



CATIA Team PDM

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Preface

Overview of PDM

The Need

Next to people, information is your company's most important resource. However, many companies are realizing that they have become so overwhelmed with the volume of data that need to be stored and managed that they need to find comprehensive solutions to overcome their data disorganization. This problem is particularly acute in CAD design shops, where many complex designs and documents undergo several stages of revision as part of their development cycle. Incorrect revision of outdated designs results in the need for additional changes later in the design process.

As your company grows, so do the number of designs, specifications and proposals that are created in the course of project development, and over a project's entire life cycle. Organizing and managing these designs in a secure database is of primary importance. In addition, designers and engineers need to ensure that the correct revision of a part is correctly identified and used during the design stages of a project.

What are the designer's Design Tasks?

The Design Engineer tasks are varied and manifold. The high-pressure world of marketplace deadlines requires the design engineer to work efficiently and effectively in accomplishing his or her engineering goals.

CATIA is a mechanical CAD application that provides the means to accomplish the design engineer's primary tasks:

- To create new parts
- To assemble component parts into a complete assembly
- To create drawings

What are the basic Data Management tasks?

The engineer designs, redesigns, and ultimately releases documents during the life cycle process. All of these documents must be managed during all stages of their development and undergo three basic *Data Management tasks*:

- File Management: Any file transaction where you save or retrieve any information from a library, a file manager or a database.
- Revision Management: The tracking of the change process and maintenance of versions during the document's lifetime.
- Acceptance and Release: Accepting a design and authorizing its use for production.

The Solution: PDM

Product Data Management (PDM) enables engineers to manage their documents during all stages of their development. Using PDM, you should be able to manage your documents electronically to maintain, access and modify designs and technical related documents quickly, securely and cost effectively. A PDM should include the following requirements:

- Electronic data storage and management
- Flexible file organization
- Support of multiple file formats

- Document security
- Timely access to information
- Workflow automation

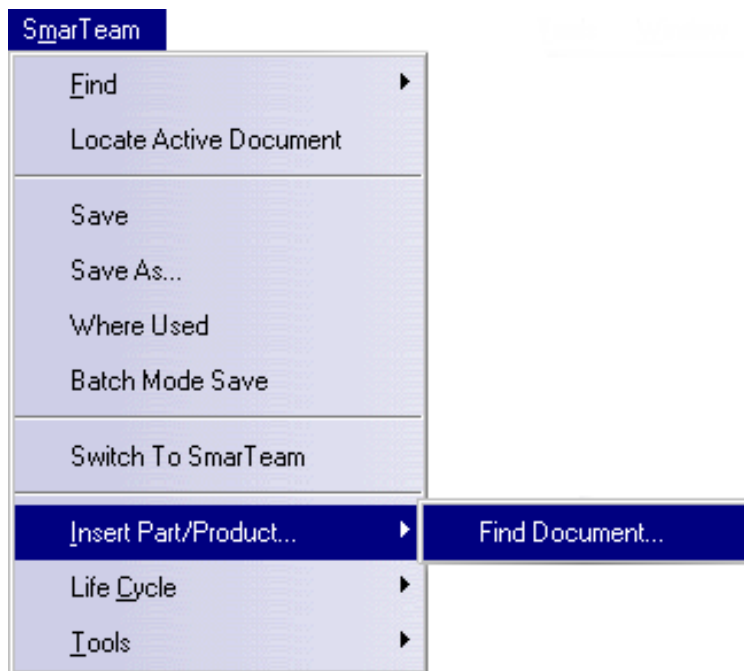
CATIA - TeamPDM is a product data management solution that aids the engineer in integrating and synthesizing the multitude of process events into two fundamental tasks:

- Design Tasks
- Data Management Tasks

Design Tasks

As the engineer designs Parts, Products, Drawings, the engineer can use the **CATIA - TeamPDM** options to enhance the CATIA capabilities.

For example: The **Insert Component** option enables the engineer to insert an existing Part or Product into a new Assembly as a component.



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 **CATIA - TeamPDM**.

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

Using **CATIA - TeamPDM**, the engineer functions within an *open environment*, that is, being able to manage all engineering data within the CATIA environment.

The **CATIA - TeamPDM** 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.

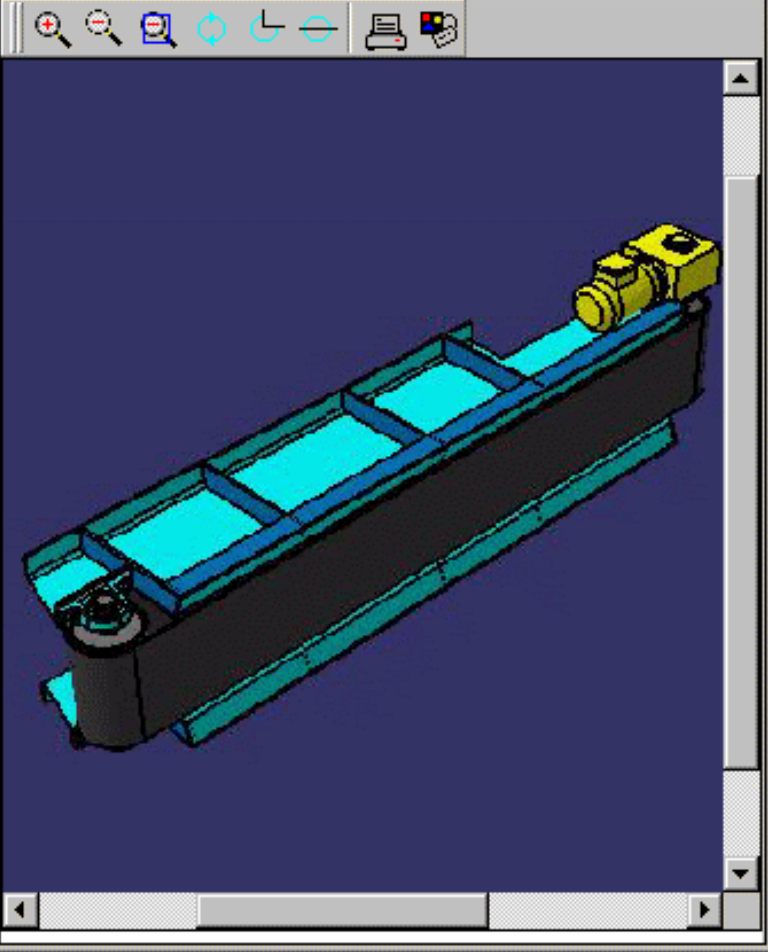
The example below demonstrates the power of **CATIA - TeamPDM** functionality:

FIND

All CATIA: CATIA

ID	State	Creation Date	Description
1		05/10/1999 1	PN20000
2		05/10/1999 1	CONNECT
3		05/10/1999 1	PN30000B
4		05/10/1999 1	PN53000
5		05/10/1999 1	PN51000
6		05/10/1999 1	PN52000
7		05/10/1999 1	PN40000
8		05/10/1999 1	PN42000
9		05/10/1999 1	PN43000
10		05/10/1999 1	PN41000
11		05/10/1999 1	PN10000
12		05/10/1999 1	PN14000
13		05/10/1999 1	PN15000
14		05/10/1999 1	PN13000
15		05/10/1999 1	PN11000
16		05/10/1999 1	PN12000
17		05/10/1999 1	PN60000
18		05/10/1999 1	PN62000
19		05/10/1999 1	PN63000
20		05/10/1999 1	PN61000
21		05/10/1999 1	Machine
22		05/10/1999 1	PN70000
23		05/10/1999 1	PN71000
24		05/10/1999 0	PN72000

Profile Card | Links | Notes | Revision | Viewer



User login: joe

SELECT OPERATION

All CATIA: CATIA

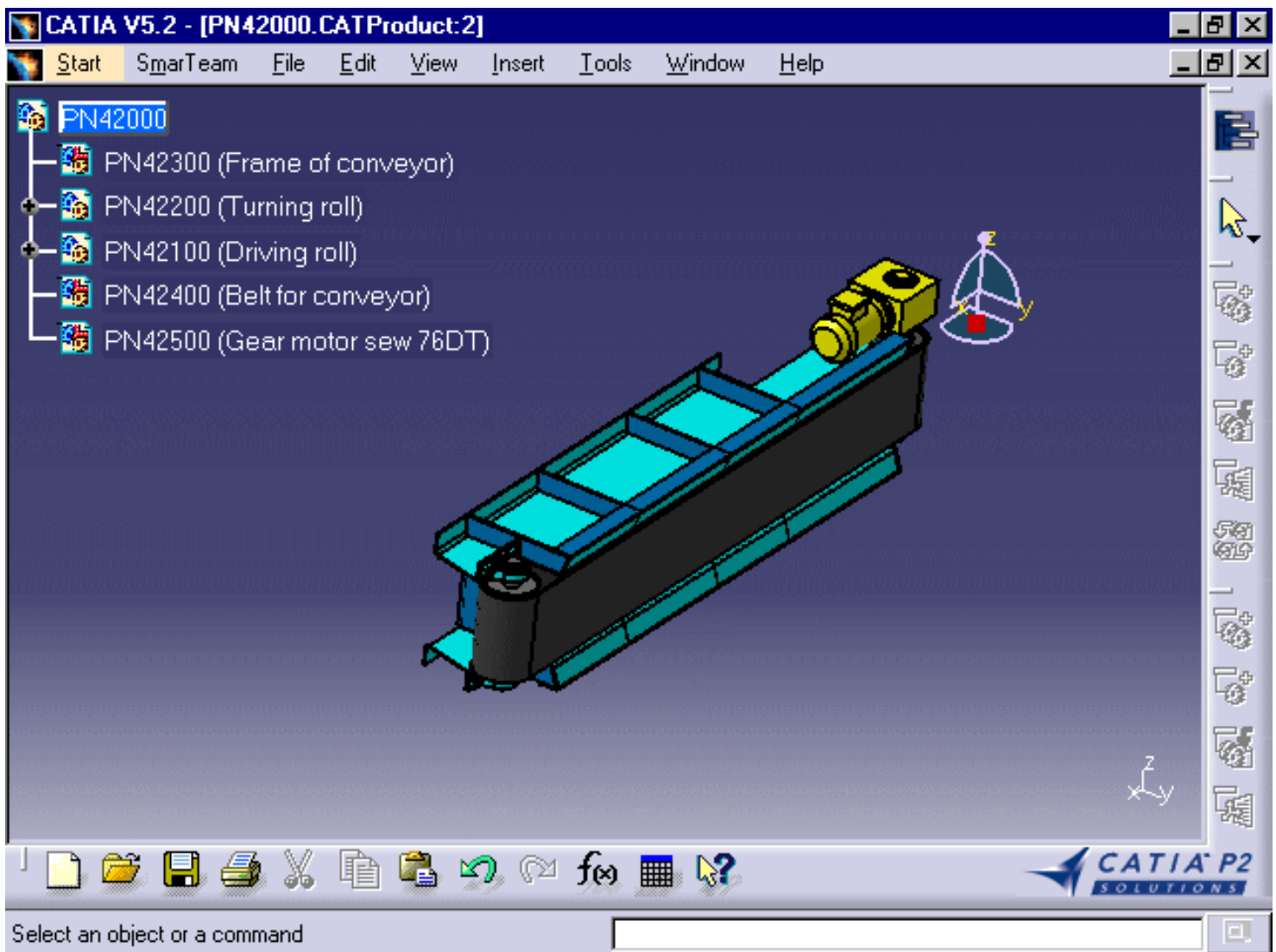
ID	State	Creation Date	Description
1		05/10/1999	1 PN20000
2		05/10/19	
3		05/10/19	
4		05/10/19	
5		05/10/19	
6		05/10/19	
7		05/10/19	
8		05/10/19	
9		05/10/19	
10		05/10/19	
11		05/10/19	
12		05/10/19	
13		05/10/19	
14		05/10/19	
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21		05/10/19	
22		05/10/19	
23		05/10/19	
24		05/10/19	

Profile Card | Links | Notes | Revision | Viewer

- Add
- Update
- Delete
- Add as Copy
- Register
- Check Out
- Check In
- Release
- New Release
- Obsolete
- Undo Check Out
- Edit**
- View**
- Redline
- Print
- Copy File
- Initiate Process
- Send to e-mail recipient...
- Send to SmarTeam recipient...
- Open Views
- Save as View...
- Grid Property

User login: joe

VIEW



What Is CATIA - TeamPDM?

CATIA - TeamPDM 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. The **CATIA - TeamPDM** 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.

CATIA - TeamPDM 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.

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

CATIA - TeamPDM comes with a complete database structure designed specifically for CATIA. It also provides CATIA users with the ability to perform advanced searches, maintain revisions and preview documents without opening the application that created them.

Features and Benefits

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

- Browse through any **CATIA - TeamPDM** window to view a hierarchical listing of **CATIA - TeamPDM** 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 **CATIA - TeamPDM** 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 sub-Assemblies (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.

CATIA - TeamPDM Design Environment

The **CATIA - TeamPDM** design environment provides engineers with quick and easy access to the powerful **CATIA - TeamPDM** tools.

The *SmarTeam* menu provides powerful functionality and enables you to perform the following **CATIA - TeamPDM** operations.

Locate Component

Displays the Profile Card of the selected component.

Where Used

Displays a list of the parents of the document.

Check Out

Enables you to check out the document from the vault.

Check In

Enables you to check in the document to the vault. A copy of the document remains at your desktop (in read-only mode).

Release

Enables you to release your document, thereby checking it into the *Released* vault. A copy of the document remains at your desktop (in read-only mode).

New Release

Enables you to check out a new release version of your document from the *Released* vault.

Open




Opens the selected Part or Product in a CATIA single Part/Product window.

CATIA - TeamPDM icons are displayed in various **CATIA - TeamPDM** windows. Each icon displays the state of the document and additional information as follows:












States

	No state
	New
	Checked In
	Checked Out
	Released
	Obsolete

Additional info

	Not latest *
	By other
	Under operation
* Recommended not to Check Out/New Release	

The following table describes the **CATIA - TeamPDM** icons:

Icon	Name	Description
	New	A document has been saved but not yet registered.
	New By Other	A document has been saved, and it has not yet been registered. The document is currently being modified
	Checked Out	A document has been checked out. Its current state is <i>Being Modified</i>
	Checked Out By Other	A document has been checked out. Another user has copied it to his or her desktop
	Checked In	A document has been checked in to the vault. Its current status is <i>Being Modified</i>
	Checked In, Not Latest	A document has been checked in to the vault, but a later version of the document exists
	Checked In, Under Operation	A document has been checked in to the vault. Another version of the document is being modified
	Released	A document has been checked in to the <i>Released</i> vault. Its current status is <i>Released</i>
	Released, Not Latest	A document has been checked in to the <i>Released</i> vault, but a later version of the document exists.
	Released, Under Operation	A document has been checked in to the <i>Released</i> vault. Another version of the document is being modified.
	Obsolete	A document has been moved to the <i>Obsolete</i> vault.

[Preferred Road Maps](#)



[Where to Find More Informat](#)



Preferred Road Maps

This book is intended to the user who needs to become quickly familiar with *CATIA - TeamPDM* product. The user should be familiar with basic CATIA Version 5 concepts such as document windows, standard and view toolbars.

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



Where to Find More Information

Prior to reading this book, we recommend that you read the *CATIA - Infrastructure User's Guide*



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[Preferred Road Maps](#)






[Where to Find More Information](#)

Getting Started

 **CATIA - TeamPDM** provides the tools that allow the engineer to effortlessly add Parts and Products to the engineering database. These tools are *in-process* and keep you in the design environment at all times. This chapter provides brief instructions for adding Parts and Products to the **CATIA - TeamPDM** database, locating documents and launching them into CATIA.

 In order to use the **CATIA - TeamPDM** 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) with no password..

Task	
Quick Overview	
Adding Parts and Assemblies	
Modifying Parts and Assemblies	

Quick Overview



Each time you create a Part or Product in CATIA, the Part or Product should then be saved in the **CATIA - TeamPDM** database. After the Part or Product is saved in **CATIA - TeamPDM**, you can use all the **CATIA - TeamPDM** features to store and control the documents.

The **CATIA - TeamPDM** (*SmartTeam*) integrated menu provides two methods for saving documents:

- **Save:** Saves the document into the **CATIA - TeamPDM** database.
- **Save As:** Saves the document into the **CATIA - TeamPDM** 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 **Standard Parts** folder in the **Convoyer** project.

After you save a new Part or Product:

- A **CATIA - TeamPDM** window is automatically displayed which shows the new document that has been added to the **CATIA - TeamPDM** database.



Adding Parts and Products to the CATIA - TeamPDM Database



This section provides brief instructions for adding new and existing Parts/Products to the **CATIA - TeamPDM** database.

The first time a Part/Product is added to the database, a Profile Card for the Part/Product is created, and you can fill in the attributes of the document. Each subsequent time that you save the Part/Product, the Profile Card is updated accordingly.



In order to use the **CATIA - TeamPDM** 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) with no password.

After a Part/Product is added to the **CATIA - TeamPDM** database, you can use the many **CATIA - TeamPDM** features to manage your documents.

Adding a New Part	
Checking Out and Checking In a Document	
Copying Documents To Your Project Desktop	
Adding an Existing Part	
Saving a Part Again	
Adding a New Assembly	
Using the Batch Method	



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[Adding Parts and Assemblies](#)



[Modifying Parts and Assemblies](#)

Adding a New Part to CATIA - TeamPDM



After you create a new Part in CATIA V5, you should save it to the **CATIA - TeamPDM** database.



1. Create a new Part in CATIA.
2. From the *SmarTeam* menu, choose **Save**.
3. A CATIA *Save As* window appears. Name the file and click **Save**.
4. A **CATIA - TeamPDM** *Save As* window appears. In this window you can define the project and the parent to which the document belongs.



If this window is not displayed, your administrator has disabled it. You can define a project and parent for the document by choosing the **Save As** option.

- Choose a project. The Part will be saved as a document in the selected project.

Project:

- Choose a parent folder. The Part will be saved as a child of the selected folder in the selected project.

Folder selection tree:

- Fold-0021 Plate Assembly
- Fold-0018 Standard Parts
- Fold-0020 Standard Parts
- Fold-0028 Specifications
- Fold-0011 Jig & Fixtures
- Fold-0019 Utility parts
- Fold-0004 Project Specs
- Fold-0017 Specification

- The CATIA file name is displayed.

File name:

- Click **Save**.

5. A Part Profile Card is displayed in which you can add relevant information.
 - Enter a name for the Part in the **Description** attribute.
 - Select the tabs to review default information about the file.
6. Click **OK**. **CATIA - TeamPDM** automatically saves your file into the database, giving it a unique identity, where it can be easily found for later use.
7. If you want to register the document:
 - From the *SmarTeam* menu, choose **Register**. The *Register* window is displayed.
 - Fill in the fields (optional) and click **OK**.

- The CATIA window is closed and the document is registered and placed into the SmarTeam vault.
- To check the document out or copy its file to your desktop, refer to "[Checking Out and Checking In a Document](#)".



Checking Out and Checking In a Document



After a Part is registered, you must check it out in order to modify it. You may then check it back into the vault for safekeeping.

You also have the option of copying the file to your desktop. You can view the Part, but you cannot modify it. You may then check it out at a later time (Refer to "[Copying Documents To Your Project Desktop](#)" for details).



1. From the *SmarTeam* menu, run a search to locate the document (Refer to "[Where's the Right document ?](#)" for details).
2. Right-click on the document and click **Edit**. A window is displayed prompting you to check out the document. Click **OK**.
3. In the displayed *Check Out* window, fill in the fields and click **OK**. The document is then displayed at your desktop and you may modify it.
4. When you have completed your modifications, choose the **Save** option from the *SmarTeam* menu.
5. Then, check the document back into the vault for safekeeping:
 - From the *SmarTeam* menu, choose Check In.
 - Fill in the fields and click **OK**. Your document is checked in to the vault and a copy of it remains at your desktop.
6. If you wish to check out the document again, choose **Check Out** item from the *SmarTeam* menu.



Copying Documents To Your Project Desktop



Instead of checking a document out of the vault, you can copy it to your project desktop. The document is then displayed at your desktop in read-only mode.



1. Run a search to locate your document. (Refer to "[Where's the Right Document ?](#)" for details).
2. Right-click on the document and choose **Copy File**. The file is copied to your desktop in read-only mode.
3. You can then check out the document in order to modify it, as follows:
 - Select **Check Out** item from the *SmarTeam* menu.
 - Fill in the fields in the *Copy File* window (optional) and click **OK**. The desktop is refreshed and you can now modify the document.



Adding an Existing Part to CATIA - TeamPDM



You may just want to add a file that already exists in CATIA.

1. Open a Part into CATIA using the CATIA **File** menu or the folder icon on the tool bar. The Part is displayed in the **CATIA V5** window.
2. From the *SmarTeam* menu, choose **Save**.
3. A **CATIA - TeamPDM Save As** window appears. In this window you can define the project and the parent to which the document belongs.
4. A Part Profile Card is displayed in which you can add relevant information.
 - It is recommended to enter a name for the Part in the **Description** attribute.
 - Select the tabs to review default information about the file.
5. Click **OK**. **CATIA - TeamPDM** automatically saves your file into the database, giving it a unique identity where it can be easily found for later use.



Saving a Part Again



Each time you modify a Part in CATIA, you should save the Part using the **CATIA - TeamPDM Save** (or **Save As**) option:

- Click **Save**: The Part is saved in the **CATIA - TeamPDM** database, with all its modifications. Its Profile Card is not displayed at this time.
- Click **Save As**: The **CATIA - TeamPDM Save As** window is displayed. In this window, you can define hierarchical links for the Part, namely the project and parent to which the Part belongs. The Part (together with its new hierarchical links) is saved in the **CATIA - TeamPDM** database.



1. Open an existing Part in CATIA.



You can run a search to locate the Part and then launch it directly into CATIA V5, as described in "[Launching a Document](#)".

2. From the *SmarTeam* menu, choose **Save**. The Part is saved in **CATIA - TeamPDM** and its Profile Card is updated accordingly.

or

Choose **Save As**. The **CATIA - TeamPDM Save As** window is displayed.

- Choose a project. The Part will be saved as a document in the selected project.

Project:

- Choose a parent folder. The Part will be saved as a child of the selected folder.
- Click **Save**. The Part (together with its hierarchical links) is saved in **CATIA - TeamPDM** and its Profile Card is updated accordingly.



Adding a New Assembly to CATIA - TeamPDM



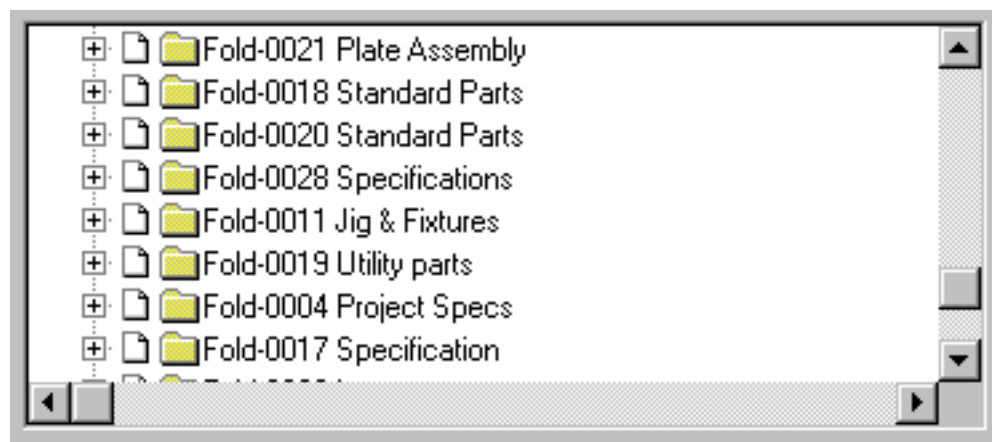
Adding an Assembly to the **CATIA - TeamPDM** database is similar to adding a Part.



1. Display the Assembly in CATIA.
2. From the *SmarTeam* menu, choose **Save**.
3. A CATIA *Save As* window appears. Name the file and click **Save**.
4. A CATIA - TeamPDM *Save As* window appears. In this window you can define the project and the parent to which the document belongs.
 - Choose a project. The Assembly will be saved as a document in the selected project.



- Choose a parent folder. The Assembly will be saved as a child of the selected folder.



- The CATIA file name is displayed.
 - Click **Save**.
5. 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.
 - Select the tabs to review default information about the file.
 6. Click **OK**. **CATIA - TeamPDM** automatically saves your Product into the database, giving it a unique identity where it can be easily found for later use.

In the **CATIA - TeamPDM** 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.

7. If the components of the Assembly have not yet been saved into **CATIA - TeamPDM**, a Profile Card for each component is displayed, one by one. In each Profile Card, name the component (in the **Description** field) and click **Add** to save the new Parts/Products into the **CATIA - TeamPDM** database.

For example: If a Product has 3 Parts as its components, and these components have not yet been saved in **CATIA - TeamPDM**, a Profile Card for each Part is displayed.



Adding a Product Using the Batch Method



The above method could be very time consuming when saving large assemblies. For this reason, **CATIA - TeamPDM** 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 **CATIA - TeamPDM** 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.



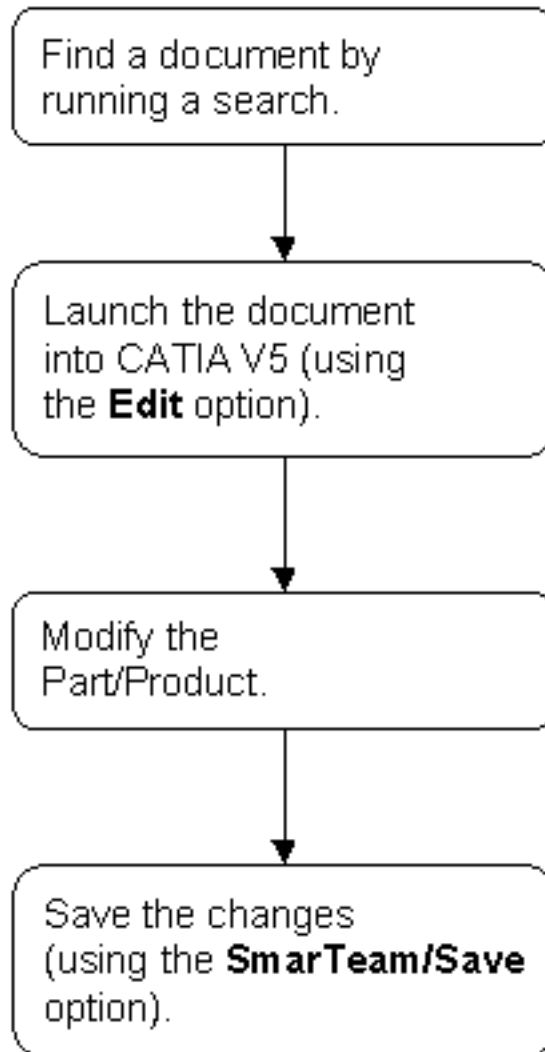
1. Display the Assembly in CATIA.
2. From the *SmarTeam* menu, choose **Batch Mode Save**. A checkmark is displayed to the left of the **Batch Mode Save** option. This action toggles between a **Batch Mode Save** and a regular **Save**.
3. From the *SmarTeam* menu, choose **Save**. The Assembly and its components are saved into the **CATIA - TeamPDM** database.



Modifying CATIA Parts and Products

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. **CATIA - TeamPDM** enables you to locate a document and launch it into CATIA quickly and easily.

The following steps illustrate how **CATIA - TeamPDM** can assist you in locating and launching documents:



In addition, **CATIA - TeamPDM** enables you to locate all the parents of any document (using the **Where Used** option). Refer to "[Where Has This Part/Product Been Used ?](#)" for details. This is particularly helpful when working with large Assemblies with many components.

Where's the Right Document?



Launching a Document



Where Has This Document Been Used ?



Up



Quick Overview




Adding Parts and Assemble



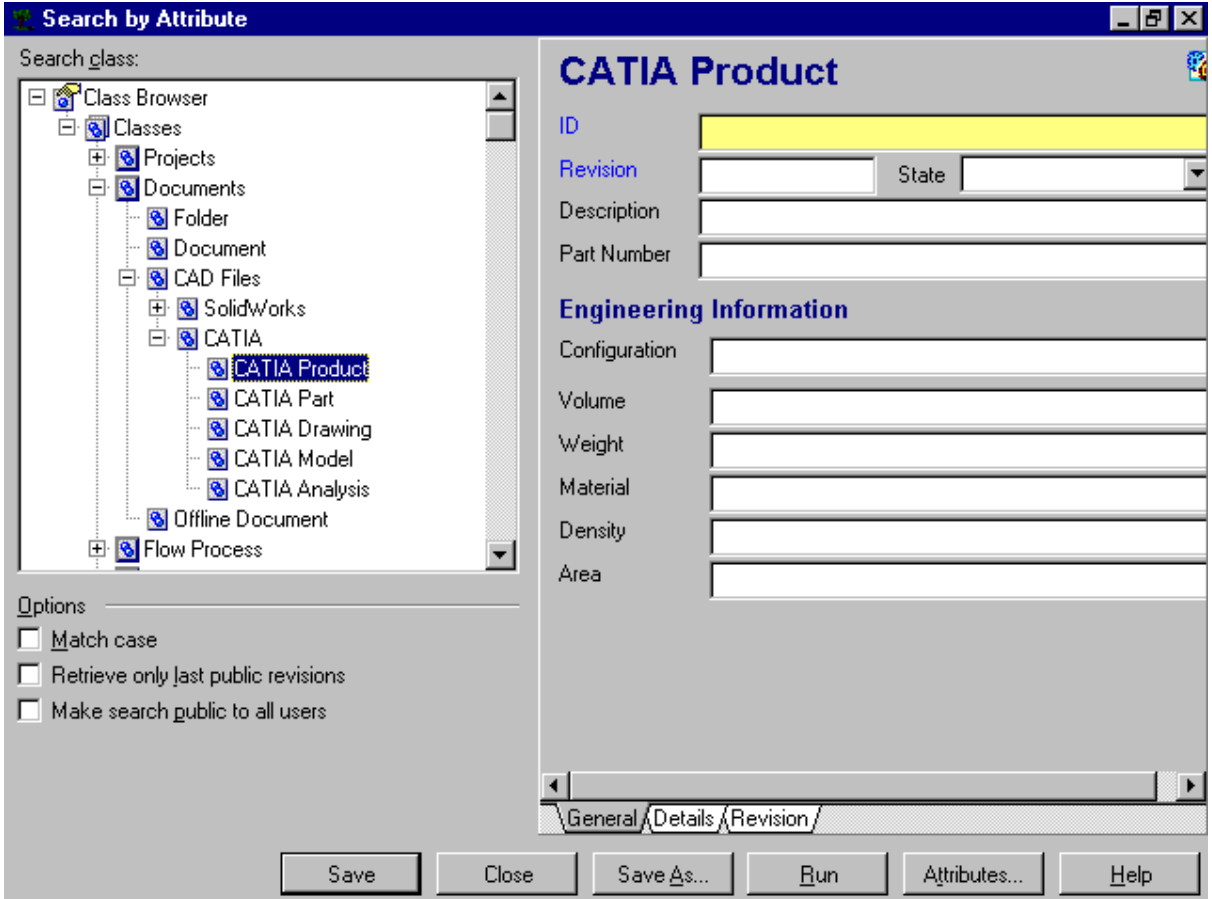
Modifying Parts and Assemk



Where's the Right Document ? Finding a Document

 **CATIA - TeamPDM** search tools enable you to find the exact Part or Product that you wish to modify.

1. From the *SmarTeam* menu, point to **Find** and choose **Find Document....**
2. The **Search Editor** window is displayed. Select one of the predefined searches or create a new search.
3. To create a new search:
 - Point to **Add** and choose **By Attribute**.
 - In the displayed *Search Details* window:
 - Choose a class of documents




- In the right side of the window, define search words in one or more attribute fields.
- Click **Save As**
- Enter a name for the search.

Search name:


4. Click **Run**. **CATIA - TeamPDM** searches for the documents that match the search criteria and displays these documents in a list.

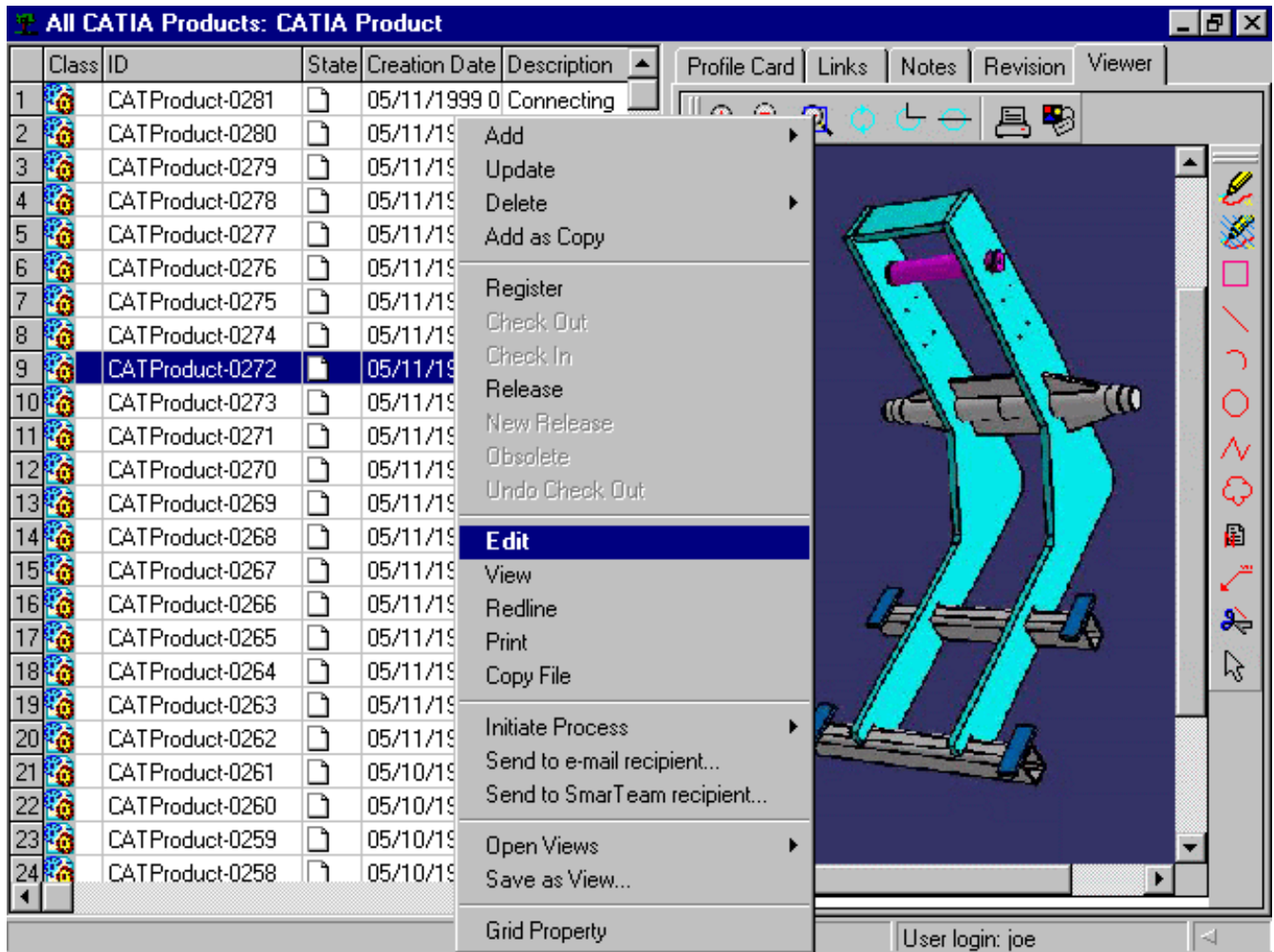


Launching a Document


 Now that you've located the documents that match the search criteria, you can browse through the list of documents. You can view the Profile Card for each document as well as a thumbnail image of each.

When you locate the exact document that you wish to modify, launch it into CATIA. Modify the Part/Product in CATIA and then save it again (using the **SmartTeam Save** option).

-  1. In the search results list, select a document to view its Profile Card.
2. Right-click on a document to display a dropdown list of options and choose **Edit**. The document is launched into CATIA.



Class	ID	State	Creation Date	Description
1	CATProduct-0281		05/11/1999	0 Connecting
2	CATProduct-0280		05/11/19	
3	CATProduct-0279		05/11/19	
4	CATProduct-0278		05/11/19	
5	CATProduct-0277		05/11/19	
6	CATProduct-0276		05/11/19	
7	CATProduct-0275		05/11/19	
8	CATProduct-0274		05/11/19	
9	CATProduct-0272		05/11/19	
10	CATProduct-0273		05/11/19	
11	CATProduct-0271		05/11/19	
12	CATProduct-0270		05/11/19	
13	CATProduct-0269		05/11/19	
14	CATProduct-0268		05/11/19	
15	CATProduct-0267		05/11/19	
16	CATProduct-0266		05/11/19	
17	CATProduct-0265		05/11/19	
18	CATProduct-0264		05/11/19	
19	CATProduct-0263		05/11/19	
20	CATProduct-0262		05/11/19	
21	CATProduct-0261		05/10/19	
22	CATProduct-0260		05/10/19	
23	CATProduct-0259		05/10/19	
24	CATProduct-0258		05/10/19	

 If the document is checked into a vault, you must check it out of the vault in order to open it in CATIA V5. A message is displayed prompting you to check out the document. Click **OK**. In the displayed *Check Out* window, click **OK** to check the document out of the vault. The document is then launched into CATIA.

3. Modify the Part/Product in CATIA.
4. From the *SmartTeam* menu, choose **Save** to update the Profile Card and save the document in the **CATIA - TeamPDM** database.




[Where's the Right Document](#)

[Launching a Document](#)


[Where Has This Document E](#)

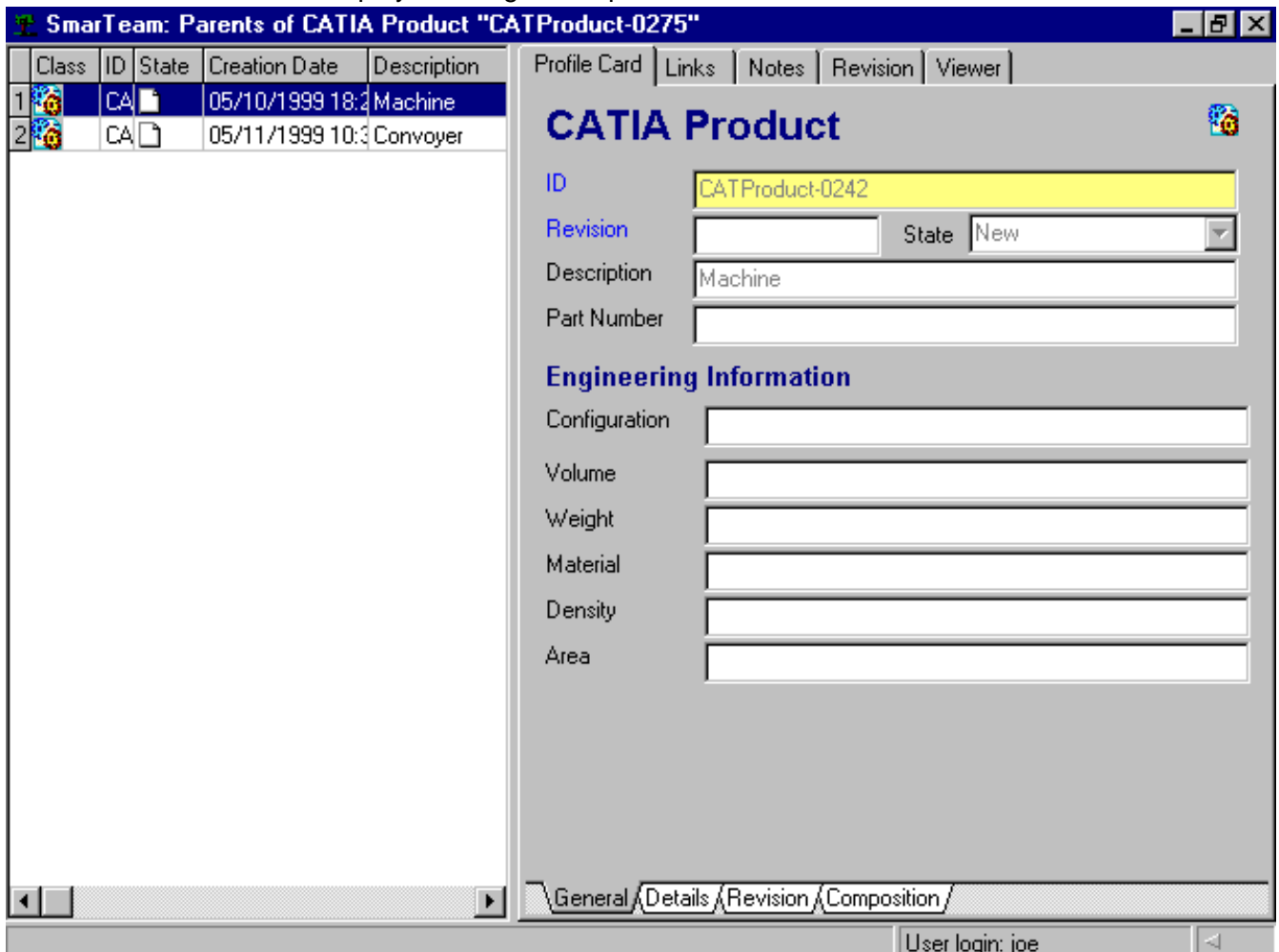


Where Has This Part/Product Been Used ?

 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. **CATIA - TeamPDM** 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.

When you choose the **Where Used** option, **CATIA - TeamPDM** displays a window which lists all the parents of the document. You can then browse through the list to view the parents (Assemblies) of the Part/Product, and you can launch any of these documents into CATIA.

- 
1. Display a Part/Product in CATIA. Remember you can launch any document using the **CATIA - TeamPDM Edit** option.
 2. Choose **Where Used** item in the *SmarTeam* menu.
A *Where Used* window is displayed listing all the parents of the Part.



	Class	ID	State	Creation Date	Description
1	CA	CATProduct-0242	New	05/10/1999 18:3	Machine
2	CA	CATProduct-0242	New	05/11/1999 10:3	Convoyer

3. You can browse through the list to view the Profile Card of each document. You can also right-click on a document and choose **Edit** to launch the document into CATIA.



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[Where's the Right Document](#)

[Launching a Document](#)

[Where Has This Document E](#)



Basic Tasks

Theme

Working with Projects



Saving Documents



Data Structure



Finding and Working With Your Documents



Securing Documents in a Vault



Document Associations and Dependencies




Working with Projects

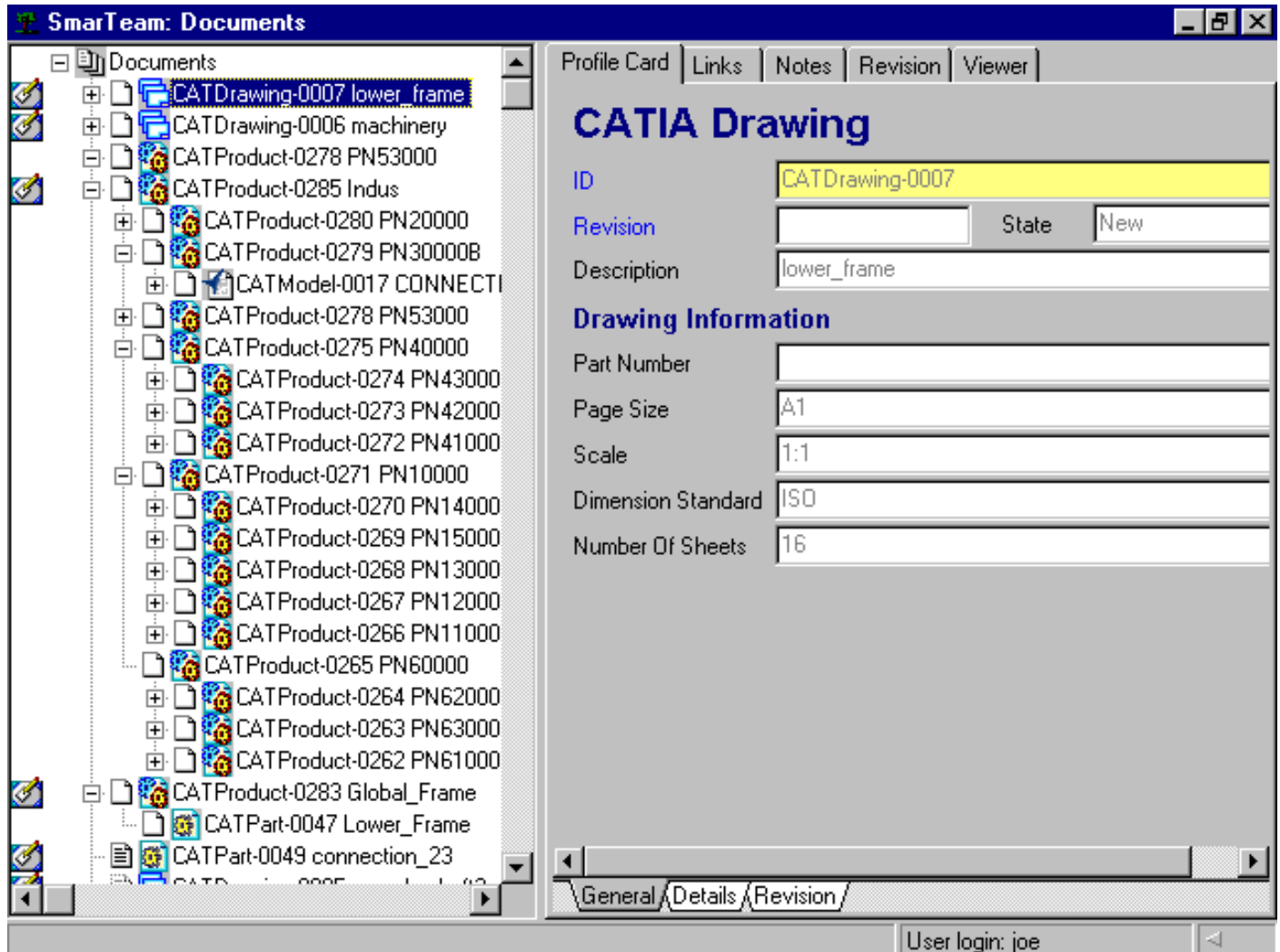
The **CATIA - TeamPDM** data structure organizes all data around projects in order to assist you to browse through the data and locate specific Parts, Drawings or Products. Projects are the highest class in the **CATIA - TeamPDM** data structure and are used to represent a product idea or concept. Each project has its own Documents tree browser which displays all the documents associated with the project in a hierarchical structure, including the CATIA documents and related documentation (such as spreadsheets and Word files).



Organizing Projects

 In general, projects are organized into folders to organize the documentation. When you begin a project, you create folders and link these folders directly to the project. When you save CATIA Parts and Products, you can save these documents into specific folders.

The **CATIA - TeamPDM** Documents tree displays a visual representation of the hierarchy, as shown below:



In the example above, the folders are the highest level in the hierarchy, directly linked to a project. The CATIA documents and associated documentation are linked as children of the folders. Each document is represented by an icon.

As you save Parts, Products and Drawings into the **CATIA - TeamPDM** database, you can save each document as a child of a specific folder. In this manner, all your documents are organized into folders and all the folders are linked to a project.



[Up](#)




[Organizing Projects](#)



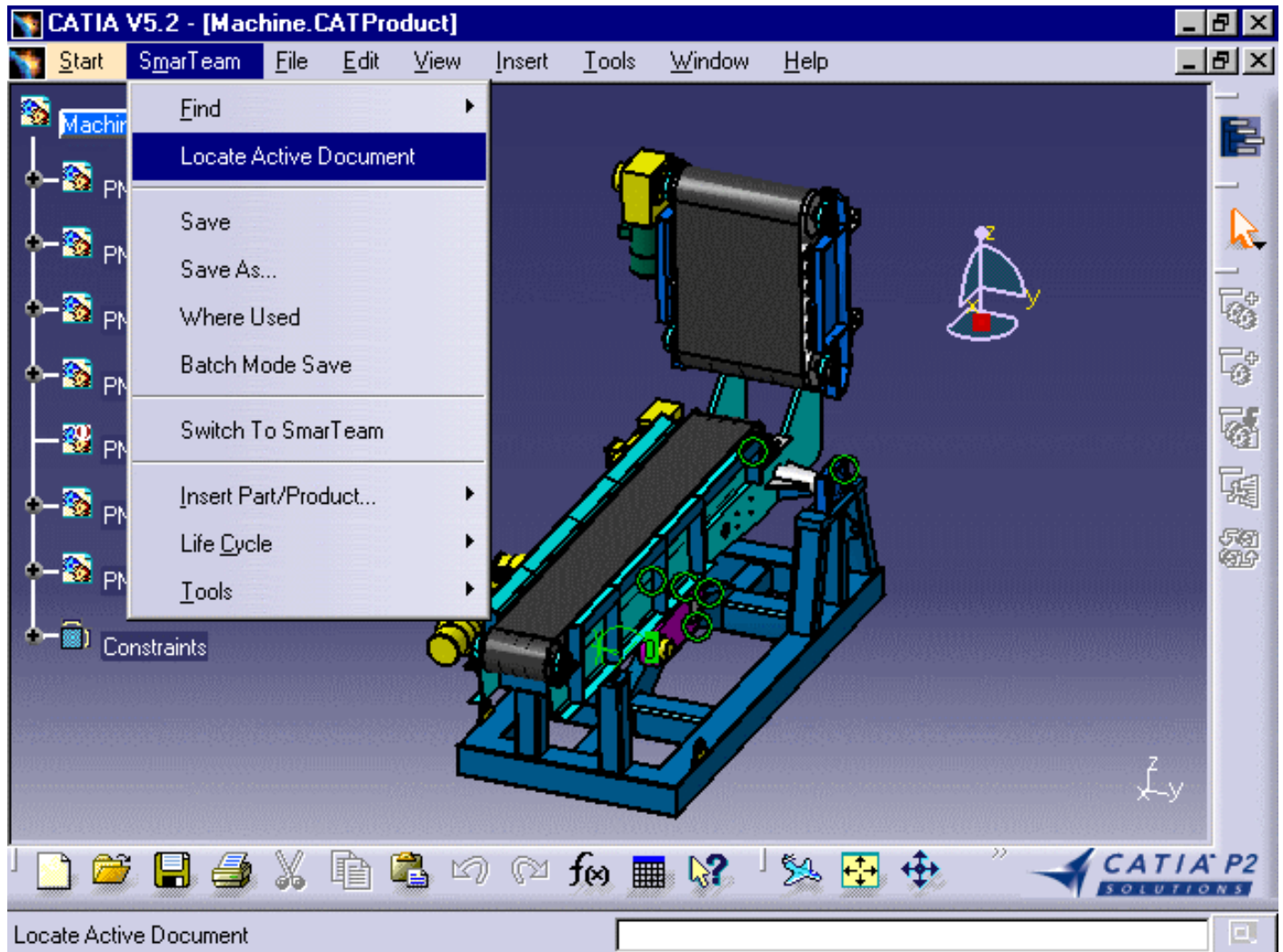
[Viewing Documents in a Pro](#)

Viewing Documents in a Project

 **CATIA - TeamPDM** enables you to save documents into a project desktop, thereby linking these documents to the selected project.

Note: When you choose the **SmarTeam Save** option (or **Save As** option) check the **Link to project desktop** checkbox in the *Object Attributes* window.

When you choose the **Locate Active Document** option from the *SmarTeam* menu, the document is displayed in a *Data Manager* window.



The **Project Desktop** icon displayed to the left of the document name shows you that the document is linked to a project, as shown below.

You can then view the projects to which the document is linked by clicking the Projects tab in the Links page of the displayed Profile Card.

SmarTeam: Documents [- [+ [X]

Documents
+ CATProduct-0242 Machine

Profile Card | Links | Notes | Revision | Viewer

	Class	Project ID	State	Creation Date	Description	Created by ...
1		Project-0015		05/10/1999 18:09	Convoyer	joe

Projects / Documents / Flow Process / Materials / Items / Contacts / Users

User login: joe



[Up](#)



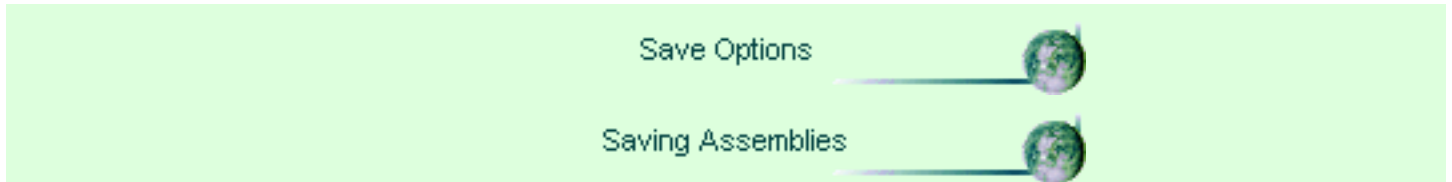
[Organizing Projects](#)



[Viewing Documents in a Pro](#)

Saving Documents

Each time you create a new CATIA Part or Product, the document should be saved into the **CATIA - TeamPDM** database. You can then use the many powerful tools offered by **CATIA - TeamPDM** to properly manage your documents.



CATIA - TeamPDM Save Options



The **CATIA - TeamPDM** integrated menu provides two methods for saving documents:

- **Save**: Saves the document into the **CATIA - TeamPDM** database.
- **Save As**: Saves the document into the **CATIA - TeamPDM** database. **and** Defines the project and the parent folder of the document. For example, you can save a new Part as a child of the **Standard Parts** folder in the **Convoyer** project.

After you save a new Part or Product:


- A **CATIA - TeamPDM** window is automatically displayed showing the new document that has been added to the **CATIA - TeamPDM** database.



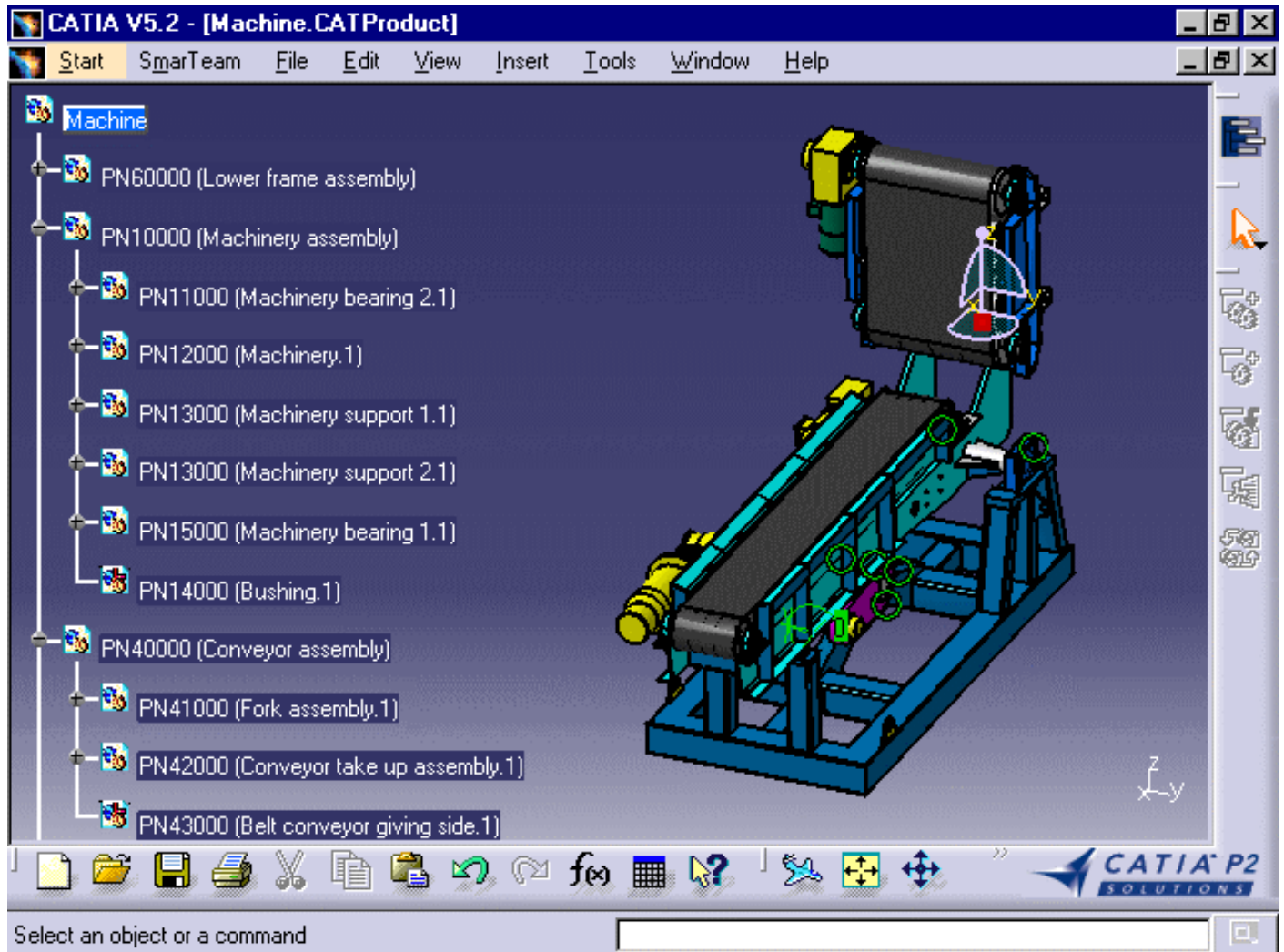
You do not need to fill in the description field in the Profile Card : by default, **CATIA - TeamPDM** will use the current name of the document.



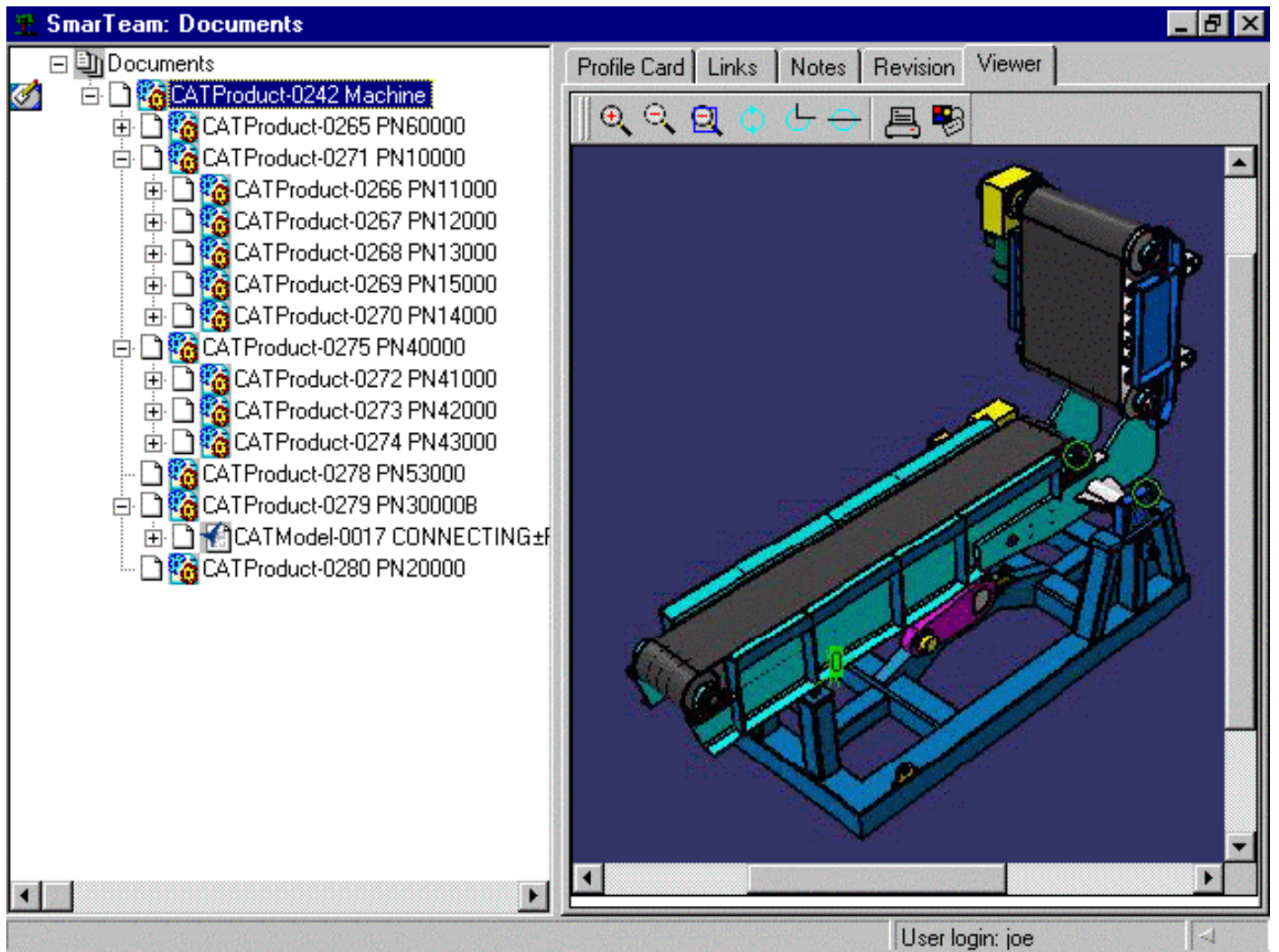
Saving Assemblies

 When you save an Assembly into the **CATIA - TeamPDM** database, its components are automatically saved as children of the parent Product. The tree hierarchy changes to reflect the new documents and the hierarchical links between them. Thus, the inherent relationship between a Product and its components is automatically maintained in **CATIA - TeamPDM**.

A Products structure within CATIA V5...



... and the corresponding documents tree within **CATIA - TeamPDM**



Up



Save Options



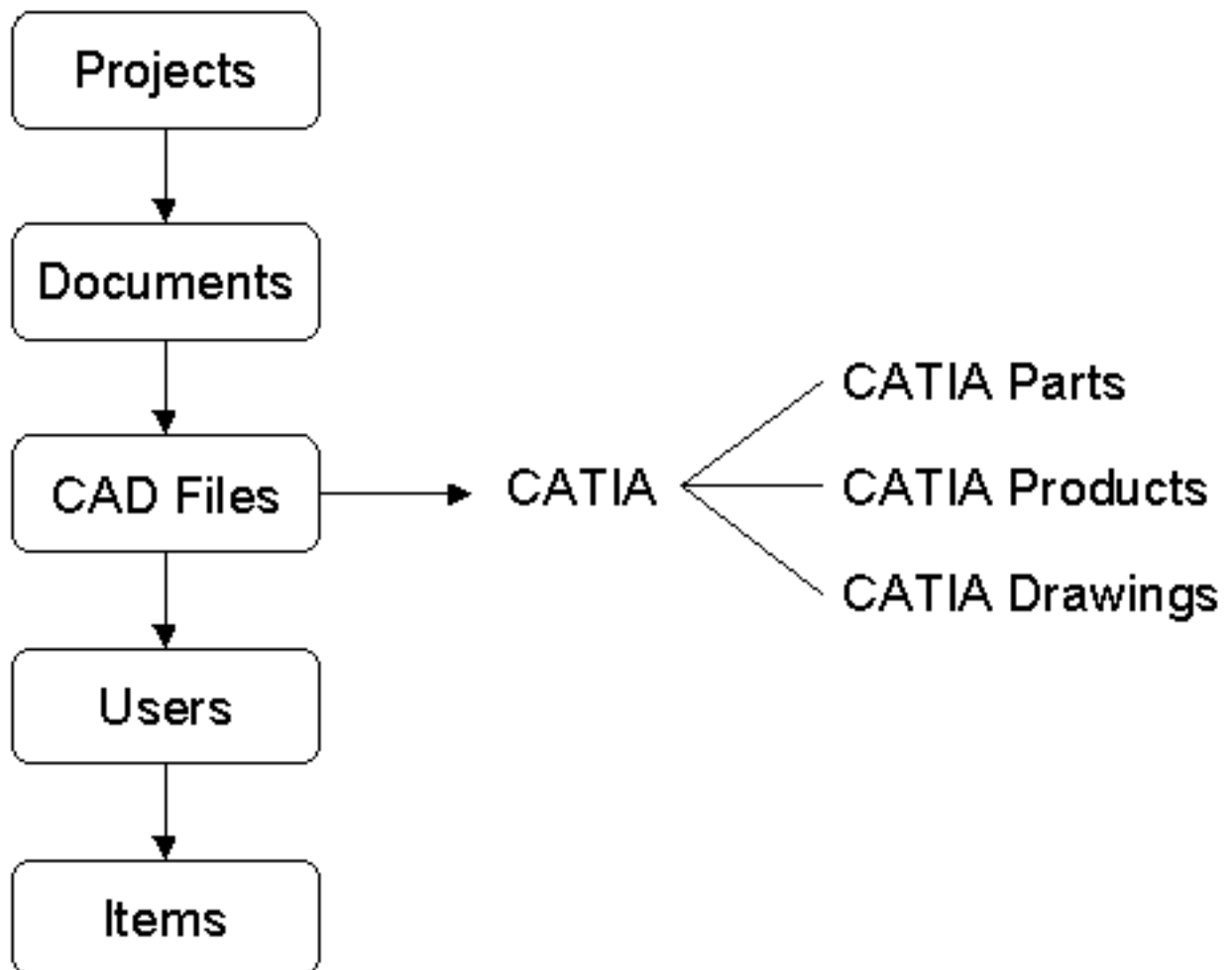
Saving Assemblies

Data Structure

CATIA - TeamPDM organizes all documents into classes. Each time you save a CATIA document, it is saved into a specific class. Each class of documents has its own Profile Card to accurately represent the attributes of the class.

For example, when you save a CATIA drawing, it is saved into the **CATIA Drawing Class**. The Profile Card for the **Drawing Class** displays attributes of importance to drawings.

CATIA - TeamPDM provides a fixed database structure:



Note: In addition to the CAD files class, some of the classes displayed above may have sub-classes.

By default, when you save CATIA documents, they are saved into the following classes:

- When you save a new CATIA Part, the Part will be saved into the **CATIA Part Class**.
- When you save a new CATIA Drawing, the Drawing will be saved into the **CATIA Drawing Class**.
- When you save a new CATIA Product, the Product is saved into the **CATIA Product Class**.

Working with Classes



Up

Data Structure



Working with Projects

Finding and Working With Yc



Document Associations and



Saving Documents



Securing Documents in a Ve

Working with Classes



Your administrator may modify the class structure to suit the needs of your organization, as follows:

- Your administrator can create additional classes (along with appropriate Profile Cards). For example, your administrator can create a class called **Beta Parts**.
- When you save a document, you can choose a class from a dropdown list of classes displayed in the Profile Card. A new Profile Card for the selected class is displayed, and the document is saved into the appropriate class.

The following scenario illustrates how you can work with several classes:

1. Your administrator defines a new class called Beta Parts.
2. You then create a new Part in CATIA V5 and save it by clicking the **SmarTeam Save** option.
3. By default, the CATIA Part Profile Card is displayed. In this Profile Card, you can select a different class.
4. If you select the **Beta Parts** class, the window refreshes to display a Beta Parts Profile Card. When you click **OK**, the new Part is saved into the **Beta Parts** class.

Each time you create a search, you must select a **CATIA - TeamPDM** class. When you run the search, **CATIA - TeamPDM** locates the documents that meet the search criteria from the specified **CATIA - TeamPDM** class.

For example: You create a search from the **CATIA Parts** class, and you define search criteria for the search. When you run the search, **CATIA - TeamPDM** will locate all the CATIA Parts which meet the specified search criteria.

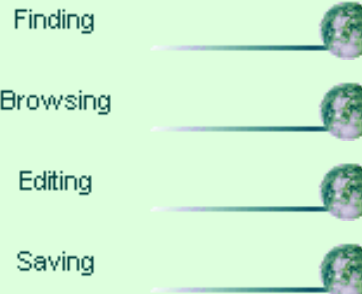


Finding and Working With Your Documents

Locating Parts, Products and Drawings is an essential task, but it can be time-consuming when creating complex Assemblies. **CATIA - TeamPDM** provides a number of powerful functions which enable you to locate a document in the **CATIA - TeamPDM** data structure.

The following **CATIA - TeamPDM** 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** Browse 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** page.
- View** When you locate the exact document that you were searching for, you can launch it directly into CATIA V5, for viewing and inspection.
- Edit** When you locate the exact document that you were searching for, you can launch it directly into CATIA V5, and modify it accordingly.
- Save** When you finish your modifications, save the document. The Profile Card (and appropriate revision history) is updated accordingly.





Up



Working with Projects



Saving Documents



Data Structure



Finding and Working With Yc



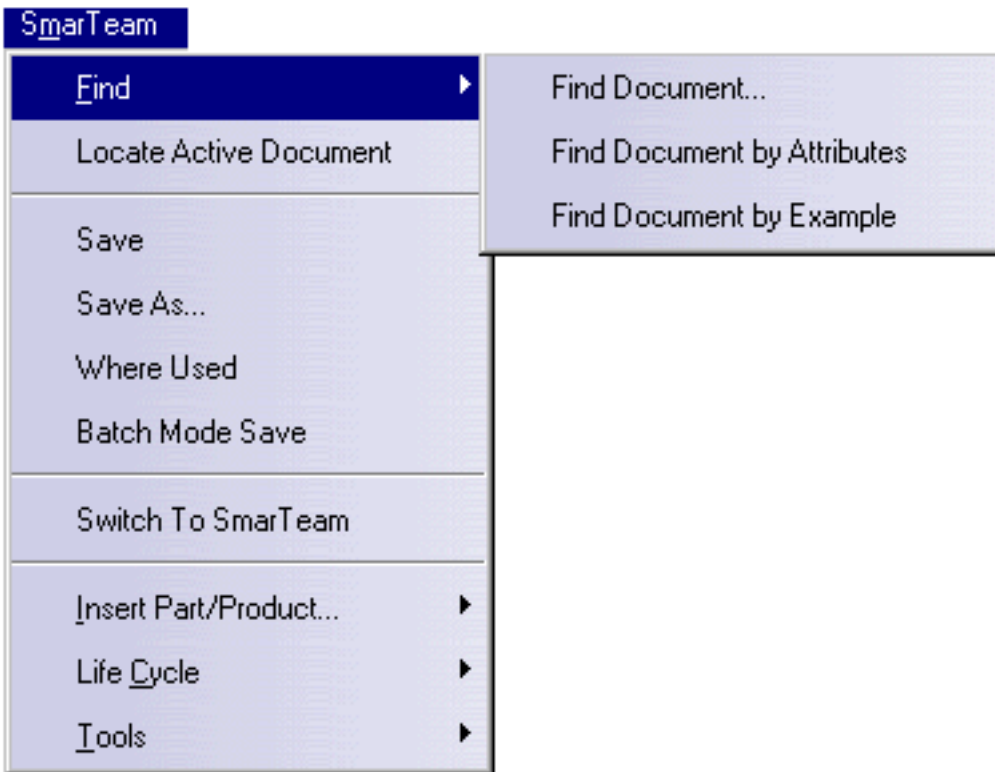
Securing Documents in a Vc Document Associations and



Finding Your Documents



CATIA - TeamPDM provides several searching tools:




- **Find Object:** 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.

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.

- **Find Object By Attribute** option: This option enables you to create a new search. You must choose a **CATIA - TeamPDM** class and enter search criteria for it. You can then run the search to locate the documents in the selected class that match the search criteria.
- **Find Object By Example** option: This option enables you to create an advanced search based on numerous search attributes. You can select two **CATIA - TeamPDM** classes and define search criteria based on the class attributes (and their link attributes). You can then run the search to locate the documents in the selected classes that match the search criteria.

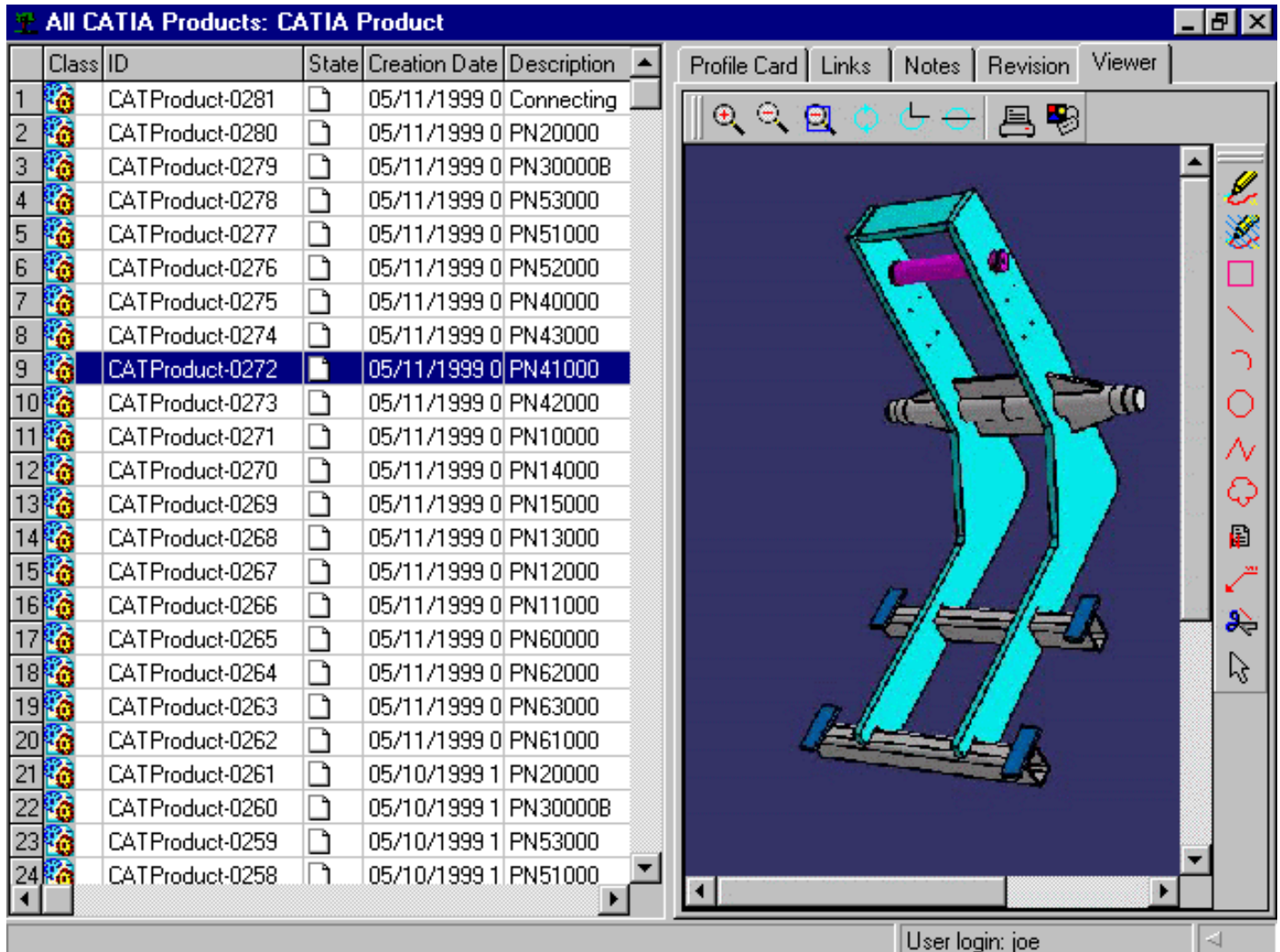


Browsing

 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.

CATIA - TeamPDM 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 it is displayed in CATIA V5.

Browsing through the *Viewer* page provides a means of searching and identifying a specific document. For example, after you run 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.




The screenshot displays the CATIA TeamPDM interface. On the left, a table titled "All CATIA Products: CATIA Product" lists search results. The table has columns for Class, ID, State, Creation Date, and Description. Row 9 is highlighted, showing CATProduct-0272 with a creation date of 05/11/1999 0 and description PN41000. On the right, the "Viewer" tab is active, showing a 3D model of a mechanical assembly in a cyan color. The interface includes a toolbar with various icons and a status bar at the bottom indicating "User login: joe".

Class	ID	State	Creation Date	Description
1	CATProduct-0281		05/11/1999 0	Connecting
2	CATProduct-0280		05/11/1999 0	PN20000
3	CATProduct-0279		05/11/1999 0	PN30000B
4	CATProduct-0278		05/11/1999 0	PN53000
5	CATProduct-0277		05/11/1999 0	PN51000
6	CATProduct-0276		05/11/1999 0	PN52000
7	CATProduct-0275		05/11/1999 0	PN40000
8	CATProduct-0274		05/11/1999 0	PN43000
9	CATProduct-0272		05/11/1999 0	PN41000
10	CATProduct-0273		05/11/1999 0	PN42000
11	CATProduct-0271		05/11/1999 0	PN10000
12	CATProduct-0270		05/11/1999 0	PN14000
13	CATProduct-0269		05/11/1999 0	PN15000
14	CATProduct-0268		05/11/1999 0	PN13000
15	CATProduct-0267		05/11/1999 0	PN12000
16	CATProduct-0266		05/11/1999 0	PN11000
17	CATProduct-0265		05/11/1999 0	PN60000
18	CATProduct-0264		05/11/1999 0	PN62000
19	CATProduct-0263		05/11/1999 0	PN63000
20	CATProduct-0262		05/11/1999 0	PN61000
21	CATProduct-0261		05/10/1999 1	PN20000
22	CATProduct-0260		05/10/1999 1	PN30000B
23	CATProduct-0259		05/10/1999 1	PN53000
24	CATProduct-0258		05/10/1999 1	PN51000



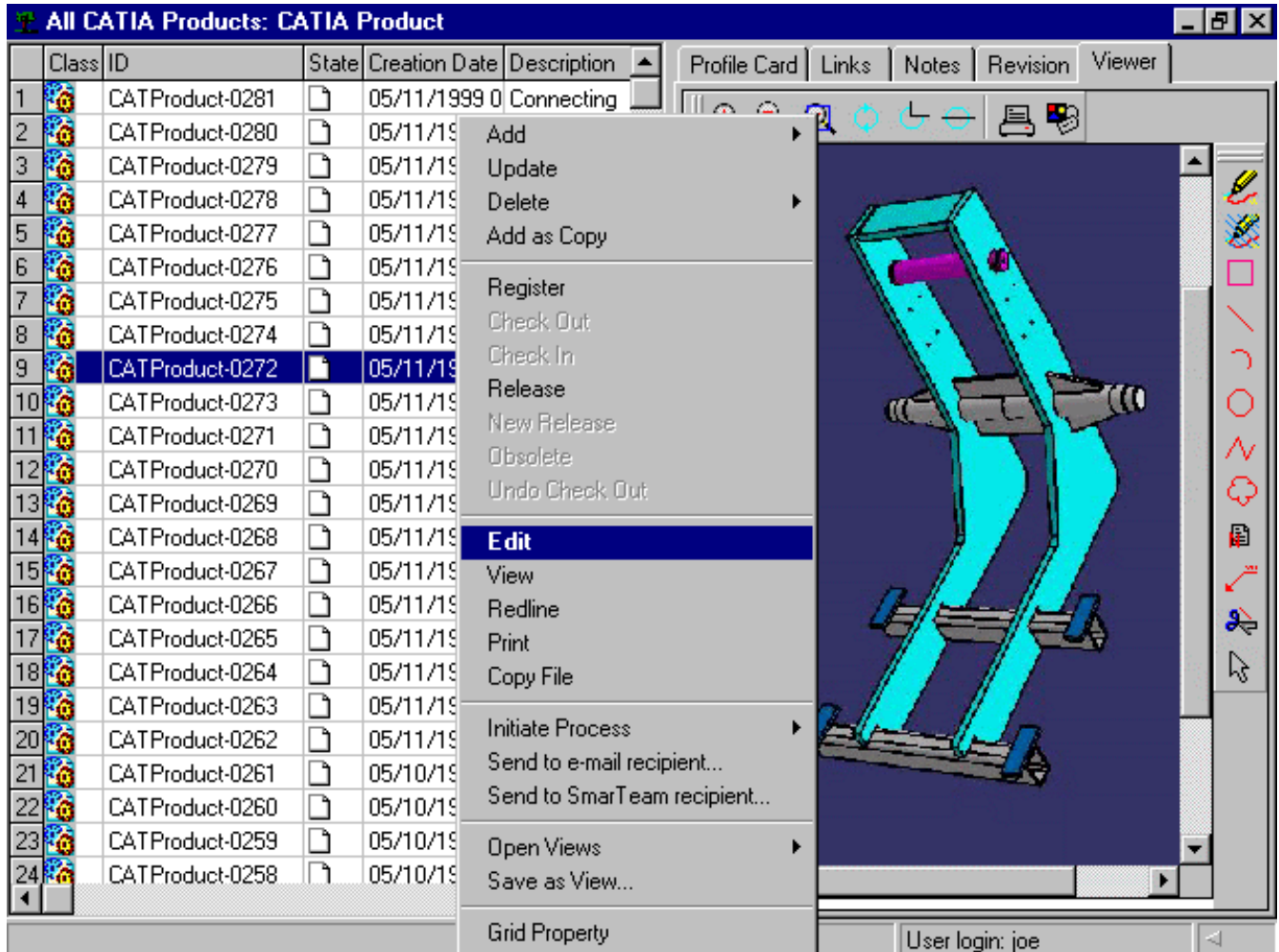
Editing

 After you locate a document, you can quickly launch it into CATIA V5, as described below:

To launch a document:

1. From any **CATIA - TeamPDM** window, select a document.
2. Right-click to display a drop-down menu and choose **Edit**.

Shortcut: Double-click on the document to launch it into CATIA V5..



The screenshot shows the 'All CATIA Products: CATIA Product' window. The table below represents the data shown in the interface:

Class	ID	State	Creation Date	Description
1	CATProduct-0281	📄	05/11/1999	0 Connecting
2	CATProduct-0280	📄	05/11/19	
3	CATProduct-0279	📄	05/11/19	
4	CATProduct-0278	📄	05/11/19	
5	CATProduct-0277	📄	05/11/19	
6	CATProduct-0276	📄	05/11/19	
7	CATProduct-0275	📄	05/11/19	
8	CATProduct-0274	📄	05/11/19	
9	CATProduct-0272	📄	05/11/19	
10	CATProduct-0273	📄	05/11/19	
11	CATProduct-0271	📄	05/11/19	
12	CATProduct-0270	📄	05/11/19	
13	CATProduct-0269	📄	05/11/19	
14	CATProduct-0268	📄	05/11/19	
15	CATProduct-0267	📄	05/11/19	
16	CATProduct-0266	📄	05/11/19	
17	CATProduct-0265	📄	05/11/19	
18	CATProduct-0264	📄	05/11/19	
19	CATProduct-0263	📄	05/11/19	
20	CATProduct-0262	📄	05/11/19	
21	CATProduct-0261	📄	05/10/19	
22	CATProduct-0260	📄	05/10/19	
23	CATProduct-0259	📄	05/10/19	
24	CATProduct-0258	📄	05/10/19	

The context menu for 'CATProduct-0272' includes the following options: Add, Update, Delete, Add as Copy, Register, Check Out, Check In, Release, New Release, Obsolete, Undo Check Out, **Edit**, View, Redline, Print, Copy File, Initiate Process, Send to e-mail recipient..., Send to SmartTeam recipient..., Open Views, Save as View..., and Grid Property. The 'Edit' option is highlighted in blue. The background shows a 3D model of a mechanical part.

- If the document is checked into the vault, you must check it out to launch it in CATIA V5.
- If the document is not saved in the vault, it is immediately launched into CATIA V5.



Saving



Make your modifications in CATIA V5 and choose one of the **Save** options from the *SmarTeam* menu. The document is saved into the **CATIA - TeamPDM** database, and the Profile Card is updated accordingly.



Safekeeping: Securing Documents in a Vault

CATIA - TeamPDM enables you to maintain and manage any information related to a document throughout its life cycle. By mirroring the physical process of document management, **CATIA - TeamPDM** 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.


Maintaining security and control



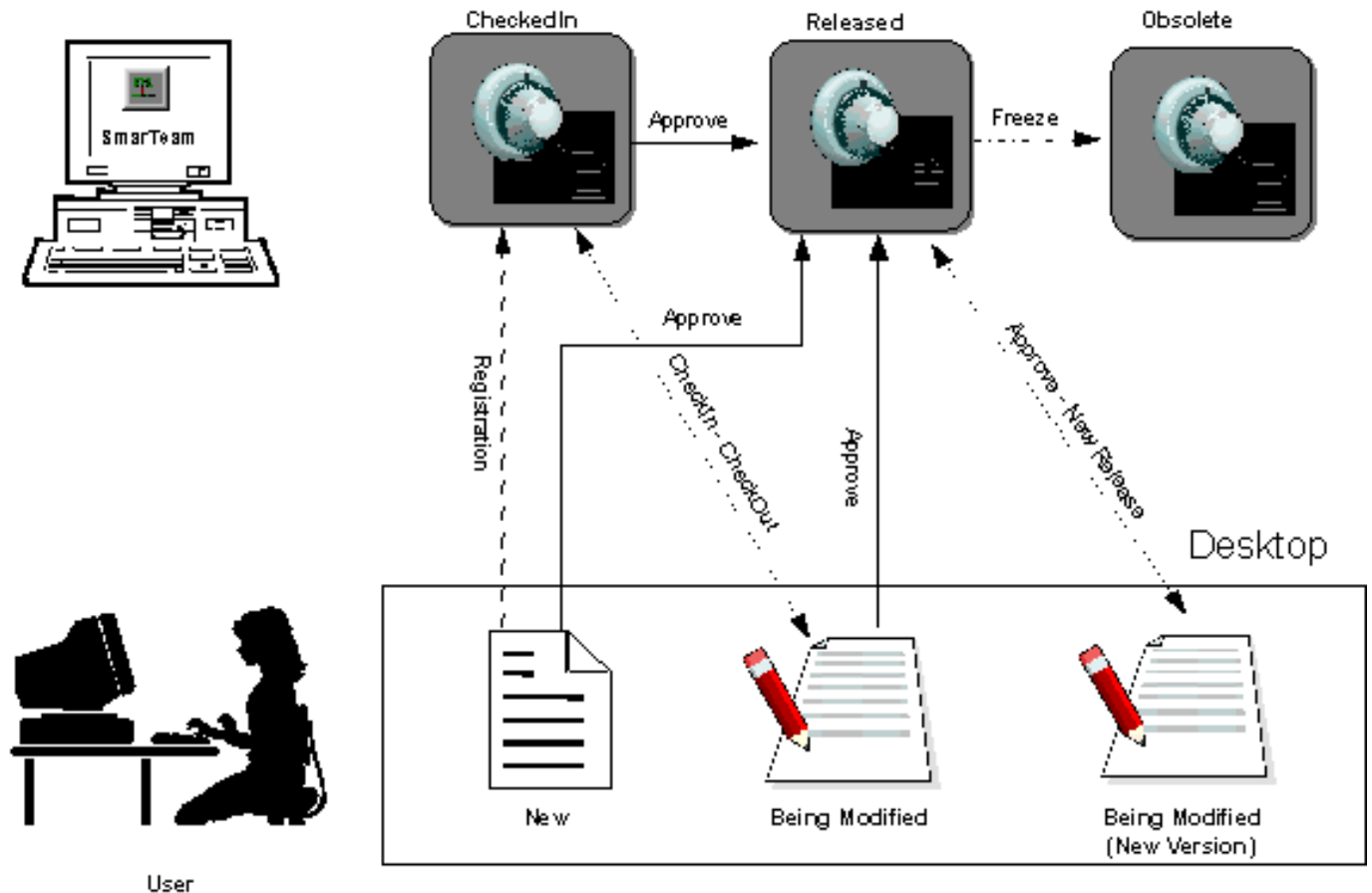
Life Cycle Menu Options



Maintaining security and control over documents

 Maintaining security and control over documents is of the utmost importance and **CATIA - TeamPDM** provides an electronic vault for this purpose. The electronic vault ensures that only those persons with access permission may access a document, and that a document cannot be accessed by more than one person at a time.

The illustration below displays the life cycle flow of a document:



CATIA - TeamPDM revision management features enables you to:

- Automate the flow of documents through different stages of development.
- Control and access the many revisions of a document.
- Save and restrict access to documents through the use of an electronic vault.

As you revise a Part, Product or Drawing, you can:

- Check out the document from the vault (using the **Check Out** option) in order to modify it and launch it in **CATIA - TeamPDM**. When a document is checked out of the vault, it can only be modified by the current user.
- You can also copy a file to your desktop in order to view it. In this case, you cannot modify the document, and the document remains in the vault. This option is helpful when another user is modifying a document, but you wish to view it in CATIA V5 at your desktop.

- Check in the document to the vault for safekeeping (using the **Check In** option). When a document is checked into the vault, it is made publicly available.
- Release the document by placing the document into the *Released* vault (using the **Release** option). The document can then be checked out for modification, (using the **New Release** option) but only as a new revision.

CATIA - TeamPDM ensures that the integrity of the Product and its children is not jeopardized by disabling life cycle operations. The following examples show how **CATIA - TeamPDM** protects the integrity of a Product.

- When you **Release** a Product, its children must be released as well. For example, if a document has 10 children, each of these child documents must be assigned the *Released* status.
- You can only move a sub-Product to the *Obsolete* vault if its parent Product is also Obsolete. For example, if a Cog is a sub-Product of a Watch, you can only move the Cog to the *Obsolete* vault if the Watch (and all its parents) are also obsolete.
- You can perform a revision operation on a Product and all its children simultaneously, using the **Propagate Operation** option. For example, if you wish to check out a Watch and all its 10 sub-Products, you can check them all out simultaneously.
- You have the option to perform a revision operation on a Product and not on its children, or perform a revision operation on a child and not on the parent Product.
- You can check out a Watch Product from the vault and leave the children in the vault.
- You can copy the children of a Product 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 Product is *Being Modified*.
- You can check a sub-Product out of the vault independently and leave the parent Product in the vault.



Up



Maintaining security and cor

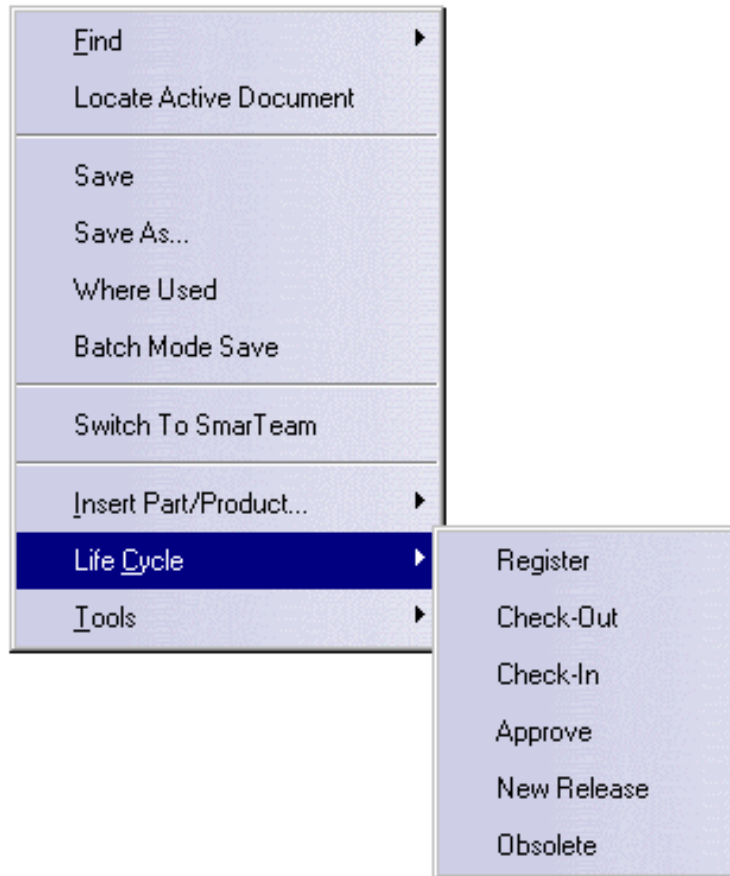


Life Cycle Menu Options

Life Cycle Menu Options



The *Life Cycle* menu contains the life cycle options which enable you to manage a Part, Product and Drawing as new revisions are created. From the *Life Cycle* menu, choose the appropriate operation in accordance with its status in the product life cycle.



All life cycle operations are controlled and managed by **CATIA - TeamPDM**, 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 into the **CATIA - TeamPDM** database, it has the *New* status. The document is not placed into a vault. When you perform the **Register** 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.

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 Option Selected	Resultant Status	Description
<p>Click Register to check a <i>new</i> document into the vault.</p> <p>or</p> <p>Click Check In to place a document that is <i>being modified</i> back in the vault.</p>	<p><i>Checked In</i></p>	<p>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.</p>
<p>Click Check Out to check out a document from the vault.</p> <p>or</p> <p>Click New Release to make a new copy of a <i>Released</i> document that was placed in the <i>Released</i> vault. The resultant document is a new revision of the source document.</p>	<p><i>Being Modified</i></p>	<p>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 <i>Checked In</i> or <i>Released</i>.</p>
<p>Click Release to transfer a document to the <i>Released</i> status.</p>	<p><i>Released</i></p>	<p>After a stage of development is completed, the document is saved in the vault of released documents.</p>

Click **Obsolete** to transfer a document to the *Obsolete* status.

Obsolete

The document can be accessed by any **CATIA - TeamPDM** user but cannot be modified. The document is saved in the vault for inactive documents.



[Up](#)



[Maintaining security and cor](#)



[Life Cycle Menu Options](#)

Document Associations and Dependencies

Dependencies/Reverse Dependencies



Revising Associated Objects



Up



Working with Projects



Saving Documents



Data Structure



Finding and Working With Yc Securing Documents in a Va



Document Associations and



Dependencies/Reverse Dependencies



CATIA V5 enables you to create a Drawing based on a Part or a Product. When you save the Drawing into the **CATIA - TeamPDM** 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*. **CATIA - TeamPDM** creates the following relationship between the two documents:

- **Dependency:** The Part/Product is a dependency of the Drawing.
- **Reverse Dependency:** The Drawing is a reverse dependency of the Part/Product.

For example: In **CATIA V5**, you create a Drawing named **Circuit Drawing** based on a Part named **Metallic Circuit**. **CATIA - TeamPDM** creates a link between these two documents as follows:

- The **Metallic Circuit** Part is a *Dependency* of the **Circuit Drawing** (since the Drawing is dependent on the Part).
- The **Circuit Drawing** is a *Reverse Dependency* of the **Metallic Circuit** Part.



Revising Associated Objects



As you revise your documents, **CATIA - TeamPDM** protects the relationship between *Associated Objects*:

- A Drawing and its associated Part/Product.

When you perform a life cycle 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:

- Dependencies are displayed in red.
- Reverse dependencies are displayed in blue.

CATIA - TeamPDM 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.



Up



Dependencies/Reverse Dep



Revising Associated Object:

Advanced Tasks

Managing Parts



Managing Assemblies



Managing Drawings



Searching for Documents



Creating and Viewing Links



Managing Parts

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

After you save the Part, check the Part into the *SmarTeam* vault by choosing the **Life Cycle/Register** option. The Part is placed into the *Checked In* vault.

In order to modify this Part in CATIA V5, you must launch the Part into CATIA V5 (by double-clicking on the Part name or choosing the **Edit** option). 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 the **Life Cycle/Check In** option).

In this manner, **CATIA - TeamPDM** manages and protects all revisions of a Part.

This section provides the following instructions:

Saving a Part



Managing the Revisions of a Part



Up



Managing Parts



Managing Assemblies



Managing Drawings



Searching for Documents



Creating and Viewing Links



Saving a Part



Every CATIA Part should be saved into the **CATIA - TeamPDM** database. After you create a new Part or modify an existing Part, choose one of the **SmartTeam Save** options.

The **CATIA - TeamPDM** integrated menu provides two methods for saving documents:

- **Save:** Saves the document into the **CATIA - TeamPDM** database.
- **Save As:** Saves the document into the **CATIA - TeamPDM** 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 **SmartTeam Save** option, a CATIA Part Profile Card is displayed in the *Object Attributes* window, as shown below. After you fill in the attribute fields and click **OK**, the Part is saved to the **CATIA - TeamPDM** database.



Your administrator may customize the Profile Card. This window will reflect the appearance of the Profile Card used in your **CATIA - TeamPDM** application.

If you choose to work in **Batch Mode Save** mode (by checking the **Batch Mode Save** option from the *SmartTeam* menu), a new Profile Card is not displayed. Instead, the Part is saved to the **CATIA - TeamPDM** 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:

- ID number: This number is assigned by **CATIA - TeamPDM** 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.

The screenshot shows a dialog box titled "Object Attributes" with a blue header bar. At the top, there is a "Select class:" label followed by a dropdown menu containing "CATIA Part". Below this, the main area is titled "CATIA Part" in large blue font. There are three input fields: "ID" with the value "CATPart-0079" (highlighted in yellow), "Revision" (empty), and "Description" (empty). To the right of the "Revision" field is a "State" dropdown menu. On the right side of the dialog, there are three buttons: "OK", "Cancel", and "Help". At the bottom, there is a tabbed interface with three tabs: "General", "Details", and "Revision", with "Revision" currently selected.

CATIA - TeamPDM enables you to define the following information in the Profile Card:

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

Saving a Part for the First Time

This process describes how to save a Part to the **CATIA - TeamPDM** database for the first time. Subsequent saves are

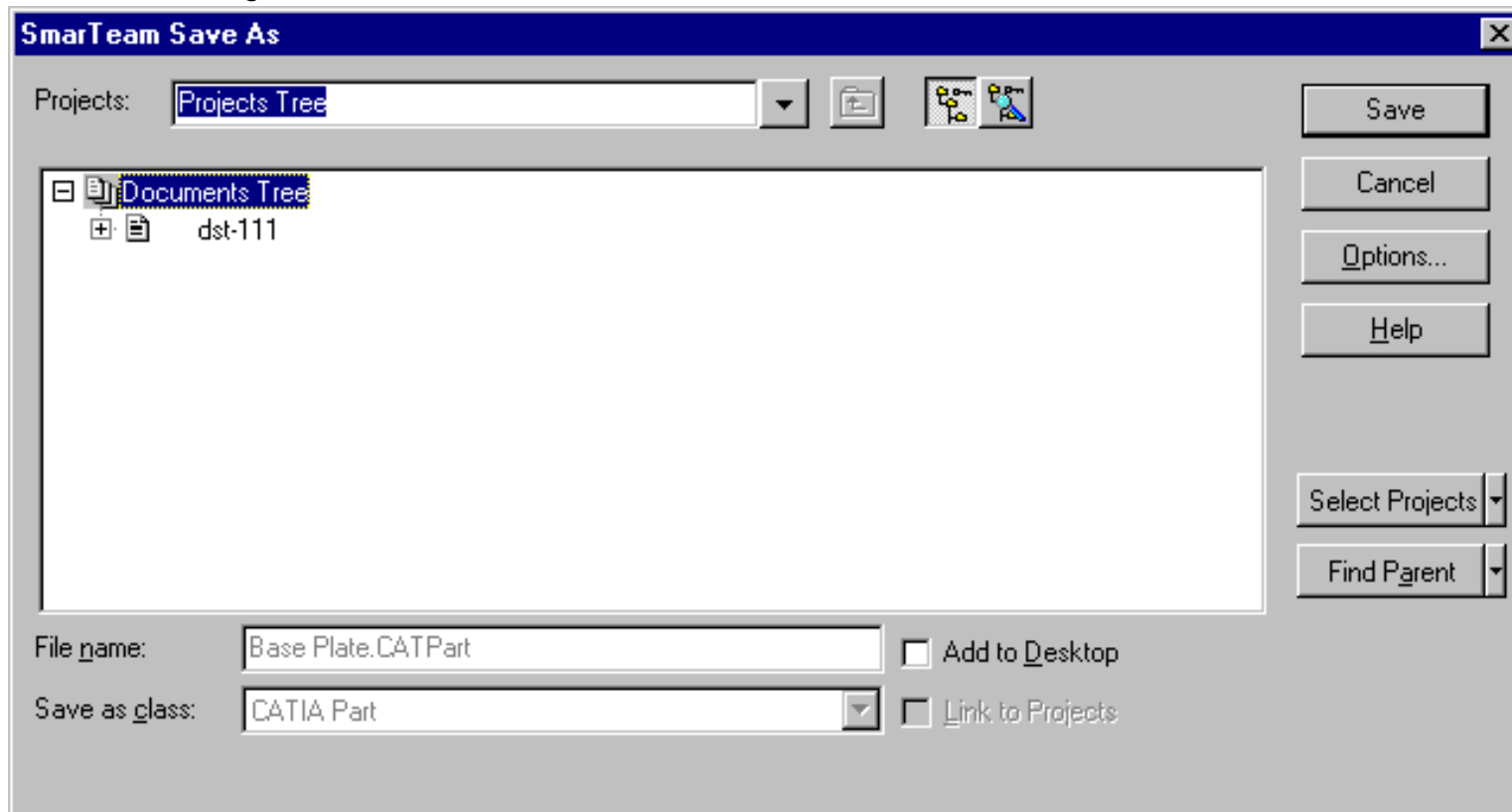
described below.

To save a Part to the CATIA - TeamPDM database:

1. In CATIA V5, create or open a Part.
2. From the *SmarTeam* menu, choose **Save**.
3. The CATIA V5 Save As window is displayed. Enter CATIA file information and click **Save**.
the CATIA V5 Save As window is not displayed if the Part has already a file name.



4. A *SmarTeam Save As* window appears. In this window you can define the project and the parent folder to which the document belongs.

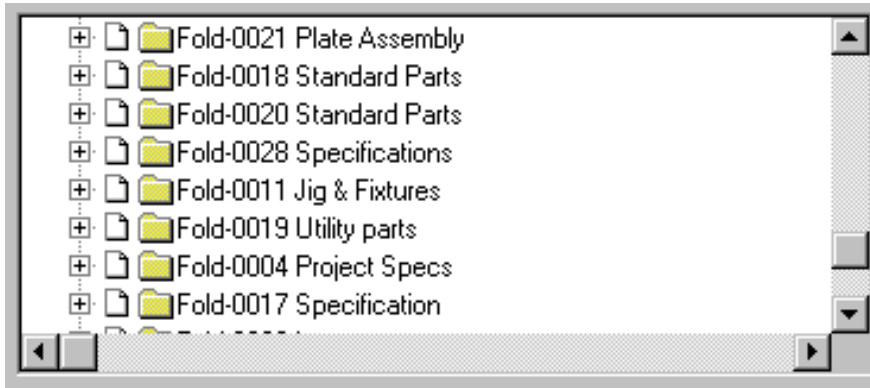


You can define the level of sub-branches displayed in the project selection tree and/or the object selection tree.

- Choose a project from the Project Tree. The Part will be saved as a document in the selected project.



- Choose a parent folder from the Object Tree. The Part will be saved as a child of the selected parent folder. For example, you can save the new Part as a child of the Beta Parts folder.



- The CATIA file name is displayed (if you already named the file in the *CATIA V5 Save As* window as described in step 3).



- Click **Save**.

5. The *Object Attributes* window is displayed with a CATIA Parts Profile Card, as shown above. If you wish to save the Part to a different class, click the arrow to the right of the **Select class** box and choose a class.

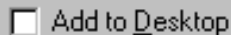


A new Profile Card is displayed in the *Object Attributes* window.



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

6. Check the **Save in Desktop** checkbox to link the Part directly to a project:



This option is not displayed in the *Object Attributes* window if you selected a project and parent link in step 4.

7. Fill in attributes for the Part and click **OK**.



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.

Saving a Part After Modifications

After a Part is added to the **CATIA - TeamPDM** 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.

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

To save a Part after modifications:

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 **CATIA - TeamPDM** and its Profile Card is updated accordingly.
or
Choose **Save As**. The *SmarTeam Save As* window is displayed.



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. Check the appropriate checkboxes.

- Choose a project from the Project Tree. The Part will be saved as a document in the selected project.

Projects:

- Choose a parent folder from the Object Tree. The Part will be saved as a child of the selected parent. For example, you can save the new Part as a child of the Beta Parts folder.
- Click **Save**. The Part (together with its hierarchical links) is saved in **CATIA - TeamPDM** and its Profile Card is updated accordingly.



Up



Saving a Part



Managing the Revisions of a

Managing the Revisions of a Part

This section provides operating instructions for managing the revisions of a Part as follows:

- Registering a Part
- Checking Out a Part
- Copying a File
- Checking In a Part
- Releasing a Part
- Checking Out a New Release
- Moving a document to the Obsolete Vault



Up



Saving a Part



Managing the Revisions of a

Registering a Part



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

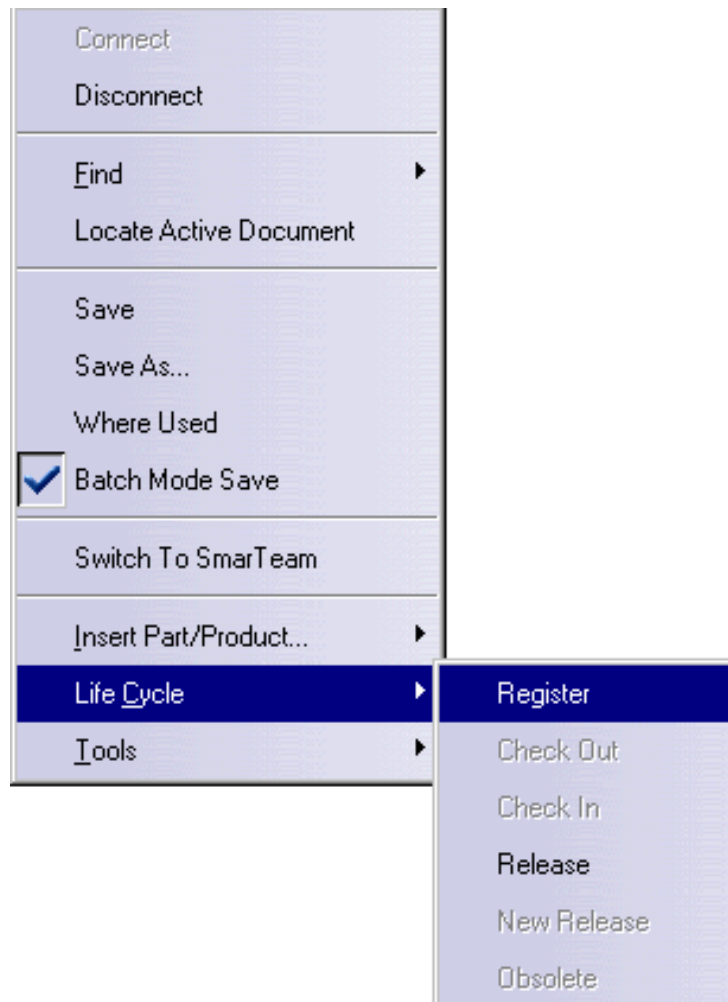
To protect the Part from modifications, place the Part into the **SmarTeam** vault by **registering** it. After the Part is registered, its status is changed to *Checked In*.

What happens next?

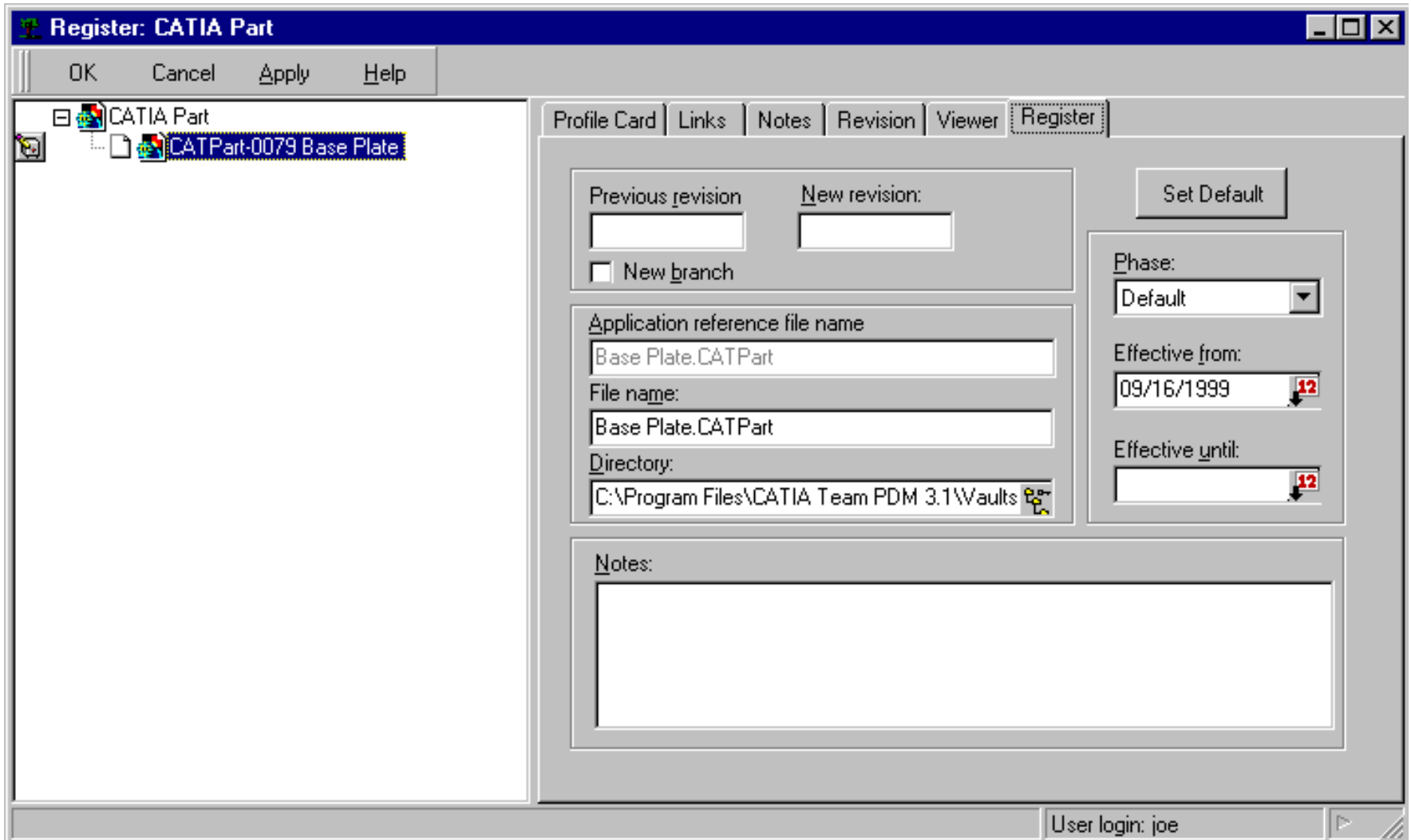
- To launch the Part into CATIA V5 and modify it, the Part must be checked out. When it is checked out, a new revision number is assigned to it.
- The Part can be copied to your desktop in read-only mode.

To register a Part

1. From the *SmarTeam* menu, point to **Life Cycle** and choose **Register**.



2. The Register window is then displayed, as shown below.



In the left side of the window, the **Register** icon appears to the left of the selected document. In the right side of the window, the *Register* window is displayed.


3. Fill in the fields in the *Register* window, as described in the table below. These fields are optional, and you may keep the default.
4. Click **OK** to register the Part and exit the **Register** view, or click **Apply** to perform the operation and remain in the view.


The Part is now placed in the *Checked In* vault. Note that the status of the document is now *Checked In*.

The following describes the attributes in the Register window:

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.
Application Reference File Name	Automatically displays the file name as it was created in a CAD system.
File Name	You may enter another file name that will be assigned to the file in the vault.
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.
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.
Notes	You may enter a comment in this field.
Set As Default	Click this option to assign the same registration information for all document revisions such as phase, effective dates, and notes.

 If you wish to modify the Part and launch it into CATIA, it must be checked out, as described on the following page.

 If you wish to copy the Part to your desktop in order to view it in CATIA as a read-only file, refer to "[Copying Documents To Your Project Desktop](#)" for instructions



Checking Out a Part



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



CATIA - TeamPDM 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. Instructions for copying a file are provided in section "[Copying a File](#)".

CATIA - TeamPDM 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 at your desktop (in read-only mode), you can perform the speedy **Check Out** operation.
- Check out a Part (from a **CATIA - TeamPDM** 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 **Edit**. 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.



Each of these procedures are described on the pages that follow.

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 section "[Checking In a Part](#)".
- You previously copied a file to your desktop (using the **Copy File** option).

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 **Check Out** item.
3. Fill in the attributes, as described below, 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 CATIA - TeamPDM window

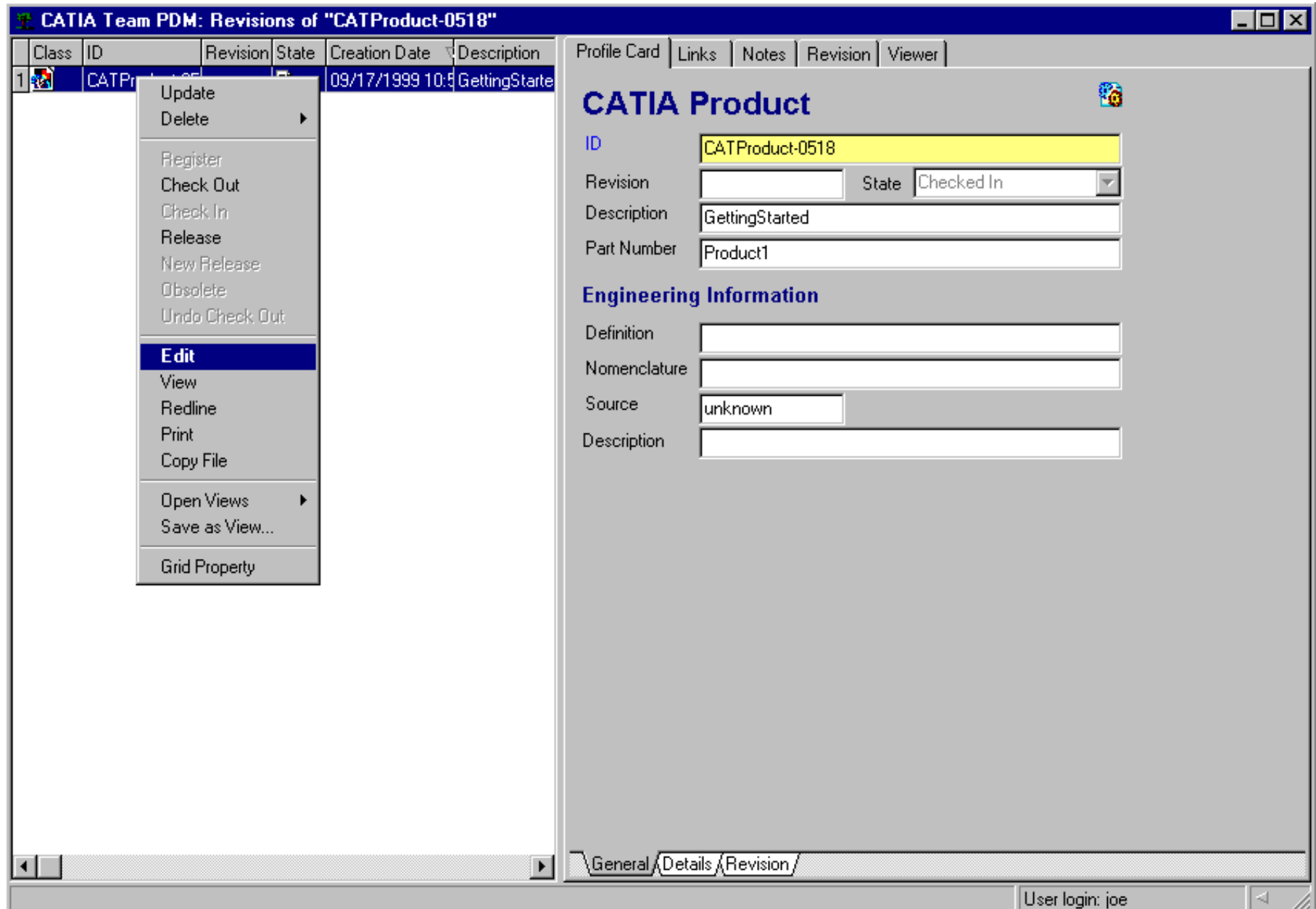
If a Part is checked in to a vault, and it is not currently displayed at your project desktop, you must locate the Part and then check it out in order to modify it in CATIA.

To check out a Part:

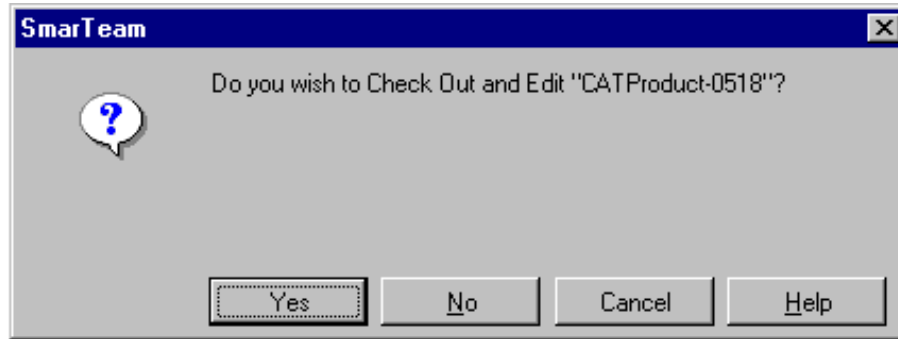
1. Run a search to locate the document that you wish to check out:
 - From the *SmarTeam* menu, point to **Find** and choose **Find Document**.
 - Select a search and click **Run**. The search results are displayed in a search results list.

- Browse through the list to locate the document you wish to modify.

2. Select the document, and right-click to display a dropdown menu. Choose **Edit**.

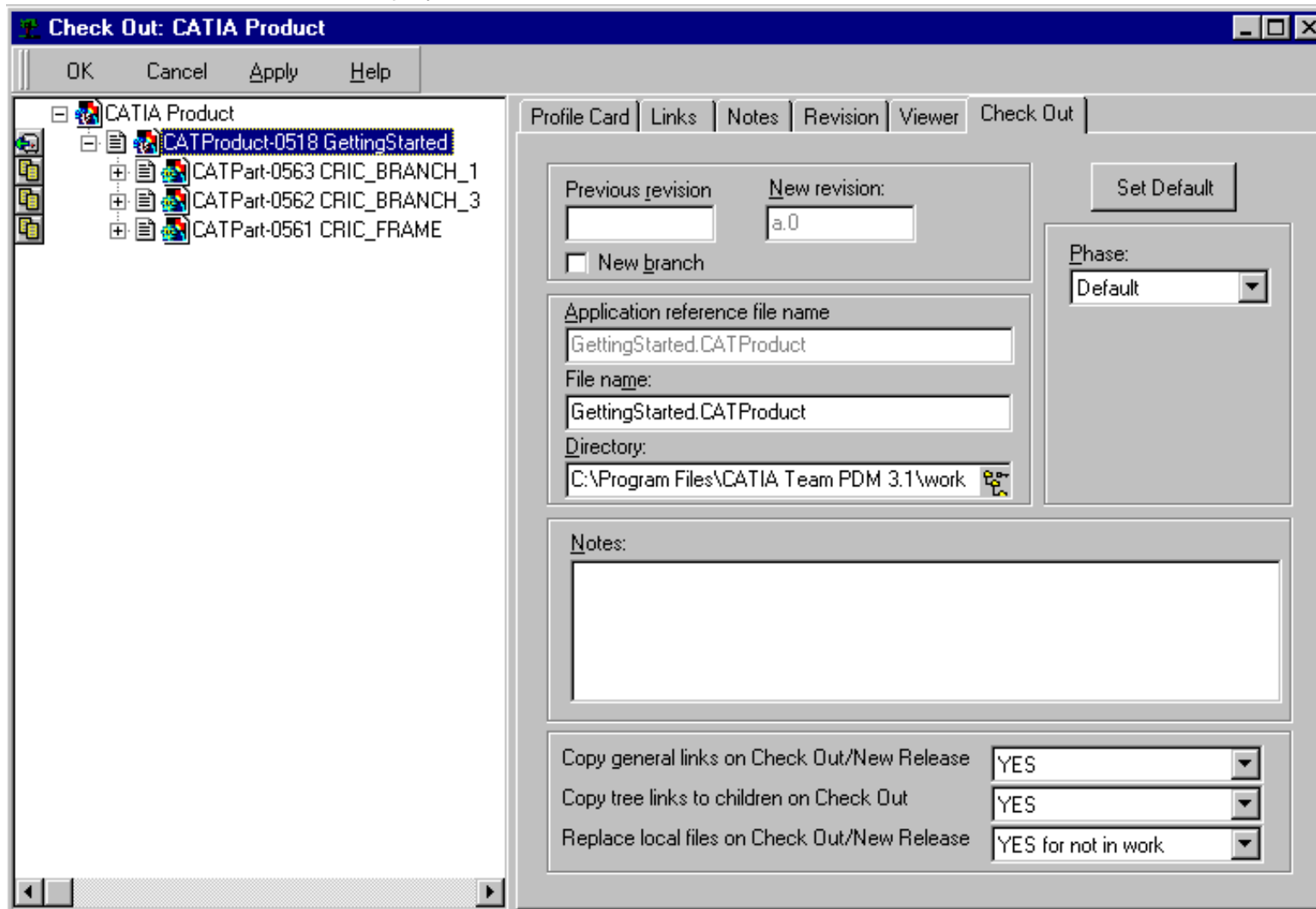


3. A message is displayed prompting you to check out the document.



Click **OK** to continue.

4. The *Check Out* window is then displayed, as shown below:





This window is also displayed when you choose the **Life Cycle/Check Out** option from the *SmarTeam* menu.

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

5. Fill in the fields in the *Check Out* window (optional) and 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.



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

The following describes the fields in the Check Out window:

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.
New Branch	<p>Click this option to enable you to create parallel branches of a revision based on the same file. Each branch will be assigned a different revision number consisting of 3 characters (a.0.1 and a.0.2 for example) instead of the standard revision number consisting of 2 character (a.0 for example).</p> <p>For example: You may have a document named Pump*55. If the Branching value is Yes, two separate users may check out the file and each can create a new revision of this file. Two different revision numbers will be assigned to these files respectively (a.0.1 and a.0.2) and they can both be modified simultaneously.</p> <p>Note: This field may be disabled according to the life cycle preferences defined by your administrator</p>
Application Reference File Name	Automatically displays the file name as it was created in a CAD system
File Name	You may enter another file name that will be assigned to the file in the vault
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
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
Notes	You may enter a comment in this field
Copy General links on Check Out/New Release	Choose Yes from the dropdown list to copy general links from the previous to the newly created revision
Copy links to children on Check out	Choose Yes from the dropdown list 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 As Default

Click this option to assign the same check out information for all document revisions such as phase, effective dates, and notes.



Up



Registering a Part



Releasing a Part



Checking Out a Part



Checking Out a New Release



Copying a File



Moving a document to the OL



Checking In a Part

Copying a File

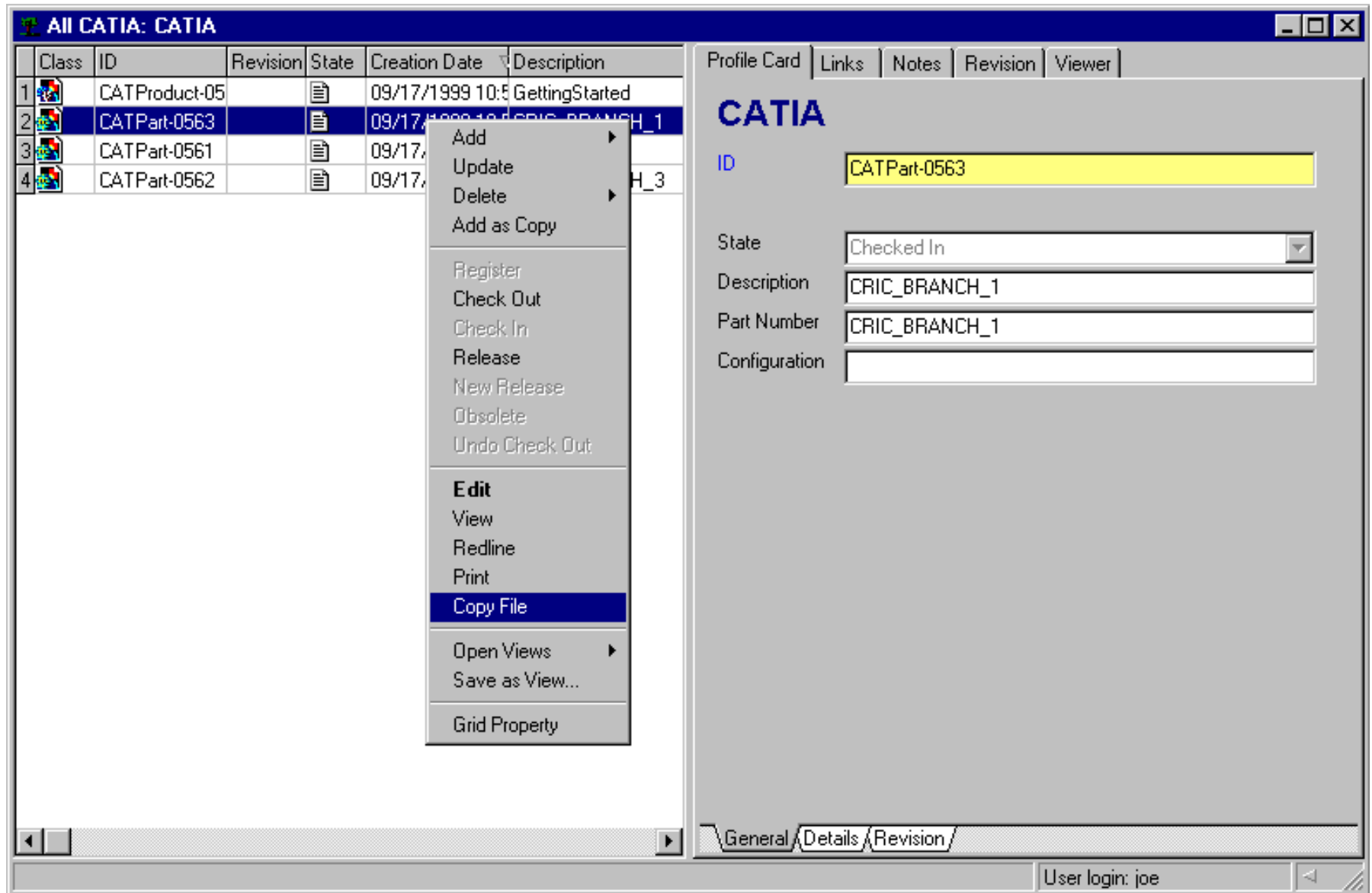


You may choose to copy a file to your desktop rather than checking it out of the vault for a number of reasons:

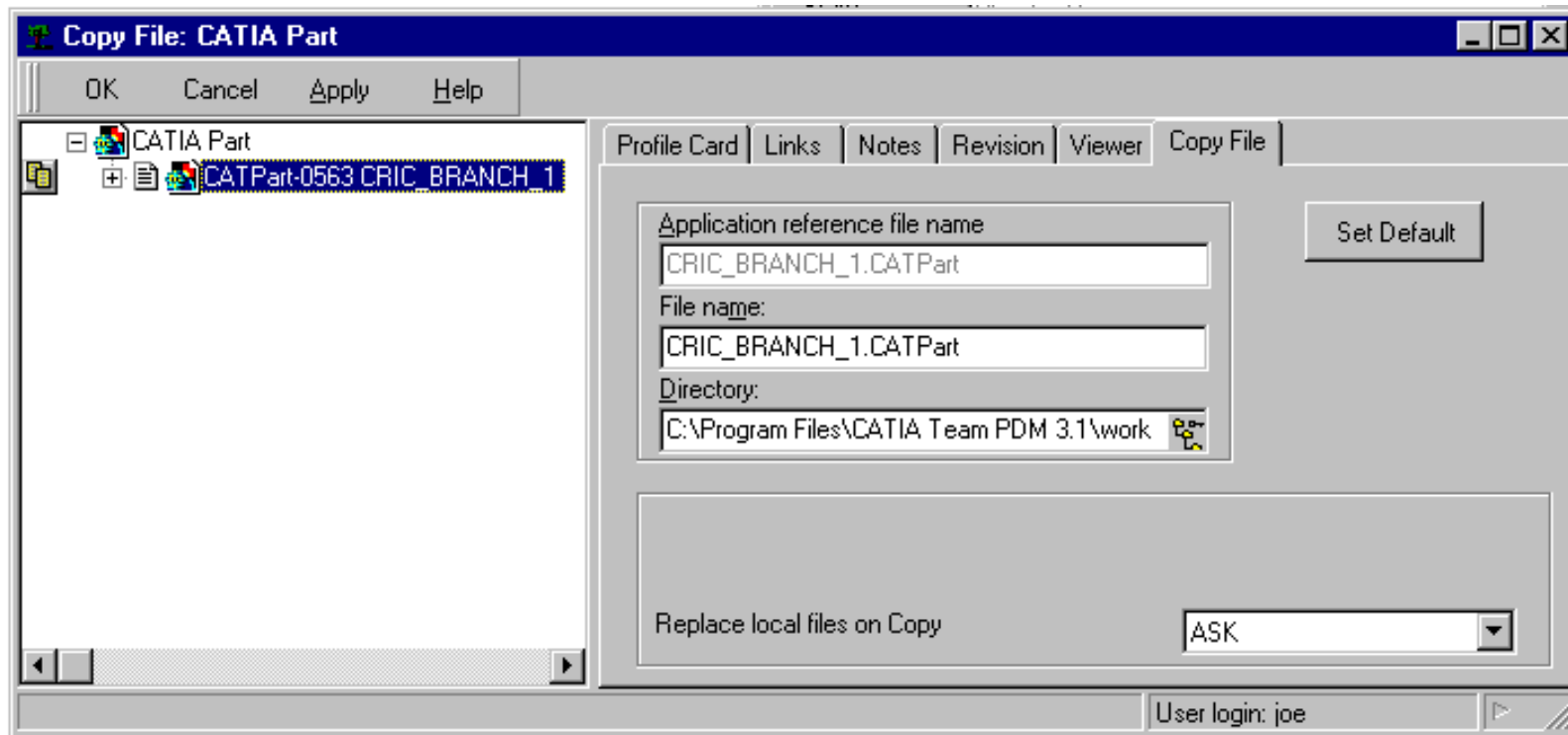
- Another user may be modifying it and you wish to work with it at your desktop as well.
- You may wish to view it so that you can see how it works together with other documents at your desktop. For example, you might copy the file of a Product to your desktop in order to view how it works together with various Parts. You can then check the Part out from the *SmarTeam* menu.

To copy a file:

1. You can run a search to locate the document that you wish to copy to your desktop. The search results list displays the documents that match the search criteria.
2. From any **CATIA - TeamPDM** window, right-click on the document and choose **Copy File**.



3. The *Copy File* window is displayed.



Fill in the fields in the window and click **OK** to copy the file to your desktop. The status of the drawing remains *Checked In*.



- Up
- Registering a Part
- Releasing a Part
- Checking Out a Part
- Checking Out a New Release
- Copying a File
- Moving a document to the OI
- Checking In a Part

Checking In a Part



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

- **Check In**, 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 section "[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.

To check in a Part:

1. Activate the document containing the Part you want to check-in
2. In the *SmarTeam*, select **Check In** item. The *Check In* window is displayed, as shown below.



You may add comments that will be linked to the part or change conditions in the life cycle operations.

3. Fill in the fields and click **OK**. The status of the document is automatically changed to *Checked In*.



The Part is now checked into the vault for safekeeping. A copy of the file remains at your desktop in read-only mode. You can check it out again in order to edit the Part, as described in section "[Checking Out a Part](#)".



Releasing a Part



When a Part is ready to be moved to the next phase of development, 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.

To release a Part:

1. Display the Part in CATIA.
2. In the *SmarTeam* menu, select **Release** item.
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 and click **OK**. The status of the document is automatically changed to *Released* in the **State** field.



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



Checking Out a New Release



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



CATIA - TeamPDM 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. Instructions for copying a file are provided in section "[Copying Documents To Your Project Desktop](#)".

CATIA - TeamPDM provides two methods for checking a New Release out of the vault:

- Check out 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 *SmartTeam* menu.
- Check out a New Release (from a **CATIA - TeamPDM** menu): If the Part is not currently displayed at your desktop, you must find the Part by running a search. From the displayed search results window, select the Part and choose **Edit**.

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 and then check out the document. The Part is immediately launched into CATIA.



Moving a document to the Obsolete Vault



When a document becomes obsolete or inactive, you may move it to the Obsolete section of the vault. An *Obsolete* document cannot be changed or released.

To move a document to the Obsolete vault:

1. Select the Part document as active document
2. In the *SmarTeam* menu, select **Obsolete** item. The *Obsolete* window is displayed. The fields in the *Obsolete* window are identical to the fields in the *Check In* window.
3. Fill in the fields and click **OK**. The status of the document is automatically changed to *Obsolete*.



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.

CATIA - TeamPDM provides powerful tools to assist you in building your assemblies:

- **Find options:** Use the **SmarTeam Find** options to locate Parts and Products.
- **Copy File option:** As you build your Assembly, you can copy Parts/Products to your desktop to view how they fit together with the current Assembly.
- **Insert Component options:** When you wish to insert an existing Part or Product as a component of the current Product, use the **Insert Component** option. **CATIA - TeamPDM** inserts the component into the current Product. When you save the Assembly, the components are saved as children of the Assembly. In this way, **CATIA - TeamPDM** accurately reflects the structure of your Assembly.
- **Save option:** If you created new Parts as components of the Assembly, **CATIA - TeamPDM** will save each of these Parts/Products into the **CATIA - TeamPDM** 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 option:** **CATIA - TeamPDM** 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 **CATIA - TeamPDM** 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.

This section provides the following instructions:

Building an Assembly



Saving an Assembly



Managing the Revisions of an Assembly





Up



Managing Parts



Managing Assemblies



Managing Drawings



Searching for Documents



Creating and Viewing Links

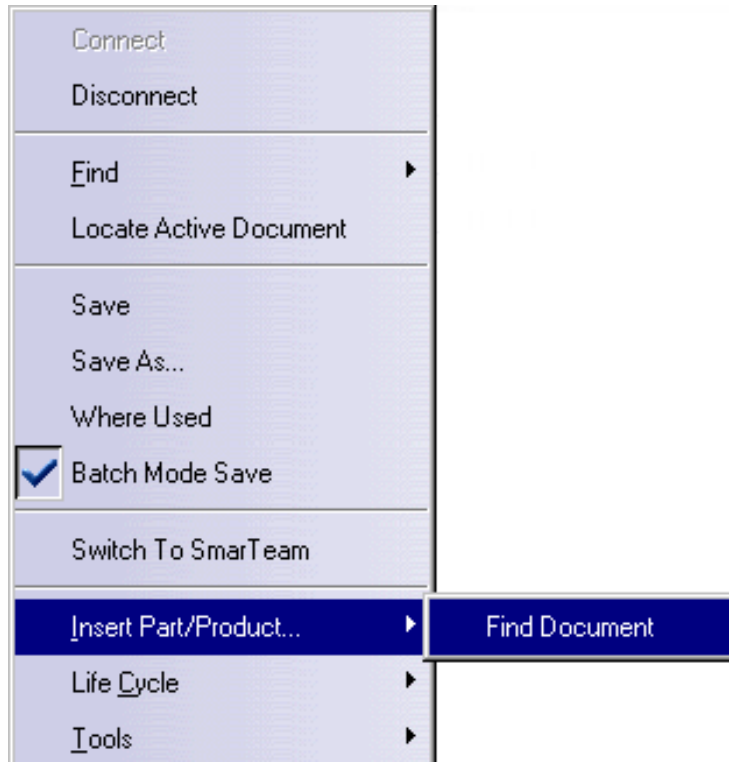
Building an Assembly



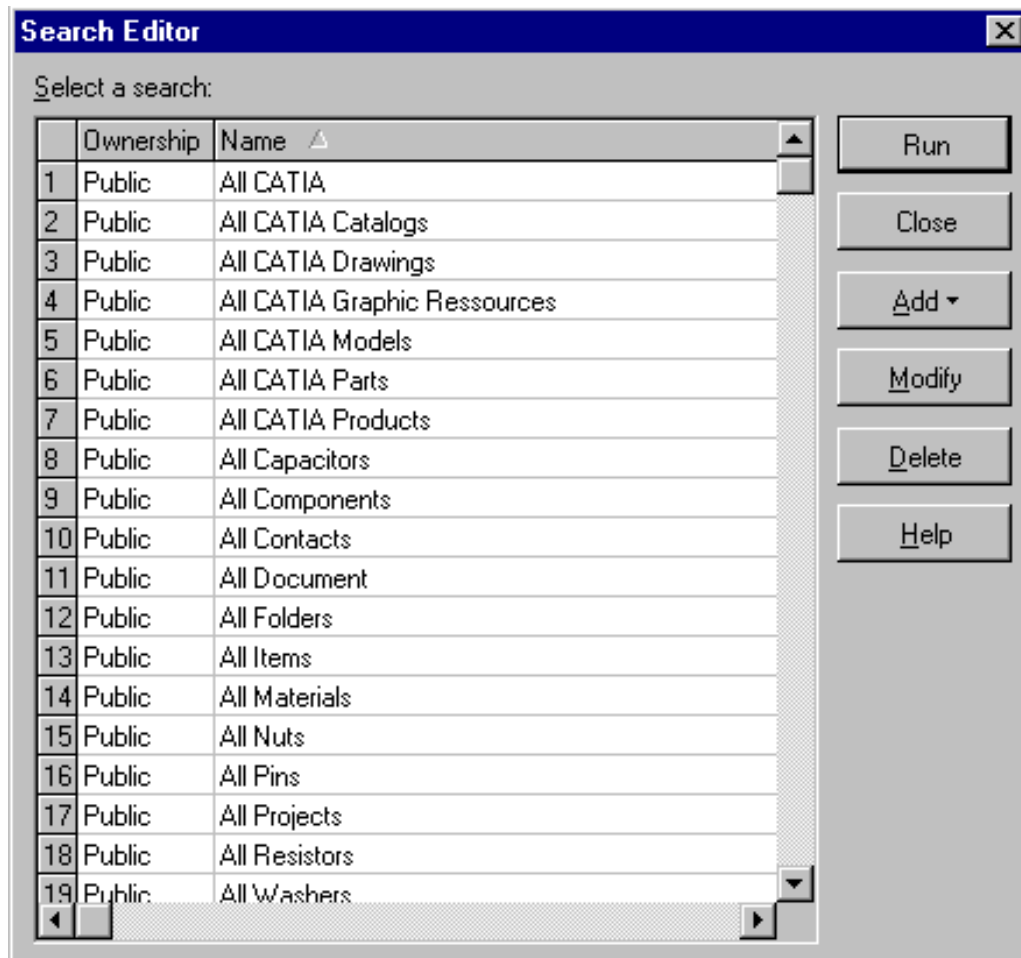
The **Insert Component** option enables you to insert existing Parts/Assemblies as components of the current Assembly.

To insert a component:

1. Display an Assembly structure in CATIA V5.
2. From the *SmarTeam* menu, choose **Insert Component** and point to **Find Document**.



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



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



You can also modify a search or define a new search. Refer to "[Searching for Documents](#)" for instructions

5. In the displayed window, choose a document and click **OK**.

The screenshot shows the 'All CATIA Parts: CATIA Part' dialog box. On the left is a table with the following data:

Class	ID	Revision	State	Creation Date	Description
1	CATPart-0566			09/17/1999 1	CRIC_BRANCH_1
2	CATPart-0564			09/17/1999 1	CRIC_FRAME
3	CATPart-0565			09/17/1999 1	CRIC_BRANCH_3

The right side of the dialog shows the 'Profile Card' for the selected part 'CATPart-0566'. The fields are:

- ID: CATPart-0566
- Revision: (empty)
- State: New
- Description: CRIC_BRANCH_1
- Part Number: CRIC_BRANCH_1

Below these is the 'Engineering Information' section with fields for Definition, Nomenclature, Source (unknown), and Description.

At the bottom, there are tabs for 'General', 'Details', and 'Revision', and a 'User login: joe' field.

6. The document is inserted into the assembly structure.



Saving a Product



Every CATIA Assembly should be saved into the **CATIA-TeamPDM** database. After you create a new Assembly or modify an existing one, choose one of the **SmarTeam Save** options. Then, register the Assembly by choosing the **Life Cycle/Register** option. The Assembly is then placed into the *Checked In* vault.

The **CATIA-TeamPDM** integrated menu provides two methods for saving Assemblies:

- **Save:** Saves the Assembly into the **CATIA-TeamPDM** database. Refer to section "[Saving a Part](#)" for details.
- **Save As:** Saves the Assembly into the **CATIA-TeamPDM** 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 **CATIA-TeamPDM** 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, section "[Saving a Part for the First Time](#)"
- To save a document for subsequent saves, section "[Saving a Part After Modifications](#)"

Saving the Components of an Assembly

When you save an Assembly, **CATIA-TeamPDM** automatically saves the components of the Assembly as well:

- If the components of the Assembly are already saved into the **CATIA-TeamPDM** 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 CATIA-TeamPDM database, then each of these components will be saved to CATIA-TeamPDM one by one.
 - A Profile Card for each component is displayed (in the *Object Attributes* window). Fill in the Profile Cards, and click **OK**.
 - After each component is saved, a Profile Card for the Assembly is displayed (in the *Object Attributes* window). Fill in the Profile Card, and click **OK**.

CATIA-TeamPDM enables you to save the components of an Assembly **without** displaying each Profile Card. This can save you a great deal of time.

CATIA-TeamPDM provides two methods for batch saving components:

- From the *SmarTeam* menu, choose **Batch Mode Save**. Then, save the Assembly using the **Save** or **Save As** option. **CATIA-TeamPDM** saves all the components but does not display a Profile Card for each one.
- From the *SmarTeam* menu, choose **Save As**. In the **Save As** window, click the **Options** tab to display the *Save Option* window. In the **Save** tab, check the checkboxes and click **OK**.



Managing the Revisions of a Product

This section provides operating instructions for managing the revisions of an Assembly as follows:



Overview



As you work with an Assembly, you can make continuous revisions to it. **CATIA-TeamPDM** 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. **CATIA-TeamPDM** ensures that the integrity of the Assembly and its children is not jeopardized by disabling life cycle operations.

In addition, **CATIA-TeamPDM** provides several life cycle options which can help you manage your Assembly. These life cycle options are described in section "[Life Cycle Options for Managing an Assembly](#)".

For example:

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



These life cycle options 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.

The following examples show how **CATIA-TeamPDM** protects the integrity of an Assembly.

- When you **Release** an Assembly, its children must be Released as well. For example, if an Assembly has 10 sub-Assemblies, each of the sub-Assemblies must be assigned the *Released* status.
- You can only move a sub-Assembly to the **Obsolete** vault if its parent Assembly is also Obsolete.
- To perform a revision operation on an Assembly and all its children simultaneously, **CATIA-TeamPDM** provides the **Propagate Operation** option. For example, if you wish to check out a Stump Preacher Guitar and all its 10 sub-Assemblies, 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 sub-Assembly out of the vault independently and leave the parent Assembly in the vault.



 [Up](#)

 [Overview](#)

 [Registering an Assembly](#)

 [Checking In/Checking Out/Re](#)

 [Life Cycle Options](#)

Registering a Product



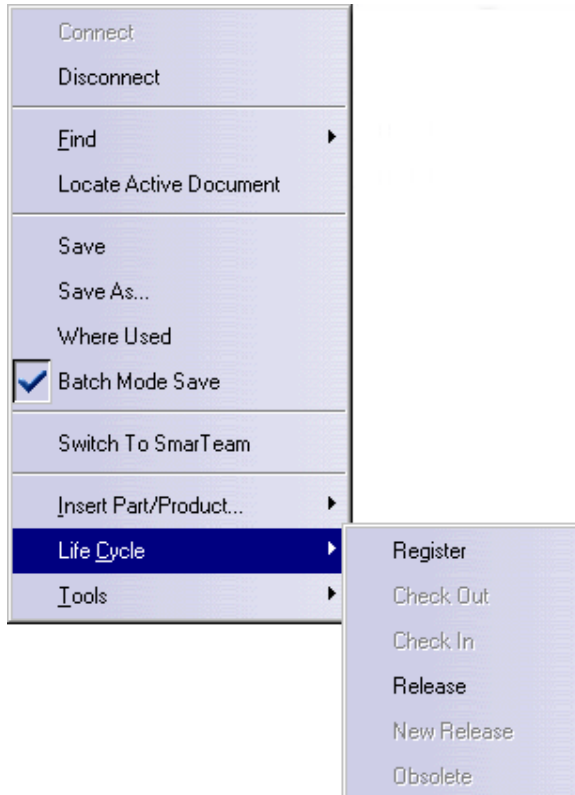
When a Product is first saved into the **CATIA-TeamPDM** database, it is automatically assigned the *New* status. This means that the Product has not yet been checked into a *SmarTeam* vault.

To protect the Product from modifications, place the Product into the *SmarTeam* vault by registering it. After the Product is registered, its status is changed to *Checked In*.

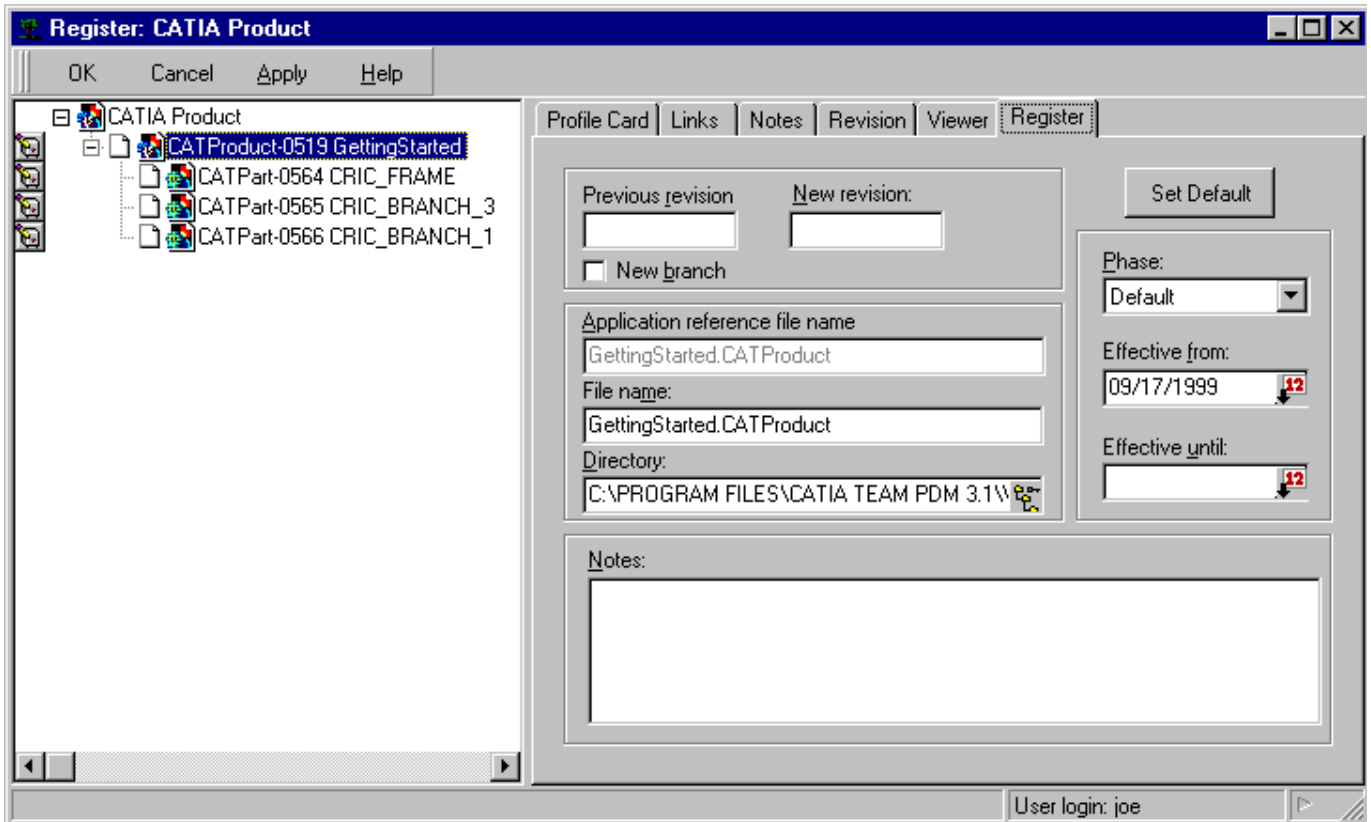
When you register the Assembly, you can register the Product and all its components simultaneously using the **Propagate Operation** option. Refer to section "[Propagate Operation](#)" for details.

To register an Assembly:

1. From the *SmarTeam* menu, point to **Life Cycle** and choose **Register**.

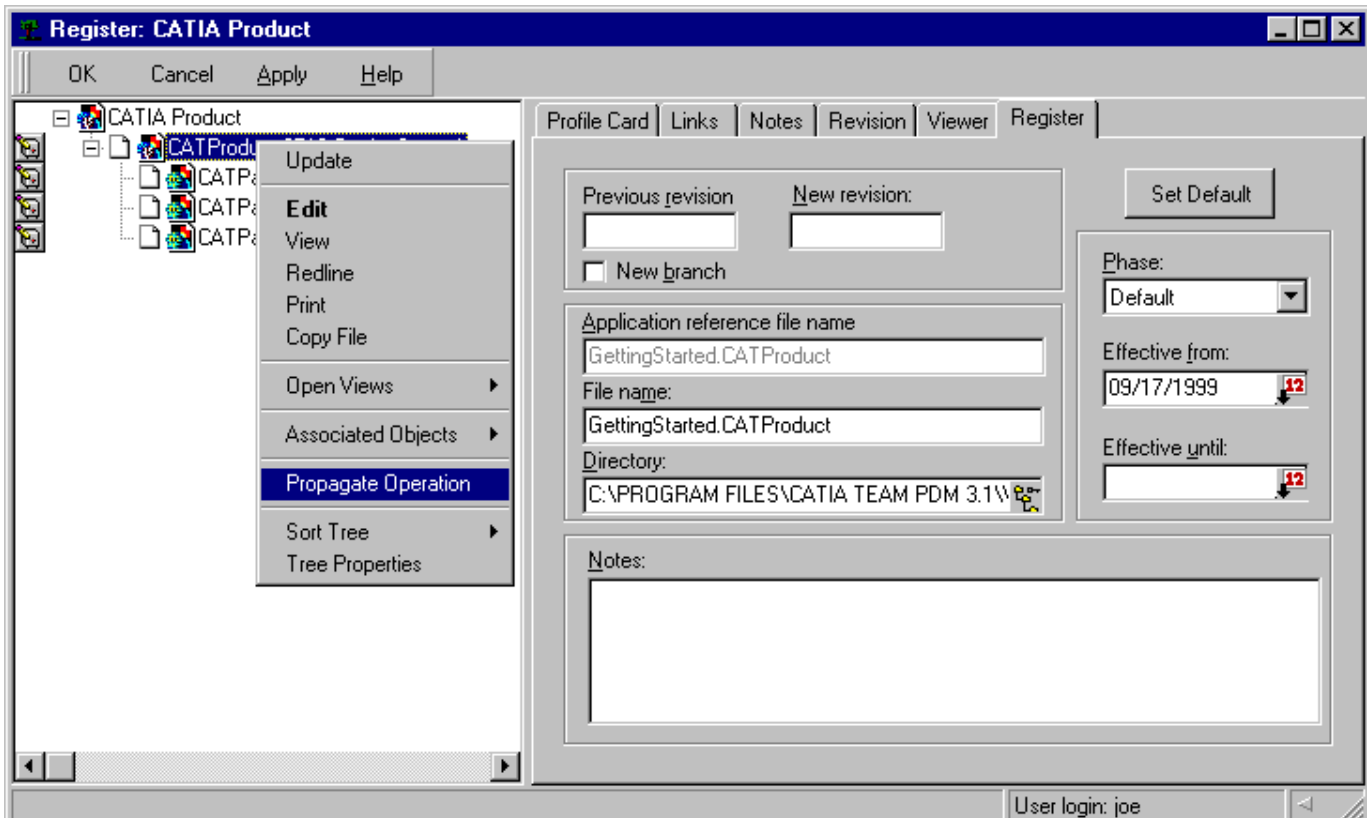


The *Register* window is then displayed, as shown below.



In the left side of the window, the Assembly and its components are displayed with the **Register** icon to the left of the selected document. In the right side of the window, the *Register* window is displayed.

- To register the Assembly and all its components together, right-click on the Assembly to display a dropdown menu and choose **Propagate Operation**.



- Fill in the fields in the *Register* window (optional) and click **OK**. Refer to section "[Registering a Part](#)" for a description of these fields.
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 **CATIA-TeamPDM** 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 10 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.

Handling Components

CATIA-TeamPDM 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 the **Propagate Operation** option, as described in section "[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** option enables you to maintain a sub-Assembly 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 **CATIA-TeamPDM**, 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. **CATIA-TeamPDM** enables you to do so.

To check out an Assembly:

1. Display the Assembly in SolidWorks (in read-only mode).

2. To check out a document: Select a document (Assembly or Part), in *SmarTeam* menu, select *Life-Cycle* then choose **Check Out**. In the displayed *Check Out* window, fill in the fields and 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 choose a **SmarTeam Save** option.

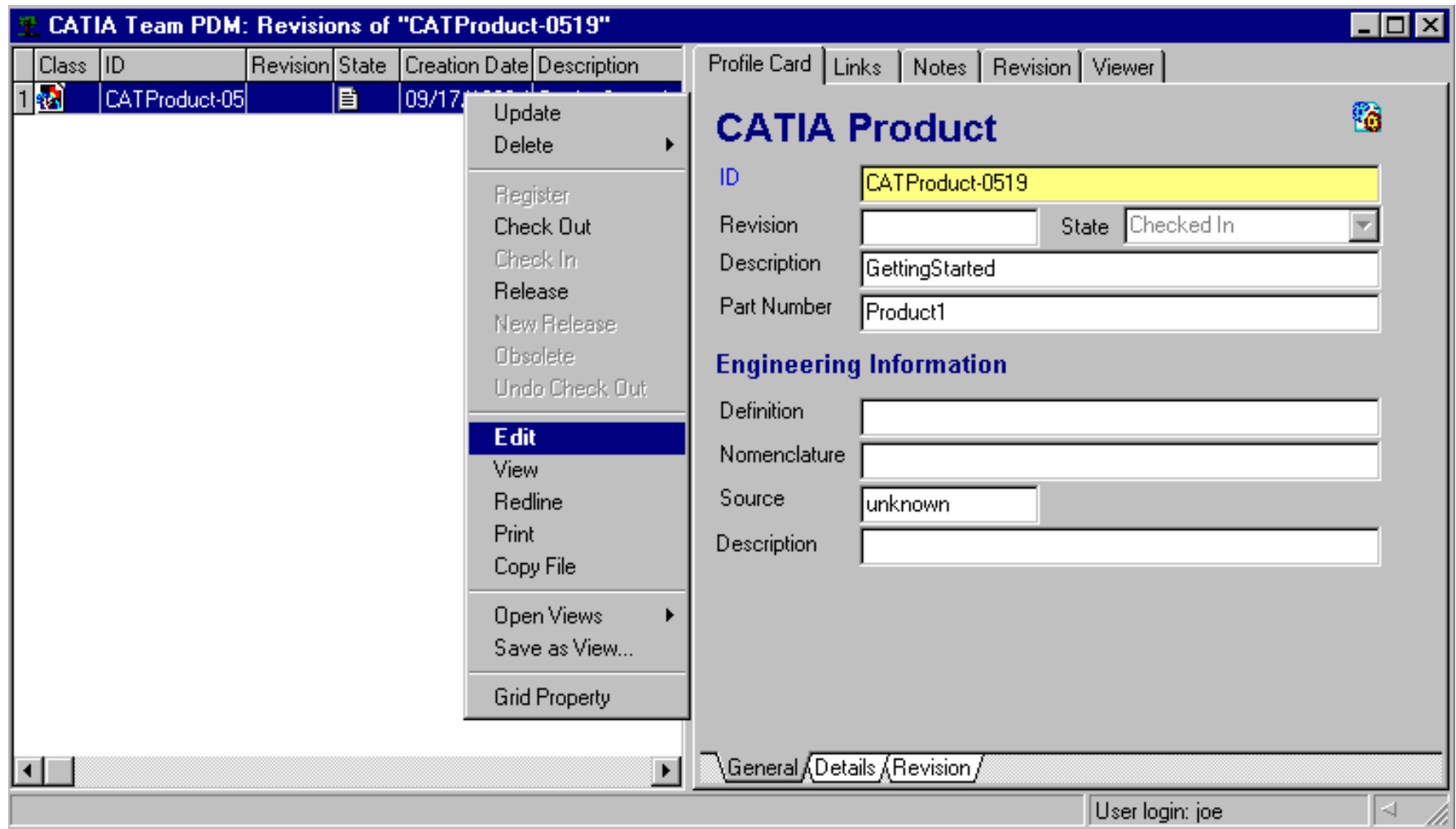
Checking Out an Assembly from a CATIA-TeamPDM window.

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 choose **Edit** in order to launch it into CATIA and modify it.

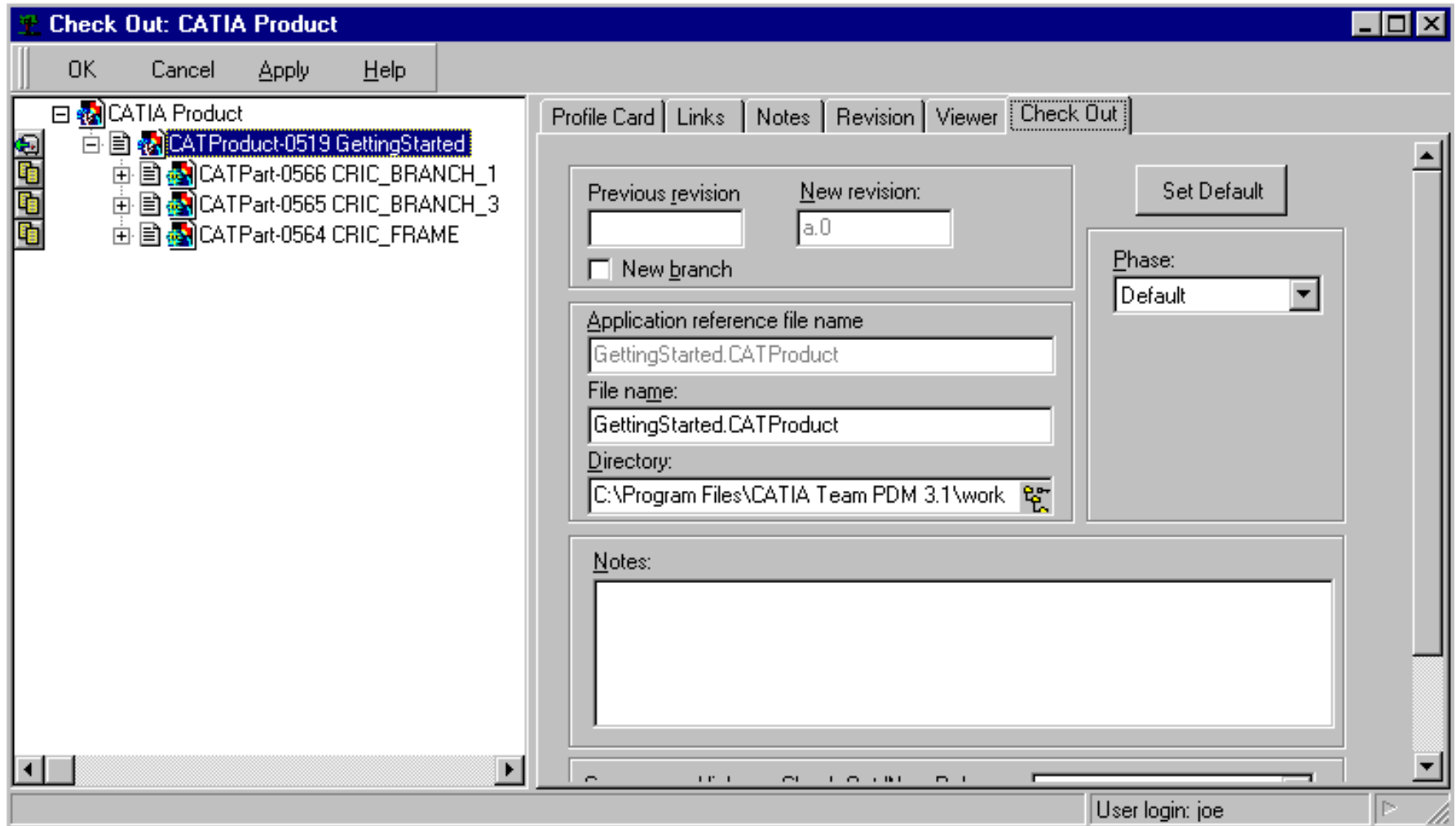
In the displayed *Check Out* window, you can right-click to display a dropdown menu which provides several life cycle options. These life cycle options can assist you in managing your Assembly. Refer to section "[Life Cycle Options for Managing an Assembly](#)" for details.

To check out an Assembly:

1. Run a search to locate the Assembly that you wish to check out.
2. Select the document, and right-click to display a dropdown menu. Choose **Edit**.



3. A message is displayed prompting you to check out the document. Click **Yes** to continue.
4. The *Check Out* window is then displayed, as shown below:



This window is also displayed when you choose the **Life Cycle/Check Out** option from the *SmarTeam* menu.

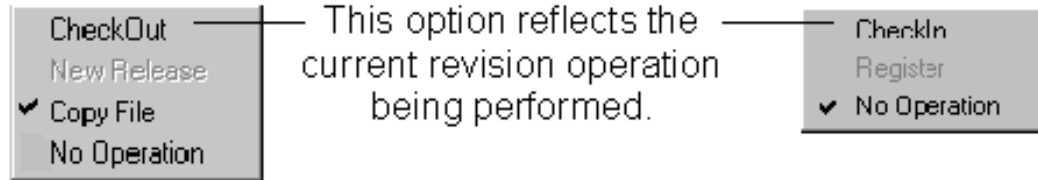
In the left side of the window, the **Check Out** icon appears next to the Assembly and the **Copy File** icon appears next to the components.



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

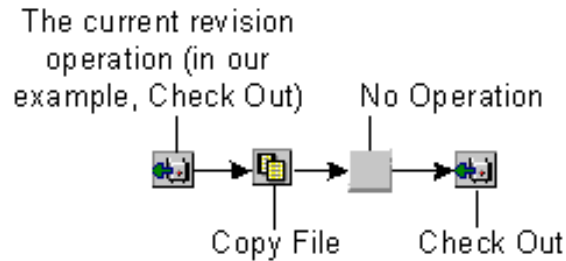
5. You can define an operation for each component as follows:

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



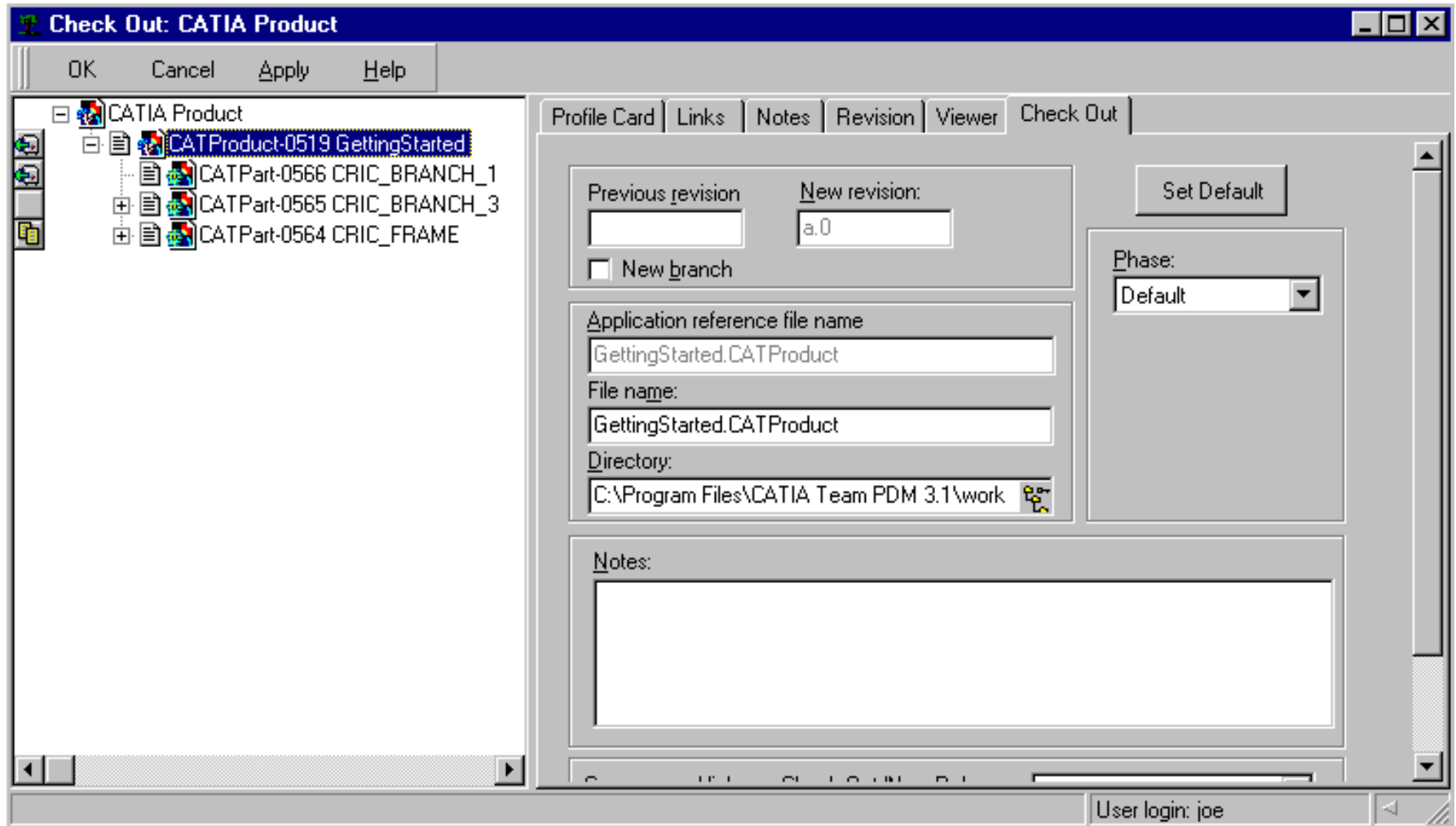
Or

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



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

The following example shows how sub-Assemblies are handled individually during the **Check Out** operation:



Checking In/ Releasing an Assembly

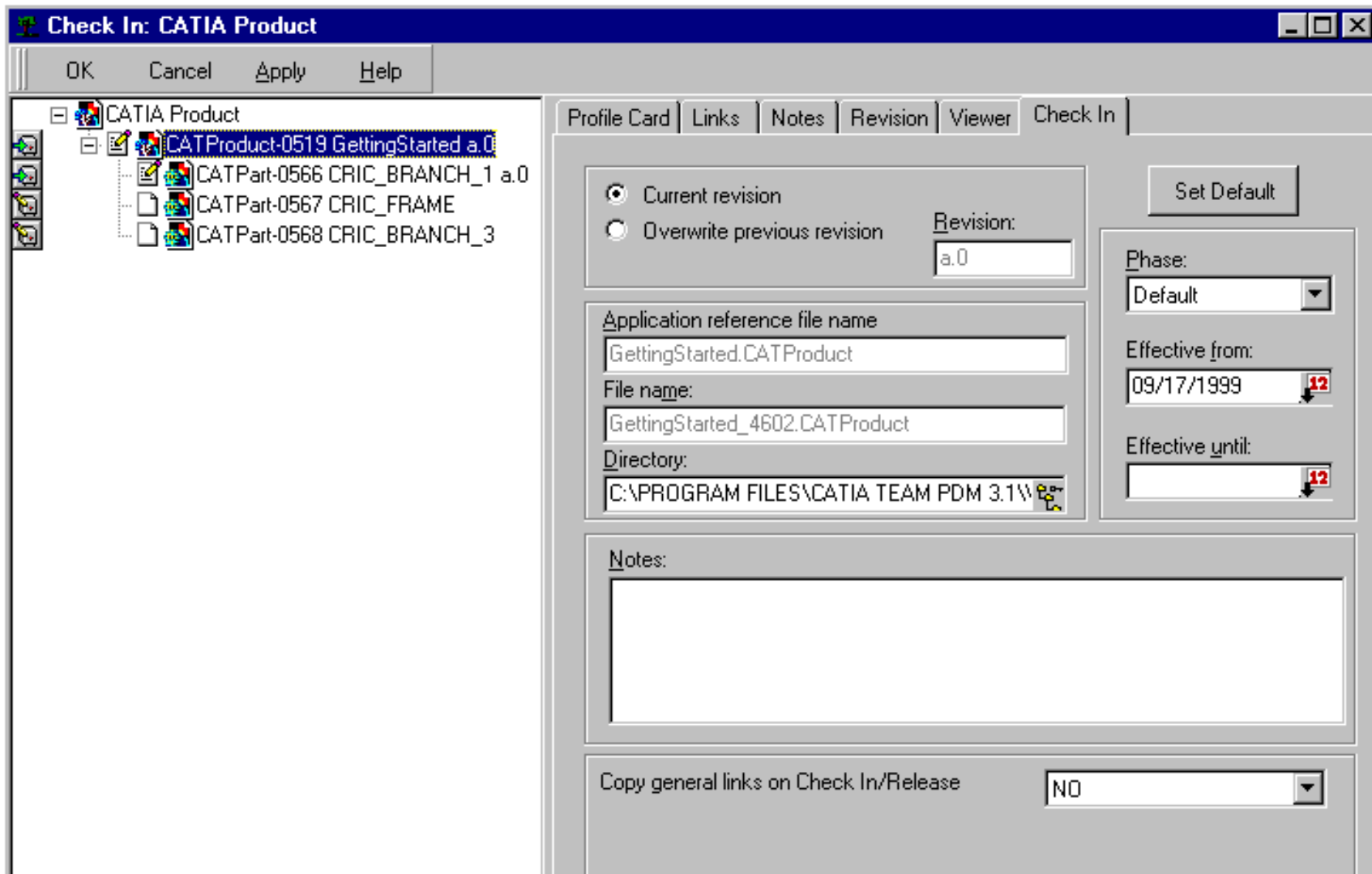
CATIA-TeamPDM provides two methods for checking in or releasing an Assembly:

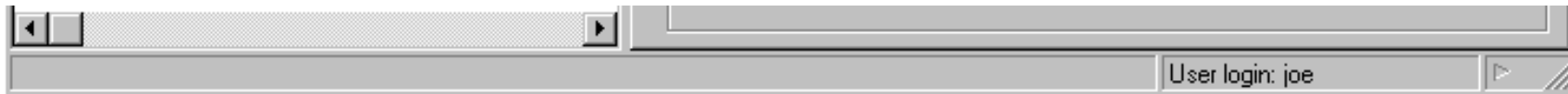
- Check in the Assembly and its components from any SmarTeam window. You must select each document one by one and check it in (or Release it). This method is identical to the checking in a Part as described in section "[Checking In a Part](#)". Below is a quick summary.
 1. Select the Part or Assembly and choose **SmarTeam/Check In** or **Release**. The *Check In (Release)* window is displayed, as shown in section "[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**) option (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 section "[Life Cycle Options for Managing an Assembly](#)".

To check in an Assembly:

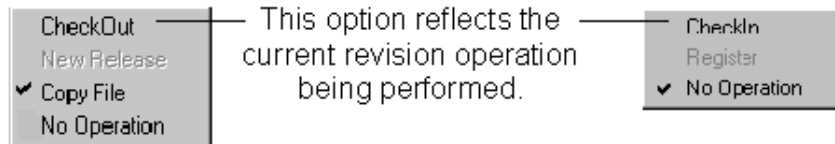
From the *SmarTeam* menu, point to **Life Cycle** and choose **Check In**. The *Check In* window is displayed as shown below:





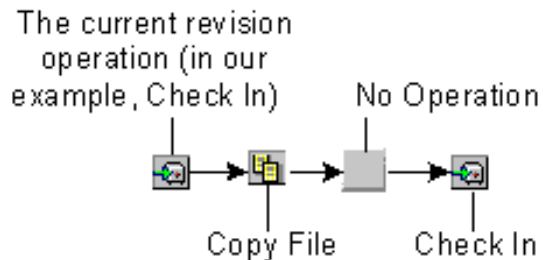
On the left side of the window, the Assembly and its components are displayed. You can

- Check in all the documents together:
 1. Right-click and choose **Propagate Operation**.
 2. The **Check In** icon is displayed next to each component in the tree.
 3. Fill in the fields in the *Check In* window (optional) and click **OK**.
 4. The Assembly and its components are checked in to the vault. A copy of these documents does not remain at your desktop.
- Handle each component individually:
 1. 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


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



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

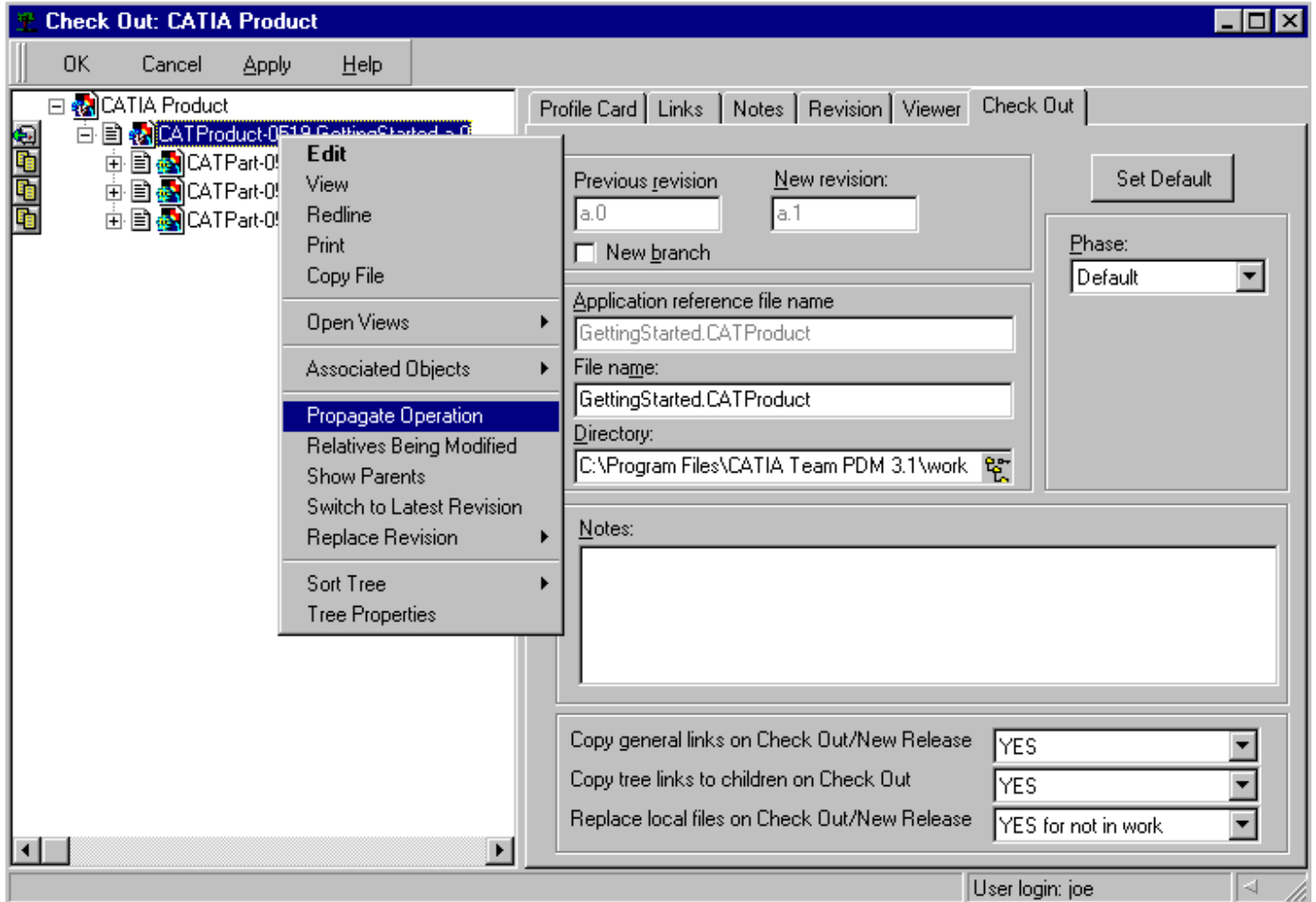


Life Cycle Options for Managing an Assembly

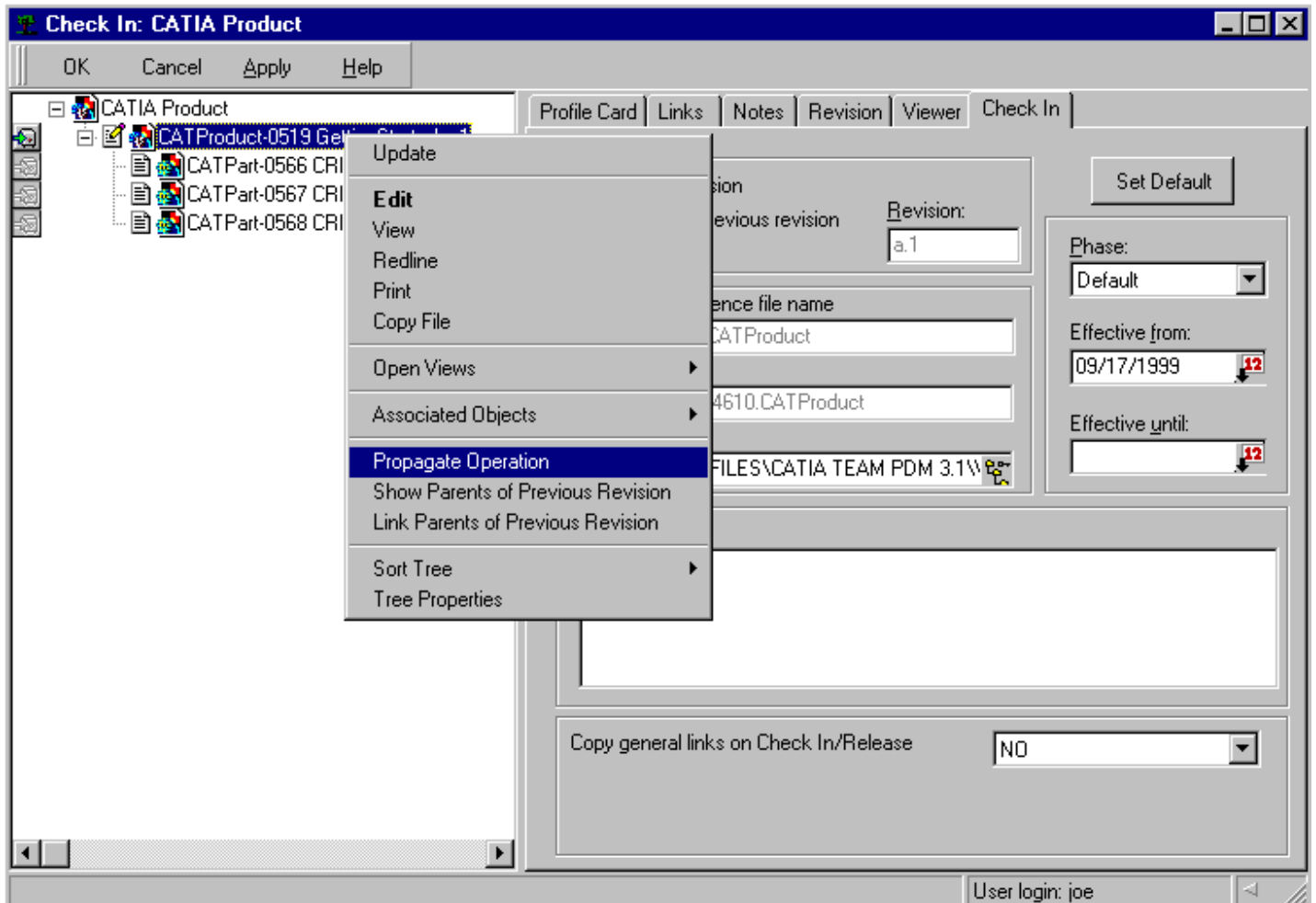
 Your Assemblies are often complex with numerous sub-Assemblies. As you manage the revisions of the Assembly, you must keep track of the status of the revisions of the parents and children of each Assembly. **CATIA-TeamPDM** provides several life cycle options that can assist you in this task.

Overview

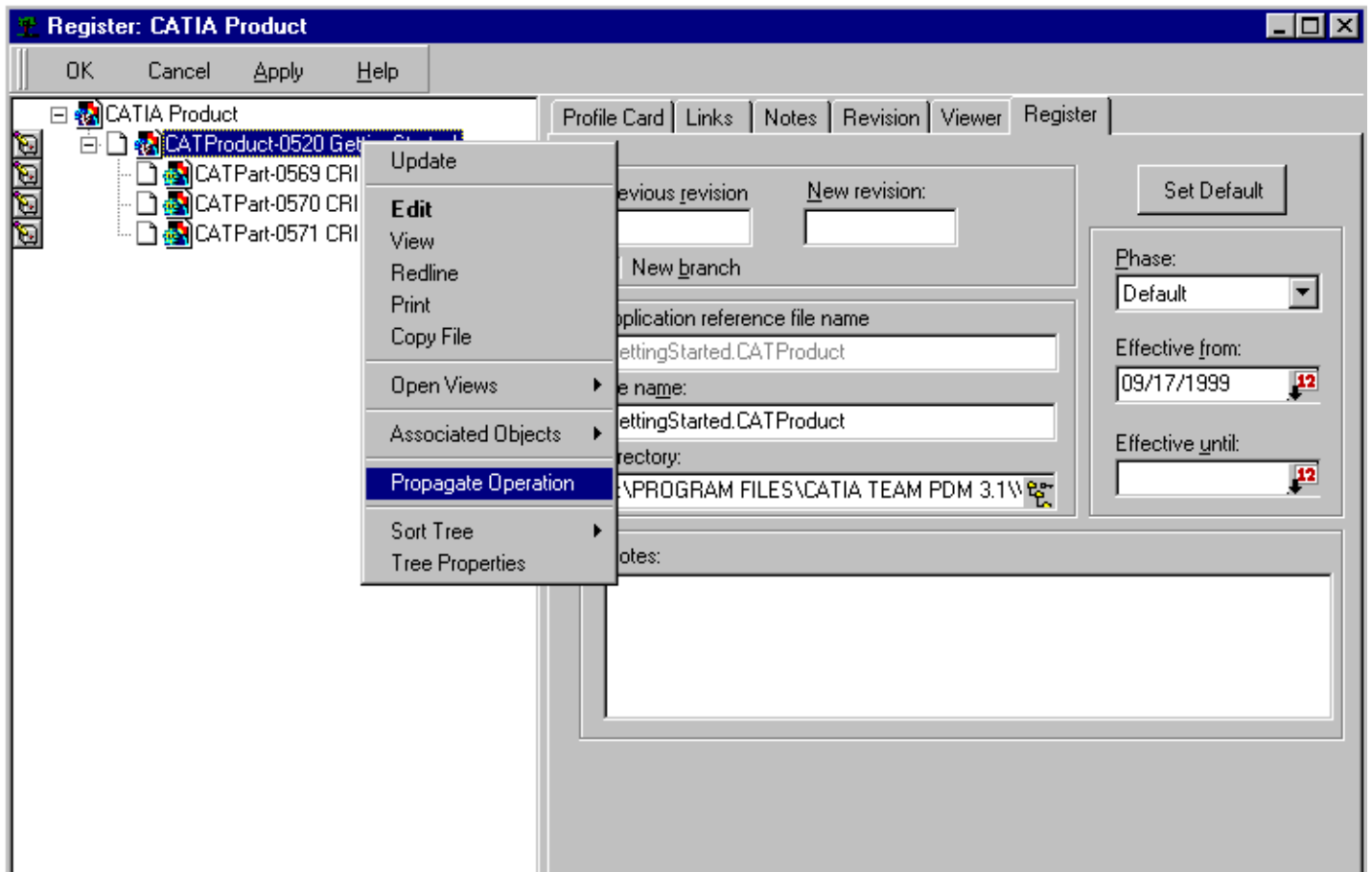
From the *SmarTeam* menu, point to **Life Cycle** and choose a life cycle operation (**Check Out** for example). In the displayed window, right-click on a document to display a dropdown menu. A list of life cycle options is displayed. This list differs according to the life cycle window being displayed, as shown below:



These life cycle options are displayed in the *Check Out/New Release* Window.



These life cycle options are displayed in the *Check In/Approve* window.





These life cycle options are displayed in the *Register* window.

The life cycle menus include the following life cycle options:

Feature	Description	Example
Associated Objects	Enables you to view those documents that are linked as dependencies or reverse dependencies to the selected document. You can then perform life cycle operations on these Associated Objects. Refer to the " Managing the Revisions of a Drawing " section for a complete description.	In CATIA-TeamPDM , 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 dependency (the Stump Preacher Guitar Drawing).
Propagate Operation	Performs the same life cycle operation on the Assembly and all its children. Refer to section " Propagate Operation " for operating instructions.	When you register an Assembly, you can register all the children simultaneously.
Relatives Checked Out	a document that are currently <i>Checked Out</i> . Refer to section " Relatives Being Modified " for instructions.	When you manage an Assembly, you can view a list of its parents or children that are currently being modified (checked out) in the <i>Relatives Being Modified</i> window. This is particularly helpful when you wish to release an Assembly, since all its documents must first be checked in.
Show Parents	Displays the parents of a document in a separate window. You can then select the parents that you wish to check out as well. Refer to section " Show Parents " for instructions.	When you check out a Part that is a child of a few different Assemblies, the <i>Show Parent</i> window displays the parents of the selected document and their revision status. You can then choose the Assemblies that you wish to check out together with the Part.
Switch to Latest Revision	Enables you to check out the latest revision of a document. Refer to section " Switch to Latest Revision " for instructions.	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 sub-Assemblies to the Assembly being checked out. Refer to section " Replace Revision " for instructions.	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 sub-Assemblies.
Link Parents of Previous Revision	Enables you to link the current Part/Assembly to a different Parent (with a previous revision number).	When you check in a Part after modifying it, you can link the Part to a previous revision of the parent Assembly (rather than the current revision of the parent Assembly).

Each of these life cycle options is described below.

Propagate Operation: *Performs the same life cycle operation on the Assembly and all its children.*

To propagate an operation:

1. In any life cycle window, right-click on an Assembly and choose **Propagate Operation**. The icons of all the children change to reflect the revision operation to be performed.
2. Fill in the fields in the revision window (optional). 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.*

To show the relatives being modified:

From the *Check Out/New Release* window, right-click on a document (not its icon) to display a dropdown menu, and choose the **Relatives Being Modified** option. 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.*

To show (and check out) the parents of a document:

1. From any *Check Out/New Release* window, right-click on a document (not its icon) to display a dropdown menu, and choose the **Show Parents** option. The *Select Parents* window displays the parents of the selected document.
2. To check out the parents, select the parents in the *Select Parents* window and click **OK**.



Use the Shift Key to select multiple parents.

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

Switch to Latest Revision: *Checks out the latest revision of a document.*

To switch to the latest revision:

In the *Check Out/New Release* window, the **Switch to the Latest Revision** option enables you to check out the latest revision of a document.

A window 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 sub-Assemblies 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** option. 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.

To replace a revision:

1. From the *Check Out/New Release* window, 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. 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.

Link Parent of Previous Revision: *Enables you to link the current Part/Assembly to a different Parent (with a*

previous revision number).

To link a Part to a previous revision of its parent:

1. From the *Check In/Release* window, right-click on a document (not its icon) to display a dropdown menu and choose **Link Parent of Previous Revision**.
2. A window is displayed with a list of the previous revisions of the parent Assembly for the Part current being checked in.
Select a parent and click **OK**. The *Check In* window is refreshed to display the selected parent Assembly. When you check in the Part, it will be linked to the selected parent Assembly.



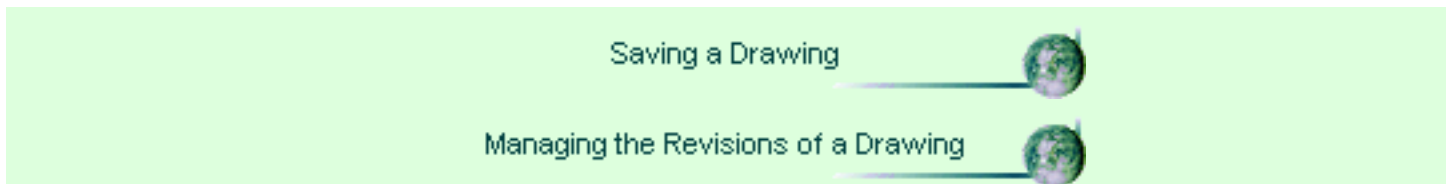
Managing Drawings

CATIA - TeamPDM provides powerful tools to assist you in creating, saving and managing your Drawings.

- When you save the Drawing into the **CATIA - TeamPDM** 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 **CATIA - TeamPDM** protects the integrity of the Drawing.

This section provides the following instructions:



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**).



Refer to section "[Saving a Part for the First Time](#)" for detailed instructions on using the **Save** option, or to section "[Saving a Part After Modifications](#)" for detailed instructions on using the **Save As** option. This process is identical to saving a Part.

2. A CATIA Drawing Profile Card is displayed (in the *Object Attributes* window).
3. Fill in the fields in the Profile Card and click **OK**.

The Drawing is saved to the **CATIA-TeamPDM** database, and a general link is created linking the Drawing to the Part/Assembly on which it was based.

In this way, **CATIA-TeamPDM** 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.



Managing the Revisions of a Drawing



CATIA-TeamPDM 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 the **SmartTeam Save** option, a link is created between these two documents in **CATIA-TeamPDM**. These linked documents are called *Associated Objects*.

CATIA-TeamPDM provides three types of Associated Objects:

- **Dependency:** A CATIA-TeamPDM Part or a Product is a *dependency* of a **CATIA-TeamPDM** Drawing (since the Drawing is based on the Part/Assembly).
- **Reverse Dependency:** A **CATIA-TeamPDM** drawing is a *reverse dependency* of a **CATIA-TeamPDM** Part/Product.

As you create revisions, you can view and manage these Associated Objects (dependencies or reverse dependencies), using the **Associated Objects/Dependencies** or **Reverse Dependencies** option.



In addition, **CATIA-TeamPDM** color indicates each of the Associated Objects, so that you can clearly identify a document's dependencies and reverse dependencies. The default color settings are:

- Dependencies are displayed in red.
- Reverse dependencies are displayed in blue.

Working with Dependencies and Reverse Dependencies

Each time you perform a life cycle operation on a document, you can view its Associated Objects, meaning its dependencies and reverse dependencies. 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 dependency (the Part/Assembly) is automatically checked out together with the Drawing.



If the administrator set the default to **Copy File**, then the dependencies will be copied to the desktop, not checked out.

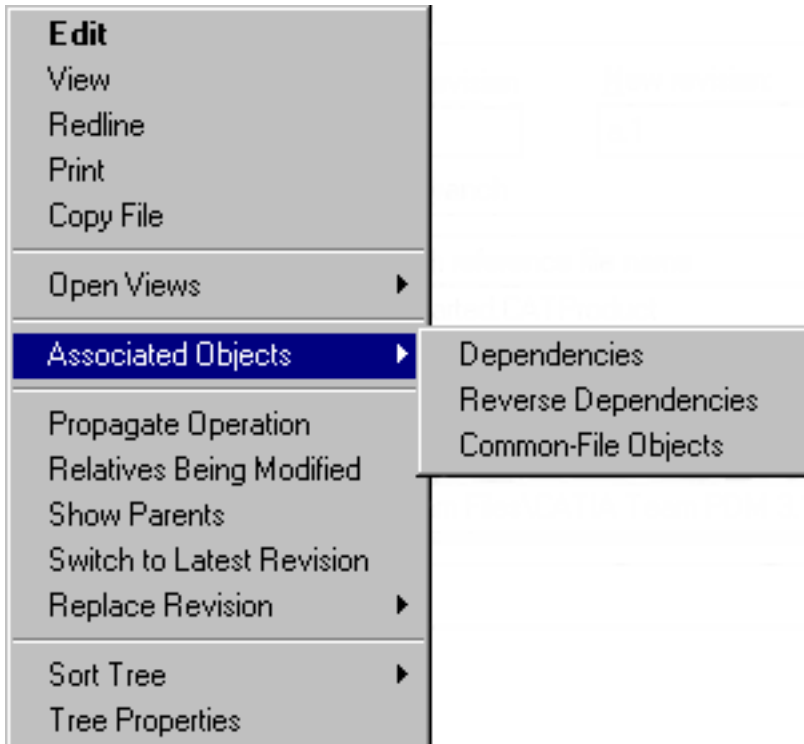
You can view the dependencies of a document and choose a life cycle operation for each dependency. **If you do not view the dependencies of a Drawing, the dependencies will automatically be checked out/in together with the Drawing (if that is the default setting).**

You can view the reverse dependencies of a document and choose a life cycle operation for them. **If you do not view the reverse dependencies of a document, then no life cycle 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 dependency, the **Stump Preacher Guitar Assembly** will be checked out with it (unless you display the dependency and choose a different life cycle operation). When you check out the **Stump Preacher Guitar Assembly**, you may view and check out its reverse dependency, the **Stump Preacher Explode A4 Drawing**.

To view Associated Objects:

1. From any life cycle window (such as *Register* or *Check Out*), right-click on a document to display the dropdown menu.
2. Point to **Associated Objects** to display the list of view options. A checkmark to the left of the option shows that the Associated Object is already displayed in the tree.



3. Click on an option to add or remove the checkmark. Removing the checkmark will remove the display of the Associated Object from the tree. Adding a checkmark will display the Associated Objects in the tree.



The default color settings can be changed, as described in section "[To customize color settings for Associated Objects](#)".

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 dependencies, even if you do not display them. This is not the case for reverse dependencies: You must choose to display reverse dependencies in order to perform life cycle operations on them.

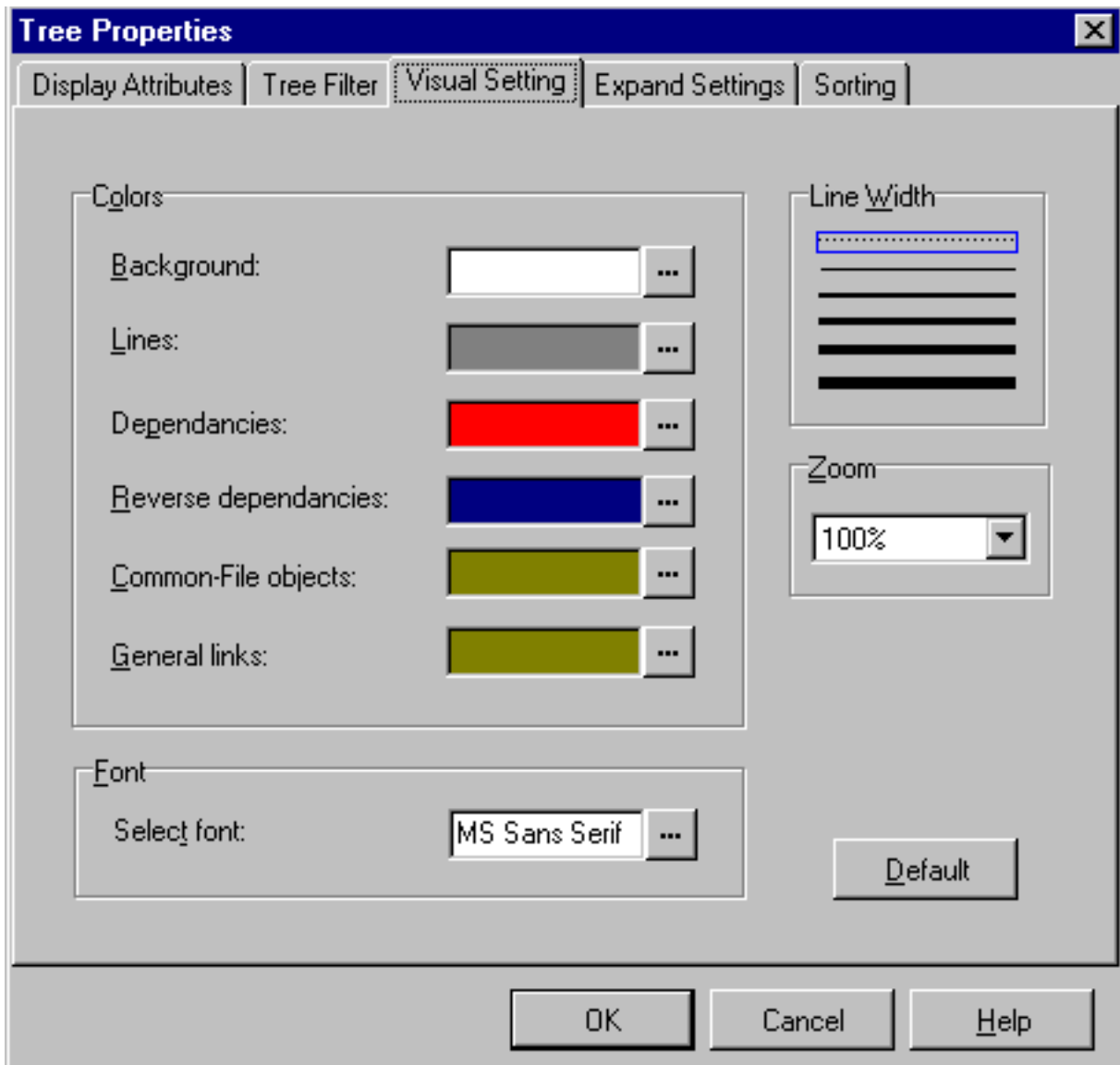
To customize color settings for Associated Objects:

CATIA-TeamPDM color indicates each of the Associated Objects, so that you can clearly view a document's dependencies and reverse dependencies. The default color settings are:

- Dependencies are displayed in red.
- Reverse dependencies are displayed in blue.

You can change these default settings, as described below.

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



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



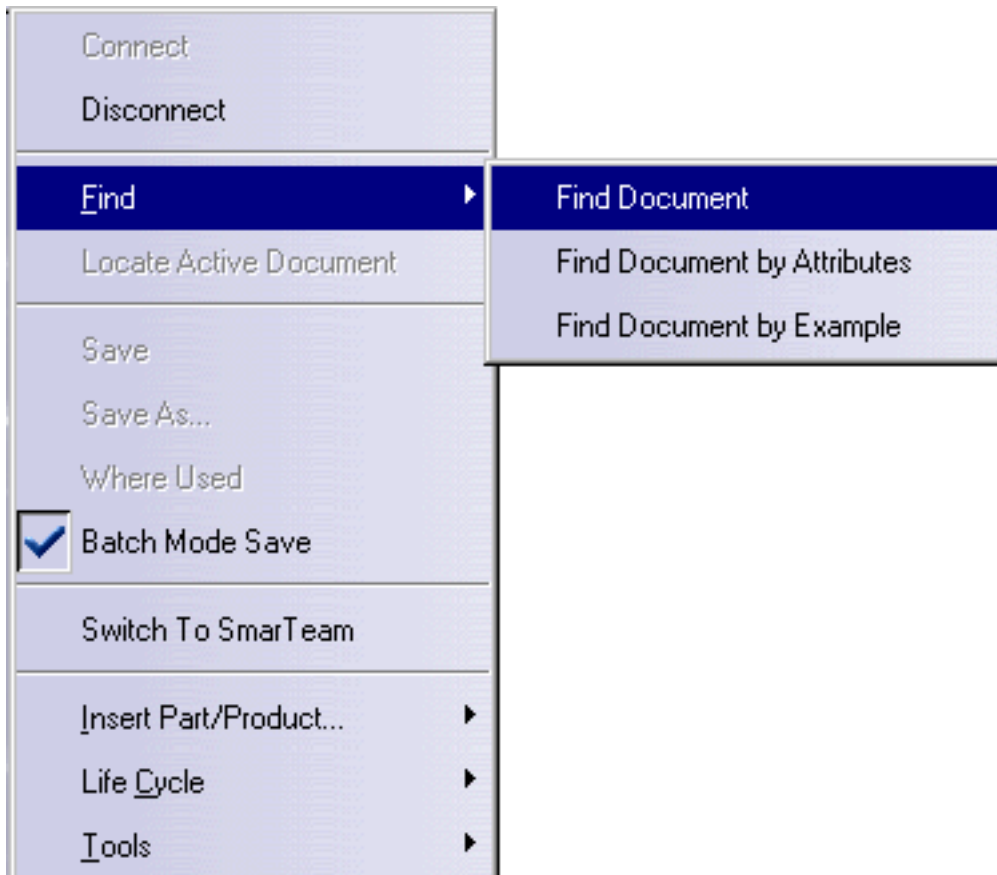
4. Choose a new color and click **OK**.
5. In the *Visual Settings* window, click **OK**.



Searching for Documents

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

CATIA - TeamPDM provides several searching tools:



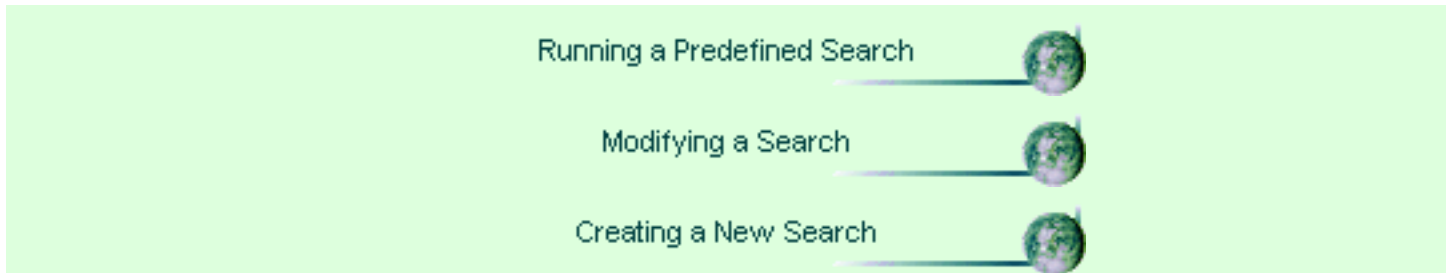
- **Find Document** option: This option enables you to view the previously defined searches. From the *Search Editor* window, you can:
 - Run a previously defined search.
 - Modify a search.
 - Create a new search.

Each search may contain numerous 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 select a document and launch it into CATIA.

- **Find Document By Attributes** option: This option enables you to create a new search. You must choose a **CATIA - TeamPDM** class and enter search criteria for it. You can then run the search to locate the documents in the selected class that match the search criteria.

- **Find Document By Example** option: This option enables you to create an advanced search from multiple classes, based on the attributes in these classes as well as the attributes in the relationship (link) between these classes. You must choose **CATIA - TeamPDM** classes, select attributes and define search criteria. You can then run the search to locate the documents in the selected classes that match the search criteria.

This section provides the following instructions:



Running a Predefined Search



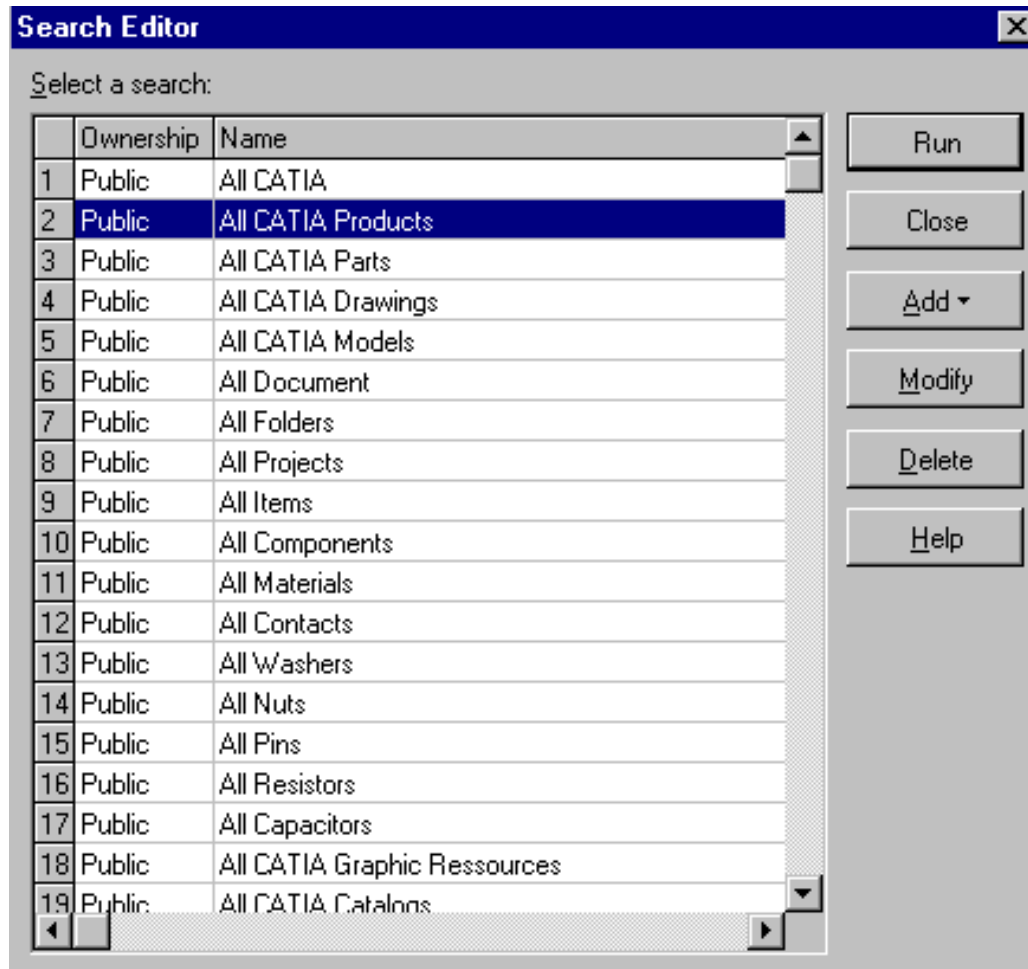
Once a search is defined and saved, you can run it over and over again. This powerful search tool can help you find your documents quickly and efficiently.

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.

After you run a search, the search results are displayed in a search results list. You can browse through the list to locate a specific document. You can then launch the document right into CATIA (by right-clicking on the document and choosing **Edit**).

To run a search:

1. From the *SmarTeam* menu, point to **Find** and choose **Find Document**.
The *Search Editor* window is displayed, as shown below:



From the *Search Editor* window, you can modify, delete or run a previous search, or create a new search.

- Click **Run** to run the selected search and display the search results, as described in step 2.
- Click **Modify** to modify the attributes of a previously defined search, as described in [Modifying a Search](#).
- Click **Add** to create a new search, and point to **By Attributes** or **By Example** to define a new search. Refer to [Creating a New Search](#) for details.
- 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:

All CATIA Products: CATIA Product

Class	ID	Revision	State	Creation Date	Description
1	CATProduct-05	a.0		09/17/1999 1	GettingStarted
2	CATProduct-05			09/17/1999 1	GettingStarted

CATIA Product

Profile Card | Links | Notes | Revision | Viewer

ID

Revision State

Description

Part Number

Engineering Information

Definition

Nomenclature

Source

Description

General / Details / Revision /

User login: joe

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



Modifying a Search



You can modify a search by changing the search criteria. This enables you to customize searches to help you narrow down a search and find the exact documents that you need.

For example: A predefined search displayed in the *Search Editor* window is called **All CATIA Parts**. You can modify this search to locate all CATIA Parts with the word **Cog** in the **Description** attribute or that are **Approved**. In this manner, you can narrow the search to locate the documents that you need.

To modify a search:

1. From the *SmarTeam* menu, point to **Find** and choose **Find Document**. The *Search Editor* window displays the previously defined searches.
2. Choose a search and click **Modify**. The *Search by Attribute* window appears with the current search attributes.
3. Modify the search attributes and click **Run**.



You can save the modified search as a new search: In the *Search by Attribute* window, select the **Save As** button name, then enter the new search in the *Search name* field and click **Save**.



Up

Running a Predefined Search

Modifying a Search



Creating a New Search



Creating a New Search

CATIA - TeamPDM enables you to create your own search by defining search attributes for a specific class of documents. **CATIA - TeamPDM** provides two options for creating new searches:

- **Find Document By Attributes** option: This option enables you to create a search from one class of documents. You must define search criteria for it by entering search information in the attributes fields. You can then run the search to locate the documents in the selected class that match the search criteria.
- **Find Document By Example** option: This option enables you to create an advanced search from multiple classes, based on the attributes in these classes as well as the attributes in the relationship (link) between these classes. You must choose **CATIA - TeamPDM** classes, select attributes and define search criteria. You can then run the search to locate the documents in the selected classes that match the search criteria.

Creating a Search by Attribute



Creating a Search by Example



Up



Running a Predefined Search



Modifying a Search



Creating a New Search

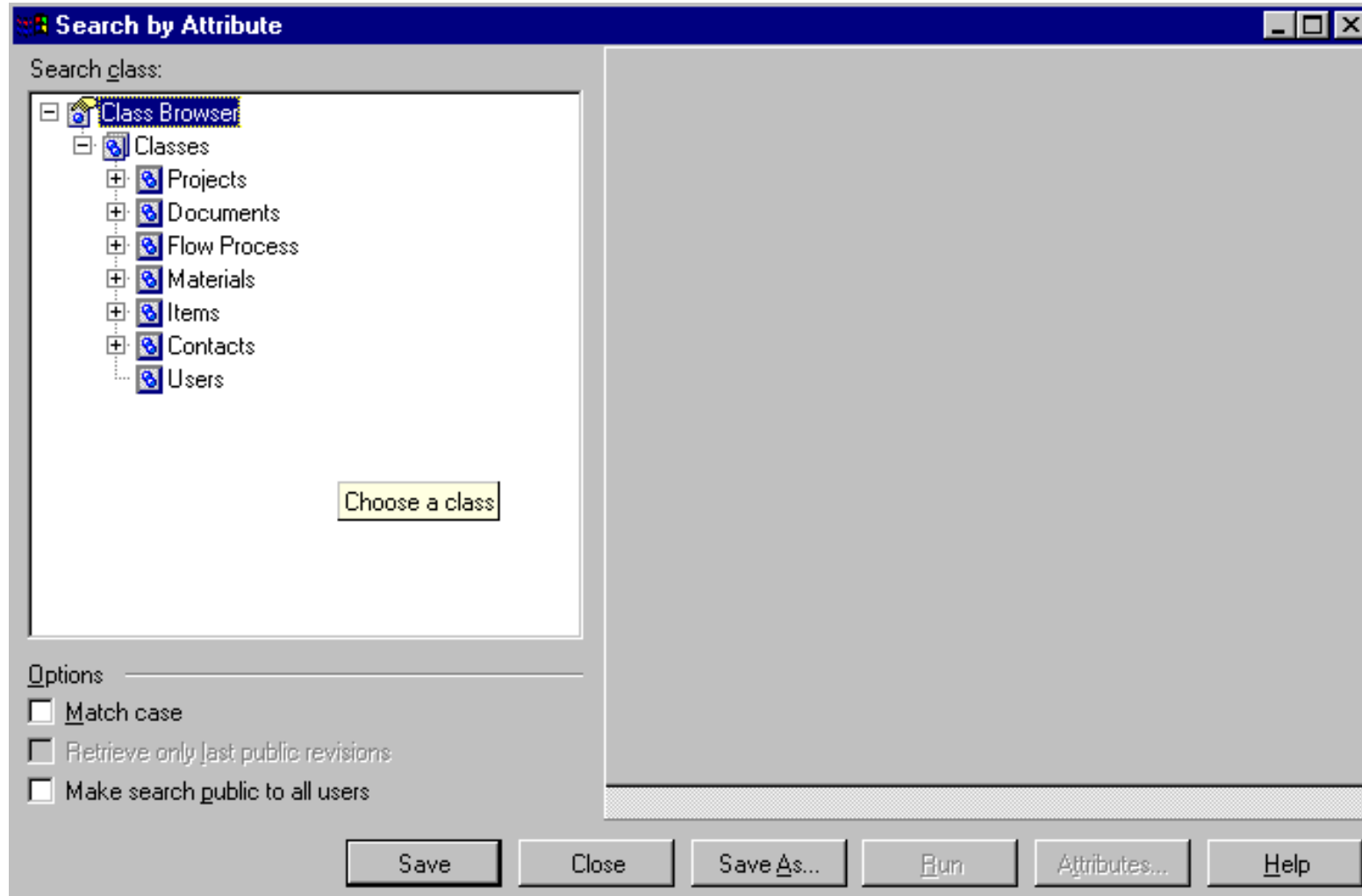


Creating a Search by Attribute



To create a search (by attribute):

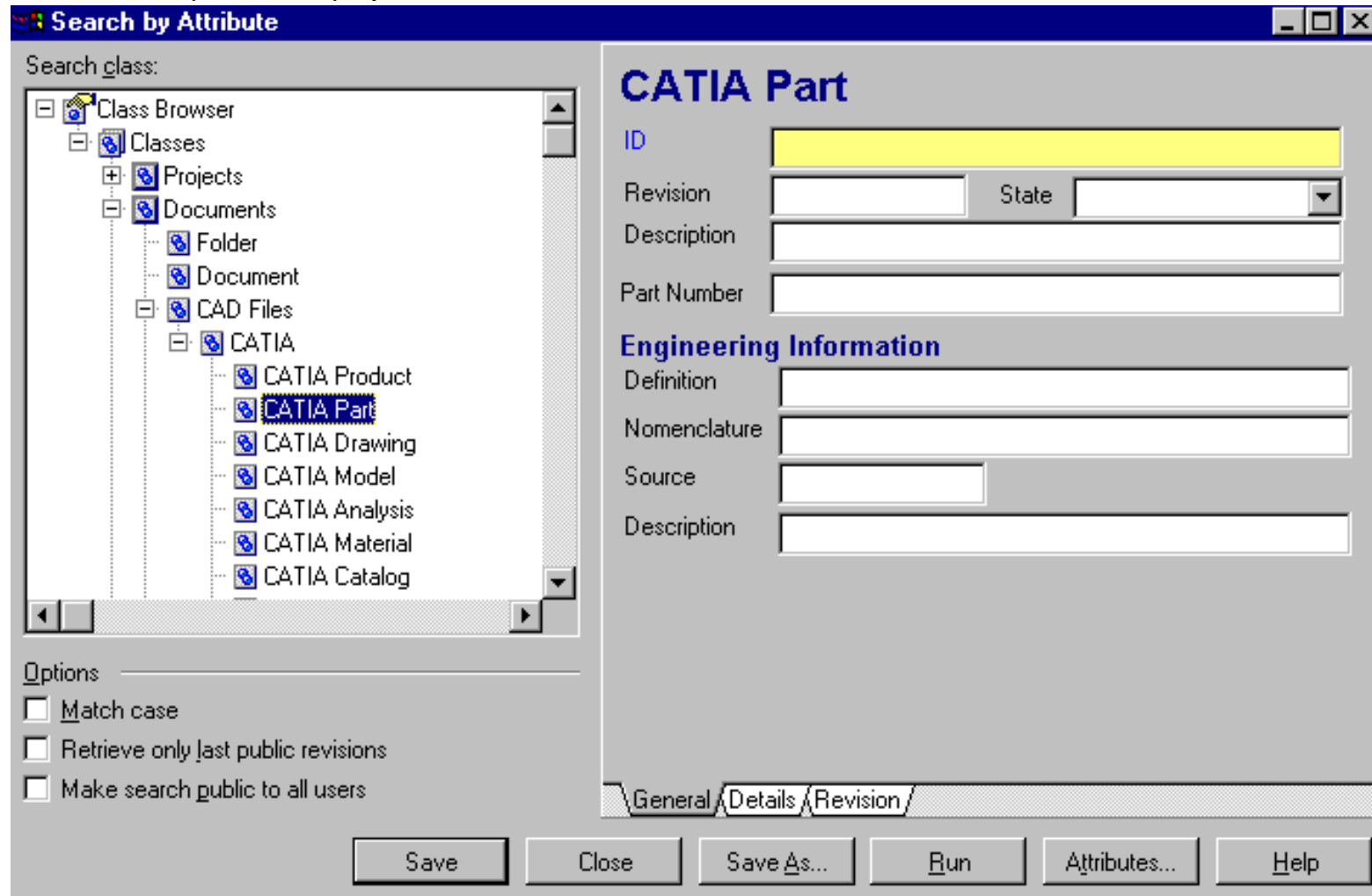
1. From the *SmartTeam* menu, point to **Find** and choose **Find Document By Attributes**. The *Search Details* window is displayed, as shown below:




The *Search Details* window is also displayed when you click the **Add** button in the *Search Editor* window and choose **By Attribute**.

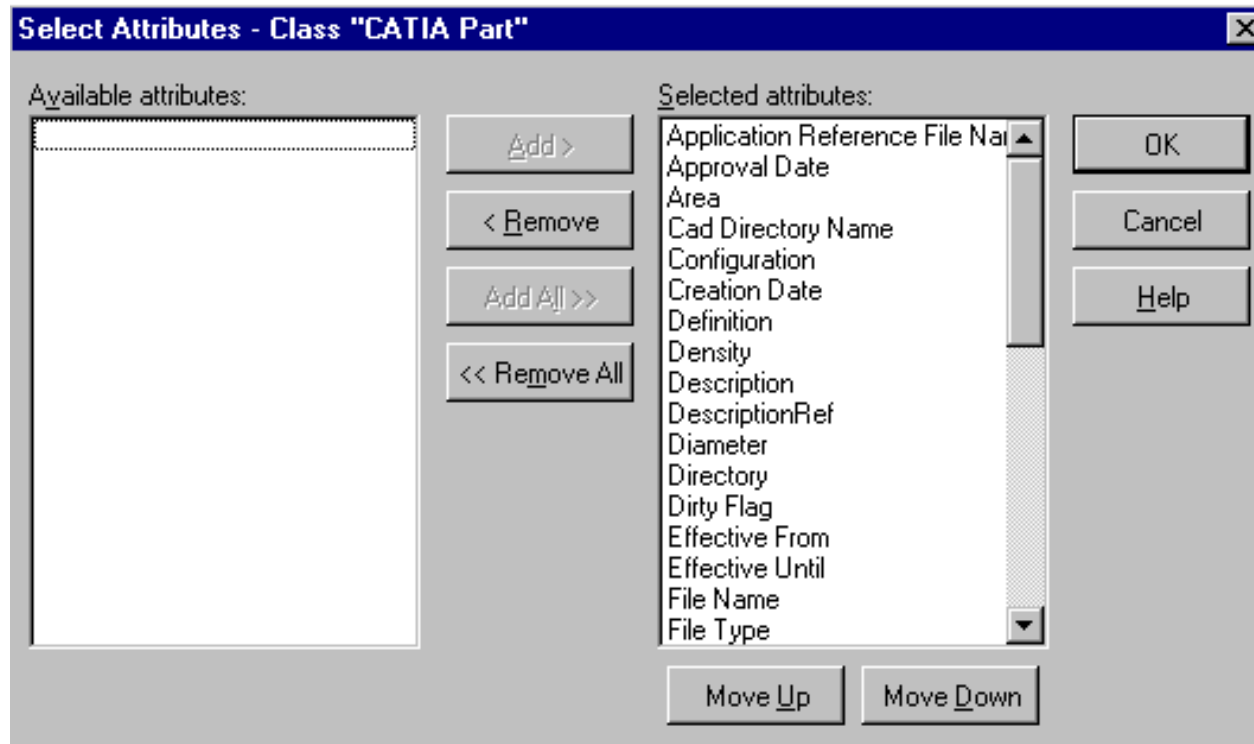
2. Select a class. When you run the search, **CATIA-TeamPDM** will locate the documents that match the search criteria from the selected class.

The window opens to display the Profile Card of the selected class, as shown below:



3. Define search options by checking the appropriate checkboxes.
- In the **Match Case** field, check the checkbox if you want the search results to match cases (lower or upper case) with the information provided in the search criteria fields. This is important since database items are case sensitive.
 - In the **Retrieve Only Last Public** field, check the checkbox if you want the search to display only the last public revision of the documents that meet the search criteria (the latest documents that are checked in or released).

- In the **Make a Search Public for All Users** field, check the checkbox if you wish to make this search public. All users can view and run this search from the *Search Editor* window. If you do not check this checkbox, the search will only appear in the *Search Editor* window for the user who created it.
4. Enter the properties of the search in the displayed Attributes fields. Follow these guidelines:
- If you enter information in two or more fields, the search will locate those documents that satisfy the entered criteria.
 - You can enter multi-field search conditions, enabling more than one value per field. For example, you can enter in the **Project ID** field **1..6|12..18**, which means that the search will find the project IDs that range from 1 to 6 *or* 12 to 18.
 - You can choose to leave these fields blank. The search will then display all the documents in the selected class.
5. Click  to choose those attributes that will be displayed in the resulting *Search Results* window. The following window is displayed:



- The **Attributes List** box shows all the attributes for the class.

- The **Selected Attributes** box shows the selected attributes that will be displayed in the search results list.
 - Use the arrow buttons to move attributes into the **Selected Attributes** list.



- Use the **Move Up/Move Down** buttons to determine the order in which these fields will be displayed.



- Click **OK**.

6. Click **Run** to run the search. The results are displayed in a *Search Results* window. However, the search definition has not yet been saved.

7. To save the search:

- Click **Save** to save the search. This search (as it was named in step 2) will subsequently be displayed in the *Search Editor* window.
- Click **Close** to exit without saving.



Creating a Search by Example



To create a search (by example):

1. From the *SmarTeam* menu, point to **Find** and choose **Find Document By Example**. The *Search By Example* window is displayed, as shown below:

Search By Example

Class tree:

- root
 - Classes
 - Projects
 - Project
 - Documents
 - Folder
 - Document
 - CAD Files
 - CATIA
 - CATIA Product**
 - CATIA Part
 - CATIA Drawing
 - CATIA Model
 - CATIA Analysis
 - CATIA Material
 - CATIA Catalog
 - CATIA Graphical Ressou
 - Offline Document
 - Flow Process
 - General Process
 - Engineering Change
 - ECP

Participating classes:

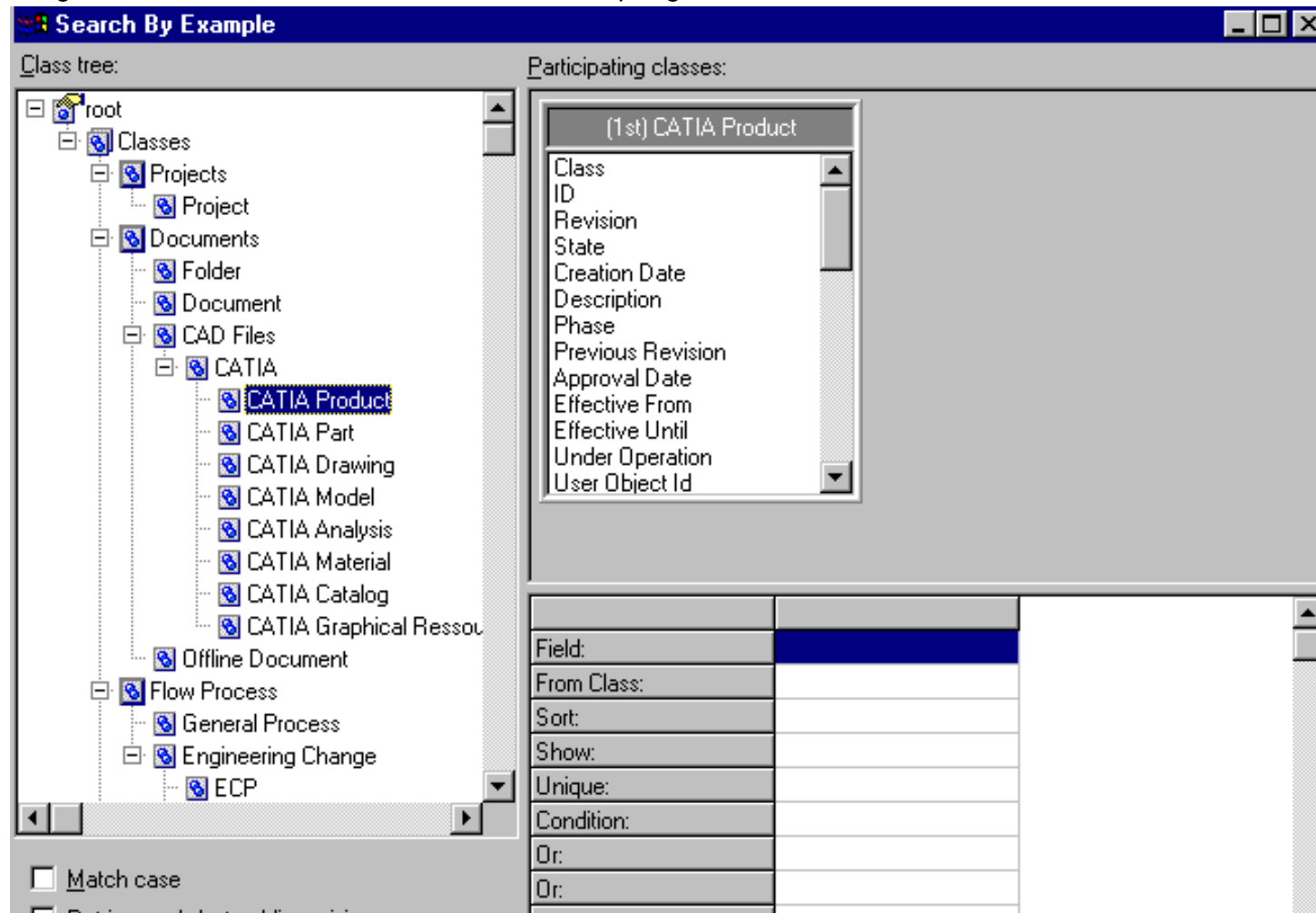
Field:	
From Class:	
Sort:	
Show:	
Unique:	
Condition:	
Or:	
Or:	
Or:	
Or:	
Or:	
Or:	
Or:	

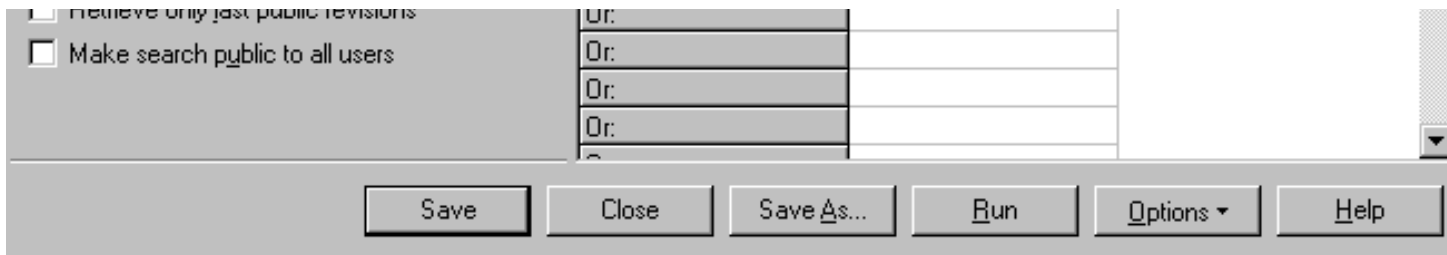
Match case
 Retrieve only last public revisions
 Make search public to all users

Save Close Save As... Run Options Help

2. Name the search in the **Search Name** field.
3. Define search options by checking the appropriate checkboxes:
 - In the **Match case** field, check the checkbox if you want the search results to match cases (lower or upper case) with the information provided in the search criteria fields. This is important since database items are case sensitive.
 - In the **Retrieve only last public revisions** field, check the checkbox if you want the search to display only the last public revision of the documents that meet the search criteria (the latest documents that are checked in or released).
 - In the **Make search public for All users** field, check the checkbox if you wish to make this search public. All users can view and run this search from the *Search Editor* window.

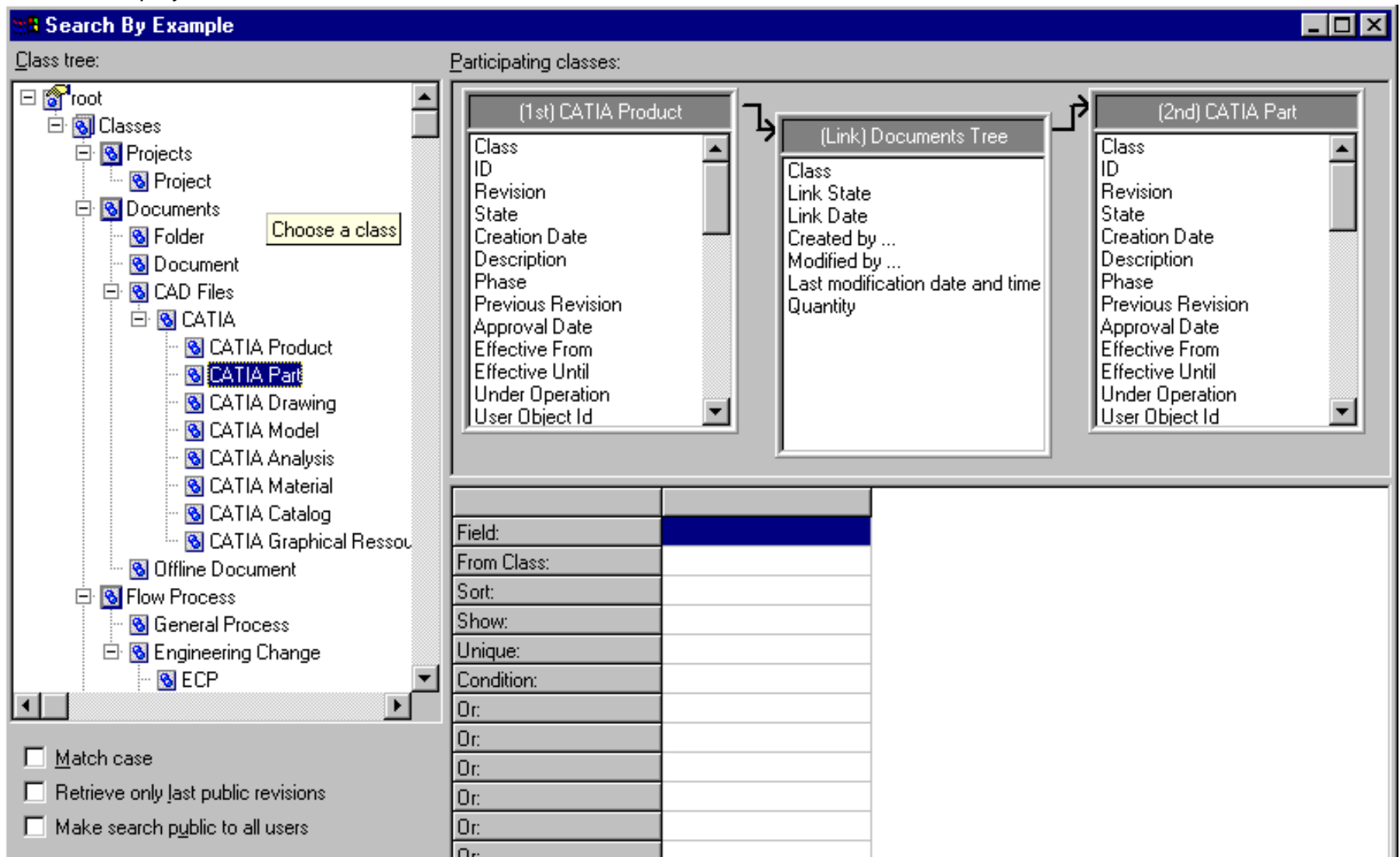
If you do not check this checkbox, the search will only appear in the *Search Editor* window for the user who created it.
4. From the tree on the left side of the window, choose a class.
5. Drag the class to the work area, located at the top, right area of the window. The attributes of the class are displayed in a list.





6. You can select a second class as follows:

From the tree on the left side of the window, choose a second class, and drag it to the work area. The attributes of the second class are displayed in the work area as well as the attributes of the link between the two classes.



Or:

Save

Close

Save As...

Run

Options ▾

Help



In the Link Attribute list, you can display general link attributes or the hierarchical link attributes: Right-click on the list and choose the appropriate option.

7. From an attribute list, select an attribute and drag it to the grid at the bottom, right area of the window.

Search By Example

Class tree:

- root
 - Classes
 - Projects
 - Project
 - Documents
 - Folder
 - Document
 - CAD Files
 - CATIA
 - CATIA Product
 - CATIA Part**
 - CATIA Drawing
 - CATIA Model
 - CATIA Analysis
 - CATIA Material
 - CATIA Catalog
 - CATIA Graphical Ressou
 - Offline Document
 - Flow Process
 - General Process
 - Engineering Change
 - ECP

Participating classes:

(1st) CATIA Product

- Class
- ID
- Revision
- State**
- Creation Date
- Description
- Phase
- Previous Revision
- Approval Date
- Effective From
- Effective Until
- Under Operation
- User Object Id

(Link) Documents Tree

- Class
- Link State
- Link Date
- Created by ...
- Modified by ...
- Last modification date and time
- Quantity

(2nd) CATIA Part

- Class
- ID
- Revision
- State
- Creation Date
- Description
- Phase
- Previous Revision
- Approval Date
- Effective From
- Effective Until
- Under Operation
- User Object Id

Field:	State
From Class:	1st
Sort:	
Show:	✓
Unique:	
Condition:	Released
Or:	Checked In
Or:	
Or:	
Or:	
Or:	
Or:	
Or:	

Match case

Retrieve only last public revisions

Make search public to all users



The selected attribute is displayed in the **Field** row.

8. Define search conditions for the attribute as follows:

- Place a checkmark in the **Sort** row to sort the search results list according the selected attribute.
- Place a checkmark in the **Show** row to show the selected attribute in the search results list.
- Place a checkmark in the **Unique** row to filter the search results so that only unique documents are displayed: If more than one document meets the specified conditions, only the first document is displayed.
- Enter search criteria in the **Condition** row for the selected attribute.
For example: For the CATIA Product class, the **State** attribute is dragged to the grid. In the **Condition** field, you enter **Released**. The search will find the Products whose state is Released.
- Enter additional search criteria in the **Or** field.
For example: Enter **Checked In** in the **Or** field. The search will find those Products whose state is **Released** or **Checked In**.

9. Select additional attributes and define search conditions for them. When you run the search, **CATIA-TeamPDM** will find those documents that meet the search criteria for all the attributes displayed in the grid (meaning an **And** condition exists between each column of attributes).

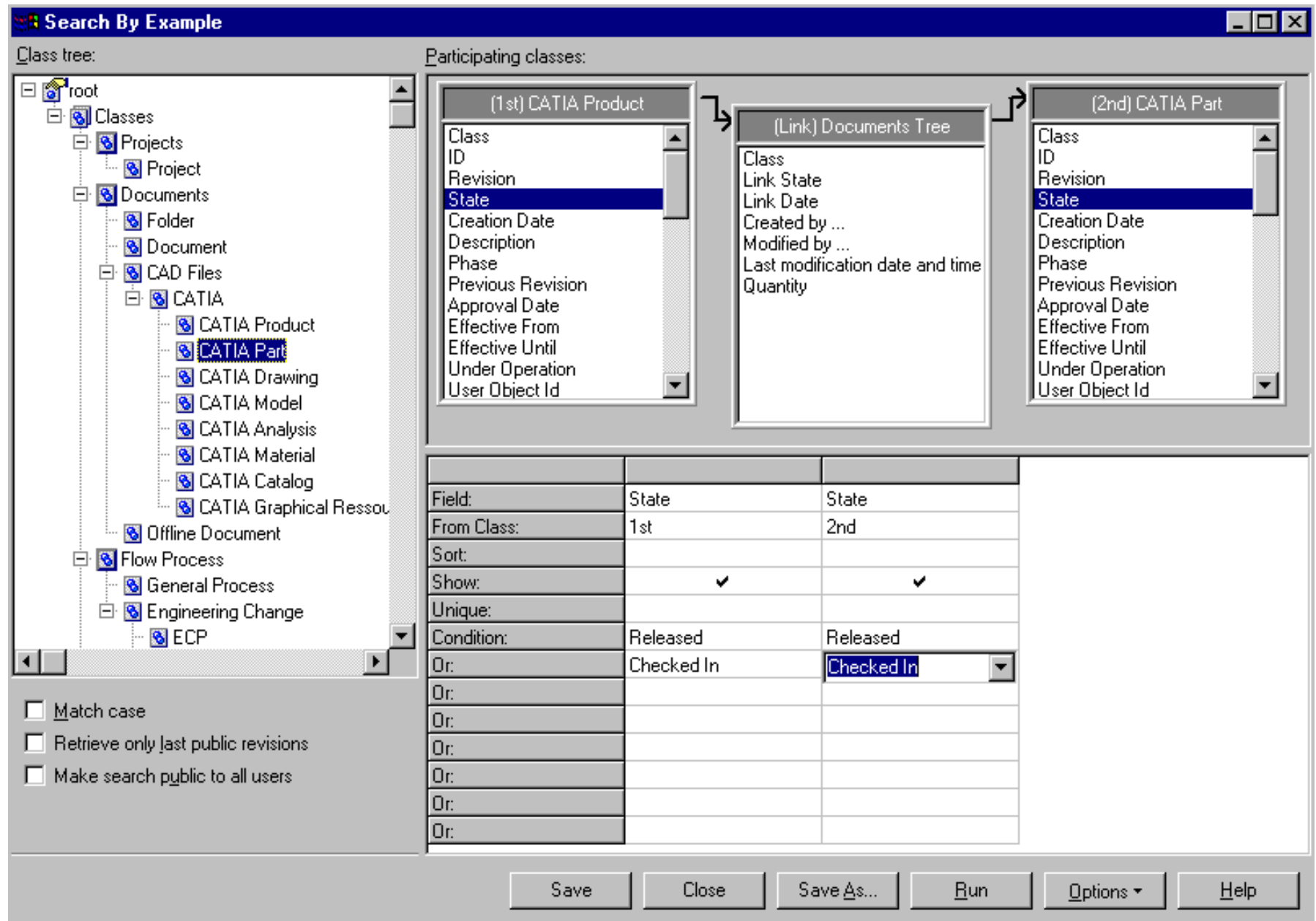
For example:

- From the CATIA Product class, the **State** attribute is selected and dragged to the grid. In the **Condition** field, you enter **Released**. In the **Or** field, you enter **Checked In**.
- From the CATIA Product class, the **Description** attribute is selected and dragged to the grid. In the **Condition** field, you enter ***cog**.
- When the search is run, the search results list will display those documents from the CATIA Product class whose state is **Released** or whose state is **Checked In and** whose description includes the word **cog**.

If you select attributes from more than one class, the search will locate only those documents that meet the search conditions for both classes.

For example:

- From the CATIA Product class, you drag the **State** attribute to the grid and define the **Condition** as **Checked In**. In the **Or** field, you enter **Released**.
- From the CATIA Part class, you drag the **State** attribute to the grid and define the **Condition** as **Checked In**. In the **Or** field, you enter **Released**.



When the search is run, the search results list will display those Products that are released or checked in whose Parts are also released or checked in.

10. After the search is defined:

- Click **Run** to run the search. The results are displayed in a search results list. You can browse through the documents in the list to display the Profile Card for each document. You can also launch a document directly into CATIA.
- Click **Save** to save the search. This search (as it was named in step 2) will subsequently be displayed in the *Search Editor* window.
- Click **Close** to exit without saving.



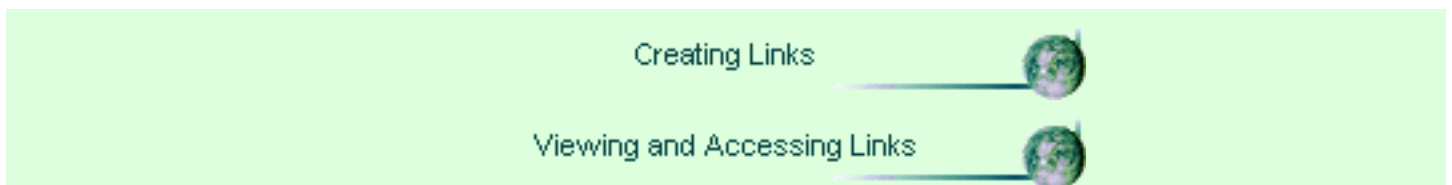
Creating and Viewing Links

CATIA-TeamPDM enables you to create links between Parts, Drawings and Products. You can view these links in the *Links* page and access the linked documents. These links enable you to:

- Define the hierarchical structure of a Part, Drawing or Product by linking it to a project or parent.
- Locate related documents in order to view or modify them. For example, if you modify a Part, you can locate and modify all its parents as well as other documents linked to it (such as a spreadsheet or a proposal).
- Check a Product into and out of the vault for modification together with its sub-Products. The hierarchical links protect the integrity of the Assembly by reflecting the hierarchical CATIA structure.
- Check a Drawing into and out of the vault for modifications together with the Part or Product on which it was based (and vice versa).

CATIA-TeamPDM automatically creates the following links:

- When you create a Product (using the **Insert Component** option) and save it (using the **SmarTeam/Save** option), the Product and all its children are saved in a hierarchical structure. A parent-child link is created between the Product and each of its components.
- When you create a Drawing based on a specific Part or Product, the Drawing is linked to the Part or Product on which it was based.



Creating Links

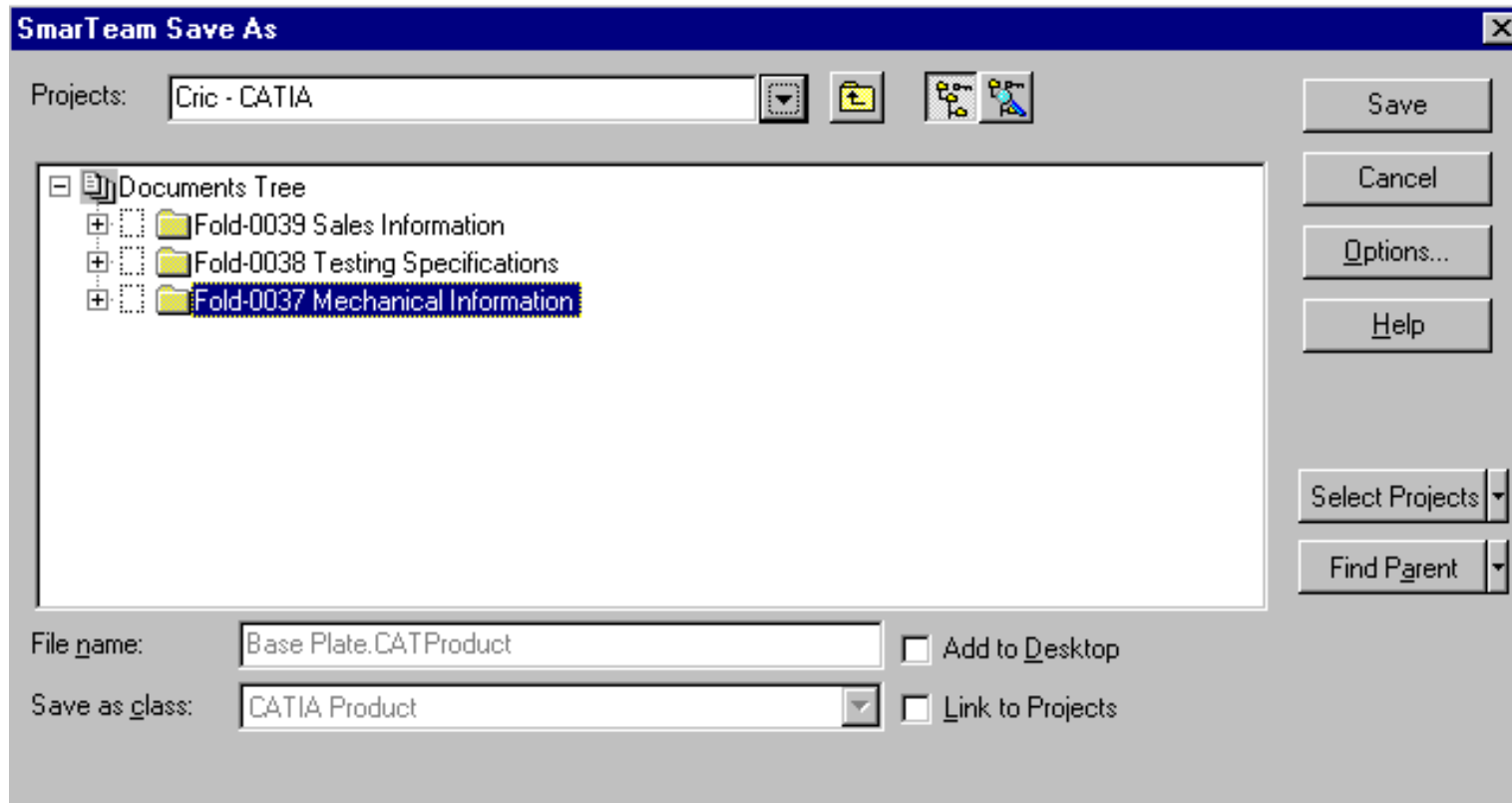


You can create general links between documents, regardless of their class or hierarchical structure. These links provide a quick tool for finding related documents. For example, you may wish to link an Assembly to a marketing document. The general links are displayed in the **Documents** tab of the *Links* page. Refer to [Creating General Links](#) for instructions.



CATIA-TeamPDM enables you to link any document to a project or a parent in the *Save As* window. This window is displayed:

- When you first save a document (using the **Save** or **Save As** option).
- and
- When you choose the **Save As** option for subsequent saves.



A Project Tree is displayed when you click the dropdown arrow to the right of the Project box. In this tree, you can select a project.

The Object Tree is displayed in the middle of the window. You can expand and collapse the tree to display its branches. In this tree, you can select a parent for the document, such as an Assembly or a Folder.

CATIA-TeamPDM enables you to modify the displayed view of these trees in several ways:

- Define the level of sub-branches displayed in the trees, as described below.
- Define the general tree properties, such as whether to display all revisions or the last public revisions. These properties are defined in the Documents tree of a project when you right-click and choose Tree Properties.

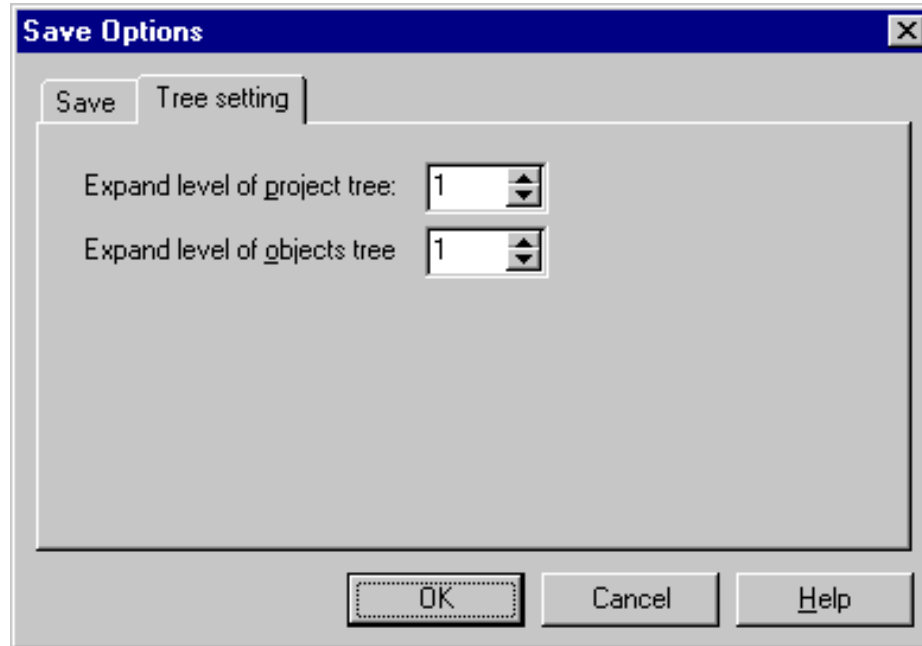


From the *SmarTeam* menu, choose **Switch to SmarTeam** and locate the Documents Tree for a specific project.

For example, in the **Tree Filter** tab of the *Tree Property* window, you can define whether to display all revisions of the documents in a tree, or only the last public revisions of the documents in a tree.

To define the level of sub-branches displayed in the **Save As** window:

1. From the *SmarTeam* menu, choose **Save As** to display the *Save As* window.
2. Click the **Options** menu to display the *Save Options* window. Click the **Tree settings** tab.



3. Define the level of branches that will be displayed in the Project Tree and the Object Tree.

Creating General Links

You can create general links between documents, regardless of their class or hierarchical structure. These links provide a quick tool for finding related documents. For example, you may wish to link a Product to a marketing document. The general links are displayed in the **Document** tabs of the *Links* page.

To create a general link:

1. Display the document in a **CATIA-TeamPDM** window:
 - From CATIA, click on the *SmarTeam* menu and choose **Locate Active Document**.
or
 - Run a search to locate the document.
2. Open the *Links* page of the document to which you want to link another document.
3. In the *Links* page, select a tab (**Projects** or **Documents** for example) from the bottom of the window to display a specific *Links* sub-page. This *Links* sub-page must display the same class as the document that you will choose in step 4.
4. Open any **CATIA-TeamPDM** window, and select the document that you wish to link to the document selected in step 1.
5. Drag and drop the selected document (selected in step 3) to the *Links* page of the document selected in step 1 to create a link.
6. A message confirming the link is displayed.
Click **Yes** to confirm a single link.
Click **All** to confirm multiple links.
The *Links* page is automatically updated to reflect these additional links.



Viewing and Accessing Links



CATIA-TeamPDM enables you to view the various links of a document by clicking through the *Links* sub-pages. You can then access a linked document, view its Profile card and launch it into CATIA.

- The **Composed Of** sub-page lists all the children (components) of a document.
- The **Where Used** sub-page for the Product lists all the parents (folders and Products) of a document.



The **Where Used** list is also displayed when you select the *SmarTeam* menu and then choose the **Where Used** option.

- The various class tabs display the general links of a document (according to classes).

To view links:

In the *Links* page, you can view the links of any **CATIA-TeamPDM** document as follows:

1. Select a document from any **CATIA-TeamPDM** window.



If the document is currently displayed in a CATIA window, you can choose the **Locate Active Document** option from the *SmarTeam* menu

2. Click the **Links** tab at the top of the Profile Card to display the *Links* page.
3. Click the appropriate tab at the bottom of the page to display a sub-page of links:
 - The **Composed of** sub-page lists all the children of a document.

Class	ID	Revision	State	Creation Date	Description	Phase
1	CATPart-0569			09/17/1999 19:13	CRIC_FRAME	Default
2	CATPart-0570			09/17/1999 19:13	CRIC_BRANCH_3	Default
3	CATPart-0571			09/17/1999 19:13	CRIC_BRANCH_1	Default

- The **Documents** sub-page lists the link between a Drawing and a Part or Product. It also lists the general links for the Part, Drawing or Product classes. The remaining sub-pages list the links according to classes.
- The **Where Used** sub-page lists all the parents of a document. These links are extremely useful in locating all the documents to which a specific document is linked as a child. For example, if you have modified a Part, you can quickly access all the Products which contain this Part as a sub-Assembly, and modify them accordingly.



You can also view the parents of a document by selecting the *SmartTeam* menu, then choosing the **Where Used** option, as shown below.

CATIA Team PDM: Parents of CATIA Part "CATPart-0569"

Class	ID	Revision	State	Creation Date	Description	Phase
1	CA			09/17/1999 19:1	GettingStarted	Default
2	CA	a.0		09/17/1999 19:2	GettingStarted	Default
3	CA	a.1		09/20/1999 11:0	GettingStarted	Default

Profile Card | Links | Notes | Revision | Viewer

CATIA Product

ID: CATProduct-0520

Revision: State: Checked In

Description: GettingStarted

Part Number: Product1

Engineering Information

Definition:

Nomenclature:

Source: unknown

Description:

General / Details / Revision / Composition

User login: joe

To access linked documents:

CATIA-TeamPDM enables you to browse through and launch into CATIA any of the documents listed on the *Links* page.

1. In the *Links* page, right click to display a drop-down menu.
2. Point to **Open Views** and then click **Separate View**. A window is displayed which lists all the links of the selected Part on the left and displays pages on the right.

