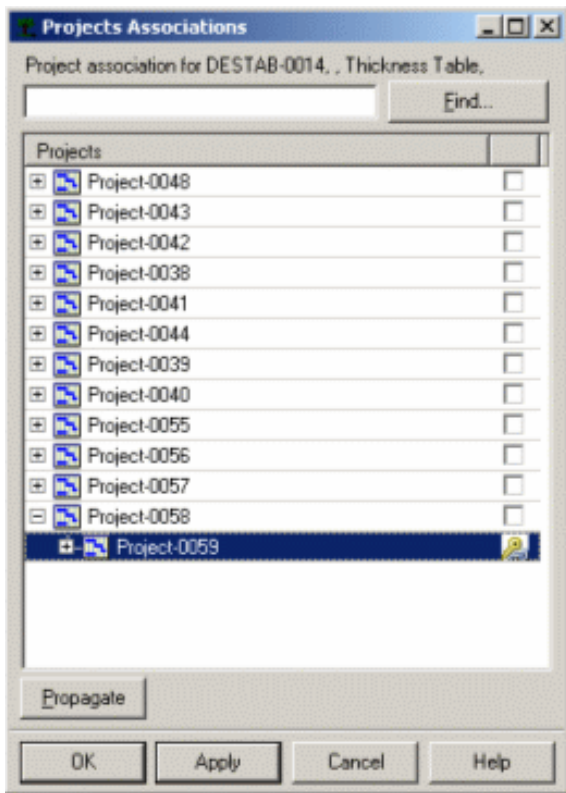


10. Release the bend tables.  
This is optional: Keeping them as checked-in documents should be sufficient.  
The Release operation automatically releases all documents.

## Associating the Bend Tables to the Project



11. To secure the bend tables thru the "Sheet metal Bend Tables" project, associate the bend tables to the project. To do so, use **Associate with project**.



## Setting Up a Shared Directory for CATIA Bend Tables

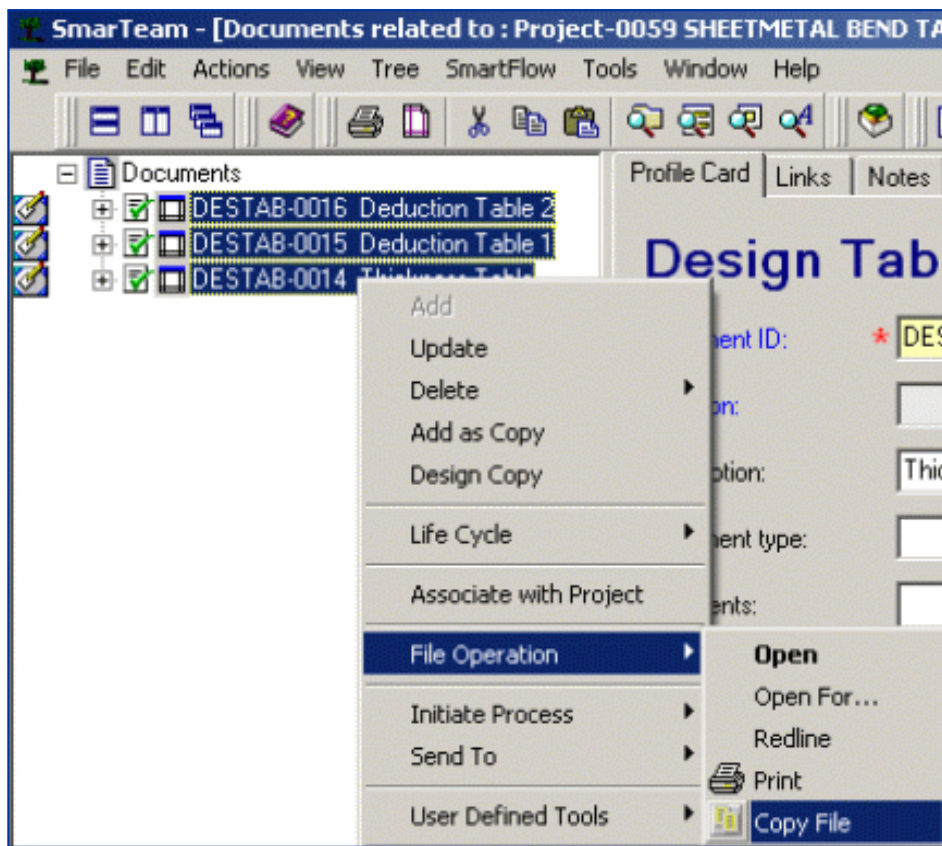
Sheet metal bend tables being meant to be widely used, a good way to dispatch them is to place them in a shared directory.

12. Set up a shared directory folder as default.  
A shared directory is just a directory on a server accessible to everybody. To keep this shared directory clean and secured, it is strongly recommended to assign the Windows permissions as below:
  - The administrator has administration rights
  - Users are assigned read-only rights.

## Copying the Bend Tables into the Shared Directory

Copying the Bend Tables into the shared directory is the final step you need to do.

1. Search for your bend tables in SmarTeam.
2. Multi-select the tables to be copied.
3. Right-click and select **File Operation > Copy File**.



4. Paste the files into the shared directory.

The administrator's work is now complete: the bend tables are in the electronic vault and the shared directory contains a copy of them. Design engineers can then [access Sheet Metal Bend Tables](#) to design sheet metal parts.

See the *SmarTeam Editor Administration Guide* for further information.



# Saving the Mass Attribute in SmarTeam



This section provides operating instructions to system administrators who need to integrate CATIA mass attributes in the SmarTeam database. System administrators need to perform three main operations:

- [Creating the CN\\_Mass Attribute](#)
- [Customizing CATIA Profile Cards](#)
- [Mapping CATIA Mass Attribute](#)

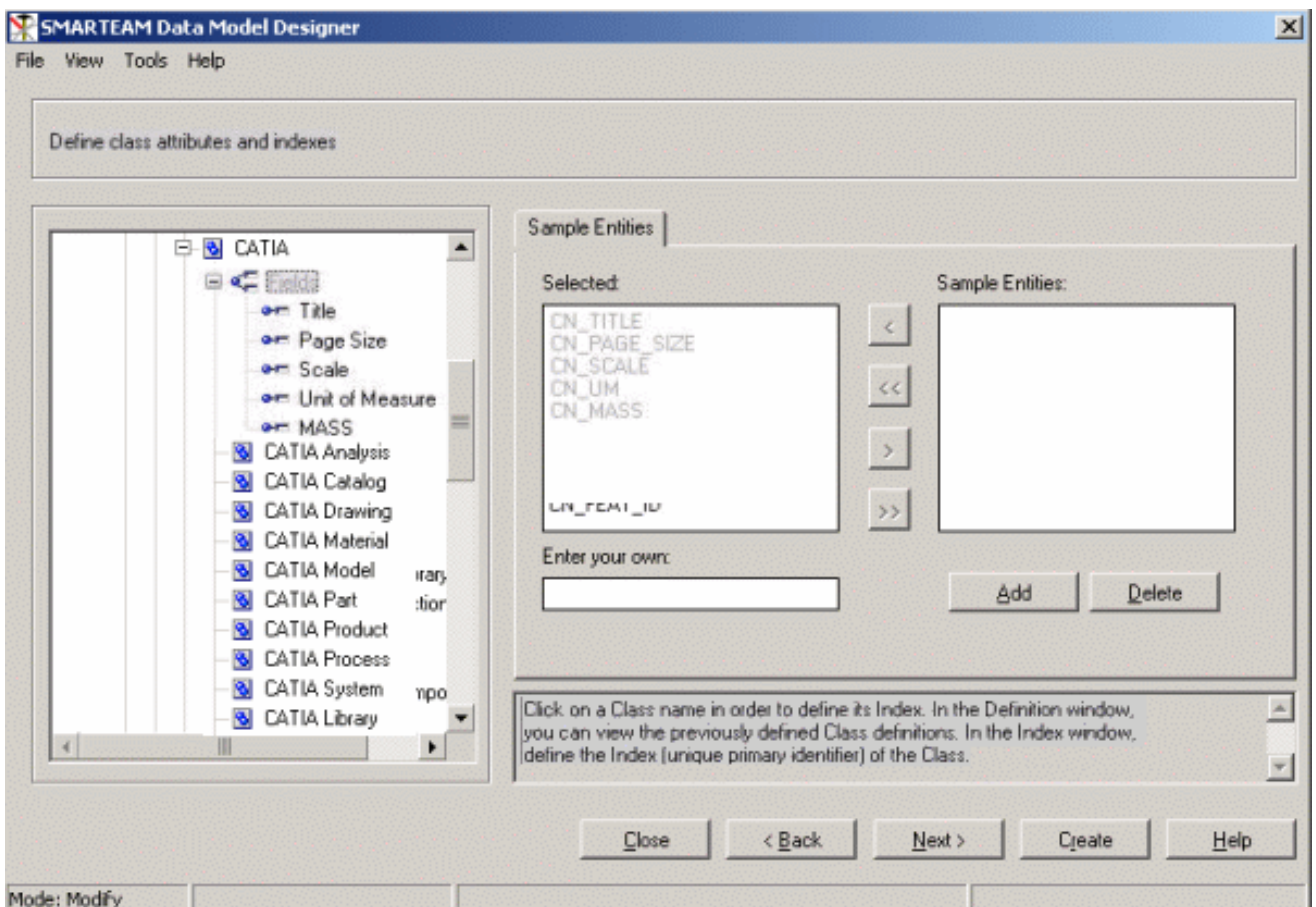
Instructions for designers working in the Generative Sheet Metal workbench are provided in [Creating the Mass Parameter in the Generative Sheet Metal Workbench](#).



## Creating the CN\_MASS Attribute

First of all, as a system administrator you need to create a new relationship between CATIA and SmarTeam. To do so, use the Data Model Designer utility to define an attribute related to the mass property.

1. From the SmarTeam Data Model Designer window, create the CN\_MASS attribute for any CATIA object class. You can create the attribute on CATIA Part class if necessary.



For reference information about SmarTeam Data Model Designer, see the *SmarTeam-Editor documentation*.

## Customizing CATIA Profile Cards

Showing the CN\_MASS attribute to designers means displaying values in profile cards. This involves

customizing CATIA Profile Cards. To do so, use the Form Designer utility.

1. Once the Form Designer utility is launched, from the Open Profile Card window select the CATIA Part class.
2. Add a new TextEdit field to CATIA Profile cards.

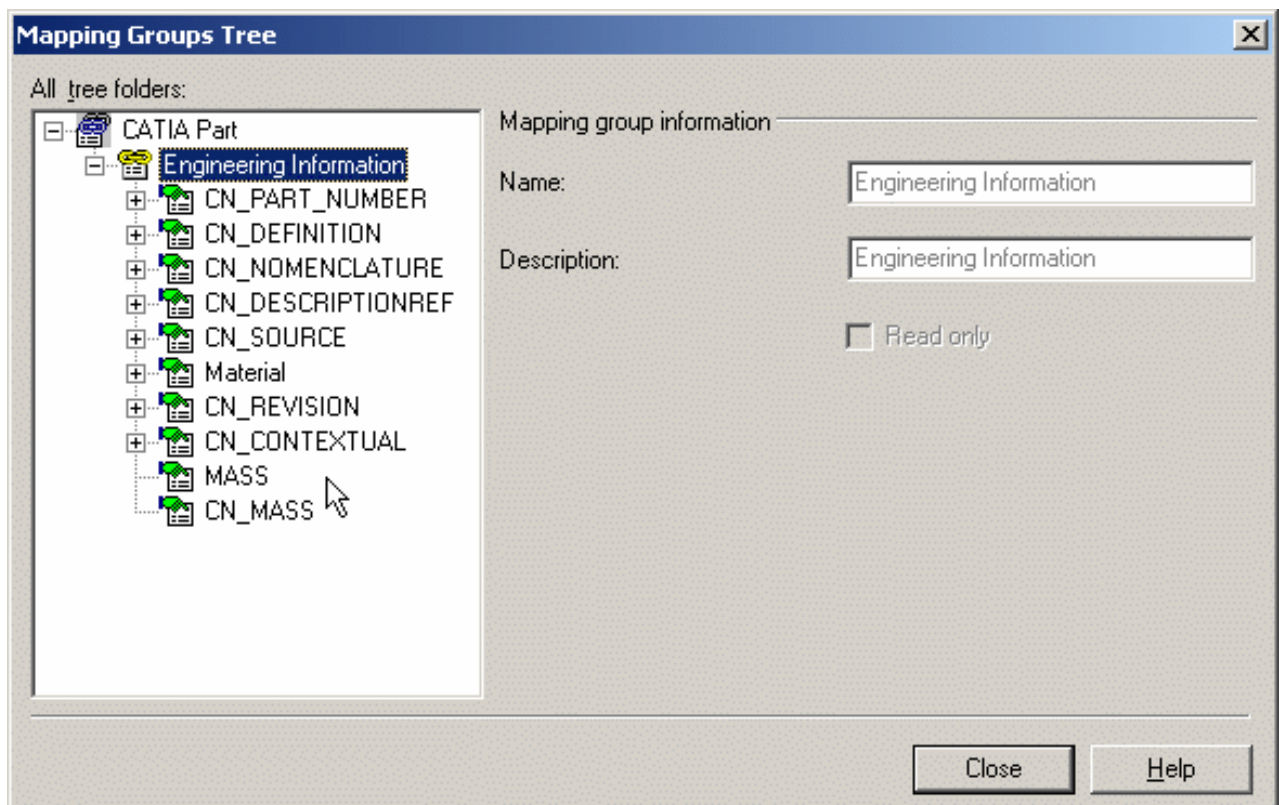
When done, when opening a CATIA Profile Card, the linked information (CN\_MASS attribute) from the database is automatically displayed in this field.

For reference information about Form Designer, see the *SmarTeam-Editor documentation*.

## Mapping CATIA MASS Attribute

The final step consists in mapping the CATIA MASS attribute with ST attribute CN\_MASS. To do so, use the [Integration Tools Setup](#) utility.


1. From the Integration Tool Setup window, select CATIA Part and run **Open groups tree**.
2. Once in the Mapping Groups Tree window, use the **add** contextual command to add the MASS attribute.
3. Map the MASS attribute with SmarTeam CATIA Part.CN\_MASS attribute.



For reference information about the Integration Tools Setup, see the *SmarTeam-Editor documentation*.



# Managing CATAnalysis Documents

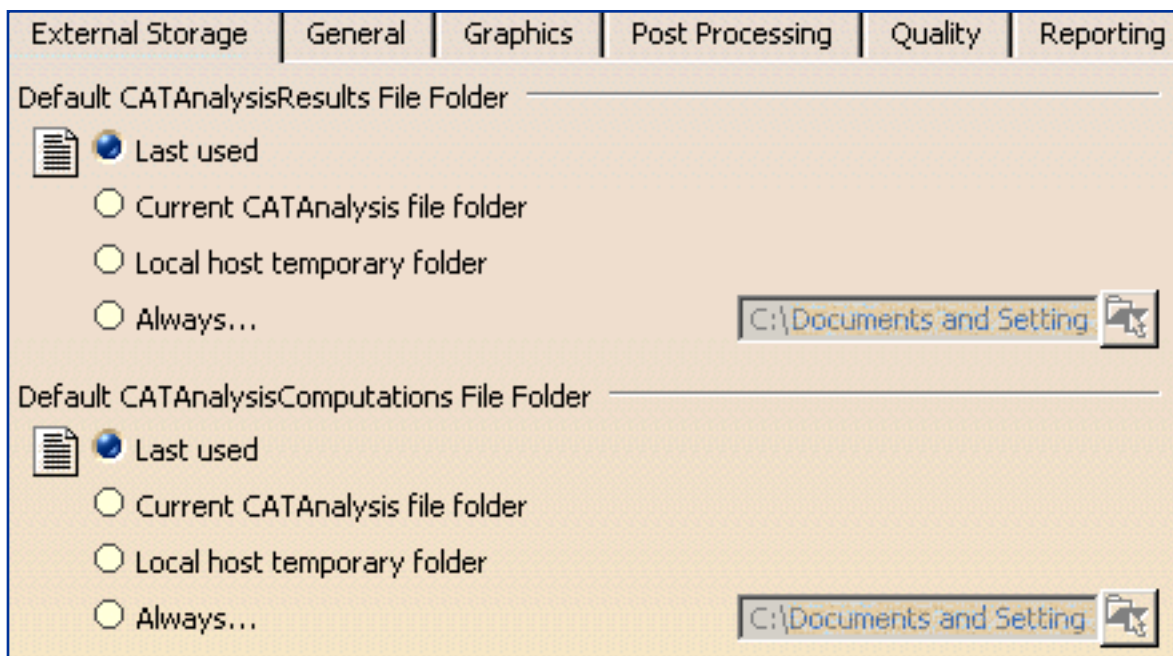
 This section is dedicated to system administrators who need to integrate CATAnalysis documents into the SmarTeam database. Instructions for design engineers are documented in [Handling CATAnalysis documents](#).

For information about mechanical analyses for 3D systems, see the *CATIA Generative Structural Analysis User's Guide*.

## Setting Directory for CATAnalysisResults and CATAnalysisComputations files.

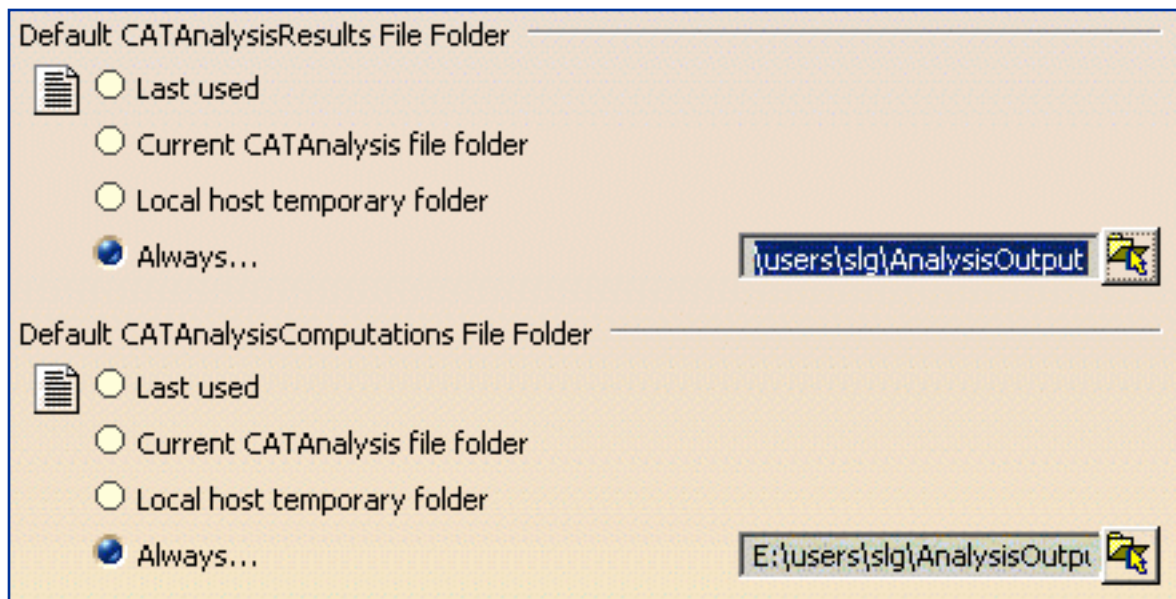
It is highly recommended to change the CATIA default setting for the location of the CATIA CATAnalysisResults and CATAnalysisComputations files. This is performed as follows:

1. Select **Tools > Options**.
2. From the Options dialog box that is displayed, click the **Analysis & Simulation** category to the left, and then the External Storage tab. The first two options let you specify the directories in which you want to save the CATAnalysisResults and CATAnalysisComputations files. By default, the **Last used** preferences are enabled in both cases. Using these default options, you would save the analysis files in a temporary directory, which is not recommended for working with SmarTeam.



3. For each option, check **Always...**
4. And click the icon to the right to navigate and select the directory of interest.


When done, the paths to the new directories are displayed:



5. Just click **OK** to confirm and close the dialog box.



# Integrating CATIA Equipment and Systems Engineering Workbenches

 This task can only be made at system administrator level. It shows you how to manage project resources when working with CATIA Equipment and Systems Engineering workbenches and SmarTeam. More precisely, it explains how to customize the Project Resource Management (PRM) file so as to reference SmarTeam resources.

This task assumes that catalogs data is already stored in SmarTeam. Additionally, you have copied all the catalogs along with all the pointed documents in shared directories. For more information on catalogs, see [Creating and Saving a Catalog](#).

## Using Project Resource Management (PRM) File

The project resource management (PRM) file identifies resources to the application. Specifically, the PRM file identifies each resource and its location (directory path). The PRM file also organizes the resources by discipline and application, associating resources to specific applications. Therefore, you get the correct resource (equipment catalog) for the resource designers are working in.

For more information on PRM, see the User's Guide dedicated to your Equipment & Systems Engineering application.

1. The project resource management (PRM) file is named **Project.xml**. Copy **Project.xml**. The default location of this file is:

```
...\intel_a\startup\EquipmentAndSystems\ProjectData
```

2. Rename and paste this file in some local folder so as to ensure that it will be shared by designers. For example, paste it at:

```
E:\machine_name\home\E&S\Project.xml
```

3. Modify the CATEnv file: set the CATDisciplinePath' CATIA Environment variable to `E:\machine_name\home\E&S\Project.xml`

This allows designers to use the Project.xml files while working with E&S. They will just need to run **Tools > Project Management > Select/Browse...** and select the appropriate .xml file.

## Modifying the .xml File

1. Change the .xml file rights to Read/write.

2. Open the file with the editor of your choice.
3. Change the Startup Directory value from ..\.. to CATIA startup path.  
`C:\Program Files\Dassault Systemes\B16\intel_a\startup`
4. Modify .xml path for all resources designers will use. To modify the .xml file for a particular resource, search the .xml file for that resource and then replace Location of resource from default value with the shared directory path where that resource (catalog) is present, give the path up to file name including file extension.  
For example, to modify the resource path for StructureSectionsCatalog, change location from default value.

```
> ${Components_Directory}\StructuralCatalogs\AISC\AISC_Resolved.  
catalog
```

with

```
\Shared_Directory\AISC_Resolved.catalog
```


Note that catalogs have been stored in SmarTeam but are extracted in shared directories.

5. Similarly, modify the locations of all resources that you want to be used from the shared directory.



# Defining Property Mapping


This section shows you how to read or set a property both on the CATIA and the SmarTeam side.

Three scenarios show you how to define property mapping between CATIA and SmarTeam, based on the CATIA Formula property  and the SmarTeam attribute.

You will learn how to map the set of CATIA properties of a CATDrawing document with the corresponding SmarTeam attributes of a SmarTeam CATIA Drawing class.

For quick, easy mapping of a drawing text with any of the CATIA Drawing attributes in the SmarTeam database you are better advised to follow the steps contained in [Displaying a CATIA Drawing SmarTeam Attribute in a Title Block](#). This particular task can be performed only by the user with administrative privileges. However, there are other tasks dealing with property mapping for which such privileges are not required ([Using Mapped Product Properties](#) for example).

# Property Mapping Basics

 The **Integration Tool Setup** utility enables the administrator to map properties from CATIA directly to an object's Profile Card. This provides users with the highest level of integration, enabling a bi-directional link from a CATIA property to the object's Profile Card and from a Profile Card to CATIA property.

This section provides an overview of property mapping. It deals with the following topics:

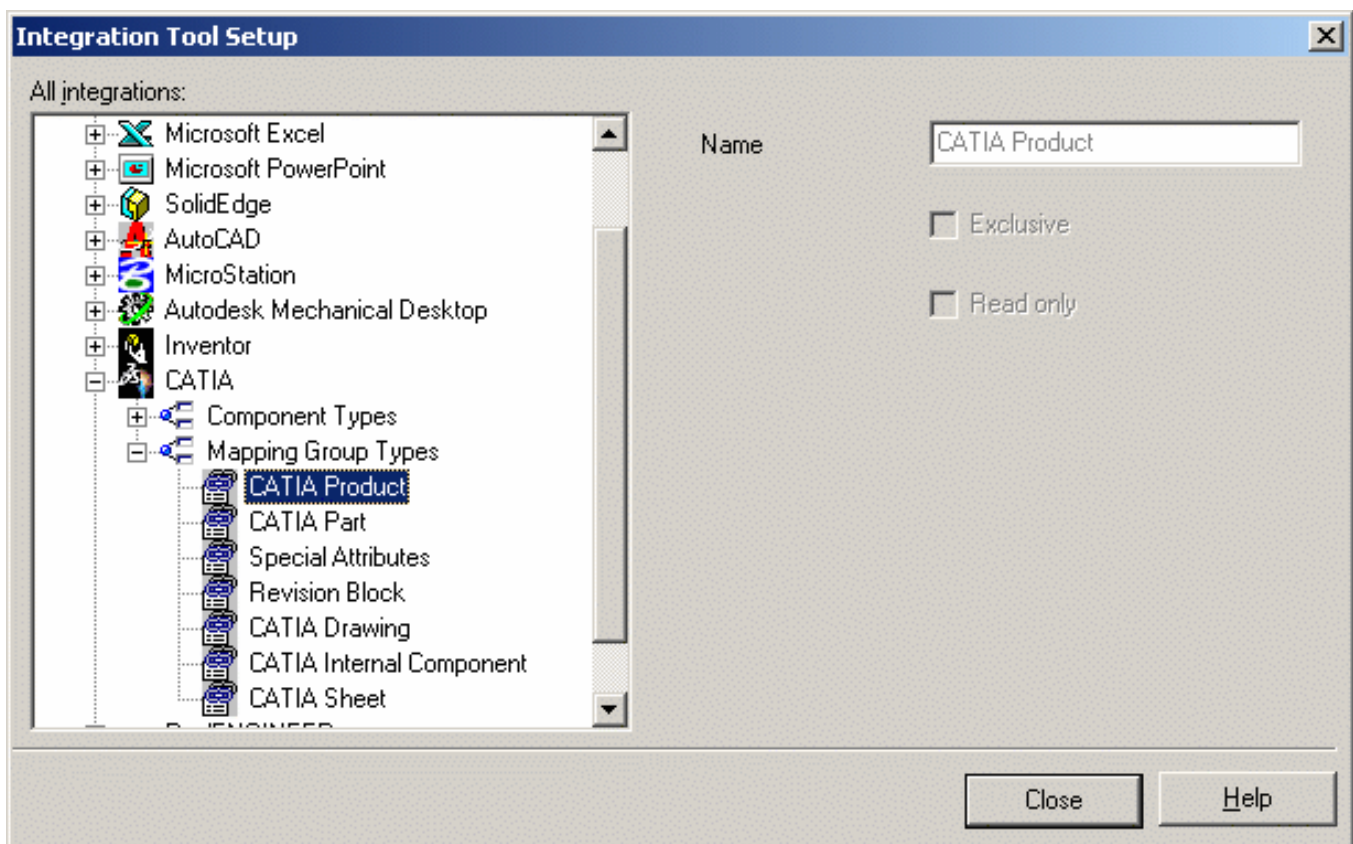
- [Integration Tool Setup Utility](#)
- [Mapping Direction](#)
- [Hard-coded Property Names](#)
- [Map a Text Value...](#)
- [Performance and Attribute Mapping](#)

For reference information on the Integration Tool Setup utility, refer to the SMARTEAM documentation.

## Integration Tool Setup Utility

The **Integration Tool Setup Utility** is used to:

- Define new relationships between SmarTeam and CATIA.
- Add, update and delete mapping groups, mapping properties and mapping attributes.



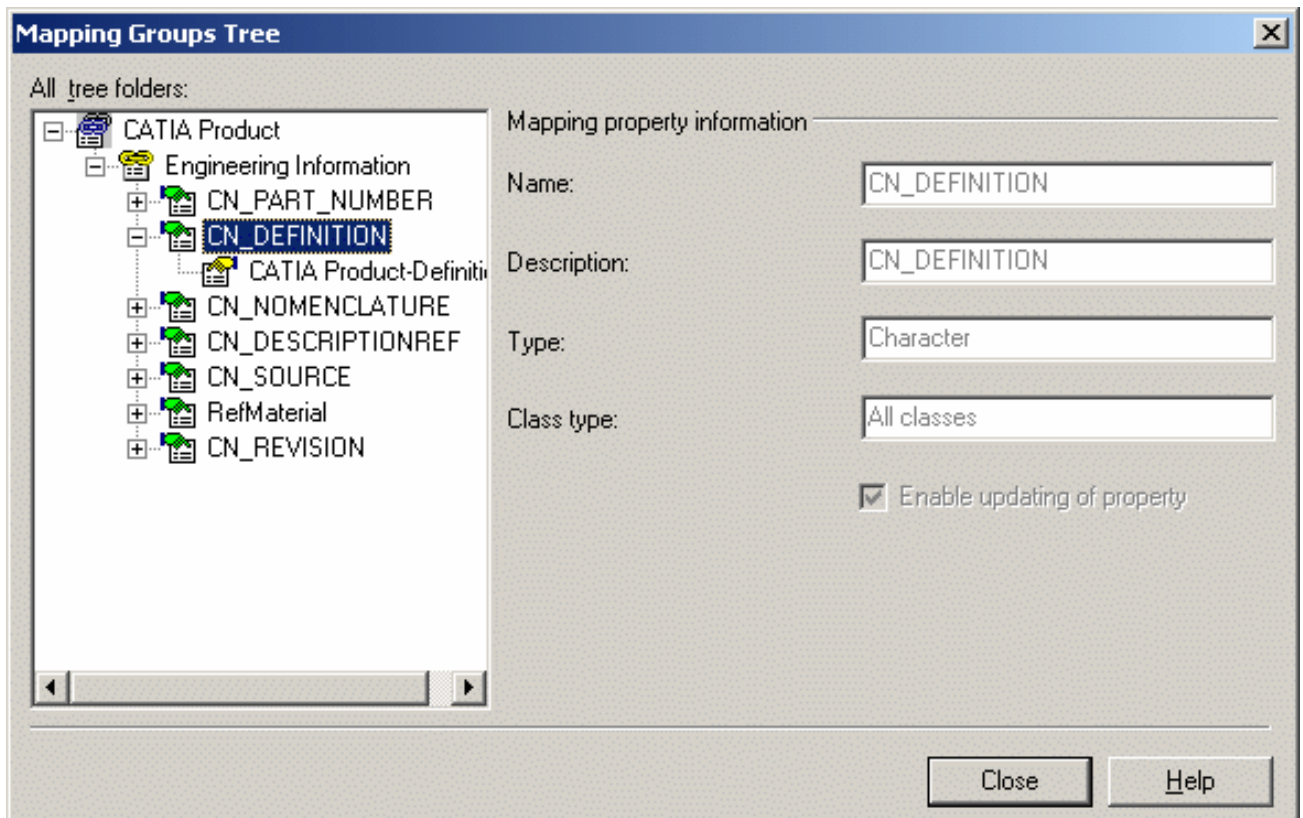
- **CATIA Product**: This indicates that the associated set of mappings are related to documents stored in the CATIA Product class.
- **Read only** : when you define a mapping you also define its direction. **Read only** is one of the options determining the [mapping direction](#).

## Mapping Groups Tree Window

Properties are mapped from the **Mapping Group Tree** window.

1. To access the **Mapping Group Tree** window, select **SmarTeam > Tools > Property Management**.

All property types (or group types) are mapped in the same way.



- **CN\_DEFINITION**: The mapping property name corresponds either to a CATIA formula or a hardcoded CATIA attribute.
- **Enable updating of property**: This is one of the options determining the mapping direction.

## Mapping Direction



For each mapped property, the mapping direction has to be defined:


- **Database > CATIA** i.e. CATIA property values come from the database
- **CATIA > Database** i.e. CATIA property values are written to the database

Note that you can select both mapping directions.

These options determine the mapping direction:

- the CATIA value is stored in the SmarTeam database or,
- The SmarTeam values are applied to CATIA
- The mapping is bidirectional

In the table below you will notice that the property mapping information relates to both the title block **and** the revision block.

Property Name in CATIA 	Attribute Name in SmarTeam	Attribute Type in SmarTeam	Update Direction
Type	Document Type	character	Database > CATIA
Designer	ID	character	Database > CATIA
Date	Creation Date	character	Database > CATIA
Drawing_Title	Title	character	Database > CATIA
Scale	Scale	character	CATIA > Database
Revision	Revision	character	Database > CATIA
Comment	Note	character	Database > CATIA
ApprovalDate	Approval Date	character	Database > CATIA
Authorized	Approved By	character	Database > CATIA

## Hard-coded Property Names



Here is a list of the hard-coded names accessing product properties and sheet information.

Some of them can only be read from the CATIA document. In other words, defining a mapping from SmarTeam to CATIA will not work because these properties are read only on the CATIA side.

Property Name	Read/Write?	Description	Relevant Class
CN_PART_NUMBER	read/write	Contents of the <i>Part Number</i> field	CATIA Part and CATIA Product
CN_REVISION	read/write	Contents of the <i>Revision</i> field	CATIA Part and CATIA Product
CN_DEFINITION	read/write	Contents of the <i>Definition</i> field	CATIA Part and CATIA Product
CN_NOMENCLATURE	read/write	Contents of the <i>Nomenclature</i> field	CATIA Part and CATIA Product
CN_SOURCE	read/write	Contents of the <i>Source</i> field	CATIA Part and CATIA Product
CN_DESCRIPTIONREF	read/write	Contents of the <i>Description</i> field	CATIA Part and CATIA Product
CN_CONTEXTUAL	read	File name of the referenced CATIA Product for a contextual Part	CATIA Part
CN_SHEET_NAME	read	Name of the sheet	CATIA Sheet
CN_SHEET_FORMAT	read	Format of the sheet	CATIA Sheet

CN_VERSION_TO_READ	read	Minimal CATIA version which can be used to open the document  Values are displayed like CATIAVxRy where: x=5 y=6, 7, 8, 9, 10, and so on.	All classes used to store V5 documents
--------------------	------	---	--

Note that surface, mass and volume mapping are no longer documented as it has a negative effect on saving performance.

If such mapping has already been done, remove it from the database as follows:

1. Run the **Integration Tools Setup** utility.
2. In the tree displayed, select **CATIA** then right-click on **Mapping group types**.
3. Right-click on each class one by one (such as **CATIA Product** or **CATIA Part**) and select **Open Group Tree**.
4. Check that neither *CN\_MASS* nor *CN\_VOLUME* nor *CN\_SURFACE* are under **Engineering Information**. If they are, delete the appropriate mapping property information.

## Map a Text Value...



Before using **SmarTeam > Properties > Map a Text Value** (for an example, see [Displaying a CATIA Drawing SmarTeam Attribute in a Title Block](#)), create a group-type corresponding to the name of the class used to store the drawing (this group-type is created using [Integration Tools Setup](#)).

## Performance and Attribute Mapping



Attribute mapping is one of the major element in terms of performance implied for the Save and Open transactions.

Thus, it is advised to clearly identify the needed mapping:

- A previous section describes the importance not to map the mass, volume and surface
- The user should determine whether each type of mapping is needed from CATIA to SmarTeam, from SmarTeam to CATIA or from both directions:
- If required, the revision should be mapped only from SmarTeam to CATIA
- The parameter attributes can be mapped from (and to) CATIA to (and from) SmarTeam but most often only the CATIA to SmarTeam mapping direction is needed.
- The same consideration can be performed on the *CN\_MATERIAL* attribute. Most often, only the CATIA to SmarTeam mapping direction is needed.

Note

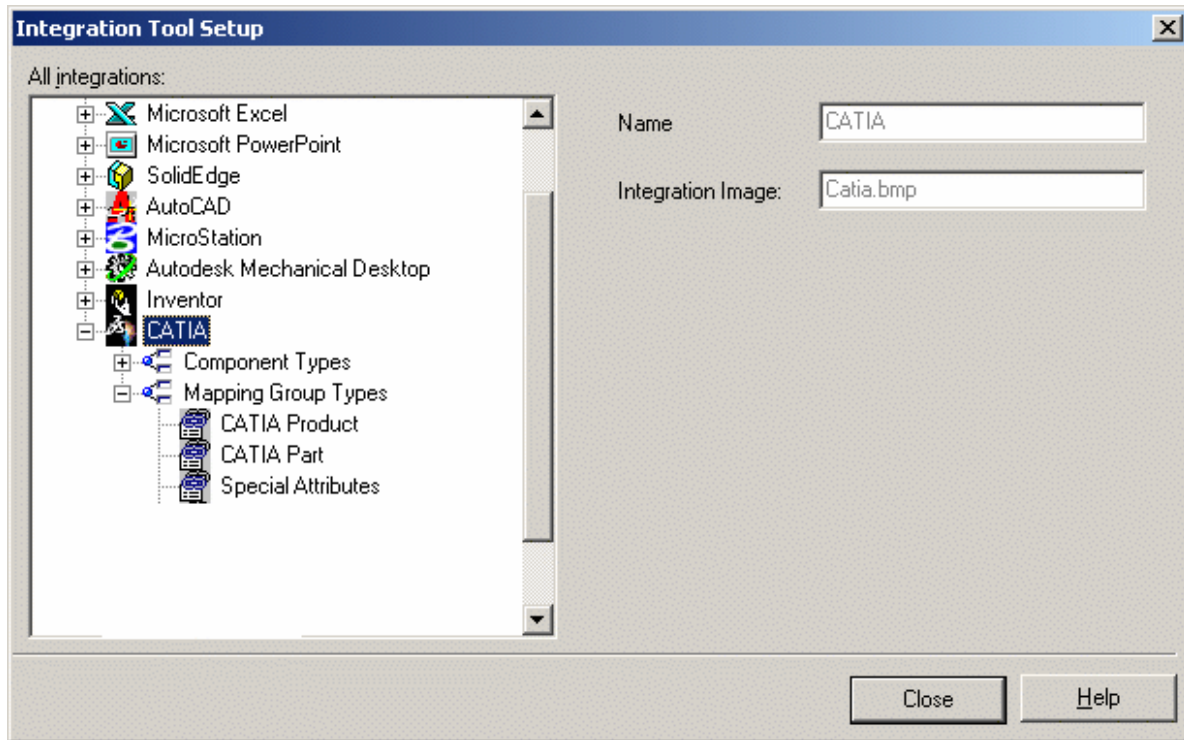
Some attribute mapping operations may require the loading of documents.



# Defining Property Mapping for the Title Block

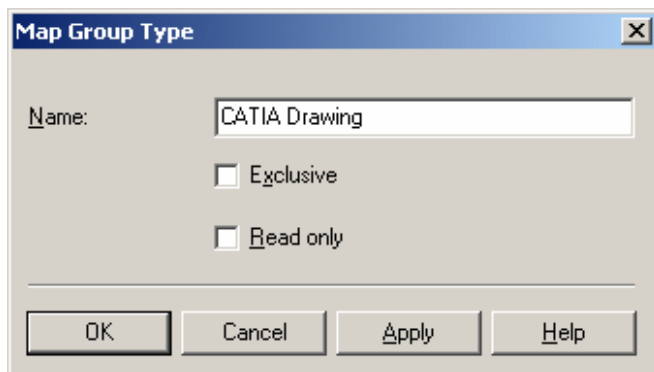
This section shows you how to define property mapping for title blocks.

1. Launch the [Integration Tool Setup](#) utility.  
The Integration Tool Setup dialog box appears:



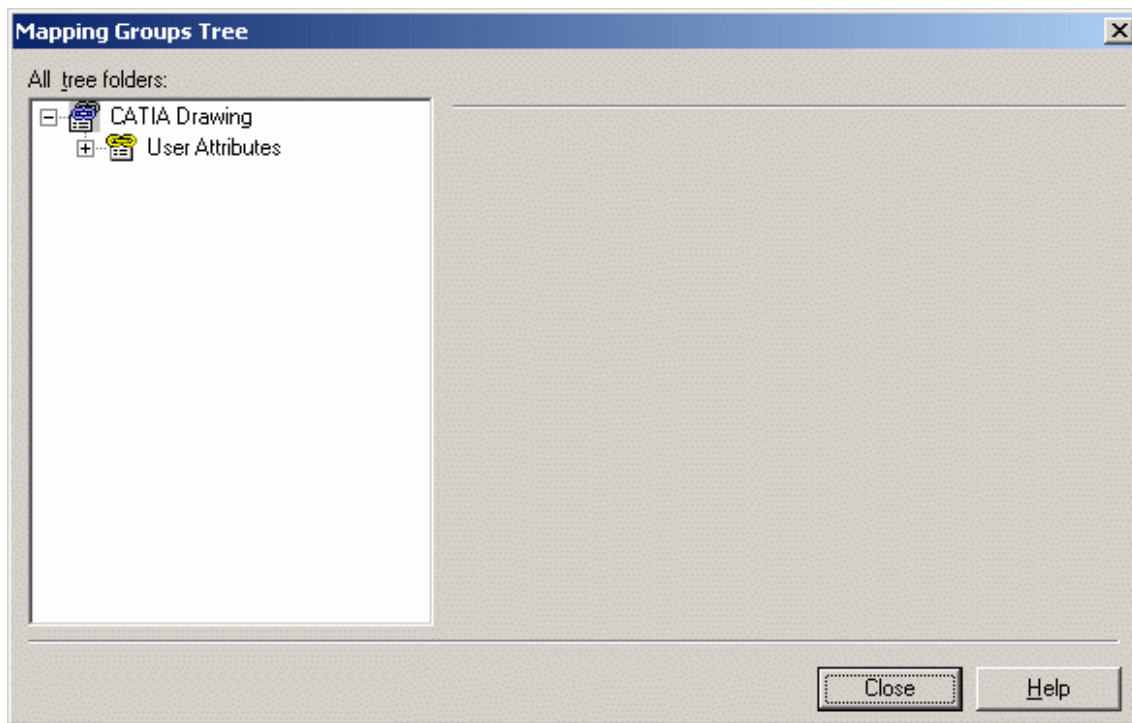
A **mapping group type** i.e. a set of mapping groups, applicable to a specific SmarTeam class must now be defined that corresponds to the CATIA Drawing class.

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 **CATIA Drawing** as shown:

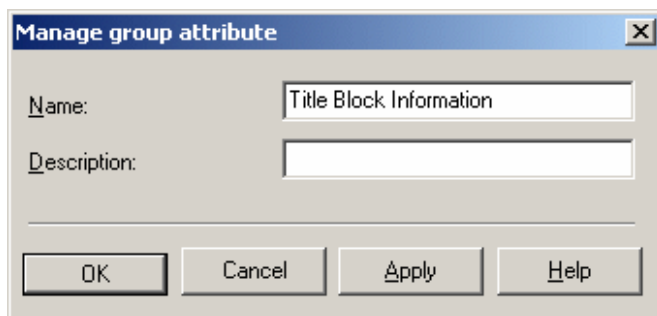


The name of the mapping group type must be identical to that of the SmarTeam Class it applies to.



4. Click **OK**.
5. In the tree, right-click on the **CATIA Drawing** item and select **Open groups tree**.  
The Mapping Groups Tree window appears.




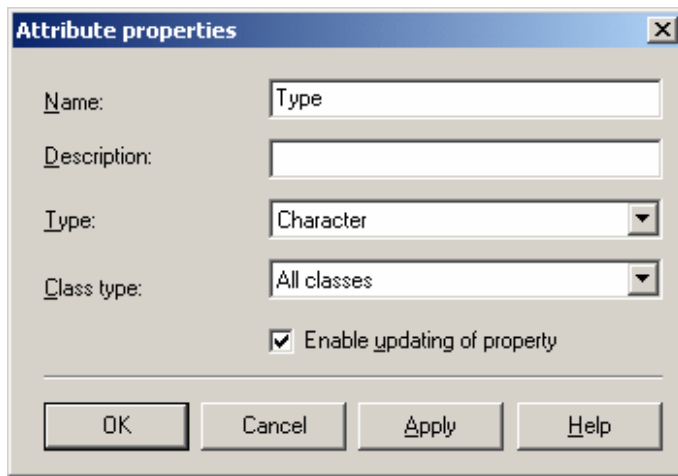
6. In the tree, right-click on **CATIA Drawing groups** and select **Add**.  
The Manage group type window appears.
7. In the **Name:** field, enter a meaningful name, for example *Title Block Information*:



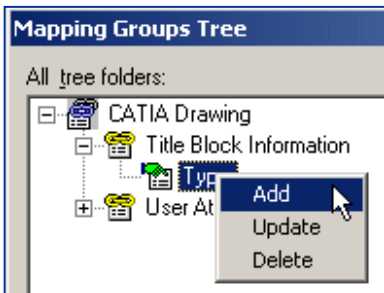
The name used should help you organize the different mapped properties you have to manage.


8. Click **OK**.
9. To define the CATIA Formula property  as *Type*, go to the tree and right-click on *Title Block Information*
10. Select **Add**.  
The Add Title Block Information property dialog box appears.
11. Enter *Type* in the **Name:** field.  
The name of the mapping property must be identical to the corresponding CATIA Formula property .
12. Select **Enable updating of property** if not already selected.

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

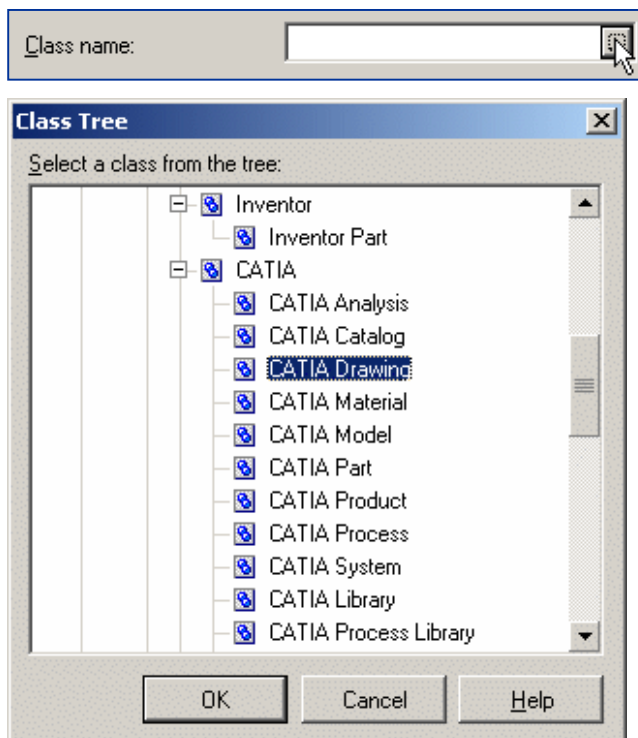


13. Click OK.
14. Go to the tree and right-click on **Type** then select **Add**.

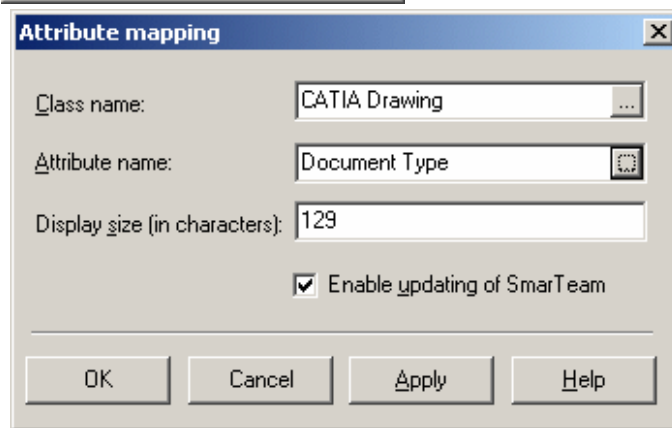
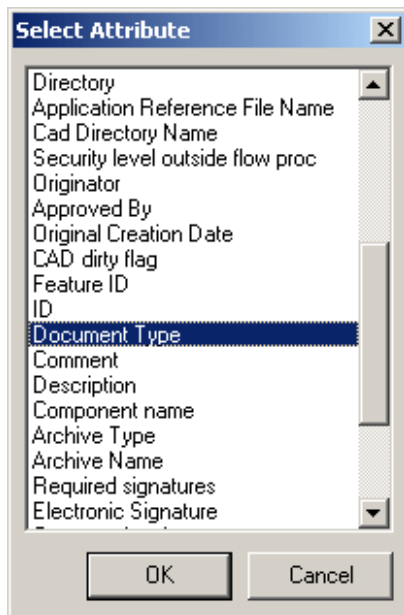


This defines as *Drawing Type* the SmarTeam attribute to be linked to the CATIA Formula property . The Attribute mapping dialog box appears.

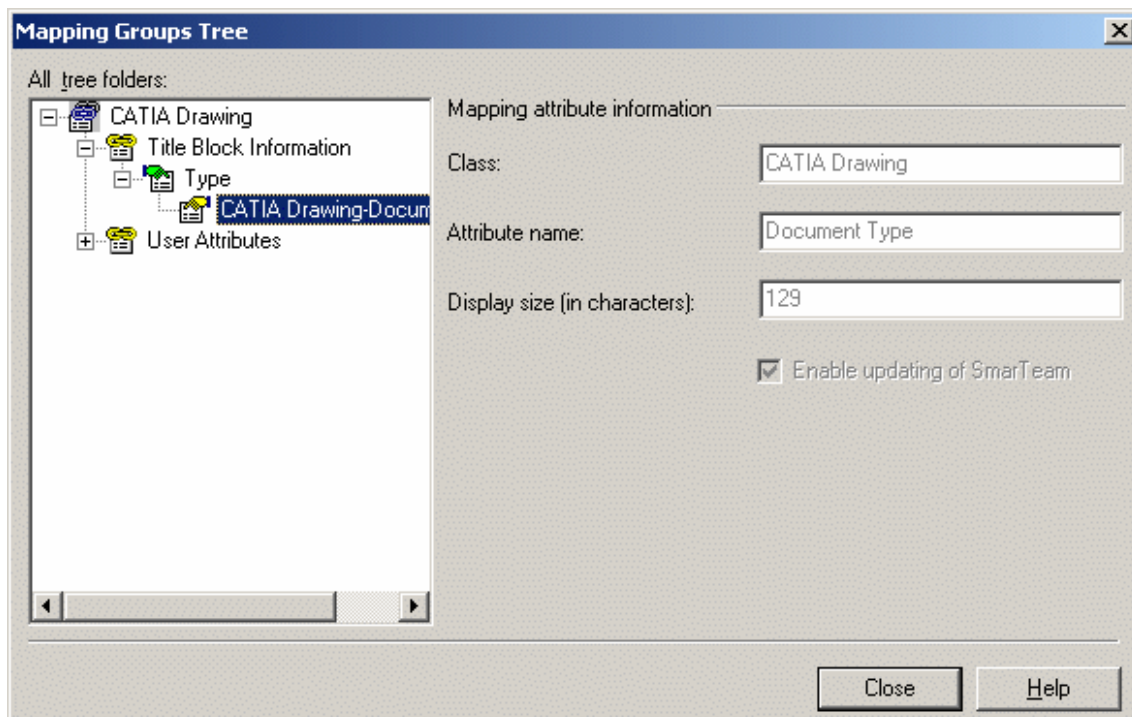
15. Click the button to the right of the **Class name** field and select the class name **CATIA Drawing** from the Class Tree window that appears.



16. Repeat the previous operation for the right of the **Attribute name** field: select the attribute name **Document Type** from the Select Attribute window that appears.  
Note that the list of the possible attributes is specific to the selected class name.



17. Click OK.  
The Mapping Groups Tree window appears:




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

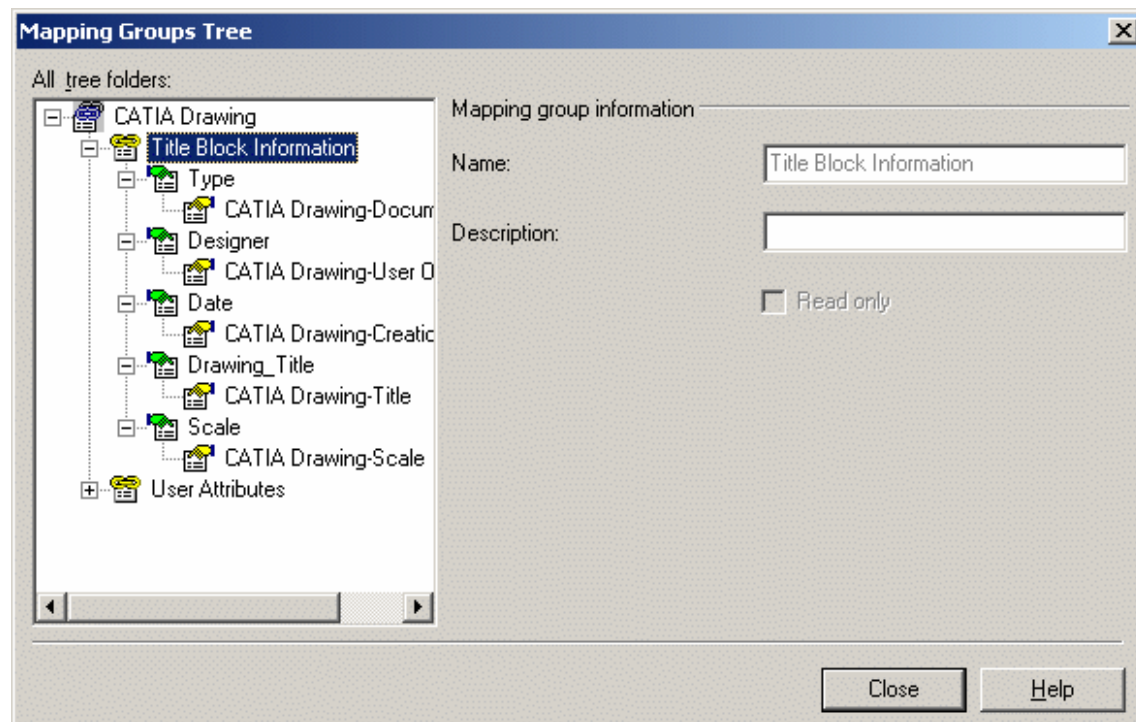
Mapping Property Name	Enable Property Update	Mapping Attribute Name	Enable SmarTeam Update
Type	yes	Document Type	

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
Designer	yes	ID	
Date	yes	Creation Date	
Drawing_Title	yes	Title	
Scale		Scale	yes


Once this has been done the CATIA Drawing 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.

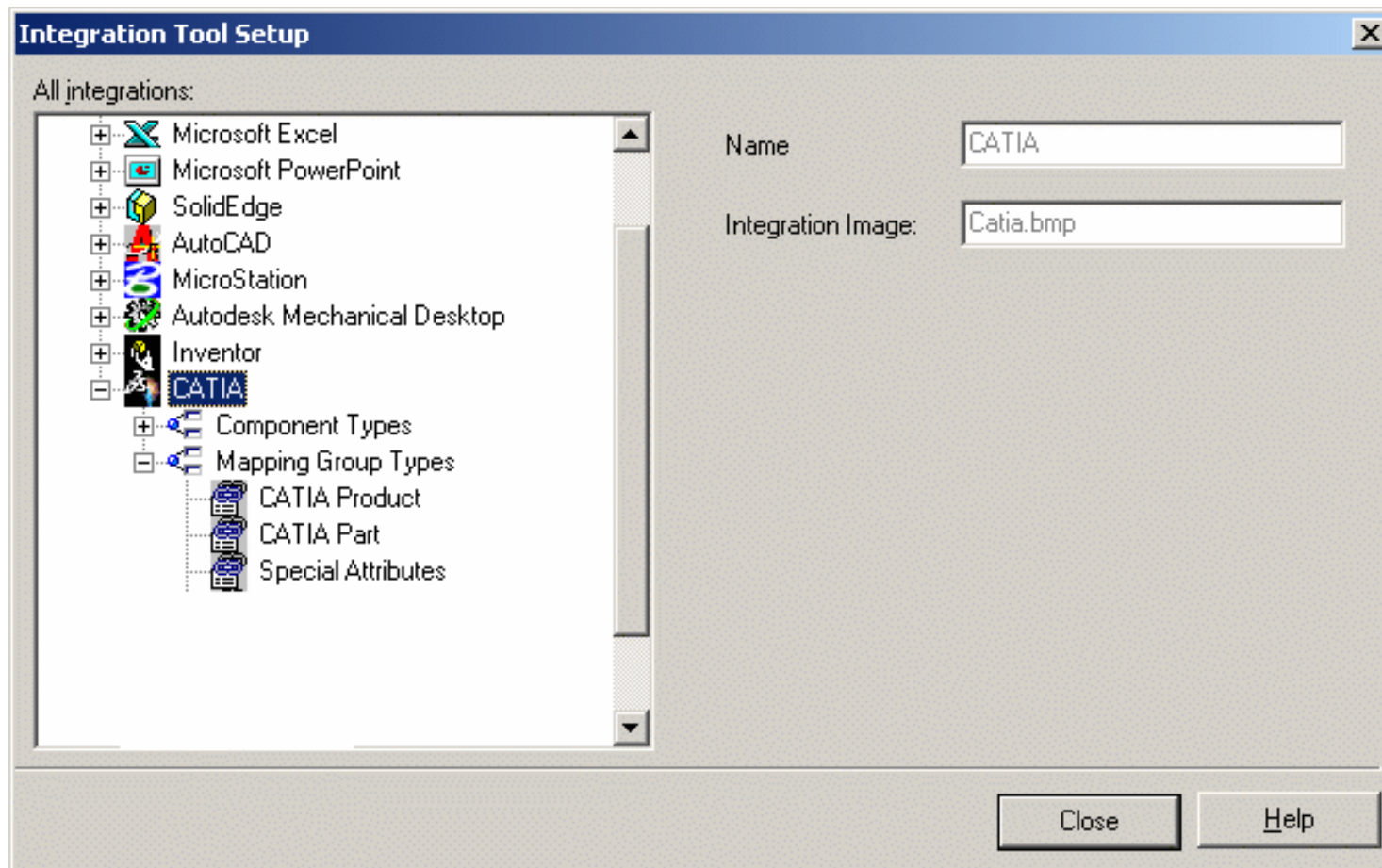


# Defining Property Mapping For the Revision Block

 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:

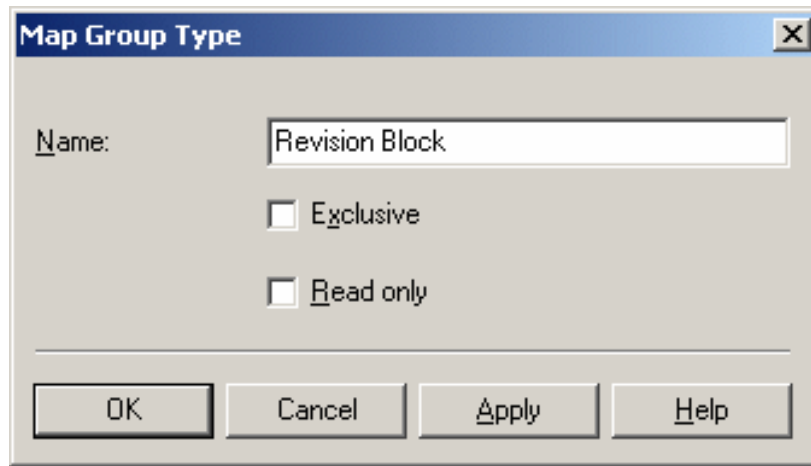


A **mapping group type** i.e. a set of mapping groups, applicable to a specific SmarTeam class must now be defined that corresponds to the CATIA Drawing class.

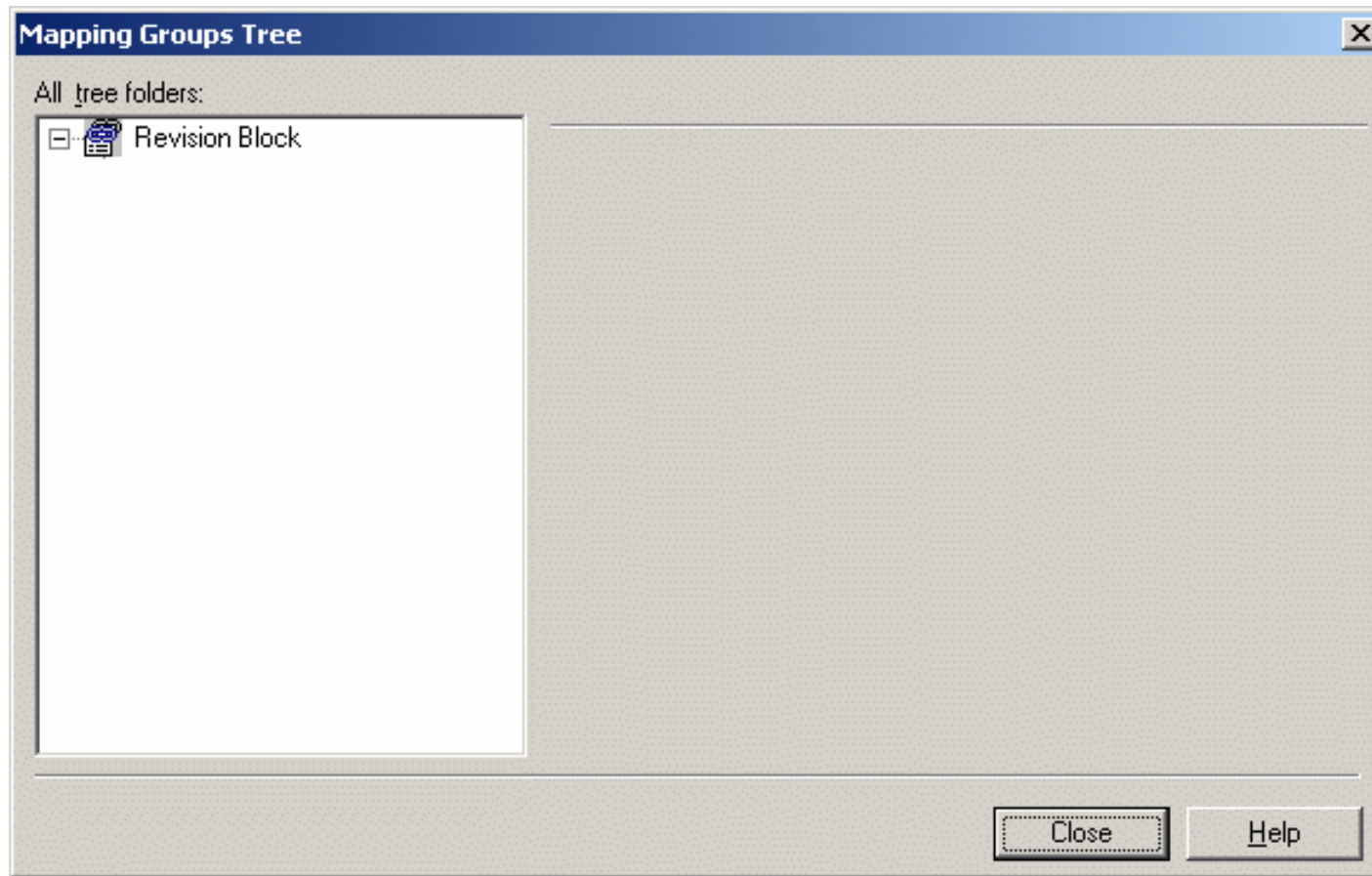
2. In the tree displayed, right-click on **Mapping Group Types** and select **Add Mapping group type**.

The Map Group Type dialog box appears.

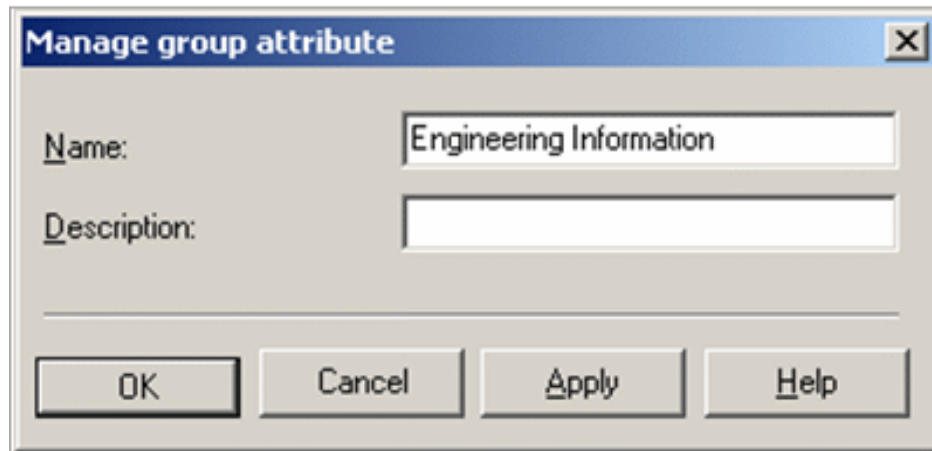
3. In the **Name:** field, enter the group type name *Revision Block* as shown:





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.

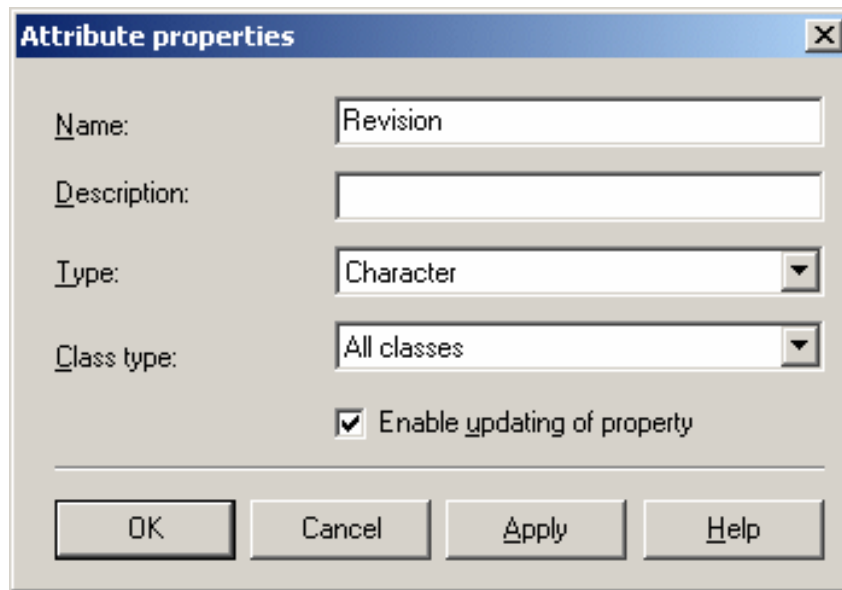


6. In the tree, right-click on **Revision Block groups** and select **Add**.  
The Manage Group Attribute dialog box appears.
7. In the **Name:** field, enter *Engineering Information*:



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 Windows-style dialog box titled "Attribute properties". It has a standard title bar with a close button (X). The dialog contains four labeled text fields: "Name:" with the value "Revision", "Description:" which is empty, "Type:" with a dropdown menu showing "Character", and "Class type:" with a dropdown menu showing "All classes". Below these fields is a checked checkbox labeled "Enable updating of property". At the bottom of the dialog are four buttons: "OK", "Cancel", "Apply", and "Help".

**Attribute properties**

Name: Revision

Description:


Type: Character

Class type: All classes

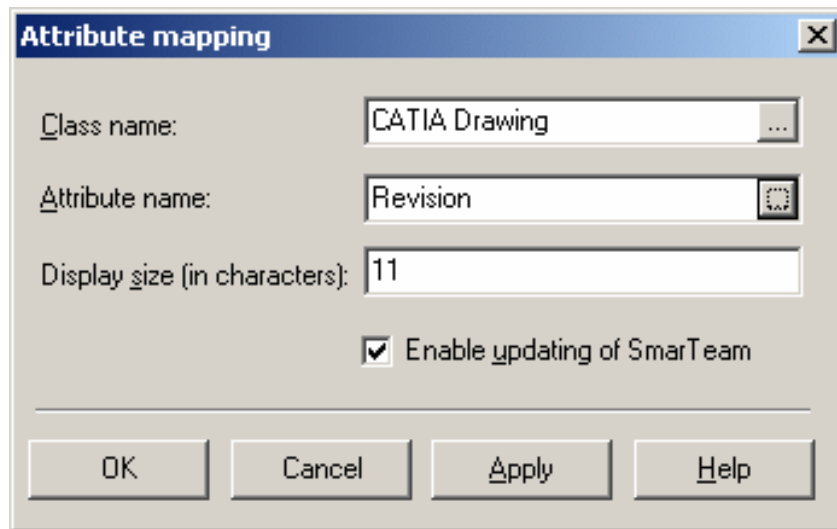
☒ Enable updating of property

OK Cancel Apply Help

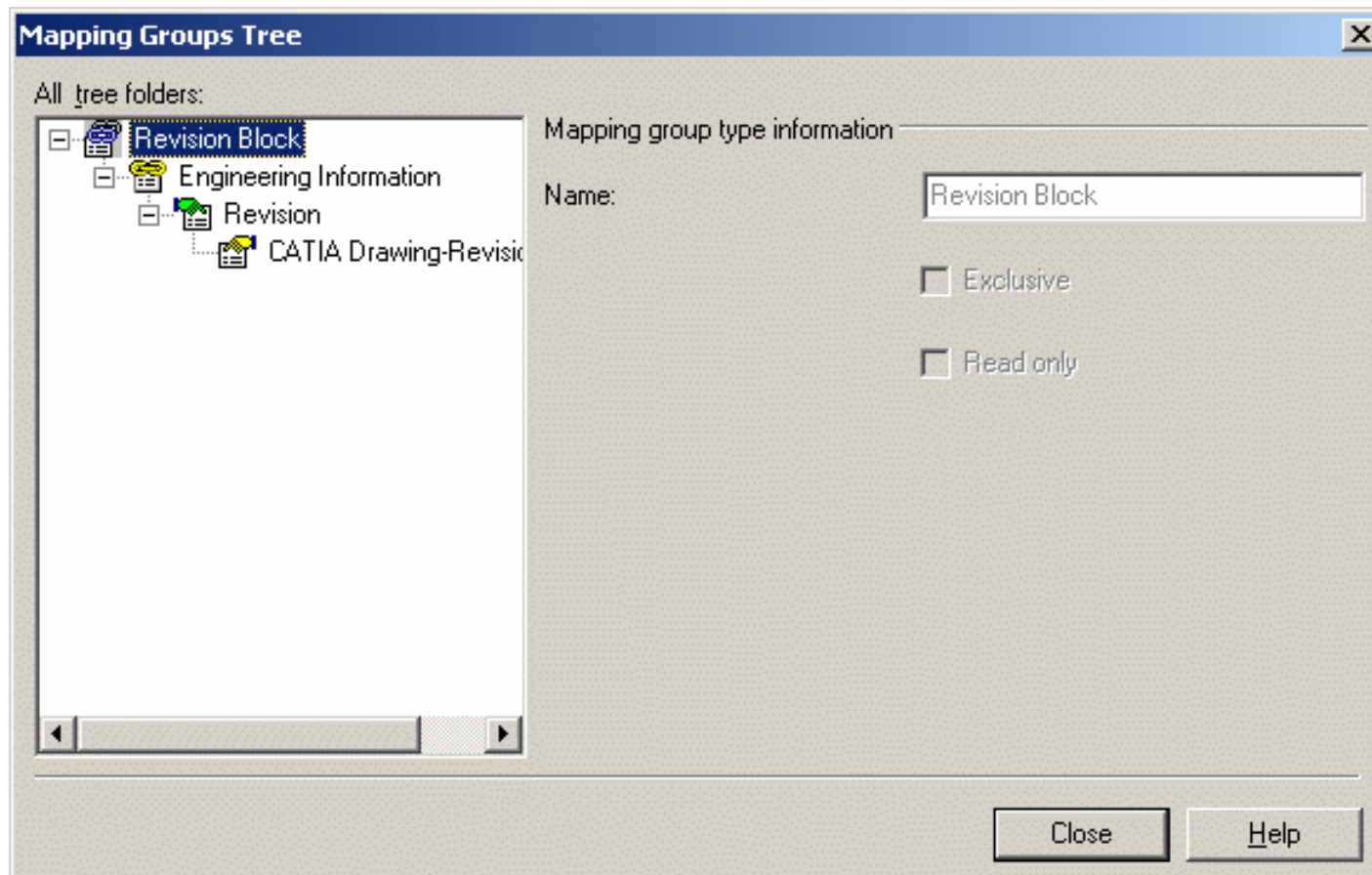
13. Click **OK**.
14. Go to the tree and right-click on **Revision** then select **Add**.

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	

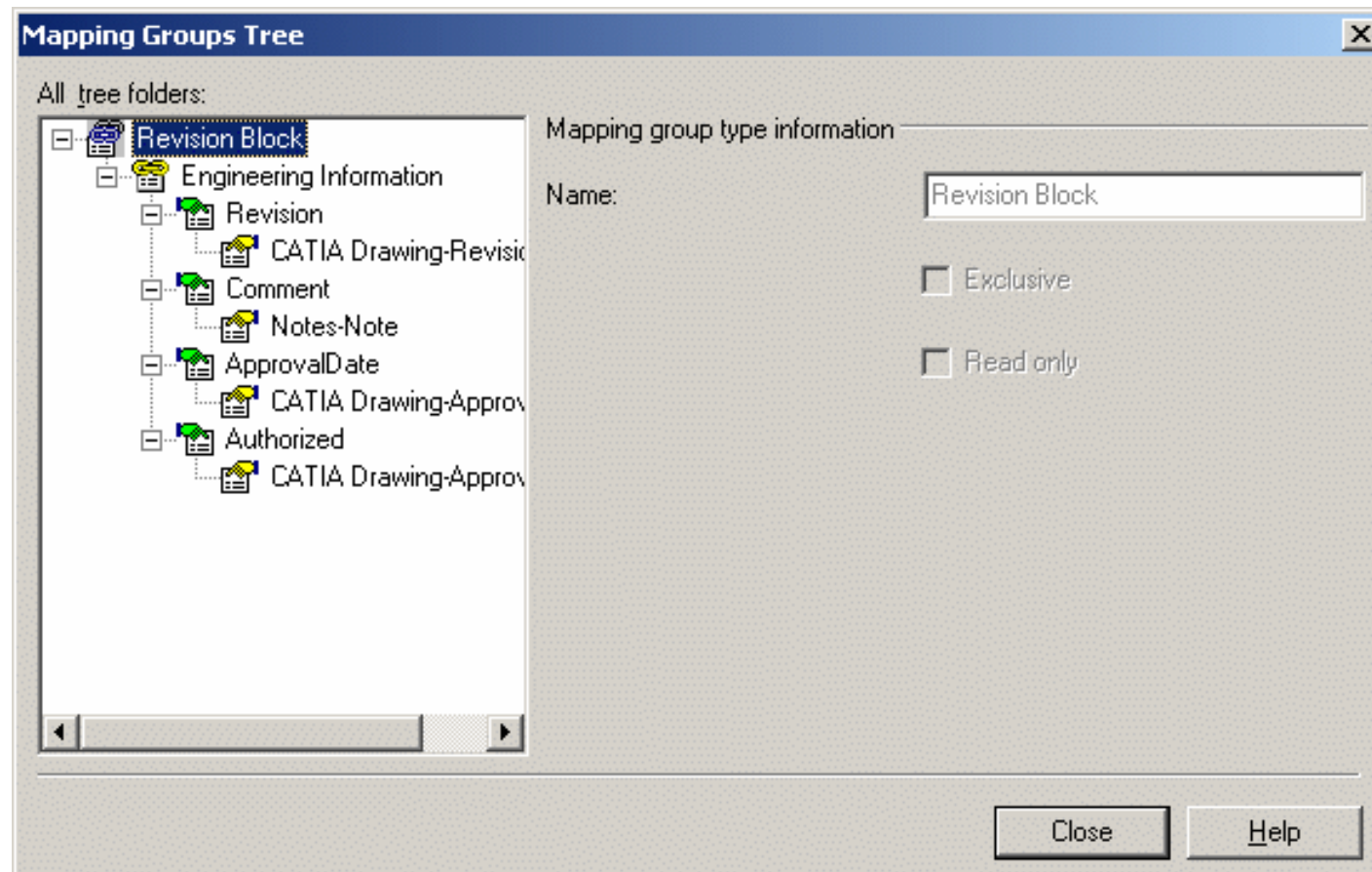
18. Repeat steps 9 through 16 for all other mapped properties.

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.



# Defining Property Mapping For CATIA Products and CATIA Parts



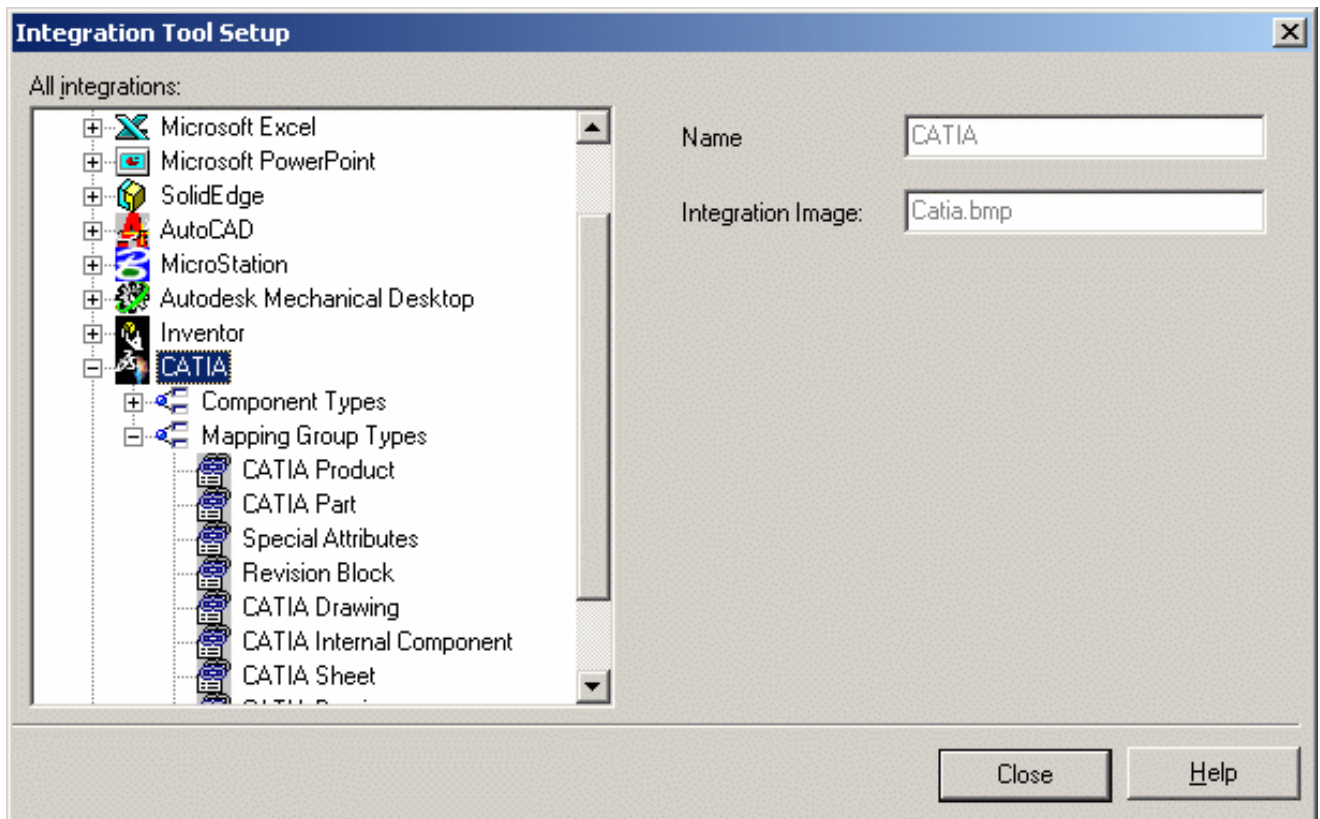
This section describes how to define property mapping for CATIA products and CATIA parts.



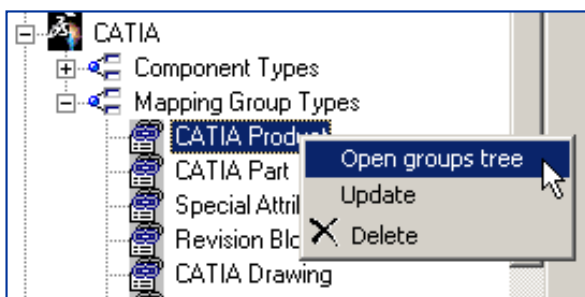
It is assumed that the properties used for this task have already been created in the database under the *CATIA Product* and *CATIA Part* classes.



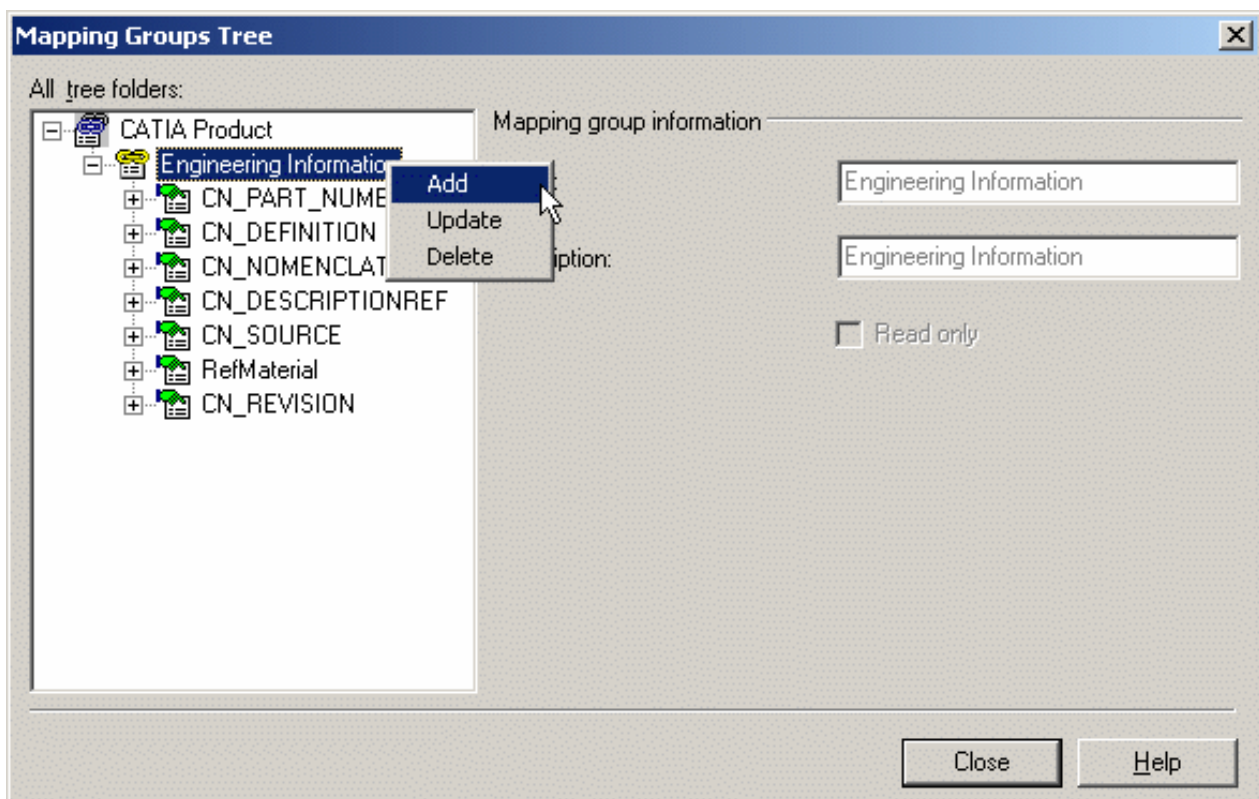
1. Launch the [Integration Tool Setup](#) utility.  
The **Integration Tool Setup** dialog box appears.
2. Double-click on CATIA or select the "+" sign to display the options associated with CATIA.
3. Double-click **Mapping Group Types** or select the "+" sign in front of it.



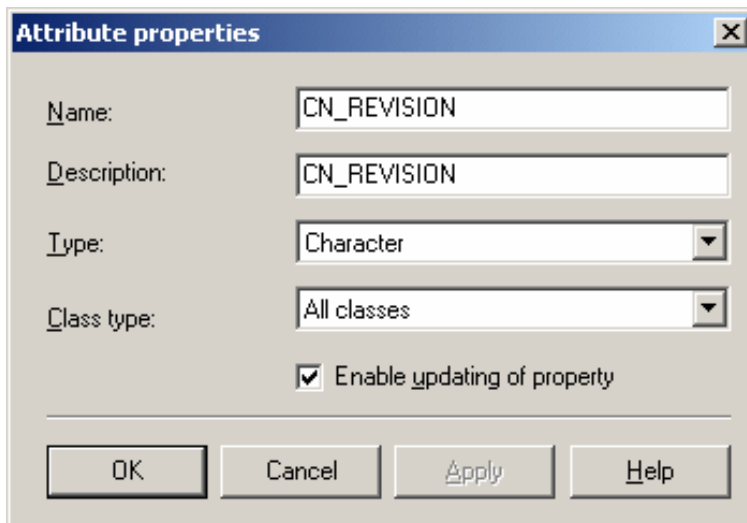
4. Select the **CATIA Product** mapping group type then right-click and select **Open groups tree**:



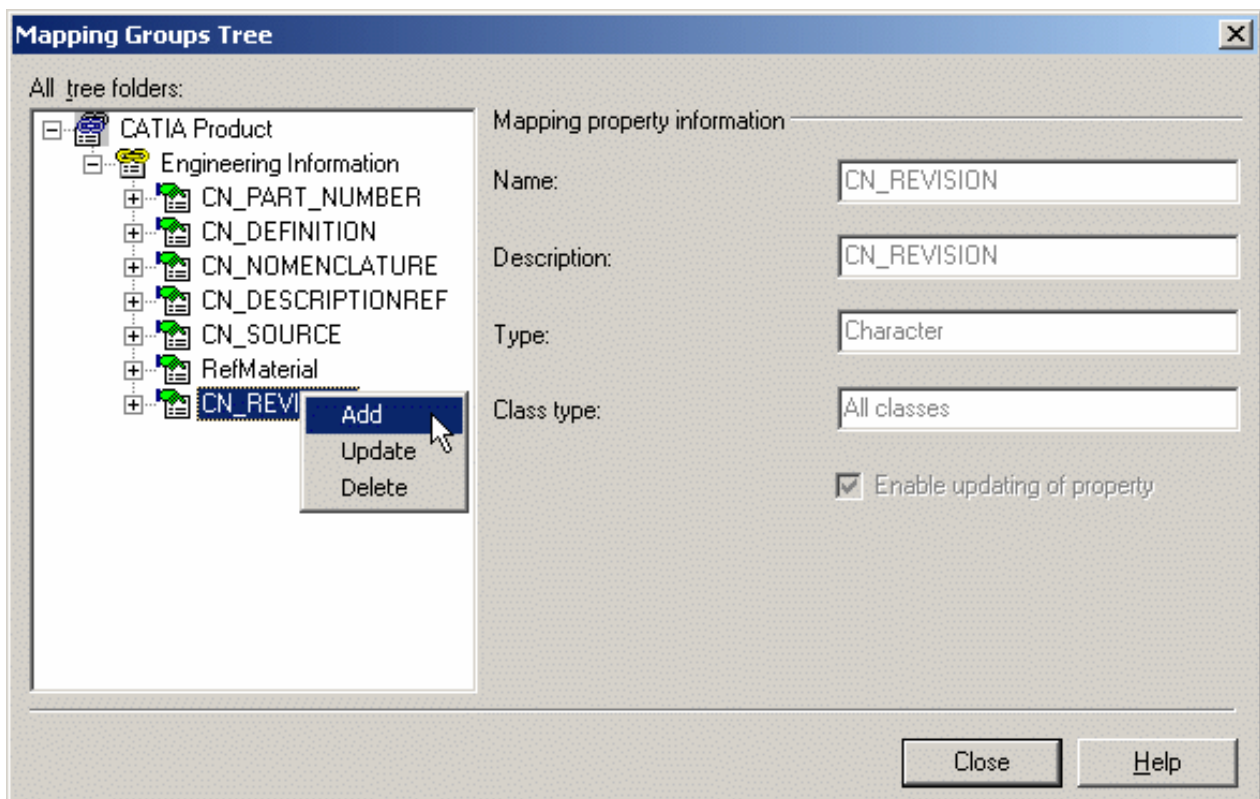
5. In the **Mapping Groups Tree** window, select the **Engineering Information** item then right-click and select **Add**.



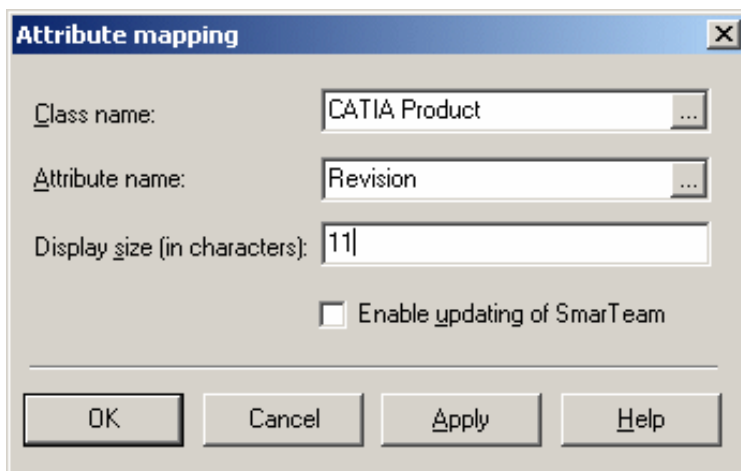
6. In the **Attribute Properties** dialog box, complete all fields as shown below, then click on **OK**:



7. Back in the **Mapping Groups Tree** dialog box, right-click *CN\_REVISION* then select **Add**.



8. In the **Attribute Mapping** window, complete the **Class name** and **Attribute name** field as shown below then click on **OK**.



9. **Recommendation:** When you assign a component type such as "CATIA Part" to two classes, e.g., CATIA Part and CATIA Standard Part in **Integration Tools Setup**, attribute mapping is performed for all of the attributes of the assigned classes (duplicates inclusive), regardless of the "Save as" class chosen during the Save operation.

If the mapping direction of an attribute in one class is disabled and the same attribute exists in the other class with a mapping direction enabled, the mapping direction of the latter is considered.

Note: This behavior occurs only during the first Save of the document in SmartTeam.



# Recommendations

The purpose of this section is to provide a certain number of recommendations to system administrators so as to facilitate the integration of SmarTeam with CATIA.

# Lifecycle Settings



## Lifecycle in SmarTeam

SmarTeam enables you to maintain and manage any information related to a revision manageable object throughout its Lifecycle. By mirroring the physical process of product management, SmarTeam uses vaults, check in, check out, and release functions to manage the Lifecycle revision manageable objects. It creates new versions of a file and protects it from unauthorized modifications.

## Lifecycle for CATIA Designers

CATIA designers use lifecycle for a number of purposes:

1. Sharing between members of a Design group.
2. Saving of modification history.
3. Approving and releasing.

SmarTeam lifecycle has been designed to allow working with large assemblies with various references between different CATIA documents. It takes into consideration the designer's need to receive the necessary documents with the correct revisions.

## Lifecycle Settings

In the Lifecycle Settings, you can define different company rules for designers. These rules define how designers should perform lifecycle operations for their design and reference documents.

The following preferences can be defined within the Lifecycle Settings:

### General Settings

- Enable/disable user-defined revisions.
- Lifecycle dialog type (no dialog, light lifecycle screen, advanced lifecycle screen)
- Last public revision note

### In Vault Operations

- Allow replacement of previous revision
- Enable/disable the Always Check in/Release of the latest available revision
- Show/hide the revision report when a revision is replaced in the lifecycle

## Out of Vault Operations

- Restrict new release operation when derived revision exists
- Allow branching
- File override policy in user's work directory
- Enable/disable check out of latest available revision

## Defining Lifecycle Settings

CATIA users depend on the document lifecycle and therefore proper settings for the lifecycle management will assist in streamlining and optimizing their work.

To obtain optimum behavior for file dependencies, is recommended to work with SmarTeam CATIA Integration V5R12 or later.

Lifecycles are part of the daily work of a CATIA user. The correct setting of lifecycle parameters can greatly assist the streamlining of engineers work.

## Lifecycle Settings Review

### Working with latest revision

Depending on the user's role and stage of the product development, users might have a different requirement for seeing by default a revision of a CATIA document. For example, design engineers usually need to see the latest revision, where production engineers needs to see the latest released revision.

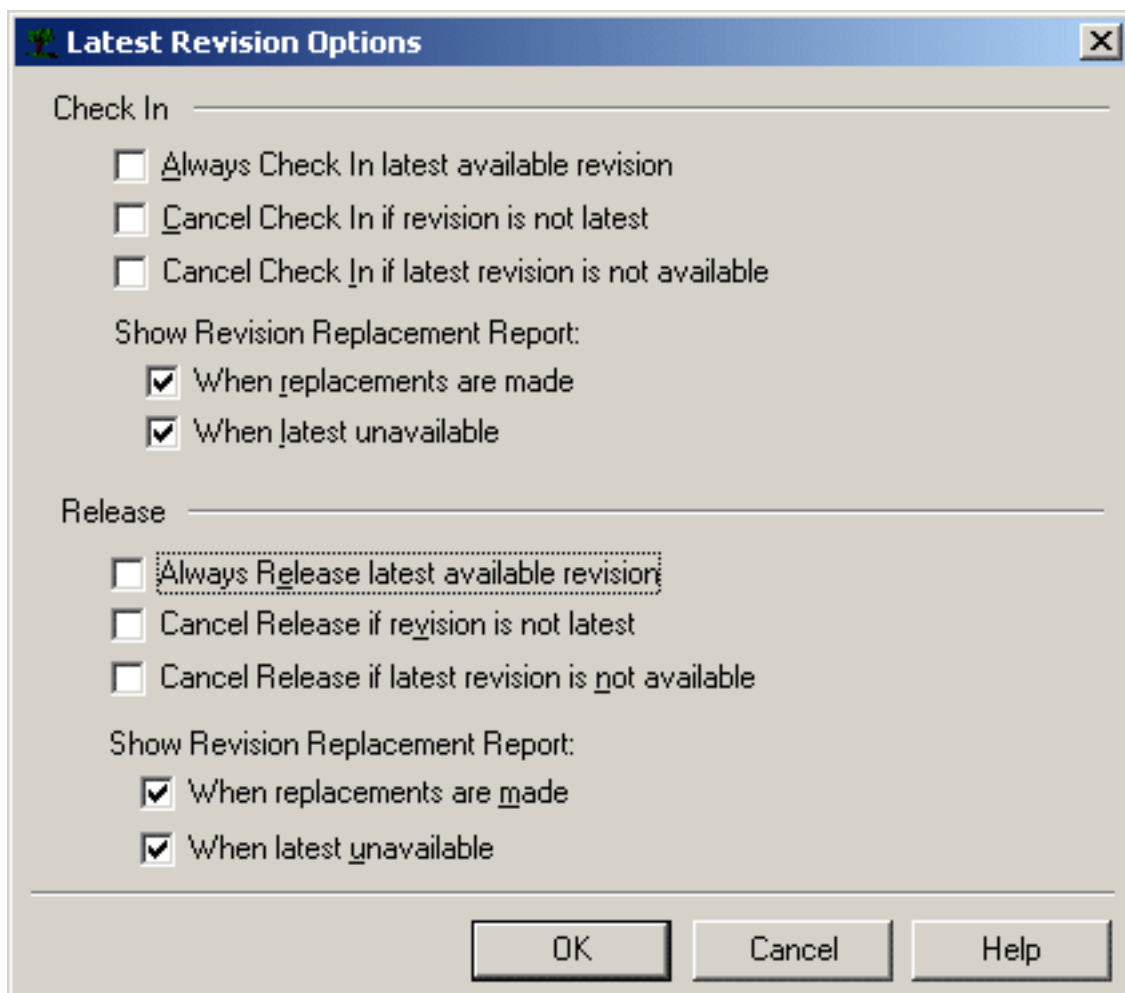
If the **Always Check in Latest Available Revision** preference is selected, automatic replacement will be done. However, the CATIA product will reference the components that may never have been saved in CATIA. This may lead to problems at the CATIA level.

**Recommendation:** To ensure integrity of the vaulted CATIA products documents, do not allow automatic Replace to Latest Revision for check in and release operations. The user should update his product to the latest revisions of the sub-components before checking it into the vault. Before performing a check in/release operation, the user can replace all Not Latest products/parts in


the product using Replace with selected revision operation from SmarTeam/ Assembly Management menu in CATIA.

## How to setup Latest Revision Options

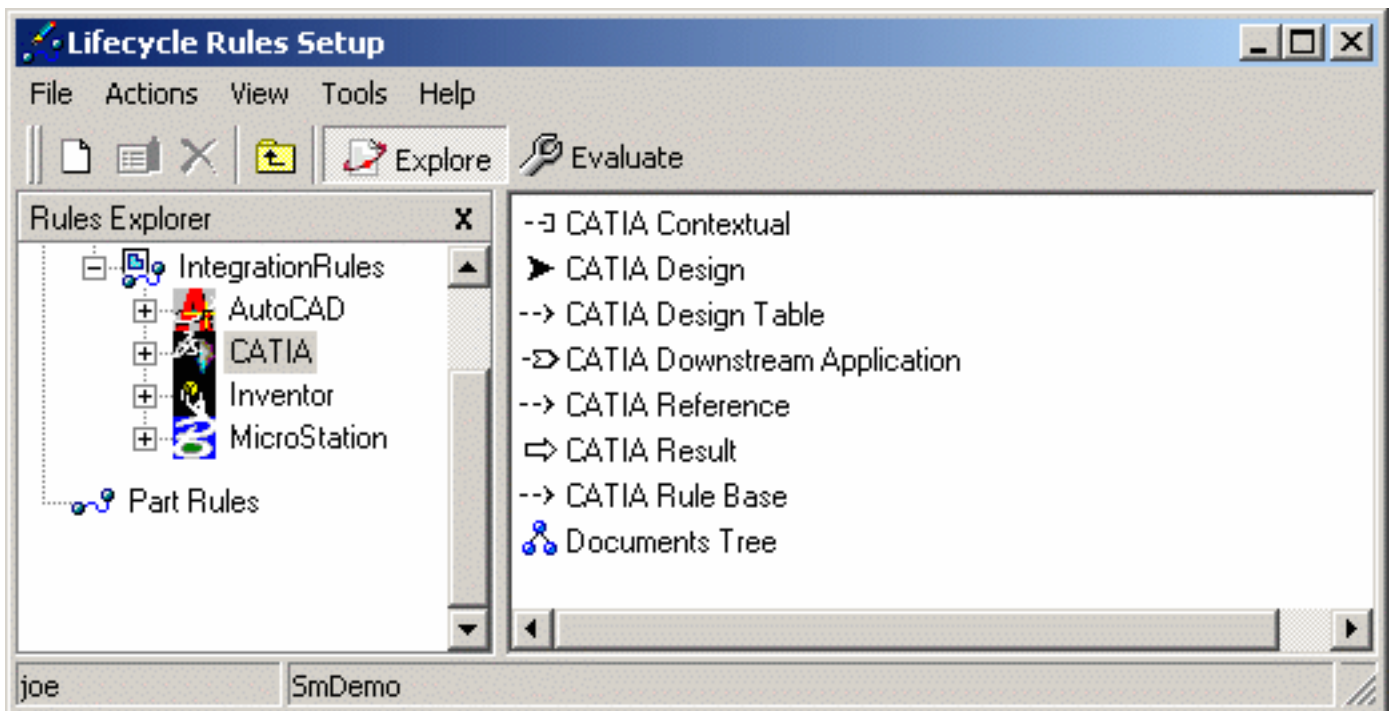
1. From administrator options, select **Lifecycle Options**.  
The **Life Cycle Options** dialog box is displayed.
2. Click the **Into vault** tab.
3. Click **Latest Revision...**  
The **Latest Revision Options** dialog box is displayed.
4. Unselect:
  - **Always Check In latest available revision**
  - **Always Release latest available revision.**



# Lifecycle Rules Setup

 **Lifecycle Rules Setup** defines rules for the Lifecycle operations on linked objects throughout a Lifecycle Operation (as a result of an operation performed on a specific object). For example, an administrator can define that if a user performs a check in operation on a CATIA Drawing, the CATIA Part will automatically be set to check in.

**Lifecycle Rules Setup** is an administrative application tool that enables users to define and manage rules for operations performed on linked objects.



Using this tool, you can:

- Review default rules
- Modify rules provided by specific Integration applications
- Add, remove or modify custom rules

## Recommendations

Lifecycle rules depend on customer processes and have to match and reflect these processes. To know the different types of lifecycle operations which can be performed on a CATIA object, see [Lifecycle Operations](#).

However, some settings are mandatory. You must set lifecycle rule setup options for **Copy File** and **Check Out** operations as indicated in [Life-Cycle Rule Set Up Options For Copy File and Check Out Operations](#).



# Lifecycle Operations



All lifecycle operations on CATIA documents must be performed through the CATIA user interface. Lifecycle operations from SmarTeam are not supported and can affect the session and data consistency.

As defined via the [Lifecycle Rules Setup](#) tool, the following types of lifecycle operations (as shown in the Lifecycle Rule Properties window) can be performed on a CATIA object:

These properties are described below:

Field	Description
Link Class	Defines the link between two SmarTeam classes.
Direction	Defines the direction of the link object, which can be: <ul style="list-style-type: none"><li>• Normal or</li><li>• Reverse</li></ul>
Original Component Type or SmarTeam Class Destination Component Type or SmarTeam class Original Operation	Defines the class or component type of the object on which you perform the Lifecycle operation. Defines the class or component type of the dependent object  Defines the operation on the source object. All SmarTeam Lifecycle operations are available.

<b>Destination Operation</b>	Defines the operation on the dependent object as a result of the operation on the original object. The availability of the destination operation depends on the selected original operation. For example, if the original operation is Check Out then the available destination operations are Check Out, Copy File or No Operation.
<b>Link Operation (Copy Link / No operation)</b>	<p>This parameter specifies how the system should handle linked objects between two objects when new revisions are created. <b>Copy Link</b> means copy the link between old revisions with all its attribute values to new objects, and <b>No operation</b> means create new revisions without creating a link between them.</p> <p>This option is available only when the original operation is in <b>Check Out</b> or <b>New Release</b> mode.</p> <ul style="list-style-type: none"> <li>• When you define <b>No operation</b>, SmarTeam will not link between the original and destination object. This option works in addition to SmarTeam options for Copy hierarchical link and Copy general link.</li> <li>• When you set <b>Link Operation</b> to <b>Copy Link</b>, SmarTeam enables you to perform this operation between the original and the destination objects.</li> </ul>
<b>Switch to Latest</b>	<p>Defines whether to allow switching to the latest revision of the object. If a new revision already exists, this option works in addition to SmarTeam 'S Lifecycle options (Check In and Check Out from the vault). When this option is set to <b>Allowed</b>, SmarTeam allows you to switch to the latest revision of the object.</p> <p>This option is not available when the original operation is <b>Obsolete</b>.</p>
<b>Propagation Allowed</b>	<p>Defines whether to allow the Propagation operation to the destination object, if you want to perform a propagation operation during the Lifecycle operation.</p> <p>This option is not available when the original operation is <b>Obsolete</b>.</p>
<b>Check Destination Object Status</b>	Defines whether to check the status of the destination object. This option replaces previous versions of SmarTeam Lifecycle options, of the Child level. This option is available only when the original operation is <b>Check In</b> or <b>Release</b> mode.

**NOTE:** To perform lifecycle operations, the user must have read-write permission to the location in which the file resides. If the directory is read-only, the file catalog cannot be created and the file is not recognized by SmarTeam.



**Releasing a checked-in document from CATIA requires SmarTeam Update rights on that Checked In document.**

This right is only required if some property mapping is defined from SmarTeam to CATIA. In such a case, before releasing the document, CATIA needs to update the CATIA mapped properties according to the modified SmarTeam attributes. Then the document modification has to be reported in SmarTeam by updating the hidden TDM\_CAD\_DIRTY\_FLAG attribute in order to insure integration consistency.



# Reviewing Lifecycle Rules for CATIA Links



## Types of Links

There are different types of links in the lifecycle rules:

- Documents Tree
- Design Link
- Contextual Link
- Downstream Application

## Documents Tree

The Documents Tree builds the product structure and links between the CATIA product and the CATIA sub-products and parts.

- **Check Out:** When a user checks out a CATIA product, all CATIA parts and sub-products that build the structure of the product must be copied. To do this, the administrator needs to set the designation operation to Copy File.
- **Check In:** When a user checks in a CATIA product, all parts and sub-products in this structure must be checked in. To do this, the administrator needs to set the designation operation to Check In.
- **Release:** When a user releases a CATIA product, all parts and sub-products in this structure must be released. To do this, the administrator needs to set the designation operation to Release.

Scenario:

In a company, an individual user uses parts of a CATIA product that belong to other users.

If the user performs a Release operation on a product, it automatically performs a Release operation on the downstream parts, and this may lead to the release of CATIA parts or products designed by another user.

In order to avoid this situation, upon a Release operation, set the designation operation to No Operation and mark the Designation Status check box. When the user tries to perform a Release operation, he is informed that he must manually release each downstream part. If he does not do this, an error will appear.

This ensures that the user is aware of each document he releases and is not simply performing an automatic release.

- **New Release:** When a user performs a new release a CATIA product, all CATIA parts and sub-products that build the structure of the product must be copied. To do this, the administrator needs to set the designation operation to Copy File.
- **Copy File:** When a user performs a Copy File of a CATIA product, the administrator needs to set the designation operation to Copy File.

## Design Link:

- **Check Out:** When a user checks out a CATIA document, all related documents must be copied. To do this, the administrator needs to set the designation operation to Copy File. This ensures that the latest revisions of the files are received.
- **Check In:** When a user checks in a CATIA product, all parts and sub-products in this structure must be checked in. To do this, the administrator needs to set the designation operation to Check In. This ensures that both documents will be checked in. In some organizations, this constraint is too high as

they need to check in the CATIA document without checking in related documents. In this case, the administrator can set the designation operation to No Operation.

- **Release:** When a user checks in a CATIA product, all parts and sub-products in this structure must be checked in. To do this, the administrator needs to set the designation operation to Check In. This ensures that both documents will be checked in.
- **New Release:** When a user checks out a CATIA document, all related documents must be copied. To do this, the administrator needs to set the designation operation to Copy File. This ensures that the latest revisions of the files are received.

## Contextual Link

CATIA contextual links create dependencies between files such that a part feature may be dependent on information stored in a product.

In the example above, notice that a check out operation in the normal direction results in the linked object being copied (Copy File). In the reverse direction, the check out operation results in no operation on linked object (No Operation).

The Copy Link setting allows you to establish a link to the object in the context of the component built. This is important if you want to automatically replace the object with the latest revision of the linked component.

- **Check Out:** When checking out, set the designation operation to Copy File to ensure that links will not be lost when checking in again.
- **Check In:** When a user checks in a CATIA product, all parts and sub-products in this structure must also be checked in. To do this, the administrator needs to set the designation operation to Check In. This ensures that both documents will be checked in. In some organizations, this constraint is too high as they need to check in the CATIA document without checking in related documents. In this case, the administrator can set the designation operation to No Operation.
- **Release:** When a user checks in a CATIA product, all parts and sub-products in this structure must also be checked in. To do this, the administrator needs to set the designation operation to Check In. This ensures that both documents will be checked in.
- **New Release:** When checking out, set the designation operation to Copy File to ensure that links will not be lost when checking in again.

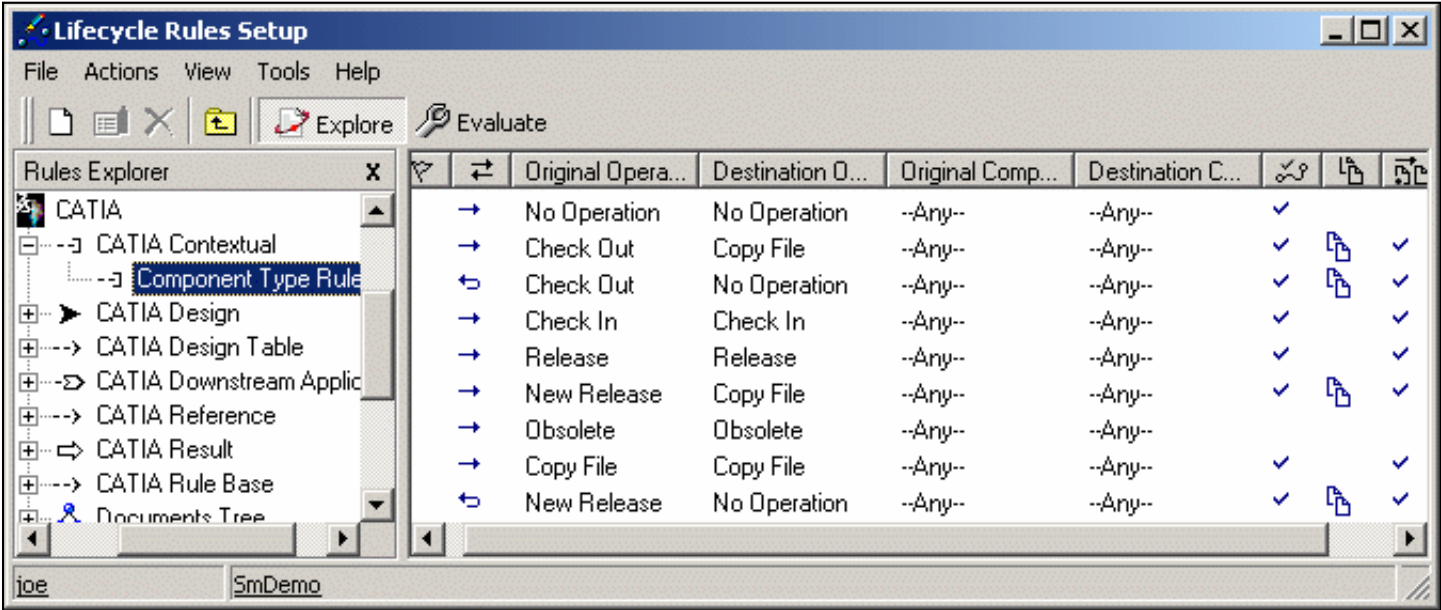
## Downstream Application

- **Check Out:** When a user checks out a CATIA Drawing, all CATIA parts and sub-products that build the structure of the drawing must be copied. To do this, the administrator needs to set the designation operation to Copy File.
- **Check In:** When a user checks in a CATIA product, all parts and sub-products in this structure must be checked in. To do this, the administrator needs to set the designation operation to Check In.
- **Release:** When a user releases a CATIA product, it is necessary to release all parts and sub-products in this structure. To do this, the administrator needs to set the designation operation to Release.
- **New Release:** When a user checks out a CATIA Drawing, all CATIA parts and sub-products that build the structure of the drawing must be copied. To do this, the administrator needs to set the designation operation to Copy File.
- **Copy File:** When a user performs a Copy File of a CATIA product, the administrator needs to set the designation operation to Copy File.
- **Reverse Link:** Some organizations require that when a user checks out a CATIA part or product, the related drawings must also be checked out to reflect modifications in the drawing. To ensure this, the administrator can set reverse links for Check Out and new release operations for downstream applications. For downstream applications, if a check out or new release operation is performed on a part; its drawings must also be checked out automatically. Therefore, the designation operation must be set to Check Out for checking out and to New Release for new release.

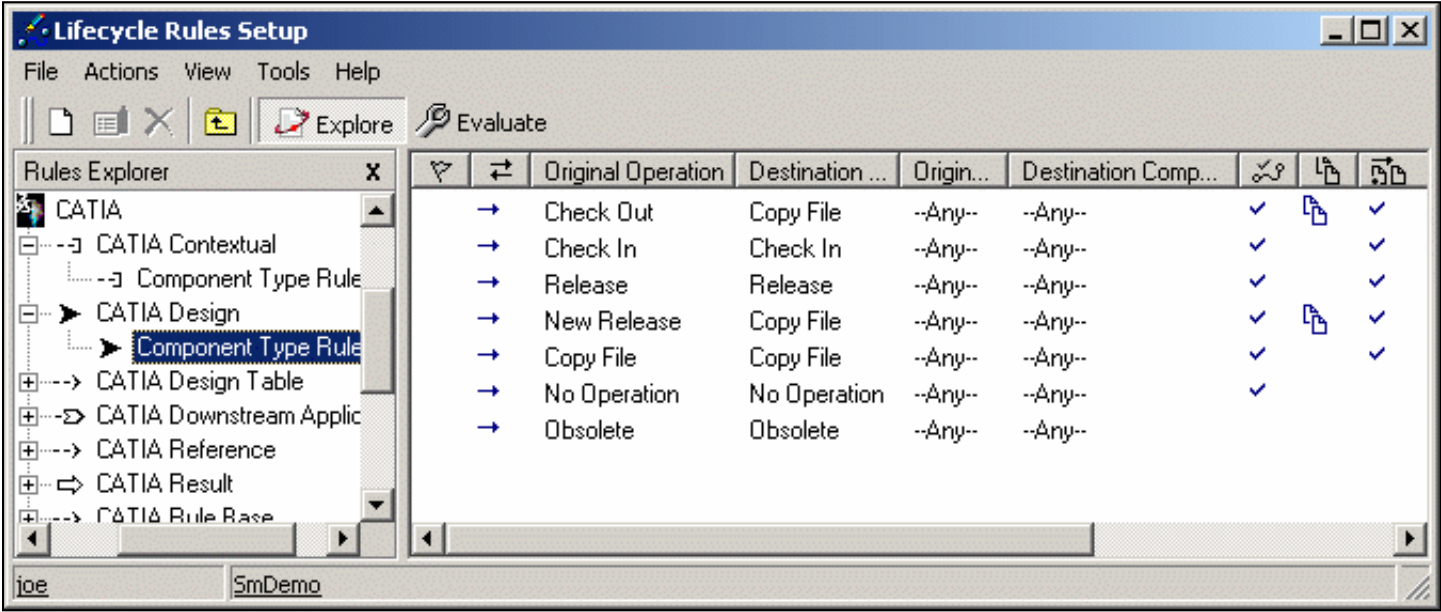
## CATIA Link Default Settings

The following settings are defined in the supplied SmDemo database.

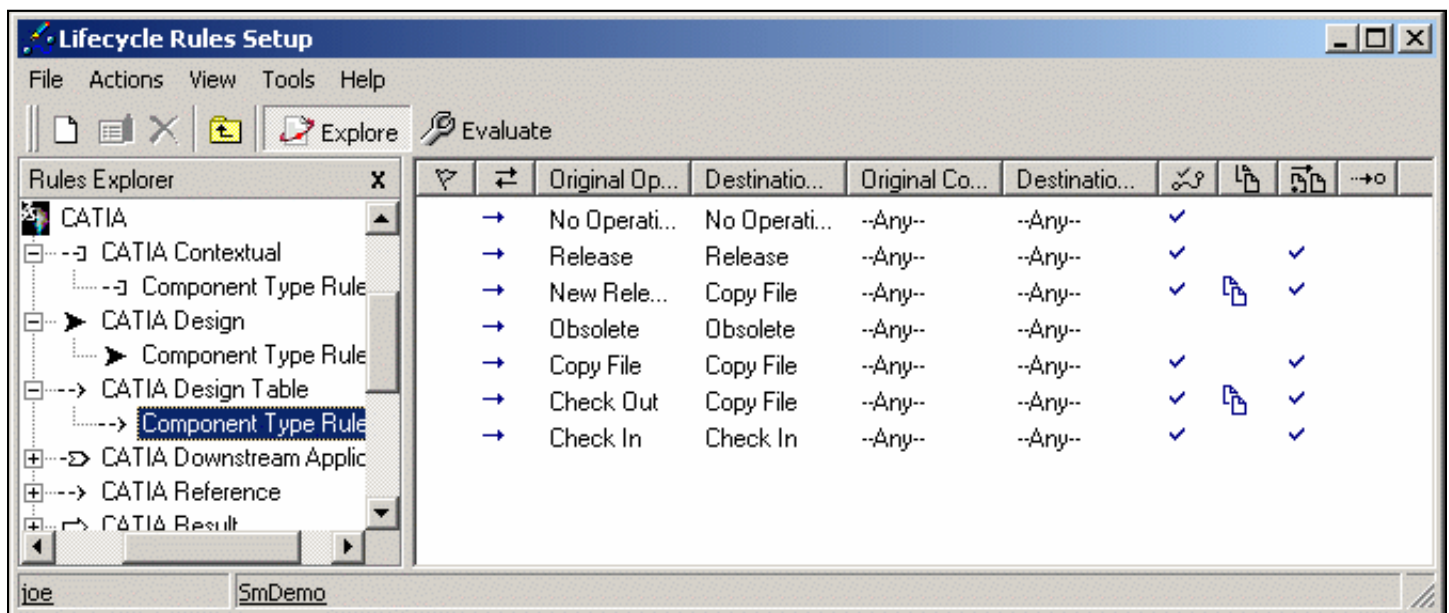
CATIA Contextual



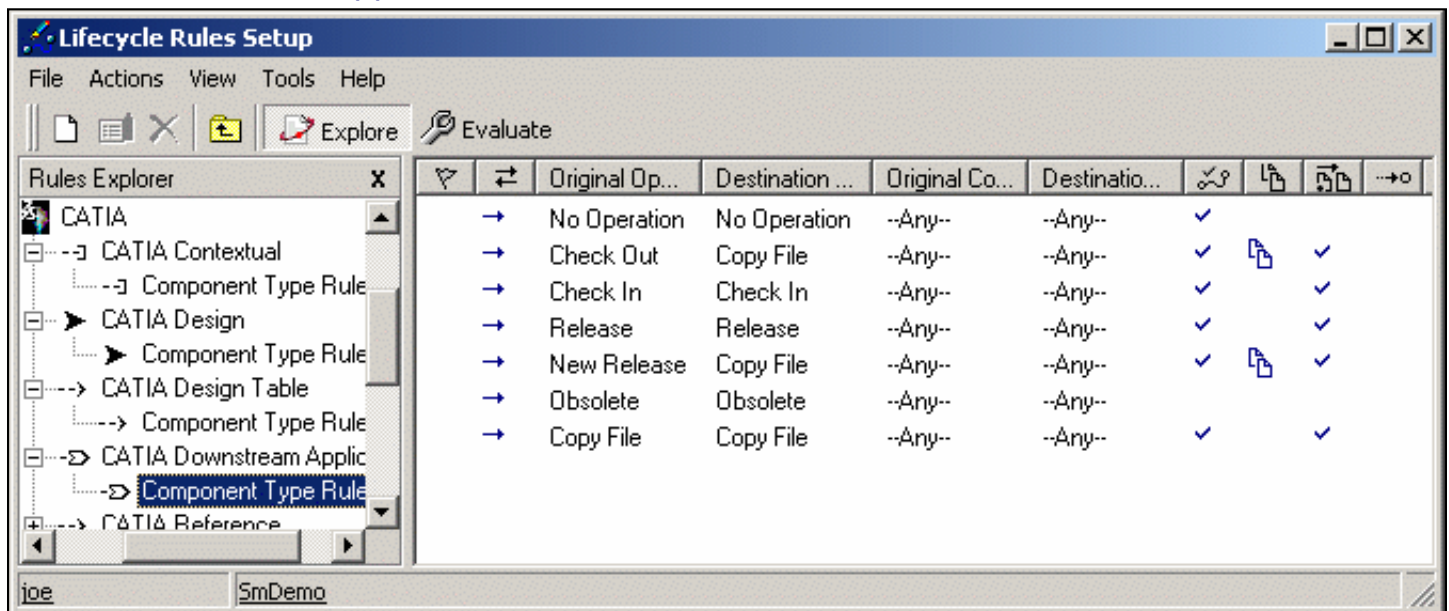
CATIA Design



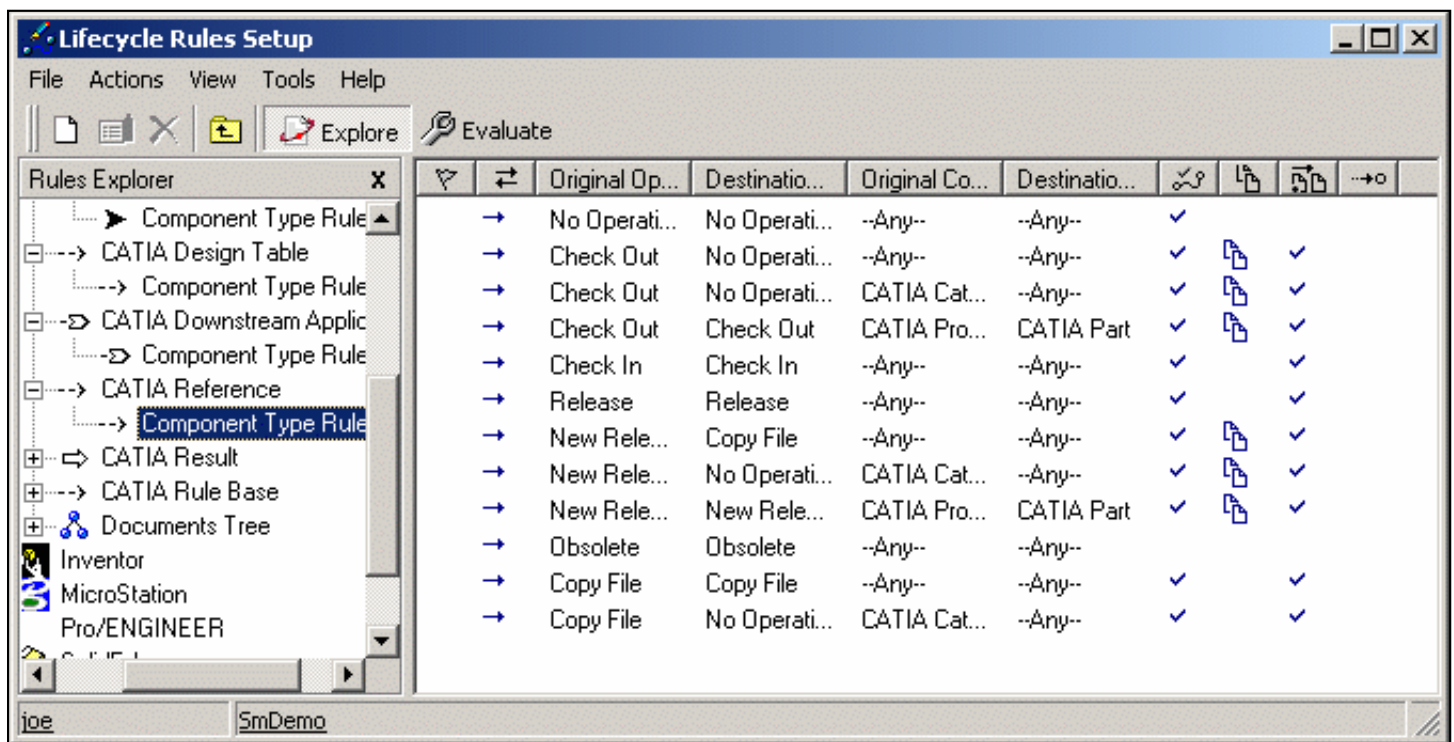
CATIA Design Table



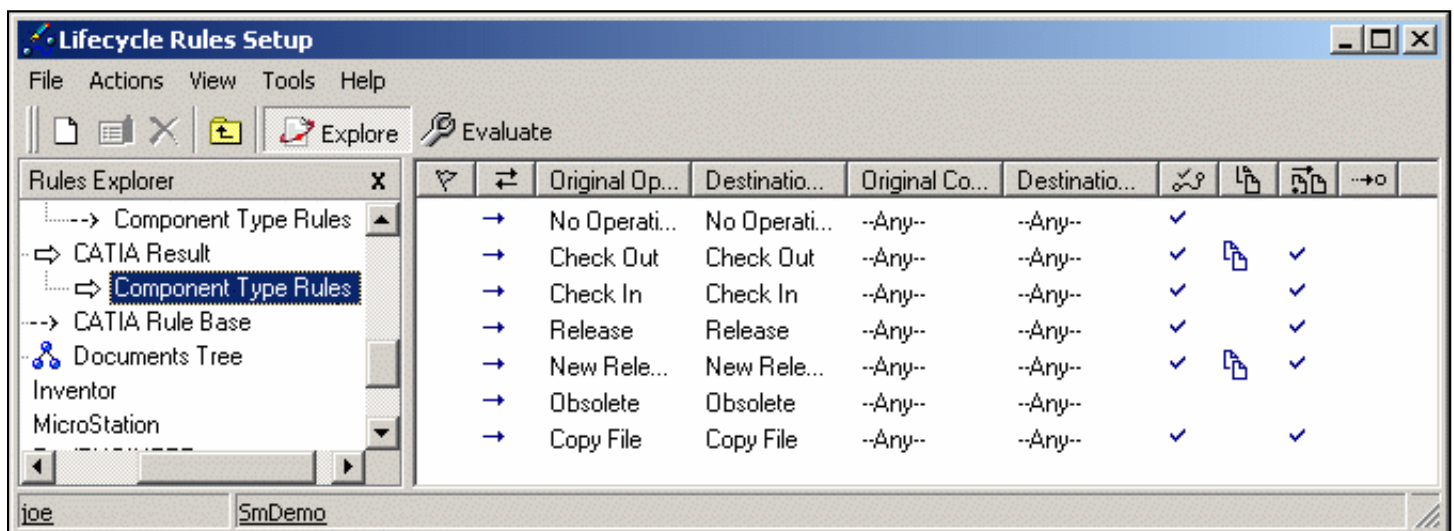
## CATIA Downstream Application



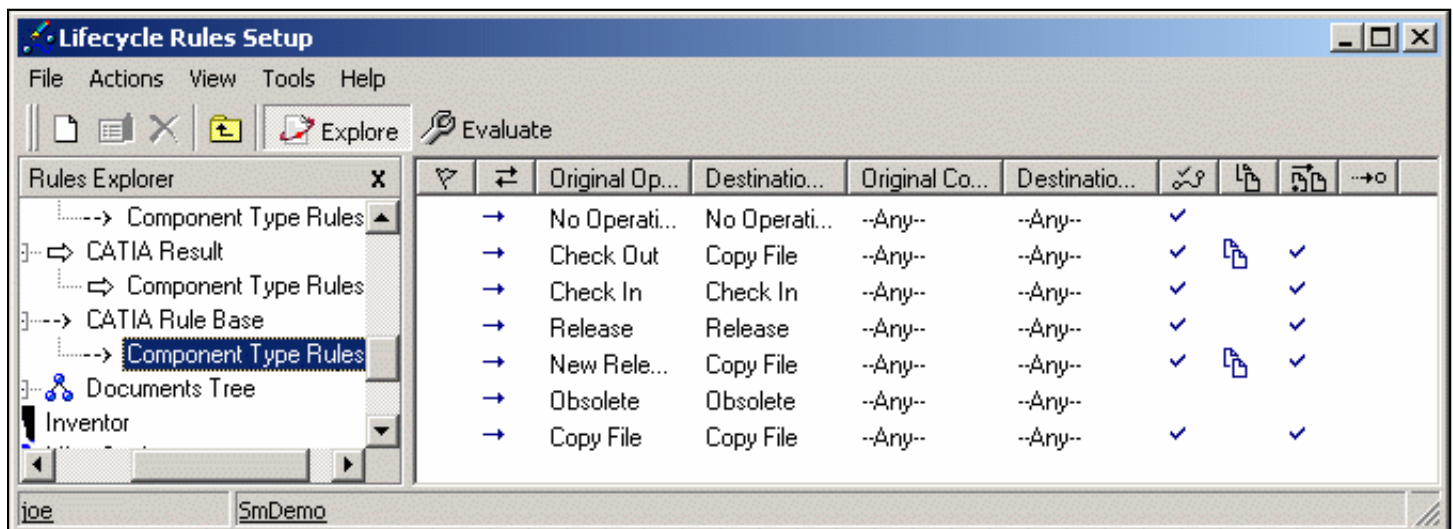
## CATIA Reference



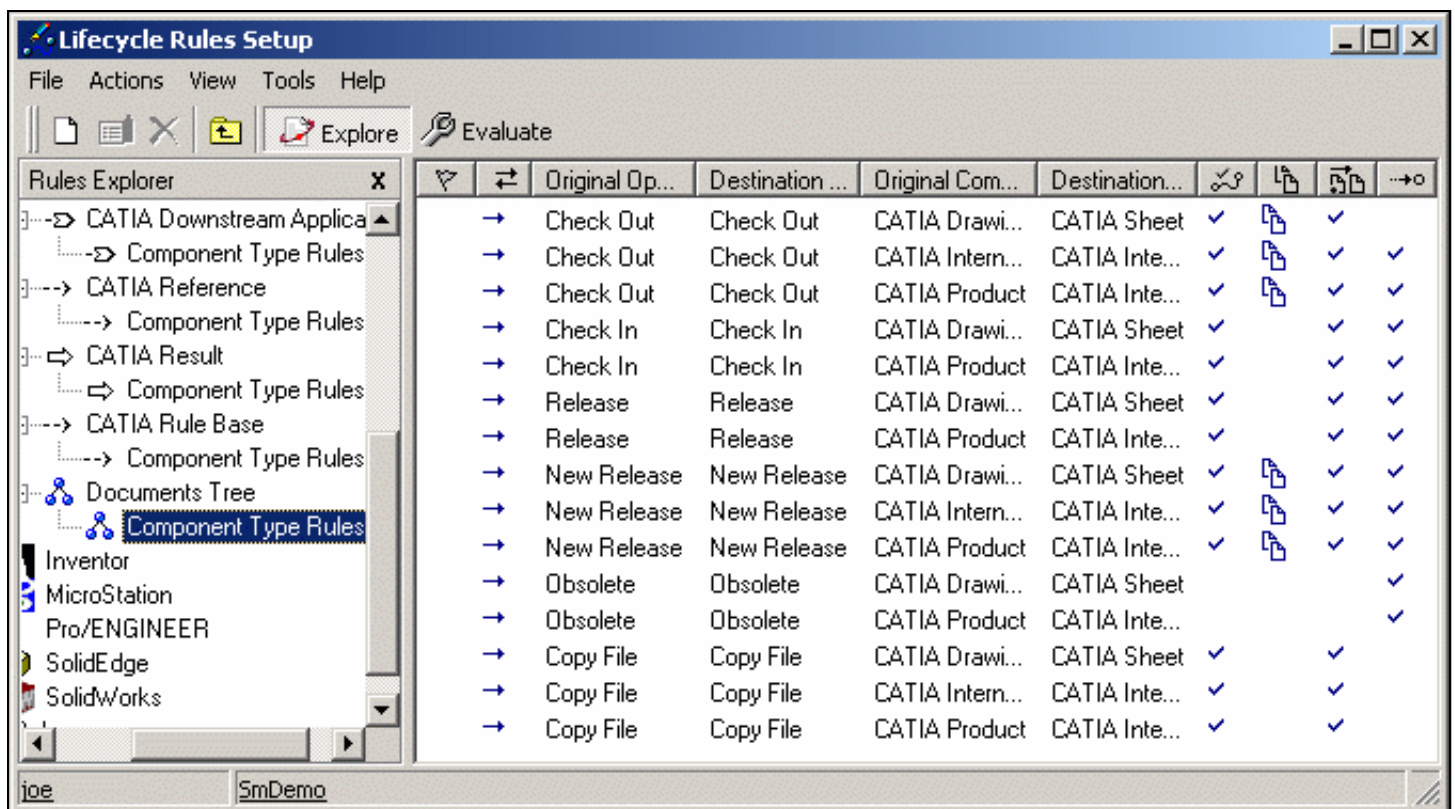
## CATIA Result




## CATIA Rule Base

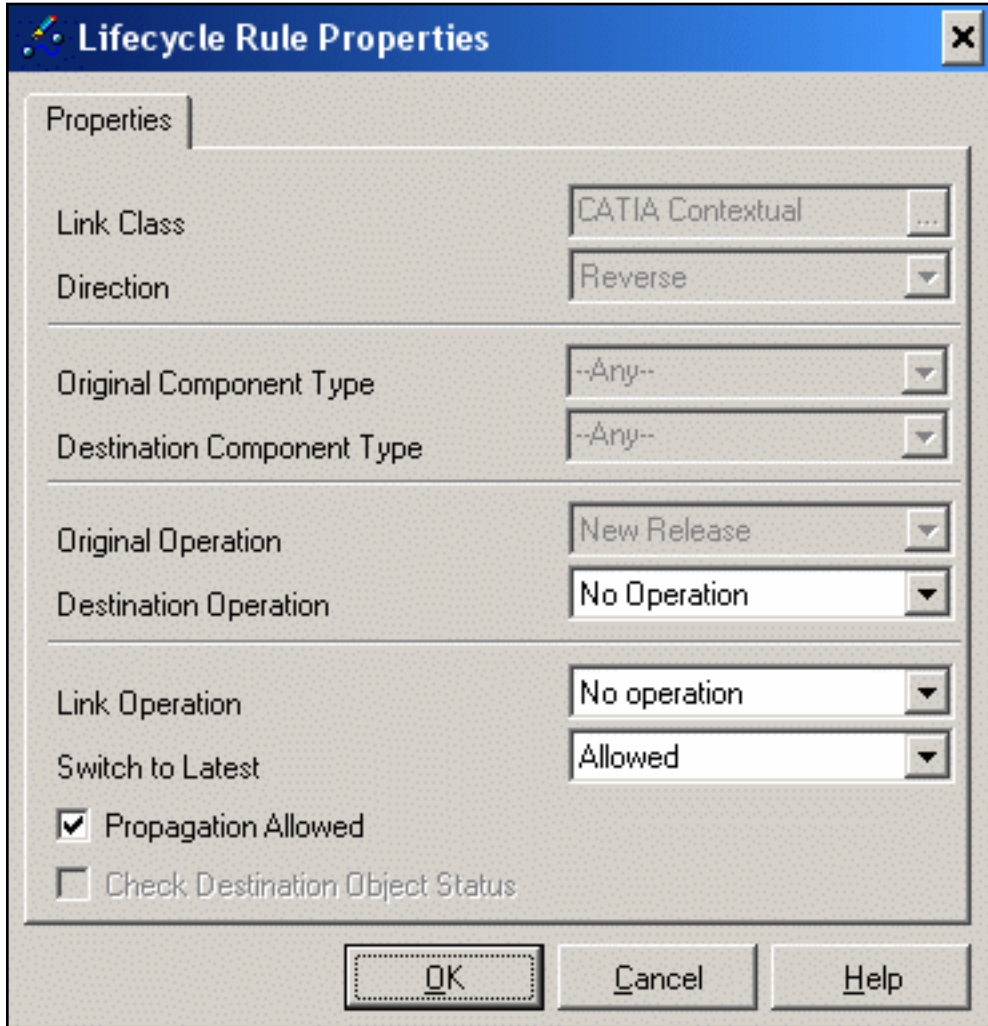
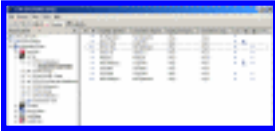


## CATIA Documents Tree



# Managing Contextual Links

 Here are the settings we recommend for working with reverse contextual links:




The 'Lifecycle Rule Properties' dialog box is shown with the following settings:

Property	Value
Link Class	CATIA Contextual
Direction	Reverse
Original Component Type	--Any--
Destination Component Type	--Any--
Original Operation	New Release
Destination Operation	No Operation
Link Operation	No operation
Switch to Latest	Allowed
Propagation Allowed	<input checked="" type="checkbox"/>
Check Destination Object Status	<input type="checkbox"/>

Buttons: OK, Cancel, Help



# Last Public Revision

 When working as a team on a large design, users need to access each other's documents. When a team member checks out a document, other team members who need to reference it will use an earlier revision by copying it to their private work areas.

When a checked out model is checked in (or released), other team members who might be referencing an older revision of it, can get the latest copy.

You can update your local copies to the latest available when performing a lifecycle operation as well as directly from within the CATIA session.

The definition of the last public revision influences the user experience relating to loading the most recently available revisions of sub-components.

## Recommendation


Set up the **Last Public Revision Mode** to [Check In and Release](#) .

Since a document can have numerous revision numbers, **SmarTeam - Editor** enables you to define which revision is considered the last revision. For example, you can define that only objects that are Released (using the **Release** option) are assigned the last revision status.

The last revision status affects the following SmarTeam - Editor operations:

- Tree Properties
- Searches
- Branching

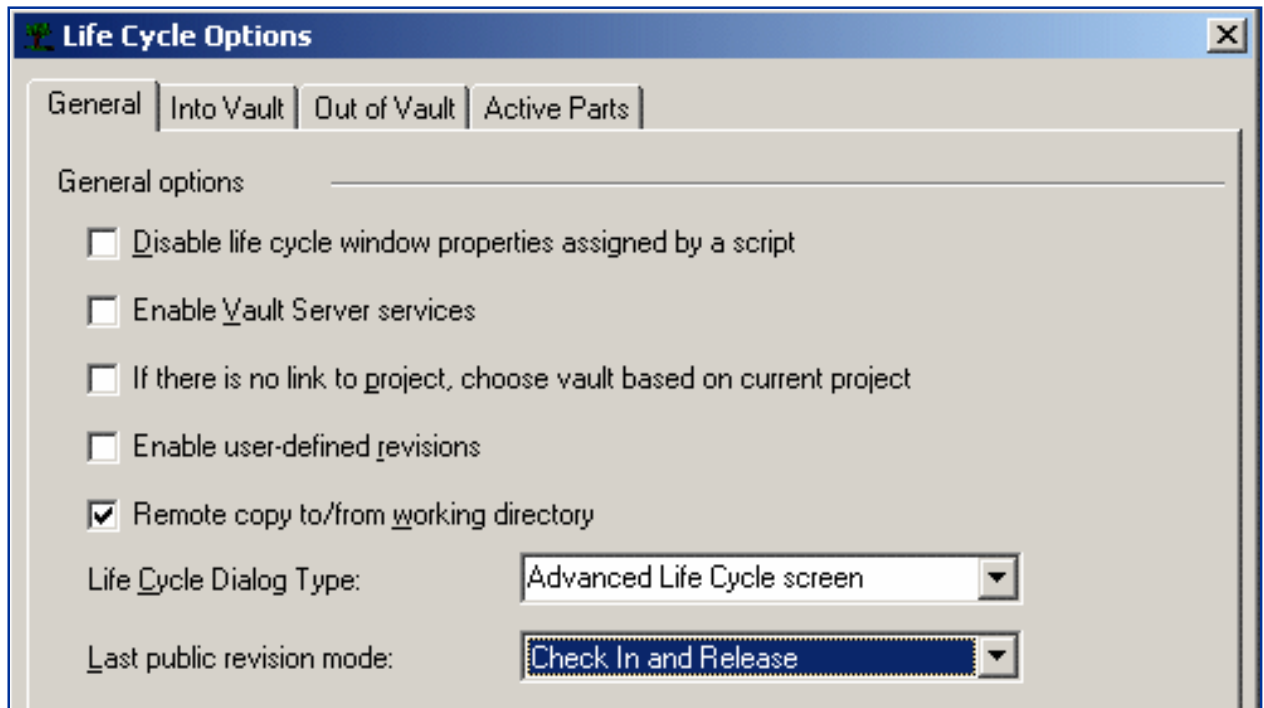
## Setting up Last Public Revision Mode

-  1. In SmarTeam, select **Administrator Options > Lifecycle Options**.
- For **Last public revision mode**, three options are available from the dropdown list in order to determine the last revision status of all objects in SmarTeam - Editor:
- [Release Only](#)
  - [Check In](#)
  - [Check In and Release](#)

### Check In and Release

2. Select **Check In and Release**.
- The last revision status is assigned to both a Released and Checked In revision of

an object. If you select this option, then two revisions of a document (Released revision and Checked In revision) may be displayed in a tree browser or Search window.



## Release Only

The last revision status is assigned to the last Released revision of an object. In cases where a Released revision does not yet exist, the last revision status is assigned to the last Checked In revision of that object. If an object was Released (as b.0 for example) and then checked back into the vault (as b.1), the last revision status is assigned to the former revision (b.0), even though it is an earlier revision. Choose this option if you wish to display only those objects that were Released.

## Check In

The last revision status is assigned to the last Checked In revision of an object. If an object was Released (as b.0 for example) and then checked back into the vault (as b.1), the last revision status is assigned to the latter revision (b.1), even though it has not been Released.

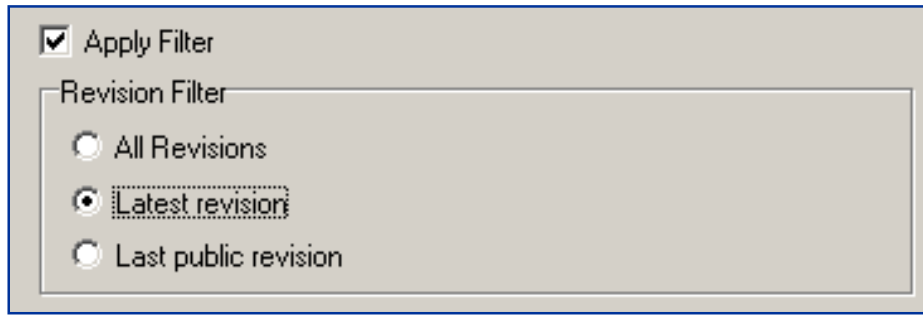
Note: The last revision of an object is significant in the following operations:

## Tree Properties

You can define that the tree will only display the last revision of a document, by proceeding as follows:

1. In SmarTeam, select **Tree > Tree Properties...**

2. From the **Tree Properties** dialog box that appears, click the **Tree Filter** tab.
3. Click in the **Latest Revision** option button located in the **Revision Filter** section.




## Lifecycle Options General window

If in the **Lifecycle Options General** window you selected:

- **Release Only**, the tree will display the last Released revision of an object.
- **Check In**, the tree will display the last Checked In revision of an object.
- **Check In and Release**, the tree will display both the last Released revision and the last Checked In revision of an object.



# Override Previous Revision

 This option enables you to determine whether a file that is copied to a user's desktop will override a duplicate copy of the same file. For example, a user can be modifying an object called oil232.drw. The user can then copy an Assembly to the desktop that contains the same object as one of its sub-Assemblies.

This option determines if the second copy of this object will overwrite the existing copy of the object. Click on the dropdown arrow to select one of the following

SmarTeam - Editor will overwrite a read-only file. If the file on the user's desktop is a read-only file, then the new copy of the file will overwrite the existing copy. For example, a user could have copied an Assembly to the desktop that contains the object oil.drw as a sub-Assembly. This sub-Assembly file is a read-only file, as it was copied to the desktop as part of an Assembly only. If the user then decides to check out and modify this specific object (oil.drw), the new file will overwrite the existing one.



# Link to Parents of Previous Revision



Upon check in, a user may request to link the checked in, new revision to the parents of its previous revision. This allows the updating of referencing documents with the new updates.

## Recommendation

Use Replace in parents of previous revision only in the WIP (Work In Process) stage of the design. This functionality also makes all referencing documents dirty. Therefore, as soon as these are opened in CATIA, a rebuild is needed (check out).

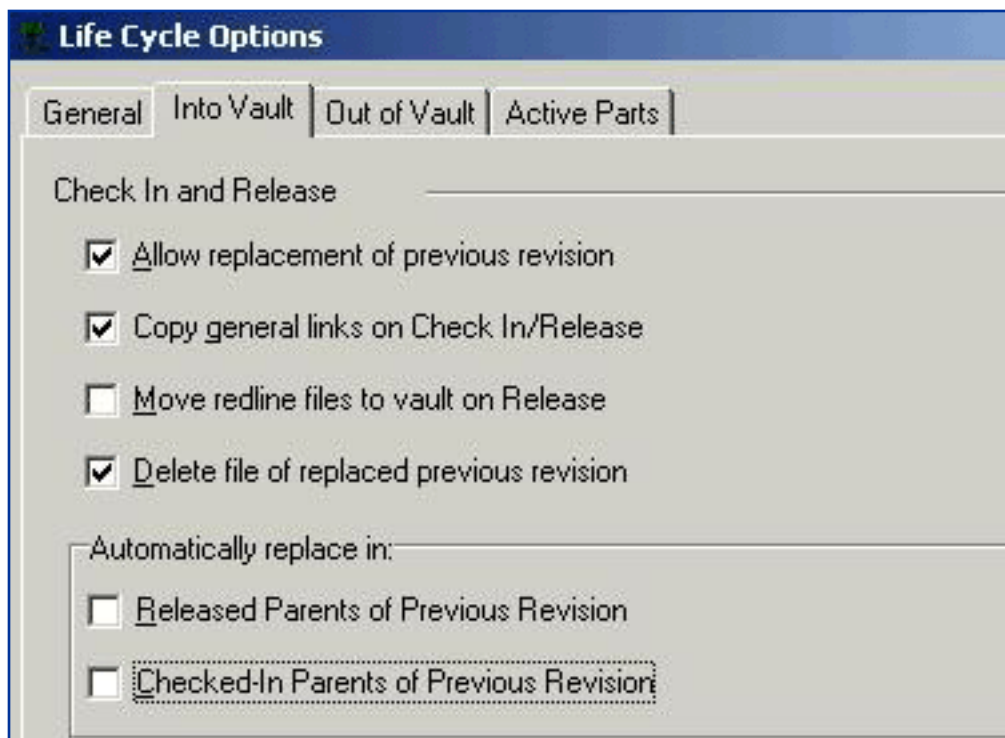
This advanced feature is not for daily use, it involves same issues as Overwrite Previous Revision regarding the updating of private working areas of other team members.

## Setting up Replace in parents of previous revision

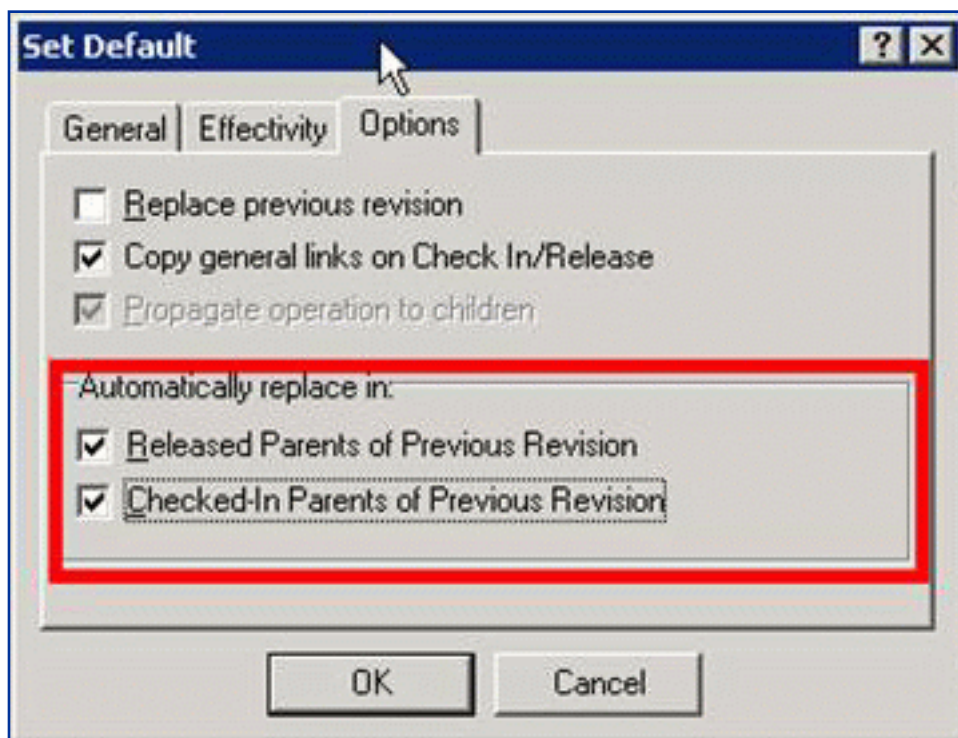


Starting from SmarTeam V5R13, the administrator can set automatic replace in parents of previous revision as shown below.

1. From Administrator Options, select **Lifecycle Options**.
2. Click the Into Vault tab.
3. Check **Released Parents of Previous Revision** and **Checked-in Parents of Previous Revision**.




- If **Allow replacement of previous revision** is checked, the user can ask the system to automatically replace the previous revision by the new one in the current operation using the Set Default dialog.
- If **Automatically replace in** is checked, the system will automatically set this option in the Set Default dialog. These options are not recommended.





# Rules for Overwriting Local Files

 One of the cornerstones of CATIA usage is the re-usability of components. Additional advantages are the ability to create revisions throughout time, the need to work concurrently with other members of the team, in-context modeling and other features. This sometimes results in a situation where a file that is brought from the vault already exists in the destination directory.

When the file that is brought from the vault differs from the one in the directory, a question arises whether or not SmarTeam should overwrite the local copy. The answer depends on the context of the operation you are performing and the reason why the local copy already exists.

If you are trying to print or view a drawing exactly as it was released, you need the exact copies of referenced documents as at the time of the release. When one of the referenced documents already exists in the destination directory, you should answer yes to overwrite the local copy. However, the local copy that is in your working area is probably there due to a different referencing product or model, and copying an older version of the document may affect that parent. In certain cases, the local copy may be checked out and worked on, and overwriting it may cause the loss of valuable work.

Due to the fact that products may include hundreds to thousands of sub-components, it is not recommended to setup SmarTeam to prompt you each time it runs into the overwriting problem. To solve the 'multiple questions' scenario, there is an error report that summarizes all the issues encountered.

There are several SmarTeam preferences that affect the system behavior regarding the overwriting of local files.

## Recommended Setup

The recommended setup is based on the following two assumptions:

1. A local file that is checked out or has been modified compared to its vaulted version will NOT be overwritten (this ensures that you do not lose any valuable work you have done). To overwrite the local file, the user has to respond to the prompt.
2. Newer versions take priority over older versions, i.e., by default an older version does not overwrite a newer, local version. In these cases, a user can request to be notified in order to take an action that overrides this 'default' rule.

The recommended setup is:

- If a mandatory rule in your CATIA design is to work with the latest revision of all sub-components at all times, setup the out of vault preferences to automatically switch to latest revisions.  
In this setup, SmarTeam overwrites local copies only if the retrieved copy is newer. No prompt or warning is presented.
- If users want to be able to define whether a local copy should be overwritten only if it is newer, set up the preferences to compare revisions.
- If users want to be able to define whether a local copy should be overwritten or not (whether it is newer or older), setup the system to ask on replacing local copy on copy and on check out new release.



# 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.

**Note:** When generating the file name, SmarTeam adds an additional directory and therefore adds characters to the file name. Thus, in the case of a long name, the full file name may exceed the limitation of 256 characters for a full path.



# Data Model Considerations: Defining Classes



The system data model is a key factor in the ease of use, performance and usability of the system. The fact that the SmarTeam system has a flexible data model is a huge advantage, but this advantage should be used correctly, taking into consideration how the CATIA Integration is affected.

This page deals with the definition of classes. For information about attributes, see [Defining Attributes](#).

## Number of Classes

A CATIA class represents data of CATIA documents in SmarTeam database. The following table describes the CATIA component types treated by SmarTeam - CATIA Integration:

CATIA Component Type	Description	Remarks
CATIA Product	CATIA Product	
CATIA Internal Component	CATIA Internal Component	Use only if Exposed Mode set to TRUE
CATIA Part	CATIA Part	
CATIA Drawing	CATIA Drawing	
CATIA Sheet	CATIA Sheet	Use only if Exposed Mode set to TRUE
CATIA Catalog	CATIA Catalog	
CATIA Material	CATIA Material	
CATIA Process	CATIA Process	
NC	NC	
CATIA V4 Model	CATIA V4 Model	
CATIA Representation	CATIA Representation	
CATIA Analysis Results	CATIA Analysis Results	
CATIA Analysis	CATIA Analysis	
CATIA Analysis Computations	CATIA Analysis Computations	
CATIA Analysis Input	CATIA Analysis Input	
CATIA System	CATIA System	
CATIA Feature Dictionary	CATIA Feature Dictionary	
CATIA Document	CATIA Document	
CATIA Process Library	CATIA Process Library	
Design Table	Design Table	
CATIA CADAM	CATIA CADAM	

We recommend to avoid a large number of classes, as it affects general performance and from the point of view of usability and understanding by the user, the interface may become too complicated. In the case of a specific data model for a customer, use only the classes that the user needs and remove unused classes in SmDemo (in case this database is used as a starting point for the implementation).



## Recommendation

- Define only SmarTeam classes that correspond to component types that you use in CATIA.
- Alternatively, if certain component types may be used in the future; remove user's permissions from these classes to ensure that they do not confuse the end users.

## Defining Composition Rules

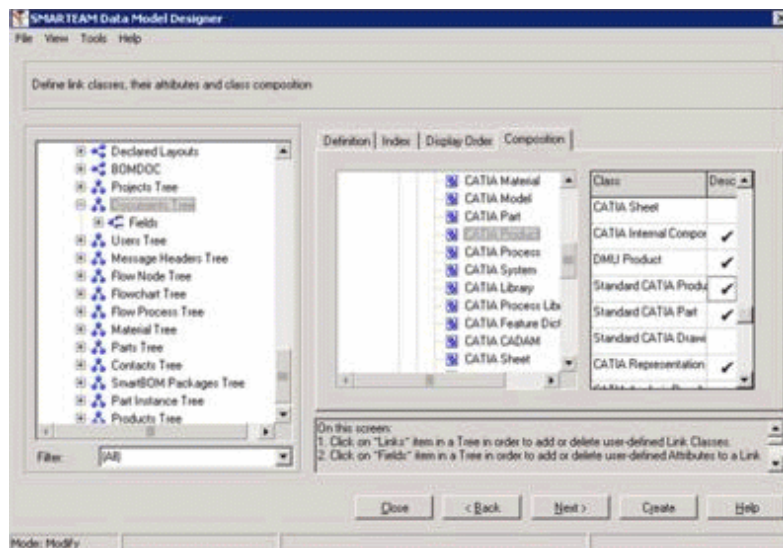
The **SmarTeam Data Model Designer** allows you to define composition rules. It is recommended to allow only the relevant compositions as this has direct effect on performance. The following table displays the recommended composition for CATIA classes. For component types that do not appear in the table, do not set any composition.

Class		CATIA Product	Standard CATIA Product	CATIA Internal Component	CATIA Part	Standard CATIA Part	CATIA Model
Class	Component Type						
CATIA Product	CATIA Product	V	V	V	V	V	V
Standard CATIA Product	CATIA Product	X	V	X	X	V	X
CATIA Internal Component	CATIA Product	V	V	V	V	V	V
CATIA Part	CATIA Part	X	X	X	X	X	X
Standard CATIA Part	CATIA Part	X	X	X	X	X	X
CATIA Model	CATIA Model	X	X	X	X	X	X

### How to define composition

In the Data Model Designer, define the link classes, their attributes and class composition. To do this, proceed as follows:

1. From the **SmarTeam Data Model Designer** window, select **Document Tree**.
2. Click the **Composition** tab.
3. Set the appropriate compositions.



## Defining Classes as Desktop Objects

When defining classes in the **Data Model Designer**, you may allow/not allow associating objects of this type with the project desktop by setting the **Add as Top Level** flag.

Assigning a Class to be top level has performance implications due to the special actions executed by SmarTeam during a lifecycle. When Out of Vault operations (Check Out, New Release) are performed on objects of classes that may be desktop objects, the system checks whether the particular object is a desktop object and re-link the

newly-created version to the root of the projects to which it was previously linked.

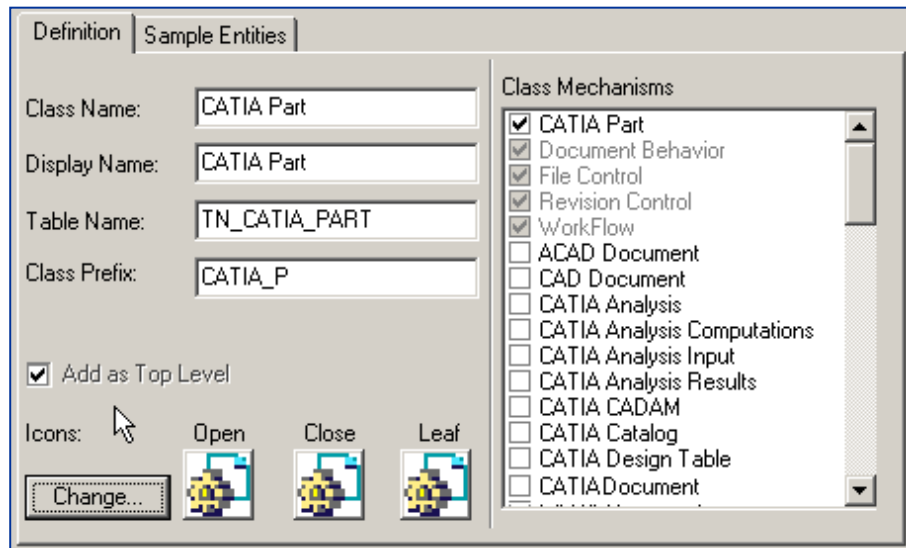


## Recommendation

Decide which classes should be allowed on the desktop and define only these classes as desktop objects. Try to minimize the number of classes allowed to be desktop objects.

### How to allow a class to be a desktop object

1. Check **Add as Top Level** for the classes as illustrated below:



## Standard Parts

Standard parts are widely used by designers. The major advantage of using standard parts in CATIA is the reusability of the parts and the time it saves. It is critical that parts that fall into the category of standard parts do not change often, and will be correctly managed and controlled as they may affect almost all your designs

For more details, see the *Standard Parts Library Methodology Guide*. See also [Managing Catalogs](#).

### STANDARD PARTS REVISION MANAGEMENT

The following table provides a brief summary of the advantages and disadvantages of revision management of standard parts.

Using revision-managed parts	
Advantages	Disadvantages
Secure vault storage	Need to always copy standard part along with assembly to user work area
Multi-site replication along with vault	
Using non-revision-managed parts	
Advantages	Disadvantages
No need to move standard parts to user work area	Standard parts reside in non-secure area, where users need write permission
	Cannot replicate standard parts in the Multi-site, as the parts do not reside in the vault



## Recommendation

Although there are advantages and disadvantages to each approach, the most effective way to handle standard parts is in a revision-managed fashion even if no new revisions are generated for the standard part.

## Assigning a Dedicated Class for Standard Parts

The SmarTeam administrator can assign either standard part class or distinguished standard parts by attributes. The following table summarize of advantages and disadvantages of both methods.

Standard part class	
Advantages	Disadvantages
Lifecycle rules can be set differently for standard parts to support company rules	Cannot transfer regular parts to standard parts
To avoid unauthorized modification, authorization can be applied for standard parts	
Visualization by icon in the SmarTeam tree for standard class part	
Attribute based standard parts	
Advantages	Disadvantages
Any type of part can be transferred to a standard part	Cannot define Lifecycle rules for standard parts
	No authorization for standard parts
	No visualization in the SmarTeam tree



### Recommendation

Currently, it is recommended to use the standard parts class to be able to define lifecycle rules for standard parts.

## File Control Mechanism

In some cases, objects in certain classes are not file-managed, for example, folders in the documents tree. Removing the file control mechanism from a superclass and assigning file control to relevant classes only may have serious performance implications.

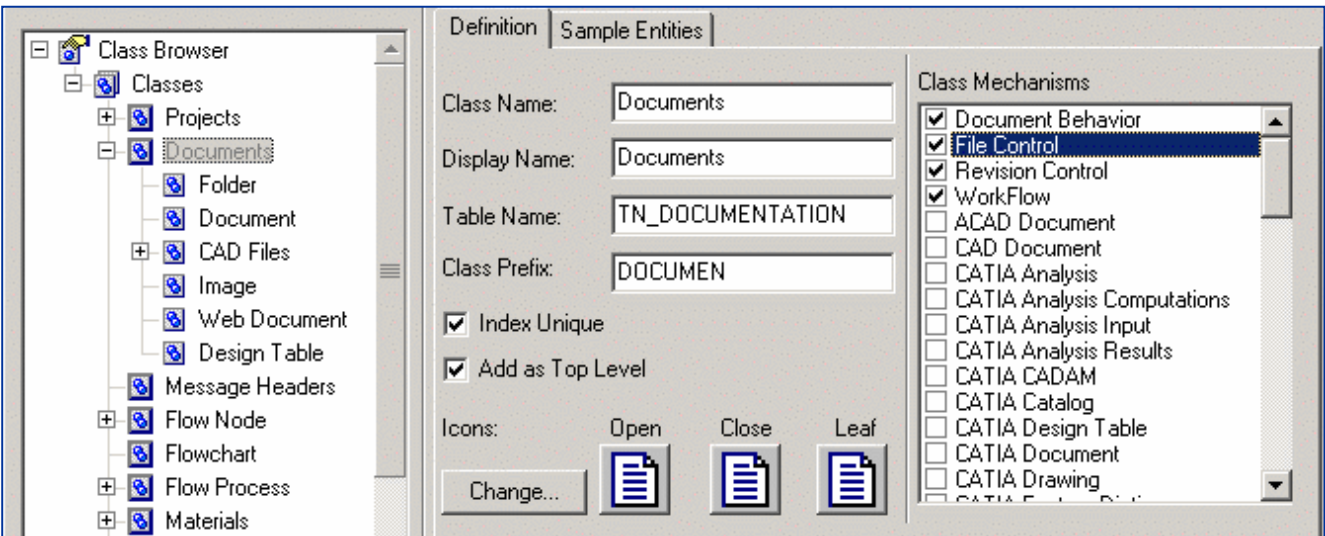


### Recommendation

To ensure good performance for the CATIA integration, it is highly recommended that file management is defined at the super-class level. This should also be done for classes in your Documents super class, that do not manage any files. (These classes will not have any file attributes filled and the attributes will not be shown on the profile card).

### How to define the File control mechanism

1. In the **Data Model Designer**, set the **File Control** class mechanism for the **Documents** super class.



## Working with Sheets and Internal Components

The SmarTeam - CATIA integration allows the creating of distinct objects for internal components in CATIA

Product, and sheets in CATIA Drawing. In some cases, sheets and internal documents do not need to be managed separately.



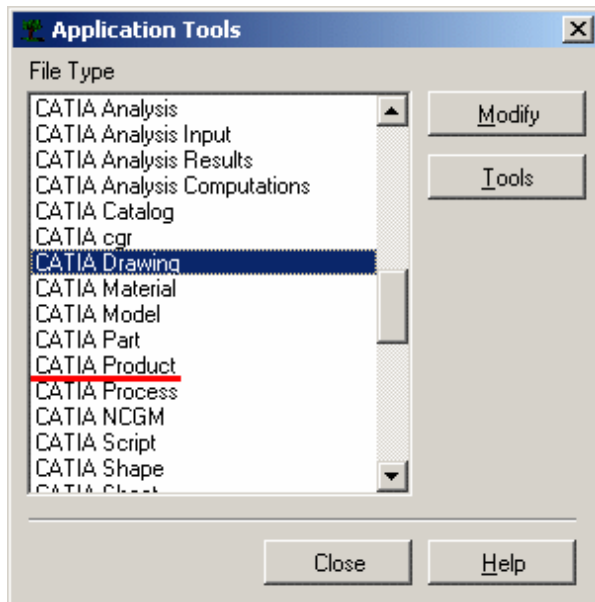
## Recommendation

Unless there is a specific need to manage internal components and/or drawing sheets as distinct objects, it is recommended to disable creating objects.

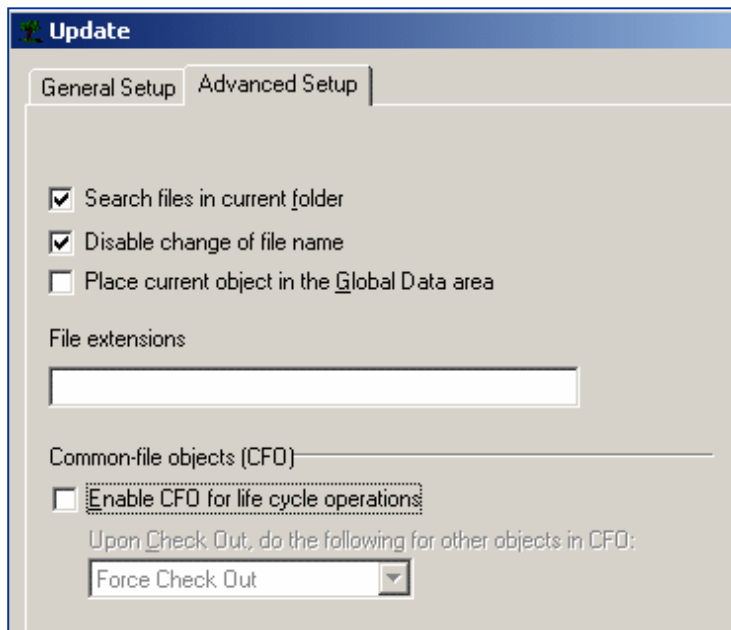
### How to disable creating objects for internal components and drawing sheets

1. Using the **System Configuration Editor**, set **CATIA.ExposeMode** to **FALSE**.
2. Disable the CFO mechanism for CATIA Product and CATIA Drawing file types as follows: in SmarTeam, select **Tools > Applications Setup**.


This displays the **Application Tools** window:



3. For CATIA Product and CATIA Drawing file types, click the **Tools** button.
4. From the dialog box that appears, click **Modify**.
5. From the **Update** dialog box that appears, click the **Advanced Setup** tab and just clear **Enable CFO for life cycle operations** to disable the CFO mechanism for CATIA Product and CATIA Drawing file types.



# Data Model Considerations: Defining Attributes

 This page deals with attribute sizes and provides information on the TDM\_CAD\_DIRTY\_FLAG Attribute. For information about classes, see [Data Model Considerations: Defining Classes](#).

## Attribute Sizes

The size of certain attributes in the system is critical. The methods your company uses for file naming may require fine-tuning of these attribute sizes.

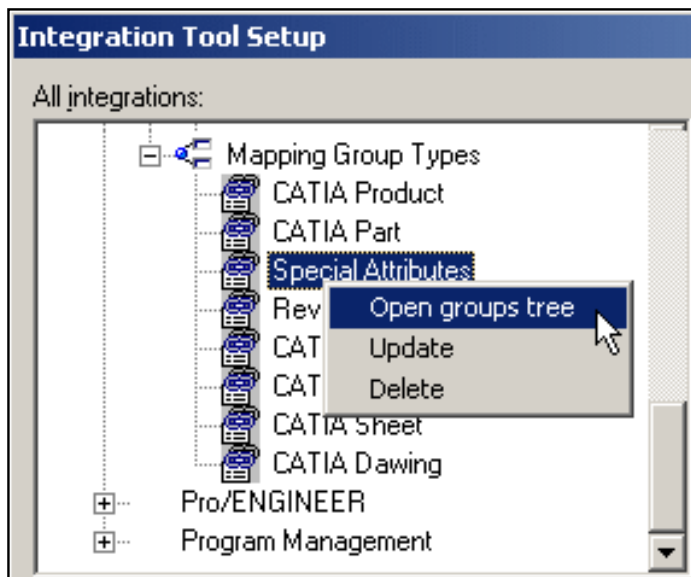
File names and directory path attributes (FILE\_NAME, DIRECTORY, CAD\_REF\_FILE\_NAME, CAD\_REF\_DIRECTORY) must be long enough to store data about your files. If the field size is too small, the system may fail to function properly.

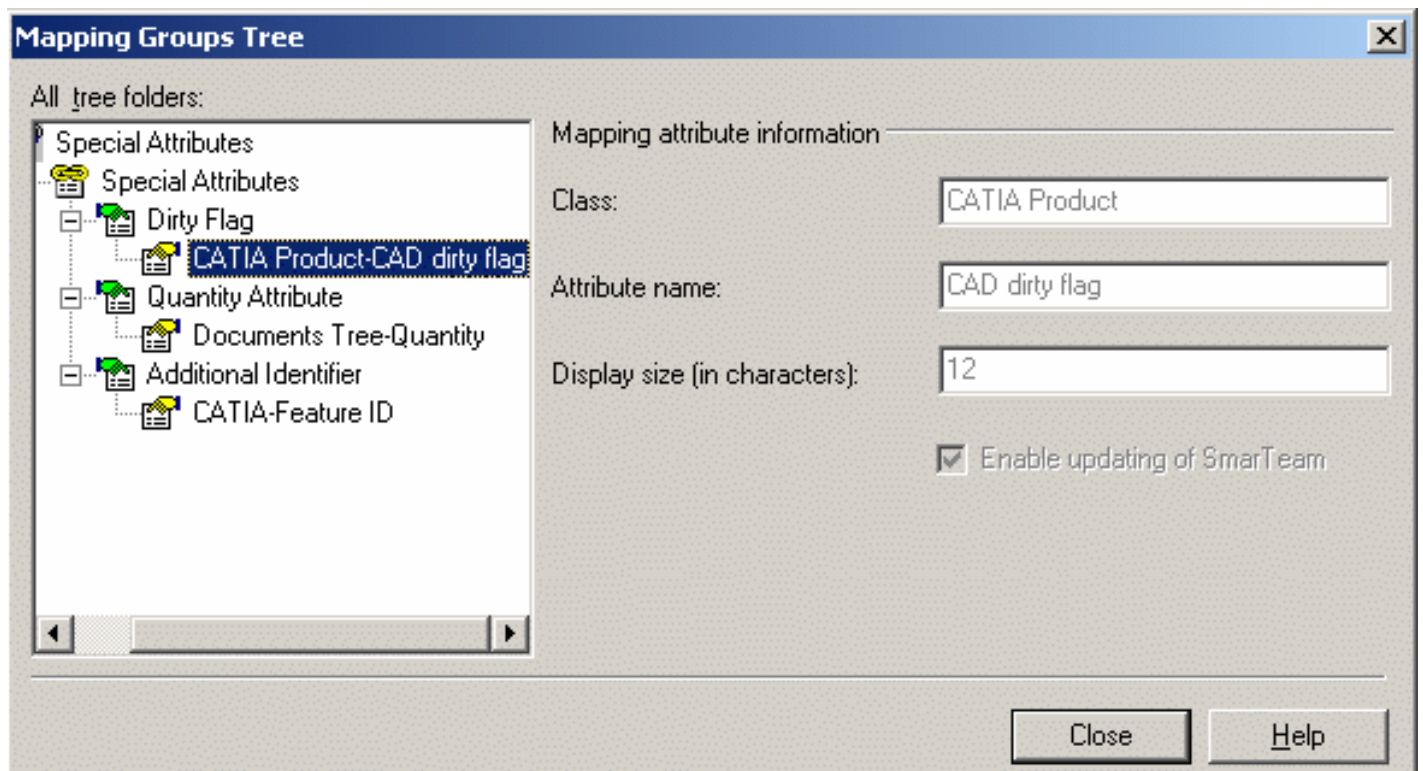
**Note:** The CAD\_REF\_FILE\_NAME is a system attribute managed by the CATIA integration, and must not be modified by the user.

However a long field size influences the performance in a negative way, so try to find an optimum solution. Examine the naming conventions and directory structure that the users will be using carefully before deciding the field size for file-managed attributes.

## TDM\_CAD\_DIRTY\_FLAG Attribute

The dirty flag attribute is assigned automatically to all classes that are defined as file-managed. Existing customers should check that the TDM\_CAD\_DIRTY\_FLAG exists in their system, and that it is used in the [Integration Tools Setup](#) as shown below.





For large assemblies, dirty flag mapping increases the Smarteam save time in CATIA, in cases in which some components were not modified.



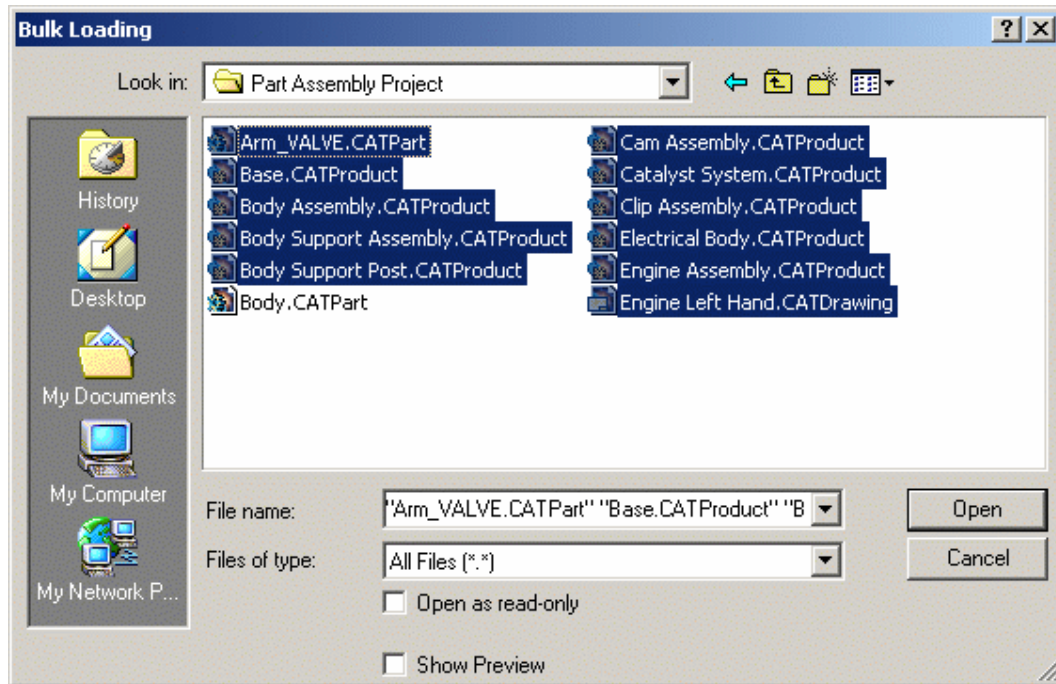
# Importing CATIA Data Inside SmarTeam

This task shows you how to use the **Bulk Loading** command which allows you to save a large number of files in the database. The advantages of this are as follows:

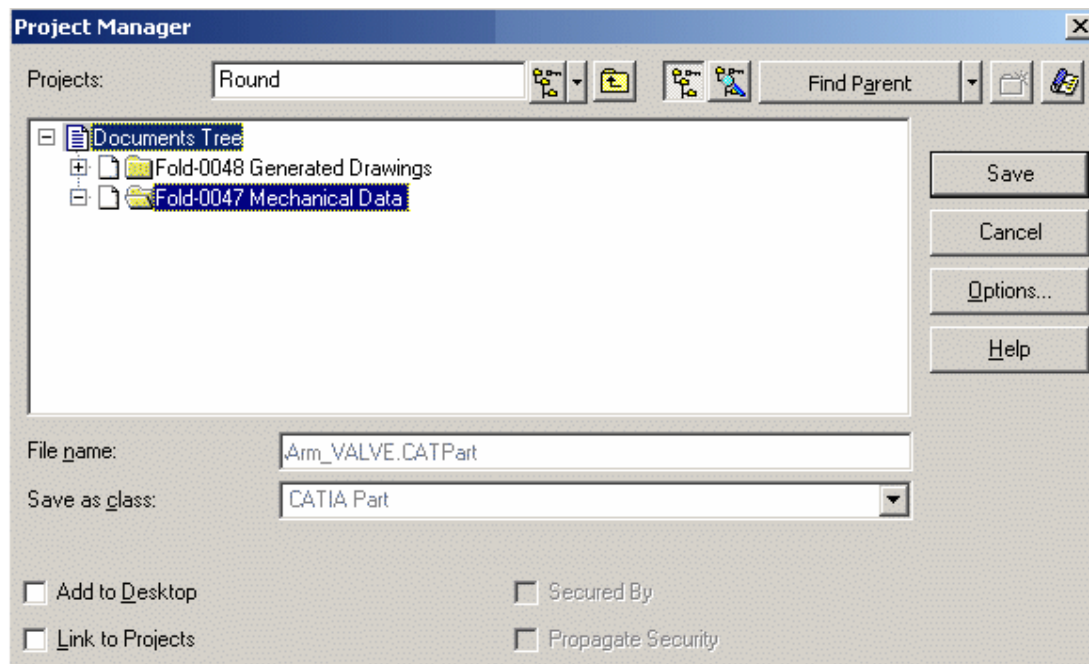
- The save operation is quick.
- Links between documents are kept.
- The attribute mapping mechanism is used during the save operation.
- Only a limited number of user interactions are required regardless of the number of files.

You are already connected to SmarTeam.

1. In your CATIA session, Select **SmarTeam > Bulk Loading...**
2. In the **Bulk Loading** dialog box that appears, multi-select the files you want to save.



3. Click on **Open**.
4. In the **Project Manager** dialog box that appears, select the project with which you wish to associate the selected files:



5. Click **Save**.

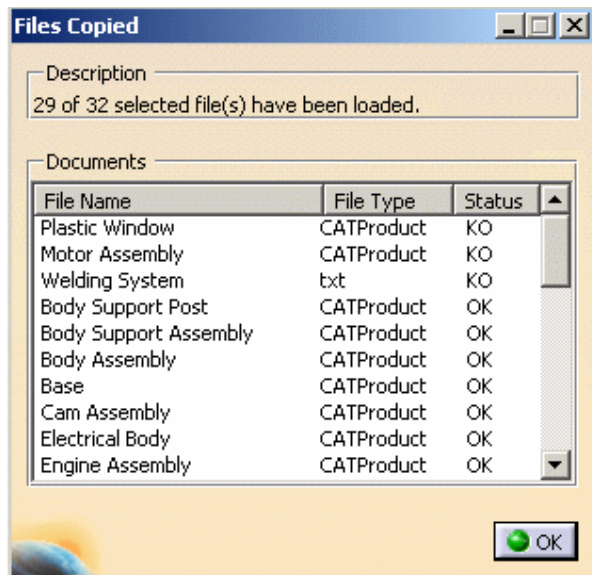
The operation you have just launched is in fact a batch mode save. A progress bar appears showing the number of files still to be processed and the estimated time required for completion. If no problem is encountered during the save operation, the following message is displayed:



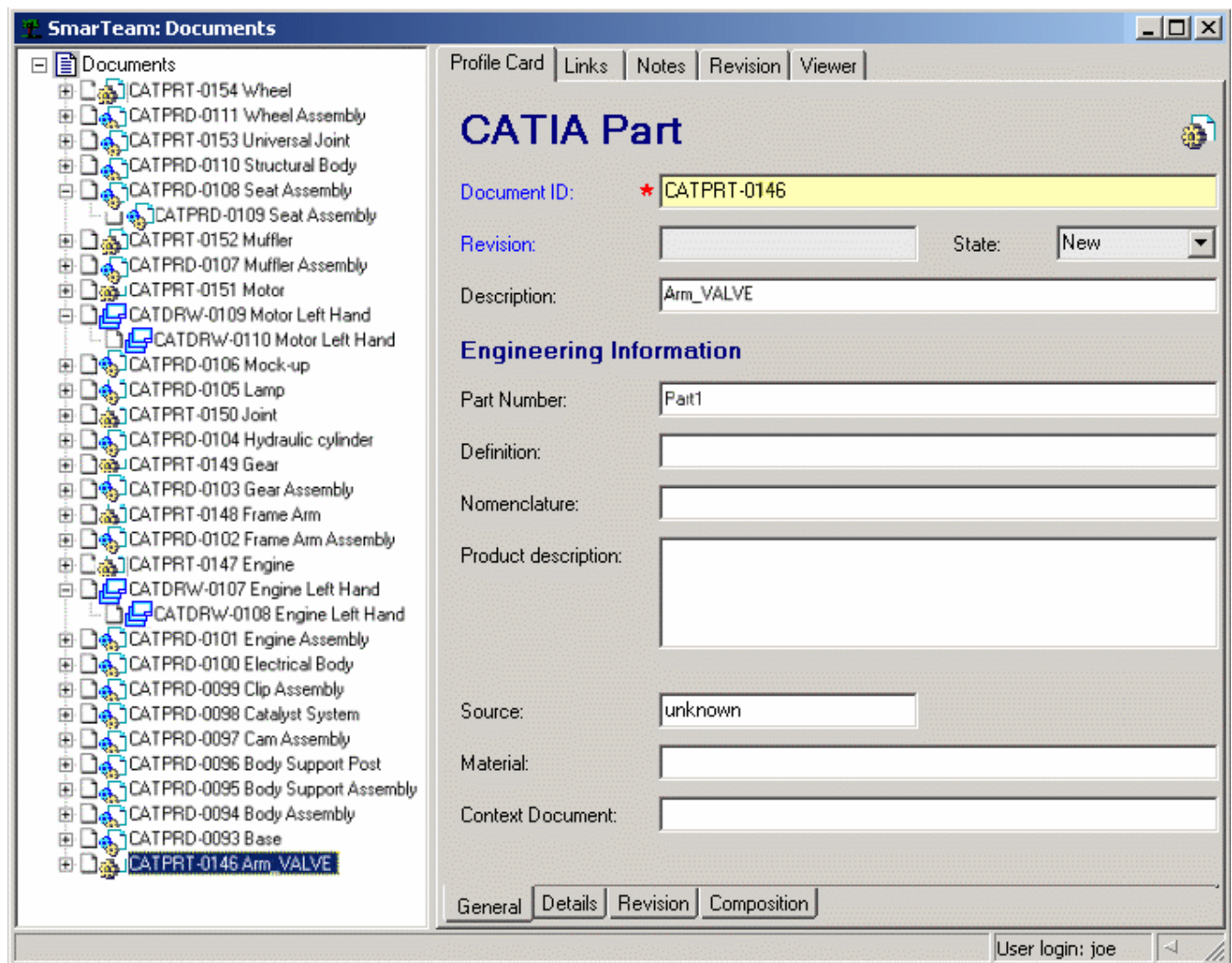
Should any problems arise during the save operation such as:

- o a document that cannot be opened in CATIA
- o no file-type integration can be defined

a dialog box appears, showing the status:



6. Once the operation is completed a dialog box appears showing you what documents have been saved and what links exist between the different documents:



Once your documents have been correctly saved you can:

- subsequently retrieve them without any difficulty as they are located in a single project or folder
- apply a [lifecycle](#) operation to all of them.

## Customizing the Bulk Loading Command

You can customize the Bulk Loading command as explained in [Customizing the Bulk Loading Command](#).

## Saving CATIA documents in SmarTeam Database

Although it is possible to save different document types in the SmarTeam database using SmarTeam Save Editor or Web Editor capabilities, you must use CATIA SmarTeam > Save to save the different CATIA documents in the database.



# Updating the Database to Use 3D Live Functionalities



## Concept

Using the **SmarTeam 3D Live Open In Catia** functionality, a 3D Live user can load specific parts/products from an assembly in a CATIA session. In order to work, the instances information of the assembly must be stored in the SmarTeam database. After the information is stored, CATIA can provide SmarTeam with this information when saving an assembly. However, the instances information of assemblies that were already saved in SmarTeam with previous releases is not stored. This functionality provides a tool to save the instances information of several CATIA products selected in SmarTeam by the user. The tool may be automated to run on the entire database.

For each CATIA Product selected by the user in SmarTeam, the links between the corresponding **CATDocument** file and its children are updated with the instances information (instance name + positioning matrix + instance activity). The pre-existing information already stored in SmarTeam before the operation on this item remains unchanged.

The process eventually goes on through the item's children if this operation is specified when the operation is launched.

Before the implementation of this functionality, the user had to manually check out the assembly from SmarTeam, open it in CATIA, make a modification and save it again to store the instances information. This functionality allows:

- Automation of this process
- Process of several assemblies in one loop



## Procedure

In order to implement this functionality the following steps must be performed:

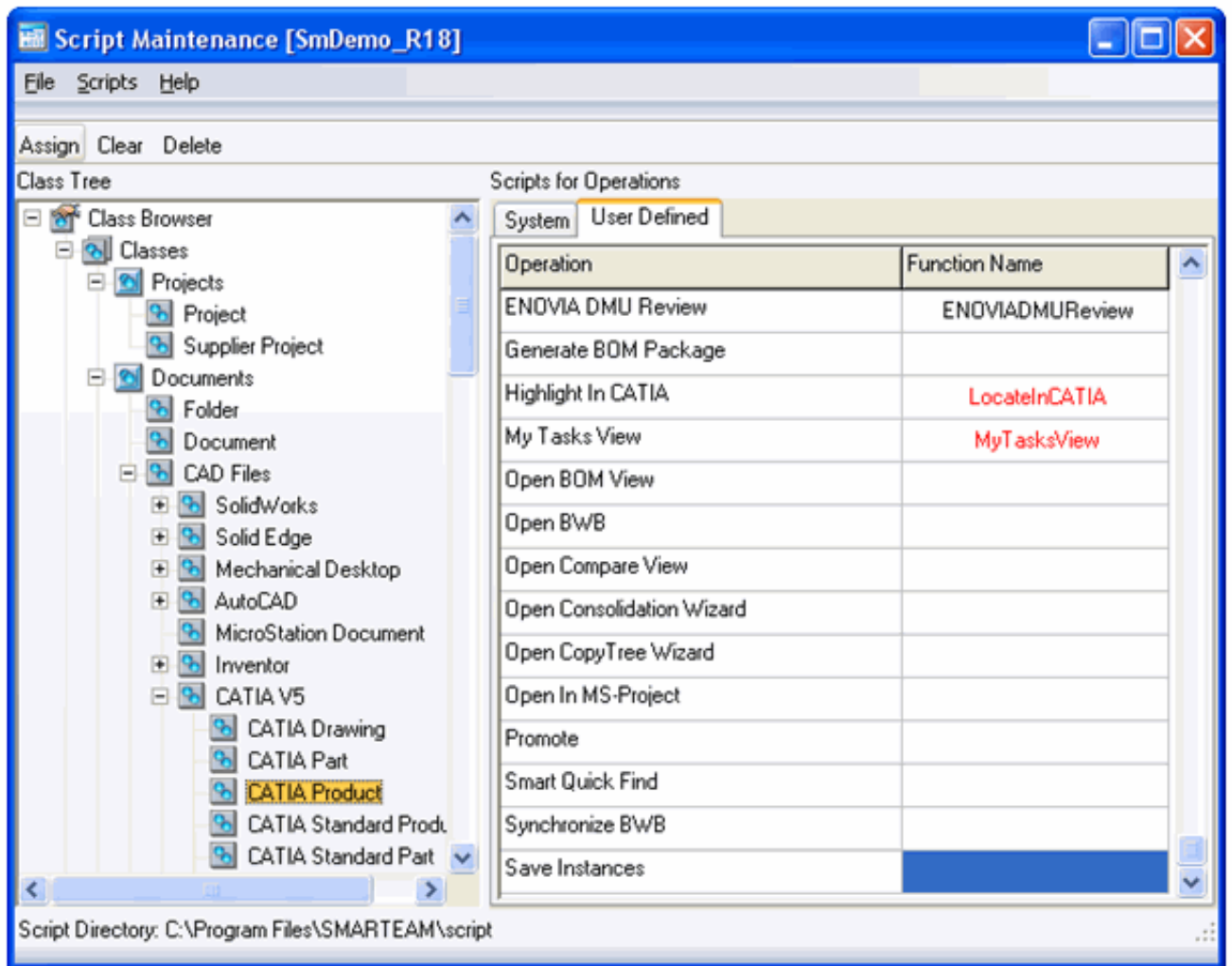
- [Add the function to the contextual menu](#)
- [Set the environment](#)
- [Run the script](#)



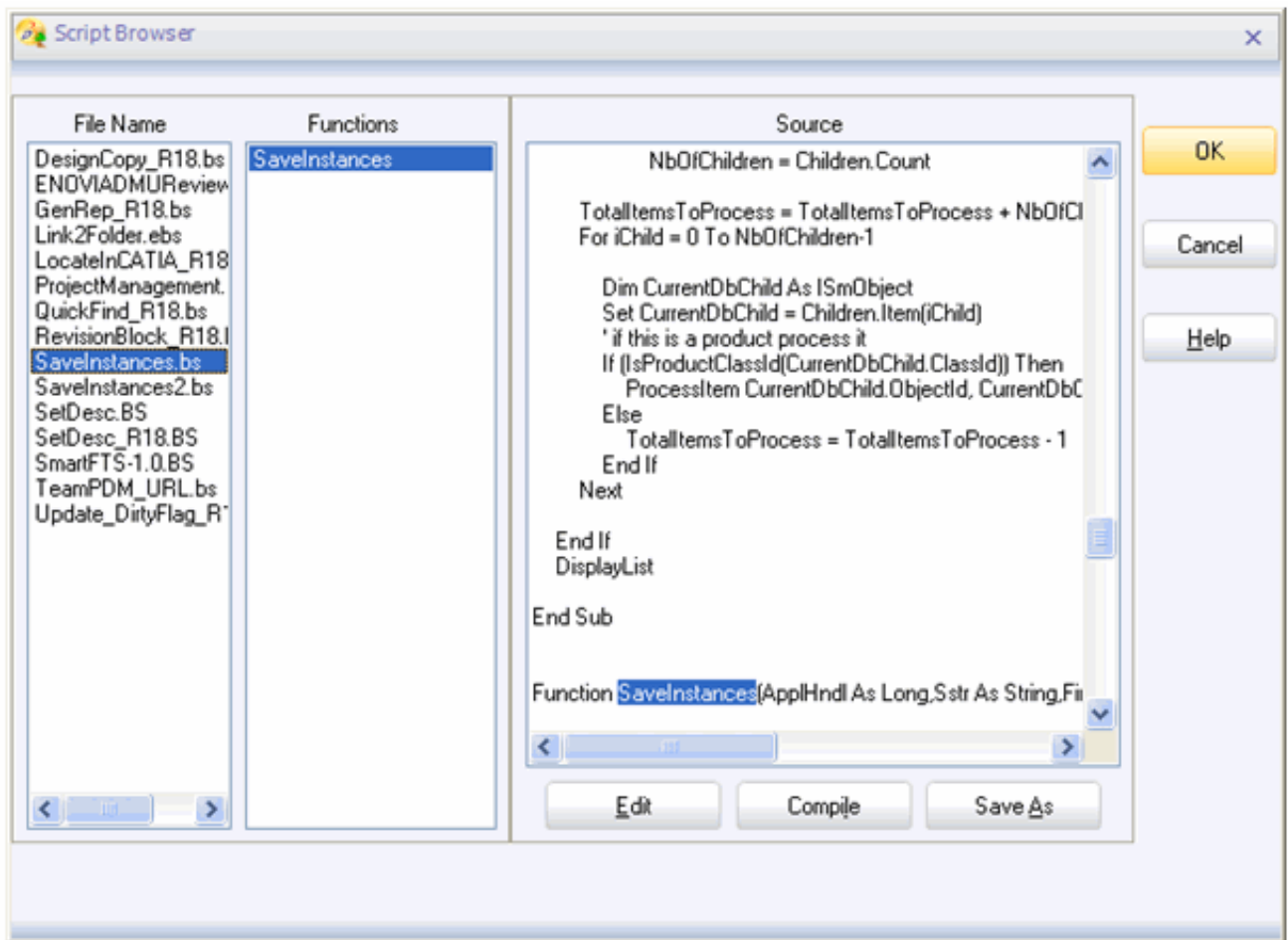
## Add the function to the contextual menu

To run the batch, add the script to the SmarTeam User Defined Tools Menu as described below. This operation has to be done only the first time the script is run:

1. In the SmarTeam Script Maintenance tool, select **CATIA Product** from the **Documents** class. In the right pane, open the **User Defined** tab.
2. In the Operation column add a key: **Save Instances**.



3. Double-click in the Function Name cell of the **Save Instances** operation. The Script Browser window opens.
4. In the **File Name** column, select **SaveInstances.bs**.
5. In the **Functions** column, select **SaveInstances**.



6. After clicking OK, the **SaveInstance** script is hooked to the database.



## Set the environment

To allow the storing of the **instance information** (name + position matrix) in SmarTeam:

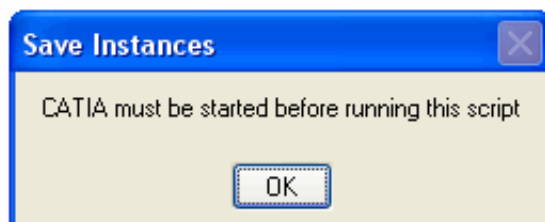
1. Launch the SmarTeam System Configuration Editor (see the SmarTeam - Foundation Administration Guide for details).
2. Search for the CATIA application.
3. Set the key **CATIA.SaveInstanceData** to **TRUE**.
4. Click the **Save Changes** button to confirm the changes.



## Run the script

### Prerequisites

- CATIA must be running. If CATIA is not started, the following error message appear and the script will exit:



- The CATIA version must be V5R19 SP2 or above. If this is not the case, the following error message appear and the script will exit:

## Save Instances

CATIA V5R18SP6 detected => this script is not supported before V5R19SP2

OK

- If CATIA is not connected to SmarTeam, the batch will handle this (using the currently running SmarTeam session).

To run the script:

- Select the checked in CATIA Products whose instances information you want to save.
- Right-click to open the contextual menu, and then select **User Defined Tools > Save Instances**.

CATIA Product: CATIA Product

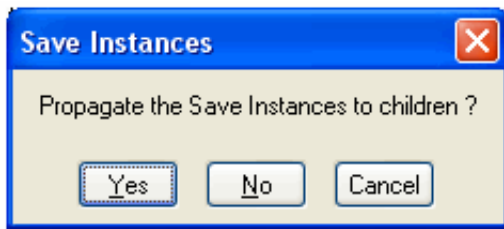
Data Chart

	Class	ID	State	Description	Creation Date	Last modification date and time
1		CATPRD-0257		Product3	12/03/2007 13:58	12/03/2007 13:59
2		CATPRD-0258		Product4	12/03/2007 13:58	12/03/2007 13:59
3		CATPRD-0256		Product2	12/03/2007 13:58	12/03/2007 13:59
4		CATPRD-0255			12/03/2007 13:58	12/03/2007 13:59
5		CATPRD-0254			12/30/2007 15:10	11/30/2007 15:10
6		CATPRD-0253			12/30/2007 15:10	11/30/2007 15:10
7		CATPRD-0252			12/30/2007 14:11	11/30/2007 14:12
8		CATPRD-0251			12/30/2007 14:11	11/30/2007 14:12
9		CATPRD-0250			11/23/2007 10:55	11/23/2007 10:55
10		CATPRD-0248			11/20/2007 17:35	11/20/2007 17:35
11		CATPRD-0249			11/20/2007 17:35	11/20/2007 17:35
12		CATPRD-0247			11/20/2007 17:35	11/20/2007 17:35
13		CATPRD-0246			11/19/2007 16:23	11/19/2007 16:23
14		CATPRD-0245			11/19/2007 12:07	11/19/2007 12:07
15		CATPRD-0244			11/19/2007 12:07	11/19/2007 12:07
16		CATPRD-0243			11/16/2007 16:16	11/16/2007 16:16
17		CATPRD-0242			11/16/2007 16:16	11/16/2007 16:16
18		CATPRD-0240			11/16/2007 15:12	11/16/2007 15:12
19		CATPRD-0241			11/16/2007 15:12	11/16/2007 15:12
20		CATPRD-0239			11/16/2007 15:12	11/16/2007 15:12
21		CATPRD-0238			11/16/2007 14:42	11/16/2007 14:42
22		CATPRD-0237			11/16/2007 14:42	11/16/2007 14:42
23		CATPRD-0236			11/16/2007 14:01	11/16/2007 14:01
24		CATPRD-0235			11/16/2007 14:01	11/16/2007 14:01
25		CATPRD-0234		Product2	11/16/2007 12:06	11/16/2007 12:07
26		CATPRD-0233		Product1	11/16/2007 12:06	11/16/2007 12:07
27		CATPRD-0232		Product2	11/16/2007 11:34	11/16/2007 11:35
28		CATPRD-0231		Product1	11/16/2007 11:34	11/16/2007 11:35
29		CATPRD-0230		Product2	11/16/2007 11:10	11/16/2007 11:13

Contextual menu options:

- Add
- Delete
- Update
- Add As Copy
- Design Copy
- Life Cycle
- Associate with Project...
- File Operation
- Initiate Process
- Send To
- User Defined Tools
  - ENOVIA DMU Review
  - Highlight In CATIA
  - Design Copy
  - My Tasks View
  - Save Instances
- BOM from Documents
- Open Views
- Grid Properties

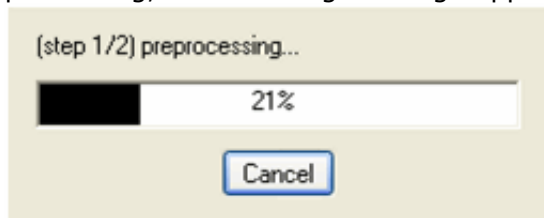
A prompt appears asking whether or not to propagate the operation on to the items children.



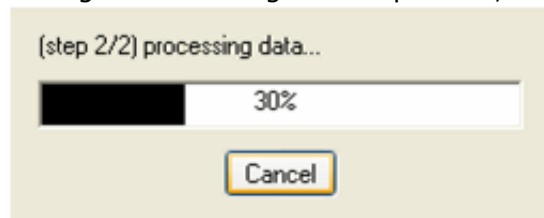
3. Click **Yes** to propagate or **No** to save the instances of the selected CATIA Product only.
4. If CATIA is not connected to SmarTeam, the batch connects CATIA to SmarTeam.

connecting CATIA to SmarTeam...

5. The batch then creates a temporary directory named **BatchTemp** inside the user's temporary directory (the directory is deleted if it already exists). This is where the CATIA products will be copied from the vault.
6. The process browses each selected CATIA Product and adds it to the list of items to process. During processing, the following message appears:

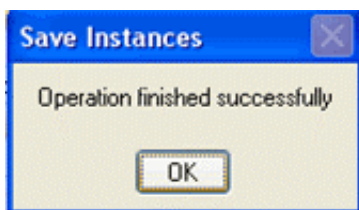


7. During the next stage in the process, the following message appears:



**Note:** Clicking **Cancel** at this point stops the process, but the process cannot be undone for all items that have already been processed.

After the operation has finished, the following message appears:



# Methodology

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

# Concurrent Engineering

SmarTeam - CATIA Integration facilitates working concurrently. This section discusses the different tools provided to help you work in a collaborative environment.

# 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:**
  - **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>LifeCycle>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)
  - the use of the **Global Refresh** command (to reload the latest versions of documents from the SmarTeam database).
- through easy access to the Product Structure tree and, in File Desk, to the *read-only and modified* status.

## Displayed Information

### Non-Latest Revision Status

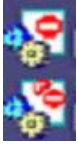
If a document is opened in CATIA with a revision that is not the latest, the SmarTeam information is displayed in the Product Structure and File-Desk trees.



### Modified and Read-Only Status

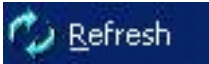
If a check-in or release document is opened in CATIA and is modified, a specific

icon mask is displayed in the CATIA Product Structure and File Desk trees.



## Operations

### Tree Refresh

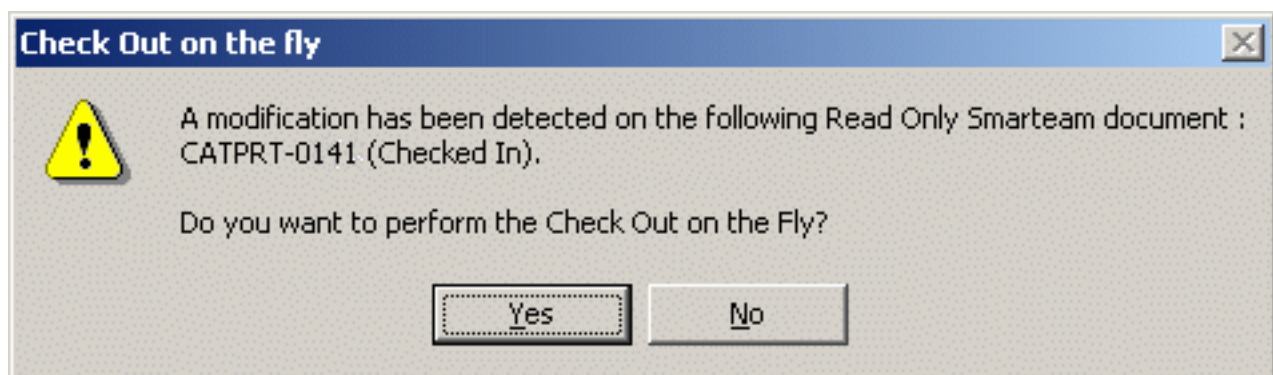
The CATIA **SmartTeam > Tools > Refresh** command  updates the Product Structure and File Desk **icons** with regard to SmarTeam information.

This is particularly useful for concurrent engineering. For instance, if you are using checked-in parts within their assembly, you can use the **Tree Refresh** command 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.

The **Tree Refresh** capability updates the display of icons representing documents in the specification tree, not your session, contrary to the [Global Refresh](#) capability.

### 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.




Check out on the fly windows displays SmarTeam information as set in the **Tree Properties** dialog box. For more information, see [Customizing SmarTeam Document Display Information](#).

## Global Refresh

The [Global Refresh](#) capability enables design engineers to update their CATIA sessions to reflect the latest versions of the documents as saved in the SmarTeam vault.



# Global Refresh

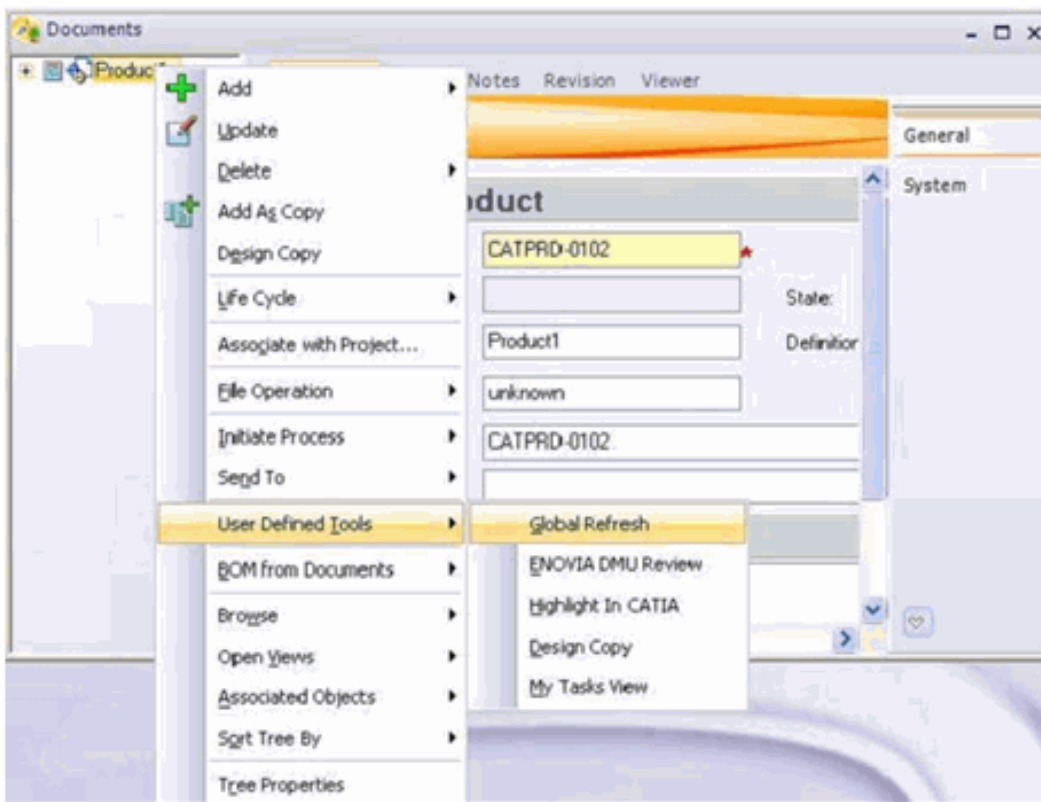
 The SmarTeam Global Refresh command allows you to update a complete CATIA product, applying the latest available revisions to all of the product's children.

Available from the SmarTeam right-click menu, the **Global Refresh** functionality launches the following actions automatically:

- Performs a Check Out operation on the selected product and forces a sub-assembly to be checked out if its children are not the latest
- Opens the selected product in a CATIA session to calculate all new CGRs
- Performs a Check In operation to validate all the changes made in the database.

 To use Global Refresh, do the following:

1. Start CATIA and then connect to SmarTeam.
2. Disable the option "*Display Check-Out/New-Release dialog box on first modification*" in order to avoid Check Out on the fly panels.
3. Select a CATIA Product in SmarTeam. Note that selection of multiple objects is not supported.
4. From the right-click menu, select **User Defined Tools** and then select the **Global Refresh** command. A new version of the CATProduct is created and linked to the latest version of all of the Product's children.



**Notes:**

- Following standard SmarTeam – CATIA integration behavior, the Global Refresh operation creates a new version each time it is performed after a modification is made.
- The Global Refresh operation works only if all children of the selected CATIA product are in “Check In” or “Release” status; i.e., if one or more of the child objects is in Checked out or in New Release state, the command cannot be performed.



# Using Global Refresh

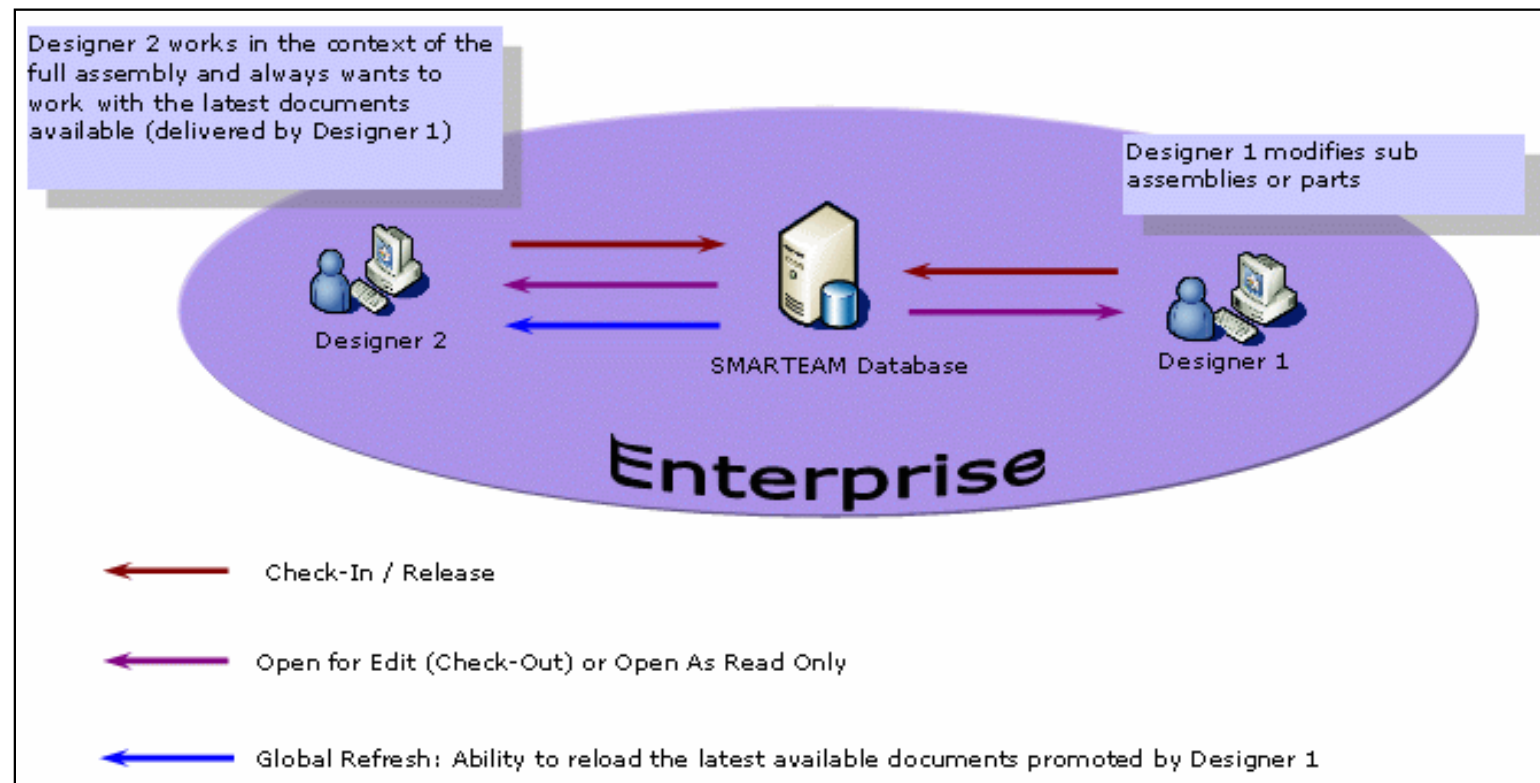


In most enterprises, design engineers work concurrently to design their products. Working concurrently often means that they are responsible for subassemblies which they will all gather later on to constitute their final assembly.

Some designers need to reference to the latest documents available they are not responsible for. In fact, a certain number of these documents will be delivered by other designers in real time.

Because they need to benefit from the most recent versions of these documents, SmarTeam CATIA Integration provides a dedicated tool to help them work in a collaborative environment: the **Global Refresh** capability enables them to update their CATIA sessions to reflect the latest versions of the documents as saved in the SmarTeam vault. Among the benefits provided by this new tool is a significant rework costs reduction.

The following illustration shows when a Global Refresh Operation is to be performed:



In this section, you will find the following information:

- [Basic Concepts](#)
- [How to use Global Refresh](#)
- [After a Global Refresh](#)
- [Note](#)

## Basic Concepts

Global Refresh ensures that all the documents in the CATIA session are having the latest available revisions in SmarTeam database. The tedious process of manually managing the documents in the session is made one click away, which saves efforts and improves efficiency by reducing risks arising due to human interaction.

To take full advantages of Global Refresh, you should keep in mind that the capability enables you to update your session **provided that the latest revisions of documents are placed in the vault.**

For example, if a member of your team is modifying a document he/she checked it out, performing a Global Refresh in your CATIA session containing a reference to that document, proves useless as long as the modified document is not placed back in the vault via a Check In or New Release operation. Once you are informed thru the red dot on the document's icon that your document is no longer the latest one, you just have to wait for the user to check in or release the document. Then, at that point, you will be able to perform a successful Global Refresh.



## How to Use Global Refresh

The scenario below provides the necessary steps to follow to perform a **Global Refresh** operation. It assumes that the assembly displayed in your CATIA session is made of different sub-assemblies, some of which were modified by other members of your team and then checked-in in SmarTeam. You want your session to contain the most recent versions of these documents.

1. Prior to running **Global Refresh**, take a closer look at the icons in your specification tree .  
There are two possible displays:

- o Dedicated icons indicate that some documents are no longer up-to-date.

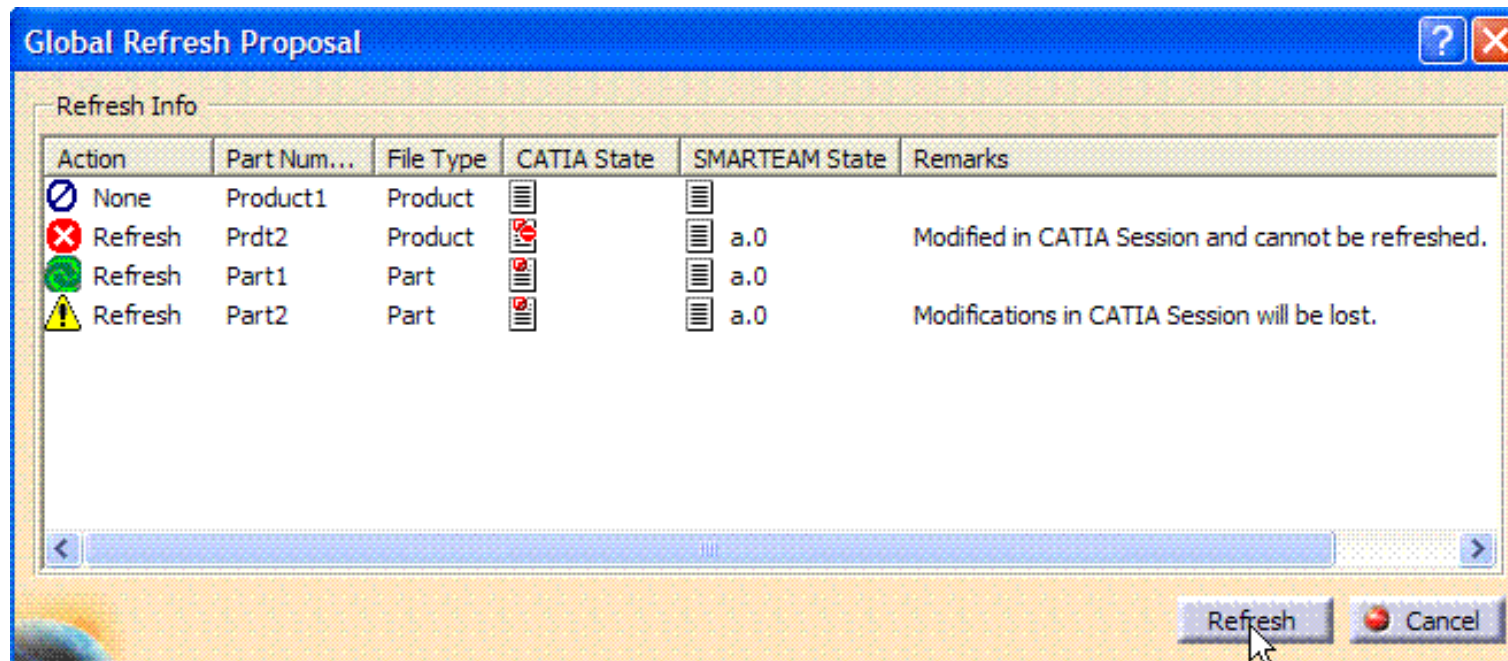


or

- o apparently, your session is up-to-date, but because you need to be sure of that, you prefer to Refresh that display. To do so, just select **SmartTeam>Tools>Refresh** . For more information, see [Tree Refresh](#).





At this point, you know that your session does not use the latest revisions of all of your documents and therefore you decide to refresh it to reflect the changes made by other users.

2. Click the **Global Refresh**  icon or select **SmarTeam > Collaboration> Global Refresh**. The Global Refresh dialog box is displayed.



## Action

The Action column displays the different operations that will be performed for each document. These operations can be:

	<b>None</b>	<p>The document in the current CATIA session is the latest and no modifications are done on it. So, no action is required on this document.</p> <p>Note1: Though the action is None, sometimes, it may require to Check Out the document. For example, the Product document is identical in CATIA session and SmarTeam. But since the child part document is updated, the product needs to be checked out to save the modifications in SmarTeam.</p> <p>Note2: The icon is only visible when the following CATIA environment variable is set: CATIA_TEAM_PDM_GR_SHOW_ALL_ACTIONS=1</p>
	<b>Refresh</b>	A more recent revision of this document is available in the SmarTeam database. Refresh should be applied successfully (the document has not been modified in the CATIA session).
	<b>Insert</b>	This document will be imported into the refreshed assembly. Refresh should be applied successfully.
	<b>Remove</b>	This document will be removed in the refreshed assembly. Refresh should be applied successfully (the document has not been modified in the CATIA session).
	<b>Refresh</b>	The document has been modified in the CATIA session AND more recent revisions are available in the SmarTeam database. The document will be updated with the latest revision in the SmarTeam database (which may result in some changes in the CATIA session being lost).
	<b>Remove</b>	The document has been modified in the CATIA session AND the document has been removed in the SmarTeam database. The document will be updated with the latest revision in the SmarTeam database (which would result in the changes in the CATIA session being lost).
	<b>Refresh</b>	The document has been modified in the CATIA session AND more recent revisions are available in the SmarTeam database. The document can't be updated with the latest revision in the SmarTeam database.
	<b>Remove</b>	The document has been modified in the CATIA session AND has been removed in the SmarTeam database. The document can't be updated with the latest revision in the SmarTeam database.

## Documents you can reload

**Global Refresh** enables you to reload CATPart, CATShape or CATProduct documents imported inside CATProduct documents. The following documents cannot be reloaded from SmarTeam database:

- MODEL, cgr or multiCAD referenced document,
- CATPart pointed by a CATPart (i.e. Ref link),
- Design Tables, CATMaterial documents, jpeg files (referenced by a CATPart or CATProduct document),
- A document that appears in a unloadable state.

## Part Number

The Part Number column displays the Part number of the document or the filename of the document (if the document is not in the CATIA session).






## File Type








The File Type column displays the type of the document. This type can be:

<b>Product</b>	For CATProduct document
<b>Part</b>	For CATPart document
<b>Shape</b>	For CATShape document

## CATIA State

The CATIA State column displays the SmarTeam state of the document (as displayed in the table below) and the SmarTeam revision of the document that is currently loaded in the CATIA session.

	Not in Database
	New
	Checked In
	Checked In and Modified
	Checked In and Not Latest

	Checked In and Modified and Not Latest
	Checked Out
	Released
	Released and Modified
	Released and Not Latest
	Released and Modified and Not Latest
	Obsolete

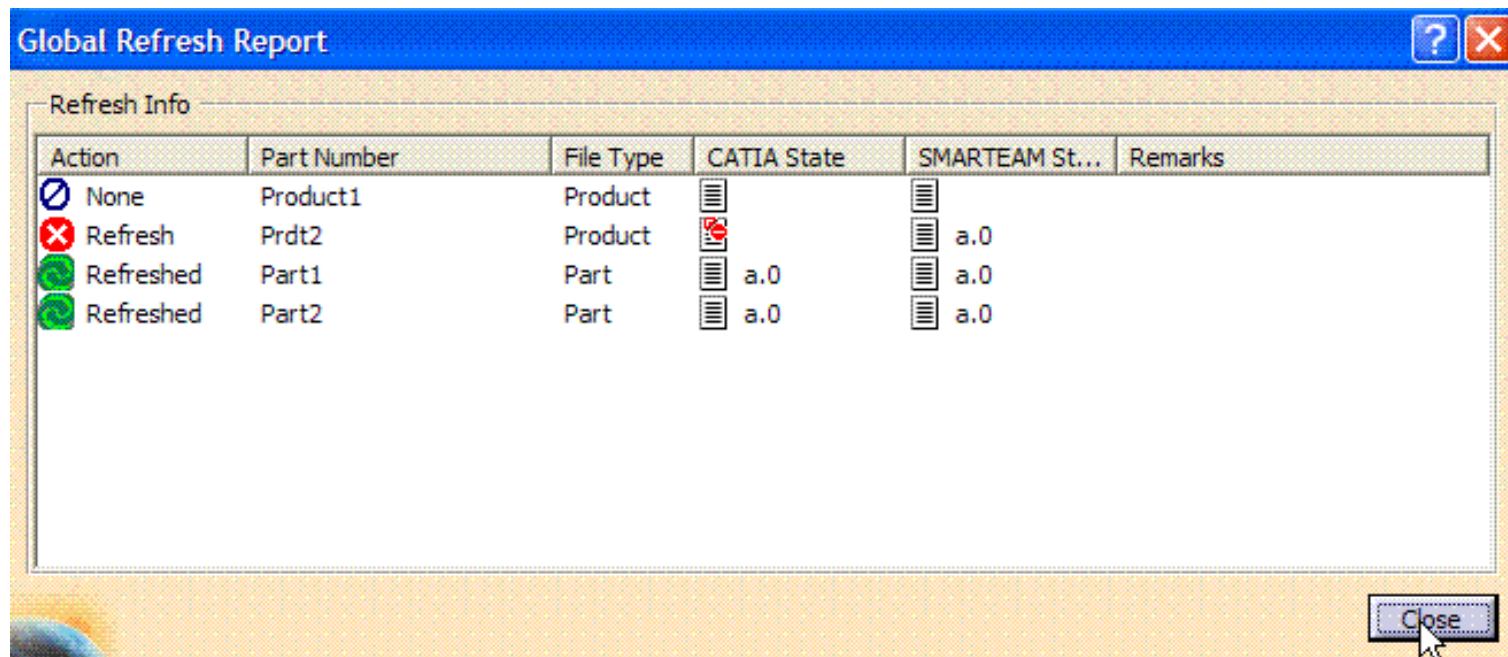
## SMARTeam State

The Revision in Database column displays the latest available revision of the document in the SmarTeam database.

## Remarks




The Remarks column displays warning and error messages.

3. Click **Refresh** to run the operation.  
Once the operation is complete, the Global Refresh Report dialog box is displayed.



## Action

The Action column displays the results of the operation for each document concerned. These results can be:

	None	No action was required on the document
	Refreshed	The document in the current CATIA session has been replaced with the latest available document in the SmarTeam database
	Inserted	The document has been inserted in the current CATIA session
	Removed	The document has been removed from the current CATIA session
	Refresh	The document in the current CATIA session could not be replaced with the latest available document in the SmarTeam database.
	Insert	The document could not be inserted into the current CATIA session
	Remove	The document could not be removed from the current CATIA session

## File Type

The File Type column displays the type of the document. This type can be:

Product	For CATProduct document
Part	For CATPart document
Shape	For CATShape document

## CATIA State

The State column displays the new state of documents once the global refresh is complete.

## SMARTEAM State

The SMARTEAM State column displays the SmarTeam revision of the document. For an object that was refreshed, the value should be the same as the value in the CATIA State column.

4. When the operation is complete, the changes won't be saved automatically but then the command prompts for prompts for a Check Out on the fly operations. Performing the Check Out changes the revision if the product and this needs not to be saved.



Refer to [Global Refresh from SmarTeam](#) for more information on configuring the **Global Refresh** operation.

## After a Global Refresh

Once you have applied a Global Refresh to your CATIA session, you cannot undo the operation.


## Note

When manipulating your documents in a collaborative environment, you should remember two key ideas:

- The Global Refresh capability affects your CATIA session only.
- To control the integrity of your data in SmarTeam, when the CATIA specification tree indicates that one or more documents are "dirty", keep in mind that to make changes to the SmarTeam database, you need to check-out and then perform a SmarTeam save on the appropriate documents.



# Keeping the Integrity of Vaulted CATIA Documents


 Because collaborative design frequently involves a large number of designers, it is important to ensure that the designer's team can share documents efficiently.

Most of the time, sharing documents means that design team members reference each other's designs. Although they work on different parts of the final assembly, they need to be informed about the different versions of documents that gradually become available. This is particularly true when certain phases of design require that a large number of lifecycle operations is carried out.

## Tracking Link Modifications

From V5R15 onward, each designer can track in their sessions link modifications resulting from **lifecycle** or **replace with selected revision** operations carried out by another member of the team. This is now made possible by CATIA which performs a document link analysis across revisions helping you verify version reliability.

Concretely speaking, the gray read-only modified symbol

 Product5 (Product5.1) appears on documents' icons in CATIA specification tree whenever the status of these documents as shown in your CATIA session is not consistent with the SmarTeam database contents.

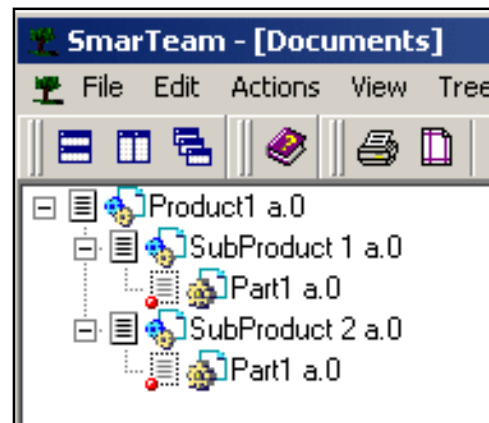
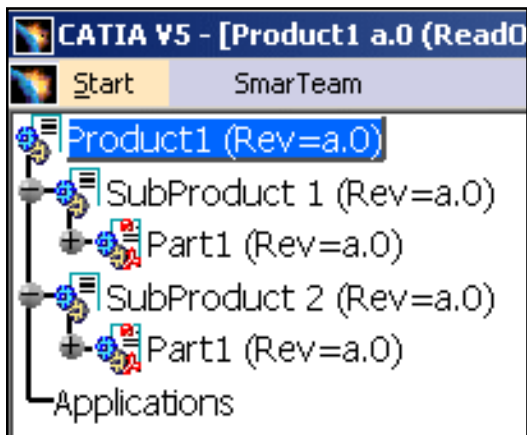
Displaying this type of information ensures that designers are informed that they need to perform check out operations to receive the most updated version of documents referencing sub-components.

## Example

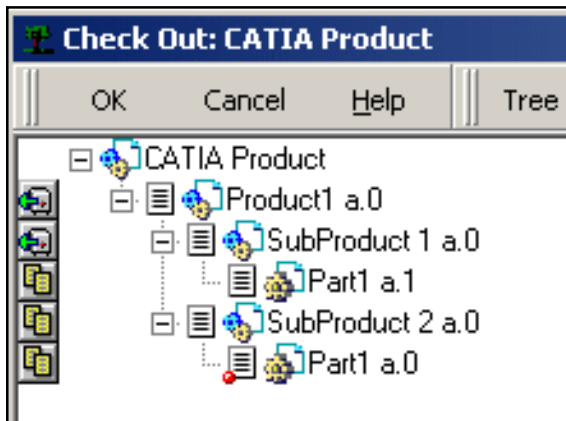
To better comprehend when the application behaves as explained above, suppose a CATIA session displays an assembly, Product1, which is made of two sub-components referencing the same part, Part1. When opening the assembly document, red dots on Part.1's icon indicate that this document is not the latest revisions available in SmarTeam.

The assembly displayed in CATIA

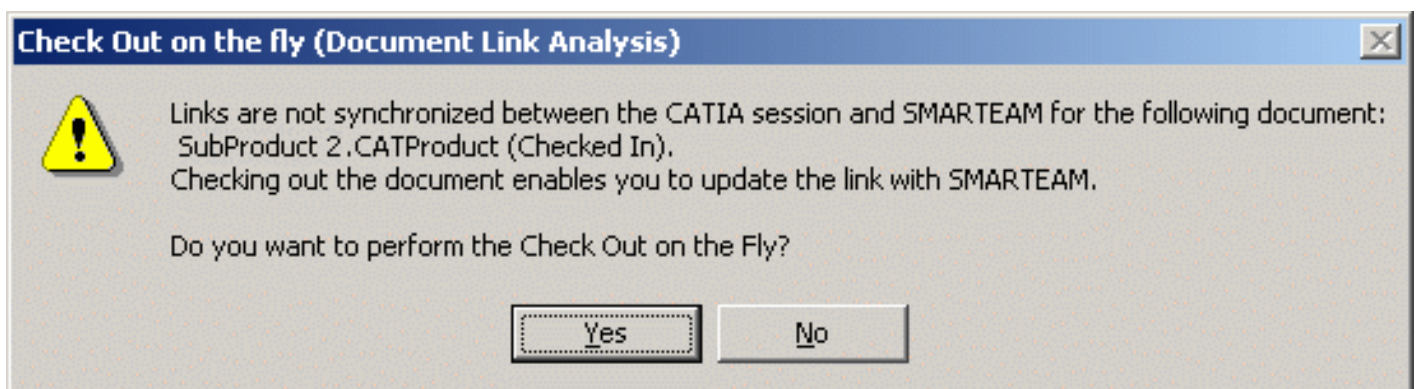
The assembly displayed in SmarTeam



To make SubProduct 1 points to the latest revision available in SmarTeam, it is necessary to apply the **Open For Edit** command onto Product1. This operation establishes a link between SubProduct 1 and the latest revision of Part1, restoring therefore the integrity of data in SmarTeam. Once the checkout operation is complete, this is the database status, where you can notice that SubProduct 2 still points to an old version of Part 1:

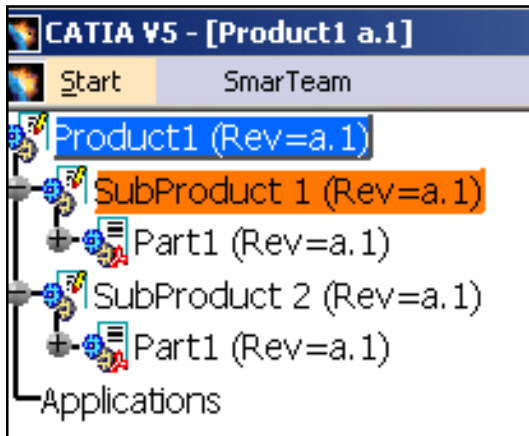


Applying the **Open for Edit** command on Product 1 both checks out the document and opens it in CATIA. Only one revision of Part1 is loaded in CATIA (only one revision is located in the work directory): this is the last revision, that is Part a.1. Consequently, SubProduct2 a.0 is in a state different from the database one: In CATIA, SubProduct2 points to Part1a.1 whereas it points to Part1a.0. in SmarTeam. To warn the user of that difference, the application proposes a **Check Out on the fly** operation for SubProduct 2:



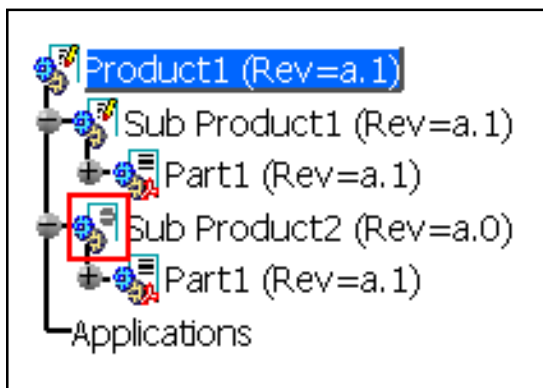
Accepting a **Check Out on the fly** operation allows to create a new revision of SubProduct2 pointing to Part1a1.

After performing the check-out operations, CATIA specification tree eventually shows that all products point to the latest revision of Part 1..



## Not Performing the Check-out on the fly

In case the **Check Out on the fly** operation is not accepted, the read-only modified icon appears on SubProduct2 in CATIA specification tree. SubProduct2 will not be linked to the latest revision of Part.1 until the user performs a checks out SubProduct2.



## Authorizing Check Out on the Fly

The ability for the user to perform a check-out on the fly is defined by the system administrator. For more information, see [System Variables for CATIA and SmarTeam](#).



# Design Compare



This section describes the functionalities related to **Design Compare**. This part of the solution groups several scripts and dll files in order to load different revisions of the same CATIA Part or CATIA Product at the same time.

A CATIA Document is identified by its Universal Unique Identifier (UUID) and by its File Name. The combination of these two identifiers prevents duplicate loading of the same file. Similarly, different product/part revisions of the same file cannot be loaded in the same CATIA session.

However, the Design Compare option offers the following functionalities to you to perform a comparison of different revisions of a file:

**Part Comparison:** Allows you to load different revisions of the same CATIA Part at the same time in one CATIA session. **Product Comparison:** Allows you to load different revisions of the same CATIA Product at the same time in one CATIA session.

Both of these functionalities copy the document and change its file name. Selected parts are copied into the work directory and are renamed automatically based on the following rules:

- **File Name** is renamed to **FileName\_RevisionNumber**.
- **Part Number** is renamed to **PartNumber\_RevisionNumber**.

**Note:** "Part Number" is also changed to facilitate recognition of different revisions.



For more information, refer to [Part and Product Comparison](#).



# Part and Product Comparison



## Part and Product Comparison

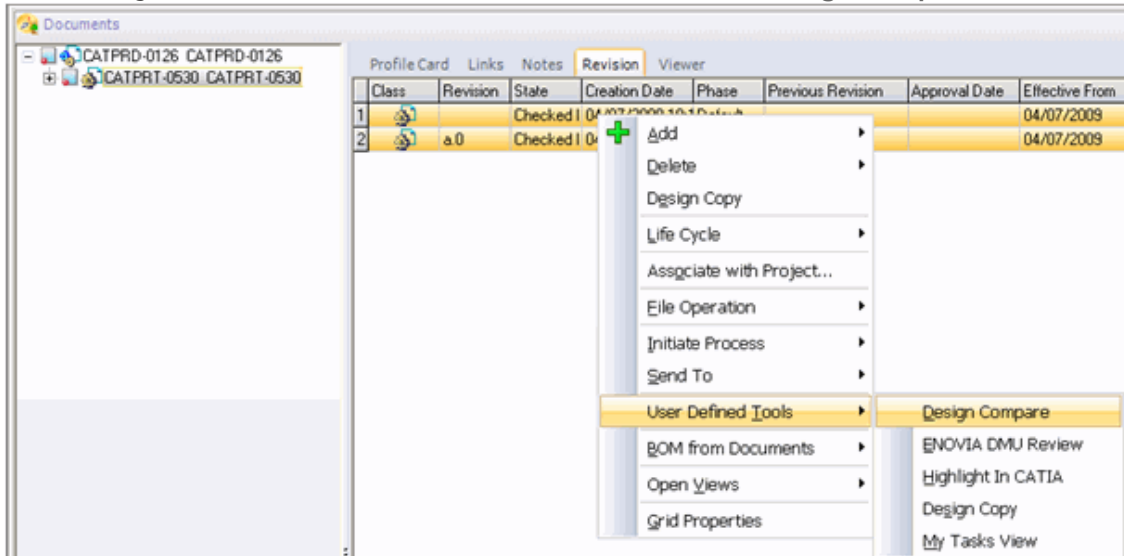
The **Design Compare** command helps designers to load several versions of the same Part or of the same assembly in CATIA and to analyze the geometrical differences through standard CATIA tools.



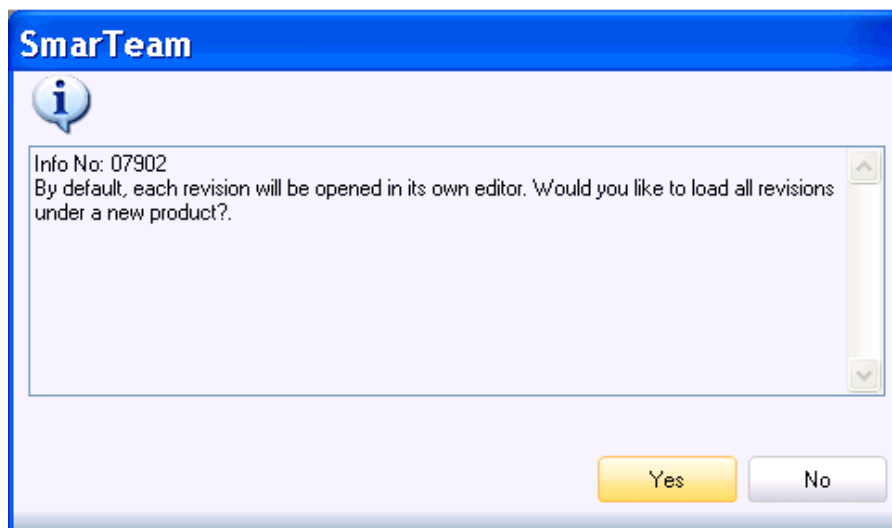
## Launching *Design Compare* through SmarTeam - Editor

CATIA must be launched and connected to SmarTeam before performing the following steps:

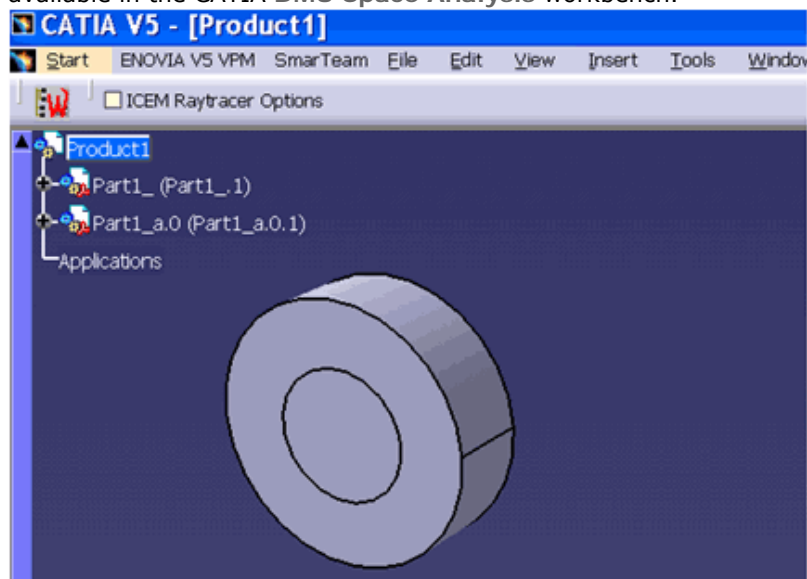
1. Select a CATIA Part in the Document Tree.
2. Select one or more revisions of this Part in the **Revision** tab.
3. From the right-click menu, select **User Defined Tools** and then **Design Compare**.



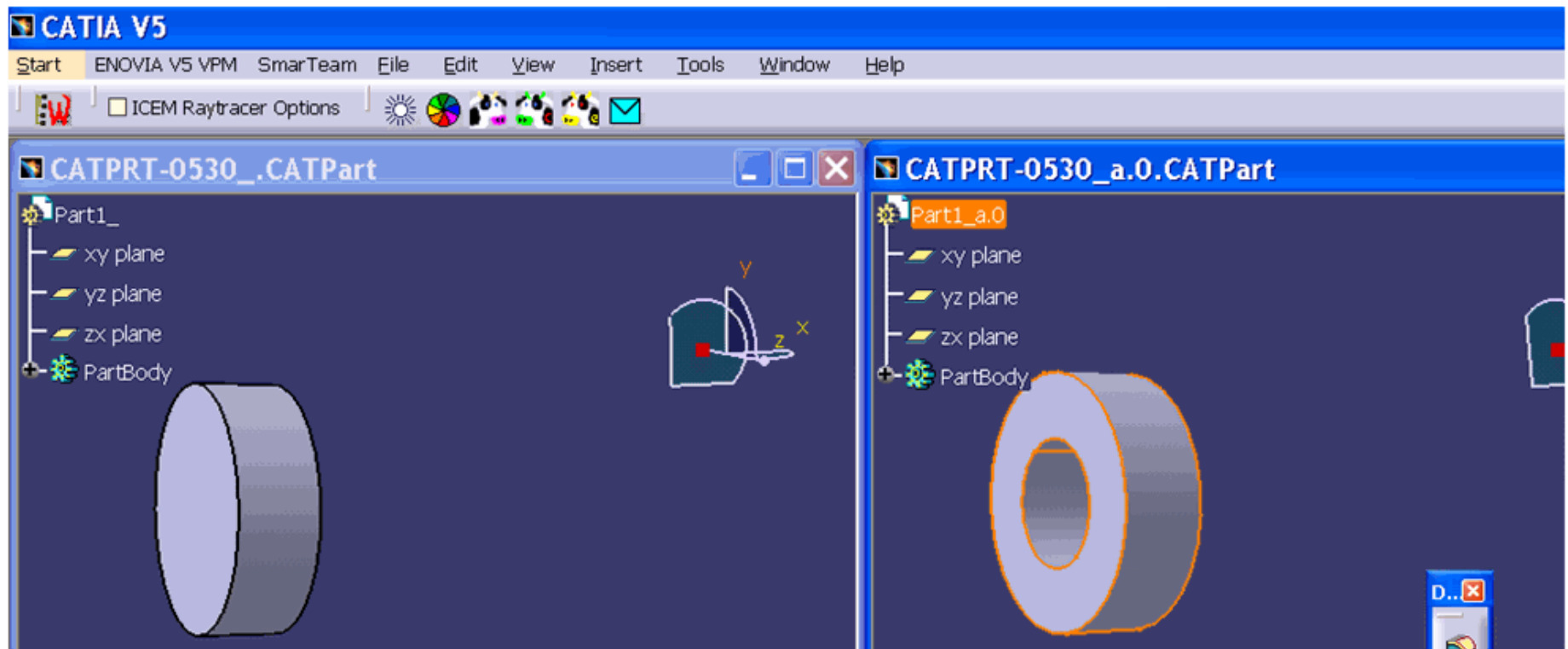
A message is displayed, asking what kind of comparison the designer wishes to perform:



**When you click YES:** All Part revisions will be added into a new root Product. To facilitate the "Geometrical comparison", you can use the comparison tools available in the CATIA DMU Space Analysis workbench.



**When you click NO:** CATIA Parts will be loaded as separate windows in a CATIA session. To facilitate the Visual Comparison, you can arrange these windows with a "Tile Vertically" configuration:

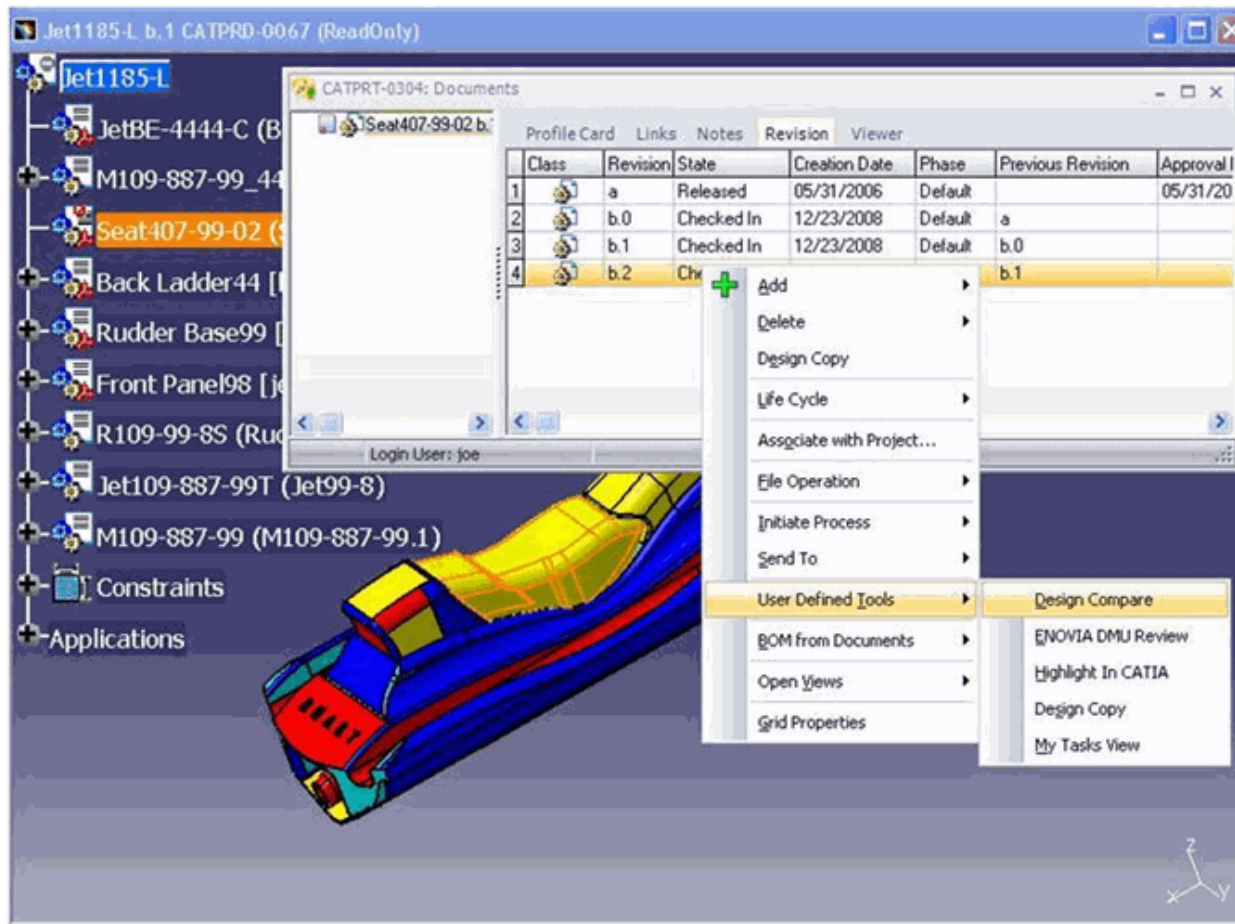


## Launching *Design Compare* through CATIA

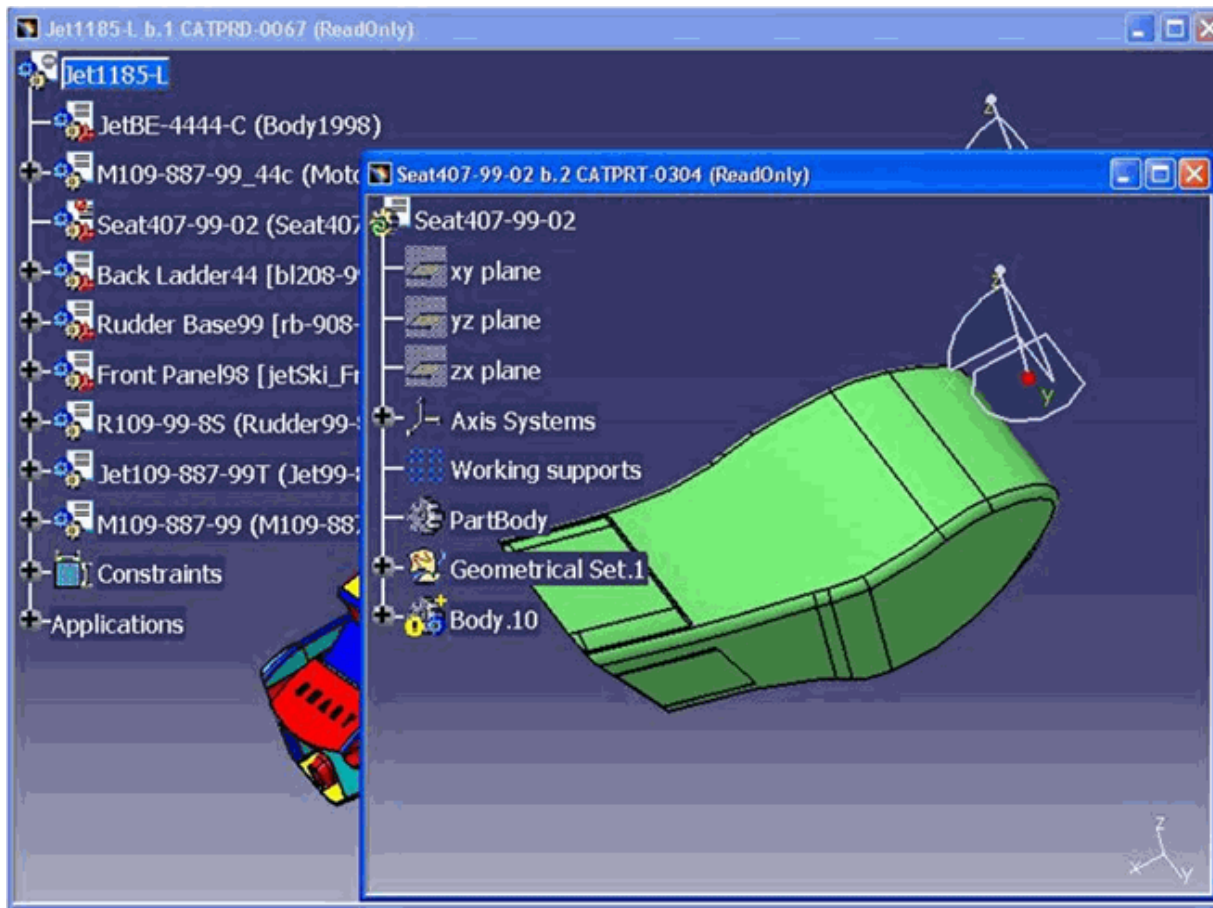
Using the Design Compare command through CATIA helps the designer to compare, for example, the latest version with a previous one already loaded into the CATIA session.

Activate CATIA and then perform the following steps:

1. Select a Part or Product in the CATIA tree and then select **Show Profile Card**. The SmarTeam profile card is displayed in the CATIA session.
2. Select the latest version of this Part or Product in the **Revision** tab.
3. From the right-click menu, select **User Defined Tools** and then **Design Compare**.



The latest version is automatically loaded in a new window into the CATIA session:



In order to load different versions of the same document, Design Compare must rename the documents. In the case of a CATProduct, all children are also renamed. Renamed documents are unknown by SmarTeam (SmarTeam icons are not displayed) and are considered as new CATIA documents. Performing a SmarTeam Save on such documents is not permitted.



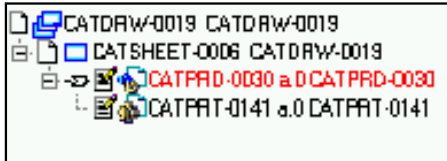
# Enriched Decision Support with All V5 Links



Interdocument relations are part of the CATIA/DELMIA/ENOVIA V5 architecture. (There is a large number of documents and types of relations between these documents).

The native SmarTeam integration of CATIA link semantics enables dedicated behavior on each link type:

- easier navigation:
  - link type display (icon)
  - filter document relation filtering by link type



- improved impact analysis capabilities
- lifecycle rules
- key operations:  
Edit/Check-Out, View, Check-In, Release, New Release, Obsolete.

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.

## Examples

- The view of a .catalog document will not bring onto the local disk the referenced Parts.
- The edit of a CATAnalysis file will by default check-out and copy to the local disk the results (*CATAnalysisResult* & *CATAnalysisComputation*).

## CATIA Links



Enriched decision support is provided by means of the native SmarTeam support of V5 links.

icon	LINKS
	CATIA Product Link (P)
➤	CATIA Design Link (D)
-->	CATIA Rule Base Link (RUL)
-->	CATIA Design table Link (DT)
-D	CATIA Downstream Application Link (DA)
-->	CATIA Reference Link (REF)
-C	CATIA Contextual Link (C)
-->	CATIA Result Link (RES)
	CATIA Is Composed Of (IS)

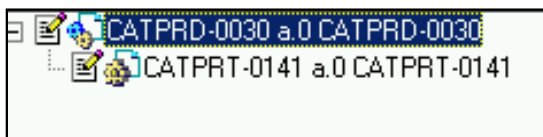
Here is the list of CATIA links natively supported in SmarTeam:

- [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:

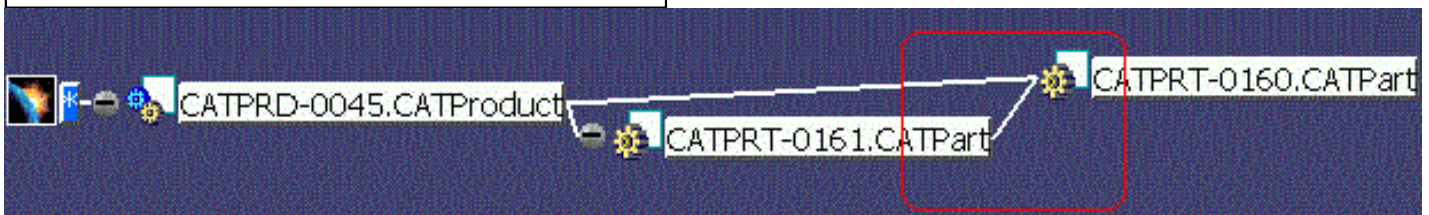
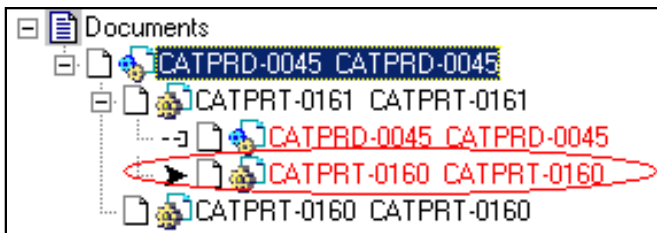
*CATDrawing>Sheet, Catalog>catalog*

- 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.

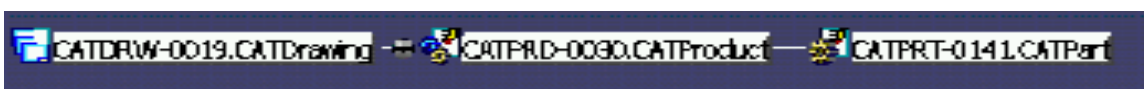
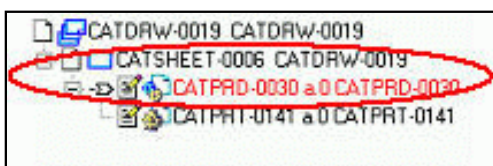


- Downstream Application link

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

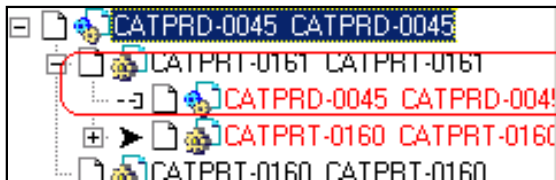
Example:

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



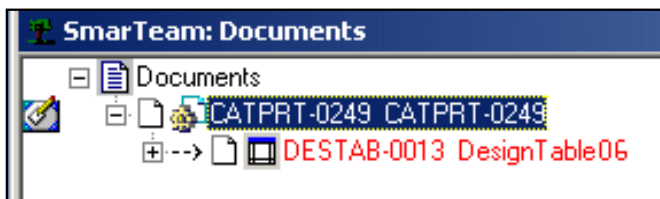
- Contextual link

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)

## Display Impact



The impact analysis can be performed by displaying the links of a given object. You can decide to display/filter CATIA links within SmarTeam by means of the SmarTeam **Tree Properties** command. For more information, see [Customizing the Links Display](#).



# Managing Multiple CATIA Environments



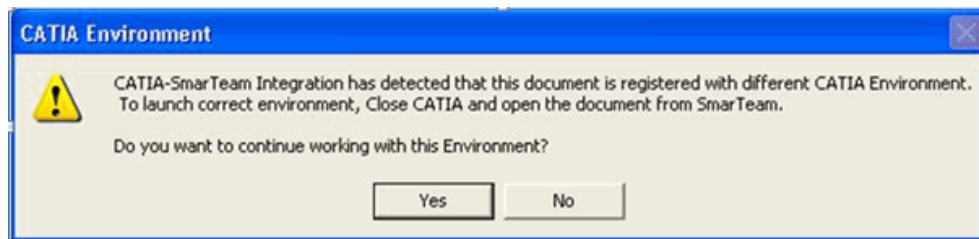
## Concept

The Multiple CATIA Environments facility has been created for the Supply Chain, and has been added to facilitate OEM requests to handle multiple CATIA releases and settings through a single SmarTeam installation.

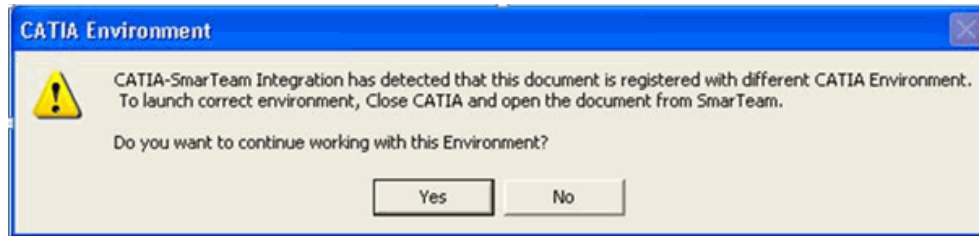
This allows the administrator to manage all CATIA data seamlessly in the proper environment in order to prevent saving data in the wrong CATIA version, wrong Drawing standards, etc.

The administrator builds a suitable environment for CATIA. When the user uses this environment and saves an object to SmarTeam, the object is saved with the correct mapping of the CATIA Environment Name.

When opening a CATIA document from SmarTeam, the appropriate CATIA environment is opened for editing the document. If a different CATIA environment is already open, a warning message notifies the user that the current environment is not the correct environment for editing the document.



When saving a CATIA document in a CATIA environment other than the one in which it was last saved, a warning message notifies the user.



## Configuration

The Multiple CATIA Environment configuration comprises two main stages:

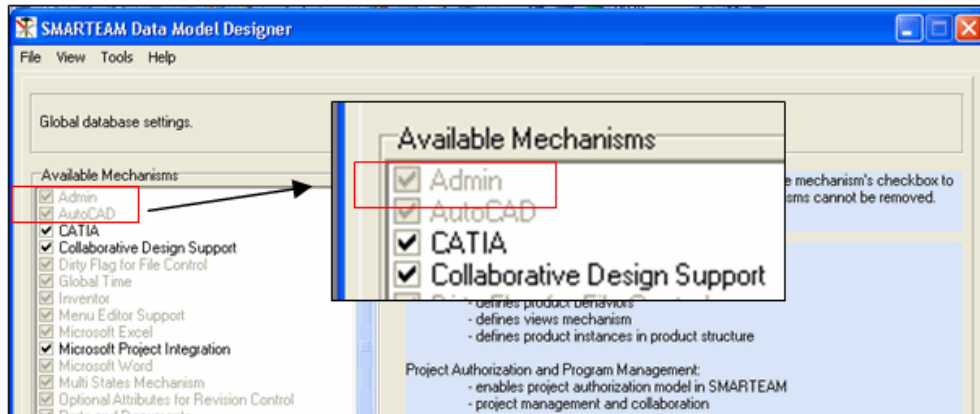
- Database Preparation
  1. Run the Data Model Designer utility.
  2. Update the CATIA document profile cards through the Form Designer.
  3. Define the mapping using Integration Tools Setup.
- CATIA Environment Definition and Registration in SmarTeam
  1. Customize the CATIA environment through the CATIA Environment Editor.
  2. Create relevant CATIA environments in SmarTeam.
  3. Modify SmarTeam keys using SmarTeam System Configuration Editor.

## Database Preparation

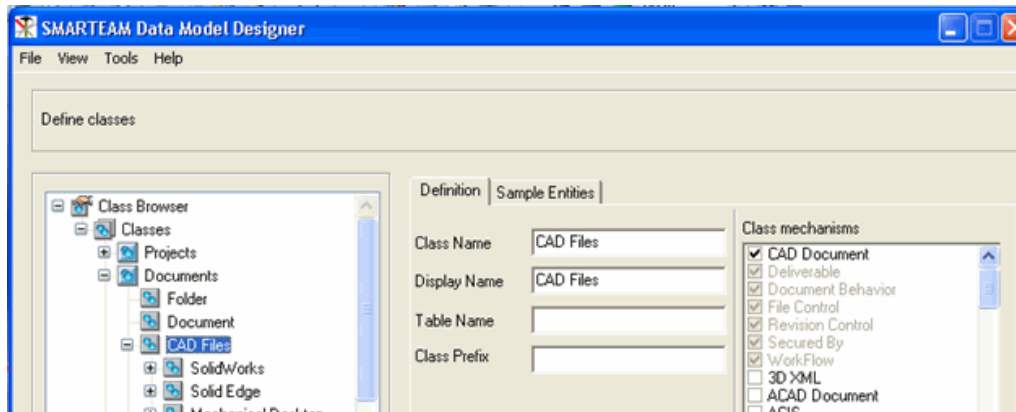
This section includes running the Data Model Designer, updating the CATIA documents profile cards and defining a mapping between CATIA properties and Smarteam attributes.

To prepare the database:

1. Launch the SmarTeam Data Model Designer.
2. From the File menu, select **Modify Database Structure**.
3. Select the R19 Database.
4. Validate the warning message and log in as **joe**.
5. Check that the Admin Mechanism is selected in the list of Available Mechanisms.



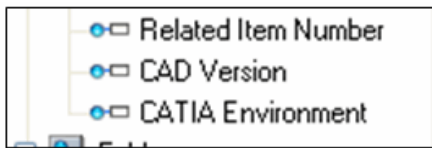
6. Click on **Next** and enable the **CAD Document** behavior for the CAD files.



7. Verify that under the Admin Superclass, CAD Environment class with CATIA Environment sub-class is present.



8. Click on next and verify that two new attributes, **CAD Version** and **CATIA Environment** are present in the List of Attributes for the **Class Documents**.



9. Click **Create**.

## Updating the Profile Card

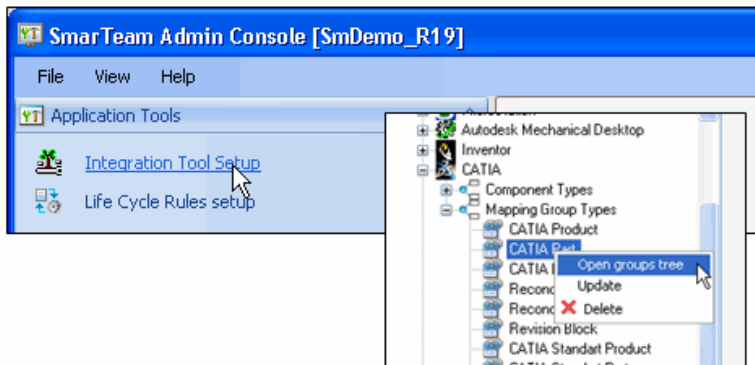
In Form Designer, update the CATIA document profile cards.

1. Launch the Form Designer utility through SmarTeam Admin Console.
2. Modify the CATIA Documents profile card and add the following two fields:
  - **CAD Version** (TextEdit on TDM\_CAD\_VERSION attribute)
  - **CATIA Environment Name** (Ref to Class on TDM\_CAT\_ENVIRONMENT attribute)

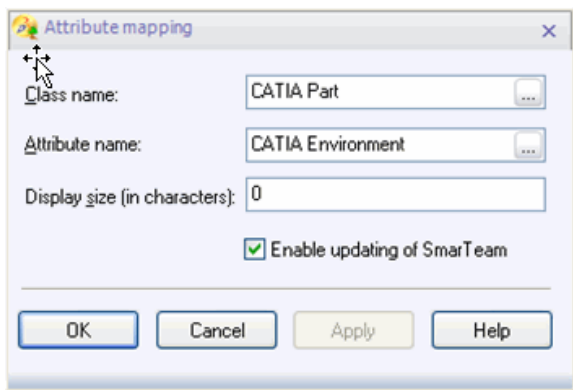
## Integration Tools Setup

In the SmarTeam Integration Tools Setup, define the CATIA Environment Attribute to map to the OEM environment.

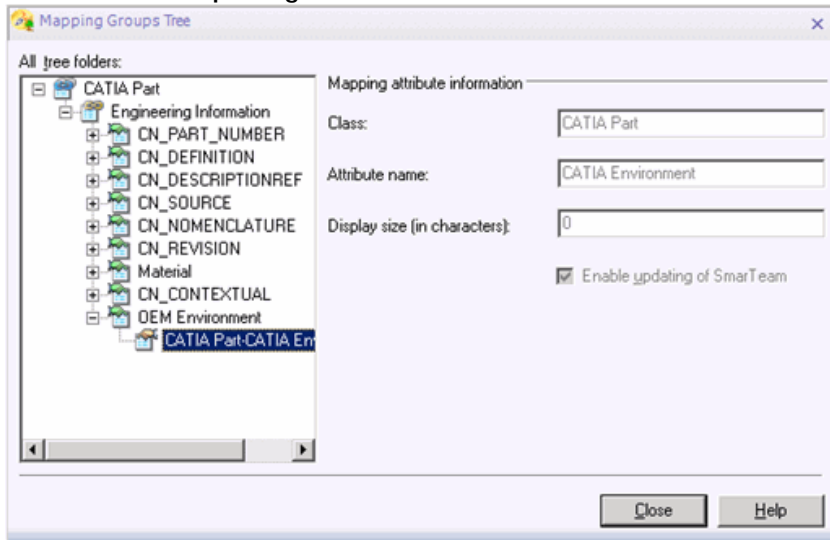
1. In SmarTeam, go to Admin Console and launch Integration Tool Setup.
2. Under CATIA, select Mapping Group Types and then CATIA Part.
3. Right-click and select **Open groups tree**.



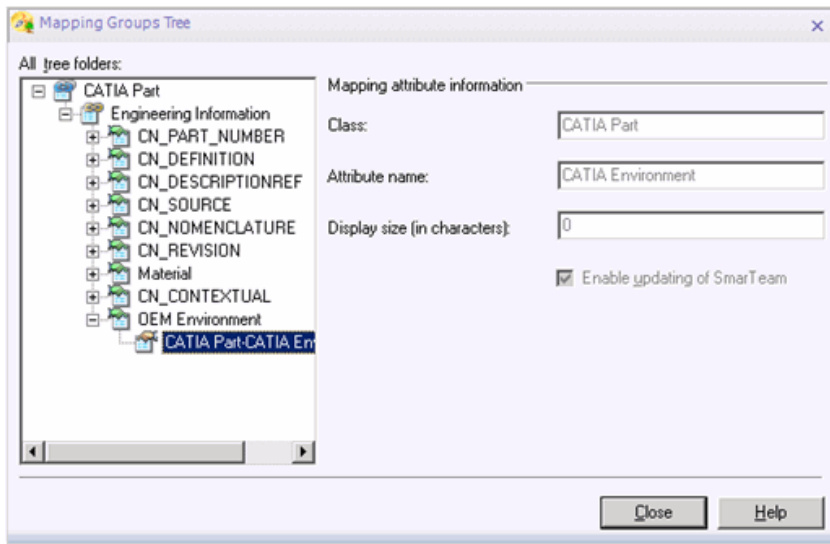
4. Under Engineering Information, add a new property named **OEM Environment**.
5. Map from **CATIA Part** to **CATIA Environment** as shown.



6. Select the **Enable updating of SmarTeam** checkbox and click **OK**.



7. Similarly, map from **CATIA Product** to **CATIA Environment** and **CATIA Drawing** to **CATIA Environment**.
8. In the same way, define the CAD Version attribute to map to the **CN\_VERSION\_TO\_READ** hard-coded property name.
  - a. Under CATIA, select **Mapping Group Types** and then **CATIA Part**.
  - b. Right-click and select **Open groups tree**.
  - c. Under Engineering Information, add a new property named **CN\_VERSION\_TO\_READ**.
  - d. Map to **CATIA Part / CAD Version**.
  - e. Select the **Enable updating of SmarTeam** checkbox and click **OK**.



9. Similarly, define the mapping for **CATIA Product** and **CATIA Drawing**.

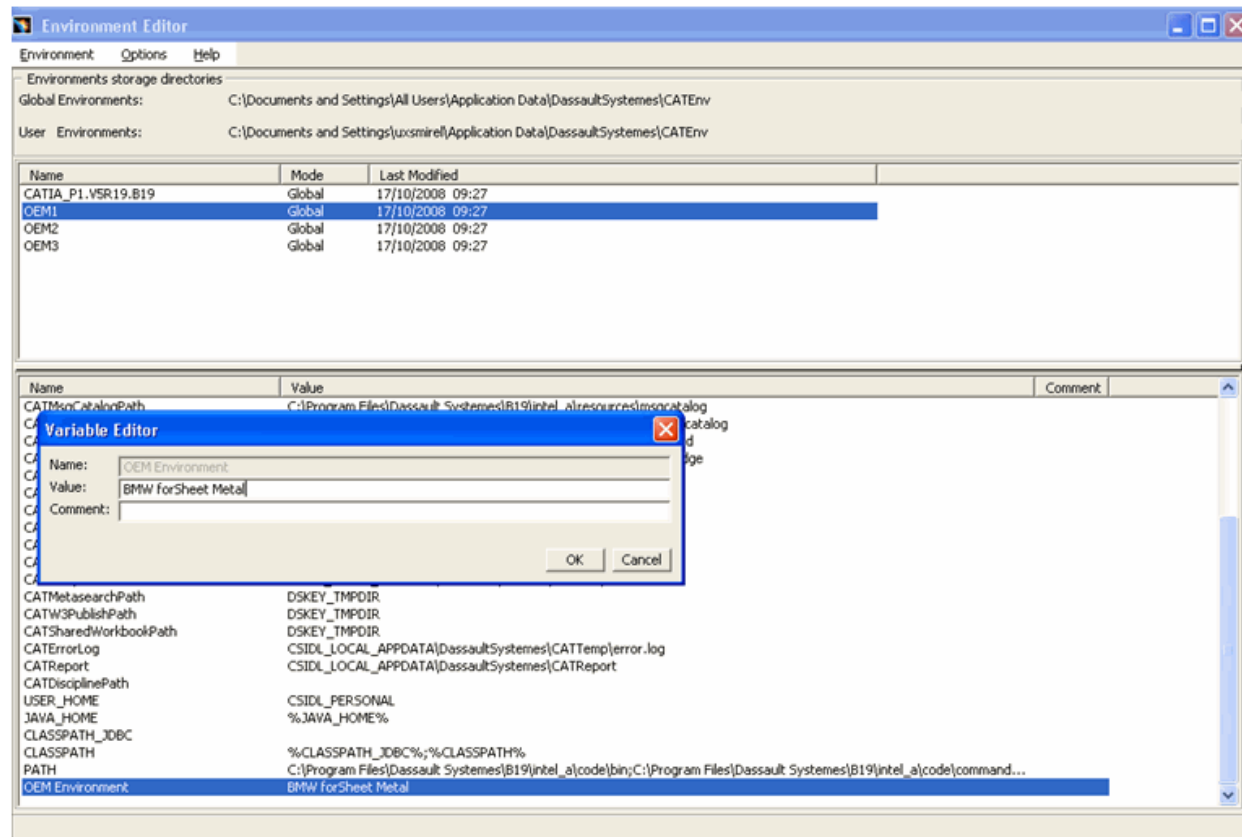
## CATIA Environment Definition and Registration in SmarTeam

### CATIA Environment Definition

On the CATIA side, the administrator must define and set a new variable named **OEM Environment** for each CATIA environment. This variable is the connection between the OEM Environment and the SmarTeam profile card.

For more information about customizing environments, refer to Customizing Your Environment on Windows.

1. Select the **Start> Programs> CATIA> Tools> Environment Editor V5R19** command.
2. Define and set a new variable as follows:
  - o Name: **OEM Environment** (This name is mandatory.)
  - o Value: OEM Environment Name



This variable can be added manually in the CATIA CATEnv file.

```
OEM1.txt - Notepad
File Edit Format View Help

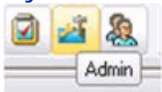
-----
DASSAULT SYSTEMES - V5 ENVIRONMENT FILE
-----
! MODE : User
! TYPE : CATIA_P3
! TMSTMP : 1238164226
! ARGS : -icon yes -menu yes -desktop no -cs CATIA_P3 -a user
-----

CATInstallPath=C:\Program Files (x86)\Dassault Systemes\B19\intel_a
CATDLLPath=C:\Program Files (x86)\Dassault Systemes\B19\intel_a\code\b1
CATICPath=C:\Program Files (x86)\Dassault Systemes\B19\intel_a\code\pro
CATCommandPath=C:\Program Files (x86)\Dassault Systemes\B19\intel_a\cod
CATDictionaryPath=C:\Program Files (x86)\Dassault Systemes\B19\intel_a\
CATDocView=C:\Program Files (x86)\Dassault Systemes\B19\intel_a\doc
CATReffFilesPath=C:\Program Files (x86)\Dassault Systemes\B19\intel_a\re
CATFontPath=C:\Program Files (x86)\Dassault Systemes\B19\intel_a\resour
CATGalaxyPath=C:\Program Files (x86)\Dassault Systemes\B19\intel_a\reso
CATGraphicPath=C:\Program Files (x86)\Dassault Systemes\B19\intel_a\res
CATMsgCatalogPath=C:\Program Files (x86)\Dassault Systemes\B19\intel_a\
CATFeatureCatalogPath=C:\Program Files (x86)\Dassault Systemes\B19\inte
CATDefaultCollectionStandard=C:\Program Files (x86)\Dassault Systemes\B
CATKnowledgePath=C:\Program Files (x86)\Dassault Systemes\B19\intel_a\r
CATStartupPath=C:\Program Files (x86)\Dassault Systemes\B19\intel_a\sta
CATW3ResourcesPath=C:\Program Files (x86)\Dassault Systemes\B19\intel_a
CATReconcilePath=
CATReferenceSettingPath=
CATUserSettingPath=CSIDL_APPDATA\DassaultSystemes\CATSettings
CATCollectionStandard=
CATTemp=CSIDL_LOCAL_APPDATA\DassaultSystemes\CATTemp
CATMetasearchPath=DSKEY_TMPDIR
CATW3PublishPath=DSKEY_TMPDIR
CATSharedworkbookPath=DSKEY_TMPDIR
CATErrorLog=CSIDL_LOCAL_APPDATA\DassaultSystemes\CATTemp\error.log
CATReport=CSIDL_LOCAL_APPDATA\DassaultSystemes\CATReport
CATDisciplinePath=
USER_HOME=CSIDL_PERSONAL
JAVA_HOME=%JAVA_HOME%
CLASSPATH_JDBC=
CLASSPATH=%CLASSPATH_JDBC%;%CLASSPATH%
PATH=C:\Program Files (x86)\Dassault Systemes\B19\intel_a\code\bin;C:\P
OEM Environment=BMW for Sheet Meta
```

## Registering a CATIA Environment in SmarTeam

The administrator creates an object for each environment in the CAD Environment class. The **name** must be identical to the **OEM Environment** value defined in the CATIA Environment Editor.

1. **style="font-weight: 400">Launch SmarTeam** and select the **Admin** button on the Browser toolbar.



2. After clicking the **Admin** button, add CATIA Environment. The CATIA Environment Profile Card appears.

3. Fill in the necessary information in the Profile Card as shown for the new CATIA Environment OEM1.
4. In the Attributes1 tab, fill in the complete string for the Environment Command using the following format:  
`-f '%1' -e '<Environment Name><Environment Name>' -c '<Environment Command><Environment Command>'`

Note the syntax of the command:

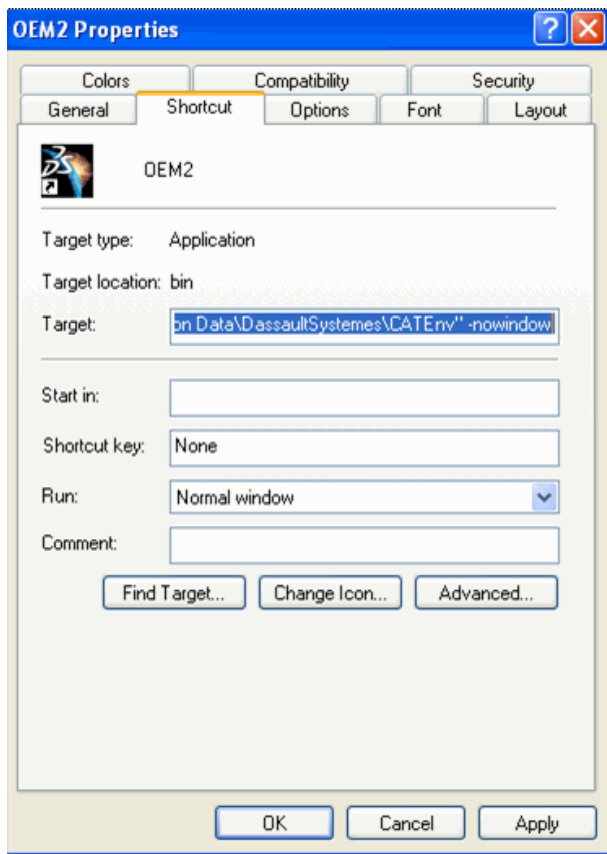
`-f<space>'%1'<space>-e<space>'<Environment Name>'<space>-c<space>'<Environment Command>'`

The String corresponding to `-f`, `-c` & `-e` must be enclosed within single quotation marks ( ' ' )

`-f '%1'` represents the name of the file to be opened in CATIA

`-e '<Environment Name>'` represents the Environment Name

`-c '<Environment Command>'` represents the Environment Command as it appears in the shortcut (Target field of environment properties)



An example of Environment Command:

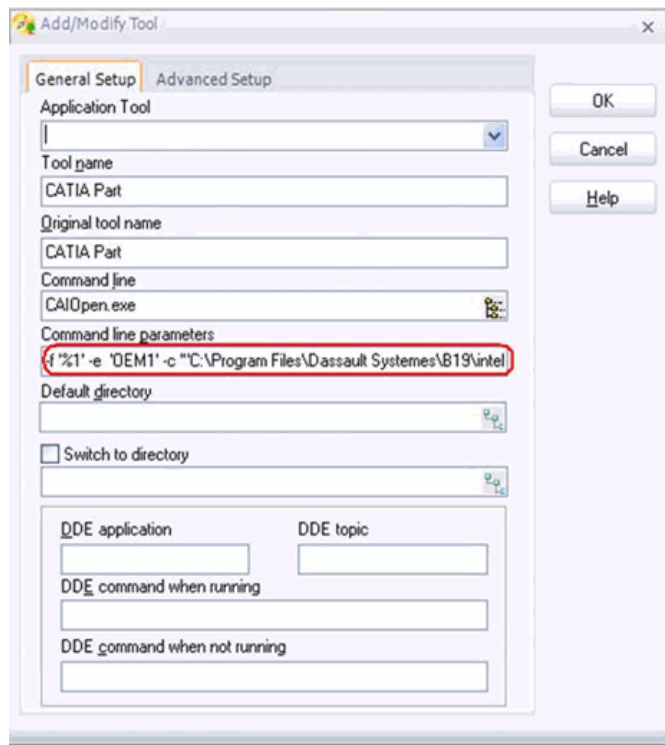
```
-f '%1' -e 'BMW for Sheet Metal' -c ""C:\Program Files\Dassault Systemes\B19\intel_a\code\bin\CATSTART.exe" -run "CNEXT.exe" -env OEM1 -direnv "C:\Documents and Settings\davidd\Application Data\DassaultSystemes\CATEnv" -nowindow""
```

5. Similarly, add two more environments and fill in the information accordingly in the Profile Card for OEM2 and OEM3.

### Application Setup

The administrator set application sets, in Command line parameters, the default environment to be launched in following cases:

- If the Environment Name is empty in the CATIA document object
- If the Environment object with the Environment Name does not exist



## System Keys

Two new system keys are provided to control whether or not the CAD Level attributes are checked when launching or saving in CATIA Integration.

### CheckCATIAEnvironmentOnSave

**Name of Key:** CATIA. CheckCATIAEnvironmentOnSave

**Description:** Specifies whether or not to display warning messages when saving a CAD document with the wrong CATIA environment

**Possible Values:** True/False True = Display a warning message False = Do not display a warning message

**Default Value:** False

**Override level:** Domain

### CheckCATIAEnvironmentOnOpen

**Name of Key:** CATIA. CheckCATIAEnvironmentOnOpen

**Description:** Specifies whether or not to display warning messages when saving a CAD document with the wrong CATIA environment

**Possible Values:** True/False

Where True = Display a warning message; False = Do not display a warning message

**Default Value:** False

**Override level:** Domain



## Cross-version compatibility

Multiple CATIA environment functionality is available for Smarteam V5R19/CATIA V5R18 SP5 and later.



# Toolbar Customization



## Concept

Toolbar customization allows the administrator to customize the toolbars such that only functions that are relevant for the current user are shown. A customization option also allows the administrator to remove the SmarTeam menu from the CATIA interface.



## Configuration

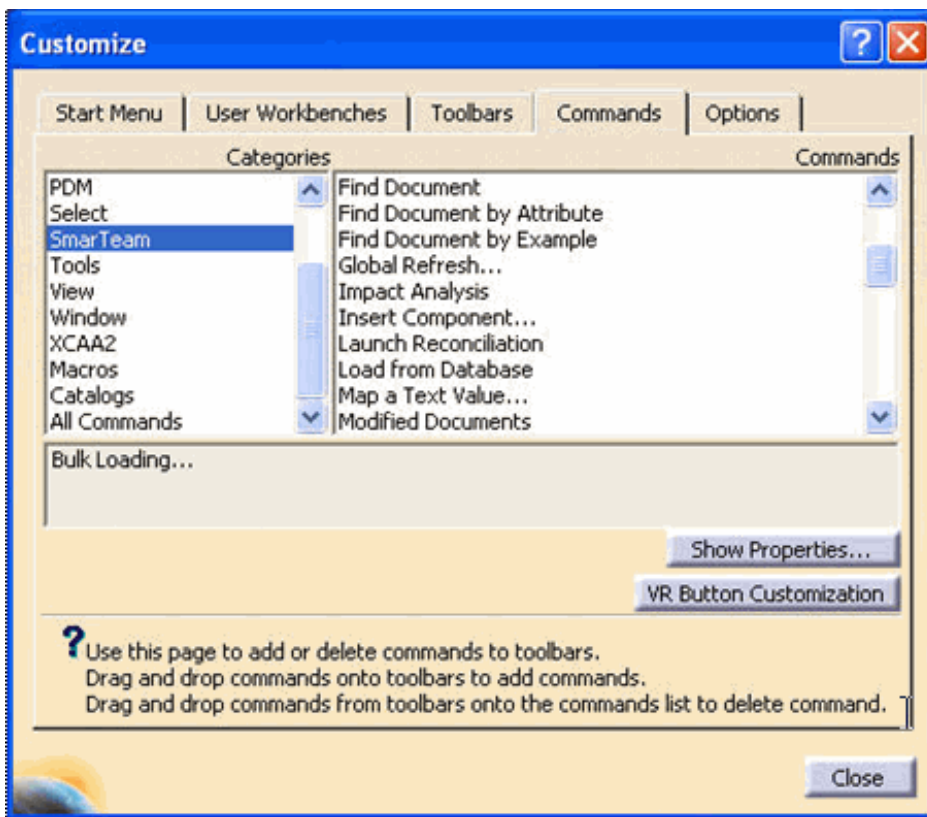
The administrator can configure the toolbar in the standard way.

### SmarTeam Category

In CATIA, a new category named SmarTeam has been added to facilitate easy selection of SmarTeam commands.

To access the SmarTeam category:

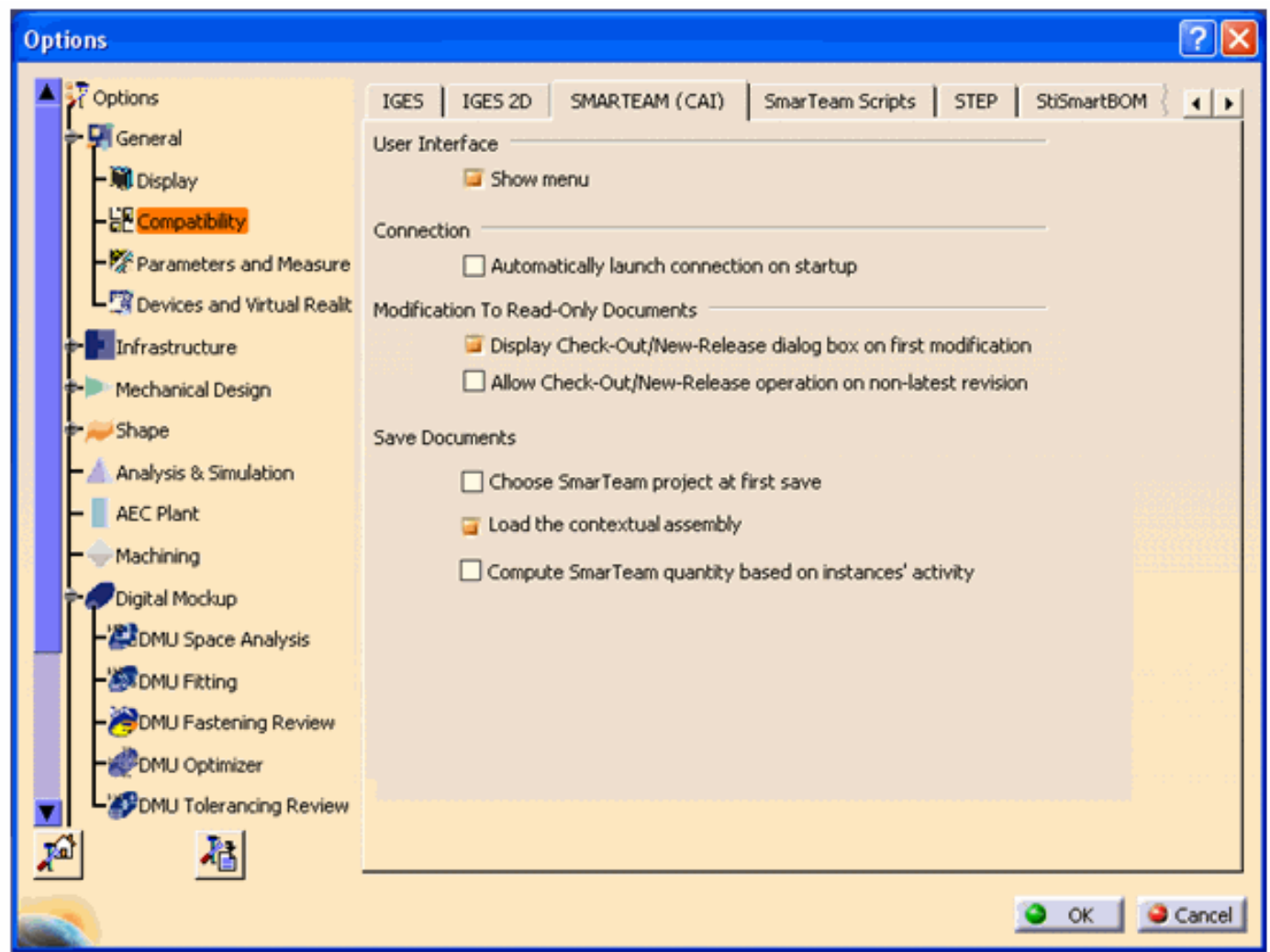
- From the **Tools > Customize > Commands** tab, select **SmarTeam**.  
The SmarTeam commands appear in the right pane.



Using this category, the administrator can create a SmarTeam toolbar which includes the required SmarTeam functions for each role. For example, options such as Save, Check In and Check Out can be assigned to the toolbar.

The SmarTeam menu can be removed:

- From the CATIA **Tools** menu, select **Options** to open the Options dialog box:



The SmarTeam (CAI) tab contains the following options:

**User Interface> Show menu**

For details see [Show Menu](#).

**Connection> Automatically launch connection on startup**

For details see [Automatic Launch Connection](#).

**Modifications to Read-Only Documents**

For details see [Modifications to Read-Only Documents](#).

**Save Documents**

For details see [Save Documents](#).



# Automatic Launch Connection



## **Connection> Automatically launch connection on startup**

When selected, SmarTeam is automatically launched when starting CATIA. If SmarTeam is not yet running, the SmarTeam login window is displayed.



# Modification to Read-Only Documents

## **Modification to Read-Only Documents > Display Check-Out/New-Release dialog box on first modification**

When selected, the Check Out/New Release windows are shown when an initial change is made.

## **Modification to Read-Only Documents > Allow Check-Out/New-Release operation on non-latest revision**

When selected, check out and new release operations can be performed on a revision of a document that is not the latest revision.



# Save Documents



## **Save Documents> Choose SmarTeam project at first save**

- Clear the check box to display the Project Manager window only when the root assembly is saved in SmarTeam for the first time.
- Mark the check box to display the Project Manager the first time a **Save to SmarTeam** operation is performed.

**Note:** When performing **Save to SmarTeam** on a part under a root object, the PM window will appear even if the root object was previously saved.

## **Save Documents> Load the contextual assembly**

- Clear the check box to prevent the SmarTeam Save operation from loading the context during the update of design links, in order to improve performance and memory consumption.
- Mark the check box to load the context during the update of design links.

## **Save Documents> Compute SmarTeam quantity based on instances activity**

- Clear the check box to ignore the instances activity information.
- Mark the check box to send the instances activity information to SmarTeam to allow it to compute the Quantity attribute based on this information.



# Text ID Mapping



## Concept

Title Block mapping allows users to map data more easily. It also allows you to map legacy data. Text in the Title Block can now be mapped directly from/to CATIA.

Until now, the Title Block mapping was based on a CATIA script. In V5R19, the only constraint is that the text id must be defined according to SmarTeam rules; the mapping is performed automatically.

In a CATIA drawing, text is mapped to SmarTeam attributes that can be displayed in the SmarTeam profile card. The user can update the text that is mapped to SmarTeam in the drawing by modifying the text directly or by updating the drawing based on SmarTeam attribute values.



## Procedure

The user uses the Title Block defined by the Administrator. The user can change the text in the Title Block and, after a Save operation is performed, the Title Block is updated.

Note: The mapping only takes place if the Title Block is located in the background of the Drawing sheet.

### Mapping Order

If the same attribute has been mapped in different ways from/to CATIA the mappings take place in the following order:

1. File properties mapped from CATIA to SmarTeam
2. CATIA Text name mapped from CATIA to SmarTeam
3. CATIA knowledge parameters mapped from CATIA to SmarTeam
4. CATIA knowledge parameters mapped from SmarTeam to CATIA
5. CATIA Text name mapped from SmarTeam to CATIA
6. File properties mapped from SmarTeam to CATIA

### Save and Lifecycle Operations

The mapping is performed from CATIA to SmarTeam and then from SmarTeam to CATIA. Mapping is automatically performed in the following events:

- Open a drawing in CATIA
- Save drawing to SmarTeam
- Check in
- Release
- Obsolete
- User clicks SmarTeam menu in CATIA
  - Properties > Load from Database – map from SmarTeam to CATIA Drawing
  - Properties> Save to Database – map from CATIA Drawing to SmarTeam

Release operation can be performed when the drawing is checked out or checked in state.  
Current behavior of mapping revision – related attributes must be preserved as they are for other mappings

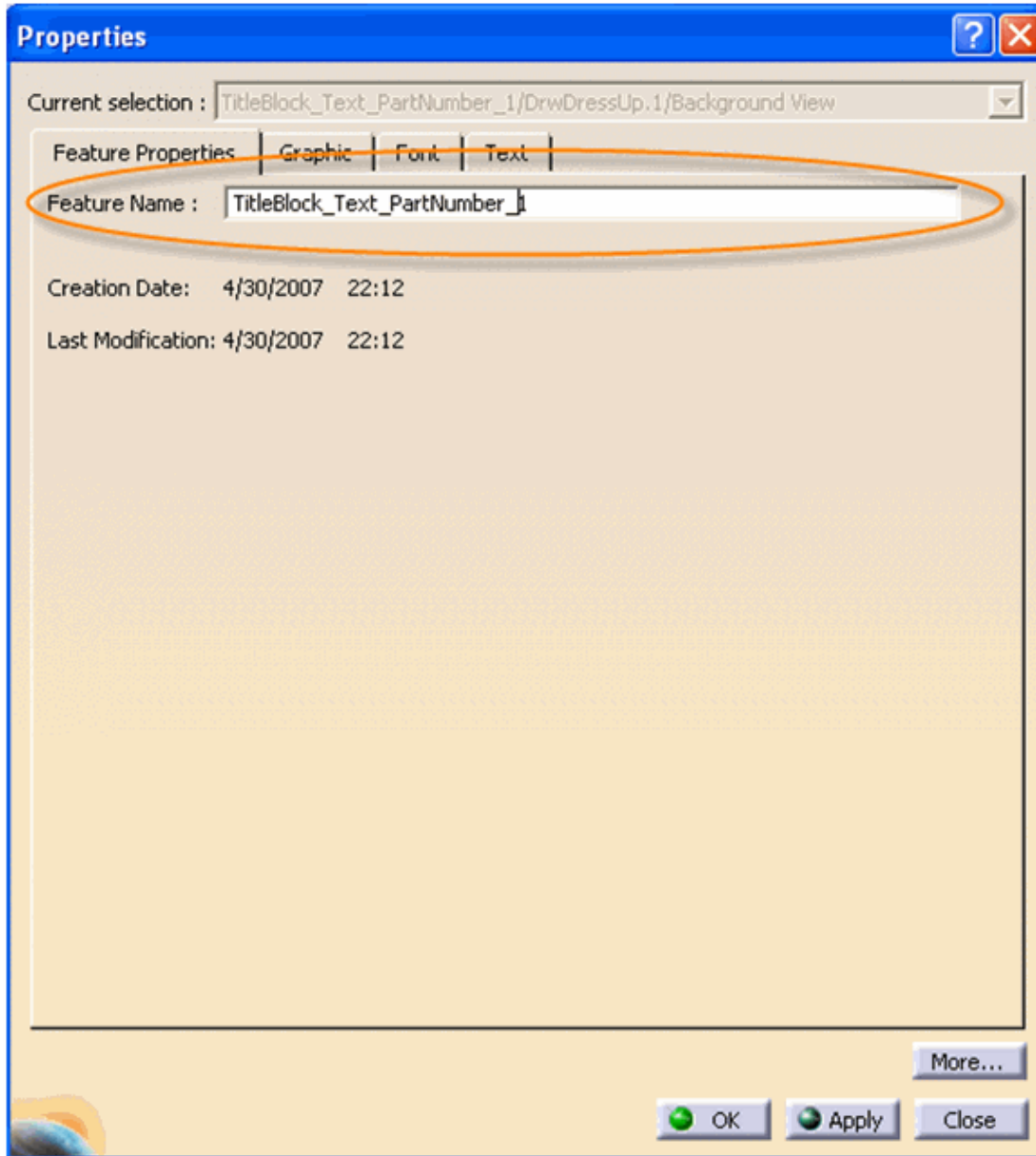
## Configuration

Configuration is done by the Administrator during the setup. It comprises of two steps:

- Defining the Feature Name
- Defining Mapping in SmarTeam Integration Tool Setup

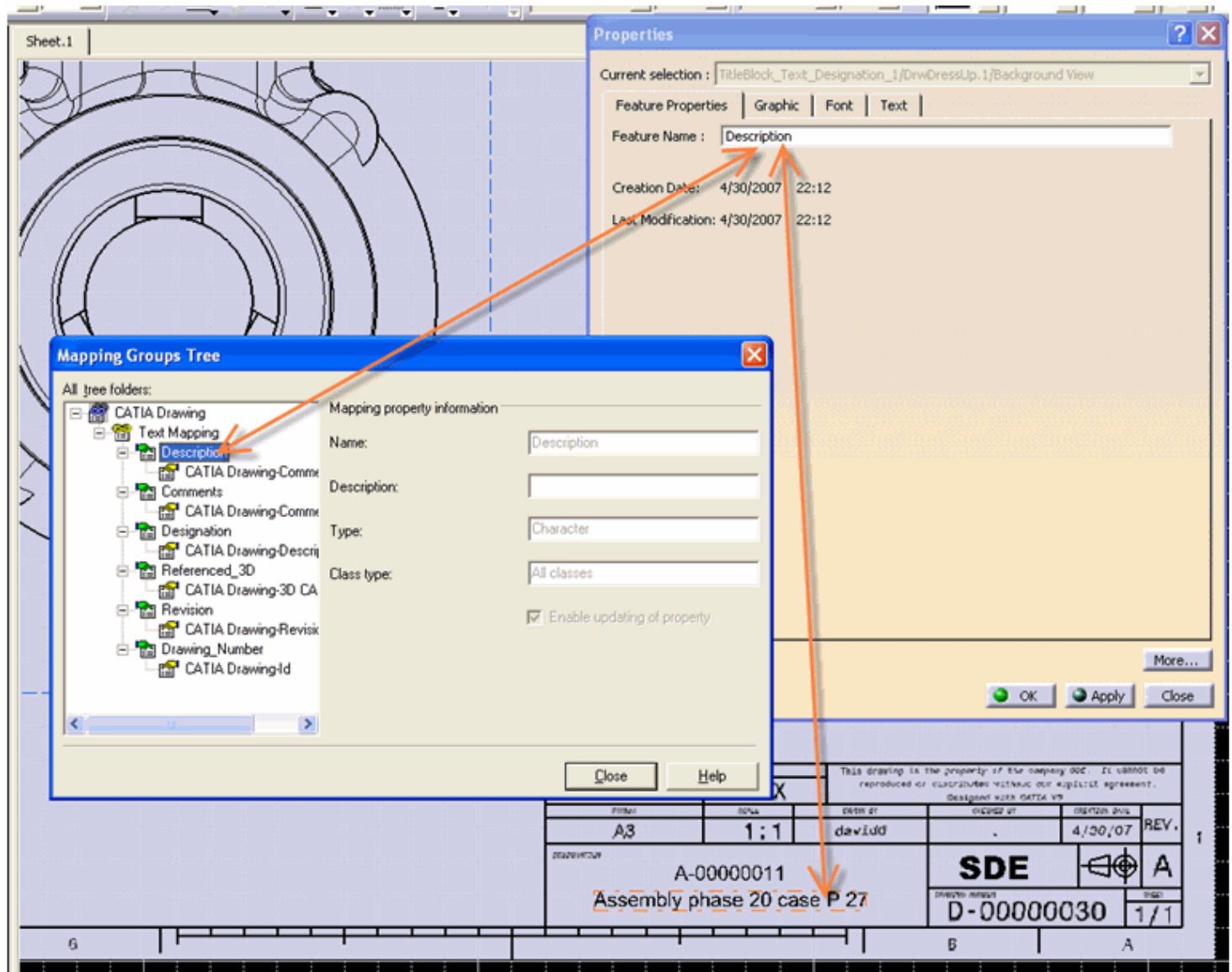
## Defining the Feature Name

In the Title Block Properties window, the administrator types in a specific Feature Name in the Feature Name field.



## Defining Mapping in SmarTeam Integration Tool Setup

The feature name is mapped to SmarTeam using the Integration Tool Setup. In the Integration Tool Setup, the administrator defines mappings between the CATIA Drawing text and a SmarTeam attribute, as shown in the example below:



**Note:** When SmarTeam attributes are mapped to text values containing carriage returns, the carriage return will not be correctly displayed in the title block of a drawing. Drawings ignore carriage returns in text.

The administrator can define the mapping direction by enabling/disabling the updating of properties. For details, see the Integration Tool Setup documentation in the SmarTeam – Editor Online Help.

When Exposed Mode is set to No, the mapping will be defined in the mapping group with the name of the class to which the CATIA Drawing is mapped.

When Exposed Mode is set to Yes, the mapping will be defined in the mapping group with the name of the class to which the CATIA Sheet is mapped.



# CATIA Workbenches Integration

Intended to design engineers, this section provides the methodology for working with data specific to certain CATIA workbenches. The following topics are illustrated:

Instructions for system administrators are described in the [CATIA Workbenches Integration](#) chapter of the Administration Tasks.

# Accessing CATIA Catalogs







Design engineers who need to instantiate catalog components (parts, products, V4 model etc.) can access their catalogs using one of the following methods:

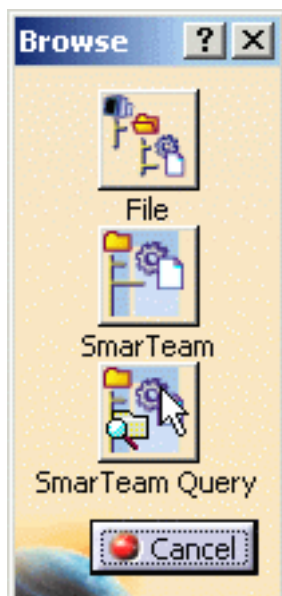
- [Searching in the SmarTeam Vault](#)
- [Searching in the Shared Directory](#)



## Searching in the SmarTeam Vault

Searching in the SmarTeam Vault is a way of accessing catalogs.

1. Click the **Catalog Browser**  icon.
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 perform a SmarTeam query: just click the **SmarTeam** icon   
icon or **SmarTeam Query**  from the **Browse** dialog box.



Depending on on how your administrator configured your CATIA session, the **File Selection** dialog box may appear. In this case, just click the **Cancel** button to close it and enable the **SmarTeam Query** capability in the **Browse** dialog box.

3. Open the document of interest.

The CATIA Part will be copied into your working directory and the CATIA product in your assembly session will reference it. However, once the CATIA Product is checked-in into the SmarTeam vault, the next time you retrieve a CATIA Product into the working directory (check out operation), standard CATIA Parts will not be copied into your working directory and will not be referenced to the shared workspace.

## Searching in the Shared Directory




If your librarian has created shared directories:

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

**Browser**  icon.

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 the **File** icon  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.

For information about shared directories, see [Creating and Saving a Catalog](#). To get precisions about instantiating catalog components, see the *Component Catalog Editor User's Guide*.



# Accessing Sheet Metal Bend Tables



While creating Sheet Metal parts as explained in the *Generative Sheet Metal User's Guide*, design engineers need to use bend tables defined for their companies. This task shows how to access them.



1. Once in the Sheet Metal workbench  , from the Browse dialog box, click the **File** icon

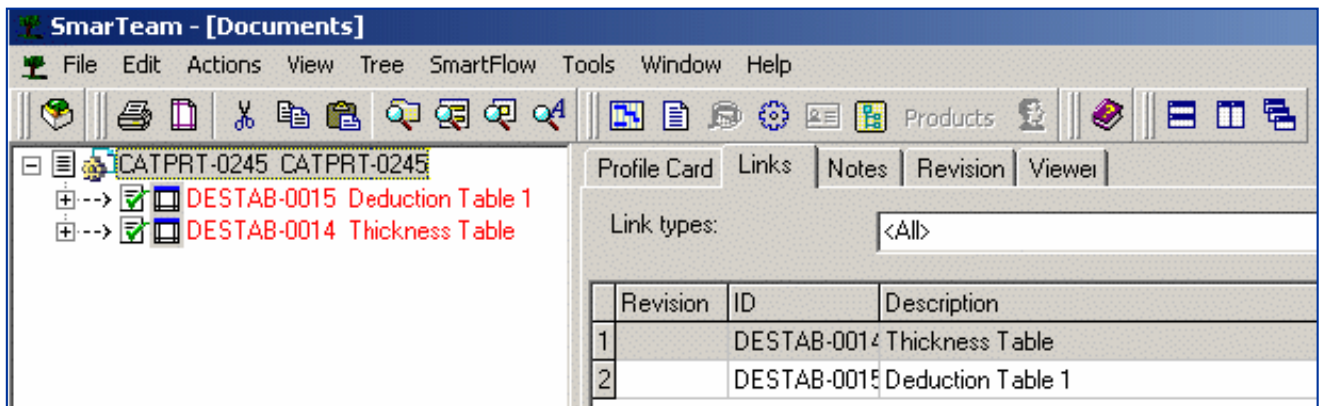


to open the File Selection dialog box.


2. From the File Selection dialog box that appears, navigate to access the bend tables saved in the [shared directory](#) defined by your system administrator.
3. Open the bend table of interest.

As soon as you save your Sheet Metal part, SmarTeam creates a link between this part and the bend table you used to create it.

The links to the thickness table and the bend table are saved.




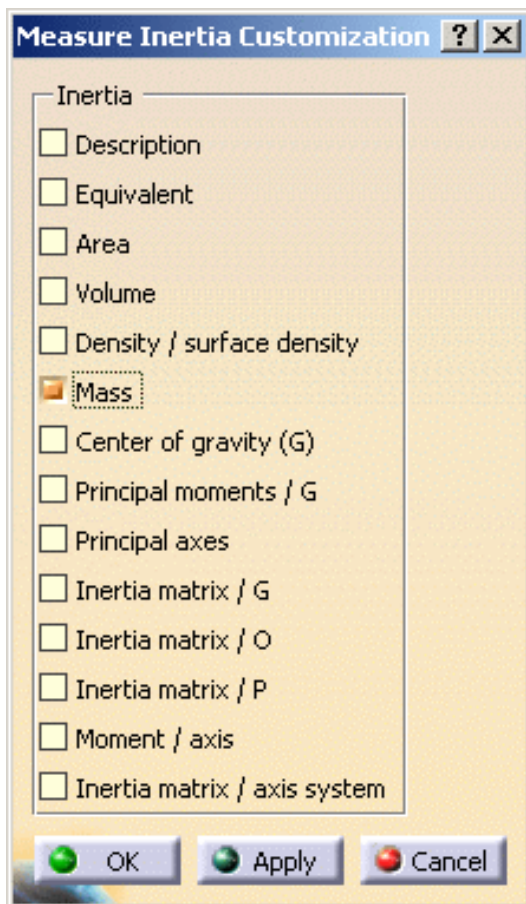
# Creating a Mass Parameter in the Generative Sheet Metal Workbench

 This section shows you how to create a mass parameter in the Generative Sheet Metal workbench. This involves three operations:

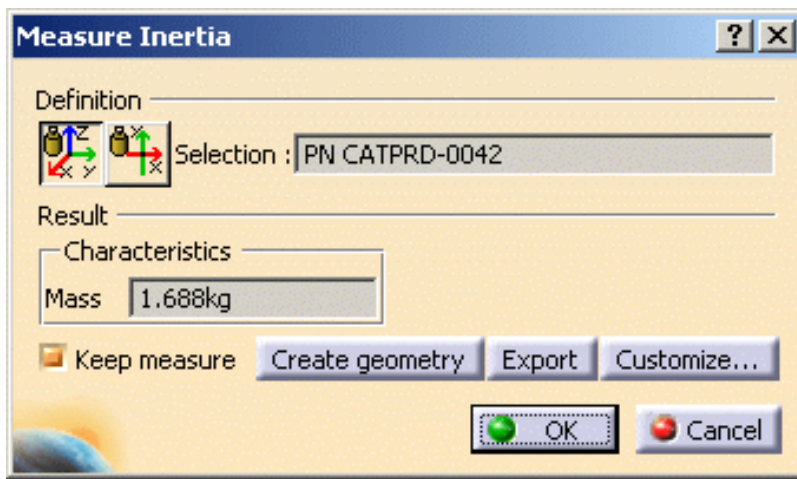
- [Calculating a Mass Value](#)
- [Creating a Formula](#)
- [Saving the Part](#)

## Calculating a Mass Value

1. Once you have created your part, click the **Measure Inertia** icon .  
The dialog box that appears lets you measure several properties of the geometrical element you want. The mass property is one these properties that can be calculated.
2. Ensure that the **Keep Measure** option in the dialog box is selected. This lets you keep measures as features in the specification tree. The mass measure kept as a feature can be used as a parameter.
3. From the **Measure Inertia** dialog that appears, click **Customize...** to indicate that you want to compute and display the **Mass** parameter in the **Measure Inertia** dialog box.




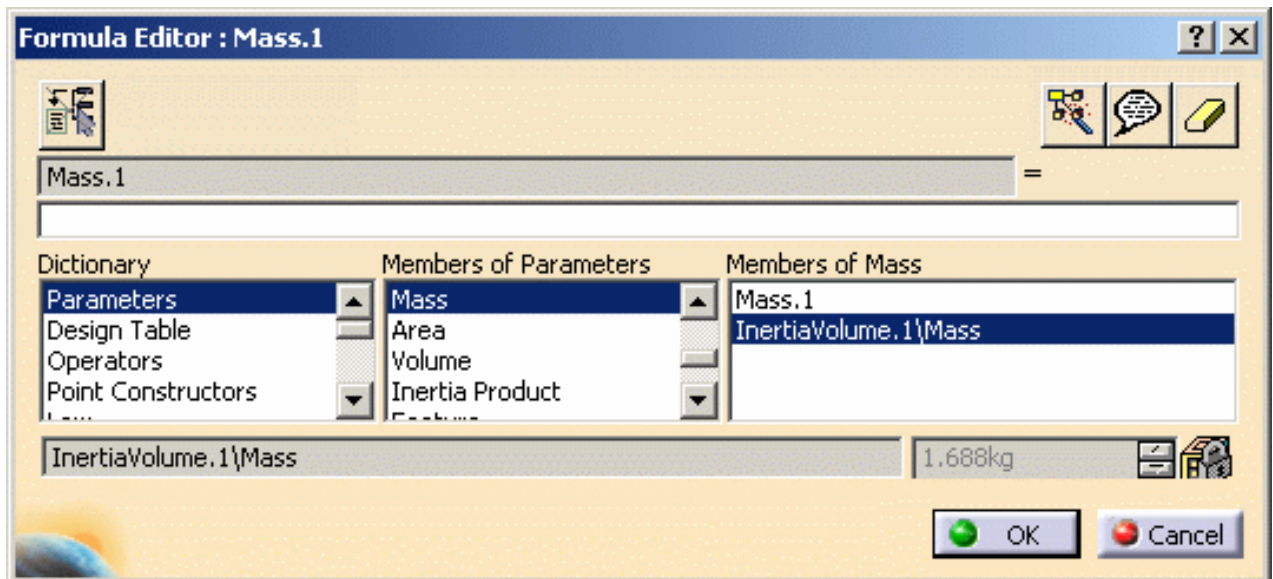
When done, the **Mass** value is displayed in the dialog box.



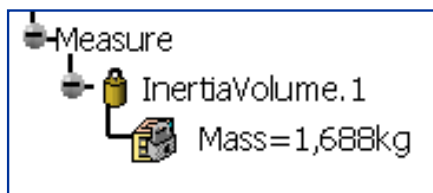
## Creating a Formula

Using the measure you have just created, you need to define a formula parameter that you will rename as Mass.

4. Click the **Formula**  icon.
5. From the **Formulas** dialog box that appears, click **Add Formula** and create the Mass.1 parameter in the **Formula Editor** window:



When done, as you checked the Keep measure option in the **Measure Inertia** dialog box, the mass is kept as a feature and your specification tree will look something like this:



## Saving the Part

As soon as you save the part in SmarTeam, the **Mass** attribute is displayed in the Profile Card

corresponding to this part.



# 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.



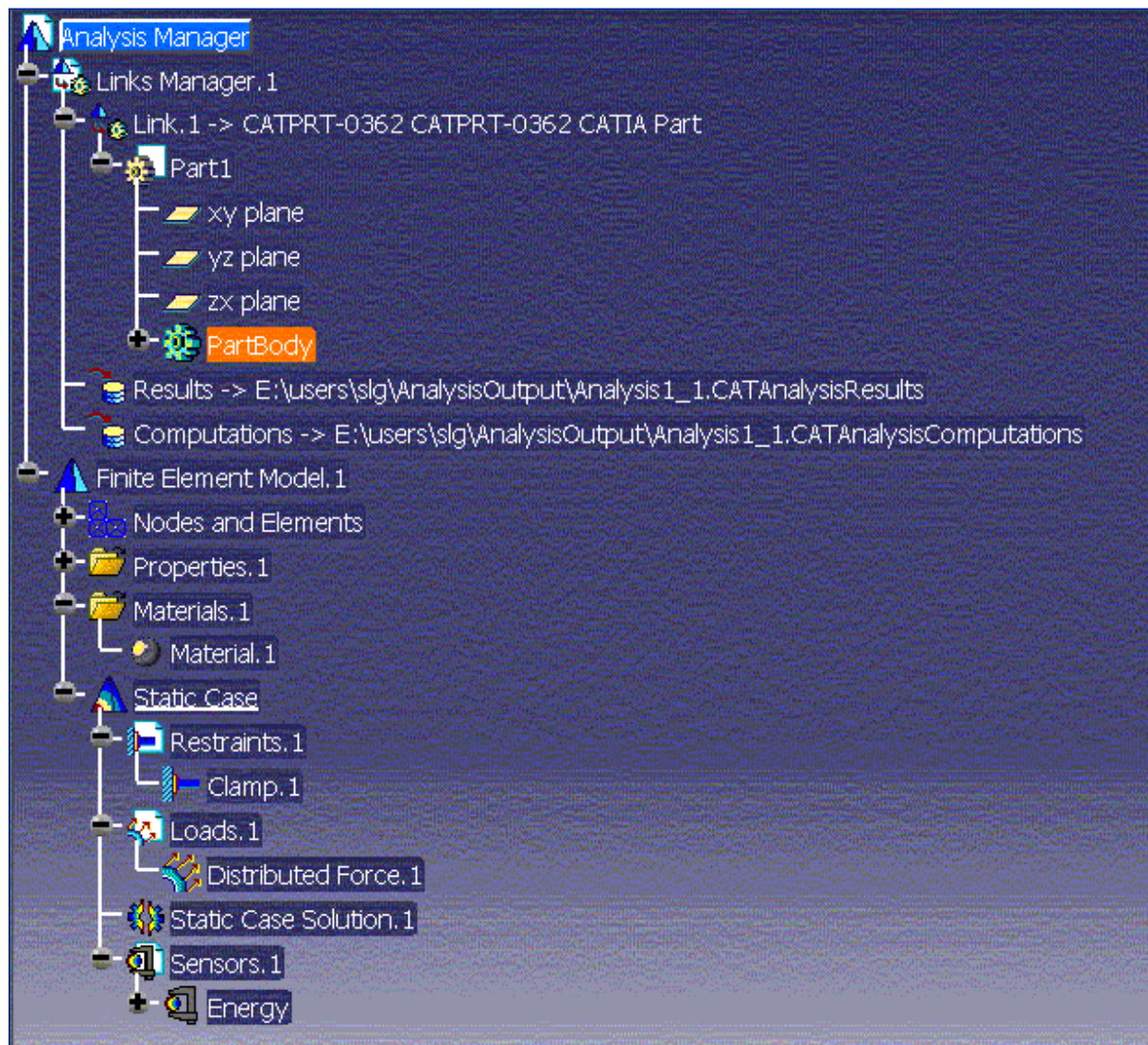
4. Launch the computation .  
Among other things, two files are generated and their paths are also displayed in the specification tree. You obtain a result similar as this:

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. Refer to [CATAnalysis Documents Management](#) for further information.
- have a name which will not be unique in the SmarTeam database.

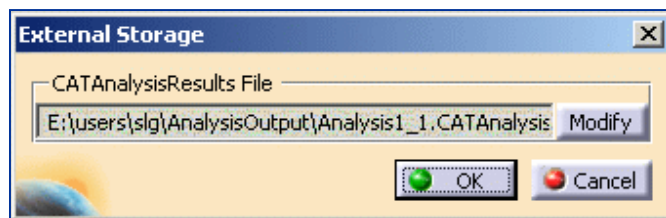


## Renaming the Files Generated

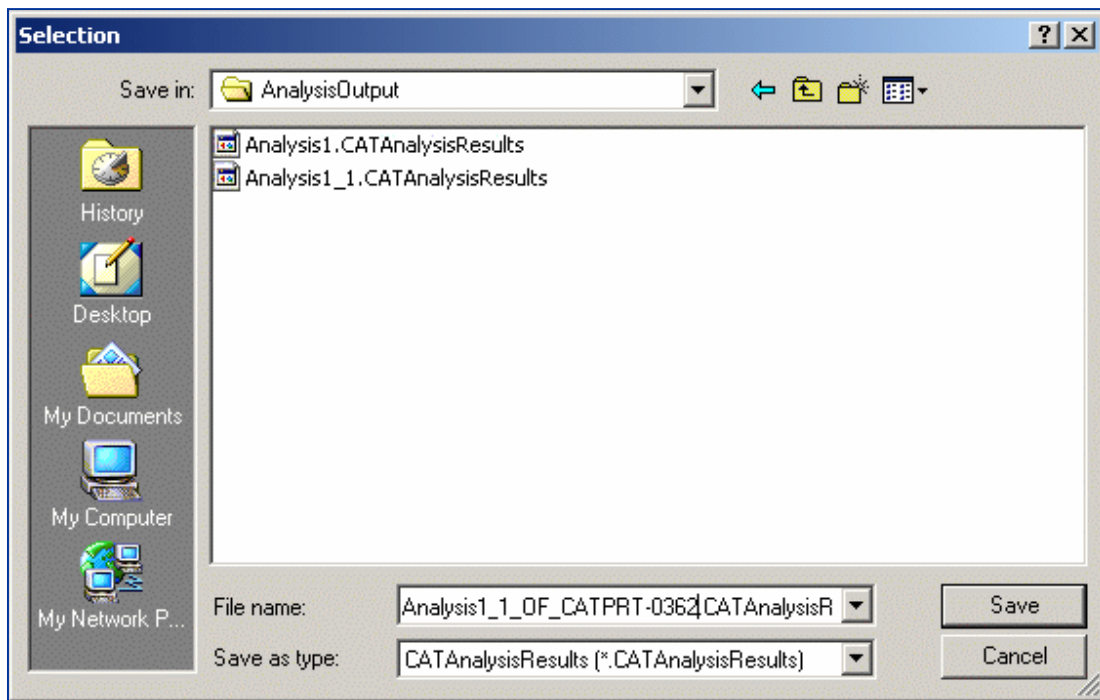
You may have noticed that the application provides names that cannot be easily reused in the SmarTeam database.

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.




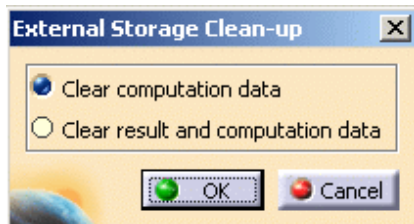
When done, the specification tree reflects the modifications. The name you have just modified are displayed.



## Saving the CATAnalysis Document in SmarTeam

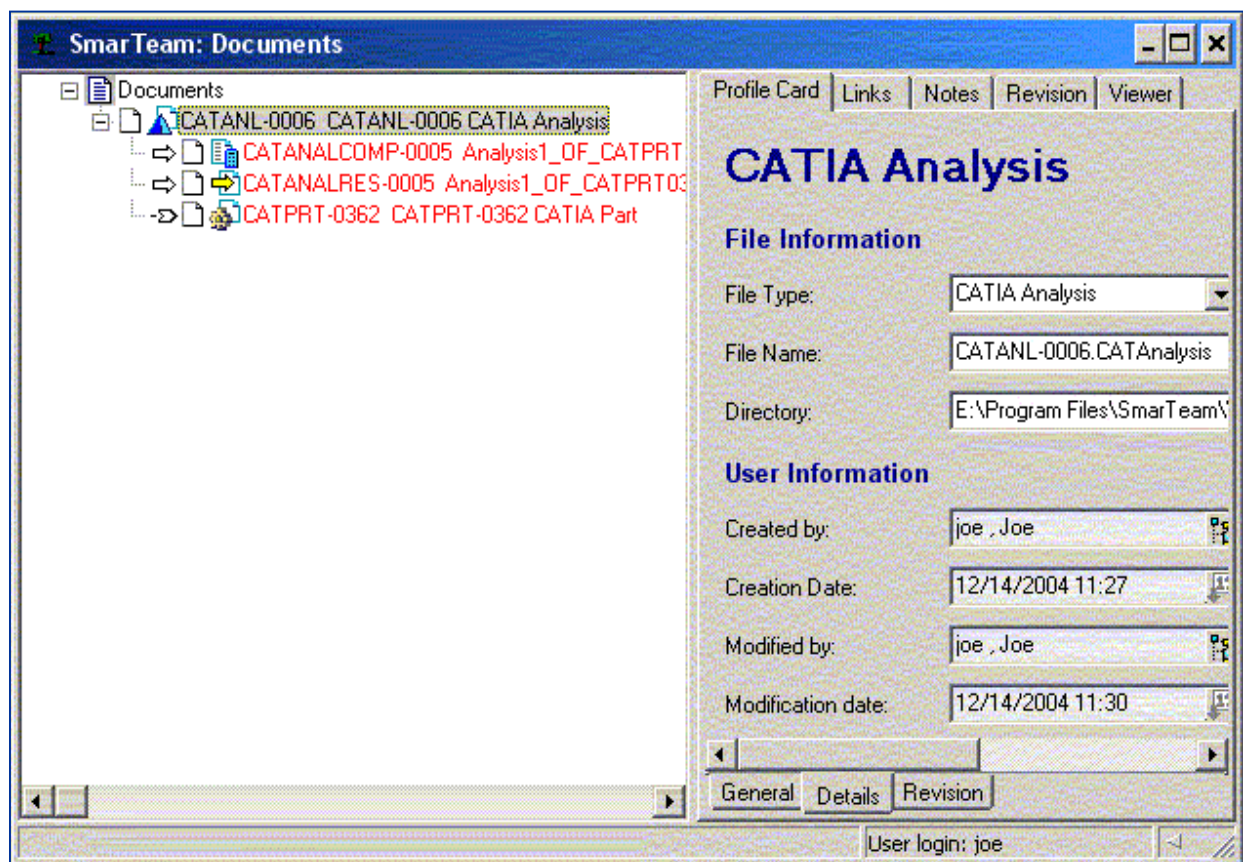


7. Prior to saving CATAnalysis documents in SmarTeam, if you wish not to save the computation data in the vault, click  to clear this data.



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



10. 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.

**Check Out: CATIA Analysis**

OK Cancel Help Tree Views Actions Set Default

CATIA Analysis

- CATANL-0006 CATANL-0006 CATIA Analysis
  - CATANALRES-0005 Analysis1\_OF\_CATPRT
  - CATANALCOMP-0005 Analysis1\_OF\_CATPP
  - CATPRT-0362 CATPRT-0362 CATIA Part

Profile Card Links Notes Revision Viewer

General Effectivity Options

### Check Out

Comment:

Current revision:

Next revision: a.0

File name: CATANL-0006.CATAnaly

Destination directory: E:\Program Files\Smarte

☐ Do not get the file from the vault

User login: joe



# 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 (check in/ check out) or shared workspace (if collaborative design is enabled)

## 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 consulting your system/network administrator and a SmarTeam consultant regarding to this issue. It has been proven that correct setup can significantly improve performance.

## Recommendations

- It is highly recommended to perform a backup of the users 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 from the performance, as well as the usability point of view, due to the redundant revisions that are exposed to team members.



# Support of DMU Scenarios

This section shows you how the application supports DMU Review and DMU batch processes such as CATDMUUtility or CATDMUBuilder.

# Support of DMU Review Process

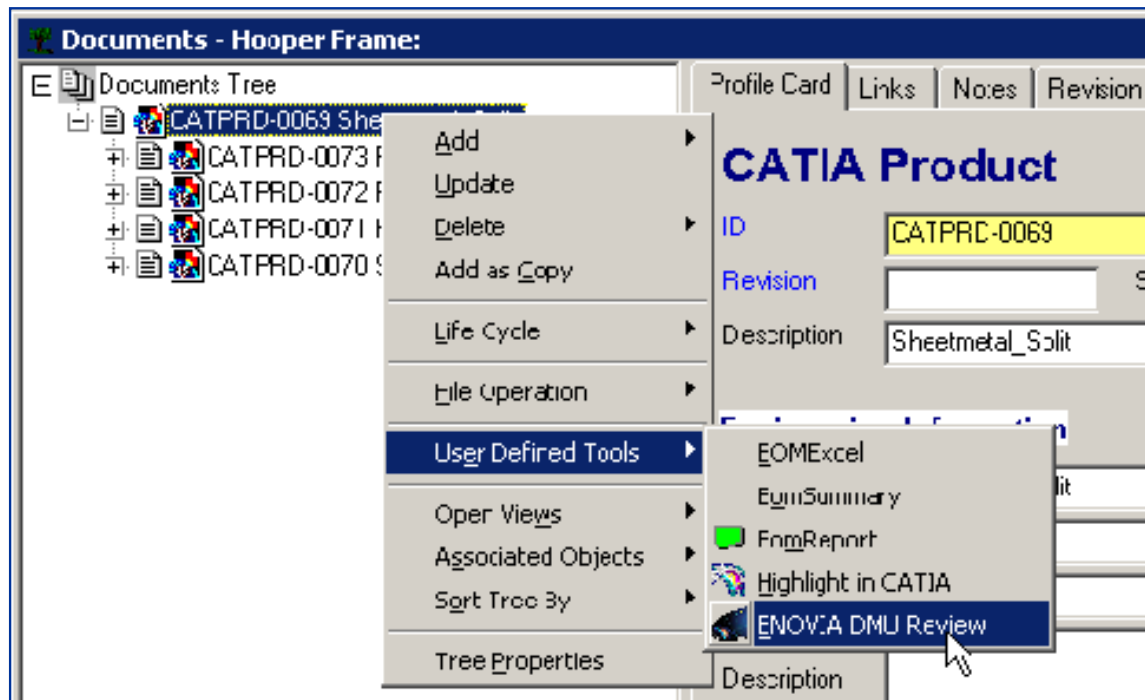
The following scenario shows you how SmarTeam manages documents stored in the database and reviewed in DMU sessions. It is divided into three parts:

- **Step 1:** in a DMU session, a product manager performs a first review of a CATProduct document stored in SmarTeam.
- **Step 2:** later on, a designer performs his/her own review and design changes to the CATProduct document.
- **Step 3:** the product manager performs another review of the CATProduct document.

## Step 1

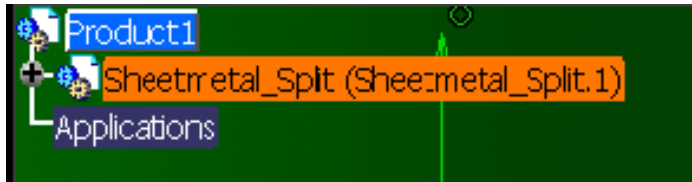
As product manager, you perform a first review of a CATProduct document in a DMU session. The document to be reviewed is stored in SmarTeam.

1. In **SmarTeam Documents Tree**, right-click the product you want to review and select **User Defined Tools > ENOVIA DMU Review**. For the purpose of our scenario, select CATPRD-0069 Sheetmetal\_Split.

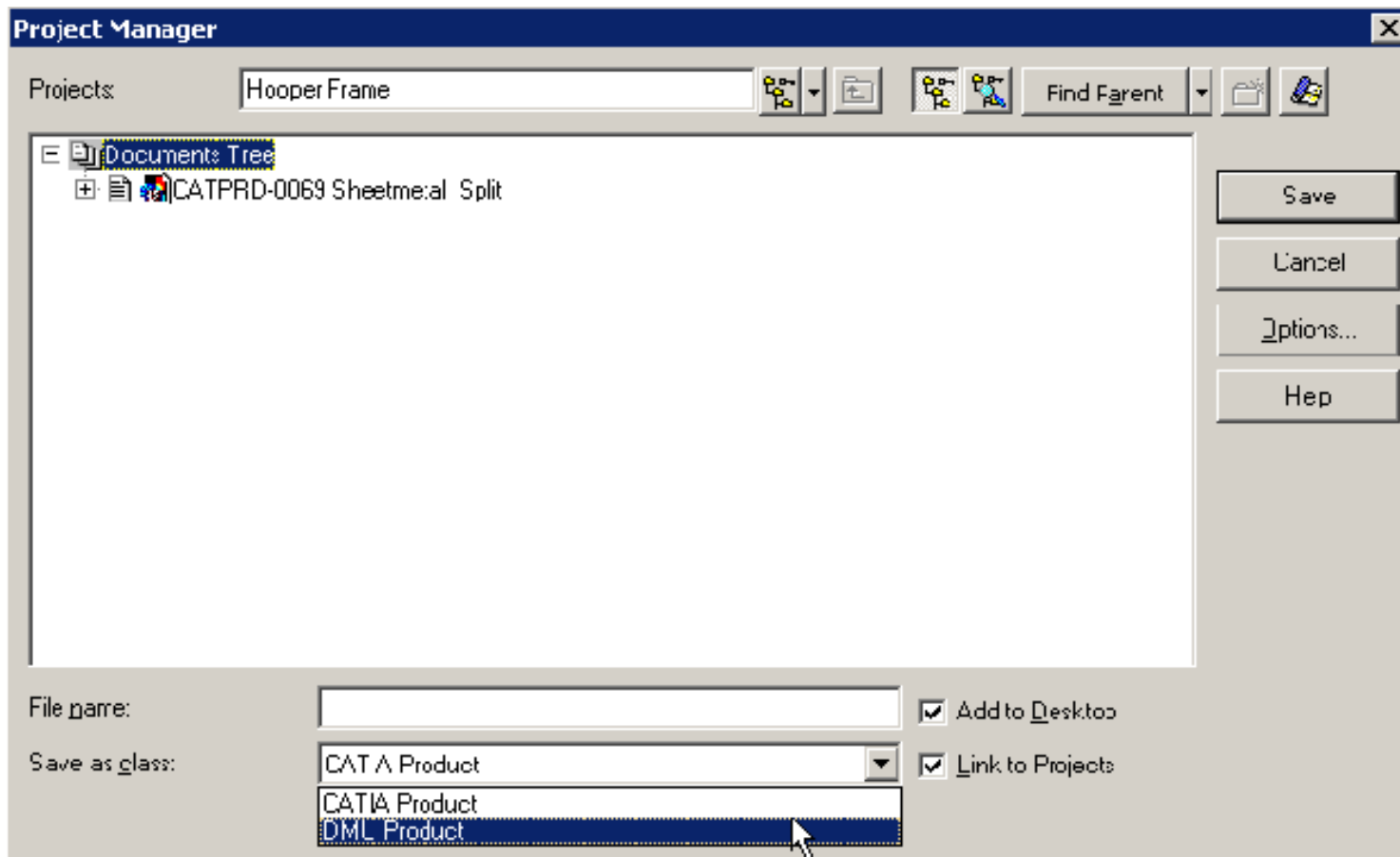


A new product opens in a DMU session. The **ENOVIA DMU Review** command has both created a new product and

inserted related parts or products as if you had used **New** and **Insert Component** from SmarTeam. You can note that the icons in the specification tree indicate that the DMU session contains new documents, that is documents that are not saved in SmarTeam.



2. In the DMU session, perform the review of your choice and make the possible necessary changes (for example, create annotations, modify graphic properties, etc.).
3. When done, select **SmarTeam** > **Save** to save the new review document in SmarTeam.
4. In the **Project Manager** window that appears, select **DMU Product** as the class in which you want to save the document.



5. Click **Save** when done.

6. Enter useful information in the related profile card that appears. For the purpose of our scenario, enter **Review Hooper** in the **Description** field.

The 'Profile Card' dialog box is shown with the title bar 'Profile Card'. It contains a 'Select class:' dropdown menu set to 'DMU Product'. Below this, the 'DMU Product' section is displayed. It includes an 'ID' field with the value 'DMUPRD-0007', a 'Revision' field, a 'State' dropdown menu, and a 'Description' field containing the text 'Review Hooper'. On the right side of the dialog, there are three buttons: 'OK', 'Cancel', and 'Help'. A mouse cursor is pointing at the 'OK' button.

7. Click **OK** to confirm the operation.  
When done, the **SmarTeam Documents** window shows that the review has been stored as DMUPRD-007 Review Hooper and that it is linked to the CATPRD-0069 Sheetmetal\_Split document:

The 'SmarTeam: Documents' window is shown with the title bar 'SmarTeam: Documents'. It features a tree view on the left under the 'Documents' folder, showing 'DMUPRD-0007 Review Hooper' and 'CATPRD-0069 Sheetmetal\_Split'. On the right, the 'Profile Card' tab is active, displaying the 'DMU Product' section. It includes an 'ID' field with the value 'DMUPRD-0007', a 'Revision' field, a 'State' dropdown menu set to 'New', and a 'Description' field containing the text 'Review Hooper'.

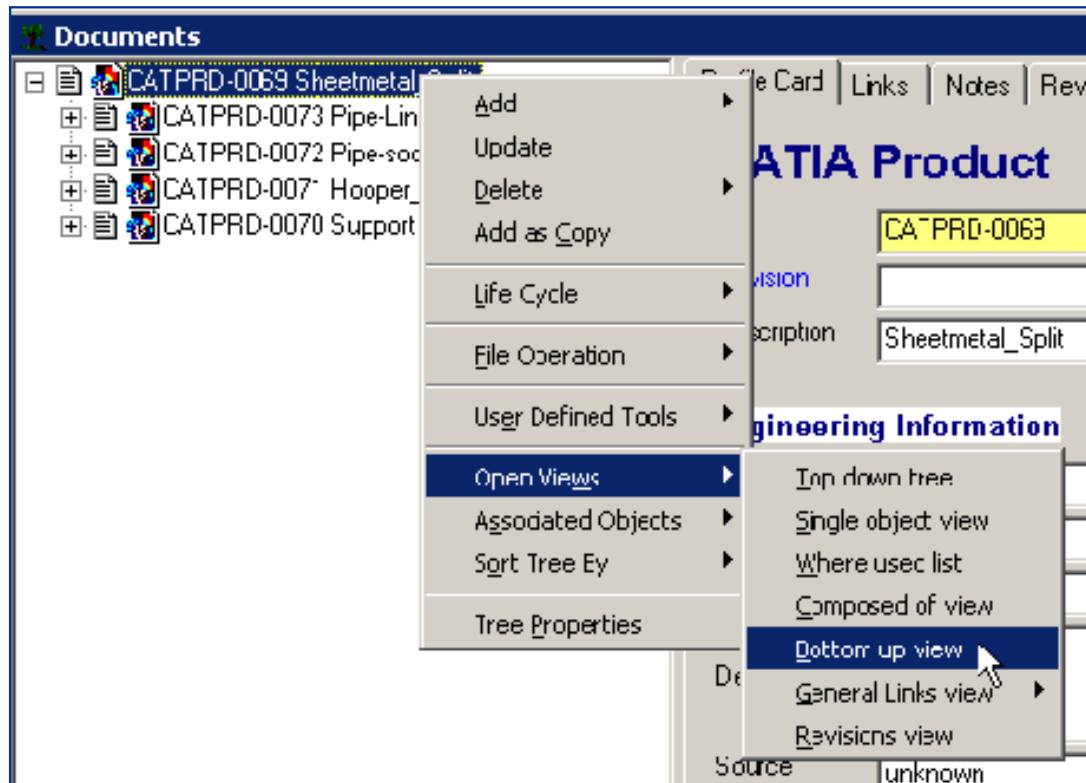
8. Check In the document.

## Step 2

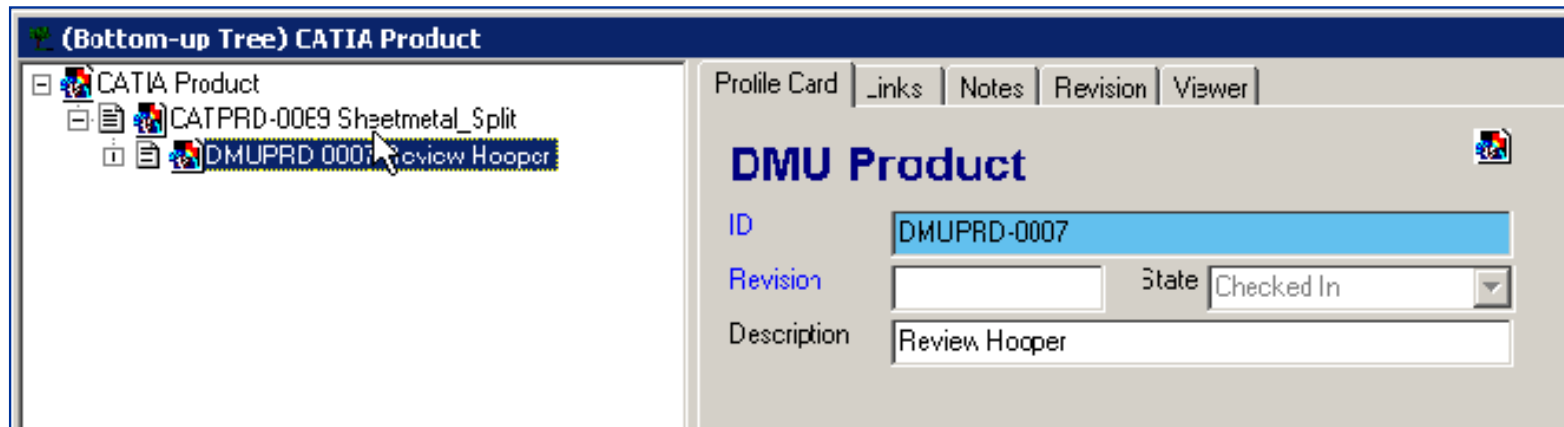
As a designer, you open the DMU document as you want to review it.

1. In the **Documents** window, right-click the CATPRD-0069 Sheetmetal\_Split document and select **Open Views > Bottom**

up view.



2. This view shows that the review is linked to the CATPRD-0069 Sheet document.



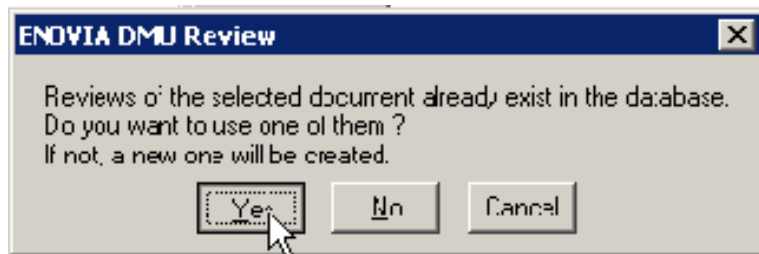
3. Right-click the DMU document, DMUPRD-007 Review Hooper, and select **Open**.  
The application asks you if you want to check out the document.
4. You can later on check the document out and modify the design. When done, **Save** and **Check in** the document.

## Step 3

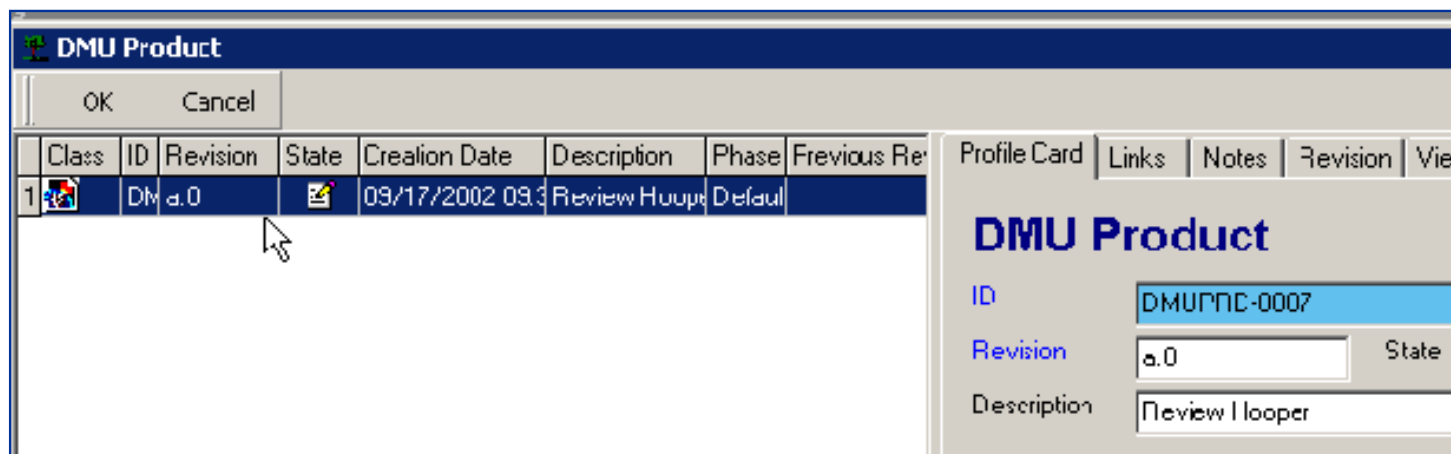
As product manager, for the second time you perform a review of the CATPRD-0069 Sheetmetal\_Split document.

1. Once connected to SmarTeam, in **SmarTeam Documents Tree** right-click CATPRD-0069 Sheetmetal\_Split and select **User Defined Tools > ENOVIA DMU Review**.

This time, a warning message is displayed, informing you that different reviews of the selected document exist in SmarTeam. You need to choose between using one of these review documents (clicking **Yes**) or creating a new one (clicking **No**).



2. Click **Yes** to open an existing review.  
The **DMU Product** window is displayed to let you select the review of your choice.



3. The only review available is selected: just click **OK** to confirm the operation.  
The document opens in a DMU session.
4. Review the document as you want and when done, **Save** and **Check in** the review document.



# Support of DMU Batches: Creating the TEAM PDM URL Command for End Users



The **TEAM PDM URL** command displays the CATIA SmarTeam identifier for a given SmarTeam document. Once this identifier has been retrieved, it is then possible to use a DMU batch program (CATDMUUtility, CATDMUBuilder etc.) to process a document stored in SmarTeam.

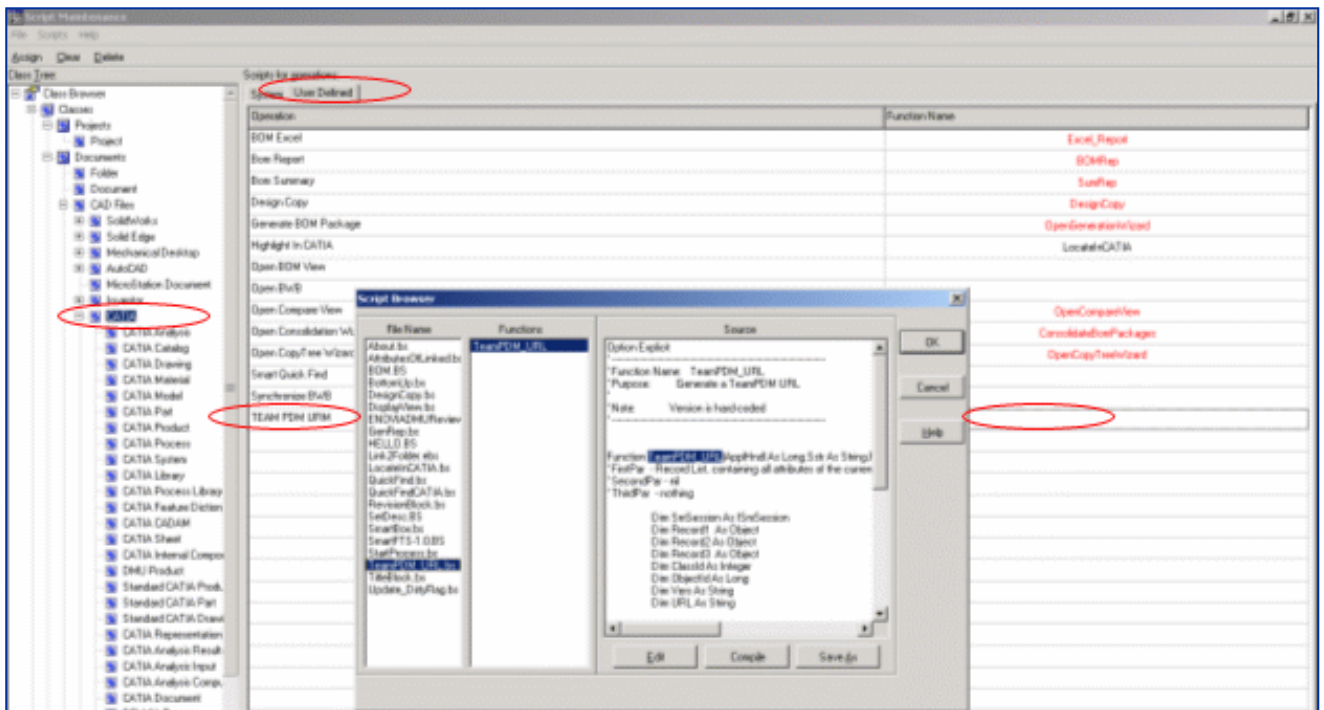
This task describes how system administrators must add this command to make it available to end users. To know how to use it, see [Support of DMU Batches: Using the TEAM PDM URL Command](#).

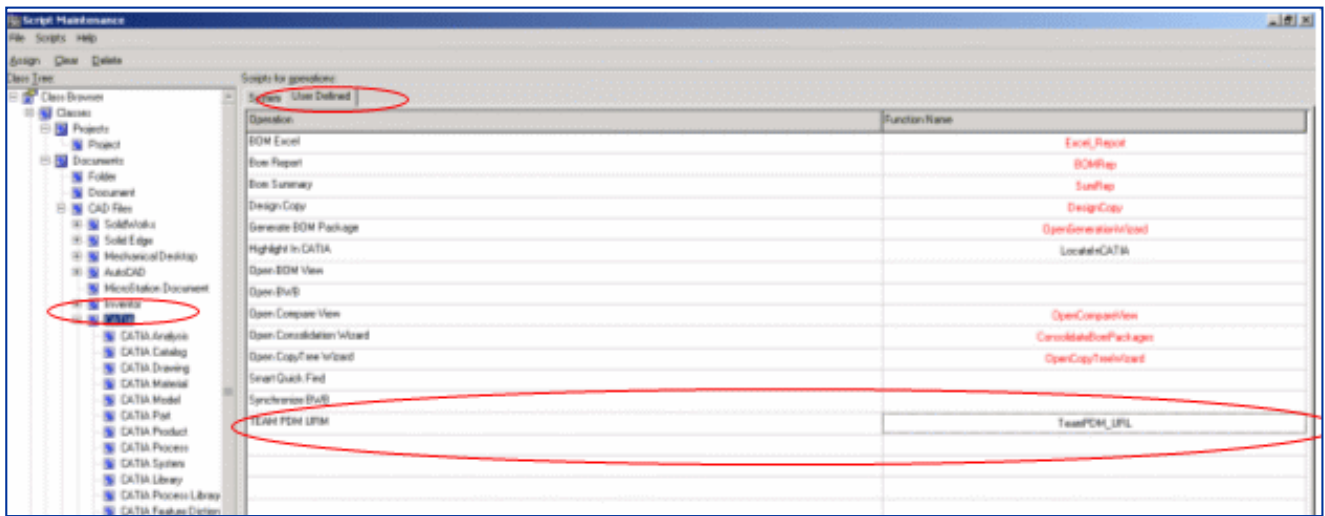


## Adding the TEAM PDM URL Command

You need to use the **SmartBasic Script Maintenance** application to add the **TEAM PDM URL** command.

1. Select **Start > Programs > SmarTeam > Administrative Tools > SmartBasic Script Maintenance**.
2. Select **CATIA Class** in Class Tree.
3. Select the **User Defined** tab and add the **TEAM PDM URL** command.



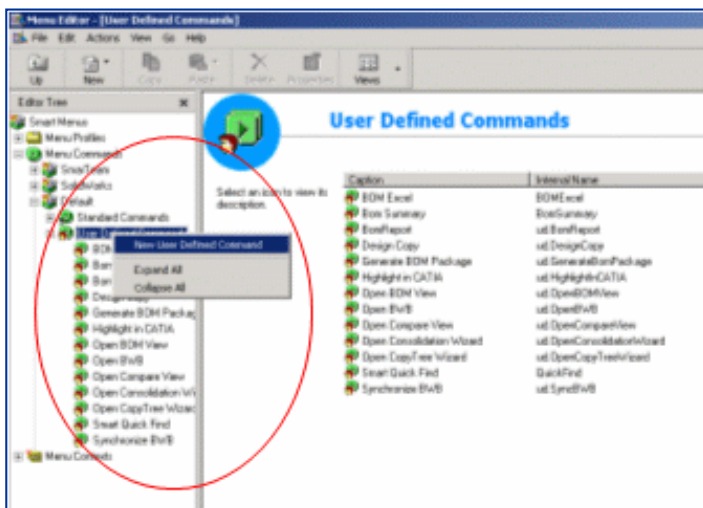


4. Save the changes.
5. Exit SmartBasic Script Maintenance.

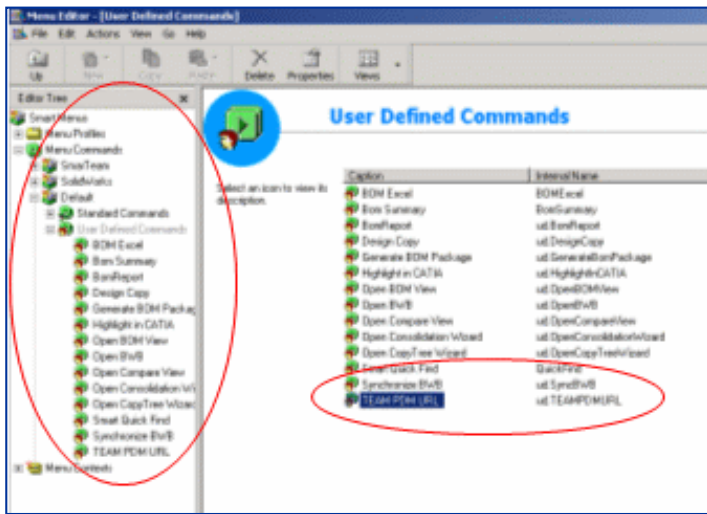
## Making the New Command Accessible to End Users

You need to use the Menu Editor application to make the TEAM PDM URL command accessible to end users.

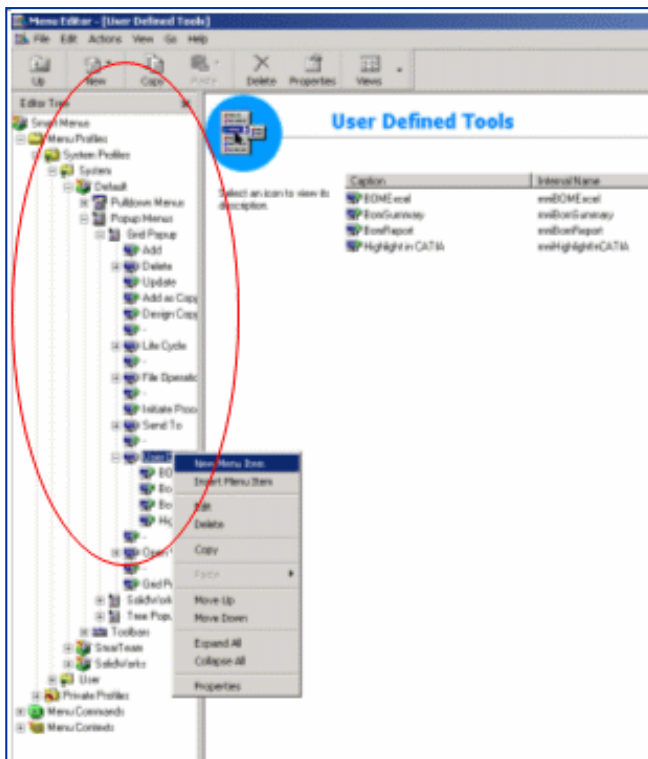
1. Select Start > Programs > SmartTeam > Administrative Tools > Menu Editor.
2. Use Start Menus > Menu Commands > Default > User Defined Commands and New User Defined Commands to create TEAM PDM URL.



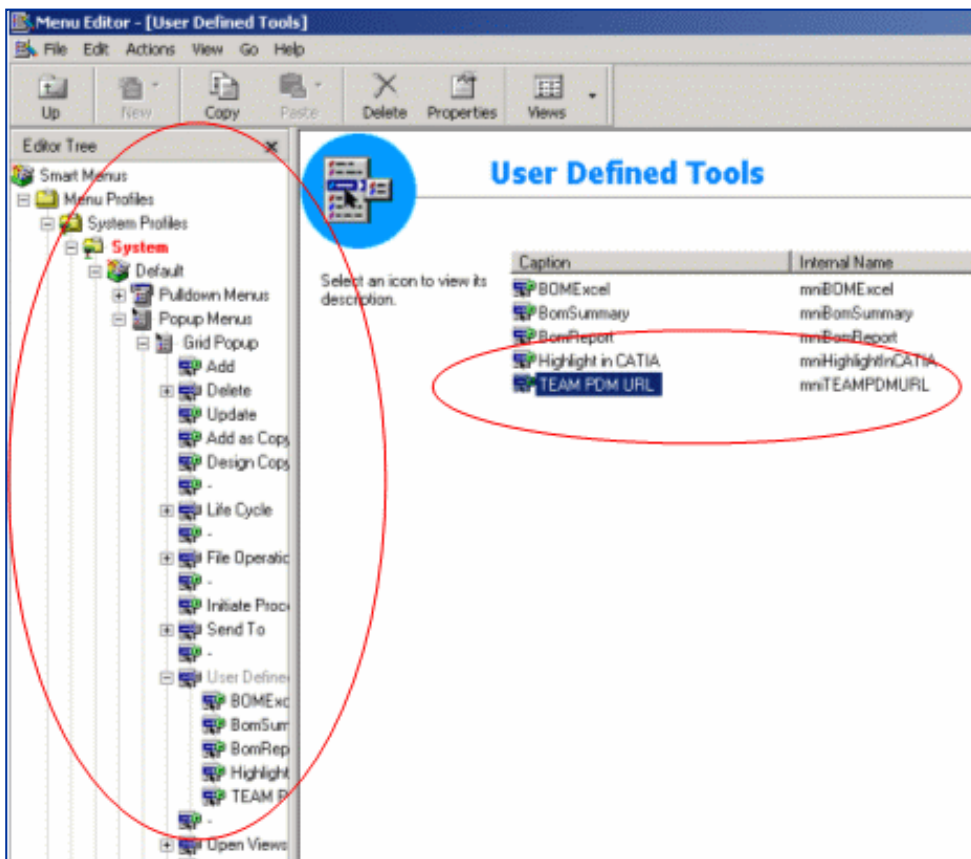
3. Select TEAM PDM URL and click OK.



4. To add the TEAM PDM URL command to the default profiles, get down to User Defined Tools in the Editor Tree. User Defined Tools is located in Start Menus/Menu Profiles/System Profiles/System/Default/Pop-Up Menus/Grid Popup.
5. Right-click User Defined Tools and select New Menu Item.



6. Save the changes.



7. Exit Menu Editor.



# Support of DMU Batches: Using the TEAM PDM URL Command



DMU batch processes such as CATDMUUtility, CATDMUBuilder etc., support documents stored in SmarTeam.

The CATDMUUtility is a batch process enabling the generation of cgr, 3dmap, hcg, wrl and NCGM formats from a CATIA file stored in a SmarTeam database. The CATDMUBuilder creates CATProducts corresponding to product stored in SmarTeam and stores them on dedicated local directories

To run a DMU batch process, you need an input parameter that corresponds to the location of a SmarTeam document. To retrieve this TeamPDM URL, you first need to use the TEAM PDM URL command.

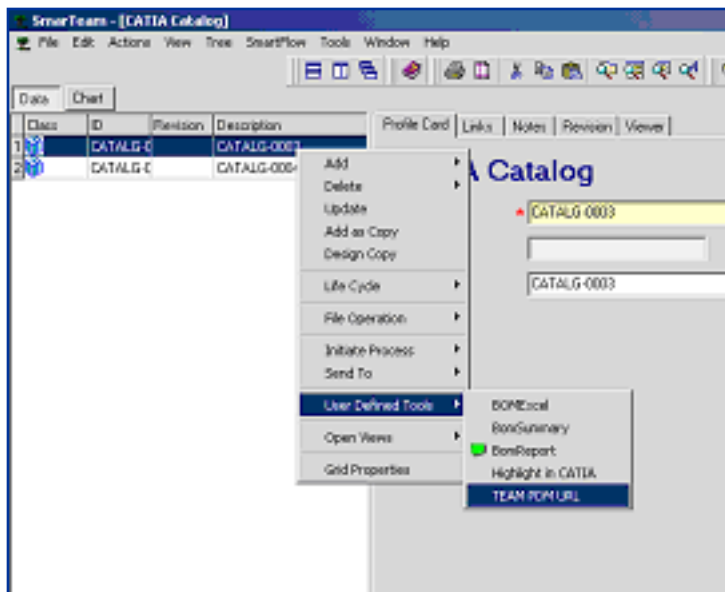
This task assumes that the system administrator has added the TEAM PDM URL command via SmarTeam Script Maintenance. For details, see [Support of DMU Batches \(Creating the TEAM PDM URL Command\)](#).



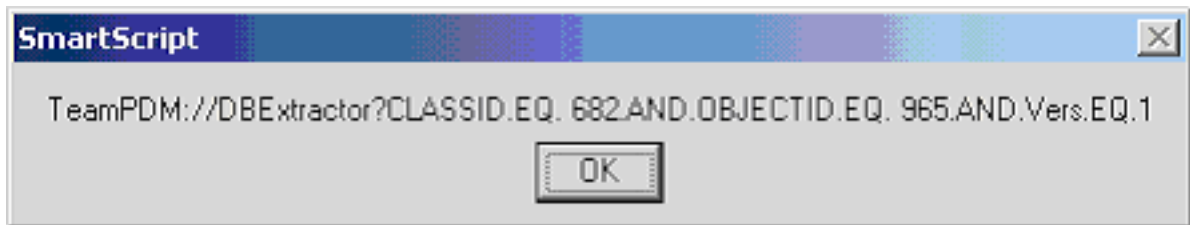
## Retrieving the Document's Identifier

The first step of the process consists of retrieving the identifier of the input document that needs to be processed with a DMU batch.

1. Launch SmarTeam.
2. Search for the CATIA document you want to convert with the DMU batch.
3. Right-click the document and select **User Defined Tools > TEAM PDM URL**.



4. Retrieve the information displayed.



The URL is always as follows:

TeamPDM://DBExtractor?CLASSID.EQ.xxx.AND.OBJECTID.EQ.xxx.AND.Vers.EQ.1

The numbers corresponding to the document "class id" and "object id" are the only values you can modify.

The output document is located on the File system and can be integrated in SmarTeam using the CATIA CAI Automation APIs.

## Launching the DMU Batch

Once the identifier of the input document has been retrieved, you can use the DMU batch you want.

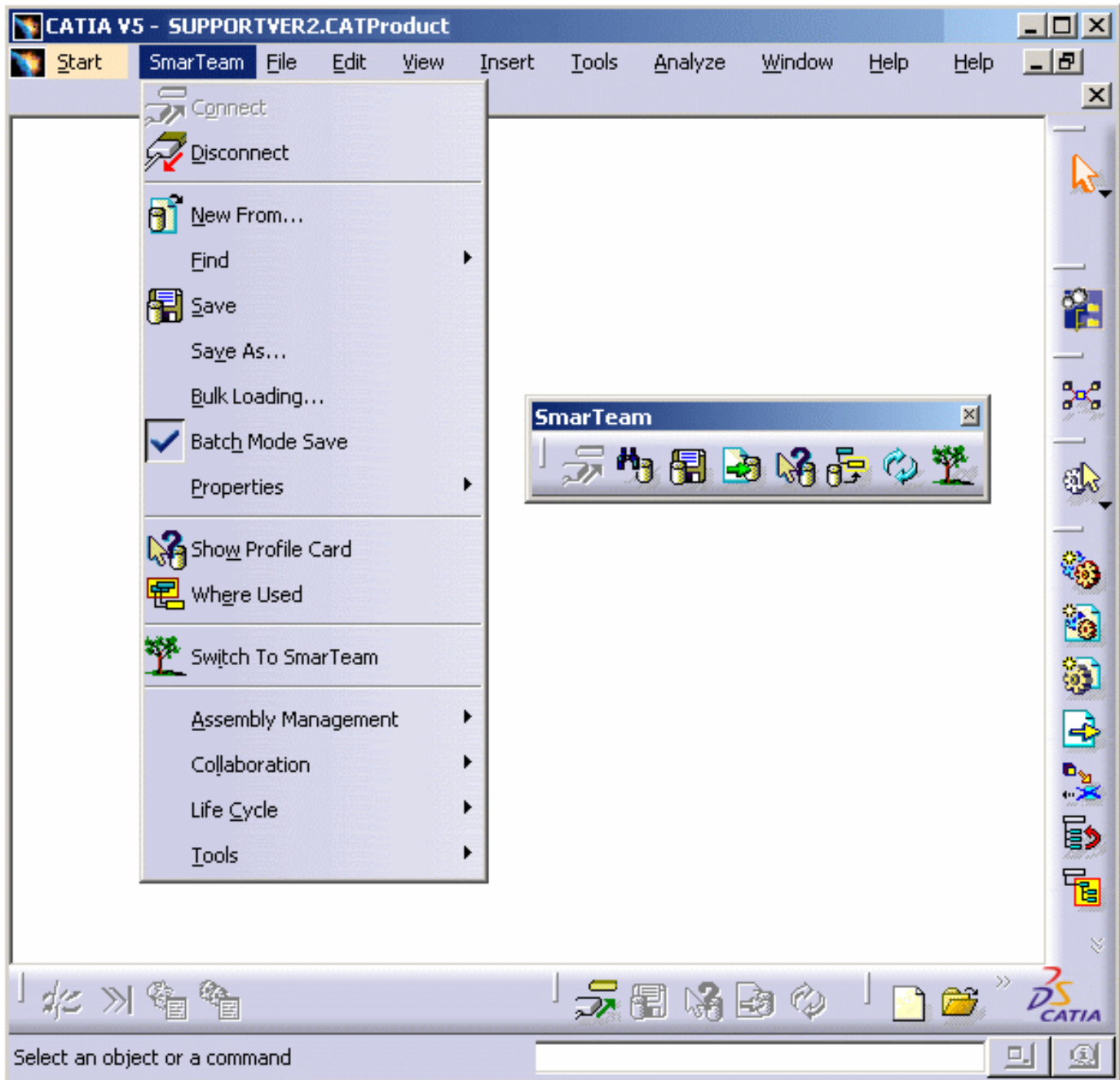
1. Launch the DMU batch using CATStart.  
For more information on how to launch the DMU batch, see the DMU Navigator Version 5 User's Guide.
2. Use **TeamPDMURL** as the input parameter along with the **-id** option and **-db TeamPDM**. For example:

```
CATDMUUtility -id TeamPDM://DBExtractor?CLASSID.EQ.682.AND.OBJECTID.EQ.965.AND.Vers.EQ.1 -cache -db TeamPDM -server none -user joe
```



# Workbench Description

The SmarTeam - CATIA Integration user interface looks something 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 sections below.

# SmarTeam Menu

Start ENOVIA V5 VPM SmarTeam File Edit View Insert Tools Windows Analyze Help

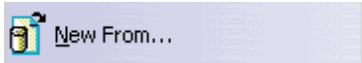
## SmarTeam



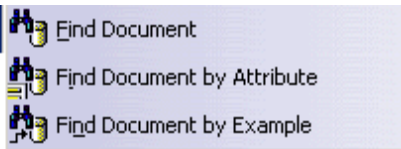
See [Connecting to the SmarTeam Database](#).



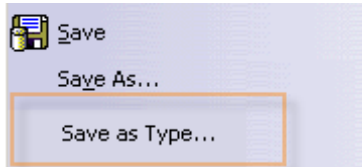
In a CATIA session, disconnects the SmarTeam database.



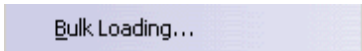
Creates, in the CATIA session, a new document based on an existing SmarTeam database document.  
For an example of use, see [Creating a Drawing Document from a Template](#).



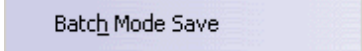
- **Find Document**  
See [Finding](#).



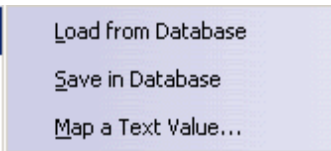
See [Saving a Part](#), [Saving an Assembly](#) or [Saving a Drawing](#).



See [Importing CATIA Data Inside SmarTeam](#).



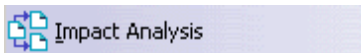
See [Saving a Part](#) or [Saving an Assembly](#).



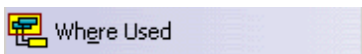
- **Load from Database**  
Forces the update of all the mapped properties of the document in the CATIA session.
- **Save in Database**  
Forces the update of all the mapped attributes of the document in the SmarTeam database.
- **Map a Text Value...**  
For an example of use, see [Displaying a CATIA Drawing SmarTeam Attribute in a Title Block](#).



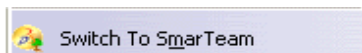
Provides the database view of the document currently displayed in a CATIA session.  
See [Showing Profile Cards](#).



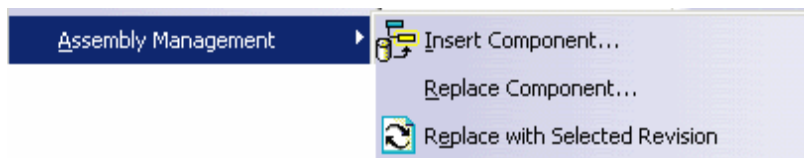
See [Analyzing the Impact of a Change](#)



See [Finding Out Where a Document Is Used](#).



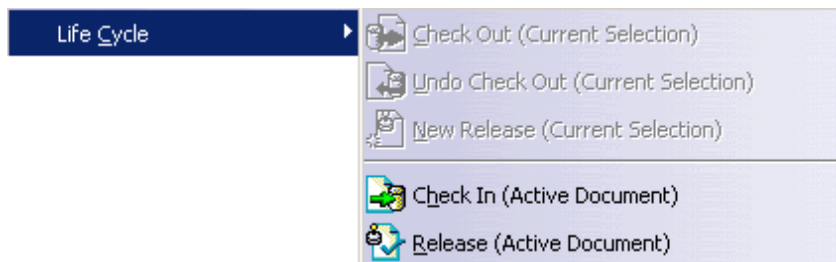
Switches from the CATIA session to the SmarTeam application.



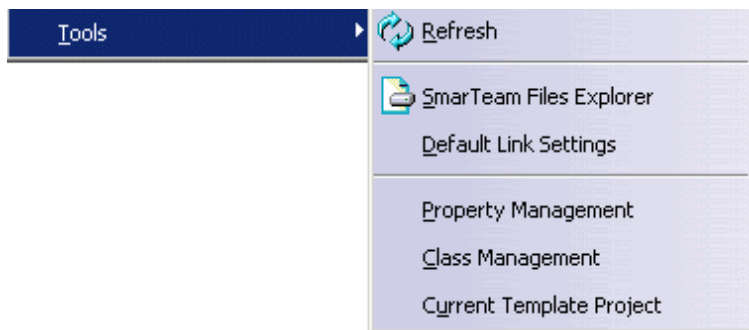
- **Insert Component...**  
Inserts a Part or Product stored in the SmarTeam database into an Assembly. For an example of use, see [Building an Assembly](#).
- **Replace Component...**  
Replaces an existing Part or Product or a component of the current Product. For an example of use, see [Modifying a Checked In Assembly](#)
- **Replace with Selected Revision**  
Replaces the current revision of a document with another. For more information, see [Managing Assemblies](#).



See Using [Global Refresh](#)



See [Lifecycle Menu Commands](#).



- **Refresh**  
Updates the Product Structure and File Desk icons with regard to SmarTeam information. See [Tree Refresh](#).
- **SmarTeam Files Explorer**  
Lists all files copied to the *view* and *work* directories. For more information, see the SmarTeam documentation.
- **Default Link Settings**  
Allows you to set the default project that is proposed when storing a new document.
- **Property Management**  
Manages mapping group types and associated information.
- **Class Management**  
Declares all classes that can be used when saving a document of the corresponding file type.
- **Current Template Project**  
Sets the template project as current. Once defined, selecting **File>New From...** displays the content of this project, allowing you to create a new

document from one of those contained in this project.

**Note:** if no templates project is defined, **File>New From...** launches the standard SmarTeam open dialog allowing you to select any document from the database.

To use this option, the user must have permissions for the Project class.

The SmarTeam administrator may customize the interface, removing the SmarTeam menu and allowing users to work with toolbars only. In addition, it is possible to customize the toolbars so that only those functions relevant for the current user are displayed. For more information, refer to [Toolbar Customization](#).

# SmarTeam Toolbar



**Connect**

See [Connecting to the Database](#)



**Find Document**

See [Finding](#)



**SmarTeam Save**

See [Saving a Part](#), [Saving an Assembly](#) or [Saving a Drawing](#)



**Check In (Active Document)**

See [Checking In a Part](#), [Checking In an Assembly](#)



**Show Profile Card**

See [Showing Profile Cards](#)



**Insert**

See [Building an Assembly](#)



**Refresh**

See [Tree Refresh Command](#)



**Switch to SmarTeam**

Switches from the CATIA session to the SmarTeam application



**Impact Analysis**

[Analyzing the Impact of a Change](#)

# SmarTeam Collaboration Toolbar



See [Using Global Refresh](#)

# Glossary



## Symbols

### 3D XML

EXtensible Markup Language. A universal, lightweight, XML-based format that enables users to capture and share live, accurate 3D data quickly and easily.

3D XML compresses highly complex data, provides rapid file transmission and shorter load times while maintaining the exact geometry of the files exchanged.

## B



### bend table

A table defining standards for sheet metal parameters in CATIA Generative Sheet Metal Design workbench. They are used to reflect what can be manufactured in a given company.

## C



### CGM

Computer Graphics Metafile. An ANSI/ISO standardized platform-independent format used for the interchange of vector and bitmap data.

### CGR

CATIA Graphical Representation. A data format containing a graphical representation of the geometry only, which is available with the Visualization mode (as opposed to the exact geometry, which is available with the Design mode). With CGR, only the external appearance of the component is used and displayed; the geometry is not available. The corresponding .cgr file, if it exists, is inserted from the cache system.

### 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.

### class

A set of objects that share common structure and common attributes. Every engineering entity is an object that can be classified.

## D



<b>default</b>	An operation or value that the system or application assumes, unless a user makes an explicit choice.
<b>desk</b>	A CATIA tool that lets you view the relationships between different documents and obtain information about their properties.
<b>design table</b>	A table containing values to be potentially applied to a document to manage its parameter values. It can be created from the document parameters or from an external file. A design table is a feature. In the document specification tree, it is displayed as a relation that can be activated or deactivated. Like any feature, a design table can be manipulated from its contextual menu. The evaluation method is based on a table (Excel or .txt file).
<b>dialog box</b>	A window that gathers additional information from a user. A dialog box usually contains one or more controls, such as buttons, list boxes, combo boxes, and edit boxes, with which the user enters text, chooses options, or directs the action of the command.
<b>document</b>	A common unit of data (typically a file) used in user tasks and exchanged between users.

## G



**geometry area** Area of a document window in which application data are displayed and edited.

## M



**model** A CATIA Version 4 document.

## O



**object** An entity or component identifiable by a user that can be distinguished by its properties, operations, and relationships.

**obsolete** An object assigned as Obsolete can no longer be activated, released or modified.

## P



**part** A 3D entity obtained by combining different features.

## R



**release lifecycle operation** 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.

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