Student Notes:

SMARTEAM Training
Foils

SMARTEAM- CATIA Integration

Version 5 Release 14
September 2004

EDU-SMT-E-TDI-FF-V5R14

Student Notes:

SMARTEAM - CATIA Integration

Objectives of the course

At the end of this course you will be able to store and retrieve the CATIA V5 documents and also analyze the links between them in SMARTEAM, during the entire lifecycle of the Product.

Targeted audience

CATIA V5 Users

Prerequisites

CATIA Basics, SMARTEAM - Editor User course

1.0 day

Student Notes:

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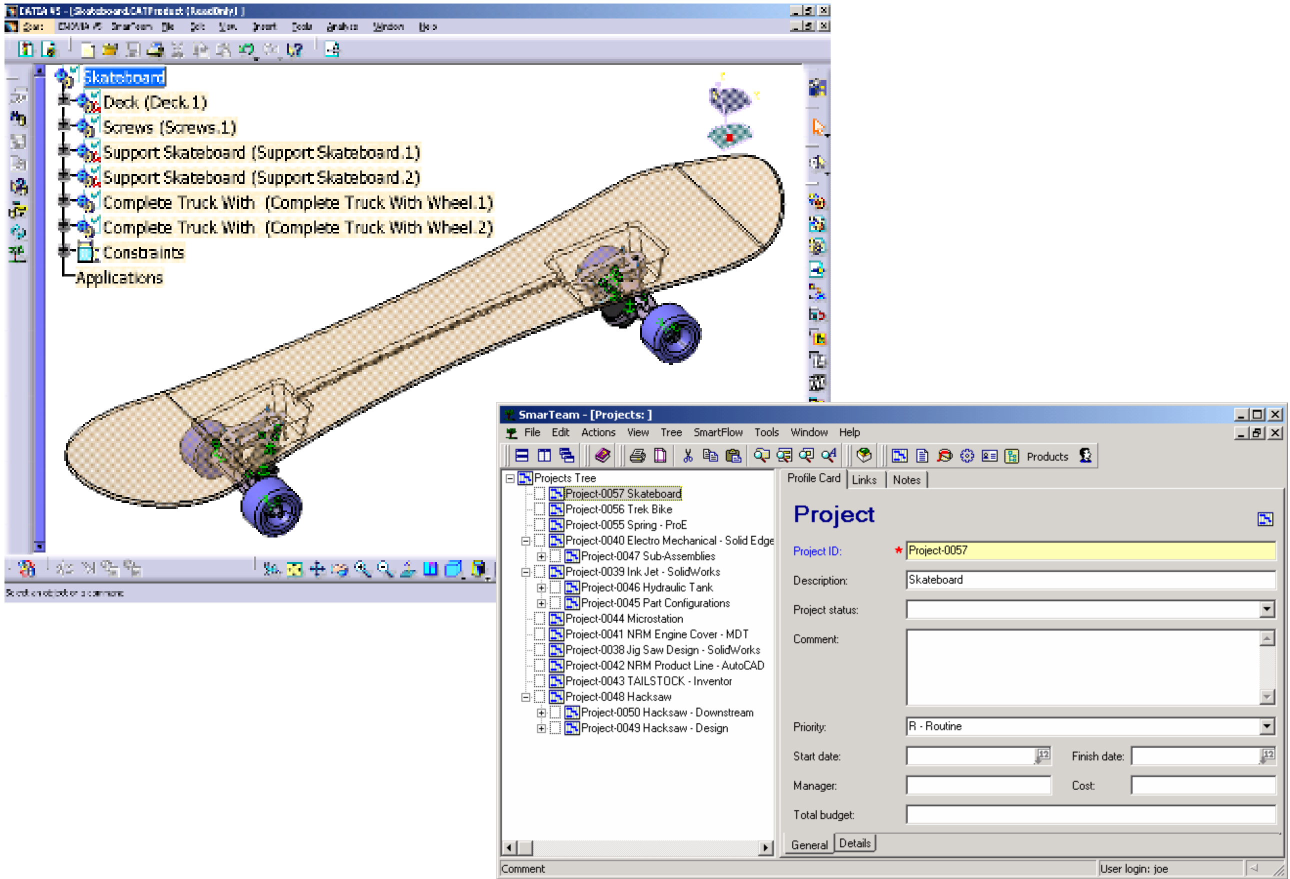
Student Notes:

Introduction to CATIA Integration

You will become familiar with the concepts used in SMARTEAM - CATIA Integration.



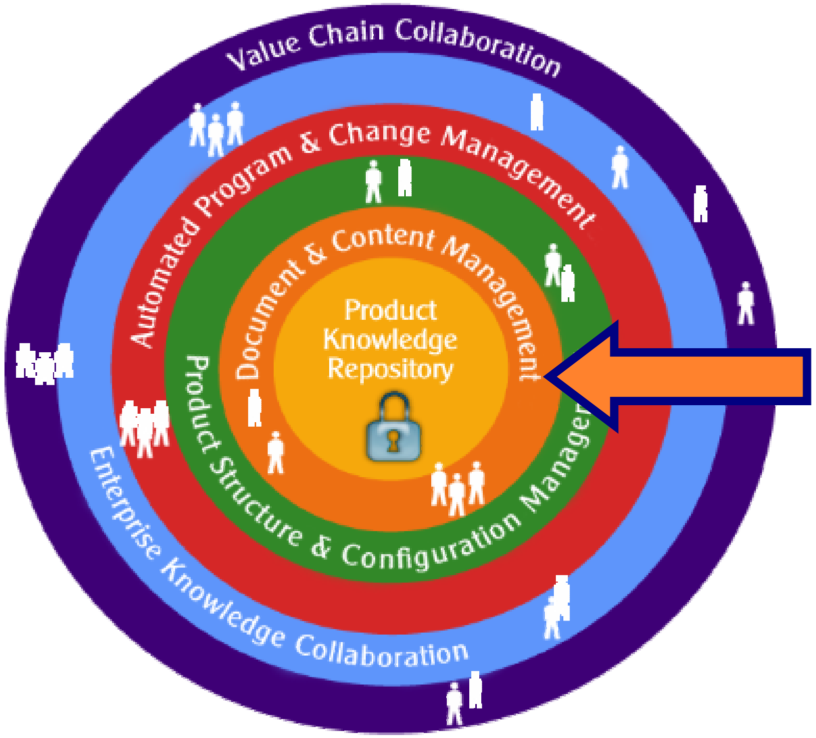
Student Notes:

SMARTEAM Industry-driven PLM Solutions

SMARTEAM has tight integrations with various authoring tools among the CAD legacy systems which makes it attractive to design-centric organizations.

By efficiently consolidating product data, facilitating data access and exchange and optimizing the processes to promote collaboration and reuse, SMARTEAM drives value from the earliest stage of product development.

In the coming chapter you will learn more about the CATIA Integration which is one of the more advanced one.



The capture, management and reuse of product IP through embedded, robust multi-CAx integrations, empowered with comprehensive search and collaboration functionality

Student Notes:

What is SMARTEAM - CATIA Integration

The SMARTEAM - CATIA integration (CAI) is the cornerstone of V5 Concurrent Engineering, enabling all Engineering disciplines to collaborate on product design, optimization and lifecycle management in an easy, dynamic and secured environment.

Here is what SMARTEAM's V5 PLM solutions can deliver around CAI to ensure that right users have the right information at the right time:

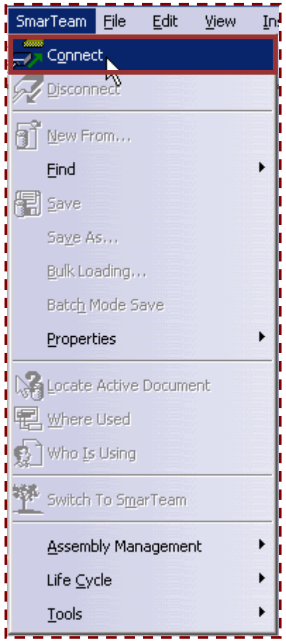
- Product knowledge Capture
  - Maximizing product and knowledge re-use
  - Remove Redundant activities
  - Capture product knowledge
- Collaboration
  - Real time co-design collaboration through the vault
  - Local team collaboration through shared workspaces.
  - Distributed world wide product design collaboration through the vault
  - PLM on demand inside and outside the office.
- Concurrent multi-disciplinary product optimization
  - FEM Analysis
  - Virtual simulations
  - Manufacturing
  - Technical publications

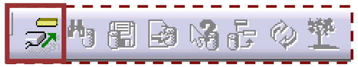
Student Notes:

The SMARTEAM - CATIA Integration Menu (1/2)

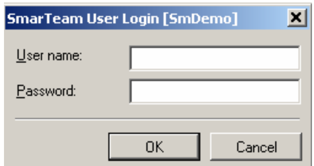
With the SMARTEAM-CATIA license installed and configured, you can connect to the SMARTEAM database as indicated below:

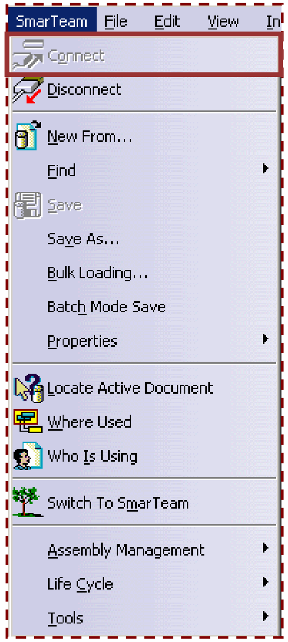
From the 'SmarTeam' menu, select Connect. **OR** **In the SMARTEAM toolbar, click on the Connect icon.**





Enter User Name and Password in the 'SmarTeam User Login' dialog box.

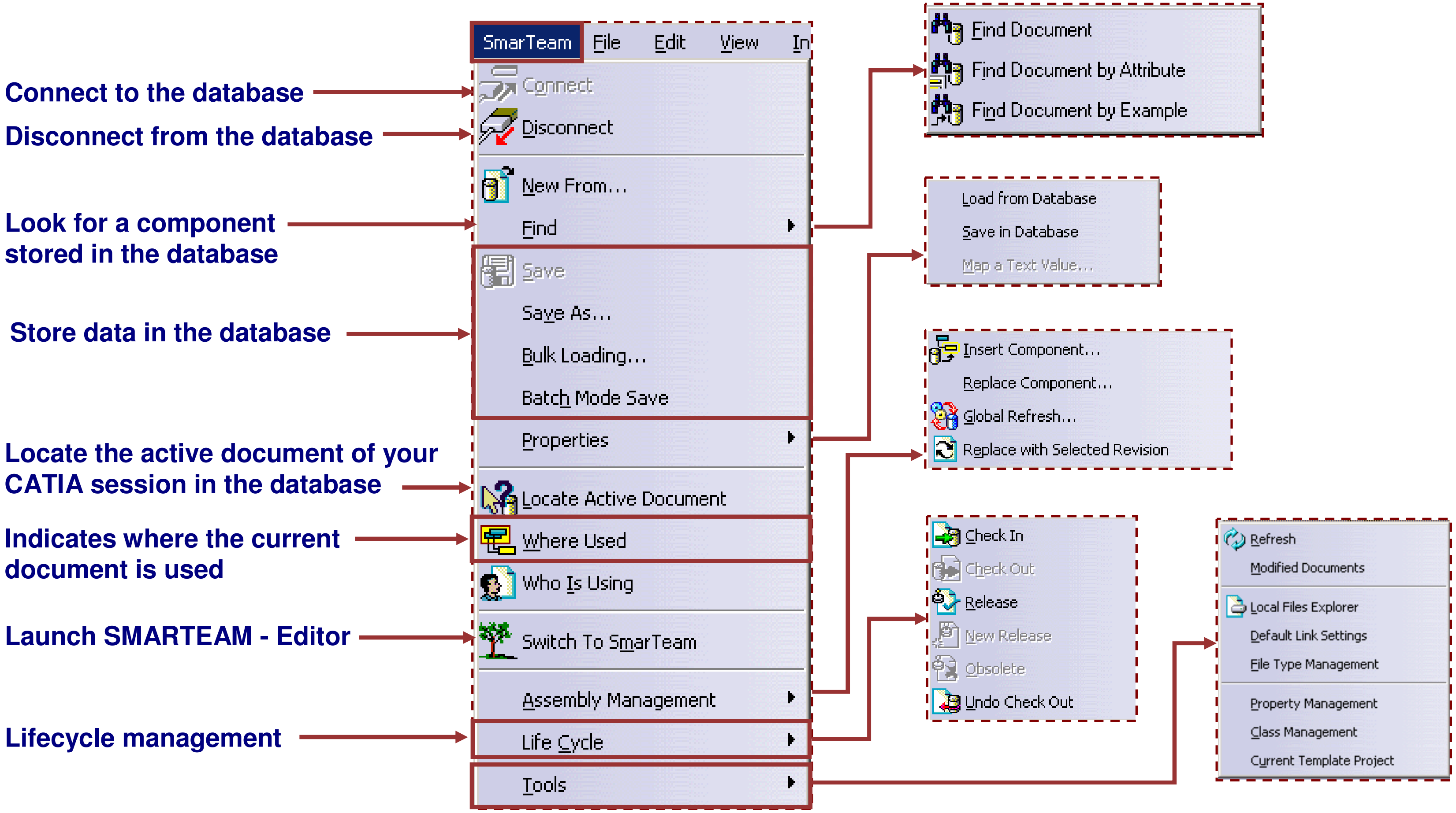




The Connect icon is disabled, and the SMARTEAM - CATIA Integration functions are activated.

Student Notes:

The SMARTEAM - CATIA Integration Menu (2/2)

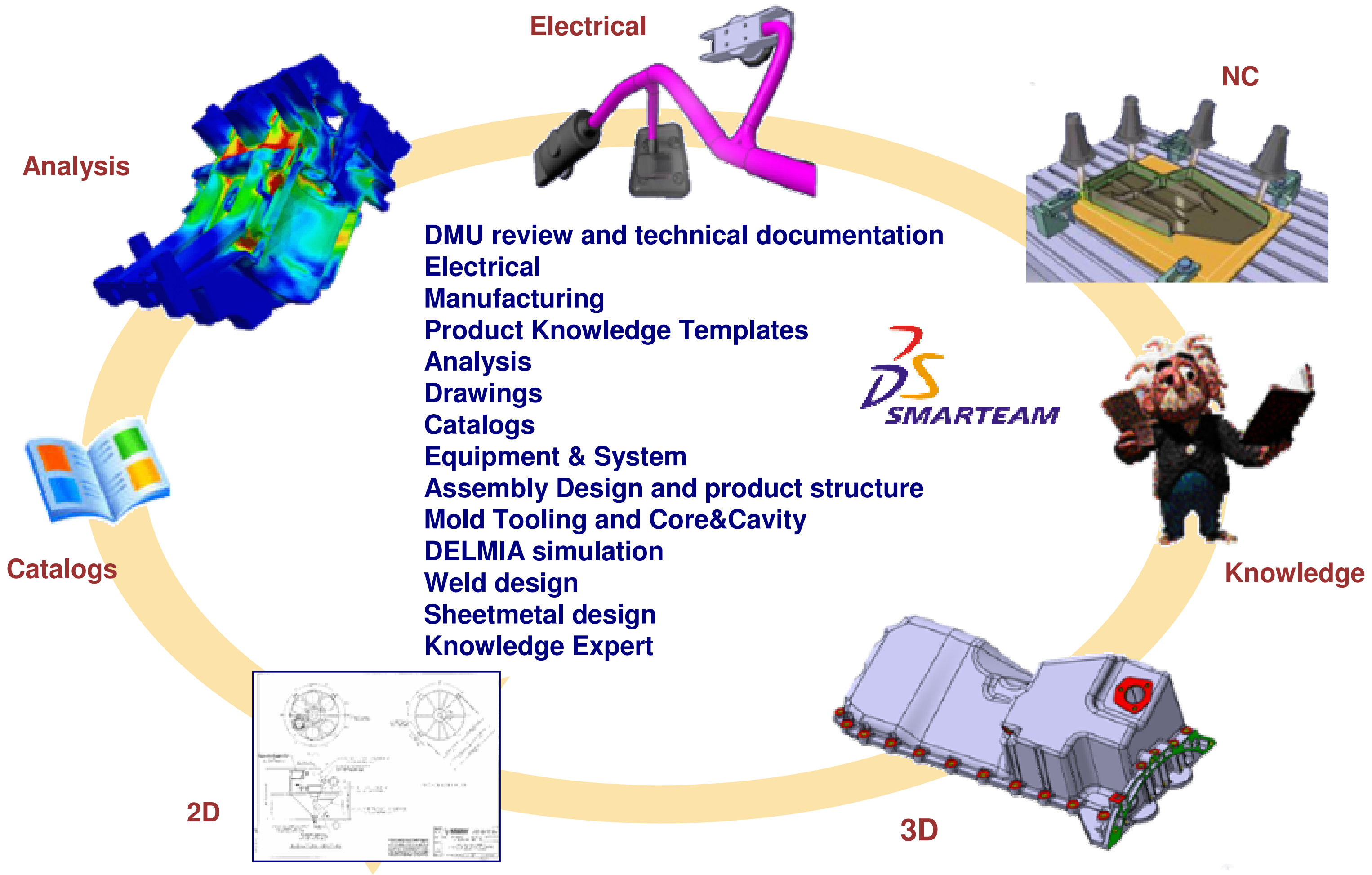


'SmarTeam' menu in CATIA V5

Student Notes:

Power of SMARTEAM - CATIA Integration

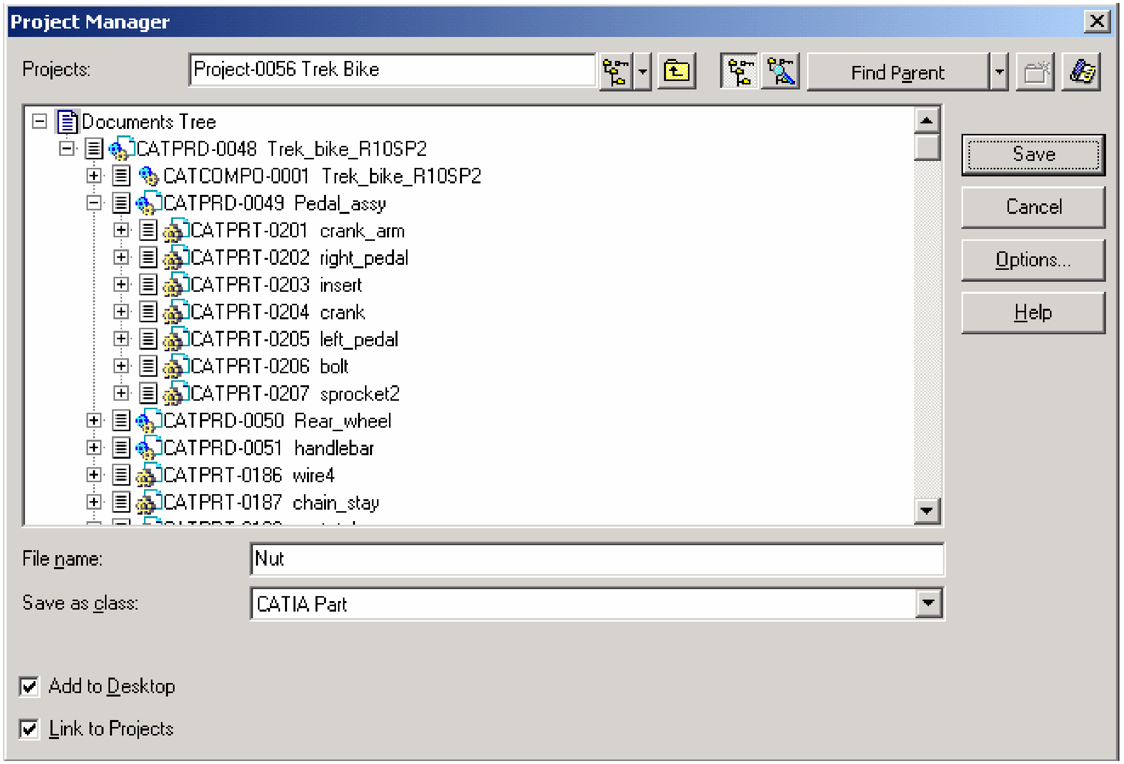
SMARTEAM - CATIA integration intuitively supports multi-discipline concurrent engineering.



Student Notes:

Saving CATIA V5 Documents

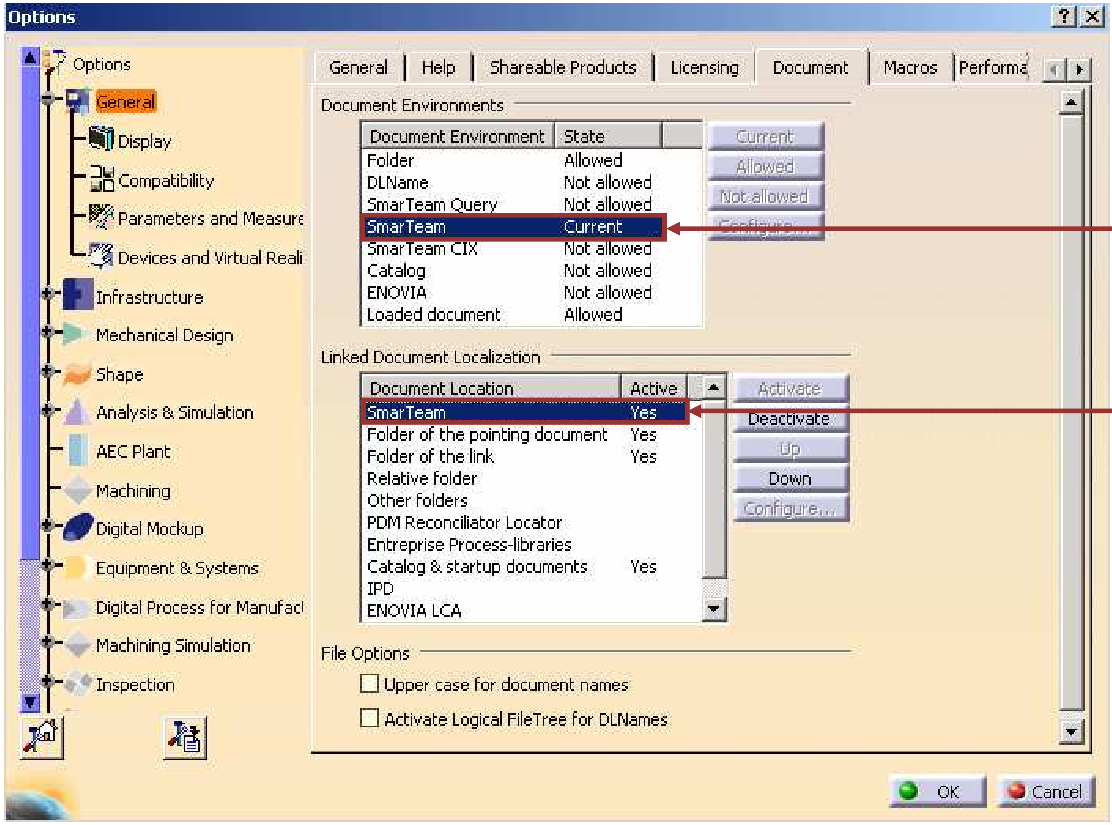
You will discover the methods to save CATIA objects in SMARTEAM.



Student Notes:

SMARTEAM - CATIA Integration Settings

The recommended settings for using SMARTEAM with CATIA can be put in place through Tools > Options > Document tab.



Set 'SmarTeam' as the allowed and current Document Environment.

Set 'SmarTeam' in the active state document localization by selecting Yes, and as the first priority using the Up button. Deactivate other options in the list as indicated in the adjoining screenshot and maintain the sequence with the Up and Down buttons.

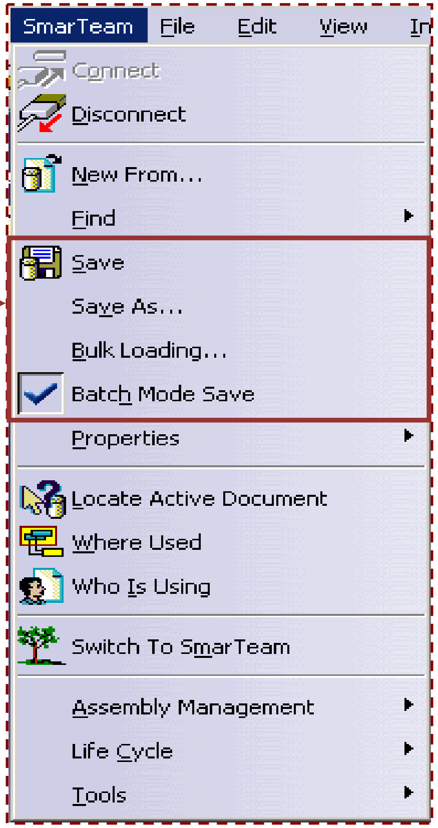
Student Notes:

About Saving Objects

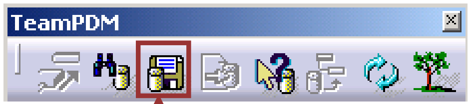
- CATIA V5 Interface is enriched by a SMARTEAM menu and a toolbar. You will find there all the needed commands.
- These commands allow you to rapidly save CATIA V5 documents in SMARTEAM and to act on them.

'SmarTeam' menu in CATIA V5

Save Commands



SMARTEAM Toolbar in CATIA V5



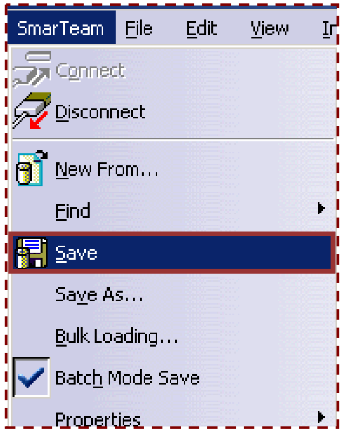
Save icon

Student Notes:

Saving a Document (1/3)

The SMARTEAM integrated menu provides two methods for saving CATIA documents:

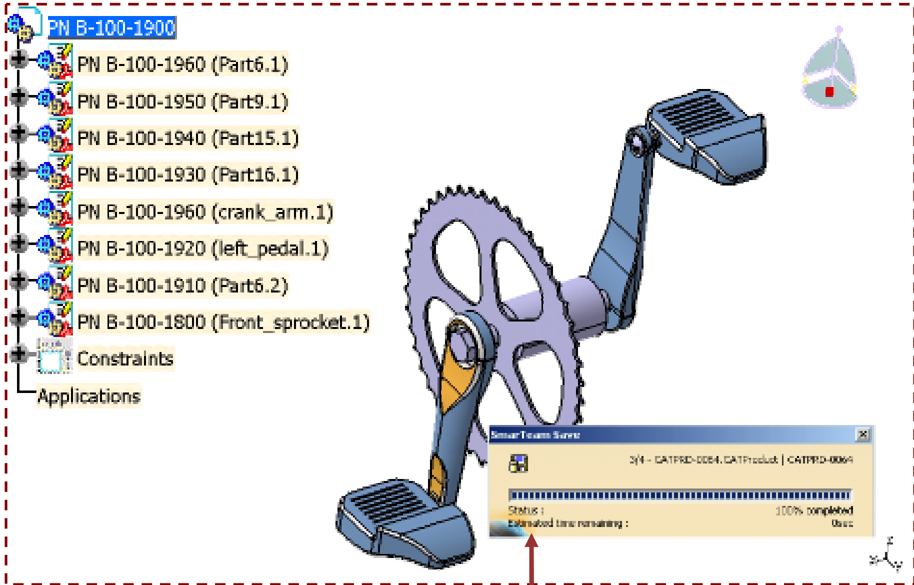
- **Save: Saves the document into the SMARTEAM database. When a save is executed for the first time on a CATIA object the Project Manager and the Profile card (based on whether the Batch mode save option is selected or not) are displayed. Forthcoming saves do not display this panels.**



Select Save from the 'SmarTeam' menu.

OR

Click on Save icon in the SMARTEAM toolbar.



The system updates the objects in the database.

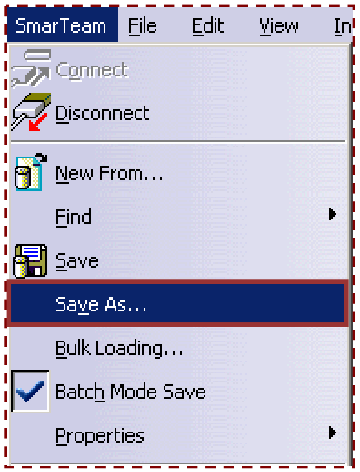
Student Notes:

Saving a Document (2/3)

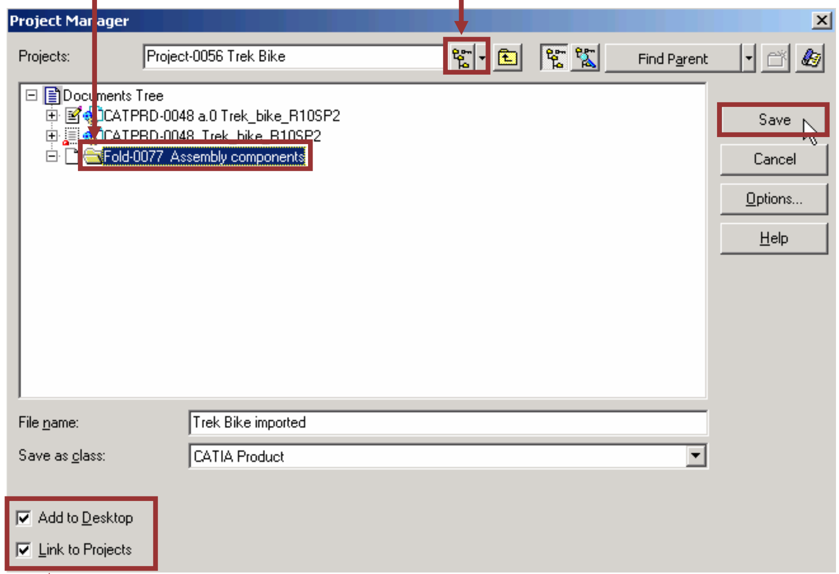
- **Save As: Saves the document into the SMARTEAM database and lets you define the project and the parent folder for the document every time the command is executed. Thus generating new Document ID each time.**

The object will be saved as a child in the selected folder under the selected project.

Click to choose the project and the parent folder of the document.



Select 'Save As…' from the 'SmarTeam' menu.

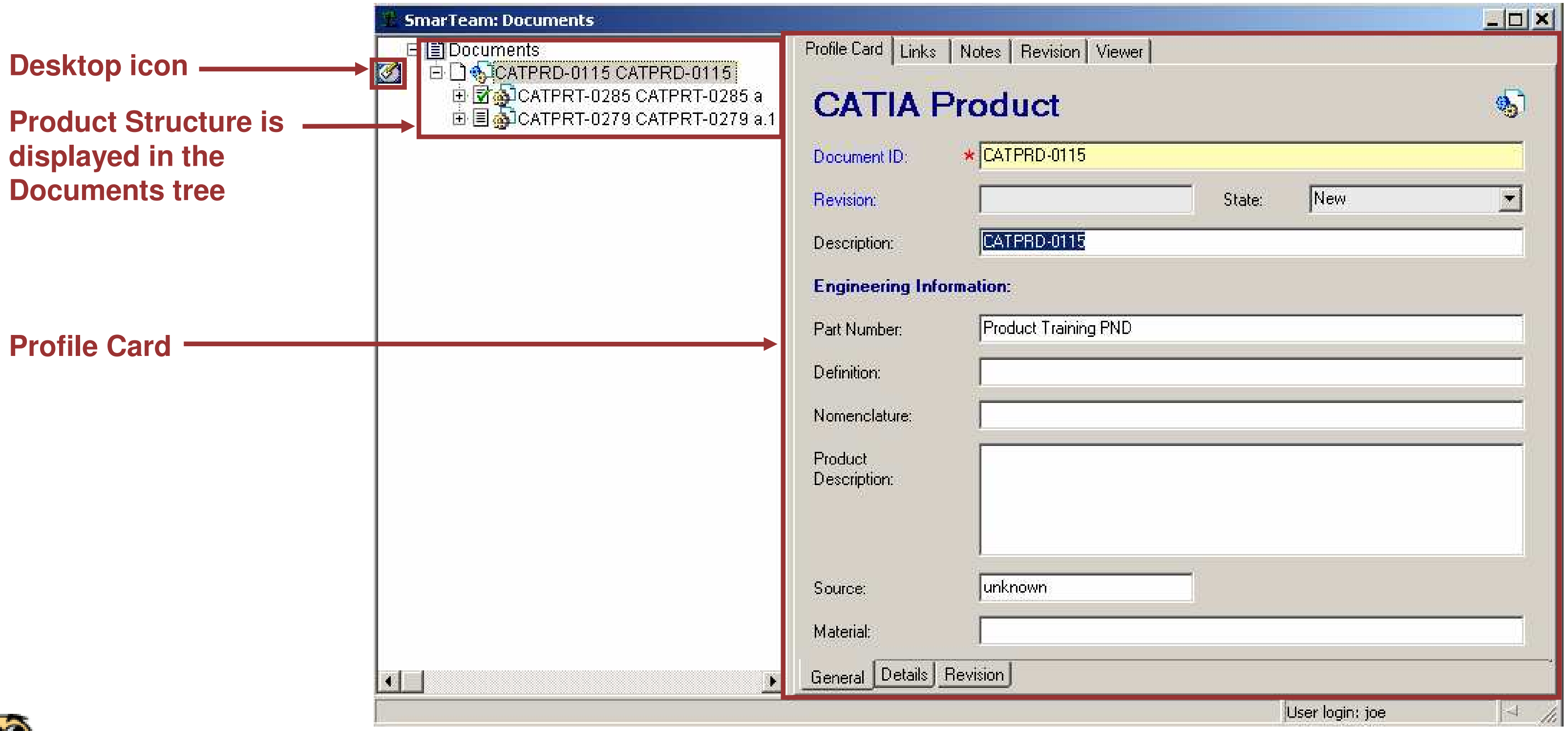


Check or Uncheck to choose the adapted option.

Student Notes:

Saving a Document (3/3)

- A Profile Card is displayed when the document has not already been saved in case of a normal SMARTEAM Save operation with the Batch Mode Save option unchecked.
- Enter a name for the part in Description Attribute.
- In the Details tab, you can name the file. If you do not, the document ID will be automatically set as file name.

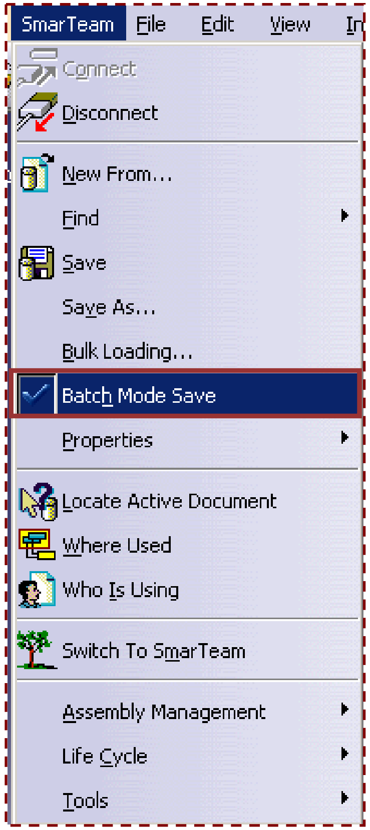


A SMARTEAM documents window is automatically displayed showing that the new document has been added to the database.

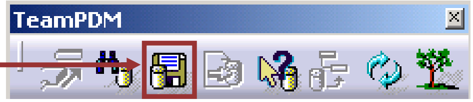
Student Notes:

Batch Mode Save

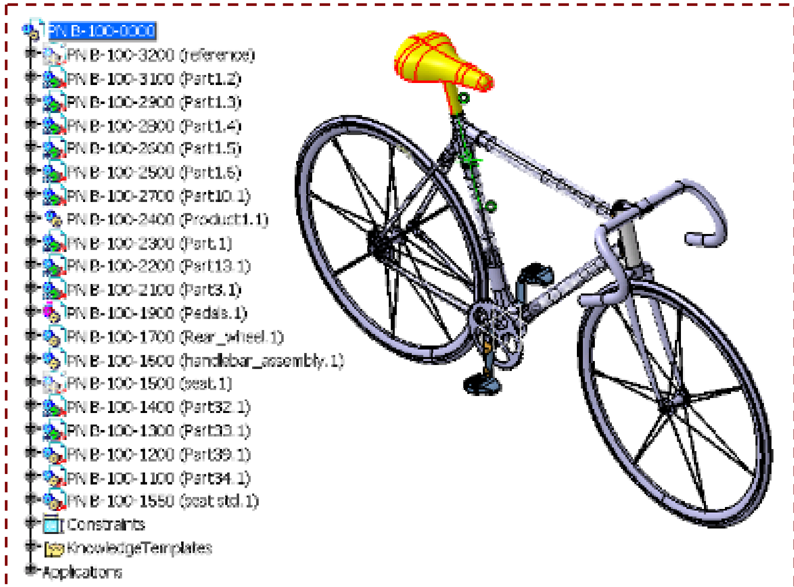
- Check or uncheck of the Batch Mode Save gives you choice of displaying or not displaying the Object Attributes panel for each component during an Assembly save
- This method allows you to save large assemblies in a shorter time.
- Each component is saved in the database with a unique SMARTEAM ID number.



Save icon



Click on Batch Mode Save to check or uncheck this option



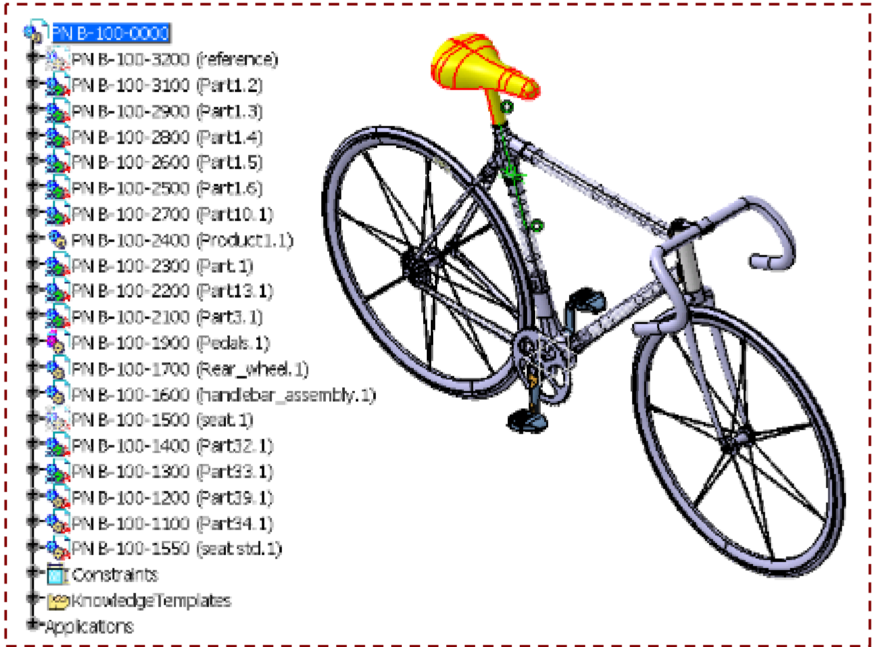
You can open a Profile Card for a component later for « update » after the save is completed. Information in the attribute fields can thus be modified.

Student Notes:

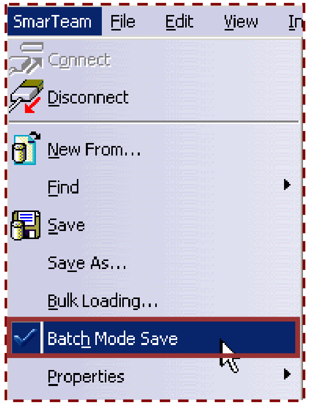
How to Save an Assembly (1/2)

In the SMARTEAM - CATIA Integration hierarchy, all of 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
The procedure is the same as adding a Part in SMARTEAM.

Shown below is an Assembly loaded in CATIA V5 session.



1 **Select the Batch Mode Save option. Object Attributes panel for each component will thus not be displayed during the save operation.**



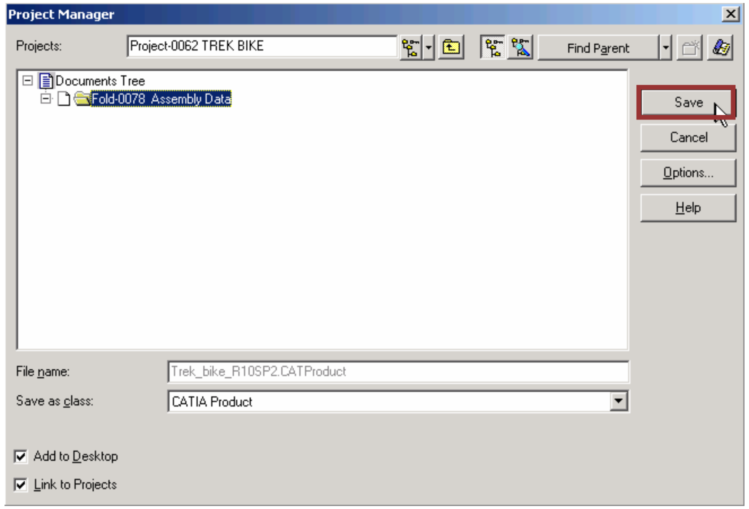
2 **Click on the Save icon in SMARTEAM toolbar.**

Student Notes:

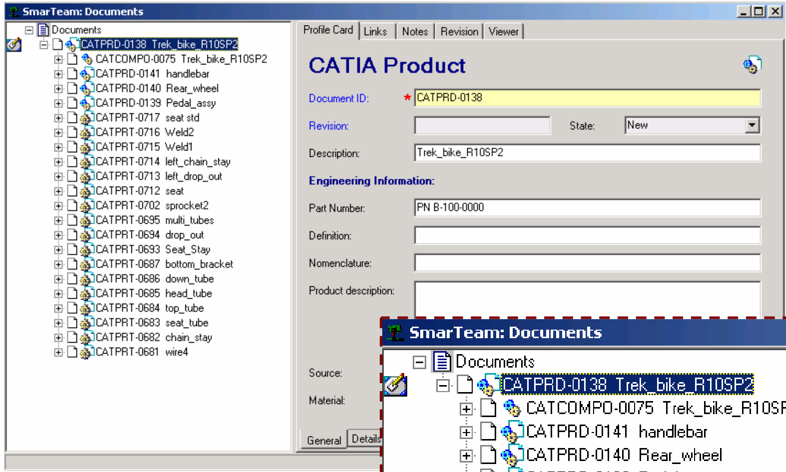
How to Save an Assembly (2/2)

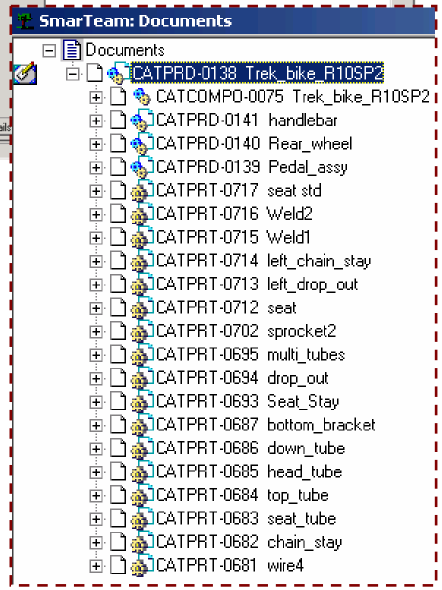
3 Choose a project and a parent folder.
The CATIA file name is displayed.

4 Click Save.



SMARTEAM Project Manager window





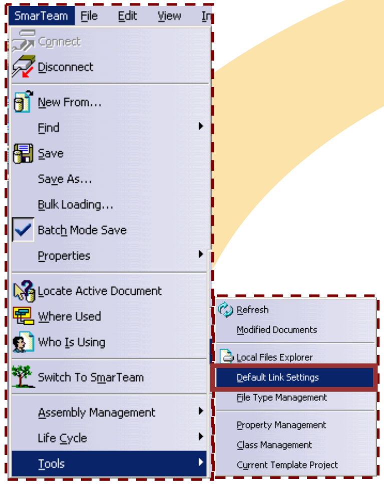
The inherent relationship between a Product and its components is automatically maintained and displayed in the SMARTEAM Documents window.

Student Notes:

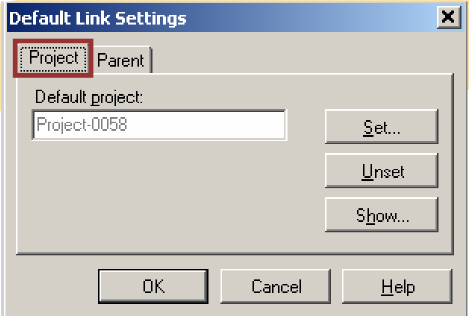
How to Choose a Default Project and Parent

To avoid choosing the Project and Parent for every *Save As* operation, you can choose to customize your defaults options

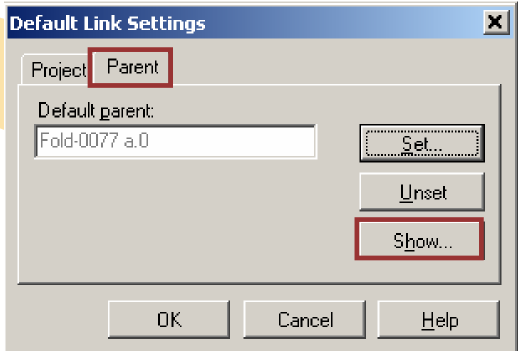
1 In SMARTEAM menu, select Tools / Default Link Settings



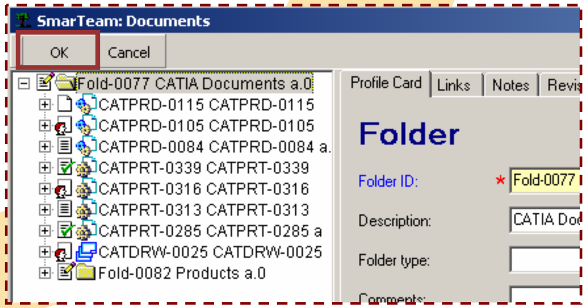
2 Select the Default Project



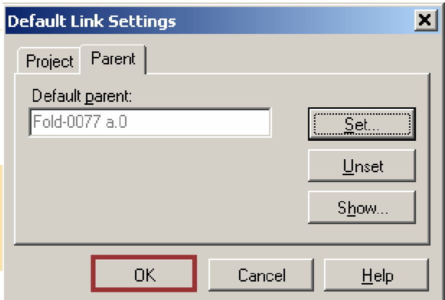
3 Select the Default Parent



4 You can locate the folder in SMARTEAM by clicking on Show button



5 Validate by "OK"



After clicking on OK, when saving a document, the Default Project and Parent are automatically selected

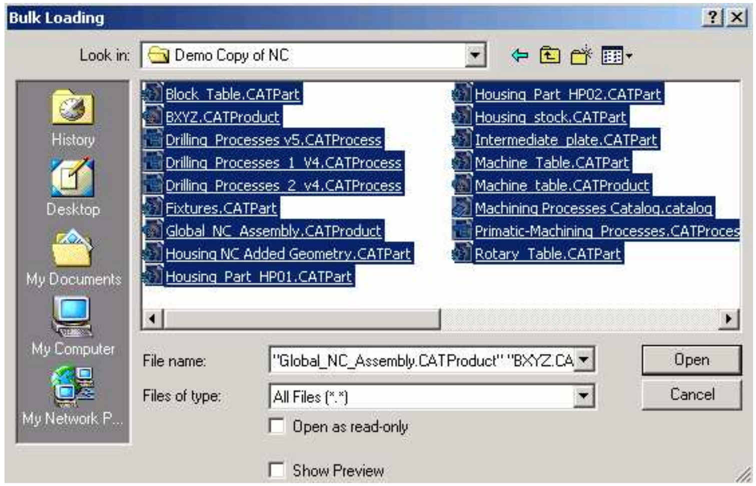
Bulk Loading

Whenever there are huge assemblies created in CATIA (locally saved) to be saved into the SMARTEAM database, the Bulk Loading command comes very handy.

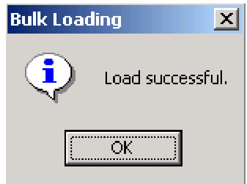
While executing Bulk Loading, the assembly is not opened in CATIA, thus saving a time required to load it into the CATIA session.

This command performs three operations at the same time – open, save and close and has the following advantages:

- **The save operation is quick**
- **Links between documents are kept**
- **The attribute mapping mechanism is used during the save operation**



CATIA files which are saved locally are selected in the panel for performing the Bulk Loading



Message indicates that the Bulk Loading is successful

Checking or Unchecking of the Batch Mode Save option has no effect during Bulk Loading.

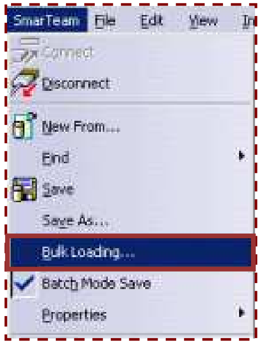
Student Notes:

Student Notes:

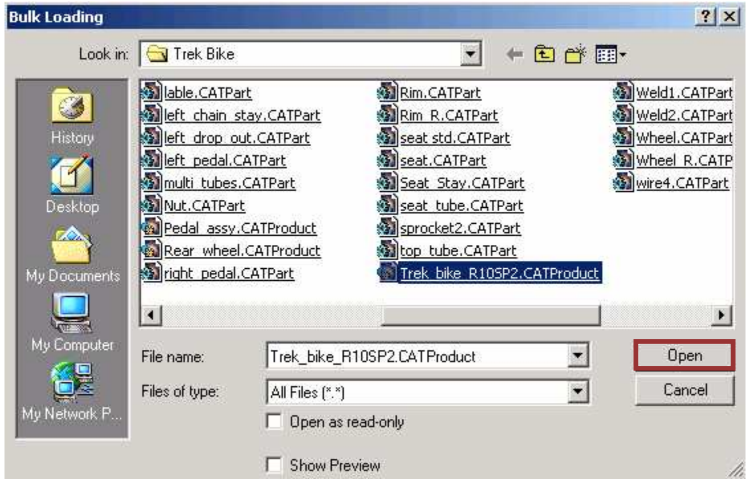
How to Use Bulk Loading (1/2)

The Bulk Loading command allows you to save a large number of files in the database without much of user intervention.
While executing Bulk Loading, the assembly is not opened in CATIA, thus saving a time required to load it into the CATIA session.

1 From SMARTEAM menu, select Bulk Loading

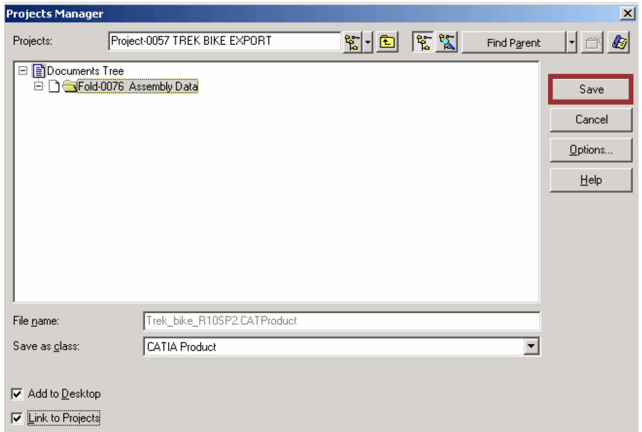


2 Select the CATIA Document/ Documents in the directory on your computer.



3 Choose the project and the parent folder for the document/ files.

4 Click on Save

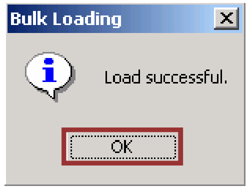


Selecting a single root CATProduct in the directory while executing the Bulk Loading ensures that all the linked CATParts and Subassemblies present in the directory are saved in the SMARTEAM database.

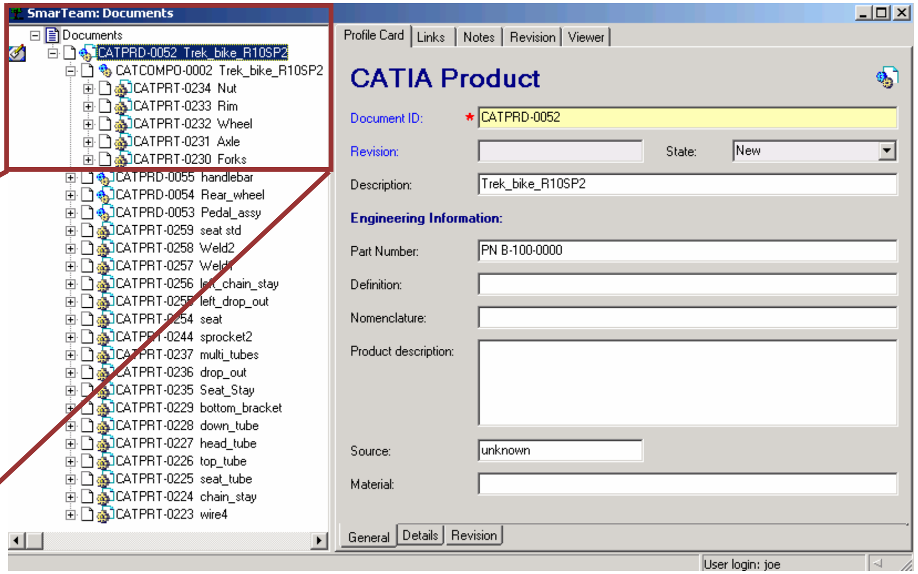
Student Notes:

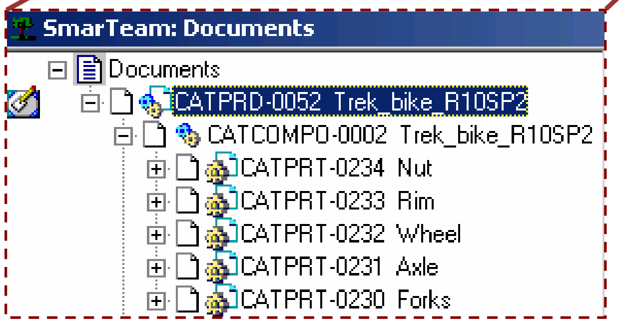
How to Use Bulk Loading (2/2)

5 A panel appears to confirm save in database successful. Click on “OK” to continue



The SMARTEAM Document window if open (minimized). All components previously selected for bulk loading, are displayed in the Documents tree



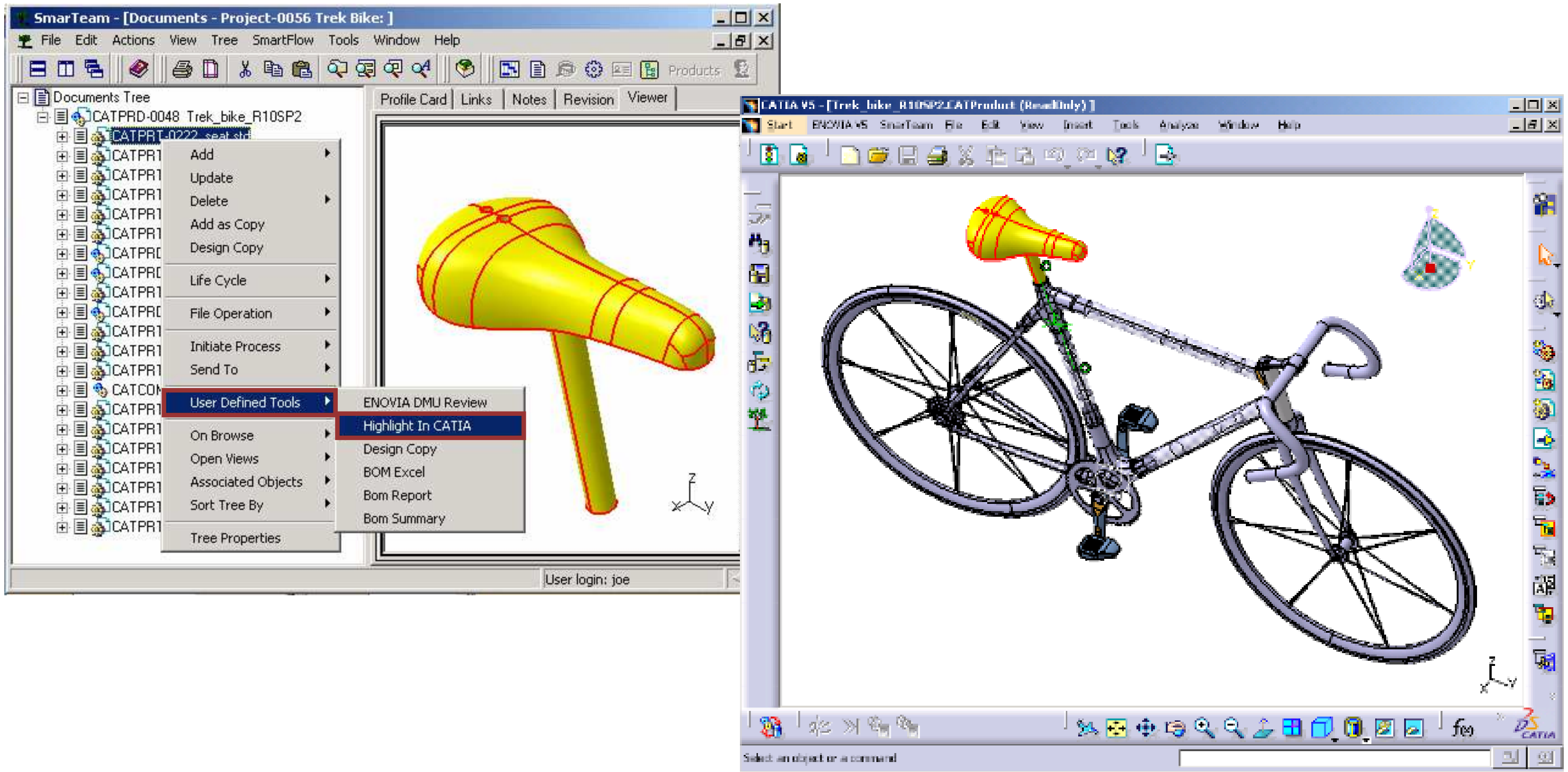


Student Notes:

Highlight in CATIA

By using this command you will be able to retrieve in CATIA, a document selected in the SMARTEAM Documents tree. The Highlight in CATIA command highlights the document in the CATIA session, assuming that the object selected in the SMARTEAM window is open in the CATIA session.

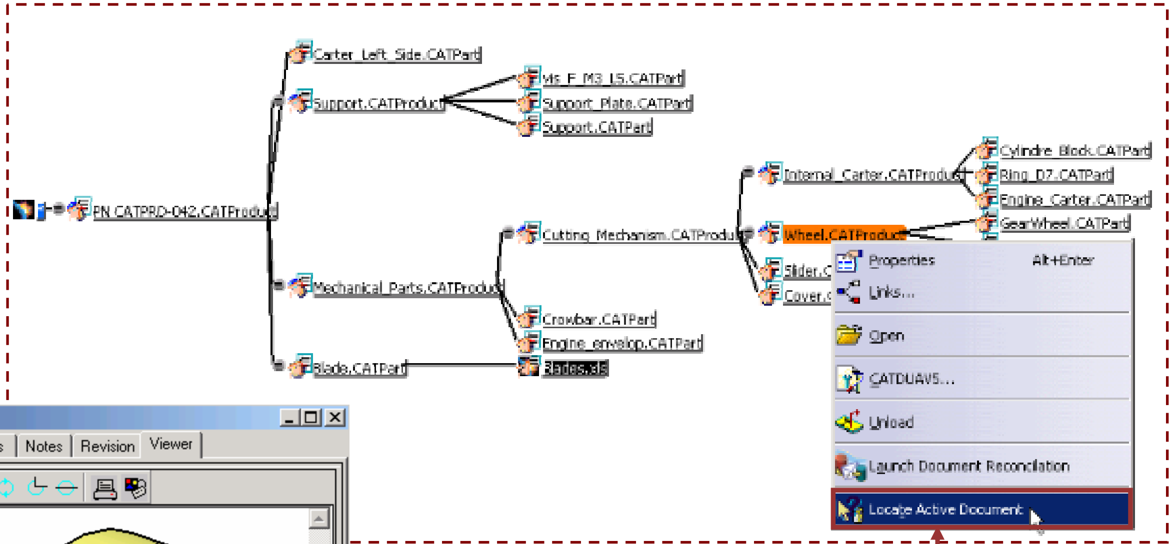
SMARTEAM

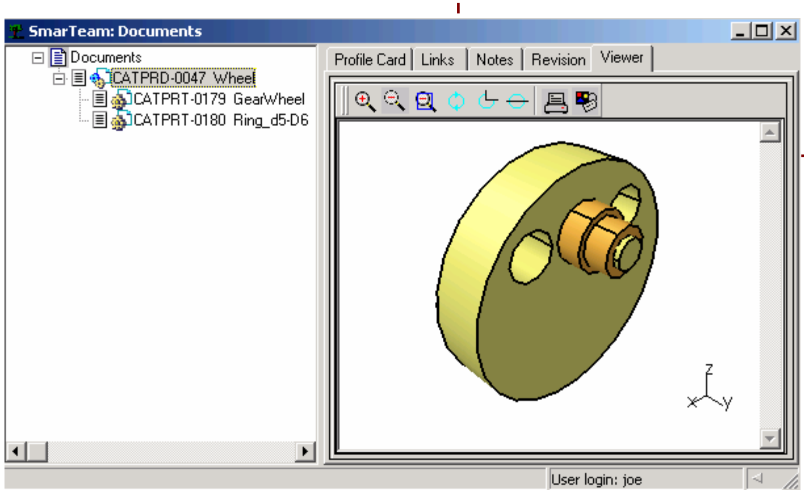


Student Notes:

Locating Active Document through the Desk Tree

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





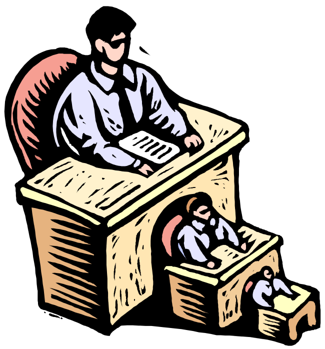
Locate the selected Document in SMARTEAM.

Profile card of the selected Document appears.

Student Notes:

Project Template

- A template is a ready-to-use data structure. SMARTEAM - Editor provides numerous business template; and each template is geared to a different environment, such as mechanical, maintenance and office.
- By providing business templates, SMARTEAM - Editor enables you to select a template that is applicable to your environment and use it "as is". You also have the option to customize these templates to suit your exact needs using the Data Model Designer, or to create a template from scratch and use it for your Project.
- Using Project Templates help in bringing about uniformity in work along with reducing time and efforts.



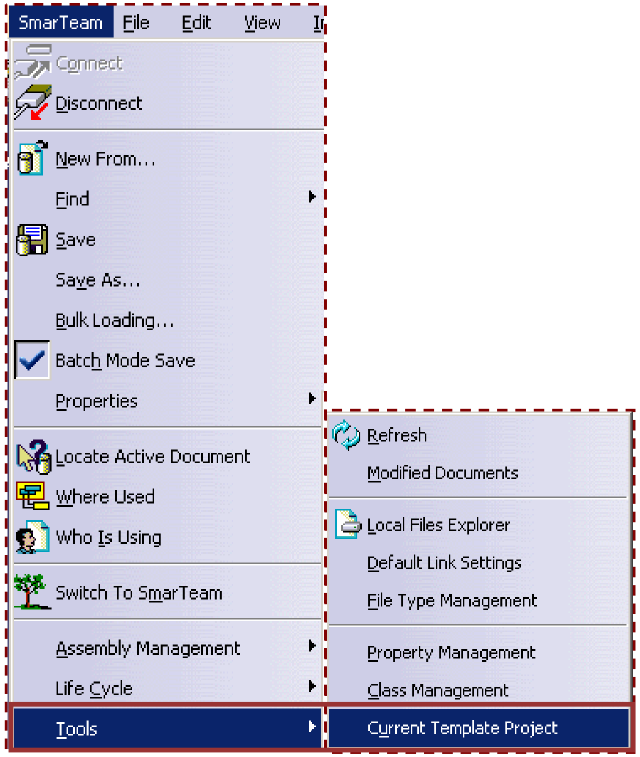
Student Notes:

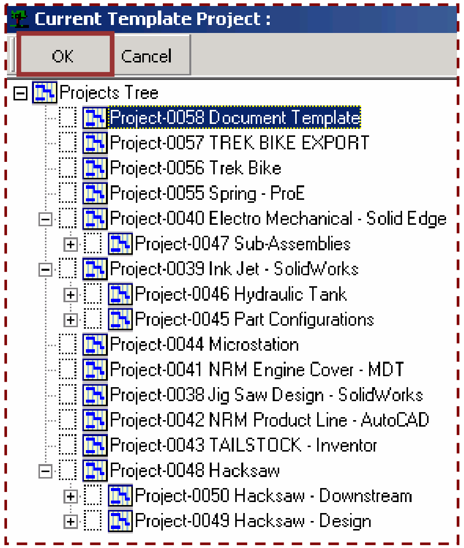
How to Use a Project Template

You can set a Project as Template Project and use the files linked to this Project as starter files.

1 **From SMARTEAM menu, select Tools / Current Template Project**

2 **Select the Project where all templates will be saved and validate "OK"**





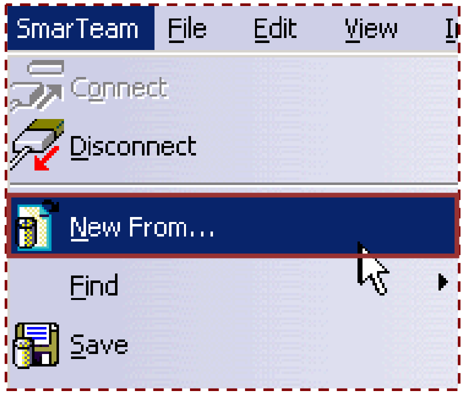
Now, you are ready to create a document from a template

Student Notes:

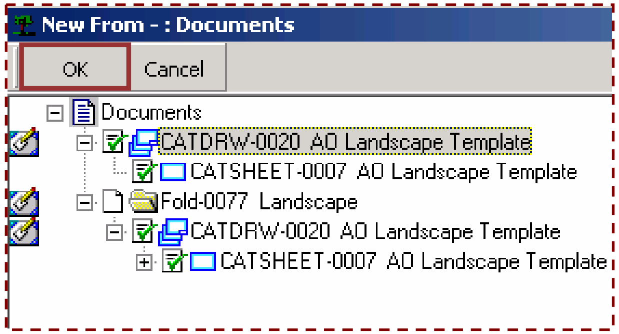
How to Use ‘New From’ from a Project Template

New From functionality allows you to start a document from one already stored in the “Template Project” defined in the CATIA Integration.

1 **From SMARTEAM menu, select New From…**

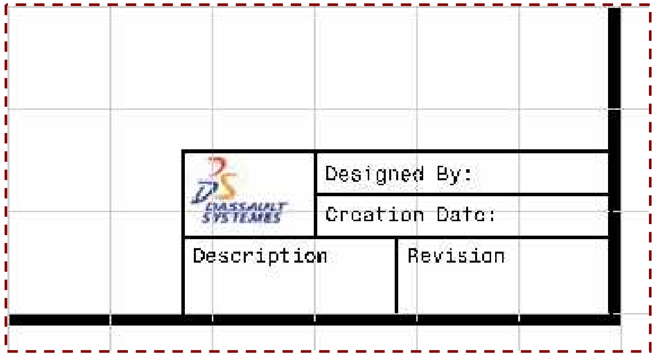


2 **Select the file in the list that you want to make a new copy of and validate ‘OK’**



A new copy of the CATIA file is opened. You are ready to work with CATIA V5.

In this case it is a blank drawing with a formatted Title block which is opened.



Student Notes:

To Sum Up

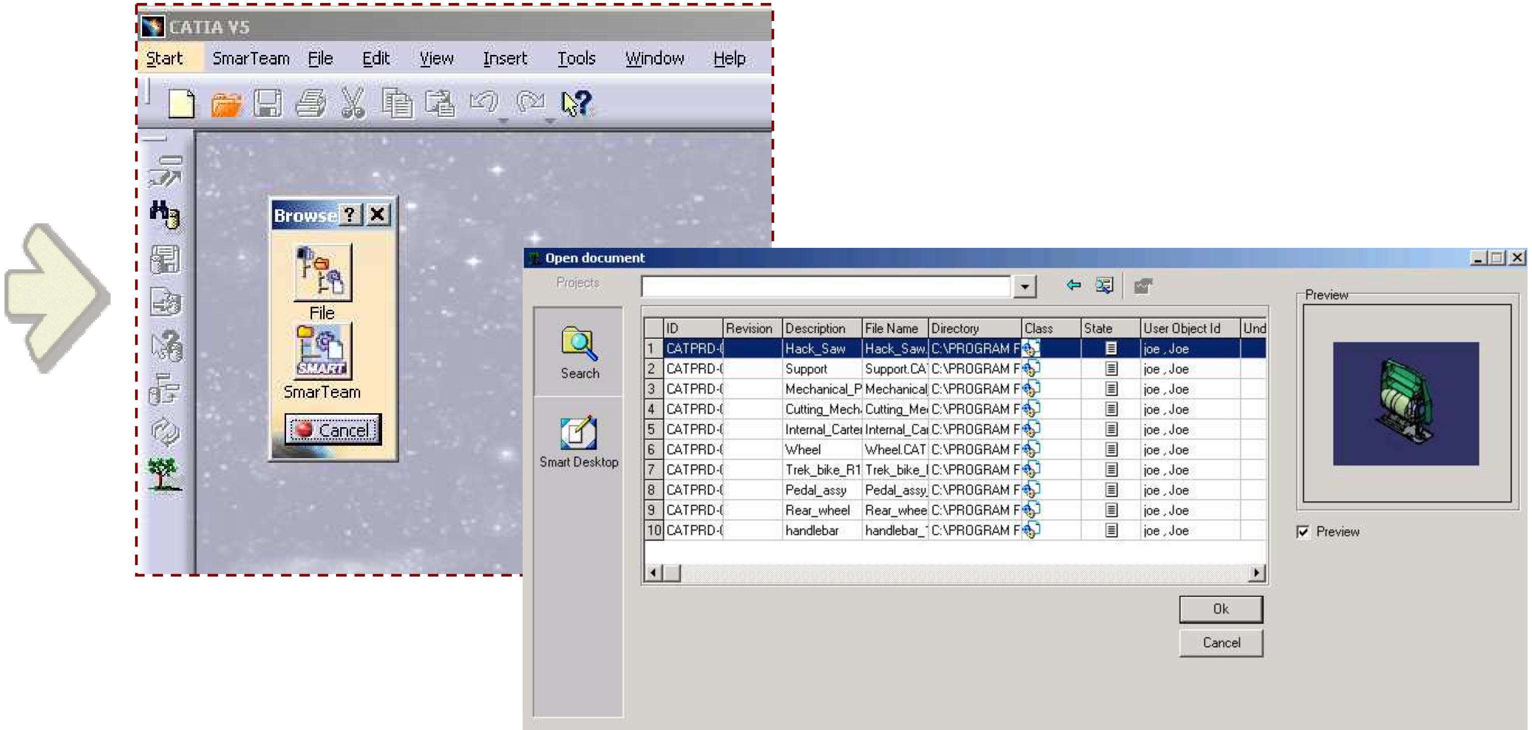
You have learned about

- SMARTEAM - CATIA Integration Settings
- Saving CATIA documents in SMARTEAM
- Highlight SMARTEAM objects in CATIA
- Locating Active Document through the Desk Tree
- Using Project Templates

Student Notes:

Manipulating Object

You will learn to open objects by using Open command from CATIA V5.

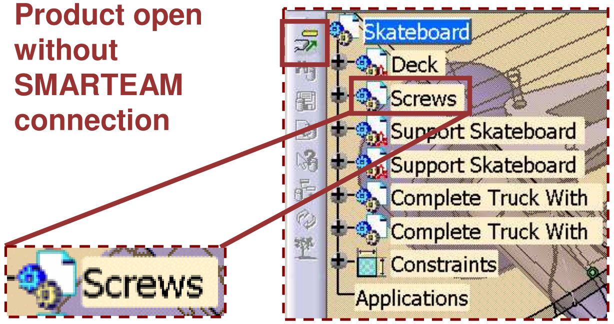


Student Notes:

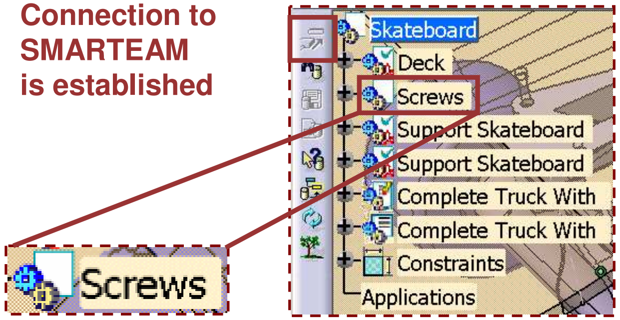
Displaying CATIA Document Status in Specification Tree

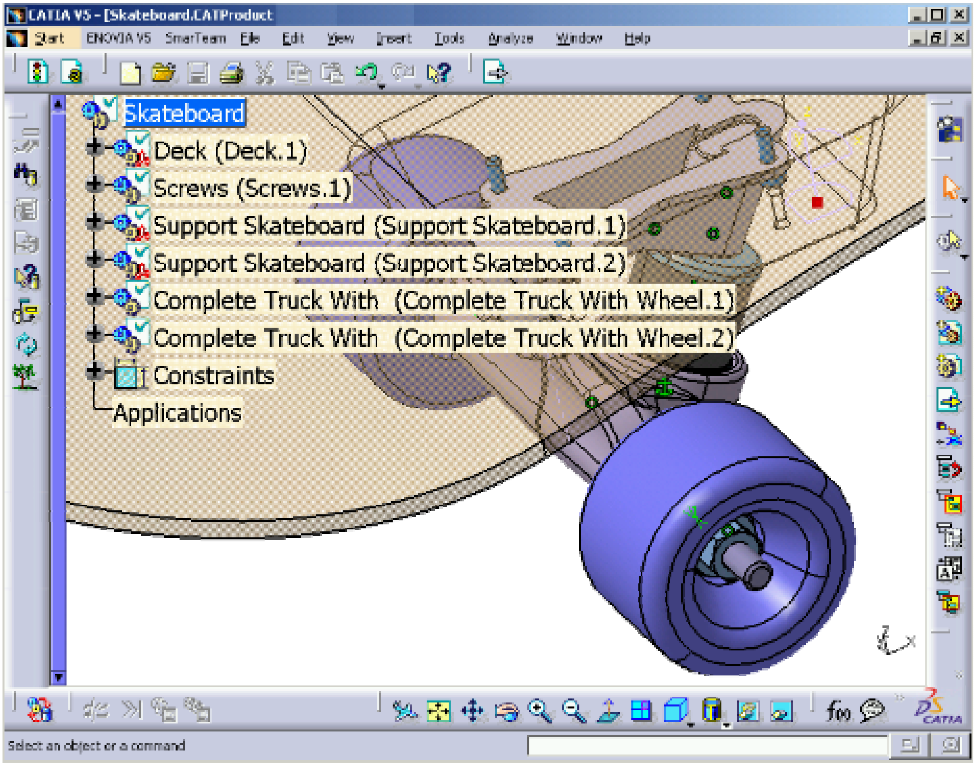
Each SMARTEAM object is displayed in CATIA V5 specification tree with an icon reflecting its state (i.e., New, Check In, Check Out, Release).

Product open without SMARTEAM connection



Connection to SMARTEAM is established





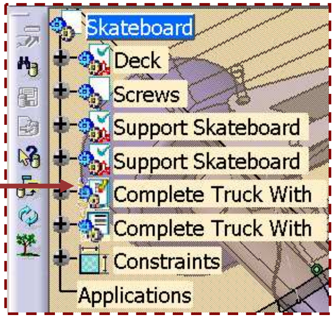
When there is no connection between CATIA-V5 and the SMARTEAM database, the SMARTEAM Lifecycle states for the CATIA objects in SMARTEAM are not indicated in the CATIA specification tree. The icon displayed is typically a page which has the top right side folded.

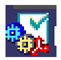
Student Notes:

Status List

- As shown in the list you can see all the lifecycle status types available in the specification tree from CATIA V5
- The status of the SMARTEAM object will change during :
  - SMARTEAM save
  - SMARTEAM Lifecycle operations like Check in, Check out, Release and Obsolete
  - Usage of «Open for edit» command from SMARTEAM
- The status of the SMARTEAM object will not change during :
  - Open as « Read only »
  - Open as « Temporary copy »

Lifecycle states indicated in the CATIA spec tree.



Lifecycle State	Representative icon in CATIA
New	
Checked In	
Checked In, Not Latest	
Checked In,Dirty	
Checked In, Not Latest, Dirty	
Checked Out	
Released	
Released, Not Latest	
Obsolete	

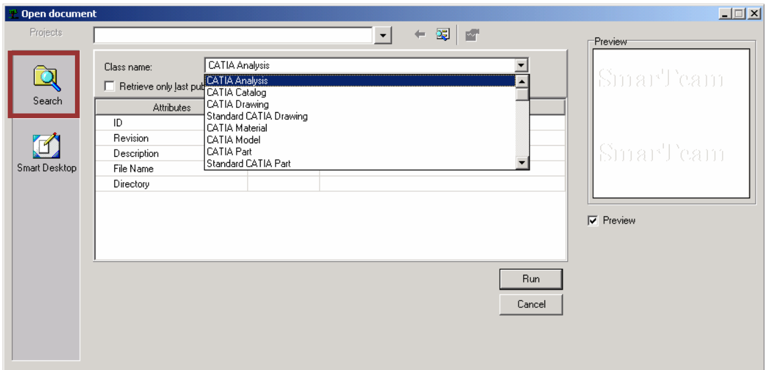
The CATIA specification tree always reflect the lifecycle status of the object if a connection between SMARTEAM and CATIA is established.

Student Notes:

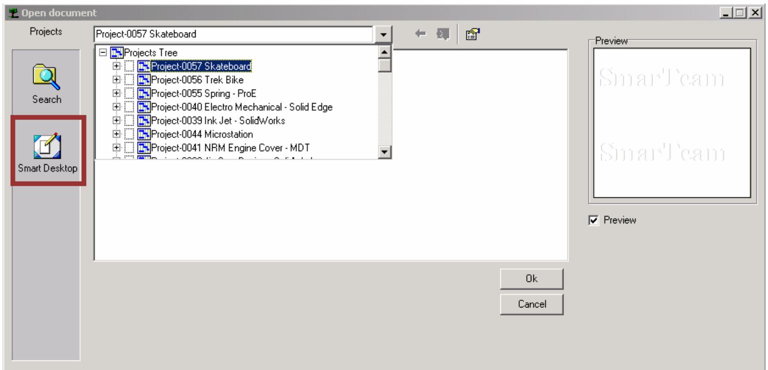
Open Document in CATIA (1/2)

- **File Open is generalized to give the ability to select a document from SMARTEAM instead of the file system location.**
- **A SMARTEAM 'Open Document' panel is displayed by using the following commands only when the recommended settings good for using SMARTEAM with CATIA are got in place.**

File Open



OR



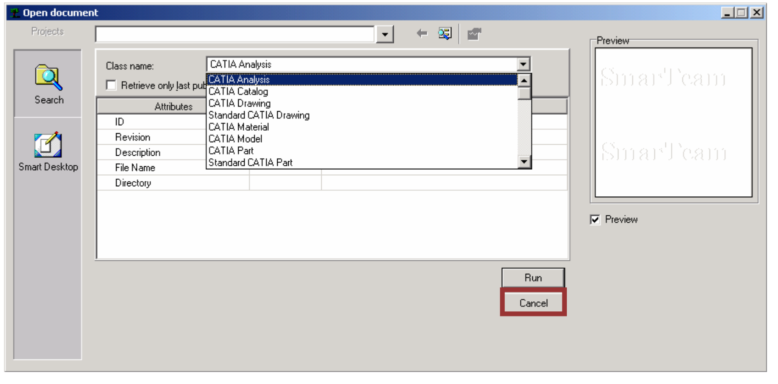
The first interface allows you to search objects through a query in SMARTEAM.

The second interface allows you to search object directly in SMARTEAM.

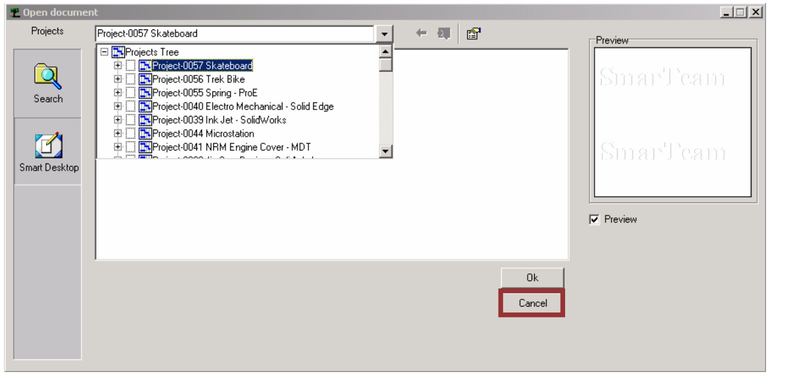
Student Notes:

Open Document in CATIA (2/2)

- By clicking on the “Cancel” button the “Browser” toolbar is displayed, thanks to the CATIA Document Environment settings in Tools > Options.
- The “Browser” toolbar gives you the options to re-open the documents from SMARTEAM or File location.



OR



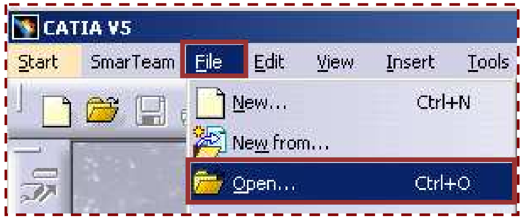
By clicking on ‘SmarTeam’ icon, you re-open the pointed document panel.

Student Notes:

How to Open an Object from CATIA V5

A SMARTEAM object pertinent to a CATIA class can be searched and opened in a CATIA session. Having launched CATIA and connected to SMARTEAM database

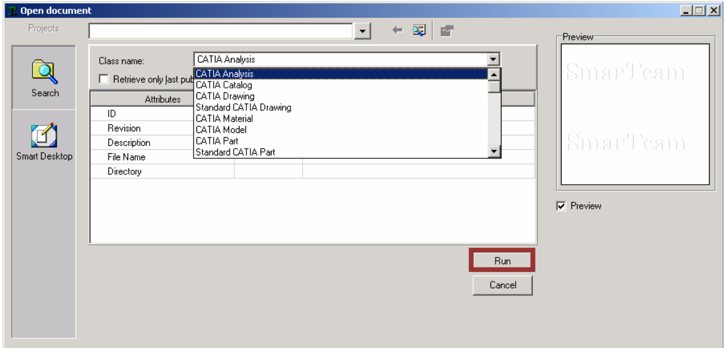
1a From the CATIA V5 menu, select File > Open…



OR

1b Click on Open icon in the Standard toolbar.

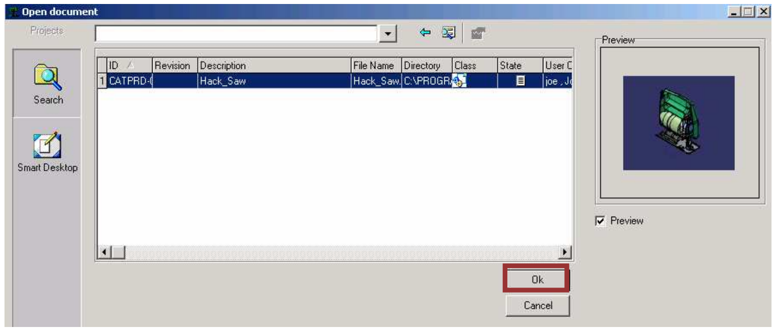
2 SMARTEAM Open document panel appears. Choose the class in dropdown list and click on Run.



3 Enter the value of the Description in the table

Attributes	Conditions	Values
ID		
Revision		
Description	Equals	hack*
File Name		
Directory		

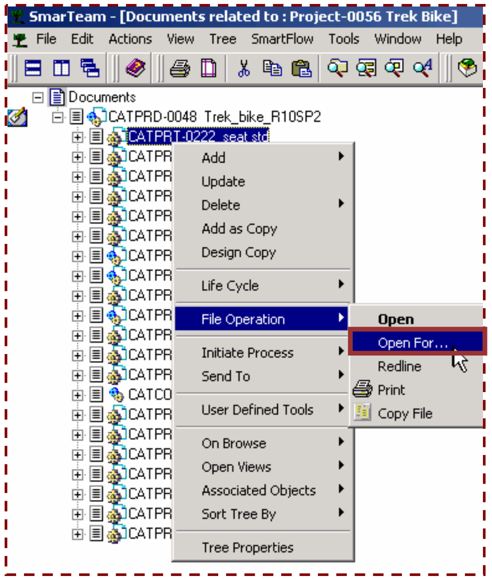
4 The result displays in the same panel. Now, select the document and click on OK to open CATIA V5



Student Notes:

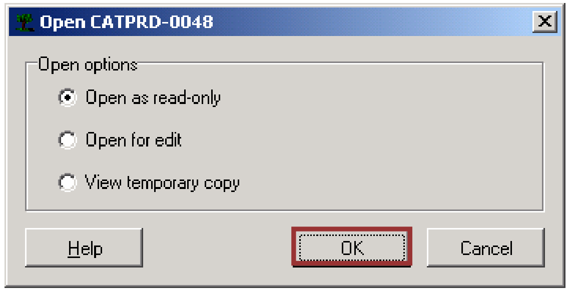
Performing File Operation

- In the Documents window of SMARTEAM, you can select a Document and with the contextual menu, execute the following file operation options :
  - Open as read only
  - Open for edit
  - Open to View the temporary copy.
- Based on the option chosen, the SMARTEAM object is opened in CATIA V5 session with Read or Write controls.



SMARTEAM Documents window

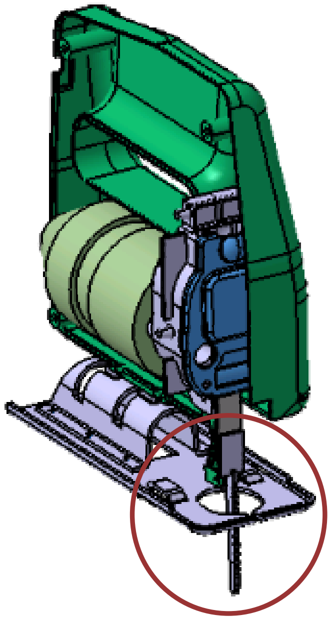
You can select any one option and click OK.

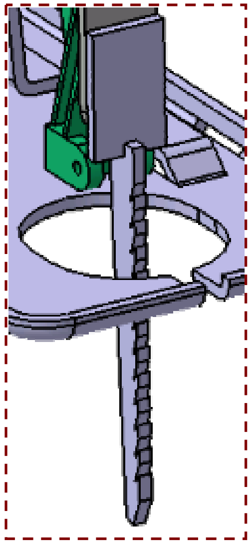


Student Notes:

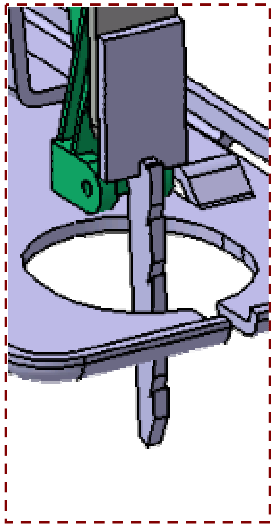
Replace Component

- **The new (replacing) component is chosen from the objects stored in the database.**
- **If the object type is not a CATPart or a CATProduct, a message is displayed and the operation is cancelled.**
- **This function is accessible from the SMARTEAM menu or in the contextual menu of CATIA thanks to the generalized ‘file open’.**



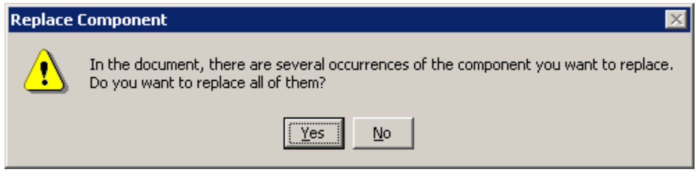


Blade in use initially



Blade after « Replace component »

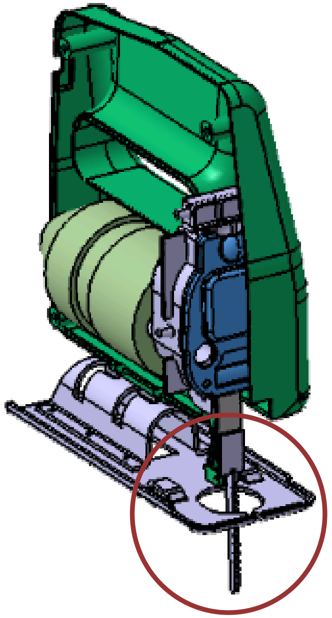
If there is more than one occurrence in the assembly, you can replace either the current one, or all of them.

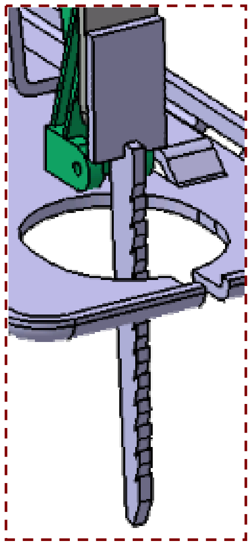


Student Notes:

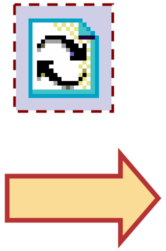
Replace with Selected Revision

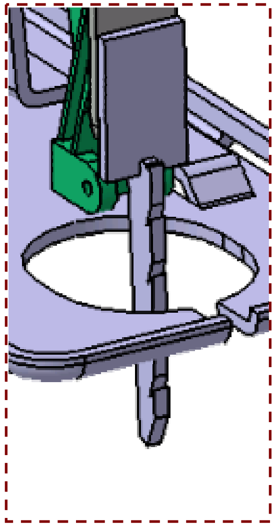
- If the design of a CATPart or a CATProduct has evolved across the Lifecycle of the CATPart or CATProduct, « Replace with selected Revision » comes handy to swap the usage of Revisions (configurations) in the Assembly using
- Below is indicated a case wherein Revision a.0 of the Blade is used in the Assembly. The design of the blade is still evolving until Revision a.
- You can select the CATPart in the CATIA spec tree and through the "SmarTeam" menu perform Assembly Management> Replace with selected Revision. Selecting and validating use of Revision a in the SMARTEAM Revisions window to instantiate the new Blade configuration in to the Assembly.





Blade configuration 1
(Revision a.o)



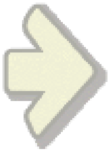


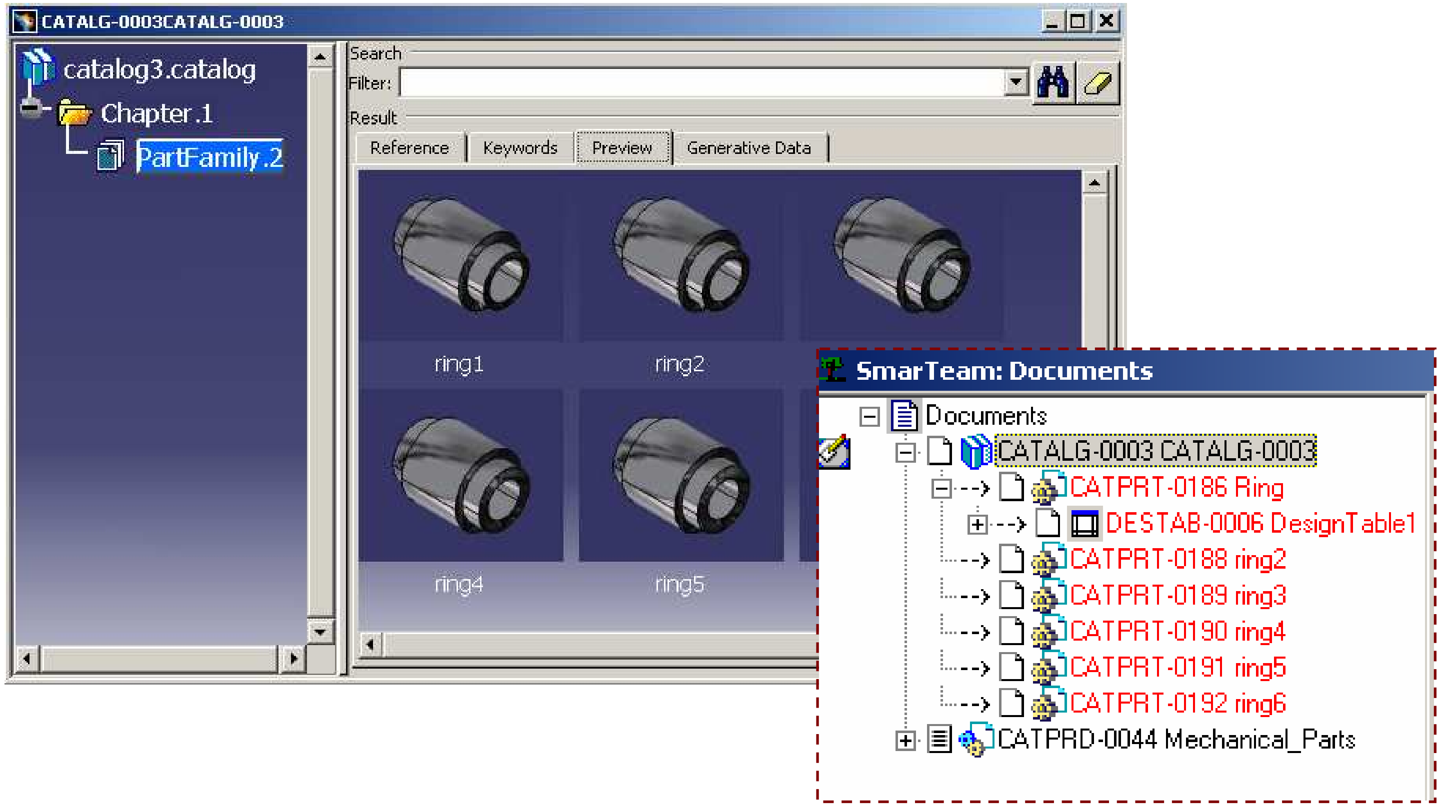
Blade configuration 2
(Revision a)

Student Notes:

Using Catalogs

After this lesson, you will be able to create, populate a catalog and instantiate a component in an assembly.

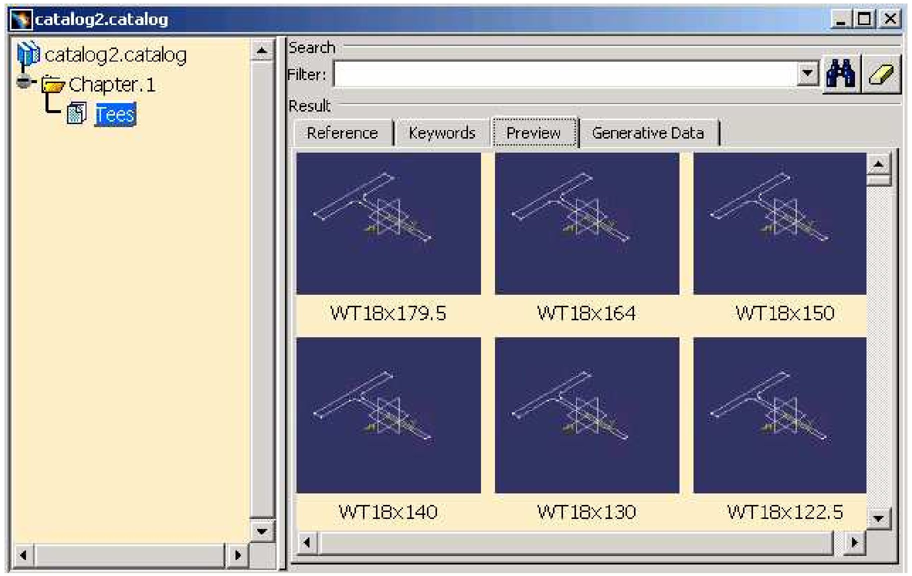




Student Notes:

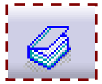
What is a Catalog

- A catalog is a file which provides you fast access and preview of CATIA Document, and features. It also classifies the documents or features with Chapters and Subchapters, and Keywords. The catalog file with its content are stored in the SMARTEAM database.
- From CATIA Catalog Editor workbench, you can define a catalog and its structure (chapters, families, part families…) and you can query catalogs to find the required components.
- From CATIA Catalog Browser panel, you can navigate through catalogs, instantiate catalog components, get a preview of any component. From several workbenches, Open Catalog icon is displayed and allows you to instantiate CATIA objects from the Catalog.

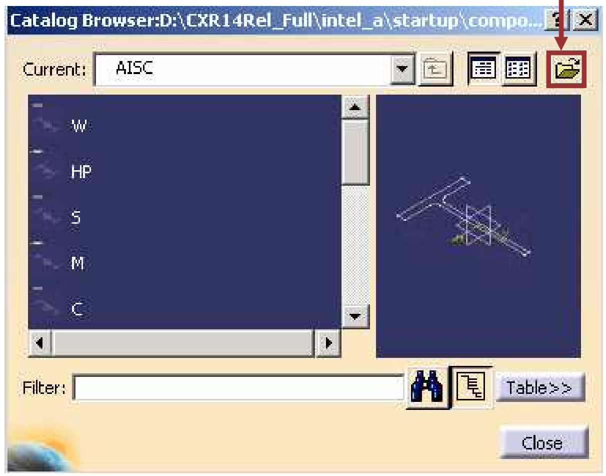


Catalog Editor Workbench

Open Catalog



Catalog Browser icon



Student Notes:

Classification Benefits

Address engineering, manufacturing and supply chain issues at the earlier stages of product design with products that integrate design tools with components obsolescence and supply chain data.

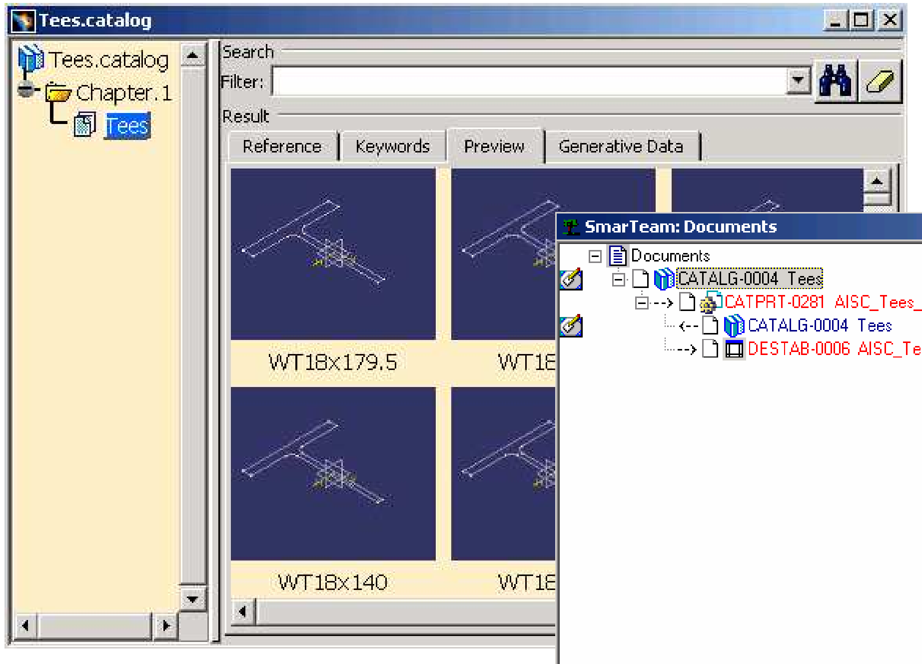
- **Give engineers fast visibility to all components that meet their technical requirements**
  - **Avoid design iteration related to non appropriate components usages**
  - **Standardize product development**
  - **Reduce inventory write-off**
- **Limit part proliferation**
  - **Reduce inventory write-off**
  - **It's easier for an engineer to create a new part than find an existing one**
  - **Every new off the shelf part introduced into a corporation costs about $10,000**
- **Reduce overall product cost**
  - **Reduce inventory write-off**
  - **Strategic sourcing**
- **Early involvement of the procurement, manufacturing, EMS and components experts in the design process**

Student Notes:

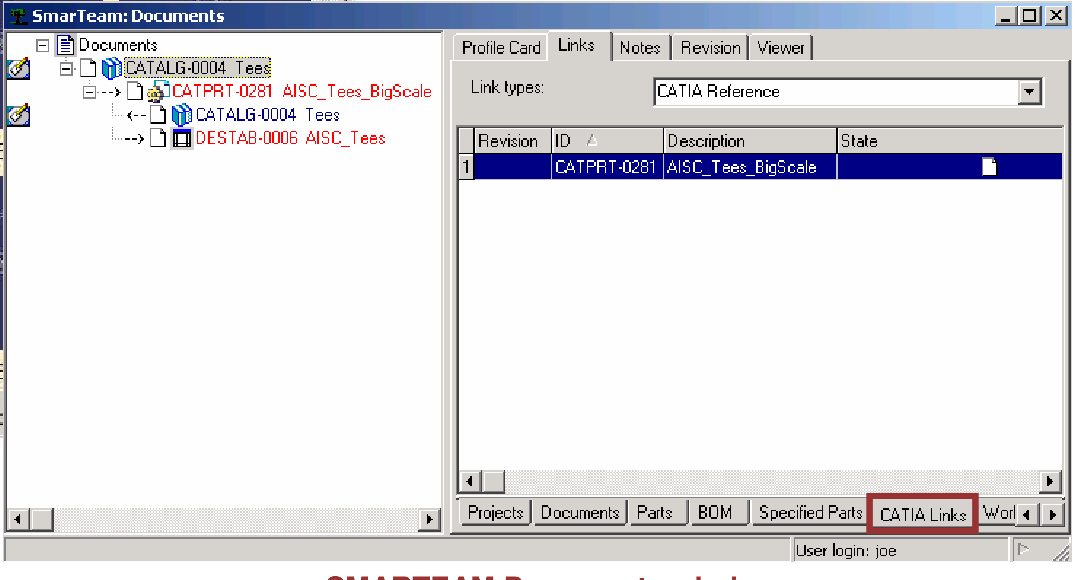
Displaying Results of Catalog Save in SMARTEAM

In SMARTEAM, after saving, the system stores the catalog in the Documents class like a container of CATIA parts. If the Visual Settings are defined for the Links (red links), all parts display in the Documents Tree under the Catalog.

The view Links tab/ CATIA Links sub-tab is another mean to visualize the CATIA links. In the catalog case, SMARTEAM shows CATIA Reference links in this view



CATIA Catalog Editor

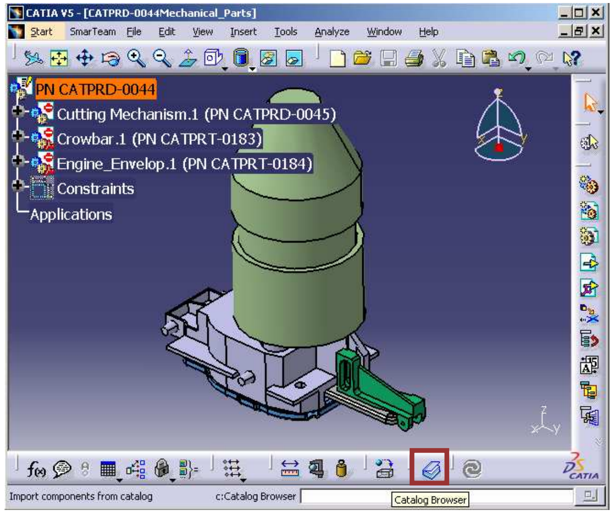


SMARTEAM Documents window

How to Use Catalog Components in CATIA V5 (1/2)

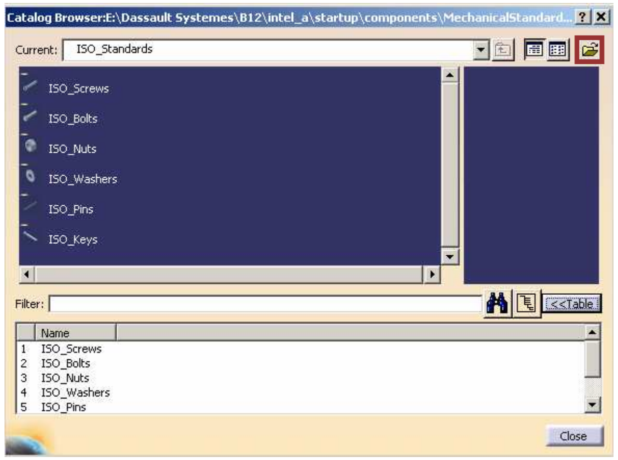
In CATIA V5, this methods allows you to instantiate a CATPart from the Catalog Browser panel.

1 Click on the Catalog Browser icon

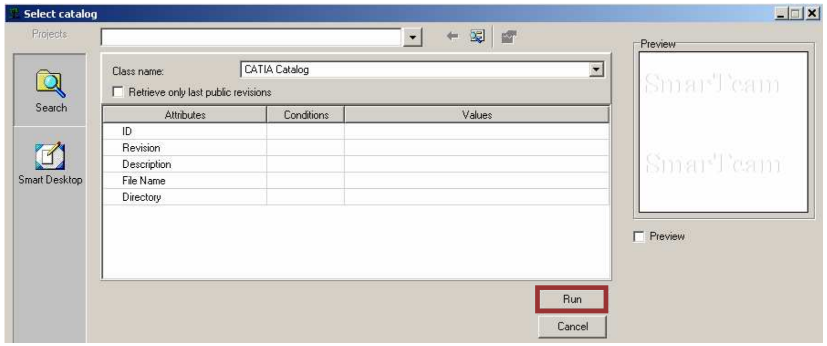


A Product is opened in CATIA V5

2 The Catalog Browser panel appears. Click on Browse another catalog icon



3 Automatically an intermediate panel appears to search a right catalog. Click on Run button to search all catalogs in SMARTEAM



Student Notes:

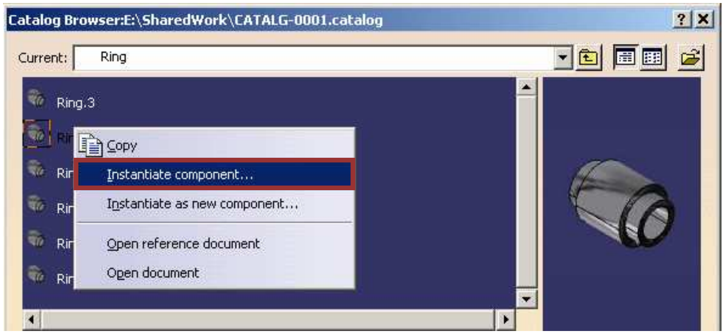
Student Notes:

How to Use Catalog Components in CATIA V5 (2/2)

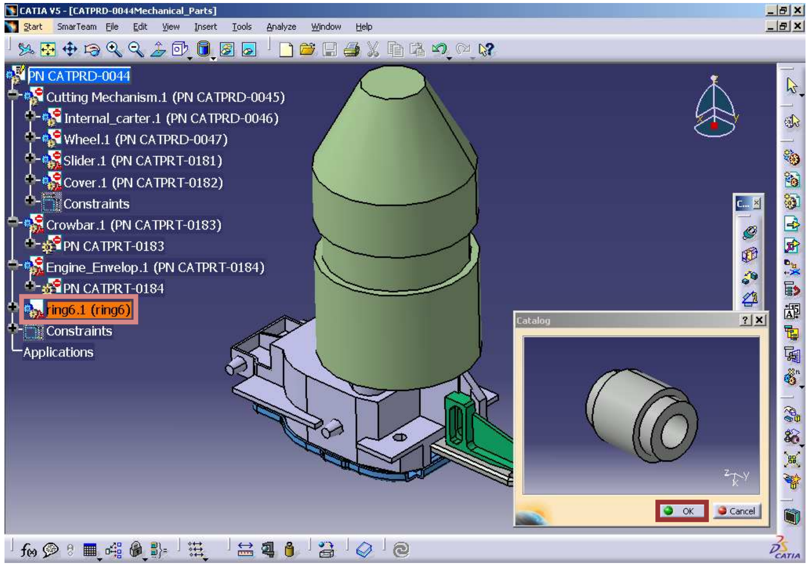
4 The result displays in the table. Select the adapted catalog and click on OK to load.

	ID	Revision	Description	File Name	Directory
1	CATALG-0		CATALG-0001	CATALG-0001.catalog	E:\SharedWork
2	CATALG-0		CATALG-0002	CATALG-0002.catalog	E:\SharedWork
3	CATALG-0		CATALG-0003	CATALG-0003.catalog	E:\SharedWork

5 The Catalog appears in the Catalog Browser panel. Navigate in the structure by double-clicking and when the right part display, right-click on the part name. Choose 'Instantiate component…' option.



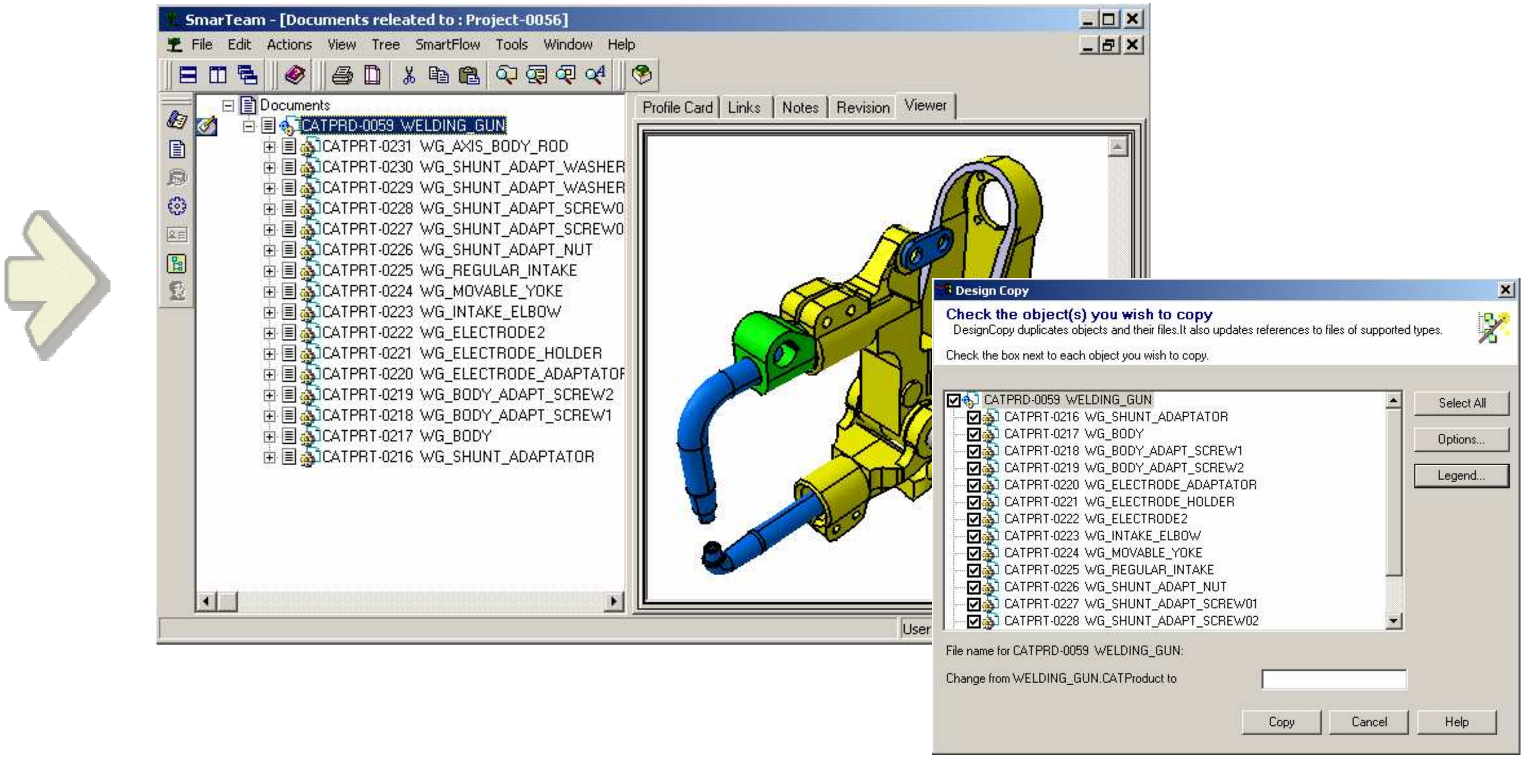
6 The Instantiate Component appears in the Specification Tree. Click OK.



Student Notes:

Design Copy Tool

You will learn how to copy a product structure to another project.



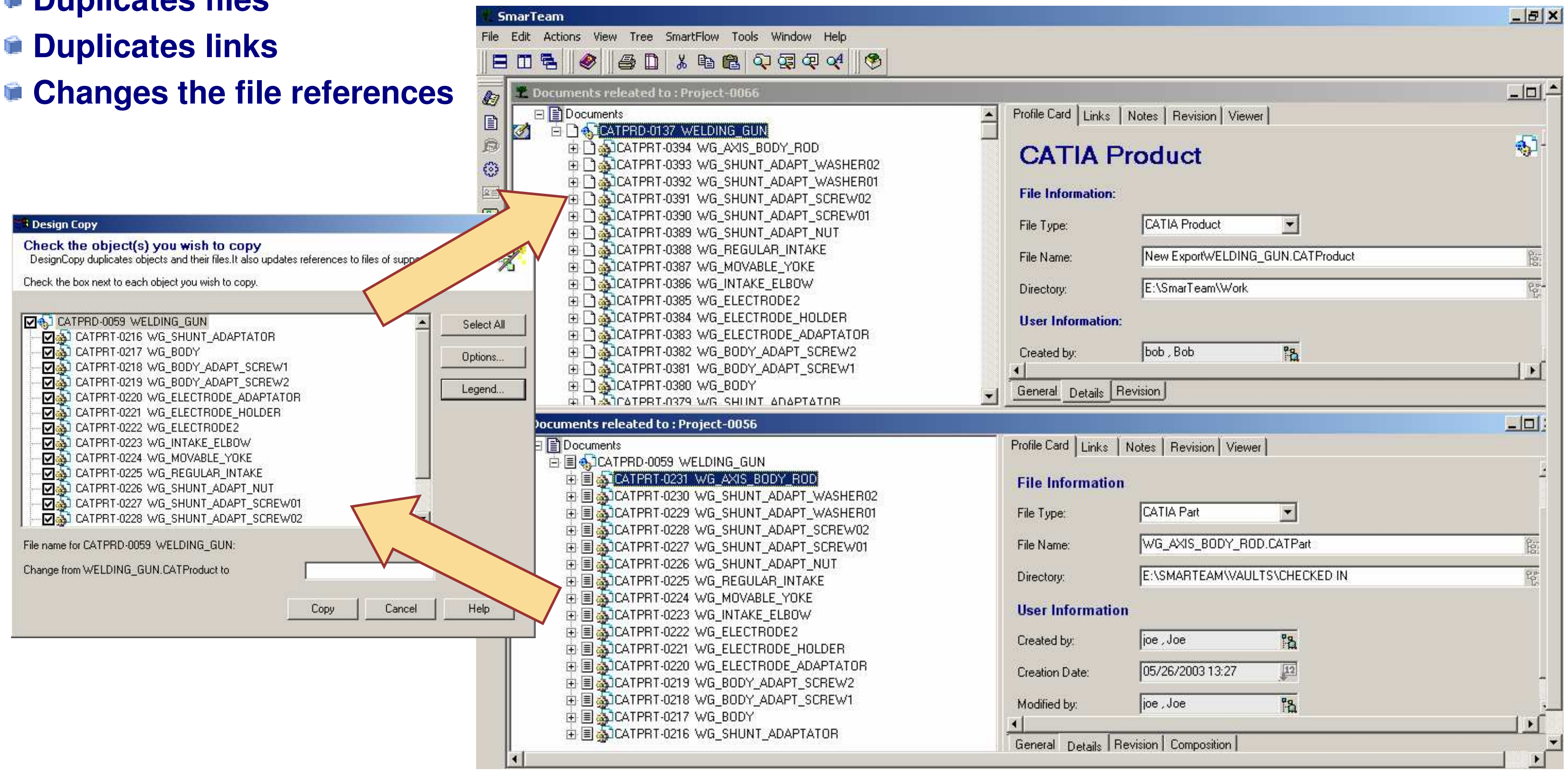
Student Notes:

About Design Copy

The Design Copy tool allows you to create a new SMARTEAM documents object structure by copying selected objects from an existing assembly.

This methods :

- **Duplicates meta-data**
- **Duplicates files**
- **Duplicates links**
- **Changes the file references**



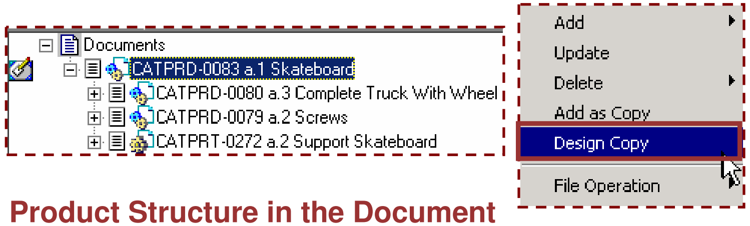
Student Notes:

How to Do the Design Copy (1/2)

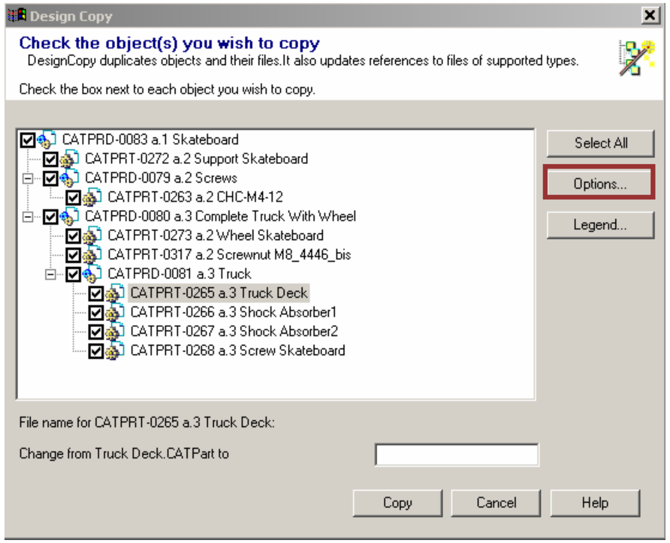
The Product structure of which the Design copy is intended to be made, is opened in the SMARTEAM Documents window

1 **Select the Document and with the contextual menu select « Design Copy »**

2 **Click on Options**



Product Structure in the Document Brower Tree



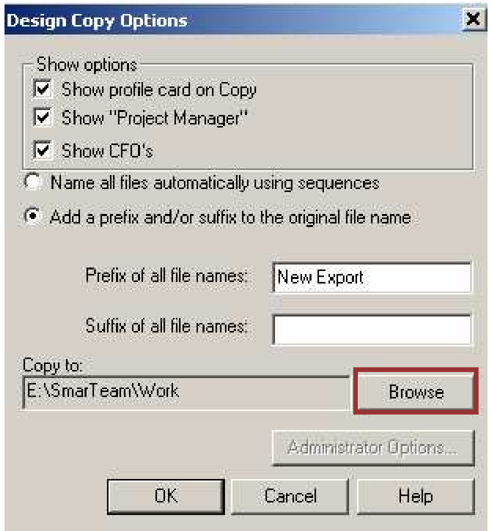
Design Copy panel

In Design Copy options, just before copying objects, you can change the Work directory to share files with another user or department.

Student Notes:

How to Do the Design Copy (2/2)

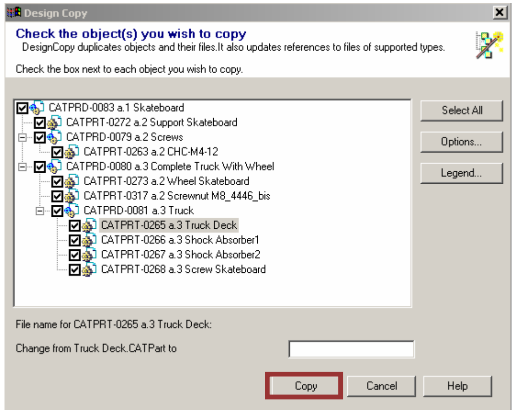
3 **Click Browse**



Design Copy options

4 **The shared work directory is displayed in this panel. Select the intended folder and validate OK.**

5 **Click Copy**



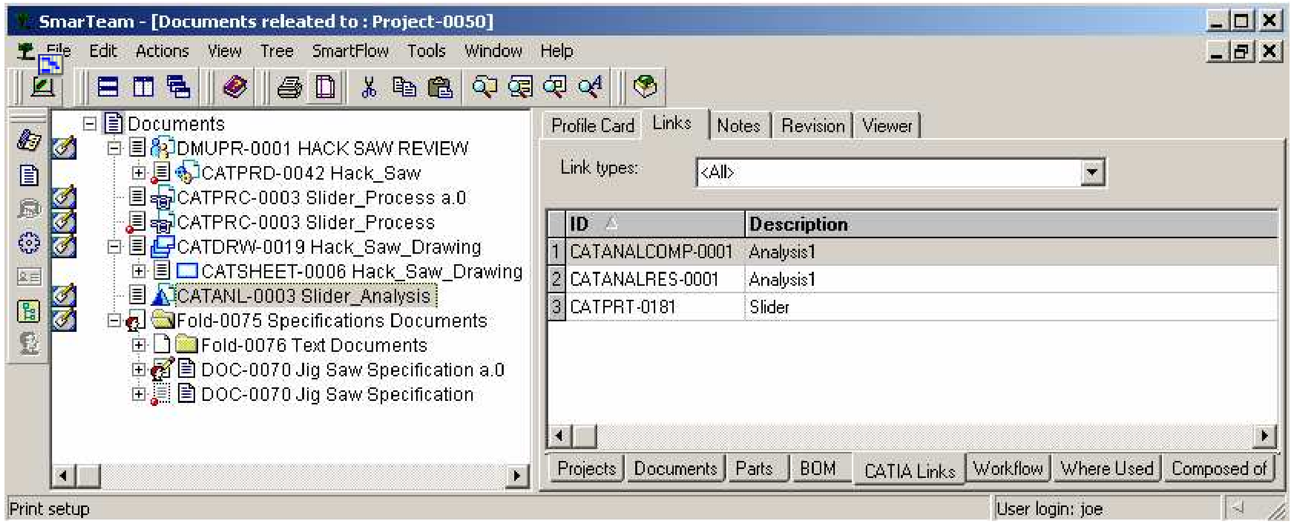
Design Copy panel

The SMARTEAM lifecycle state of the objects copied is 'New' immaterial of the state of the objects used for copy.

Student Notes:

CATIA Links

You will see how to show and check CATIA links in SMARTEAM.



Student Notes:

CATIA Links

This lesson will cover following topics:

- About CATIA links
- CATIA Links List
- Displaying Link Impact
- CATIA Drawing Links
- CATIA Analysis Links
- CATIA Process Links

Student Notes:

About CATIA Links

SMARTEAM - Editor stores a large number of documents and types of relations between these documents. The documents created in several applications such as CATIA/DELMIA/ENOVIA V5.

The native SMARTEAM integration of CATIA link semantic enables dedicated behaviors on each type of link:

- **Eased navigation**
- **Display links type (icon)**
- **Filter document relations by link type**
- **Improved Impact Analysis capabilities**

V5 Links	icon
CATIA Product Link (P)	
CATIA Design Link (D)	
CATIA Rule Base Link (RUL)	
CATIA Design Table (DT)	
CATIA Downstream Application Link (DA)	
CATIA Reference Links (REF)	
CATIA Contextual Links (C)	
CATIA Results Links (RES)	
CATIA Is composed of (IS)	

The nine 'CATIA links' managed by SMARTEAM

CATIA Links List 1/4

Student Notes:

Through the list below, you will discover all CATIA links exposed in SMARTEAM

- **Product Structure Link: This is the main link in SMARTEAM. It define the Bill Of Material structure.**
  - **CATProduct->CATProduct, CATProduct->CATPart, CATProduct->Internal Component…**

Examples

Link	Origin Type	Icon	Destination Type
CATIA Product Link	CATIA Product		CATIA Product
CATIA Product Link	CATIA Product		CATIA Internal Component
CATIA Product Link	CATIA Product		CATIA Part
CATIA Product Link	CATIA Product		CATIA Representation
CATIA Product Link	CATIA Product		CATIA model
CATIA Product Link	CATIA Internal Component		CATIA Product
CATIA Product Link	CATIA Internal Component		CATIA Part
CATIA Product Link	CATIA Internal Component		CATIA Representation

- **Design Link: The 'Design Link' directly participates in the Design process to describe the relationship created by the designers between two parts. These links capture the important design intent information and are used extensively for impact analysis.**
  - **Link to external parameters (formulas)…**

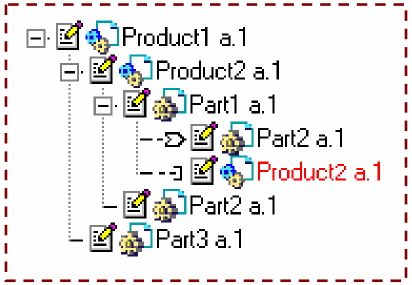
Examples

Link	Origin Type	Icon	Destination Type
CATIA Design Link	CATIA Product		CATIA Product
CATIA Design Link	CATIA Product		CATIA Internal Component
CATIA Design Link	CATIA Product		CATIA Part
CATIA Design Link	CATIA Product		CAD Document
CATIA Design Link	CATIA Internal Component		CATIA Product
CATIA Design Link	CATIA Internal Component		CATIA Part
CATIA Design Link	CATIA Internal Component		CAD Document
CATIA Design Link	CATIA Part		CATIA Part
CATIA Design Link	CATIA Part		CATIA Product
CATIA Design Link	CATIA Part		CATIA model
CATIA Design Link	CATIA Part		CAD Document
CATIA Design Link	CATIA Analysis		CATIA Product
CATIA Design Link	CATIA Analysis		CATIA Part
CATIA Design Link	CATIA Drawing		CATIA Drawing
CATIA Design Link	CATIA Drawing		CATIA Part
CATIA Design Link	CATIA Drawing		CATIA Product
CATIA Design Link	CATIA Drawing		CATIA Representation
CATIA Design Link	CATIA Sheet		CATIA Drawing
CATIA Design Link	CATIA Sheet		CATIA Part
CATIA Design Link	CATIA Sheet		CATIA Product
CATIA Design Link	CATIA Process		CATIA Part
CATIA Design Link	CATIA Process		CATIA Product
CATIA Design Link	CATIA Process		CATIA Process
CATIA Design Link	CATIA Document		CATIA Document

Student Notes:

CATIA Links List 2/4

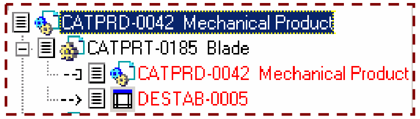
- **Downstream Application Link: The 'Downstream Application Link' corresponds to a link between a downstream application and the design data. It describes the relationship between a downstream application document (Drawing, NC,…) and a CATPart or CATProduct**
  - **CATDrawing->CATProduct/CATPart, CATProcess->CATProduct/CATPart, CATAnalysis->CATProduct/CATPart**



Examples

Link	Origin Type	Icon	Destination Type
CATIA Downstream Application Link	CATIA Analysis	->	CATIA Product
CATIA Downstream Application Link	CATIA Analysis	->	CATIA Part
CATIA Downstream Application Link	CATIA Analysis	->	CATIA Analysis Input
CATIA Downstream Application Link	CATIA Drawing	->	CATIA model
CATIA Downstream Application Link	CATIA Drawing	->	CATIA Part
CATIA Downstream Application Link	CATIA Drawing	->	CATIA Product
CATIA Downstream Application Link	CATIA Drawing	->	CATIA Representation
CATIA Downstream Application Link	CATIA Sheet	->	CATIA model
CATIA Downstream Application Link	CATIA Sheet	->	CATIA Part
CATIA Downstream Application Link	CATIA Sheet	->	CATIA Product
CATIA Downstream Application Link	CATIA Sheet	->	CATIA Representation
CATIA Downstream Application Link	CATIA Process	->	CATIA Part
CATIA Downstream Application Link	CATIA Process	->	CATIA Product

- **Design Table Link: The 'Design Table Link' is a link between a CATIA document and its design table (excel or text file) and describes the relationship between them.**



Examples

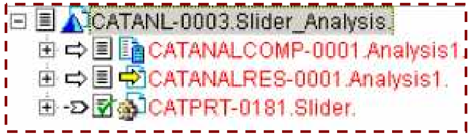
Link	Origin Type	Icon	Destination Type
CATIA Design Table Link	CATIA Internal Cr-->pone		Design Table
CATIA Design Table Link	CATIA Part	-->	Design Table
CATIA Design Table Link	CATIA Analysis	-->	Design Table
CATIA Design Table Link	CATIA Drawing	-->	Design Table
CATIA Design Table Link	CATIA Sheet	-->	Design Table
CATIA Design Table Link	CATIA Process	-->	Design Table
CATIA Design Table Link	CATIA Document	-->	Design Table

Student Notes:

CATIA Links List 3/4

- **Result Link: The ‘Result Link’ corresponds to CATIA output links.**
  - **CATAnalysis->CATAnalysisResult, CATAnalysisComputation…**
  - **CATProcess->NC (aptsource, CATNCCode, tlp)**
  - **CATProcess documentation link (html)**
  - **Knowledge optimization link (CATIADocument->Excel/text)**

Examples



Link	Origin Type	Icon	Destination Type
CATIA Result Link	CATIA Analysis	⇨	CATIA Result
CATIA Result Link	CATIA Analysis	⇨	CATIA Analysis Computation
CATIA Result Link	CATIA Process	⇨	NC
CATIA Result Link	CATIA Process	⇨	Web Document

- **Rule Base Link: The ‘Rule Base Link’ describe a link between an instance of a rule base and its reference.**

Examples

Link	Origin Type	Icon	Destination Type
CATIA Rule Base Link	CATIA Product	-->	CATIA Product
CATIA Rule Base Link	CATIA Product	-->	CATIA Part
CATIA Rule Base Link	CATIA Product	-->	CATIA Representation
CATIA Rule Base Link	CATIA Part	-->	CATIA Part
CATIA Rule Base Link	CATIA Part	-->	CATIA Product
CATIA Rule Base Link	CATIA Drawing	-->	CATIA Drawing
CATIA Rule Base Link	CATIA Drawing	-->	CATIA Part
CATIA Rule Base Link	CATIA Drawing	-->	CATIA Product
CATIA Rule Base Link	CATIA Drawing	-->	CATIA Representation
CATIA Rule Base Link	CATIA Process	-->	CATIA Part
CATIA Rule Base Link	CATIA Process	-->	CATIA Process

Student Notes:

CATIA Links List 4/4

- **Reference Link: The 'Reference Link' includes all the CATIA links and describes the relationship between a CATPart or CATProduct with external documents (other than Excel).**
  - **CATDrawing->Image/OLE**
  - **CATProduct/CATPart->CATMaterial**
  - **CATProcess->NC Manufacturing Part (CATPart)**

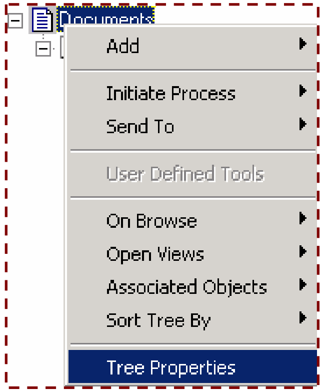
Link	Origin Type	Icon	Destination Type
CATIA Reference Link	CATIA Product	-->	Design Table
CATIA Reference Link	CATIA Product	-->	CATIA Material
CATIA Reference Link	CATIA Product	-->	CATIA Image
CATIA Reference Link	CATIA Internal Component	-->	CAD Document
CATIA Reference Link	CATIA Internal Component	-->	CATIA Material
CATIA Reference Link	CATIA Internal Component	-->	CATIA Image
CATIA Reference Link	CATIA Part	-->	CATIA Material
CATIA Reference Link	CATIA Part	-->	CATIA Image
CATIA Reference Link	CATIA Catalog	-->	CATIA Drawing
CATIA Reference Link	CATIA Catalog	-->	CATIA Material
CATIA Reference Link	CATIA Catalog	-->	CATIA Part
CATIA Reference Link	CATIA Catalog	-->	CATIA model
CATIA Reference Link	CATIA Catalog	-->	CATIA Representation
CATIA Reference Link	CATIA Catalog	-->	CATIA Product
CATIA Reference Link	CATIA Catalog	-->	CATIA Process
CATIA Reference Link	CATIA Catalog	-->	CATIA Image
CATIA Reference Link	CATIA Drawing	-->	Office
CATIA Reference Link	CATIA Drawing	-->	CATIA Image
CATIA Reference Link	CATIA Sheet	-->	Office
CATIA Reference Link	CATIA Sheet	-->	CATIA Image
CATIA Reference Link	CATIA Material	-->	CATIA Image
CATIA Reference Link	CATIA Process	-->	CATIA Document
CATIA Reference Link	CATIA Document	-->	CATIA Document

Student Notes:

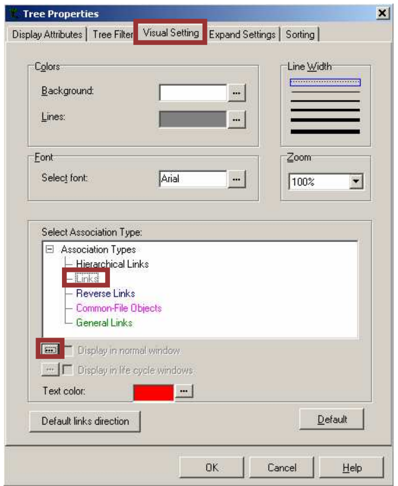
Displaying Link Impact

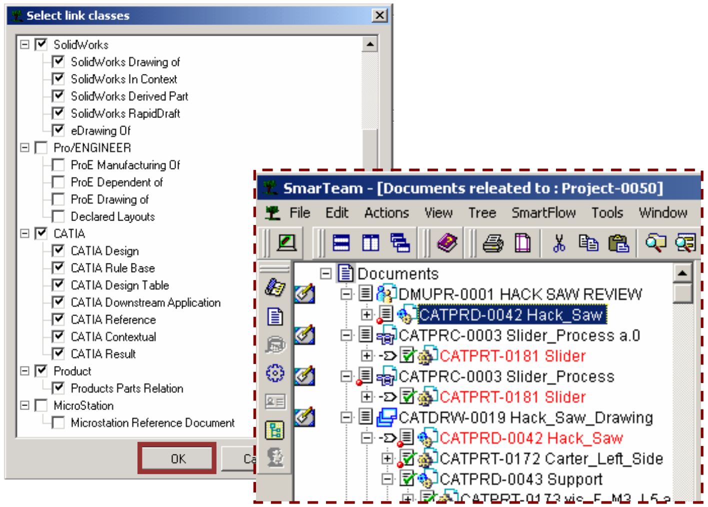
- From CATIA V5, after the Save command, the hierarchical structure is created in Super Class 'Documents'. SMARTEAM automatically generates the CATIA documents structure.
- All kind of links generated in CATIA V5 are saved and exposed in SMARTEAM.
- The impact analysis can be performed by displaying the links of a given object. Through the Tree Properties panel, SMARTEAM - Editor allows you to display/filter CATIA links in the Tree Browser.

You will retrieve in SMARTEAM all the link shown in the following panels.



From the Tree Properties.



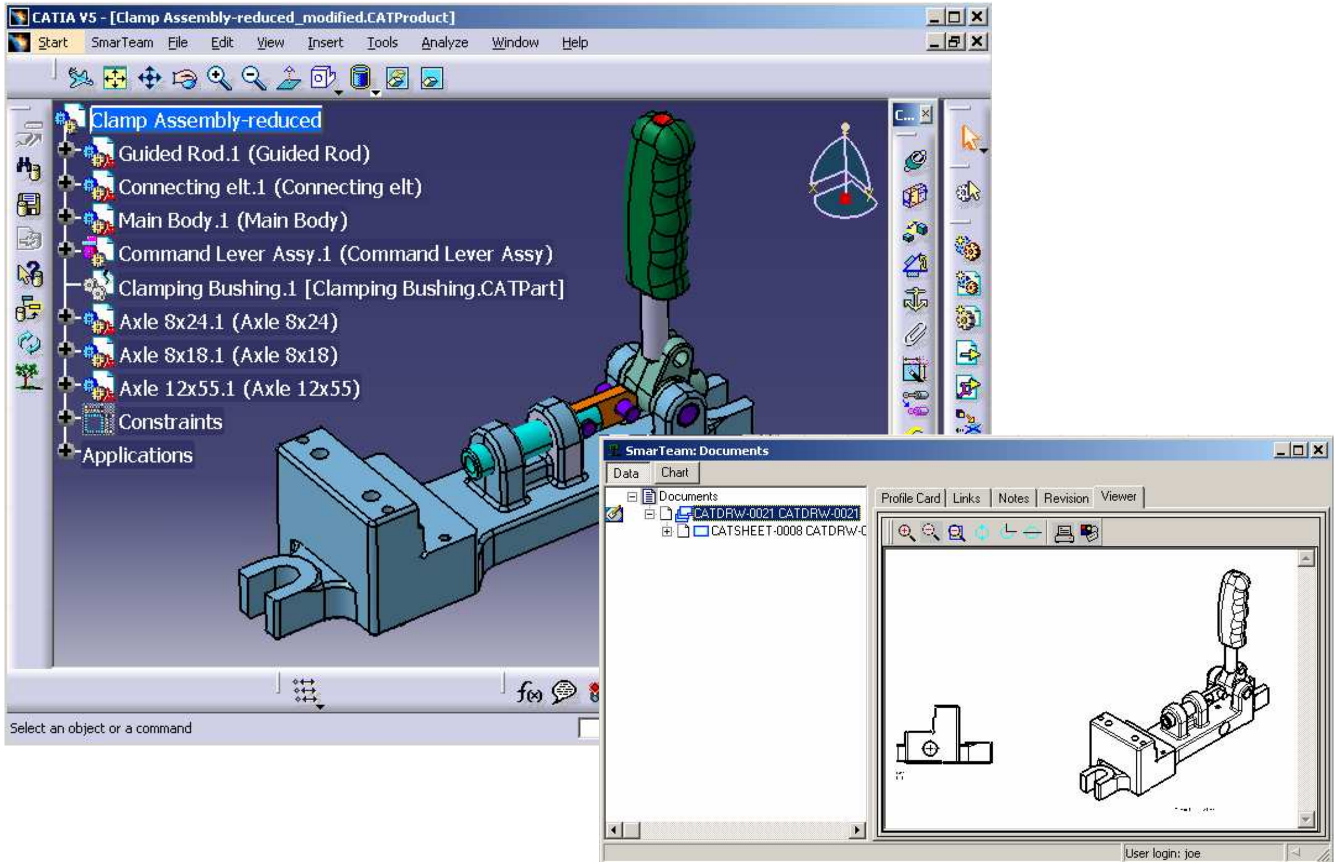


Through this panel, you can display the links integration in the Documents tree.

Student Notes:

CATIA Drawings Links

You will learn how to analyze CATIA drawings in SMARTEAM.



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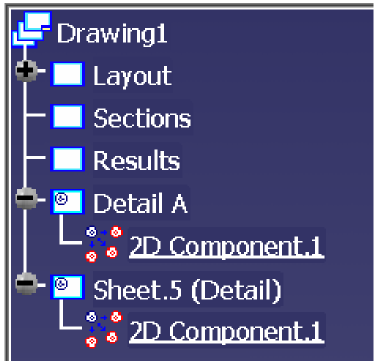
Student Notes:

About CATIA Drawings Links

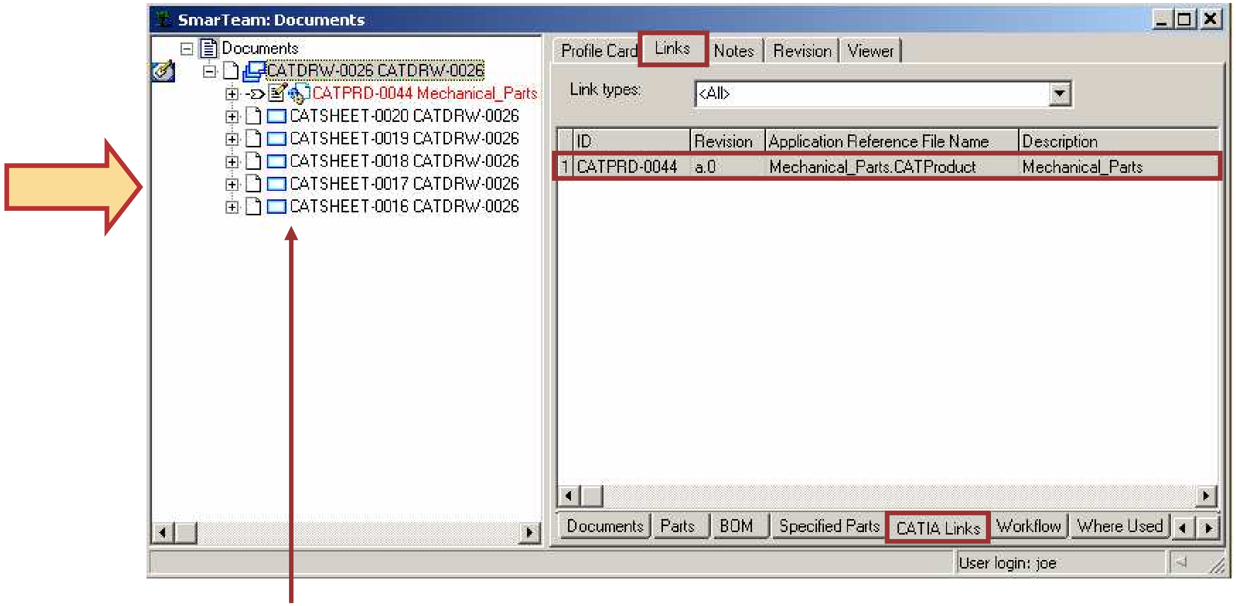
In CATIA, a drawing can be generated at any time from a Product or a Part. If one of them is modified, the drawing will be automatically updated thanks to the associative link.

In SMARTEAM, this mechanism is controlled by the Life Cycle mechanism. When you check out a CATDrawing, the system asks you if you want to replace the parts by the latest available if there is some.

A CATDrawing file contains a structure listing of all the sheets and views contained in the document. After saving in SMARTEAM, all sheets are displayed in the Documents Tree if the Visual Settings are defined for the Links (red links).



CATSheets available in the Specification tree in a CATDrawing

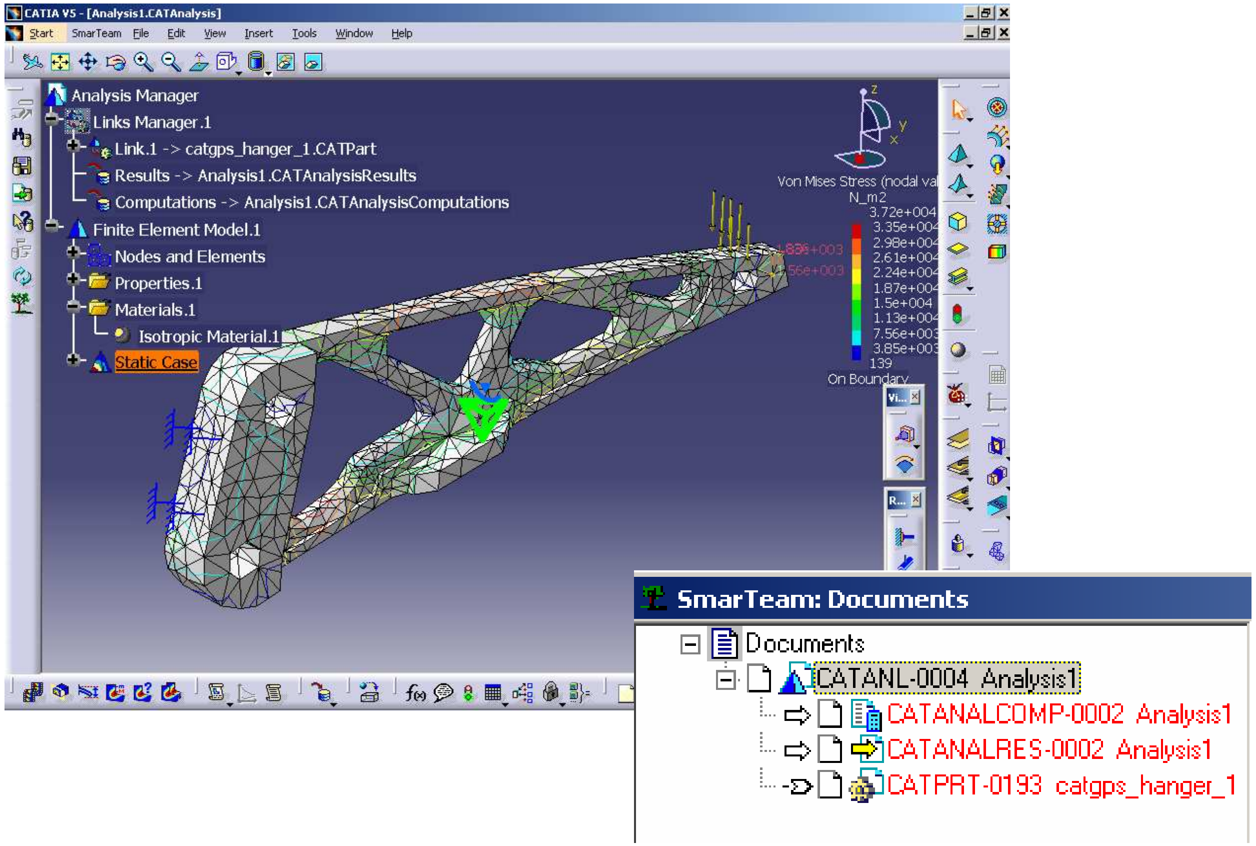


CATSheets stored in SMARTEAM and displayed in Documents Tree

Student Notes:

CATIA Analysis Links

You will learn how to handle CATIA analysis links in SMARTEAM.



Student Notes:

About CATIA Analysis Links

SMARTEAM is able to save all CATIA Analysis links. These links are visible in the Documents tree or in the CATIA Links tab.

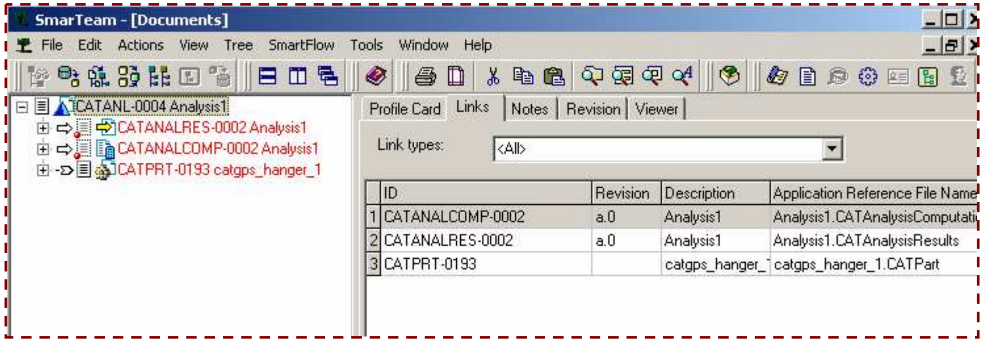
In CATIA, the composition of a CATAnalysis file displayed in the Specification tree is defined by:

- **Links Manager folder**
- **Finite Element folder**

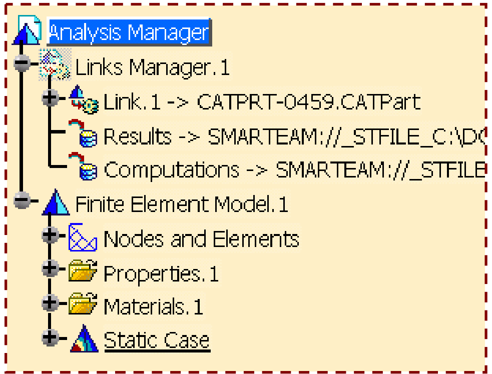
In the Links Manager folder, you can see:

- **A persistent CATPart or CATProduct link. A CATAnalysis file is always linked to a geometry.**
- **An occasional CATAnalysisResults and CATAnalysisComputations links. These files are created when the user launches a computation.**

In Finite Element folder, you can see all parameters applied to define the Structural Analysis such as Mesh, Forces, materials …



Documents Tree in SMARTEAM

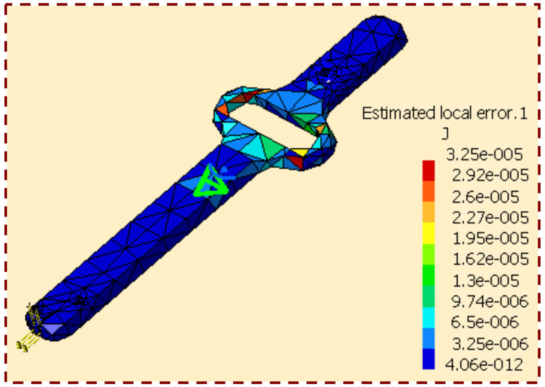


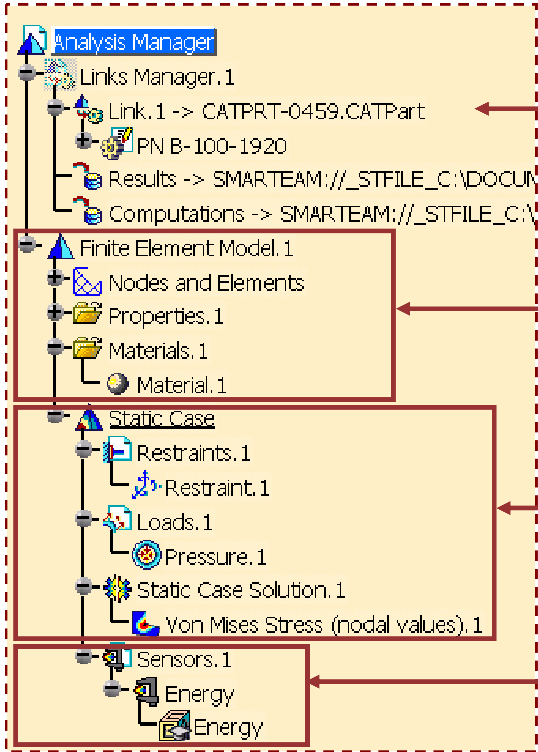
Specification Tree in CATIA V5

Student Notes:

Breakdown Structure of CATIA Analysis File

The Specification tree for a CATIA Analysis file is detailed here below:





Link to the part

Link to the Result file

Link to the Computation file

Kind of Meshes and Properties that are applied to your part

Here you will find the restraints, loads etc… you have applied to your parts

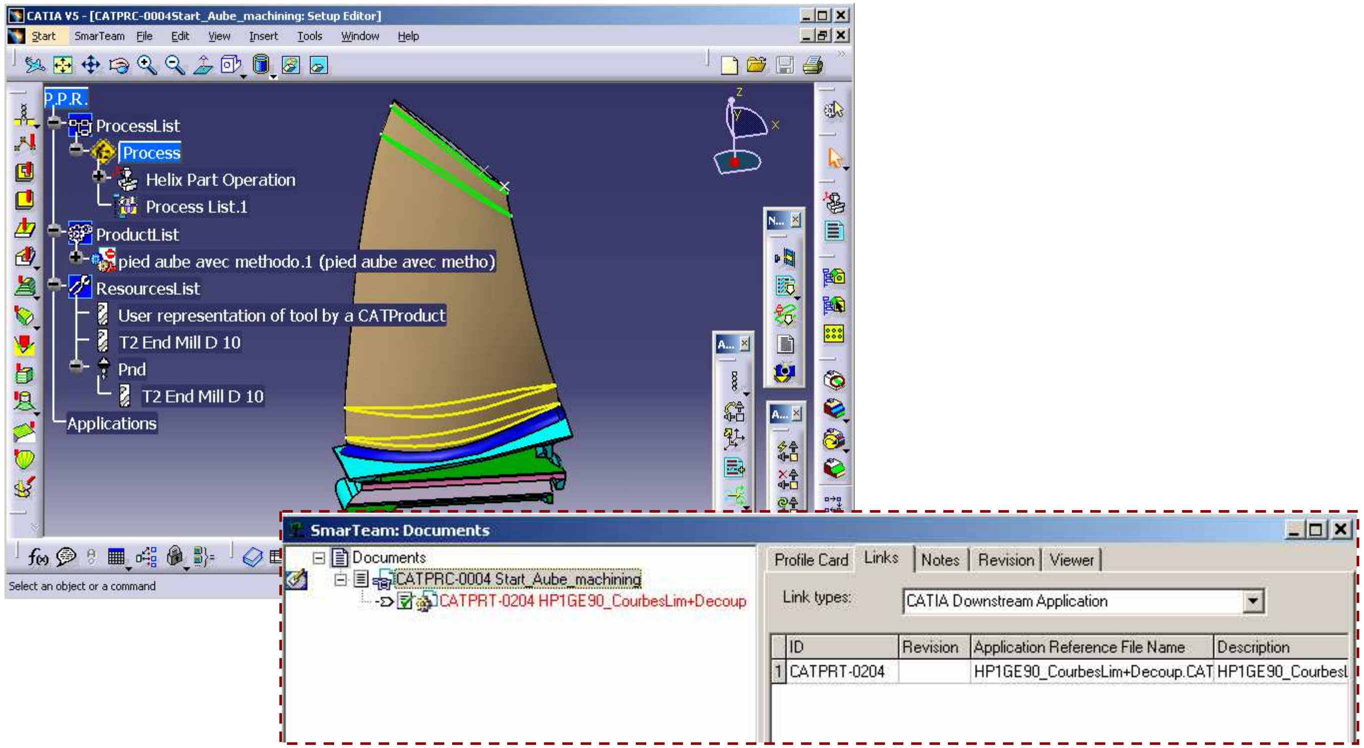
Before computing an Analysis, you can apply sensors on predefined parameters

Specification Tree of CATAnalysis file

Student Notes:

CATIA Process Links

You will discover how to review CATIA Process links.

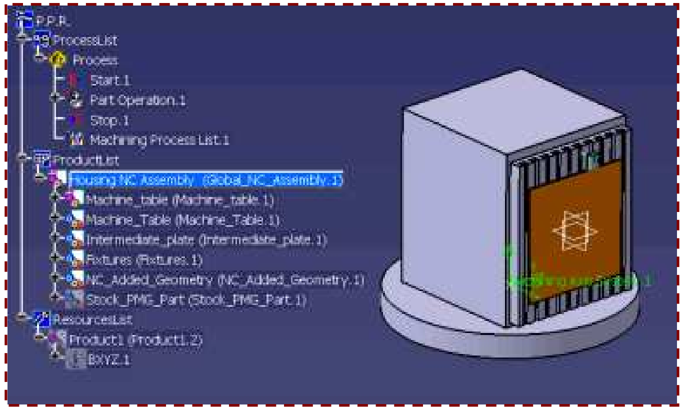


Student Notes:

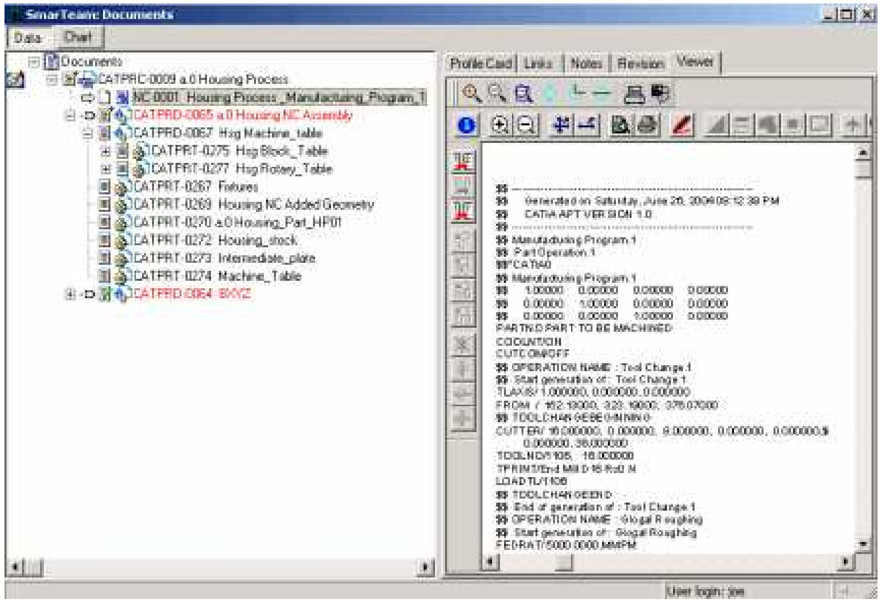
About CATIA Process Links

When a CATProcess is initiated a CATPart or a CATProcess are automatically associated and an instance is created in the Product List folder in the Specification Tree in CATIA

NC CATProcess document and related linked documents (such as APT source, NC code, tool user representation, machine) can be automatically managed in SMARTEAM.



Specification Tree in CATIA V5

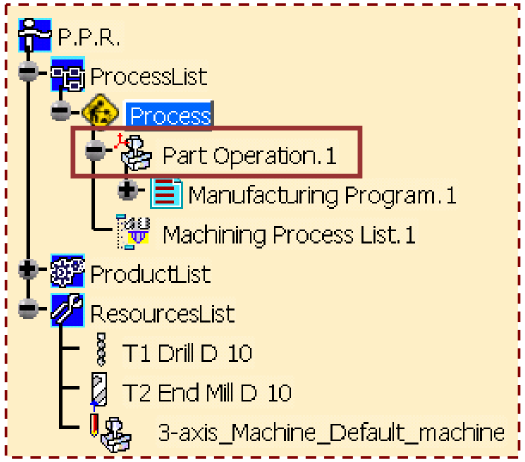


Documents Tree in SMARTEAM

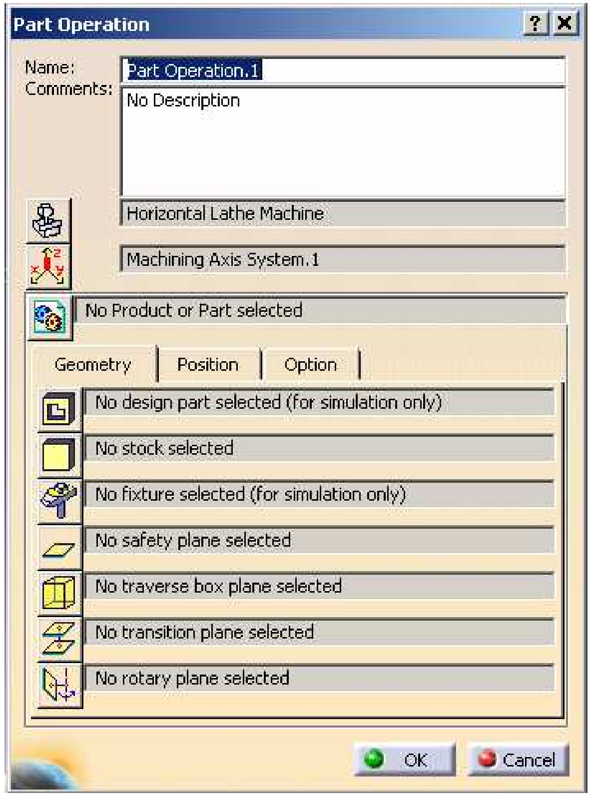
Student Notes:

Interoperability with Part Operation Panel (1/4)

Through the Part Operation panel, it's possible to search a machine, a Part or a Product directly in SMARTEAM



Specification Tree in CATIA Process

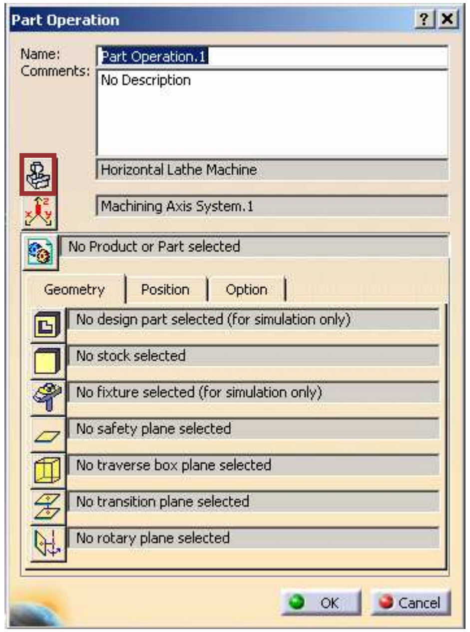


Part Operation panel

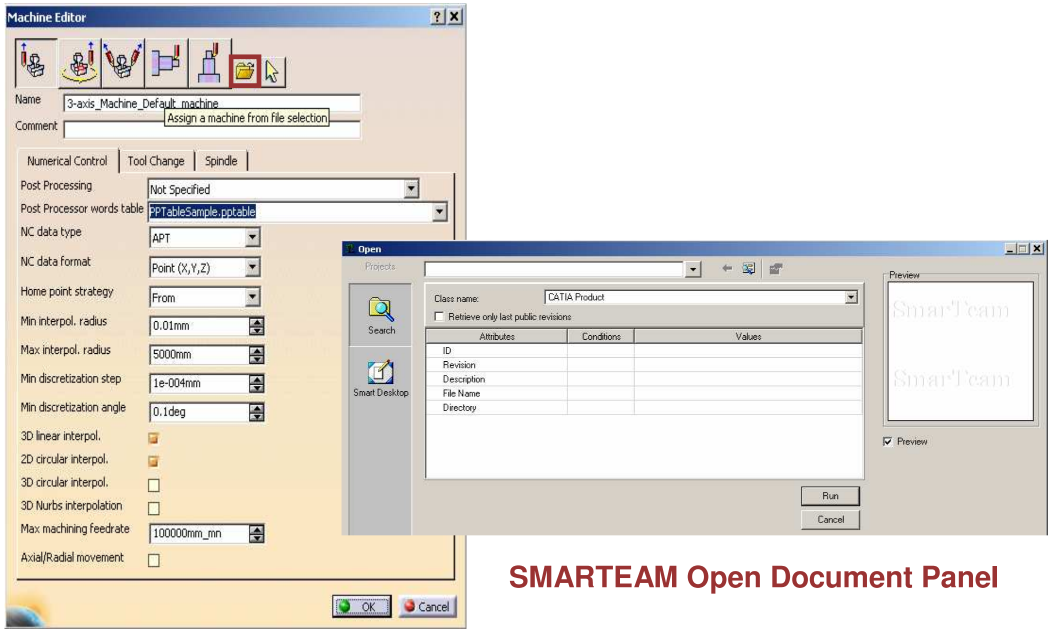
Student Notes:

Interoperability with Part Operation Panel (2/4)

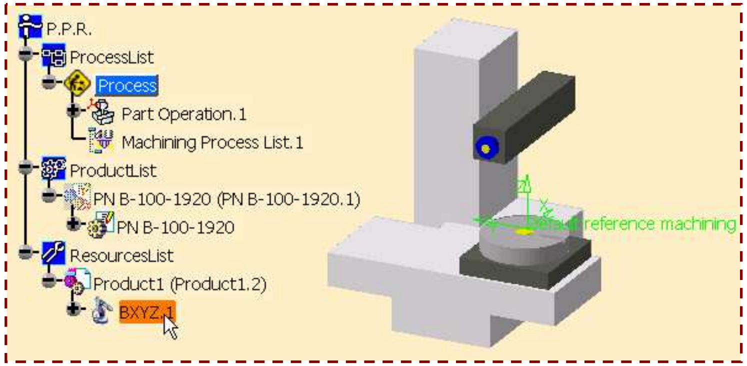
In this example, you can see how to assign a machine from file selection option



Part Operation Panel



SMARTEAM Open Document Panel

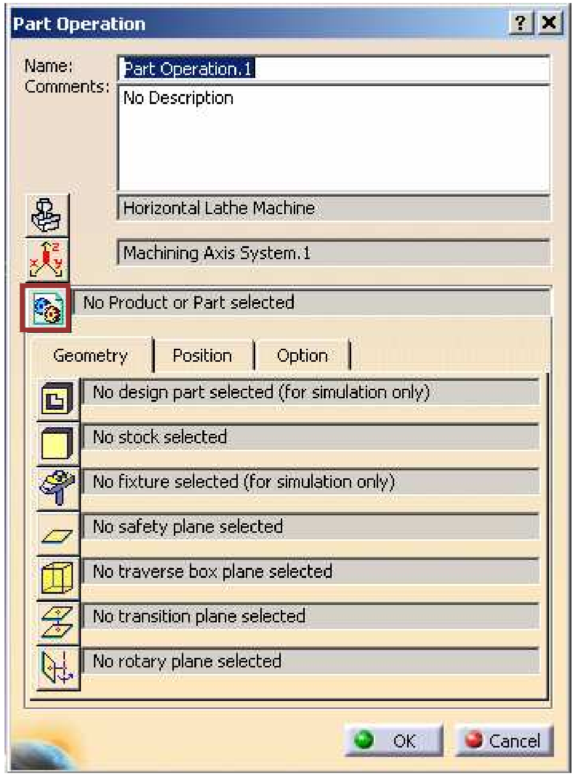


The Machine (CATProduct) is instantiated into the Process and user can proceed further on.

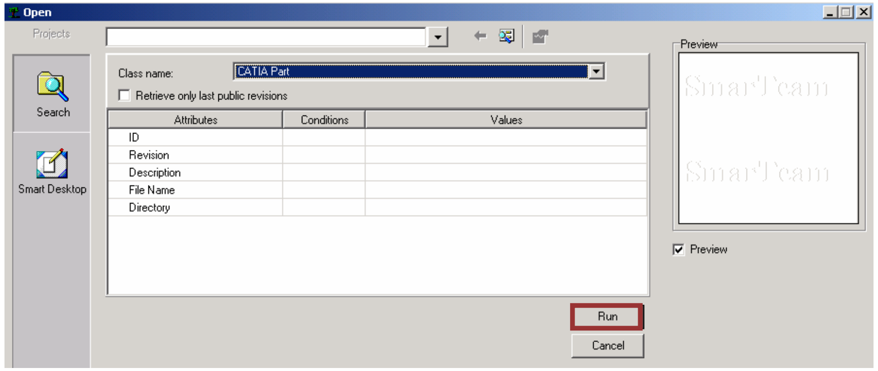
Student Notes:

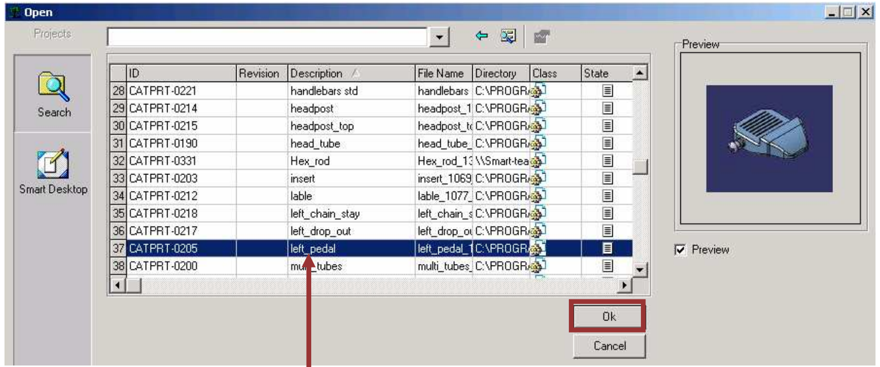
Interoperability with Part Operation Panel (3/4)

In this example, you can see how to assign a CATIA Product or a Part to a Process.



Part Operation Panel

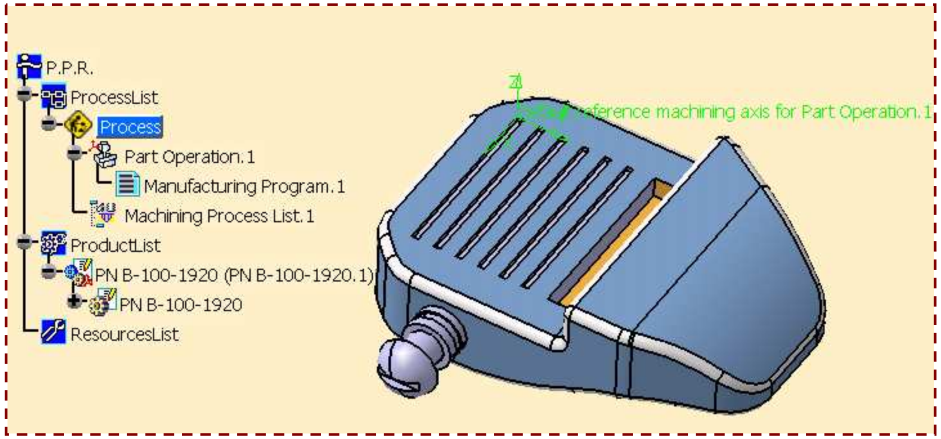




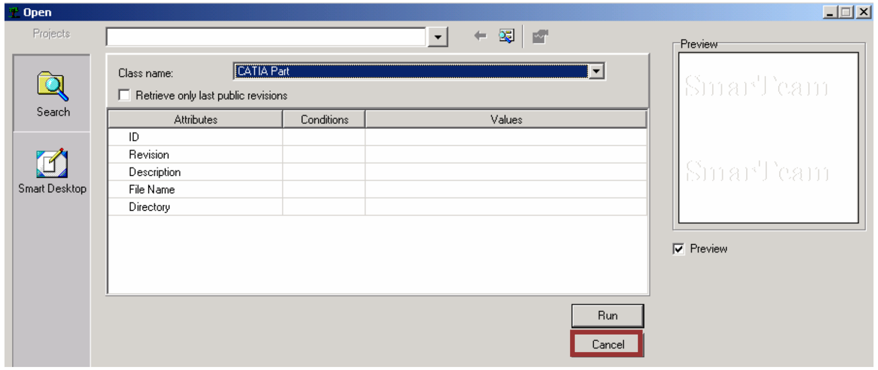
The CATIA Part to be instantiated into the Process is selected in the SMARTEAM Open Document Panel and validated OK.

Student Notes:

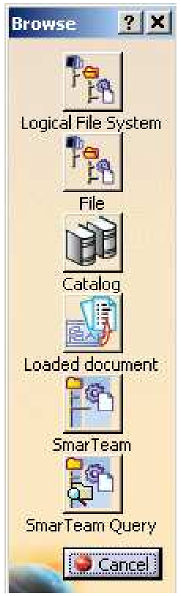
Interoperability with Part Operation Panel (4/4)



The Part is instantiated into the Process and user can proceed with including the Manufacturing Program



SMARTEAM Open Document Panel



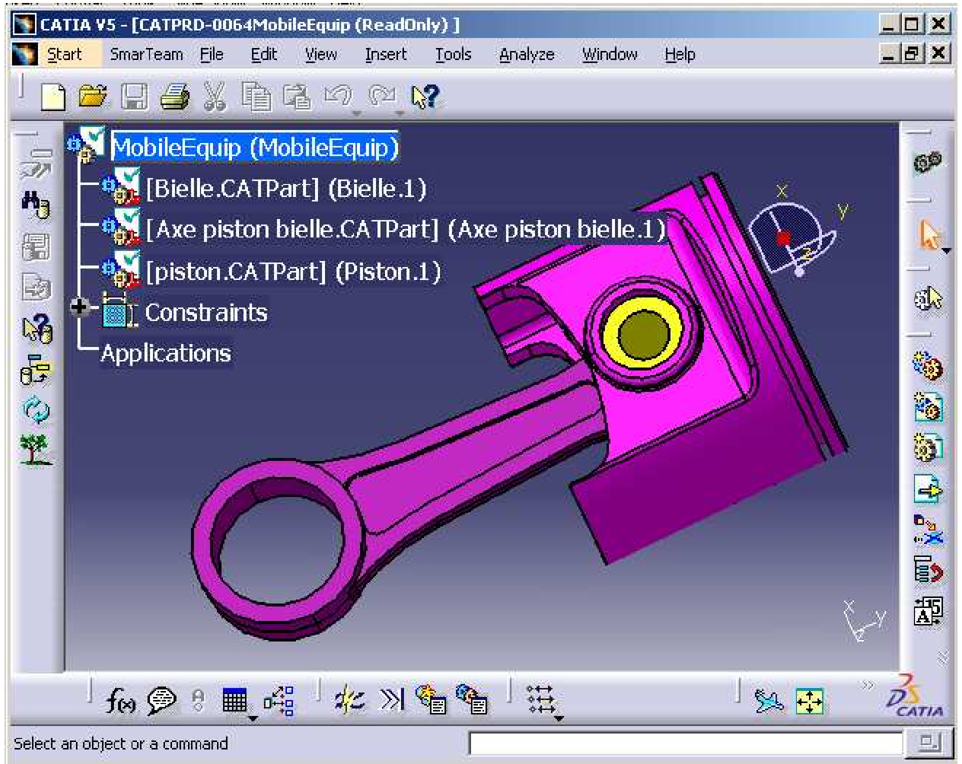
Browse toolbar

If you cancel the search in SMARTEAM Open Document Panel, the Browse toolbar is displayed.

Student Notes:

Using Cache Management

You will learn how to open a large assembly in light representation.

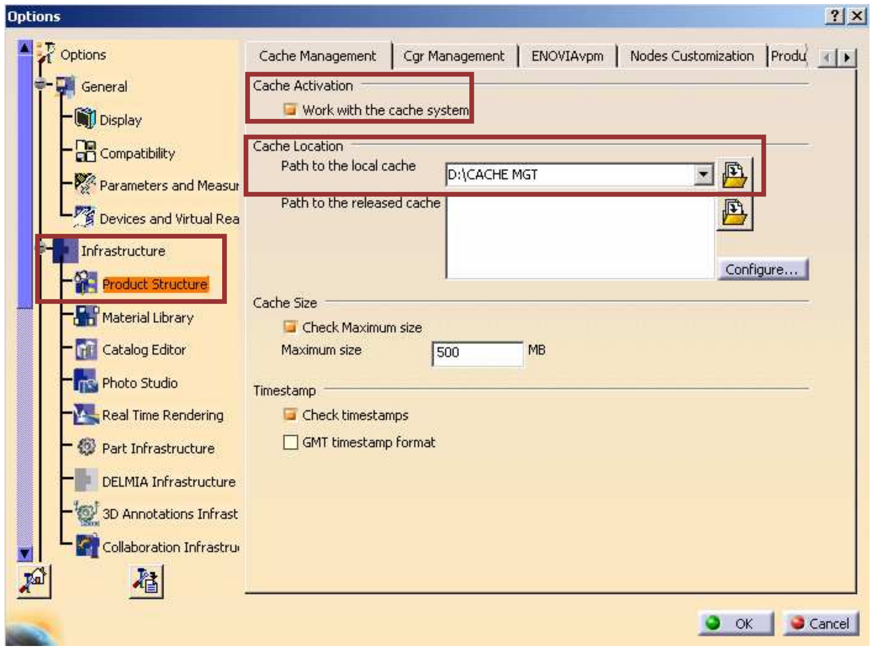


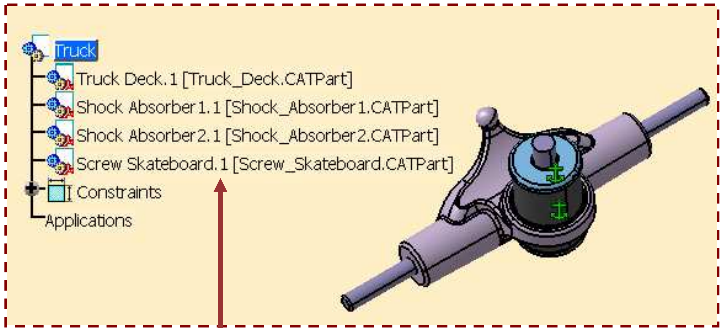
Student Notes:

About a Cache System

Using a Cache System enables the user to manage heavy assemblies, by checking into the vault only the relevant documents. Therefore, it reduces the time required to load data.

Automatically during open, CATIA V5 creates a *.cgr* file in a specific folder. This created file is called CATIA Graphical Representation.





The CATIA Product is opened in visualization mode with no geometry loaded (observe the CATIA specification tree)

The cache system options are defined via the Cache Management tab in the CATIA V5 Tools/ Options panel

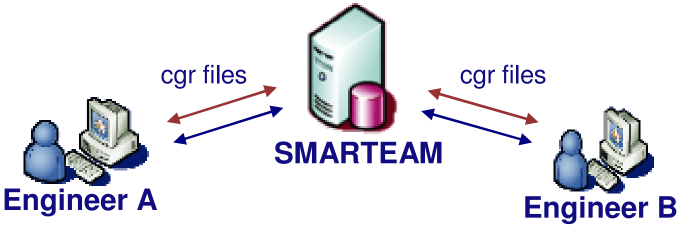
A Part opened in visualization mode can be easily switched to design mode by a simple double click.

Student Notes:

CATIA Cache Management with SMARTEAM

- CATIA Cache management can be generated and managed by SMARTEAM for data saved under SMARTEAM.
- The process is that CATIA copies the .cgr files from work or vault folder and puts in Cache folder with another name. It is a simple copy process not a creation process.

- If a user checks in a CATPart file, the associated cgr files will also go to the Check In vault.
- If the same or another user opens a CATProduct file referencing the CATPart files, the associated cgr will be used (in visualization mode) instead of the CATPart file.



When the cache is active:

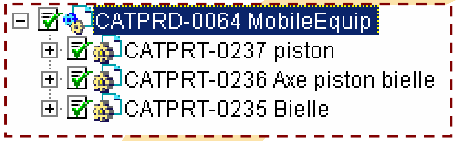
- The representation is checked in the local cache and the possible cache releases.
- If the representation is not found in the local cache or if the time stamp is not valid, the representation is extracted from the vault and copied back in the local cache.
- If the representation is not found in the vault, the master document is loaded in CATIA V5 in order to be tessellated and updated in the local cache.

Student Notes:

How to Use Cache Management

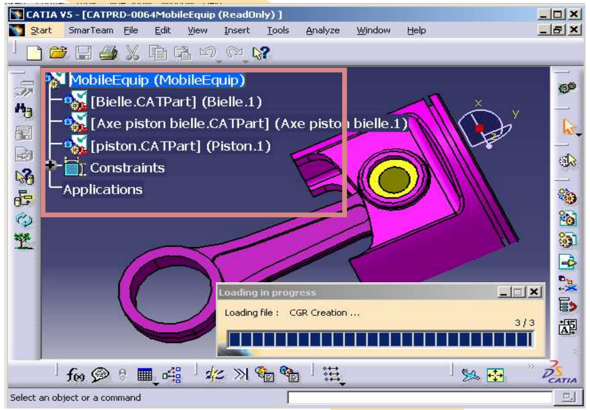
This function allows the user to manage the cache of data saved under SMARTEAM; cgr files won't be generated under CATIA cache folder

1 Select a product in Documents tree

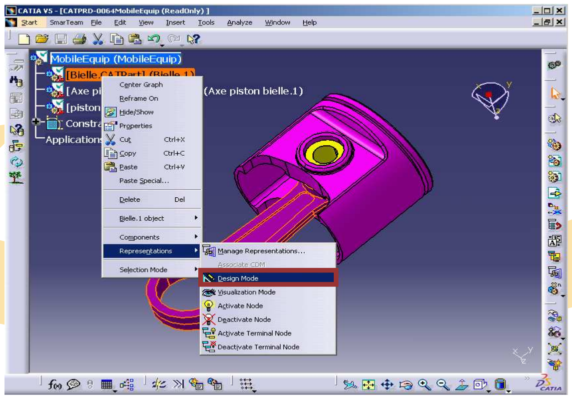


2 Select File Operation/Open to load the product Read Only

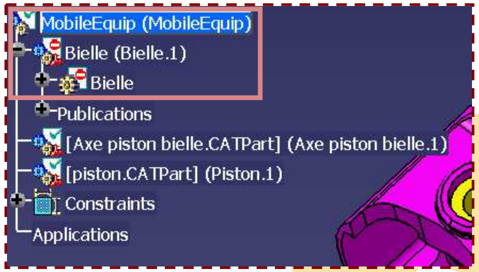
3 The product is opened in CATIA V5.The Visualization Mode uses documents in cgr format. The geometry is not available.



4 Select Design Mode option in the contextual menu on the selected part to modify geometry



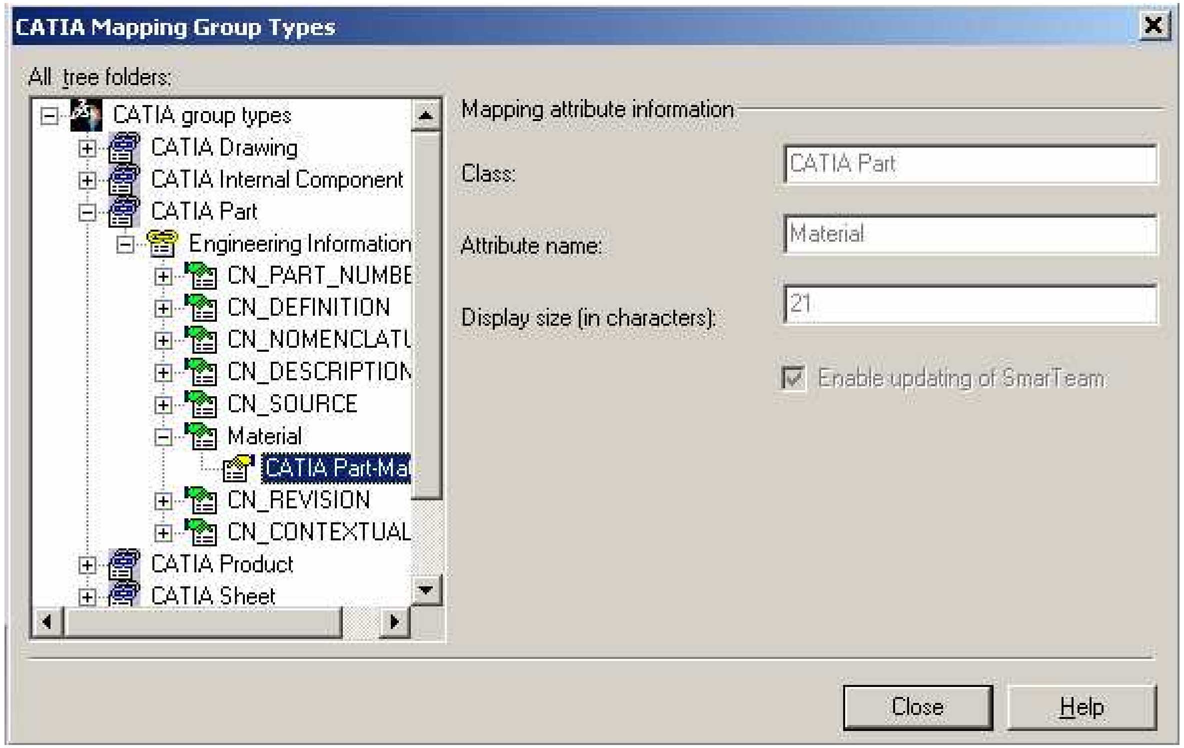
5 The geometric data is available in Specification Tree. All workbench commands are available if this mode is activated.



Student Notes:

Mapping of Properties

You will see how to map properties between CATIA and SMARTEAM.



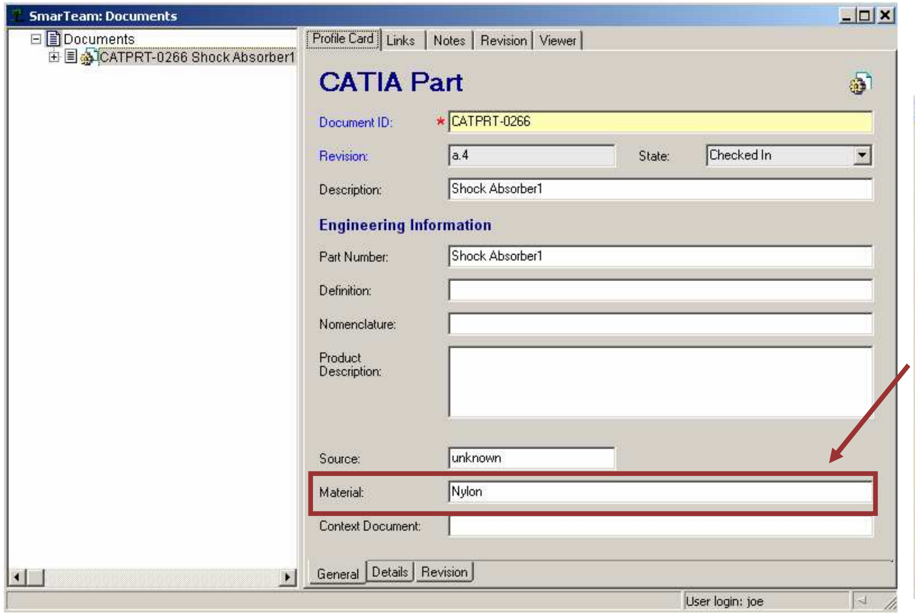
Student Notes:

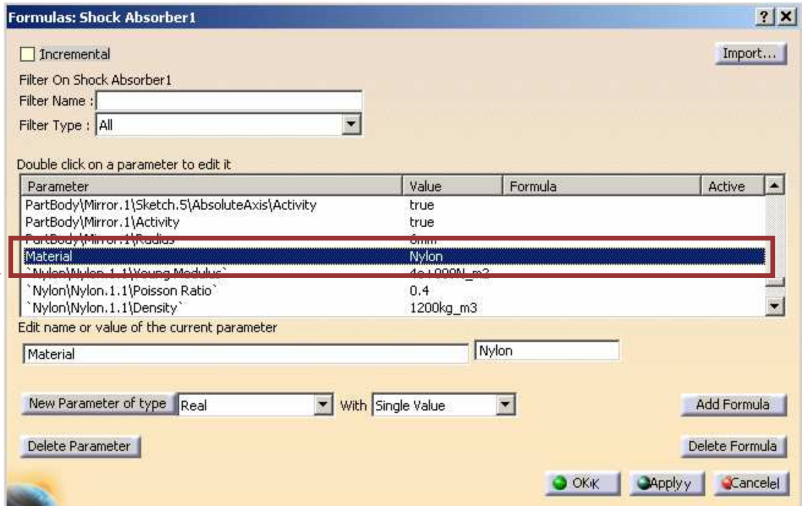
About Mapping of Properties

The Mapping has the ability to connect properties made in CATIA V5 with attributes defined in SMARTEAM database.

For each mapped property, the mapping direction has to be defined:

- **SMARTEAM =>CATIA : CATIA property values come from the SMARTEAM database**
- **SMARTEAM <=CATIA: CATIA property values are written to the SMARTEAM database**
- **CATIA ⇔ SMARTEAM : Both mapping directions are available**





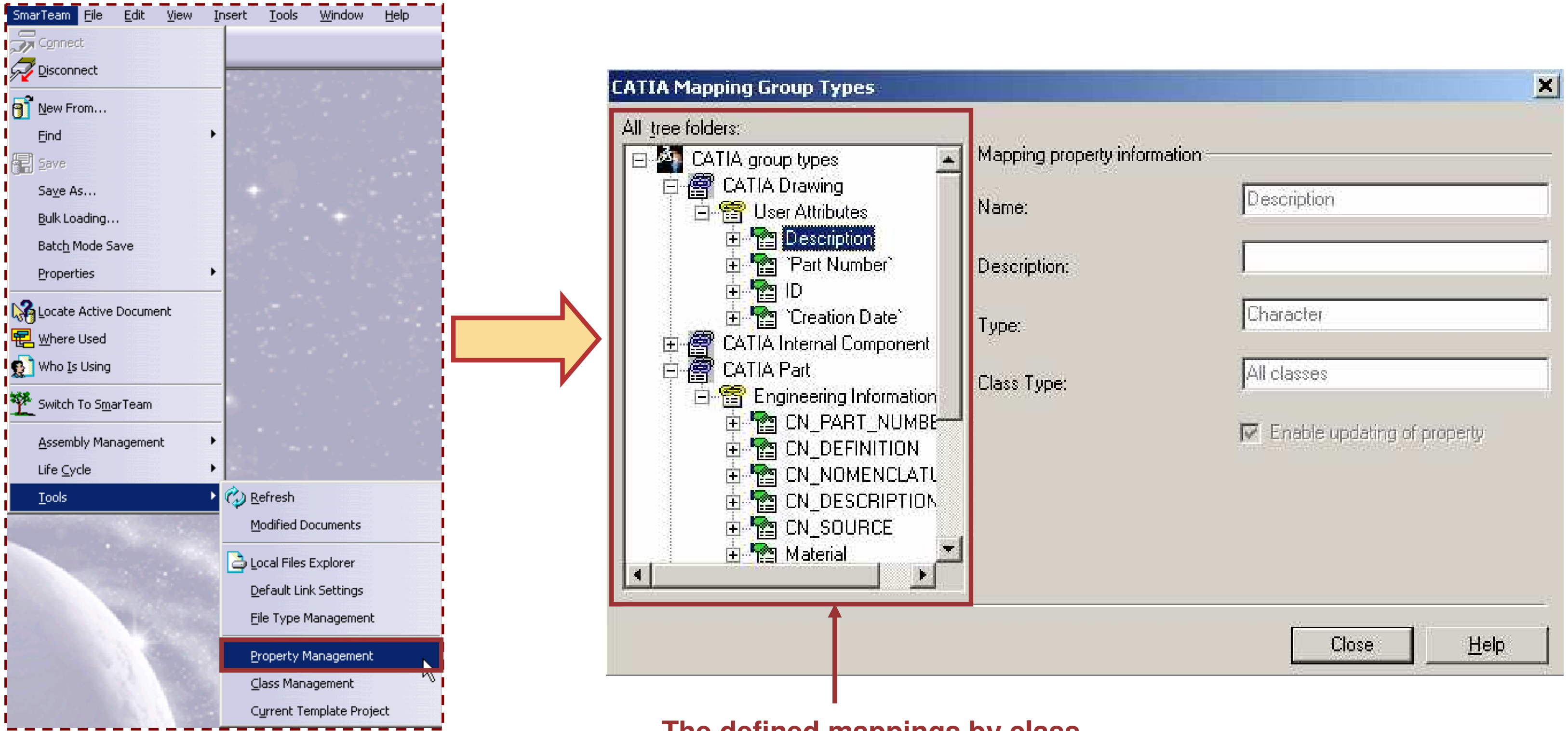
This task can only be performed by the administrator.

Student Notes:

Mapping Properties (1/2)

The definition of the mapped attributes and properties, as well as the direction, is defined by the Administrator. However, any user can see the defined mapping.

You can see which properties and attributes are mapped:



Select 'SmarTeam' menu > Tools > Property Management

The defined mappings by class

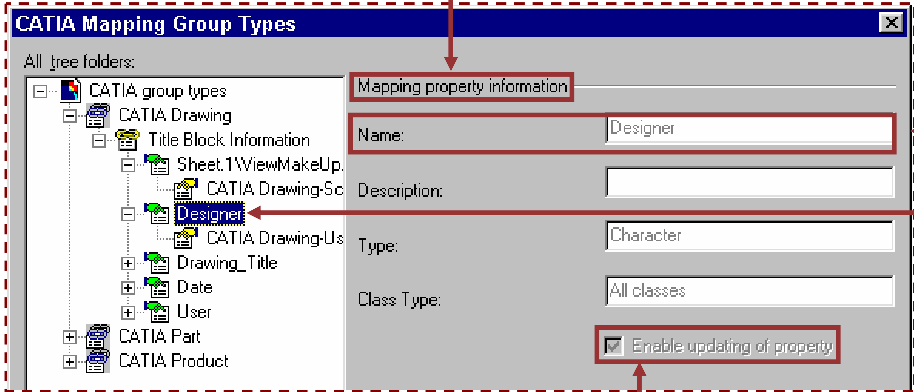
Student Notes:

Mapping Properties (2/2)

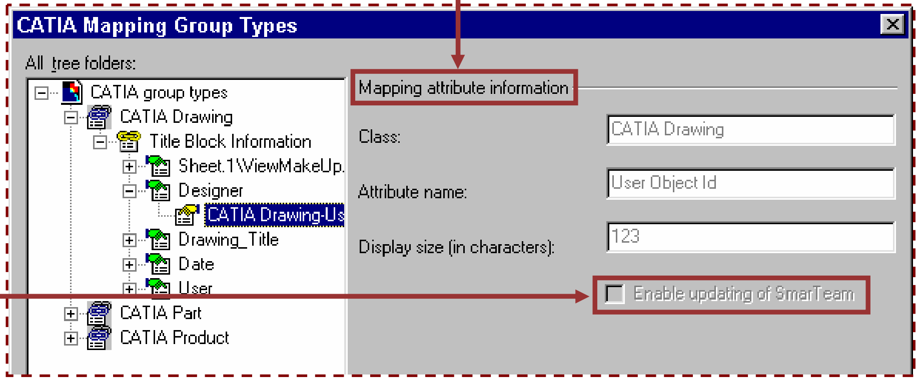
Some useful details like Properties and attributes which are mapped are easy to identify.

The CATIA Property being mapped

Name of CATIA Property



Corresponding SMARTEAM attribute

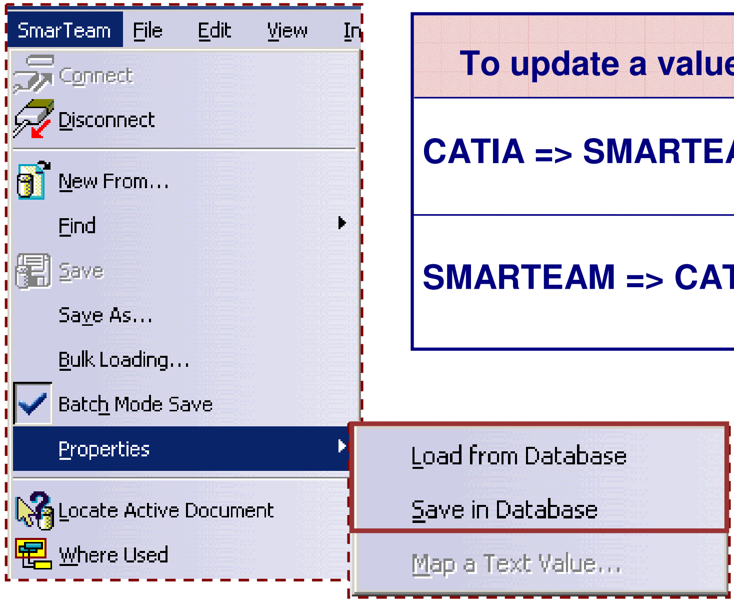


Indicates the direction in which the mapping can be made. Enable updating of Property or Enable updating of SMARTEAM.

Student Notes:

Updating Properties or Attributes

When performing a modification in either CATIA or SMARTEAM, there are different ways to update the object or document which was modified.



Menu to update the attribute value

To update a value	Command to be used
CATIA => SMARTEAM	'SmarTeam' / Save in SMARTEAM / Properties / Save in Database
SMARTEAM => CATIA	'SmarTeam' menu / Properties / Load from Database

Load from Database or Save in Database are allowed only if the authorization has been granted by the administrator.

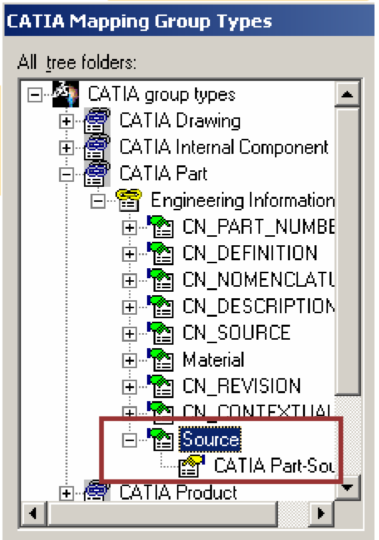
Student Notes:

How to Map Properties from CATIA to SMARTEAM (1/2)

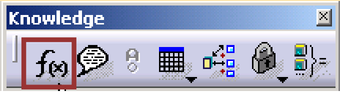
You are going to define the mapping between a CATIA parameter and an attribute visible in a CATIA Part Profile Card - 'Source'

In the CATIA document, the Property has to exist with the same name as the one entered in Property Management.

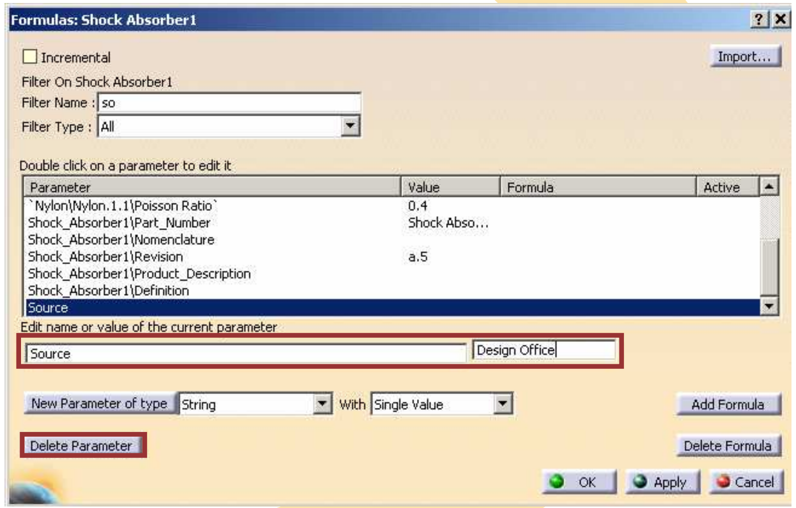
1 In CATIA Mapping Group Types panel under CATIA Part class, create a property



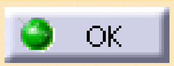
2 Click on Formula icon in Knowledge toolbar



3 Click on New Parameter of type button and enter the description defined before in CATIA Mapping Group Types



4 Confirm by clicking on "OK"

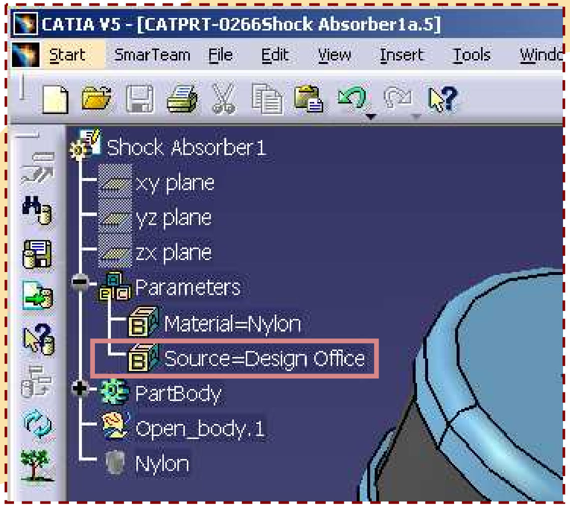


Student Notes:

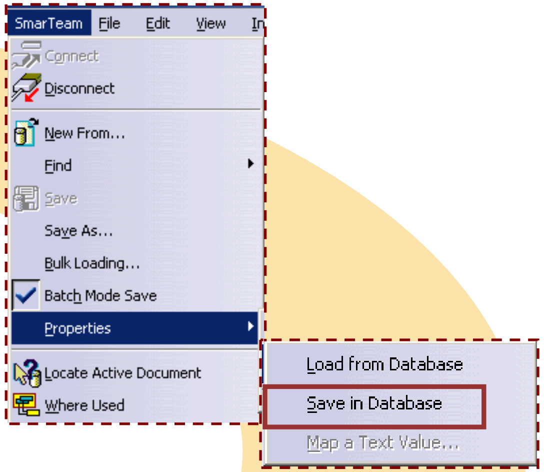
How to Map Properties from CATIA to SMARTEAM (2/2)

In the CATIA document, the Property has to exist with the same name as the one entered in Property Management

5 The parameter appears in Specification Tree

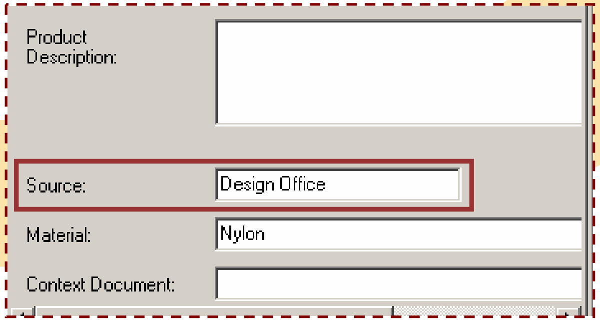


6 Save from SMARTEAM toolbar



7 Menu to update value in SMARTEAM

8 Check the Profile card for CATIA Part, the value of attribute is updated

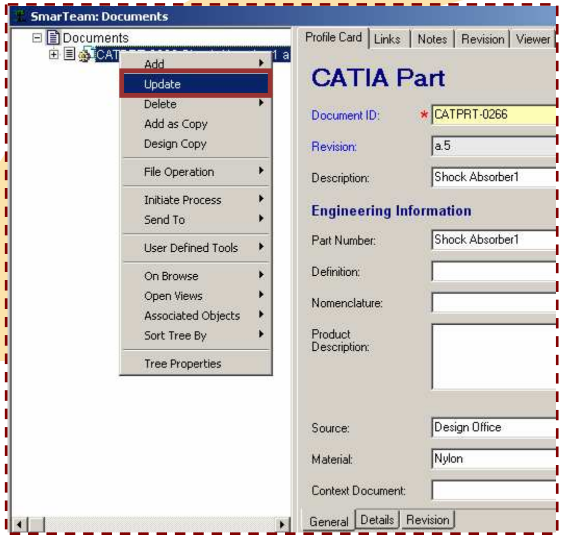


How to Map Properties from SMARTEAM to CATIA

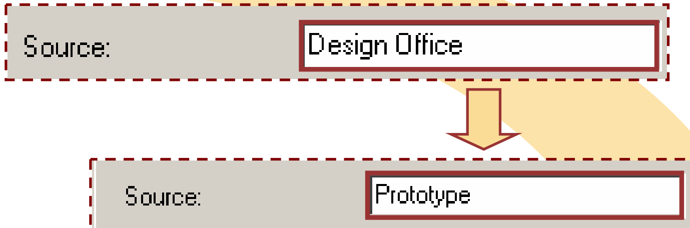
When you have the attribute mapping in place between CATIA V5 and SMARTEAM, you can change the attribute value in SMARTEAM and have it retrieved in CATIA V5.

Warning: the name of the mapped property has to correspond to the name entered by the administrator in the CATIA Mapping Group Types.

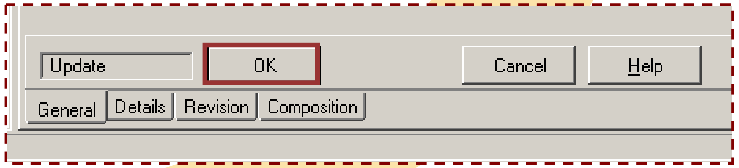
1 A component has been saved in the database. Update its Profile Card to add or modify information.



2 Change the value of the mapped attribute.



3 In the Profile card for the CATPart, confirm by clicking OK.



4 To retrieve the information in the CATIA document,
if the document was not opened: File Operation > Edit;
OR
if the document is already opened in session:
SmarTeam > Properties > Load from Database.

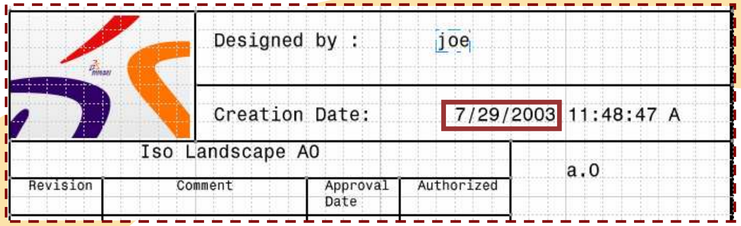
Student Notes:

Student Notes:

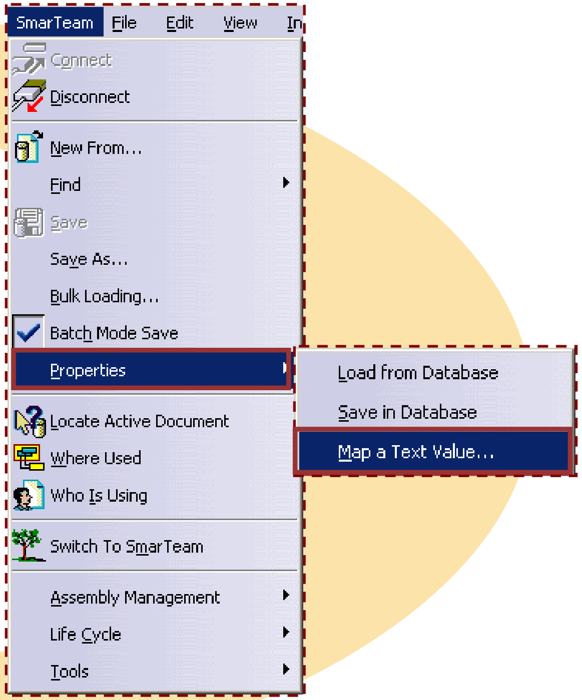
How to Manage Drawing's Data

The SMARTEAM Administrator can map a drawing's text with any of the CATIA Drawing attributes in the SMARTEAM database.

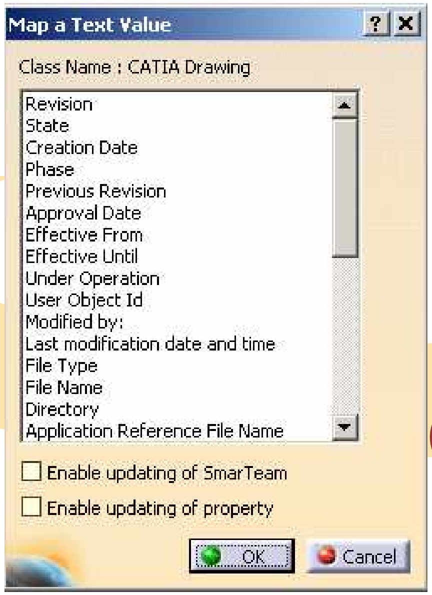
1 Select a text in the drawing



2 In SmarTeam menu, select Properties – Map a Text Value



3 The Map a Text Value panel displays. Select the adapted attribute define in CATIA Drawing class

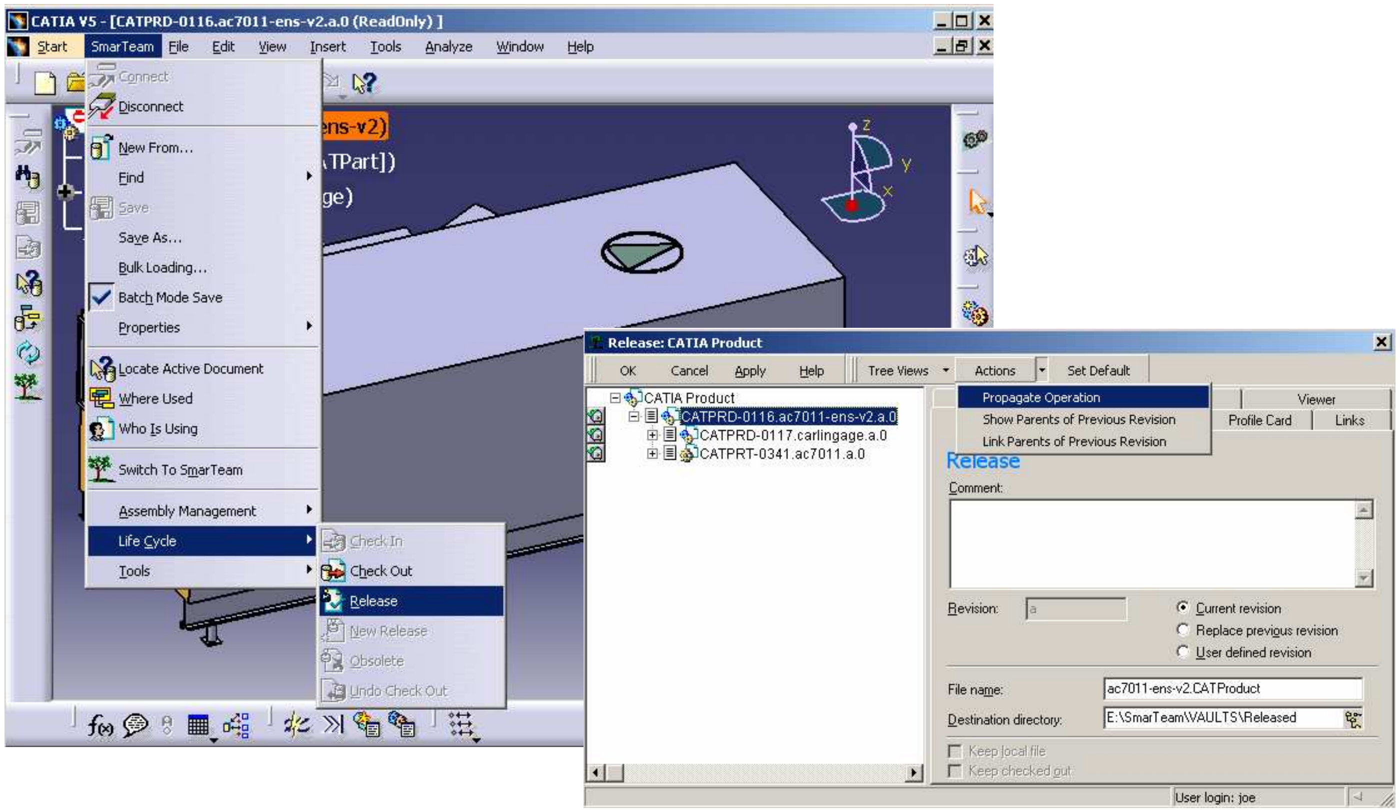


4 A CATIA property is created inside the Drawing

Student Notes:

Lifecycle Operation

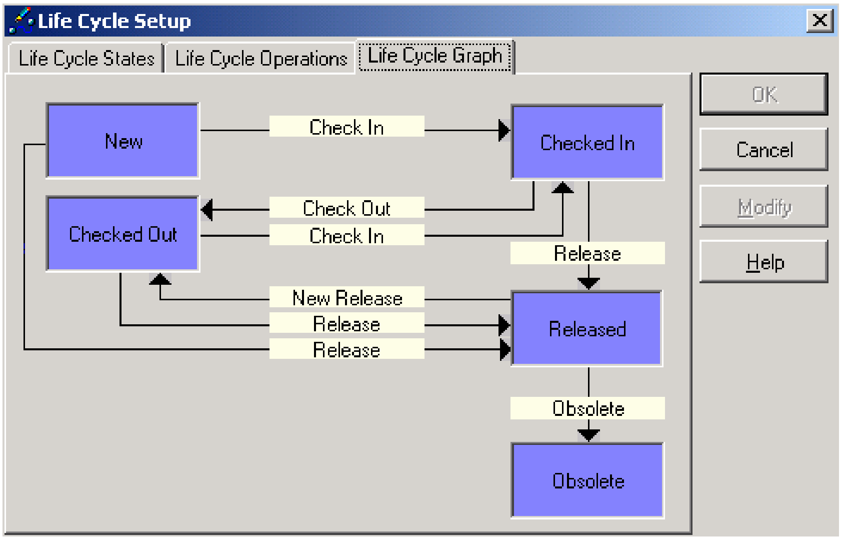
You will learn about the different Lifecycle Operation modes and ways to manage Assemblies.



Student Notes:

About Lifecycle Operations

- Maintaining security and control over files is of utmost importance in an enterprise and SMARTEAM provides an electronic vault for this purpose. New versions of each file as it is revised are created and this helps in protection from unauthorized modifications.
- The electronic vault ensures that only those persons with access permission may access a file, and that a file cannot be accessed by more than one person at a time.
- By mirroring the physical process of product management, SMARTEAM - Editor uses the vaults, check in, check out, and release functions to manage the Lifecycle of revision manageable objects.



Lifecycle Rules Setup panel

Student Notes:

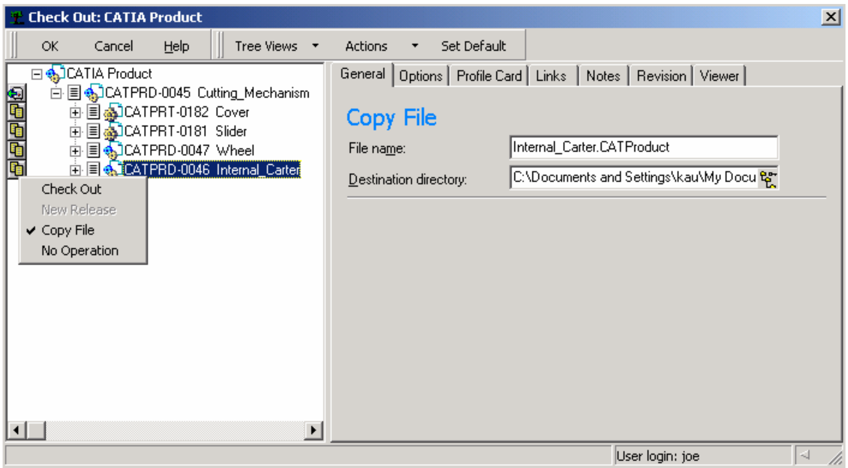
Managing an Assembly

SMARTEAM provides for the Lifecycle Management with which you can perform any revision operation on an object and all its children simultaneously using the Propagate Operation option. This is essential to protect the integrity of an assembly together along with all of its children.

For example, if you have an Engine that has many children, you can perform any revision operation (such as Check In, Check Out, Release) on the Engine together with all its components simultaneously.

You can change the lifecycle operation on the children based on the choice executed on the root Product:

Check Out	**Copy File**
No Operation	**Release**
New Release	**Obsolete**



Change operation by:

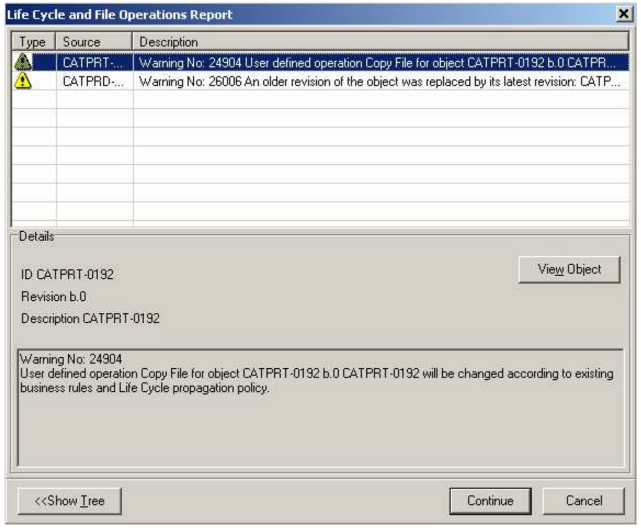
- **Clicking on the icon**
- **Contextual menu on the icon**

On a Check In or Release operation, the operation on the Product is automatically propagated to all of its children (administrator role).

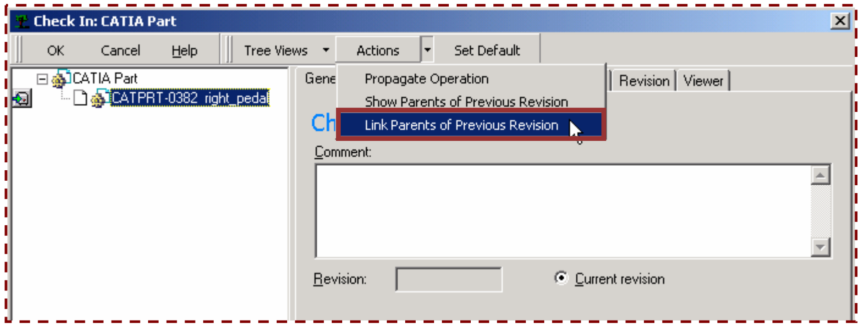
Modifications on an Assembly

When a component of an assembly is modified, you have two methods to reconcile the assembly with the latest revision of its components.

First method:
- **Check Out the Component**
- **Perform the modification**
- **Check it back In**
  - **Check Out the assembly: the latest version of the component is automatically selected**
  - **A « Revision Replacement Report » is generated**



Second method:
- **Check Out the component**
- **Perform the modification**
- **On Check In operation, select 'Action / Link Parents of Previous Revision'**
  - **The parent assembly points to this version of the component, without performing a Check Out of the assembly**



Student Notes:

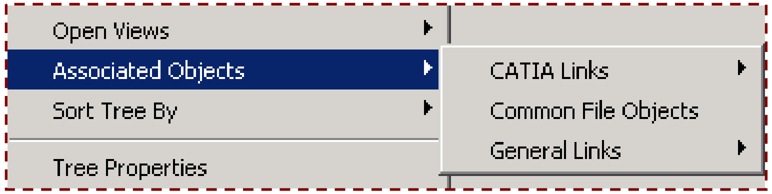
Student Notes:

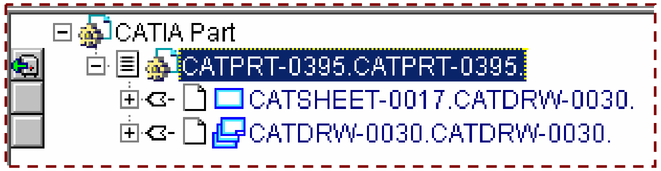
How to Revise Associated Objects

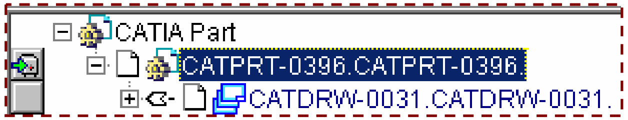
As you revise your documents, SMARTEAM protects the relationship between Associated Objects.

When you perform a lifecycle operation on a document (such as Check In, Check Out and Release) you can display and manage the Associated Objects.

To display Associated Objects:
From advanced lifecycle window, right-click to display the contextual menu.
Select Associated Objects and choose the object type you wish to display.







The CATSheet and the CATDrawing are exposed in the tree

There is only the CATDrawing exposed in the tree

Each Associated Object is color-coded for easy recognition.
The administrator can force the system to display a CATDrawing.

Student Notes:

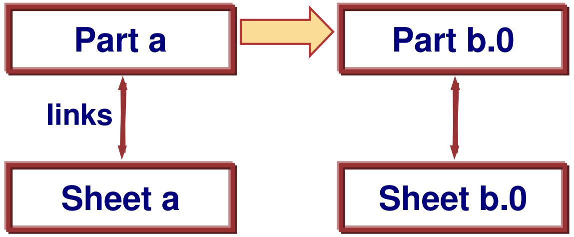
How to Manage Associated Objects During Lifecycle (1/2)

How is a Part and its associated Drawing handled during Lifecycle operations?

If a CATIA Part and its associated Drawing and Sheet are released, you have two scenarios for modifications

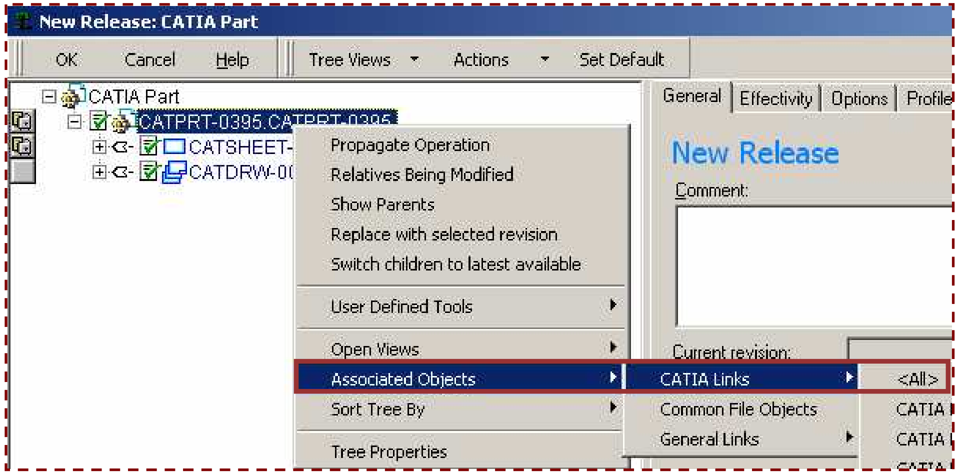
1. **Make ‘New Release’ of drawing and part**
2. **Modify the part, and synchronize the drawing later**

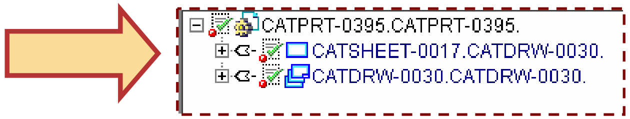
In the tree, you have the choice to select the drawing or the sheet. This operation propagates on the two attached objects.



1 **Make ‘New Release’ of both sheet and part together.**

In ‘New Release’ panel, select Associated Objects > CATIA Links > All and choose ‘New Release’ for the sheet as well as for the part.





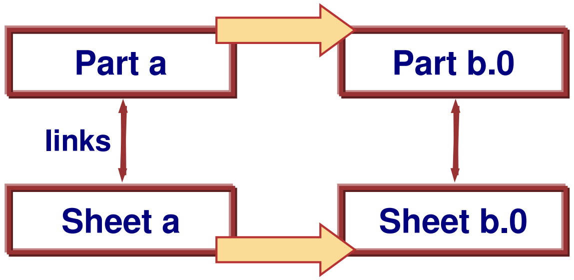
The Documents tree is updated

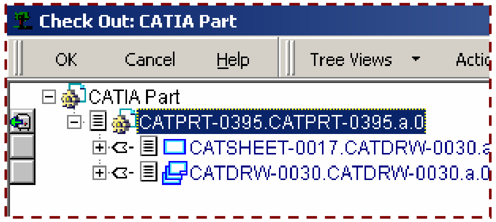
Student Notes:

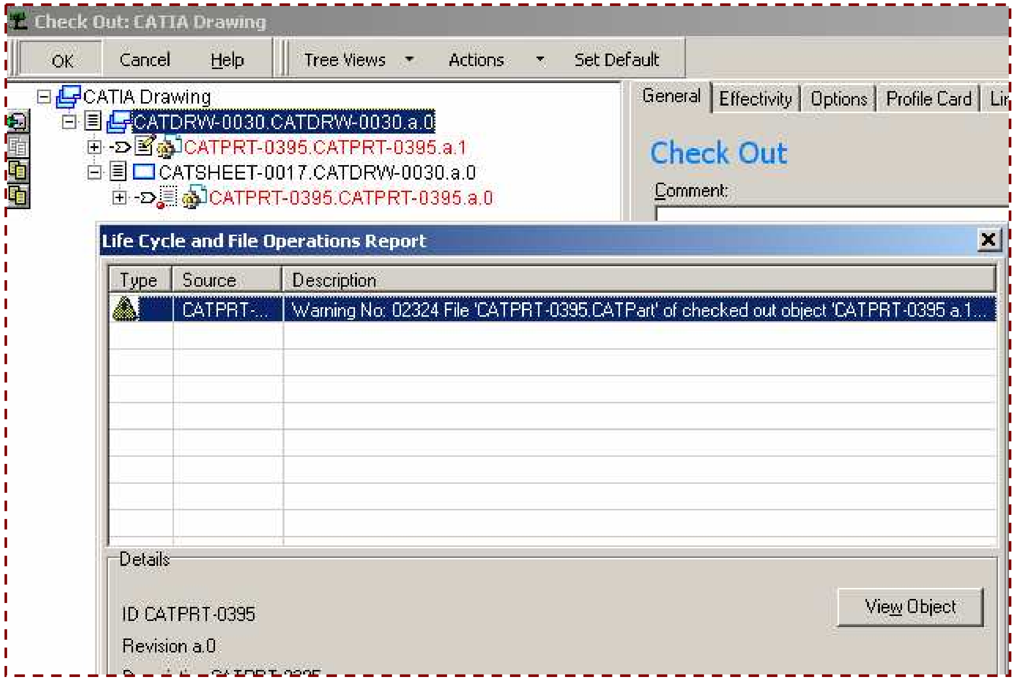
How to Manage Associated Objects During Lifecycle (2/2)

2 Modify the part, and synchronize the sheet after:

- New Release of the Part, and make Modifications.
- New Release of the Sheet. During New Release Operation, switch to the new release of the Part as Parent.







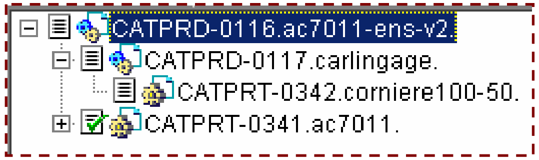
The report menu appears to show the reroute.

Student Notes:

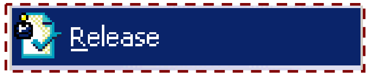
Examples (1/2)

Here you will see some examples which illustrate that the integrity of a product is protected during all lifecycle operations.

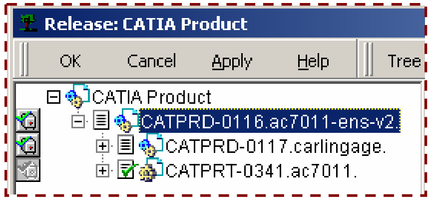
- **When releasing a Product all its children will also be released.**



Documents Tree window

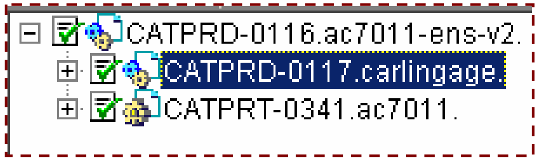


'Life Cycle' command in SMARTEAM menu



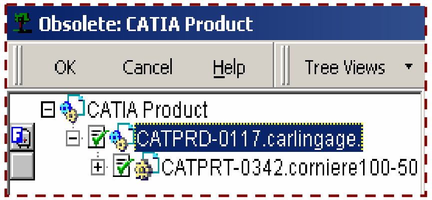
Release window

- **You can move a sub-product to the Obsolete Vault only if its parent is already obsolete.**



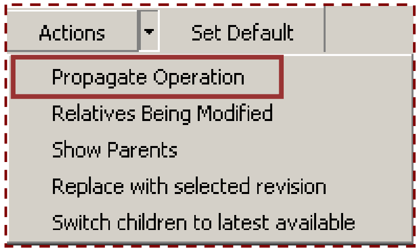
Documents Tree window

'Life Cycle' command in SMARTEAM menu



Obsolete window

- **You can make a revision operation on a Product and all its children simultaneously, using the Propagate Operation option in the intermediate panel by using 'Life Cycle' command.**

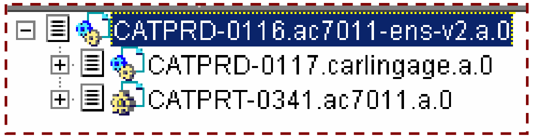


Student Notes:

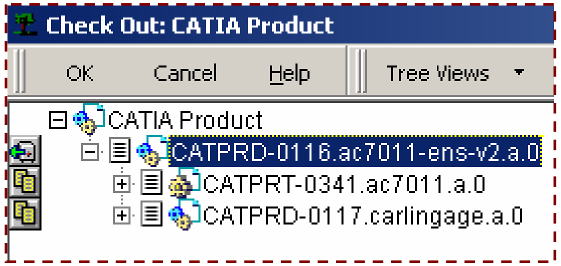
Examples (2/2)

Here you will see some examples which illustrate that the integrity of a product is protected during all life cycle operations.

- **You can perform a revision operation on a product and just copy its children to the desktop**
  - **On a Product, select Check Out**
  - **In the Check Out panel, display a Top Down Tree of the Assembly**
  - **The icons on left side indicate which objects will be checked out or copied**



Documents Tree window



Check Out window

- **You can check out a child independently, and leave the parent product in the vault**
  - **On a part, select Check Out.**