



## V4 Integration

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# Preface

The Extended Enterprise can at last become reality! The **V4 Integration** product includes unique two-way interoperability between CATIA Version 5 and CATIA Version 4 data. This means that:

- V4 designs can be accessed, exchanged and processed thus ensuring continuity and protecting V4 customer investment while taking advantage of Version 5's full functionality.  
Using an integration tool called the **CATIA Site Navigator** in CATIA Version 5, CATIA Version 4 data can be read and, if actual editing is required inside CATIA Version 5 application workbenches, this data can also be converted into V5 format. This can be done either in the Windows or UNIX environment.
- CATIA Version 5 data can be loaded and processed in a CATIA Version 4 session enabling V5 data to benefit from the breadth of the CATIA Solutions Version 4 portfolio.

As CATIA Version 5 comprises an integrated package including interoperability between CATIA Version 4 data and the application portfolio V4/V5 interaction can take many forms:

- a Version 5 design using Version 4 data (see [Copying CATIA Version 4 Model Data to CATIA Version 5](#))
- a V5 drawing using V4 data (see [Converting CATIA Version 4 Drawing Data into CATIA Version 5 Data](#))
- a V5 assembly combining both V4 and V5 data (see [Inserting Existing Components](#) in the *CATIA - Assembly User's Guide*).

As an open solution, it includes interfaces with the most commonly used data exchange industry standards.

Managing V4 Models, PRJ Fi Where to Find More Informat



# Managing CATIA Version 4 Models, PROJECT Files and Library Objects in CATIA Version 5

To ensure CATIA Version 4->CATIA Version 5 interoperability, CATIA Version 5 provides the **CATIA Site Navigator** allowing you to read, in a CATIA Version 5 session, CATIA Version 4 models on both Windows NT and UNIX and both PROJECT files and library objects on UNIX. Any geometric element (whether the model involved has an internalized PROJECT file or is linked to an external PROJECT file) can be read and copied to a CATIA Version 5 document.

In a CATIA Version 5 session, CATIA Version 4 models can be displayed on both NT and UNIX and library objects can be displayed on UNIX. Basic **read-only** operations can then be performed, such as:

- displaying and selecting geometric elements and workspaces, in the geometry area and/or the specification tree (except for the HLR mode)
- displaying graphic properties (color, show/hide, layers, filters, pick/nopick)
- zooming, rotating and panning
- printing (except for the HLR mode)
- applying, creating, deleting, and modifying layer filters
- verifying the geometry (and, in the case of exact solids, the specifications) of one or more Version 4 elements prior to copy-pasting it into a CATIA Version 5 workbench.

In CATIA Version 5, you **cannot edit** CATIA Version 4 models or library objects but you can paste all or part of a model or library object into a CATIA Version 5 document and then edit it. Once this has been done, it is of course no longer a Version 4 model, but a CATIA Version 5 document.



# Where to Find More Information

Prior to reading this book, we recommend that you read the [CATIA - Infrastructure User's Guide](#).



Up



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# What's New?

Enhanced: [Connecting an ORACLE or DB2 Database to Your CATIA Version 5 Environment \(on UNIX Only\)](#)

New: [Preparing the VPM and CATIA Environments for the Use of VPM Data in CATIA Version 5](#)

New: [Creating and Saving a CATIA Version 5 Product for Save or Commitment in VPM1.1](#)

Enhanced: [Selecting Geometric Elements on CATIA Version 4 Models](#)

Enhanced: [Copying CATIA Version 4 Drawing Data to CATIA Version 5](#)

New: [Saving CATIA Version 5 CATPart Documents As CATIA Version 4 Models](#)

New: [Converting CATIA Version 4 Libraries into CATIA Version 5 Catalogs](#)

# Getting Started



This task shows you how to open a CATIA Version 4 model in CATIA Version 5.




This task should take approximately 1 minute.



Open the document \Online\Samples\V4Integration\xwings.model.



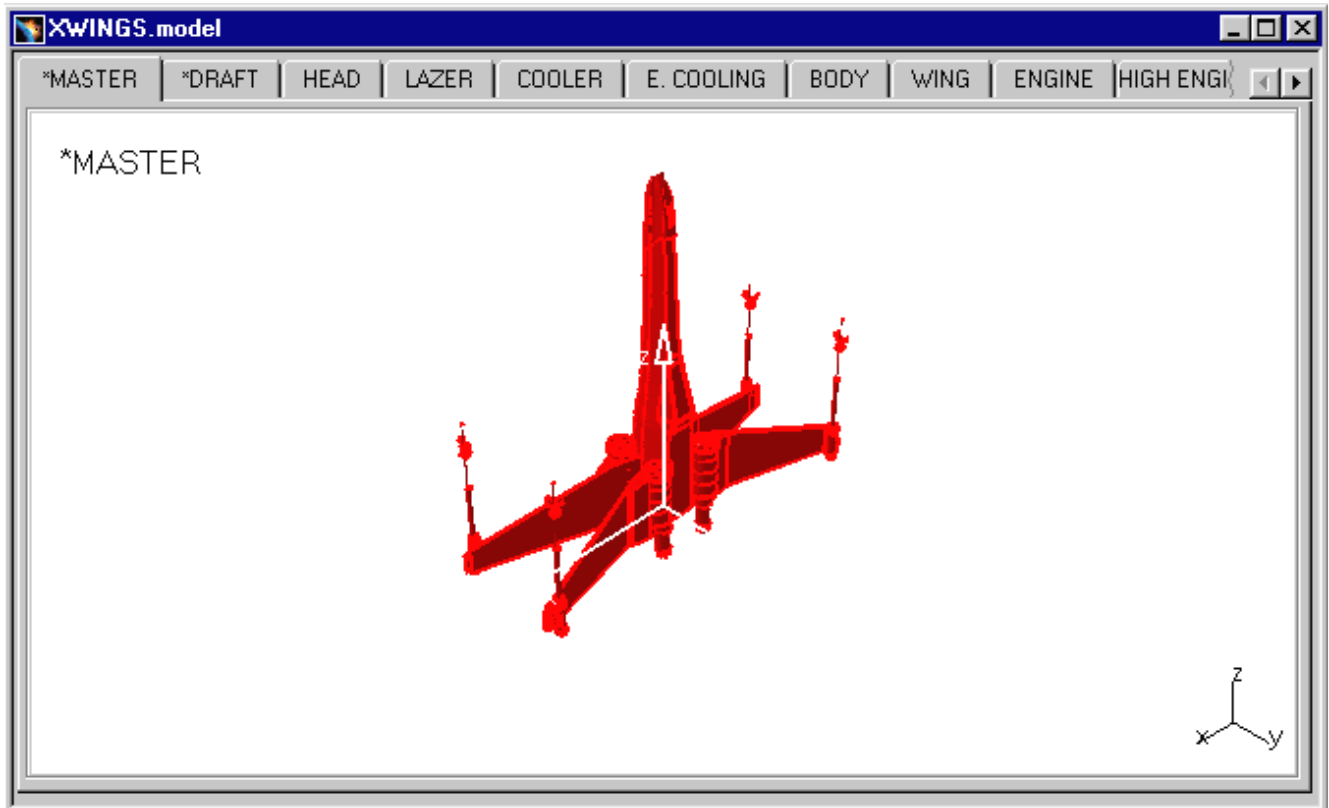
1. Click the Open icon  or select the File->Open command.

2. In the File Selection Box, select the file location.

3. Select the model.

(Where there are many different types of files, it may be useful to click the Files of type list and specify model as the document type.)

The following document will appear:



For other ways of opening existing Version 4 models, see "[Opening Existing Documents](#)" and "[Opening Most Recently Used Documents](#)" in the *CATIA - Infrastructure User's Guide*.



# Basic Tasks

<b>Theme</b>	<b>Purpose</b>
<a href="#"><u>Manipulating CATIA Version 4 Models in CATIA Version 5</u></a>	Perform basic operations on CATIA Version 4 models in CATIA Version 5
<a href="#"><u>Copying CATIA Version 4 Models to CATIA Version 5</u></a>	Copy CATIA Version 4 models to CATIA Version 5 and perform associated operations
<a href="#"><u>Manipulating CATIA Version 5 Data in CATIA Version 4</u></a>	Specify requirements for the use of CATIA Version 5 data in CATIA Version 4
<a href="#"><u>Using CATIA Version 4 Library Objects (on UNIX Only)</u></a>	Perform basic operations on CATIA Version 4 library objects

# Manipulating CATIA Version 4 Models in CATIA Version 5

	<b>Purpose</b>
<a href="#">Opening CATIA Version 4 Models in CATIA Version 5</a>	Open CATIA Version 4 models in CATIA Version 5
<a href="#">Selecting Geometric Elements on CATIA Version 4 Models</a>	Display and select Version 4 model workspaces and their geometric elements
<a href="#">Managing Layer Filters on CATIA Version 4 Models</a>	Create, apply, modify and delete layer filters on Version 4 models
<a href="#">Enabling Read Only Access to CATIA Version 4 Data in CATIA Version 5</a>	Provide CATIA Version 5 with information it requires to locate and process Version 4 data adequately
<a href="#">Opening CATIA Version 4 Models Referencing an External PROJECT File</a>	Open a CATIA Version 4 model referencing an external PROJECT file
<a href="#">Displaying Faces and Surfaces Visible in Shading Mode in CATIA Version 4</a>	Display all faces and surfaces that were visible in CATIA Version 4



# Opening CATIA Version 4 Models in CATIA Version 5




This task shows you how to open a CATIA Version 4 model in CATIA Version 5.



Open the document \Online\Samples\V4Integration\XWINGS.model.

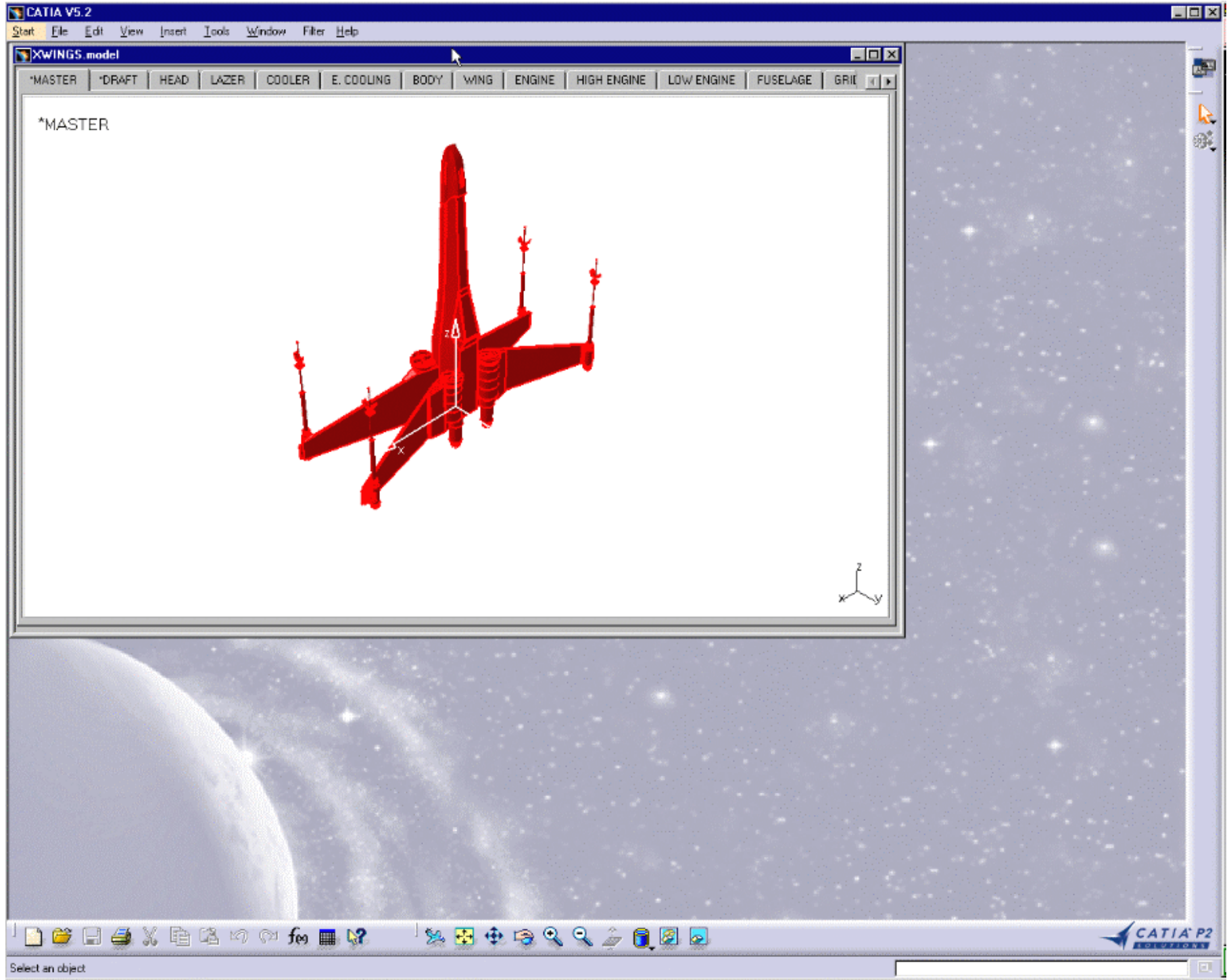
You should have already completed the task [Enabling Read Only Access to CATIA Version 4 Data in CATIA Version 5](#).



1. Click the Open icon  or select the File->Open command.
2. In the File Selection Box, select the file location.
3. Select the model.

(Where there are many different types of files, it may be useful to click the Files of type list and specify model as the document type.)

A document like this will appear:





If, in CATIA Version 4, you defined specific views for a model by means of IMAGE/WINDOW+DEFINE, you can still use these views by clicking the View->Defined Views... command.



For other ways of opening existing CATIA Version 4 models, see "[Opening Existing Documents](#)" and "[Opening Most Recently Used Documents](#)" in the *CATIA - Infrastructure User's Guide*.

V4 models residing on UNIX can be accessed from Windows using the http protocol. (Make sure beforehand that an http server has been installed on the machine where the models reside.)

The address to be specified should look something like this:

http://UNIXserver:port/V4modellocation



# Selecting Geometric Elements on CATIA Version 4 Models



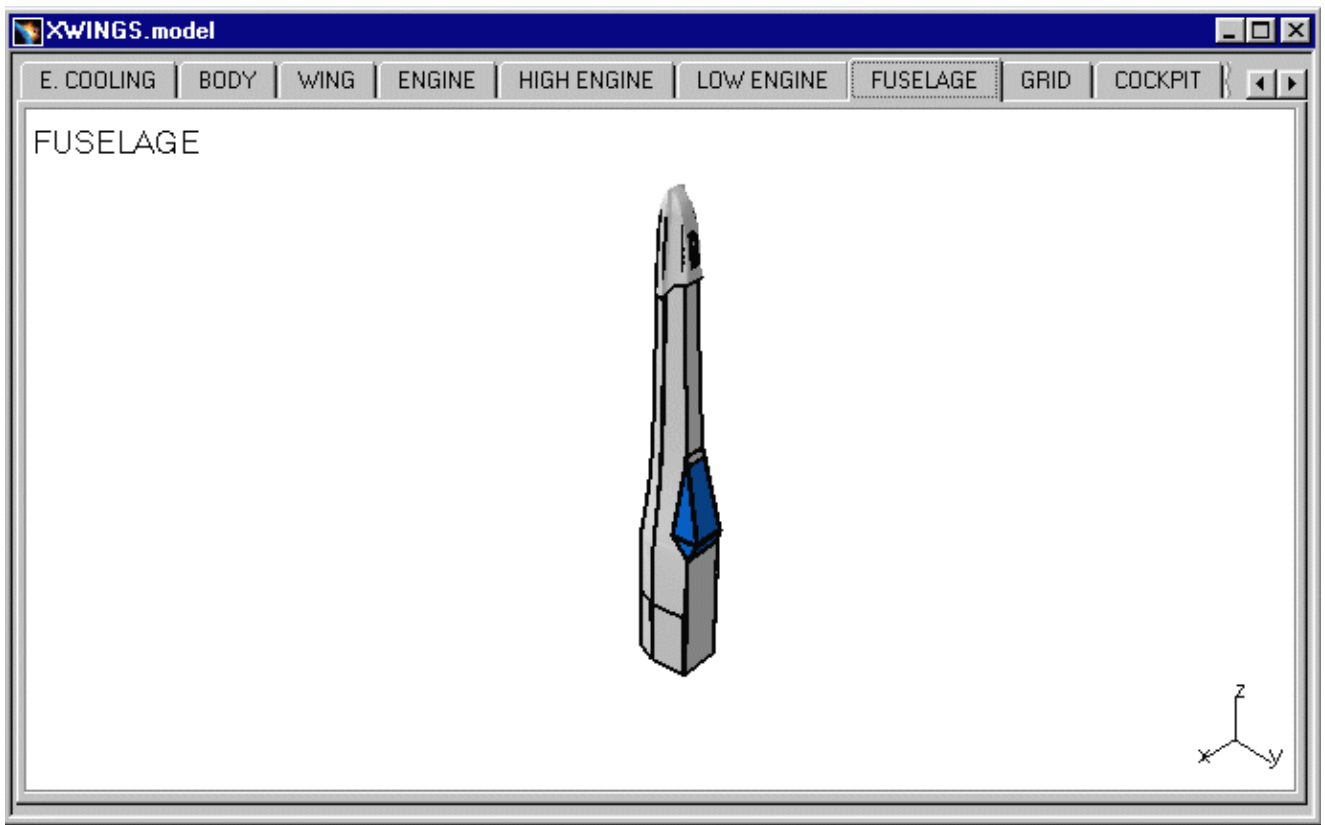
This task shows you how to display a CATIA Version 4 model workspace and select its geometric elements in CATIA Version 5.



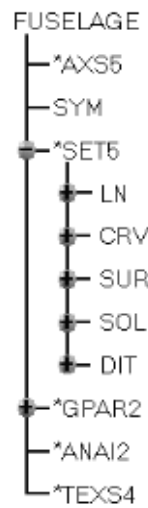
Open the document \Online\Samples\V4Integration\XWINGS.model.



1. Above the geometry area where the Version 4 model is displayed, click the workspace tab you wish to display (the FUSELAGE workspace in the model shown below, for example), The tab selected then "rises" relative to the other tabs.



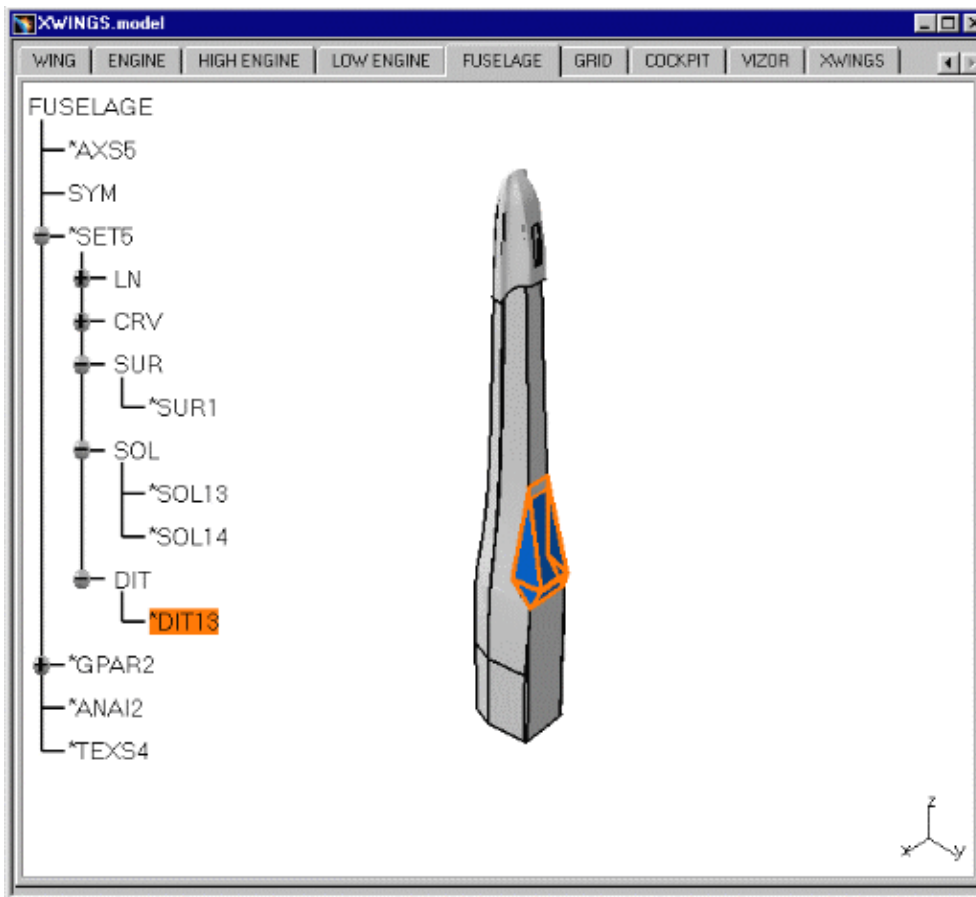
2. If you then wish to select a geometric element, expand the specification tree by clicking on the FUSELAGE item and then on one of the items with a plus sign, \*SET5, for example. The specification tree will then look like this:



A set is made up of subsets each containing lines, curves, surfaces, solids, dittos, etc. In the specification tree shown above these are easily identifiable (LN, CRV, SUR, etc.). A subset can be selected just like any other item for copy/paste, show/no-show operations, etc.

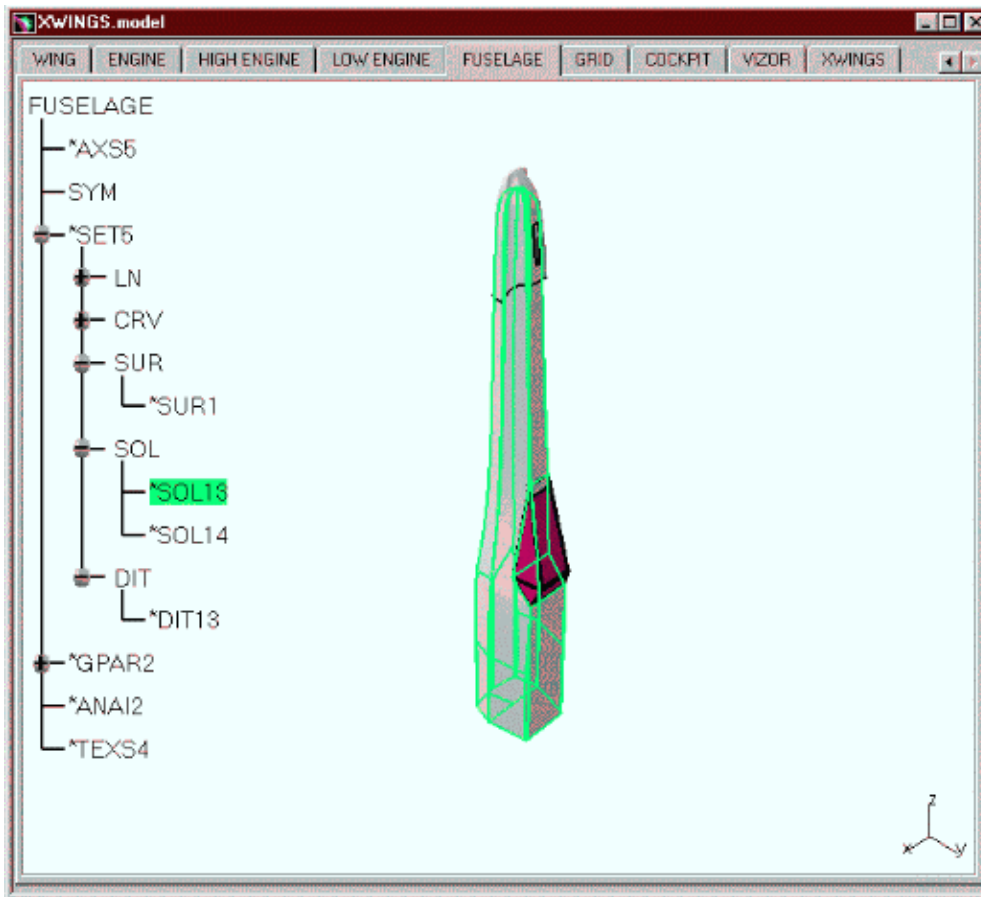
However, you can of course select one or more of the lines, curves, solids, etc. contained in the subsets.

3. Click on the plus sign of the DIT subset and select the contents, \*DIT13. Notice that the corresponding part of the model in the geometry area is highlighted as shown below.



You can of course select an element by clicking on it in the geometry area.

4. Click on the central part of the fuselage (not on the blue component shown above). The corresponding solid is highlighted in the specification tree:




As you can see above, in the specification tree, double-clicking boxes with a plus sign shows the component elements of that particular workspace or set. Clicking the minus sign hides the elements contained in the workspace or set.




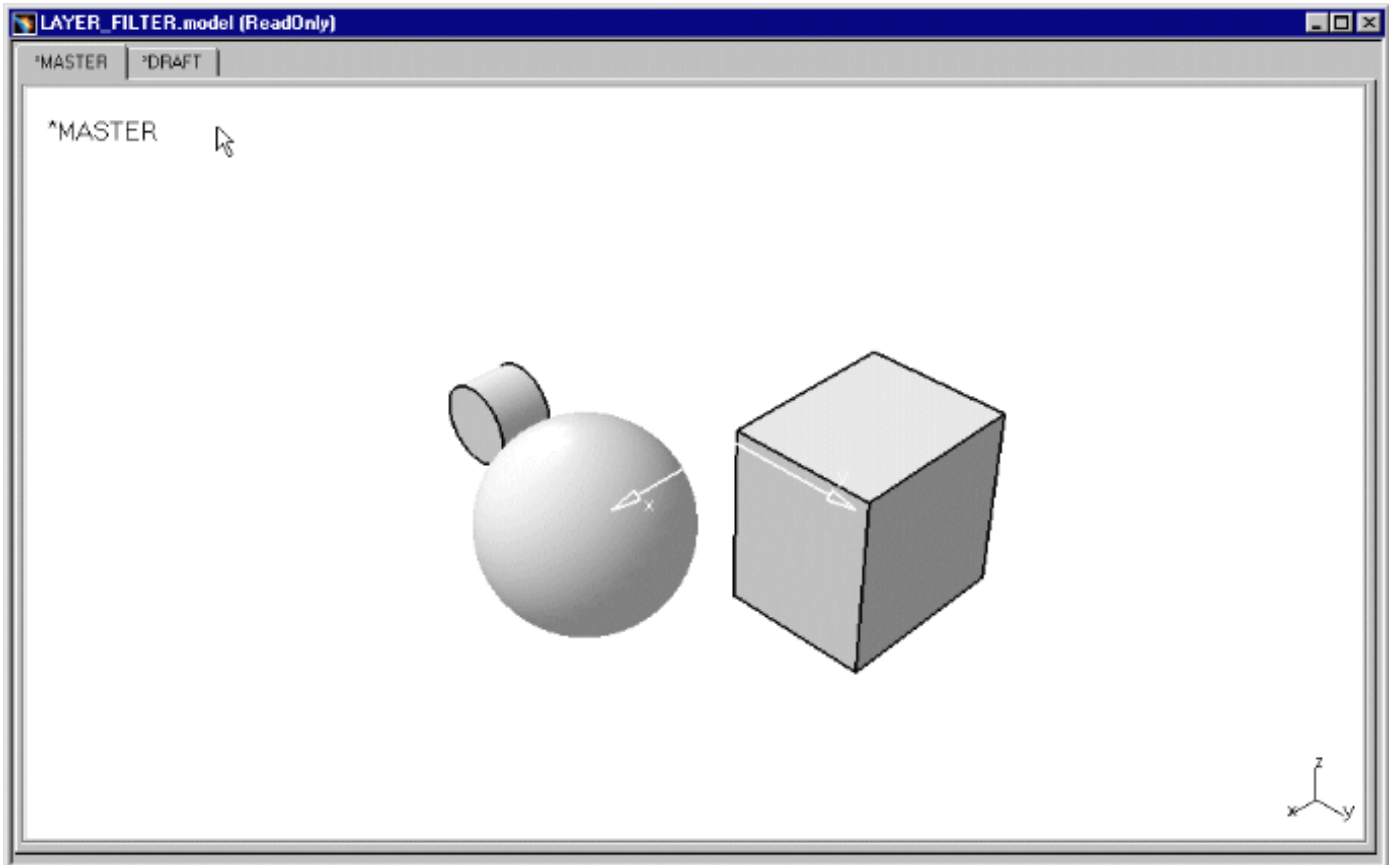
- [Up](#)
- [Managing Layer Filters on V](#)
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- [Displaying All Faces and Sur](#)

# Managing Layer Filters on CATIA Version 4 Models

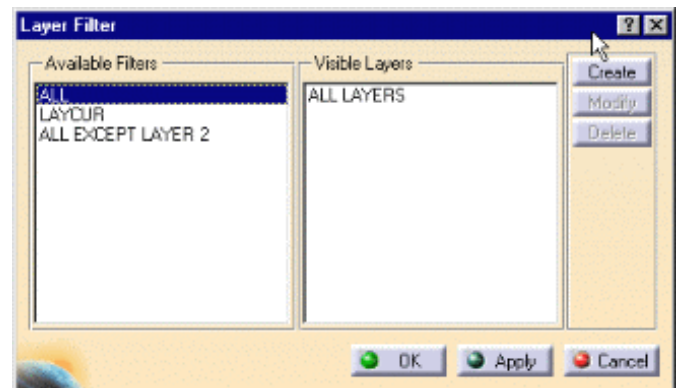
 This task shows you how to apply, create, modify and delete layer filters on a CATIA Version 4 model.

 Open the document \Online\Samples\V4Integration\LAYER\_FILTER.model.

 1. Open the model.

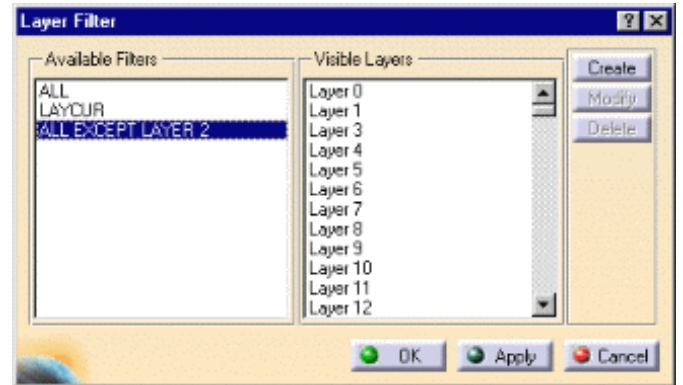


2. To **apply**, **create**, **modify** or **delete** an existing filter that was defined in Version 4, select the Filter->Layer Filter... command. The Layer Filter dialog box appears.



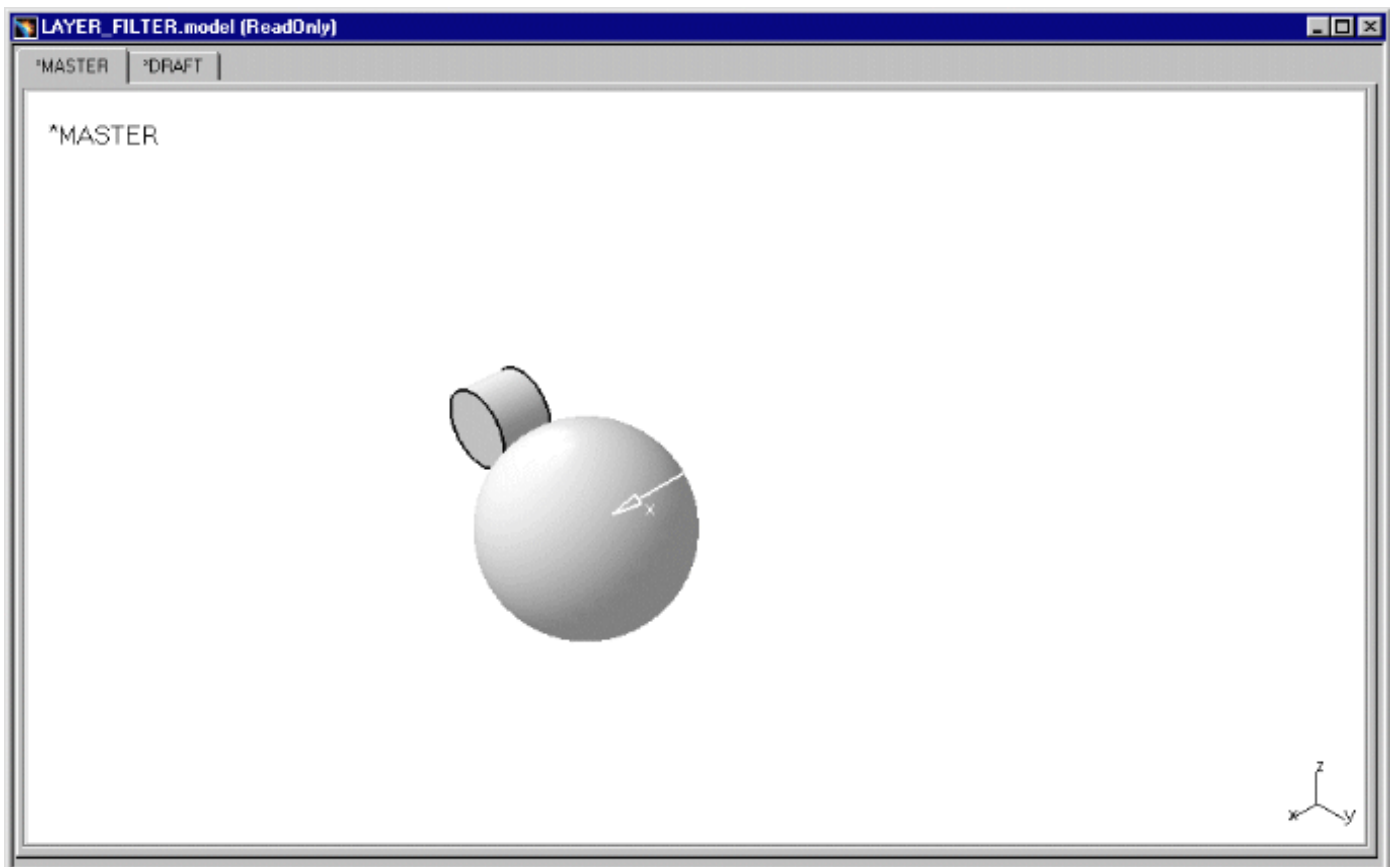
The first two filters in the Available Filters list are always ALL and LAYCUR (current layer).

3. To **apply** an existing filter, select the filter ALL EXCEPT LAYER 2 in the Available Filters list. and click the OK or Apply button.

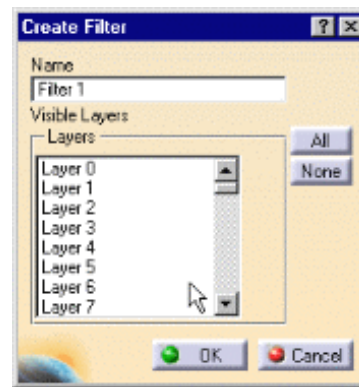


In the Layer Filter dialog box, click the Apply button if you have other operations to perform. If not, click the OK button.

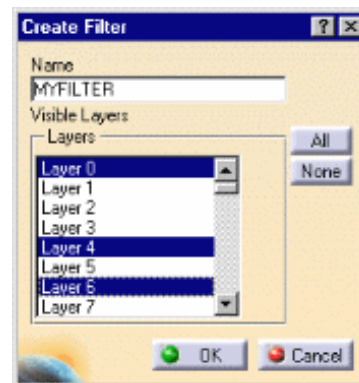
The model then looks like this:



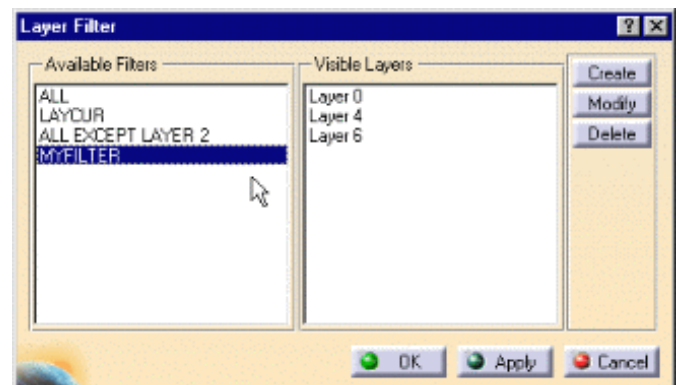
4. To **create** a new filter, select, in the Layer Filter dialog box, the Create button. The Create Filter dialog box appears.



5. In the Create Filter dialog box, select one or more of the layers in the list displayed. If you want to give the new filter a name you can enter it in the Name field. Click the OK button.

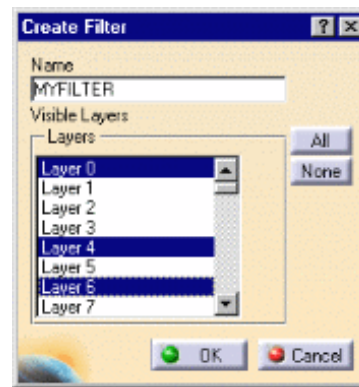


6. The filter created is added to the Available Filters list in the Layer Filter dialog box.

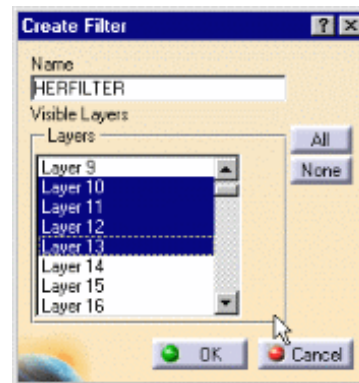


7. In the Layer Filter dialog box, click the Apply button if you have other operations to perform. If not, click the OK button.

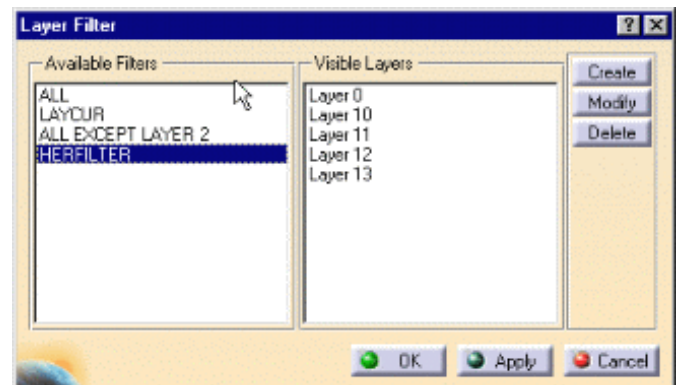
8. To **modify** a filter, select, in the Layer Filter dialog box, a filter that you have created. Click the Modify button. The Modify Filter dialog box appears.



9. Make the changes. You may wish to change the name of the filter(s) and/or modify filter selection. Click the OK button.



10. The filter modified is added to the Available Filters list in the Layer Filter dialog box.



11. In the Layer Filter dialog box, click the Apply button if you have other operations to perform. If not, click the OK button.



Bear in mind that you cannot modify filters defined in CATIA Version 4 (the Modify button is grayed out). Only those filters defined in CATIA Version 5 using the Create button can be modified.

12. To **delete** a filter, select one or more filters in the Layer Filter dialog box.

13. Click the Delete button.

14. In the Layer Filter dialog box, click the Apply button if you have other operations to perform. If not, click the OK button.





Bear in mind that you cannot delete filters defined in CATIA Version 4 (the Delete button is grayed out). Only those filters defined in CATIA Version 5 using the Create button can be deleted.



 Up	 Opening V4 Models in V5	 Selecting Geometric Element
 Managing Layer Filters on V	 Enabling Read Only Access	 Displaying All Faces and Sur
	 Opening V4 Models with an	

# Enabling Read Only Access to CATIA Version 4 Data in CATIA Version 5

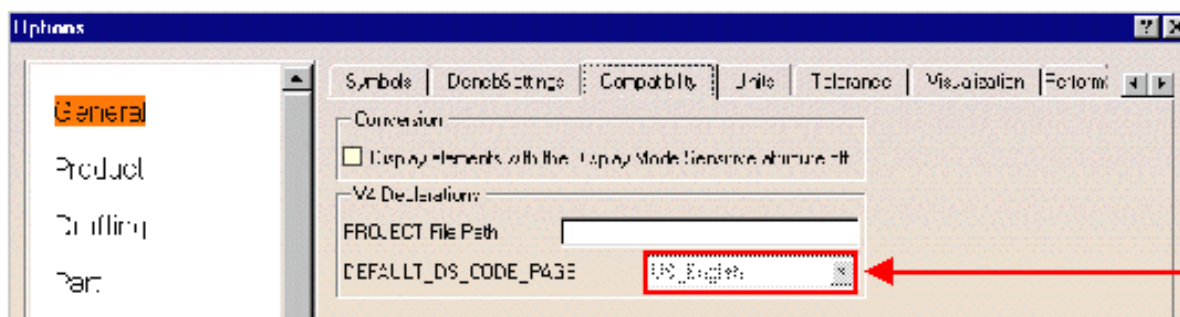
 This task shows you what preparatory steps to perform to ensure access to CATIA Version 4 data i.e. a model or library object in a CATIA Version 5 session.

 If a Version 4 user, you will remember that the **declaration parameter** catsite.DEFAULT\_DS\_CODE\_PAGE declares the language to identify the data read if this data is not labeled (i.e. if it is just labeled EBCDIC or ASCII, and not labeled with a standard code page such as ISO8859-x, IBM-392, EUC-KR, EUC-CN, and so forth).

In Version 4, information such as the language used to identify unlabeled readable data was specified by means of the parameter settings in the declaration files. These declaration parameters are no longer supported in CATIA Version 5 and there is no way to transfer them automatically to CATIA Version 5. Such information must be provided by means of the dialog boxes described below, **before** attempting to read Version 4 data.



1. Select the Tools->Options... command. The Options dialog box appears with the General category selected in the left-hand column.
2. Click the Compatibility tab.



Click here for  
list of languages  
identifying  
unlabeled data

For Version 4 data that is not labeled with a standard code page (for example, ASCII-DS-xxx or EBCDIC-DS-xxx) other than US English, you must specify the appropriate language from the list provided.


3. To do this, click on the DS\_DEFAULT\_CODE\_PAGE list in the V4 Declarations part of the dialog box and select the appropriate language.





4. Click OK to confirm.



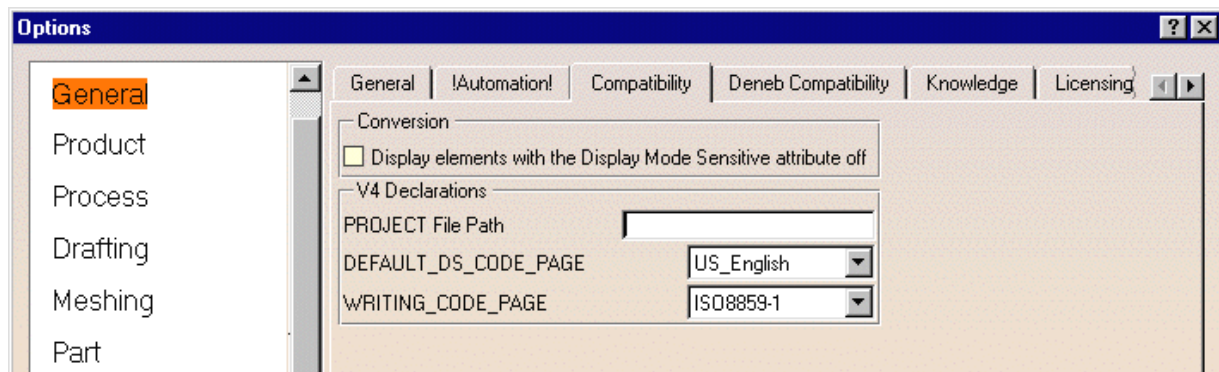
# Opening CATIA Version 4 Models Referencing an External PROJECT File

 This task shows you how to display a Version 4 model referencing an external PROJECT file.

 If a Version 4 user, you will remember that certain model data must be contained in a PROJECT file which can either be internal to the model or external. If it is external, the only way to access such data is to provide Version 5 with precise information about the PROJECT file's whereabouts.


 1. Select the Tools->Options... command. The Options dialog box appears with the General category selected in the left-hand column.


2. Click the Compatibility tab.



3. Enter, in the field indicated above, the location and name of the PROJECT file referenced by the V4 model you wish to display.

Make sure you complete this field **before** displaying a Version 4 model.

 If you do not specify the PROJECT file path before opening the model, a warning message will appear.

 You can display such models on Windows or on UNIX. However, the external PROJECT file referenced by the model can **only** be on UNIX. Also, you must first make sure an http server has been installed on the machine where the PROJECT file resides.

- On Windows, the address to be specified should look something like this:


http://UNIXserver: port/PRJlocation

- On UNIX, just specify the path, for example:

/u/users/username/PRJname

The following reference tables in the PROJECT files can now be accessed in CATIA Version 5:


- attribute and class tables
- annotations/dimensions.

 There are no restrictions as regards the PROJECT file's code page. However, you must make sure that the code pages of the model and the PROJECT file are compatible.



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-  [Opening V4 Models in V5](#)
-  [Enabling Read Only Access](#)
-  [Selecting Geometric Element](#)
-  [Displaying All Faces and Sur](#)
-  [Opening V4 Models with an](#)

# Displaying Faces and Surfaces Visible in Shading Mode in CATIA Version 4

 This task shows you how to display all faces and surfaces that were visible in CATIA Version 4.

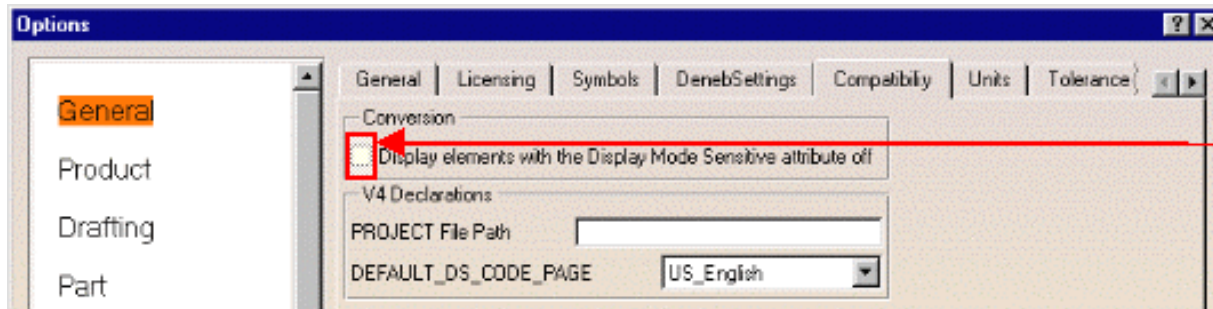
 If a Version 4 user, you will remember that the **display attribute** CURRENT DISPLAY MODE SENSITIVE enables you to decide whether to display the hidden parts or not.

In Version 5, by default, only faces and surfaces that were in Shading mode in Version 4 are shown in models displayed by means of the CATIA Site Navigator. To display all faces and surfaces, you must therefore disable the CURRENT DISPLAY MODE SENSITIVE attribute.



1. Select the Tools->Options... command. The Options dialog box appears with the General category selected in the left-hand column.

2. Click the Compatibility tab.



Check this box

3. Check the box indicated by the arrow above.

You can now visualize in Version 5 all Version 4 faces and surfaces, even those that were not in Shading mode.





Up



Opening V4 Models in V5



Selecting Geometric Element Managing Layer Filters on V



Enabling Read Only Access



Displaying All Faces and Surfaces Opening V4 Models with an




# Copying CATIA Version 4 Models to CATIA Version 5

Task	Purpose
<a href="#">About Copying CATIA Version 4 Models to CATIA Version 5</a>	Provide hints and tips in the event of difficulty copying CATIA V4 models into CATIA Version 5
<a href="#">Checking CATIA Version 4 Model Data Before Copying It to CATIA Version 5</a>	Check CATIA Version 4 model data before copying it to CATIA Version 5
<a href="#">Copying CATIA Version 4 Model Data to CATIA Version 5</a>	Copy CATIA Version 4 model specifications and geometry to CATIA Version 5
<a href="#">Copying CATIA Version 4 Drawing Data to CATIA Version 5</a>	Copy CATIA Version 4 drawing data to CATIA Version 5
<a href="#">Interoperability with V4 Textures</a>	Apply a CATIA Version 4 2D texture to a CATIA Version 5 material
<a href="#">Converting CATIA Version 4 Kinematic Data into DMU Kinematic Version 5 Data</a>	Apply a CATIA Version 4 2D texture to a CATIA Version 5 material



# About Copying CATIA Version 4 Models to CATIA Version 5

 If you encounter any difficulty copying CATIA V4 models to CATIA Version 5, there are a certain number of precautions you should take prior to the copy operation.

## Precautions to be taken in CATIA Version 4 prior to the copy operation

- If the model to be copied is an exact solid it must be processed using the Force Update option in the Smart Solid mode. This operation must be done on CATIA 418 or higher.
- It is also advisable to run the CATCLN utility on the model to be copied. Again, this operation must be done on CATIA 418 or higher.
- Remember to run the specification check on all the V4 models you wish to copy to CATIA Version 5 as it provides you with the following invaluable information:
  - identification of the primitives which are converted to datums, indicating loss of canonical information (see the Restrictions section below for the complete list).  
This is shown in the Message column of the Check Validity dialog box, for example:  
2 primitives checked as geometry only
  - identification of the operations which are not supported in V5R2 and MUST BE DELETED before proceeding with the copy operation (see the Restrictions section below for the complete list).  
This is shown in the Message column of the Check Validity dialog box, for example:  
2 operations not implemented
  - identification of the non-supported primitives (see the Restrictions section below for the complete list).

For more detailed information, you only have to double-click on the appropriate message in the rightmost column of the Check Validity dialog box. This displays the contents of the model in the form of a specification tree (see [Checking CATIA Version 4 Model Data Before Copying It to CATIA Version 5](#)).

- If, in CATIA Version 5, you intend to make only cosmetic changes to a Version 4 solid with no impact on the specification tree you can simply copy the solid's geometry. However, if your modifications are major and will considerably alter the specification tree you would be better advised to copy the solid's specifications (see [Copying CATIA Version 4 Model Data to CATIA Version 5](#)).
- Make sure that the update option selected in the General tab of the Options dialog box (displayed using the Tools->Options command) is NOT set to Automatic.
- When updating, make sure that you select the Inactivate option in the Update Diagnosis dialog box to deactivate any troublesome operations or primitives in CATIA Version 4. Do NOT delete them as such data, although not copied, can, once the rest of the model has been successfully copied, be recreated in CATIA Version 5 in a more satisfactory way. In a future version of CATIA Version 5, this requirement will no longer apply.

## Restrictions

As mentioned above, certain restrictions apply when copying models from CATIA Version 4 to CATIA Version 5:

The primitives converted to datums are:

- pyramids
- sweep spines, sweeps with a non-close profiles, until a sweep
- import primitives (linked to a solid in another model).

The non-supported operations are:

- certain draft types, for example, keep edges with more than two neutrals
- certain fillet types, for example, rolling edges.

The non-supported primitives are:

- macroprimitive multibodies
- non-isometric transformations.

The elements copied as geometry only i.e. not as history are:


- all elements (including SKD) with the exception of exact and mockup solids.



# Checking CATIA Version 4 Model Data Before Copying It to CATIA Version 5

 This task shows you how to check CATIA Version 4 model data before copying it to CATIA Version 5.

 Open the document \Online\Samples\V4Integration\CheckGeometry Specifications.model.

 Before checking either the geometry or specifications of Version 4 data, you should understand the distinction between the two:

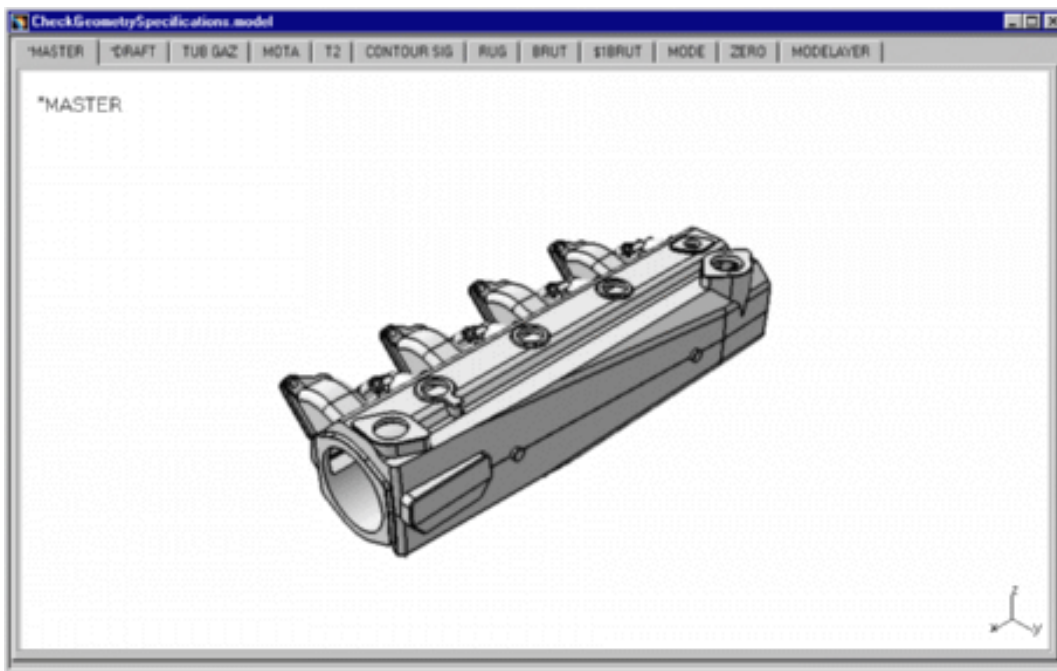
The **geometry** is just the three-dimensional representation of the elements contained in the data. An example would be the set of faces that make up a solid. It is roughly equivalent to the Brep representation used in CATIA Version 4.



The **specifications** are made up of the entire history of the actions performed to obtain the data. They are shown in the form of a tree and are roughly equivalent to the CSG tree used in CATIA Version 4.

Note that a specification check is only meaningful when applied to **exact and mockup solids**.

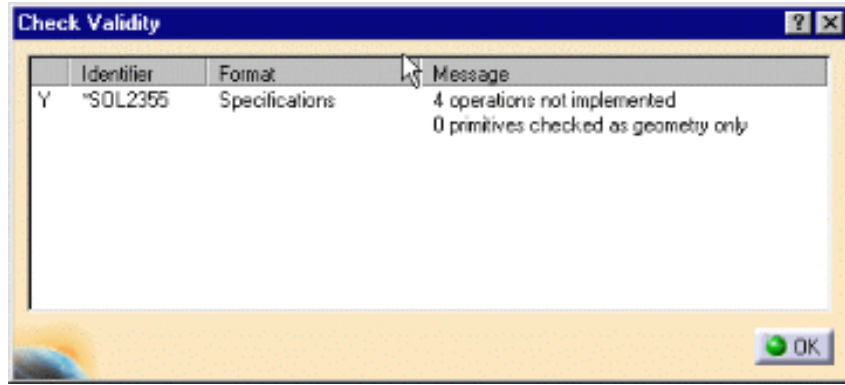


1. Select one or more geometrical elements in the geometry area.



This activates the Check Geometry  and Check Specifications  icons in the Version 4 model toolbar.

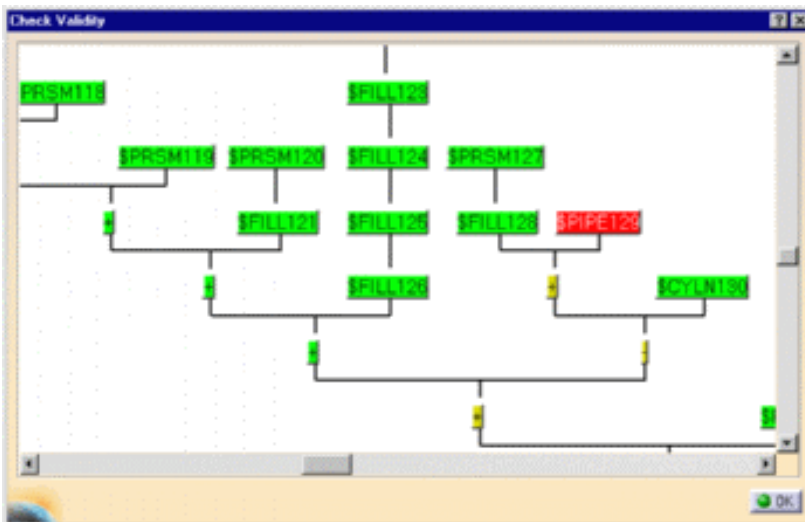
2. In the Version 4 model toolbar, click the Check Specifications or Check Geometry icon. The Check Validity dialog box opposite appears:



In the Check Validity dialog box there are four columns:

- the first column showing the overall result, color-coded as follows:
  - G (green): will be copied as is to CATIA Version 5
  - R (red): will NOT be copied to CATIA Version 5 as is
  - Y (yellow): will NOT be copied to CATIA Version 5 as is because of a link to a red-coded item.
- the Version 4 identifier
- the format i.e. Specifications
- the Message column providing the number and type of problems.

3. To view the results of a specification check, double-click on the contents of any of the columns in the Check Validity dialog box. The tree structures of some models are very large. Use the scrollbars to locate any red elements. There are several in the model shown below. Here is just one example



4. Right-click the red element to find out exactly what the problem is.





Clicking on the background and not on an element displays a menu containing two options:

- Reframe graph, which centers the tree structure
- Print whole, which prints it (for details, see "[Printing Documents](#)" in the *CATIA - Infrastructure User's Guide*).



# Copying CATIA Version 4 Model Data to CATIA Version 5



This task shows you how to copy the specifications or geometry of a CATIA Version 4 model to CATIA Version 5.



The following data can be copied from CATIA Version 4 to CATIA Version 5:

- surfaces (both polynomial and BSpline)
- faces
- volumes
- skins and exact solids
- mockup solids (see remarks regarding [copy/pasting mockup solids](#) below)
- polyhedral surfaces and solids
- circles
- ellipses
- points
- lines
- planes
- clouds of points
- edges
- parabolas
- hyperbolas
- curves (both polynomial and BSpline)
- CCVs
- NURBs (curves and surfaces).

The following task describes how an entire model is pasted from Version 4 to Version 5. You can also select the geometric elements listed above and insert them into an already existing Version 5 document.



Open the document \Online\Samples\V4Integration\LAMP.model.

You should have already completed the tasks [Enabling Read Only Access to CATIA Version 4 Data in CATIA Version 5](#) and [Checking CATIA Version 4 Model Data Before Copying It to CATIA Version 5](#).





1. Open a new CATIA Version 5 CATPart document. To do this, refer if necessary to "[Creating New Documents](#)" in the *CATIA - Infrastructure User's Guide*.


2. In the specification tree or geometry area where the Version 4 model is displayed, select the geometrical element or elements you wish to convert.

3. If you intend to copy the **geometry** you can either:


- drag and drop the element(s) onto the appropriate location in the CATIA Version 5 document.

The cursor changes slightly i.e. the symbol  appears indicating where a drop is allowed. If the cursor changes to the symbol , the drop is not allowed in that location.

- or:


a. Put the element(s) you have selected in the clipboard by clicking the Copy icon , select the Edit->Copy command or select the Copy command in the contextual menu.

b. In the specification tree of the CATIA Version 5 document, select the appropriate item (for example, PartBody or Body.1, Body.2, etc. in the PartDesign workbench).

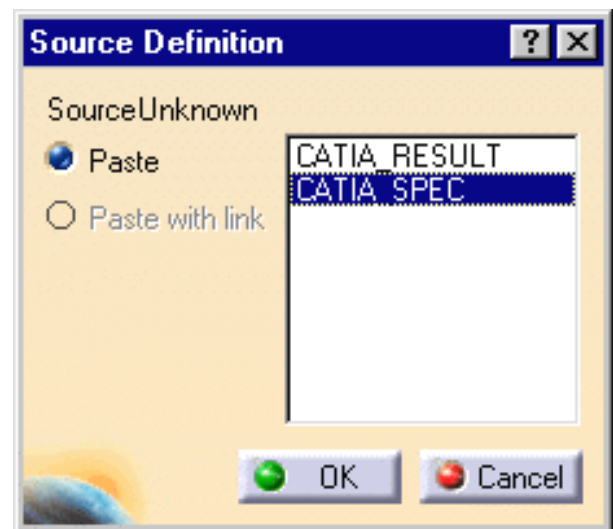
c. Click the Paste icon  or select the Edit->Paste command or select the Paste command in the contextual menu.


This operation recovers the specifications previously put in the clipboard.

4. If you intend to copy the **specifications**:


- a. Put the element(s) you have selected in the clipboard by clicking the Copy icon  , selecting the Edit->Copy command or selecting the Copy command in the contextual menu.
- b. In the specification tree of the CATIA Version 5 document, select the appropriate item (for example, PartBody or Body.1, Body.2, etc. in the PartDesign workbench).
- c. Select the Edit->Paste Special... command or select the Paste Special... command in the contextual menu.

The dialog box opposite appears. Select CATIA\_SPEC and click OK. This operation recovers the specifications previously put in the clipboard.

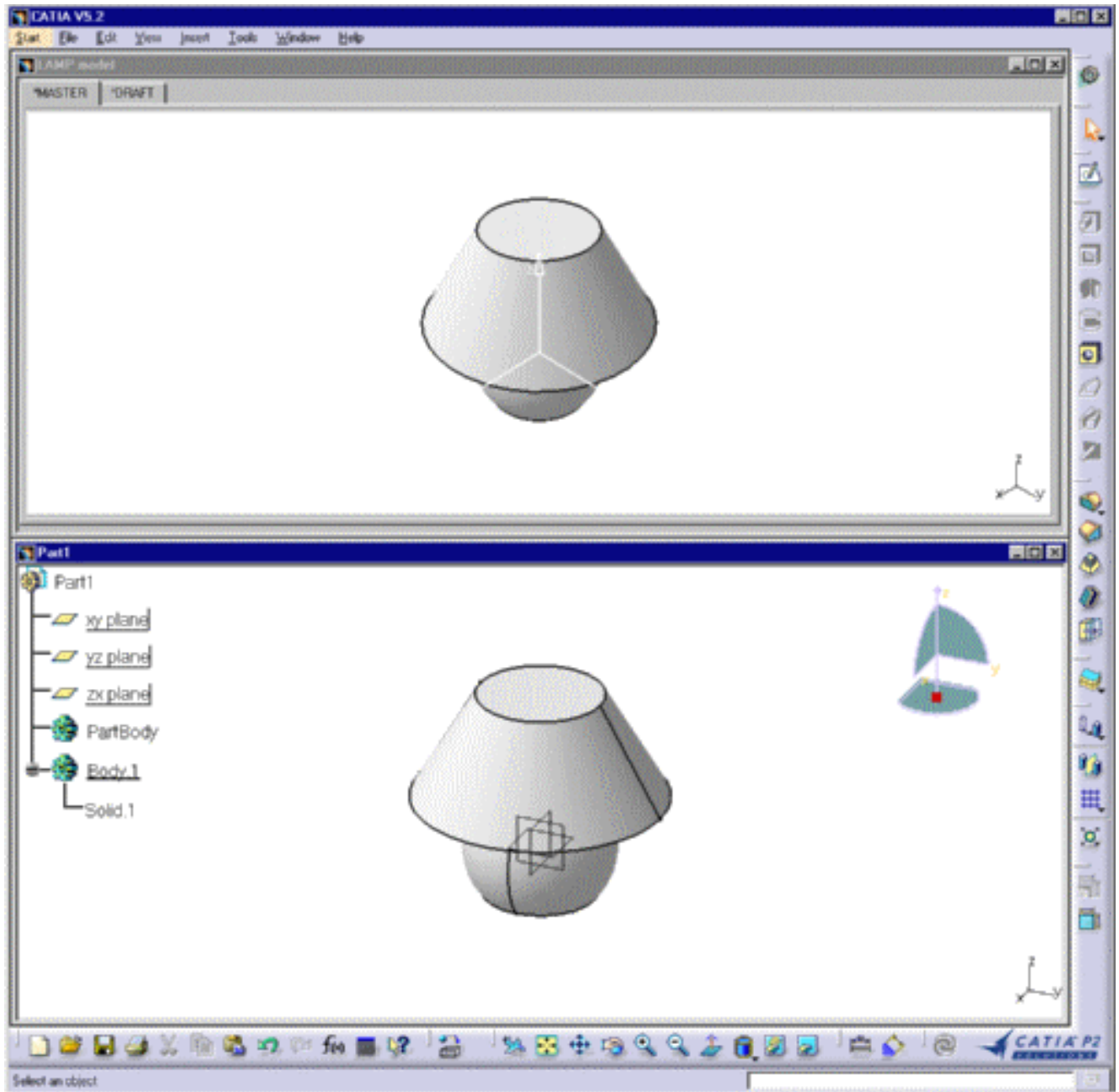


5. Click the Update icon  to view the copied data.

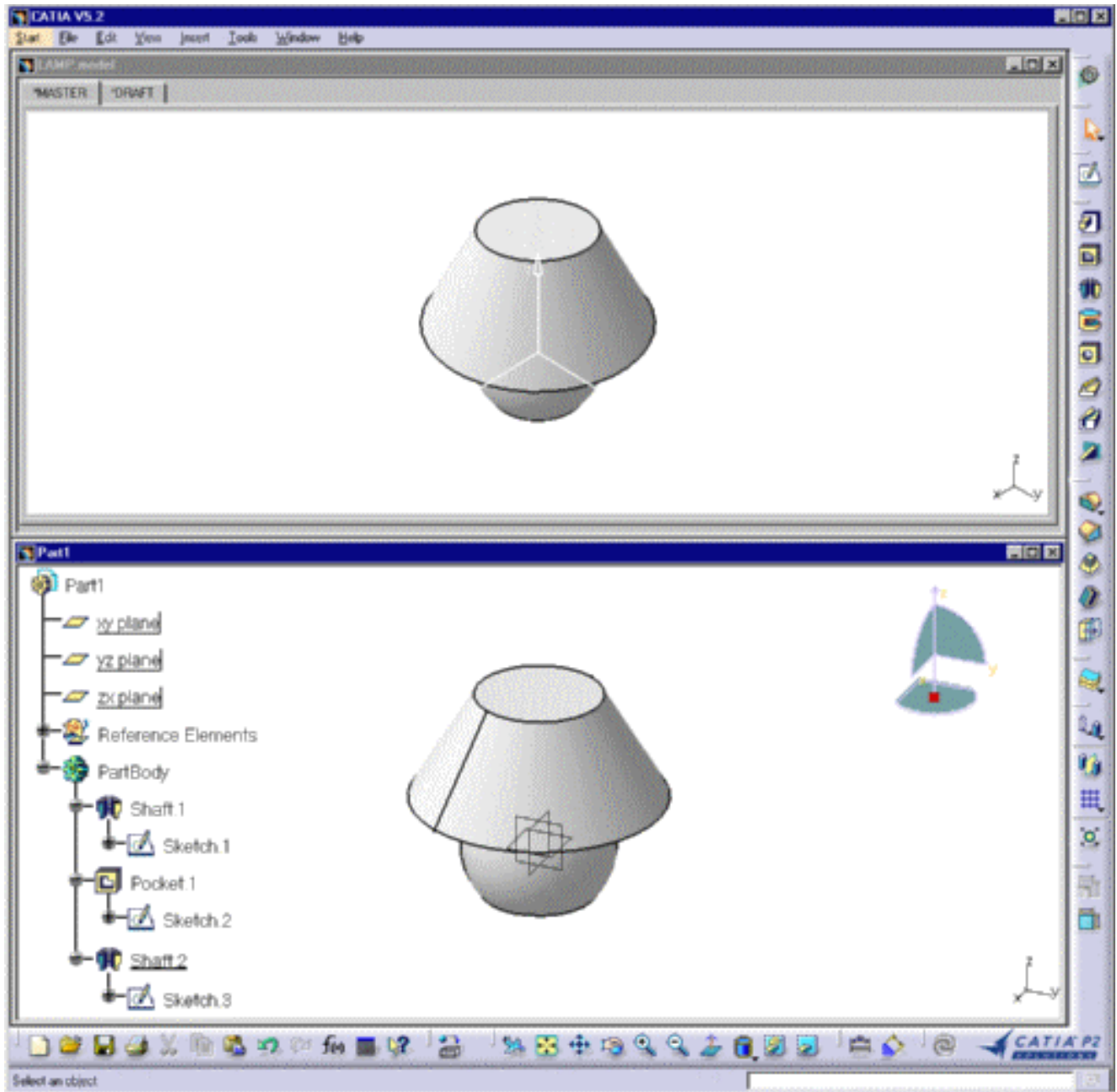
6. You may want to click the Fit All In icon  to fit all data in the window.

 Notice that the toolbars change depending on whether a CATIA Version 4 model or a CATIA Version 5 document is selected.

If you copied the **geometry** the result should look something like this:



If you copied the **specifications** the result should look something like this:



Bear in mind the following when copy/pasting **mockup solids**:

- If the solid has a history then the V5 specifications are created.
- However, if the solid has no history or if the CATIA\_RESULT option is selected(using the Paste Special... command) then a cgr file is generated containing the visualization information of the solid. The name of this file is "mymodel\_SOLMxxx" and is located in the same directory as the V4 CATIA model. This file can be visualized separately or inserted into the Product Structure application.





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Copying V4 Model Data to V Copying V4 Drawing Data to



# Copying CATIA Version 4 Drawing Data to CATIA Version 5



This task shows you how to convert the data of a CATIA Version 4 drawing into CATIA Version 5.



Open the document \Online\Samples\V4Integration\DraftingInterop.model.



The following data can be copied from CATIA Version 4 to CATIA Version 5:

- circles
- dittos, symbols (exploded in V5 geometry)
- ellipses
- points
- lines
- parabolas
- hyperbolas
- curves



- AUXVIEW2 views as result i.e. as geometry only with no associativity with 3D
- texts and dimensions.



The text and dimension types supported by V5R3 are converted into V5 geometry. Those types not supported by V5R3 appear as broken i.e. non-associative dimensions.






You cannot directly migrate V3 texts and dimensions into V5. They must first be migrated to V4.

The following task shows how drafting data is pasted from an existing Version 4 model to a new Version 5 document. You can of course also insert the V4 data into an existing Version 5 document alongside V5 drafting data.



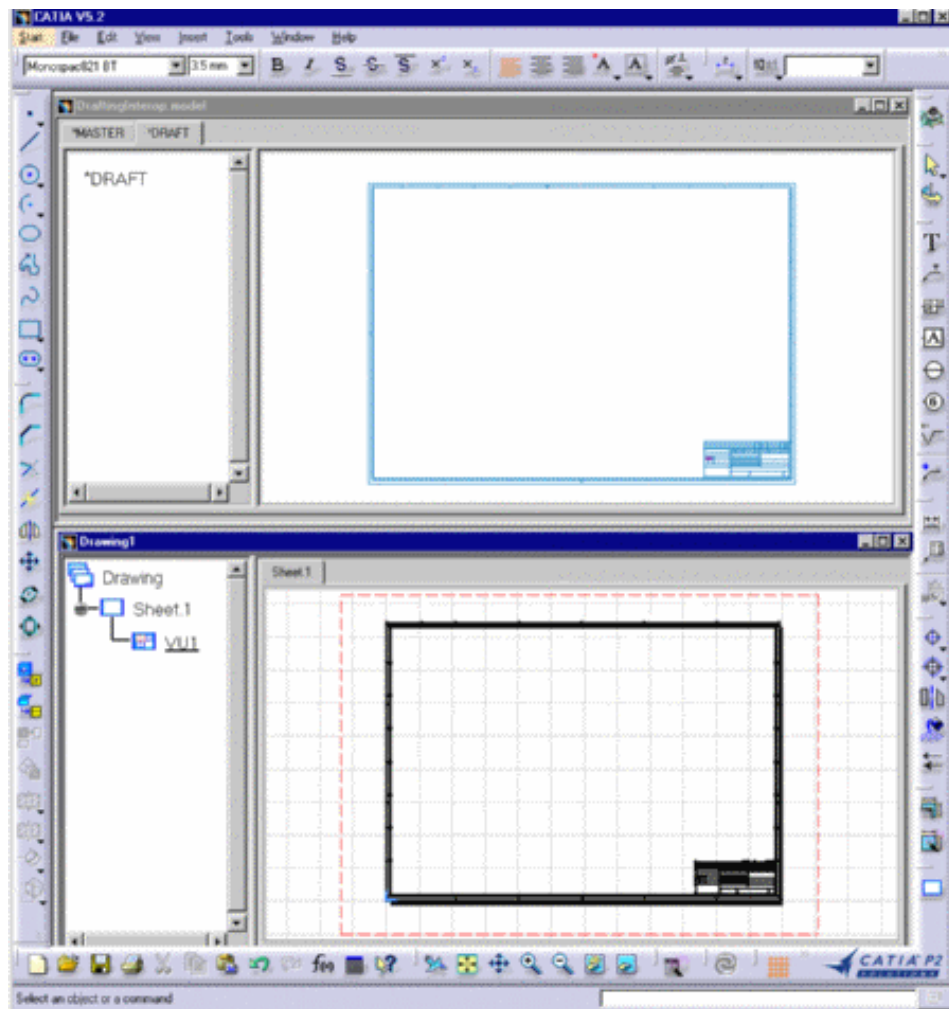
1. Open the model.
2. Open a CATDrawing document.
3. In the specification tree or geometry area where the Version 4 drawing is displayed, select the view you wish to copy into CATIA Version 5. You can select several views if you prefer.

4. Put the view(s) you have selected in the clipboard. To do this, either click the Copy icon , select the Edit->Copy command or select the Copy command in the contextual menu.
5. In the specification tree of the CATIA Version 5 CATDrawing document, select the appropriate sheet.
6. Now either click the Paste icon , select the Edit->Paste command or select the Paste command in the contextual menu. This operation recovers the view previously put in the clipboard.
7. You may want to click the Fit All In icon  to fit all data in the window.



Notice that the toolbars change depending on whether a CATIA Version 4 model or a CATIA Version 5 document is selected.

The result should look something like this:





When copying V4 drawing data to V5, bear the following in mind:

- Whatever the standard of the V4 view was prior to being copied into CATIA Version 5, once in V5 its standard is that selected when you open the CATDrawing document.
- The smallest unit that you can copy is the view. All the elements that go to make up this view are included in the copy
- In the V5 Drafting mode Working Views (Edit->Working Views) the copy described above creates a V5 view with the same name as in V4.  
In the V5 Drafting mode Background (Edit->Background) the V4 elements are copied into the background view of the V4 view.



# Manipulating CATIA Version 5 Data in CATIA Version 4

Task	Purpose
<a href="#">About Manipulating CATIA Version 5 Data in CATIA Version 4</a>	Specify requirements for the manipulation of CATIA V5 data in CATIA Version 4
<a href="#">Saving CATIA Version 5 CATPart Documents As CATIA Version 4 Models</a>	Save CATIA Version 5 CATPart documents as CATIA Version 4 models



# About Manipulating CATIA Version 5 Data in CATIA Version 4



In order to read and use CATIA Version 5 CATPart documents in CATIA Version 4, the following steps must be carried out:

1. First of all, CATIA Version 4 Release 3.1 must be installed on a UNIX machine.
2. You must then install a CATIA V5R3 configuration which includes the V4 Integration product (V4I), for example the MD2 configuration (Mechanical Design 2) and set up the appropriate licenses.
3. In order to be able to access the CATIA Version 5 software from a CATIA Version 4 session you must add to the USRENV.dcls file, both in the user and administrator environments, the following:

```
CATIA.ENVTV5 : STRING;  
CATIA.ENVTV5 = `$HOME/CATENV/CATIA.V5R3.B03.sh';  
CATIA.MACHV5 : STRING;  
CATIA.MACHV5 = 'my_server_machine';
```

where \$HOME is the root \$HOME and where CATIA.V5R3.B03.sh is the shell downloaded to \$HOME/CATEnv/ using the CATIA Version 5 installation procedure.



Make sure that:

- the path used to access the CATPart document is the same and the file system containing the CATPart and the file system containing the CATPart is shared between the V4 and V5 machines
- the user \$HOME is the same on the V4 and V5 machines and is shared between both.



Once the above statement is added to the USRENV.dcls file it is possible to use Version 5 data in a Version 4 session within a client/server environment with the appropriate V5 license set up on the server machine. However, if the last two lines of the declaration above are not specified it is assumed by the system that both V4 and V5 are installed on a local machine.


This avoids the error message "Dynamic storage cannot be allocated" displaying when you attempt to read CATIA Version 5 data.


4. You are now ready to manipulate CATIA Version 5 data in CATIA Version 4. You can, for example:

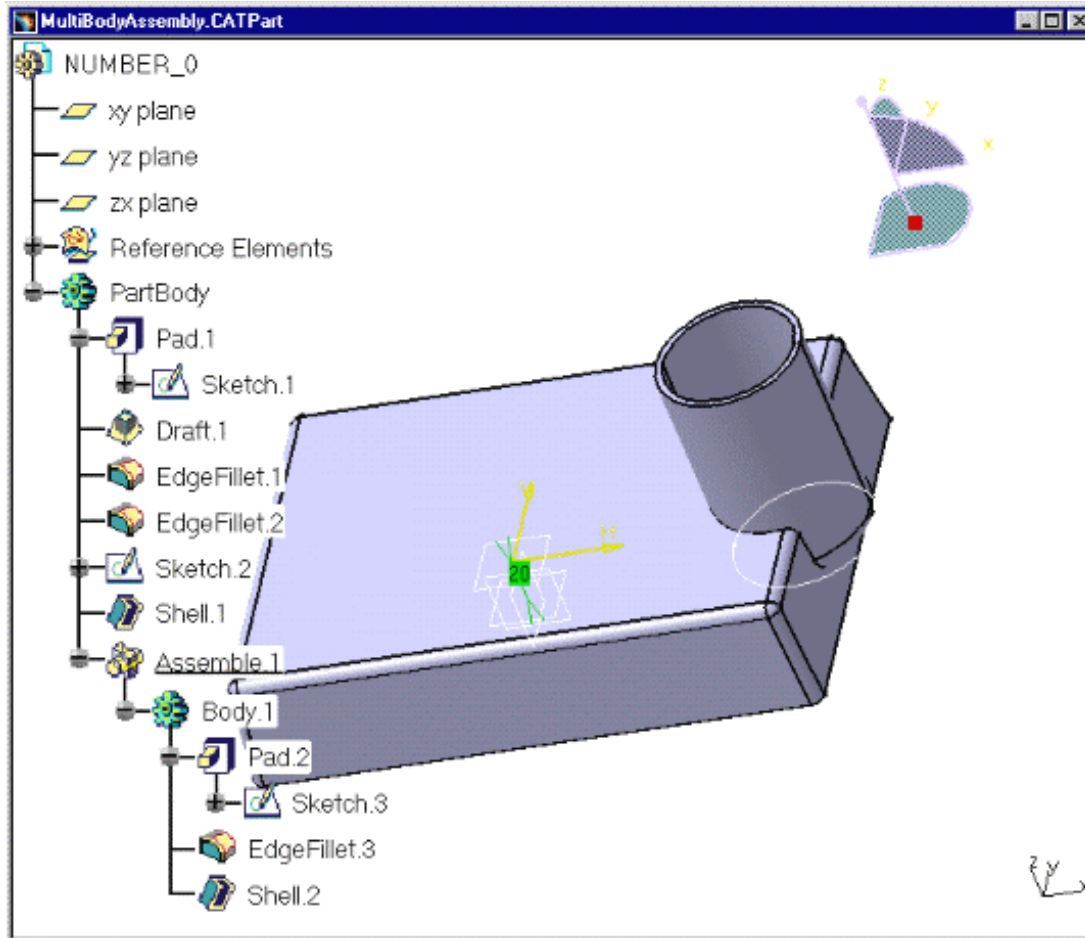
- browse V5R3 documents as passive models using FILE/OPEN
- read a V5R3 CATPart document and copy exact solids and surface entities into CATIA Version 4 using MODELS/COPY.




# new Saving CATIA Version 5 CATPart Documents As CATIA Version 4 Models

 This task shows you how to save CATIA Version 5 CATPart documents as CATIA Version 4 models.


 Open the document \doc\samples\V4Integration\MultiBodyAssembly.CATPart. It looks like this:

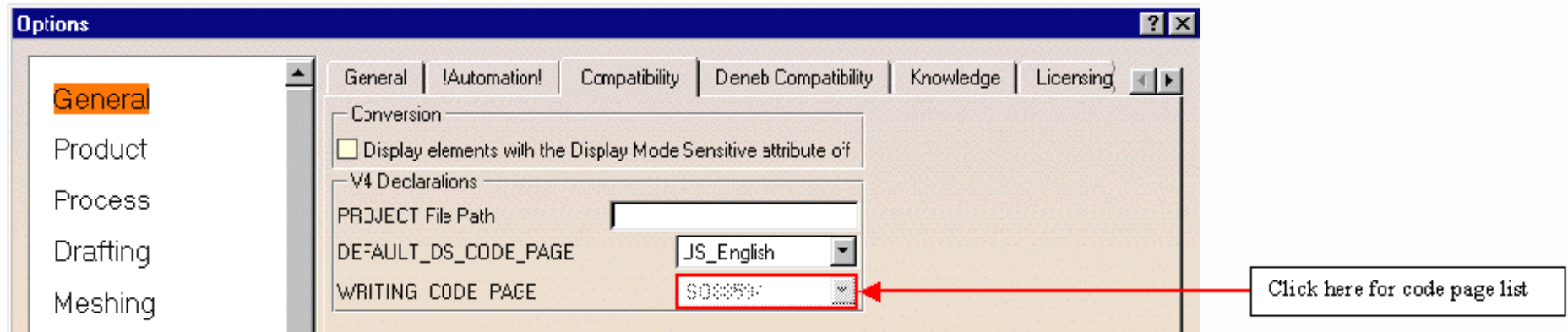


 You will remember that in Version 4 the **declaration parameter** `catsite.WRITING_CODE_PAGE` declares the code page to be stored in the CATIA data to be written. Such information such as the writing code page was specified in V4 by means of the parameter settings in the declaration files. These declaration parameters are no longer supported in CATIA Version 5. Such information must therefore be provided by means of the dialog boxes described below, **before** attempting to save a V5 CATPart document as a V4 model.

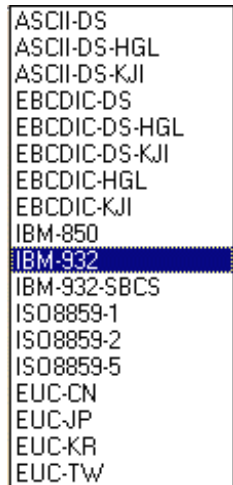
If you want to use the writing code page ISO-8859-1 go straight to step 4. It is the default value so normally, unless another code page was already specified, you can go ahead with the save.

However, if you want to use a writing code page other than ISO-8859-1 start with step 1.

- 
1. Select the Tools->Options... command. The Options dialog box appears with the General category selected in the left-hand column.
  2. Click the Compatibility tab. The following dialog box appears:



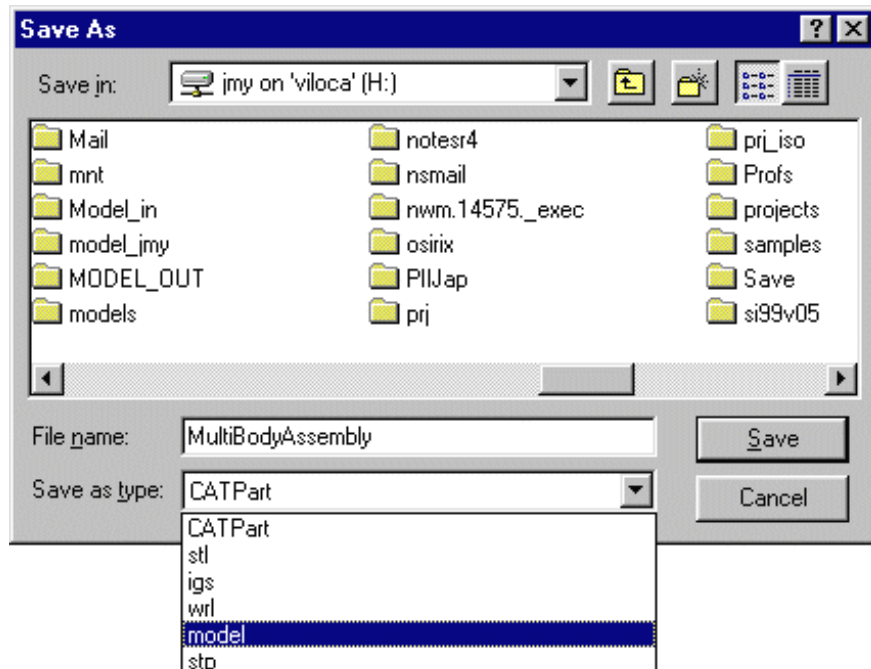
3. Open the `WRITING_CODE_PAGE` list in the V4 Declarations part of the dialog box (indicated by the arrow above), select the appropriate code page and click OK.



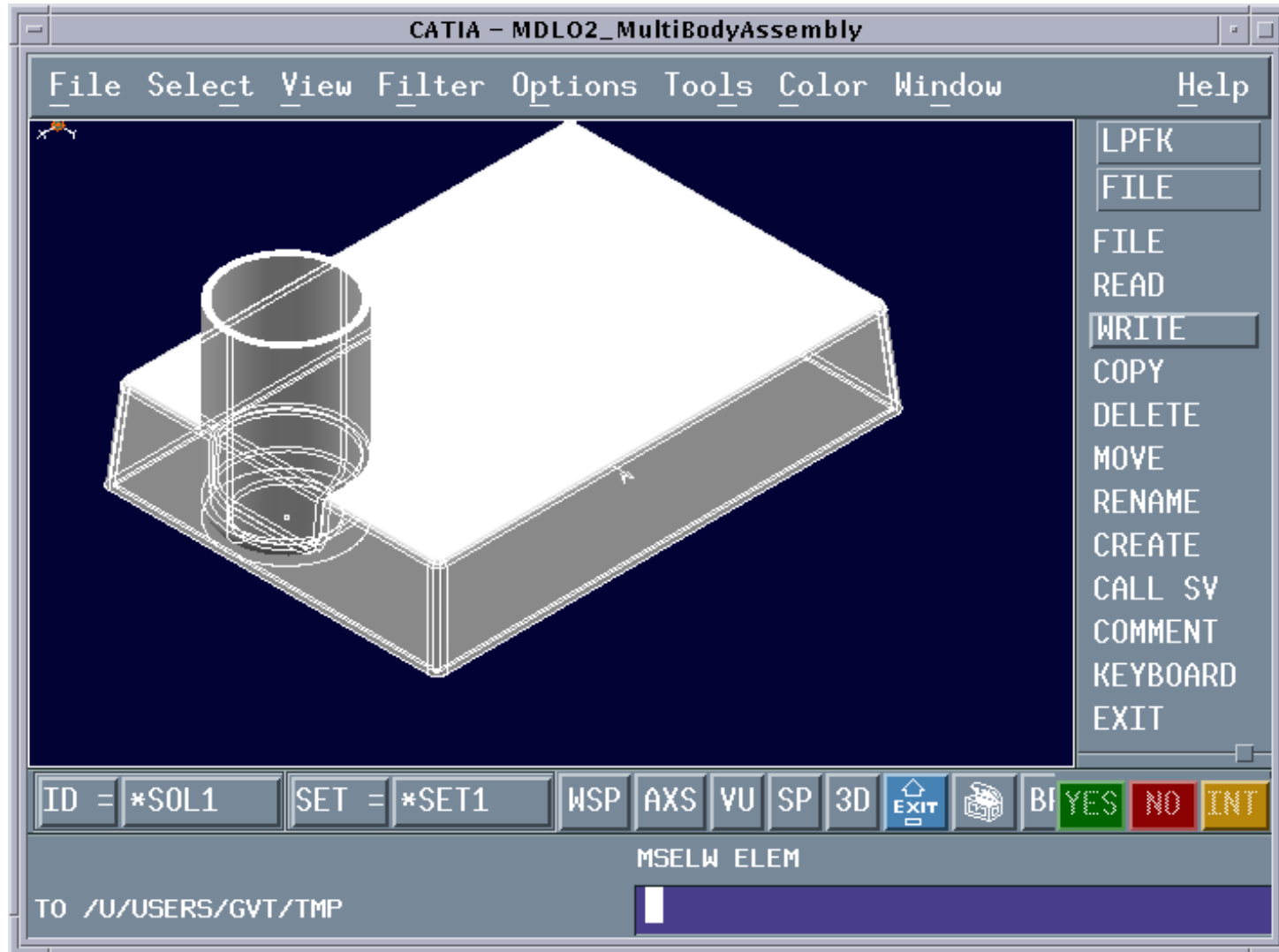
4. Select the File->Save As... command.

5. In the Save As dialog box, select the location of the .model document to be saved and rename it (or not) as required.

6. Click the Save as type: list and select model in the list displayed as shown below.



7. In the same box, click on Save. The MultiBodyAssembly.model just created can now be opened in CATIA Version 4 and will look something like this:





Up



About Manipulating V5 Data




Saving a V5 Part As a V4 M

# Using CATIA Version 4 Libraries (on UNIX Only)


Task	Purpose
<a href="#">Opening CATIA Version 4 Library Objects in CATIA Version 5 on Windows</a>	Open CATIA Version 4 library objects in CATIA Version 5 on Windows
<a href="#">Opening CATIA Version 4 Library Objects in CATIA Version 5 on UNIX</a>	Open CATIA Version 4 library objects in CATIA Version 5 on UNIX
<a href="#">Displaying CATIA Version 4 Library Keywords in CATIA Version 5</a>	Display CATIA Version 4 library keywords in CATIA Version 5
<a href="#">Applying Filters to CATIA Version 4 Library Families</a>	Apply filters to CATIA Version 4 library families
<a href="#">Converting a CATIA Version 4 Library Objects into CATIA Version 5 CATPart Documents</a>	Convert a CATIA Version 4 library object into a CATIA Version 5 Part document
<a href="#">Converting CATIA Version 4 Libraries into CATIA Version 5 Catalogs</a>	Convert a CATIA Version 4 library into a CATIA Version 5 catalog



# Opening CATIA Version 4 Library Objects in CATIA Version 5 on Windows

 This task shows you how to open a CATIA Version 4 library object in CATIA Version 5 on Windows.


 Make sure that an http server has been installed on the machine where the objects reside.

 A CATIA Version 4 library is a means of storing objects such as details, symbols, NC mill and lathe tools and beam sections. A library is made up of a number of **families** each of which contains **objects**. Keywords are attributed to these families to enable classification and easy retrieval of the objects contained in them.

Just like Version 4 models, Version 4 library objects cannot be edited as such. However, it is possible to convert the library objects into CATPart documents. Read-only operations are of course allowed. Accessing CATIA Version 4 library objects does not modify these objects.

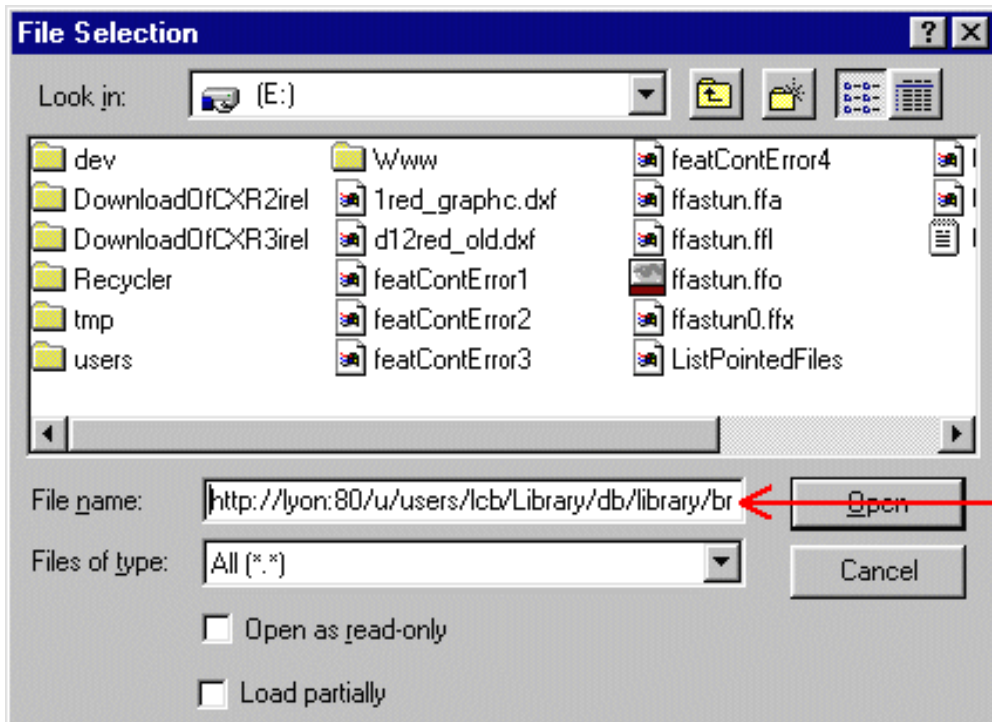
Only SPACE details can be read in CATIA Version 5. Other library objects such as DRAW details, symbols, sections and tools cannot be read in CATIA Version 5. A SPACE detail that points to external library objects can be read only if all the objects pointed to are SPACE details. This means, for example, that no detail containing structure pointing to a section can be read.



1. Click the Open icon  or select the File->Open command.
2. In the File Selection dialog box, enter, in the field File name, the whole address of the library object you want to open, including the library name, as shown below. If you enter the wrong location or if the library name cannot be found, exit the dialog box and start over.

The address to be specified should look something like this:

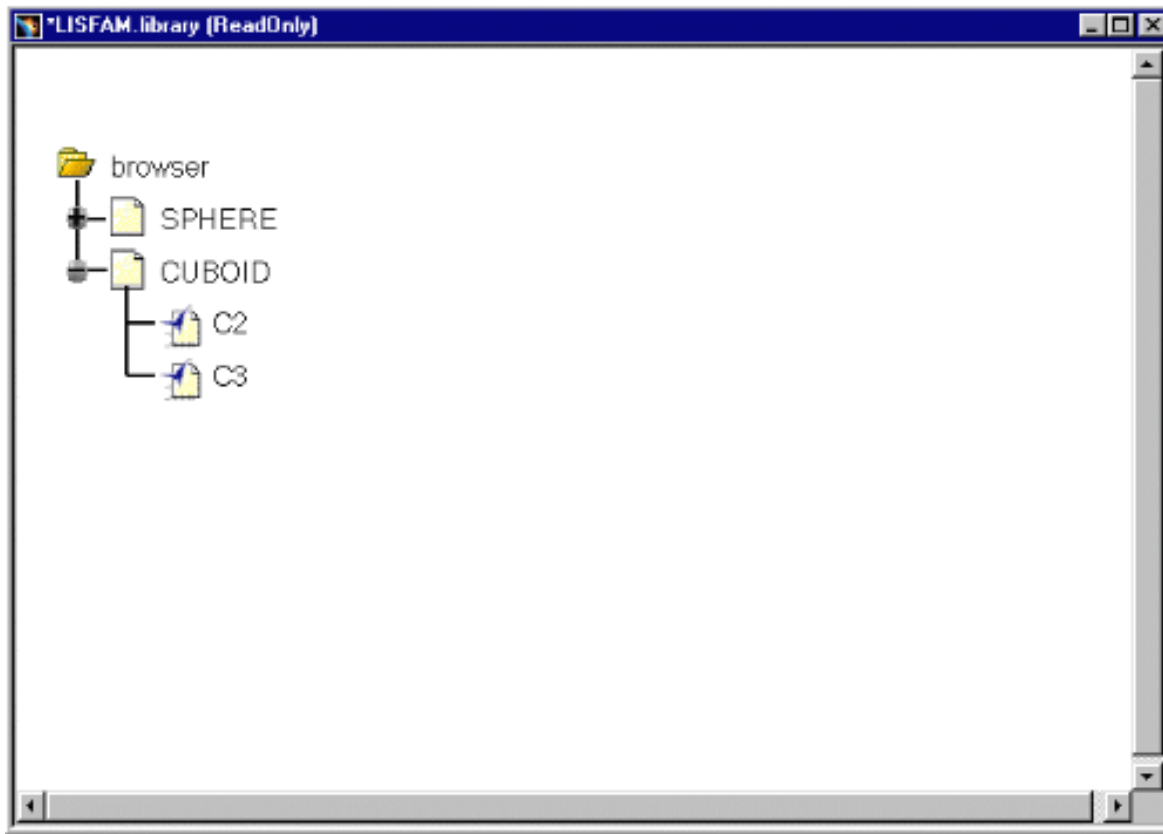
`http://UNIXserver: port/libraryobjectlocation/libraryname`



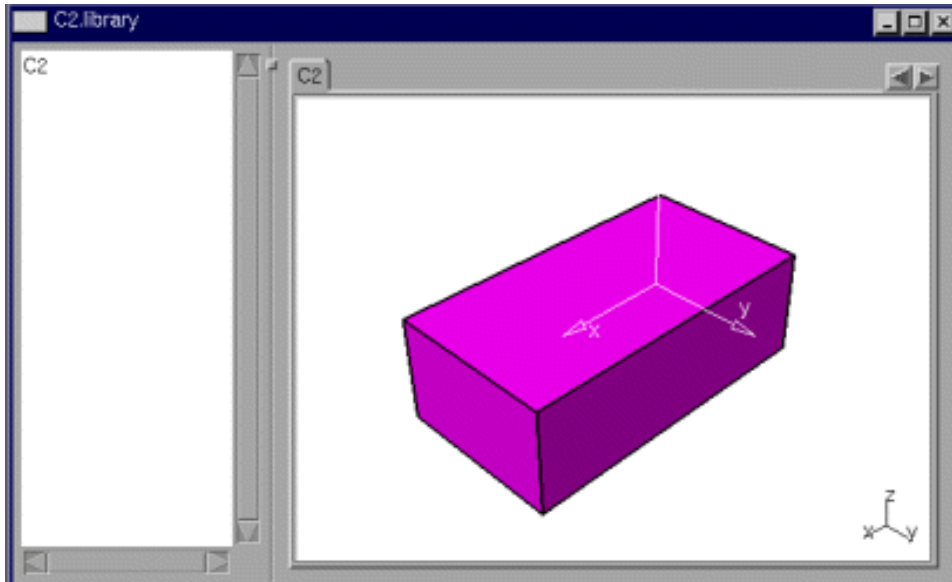
Enter the full address here including the library name


The name of the library appears.

4. Double-click on the library name to display the library families. Then double-click on one of the objects belonging to these families, CUBOID in the example shown below, to see the whole structure of the library. A dialog box similar to this is then displayed:



5. To open a library object, double-click on it. Alternatively, select the library object, right-click and select Open. The result, providing you with CATIA Site Navigator capabilities, is as follows:




 Library objects residing on UNIX can of course also be accessed from UNIX. To see how, refer to [Opening CATIA Version 4 Library Objects in CATIA Version 5 on UNIX](#).




# Opening CATIA Version 4 Library Objects in CATIA Version 5 on UNIX

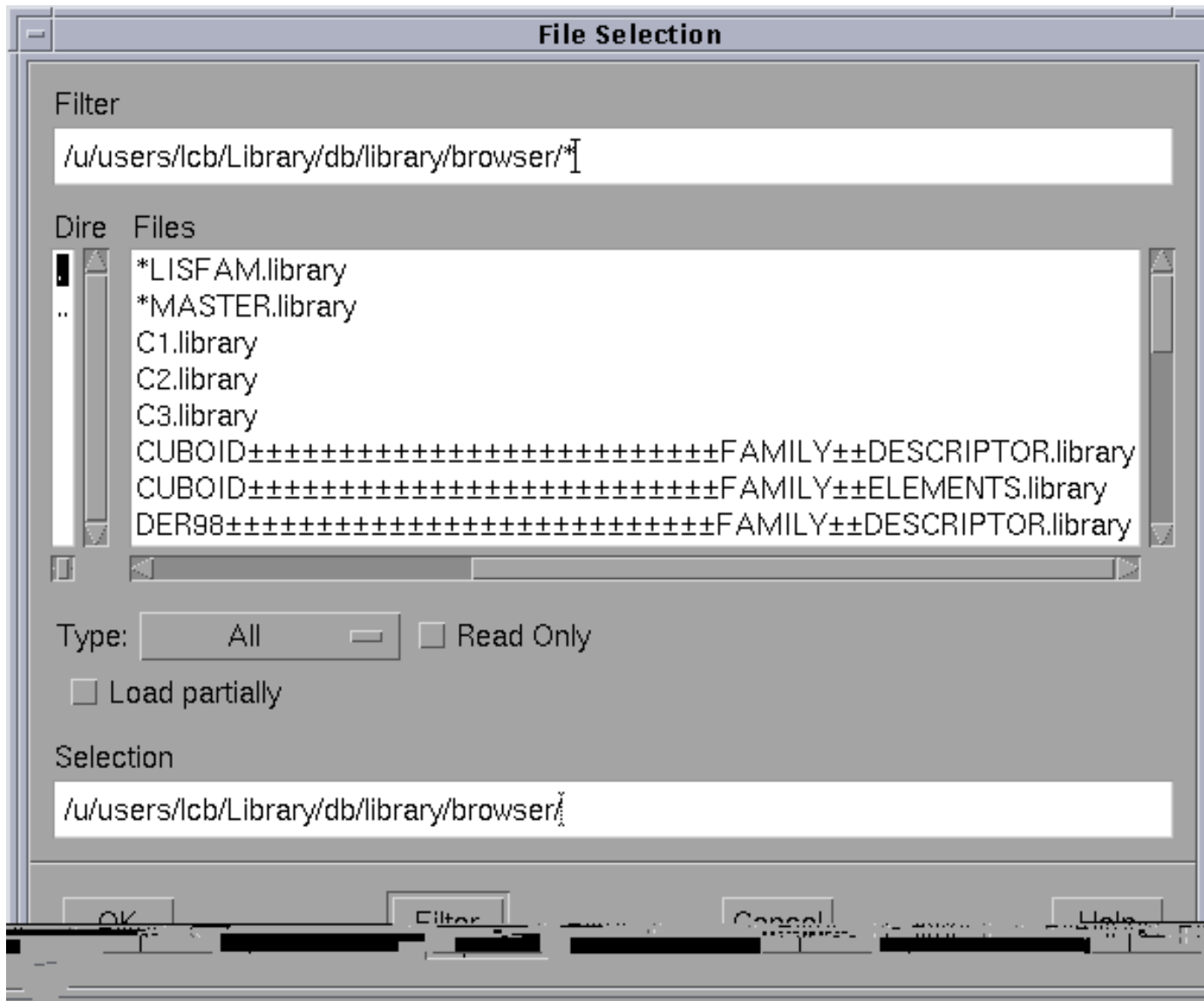
 This task shows you how to open a CATIA Version 4 library object in CATIA Version 5 on UNIX.

 Just like Version 4 models, Version 4 library objects cannot be edited as such. However, it is possible to convert the library objects into CATPart documents. Read-only operations are of course allowed. Accessing CATIA Version 4 library objects does not modify these objects.

Only SPACE details can be read in CATIA Version 5. Other library objects such as DRAW details, symbols, sections and tools cannot be read in CATIA Version 5. A SPACE detail that points to external library objects can be read only if all the objects pointed to are SPACE details. This means, for example, that no detail containing structure pointing to a section can be read.



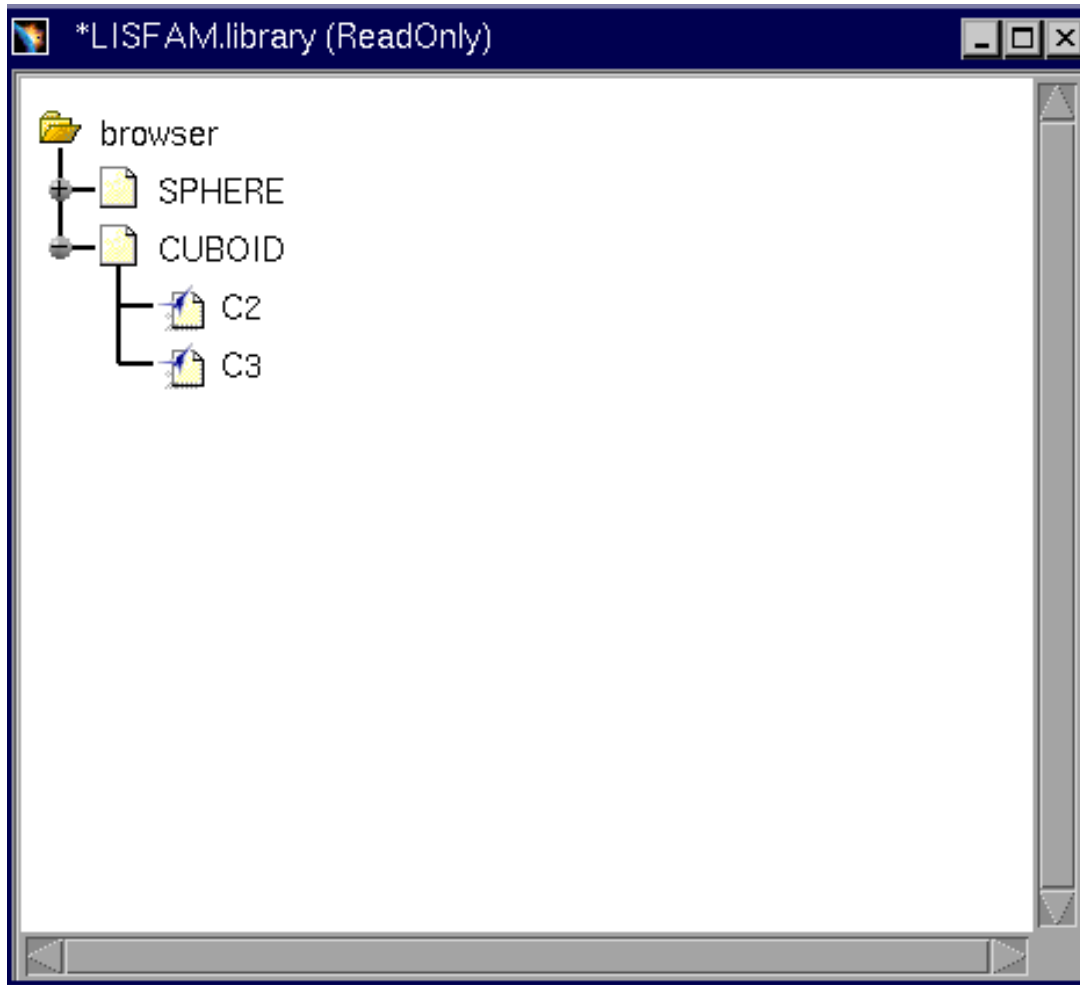
1. Click the Open icon  or select the File->Open command.
2. In the File Selection Box, select the file location.
3. Click the Files of type: list, select library as the document type. The File Selection dialog box is displayed.



Note that the items displayed in this dialog box do not correspond to the hierarchical structure of the library. Families and library objects are displayed at the same level. For more information about this, refer to the relevant CATIA Version 4 documentation.

4. To access the library family list, double-click on either \*LISFAM.library or \*MASTER.library. Both these files are required for library management purposes.

The name of the library appears. Double-click on the library name to display the library families. Then double-click on the objects belonging to these families to see the whole structure of the library. The following dialog box is then displayed:

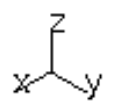
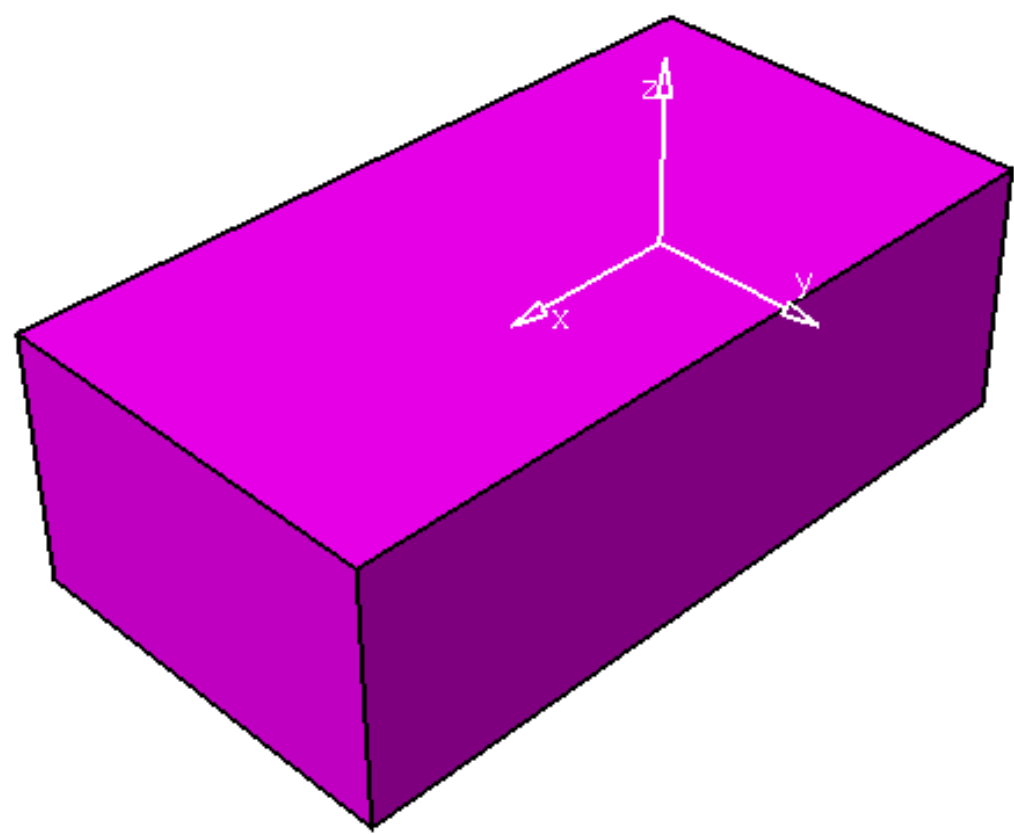


5. To open a library object, double-click on it. Alternatively, select the library object, right-click and select Open. The result, providing you with CATIA Site Navigator capabilities, is as follows:

C2



C2



6. To access a family directly, double-click one of the following:

- *family\_name*\_\_\_\_\_ FAMILY\_\_\_\_\_ DESCRIPTOR.library
- *family\_name*\_\_\_\_\_ FAMILY\_\_\_\_\_ ELEMENT.library.

Both these files are required for library management purposes.

A dialog box appears showing the name of the family. Double-click on this family name to display the whole structure of the family then double-click on a library object to open it. Alternatively, select the library object, right-click and select Open. The result is the same as above.

7. To access a library object directly, select it in the File Selection dialog box. The result is the same as above.



Library objects residing on UNIX can also be accessed from Windows using the http protocol. To see how this is done, refer to [Opening CATIA Version 4 Library Objects in CATIA Version 5 on Windows](#).



# Displaying CATIA Version 4 Library Keywords in CATIA Version 5



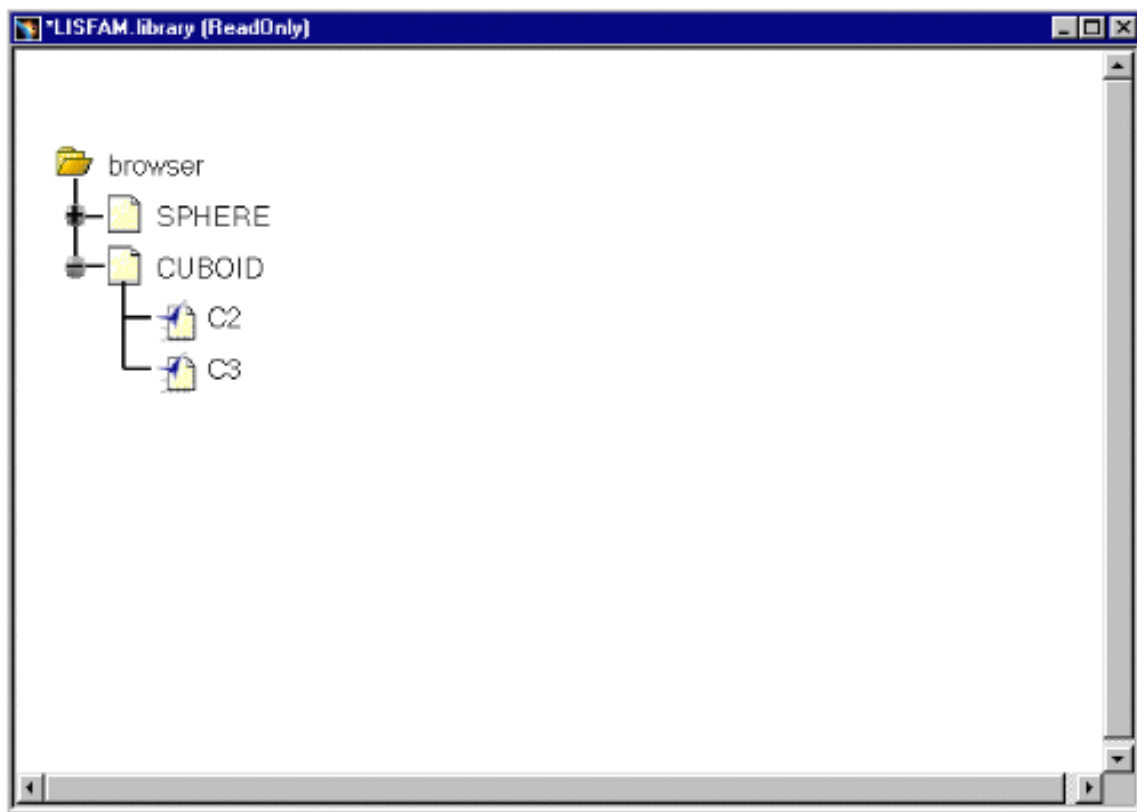
This task shows you how to display CATIA Version 4 library keywords in CATIA Version 5.



Remember that in CATIA V4, some properties associated with a family, called keywords, can be assigned to objects of the given family. For more information, see the relevant CATIA Version 4 documentation.

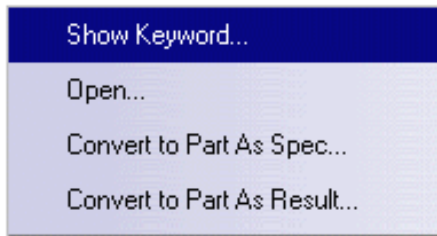


Have a library structure open in CATIA Version 5 showing the families and objects it contains as shown below.

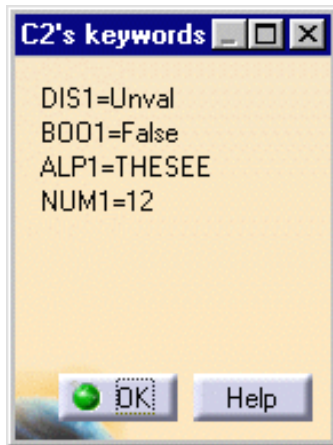


1. Select the library object whose keywords you want to display (C2, for example) and right-click.
2. In the menu that appears, select Show Keyword...


2. In the menu that appears, select Show Keyword...



3. There appears a list of the keywords for the library object selected.




4. Click OK to close the list.

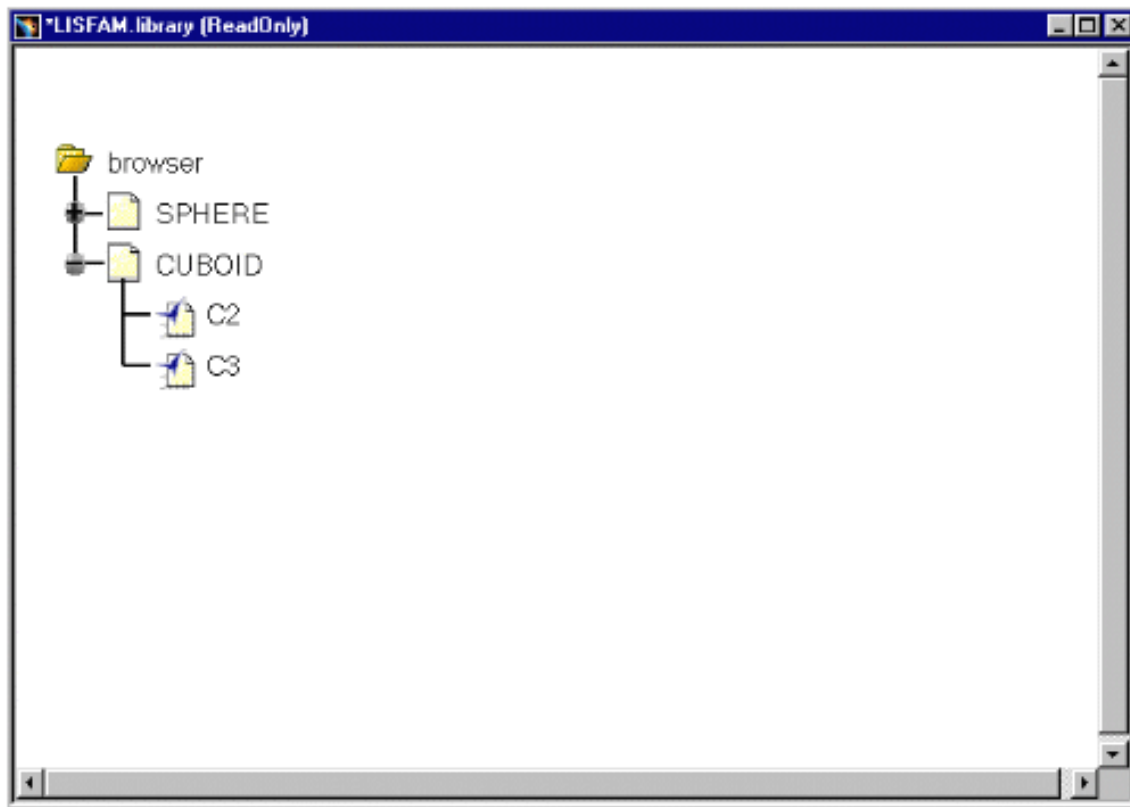
 If a family's keywords are linked to PROJECT file attributes make sure you specify the PROJECT file path in the Compatibility tab of the Options dialog box (displayed using the Tools->Options command).




# Applying Filters to CATIA Version 4 Library Families

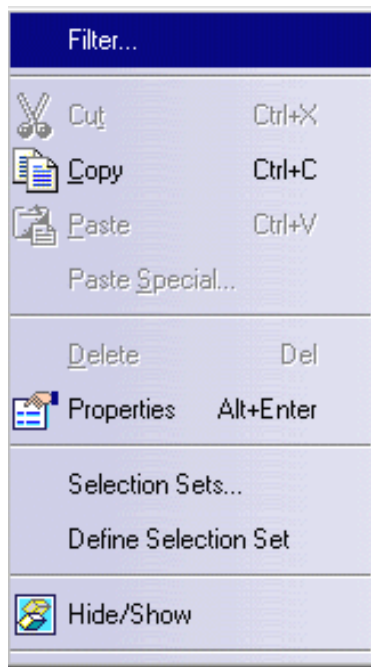
 This task shows you how to apply a filter to a CATIA Version 4 Library family in CATIA Version 5.

 Have a Version 4 library and its contents open as shown below.

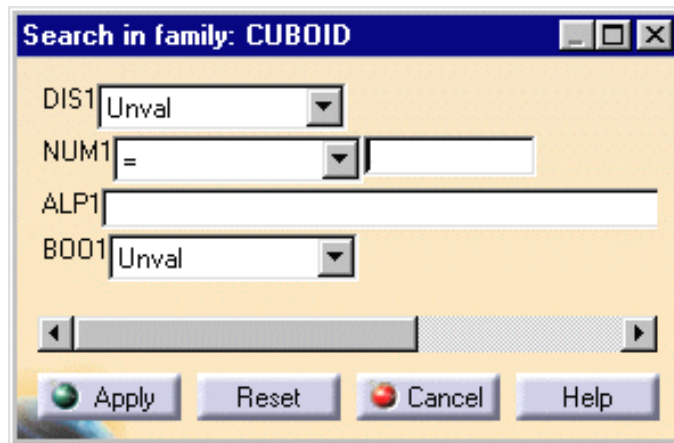


 1. In the dialog box above, display all the objects contained within a family, CUBOID for example, by clicking on the plus sign opposite the family name, select the family name and right-click.

2. Select Filter in the menu that appears.



3. The dialog box opposite is displayed.




4. Complete the fields, then click Apply to redisplay the whole family structure.

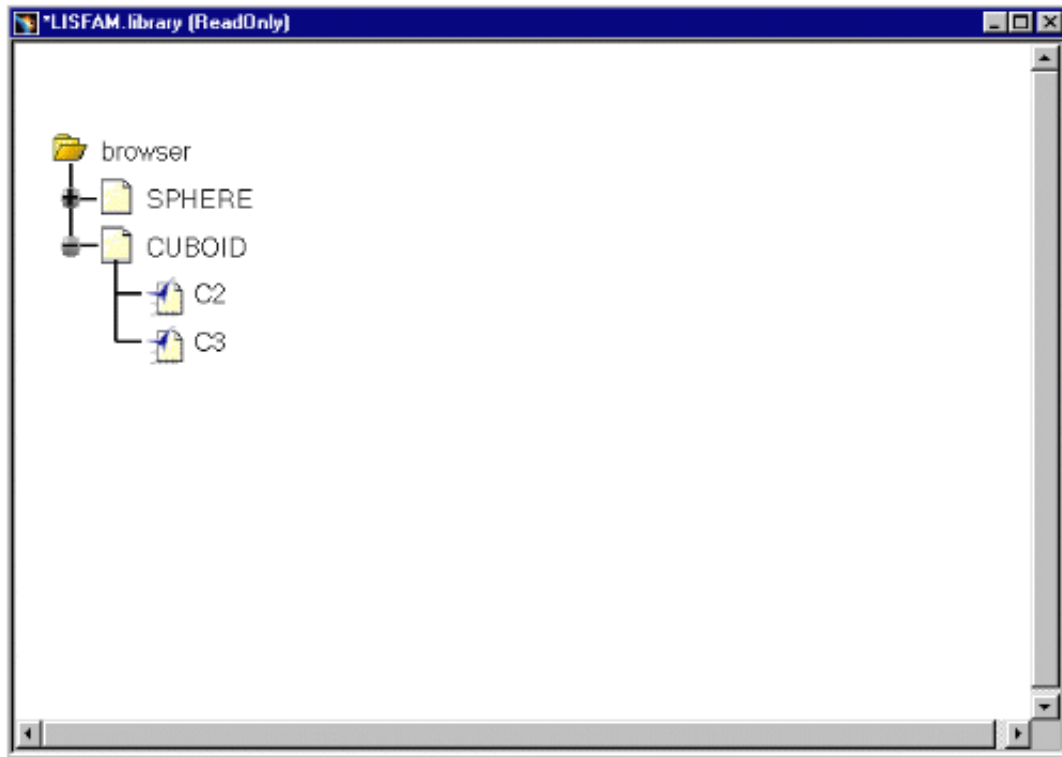
5. Click Reset to display the whole family structure as it was prior to the filter application.



# Converting CATIA Version 4 Library Objects into CATIA Version 5 CATPart Documents

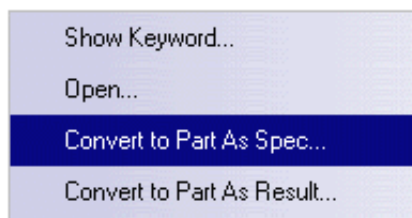
 This task shows you how to convert a CATIA Version 4 library object into a CATIA Version 5 Part document.

 Have a library structure open in CATIA Version 5 showing the families and objects it contains.



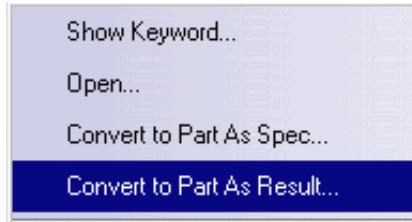
1. Select the library object and right-click.
2. In the menu that appears, select either:

Convert to Part As Spec... to convert the object into a V5 CATPart document as specifications (identical to the operation described in [Copying CATIA Version 4 Model Data to CATIA Version 5](#)).





or:

Convert to Part As Result... to convert the object into a V5 CATPart document as geometry (identical to the operation described in [Copying CATIA Version 4 Model Data to CATIA Version 5](#)).



A new window is displayed showing the converted object. The selected library object is now displayed as a new Part (equivalent to creating a Part and performing a copy/paste operation) and you are provided with all the Part Design workbench capabilities.

You may now want to click the Fit All In icon  to fit all data in the window.

 Alternatively, you can of course:

1. open the library object
2. copy/paste it into a Part already open in your CATIA session.

In both cases, only solid-type elements (exact solids i.e. Version 4 CATGEO 17-1 and polyhedral solids i.e. Version 4 CATGEO 17-2) can be pasted into a Part document. If details reference other subdetails the whole of the geometry is exploded in one single CATPart document.



# **new** Converting CATIA Version 4 Libraries into CATIA Version 5 Catalogs



This task shows you how to convert a CATIA Version 4 library into a CATIA Version 5 catalog.



Before embarking on any conversion make sure you are familiar with the concepts described in "[About Catalogs](#)" in the *CATIA Version 5 Infrastructure User's Guide*.

It would also be instructive to read "[Converting CATIA Version 4 Library Objects into CATIA Version 5 CATPart Documents](#)".

Have the documents table.txt and ConvertLibrary.CATScript in the directory \Online\Samples\V4Integration.



## **Migrating the data structure**

The batch operation creates a catalog chapter for each **DETAIL** type library family (see "[Creating a Catalog](#)" in the *CATIA Version 5 Infrastructure User's Guide*).

Other types of families (SYMBOL, SECTION, NCMILL, NCLATHE) **cannot** be converted.

The keywords relating to the library family are converted into catalog keywords as shown in the table below. These keywords define the associated chapter.

V4 Keywords	V5 Keywords
Discrete	String
Alphanumeric	String
Numeric	Real (no unit retrieval)
Boolean	Boolean

The batch operation creates a chapter referencing each of the previously described terminal chapters. The chapter created becomes the main entry point of the library converted into a catalog.

## **Migrating the actual data**

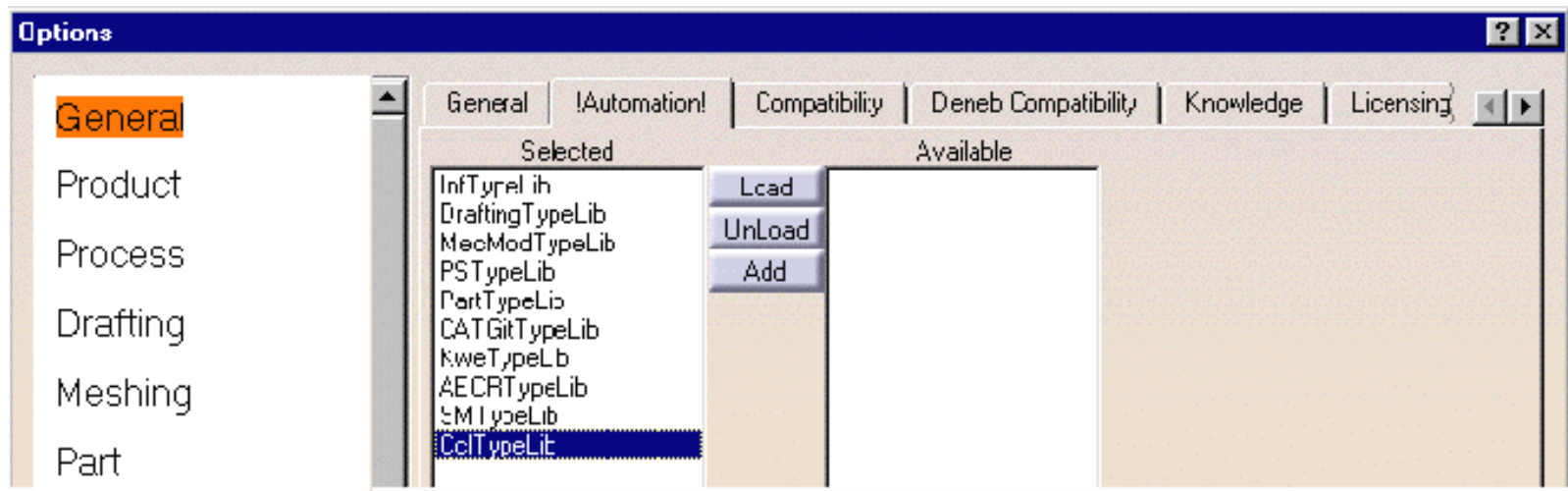
Bear in mind the following:

- No V4 link is kept between the V4 library and the new V5 catalog.
- Only SPACE details can be migrated to CATIA Version 5 i.e. converted into CATPart documents. Other library objects such as DRAW details, symbols, sections and tools cannot be migrated.
- Only solid-type elements (exact solids i.e. Version 4 CATGEO 17-1 and polyhedral solids i.e. Version 4 CATGEO 17-2) can be pasted into a Part document.  
If details reference other subdetails the whole of the geometry is exploded in one single CATPart document.
- Only DETAIL type families are converted into a catalog chapter.



1. Before being able to use the CATScript file in order to run the batch operation you must make sure the CclTypeLib library is loaded. This library declares the library conversion method. To do this, select the Tools->Options... command. The Options dialog box appears with the General category selected in the left-hand column.

2. Click the !Automation! tab. The following dialog box appears.



3. If the CclTypeLib library is not in the Selected list click on the Add button and retrieve the library from intel\_a/code/bin. Make sure it is in the Selected list before clicking OK.



If the batch operation is unsuccessful check to make sure that CclTypeLib has been declared.

4. Open the file ConvertLibrary.CATScript: and select the appropriate setting where you see an arrow.

```
Language="VBSCRIPT"

Sub CATMain()

'Defines the conversion format option of the space details in parts
AS_SPEC = 0
AS_RESULT = 1
'Choose the conversion format you want of details in parts: as spec or as result

CONVERSION_FORMAT = AS_SPEC
'Defines the batch mode option
MIGRATION = 0
SIMULATION = 1
'Choose the batch mode you want

BATCH_MODE = MIGRATION
'Specify the absolute UNIX directory of the library
'(for access from Windows use the http protocol)

libraryDirectory = "http://..."
'Specify the absolute directory of the catalog

catalogDirectory = "..."
'Specify the absolute UNIX directory of the project if the library needs one.
'If not, specify an empty location (projectDirectory=" ")

projectDirectory = "http://..."
```

```
'Specify the absolute path of the character conversion table
'to convert V4 file names of details and families
'into V5 file names of parts and chapters. If the default rules are sufficient
'(see Consequences of migration on filenames below) specify an empty location (tablePath=" ")
```

```
tablePath = "..."
```



```
'Creates a CatalogDocument type document
```

```
Dim Catalog As Document
```

```
Set Catalog = CATIA.Documents.Add("CatalogDocument")
```

```
'Calls CreateCatalogFromLibrary method on Catalog
```

```
Catalog.CreateCatalogFromLibrary libraryDirectory, projectDirectory, catalogDirectory, tablePath,
CONVERSION_FORMAT, BATCH_MODE
```

```
'Closes the document
```

```
Catalog.Close
```

```
End Sub
```

As you can see, this file, written in the VBScript language, contains the information required to batch-convert a V4 library into a V5 catalog file in the location you specify. It can be used on both Windows and UNIX. (Do not forget to use the http protocol on Windows to access the library and the PRJ file.)

In the example above the CATScript file contains just one set of instructions to batch-convert a single library. However, a CATScript file can of course include several series of instructions for the conversion of several libraries.

5. Select the appropriate setting where you see an arrow and close the file when you have finished.



## Consequences of migration on filenames

On Windows, it is extremely important to be aware of the consequences of migration on filenames. Before reading any further see "[About Filenames](#)" in the *CATIA Version 5 Infrastructure User's Guide*. In view of the restrictions described in this section, the following default conversion table will apply to new CATPart documents converted from library objects:

Original Character's ASCII Code	Character to be Replaced	New Character String
0x22	"	_Inch
0x2a	*	x
0x2f	/	_
0x3a	:	_
0x3c	<	_
0x3e	>	_
0x3f	?	_
0x5c	\	_
0x7c		_

National characters will be replaced by default by \_xxx where xxx is the ASCII code of the character replaced. The default correspondence table can be modified but you cannot avoid the replacement of these characters. If, for example, you delete the line |"\*" the table will be declared invalid. The two characters most likely to occur are:

- 0xb1
- 0xb4

which correspond to the characters used in V4 model names as replacements for the blank character " " and slash character "/" respectively. To avoid excessively long document names it is advisable to modify the default conversion table.



Only national characters and those characters not supported by Windows can be declared in the correspondence table. For example, you cannot decide that you want to change all characters from "a" to "w".

6. If you now wish to act on the information above and modify the string to replace one of the characters in the default conversion table, open the file table.txt and make the changes required.

7. You are now ready to run the batch operation. Select the Tools ->Macro->Macros command.

8. In the dialog box that appears, select the appropriate ConvertLibrary.CATScript file. Make sure that at the bottom of the box External File is selected indicating the location of the macro. Click on Run.

If you chose SIMULATION the batch operation will:

- generate a report in the same location as the catalog file, allowing you to verify whether migration would be successful or not
- detect errors in data migration
- detect possible occurrences of V5 names following the conversion of V4 names into V5 names.



If more than one occurrence of the same V5 name is detected the older CATPart documents are overwritten. You are therefore strongly advised to modify the conversion table so that CATParts are not created with the same name.

If you chose MIGRATION the batch operation will generate:

- a report in the same location as the catalog file, allowing you to verify successful migration
- and a catalog with the name of the library.

9. If you want to open the catalog you have just created see "[Navigating Through a Catalog](#)" in the *CATIA - Infrastructure User's Guide*.



If a library is migrated to the same directory twice the catalog (both catalog and CATParts) generated by the previous migration is overwritten by the new one.



# Advanced Tasks

Theme	Purpose
<a href="#">Using CDMA Data in CATIA Version 5</a>	Use CDMA data in CATIA Version 5 and perform associated operations
<a href="#">Using VPM Data in CATIA Version 5</a>	Use VPM data in CATIA Version 5 and perform associated operations

# Using CDMA Data in a CATIA Version 5 Assembly (on UNIX Only)

Task	Purpose
<a href="#">Connecting an ORACLE or DB2 Database to Your V5 Environment</a>	Connect an ORACLE or DB2 database to your V5 environment
<a href="#">Mapping CDMA and Assembly Attributes on UNIX</a>	Map CDMA and Assembly attributes on UNIX
<a href="#">Adding a CDM Representation to a Product on UNIX</a>	Add a CDM representation to a product on UNIX
<a href="#">Adding a CDM Product to a Product on UNIX</a>	Add a CDM product to a product on UNIX



Up



Using CDMA in V5 (on UNIX)



Using VPM Data in V5 (on UI)

# Connecting an ORACLE or DB2 Database to Your CATIA Version 5 Environment (on UNIX Only)



This task shows you how to connect an ORACLE or DB2 database to your CATIA Version 5 environment, so that you can later use CDMA data as representations or products in a CATIA Version 5 assembly.



Have your CATIA Version 5 environment prepared for receiving Version 4 CDMA data. Bear in mind the following:

- Before importing CDMA data residing on ORACLE databases, you should check that these databases are properly described in the tsnames.ora UNIX file and export the TNS\_ADMIN variable by specifying the path of the tsnames.ora file directory. When using ORACLE8, you must export the ORA\_NLS33 variable by specifying the path of the directory containing the lx... .nlb files (normally located in /\$ORACLE\_HOME/common/nls/admin/data).
- Before importing CDMA data residing on a DB2 database, you should check that the client DB2 database installation has been performed for the database you plan to use.

For further information, see the appropriate CDMA, Assembly, ORACLE or DB2 documentation.



Only data from CDMA databases in the RAW\_COID = TRUE mode (binary COID type attributes) and the NEW\_MATRIX = 'YES' mode (positioning matrices in FLOAT format) can be imported.

Models intended to be used as representations should have been stored by using the:

- DBLFCAT
- DBLFAIX
- DBLFCDM

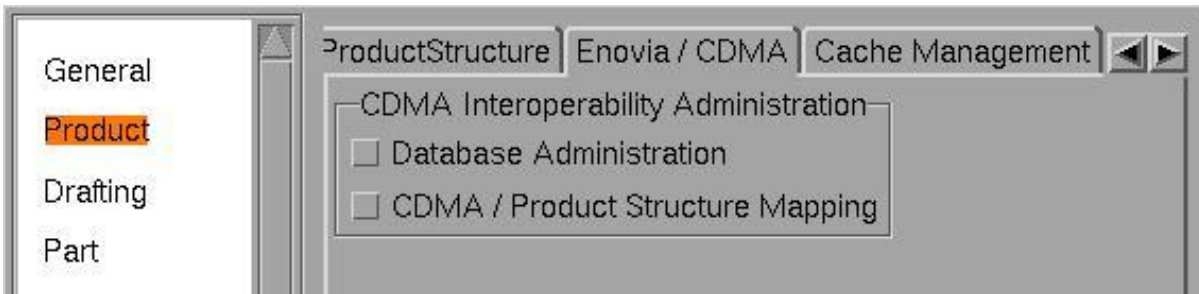


access methods. CATIA Version 5 cannot access models stored by means of any other method.

The product generated by adding some CDM representations and/or models can be modified just like any CATProduct document in CATIA Version 5. However, modifications cannot be written back to the CDMA database.



1. Select the Tools->Options... command. The Options dialog box appears.
2. Select the Product category in the left-hand column and click on the Enovia/CDMA tab.



3. Select the option Database Administration to declare the logical name of your database in your Version 5 environment and click on the Apply button.

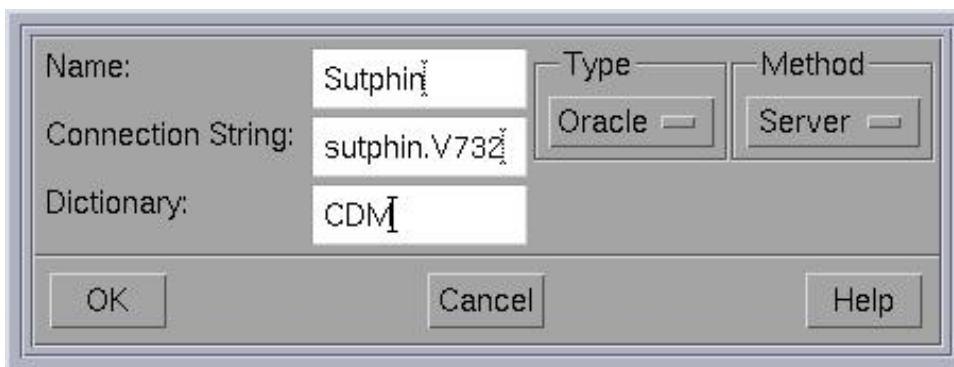
This step is intended to define the parameters for connecting to the CDMA databases you wish to use.

The Database Server Administration dialog box is displayed.



The Known Servers frame should initially be empty.

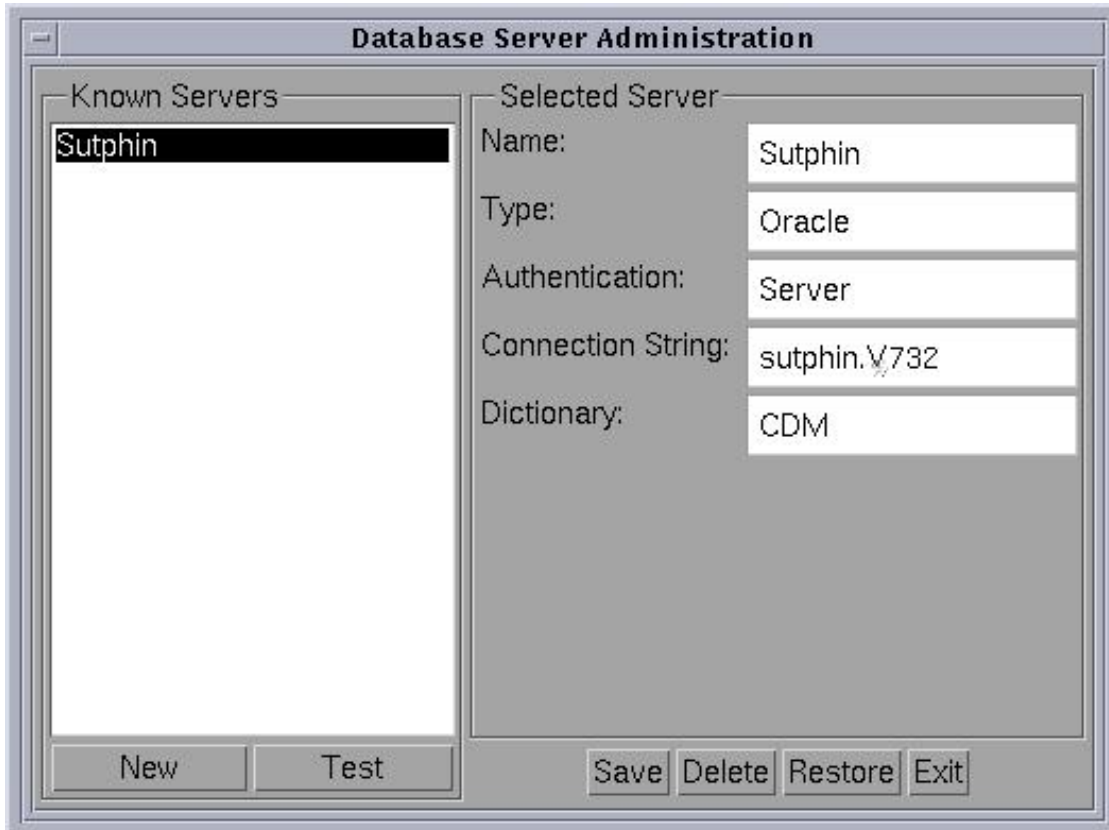
4. Click on New to declare a new server. The dialog box below is displayed.



5. In this dialog box, enter the server's logical name in the Name field.

6. In the Connection String field, enter connection string declared in the tsnames.ora file (ORACLE) or the database alias declared by the configuration utility (DB2).

7. In the Dictionary field, enter the CDMA dictionary name and click OK. The updated Database Server Administration dialog box is displayed. The name of the server you have just declared should appear in the Known Servers box as well as the description of the server in the Selected Server box.



8. Click on Save to save your new server configuration.

9. Click on Test to see whether the required server can be properly connected. The following dialog box is displayed.



10. Complete the User Name and Password Name fields. Click OK. The following dialog box is displayed.



11. Click OK to terminate the Database Administration dialog.

You have just completed the first administration step. All the declarations required to connect your database have been made.



# Mapping CDMA and Assembly Attributes on UNIX



This task shows you how to map CDMA and Assembly attributes.



You should have already connected an ORACLE or DB2 database to your CATIA Version 5 environment (see [Connecting an ORACLE or DB2 Database to Your CATIA Version 5 Environment](#)).



1. In the Options dialog box, click on the CDMAInterop tab and select the option CDMA / Product Structure Mapping to define the mapping between the CDMA and Assembly attributes. Click on the Apply button.



As Assembly imposes a predefined set of attributes on a product and CDMA lets you define your own attribute schema, a mapping is proposed to ease recognition of CDMA extracted parts in CATIA Version 5.

The following dialog box is displayed:

CDMA\_MAPPING

Known Mappings

Server	Environment
Sutphin	CDAENV

Selected Mapping

Product Attributes

Attribute used for query

PartNumber:

Revision:

Definition:

Nomenclature:

DescriptionRef:

Model Attributes

ShapeType:

Value for Master Shape:

New Delete Save Restore Exit

No mapping should initially appear in the Known Mappings box.

2. Click on New to create your mapping.

The Context dialog box is displayed.

3. Enter in the Server field the logical name you declared using the Database Administration utility. Specify in the Environment field the name of the CDMA environment you want to access. Then enter your CDMA user name and password. Click OK.



The image shows a 'Context' dialog box with a grey background and a red border. It contains four text input fields: 'Server' with the text 'Sutphin', 'Environment' with the text 'CDAENV', 'User', and 'Password'. The 'Environment' field is highlighted with a red border. At the bottom, there are three buttons: 'OK', 'Cancel', and 'Help'.

The dialog box opposite is displayed.

4. Using the selector list, associate each Assembly attribute with a CDMA attribute.

Note that for a given Assembly attribute, a number of CDMA attributes may be proposed. The PartNumber attribute is used to perform the query operations when using the Assembly product and is the only

attribute that cannot remain unmapped whereas the other attributes can remain **\*\* NOT\_MAPPED \*\***.

The Shape Type and Value for Master Shape fields allow you to define which models will be associated with a product as a Master Representation. When a model whose CDMA attribute associated with Shape Type is assigned the value declared in the Value for Master field, this model will be associated with a product as a master representation. Master Shape is the preferential shape handled by Assembly. Otherwise, it will be considered as an Alternate Representation. Note that only CDMA attributes comparable to character strings are proposed for PartNumber and Shape Type mappings.

The dialog box is titled "Mapping Administration" and is divided into three main sections:

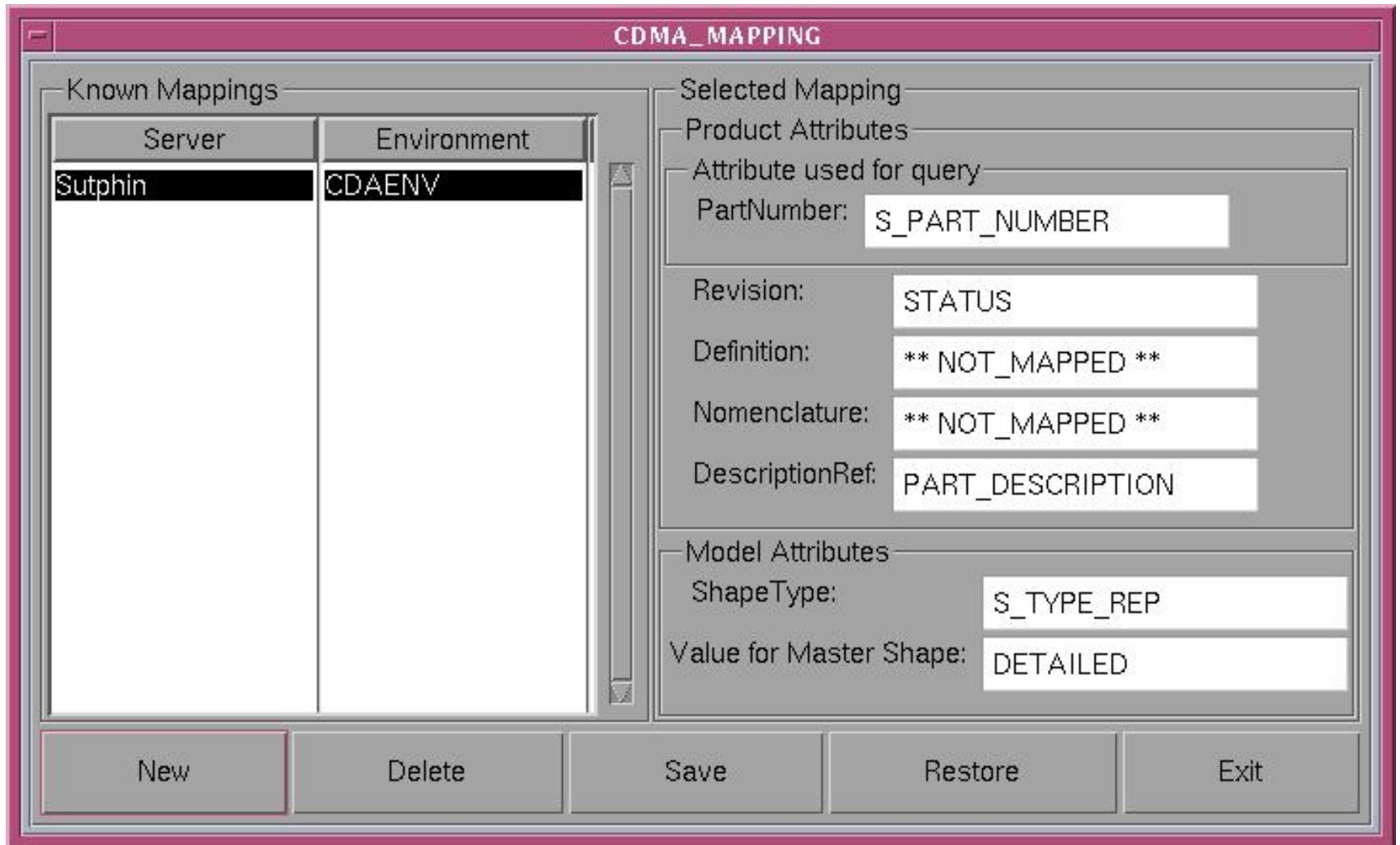
- Context:** Server: Sutphin; Environment: CDAENV.
- Product Attributes:** Attribute used for query: PartNumber (dropdown menu showing S\_PART\_NUMBER); Revision: \*\* NOT\_MAPPED \*\*; Definition: \*\* NOT\_MAPPED \*\*; Nomenclature: \*\* NOT\_MAPPED \*\*; DescriptionRef: \*\* NOT\_MAPPED \*\*.
- Model Attributes:** ShapeType: \*\* NOT\_MAPPED \*\*; Value for Master Shape: (empty text field).

Buttons: OK, Cancel, Help.

For example, in the dialog box above, all the models whose CDMA attributes S\_TYPE\_REP are assigned the DETAILED value will be considered as Master Representations.

5. Click OK to confirm your mapping declarations.

An updated Mapping Administration dialog box is displayed. A new line is added in the Known Mappings group box.



6. Click on the Save button to save your declared mapping before exiting.

7. To modify mapping, double-click on it in the Known Mappings group box. You will then be able to repeat the mapping operation.

You have just completed the second administration step which consists in declaring a mapping between CDMA and Assembly attributes.



# Adding a CDM Representation to a Product on UNIX



This task shows you how to use CDMA data stored as CATIA Version 4 model representations in a CATIA Version 5 assembly.



Have your CATIA Version 5 environment prepared to receive CDMA data and have a CATProduct document open.



1. In the CATProduct document, select the product you want to associate a CDM representation with. In the contextual menu, select the Representation->Associate CDM command.

A Connection dialog box similar to the one below is displayed.

The image shows a 'Connection' dialog box with the following fields and controls:

- Server : [Text input field]
- Environment : [Text input field]
- User : [Text input field]
- Password : [Text input field]
- OK button
- Cancel button

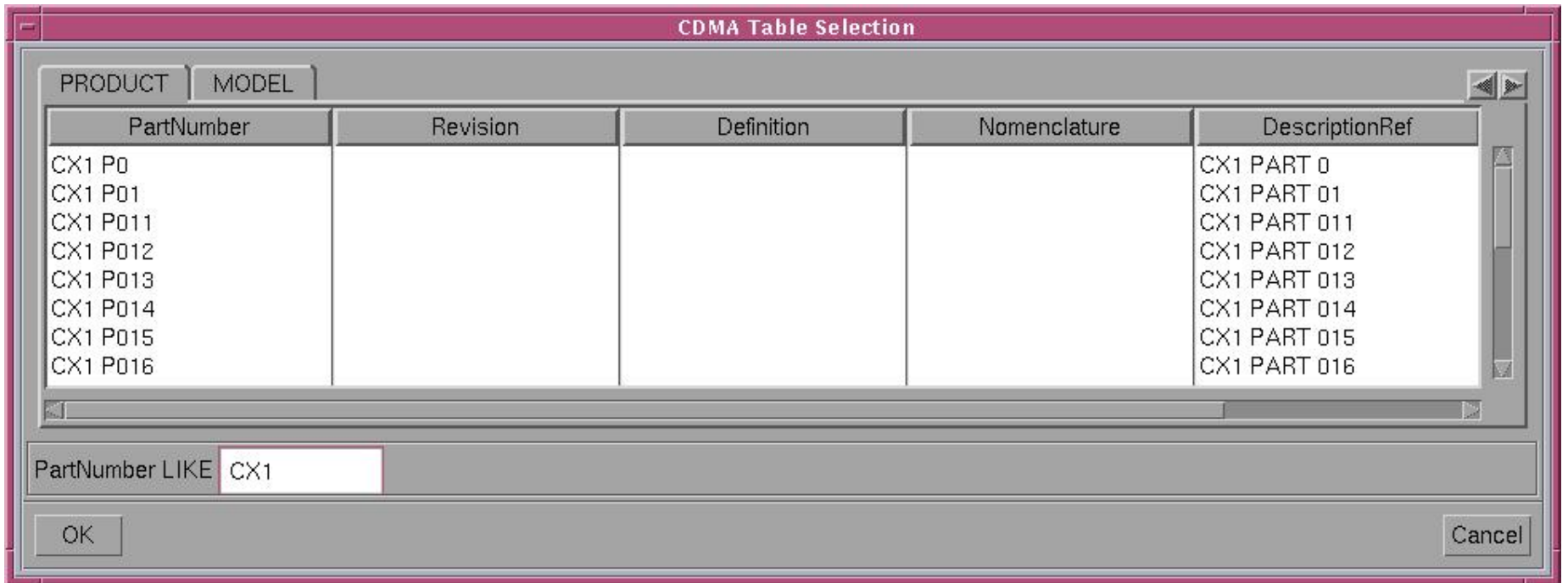
2. Complete the fields, then click OK.

Note that the string to be entered in the Server field is the logical name of your CDM database (the one declared by means of the Database Administration utility).

The CDMA Table Selection dialog box is displayed. This dialog box should initially be empty.

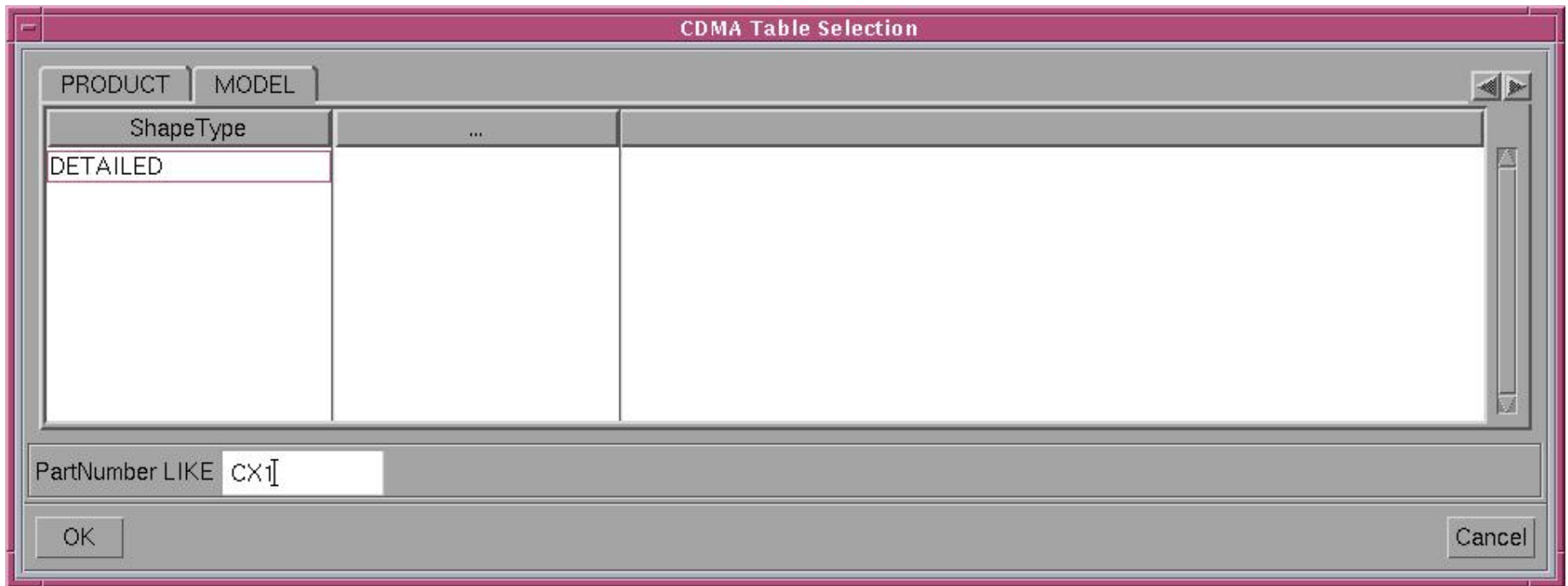
3. At this stage, you can either:

- click OK to display the list of products (which may be time-consuming if there are too many parts in the database), or
- specify a filter (CX1, for example) in the PartNumber LIKE field to display a restricted list of products and click OK.  
Any part for which the value of the CDMA attribute mapped on "PartNumber" contains that string as a substring will be displayed.



4. In the CDMA Table Selection dialog box, select a product.

A new CDMA Table Selection dialog box displays the list of the models for the selected product.





5. Select a model and click OK.


The CDMA representation is added to your product.



# Adding a CDM Product to a Product on UNIX

 This task shows you how to use CDMA assemblies as products in a CATIA Version 5 assembly.

 Have your CATIA Version 5 environment prepared to receive CDMA data and have a CATProduct document open.

 1. In the CATProduct document, select the product you want to associate a CDM product with. Right-click to select the New CDM Component command.

A Connection dialog box similar to the one opposite is displayed.

2. Complete the fields, then click OK.

Note that the string to be entered in the Server field is the logical name of your CDM database (the one declared by means of the Database Administration utility).

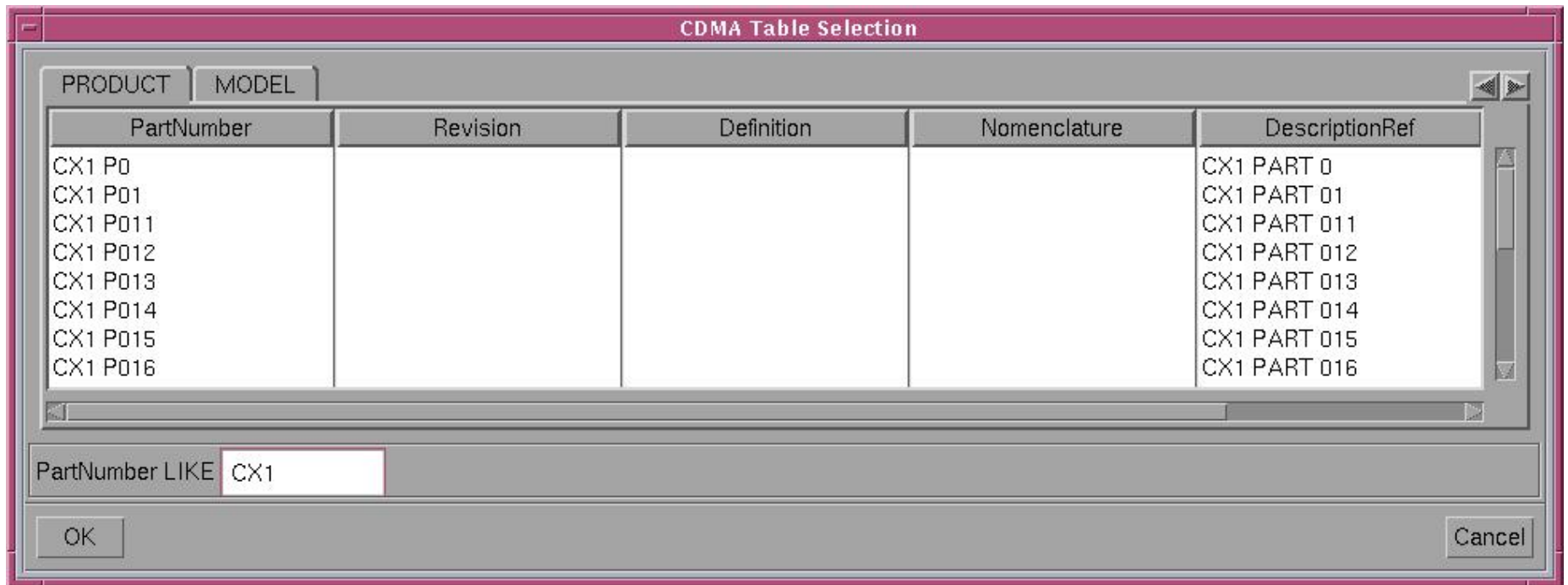


The image shows a dialog box titled "Connection". It contains four input fields: "Server :", "Environment :", "User :", and "Password :". Each field has a small vertical icon to its right. At the bottom of the dialog box, there are two buttons: "OK" and "Cancel".

The CDMA Table Selection dialog box is displayed. This dialog box should initially be empty.

3. At this stage, you can either:

- click OK to display the list of products (which may be time-consuming if there are too many parts in the database), or
- specify a filter (CX1, for example) in the PartNumber LIKE field to display a restricted list of products and click OK. Any part for which the value of the CDMA attribute mapped on "PartNumber" contains that string as a substring will be displayed.



4. In the CDMA Table Selection dialog box, select a product.

The selected product is added to the assembly structure.



# Using VPM Data in CATIA Version 5 (on UNIX Only)

Task	Purpose
<a href="#">Preparing the VPM and CATIA Environments for the Use of VPM Data in CATIA Version 5</a>	Describes the preparatory steps to be performed by the administrator enabling the building of a CATIA Version 5 product from a VPM1 PSN window (on UNIX only)
<a href="#">Building a CATIA Version 5 Product from a VPM1 PSN Window</a>	Build a CATIA Version 5 product from a VPM1 PSN window (on UNIX only)
<a href="#">Creating and Saving a CATIA Version 5 Product for Save or Commitment in VPM1.1</a>	Create and save a CATIA Version 5 Product for subsequent save or commitment in VPM1.1 (on UNIX only)



Up

[Using CDMA in V5 \(on UNIX](#)



[Using VPM Data in V5 \(on UI](#)



# **new** Preparing the VPM and CATIA Environments Before Using VPM Data in CATIA Version 5 (on UNIX Only)



This task shows you how to prepare the VPM and CATIA environments before building a CATIA Version 5 product from a VPM1.1 PSN window (see [Building a CATIA Version 5 Product from a VPM1.1 PSN Window](#)).



1. On the **VPM** side, add the following declaration parameters to the USRENV.dcls declaration file:

```
CATCDMA.METHOD_LIST(*).CATAB = 'CATIA_MODEL';  
CATCDMA.METHOD_LIST(*).TYPE = 'Reserved';  
CATCDMA.METHOD_LIST(*).COMMAND = 'CATIAV5';  
CATCDMA.METHOD_LIST(*).WHERE = 'PSN';  
CATCDMA.METHOD_LIST(*).ROLE = 'Tools';  
CATCDMA.METHOD_LIST(*).TAG = 'CdmaPrincPanelOpenCATIAV5';
```

where \* is the number just above that of the last viewer declared.

2. If you do not know the number of the last viewer declared, enter `catpath CATCDMA*.WHERE -A -l`  
This displays a list of all declared viewers.

3. Use the buttons at the bottom of the Database Access window to choose whether or not to display the models using CATIA V5, DMUNavigator and Deneb.

4. On the **CATIA V5** side, export the variables required to enable you to connect to DB2 (to be valuated in the CATEnv of the installation):

- export DB2INSTANCE\_HOME=/.../  
(name of the DB2 instance user's Home directory)
- source \$DB2INSTANCE\_HOME/sqllib/db2profile  
(DB2 shell execution)
- export LIBPATH=\$DB2INSTANCE\_HOME/sqllib/lib:\$LIBPATH  
(DB2 library path)

4. Export the variables required to enable you to connect to ORACLE (to be valuated in the CATEnv of the root):

- export ORACLE\_HOME=/.../  
(ORACLE user's home directory, where the ORACLE libs are)
- export TNS\_ADMIN=/.../  
(tnsnames.ora file's directory)
- export LIBPATH=\$ORACLE\_HOME/lib:\$LIBPATH  
(ORACLE library path: libclntsh.xxx where xxx is OS-dependent)
- export ORACLE\_NLS33=\$ORACLE\_HOME/ocommon/nls/admin/data  
(Only necessary if ORACLE Version 8 is used:  
directory containing lx....nlb type files)

If an ORACLE version lower than 8.04 is used (for example, V7.3.2 which is the version referenced by VPM1.1) a libDbuOracleStubs.xxx library must be rebuilt for V5. To enable this, a shell called oracleglue.sh is supplied in the same location as the startup shell i.e. /code/commands. Do not forget that the IRIX operating system requires the n32 libraries. Make sure that the ORACLE local client server is running and that the V5 and ORACLE variables are correctly valuated. A library called CATDbuStubsOracle is then recompiled with the correct ORACLE level.



Up





Preparing VPM and CATIA E Building a V5 Product from a Creating and Saving a V5 Pr



# Building a CATIA Version 5 Product from a VPM1.1 PSN Window (on UNIX Only)


 This task shows you how to build a CATIA Version 5 product from a VPM1.1 PSN window.

 This means in effect that CATIA Version 5 is used as an external viewer to load models stored in VPM1.1 and thus benefit from V5 functionality. For more information about how to use VPM1.1, refer to the VPM1.1 User's Guide.

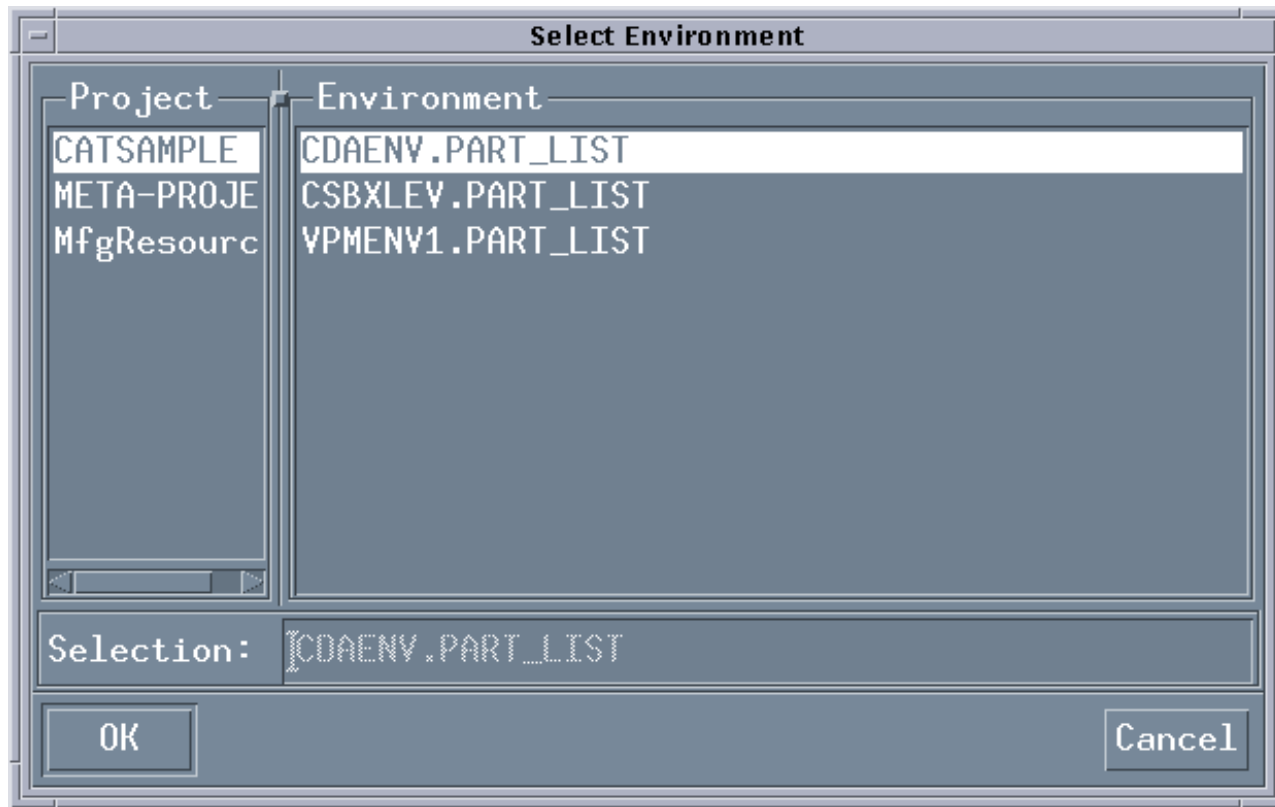
 On the CATIA Version 5 site, the support of interoperability with ENOVIA VPM1.1 requires (depending on whether the database is an IBM DB2 or an ORACLE server):

- IBM DB2 Universal Database Version 5.2 (for UNIX clients)
- ORACLE Version 8.0.4 (for AIX, HP-UX and Solaris clients) and ORACLE Version 8.0.x (BD for IRIX clients).  
If you have a lower version of ORACLE refer to the relevant CATIA installation documentation.

Your administrator should have already prepared the VPM and CATIA environments to enable this task to be performed.

 Have open the VPM1.1 main panel and a Product Structure Workbench in a CATIA V5R2 session.

1. In the VPM1.1 main panel, double-click on the Environment field. The Select Environment dialog box will appear:
2. Select the project name in the Project list and the VPM environment in the Environment list as shown below. Click OK. The name of the environment selected now appears in the Environment field in the VPM main panel.



3. In the VPM main panel, make sure that PART is selected in the Object menu.

4. In the top right-hand corner of the panel, specify a filter (CX1, for example)

in the second of the Simple query: fields.



This displays a restricted list of products in the main panel:



Environment :

CDAENV

Simple query:

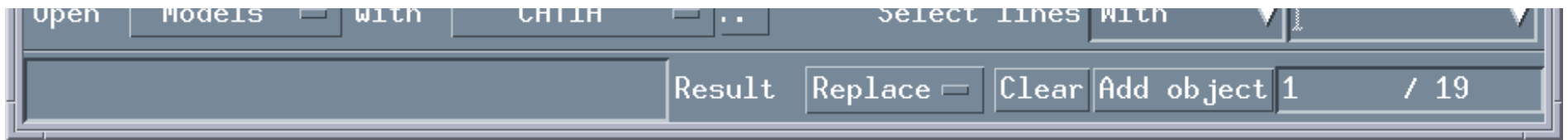
With

CX1

Search Now

 Parts

#	Part Number	Eng. Change	Description	Date
1	CX1 P0	1	CX1 PART 0	1998-04-21-16.47.02.
2	CX1 P01	1	CX1 PART 01	1998-04-21-16.47.11.
3	CX1 P011	1	CX1 PART 011	1998-04-21-16.47.16.
4	CX1 P012	1	CX1 PART 012	1998-04-21-16.47.22.
5	CX1 P013	1	CX1 PART 013	1998-04-21-16.47.26.
6	CX1 P014	1	CX1 PART 014	1998-04-21-16.47.29.
7	CX1 P015	1	CX1 PART 015	1998-04-21-16.47.32.
8	CX1 P016	1	CX1 PART 016	1998-04-21-16.47.36.
9	CX1 P0161	1	CX1 PART 0161	1998-04-21-16.47.39.
10	CX1 P0162	1	CX1 PART 0162	1998-04-21-16.47.42.
11	CX1 P017	1	CX1 PART 017	1998-04-21-16.47.46.
12	CX1 P02	1	CX1 PART 02	1998-04-21-16.47.49.
13	CX1 P021	1	CX1 PART 021	1998-04-21-16.47.52.
14	CX1 P022	1	CX1 PART 022	1998-04-21-16.47.55.
15	CX1 P03	1	CX1 PART 03	1998-04-21-16.47.59.
16	CX1 P031	1	CX1 PART 031	1998-04-21-16.48.02.
17	CX1 P032	1	CX1 PART 032	1998-04-21-16.48.05.
18	CX1 P033	1	CX1 PART 033	1998-04-21-16.48.08.
19	CX1 P04	1	CX1 PART 04	1998-04-21-16.48.11.



5. Double-click on one of the part numbers displayed, CX1 P0 for example. This displays the Product Structure Navigator (PSN) window showing the selected part number in the form of a box. Fully expand this box. In the lower left-hand corner of the window, make sure that Models and CATIA are selected as shown below:






6. Now, before being able to open any VPM data in CATIA Version 5, you must establish the connection between VPM 1.1 and CATIA V5. To do this, go into your V5 session. If you have opened the Product Structure workbench, you will see the toolbar shown opposite.





7. Click on the  icon or select the command Tools->New VPM Session. This creates a new product from the PSN with the default name DbProduct1.

The dialog box opposite appears. Enter the name of your server, your user identification and the appropriate password. Click on OK.



The No Connection icon




changes to the icon

confirming connection to the Product Structure Navigator (PSN).

The image shows a dialog box titled "Connection". It has three input fields: "Server:" with a cursor in the text box, "User:" with a password mask (dots), and "Password:" with a password mask (dots). At the bottom, there are two buttons: "OK" and "Cancel".



If the No Connection icon  is still visible this means that the connection has not yet been made. Click on this icon or select the command Tools->New VPM Session.



The icon  now appears confirming connection to the Product Structure Navigator (PSN).

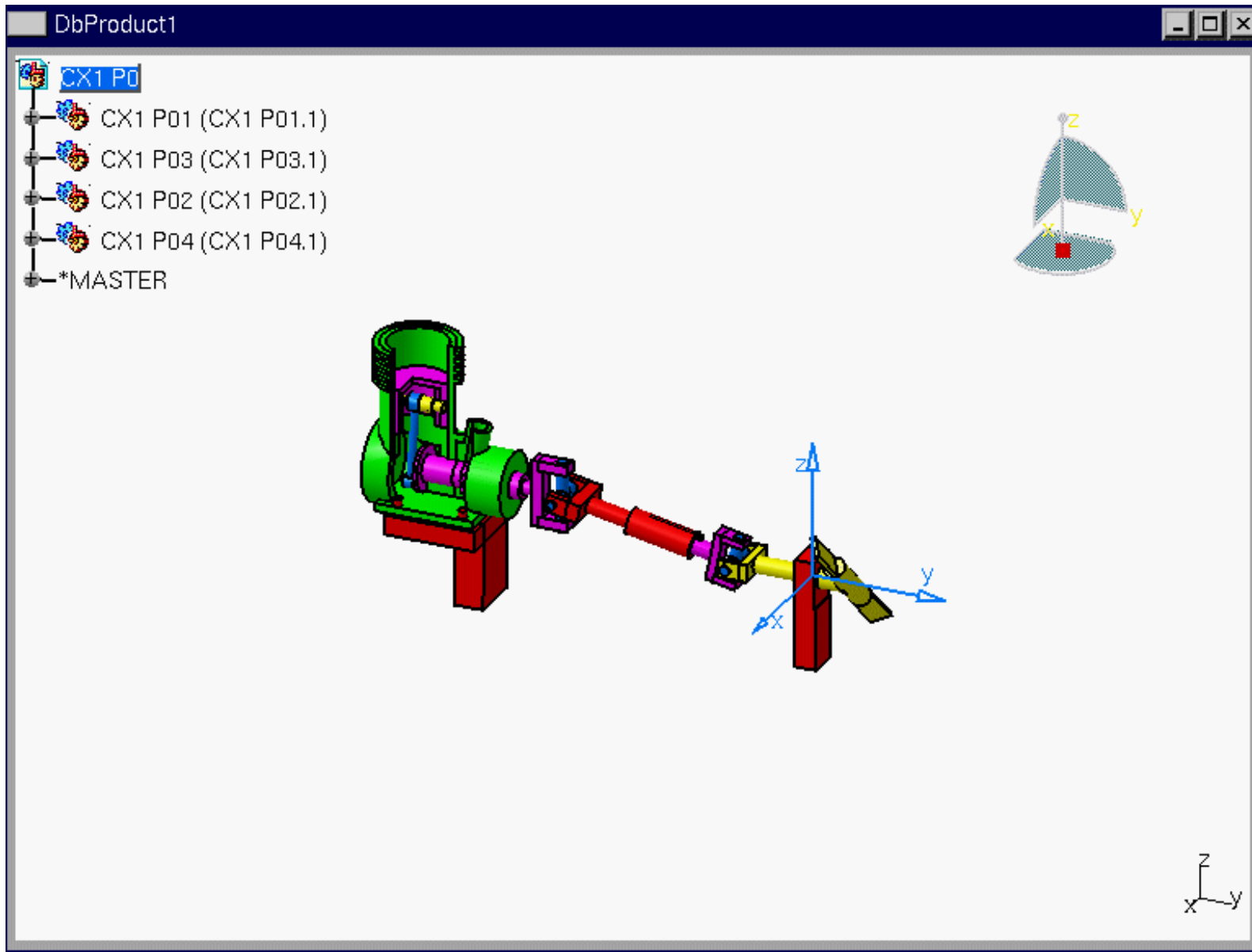
From now on, make sure that the appropriate windows in both VPM and CATIA are active each time you perform an operation.

8. In the dialog box that now appears, choose ADD to view the model in your CATIA V5 session. (Selecting REPLACE resets the current viewer.)




9. In the PSN window, click on the model you want to view in CATIA V5. The color of the model selected changes to orange. Now either click on the CATIA V5 icon in the top left-hand corner of the PSN window or right-click and select Open.

The model viewed in CATIA V5 could look like this:




Note that this image reflects operation in Design Mode (hence the \*MASTER item in the tree).

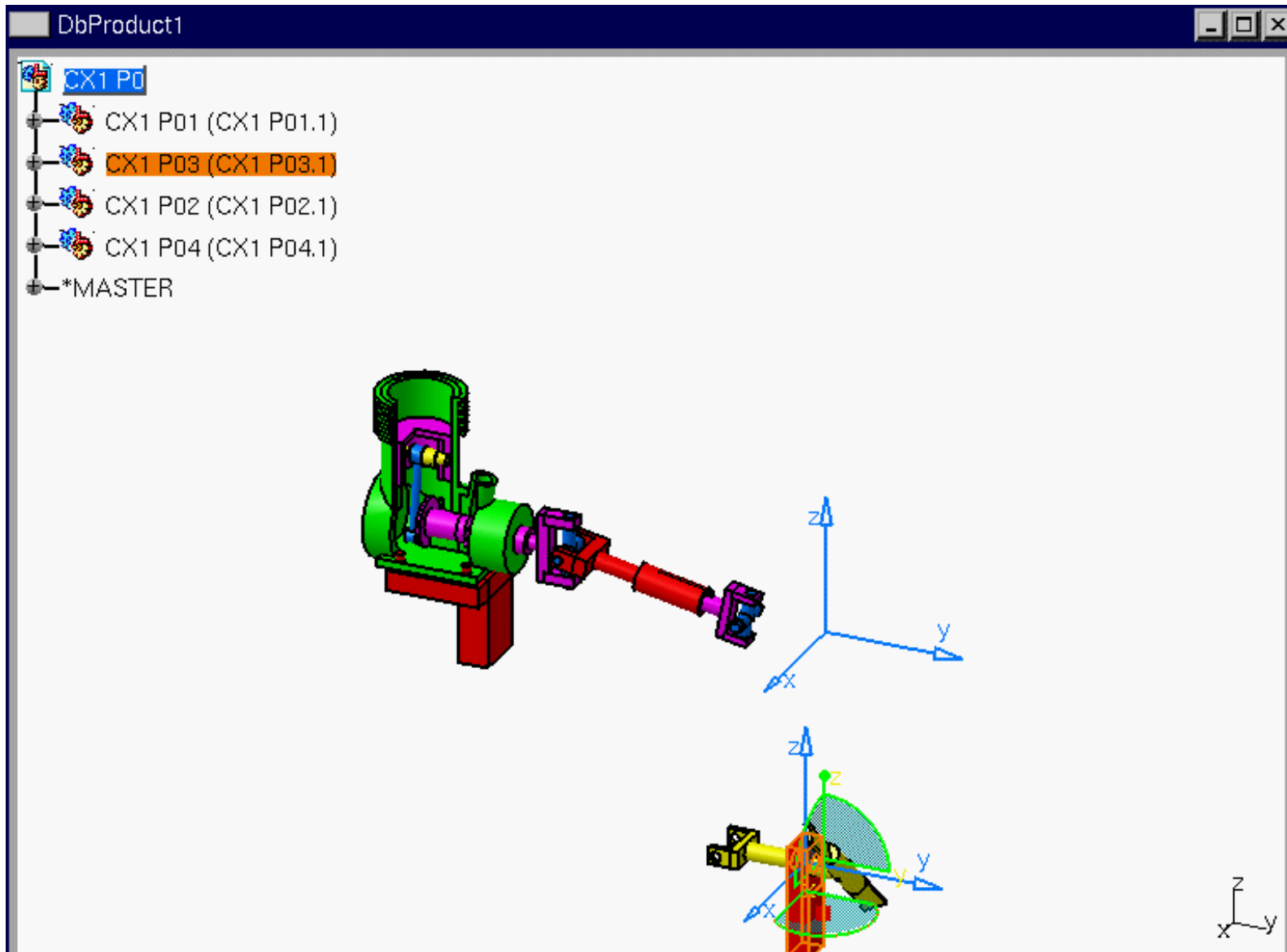


You can also open a PSN graph from the CATIA V5 session by clicking the  icon. Once in the PSN graph, you can right-click on a component to use the Cut function i.e. delete a branch of the structure displayed. (All other functions in this menu are VPM-specific and have no corresponding functions in CATIA V5.)



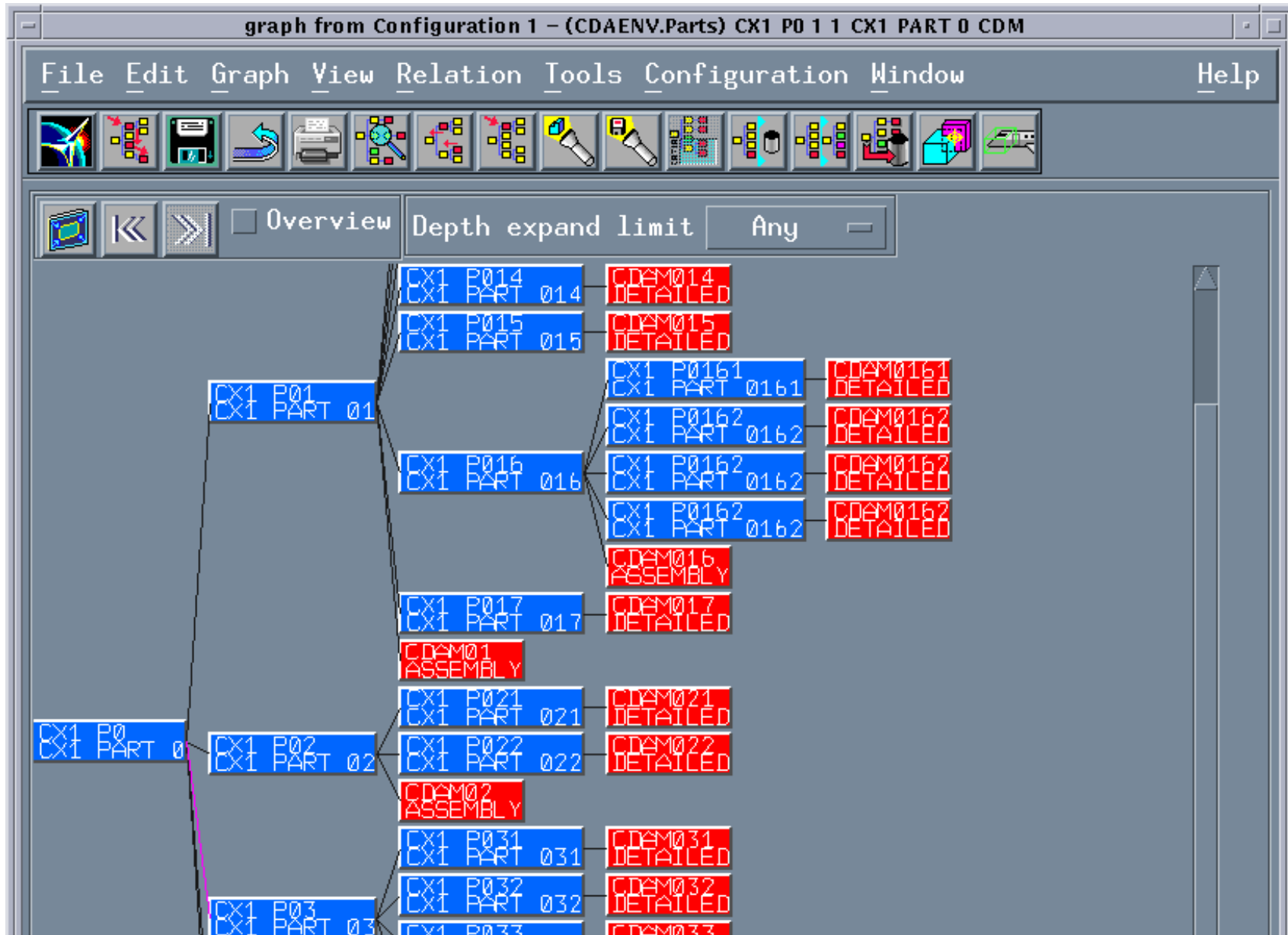
10. Select a model in the CATIA V5 session. This activates the  icon. Click on this icon to highlight the corresponding model in the active PSN graph. Alternately, if you select a model in the PSN graph and click on this icon the corresponding model in the CATIA V5 session will be highlighted. This can be customized to set systematic highlighting. See [Customizing Enovia/CDMA Settings \(on UNIX Only\)](#) in the *CATIA Version 5 Infrastructure User's Guide*.

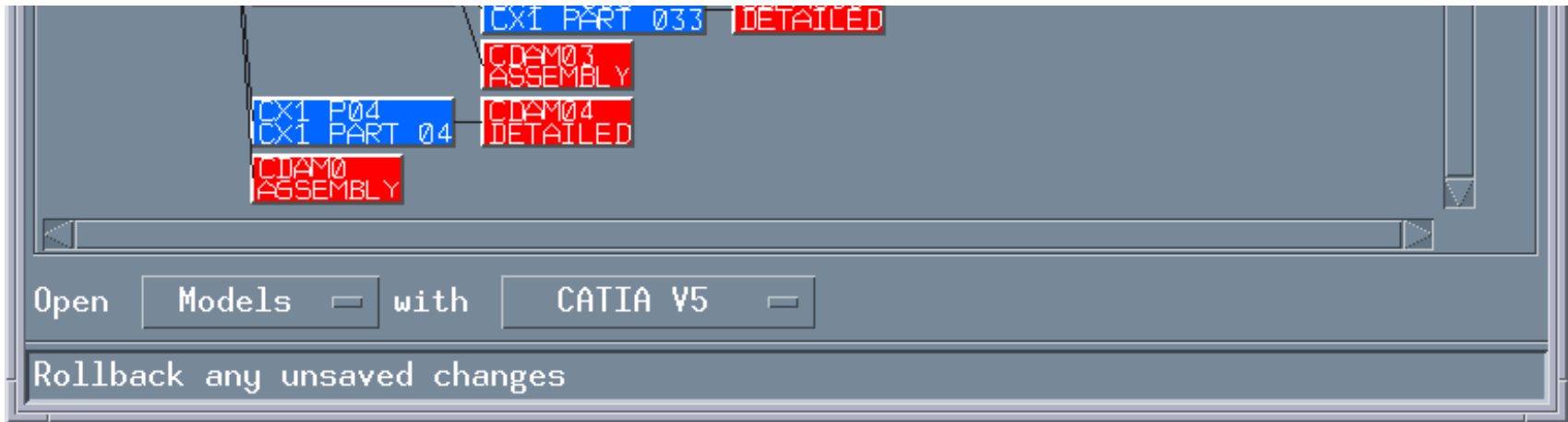
11. Perform a move on one or more components of the model displayed in the CATIA V5 session, as shown below:



Note that this image reflects operation in Design Mode (hence the \*MASTER item in the tree).

If the move is successful you will see that the PSN graph is updated accordingly i.e. the color of the corresponding link changes to purple, in this particular case between the part numbers CX1 P0 and CX1 P03.





For the move to be taken into account you must have the required write authorization (see the appropriate VPM documentation).

You can perform a commit in the database from the PSN window.



Bear in mind that the V5 model is non-persistent and that any changes made in the CATIA Version 5 session are not necessarily reflected in VPM.



# Creating and Saving a CATIA Version 5 Product for Save or Commitment in VPM1.1 (on UNIX Only)



This task shows you how to create and save a CATIA Version 5 Product for subsequent save or commitment in VPM1.1 on UNIX.

1. In the VPM main panel, make sure that DOCUMENT is selected in the Object menu.


The Create & Save dialog box appears.

2. Provide the required information about the product created putting CATIA V5 in the Source field.

3. Make sure that the document you want to save is active in the V5 session.

4. Click OK.

In the VPM Access panel you will see that a row appears for the new product created.

5. In the CATIA V5 session, you must now either select the Tools->Save in Database command to **save** the product in the database or click on the  icon to **commit** the product to the database.



**Committing** the document saves it inside another active document.

**Saving** the document saves the active document of the current document.



The document created can neither point to another document in the database nor can it be pointed to.



# Customizing Settings

## Settings

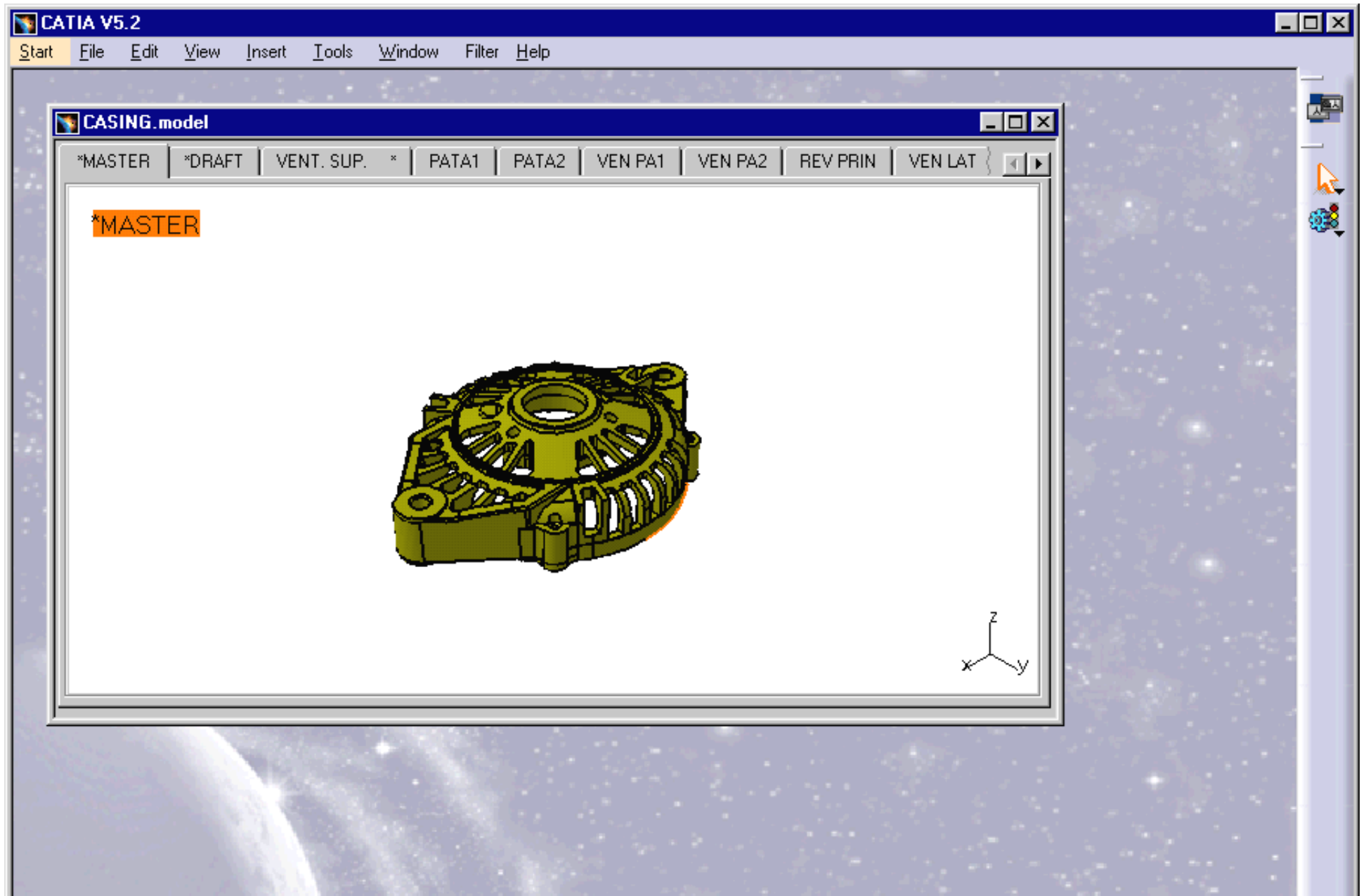
[Compatibility](#)

[IGES](#)

[Enovia/CDMA \(on UNIX only\)](#)

# Workbench Description

The V4 integration user interface looks like this:





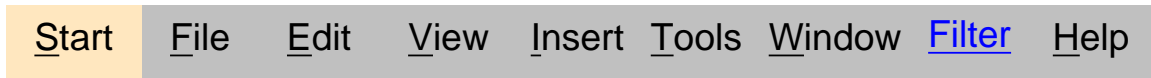
The section is organized as follows:

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# V4 Integration Menu Bar

This section presents the V4 integration menu bar available whenever you open a Version 4 model:



Note that the V4 Integration menu bar is only slightly different from the main menu bar described in the [CATIA - Infrastructure User's Guide](#).

## Filter



Filter

For...

See...



Layer Filter [Managing Layer Filters on CATIA Version 4 Models](#)



# Toolbar

The only toolbar that is specific to the V4 Integration workbench is the Geometry and Specification Check toolbar:



For more information about either of these icons, see [Checking CATIA Version 4 Model Data Before Copying It to CATIA Version 5](#)



Up



Menu Bar



Toolbars

# Glossary

## A

- active** The state when an object is the focus of user input and its operations are available.
- active object** Object currently being edited.
- active window** The window in which a user is currently working or directing input. An active window is typically at the top of the Z order and is distinguished by the color of its title bar.
- application window** Window containing the CATIA Version 5 application.
- apply** To commit a set of changes or pending transactions made in a secondary window, typically without closing that window.

## C

- cancel** To halt an operation or process and return to the state before it was invoked.
- CATIA Site Navigator** A viewer allowing you to display CATIA Version 4 data in the CATIA Version 5 environment.
- click** (v.) To position the pointer over an object and then press and release a mouse button.  
(n.) The act of clicking.
- clipboard** The area of storage for objects, data or their references after a user carries out a Cut or Copy command.
- close** To remove a window.
- character set** A character is any symbol used for the organization, control, or representation of data. A group of such symbols used to describe a particular language. Each language (or group of languages) has its **character set**
- code page** A collection of characters that make up a character set.
- code set** A **code page** (or **code set**) is a table mapping a given character (from a given character set) to a hexadecimal code position, or **code point**, for that code page. It contains the encoding values for a character set or several character sets. The encoding values in a code set provide the interface between the system and its input and output devices.
- current** State of an object when selected.

**cursor** A generic term for the visible indication of where a user's interaction will occur.

## D

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

**dialog box** A secondary window that gathers additional information from a user. A dialog box usually contains one or more controls, such as buttons, list boxes, combo boxes, and edit boxes, with which the user enters text, chooses options, or directs the action of the command.

**document** A common unit of data (typically a file) used in user tasks and exchanged between users. When saved on disk, a document is given a unique filename by which it can be retrieved.

**document window** A window that provides a primary view of a document (typically its content).

**double-click** (v.) To press and release a mouse button twice in rapid succession.

(n.) The act of double-clicking.

## E

**edit field** See **text box**.

**enter** (v.) To type a character from the keyboard.

(n.) A classification of an object based on its characteristics, behavior, and attributes.

## G

**geometry** Three-dimensional representation of the elements contained in the data. An example would be the set of faces that make up a solid. Roughly equivalent to the Brep representation used in CATIA Version 4.

**geometry area** Area of a document window in which application data are displayed and edited.

## H

**hold down** To continue pressing a keyboard key, or mouse button.

## I

<b>icon</b>	A pictorial representation of an object.
<b>inactive</b>	The state of an object which it is not the focus of a user's input.
<b>inactive window</b>	A window in which a user's input is not currently being directed. An inactive window is typically distinguished by the color of its title bar.
<b>insertion point</b>	The location where text or graphics will be inserted (also referred to as the caret). Also used for text box controls to indicate input focus.
<b>interoperability</b>	<ul style="list-style-type: none"><li>● Ability to exchange data between CATIA Version 4 and CATIA Version 5.</li><li>● Ability to exchange CATIA Version 5 data between CATIA Version 5 workshops</li><li>● Ability to exchange data between CATIA Version 5 and OLE-compliant applications.</li></ul>

## L

<b>link</b>	(v.) To form a connection between two objects. (n)) A reference to an object that is linked to another object.
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## M

<b>menu</b>	A list of textual or graphical choices from which a user can choose.
<b>menu bar</b>	A horizontal bar at the top of a window, below the title bar, that contains menus.
<b>menu button</b>	A command button that displays a menu.
<b>menu item</b>	A choice on a menu.
<b>menu title</b>	A text or graphic label that designates a particular menu. For drop-down menus, the titles is the entry in the menu bar; for cascading menus the menu title is the name of its parent menu item.
<b>message box</b>	A secondary window that is displayed to inform a user about a particular condition.
<b>mode</b>	A particular state of interactions, often exclusive in some way to other forms of interactions.
<b>model</b>	CATIA Version 4 model.
<b>model document</b>	Document containing a CATIA Version 4 model.

**mouse** A commonly used input device that has one or more buttons used to interact with a computer. It is also used as a generic term to include other pointing devices that operate similarly (for example, trackballs and headpointers).

## O

**object** An entity or component identifiable by a user that can be distinguished by its properties, operations, and relationships.

**operation** A generic term that refers to the actions that can be done to or with an object.

**option button** A standard Windows control that allows a user to select from a fixed set of mutually exclusive choices (also referred to as a radio button).

## P

**point** (v.) To position the pointer over a particular object an location.  
(n.) A unit of measurement for type (1 point equals approximately 1/72 inch).

**pointer** A graphic image displayed on the screen that indicates the location of a pointing devices (also referred to as a cursor).

**press** To press and release a keyboard key.

**PROJECT file** A CATIA Version 4 entity containing project standards. This is the heart of the CATIA Version 4 site without which no significant operations can be carried out. The "building blocks" of a project are stored in the form of tables, each of which corresponds to a member of the PROJECT file.

**property** Attribute or characteristic of an object that define its state, appearance, or value.

## R

**right-click** Click using the right mouse button (to display contextual menu).

## S

**scroll** To move the view of an object or information to make a different portion visible.

**scroll bar** A standard Windows control that supports scrolling.

**select** To identify one or more objects upon which an operation can be performed.

**selection** An object or set of objects hat have been selected.

<b>settings</b>	Set of setup parameters and user preferences stored in non-editable files.
<b>specifications</b>	Made up of the entire history of the actions performed to obtain the data. Shown in the form of a tree and roughly equivalent to the CSG tree used in CATIA Version 4.
<b>specification tree</b>	Area of the document window reserved for viewing the design specifications of a part, presented in the form of a tree structure.
<b>status bar</b>	An area that allows the display of state information of the information being viewed in the window, typically places at the bottom of a window.
<b>stop</b>	To halt a process or actions, typically without restoring the state before the process began.

## T

<b>text box</b>	A standard Windows control in which a user can enter an edit text (also referred to as the edit field).
<b>title bar</b>	The horizontal area at the top of a window that identifies the window. The title bar also acts as a handle for dragging the window.
<b>toolbar</b>	A frame or special area that contains a set of other controls.

## U

<b>unavailable</b>	The state of a control or data whose normal functionality is not presently available to a user (also referred to as dimmed).
<b>undo</b>	To reverse one operation performed on an object.

## W

<b>window</b>	A standard Windows object that displays information. A window is a separately controllable area of the screen that typically has a rectangular border.
<b>workbench</b>	Set of tools for completing specific tasks. Each type of document can be edited with a document-specific set of tools.

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