SMARTEAM CATIA Integration

User's Guide

Version 5 Release 16

Special Notices

SMARTEAM CATIA Integration

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, refer to the *Conventions* section.

Conventions

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

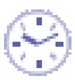
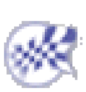
Graphic Conventions

The three categories of graphic conventions used are as follows:

- Graphic conventions structuring the tasks
- Graphic conventions indicating the configuration required
- Graphic conventions used in the table of contents

Graphic Conventions Structuring the Tasks

Graphic conventions structuring the tasks are denoted 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 Indicating the Configuration Required

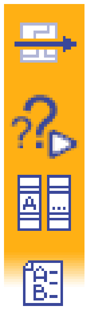
Graphic conventions indicating the configuration required are denoted as follows:

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 denoted as follows:

This icon...	**Gives access to...**
	Site Map
	Split View Mode
	What's New?
	Overview
	Getting Started
	Basic Tasks
	User Tasks or Advanced Tasks
	Interoperability
	Workbench Description
	Customizing
	Administration Tasks
	Reference



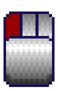
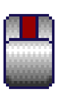
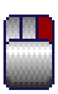
Text Conventions

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.

How to Use the Mouse

The use of the mouse differs according to the type of action you need to perform.

Use this mouse button...	Whenever you read...
	• Select (menus, commands, geometry in graphics area, ...) • Click (icons, dialog box buttons, tabs, selection of a 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)

What's New?

New Functionality

Analyzing the Impacts of a Change

The **Impact Analysis** functionality 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 help you choose the solution to implement with the least modification cost.

Enhanced Functionality

SMARTEAM Document Display

The display of SMARTEAM objects in Check Out, Check Out on the Fly windows and Not Saved warnings can be customized thru **Tree Properties**.

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:

Before Performing the Getting Started Scenario
Connecting to the SMARTEAM Database
Copying a Part Stored in the SMARTEAM Database
Editing the Copy in the Part Design Workbench
Storing the New CATPart Document in the SMARTEAM Vault
Modifying a Checked In Assembly
Releasing the Modified Assembly

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

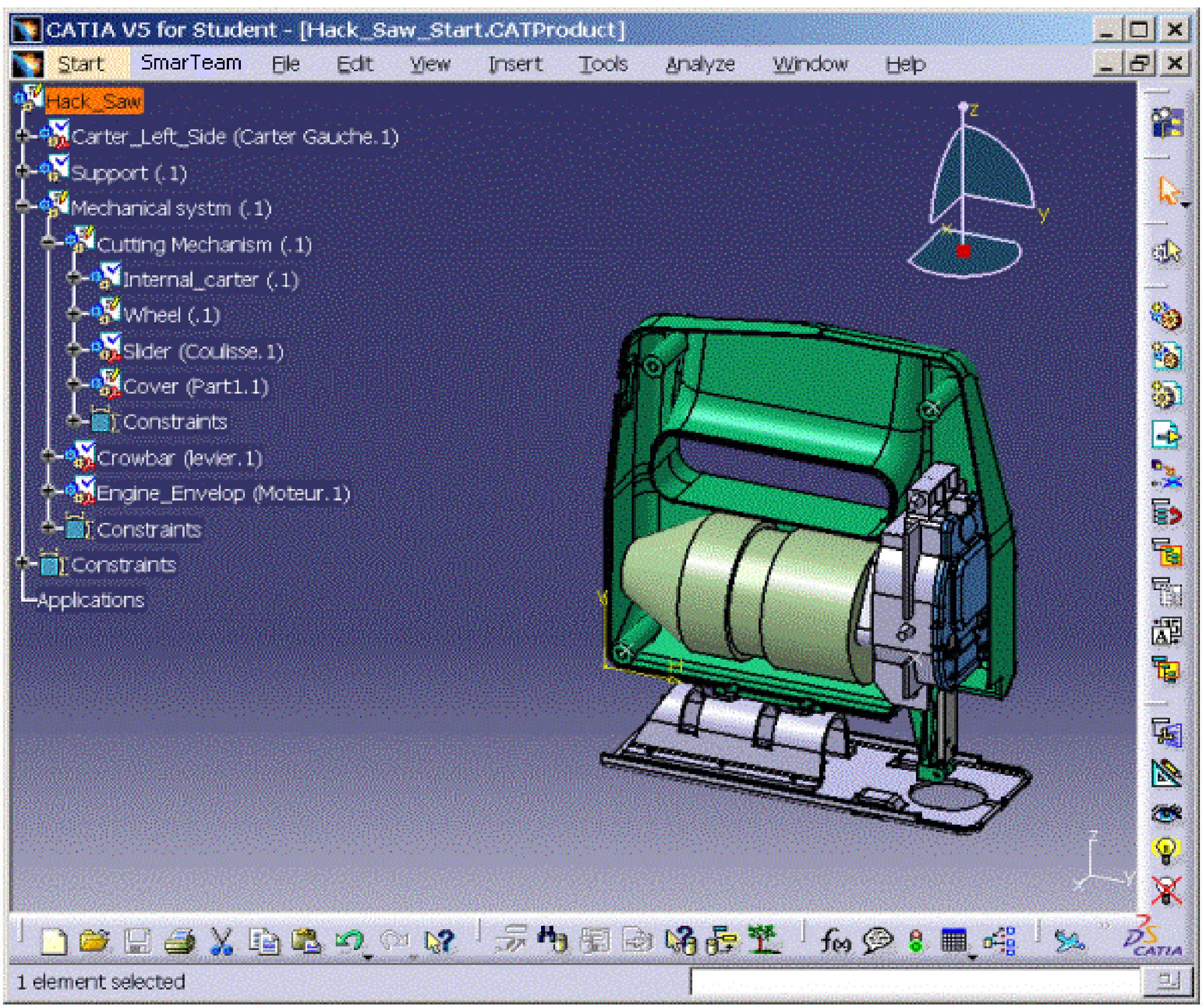
The Product Structure tree, the drafting workbenches and the Desk tree show the SMARTEAM status for each document. The status is displayed on the icons of the documents.

This section deals with the following information:

- Product Structure Tree
- Drafting Workbench
- Desk Tree
- Displayed Statuses

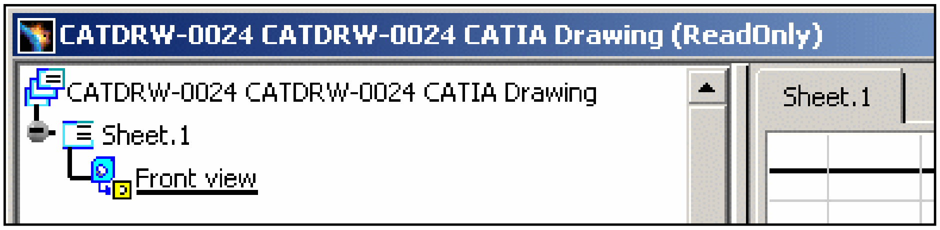
Product Structure Tree

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.



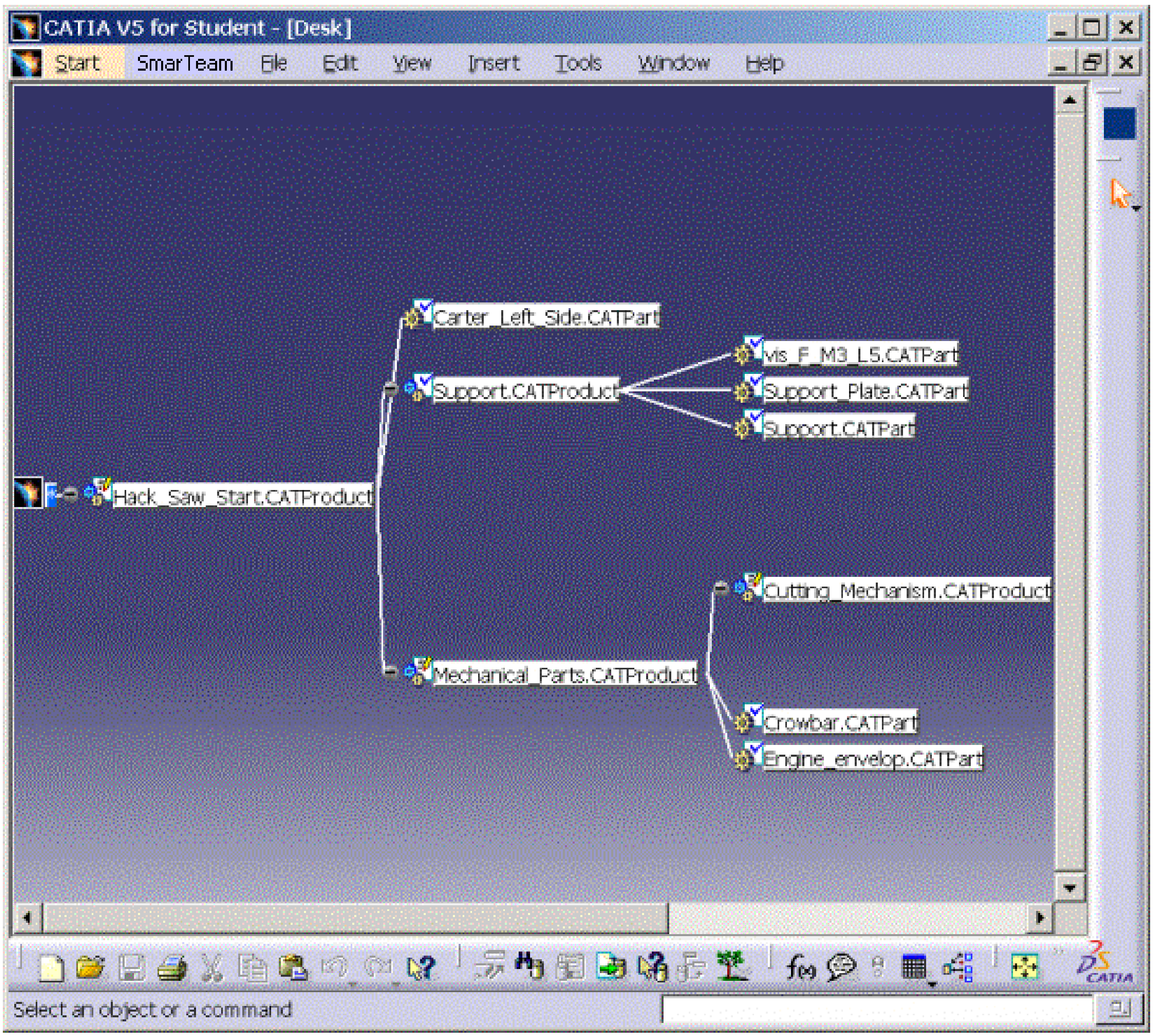
Drafting Workbench

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



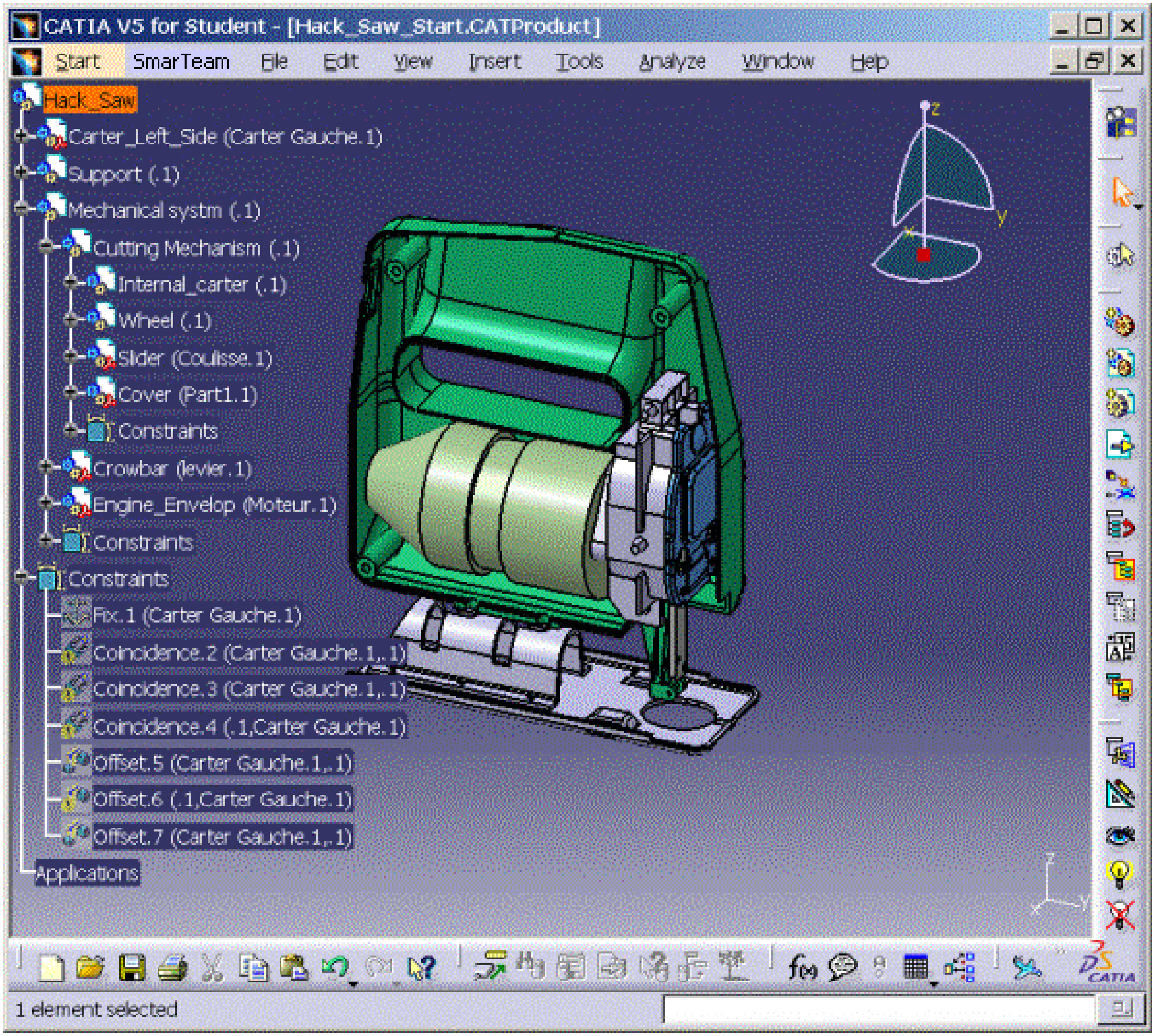
Desk Tree

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

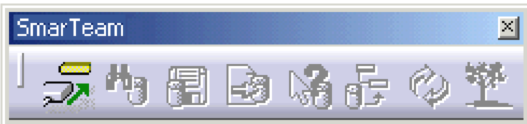
The statuses displayed by CATIA are as follows:

- Unknown
- New
- Checked In
- Checked In, Not Latest
- Checked In, Dirty
- Checked In, Not Latest, Dirty
- Checked Out
- Released
- Released, Not Latest
- Released, Dirty
- Released, Not Latest, Dirty
- Obsolete
- Obsolete, Dirty
- Read-only, Modified

Connecting to the SMARTEAM Database

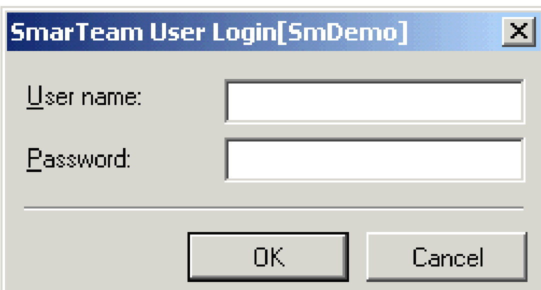
Your first task consists in connecting to the SMARTEAM Database.

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 the **Connect** icon .

   or select **SMARTEAM->Connect**.

   The SMARTEAM User Login window appears:

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.

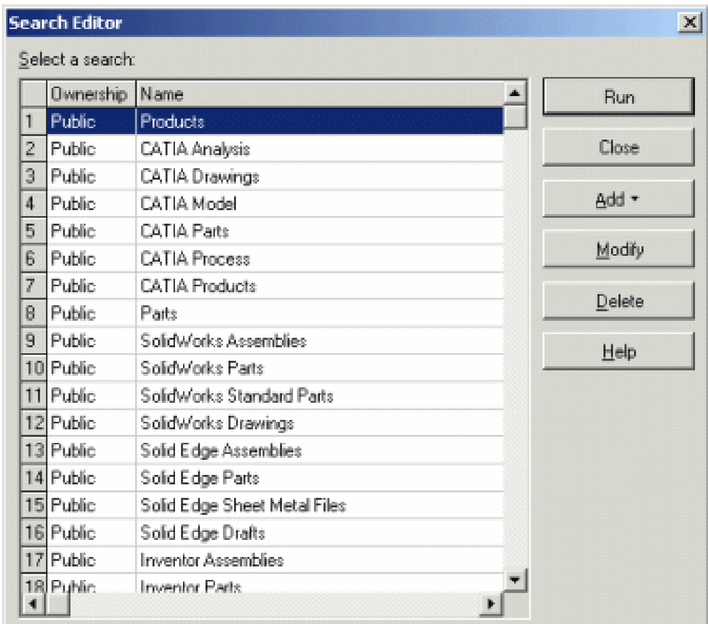
This task is made up of the following stages:

- 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.

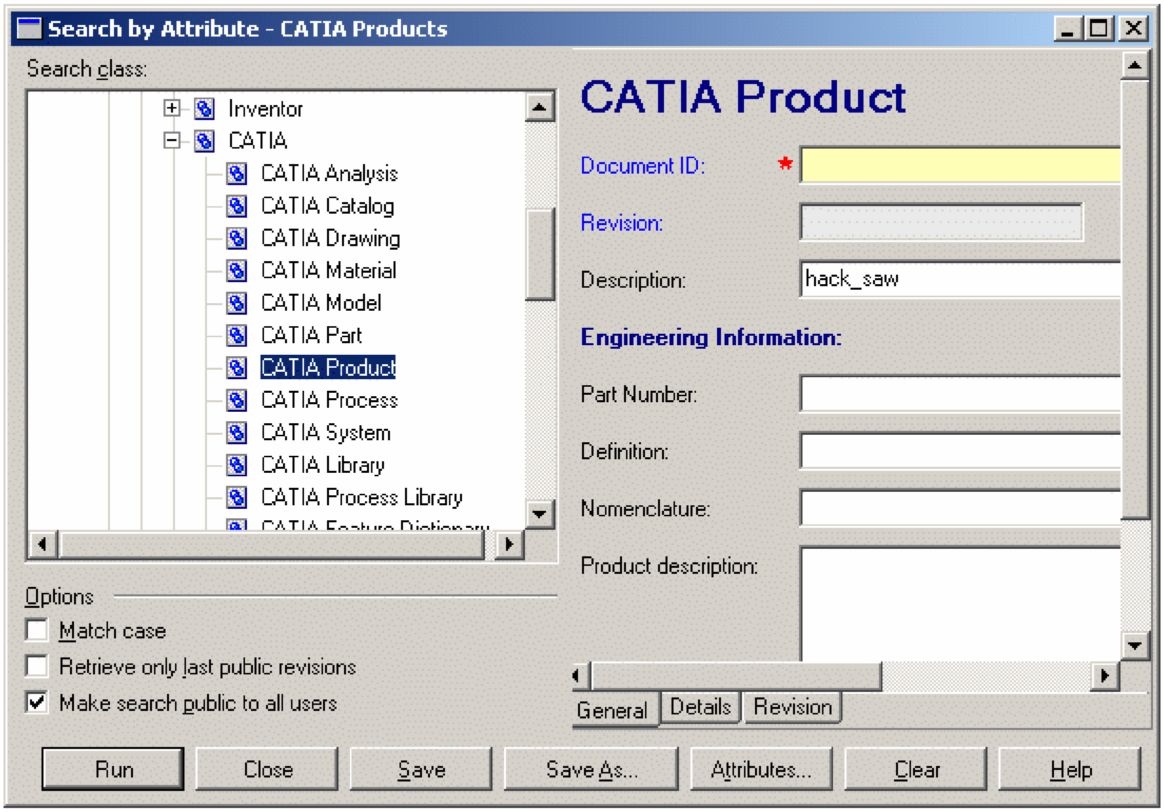


2. Select CATIA Products from the list.

3. Click **Modify** to perform the type of search that best meets your needs.

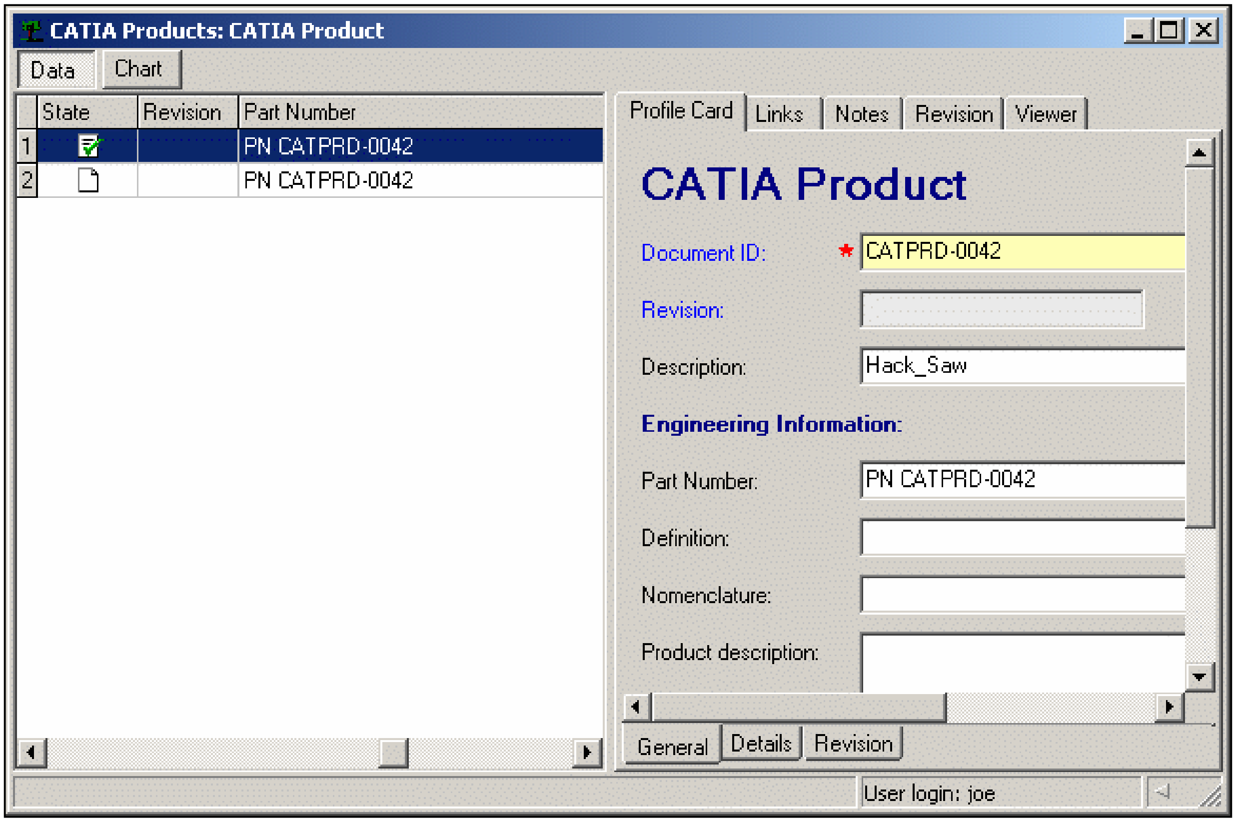
   The list of all CATProduct documents contained in the base is displayed in a new window.

4. In the **Description** field, enter hack_saw.



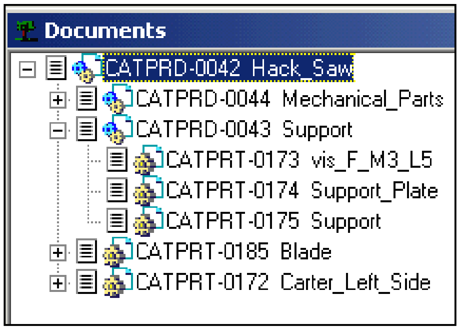
5. Click **Run** to confirm the search.

A list of CATIA Products referencing the part we want is displayed.



6. Right-click PN CATPRD-0042 and select **Open Views -> Top down tree**.

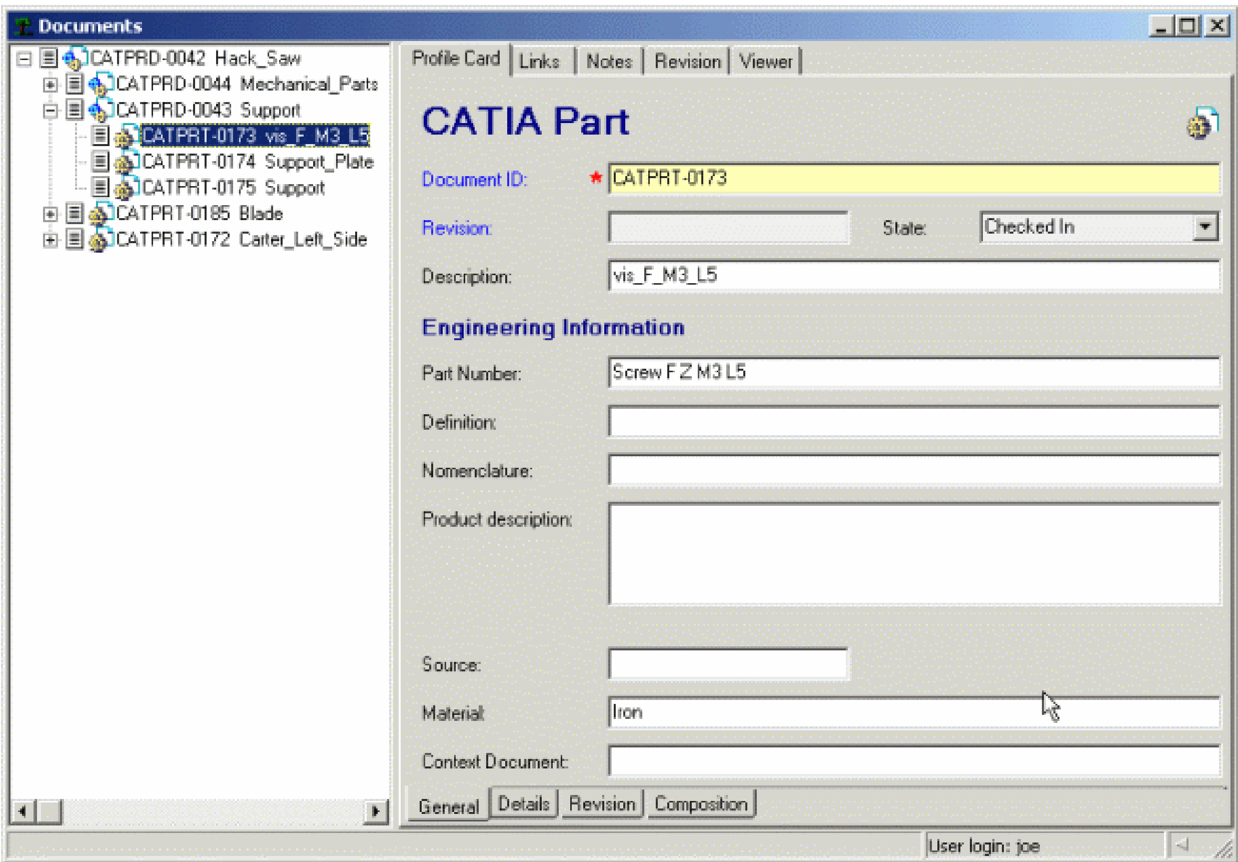
This displays in a new window all documents linked to the product you have just chosen.



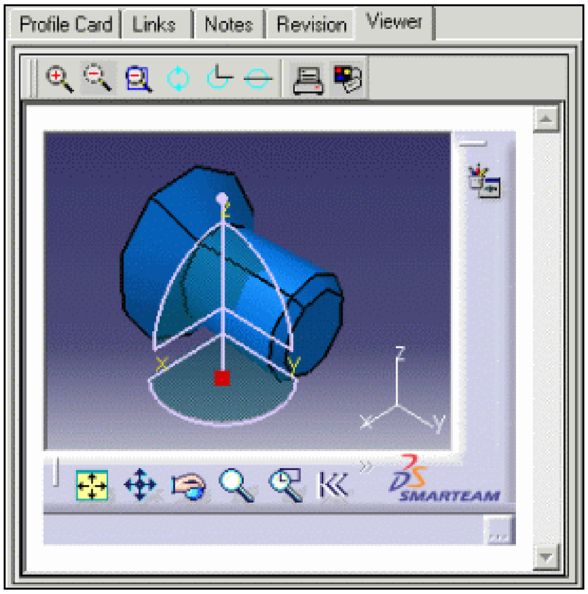
7. Select the + sign to the left of CATPRD-0043 Support to see which documents are associated with it.

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.



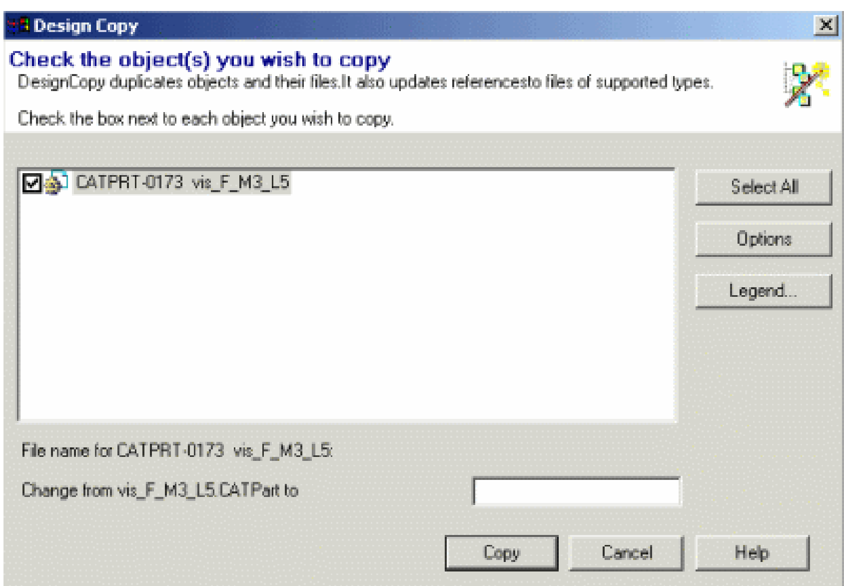
9. Optionally, click the **Viewer** tab to check that it corresponds to the part you want.



Creating a Copy from the CATPart Document

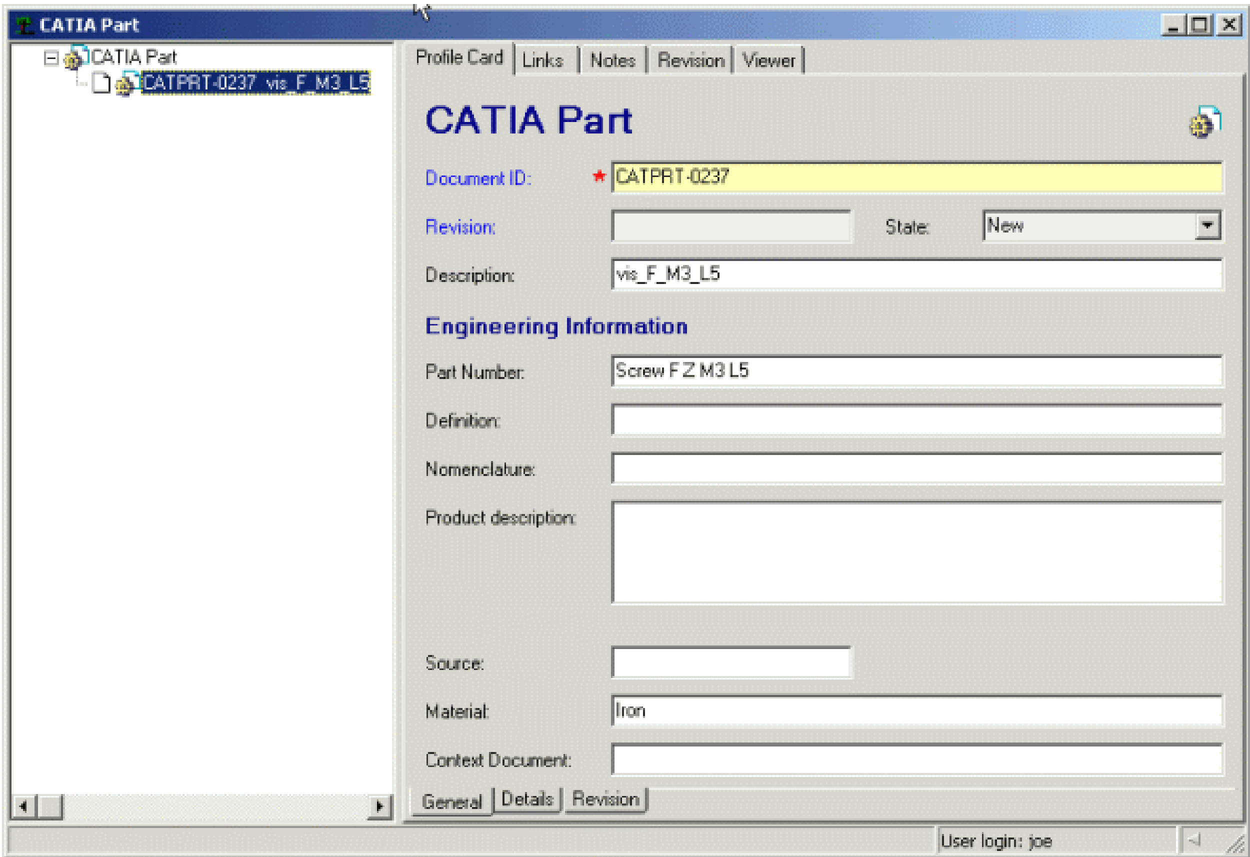
8. Right-click on CATPRT-0173 vis_F_M3_L5. and select **Design Copy**.

   The Design Copy dialog box appears.



9. Click the **Copy** button.

   Note that the CATIA Part window that appears indicates that the part is a new feature:



10. 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 the Design Copy command, refer to Copying Documents.

Editing the Copy in the Part Design Workbench

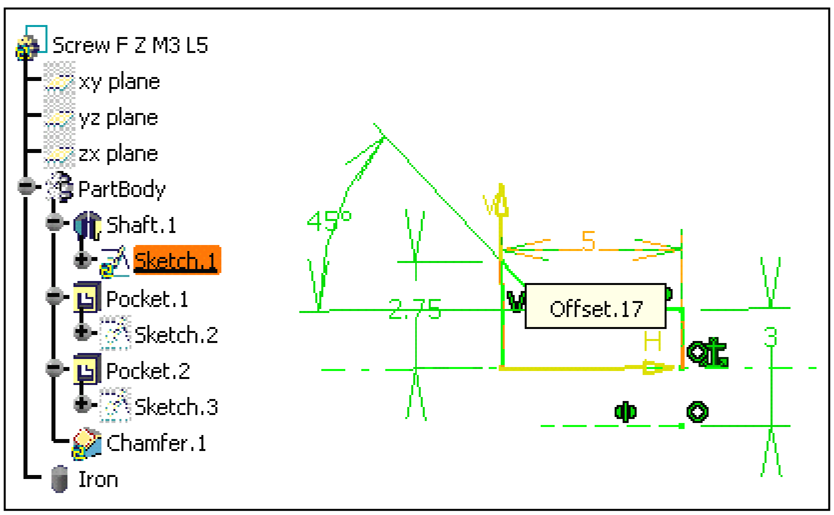
Now that you have loaded the copy in the Part Design workbench, you can modify the part's geometry. This task proposes you to perform three modifications, then it shows you how to store this new part in the SMARTEAM database.

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

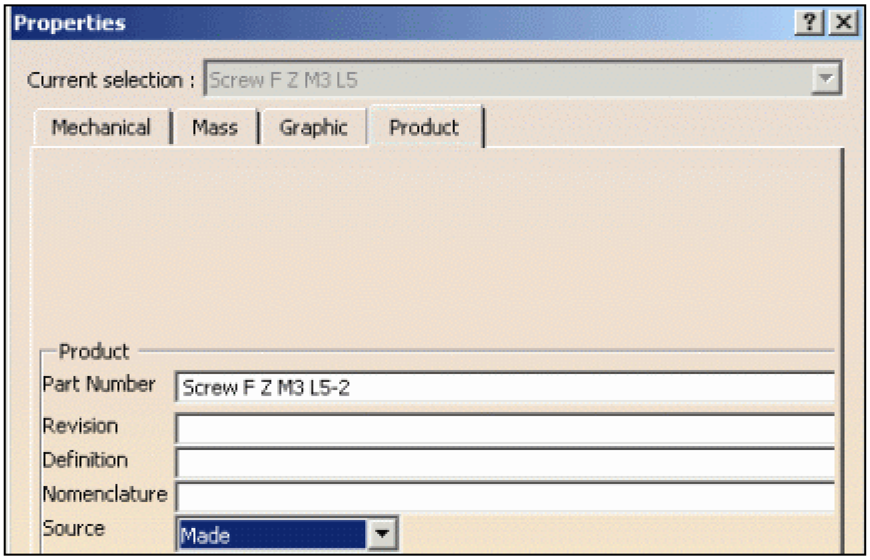
1. Double-click Sketch 1 to edit Offset.17.



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

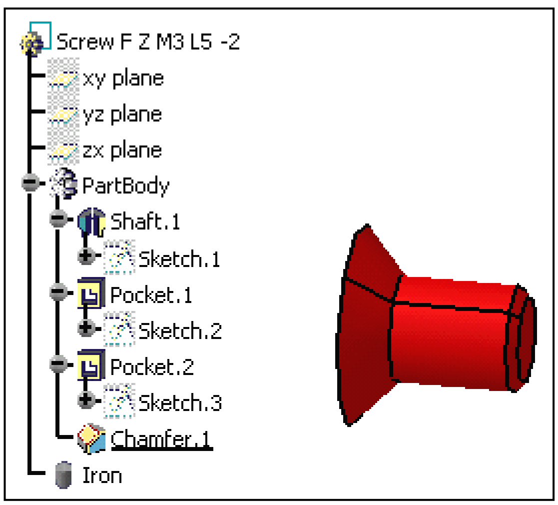
Applying a New Color to the Part

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 and select **Properties**.
7. Click the **Product** tab.
8. Add -2 to the **Part Number** displayed, and set the **Source** field to **Made**.



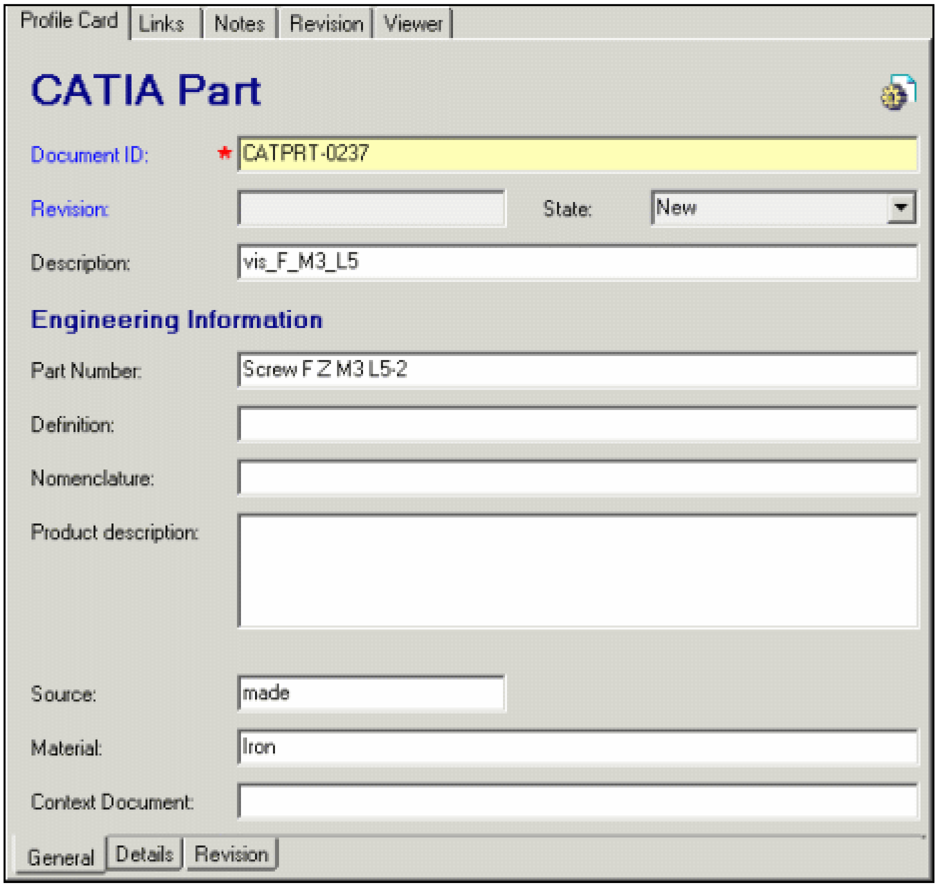
9. Click **OK** to confirm these operations.

   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.



Applying a Material to the Part

11. Back in CATIA, select the part in the specification tree.

12. Click on the **Apply Material** icon .

13. Select the **Metal** tab from the dialog box that appears, then select **Steel** as the new material to be applied to the copy.

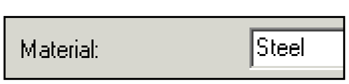
14. Select the **OK** button.

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**).



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

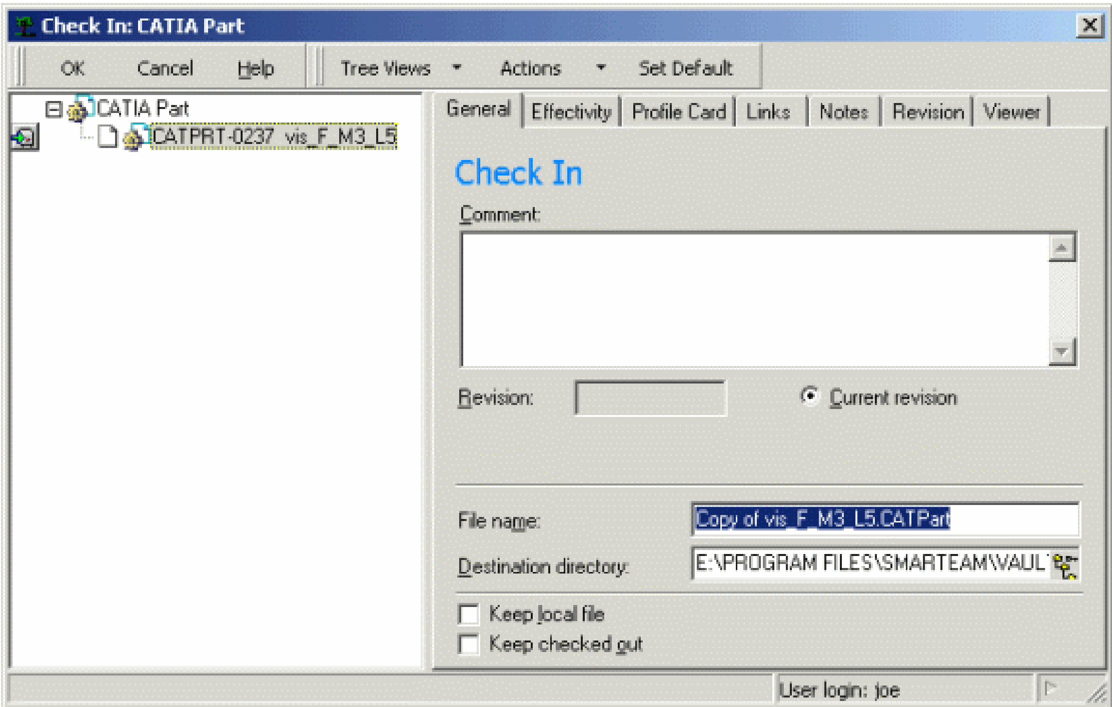
Storing the New CATPart Document in the SMARTEAM Vault

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.

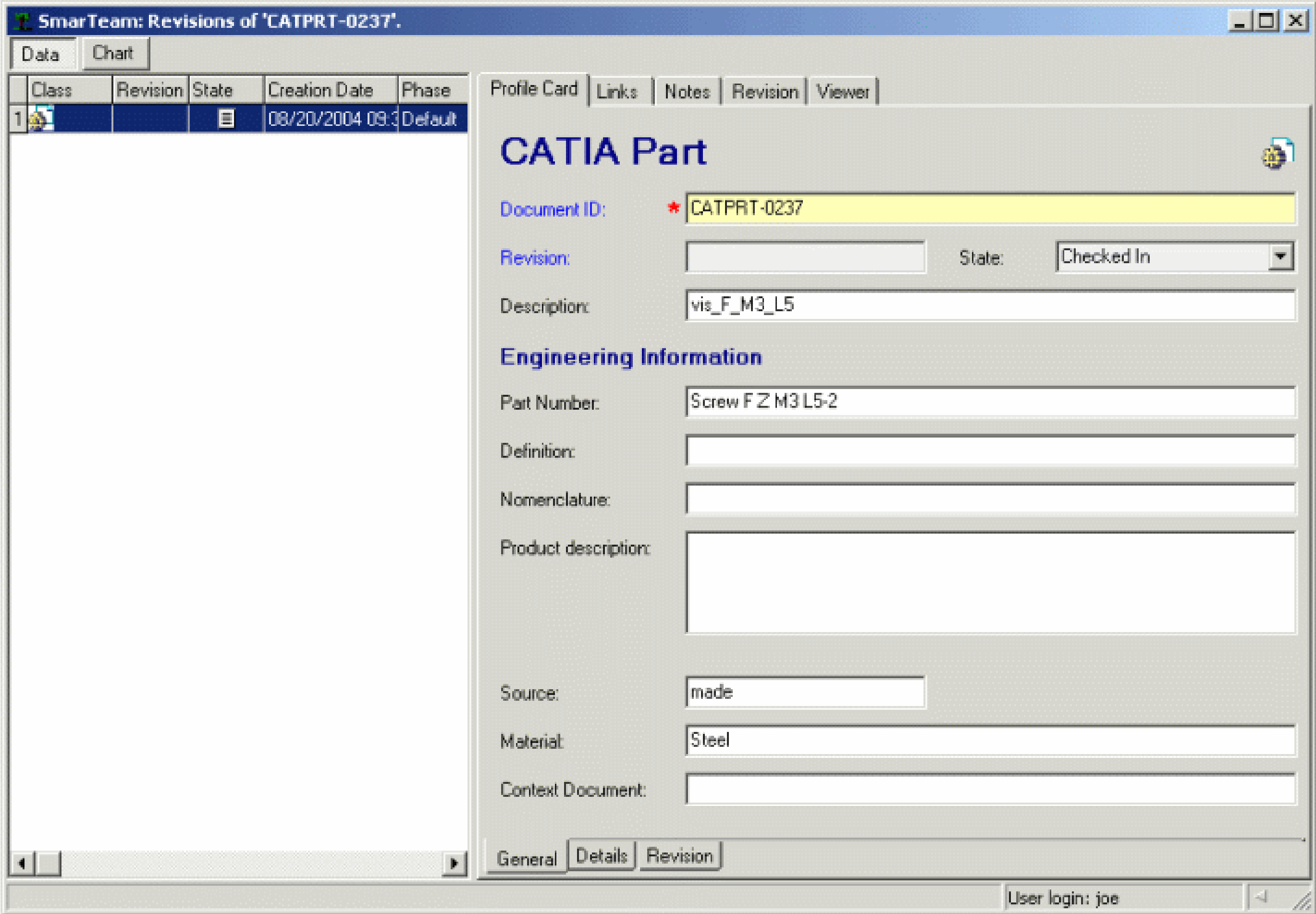


2. Optionally, enter some notes in the **Comment:** area.

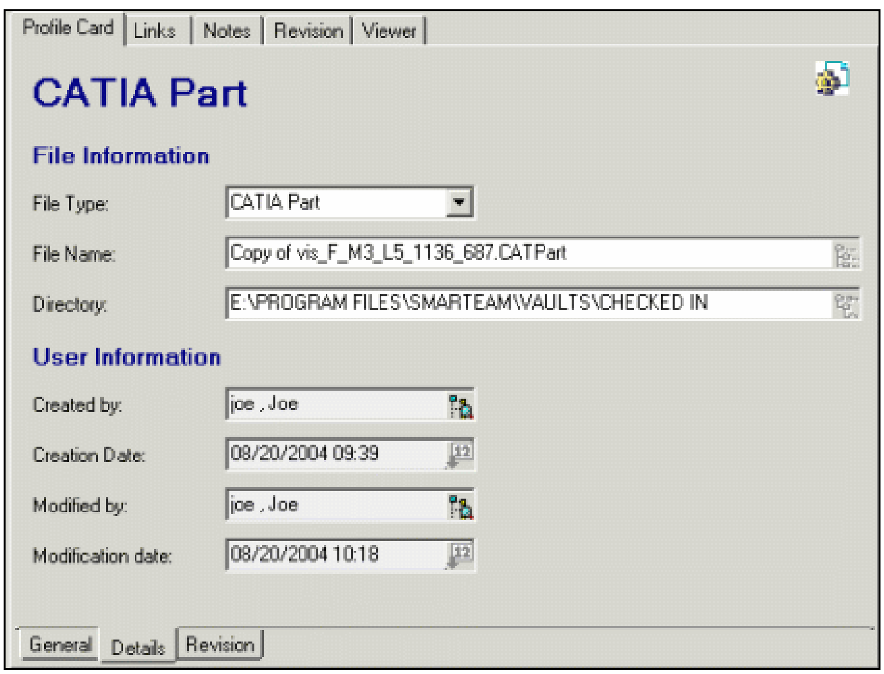
3. Click **OK** to confirm.

   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**.



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



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 the **Find Document** icon 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.

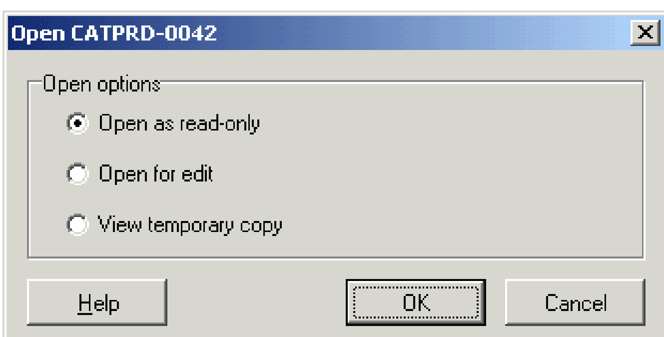
   The list of all CATProduct documents contained in the base is displayed.
4. Select CATPRD-0042 Hack_Saw.
5. Right-click CATPRD-0042 Hack_Saw and select **Open Views->Top down tree**.

   A Documents window is now displayed showing the whole CATPRD-0042 Hack_Saw assembly.

Opening the Assembly

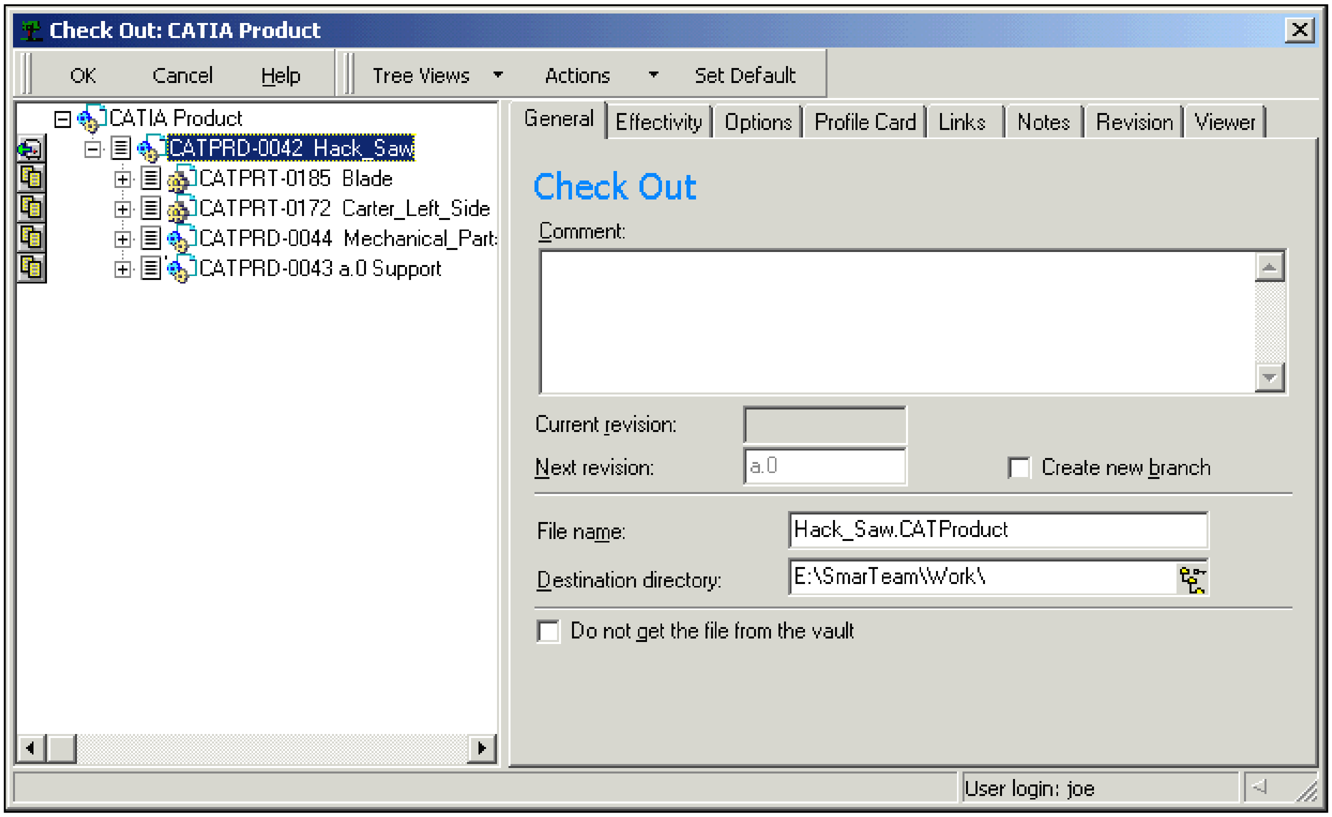
1. Right-click the assembly and select **File Operation-> Open For...** .

   The Open CATPRD-0042 dialog box is displayed. By default, the Open as read-only option is selected.



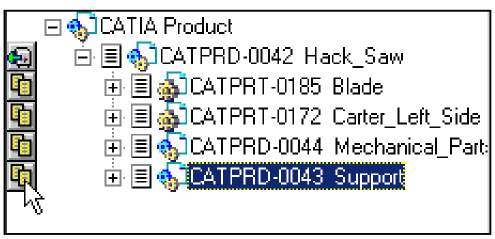
2. Because you want to modify the assembly, check the **Open for edit** option.
3. Click **OK.**

   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.

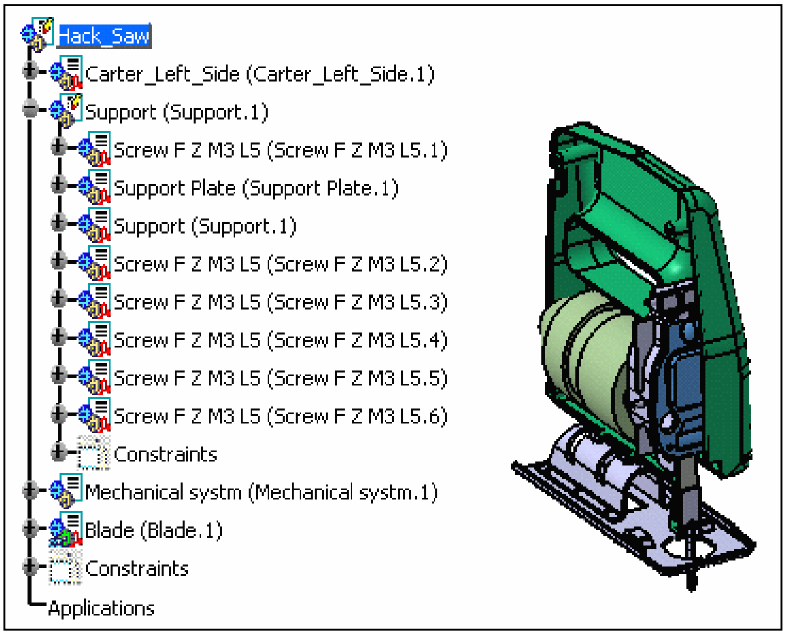
4. To check out not only CATPRD-0042 Hack_Saw but also one of its components, you need to click on the icon close to that component. For the purpose of our scenario, click the icon in front of CATPRD-0043 a.0 Support.



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.



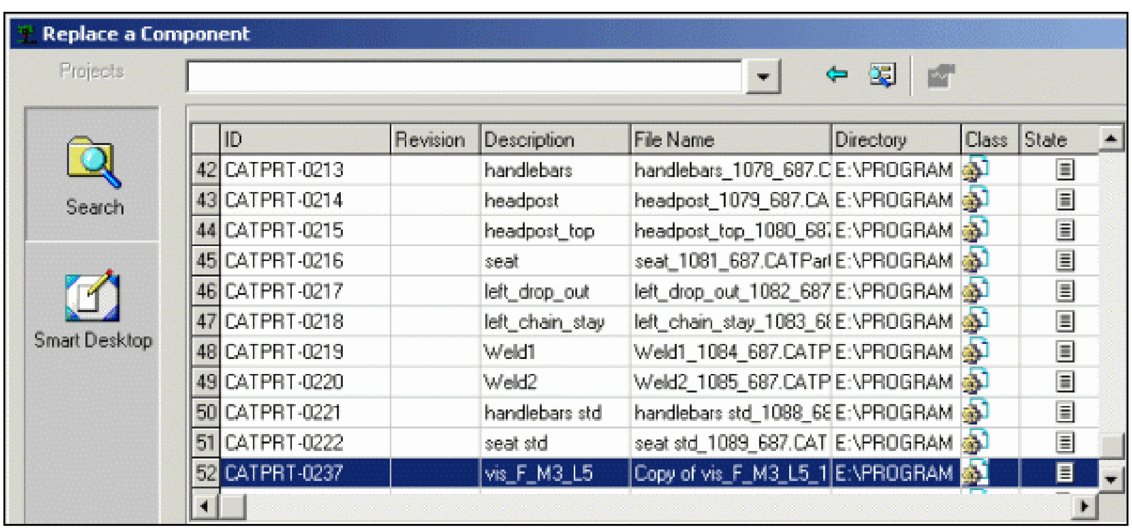
Note that the icons close to Hack_Saw and Support (Support.1) indicate that you can edit these documents, which is not the possible for the other documents.

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...**

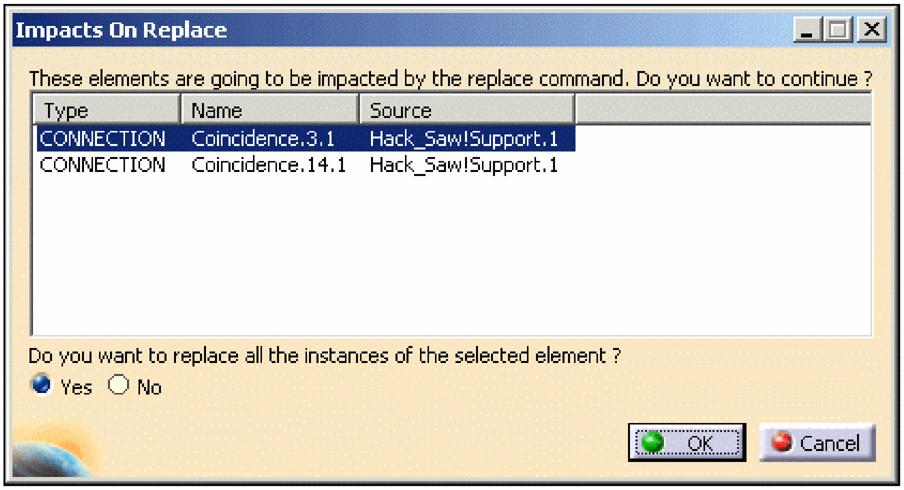
   The Replace a Component window displays.
3. Set the Class name combo list to CATIA Part.

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



6. Click **OK.**

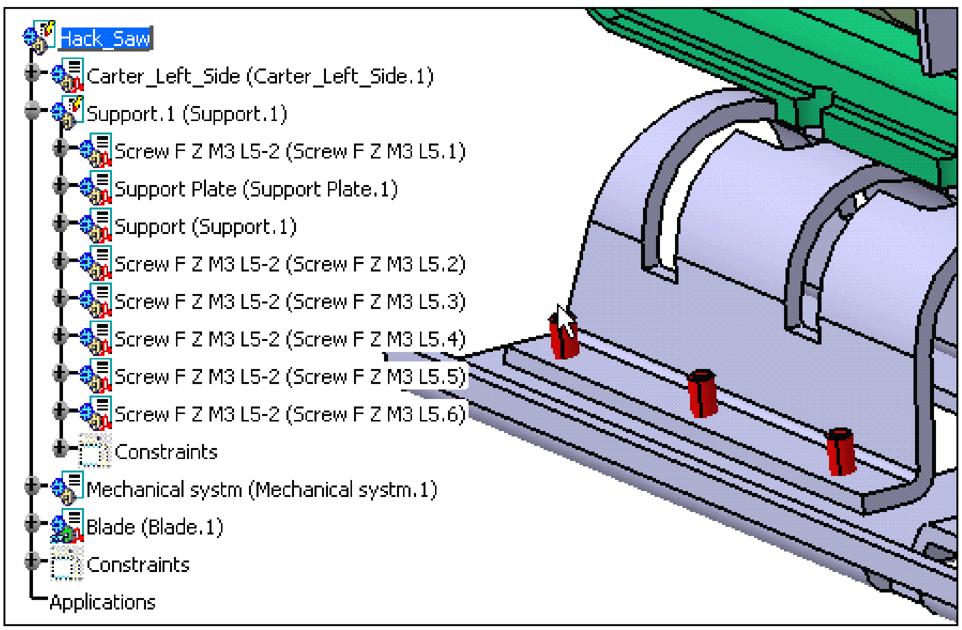
   The Impacts On Replace dialog box appears, showing the elements that will be affected by the change.



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

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

The original screw has been replaced as shown here:



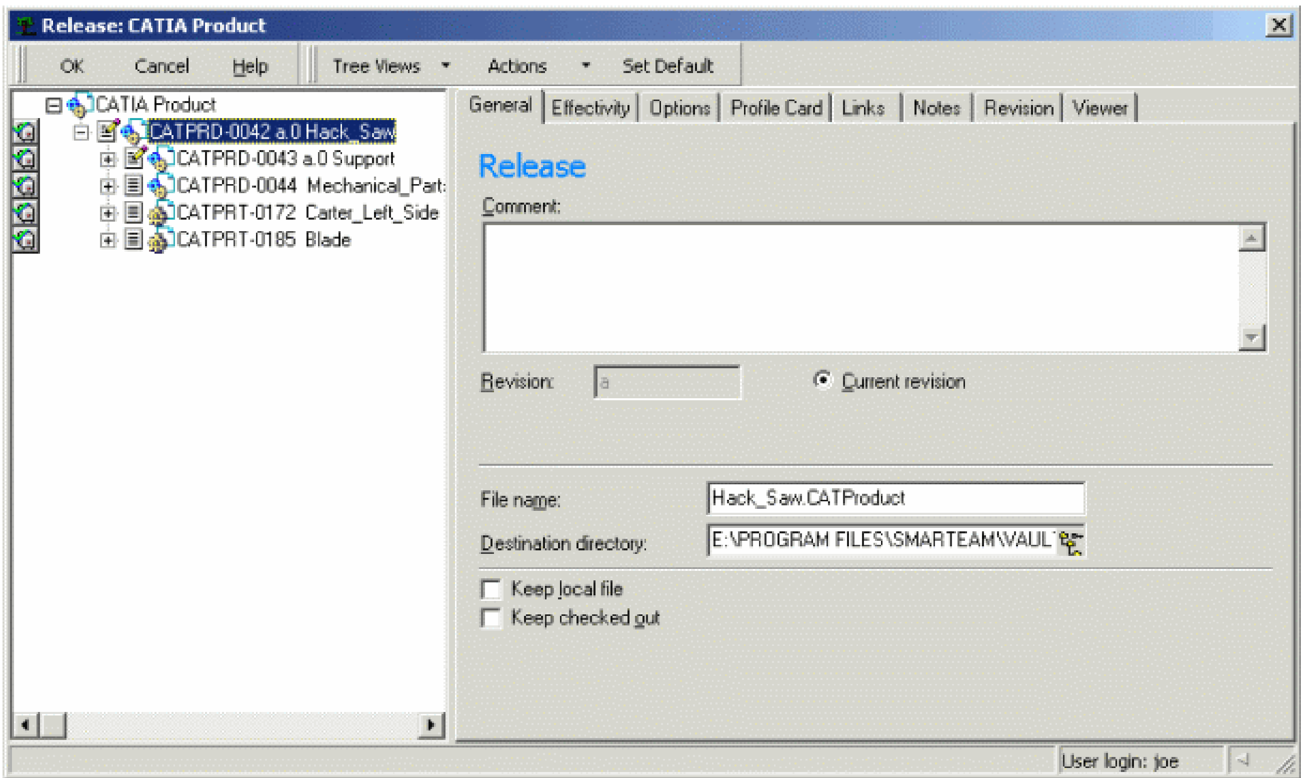
Now let's go on to Releasing the Modified Assembly, which is our last task.

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:

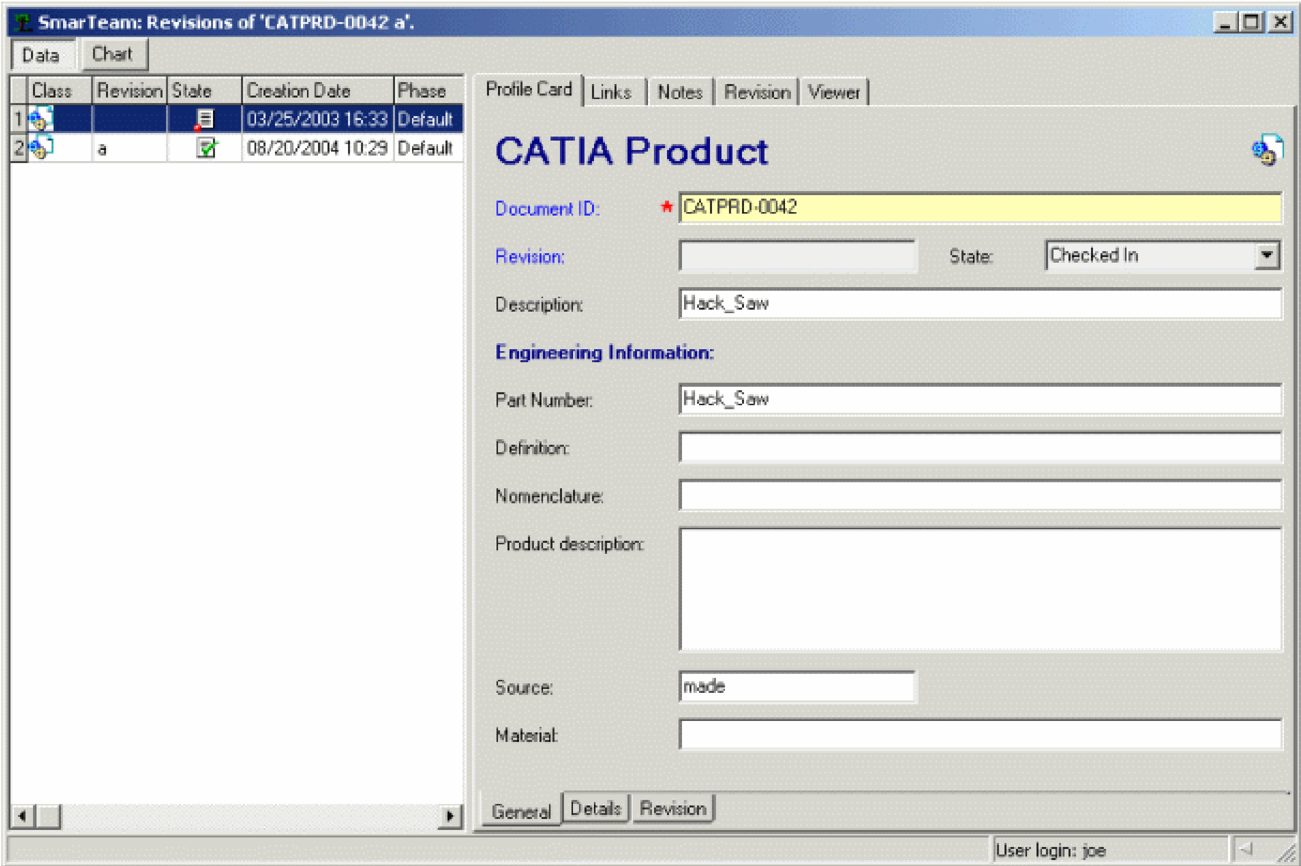


   Note that the Release operation automatically releases the documents that you did not check out for replacing the original screw, as indicated by this icon  displayed close to each document.

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


SmarTeam: Revisions of 'CATPRD-0042 a'.
Data
Chart
Class
Revision
State
Creation Date
Phase
1
03/25/2003 16:33
Default
2
a
08/20/2004 10:29
Default
Profile Card
Links
Notes
Revision
Viewer
CATIA Product
Document ID:
CATPRD-0042
Revision:
State:
Checked In
Description:
Hack_Saw
Engineering Information:
Part Number:
Hack_Saw
Definition:
Nomenclature:
Product description:
Source:
made
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:

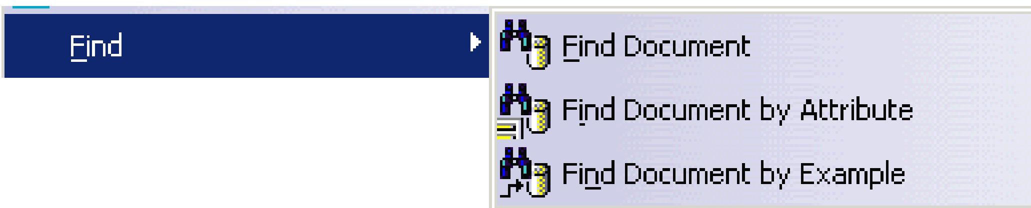
Find	Use the **Find** options to run a search and locate specific documents that match the search criteria. These documents are listed in a search results list.
Browse	Browses through each document in the list. Each time you select a document, its Profile Card is displayed. You can view general attributes of the document as well as its revision history. In addition, you can view a thumbnail image of the document in the **Viewer** tab.
Find Out Where a Document Is Used	Keeps track of all the Assemblies that use a particular document as a component before you begin to modify the Part/Product.
Open as read-only	When you locate the exact document that you were searching for, you can launch it directly into CATIA V5, for viewing and inspection.
Open for edit	When you locate the exact document that you were searching for, you can launch it directly into CATIA V5, and modify it accordingly. See Editing.
View temporary copy	Use **View Temporary Copy** 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 Viewing "exactly as released".
Save	When you have finished your modifications, save the document. The Profile Card (and appropriate revision history) is updated accordingly. See Saving a Part, Saving an Assembly or Saving a Drawing.

Finding/Browsing/Editing
Viewing "exactly as released"
File->Open Integration
Running a Predefined Search
Finding Out Where a Document Is Used
Showing Profile Cards
Duplicating an Existing Document

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:



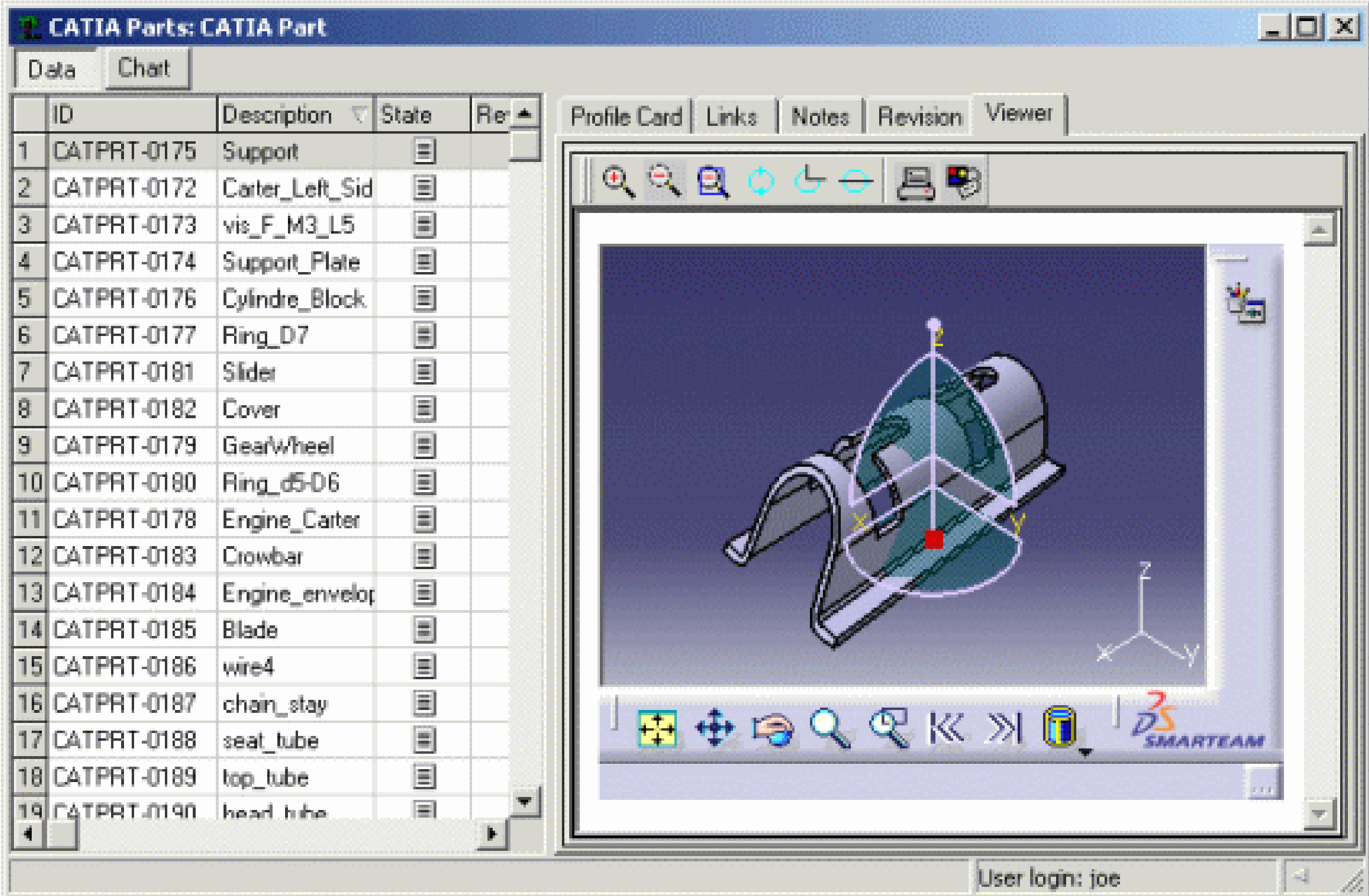
- **Find Document** : This option enables you to view the previously defined searches. From the Search Editor window, you can:
  - run a previously defined search (see Running a Predefined Search)
  - modify a search.
  - create a new search.
- Each search may contain numerous search criteria. The results 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.



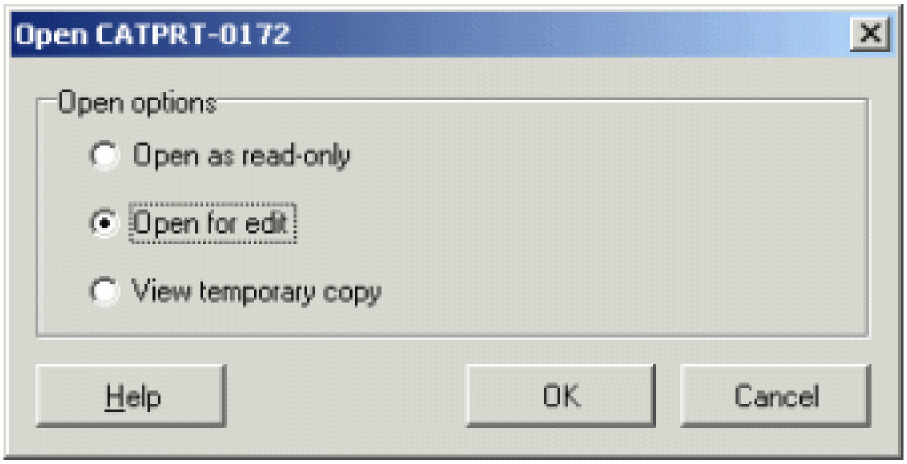
Editing

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 then 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.

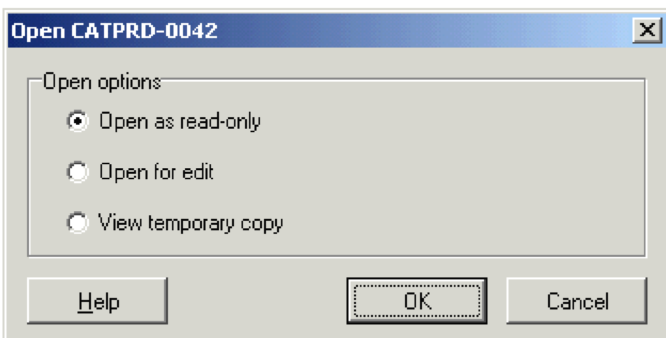
Recommendations

- 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.

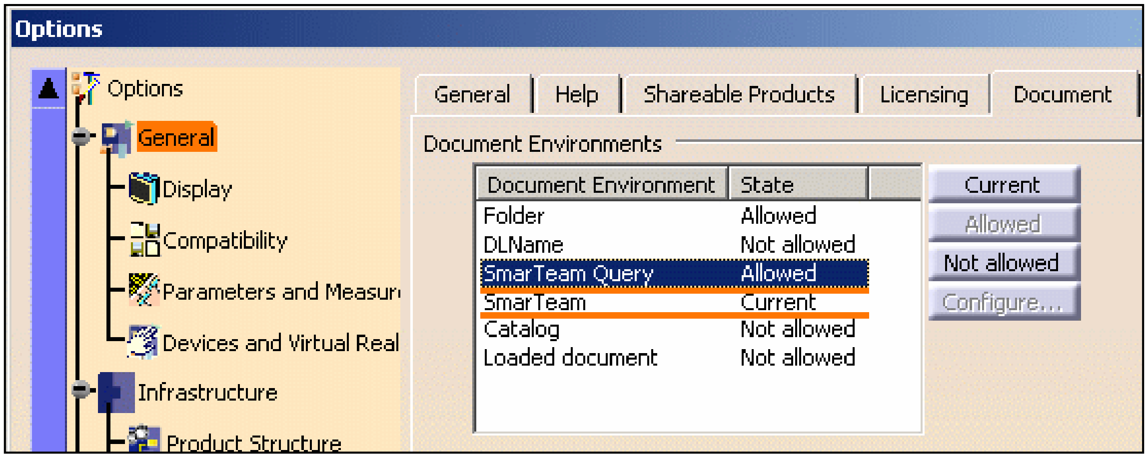
File->Open Integration

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

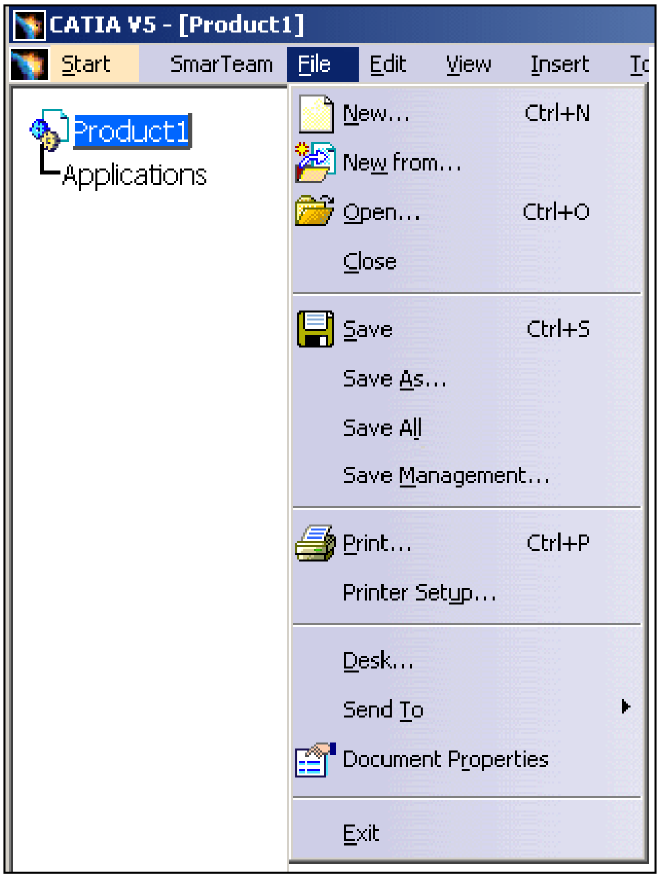
A SMARTEAM dialog box can be displayed after running the following commands:

- **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.



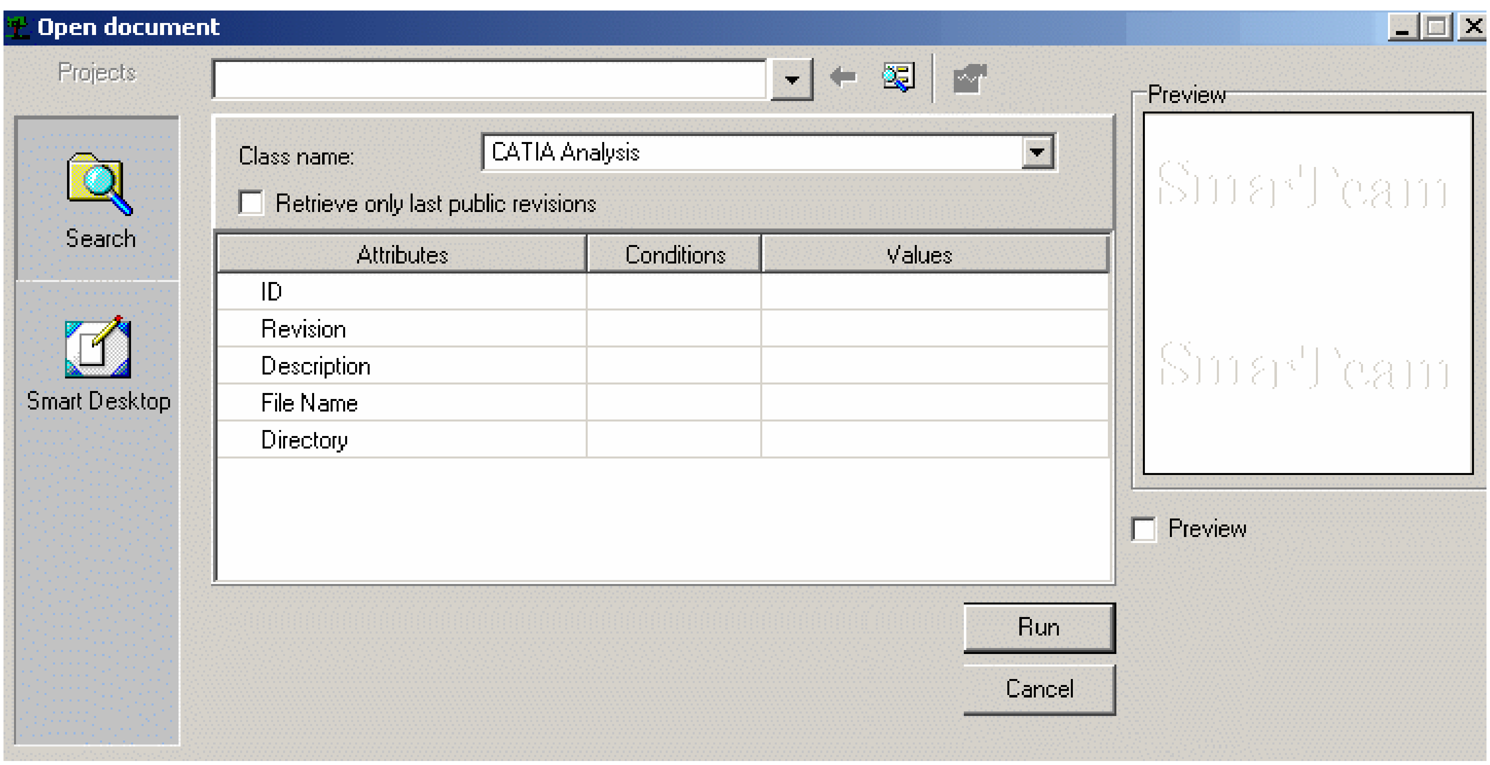
2. From the SMARTEAM toolbar, click the **Connect** icon .
3. Select **File-> Open...**.



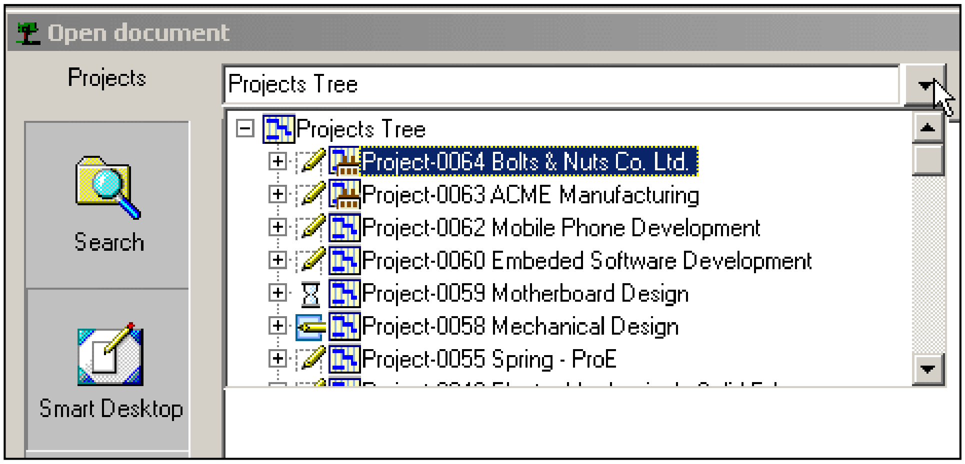
SmarTeam Set as Current

If SmarTeam is set as current in the Document Environments list, Open document dialog box is displayed

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, refer to the SMARTEAM documentation.

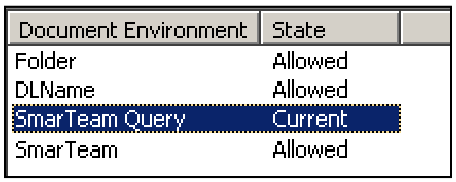


Clicking on Smart Desktop enables you to search from the list of projects:

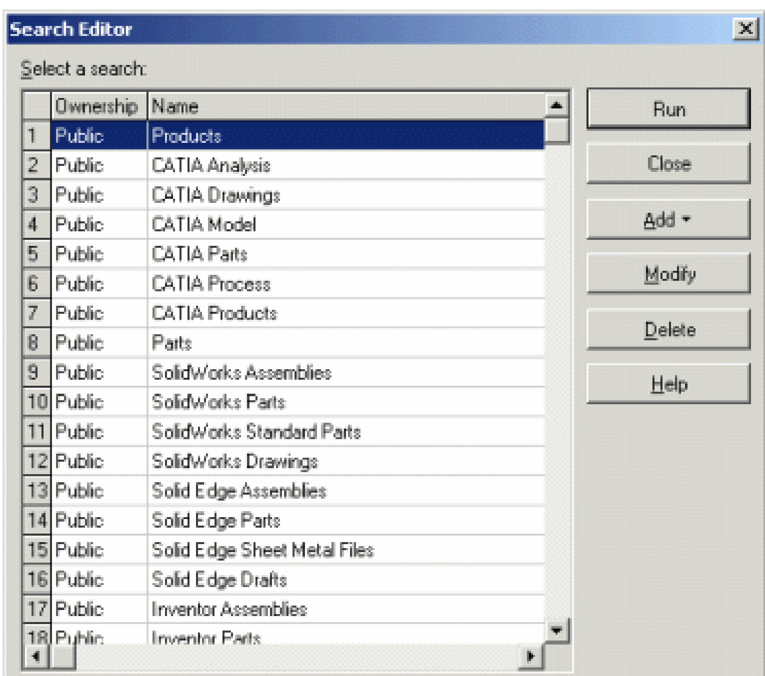


SmarTeam Query Set as Current

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



Running a Predefined Search

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

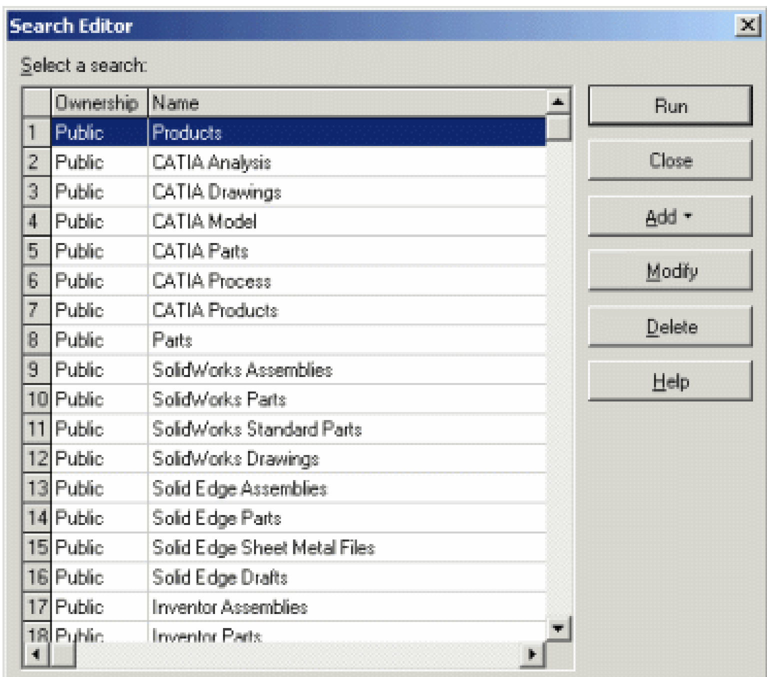
- run a previously defined search.
- modify a search.
- create a new search.

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:

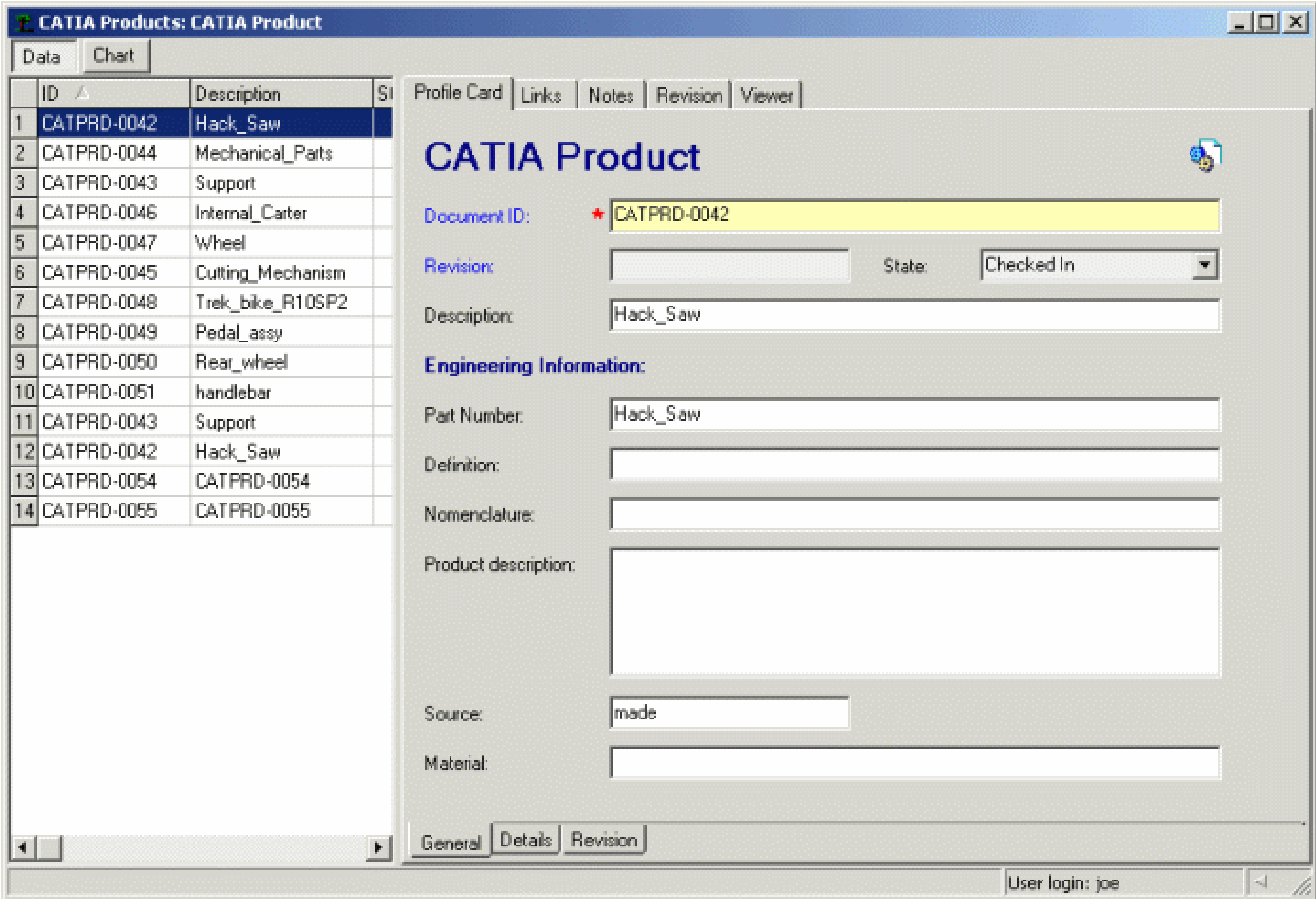


   Note that the Folder Document Environment must be set to **Allowed** to be able to use the Find capabilities. For more information, refer to Documents Environments. From the Search Editor window, you can:

   - 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:



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

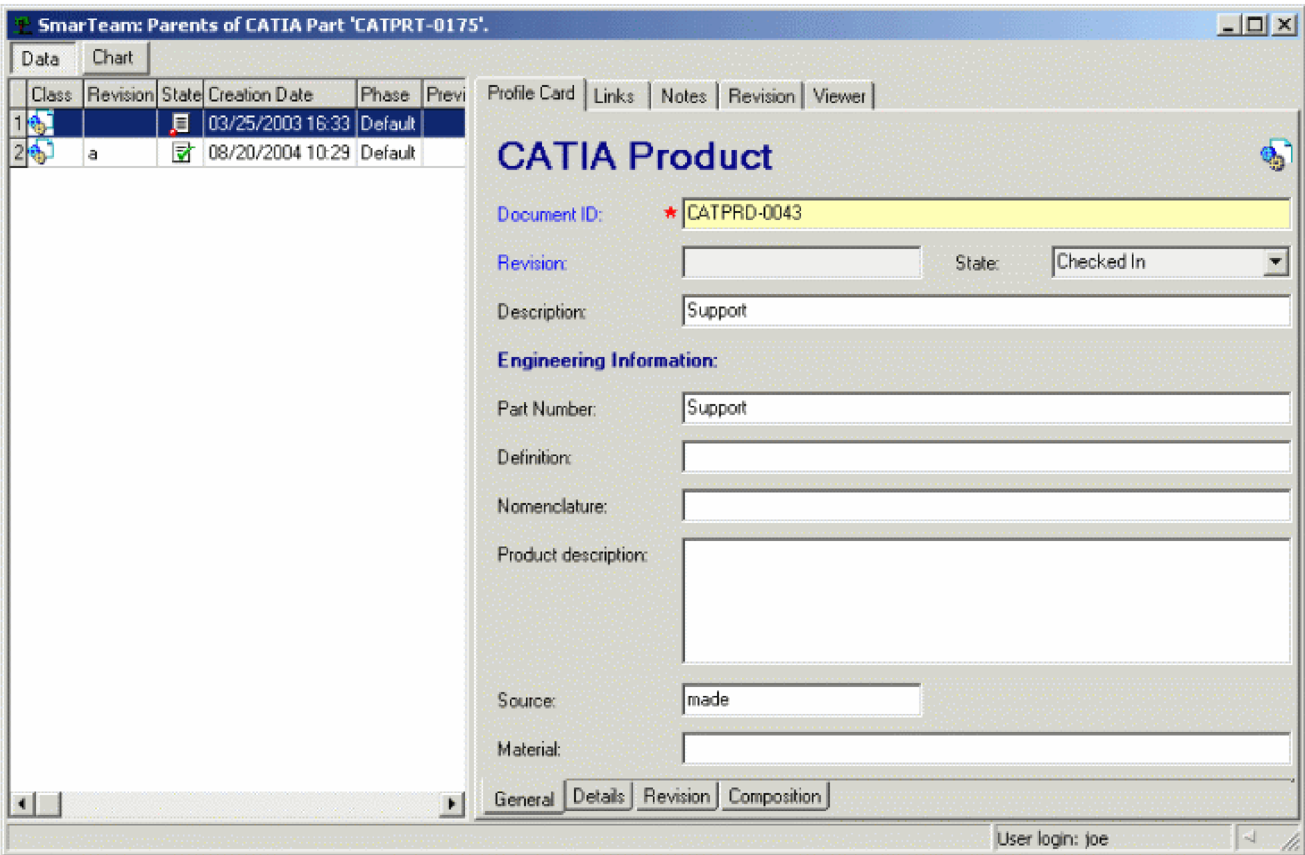
Hierarchical Links (mainly Product Structure relationships)

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.



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 checks 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, refer to Enriched Decision Support with All V5 Links.

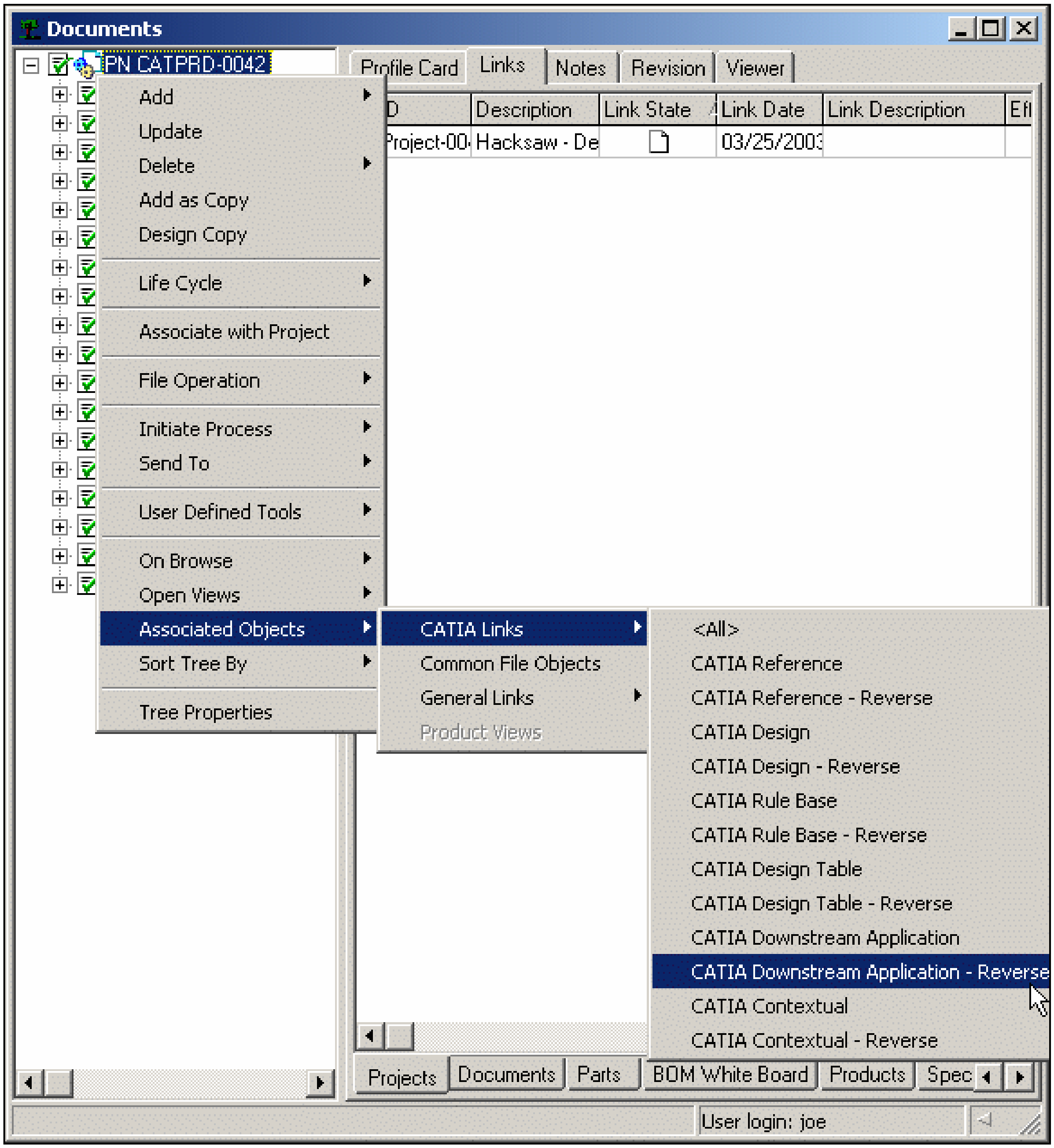
Let's take the example of a CATDrawing referencing a CATProduct document. In that case, the link is a **CATIA Downstream Application** link.

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

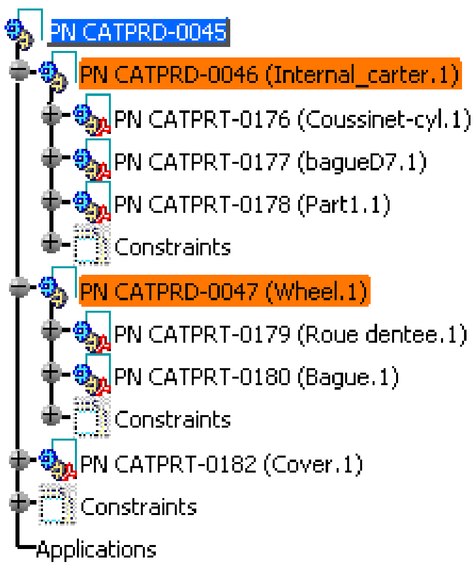
If you wish to view the relationships between documents open in your CATIA session and other documents as saved in SMARTEAM, you can use **Show Profile Card.** This capability provides the database view of the document currently displayed in a CATIA session. It is available for all documents and can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

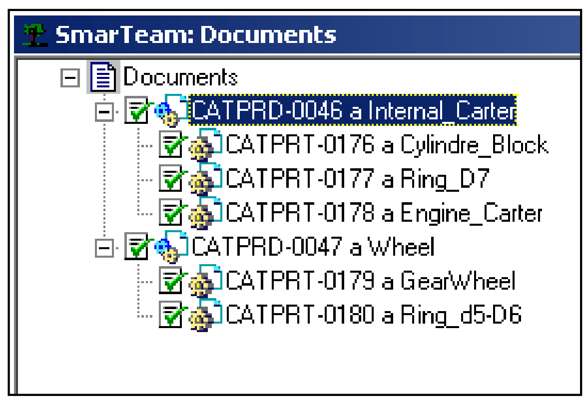
- From CATIA specification tree
- From the CATIA Desk tree
- From the 3D area

From CATIA Specification Tree

1. Select the document of interest.

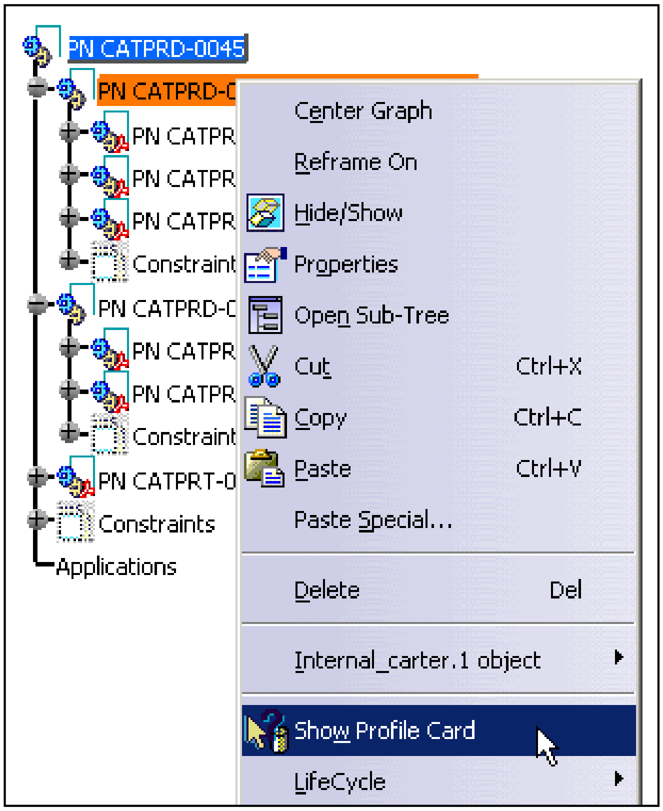
2. Click the **Show Profile Card** icon or select **SMARTEAM-> Show Profile Card**.
   Note that the command applies to multi-selected documents:





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.

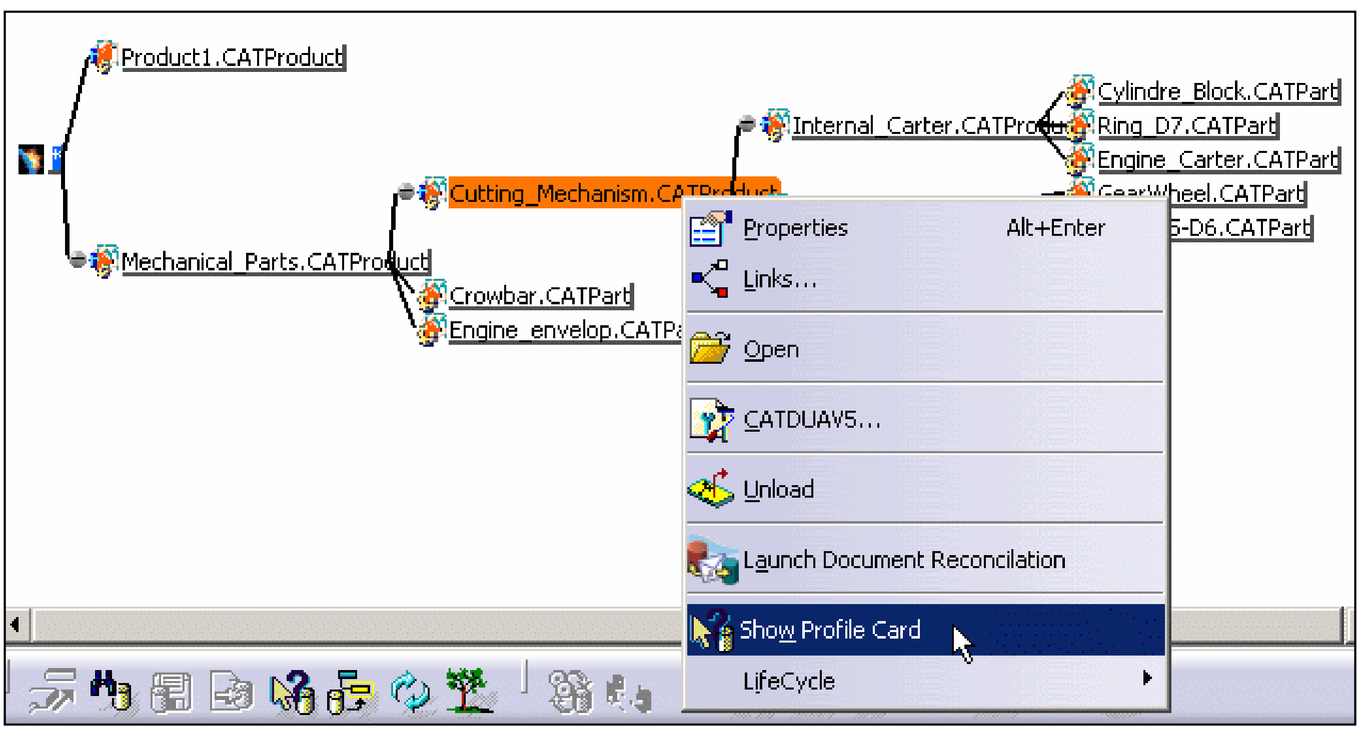
- From V5R15 onward, you can run **Show Profile Card** via contextual menus available from the documents you select. The command applies to the documents you select, not to the current document. Just as a reminder, the application distinguishes **selected documents** from **active documents**.
  - Selected 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.

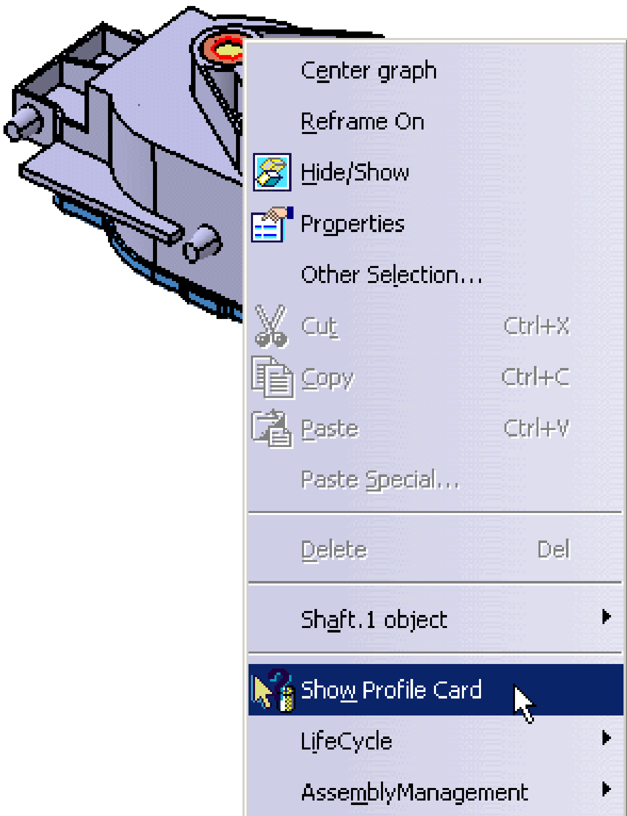
1. Right-click the document of interest.
2. Select **Show Profile Card** contextual command:



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.



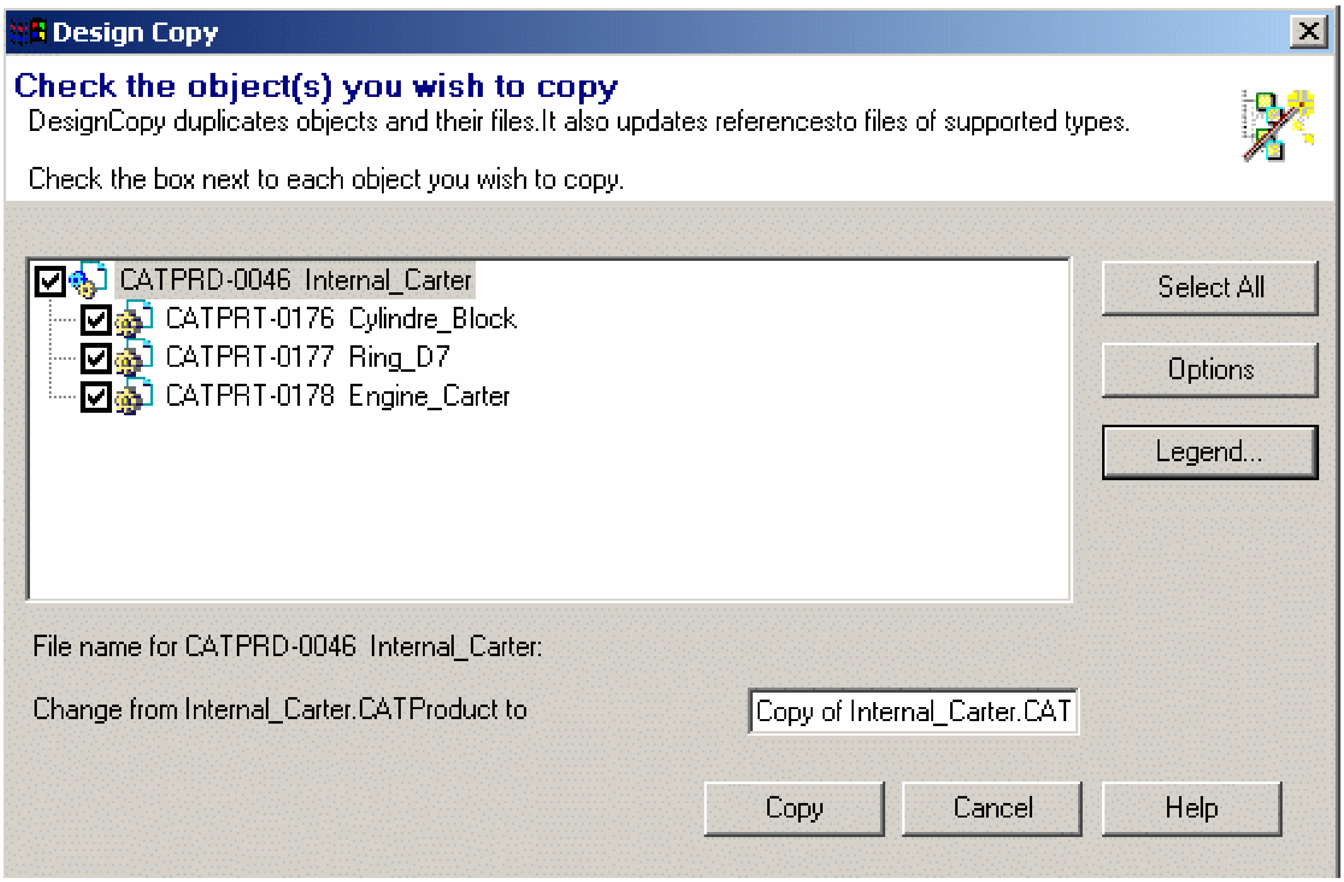
Duplicating an Existing Document

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.



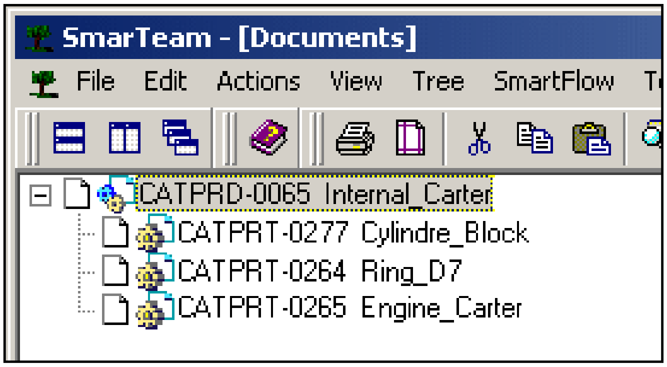
2. Check the documents you wish to copy, if not already done.

   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.



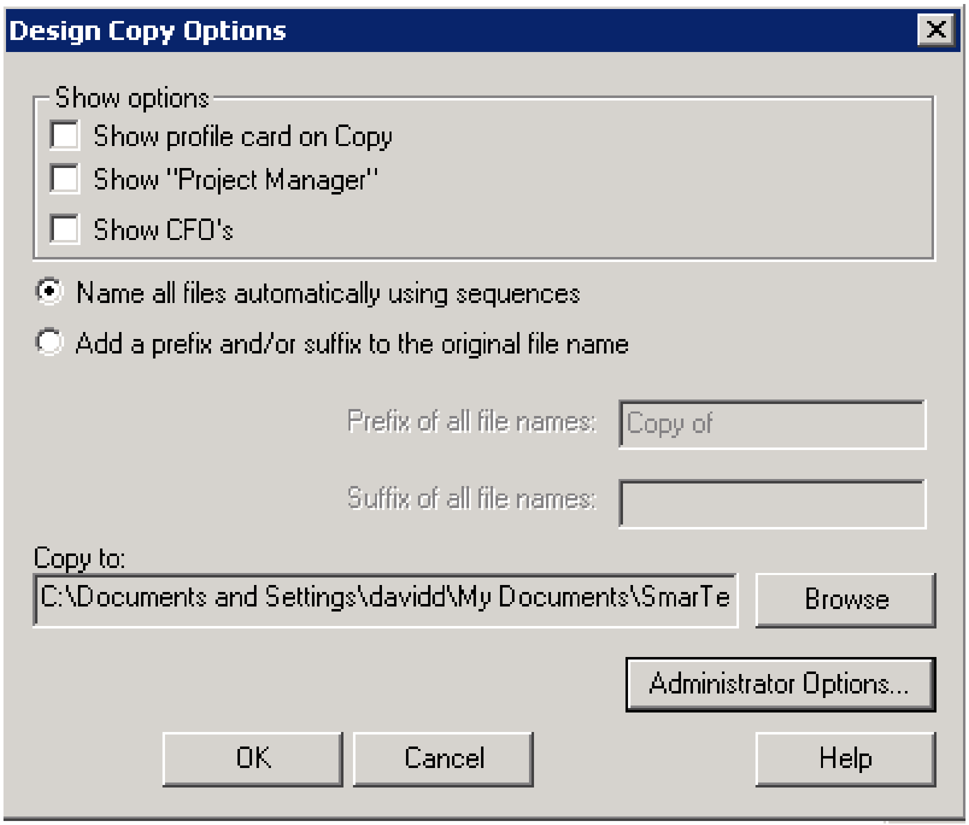
Select All

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.

Options

5. Click **Options**.

   The Design Copy Options window appears.



6. 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.

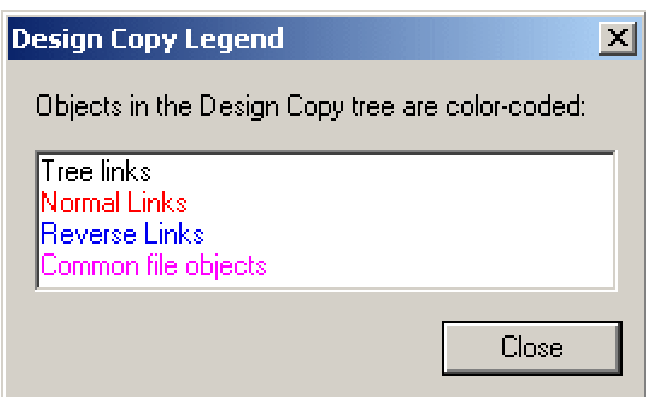
The file is copied to the destination entered in the **Copy to** field.

Administrator Options...

The **Administrator Options** is disabled. It is only enabled if you are an administrator. If so, refer to 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.

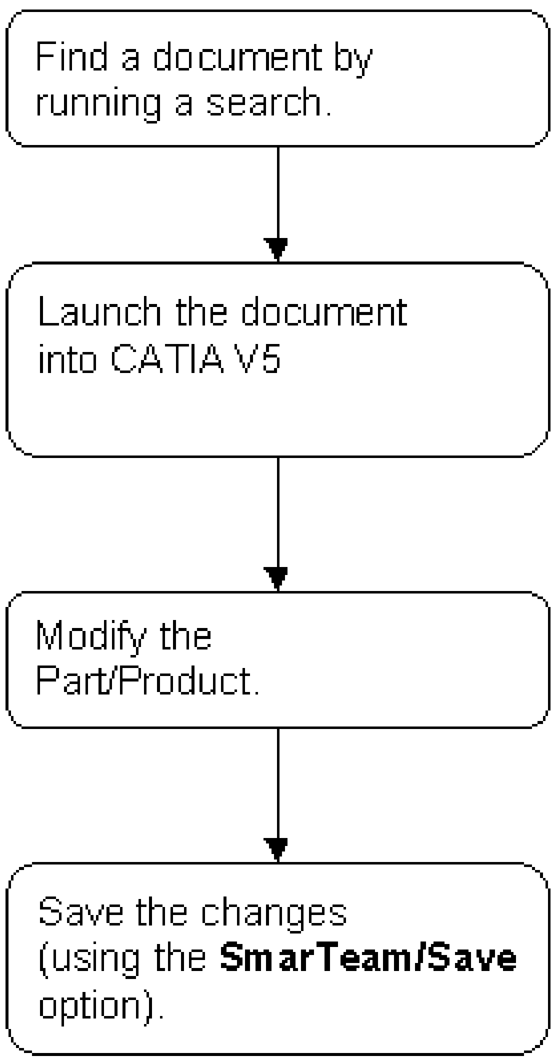
Managing CATIA Parts

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**). Refer to "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.

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?

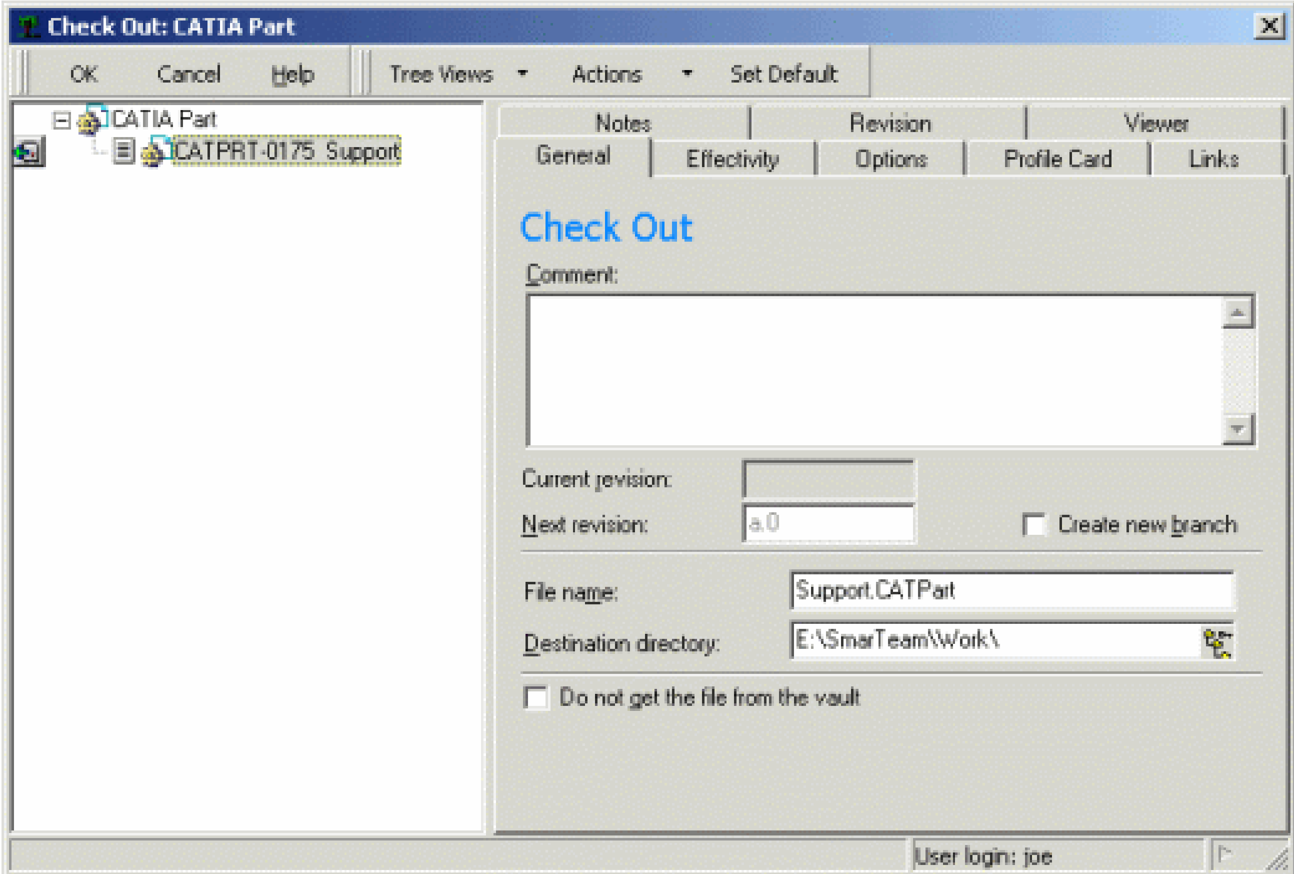
- You previously checked in the Part, and a copy or the Part remained at your desktop in read-only mode, as described in Checking In a Part.
- You previously copied a file to your desktop (using the **Copy File** contextual command).

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.**

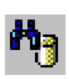
   The Part is re-displayed at your desktop and it can now be modified.

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:

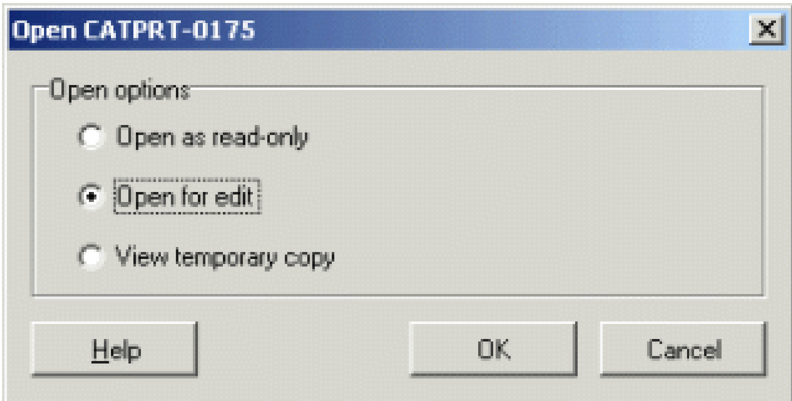
1. Run a search to locate the document that you wish to check out:

   - 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.

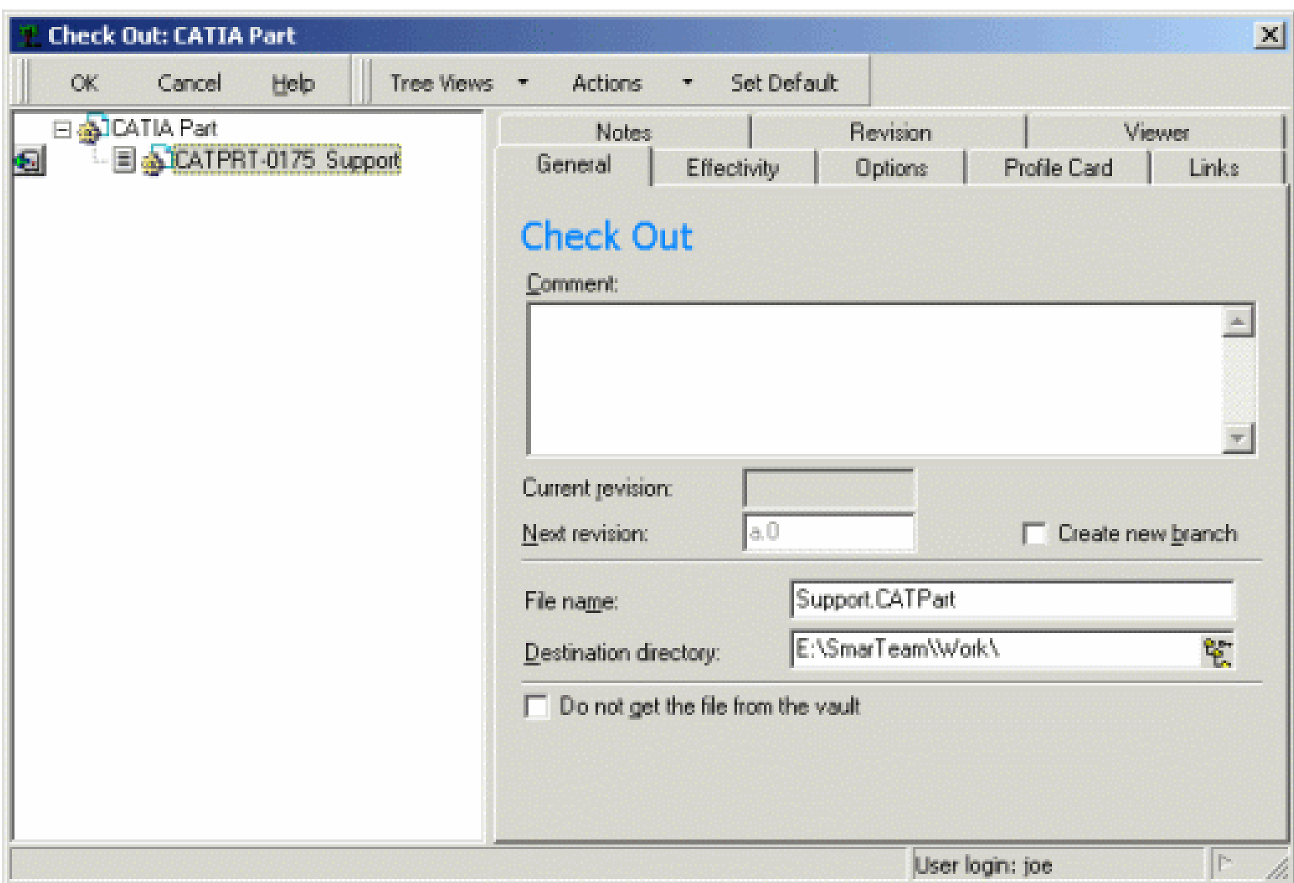
3. Check the **Open for edit** option.



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
General tab	
Current revision **Next revision**	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.
Create new branch	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). 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. **Note**: This field may be disabled according to the life cycle preferences defined by your administrator.
Destination 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.
Effectivity tab	
Phase	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.
Comments	You may enter a comment in this field.
Options tab	

Copy General links on Check Out/New Release	Click on this option to copy general links from the previous to the newly created revision.
Copy links to children on Check out	Click on this option to copy all the links to children in the current revision to the new revision.
Replace local files on Check Out from vault	Choose an option from the dropdown list: • **Yes** - to replace all local files that were copied during the Check Out operation. • **Yes for copied files** - to replace the file if a copy was made of this file during an earlier operation. • **Ask** - to prompt the user before making a copy of the file.
Set Default	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.

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.

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 with its Associated Design Table

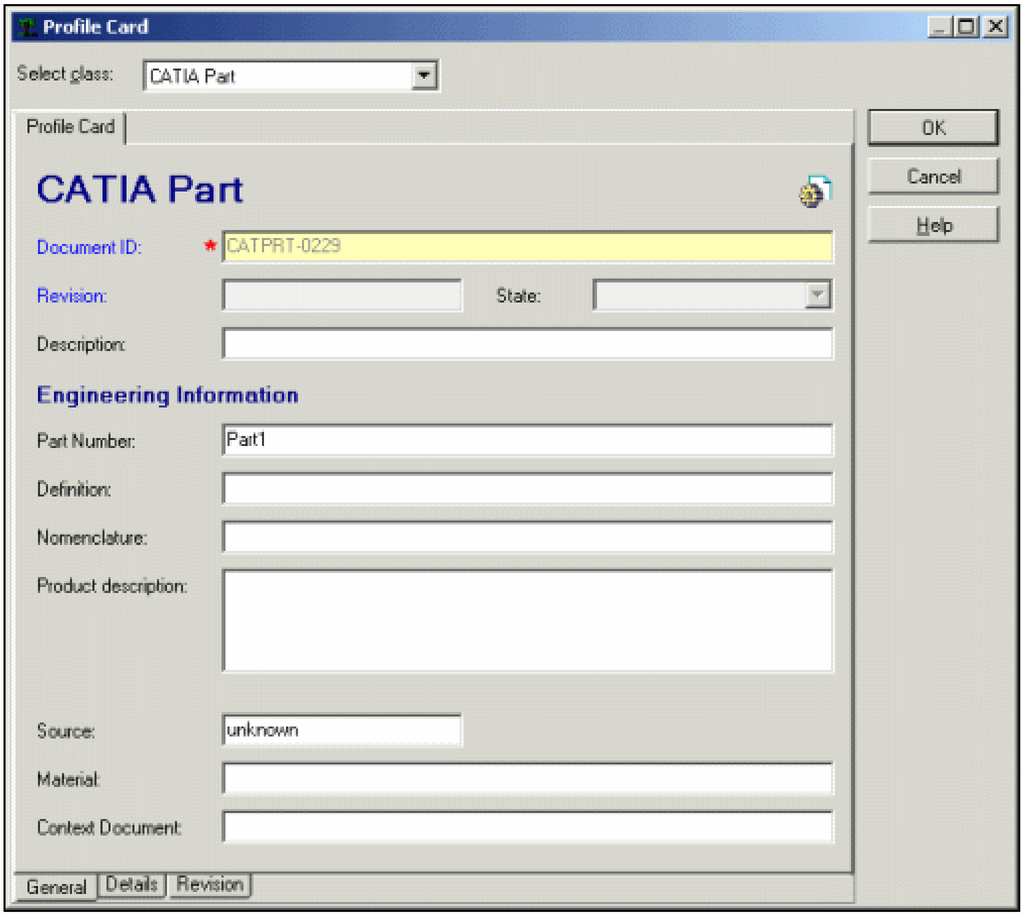
Profile Card

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.

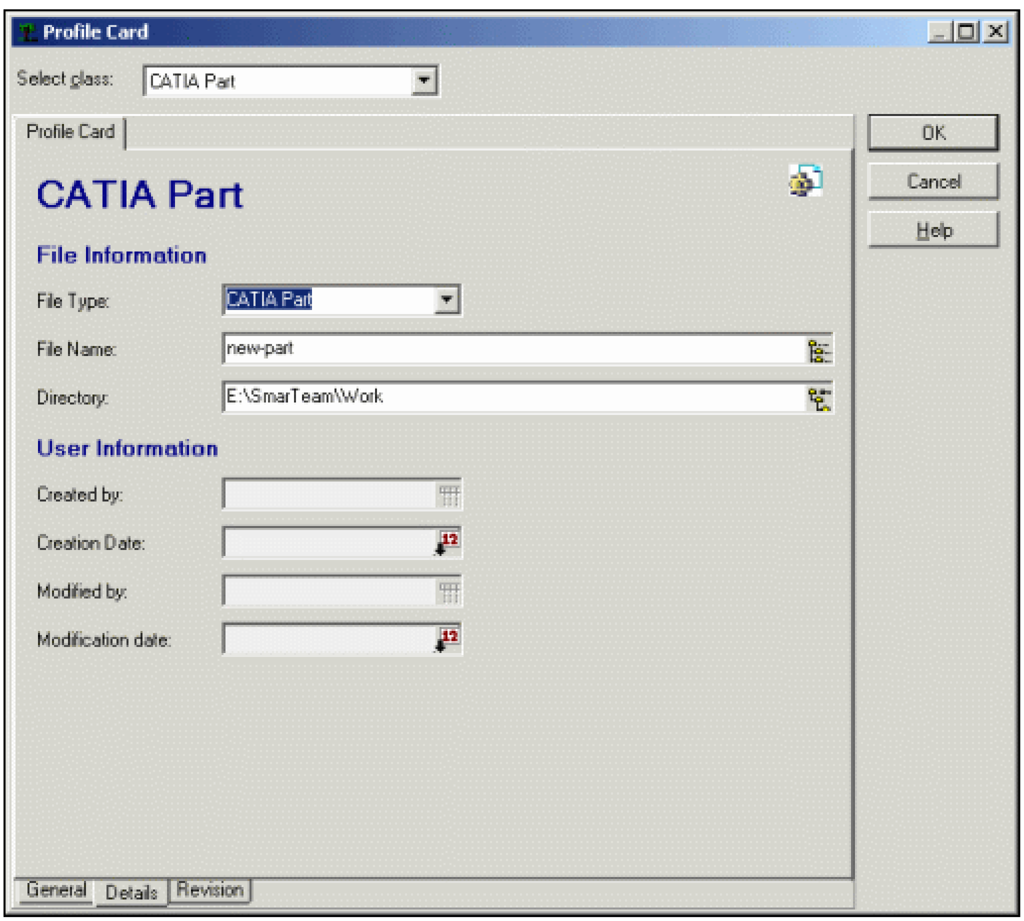
If you choose to work in **Batch Mode Save** mode (by checking the **Batch Mode Save** option from the SMARTEAM menu), a new Profile Card is not displayed. Instead, the Part is saved in the SMARTEAM database with the default attributes. You can update the Profile Card attributes at any time.

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.



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

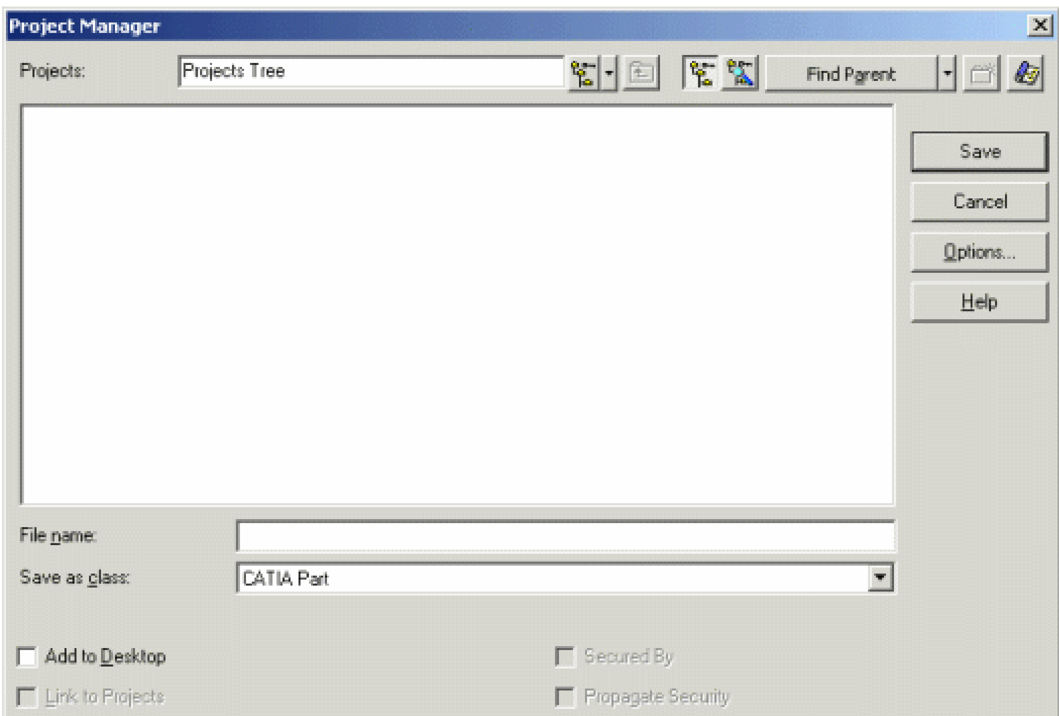


SMARTEAM enables you to define the following information in the Profile Card:

- 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.

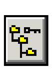
Saving a Part for the First Time

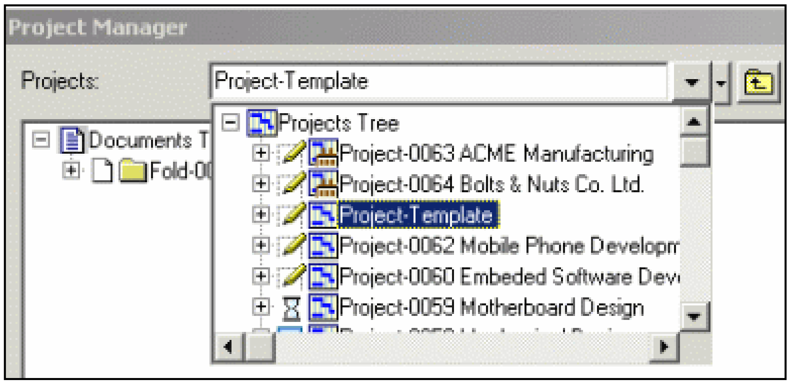
1. In CATIA V5, create a new Part.
2. From the **SMARTEAM** menu, choose **Save**.
   A Project Manager dialog box appears. In this window you can define the project and the parent folder to which the document belongs.



   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.

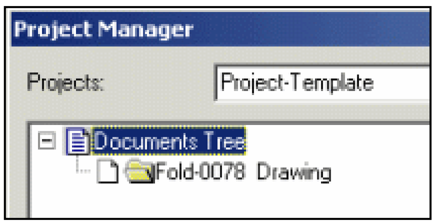
   If you leave the file name field empty, when clicking **Save** an automatic file name will be generated.

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.



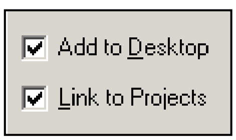
4. Choose a parent folder from the Document Tree. The Part will be saved as a child of the selected parent folder.

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



6. Check the **Add to Desktop** checkbox.

7. Check the **Link to Projects** checkbox to link the Part directly to a project.



8. 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.

9. 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.

10. Select the tabs to review default information about the file.

11. 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.

Saving a Part After Modifications

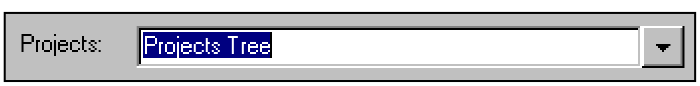
After a Part is added to the SMARTEAM database, you are likely to modify the Part in CATIA many times. Each time you modify the Part, you must save it using one of the **SMARTEAM Save** options.

- **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 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**.

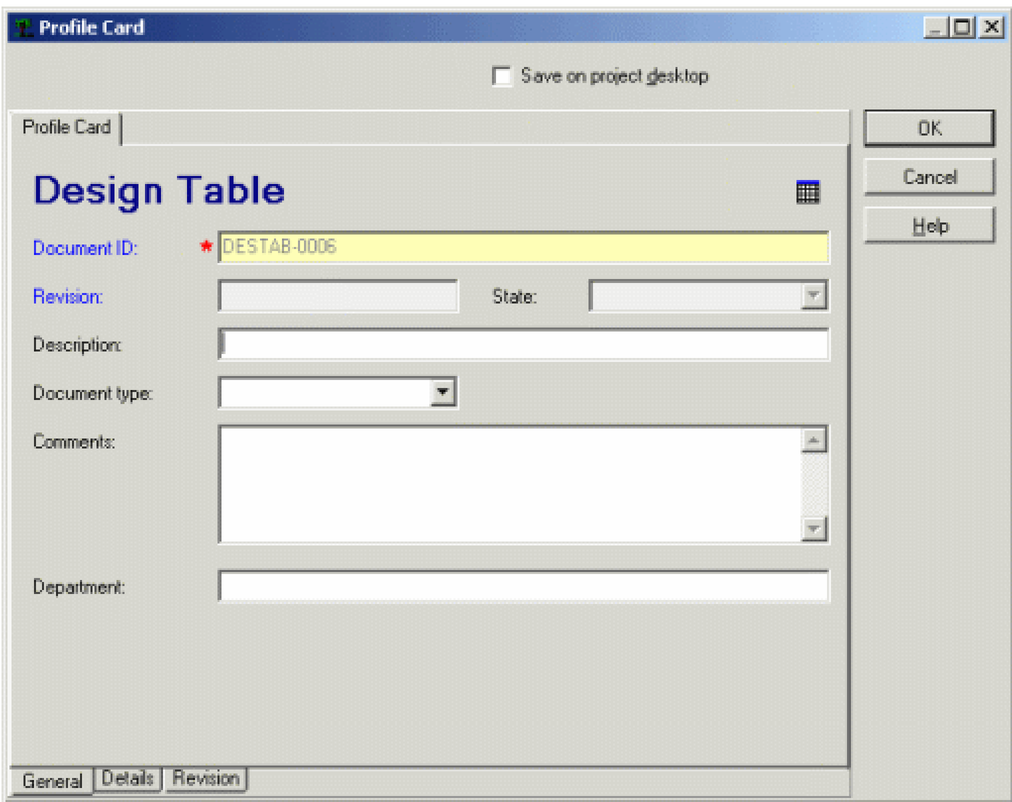
   This declares the two documents i.e. the Part and the Design Table, and saves them in the database. The Project Manager dialog box appears.

5. Click **Save**.

   The Profile Card dialog box appears for the Part.

6. Click **OK**.

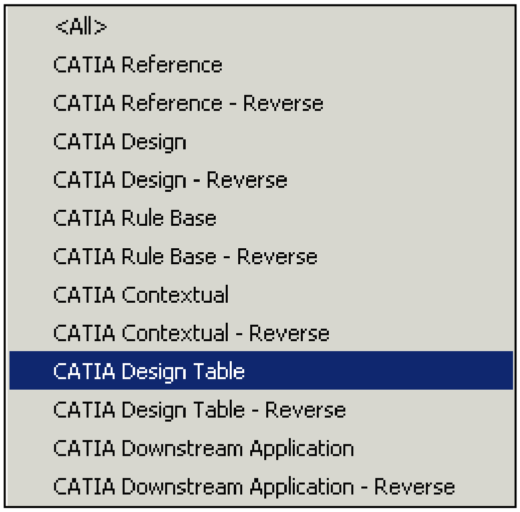
   The Profile Card dialog box now shows information specific to the Design Table:



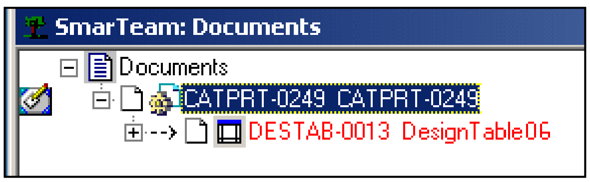
7. Click **OK**.

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:



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.

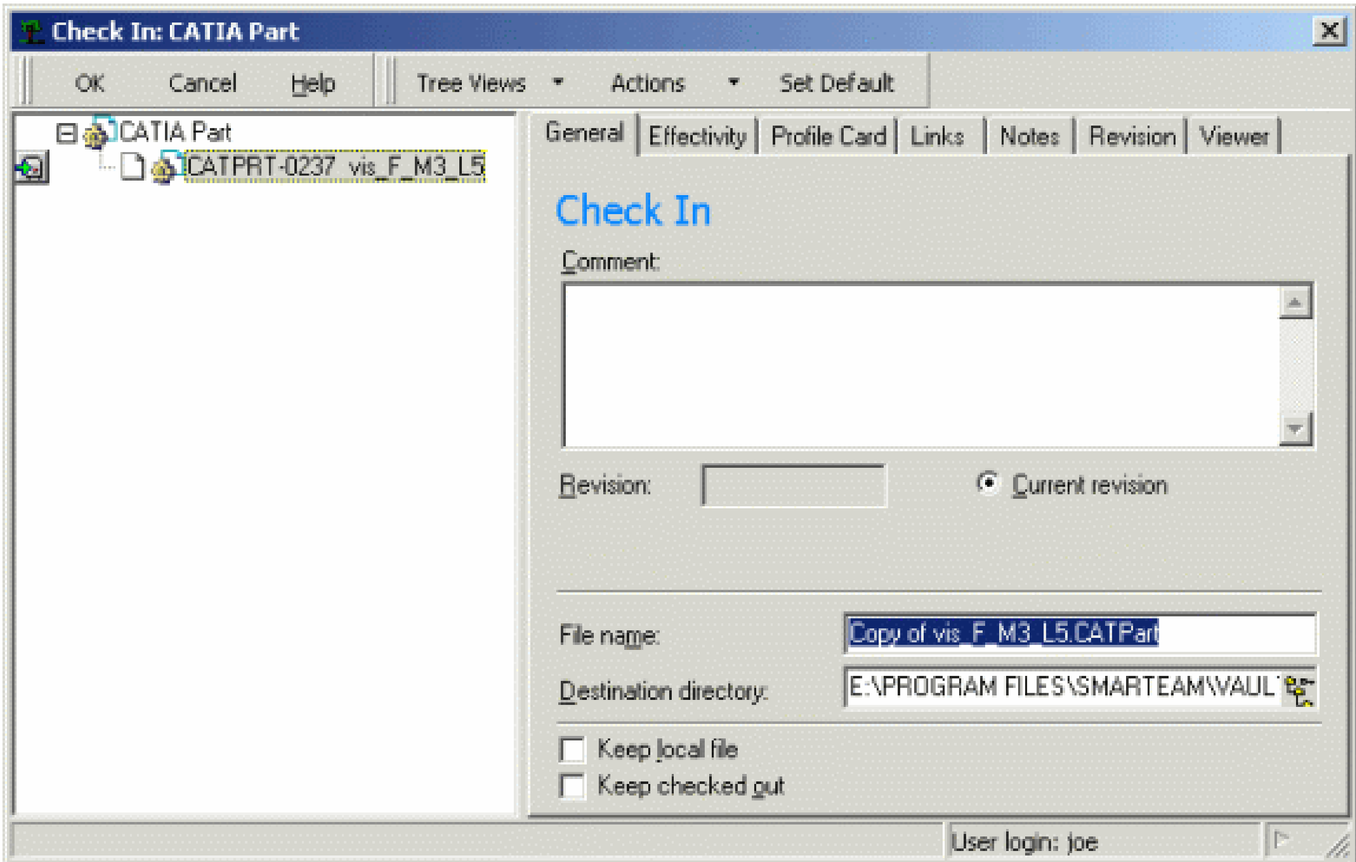
Checking In a Part

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.
Comments	You may enter a comment in this field.
Set Default	Click this option to assign the same registration information for all document revisions such as phase, effective dates, and notes.
Effectivity tab	
Phase	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.
Effective From / Effective Until	Click the **Date** 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.

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

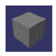
This section discusses the following topics:

- Saving Properties in SMARTEAM
- Updating a CATIA Property Following a Change in SMARTEAM

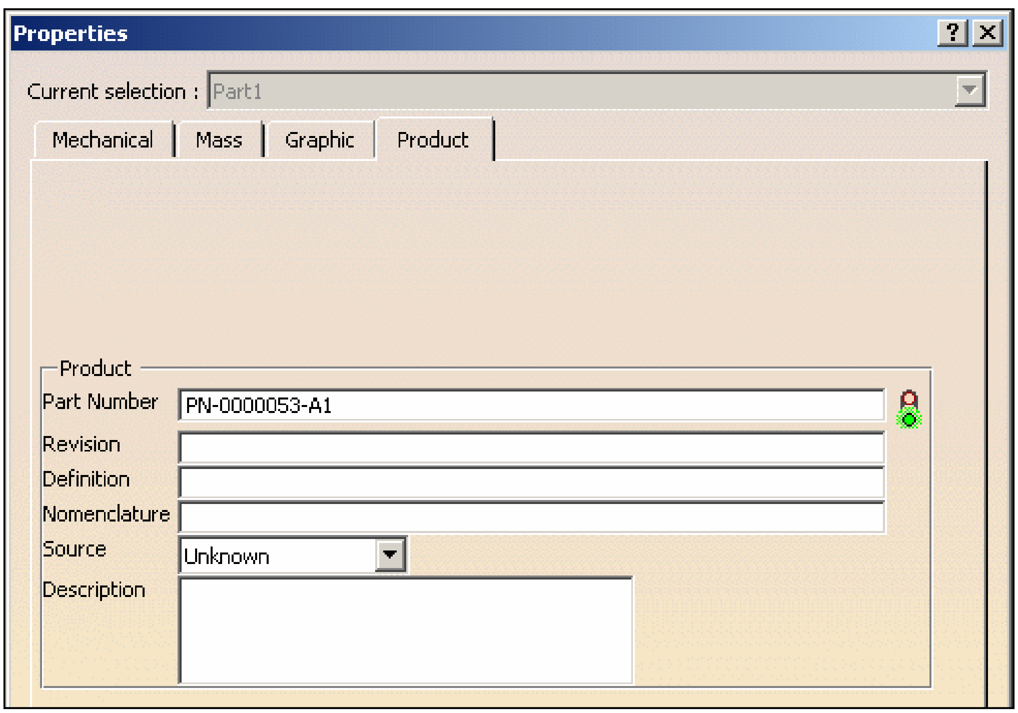
Prior to performing the scenarios, some properties will already have been mapped in CATIA and SMARTEAM by the administrator. This is described in Defining Property Mapping.

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.

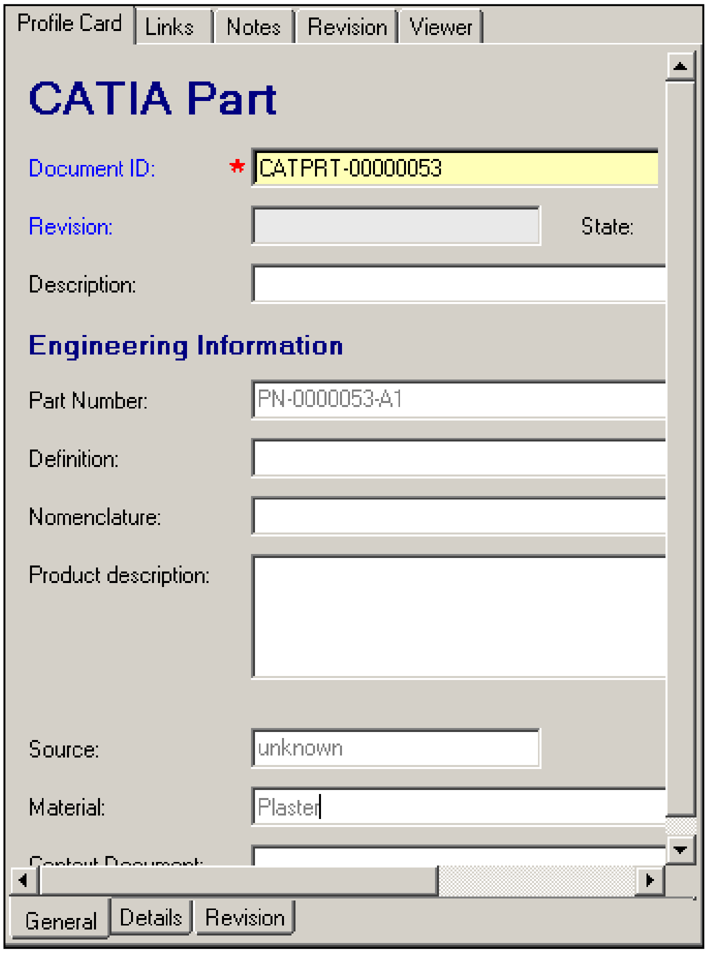
1. Select the Part in your CATIA session.
2. Click on the **Apply Materials**  icon to display the Library dialog box.
3. For example, click on the **Plaster** item  and click **OK**.

   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:



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:



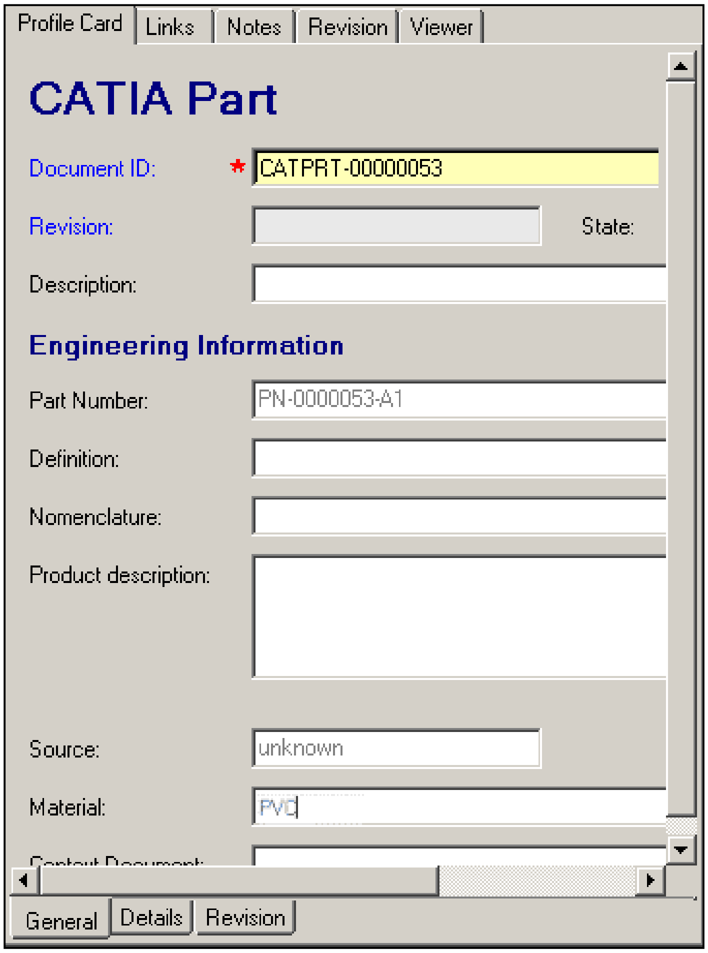
Should you subsequently modify any CATIA properties you can update the database using **SMARTEAM->Properties->Save in Database**.

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 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	• **SMARTEAM->Save** (the **Save** icon ) or • **SMARTEAM->Properties->Save in Database**
from SMARTEAM to CATIA	**SMARTEAM->Properties->Load from Database**

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:

- 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.
- **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.
    This operation can only be performed on a document in *Checked In* status. You are free to use any one of the document's revisions.
    Be aware that the revision is replaced at **all** tree locations.
    From V5R15 onward, **Replace with Selected Revision** is available from contextual menus.
- 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.

This section contains the following tasks:

Building an Assembly

This task shows you how to use the **Assembly Management** commands. These enable you to insert existing Parts/Assemblies as components of the current Assembly and to replace an existing Part or Product or a component of the current Product.

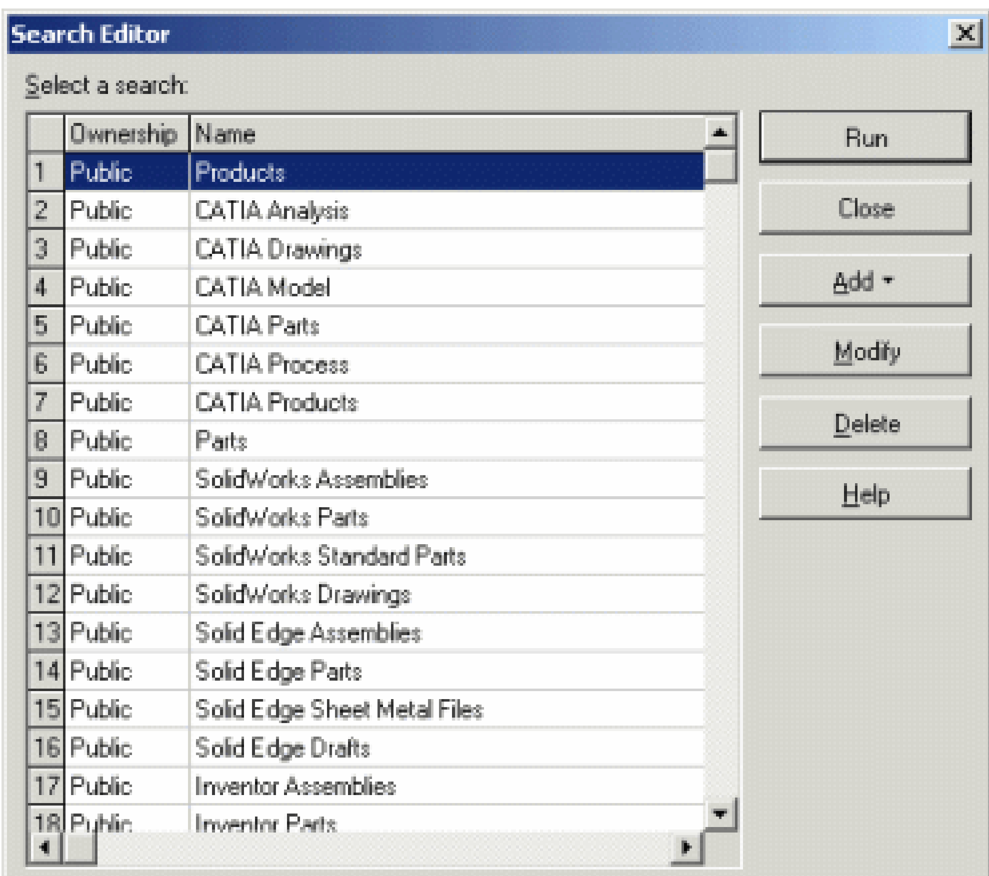
Inserting a Component

1. Display an Assembly structure in CATIA V5.

2. Click the **Insert Component** icon .

2. 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.

Replacing a Component

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

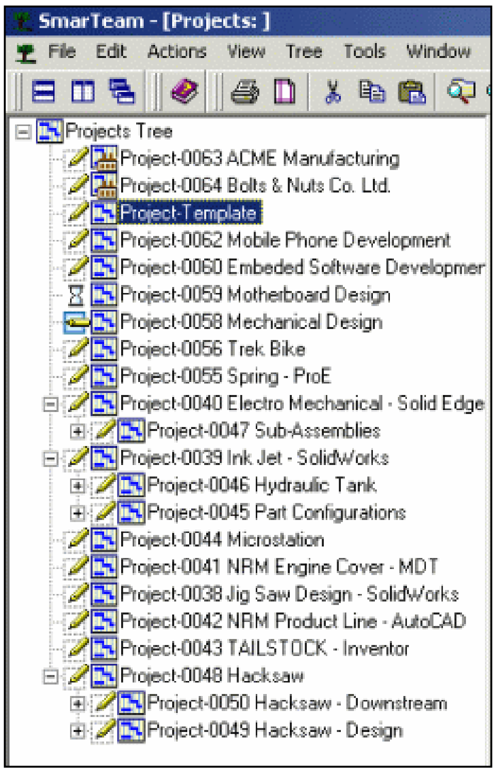
Adding a New Assembly to SMARTEAM

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 SMARTEAM 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.

Since the process of saving a Part is identical to that of saving an Assembly, refer to 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**.

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.

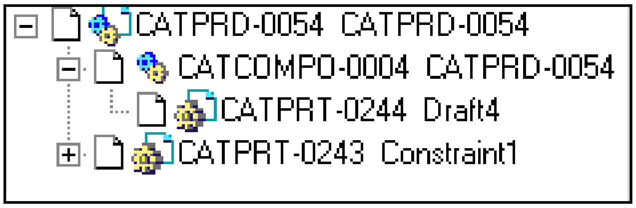
   SMARTEAM saves all the components but does not display a Profile Card for each one.

Or

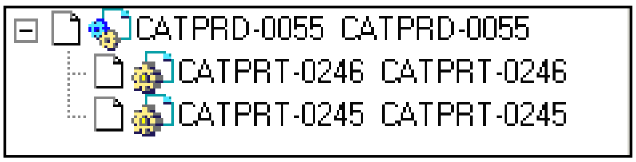
1. Select **SMARTEAM ->Save As**.
2. In the Save As window, click the ***Options*** tab to display the Save Option dialog box.
3. In the **Save** tab, check the checkboxes
4. Click **OK**.

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, refer to 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 *C*heck 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

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**.

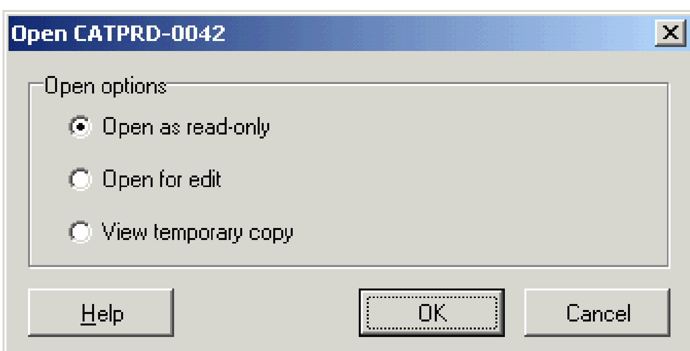
At your desktop, you can view the assembly and its components. Those documents that were checked out may be modified. After you modify the document, remember to apply the **SMARTEAM->Save** command.

Checking Out an Assembly from a SMARTEAM Dialog Box

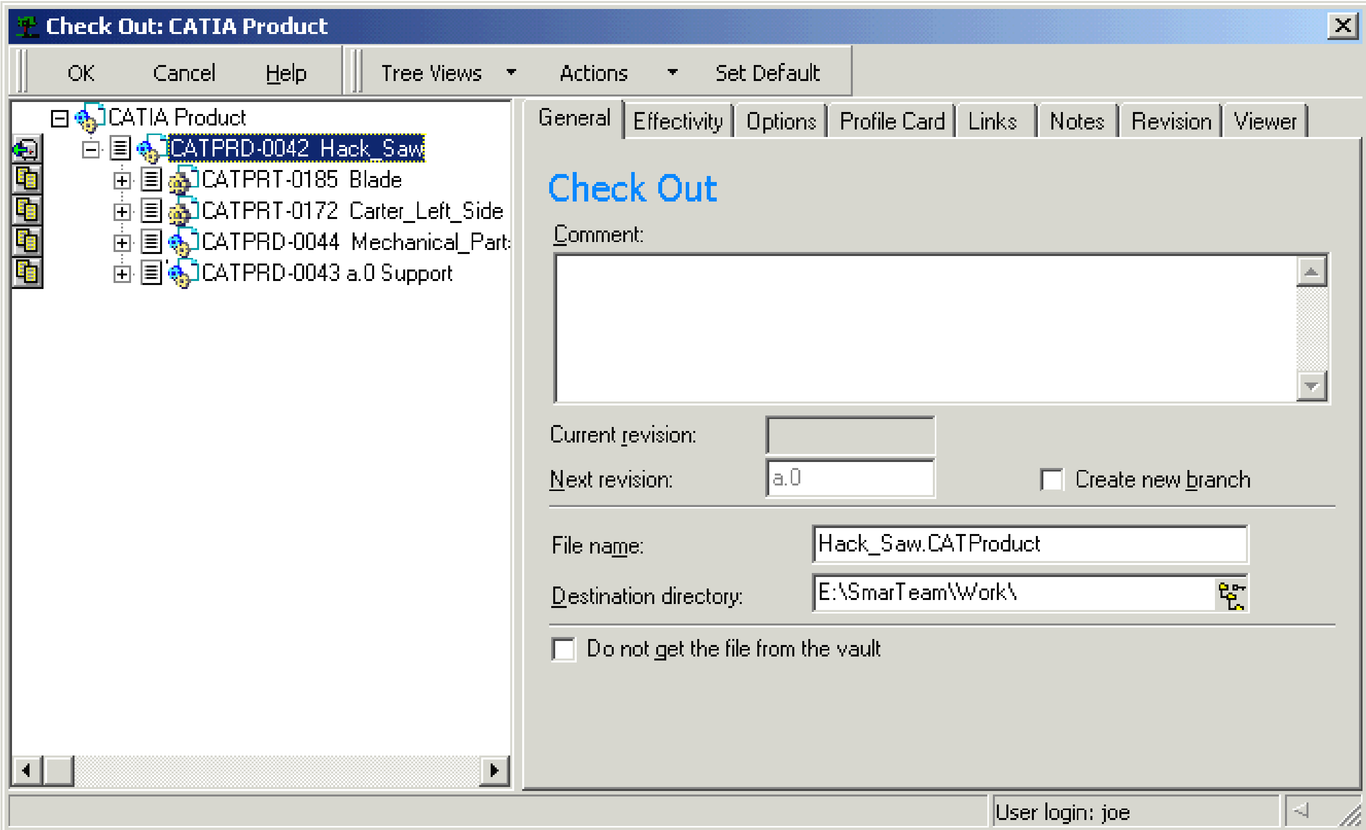
If an Assembly is checked in to a vault, and it is not currently displayed at your desktop (in read-only mode), you must locate the Assembly and then select **File Operation-> Open For...** contextual command to launch it into CATIA and modify it.

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. Refer to 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

5. 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. For more information, refer to Life Cycle Options.

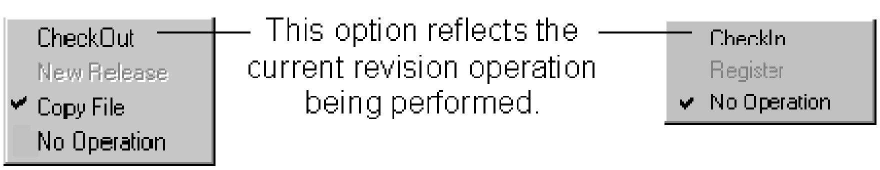
Checking out Components Individually

5. 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.

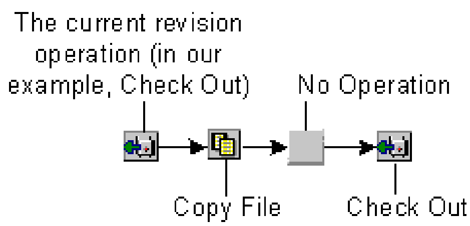
     This list differs for each life cycle operation, as shown below:



   Or, more simply,

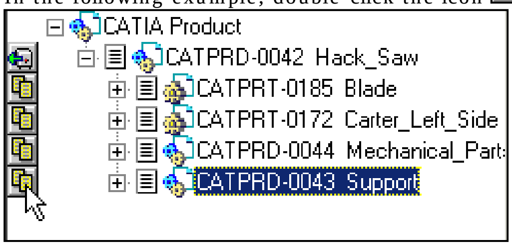
   - Click on the icon of the document to choose an operation.

     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.

6. 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.

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.

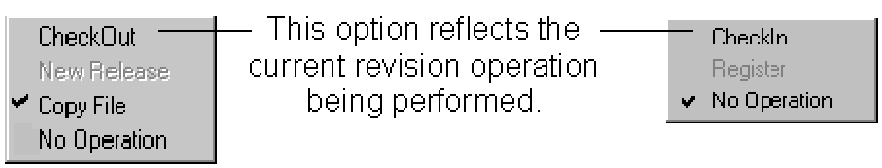
Checking in Components Individually

1. To handle each component individually:

   Either:

   - Click on the icon of a document to display a list of options.

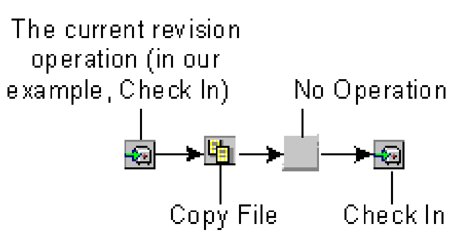
     This list differs for each life cycle operation, as shown below:



   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**
Associated Objects	Enables you to view those documents that are linked to the selected document. You can then perform lifecycle operations on these Associated Objects. See Managing the Revisions of a Drawing for a complete description.	In SMARTEAM, a Drawing (**Stump Preacher Guitar Drawing**) was created based on a **Stump Preacher Guitar Drawing**. When you check out the **Stump Preacher Guitar Drawing** you can view and manage its link (the **Stump Preacher Guitar Drawing**).
Propagate Operation	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.
Relatives Checked Out	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 *Relatives Being Modified* 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 *Show Parent* 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 **Switch to Latest Revision** 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.

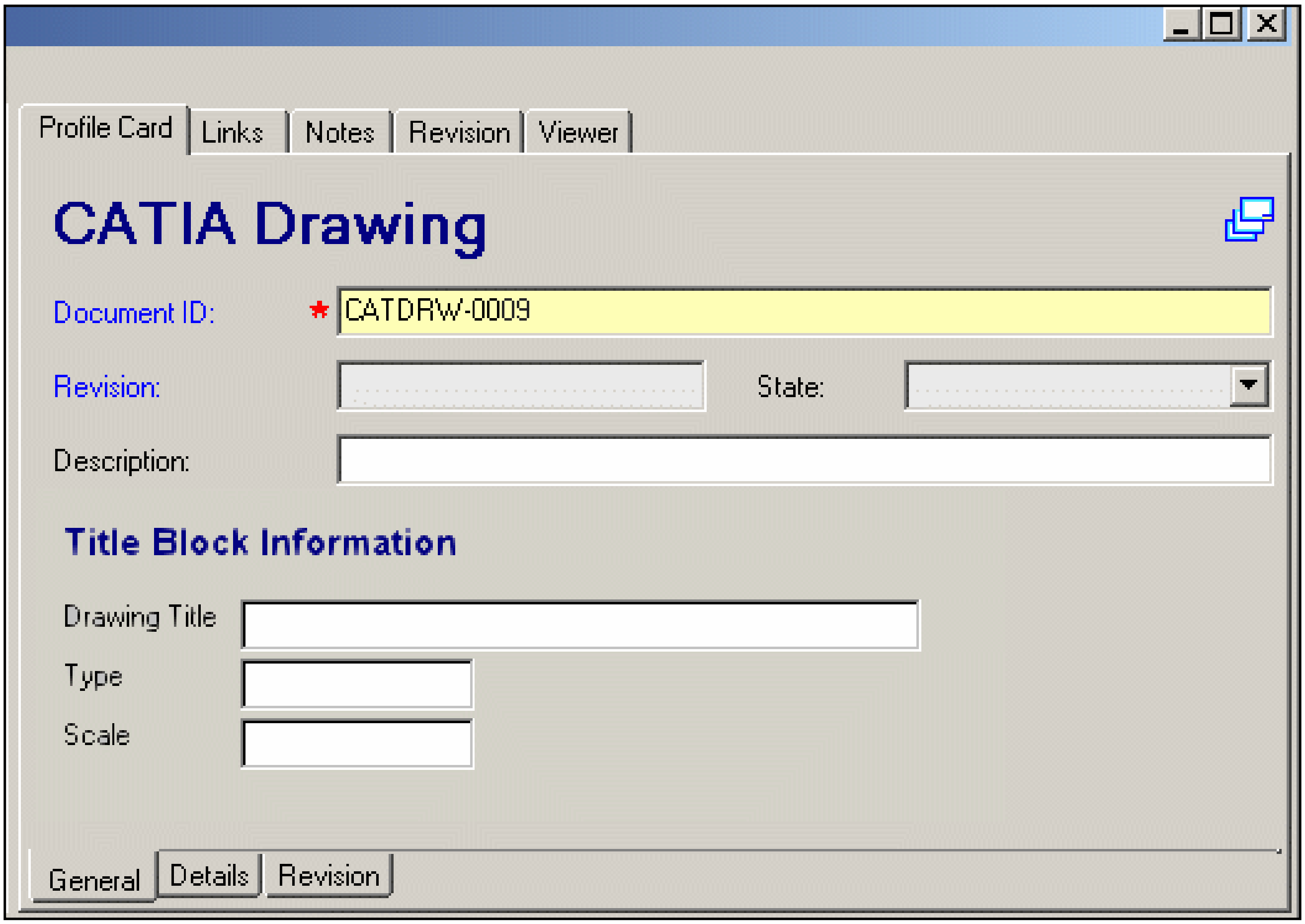
This section contains the following tasks:

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.



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.

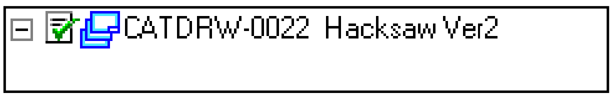
   In this way, SMARTEAM reflects the nature of the Drawing. As you manage the revisions of the Part/Assembly, you can manage the revisions of the Drawing in parallel.

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, refer to the SMARTEAM documentation.

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.

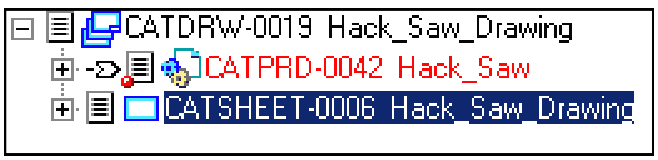
Viewing Associated Objects

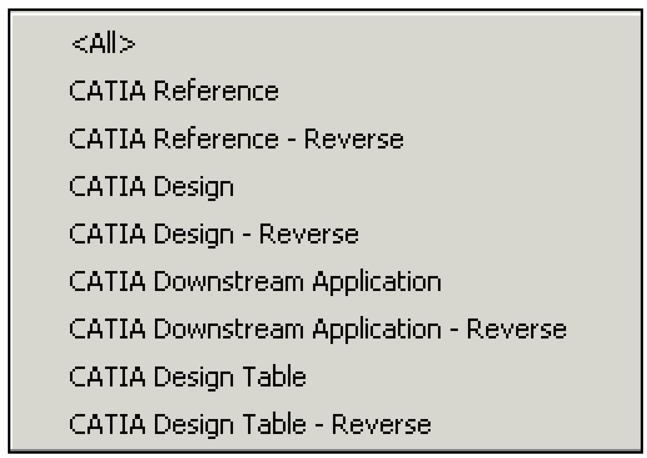
You can view the links of a document and choose a lifecycle operation for each associated document. If you do not view the links of a Drawing, the associated documents will automatically be checked out/in together with the Drawing (if that is the default setting).

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 .

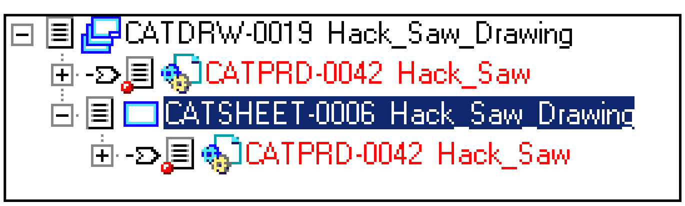
1. From any life cycle window (such as **Check In** or **Check Out**), right-click a drawing document and select **Associated Objects->CATIA Links** to display the list of view options which are as follows:





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.

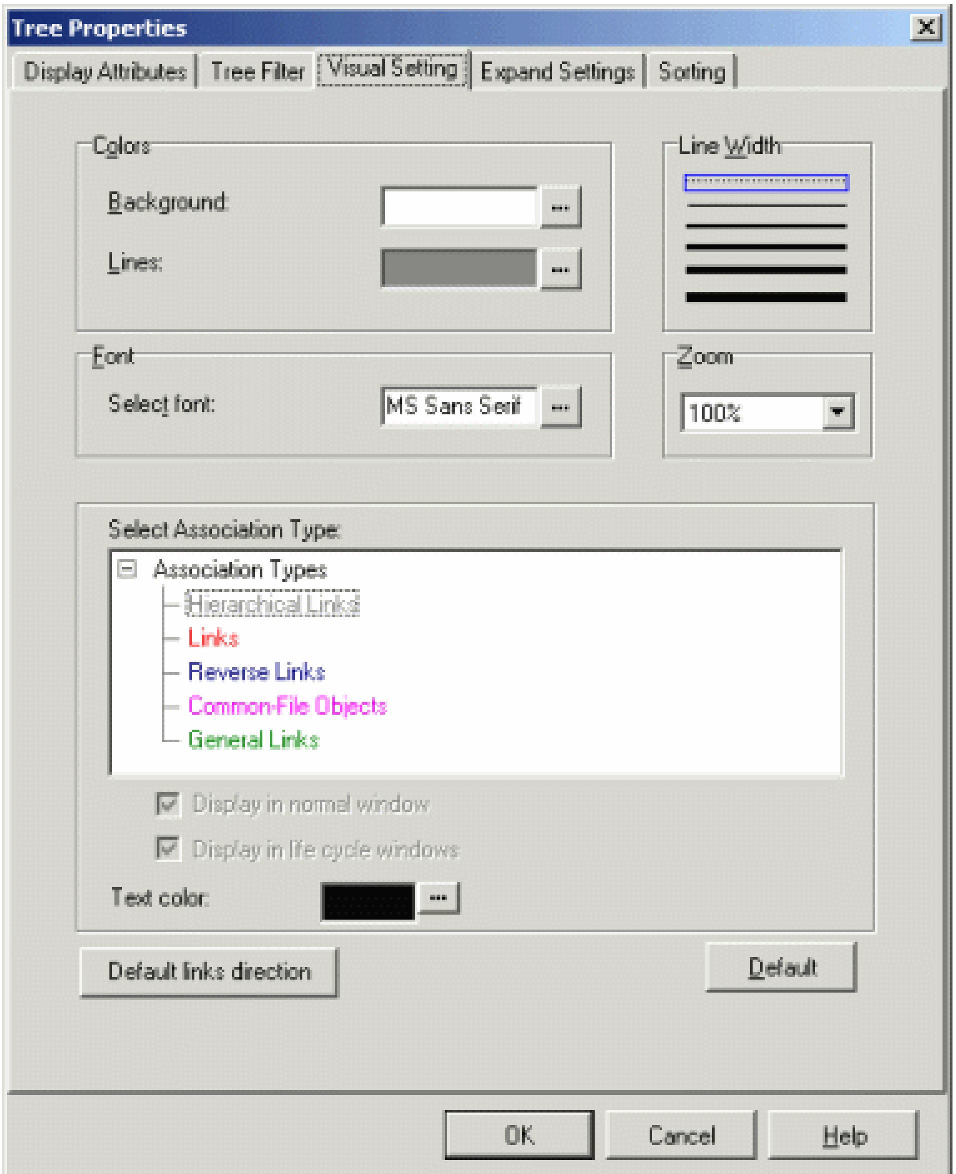
Customizing Color Settings for Associated Objects

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, refer to 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.

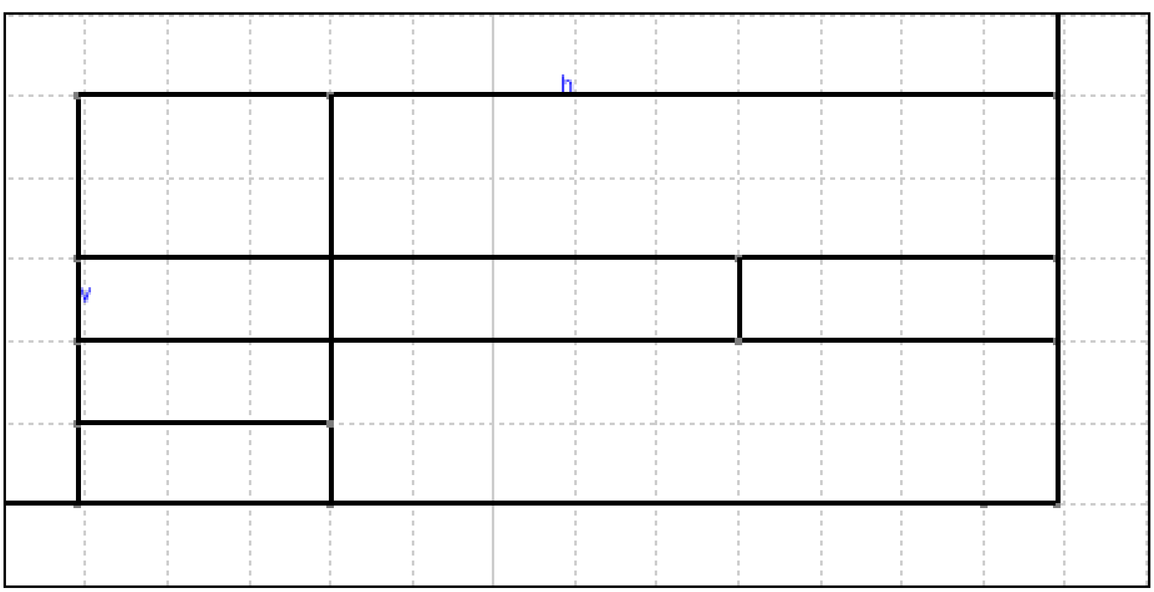
This task is divided into the following stages:

- 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.

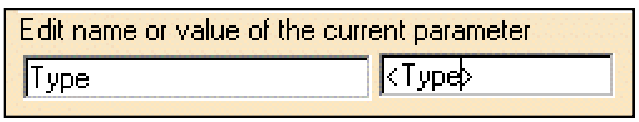
3. Click the **New parameter of type** button.

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.



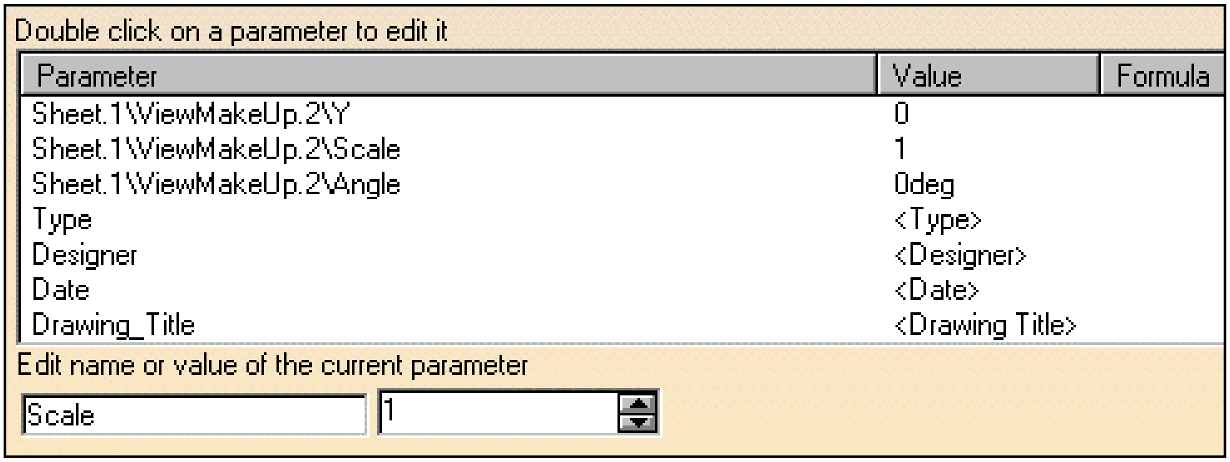
6. Click on the **Apply** button.

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

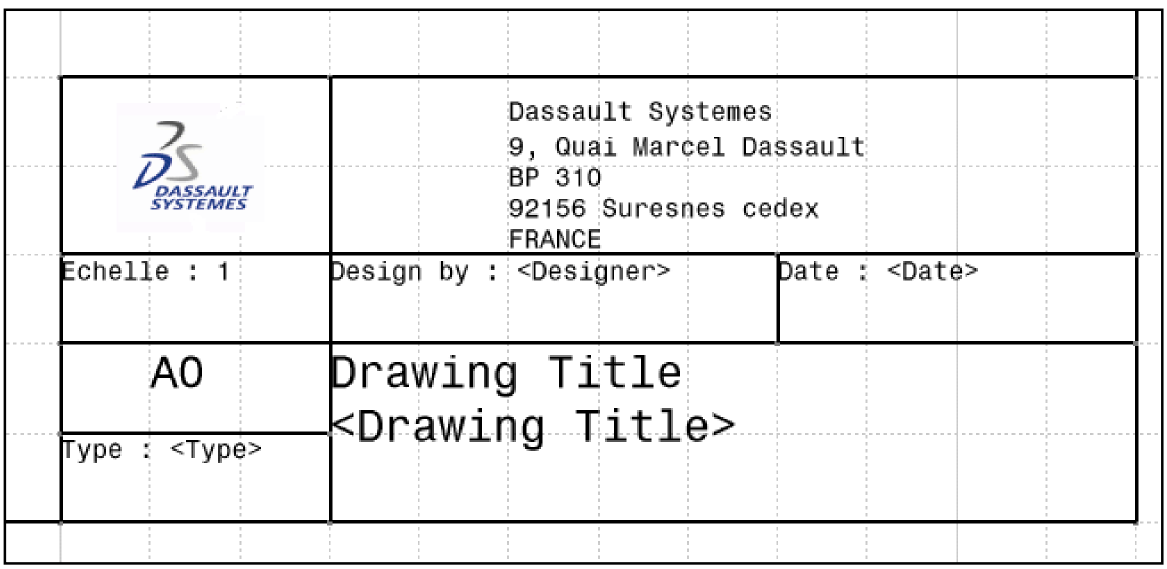
8. Locate the Scale property i.e. *Sheet1\ViewMakeUp.1\Scale* and rename it to **Scale** as shown below:



9. Click **OK**.

Creating the Text Attributes in the Title Block

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



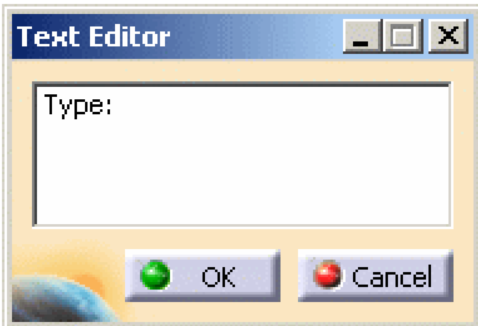
For more information see "Adding Attribute Link to Text" in the *CATIA - Interactive Drafting User's Guide*.

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:

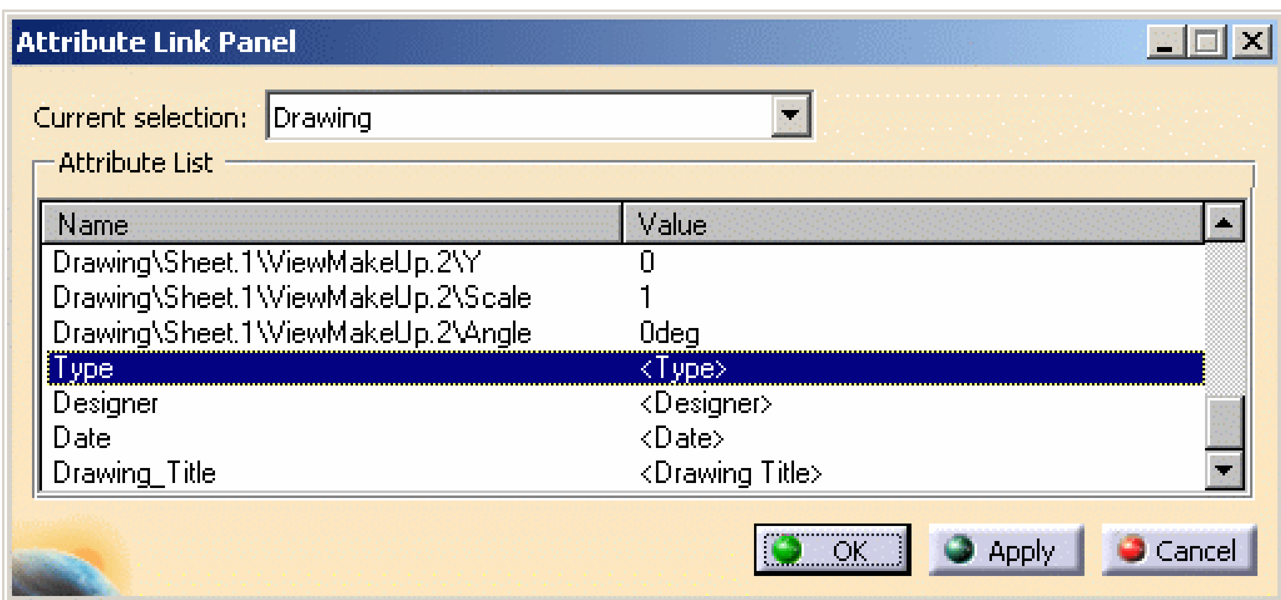


4. Right-click on the appropriate location in the title block and select **Attribute Links**.

5. In the specification tree of the drawing, click on the **Drawing**.

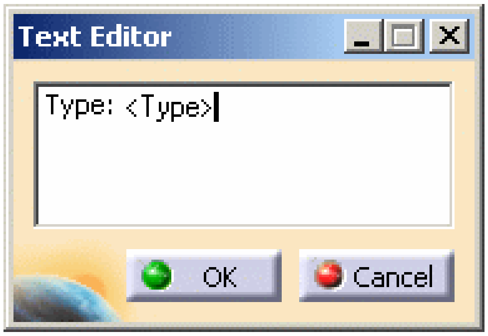
   The Attribute Link dialog box appears.

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:



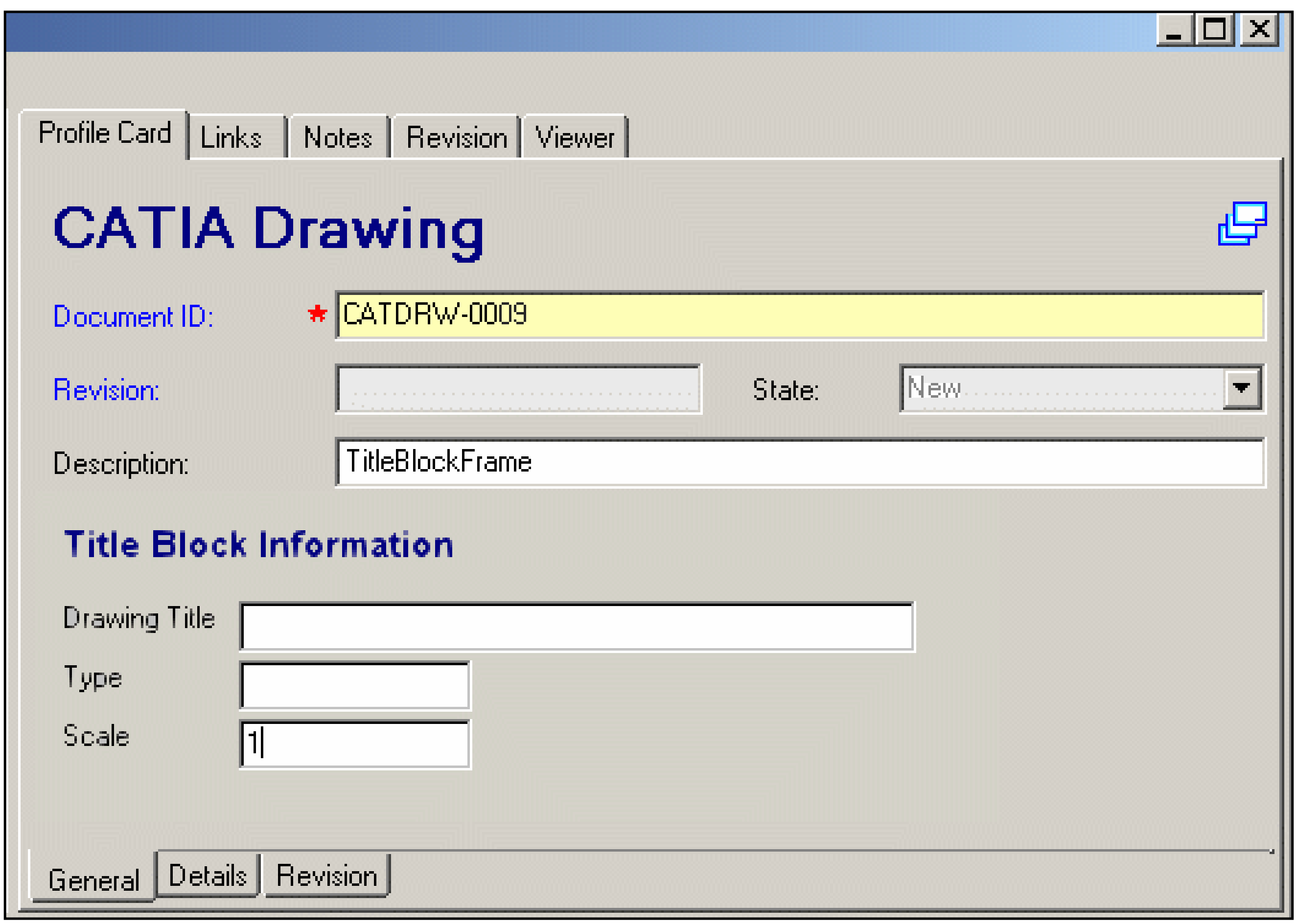
8. Click **OK**.

9. Repeat steps 1 through 8 for all texts.

Saving the Document in SMARTEAM

1. Click the **Save** icon or select **SMARTEAM->Save**.

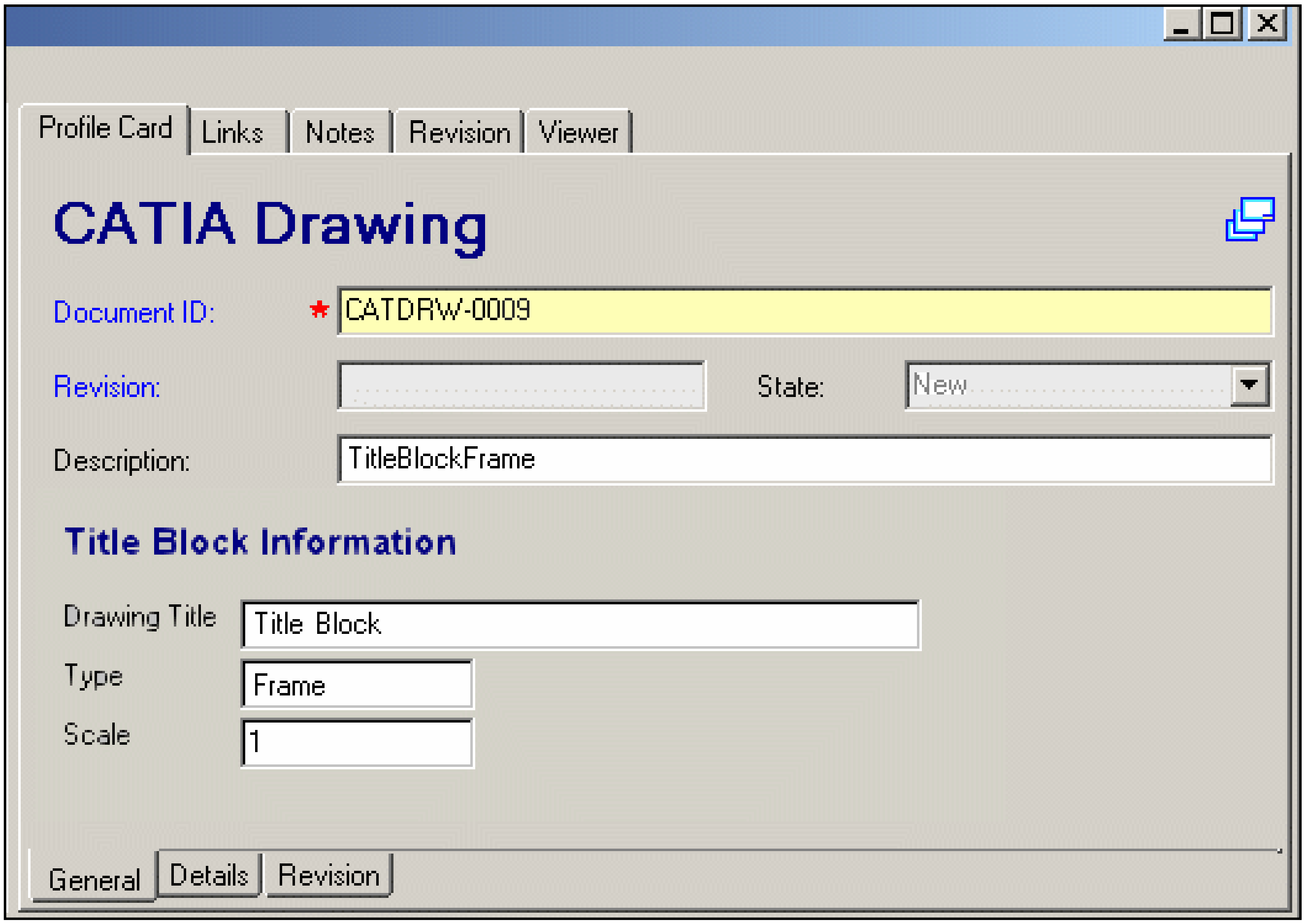
   This declares the document and saves its properties in the database. In the SMARTEAM profile card corresponding to the document, you can see that the fields are completed:



Updating a CATIA Property in SMARTEAM

1. Open the SMARTEAM: Documents dialog box.
2. Right-click on the document and select **Update**.

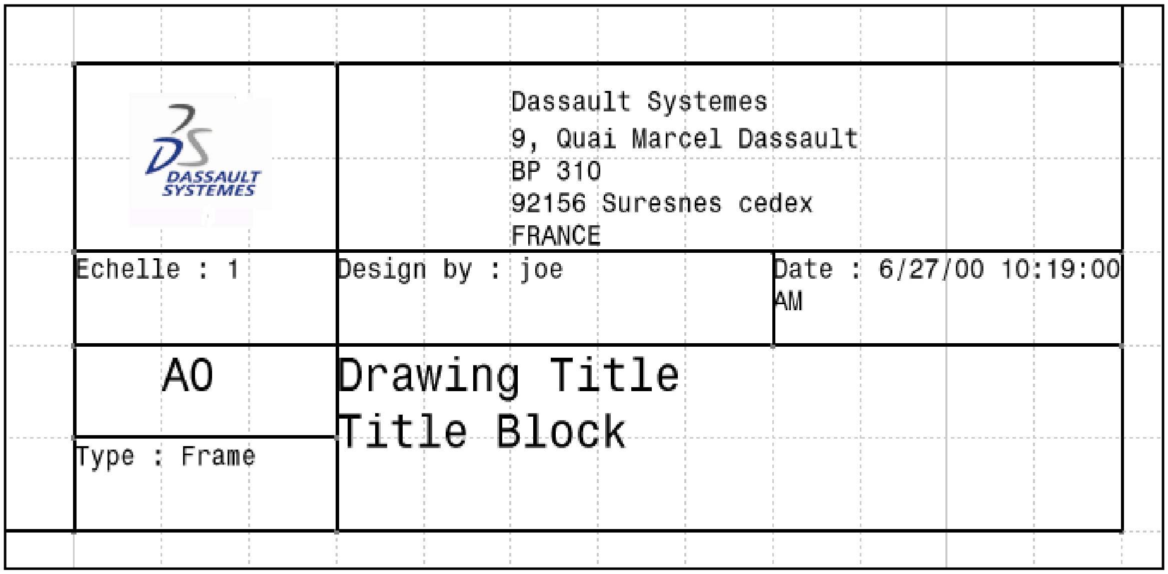
   The profile card is now ready to be changed.
3. Modify the values of Drawing Title and Type.



4. Click **OK**.

   The property has been modified in the SMARTEAM database.
5. Back in CATIA, select **SMARTEAM->Properties->Load from Database**.

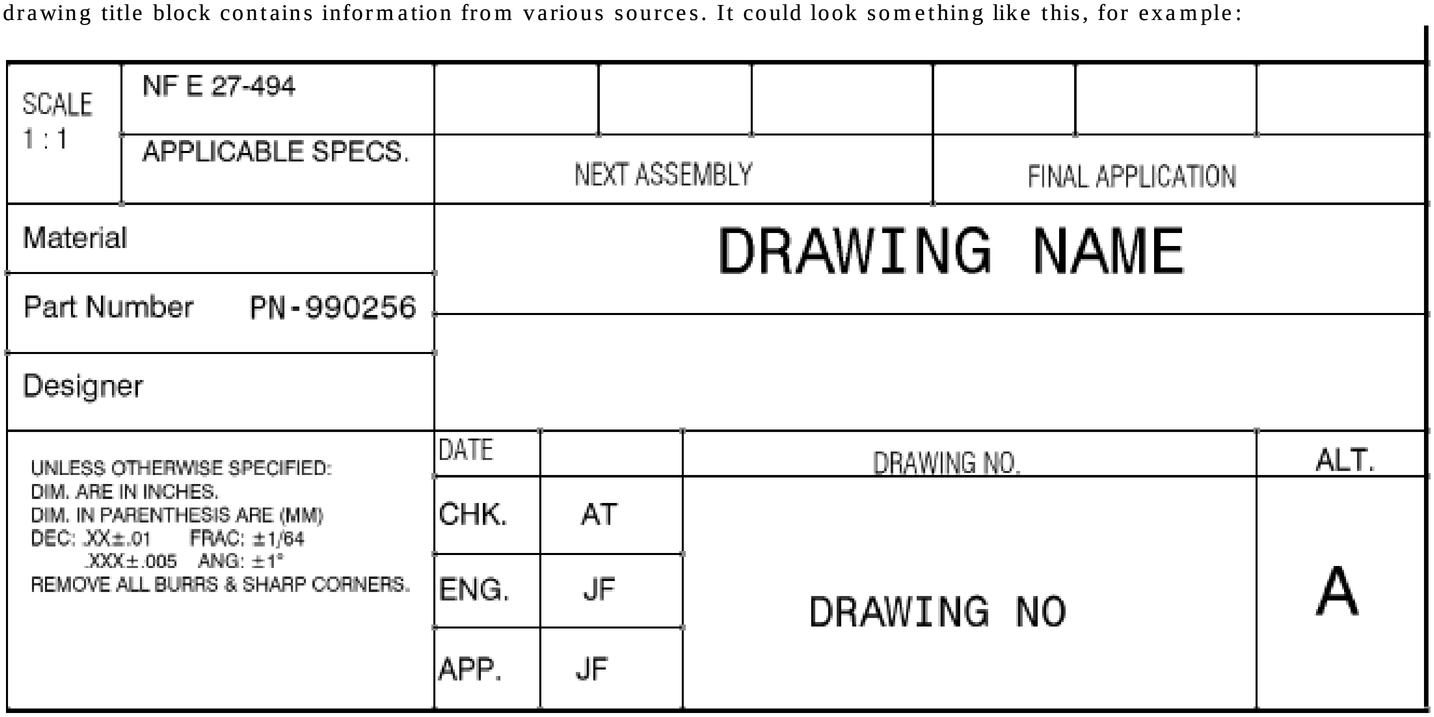
   Still in CATIA, you can now see the new values attached to the Drawing title block.


DASSAULT SYSTEMES
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
Title Block
Type : Frame

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:



- 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.

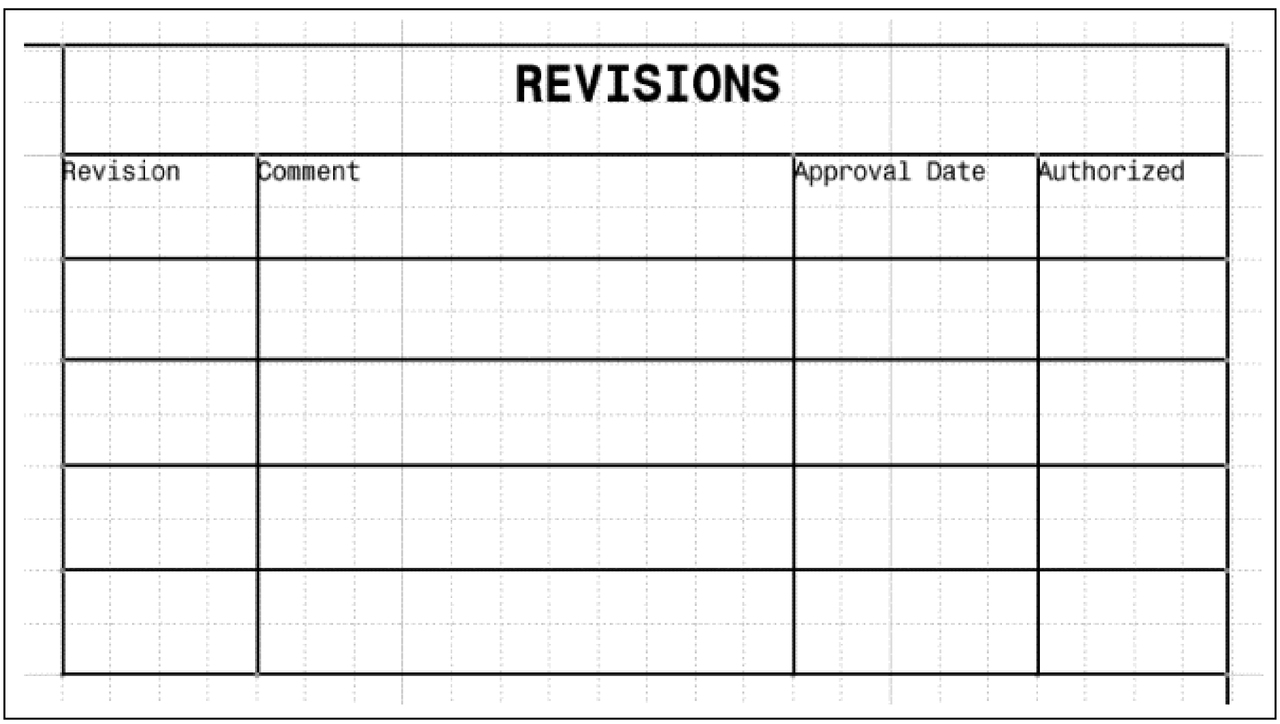
This task is divided into the following stages:

- 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
- Associating the *RevisionBlock.bs* Script with a SMARTEAM Event

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:



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.

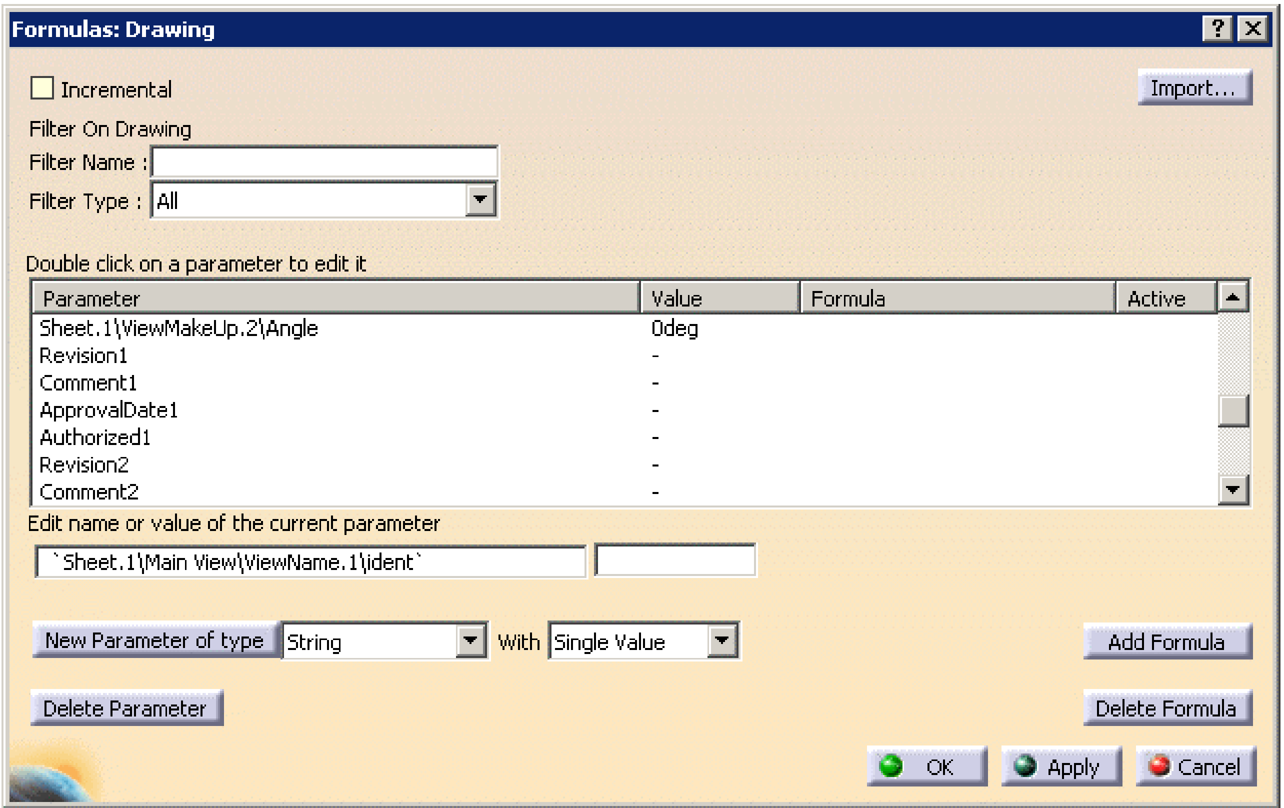
   You must now use the name defined in your mapping property. In our example we will use Revision, ApprovalDate, etc.

4. For example, change the name to *Revision1* 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. : Revision1 = e, Revision2 = d, Revision3 = c, and so on.
5. In the field on the right specifying the value of the current parameter, enter the "-" symbol.

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:

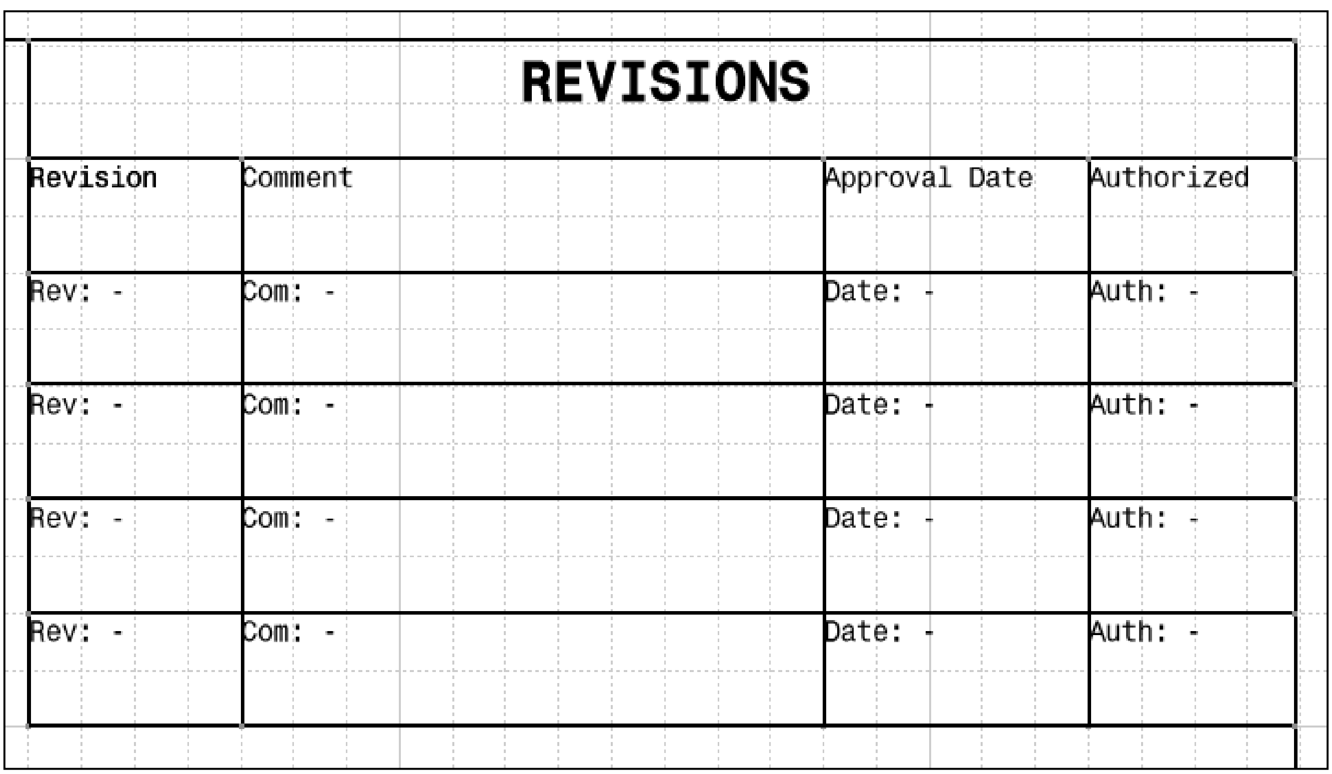
Property Name in CATIA	**Property Value**	**Property Type**
Revision2, Revision3, Revision4	-	String
Comment1, Comment2, Comment3, Comment4	-	String
ApprovalDate1, ApprovalDate2, ApprovalDate3, ApprovalDate4,	-	String
Authorized1, Authorized2, Authorized3, Authorized4	-	String



8. Click **OK.**

Creating the Text Attributes in the Revision Block

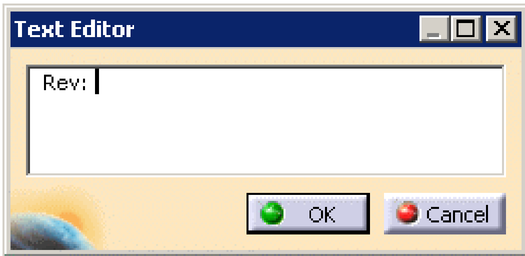
Here is what you will obtain at the end of this task:



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

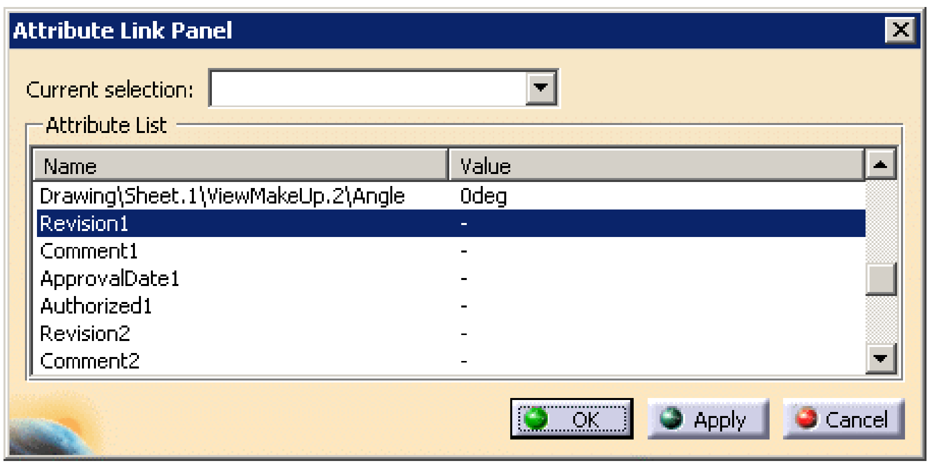
1. Select **Insert->Annotations->Texts->Text**.
2. Click inside the drawing to indicate where you want to display the drawing type. (If necessary, refer to the revision block design.)

   A text editor window appears.
3. Enter *Rev:* in this window:



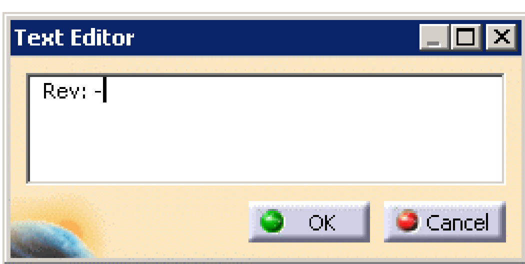
4. Right-click on the appropriate location in the revision block and select **Attribute Links** in the contextual menu.
5. In the specification tree of the drawing, click on the **Drawing**.

   The Attribute Link dialog box appears.
6. Select **Revision1** 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:



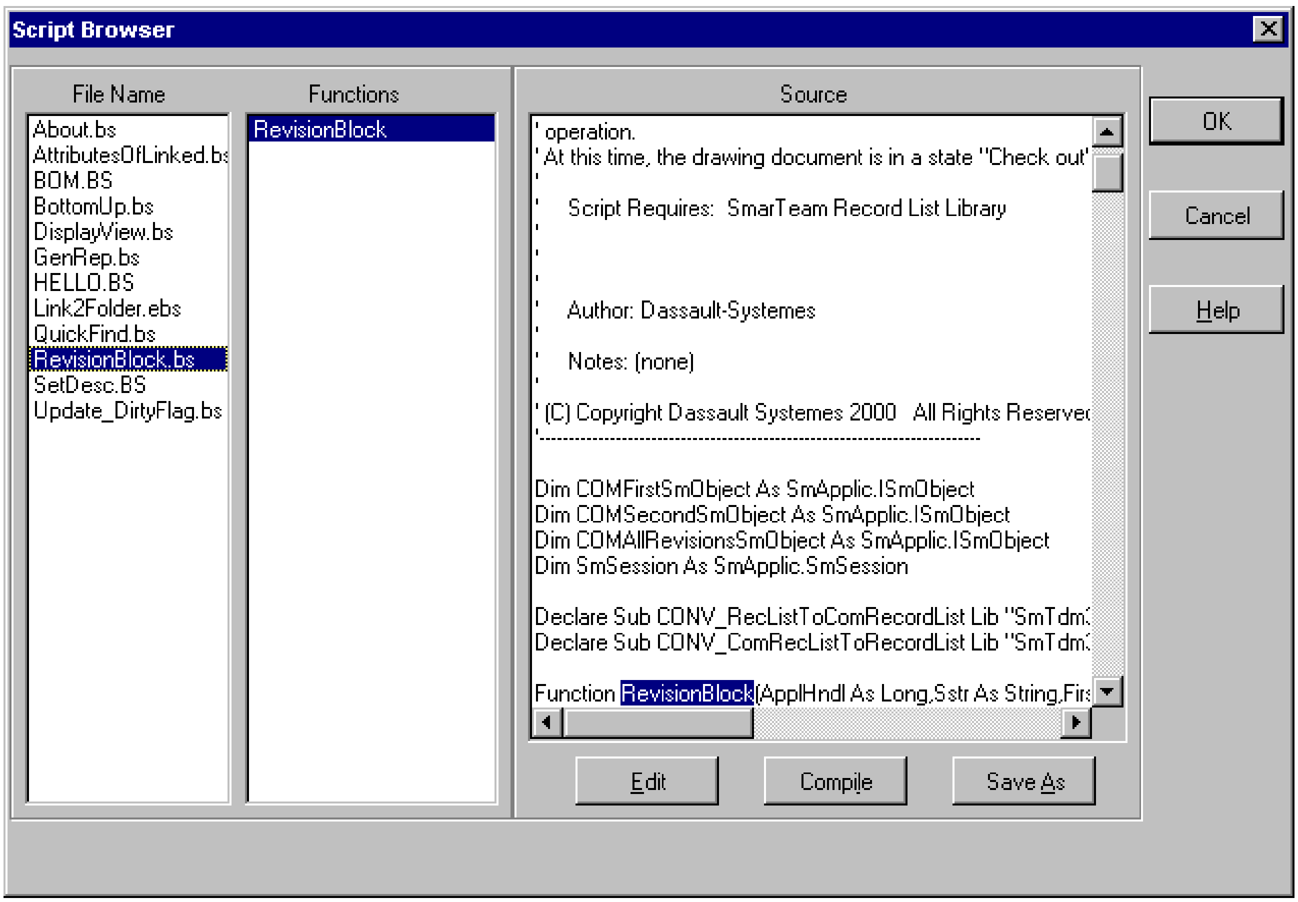
8. Click **OK**.
9. Repeat steps 1 through 8 for all the cells.

Associating the *RevisionBlock.bs* Script with a SMARTEAM Event

This step has to be done using the *SmartBasic Script Maintenance* utility.

1. Run the **SmartBasic Script Maintenance** utility.
2. Enter your login and password with administrative privileges. (Note that the **SmartBasic Script Maintenance** utility runs automatically on the default database.) If you want to modify a database other than the default one, you must first set it as the default database (using the **File->Switch to Database** menu item in the SMARTEAM application).
3. In the class tree, select the *CATIA Drawing* class then, on the right-hand side, double-click on the cell at the intersection of the **Life Cycle Stage 2 row** and the **Before** column.

   The Script Browser dialog box is now displayed.
4. Select **RevisionBlock.bs.**
5. Click on the **Compile** button.



6. Click on the **OK** button.
7. Back in the Script Maintenance dialog box, select **File->Exit**.

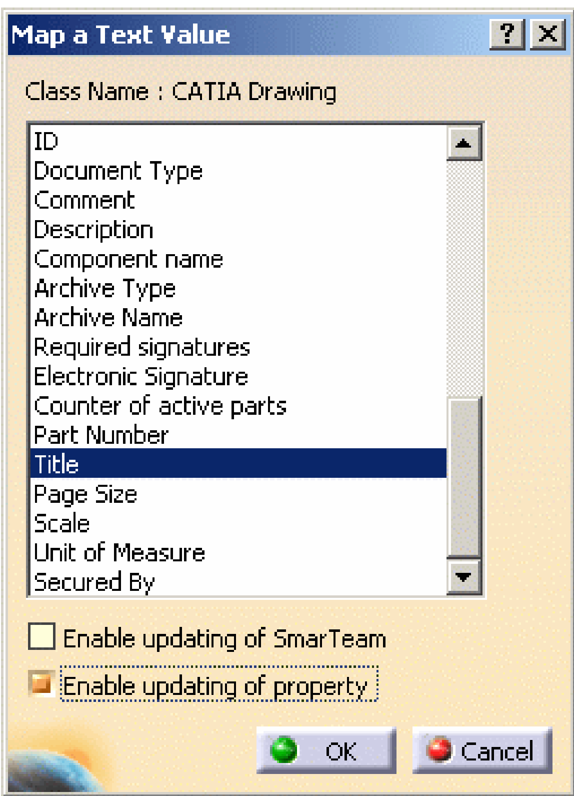
Displaying a CATIA Drawing SMARTEAM Attribute in a Title Block

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.

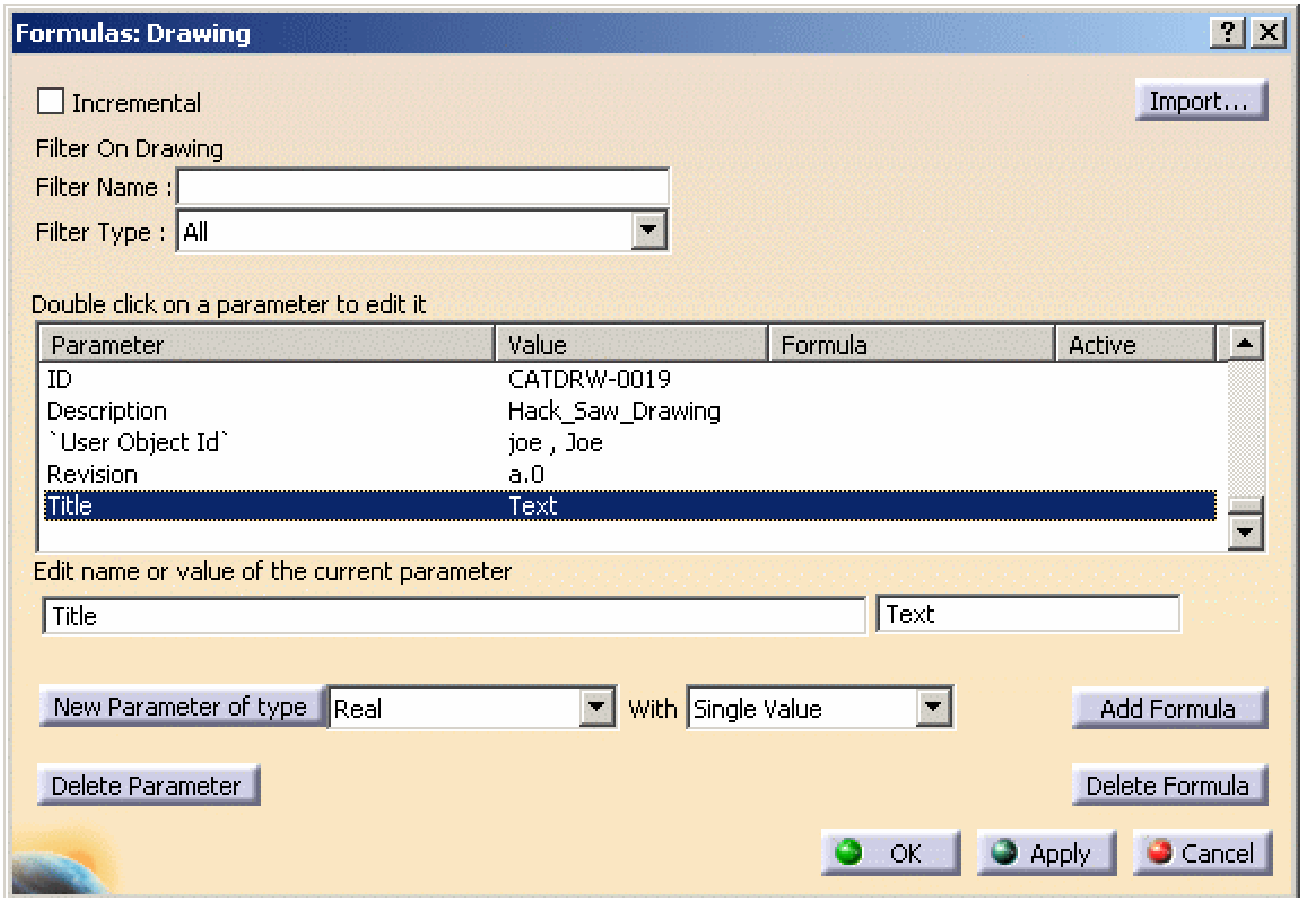
1. In the CATIA session, open a drawing document that has already been saved in the SMARTEAM database.
2. Select a text of the drawing.
3. Select **SMARTEAM->Properties->Map a Text Value...**

   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:

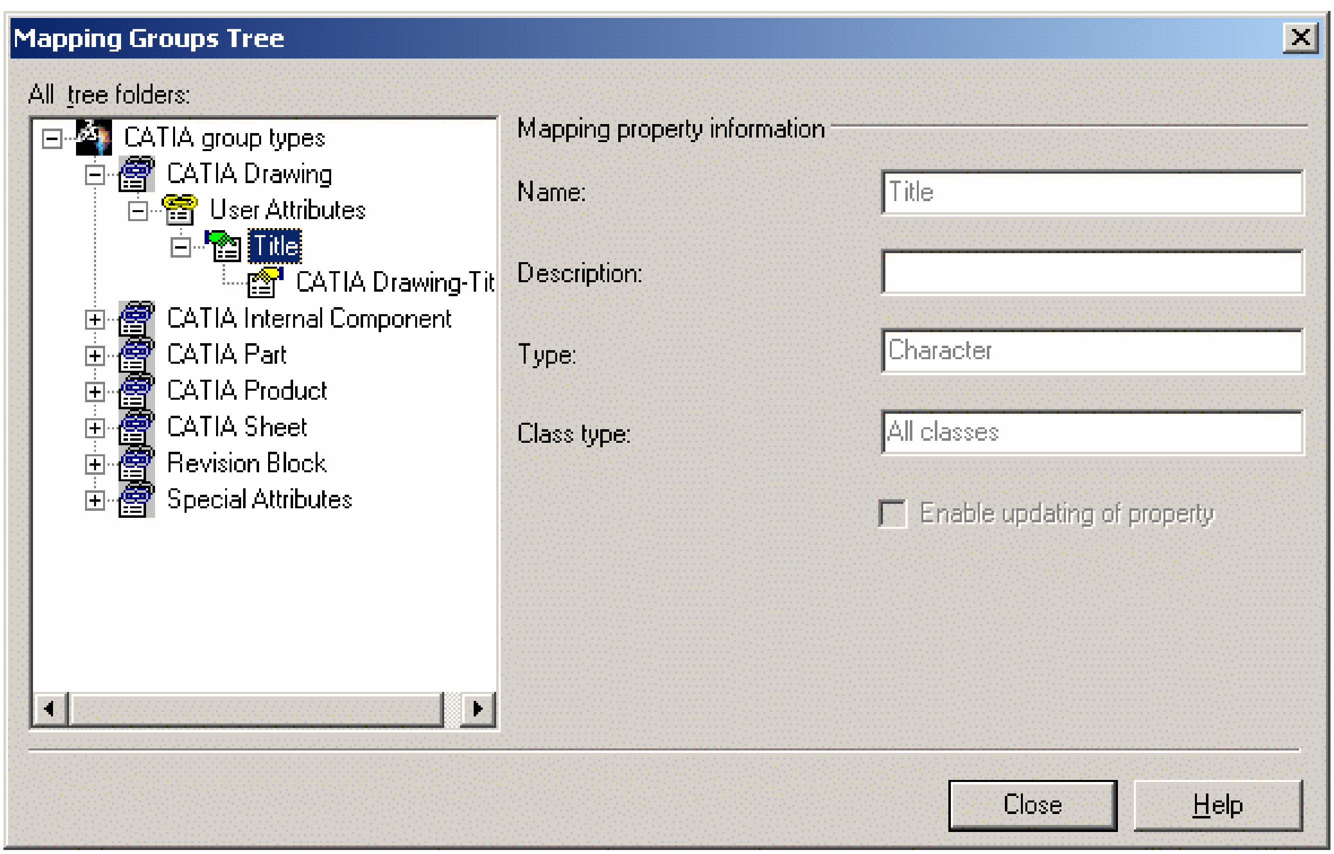
   

   The Map a Text Value dialog box shows the current mapping for the selected text. In other words, if you have already used the **Map a Text Value...** command for the same text, the attribute name and the update direction(s) you originally selected are kept.
6. Click **OK**.

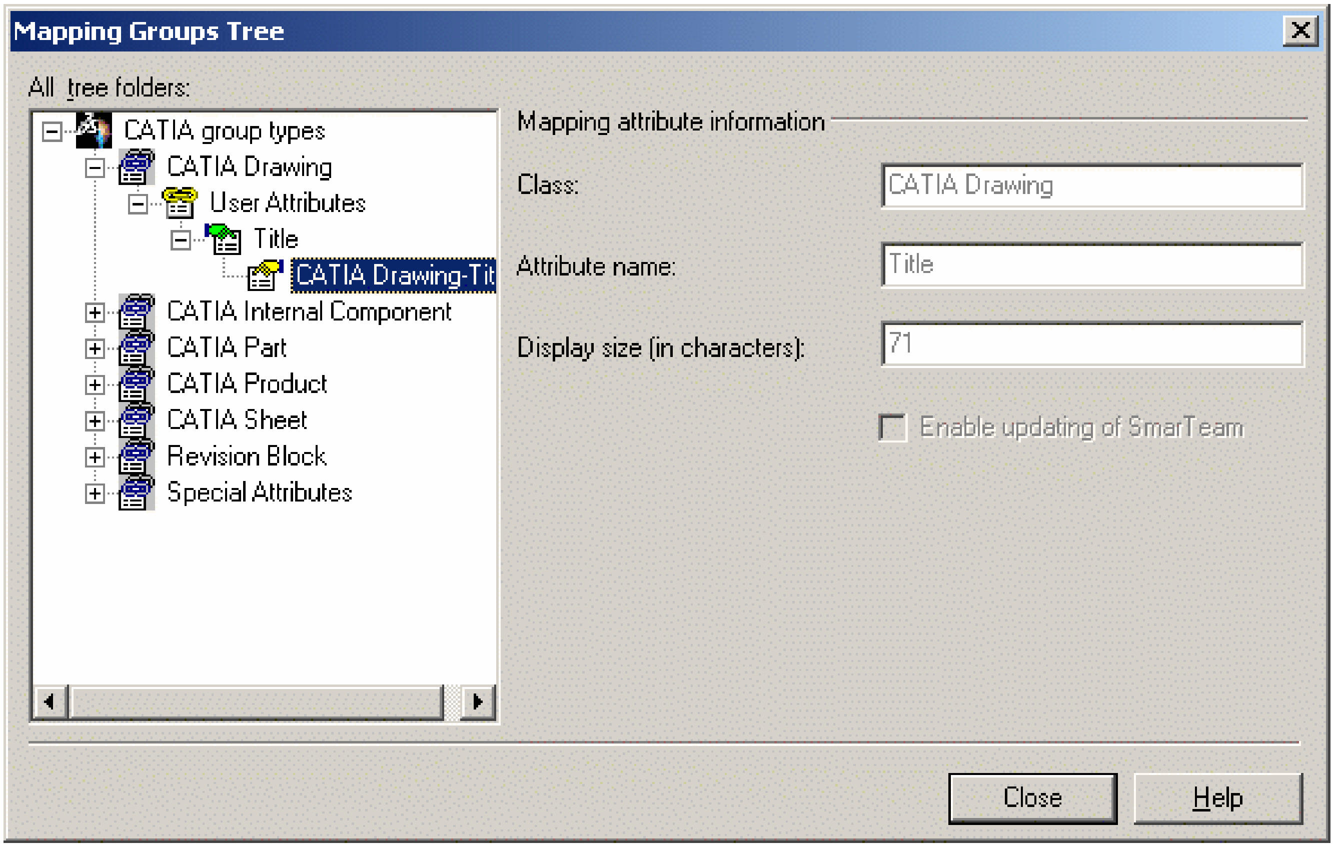
   A CATIA property is created inside the drawing as shown in the Formulas dialog box, for example:



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 draw the views of a CATPart document.

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.

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.

4. Click **OK**.
   A new drawing is created starting from the selected template.
5. 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*.

Managing the Document Lifecycle

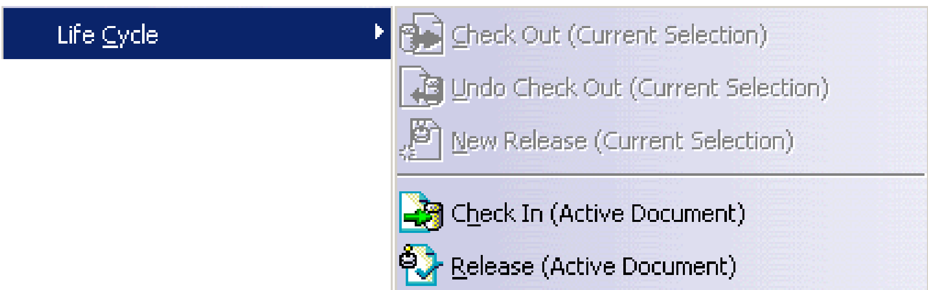
SMARTEAM enables you to maintain and manage any information related to a document throughout its life cycle. By mirroring the physical process of document management, SMARTEAM uses vaults, check-in, check-out, and approval functions to manage the life cycle of your Product, Part or Drawing. It creates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

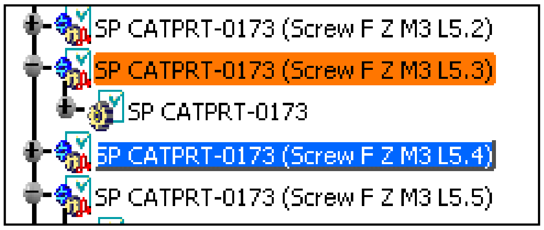
From V5R15 onward, you can access the following Lifecycle commands from contextual menus:

- **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 **Check In** to to check a *new* document into the vault or to place a document that is **being modified** back in the vault.	**Checked In**	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 **Check Out** to check out a document from the vault. or Click **New Release** to make a new copy of a **Released** document that was placed in the **Released** vault. The resulting document is a new revision of the source document.	**Being Modified**	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 **Checked In** or **Released.**
Click **Release** to transfer a document to the **Released** status.	**Released**	The document is saved in the vault of released documents.
Click **Undo Check Out** to cancel the check-out of a document from the vault.	**Checked In**	Any changes made following the check-out are lost. The new revision is deleted. The document status reverts to **Checked In** status.

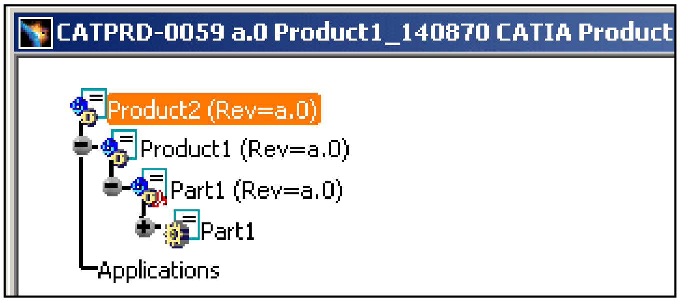
Analyzing the Impacts of a Change

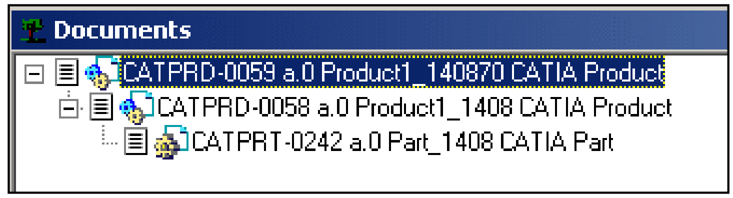
The **Impact Analysis** function displays all the documents pointing onto a selected object. This tool allows you to evaluate the impacts brought by the modifications made to the current object and therefore helps you choose the solution to implement with the least modification cost.

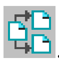
We recommend you use this tool prior to checking out and modifying documents.

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

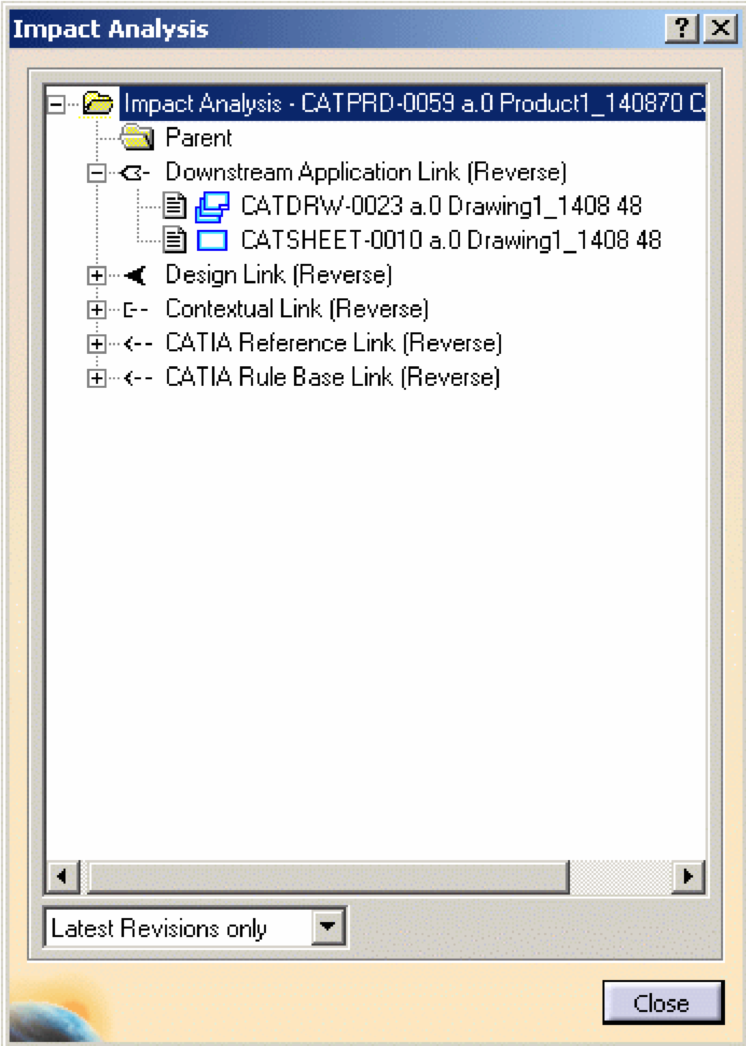
3. From the drop-down menu in the lower part of the dialog box, set **Latest Revisions only**.

   This option displays only the latest revisions of the documents that will be impacted by the current revision of the selected object.

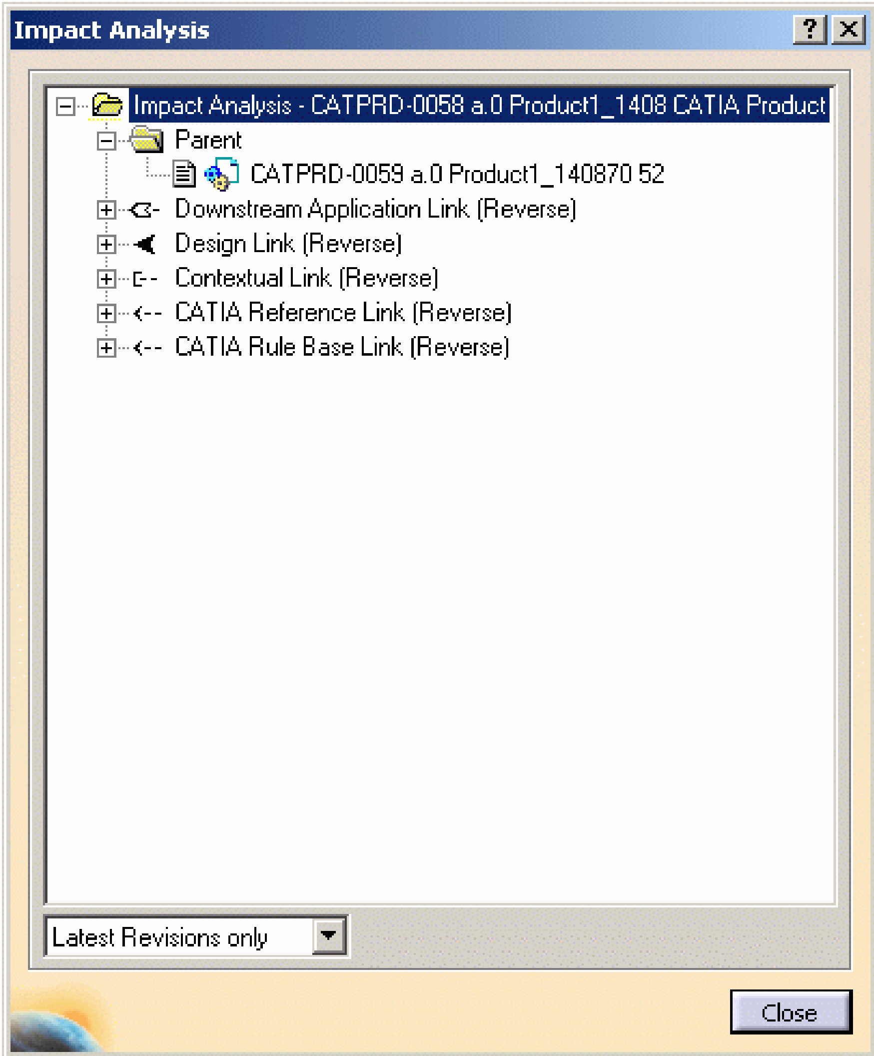
4. Click the + sign in front of the **Downstream Application Link (Reverse)** category.

   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:

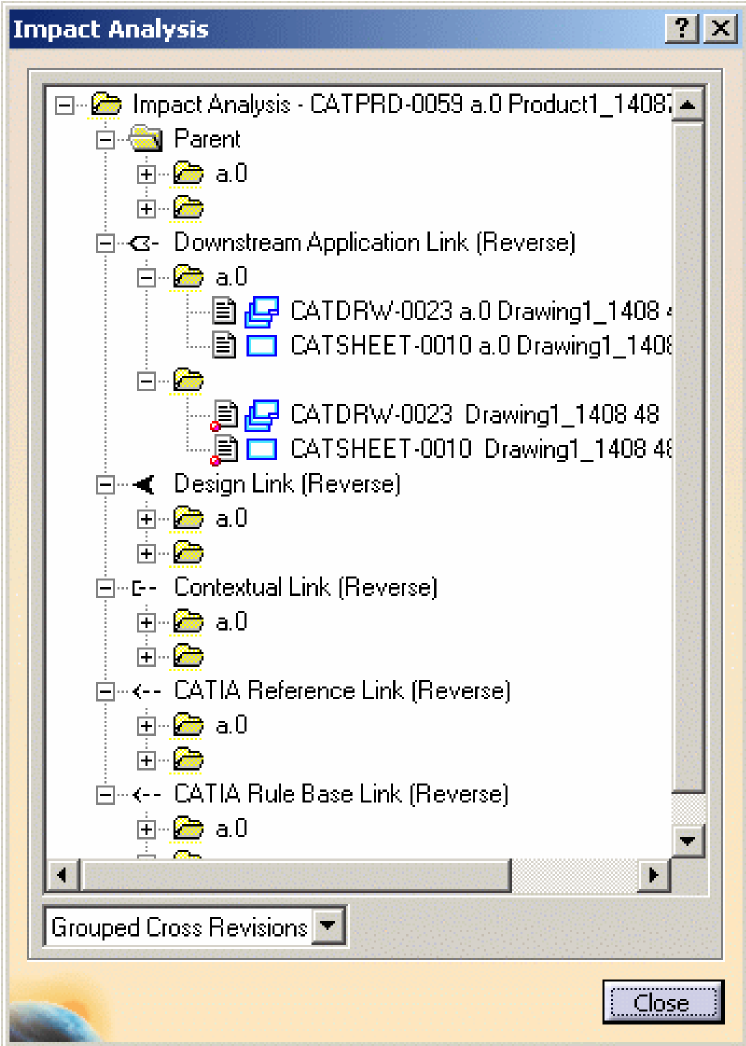


Grouped Cross Revisions

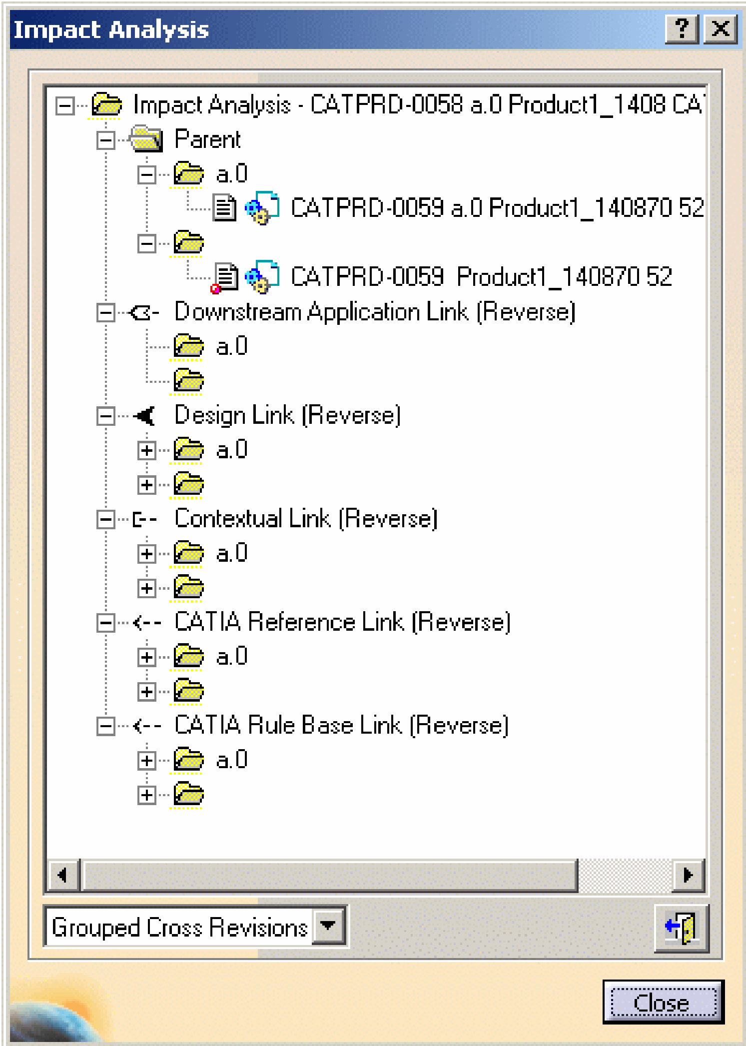
3. Still using **Product2** as the selected document you are analyzing, from the drop-down menu, set **Grouped Cross Revisions.**
4. Click the + sign in front of the **Downstream Application Link (Reverse)** category.

   Grouped Cross Revisions displays also the impacted documents as well as all their revisions.

   You obtain this view:



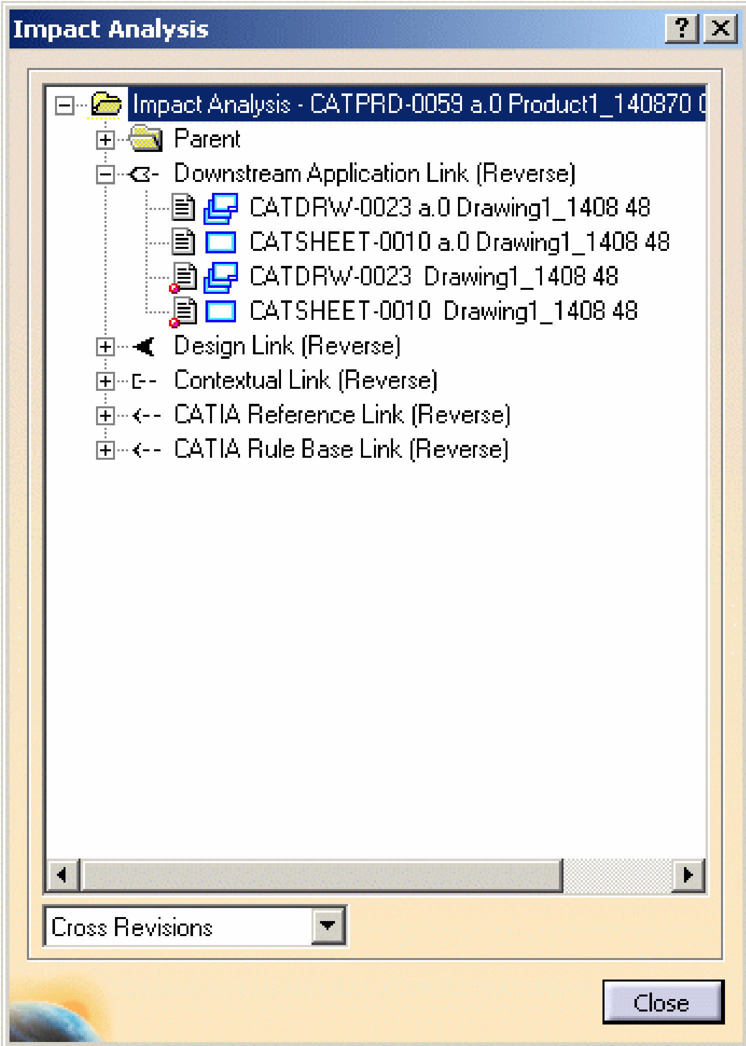
By applying **Impact Analysis** onto the sub-assembly, you could see in the **Downstream Application Link (Reverse)** category that an earlier version exists for its parent CATPRD-0058 a.0 Product1_1408 CATIA Product. You would obtain this view:



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 two major operations a system administrator should perform prior to using SMARTEAM CATIA Integration. Both 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.

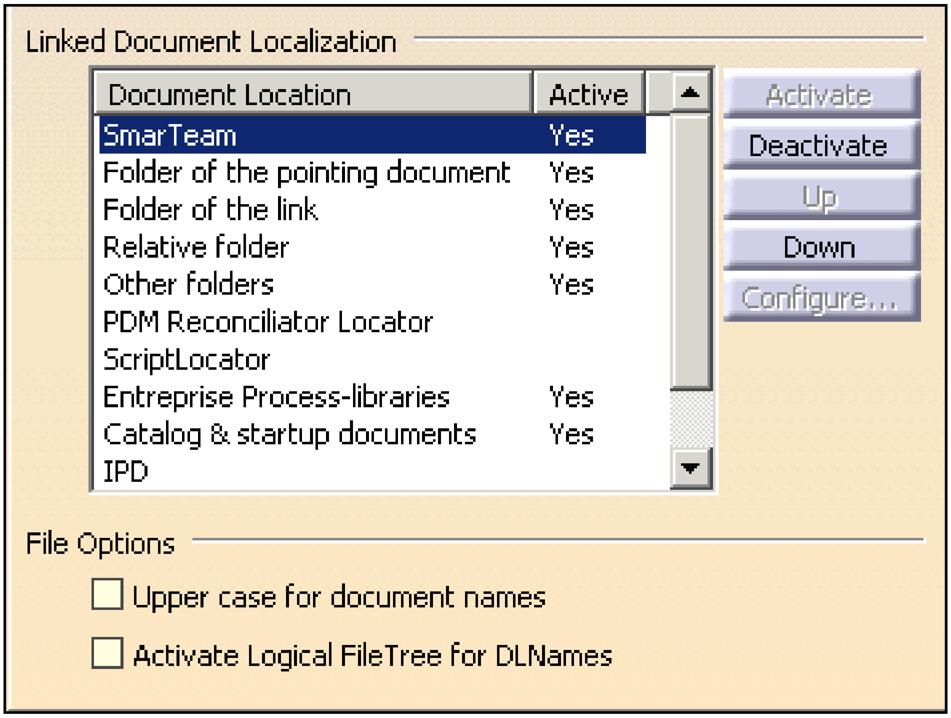
There are two main folders:

- 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 the SMARTEAM folder of the link and folder of the pointing document strategy. Ensure that **SMARTEAM** is the first item in the list and that **Folder of Link** is the second one.
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. Ensure that the SMARTEAM Document environment is allowed and is in the current state in the **Document Environment** selection list.



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

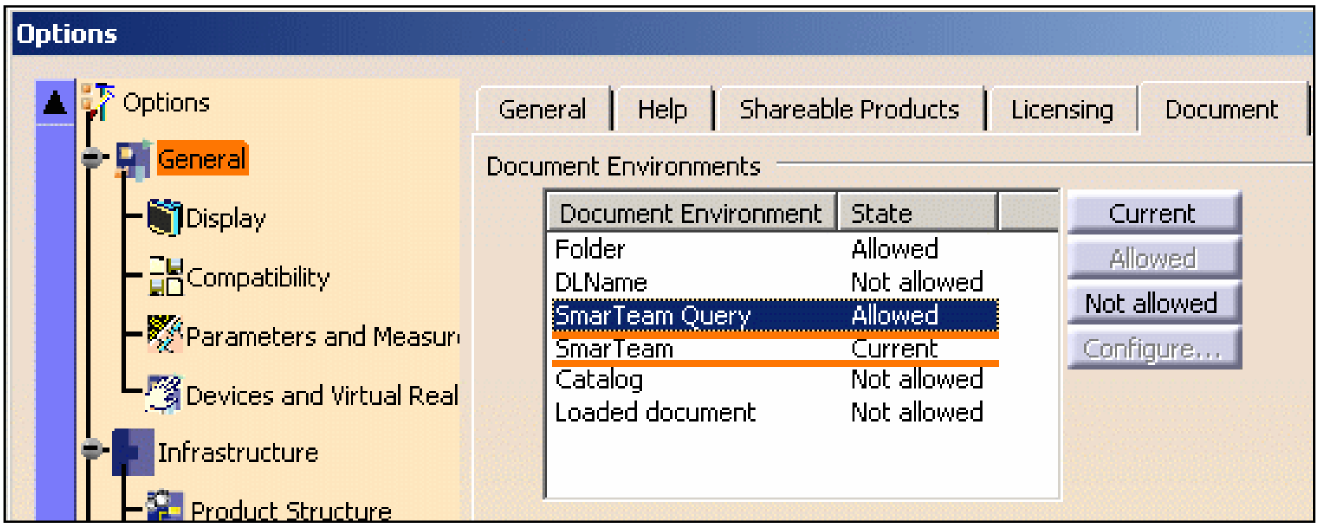
Note

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). In this specific case, the **Folder of the pointing document** strategy will not be able to recover the linked documents. Another strategy, called **SMARTEAM Database** has to be used. This strategy will ask the SMARTEAM database where the linked document has been copied.

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 form 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 (thru predefined queries or precise queries). Additionally, ensure that to be able to use the Find capabilities, the Folder Document Environment must be set to Allowed.

Customizing

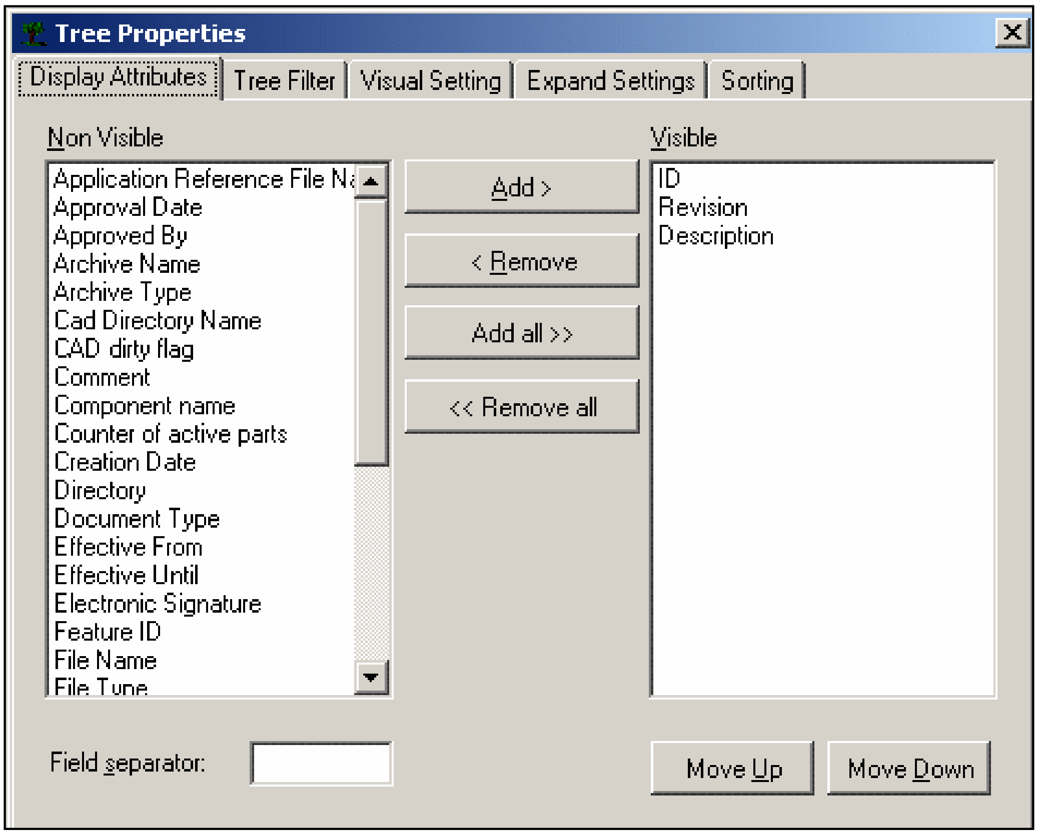
This section discusses different ways of customizing your SMARTEAM CATIA Integration session.

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**.

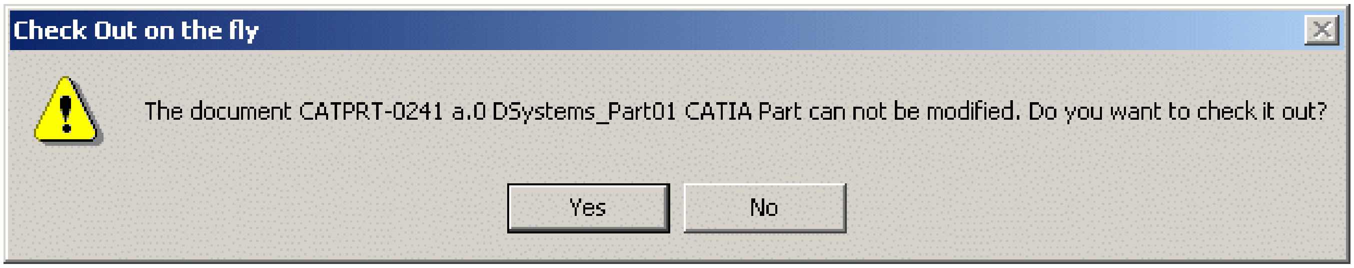
   The **Tree Properties** dialog box that appears contains various tab pages to customize the appearance and tree content. The **Display Attributes** tab enables you to specify which attributes will comprise the name of the objects as they are displayed in the tree.



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

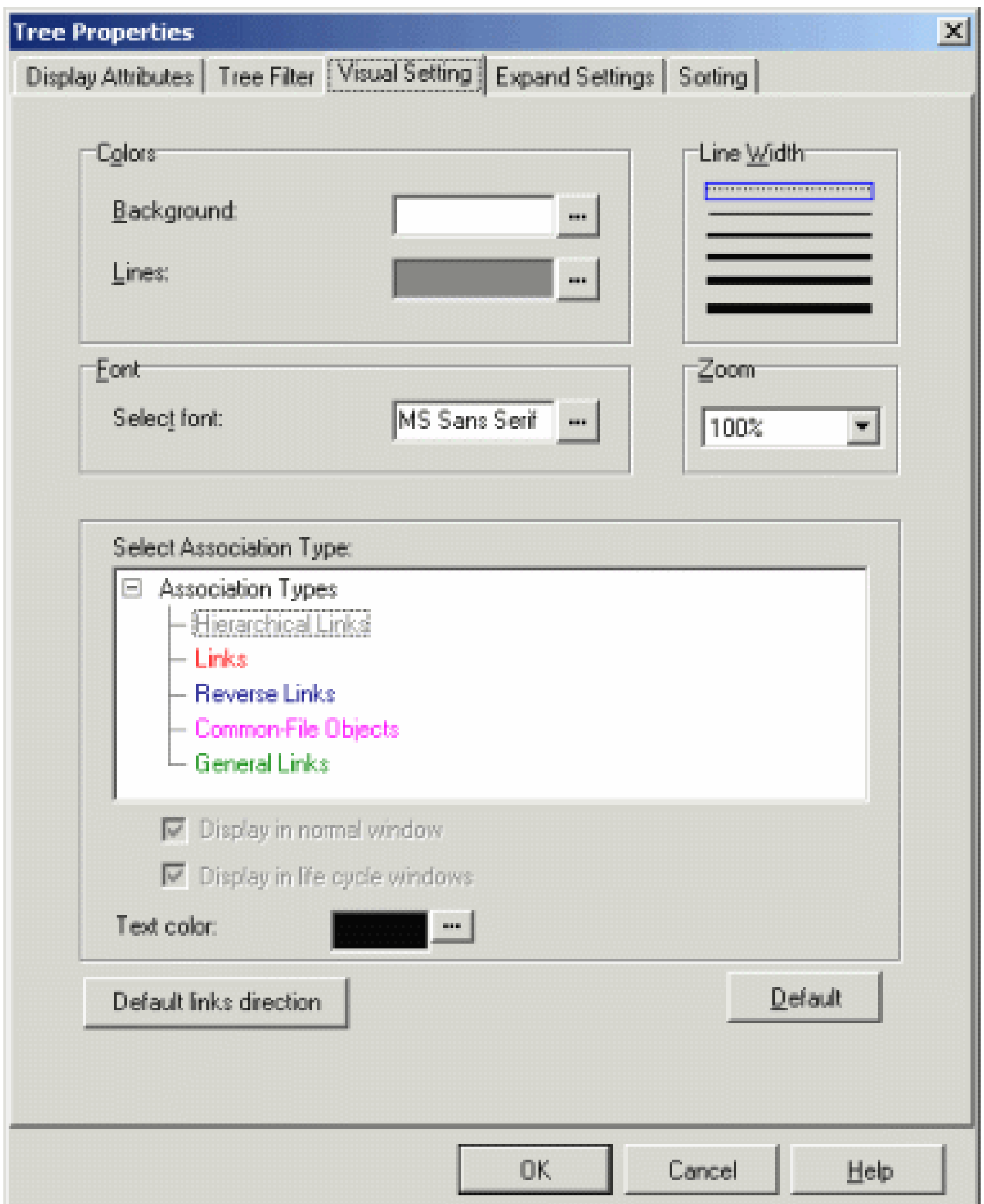

Check Out on the fly
The document CATPRT-0241 a.0 DSystems_Part01 CATIA Part can not be modified. Do you want to check it out?
Yes
No

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.

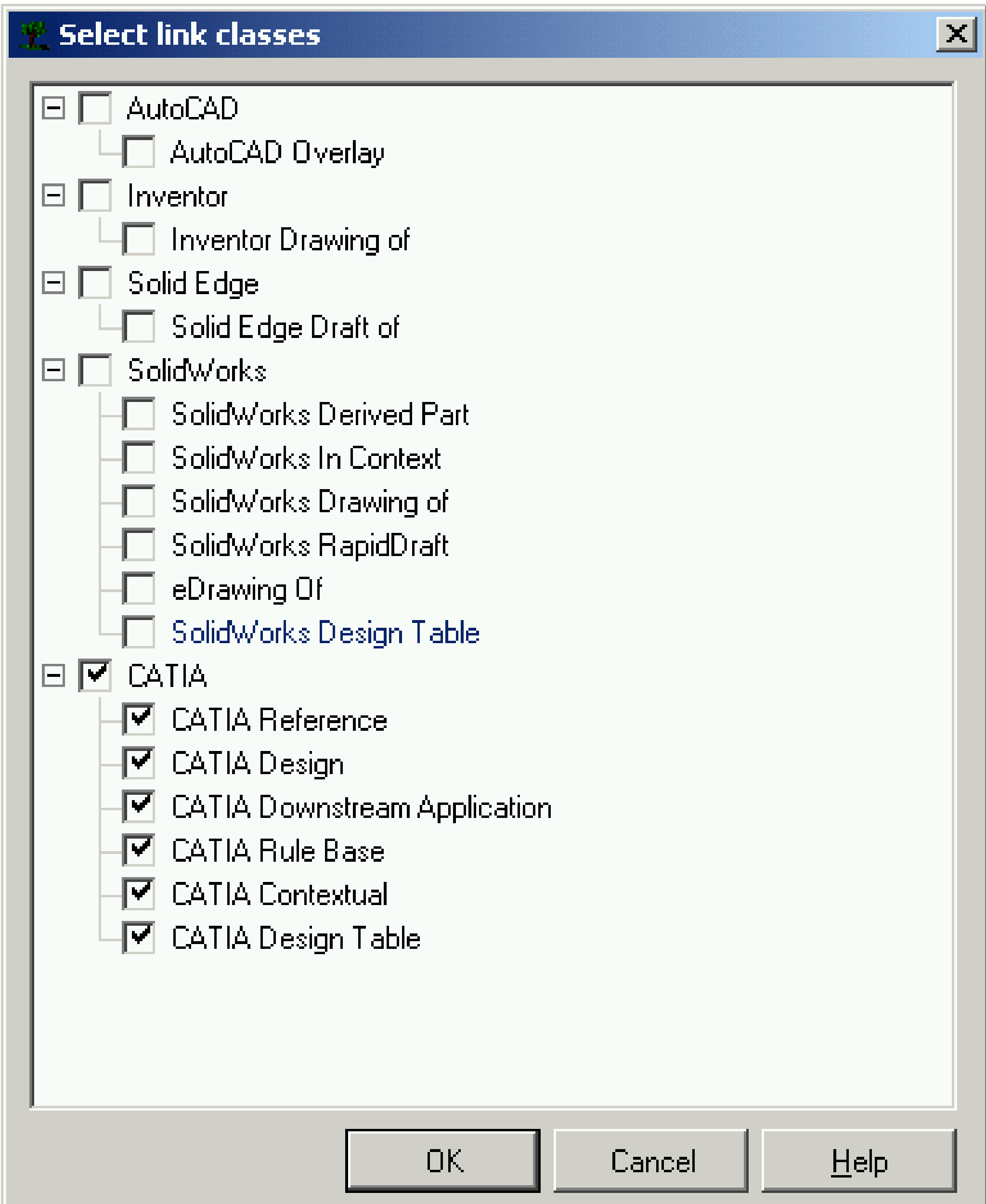
1. Select your document in SMARTEAM.
2. Right-click **Tree Properties**.
3. Click the **Visual Setting** tab.

   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.

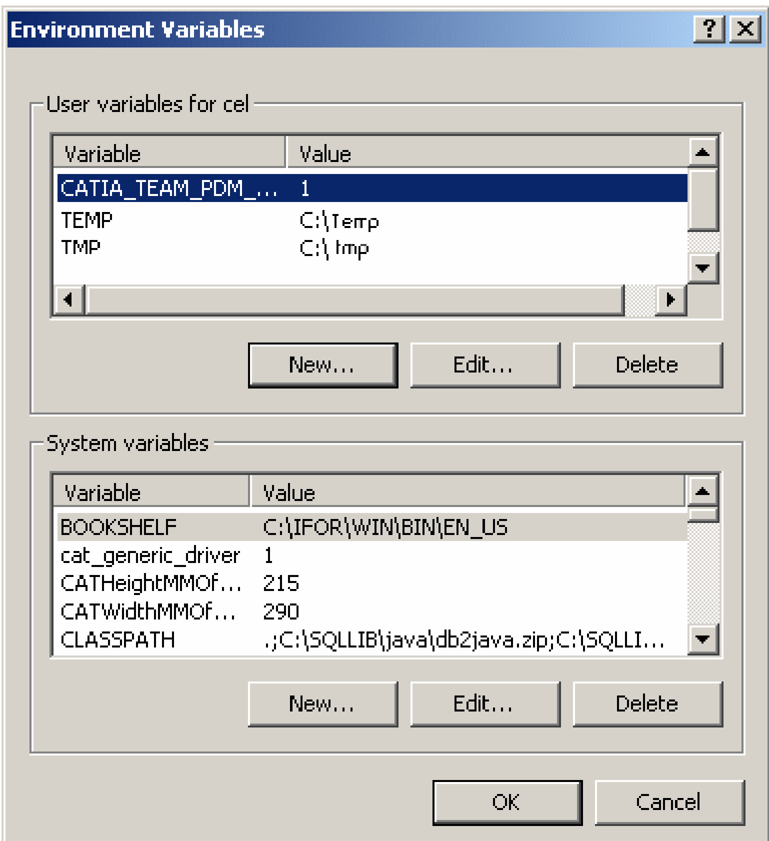
6. Click **OK** to validate and close the dialog box.

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.



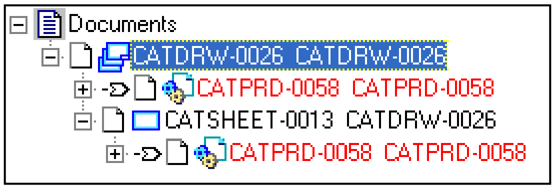
The following variables can be set in the Environment Variables of a user's workstation

- *CATIA_TEAM_PDM_AUTO_CONNECT*=1
  Allows automatic login to SMARTEAM when CATIA is launched.
- *CNEXTSPLASHSCREEN* = NO
  This system variable is to be used if you set *CATIA_TEAM_PDM_AUTO_CONNECT*.
  It prevents CATIA splash screen from displaying while you are connecting

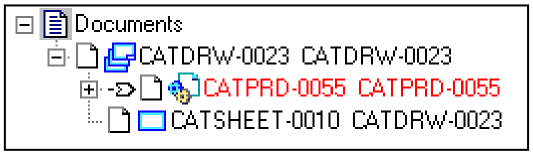
  to SMARTEAM.

6. *CATIA_TEAM_PDM_NOCOOTF*=1
Disables check out on the fly operations. It is recommended not to set this variable.
7. *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, refer to Document Associations and Dependencies.
8. *CATIA_TEAM_PDM_SPCLASS* = NO
The specific class mode is not used (specific class for the internal component or sheet objects).

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

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.

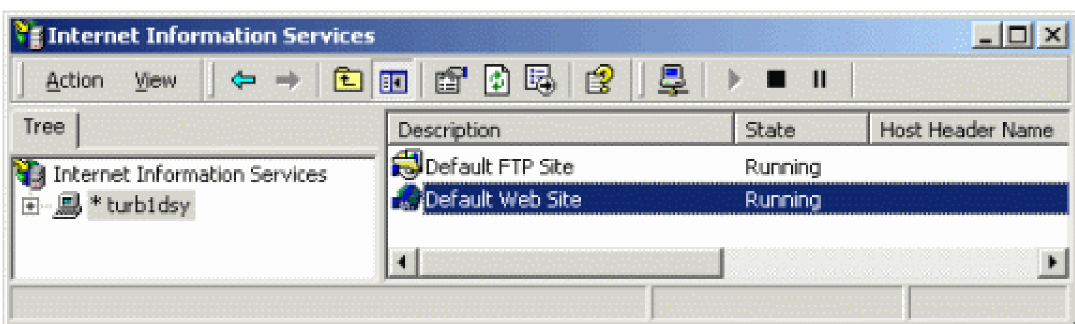
This functionality is useful when saving a large amount of documents of one given file type. When saving lots of documents of various CATIA file types, the rule to save the objects will follow the SmTeam32 information. It is up to the user to check that the targetted class is consistent. When saving 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**, refer to Importing CATIA Data Inside SMARTEAM.

SMARTEAM System Configurator

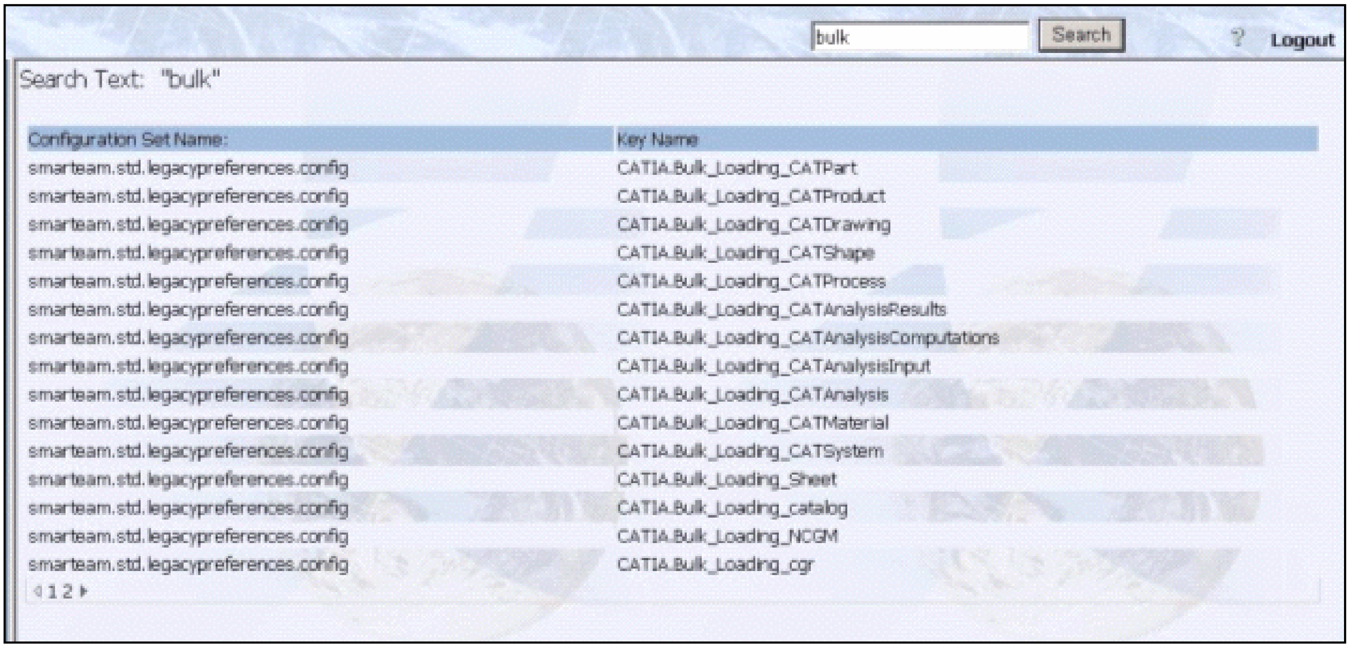
In order to set the classes to be used during the Bulk Loading operation, you should add/modify dedicated SMARTEAM options using the SMARTEAM System Configurator tool and proceeding as follows:

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:

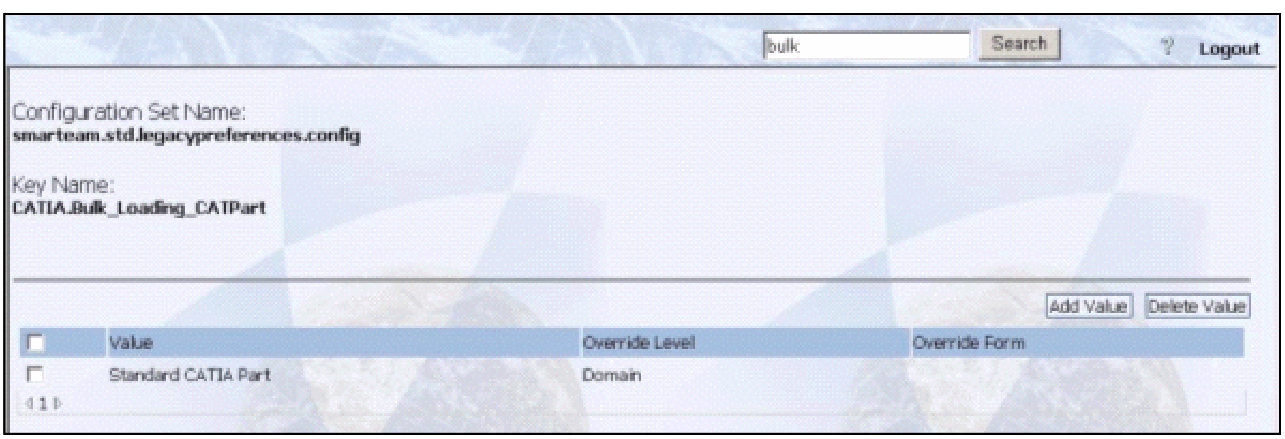


5. Select one item from the list. For instance, CATIA_Bulk_Loading_CATPart.

6. Set the appropriate value: for example, choose Domain from the Override Level combo box and enter Standard CATIA Part in the **Value** field.

7. Click the **Save Changes** button to confirm the changes.

Once saved, the dialog box looks like this:



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

For more information about the SMARTEAM System Configurator, refer to the *SMARTEAM documentation*.

Customizing Design Copy

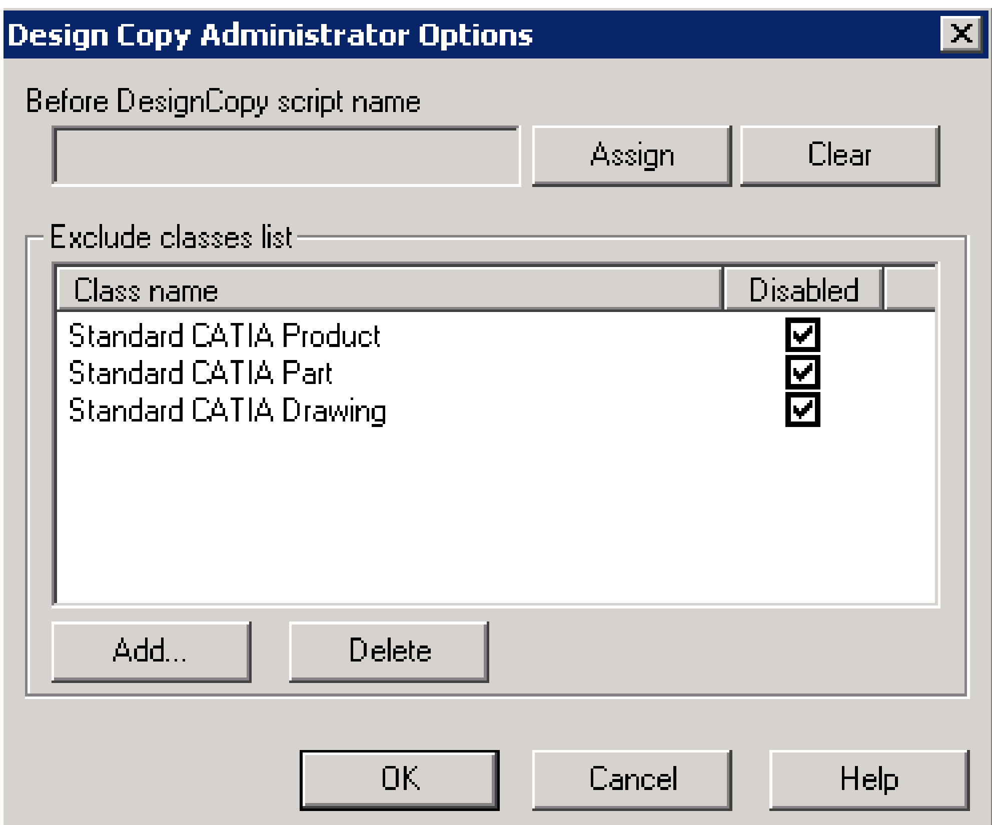
This section describes the different ways of customizing the Design Copy command. The topics discussed are:

- 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 not checked in the Design Copy window. You can change the status to checked in the Design Copy window.

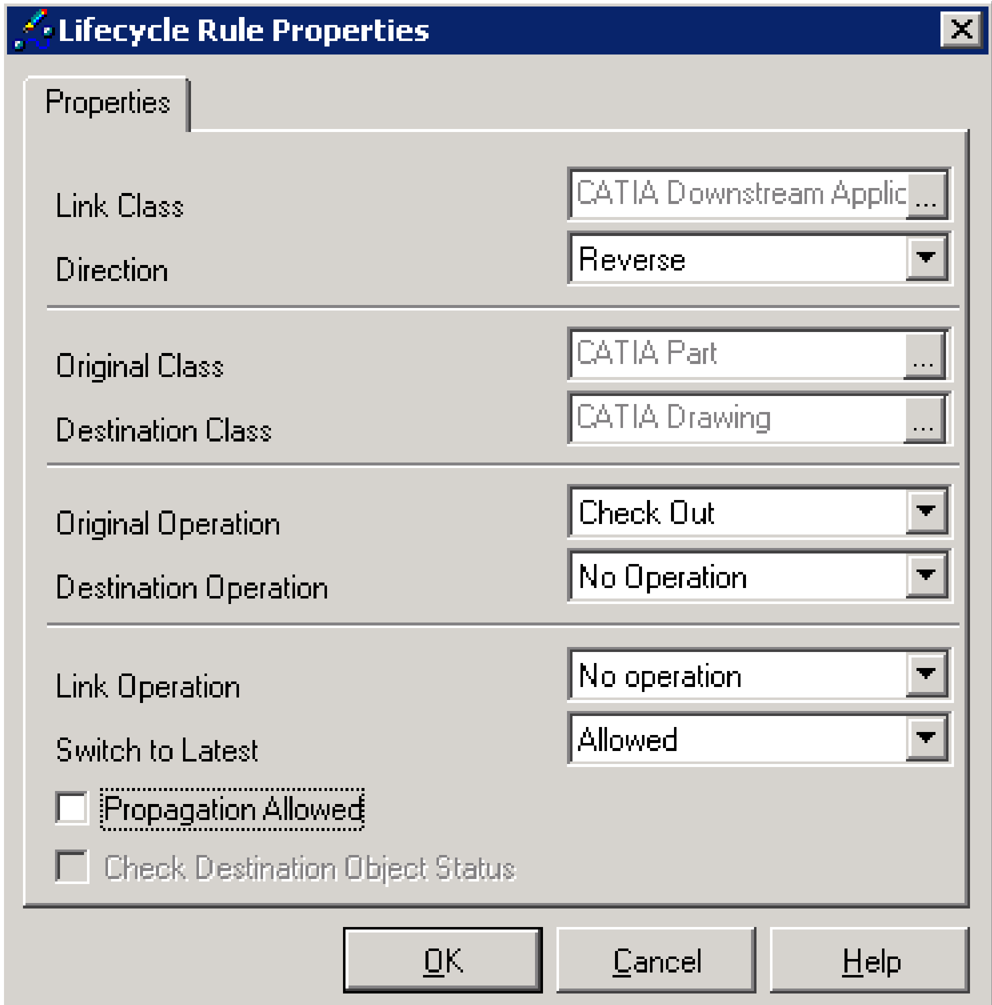
4. Restart the Design Copy tool to apply the changes made to the Design Copy Administrator Options.

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.



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.

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

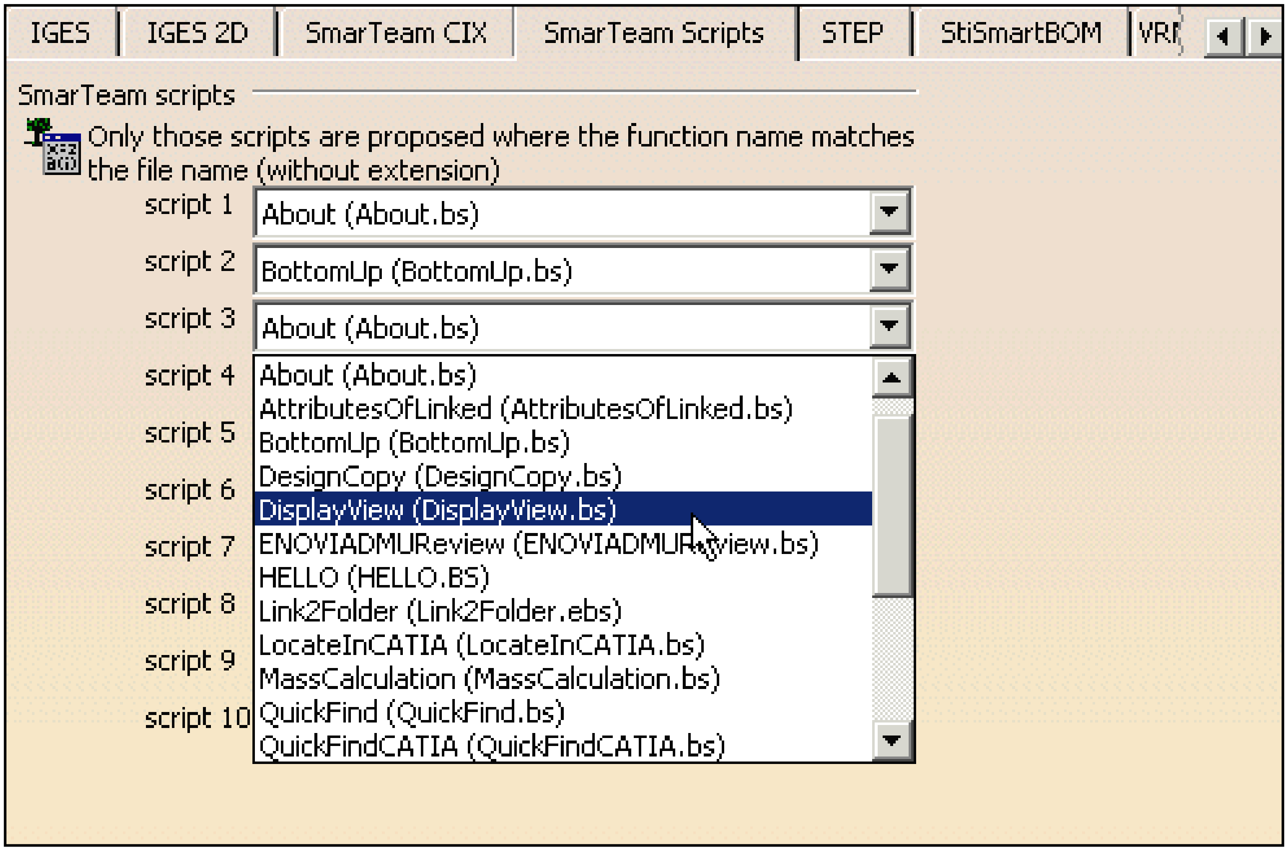
1. Run a CATIA session.
2. Select **Tools->Options...**.

   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.

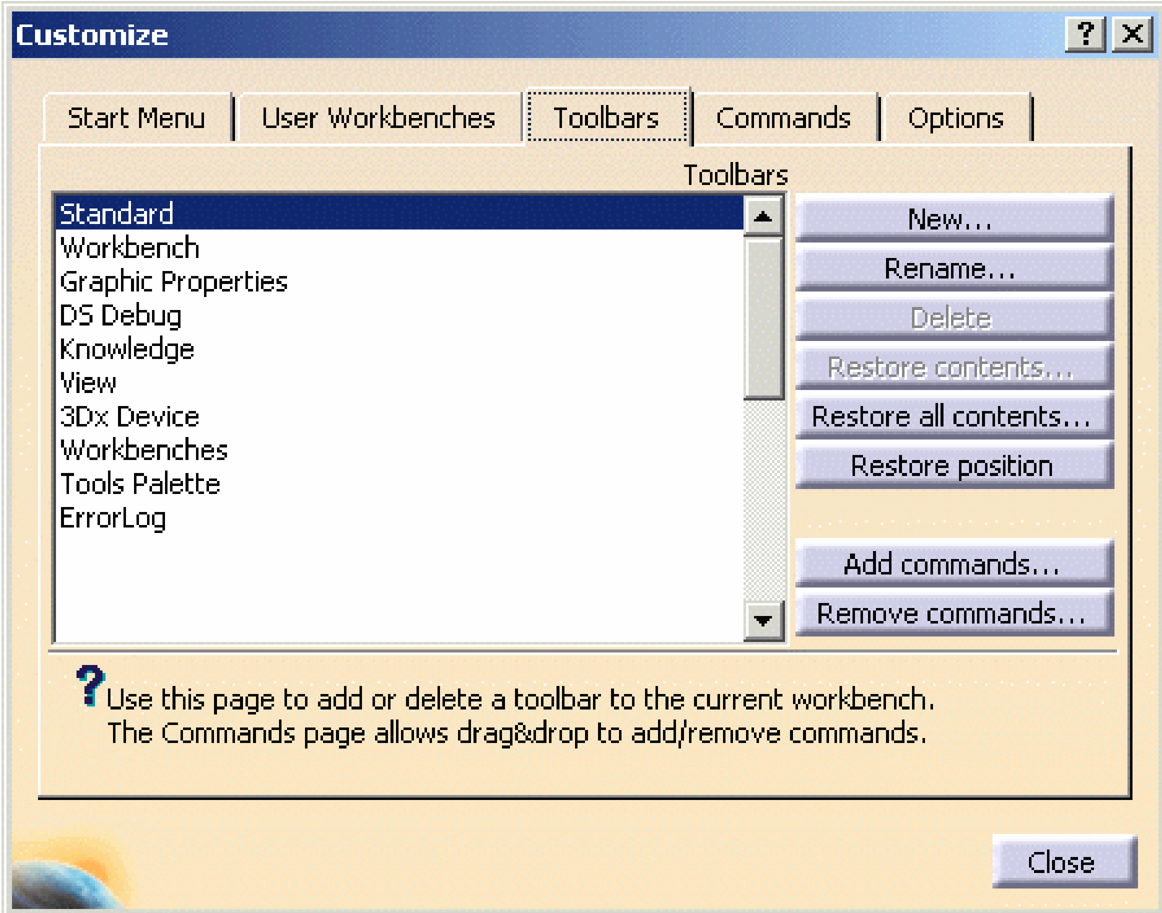
Placing the Commands in the CATIA Graphic User Interface

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.

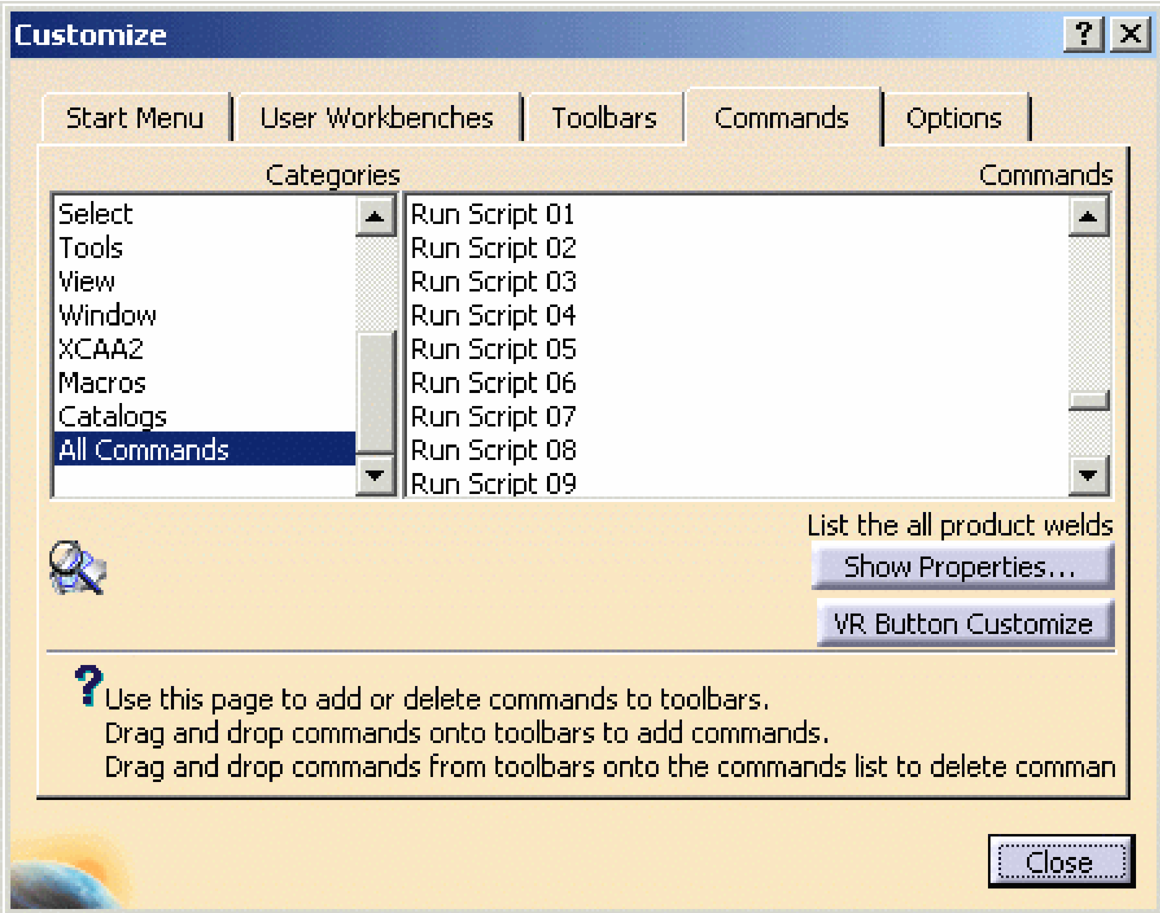
2. Select the **Toolbars** tab:



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 SMARTEAMs 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.

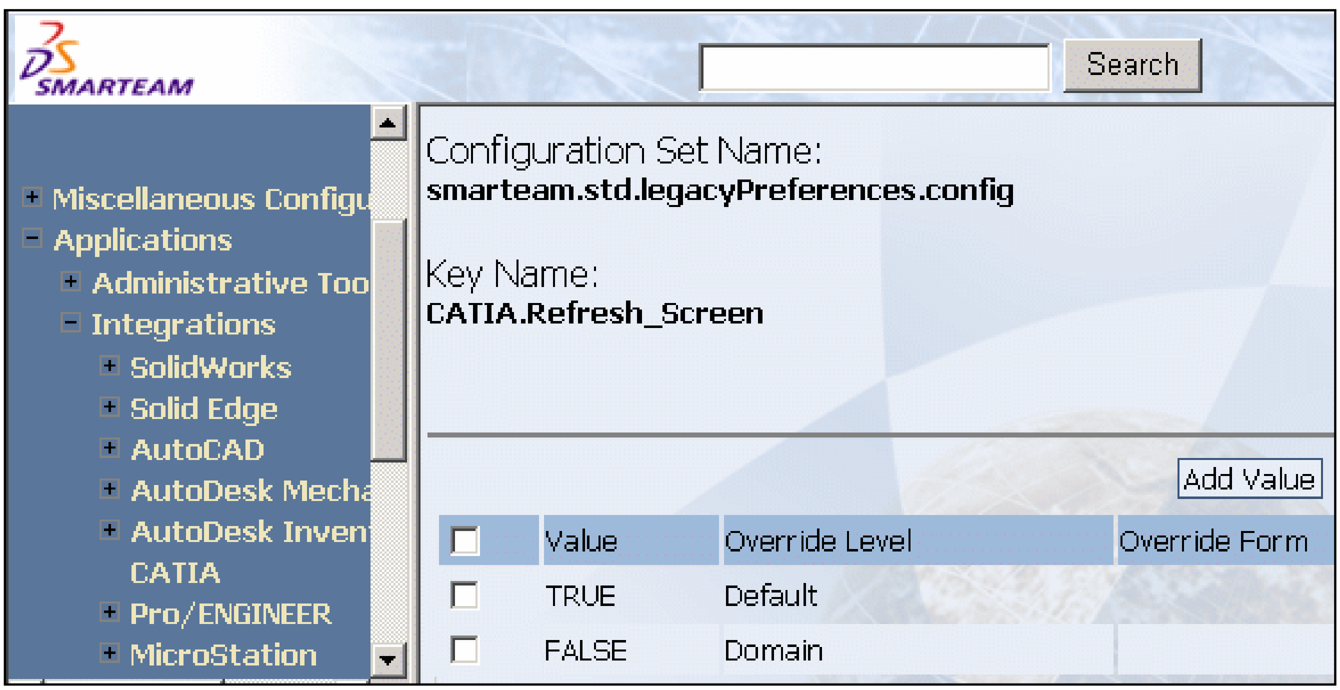
   To know how to use this tool, refer to the SMARTEAM documentation.

2. Search for CATIA application.

3. Set CATIA.Refresh_Screen to FALSE.

4. Click the **Save Changes** button to confirm the changes.

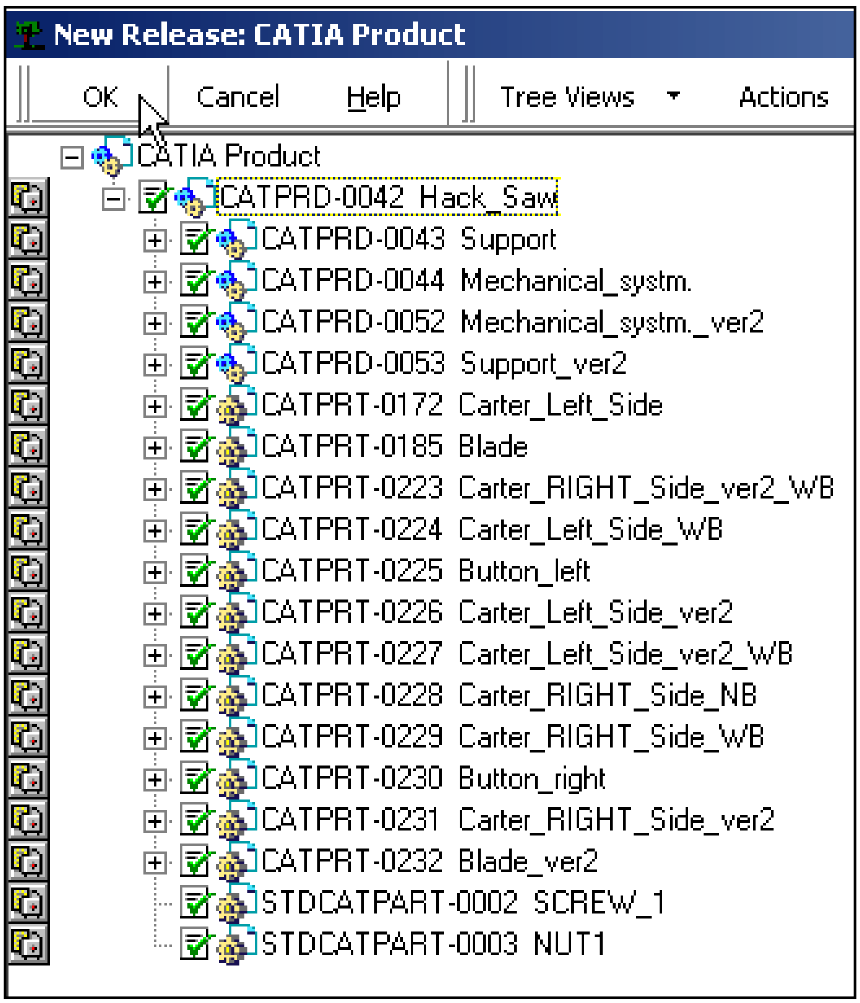
   Once saved, the dialog box looks like this:



Simplifying Lifecycle Operations

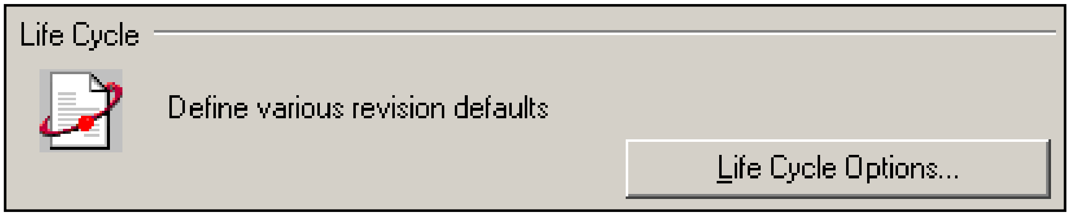
To make end users gain time, system administrators can now prevent the application from displaying all windows that appear during lifecycle operations (Check In, Check Out, Release, New Release, Undo Check Out) performed in CATIA or SMARTEAM. This capability is available from SMARTEAM V5R14 SP5 onward.

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:

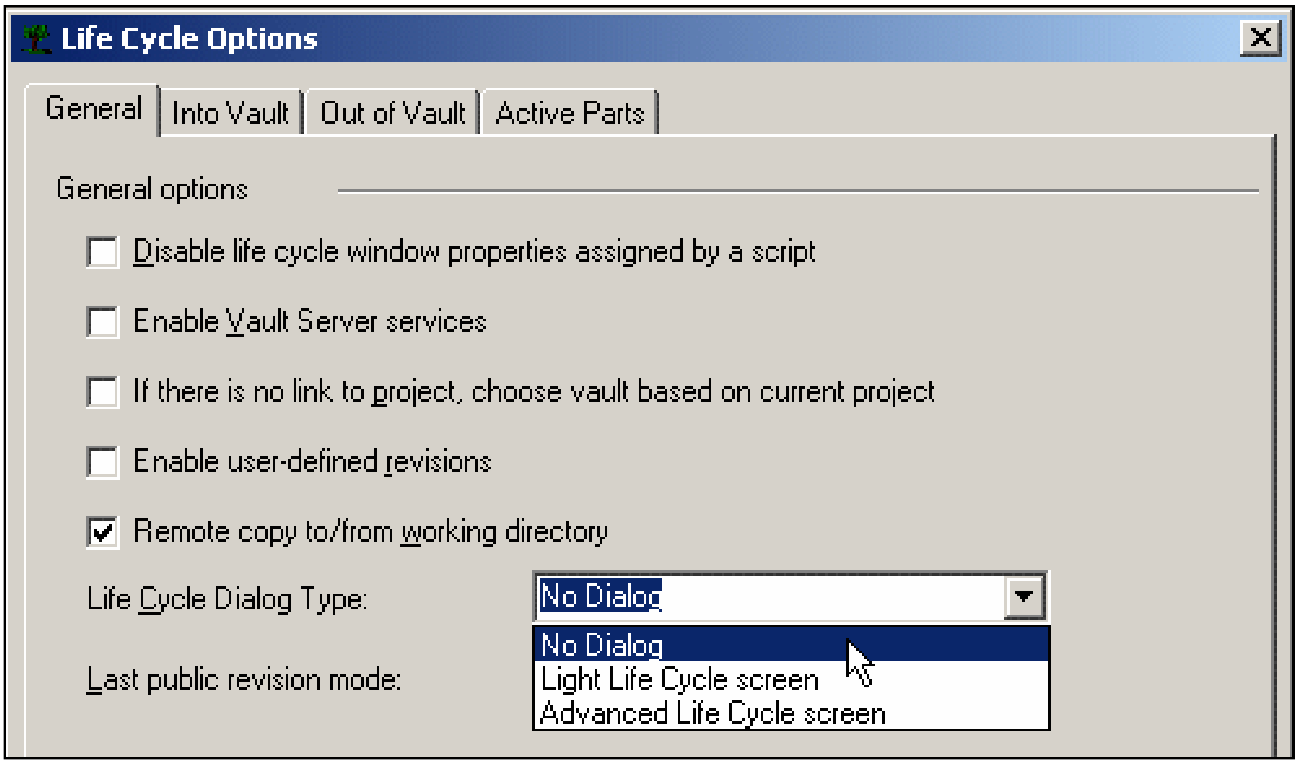


To eliminate the display of the different lifecycle windows, proceed as follows:

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.

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.

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

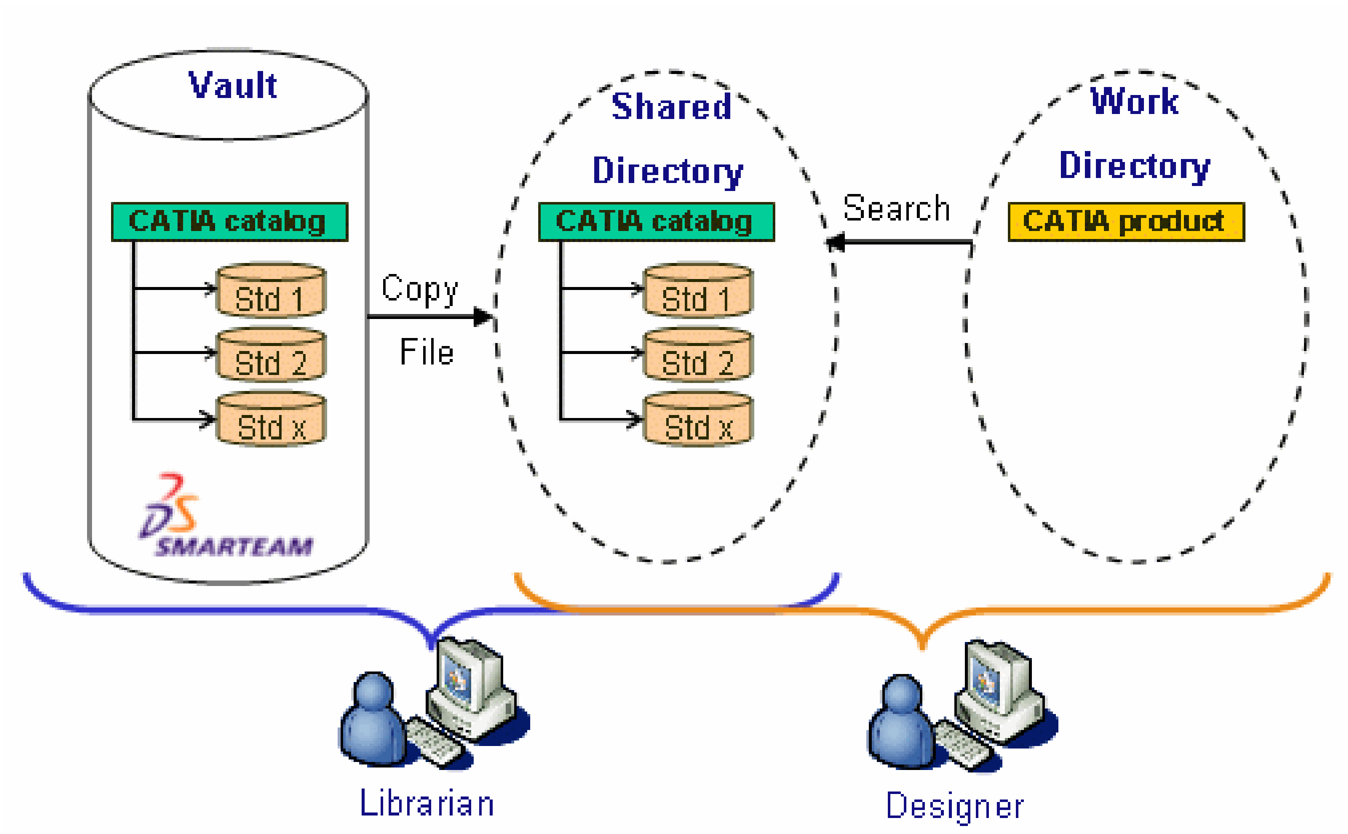
Remove permissions for all lifecycle operations for design engineers.

Only a librarian must have permissions for lifecycle operation on Standard CATIA Parts.

Setting Up a Shared Directory for CATIA Catalogs

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

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



Creating and Saving CATIA Catalogs

The procedure described below is a simplified version of the process. For detailed instructions, refer 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).

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 file contains the information required to create a complete chapter referencing subchapters

For more about the Catalog Workbench, refer to 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.

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.

   Note:

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

Steps in SMARTEAM

Saving a CATIA V5 catalog inside SMARTEAM

5. Define classes in which you will save your documents.

   For detailed instructions, refer to 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. Check out the catalog containing the documents to be modified.
2. Check out the documents you wish to modify.
3. Modify the documents in your CATIA session.
4. Save the documents in the SMARTEAM database.

Releasing the CATIA V5 catalog inside SMARTEAM

5. 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

6. Set the shared directory folder as the default shared directory folder.
7. Search for your catalog using the **Find** command.
8. Right-click and select **Copy File**.
9. 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, refer to 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, refer to 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

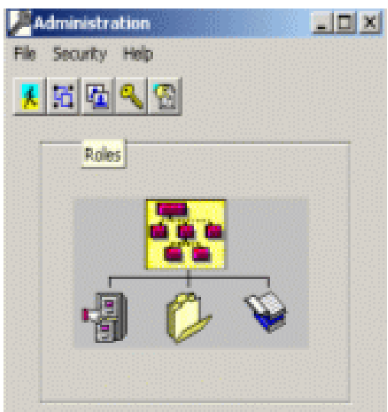
This task provides the main instructions for integrating CATIA sheet metal bends into your SMARTEAM database. This procedure can be divided into the following main steps, all of them to be performed in SMARTEAM.

- 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

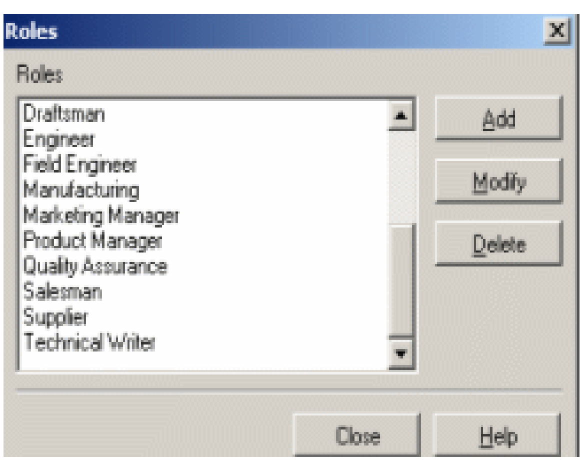
Creating a Role

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:



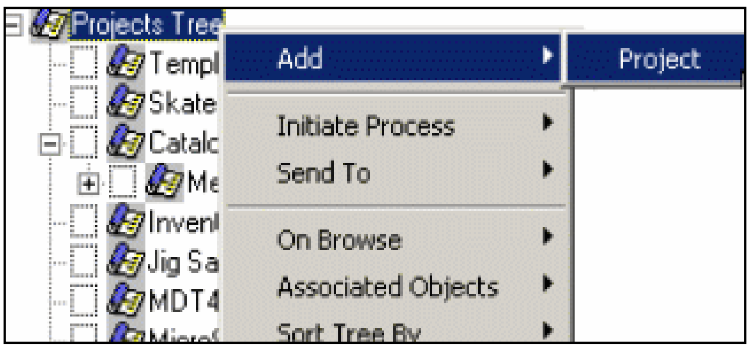
3. Create the role of interest. For example, name it as "Read Only on Standards".

For more information on defining roles, refer to *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.

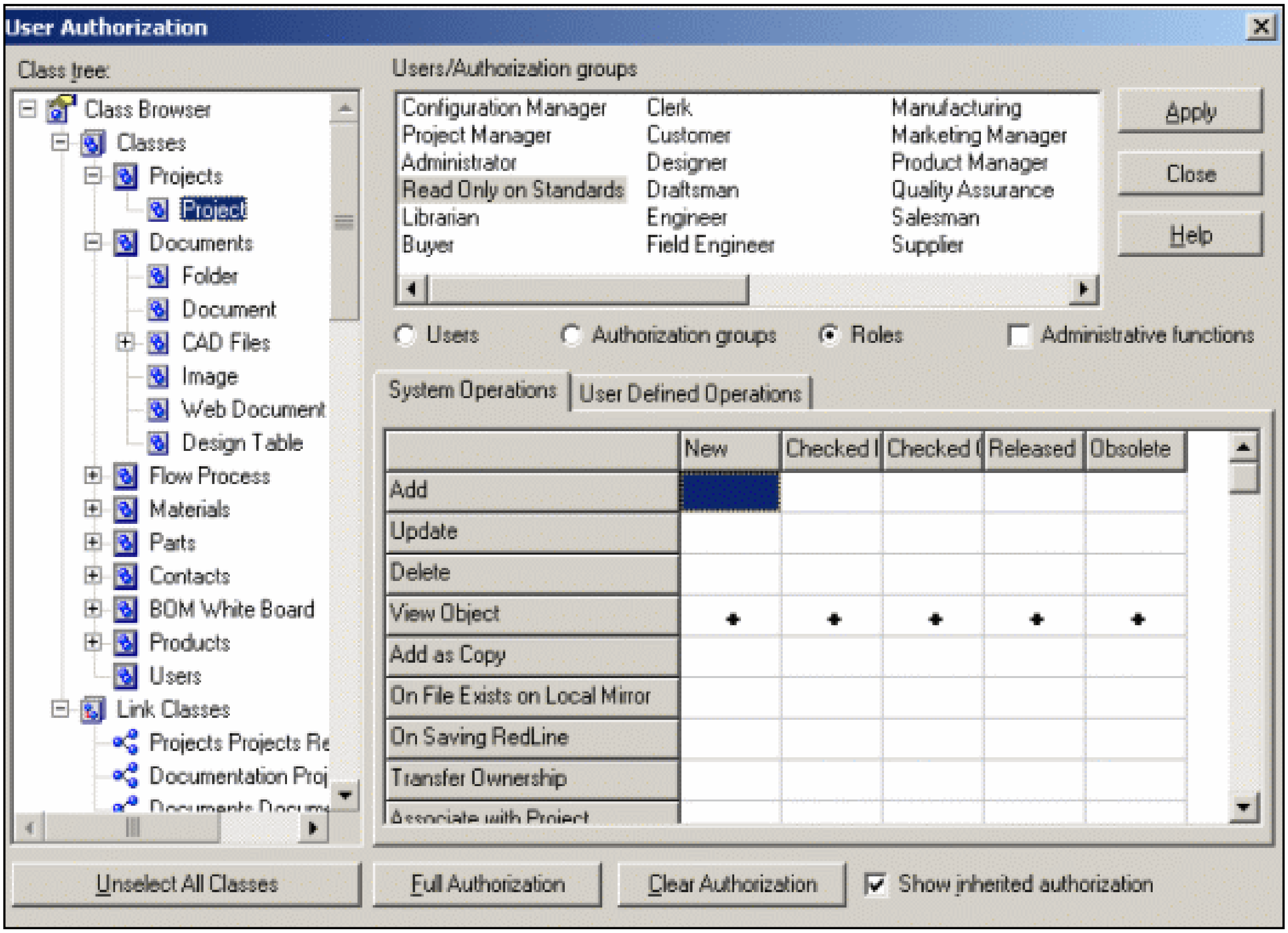
4. Add a project to which standard sheet metal tables will be linked.



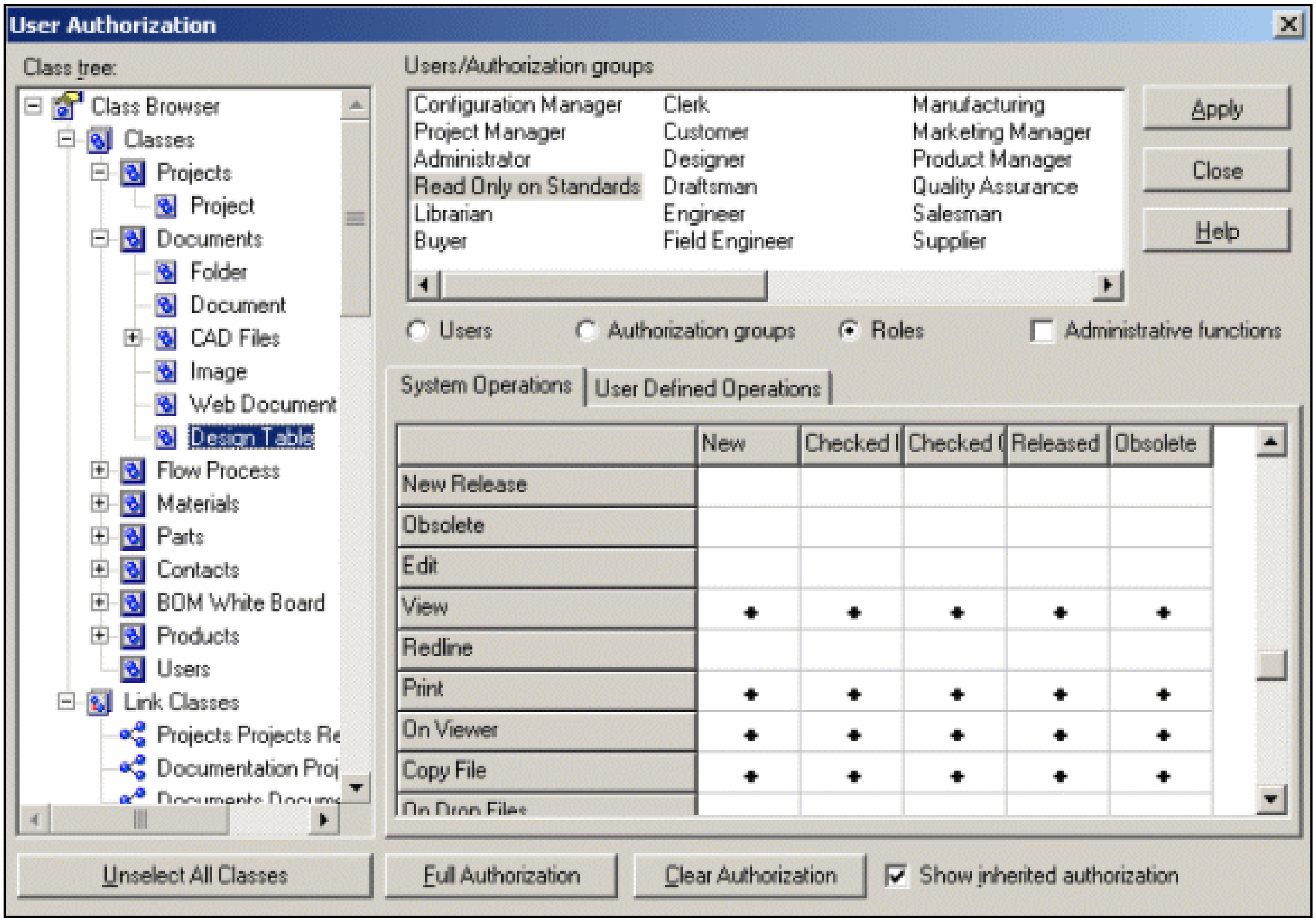
Assigning Authorizations

You need to assign authorizations for design engineers.

5. Define appropriate authorizations for the Read Only on Standards role, including:
   - **View Object** on project



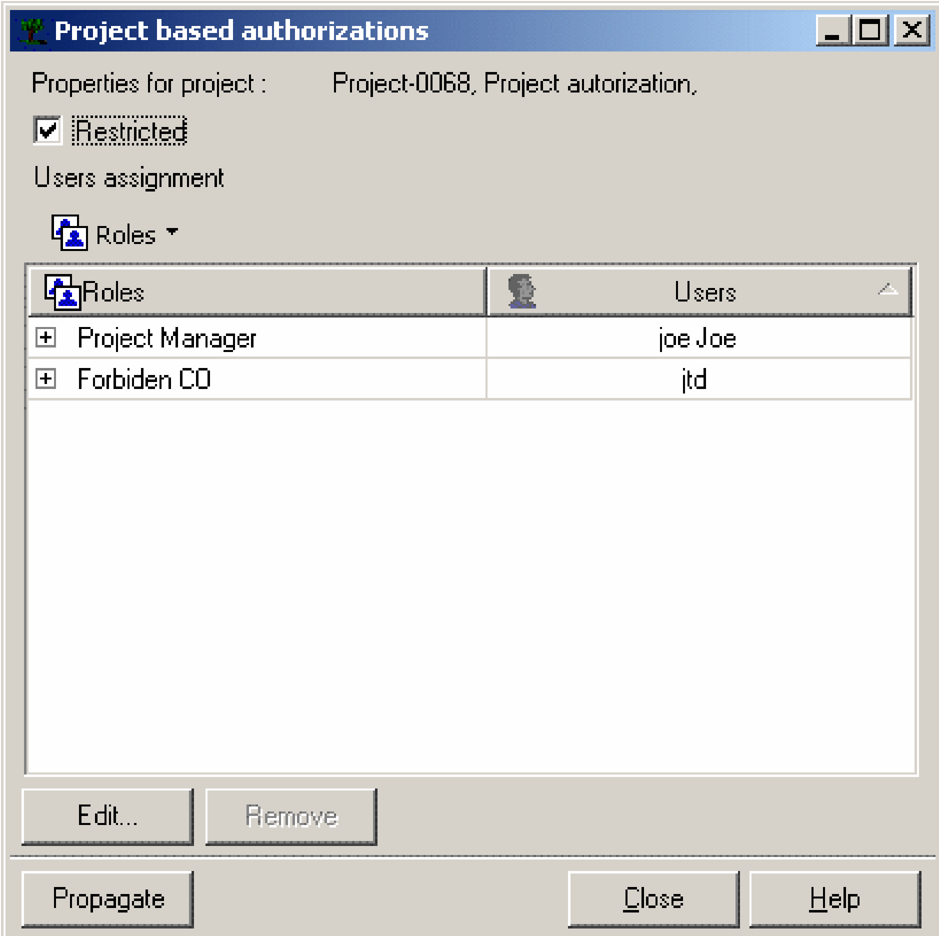
- **View Object**, **View**, **Print**, **On Viewer** and **Copy File** only for Design Tables:



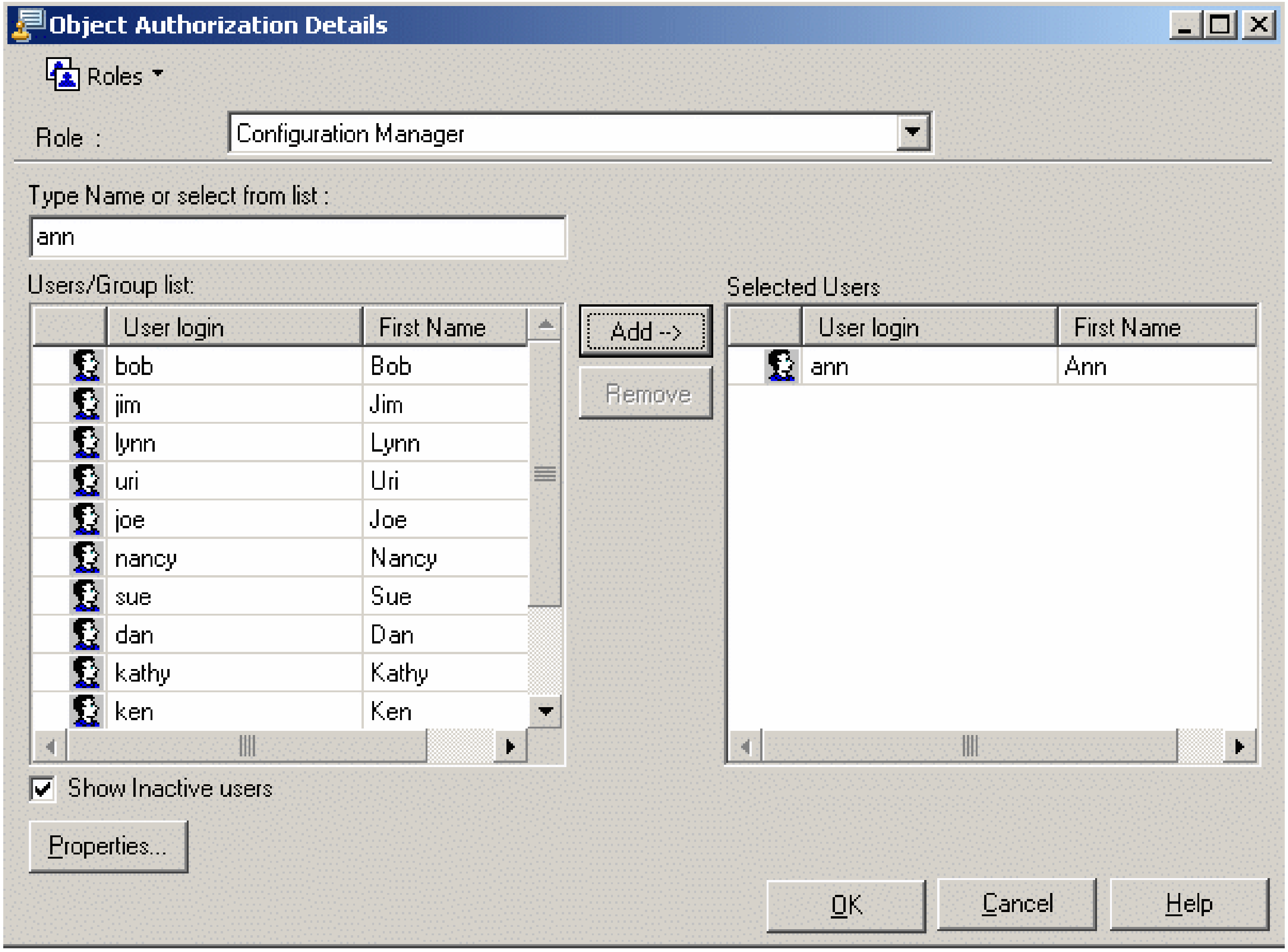
- **View Object**, on **Projects Projects**, **Documentation Projects** and **Documents Documents** relations

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.



Refer to 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.

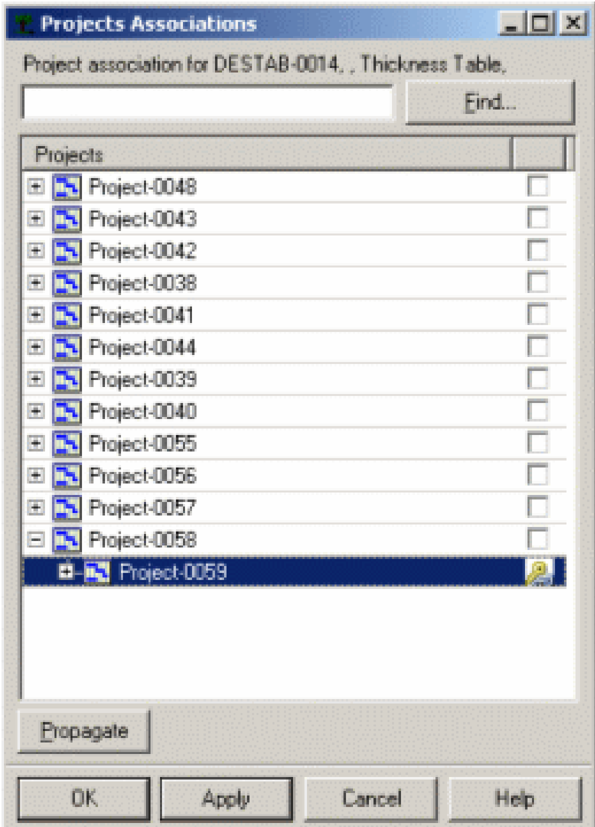
    This is optional: Keeping them as checked-in documents should be sufficient.

    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.

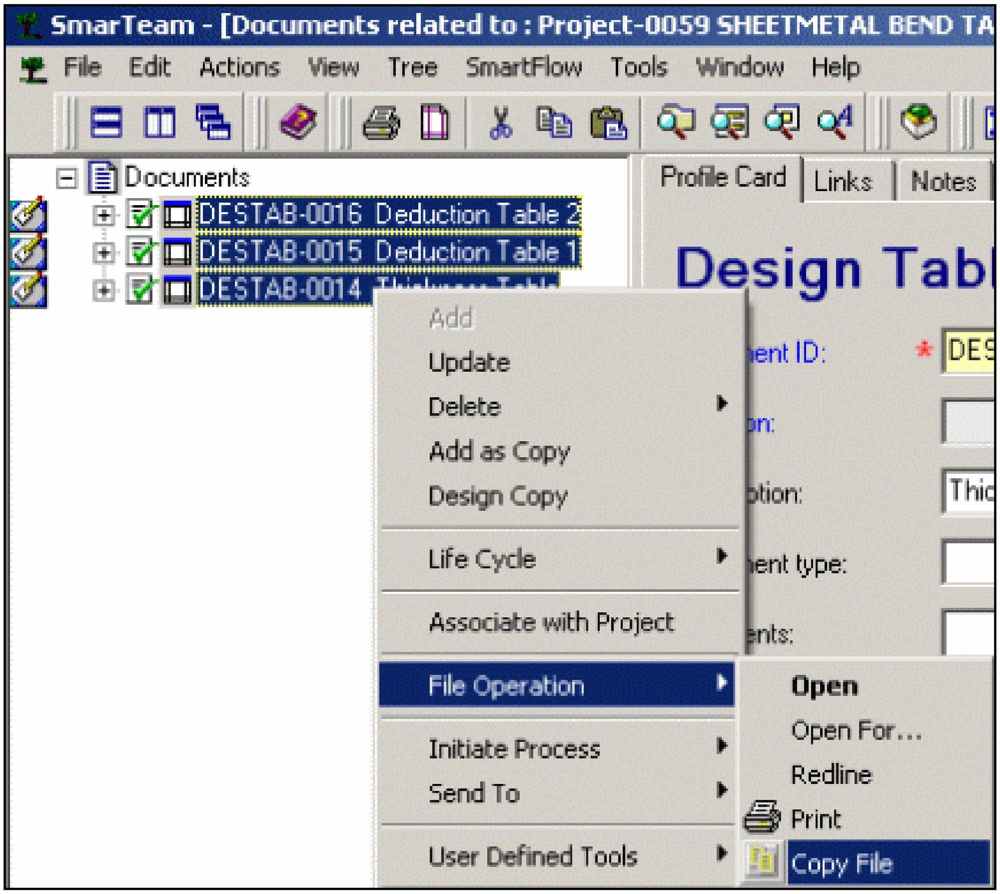
    A shared directory is just a directory on a server accessible to everybody. To keep this shared directory clean and secured, it is strongly recommended to assign the Windows permissions as below:

    - 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.

Refer to 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:

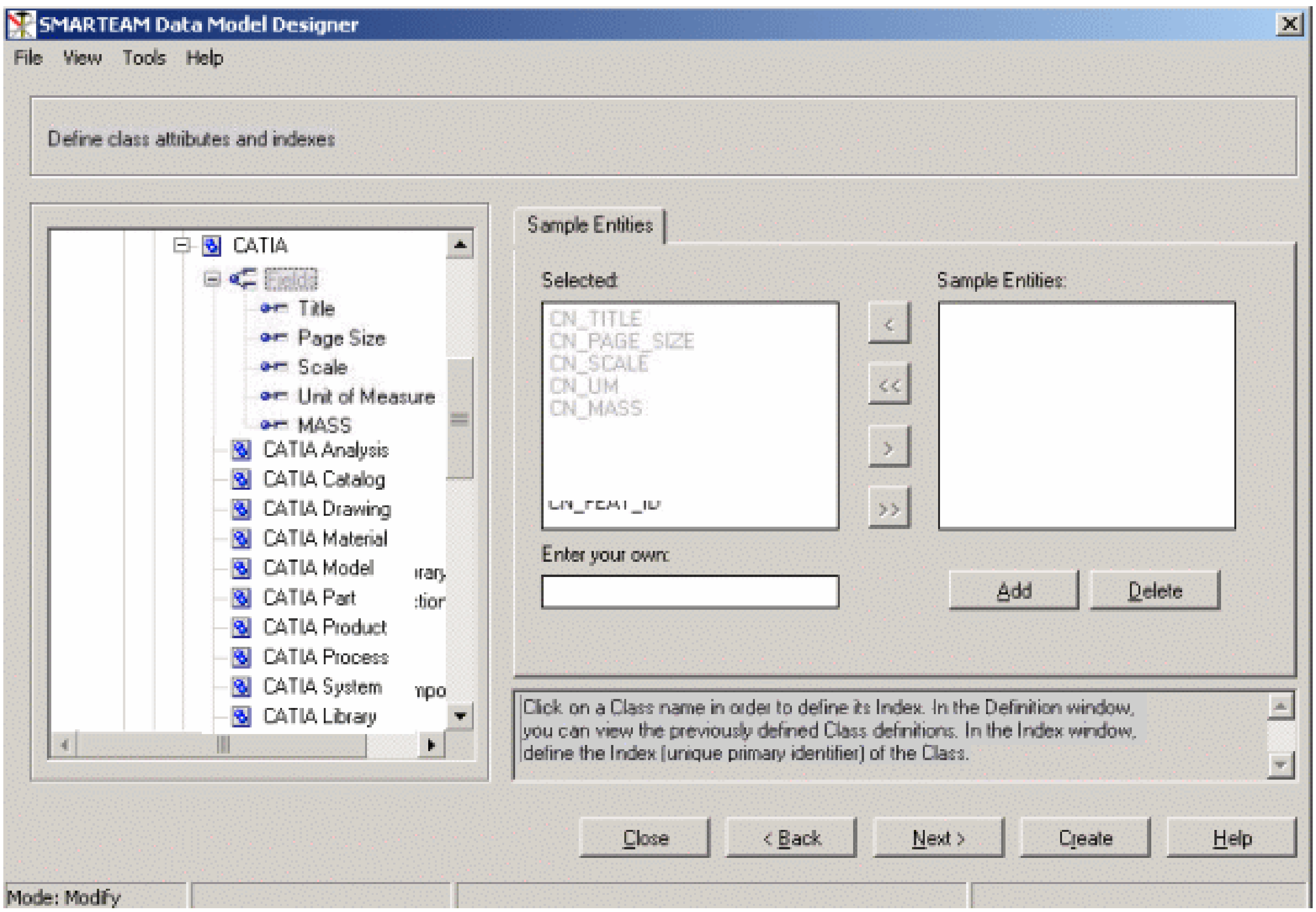
- Creating the CN_Mass Attribute
- Customizing CATIA Profile Cards
- Mapping CATIA Mass Attribute

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.



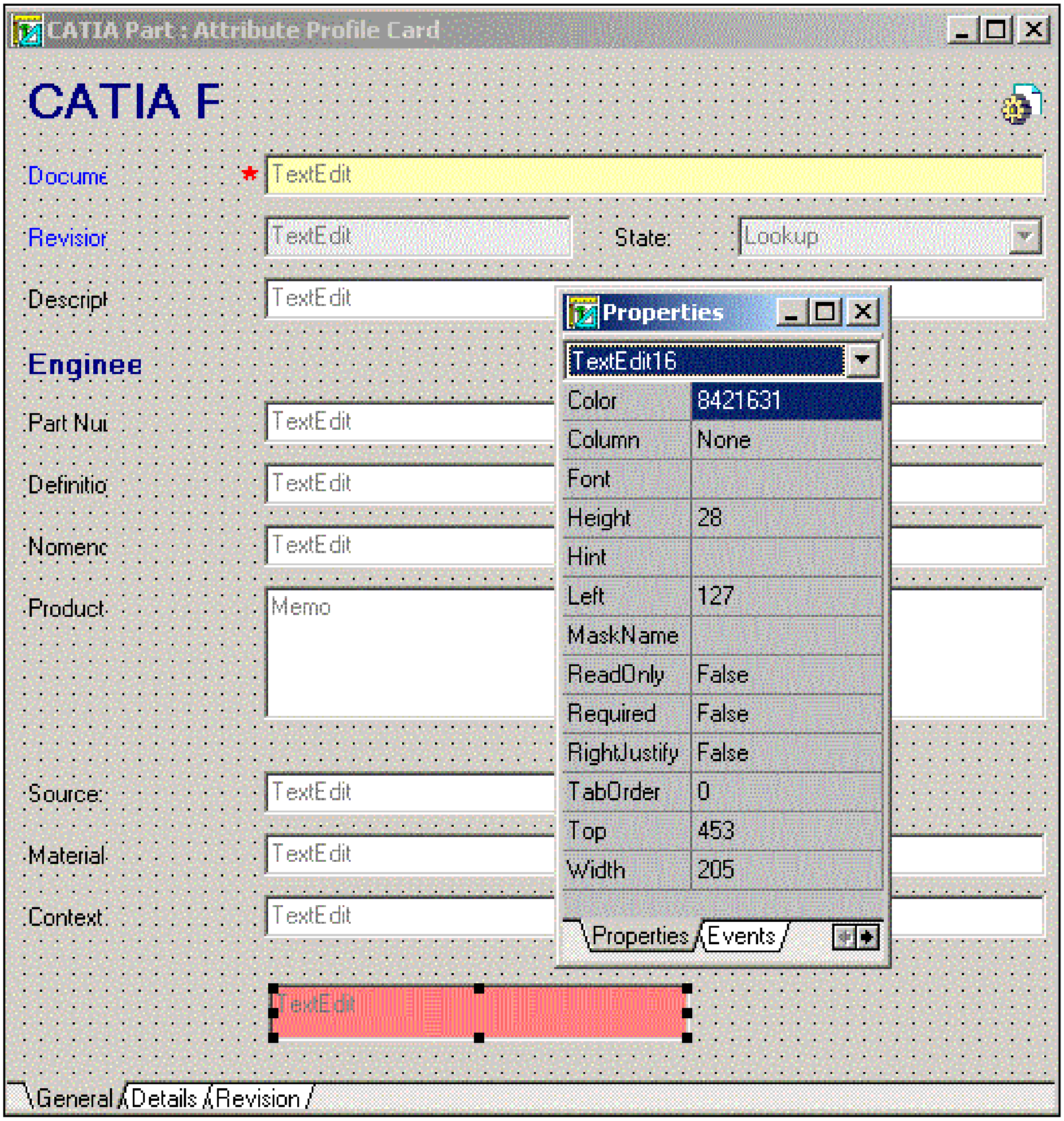
For reference information about SMARTEAM Data Model Designer, refer to the *SMARTEAM-Editor documentation*.

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, refer to the *SMARTEAM-Editor documentation*.

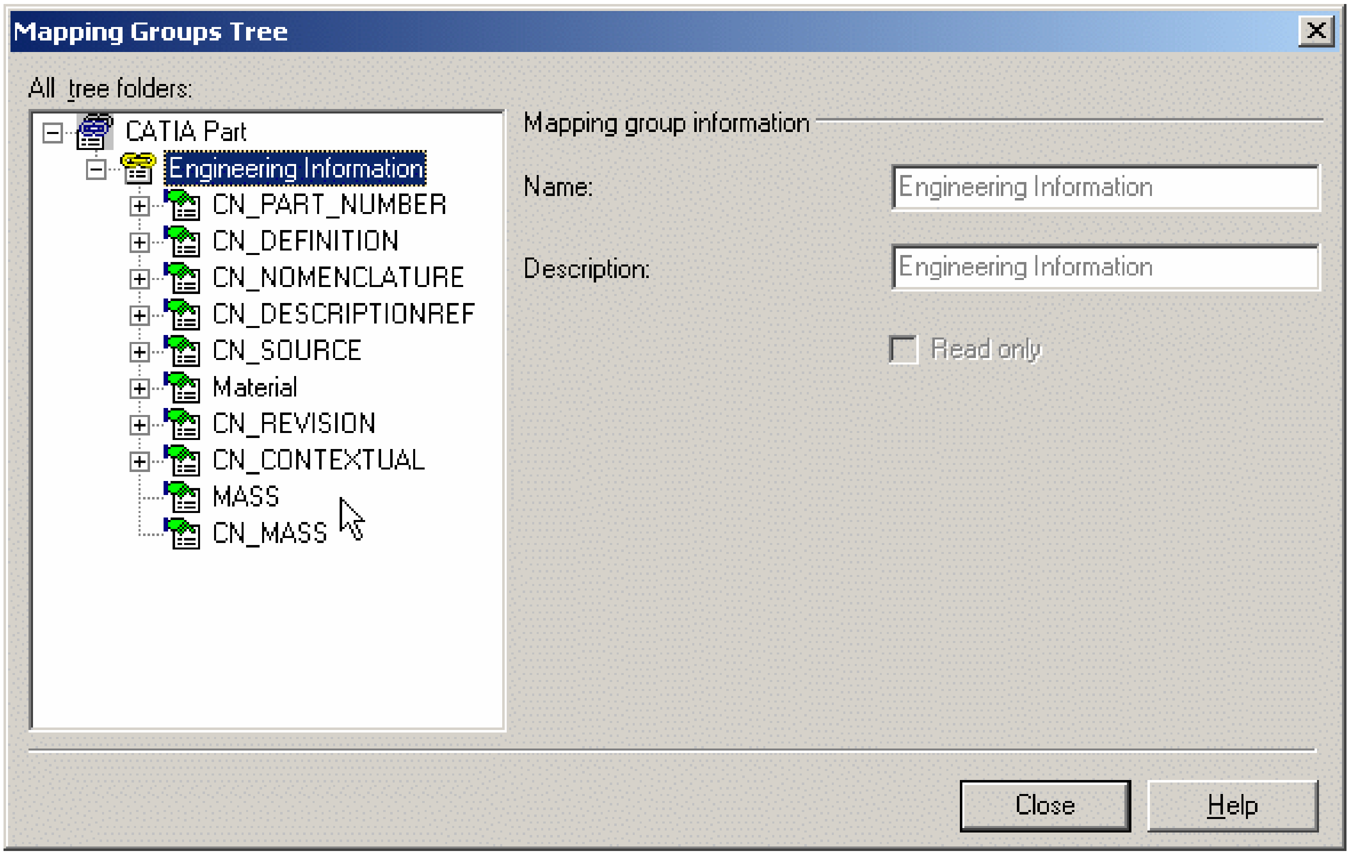
Mapping CATIA MASS Attribute

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, refer to the *SMARTEAM-Editor documentation*.

Managing CATAnalysis Documents

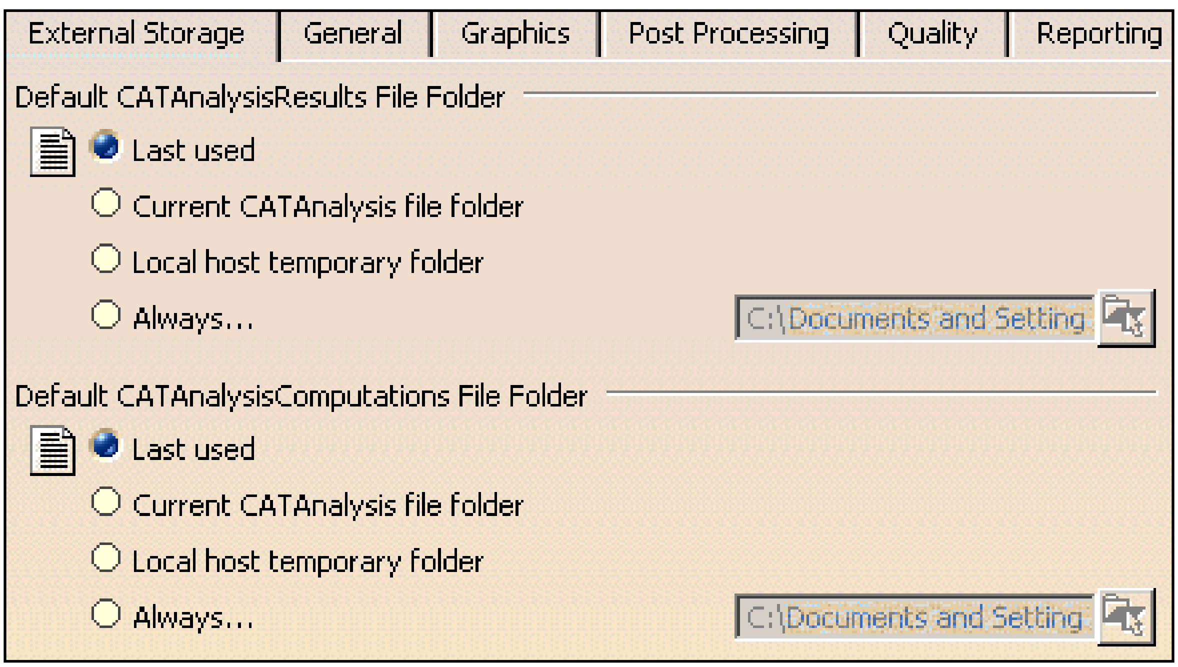
This section is dedicated to system administrators who need to integrate CATAnalysis documents into the SMARTEAM database. Instructions for design engineers are documented in Handling CATAnalysis documents.
For information about mechanical analyses for 3D systems, refer to the *CATIA Generative Structural 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:

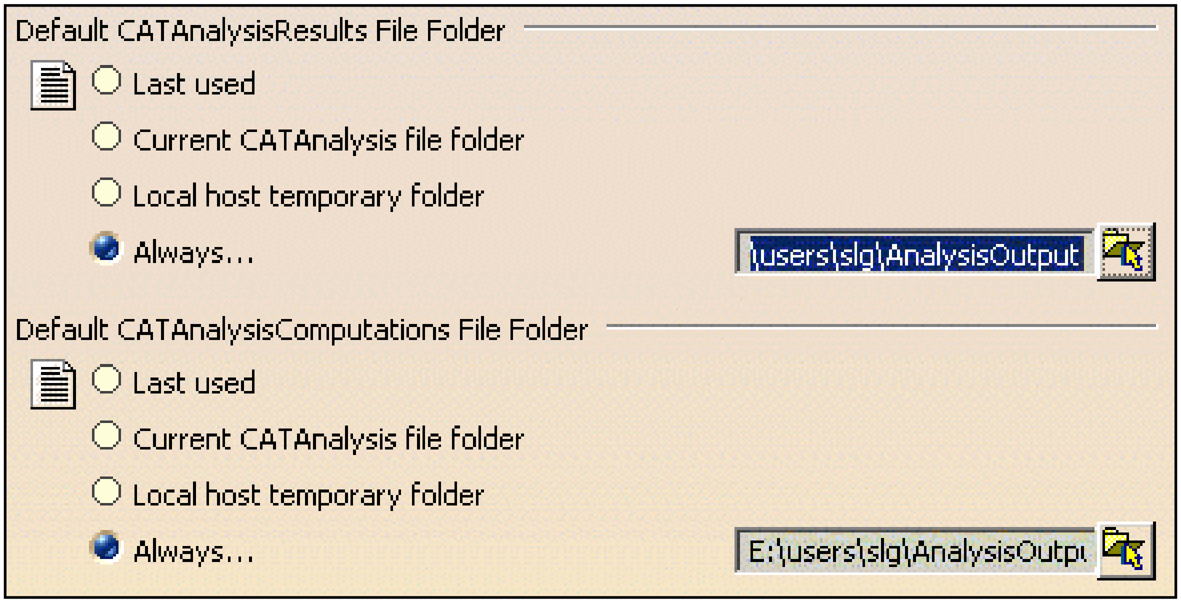
1. Select **Tools-> Options**.
2. From the Options dialog box that is displayed, click the **Analysis & Simulation** category to the left, and then the External Storage tab.

   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.

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:

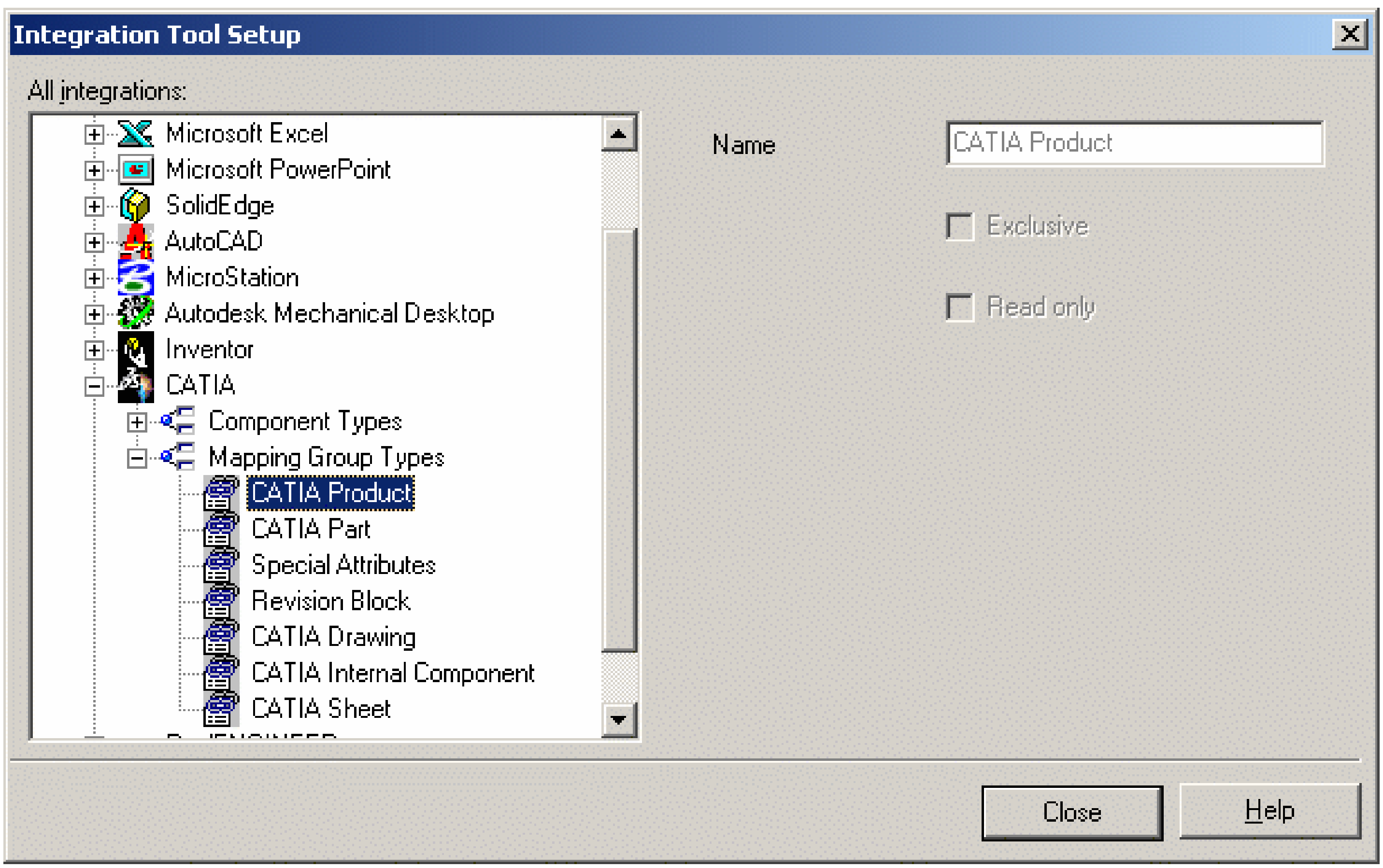
- Integration Tool Setup Utility
- Mapping Direction
- Hardcoded Property Names
- Map a Text Value...
- Performances and Attribute Mapping

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.



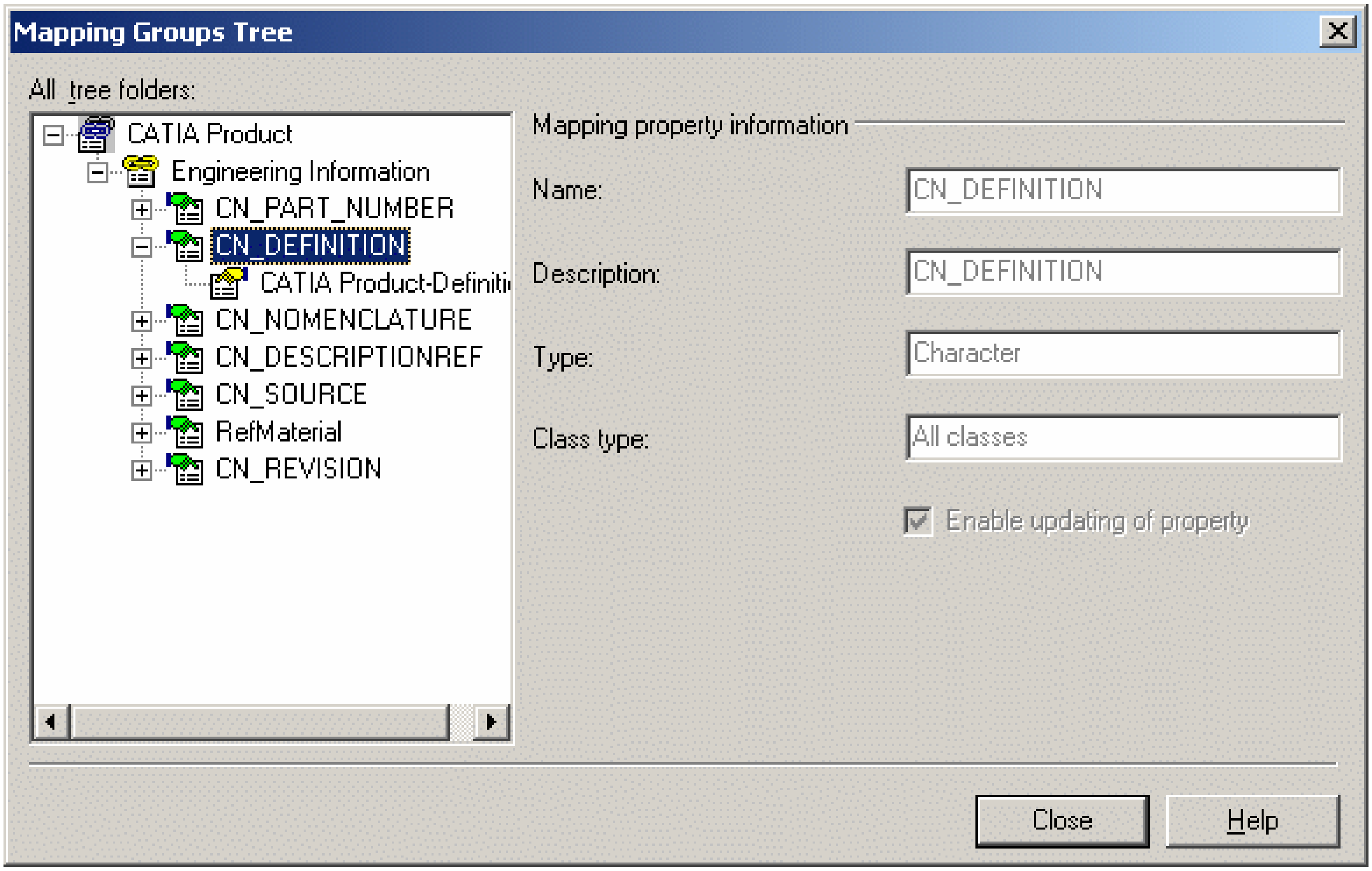
- **CATIA Product**: This indicates that the associated set of mappings are related to documents stored in the CATIA Product class.
- **Read only** : when you define a mapping you also define its direction. **Read only** is one of the options determining the mapping direction.

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

Hardcoded Property Names

Here is a list of the hardcoded 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 *Part Number* field	CATIA Part and CATIA Product
CN_REVISION	read/write	Contents of the *Revision* field	CATIA Part and CATIA Product
CN_DEFINITION	read/write	Contents of the *Definition* field	CATIA Part and CATIA Product
CN_NOMENCLATURE	read/write	Contents of the *Nomenclature* field	CATIA Part and CATIA Product
CN_SOURCE	read/write	Contents of the *Source* field	CATIA Part and CATIA Product
CN_DESCRIPTIONREF	read/write	Contents of the *Description* 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

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).

Performances and Attribute Mapping

Attribute mapping is one of the major element in terms of performances implied for the Save and Open transactions.

Thus, it is advised to clearly identify the needed mapping:

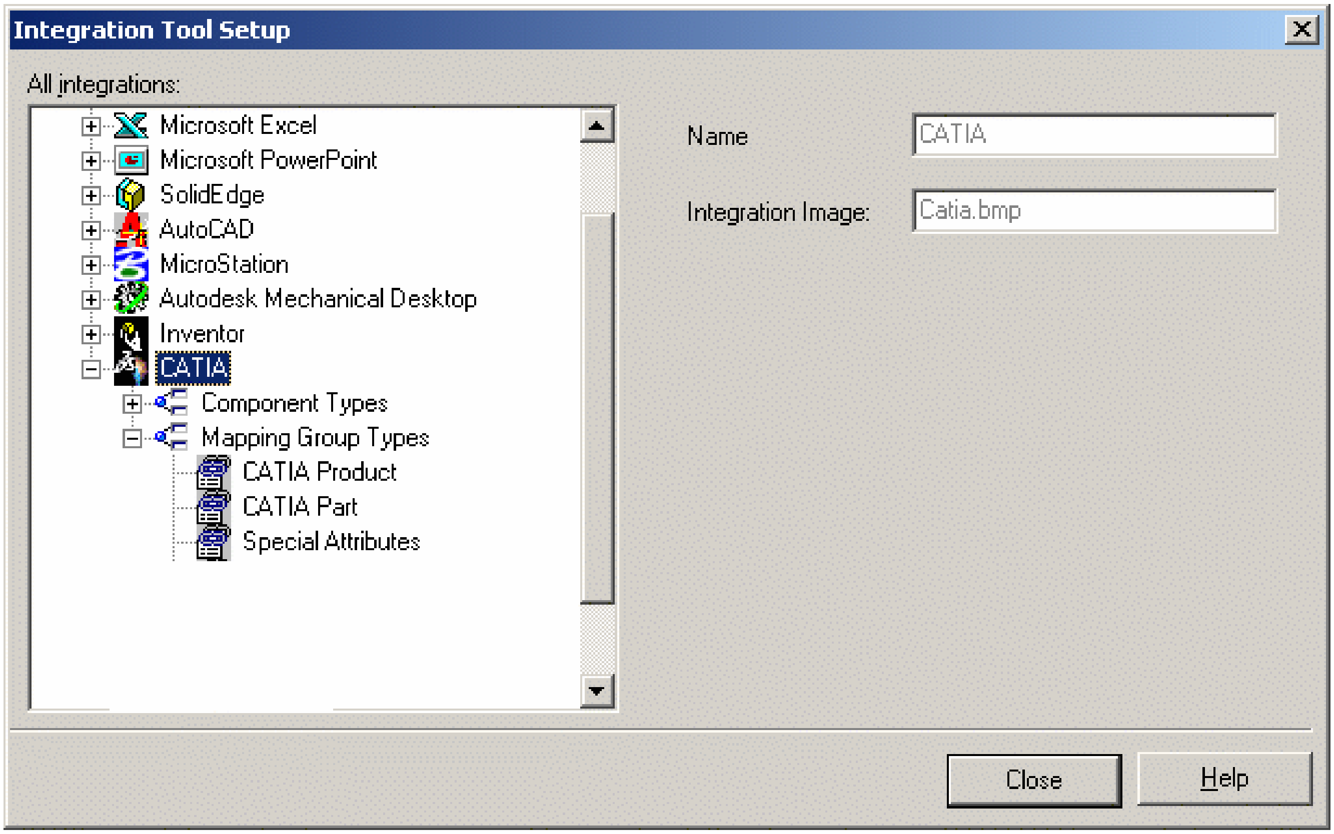
- A previous section describes the importance not to map the mass, volume and surface
- The user should determine whether each type of mapping is needed from CATIA to SMARTEAM, from SMARTEAM to CATIA or from both directions:
- If it is really required, the revision should be mapped only from SMARTEAM 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.

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:

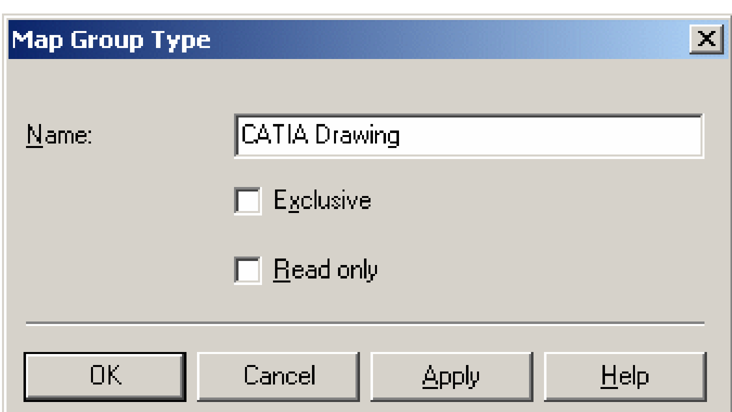


   A **mapping group type** i.e. a set of mapping groups, applicable to a specific SMARTEAM class must now be defined that corresponds to the CATIA Drawing class.

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:

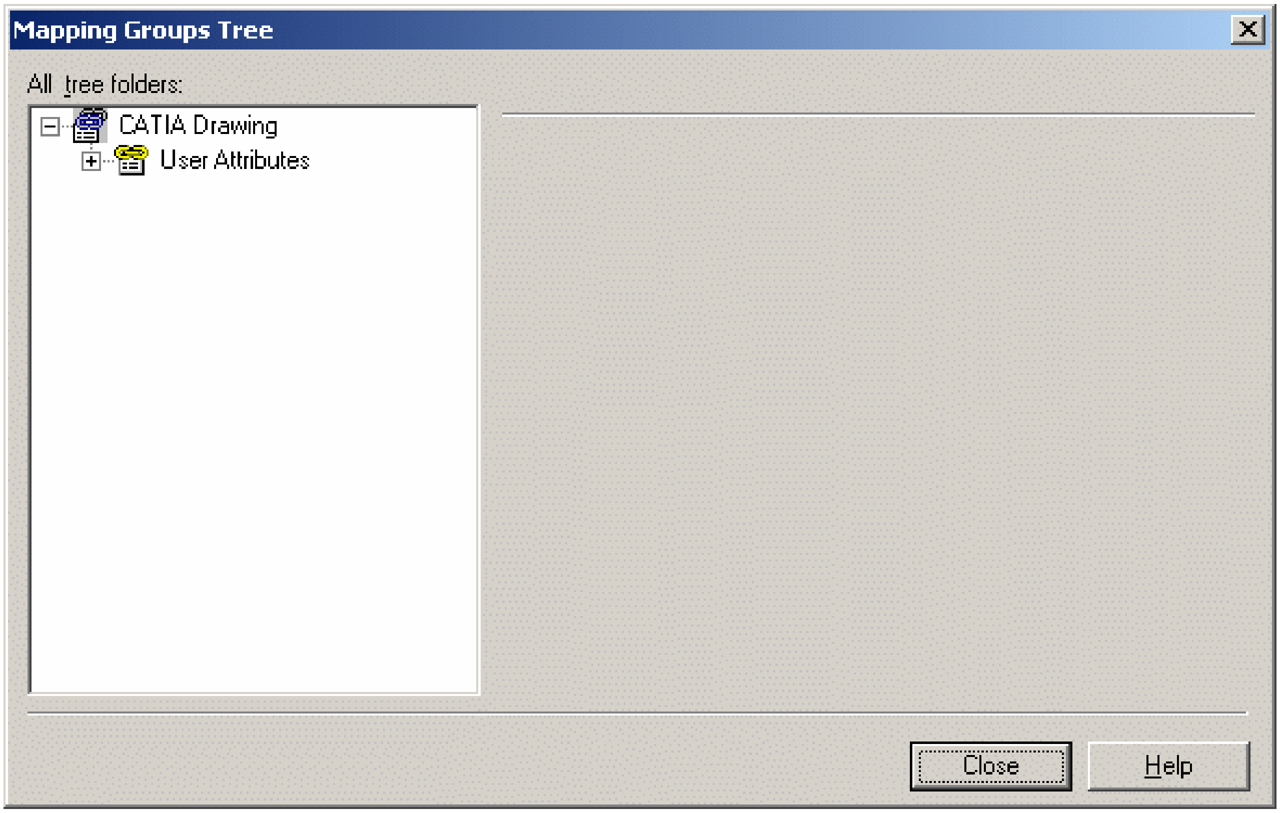


   The name of the mapping group type must be identical to that of the SMARTEAM Class it applies to.

4. Click **OK**.

5. In the tree, right-click on the **CATIA Drawing** item and select **Open groups tree**.

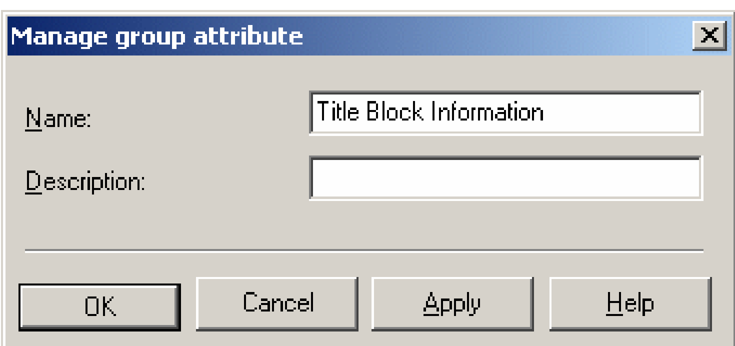
   The Mapping Groups Tree window appears.



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**.

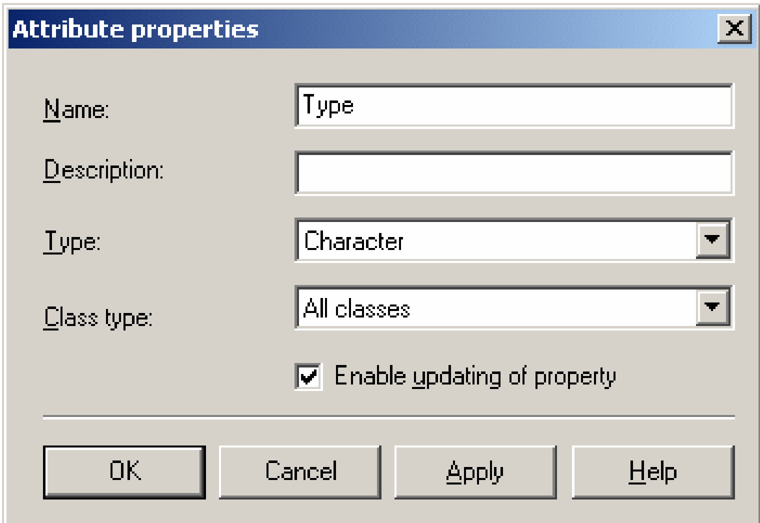
    The Add Title Block Information property dialog box appears.

11. Enter Type in the **Name:** field.

The name of the mapping property must be identical to the corresponding CATIA Formula property .

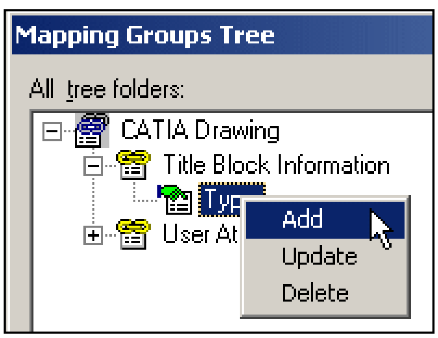
12. Select **Enable updating of property** if not already selected.

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



13. Click **OK.**

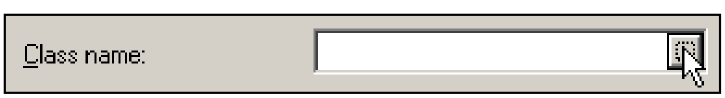
14. Go to the tree and right-click on **Type** then select **Add**.

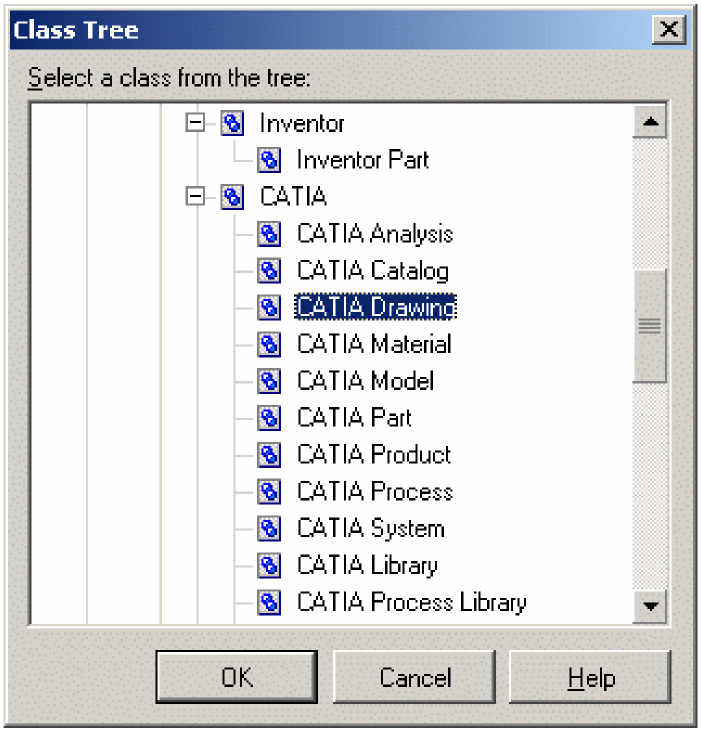


This defines as *Drawing Type* the SMARTEAM attribute to be linked to the CATIA Formula property .

The Attribute mapping dialog box appears.

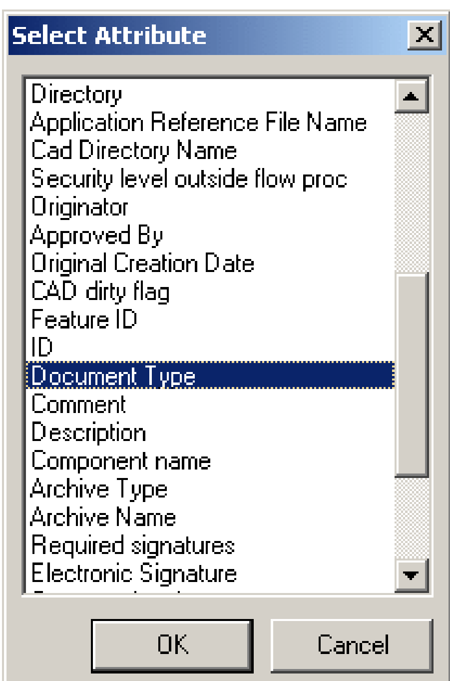
15. Click the button to the right of the **Class name** field and select the class name **CATIA Drawing** from the Class Tree window that appears.

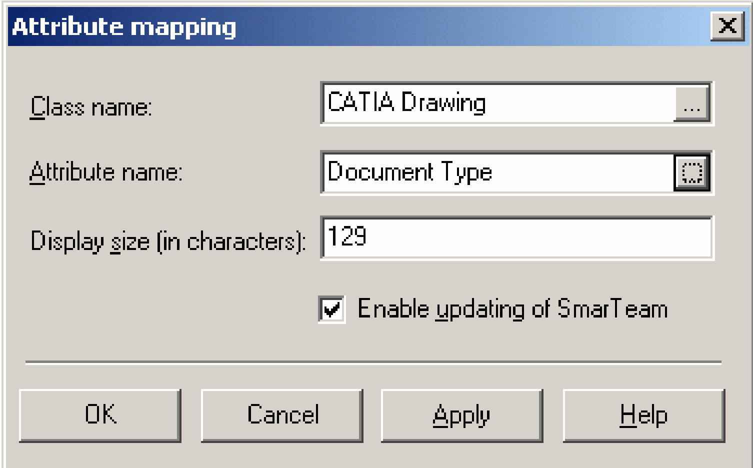




16. Repeat the previous operation for the right of the **Attribute name** field: select the attribute name **Document Type** from the Select Attribute window that appears.

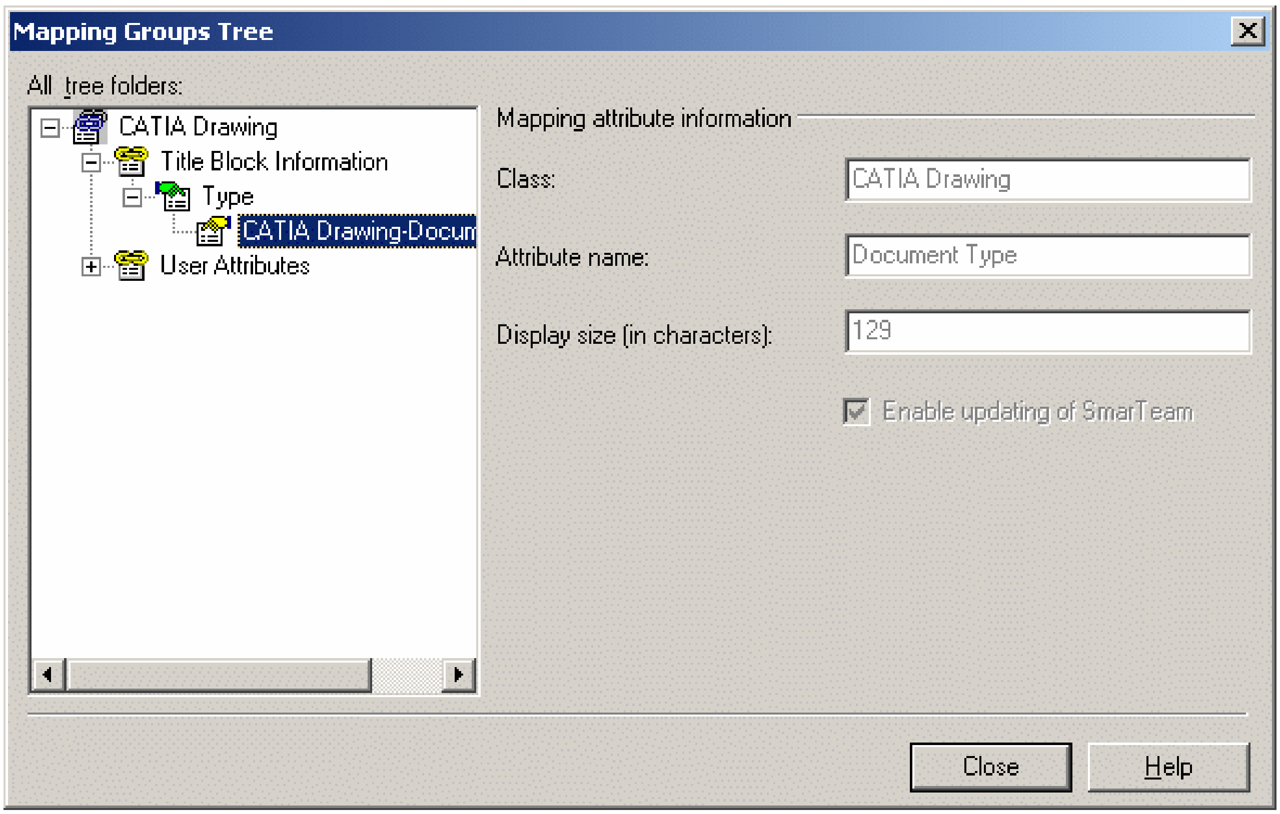
Note that the list of the possible attributes is specific to the selected class name.





17. Click **OK**.

The Mapping Groups Tree window appears:



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

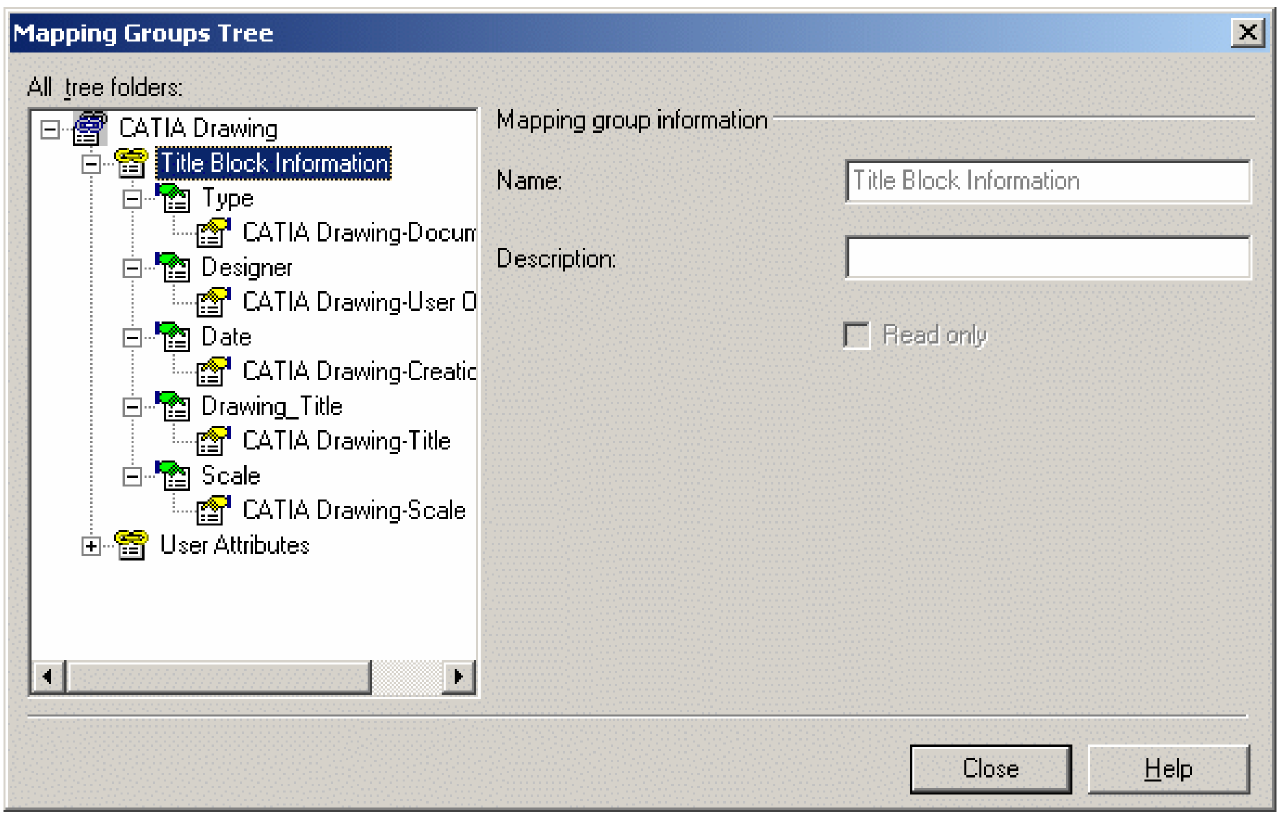
Mapping Property Name	**Enable Property Update**	**Mapping Attribute Name**	**Enable SMARTEAM Update**
Type	yes	Document Type	

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:



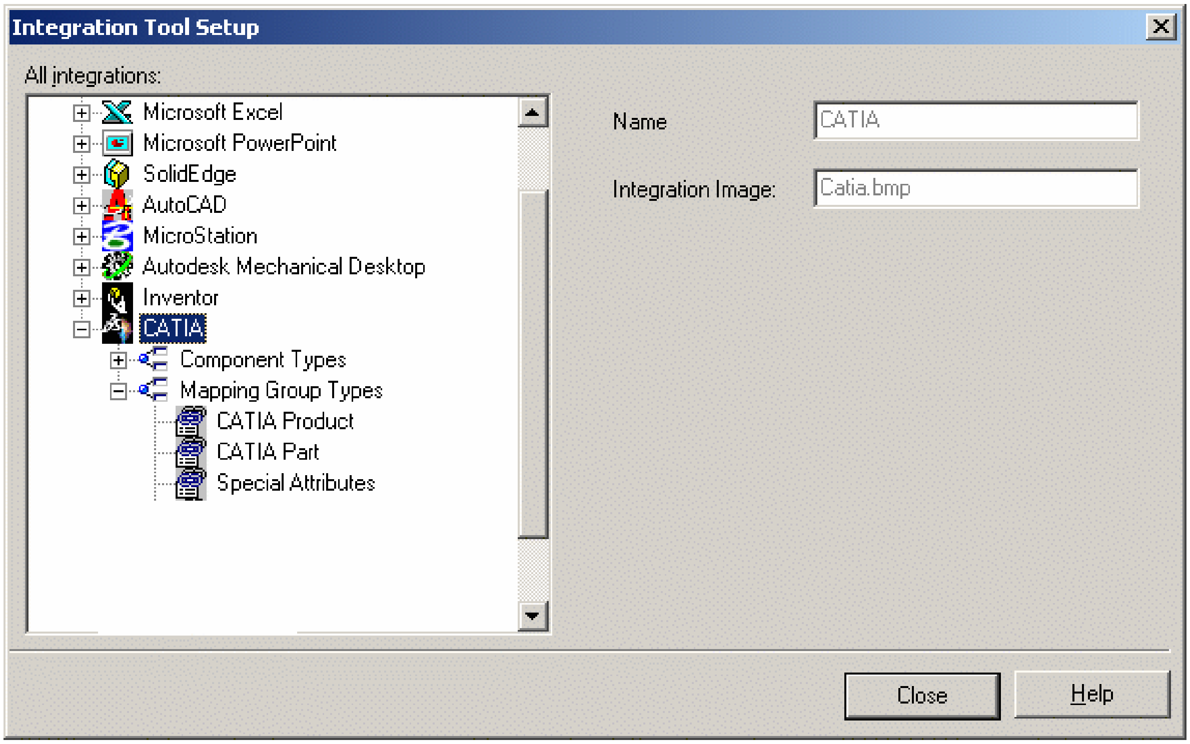
As you can see, mapping of the CATIA properties and the SMARTEAM attributes has now been completed.

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:

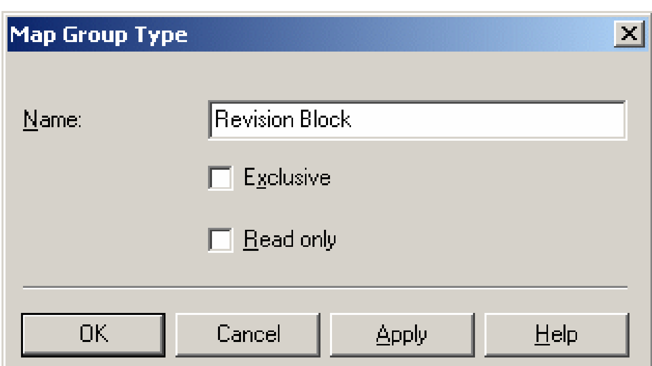


   A **mapping group type** i.e. a set of mapping groups, applicable to a specific SMARTEAM class must now be defined that corresponds to the CATIA Drawing class.

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:

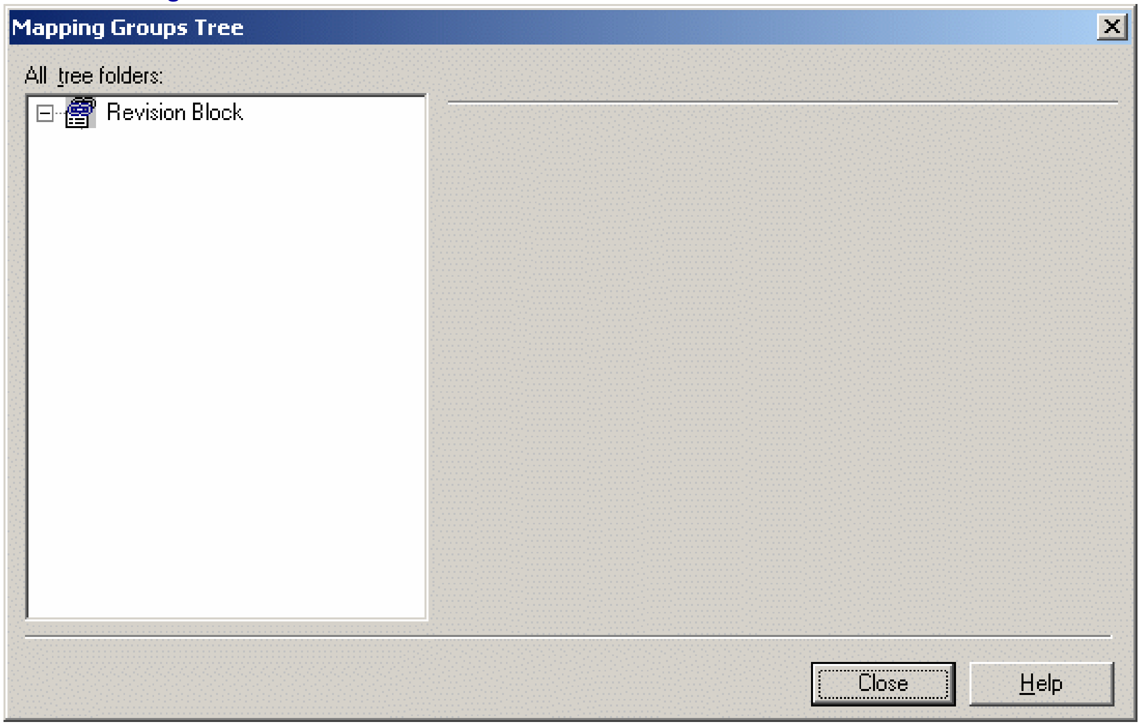


   The name of the mapping group type must be identical to that of the SMARTEAM Class it applies to.

4. Click **OK**.

5. In the tree, right-click on the **Revision Block** item and select **Open Groups Tree**.

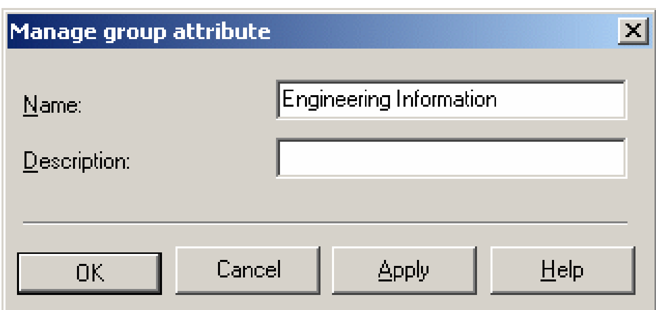
   The Mapping Groups Tree dialog box appears.



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 a meaningful name, for example *Engineering Information*:



   The name used (*Engineering Information* in this particular case) is referenced in the RevisionBlock.bs script. If you want you can subsequently change this name.

8. Click **OK**.

9. To define the CATIA Formula property  as **Type**: Go to the tree and right-click *Engineering Information*

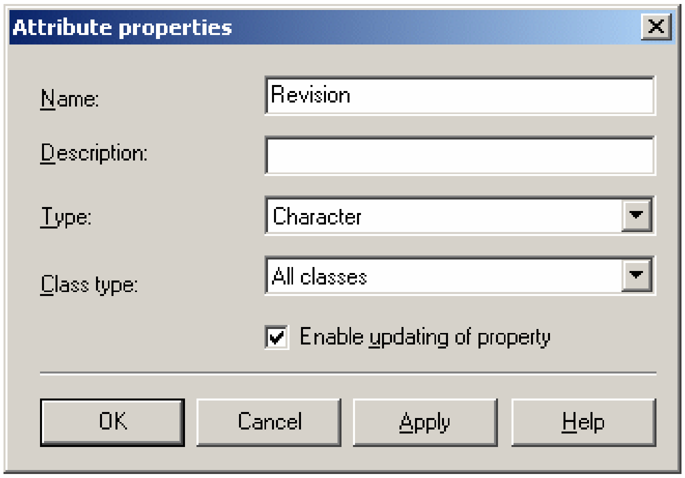
10. Select **Add**.

    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.



13. Click **OK**.

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

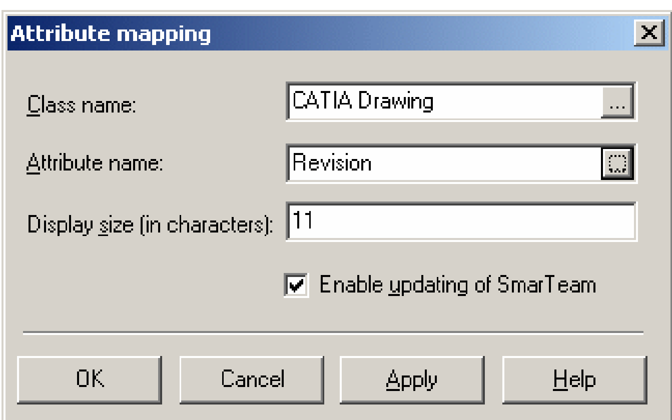
    This defines as **Drawing Type** the SMARTEAM attribute to be linked to the CATIA Formula property 

    The Attribute Mapping dialog box appears.

15. Select the class name **CATIA Drawing**.

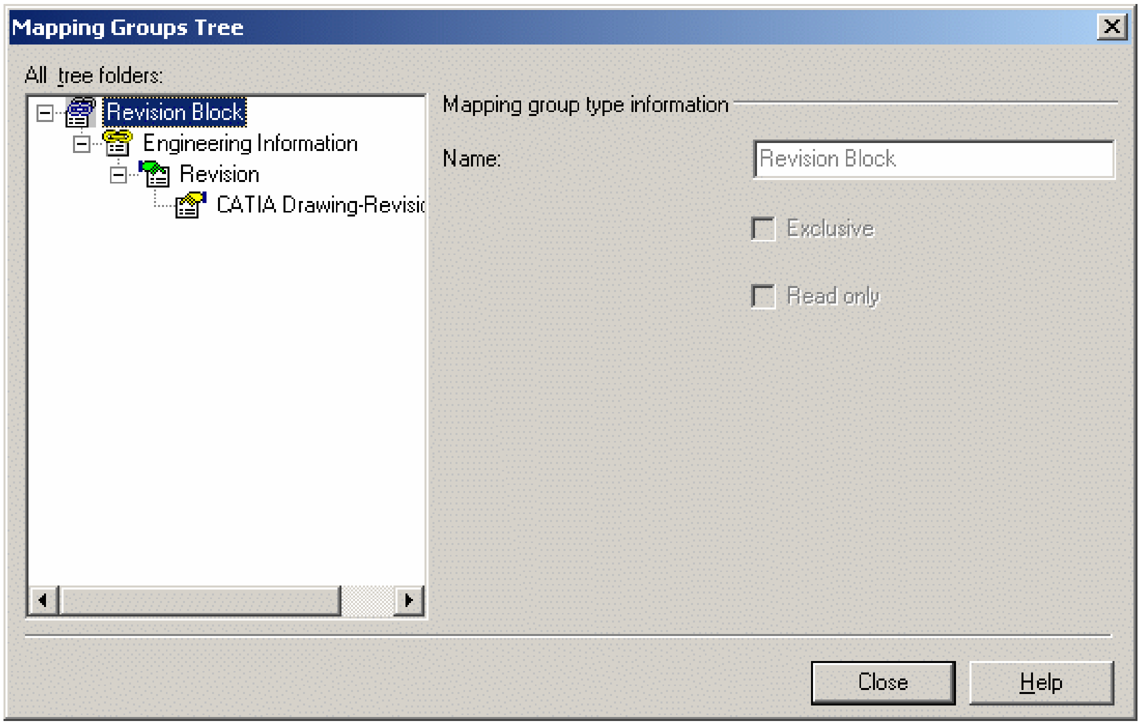
16. Select the attribute name **Revision**.

    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.

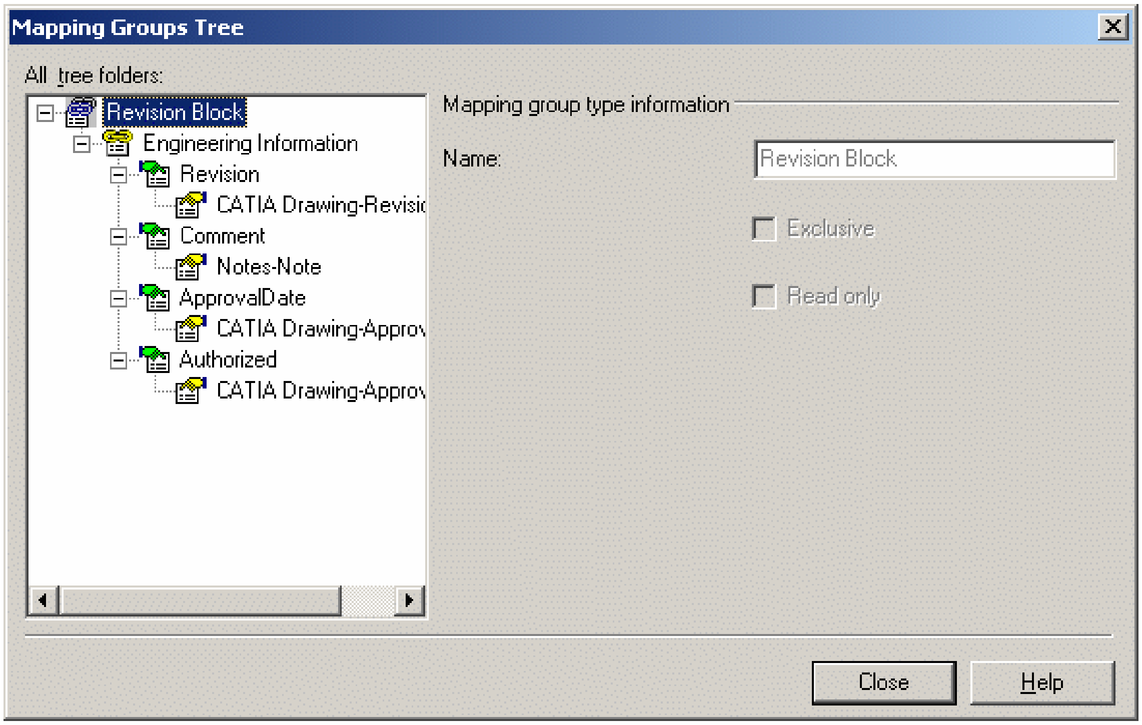
Mapping Property Name	Enable Property Update	Mapping Attribute Name	Enable SMARTEAM Update
Revision	yes	Revision	

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.

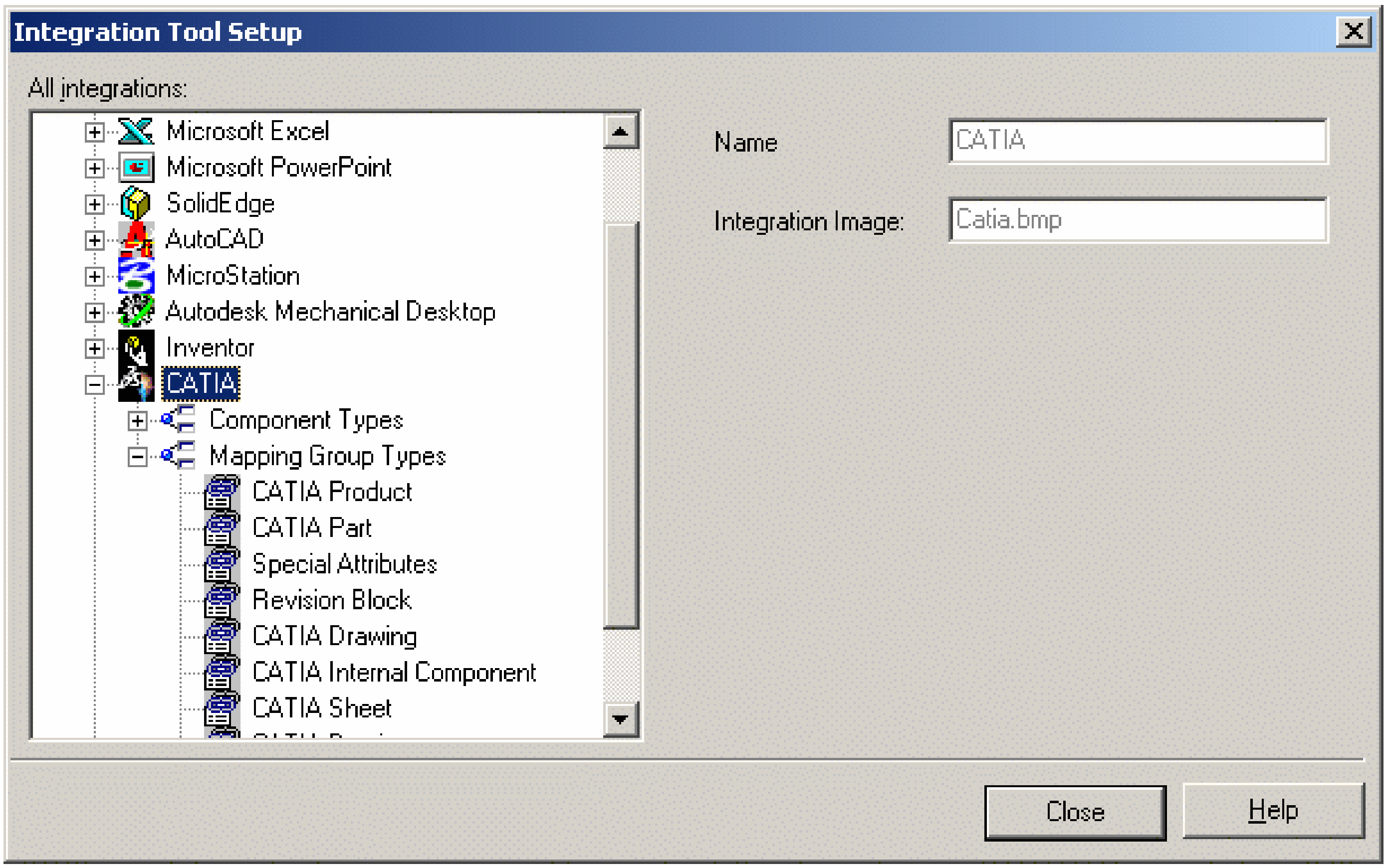
It is assumed that the properties used for this task have already been created in the database under the *CATIA Product* and *CATIA Part* classes.

1. Launch the Integration Tool Setup utility.

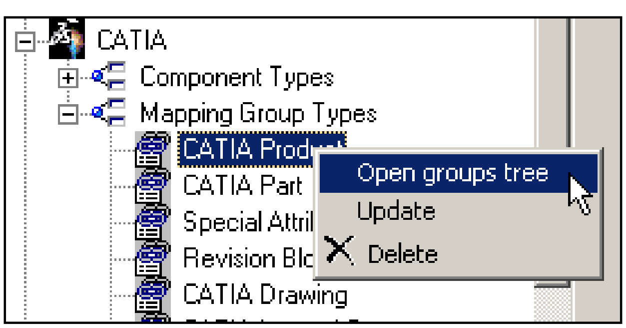
   The Integration Tool Setup dialog box appears.

2. Double-click on CATIA or select the "+" sign to display the options associated with CATIA.

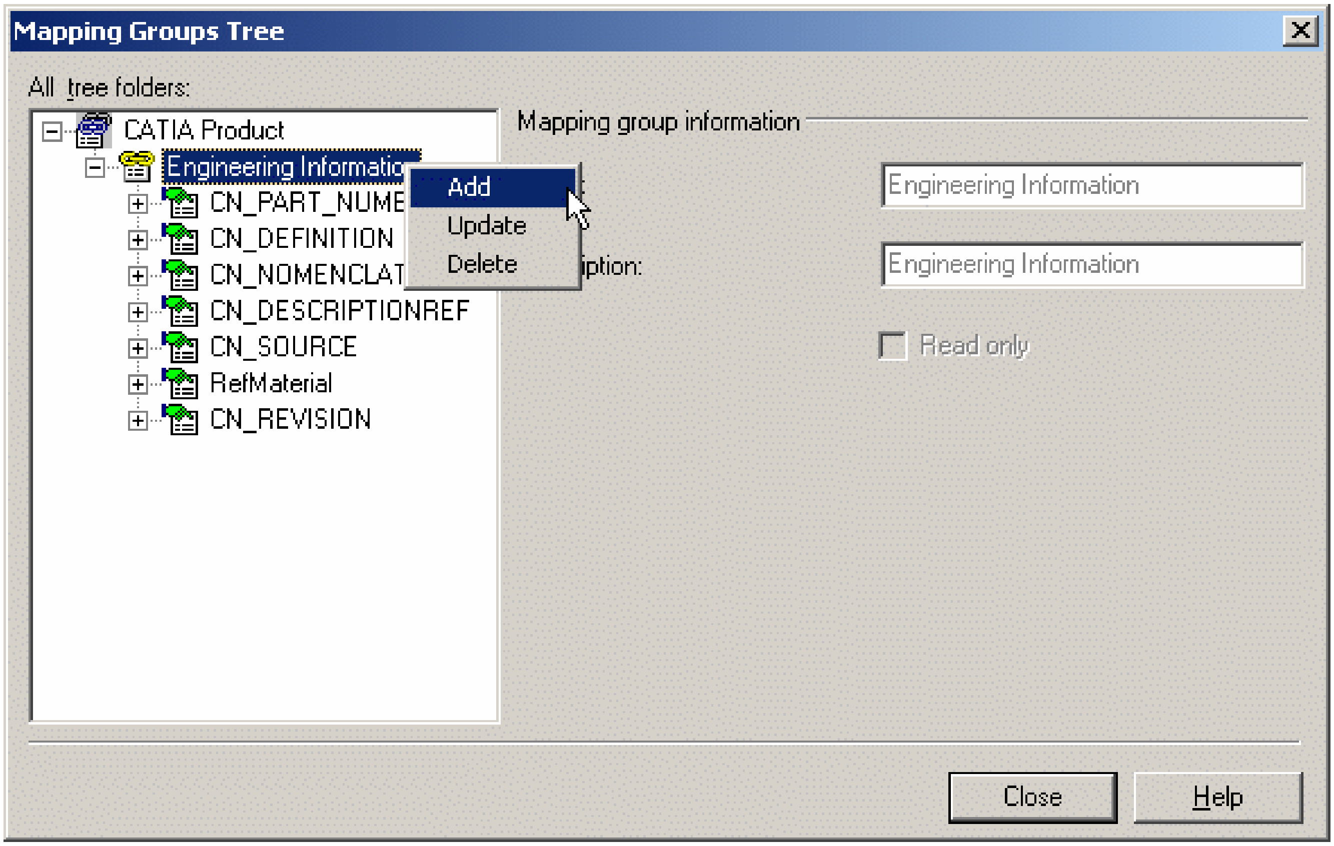
3. Double-click **Mapping Group Types** or select the "+" sign in front of it.



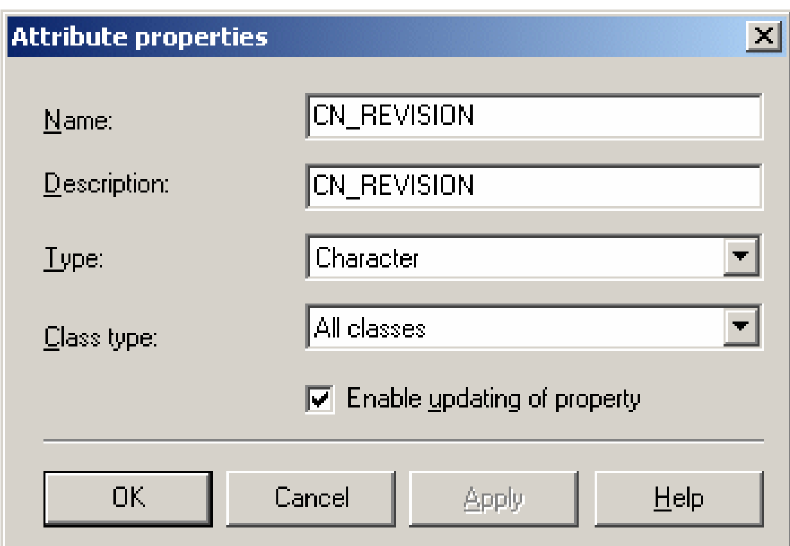
4. Select the **CATIA Product** mapping group type then right-click and select **Open groups tree**:



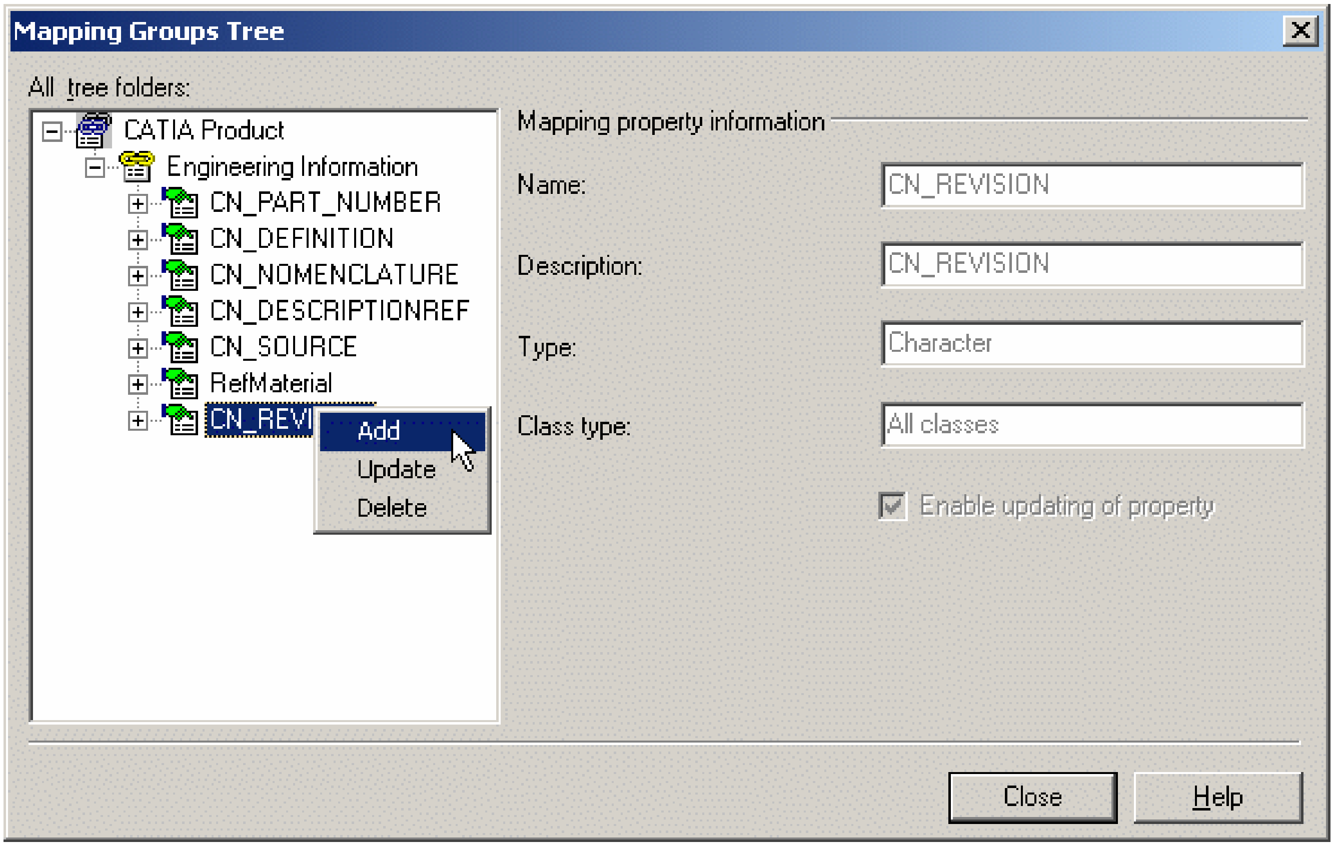
5. In the Mapping Groups Tree window, select the **Engineering Information** item then right-click and select **Add**.



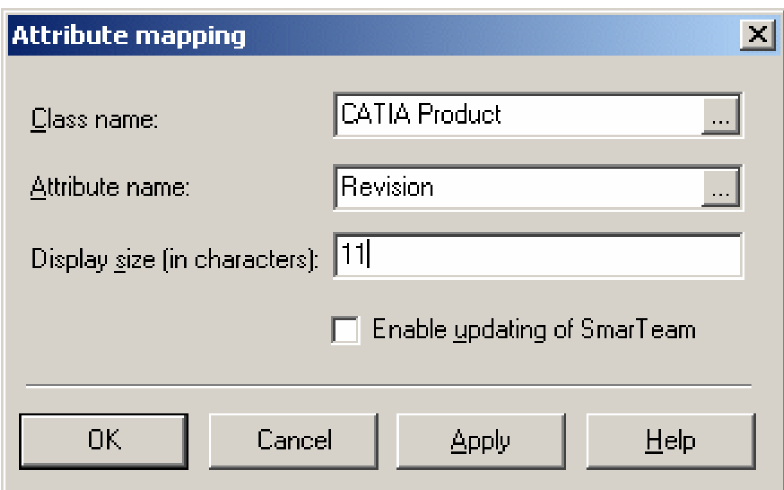
6. In the Attribute Properties dialog box, complete all fields as shown below, then click on **OK**:



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**.



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.

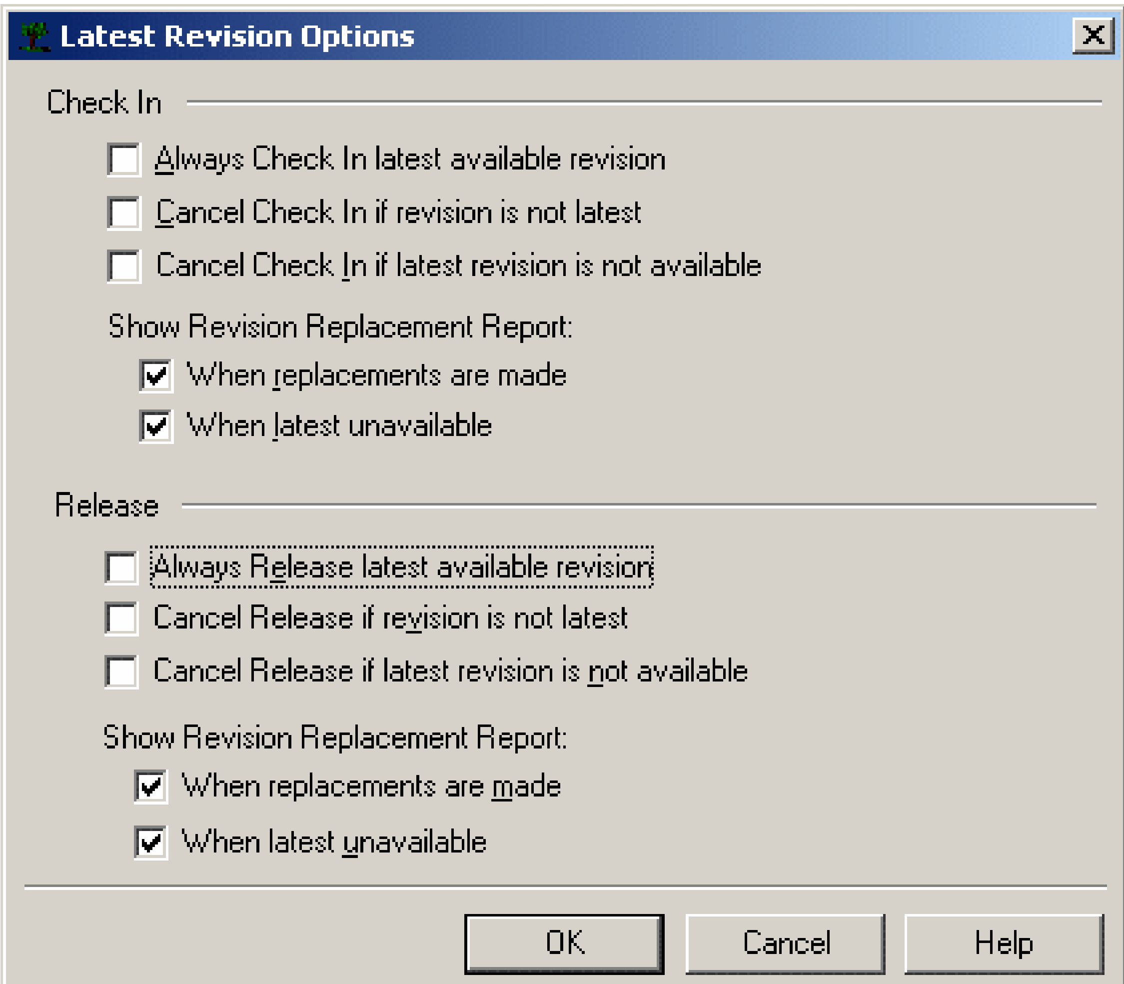
2. Click the **Into vault** tab.

3. Click **Latest Revision...**

   The Latest Revision Options dialog box is displayed.

4. Unselect:

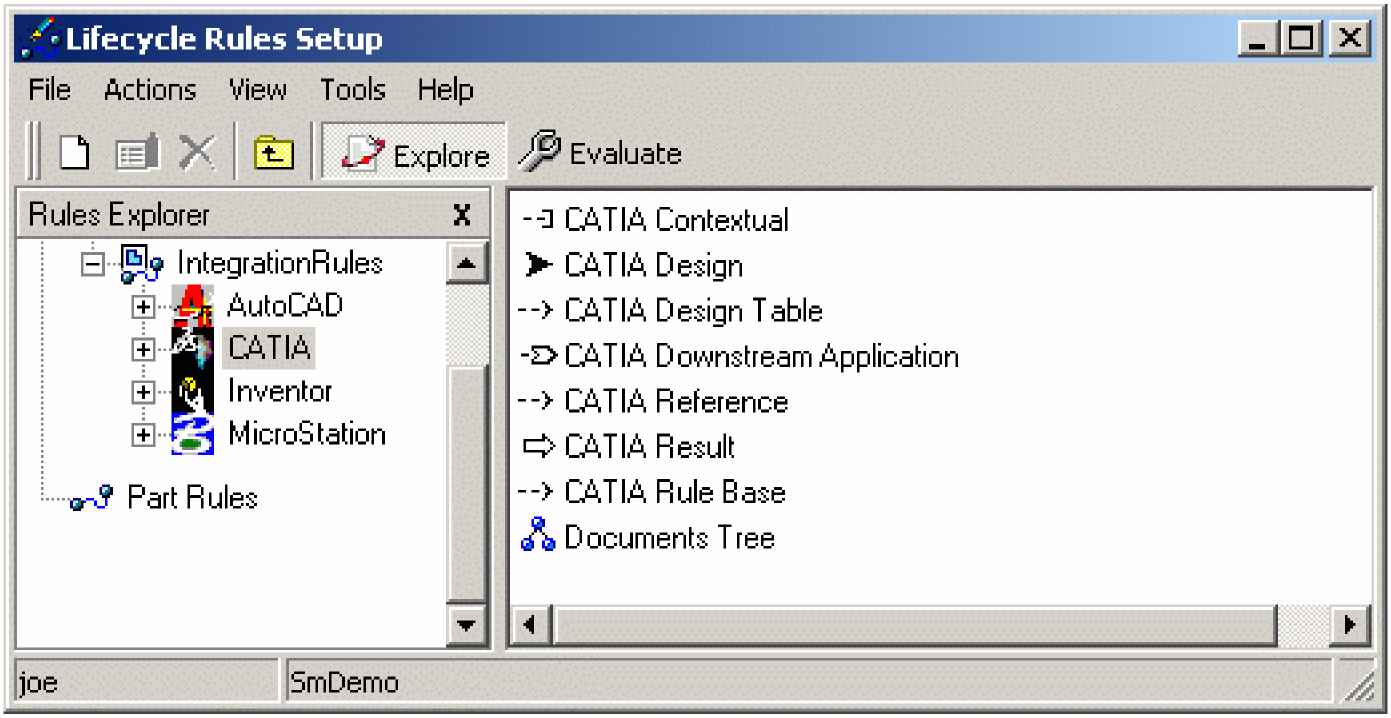
   - **Always Check In latest available revision**
   - **Always Release latest available revision**.



Lifecycle Rules Setup

The 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.

The 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:

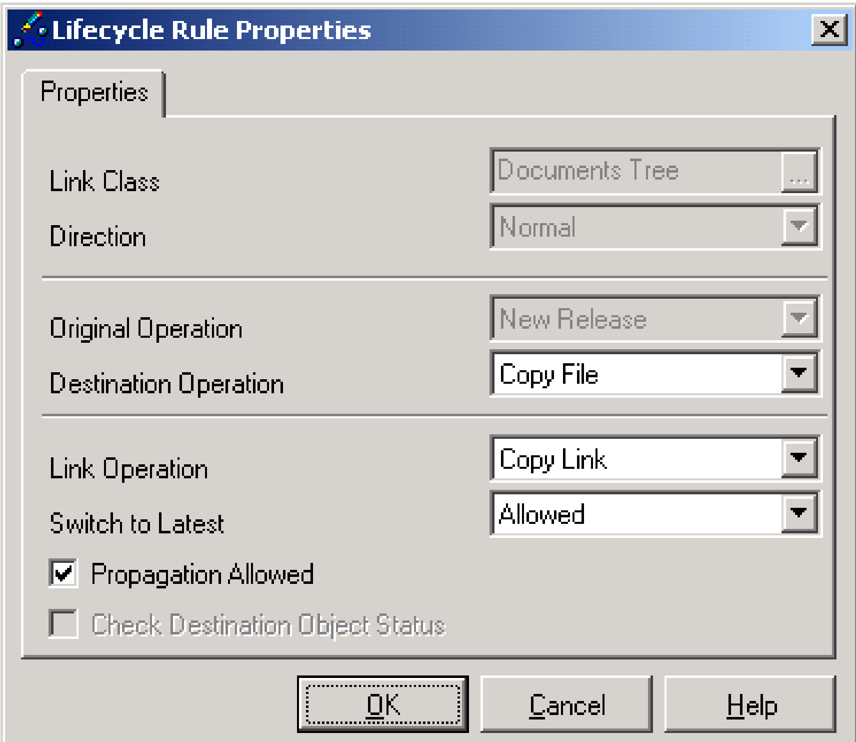
- Review default rules
- Modify rules provided by specific Integration applications
- Add, remove or modify custom rules

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, refer to Lifecycle Operations.

Lifecycle Operations

All life cycle operations on CATIA documents must be performed through CATIA user interface. Life Cycle 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: • **Normal** or • **Reverse**
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.

Destination Operation	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.
Link Operation (Copy Link / No operation)	This parameter specifies how the system should handle linked objects between two objects when new revisions are created. **Copy Link** means copy the link between old revisions with all its attribute values to new objects, and **No operation** means create new revisions without creating a link between them. This option is available only when the original operation is in **Check Out** or **New Release** mode. • When you define **No operation**, 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. • When you set set **Link Operation** to **Copy Link**, SMARTEAM enables you to perform this operation between the original and the destination objects.
Link Operation (Copy link / No operation)	This parameter specifies how the system should handle linked objects between two objects when new revisions are created. **Copy Link** means copy the link between old revisions with all its attribute values to new objects, and No operation means create new revisions without creating a link between them. This option is available only when the original operation is in Check Out or New Release mode. • When you define **No operation**, 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. • When you set **Link Operation** to **Copy Link**, SMARTEAM enables you to perform this operation between the original and the destination objects.
Switch to Latest	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 **Allowed**, SMARTEAM allows you to switch to the latest revision of the object. This option is not available when the original operation is **Obsolete**.

Propagation Allowed	Defines whether to allow the Propagation operation to the destination object, if you want to perform a propagation operation during the Lifecycle operation. This option is not available when the original operation is **Obsolete**.
Check Destination Object Status	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 **Check In** or **Release** mode.

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 to 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.

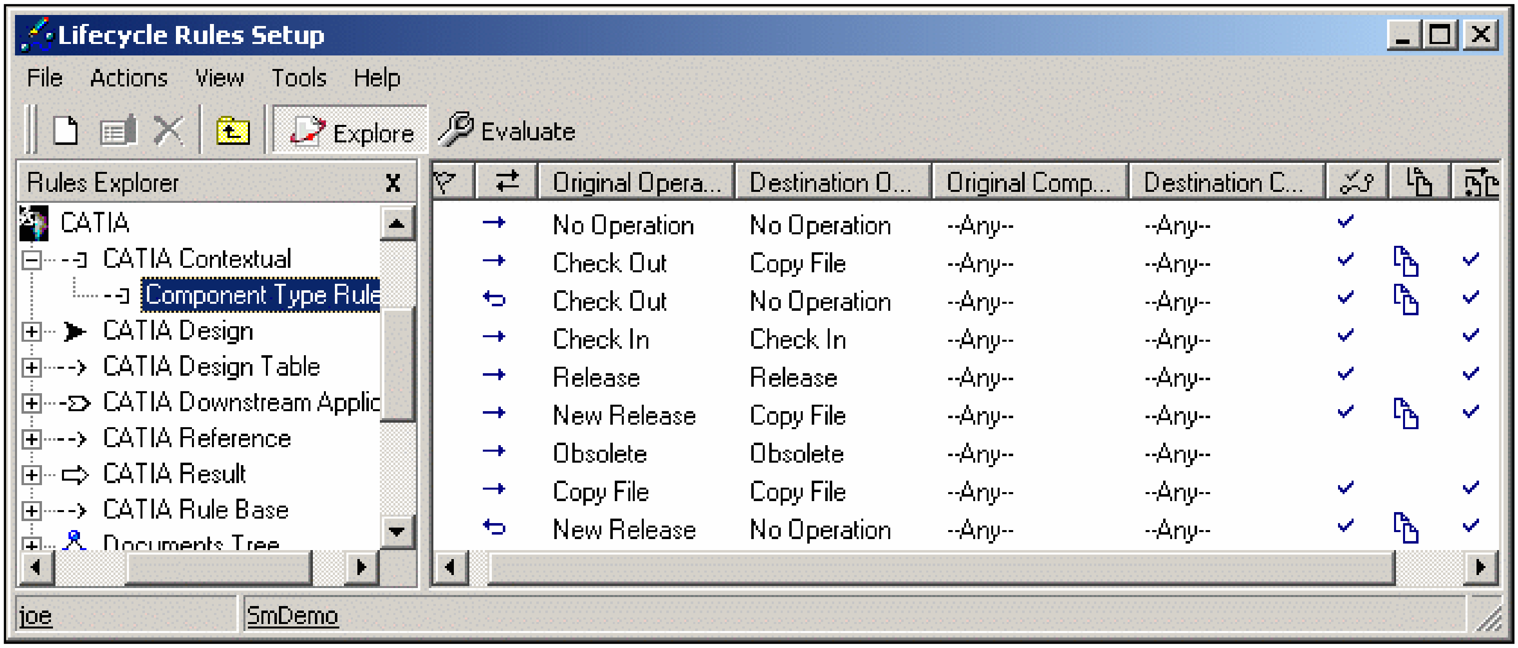
Downstream Application

- **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 to Check Out for checking out and to New Release for new release.

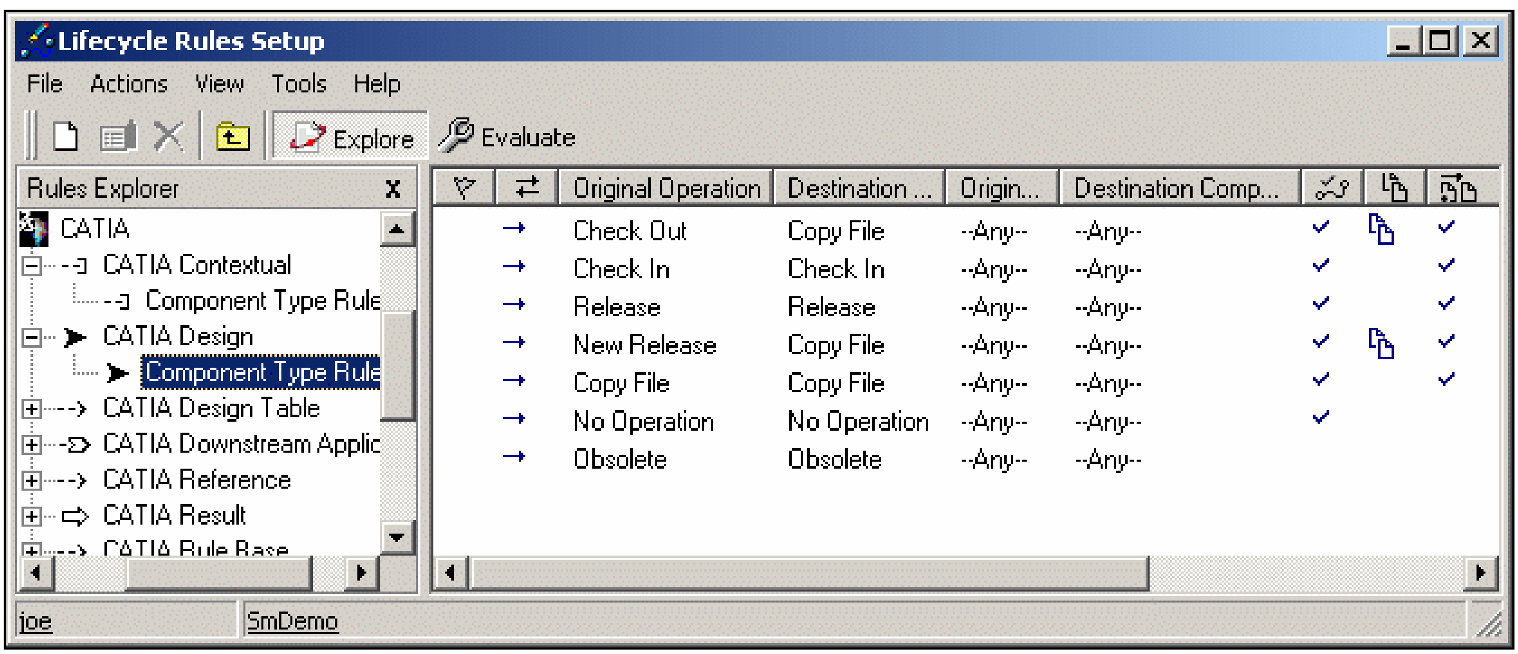
CATIA Link Default Settings

The following settings are defined in the supplied SmDemo database.

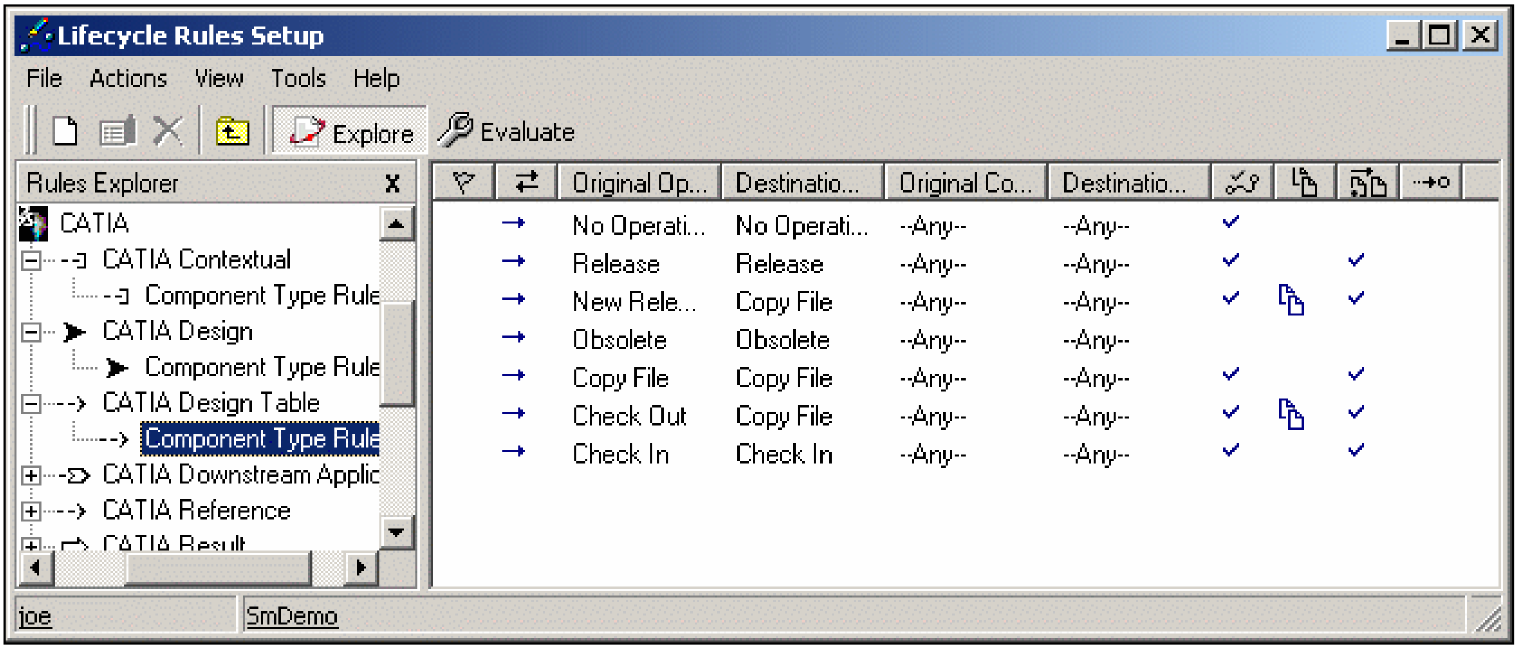
CATIA Contextual



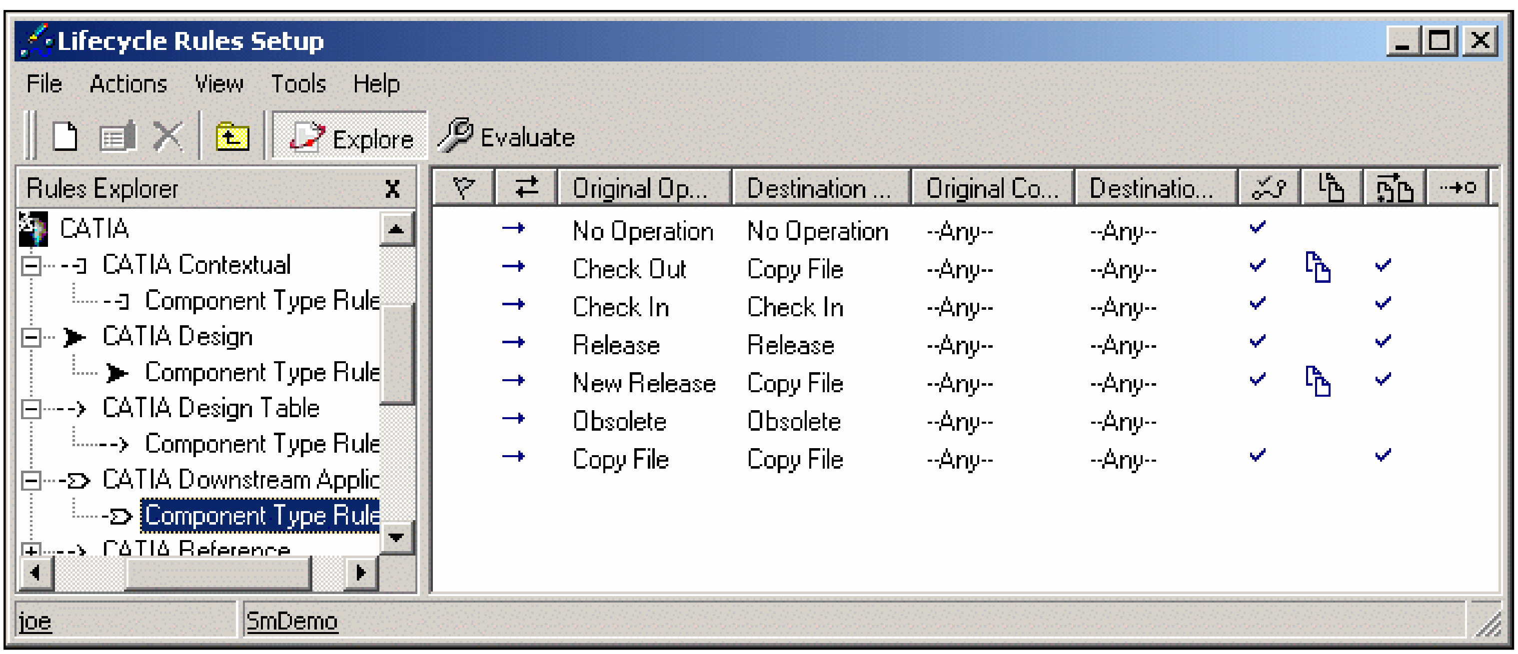
CATIA Design



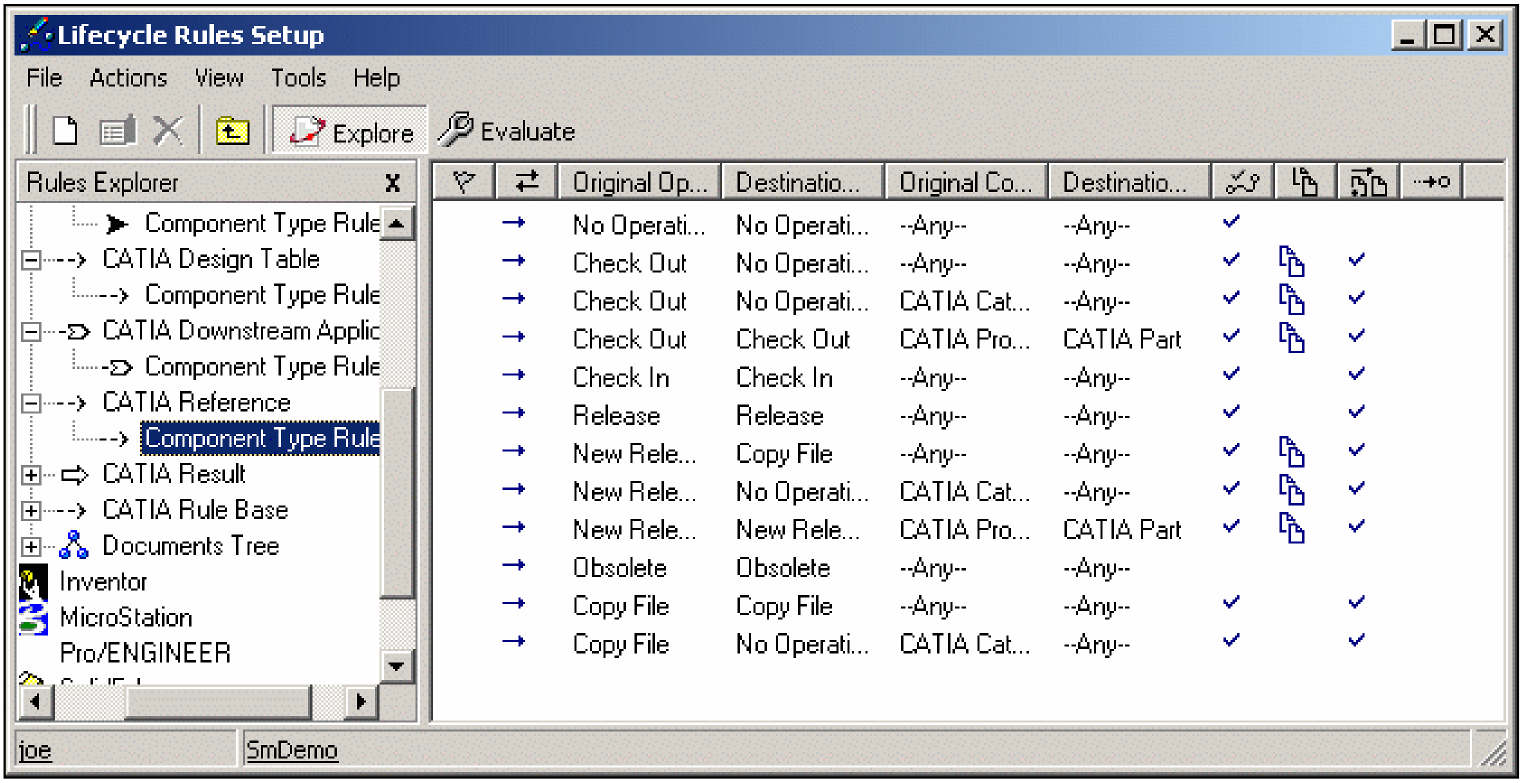
CATIA Design Table



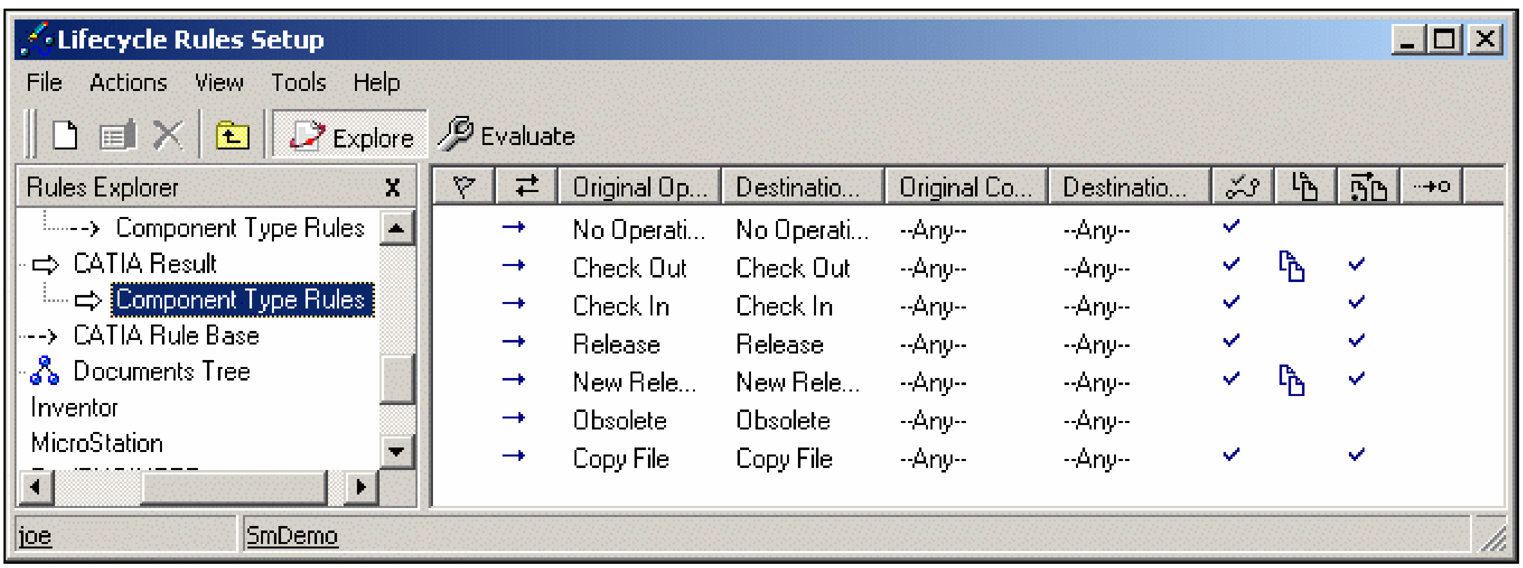
CATIA Downstream Application



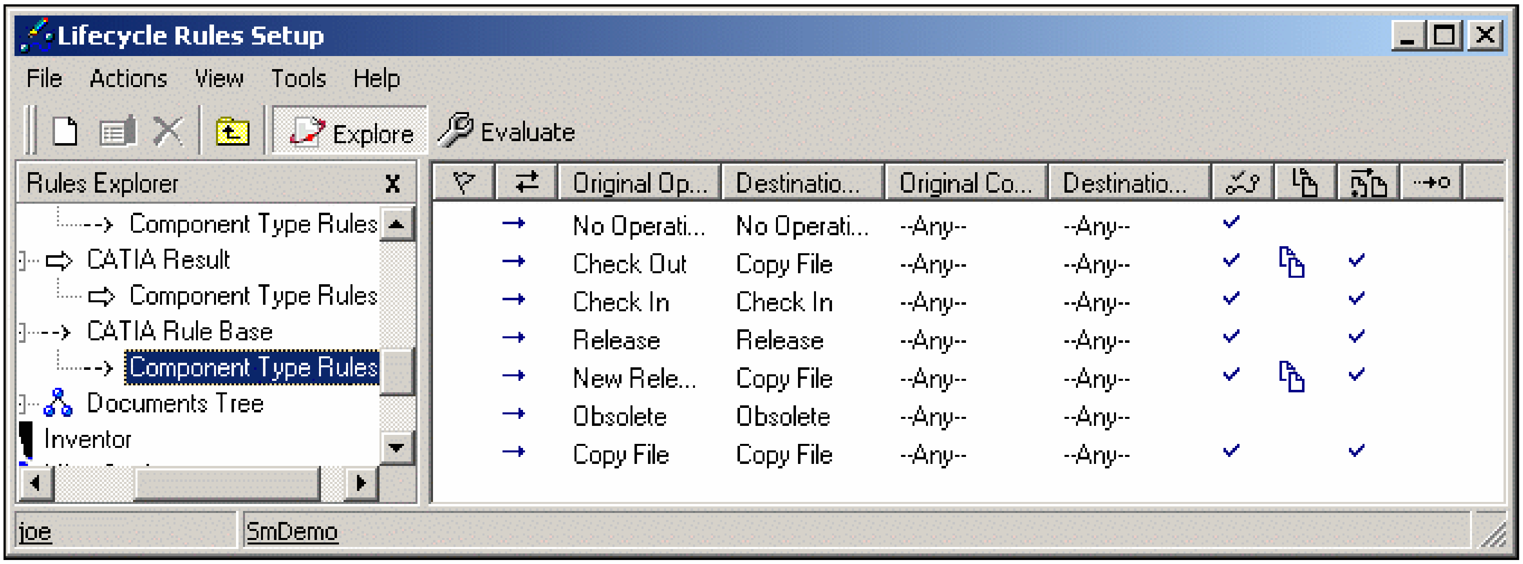
CATIA Reference



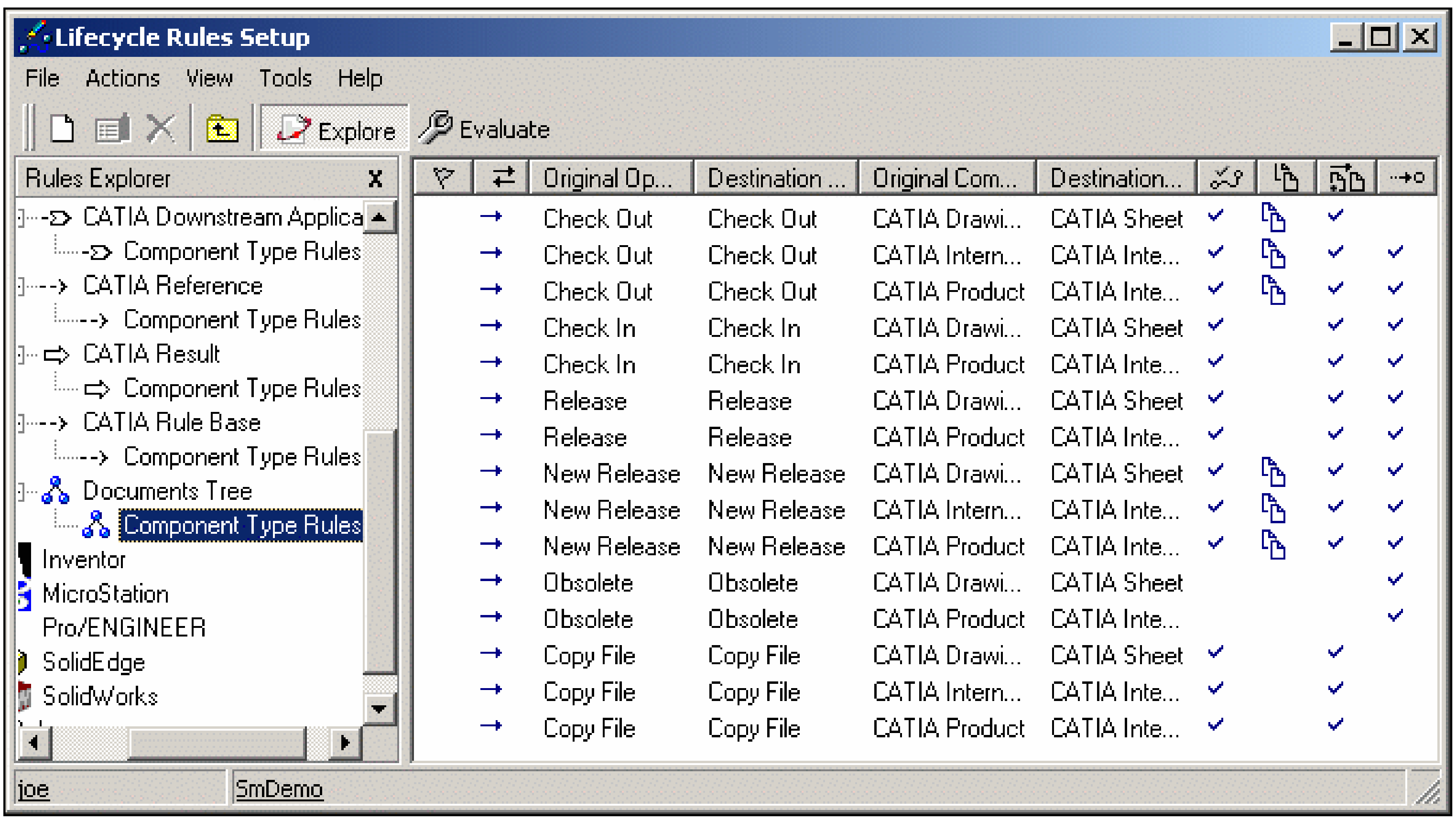
CATIA Result



CATIA Rule Base

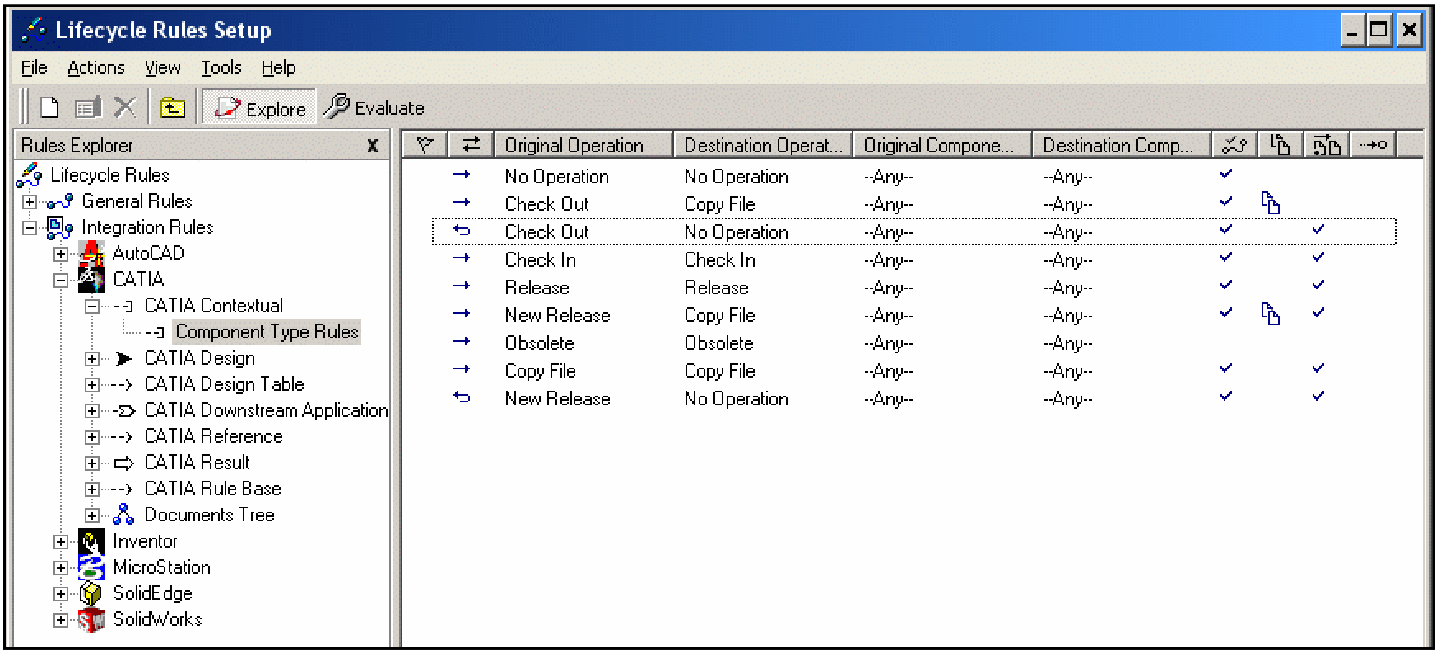


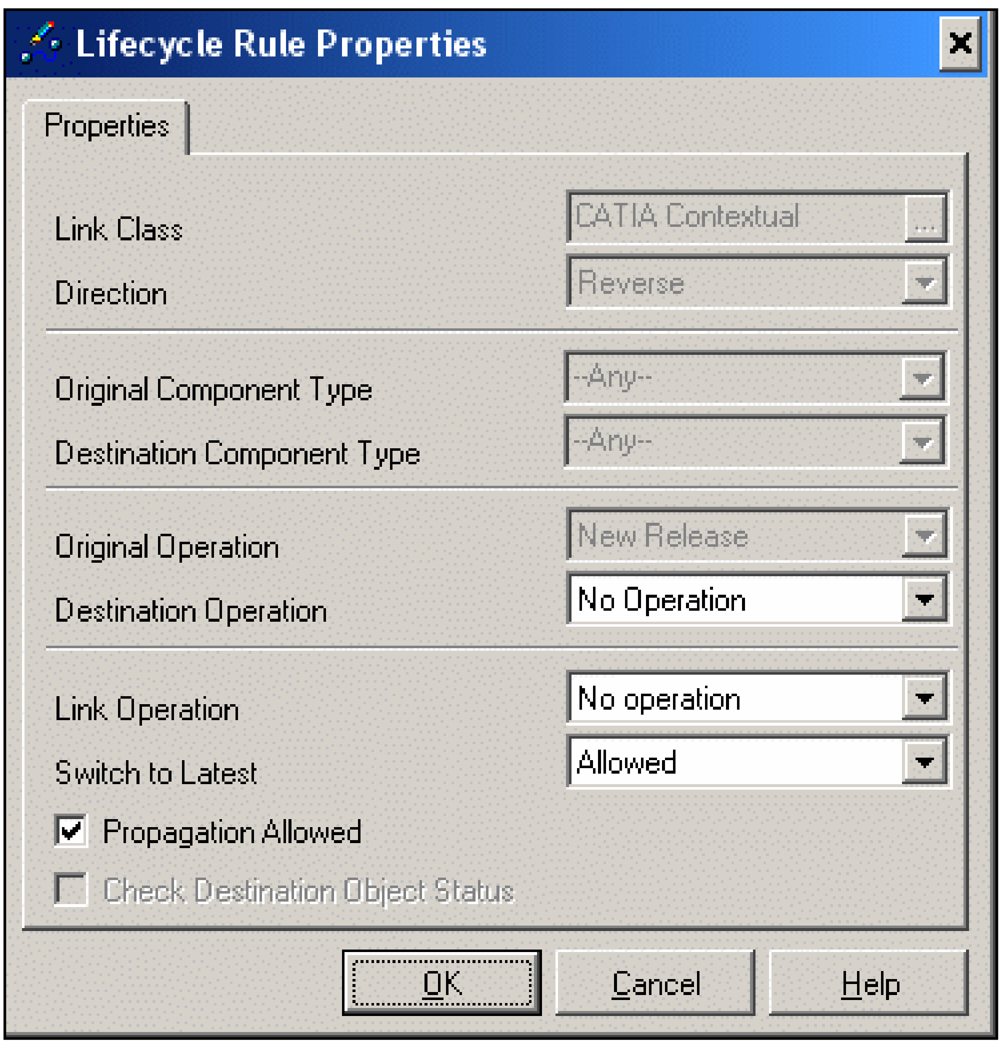
CATIA Documents Tree

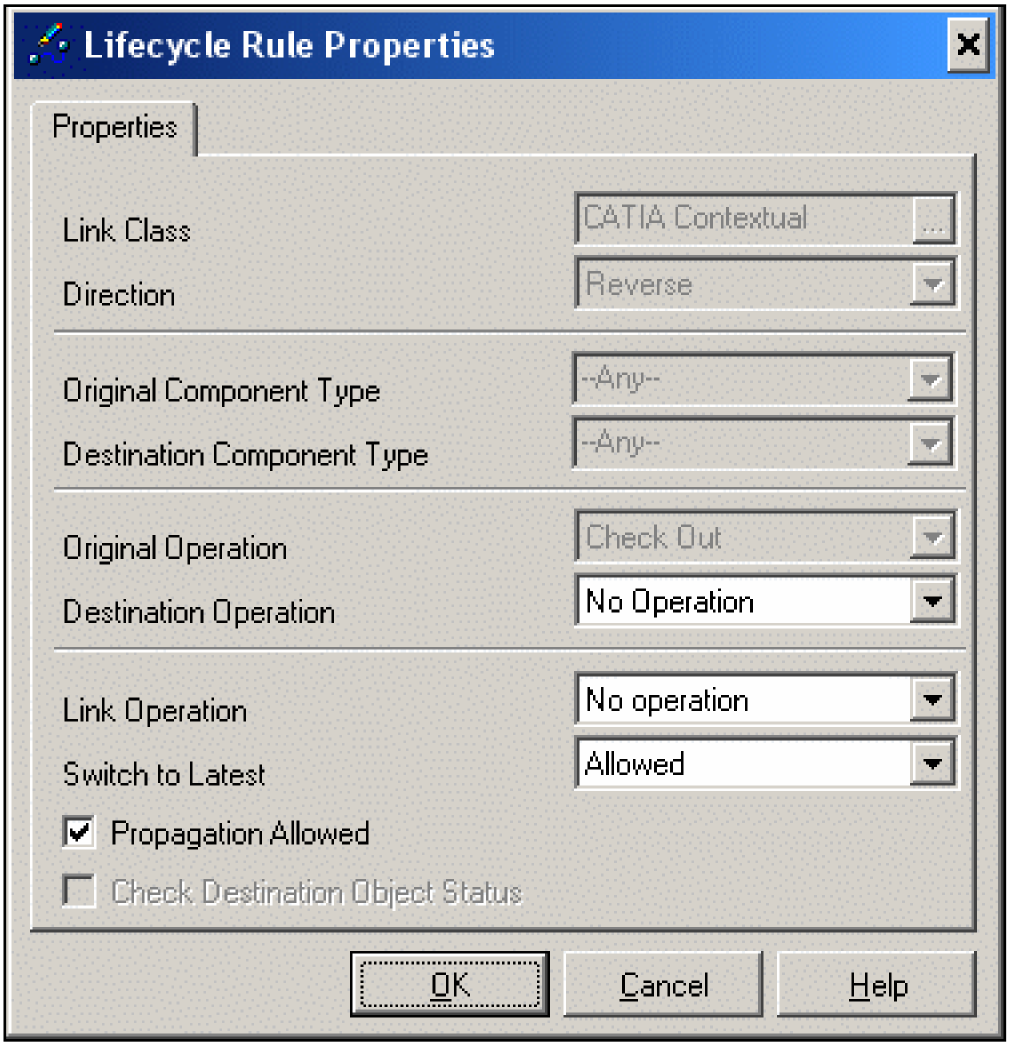


Managing Contextual Links

Here are the settings we recommend for working with reverse contextual links:






Lifecycle Rule Properties
Properties
Link Class
CATIA Contextual
Direction
Reverse
Original Component Type
--Any--
Destination Component Type
--Any--
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

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.**

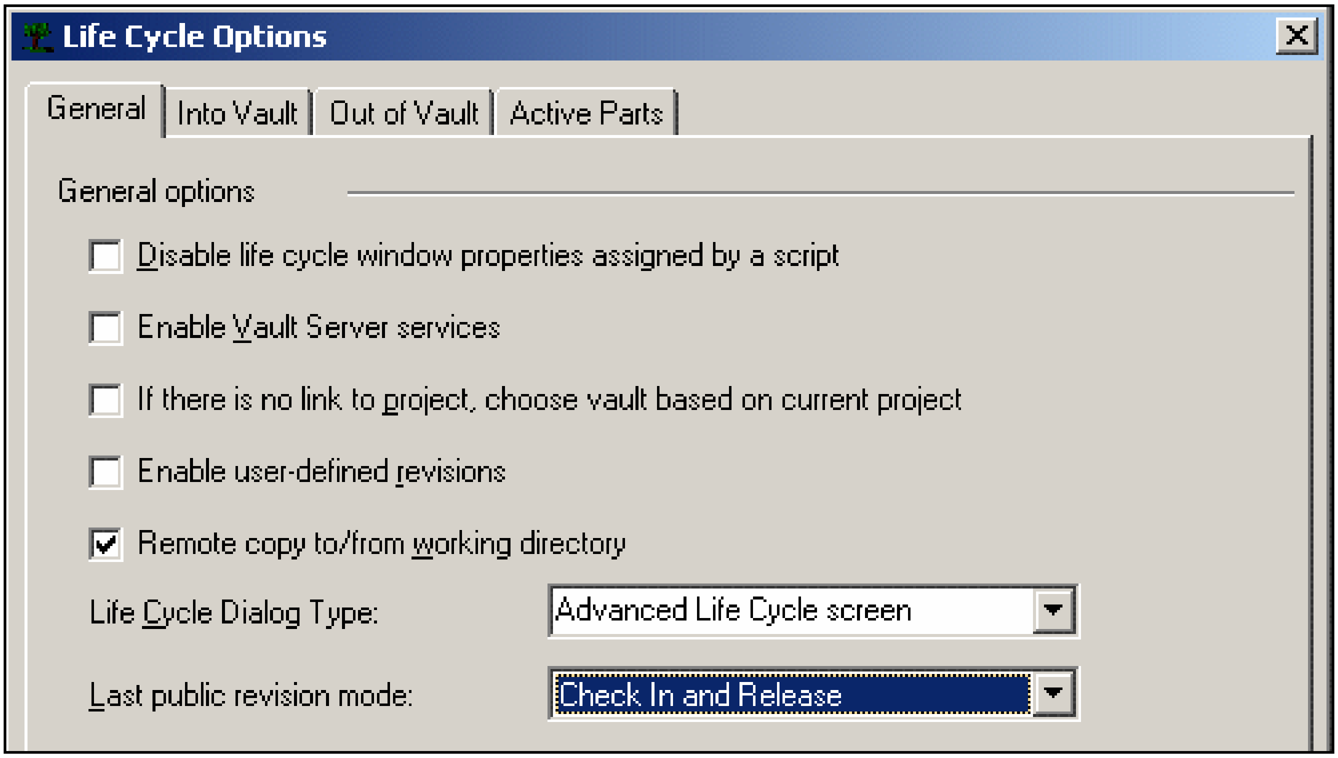
   For **Last public revision mode**, three options are available from the dropdown list in order to determine the last revision status of all objects in SMARTEAM - Editor:

   - 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:

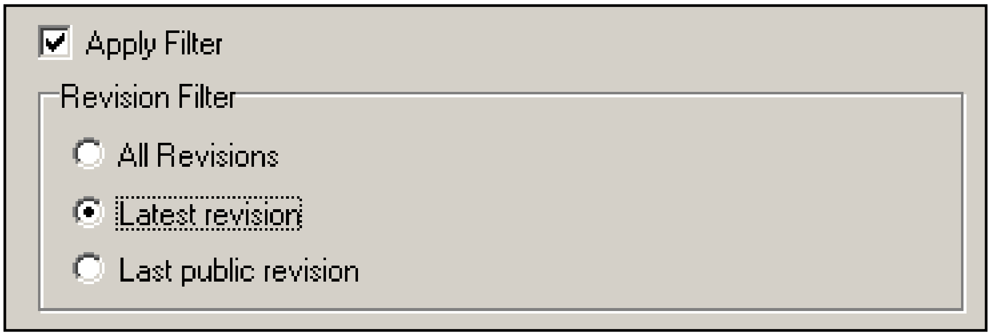
Tree Properties

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

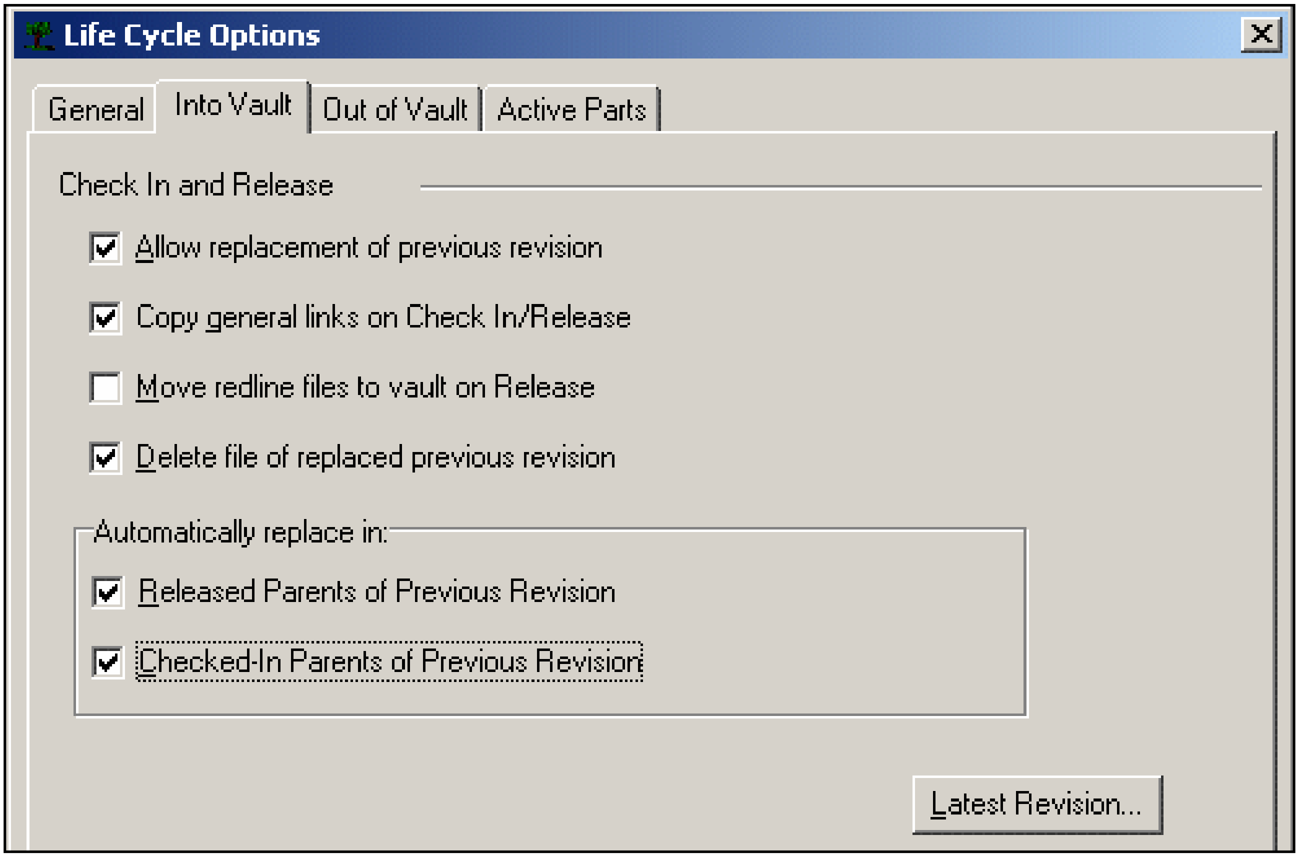
Use Replace in parents of previous revision only in the WIP (Work In Process) stage of the design. This functionality also makes all referencing documents dirty. Therefore, as soon as these are opened in CATIA, a rebuild is needed (check out).

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

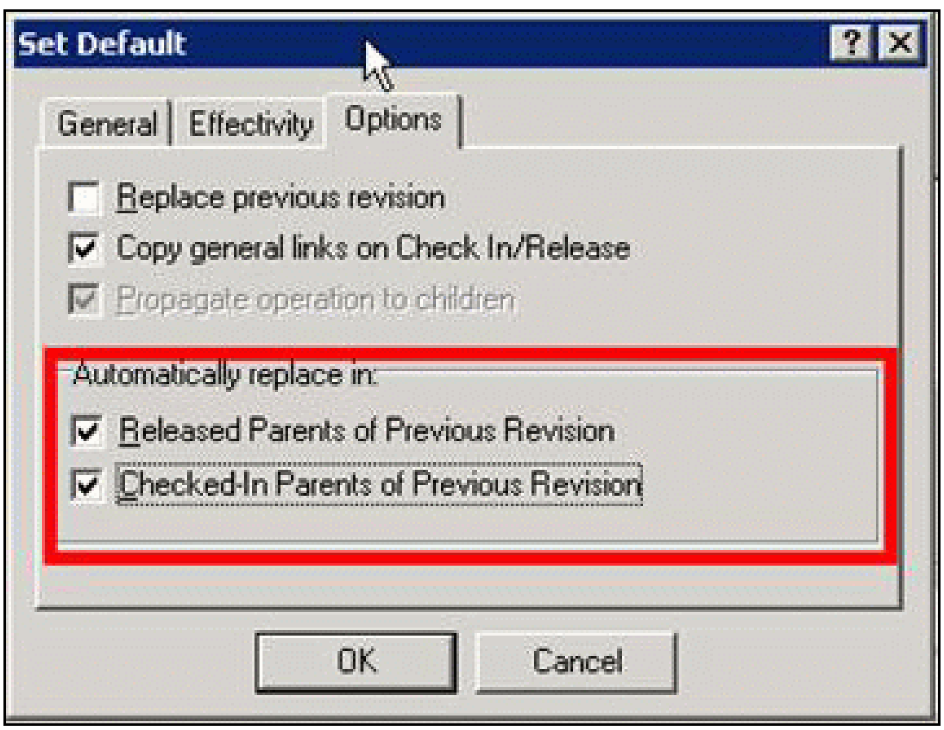
Starting from SMARTEAM V5R13, the administrator can set automatic replace in parents of previous revision as shown below.

1. From Administrator Options, select **Lifecycle Options.**
2. Click the Into Vault tab.
3. Check **Released Parents of Previous Revision** and **Cheked-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.



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.

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, refer to 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

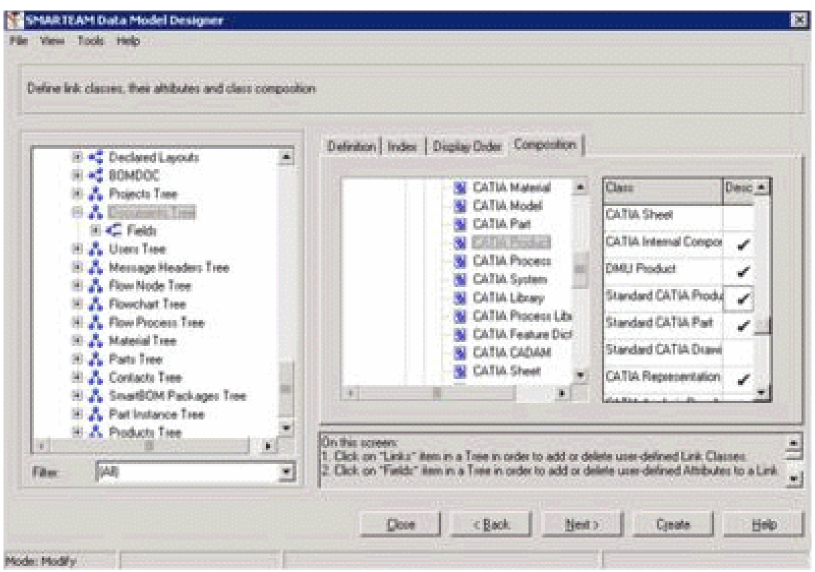
The SMARTEAM Data Model Designer allows you to define composition rules. It is recommended to allow only the relevant compositions as this has direct effect on performance. The following table displays the recommended composition for CATIA classes. For component types that do not appear in the table, 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.

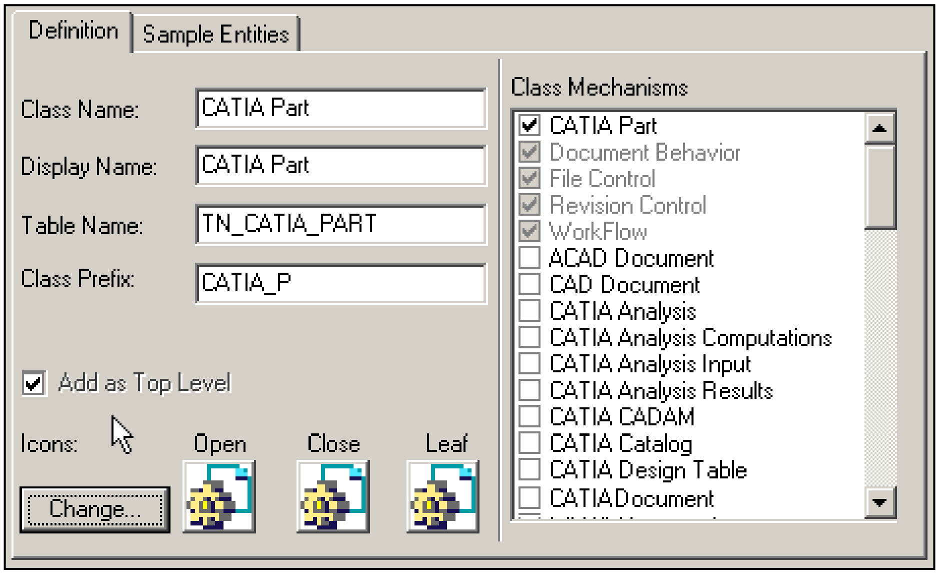
Assigning a Class to be top level has performance implications due to the special actions executed by SMARTEAM during a lifecycle. When Out of Vault operations (Check Out, New Release) are performed on objects of classes that may be desktop objects, the system checks whether the particular object is 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, refer to 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

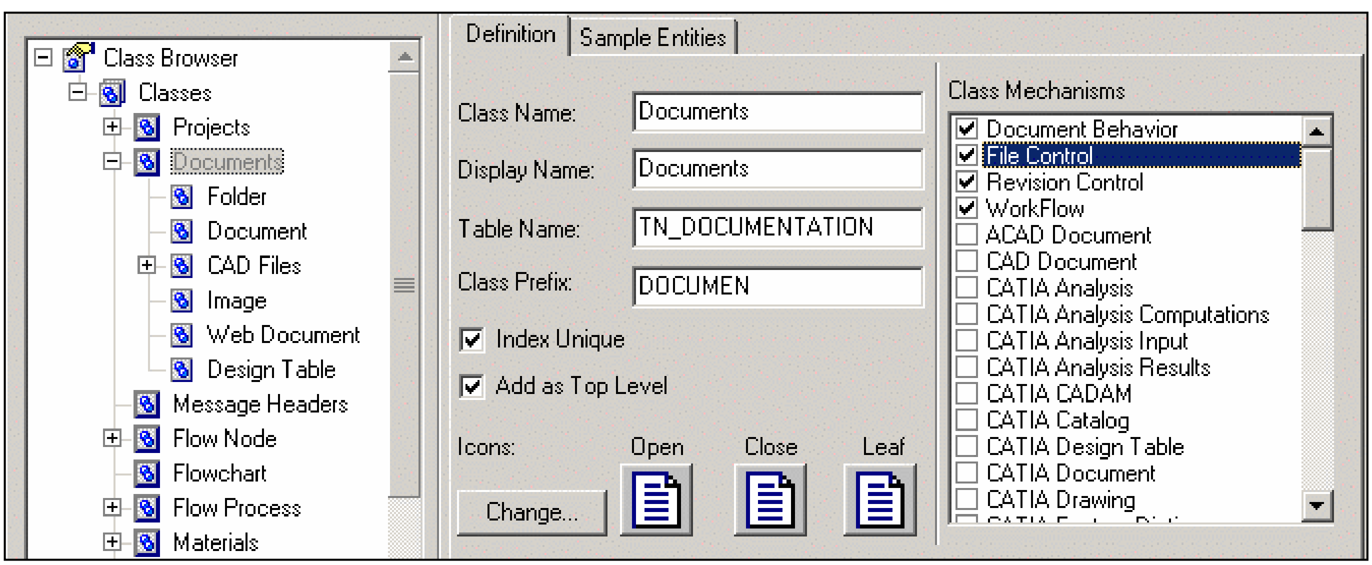
In some cases, objects in certain classes are not file-managed, for example, folders in the documents tree. Removing the file control mechanism from a superclass and assigning file control to relevant classes only may have serious performance implications.

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.

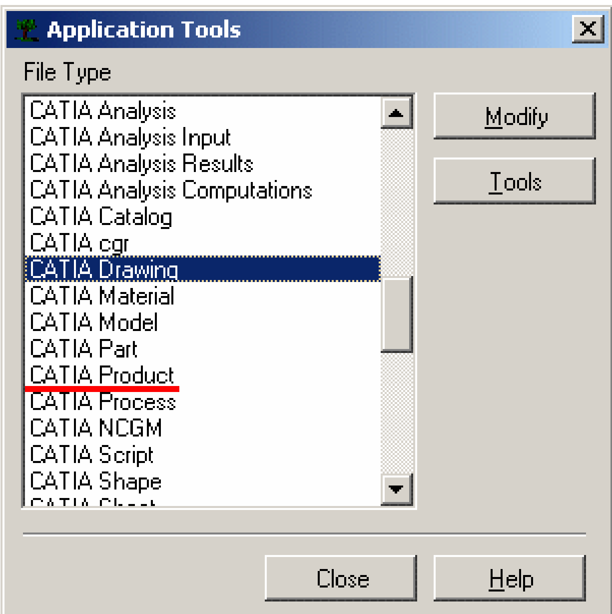
Recommendation

Unless there is a specific need to manage internal components and/or drawing sheets as distinct objects, it is recommended to disable creating objects.

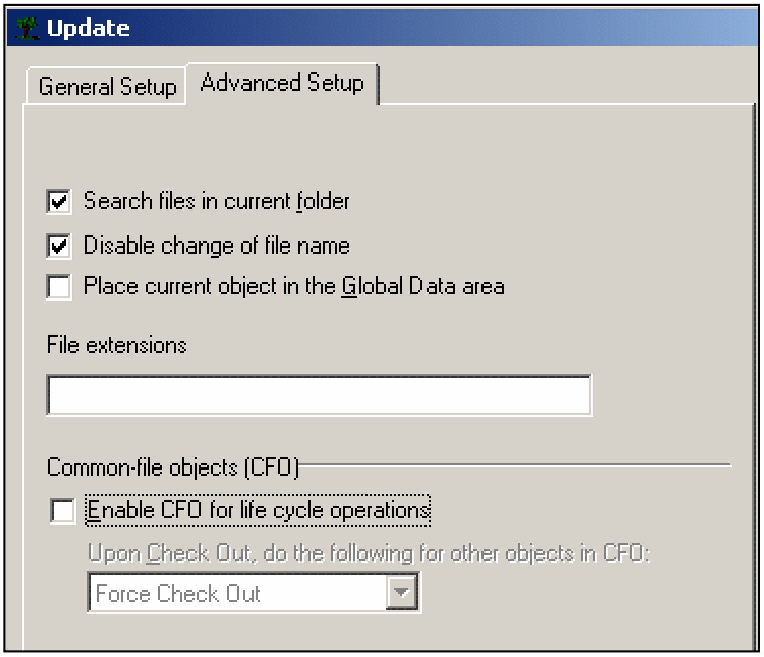
How to disable creating objects for internal components and drawing sheets

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 unselect **Enable CFO for life cycle operations** to disable the CFO mechanism for CATIA Product and CATIA Drawing file types.


Update
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:
Force Check Out

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, refer to Data Model Considerations: Defining Classes.

Attribute Sizes

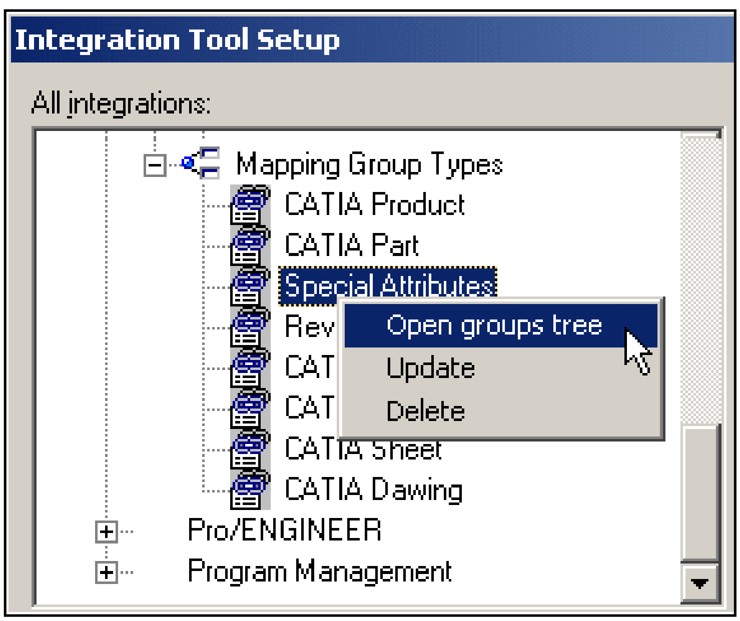
The size of certain attributes in the system is critical. The methods your company uses for file naming may require fine-tuning of these attribute sizes.

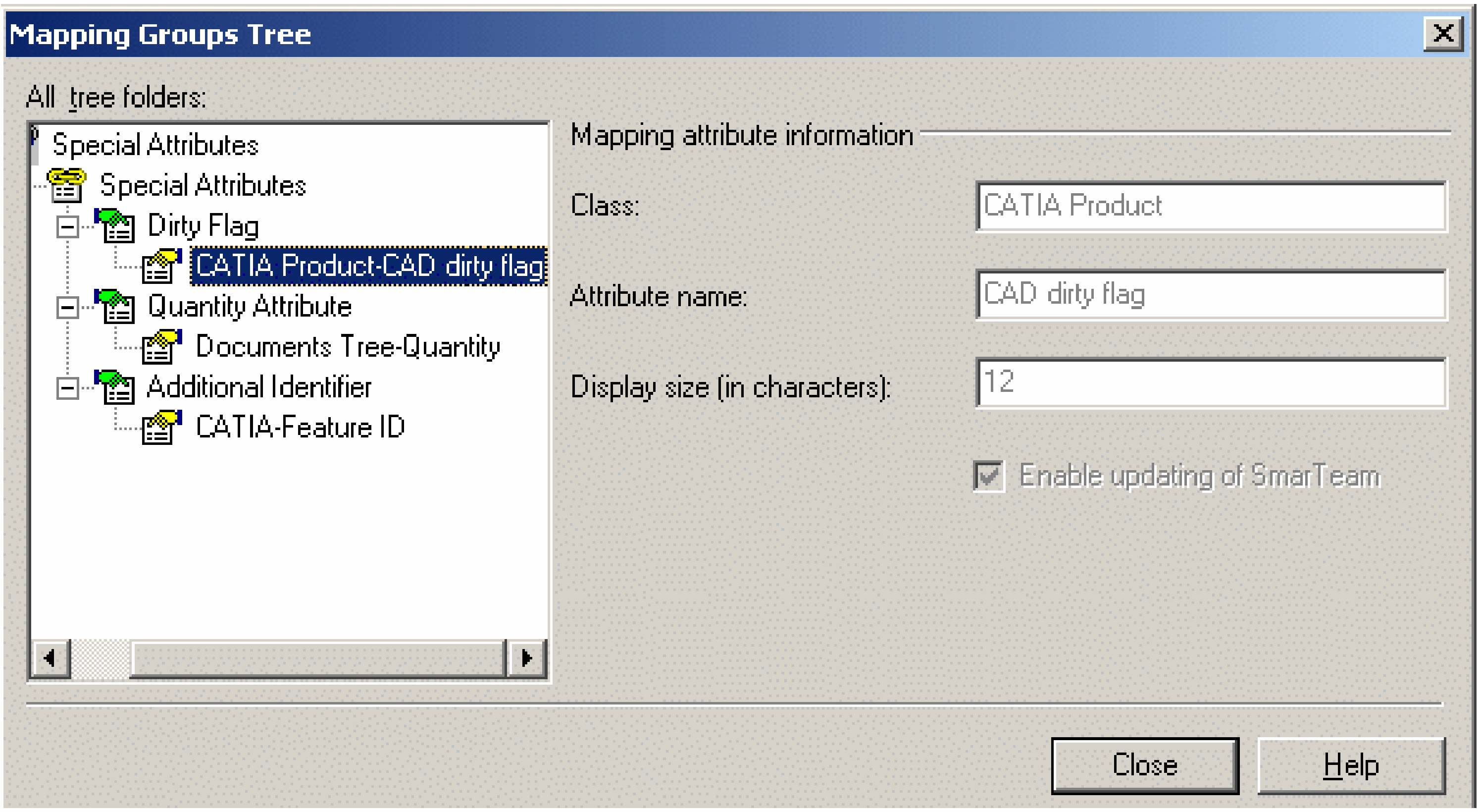
File names and directory path attributes (FILE_NAME, DIRECTORY, CAD_REF_FILE_NAME, CAD_REF_DIRECTORY) must be long enough to store data about your files. If the field size is too small, the system may fail to function properly.

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.

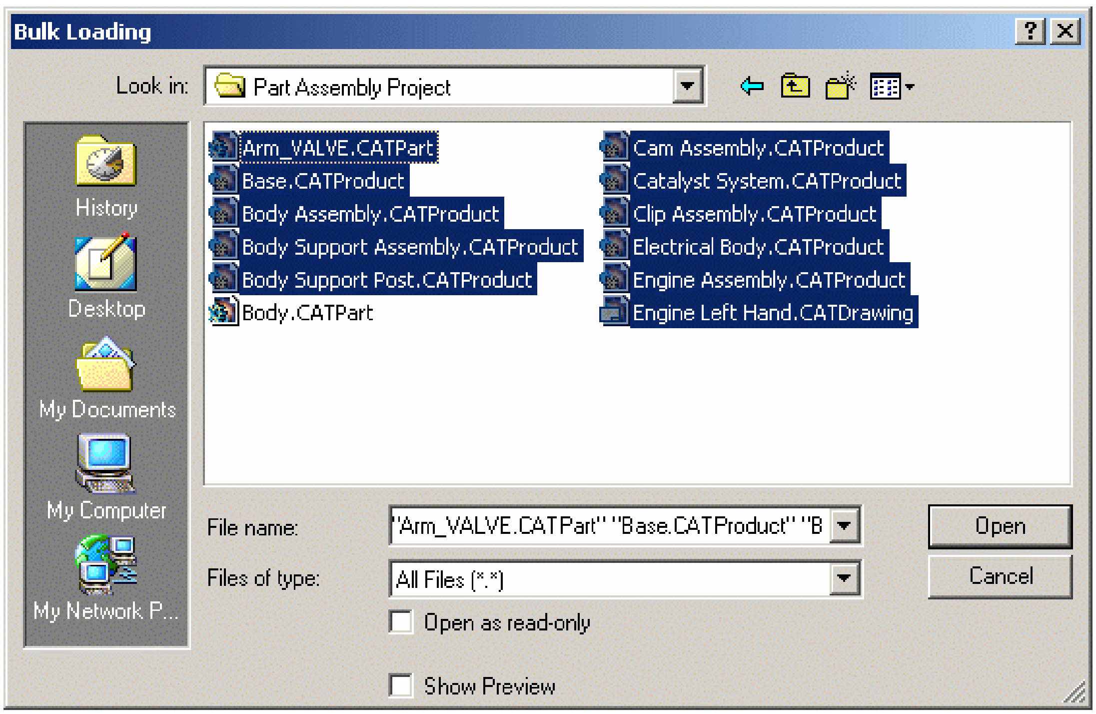
Importing CATIA Data Inside SMARTEAM

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:

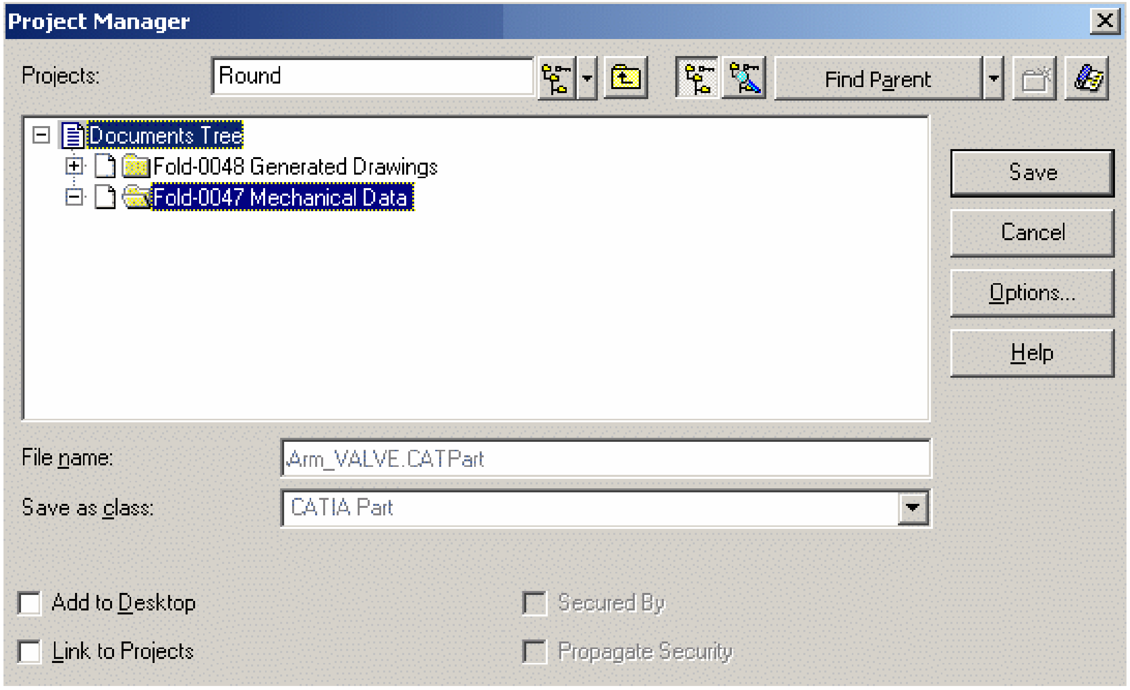
- The save operation is quick.
- Links between documents are kept.
- The attribute mapping mechanism is used during the save operation.
- Only a limited number of user interactions are required regardless of the number of files.

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:



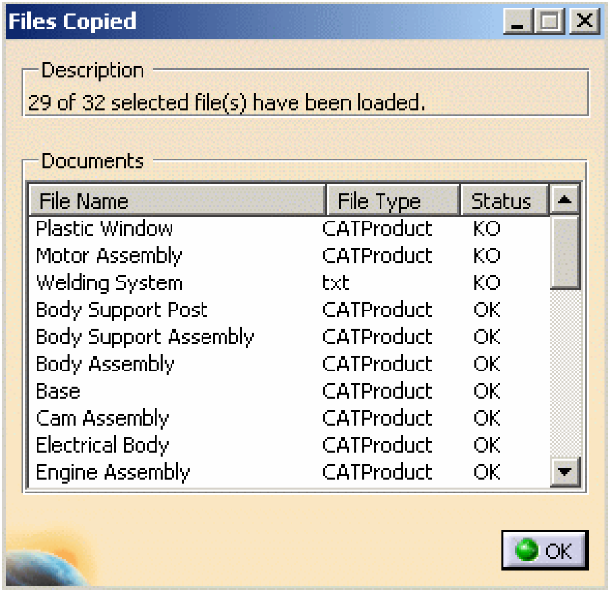
5. Click **Save**.

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:

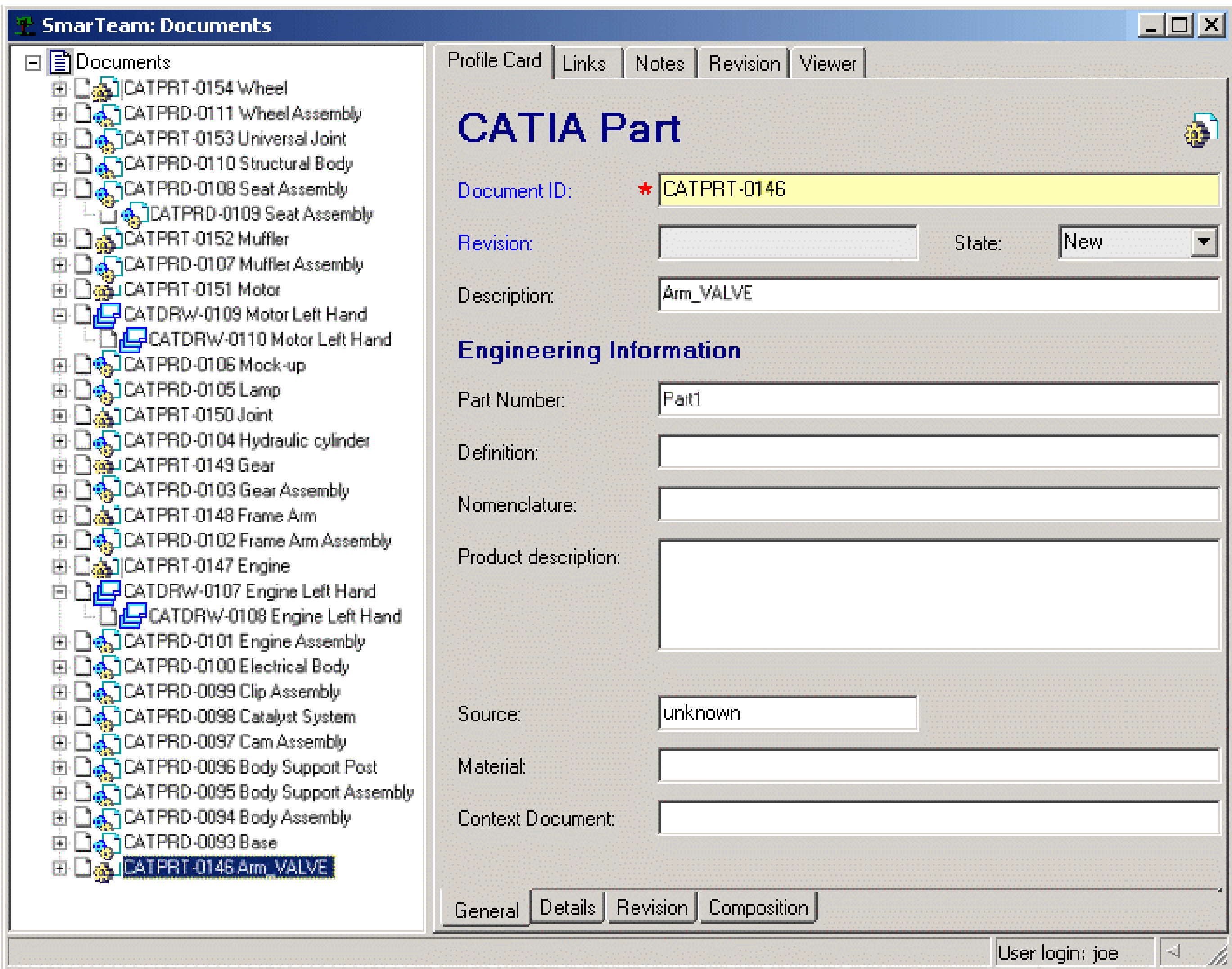
Should any problems arise during the save operation such as:

- a document that cannot be opened in CATIA
- 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.

SMARTEAM Upgrade/Migration

This section provides operational instructions for updating the CATIA-SMARTEAM system when used with Oracle databases on Windows machines. It covers migrations from releases that are still supported by the vendors to the newest ones or intermediate ones, ranging today from SMARTEAM V4.0.5.5/CATIA V5R9 to CATIA/SMARTEAM V5R13 or R14.

Compatibility Matrix

This simplified compatibility matrix corresponds to recommended configurations. It assumes that for each release, the last service pack is used.

	2002	2002	2003	2003
CATIA	V5R9	V5R10	V5R11	V5R12
SMARTEAM	V4.05.5	V5R10	V5R10/R11/R12/R13	V5R10/R11/R12/R13/R14
ORACLE	8.1.6+ (for ST V4.05.5)	8.1.6+ (for ST R10)	8.1.7+ (for ST R11)	8.1.7+ (for ST R12)
OS	WIN 2000 (for ST V4.05.5)	WIN 2000 (for ST R10)	WIN 2000 (for ST R11)	WIN 2000/3 (for ST R12)

	2004	2004	2005
CATIA	V5R13	V5R14	V5R15
SMARTEAM	V5R11/R12/R13/R14	V5R12/R13/R14	V5R13/R14
ORACLE	9.2 (for ST R13)	9.2 (for ST R14)	9.2 (for ST R14)
OS	WIN 2000/3 (for ST R13)	WIN 2000/3 (for ST R14)	WIN 2000/ (for ST R14)

Reference documents

The following reference documents provide more information about the different topics of the procedure described here. You can access them on the SMARTEAM documentation which is available on GA CDs or in SMARTEAM.

- *Oracle Installation*
- *Oracle Fine Tuning for SMARTEAM Implementers*
- *SMARTEAM Hardware and Software requirements*
- *SMARTEAM Releases upgrade & What's New documents*

Steps to follow

Depending on your present configuration and the targeted one for the update you will need to apply the migration tasks related to CATIA, SMARTEAM and possibly Oracle. If so, the Oracle update is the first step to perform (See Oracle Migration) . CATIA software update is not so critical, we recommend to install the targeted release just after Oracle Migration, and then to continue with SMARTEAM update (See SMARTEAM Migration) . If you install CATIA R12 you will have advantage to proceed to the files update to take full benefits of this release. This is explained on the last paragraph, as the last step to perform.
The tasks described are as the following ones:

- Oracle Migration
  - Export the SMARTEAM database to a dump file
  - Create the new database
  - Import the SMARTEAM dump file to the new database
- SMARTEAM Migration
  - V5R10+ specific upgrade
  - V5R11+ specific upgrade
  - V5R12+ specific upgrade

- V5R13+ specific upgrade
- Post-activities

Time critical points

These tasks are time-consuming activities and are given with typical delays: SMARTEAM automatic database upgrade: about 2 hours for 10000 Db objects. Test the expected delay with your data, because this depends of the files and the database complexity. Then schedule your production migration taking these constraints into account.

Prerequisites

1. SMARTEAM scripts are written using COM SMARTEAM API, API functions and low-level database objects used are still supported (see appendix of upgrade documents).
2. LUM licenses for SMARTEAM are available through server or node-locked.
3. Check-in all in-work files in the vault.
4. Disconnect all clients from SMARTEAM Servers. CATIA should be used in local during this migration period
5. Up to date Production database, SMARTEAM directory, and vaults are backup.
6. Apply first the following procedure to a test environment (including database and vaults) that is a copy of the production one.
7. Modify the test database to point to the test vaults (use Vault Server Setup utility).

Oracle Migration

The chosen update procedure consists of installing the totally new database on another directory on the server, and then migrating the data from the former database to the new one. The old database should be deleted only at the end of the migration process. This method allows us to go directly to V9.2 when coming from v8.1.6.

To migrate the data, we choose to use the standard Export and Import mechanisms of Oracle.

Export the SMARTEAM database to a dump file

Oracle Setting for data management

1. First verify that Oracle Enterprise Management Tools are installed.
2. Verify the Oracle Management Server service is running under a user who is granted the Log On as a service right.
3. Run Oracle Enterprise Manager Console.
4. Set System/Preferences/Preferred Credentials user for your database or default value for Node Service Type.
   The user should be the same as the one declared to run the Oracle Management Server service. It must have the right to the directory where the file you want to import resides. User information should be entered without domain information.

Export Operation

1. Right-click Database / Data Management/ Export.
2. In the Export File window, set the dump filename you want to export to.
3. In the Export Type window, select Table.
4. In the Table selection Window, click on the user that host the current SMARTEAM schema, and select all tables under him.

5. In the Associated Objects window, disable Indexes on Tables.

6. In the schedule window, run the job immediately then finish.

7. Verify on the job window the status of the job. Wait for it to be completed.

For further information on exports, refer to *SMARTEAM - Editor Administration Guide*.

Create the new database

1. Install the software components of the new Oracle database engine. Refer to the instructions of the Oracle Installation Guide provided in the SMARTEAM documentation. If you change the block size as recommended from 8 to 4096, do not forget to change the units too: Kbytes to bytes.

2. Create a new database following the instruction of the same guide.

3. Create a user to host the schema of the SMARTEAM data (Go to the Security tab). Grant him the Resource role. Assign it the same table space as that used in the previous database.

4. Verify that you can access this newly created database by using the SmartDBExplorer utility. This database is empty at this point.

Import the SMARTEAM dump file to the new database

Oracle Setting for data management

Apply the same setting that the ones explained in the export paragraph (see Oracle Setting for data management)

Import Operation

1. Right-click database->data Management->Import.

2. In the Import File window, Set Dump directory and filename.

3. Read Import File and select what objects you want to import.

4. In the Import Type window, select Table.

5. In the Tables selection window, select all tables from the shown user.

6. In the User mapping window, map old user with the new one.

7. On the Associated objects, disable indexes import.

8. On the schedule window, run job immediately, then finish.

9. Verify on the job window the status of the job. Wait it to be completed.

10. Create New Database registration for the new database using DBRegistration.exe in the bin folder of your SMARTEAM directory.

11. Repair DB indexes using SmartDBRepairing. Apply Indices Repairing.

12. Open the new database with SMARTEAM and verify that all is OK.

SMARTEAM Migration

Each specific release upgrade procedure is not cumulative. It means that if you are running SMARTEAM V4 and plan to migrate to V5R12, you have to apply R10 specific task, R11 specific tasks then R12 specific tasks. One chosen client machine will be dedicated to these migration tasks.

V5R10+ specific upgrade

We assume that your current version of the SMARTEAM database is at least 4.0.3, look at the TDM_DB_UPGRADES and TDM_DB_VERSION tables to check this point with the SmartDBExplorer utility.

Product Installation

1. Install R10 GA on the identified upgrade machine (Select Editor Installation).

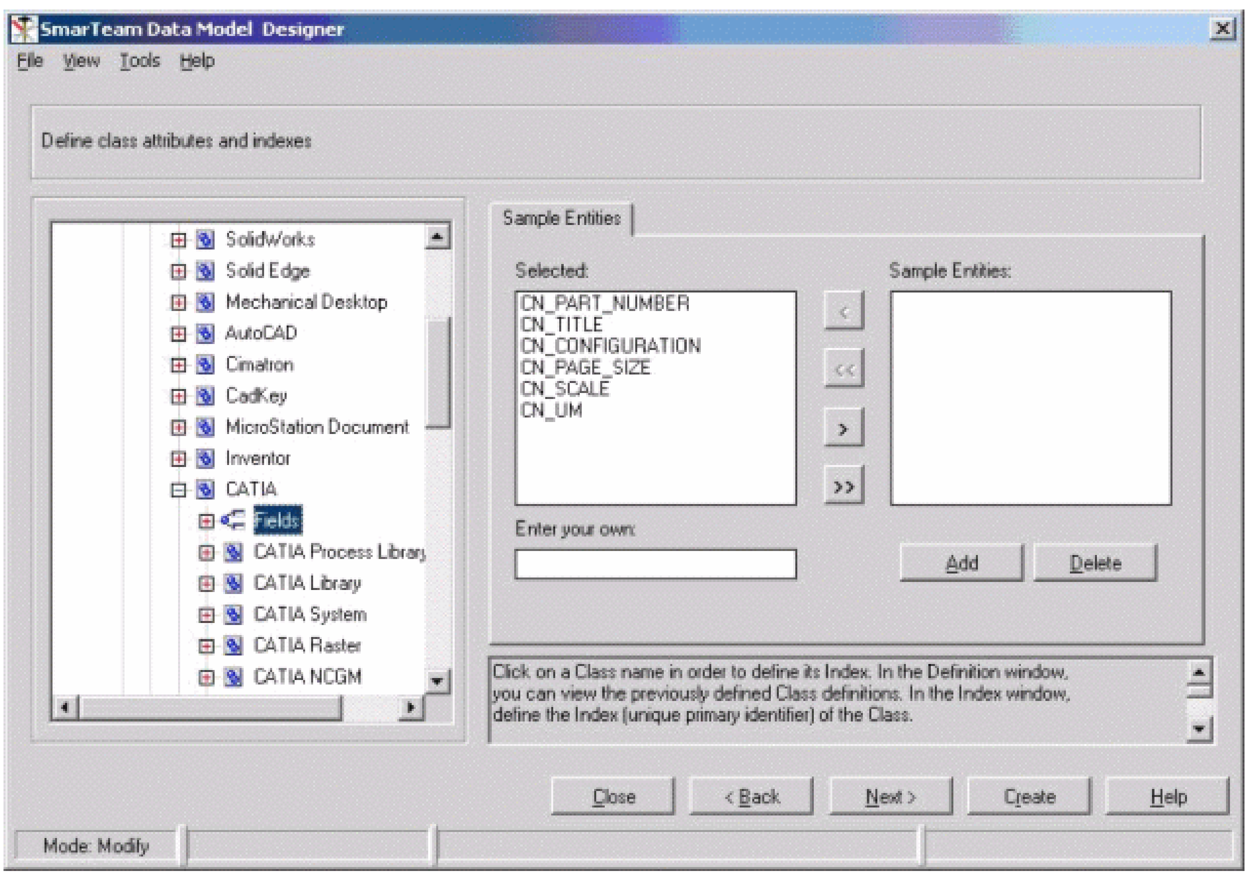
>What you' need to check

Case 0, you're using PartNumbers to localize documents (CATIA_TEAMPDM_UUID=NO in environment variables)

If CATIA_TEAMPDM_UUID=YES in your environment variables or does not exist at all, launch SMARTEAM data model designer

1. Do File / Modify database structure and choose your database.
2. Go to Define class attributes and indexes.
3. Check the fields below CATIA class.

Case 1, Feature ID does not exist below this class.



Case 2, the Feature ID has existed below CATIA class

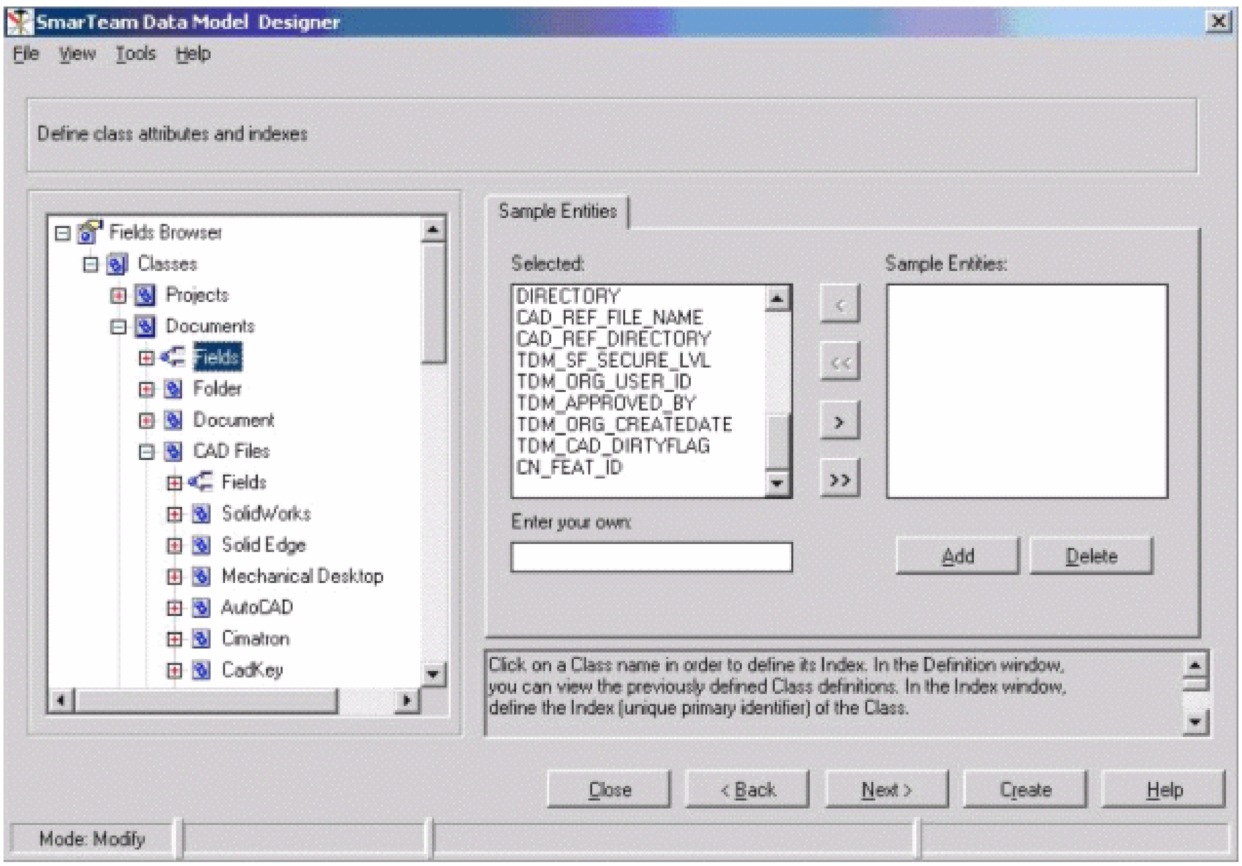
or even below since you began to use the CATIA Integration (ie your database has always been designed like that):

What you' need to do

Case 0

In this part, we will delete the CATIA component unique identifier (UUID or Feature ID, named CN_FEAT_ID in the database) from all CATIA classes. And map this again to CATIA class to delete all the records to PartNumbers.

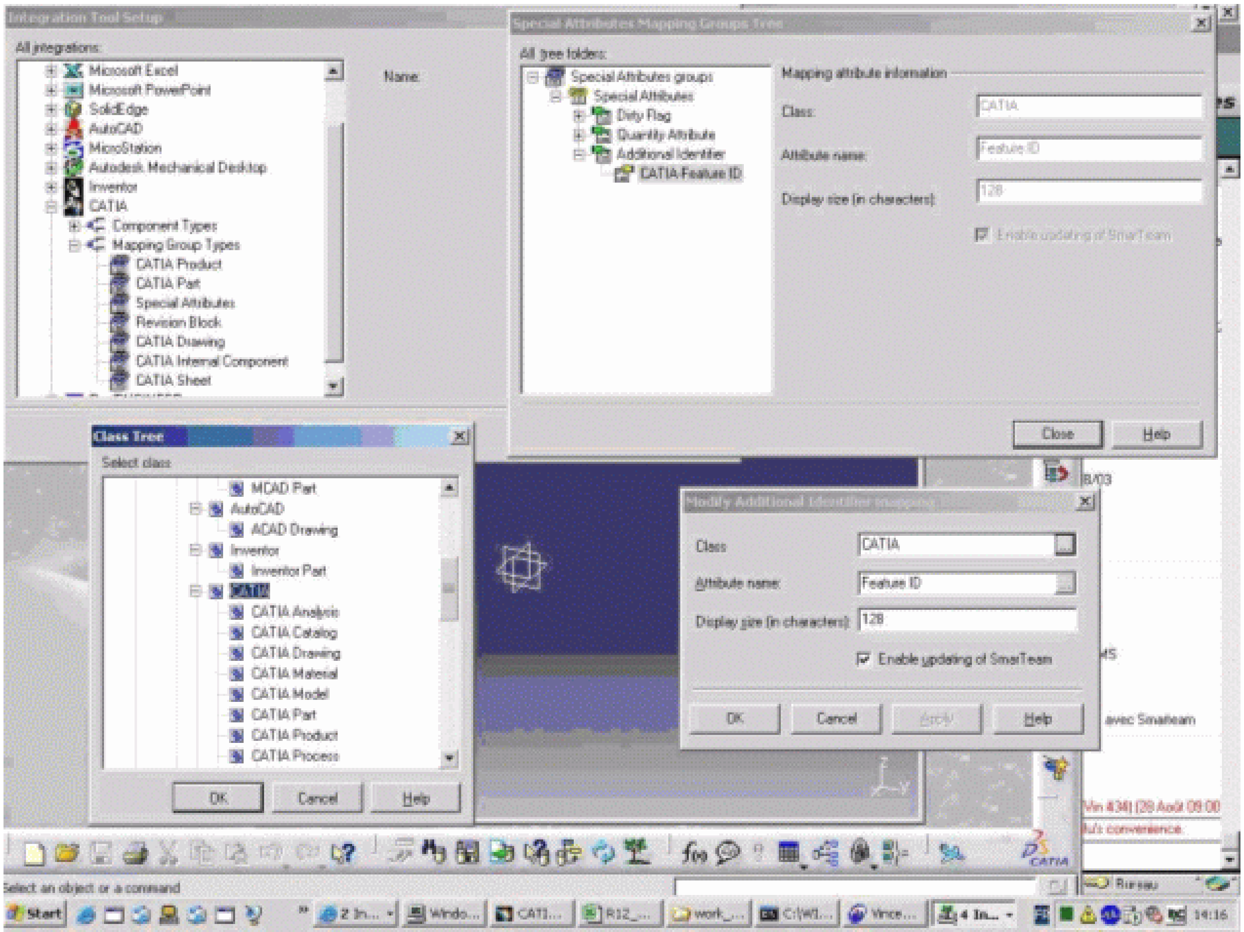
1. Launch the SMARTEAM Data Model Designer.
2. From the File menu, select the Modify Database Structure option and choose your database. Then Go to the Define Class Attributes and Indexes. Identify all CATIA Classes where the Feature ID attribute appears (e.g., CATIA Product, CATIA Internal Component, CATIA Drawing, CATIA Sheet).
3. Remove now the *CN_FEAT_ID* attribute from any of the previously selected classes.
4. Recreate this attribute directly on the *Documents* super class (Name = CN_FEAT_ID, type = char, length = 127, Display Name and Description = Feature ID)



5. Click on Create to save the changes to the database.
6. Concerning the drawings, check if *CATIA Sheet* class exists. If not ensure that CATIA Drawing class supports *CATIA Sheet* class behavior and that you have the EXPOSEMODE=NO in CATIA section of SmTeam32.ini.
7. Concerning the drawings, verify the existence of the two attributes:
   CN_SHEET_NAME (type = char, length = 127, Display Name and Description = Sheet Name) and *CN_SHEET_FORMAT* (type = char, length = 20, Display Name and Description = Sheet Format) under the *CATIA Sheet* class or under the *CATIA Drawing* class if the CATIA Sheet class doesn't exist.

8. In the Integration Tools Setup, select CATIA / Mapping group types / Special Attributes.

9. Create or modify the following mapping: *Additional Identifier* with *CATIA-Feature ID* (Class = Catia, Attribute = Feature ID). Enable update of SMARTEAM.



10. Modify or create now the mappings relative to the new drawing attributes:

Catia *CN_SHEET_NAME* with SMARTEAM *CN_SHEET_NAME* and Catia *CN_SHEET_FORMAT* with SMARTEAM *CN_SHEET_FORMAT* in CATIA / Mapping Group Types / CATIA Sheet (or CATIA Drawing if CATIA Sheet class doesn't exist). For these mappings, enable update of SMARTEAM.

Case 1

In this part, we will transfer the CATIA component unique identifier (UUID or Feature ID, named CN_FEAT_ID in the database) from the CATIA leaf classes to the Documents superclass. And map this to CATIA class.

1. Launch the SMARTEAM Data Model Designer.

2. From the **File** menu, select the **Modify Database Structure** option and choose your database.

3. Then go to the Define Class Attributes and Indexes.

4. Identify all CATIA Leaf Classes where the Feature ID attribute appears (e.g., CATIA Product, CATIA Internal Component, CATIA Drawing, CATIA Sheet).

5. For each of these classes note the table name (it is indicated on the right dialog box, typically, TN_CATIA_PRODUCT, TN_CATIA_DRAWING, TN_CATIA_SHEET).

6. Apply the following SQL statements for each of the selected tables (Launch your SQL command Interpreter (e.g. SQL +), and type

or copy and paste the following text , replacing *table_name* by the actual table name, if necessary replace also the table name of the Documents super-class here TN_DOCUMENTATION by the actual one):

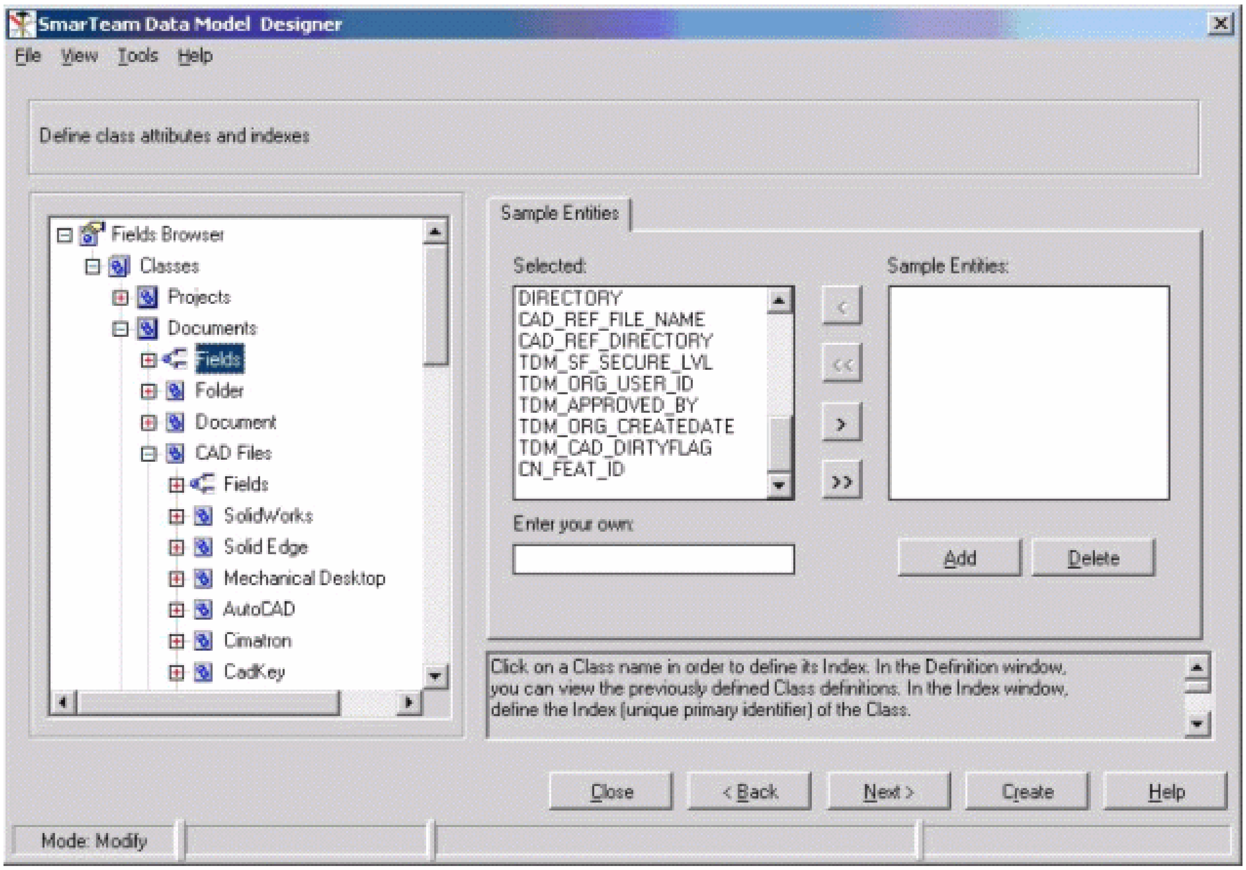
```
ALTER TABLE TN_DOCUMENTATION ADD CN_FEAT_ID_TEMP VARCHAR(127);

CREATE VIEW MY_VIEW AS SELECT TN_DOCUMENTATION.OBJECT_ID, CN_FEAT_ID_TEMP, CN_FEAT_ID FROM
TN_DOCUMENTATION LEFT OUTER JOIN table_name ON TN_DOCUMENTATION.OBJECT_ID = table_name.OBJECT_ID;

UPDATE MY_VIEW SET CN_FEAT_ID_TEMP = CN_FEAT_ID WHERE CN_FEAT_ID IS NOT NULL;

DROP VIEW  MY_VIEW;
```

7. Coming back to the Data Model Designer, remove now the *CN_FEAT_ID* attribute from any of the previously selected classes.
8. Recreate this attribute directly on the *Documents* super class (Name = CN_FEAT_ID, type = char, length = 127, Display Name and Description = Feature ID)



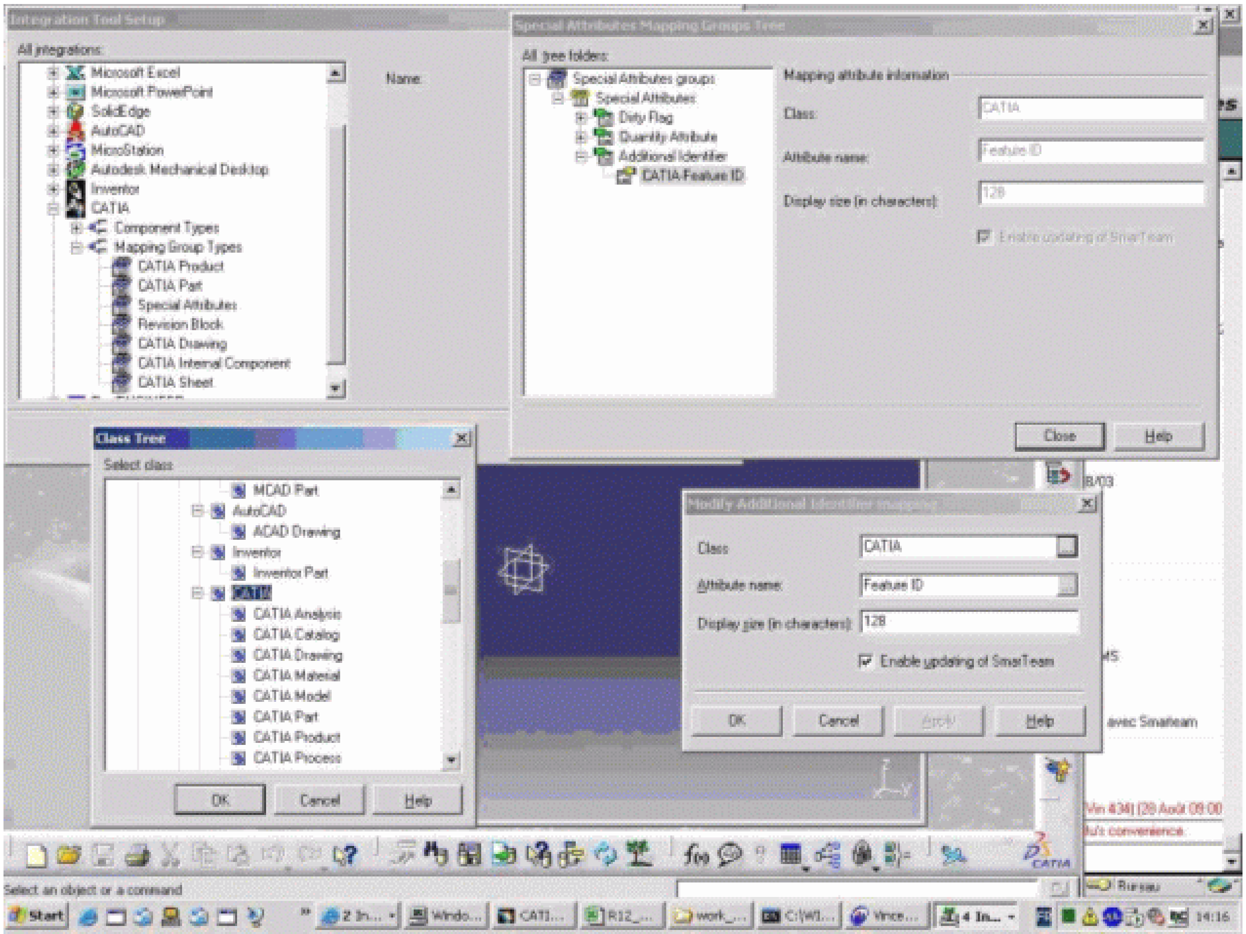
9. Click on **Create** to save the changes to the database.
10. To finish this UUID migration, type the following statements in SQL+

```
UPDATE TN_DOCUMENTATION SET CN_FEAT_ID = CN_FEAT_ID_TEMP;

ALTER TABLE TN_DOCUMENTATION DROP COLUMN CN_FEAT_ID_TEMP ;
```

11. Concerning the drawings, check if *CATIA Sheet* class exists. If not ensure that CATIA Drawing class supports *CATIA Sheet* class behavior and that you have the EXPOSEMODE=NO in CATIA section of SmTeam32.ini.
12. Concerning the drawings, verify the existence of the two attributes *CN_SHEET_NAME* (type = char, length = 127, Display Name and Description = Sheet Name) and *CN_SHEET_FORMAT* (type = char, length = 20, Display Name and Description = Sheet Format) under the *CATIA Sheet* class or under the *CATIA Drawing* class if the CATIA Sheet class doesn't exist.
13. In the Integration Tools Setup, select CATIA / Mapping group types / Special Attributes.

14. Create or modify the following mapping: *Additional Identifier* with *CATIA-Feature ID* (Class = Catia, Attribute = Feature ID). Enable update of SMARTEAM.



15. Modify or create now the mappings relative to the new drawing attributes:

Catia *CN_SHEET_NAME* with SMARTEAM *CN_SHEET_NAME* and Catia *CN_SHEET_FORMAT* with SMARTEAM *CN_SHEET_FORMAT* in CATIA / Mapping Group Types / CATIA Sheet (or CATIA Drawing if CATIA Sheet class doesn't exist).

16. For these mappings, enable update of SMARTEAM.

Case 2.

1. Launch the SMARTEAM Data Model Designer.
2. From the File menu, select the **Modify Database Structure** option and choose your database.
3. Go to the Define Class Attributes and Indexes window and from the CATIA class, check if *CATIA Sheet* class exists. If not ensure that CATIA Drawing class supports *CATIA Sheet* class behavior and that you have the EXPOSEMODE=NO in CATIA section of SmTeam32.ini.
4. Concerning the drawings, verify the existence of the two attributes:

   CN_SHEET_NAME (type = char, length = 127, Display Name and Description = Sheet Name) and *CN_SHEET_FORMAT* (type = char, length = 20, Display Name and Description = Sheet Format) under the *CATIA Sheet* class or under the *CATIA Drawing* class if the CATIA Sheet class does not exist.
5. In the Integration Tools Setup, select **CATIA->Mapping group types->Special Attributes**.
6. Modify or create now the mappings relative to the new drawing attributes: Catia *CN_SHEET_NAME* with SMARTEAM *CN_SHEET_NAME* and Catia *CN_SHEET_FORMAT* with SMARTEAM *CN_SHEET_FORMAT* in CATIA / Mapping Group Types / CATIA

Sheet (or CATIA Drawing if CATIA Sheet class doesn't exist). For these mappings, enable update of SMARTEAM.

V5R11+ Specific upgrade

If you already are running V5R11 or a more recent version, skip this paragraph and go directly to the V5R12 specific upgrade.

Product installation on the upgrade machine

1. Install R11 standalone and administrative tools (select Editor Installation)
2. Install last available service pack (at least SP8)
3. Install SmartDBUpgradeV5R11SPx from your Kits\SmartDBUpgrade folder of your service pack disk.
4. Install SmartDBRepairingV5R11SPx from your Kits\SmartDBRepairing folder of your service pack disk (at least SP8, oracle issue with previous releases)

Automatic Database upgrade

Once R11 upgraded the database schema will be compatible with R12 as well as with R13.

Prepare ini files

1. Copy the reference SMASSCLS.INI to the SMARTEAM directory of the upgrade machine.
2. Copy the reference SmTeam32.ini file to the LocalConfig folder of the SMARTEAM directory of your upgrade machine. This file is the relevant merge of admin and client files. Include also the [Tree-VisualSetting] section of the reference user ini file (located in the LocalConfig\[db]\[username] subfolder)
3. If you use customized integration for specific files, register these files in the [AdditionalFileTypes] section of the UpgradeSmartDatabase.ini file located in the SMARTEAM *LocalConfig* directory by specifying corresponding CAD Component Type. Add a line under the section using this convention:
   Customized File Type = CAD Component Type
   NB: the list of all available component types may be found in the Integration Tool Setup, and correspond to the class behaviors of the Data Model designer, while Customized File Type is a custom entry of the SMARTEAM *File Type* lookup table.

Oracle server temporary Tuning for upgrade

1. Check the database for compliance with the memory specifications in the Oracle Fine Tuning for SMARTEAM Implementers Guide.
2. Set SORT_AREA_SIZE = 524288 (for 8.1.7) (Instance / Edit Database / General / Initialization Parameters
3. Turn ARCHIVE mode OFF.
4. Check that your SMARTEAM related tablespace and SYSTEM tablespace have at least 40% free space. The TEMP tablespace must have at least 100 MB. Check you have sufficient disk space on the database server.
5. Type In SQL+ as SYSTEM (*TABLESPACE_NAME* is the name of your SMARTEAM related tablespace):

   ALTER TABLESPACE *TABLESPACE_NAME* NOLOGGING

6. In SQL+ as a SMARTEAM user, set the following statement:

   spool c:\nologging.sql

```
select 'ALTER TABLE '||TABLE_NAME||' NOLOGGING;' from USER_TABLES;

spool off

@c:\nologging;

/
```

Automatic upgrade of the database

1. Launch UpgradeSmartDatabase.exe from the Upgrade folder of your SMARTEAM directory. Warning: this step may be more than one hour long.

Oracle server tuning after upgrade

1. Type In SQL+ as SYSTEM (Where *TABLESPACE_* NAME is your SMARTEAM related database tablespace)

   ALTER TABLESPACE *TABLESPACE_NAME* LOGGING;

2. In SQL+ as a SMARTEAM user, set the following:

```
spool c:\nologging.sql

select 'ALTER TABLE '||TABLE_NAME||' LOGGING;' from USER_TABLES;

spool off

/
```

3. Turn ARCHIVE mode ON if necessary.

Indices repairing

1. Perform now SmartDBRepairing.exe from your bin folder of your SMARTEAM directory, choose indices Repairing.

WizSrc Upgrade

1. Run *SMARTEAMWizSrcUpgradeWizard.exe* from the upgrade folder of your SMARTEAM directory.

Registering Files out of the vault

This step is only needed if you use File control-classes that are not revision managed or if you performed the migration with remaining new or checked out files (which was not recommended)

1. In the SmTeam32.ini file located in *<SMARTEAM>\LocalConfig\<database>\SmTeam32.ini*, add the following entry:

   [Migration]

   ImportStandardParts=TRUE

2. Ensure that the Administrator has read-write permissions in the area where the files of non-revision managed object reside.
3. Administrator should log into SMARTEAM, the file catalog is created then once. This may take some time, depending on the number of files.

Viewer Setting and CGR generation

Change the file type to be viewed for CATIA Part and CATIA Product classes.

1. Select **SMARTEAM->Tools->Applications** Setup.
2. From the Applications Setup window that is displayed, select **CATIA class**.
3. Click on **Tools**.
4. Click the **Embedded Viewer** tab.
5. Select **Modify**.
6. Replace .CATProduct.hcg with .CATProduct.cgr.

Setting Update Rights to SMARTEAM users

Users need update rights to ensure that CATIA can upgrade TDM_COMPONENT_NAME on the fly when opening any CATIA document.

1. Launch **Users Maintenance**.
2. Click the **Authorization** icon.
3. For users likely to use CATIA documents, set update rights for all states on CATIA documents.

V5R12+ Specific Upgrade

If you are already running V5R12 skip this paragraph and go to V5R13 specific upgrade, otherwise:

Important note: if R12 is not your target release, but R13 is, the best is to apply the following with SMARTEAM R13 installed rather than R12.

Product Installation

1. Install R12 core services on a server (dedicated or not) (select foundations).
2. Install last available SP (currently SP6) on this server.
3. Install R12 GA on the upgrade machine.
4. Install last available SP (currently SP6) on this machine.
5. Install SmartDBUpgradeV5R12SPx.exe from the kits\SmartDBUpgrade directory of the service pack CD (to run later Wizsrc update only)

Session Management Configuration

1. On the server Run the Authentification Manager Utility (SMARTEAM \ bin \ SMARTEAM.Std.AuthenticationManager.exe), choose SMARTEAM Authentification protocol, check "Authentificate using the following database" and select your database in the list.

NLS Configuration

1. Verify that the reference tdmerror.err, tdmwizard.err, tdmvlsrv.msg are on the SMARTEAM root directory of the upgrade machine.
2. Run the NLS Extractor Utility smartNLSextractor.exe from the upgrade machine (Note: Data will be extracted to XML files in the ...\SMARTEAM\NLS\<language RFC id>\CUSTOM folder. For example <language RFC id> = fr for the French language. For further information on RFC codes, and the official list of language codes, refer to: http://www.oasis-open.org.

3. Copy the NLS folder created by the NLS Extractor Utility (located under SMARTEAM) to the Core Services Server.

4. Define the copied folder as Shared for Read/Write access for all users of SMARTEAM products.

5. In the System Configuration Utility, update the location of the <rootPath> key for the copied NLS folder, located in: \SMARTEAM\ConfigurationSettings\Data\Domain\SMARTEAM.std.nls.xml to point to the NLS folder of the core services server.

WizSrc Upgrade

This step is optional if your target release is R13 and not R12.

1. Run SMARTEAMWizSrcUpgradeWizard.exe from the Upgrade Folder of your SMARTEAM directory.

V5R13+ specific upgrade

Product Installation

1. Install R13 core services on a server (dedicated or not) (select foundations).

2. Install last available SP on this server.

3. Install R13 GA on the upgrade machine.

4. Install last available SP on this machine.

5. Install SmartDBUpgradeV5R13SPx.exe from the kits\SmartDBUpgrade directory of the service pack CD (to run later Wizsrc update only).

System Configuration

1. Get the reference configuration files and put them on the upgrade machine:

   SmTeam32.ini - genrep.ini - ServerSafeScripts.ini - SmarDESK.ini

   SmartDbExplorer.ini - SmartQuickReport.ini - SmERPSyncServer.ini

   Smupgsrc.ini - Smvlt32.ini - SMWIZA32.INI - SmWorkFlow.ini

Note 1 :If you are using both administrator and client SmTeam32.ini, merge the two on one single reference file.

Note 2 : The user specific configuration will be automatically migrated on first user log-on.

2. Upgrade the system configuration using the Configuration migration wizzard.

3. Verify the migration with the Configuration Manager Editor (Start \ Program Files \ SMARTEAM \ Administrator Tools).

WizSrc Upgrade

1. Run the SMARTEAMWizSrcUpgradeWizard.exe.program from the Upgrade folder of your SMARTEAM directory.

Web Editor

The user-defined configuration in Web Editor must be redefined manually in R13

Web Viewer/Markup monitor

Uninstall then reinstall these components.

Post-activities

Deployment

1. Upgrade now all server and client machines by installing all pre-installed products of your SMARTEAM target release.
2. Install on all SMARTEAM machine the last available service pack.

Verifying functionality

1. Check CAD Integration existing and new objects with links, including drawings, large assemblies, parts within assembly and single parts, design tables -
2. Check Data Model Designer, browse your updated database schema.
3. Check Sequence Designer, create and test new objects that uses sequences.
4. Check user scripts functionality.

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 

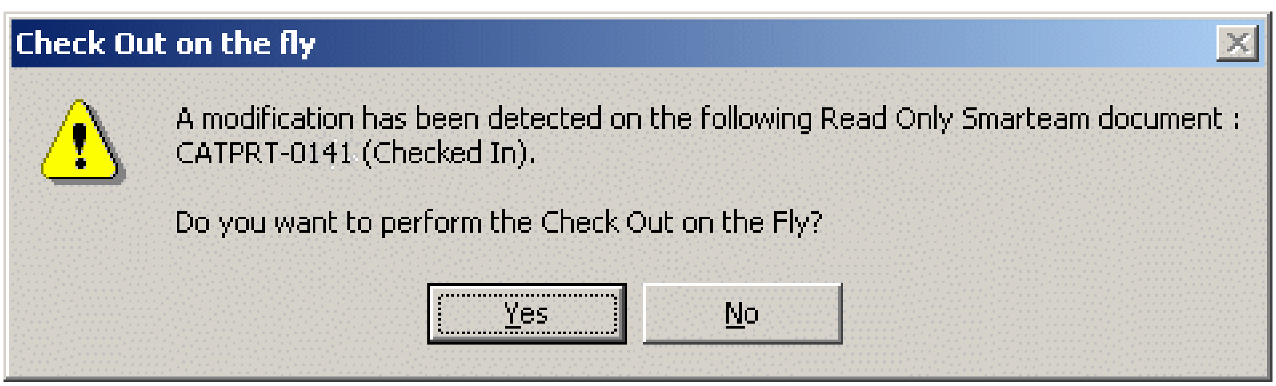
updates the Product Structure and File Desk **icons** with regard to SMARTEAM information.

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.

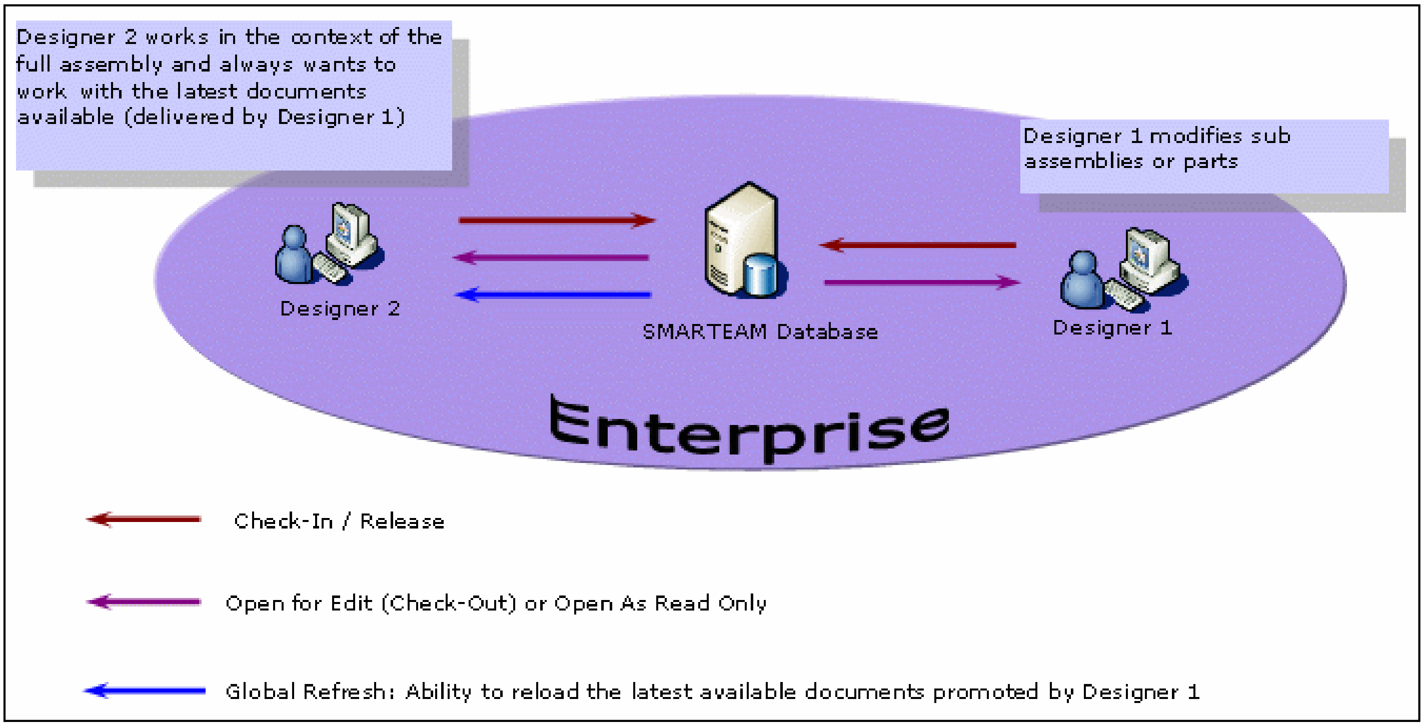
Using Global Refresh

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 out 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:

   - Dedicated icons indicate that some documents are no longer up-to-date. 

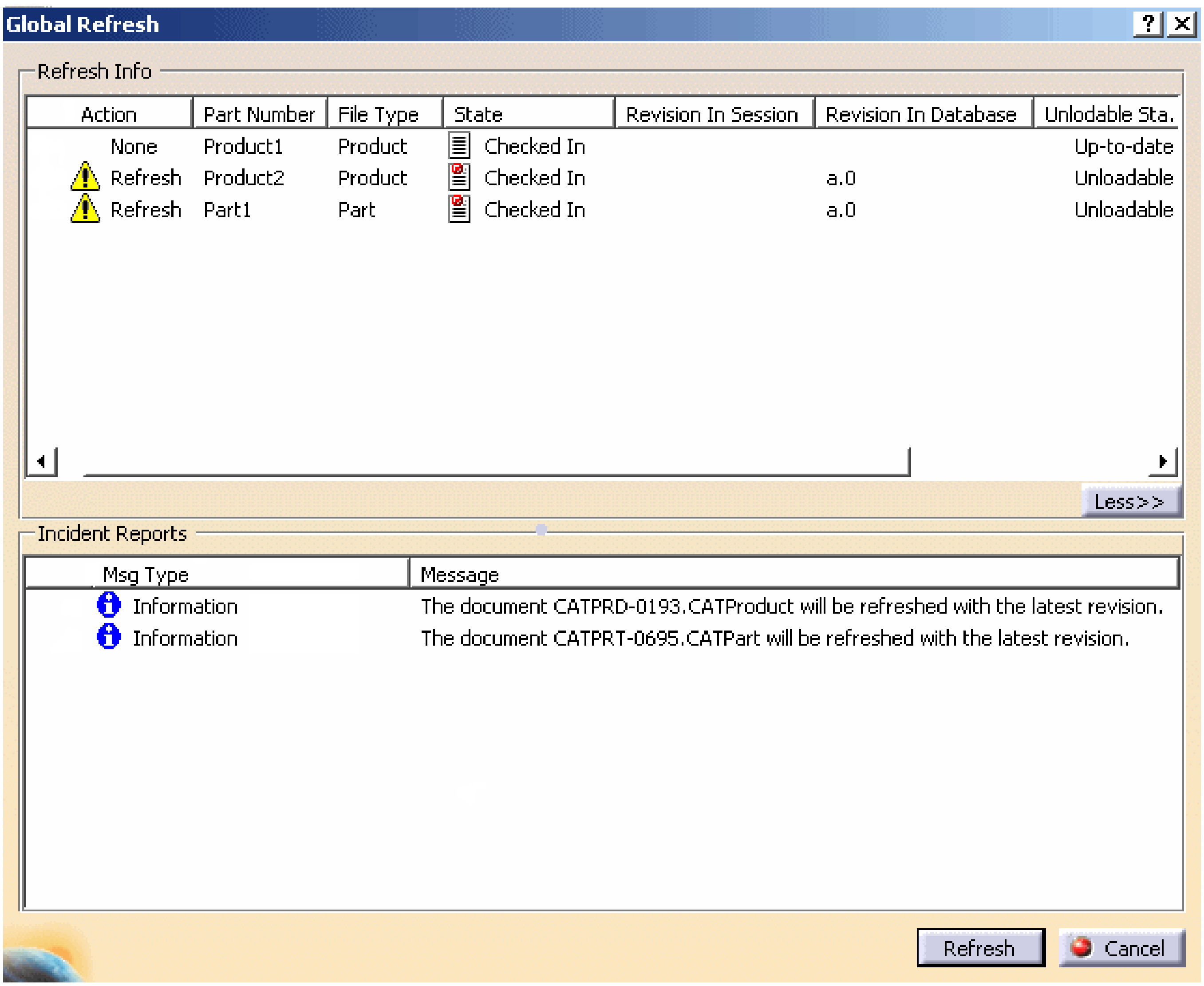
   or

   - 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, refer to 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:

None	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. Note: 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.
Refresh	More recent revision is available in the SMARTEAM database than that present in the current CATIA session. So the document will be refreshed with the available latest revision from the database.
Reload	The more recent revision of the document is available in the shared workspace than that present in the current CATIA session. So the current document will be reloaded with the latest revision of the document from the shared workspace. This typically happens when two or more designers are modifying the same document. Reload action is not related with SMARTEAM. Currently, it supports only those documents that are stored in SMARTEAM.

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)

Part Number

The Part Number column displays the Part number of the document

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

State

The State column displays the SMARTEAM state of the document. Following are the valid values for State:

New	The document is saved in SMARTEAM. No Life Cycle operation is performed on it.
Checked In	The document is in Checked-In state in SMARTEAM database.
Checked Out	The document is Check-In in SMARTEAM database and is currently checked out for modifications.
Released	The document is in released state in SMARTEAM database.
New Release	The document is in New Release state in SMARTEAM database.
Obsolete	The document is in obsolete state in SMARTEAM database.
Not in Database	The document is not stored in SMARTEAM.

Revision in Session

The Revision in session column displays the SMARTEAM revision of the document present in the current CATIA session.

Revision in Database

The Revision in Database column displays the latest available revision of the document present in the current CATIA session.

Unloadable State

The Unloadable State column displays the document state with respect to the CATIA session. Valid values are as follows:

Up-to-date	The document is up-to-date and does not need to be unloaded from the session.
Unloadable	The document is not the latest one and can be loaded in order to reload the latest revision of the document.
Modified	The document needs to be unloaded but cannot be unloaded because it is under modification in the current session. To make it unloadable, you need to save the document. Otherwise, the document will be skipped from the refresh list.
Undo/Redo	The document is present in the Undo log and Undo Redo mechanism hence cannot be unloaded. In order to make it unloadable, user has to clear the Undo log history. Otherwise, document will be skipped from the refresh list.

File Name

The File Name column displays the name of the document.

3. Click **More**.

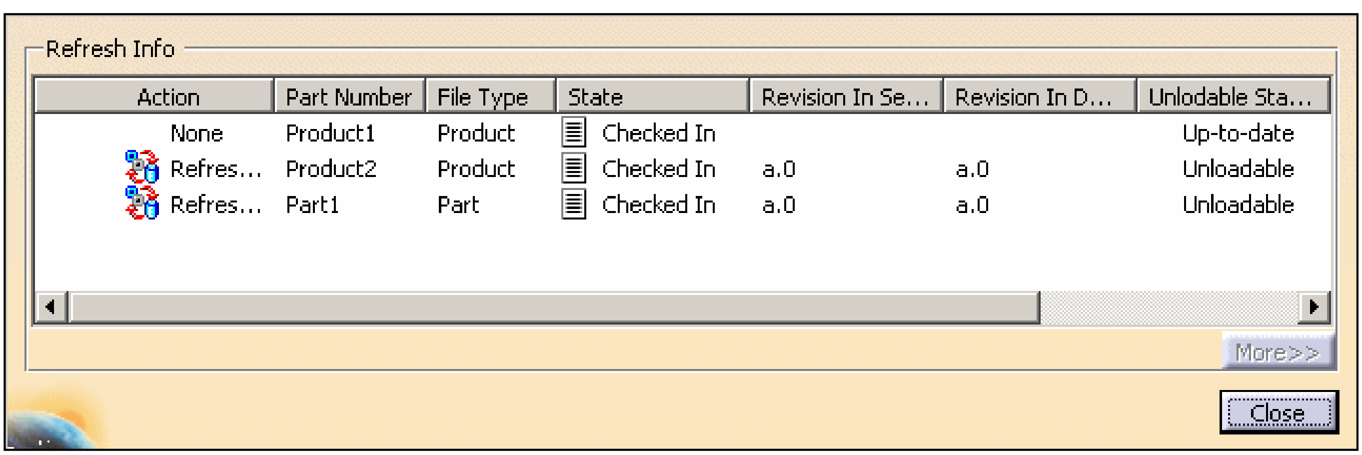
The dialog box expands and displays additional information on session documents requiring a particular attention.

Messages that can be displayed are as follows:

Msg Type	Message
Error	Document modified in Shared workspace, but not stored in SMARTEAM. Therefore it will not be taken into consideration during the Global Refresh operation.
Error	Document modified or locked in session and cannot be unloaded from session. Therefore it will not be taken into consideration during the Global Refresh operation.
Information	Document is going to be refreshed with the latest available revision
Warning	Document new in assembly
Information	Document does not exist in SMARTEAM database.

4. 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
Reloaded		The document in the current CATIA session has been replaced with the latest available document in the shared workspace.

State

The State column displays the new state of documents once the global refresh is complete.

Revision in Session

The Revision in session column displays the SMARTEAM revision of the document in the current CATIA session after the global refresh. For a SMARTEAM object, the value should be the same as the value in the Revision in Database column.

5. Click **More**.

   The dialog box expands and provides a report on the global refresh operation. Warning or error messages can be displayed.

   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.

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 

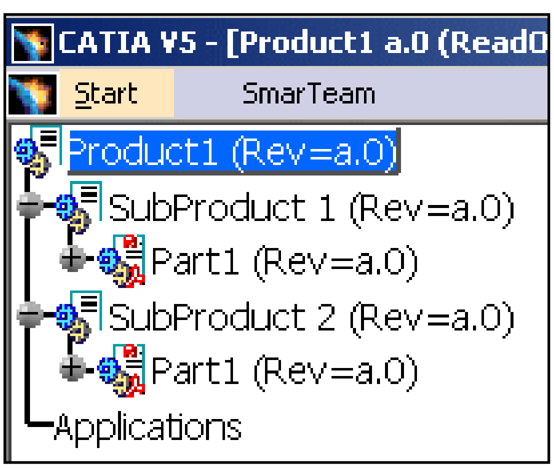
appears on documents' icons in CATIA specification tree whenever the status of these documents as shown in your CATIA session is not consistent with the SMARTEAM database contents.

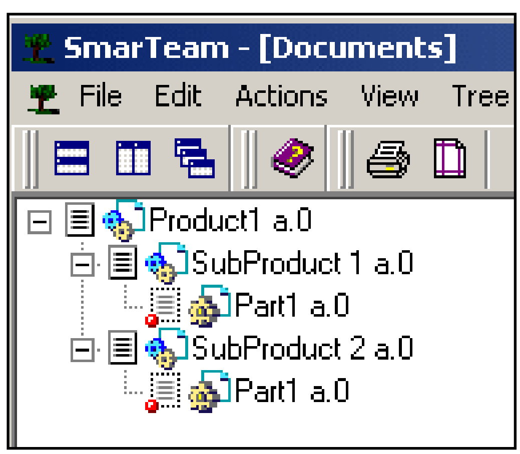
Displaying this type of information ensures that designers are informed that they need to perform check out operations to receive the most updated version of documents referencing sub-components.

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.

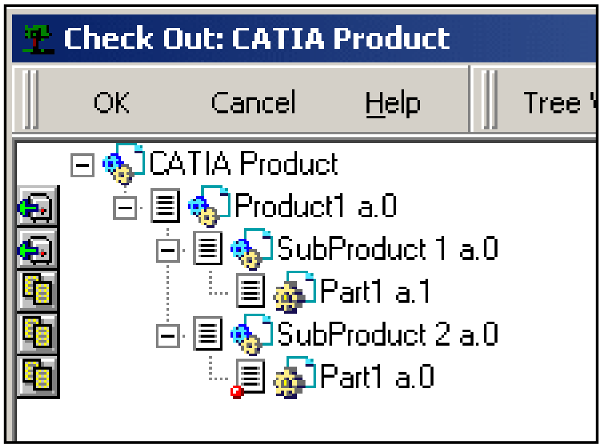
The assembly displayed in CATIA The assembly displayed in SMARTEAM





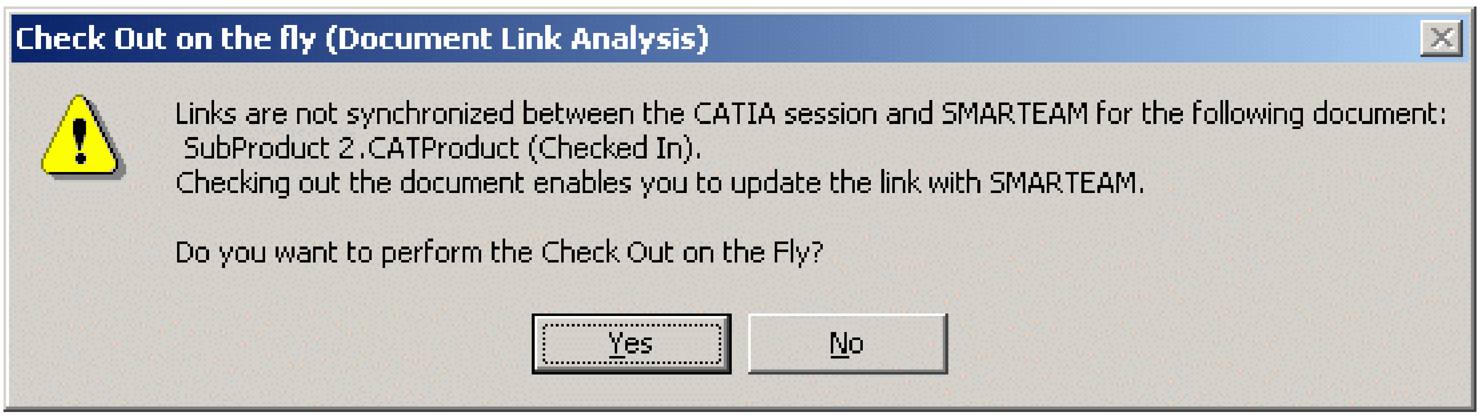
To make SubProduct 1 points to the latest revision available in SMARTEAM, it is necessary to apply the **Open For Edit** command onto Product1.

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:



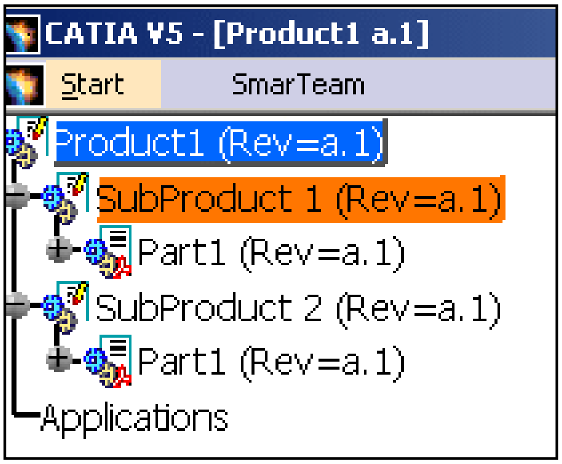
Applying the **Open for Edit** command on Product 1 both checks out the document and opens it in CATIA. Only one revision of Part1 is loaded in CATIA (only one revision is located in the work directory): this is the last revision, that is Part a.1. Consequently, SubProduct2 a.0 is in a state different 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:



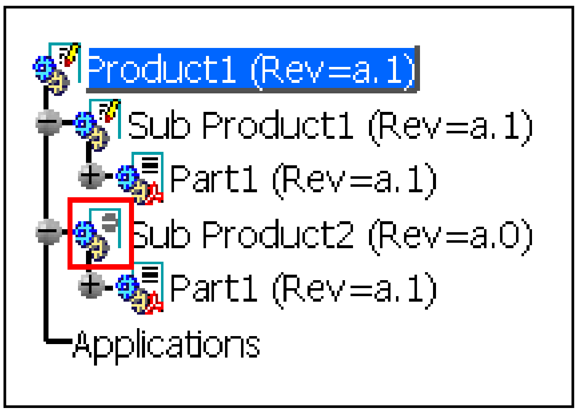
Accepting a **Check Out on the fly** operation allows to create a new revision of SubProduct2 pointing to Part1a1.

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, refer to System Variables for CATIA and SMARTEAM.

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

The ability for the user to perform a check-out on the fly is defined by the system administrator. For more information, refer to System Variables for CATIA and SMARTEAM.

Enriched decision support is provided by means of the native SMARTEAM support of V5 links.

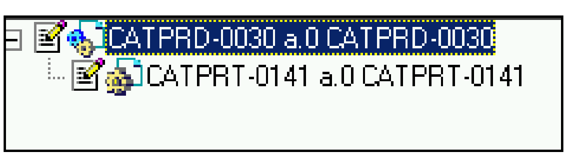
icon	LINKS
	CATIA Product Link (P)
	CATIA Design Link (D)
	CATIA Rule Base Link (RUL)
	CATIA Design table Link (DT)
	CATIA Downstream Application Link (DA)
	CATIA Reference Link (REF)
	CATIA Contextual Link (C
	CATIA Result Link (RES)
	CATIA Is Composed Of (IS)

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

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

- ## Product Structure link

The *Product Structure* link plays a part in the BOM structure. It is displayed as a standard hierarchical link. Example:
CATProduct->CATProduct, CATProduct->CATPart, CATProduct->Internal Component...



- ## Is Composed Of link

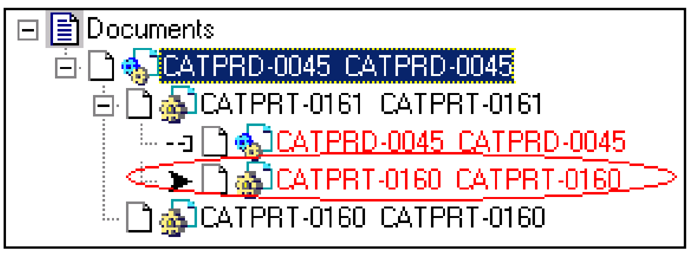
The *Is Composed Of* link is an aggregation link but which plays no part in the BOM structure. It is displayed as a standard hierarchical link.

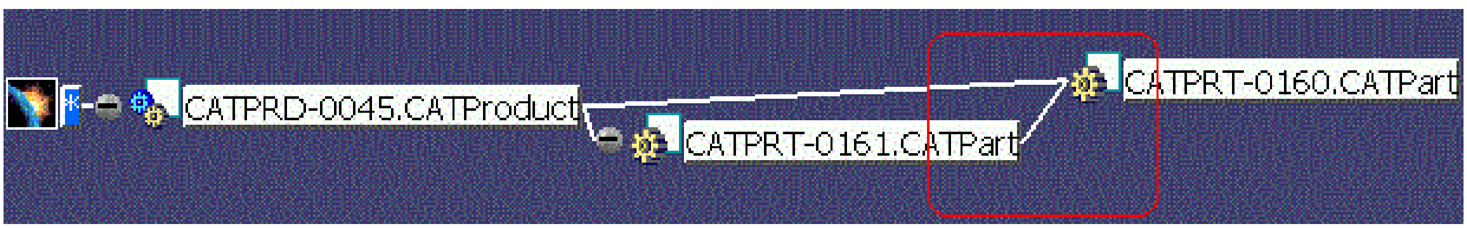
Example:
CATDrawing->Sheet, Catalog->catalog

- ## Design link

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

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

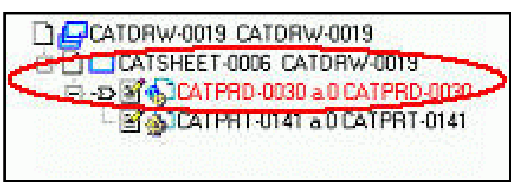




- ## Downstream Application link

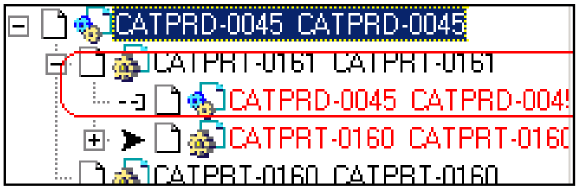
The *Downstream Application* link is a link between a downstream application and the design data

Example:
CATDrawing->CATProduct/CATPart, CATProcess->CATProduct/CATPart, CATAnalysis->CATProduct/CATPart



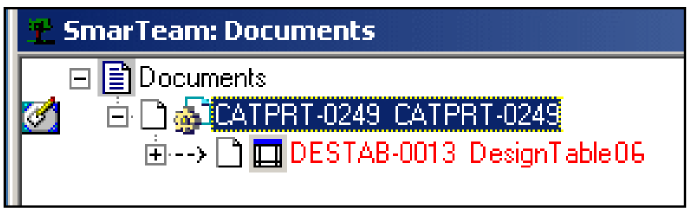
- ## Contextual link

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



- ## Design Table link

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



- ## Result Link

The *Result* link corresponds to CATIA output links:

Example:
CATAnalysis->CATAnalysisResult, *CATAnalysisComputation*...
CATProcess->NC (aptsource, 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, refer to Customizing the Links Display.

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.

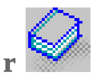
Accessing CATIA Catalogs

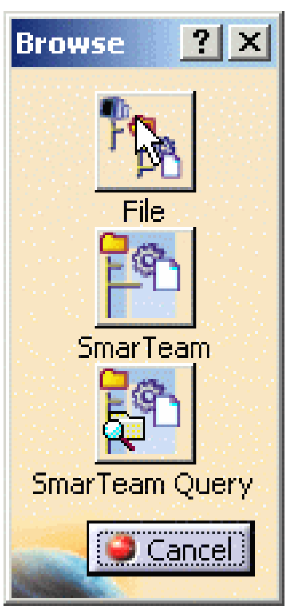
Design engineers who need to instantiate catalog components (parts, products, V4 model etc.) can access their catalogs using one of the following methods:

- Searching in the Shared Directory defined by your librarian
- Searching in the SMARTEAM Vault

Searching in the Shared Directory

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

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

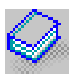
4. Open the document of interest.

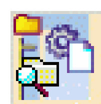
   Using this method, the CATIA Product creates a reference to Standard CATIA Parts in the shared workspace.

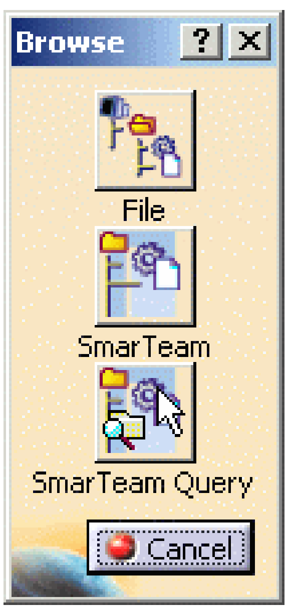
   For information about shared directories, refer to Creating and Saving a Catalog. To get precisions about instantiating catalog components, refer to the *Component Catalog Editor User's Guide*.

Searching in the SMARTEAM Vault

Searching in the SMARTEAM Vault is an alternative way of accessing catalogs. Although we do not recommend this method, we comment it for users who need to work in nomad mode.

1. Click the **Catalog Browser**  icon.

2. Perform a SMARTEAM query: just click the **SMARTEAM Query**  icon from the Browse dialog box.

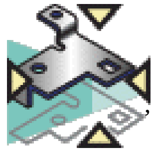
   Depending on on how your administrator configured your CATIA session, the File Selection dialog box may appear. In this case, just click the **Cancel** button to close it and enable the SMARTEAM Query capability in the Browse dialog box.

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.

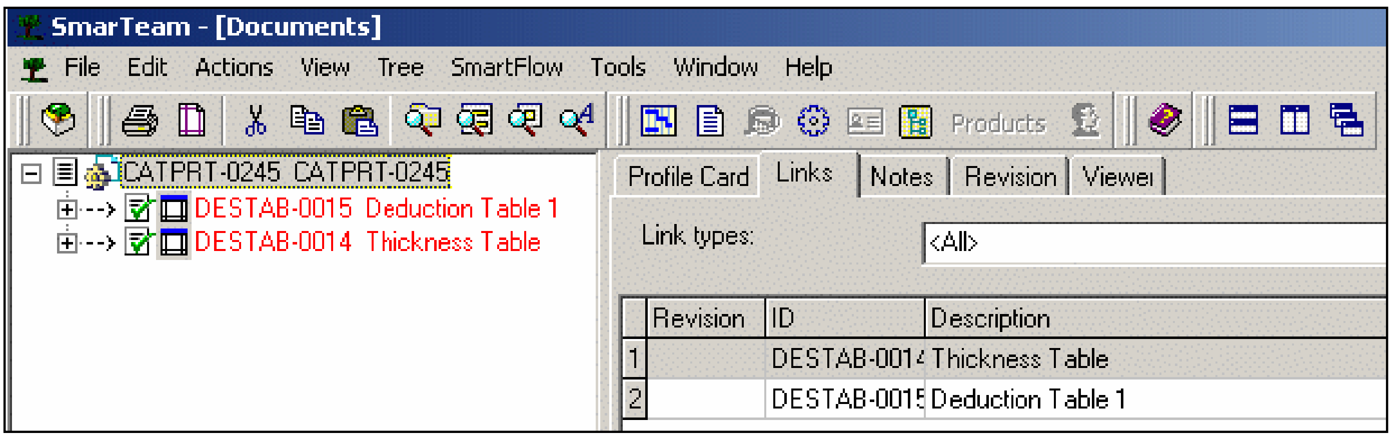
Accessing Sheet Metal Bend Tables

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:

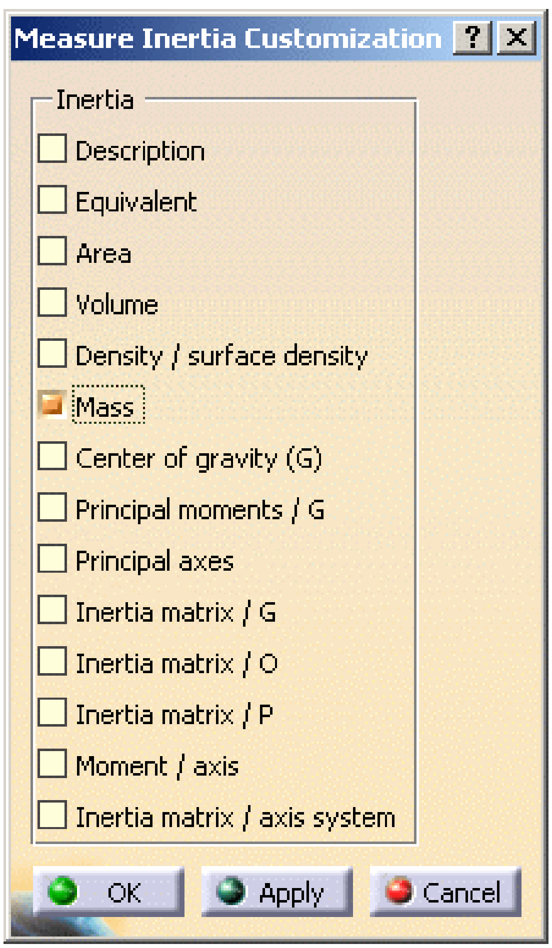
- Calculating a Mass Value
- Creating a Formula
- Saving the Part

Calculating a Mass Value

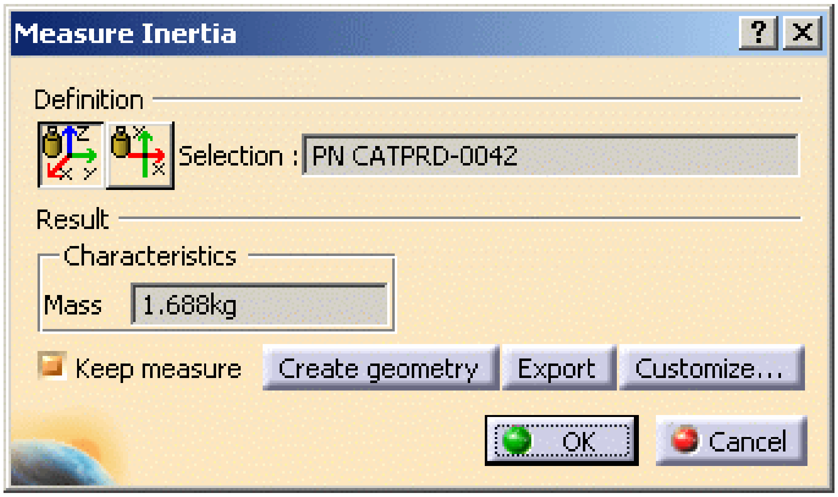
1. Once you have created your part, click the **Measure Inertia** icon .
   The dialog box that appears lets you measure several properties of the geometrical element you want. The mass property is one these properties that can be calculated.

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.



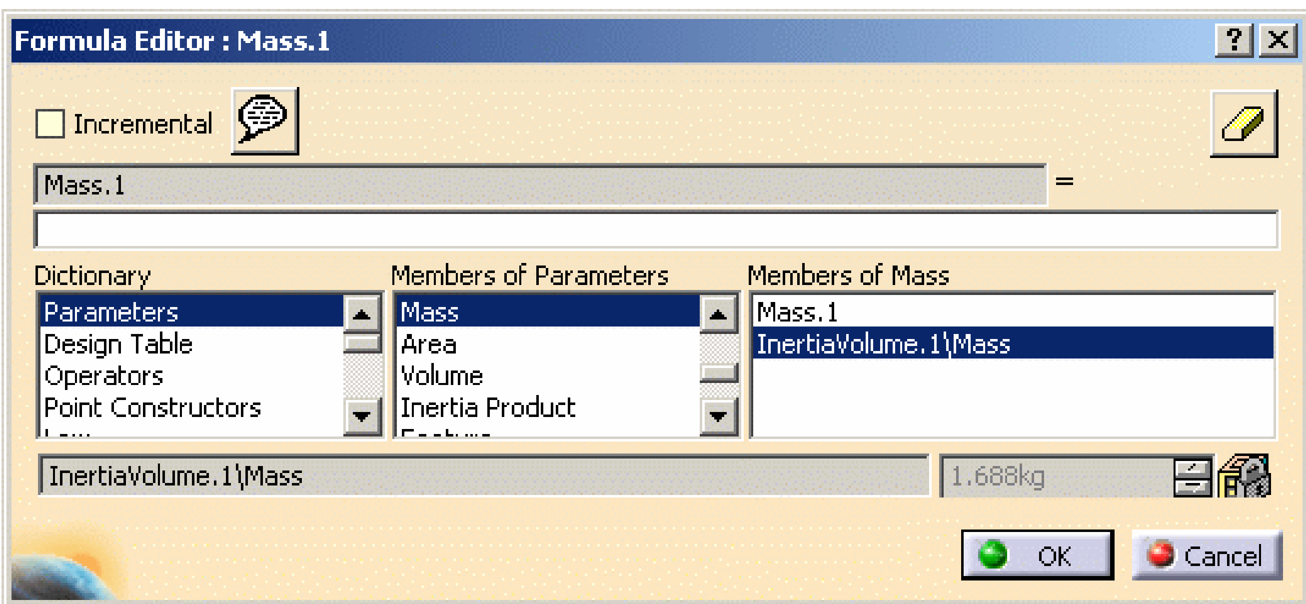
When done, the Mass value is displayed in the dialog box.



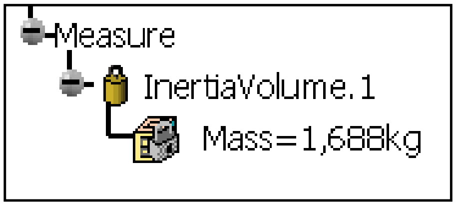
Creating a Formula

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

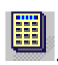
This scenario outlines the different operations you need to perform when handling CATAnalysis documents in the Generative Structural Analysis workbench  .

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 .

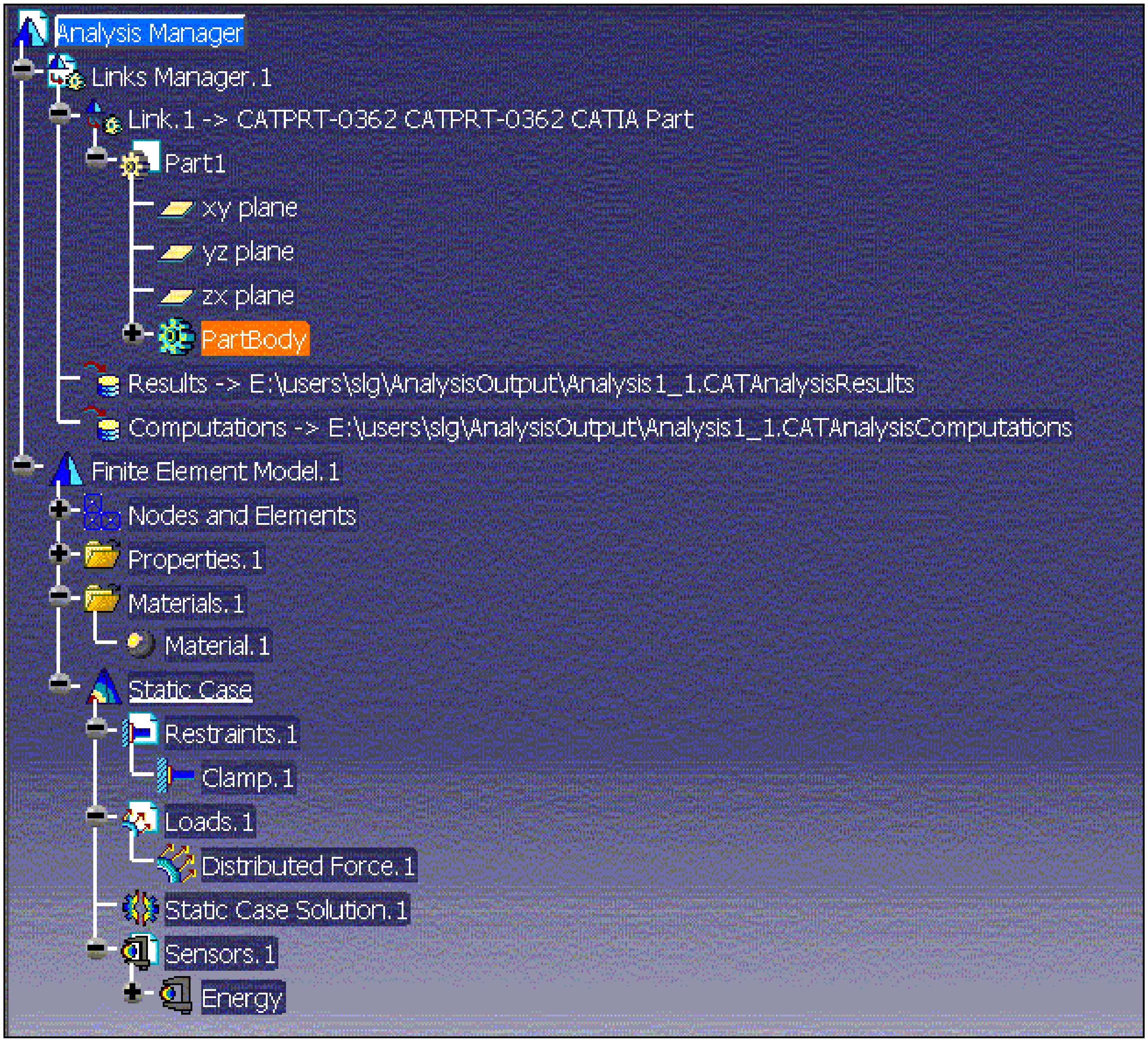
   Among other things, two files are generated and their paths are also displayed in the specification tree. You obtain a result similar as this:

   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.

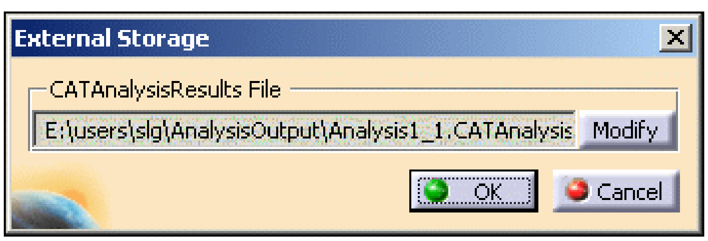


Renaming the Files Generated

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

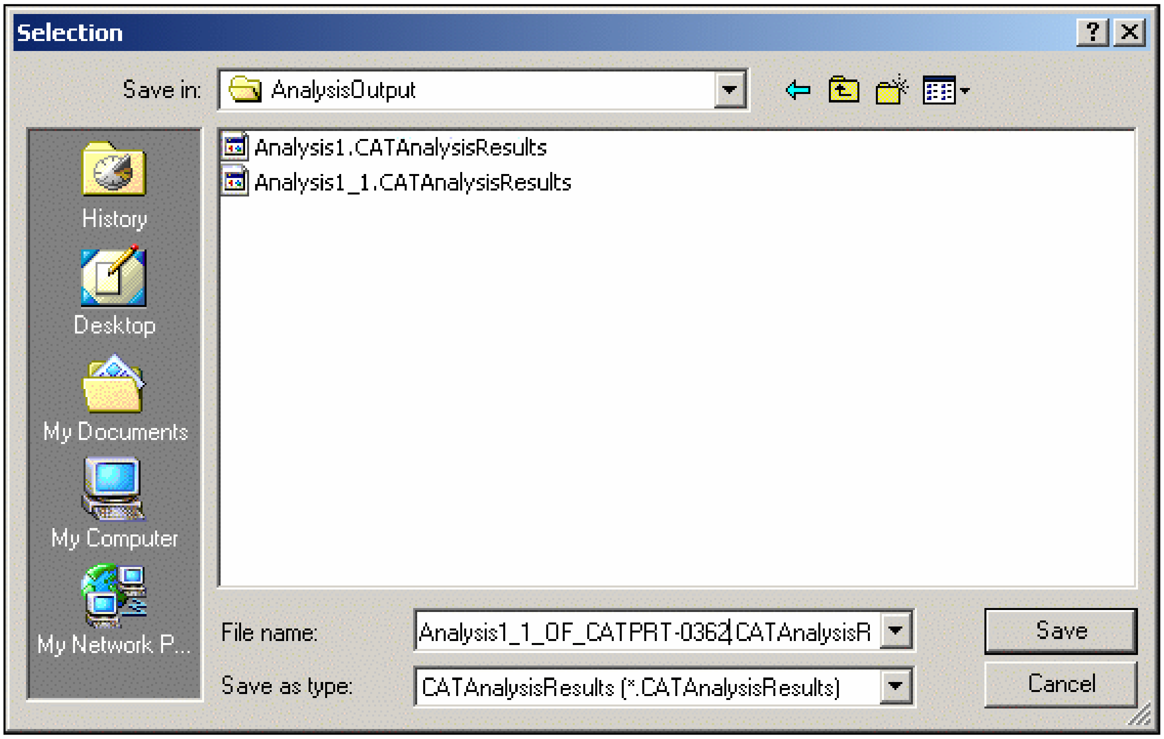
5. To ensure that both files will be identified with a unique name in the database, you need to rename them. To do so, right-click the file to be renamed and select **Results object->Definition...**

   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.

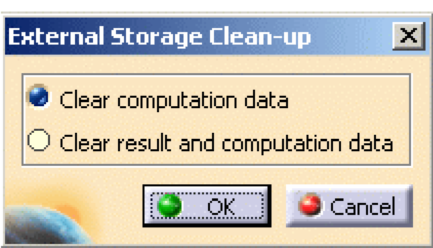
   We recommend you use the names of the referenced Part or Product.



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.

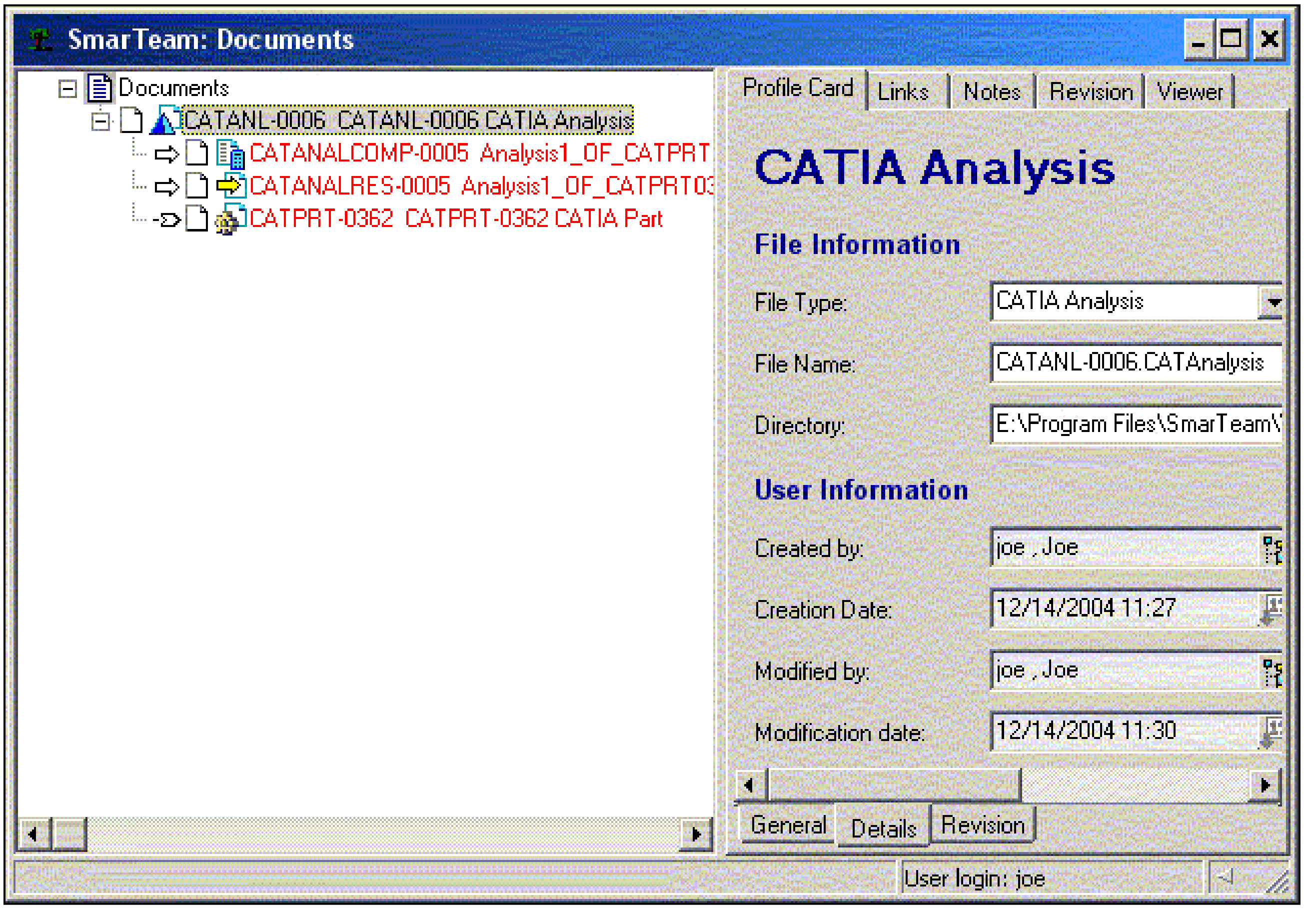


8. Save the documents in the SMARTEAM database.

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.

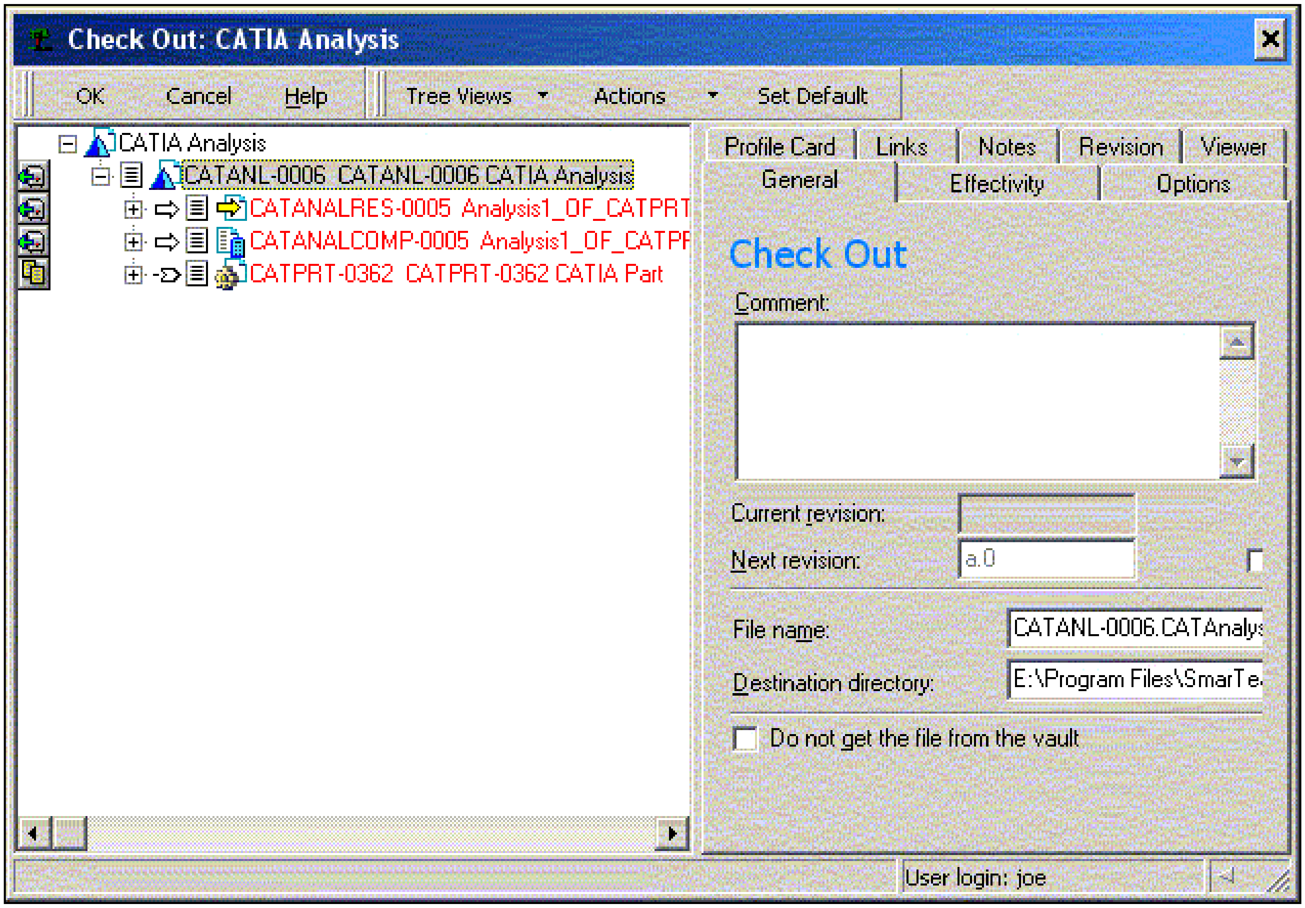


9. To secure the documents in the electronic vault, select **SMARTEAM->Life Cycle->Check-In.**

Checking Out the CATAnalysis Document

9. In case you wish to modify the CATAnalysis document later on, perform an Open for Edit operation.

You will note that a check-out of the analysis proposes by default the check-out of the result files.


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_CATPI
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.

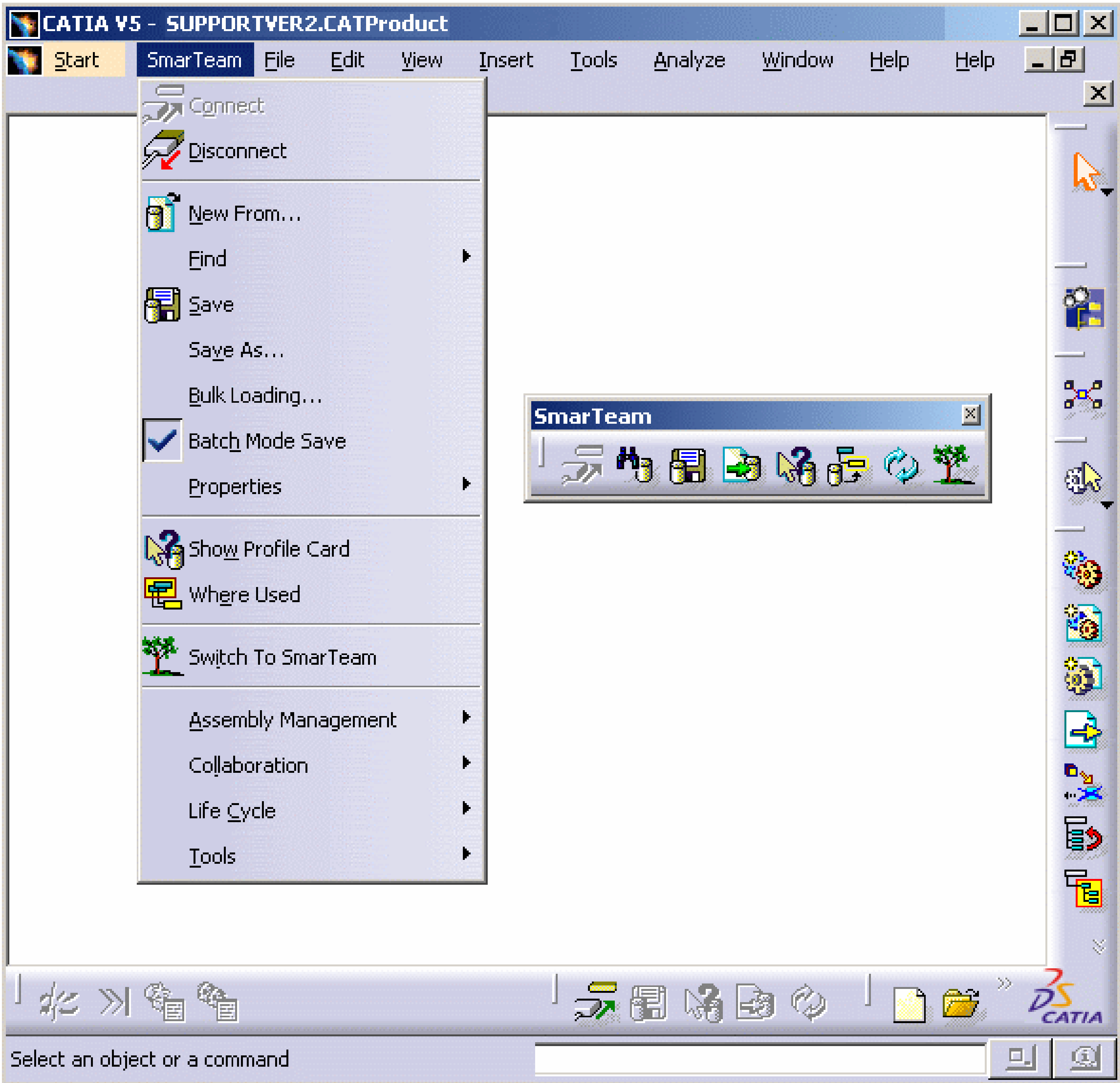
We suggest consulting your system/network administrator and a SMARTEAM consultant regarding to this issue. It has been proven that correct setup can significantly improve performance.

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.

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
SMARTEAM Toolbar
SMARTEAM Collaboration Toolbar

SMARTEAM Menu

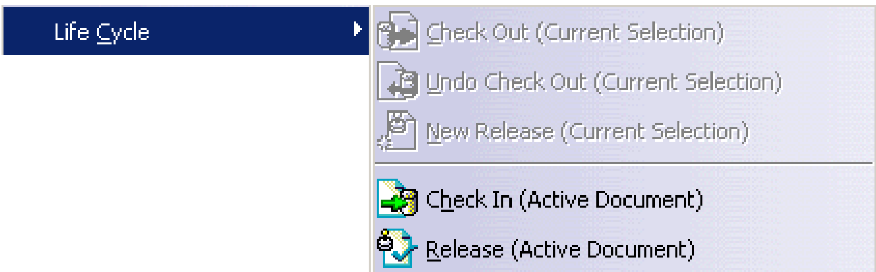
Start | SMARTEAM | File | Edit | View | Insert | Tools | Windows | Help

SMARTEAM

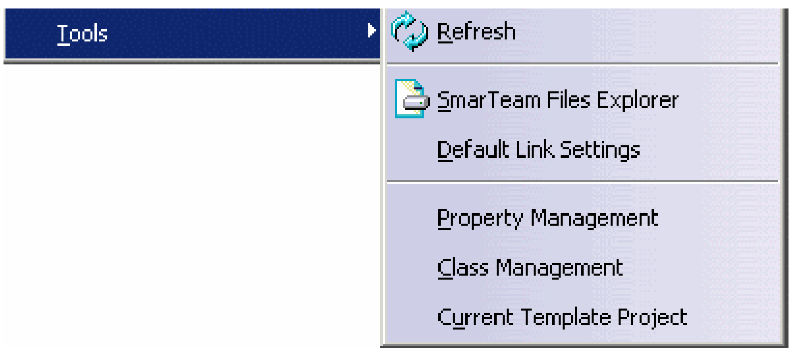
Connect	See Connecting to the SMARTEAM Database.
Disconnect	In a CATIA session, disconnects the SMARTEAM database.
New From...	Creates, in the CATIA session, a new document based on an existing SMARTEAM database document. For an example of use, see Creating a Drawing Document from a Template.
Find ▸ Find Document / Find Document by Attribute / Find Document by Example	• **Find Document** See Finding.
Save Save As...	See Saving a Part, Saving an Assembly or Saving a Drawing.
Bulk Loading...	See Importing CATIA Data Inside SMARTEAM.
Batch Mode Save	See Saving a Part or Saving an Assembly.
Properties ▸ Load from Database / Save in Database / Map a Text Value...	• **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.
Show Profile Card	Provides the database view of the document currently displayed in a CATIA session. See Showing Profile Cards.
Impact Analysis	See Analyzing the Impact of a Change
Where Used	See Finding Out Where a Document Is Used.
Switch To SmarTeam	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



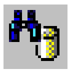
See Lifecycle Menu Commands.



- **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, refer to 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

SMARTEAM Toolbar

 See Connecting to the Database

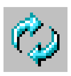
 See Finding

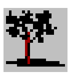
 See Saving a Part, Saving an Assembly or Saving a Drawing

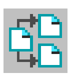
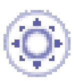
 See Checking In a Part, Checking In an Assembly

 See Showing Profile Cards

 See Building an Assembly

 See Tree Refresh Command

 Switches from the CATIA session to the SMARTEAM application

  Analyzing the Impact of a Change

SMARTEAM Collaboration Toolbar

 See Using Global Refresh

Glossary

B C D G M O P R S

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

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

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

desk A CATIA tool that lets you view the relationships between different documents and obtain information about their properties.

design table 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).

dialog box 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.

document 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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Symbols

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