

# CATIA Infrastructure User Guide

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ENOVIAVPM / CATIA V5 Knowledgeware Integration

ENOVIAVPM / CATIA V5 Knowledgeware Integration

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# Knowledgeware Applications Interoperability



In this section, you will find information about interoperability between *CATIA* Knowledgeware Applications and other applications listed below:

[ENOVIA LCA Interoperability](#)  
[ENOVIAvpm Interoperability](#)



# ENOVIA LCA Interoperability





# Working with Design Tables in ENOVIA LCA Using VPM Navigator

It is possible to save design tables in ENOVIA LCA and to apply design tables saved in ENOVIA LCA to existing parts using VPM Navigator.

Note that:

- To modify a design table saved in ENOVIA, the sheet must be synchronized with the design table saved in ENOVIA LCA. To do so, right-click the design table in CATIA and select the **Synchronize (ENOVIA)** command. The design table should also be checked out/locked in ENOVIA.
- To share a design table, see [Saving a Design Table in ENOVIA LCA Using the VPM Navigator](#)

# Saving a Design Table in ENOVIA LCA Using the VPM Navigator



This task is designed to show the user how to save a design table and its external file in ENOVIA LCA by using the VPM Navigator and how to use the external file in another file (it is possible to share a .xls file with other design tables). This scenario is divided into the following steps:

- In CATIA, the user creates the design table using a pre-existing file.
- He saves the file in ENOVIA LCA.
- He creates another part in CATIA, saves it and associates the design table external file stored in ENOVIA LCA.




To know more about the VPM Navigator, see the *VPM Navigator User's Guide*.



Prior to carrying out your scenario, enable the **Enovia** option in the **Tools->Options->General->Document** tab.



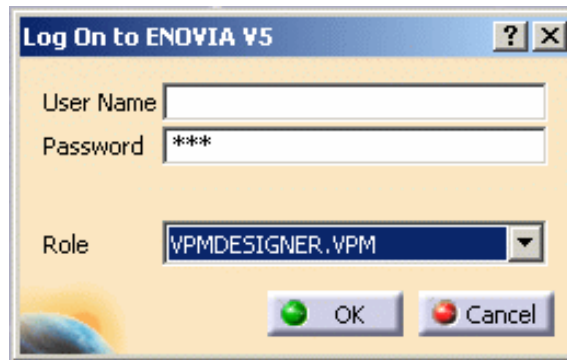
## Creating the design table

1. Open the [BallBearing.CATPart](#) file.
2. Click the **Design Table** icon () in the Knowledge tool bar. The **Creation of a Design Table** window displays.
3. Click the **Create a design table from a pre-existing file** option and click **OK**. The File Selection dialog box opens.
4. Select the [BearingDesignTable.xls](#) file and click **Open**. Click **Yes** when asked if you want to associate the columns of the table with the parameters.
5. Click **OK** to apply the default configuration. DesignTable.1 displays below the Relations node.

## Saving the files in ENOVIA LCA

6. Save your files in ENOVIA LCA. To do so, proceed as follows:
  - Choose the **Start -> Infrastructure -> VPM Navigator** menu item.

- o Click the **Connect** icon (  ) in the **ENOVIA V5** toolbar. The Log On to dialog box displays:




- o Click **OK** when done.

- o Click the **Save data in Enovia V5** icon (  ). The Save in Enovia V5 dialog box displays. Click OK to save the files. Close the file in *CATIA*.

The BearingDesignTable.xls file and the BallBearing.CATPart file are now both saved in ENOVIA LCA.

## Applying the Design Table file saved in ENOVIA LCA to another part

7. Open the [New\\_BallBearing.CATPart](#) file. Click the **Save data in Enovia V5** icon (  ). The Save in Enovia V5 dialog box displays. Click **OK** to save the file.

8. Click the Design Table icon (  ) to associate a design table to the part.

7. Click the **Create a design table from a pre-existing file** option and click **OK**. The File Selection window displays as well as the Browser (see picture opposite).

8. Click **Cancel** in the File Selection window and click the Enovia button in the browser.



7. The Search Conditions window displays.
8. In the **Creator** field, enter the creator's name and click **Search**. The results display in the Search Result window.
9. Select the BearingDesignTable.xls file and click **Open**.
10. Click **Yes** when asked if you want to associate the columns of the table with the parameters.

11. Click **OK** to apply the default configuration. DesignTable.1 displays below the Relations node.



# ENOVIAvpm Interoperability

# ENOVIAvpm / CATIA V5 Knowledgeware Integration



Note that for the Knowledgeware integration to work properly, VPM1.5 PTF13 is required.

The VPM integration of Knowledgeware information created at the product level is guaranteed for Knowledge Advisor, Product Engineering Optimizer and for the infrastructure functionalities (parameters, formulas, laws, Design Tables).

The following Knowledgeware objects are supported:

- Parameters sets
- Formulas, checks, rules, sets of equations
- Laws
- Design Tables (The link between design tables and their external files is not supported. The file should be located in a shared directory).
- Macros with arguments, Knowledge Advisor actions
- Reactions
- Equivalent Dimensions
- Comments and URLs
- Optimizations, Design of Experiments, Constraints Satisfaction.

These objects, considered as connections, support the following pointings:

- To Part parameters and objects
- To published parameters
- To Product users parameters
- To User properties of a product
- To assembly constraints parameters
- To Knowledge objects at the Product level

Creating and Storing a Knowledgeware Relation  
Useful Tips

# Creating and Storing a Knowledgeware Relation



In the scenario described below, we assume that the user starts from scratch. The scenario is divided into the following steps:

- Creating documents in VPM
- Creating the Assembly in VPM
- Applying constraints to the assembly in Catia
- Creating a formula in Catia
- Loading the Assembly in Catia



To carry out this scenario, you will need the following files:

[TowHook.CATProduct](#)  
[M39.CATPart](#)  
[Axis\\_Step3.CATPart](#)  
[Support.CATPart](#)  
[Liner\\_Step3.CATPart](#)



In Catia, from the **Tools->Options** menu, select **Infrastructure->Product Structure** and make sure that the **Work with the cache system** option is unchecked.



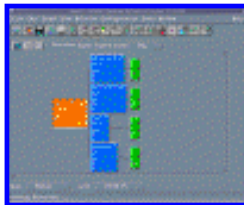
## Creating Documents in VPM

- 1.** In VPM, select the **Object->Document** command to specify the type of file that you want to create.
- 2.** Select the **File->Create & Save** command. The **Create&Save** window displays.
  - In the **Part Number** field, enter a string of characters. Note that this string will be visible in Catia when loading the part: M39 for example.
  - In the **Eng Change** field, enter 1 (version number).
  - In the **Type** field, enter Part.
  - Click the **Source** field and select the [M39.CATPart](#) file.
  - Enter 1 in the remaining fields.
  - Click **Ok** when done. The document is now created in VPM.
- 3.** **Double-click the Document to open the PSN editor.**
- 4.** Right-click the sheet corresponding to the created document and select **Open**. The Part opens in *CATIA*.
- 5.** Repeat the 4 steps described above for the following parts:

- [Axis\\_Step3.CATPart](#)
- [Support.CATPart](#)
- [Liner\\_Step3.CATPart](#)

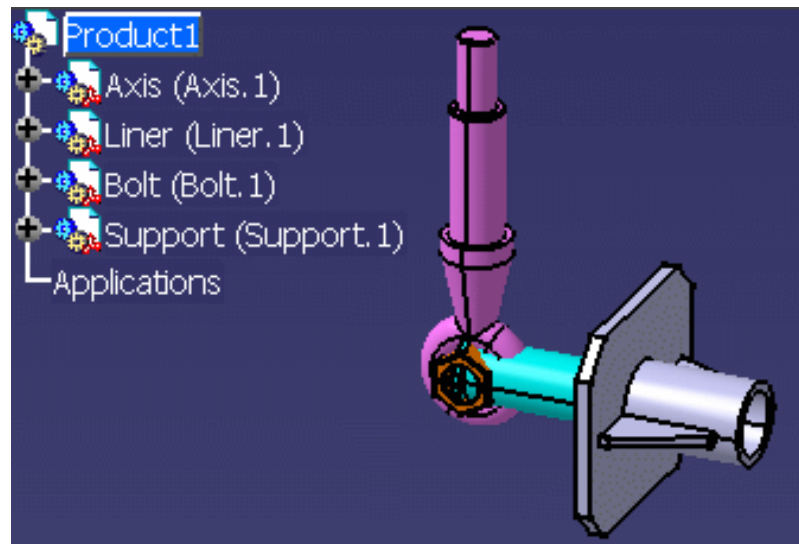
## Creating the Assembly

- 6.** In VPM, select the **Object->Part** command to specify the type of file that you want to create.
- 7.** Select the **File->Create & Save** command. The **Create&Save** window displays.
  - In the **Part Number** field, enter TowHook.
  - In the **Eng Change** field, enter 1 (version number).
  - In the **Type** field, enter **Product**.
  - Click **OK** when done and **Yes** to confirm.
- 8.** Specify that the TowHook is made up of M39.CATPart, Axis\_Step3.CATPart, Support.CATPart, and Liner\_Step3.CATPart. To do so, proceed as follows:
  - Use the shift button to select the 4 documents listed above.
  - Press the Ctrl+C keys.
  - Click the TowHook file.
  - Press the Ctrl+V keys. The assembly is created.
- 9.** Double-click TowHook in VPM. The editor displays.
- 10.** In the PSN editor, expand the documents and, holding the shift key, right-click the 4 documents and select the **Open** command.



- 11.** Select **Add** when asked if you want to replace or add the models in your current viewer session. The assembly displays in Catia.



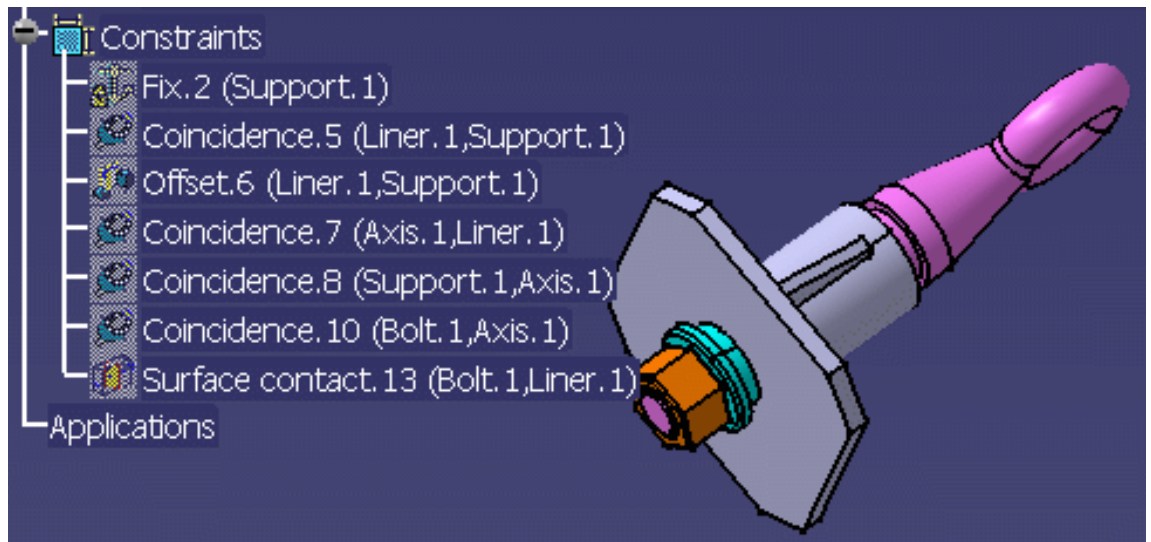


## Applying constraints to the assembly in Catia

**12.** Apply constraints to build the assembly. To do so, proceed as follows:

- From the **Start** menu, select the **Mechanical Design->Assembly Design** workbench.
- Apply a fix constraint to the Support.
- Apply a coincidence constraint between Liner.1 and Support.1.
- Apply an offset constraint (10 mm) between Liner.1 and Support.1.
- Apply a coincidence constraint between Axis.1 and Liner.1.
- Apply a coincidence constraint between Support.1 and Axis.1.
- Apply a coincidence constraint between Bolt.1 and Axis.1.
- Apply a contact constraint between Bolt.1 and Liner.1.
- Do not hesitate to use the Manipulations parameters tool available in the Assembly workbench to move the assembly components before applying the constraints.

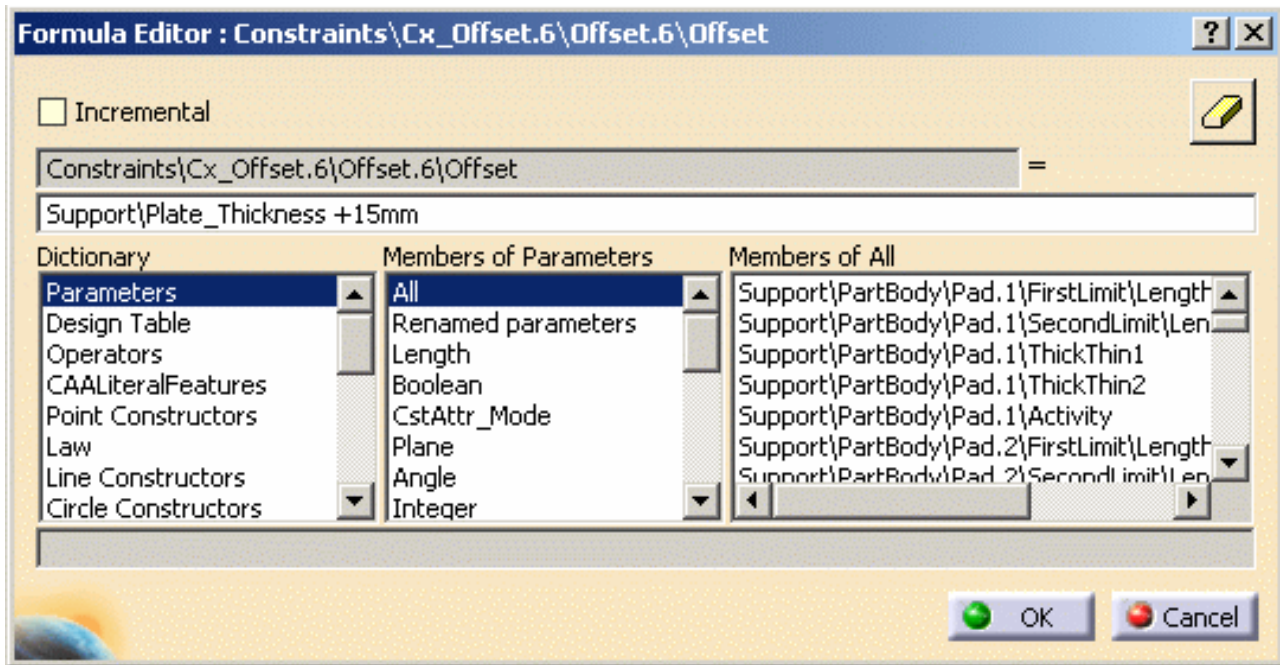
The constraints are created (see picture opposite.)



### Creating a formula

**13.** Create a formula applied on the Offset constraint. To do so, proceed as follows:

- 
- Click the f(x) icon. The Formula Editor displays.
- In the specification tree, select the offset constraint and click the Add formula button.
- In the editor, double-click Support\Plate\_Thickness and add "+ 15mm". (See graphic below).

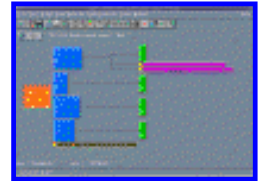


**14.** Click **OK** twice. The formula is created.

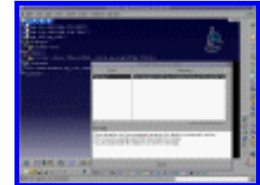
**15.** Save your .CATProduct file. Your file is saved in VPM.

## Loading the Assembly in Catia

16. Go back to VPM and right-click M39.CATPart, Axis\_Step3.CATPart, TowHook, and Liner\_Step3.CATPart and select the **Open** command.

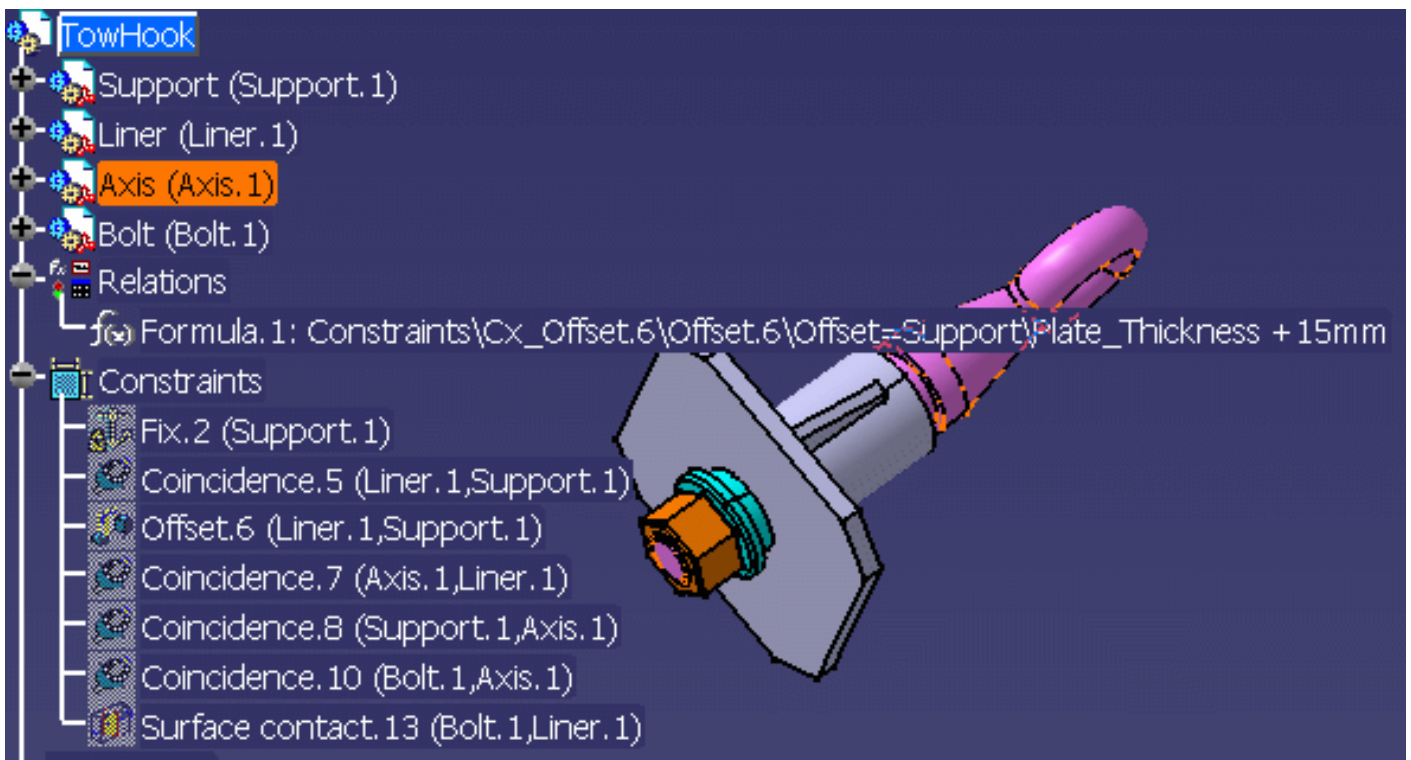


- A message is fired by Knowledge indicating that the relation cannot be solved.
- The formula displays with an interrogation mark.
- The formula cannot be edited.
- The formula can be deactivated.
- Some parameters are seen as "unfound".



The constraints applied to the Support document do not display.

17. Go back to PSN, right-click the Support document and select **Add**: The formula applies correctly and the constraints all display.



# Useful Tips

## Replacing a document publishing parameters

In VPM 4, if you want to replace a document publishing parameters with another parameter, do it in *CATIA*: Before replacing the assembly, send it to *CATIA* as well as the document that will replace the existing one, perform the "replace" operation in *CATIA* and save it.

Replace operations performed from VPM are supported with one restriction: In assemblies, you can only replace part instances directly located below the root product.

# Workbench Description

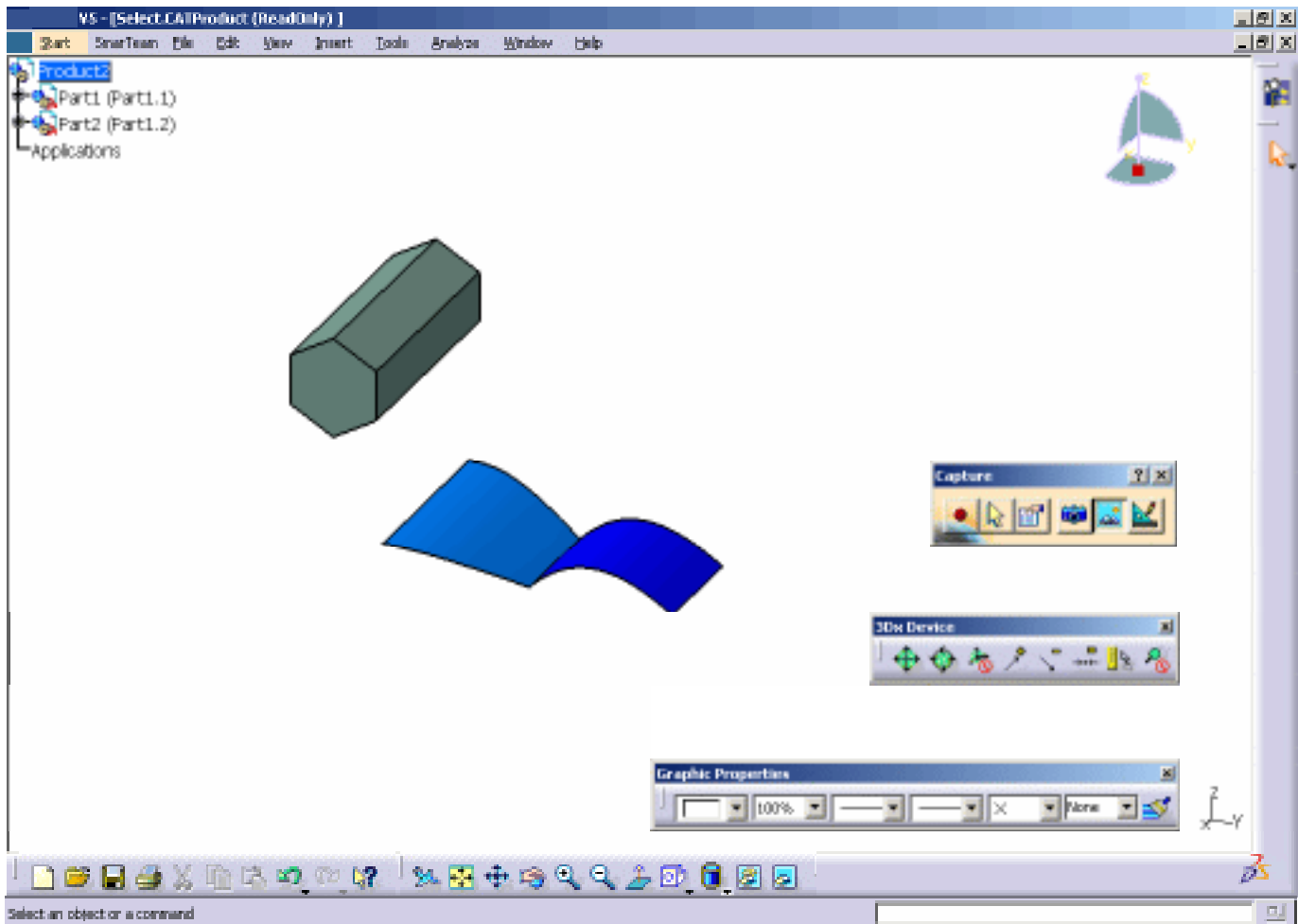
Version 5 lets you create and manage parts, assemblies, drawings and CATIA Version 4 models in **documents**.

Version 5 uses a multiple document interface (MDI), meaning that you can manipulate several documents at the same time. You can even view the contents of the same document in several windows at the same time.

Opening a specific type of document activates the **workbench** and the associated **workbench toolbar** containing all the tools you need to edit the document, so the Version 5 application window looks a little different depending on what type of document you are editing. The same applies to the contents of the menu bar and the commands on pulldown menus.

This section contains the description of the icons and menus specific to the CATIA Object Manager workbenches. These commands are discussed in greater details elsewhere in the guide.

The main application window looks like this (click the sensitive areas to see the related documentation):



The section is organized as follows:

- Menu Bar
- Standard Toolbar
- View Toolbar
- Capture Toolbar
- Graphic Properties Toolbar
- 3Dx Device Toolbar
- Desk Toolbars
- Symbols
- Viewing and Hiding Toolbars
- Viewing the Commands List
- Keyboard Shortcuts
- Knowledgeware Toolbar
- Feature Dictionary Editor Toolbar

# Infrastructure Menu Bar

This section presents the main menu bar available when you run the application and before creating or opening a document.



## Start

The Start menu is a navigation tool intended to help you toggle between different workshops. The contents of the Start menu vary according to the configurations and/or products installed. For more information about the Start menu, refer to ["Accessing the Navigation Tools"](#).

## File

File	For...	See...
New... Ctrl+N	New...	<a href="#">Creating New Documents</a>
New from...	New from...	<a href="#">Creating a New Document from an Existing One</a>
Open... Ctrl+O	Open...	<a href="#">Opening Existing Documents</a>
Close	Close	<a href="#">Closing Documents</a>
Save	Save	<a href="#">Saving Existing Documents</a>
Save As...	Save As	<a href="#">Saving Documents For the First Time or Under Another Name</a>
Save All	Save All	<a href="#">Saving Documents In Other Formats</a>
Save Management...	Save All	<a href="#">Saving All Documents</a>
Print... Ctrl+P	Save Management	<a href="#">Managing Document Save</a>
Printer Setup...	Print...	<a href="#">Customizing Print Settings Before Printing Your Documents</a>
Desk...	Printer Setup...	<a href="#">Setting Up Your Printers on UNIX and Windows</a>
Send To	Desk...	<a href="#">Using the FileDesk Workbench</a>
Exit	Send To	<a href="#">Transferring Version 5 Data</a>

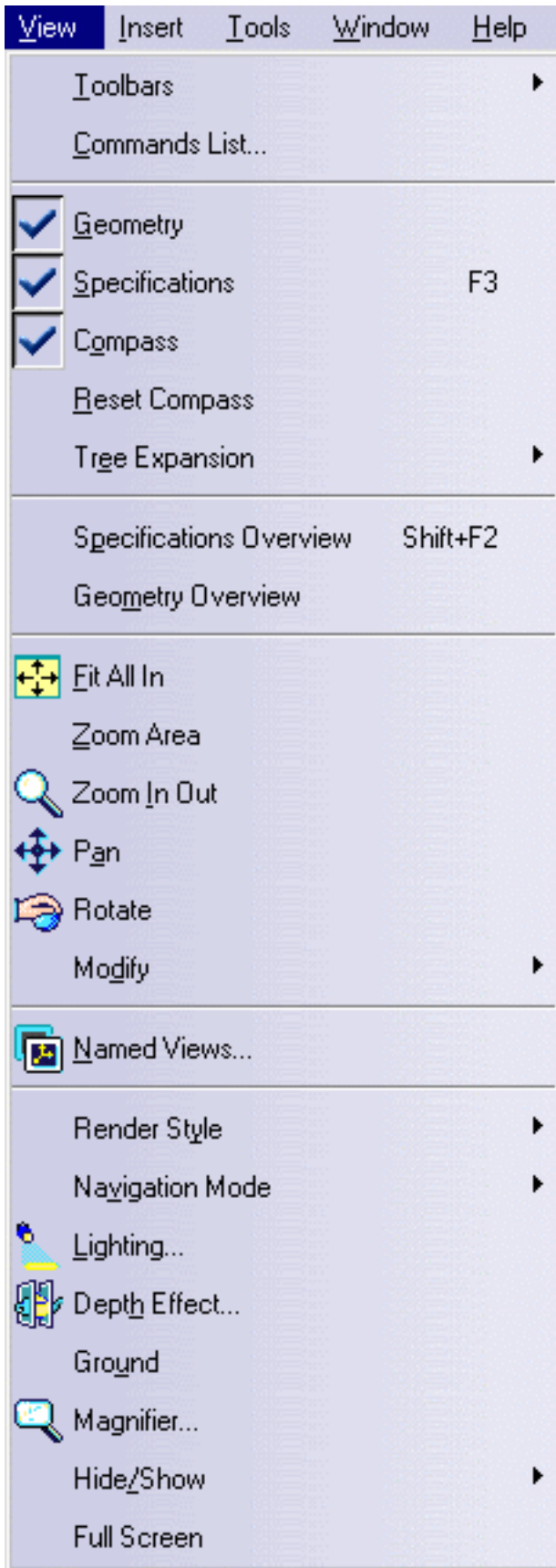
# Edit

<b>Edit</b>	<b>View</b>	<b>Insert</b>	<b>Tools</b>	<b>Analyze</b>	<b>For...</b>	<b>See...</b>
 <b>U</b> ndo Select element					<b>Undo</b>	<a href="#">Undoing Actions</a>
 <b>R</b> epeat					<b>Redo/Repeat</b>	<a href="#">Redoing and Repeating Actions</a>
 <b>C</b> ut					<b>Cut</b>	<a href="#">Cutting and Pasting Objects</a>
 <b>C</b> opy					<b>Copy</b>	<a href="#">Copying and Pasting Objects</a>
 <b>P</b> aste					<b>Paste</b>	<a href="#">Cutting and Pasting Objects</a> <a href="#">Copying and Pasting Objects</a>
Paste <b>S</b> pecial...					<b>Paste Special...</b>	<a href="#">Using the Paste Special... command</a>
<b>D</b> elete					<b>Delete</b>	<a href="#">Deleting Objects</a>
 <b>S</b> earch...					<b>Search...</b>	<a href="#">Selecting Using the Search... Command</a>
 <b>S</b> election Sets...					<b>Selection Sets...</b>	<a href="#">Storing Selections Using Selection Sets</a>
 <b>S</b> election Sets Edition...					<b>Selection Sets Edition...</b>	<a href="#">Editing Selection Sets</a>
 <b>F</b> ind Owing Selection Sets...					<b>Find Owing Selection Sets...</b>	<a href="#">Storing Selections Using Selection Sets</a>
 <b>L</b> inks...					<b>Links...</b>	<a href="#">Editing Document Links</a>
 <b>P</b> roperties					<b>Properties...</b>	<a href="#">Displaying and Editing Graphic Properties</a>
<b>O</b> ther Selection...					<b>Other Selection...</b>	<a href="#">Selecting Using the Other Selections... Command</a>
<b>C</b> omponents						
<b>R</b> epresentations						
<b>P</b> ad.1 object						

# View

<b>For...</b>	<b>See...</b>
<b>Toolbars</b>	<a href="#">Viewing and Hiding Toolbars</a>
<b>Commands List...</b>	<a href="#">Viewing the Commands List</a>
<b>Geometry</b>	<a href="#">Setting Document Window Layout Preferences</a>





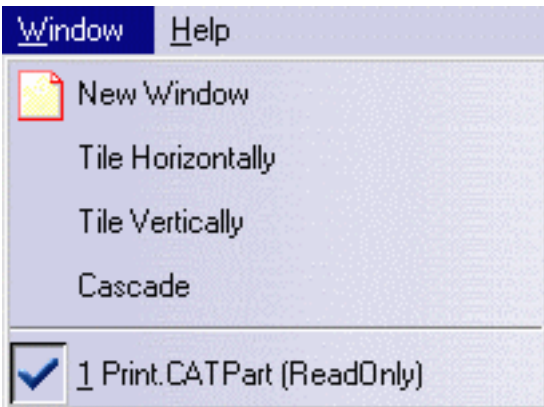
<b>Specifications</b>	<a href="#">Setting Document Window Layout Preferences</a>
<b>Compass</b>	<a href="#">About the 3D Compass</a>
<b>Reset Compass</b>	<a href="#">Manipulating Objects Using the Mouse and Compass</a>
<b>Tree Expansion</b>	<a href="#">Expanding and Collapsing the Spec Tree</a>
<b>Specifications Overview</b>	<a href="#">Using the Overview on the Specification Tree</a>
<b>Geometry Overview</b>	<a href="#">Using the Geometry Overview</a>
<b>Fit All In</b>	<a href="#">Fitting All Geometry in the Geometry Area</a>
<b>Zoom Area</b>	<a href="#">Zooming In On An Area</a>
<b>Zoom In Out</b>	<a href="#">Zooming In</a>
	<a href="#">Zooming Out</a>
<b>Pan</b>	<a href="#">Panning</a>
<b>Rotate</b>	<a href="#">Rotating</a>
<b>Modify-&gt;Look At</b>	<a href="#">Looking At Objects</a>
<b>Modify-&gt;Turn Head, Zoom In, Zoom Out, Normal View</b>	<a href="#">Turning Your Head To View An Object</a>
	<a href="#">Zooming In</a>
	<a href="#">Zooming Out</a>
	<a href="#">Viewing Along a Normal to a Plane</a>
<b>Named Views...</b>	<a href="#">Using Standard and User-Defined Views</a>
<b>Render Style</b>	<a href="#">Using Rendering Styles</a>
<b>Navigation Mode</b>	<a href="#">Navigating</a>
<b>Lighting...</b>	<a href="#">Setting Lighting Effects</a>
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<b>Ground</b>	<a href="#">Viewing Objects against the Ground</a>
<b>Magnifier...</b>	<a href="#">Magnifying</a>
<b>Hide/Show</b>	<a href="#">Hiding and Showing Objects</a>
<b>Full Screen</b>	<a href="#">Using the Full Screen</a>

## Tools



<b>For...</b>	<b>See...</b>
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<b>Conferencing</b>	<a href="#">Conferencing</a>

## Window



<b>For...</b>	<b>See...</b>
<b>New Window</b>	<a href="#">Using Document Windows</a>
<b>Tile Horizontally</b>	<a href="#">Using Document Windows</a>
<b>Tile Vertically</b>	<a href="#">Using Document Windows</a>
<b>Cascade</b>	<a href="#">Using Document Windows</a>

## Help

<b>For...</b>	<b>See...</b>
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<b>CATIA User Companion</b>	
<b>Contents, Index and Search</b>	<a href="#">Accessing the Online Help Library</a>
<b>What's This?</b>	<a href="#">Using the What's This? Command</a>



**User Galaxy**




[Accessing the Dassault Systèmes User Galaxy](#)

**About CATIA V5**

[Displaying Copyright Information](#)

# Standard Toolbar



-  See [Creating New Documents](#)
-  See [Opening Existing Documents](#)
-  See [Saving Existing Documents](#)
-  See [Printing a Document Quickly without Customizing Print Settings](#)
-  See [Cutting and Pasting Objects](#)
-  See [Copying and Pasting Objects](#)
-  See [Cutting and Pasting Objects](#), and [Copying and Pasting Objects](#)
-  See [Undoing Actions](#)
-  See [Undoing Actions](#)
-  See [Redoing and Repeating Actions](#)
-  See [Redoing and Repeating Actions](#)
-  See [Using the What's This? Command](#)

# View Toolbar

The View toolbar appears in three different configurations.

This is the default *View* toolbar in *Examine* mode:



**Note:** the visualization mode icons always show a black arrow within a white square to indicate a full pop-up. This means that clicking the icon displayed in the toolbar will pop-up the whole icon box to let you choose the appropriate rendering style (wireframe, shading, etc.). As it is a visualization mode command, it would be useless to let you run the command currently displayed without displaying the other available modes.

This is the View toolbar in *Walk* mode:



This is the View toolbar in *Fly* mode:



See [Navigating in Fly Mode](#)



See [Navigating in Fly Mode](#)



See [Navigating in Walk Mode](#)



See [Navigating in Examine Mode](#)



See [Fitting All Geometry in the Geometry Area](#)



See [Turning Your Head To View An Object](#)

 See [Navigating in Fly Mode](#), [Navigating in Walk Mode](#)

 See [Navigating in Fly Mode](#), [Navigating in Walk Mode](#)

 See [Viewing Along a Normal to a Plane](#)

 See [Using Standard Views](#)

 See [Using Standard Views](#)

 See [Using Standard Views](#)

 See [Using Standard Views](#)

 See [Using Standard Views](#)

 See [Using Standard Views](#)

 See [Using Standard Views](#)

 See [Panning](#)

 See [Rotating](#)

 See [Zooming In](#)

 See [Zooming Out](#)

 See [Shading \(SHD\)](#)

 See [Shading with All Edges](#)

 See [Shading with Edges without Smooth Edges](#)

 See [Shading with Edges and Hidden Edges](#)

See [Shading with Material](#)

 See [Customizing the View Mode](#)

 See [Hiding Objects](#)

 See [Displaying Hidden Objects](#)

# Capture Toolbar


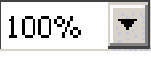

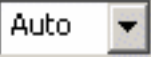
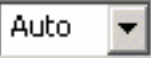





-  See [Capturing Simple Images](#)
-  See [Capturing Selected Areas of Images](#)
-  See [Capturing Simple Images](#)
-  See [Capturing Selected Areas of Images](#)
-  See [Capturing Simple Images](#)
-  See [Capturing Simple Images](#)

# Graphic Properties Toolbar

The Graphic Properties toolbar is hidden by default. it can be displayed using the **View->Toolbars->Graphic Properties** commands.











-  See [Using the Graphic Properties Toolbar](#)
-  See [Using the Graphic Properties Toolbar](#)
-  See [Using the Graphic Properties Toolbar](#)
-  See [Using the Graphic Properties Toolbar](#)
-  See [Using the Graphic Properties Toolbar](#)
-  See [Using the Graphic Properties Toolbar](#)
-  See [Using the Graphic Properties Toolbar](#)
-  See [Using the Graphic Properties Toolbar](#)



# 3Dx Device Toolbar

The 3Dx Device toolbar is hidden by default. It can be displayed using the **View->Toolbars->3DxDevice** command.



-  See [Customizing Buttons](#)
-  See [Customizing Buttons](#)
-  See [Customizing Buttons](#)
-  See [Customizing Buttons](#)
-  See [Customizing Buttons](#)
-  See [Customizing Buttons](#)
-  See [Customizing Buttons](#)
-  See [Customizing Buttons](#)

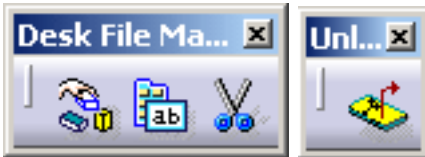
# Desk Toolbars


The Desk toolbar is hidden by default. To display it, select the **View->Toolbars** command then check **"Desk (basic)"**:



-  See [Using the FileDesk Workbench](#)
-  See [Using the FileDesk Workbench](#)
-  See [Using the FileDesk Workbench](#)
-  See [Using the FileDesk Workbench](#)
-  See [Using the FileDesk Workbench](#)

The Desk File Management and Unload toolbars are available from the Desk window:




-  See [Using the FileDesk Workbench](#)
-  See [Using the FileDesk Workbench](#)
-  See [Using the FileDesk Workbench](#)
-  See [Using the FileDesk Workbench](#)

# Knowledgeware Toolbar



 See [Using Knowledgeware Capabilities](#)

 See [Using Knowledgeware Capabilities](#)

 See the *Knowledge Advisor User's Guide*

# Feature Dictionary Editor Toolbar



See [Creating a New Object Class](#) and [Creating a New Feature Dictionary](#)



See [Modifying the Object Naming Rules](#)



See [Opening a Reference Dictionary](#)



See [Creating a New Object Class](#)



See [Adding Properties to an Object Class](#)



See [Modifying the Object Naming Rules](#)



See [Generating a Report](#)

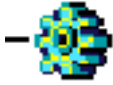
# Symbols Used in the Specification Tree

Depending on the operations you perform, you may encounter new symbols on your features in the specification tree. This table provides a list with a brief explanation about the use of these symbols.

Miscellaneous Symbols  
Product Structure Symbols  
Symbols reflecting an incident in the Geometry building  
Referenced Geometry  
Analysis Symbols

# Miscellaneous Symbols

## Miscellaneous



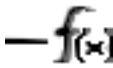
PartBody

A part body. You have access to the part's features, the way it is organized (Pad, Pocket, Intersection, etc.).



xy plane

xy plane, yz plane or zx plane. You can click the desired reference plane either in the geometry area or in the specification tree.



Point.1

Formula defined for Point.1. For more about formulas, please refer to *Infrastructure User's Guide Version 5*.



BODY1

A model with geometrical representation.



Sketch.1

Sketch. For more information about Sketcher Workbench, refer to : Entering the Sketcher Workbench in *Sketcher User's Guide*.



AbsoluteAxis

Absolute Axis: contains information about Origin, HDirection and VDirection.



Origin

Origin.



HDirection


HDirection or VDirection.

-  Geometry

Geometry (Point, Line,...): Wireframe and Surfaces features.

-  Constraints

Constraints: Parallelism, Perpendicularity, etc.

-  face

Publication : a CATPart or CATProduct element is published that is to say its geometrical data is exposed. For more information refer to Managing a Product Publication in *Assembly User's Guide*.

-  Hole.1


Assembly hole. For more about Assembly features, please refer to *Assembly Design User's Guide Version 5*.

-  Open\_body.1

External references branch of the part : external geometry (a face, a point or a line) is copied/imported from driving parts to contextual parts that are being driven (Design in context). You can customize External References in **Tools -> Options -> Mechanical Design -> Part Design**, select the **General** tab and check the box **Keep links with selected object**.

-  Product4

A product in NO SHOW. By clicking in the CATProduct's contextual menu or by using the

Hide/Show icon , you can put the product in the SHOW or NO SHOW area, the product's geometry is hidden. As a consequence, the documents under it, for instance the CATPart is in the NO SHOW space as well.

For more information about the SHOW/NO SHOW modes, see Displaying Hidden Objects in *Infrastructure's User Guide*.



Part5

A part in NO SHOW.



The Sketcher symbol is by default in NO SHOW. By this means the geometry lighter. But you can reactivate the Sketcher representation by clicking in the contextual menu or by using the Hide/Show icon



. By double-clicking on this symbol you can return into the Sketcher workbench.



# Product Structure Symbols

## Product Structure




A product. For more information, refer to Insert a New Product in *Product Structure User's Guide*.



A component or sub-product. For more information, refer to Insert a New Component in *Product Structure User's Guide*.



The purple little wheel to the left corner of the CATProduct icon and the light bar  identify a flexible sub-assembly. For more information, refer to Soft Sub-Assemblies in *Product Structure User's Guide*.



Instance of a part. This symbol  means that there is a geometrical representation of the part and that it is activated.



The representation of this part is deactivated. This symbol means the geometric representation is deactivated. Before opening a document, you choose the activate or deactivate Shape representation in **Tools->Options->Infrastructure**, select the Product Structure tab and check the box entitled **Do not activate default shapes on open**. For a particular instance in the document, you can deactivate or activate it by selecting the **Representations -> Deactivate Node / Activate Node** contextual commands.




The representation of this component is deactivated.

**Contextual parts:**


For contextual parts, the reference keeps a link with the definition instance. For each parts, every instance keeps a link with its reference. But their first reference has only one link, with a single instance which is contextual. This unique link allows you to know the name of the document (CATProduct) on which the part 's external geometry rests. Three types of contextual parts exists:

 Definition Instance

This is a contextual part and more precisely a Definition Instance. The green gear and the blue chain signify the "original" instance of a part that is contextual (driven by another part, built with another part's data) in a CATProduct.

 Instance of the Definition Instance


This contextual part is the Instance of the Definition Instance represented by the white gear and the green arrow. The geometry of this instance is connected with the Definition Instance (contextual link). You can edit this contextual part.

 Other Instance of the Contextual Part

The brown gear and the red flash signify the second or subsequent instance of a part that is contextual. There is a distinction between the "original" and subsequent instances of contextual parts because the geometrical definition of contextual parts (Skillsets) is dependant upon neighboring components (support) in the Assembly. The geometry of this contextual part depends upon another instance. This symbol can appear when you import this part into another CATProduct.

 Part1

Reference of a part. For more information, refer to Insert a New Part in *Product Structure User's Guide*.

 Deactivated\_Component

A deactivated component. The shape representation is deactivated; its geometry is not visible. This functionality can occur simultaneously on several documents containing this component, especially when this component is the instance of a reference. This operation is equivalent to the **Delete** operation because the reference of the component no longer exists within the Bill Of Material.

 Deactivated\_Product

A deactivated product.

 Unloaded\_Product

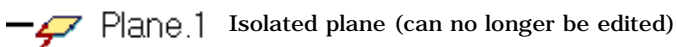
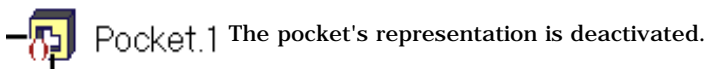
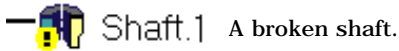
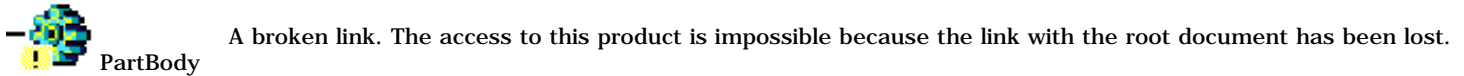
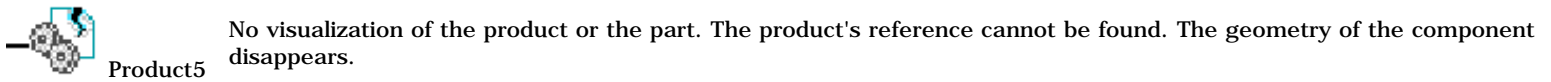
The geometry of the component disappears. The product is downloaded, its references are missing but the user is able to find them back.

# Symbols reflecting an incident in the Geometry building

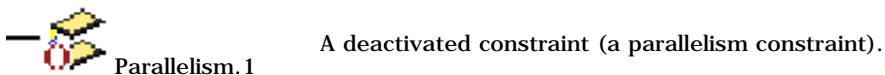
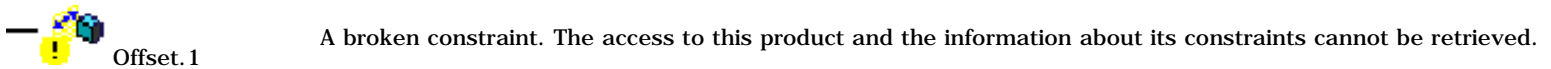
Miscellaneous Incidents

Incidents on Constraints

## Miscellaneous Incidents



## Incidents on Constraints





Perpendicularity.1 A constraint to be updated (a perpendicularity constraint).

# Referenced Geometry

## Referenced Geometry

 Solid.1

Geometry copied from a document different from the CATPart document in which it is pasted.

 Solid.1

Initial geometry has undertaken modifications in the original CATPart document: solid to be synchronized.

 Solid.1

Initial geometry has been deleted in the original CATPart document or the original CATPart document has not been found

 Solid.1

Pointed document found but not loaded (use the Load contextual command or the Edit -> Links command)

 Solid.1


External link deactivated so that geometry cannot be synchronized during the update of the part (even if the option "Synchronize all external references for update" is on).

 Solid.1

Geometry pasted (using the As Result with Link option) within the same CATPart document from which it is has been copied

 Point.4(..!Part3.1!PublishedPoint)

Point referenced in the CATPart document is a published element.

 Sketch.1(..!Part5.1!Sketch.1)

Sketch referenced in the CATPart document is a published element. The published point has undertaken modifications so that a synchronization is required.

# Analysis Symbols

Depending on the operations you perform, you may encounter new symbols on your features in the specification tree. This table provides a list with a brief explanation about the use of these symbols.

[Analysis Manager](#)

[Links Manager](#)

[Nodes and Elements](#)

[Properties](#)

[Property Connections](#)

[Virtual Parts](#)

[Restrains](#)

[Loads](#)

[Mass](#)

[Adaptivity Process](#)

[Sensors](#)

[Analysis Connection Manager](#)



**Analysis Manager**

All the objects making up an analysis document.



Analysis Manager



Links Manager



Finite Element Model



Adaptivity Process



**Links Manager**

All the links managed from the CATAnalysis document to other documents: a part, result data or computation data.



**Finite Element Model:**

Models with representations used for performing computer-aided engineering analysis (CAEA) of products. They are complementary to computer-aided design (CAD) models, which are mainly geometric representations of products.



**Adaptivity Process**

Any specification required for managing the process that will let you perform adaptativity computation (re-meshing).



## Links Manager

All the links managed between the CATAnalysis document and other documents: a part, result data or computation data.



### Results

Path to an external storage file directory (result data).



### Computations

Path to an external storage file directory (computation data).



### Link

Any link to a part or a product (assembly).



## Finite Element Model

Models with representations used for performing computer-aided engineering analysis (CAEA) of products. They are complementary to computer-aided design (CAD) models, which are mainly geometric representations of products.



### Static Case

A procedure for the computation of the system response to applied static loads under given restraints.



### Frequency Case

A procedure for the computation of the system vibration frequencies and normal modes for a given non-structural mass distribution under given restraints.



### Buckling Case

A procedure for the computation of the system buckling critical loads and buckling modes for a given Static Analysis Case.



## Nodes and Elements

Nodes and elements that can be created are either Smart Surface Mesh (FMS) or OCTREE (GPS) type.



### OCTREE Tetrahedron Mesh

Automatic mesh specifications generating tetrahedron mesh elements and using OCTREE methods.



## Properties

Any specification applied to physical properties (geometry): material and thickness (surface), connections, virtual connections and virtual parts.



## Material Property3D

A link to the material (either 2D or 3D) assigned to the part: name, support and thickness.

## Property Connections

Property Connections: assembly connections used to specify the boundary interaction between bodies in an assembled system.



### Fastened Connections

Link between two bodies which are fastened together at their common boundary, and will behave as if they were a single body.



### Slider Connections

Link between two bodies which are constrained to move together in the local normal direction at their common boundary, and will behave as if they were allowed to slide relative to each other in the local tangential plane.



### Contact Connections

Link between two part bodies which are prevented from inter-penetrating at their common boundary, and will behave as if they were allowed to move arbitrarily relative to each other as long as they do not come into contact within a user-specified normal clearance.



### Pressure Fitting Connections

Link between two bodies which are assembled in a Pressure Fitting configuration, more precisely when there are interferences or overlaps between both parts.



### Bolt Tightening Connections

Connection that prevents bodies from penetrating each other at a common interface.



### Virtual Rigid Bolt Tightening Connections

Connections used to specify the boundary interaction between bodies in an assembled system. This connection takes into account pre-tension in a bolt-tightened assembly in which the bolt is not included.



### Virtual Spring Bolt Tightening Connections

Connections are used to specify the boundary interaction between bodies in an assembled system. This connection takes into account pre-tension in a bolt-tightened assembly in which the bolt is not included.



### Rigid Connections

Connection that fastens bodies together at a common rigid interface.



### Smooth Connections

Connection that fastens bodies together at a common soft interface.



### Spot Welding Connections

Connection that fastens bodies together at a common soft interface.

## Virtual Parts

Structures created without a geometric support. They represent bodies for which no geometry model is available, but which play a role in the structural analysis of single part or assembly systems. Virtual Parts are used to transmit action at a distance.



### Rigid Virtual Parts

Generates a stiff transmission rigid virtual part.



### Smooth Virtual Parts

Generates a soft transmission rigid virtual part.



**Contact Virtual Parts**

Generates a contact transmission rigid virtual part.

**Rigid Spring Virtual Parts**

Generates a stiff transmission elastic spring virtual part.

**Spring Smooth Virtual Parts**

Generates a soft transmission elastic spring virtual part.

**Restraints**

Generates restraining joins, either on a geometry selection or on a virtual part or still various types of degree of freedom restraints.

**Restraint**

Any combination of degrees of freedom on a geometry selection.

**Isostatic**

Statically determinate supports generated on a part.

**Clamp**

Fixes all degrees of freedom on a geometry selection.

**Surface Slider**

Generates surface constraint joins, which allow points of a surface to slide along a coinciding rigid surface (fixes the translation degree of freedom for a surface in the direction of the local normal).

**Slider**

Generates prismatic joins (sliders), which allow a rigid body to translate along a given axis (fixes all degrees of freedom of a point, except for one translation).

**Sliding Pivot**

Generates cylindrical joins (actuators) which allow a rigid body to translate about and rotate around a given axis (fixes all degrees of freedom of a point, except for one translation and one rotation).

**Ball Joint**

Generates spherical joins (balls), which allow a rigid body to rotate about a given point (fixes all translation degrees of freedom of a point).

**Pivot**

Generates conical joins (hinges), which allow a rigid body to rotate around a given axis (fixes all degrees of freedom of a point, except for one rotation).

**Loads**

Generates distributed force systems equivalent to given static resultants, force densities of given intensity or acceleration fields.

**Pressure**

Generates pressure loads over a surface.

**Distributed Force**

Generates a distributed force system equivalent to a pure force at a point (given force resultant and zero moment resultant).

**Moment**

Generates a distributed force system equivalent to a pure couple (given moment resultant and zero force resultant).

**Bearing Load**

Simulate contact loads applied to cylindrical parts.

**Acceleration**

Generates a uniform acceleration field over a part.

**Rotation Force**

Generates a linearly varying acceleration field over a part.

**Line Force Density**

Generates a line force field of given uniform intensity on a part edge.

**Surface Force Density**

Generates a surface traction field of given uniform intensity on a part face.

**Body Force**

Generates a volume body force field of given uniform intensity on a part.

**Enforced Displacements**

Assigns non-zero displacement values to restrained geometric selections.

**Mass**

Non-structural mass densities of given intensity.

**Distributed Mass Equipment**

Non-structural lumped mass distribution equivalent to a total mass concentrated at a given point.

**Line Mass Densities**

Scalar line mass field of given uniform intensity on a curve geometry.

**Surface Mass Densities**

Scalar surface mass field of given uniform intensity on a surface geometry.

**Adaptivity Process****Adaptivity Convergence**

Creating global adaptive mesh refinement specifications and computing adaptive solutions.

**Adaptivities**

Selectively refining the mesh in such a way as to obtain a desired results accuracy in a specified region.

**Adaptivity Boxes**

Local specifications relative to the maximum error in the approximate computed solution relative to the exact solution.

**Sensors****Sensors**

A synthesis of analysis results which provide measures that can be re-used in Knowledgeware in order to set rules, checks and formulas.

**Sensor sets**

Different types of object sets can be generated:

- **Static case:** **misesmax** (Maximum von Mises), **dispmax** (Maximum Displacement), **reaction** (Reaction on geometry associated to restrain and connection specifications) and **globalerror**
- **Frequency case:** **Frequency** (Represents the frequency value) and **Frequencies** (Represents the list of the frequency values)
- **Buckling case:** **Buckling Factor**



## Analysis Connection Manager



### **Analysis Connections**

All the connections that you choose not to create using the **Assembly Design** workbench.



### **Face Face Analysis Connections**

Connections used for connecting either faces that are parallel to each others or concentric cylinders, on an assembly model.



### **Distant Analysis Connections**

Connections used for connecting any part from an assembly with or without point type geometrical elements, on an assembly model.



### **Spot Welding Analysis Connections**

Connections used for projecting welding points onto two parallel faces, on an assembly model.

# Viewing and Hiding Toolbars

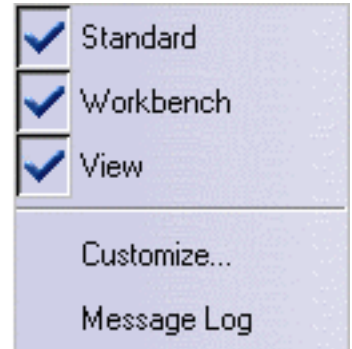


## Method 1



### 1. Select **View->Toolbars**.

The list of current toolbars is displayed. Currently visible toolbars are indicated by a tick symbol to the left of the toolbar name.



### 2. In the list, click the toolbar you want to view or hide.



Note that:

- you can detach toolbars from the application window border by dragging the double line to the left of the toolbar: you can drag the toolbar anywhere around the screen, then dock the toolbar in the same or in another location by dragging it onto the application window border
- you can drag a tool icon from a workbench toolbar, and drop it onto a selected object: this is a quicker way to run a command.

You can also drag and drop an object onto an object: for example, copy a fillet on a part.



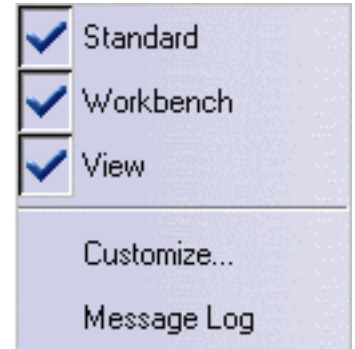


**Method 2**



1. Right-click any icon in any toolbar.

The list of current toolbars is displayed. Currently visible toolbars are indicated by a tick symbol to the left of the toolbar name.



2. Select the **Customize...** command to display the Customize dialog box.
3. Click the Toolbars tab, then toggle the Hide or Show button to hide or show the toolbar.



# Viewing the Commands List

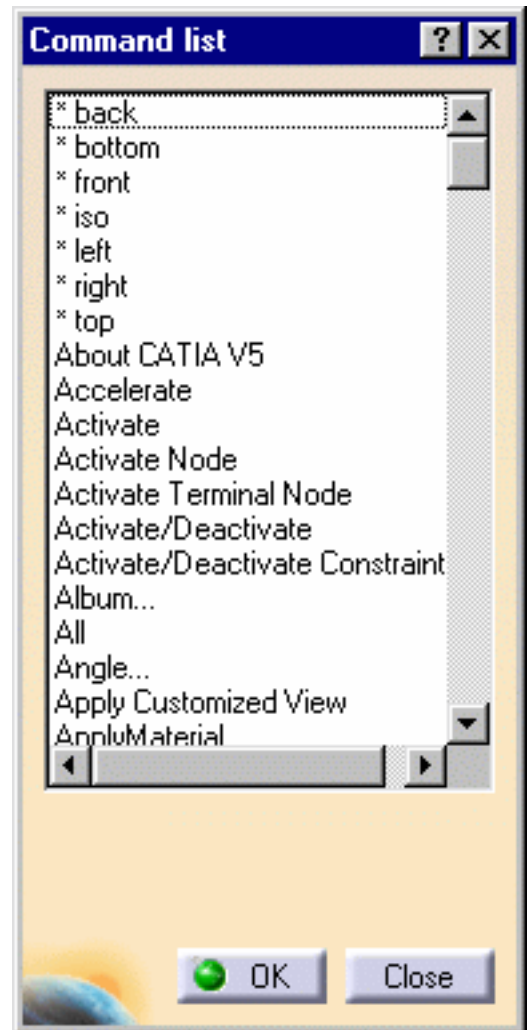


This task illustrates how to access a list of all available commands.



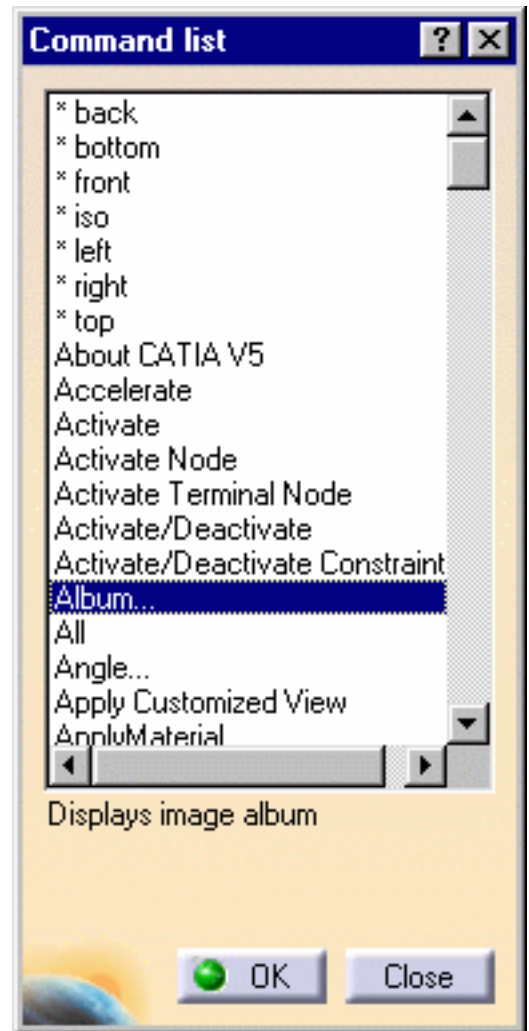
## 1. Select **View -> Commands List...**

The Command List... dialog box is displayed:



## 2. Select a command in the list.

Help about the command is displayed in the dialog box:



3. With the command still selected, click **OK**.

The selected command is accessed if the appropriate workbench is already activated.



To access a command rapidly, double-click the name of the command in the list.

If you know the first letter of the command, select any command in the list, then type the first letter to go to the commands starting with the letter you typed.



# Keyboard Shortcuts

Use this keyboard key (or combination)...	To...
Escape	Exit the current dialog box (when there is one)
F1	Get contextual online help
Shift + F1	Get help on toolbar icons
Shift + F2	Toggle the specification tree overview on and off
F3	Toggle specification tree display on and off
Alt + F8	Run macros
Shift + F3	Activate the graph is the model is active and inversely
Home	Display the top of the graph
End	Display the bottom of the graph
Page Up	Relocate the graph one page up
Page Down	Relocate the graph one page down
Ctrl + Page Up	Zoom In the graph
Ctrl + Page Down	Zoom Out the graph
Up arrow	Relocate the graph 1/10th (one tenth) of a page to the top
Down arrow	Relocate the graph 1/10th (one tenth) of a page to the bottom
Left arrow	Relocate the graph 1/10th (one tenth) of a page to the left
Right arrow	Relocate the graph 1/10th (one tenth) of a page to the right
Ctrl + Tab	Swap active document windows
<b>Alt + Enter</b>	Run the Properties... command
<b>Ctrl + C</b>	Run the Copy command
<b>Ctrl + F</b>	Run the Search... command
<b>Ctrl + G</b>	Run the Selection Sets... command
<b>Ctrl + N</b>	Run the New... command
<b>Ctrl + O</b>	Run the Open... command
<b>Ctrl + P</b>	Run the Print... command
<b>Ctrl + S</b>	Run the Save... command
<b>Ctrl + V</b>	Run the Paste command
<b>Ctrl + X</b>	Run the Cut command
<b>Ctrl + Y</b>	Run the Redo command
<b>Ctrl + Z</b>	Run the Undo command



**Another useful shortcut**

Clicking an icon lets you run the command associated with that icon only once. However, double-clicking an icon lets you use the associated command as many times as you want without having to click on the icon several times.

# Customizing

- Customizing Toolbars
- Customizing Standards
- Customizing Settings

# Customizing Toolbars and Workbenches

[Customizing a Toolbar by Dragging and Dropping](#)

[Managing User-Defined Toolbars](#)

[Creating User-Defined Workbenches](#)

[Customizing Command Properties](#)

[Customizing Options](#)

[Customizing Buttons](#)

# Customizing a Toolbar by Dragging and Dropping

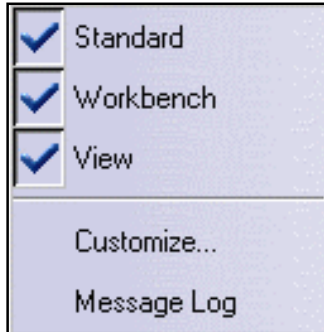


This task shows how to customize a toolbar by dragging and dropping a command onto the toolbar to add the command, and dragging a command away from a toolbar to delete the command.



1. Right-click any icon in any toolbar to display the list of toolbars.

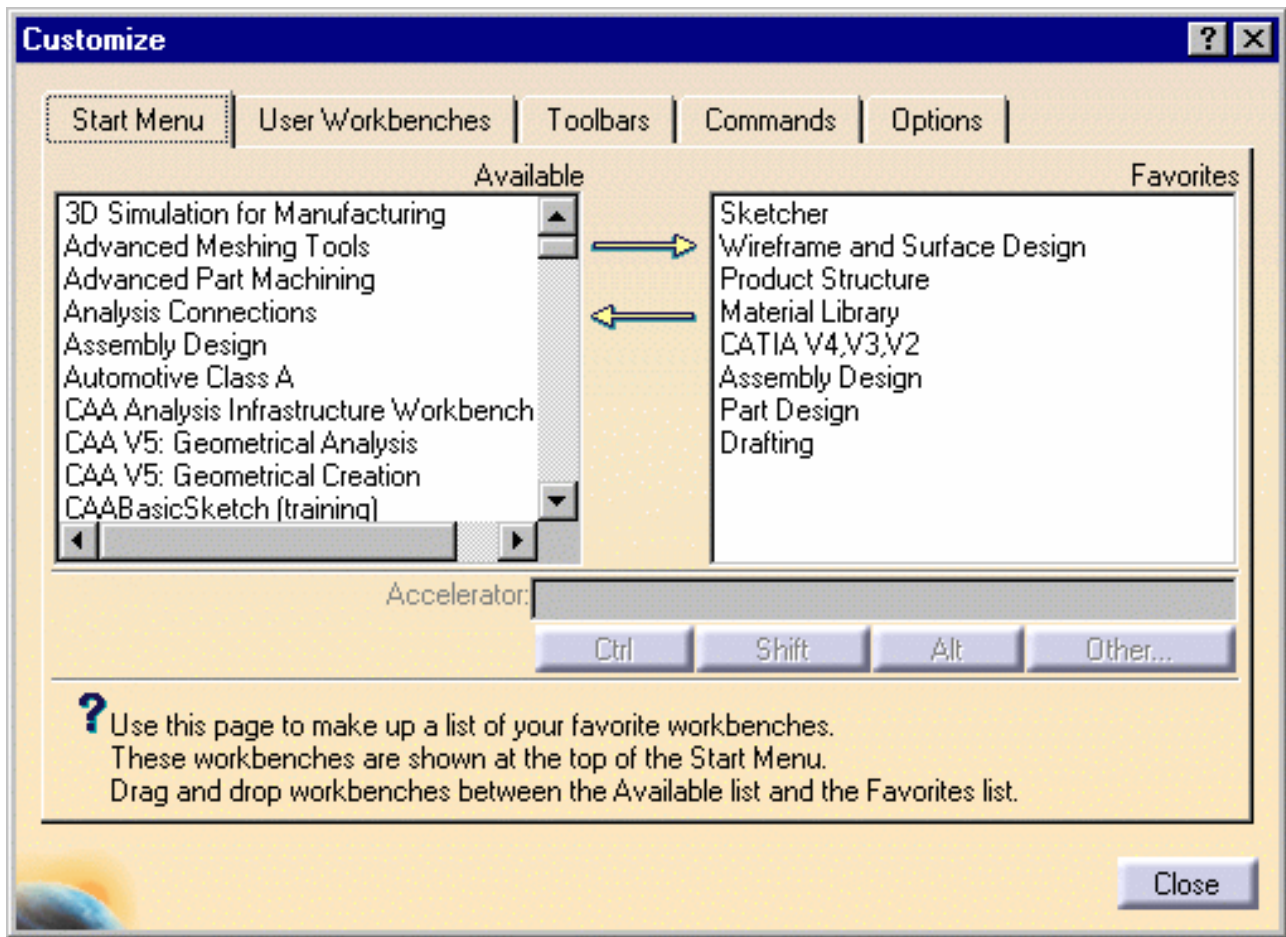
For example, the following toolbar list appears when no documents are open:



2. Select the **Customize...** command to display the Customize dialog box.

The dialog box contains the following tabs:

- **Start Menu:** customizes the Start menu and workbench access icons (as described in "Accessing the Navigation Tools")
- **User Workbenches:** lets you create your own workbenches
- **Toolbars:** lists the currently visible toolbars (default)
- **Commands:** lists the commands you can drag and drop onto a toolbar
- **Options:** contains general customization options.

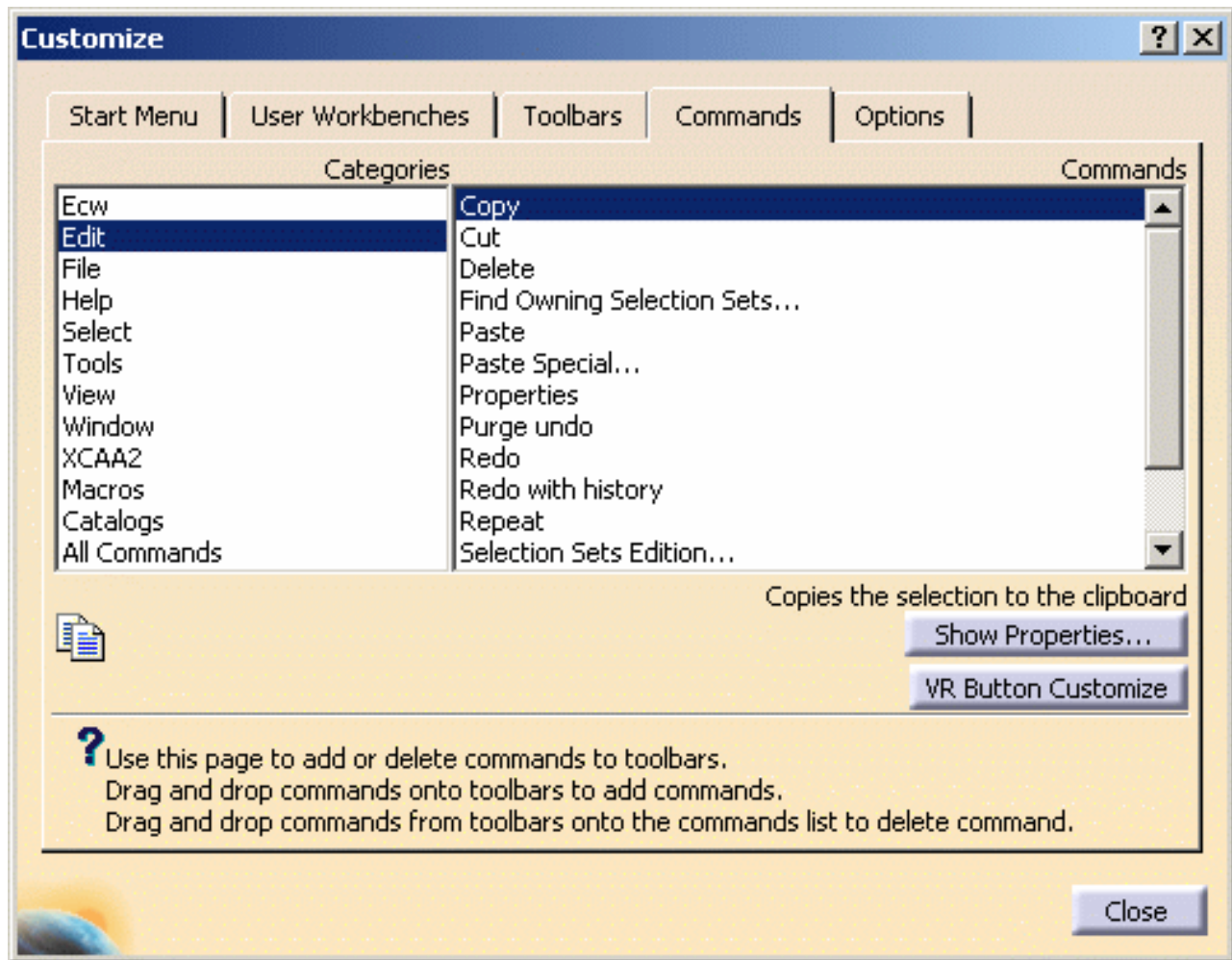


With the Toolbars tab open, double-clicking any icon highlights, in the toolbars list, the toolbar the icon belongs to. Double-clicking any toolbar also highlights the toolbar name, allowing you to identify it.

3. Click the Commands tab to list the commands available.

The Categories list allows you to filter the listed commands by category: the category is the name of the menu in the menu bar.

For example, selecting the Edit category in the Categories list displays in the commands list all the commands likely to appear on the Edit menu.

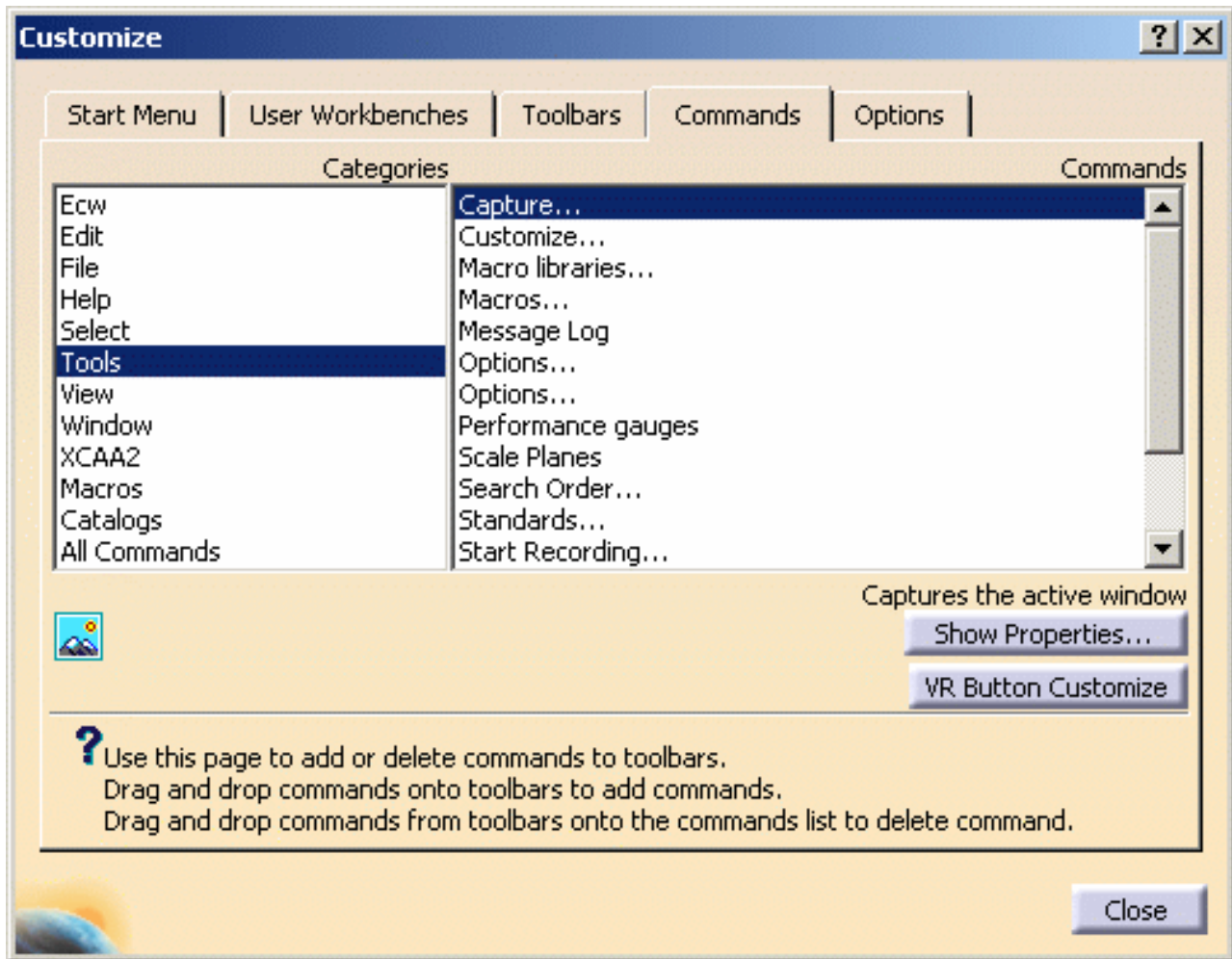


The category All Commands lists all commands available. You can also select the Select command.

If you created macros, the macro names are also listed. You can then add macros to a toolbar.

**4.** Select a command from the command list.

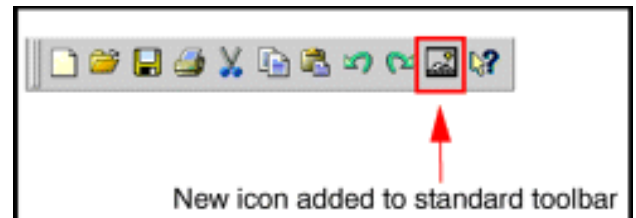
In this example, the **Capture...** command has been selected. Note that the icon for the command is displayed at the bottom of the dialog box, along with a short help message explaining the role of the icon.



5. Drag the command from the command list to the toolbar to which you want to add the command.

6. Drop the command onto the desired toolbar.

In our example, the icon for the Image Capture... command has been added to the standard toolbar:



Note that this does not apply to icon boxes which cannot be customized.

7. To delete a command from a toolbar using the drag-and-drop mechanism, drag the command and drop it back inside the list of commands in the Customize dialog box.

You can also drag and drop commands which do not have icons: in this case, the command name appears in the toolbar.

Your customization is stored automatically in the FrameConfig.CATSettings file: you recover it if you exit and restart. For more information about settings, refer to "[About Settings](#)".



## Working with ENOVIA LCA: Optimal CATIA PLM Usability

The Safe Save mode prevents the user from creating or editing data in CATIA that cannot be correctly saved in ENOVIA V5.

Working in this mode means that certain commands are unavailable (i.e. grayed) as you enter the workbench.

However, the list of commands displayed using the **Customize...** command does not gray these unavailable commands. As a consequence, if you drag and drop one of these commands onto a toolbar then use it, bear in mind that the created data will not be saved in ENOVIA V5.


For detailed information on the interoperability between CATIA and ENOVIA, refer to the *Version 5 ENOVIA-CATIA Interoperability User's Guide*.





# Managing User-Defined Toolbars

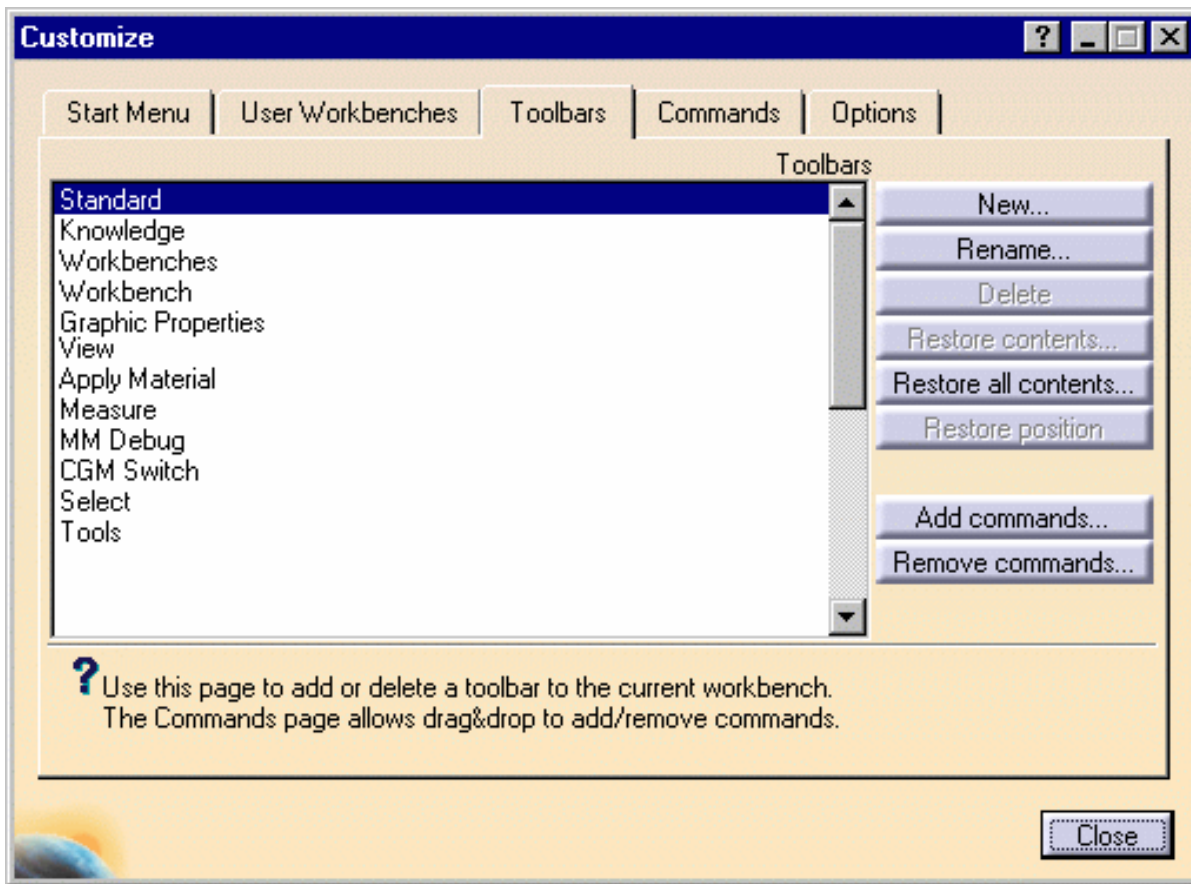
 This task shows how to create, rename, delete, restore, hide and show user-defined toolbars.

-  **1.** Right-click any icon in any toolbar to display the list of toolbars and associated commands.

For example, the following toolbar list appears when no documents are open:



- 2.** Select the **Customize...** command to display the Customize dialog box, and click the Toolbars tab if it is not already activated.

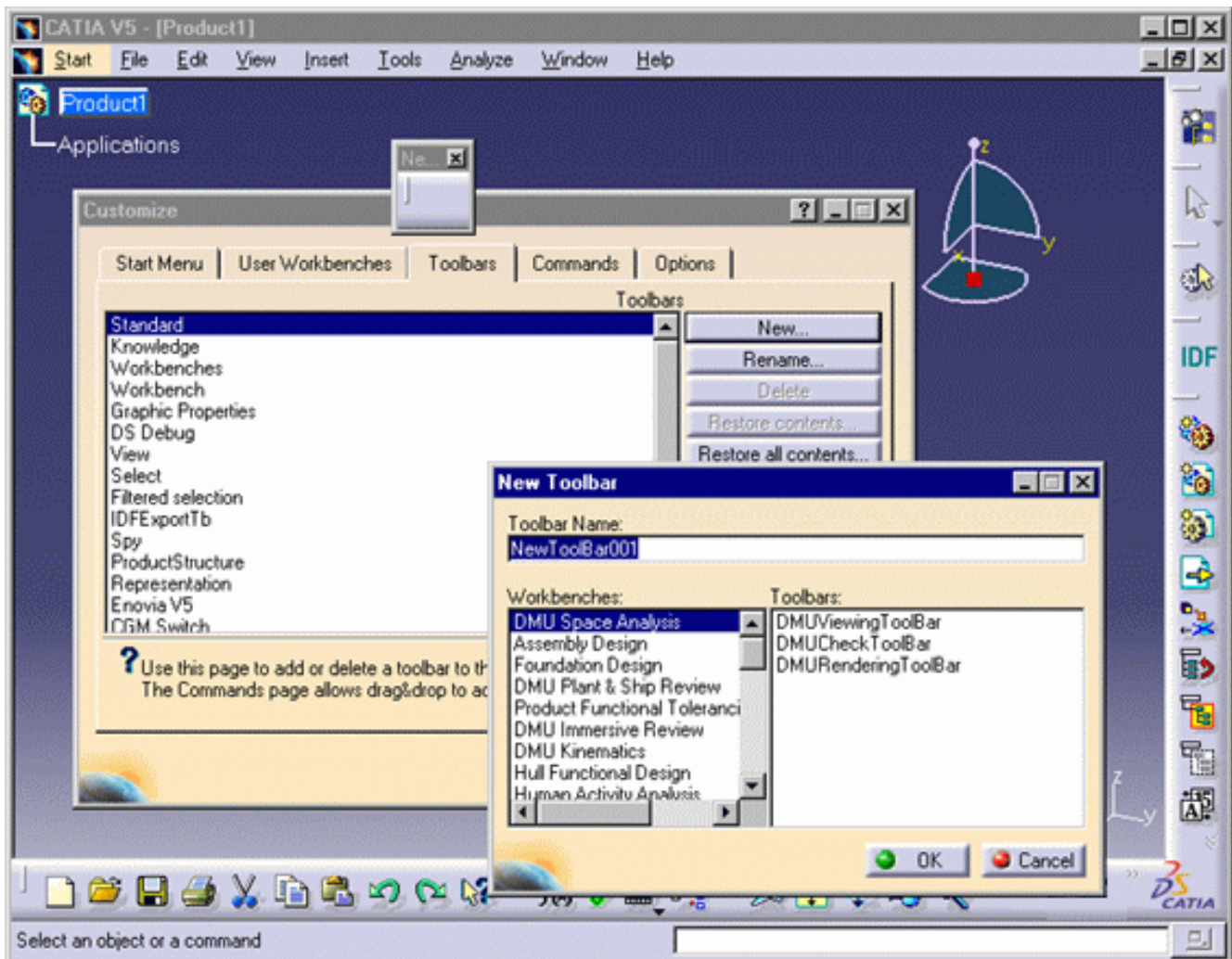


The Toolbars tab lists the currently available toolbars, and provides the commands for:

- creating new toolbars
- renaming toolbars
- deleting toolbars
- restoring the original content of selected toolbars
- restoring the original content of all toolbars
- restore the toolbars to their original position in the current workbench
- add commands to the selected toolbar
- remove commands from the selected toolbar.

3. To create a new toolbar, click the New... button.

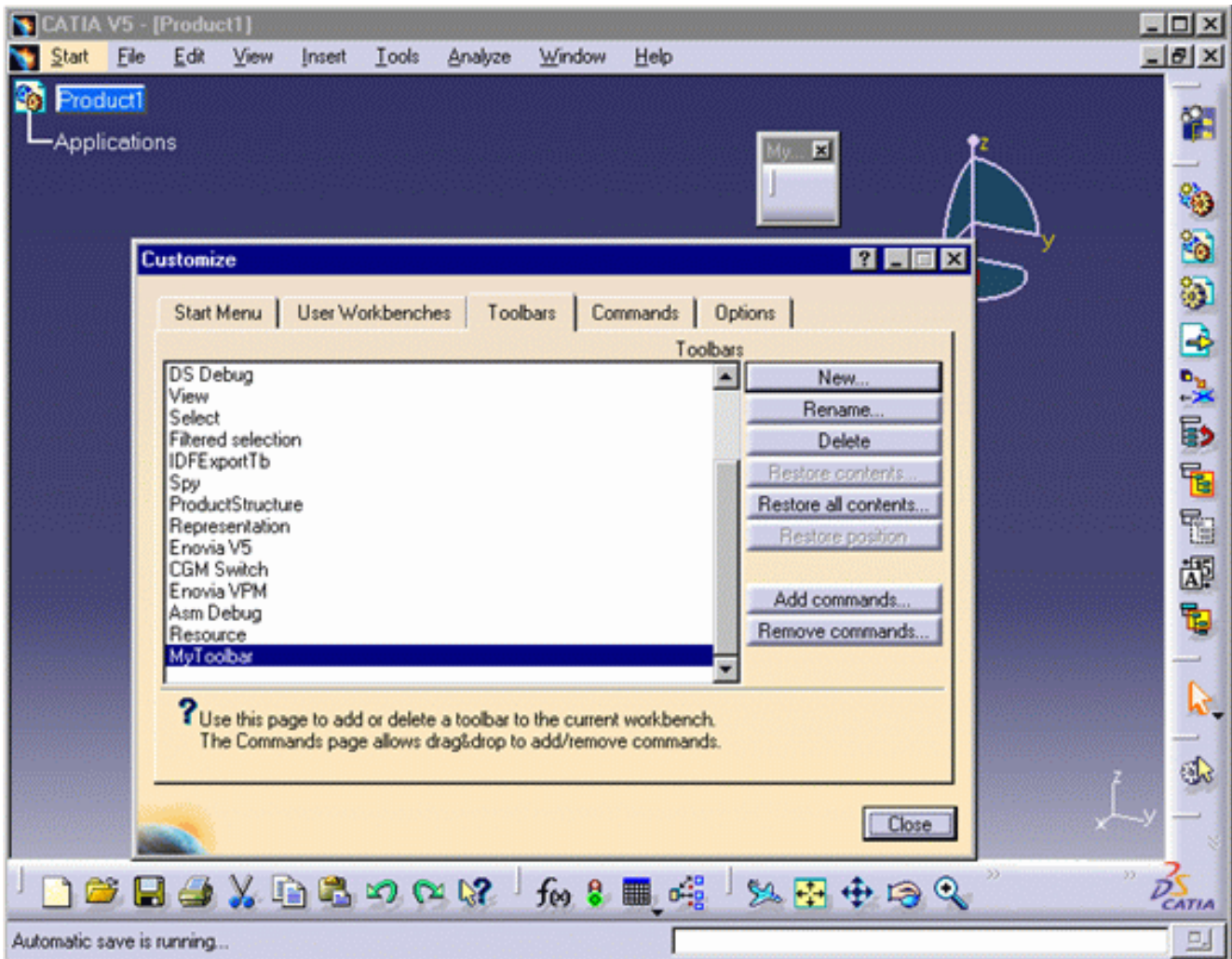
The New Toolbar dialog box appears with a default toolbar name (NewToolBar001), and an empty toolbar appears:



4. Enter the name of the new toolbar, then click **OK**.

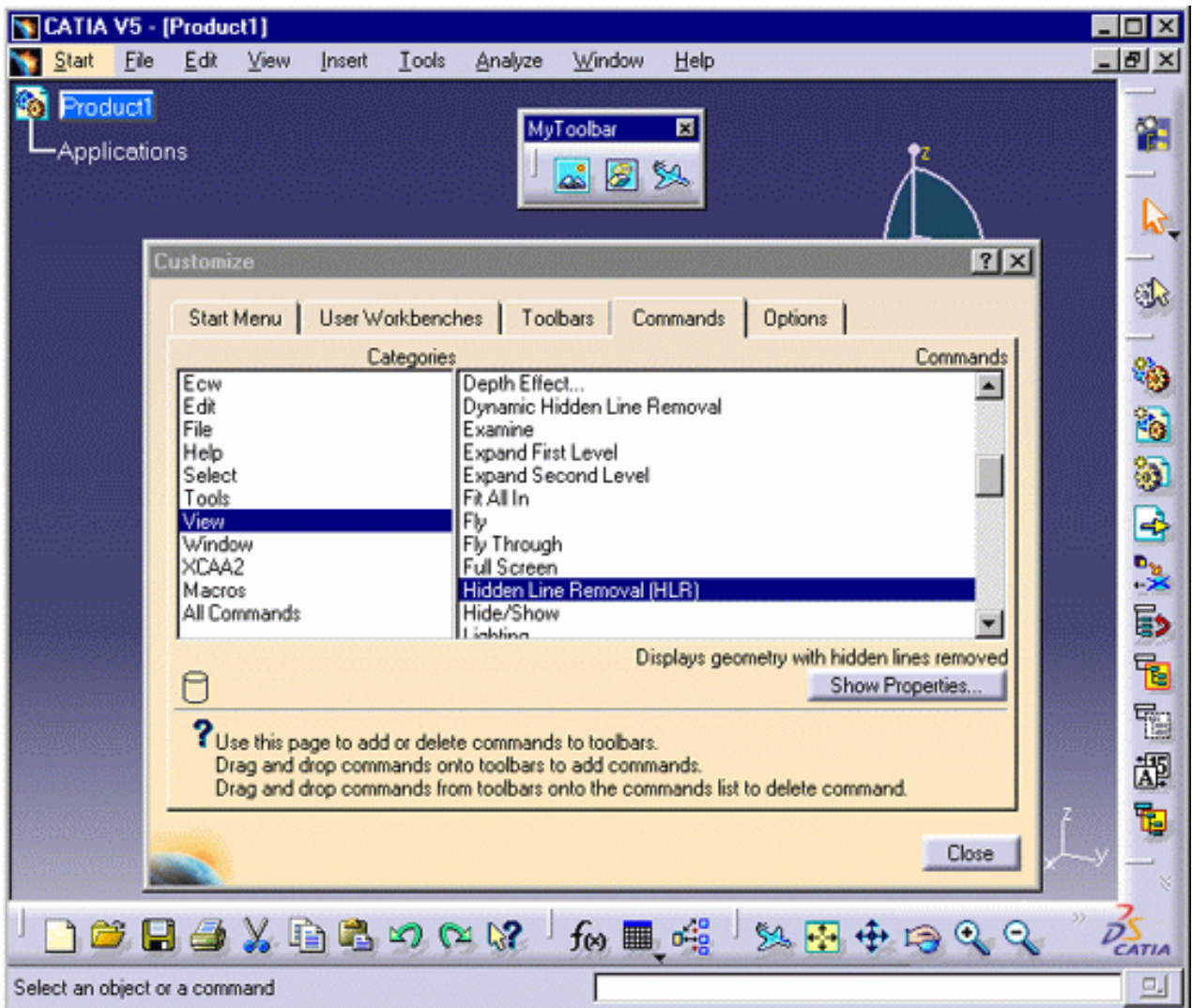
The new toolbar is added to the bottom of the list, and an empty toolbar is added to the right of the main application window.

In our example, the toolbar "MyToolbar" has been added to the list:



5. Click the Command tab and drag and drop commands onto the new toolbar.

In our example, the toolbar "MyToolbar" contains three icons:



6. To rename a user-defined toolbar, click the **Toolbar** tab again, select a toolbar, click the **Rename...** button and enter the new name, then click **OK**.
7. To delete a user-defined toolbar, click the **Toolbar** tab again, select a toolbar, click the **Delete...** button, then click **OK**.
8. To restore the original contents of a toolbar, click the **Toolbar** tab again, select a toolbar, click the **Restore...** button, then click **OK**.

The **Restore All...** button restores the original contents of all toolbars.

9. To hide or display a user-defined toolbar, click the **Toolbar** tab again, select a toolbar, and click the **Hide** or **Show** button to toggle toolbar display on or off, then click **OK**.

## Working with ENOVIA LCA: Optimal CATIA PLM Usability


The Safe Save mode prevents the user from creating or editing data in CATIA that cannot be correctly saved in ENOVIA V5.


Working in this mode means that certain commands are unavailable (i.e. grayed) as you enter the workbench.

However, the list of commands displayed using the **Customize...** command does not gray these unavailable commands. As a consequence, if you drag and drop one of these commands onto a toolbar then use it, bear in mind that the created data will not be saved in ENOVIA V5.



# Creating User-Defined Workbenches

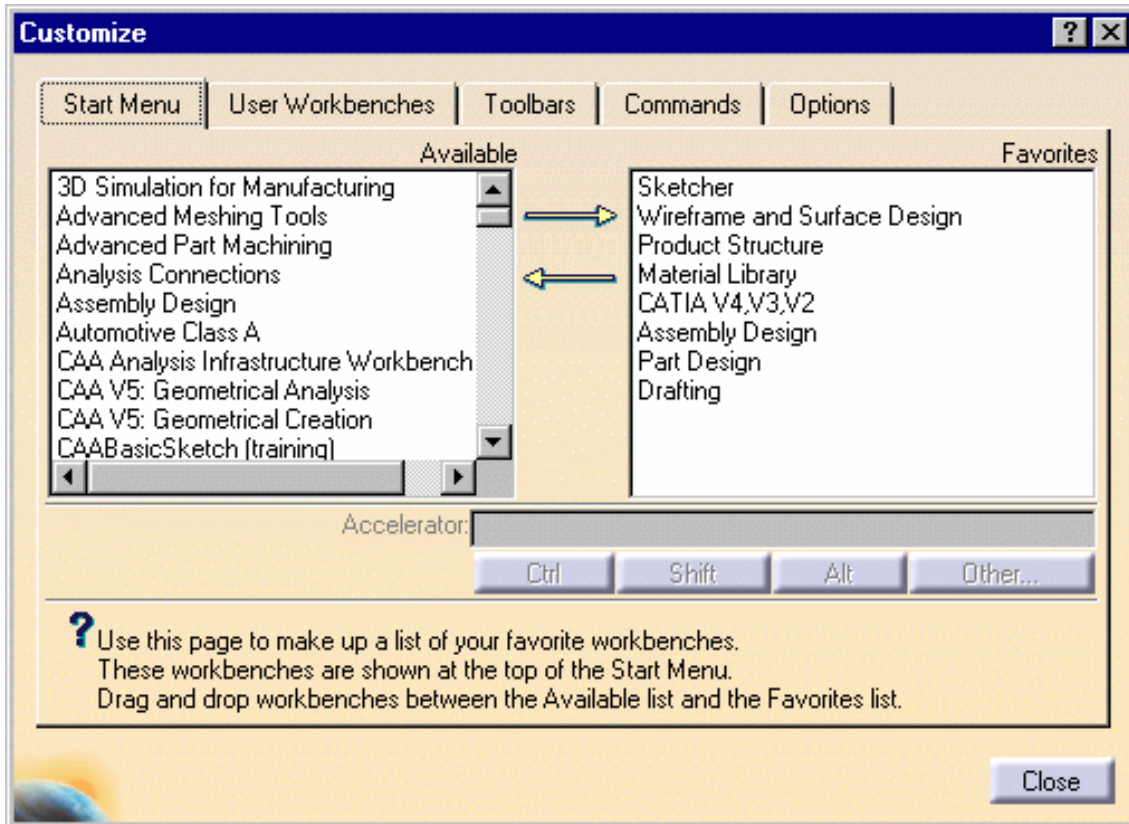
 This task explains how to create and customize workbenches.

-  **1.** Right-click any icon in any toolbar to display the list of toolbars.



For example, the following toolbar list appears when no documents are open:

- 2.** Select the **Customize...** command to display the Customize dialog box.



- 3.** You can select a workbench then click the **Ctrl**, **Shift**, **Alt** or **Other...** buttons to include them in the shortcut displayed in the Accelerator field.

The **Other...** button lets you see the list of available keys. Just select a key then click the **Add** button, or double-click a key.

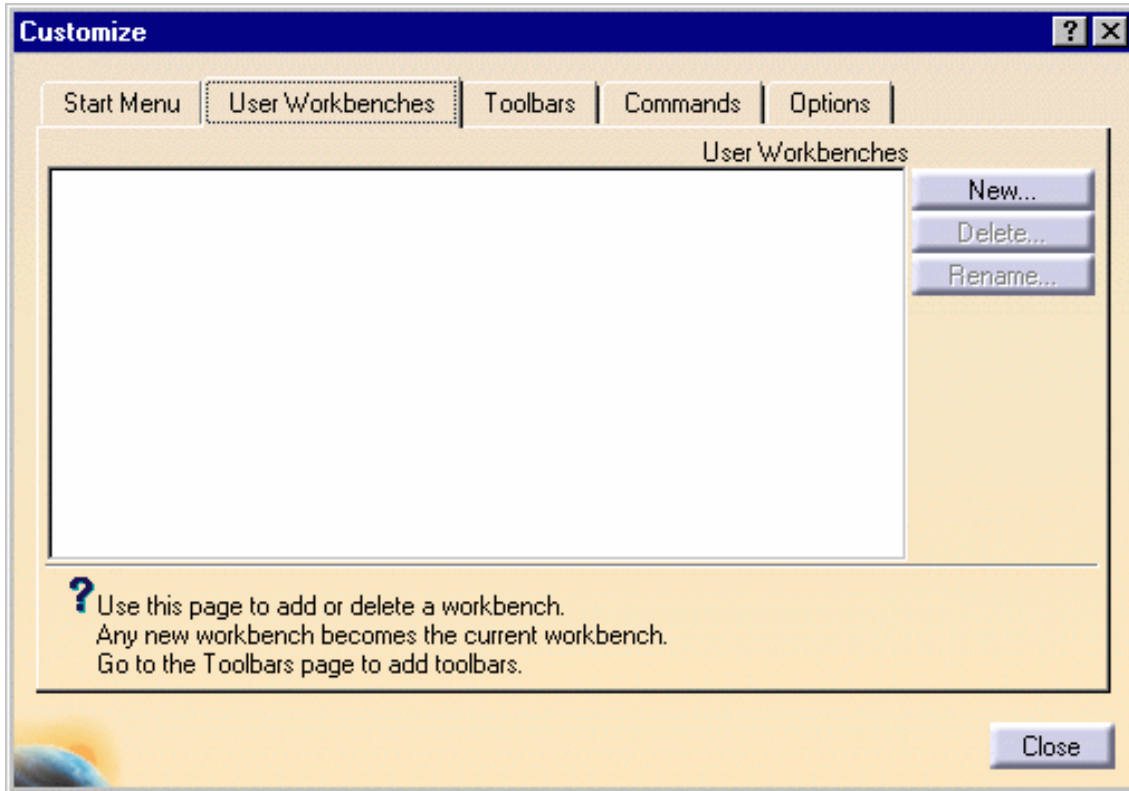
Whichever method you use, the corresponding choice is displayed in the Accelerator field, for example like this: Ctrl+.



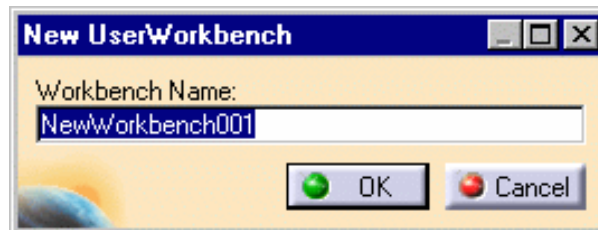
The shortcut you define will appear besides the workbench name in the Start menu.

4. Click the User Workbenches tab.

Note that the **New...** option will be available only if there is an active workbench: if no document is open, the **New...** option will be grayed out.

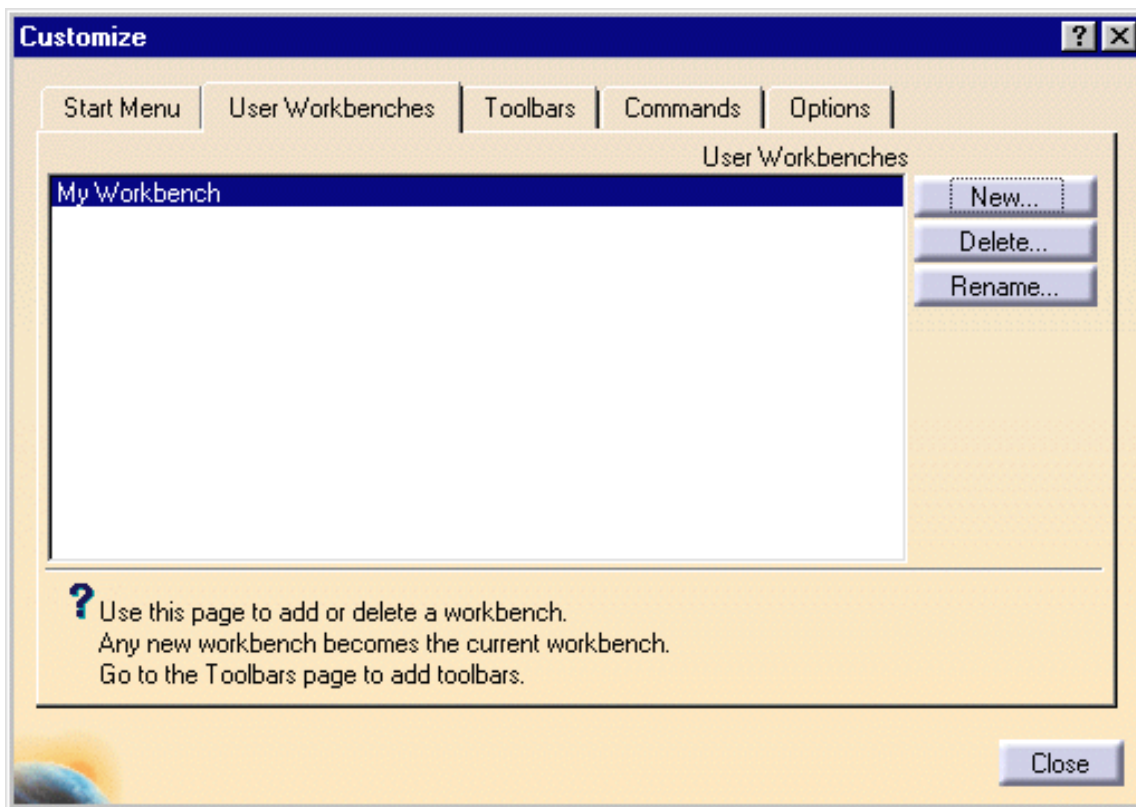


5. Click the **New...** option to display the New User Workbench dialog box:



6. Enter the name for your workbench, then click the **OK** button.

In our example, let's assume the name of the user workbench is "My Workbench". Clicking the **OK** button adds the workbench name to the list...



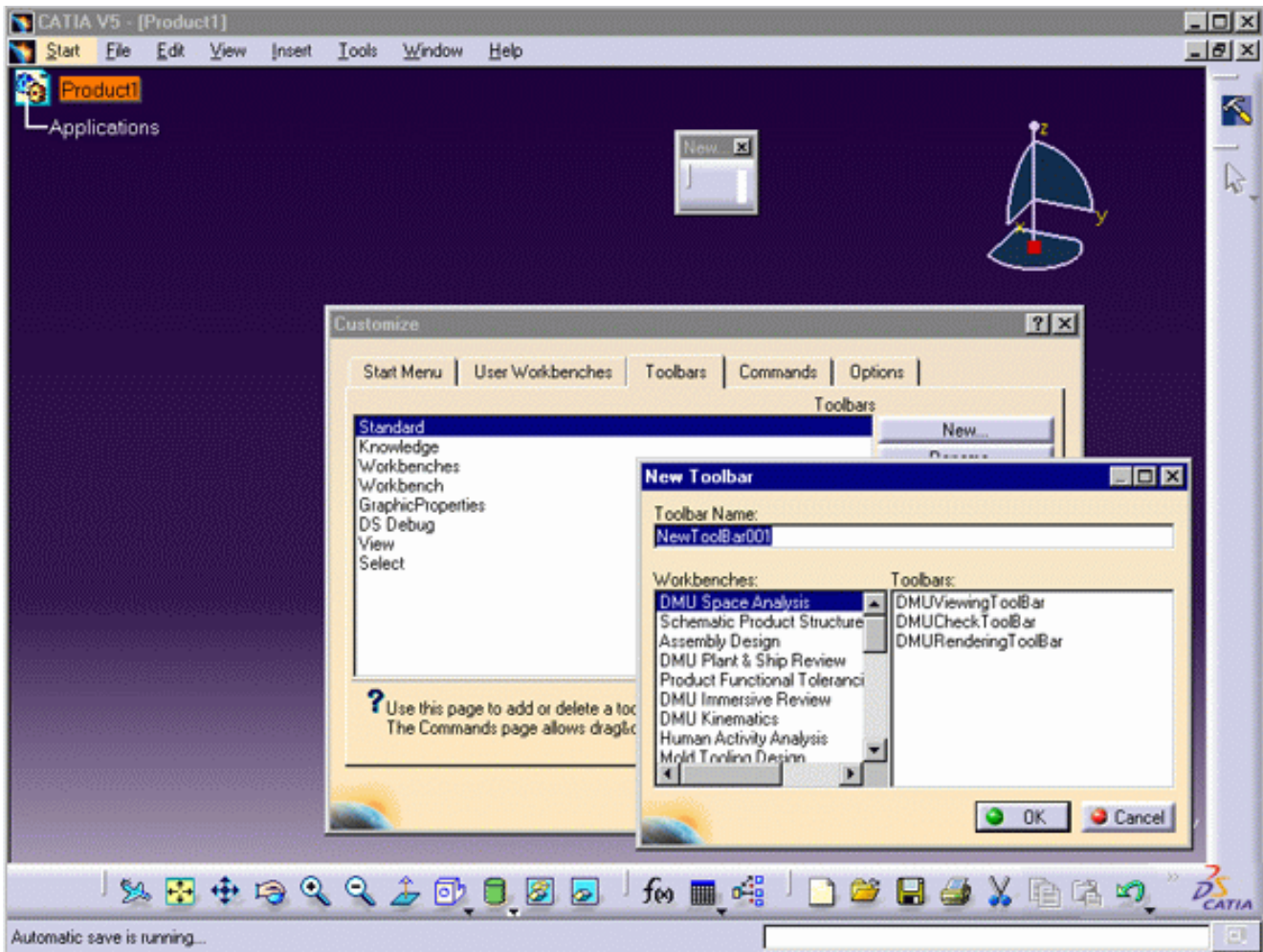
... deactivates the current workbench, and activates "My Workbench", represented by the icon top right:



7. Click the Toolbars tab, then click the **New...** option to start adding toolbars to "My Workbench".

The New Toolbar dialog box and an empty toolbar are displayed:





In the New Toolbar dialog box:

- the Workbenches list contains a list of all workbenches related to the current working context: in our example, a CATPart document was open when we created "My Workbench", so the list contains all workbenches that could possibly be activated in a CATPart document
- the Toolbars list contains the toolbars belonging to the selected workbench.

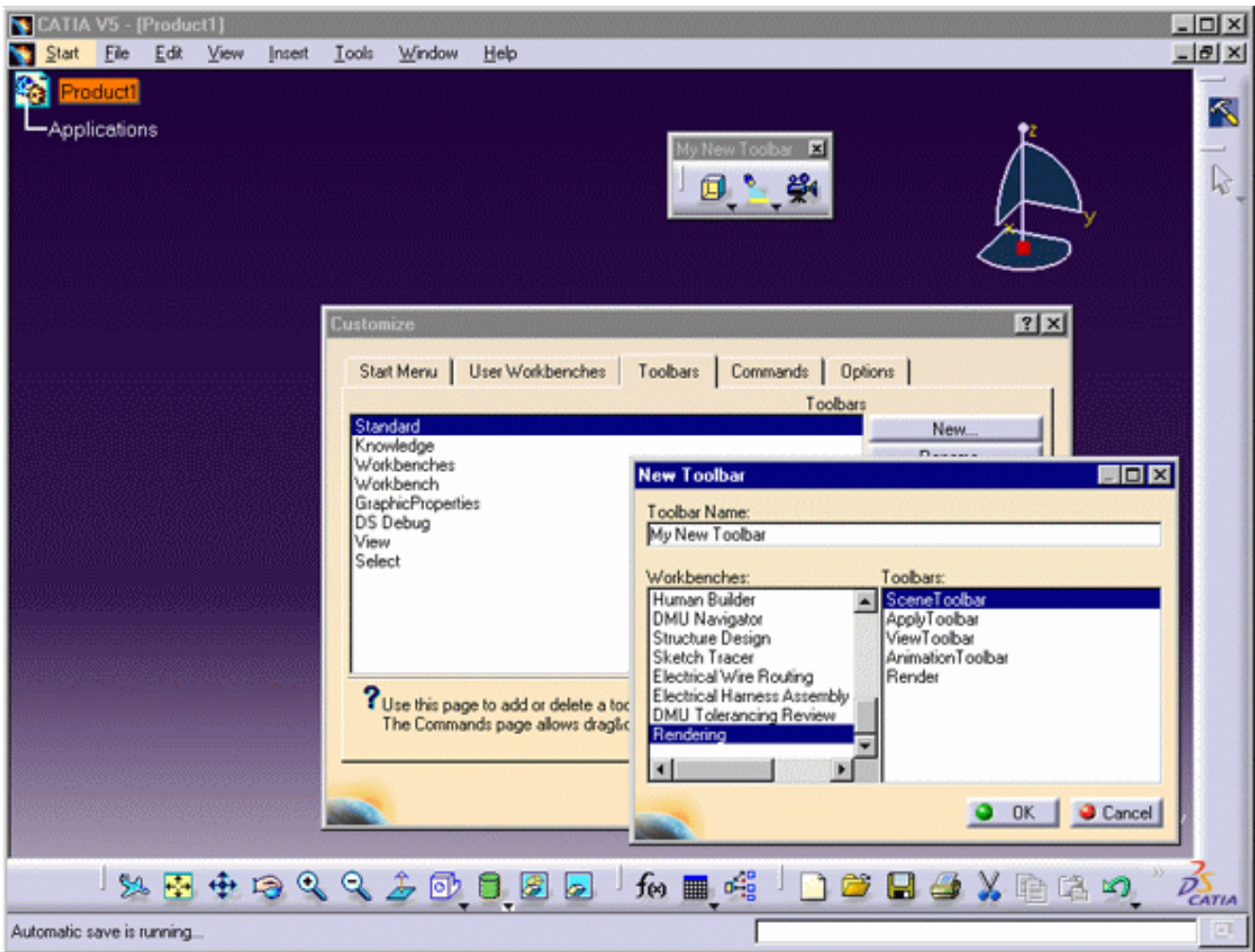
**8.** Name the new toolbar to be included in your workbench.

**9.** Select a workbench from the Workbench list.

The toolbars belonging to the selected workbench are displayed in the Toolbars list.

**10.** Select a toolbar.

The contents of the toolbar are added to your empty toolbar, for example:



11. Click **OK**, and close the Customize dialog box.

You can then drag your new toolbar and dock it to the right of the application window so that your workshop now looks like this:



**Note:** your user-defined workbench will remain active after it is created, until you activate another workbench.

Your new workbench is added to:

- the top of the Start menu (and to the Favorites list in the Start Menu tab)
- the "Welcome to CATIA V5" dialog box, which appears when you start a session, or when you click the icon representing the current workbench
- the list of workbench icons accessed by right-clicking the current workbench icon.

You can also insert your favorites workbenches in the Workbenches toolbar.

This toolbar is activated when you select Workbenches from the toolbar list:



The Workbenches toolbar displays the workbenches selected as favorites in the Start Menu tab of the Customize dialog box.

In the example below, we selected the Product Structure, Real Time Rendering and Catalog Editor workbenches as favorites. Thus, the Workbenches toolbar will look like this:



Just click one of the icons to start the corresponding workbench.



**Note:** when removing a dynamic (i.e. non-static and shareable) license during a session, the corresponding favorite workbench disappears from the list of favorites. From now on, when the removed license is acquired once again, the workbench automatically reappears in the list of favorites. You do not need to add it manually anymore.



# Customizing Command Properties

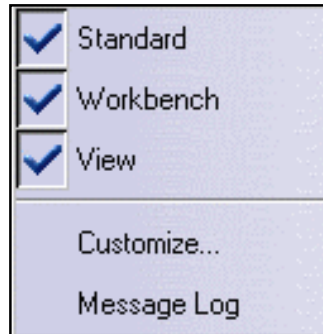


This task explains how to customize command properties by assigning keyboard shortcuts or icons to the commands.

Note that this does not apply to icon boxes which cannot be customized.



1. Right-click any icon in any toolbar to display the list of toolbars and associated commands.

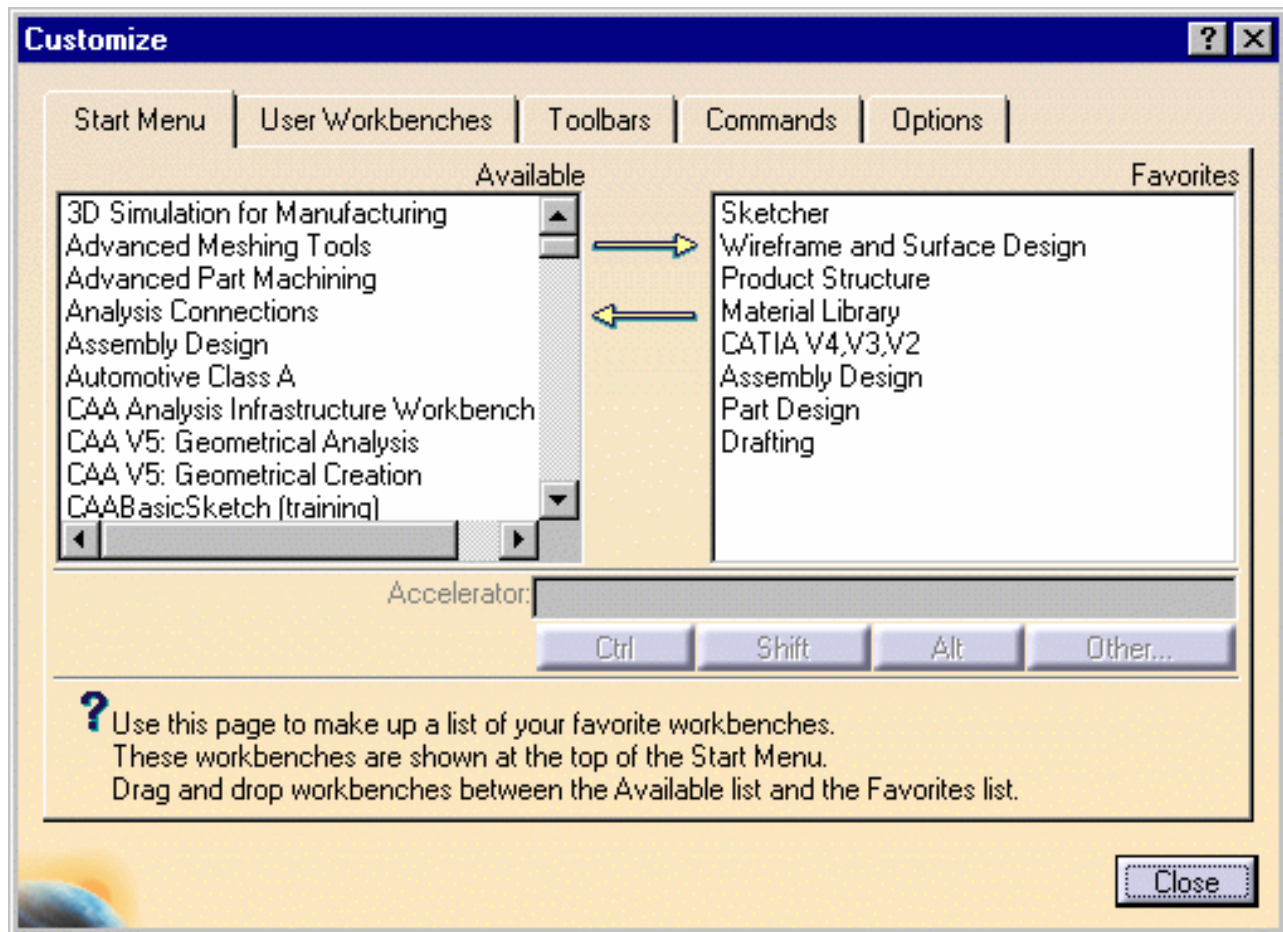


For example, the following toolbar list appears when no documents are open:

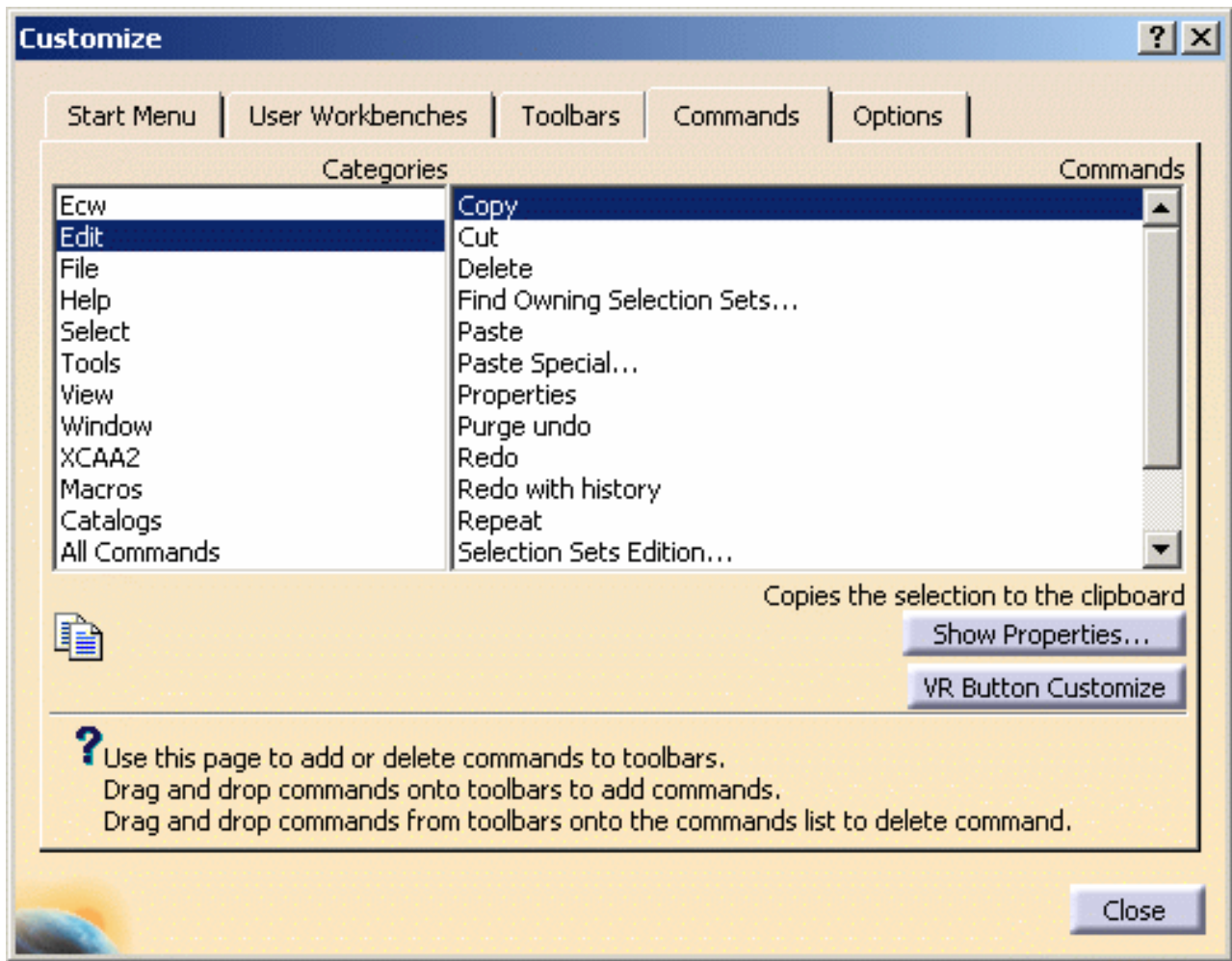
2. Select the **Customize...** command to display the Customize dialog box.

The dialog box contains three tabs:

- Start Menu: customizes the Start menu and workbench access icons (as described in "Accessing the Navigation Tools")
- User Workbenches: lets you create your own workbenches
- Toolbars: lists the currently visible toolbars (default)
- Commands: lists the commands you can drag and drop onto a toolbar
- Options: contains general customization options.

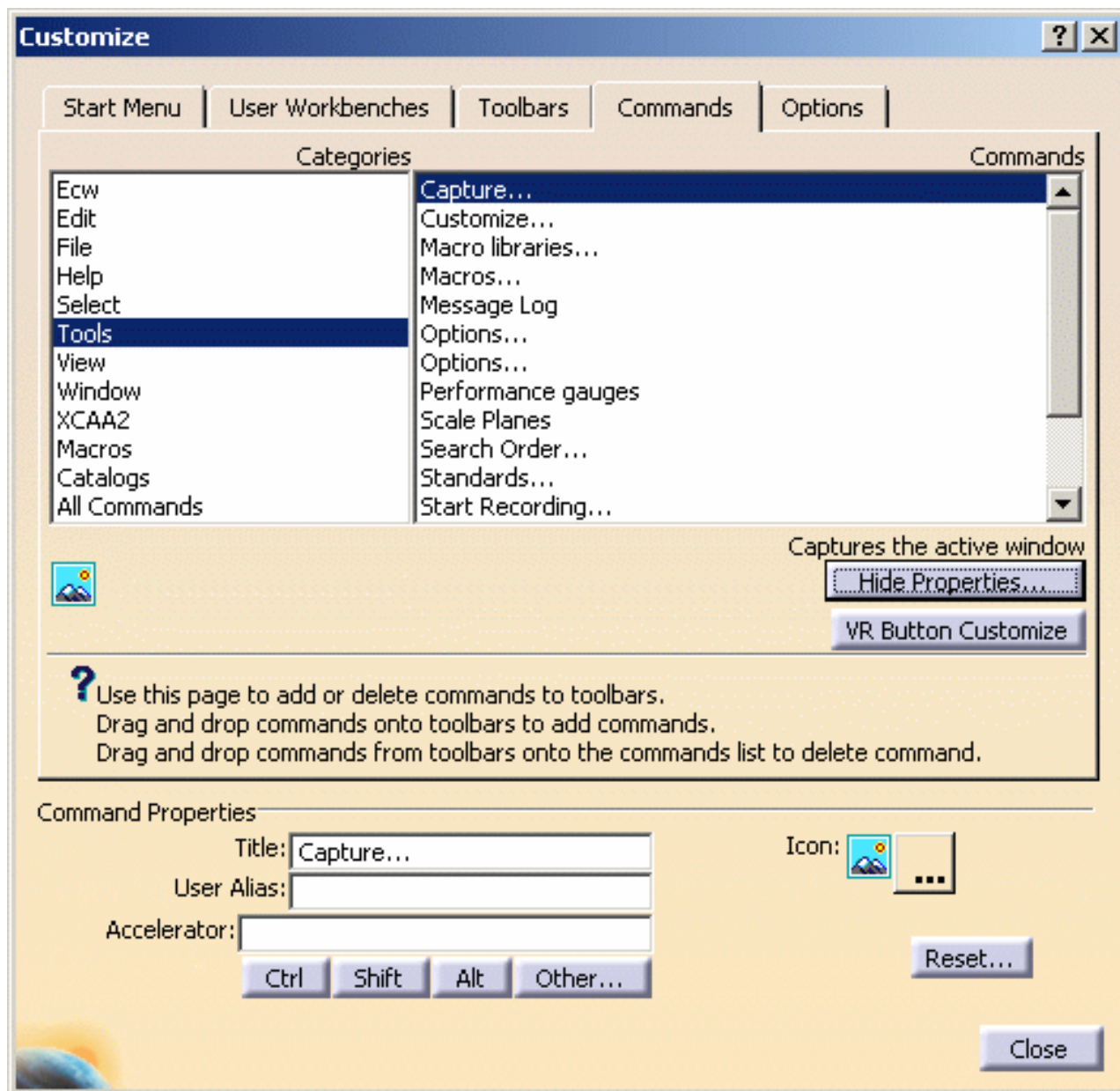


3. Click the Commands tab to list the commands available.



4. Select a category (menu name), then double-click a command, or select it and click the **Show Properties...** button.

The Command Properties frame is added to the bottom of the dialog box:



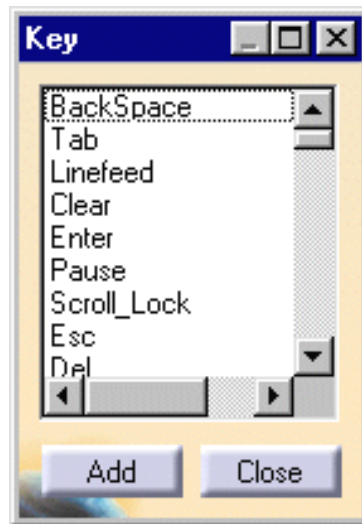
5. Enter a new name for the command, if needed, in the title field.
6. Click the Ctrl, Shift or Other buttons to include them in the shortcut displayed in the Accelerator field.



If the command you are customizing is not represented in a toolbar, a menu bar, etc., the Accelerator field is grayed to prevent you from defining a new accelerator or from using an already existing accelerator. In that case, simply drag and drop the command onto any toolbar to activate the Accelerator field.

The Meta button is now named "Other".

7. If you want to use another key, click the **Other...** button to see the list of available keys.



8. Select a key then click the **Add** button, or double-click a key.

Whichever method you use, the corresponding choice is displayed in the Accelerator field, for example like this: Ctrl+.

The User Alias field lets you assign an alias to the selected command. After clicking the Close button, you just have to type **c:alias\_name** in the PowerInput field to run the command. For example:

enter *cap* in the User Alias field

then

enter **c:cap** in the PowerInput field

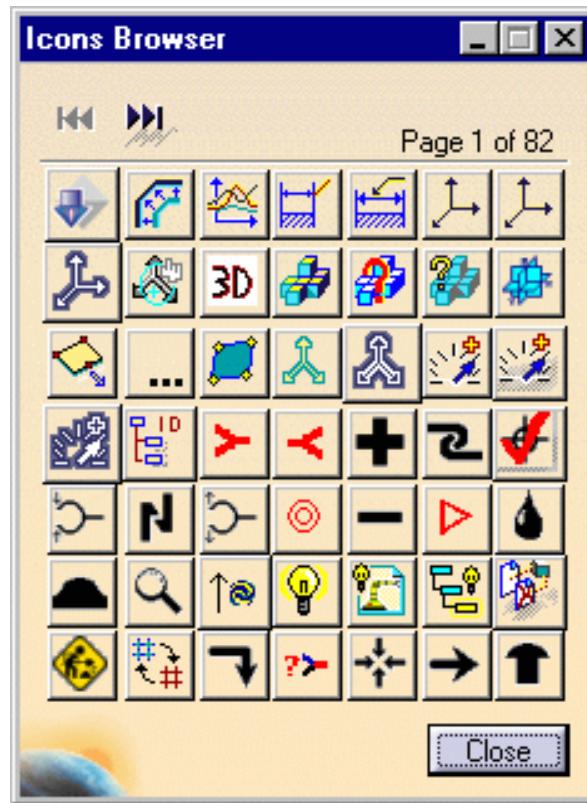
press ENTER.

The Capture command starts.

9. Click the button to the right of the Icon option (bottom right).

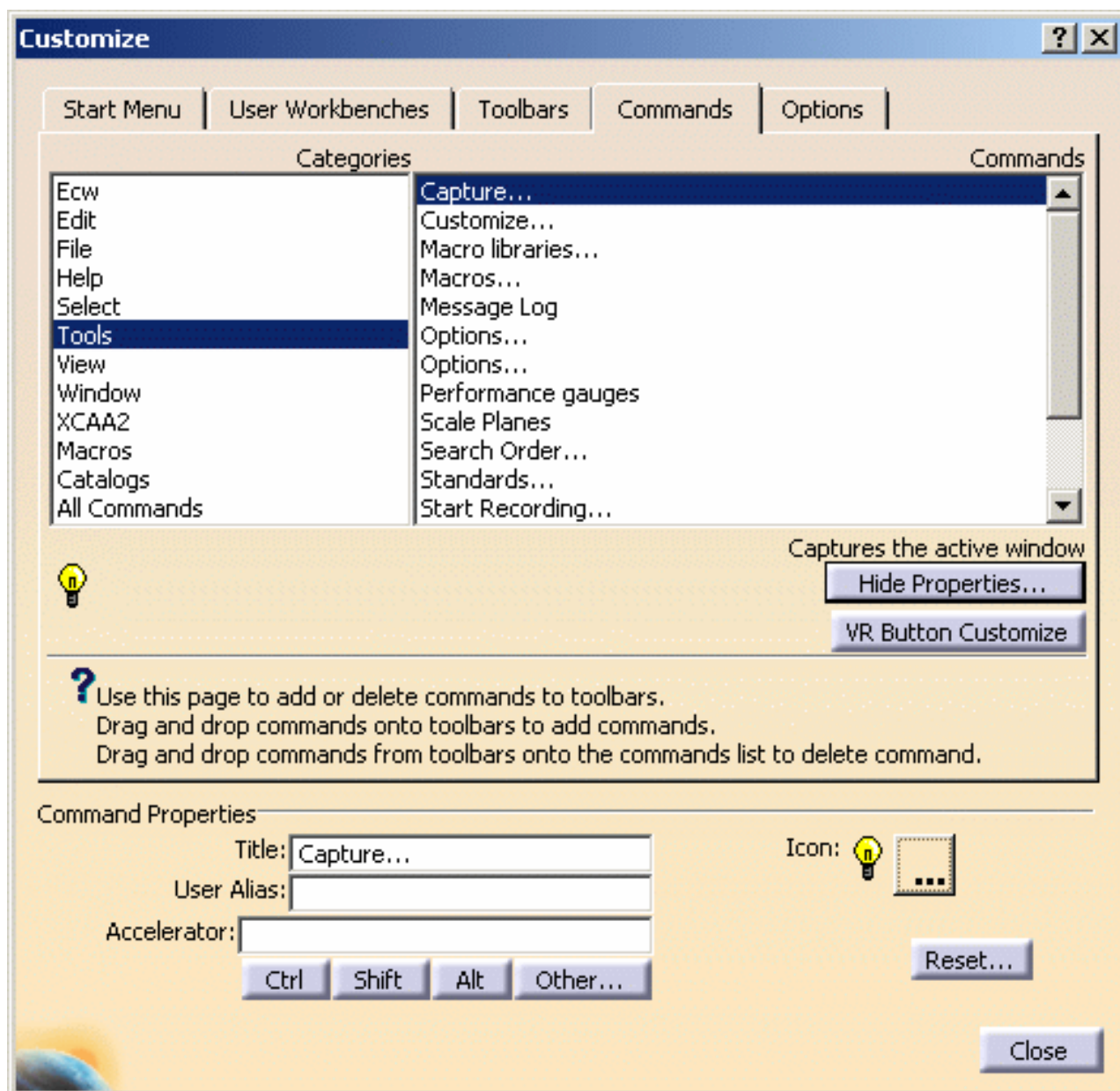


This displays the Icon Browser which lets you access all the graphic icons installed with the Version 5 software, then browse to select the appropriate icon.



**10.** Click an icon and click the **Close** button (if you used the Icon Browser).

The icon is now displayed for the command in the Commands tab, and the **Show Properties...** button changes to **Hide Properties...**:



**11.** Click the **Reset...** button to access the reset options.

You can:

- restore the original settings of the present command
- or restore the original settings of all commands in the present category.

**12.** Click the **VR Button Customize** button to access the Button Customize window which lets you assign any command or macro to a device button.

For detailed information, refer to [Customizing Buttons](#) in this guide.

**13.** Click the **Close** button.



# Customizing Options



This task explains how to customize general options.



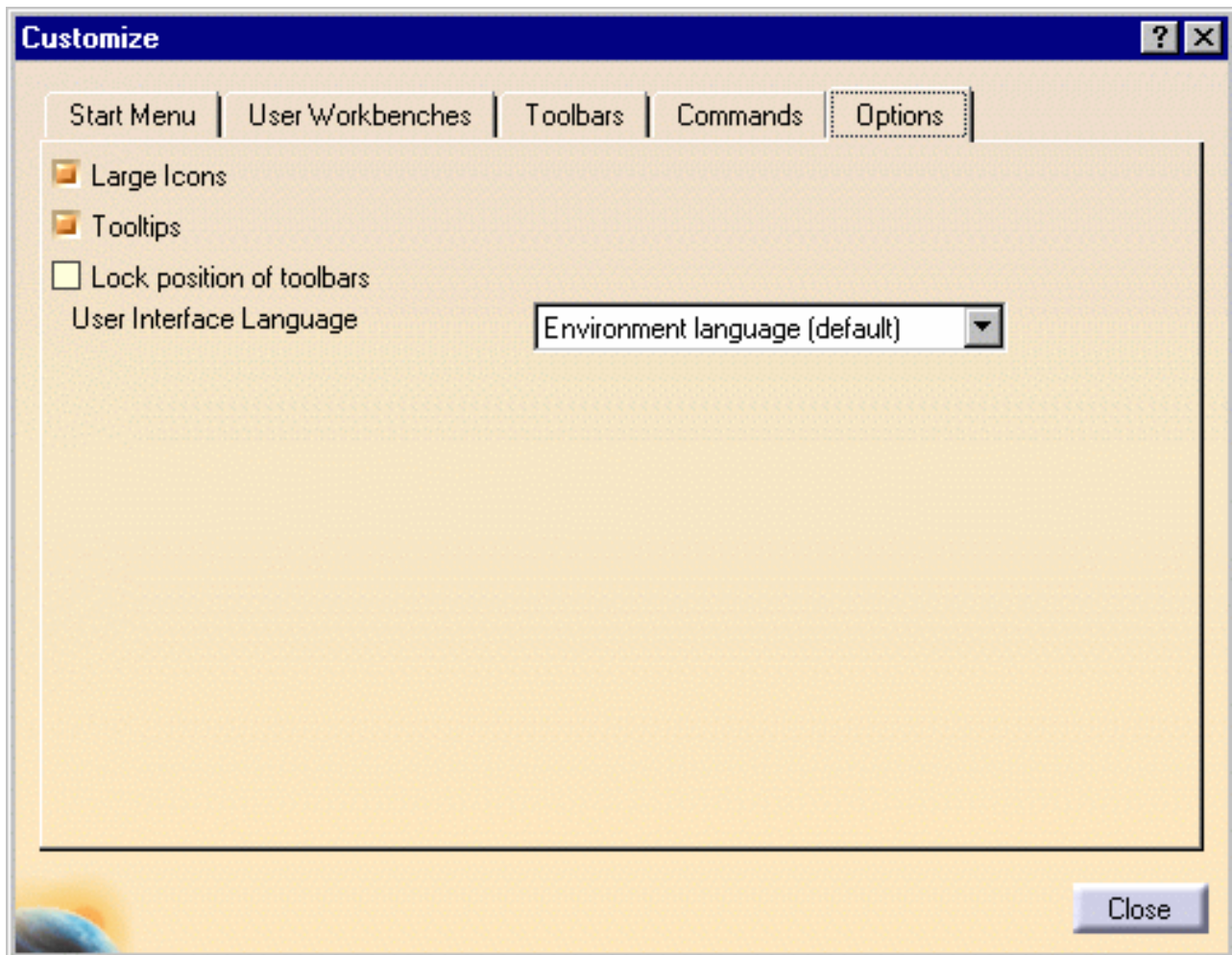
1. Right-click any icon in any toolbar to display the list of toolbars and associated commands.

For example, the following toolbar list appears when no documents are open:



2. Select the **Customize...** command to display the Customize dialog box.

3. Click the Options tab:



4. Check the Large Icons option if you want large icons displayed.

The default icon size is large. Uncheck this checkbox if you want the icons to be normal size.

5. Check the Tooltips option to switch tooltips on or off.

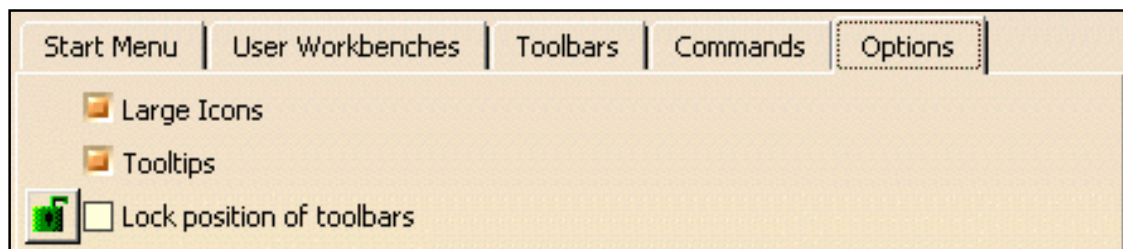
Tooltips are short help messages displayed when you point to icons, as explained in "[Displaying Tooltips and Help Messages](#)".


6. Check the Lock position of toolbars option to lock the position of all toolbars so that they cannot be moved.

Note that you do not need to restart the session to take the lock into account.



When logged as administrator, a green lock button is displayed to the left:

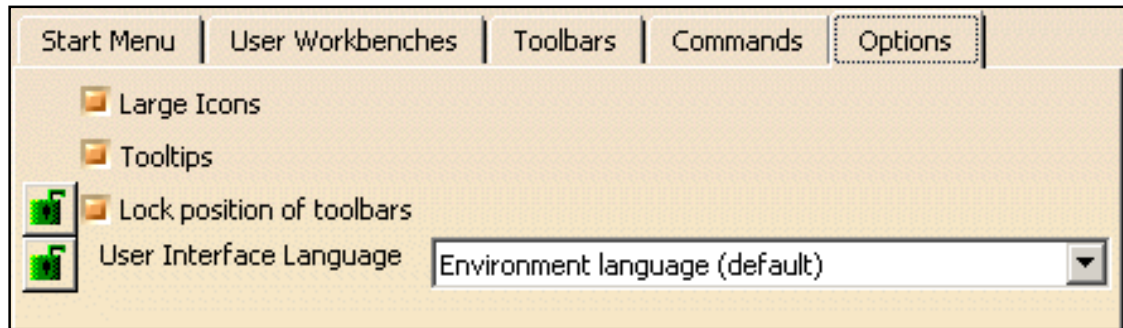



Clicking this green lock (which then turns ) lets you lock the position of toolbars for all users. As a consequence, the Lock position of toolbars option will be disabled when running a session as non-administrator.

7. The User Interface Language option lets you define a Version 5 user interface language independently from any other localized languages that may be used in other applications. Once you have selected the desired language, you have to restart your Version 5 session to take the new language into account.

**Note:** the common use is to leave the field to "Environment language (default)". This default option corresponds to the language you define in the [Regional Settings tab](#) (on Windows) or in the [LANG variable](#) (on UNIX).

When logged as administrator, you have the ability to lock this option to prevent users from modifying the user interface language:



Click the green lock displayed to the left of "User Interface Language": the lock will turn  and the option will be disabled when running a session as non-administrator.



# Customizing Buttons



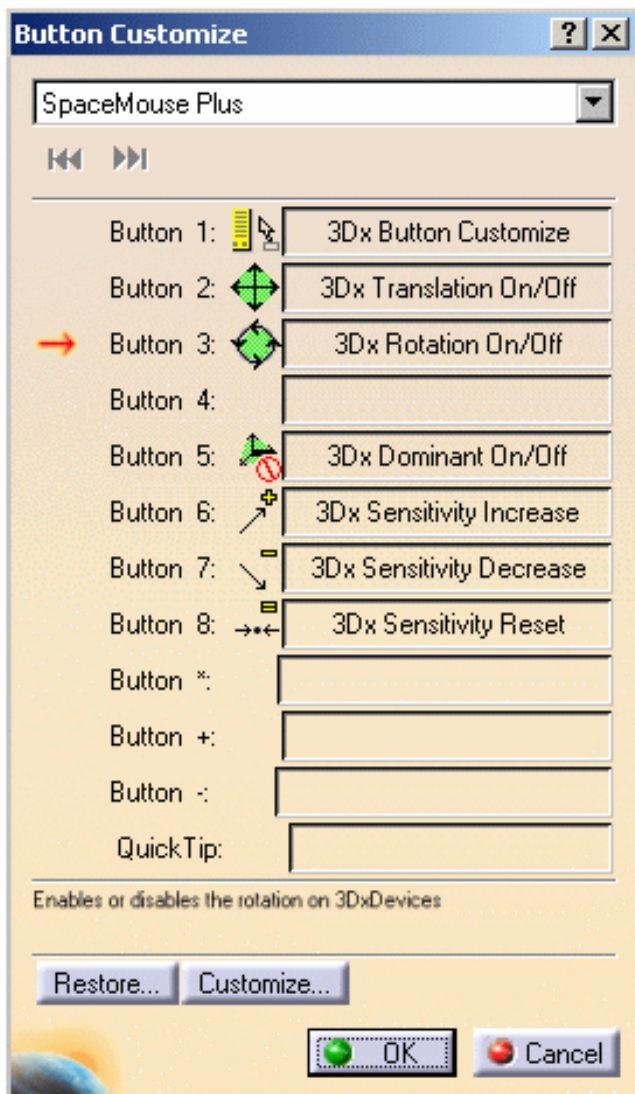
This task explains how to customize buttons which allows you to assign (i.e. map) any given command or macro a device button.

This functionality is available for any application-supported hardware providing the necessary buttons, e.g. Space Mouse and the Spaceball range of 3D input devices, joysticks, Intersense wand...



1. Select the **Opens the Button Customize dialog**  icon from the 3DxDevice toolbar to open the Button Customize dialog box.

The 3DxDevice toolbar can be accessed by selecting **View->Toolbars->3DxDevice**.



Just a few words about the 3Dx Device toolbar which looks like this:



This toolbar contains native integrated commands which let you modify the mode of the 3Dconnexion® device you are currently using. For detailed information, you can browse the following Internet site:

<http://www.3dconnexion.com>

Clicking:



toggles the translation mode on/off



toggles the rotation mode on/off



toggles the dominant mode on/off



increases the sensitivity of the device



decreases the sensitivity of the device



resets the sensitivity of the device to its original state



toggles the fast zoom mode on/off. Note that when the fast zoom mode is on, all other manipulation commands are disabled.

Note that when entering the Sketcher workbench, the rotation mode is automatically switched to "off".

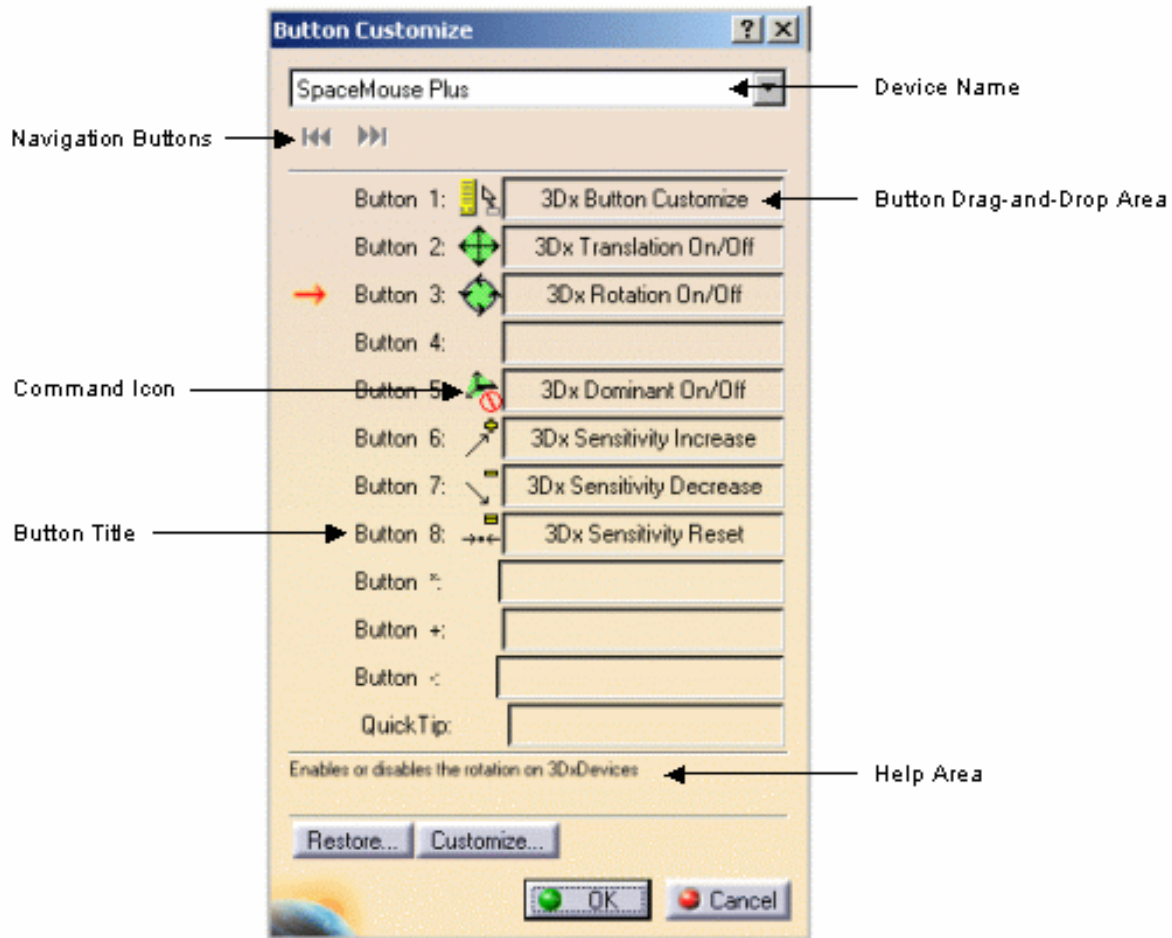
You can also access the Button Customize dialog box using one of the methods below:



- select the **Tools->Customize** command, choose the Commands tab then click the **VR Customize button**
- select the **View->Commands List** command then select the VR Button Customize command
- type **c:VR Button Customize** or **c:3Dx Button Customize** in the power input field then press **ENTER**
- press button "1" of your Space Mouse or Spaceball (if you are using original settings).

2. Use the input device name area on top of the dialog box to select the desired device from the pulldown list in order to modify button mappings.

In our example, the device is a Space Mouse Plus.

Let's have a look at the various information displayed in this dialog box. Click a text to access the corresponding information:



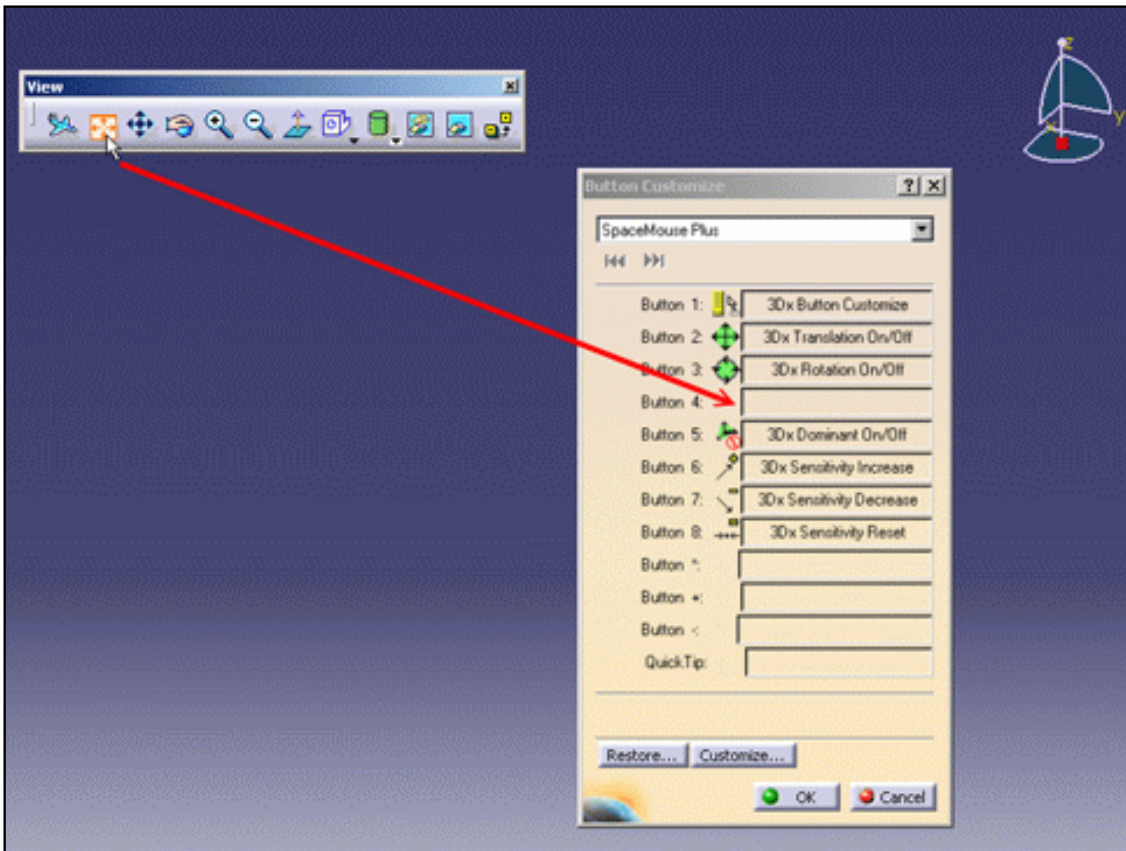
- the navigation buttons will not be grayed out to let you skip backward  and forward  through the pages if the device you chose has more than 12 buttons
- the button title displays the device button title. For instance, with a SpaceBall 4000FLX the tenth button from the top is shown as "Button A"
- the button drag-and-drop area to the right displays the title of the command or macro which is currently assigned to the corresponding device button. For instance, Button 2 has been assigned 3Dx Rotation On/Off
- the command icon displays the mapped command icon (if available) as it is shown on a toolbar
- the help area in bottom part of the dialog box displays a help text for the corresponding mapped command when a device button is pressed (a red arrow identifies in the dialog box the button you press)

3. Map the desired device button to a particular command or macro using one of the four methods detailed below:

**Dragging and dropping from a toolbar**

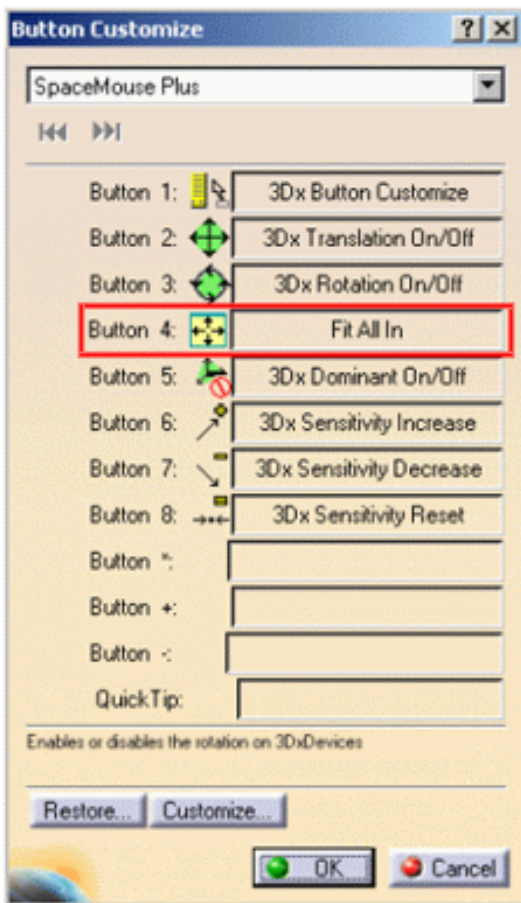


1. Make sure that the toolbar containing the command to be mapped is visible.
2. Press and hold down the left-mouse button on top of the command icon.
3. Drag the icon to the top of the button drag-and-drop area:



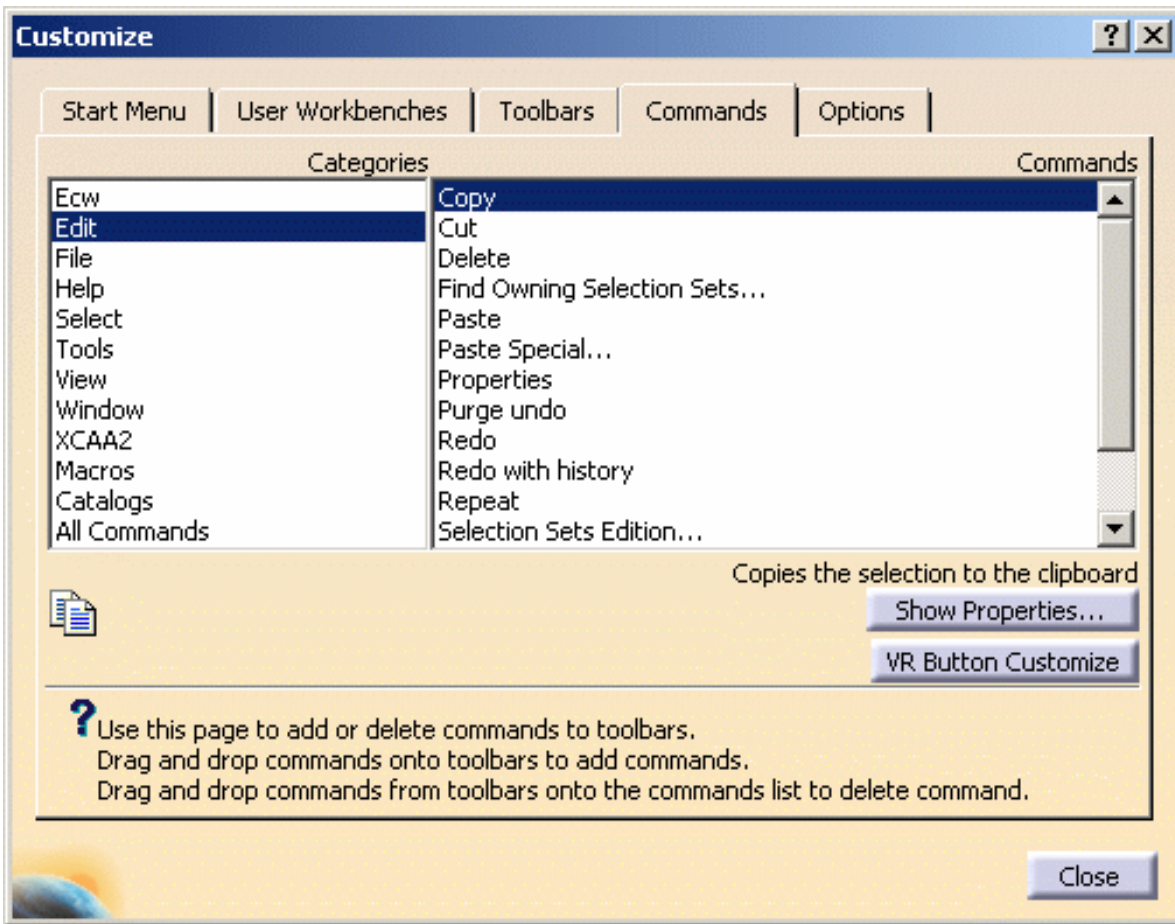
4. Release the left-mouse button.

The command is assigned to the device button:



### Dragging and dropping from the Customize dialog box

1. In the Button Customize dialog box, click the **Customize...** button or select the **Tools** > **Customize...** command to open the Customize dialog box.
2. Select the Commands tab:



3. In the Categories list, select a category to map a command, or select the "Macros" category to map a macro.
4. Select the command or macro to be mapped from the list of commands.
5. Press and hold down the left-mouse button on top of the command or macro entry.
6. Drag the icon to the top of the button drag-and-drop area.
7. Release the left-mouse button.

The command is assigned to the device button.

Note that you can now assign the 3Dx Swap view/tree command to one of the buttons of your Space Mouse or Spaceball in order to swap the focus between the specification tree and the geometry.

### **Dragging and dropping from another button**

You can copy a command or macro already assigned to a device button. To do so:

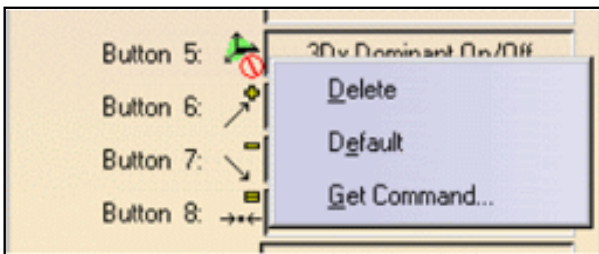
1. Press and hold down the left-mouse button on top of the button drag-and-drop area of the button mapping to be copied.
2. Drag the button mapping to the other button drag-and-drop area the command or macro is to be assigned to.
3. Release the left-mouse button.

The command or macro is now assigned to two different device buttons.

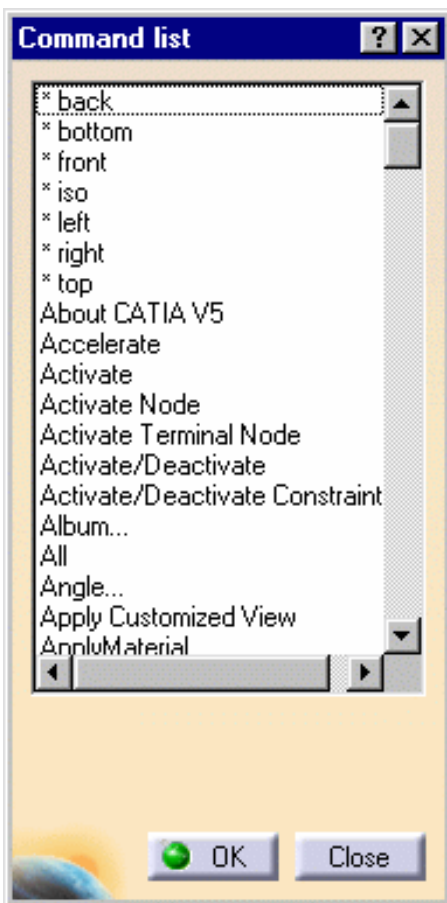
### Using the contextual menu

As an alternative to the above-detailed drag-and-drop methods, it is also possible to use the contextual menu to access the complete list of available commands.

1. Right-click the desired button drag-and-drop area to display the contextual menus:



2. Select **Get Command...**. The Command list dialog box listing the commands available is displayed.



3. In the Commands list dialog box, select the command to be mapped.
4. Click **OK** to validate.

The button drag-and-drop area now shows the title of the mapped command.

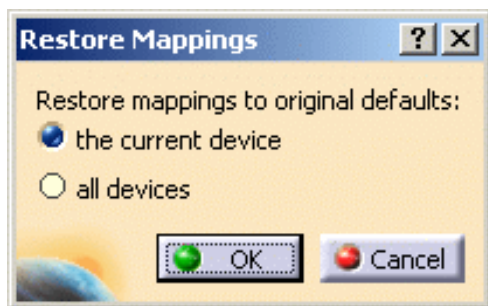


If you wish to remove a button mapping, simply right-click the desired button drag-and-drop area then select the **Delete** contextual command. This will empty the button drag-and-drop area to indicate that no macro or command is currently assigned to this device button.

The contextual menu also lets you restore the selected button to its original settings by selecting the **Default** contextual menu. In case the selected button has no original settings but only a mapped command or macro, the button drag-and-drop area will be emptied (mapped commands or macros are not considered as original settings).

4. Repeat step 3. as many times as needed to map your device buttons.
5. Press the desired device button to activate the command you assigned to it.

**Note:** the **Restore...** button lets you open the Restore Mappings dialog box:



You can select two options:

- *the current device* to restore original button mappings for the device you are currently using
- *all devices* to restore button mappings for all devices to their original commands.

In both cases, if the button mappings have no original settings but only mapped commands or macros, the button drag-and-drop areas will be emptied since mapped commands or macros are not considered as original settings.



# Customizing Standards

About Standards

Customizing Standards for Layers and Filters

# About Standards

This section aims at giving a general overview of standards. For detailed information on standards which are specific to a workbench, you will be asked to refer to the corresponding documentation.

The second part of this section shows a basic example of how standards impact the documents you create.

## Introduction

Standards are defined by your administrator in XML (Extensible Markup Language) files and let you set default values for element properties.

Standards can be customized afterwards according to your needs but bear in mind that unless you are logged as administrator, you will not be able to modify standards.

There are two types of standards:

- default standards provided by Dassault Systèmes. These standards can be customized using the interactive standard editor.

For detailed information on each type of standards, you will have to refer to the corresponding documentation. These default standards are for:

- **General parameters** including
  - default graphic attributes (for Solids, OpenBodies, Points, Lines, Planes and Surfaces) in CATStdMechanicalDesign.xml
  - attributes for layers and filters in
    - CATStdLayerAndFilter.xml and CATStdTypeLayerAndFilter.xml for CATPart documents
    - CATDftStdLayerAndFilter.xml and CATDftStdTypeLayerAndFilter.xml for .CATDrawing documents

Refer to [Customizing Standards for Layers and Filters](#) in this guide.

- attributes for line thickness in CATStdLineThickness.xml.
- **Drafting parameters:** as far as .CATDrawing documents are concerned, the use and administration of their specific standards is detailed in the *Version 5 Interactive Drafting User's Guide - Administration Tasks*
- **Generative parameters:** for detailed information on generative view styles, refer to the *Version 5 Generative Drafting User's Guide - Administration Tasks*.

As far as default standards are concerned, their file name cannot be modified.



- user standards which are created and managed by the administrator.

In both cases, the administrator controls the access to the standard files and can, for instance, prevent users to edit one or more of these files.


Standard files are identified by two runtime environment variables which are:

Environment variable	Description
<b>CATDefaultCollectionStandard</b>	Indicates the name and path of the sub-directory (or sub-directories) containing the default standards provided by Dassault Systèmes. This variable is set by default to <code>installation_folder\resources\standard</code> during the installation procedure
<b>CATCollectionStandard</b>	Indicates the name and path of the sub-directory (or sub-directories) containing the standards customized by the administrator. This means that you should save your customized standards in these sub-directories

For detailed information on environment variables, refer to [What is An Environment?](#) in the *Version 5 Installation and Administration Guide*.

For detailed information on how to set variables, refer to [Customizing Your Environment on Window](#) or [Customizing Your Environment on UNIX](#) (depending on your operating system).

## Impact of standards - Example

 This scenario will show you how to access the pre-defined values for information only using the interactive standard editor and then, how these standards impact the new documents you will create.

 Bear in mind that if you want to be able to modify standards, you must be logged as administrator. An example of customization is given in [Customizing Standards for Layers and Filters](#).



Our example deals with default graphic attributes and thus, this scenario assumes you set up the corresponding CATStdMechanicalDesign file. To do so:

1. Access the following folder to copy the CATStdMechanicalDesign file:

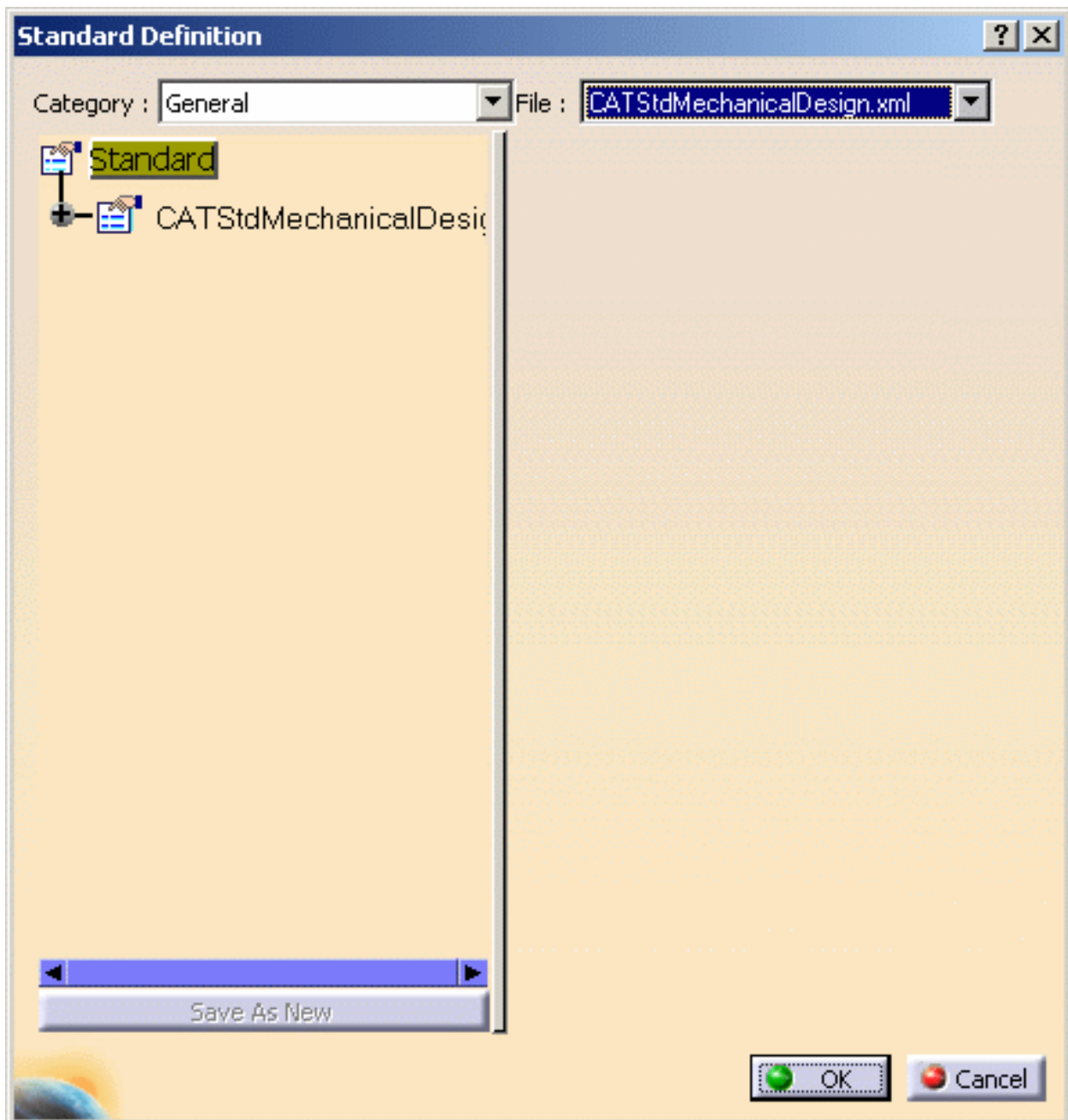
C:\Program Files\Dassault Systemes\B13\intel\_a\startup\templates

2. Paste the CATStdMechanicalDesign file in the following folder:

C:\Program Files\Dassault Systemes\B13\intel\_a\resources\standard

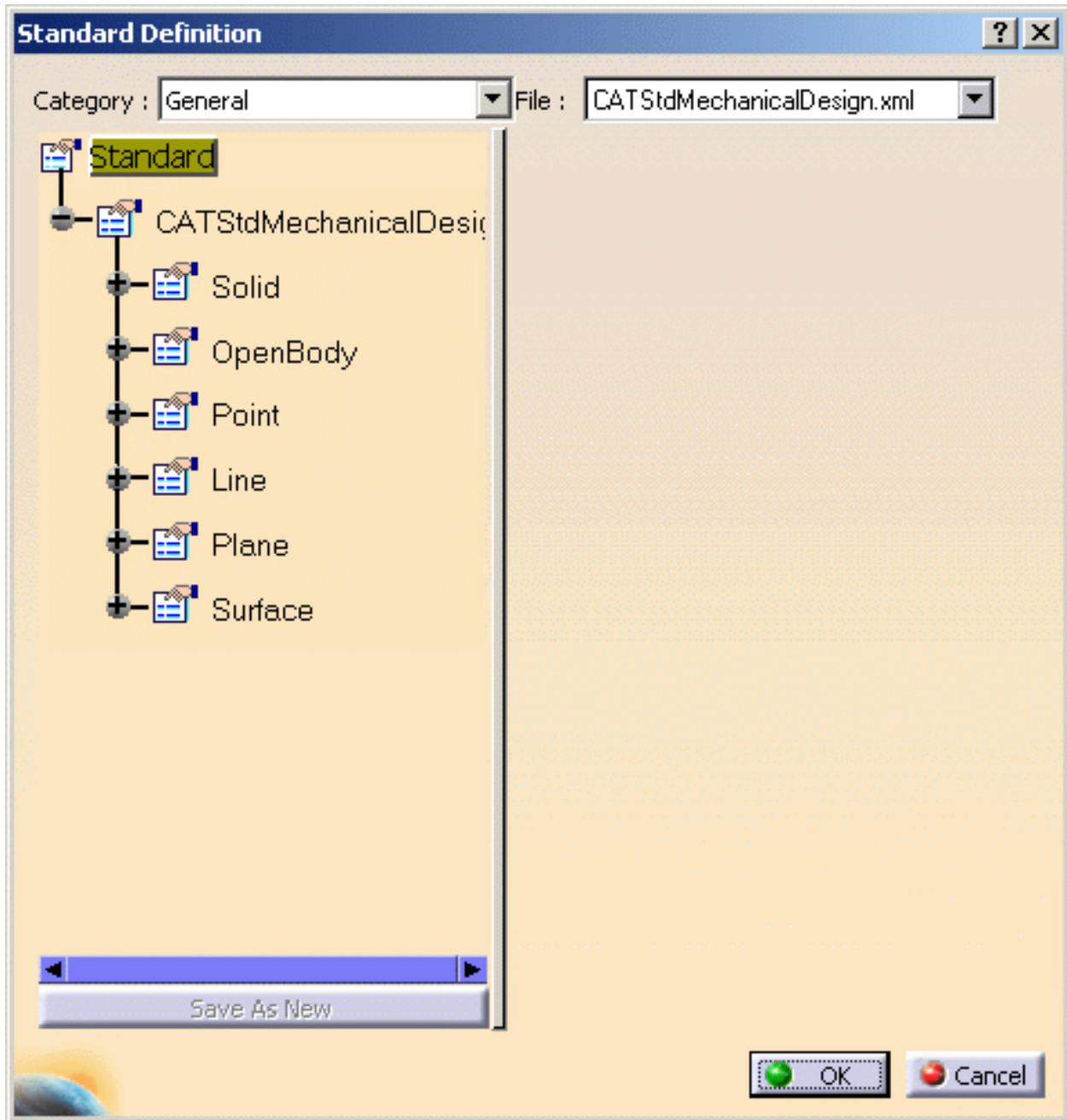


1. Select the **Tools->Standards...** command to access the interactive standard editor:



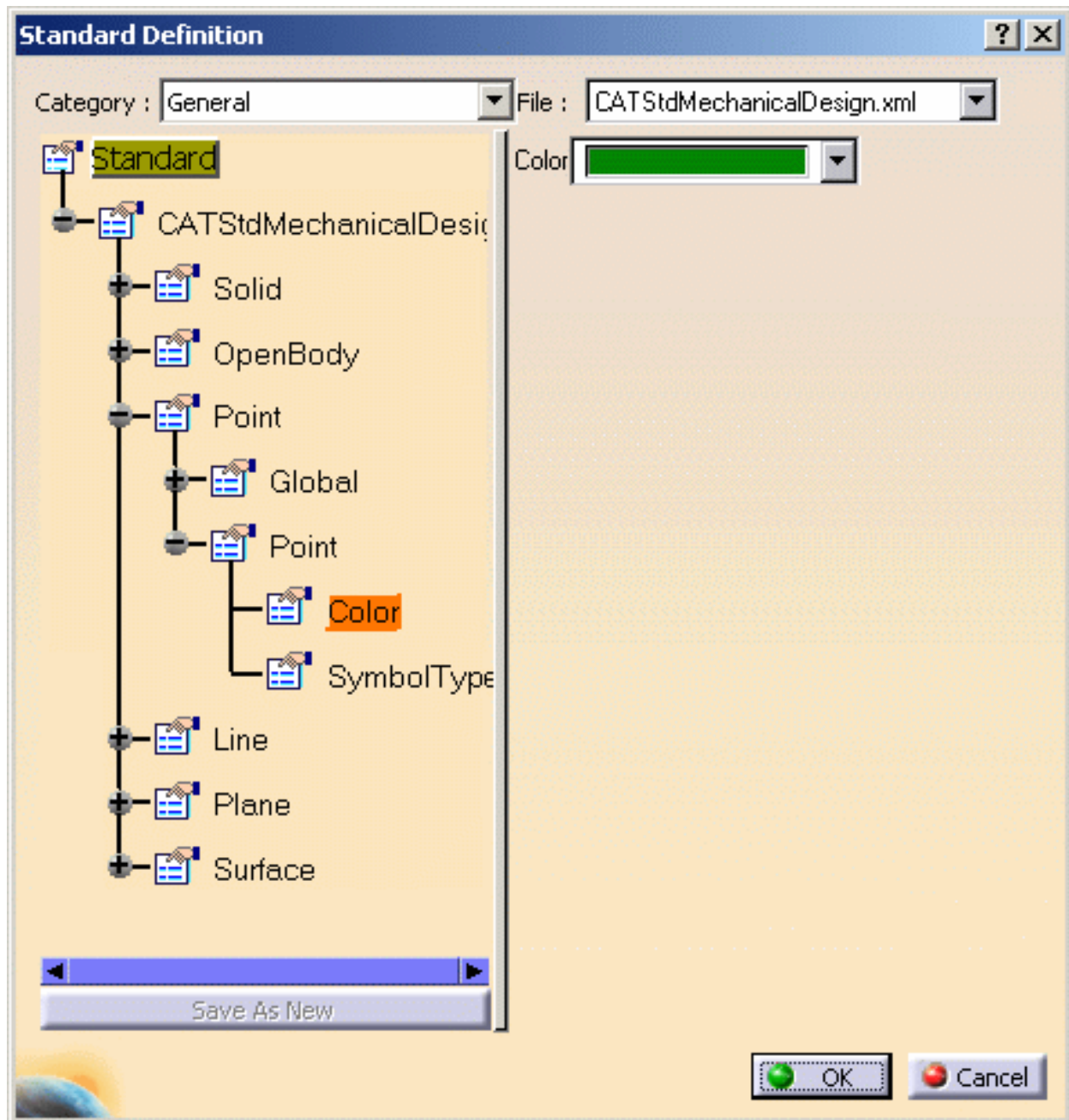
Bear in mind that you cannot modify the data, you access the editor for information only.

- 2.** Select the Category you wish to display: either General or Drafting-specific. Once a category is selected, the corresponding .xml file names are displayed in the File list. For instance, CATStdLayersAndFilters.xml lets you define default properties for the current layer and filter.
- 3.** From the file list, select the CATStdMechanicalDesign.xml file to display the default visualization properties defined by your administrator.
- 4.** Click the plus sign next to CATStdMechanicalDesign node then repeat this step for each node to display the complete xml tree structure. As you can see it in the dialog box below, standards can be defined for Solids, OpenBodies, Points, Lines, Planes and Surfaces:




5. Select the desired item in the tree to expand the node then select the desired element to display the corresponding values on the right.

In our example, we display standards applied to the point default color but you can display standards for all 3D elements such as line type, thickness, etc.



The interactive standard editor shows that the default color for points is green which means that in any new document you will create, points will be created with a green color.

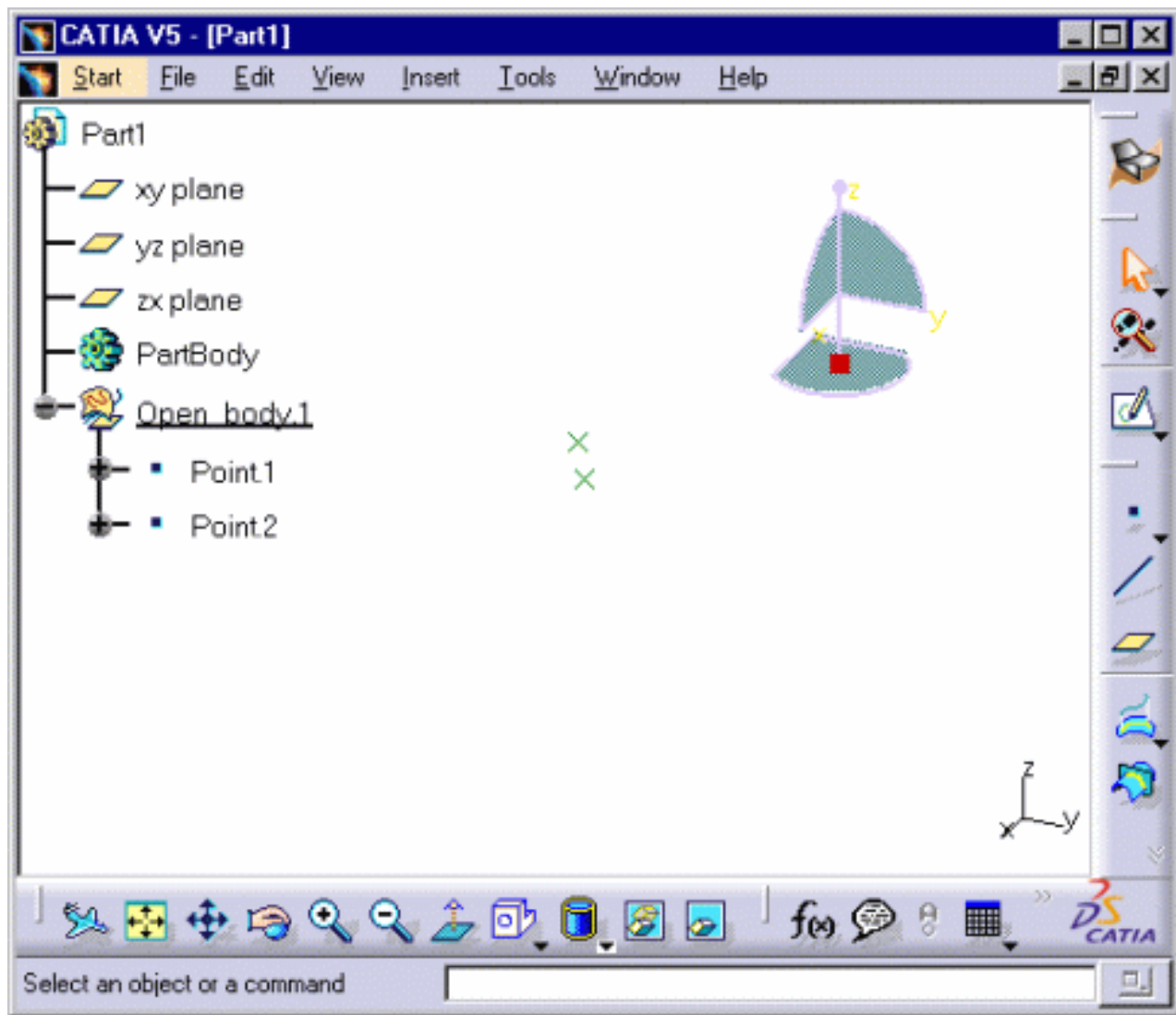
6. Click **OK** to close the interactive editor.

7. Suppose you create a new part document by clicking the **New** icon  or by selecting the **File->New...** command.

Standards impact new CATPart documents only. They do not apply to other documents you may have previously created.

For detailed information on document creation, refer to [Creating New Documents](#) in this guide.

8. From the Start menu, select Shape Design then access the Generative Shape Design workbench.
9. Create as many points as you wish by clicking the Point icon then entering the desired coordinates:



The color of the created points matches the color displayed in the interactive editor.



# Customizing Standards for Layers and Filters



As explained in [About Standards](#), default standards for layers and filters are provided by Dassault Systèmes and referenced by the `CATDefaultCollectionStandard` environment variable which is set to `installation_folder\resources\standard` by default.

Two sets of .xml files are provided:

- `CATStdLayerAndFilter.xml` and `CATStdTypeLayerAndFilter.xml` for part documents
- `CATDftStdLayerAndFilter.xml` and `CATDftStdTypeLayerAndFilter.xml` for .CATDrawing documents.

The default values set in these samples can easily be customized to address your needs, provided that you are logged as administrator.

This scenario aims at explaining how to modify the standard parameters for layers and filters either by using the [interactive editor](#), or by [editing the .xml standard file](#).

We will take `CATStdLayersAndFilters.xml` as an example but the method is identical for editing the other XML files available in the General category.



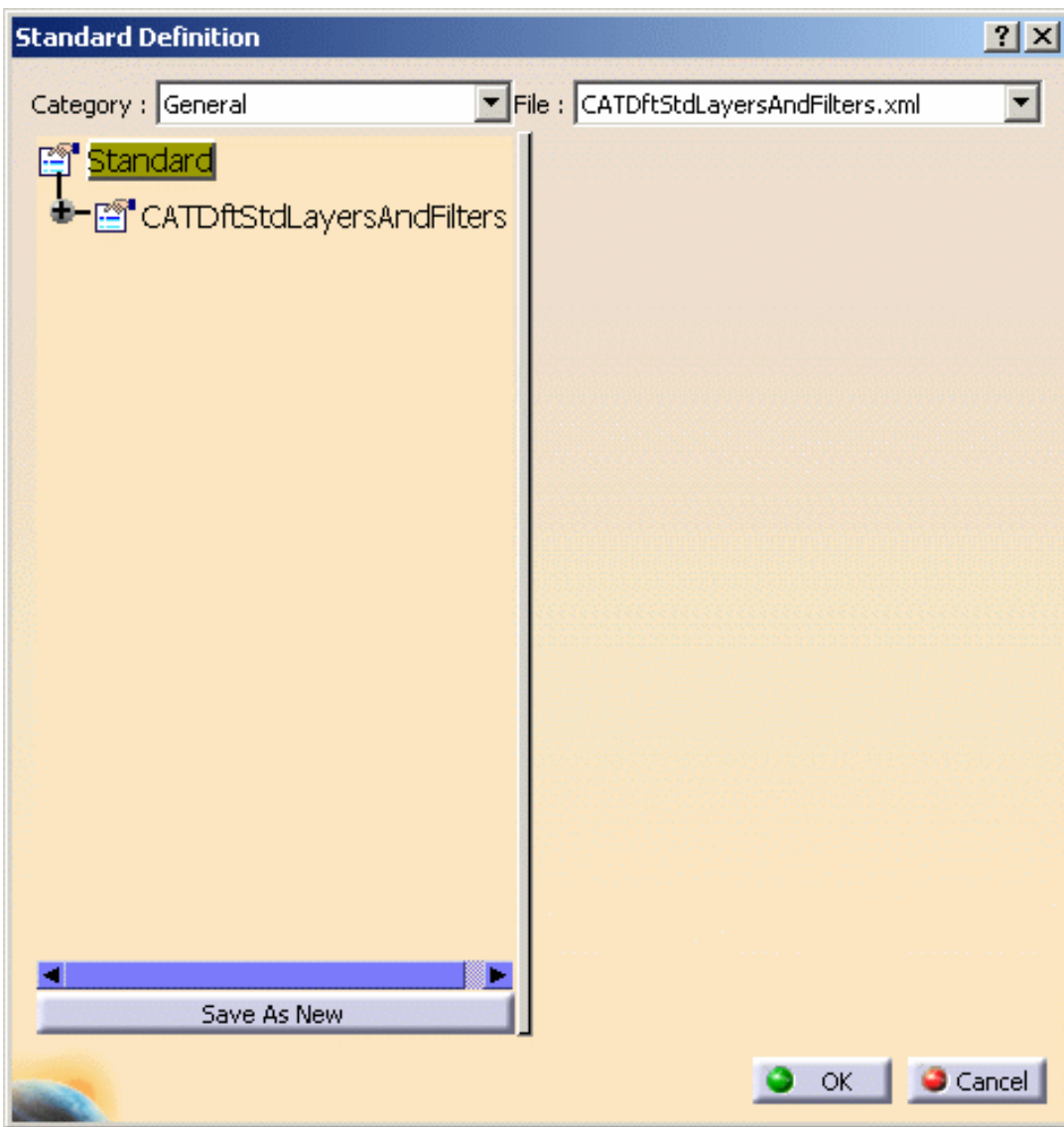
Prior to modifying standards, you need to set the `CATCollectionStandard` environment variable to identify the location where your customized standards will be saved. For detailed information on how to set environment variables, refer to [Customizing Your Environment on Windows](#) (or [on UNIX](#) depending on your operating system).

## Using the interactive editor



1. Select the **Tools->Standards...** command to access the interactive standard editor:



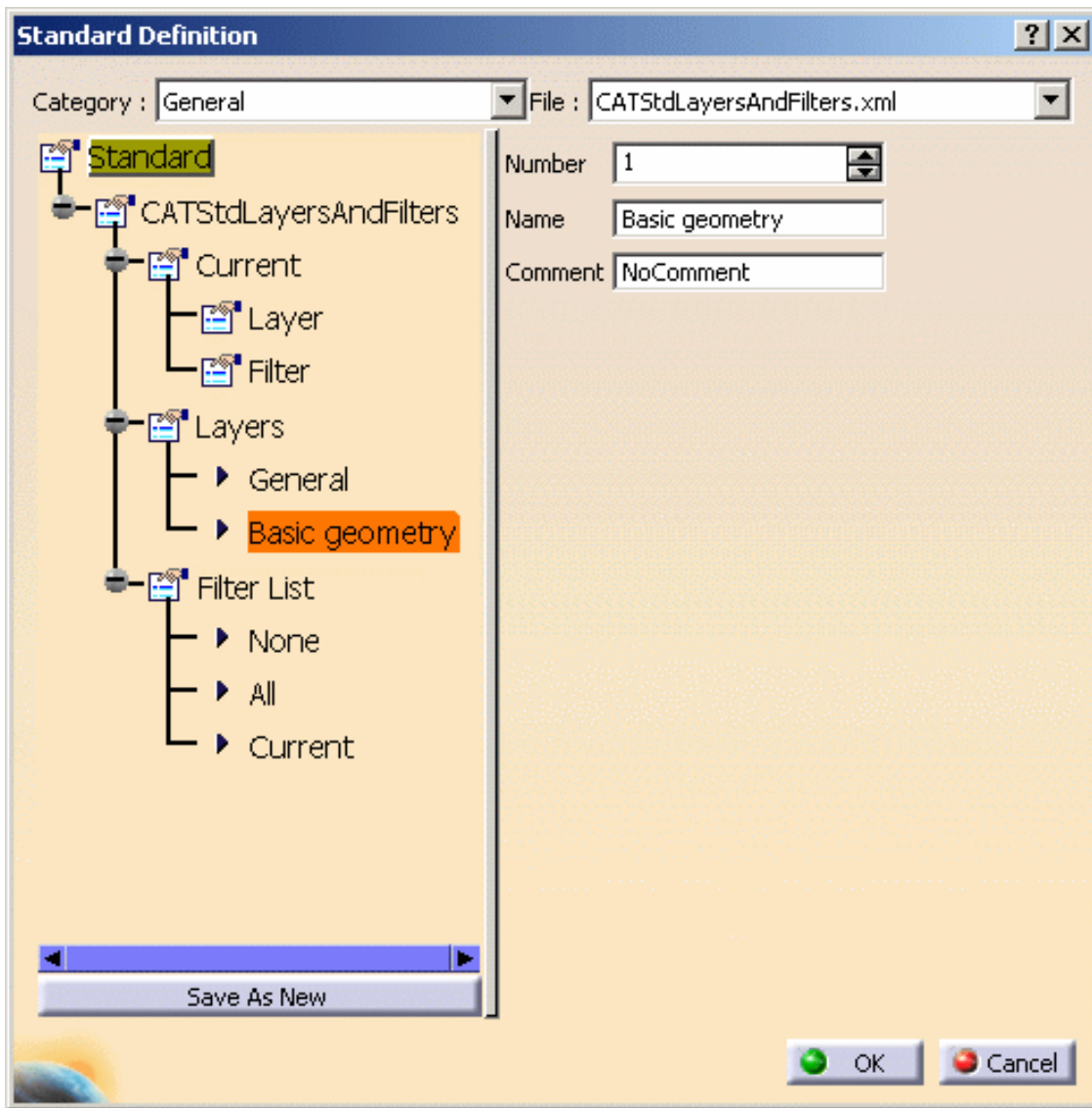


2. Check that the General category is selected then from the file list, select the CATStdLayersAndFilters.xml file.
3. Click the plus sign next to CATStdLayersAndFilters node then repeat this step for each node to display the complete xml tree structure with the values defined by your administrator.

As you can see it in the picture below, standard properties have been defined for the following instances:

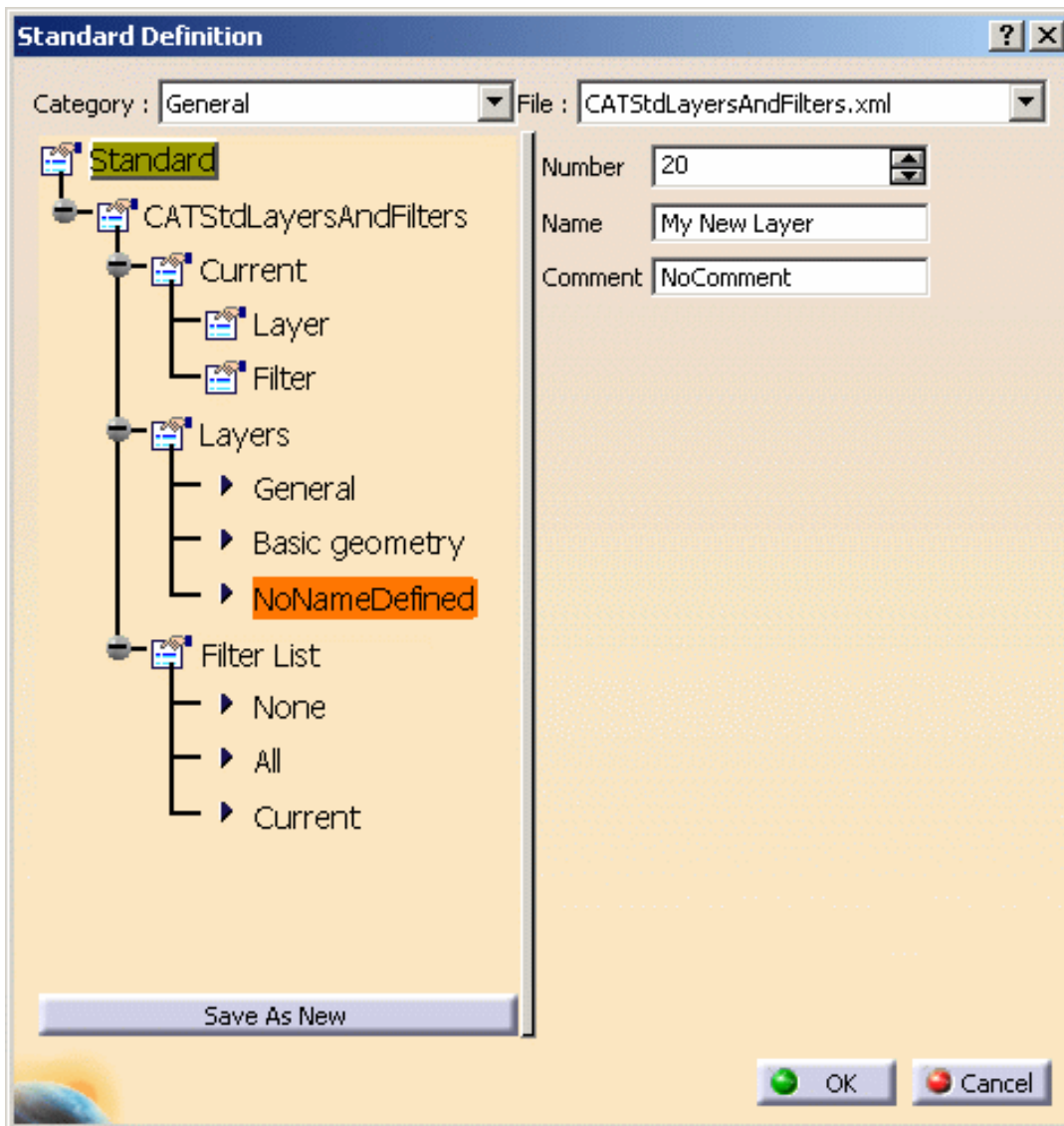
- current layer and filter
- 2 layers named "General" and "Basic geometry".  
Refer to [Assigning Objects to Layers](#) for detailed information on how to use layers
- 3 filters named "None", "All" and "Current".  
Refer to [Using Visualization Filters](#) for detailed information on layer filters.

More precisely, our example shows that the layer Basic geometry has been assigned number "1" without any comment:

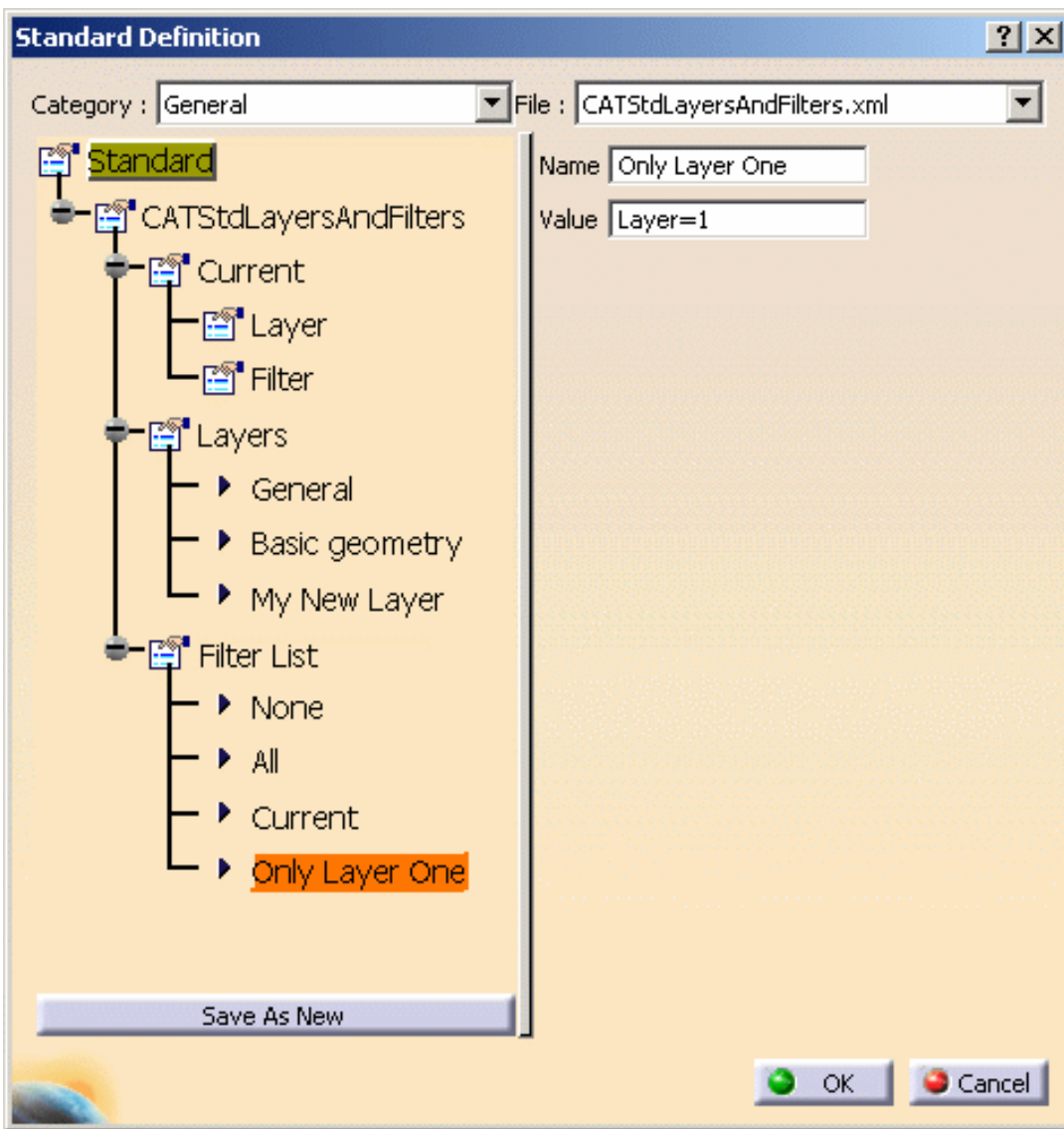


Now let's suppose you want to add a new layer. To do so:

4. Select the Layers node from the tree structure then click the *Add instance* button to the right.  
This creates a new instance in the tree structure which is named "NoNameDefined" by default.
5. Select the created instance, i.e. "NoNameDefined" then assign a number (the value must be comprised between 0 and 999) to the layer, a name of your own choosing as well as a comment (if necessary) in the corresponding fields as shown below:



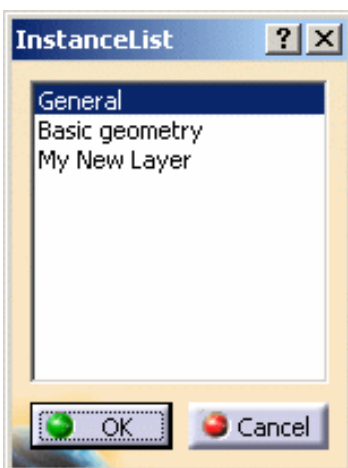
6. To create a new filter, select the Filter List node then click the *Add instance* button.
7. Select the created instance (NoNameDefined) then enter a name and a value:



You can modify any field value anytime simply by selecting the desired instance then entering the new value in the corresponding field.

8. In case you are not satisfied with an instance, select the node under which you want to remove an instance then click the *Remove instance* button (in our example, we choose the Layers node).

This opens the following dialog box which displays the list of available instances for the selected node:



Just select the instance to be deleted then click **OK** to validate.

9. When finished, you can:

- click **OK** to save the modifications in the `CATStdLayersAndFilters.xml` file. A pop-up message will inform you that the modified standard file has been saved in the directory specified when setting the `CATCollectionStandard` variable
- click **Save As New** to save the modifications in a new `.xml` file. Clicking this button opens the Save As dialog box which places you in the directory you specified when setting the `CATCollectionStandard` variable and lets you give a name to the file. This new file will then be accessible from the File pulldown list.

Note: you cannot change the name of standard files included in the General category. For these files, the Save As New button can only be used to generate a new file under another directory but still with the same name.

## Editing the `.xml` file

Standard files are stored in a format called XML (Extensible Markup Language) used to exchange data very easily. XML files can be customized in a simple way, for instance to add new layers and filters, provided that you respect some syntax rules detailed hereafter.



Be careful not to edit the `CATStdTypeLayerAndFilter.xml` file since it contains the layer and filter type definition used in the `CATStdLayerAndFilter.xml` file:

```
<?xml version="1.0" ?>
- <std:typedef name="CATLayer">
  <std:intval name="Number" minval="0" maxval="999">0</std:intval>
  <std:strval name="Name">NoNameDefined</std:strval>
  <std:strval name="Comment">NoComment</std:strval>
</std:typedef>
- <std:typedef name="CATLayersFilter">
  <std:strval name="Name">NoNameDefined</std:strval>
  <std:strval name="Value">*</std:strval>
```



1. Open the `CATStdLayerAndFilter.xml` file in your XML editor. This file is made up of the following parts:

- XML starting tags indicating the format as well as the name and type of the file:

```
<?xml version="1.0" ?>
<!DOCTYPE std [
  <ENTITY TypeDefinitionFile SYSTEM "CATStdTypeLayerAndFilter.xml">
]>
<std:node name="CATLayersAndFilters"
xmlns:std="http://www.dsweb.com/std">&TypeDefinitionFile;
```

- data related to the "**Current**" layer and filter, i.e. the layer to which objects are assigned and the visualization filter set as current. We recommend that you leave these data unchanged to avoid any trouble or misunderstanding. In our example, the value "-1" corresponds to the layer "None" which means that there is no current layer:

```
- <std:node name="Current">
- <std:node name="Layer">
  <std:intval>-1</std:intval>
</std:node>
- <std:node name="Filter">
  <std:strval>All</std:strval>
</std:node>
</std:node>
```

- the list of layers declared in this standard file. This list is surrounded by `<std:node name="Layers">` at the beginning and `</std:node>` at the end. Two layers have been defined in our example, each layer definition containing a **Number** and a **Name** (a **Comment** may be added, if necessary):

```
- <std:node name="Layers">
- <std:typeval name="CATLayer">
  <std:intval name="Number">0</std:intval>
  <std:strval name="Name">General</std:strval>
</std:typeval>
- <std:typeval name="CATLayer">
  <std:intval name="Number">1</std:intval>
  <std:strval name="Name">Basic geometry</std:strval>
  <std:strval name="Comment">NoComment</std:strval>
</std:typeval>
</std:node>
```

- the list of layer filters. This list is surrounded by `<std:node name="LayersFilters">` at the beginning and `</std:node>` at the end. Three filters have been defined in our example, each filter definition containing a **Name** and a **Value**:

```

- <std:node name="LayersFilters">
  - <std:typeval name="CATLayersFilter">
    <std:strval name="Name">None</std:strval>
    <std:strval name="Value" />
  </std:typeval>
  - <std:typeval name="CATLayersFilter">
    <std:strval name="Name">All</std:strval>
    <std:strval name="Value">*</std:strval>
  </std:typeval>
  - <std:typeval name="CATLayersFilter">
    <std:strval name="Name">Current</std:strval>
    <std:strval name="Value">$</std:strval>
  </std:typeval>
</std:node>

```

- the file ends with the tag `</std:node>`.

2. To declare a new layer, you just have to add a sequence `<std:typeval name="CATLayer"> ... </std:typeval>` to the node "Layers". For instance:

```

<std:typeval name="CATLayer">
  <std:intval name="Number">50</std:intval>
  <std:strval name="Name">My new layer name</std:strval>
</std:typeval>

```

Note that the layer number must be comprised between 0 and 999.

3. To declare a new filter, add a sequence `<std:typeval name="CATLayersFilter"> ... </std:typeval>` to the node "LayersFilters".

For instance:

```

<std:typeval name="CATLayersFilters">
  <std:strval name="Name">Only Layer One</std:strval>
  <std:strval name="Value">Layer= 1</std:strval>
</std:typeval>

```

As you can see it in the above example, a filter must be created using a layer number. However, neither filter value nor filter definition needs the layer to be declared.

For instance, you can enter:

```
<std:typeval name="CATLayersFilters">  
  <std:strval name="Name">My Filter</std:strval>  
  <std:strval name="Value">Layer=0 + Layer=50 + Layer=100</std:strval>  
</std:typeval>
```

even if Layer 100 does not exist.

You can even use more complex expressions such as "Layer=0 + (Layer>50 & Layer<100)" but the simpler, the better.

4. When finished, save the modified .xml file.

The new standards for layers and filters will be applied to any documents you will create from now on.





# Customizing Settings

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Analysis & Simulation

AEC Plant

NC Manufacturing

Digital Mockup

Equipments and Systems

Knowledgware

Digital Process for Manufacturing

Ergonomics Design & Analysis

# About Settings



## Types of Data Created by Version 5

Version 5 creates two types of data:

- application data contained in the documents you create
- and setting files which are non-editable

There are two types of settings:

- temporary settings
- permanent settings.

## What Do Settings Files Contain?

*Temporary settings* contain settings of a temporary nature (album screen captures, roll file information,...)

CATTemp contains two folders or directories:

- Album: contains screen captures created using the **Tools->Image->Capture...** command
- CNext01.roll: roll file.

Temporary settings are created in a location referenced by the CATTemp variable.

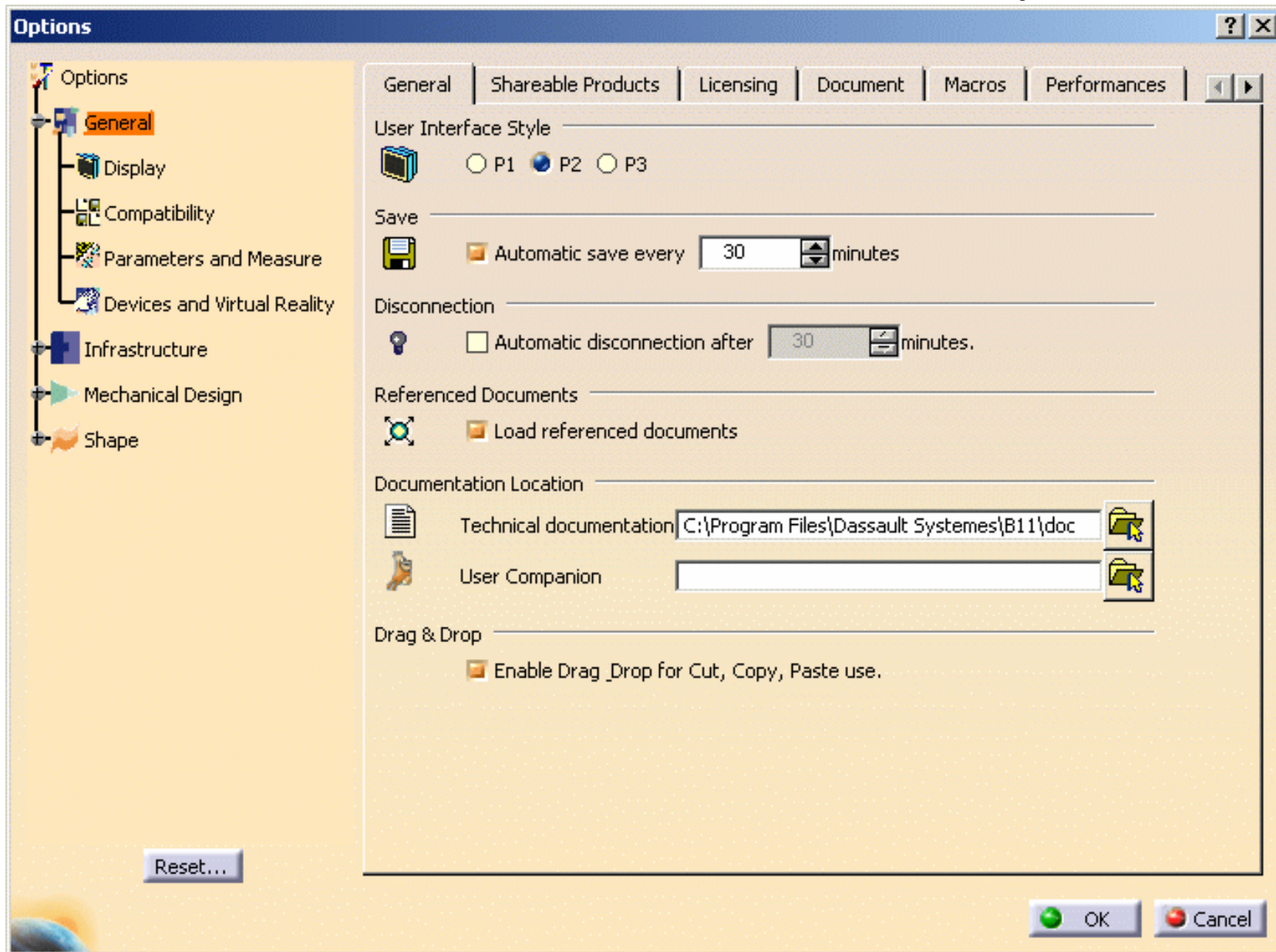
*Permanent setting* files store customization you perform mainly using the various tabs provided by the **Tools->Options...** command. For example, application window customization, background colors, part and print settings, etc.

Permanent setting files are identified by the suffix: \*.CATSettings, and are created in a location referenced by the CATUserSettingPath variable. Deleting either types of files deletes your customization.



## How Do You Specify Settings?

You specify settings using the **Tools->Options...** command which displays the Options dialog box:



The left-hand column contains a list of categories used for organizing the different groups of settings. There are general settings for all configurations and products, and settings for each type of configuration installed. The category names are the same as those listed on the **Start** menu.

To access the settings for a specific configuration, click the "+" to display the subcategories. Clicking on the subcategory displays the settings tabs for that subcategory.

## Where Are Settings Files Located on Windows?

The location of settings files on Windows platforms is inspired by the general data and settings management requirements operating on the Windows 2000 platform, which provides an underlying infrastructure allowing you to separate user data, user settings and computer settings.

The mechanism used is the CSIDL value mechanism. This implementation allows:

- your permanent settings (CATSettings) to roam as part of your user profile (CSIDL\_APPDATA)
- your temporary settings (CATTemp, etc.) to be still stored in the user profile, but prevents them from roaming (CSIDL\_LOCAL\_APPDATA).

The following table will help you determine where your settings are now located:

### Windows 2000/XP

VARIABLE	LOCATION
CATUserSettingPath	C:\Documents and Settings\user\Application Data\DassaultSystemes\CATSettings
CATTemp	C:\Documents and Settings\user\Local Settings\Application Data\DassaultSystemes\CATTemp
CATCache	Obsolete
CATReport	C:\Documents and Settings\user\Local Settings\Application Data\DassaultSystemes\CATReport
CATErrorLog	C:\Documents and Settings\user\Local Settings\Application Data\DassaultSystemes\CATTemp\error.log
CATMetasearchPath	C:\Documents and Settings\user\Local Settings\Application Data\DassaultSystemes\CATTemp
CATW3PublishPath	C:\Documents and Settings\user\Local Settings\Application Data\DassaultSystemes\CATTemp

### **CSIDL Values in Environment Variable Paths**

The values:

`C:\Documents and Settings\user\Application Data`

are the defaults on these Windows platforms for the CSIDL\_APPDATA values.

The values:

`C:\Documents and Settings\user\Local Settings\Application Data`

are the defaults on these Windows platforms for the CSIDL\_LOCAL\_APPDATA values.

### **Location of Settings Files on UNIX**

Permanent settings are stored in the CATSettings directory in your home directory; temporary settings are stored in the CATTemp directory, also in your home directory.

### **Administrator Settings**

If you start a session in administrator mode using a specific environment, you can lock settings so that other users running a session with the same environment inherit those settings **and cannot change them**. For full details how to do so, refer to [Locking Settings](#).

### **Default Settings and Recommended Settings**

Default settings are provided by applications. However, we recommend the following settings to enhance performance:

- General category, Performance tab:
  - deactivate occlusion culling
  - level of details (static) = 1 to 1.2
  - level of details (while moving) = 10 to 11
- General category, Visualization tab:
  - deactivate the graduated color background
  - deactivate preselect highlighting
  - deactivate the manipulation bounding box
- Product category, Cache Management tab:
  - work with cache management
  - cache size = 600MB

Selecting **View->Render Style->Shading** to work with shading without edges will also enhance performance.

### **Accessing Settings Without Running a Session**

You do not need to start a session to be able to access your settings.

#### **On Windows**

1. Change to the default folder in which you installed the product.

On Windows, the default folder is:

`C:\Program Files\Dassault Systemes\B13\intel_a\code\bin`

2. Enter the command:

`CATOptionsMgt`

You can also access the Options dialog box using the **Start->Programs->CATIA Tools** menu, and running the **Settings Management V5R13** command.

The Options dialog box is displayed.

This function is useful for administrators because it allows you to set up user settings without having to start an interactive session first.

Specify your settings as usual.

**On UNIX**

1. Log on as root or end user.
2. Enter the command:

```
/usr/DassaultSystemes/B13/OS_a/code/command/catstart -run CATOptionsMgt
```

# General

[General](#)  
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[Printers](#)  
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[Devices and Virtual Reality](#)



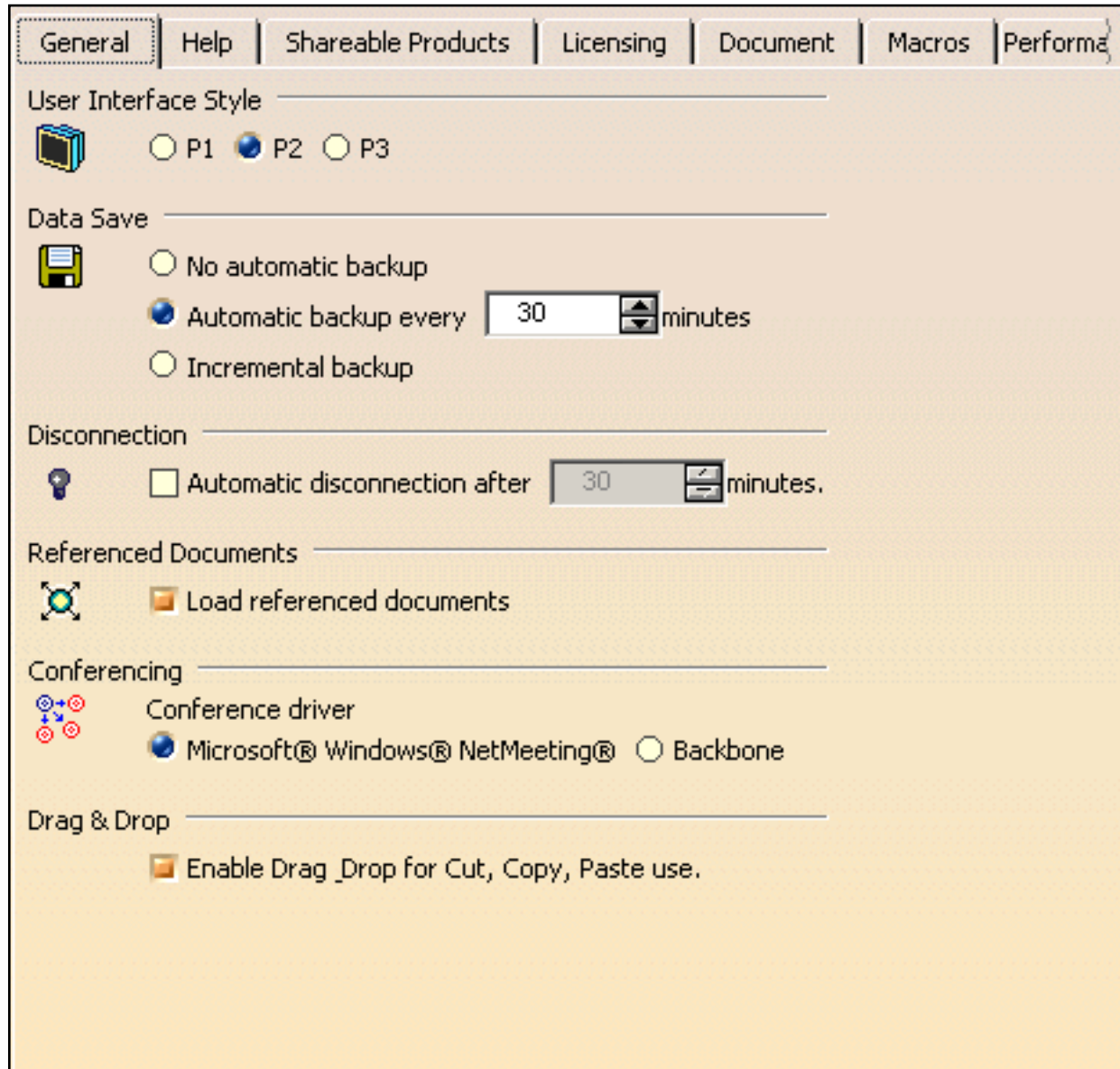
# General



This task explains how to customize general-purpose settings.



1. Select the **Tools->Options...** command.
2. Select the General category (in the tree to the left), then the General tab:



## User Interface Style

- CATIA - P1: sets the P1 user interface style, and is available for both CATIA - P1 and CATIA - P2 users
- CATIA - P2: sets the P2 user interface style, and is available for CATIA - P2 users only
- CATIA - P3: sets the P3 immersive user interface style and features, and is available for P3 products/configurations only.

## Data Save

The option **Automatic backup every ... minutes** allows you to specify a save frequency other than the default frequency (30 minutes). This option is activated by default.

The purpose of this option is to automatically save your data to allow you to recover it after a crash. When a crash occurs, and you restart your session, the Warm Start dialog box asks the following question:

Do you want to recover previous session data?

If you choose "Yes", your documents will be recovered in the state in which they were last saved by the automatic save option. Any data created after the last automatic save will not be restored.

If you choose "No", a new document will be created when opening the session.

**No automatic backup:** documents are not saved automatically.



You may encounter, when working on a document, a dialog box containing the following message:

Command interrupted

When this occurs, we recommend that you click OK to continue (a check is launched on your data in the background). Then, you should immediately save your document(s) and exit then restart your session.

We also recommend the same method if a dialog box appears containing the message: "This handler is null"

## Incremental backup

### Autosave Feature Constraints

Until now, warm start meant simply automatically saving all documents every xx minutes. Autosave operations on large amounts of data can take a long time, even lasting several minutes, and all interactions since the last autosave are lost. The autosave feature can also be deactivated by users who prefer to save their documents frequently and manually to avoid losing data.

### Incremental Warm Start Principles

All open documents are stored in a temporary directory, and all modifications to the document are logged in a log file. These operations involve a minimum inconvenience for the end user when opening documents (direct copy without loading), and the logging of document modifications at each interaction is not perceptible.

Regarding the restore behavior:

- after a crash, all documents are opened from the temporary directory
- all modifications stored in the log are replayed
- all necessary editors are opened
- restoring data could take a long time:
  - all modifications made by users (including test/try, mistakes, etc.) are replayed
  - time is needed to replay necessary updates.

A slight risk is involved: the modification log needs to be complete, otherwise there is a risk of restoring corrupted data.

The incremental warm start mechanism is available for the CATPart, CATProduct and CATDrawing document workbenches.

**Note:** movements of components using compass are not retrieved when translation/rotation is the last operation before a crash.

### Example

1. Start a Version 5 session, select the **Tools->Options...** command, click the General tab, and check the Incremental backup option, then click OK to save, and restart Version 5.
2. Continue making modifications to the current document until a crash occurs.
3. Kill your Version 5 session.

4. Restart Version 5.
5. After restarting, you are prompted as follows:

Previous session abnormally ends,  
do you want to recover data?

Click OK to recover data.

A progress bar is displayed while the data is being restored, and a confirmation window is displayed after the data has been restored:

Data successfully restored, please save all to activate incremental warmstart again

6. Check visually that your data was correctly restored.
7. You must save all data to reactivate the incremental warm start feature.  
If you do not do so, the incremental warmstart feature will be deactivated.
8. Continue working.

## Partial Activation

As specified above, certain workbenches do not support the incremental warm start feature. When you switch to a workbench which does not support it, a message will inform you that the workbench does not support this feature, and warn you that it is deactivated.

The same also applies to certain commands: when you start the command, you will be warned that the incremental warm start feature is deactivated.

To reactivate the warm start feature, go into a workbench supporting the incremental warm start feature, and save all documents. If you close all documents, the warm start feature will be activated for the next opened document supporting the incremental warm start. You can close or save one V5 document, but if there is always only one dirty document in the session, the warm start will not be reactivated.

Generally speaking, if in doubt, save all documents to be sure the warm start is reactivated. It enables you also to empty the modification logging files which can be large after a long time.

## Disconnection

The option **Automatic disconnection after ... minutes** lets you activate automatic disconnection of your Version 5 session after a user-defined duration (in minutes) without program use.

## Referenced Documents

By default, the **Load referenced documents** option is checked. This means that when a father document is loaded, the child documents it points to are also loaded. However, if you uncheck this option, only the father document is actually loaded when you open it. This may be useful for reasons of performance and efficiency. A typical case of this would be a CATProduct or CATDrawing document, i.e. the father document pointing to CATPart or model documents i.e. the child documents.

Once a father document has been loaded you cannot load or unload its child documents as changing the option cannot apply retroactively. You must close the father document, check or uncheck the **Load referenced documents** option then reopen the father document.

If you open a father document containing unloaded child documents you can load one or more of these documents using the File->Desk command (see [Using the FileDesk Workbench](#)).



Note that when loading objects pasted "As Result With Link" or external references, the pointed documents containing the referenced geometry are not loaded and this, even if the **Load referenced documents** option is checked. These pointed documents will be

identified by the symbol  in the specification tree to indicate that they have been found but not loaded.

## Conferencing

If you installed a license for a DMU Navigator product, you will be able to run conferencing sessions: a conference host initiates a conference and invites other users to join the conference as guests.

The actions replicated during a conference are the following:

- workbench transition
- object selection
- viewpoint modification (zoom, rotation)
- 3D annotations (creation, modification and deletion)
- 2D annotations (linked to cameras) creation, modification and deletion.

Conferencing is available on the Windows and UNIX platforms, and is based on two underlying prerequisites:

- NetMeeting (provided with Windows)
- the Communications Backbone (provided with the Version 5 infrastructure), required for conferencing on UNIX.

For more information, refer to the *DMU Navigator User's Guide*.

## Drag & Drop

Enables dragging and dropping in the viewers (for copying, pasting or cutting purpose only).

For more information, refer to [Dragging and Dropping Icons onto Objects](#) and [Dragging and Dropping Objects onto Objects](#) in this guide.

**Note:** this drag and drop option does not apply to toolbar customization.

3. Click **OK** to confirm.



# Help

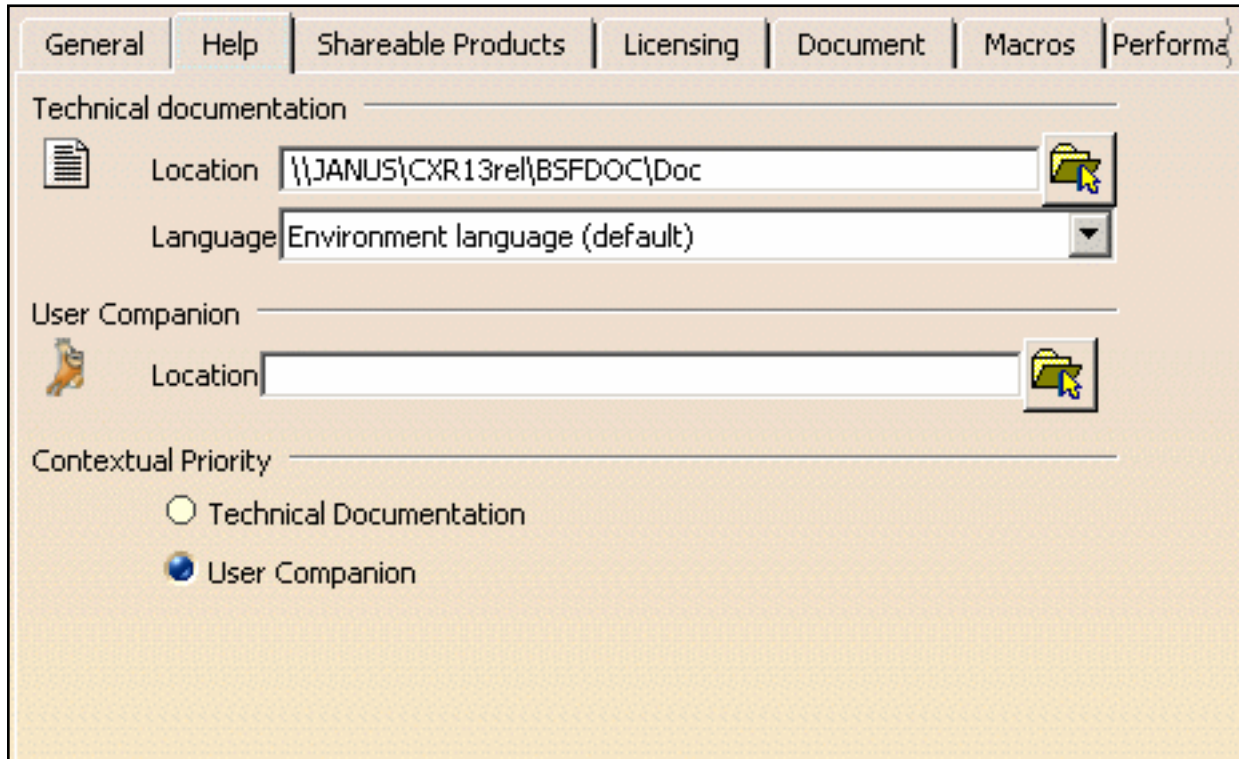


This task explains how to customize contextual help-purpose settings.



1. Select the **Tools->Options...** command.


2. Select the General category (in the tree to the left), then the Help tab:




## Technical documentation




The **Location** field displays the current location of the online documentation. By default, the online documentation is located in:

C:\Program Files\Dassault Systemes\B13doc

Enter the path where the documentation is installed or click the Path selection  icon

to navigate to the desired location then click the  icon to add the selected directory or file to the path (displayed to the right).

Once a path has been added to the list, you can select it then click the following icons to:

-  move it up in the list
-  move it down in the list
-  remove it from the list.

Please note the following information:

- if you install the documentation on the network, you can map a drive and enter the appropriate path in this field. UNC (Universal Naming Convention) syntax is also supported. Refer to [Doc Installation Path](#) for more details.
- if the online documentation is installed on a http server, the installation path must be surrounded by quotes, for instance "[http://myserver/B13doc](#)".

Note also that the Path Selection dialog box lets you select several different paths to interactively concatenate them. On Windows, you can perform the concatenation manually. In that case, the separator to be used is ; and not : . Furthermore, when you select several paths using the Path Selection dialog box, the concatenated paths are represented between " ", for example, like this:

"J:\V5R13DOC\Doc";"F:\V5R13DOC\Doc"

If you indicate a wrong path, an error message is displayed to inform you that the previous path will be kept.

The **Language** pulldown list lets you select your documentation language: "Environment language (default)" or any language you selected when installing the software. For detailed information on installing Version 5, refer to [Unloading Your Product Code on Windows](#) (or [on UNIX](#)).

The default environment language corresponds to the language you define in the [Regional Settings tab](#) (on Windows) or in the [LANG variable](#) (on UNIX).

To access a documentation in the same language as the software, you need to install the online documentation in the appropriate language. Refer to [Installing the Online Documentation After Installing the Software on Windows](#) (or [on UNIX](#)).


However, note that the documentation language you choose is independent from the language of your session or of your operating system. For instance, you may run a session in Japanese and access an English documentation, provided that you installed it on your computer.



## User Companion

The **Location** field displays the path to the User Companion (provided that the Computer-Based Training module and the Companion have been previously installed). You can enter

another path directly in this field or click the Path selection  icon to navigate to the

desired location then click the  icon to add the selected directory or file to the path (displayed to the right).

## Contextual Priority

This area enables you to specify which documentation ("Technical Documentation" or "User Companion") will be accessed in priority when pressing the F1 key to get contextual help.

By default, the User Companion takes priority.

3. Click **OK** to confirm.



# Licensing



Refer to [Reserving Static Licenses Using the License Manager](#) for detailed information about how to reserve configuration and product licenses using the License Manager.

# Document

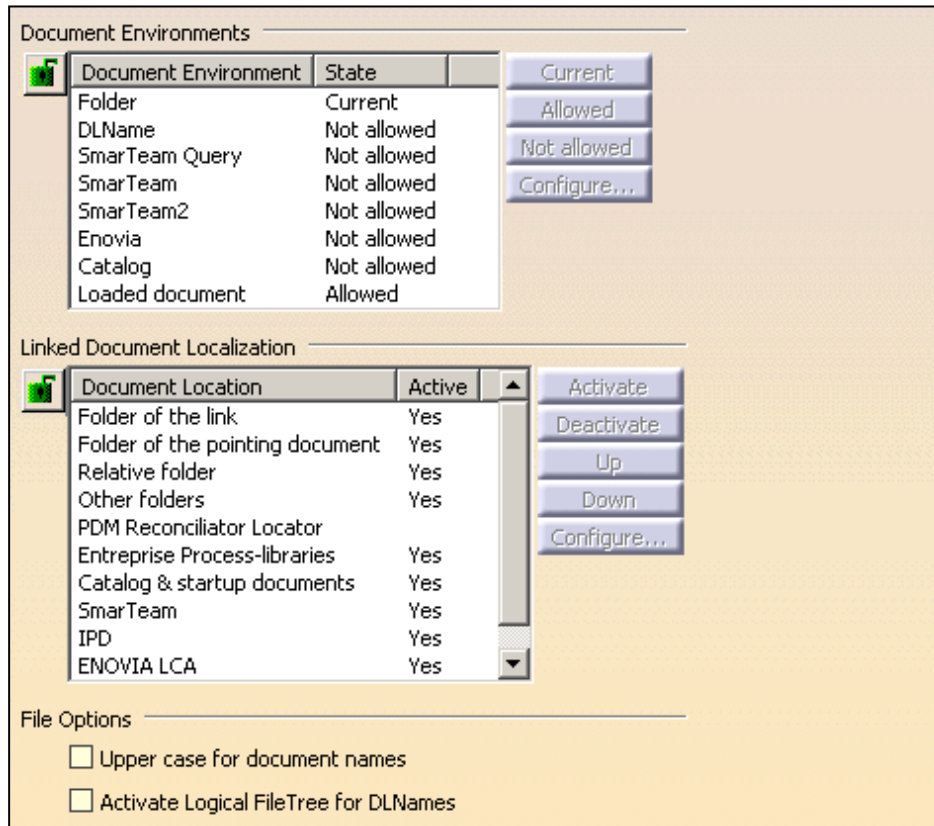


This task explains how to manage [document environments](#) as well as [linked document localization](#).

For detailed information about the administration of document environments based on the DLName mechanism, refer to [Administering Data Using the DLName Mechanism](#).



1. Select the **Tools->Options...** command.
2. In the **General** category, click the **Document** tab:



## File Options

You can activate at anytime the "Upper case for document names" option which converts file names to upper case when saving documents.

This conversion will be taken into account when using the following commands:

- **File->New...**
- **File->Save As...**
- **File->Save Management...**
- **File->Send To->Mail/Directory.**

The File Options group box also lets you organize DLNames in a logical way. Refer to [Activating Logical FileTree for DLNames](#) further in this scenario for detailed information.

## Document Environments

The Document Environment is a way of accessing your documents. When you access the Document tab, the following document environments are displayed:

- **Folder:** default document environment in which the dialog access any folder containing the documents
- **DLName:** document environment which lets you restrict the access to specific folders referenced by logical names ("DLNames"). Each folder is assigned a logical name.  
Any modification of a path impacts the DLName once, thus avoiding to search for and modify manually any occurrence of this path.  
The administrator may also define the folders in which users are allowed to work
- **SmarTeam:** lets you access SmarTeam projects stored in SmarTeam - CATIA Integration
- **SmarTeam Query:** lets you access SmarTeam projects stored in SmarTeam - CATIA Integration using queries
- **SmarTeam2:** lets you access documents stored in SmarTeam - CATIA Xtended Integration
- **Enovia:** lets you access documents stored in ENOVIA V5
- **Catalog:** lets you access documents stored in catalogs
- **Loaded document:** lets you access documents loaded in the current session.

A document environment may be assigned one of the following states:

- **Current:** sets the selected environment as the current document environment and defines the dialog box (i.e. File->Open, File->Save, Edit->Links, etc.) that will be displayed when accessing your documents
- **Allowed:** indicates that the environment may be set as "Current" or "Not allowed"
- **Not allowed:** indicates that the environment may not be set as "Current". You have to assign it the "Allowed" state before selecting it as your "Current" environment.

Note that the *Catalog* and *Loaded document* environments cannot be set as "Current".



If your administrator has locked the document environments, the **Allowed** and **Not allowed** buttons are grayed and thus cannot be selected by the user. Only the **Current** and **Configure...** buttons are active.

### Setting the DLName environment as Current

3. In the Document Environments column, select "DLName".

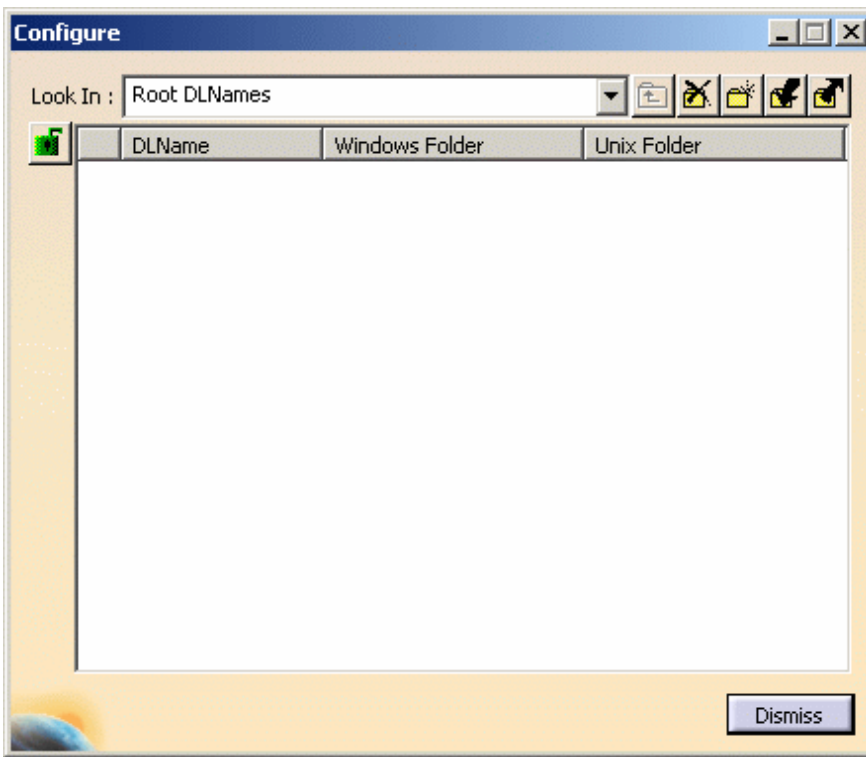
4. Select successively the **Allowed** and **Current** buttons.

DLName is now defined as your current document environment as indicated by the "Current" value in the State column.

### Defining DLNames

Now that you have set the DLName environment as your current environment, you have to create the DLNames you will use.

5. Click the **Configure...** button to open the Configure dialog box which lets you add or remove DLNames:



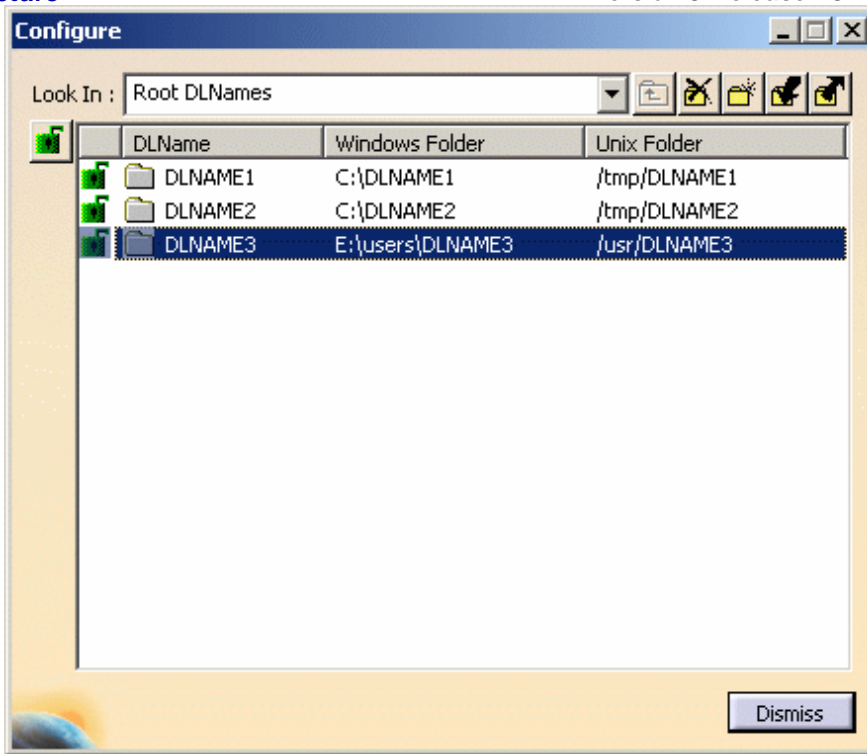
**Note:** the UNIX Folder column is displayed on Windows only when running Version 5 in administrator mode. On UNIX, only the Windows folder is displayed. When running Version 5 in user mode, only the column corresponding to your operating system will be displayed (for instance, "Windows Folder" when working on Windows).

The Lock column is displayed in both administrator and user modes:

- means "locked" (in user and administrator mode)
- means "locked at administrator level" (in administrator mode)
- means "no lock" (in administrator mode). In user mode, there is no lock symbol.

6. Click the button or right-click then select the **New** command to create a new DLName.

A default name and a default folder are assigned to the new DLName as shown below (you can create as many DLNames as necessary):



7. If you are authorized by your administrator (i.e. when no lock appears opposite the DLName), you can modify the values displayed. To do so, click the desired value to be able to edit it then type the new data.

In **administrator mode**, you can modify the DLName and the associated paths as you wish.

In **user mode**, you can create a root DLName. However, do not forget that some rules apply if you want to modify names and folders:

- when changing the folder of a sub-DLName, the new folder must correspond to a folder of the father DLName
- the name of the last folder (on UNIX and Windows) must be the one of the DLName.

Therefore, be extra careful when renaming a DLName since this modification will impact the name of the folder and of the sub-folder(s), if any!

Any modification of a root DLName path is controlled by the administrator which may prevent you from doing so.

As far as sub-DLNames are concerned, you cannot modify the path of a sub-DLName because this path contains the name of the sub-DLName and depends on the location of the parent DLName.

To modify the Windows folder of a DLName, just select the folder to be modified in the Windows column then choose the **Browse** contextual command: this opens a dialog box that lets you navigate to the new folder. After clicking **OK**, this new folder will then replace the former folder in the Windows column.





Note that you can include variables in DLNames using the syntax `${VARIABLE}`.


For instance:


`C:\users\${MODEL}\publish` where `${MODEL}` is a user-defined variable.


- 8. If you are running in administration mode, you can lock or unlock each DLName individually simply by clicking the green symbol opposite the DLName.

Locking a DLName changes its state from  to  (and inversely when you unlock the DLName).

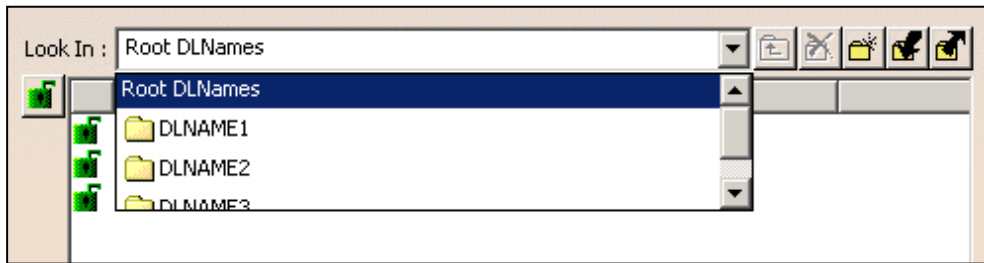
As an administrator, you can also click the general lock to prevent end users from defining and modifying DLNames. Therefore, the following commands will not be available: **New**, **Browse**, **Delete**, **Import** and **Export**.

- 9. When finished adding DLNames to your list, you can then click the  button or use the **Export** contextual command to save your list of DLNames as a .txt file in the appropriate location using the Export DLNames dialog box.

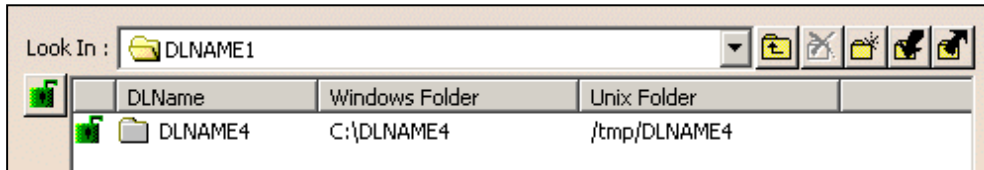
You can make as many lists as you like and, for further use, import the list whenever you want to. To do so, click the  button or use the **Import** contextual command then select the list to be imported in the Import DLNames dialog box.

- 10. If you want to remove a DLName from the list, select it then click the  button or use the **Delete** contextual command.


Once DLNames have been created, they are displayed in the "Look in" pulldown list:



You can then use this list to display sub-DLNames, if any. In our example, selecting DLNAME 1 in the list shows that it contains a sub-folder "DLNAME4":



Note that you can also display sub-DLNames either by double-clicking the father DLName in the DLName column or by selecting the father DLName (still in the DLName column) then the **Down** contextual command.

To re-access the upper level, click the  button.

- 12. Click **Dismiss** to close the Configure dialog box then **OK** to exit the Document tab.

Once you have defined your DLNames, you are allowed to put them in a search order. For more information, refer to [Localizing Linked Documents](#) further in this task.

Defining DLNames also impacts the commands in which they are integrated. Refer to the [DLName Integration](#) section for detailed information.



## Activating the Logical FileTree for DLNames



This task aims at explaining how to organize your DLNames in a "logical tree" so that they are easier to access and how this will impact all commands providing access to DLNames through file or folder selection.



Prior to organizing DLNames, you must define them. Refer to [Defining DLNames](#) for detailed information.

Once your .txt list of DLNames has been created, some additional modifications are necessary to create the required structure with respect of rules that will be detailed hereafter:

- each DLName has a unique name inside the whole logical structure
- each DLName has one logical DLName "father", which can be an already defined DLName or the abstract "Root DLName"
- there is no relation between the logical structure and the physical one. When working in administrator mode, you can place the DLName wherever you want and give it the name of your choice.  
However, in user mode, bear in mind that when you decide to define a DLName as a sub-DLName of another existing DLName:
  - the name of its physical sub-folder will be the one of the sub-DLName
  - the location of its physical folder will be a sub-folder of the parent DLName's physical folder.



**1.** Open your .txt list to define your structure using the following format:

- one line per DLName
- each line has four fields separated each with a semi-colon as shown: Logical Name;Windows Folder;UNIX Folder; Father Logical Name. The first three fields correspond to the first three fields displayed when configuring DLNames
- each field ends with a semi-colon
- a blank space between the last two semi-colons means that the Father Logical Name will be "Root DLName".

Let's have a look at the following example...

```
DLName1;E:\users\ajt\;tmp; ;
DLName2;E:\users\ajt\DLName2;/tmp/DLName2;DLName1;
DLName3;E:\temp;/tmp; ;
```

... and more precisely at the meaning of each line:

```
DLName1;E:\users\ajt\;tmp; ;
```

- the logical name is named "DLName1"
- the logical name is located in "E:\users\ajt" on Windows
- the logical name is located in "/tmp" on UNIX
- the logical father name is "Root DLName".

```
DLName2;E:\users\ajt\DLName2;/tmp/DLName2;DLName1;
```

- the logical name is named "DLName2"
- DLName2 is a sub-directory of DLName 1. This sub-directory is named "DLName2" after the DLName it corresponds to. As a consequence, the logical name is located in "E:\users\ajt\DLName2" on Windows
- DLName2 is a sub-directory of DLName 1 and thus, the logical name is located in "/tmp/DLName2" on UNIX
- the logical father name is "DLName1".

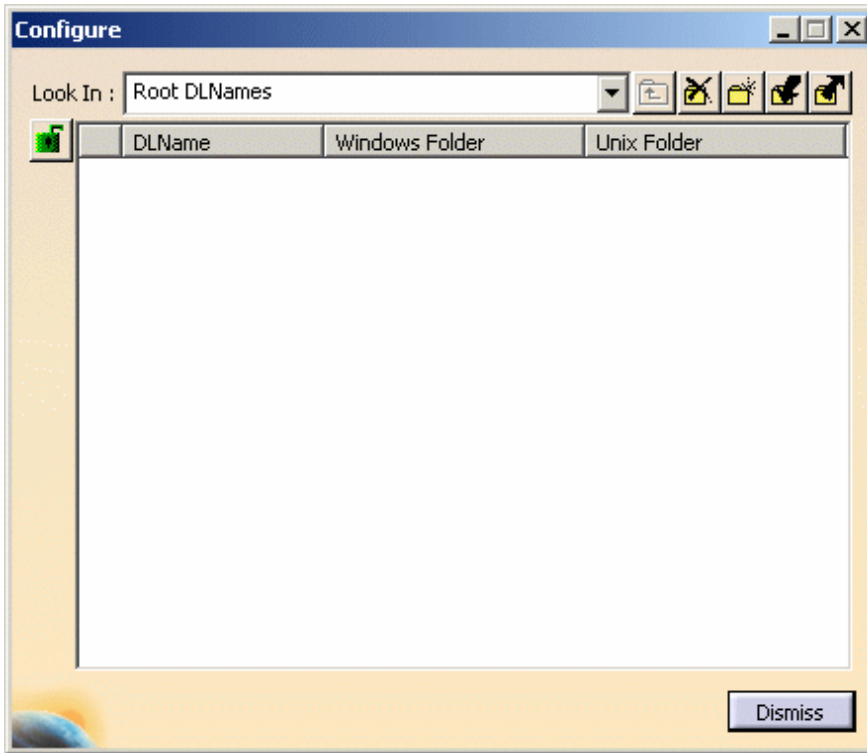
```
DLName3;E:\temp;/tmp; ;
```


- the logical name is named "DLName3"
- the logical name is located in "E:\temp" on Windows
- the logical name is located in "/tmp" on UNIX
- the logical father name is "Root DLName".



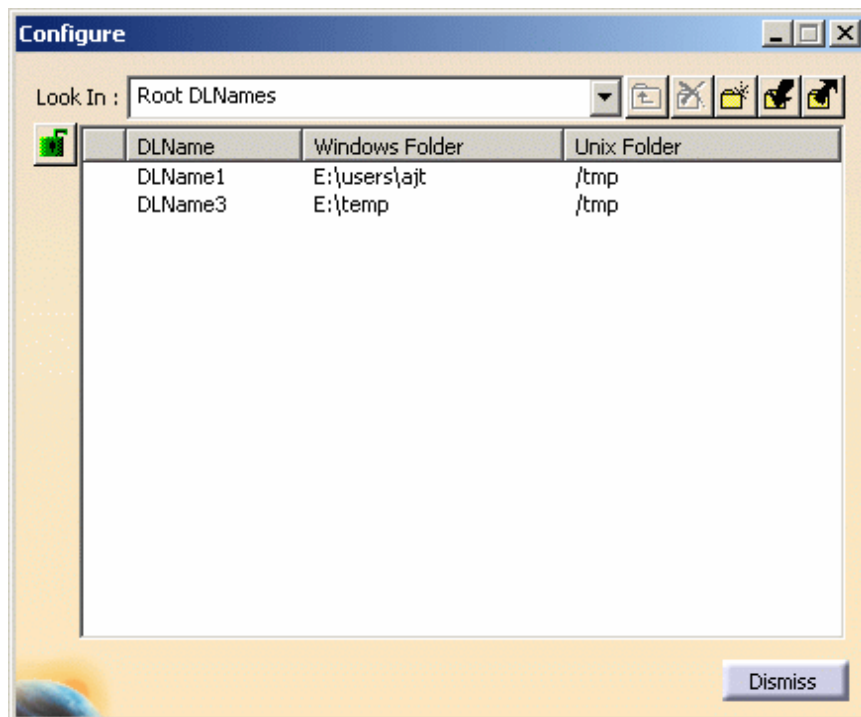
2. Save your .txt file.

3. Access the **Document** tab in **Tools->Options...->General** then click the **Configure...** button.




4. Click the  button to import the logical structure you have created in step 1.

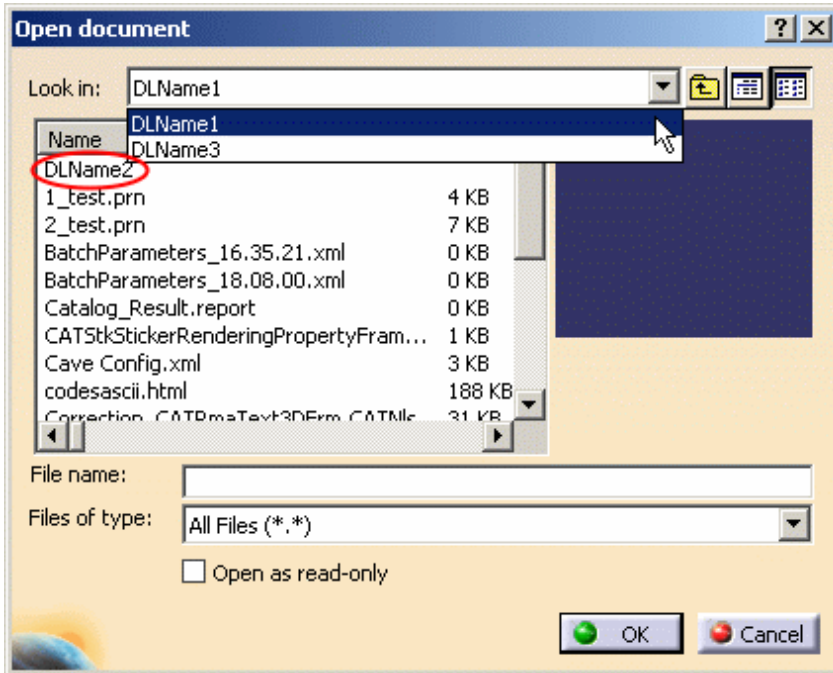
5. Navigate to the .txt file to be imported then click **Open**. The logical structure is imported:



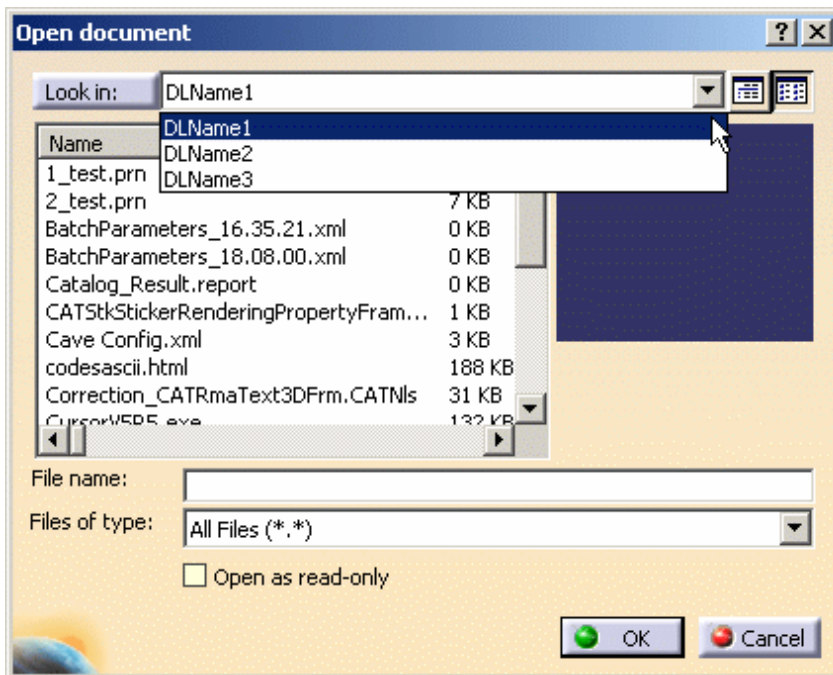
6. Click **OK** to validate.
  
7. Check the "Activate Logical FileTree for DLNames" option.

8. Click the **Open** icon  or select the **File->Open...** command. You will see that the Open document dialog box reflects the hierarchical organization and differs slightly from the Open document dialog box displayed when the logical filetree is not activated.

Example 1 - logical filetree is activated:

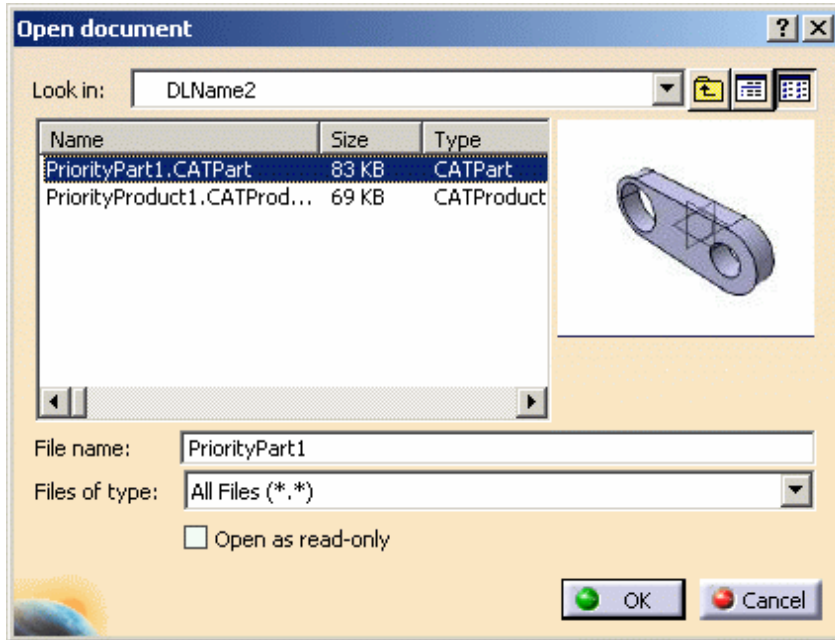



Example 2 - logical filetree is not activated:



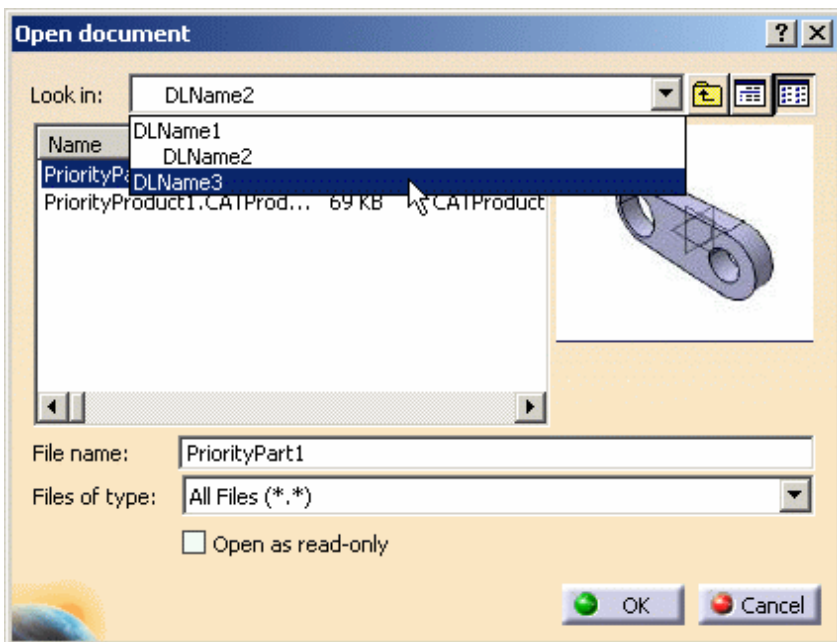
The Open document dialog box opens simultaneously with the Browse window. For more information on this window, refer to [Opening Existing Documents Using the Browse Window](#).

- 9. To access the sub-directory "DLName2", just double-click it from the list. You can, of course, display the preview of the desired element by selecting it:

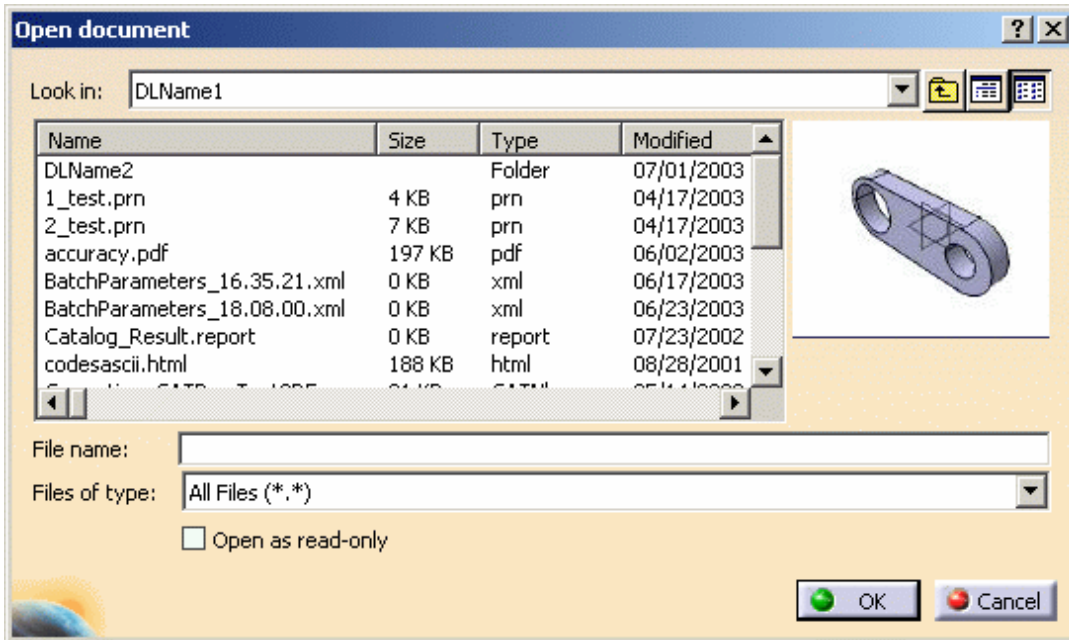


When you are in a sub-directory as this is the case in our example, you can click the icon  to move up in the hierarchy. In our case, we can use it to access the father directory "DLName1".

- 10. Clicking the black arrow next to "Look in" displays a pulldown list containing the hierarchical structure from which you can click the directory you want to access:



Would you need to display more information on the files displayed, click the Details icon. This display mode gives you information on the size, the type and the last modification date and time:




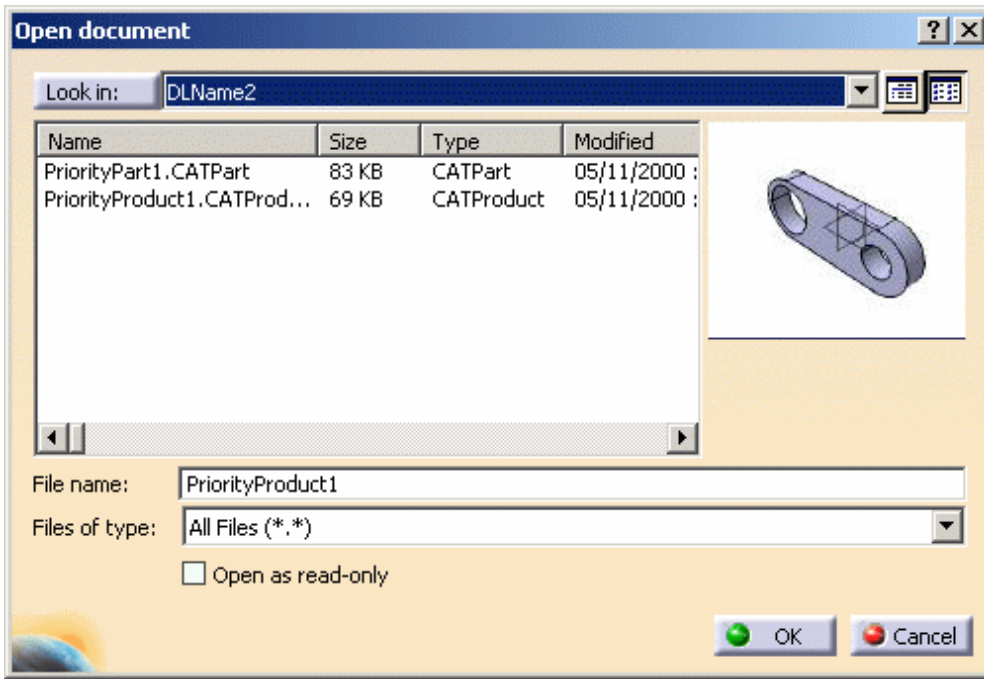
If you want to revert to the List mode, click the List icon.

11. Click **OK** to open the selected document.

## DLName Integration

 This task aims at giving you one example of the DLName integration with the **File->Open** command.

1. Click the **Open** icon  or select the **File->Open...** command. Instead of the usual File Selection dialog box, the following dialog box opens:

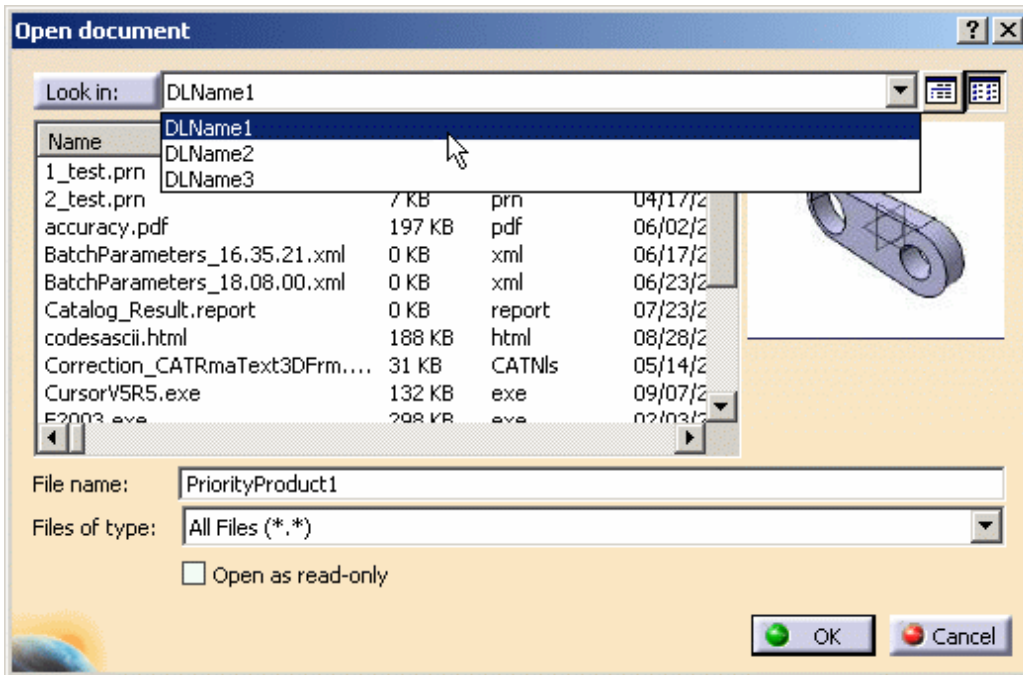


The Open document dialog box opens simultaneously with the Browse window. For more information on this window, refer to [Opening Existing Documents Using the Browse Window](#).

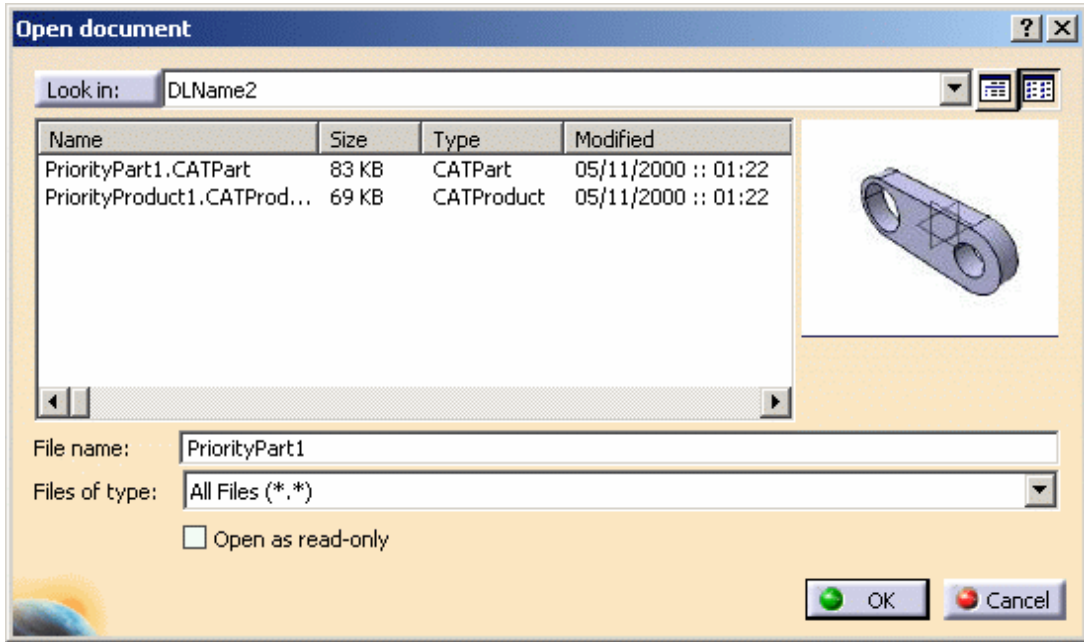


In case no DLName has been previously defined, a warning dialog box opens when running the command.

Clicking the black arrow next to "DLName1" displays a pulldown list containing the DLNames you defined in the previous steps:



Should you need to display more information on the files displayed, click the Details icon. This display mode gives you information on the size, the type and the last modification date and time:

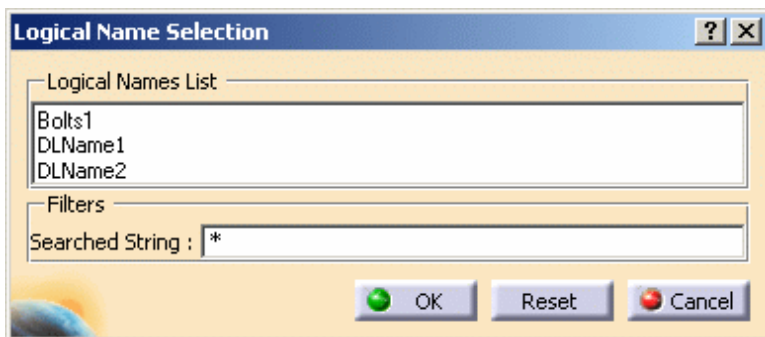


If you want to revert to the List mode, click the List icon.

**Note:** this Open document dialog box is very similar to the File Selection displayed when opening existing documents since you can, for instance, select the document type. For more information, refer to [Opening Existing Documents](#) in this guide.

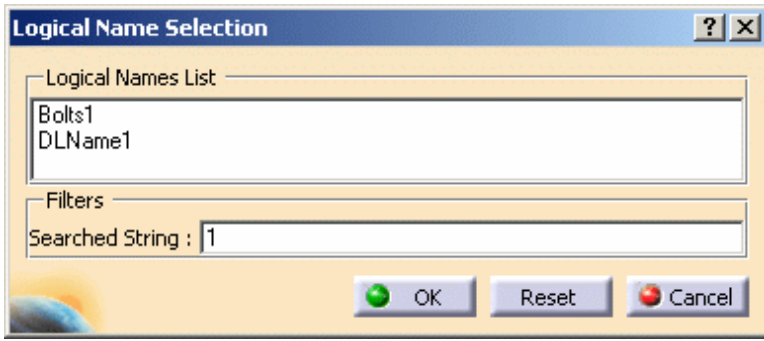
2. Select the desired File name and type from the list

3. If you want to filter the list of DLNames to be displayed, click the *Look in:* button to open the Logical Name Selection dialog box:



This dialog box lists existing DLNames in alphabetical order. To filter the list, just enter the desired string in the Searched String field then press ENTER to validate.

For instance, entering "1" displays only DLNames containing the string "1" as shown below:



You can reset the filtered list to its original state by clicking the Reset button. When finished, just click OK to close the window and go back to the Open document dialog box.

4. Click **OK** to open the selected document

DLNames are also integrated in the following commands:

- [File->Save](#) (included [Save As...](#), [Save All](#) and [Save Management...](#))
- [File->Desk](#)
- [File->New from...](#)
- [File->Send To](#)
- [Edit->Links...](#)
- [Search order](#) ("Other folders" option)
- Browse another catalog (refer to the *Version 5 Component Catalog Editor User's Guide - [Browsing a Catalog using the Catalog Browser](#)* for detailed information)
- Save in catalog (refer to the *Version 5 Generative Shape Design User's Guide - [Saving PowerCopies into a Catalog](#)* for detailed information)

For more information on these commands, refer to the corresponding task in this guide by clicking one of the above hyperlinks.



## Localizing Linked Documents



The "Linked Document Localization" replaces the "Search Order" command.

This task shows you how to define a personal strategy for link resolution.

The primary aim of the Linked Documents Localization... function is to resolve document links and to manage in an easy way the strategy that will be used to locate your linked documents. Its purpose is different from that of the **Edit -> Links** command (see [Editing Document Links](#)).



A default strategy will always take an already loaded document with the same identification to resolve document links. In a file-based environment, a document is identified by its name and an internal identifier.



1. Select the **Tools->Options...** command then in the **General** category, click the **Document** tab.

The following dialog box appears:

Document Environments

Document Environment	State	
Folder	Current	Current
DLName	Not allowed	Allowed
SmarTeam Query	Not allowed	Not allowed
SmarTeam	Not allowed	Configure...
SmarTeam2	Not allowed	
Enovia	Not allowed	
Catalog	Not allowed	
Loaded document	Allowed	

Linked Document Localization

Document Location	Active	
Folder of the link	Yes	Activate
Folder of the pointing document	Yes	Deactivate
Relative folder	Yes	Up
Other folders	Yes	Down
PDM Reconciliator Locator		Configure...
Enterprise Process-libraries	Yes	
Catalog & startup documents	Yes	
SmarTeam	Yes	
IPD	Yes	
ENOVIA LCA	Yes	

File Options

Upper case for document names

Activate Logical FileTree for DLNames

The Document Location column contains the various strategies you can choose to localize your linked documents. The Active column indicates whether a strategy has been activated ("Yes" value) or deactivated (blank value).



If your administrator has locked the Linked Document Localization, all buttons except **Configure...** are grayed.

2. Choose a strategy from the proposed list:

Linked Document Localization

Document Location	Active	
Folder of the link	Yes	Activate
Folder of the pointing document	Yes	Deactivate
Relative folder	Yes	Up
Other folders	Yes	Down
PDM Reconciliator Locator		Configure...
Enterprise Process-libraries	Yes	
Catalog & startup documents	Yes	
SmarTeam	Yes	
IPD	Yes	
ENOVIA LCA	Yes	

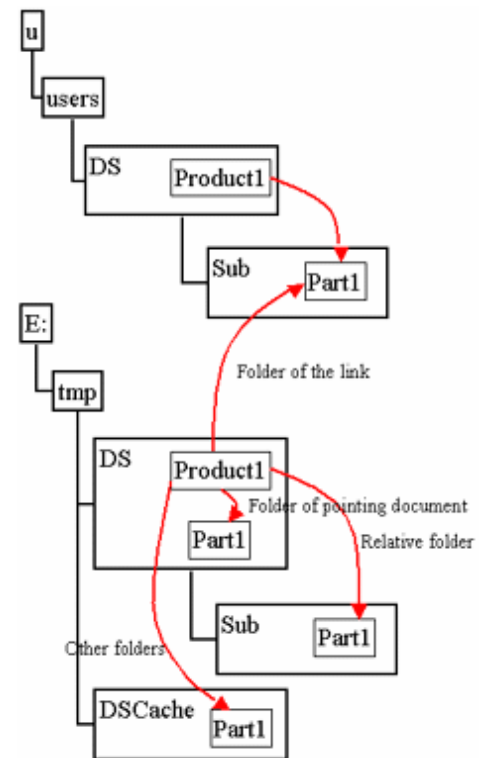


- **Folder of the link:** provides the absolute path which was saved in the link, i.e. the path used when you saved your document
- **Folder of the pointing document:** provides the current folder of your document
- **Relative folder:** provides a sub-folder with the same starting path
- **Other folders:** provides a user-defined list of folders (former "Search Order")
- **PDM Reconciliator Locator:** enables to reroute document links by providing the path to documents moved from one PDM to another (for instance, from ENOVIA VPM to ENOVIA V5)
- **Enterprise Process-libraries:** provides the path to folder containing process libraries (.act files)
- **Catalog & Startup Documents:** provides the absolute path to catalogs and documents saved in the startup installation folder (for instance c:\Dassault Systemes\B12\SOS\startup\...). When migrating from one release level to another, any link to a catalog or document located in the installation folder will be automatically updated to match the new startup path
- **SmarTeam Database** (appropriate license is required) for documents stored in SmarTeam
- **ENOVIA LCA** (appropriate license is required) for documents stored in ENOVIA LCA
- **ENOVIA VPM database** (appropriate license is required) for documents stored in ENOVIA VPM.

The following example illustrates more precisely the behavior of the four first strategies:

First of all, we created a product named Product1.CATProduct in the folder /u/users/DS/. This product is linked to Part1.CATPart, which is stored in the folder /u/users/DS/Sub/.

Then, we moved Product1 to another folder, i.e. E:\tmp\DS/. As this product is linked to Part1.CATPart, you need to select a link resolution strategy. Depending on the strategy you select, the behavior will be:



- **Folder of the link:** try to locate Part1 in /u/users/DS/Sub
- **Folder of the pointing document:** try to locate Part1 in E:\tmp\DS
- **Relative folder:** try to locate Part1 in E:\tmp\DS\Sub
- **Other folders:** try to locate Part1 in user-defined folders.

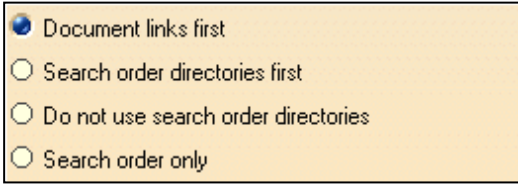
3. Once the strategy is selected, click one of the buttons displayed in the right part of the dialog box. These buttons are grayed out depending on the strategy you selected:

- "Activate" lets you activate a strategy that has been deactivated
- "Deactivate" enables you to deactivate a strategy
- "Up" and "Down" let you customize the strategy order
- "Configure" is available for the *Other folders* strategy only.

**Note:** whichever action you choose for the strategy, you must restart your Version 5 session to take this modification into account. A warning message will be displayed after clicking one of the above buttons.

Here is a comparison between the former Search Order command and "Linked Document Localization":

**Tools->Search Order**

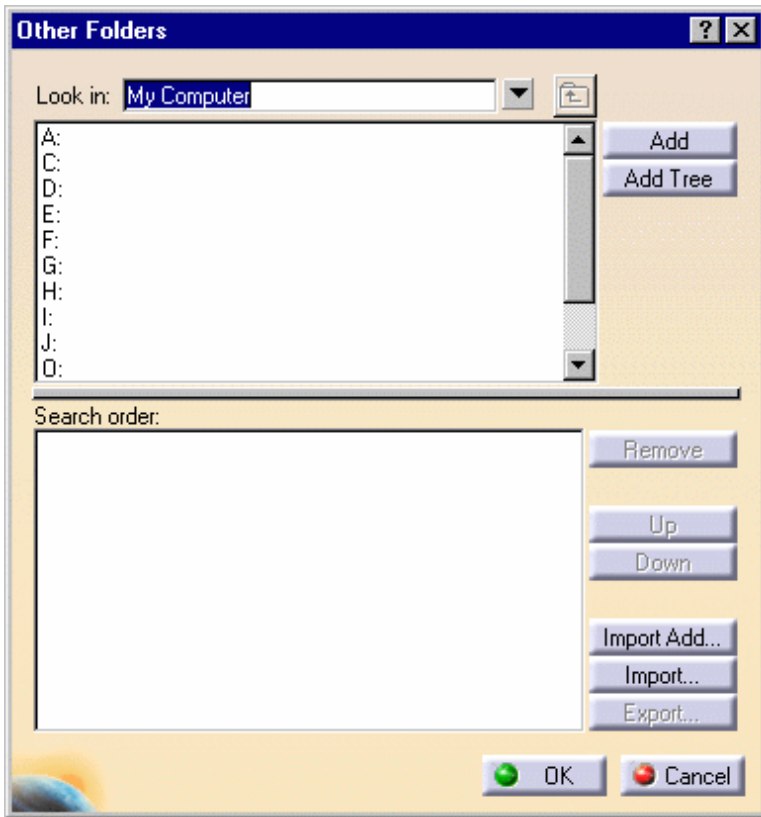


**Tools->Options->Document->Linked Document Localization**

- = "Folder of the link" first
- = "Other folders" first
- = Deactivate "Other folders"
- = Only "Other folders" is activated

**Note:** "Relative folder" is used for catalog component links.

When you click the "Configure..." button, a dialog box appears with a user interface depending on your current document environment. In our example, we are working in a Folder environment. For more information on managing document environments, refer to the [Document Environments](#) part in this task.



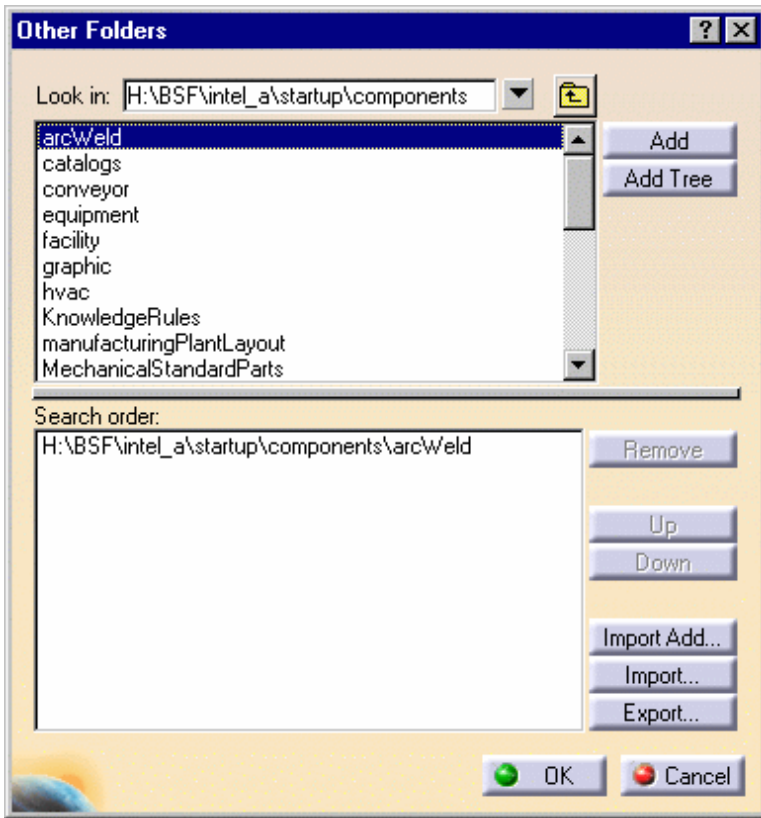
**Folder environment**



4. Select a directory in the "Look in" list then click the **Add** button.

In case at least one document is already opened in your current session, a warning window will be displayed to ask you to close then reopen the document(s) to take into account your modifications. Thus, it is recommended to close any opened documents before modifying the Other folders strategy.

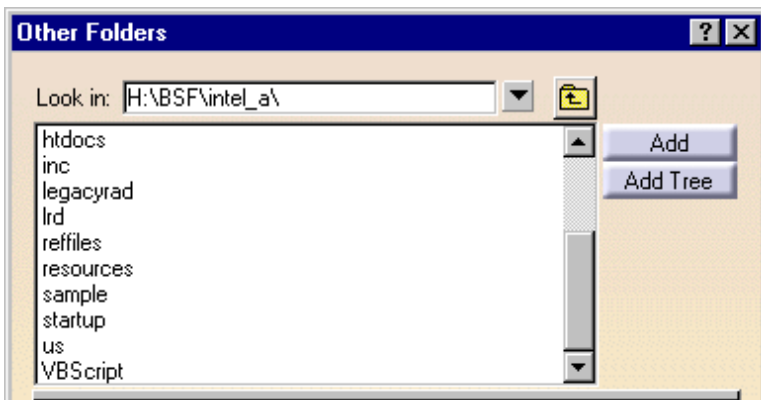
**Note:** you can also click the **Add Tree** button to add the folder tree underneath the specified directory. However, bear in mind that the more elements, the longer the duration. This button is available for the Folder environment only. When working in a DLName environment, you can copy a DLName but not its folders.

This puts the directory into the Search order list as shown below:

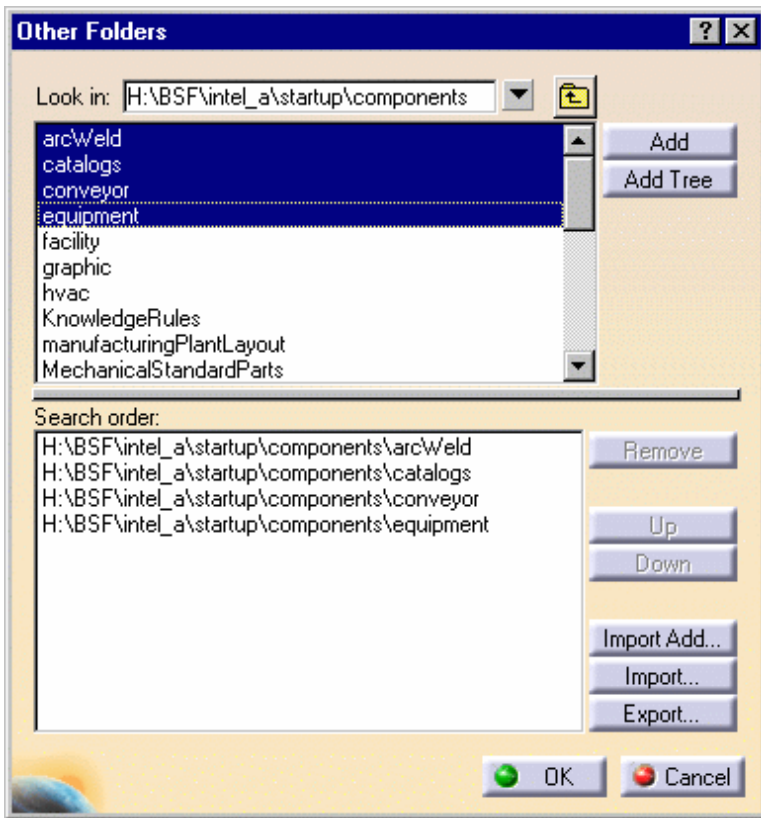


The  icon lets you access your various directories and the  icon lets you move up in the directory hierarchy.

You do not need to enter the whole path in the "Look in" field. Just enter the main directory and press Enter. Any subdirectories are then displayed in the list. Select the appropriate subdirectory and click the **Add** button.

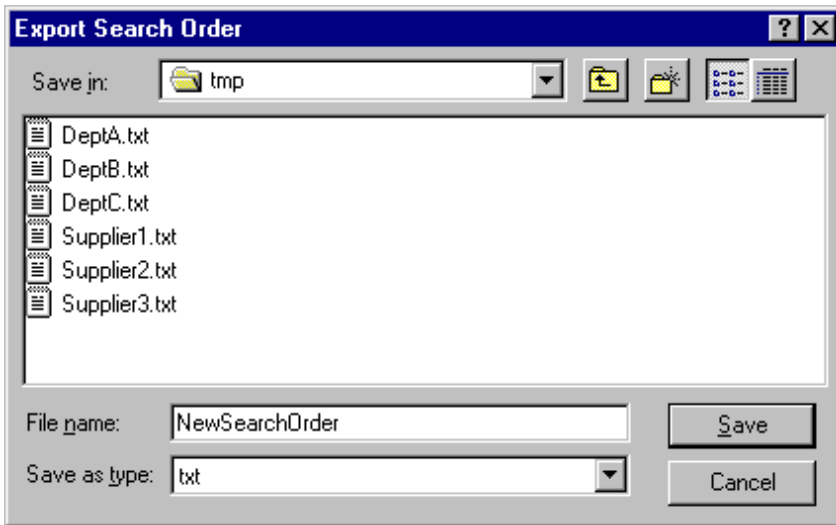


Note that you can use the multiselection to add your folders to the search order:

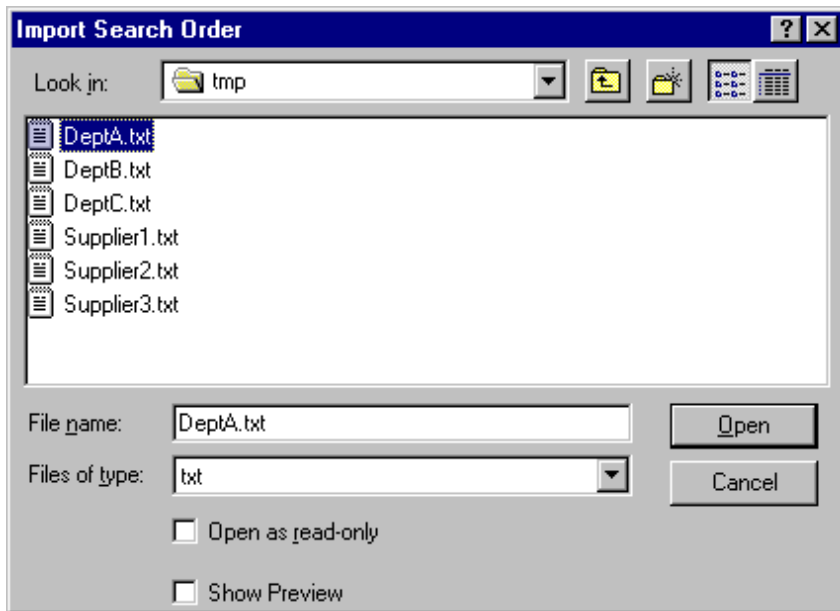


The Search order list is intended to contain the directories in which your linked documents are saved. You may wish to use different search order lists for particular uses and for different circumstances.

- Continue adding your most frequently used directories to the list. When you have finished click the **Export...** button and save the list as a .txt file in the appropriate location in the Export Search Order dialog box:



- You can make as many lists as you like. When you want to use one of these lists just click the **Import...** button. The Import Search order dialog box appears.



Right-clicking on any of the .txt files allows you to edit these files. If you then click the **Open** button the contents of the .txt file you chose is placed in the list and replaces the elements previously contained, if any.



The **Import Add...** button lets you import and add a search order contained in a text file to the current Search order list.

- 7. The search order lists can easily be updated. To change the priority of a directory i.e. move it up or down in the list, select it then click the corresponding button: **Up** or **Down**.  
Note that the multi-selection is also allowed.

- 8. To remove a directory, simply select it and click the **Remove** button.

The **Remove**, **Up** and **Down** buttons are activated only when at least one item in the list is selected.



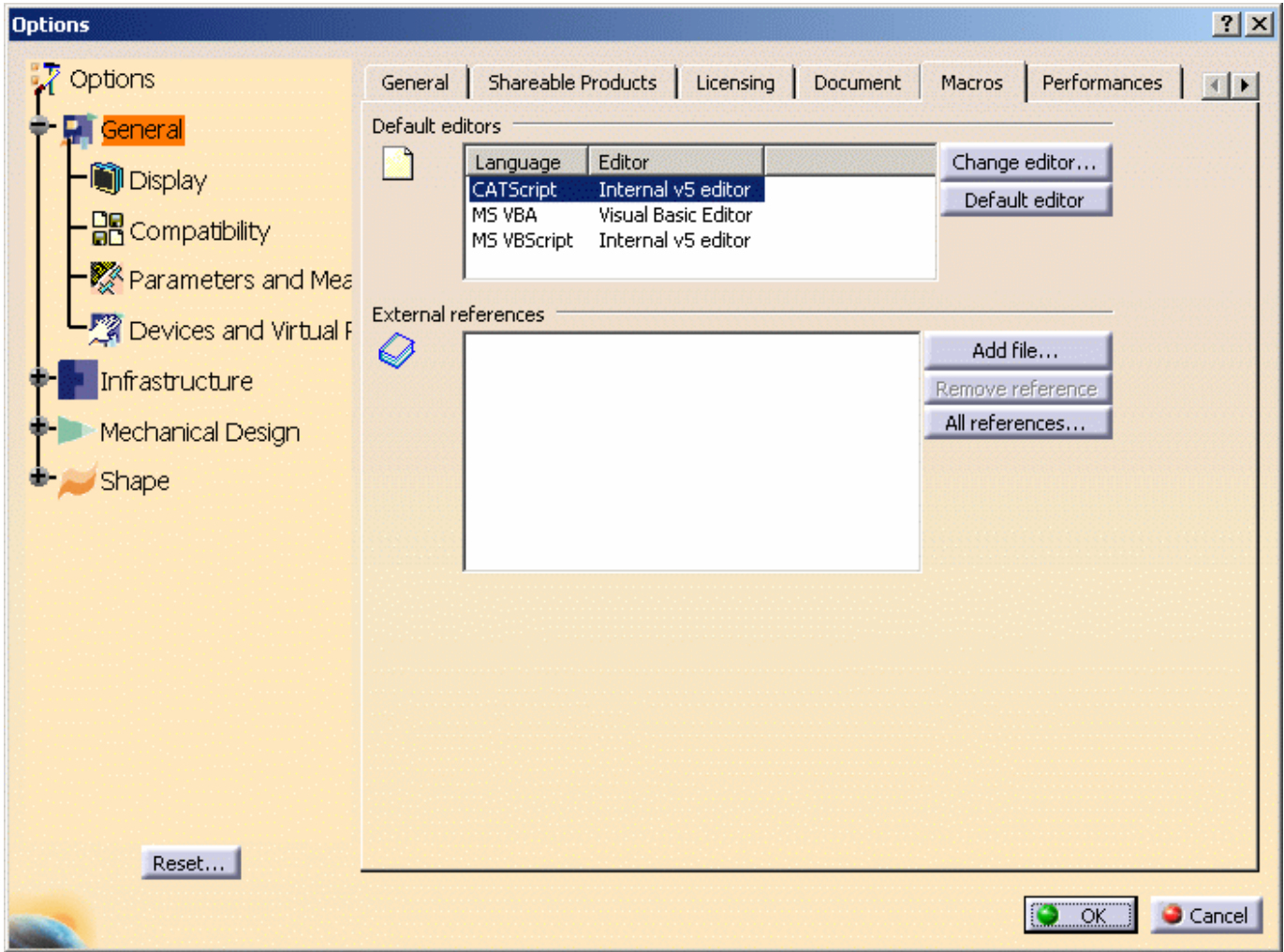
# Macros



This task explains how to use the Macros tab for accessing the macro development environment for automation purposes.



1. Select the **Tools->Options...** command.
  
2. Select the General category (in the tree to the left), then the Macros tab:



### Default Editors

The default editors list is as follows:

- CATScript: internal V5 editor
- VBScript: internal V5 editor
- VBA: Visual Basic Editor.

If you are using CATScript or VBScript, click the Change editor... button and select the editor you want to use. You cannot change the editor if you are using VBA.

After choosing an external editor, you can restore the default editor by selecting the language and clicking the Default editor button.

### External References

External references are only used when your macro script is used to control external applications. You have identify and select the corresponding typelibs.

By default, the external references list is empty. Clicking the All references... button displays the Current references dialog box containing a list of all the Version 5 typelibs because they are installed automatically.

To add external typelibs, click the Add reference... button and navigate to select the typelibs. If you click the All references... button again, you will see the external references at the bottom of the list in the Current references dialog box.

To remove external references, select them from the list and click the Remove reference button.

**3.** Click **OK** to confirm.



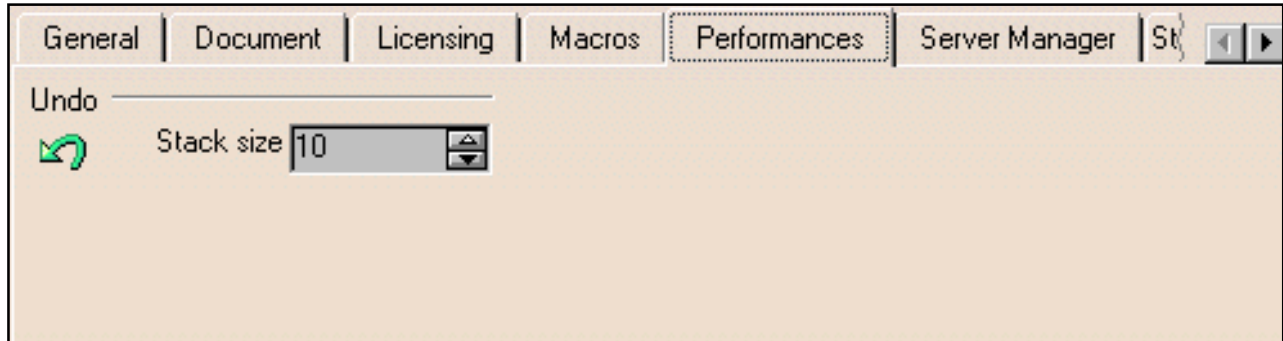
# Performances



This task explains how to customize settings for undoing commands.



1. Select the **Tools->Options...** command.
2. In the **General** category, select the **Performances** tab:



3. In the Stack size field, indicate the number of commands (from 0 to 99) which can be undone for each document.


Whatever the value, it is always possible to undo internal transitions (also called "local actions") inside the current command ("global action").

4. Click **OK** to validate.



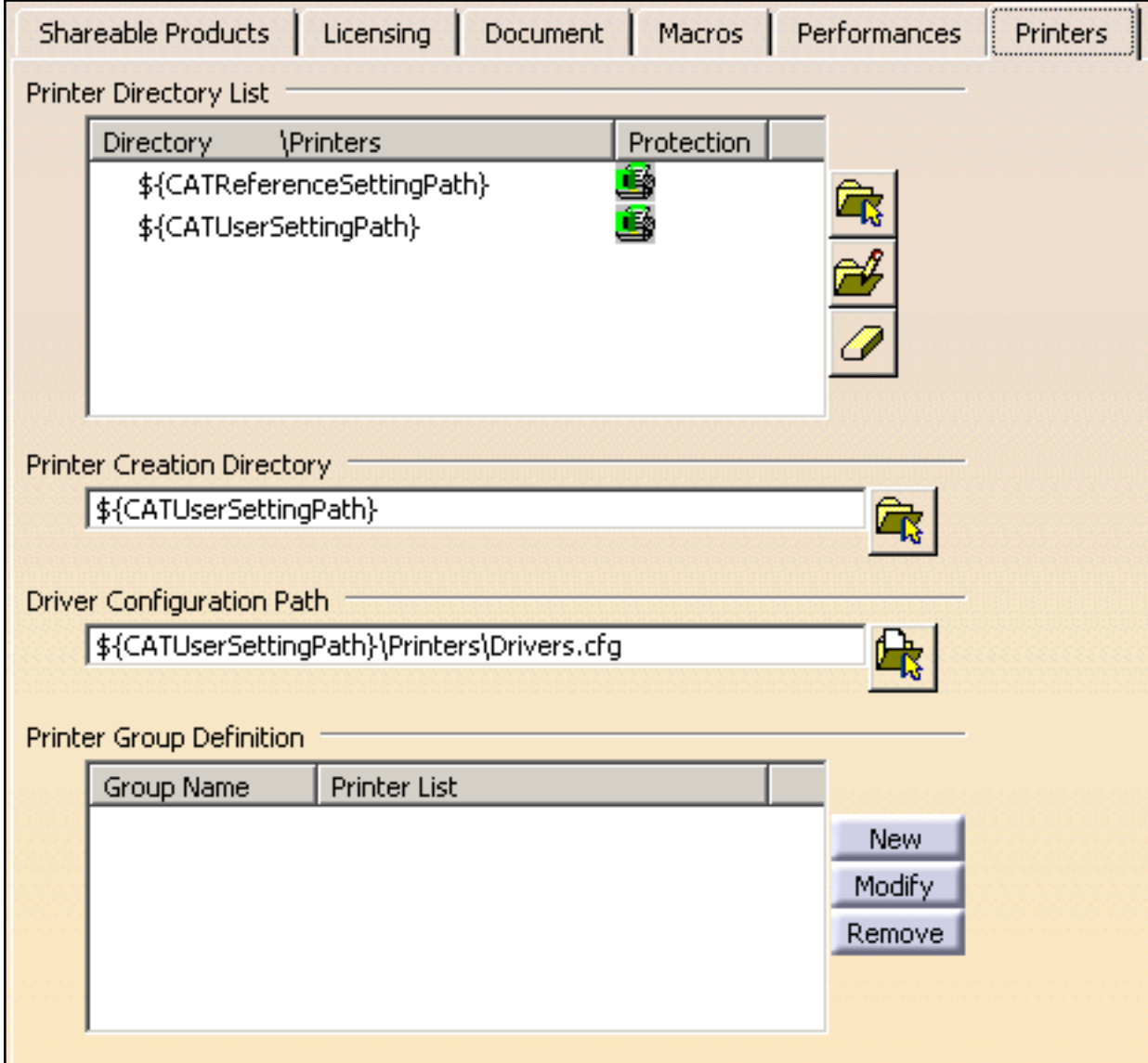


# Printers

 This task explains how to customize settings for printers directories.

 **1.** Select the **Tools->Options...** command.


**2.** In the **General** category, select the **Printers** tab:

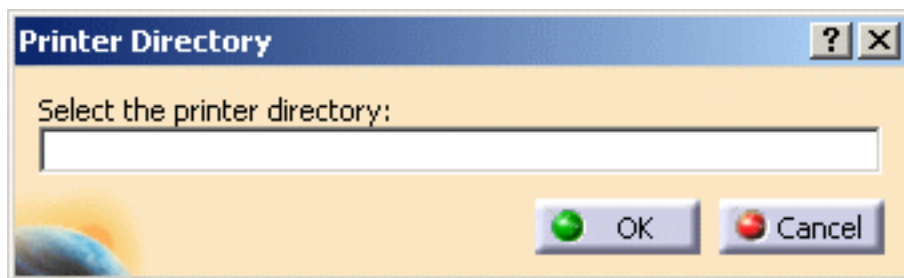


The screenshot shows the 'Printers' tab in the 'Options' dialog. It features several sections for configuring printer settings:

- Printer Directory List:** A table with columns 'Directory' and 'Protection'. It lists two entries: `${CATReferenceSettingPath}` and `${CATUserSettingPath}`, both with protection icons. To the right are icons for adding, deleting, and editing entries.
- Printer Creation Directory:** A text field containing `${CATUserSettingPath}` with a folder icon to its right.
- Driver Configuration Path:** A text field containing `${CATUserSettingPath}\Printers\Drivers.cfg` with a folder icon to its right.
- Printer Group Definition:** A table with columns 'Group Name' and 'Printer List'. To the right are 'New', 'Modify', and 'Remove' buttons.

Please note that you can use variables other than `${CATReferenceSettingPath}` or `${CATUserSettingPath}`; user-defined variables are supported.

To edit a user-defined variable, click the  button then enter the new value in the Printer Directory dialog box that opens:



When finished, click **OK** to validate.

## Printer Directory List

This area provides a list of directories that will be used to store the configuration files of your 3DPLM printers. You can add your own directories and also lock or unlock the access to the desired directories when logged as administrator.


3. When logged as administrator, you can select the desired directory from the list then use the

**Protect Printers** contextual command to lock the access to this directory (the green lock



under the Protection column turns red).

Inversely, once a directory has been locked, you can make it accessible using the **Free**

**Printers** contextual command (the red lock  turns green).

Note: you can also use the above-mentioned commands when logged as a standard user but it is recommended to be an administrator.


"Locking" a printer directory means that when accessing the Printer Properties dialog box to configure a printer, the parameters defined by your administrator will be displayed but you will not be able to modify them.


4. To add a directory to the list, click the  button then browse your filetree.

If no directory has been added, `${CATUserSettingPath}` and `${CATReferenceSettingPath}` are automatically added to the list. These two variables reference in the Version 5 environment the directories where setting files are stored:


- `CATReferenceSettingPath`: points to the directory (or directories) where administrator settings are stored
- `CATUserSettingPath`: points to the directory where user settings are stored.

Note that any directory you add should have a father directory named "Printers". This father directory must be located at the end of the path, for instance "e:\users\Printers". If no such directory exists, the application automatically adds it to the path.

5. To remove a directory from the list, select it then click the  button.

 When a printer directory has been locked by your administrator, you can still add printers from this directory but you cannot modify them.

## Printer Creation Directory

6. The Printer Creation Directory field lets you navigate to the directory where the configuration files for your new 3DPLM printers will be stored by clicking the  button.

By default, [3DPLM Printer configuration files](#) are stored in the permanent setting files which are created in a location referenced by the `CATUserSettingPath` variable. This location is:

- `$HOME/CATSettings/Printers/PLOTxxxx.xml` (on Windows)
- `$HOME\CATSettings\Printers\PLOTxxxx.xml` (on UNIX).

where "\$HOME" represents the `CATUserSettingPath` variable.

If no directory has been specified, the field displays `${CATUserSettingPath}` to indicate that the default location will be used.

Once a directory has been specified in the Printer Creation Directory field, any new 3DPLM printer you add afterwards (refer to [Adding a Printer](#) for detailed information) will be

added to this directory.

**Note:** any printer creation directory you specify should have a father directory named "Printers". If no such directory exists, the application automatically adds it to the path.



When working in administrator mode, new printers are created in `${CATUserSettingPath}` by default but you can change it to `${CATReferenceSettingPath}` if needed. In that case, once printers have been created, do not forget to change `${CATReferenceSettingPath}` back to `${CATUserSettingPath}` to provide a user environment to end users.

## Driver Configuration Path

This area is relevant when working in a CAA environment only since it lets you customize the path to external drivers that may have been defined with CAA Version 5 APIs.

7. Enter the path to the configuration file where your driver settings are stored or click the



button to navigate to the desired location.

This file is identified by the suffix `*.cfg`.

By default, the external driver configuration file is stored in:

- `$HOME/CATSettings/Printers/Drivers.cfg` (on Windows)
- `$HOME\CATSettings\Printers\Drivers.cfg` (on UNIX).

where "\$HOME" represents the `CATUserSettingPath` variable.

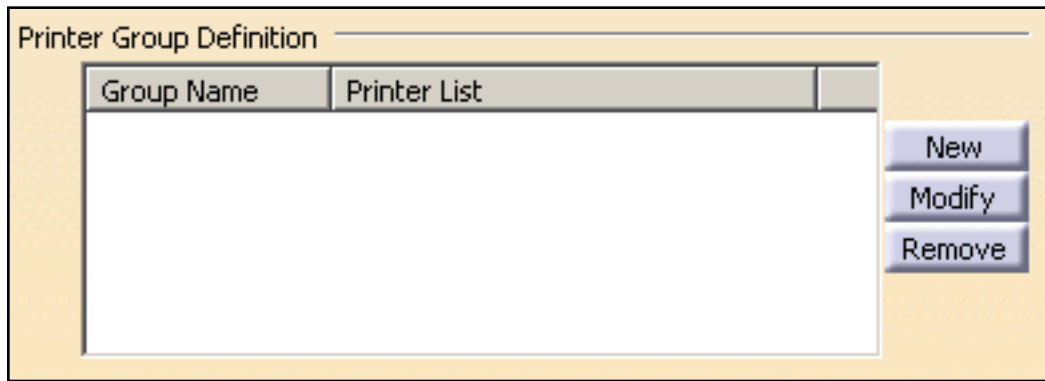
If no path is specified, the field displays `${CATUserSettingPath}\Printers\Drivers.cfg` to indicate that the default location will be used.



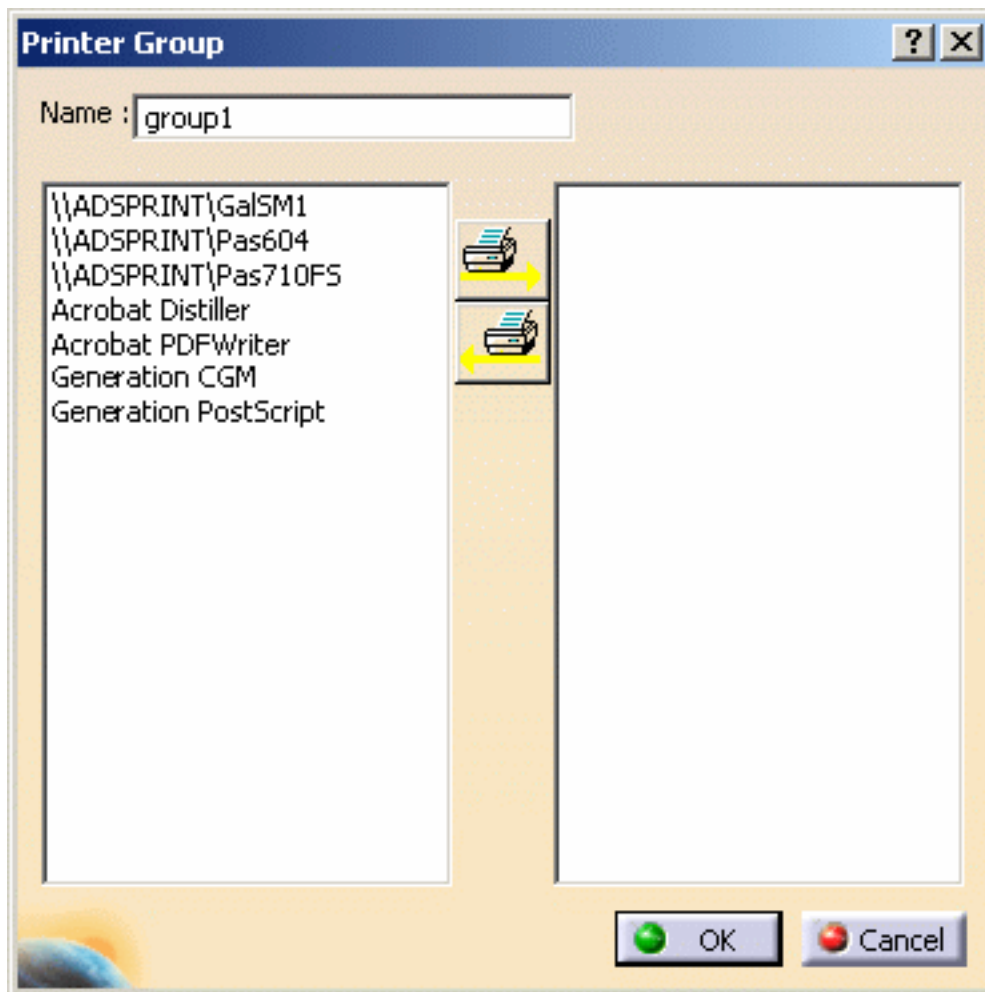
## Printer Group Definition

In this area you can define, modify and delete groups of printers. These groups will then enable you to filter the printers that will be displayed in the Print dialog box and therefore, select the desired printer more easily.

Note that the printer groups you define in this tab are saved in the `Printers.CATSettings` file (stored in the `CATSettings` directory).



8. Click the **New** button to open the Printer Group dialog box:




The left column displays the printers that have already been set up.

The group name is displayed in the **Name** field and can be modified if needed. The group number is incremented for each new group you create.



- In the left column, select the printers to be added to the new group then click the button to add them to the right column of the dialog box.

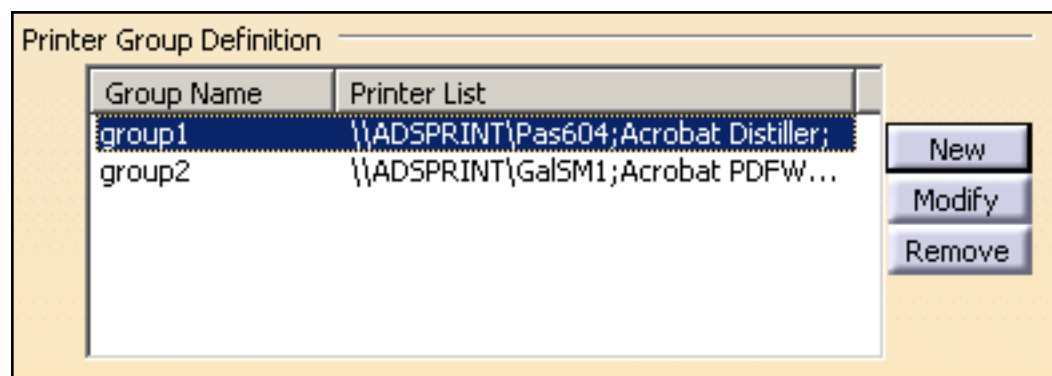
Once a printer has been added to a group, you can remove it by selecting it in the right

column then clicking the  button: in that case, the removed printer will reappear in the left column.

**Note:** you use the multi-selection to select your printers in the left and right columns.

- Click **OK** to close the Printer Group dialog box.

The Printer Group Definition area now looks like this:



- You can then carry out the following actions on the created groups:

- modification: select a group then click the **Modify** button. This opens the Printer group dialog box detailed above and you can then add or remove printers
- removal: select a group then click the **Remove** button to remove it from the group list.

Once printer groups have been defined, you use them to filter the printers displayed in the [Print](#) dialog box using a specific combo box.

- When finished, click **OK** to validate your settings.



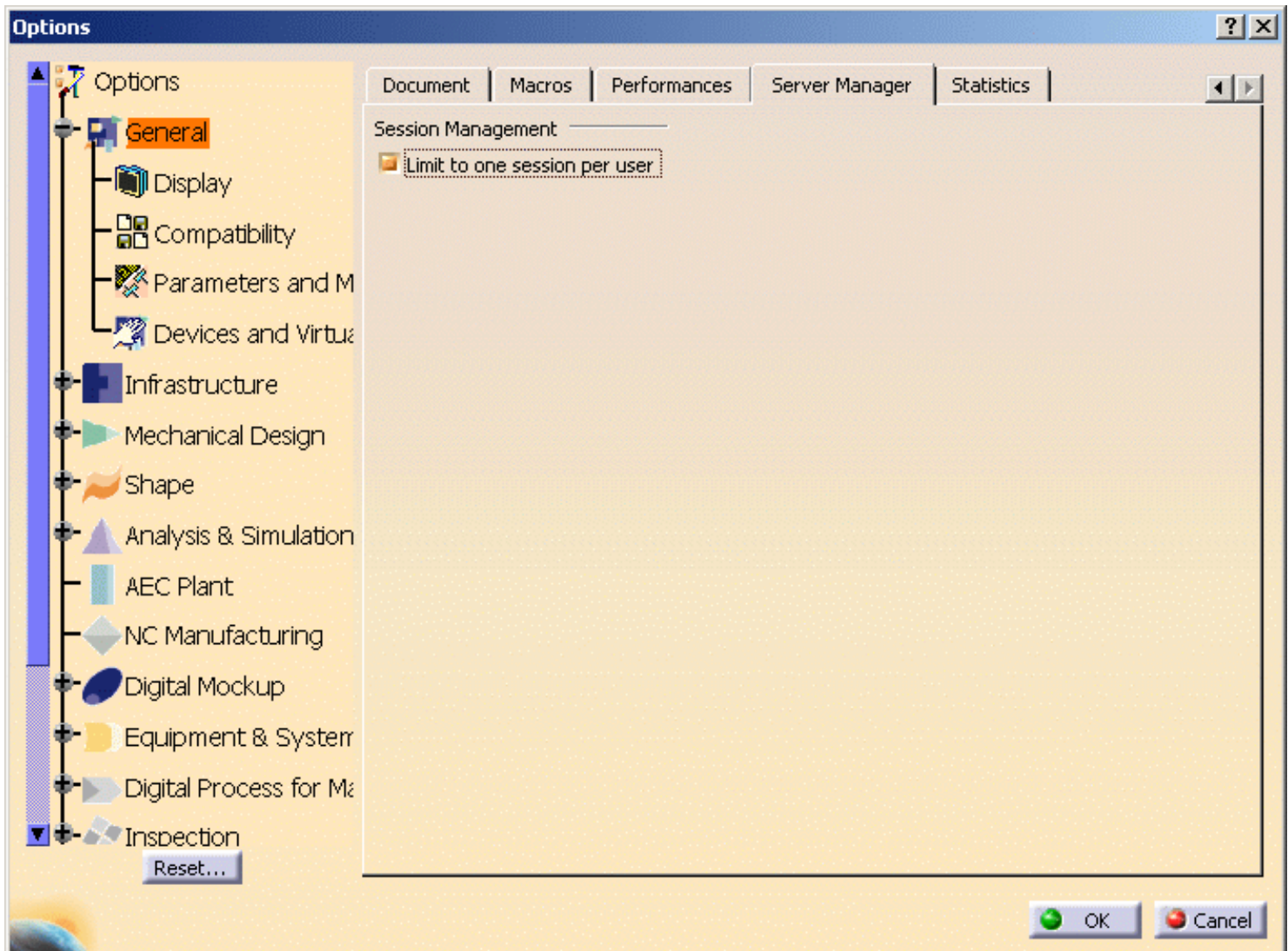
# Server Manager


## Creating a Limit of One Session per User

The following setting enables you to limit each user to one session. In the case that the user already has a session opened, that session will be aborted.



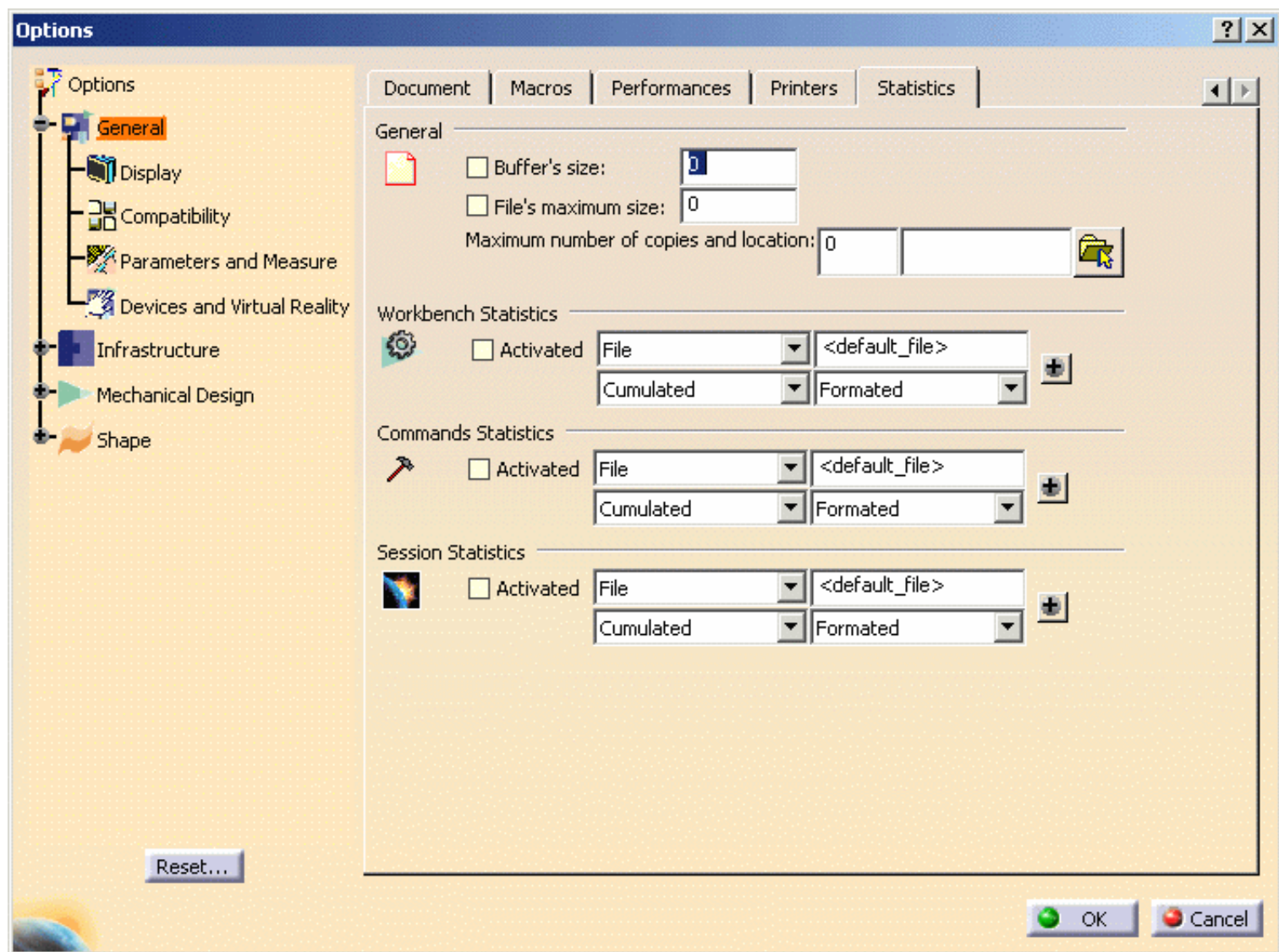
1. In the **General** category, click the **Server Manager** tab.
2. Check the **Limit to one session per user** checkbox.
3. Click the **OK** button to validate.



 This task explains how to obtain workbench and command use statistics, to log the time spent in workbenches, specific commands and server and session statistics.

 **1.** Select the **Tools->Options...** command.

**2.** Select the **General** category, then the **Statistics** tab:





The General settings control for all log files:

- **B:** size of buffer file in Kbytes; the log file is only written to disk once the log file size exceeds the buffer size you set
- **File's maximum size:** sets the maximum log file size; each time this size is exceeded, the file is written to the location you specify by navigating using the file selection box in the **Maximum number of copies and location** option, which also controls the number of copies written to the disk: this helps to optimize disk space. Once the maximum number of copies is reached, the oldest log file is deleted.

You can log statistics for the following types of activities:

- time spent in workbenches
- time spent using specific commands in those workbenches
- server statistics
- session statistics.

There are two log display modes:

- **File:** the log is written to the file whose default name is *Workbench.log* or *Command* when you exit the session, in the location pointed to by the CATErrorLog variable. Consequently, the files are written by default to:

```
CSIDL_LOCAL_APPDATA\DassaultSystemes\CATTemp\Statistics\Workbench.log
CSIDL_LOCAL_APPDATA\DassaultSystemes\CATTemp\Statistics\Command.log
```

- **Console:** the log is written to a window.

There are two log update modes:

- **Add:** the log is written to the existing log
- **Session:** the existing log is overwritten by the log of the current session.

The log can be **formatted** or **non-formatted**.

## Logging Workbench Activity

Example of formatted traces for a workbench log:

...

```
them=WORKBENCH: time=Wed May 22 10.29.02 2002: elps=0.000000: rtim=0.000000: cpus=0.000000: Session=Start: User=SES: Host= JANEDSY
WORKBENCH: Wed May 22 10.38.57 2002: 0.000000: 0.000000: 0.000000: Start: SES: JANEDSY
them=WORKBENCH: time=Wed May 22 10.39.32 2002: elps=5888.000000: rtim=861.000000: cpus=370.000000: Workbench= CATDrwDrwWkb: NLS= Drafting
them=WORKBENCH: time=Wed May 22 10.39.38 2002: elps=3213621.000000: rtim=901.000000: cpus=170.000000: Workbench= CATPstProductWkb: NLS=
Product Structure
them=WORKBENCH: time=Wed May 22 11.33.12 2002: elps=4567.000000: rtim=270.000000: cpus=130.000000: Workbench= CATCuiCATALOGWks: NLS= Catalog Editor
them=WORKBENCH: time=Wed May 22 11.33.16 2002: elps=3866.000000: rtim=100.000000: cpus=60.000000: Workbench= CATMaterialWorkshopWks: NLS=
Material Library
them=WORKBENCH: time=Wed May 22 11.33.20 2002: elps=3585.000000: rtim=100.000000: cpus=90.000000: Workbench= CATAifAssemblyWkb: NLS= Assembly Design
them=WORKBENCH: time=Wed May 22 11.33.24 2002: elps=50.000000: rtim=50.000000: cpus=40.000000: Workbench= CATMaterialWorkshopWks: NLS= Material
Library
them=WORKBENCH: time=Wed May 22 11.33.24 2002: elps=40.000000: rtim=40.000000: cpus=30.000000: Workbench= CATCuiCATALOGWks: NLS= Catalog Editor
them=WORKBENCH: time=Wed May 22 11.33.24 2002: elps=1833.000000: rtim=120.000000: cpus=110.000000: Workbench= CATAifAssemblyWkb: NLS= Assembly
Design
```

Example of non-formatted traces:

```
WORKBENCH: Wed May 22 11.36.17 2002: 48690.000000: 341.000000: 300.000000: Workbench= CATAifAssemblyWkb: NLS= Assembly Design
WORKBENCH: Wed May 22 11.37.06 2002: 2173.000000: 481.000000: 290.000000: CATPstProductWkb: Product Structure
WORKBENCH: Wed May 22 11.37.08 2002: 11517.000000: 1832.000000: 1451.000000: CATPcfPartWkb: Part Design
WORKBENCH: Wed May 22 11.37.21 2002: 4877.000000: 471.000000: 430.000000: CATDrwDrwWkb: Drafting
WORKBENCH: Wed May 22 11.37.26 2002: 4747.000000: 491.000000: 120.000000: CATHybridPartWorkbenchWkb: Hybrid Part Workbench
WORKBENCH: Wed May 22 11.37.31 2002: 50.000000: 50.000000: 30.000000: CATDrwDrwWkb: Drafting
WORKBENCH: Wed May 22 11.37.31 2002: 1292.000000: 70.000000: 60.000000: CATHybridPartWorkbenchWkb: Hybrid Part Workbench
```

Note that:

- Workbench = internal workbench name
- NLS = NLS name of the workbench (the name which appears in the Start menu).

When you change an option, exit the session and restart to take your settings into account.

## Logging Command Activity

Here is the log syntax:

**Them=COMMAND:time=**Date and time:**elps=**elapsed time:**cpus=**cpu time:**Workbench=**Internal current workbench name:**NLS=**Workbench title:**Command=**Internal command name:**NLS=**Command title

Elps: elapsed time spent in the command.

rtim: "response time", which is the elapsed time minus user wait time.

Cpus: cpu time spent in the command.

This trace will be logged when you exit the command. The workbench name is the current workbench when you exit the command.

Note that certain auxiliary commands (for example, the Search command) are not logged.

### Example of formatted traces for a command log

```
them=COMMAND:time= Wed May 22 10.05.14 2002: elps= 0.000000: rtim= 0.000000: cpus= 0.000000: Session= Start: User= SES: Host= JANEDSY
them=COMMAND:time= Wed May 22 10.05.17
2002: elps= 8993.000000: rtim= 7491.000000: cpus= 630.000000: Command= FileNew: NLS= New...: CurrentWorkbench= CATPstProductWkb: NLS_CurrentWorkbench=
Product Structure: Origin_Header= Global: mode= ForegroundCmd
them=COMMAND:time= Wed May 22 10.05.14
2002: elps= 11998.000000: rtim= 20.000000: cpus= 20.000000: Command= Select: NLS= Select: CurrentWorkbench= CATPstProductWkb: NLS_CurrentWorkbench= Product
Structure: Origin_Header= Global: mode= ForegroundCmd
them=COMMAND:time= Wed May 22 10.05.26
2002: elps= 779110.000000: rtim= 80.000000: cpus= 60.000000: Command= Select: NLS= Select: CurrentWorkbench= CATPcfPartWkb: NLS_CurrentWorkbench= Part
Design: Origin_Header= undef: mode= ForegroundCmd
them=COMMAND:time= Wed May 22 10.20.23
2002: elps= 79014.000000: rtim= 16935.000000: cpus= 480.000000: Command= OpenSettings: NLS= Options...: CurrentWorkbench= CATAifAssemblyWkb:
NLS_CurrentWorkbench=
Assembly Design: Origin_Header= undef: mode= ForegroundCmd
them=COMMAND:time= Wed May 22 10.21.42
2002: elps= 2143.000000: rtim= 20.000000: cpus= 20.000000: Command= Select: NLS= Select: CurrentWorkbench= CATAifAssemblyWkb: NLS_CurrentWorkbench= Assembly
Design: Origin_Header= undef: mode= ForegroundCmd
them=COMMAND:time= Wed May 22 10.19.45
```

```
2002: elps= 119642.000000: rtim= 80.000000: cpus= 20.000000: Command=AnalyserCmd: NLS=AnalyserCmd: Origin_Header=Global: mode=BackgroundCmd
them=COMMAND: time=Wed May 22 10.21.44
2002: elps= 50.000000: rtim= 50.000000: cpus= 50.000000: Command=FileExit: NLS=Exit: CurrentWorkbench= CATAifAssemblyWkb: NLS_CurrentWorkbench= Assembly
Design: Origin_Header=undef: mode=ForegroundCmd
```

### Example of non-formatted traces for a workbench log

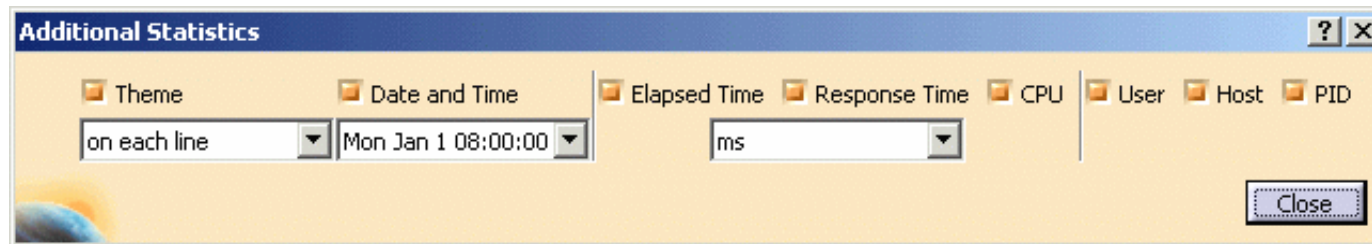
```
COMMAND: Wed May 22 10.29.14
2002: 7051.000000: 171.000000: 160.000000: Command=OpenSettings: NLS=Options...: CurrentWorkbench= CATAifAssemblyWkb: NLS_CurrentWorkbench=
Assembly Design: Origin_Header=Global: mode=ForegroundCmd
COMMAND: Wed May 22 10.29.21 2002: 2673.000000: 30.000000: 30.000000: Select: Select: CATAifAssemblyWkb: Assembly Design: Global: ForegroundCmd
COMMAND: Wed May 22 10.29.24 2002: 1392.000000: 31.000000: 30.000000: PasteSpecial: Paste Special...: CATAifAssemblyWkb: Assembly Design: Global:
ForegroundCmd
COMMAND: Wed May 22 10.29.25 2002: 2003.000000: 21.000000: 10.000000: Select: Select: CATAifAssemblyWkb: Assembly Design: Global: ForegroundCmd
COMMAND: Wed May 22 10.29.00 2002: 27429.000000: 70.000000: 10.000000: Command=AnalyserCmd: NLS=AnalyserCmd: Origin_Header=Global: mode=
BackgroundCmd
COMMAND: Wed May 22 10.29.27 2002: 50.000000: 50.000000: 40.000000: FileExit: Exit: CATAifAssemblyWkb: Assembly Design: Global: ForegroundCmd
```

## Logging Session Statistics

The Session Statistics is a new statistics theme which records information about application sessions (duration, start date, exit status). This information is collected via a statistics theme so as to take advantage of the statistics infrastructure (for example, the automatic copy and relocation of statistics files exceeding a determined size).

This information is limited to one event per session.

As such, it will help keep track of users' session durations and ending status, providing a supplementary tool to measure MTBF. More advanced statistics can be obtained by clicking the "+" option:



When activated, the Session Statistics generates one output line by session on the console or in a file. The default file is:

CATTemp/Statistics/Stats\_SESSION\_user\_HOST

Example of output:

```
them=SESSION: time=Mon Apr 14 09.12.06 2003: elps= 131690.000000:
rtim= 1112.000000: cpus= 110.000000: MainName=CATIA: ExitStatus=normal_end
```

No output is generated for the session during which the session statistics are activated. You have to exit then restart your Version 5 session, and from then on you will obtain session statistics output for all further sessions.

This new functionality is mainly useful for Version 5 administrators. Session Statistics list all the sessions, their duration, and exit conditions for any or all users, easily providing them with a comprehensive view of the average duration of user sessions and allows a global view of the usage of Version 5 by any user or group of users.

It also provides an idea of the percentage of sessions where no failure occurred.

Session statistics add a global view in addition to the more detailed views already provided by the existing COMMAND and WORKBENCH statistics.

## Notes about output logs

Event dates in the output file are not organized in chronological order. A statistics event is always logged when it is terminated. The start date and duration are specified. Events are generated in event termination chronological order. Because the statistics software allows interlaced or embedded events, events do not necessarily end in the order in which they began.


The two start and end slots indicate the same time. A statistics event is always logged when it is terminated. The start time and duration are specified. The corresponding session event is generated at the end of the session: it started at the beginning of the session and lasts the duration of the session. Due to the possibility of having several successive session statistics in the same session, it is necessary to physically log the start of a session. This is the session start event (whose duration is null). The two events are generated at the same time in the code. To determine the end time, information is provided for the start date and duration.



# Display

- Tree Appearance
- Tree Manipulation
  - Navigation
  - Performance
  - Visualization
- Thickness & Font
- Layer Filters
- Linetype

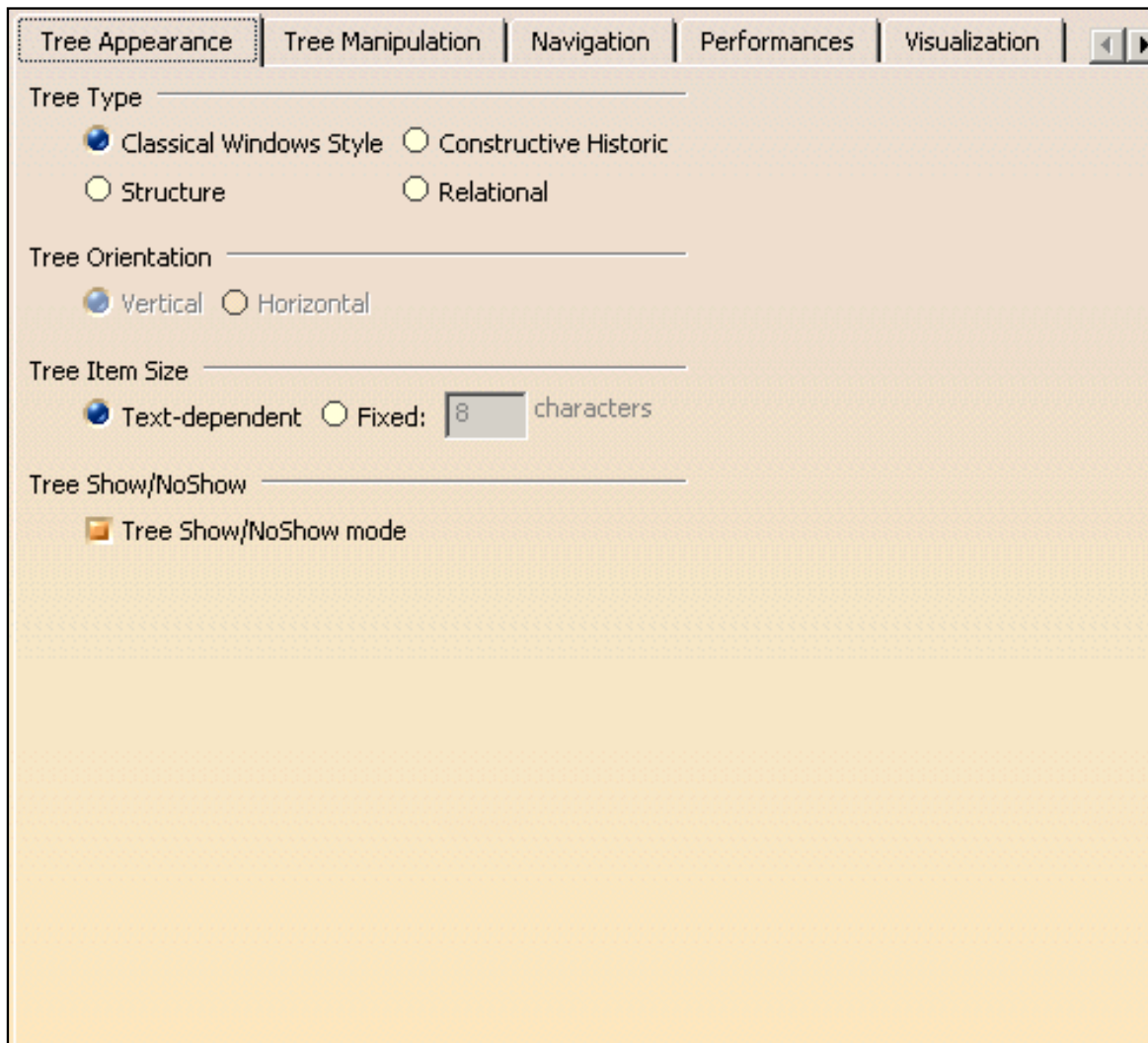
# Setting Specification Tree Appearance Options

 This task explains how to set specification tree display options.

 1. Select the **Tools->Options...** command.

The Options dialog box appears with the **General** category selected in the left-hand column.

2. In the **Display** sub-category, select the **Tree Appearance** tab to display the document layout options.

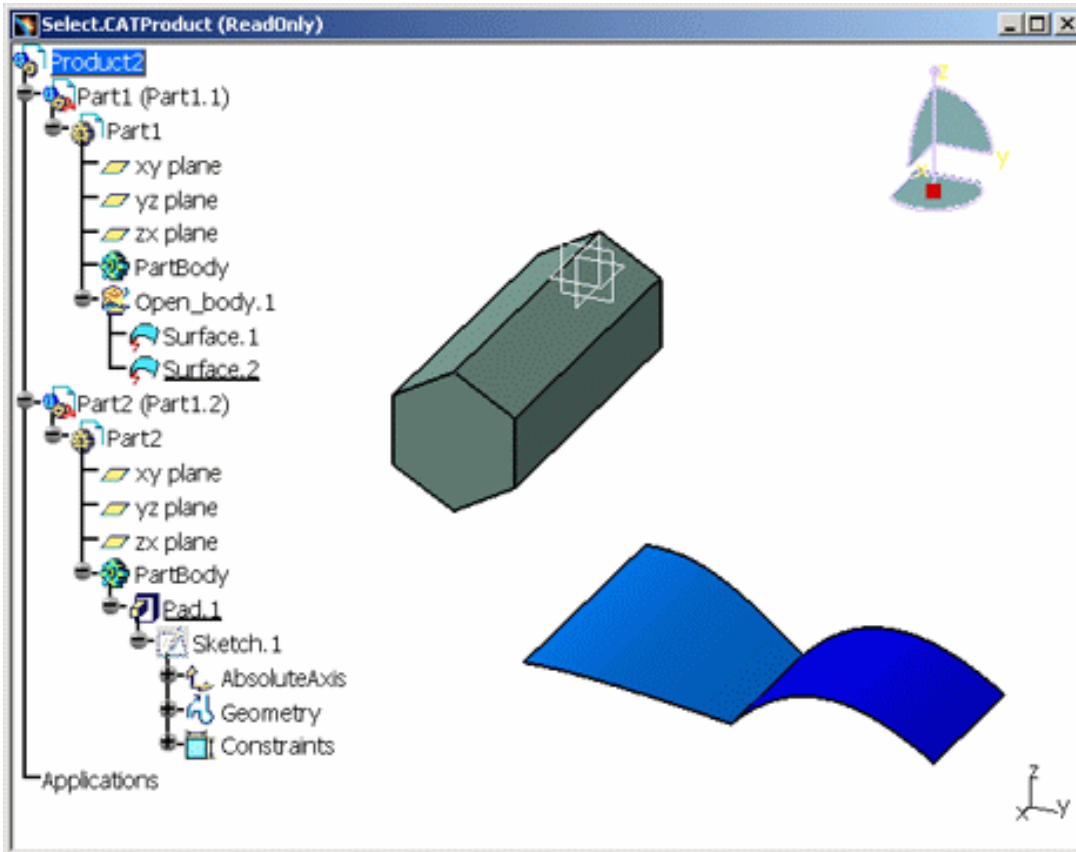


## Tree Type

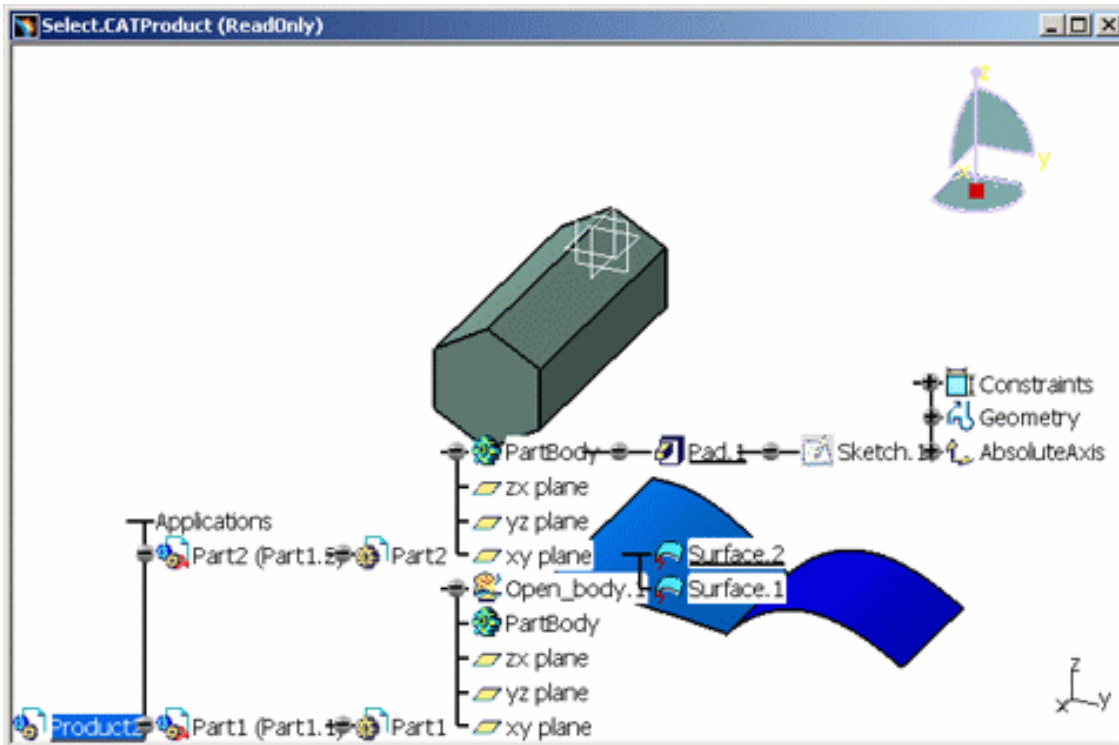
In the **Tree Type** section of the dialog box, you will notice that the option Classical Windows Style is selected by default. This is the conventional way of displaying the specification tree.

• **Classical Windows Style**

This option might, for example, be useful when working on solids and assemblies.



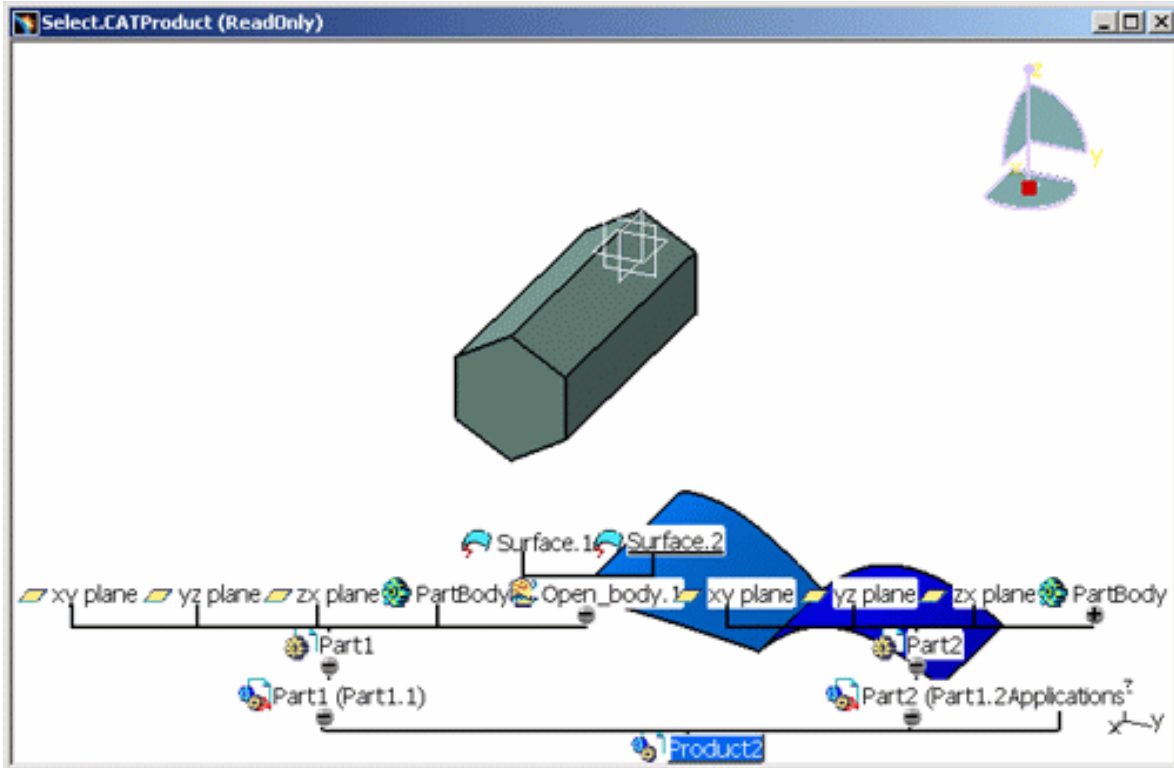
• **Structure**



- **Constructive Historic**

If the whole of the tree is not displayed, you may have to enlarge the window.

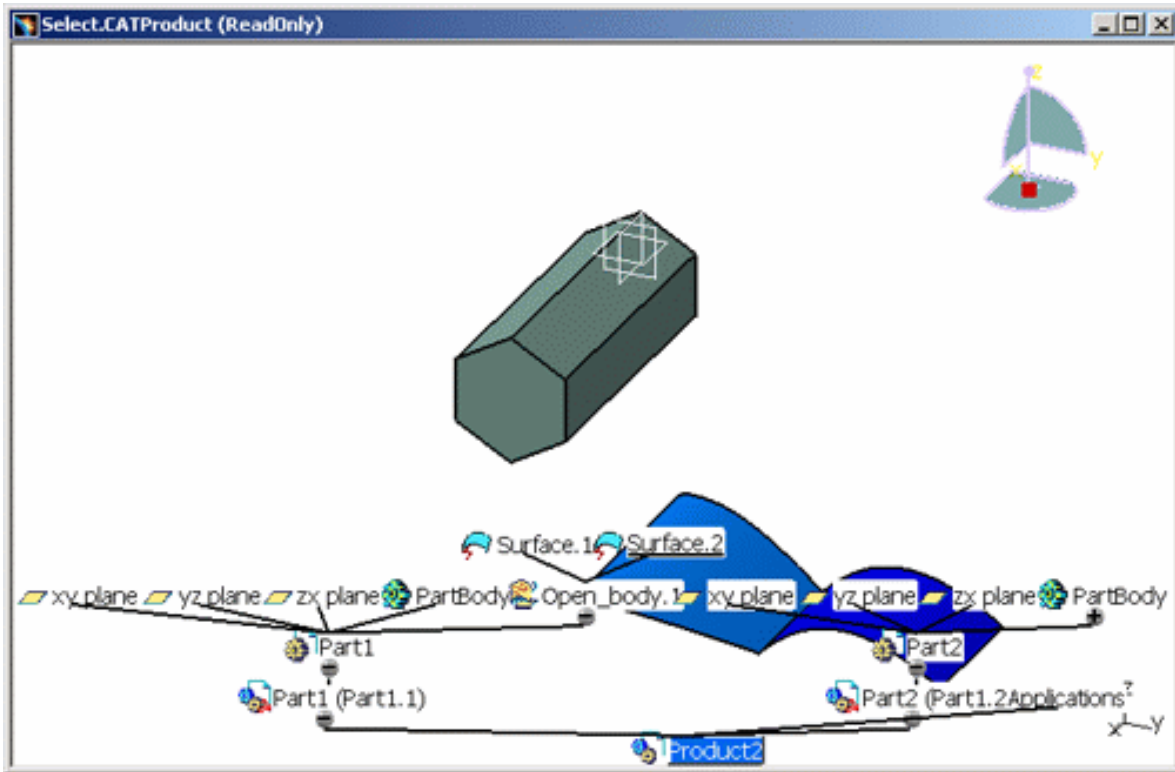
Alternatively, you can reduce the length of tree item names. To do this, go back to the **Tools->Options...** dialog box and select the Fixed Size option in the Tree Item Size section. The default number of characters is 8. You can of course specify the number you wish.



- **Relational**

To do this, go back to the **Tools->Options...** dialog box and select the Relational option in the Tree Type section. The tree will then look something like this:



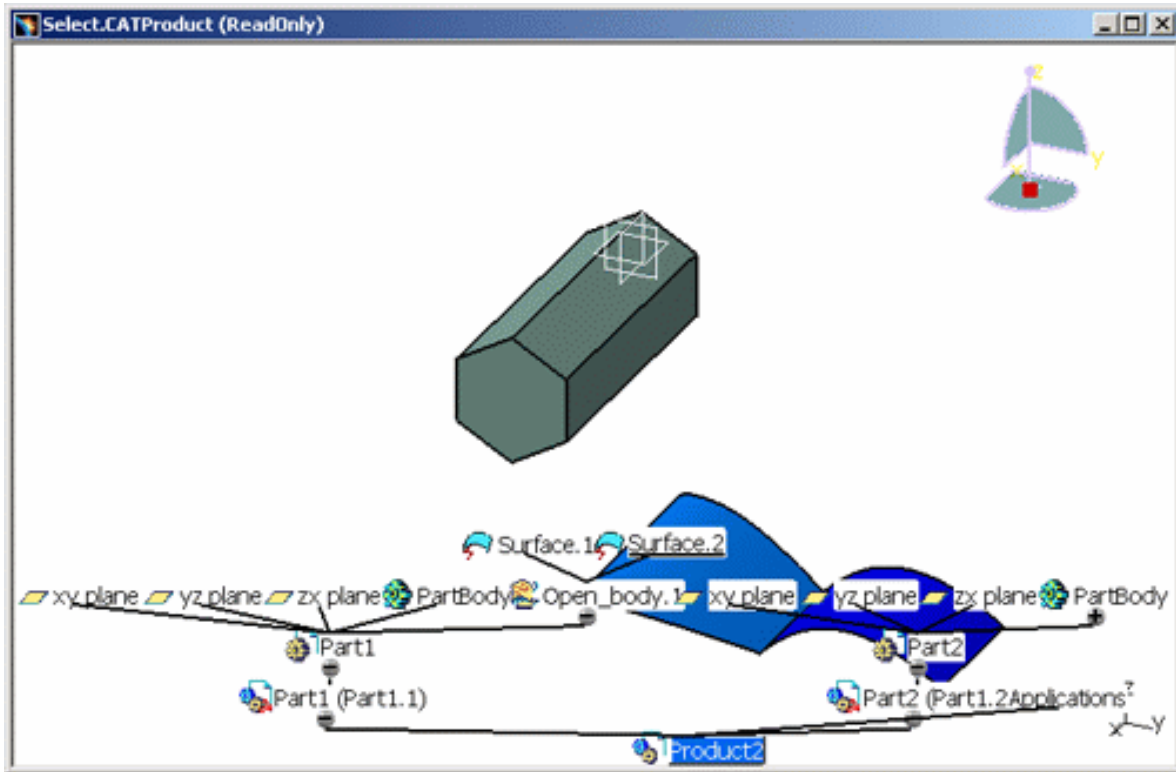


This option might, for example, be useful when working on solids and surfaces.

## Tree Orientation

Note that the tree orientation can be modified only when displaying the specification tree in "Constructive Historic" or "Relational" mode.

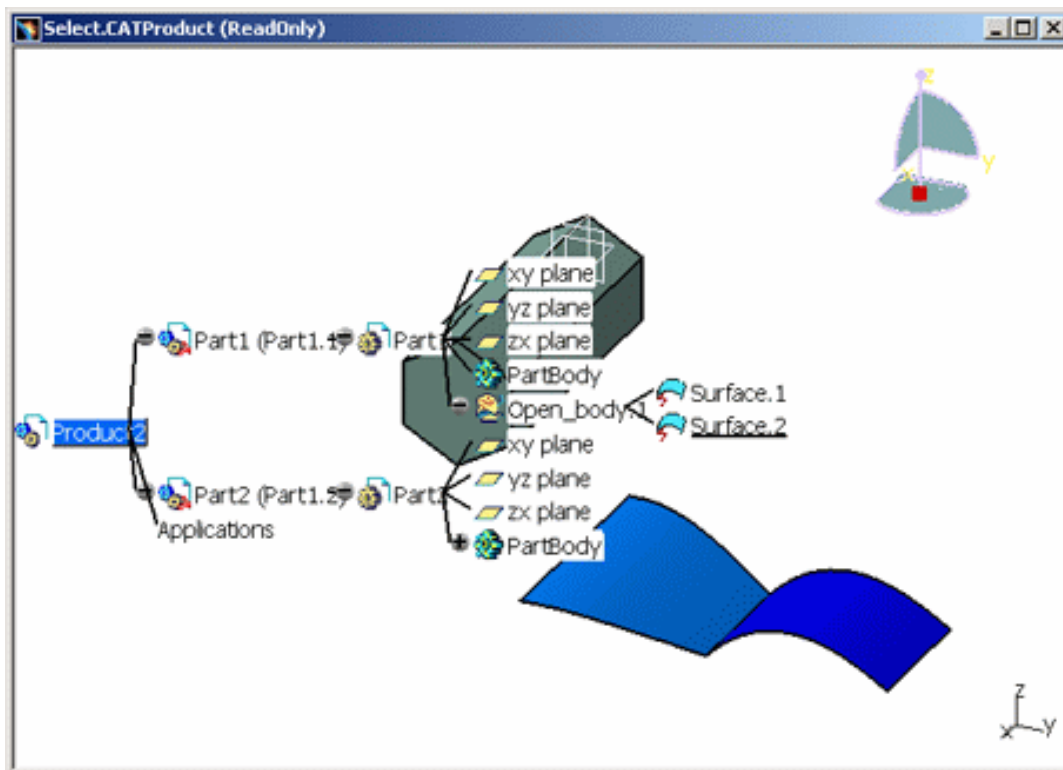
- Vertical



As you can see, this option shows all possible relations between the different items that go to make up the part.

- **Horizontal**

To do this, go back to the **Tools->Options...** dialog box and select the Horizontal option in the Tree Orientation section:



This option could, for example, be used for surface creation.

You can also change the background colors using the Visualization tab via the **Tools->Options...** function.




If the specification tree prevents you from seeing the geometry, you can move the tree by first clicking the reference axis (the geometry is displayed in low highlighting mode), then dragging the tree to a new location using the middle mouse button.


## Tree Item Size

This option enables you to specify the number of characters used for the names of the items displayed in the specification tree.

The default option is "Text-dependent", which means that each item name will be displayed as a whole. If you check the "Fixed" option, the field displayed to the right is activated to let you enter the fixed number of characters that should be displayed for item names.

## Tree Show/No Show

When this option is checked, if you select an item in the tree, then select the **Hide/Show**  icon, the item is grayed out in the tree.

If this option is unchecked, you will not be able to gray out an item in the tree. Clicking the **Hide/Show**  icon will have no effect.

When the size of the tree exceeds the window size, a scrollbar appears: you can then move the tree and navigate using the scrollbar.

3. Click **OK** to confirm.



# Setting Specification Tree Manipulation Options



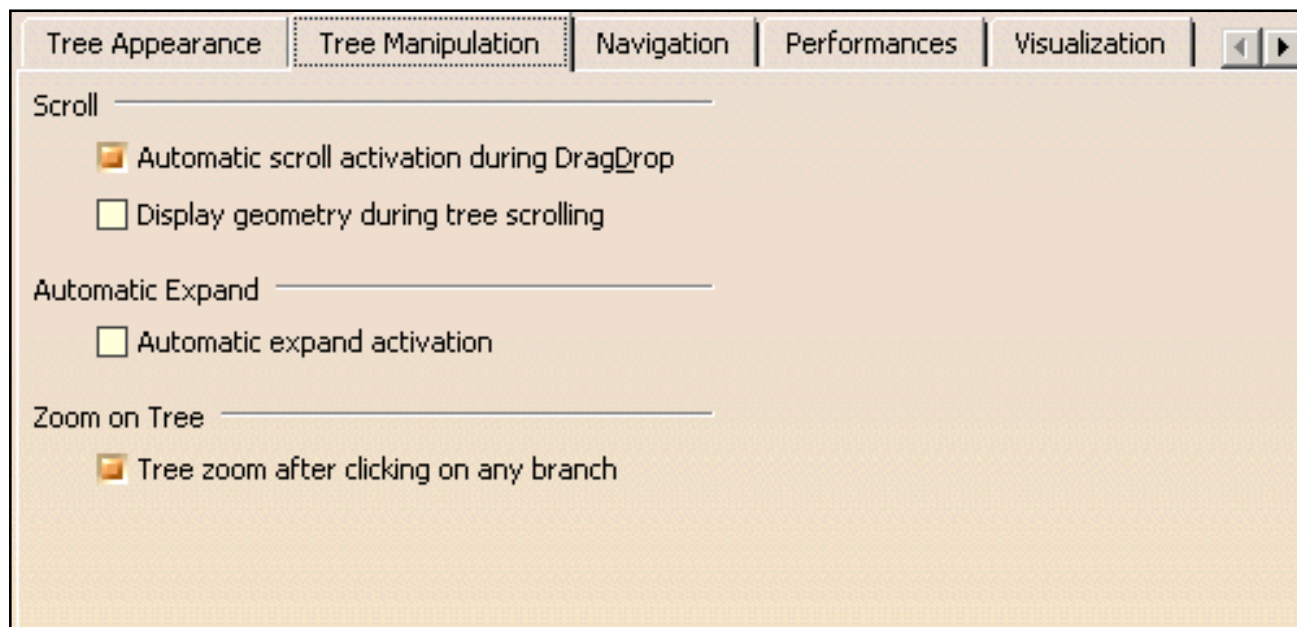
This task explains how to set specification tree manipulation options.



1. Select the **Tools->Options...** command.

The Options dialog box appears with the **General** category selected in the left-hand column.

2. In the **Display** sub-category, select the **Tree Manipulation** tab to display the document layout options.



## Scroll

- **Automatic scroll activation during drag and drop**  
When checked along with the "Activate the Visualization of Show/No Show" option, this option lets you scroll the tree automatically when dragging and dropping icons or objects onto it.  
The tree is scrolled until you remove the mouse from the scroll area.

For more information on dragging and dropping, refer to [Dragging and Dropping Icons and Objects](#) in this guide.

- **Display geometry during tree scrolling**

When checked, this option enables you to see the geometry while you are scrolling the specification tree using the scroll bar displayed to the left.

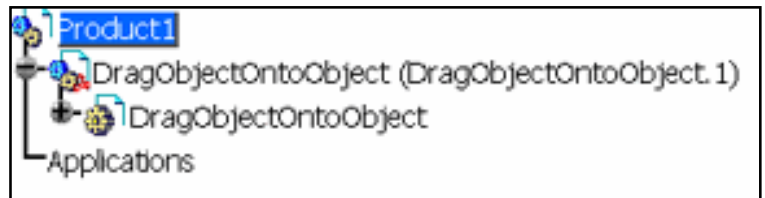
Note that this option has no impact when scrolling the specification tree using the mouse scroll button or when dragging and dropping icons with the "Automatic scroll activation during Drag and Drop" option activated: in both cases the geometry is displayed.

## Automatic Expand

When checked, this option activates the automatic expansion of the specification tree up to the second level as shown below:

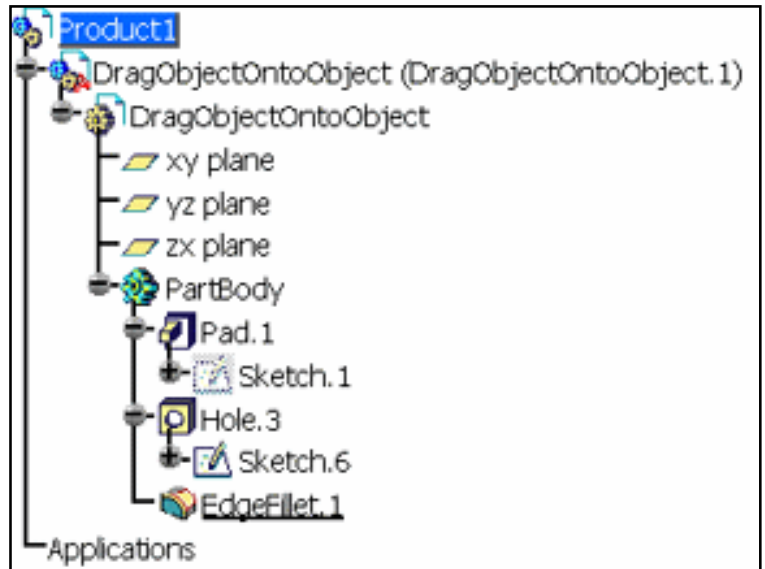
### Example 1

When clicking the + sign next to the first "DragObjectOntoObject" item with the automatic expand not activated:



### Example 2

When clicking the + sign next to the first "DragObjectOntoObject" item with the automatic expand activated:



When the automatic expand is activated, a message informs you that the automatic expand of the specification tree will be taken into account the next time you open a document.

## Zoom on Tree

When this option is checked, if you click any branch in the tree, the geometry is dimmed and only the tree is active, enabling you to manipulate it on its own (zoom it up and down, etc.). This option is activated by default.

When deactivated, the only way to manipulate the graph is to click the reference axis in the bottom right corner of the document.

3. Click **OK** to confirm.



# Navigation

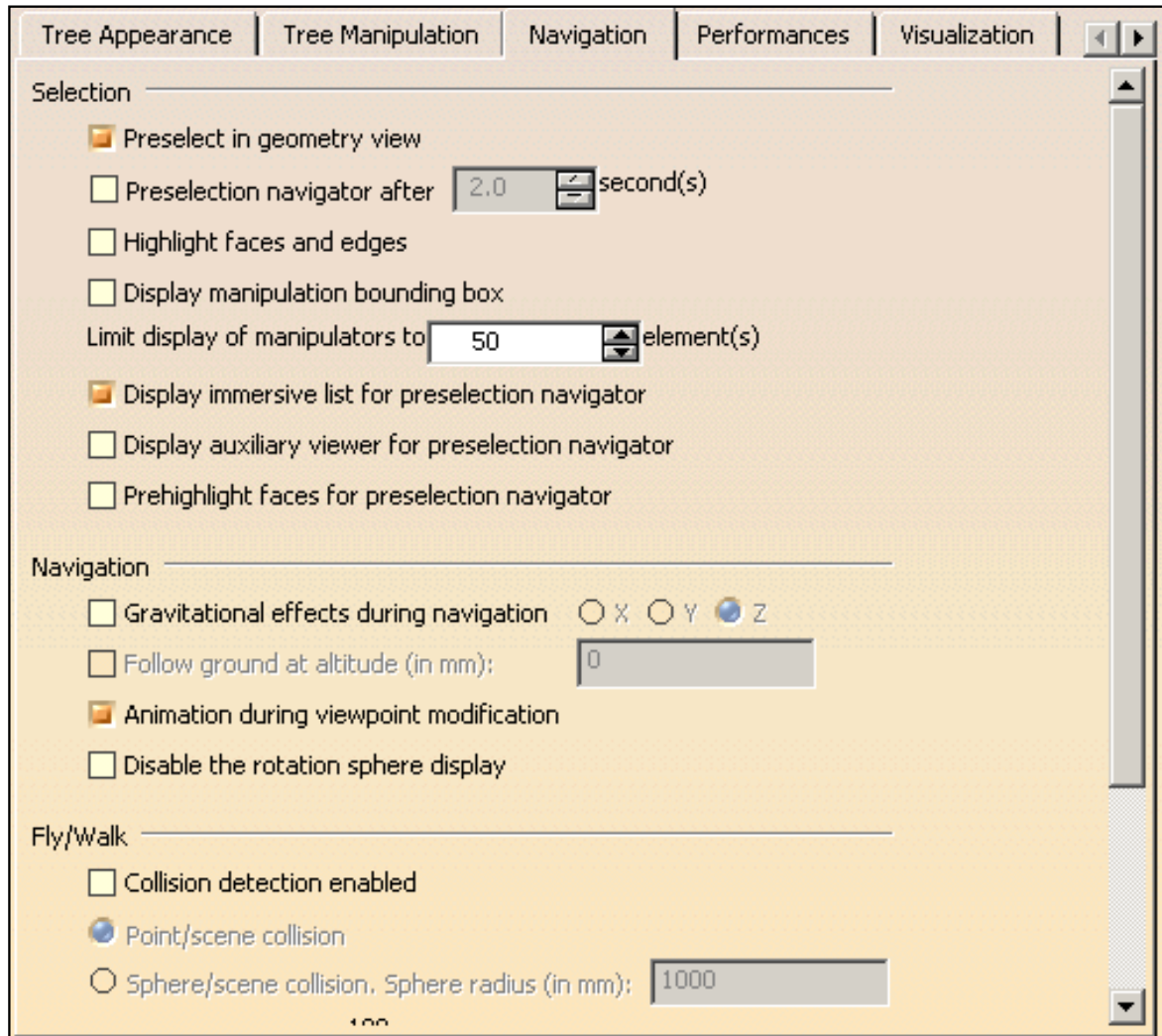


This task explains how to customize display navigation settings.



1. Select the **Tools->Options...** command.

2. In the **General** category, select the **Display** sub-category then the Navigation tab:



The Navigation tab lets you customize:

## Selection

- **Preselect in geometry view:** activates preselection highlighting. As you point to objects, different parts of the objects are highlighted in the geometry area, and the object name is highlighted in the specification tree. However, note that deactivating this option impacts all commands using prehighlight, especially the **Other Selection...** command that does not work anymore when it is based on prehighlight.
- **Preselection navigator after ... second(s):** sets the amount of time, in seconds, which elapses before the preselection navigator appears after pointing at an object. Refer to ["Selecting Using the Preselection Navigator"](#) for more information about how to use the preselection navigator. For more information about navigating in fly mode, refer to ["Navigating in Fly Mode"](#).
- **Highlight faces and edges:** controls the way faces and edges are prehighlighted and highlighted.

How elements are prehighlighted and highlighted:

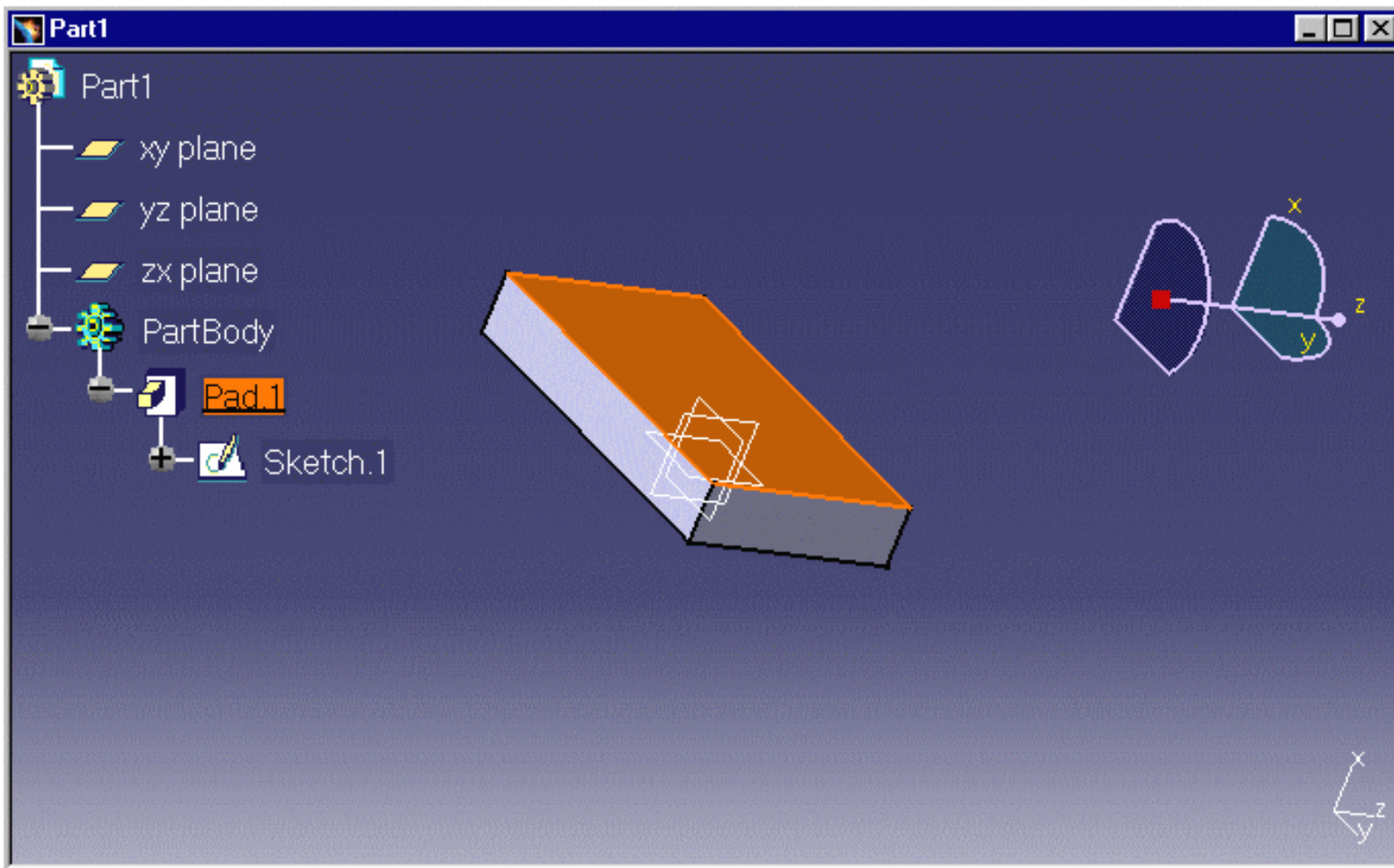
- is determined by whether you are in Design mode (editing an element in the context of an assembly) or in Visualization mode
- and varies according to the current visualization mode: shading with edges, shading, wireframe (NHR), dynamic hidden line removal (HRD). Note that some minor differences in the way elements are highlighted are noticeable using the HRD mode.

The default prehighlight and highlight colors are different, and can be customized using the Visualization tab. By default, faces and edges are highlighted. Depending on the element type, elements may or may not be displayed using the Z-buffer.

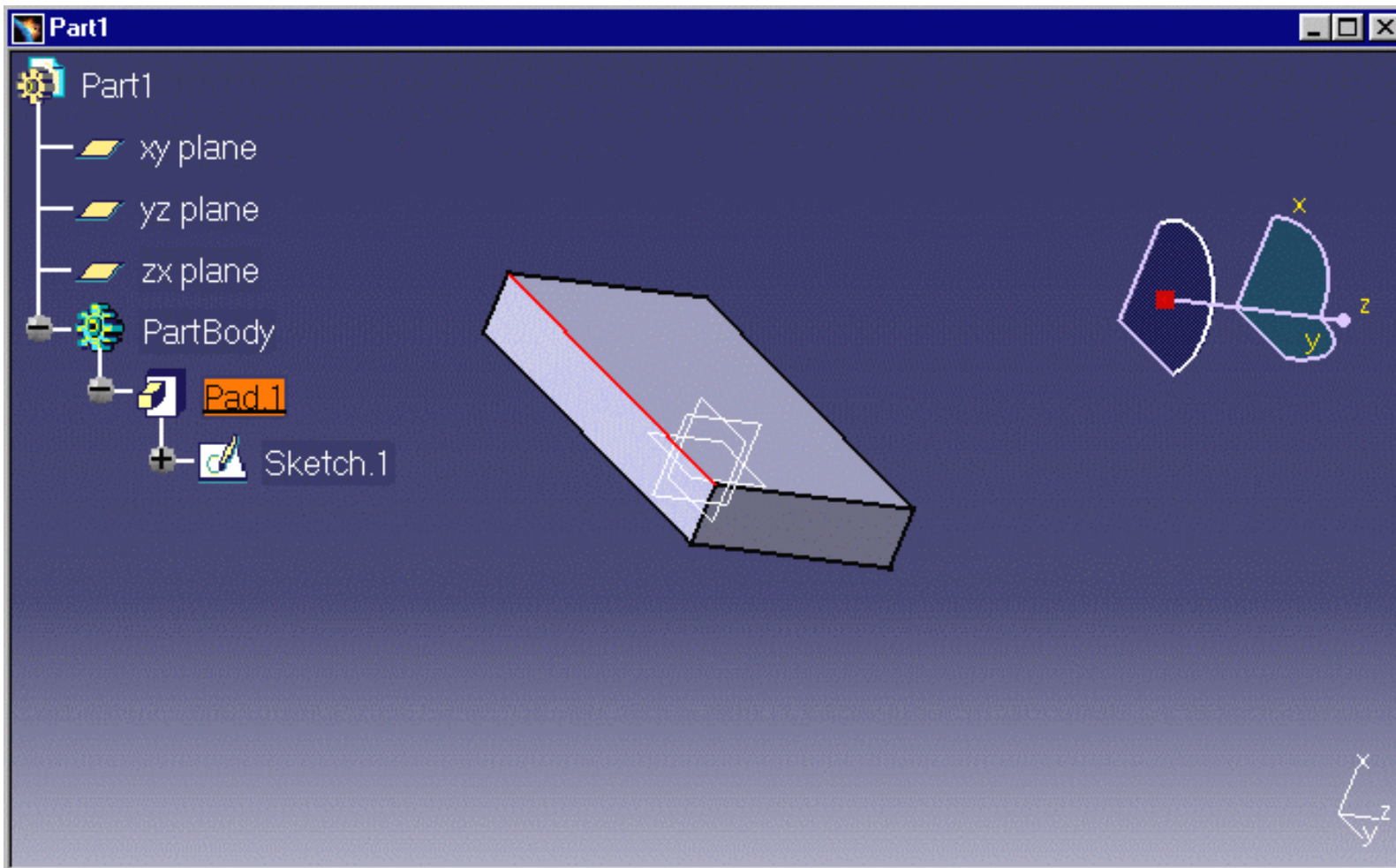
The following examples use the shading mode.

When you are editing an object, a selected face is highlighted like this:

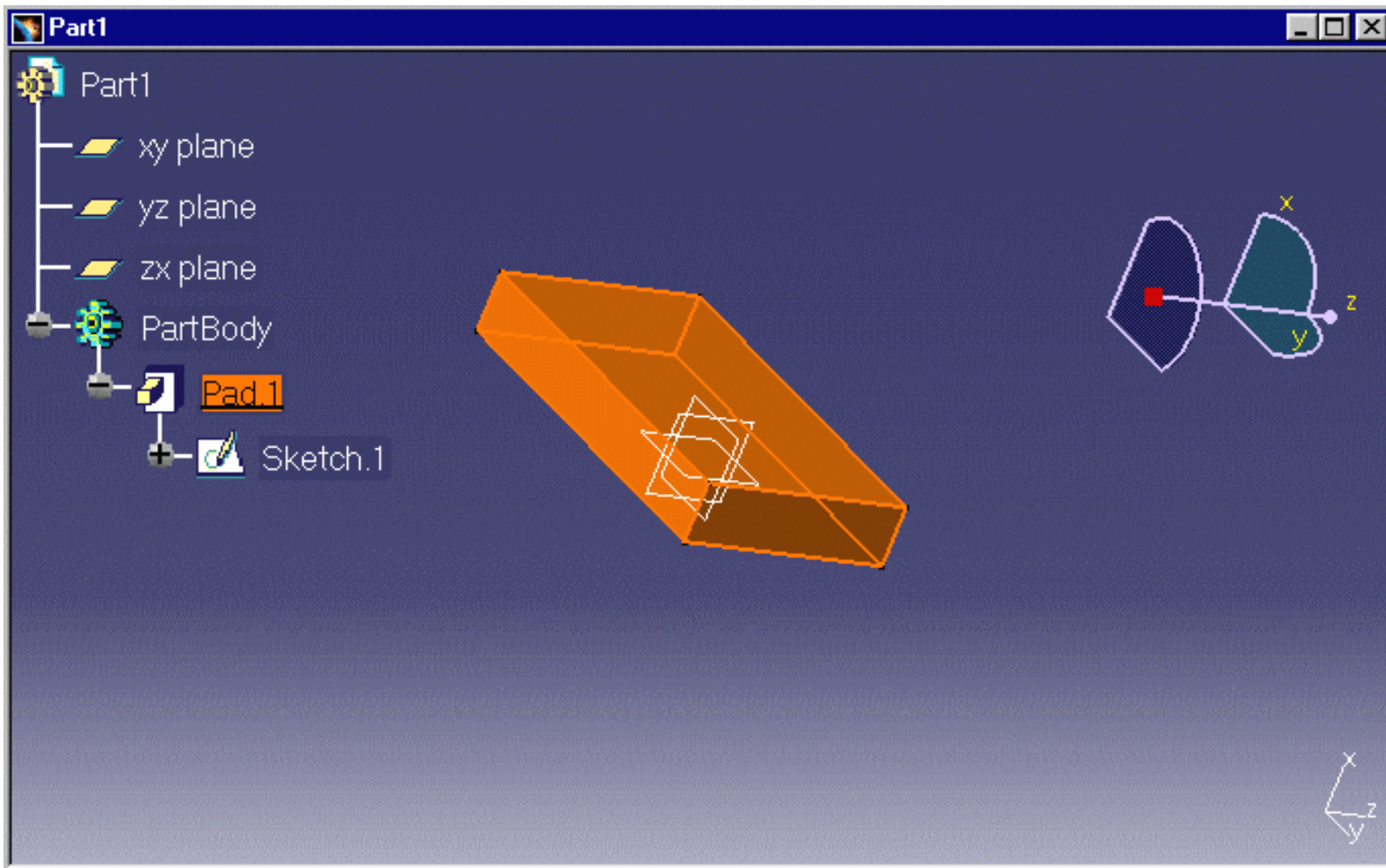




... and a selected edge is highlighted like this:

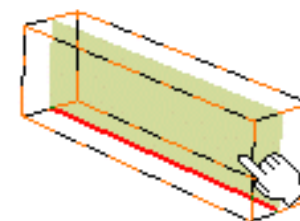


If you select the Pad or the PartBody in the specification tree, the whole object is highlighted like this:



If you are in Visualization mode in an assembly, the whole object is highlighted.

- **Display manipulation bounding box:** when clicking on an object, displays a box around the selected object if it uses manipulators:



Objects using manipulators can be manipulated by the 3D compass. For more information about manipulating objects using the 3D compass, refer to "[Manipulating Objects Using the Mouse and Compass](#)".

- **Limit display of manipulators to ... element(s):** sets a limit on the number of elements selected in multi-selection mode (using the selection trap) on which manipulators can be displayed. In case you select more elements than the limit (for example, by using the CTRL key to extend the selection), no element can be manipulated.

This option is activated when working in a 2D context (objects with handles such as texts and arrows, for instance) or 3D context.

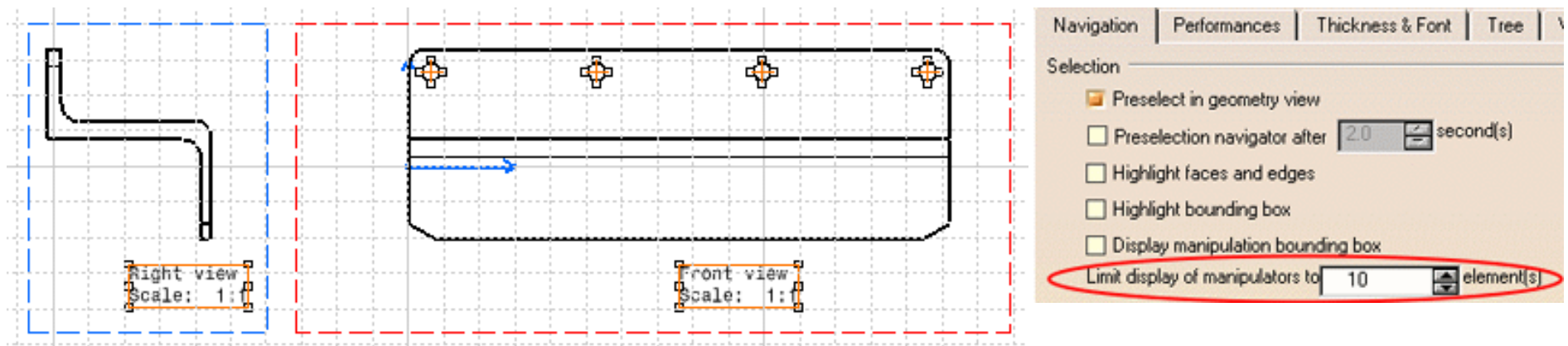
The following scenario uses the Drafting application as an example:

1. Enter a value in the Limit display of manipulators field, "10" for instance then click OK to validate.

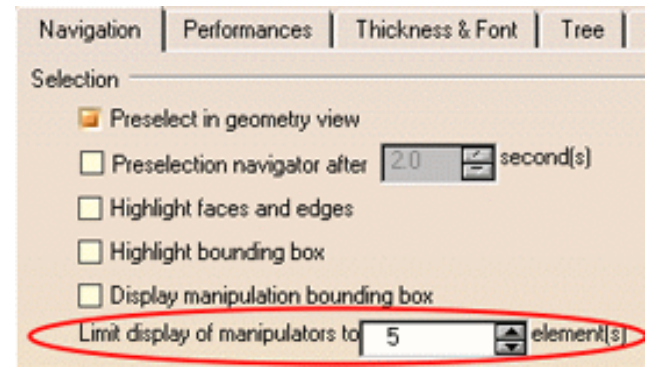
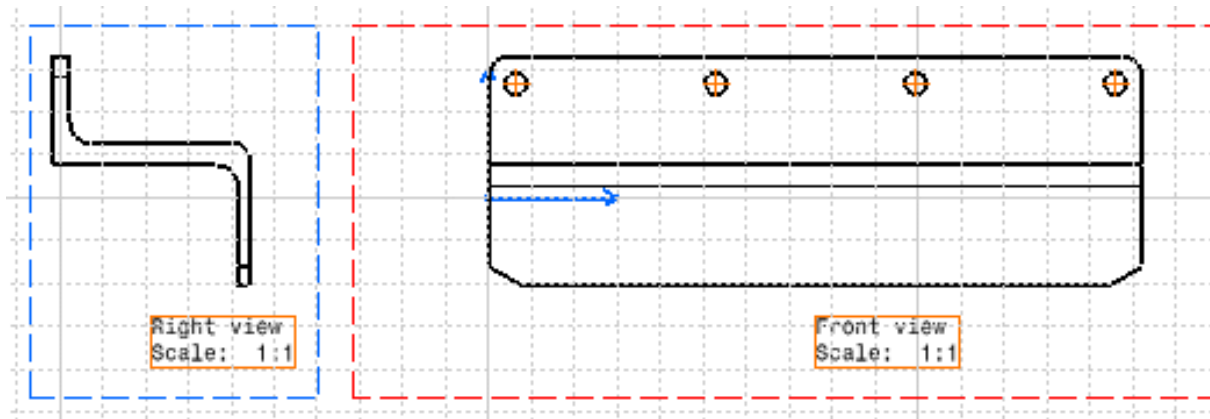
Note that setting a high number (greater than "50") will have an impact on the content of the contextual menu displayed when right-clicking a selected element.

2. In the geometry area, select six elements either using the selection trap or the CTRL key.

As you can see it below, manipulators are displayed on the selected elements to let you modify them:



If you set the limit to "5" instead of "10", no manipulator will be displayed since the number of selected elements (i.e. six) exceeds the limit:



- Display immersive list for the preselection navigator:** this option is activated by default to display the list of all the elements you can preselect when using the [preselection navigator](#).  
 If you do not want to display the immersive list, uncheck the option.
- Display auxiliary viewer for preselection navigator:** checking this option activates an auxiliary viewer displaying the preselected object when using the [preselection navigator](#). This viewer enables you to perform viewing operations with the mouse such as zoom, rotate or pan.
- Prehighlight faces for preselection navigator:** this option lets you activate the highlight of the face(s) you preselect using the [preselection navigator](#).




## Navigation

- Gravitational effects during navigation:** fixes the X, Y or Z axis during navigation. While turning in Fly mode, this creates the impression that the user viewpoint tilts or banks with respect to the fixed axis, as in a real plane.

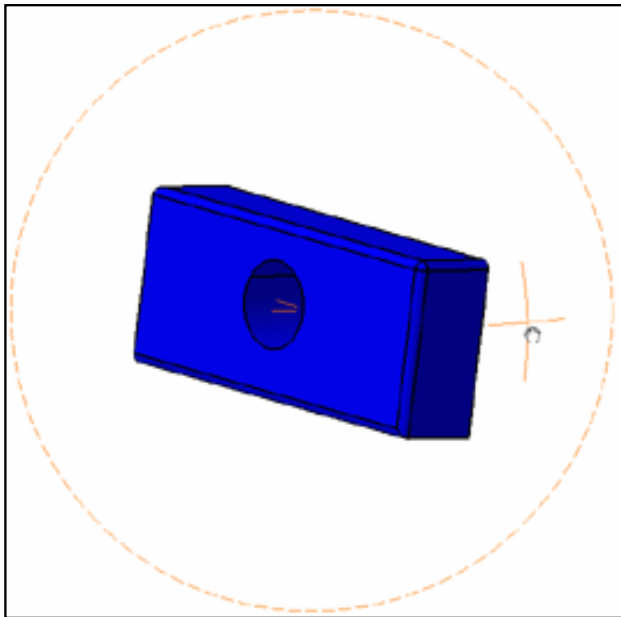
- **Follow ground at altitude (in mm):** first check "Gravitational effects during navigation" to be able to activate this option. When flying and walking, this gravitational effect makes you automatically fall if you do not reach the specified altitude (in millimeters). This allows you to follow the ground relief, such as stairs.  
An altitude set to "0" means that the eye level will be set at ground level, whereas an altitude higher than "0" will set the eye level above ground level.

Note that the "Follow ground" mode is automatically deactivated when it is not relevant, i.e. when no ground is detected.

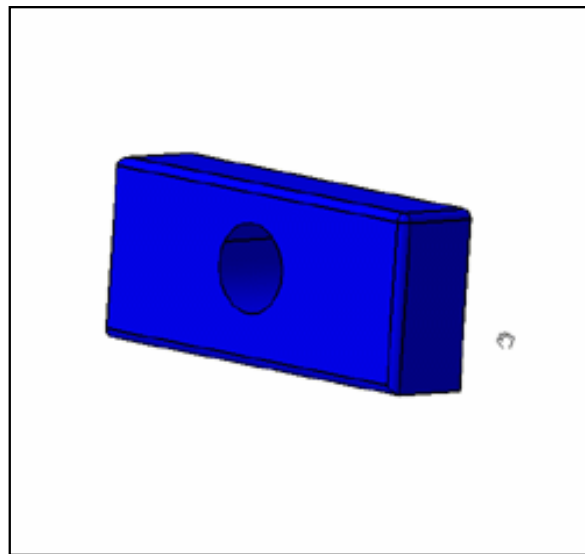
- **Animation during viewpoint navigation:** set this option if you want viewpoint changes in certain contexts to be animated. To see an example of the effect of this option, check the option, then select a plane and click the Sketcher  icon. The selected plane is slowly rotated until parallel to the screen just like during an animation. If you do not check this option, the selected plane is set parallel to the screen immediately (without the animation effect).

**Note:** the visualization time of the product impacts the number of animation steps. The longer the duration, the less steps. In case the visualization takes too much time, there is no viewpoint animation at all.

- **Disable the rotation sphere display:** hides the rotation sphere usually displayed when [rotating](#) an object. The result is as follows:



Rotation sphere activated (default mode)

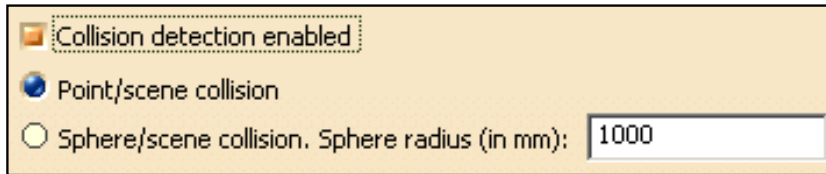


Rotation sphere deactivated

## Fly/Walk

- **Collision detection enabled**

Checking this option activates the following two options:



- *Point/scene collision*: when flying and walking, detects if you collide with an object, so that you bounce off the object instead of going through it. This is the default option.
- *Sphere/scene collision*: this options works the same way as the "Point/scene collision" option but this time, the user is modeled as a sphere whose radius can be defined by entering a value (in millimeters) in the field to the right. When flying and walking, the option detects if the sphere collides with an object, so that you bounce off the object. As no object can break through the sphere, this provides a more "immersive" experience.

Note that this detection collision mode works with complex geometry since it is based on visualization triangles. Note also that, when used in combination with the "Follow ground" option, this option can provide video game Quake-like navigation effects.

However, bear in mind that there is a price to pay in performance when using large models.

- **Mouse Sensitivity**: sets the mouse sensitivity when flying and walking: if you set a low value, pointing the cursor produces only slow, minor changes in direction; if you set a high value, pointing the cursor produces rapid, significant changes in direction.
- **Start Speed**: when you begin to fly, this option lets you choose between two options:
  - *Automatic*: you will begin to fly at the default speed, i.e. the speed at which you will first approach the object will depend on the initial distance from the object and will be calculated automatically
  - *Custom*: you can define by yourself the speed at which you will first approach the object using the slider.


- **Mouse Speed:** sets the time interval (from 0 to 100 milliseconds) during which mouse movements will not be taken into account for prehighlight purposes: the higher the speed, the fewer elements are prehighlighted.
3. Click **OK** to confirm, or **Reset...** and **OK** again to reset default settings.

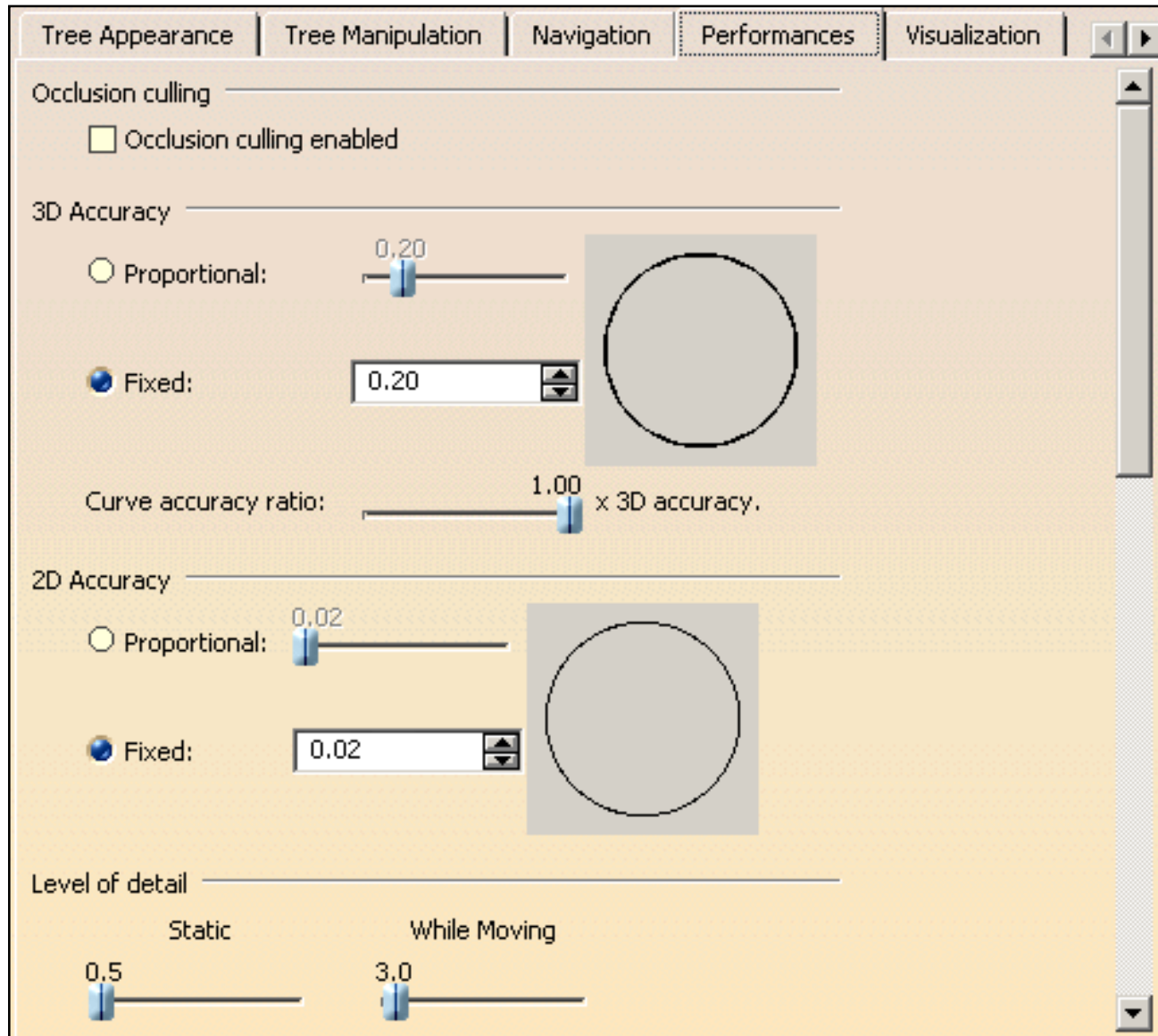




# Performance

 This task explains how to customize display performance settings.

-  1. Select the **Tools->Options...** command.
2. In the **General** category, click the **Display** category then the **Performances** tab:

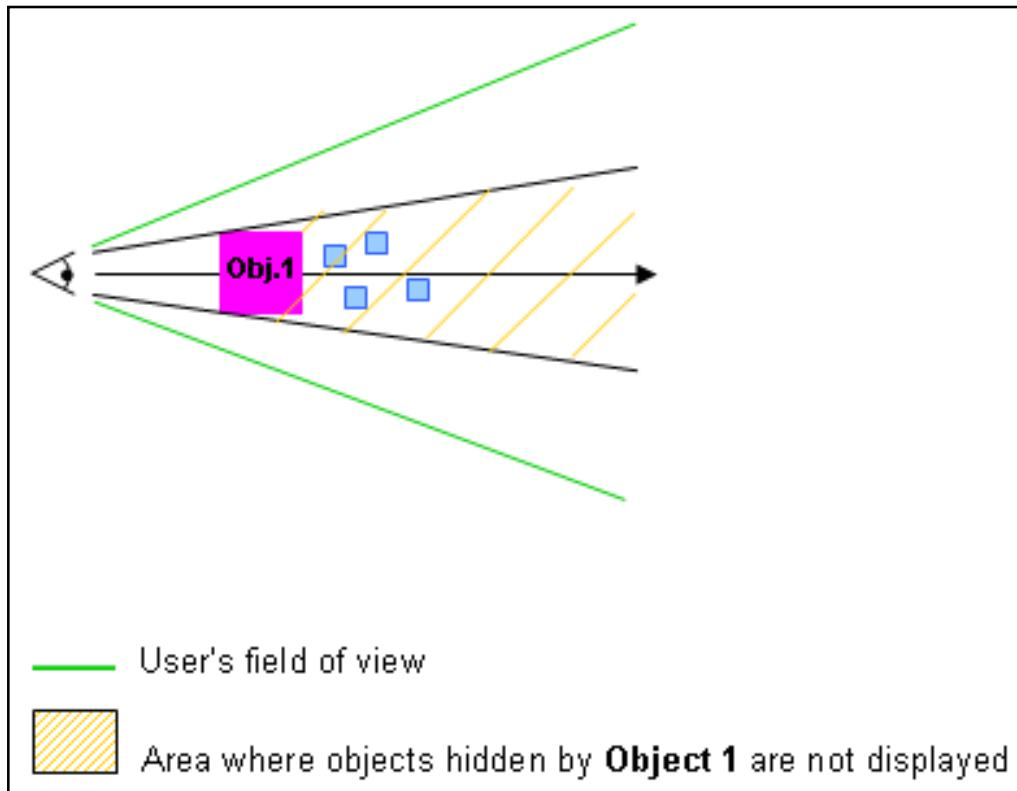


## Occlusion Culling

Click "Occlusion culling enabled" to activate occlusion culling.

Occlusion culling avoids redisplay of hidden elements, particularly useful when viewing highly compartmented scenes such as plants and buildings but it does not improve display performance.

Let's have a look at the picture below to see what happens when occlusion culling is activated:



## 3D Accuracy and 2D Accuracy

The accuracy setting controls the tessellation of surfaces. "Tessellation" means that the surfaces of your geometry are built using triangles. A triangulation is computed to describe the neighborhood relation of all points.

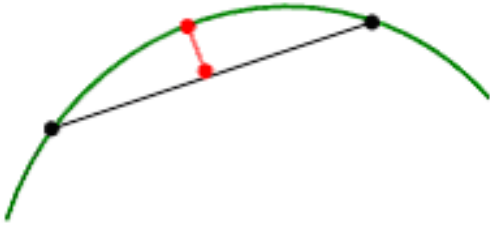
Note that 2D accuracy settings are the same as for 3D.

You have two choices (the preview area to the right shows you the effect of each setting):

- **Fixed:** sets a fixed sag value (from 0.01 to 10) for calculating tessellation on all objects, which does not vary with object size.

The sag value defines the chordal deviation for curves and surfaces.

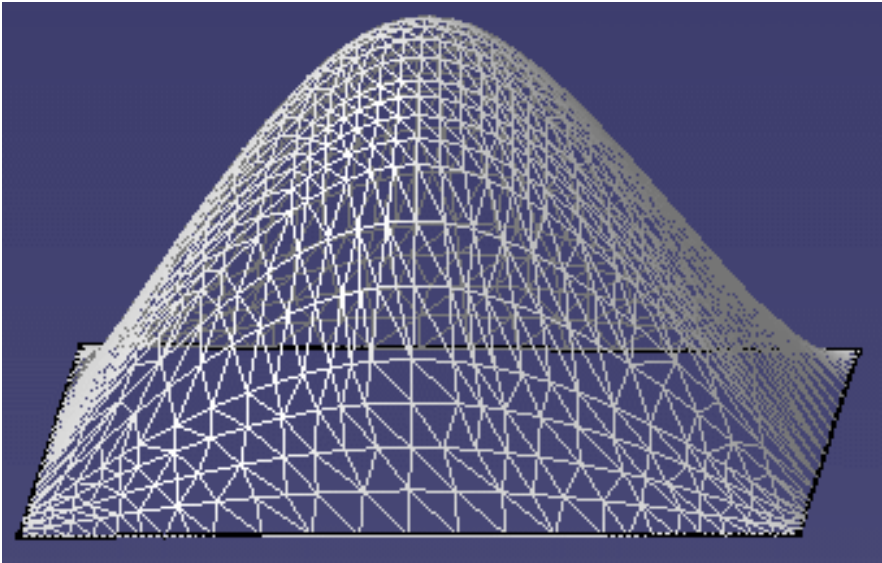
The "curve chordal deviation" represents the maximum distance between a polyline ("chord") whose end points lie on a curve and a point on this curve:



●● = Distance

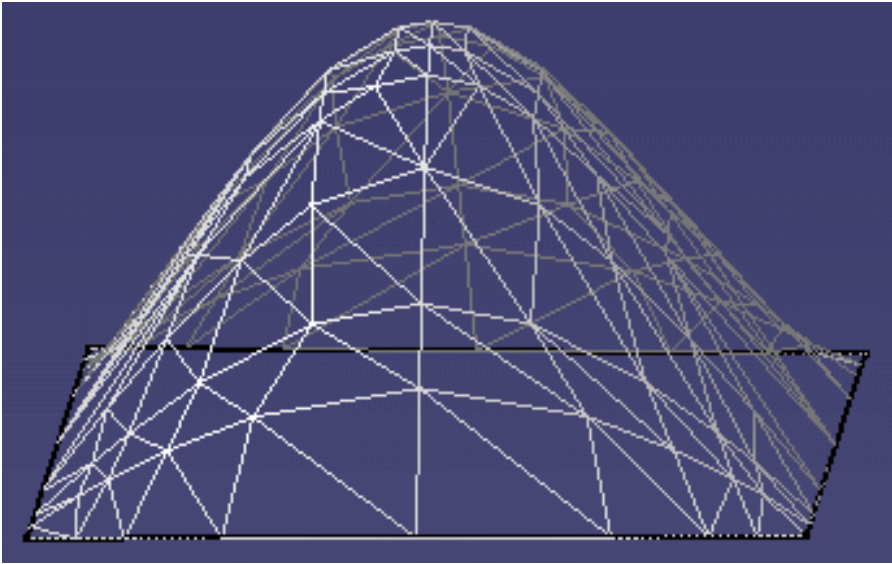
The "surface chordal deviation" represents the maximum distance between the tessellation triangles and the surface.

- a low value means that a very fine mesh is used to render surfaces because the distance between the geometry and the triangles in the tessellation is very low. However, the drawback is that geometry will be redrawn more slowly when using the viewing tools.



Example with default fixed sag value ( = 0.20)

- a high value increases the distance between the geometry and the triangles and thus, decreases the number of triangles that will be computed on the object. This means that a very coarse mesh is used, but the advantage is that geometry will be redrawn more quickly.



Example with default fixed sag value set to 8.5

- **Proportional to element size**

Tessellation is calculated according to object size: the larger the object, the coarser the tessellation. For the same sag value, the tessellation on small objects will always be finer than on large objects.

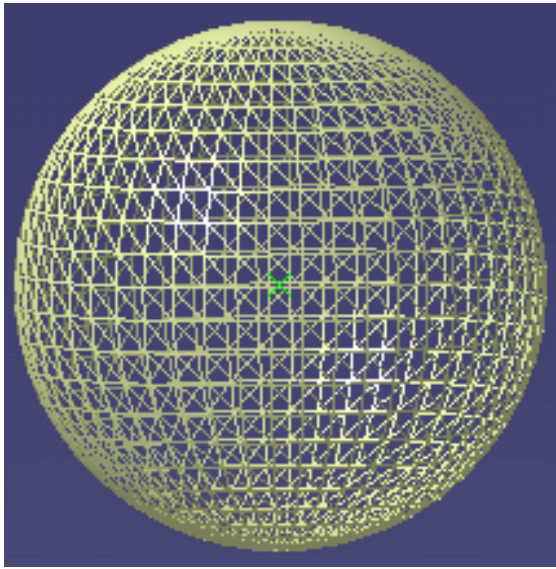
The sag value used to calculate the tessellation of each object is calculated is as follows:

$$\text{sag} = \text{coeff.} \times \text{radius of sphere} / 100$$

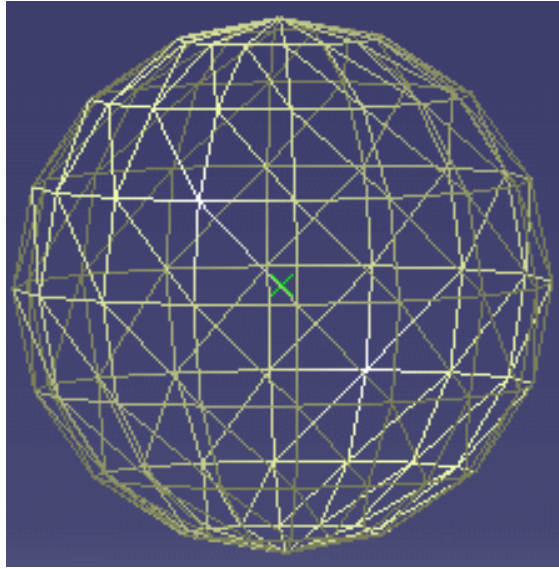
where:

- "coeff." is the value you set using the slider (between 0.1 and 1)
- "radius of sphere" is the radius of a sphere encompassing the object entirely (this value is obviously higher on larger objects).

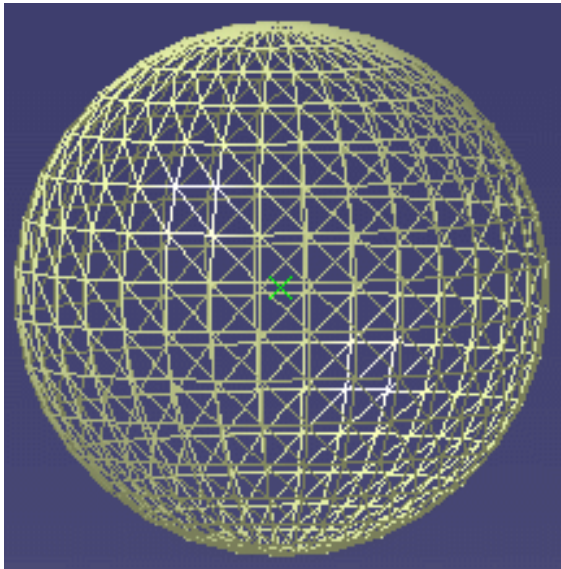
The following examples show the difference between a fixed and a proportional sag value applied to two spheres with different radiuses:



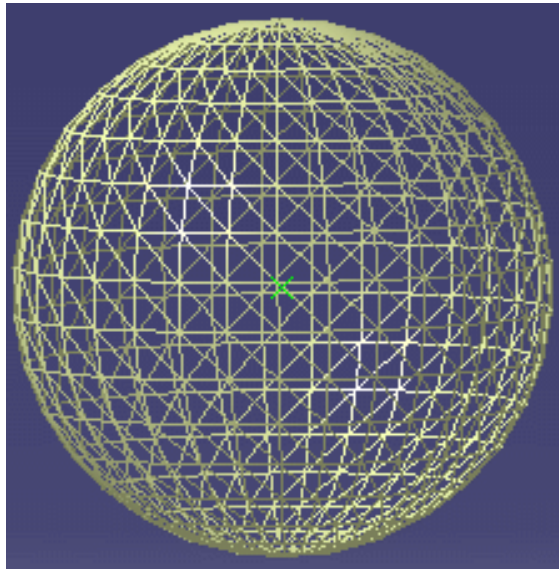
Radius = 100 mm  
Fixed sag value = 0.20



Radius = 10 mm  
Fixed sag value = 0.20



Radius = 100 mm  
Proportional sag value = 0.20



Radius = 10 mm  
Proportional sag value = 0.20



Start by setting a high fixed value in order to decrease the number of tessellation triangles and thus, pay a lower price in performance.

**Warning:** when working with V4 models, be careful not to set a too high sag value for 3D accuracy, otherwise you will not be able to visualize them.

- **Curve accuracy ratio**

This option lets you control the curve accuracy using the slider displayed to the right (you can choose a value between 0.1 and 1).

The curve accuracy is a ratio of the 3D accuracy you define using the above-detailed 3D Accuracy option.

The curve accuracy is calculated as follows:

$$\text{sag} = \text{3D accuracy} \times \text{ratio}$$

As a consequence, the tessellation on curves will be finer than the tessellation on surfaces.

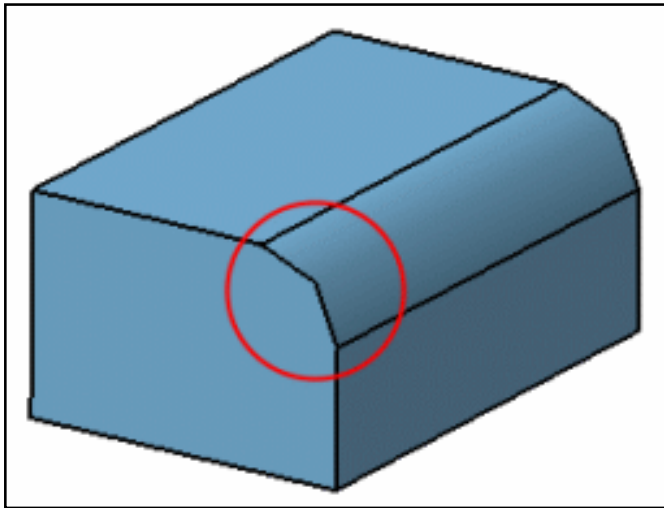
For instance, setting the 3D sag value to 0.20 and the curve accuracy ratio to 0.10 means that:

- faces will be tessellated with a 0.20 sag
- curves will be tessellated with a 0.02 sag (0.20 x 0.10).

**Example 1**

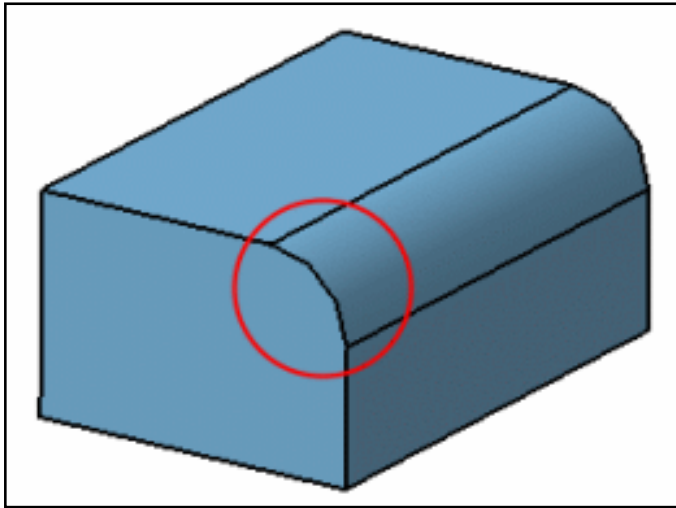
Fixed 3D accuracy = 10

Curve accuracy ratio = 1

**Example 2**

Fixed 3D accuracy = 10

Curve accuracy ratio = 0.10



## P2

### Level of Details

You do not always need to view a high level of detail in your geometry all the time. You can use the Static and While Moving settings to add or remove display quality:

- **Static:** even if you do not want to move geometry, it is often useful to remove details you do not need to see. Set a low value if you want to see all the details, or a high value to remove details
- **While Moving:** you will be able to move large parts more quickly if you set While Moving to a high value. When you release the mouse after moving the part, the normal level of detail will be redisplayed.

In both cases, the higher the value, the lower the level of detail. Normally, you set Static to a low value, and While Moving to a high value. The added value is increased display performance.

Note that if you are using the Level of Details option with Product Structure or DMU Navigator functions, you must activate the cache for your LOD settings to be taken into account.

### Pixel Culling

This option is similar to Level of Details, but in that case, it lets you define the size in pixels of objects to be hidden/displayed in your geometry using the Static and While Moving settings.

- **Static:** even if you do not want to move geometry, it is often useful to remove details you do not need to see. To do so, set a high value to remove these details. On the contrary, setting a low value displays the details. For instance, setting "2" means that objects whose size on screen is lower than 2 pixels will be static
- **While Moving:** setting a high value will enable you to move large parts more quickly. When you release the mouse after moving the part, the Static size will be redisplayed.

## Transparency Quality

- **Low (Screen Door)**: similar to viewing an object through a mesh or a screen. Use this setting when you need to look at objects through another transparent object. This mode is also recommended for increased display performance.  
When checked, the transparency is not impacted whatever value you set (between 1 and 255).
- **High (Alpha Blending)**: similar to looking through clear glass. Use this setting when you need to view several transparent objects located at different depths of a scene. For example, looking through a car windscreen at other opaque objects inside the car. Note, however, that this mode is computation-intensive and consequently has an adverse affect on display performance.  
When checked, the transparency is impacted according to the value you set.

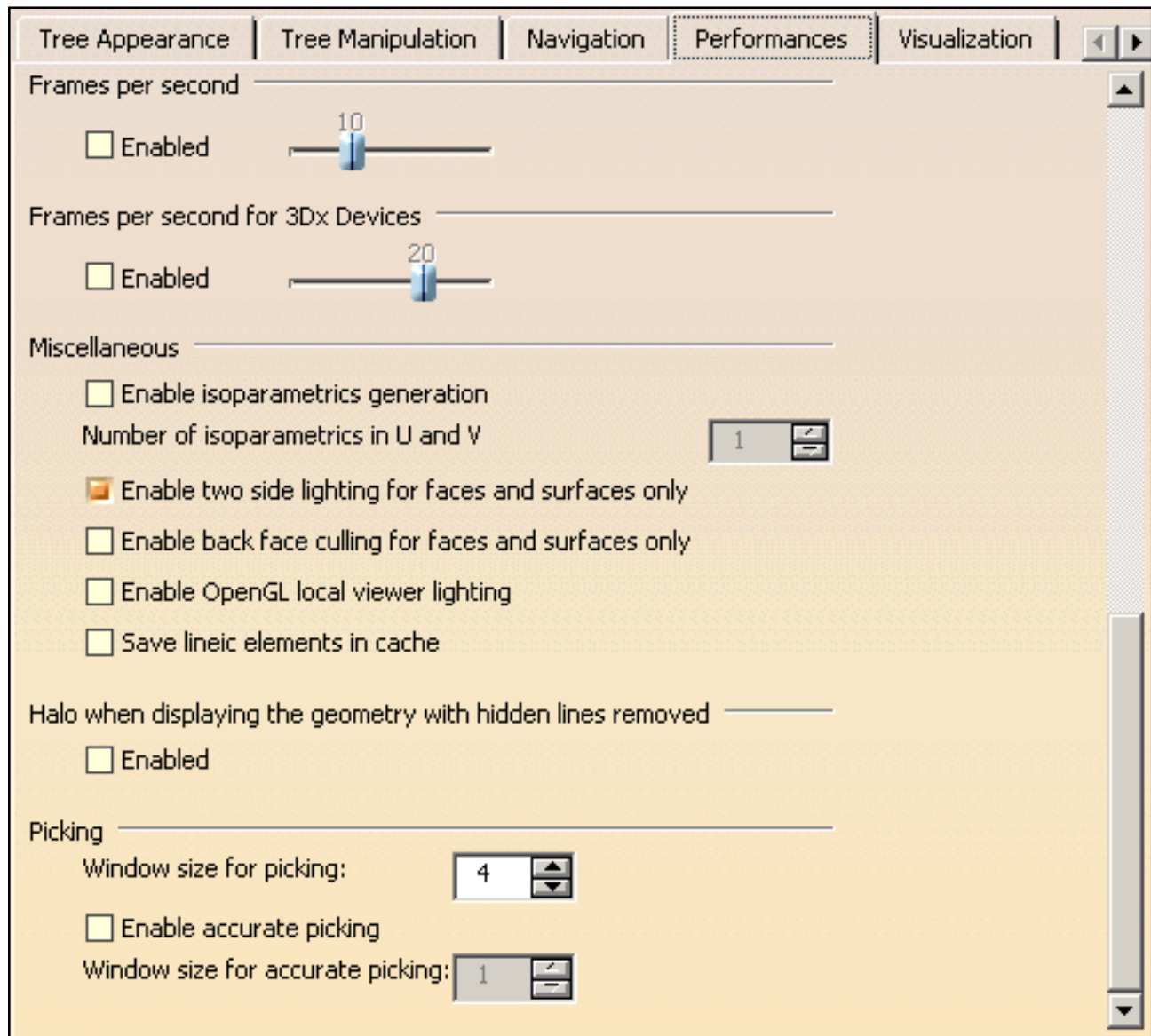
Note that you also need to set the transparency coefficient on selected objects using the **Edit->Properties** command or **Properties** command on the contextual menu, by dragging the Transparency slider on the Graphic tab. Refer to [Displaying and Editing Graphic Properties](#) for more information.

## Frames per second

Check the **Enabled** button if you want to control the minimum number of frames per second (frame rate) during animations (zooming, moving, flying,... etc.) when using a regular mouse. The frame rate varies from 1 to 30. Setting a low frame rate keeps a maximum number of details visible, but animations are less smooth and fluid; setting a high frame rate limits visible details, but provides smoother, more fluid animations.

This option is particularly useful, for example, in Fly mode: when flying within large objects, you may not need to see all the details, but you want to navigate through the object in as smooth a manner as possible.





## Frames per second for 3Dx Devices

Check the **Enabled** button if you want to control the minimum number of frames per second (frame rate) during animations when using a 3Dx device such as a Space Mouse. This option works the same way as the above-detailed "[Frames per second](#)", the only difference is that it is specific to 3Dx devices.

## Miscellaneous

**Show Hidden Edges (whenever possible):** note that the display of hidden edges is now activated in the [Custom View Modes](#) dialog box.

## Enable isoparametrics generation

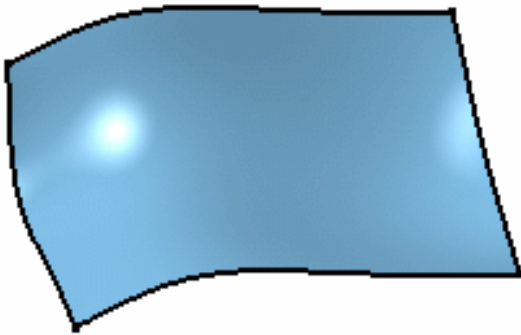
This option enables you to generate (in this context, it means to "display") the topological elements defined as being isoparametrics.

By default, the option is not checked for performance reasons.

To use this option, just follow the steps below:

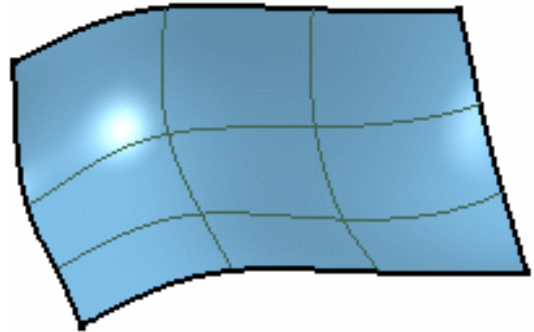
1. Check the "Enable isoparametrics generation" option: you are then asked to restart your Version 5 session
2. Restart your Version 5 session
3. Re-access the Performances tab then, if desired, specify the number of isoparametrics to be generated for u and v axes using the field displayed to the right. You can generate up to 10 isoparametrics.
4. Select the **View->Render Style->Customize View** command then check the "Isoparametrics" option which is now activated.

The isoparametrics are displayed:



Example 1

"Enable isoparametrics generation" and "Isoparametrics" options are off.

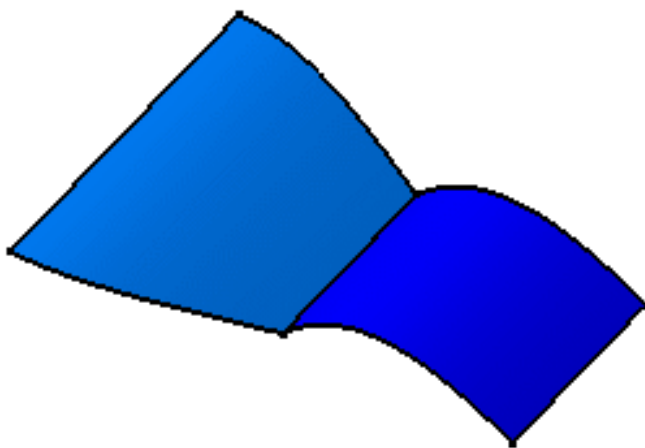


Example 2

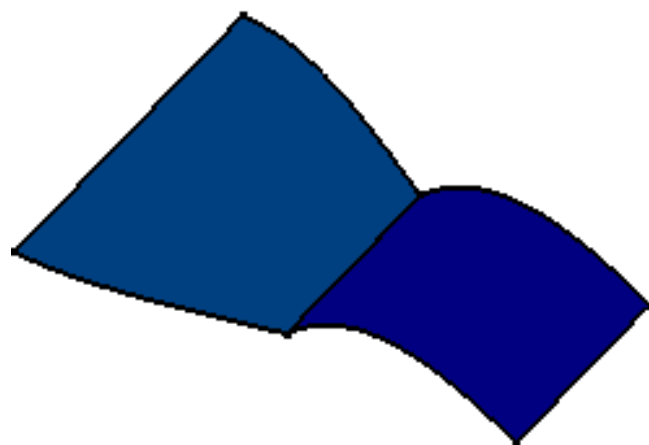
"Enable isoparametrics generation" and "Isoparametrics" options are on. 2 isoparametrics are displayed for u and v axes since we entered the value "2" in step 3.

## Enable two side lighting for faces and surfaces only

This option helps you viewing the two sides (i.e. front side and back side) of a face or surface by lighting them.



Example with option on



Example with option off

**Enable back face culling for faces and surfaces only**

Check this option to avoid redisplay of back sides of faces or surfaces.

Note that when this option is checked along with the "Enable two side lighting for faces and surfaces only" option, only front faces will be lightened.

**Enable OpenGL local viewer lighting**

When checked, this option provides a better lighting quality. However, bear in mind that there is a price to pay in performance.

## Save lineic elements in cache

This option lets you save lineic elements (such as wireframe elements, for instance) when saving files of type cgr.

## Enable OpenGL Shader

This option lets you apply OpenGL materials when working with the Real Time Rendering 2 product. These advanced materials are used to create textures such as paint, wood or marble.

For detailed information, refer to the section entitled "Advanced Materials" in the *Version 5 - Real Time Rendering User's Guide*.

To be able to use this advanced material, you need to download the OpenGL Shader™ development kit. To do so:

- if you are working on IRIX, browse the following site:

<http://www.sgi.com/software/shader/downloads.html>

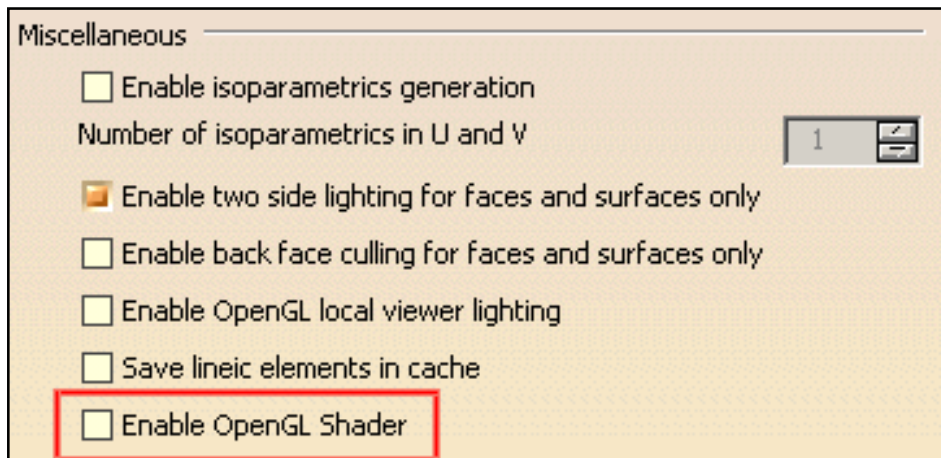
then download the corresponding kit.

- if you are working on Windows, browse the following site:

<http://www.sgi.com/industries/manufacturing/partners/catia/>

then follow the instructions.

The option will be displayed when re-accessing the Performances tab:

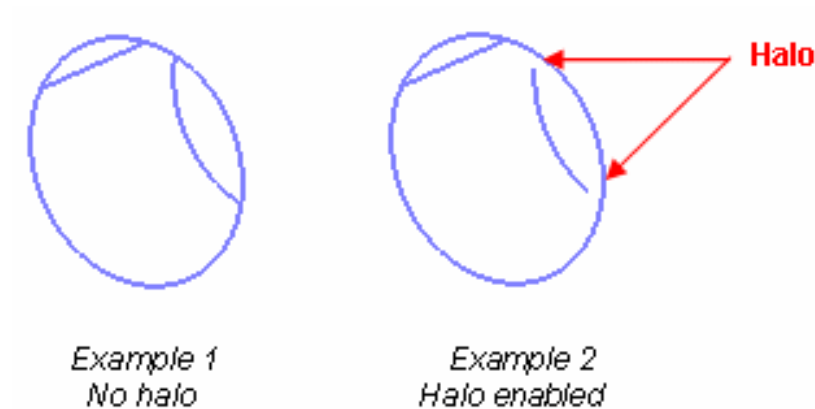


## Halo when displaying the geometry with hidden lines removed

This option lets you display a halo around intersecting edges to create a perspective effect.

Prior to using it, you need to switch to Hidden Line Removal mode otherwise, you will not be able to use this option. To do so, select the **View->Render Style->Customize View** then activate the "Dynamic hidden line removal" option before clicking **OK** to validate.

The following picture shows an example of halo:



## Picking

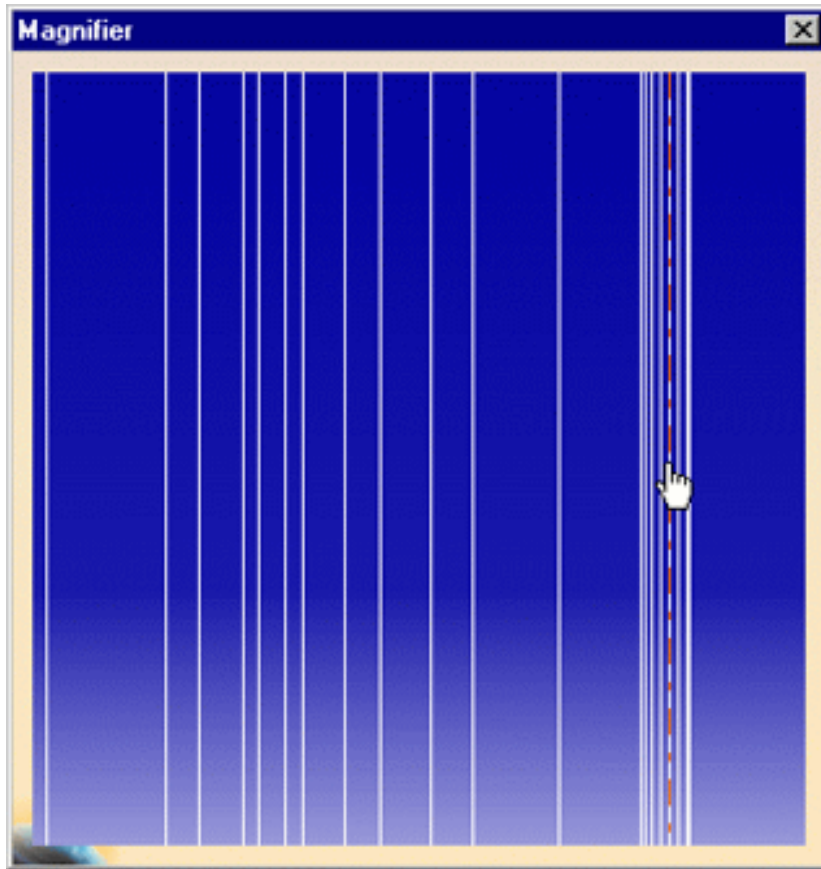
This area enables you to pick elements, i.e. select elements when pointing them. In "normal" picking mode, you need to move your mouse to select an element whereas in accurate picking mode, the slightest move (one pixel, for instance) is enough to select.

The "Window size for picking" field lets you specify the size (in pixels) of the window you will use for picking elements in normal mode. You can set a value comprised between 1 and 100, the default value being 4 pixels.

The smaller the value, the more precise the picking.

The "Enable accurate picking option" helps you select more precisely elements that are very close to each other when displaying the geometry.

It is recommended to work with a magnified view of your document by selecting the **View->Magnifier...** command as shown below:



For more information, refer to [Magnifying](#) in this guide.

When "Enable accurate picking option" is checked, the "Window size for accurate picking" option is activated to let you define the size of the window (in pixels) used for accurate picking.

The maximum value you can enter corresponds to the maximum value defined for the "normal" picking, i.e. the value entered in "Window size for picking".

Note that if you this option enables you to use normal picking as well.



Checking this option may have a negative impact on the performances when using big models.

When working on UNIX workstations, you can use the "Double click latency" slider to manage the maximum duration (in milliseconds) between two mouse clicks so that they could be considered as a double-click.

The default value is 500 ms but you can increase it up to 2 seconds.

**3.** Click **OK** to confirm.



# Customizing Visualization Settings

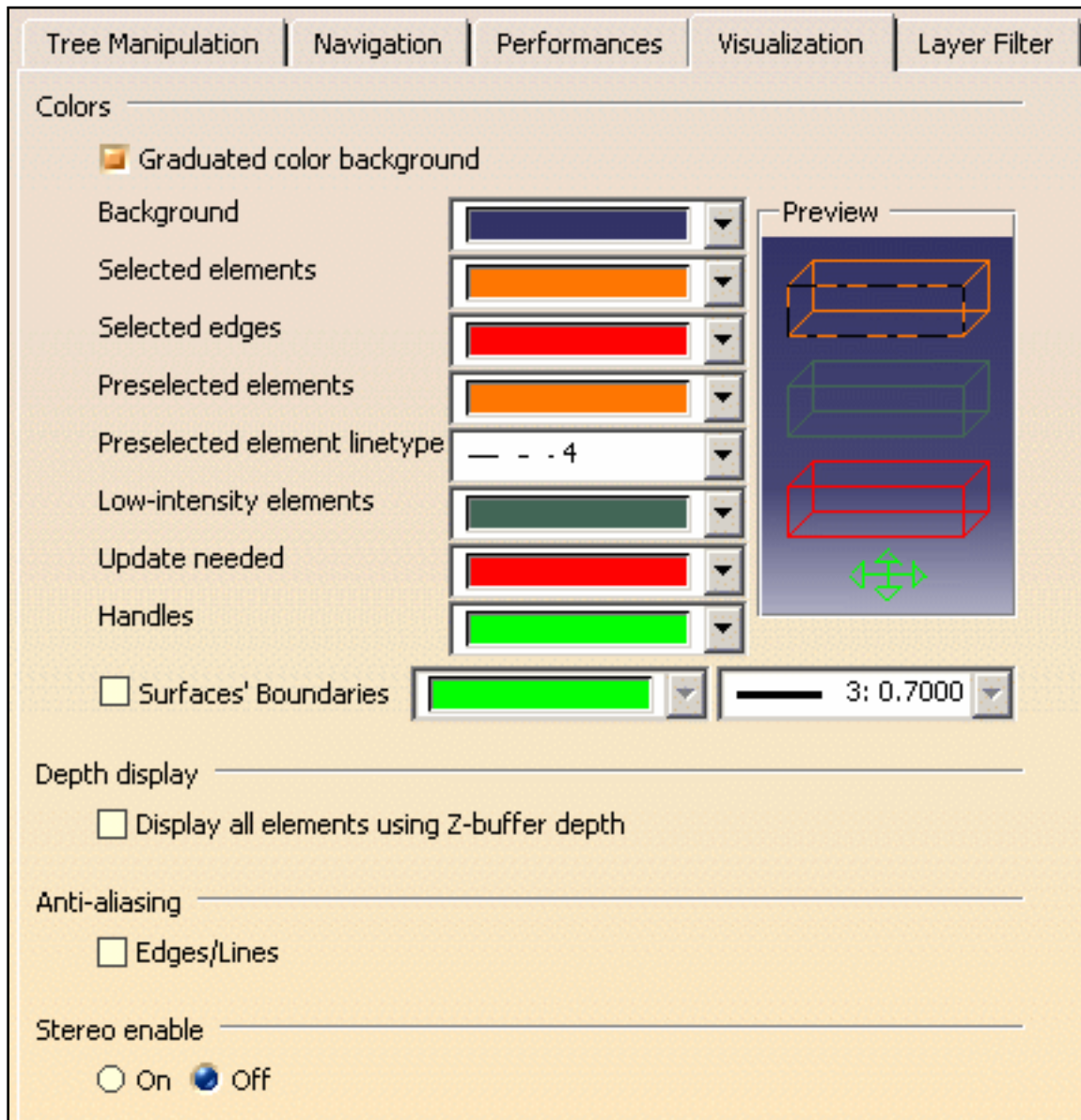


This task explains how to customize display visualization settings.



1. Select the **Tools->Options...** command.

2. In the **General** category, select the **Display** sub-category then the Visualization tab:



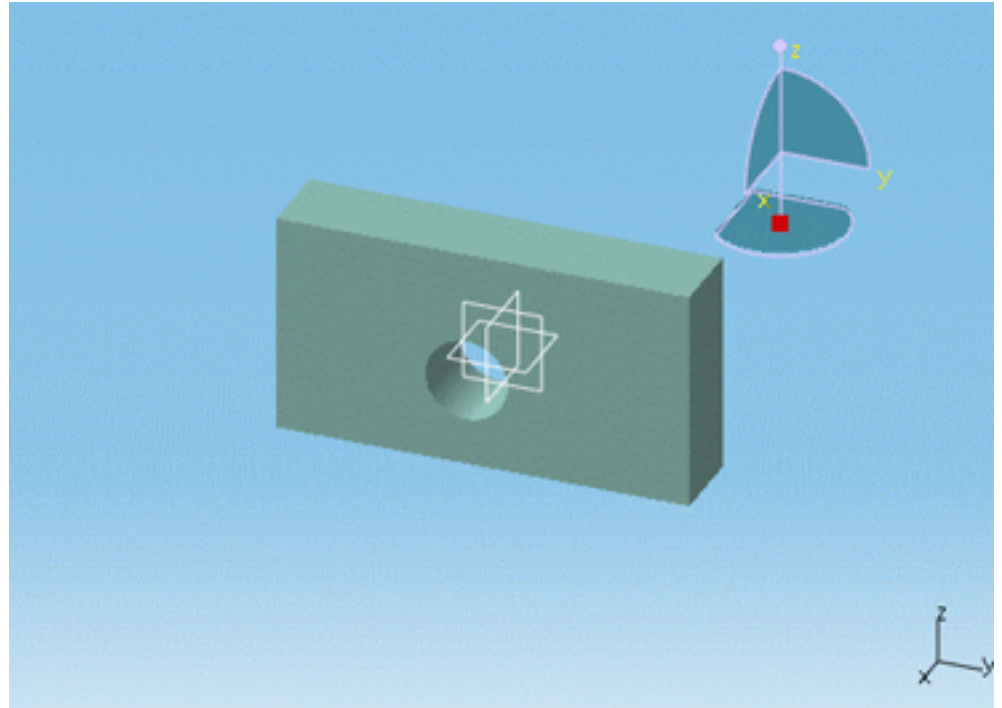
The Visualization tab lets you customize:

- **Graduated color background**

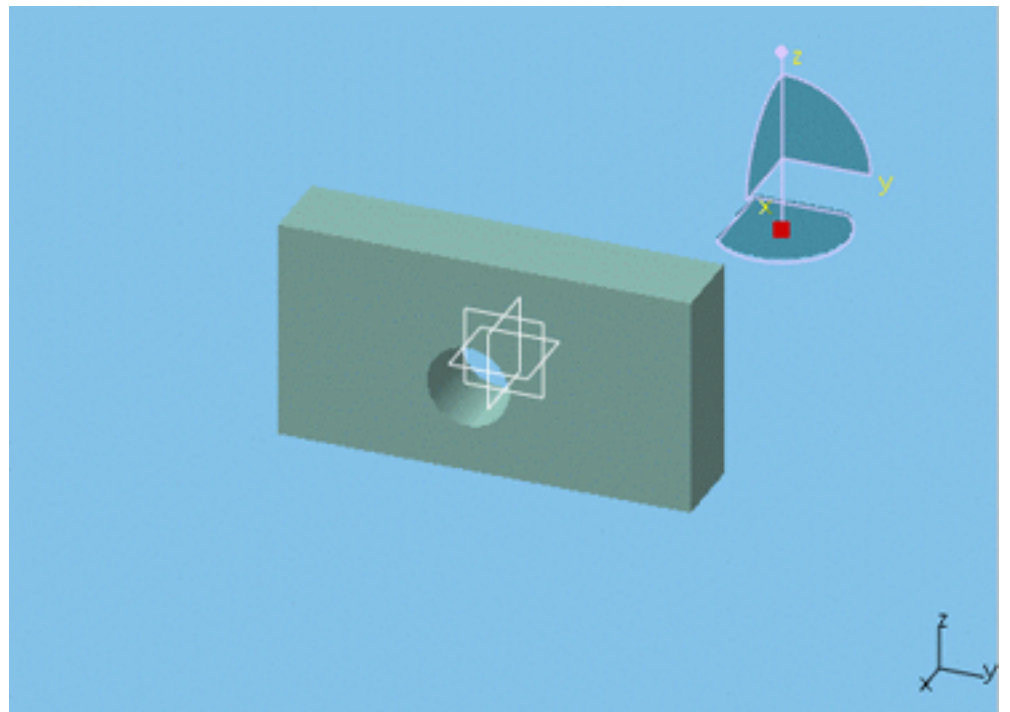
Note that this option is also available for P1 users.

This option activates a graduated color background in all open documents, and in the Preview and workbench list areas of the Visualization tab itself.

For example, this document uses a graduated color background. The color becomes gradually lighter towards the bottom of the document:



...but this document uses a normal color background:

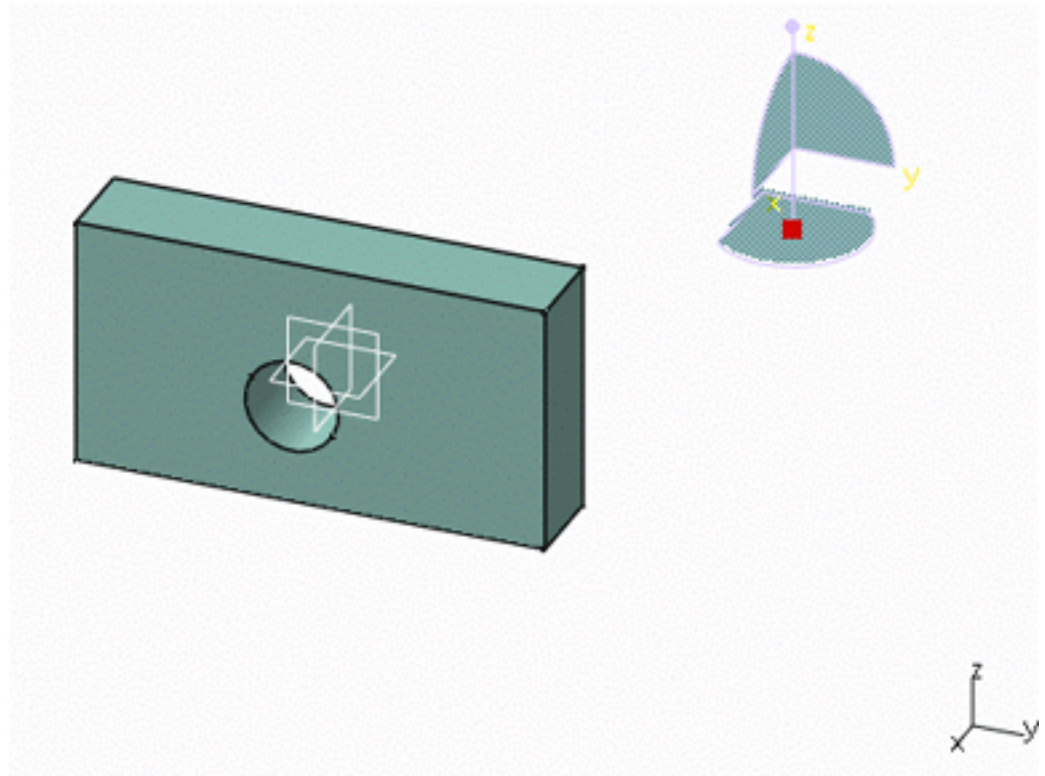




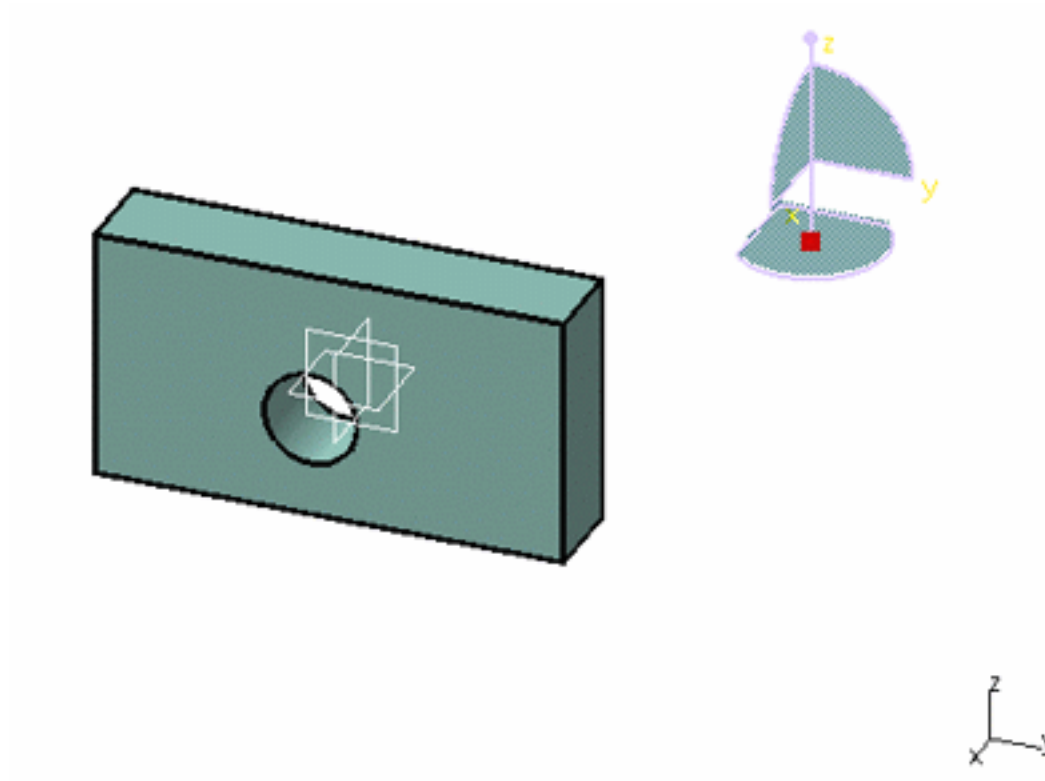
- **Background:** sets background color in all open document windows.  
Note that the default graduated background color can be reset simply by selecting it from the pulldown list.
- **Selected elements, Preselected elements, Low-intensity elements and Preselected element linetype:** set colors of selected elements, preselected elements, low-intensity elements, and the linetype used to display preselected elements.  
  
Note: low-intensity elements are elements such as parts, products, drawings, etc. which can be neither filtered, nor removed. The default color for low-intensity elements is "dark green" (as indicated in the picture of the Visualization tab above).  
  
Refer to "[Preselecting and Selecting Using the Pointer](#)" for a discussion of preselection and selection techniques.
- **Update needed:** sets the color of objects to be updated; refer to the Part Design documentation for more information about updating objects.
- **Handles:** sets manipulator handle colors. Graphic manipulators are displayed on certain objects (planar patches, holes, ...) for the purpose of moving the objects more easily. Refer to "[Moving Objects Using the 3D Compass](#)" for more information about how the compass is used on examples of where manipulators are used.
- **Selected edges:** set colors of selected edges.
- **Surface boundaries:** check this option to set the color and thickness of surface boundaries.
- **Smooth Edges:** note that the visualization of smooth edges is now defined in the [Custom View Modes](#) dialog box.
- **Display all elements using Z-buffer depth:** activates the Z-buffer so that all elements hide each other. When activated, all elements including lines and planes (usually displayed in front of other elements) are displayed at their true depth in the 3D scene. When deactivated, lines and planes are always displayed in front of other elements.

- **Anti-aliasing:** activates anti-aliasing on all edges and lines.

Anti-aliasing makes jagged lines and edges appear smoother:



... and when switched off, edges and lines look jagged:



- **Stereo enable:** enables or disables stereoscopic visualization of graphical data with a perception of realistic, three-dimensional images:
  - On: enables stereoscopic vision
  - Off: disables stereoscopic vision.

For more information, refer to "[Stereoscopic Viewing](#)".

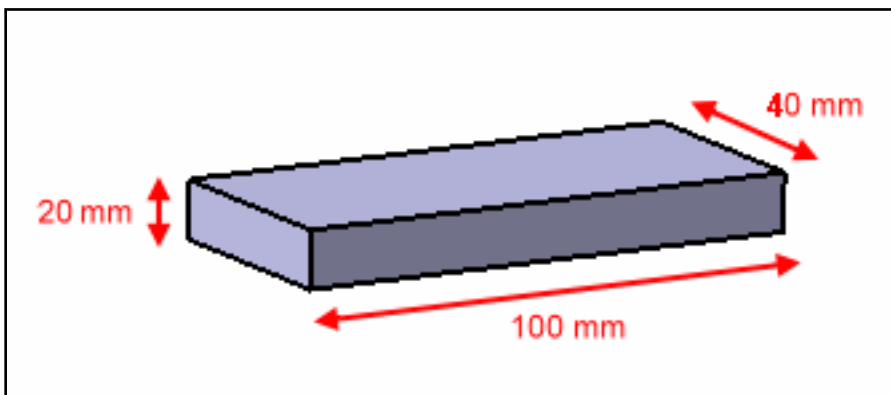
- **Display current scale in parallel mode:** when set to "On", this option activates the display of the scaling factor (at the bottom right of the geometry area) when displaying objects in a parallel view (using the **View->Render Style->Parallel** command).

The option helps you to know the real size of an object using the following formula:

$$\text{real size} = \text{size on screen} / \text{scaling factor}$$

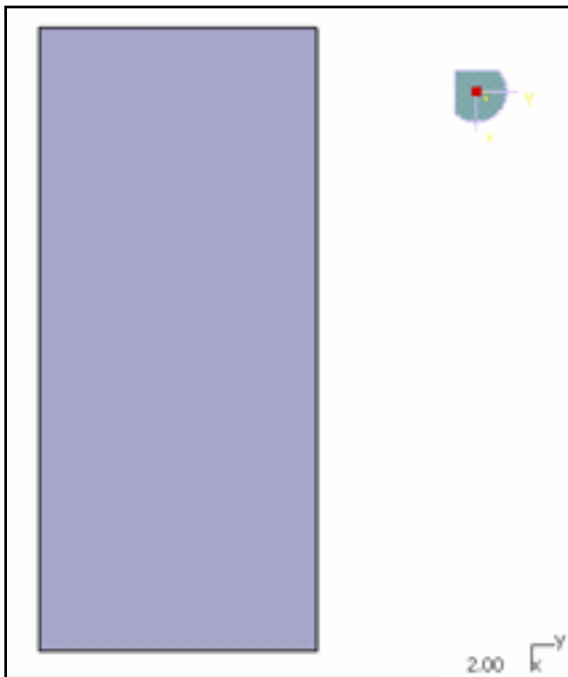
Note that the scale display is automatically refreshed when zooming.

To illustrate this, let's suppose a pad with the following dimensions:



Now, follow the steps below:

1. Display this pad in a parallel view then set the viewpoint so that you see one face of the pad.
2. Set the "Display current scale in parallel mode" option to "On".
3. Zoom until the scale factor is equal to 2.00. The result should look like this:



According to the above-mentioned formula, you know that the pad real dimensions are:

Length = 50 mm (100 mm / 2)

Width = 20 mm (40 mm / 2)

Height = 10 mm (20 mm / 2)

3. Click **OK** to confirm, or **Reset...** and **OK** again to reset default settings.



# Thickness & Font



This task explains how to set thickness properties.

**1.** Select the **Tools->Options...** command.

**2.** In the **General** category, click the **Display** sub-category then the **Thickness** tab:

The screenshot shows the 'Thickness & Font' dialog box in CATIA. The 'Thickness' section contains a table with the following data:

Index	Size in pixel	Size in mm
1	1	0.13
2	2	0.35
3	3	0.70
4	4	1.00
5	5	1.40
6	6	2.00
7	7	2.30
8	8	2.60

The 'Font Option' section has a checked checkbox for 'Use system TrueType fonts in CATIA.'

3. In the Index column of the **Thickness** area, select a thickness type from the 55 values contained in the list.
4. If necessary, modify the Size in pixels or in millimeters by selecting the value to be modified then clicking it once.

The Size in pixels reflects the result displayed on screen and the Size in mm corresponds to the printed version.



You cannot assign a size greater than 16 pixels.

5. The "Use system TrueType fonts in CATIA" option enable you to use TrueType fonts provided by Windows and thus, add them to the font chooser.

This default list of fonts can be customized. To do so, refer to [Customizing Fonts for Displaying Geometry Area Texts](#) in this guide.

6. Click **OK** to confirm.



# Customizing Settings for Layer Filters

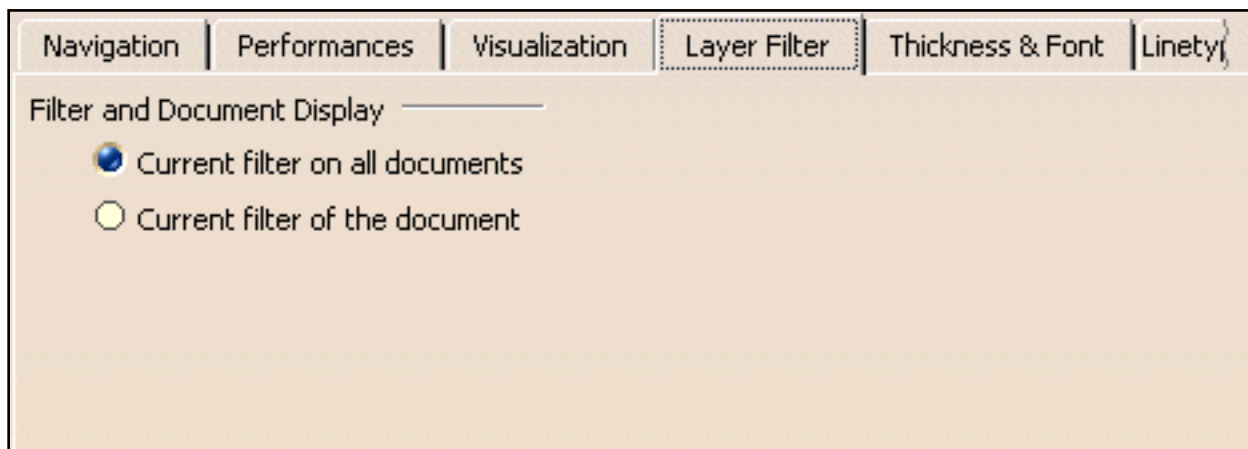


This task explains how to customize settings for layer filters.



1. Select the **Tools->Options...** command.

2. In the **General** category, select the **Display** sub-category then the Layer Filter tab:




3. Select the type of filter to be applied in the Filter and Document Display area which lets you choose between two options:


- **Current filter on all documents** means that the current filter will be applied to all documents opened in your Version 5 session
- **Current filter of the document** means that each document will use its own current filter (if there is one).  
Note that there is a price to pay in performance when using this filter mode.

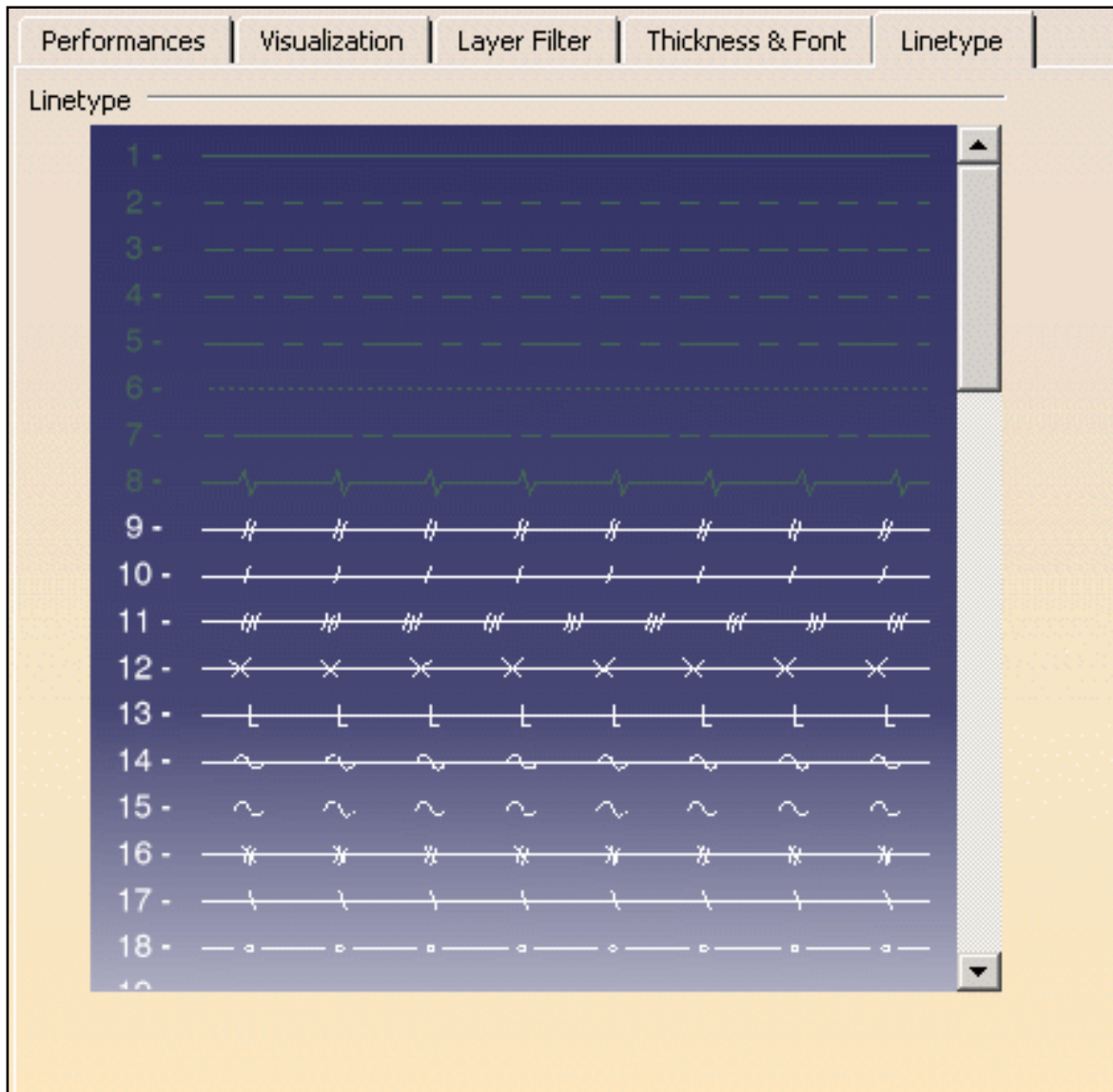
4. Click **OK** to validate and apply the selected filter.



# Linetype

 This task explains how to set linetype properties. This functionality is relevant for Drafting products only.

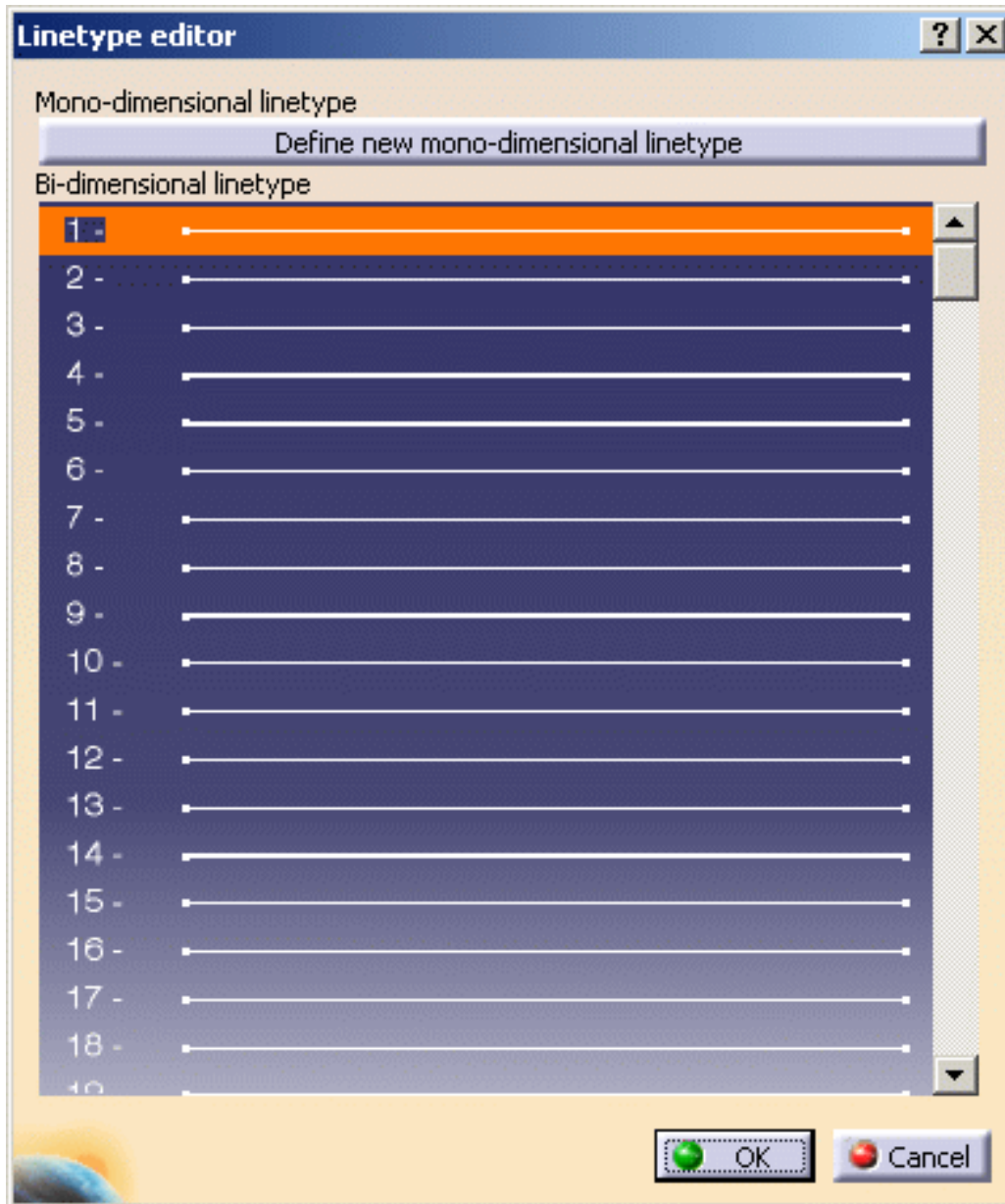
-  **1.** Select the **Tools->Options...** command.
- 2.** In the **General** category, click the **Display** sub-category then the **Linetype** tab:



The available linetypes are displayed in a list, the eight first types being grayed as they correspond to the default linetypes (displayed in the Graphic Properties toolbar) and cannot be modified.

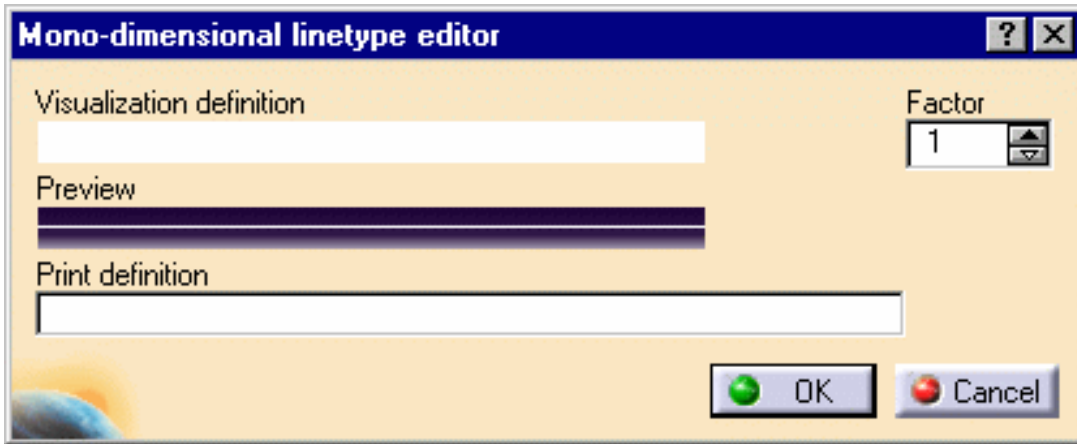


3. Double-click the linetype to be modified. The Linetype Editor opens:



You can either define a mono-dimensional linetype or a bi-dimensional linetype.

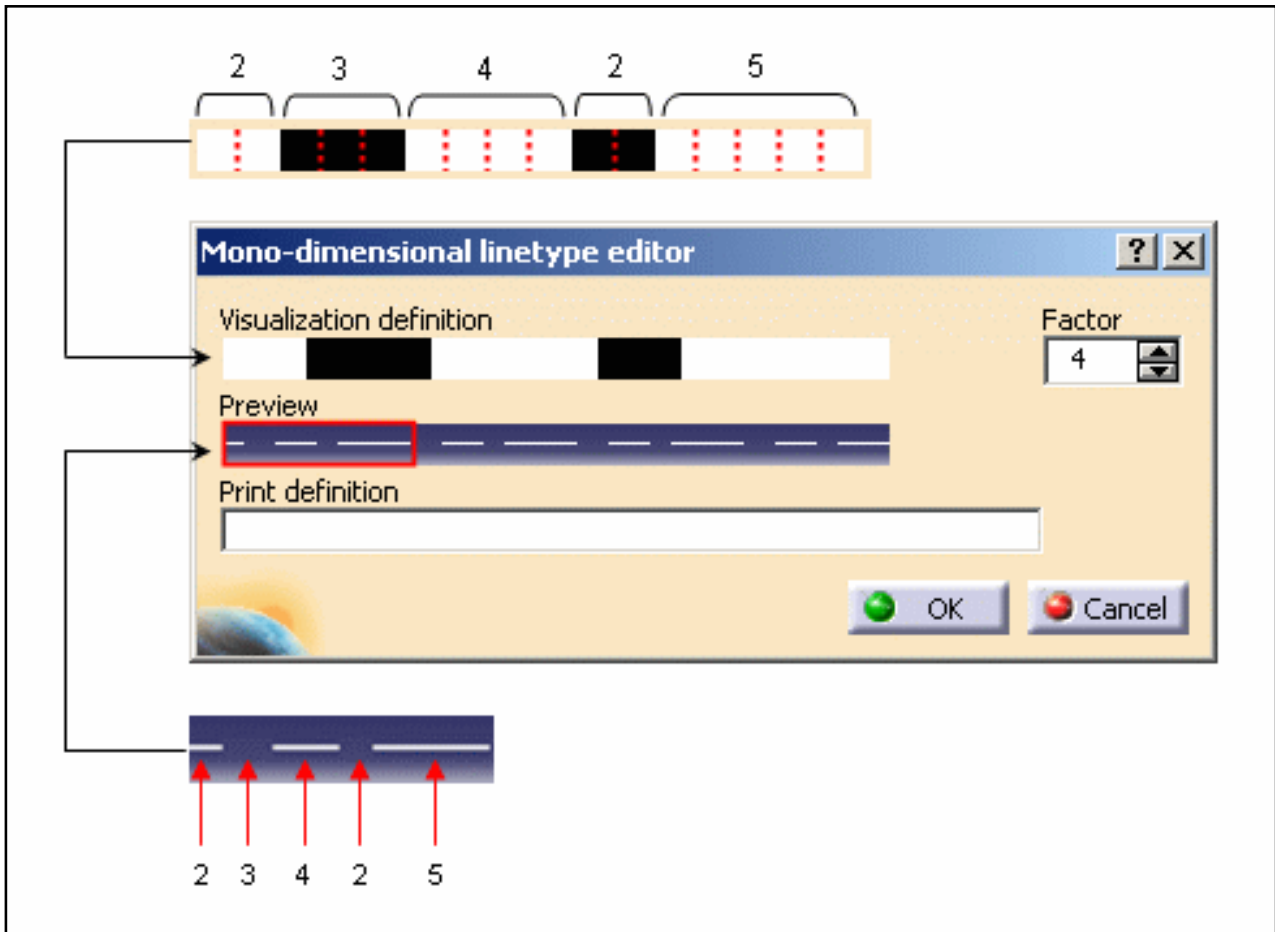
4. To define a mono-dimensional linetype, click the **Define a new mono-dimensional linetype** button. The Mono-dimensional linetype editor dialog box opens:



This dialog box lets you define the linetype for visualization and print purposes.

5. In the Visualization definition field, click the desired squares to define the linetype pattern: a white square corresponds to a drawing, a black square to a blank. You can select up to 16 squares.

The Preview will be updated accordingly as shown below:



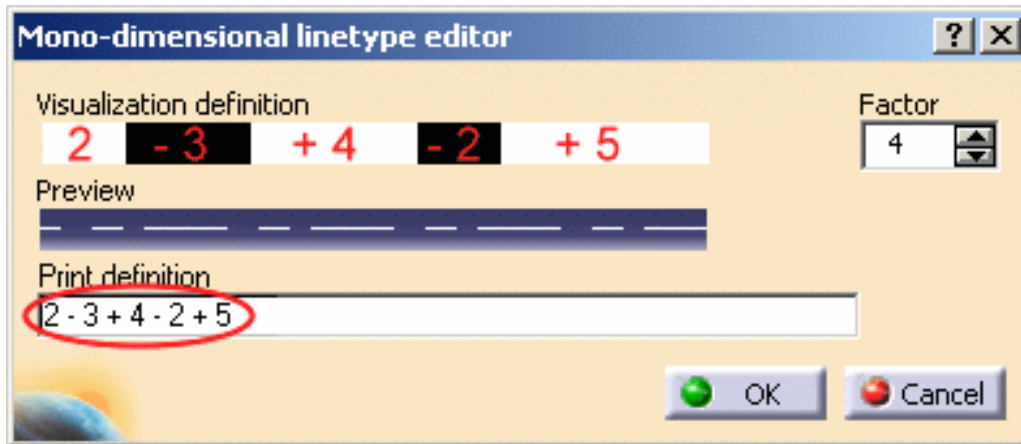
6. Use the Factor field to stretch the pattern by increasing the value as shown below:



7. In the Print definition field, enter the length in millimeters for each pattern segment (do not forget to leave a blank space after each character you type).

A positive value indicates a drawing and a negative value indicates a move without drawing.

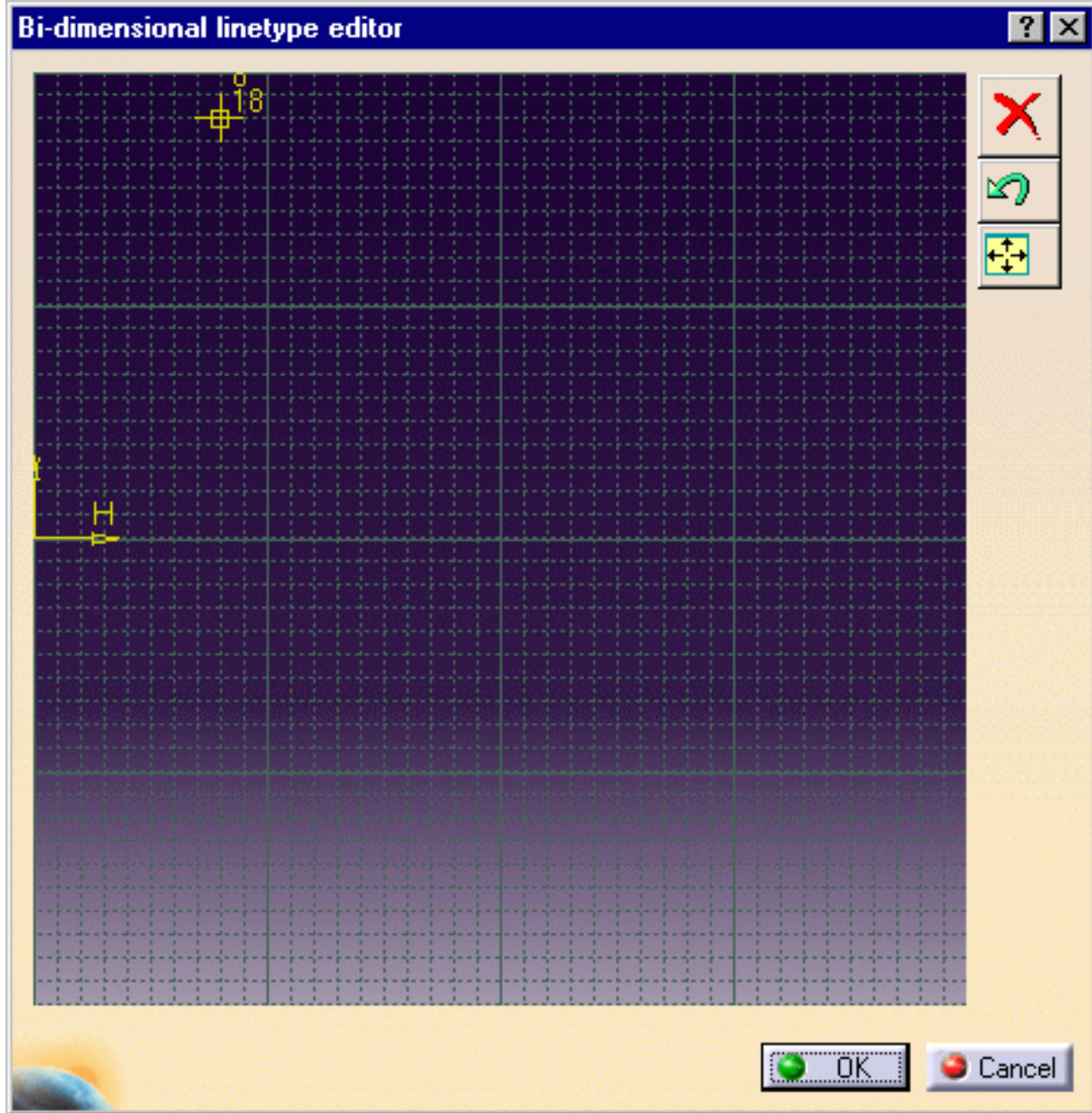
The capture below illustrates the print definition of the pattern defined in step 5:



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

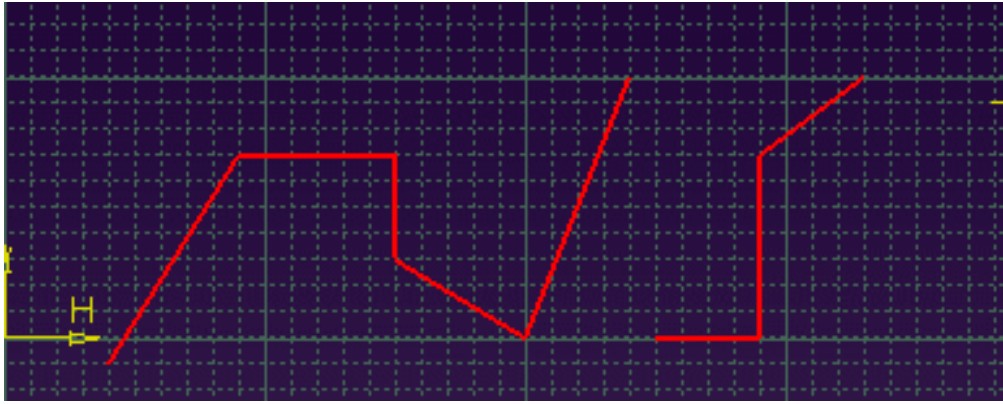
The new linetype replaces the selected line in the Linetype editor list. If you do not wish to go on defining linetypes, just click **OK** once again to close this dialog box and the new linetype will be displayed instead of the selected line at the end of the Linetype list.

9. To define a bi-dimensional linetype, double-click a line from the Bi-dimensional linetype list in the Linetype editor. The Bi-dimensional linetype editor window opens:






10. Move the cursor then click to create a segment and repeat these steps as necessary to define the desired pattern.

You can create segment breaks by double-clicking a point then moving the cursor to draw a new segment:



11. Use the following icons to:

-  delete the whole pattern
-  delete the last segment
-  reframe the bi-dimensional line editor.

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

The linetype is displayed instead of the selected line at the end of the Linetype editor.

13. Click **OK** once again to close the Linetype editor and display the new linetype instead of the selected line at the end of the Linetype list.



# Compatibility

V4 Data Reading

Saving As V4 Data

V4/V5 DRAW

V4/V5 SPACE

V4/V5 SPEC

Migration Batch

DELMIA/Deneb

DXF/DWG

Electrical Migration

ENOVIA LCA

External Formats

IGES 3D


IGES 2D

SmartBOM

STEP

VRML

# V4 Data Reading

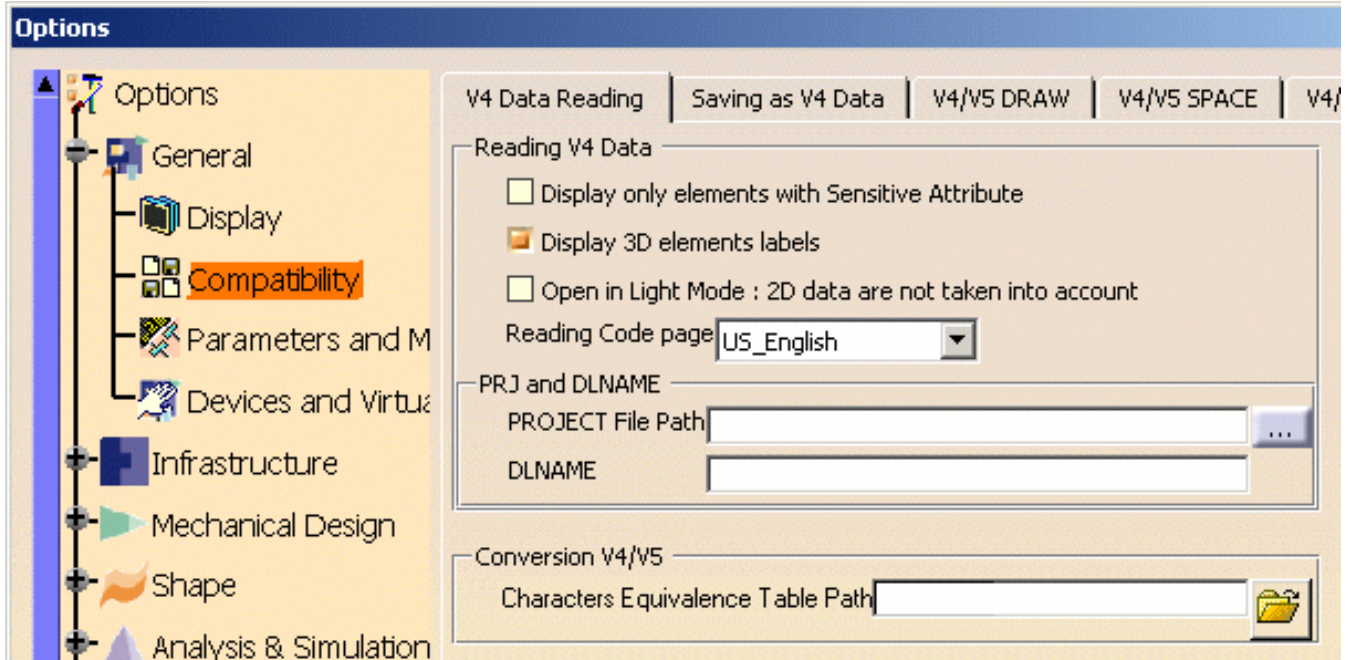
 This task shows you how to customize Compatibility settings.



1. Select the **Tools -> Options...** command.

The Options dialog box appears with the category tree in the left-hand column.

2. In the **General** category of the Options Tree, select **Compatibility** and click the **V4 Data Reading** tab. The following dialog box appears:



In CATIA Version 4, certain information was specified by means of the parameter settings in the declaration files. These declaration parameters are no longer supported in Version 5 and there is no way to transfer them automatically to Version 5. Such information must be provided by means of the Options dialog box shown above, **before** attempting to read Version 4 data.

The Options dialog box lets you:

- **display faces and surfaces** visible in shading mode in CATIA Version 4.
- open CATIA Version 4 models referencing an external **PROJECT file**.
- specify the **Initial Model File Path**: to translate V5 CATPart documents into V4 Models using V4 standards defined in the V4 Initial Model.
- specify the **writing code page** identifying unlabeled data.
- declare the **code page** to be stored in the CATIA data to be written when saving Version 5 CATPart documents as CATIA Version 4 models.
- declare the **Model Dimension** when saving V5 CATPart documents as V4 Models.
- declare the **Model Unit** when saving V5 CATPart documents as V4 Models.
- declare the location of a **character equivalence table**.
- choose options for **Migration Batch**: Conversion Mode and Format.

## Reading V4 Data

- **Display only elements with Sensitive Attribute:**

In CATIA V4, the "**Display only elements with Sensitive Attribute**" mode enables you to decide whether to display the faces and surfaces sensitive to the shading mode in CATIA Version 4.

To display only the faces and surfaces that were sensitive to the shading mode in Version 4, you must therefore activate this option. In CATIA Version 5, by default, this box is deactivated.

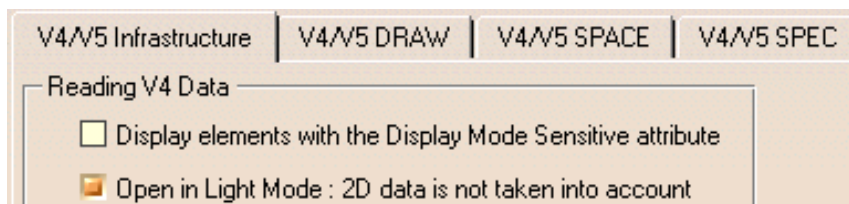
- **Display 3D elements labels**

This option is checked by default. It means that when you download a model in CATIA V5, the visualization of the V4 elements' Id or 3D text is recalculated. For instance, the Id \*FAC can appear near the 3D Face in the 3D space.

If you do not check this option, the 3D text associated to the V4 elements will not be built. They will not appear in the 3D space in CATIA V5.

- **Open in Light Mode: Open a V4 Model without DRAW Data**

If you activate this setting, you can disable the reading of DRAW data.

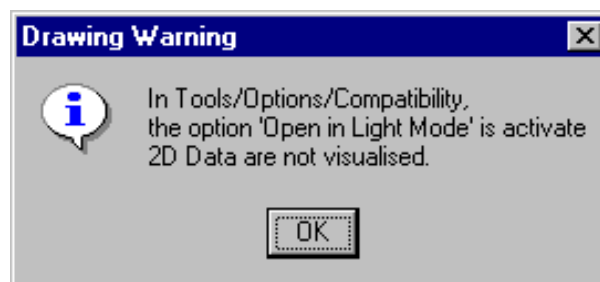


The integration of V4 models in V5 applications generally uses only the 3D Master representation of the model. On the other hand, the management of V4 DRAW data brings several critical issues:

- Stability issue: PRJ has to be referenced correctly, else CATIA may crash
- Performance issue: data conversion (CATAIX/CATUNIT) of DRAW data is very expensive.
- Memory consumption.

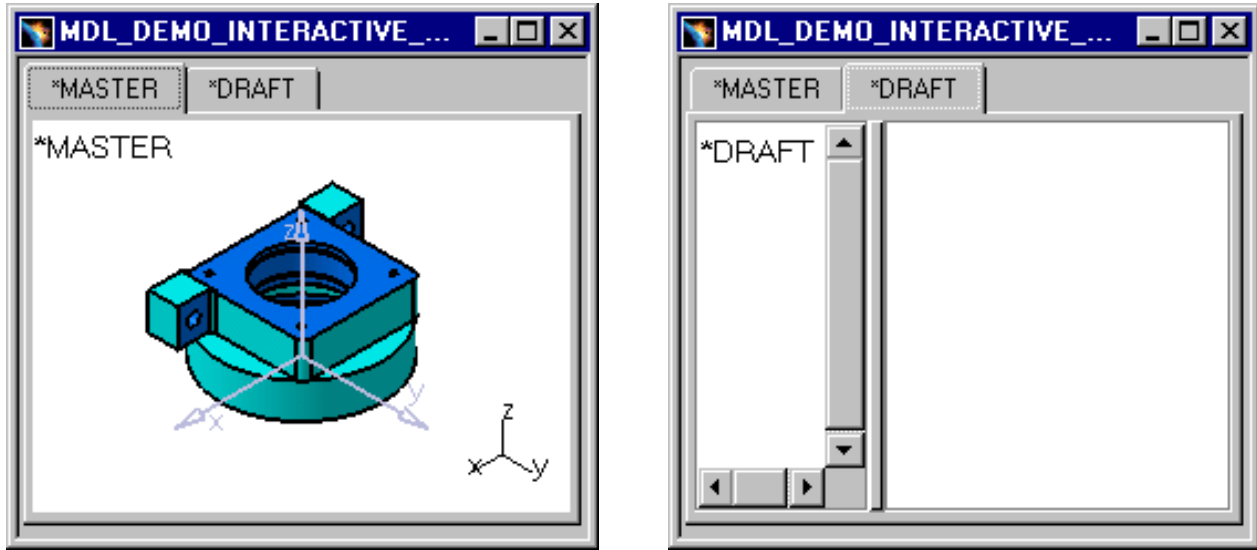
Skipping DRAW data when opening the model will not bring any restriction when working in Assembly or DMU workbenches for instance. On the contrary, it will be a major stability / performance / memory cost upgrade.

The DRAW data is suppressed before any operation (internal or external) can be performed on the model. However the DRAFT tab page is still available, but swapping to this tab page will generate a warning message indicating that the model has been opened in light mode:



You only get SPACE elements in the \*MASTER tab; the \*DRAFT tab is empty:





This setting can also be effective with CATProducts and Sessions containing V4 models.

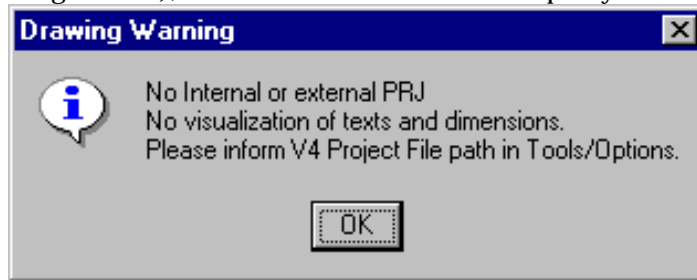
Note:

- the warning is displayed only once per tab page.
- some functionalities open directly the models in 2D mode (for example: ENOVIA DMU / Send to V5 2D Viewer). The warning should be sent in that case too.



If a user wants to reread a model and browse V4 geometry, it is necessary to uncheck the setting and to reopen the model. If this setting is activated, the V4-V5 Migration Batch will not be able to migrate V4 Drawing data. Therefore, if you want to convert V4 drawing elements in CATIA V5, it is necessary to uncheck this option. For more information about the Migration Batch, please refer to the *CATIA - V4Integration User's Guide*.

If the model has an external PRJ File which is not specified in the Tools-Options, a second message appears (even if the PRJ File Path is not necessary when you choose the Light Mode). However, this warning is useful when the user decides to go back to the normal mode (not the Light Mode), at the same time he can also specify the PRJ File Path in the Tools-Options.



This mode is also available in PDM Application when sending a model from PDM Application into CATIA V5.

• **Specifying the code page identifying unlabeled data**

In V4, 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 Version 5 and there is no way to transfer them automatically to Version 5. Such information must be provided by means of the dialog boxes described below, **before** attempting to read Version 4 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.

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.



- **PROJECT File Path: Opening CATIA Version 4 models referencing an external PROJECT File**

In V4, 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.

Enter, in the **PROJECT File Path** field, the location 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.



Bear in mind the following:

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

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

- attribute and class tables
- annotations/dimensions.

**On UNIX and NT**, you can reference an external PROJECT File using the http protocol, the address to be specified should be like this:

**http://serverName/path/PRJDirName**



Make sure beforehand that an http server has been installed on the machine where the V4 data resides. For more information, you can refer to About Data Sharing between Windows and UNIX, in *CATIA - Infrastructure User's Guide*.

**On UNIX only**, you can also reference PRJ files with a UNIX path:

**/u/path/PRJDirName**

**On NT only**, if you have access to PRJ files installed on the NT disk, you can reference it with an NT path:

**E:\path\PRJDirName**

In that case, you need to bring these Project files referenced by the model into an NT directory and rename these files according to the conversion table. In the PROJECT File path, you refer to the directory containing the Project Files.



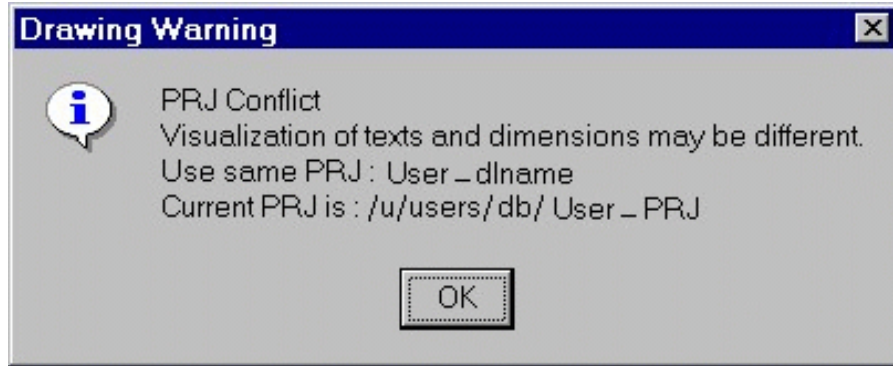
For more information about character conversion, refer to the Conversion Table in the Model Naming chapter in *CATIA - V4Integration User's Guide*.

Thus, in the PROJECT File path you refer to the directory containing the Project Files. For more information about the methodology of copying your Project File on NT, refer to Having Access to PRJ File on NT in *CATIA - V4Integration User's Guide*.

- **DLNAME**

In the V4/V5 Infrastructure settings, the only way to locate a PROJECT File physically is to fill in the PROJECT File Path. The DLNAME setting does not allow this functionality. This setting only allows to compare the Model DLNAME (when the Model references a DLNAME instead of a PRJ File Path) with the specified one within the setting.

When the user has only specified the PRJ File Path (User\_PRJ) and not the DLNAME, whereas the model references only a DLNAME (User\_dlname), the following Warning message appears:



Whatever the value of the DLNAME field is, the PROJECT File Path is taken into account (reference to locate the PROJECT File). This is the reason why the DLNAME option is automatically deleted when the PROJECT File Path is not specified and when you click OK to close the panel.

## Conversion V4/V5

- **Characters Equivalence Table Path:**

The Characters Equivalence Table is taken into account when using the following functionalities:

1. File / Open a Session or a .asm document (documents are read with V5 compliant names).

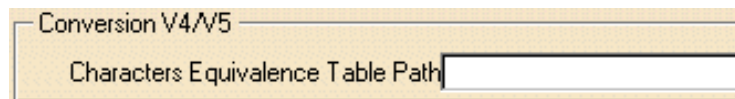
For more information about this functionality, read Opening a CATIA V4 Session in CATIA V5, in *CATIA - V4Integration User's Guide*.



2. V4 to V5 Migration Batch (documents are migrated with V5 compliant names).

The Characters Equivalence Table allows to convert characters contained in CATIA V4 documents. It is mainly used to generate V5 compliant names from V4 names by replacing special characters (" , \* , / , etc...) by standard characters.

A default table exists (table path empty) but the user can choose a different table by specifying its NT or UNIX path in the option box:



The table file is a .txt document. By default, the [conversion table](#) will be applied to special characters:

ASCII Code of the Original Character	Character to be Replaced	New Character String
0x22	"	_Inch
0x2a	*	x
0xB1	±	—
0x2f	/	—
0x3a	:	—
0x3c	<	—
0x3e	>	—
0x3f	?	—
0x5c	\	—
0x7c		—

For instance, the double-quotes (0x22 in ASCII code) will be replaced by "\_Inch".

If you want to display it on Windows, the address to be specified should look something like this:  
**home/path/table.txt.**

CATIA V4 documents, containing the special characters space and slash, are saved with a name which does not have the space or slash character. The space is automatically converted in (0xB1) and the slash is automatically converted in (0XB4).

In CATIA V4, if a pointed document is renamed or moved, the multi-model link is broken.

If a document name contains a space, it is a result of a manual renaming procedure, so the pointed document is considered to be not the same as the one corresponding to the link name.

When V4 documents dependencies are scanned for renaming, V4 links (if exist) are analyzed, and the CATV4ToV5CompatibilityName try to find the pointed documents. If their name must be changed, they are added to the list of documents to rename. But, if the name of the pointed document does not match the saved pointed document (with ± and ') the dependency is not taken into account. And the pointed documents are considered as not found.

In the multi-model links, the conversion table is applied on:

- the document name (in the link) for the slash
- the saved name for the space.

If the conversion table is not defined, the default character for space and slash for the CATV4ToV5CompatibilityName is underscore.

If the conversion table is defined, it is used to convert the V4 documents.

Here is an example with a pointed document or a model name:

- Link name with space and slash
  - Sample ZZZ/ZZZ.model
- Pointed document on disk:
  - Sample ±ZZZ 'ZZZ.model

- With no conversion table:
  - new name: Sample\_ZZZ\_ZZZ.model (± and ' become \_)
- With Conversion table: (if ± converted into A and / converted into B)
  - new name: SampleAZZZBZZZ.model



For more information about this functionality, read Migration in Batch Mode, in *CATIA - V4 Integration User's Guide*.

Click **OK** to confirm.



# Saving as V4 Data



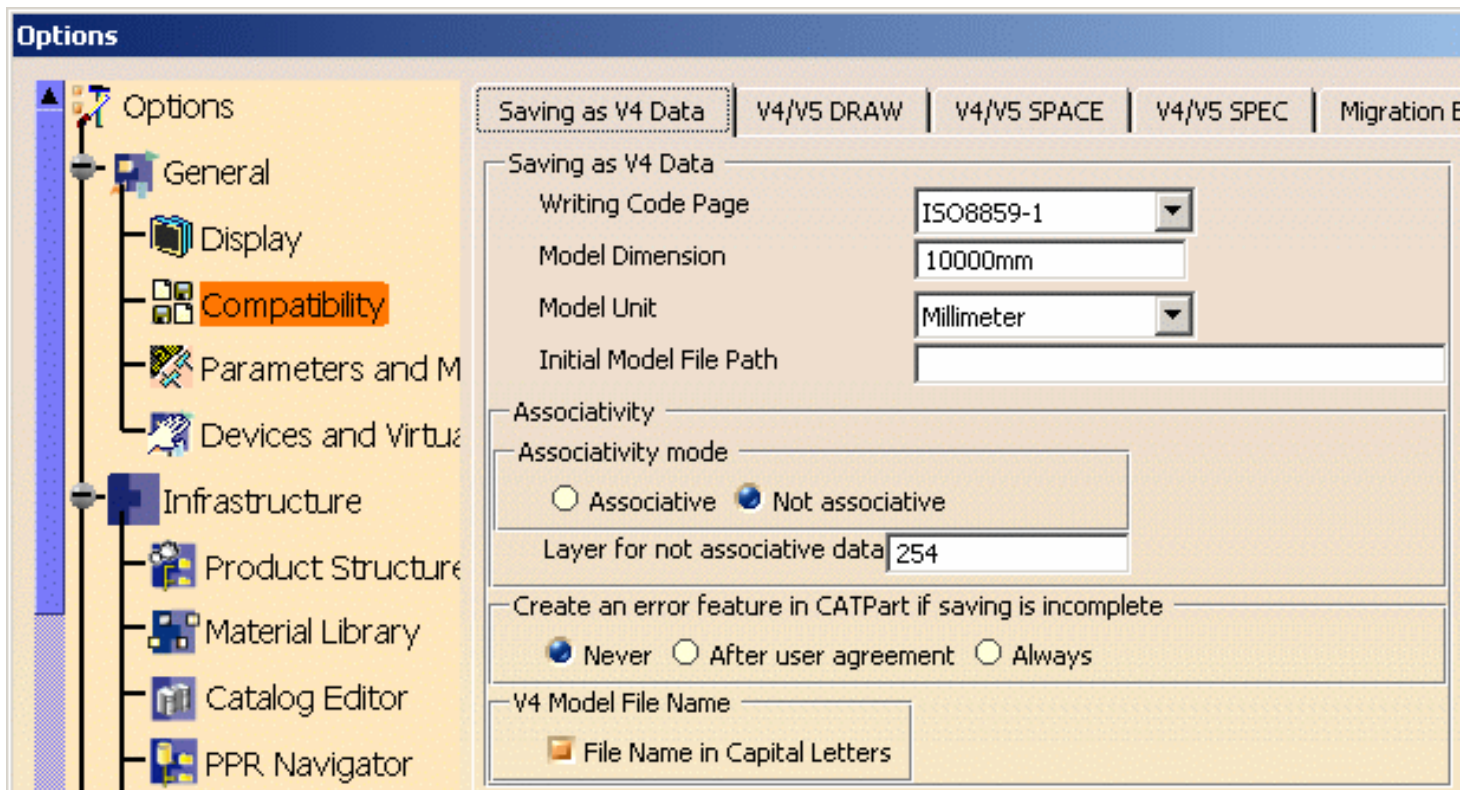
This task shows you how to customize Compatibility settings, and more particularly for the V5 to V4 Migration (Save As Model).



1. Select the **Tools->Options...** command.

The Options dialog box appears with the category tree in the left-hand column.

2. In the **General** category of the Options Tree, select **Compatibility** and click the **Saving as 4 Data** tab. The following dialog box appears:



## Saving as V4 Data

- **Declaring the writing code page when saving Version 5 CATPart documents as CATIA Version 4 models**

For a full scenario illustrating this functionality see "Saving Version 5 CATPart Documents As CATIA Version 4 Models" in the *CATIA - V4 Integration User's Guide*.

In Version 4, the **declaration parameter** `catsite.WRITING_CODE_PAGE` declares the code page to be stored in the Version 5 data to be written. The writing code page ISO-8859-1 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, 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.

ASCII-DS
ASCII-DS-HGL
ASCII-DS-KJI
EBCDIC-DS
EBCDIC-DS-HGL
EBCDIC-DS-KJI
EBCDIC-HGL
EBCDIC-KJI
IBM-850
<b>IBM-932</b>
IBM-932-SBCS
ISO8859-1
ISO8859-2
ISO8859-5
EUC-CN
EUC-JP
EUC-KR
EUC-TW

- **Declaring the Model Dimension when saving Version 5 CATPart documents as CATIA Version 4 models**

It is possible to customize Model Dimension for V4 models generated in V5.

V4 tolerances will be computed according to V4 recommended values from this Model Dimension. In V4, model tolerances can be modified through **Standards** -> **Model Function**. The Model Dimension Parameter, which can be modified in CATIA V4, has an impact upon the precision of geometrical calculation. The value by default is 10000mm and it corresponds to the V4 value.

Before saving a V5 Part as a Model, you can change the Model Dimension parameter to fit your V4 standards. The whole set of tolerances will be computed from this Model Dimension according to the V4 recommended values.



Be aware that modifying this value without consideration can affect dangerously the V4 geometry. The modification of the Model Dimension parameters should only be used to respect the V4 standards.

The management of the resolution is different in V5 and in V4 :

In V5 the main concept for tolerances is the Resolution which defines the minimum length of a valid object. It is fixed to  $10^{-3}$ mm. The management of confusions ("Do two objects have the same geometry?") is a direct consequence of the resolution: if the distance between to geometric points is less than the resolution, the two points are considered to be geometrically at the same location.

In V4, tolerances are driven by the Model dimension. The most appropriate value to fit V5 resolution is 10000mm. Note that this is also the V4 default value.

- **Declaring the Model Unit when saving Version 5 CATPart documents as CATIA Version 4 models:**

You can apply a scale factor between the imported file and what you want to get from the original Model. You can choose a V4 Unit in this list:



### • Writing the Initial Model File Path

With this option, you have the possibility to translate V5 CATPart documents into V4 Models using V4 standards defined in the V4 Initial Model.

You may use an Initial Model; this means that you define a reference model which includes default values. To customize the environment (initialization values to modal parameters and management parameters), you need to create an Initial Model. Such a Model is often used to ensure that everyone in a company works with the same predefined standard values.

The **Save As Model** operation is able to take into account a V5 setting: a V4 Initial Model file path. Before saving a V5 CATPart as a V4 model, you can specify an initial Model file path in this setting **Tools->Options...->General->Compatibility->V4/V5 Infrastructure**.

Initial Model File Path

The "Initial Model file path" setting can replace the following settings:

- The "**WRITING\_CODE\_PAGE**" setting
- The "**Model Dimension**" setting
- The "**Model Unit**" setting.

And the "Initial Model file path" setting defines other V4 standards:

- The layer filters
- The colors
- Standards for SPACE elements: GENERAL and HLR standards can be taken into account. SPEC ELEMENTS standards cannot be defined.
- Standards for DRAW elements

This information will be extracted from the Initial Model and added to the V5-generated model.





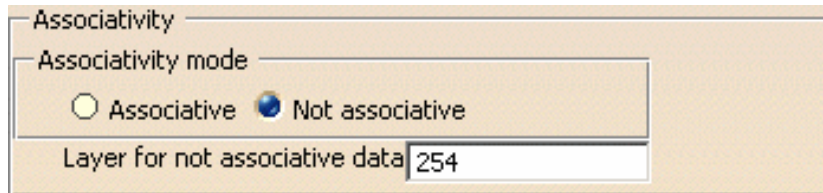
Initial Models referenced in CATIA V5 should not contain:

- any geometry, SPACE or DRAW
- any applicative data
- Internalized Project File

Therefore, Initial Models used in CATIA V4 will not always be reusable directly in V5, and it will be generally easier to recreate a dedicated Initial Model for CATIA V5.

If the Initial Model contains elements that are not supported, the Save As Model operation will be aborted.

- **Associativity Mode**



You can create a model in associative mode during the Batch migration or interactively. Applying a Save As Model operation on a Volume or a Surface means that the model keeps in memory the path of the Part and you can resynchronize the model after modifying the Part. You can choose either the Associative Mode or the Non-Associative Mode.

Previously, associativity could be maintained only on the PartBody (on volumic components). A message was displayed whenever the migration could not take place for non-volumic elements.

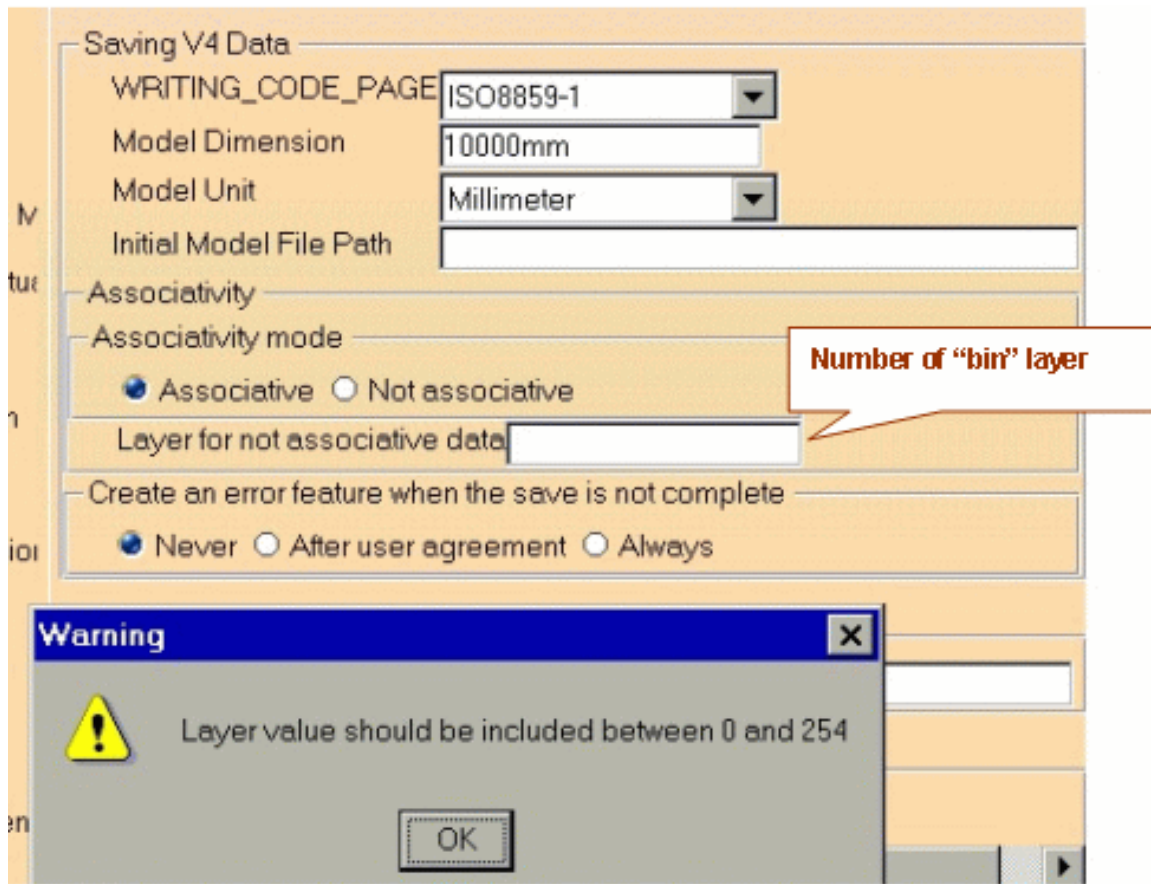
Now, you can launch the associative mode whatever the CATIA components' nature is (volumic or not). Therefore, a standard Save as Model can be applied on non-volumic or non-surfacic elements.

### **Synchronization of the model:**

At each stage of synchronization, the non-associative elements created during the previous migration are added in the "bin" layer. And the non-associative elements of the current CATPart's geometry are entirely re-created. In this way, the user, if he wants, can delete all the old non-associative geometry by selecting the "bin" layer in CATIA V4.

### **Number of the layer for non-associative data:**

The number of layer can be specified by the user thanks to the **Tools->Option->Compatibility** Settings. Note that in V4 model, a number of layer can have a value between 0 and 254. If this setting is not informed, a default layer number is used: layer number 254. If you enter 255 for instance, a warning is displayed. The layer is visible in the model. The user can put it in the No Show space or delete it if he wants.



**Identifiers of the non-associative elements:**

During the stage of synchronization, non-associative elements issued from the previous migration are moved into the "bin" layer and their identifiers are modified in order to know which migration number they come from. Each identifier is modified in that way: RxxxIDENT, where xxx is the migration number.

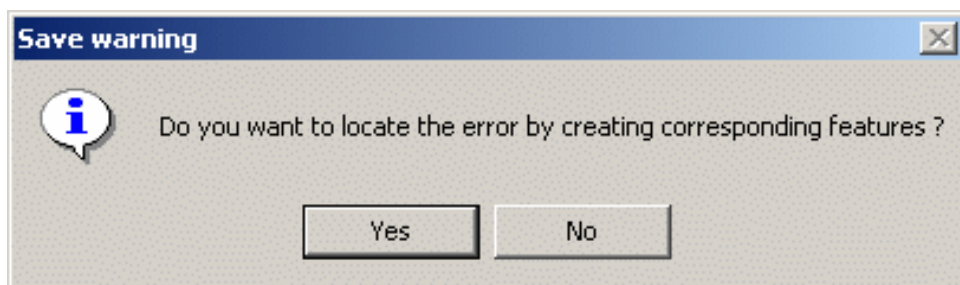
For instance, the identifier LINE.1 becomes R001LINE.1 at the first synchronization of the model. If we make two synchronizations, the model contains LINE.1 and the following identifiers R001LINE.1 and R002LINE.1.

- Create an error feature in CATPart if saving is incomplete

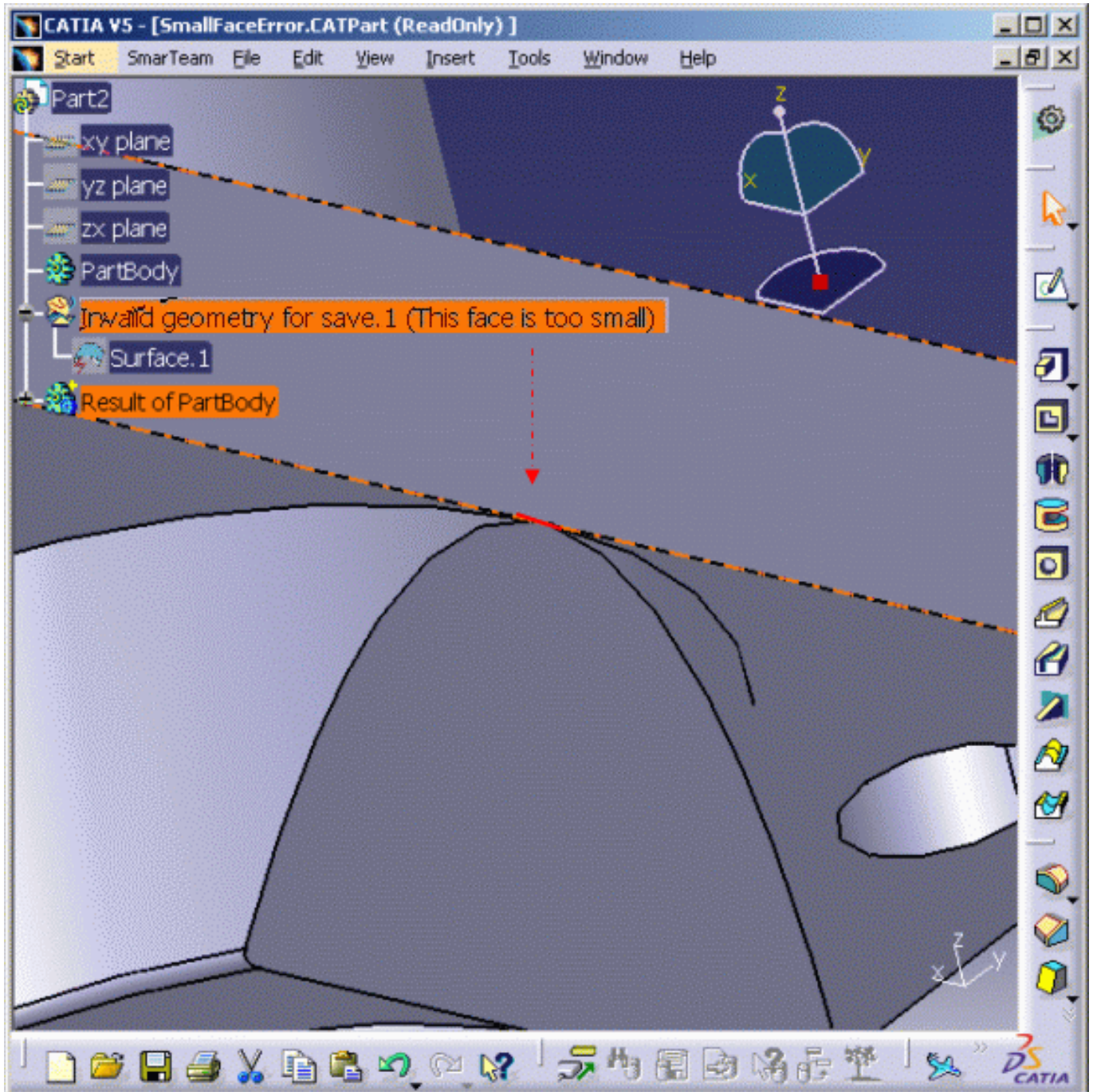
When some V5 elements cannot be migrated into CATIA V4, a warning message can be displayed and an error feature appears in the CATPart's Specification Tree.

**Never:** if the user does not want to see this message and the error feature, he can click this option.

**After user agreement:** if he chooses this option, a panel appears, asking if he wants to see the corresponding error feature in the Specification Tree.

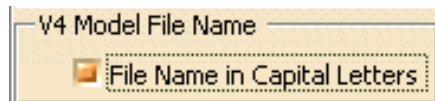


If the user click Yes, an error feature indicates the V5 component that could not be migrated: "Invalid BRep Geometry.1 (This face is too small)". In the Geometry, the face is highlighted in red and the user can modify it.



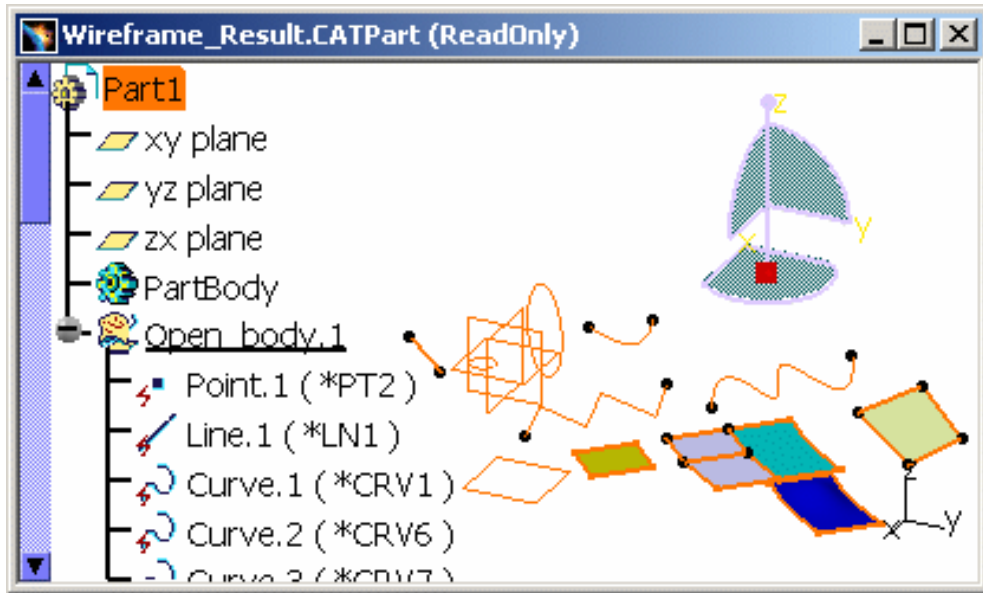
**Always:** no panel is displayed and the error feature automatically appears in the CATPart's tree. Click **OK** to confirm.

- **V4 Model File Name**

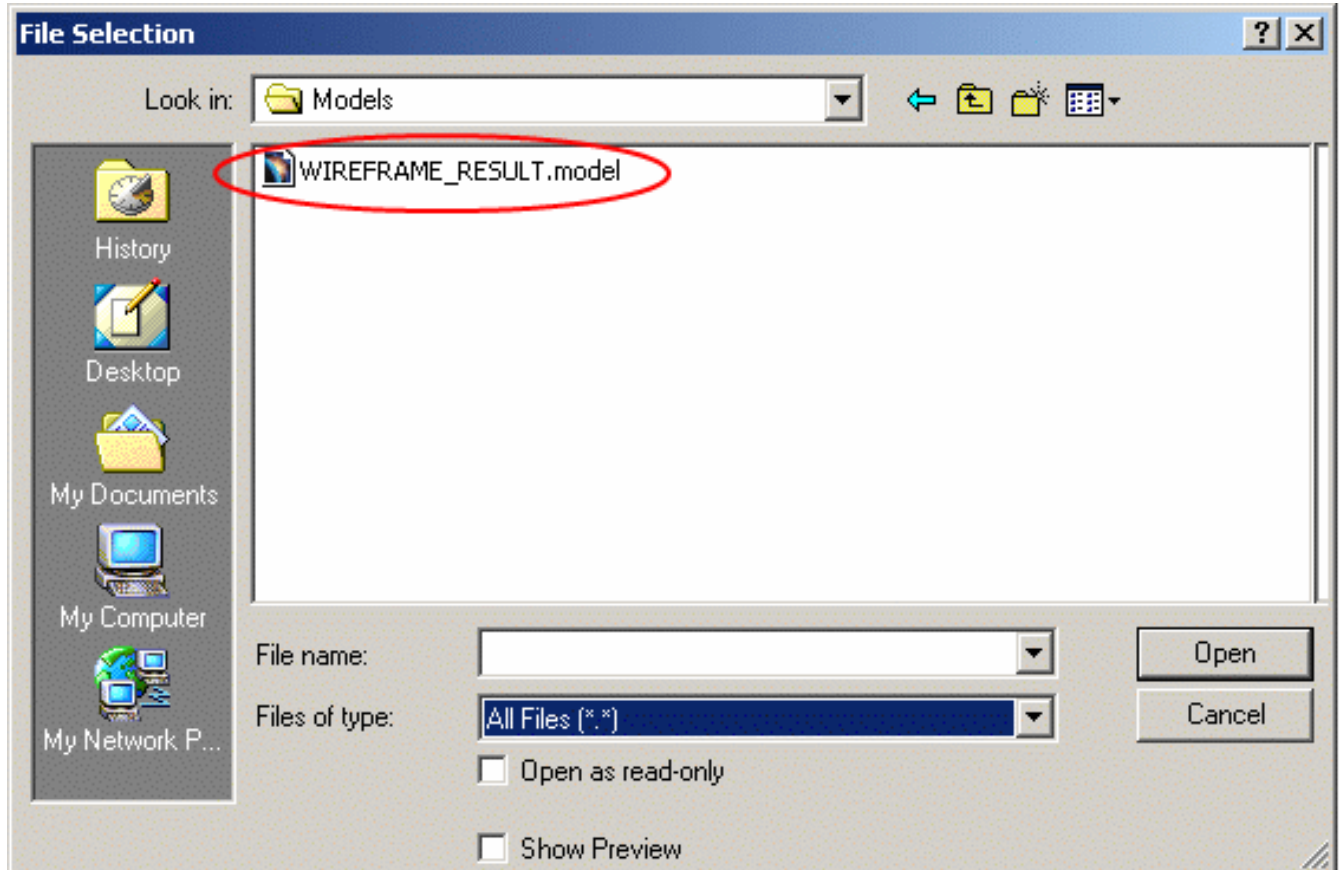


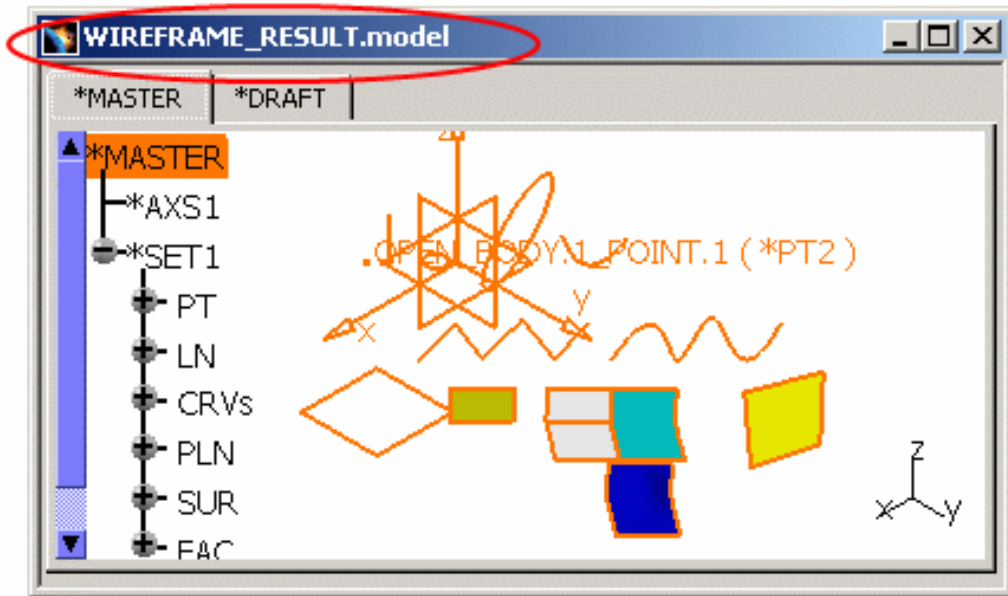
Before migrating a CATPart into a .model, you can specify in the Saving as V4 Data settings that the resulting .model must have Capital Letters.

1. Check the option: File Name in Capital Letters.
2. Open a CATPart in CATIA V5.




3. Save the CATPart as a .model.
4. Open the .model in CATIA V5. In the File Selection panel and in then in CATIA V5, the model's name is in Capital Letters:



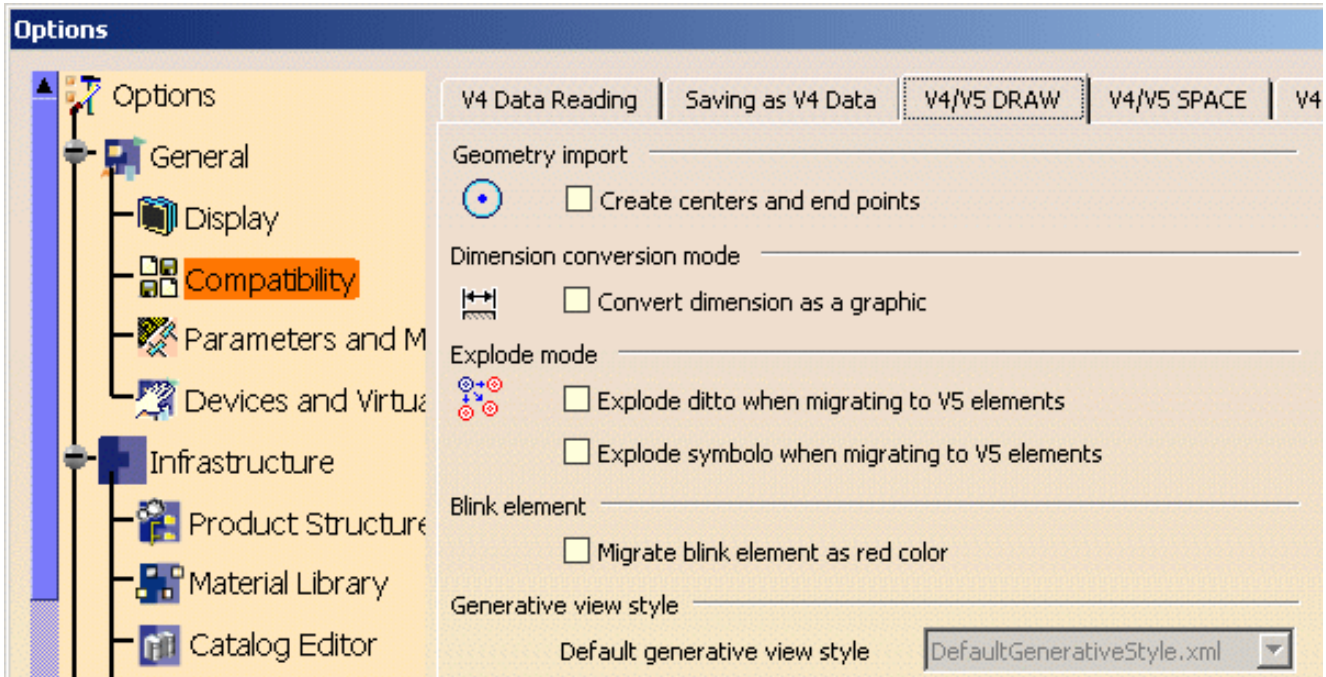


# V4 / V5 DRAW

 You can customize given options when using interoperability from version 4 to version 5.

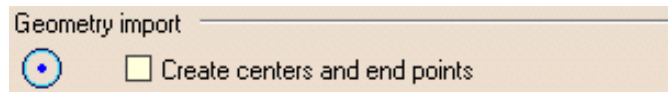


1. Select the **Tools -> Options...** command.
2. Click **Compatibility** in the list of objects to the left of the **Options** dialog box.
3. Select the **V4/V5 DRAW** tab.



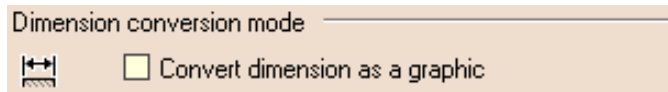
• **Geometry import**

When importing geometry, you can decide that you do or do not create centers and end points.



• **Dimension conversion mode**

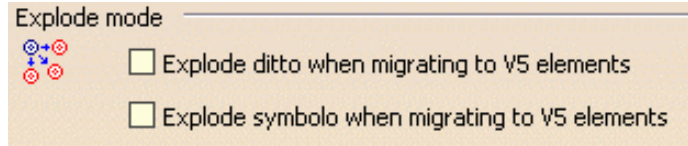
When converting dimensions, the resulting dimensions can or cannot be edited (modified).



- **Explode mode**

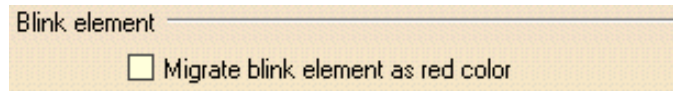
When importing a ditto (or component), this ditto is automatically exploded. As result each element in that ditto can be modified separately.

By default, you can convert the V4 Symbolo into Ditto in CATIA V5, but when this setting is activated you can keep the former mode, Explode symbolo mode.



- **Blink element**

When this setting is activated, you obtain a red element in CATIA V5 corresponding to the V4 Blink element. If you do not activate this option, the migrated Blink element is displayed in highlight.



- **Generative View style**

This setting specifies the generative view style that should be used when migrating AUXVIEW2 views from V4 to V5. Select an XML file from the list. The generative view style parameters defined in this file will be used for the migration.

If this setting is unavailable, you need to uncheck the Prevent generative view style creation setting from **Tools -> Options -> Mechanical Design -> Drafting -> Administration** tab.

For more information on generative view styles, refer to the *Generative Drafting User's Guide*.

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



# V4 / V5 SPACE



You can customize given options when using interoperability from CATIA version 4 to CATIA version 5. You can choose different parameters for pasted Surfaces/Curves **Continuity** and **Segmentation**. In order to optimize Continuity, this panel offers you 3 settings allowing you to control the deformation of V4 Geometry in CATIA V5.

In this page, there are two sections: explanations about the concepts followed by three examples for practice.

## Concepts: Controlling the Deformation of V4 Geometry in CATIA V5.

### Gap Healing

Curvature Improvement (Maximum Deformation)

Surface and Curve Sub-elements (Segmentation)

Migrating DETAILS SPACES used by Dittos

### Examples:

[How to use Gap Healing](#)

[How to use Curvature Improvement](#)

[How to keep Segmentation](#)

## Other Settings:

In this panel, you can also customize the migration of isolated Geometry from Details / Dittos by using a variable and the migration of Mock-Up Solids as PartBody:

### Interactive Migration of unused Geometry from Details

Isolated Geometry or unused Geometry is Geometry not in use for Geometry or family building. For more information, please refer to: [Optimizing V4 Migration: Deactivation of the V4 Parents' Filter in CATIA - V4 Integration](#).

### Isolated Mock-Up Solids Migration

Mock-Up Solids can be converted (as Result) into CATIA V5 as CGR or as PartBody.



CATIA V5 requires its geometry to be C2-continuous. When non C2-continuous geometry must be imported from CATIA V4 to V5, this geometry (Curves, Surfaces) is first optimized and then broken down into a set of contiguous geometries (Faces, Edges), each of them being C2-continuous.

This panel allows you to optimize and control the quality of the geometry and to obtain C2 Continuity (**deformation** is possible).

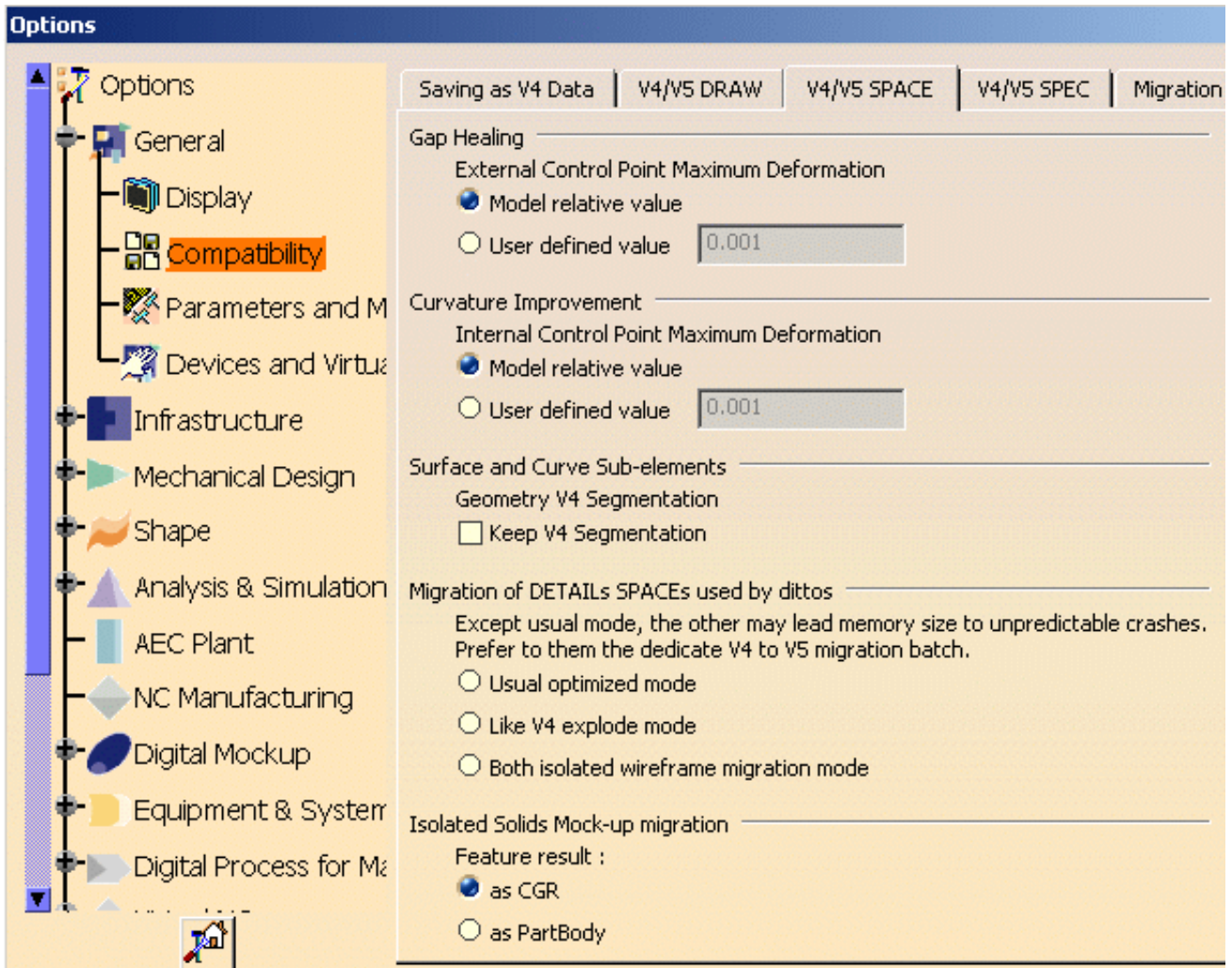


Open the .model of your choice.



1. Select the **Tools** -> **Options...** command.
2. Click **Compatibility** in the list of objects to the left of the **Options** dialog box.
3. Select the **V4/V5 SPACE** tab.



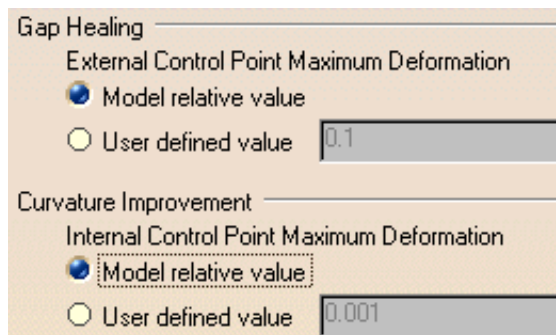


You can customize V4->V5 SPACE settings:

## Concepts:

# Gap Healing

Gap Healing has an impact on **External Control Points** and not on Inner Control Points. External Control Points are modified according to the following options. In this category you have the choice between two options:



- **Model relative value (corresponding to the Identical Curves Value defined by the model)**

External Control Points are modified according to the model's value or tolerance.

- **User defined value (External Control Point Maximum Deformation)**

Here is a schema to explain the position of the External Control Points, before activating this option:



and the displacement of the External Control Points after activating this option:

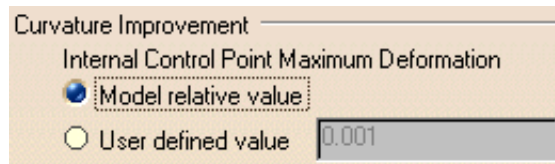


When this option is activated, it allows you to fill in the gaps between two **arcs/patches**, and therefore to optimize the geometry. Both patches are joined because the External Control Points have merged. There is however a **deformation tolerance** and this process can give the Surfaces a  $C^0$  Continuity. The examples in [How to use Gap Healing](#) will help you to understand the results.

## Curvature Improvement (Maximum Deformation)

For a Surface/Curve, the number of generated V5 Faces/Edges varies according to another parameter: the **maximum value** allowed for Control Point **deformation**.

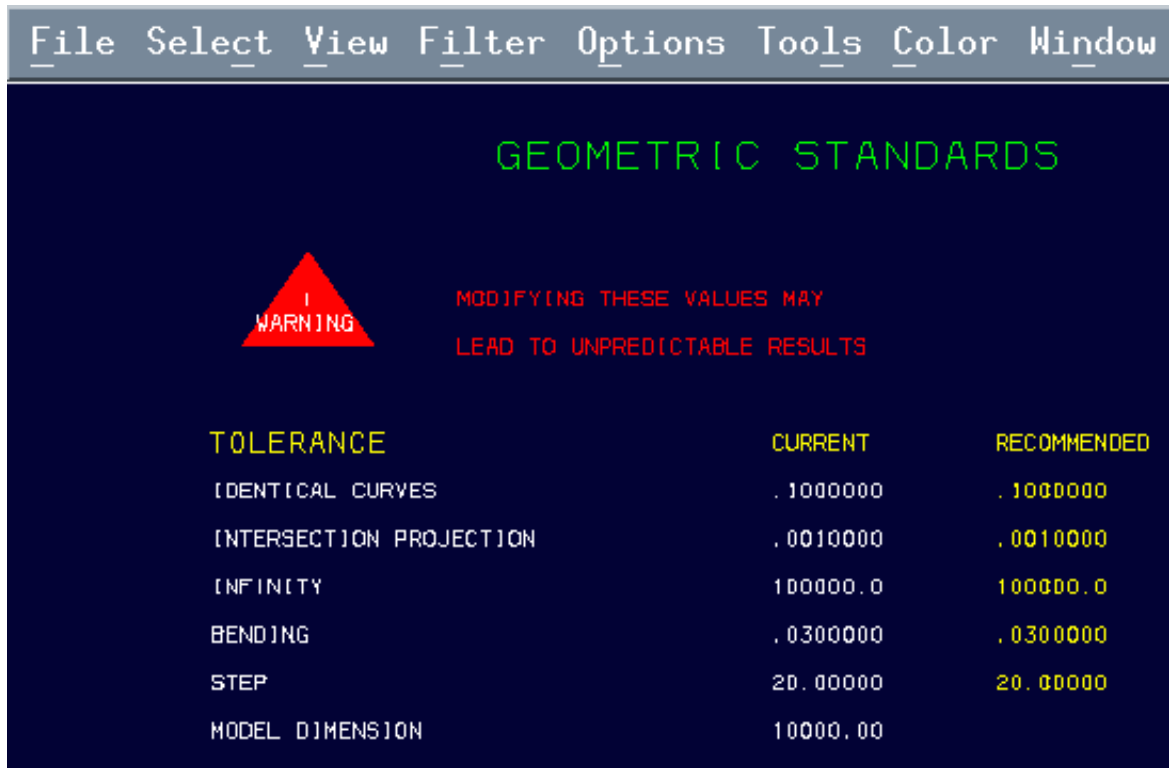
Curvature Improvement has an impact on **Inner Control Points**. Inner Control Points are modified (displaced) according to the value defined by the following options. If you select the **Inner Control Point Maximum Deformation** option, the Curvature Improvement value can be modified.



- **Use the default value (corresponding to the Projection Point tolerance defined by the model)**

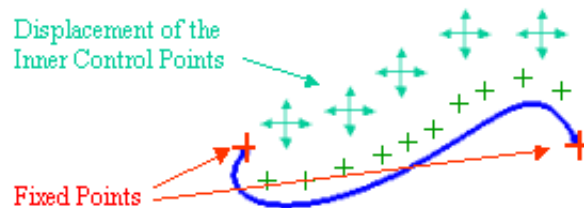
If you choose this option, the value which is taken into account is the maximum value between the V5 resolution value ( $10^{-3}$ ) and the Projection Point tolerance value ( $10^{-3}$  for instance, in V4) for a model dimension of 10 000m.

For a model dimension of 10m, the standard value of the Curvature Improvement is 0.001mm (corresponding to the V4 model's value). In CATIA V4, the geometric standards are listed in the following dialog box:



In order to understand the Curvature Improvement results, please refer to the paragraph entitled [How to use Curvature Improvement](#).

- **Inner Control Point Maximum Deformation (User Value)**

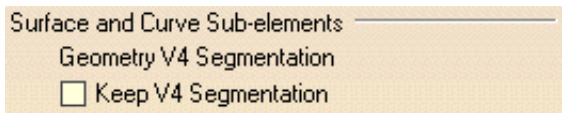


If the Inner Control Point Maximum Deformation value is equal to 0, the geometry is broken and an edge is created for each patch limit.

To decrease the number of cells/patches in the geometry, you need to select this option (default value) and to increase the tolerance/curvature improvement. The unit is the millimeter (V4 model unit in general).

# Surface and Curve Sub-elements (Segmentation)

When V4 Surfaces/Curves are migrated into CATIA V5, some Faces/Edges are created in V5. In order to keep a C<sup>2</sup> continuity with the geometric objects, Surface and Curve sub-elements may be deformed and cut into several pieces (where they are not C<sup>2</sup>).



- **Keep Segmentation**

By default, the **Keep Segmentation** option is not selected, the simplification of the geometry is active. If you want to reduce the geometry data size, select this option. If you want to keep the same number of sub-elements, deselect this check-box.

For instance, two arcs and two surfaces with a C<sup>2</sup> Continuity can be transformed into a single patch or arc in order to simplify the geometry. But if you keep the patch segmentation, these arcs and surfaces are not simplified.

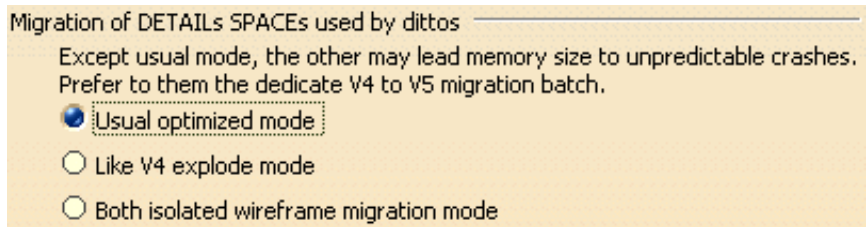
You can find an example in the section entitled [How to Keep Segmentation](#).

## Migrating DETAILS SPACES used by Dittos

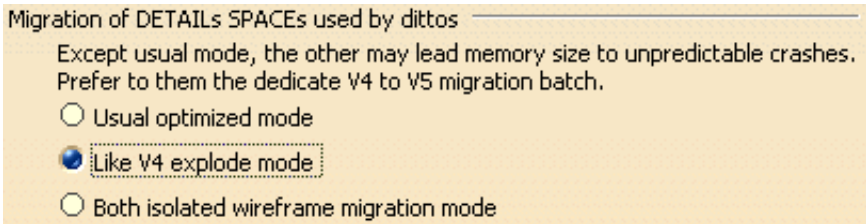


If you want to make a special migration for Dittos, you can add select one of the following settings:

- Usual optimized mode: corresponds to the normal and advised mode (with no setting). You can select this option if you want to see the geometry (curves, lines, surfaces) non-used by the Solid and contained in the Detail's area. This is what we call "Isolated Wireframe Geometry".



- Like V4 explode mode: before it was activated by a setting "set V4V5ForceDetailToExplode= 1" on NT and "export V4V5ForceDetailToExplode= 1" on UNIX. If you want everything to be migrated without CARWL, like the explode of a V4 model, you can select this option.



Both isolated wireframe migration mode: before it was activated by a setting "set V4ToV5DetailGeomRequired= 1" on NT and "export set V4ToV5DetailGeomRequired= 1" on UNIX.

Migration of DETAILS SPACES used by dittos

Except usual mode, the other may lead memory size to unpredictable crashes. Prefer to them the dedicate V4 to V5 migration batch.

Usual optimized mode  
 Like V4 explode mode  
 Both isolated wireframe migration mode

**Note:**

- be careful, use these functionalities only when it is really necessary. Do not use it systematically, otherwise there could be memory crashes.
- Use one or the other setting, but not both at the same time.
- If you choose one of these settings, be careful because it will be also activated in the interactive mode and in the Batch mode (CATV4ToV5Migration). For information, the batch offers another way to migrate workspaces.

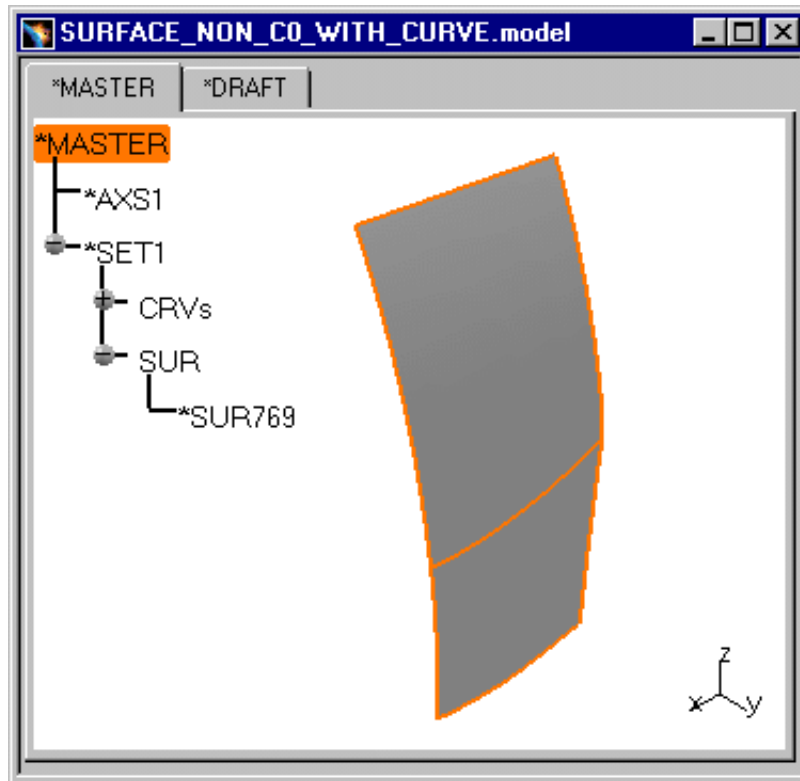


# Examples:

## How to use Gap Healing



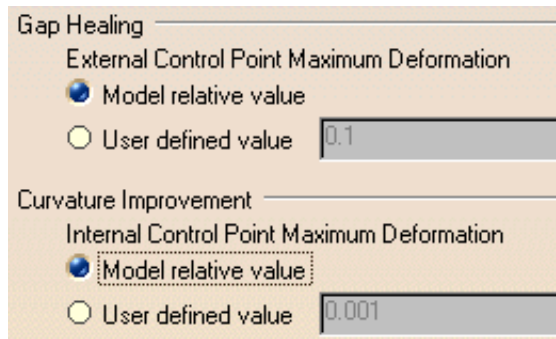
Open [SURFACE\\_NON\\_CO\\_WITH\\_CURVE.model](#).



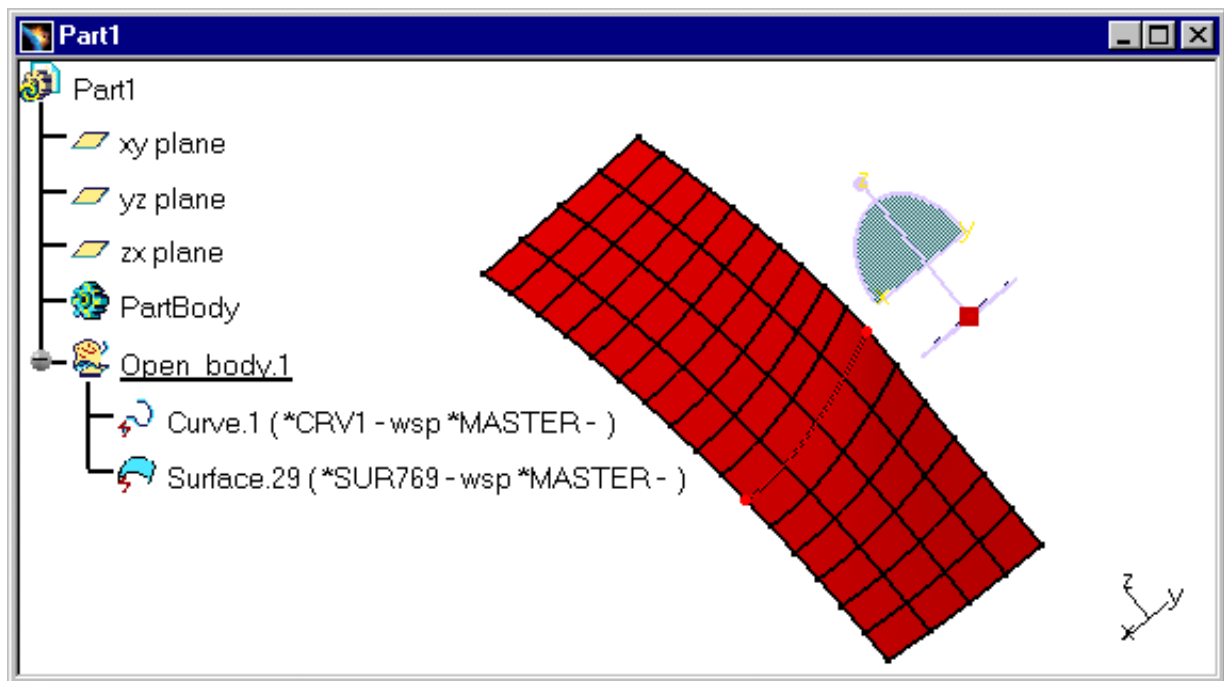
In all these examples, the option Keep Segmentation is not activated.

**Example 1:**

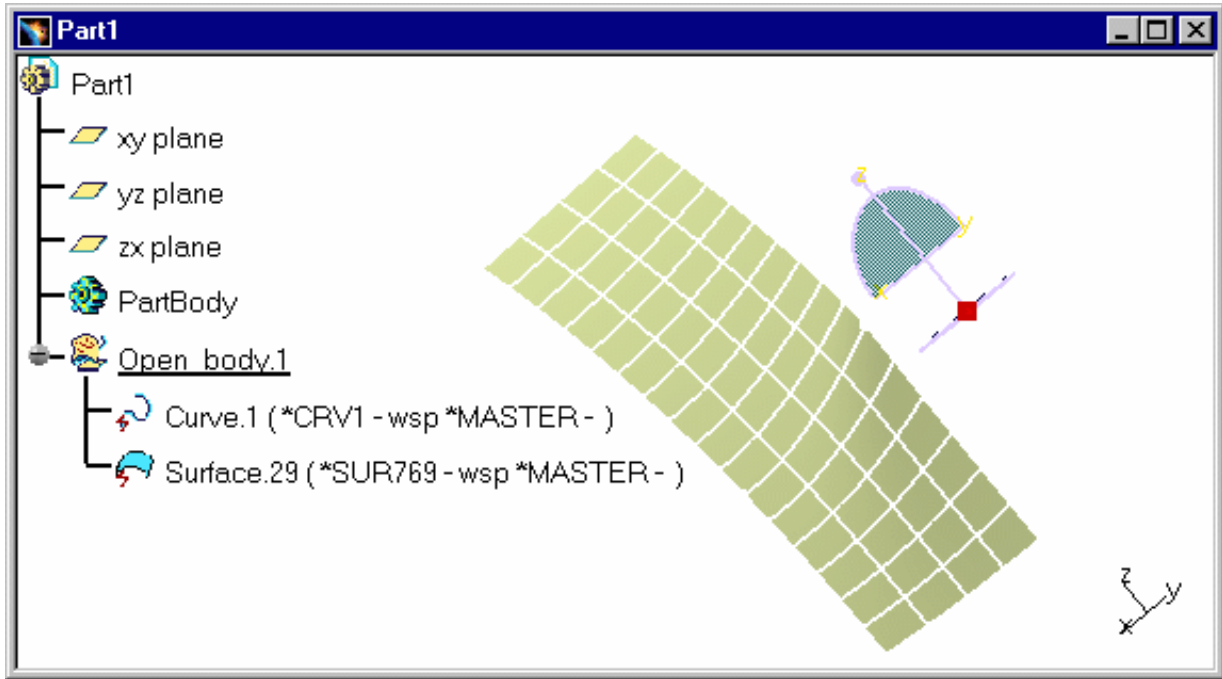
1. Select the following settings:



2. Copy / Paste the .model document into a CATPart and you obtain:



3. Update the CATPart:

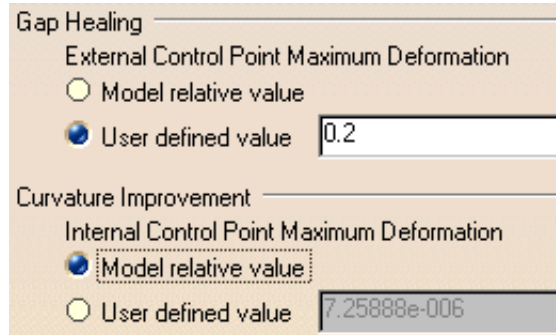


External Control Points are modified according to the model's value or tolerance.

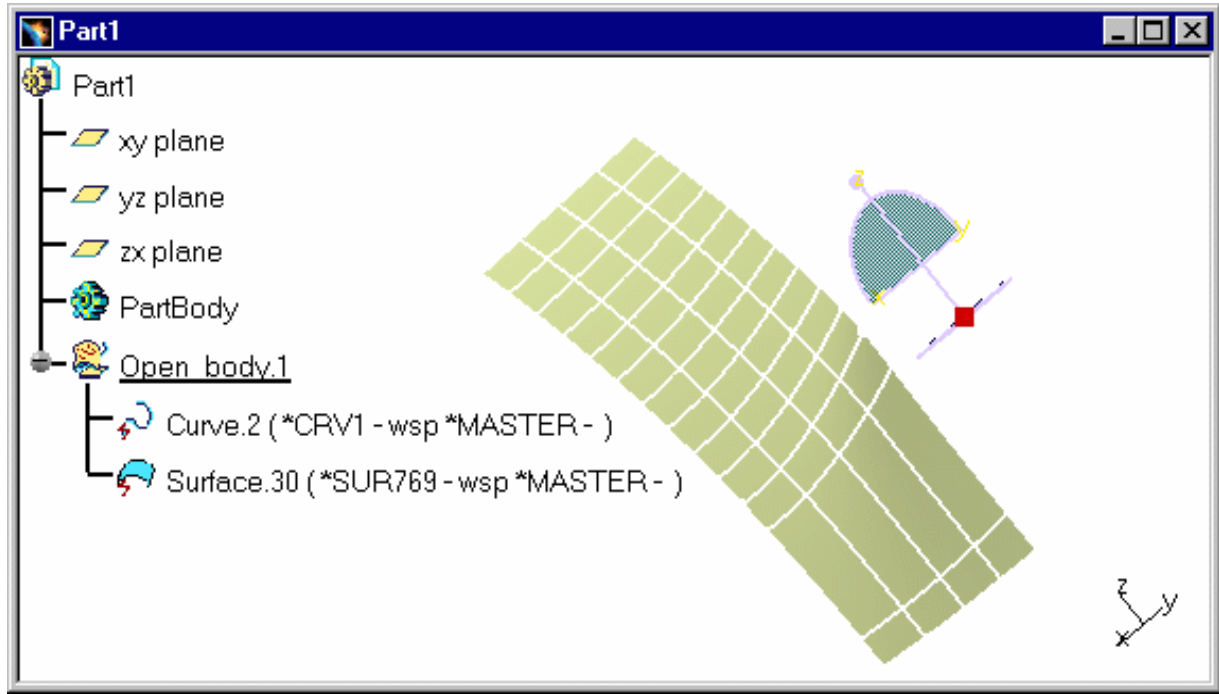


**Example 2:**

1. Select the following settings:



2. Copy / Paste the .model document into a CATPart and press the Update button.

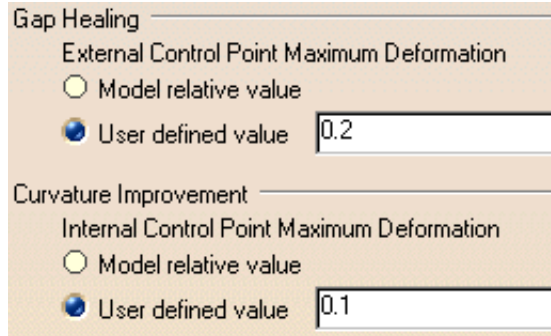


The number of cells is reduced because there is an optimization of the geometry.



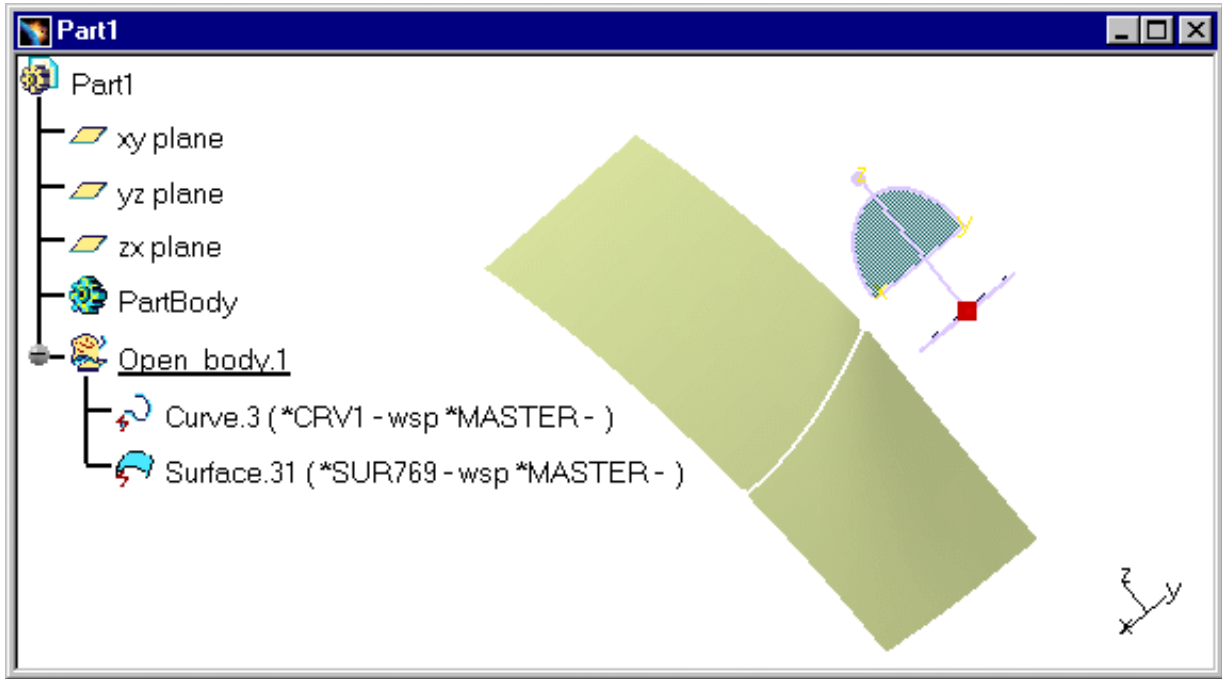
**Example 3:**

1. Select the following settings:



2. Copy / Paste the .model document into a CATPart and press the Update button.





The patches are joined because the External Control Points have merged.



## How to use Curvature Improvement



Open [SURFACE\\_CURVATURE.model](#)

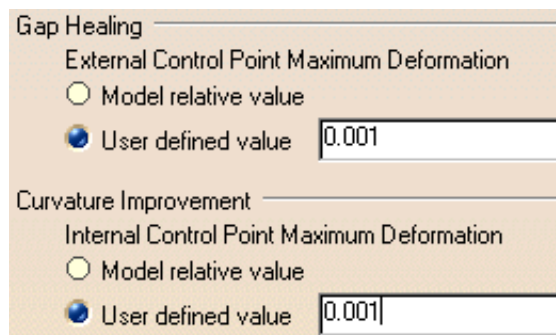
Enter the parameters in the Curvature Improvement (Maximum Deformation) area, according to the maximum value allowed for deformation. In all these examples, the option Keep Segmentation is not activated.

If the Curvature Improvement value is positive or equal to 0, the value appearing in the V4/V5 SPACE panel is recommended as Maximum Deformation.

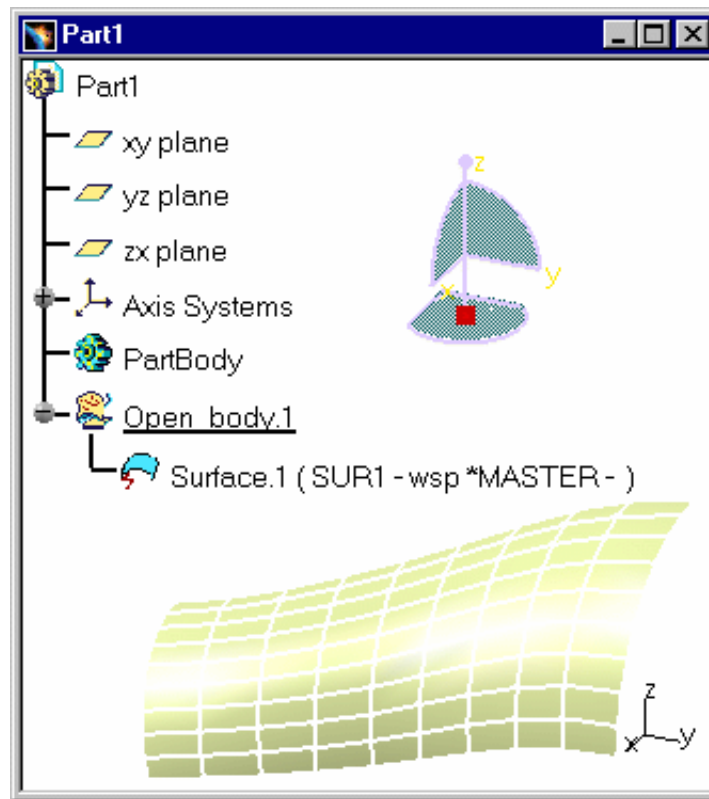
These examples reveal the influence of Curvature Improvement variations.

### Example 1:

1. Select the following settings:

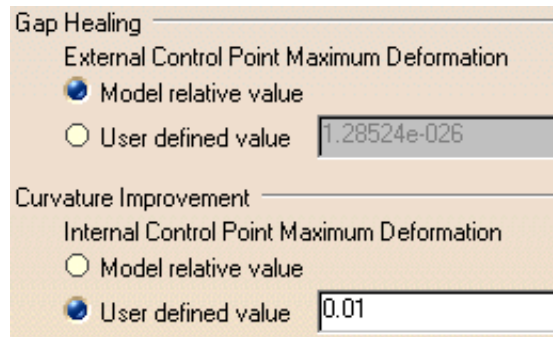


2. Copy / Paste the .model document into a CATPart and press the Update button.

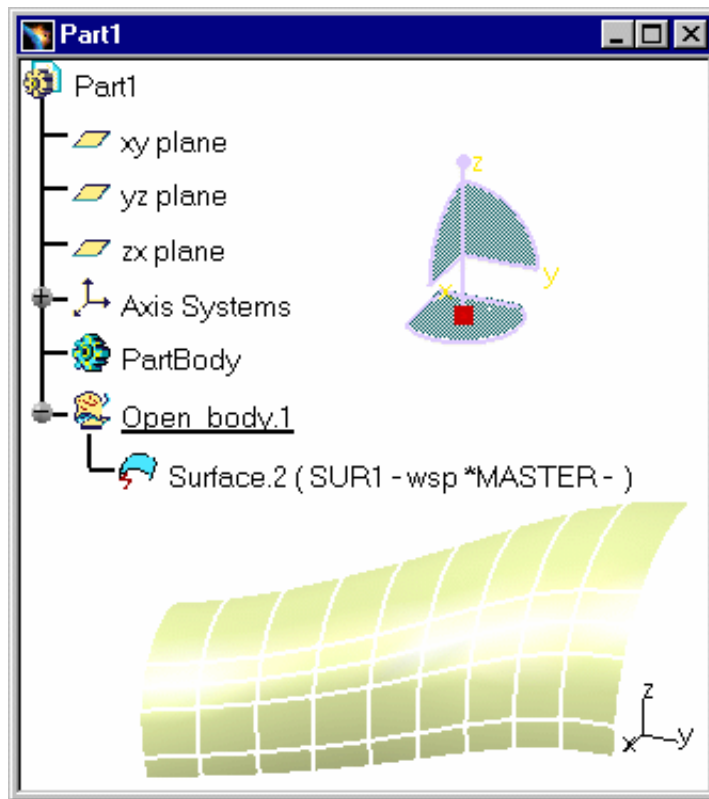


**Example2:**

1. Select the following settings:

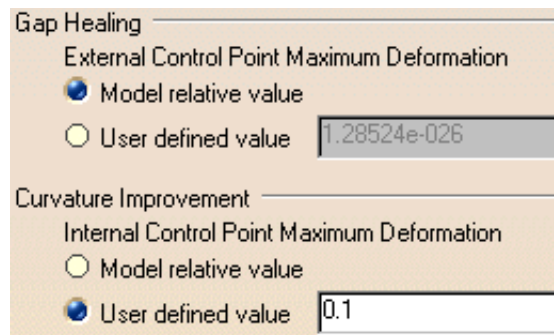


2. Copy / Paste the .model document into a CATPart and press the Update button.

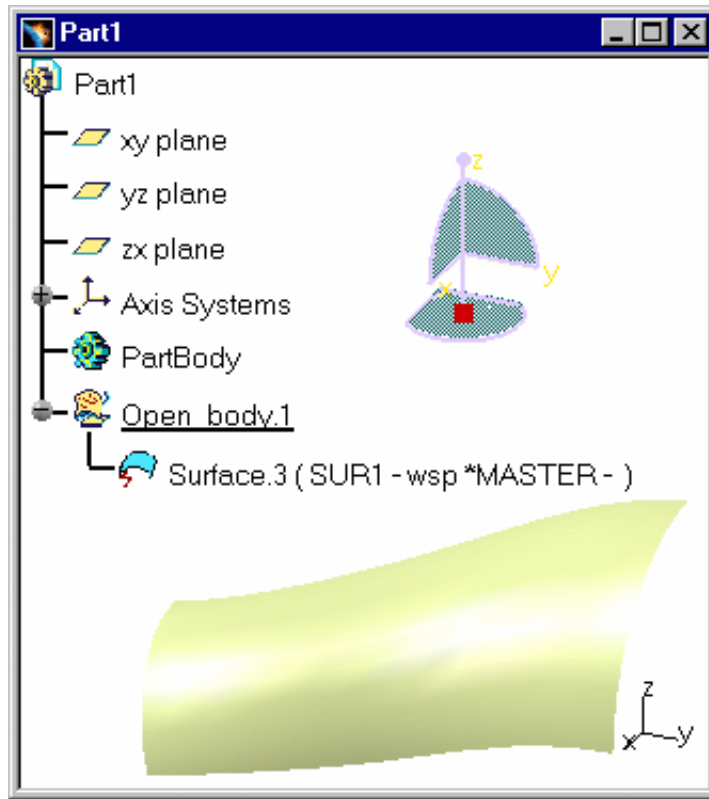


**Example3:**

1. Select the following settings:



2. Copy / Paste the .model document into a CATPart and press the Update button.



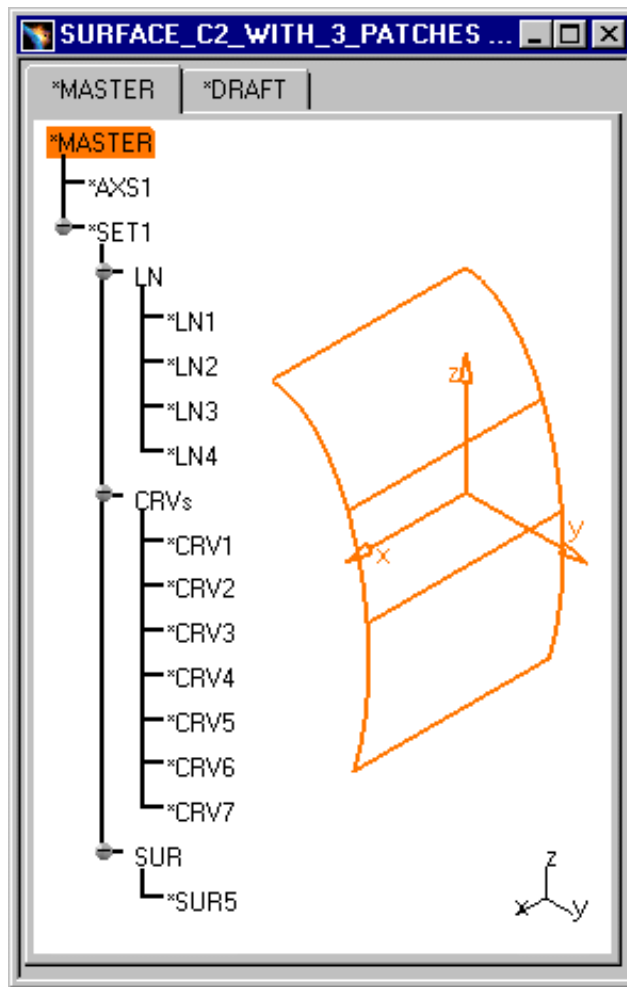
The higher the Curvature Improvement value is, the fewer Faces there are. **Tolerance** is more important when Curvature Improvement is high, which means that a slight variation is allowed between the original document and the new one.




## How to Keep Segmentation

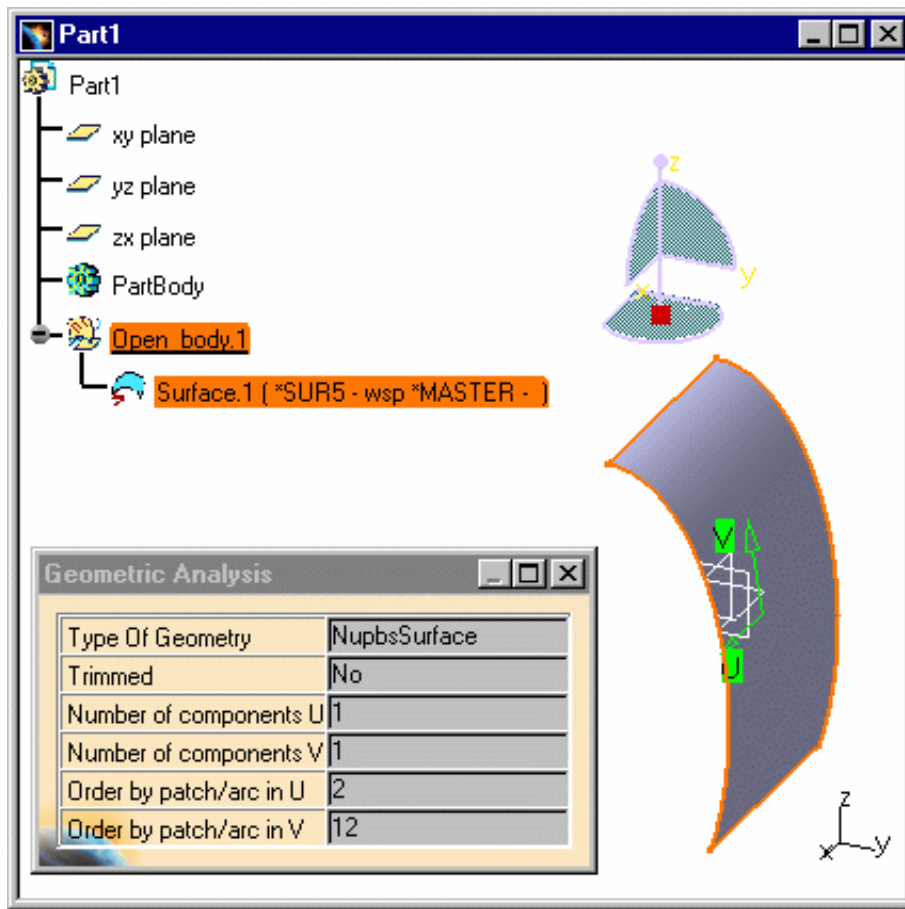


Open [SURFACE\\_C2\\_WITH\\_3\\_PATCHES.model](#) and copy / paste it into a V5 CATPart and update the V5 CATPart.



Here are two examples illustrating the behavior of this setting:

1. When the **Keep Segmentation** option is not activated and the Curvature Improvement (Maximum Deformation) is 0.1mm, go into the FreeStyle Shaper workbench and copy/paste \*SUR5 into a new Part. Select Surface.1 (\*SUR5) and click the **Geometric Information** icon . The Geometric Analysis dialog box appears:



Geometry Analysis delivers the following information:

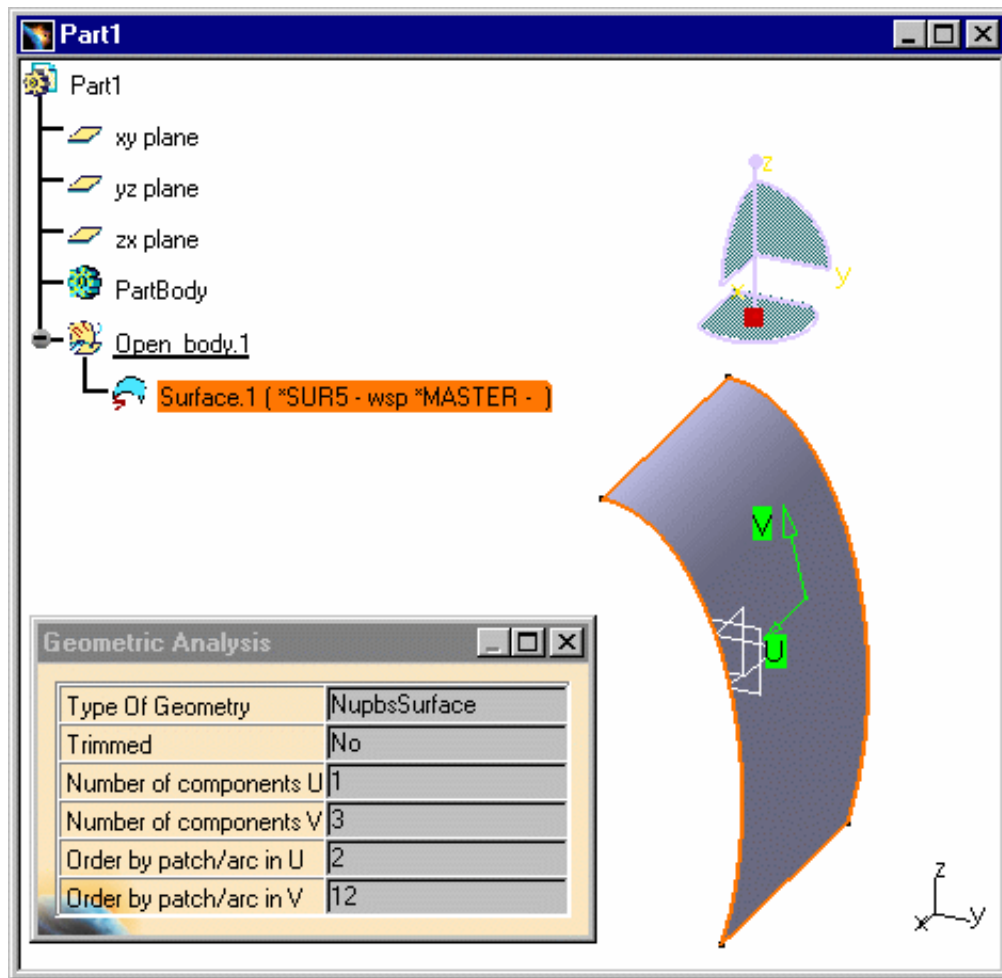
Number of components U: 1

Number of components V: **1** -> only one component because you do not have selected the **Keep Segmentation** option. The **default value** of this option is taken into account and the Surface has been simplified.

Without selecting the **Keep Segmentation** option, the nodes are removed and geometry is simplified within the tolerance.

- When the **Keep Segmentation** option is activated and the Curvature Improvement (Maximum Deformation) is 0.1mm, go into the FreeStyle Shaper workbench and copy/paste \*SUR5 into a new Part. Select Surface.1 (\*SUR5) and click the

**Geometric Information** icon . The Geometric Analysis dialog box appears:



Number of components U: 1  
 Number of components V: **3** -> 3 components because you have selected the **Keep Segmentation** option and **it forces the numbers and boundaries of the arcs and patches to be kept.**

## Copy/Paste Details and Dittos: Migration of unused Geometry from Details



This functionality allows you to get round the constricted migration of isolated Geometry from Details. The result is the same for the Geometry as if you had exploded all the Dittos in CATIA V4 and then migrated the \*MASTER, but in our example below everything is made in CATIA V5. The Copy / Paste As Result With Link (CPARWL) is not used for the Geometry. And the process is unchanged for the Solids.



On Windows and UNIX, before opening CATIA V5, you need to launch this option by entering the command (to make it visible in Tools->Options):

- **set V4ToV5DetailGeomRequired**



If you want to make a special migration for Dittos, you can add one of the following settings in the CNEXT launching shell:

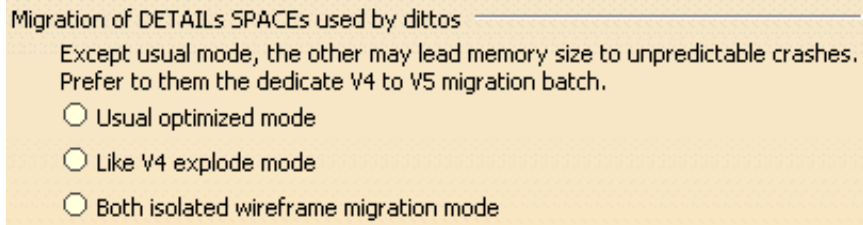
- **set V4ToV5DetailGeomRequired = 1**: if you want to see the geometry (curves, lines, surfaces) non-used by the Solid and contained in the Detail's area. This is what we call "Isolated Wireframe Geometry".
- **set V4V5ForceDetailToExplode = 1**: if you want everything to be migrated without CARWL, like the explode of a V4 model.

**Note:**

- be careful, use these functionalities only when it is really necessary. Do not use it systematically, otherwise there could be memory crashes.
- Use one or the other setting, but not both at the same time.
- If you add a setting in the CNEXT launching shell, be careful because it will be also activated in the interactive mode and in the Batch mode (CATV4ToV5Migration). For information, the batch offers another way to migrate workspaces.



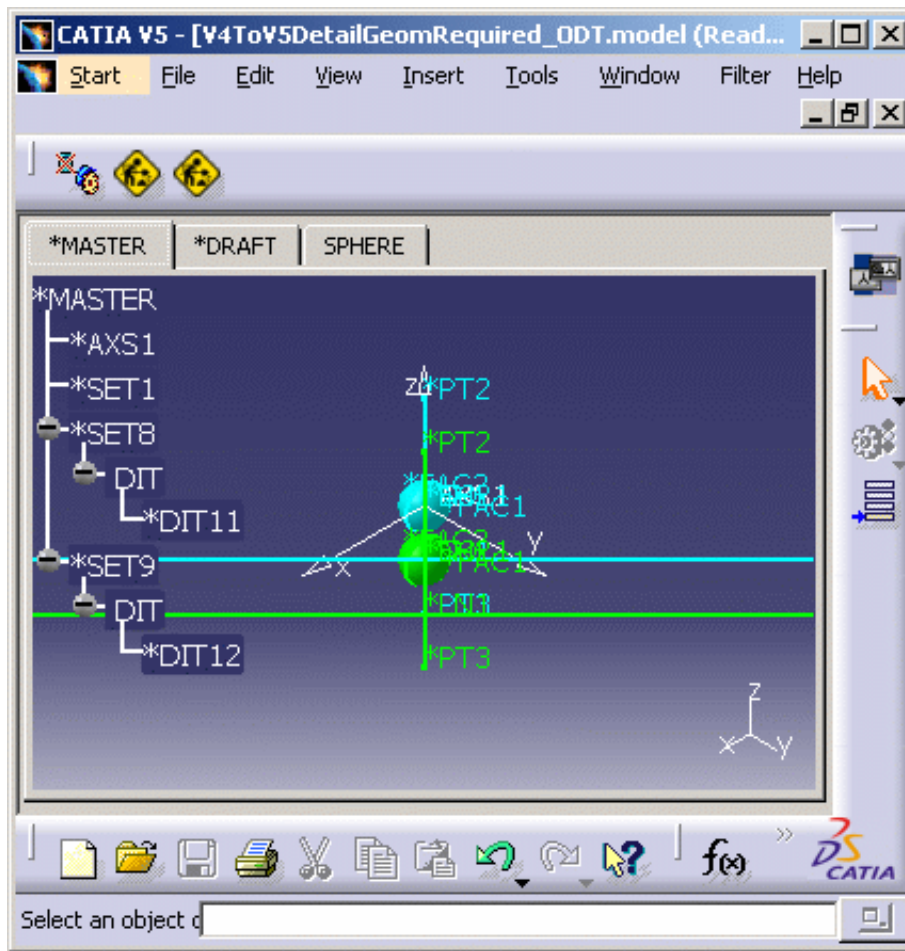
1. Select the **Tools -> Options...** command.
2. Click **Compatibility** in the list of objects to the left of the Options dialog box.
3. Select the **V4/V5 SPACE** tab.



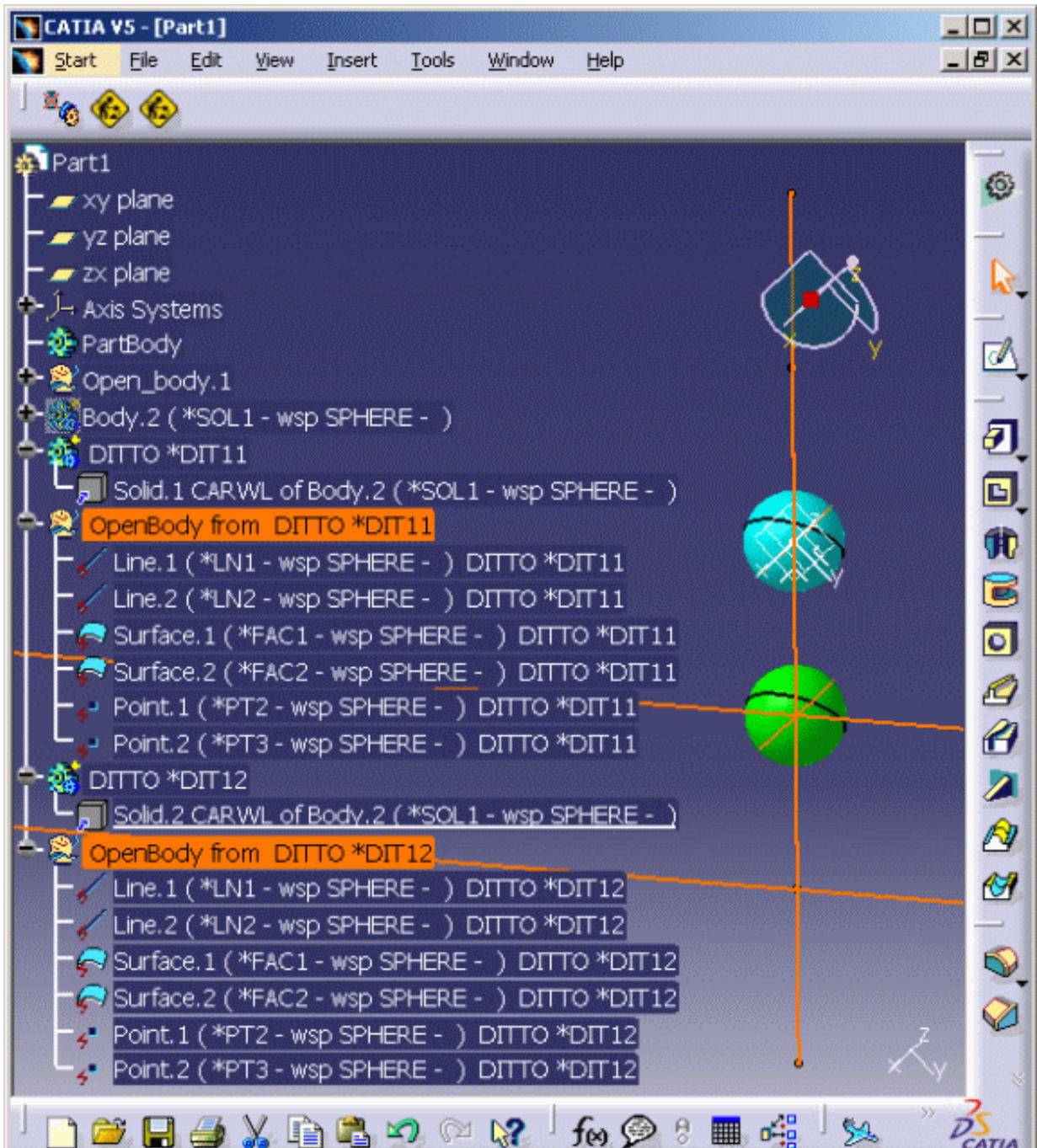
You can customize the following setting:

4. Check the option: "Migration of unused geometry from Details required".
5. Open V4ToV5DetailGeomRequired\_ODT.model in CATIA V5:





6. Copy/Paste (As Spec or As Result) \*MASTER into a V5 CATPart. And you obtain: [V4ToV5DetailGeomRequired.CATPart](#).



Thanks to this option, all the Geometry of the Dittos is migrated, it is gathered under the corresponding OpenBody: for instance, the isolated Geometry of DIT12 is under OpenBody from DITTO \*DIT12 in CATIA V5.

If you do not use the option, the isolated Geometry of the Dittos is not migrated within a single Part but under the Detail.



When you have finished, **do not forget to uncheck this option.**



## Isolated Mock-Up Solids Migration

Isolated Mock-Up Solids (SolidM) means that they do not have any history, any specifications.

For their migration into CATIA V5, you can choose between:

- the SolidM conversion "as CGR" meaning that you will only get the meshing view (visualization mode)
- or the SolidM conversion "as PartBody" meaning that you will be able to see the Geometry of the Solid, to edit it and you obtain an exact Solid.

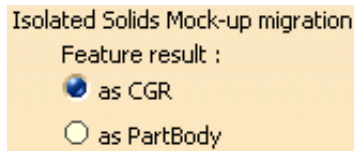


Launching the "As PartBody" migration mode:

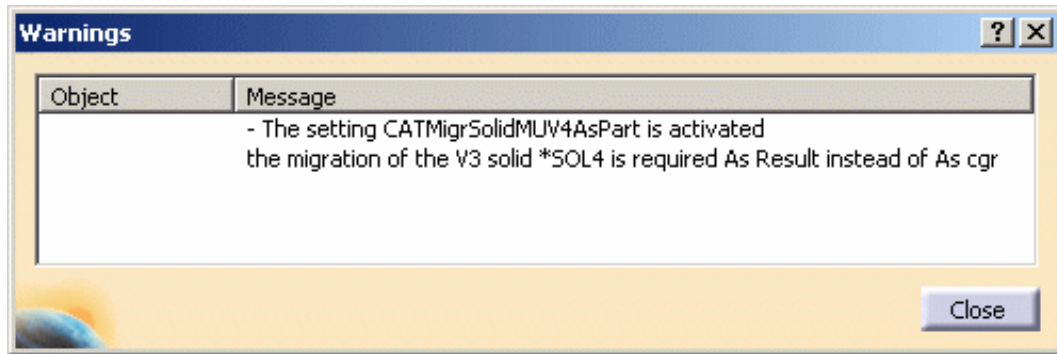
1. On NT, enter the following command: **set CATMigrSolidMUV4AsPart= 1**

On UNIX, enter the following command: **export CATMigrSolidMUV4AsPart= 1**

2. Or activate the setting "**as PartBody**" in the option entitled "Isolated Solids Mock-Up Migration - Feature Result" in **Tools -> Options -> General -> Compatibility -> V4/V5 Space** (by default, the "as CGR" option is checked):



If you select the "as PartBody" option or use the variable, you will also have the following message after the copy/paste of the SolidM into a CATPart:





If you do not use the variable nor the setting, the SolidM will be migrated as CGR.




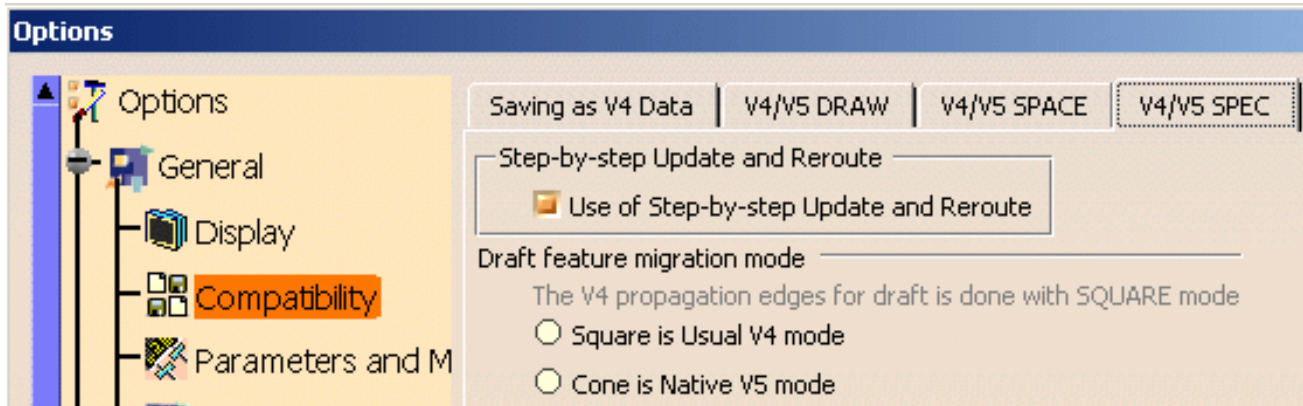
For more information about the Conversion of SolidM, please refer to the scenario entitled *Conversion of SolidM entities as Result (migrated as a PartBody) in CATIA - V4 Integration User's Guide*.

# V4 / V5 SPEC


 You can customize given options when using interoperability from version 4 to version 5.

 Open the .model of your choice.

-  **1.** Select the **Tools -> Options...** command.
- 2.** Click **Compatibility** in the list of objects to the left of the **Options** dialog box.
- 3.** Select the **V4/V5 SPEC** tab.




**Use of Step-by-step Update and Reroute:**

- Step-by-step Update: If you select this option, you do not need to update your document after the copy/paste of a .model into a CATPart or a CATProduct. The update operation is automatic and you no longer need to press the update button: .
- Automatic Reroute: it is a means to use the geometry in order to recover the Edges and Faces. This functionality is applicable only with updated EXACT or SMART Solids.

In interactive mode, this option is deselected by default.

In batch mode, this check-box is selected by default.

 What is the reroute mechanism?

When a contextual primitive (a Fillet, a Chamfrain, a Support or a Feature Draft) is unsolved in V5, the Modeler (in Part Design) is able to visualize the former geometric shape and the Fillet's Specifications.

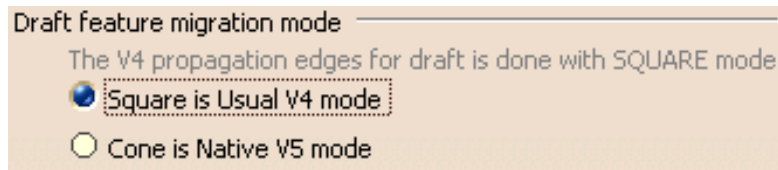
**Draft feature migration mode:**

The V4 propagation edges for draft is done with SQUARE mode:

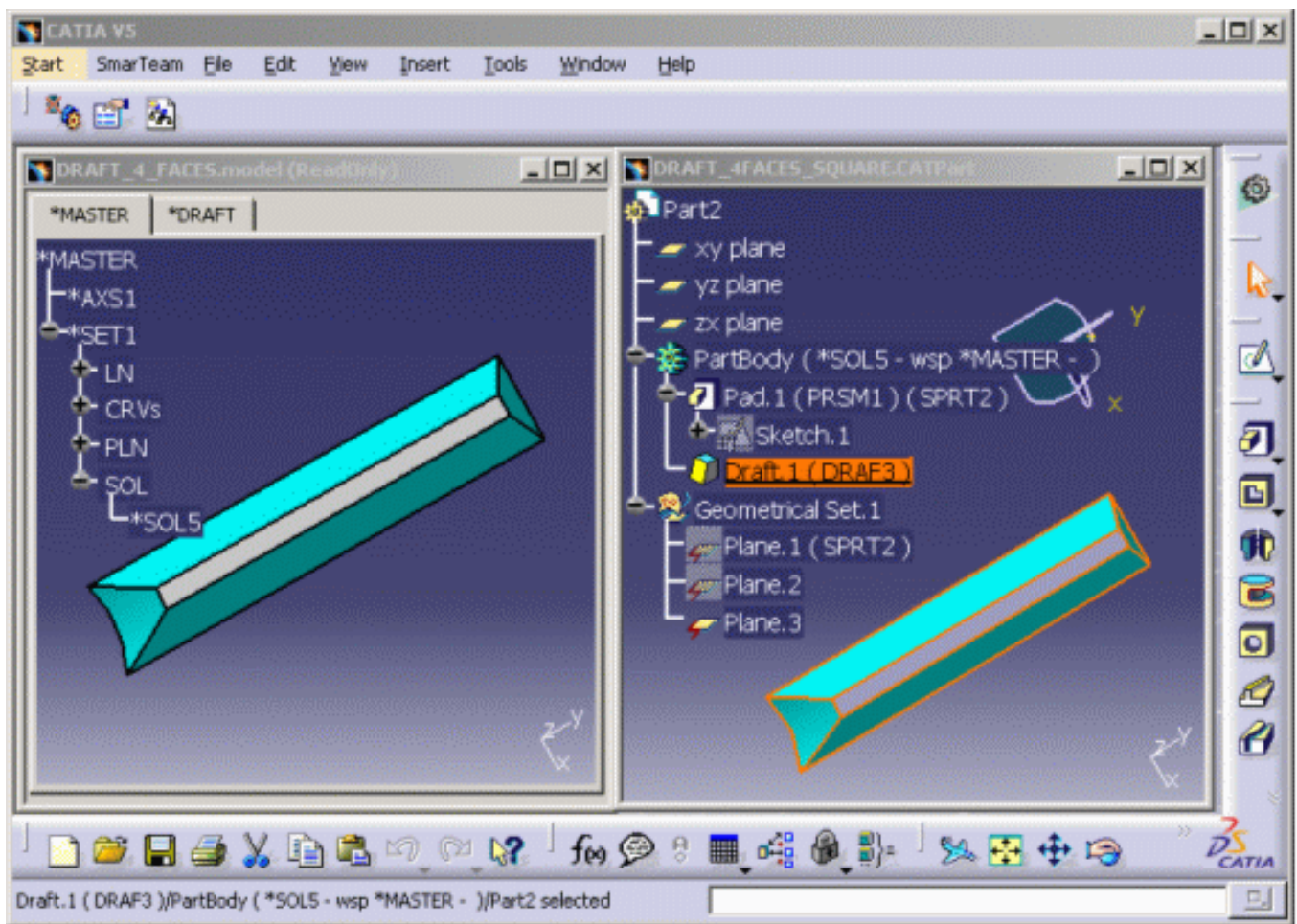
In order to get the best V5 results (close to CATIA V4) from copying / pasting As Spec a model containing Drafts into CATIA V5, it is advisable to choose the **Square** mode (formerly it was activated with: set DraftEquerre= 1 on Windows or export DraftEquerre= 1 on UNIX). Indeed, this is the mode used in CATIA V4.

 Open [DRAFT\\_4\\_FACES.model](#).

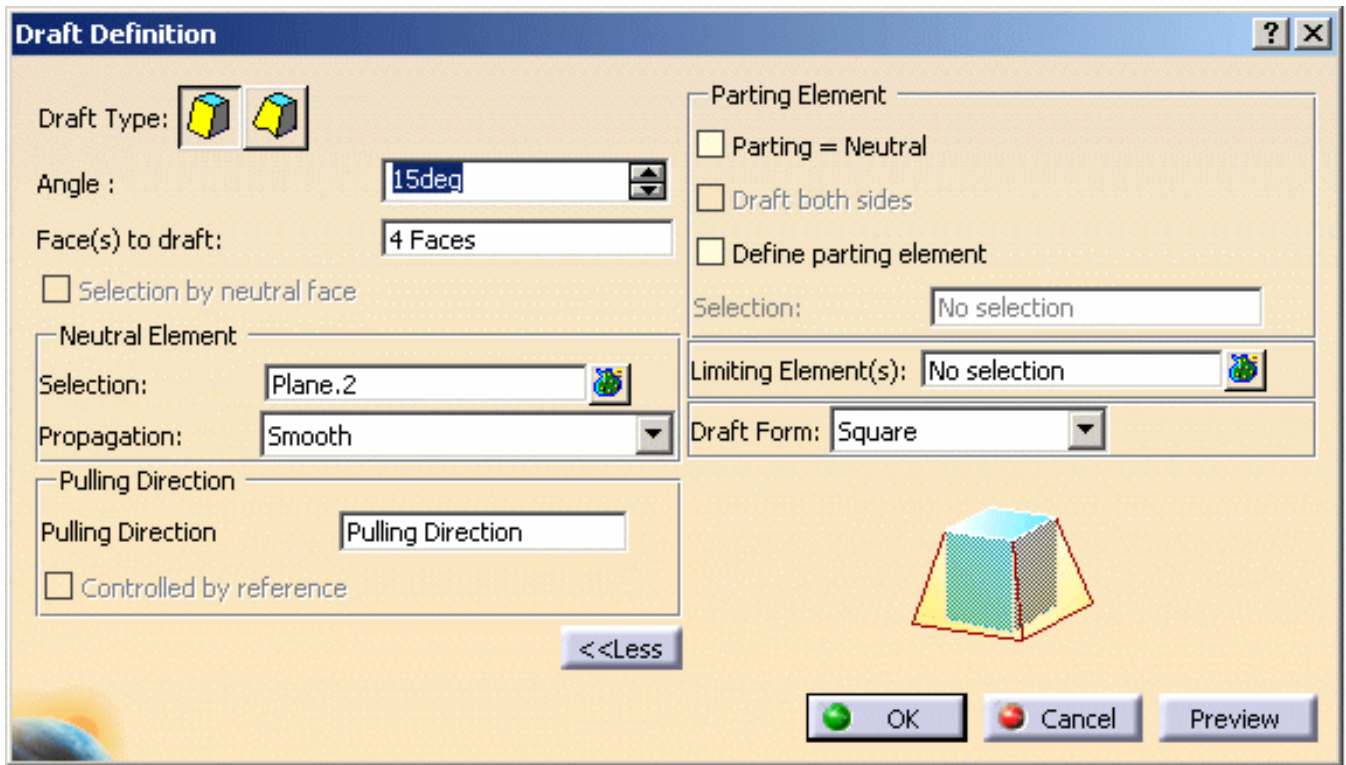
1. Select the **Square** option and click OK, all the migrated Drafts (during the copy / paste As Spec from CATIA V4 to CATIA V5) will be migrated in this mode.



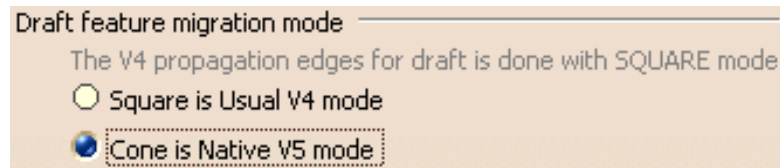
2. Select \*MASTER and copy / paste it in a CATPart.
3. In the CATPart, **DRAFT\_4FACES\_SQUARE.CATPart**, double-click Draft.1 in order to see the Draft Definition panel.



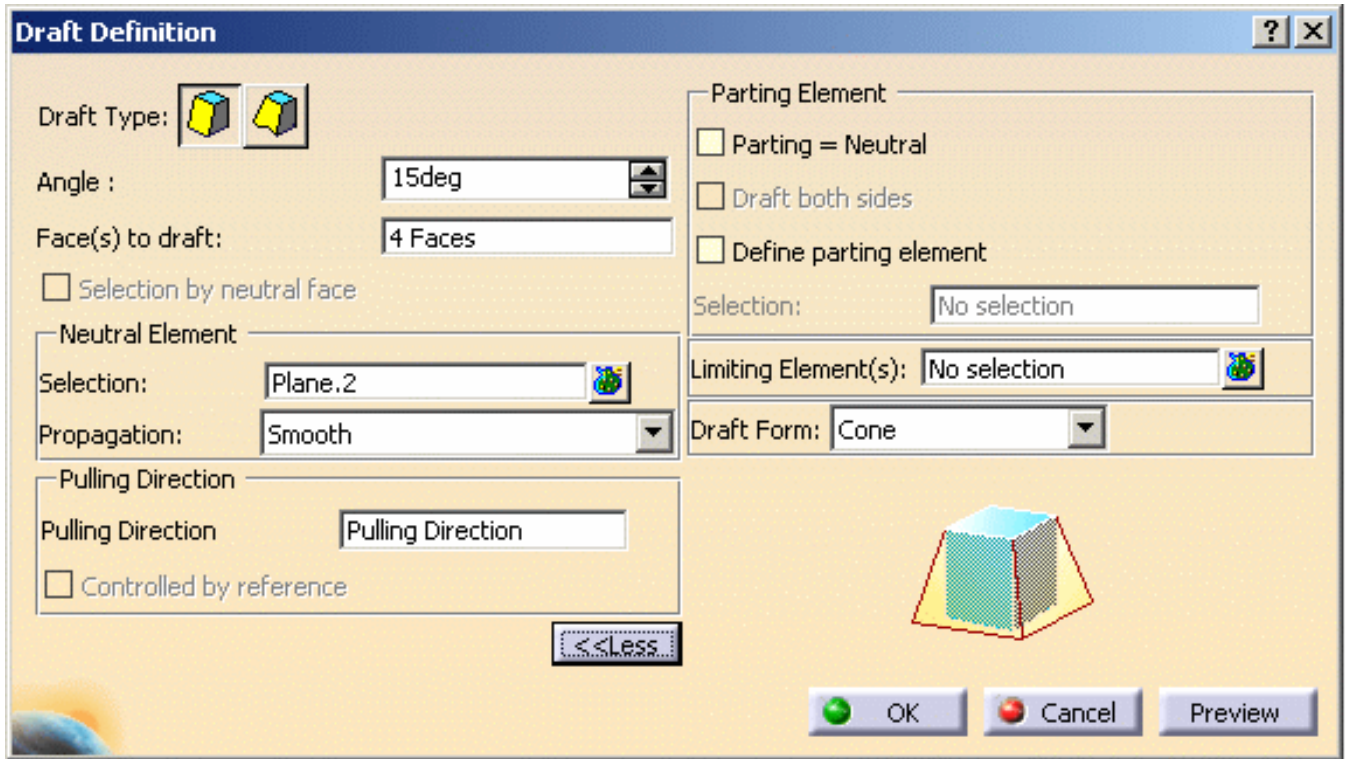
In the Draft Definition panel, you can see the property "Square" in the Draft Form field.



4. If the paste of a Draft or the update of the resulting Draft goes wrong, select the **Cone** mode (formerly it was activated with: set DraftCone= 1 on Windows or export DraftCone= 1 on UNIX). It will allow you to paste the Draft:



5. Select \*MASTER and copy / paste it in a new CATPart.
6. In the CATPart, **DRAFT\_4FACES\_CONICAL.CATPart**, double-click Draft.1 in order to see the Draft Definition panel. This time, the Draft Form is "Cone".



However, after you have copied / pasted the Draft in **Cone** mode and before leaving your CATIA V5 session, you should not forget to re-select the **Square** mode so that the **Cone** mode is not chosen by default in further sessions.



# Migration Batch



This task shows you how to customize Migration Batch settings.



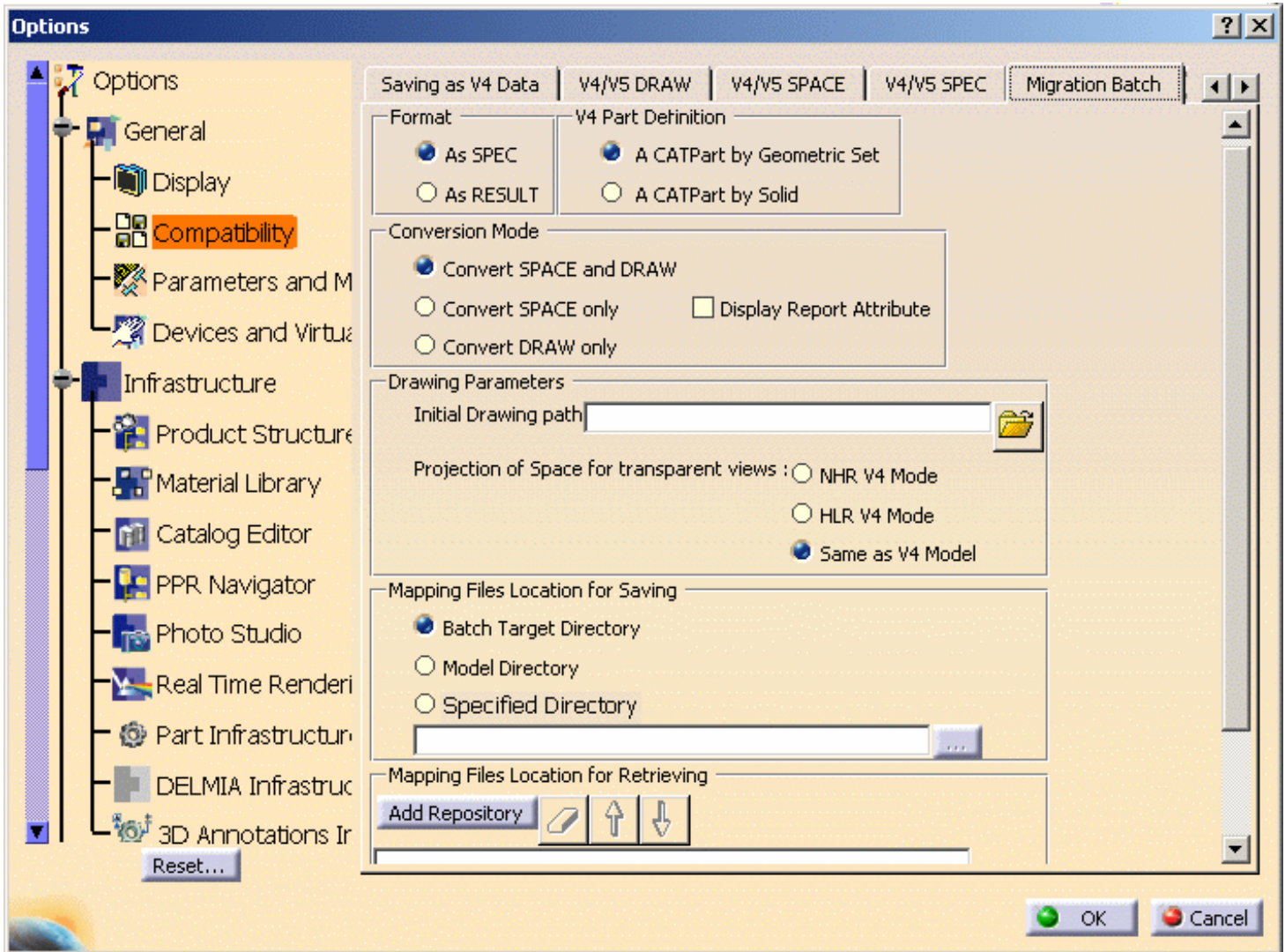
1. Select the **Tools -> Options...** command.

The Options dialog box appears with the category tree in the left-hand column.

2. In the **General** category of the Options Tree, select **Compatibility** and click the **Migration Batch** tab.

Or when the Migration Batch panel is open:

1. Select the **Options** button. The Batch Options dialog box appears with five tabs.
2. Click the **Migration Batch** tab.



## Migration Batch

The following options are only useful in the Migration Batch context, not with interactive CATIA V5.



- **Format: Selecting the format of Migration in Batch Mode (AS SPEC or AS RESULT)**

By default, the format of this migration is AS SPEC, but the user can select the AS RESULT option.

With the AS SPEC format, you can get the model's specifications in a first place and the geometry in a second place (after updating the document). Then you can modify the elements you have converted into CATIA V5.

The AS RESULT format allows you to migrate the V4 elements that cannot be converted AS SPEC. And this operation is more efficient in terms of performance.

- **V4 Part Definition**

A CATPart by Geometric Set: if you check this option, the "V4 Part" definition is the Geometric Set. Therefore, the subsets will be migrated into a V5 CATPart document.

A CATPart by Solid: The "V4 Part" definition can also be a Solid. The Solids will be migrated into CATParts and the non-solid elements (for instance: Wireframe) into a CATPart as well.

- **Conversion Mode:**

**Selecting the Migration of SPACE / DRAW data in Batch Mode:**

According to its content, a model is migrated into several documents : a CATPart and / or a CATProduct and / or a CATDrawing. It is possible to separate the treatment of SPACE data and DRAW data. In the Batch window, there is an **Options** button giving you access to this Tools-Options panel and you can choose between Space And Draw, Space, and Draw.

When a model contains **DRAW data**, they are migrated as a unique document : a CATDrawing.

Some entities undergo a few changes during the V4 to V5 conversion:

- V4 DRAFT -> V5 Sheet
- V4 Detail -> V5 Detail Sheet
- V4 Views -> V5 Views

In the Batch Mode, all Draw and Detail workspaces are migrated in CATIA V5. On the contrary, if you Copy / Paste a Draft or a View interactively, only referenced Details are migrated.

**Batch Migration of SPACE data:**

There are 2 Part definitions; the migration of each Set respects the definition determined by the "V4 Part definition" a few lines above.

Every Workspace containing several Dittos or Set instances is converted as a CATProduct. Under this CATProduct, there are as many components as Dittos (Part or Product instances) and Sets.

The impacts of the V4 to V5 conversion on the entities:

- V4 Set -> V5 Part
- V4 Detail -> V5 Part or Product
- V4 Model -> V5 Part or Product and/or CATDrawing
- V4 Ditto -> V5 Product component



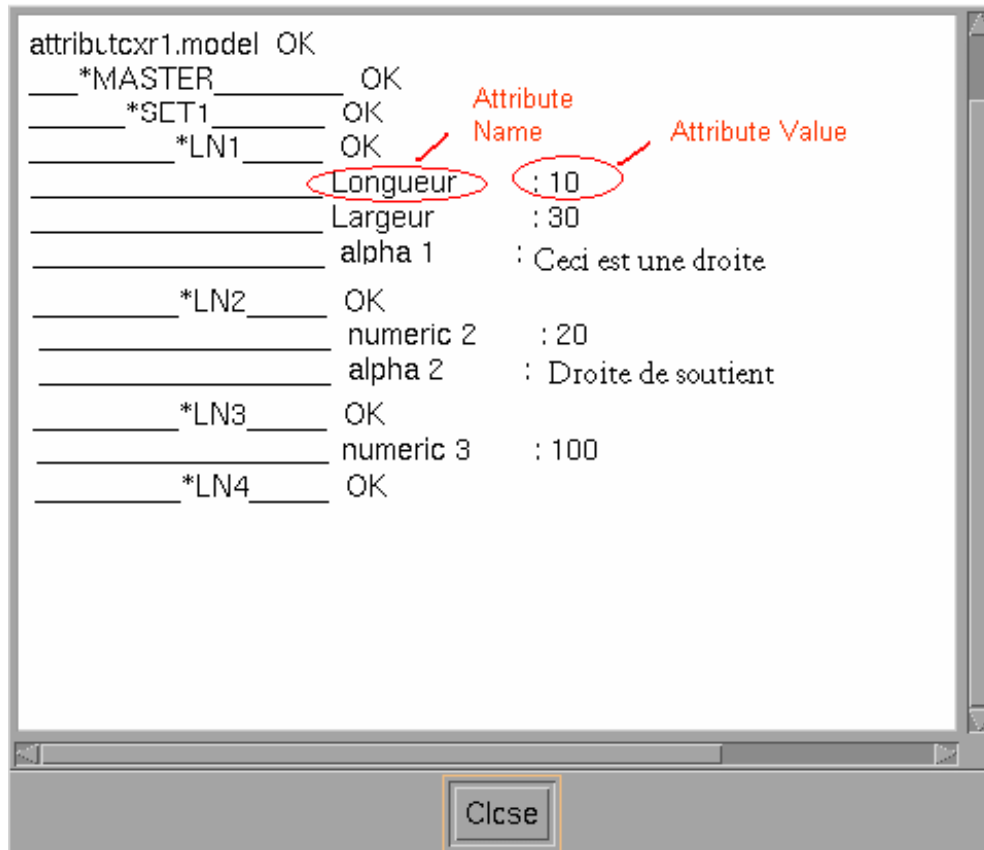
### Display Report Attribute:

This functionality allows the visualization of 3D elements attributes in the Migration Report. For each element, the list of its attributes is displayed with the following data:

- The name of the attributes
- The value of the attributes

The CATV4ToV5Migration interface is unchanged.

The Report panel displays attributes information.



There are some limitations:

- Display of geometrical elements' attributes (2D and 3D elements) only. You can find the information about the 3D elements in the interactive report and about the 2d elements in the .rpt file.
- Display of only the attributes which have a value. For example, empty words that is to say words with no number are not displayed.

### • Drawing Parameters

**Initial Drawing Path:** Select a .CATDrawing document which will serve as a template. The standard used by this document will be used during the migration. If you leave this field blank, the standard used by default during the migration will be the last one you used when creating a drawing.

**Projection of Space for transparent views:** Specify what kind of projection mode you want to use for transparent views during the migration: the NHR V4 mode, the HLR V4 mode or the same projection mode as the V4 model.


• **Mapping Files Location for Saving (to retrieve Multi Model Links)**

This functionality allows you to find pointed entities in V4 MML Solids and to retrieve associativity in CATIA V5. You can store them in the following directories:

- Batch Target Directory
- Model Directory: a mapping table per model is available; it is stored at the same place where the associated model is.
- Source Directory: you can specify the directory path of your choice.


• **Mapping Files Location for Retrieving**

When you migrate the pointing model, you need to know where the mapping table is, in order to retrieve the V5 entities.

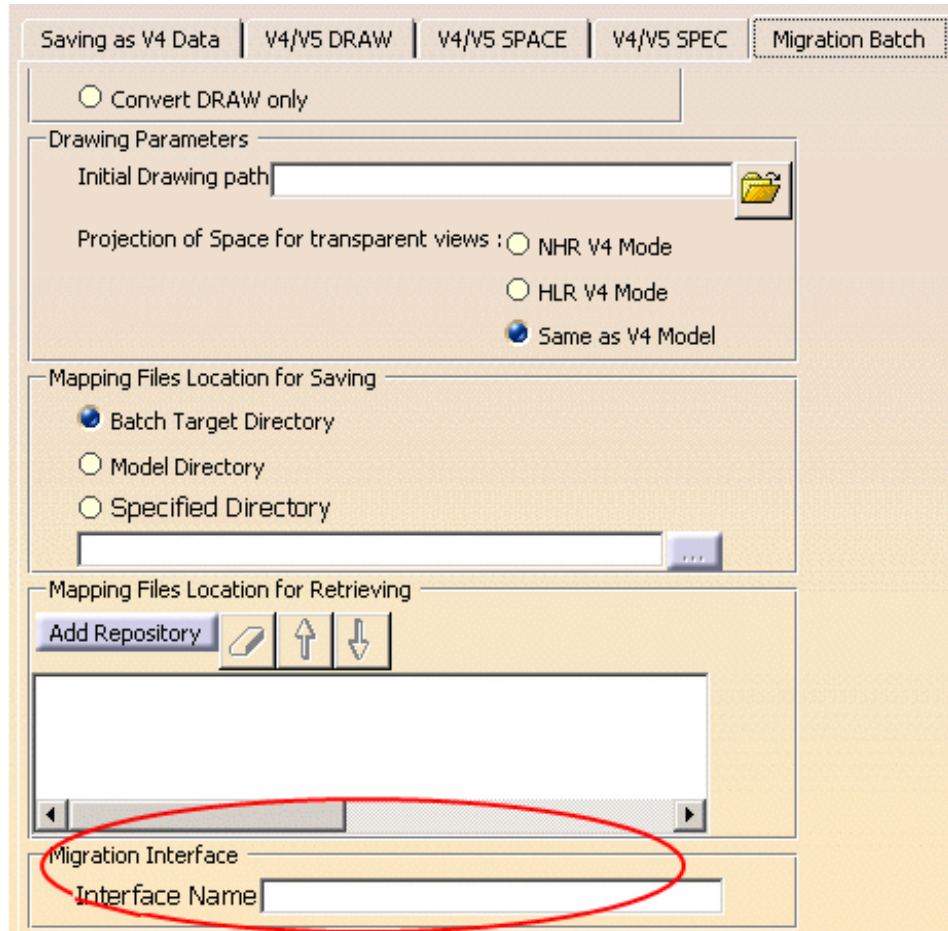
 The mapping table has the same name as the model; this is why it is important not to change the name of the mapping table. It is also not recommended to rename V5 documents otherwise you will not be able to find them in the next migrations, and therefore to recreate the V5 associativity.

If you cannot find the mapping table, a Migration As Result will take place: first, the pointed models are migrated and then the pointing models (asynchronous migration).

If you cannot find the pointed element, the link between the models cannot be reproduced.

Click **Add repository** to find the mapping table location. With the arrows (Up and Down), you can move in the list and with the rubber button , you can erase a file location.

**Migration Interface:**



- **Interface Name:**

This option allows you to customize Applications' migration from CATIA V4 to CATIA V5. You can specify a particular behavior for this migration. You can choose how your applicative data will be migrated by writing source code (the name of the interface) in this empty field.

Click **OK** to confirm.



# Setting the DELMIA/Deneb Compatibility Tab



The following procedure describes how to set the DELMIA/Deneb Compatibility tab. Setting this tab enables your V5 system to use D5 data created with DELMIA (formerly Deneb Robotics) software.

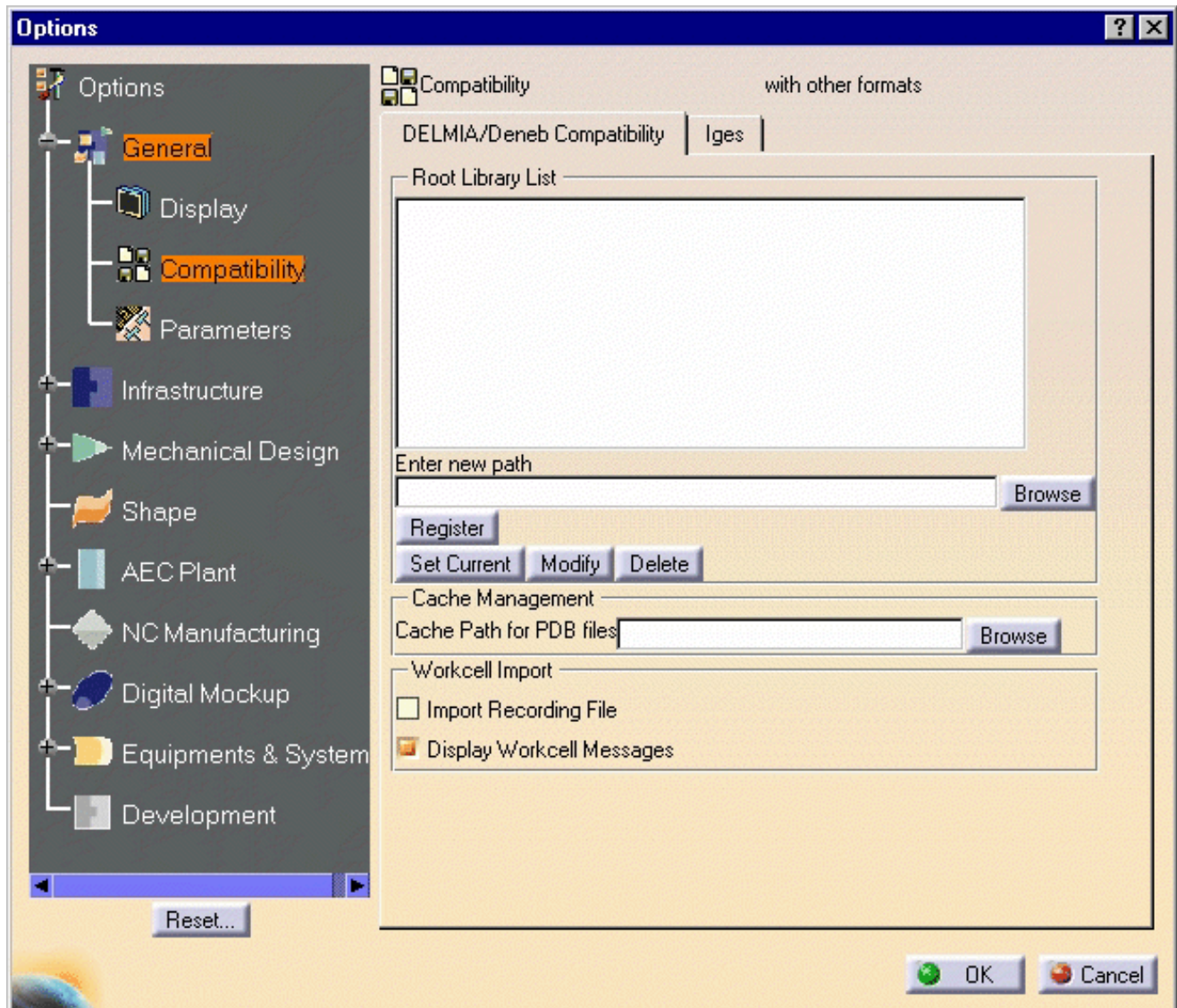


1. Select **Tools->Options**.

By default, the General option is highlighted.

2. If the gray ball next to General (in the left bar) has a plus (+) sign next to it, and does not show its three sub-options, click on the ball to expand the tree.

The compatibility option appears, as shown in the figure below.



3. Select the "DELMIA/Deneb Compatibility" tab.
4. Enter the full path name of the project library in the PATH block.
5. Click the Register button to register this path in the root library list.



Several project libraries can be entered in the root library list. If more than one library is currently in the list, select the one which you would like to be the current project and click the Set Current button. This will place the selected library at the top of the root library list.

6. Press the OK button.



Whenever work is saved or imported, the system looks in the current project directory. The last project selected by the Set Current button remains the current project until it is changed again.



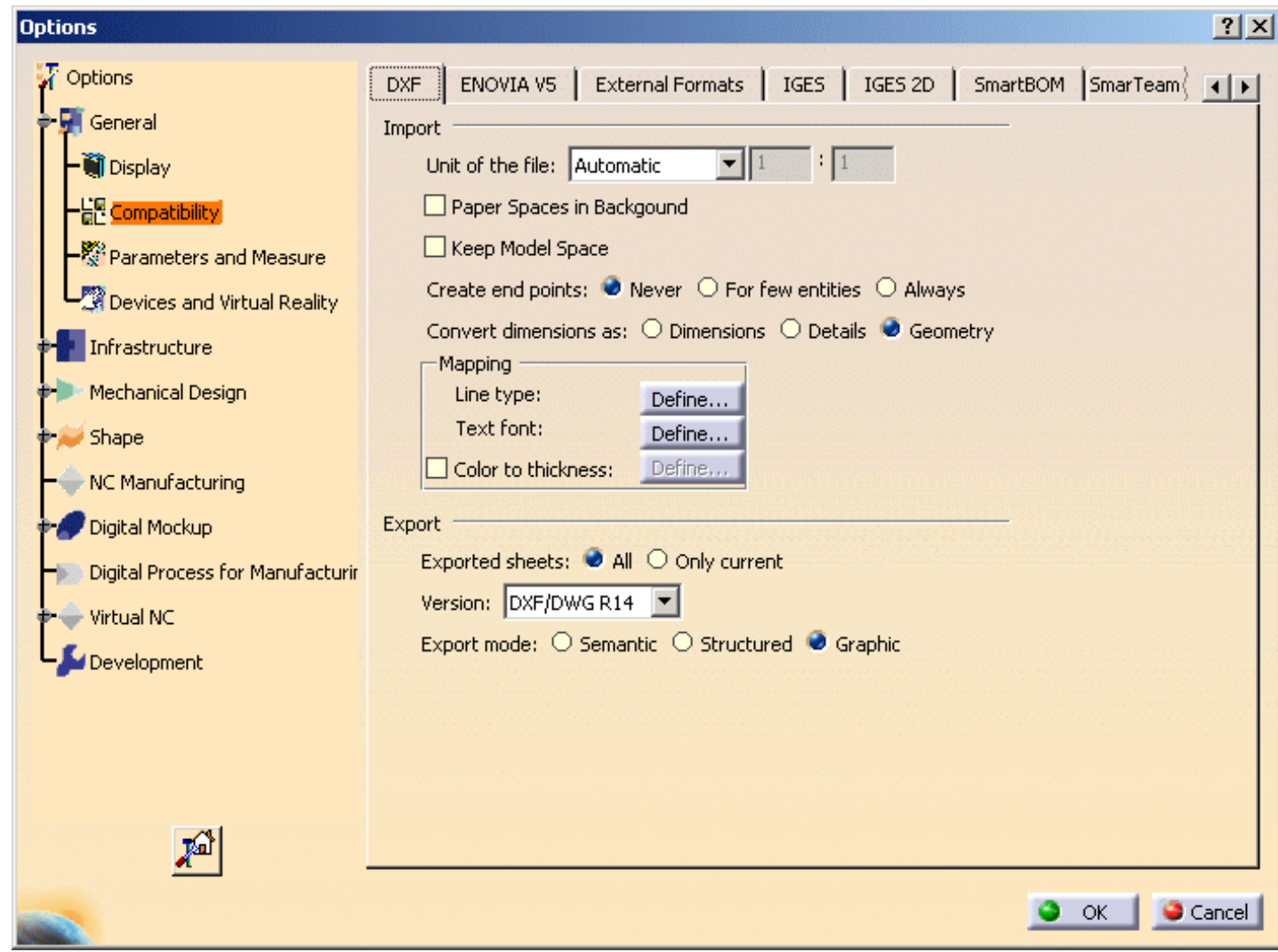
# DXF/DWG Settings



This task shows you how to customize DXF/DWG settings

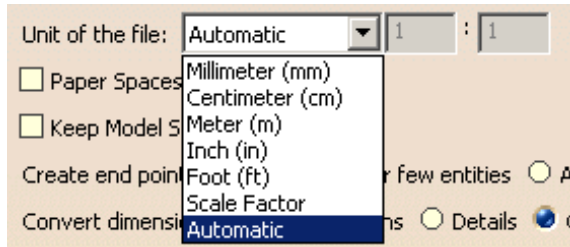


1. Select the Tools->Options... command.
2. Select the General category, then the Compatibility category, then click the DXF tab.



## Import

 Unit of the File



By default, the option is set to **Automatic** and the unit of import is determined automatically (either millimeter or inch) for the best possible resulting drawing. However, in some cases the resulting drawing is not satisfactory and requires another unit. Select this unit in the list. Then restart the import.

If you have selected **Scale Factor**, enter the value of the scale factor between the imported file and what you want to get from the original Drawing in the fields on the right.

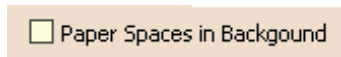
### Paper Spaces and Model Space

The Interactive Drafting workbench provides a simple method to manipulate a sheet. A sheet contains:

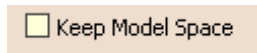
- a main view: a view which supports the geometry directly created in the sheet
- a background view: a view dedicated to frames and title blocks
- interactive or generated views.

An AutoCAD file is usually made of:

- a model space that contains the geometry,
- a paper space (or several in AutoCAD 2000).  
AutoCAD recommends that the paper space contains the title box and one or several viewports (A viewport is a window to the model space). However other configurations are possible.



Check this option to put the Paper Spaces in the Background view. The viewports will be created in the working view.



Check this option to keep the entire Model Space in its own sheet.

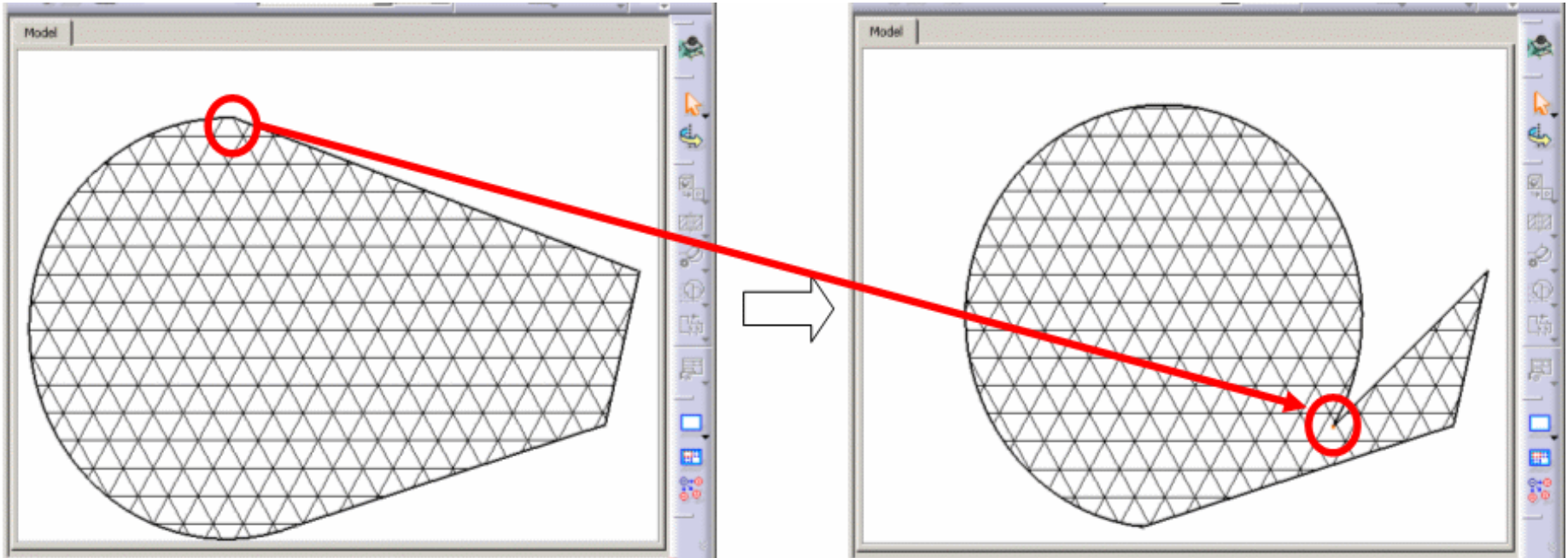


 Create end points

Create end points:  Never  For few entities  Always

It is not easy to modify and stretch geometry of imported elements the way you can do it in a V5 native elements. A solution is to create end points when needed, but to the detriment of performances. **Create end points** offers you three options to fit your needs:

- **Never:** it is the default option. It ensures the best performances.
- **For few entities:** creates end points only for hatch boundaries and mixed polylines. This is an intermediate choice between performances and edition capabilities.
- **Always:** creates end points for arcs, ellipses, lines, mlines, leaders and not standard polylines and splines. Use this option only when edition capabilities are required.



Information on the option selected is given in the report file:

```

~~~~~OPTIONS~~~~~
Unit of the file           : Millimeter (mm)
Paper Spaces in Background : Disabled
Keep Model Space          : Disabled
Create end points         : Never
Convert dimensions as     : Real Dimensions
Color to thickness        : Disabled
    
```

## Convert dimensions as

Convert dimensions as:  Dimensions  Details  Geometry

Check the required option:

- **Dimensions:**

This option preserves the semantic of AutoCAD dimensions in V5 as best as possible.

- In most cases, the whole semantic of the dimension is kept.  
This means the position, the layout and the text are preserved.  
The position, color, thickness, text caption (symbol, value, tolerance, font, color) can be edited.
- In some cases, text with redundant information  
(e.g. tolerance present both in the dimension field and in the dimension text) or dual dimension,  
or other unrecognized information will be dealt as a text with an associative link with the dimension.  
This dimension has a "fake value" that is blanked.  
To display the true dimension value, delete the associated text and enter the data in the properties of the dimension.



Limitation for the **Dimensions** option:

- Angular dimensions, problems may occur while positioning texts,
- Following attributes are not yet supported :
  - suppression of one of the dimension lines (for half dimensioning),
  - offset extension or suppression of extension lines,
  - arrowhead choice (the V5 default arrowhead is used),
  - automatic suppression of arrowheads if space is not sufficient.

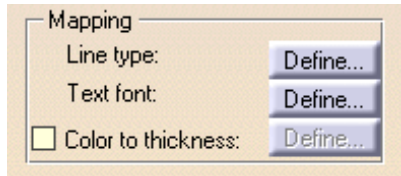
- **Geometry:**

Check this option to keep the graphical aspect. Geometry is exploded into multiple lines, arcs, texts. (this mode increase performance when loading a model).

- **Details:** dimensions are turned into details.  
This is an halfway notion between the previous two, in which the geometry is preserved and the dimension is easy to handle (it can be all selected at once).

See also the FAQ

## Mappings



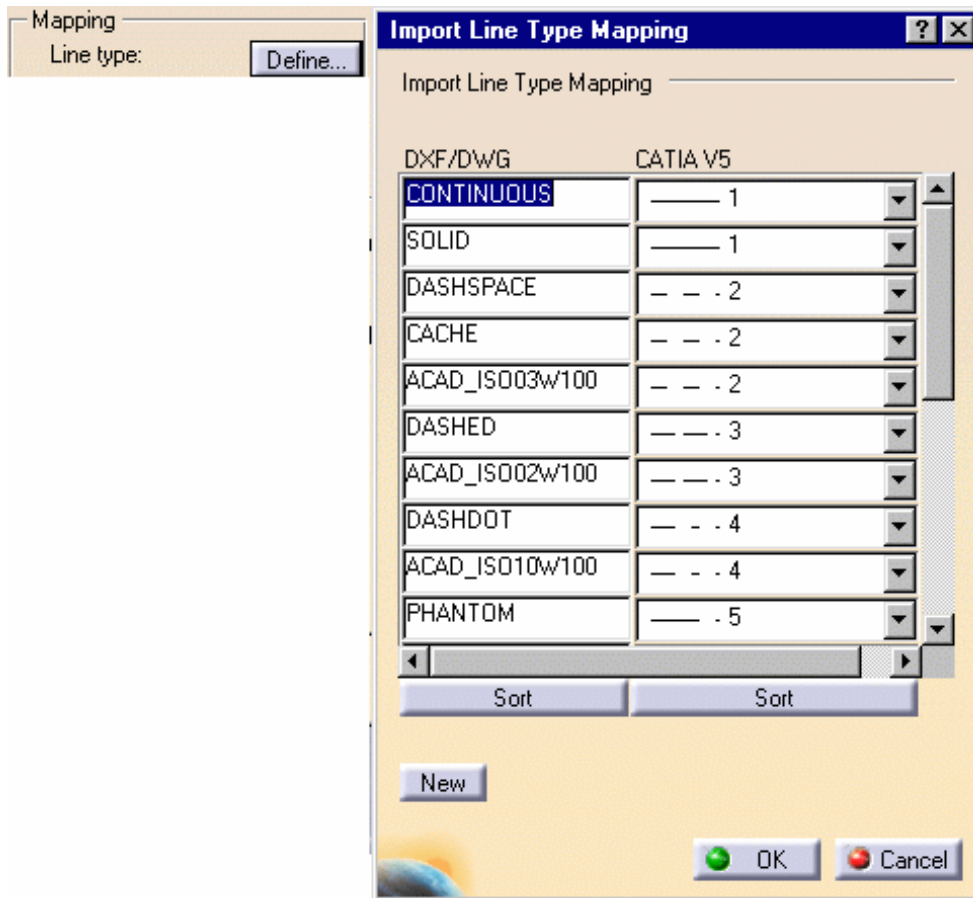
Several mapping options are available from the **Tools->Options->General->Compatibility->DXF**.

The lists of attributes are not the same in V5 and AutoCAD. Mapping options are used to come as close as possible to the AutoCAD attributes, or to switch them to V5 attributes.

### Line Type Mapping

A default mapping is available for imported lines, but you can customize the mapping of imported line types.

Push the **Line Type Define** button. The **Import Line Type Mapping** dialog box is displayed.



- A default mapping is proposed for each DXF/DWG line type.
- Use the **Sort** button of the DXF/DWG column to list the DXF/DWG line type names in alphabetical order,
- Use the **Sort** button of the CATIA V5 column to list the V5 line type by number.
- To modify the mapping for a given DXF/DWG line type, select the name of this line type in its text field and select a new V5 line type from the corresponding combo box.
- To add a new DXF/DWG line type, push the **New** button. A line is added at the bottom of the list. The name proposed for the new line type is **New\_x**. A default mapping is proposed for this new line type. Proceed as above to modify this mapping.
- To change the name of an existing line type, or of a newly created line type, select the text of the line type and enter the new name.
- To remove a line type from the list, select the name of the line type, erase it in the text field and click outside the text field.
- For better performances, do not hesitate to remove useless types from the mapping list.

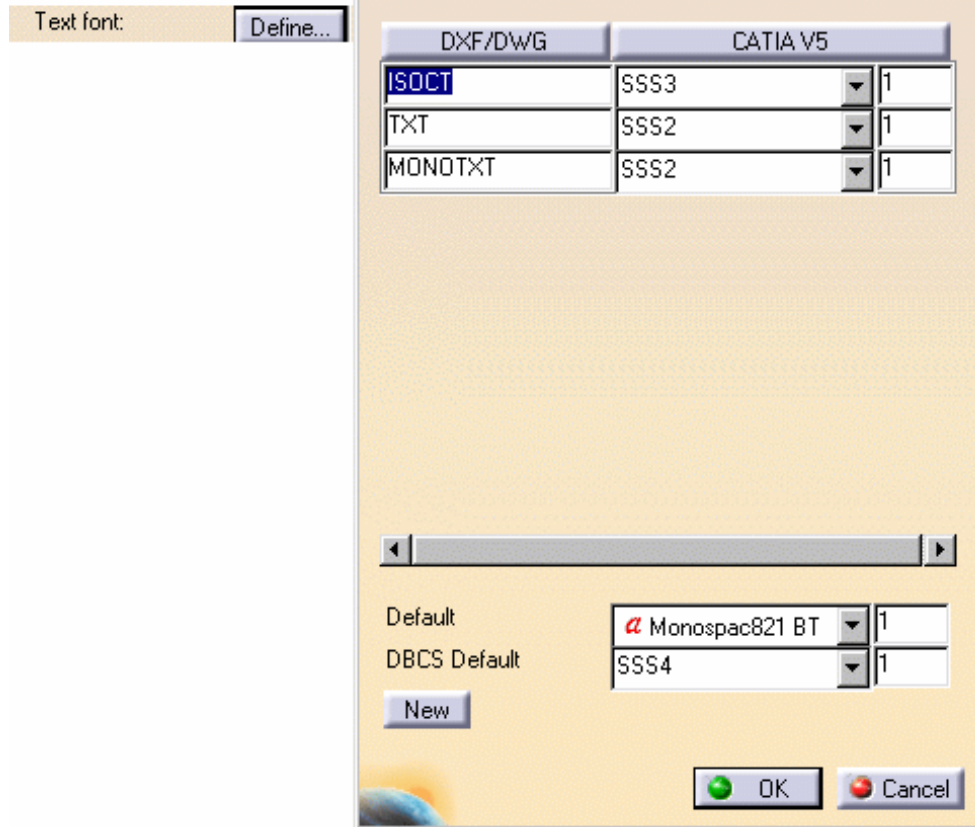
## Text Font Mapping:

By default, a mapping is done automatically between the AutoCAD font and V5.

Push the **Text Font** button to customize this mapping to:

- map any DXF font with a V5 font,
- associate a X scale factor to reframe a font with a different geometry (characters too wide or too narrow) to achieve the best possible alignment ,
- define a default font to be used when there is no mapping for a DXF font,
- define a default KANJI font, other than SSS4, to be used when the BigFont DXF font is not mapped.

The **Import Text Font Mapping** dialog box is displayed:



Under the **DXF/DWG** button, you find the DXF font to define,  
Under the **CATIA V5** button, you find combo boxes listing the V5 fonts available, and the scale factors.

You can:

- edit each column: enter the name of the DXF/DWG font, choose a V5 font from the combo box, and enter a scale factor,
- add a new mapping: push the **New** button, and repeat the step above,
- delete a mapping: erase the name of the DXF/DWG font in its field,
- sort the fonts: push the **DXF/DWG** or the **CATIA V5** button,
- define two default fonts with their scale factor at the bottom of the column.

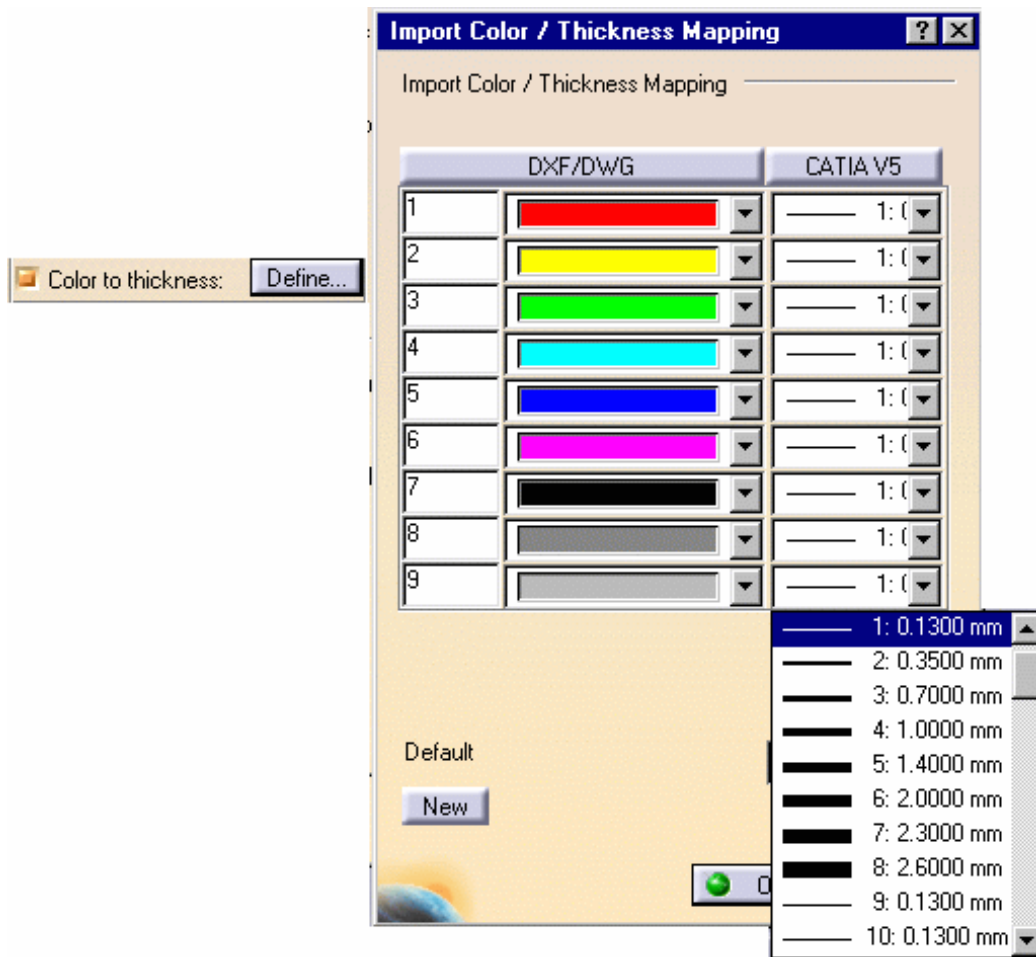
If you reset the DXF tab, the list of fonts is re-initialized with the most often mapped fonts.

### Color/Thickness Mapping:

- By default (button not checked)
- When importing an Autocad2000 DXF or DWG file (this format is automatically recognized) the "weight" property of each entity is mapped to the nearest available V5 line weight. The current V5 line weights are visible and customizable through the menu **Tools->Options->General->Display->Thickness**.
- Previous versions (up to AutoCAD 14) did not offer weight attributes for the geometry. Therefore, all the geometry is created with the default weight attribute.

- If you

1. Push the **Color to thickness Define** button. The **Import Color/Thickness Mapping** dialog box is displayed.



In the mapping dialog box, you can select DXF colors by their number (the combo of colors is automatically updated) or by selecting it in the combo of colors (the

editor of color numbers is automatically updated) and finally select the corresponding V5 thickness.

A default thickness is requested for not defined colors.

You can also:

- add mappings with the **New** button or
- delete mapping by erasing the color number in its field and clicking outside the text field and
- sort the table by colors or by thickness with the DXF/DWF or CATIA V5 push buttons.

## Export

### Exported Sheets

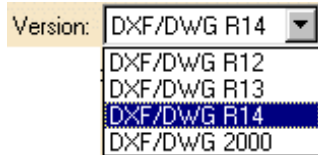
Exported sheets:  All  Only current

1. Check the required option to export either all sheets or only the current sheet of a multi-sheet drawing.

- The option "**Only current**" exports the data to a file with the name entered in the **Save as** dialog box.
- The option "**All**" exports the data to several files.

The name of each file is made of the name entered in the **Save as** dialog box and the name of the sheet (Drawing1\_sheet\_1.dxf, Drawing\_sheet\_2.dxf, ...).

### Version



1. Select the required export file format from the combo box.



Version 5 supports DXF/DWG formats version 12, 13, 14 and Autocad2000.

## Export mode

Export mode:  Semantic  Structured  Graphic

This option offers the choice between three export modes:


- **Graphic:**  
This was the only available mode up to Release 9. It is the default mode.  
It is quick and reliable. It is useful if you want to export a CATDrawing to AutoCAD and print it without modifying it.
- **Structured:**  
This mode is introduced in Release 10. The exported file can be modified. For example, in the case of a dimension exported in Structured mode, all the graphic entities representing the dimension are exported to an Insert/Block.  
The more complex Drafting elements are exported the same way whereas the basic geometric elements (lines, arcs, etc) are exported as isolated elements.
- **Semantic:**  
This mode is introduced in Release 11 and is similar to the Structured mode, with the advantage that linear, angular and circular dimensions are exported as true dimensions (with a default dimension style). "Default dimension style" entails that most graphic attributes (such as color, display format of the dimension value, type of arrow, space...) of the dimensions are lost.

For more information, see the **What About the Elements You Export?** chapter.





# Customizing Compatibility Options

 Before you start your first working session, you can customize the way you work to suit your habits.

This type of customization is stored in permanent setting files: these settings will not be lost if you end your session.



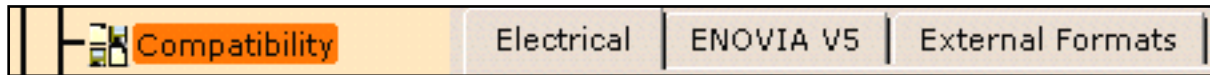
1. Select the **Tools** -> **Options** command.

The Options dialog box displays.

2. Choose the **General** category in the left-hand box.

3. Click the **Compatibility** options.

The Electrical tab displays.



This tab lets you define the [migration option](#).

4. Change this option according to your needs.
5. Click **OK** when done.



# Electrical Migration Options

Click [here](#) to see the parent page.

## Migration Options

Bundle Segment Migration Mode

Bundle segments

Multi-branchable bundle segments

Migrate V4 BNSs to V5 bundle segments.  
Each V5 bundle segment will have its own CATPart document.  
All these bundle segments will be instantiated in the geometrical bundle CATProduct document.

### Bundle segments

If this option is checked, it means that each V4 BNS is migrated to a V5 bundle segment. Each V5 bundle segment is stored in a CATPart document. They are instantiated in the geometrical bundle, a CATProduct document.

 It is the default value.

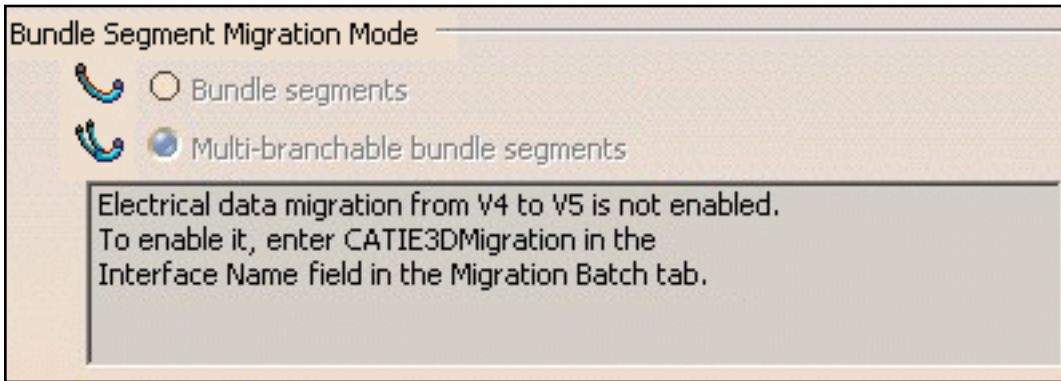
### Multi-branchable bundle segments

If this option is checked, it means that all the BNSs of a V4 GBN are migrated to one multi-branchable bundle segment in a V5 geometrical bundle. All the V5 bundle segments are stored in a single CATPart document, the multi-branchable bundle segment CATPart.

In case there are several V4 GBNs to be migrated, as many V5 geometrical bundles (CATProduct documents) are created, with one multi-branchable bundle segment (CATPart document) containing all the bundle segments.

## Migration Options Disabled





The electrical migration options can be disabled if the **Interface Name** parameter is not filled up with the **CATIE3DMigration** value in the **Migration Batch** tab.

Refer to [Migration Interface](#).

# ENOVIA LCA



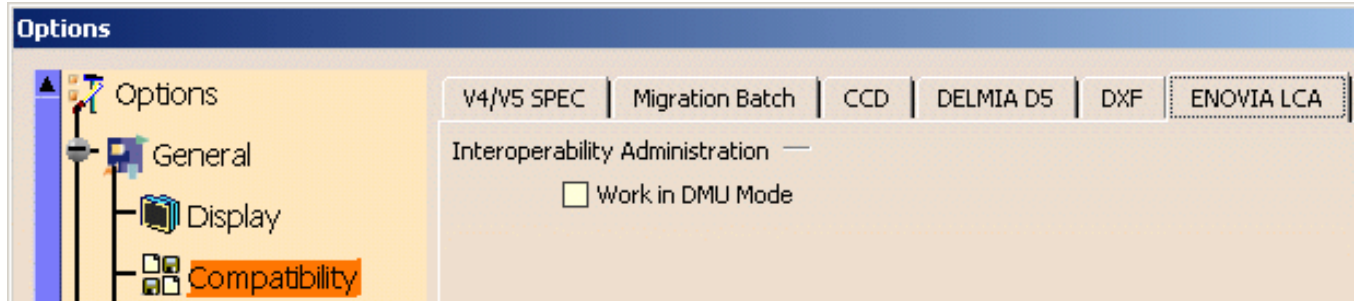
This task shows you how to customize Compatibility settings.



1. Select the **Tools->Options...** command.

The Options dialog box appears with the category tree in the left-hand column.

2. In the **General** category of the Options Tree, select **Compatibility** and click the **ENOVIA LCA** tab.



This option is useful in a CATIA/VPM interoperability context.

In CATIA V5, you can select the **DMU Mode** (Digital Mock-Up) in order to transfer a lighter version of VPM data into CATIA V5. If this check-box is selected, only the visible data will be sent to CATIA V5; the transfer is more rapid because the constraints are not downloaded during the interoperability process.



# Customizing External Formats Import

This task explains how to customize the import settings.

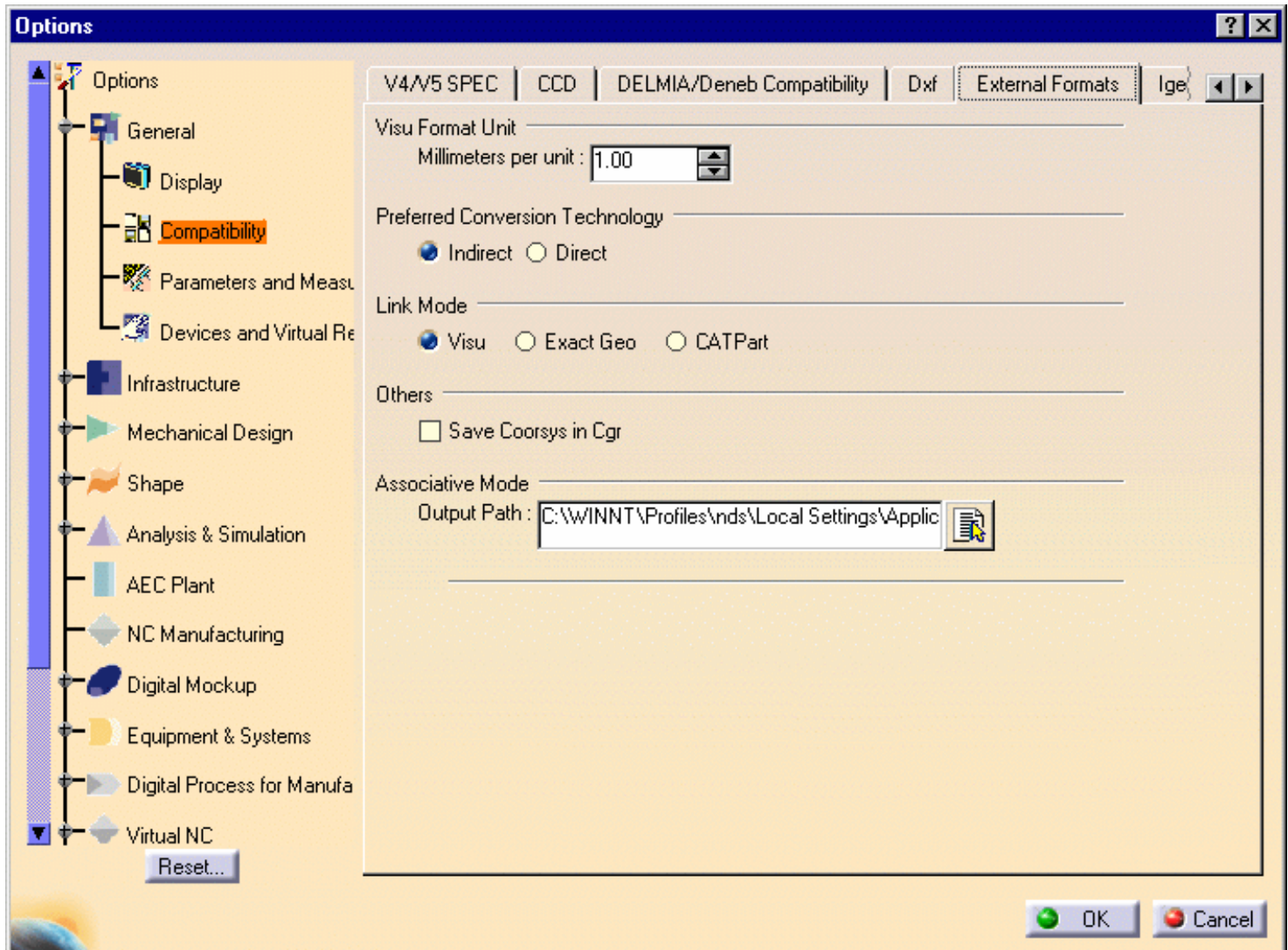


You imported a CAD Part through the **Insert ->Existing Component** command.



1. Select the **Tools->Options...** command. The Options dialog box appears.
2. Expand the **General** category in the left-hand tree and select the **Compatibility** category.
3. Click the **External Formats** tab.

If you have the SolidWorks, SolidEdge or Acis Plug-In license, the following settings will be available:



4. Modify the settings as desired.

- **Visu Format Unit :**

This setting allows you to specify the number of millimeters per unit of the model. For instance, if you have a file in inch, you need to enter 25.4 in this field.

This option is taken into account when you have the following formats: SLP, STL, OBJ, BYU, IFF and \_PS.

Note: For a file of type \_PS, the **Visu Format Unit** must be set to 10.

- **Preferred Conversion Technology**

- **Indirect :** use of CAD license (available for DELMIA product line only).
- **Direct :** no use of CAD license.  
Support of ProE20 only.  
Support of UG data up to V18.  
For Solidworks, SolidEdge, Parasolid, ACIS, DXF 3D, Inventor and VDA-FS : use the direct mode on Windows. (Note: These formats are supported on Windows only.)

- **Link Mode**

- **Visu :** tessellated data
- **Exact Geo :** recovery of CAD's exact geometry to generate V5 geometry (NCGM). After selecting this option, this message appears: "Please, restart session to take modifications into account".
- **CATPart :** link with a V5 CATPart. **Do not modify** this CATPart in CATIA V5.

- **Others**

- **Save Coorsys in cgr :** store part coordinate systems in .cgr file. It is only supported in the indirect mode. This parameter determines whether coordinate systems present in imported ProE, UG or IDEAS parts will be imported into the resulting CGR files.

- **Associative Mode**

- **Output Path :** you can customize the Output File for generated data. It specifies the location where CGR, NCGM and CATParts will be generated.

If you have the IDEAS, ProEngineer or Unigraphics Plug-In licenses, additional options will appear in the External Formats tab.

With the **IDEAS** Plug-In license, you obtain:



- **IDEAS** (indirect mode only)

- **Tessellation Parameter:** parameter of IDEAS chordal deviation. The Tessellation parameter holds the value tessellating the surfaces contained by the I-DEAS parts.
- **Project Name:** This parameter is only used when converting data through the MS8 and MS9 based converter. This parameter must hold the name of the project under which the model file is being converted, and is registered, in the I-DEAS Data Management system.
- **Library Name:** pkg import (IDEAS package file) is the name of the library to transfer.
- **Part or Assembly Name:** you can select any kind of data you want to read from the pkg file Part or Assembly and specify its name.
- **Part Number:** this setting is optional, you can specify the part number you want to transfer.
- **Revision Number:** this setting is optional, you can specify the revision number you want to transfer.

With the **ProEngineer** Plug-In license:



• **ProEngineer**

- **Quilts Read:** enables to activate the transfer of quilts surfaces inside CGR. This setting does not apply to NCGM or CATPart conversions.
- **Simplified Representation:** only part simplified representation is supported. To set the name of the part simplified representation to use before the Assembly conversion, check this box and enter the name.
- **Instance Name:** When you want to import a specific instance of a generic assembly/part, check this button and specify the name of the instance of your choice.

NOTE: For Simplified Representation and Instance Name, only upper case names are supported with the direct converter.

And with the **Unigraphics** Plug-In license:



• **Unigraphics** (indirect mode only)

- **Layers:** string indicating the layers that need to be transferred during conversion. The syntax is:  
 \* : all layers are transferred by default  
 5, 8, 27 : only layers 5, 8, 27 are transferred.  
 8-32 : all layers from 8 up to 32 (including 8 and 32) are transferred.  
 2, 5, 8-32 : layers 2, 5 and all layers from 8 up to 32 (including 8 and 32) are transferred.
- **Convert active layers only:** if this option is activated, it enables to transfer only active layers or to transfer all types of layers (active and inactive). It can be used together with the previous Layers setting, in that case only layers that are compliant with those two settings will be transferred.
- **Convert open surfaces:** when this option is activated, it enables to transfer UG Open Surfaces during the conversion.
- **Drawing Name:** this setting is used in Batch mode only, while creating a CATDrawing. In this box you can specify the name of the drawing you want to transfer.

5. Click OK to confirm your operation.



# Customizing IGES 3D Settings



This task shows you how to customize IGES settings.



1. Select the **Tools->Options...** command.

2. Select the **General** category, then the **Compatibility** category, then click the **IGES** tab.

**Options**

Options

- General
- Display
- Compatibility**
- Parameters and Measure
- Devices and Virtual Reality
- Infrastructure
- Mechanical Design
- Shape
- NC Manufacturing
- Digital Mockup
- Digital Process for Manufacturing
- Development

DXF | ENOVIA LCA | External Formats | **IGES** | IGES 2D | SmartBOM | SmarTeam Scripts | STEP

Export : Curve and surface type \_\_\_\_\_  
 Standard  BSpline

Export : Representation mode \_\_\_\_\_  
 Surface  Wireframe

Export : Show/NoShow \_\_\_\_\_  
 Save only shown entities

Export : Author's Name and Organization \_\_\_\_\_  
 Author's Name :   
 Author's Organization :

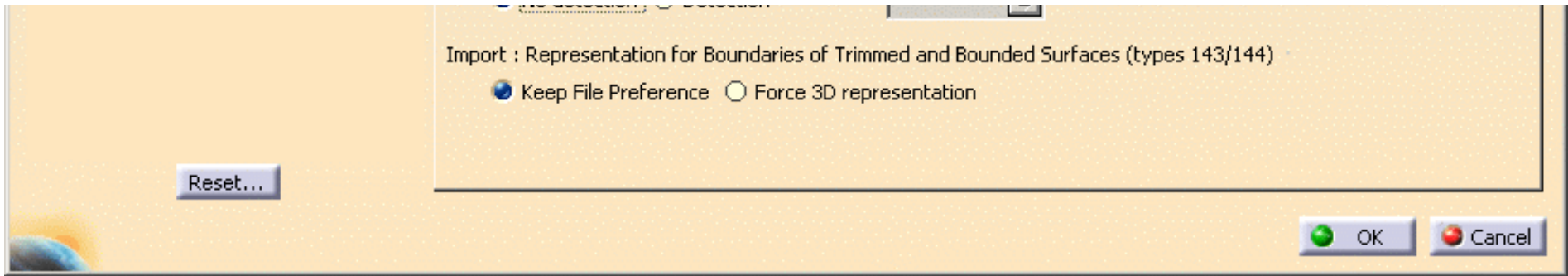
Import : Join \_\_\_\_\_  
 Join surfaces of the model   
 Join surfaces of each Group

Import : Continuity Optimization of Curves and Surfaces \_\_\_\_\_  
 No Optimization  
 Automatic Optimization  
 Advanced Optimization

Import : Detection of Invalidity in Input Geometry \_\_\_\_\_  
 No detection  Detection Tolerance :

Import : Representation for Boundaries of Trimmed and Bounded Surfaces (types 143/144) \_\_\_\_\_  
 Keep File Representation  Same as Representation





Refer to the *Version 5 Data Exchange Interfaces User's Guide - Customizing IGES 3D Settings*.

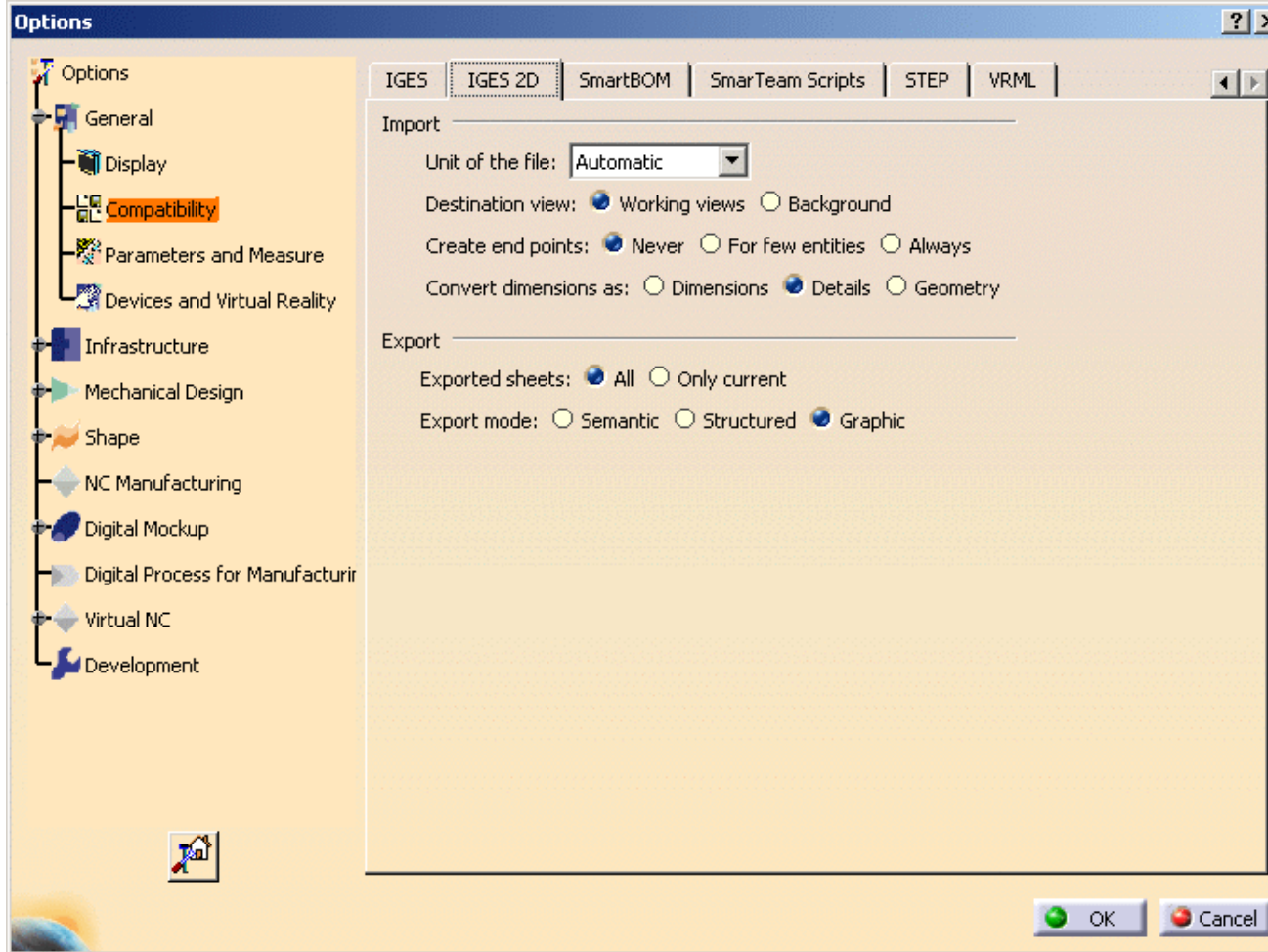




As you import an IGES 2D file, you may customize the units and the dimensions format



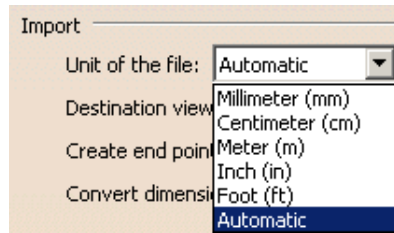
1. Select **Tools -> Options -> ...**
2. Click **General-> Compatibility** in the list of objects to the left of the **Options** dialog box.
3. Click the IGES 2D tab.



## Import



## Unit of the file



By default, the option is set to **Automatic** and the unit of import is determined automatically (either millimeter or inch) for the best possible resulting drawing. However, in some cases the resulting drawing is not satisfactory and requires another unit. Select this unit in the list. Then restart the import.

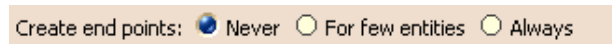
## Destination view



Check the view in which you want to import the file.

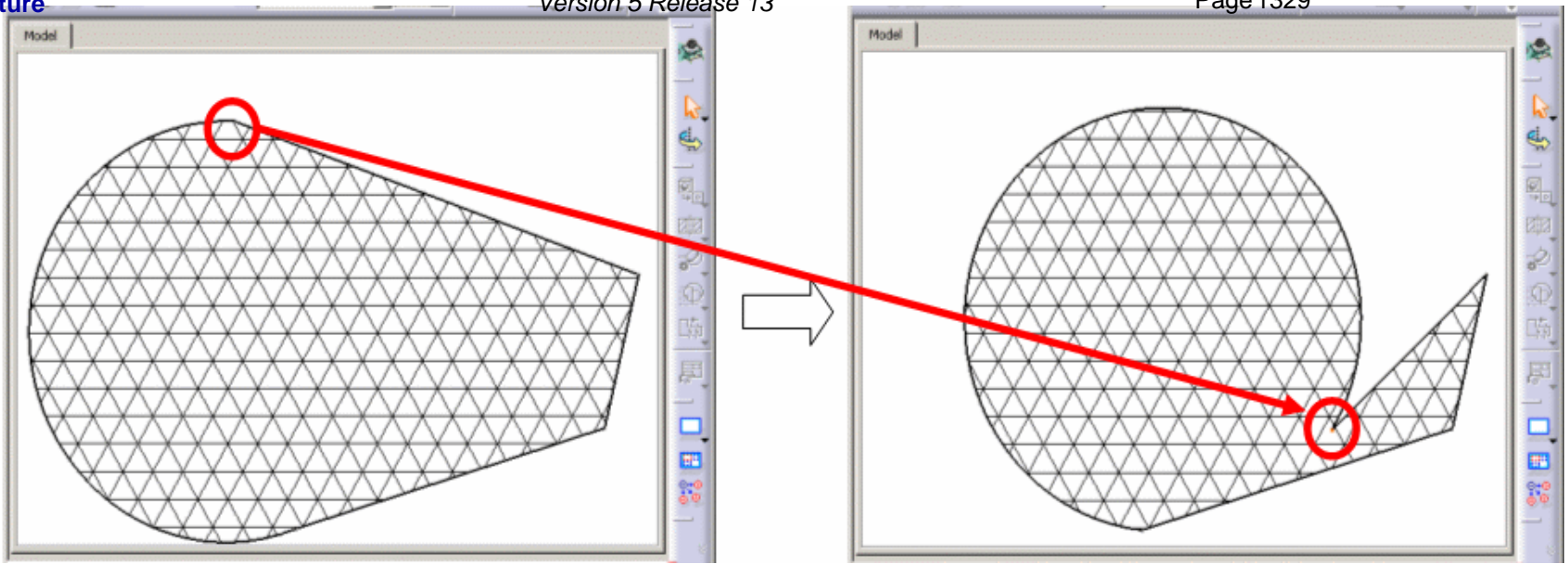


## Create end points



It is not easy to modify and stretch geometry of imported elements the way you can do it in a V5 native elements. A solution is to create end points when needed, but to the detriment of performances. **Create end points** offers you three options to fit your needs:

- **Never:** it is the default option. It ensures the best performances.
- **For few entities:** creates end points only for hatch boundaries and mixed polylines. This is an intermediate choice between performances and edition capabilities.
- **Always:** creates end points for arcs, ellipses, lines, mlines, leaders and not standard polylines and splines. Use this option only when edition capabilities are required.



Information on the option selected is given in the report file:

```

~~~~~OPTIONSS~~~~~
Unit of the file      : Automatic
Destination view     : Working View
Create end points    : Never
Convert dimensions as : Real Dimensions

```

### Convert dimensions as

Convert dimensions as:  Dimensions  Details  Geometry

If you check:

- Dimensions:** linear, angular and circular dimensions will be preserved, others (ordinate types) will be transformed into details.  
 This option keeps the semantic of the dimension. This means the position, the layout and the text are preserved. The position, color, thickness can be edited. The text of the dimension is a text that has an associative link with the dimension.  
 This dimension has a "fake value" that is blanked.  
 To display the true dimension value, delete the associated text and enter the data in the properties of the dimension.



Limitation for the **Dimensions** option:

- Linear dimensions: rotated dimensions, i.e. linear dimensions with a fixed angle other than horizontal or vertical are turned into details,
- Radial dimensions: problems may occur while positioning arrows,
- Angular dimensions, problems may occur while positioning texts,
- Following attributes are not yet supported :
  - suppression of one of the dimension lines (for half dimensioning),
  - offset extension or suppression of extension lines,
  - arrowhead choice (the V5 default arrowhead is used).
- **Geometry**: geometry is exploded into multiple lines, arcs, texts.  
Check this option to keep the graphical aspect (this mode increase performance when loading a model).
- **Details**: dimensions are turned into details. This is an halfway notion between the previous two, in which the geometry is preserved and the dimension is easy to handle (it can be all selected at once).



## Export

### Exported Sheets

Exported sheets:  All  Only current

1. Check the required option to export either all sheets or only the current sheet of a multi-sheet drawing.

- The option "**Only current**" exports the data to a file with the name entered in the **Save as** dialog box.
- The option "**All**" exports the data to several files.

The name of each file is made of the name entered in the **Save as** dialog box and the name of the sheet (Drawing1\_sheet\_1.ig2, Drawing\_sheet\_2.ig2, ...).

### Export mode

Export mode:  Semantic  Structured  Graphic


This option offers now the choice between three export modes:


- **Graphic**:  
This was the only available mode up to Release 11. It is the default mode.  
It is quick and reliable. It is useful if you want to export a CATDrawing and print it without modifying it.
- **Structured**:  
This mode is introduced in Release 11. The exported file can be modified. For example, in the case of a dimension exported in Structured mode, all the graphic entities representing the dimension are exported to an Insert/Block. The more complex Drafting elements are exported the same way whereas the basic geometric elements (lines, arcs, etc) are exported as isolated elements.
- **Semantic**:


This mode is introduced in Release 11 and is similar to the Structured mode, with the advantage that linear dimensions are exported as true dimensions (with a default dimension style). "Default dimension style" entails that most graphic attributes (such as color, display format of the dimension value, type of arrow, space...) of the dimensions are lost.

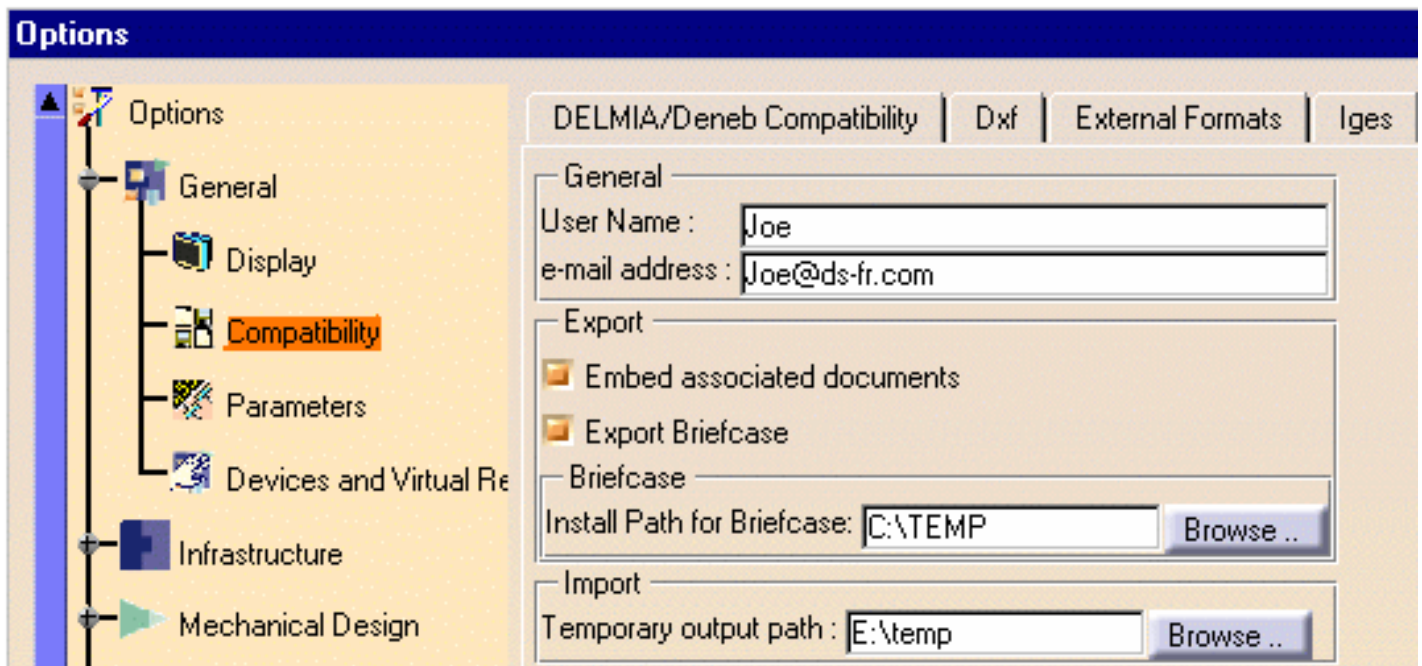
For more information, see the **What About the Elements You Export?** chapter.

# Export SmartBOM from CATIA Version 5

 This task shows you how to export SmartBOM from CATIA Version 5. In Supply Chain environment or in Project management, the Bill of Material (BOM) is one of the most precious information shared between actors. Export of BOM Briefcase enables to share such data by means of nomad application (SmartBOM Editor from SmarTeam corporate). No specific installation is required to view and edit BOM.

 Install SmartBOM Product from the SmarTeam Corporate.

-  1. In order to set the environment data, select the **Tools -> Options...** command. The option dialog box appears with the category tree in the left-hand column.
2. In the **General** category of the options tree, select **Compatibility** and click the **SmartBOM** tab. The following dialog box appears.
3. Insert the User Name and e-mail address of you choice.

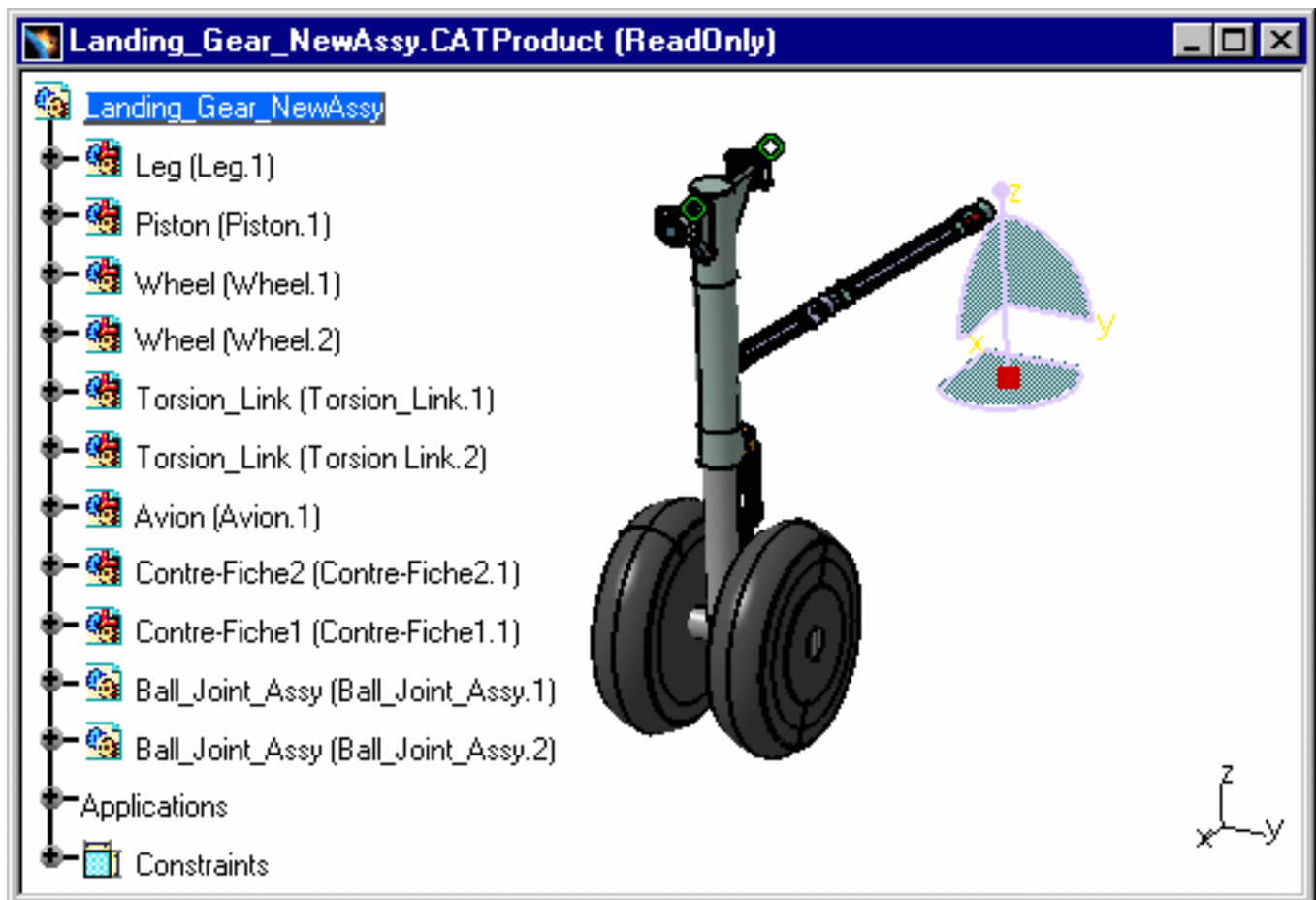


## For the **Export** operation,

4. Check the **Embed associated documents** box to attach CATIA documents (geometry and specifications) in Briefcase.
5. Check the **Export Active Briefcase** box, if you want to export BOM as a nomad application (.exe). This will enable you to browse the "Install Path". Install path is a temporary directory where the nomad application is built before being sent.

## For the **Import** operation,

6. Browse the temporary directory where imported package will be read.
7. Open a CATProduct document.



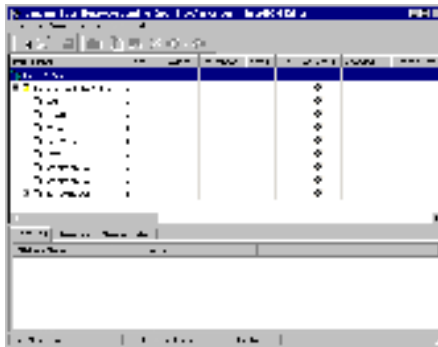
8. Save it as stbom file (or .exe file): select stbom or .exe as file format in the File/Save As dialog box.



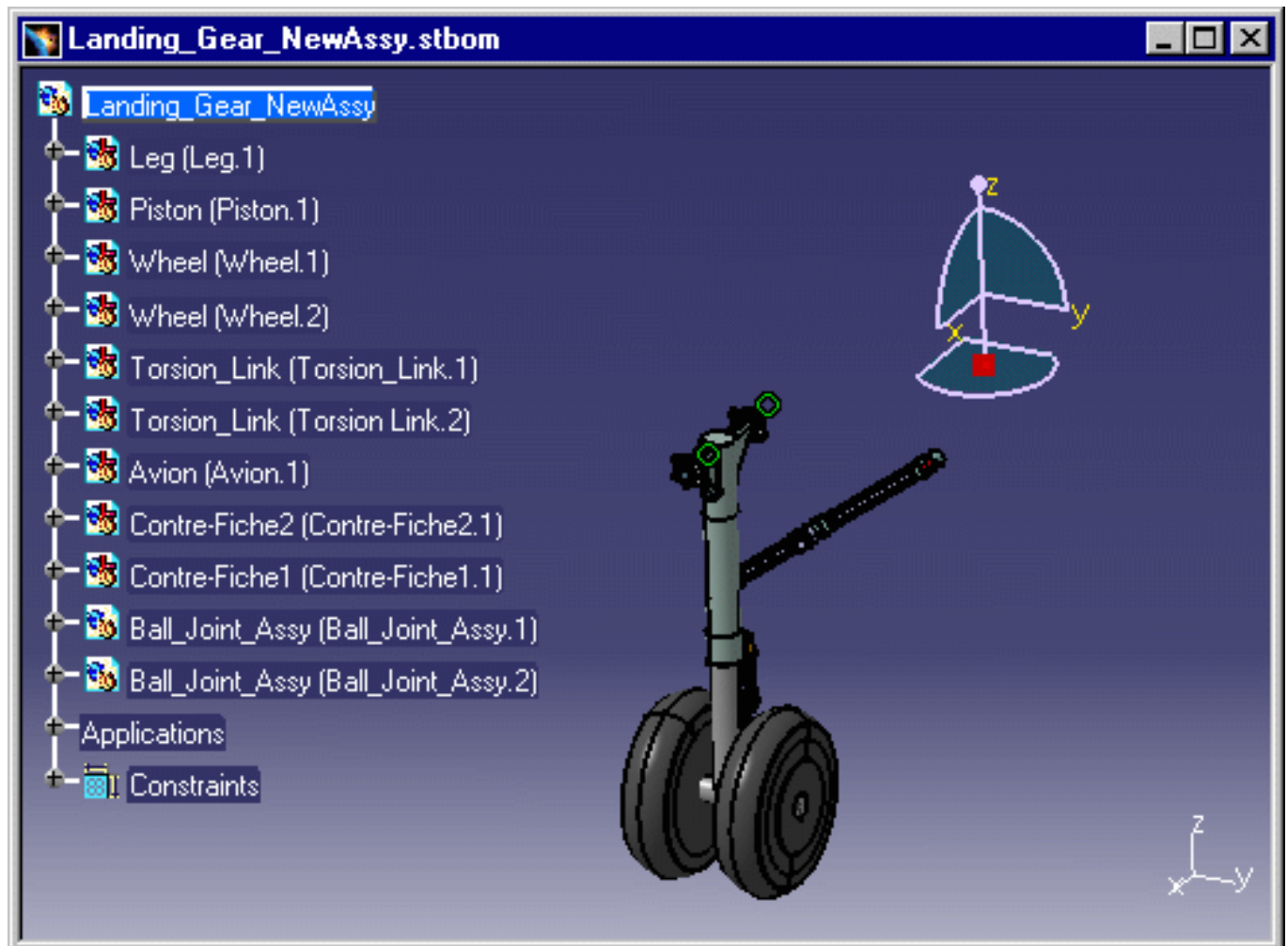
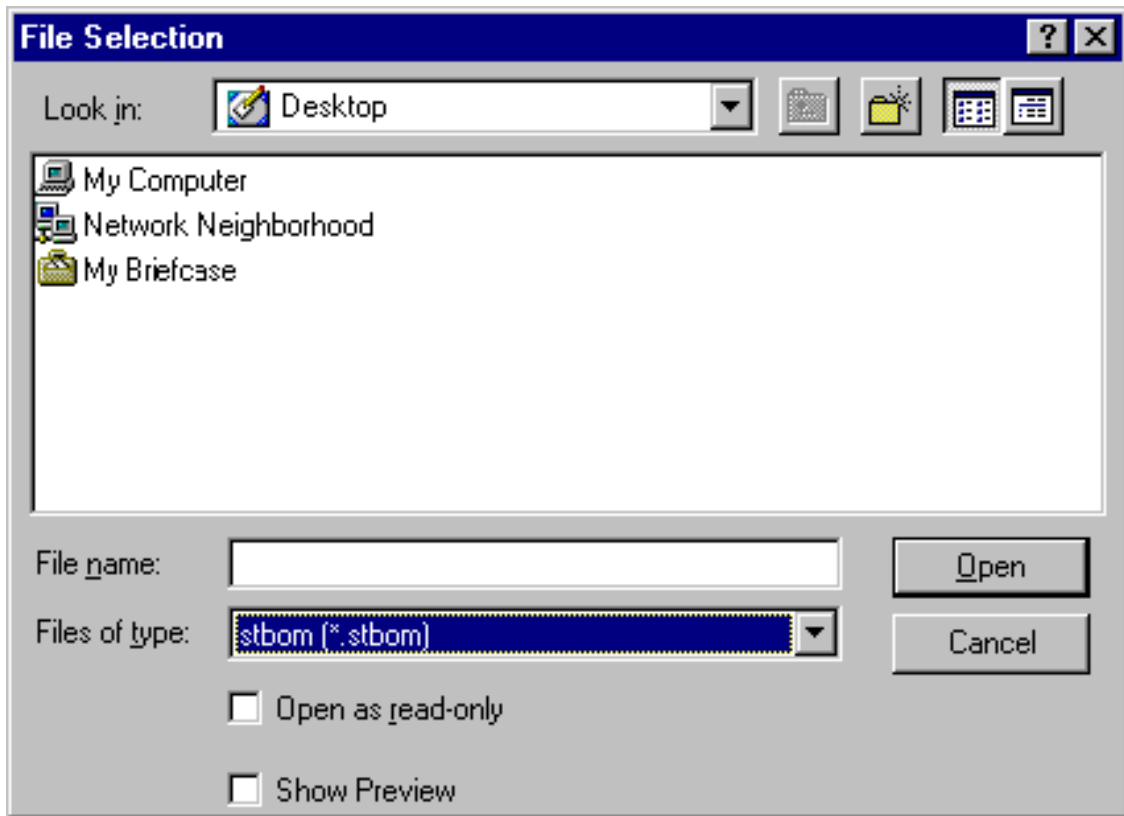


This will generate the SmartBOM Briefcase (for instance: Landing\_Gear\_NewAssy.stbom) or SmartBOM Active Briefcase with your data (.exe file). You can then send this data by e-mail or publish it on your Web site.

9. Edit the stbom file in SmartBOM Editor: double-click the stbom file (or .exe file) to edit the BOM by using SmartBOM Product:



10. Save your changes by using SmartBOM Editor (File->Save as).
11. To **import** a new SmartBOM Briefcase in CATIA V5, select **File->Open** in CATIA V5 and choose the stbom format: the new BOM is imported with the attached CATIA documents.



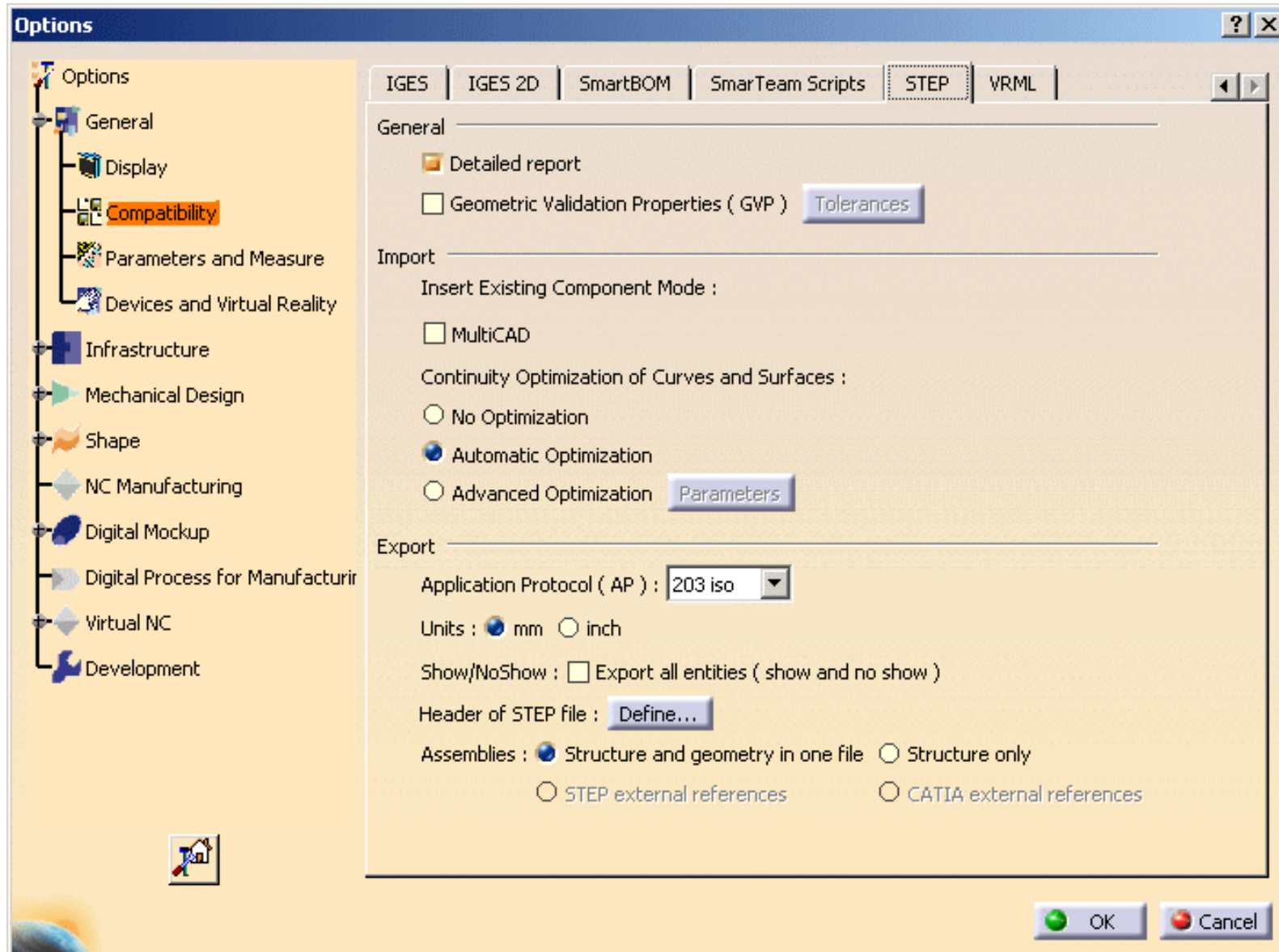
# STEP Settings



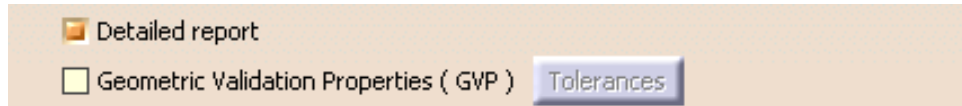
This task shows you how to customize STEP settings



1. Select the **Tools->Options...** command.
2. Select the **General** category, then the **Compatibility** category, then click the **STEP** tab.



## General



## Detailed Report

By default, the report file contains a Detailed Conversion chapter. Uncheck the **Detailed Report** option to remove this chapter from the report file.

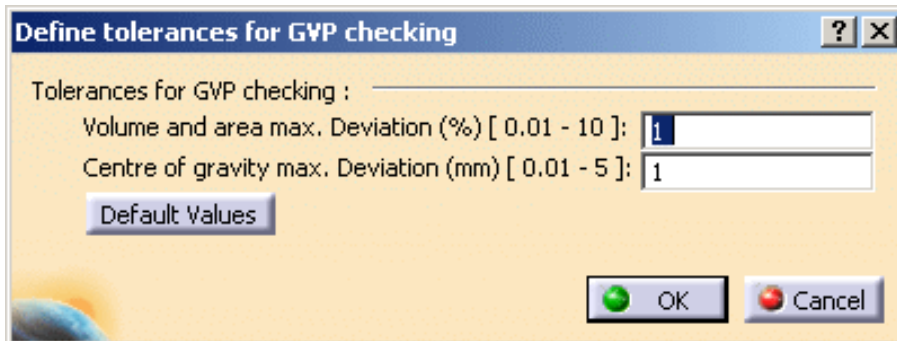
## Geometric Validation Properties



This functionality is available in STEP AP214 only.



1. Check the **Geometric Validation Properties** option (By default, they are not processed ).
2. The **Tolerances** button becomes available. It opens a dialog box where you can define the tolerances for the GVP checking.
  - The first tolerance is the percentage of variation of volume or area allowed. Default value: 1%.
  - The second tolerance is the maximum error for the center of gravity, expressed in mm. Default value: 1mm.
  - You can retrieve the default values with the **Default Values** button.



## For import:

- Geometric validation properties are computed for each solid, shell, product or instance, and this information is written in the report file.

- For each solid, shell, product or instance, the report file gives the computed geometric validation properties:
  - Centroid: coordinates of the center of gravity (applies to solid, shell, product or instance),
  - Area: area of the entity (wetted area for solids) (applies to solid, shell or product),
  - Volume: volume of the entity (for solids only) (applies to solid or product).
- If the imported STEP file contains geometric validation properties, these properties are read. This information is written in the report file.
- For each read geometric validation properties, the report file gives the status of comparison between read and computed properties, with the following information:
  - Centroid deviation error (distance measure) (applies to solid, shell, product or instance),
  - Surface area difference value and error ratio (applies to solid, shell or product),
  - Volume difference value and error ratio (applies to solid or product).
- A global status for the conversion is given, together with the maximum deviations found.

Example of report file:

~~~~~DETAILED CONVERSION~~~~~

```
#33 Manifold Brep Type: MANIFOLD_SOLID_BREP Transferred correctly
  Computed Properties : Area:70027.4 mm2 - Volume:530575 mm3 - Centroid:(-50 mm,-10 mm,0 mm)
  Read Properties     : Area:70027.4 mm2 - Volume:530575 mm3 - Centroid:(-50 mm,-10 mm,0 mm)
  Validation successful : Error Ratio: Area:0% - Volume:0% - Centroid Position Error:0 mm
#1080 Manifold Brep Type: MANIFOLD_SOLID_BREP Transferred correctly
  Computed Properties : Area:24628.3 mm2 - Volume:96858.4 mm3 - Centroid:(0 mm,20.2027 mm,14.5946 mm)
  Read Properties     : Area:24628.3 mm2 - Volume:96858.4 mm3 - Centroid:(0 mm,20.2027 mm,14.5946 mm)
  Validation successful : Error Ratio: Area:0% - Volume:0% - Centroid Position Error:0 mm
```

=====

Check validation properties of products in the assembly

```
AS1-SIMPLIFIED :
  Computed Properties : Area:119284 mm2 - Volume:724292 mm3 - Centroid:(90 mm,18.0779 mm,-75 mm)
  Read Properties     : Area:119284 mm2 - Volume:724292 mm3 - Centroid:(90 mm,18.0779 mm,-75 mm)
  Validation successful : Error Ratio: Area:0% - Volume:0% - Centroid Position Error:0 mm
```

```
L-BRACKET_ASM :
  Computed Properties : Area:24628.3 mm2 - Volume:96858.4 mm3 - Centroid:(50 mm,20.2027 mm,-35.4054 mm)
  Read Properties     : Area:24628.3 mm2 - Volume:96858.4 mm3 - Centroid:(50 mm,20.2027 mm,-35.4054 mm)
  Validation successful : Error Ratio: Area:0% - Volume:0% - Centroid Position Error:0 mm
```

=====

Check position of instances of products in the assembly

```
0 : Validation successful : Centroid Position Error:0 mm
1 : Validation successful : Centroid Position Error:0 mm
7 : Validation successful : Centroid Position Error:0 mm
8 : Validation successful : Centroid Position Error:0 mm
```

=====

Geometric validation properties check: OK

Maximum errors:

```
Centroid : 7 : 4.69362e-007 mm Volume : #1080 : 2.13343e-006 % Area : #1080 : 5.87308e-006 %
```

=====



- The status of comparison for a given solid, shell, product or instance is ok if:
  - ratios (Volume difference and Surface area difference) are lower than 1%.
  - and the centroid deviation is lower than 1 mm.
- If all status are ok, the global status is ok too.
- The maximum deviation found for each comparison (centroid, area and volume) is given in the report file with the corresponding entities (the maximum deviations found do not necessarily apply to a single object).
- This functionality involves a slight performance loss, due to the properties computation cost.

### For export:

- The exported STEP file includes geometric validation properties for each solid, shell, product or instance, according to STEP AP214 and the CAX-IF recommended practices.
- For each solid, shell, product or instance, the report file gives the computed geometric validation properties:
  - Centroid: coordinates of the center of gravity (applies to solid, shell, product or instance),
  - Area: area of the entity (wetted area for solids) (applies to solid, shell or product),
  - Volume: volume of the entity (for solids only) (applies to solid or product).



- The unit used for geometric validation properties is the STEP length user unit. See [Unit option](#).
- If the option [Assemblies/Structure only](#) is checked, the Geometric Validation Properties are not available.
- This functionality involves a slight performance loss, due to the properties computation cost.

## Import



### Insert Existing Component Mode

Insert Existing Component Mode :

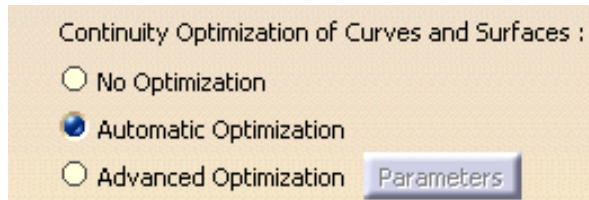
MultiCAD

This option appears only if both the *V5 - STEP AP203 Interface* and the *V5 - STEP AP214 Interface* and the *MULTICAx STEP Plug-in* exist on the machine.

If you check MultiCAD, you activate the MultiCAD mode.

## Continuity Optimization of Curves and Surfaces

This setting allows a better user control over the number of curves and surfaces that are created during the process of importing STEP data into V5:



V5 requires its geometry to be C2-continuous. When non C2-continuous geometry must be imported from a STEP file, this geometry (curves, surfaces) is broken down into a set of contiguous geometries, each of them being C2-continuous. This is what happens when the No Optimization option is chosen.

However, this can produce an increase of the size of the resulting data, because more curves/surfaces are created. In order to limit this drawback, two other modes are optionally offered.

In those modes, the STEP interface tries to limit the splitting of curves and surfaces by modifying their shape slightly, so that they become C2-continuous while remaining very close to their original shape.

In order to guarantee that the deformation is not excessive, a maximum deviation parameter is used. When in Automatic Optimization mode, this maximum deviation is read into the STEP file itself, in the STEP parameter that documents the precision of points in the file. In this mode, the value read from the STEP file is then corrected so that it remains comprised between  $10E-2$  and  $10E-3$ . This guarantees an optimization that remains compatible with the precision for the data that was set by the emitting system.

Last, if this strategy is not enough, you can choose the Advanced Optimization mode, in which an arbitrary deviation value can be entered.

By default, the Automatic Optimization is proposed:

- No approximation, thus this option does not create a significant deformation and keeps the internal BSpline structure (equations and knots).
- A continuity optimization is performed within the deformation tolerance used for optimizing BSplines, comprised between 0.001mm and 0.01mm (depending on the tolerance value defined within the imported STEP file) on:
  - BSpline surfaces,
  - BSpline boundary curves (3D and P-curves when available),
  - BSpline independent 3D curves,
- The parameters box cannot be activated

This option softens the effect C2 cutting of faces and boundaries (which is mandatory in V5) without any significant geometric deformation

If you check No optimization:

- No optimization is performed on BSplines (neither curves nor surfaces).
- Elements are cut at discontinuity points to suit the modeler (exact mathematic continuity). This may result in a dramatic number of faces and boundary



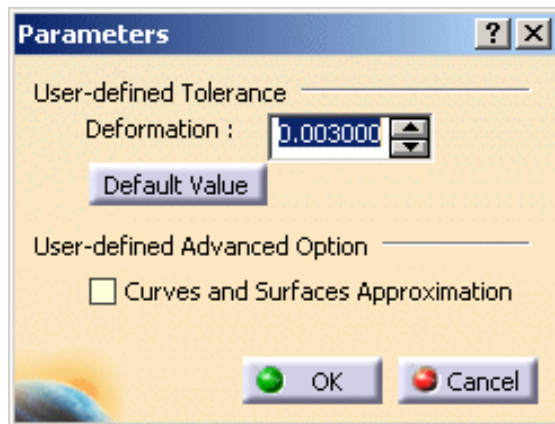
curves, data of poor quality and poor performances in further use in V5.

If you check Advanced Optimization:

- No approximation. The internal BSpline structure (equations and knots) is kept,
- A continuity optimization is performed on:
  - BSpline surfaces,
  - BSpline boundary curves (3D and P-curves when available),
  - BSpline independent 3D curves,
- but the deformation tolerance is set by the user (see [Parameters](#)).

With this option, you can enter a larger tolerance value which may enhance the optimization impact (resulting in less C2 cutting on faces).

Push the Parameters button to access advanced optimization options and tolerances.



#### User-defined Tolerance

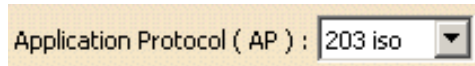
- Deformation: maximum deformation (in millimeter) allowed in the optimization of curves and surfaces. Ranges between 0.005 and 0.1 mm.
- Push the Default Values button to revert to the default value.

#### User-defined Advanced Option

- Curves and Surfaces Approximation:
  - BSpline surfaces and curves continuity is optimized,
  - In addition, Bspline curves and surfaces approximation is performed,
  - It is possible to enter a user value for Deformation,
  - This option may change the internal structure of BSplines (equations and knots),
  - This option usually results in a significant decrease in the number of faces cuttings.

## Export

### Application Protocol (AP)



#### 1. Check the required export mode:

- AP203 iso
- AP214 iso

The data contained in a CATPart or CATProduct document will be saved in STEP AP203 or AP214 formats. For more information about STEP AP203 and STEP AP214, refer to Exporting CATPart or CATProduct Data to a STEP AP203 / AP214 File.

## Units

Units :  mm  inch

1. Check the required export mode:

- mm (millimeter)
- Inch

to export a CATPart or a CATProduct in STEP format and in Inch or millimeter independently of the V5 Session unit.

## Show/NoShow

Show/NoShow :  Export all entities ( show and no show )

A CATProduct to export may contain:

- visible entities placed in the Show space,
- hidden entities placed in the NoShow space.

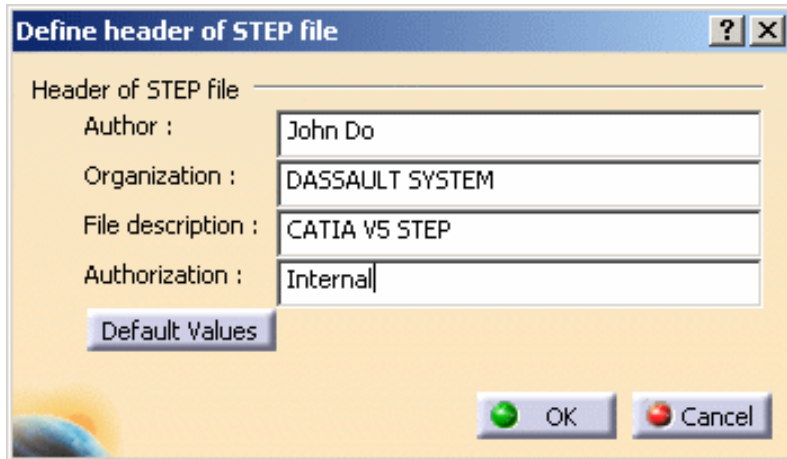
By default, i.e. when the option is not checked, only Show entities are exported.

Check the **Export all entities** option to export all entities belonging to both the "Show" and the "NoShow" spaces. Note that the "NoShow" entities are exported as if they were "Show" entities. This means that reading back a STEP file generated with the **Export all entities** option will put all the "NoShow" entities in the "Show" space.

## Header of STEP file

Header of STEP file :

1. Push the **Define...** push button. The Define header of STEP file dialog box is displayed:
2. Fill in the form as required. You can revert to the Default Values by pushing the Default Value push button.



The header of the exported file looks like this:

```
ISO-10303-21;
HEADER;
FILE_DESCRIPTION(('CATIA V5 STEP'),'2;1');

FILE_NAME('E:\\tmp\\MyFile.stp','2003-06-05T09:06:15+00:00',
|('John Do'),('DASSAULT SYSTEMES'),'CATIA Version 5 Release 11 (IN-9)',
'CATIA V5 STEP AP203','Internal');

FILE_SCHEMA(('CONFIG_CONTROL_DESIGN'));

ENDSEC;
```

## Assemblies

Assemblies :  Structure and geometry in one file  Structure only  
 STEP external references  CATIA external references

1. Check the required option to select the export mode.

- Structure and Geometry in one file: one STEP file only containing the structure and geometry of the components. This is the option by default.
- Structure only: a STEP file containing structure and entities PRODUCT\_DEFINITION\_WITH\_ASSOCIATED\_DOCUMENT which have a link with CATPart files.
- STEP external references:
  - one STEP file containing the structure
  - and a STEP file for each component  
 The STEP file names, for each component, have the same name as the components.  
 the structure and the component STEP files are generated in one shot, in the same location.
- CATIA external references: one STEP file containing the assembly structure with external links to CATPart, CATShape, model V4, .cgr, .wrl files, according to AP 214 external references mechanism.

The External References functionality is available only with AP214.

A STEP file cannot refer to a STEP assembly file.

This summary table shows you all the possible combinations within the first two frames, Export : Application Protocol and Export : Assemblies.

| <b>Frame Export AP</b>                            | <b>203</b> <sup>(1)</sup> | <b>214</b> <sup>(2)</sup> |
|---------------------------------------------------|---------------------------|---------------------------|
| <b>Frame Export Assemblies</b>                    |                           |                           |
| Structure and Geometry in one file <sup>(3)</sup> | YES                       | YES                       |
| STEP External References <sup>(4)</sup>           | NO (inactive button)      | YES                       |
| CATIA External References <sup>(4)</sup>          | NO (inactive button)      | YES                       |
| Structure only <sup>(5)</sup>                     | YES                       | YES                       |

- (1) = reference of the Application Protocol Config Control Design (AP203)
- (2) = reference of the Application Protocol Core Data for Automotive Mechanical Design Processes (AP214)
- (3) = Export of the Structure and Geometry of a CATProduct into one file only
- (4) = Export of the Structure and Geometry of a CATProduct into different files
- (5) = Export of the Structure only of a CAPProduct

If you have no STEP license, you can export a CATProduct in Structure Only mode. You can choose either AP203 or AP214, but the following buttons remain inactive:

- Structure and Geometry in one file
- External References




Dynamic Licensing:


If you release the STEP license and you access the STEP options panel, the export assemblies options may change.

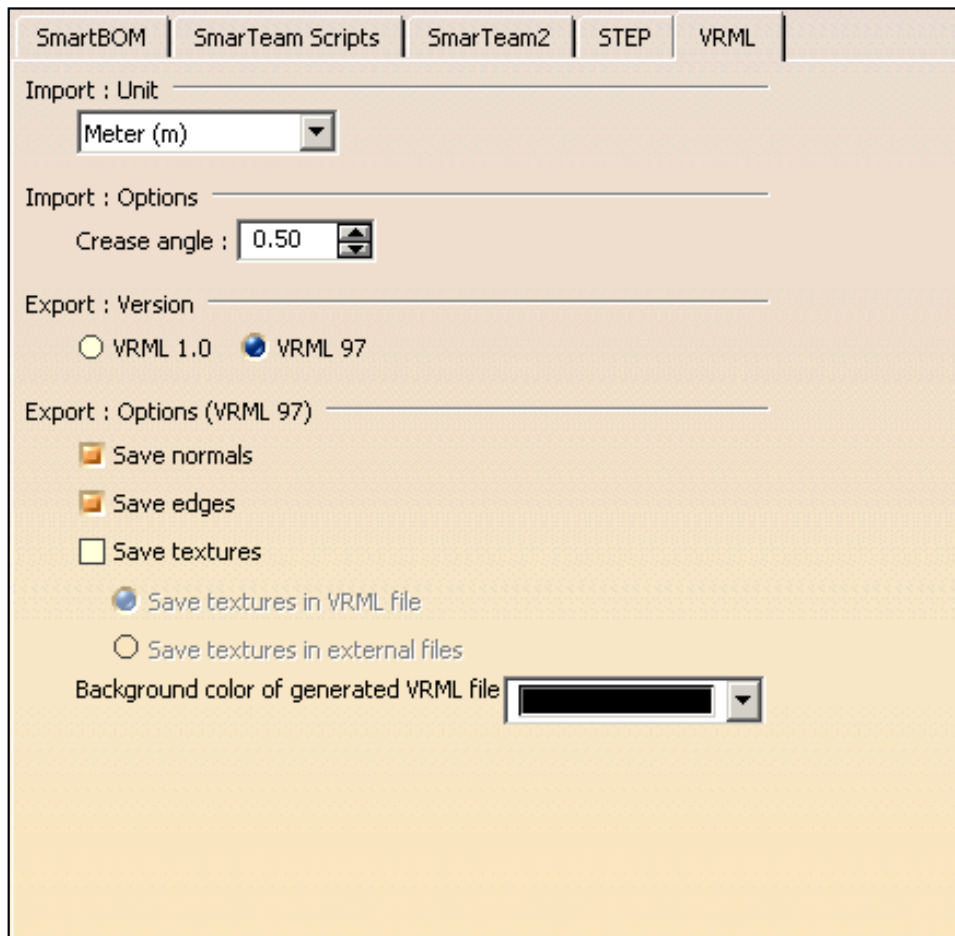
In this case, when you recover the STEP license, you have to go back to the STEP options panel to restore the good values.



# VRML

 This task explains how to customize the VRML settings when saving a document in VRML format.

-  **1.** Select the **Tools->Options...** command.
- 2.** In the left-hand tree, select the **General** category then the **Compatibility** sub-category.
- 3.** Click the **VRML** tab:



The screenshot shows the 'VRML' tab of the 'Options' dialog box. The dialog has a title bar with tabs for 'SmartBOM', 'SmarTeam Scripts', 'SmarTeam2', 'STEP', and 'VRML'. The 'VRML' tab is active. The dialog is divided into sections for 'Import' and 'Export' settings.

**Import :** Unit:

**Import :** Options: Crease angle:

**Export :** Version:  VRML 1.0  VRML 97

**Export :** Options (VRML 97):

- Save normals
- Save edges
- Save textures
- Save textures in VRML file
- Save textures in external files

Background color of generated VRML file:

- 4.** Modify the settings as needed:

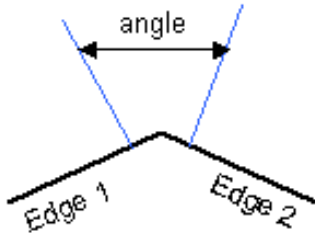
## Import

- **Unit**

From the pulldown list, select the desired unit for the VRML file to be imported: Meter, Millimeter or Centimeter.

- **Options**

The Crease angle field impacts the generation of default normals. The following picture illustrates this:



If the angle you defined in the Crease angle field is lower than the angle between the two geometric normals (shown in blue) of the adjacent edges, then a default and unique normal will be calculated so that the faces look smoother across the edges.

If the defined angle is greater than the angle between the two normals, two normals will be calculated and the faces will look less smooth across the edges.

## Export

- **Version**

You are now able to choose between the VRML 1.0 and the VRML 97 format. To do so, simply check the corresponding radio button.

Note: when saving sections in VRML 97 format, make sure that the *Save edges* export option is checked.

- **Options**

This option is relevant for VRML 97 format only and lets you save normals and/or edges contained in your model when converting it into VRML format.

Bear in mind that saving edges increases the file size significantly.

You can also activate the *Save textures* option to save your textures:

- in an external file of type .jpg
- directly in the VRML file. However, note that this will increase the file size considerably.





- use the "Background color of generated VRML file" pulldown list to select the background color to be used when visualizing the generated VRML file in a standard VRML viewer (such as Cosmo Player, for instance). The default color is "black" but you can select the desired color from the list or choose the **More Colors...** option to access the color palette. This palette lets you define more colors or create your own colors.  
Note that when reopening your VRML document in Version 5, this background color will not be taken into account.

5. Click **OK** to validate your settings.



# Parameters and Measure

Knowledge  
Language  
Measure Tools  
Parameter Tolerance  
Report Generation  
Symbols  
Units

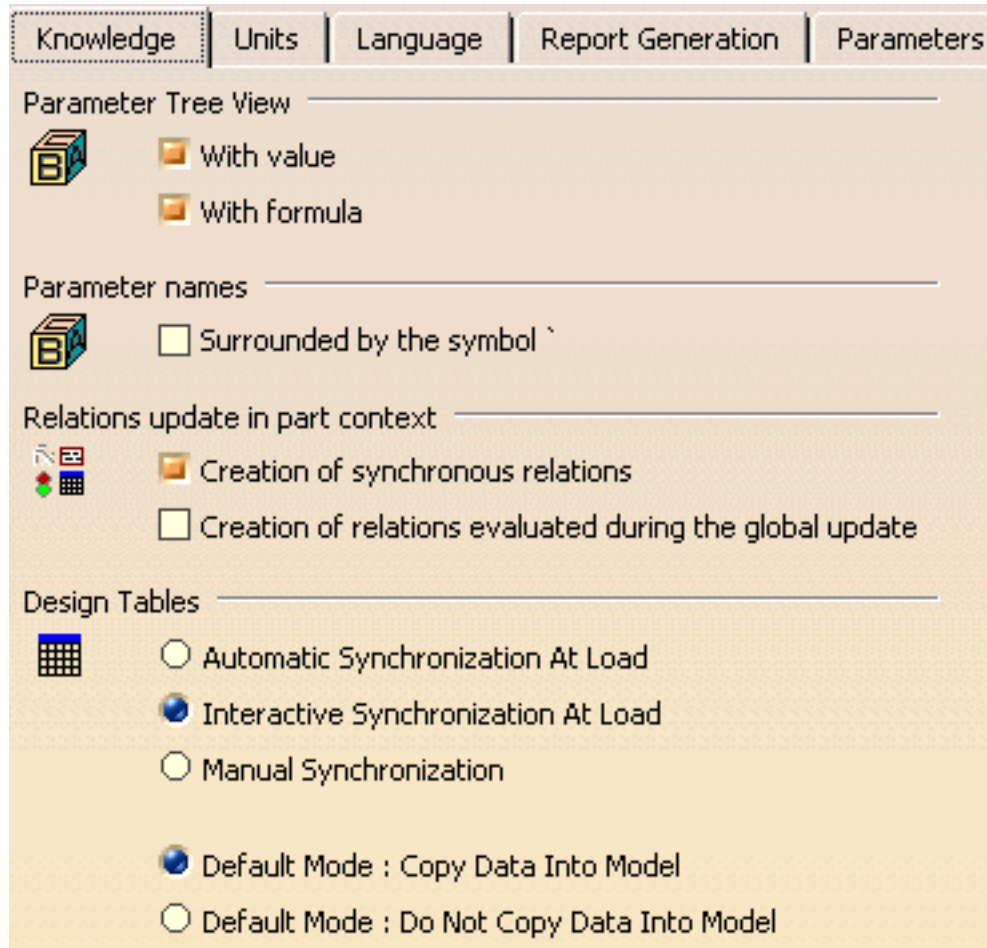
# Knowledge



This task explains how to specify the options when working with Knowledgeware relations, parameters and design tables. Refer to [Using Knowledgeware Capabilities](#) for more information.



1. Select the **Tools->Options...** command. The **Options** dialog box displays.
2. Select **General->Parameters and Measure** and click the **Knowledge** tab. This is what you can see onscreen:



## Parameter Tree View Field

- Check **Tools->Options...->General->Parameters and Measure->Knowledge->Parameter Tree View->With value** to display the parameter values in the specification tree.
- Check the **Tools->Options...->General->Parameters and Measure->Knowledge->Parameter Tree View->With formula** to display the formulas constraining the parameter in the specification tree.

## Parameter names Field

>

- Check the **Tools->Options...->General->Parameters and Measure->Knowledge->Parameter Names->Surrounded by the symbol'** option if you work with non-Latin characters. If this option is unchecked, parameter names should have to be renamed in Latin characters when used in formulas.

## Relations update in part context

Before V5R12, Knowledge relations (formulas, rules, checks, design tables, and sets of equations) used to execute as soon as one of their inputs were modified.

The user can now choose, when creating the relation, if it will be synchronous (i.e. the evaluation will be launched as soon as one of its parameters is modified) or asynchronous (i.e. the evaluation will be launched when the Part is updated). Each relation can therefore be synchronous or asynchronous.

The 2 following options enable the user to create synchronous or asynchronous relations.

### Creation of synchronous relations

Enables the user to create synchronous relations, that is to say relations that will be immediately updated if one of their parameters is modified. Relations based on parameters are the only one that can be synchronous.

### Creation of relations evaluated during the global update

Enables the user to associate the evaluation of asynchronous relations with the global update. The relations can be asynchronous for 2 reasons:

- The user wants the relations to be asynchronous
- The relation contains measures.

- Relations based on parameters: These relations can be synchronous or asynchronous.
- Relations based on geometry: These relations can only be asynchronous.
- Relations based on parameters and on geometry: For the part of the relations containing parameters, the user decides if he wants the update to be synchronous or not. For the other part of the relations, the update occurs when the global update is launched.

To get more information about the update in a Part context, see [Controlling Relations Update](#).

## Design Tables Field

|                                           |                                                                                                                                                                                                                                                                                                                                                                                                                        |
|-------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Automatic Synchronization at Load         | When loading a model containing user design tables, if the design table files have been modified and the external file data is contained in the model, the design table will be synchronized automatically if this radio button is checked.                                                                                                                                                                            |
| Interactive Synchronization at Load       | When loading a model containing user design tables whose external source file was deleted, this option enables the user to select a new source file or to save the data contained in the design tables in a new file.                                                                                                                                                                                                  |
| Manual Synchronization                    | When loading a model containing user design tables, if the design table files have been modified and the external file data is contained in the model, the design table will be synchronized if this radio button is checked. To synchronize both files, right-click the design table in the specification tree and select the <b>DesignTable object-&gt;Synchronize</b> command or the <b>Edit-&gt;Links</b> command. |
| Default Mode: Copy Data Into Model        | If checked, the data contained in the external source file will be copied into the model.                                                                                                                                                                                                                                                                                                                              |
| Default Mode: Do Not Copy Data Into Model | If checked, the data contained in the external source file will not be copied into the model.                                                                                                                                                                                                                                                                                                                          |

3. Click **OK** when done.

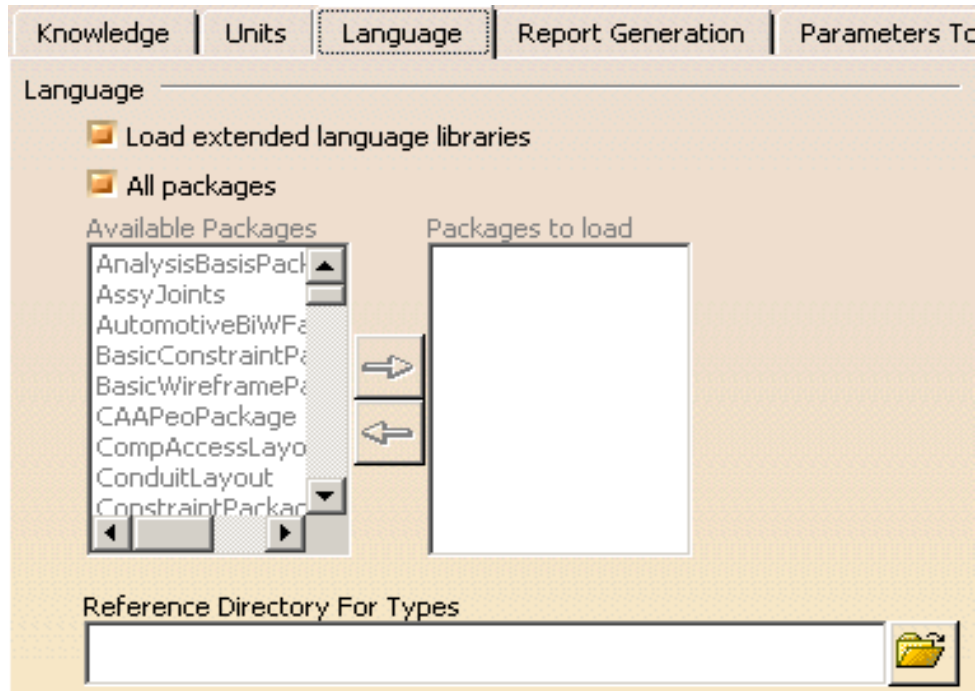
# Language



This task explains how to specify the options you may need to check when working with Knowledge browsers.



1. Select the Tools->Options... command. The Options dialog box is displayed.
2. Select General->Parameters and Measure and click the Language tab. This is what you can see onscreen:



## Language Field

This field is to be used when using *measures* in relations or *user functions*. Measures are specific functions to be used in formulas and rules.

The *Knowledge Advisor User's Guide* provides you with tasks explaining how to use measures. For how to create and use user functions, see the *CATIA Application Architecture* documentation.

- Check Tools->Options...->General->Parameters and Measure->Knowledge->Load extended language libraries and select the packages you want to load under Packages to load if you want to load a limited number of packages.



This option is particularly useful for the administrator to limit the number of packages used by the user.

It is also very useful to improve performances since only the required libraries are loaded.

- Check Tools->Options...->General->Parameters and Measure->Knowledge->Load extended language libraries and select All packages to load all the packages displayed under Packages to load.

## Reference Directory For Types field

If you want to reuse the generated type in another Catia session in the PKT workbench, proceed as follows:

- save the CATGScript file in the Directory indicated in the Reference Directory for Types field (see **Tools->Options->Parameters and Measure-> Language** tab)
- check the **Load extended language libraries** check box and select the package containing the type you created.

3. Click **OK** when done.

# Measure Tools



This task explains how to customize measure settings.

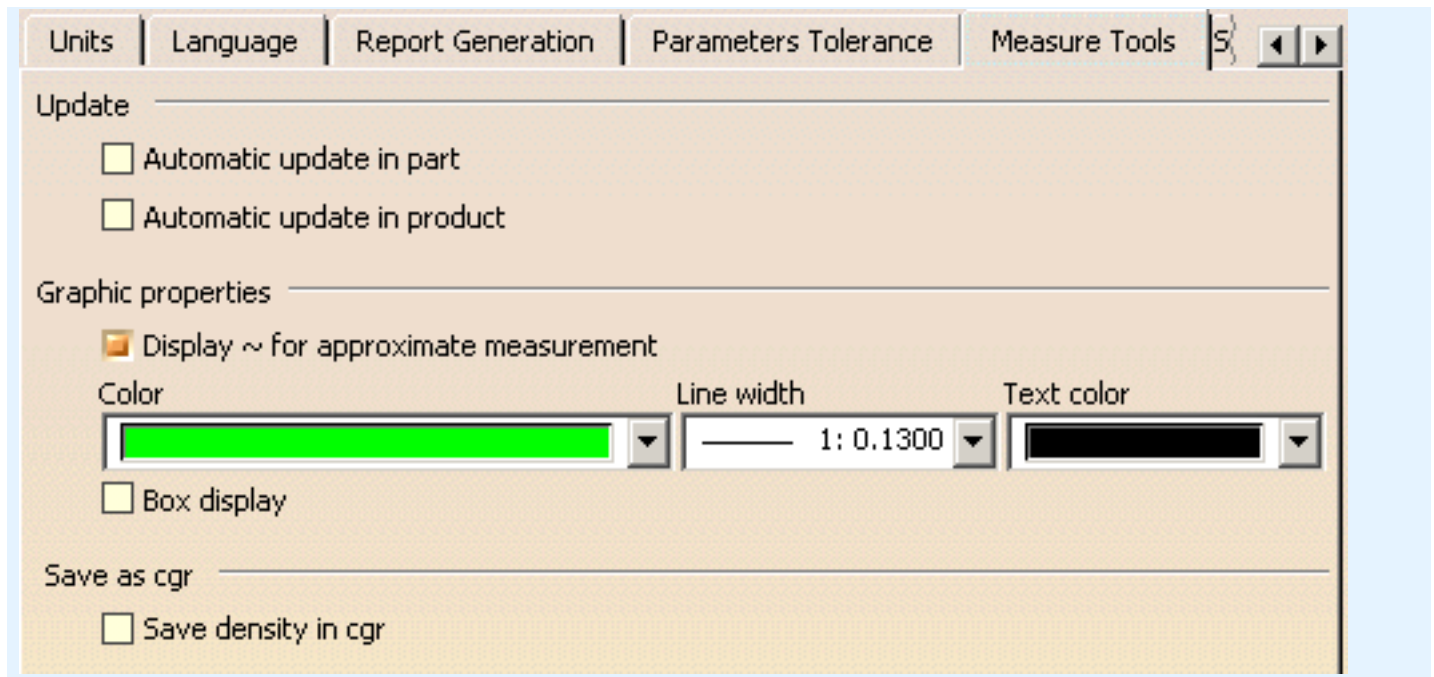


1. Select **Tools** -> **Options** from the menu bar:

The Options dialog box appears.

2. Click **General** -> **Parameters and Measure** in the left-hand box.

3. Click the **Measure Tools** tab.



The Measure Tools tab lets you customize the graphic properties and automatic update of measures.

## Update

- **Automatic update in part:** when checked, measures made on parts are automatically updated if you edit the part.  
**Note:** This option requires that the automatic update of parts option is also checked (**Tools** -> **Options, Mechanical Design** -> **Part Design** -> **General tab**).
- **Automatic update in product:** when checked, measures made on products are automatically updated if you modify geometry or move geometry in a product structure context.  
**Note:** If you edit a part, the measure is updated when you activate the product.



Only associative measures can be updated.



## Graphic properties

The graphic properties available for editing are:

- **Display ~ for approximate measurement:** sets the default display of the tilde (~) in dialog boxes and the geometry area to visually identify approximate measures.
- **Color, line width and text color:** sets fill color, measure line width and text color.
- **Box:** when checked, creates a box of the color specified around measures.

**Important:** You cannot edit the text color or measure style (i.e. choice between fill color or box).



Graphic properties set in the Options dialog box are automatically applied to documents containing measures made before Version 5 Release 8, Service Pack 1 when you open the document. You are therefore advised to customize your graphic properties before you open the document.

### Save as cgr

- **Save density in cgr:** when checked, stores the density, if any, of a part in the CGR when the part is saved as a CGR file.  
This lets you measure part density in visualization mode using the Measure Inertia command.

4. Click **OK** in the dialog box when done



# Parameter Tolerance

The Parameter Tolerance tab allows you to specify the default tolerance of Angle and Length type parameters.

Angle and Length type parameters can be assigned a tolerance. To do this, edit the parameter, right-click the value field then select the 'Add Tolerance...' command from the contextual menu. The default values which are displayed in the tolerance dialog box are those you have specified in the Parameter Tolerance tab.

Using the tolerances specified in the figure opposite:

- If you create a Length type parameter then assign a tolerance to it, the default values displayed in the tolerance edition box will be 1 mm and 0 mm.
- If you create an Angle type parameter then assign a tolerance to it, the default values displayed in the tolerance edition box will be 1 deg and 0 deg.
- If you uncheck the 'Default Tolerance' box, all value fields are grayed out and you can't specify any default values.

The screenshot shows a dialog box with four tabs: Knowledge, Parameters Tolerance (selected), Symbols, and Units. The dialog is divided into two sections: Length and Angle. Each section has two input fields: Maximum tolerance and Minimum tolerance. The Length section shows Maximum tolerance as 1mm and Minimum tolerance as 0mm. The Angle section shows Maximum tolerance as 1deg and Minimum tolerance as 0deg. At the bottom, there is a checkbox labeled 'Default Tolerance' which is currently checked.

| Parameter Type | Maximum tolerance | Minimum tolerance |
|----------------|-------------------|-------------------|
| Length         | 1mm               | 0mm               |
| Angle          | 1deg              | 0deg              |

Default Tolerance

# Report Generation



This task explains how to customize the reports generated by the Global Check Analysis tool in the Knowledge Advisor and Knowledge Expert workbenches.

You can choose to display a xml or a html report.

## Displaying a HTML report

To generate a html report when performing the check analysis, go to **Tools->Options->General->Parameters and Measure** and select the **Report customization** tab. Select **Html** in the Select **Report Visualization** area.

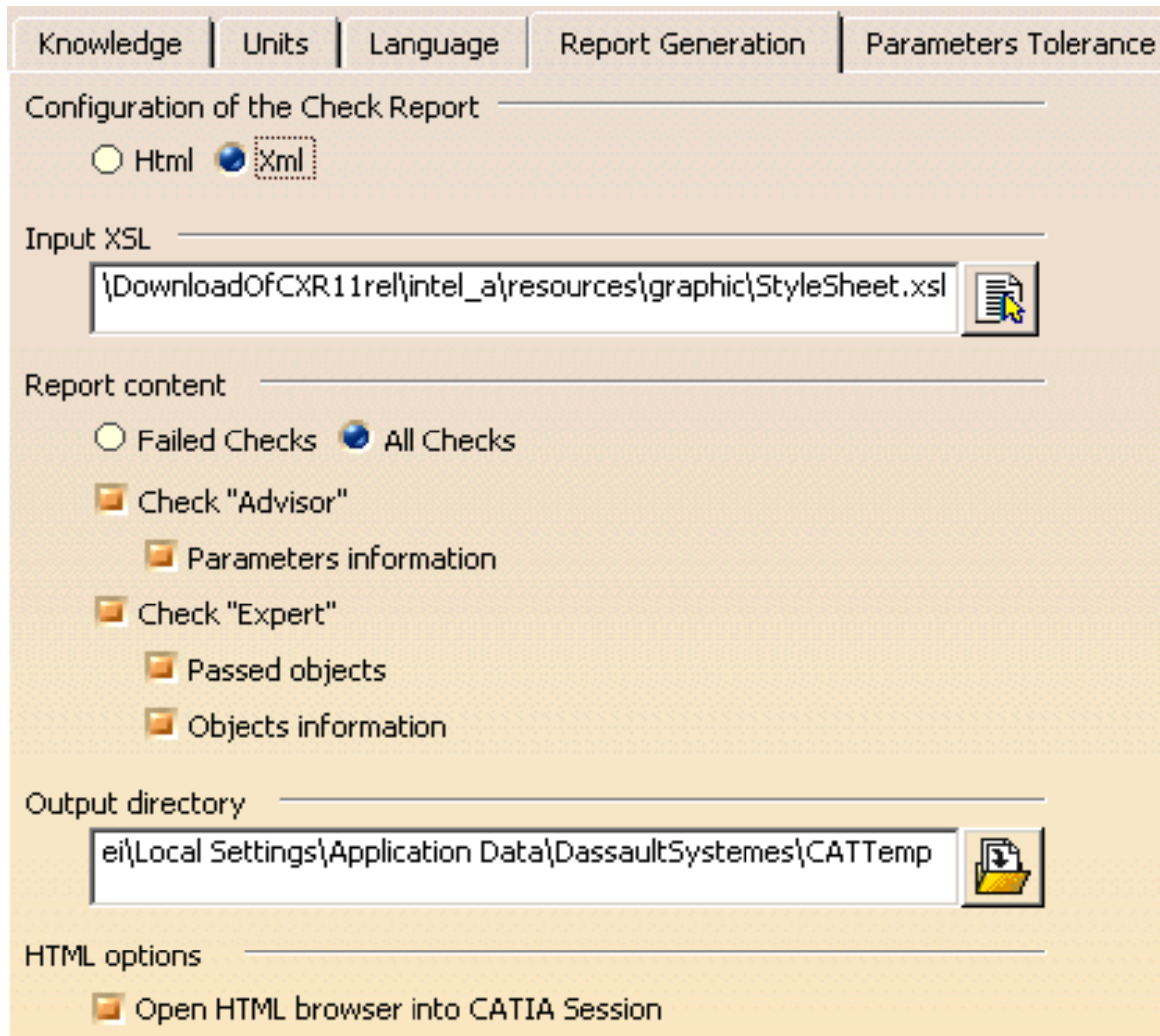
In this case, only the **Check Advisor**, the **Check expert** and the **Passed objects** options are available in the Report content area. You can specify the output directory containing the generated HTML report in the **Select output directory** field.



Select **Html** if you use a Netscape browser.

## Displaying a XML report

To display a XML report when performing the check analysis, go to **Tools->Options->General->Parameters and Measure** and select the **Report customization** tab. Select **Xml** in the Select **Report Visualization** area. . The following window opens:



The **Report Customization** tab is made up of 4 different fields: The **Select Input XSL**, the **Report Content**, the **Select output directory**, and the **HTML options** fields.

### Select Input XSL field

This field enables the user to select the XSL style sheet that will be applied to the generated XML report. The StyleSheet.xml file is the default XSL file, but you can use your own template.

### Report content field

|                      |                                                                                                                             |
|----------------------|-----------------------------------------------------------------------------------------------------------------------------|
| <b>Failed Checks</b> | If checked, the generated report will contain information about the failed checks only.                                     |
| <b>All Checks</b>    | If checked, the generated report will contain information about all the checks contained in the document.                   |
| <b>Check advisor</b> | If checked, the generated report will contain information about all the Knowledge Advisor checks contained in the document. |

|                     |                              |                                                                                                                                                                                                           |
|---------------------|------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
|                     | <b>Parameter information</b> | If checked, the generated report will contain information about the parameters of the Advisor checks.                                                                                                     |
| <b>Check expert</b> |                              | If checked, the generated report will contain information about all the Knowledge Expert checks contained in the document.                                                                                |
|                     | <b>Passed objects</b>        | If checked, the generated report will contain information about the objects that passed the Expert checks as well as information about the parameters of these objects (diameter, depth, pitch,...).      |
|                     | <b>Objects information</b>   | If checked, the generated report will contain information about all the objects contained in the Expert checks as well as information about the parameters of these objects (diameter, depth, pitch,...). |

## Select output directory field

This field enables the user to select the output directory containing the generated XML report.

## HTML options field

This option is available for Windows only. It enables the user to define if the report will be opened in a CATIA session (in this case, the check box should be checked) or if it will be opened in an Internet Explorer session (in this case, the check box should remain unchecked.)



Note that it is highly recommended not to use this report as a basis for macros or for other applications. It is only provided for information purposes.



# Customizing Constraint Appearance



This task shows you how to customize the display and style of the constraints defined in your document.



1. Select the **Tools->Options** command.

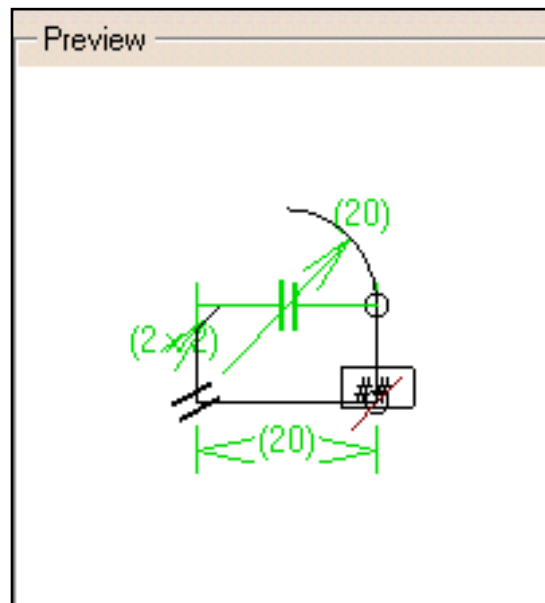
The **Options** dialog box is displayed.

2. Click **General** category, then the **Parameters** category, then click the tab **Symbols**.

This tab lets you customize:

- Constraint Style
- [Dimension Style](#)
- [Display at Creation](#)

Note that a Preview glyph displays your preferences as you are specifying them.



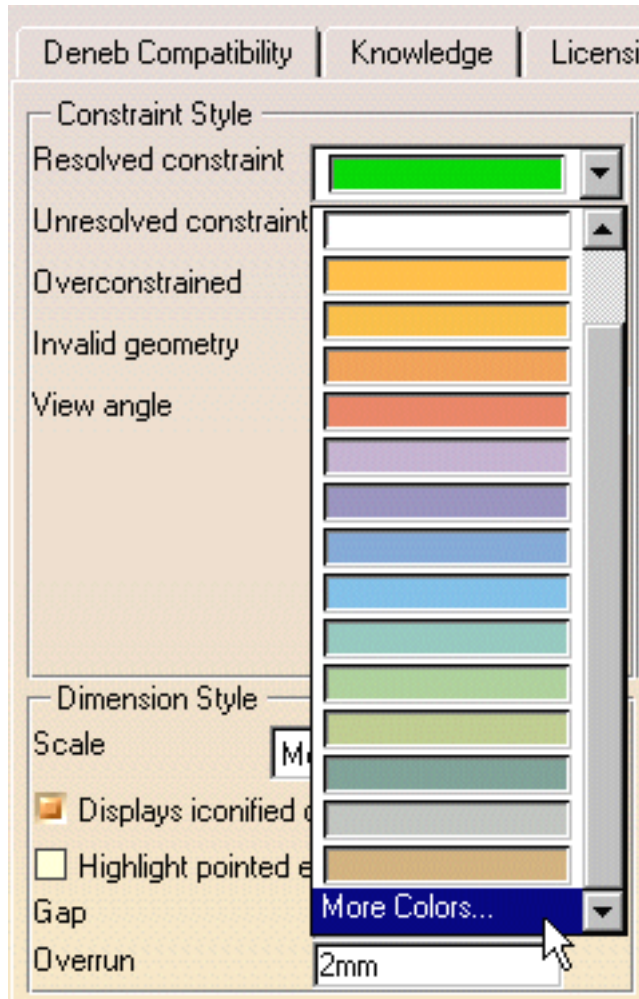
### Constraint Style

3. To set the colors where appropriate, click the arrow of the constraint type combo box.

A list appears containing the current color and the colors available by default.

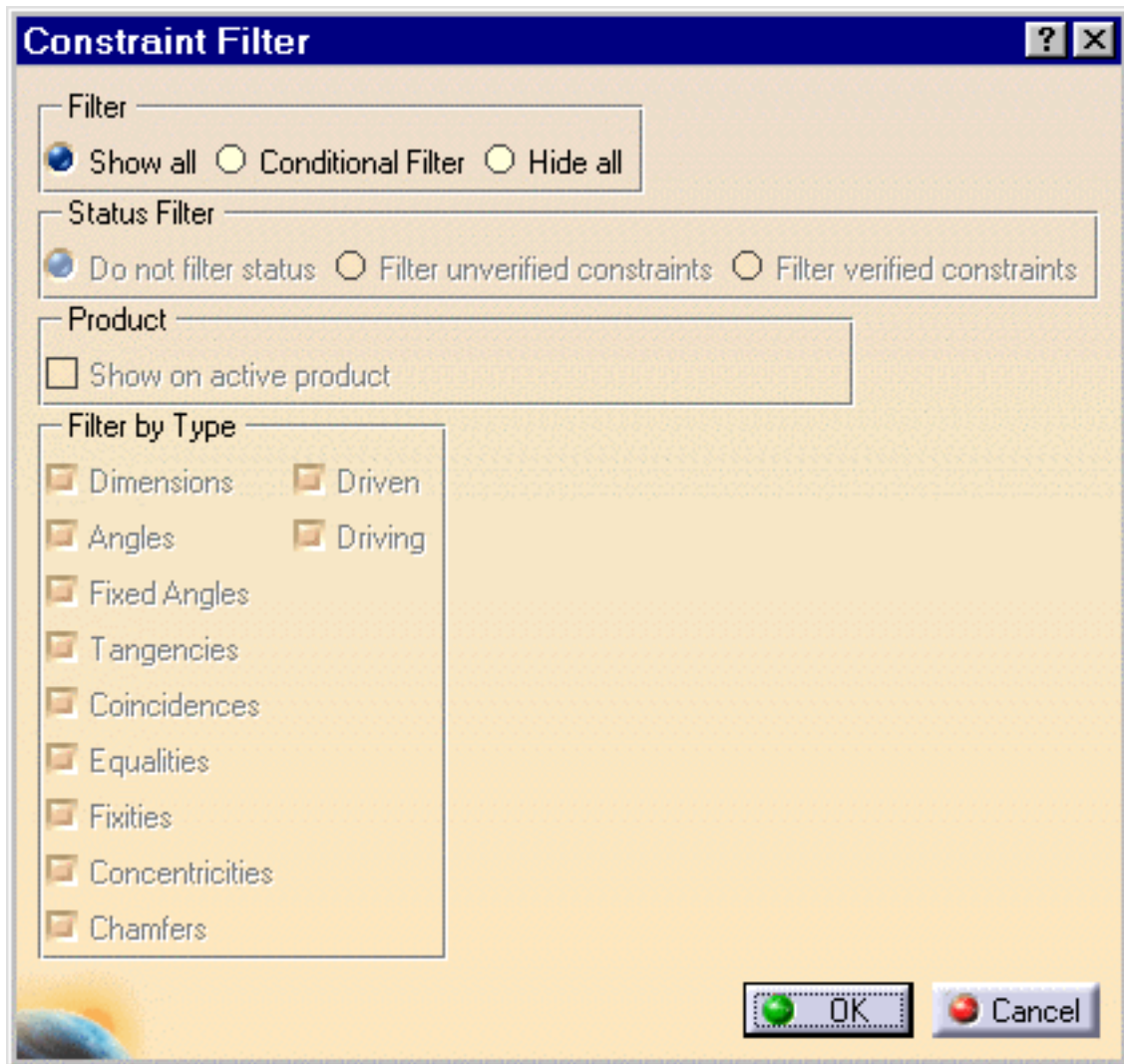
You can define a personal color too. See [CATIA](#) for more information.

4. Choose any of these colors.



5. To choose between the constraints you wish to see or not in the geometry area, click the Filter button.

The Constraint Filter dialog box contains filter options available for all the constraints that can be defined for the geometry.



The Filter frame contains three options:

- **Show all:** shows all the constraints
- **Hide all:** hides all the constraints
- **Conditional Filter:** once activated, it makes the options of the Status Filter frame available. You can then decide whether you wish to display the constraint status or not, or display unverified or verified constraints. It also makes the Filter by Type options available. You can then decide the constraint types you wish to display by checking the appropriate options.

The Filter by Type option now lets you filter out driving or driven constraints.

The Product frame contains the Show on active product option. If activated, this option displays the constraints defined on the active product. This option is specific to Assembly Design workbench.

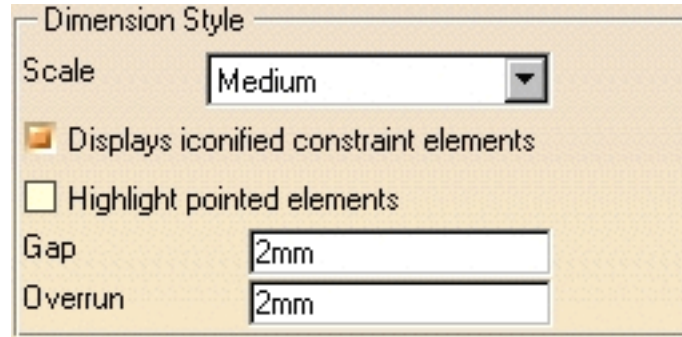
6. Uncheck the constraints you do not wish to display and click OK to confirm.



## Dimension Style

7. Set the **Scale** option to **Large**. This defines the size of the symbols for tangency and parallelism constraints. You can set:

- Small
- Medium
- Large

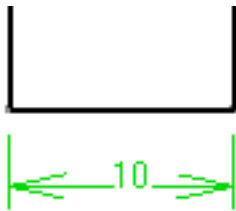


8. Check **Displays iconified constraint elements**. This option increases the number of graphic symbols you can see in the geometry area. For example, you can see symmetry axes.

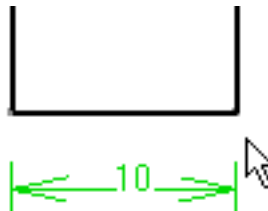
9. Check **Highlight pointed elements** to make sure the application highlights the constrained elements when their constraints are pointed to.

10. Enter a value to define the gap between construction lines and geometric elements. For example, enter 4mm.

The default gap is 2mm.

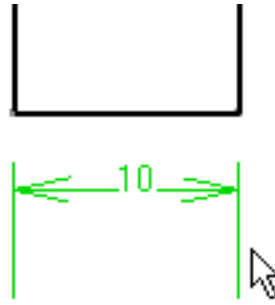


The glyph previews the new gap.

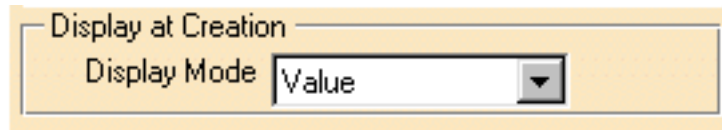


11. Enter a value to define the overrun between construction lines and geometric elements. For example, enter 9mm.

The glyph previews the new overrun.

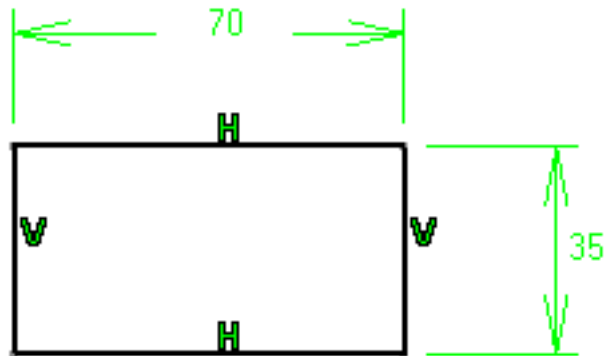


### Display at Creation

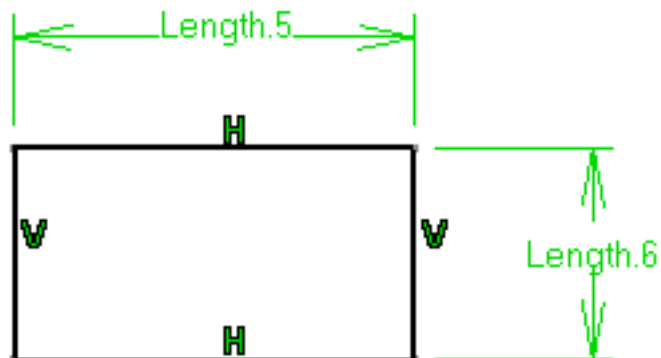


12. The Display mode option lets you choose between four options:

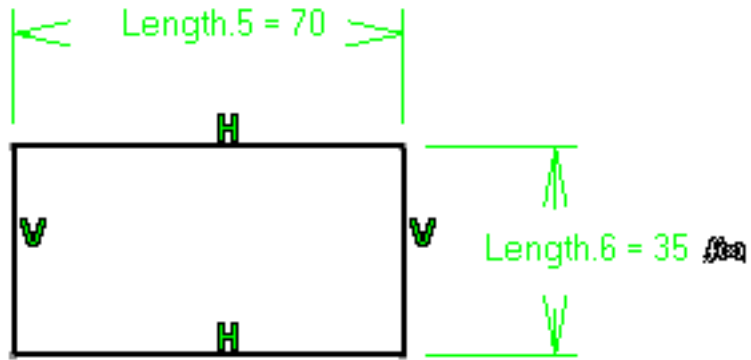
- 'Value': only the constraint (or parameter) value is displayed.



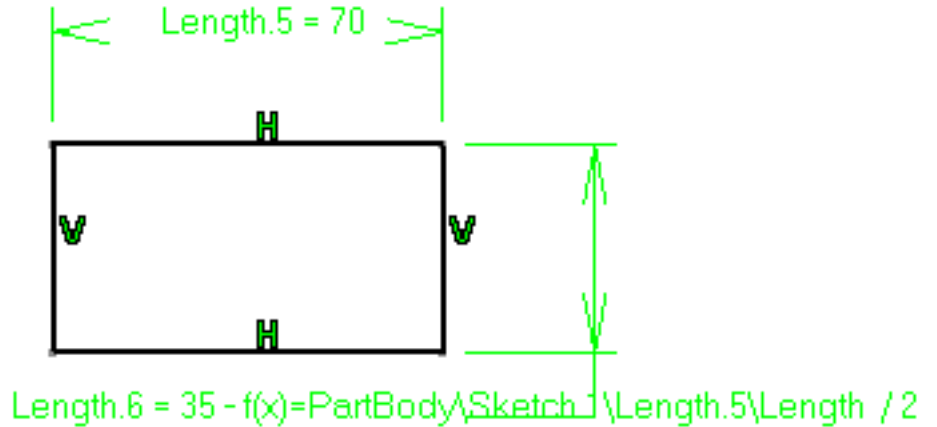
- 'Name': only the constraint (or parameter) name is displayed.



- 'Name + Value': the constraint (or parameter) name and value are both displayed.



- 'Name + Value (formula)': the constraint (or parameter) name and value are displayed as well as the possible formula defined for this constraint.



Whatever mode you choose, it applies to all constraints you are creating. However, you can edit each constraint individually by using the contextual commands available (**xxxobject -> Value Display** or **Name Display** or **Name/Value Display**).

13. Click **OK** to confirm the operation and quit the **Options** dialog box.



# Customizing Units

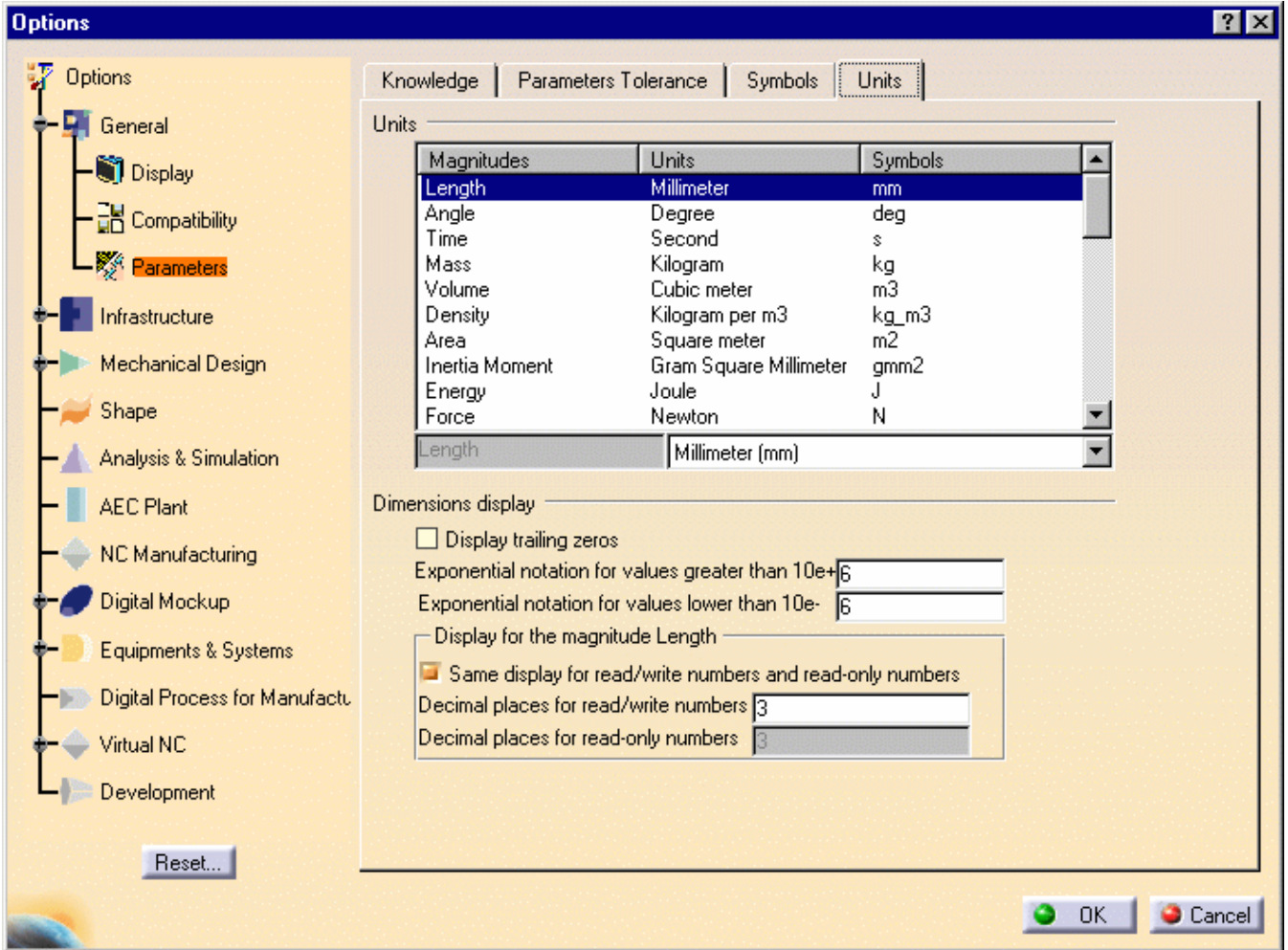


This task explains how to customize units.

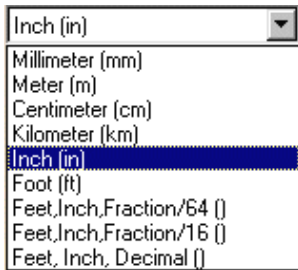


1. Select the **Tools->Options...** command.
2. In the **General** category, click the **Parameters** sub-category, then the **Unit** tab. You can define or redefine:
  - o the default unit
  - o and the display of values for magnitude type parameters.

In the upper part of the dialog box, the default unit along with its symbol is displayed for each magnitude:



3. In the lower part of the dialog box, the 'Dimension Display' check boxes allow you to define:
  - o whether you want to display trailing zeros
  - o the upper and lower limits for using an exponential notation
  - o how many decimal places to display.
4. If you want to redefine a default unit:
  1. Select the line with the magnitude whose unit is to be redefined. The list of available units for this magnitude is displayed right below the magnitude list (the magnitude name is grayed out). For example, if you want to redefine the Length unit, the list of available units is:



2. select the unit you want to define as new default unit in the selection list (in the example above, the Inch has been selected).  
The magnitude definition is updated in the magnitude list right above.
5. If you want to display trailing zeros, check the 'Display Trailing Zeros' box.  
Example: a 10 mm value will be converted in 10.000 mm if the number of decimal places is set to 3.
6. If you want to redefine the decimal places, check the appropriate options.  
Example: a 10.571 mm will be converted to 10.57 mm if the number of decimal places is set to 2.
7. If you want to define the exponential notation limits, enter the value under and below which you want to display a magnitude type parameter in an exponential form.  
Example1: 105700 mm will be displayed as 1.06e+005mm if the upper limit for an exponential notation is set to 5.  
Example2: 0.000057 mm will be displayed as 5.70e-005mm if the lower limit for an exponential notation is set to 4.
8. Click **OK** to confirm your choice.



The new settings will apply immediately both to documents already loaded and to those subsequently loaded. The default values for length, mass and time are millimeters, kilos and seconds.



# Devices and Virtual Reality

Devices  
Tablet Support

# Devices



This task explains how to set up peripheral devices.

The peripheral devices concerned are primarily:

- the joystick
- the spaceball/space mouse.

The joystick is supported on all product brands and on all supported Windows platforms.

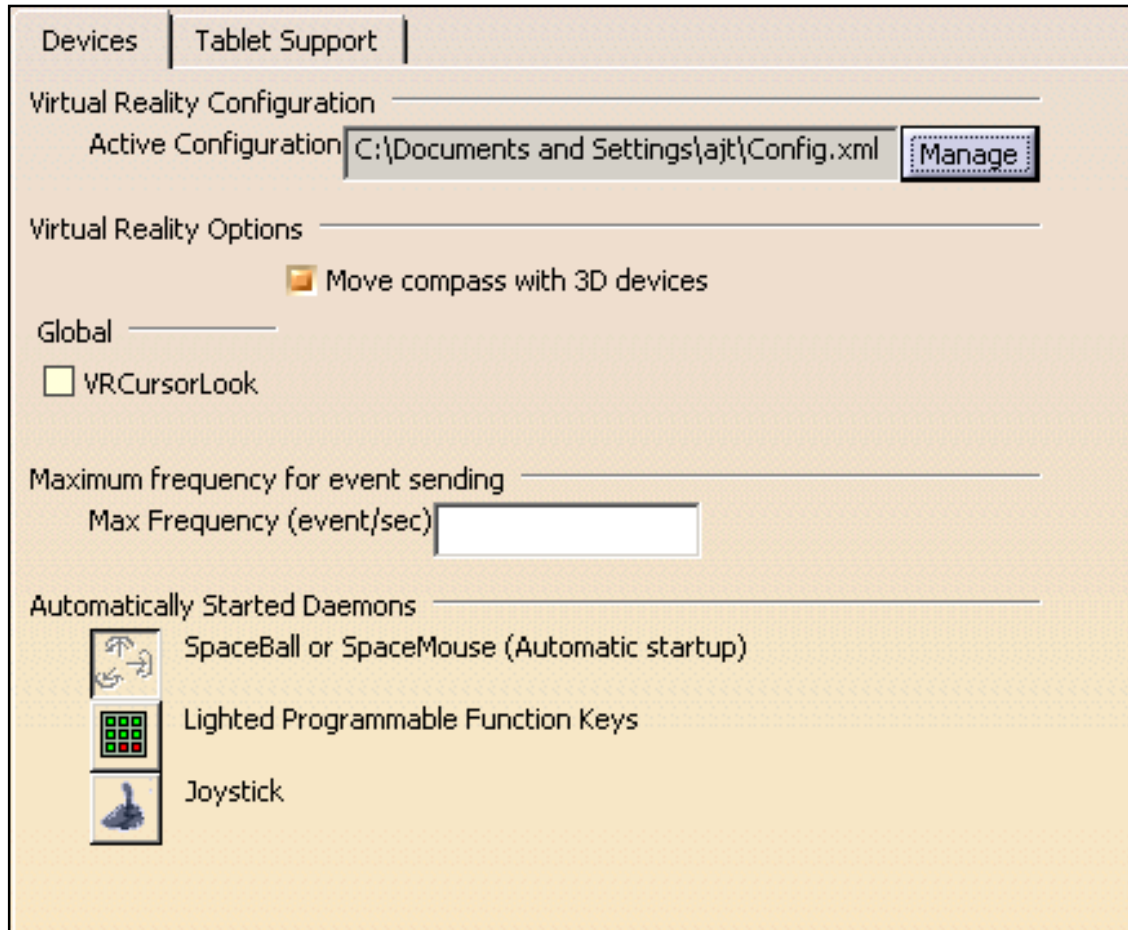
The spaceball and space mouse are supported on all P2 products on all supported platforms.



After installing Version 5, you must connect your peripheral devices and install the appropriate drivers.



- 1.** Select the **Tools->Options...** command.
- 2.** In the **General** category, select the **Devices and Virtual Reality** sub-category.
- 3.** Click the **Devices** tab:



The Devices tab lets you customize:



### Virtual Reality Configuration

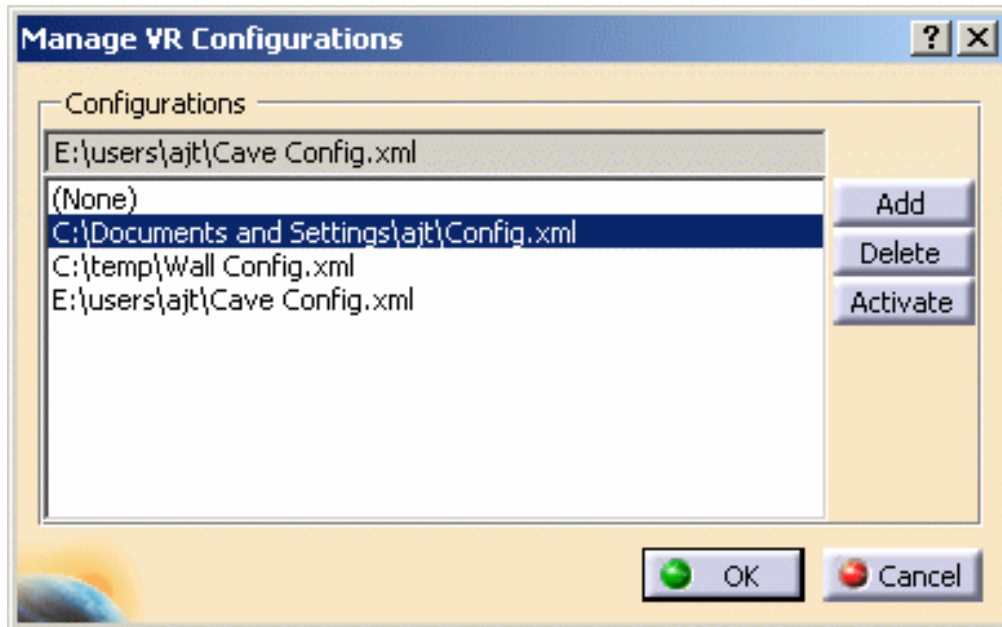
The Virtual Reality Configuration area lets you activate the configuration to be used to run your tracking devices when starting a Version 5 session.

Prior to activating a configuration, you need to generate the corresponding configuration file in the [Immersive System Assistant](#) workbench.

Each configuration is saved in a .xml file and contains all the necessary MPK display configuration, including those related to the supported drivers and brokers which are now threaded.

To set a configuration as active, click the **Manage** button to open the following dialog box:





The list contains all the configuration files you have previously generated.

Select the desired configuration in the list and click **Activate** then **OK** to validate: the selected configuration now appears in the Active Configuration field. You need to restart your Version 5 session to take the new active configuration into account.

You can also:

- remove a configuration from the list by selecting it then clicking the **Delete** button
- add a configuration to the list by clicking the **Add** button. This opens the Select VR Configuration dialog box that lets you browse your file tree to select the configuration file to be added to the list. Once a file is selected, click the **Open** button to add it to the list.

## Virtual Reality Options

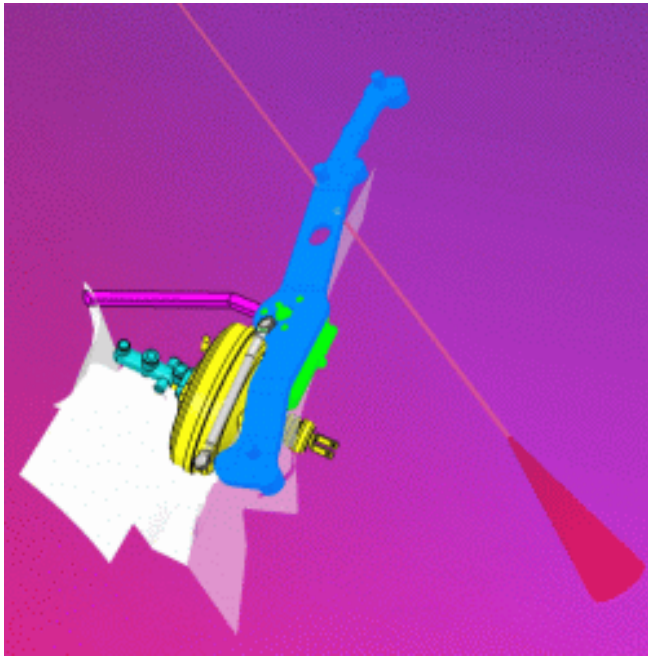
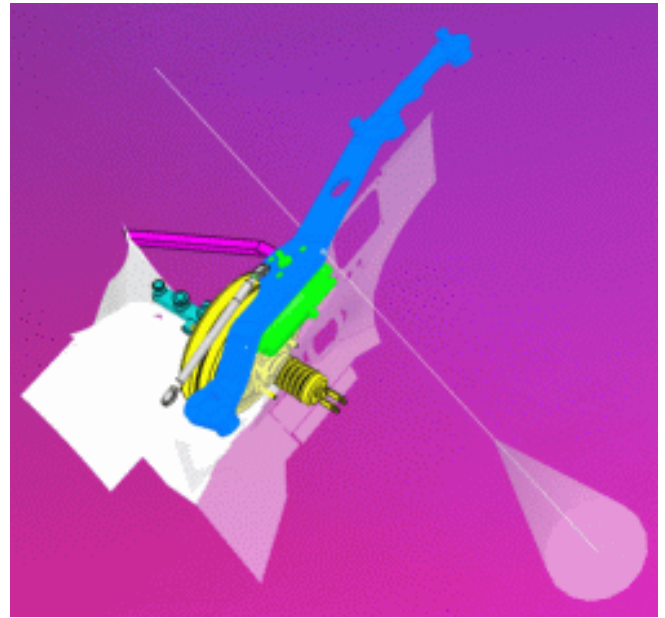
Check the "Move compass with 3D devices" option to be able to move the compass using 3D devices, i.e. a Space Mouse ou a SpaceBall for instance when put into the 3D scene.

## Global

You can customize the 3D picking device (laser beam) representation to obtain a better precision in picking and a better visualization in dark environments, all you need to do is:

Select the **VRCursorLook** check box.

the 3D picking device representation becomes a white and thin line (see images below)

**Standard representation****New representation**

### **Maximum frequency for event sending**

Maximum frequency (number of events per second) at which events are sent by peripheral devices. This value aims at limiting the frequency of emitted events for some devices.

Polhemus, for instance, is not impacted by this value.

We recommend that you use the default value, unless the peripheral device you are using has special requirements.

A process filters the mass of events generated when using I/O devices and relays only the useful events to Version 5.

The **c:VR Monitor** command toggles on a diagnostic window (named VRMonitor) displaying any drivers declared to the broker as well as any events those drivers are able to send.

## Automatically Started Daemons

This part of the tab lists the peripheral devices for which native support is provided by Version 5. Just activate the threaded drivers you need:


- **Space Daemon:** spaceball/space mouse. It is an automatic, i.e. threaded, driver which means that it can be managed by an automatic broker. This driver manages the orientation as well as button information
- **Function Keys**
- **SW\_Joystick:** joystick.

Check the corresponding option to ensure that the peripheral device will be activated and recognized the next time you start a session. This setting requires the use of a peripheral broker. If you are using the peripheral devices for which native support is provided by Version 5 (joystick, spaceball/space mouse): the peripheral broker process is started automatically when you start a session. Otherwise, you must run it manually.


4. Click **OK** to confirm, or **Reset...** and **OK** again to reset default settings.
5. Exit your session, then restart for your new settings to take effect.

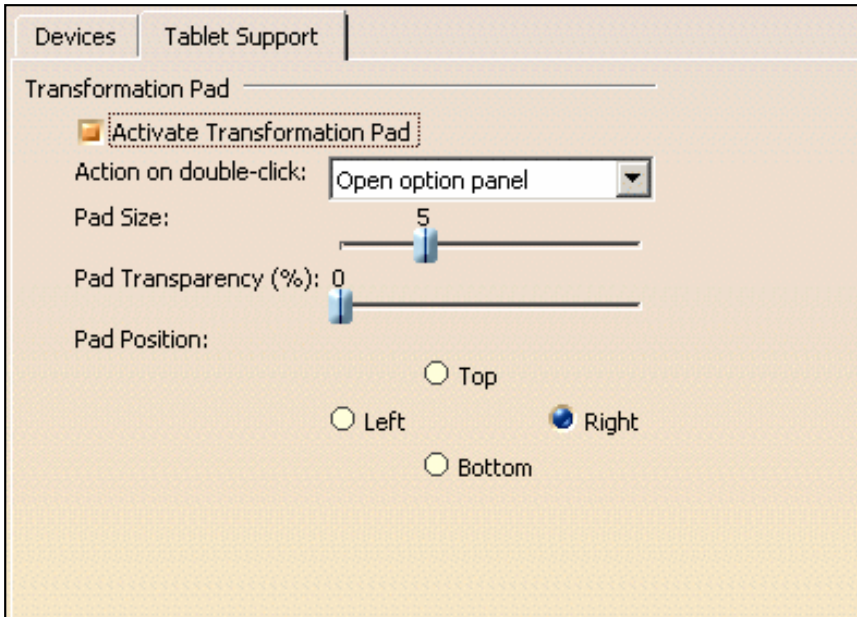


# Tablet Support

 This task explains how to set up pad options in order to perform viewing operations using the Tablet Pen on a Tablet PC. Note that you can use the pad to visualize both 2D and 3D objects.

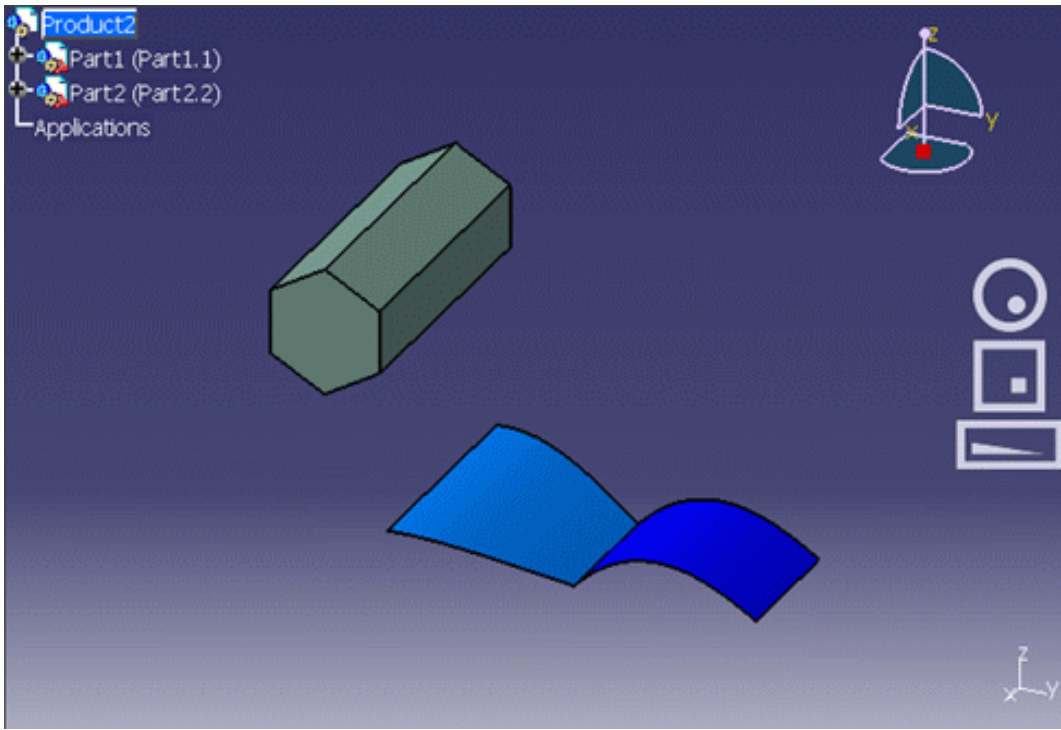
This functionality is available on Windows only.

-  **1. Select the `Tools->Options...` command.**
  
- 2. In the `General` category, select the `Devices and Virtual Reality` sub-category.**
  
- 3. Click the `Tablet Support` tab:**



The "Activate Transformation Pad" option lets you activate/deactivate the pad display in the geometry area. When working with a Tablet PC, this option is already checked.

By default, the pad is displayed to the right as shown below:



4. In the Pad Position area, activate the corresponding radio button to position the pad:

- to the top
- to the bottom
- to the left
- to the right.

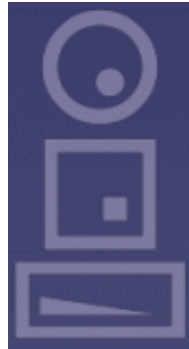
By default, the pad is located to the right part of the screen.

5. Set the pad size using the slider. You can choose a value comprised between 1 and 15.

6. Use the Pad Transparency slider to define how transparent the pad will be: the higher the value, the more transparent the pad.



*No transparency*

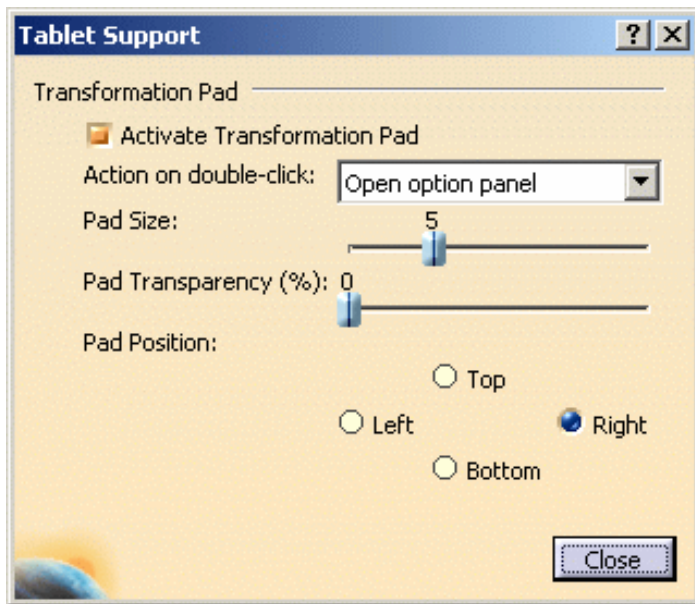


*Transparency = 60%*

7. The "Action on double-click" pulldown list lets you select the action to be triggered when double-clicking. You can choose among the following actions:

- **Open option panel**

This action opens the Tablet Support dialog box directly in the geometry area. You can then modify your pad options without having to re-access the Tablet Support tab:



- **Reframe**

Zooms in or out to fit all the geometry into the available space

- **Center location**

Centers the geometry at a specific location

- **No action**

Means that no action will be performed on double-clicking. This may be useful, for instance, to avoid involuntary actions when manipulating the Tablet Pen.

For more information on the various viewing operations you can perform with the Tablet Pen, refer to [Activating Viewing Tools](#).

8. When finished, click **OK** to validate your settings.



# Infrastructure

Product Structure  
Material Library  
Catalog Editor  
VPM Navigator  
Photo Studio  
Real Time Rendering 2  
Part Infrastructure  
3D Annotations Infrastructure

# Customizing

- Cache management
- CATIA / ENOVIA VPM Interoperability
- Nodes Customization
- Product Structure
- Product Visualization
- Tree Customization



# Customizing Cache Settings



## Working with a Cache System

Two different modes are available when a component (V4 model, V5 CATPart, V5 CATProduct, etc.) is inserted into a DMU Navigator CATProduct document:

- **Design mode:** in this mode, the exact geometry is available and the document is inserted as is
- **Visualization mode:** in this mode, a representation of the geometry only is available and the corresponding cgr file, if it exists, is inserted from the cache system.

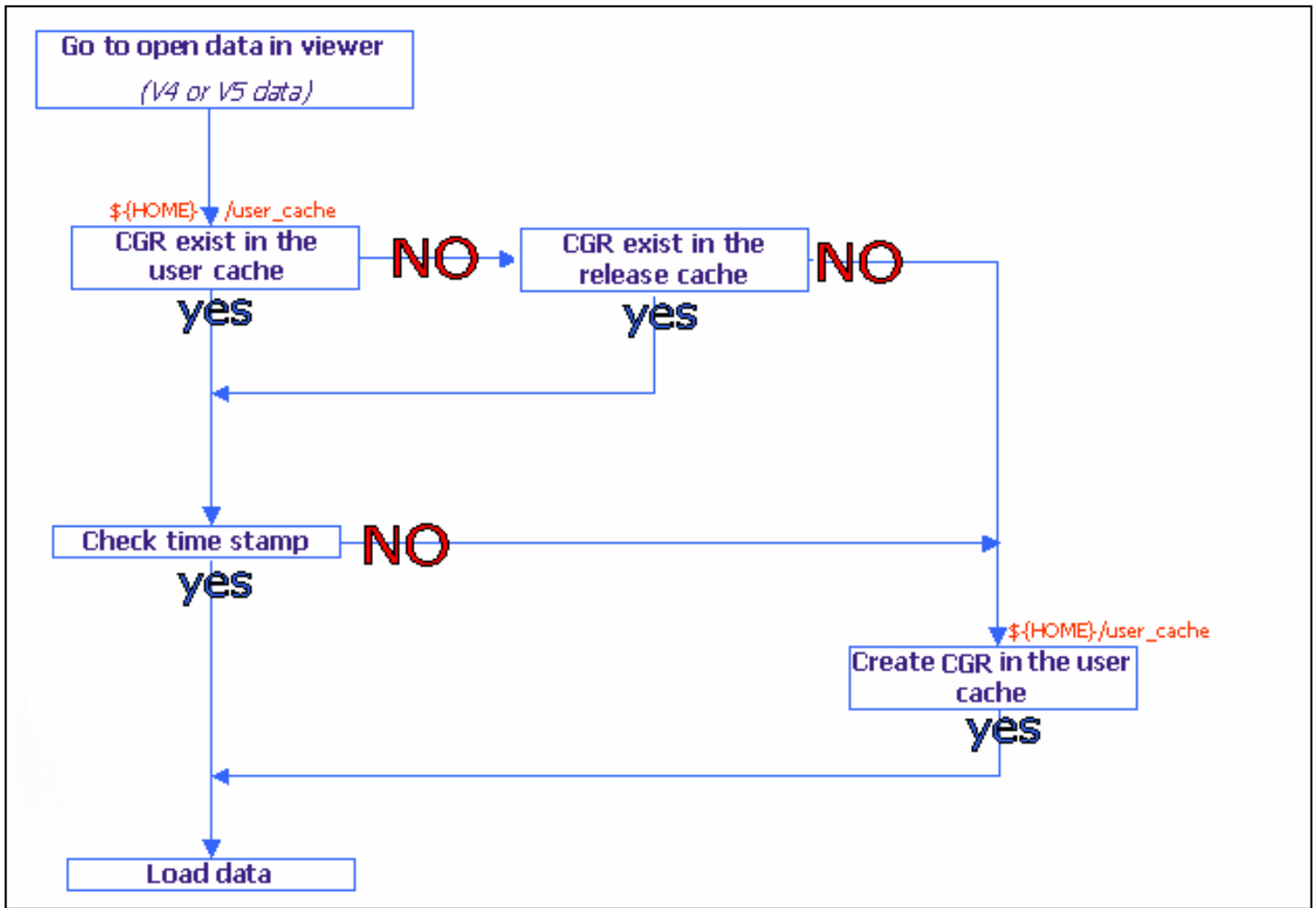
Using a cache system considerably reduces the time required to load your data. The cache system is organized into two parts:

- **Local cache:** a read/write directory located locally on your machine and used to store cgr files. The first time a component is inserted, it is tessellated. This means that the corresponding cgr file is computed and saved in the local cache as well as displayed in the document window. The next time this component is required, the cgr file which already exists (and not the original document) is automatically loaded from the local cache. The user is normally responsible for the local cache.
- **Released cache:** a read-only cache that can be located anywhere on your network. Several directories can be defined for the released cache. If a cgr file cannot be found in the local cache, the software browses the released cache directories in their listed order to see if the cgr file is located in one of them. If the cgr file is still not found, the component is tessellated and the resulting cgr file is saved in the local cache. The site administrator is normally responsible for the released cache.

A timestamp enables the verification that no modifications have been made to a document since the generation of the corresponding cgr file found in either of the above caches:

- **Timestamp:** the date and hour at which the origin document was last modified. If you activate the **Check timestamp** button, then before a cgr file is loaded into a viewer, its timestamp will be checked to verify that no modifications have been made to the document since the generation of the cgr file. If you don't activate the **Check timestamp** button, then a cgr file of a document will be loaded without any verification of its time-wise coherence with the document.

The process used for loading data using the caches is as follows:



- The user requests the loading of a document into a viewer.
- The System searches for a corresponding tessellated file (cgr file) in the user's local cache.
  - If found, then the timestamp is checked.
  - If the timestamp of the cgr file is more recent than that of the document, then the cgr file is **loaded into the viewer.**
  - If not, then the document is tessellated, the resulting cgr file is added to the user's local cache and the cgr file is **loaded into the viewer.**
- If not found, then the System searches for the corresponding tessellated file (cgr file) in the released cache.
  - If found, then the timestamp is checked.
  - If the timestamp of the cgr file is more recent than that of the document, then the cgr file is **loaded into the viewer.**
  - If not, then the document is tessellated, the resulting cgr file overwrites the existing cgr file in the user's local cache and the cgr file is **loaded into the viewer.**
  - If not found, then the document is tessellated and the resulting cgr file is added to the user's local cache and the cgr file is **loaded into the viewer.**

Reading Components from a Database: The cache system works in exactly the same way when components inserted into a CATProduct document come from a database. An additional check is run: if the cgr file is not found in the local or released caches, the DMU Navigator requests that, if the cgr file exists in the database, it be downloaded.

The cache system is managed via the Cache Management tab in the Options dialog box. All cache options can be locked.



This task explains how to customize data cache settings.



A DMU Navigator document must be opened.



It is imperative that any directory designated for use as a cache directory be reserved for this usage only, i.e. you should not manually save any other files in this directory. (When necessary, the **Check Maximum Size** option purges the least-recently-accessed files in order to maintain the cache smaller than the designated **Maximum size**. As a result, any files that you manually save to this directory could eventually be purged.)



When you are working in Cache mode, a .model or .CATPart is converted to a .cgr format. You can convert the .cgr to a .model or .CATPart format using the Design Mode command. See the task Design Mode in the Product Structure user guide.



1. Select the **Tools -> Options...** command.

The Options dialog box appears.

2. Click **Infrastructure -> Product Structure** in the left-hand box.
3. Click the **Cache Management** tab.

Cache Management | ENOVIAvpm | Nodes Customization | Product Structure | Product

Cache Activation

Work with the cache system

Cache Location

Path to the local cache: C:\Documents and Settings\jlr\Local Setti...

Path to the released cache: [Empty field]

Cache Size

Check Maximum size

Maximum size: 500 MB

Timestamp

Check timestamps

GMT timestamp format

Cache Content

Generate cgr with level of detail

### Cache Activation

4. To activate the cache, check the **Work with the cache system** checkbox. (By default, the cache is not activated.)

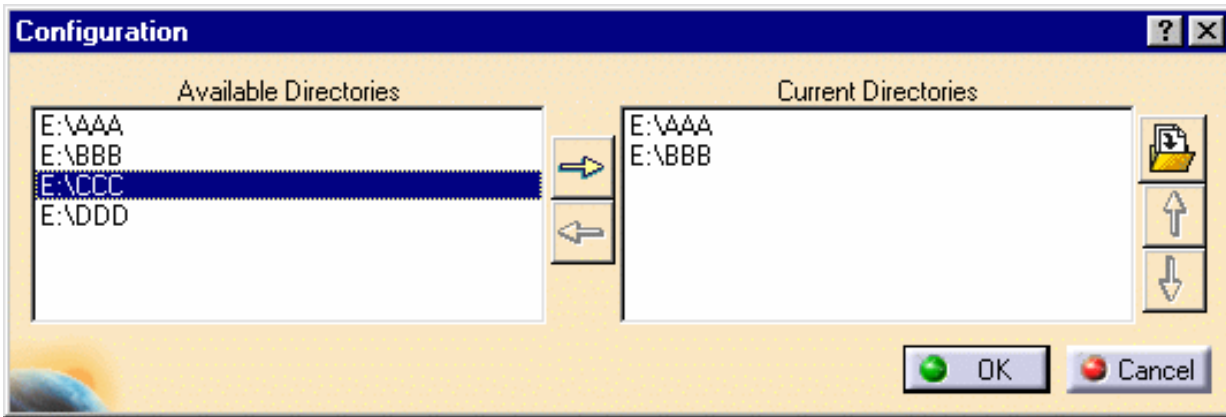
### Cache Location


New features have been added to the Cache Management that render the configuration of the local and released caches easier. The new features are:

- Local cache - a history of the last five defined paths
- Released cache - a configuration panel that enables you to:
  - select directories from a list of "accessible" directories that have been pre-defined by the administrator
  - reorder directories list
  - remove entries from the directories list

Note that it is possible to use environment variables to define a local cache or a released cache:

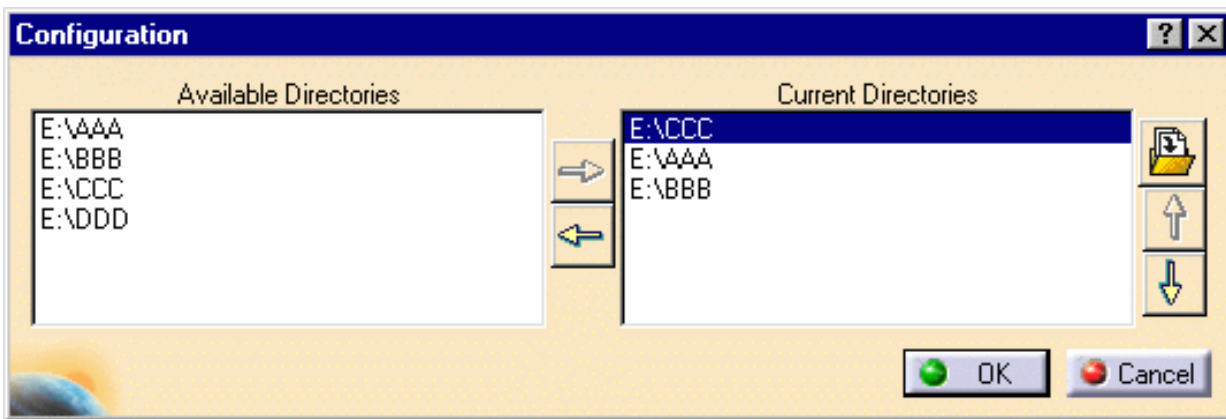
- the variable should be defined (using the **set** command on Windows NT or the **export** command on UNIX) in the script file used to launch the executable file of the interactive application
  - the variable must be correctly valuated to an existing directory
  - the entry in the local cache or released cache text-entry field should be of the form **#{ variable\_name }**
5. To define a local cache directory, enter the name of the directory in the **Path to the local cache** text-entry field or click the **Path to the local cache** selection button and select an entry from the proposed history list.
  6. To define a released cache directory, enter the name of the directory in the **Path to the local cache** text-entry field or invoke the released cache configuration panel by clicking the **Path to the released cache Configure** button.  
The Configuration panel appears.



7. To add one of the Accessible Directories to the list of Current Directories, select the entry in the Accessible Directories list and click the Right-arrow icon . The entry is added to the Current Directories list.

Note: The list of directories in the Accessible Directories list is defined by the administrator as follows:

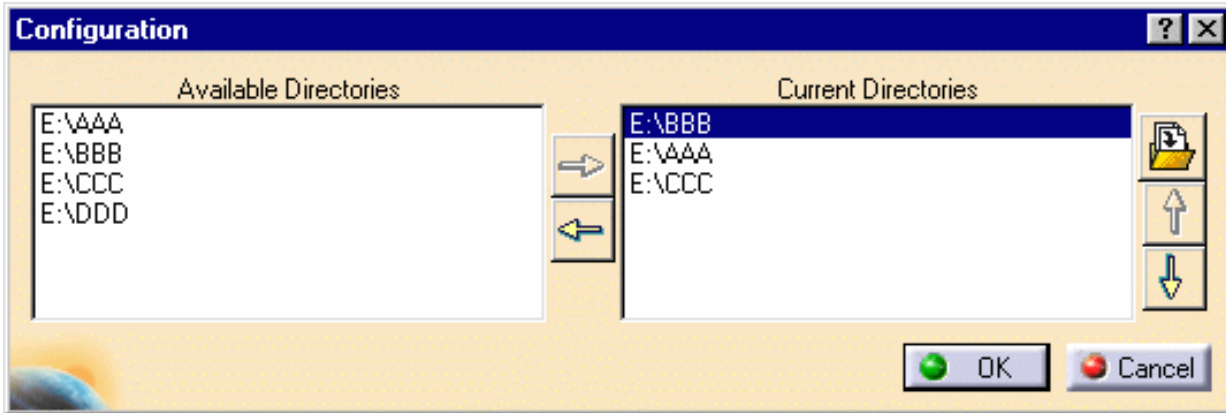
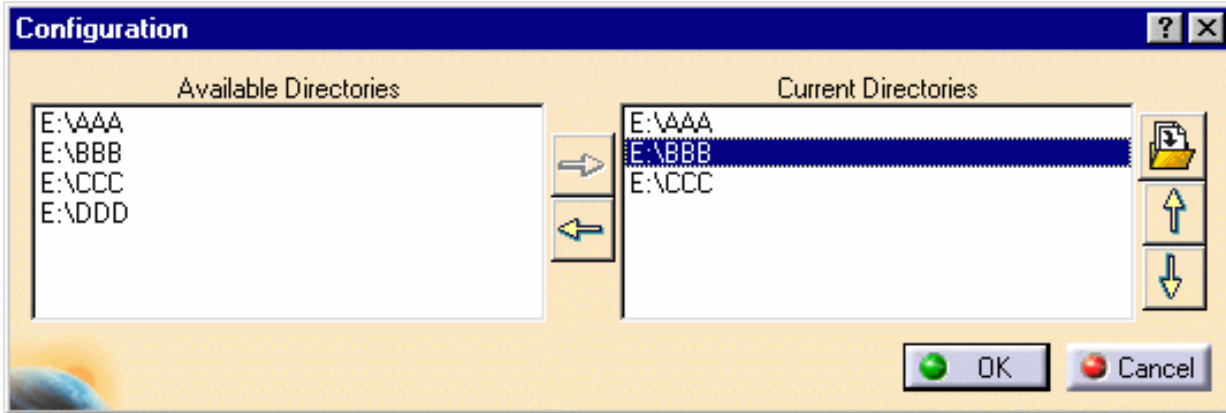
- define a .txt file of which each line contains the path to a directory that will appear in the list of accessible directories
- set and export an environment variable `AVAILABLE_CACHE_DIR_PATH` that points to the above .txt file
- the administrator can, of course, manage the access rights to the above-defined directories so as to restrict their access to groups of users or specific users



8. To move one of the entries in the Current Directories list to a higher place in the list (remember that released cache directories are searched in the order in which you see them displayed), select the entry in the Current



Directories list and click the Up-arrow icon .  
The entry is moved up in the list.



9. To delete an entry from the Current Directories list, select the entry in the Current Directories list and click the **Delete** button.

10. (Optional) Enter the paths identifying the cache locations:



The Browse icon enables you to locate the file you want. The user can enter the path to his own local cache location, and if permitted by the site administrator, one or more paths to released cache locations.

The default directory is the user's home directory under UNIX and the USERPROFILE directory under Windows.

**Cache Size**

11. (Optional) Set the maximum size for the local cache (in write mode).

When the maximum size is exceeded, the automatic deletion of .cgr files (on a first-in / first-out basis) is triggered. The default maximum size is 500 MB.

12. (Optional) Set the Check maximum size option:

When creating new cgr files in the cache, the check maximum size option serves to purge the oldest cgr files if necessary in order to respect the defined maximum cache size.

If the option is activated, the check and eventual purge will be performed.  
If the option is not activated, the check will not be performed.

### Timestamp

13. (Optional) Set the Check timestamp option:

The timestamp option serves to check whether or not the tessellated cgr file is up-to-date.

If checked, the system checks the original document's date against that of the corresponding cgr file, and if older, tessellates the document again overwriting the out-of-date version. If dates are the same, no tessellation is done and the cgr file in the data cache is loaded.

Turning this option off means that no check is run and cgr files in the cache are systematically loaded.

Until now, the cache timestamp has been based on local time. This can lead to unwanted re-tessellations if you are sharing data between sites located in different time zones.

It is now possible to base your cache on GMT time.

The following are the rules for implementing your cache based on GMT time:

- a batch job will permit you to migrate old caches to GMT time-based caches
- once a cache is migrated to or designated as GMT time-based, you can never go back to a local time-based cache
- **ALL** caches (local cache and released caches) must be of the same format

To migrate an old cache, use the following command:

```
CATSysMvCache -o UTC -inputdir input_directory -log file.txt [-outputdir output_directory]
```

If `outputdir` is not set, the cache specified in `inputdir` and the documents it contains are converted to GMT format.

Once a cache has been migrated to GMT time-based, it is no longer compatible with previous releases of ENOVIA DMU or CATIA (i.e. a migrated cache cannot be used in V5R11 or below).

14. Check the **GMT timestamp format** checkbox to designate all caches specified in **Cache Location** as GMT time-based.

Note: When the **GMT timestamp format** option is activated:

- all new caches will be based on GMT time

- all specified caches must have been updated using the provided batch job

If any of the caches specified do not conform to the rule that all caches must be of the same format, an error message will appear.

### Cache Content

When working in visualization mode, the insert component of a CATProduct will generate in the cache a cgr containing Levels of Detail (Lods). This option is activated by default.

15. To deactivate the **Generate cgr with level of detail** option, click the corresponding checkbox.
16. When you've finished defining the different options, click **OK** to confirm.





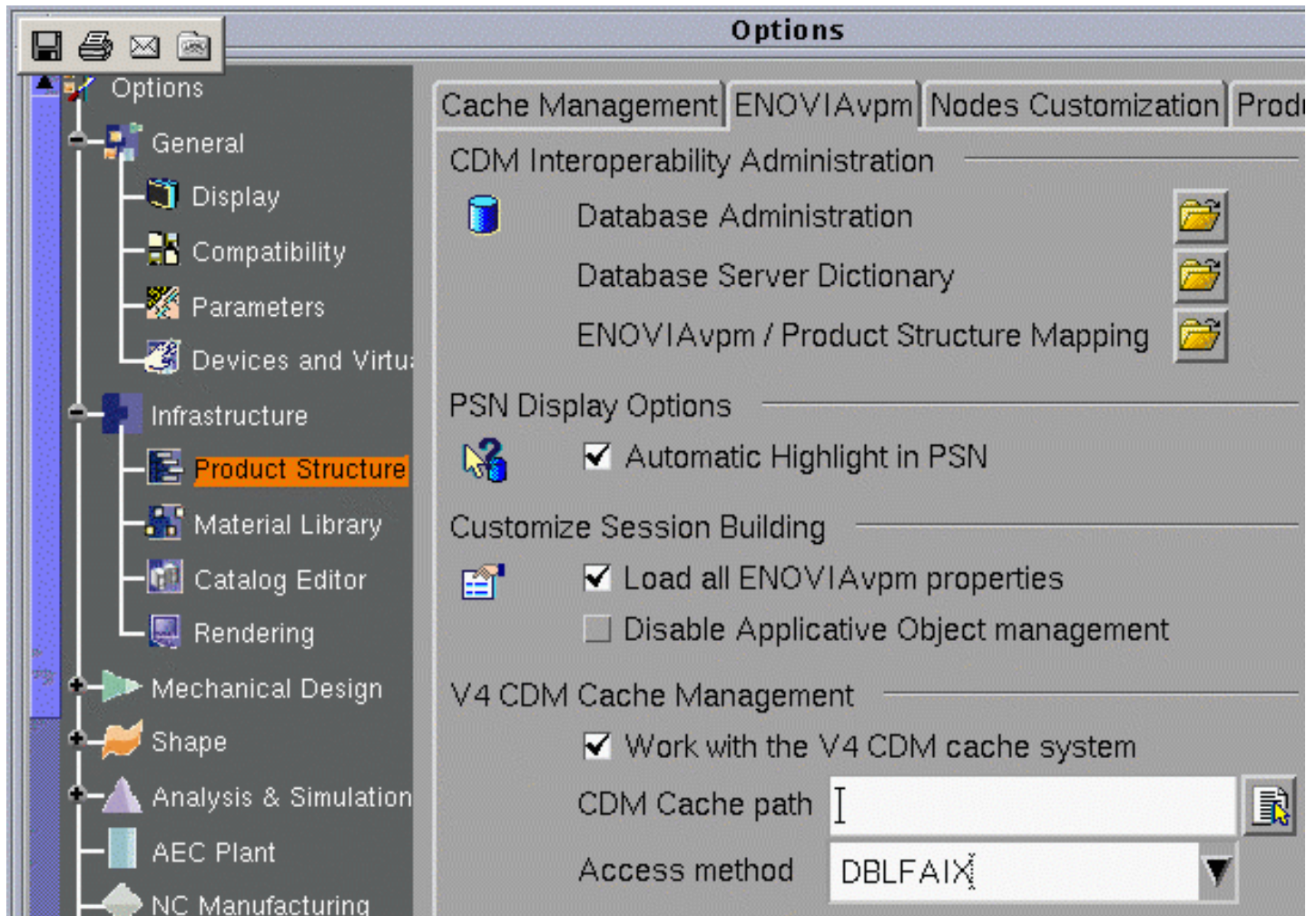
# Customizing CATIA/ENOVIAVPM Interoperability Settings



This task shows you how to customize ENOVIAVPM settings.



1. Select the **Tools->Options...** command. When it appears, select the **Infrastructure->Product Structure** category in the tree on the left.
2. Click on the *ENOVIAvpm* tab. The following dialog box appears:



As you can see, the Options dialog box is divided into four parts:

[CDM Interoperability Administration](#)

[PSN Display Options](#)

[Customize Session Building](#)

[V4 CDM Cache Management](#)

## CDMA Interoperability Administration

There are three pushbuttons in this part:

- Selecting the **Database Administration** button lets you define the parameters for connection to the CDMA databases you wish to use (see "Connecting an ORACLE or DB2 Database to Your CATIA Version 5

Environment" in the *ENOVIAVPM User's Guide*).

- Checking the **Database Server Dictionary** box lets you define a CDMA dictionary associated with a server.
- Checking the **ENOVIAvpm/Product Structure Mapping** box lets you map CDMA and assembly attributes (see "Mapping CDMA and Assembly Attributes" in the *ENOVIAVPM User's Guide*).

### PSN Display Options

There is one checkbox in this part:

- Checking the **Automatic Highlight in PSN** box sets the automatic highlight for the identification of models in a PSN graph from a Version 5 session and vice-versa. This means that whenever you select one or more models in a PSN graph they are automatically highlighted in the Version 5 session. When selected in a Version 5 session, they are automatically highlighted in the corresponding PSN graph.



If you do decide to use this option you should be aware that performance will be adversely affected.

### Customize Session Building

There are two checkboxes in this part:

- Checking the **Load all EnoviaVPM properties** box means that when a session is built all the properties visualized in VPM that have not been mapped will be loaded as user properties.



If you do decide to use this option you should be aware that performance will be adversely affected.

- Checking the **Disable Applicative Object management** box speeds up performance.



If you do decide to use this option you should be aware that V5 applicative objects such as publications, constraints, etc. will neither be saved nor stored in the database.

### V4 CDM Cache Management


There is one checkbox in this part:


- Checking the **Work with the V4 CDM cache system** box enables you to use the cache system and provide the location of the V4 local cache in the field CDM Cache path.

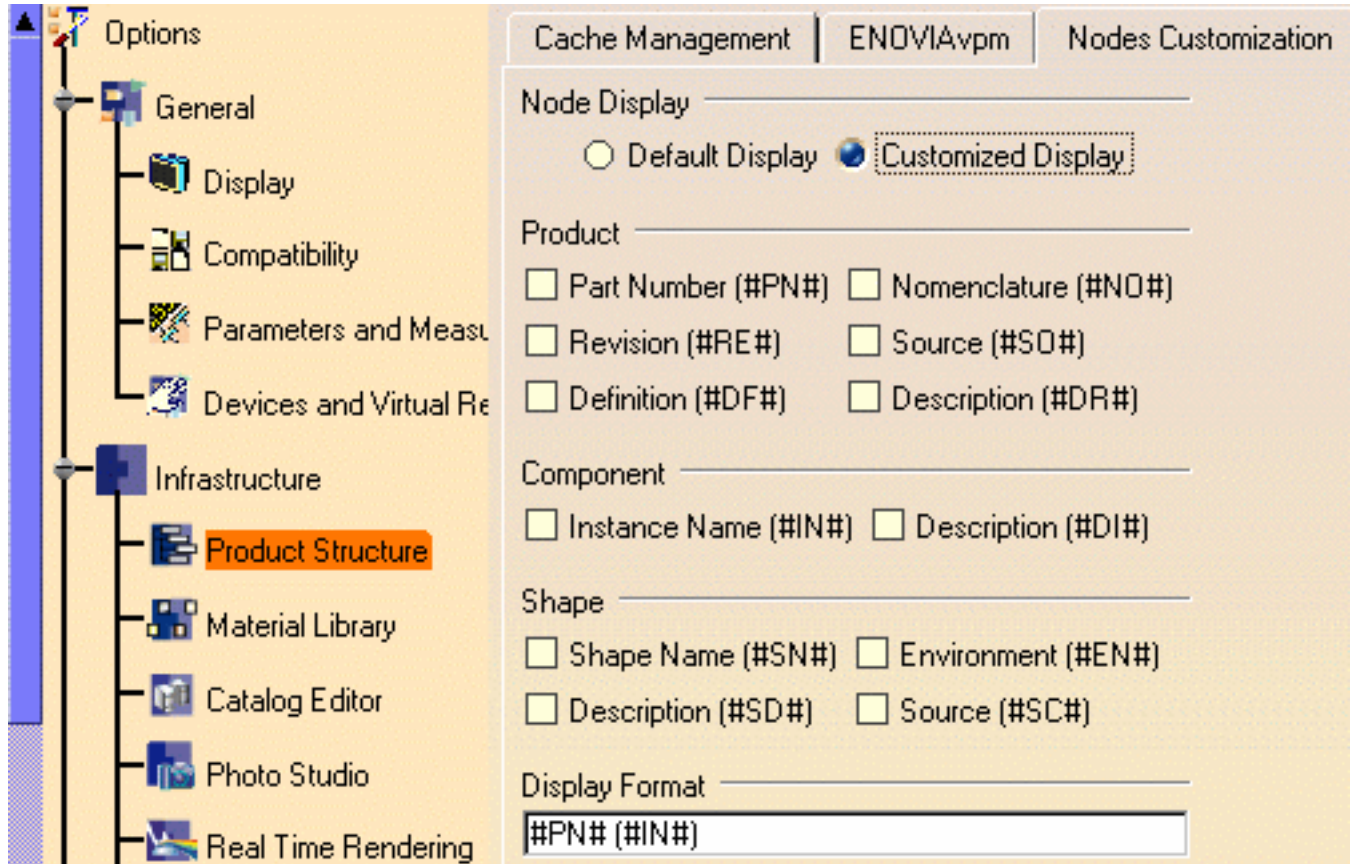
You can also specify the access method in the field **Access method**.




# Nodes Customization

 This task shows you how to give a particular name, reference or description to a product or a component.

-  1. Select the **Tools->Options->Infrastructure->Product Structure** command and click the **Nodes Customization** tab. Check the option **Customized Display** and the settings become active.



2. Check **Customized Display**. Then, you can select different options: Part number (#PN#), Revision (#RE#), Definition, Nomenclature, Source, Instance Name, Description. You can directly replace the diagram (PN, RE,...) between the two # by the term of your choice and it will appear in the Product Structure. By the same way, you can add a comma and a blank space between these parameter.

 If you are using CATIA "with cache system", many informations that you can usually display on each node are not available for the CATPart nodes when opening a CATProduct.

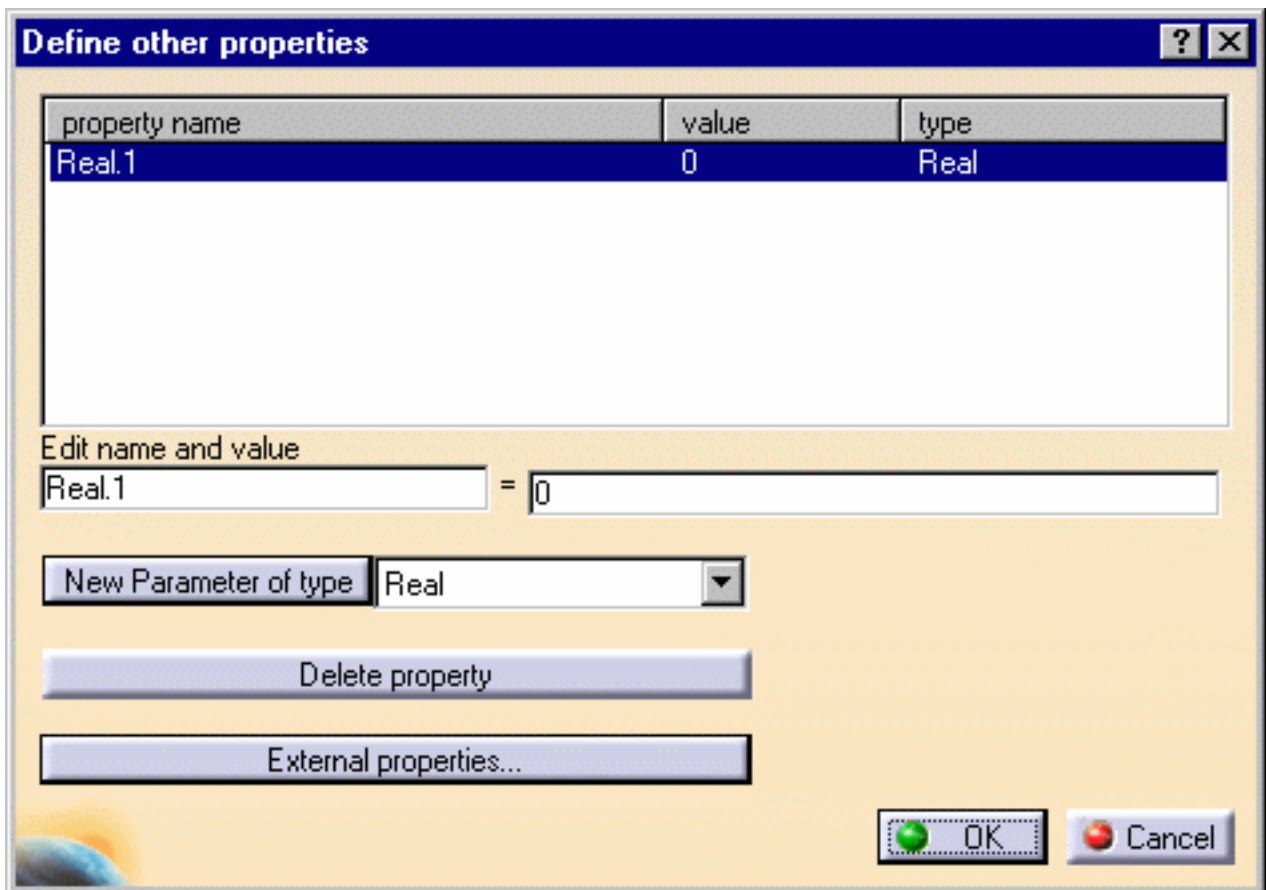
In this case, the unavailable information is replaced by the character string '[?]'. There is one exception : the "Part Number" (#PN#) which is replaced by '[NNNN]' where NNNN is the name of the file containing the CATPart (without the directory name).

Those unavailable informations are made available when you use the "Properties" command on a CATPart. At this moment, the '[?]' strings are replaced by the real values for this particular node in the specification tree.

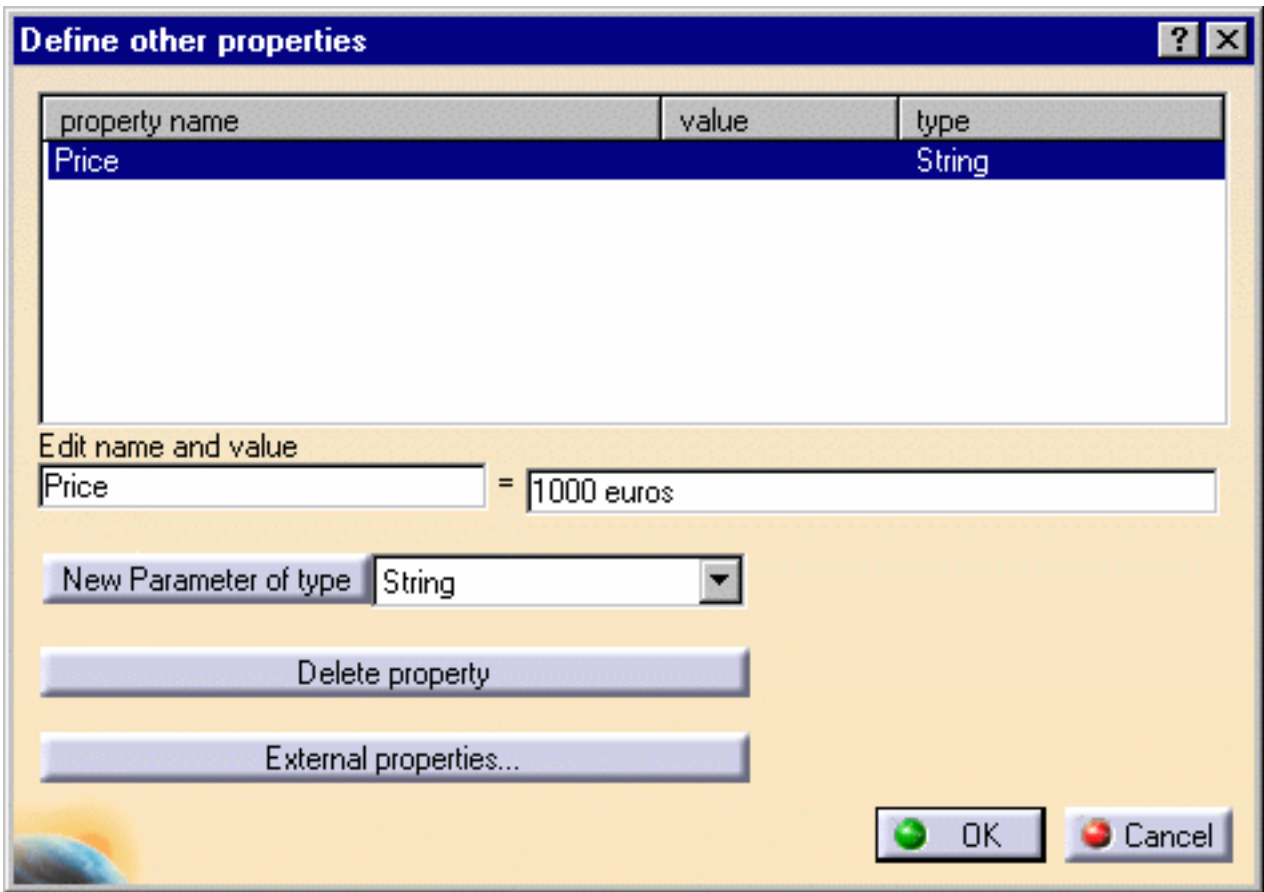
- 3. These attributes are also accessible in the component's contextual menu: **Edit->Properties**.  
For instance, open the following example [ManagingComponents01.CATProduct](#) and select the **Edit->Properties** contextual command of CRIC\_AXIS.CATPart.

 For more information about Component Properties, please refer to Modifying Component Properties in *CATIA - Product Structure User's Guide*.

- 4. Click the **Define other properties** button to add new attributes to CRIC-AXIS.CATPart and click the **New Parameter of type** button. The following dialog box appears:



You can add the "Price" parameter in the **Edit name and value** field:



And you can notice the modifications in the Properties dialog box:

Properties

Current selection : CRIC\_AXIS.1

Product | Graphic | Mechanical

Component

Instance name CRIC\_AXIS.1

Description Component Description

Visualize in the Bill Of Material

Link to Reference

CRIC\_AXIS E:\www\ndsv5r9\PstEnglish\pstug.doc\src\samples\CRI

Product

Part Number CRIC\_AXIS

Revision Revision

Definition Definition

Nomenclature Nomenclature

Source Unknown

Description Component Description

Product: Added Properties

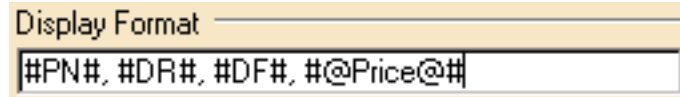
Price 1000 euros

Define other properties...

More...

OK Apply Close

5. Click OK.
6. Select the **Tools->Options->Infrastructure->Product Structure** command and the **Nodes Customization** tab. Check the option **Customized Display** and choose the following settings:



To specify the Price parameter, you need to use these characters: #@xxx@#.

7. Click OK. And you obtain in the following result in the Specification Tree:



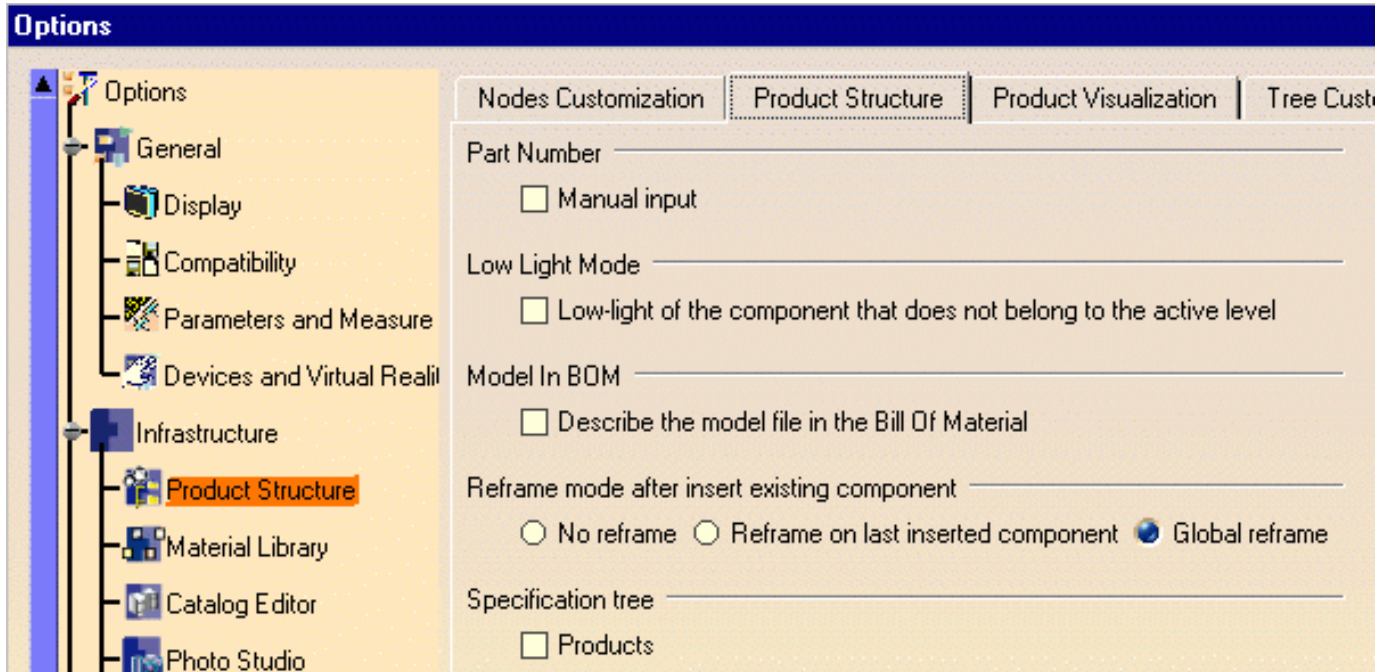
# Customizing Product Structure Settings



This task explains how to customize Product Structure settings.



1. Select the **Tools->Options...** command.
  
2. Select the **Infrastructure** category, then the **Product Structure** sub-category, then the **Product Structure** tab:



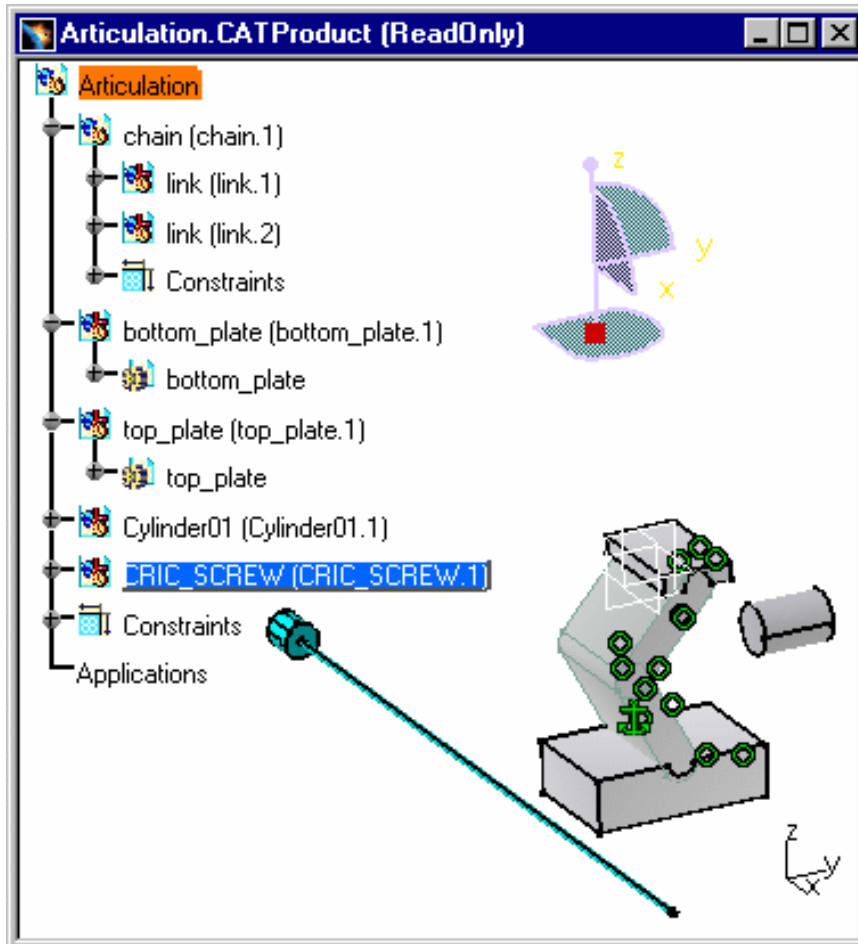
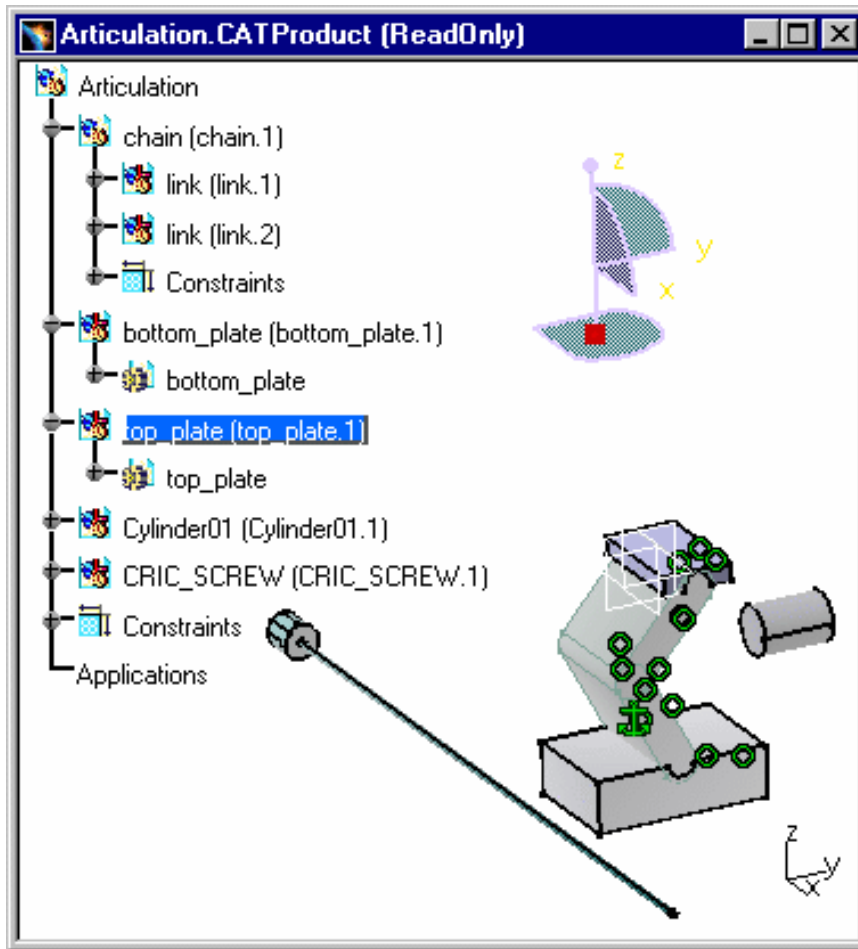
## Defining the Default Part Number of the Component to be Imported

Check **Manual input** if you wish to assign the name you wish to the component you insert into the assembly.

## Selecting the Low Light Mode of the component that does not belong to the active level

If you check the **Low Light Mode** option and select (double-click) a component in a CATProduct, it becomes UI Active (highlighted) in the Geometry Space:





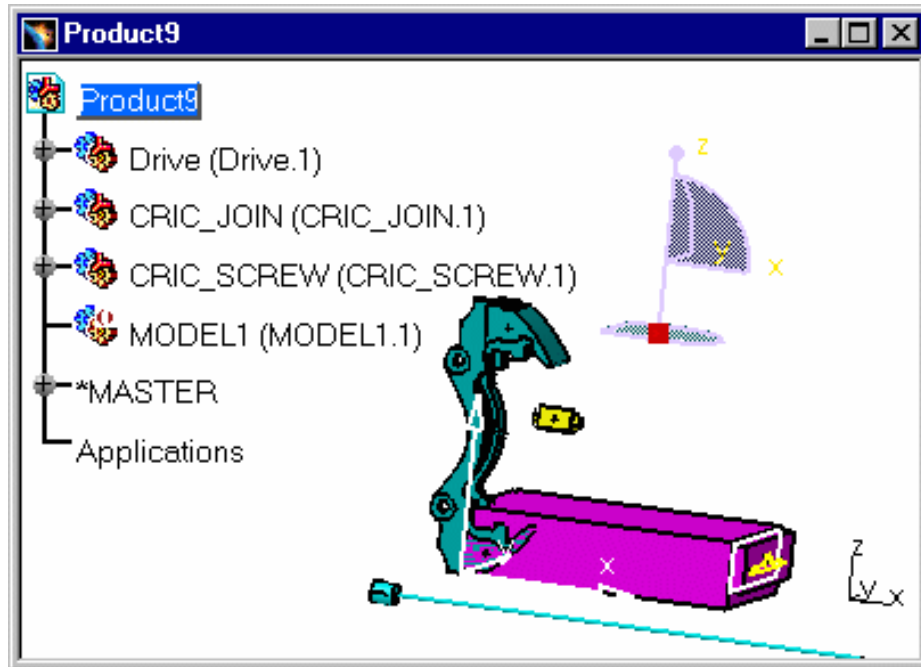
The second example shows that the operation is also possible with Models.

This is a visualization mode : the selected element remains highlighted and the other geometric components are in low light (gray-colored). If you double-click on another element, it immediately becomes highlighted and the other ones are grayed.

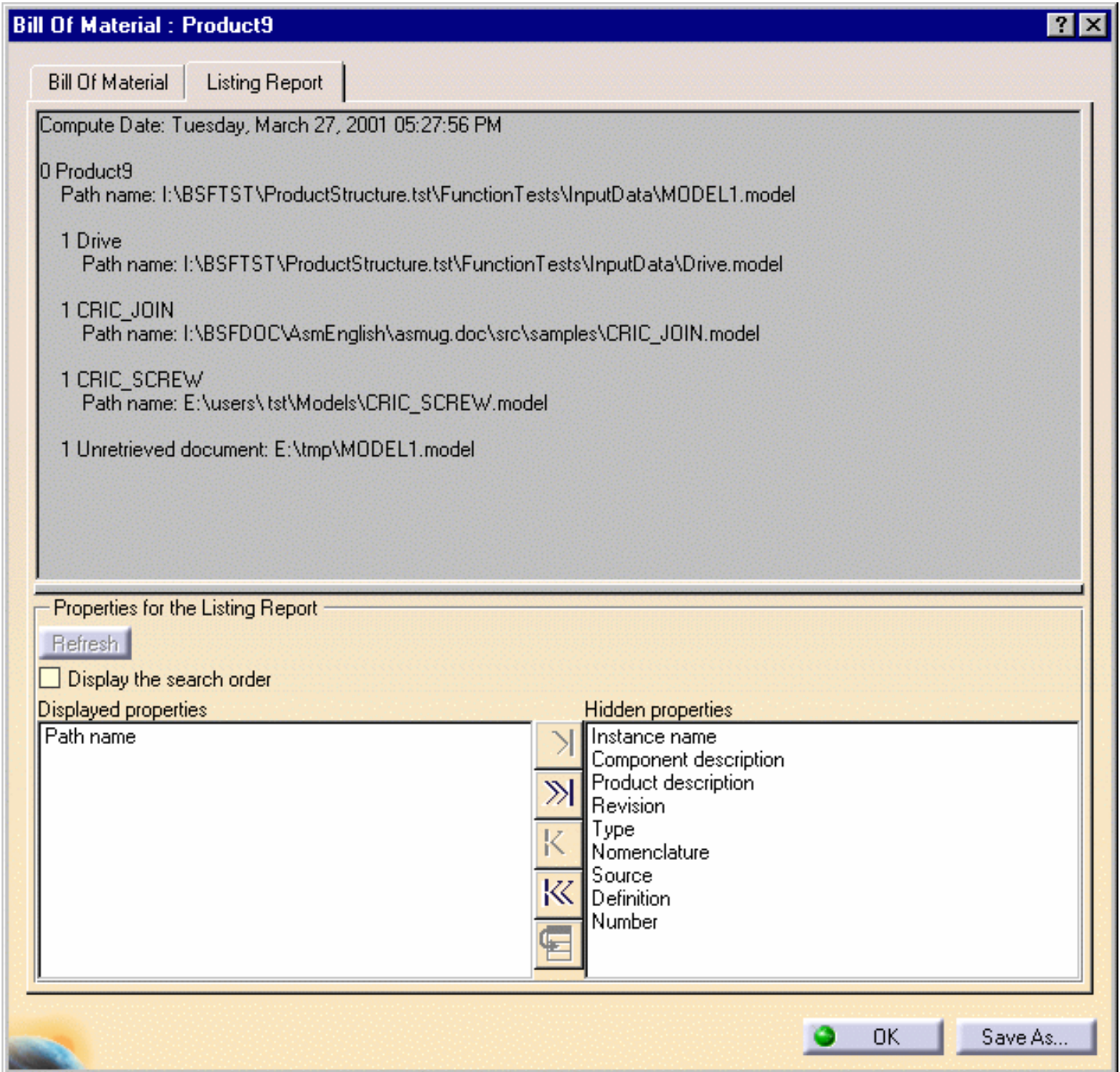
### Describing the model file in the Bill Of Material

Select the **Model In BOM** option in order to have access the Models' information (path name for instance) in the document's BOM.

1. Open a CATIA document containing one or several models.
2. Select the **Analyze -> Bill Of Material** Menu and click the **Listing Report** tab.



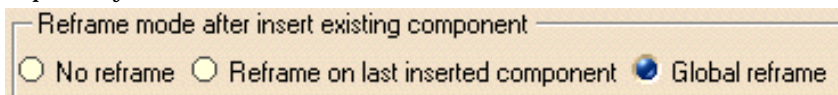
3. Click on the **Refresh** button in order to access to the components' properties (path name for example):



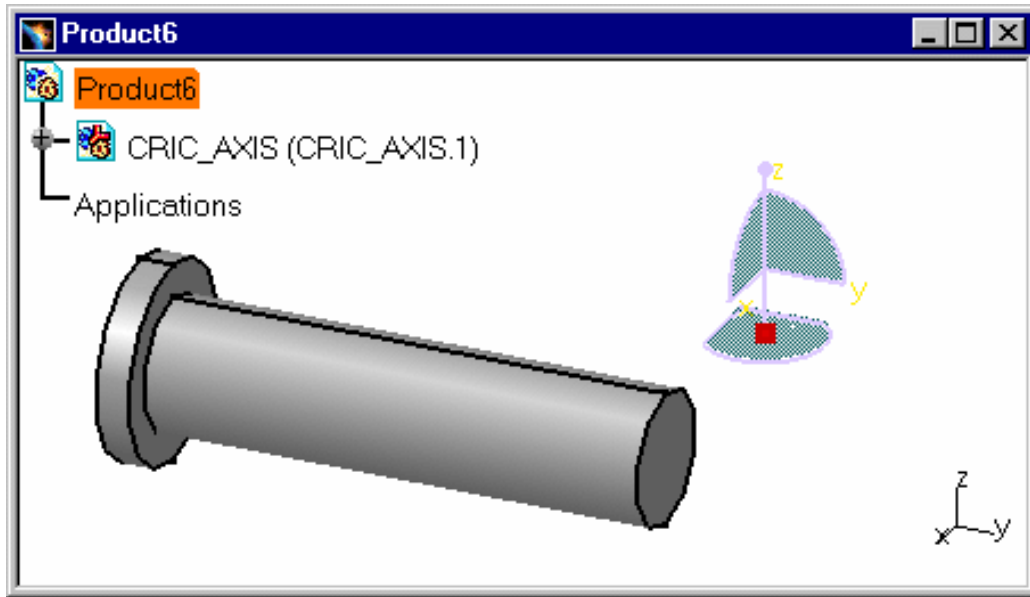
These Models have been inserted into a CATProduct and you can now visualize the Models' path name. You can find the original directories in which the models were stored. When there is a broken link, you can read the following expression : "Unretrieved document".

**Reframe Mode after insert existing component**

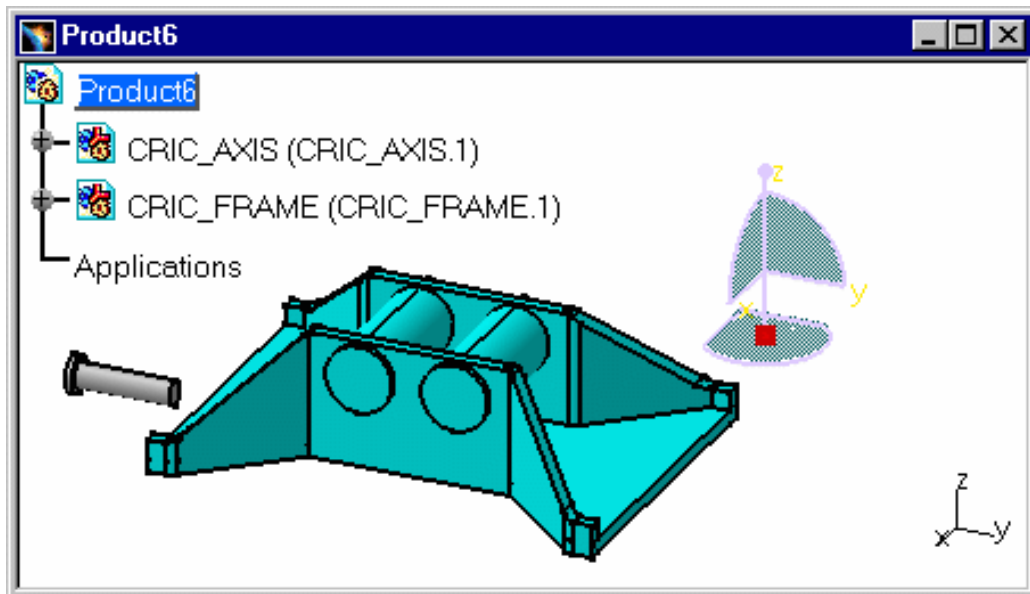
Check the Reframe Mode options you need:



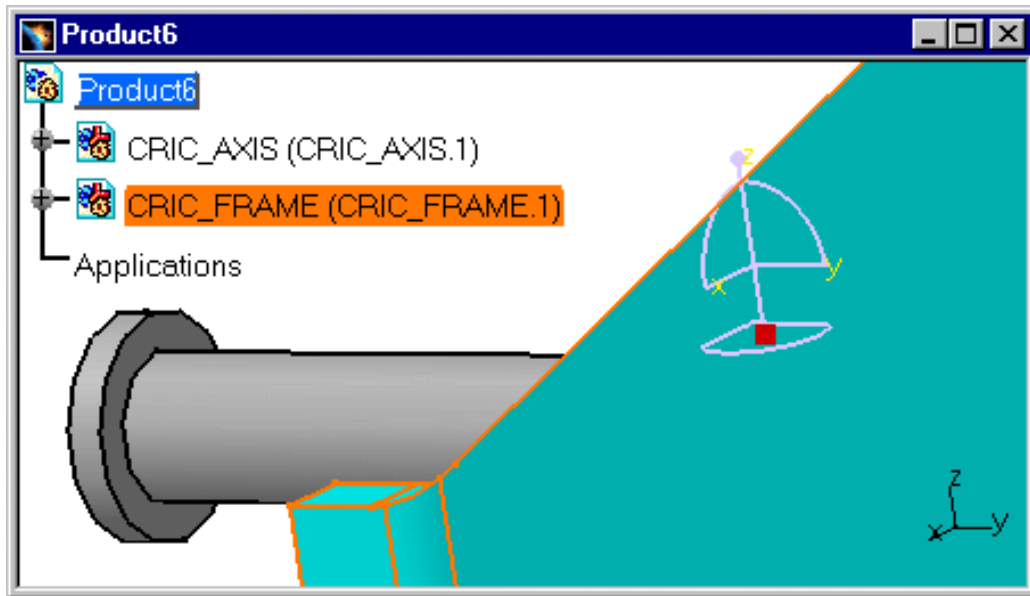
In the following examples, CRIC\_FRAME.1 has been inserted into Product6 and a different Reframe Mode has been selected beforehand. The original CATProduct looks like this:



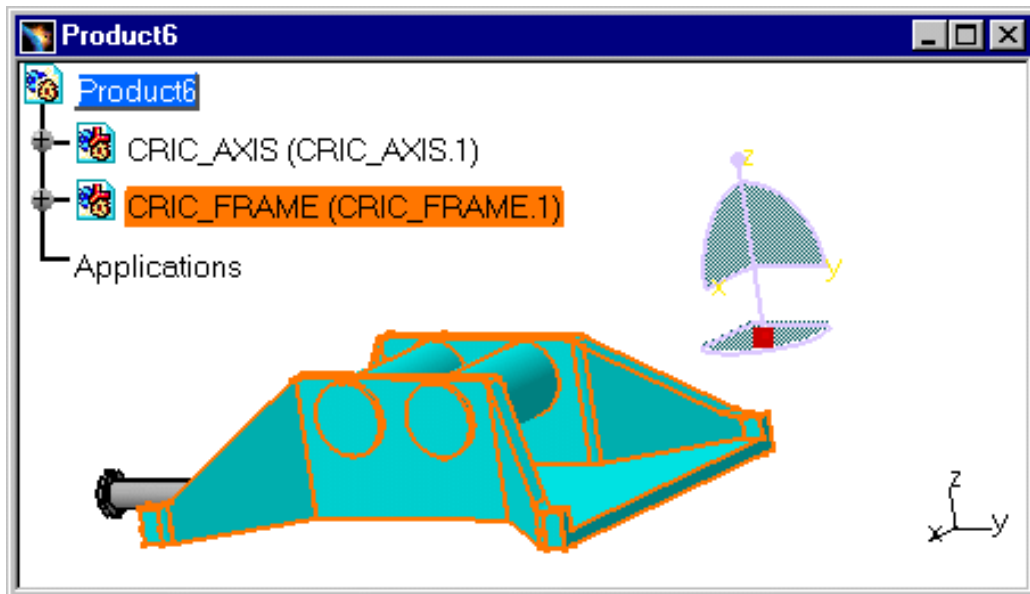
- Global Reframe: this is the default value, Global Reframe zooms on the whole geometry and it allows you to see all the components. After the insertion of CRIC\_FRAME.1 into Product6, the visualization of the geometry is re-adjusted in CATIA window so that all the components can be seen.



- No Reframe: if you insert an element in the product, you may not see the totality of this element in the CATIA window. The visualization on the already existing component, CRIC\_AXIS.1, does not change. There is no Reframe on the last inserted element.



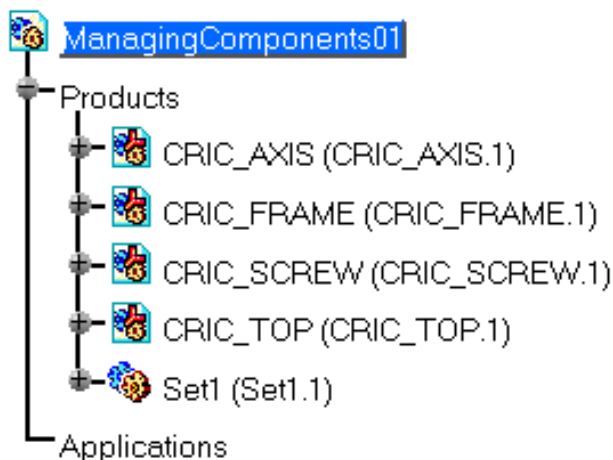
- Reframe on last inserted component: This zoom-in capability improves the visualization of the last inserted component. Therefore, in some cases, you cannot visualize the other CATIA objects because there is an automatic Reframe on the last inserted component.



**Customizing the Specification Tree**

- **Products** to display the products in the Specification Tree.

When the option is activated:

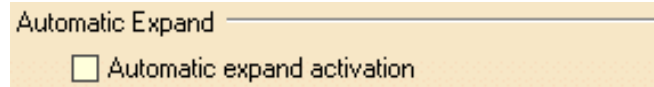


When the option is deactivated:



To display the **Constraints**, **Parameters** and **Relations**, you need go to the [Tree Customization](#) tab.

To modify the setting **Automatic expand**, you need to select the **Tools->Options->General->Display** category and the **Tree** tab:



3. Click OK to confirm.



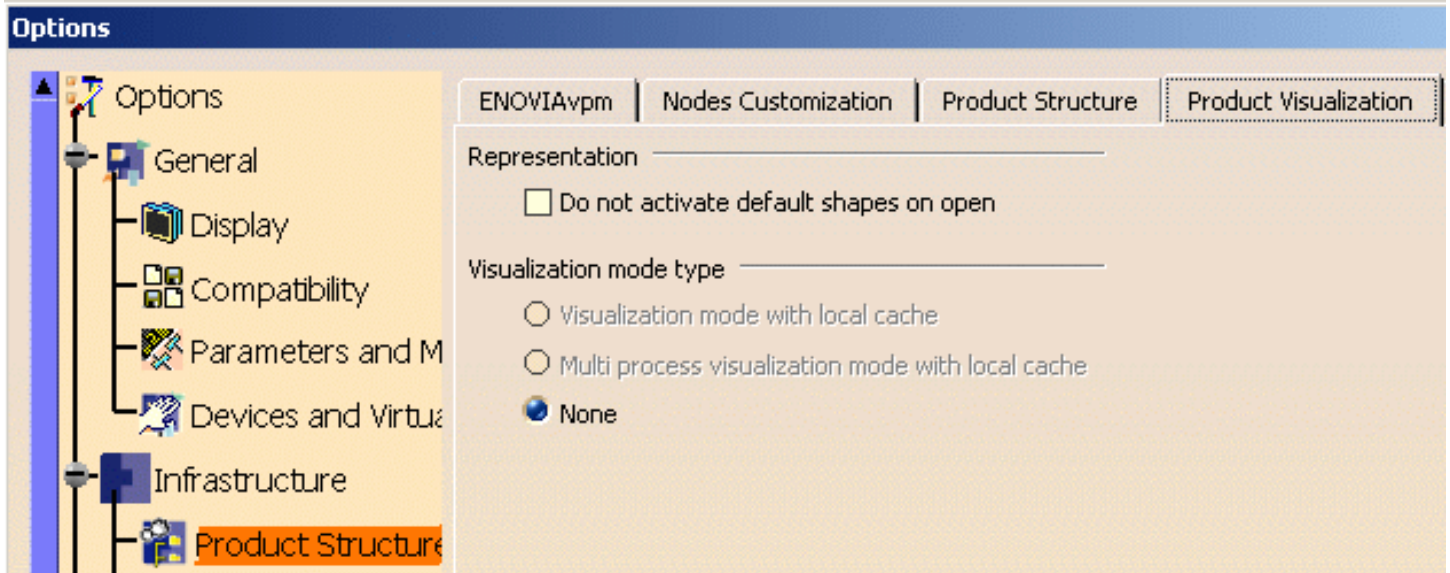
# Customizing Product Visualization Settings



This task explains how to customize Product Visualization settings.



1. Select the **Tools->Options...** command.
  
2. Select the **Infrastructure** category, then the **Product Structure** sub-category, then the **Product Visualization** tab:



## Representation

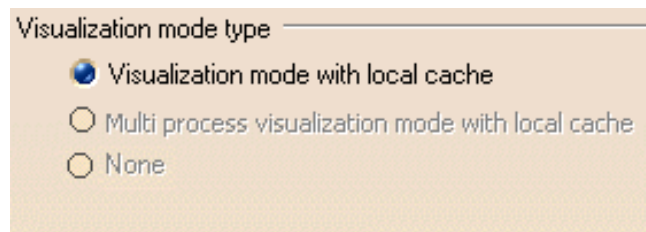
In the "Representation" area, if you click the "Do not activate shapes on open" selection box, the graphical representations of all the components will not be computed when opening a CATProduct. So the only thing that will be displayed is the specification tree.

This option may be used to speed up the opening of large CATProducts when you don't need to see all the components.

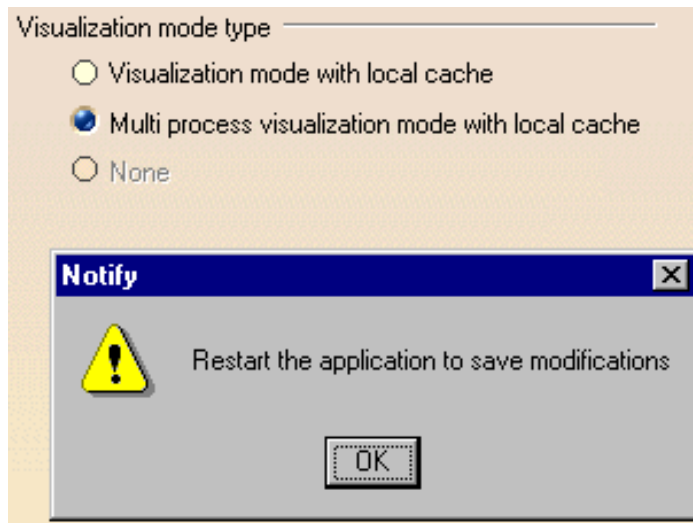
Once the CATProduct is opened, you can display the component you need to see by using the **Representation -> Activate Node** option in the contextual menu of these components.

## Visualization Mode Type

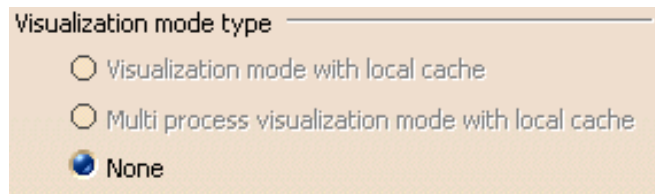
- Visualization Mode with local cache: option by default, when the [Cache](#) is activated (check the **Work with the cache system** checkbox). In this case you create all the cgr files in a local cache in mono process.



- Multi Process Visualization Mode with local cache: when the [Cache](#) is activated. You can choose the creation of Cache files in multi process (DMU context). If you decide to check "Multi Process Visualization Mode with local cache" rather than "Visualization Mode with local cache", the following message appears:



- Visualization Mode without local cache: the cgr files within local Cache cannot be visualized; they are not created in the local cache.
- None: when this option is activated, you are not in Visualization Mode but in Design Mode.




3. Click OK to confirm.



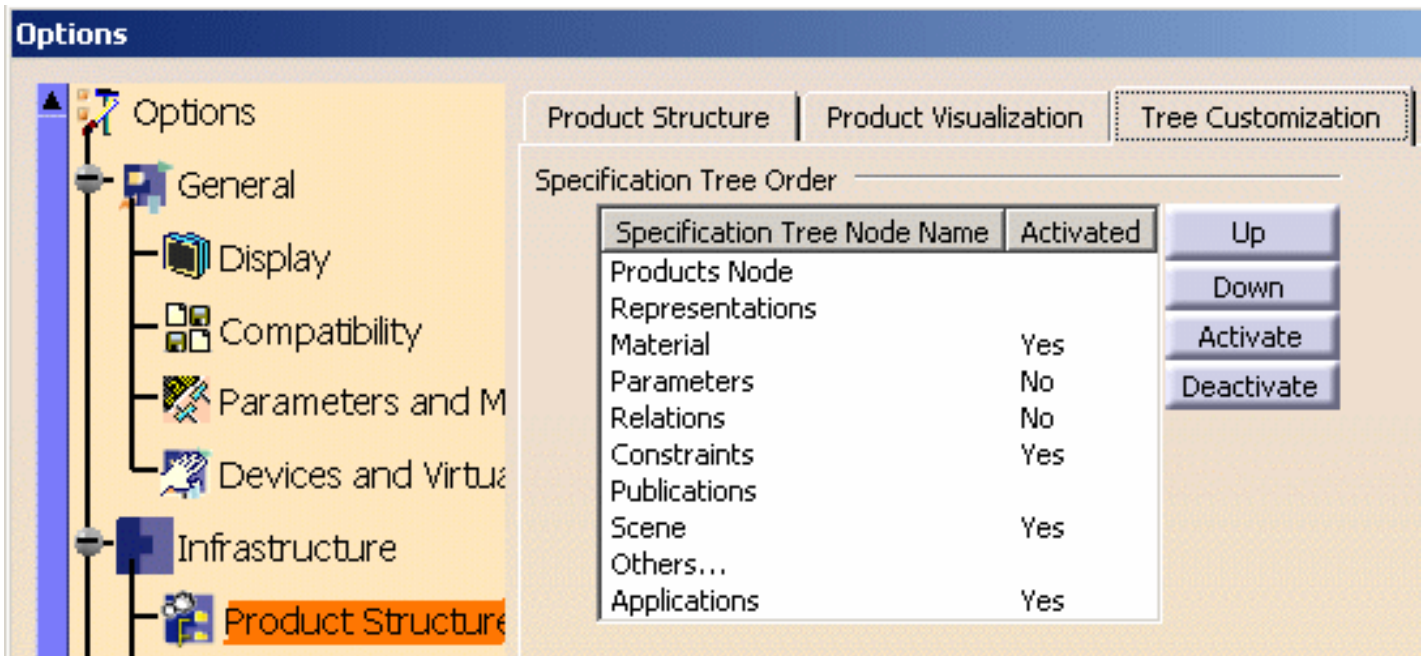


# Tree Customization

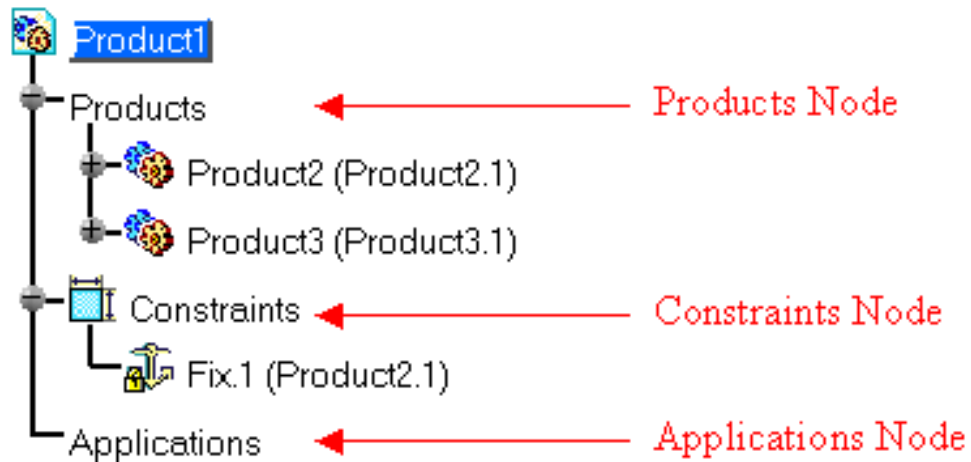
 This task explains how to customize Product Structure settings.

 1. Select the **Tools->Options...** command.

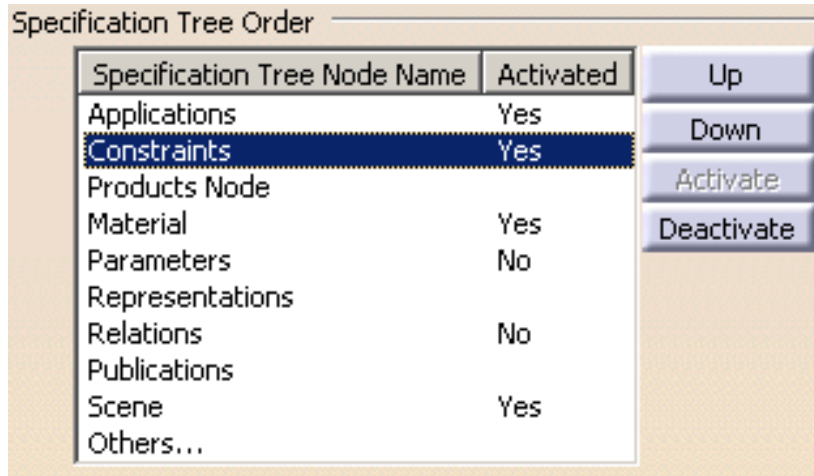
2. Select the **Infrastructure** category, then the **Product Structure** sub-category, then the **Tree Customization** tab:



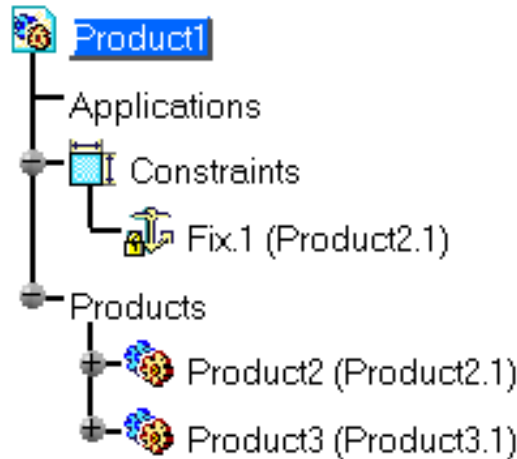
In the **Specification Tree Order** category, there is a list of CATIA attributes corresponding to the nodes in the Specification Tree. This list allows you to order nodes in the Specification Tree and to activate or deactivate their visualization.



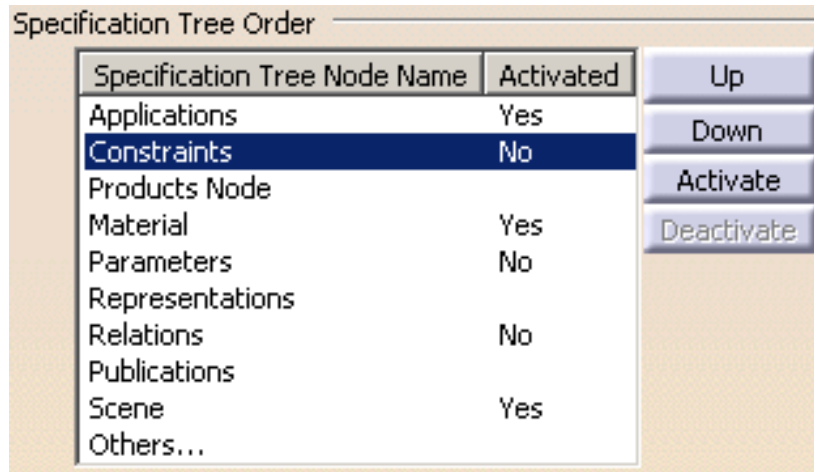
This is an order by default. You can modify this order by selecting a Specification Tree Node Name and clicking the **Up** or **Down** button. You can reorder the whole structure, for instance:



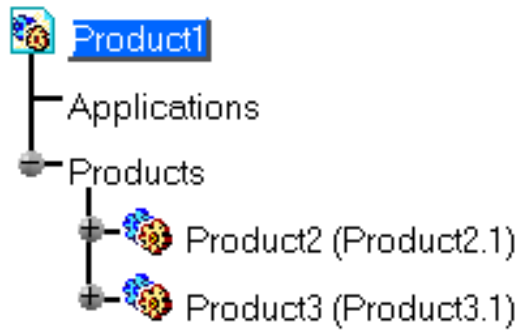
3. Click OK and the option panel is closed. The result in the Specification Tree corresponds to the order you had customized in the setting, for instance:



The column entitled **Activated** allows you to activate or deactivate one or several nodes in the Specification Tree. For instance, you can choose to deactivate the Constraints node:

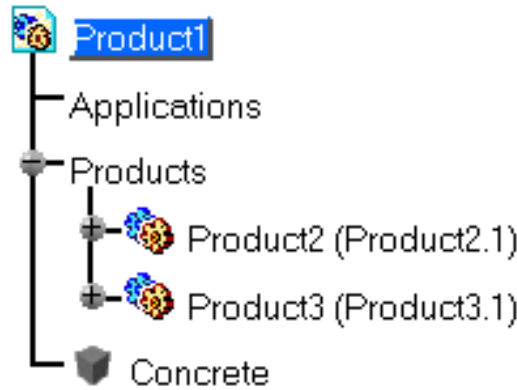


And you obtain:



 Products and Publication Nodes cannot be deactivated.

If you add a new node (for instance, a Material: Concrete) in the Specification Tree, you can activate the corresponding nodes:



You can deactivate this component by the same way:

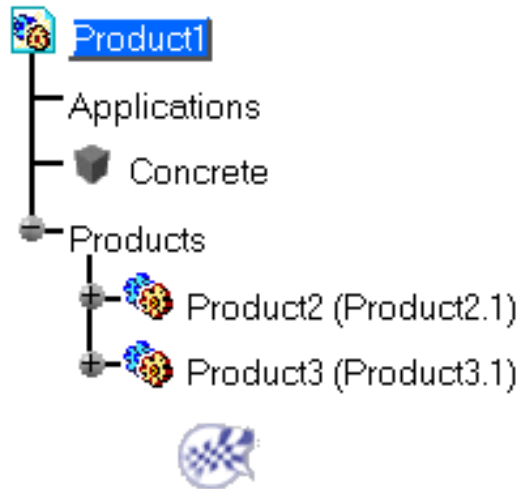
| Specification Tree Order     |           |            |
|------------------------------|-----------|------------|
| Specification Tree Node Name | Activated |            |
| Applications                 | Yes       | Up         |
| Constraints                  | No        | Down       |
| Products Node                |           | Activate   |
| <b>Material</b>              | No        | Deactivate |
| Parameters                   | No        |            |
| Representations              |           |            |
| Relations                    | No        |            |
| Publications                 |           |            |
| Scene                        | Yes       |            |
| Others...                    |           |            |

Or reorder the components within the Specification Tree:

Specification Tree Order

| Specification Tree Node Name | Activated | Up                             |
|------------------------------|-----------|--------------------------------|
| Applications                 | Yes       | Down<br>Activate<br>Deactivate |
| <b>Material</b>              | Yes       |                                |
| Constraints                  | No        |                                |
| Products Node                | No        |                                |
| Parameters                   | No        |                                |
| Representations              |           |                                |
| Relations                    | No        |                                |
| Publications                 |           |                                |
| Scene                        | Yes       |                                |
| Others...                    |           |                                |

And you obtain:



# Customizing For Real Time Rendering 1



This section describes how to customize different settings. All tasks described here deal with permanent setting customization. These tasks are:

## Setting Priority Between Part and Product Material Library

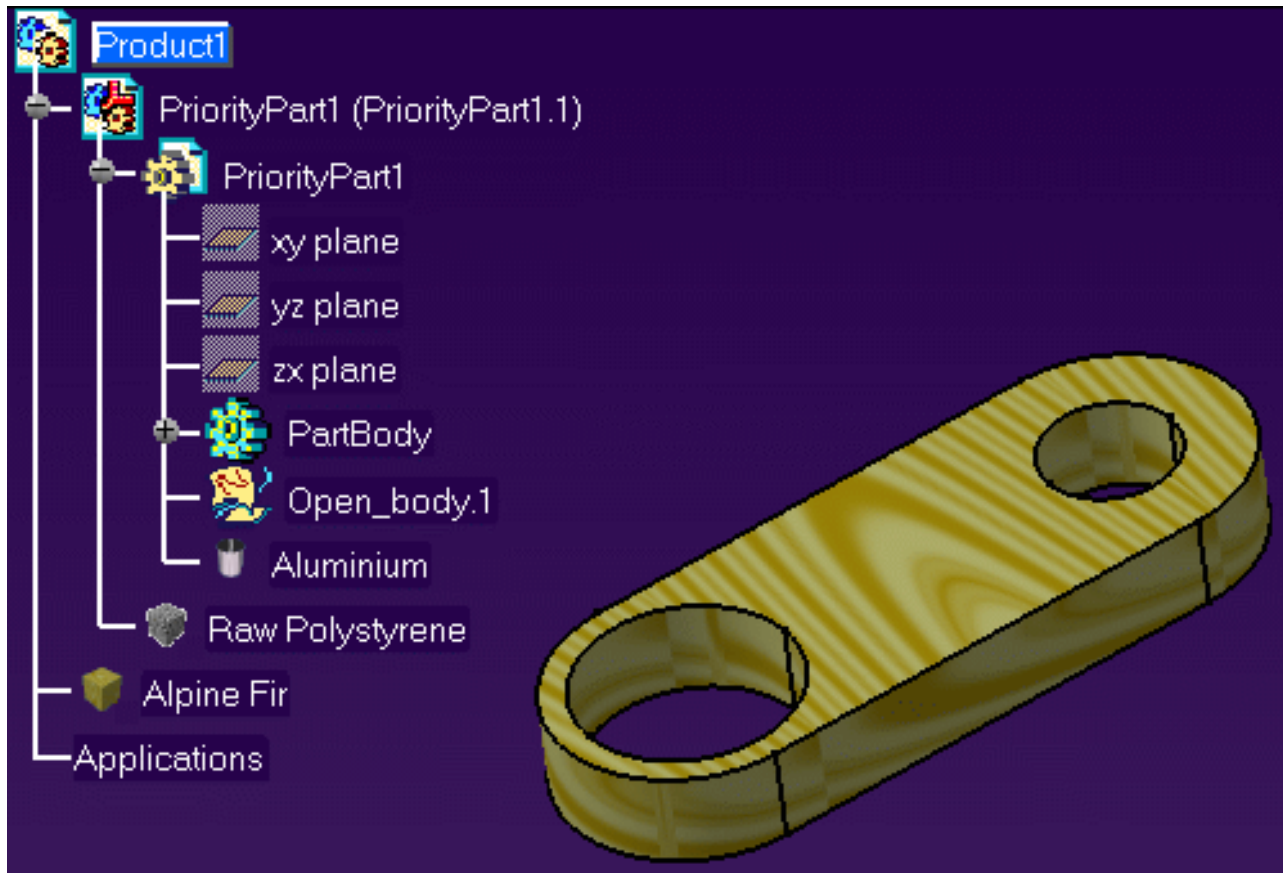
# Setting Priority Between a Part and a Product


 This task explains how to set material visualization priority between the material applied to a Part and another material applied to a Product.

 Open the [PriorityProduct1.CATProduct](#) document.

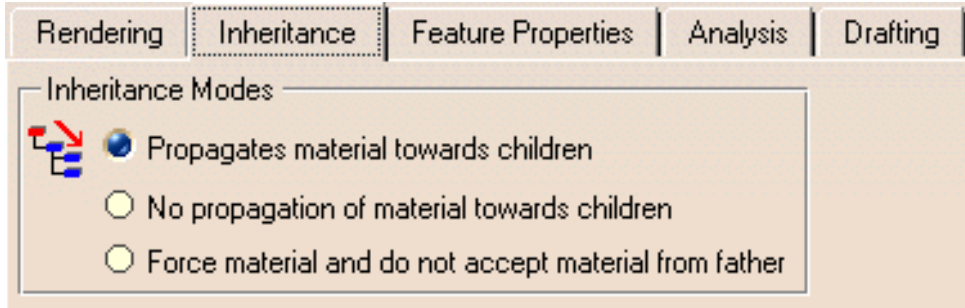
The document contains a Part, a Part instance and a Product onto which three different materials have been applied. By default, the material mapped onto the Product is displayed, having precedence over any other element into the document.

Therefore, in our example, the Alpine Fir material is visible:



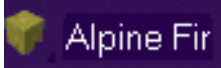
-  **1.** Right-click the Alpine Fir material in the specification tree and using the Properties menu item, display the Properties dialog box.

2. Select the Inheritance tab.




The three available inheritance modes are the following ones:


- **Propagates material towards children:** the material propagates towards lower levels in the hierarchy (default mode).

The corresponding material icon in the specification tree is .

- **No propagation of material towards children:** the material does not propagate.

The corresponding material icon in the specification tree is .

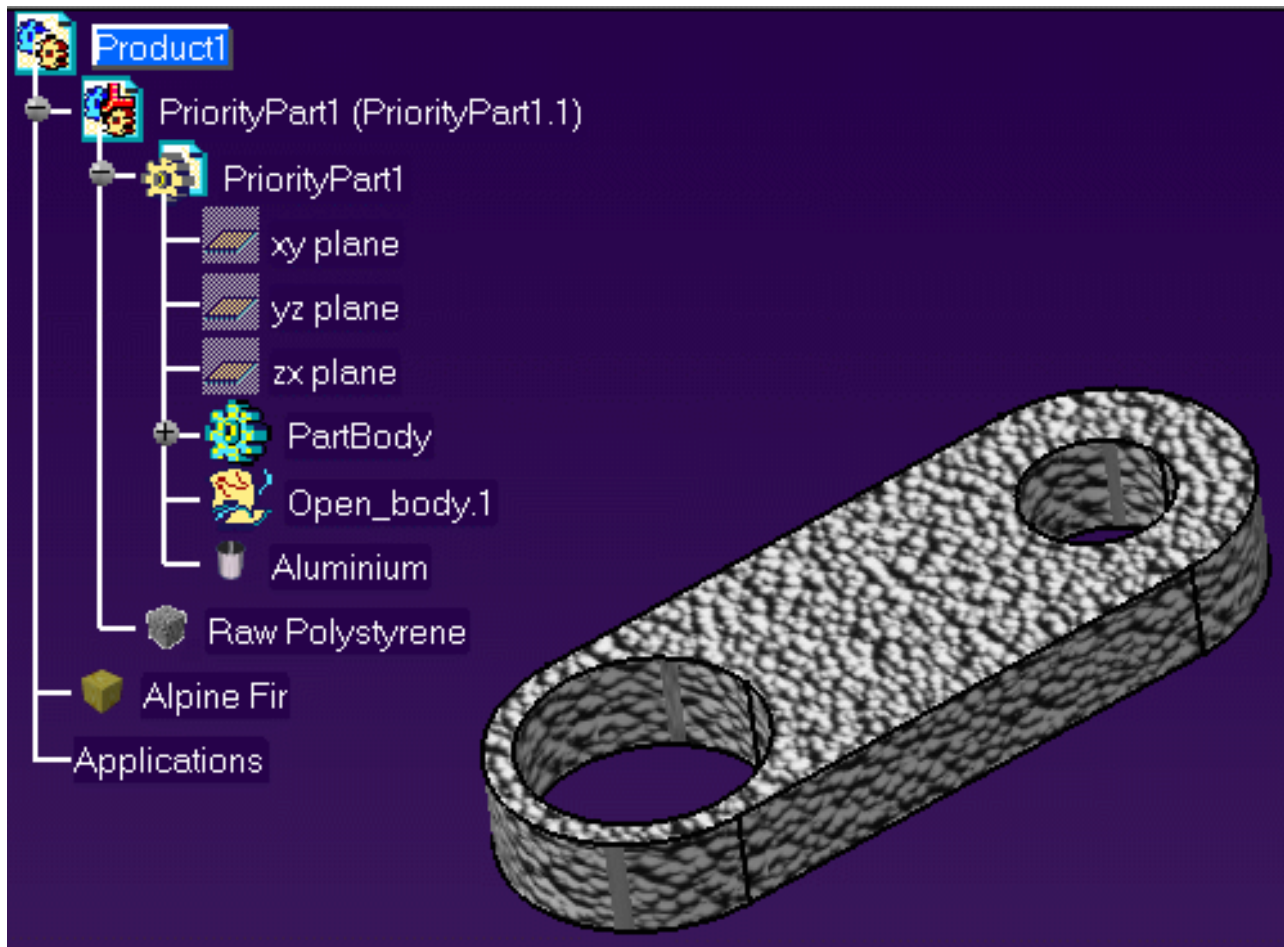
- **Force material and do not accept material from father:** the material is forced, i.e. it is visible even if an upper level material propagates.

The corresponding material icon in the specification tree is .


3. Check the "No propagation of material towards children" option.

The element visualization is instantly modified, and you can now visualize the material mapped onto the Part instance.

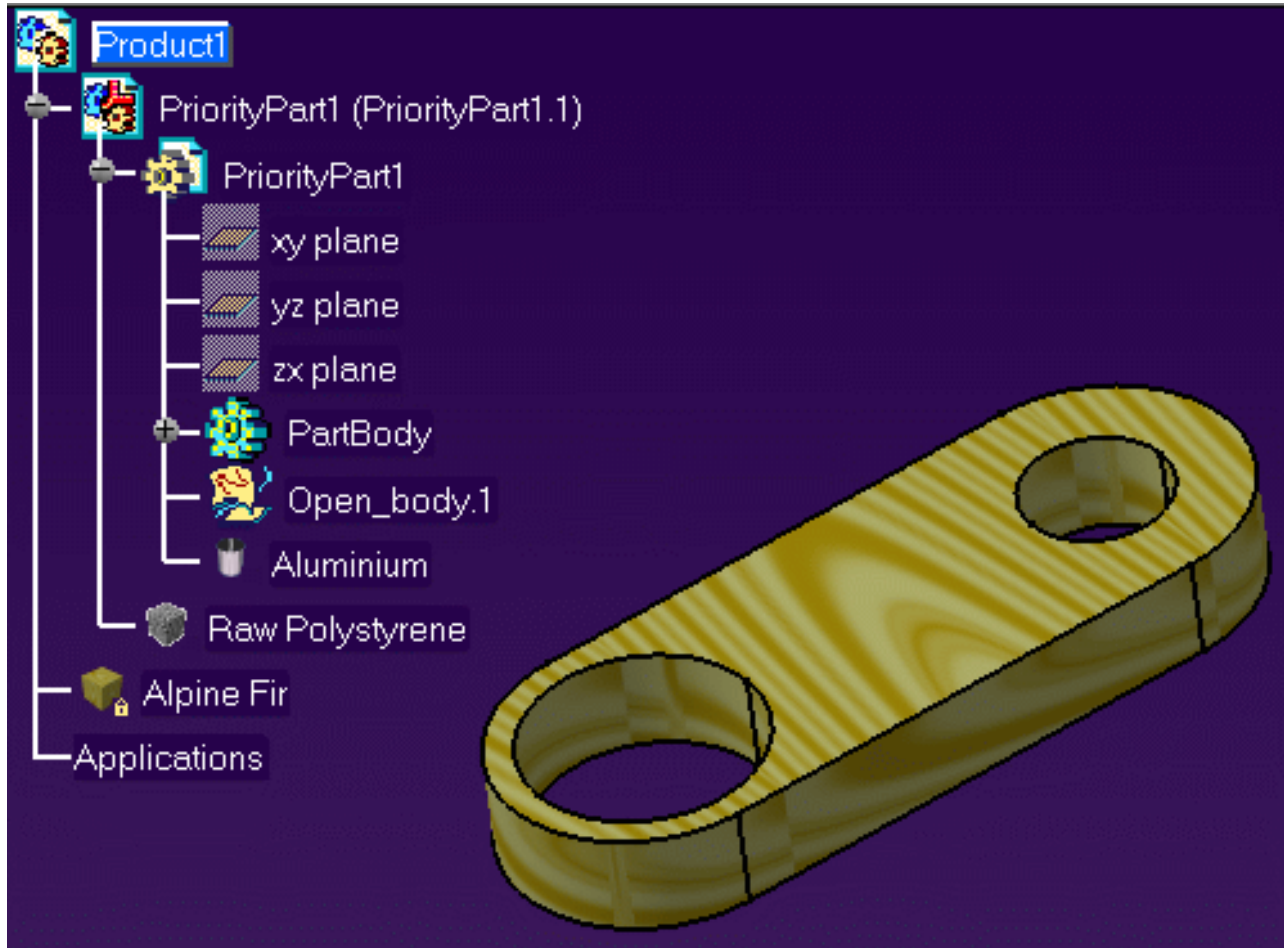
The geometry now looks like this:



4. Click **OK**.

 If you had selected the "Force material and do not accept material from father" option, the geometry would have looked like this:





To be able to visualize the material mapped onto the lowest element in the specification tree (the Part here) you only need to force the material on this element.



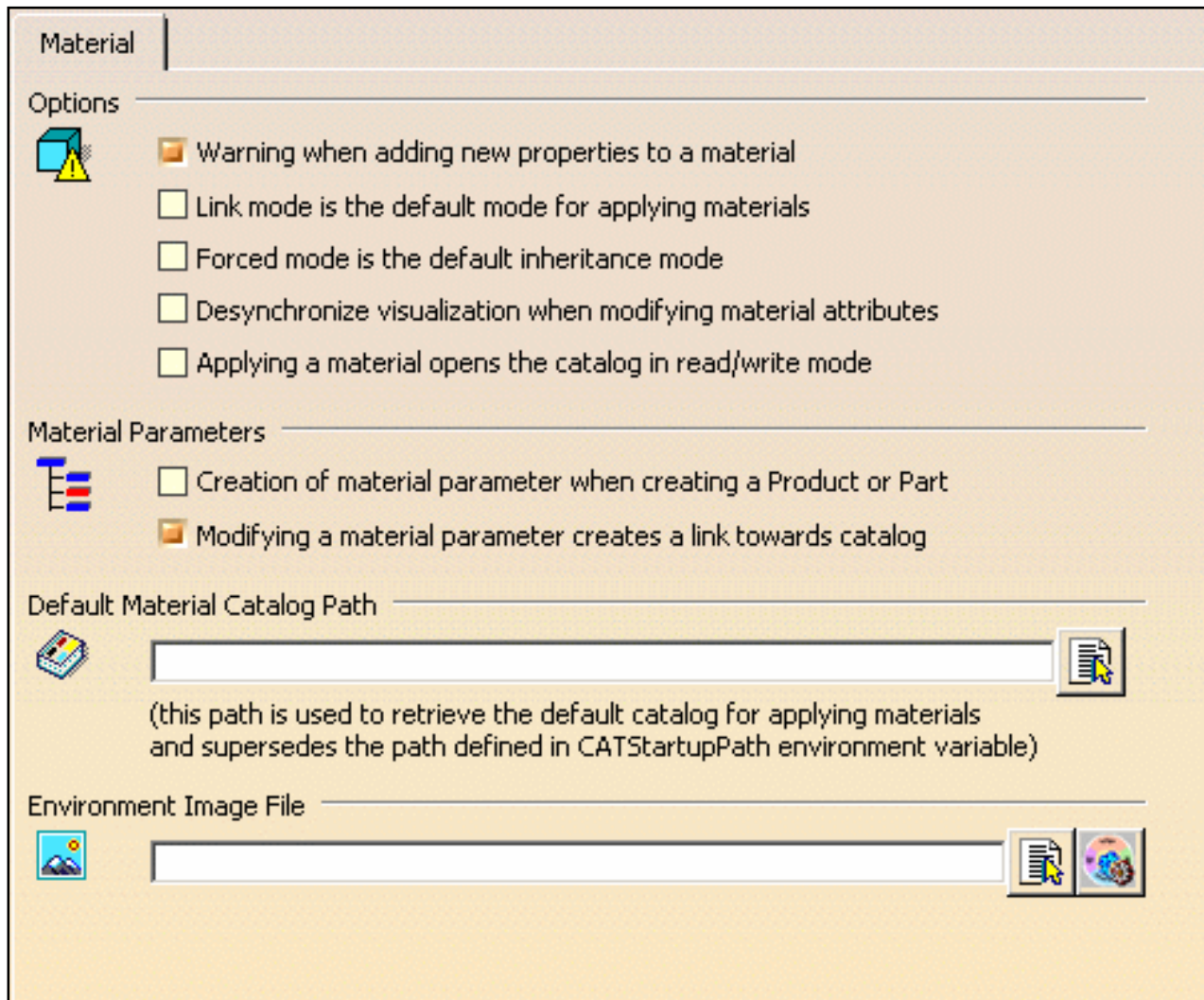
# Material Library

 This task explains how to define material general settings using the **Tools -> Options...** menu item.



**1.** Select the **Tools -> Options...** command.

**2.** Click the **Infrastructure** category then select the **Material Library** subcategory to display the Material tab:

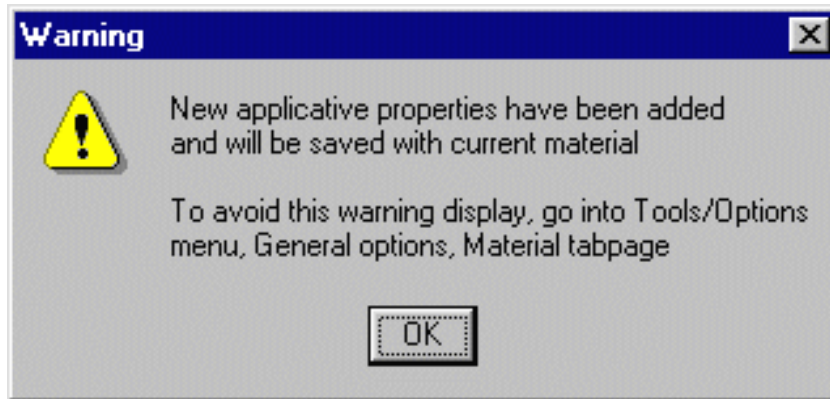


The screenshot shows the 'Material' options dialog box. It is divided into several sections:

- Options:** Contains five checkboxes:
  - Warning when adding new properties to a material
  - Link mode is the default mode for applying materials
  - Forced mode is the default inheritance mode
  - Desynchronize visualization when modifying material attributes
  - Applying a material opens the catalog in read/write mode
- Material Parameters:** Contains two checkboxes:
  - Creation of material parameter when creating a Product or Part
  - Modifying a material parameter creates a link towards catalog
- Default Material Catalog Path:** A text input field with a file icon on the right. Below it, a note states: "(this path is used to retrieve the default catalog for applying materials and supersedes the path defined in CATStartupPath environment variable)".
- Environment Image File:** A text input field with a landscape icon on the left and file and image icons on the right.

**3.** Activate the "Warning when adding new properties to a material".

Once the warning option has been activated, whenever you wish to add properties to a material (by double-clicking on it and choosing the Analysis or Drafting tab for examples) a warning message is issued:



This to let you know that the material have been modified when you accessed the analysis or drafting properties.

4. Check the "Link mode is the default mode for applying materials" option to map the selected material as a linked object and thus, have it automatically updated to reflect any changes to the original material in the library.

**Note:** the link mode is activated whichever method you use to apply the material: click **Apply Material** or use drag & drop capabilities. For more information, refer to "Applying Materials" in the *Version 5 - Real Time Rendering User`s Guide*.

5. The "Force mode is the default inheritance mode" option lets you activate the force inheritance mode, i.e. force a material when applying it onto parts, products, bodies or surfaces in order to make it visible even if a father material propagates.

For more information on inheritance modes, refer to Setting Priority between Part and Product in the *Version 5 - Real Time Rendering User`s Guide*.

6. Use the "Desynchronize visualization when modifying material attributes" option to modify the material properties independently from the visualization and thus, to avoid visualization refresh which can take a long time when working on big models.

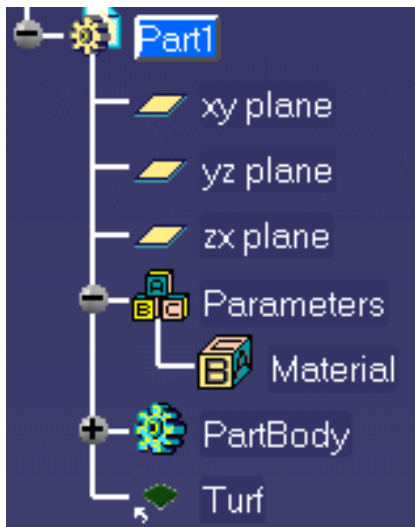
- 7. The "Applying a material opens the catalog in read/write mode" option enables you, for any applied material, to open and modify the corresponding material library.
- 8. When checked, the "Creation of a material parameter when creating a Product or a part" option automatically creates a material parameter for any part or product you create (either using the *File->New...* or the *Insert->New Part/New Product* command), even if no material is mapped onto it.

Activating this option also means that the "Material=None" parameter will appear in the specification tree.

- 9. Use the "Modifying a material parameter creates a link towards catalog" option to create (when checked) or not a linked material when modifying a mapped material using Knowledgeware capabilities.

Do not forget to check the "Parameters" option in *Tools->Options->Infrastructure->Part Infrastructure->Display* to visualize the Material parameter.

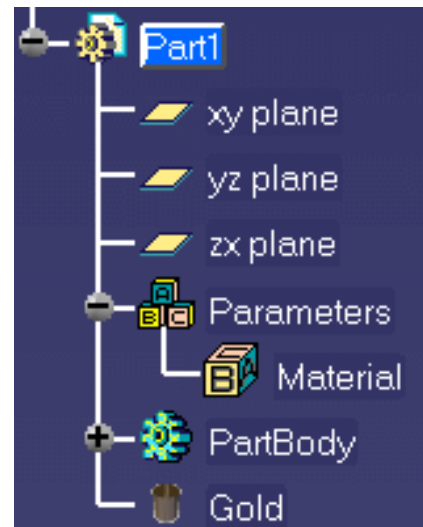
In the two examples below, the Material parameter was selected in the specification tree then modified via the Edit Parameter dialog box (using the *Material object->Definition...* contextual command):



*Example 1*

*Option "On"*


*Material applied as a linked material*



*Example 2*


*Option "Off"*

*Material applied as a non-linked material*

- 10.** The Default Material Catalog Path area lets you browse your file tree using the  icon to select the material catalog to be used by default when applying materials.


This material catalog will supersede the one defined in the CATStartupPath environment variable and thus, will be retrieved when selecting "Default Material Catalog" from the Library dialog box.

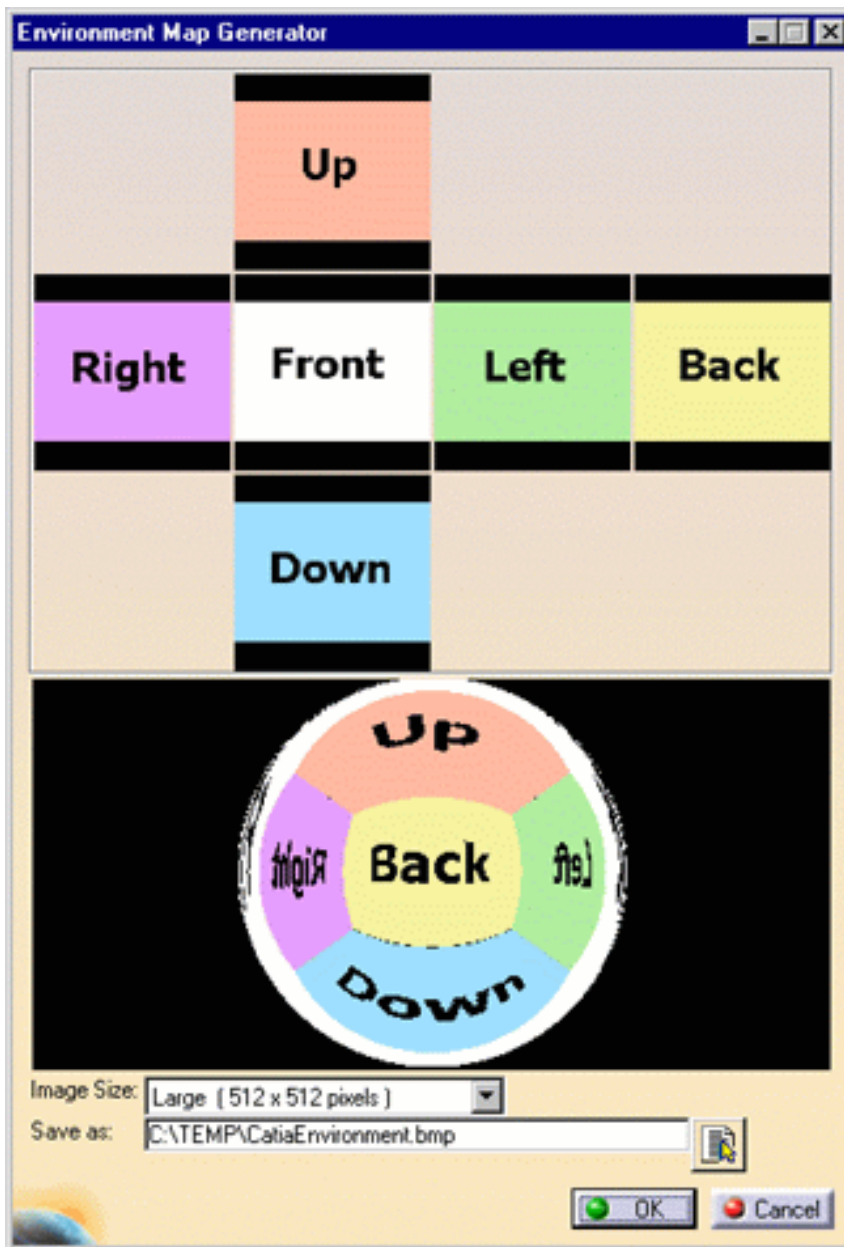
- 11.** The Environment Image area enables you to use your own environment instead of the default one provided.

Simply enter the path to the user-defined environment or use the  icon to open the File Selection dialog box which lets you browse your folders to the desired location.

**P2**

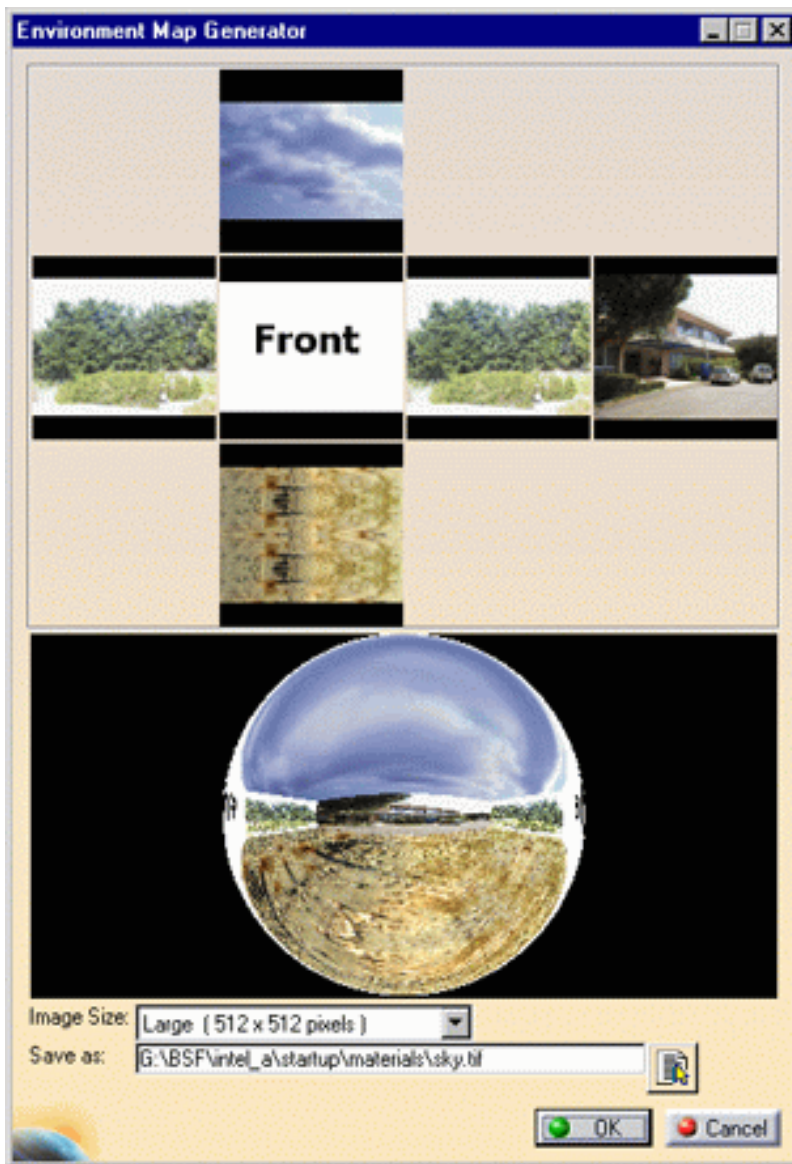
**Or**

Click the Environment image generator  icon to open the Environment Map Generator dialog box (this icon only appears when working with a Real Time Rendering 2 license):




In the upper part of the dialog box, just click each environment wall (i.e. "Up", "Back", etc.) then navigate to the desired image using the File Selection dialog box.

The resulting environment will be displayed on the environment map as shown below:



The Image Size pulldown list lets you choose a Small, Medium or Large size for your environment image.


Once the image is generated, enter its name and path in the "Save as" field or click the  icon to open the File Selection dialog box which lets you browse your folders to the desired location.

Click **OK** to validate and go back to the Material tab.


**12.** Click **OK**.

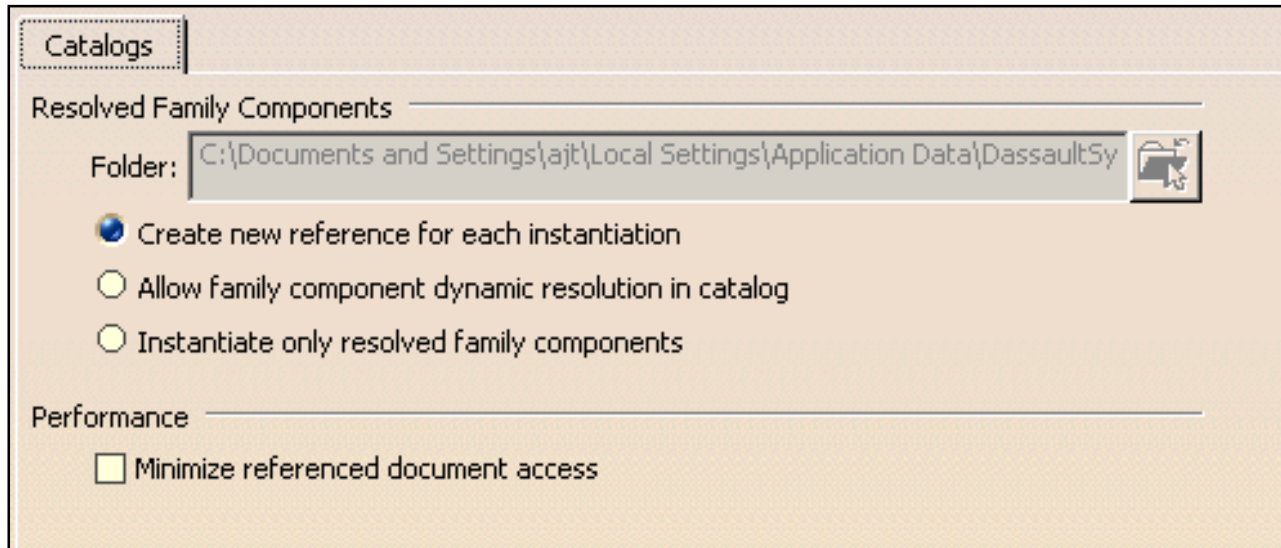


# Customizing Catalogs

 This task explains how to define Catalog Editor settings using the **Tools -> Options...** menu item.


 These settings are relevant for catalogs with design tables only.

-  **1.** Select the **Tools -> Options...** command.
- 2.** Click the **Infrastructure** category then select the **Catalog Editor** subcategory to display the Catalogs tab:



- 3.** In the Folder field, enter the path to the folder in which you will store your resolved parts along with their configurations.

This avoids recalculating the part resolution whenever you wish to use a part in a configuration.

 You can also click the icon to the right to open the Folder Chooser dialog box which lets you browse your folders to the desired location.



4. Use the "Create new reference for each instantiation" option to create a component reference from each instantiated reference part (just like the **New from...** command). In other words, each component instantiation will result in the generation of a .CATPart document saved in the Storage Folder you specified.
5. The "Allow family component dynamic resolution during instantiation" option lets you dynamically resolve components when instantiating them.
6. The "Instantiate only resolved family components" option lets you instantiate only resolved and stored components. Otherwise, an error message appears to warn you that the component cannot be resolved.
7. In the Performance area, checking the "Minimize referenced document access" option improves performance since if the preview of a catalog component is not stored in the catalog, this preview will not be displayed (either in the Catalog Browser or in the Preview tab).  
Checking this option also means that pointed catalogs will not be loaded and that the links to these catalogs will not be checked. As a consequence, you will not be aware of broken links, if any.  
However, note that you can still display the preview of a pointed element by double-clicking it in the list.
8. Click **OK** to validate.



# Customizing For Photo Studio

This section describes the different types of setting customization you can perform. All tasks described here deal with permanent setting customization.

These tasks are:



- Display
- General
- Output
- Satellites
- Stickers

# Display

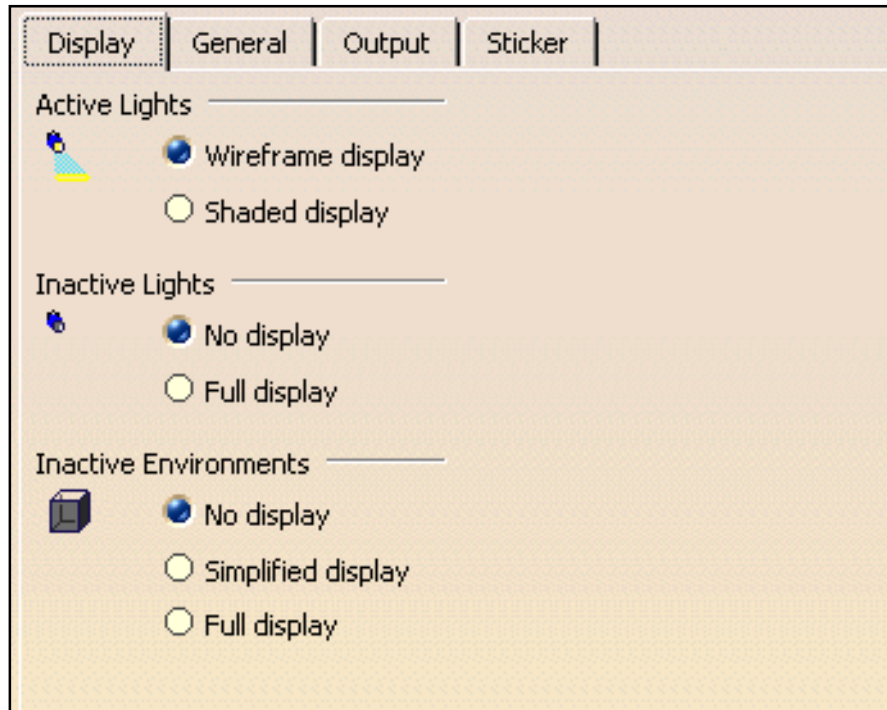


This task explains how to customize Rendering display settings.



1. Select the **Tools->Options...** command then in the **Infrastructure** category, click the **Photo Studio** sub-category.

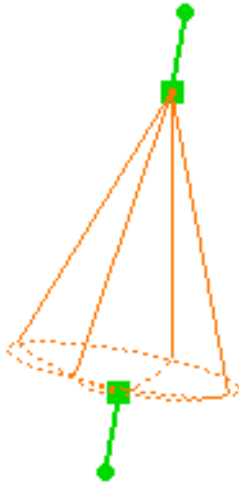
The **Display** tab appears:



## Setting Display for Active Lights

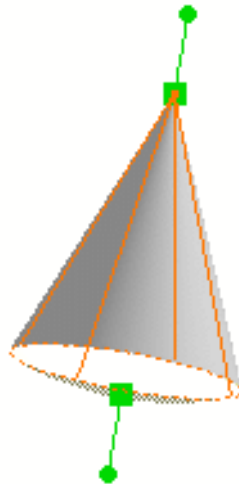
This area lets you control the display of active lights, whether they be standard or surfacic light sources:

- check the "Wireframe display" option to display the representation of the light source in wireframe mode. This is the default mode:



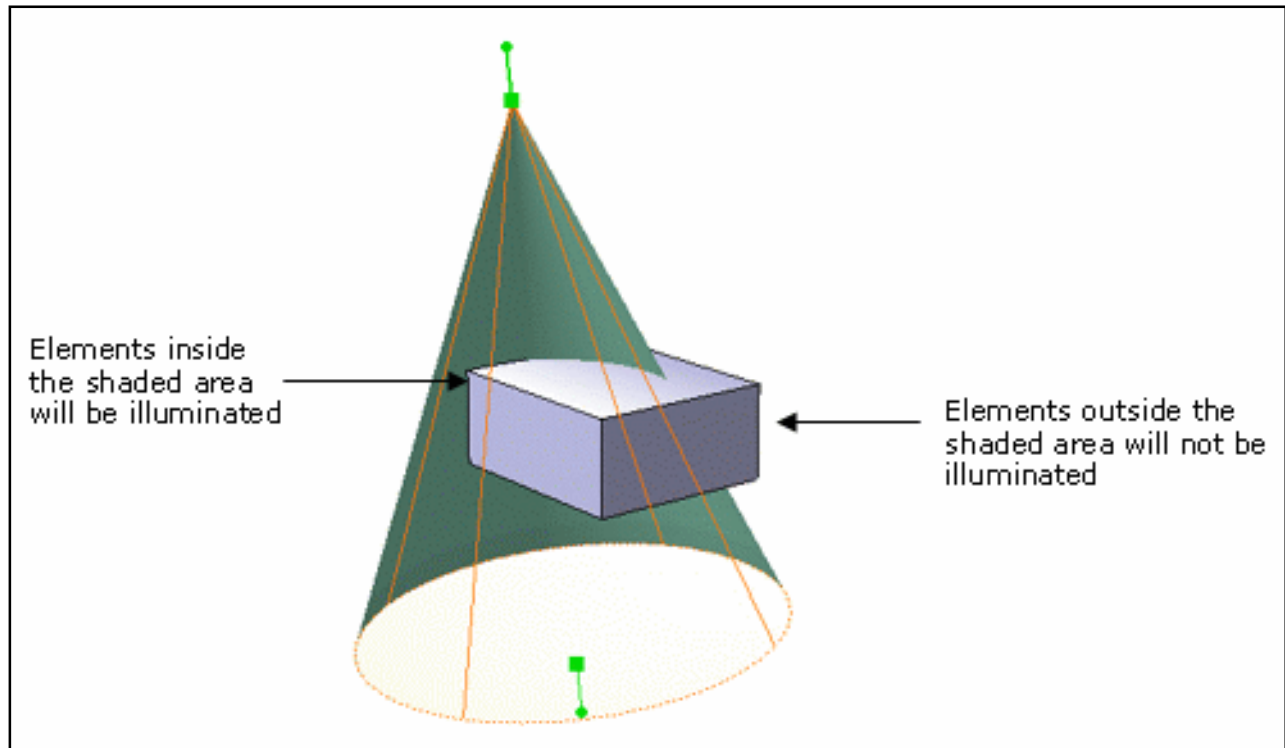
*Spot light source:  
wireframe display*

- check the "Shaded display" option to display the representation of the light source in shading mode as shown below:



*Spot light source:  
shaded display*

You can use the manipulators to interactively position the light source as you would do for the default wireframe representation. This display helps you visualize more easily which part of the object will be illuminated as shown below:



Note that the color of the shaded representation corresponds to the color defined for the light source (via the Lighting tab in the Properties dialog box). In the above example, the light intensity has been assigned a green color.

### Setting Display for Inactive Lights

This area lets you control the display of inactive lights:

- check the "No display for inactive lights" option to turn off inactive light display
- check the "Full display for inactive lights" option to turn on inactive light display.

### Setting Environment Display

This area lets you to set the type of environment display:

- check the "No display for inactive environments" option to turn off inactive environment display
- check the "Simplified display for inactive environments" option to display inactive environments in a simplified way
- check the "Full display for inactive environments" option to turn on inactive environment display.

Note that dynamic reflections are no longer supported.

2. Click **OK** to validate your settings.



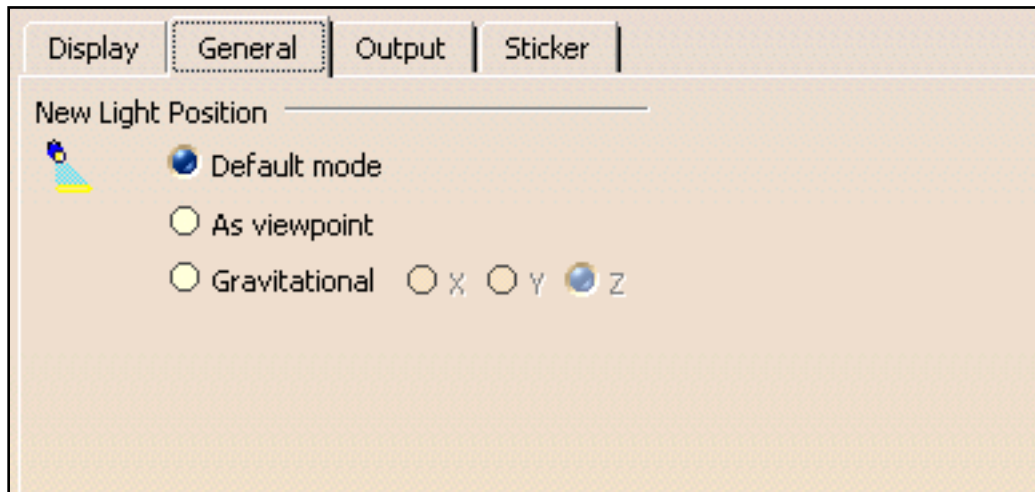
# General



This task explains how to customize general settings for light sources.



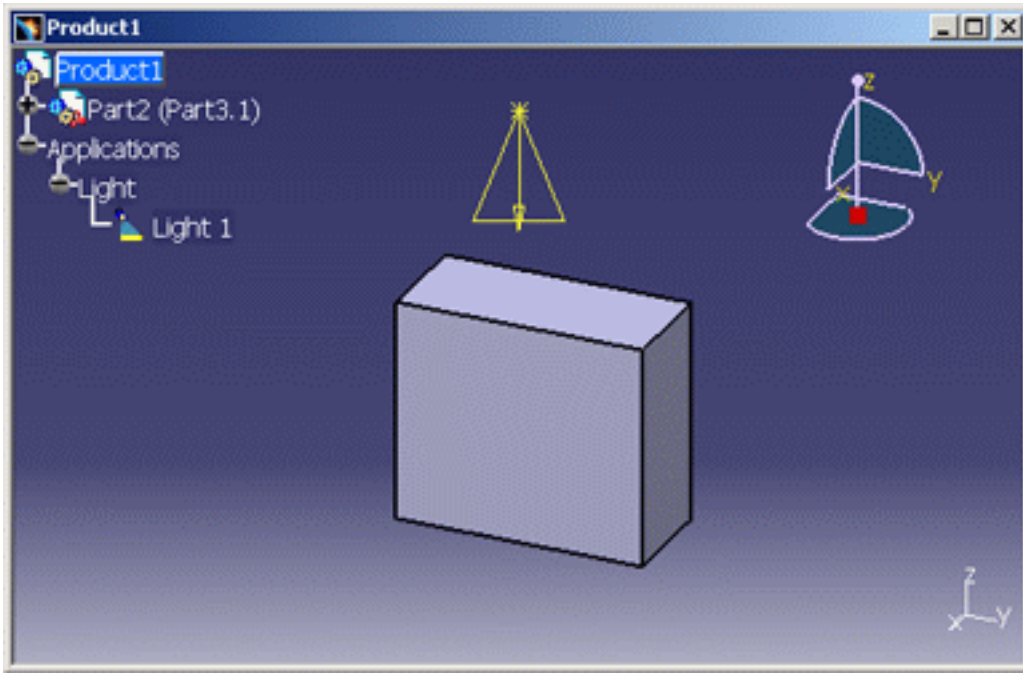
1. Select the **Tools->Options...** command then in the **Infrastructure** category, click the **Photo Studio** sub-category.
2. Click the **General** tab:



## New Light Position

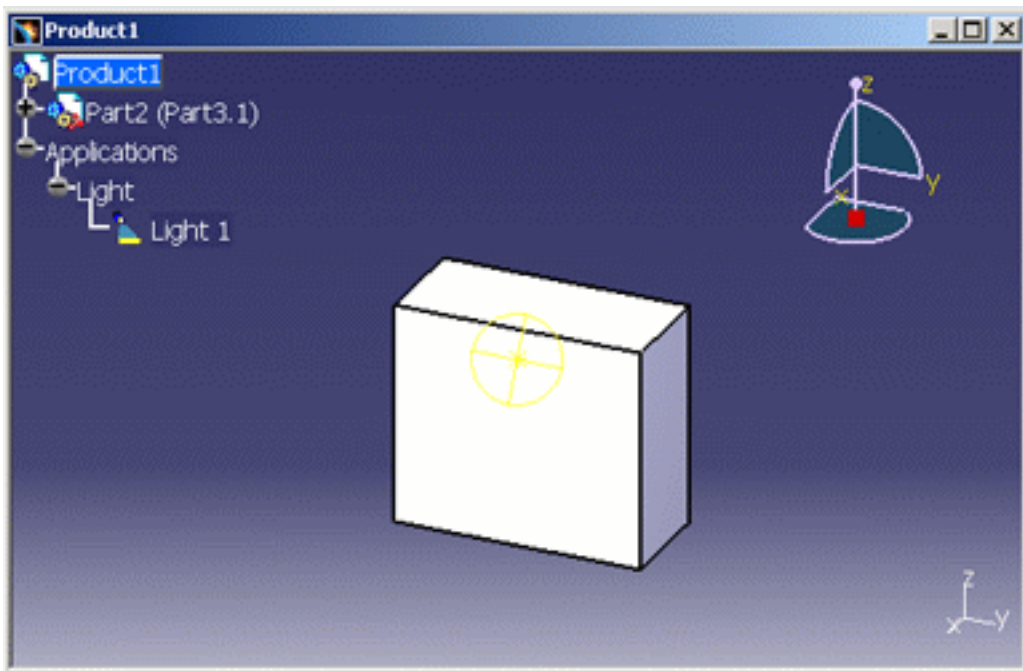
This area lets you define how the new light sources you will create will be positioned. You can choose between three modes:

- **Default mode**  
The light source will be positioned in the upper part of the geometry area, oriented down and centered:



Spot light source - "Default" mode

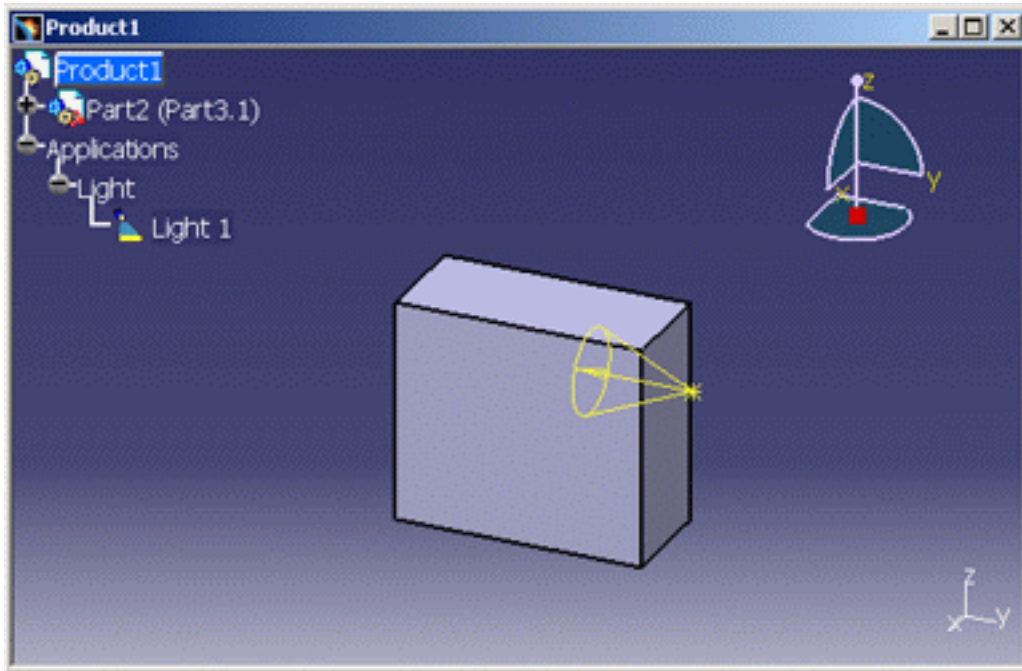
- As Viewpoint  
The light source will be positioned according to the user's viewpoint:



Spot light source - "As viewpoint" mode



- **Gravitational**  
The light source will be positioned along the X, Y or Z axis of the model. The axis to be used is defined by checking the corresponding radio button next to "Gravitational":



Spot light source - "Gravitational" mode along Y axis

3. Click **OK** to validate your settings.



# Output

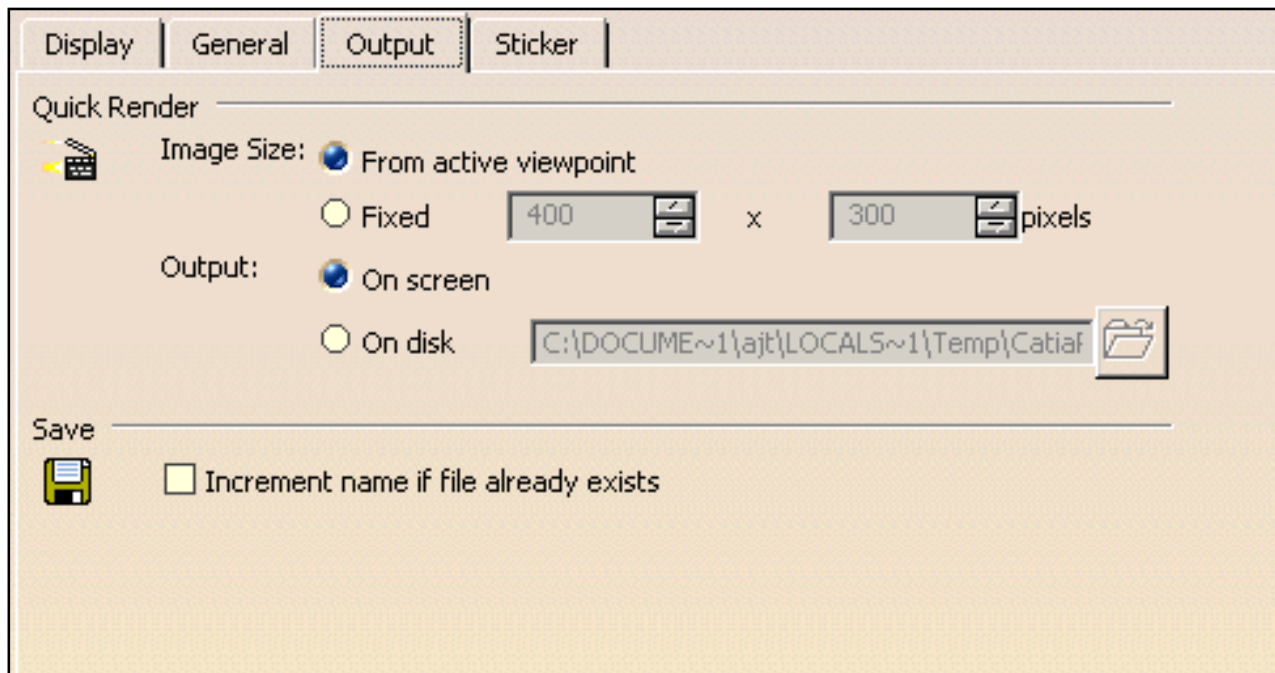



This task explains how to customize Rendering output settings.



1. Select the **Tools->Options...** command then in the **Infrastructure** category, click the **Photo Studio** subcategory.

2. Click the **Output** tab:





3. In the Image Size area, specify whether the image size is defined "From active viewpoint" or "Fixed". If you selected the "Fixed" option, you must then enter the image width and height in pixels.
4. In the Output area, indicate the output type, i.e. On screen or On disk. If you select the "On disk" option, you must then specify the new path and name of the image either by entering them directly in the field or by clicking the  icon to browse your folders to the desired location.

5. In the Save area, check the "Increment name if file already exists". In case the file name you specify already exists, this option adds the suffix "*\_ number*" to the name. The *number* will be automatically incremented as necessary, e.g. "CatiaRender\_1.tif", "CatiaRender\_2.tif", and so on.
  
6. Click **OK** to validate your settings.



# Satellites

 The satellites tab lets you define the name of the remote computers ("satellites") used in network to compute a rendered image. Distributed rendering enables to decrease rendering time since several processors are used to generate the image.

 This tab is available only when using Photo Studio Optimizer.

For detailed information on how to use this tab, refer to "Using Distributed Rendering" in the *Version 5 Photo Studio Optimizer User's Guide*.



# Stickers




This task explains how to customize the sticker default image.



1. Select the **Tools->Options...** command then in the **Infrastructure** category, click the **Photo Studio** subcategory.
  
2. Click the **Sticker** tab:



3. The Default Image area lets you replace the default sticker image with your own image.

Simply enter the path to the user-defined image or use the  icon to open the File Selection dialog box which lets you browse your folders to the desired location.

4. Click **OK** to validate.



# Customizing For Real Time Rendering 2



This section describes how to customize different settings. All tasks described here deal with permanent setting customization. These tasks are:

[Display](#)

[General](#)

[Stickers](#)

# Display

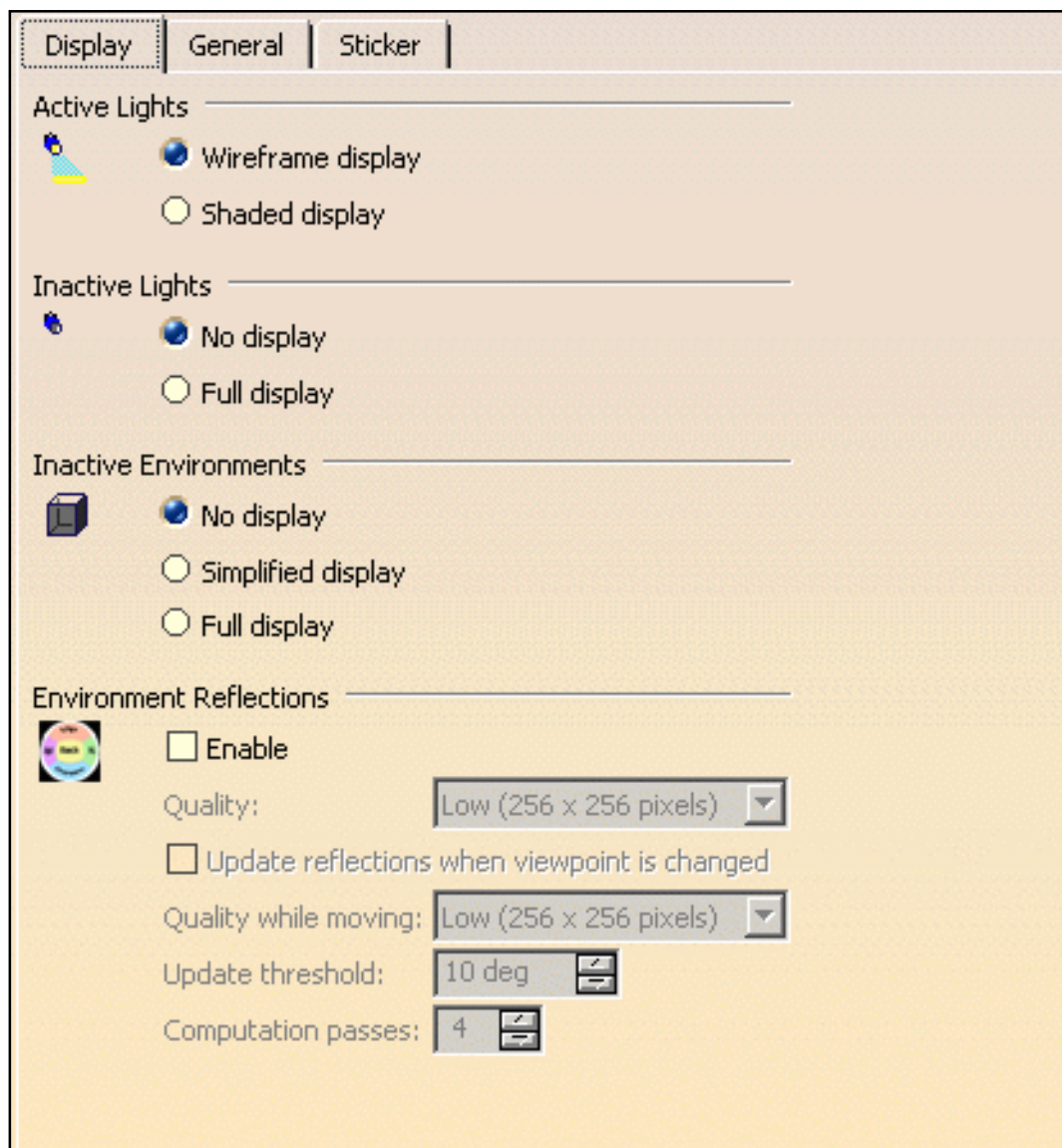


This task explains how to customize the rendering display.



1. Select the **Tools->Options...** command then in the **Infrastructure** category, click the **Real Time Rendering** sub-category.

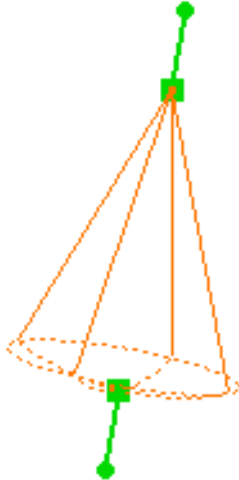
The Display tab appears:



## Setting Display for Active Lights

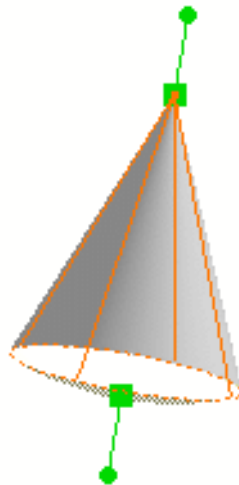
This area lets you control the display of active lights:

- check the "Wireframe display" option to display light sources in wireframe mode in the geometry area. This is the default mode:



*Spot light source:  
wireframe display*

- check the "Shaded display" option to display spot and punctual light sources in shading mode in the geometry area:

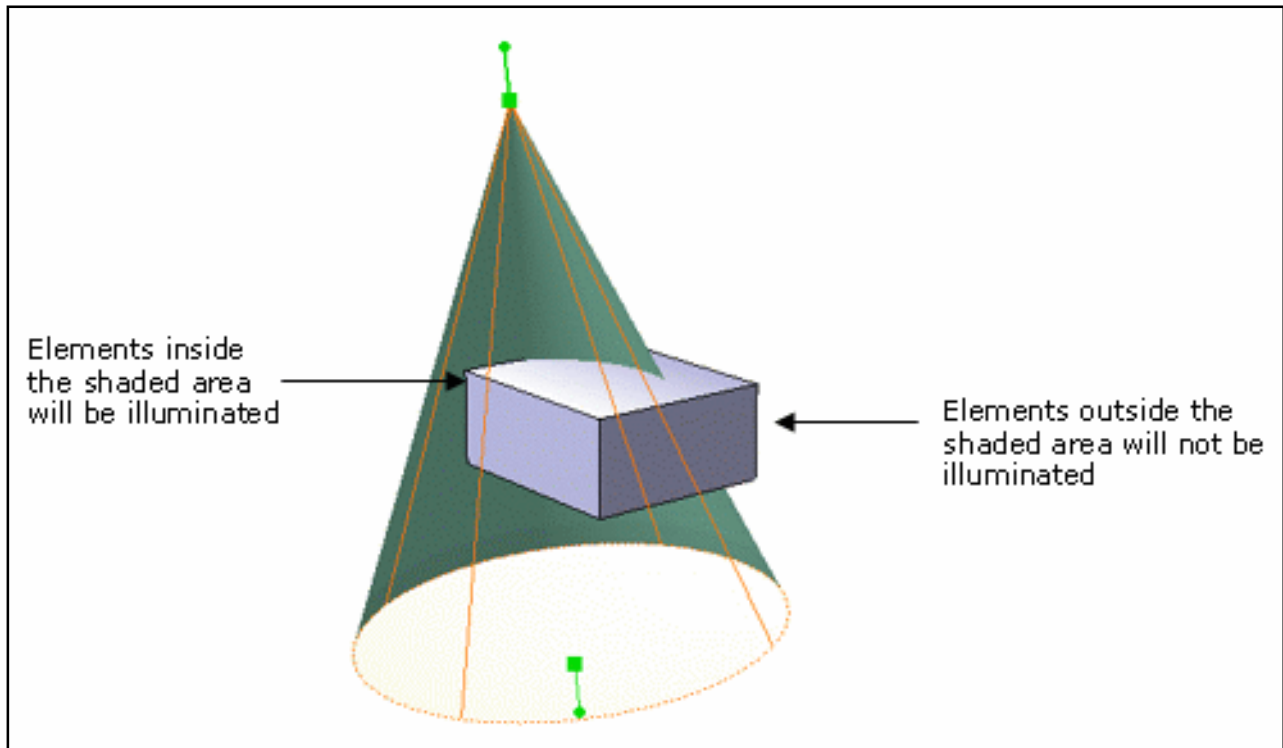


*Spot light source:  
shaded display*

You can use the manipulators to interactively position the light source as you would do for the default wireframe representation.

This display helps you visualize more easily which part of the object will be illuminated as shown below:





Note that the color of the shaded representation corresponds to the color defined for the light source (via the Lighting tab in the Properties dialog box). In the above example, the light intensity has been assigned a green color.

### Setting Display for Inactive Lights

This area lets you control the display of inactive lights:

- check the "No display for inactive lights" option to turn off inactive light display
- check the "Full display for inactive lights" option to turn on inactive light display.

### Setting Environment Display

This area lets you to set the type of environment display:

- check the "No display for inactive environments" option to turn off inactive environment display
- check the "Simplified display for inactive environments" option to display inactive environments in a simplified way
- check the "Full display for inactive environments" option to turn on inactive environment display.

## Setting Environment Reflections

This area lets you define reflection parameters:

- check the "**Enable**" option to enable active environment reflections on reflective objects.

However, bear in mind that this option does NOT deactivate the active environment reflections. Whenever you wish to restore the default reflections, you can either deactivate the environment by right-clicking it in the specification tree then unchecking the "Environment Active" option, or simply delete it.

Checking this option activates the fields displayed below:

- specify the reflection **Quality**: Low, Medium or High
- check the "**Enable reflections when viewpoint is changed**" option to update environment reflections on reflecting objects when the viewpoint is modified beyond a predefined threshold. Otherwise, the reflection update only occurs when you are working with a new or modified environment (i.e. new wall textures or colors)
- you can use the "**Quality while moving**" option to indicate the texture size used when the viewpoint is moved
- the "**Update threshold**" field lets you specify the texture update threshold in degrees when the viewpoint is rotated
- the "**Computation passes**" field lets you indicate the number of texture passes.

2. Click **OK** to validate your settings.



# General

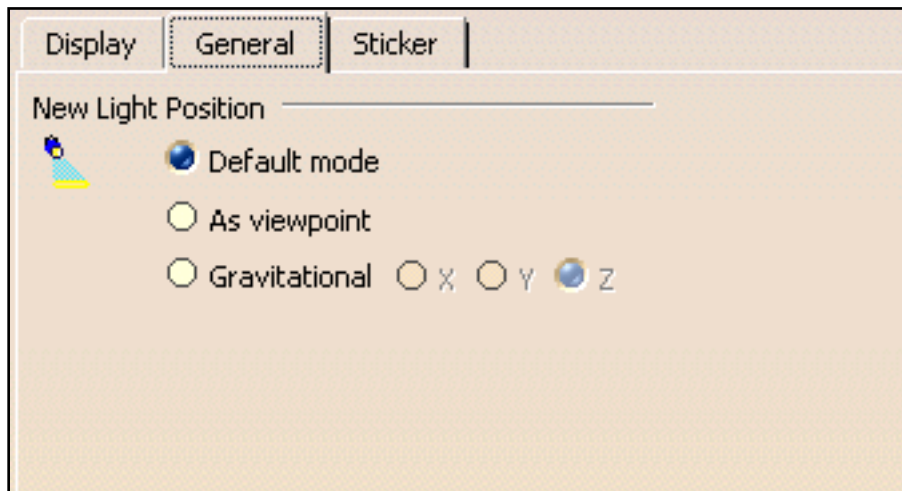


This task explains how to customize general settings for light sources.



1. Select the **Tools->Options...** command then in the **Infrastructure** category, click the **Real Time Rendering** sub-category.

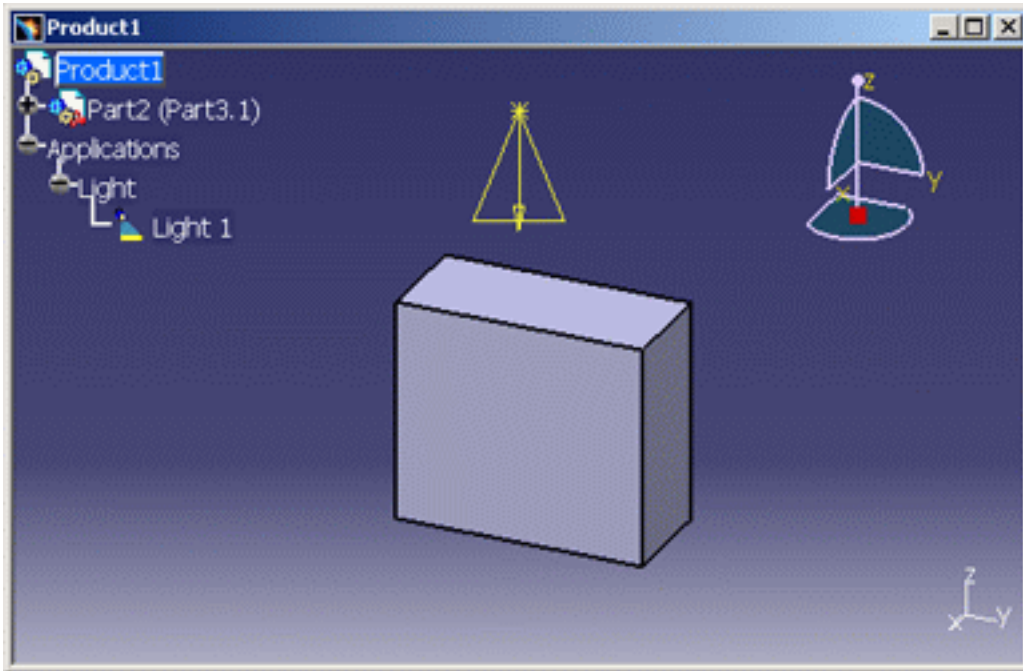
2. Click the **General** tab:



## New Light Position

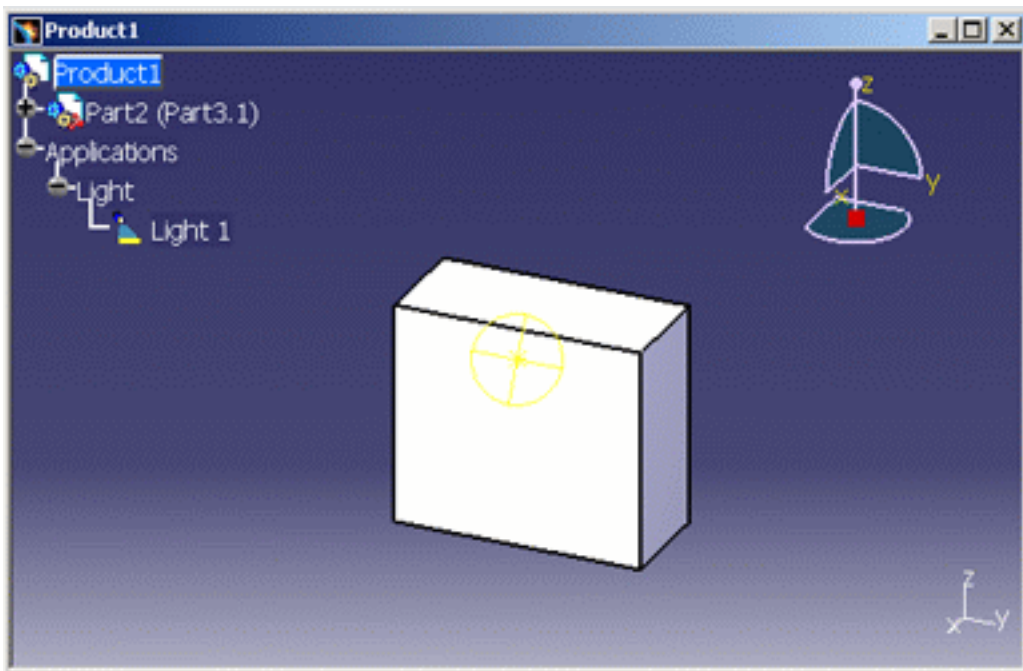
This area lets you define how the new light sources you will create will be positioned. You can choose between three modes:

- **Default mode**  
The light source will be positioned in the upper part of the geometry area, oriented down and centered:



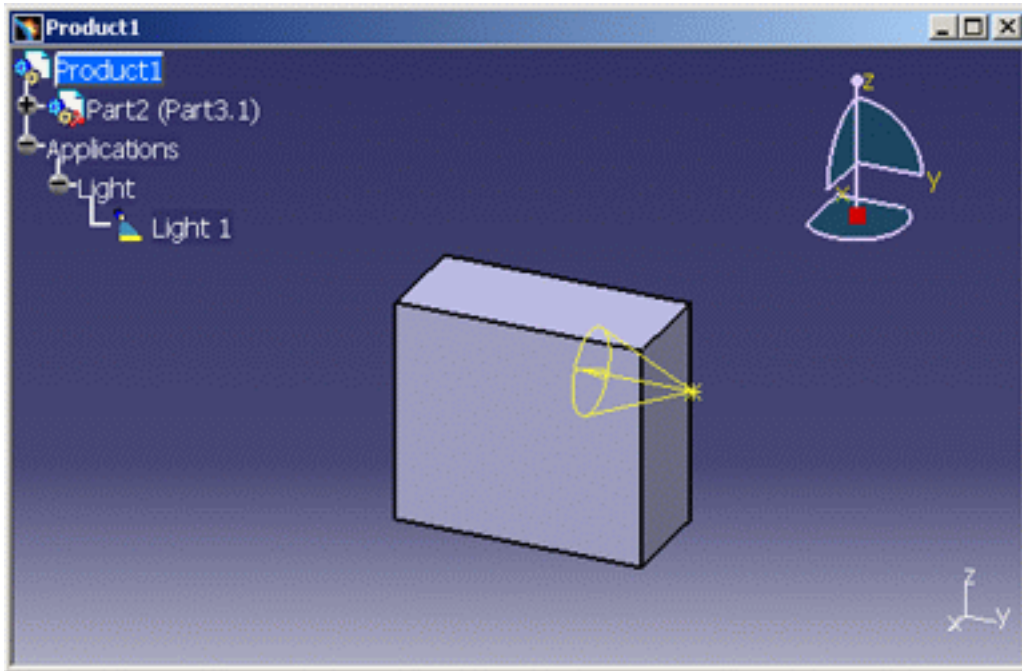
Spot light source - "Default" mode

- As Viewpoint  
The light source will be positioned according to the user's viewpoint:



Spot light source - "As viewpoint" mode

- Gravitational  
The light source will be positioned along the X, Y or Z axis of the model. The axis to be used is defined by checking the corresponding radio button next to "Gravitational":



Spot light source - "Gravitational" mode along Y axis

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



# Stickers



This task explains how to customize the sticker default image.




1. Select the **Tools->Options...** command then in the **Infrastructure** category, click the **Real Time Rendering** subcategory.

2. Click the **Sticker** tab:



3. The Default Image area lets you replace the default sticker image with your own image.

Simply enter the path to the user-defined image or use the  icon to open the File Selection dialog box which lets you browse your folders to the desired location.

4. Click **OK** to validate.



# Customizing

**Tree and Geometry View**

**General Settings**

**Constraint Appearance**

**CATPart Document**

**Tolerancing**

**Display**

**Manipulators**

**Annotation**

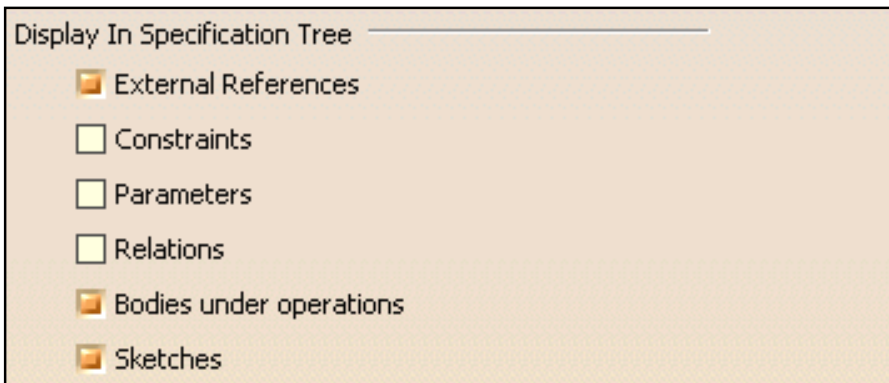
**View/Annotation Plane**

# Customizing the Tree and Geometry Views

This page deals with the these categories of options:

- [Display in Specification Tree](#)
- [Display in Geometry Area](#)
- [Checking Operation When Renaming](#)


## Display in Specification Tree



There are six types of elements you can display or not in the Specification tree. If you want them to be displayed, just check them.


### External References

Copies with links of geometry from other documents.

 By default, this option is checked.


### Constraints

Dimensional and geometrical constraints created in the CATPart document.

 By default, this option is unchecked.


### Parameters

Parameters created using the Knowledge Advisor capability. If you wish to know what parameters and relations are, refer to the *CATIA Knowledge Advisor Users Guide Version 5*.

 By default, this option is unchecked.

### Relations

Relations (formulas) created using the Knowledge Advisor capability. If you wish to know what relations are, refer to the *CATIA Knowledge Advisor Users Guide Version 5*.

 By default, this option is unchecked.




## Bodies under operations

Operations attaching bodies in different ways (Add, Assemble, Remove, Intersect, Union Trim). This option is available only with Part Design application. For more, refer to "Associating Bodies" in the *CATIA Part Design Users Guide Version 5*.

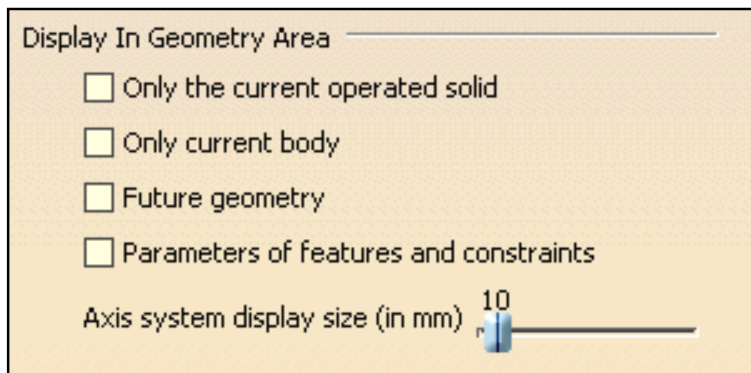
 By default, this option is checked.

## Sketches

If the **Sketches** option is checked, the sketches are displayed in the specification tree during creation. If unchecked, sketches are present in the tree but you need to use the expand capability.

 By default, this option is checked.


## Display in Geometry Area




There are five options available for customizing the **geometry** display.

### Only the current operated solid


This option is used when editing features belonging to **attached** bodies only. It lets you display only the features of the current body and greatly improves the application performances whenever you edit these features. Please note that this option is available in Part Design application as well as for geometrical and ordered geometrical sets.

**Note:** Instead of accessing this option via **Tools -> Options**, you can click this icon  available in the **Tools** toolbar.

 By default, this option is unchecked.

### Only Current Body


This option displays the geometry of the current part body or open body only.

 By default, this option is unchecked.




### Future geometry

This option displays the geometry located after the current feature only.

 By default, this option is unchecked.


### Parameters of features and constraints

This option permanently displays parameters attached to Part Design features and Sketcher constraints.

 By default, this option is unchecked.

### Axis system display size (in mm)

This option lets you define the size of axis systems in mm.

 By default, this option is unchecked.

## Checking Operation When Renaming




Checking Operation When Renaming \_\_\_\_\_

- No name check
- Under the same tree node
- In the main object

Three options let you define rules for renaming geometric elements (using the Properties command).


### No name check

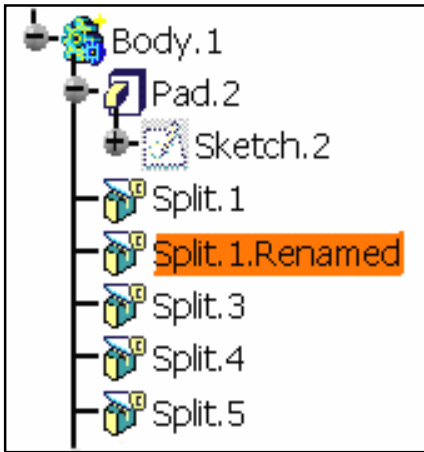
Use this option if you wish to allow all types of rename operations whatever the locations of the elements in the specification tree.

 By default, this option is checked.

### Under the same tree node

Check this option to prevent two elements belonging to a common node from having the same name. If you are giving an identical name, a warning message is issued informing you that the element you are renaming will be suffixed as 'Renamed'. The check operation is case-insensitive.

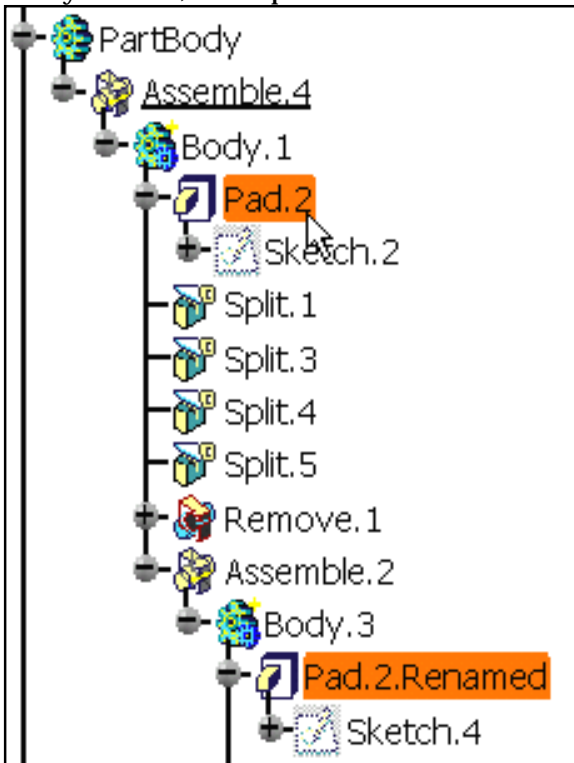
 By default, this option is unchecked.



**In the main object**

Check this option to prevent two elements belonging to the same main node from having the same name. The check operation is case-insensitive.

By default, this option is unchecked.



# Customizing General Settings

This page deals with these categories of options:

- [External References](#)
- [Update](#)
- [Delete Operation](#)
- [Replace](#)
- [Select](#)

## External References




External References

- Keep link with selected object
- Create external references as shown elements
- Confirm when creating a link with selected object
- Use root context in assembly
- Only use published elements for external selection keeping link
- Publish a face, edge, axis, vertex or extremity

### Keep link with selected object

Checking this option lets you maintain the links between external references, copied elements for example, and their origins when you are editing these elements. This option is used as you are editing parts included in assemblies. For more about designing parts in assembly context, refer to the *CATIA Assembly Design Users Guide Version 5*. If later you need to cut the link between external references and their origin, you just need to use the Isolate command.

 By default, this option is unchecked.

### Create external references as shown elements

Checking this option lets you define the visualization mode for external references or imported elements only.

 By default, this option is unchecked.


### Confirm when creating a link with selected object

Checking this option enables you to be informed that when you are pasting a part.

 By default, this option is unchecked.


### Use root context in assembly

Check this option to ensure that the root of the assembly is the context used. Uncheck this option if you prefer to use the minimal context. For more about changing contexts, please refer to the task describing the **Define Contextual Links** command in the *Product Structure User's Guide*.

 By default, this option is checked.


### Only use published elements for external selection keeping link

Check this option if you want to make only published elements valid for selection.

 By default, this option is unchecked.

### Publish a face, edge, vertex or extremity

Check this option if you wish to be able to directly select faces, edges, vertices, axes extremities when using the **Publication** command.

 By default, this option is checked.

## Update



Update

Automatic  Manual

Stop update on first error

Synchronize all external references for update

Activate local visualization


### Automatic/Manual

Check **Automatic** if you want parts to be updated automatically. Conversely, check **Manual** if you wish to control your update operations.

 By default, the **Automatic** option is checked.


### Stop Update on first error

Check this option to stop the update process as soon as the application finds an error when building the geometry.

 By default, this option is checked.


### Synchronize all external references for update

Check this option to make sure that the application updates elements copied from other parts. Synchronizing assumes that all modifications to the other parts affect external references included in your part. If this option is deactivated, the application will update your part only.

 By default, this option is checked.

### Activate local visualization

Check this option to visualize features as they are being rebuilt during the update process.

 By default, this option is unchecked.


## Delete Operation



|                                     |                               |
|-------------------------------------|-------------------------------|
| Delete Operation                    |                               |
| <input checked="" type="checkbox"/> | Display the Delete dialog box |
| <input type="checkbox"/>            | Delete exclusive parents      |

### Display the Delete dialog box


Check this option if you wish to access filters for deletion (see "Deleting Features" in the *Part Design Users Guide Version 5*).

 By default, this option is checked.

### Delete exclusive parents

Check this option if you wish to delete the parents of the features you are deleting. The parents will be deleted only if they are exclusive, which means that if they are shared by other features, they will not be deleted. When this setting is active, the option is checked in the **Delete** dialog box, but by default, the option is not checked.

Even if the option is checked in the Delete dialog box, you can uncheck it if you wish to. If the **Display the Delete dialog box** (see above) setting is not checked, this setting has no effect. For more information, refer to Deleting Features in the *Part Design Users Guide Version 5*.

 By default, this option is unchecked.



## Replace

Replace \_\_\_\_\_

Do replace only for elements situated after the In Work Object



### Do replace only for elements situated after the In Work Object

Checking this option makes the **Replace...** operation possible only for features located below the feature in Work Object and in the same branch.

 By default, this option is unchecked.



## Select

Select \_\_\_\_\_

Enable selection of not drawn geometry

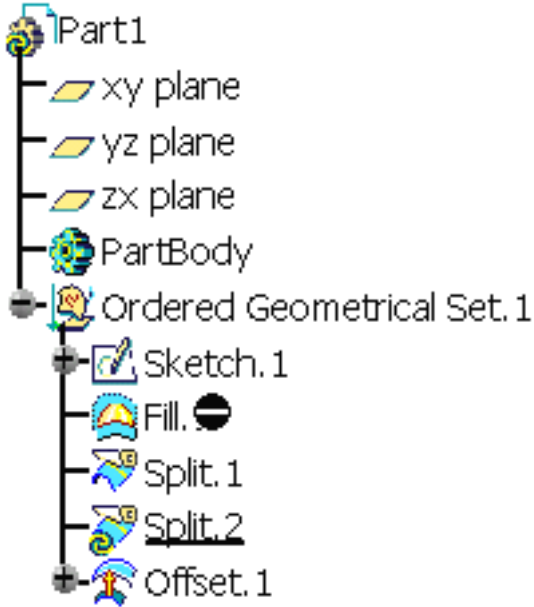
Enable selection of future geometry



### Enable selection of not drawn geometry

This option applies to the elements you select when you are creating features that will be located in an ordered geometrical set. If unchecked, selecting elements whose geometry is not visible any more, is not possible.

Here, for instance, selecting Fill.1 is not possible for defining Split.2 because Fill.1 geometry was absorbed by Split.1. A black sign indicates that this selection is not possible. Additionally, the application displays a tooltip explaining why it is not possible.



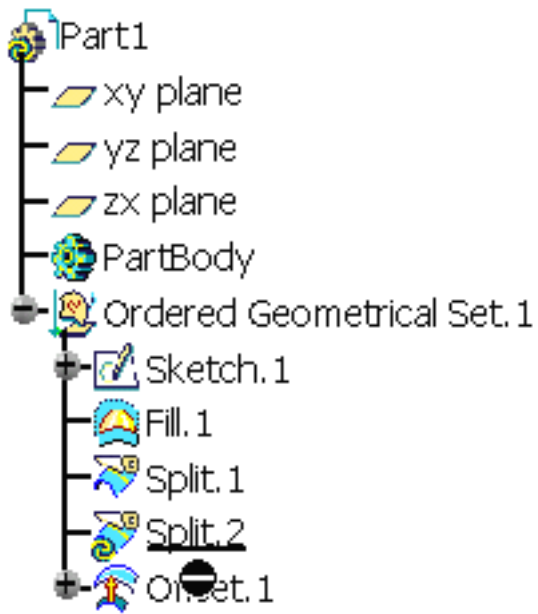
By default, this option is unchecked.


### Enable selection of posterior geometry

This option applies to the elements you select when you are creating features that will be located in an ordered geometrical set. If unchecked, the selection of elements located after the element being created is not possible.

Here, for instance, selecting Offset.1 is not possible for defining Split.2, as shown by the black sign. Additionally, the application displays a tooltip explaining why this selection is not possible.



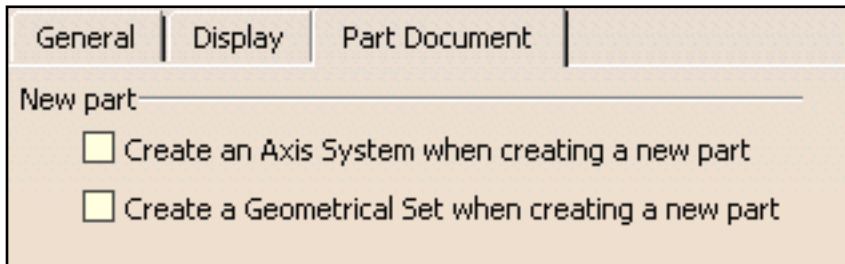


 By default, this option is unchecked.

# Customizing a CATPart document


This page deals with two options dealing with the creation of a part.

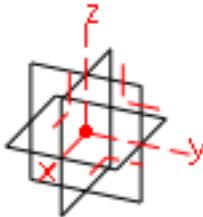
## New Part



### Create an Axis System when creating a new part

Check this option if you wish to create a three-axis system which origin point is defined by the intersection of the three default planes that is plane xy, plane yz, and plane zx. When the CATPart is open, the axis system is displayed both in the geometry and in the specification tree. For more information about the **Axis System** capability, refer to *CATIA - Part Design User's Guide*.

 By default, this option is unchecked.




**Note:** data contained in the CGR format are saved within the CATPart format when you are saving your part in order to improve performance when working in Assembly Design workbench.



### Create a Geometrical Set when creating a new part

Check this option if you wish to create a geometrical set as soon as you create a new part. For more information about the **Axis System** capability, refer to *Generative Shape Design User's Guide*.

 By default, this option is unchecked.

# Customizing for 3D Annotations Infrastructure


This section describes the various settings you can customize for 3D Annotations Infrastructure via the **Tools -> Options** command.

These settings will then be used when creating annotations via the **Annotations** toolbar available in some workbenches such as Part Design or Weld Design, for example.



Tolerancing  
Display  
Manipulators  
Annotation  
View/Annotation Plane

# Tolerancing

 This task will show you how to customize 3D Annotations Infrastructure tolerancing settings.



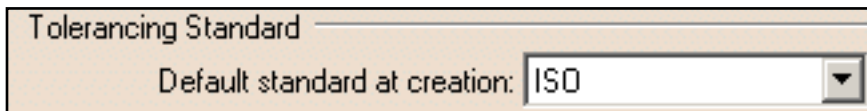
1. Select **Tools** -> **Options...**

The **Options** dialog box appears.

2. Click the **Infrastructure** category.
3. Click the **3D Annotations Infrastructure** subcategory.
4. Click the **Tolerancing** tab.

The **Tolerancing** tab appears, displaying two categories of options:

## Tolerancing Standard



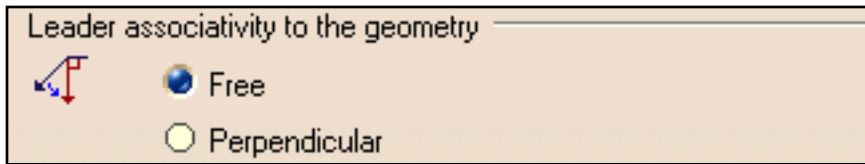
### Default standard at creation

Defines the default standard at creation.

Provides five conventional standards:

- ASME: American Society for Mechanical Engineers
- ASME 3D: American Society for Mechanical Engineers
- ANSI: American National Standards Institute
- ISO: International Organization for Standardization
- JIS: Japanese Industrial Standard

Leader associativity to the geometry

**Free**

Specifies that leader annotations are freely positioned relative to their geometrical elements.

**Perpendicular**

Specifies that leader annotations are positioned perpendicular to their geometrical elements.



# Display



This task will show you how to customize 3D Annotations Infrastructure display settings.



1. Select **Tools** -> **Options...**

The **Options** dialog box appears.

2. Click the **Infrastructure** category.
3. Click the **3D Annotations Infrastructure** subcategory.
4. Click the **Display** tab.

The **Display** tab appears, displaying two categories of options:

## Grid

| Grid                                       |  | Primary spacing: | Graduations: |
|--------------------------------------------|--|------------------|--------------|
| <input type="checkbox"/> Display           |  | H: 100 mm        | 10           |
| <input type="checkbox"/> Snap to point     |  | V: 100 mm        | 10           |
| <input type="checkbox"/> Allow Distortions |  |                  |              |

### Display

Defines whether the grid is displayed.

### Snap to point

Defines whether annotations are snapped to the grid's point.

### Allow Distortions

Defines whether grid spacing and graduations are the same horizontally and vertically.

### H Primary spacing

Defines the grid's horizontal spacing.

**H Graduations**

Defines the grid's horizontal graduations.

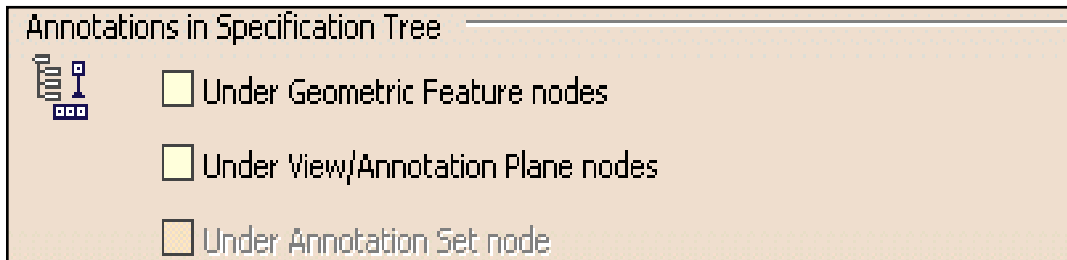
**V Primary spacing**

Defines the grid's vertical spacing (available only if Allow Distortions is checked).

**V Graduations**

Defines the grid's vertical graduations (available only if Allow Distortions is checked).

## Annotations in Specification Tree

**Under Geometric Feature nodes**

Defines that 3D annotations should be displayed under the geometric feature nodes in the specification tree. This lets you view 3D annotations under the Part Design or GSD feature nodes to which they are applied.

**Under View/Annotation Plane nodes**


Defines that 3D annotations should be displayed under the view/annotation plane nodes in the specification tree. This lets you view 3D annotations under the view node to which they are linked.

**Under Annotations Set node**

Defines that 3D annotations should be displayed under the annotation set node in the specification tree.



# Manipulators

 This task will show you how to customize 3D Annotations Infrastructure manipulators settings.



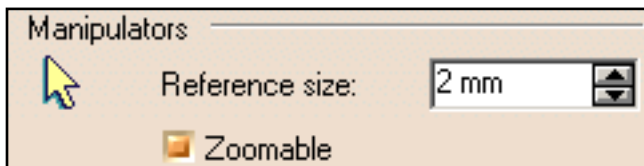
1. Select **Tools** -> **Options...**

The **Options** dialog box appears.

2. Click the **Infrastructure** category.
3. Click the **3D Annotations Infrastructure** subcategory.
4. Click the **Manipulators** tab.

The **Manipulators** tab appears, displaying one option:

## Manipulators



Reference size

Defines the annotation manipulator's size.


Zoomable

Defines whether the annotation manipulator is zoomable or not.





# Annotation

 This task will show you how to customize 3D Annotations Infrastructure annotation settings.



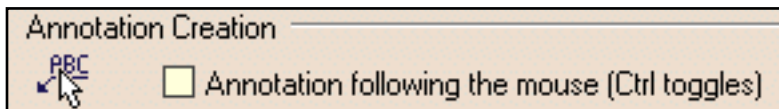
1. Select **Tools** -> **Options...**

The **Options** dialog box appears.

2. Click the **Infrastructure** category.
3. Click the **3D Annotations Infrastructure** subcategory.
4. Click the **Annotation** tab.

The **Annotation** tab appears, displaying one category of options:

## Annotation Creation




### Annotation following the mouse (Ctrl toggles)

Defines whether the annotation is positioned according to the cursor, following it dynamically during the creation process or not.



# View/Annotation Plane

 This task will show you how to customize 3D Annotations Infrastructure view/annotation plane settings.



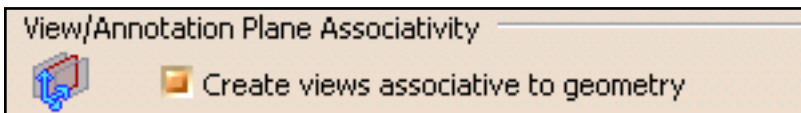
1. Select **Tools -> Options...**

The **Options** dialog box appears.

2. Click the **Infrastructure** category.
3. Click the **3D Annotations Infrastructure** subcategory.
4. Click the **View/Annotation Plane** tab.

The **View/Annotation Plane** tab appears, displaying two categories of options:

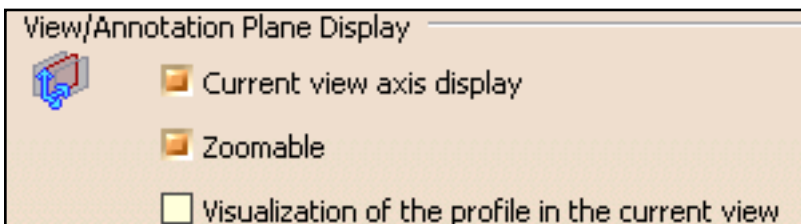
## View/Annotation Plane Associativity



### Create views associative to geometry

Creates views associative to the geometry, so that views and their annotations are automatically updated when the geometry is modified.

## View/Annotation Plane Display



### Current view axis display

Defines whether the active annotation plane axis system is displayed.

**Zoomable**

Defines whether the annotation plane axis is zoomable.

**Visualization of the profile in the current view**

Defines whether the view/annotation plane profile on the part/product is displayed or not.



# DELMIA Infrastructure

Refer to the *3D Simulation for Manufacturing User's Guide*

# Mechanical Design

Assembly Design

Sketcher

Mold Tooling Design

Structure Design

Drafting

Aerospace Sheetmetal Design

Sheetmetal Design

Functional Tolerancing and Annotation

# Customizing for Assembly Design

This section describes the different types of setting customization you can perform in Assembly Design using the **Tools -> Options** command.

All tasks described here deal with permanent setting customization.



General  
Constraints

# General



This task will show you how to customize Assembly Design constraints settings..



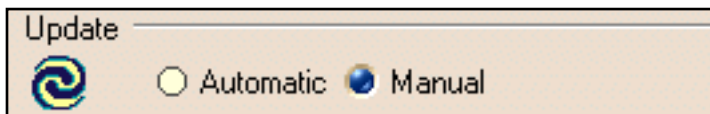
1. Select **Tools** -> **Options...**, the **Options** dialog box appears.

The **Options** dialog box appears.

2. Click the **Mechanical Design** category.
3. Click the **Assembly Design** subcategory.
4. Click the **General** tab

The **General** tab appears, displaying five categories of options:

## Update



Update affects Assembly constraints, Assembly features, Weld features, contextual design.

- **Automatic**
  - This option update your assembly automatically after each interaction.
- **Manual**
  - This option lets you decide when you need to update your assembly. It is the default mode.

## Update propagation

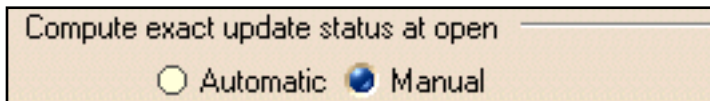


Update propagation affects Assembly constraints, Assembly features, Weld features.

- **Active level**
  - This option update child components of the active components only.
- **All the levels**
  - This option update all components under the active components.



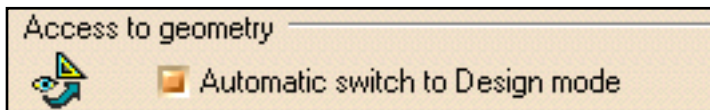
## Compute exact update status at open



This option is only take into account when you work with the **Work with the cache system** option checked, and when you are opening an assembly document or inserting an assembly component.

- **Automatic**
  - This option load the minimal data needed in assembly components to determine the update status of the assembly: updated or not.
- **Manual**
  - This option displays the **Unknown Status Update** icon when the minimal data needed are not loaded to determine the update status of the assembly.

## Access to geometry



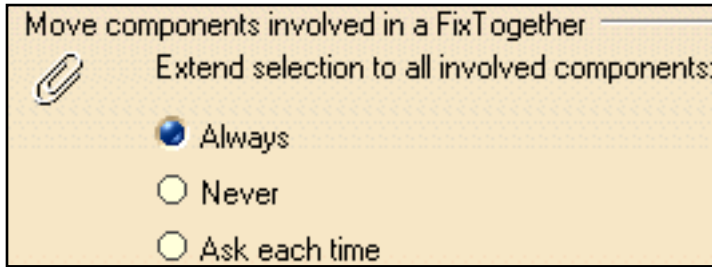
This option is only take into account when you work with the **Work with the cache system** option checked.

- **Automatic switch to Design mode**
  - This option ensures that the application will automatically launch the **Design mode** when using the following commands: constraints creation, reuse pattern and define multi-instantiation.





## Move components involved in a Fix Together



- **Always**
  - This option ensures to move all the components involved in a **Fix Together** without warning.
- **Never**
  - This option ensures to move only the selected component involved in a **Fix Together** without warning.
- **Ask each time**
  - This option ensures to display of a warning message that appears when moving components.

The message will appear whenever you will apply the following commands to components fixed together:

- **compass**
- **Snap**
- **Translation or Rotation**
- **Manipulate**



# Constraints



This task will show you how to customize Assembly Design constraints settings..



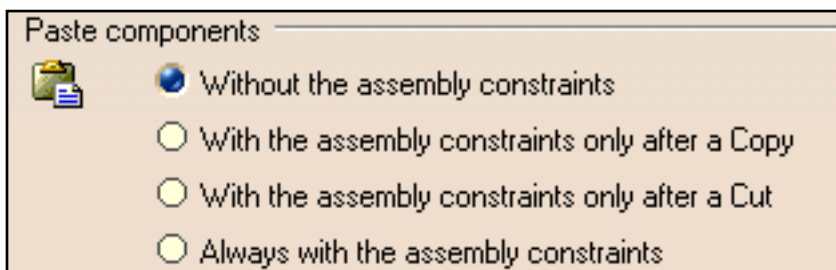
1. Select **Tools** -> **Options...**, the **Options** dialog box appears.

The **Options** dialog box appears.

2. Click the **Mechanical Design** category.
3. Click the **Assembly Design** subcategory.
4. Click the **Constraints** tab

The **Constraints** tab appears, displaying three categories of options:

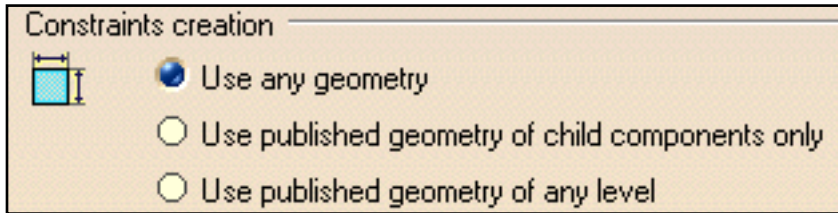
## Paste components



- **Without the assembly constraints**
  - You can paste one or several components without the assembly constraints applying to them.
- **With the assembly constraints only after a Copy**
  - You can paste one or several components with the assembly constraints applying to them, only after the **Copy** command.
- **With the assembly constraints only after a Cut**
  - You can paste one or several components with the assembly constraints applying to them, only after the **Cut** command.
- **Always with the assembly constraints**

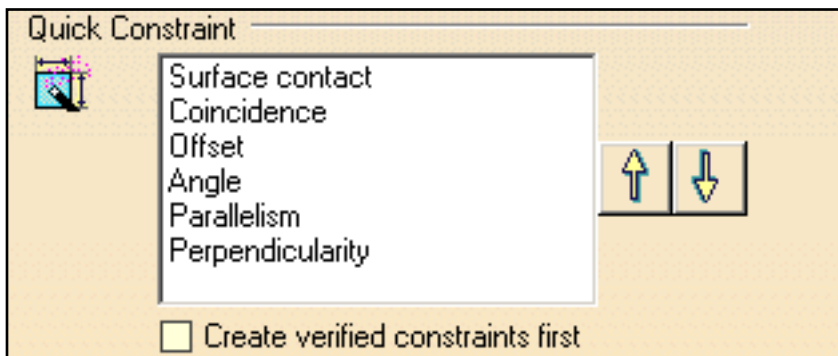
- You can paste one or several components always with the assembly constraints applying to them.

## Constraints creation



- **Use any geometry**
  - You can select any geometrical element.
- **Use published geometry of child components only**
  - You can select any published element belonging to child components. For more information about publication, refer to **Managing a Product Publication** in the **Assembly Design** documentation.
- **Use published geometry of any level**
  - You can select any published element

## Quick Constraint



- The **Quick Constraint** command is based on a ordered list of constraints to be created by the application. The setting available here lets you reorder the list of constraints having priority when applying the Quick Constraint command to the selected geometrical elements. What you need to do is select the constraint type you wish to reorder and click the arrows to the right of the selection to reorder the selected constraint.
- Additionally, you can create verified constraints if the option **Creation verified constraints first** is checked.



# Customizing for Sketcher



The Sketcher settings will help you sketch your profile with given options. For example, the grid will make it easier to sketch a profile requiring parallel lines.




**1.** Select the **Tools** -> **Options** command to display the **Options** dialog box. The **Options** dialog box appears.

**2.** Expand the **Mechanical Design** option, then click **Sketcher**. The **Sketcher** tab appears, containing the following set of options:

- [Grid](#)
- [Sketch Plane](#)
- [Geometry](#)
- [Constraint](#)
- [Colors](#)

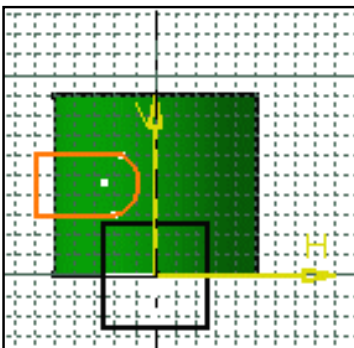
## Grid




| Grid                                                                                |                                             |                                       |                                 |
|-------------------------------------------------------------------------------------|---------------------------------------------|---------------------------------------|---------------------------------|
|  | <input checked="" type="checkbox"/> Display | Primary spacing :                     | Graduations :                   |
|                                                                                     | <input type="checkbox"/> Snap to point      | H: <input type="text" value="100mm"/> | <input type="text" value="10"/> |
|                                                                                     | <input type="checkbox"/> Allow Distortions  | V: <input type="text" value="100mm"/> | <input type="text" value="10"/> |

## Display

Check this option to display the grid in your session.



 By default, this option is unchecked.

## H; V; Primary spacing; Graduations


To define your grid, enter the values of your choice in the **H** and **V** fields. The **Primary spacing** option lets you define the spacing between the major lines of the grid. The **Graduations** field lets you set the number of graduations between the major lines of the grid, which actually consists in defining a secondary grid.



Note that in the Sketcher workbench you need to set 2D curves in NO Show mode to be able to visualize the part edges in the 3D area.


### Snap to point

Check the **Snap to point** option if your sketch needs to begin or end on the points of the grid.

 By default, this option is checked.

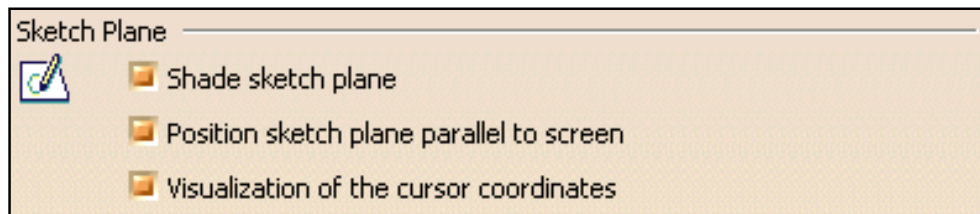
### Allow Distortions

Check this option to apply different graduations and spacing between **H** and **V**.

 By default, this option is unchecked.




## Sketch Plane

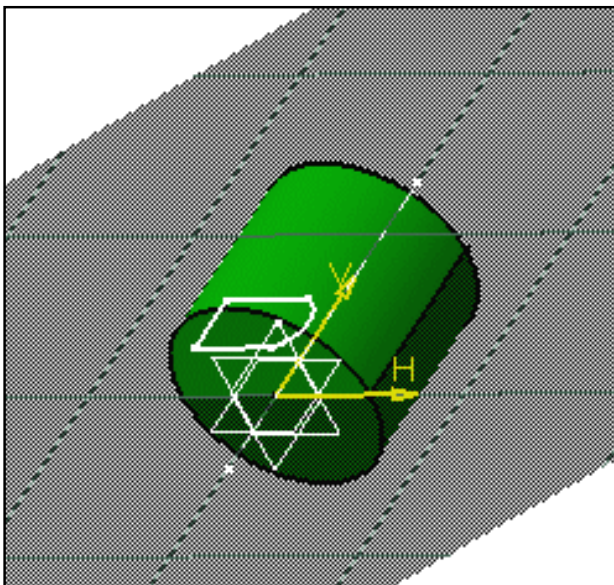


### Shade sketch plane

Check this option to view the sketch plane in the Sketcher workbench. Note that you also need to display the grid in order to be able to see the shaded plane.


The sketch plane will be shaded in the Sketcher workbench:

 By default, this option is unchecked.




### Position sketch plane parallel to screen

Check this option to ensure that the sketch plane will be positioned properly each time you enter the Sketcher workbench.

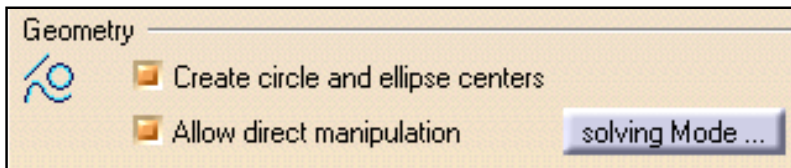
 By default, this option is checked.

### Visualization of the cursor coordinates

Check this option to have the cursor coordinates displayed, near the cursor, within the geometry.


 By default, this option is checked.

## Geometry




### Create circle and ellipse centers

You can decide whether or not you want to create centers when sketching circles or ellipses. By default, this option is activated. Just uncheck it if you do not need to create centers when sketching.

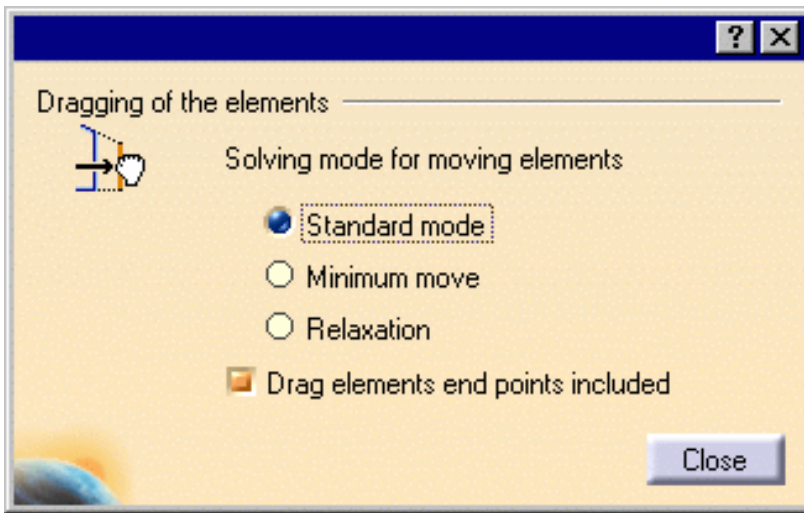
 By default, this option is checked.

### Allow direct manipulation

Select this option to be able to move geometry using the mouse. When moving geometry, you can move either the minimum number of elements, the maximum number of elements, or still the minimum number by modifying the shape of elements, if needed. Click the **Solving mode...** button to configure manipulation.

 By default, this option is checked.

### Solving mode...



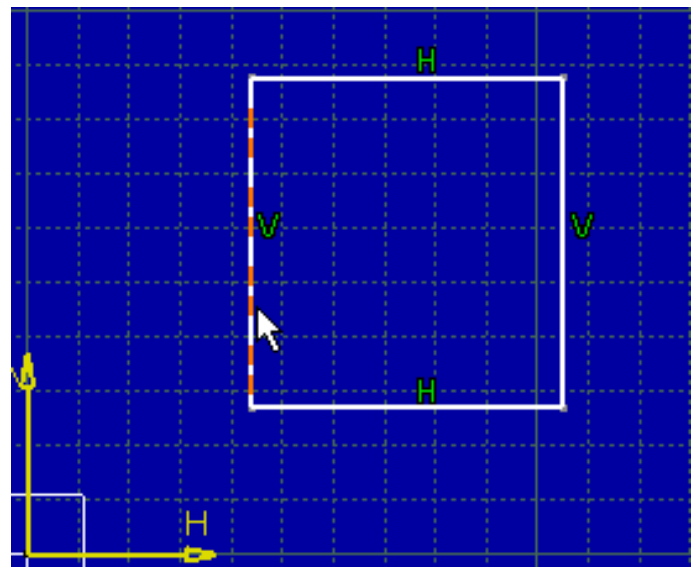
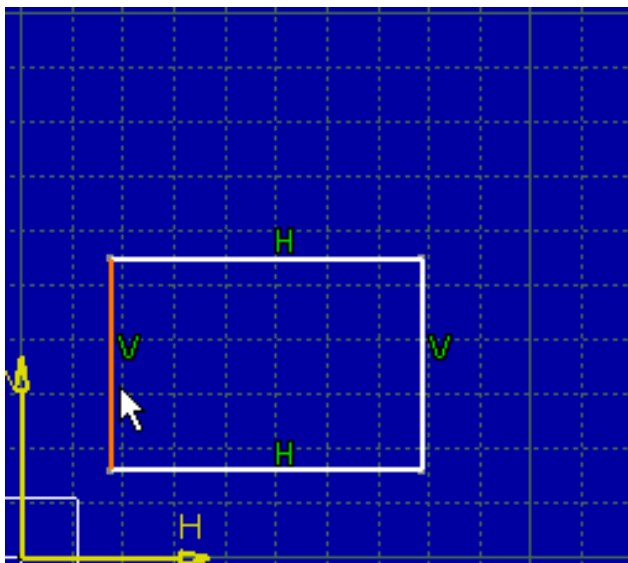
The dialog box that appears offers the following options as regards the solving mode:

**Solving mode for moving elements**

**Standard mode**

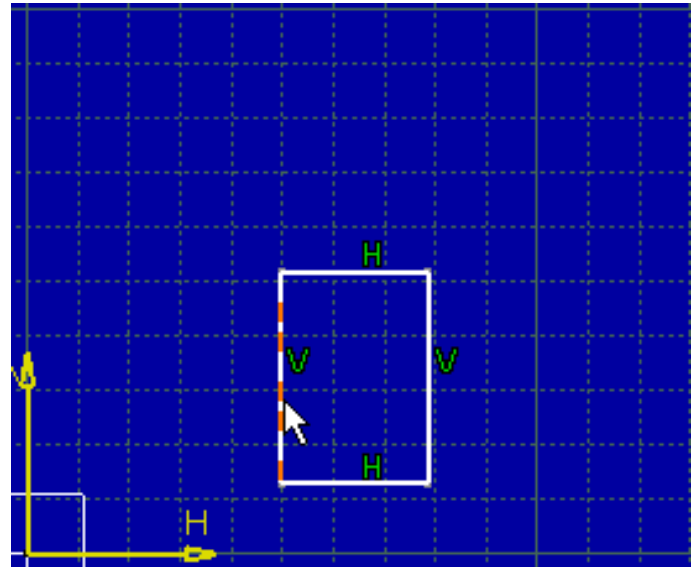
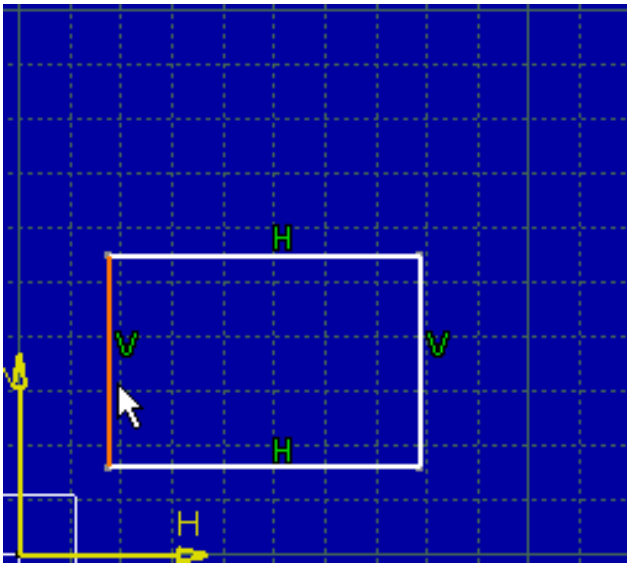
You move as many elements as possible and also respect existing constraints.

By default, this option is checked.



**Minimum move**

You move as few elements as possible and also respect existing constraints.



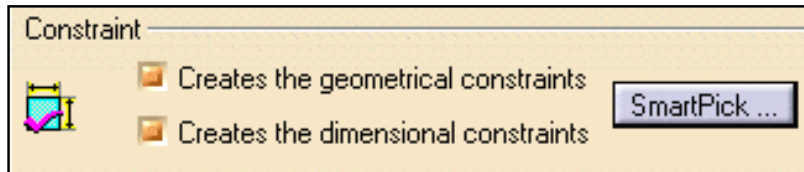
**Relaxation**

You move elements by re-distributing them over the sketch, globally speaking. This method solves element moving by minimizing energy cost.

**Drag elements end points included**


Furthermore, you can choose to drag elements along with their end points by checking this box.

**Constraint**



**Creates the geometrical constraints**


Lets you specify whether or not you want to created the geometrical constraints detected by the SmartPick tool.

 By default, this option is checked.

**Creates the dimensional constraints**

Lets you specify whether or not you want to created the dimensional constraints detected by the SmartPick tool.

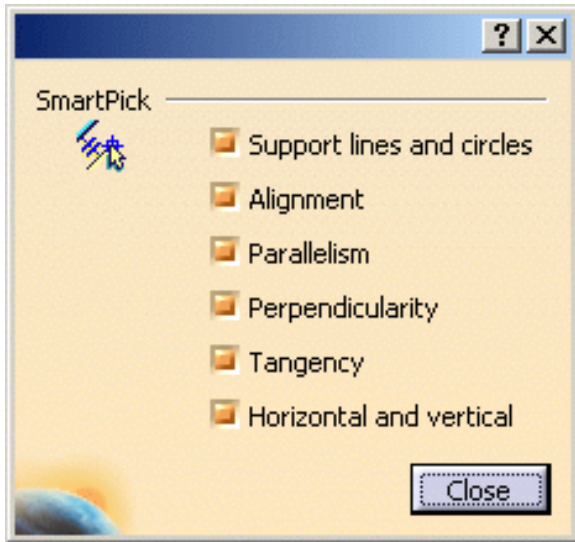
If both these detection options are unchecked, the **Create detected constraints** option will be inactive by default in the Tools toolbar. You will be able to activate it at any time.

 By default, this option is checked.

**SmartPick... (switch button)**



As you create more and more elements, SmartPick detects multiple directions and positions, and more and more relationships with existing elements. This may lead to confusion due to the rapid highlighting of several different detection possibilities as you point the cursor at different elements in rapid succession. Consequently, you can decide to filter out undesired detections by clicking the **SmartPick...** button.



In the SmartPick dialog box that appears, the following options are available.

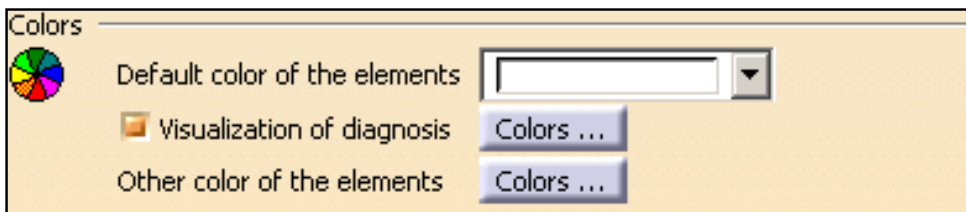
- **Support lines and circles**
- **Alignment**
- **Parallelism**
- **Perpendicularity**
- **Tangency**
- **Horizontal and vertical**

 By default, all of these options are checked.

Uncheck the elements you do not wish to detect while sketching.

Disabling SmartPick completely (i.e. unchecking all options in the SmartPick dialog box) is particularly useful when your screen is full of elements: in this case, it may be a good idea to disable SmartPick to concentrate only on the geometry.

## Colors



Two types of colors may be applied to sketched elements. These two types of colors correspond to colors illustrating:

- Graphical properties  
Colors that can be modified. These colors can therefore be modified using the **Tools->Options** dialog box.

**OR**

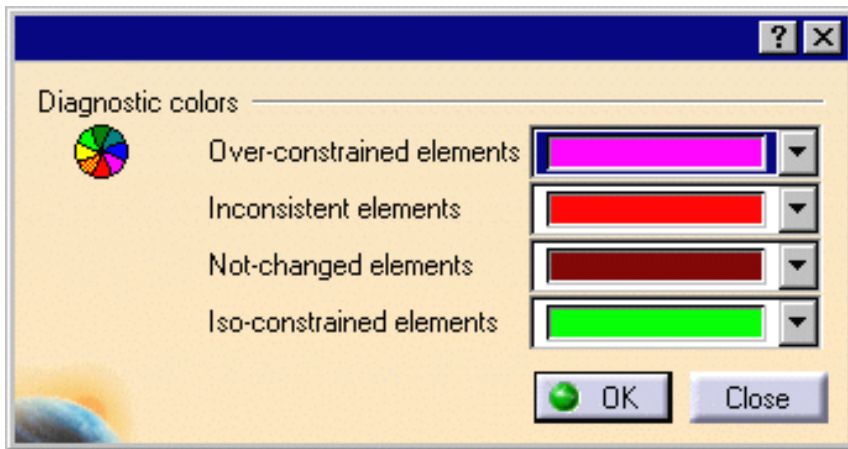
- Constraint diagnosis  
Colors that represent constraint diagnoses are colors that are imposed to elements whatever the graphical properties previously assigned to these elements and in accordance with given diagnoses. As a result, as soon as the diagnosis is solved, the element is assigned the color as defined in the **Tools->Options** dialog box.

**Default color of the elements**

From the list, choose the color that you want to use as the default for sketched elements.

**Visualization of diagnosis**

Select this option if you want over-constrained, inconsistent, not-changed or iso-constrained elements to be identified using specific colors. Then, click the **Colors...** button to configure these colors.



In the dialog box that appears, you can configure colors for the following types of elements:

**Over-constrained elements**

The dimensioning scheme is over-constrained: too many dimensions were applied to the geometry.

**Inconsistent elements**

At least one dimension value needs to be changed. This is also the case when elements are under-constrained and the system proposes defaults that do not lead to a solution.


**Not-changed elements**

Some geometrical elements are over-defined or not-consistent. As a result, geometry that depend(s) on the problematic area will not be recalculated.

**Iso-constrained elements**

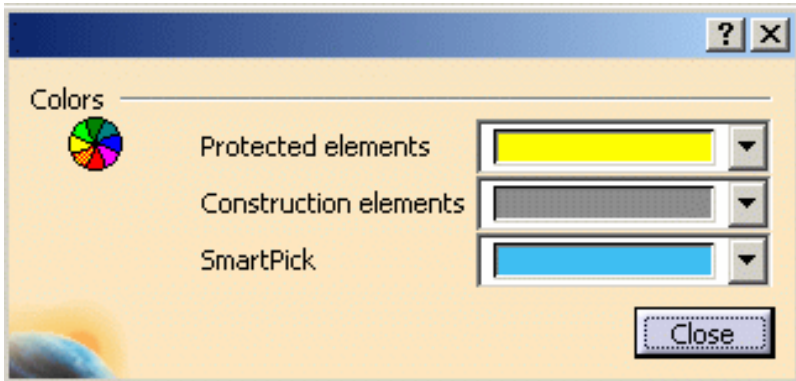
All the relevant dimensions are satisfied. The geometry is fixed and cannot be moved from its geometrical

support.

 By default, this option is checked.

### Other color of the elements

Click the **Colors...** button to configure the colors of other elements.



In the dialog box that appears, you can configure colors for the following types of elements:

#### **Protected elements**

Non-modifiable elements.

#### **Construction elements**

A construction element is an element that is internal to, and only visualized by, the sketch. This element is used as positioning reference. It is not used for creating solid primitives.

#### **SmartPick**

Colors used for SmartPick assistant elements and symbols.

**3.** Once you have chosen all of your options, click **OK** to confirm your choices and to exit the Options dialog box.



# Customizing for Mold Tooling Design

General  
Component

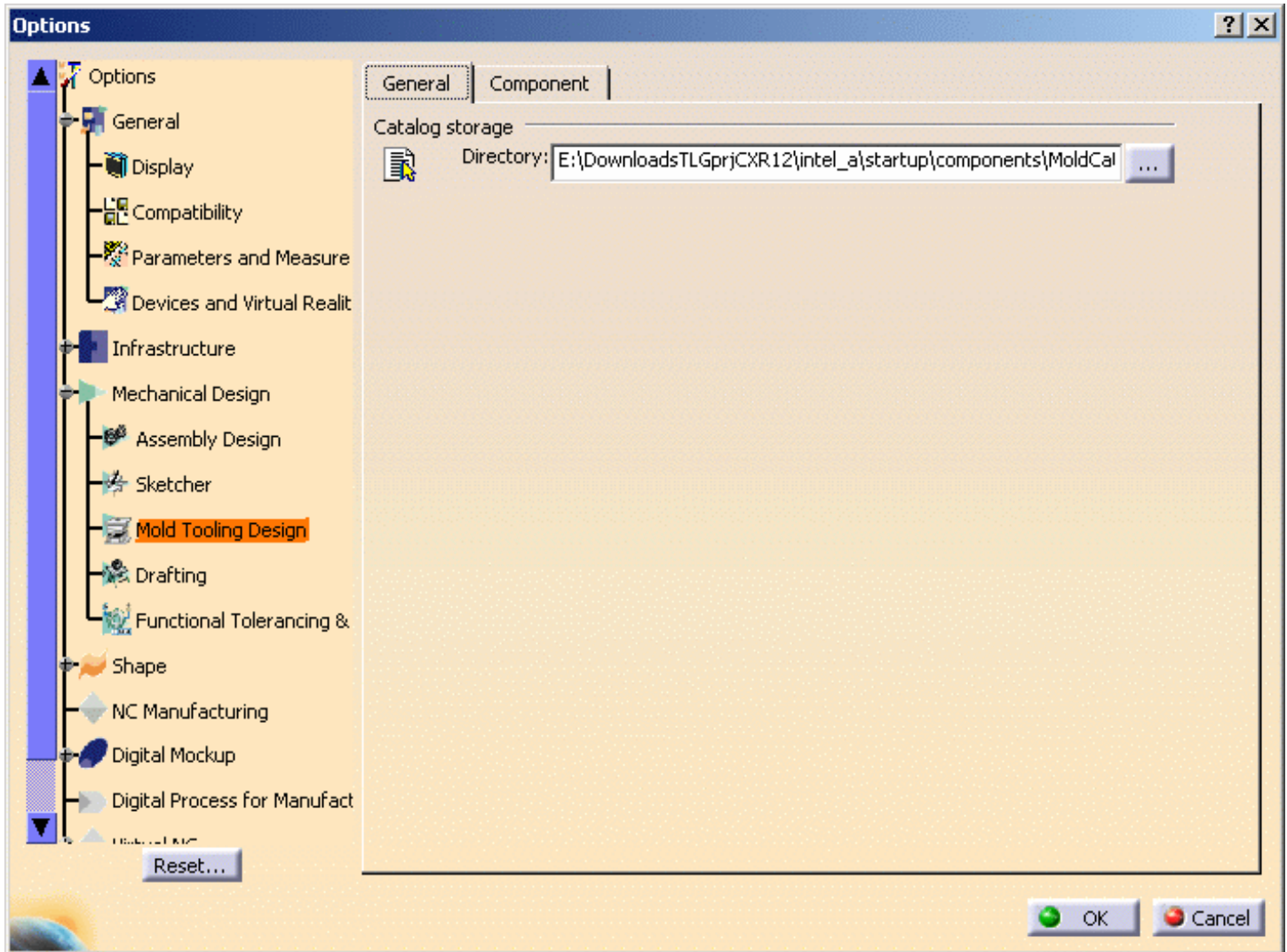
# Customizing for Mold Tooling Design



This task shows you how to customize the options of components of the Mold Tooling Design application.

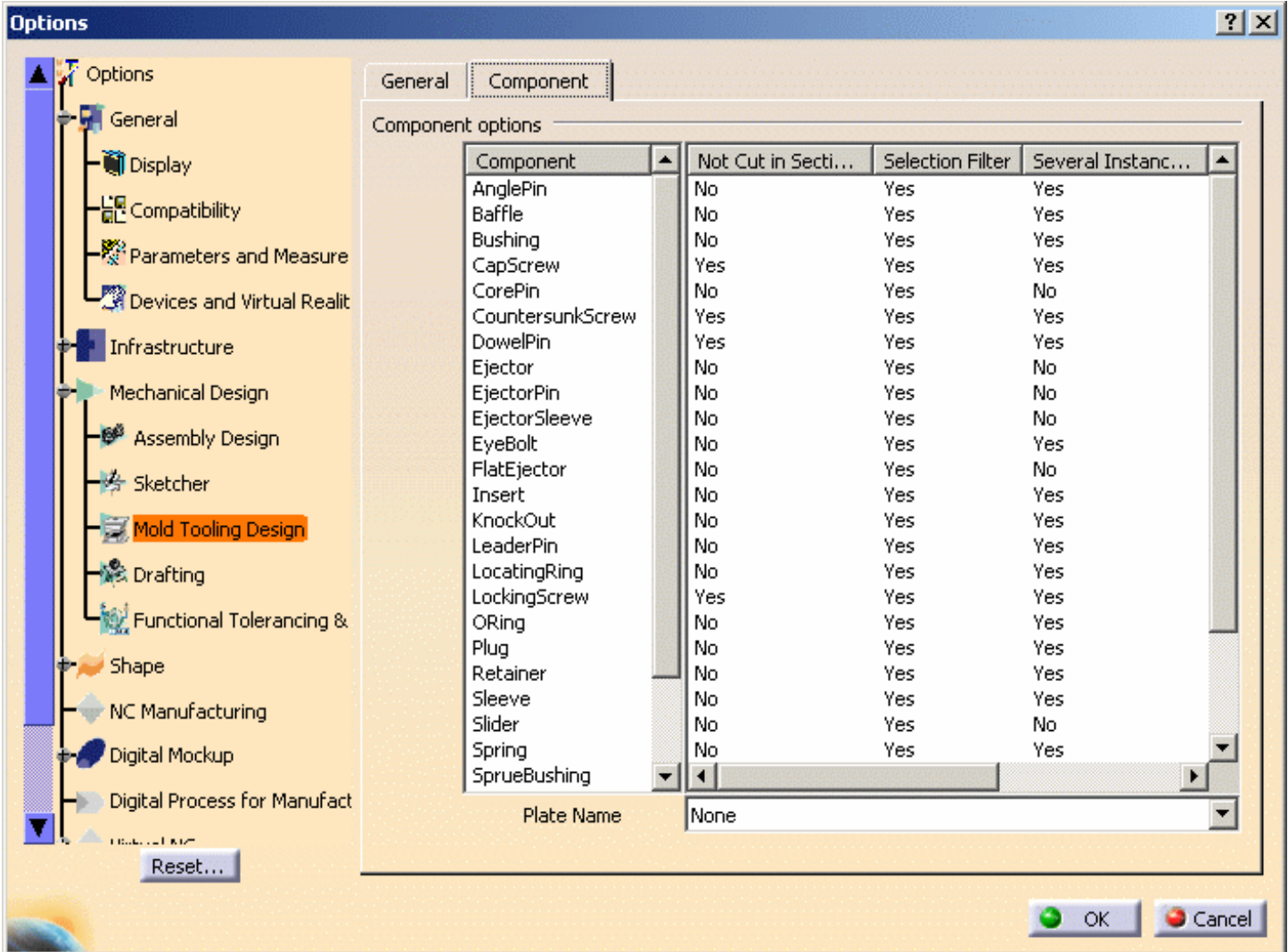


1. Select the **Tools, Options** menu, then **Mechanical Design, Mold Tooling Design** in the specification tree.



2. **Catalog storage Directory** in the **General** tab is the directory where the root catalogs are stored. This field may not be empty. A default directory is proposed. You can add other root catalog storage directories according to your needs. Separate each path by a ";".

3. Then, select the **Component** options.



4. The option 'Not cut in section views' is used to determine whether the component will be visualized in crosshatch display when a cut is being performed in its drafting.

By default, all types of screws (cap screws, countersunk screws, locking screws,...) and dowel pins are not cut; therefore the option is selected by default for these components

5. **Selection filter** allows the user to activate or not know-how rules when creating components. When activated, associated filters are applied in the Catalog Browser during component creation.

By default, the rules are automatically applied to all components:

For a:

- **bushing:** the value of the inner diameter of the bushing (InD) must correspond to the value of the leader pin 's tip diameter (D) if there is one.  
The following filter is therefore activated:  $InD = D$
- **core pin:** the overall length of the core pin (L) must be greater than or equal to H, the height between the bottom of EjectorPlateA and the top of the uncut CorePlate.  
The following filter is therefore activated:  $L \geq H$   
Also, the height of the core pin's guide hole is set by the Offset\_Parting parameter.
- **ejector:** the overall length of the ejector (L) must be greater than or equal to H, the height between the bottom of EjectorPlateA and the top of the uncut CorePlate  
The following filter is therefore activated:  $L \geq H$   
Also, the height of the ejector's guide hole is set by the Offset\_Parting parameter.
- **ejector pin:** the overall length of the ejector pin (L) must be greater than or equal to H, the height from the bottom of EjectorPlateA to the top of the uncut CorePlate.  
The following filter is therefore activated:  $L \geq H$   
Also, the height of the ejector pin's guide hole is set by the Offset\_Parting parameter.  
There is a check on the height value of the hole containing the ejector pin's shouldered part to determine whether it is consistent with the course of the ejection plates; an error message is displayed when there is a risk of collision.
- **ejector sleeve:** the overall length of the ejector sleeve (L) must be greater than or equal to H, the height from the bottom of EjectorPlateA to the top of the uncut CorePlate  
The following filter is therefore activated:  $L \geq H$   
Also, the height of the ejector sleeve's guide hole is set by the Offset\_Parting parameter.

- **flat ejector:** the overall length of the flat ejector (L) must be greater than or equal to H, the height between the bottom of EjectorPlateA and the top of the uncut CorePlate.  
The following filter is therefore activated:  $L \geq H$   
Also, the height of the flat ejector's guide hole is set by the Offset\_Parting parameter.  
There is a check on the height value of the hole containing the flat ejector's shouldered part to determine whether it is consistent with the course of the ejection plates; an error message is displayed when there is a risk of collision
  - **leader pin:** in the case of a standard mold base, use of the DLP diameter parameter (a mold base parameter which is displayed in the specification tree) is recommended by the supplier.  
The following filter is therefore activated:  $D = DLP$ ;
  - **locating ring:** in the case of a standard mold base, use of the DLR diameter (a mold base parameter which is displayed in the specification tree) is recommended by the supplier.  
The following filter is therefore activated:  $ShD = DLR$
  - **sleeve:** the inner diameter of the sleeve (InD) must correspond to the diameter of the leader pin (D) if there is one. The following filter is therefore activated:  $InD \geq D$   
The length of the sleeve (L) must be greater than the height of the riser bars (H). The following filter is therefore activated:  $L > H$ .  
One or two filters can be activated, depending on the current situation.
  - **stop pin:** in the case of a standard mold base, use of the DSP diameter (a mold base parameter which is displayed in the specification tree) is recommended by the supplier.  
The following filter is therefore activated:  $ShD = DSP$
  - **support pillar:** the length of a support pillar (L) must equal the height of the riser bars, i.e. the distance between SettingPlate and CoreSupportPlate or CorePlate.  
The following filter is therefore activated:  $L = H$
- 6. Several Instances per Reference** is active by default for all components (except for core pins, ejectors, ejector pins, flat ejectors, ejector sleeves, spring, sliders, and inserts). This is an option that allows you to create several instances of one reference component. If you deactivate this option, only one instance will be created per reference component.
- 7. Default plate:** use this option to define a default plate for a given component. By default it is set to None.

Select the component, then select a plate from the Plate name list at the bottom.

- 8. Plate Position:** use this option to define a default positioning on a plate for a given component.

By default, this option is set to **Bottom**. Click the label to change it to **Top**.

First select the plate on which the component is to be positioned. The **Top** and **Bottom** option becomes available. Check the proper one.

The next time you will create a component of this type, it will be positioned on the chosen plate, on the selected face, without selecting it.



# Customizing for Drafting

These sections explain how to customize settings for **Drafting**.

[Customizing for Interactive Drafting](#)

[Customizing for Generative Drafting](#)



# General Settings



This task shows you how to set general settings to be used in the Interactive Drafting workbench.



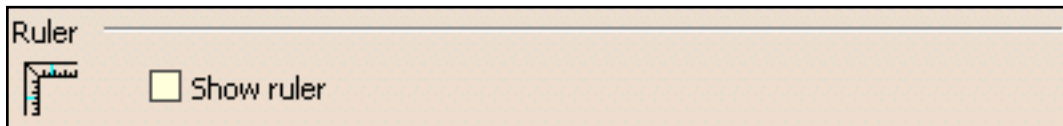
**1.** Select **Tools->Options**. The **Options** dialog box appears.

**2.** Click **Mechanical Design -> Drafting** in the list of objects to the left of the **Options** dialog box.

**3.** Select the **General** tab. It contains several categories of options:

- [Ruler](#)
- [Grid](#)
- [Rotation](#)
- [Colors](#)
- [Tree](#)
- [View Axis](#)

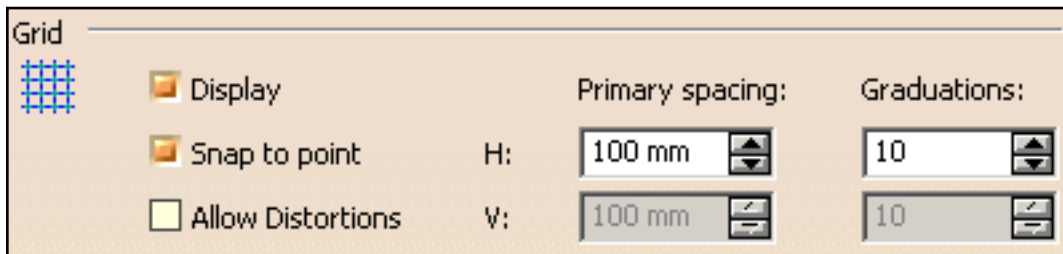
## Ruler



### Show ruler

Check this option to display the ruler in your sheet. It means you visualize the cursor coordinates as you are drawing.

## Grid



## Display

Check this option to display the grid in your session. You will note that this capability is also available via the Drafting **Options** toolbar.

## Snap to point

You need to check this option if the geometry needs to begin or end on the points of the grid.

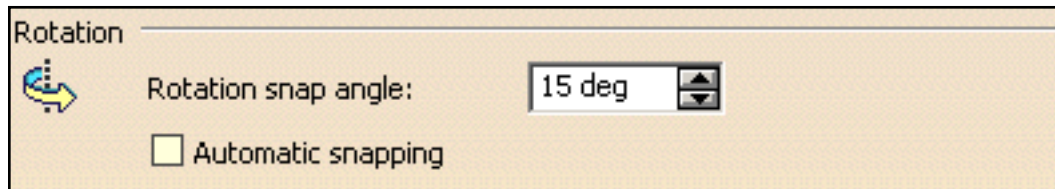
## Allow Distortions

Check this option to apply different graduations and spacing between **H** and **V**.

## H; V; Primary spacing; Graduations

To define your grid, enter the values of your choice in the **H** and **V** fields. The **Primary spacing** option lets you define the spacing between the major lines of the grid. The **Graduations** field lets you set the number of graduations between the major lines of the grid, which actually consists in defining a secondary grid.

# Rotation



## Rotation Snap Angle

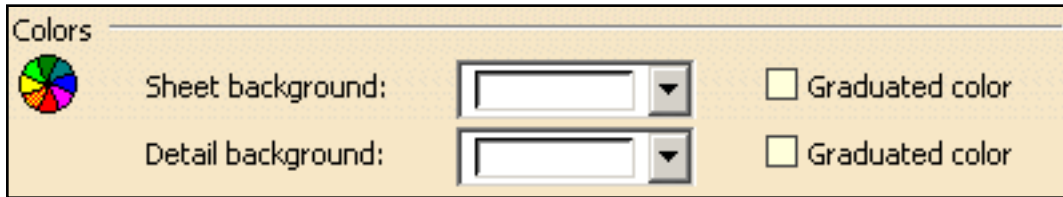
Specify the angle that should be used when rotating text elements (text, frame, or leader) using snapping. In other words, this option defines the snapping value used when rotating an element using the **Select** or **Rotate** commands.

## Automatic Snapping

This option automatically uses snapping when rotating an element.

# Colors

You can customize a number of options for modifying the drawing background color. You can do this at any time.



**Sheet background**

Choose the color that will be used for the sheet background.

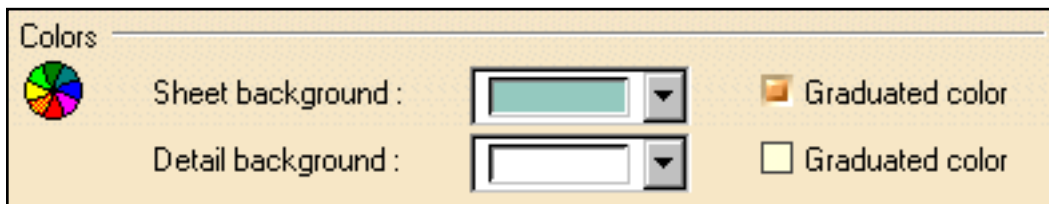
**Detail background**

Choose the color that will be used for the background of 2D components.

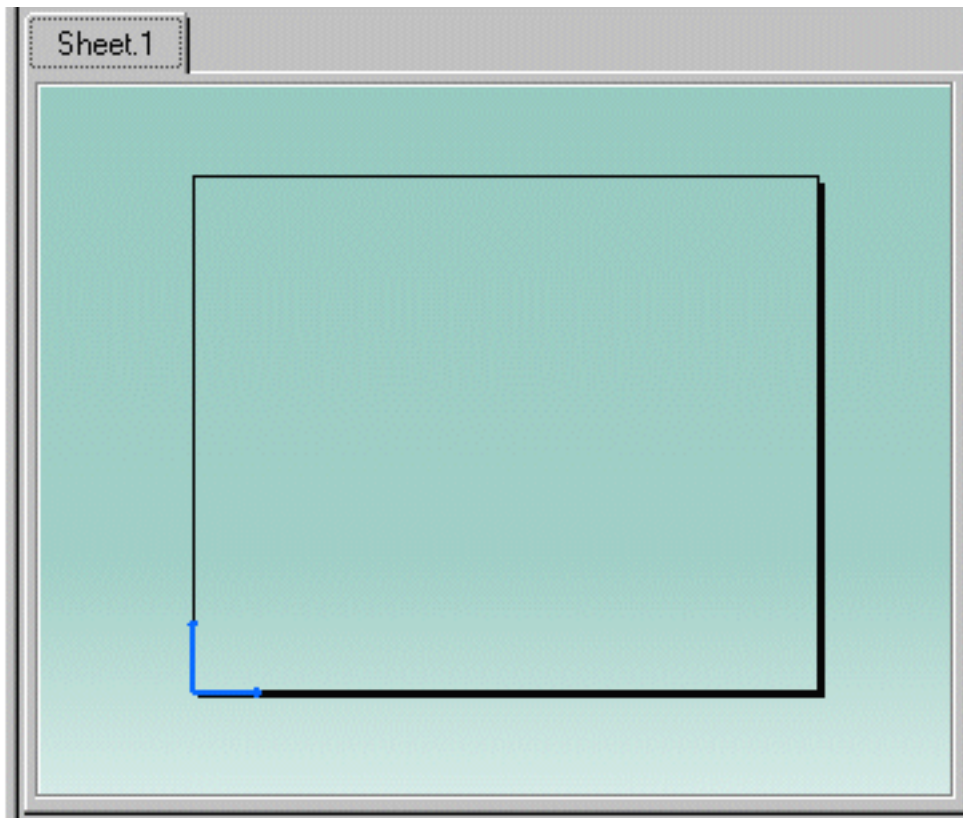
**Graduated color**

If you want the sheet background and/or the detail (i.e. 2D component) background to be graduated, check the associated box.

For example, if you customize the color type as shown below:

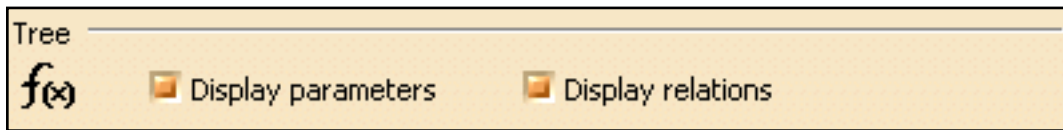


You will get this result:



## Tree

You can choose whether or not parameters and relations should be displayed in the specification tree.



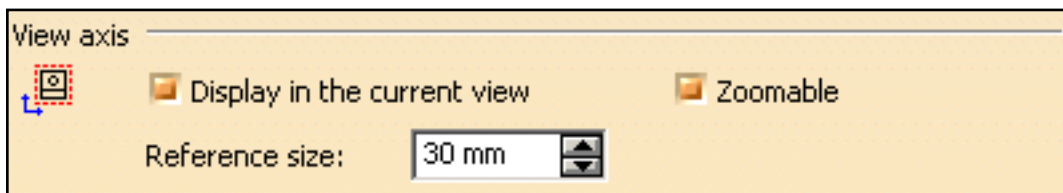
### Display parameters

Check this option to display in the specification tree the formula parameters used in the drawing.

### Display relations

Check this option to display in the specification tree the relation parameters used in the drawing.

## View axis



### Display in the current view

Check this box if you want the view axis to be displayed when you activate a view.

### Zoomable

Check this box if you want to be able to zoom view axes (as you can do with geometry).

### Reference size

Enter the size that you want to use as a reference to display view axes size.

4. Choose the options you want and then click **OK** to validate your settings.



# View and Sheet Layout Settings



You can customize given options when creating views or when adding sheets.



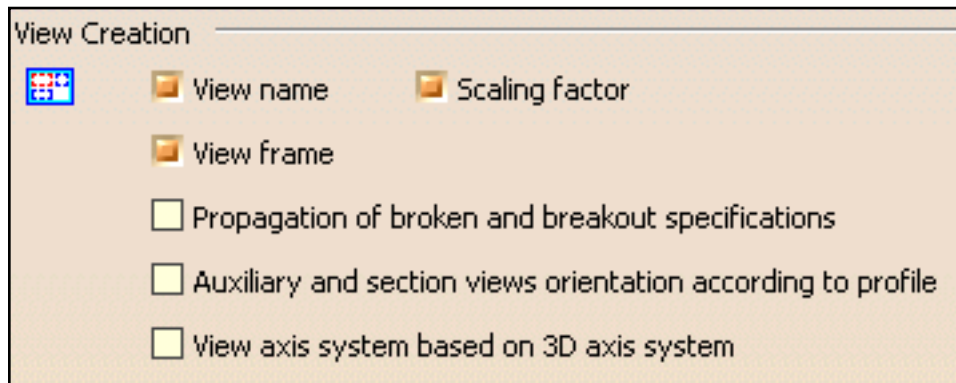
**1.** Select **Tools->Options**. The **Options** dialog box appears.

**2.** Click **Mechanical Design -> Drafting** in the list of objects to the left of the **Options** dialog box.

**3.** Select the **Layout** tab. It contains the following sets of options:

- [View Creation](#)
- [New Sheet](#)
- [Background View](#)
- [Section / Projection Callout](#)

## View creation



### View name

Check this box if you want the view name to be created automatically when creating views.

### Scaling factor

Check this box if you want the scaling factor to be created automatically when creating views.

### View frame

Check this box if you want the view frame to be created automatically when creating views.

### Propagation of broken and breakout specifications

Check this box if you want broken and breakout specifications to be reproduced.

**Auxiliary and section views orientation according to profile**

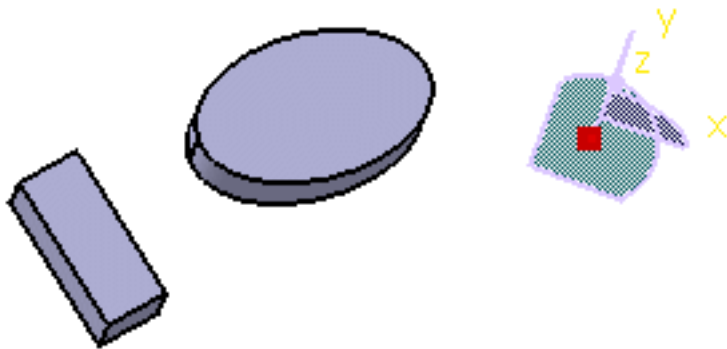
You can decide if auxiliary and section views will be oriented according to the profile. In this case, the X axis will be parallel to the profile.

**View axis system based on 3D axis system**



Check this box if you want the axis system of the generated view to be based on the axis system of the 3D part. This enables you to create views with the same orientation if, when creating two views in the same projection plane by selecting two different faces, the axis systems which are specific to these faces are different.

For example, take the following part:



With the **View axis system based on 3D axis system** option not checked, the view orientation will be different depending on the element selected in the 3D when creating the view:



*View orientation when a face of the rectangular pad is selected*



*View orientation when a face of the elliptic pad is selected*



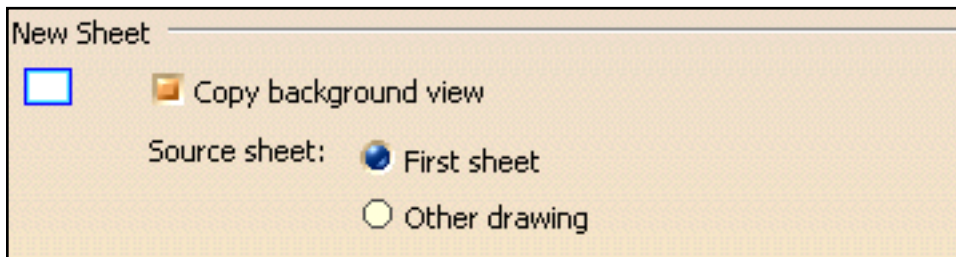
*View orientation when the absolute XY plane is selected*

With the **View axis system based on 3D axis system** option checked, the view orientation will always be the same, no matter what element is selected in the 3D when creating the view:



*View orientation when either a face of the rectangular pad, a face of the elliptic pad, or the absolute XY plane is selected*

## New sheet



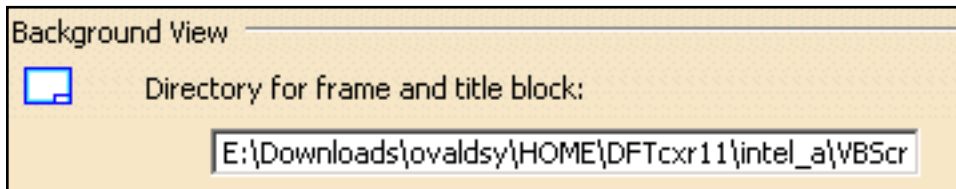
### Copy background view

Check this box if you want a background view to be copied into newly created sheets.

### Source sheet

Specify whether you want the source sheet for the background view to be the first sheet of the current drawing, or a sheet from another drawing by selecting the appropriate option.

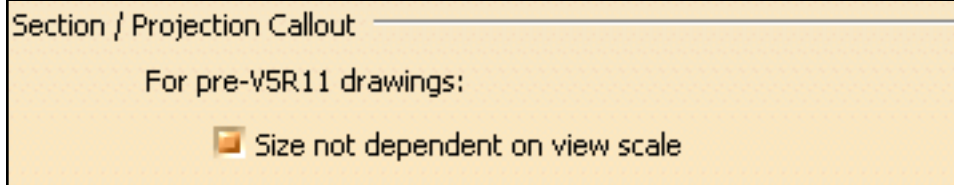
## Background view



### Directory for frame and title block

You can specify the path to the directory containing the frame and title block macros.

## Section/Projection Callout



### **Size not dependent on view scale**

Check this option if you do not want the size of projection and section callout elements to be dependent on the view scale. This option will apply to newly created callouts, i.e. selecting this option will not have any impact on existing callouts.

Note that this option only applies to drawings created with versions prior to V5 R11 (i.e. versions up to V5 R10).

4. Choose the options you want and then click **OK** to validate your settings.





# View Generation Settings

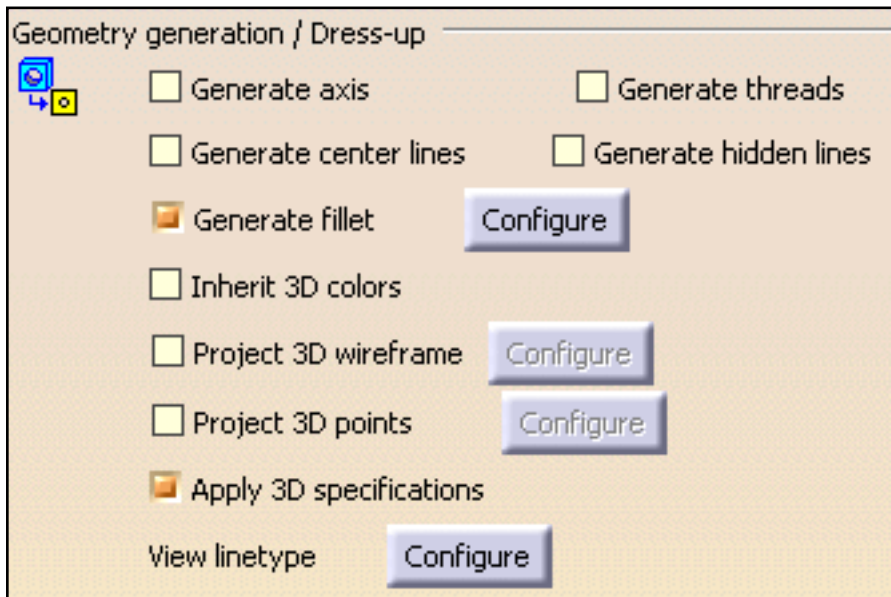


You can customize given options for controlling geometry and dress-up behavior when generating views, as well as view generation.



1. Select **Tools**->**Options**. The **Options** dialog box appears.
2. Click **Mechanical Design** -> **Drafting** in the list of objects to the left of the **Options** dialog box.
3. Select the **View** tab. It contains several categories of options:
  - [Geometry generation / Dress-up](#)
  - [View generation](#)

## Geometry generation / Dress-up



This category of options lets you specify what kind of geometry and dress-up elements you want to generate when generating views.

### Generate axis

Check this option to generate axis lines.

### Generate threads

Check this option to generate threads.

### Generate center lines

Check this option to generate center lines.

### Generate hidden lines

Check this option to generate hidden lines.

### Generate fillets

Check this option to generate fillets. Additionally, click the **Configure** button to configure fillets generation. You can choose to generate either of the following types of fillets:

#### Boundaries

Thin lines, representing the mathematical limits of the fillets.



Boundaries will not be projected if they correspond to two faces which are continuous in curvature. They will be projected only if they correspond to a smooth edge which is situated between two faces whose curvature radii vary.

This mode will be used automatically to represent a connection between two faces which are not joined by a fillet, no matter what option you select.

#### Symbolic

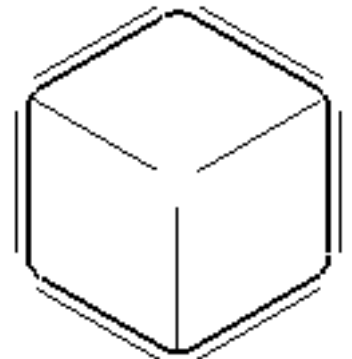
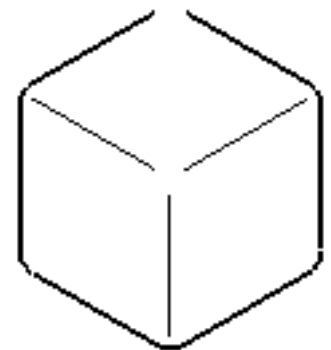
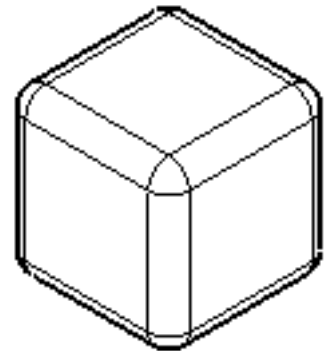
Original edges, projected in a direction that is normal to each corresponding surface.

#### Approximated Original Edges

Original edges, at the intersection of the two surfaces joined by the fillet.

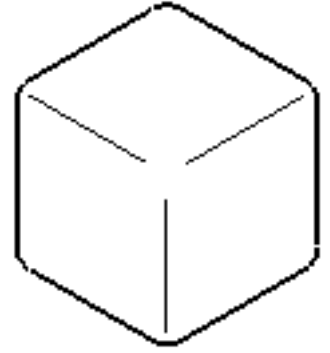
#### Projected Original Edges

Original edges, projected on fillet surfaces in the direction of the view projection.





This projection mode is equivalent to the CATIA V4 fillet projection mode.



Dimensions on **Symbolic**, **Approximated Original Edges** and **Projected Original Edges** are not associative.

### Inherit 3D colors

Check this option if you want the colors of a part to be automatically generated onto the views.



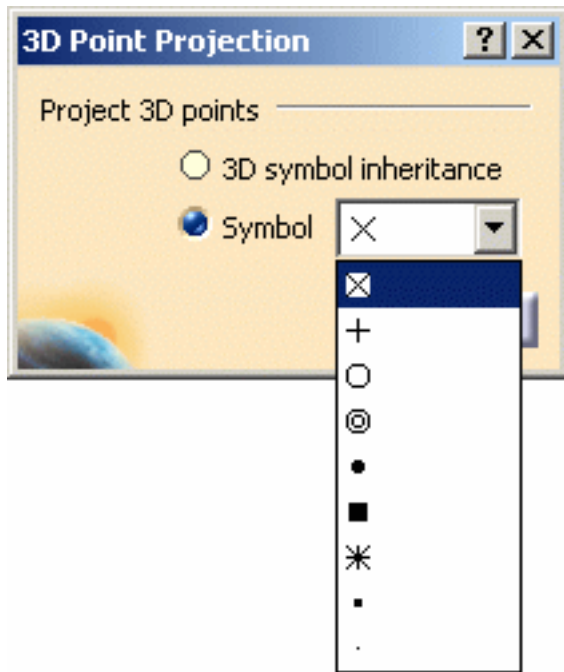
In the case of white parts, the views generated with this option selected will be white, and will therefore not be properly displayed.

### Project 3D Wireframe

Check this option to visualize both the wireframe and the geometry on generated views. Additionally, click the **Configure** button to configure the 3D wireframe projection mode. You can choose whether projected 3D wireframe *can be hidden* (in some cases, depending on the projection angle, part or all of 3D wireframe will possibly be hidden) or *is always visible* (3D wireframe will be visible in all cases, independently of the projection angle).

### Project 3D Points

Check this option to project points from 3D (no construction element). Additionally, click the **Configure** button to select the type of points visualized in the projected drawing. In the 3D Point Projection dialog box, you can choose between keeping the symbols that are used in the 3D or using a new symbol.

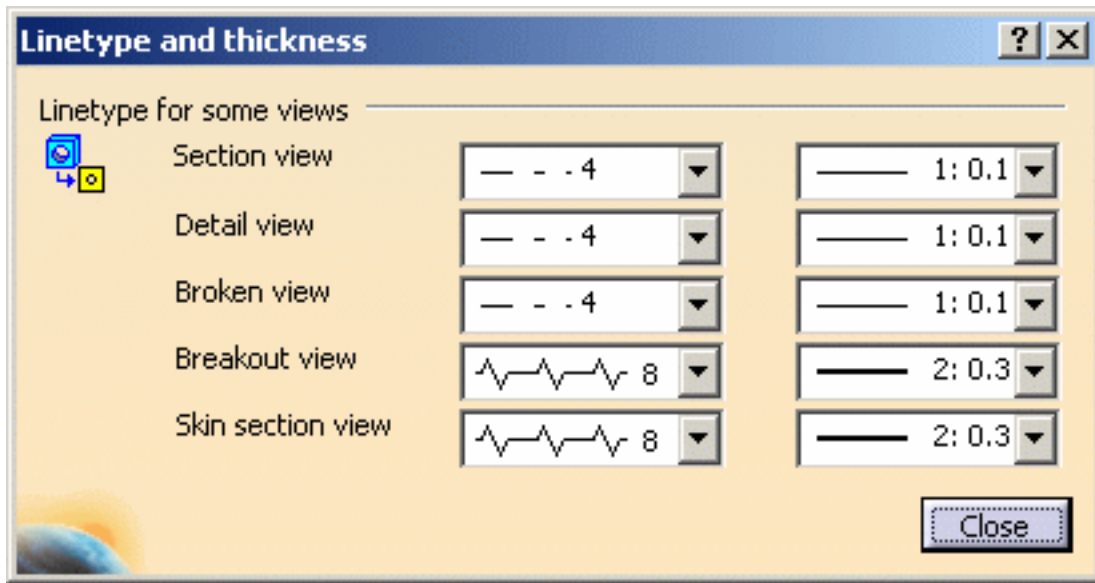


### Apply 3D specification

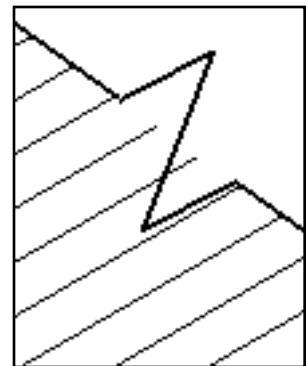
Check this option to specify that, in an assembly, given parts will or will not be sectioned into section views or breakout views (Generative Drafting workbench). For this, you select one view, then the **Edit -> Properties** command from the menu bar from the Assembly Design workbench (**Mechanical** tab, **Drafting properties** options) and either activate or de-activate the **Not cut in section views** option.

### View Linetype

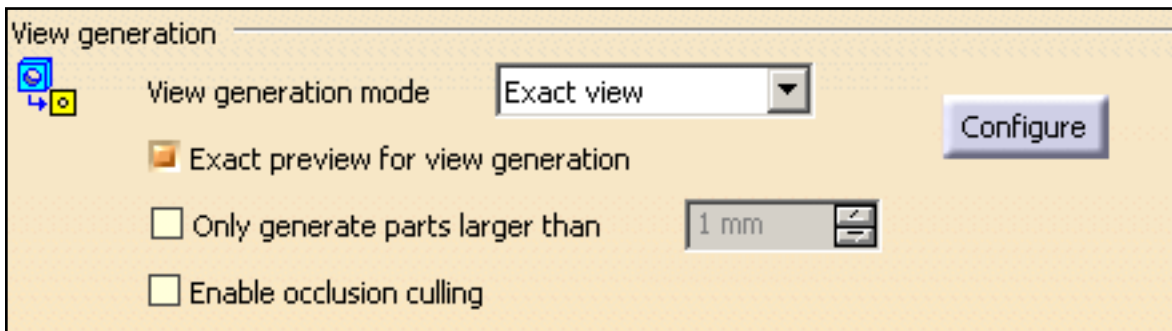
Click the **Configure** button to configure linetypes for specific types of views: section view, detail view, broken view, breakout view, skin section view (in the case of wireframes and surfaces). In the Linetype and Thickness dialog box, select the line type and the thickness you want for each type of view, from the associated fields. Click **Close** when you are done.



If you choose the zigzag linetype (linetype #8), note that this linetype is just a graphical dress-up of the view. This means that if one line is relimited on the breakout line, then it will be relimited on the theoretical line as shown here, and not on the visualized zigzag line.



### View generation



## View generation mode

From this list, select how you want to generate views.

### Exact view

Generates exact views from the Design mode, i.e. views for which the geometry is available. The exact generation mode will be the best option in most cases:

- This is the fastest generation mode.
- All types of views can be generated using this option.
- All functionalities (dress-up, dimensions, annotations, etc.) are available.

However, there are a few cases in which choosing the exact generation mode will not be appropriate:

- In the case of sophisticated products or assemblies involving large amounts of data, generating exact views may consume too much memory.
- Polyhedral elements (such as dittos, surfaces, etc.) from V4 .model documents are not supported.

### CGR

Generates views using the CGR format (CATIA Graphical Representation). CGR corresponds to a data format containing a graphical representation of the geometry only, which is available with the Visualization mode (as opposed to the exact geometry, which is available with the Design mode). With CGR, only the external appearance of the component is used and displayed; the geometry is not available. The corresponding .cgr file, if it exists, is inserted from the cache system.

CGR views are not as high in quality as exact views, but they consume much less memory during the generation. This may be useful when dealing with sophisticated products or assemblies involving large amounts of data. However, this generation mode is rather slow.

For more information about the advantages and restrictions associated with the CGR generation mode, see [Advantages and restrictions common to CGR and Approximate modes](#) below.

## Approximate

You can now generate views in Approximate mode. Although Approximate views are not as high in precision and quality as exact views, this generation mode dramatically reduces memory consumption. Performances may also be improved, depending on how you fine-tune precision. Therefore, the Approximate mode is particularly well-adapted to sophisticated products or assemblies involving large amounts of data.

The Approximate mode offers about the same advantages and restrictions than the CGR generation mode (see above). However, there are some differences:

- Approximate consumes even less memory than CGR.
- Approximate is faster than CGR, but performances largely depend on fine-tuning.
- Approximate provides more approximate results than CGR, but these results largely depend on how you fine-tune precision.

You can fine-tune the generation options according to your needs. Click the **Configure** button. In the dialog box, move the cursor to set the precision (i.e. the level of detail - LOD) with respect to the performances (i.e. generation time - Time). The level of detail corresponds to the precision with which the application determines which edges are hidden and which are not. As a result, decreasing this precision may lead to smaller geometry being visible whenever it should not be, and vice-versa.

The higher the precision, the lower the performances. In any case, memory consumption will not be impacted. Click **Close** when you are done.

For more information about the advantages and restrictions associated with the Approximate generation mode, see [Advantages and restrictions common to CGR and Approximate modes](#) just below.



### Advantages and restrictions common to CGR and Approximate

Using CGR or Approximate to generate views offers the following advantages:

- Optimize memory consumption when generating and handling projection views for large products or assemblies.
- Generate views from third-party data (such as MultiCAD), as well as from polyhedral elements (such as dittos, surfaces, etc.) in V4 .model documents.

However, the CGR or Approximate generation mode involves a number of restrictions:

- You cannot generate section views, section cuts, detail views, detail view profile, breakout views, unfolded views and views from 3D.
- You cannot project 3D elements such as wireframe, points, etc. on CGR or Approximate views.
- CGR or Approximate views cannot contain dress-up elements (axis, center lines, threads).
- Auxiliary view profiles, annotations, dimensions, etc. are not associative on CGR or

Approximate views.

- CGR or Approximate views being only a graphical representation of the geometry, only line segments are generated in such views. As dimensions are not associative, the only elements that can be dimensioned are these line segments. As a result, it is impossible to create certain types of radius or diameter dimensions in such views; to put it simply, you cannot create radius and diameter dimensions on elements other than these line segments.

As a consequence of these restrictions, selecting either the **CGR** or the **Approximate** option disables a number of other options on the View and on the Generation tab.

## Raster

Generates views as images. This enables you to quickly generate overall views for large products or assemblies, regardless of drawing quality. Such views are associative to the 3D geometry and can be updated when the part or product changes.

Raster views offer a number of restrictions:

- You cannot generate the following types of views using this option: view from 3D, section views, section cuts, detail views, breakout views, unfolded views.
- Raster views cannot contain dress-up elements (axis, center lines, threads).
- Creating dimensions is impossible.
- Generally speaking, all commands requiring the selection of geometry are not available.
- Raster views cannot be edited (you can work around this by isolating the view: double-clicking the image will then launch an image editor).

As a consequence of these restrictions, selecting this option disables a number of other options on the View and on the Generation tab.



To optimize disk space and memory consumption, it is recommended that you do not select the **Inherit 3D colors** option when generating views as images.

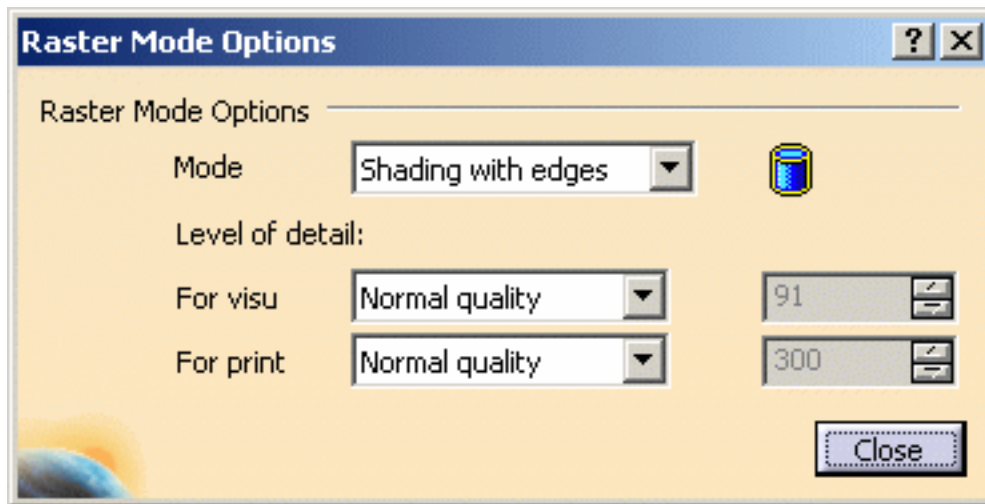



Click the **Configure** button to configure the raster mode options.


From the **Mode** list, select the mode that you want to use: Dynamic Hidden Line Removal, Shading, Shading with edges. These modes are equivalent to the 3D rendering styles. For more information, refer to Using Rendering Styles in the *Infrastructure User's Guide*.


Now, set the level of detail (i.e. the definition, in dpi) that will respectively be used to visualize and to print the drawing. You can choose between three pre-defined modes (Low quality, Normal quality and High quality) and a custom mode (Customize). If you choose to customize the definition yourself, set the dpi for visualization and for print in the appropriate fields.

Click **Close** when you are done.



 The level of detail applies to the scale of the view. In some cases (when the view would print with a considerable height or width), there may be too many pixels to generate the view. In this case, the view will be displayed as a red cross-mark. If this happens, try to reduce the scale of the view and/or the level of detail.

 If you want the colors of a part to be used when generating Raster views using the Shading or Shading with edges mode, remember to select the **Inherit 3D Colors** option. Otherwise, the view will be generated using shades of grey.

 To further improve performance when generating Raster or CGR views, we recommend you work in Visualization mode: to do this, in the Options dialog box, go to **Infrastructure** -> **Product Structure** -> **Cache Management** tab and select **Work with the cache system**. (For more information, see [Customizing Cache Settings](#) in the *Infrastructure User's Guide* and Visualization mode in the *Product Structure User's Guide*.)

## Exact preview for view generation

Make sure this option is selected if you want an exact preview when generating views. As a result, the part or product will be loaded in Design mode when previewing the view to generate, even if you are working in Visualization mode. Deselect this option to get a quick preview of the 3D document when generating views. In this case, a part or product open in Visualization mode will not be loaded in Design mode for the preview, which optimizes memory consumption.

## Only generate parts larger than

To specify that you only want to generate parts which are larger than a certain size, select this option and indicate the appropriate size by providing a value in millimeters in the appropriate field.

## Enable occlusion culling

Select this option if you want to save memory when generating exact views from an assembly which is loaded in Visualization mode (i.e. when the **Work with the cache system** option is active). This will load only the parts which will be seen in the resulting view (instead of loading all of them, which is the case by default), which optimizes memory consumption and CPU usage.

To ensure the efficiency of this option, make sure that the **Exact preview for view generation** option is not selected.

In the case of an assembly which is loaded in Design mode, or in the case of a part, the **Enable occlusion culling** option will help increase performance by reducing CPU usage.



Keep the following restrictions in mind when selecting the **Enable occlusion culling** option:

- The **Project 3D points** option will be disabled.
- The only option available for **Project 3D wireframe** is **Can be hidden**.



- If you choose to project 3D wireframe, you will need to make sure that your wireframe elements have been taken into account when the CGR data was created: this is the case if you activated the **Save lineic elements in cache** option from **Tools -> Options -> General -> Display -> Performances** before the creation of CGR data (i.e. before you launched the part or product in Visualization mode). If not, you need to activate the **Save lineic elements in cache** option and then re-create the CGR data. To do this:

1. Close all open parts and products and exit the application.
2. Delete your CGR data from the cache. (The cache location is specified in **Tools -> Options -> Infrastructure -> Product Structure -> Cache Management** tab, **Path to the local cache field**.)
3. Re-open the product in Visualization mode.

4. Choose the options you want and then click **OK** to validate your settings.

 These options are also available in the Properties dialog box for each view: from the contextual menu, click **Properties**, click the **View** tab and then select the desired options.



# Generation

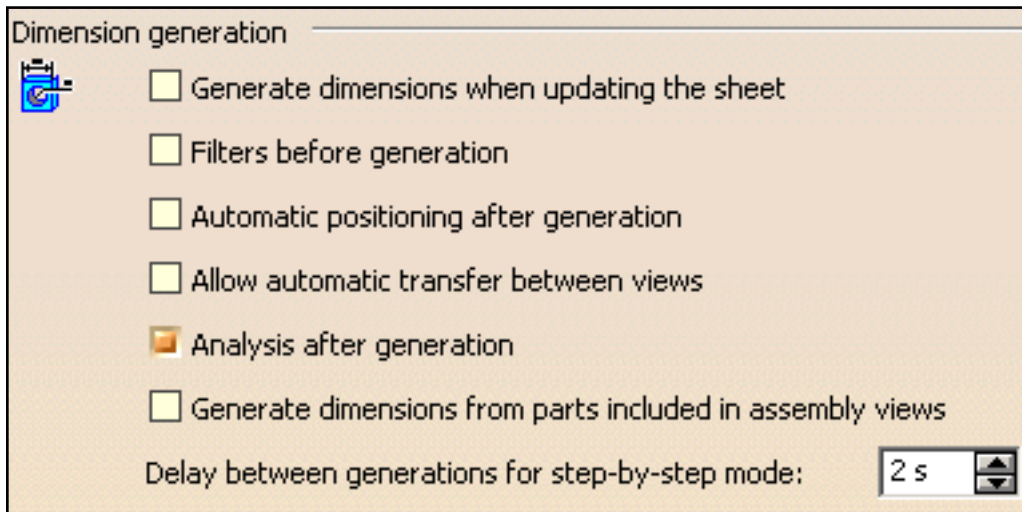


You can customize given options for controlling dimension and balloon generation in generative views.



1. Select **Tools->Options**. The **Options** dialog box appears.
2. Click **Mechanical Design -> Drafting** in the list of objects to the left of the **Options** dialog box.
3. Select the **Generation** tab. It contains several categories of options:
  - [Dimension generation](#)
  - [Balloon generation](#)

## Dimension generation



Generated dimensions are positioned according to the most representative views. In other words, a dimension will appear on a view so that it does not need to be created on another view.

The dimensions are generated on the views on the condition the settings were previously switched to the dimension generation option.

### Generate dimensions when updating the sheet

Check this option to generate dimensions automatically each time you update the sheet.

### Filters before generation

Check this option to display the Dimension Generation Filters dialog box before generation. This enables you to specify what type of dimensions you want to generate. Also, in assembly or product views, this lets you indicate what parts you want to generate dimensions for.

### Automatic positioning after generation

Check this option if you want the dimensions to be automatically positioned after generation.

### Allow automatic transfer between views


Check this option if you want dimensions to be automatically transferred to the most appropriate view when regenerating dimensions.

### Analysis after generation

Check this option to display the Generated Dimension Analysis dialog box after generation.

### Generate dimensions from parts included in assembly views

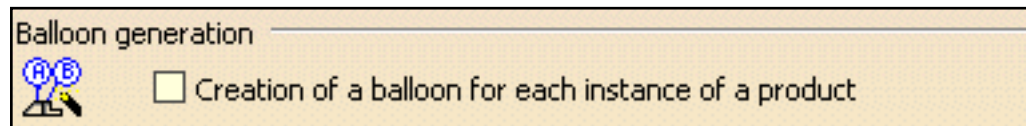
Check this option to extract 3D part constraints (on top of assembly constraints) when generating product dimensions.

 This option is particularly useful if you want to generate dimensions for all parts included in assembly or product views, without displaying the Dimension Generation Filters dialog box before dimension generation. Note that if you display the Dimension Generation Filters dialog box before generating dimensions, you will need to indicate what parts you want to generate dimensions for (whether this option is selected or not).

### Delay between generations for step-by-step mode

Specify the delay between each dimension generation when generating dimensions step by step.

## Balloon generation



### Creation of a balloon for each instance of a product


If you select this option, a balloon will be generated for each instance of a component: therefore, if a component is used two times within a product, then the balloon will be generated twice.


If you leave this box unselected, a single balloon will be generated for all instances of the same component, when a component is used several times within a part or product.

4. Choose the options you want and then click **OK** to validate your settings.



# Geometry Creation

 You can customize given options when creating 2D geometry, either or not using autodetection (or SmartPick), or still adding constraints to this geometry.

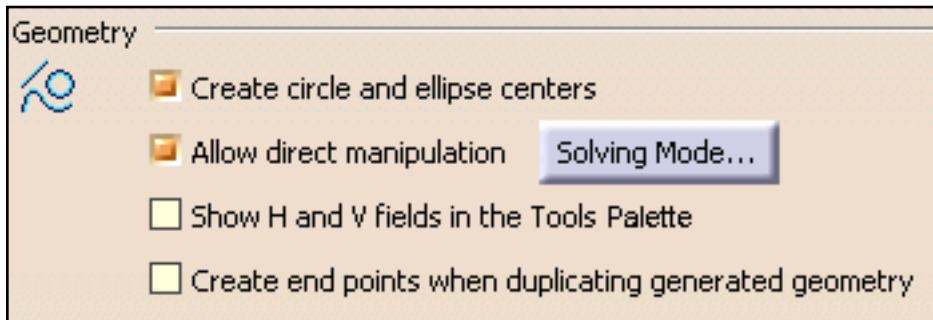
 **1.** Select **Tools->Options**. The **Options** dialog box appears.

**2.** Click **Mechanical Design -> Drafting** in the list of objects to the left of the **Options** dialog box.

**3.** Select the **Geometry** tab. It contains several categories of options:

- [Geometry](#)
- [Constraints creation](#)
- [Constraints Display](#)
- [Colors](#)

## Geometry

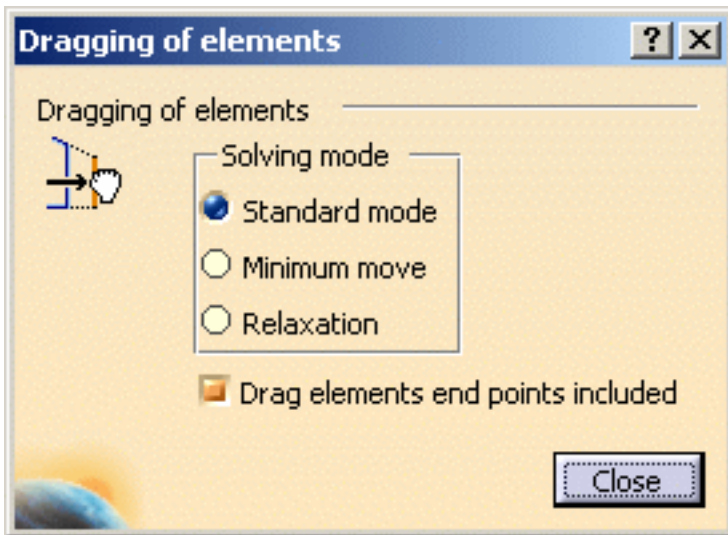


### Create circle and ellipse centers

You can decide whether or not you want to create centers when creating circles or ellipses. By default, this option is activated. Just uncheck it if you do not need to create circle and ellipse centers.

### Allow direct manipulation

Select this option to be able to move geometry using the mouse. When moving geometry, you can move either the minimum number of elements, the maximum number of elements, or still the minimum number by modifying the shape of elements, if needed. Click the **Solving mode...** button to configure manipulation.



The dialog box that appears offers the following options as regards the solving mode:

### **Solving mode for moving elements**

#### **Standard mode**

You move as many elements as possible and also respect existing constraints.

#### **Minimum move**

You move as few elements as possible and also respect existing constraints.

#### **Relaxation**

You move elements by re-distributing them over the sketch, globally speaking. This method solves element moving by minimizing energy cost.

### **Drag elements end points included**

Furthermore, you can choose to drag elements along with their end points by checking this box.

### **Show H and V fields in the Tools Palette**

You can show the H and V fields in the Tools Palette when creating 2D geometry or when offsetting elements. Leaving the option unchecked enables you to directly enter the value corresponding to the type of element you are creating: for example, the length when creating a line, the radius when creating a circle or the offset value when offsetting elements.

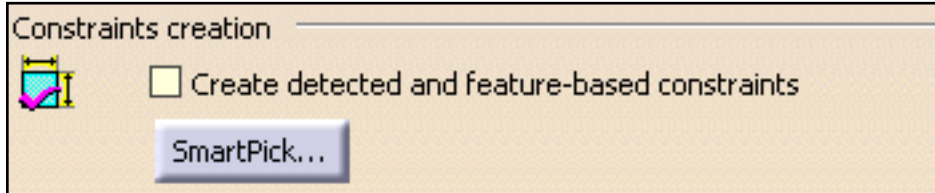


When a command (such as the Point creation command) does not have any parameters other than H and V, then these two fields will remain in the Tools Palette, whether you select this option or not.

## Create end points when duplicating generated geometry

When duplicating geometry that was generated from the 3D, you can choose to create end points for these geometrical elements.

## Constraints creation



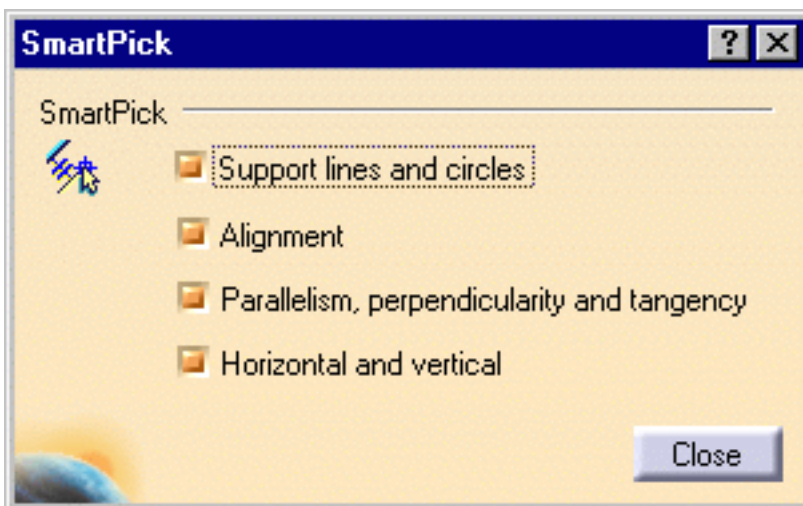
### Create detected and feature-based constraints

Select this option if you want to create the geometrical or dimensional constraints detected by the SmartPick tool. If all of the detection options are unchecked, the **Create detected and feature-based constraints** option is not available.

If this detection option is unchecked, the **Create detected constraints** option will be inactive by default in the Tools toolbar. You will be able to activate it at any time.

### SmartPick... (switch button)

As you create more and more elements, SmartPick detects multiple directions and positions, and more and more relationships with existing elements. This may lead to confusion due to the rapid highlighting of several different detection possibilities as you point the cursor at different elements in rapid succession. Consequently, you can decide to filter out undesired detections by clicking the **SmartPick...** button.





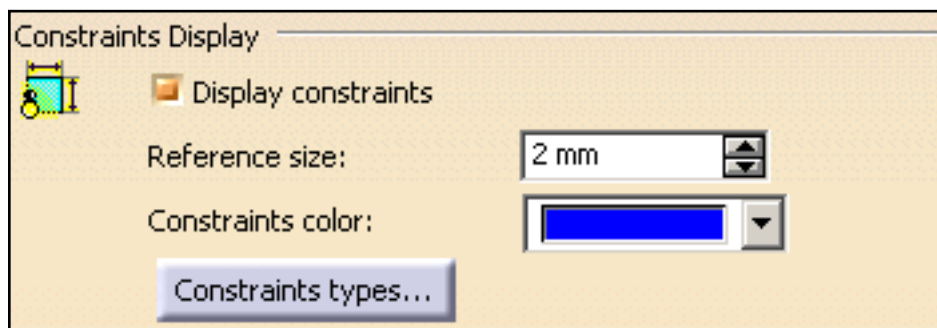
The **SmartPick** dialog box provides these options:

- **Support lines and circles**
- **Alignment**
- **Parallelism, perpendicularity and tangency**
- **Horizontality and verticality**

Uncheck the elements you do not wish to detect when sketching.

Disabling SmartPick completely (i.e. unchecking all options in the SmartPick dialog box) is particularly useful when your screen is full of elements: in this case, it may be a good idea to disable SmartPick to concentrate only on the geometry.

## Constraints Display



### Display constraints

Check this option to visualize the logical constraints specific to the elements. Note that if the **Display constraints** option is unchecked, the other options in this category are not available.

### Reference size

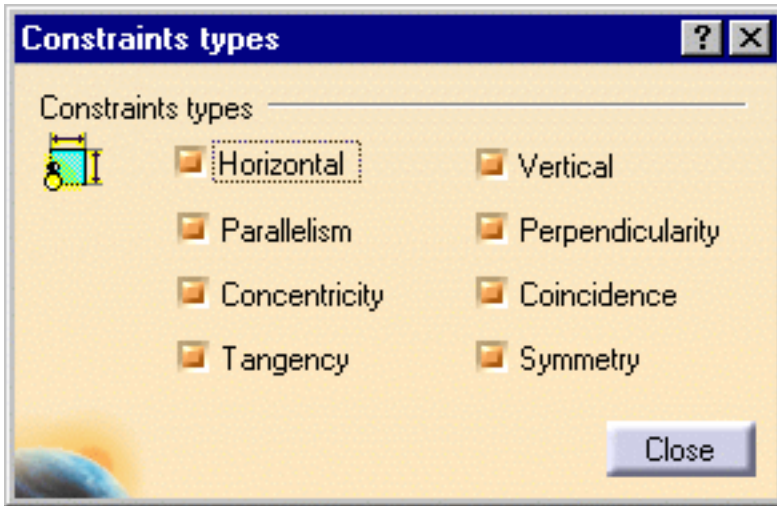
Specify the size that will be used as a reference to display constraints symbols. Changing this reference size will modify the size of all constraints representations.

### Constraints color

Choose the color that will be used to display constraints.

### Constraints types... (switch button)

Click this button to define which types of constraints you will visualize as you create the geometry.

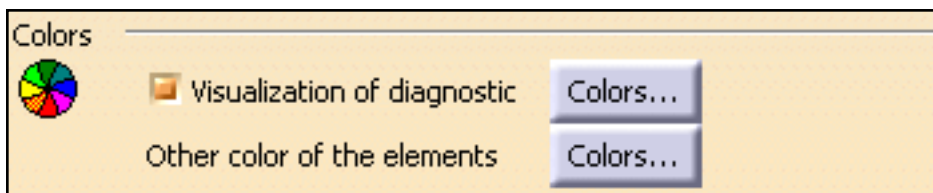


The **Constraints types** dialog box provides these options:

- **Horizontal**
- **Vertical**
- **Parallelism**
- **Perpendicularity**
- **Concentricity**
- **Coincidence**
- **Tangency**
- **Symmetry**

Uncheck the types of constraints you do not want to visualize as you create the geometry.

## Colors



Two types of colors may be applied to sketched elements. These two types of colors correspond to colors illustrating:

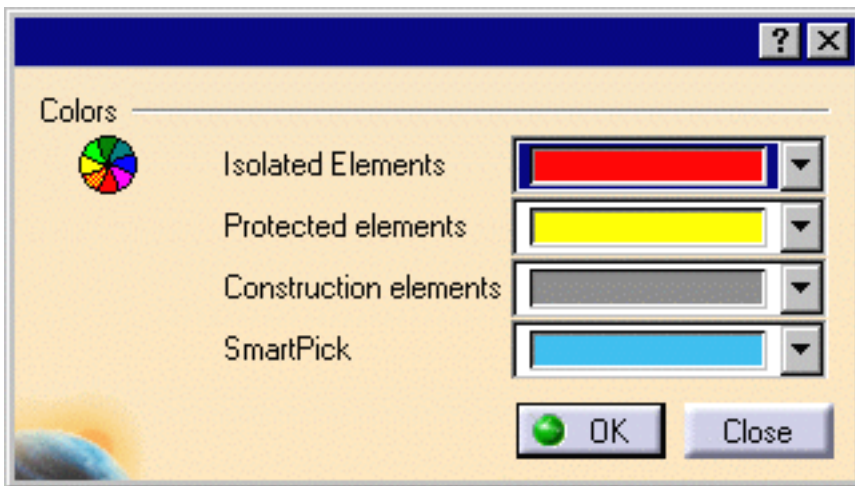
- **Graphical properties**  
Colors that can be modified. These colors can therefore be modified using the **Tools->Options** dialog box.

## OR

- **Constraint diagnosis**  
Colors that represent constraint diagnoses are colors that are imposed to elements whatever the graphical properties previously assigned to these elements and in accordance with given diagnoses. As a result, as soon as the diagnosis is solved, the element is assigned the color as defined in the **Tools->Options** dialog box.

## Visualization of diagnosis

Select this option if you want over-constrained, inconsistent, not-changed or iso-constrained elements to be identified using specific colors. Then, click the **Colors...** button to configure these colors.



In the dialog box that appears, you can configure colors for the following types of elements:

### **Over-constrained elements**

The dimensioning scheme is over-constrained: too many dimensions were applied to the geometry.

### **Inconsistent elements**

At least one dimension value needs to be changed. This is also the case when elements are under-constrained and the system proposes defaults that do not lead to a solution.

### **Not-changed elements**

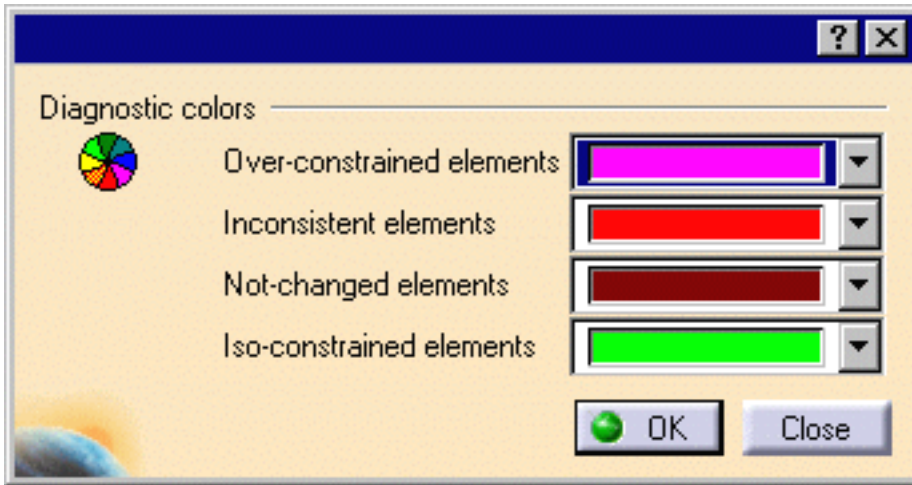
Some geometrical elements are over-defined or not-consistent. As a result, geometry that depend(s) on the problematic area will not be recalculated.

**Iso-constrained elements**

All the relevant dimensions are satisfied. The geometry is fixed and cannot be moved from its geometrical support.

**Other color of the elements**

Click the **Colors...** button to configure the colors of other elements.



In the dialog box that appears, you can configure colors for the following types of elements:

**Isolated elements**

Use-edge (projection, intersection, etc.) that does not depend on the 3D anymore.

**Protected elements**


Non-modifiable elements.

**Construction elements**

A construction element is an element that is internal to, and only visualized by, the sketch. This element is used as positioning reference. It is not used for creating solid primitives.

**SmartPick**

Colors used for SmartPick assistant elements and symbols.

 When opening a drawing, colors are not recomputed. Colors will not be displayed until you create another element or move the geometry.

4. Choose the options you want and then click **OK** to validate your settings.



# Dimension Creation



You can customize given options when creating or re-positioning dimensions.



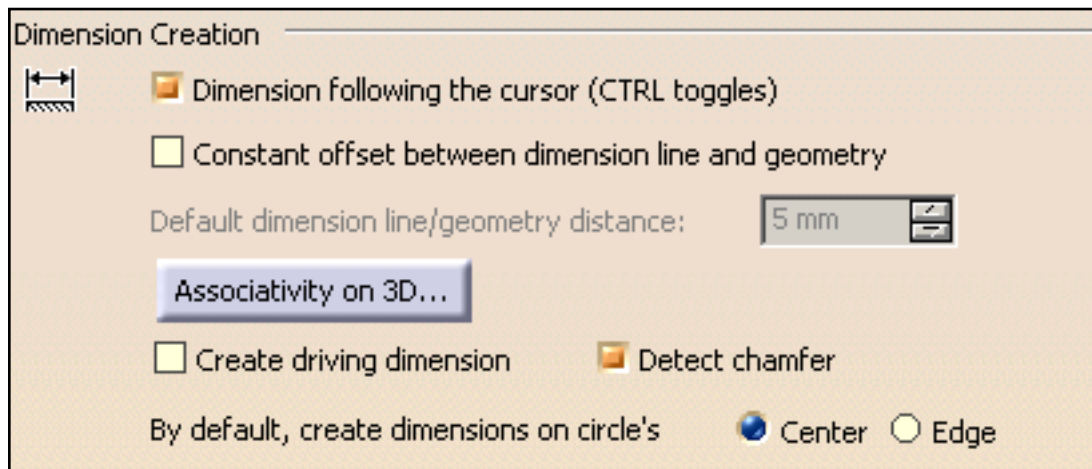
**1.** Select **Tools->Options**. The **Options** dialog box appears.

**2.** Click **Mechanical Design -> Drafting** in the list of objects to the left of the **Options** dialog box.

**3.** Select the **Dimension** tab. It contains several categories of options:

- [Dimension Creation](#)
- [Move](#)
- [Line-Up](#)
- [Analysis Display Mode](#)

## Dimension Creation



### Dimension following the mouse (ctrl toggles)

You can decide that the dimension line is positioned according to the cursor, following it dynamically during the creation process.

### Constant offset between dimension line and geometry

The distance between the created dimension and the geometry remains the same when you move the geometry.

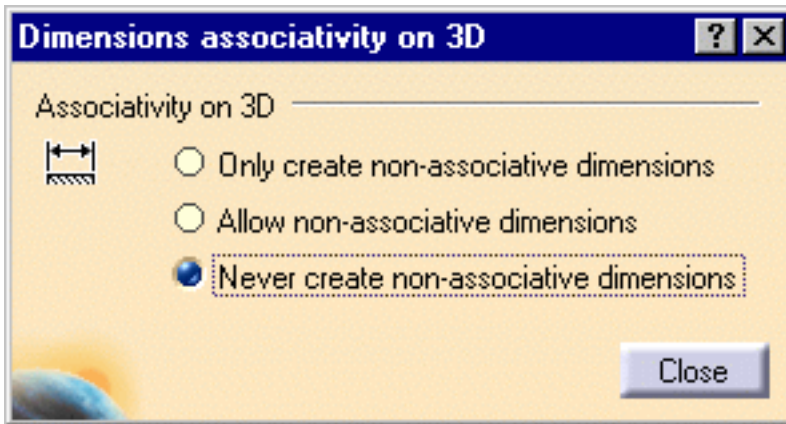
### Default dimension line/geometry distance

If you position the dimension according to the cursor, you can define the value at which the dimension is created. If you create associativity between the dimension and the geometry, you can define the value at which the dimension will remain positioned.

## Associativity on 3D

If you click the switch button, the Dimensions associativity on 3D dialog box appears.

A link can be applied between a dimension and the 3D part. As a result, when you update the drawing, the dimension is automatically re-computed. If you do not check this option, when you perform the update, you need to re-create the dimension afterwards.



## Create driving dimensions

The dimensions you will create will drive the geometry.



A new field will appear in the Tools Palette during the creation process, allowing you to enter the driving dimension value.



## Detect chamfer

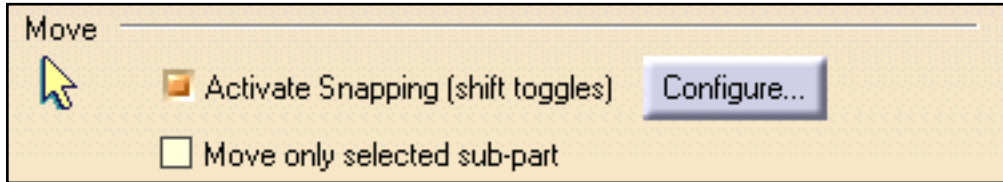
Automatically detects chamfers so that you can create chamfer dimensions in a single click.

As chamfer detection may slow performance down, you may want to deactivate this option for large products or assemblies.

## By default, create dimensions on circle's

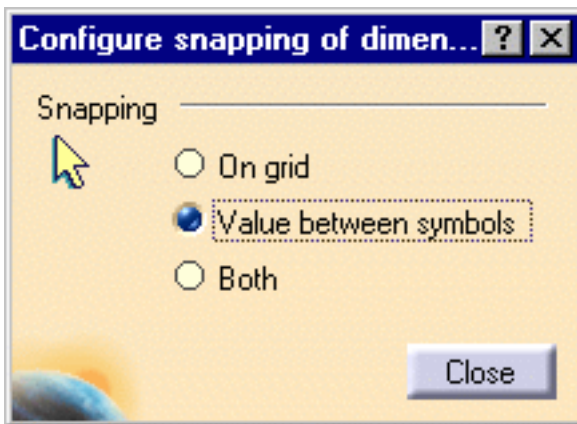
Specify whether the dimension you will create between a circle and another element should be on the circle center or on the circle edge.

## Move



### Activate Snapping (shift toggles)

Select to activate the snapping option. Click the **Configure** button. In the dialog box, specify whether the dimension should be snapped on the grid, or whether the dimension value should be located at its default position between symbols (it will work only if the cursor is between the symbols), or both.

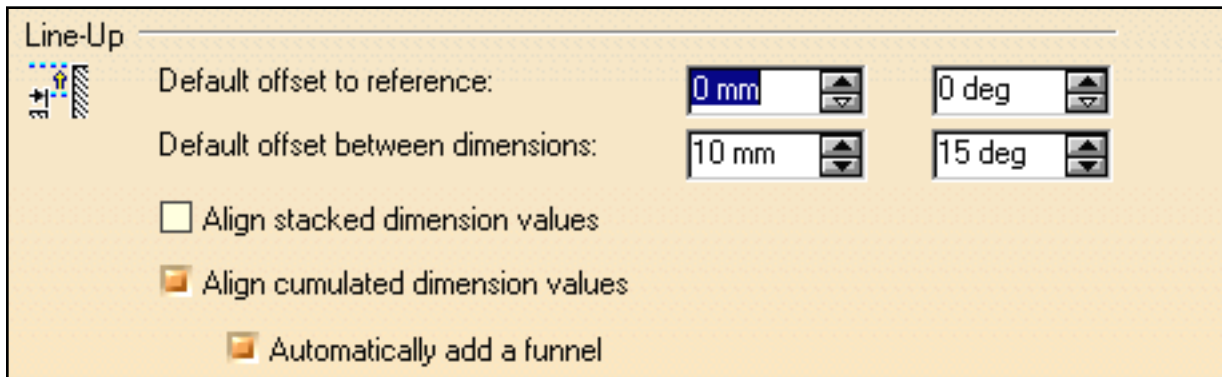


Pressing the **Shift** key allows you to temporarily deactivate or activate this mode.

### Move only selected sub-part

Activate this option if you want to move only a dimension sub-part (text, line, etc.).

## Line-Up



You can organize dimensions into a system with a linear offset. The offset will align the dimensions to each other as well as the smallest dimension to the reference element.

### Default offset to reference

This allows you to set the offset between the smallest dimension and the reference element.

### Default offset between dimensions

This allows you to set the offset between dimensions.

### Align stacked dimension values

Lets you align all the values of a group of stacked dimensions on the value of the smallest dimension of the group.

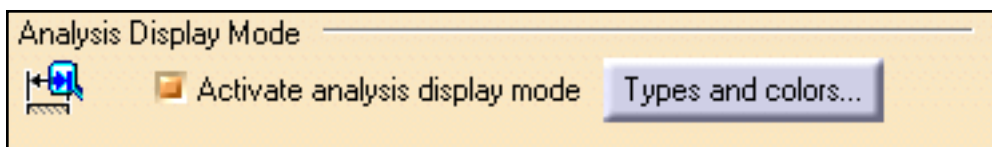
### Align cumulated dimension values

Lets you align all the values of a group of cumulated dimensions on the value of the smallest dimension of the group.

### Automatically add a funnel

Whenever the value of a cumulated dimension requires a funnel to be displayed correctly, lets you have one added automatically.

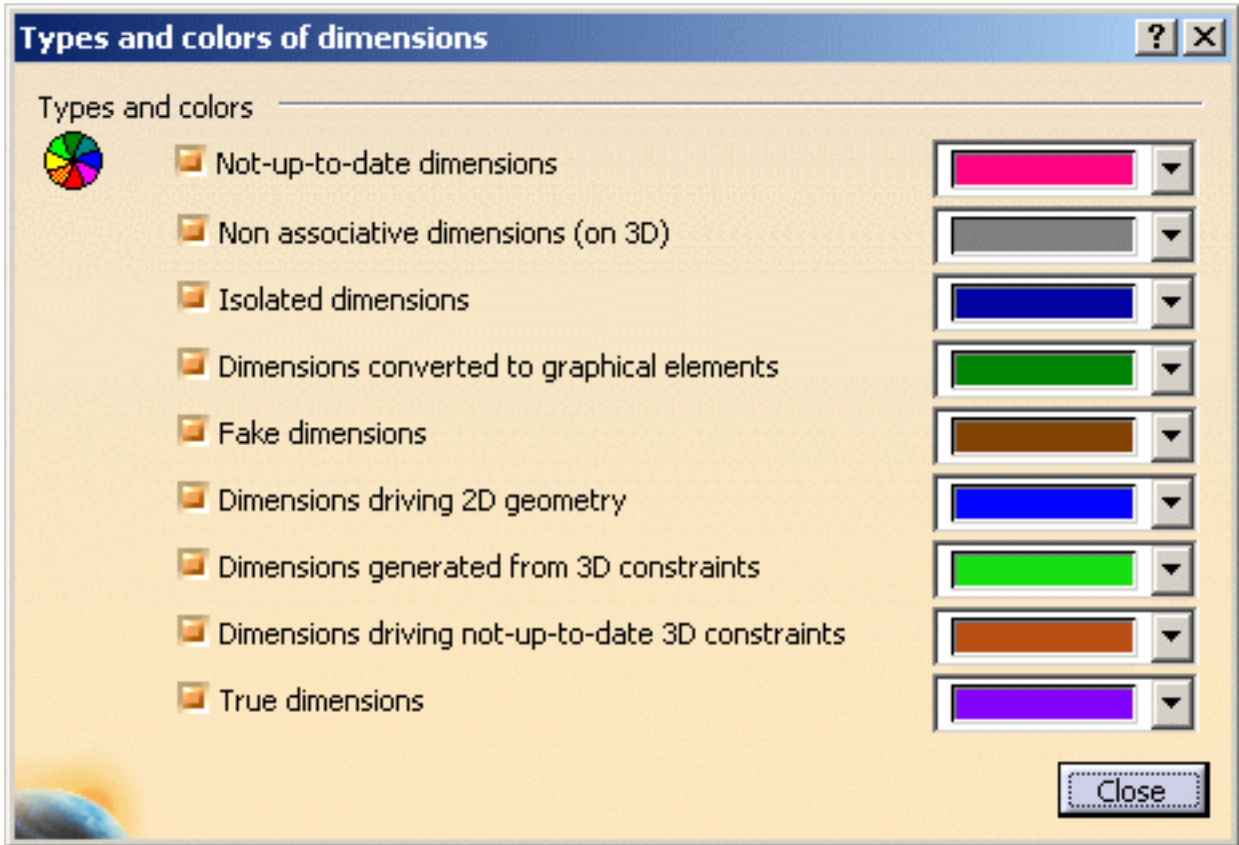
## Analysis Display Mode





**Activate analysis display mode**



Colors can be customized with this option. To activate this mode, select this option and then click the **Types and colors** button. The Types and colors of dimensions dialog box lets you assign the desired color(s) to the selected dimension types. You will then be able to visualize the different types of dimensions using their assigned colors.




4. Choose the options you want and then click **OK** to validate your settings.

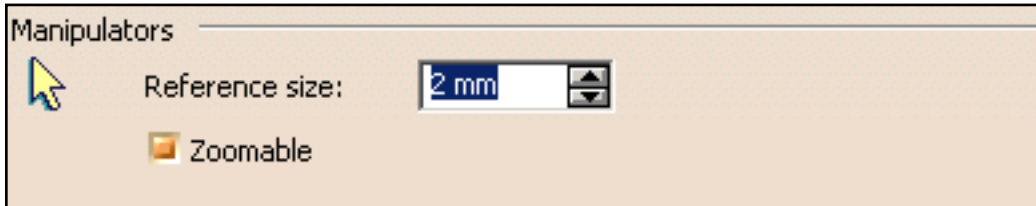


# Manipulators

-  These options apply to the **Interactive Drafting** workbench only.
-  You can decide that you will visualize given manipulators whenever creating or modifying dimensions.

-  **1.** Select **Tools->Options**. The **Options** dialog box appears.
- 2.** Click **Mechanical Design -> Drafting** in the list of objects to the left of the **Options** dialog box.
- 3.** Select the **Manipulators** tab. It contains several categories of options:
  - [Manipulators](#)
  - [Dimension Manipulators](#)

## Manipulators



These settings can be used for any type of manipulator (texts, leaders, center lines, dimensions and so forth).

### Reference size

Specify the reference size that should be used for manipulators. In the case of texts, for example, this reference size corresponds to the diameter of the rotation manipulators.



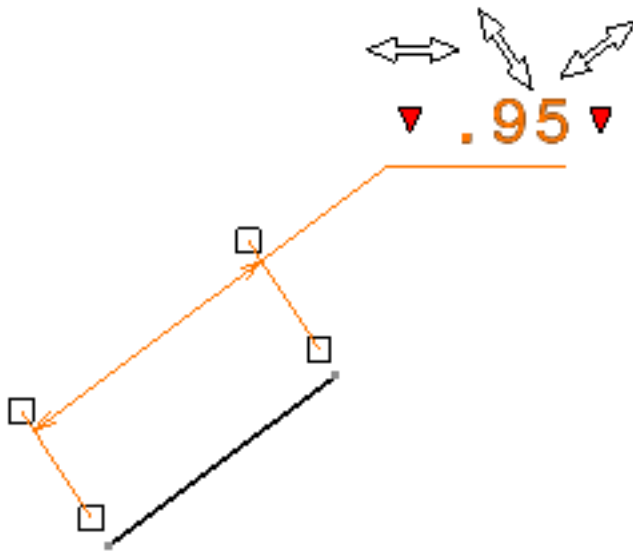
### Zoomable

Check this box if you want to make manipulators zoomable.

## Dimension Manipulators

| Dimension Manipulators              |                                     | Creation                 | Modification                        |
|-------------------------------------|-------------------------------------|--------------------------|-------------------------------------|
|                                     |                                     |                          |                                     |
| <input type="checkbox"/>            | Modify overrun:                     | <input type="checkbox"/> | <input type="checkbox"/>            |
| <input type="checkbox"/>            | Modify blanking:                    | <input type="checkbox"/> | <input checked="" type="checkbox"/> |
| <input checked="" type="checkbox"/> | Insert text before:                 | <input type="checkbox"/> | <input type="checkbox"/>            |
| <input checked="" type="checkbox"/> | Insert text after:                  | <input type="checkbox"/> | <input type="checkbox"/>            |
|                                     | Move value:                         | <input type="checkbox"/> | <input type="checkbox"/>            |
|                                     | Move dimension line:                | <input type="checkbox"/> | <input type="checkbox"/>            |
|                                     | Move dimension line secondary part: | <input type="checkbox"/> | <input type="checkbox"/>            |

These settings let you define which manipulators you will visualize and therefore use when creating and/or modifying dimensions:



### Modify overrun

If you drag select one overrun manipulator, both overrun extension lines are modified. To modify only the selected overrun extension line, use the **Ctrl** key. You can also double-click on the manipulator and enter the new value in the dialog box that appears.

### Modify blanking

If you drag select one blanking manipulator, both blanking are modified. To modify only the selected blanking, use the **Ctrl** key. You can also double-click on the manipulator and enter the new value in the dialog box that appears.

**Insert text before**

Allow inserting a text before, without using the Properties dialog box. For this, you will click on the manipulator and enter the new text in the dialog box that appears.

**Insert text after**

Allows inserting a text after, without using the Properties dialog box. For this, you will click on the manipulator and enter the new text in the dialog box that appears.

**Move value**

Allows moving the dimension value and only it.

**Move dimension line**

Allows moving the dimension line and only it by dragging to the new location.

**Move dimension line secondary part**

Allows moving the dimension line secondary part and only it by dragging to the new location.

4. Choose the options you want and then click **OK** to validate your settings.



# Annotation and Dress-up



You can customize given options that will be used when creating annotations.



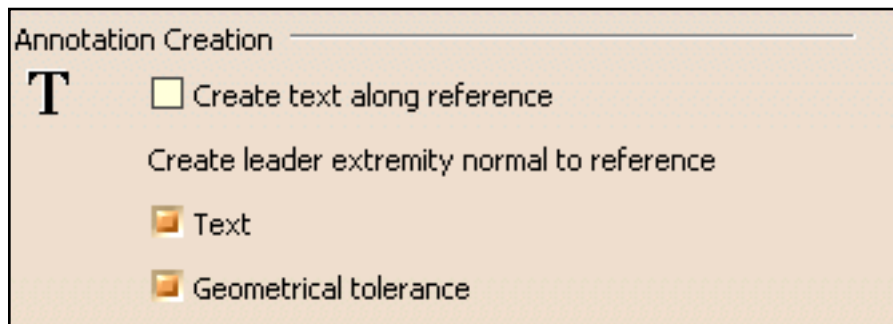
**1.** Select **Tools->Options**. The **Options** dialog box appears.

**2.** Click **Mechanical Design -> Drafting** in the list of objects to the left of the **Options** dialog box.

**3.** Select the **Annotation and Dress-Up** tab. It contains the following sets of options:

- [Annotation Creation](#)
- [Move](#)
- [2D Component Creation](#)
- [Balloon Creation](#)

## Annotation Creation



In order for these options to be taken into account, the **Activate Snapping (SHIFT toggles)** box must be checked. Note that the option selected in the Activate snapping dialog box will be taken into account. See the [Move](#) section.

### Create text along reference

Select this option if you want to create annotation texts along a reference direction. For example, if you select a line when creating a text, the text will be oriented parallel to the line.

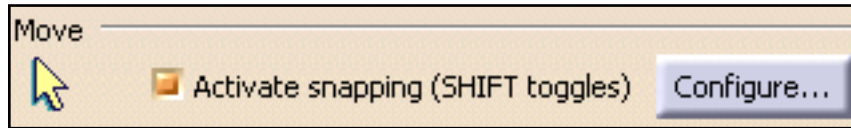
### Text

Select this option if you want to create the extremity of text leaders normal to a reference direction. For example, if you select a line when creating a text with leader, the leader will be normal to the line.

### Geometrical tolerance

Select this option if you want to create the extremity of geometrical tolerance leaders normal to a reference direction. For example, if you select a line when creating a geometrical tolerance, the leader will be normal to the line.

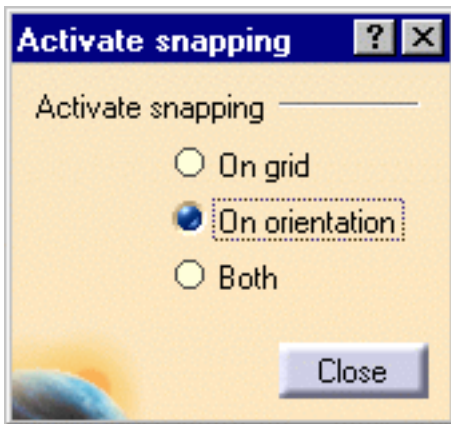
## Move



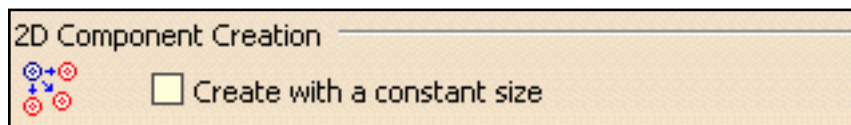
### Activate Snapping (SHIFT toggles)

Select this option to activate snapping. Click the **Configure** button.

In the dialog box that appears, specify whether you want the annotation to be snapped on the grid, according to the orientation, or both. This will apply to the annotations selected in the Annotation Creation area. To deactivate snapping when creating or moving annotations, press the **Shift** key.



## 2D Component Creation



### Create with a constant size

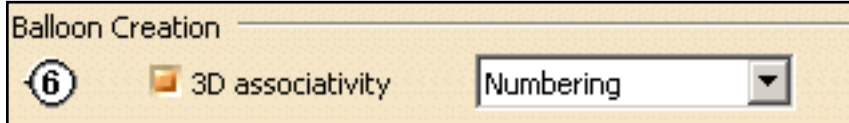
Select this option if you want all 2D component instances to have the same size when you create them, no matter what the view scale is.

This lets you create 2D component instances whose size is independent from the view scale so that they always look the same. You can use them as symbols, for example.



If you want to use as symbols 2D components with text, activate both the **Create with a constant size** setting and the **Apply Scale** property for the text (in **Edit -> Properties**): the size of both the 2D component and its text will then be independent from the view scale.

## Balloon Creation



### 3D associativity

You can specify what kind of balloons you want to create (using the Balloon command from the Annotation toolbar) or to generate (using the Generate Balloons command from the Generation toolbar).

First, select the **3D associativity** box to indicate that you want to associate balloons with information from the 3D. Then, select from the list the kind of balloons you want to create or generate: the numbering of parts within an assembly (default option), the instance name or the part number.

4. Choose the options you want and then click **OK** to validate your settings.



# Administration Settings



You can customize settings for the management of drawings.



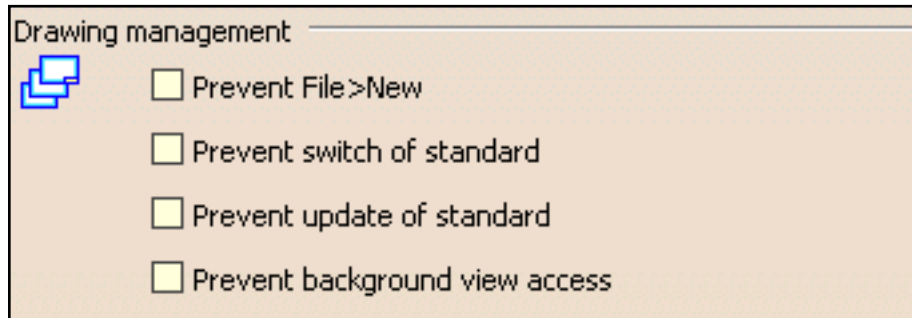
**1.** Select **Tools -> Options**. The **Options** dialog box appears.

**2.** Click **Mechanical Design -> Drafting** in the list of objects to the left of the **Options** dialog box.

**3.** Select the **Administration** tab. It contains the following sets of options:

- [Drawing management](#)
- [Style](#)
- [Generative view style](#)
- [Dress-up](#)

## Drawing management



### Prevent File>New

Check this box to make it impossible to create drawings using the **File -> New** command. All drawings will be created using the **File -> New From...** command instead.

### Prevent switch of standard

Check this box to make it impossible to change standards, i.e. to use a standard other than the one currently defined in the Page Setup dialog box.

### Prevent update of standard

Check this box to make it impossible to update standards for the current document in the Page Setup dialog box.

### Prevent background view access



Check this box to make it impossible to access the background view.

## Style

Style

Use style values to create new objects

Create new sheet from:     Style     First sheet

For pre-V5R11 drawings:

Lock "User Default" style

Prevent "Set As Default" and "Reset All Defaults"

### Use style values to create new objects

Check this box if you want dialog boxes, Properties toolbars and the Tools Palette to be pre-filled with custom style values (as defined in the Standards Editor) when creating new annotations. In this case, Properties toolbars and the Tools Palette will be disabled during the creation of the annotation.

If you leave this box unchecked, annotation dialog boxes, Properties toolbars and the Tools Palette will be pre-filled with the last entered values (except for Texts, Texts with leader, Balloons and Datum features). In this case, Properties toolbars and the Tools Palette will be active during the creation of the annotation.

If you check this box, you will be able to reset the current style values in dialog boxes at any time using the **Reset** button.

### Create new sheet from

This setting lets you specify if the properties used for creating new sheets should be those defined in the standards or those defined in the first sheet in a drawing. These properties are the scale and the projection method (first or third angle).

Select **Style** if you want the sheet to use the style defined in the standards (in **Tools -> Standards -> Drafting -> [StandardName] -> Styles -> Sheet**).

Select **First sheet** if you want the sheet to use the properties defined in the first sheet in a drawing. For example, you can use this option if you use an existing drawing to create a new one (i.e. when you want the new drawing to have the same properties as the existing drawing).

### Lock "User Default" Style

Check this box to make it compulsory to use User Defaults (i.e., user-defined values set as default). The **Styles** drop-down list will be set to **Only User Defaults** and will be inactive so that **Original Defaults** or **User Defaults** cannot be selected.

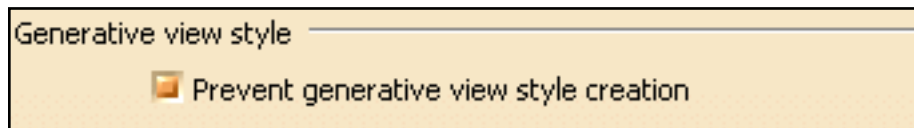
*This option applies only to drawings created with versions up to V5 R10 whose standard has NOT been updated or changed in V5 R11 and later.*

### Prevent "Set As Default" and "Reset All Defaults"

Check this box to use the current defaults and to make it impossible to create, change and reset user defaults (i.e. user-defined values). This disables the **Set as Default** and the **Reset All Defaults** commands.

*This option applies only to drawings created with versions up to V5 R10 whose standard has NOT been updated or changed in V5 R11 and later.*

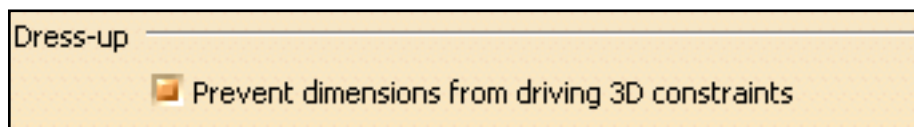
## Generative view style



### Prevent generative view style creation

Check this box if you do not want to use generative view styles when creating views. In this case, you will not be able to select a generative view style after having selected a view creation command, which means that the Generative View Style toolbar will not be displayed. (In the case of advanced front views, it is the Generative view style list in the View Parameters dialog box which will not be displayed).

## Dress-up




### Prevent dimensions from driving 3D constraints


Check this box to make it impossible to modify a 3D constraint via a 2D dimension that was generated from it.

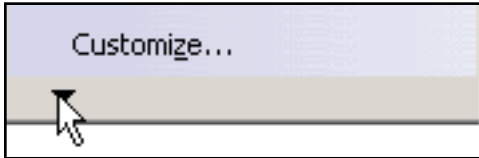
4. Choose the options you want and then click **OK** to validate your settings.



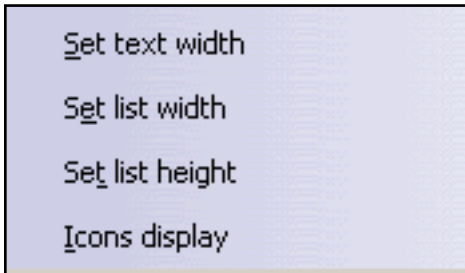
# Customizing Toolbars


 You can customize the appearance of some fields in the following properties toolbars: Styles, Graphic Properties, Text Properties, Dimension Properties.

-  **1.** Right-click the toolbar field you want to customize. A contextual menu is displayed.
- 2.** If necessary, scroll down this contextual menu to display the toolbar customization options.




The customization options that you can apply to the selected field are displayed.



 The options available depend on the selected field. For more information on what options will be available for each field, see the table below.

- 3.** Click the option you want. Depending on the option you selected, the corresponding dialog box appears.
- 4.** Enter the appropriate value in the dialog box.
  - **Set text width:** sets the width used to display the field in the toolbar, in number of characters to be displayed (based on 'W').
  - **Set list width:** sets the width used to display the drop-down list, in number of characters to be displayed (based on 'W').
  - **Set list height:** sets the height used to display the list, in number of lines to be displayed (up and down arrows will make it possible to scroll within the list).
  - **Icons display:** defines whether icons should be displayed in this field, or only in the list, when the list is collapsed.
  - **Precision:** sets the precision used to display a numerical value in this field, in number of digits after the separator.
- 5.** Click **OK** to validate.

 The table below indicates which fields you can customize in each toolbar, along with what you can customize for each field.

|                                     | <b>Set text width</b> | <b>Set list width</b> | <b>Set list height</b> | <b>Icons display</b> | <b>Set precision</b> |
|-------------------------------------|-----------------------|-----------------------|------------------------|----------------------|----------------------|
| <b>Style toolbar</b>                |                       |                       |                        |                      |                      |
| Style                               | Yes                   | Yes                   | Yes                    | Yes                  | No                   |
| <b>Text Properties toolbar</b>      |                       |                       |                        |                      |                      |
| Font Name                           | Yes                   | Yes                   | Yes                    | Yes                  | No                   |
| Font Size                           | Yes                   | Yes                   | Yes                    | No                   | Yes                  |
| <b>Dimension Properties toolbar</b> |                       |                       |                        |                      |                      |
| Tolerance Description               | Yes                   | Yes                   | Yes                    | Yes                  | No                   |
| Tolerance                           | Yes                   | Yes                   | Yes                    | No                   | No                   |
| Numerical Display Description       | Yes                   | Yes                   | Yes                    | No                   | No                   |
| Precision                           | Yes                   | Yes                   | Yes                    | No                   | Yes                  |
| <b>Graphic Properties toolbar</b>   |                       |                       |                        |                      |                      |
| Color                               | Yes                   | No                    | No                     | No                   | No                   |
| Thickness                           | Yes                   | No                    | Yes                    | No                   | No                   |
| Linetype                            | Yes                   | No                    | Yes                    | No                   | No                   |
| Point type                          | Yes                   | No                    | Yes                    | No                   | No                   |



# Customizing For Aerospace Sheet Metal Design

This section describes how to customize different settings specific to the Aerospace Sheet Metal Design workbench. The settings described here deal with permanent setting customization.

[Customizing Standards Files To Define Design Tables](#)

[Customizing General Settings](#)


[Customizing Standards Files To Define Methods for Compensations](#)

# Customizing Standards Files To Define Design Tables

This section describes how to customize settings.  
The task described here deals with permanent setting customizing.

## Using Sheet Metal Standards Files

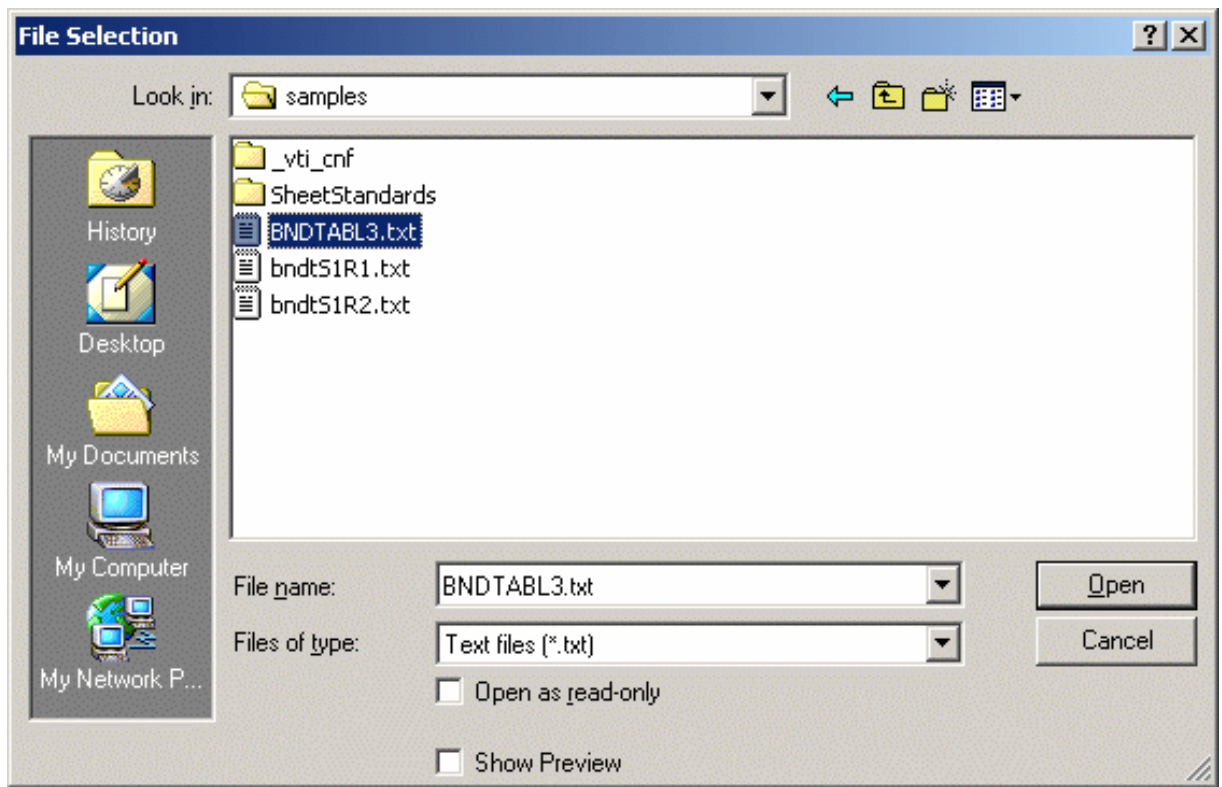
 This task explains how to access company standards files in order to access and define design tables.


 Open a new document.

 **1. Click the Sheet Metal Parameters icon** .  
The Sheet Metal Parameters dialog box opens.

**2. Select the Sheet Standards Files...** button. The File Selection window is displayed.

**3. Indicate the path to the Sheet Metal table.**



 These files are available under .txt format or .xls format (only for NT and Windows)

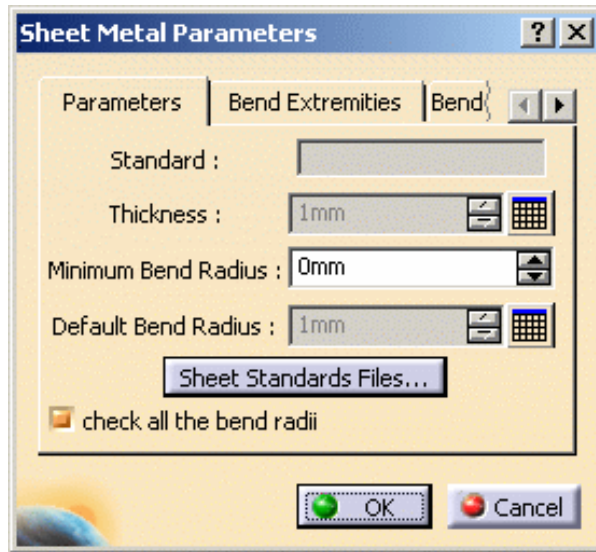
**4. Click Open.**

In the Sheet Metal Parameters dialog box, the **Design Table** icon




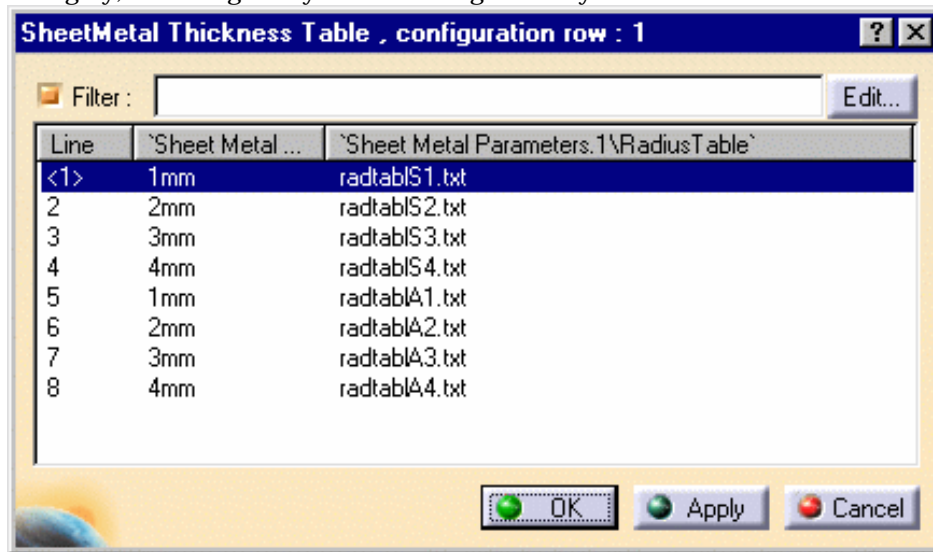
appears opposite the Thickness and Bend radius

fields.



The parameters are now in gray, indicating that you can no longer modify the values.

- Click the Thickness Design Table icon  and select the line containing the appropriate parameters (for example Line 1).




Using the **Tools -> Options -> General -> Document** tab, **Other Folders** option, you can specify where the files are located. Refer to [Document](#).

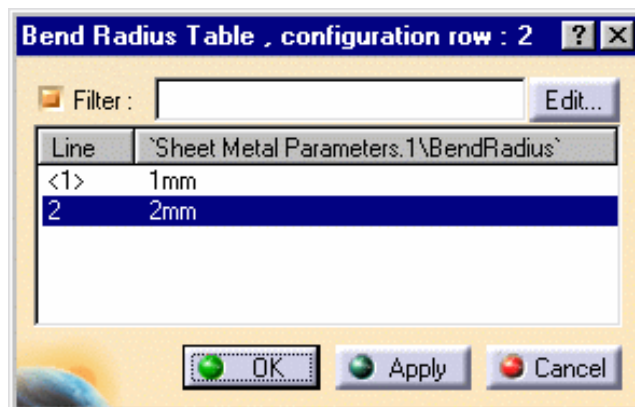
This scenario can work when the .CATPart document and all reference table files (Design & Radius) are located in the same directory. This directory is the current one when the Design table is created, and also when the .CATPart is open. However, generally speaking, you must reference the complete path indicating where the radius table files are to be found in the RadiusTable column. In this case, regardless of the current directory, the correct tables are located when re-opening the .CATPart document.

- Click **OK**.

The parameter values are updated in the Sheet Metal Parameters dialog box.

- Click the Bend Radius Design Table icon .

- Select line 2 and click **OK**. The parameter values are updated in the





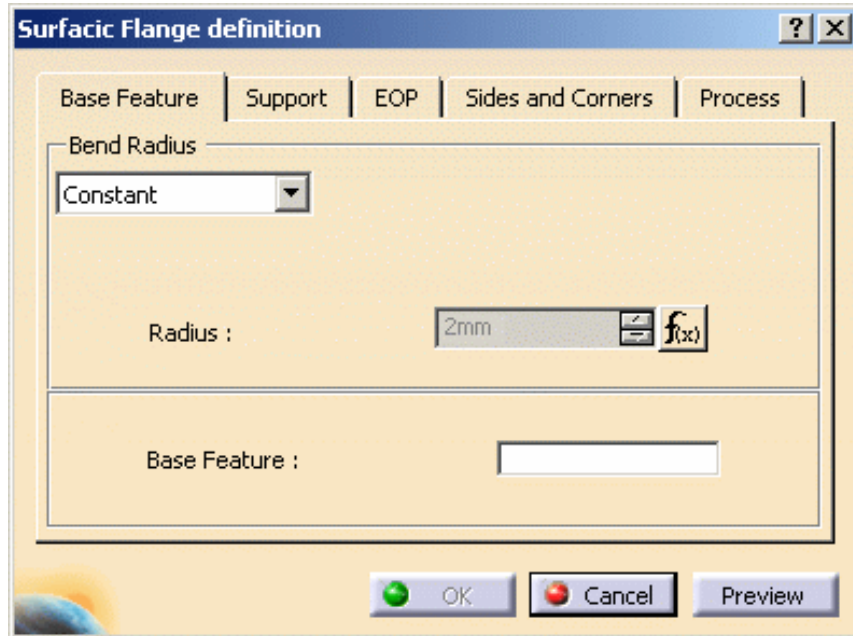
Sheet Metal  
Parameters  
dialog box.

9. Create a flange.

The Flange definition dialog box reflects the modification for the Radius.


The default mode, that is to say the formula:

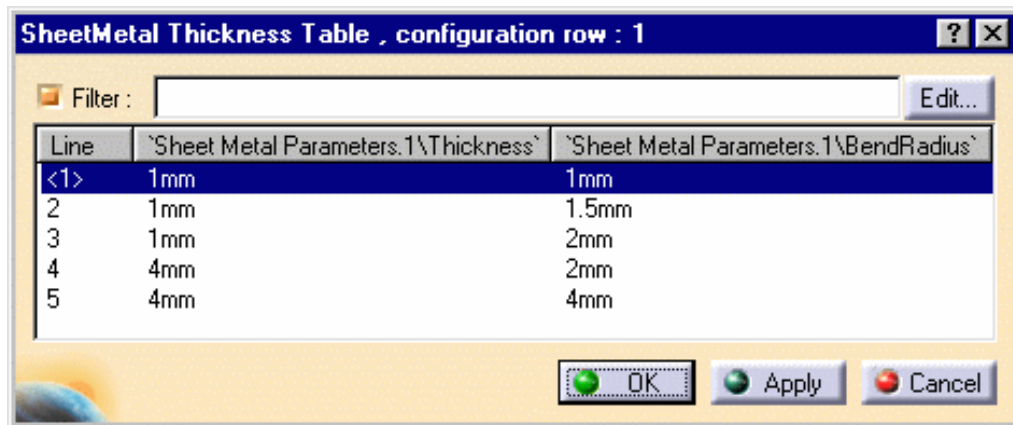
**Bend Radius = Part Radius is deactivated.**



### Using the Sheet Metal Design Tables:


Steps 1 to 4 are identical.

5. Click the Design Table icon  and select a line.




6. Click **OK**.


The parameter values are updated in the Sheet Metal Parameters dialog box.

 At that time, the parameters Thickness and Bend radius are driven by the design table.

They are now in gray, indicating that you can no longer modify the values.

Note that if you create a bend, there is no design table: it's the formula which is used.

 To disable the access to design tables:

- Select the **Tools -> Options -> Part -> Display** tab and check **Relations: the Design Table** icon  is displayed in the specification tree.
- Right-click this icon: the contextual menu appears.
- Select **SheetMetal Thickness Table object -> Deactivate**

The relation is no longer used but still exists. It can be activated at any time.



# Customizing General Settings

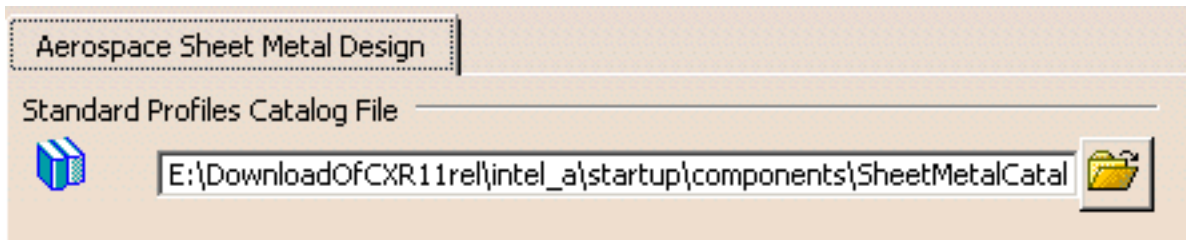


This task explains how to customize the Aerospace Sheetmetal Design General Settings.

1. Select the **Tools -> Options** menu item.

The Options dialog box appears.

2. Click **Mechanical Design category then the Aerospace Sheetmetal Design** subcategory.



The default path is displayed in the Standard Profiles Catalog Files. You can modify it by clicking

the **Browse** icon



If no catalog path has been defined prior to entering the Catalog Browser command, the default catalog is selected and its path automatically added to the Standard Profiles Catalog Files field.



# Customizing Standards Files To Define Methods for Compensations



This task explains how to access company standards files (available in .xls format) in order to define methods for joggle and sides compensations. The methods described in this task apply to joggles relying on a flange with a base feature (either a web or a flange) without joggles.



Open a new document.



1. Click the **Sheet Metal Parameters** icon . The Sheet Metal Parameters dialog box is displayed.
2. Select the **Sheet Standards Files...** button. The File Selection window is displayed.
3. Indicate the path to the Sheet Metal methods (Std\_Method1.xls or Std\_Method1\_2.xls).

Two methods are available to enable the joggle compensations. Both use Design Tables.

## Method 1



This method is the method which was used in V4.

The type of modifications performed depends on the position of the joggle with regards to:

- the end of the part:
  - near the end of part (case 1)
  - not near the end of part (case 2)
- the position of other joggles:
  - twin joggles (case 3)
  - double joggle near the end of part (case 4)
  - double joggle not near the end of part (case 5)

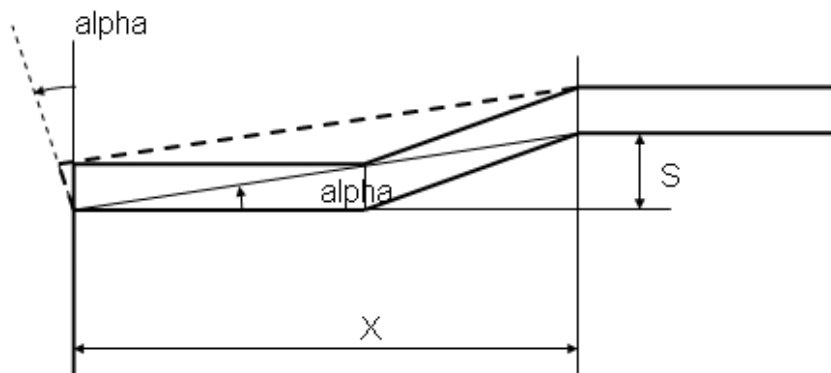
## How the different cases and the deformation are managed

Six values are used in the following description: C1, C2, C3, C4, C5, Ra. These values are defined in a design table. The path to this design table is defined in the Std\_Method1.xls file.

Here is an example of a design table:

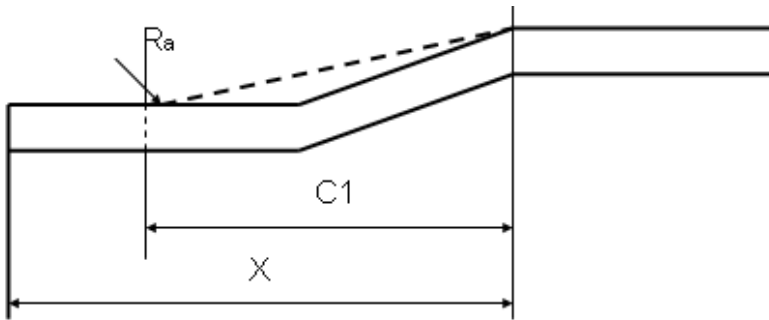
| C1 | C2 | C3  | C4 | C5 | Ra |
|----|----|-----|----|----|----|
| 75 | 5  | 150 | 10 | 75 | 3  |

## Case 1



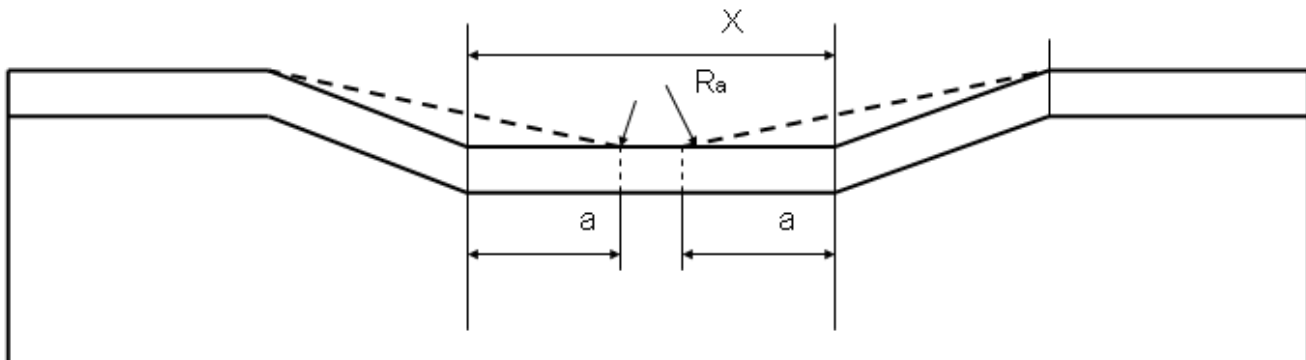
- The parameters necessary for performing the modifications are as follows:
  - The parameter of the feature is: S (joggle depth)
  - Some values are computed to define the type of deformation: alpha, X.
- The parameter definition and links are as follows:
  - This case is applied if  $X < C1$ .
  - C1 is defined with a constant value.
- This deformation is performed as follows:
  - The alpha angle is computed from S and X ( $\text{tangent}(\alpha) = S / X$ ).
  - The side and the EOP are then rotated with the alpha angle. The center of rotation is the intersection between the BTL on the flange and the side.

Case 2




- The parameters necessary for performing the modifications are as follows:
  - The values of the modification are: C1, Ra.
- The parameter definition and links are as follows:
  - This case is applied if  $X > C1$ .
  - C1 is defined with a constant value.
  - Ra is defined with a constant value.

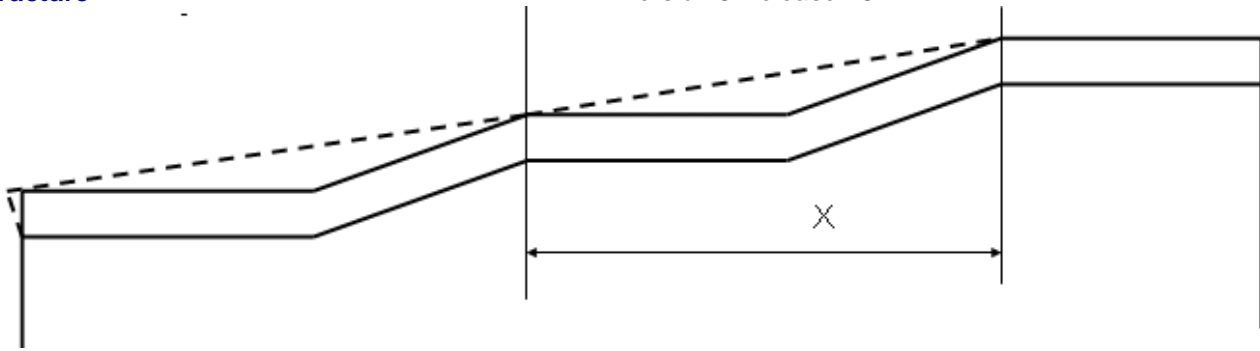
Case 3



- The parameters necessary for performing the modifications are as follows:
  - The values of the modification are: C2, C3, C4, Ra.
  - A value is computed to define the type of deformation (table) to apply: X.
- The parameter definition and links are as follows:
  - This case is applied if  $C3 > X > C2$ .
  - Then:  $a = (X - C4) / 2$ .
  - C2, C3 and C4 are defined with a constant value.
  - Ra is defined with a constant value.

Case 4

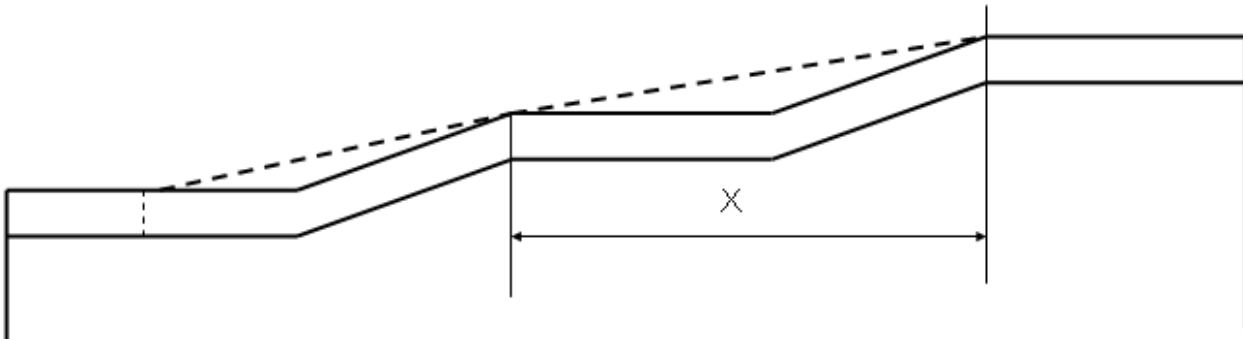
 This case applies only to the joggle positioned on the right; the joggle positioned on the left follows case 1.



- The parameters necessary for performing the modifications are as follows:
  - A value is computed to define the type of deformation (table) to apply: X.
- The parameter definition and links are as follows:
  - This case is applied if  $C4 > X$ .
  - C4 is defined with a constant value.

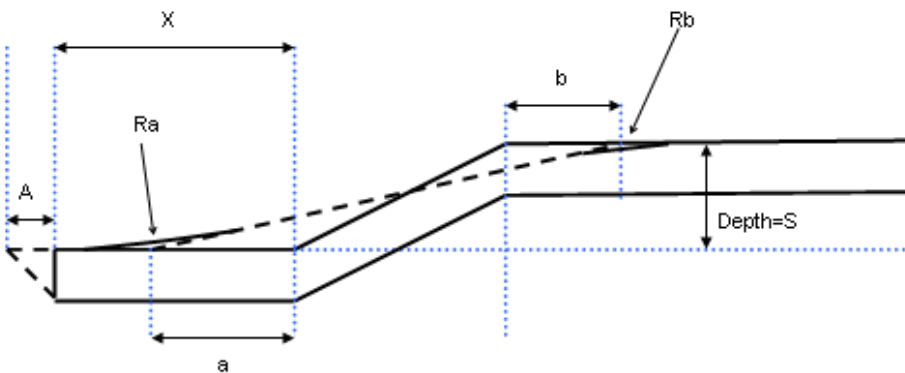
Case 5

*i* This case applies only to the joggle positioned on the right; the joggle positioned on the left follows case 2.



- The parameters necessary for performing the modifications are as follows:
  - A value is computed to define the type of deformation (table) to apply: X.
- The parameter definition and links are as follows:
  - This case is applied if  $C5 > X$ .
  - C5 is defined with a constant value.

Method 2



- The parameters necessary for performing the modifications are as follows:
  - Input parameters for joggle compensation:
    - Material Thickness:  $t$
    - Joggle parameters:  $S$  (joggle depth)
    - Distance between the joggle and the side of the flange:  $X$
  - Output parameters:
    - Offset from start of joggle:  $a$
    - Offset from end of joggle:  $b$
    - Flange side compensation:  $A$
    - Radius on start and end of joggle compensation:  $R_a, R_b$
- The parameter definition and links are as follows:
  - $A, a, b, R_a, R_b$  are defined from two design tables. The paths to these design tables are defined in the Std\_Method1\_2.xls file. Std\_Method1\_2.xls also contains the path to each defined thickness.

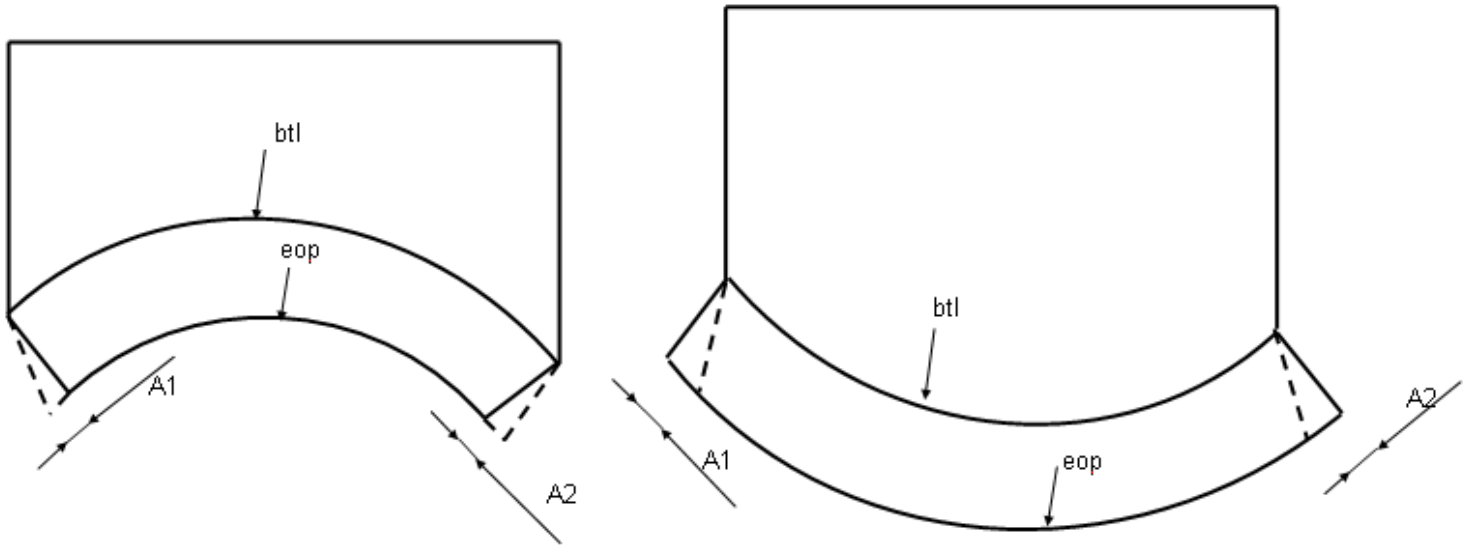
For each value of a material (therefore for each thickness) in Std\_Method1\_2.xls, two tables defining compensation values may be defined:

A first table defining  $A$ , depending on  $S$  and  $X$ :

| SMax | XMax | A |
|------|------|---|
| 0.8  | X1   |   |
| 0.8  | X2   |   |
| 0.8  | X3   |   |
| 1.1  | X1   |   |
| 1.1  | X2   |   |
| 1.1  | X3   |   |

A second table defining  $A$ , depending on  $S$ :


| SMax | a | b | $R_a$ | $R_b$ |
|------|---|---|-------|-------|
| 0.8  |   |   |       |       |
| 1.1  |   |   |       |       |
| 1.5  |   |   |       |       |



- The parameters necessary for performing the modifications are as follows:
  - The values of the modification are: A1, A2
  - Compensation can be automatic, or manual (defined in flange parameters)
    - Automatic (for symmetric flanges):  $A1 = A2 = 0.5 * (\text{length } btl - \text{length } eop)$
    - Manual: manual input for A1 and A2

4. Choose the appropriate file for the desired method.

5. Click **Open**.

In the Sheet Metal Parameters dialog box, if you chose Method 2, the **Design Table** icon  appears opposite the Thickness field. You can click this icon to edit the design table.





# Customizing For Generative Sheetmetal Design

This section describes how to customize different settings specific to the Generative Sheetmetal Design workbench. The settings described here deal with permanent setting customization.

[Customizing General Settings](#)

[Customizing Standard Files](#)

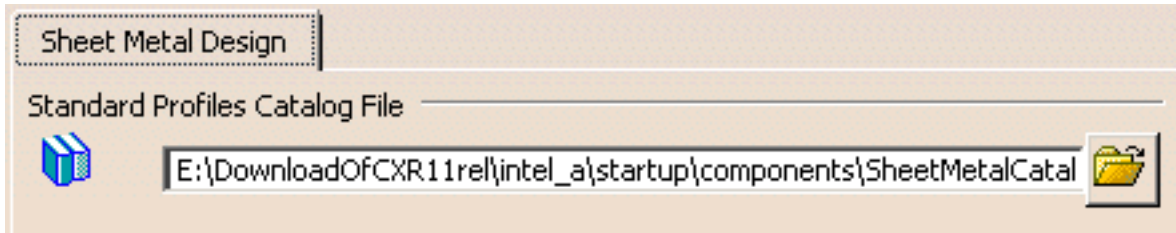
# Customizing General Settings

 This task explains how to customize the Generative Sheetmetal Design General Settings.


1. Select the **Tools -> Options** menu item.

The Options dialog box appears.

2. Click **Mechanical Design** category then the **Generative Sheetmetal Design** subcategory.



The default path is displayed in the Standard Profiles Catalog Files. You can modify it by clicking the **Browse** icon .


 If no catalog path has been defined prior to entering the Catalog Browser command, the default catalog is selected and its path automatically added to the Standard Profiles Catalog Files field.




# Customizing Standard Files

This section describes how to customize settings.  
The task described here deals with permanent setting customizing.

## Using Sheet Metal Standards Files

 This task explains how to access company standards files.

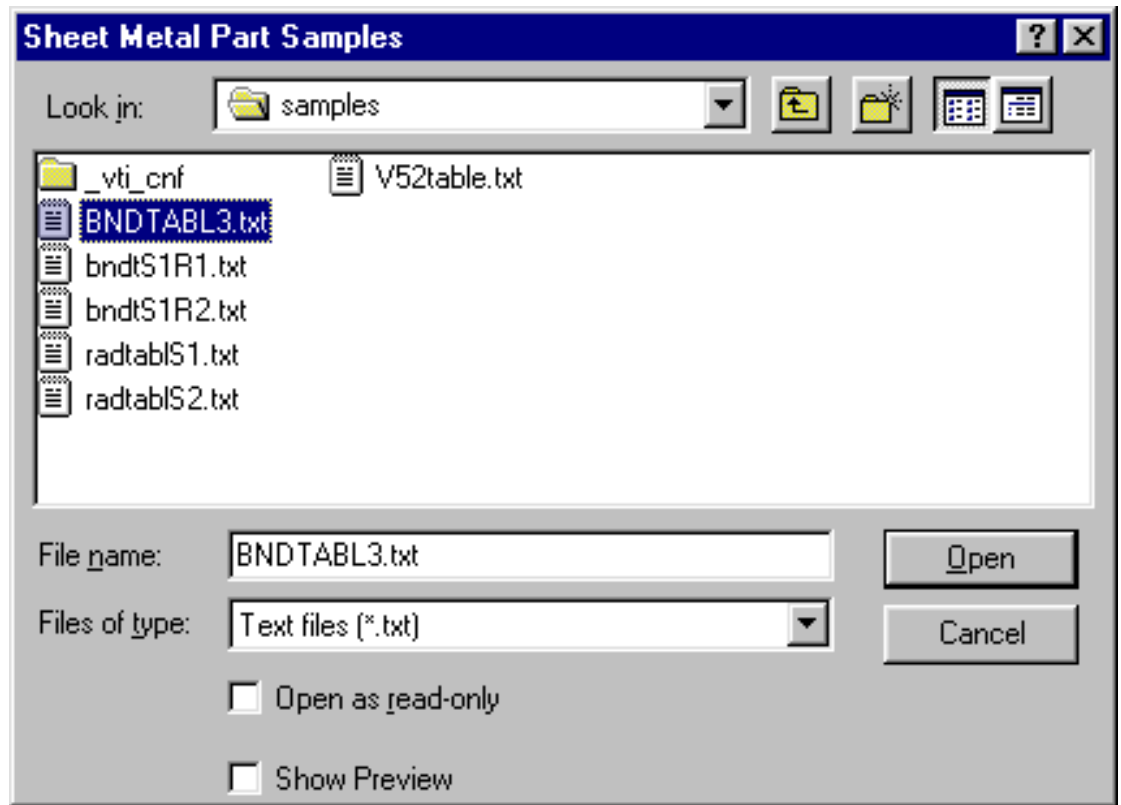
 Open a new document.

 **1.** Click the **Sheet Metal Parameters** icon .

The Sheet Metal Parameters dialog box opens.

**2.** Select the **Sheet Standards Files...** button. The Sheet Metal Part Samples window is displayed.

**3.** Indicate the path to the Sheet Metal table.



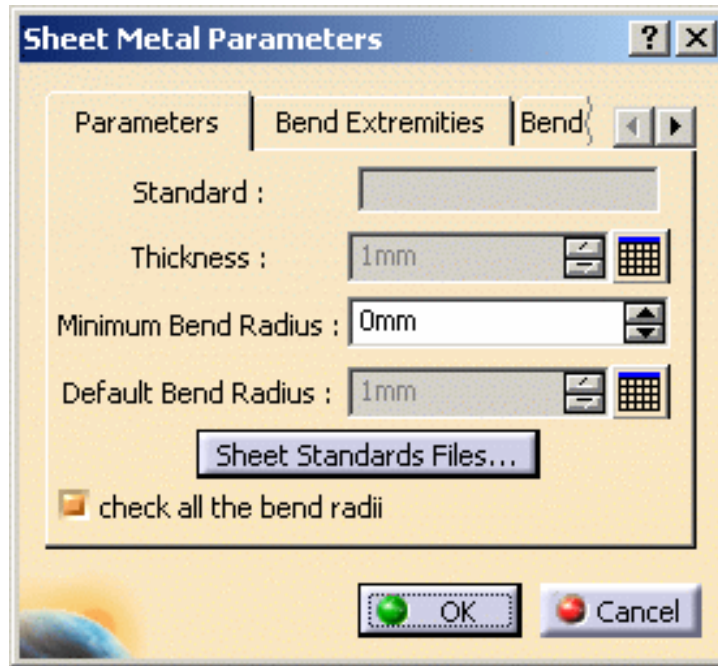
 These files are available under .xls or .txt format.

4. Click **Open**.

In the Sheet Metal Parameters dialog box, the **Design Table** icon

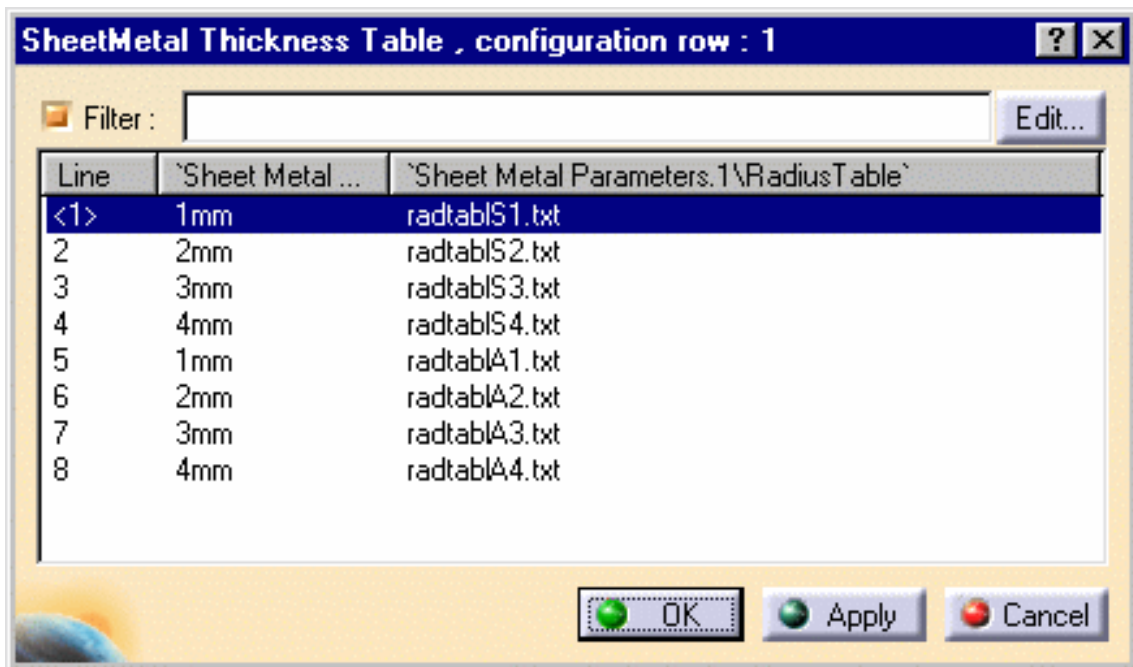


appears opposite the Thickness and Bend radius fields.



The parameters are now in gray, indicating that you can no longer modify the values.

5. Click the Thickness Design Table icon  and select the line containing the appropriate parameters (for example Line 1).



 Using the **Tools -> Options -> General -> Document** tab, **Other Folders** option, you can specify where the files are located. Refer to [Document](#).

This scenario can work when the .CATPart document and all reference table files (Design & Radius) are located in the same directory. This directory is the current one when the Design table is created, and also when the .CATPart is open.

However, generally speaking, you must reference the complete path indicating where the radius table files are to be found in the RadiusTable column. In this case, regardless of the current directory, the correct tables are located when re-opening the .CATPart document.

- 6. Click **OK**.

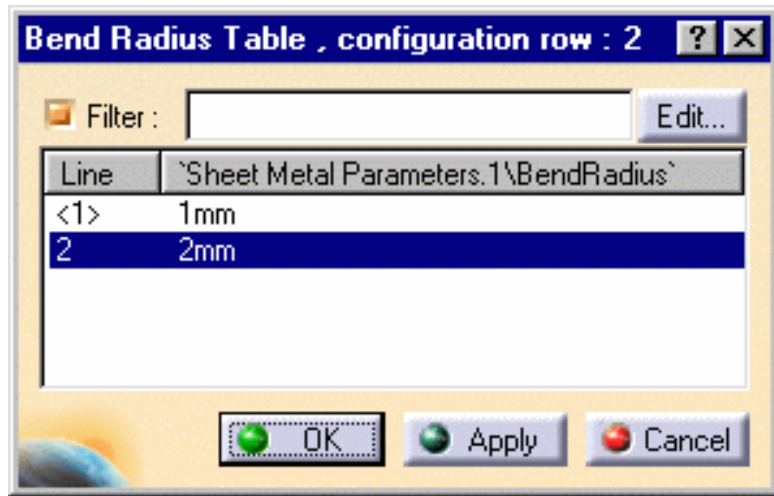
The parameter values are updated in the Sheet Metal Parameters dialog box.

- 7. Click the Bend Radius Design Table icon



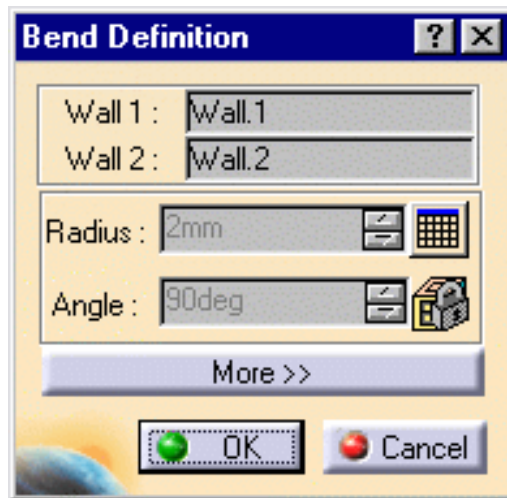
- 8. Select line 2 and click **OK**.

The parameter values are updated in the Sheet Metal Parameters dialog box.



- 9. Create a bend.


The Bend Definition dialog box displays a design table for the Bend Radius.

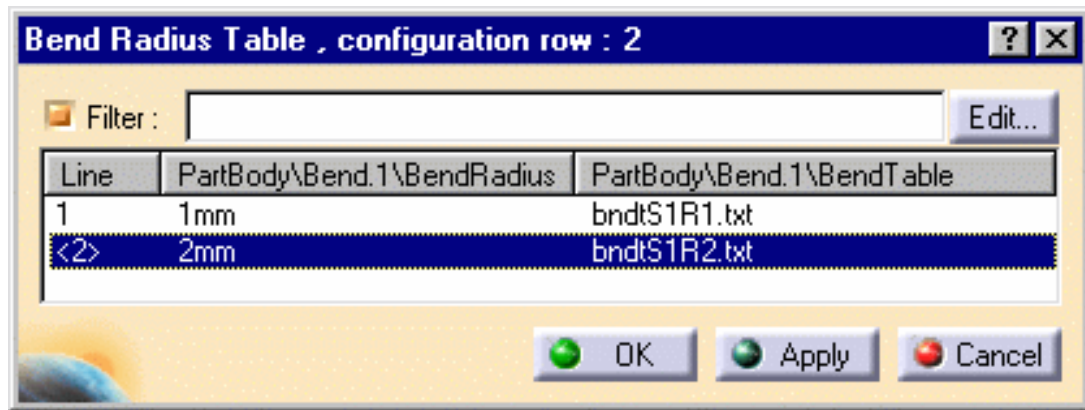


The default mode, that is to say the formula:

**Bend**

**Radius =**  
**Part**  
**Radius is**  
 deactivated.

Let's see the Bend Radius Table, using this icon .




It shows the Bend Radius and the corresponding Bend Table.

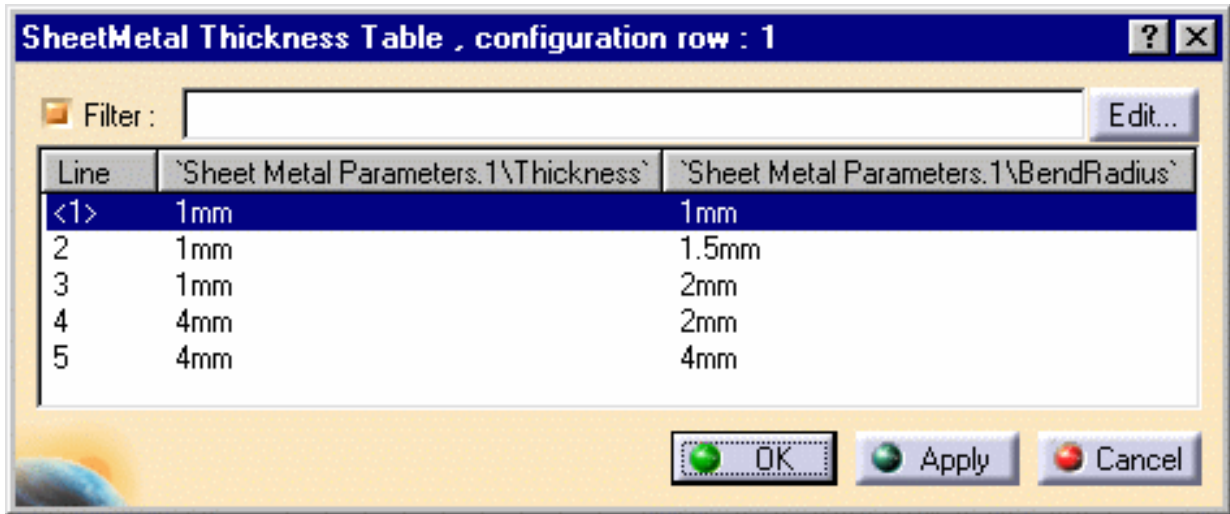
**10.** Click OK.

If the **Angle** value is contained in the **Bend Table**, the **Bend Allowance** uses the corresponding value.  
 If not, the **Bend Allowance** is computed according to the KFactor.

### Using the Sheet Metal Design Tables:


 Steps 1 to 4 are identical.


**5.** Click the Design Table icon  and select a line.




6. Click **OK**.

The parameter values are updated in the Sheet Metal Parameters dialog box.

 At that time, the parameters Thickness and Bend radius are driven by the design table. They are now in gray, indicating that you can no longer modify the values. Note that if you create a bend, there is no design table: it's the formula which is used.

 To disable the access to design tables:

- Select the **Tools -> Options -> Part -> Display** tab and check **Relations:**

the **Design Table** icon  is displayed in the specification tree.

- Right-click this icon: the contextual menu appears.
- Select **SheetMetal Thickness Table object -> Deactivate**

The relation is no longer used but still exists.

It can be activated at any time.



# Customizing for 3D Functional Tolerancing & Annotation

This section describes the various settings you can customize for 3D Functional Tolerancing & Annotations via the **Tools -> Options** command.



- Tolerancing
- Display
- Constructed Geometry
- Manipulators
- Dimension
- Annotation
- Tolerance Values
- View/Annotation Plane



# Tolerancing



This task will show you how to customize 3D Functional Tolerancing & Annotation tolerancing settings.



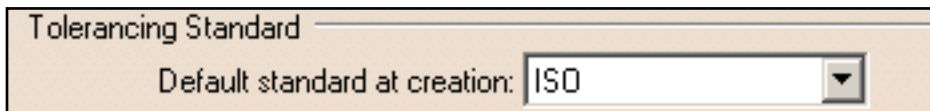
1. Select **Tools** -> **Options...**

The **Options** dialog box appears.

2. Click the **Mechanical Design** category.
3. Click the **3D Functional Tolerancing & Annotation** subcategory.
4. Click the **Tolerancing** tab.

The **Tolerancing** tab appears, displaying five categories of options:

## Tolerancing Standard



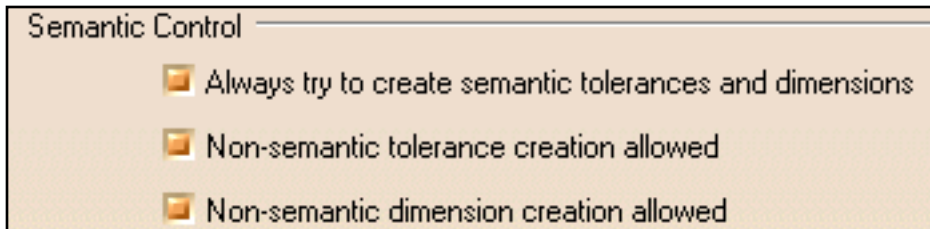
### Default standard at creation

Defines the default standard at creation.

Provides five conventional standards:

- ASME: American Society for Mechanical Engineers
- ASME 3D: American Society for Mechanical Engineers
- ANSI: American National Standards Institute
- ISO: International Organization for Standardization
- JIS: Japanese Industrial Standard

## Semantic Control



### **Always try to upgrade non-semantic tolerances and dimensions**

Defines automatic upgrade of a non-semantic annotation during its creation to a semantic annotation. If it can be upgradeable to a semantic annotation only.

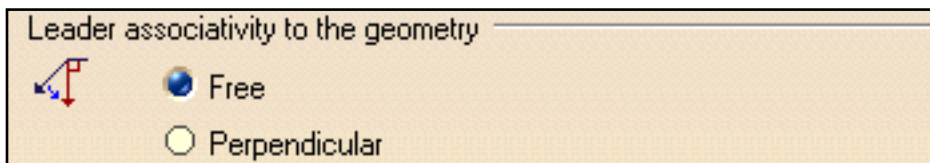
### **Non-semantic tolerance creation allowed**

Defines whether non-semantic tolerances creation is allowed or not.

### **Non-semantic dimension creation allowed**

Defines whether non-semantic dimensions creation is allowed or not.

## Leader associativity to the geometry



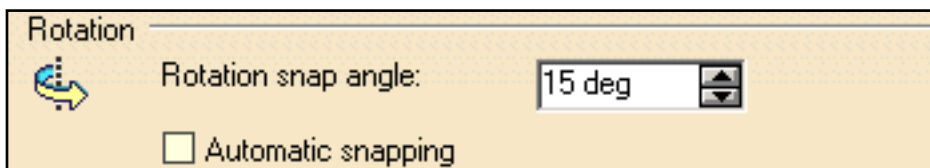
### **Free**

Specifies that leader annotations are freely positioned relative to their geometrical elements.

### **Perpendicular**

Specifies that leader annotations are positioned perpendicular to their geometrical elements.

## Rotation




### **Rotation snap angle**

Defines an angle value for rotating elements. This option is used to rotate text elements (text, frame, or leader).

### **Automatic snapping**

Defines whether the rotation will be snapped to the angle value or not.

# Note Object Attribute

|                                                                                   |                                                               |
|-----------------------------------------------------------------------------------|---------------------------------------------------------------|
| Note Object Attribute                                                             |                                                               |
|  | <input type="checkbox"/> Allow Note Object Attribute creation |

## **Allow Note Object Attribute creation**

Defines whether note object attribute may be created by user or not. Enable or disable the icon and menu item.



# Display



This task will show you how to customize 3D Functional Tolerancing & Annotation display settings.



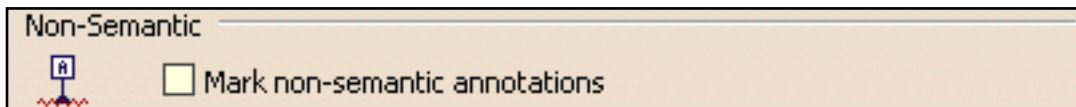
1. Select **Tools -> Options...**

The **Options** dialog box appears.

2. Click the **Mechanical Design** category.
3. Click the **3D Functional Tolerancing & Annotation** subcategory.
4. Click the **Display** tab.

The **Display** tab appears, displaying the following categories of options:

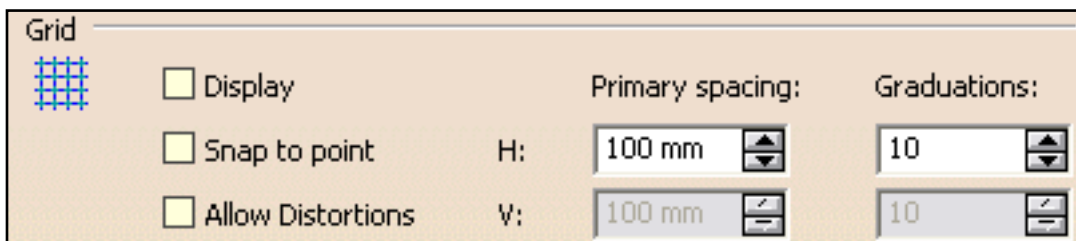
## Non-Semantic



### Mark non-semantic annotations

Defines whether non-semantic annotations (datum elements, datum targets, geometrical tolerances, linear and angular dimensions) are marked with a wavy red line in the specification tree and in the geometry.

## Grid



### Display

Defines whether the grid is displayed.

**Snap to point**

Defines whether annotations are snapped to the grid's point.

**Allow Distortions**

Defines whether grid spacing and graduations are the same horizontally and vertically.

**H Primary spacing**

Defines the grid's horizontal spacing.

**H Graduations**

Defines the grid's horizontal graduations.

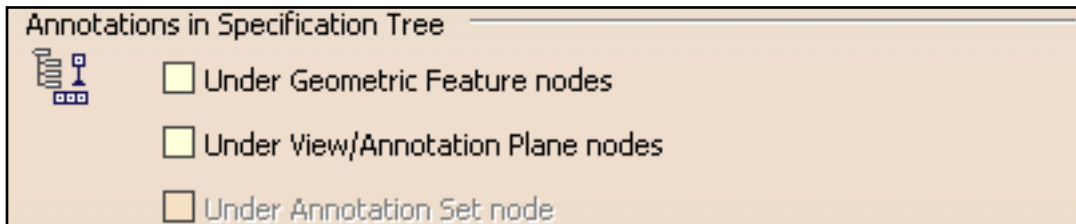
**V Primary spacing**

Defines the grid's vertical spacing (available only if Allow Distortions is checked).

**V Graduations**

Defines the grid's vertical graduations (available only if Allow Distortions is checked).

## Annotations in Specification Tree

**Under Geometric Feature nodes**

Defines that 3D annotations should be displayed under the geometric feature nodes in the specification tree. This lets you view 3D annotations under the Part Design or GSD feature nodes to which they are applied.

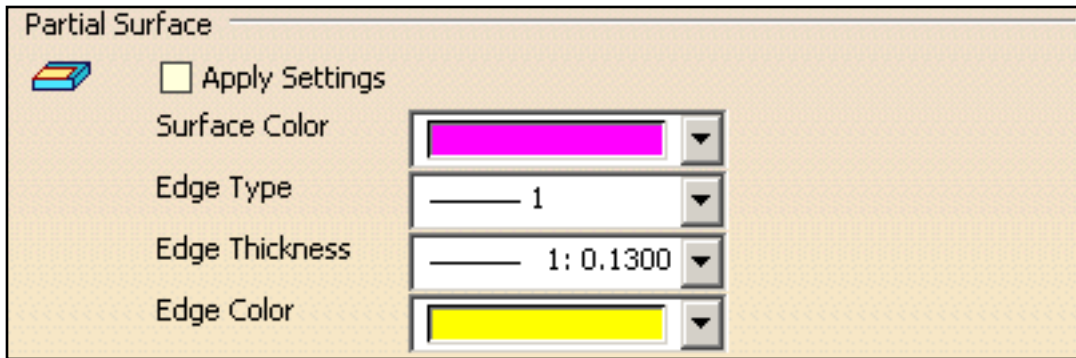
**Under View/Annotation Plane nodes**

Defines that 3D annotations should be displayed under the view/annotation plane nodes in the specification tree. This lets you view 3D annotations under the view node to which they are linked.

**Under Annotations Set node**

Defines that 3D annotations should be displayed under the annotation set node in the specification tree.

## Partial Surface



### Apply Settings

Defines whether the following settings are applied while creating a partial surface feature.

### Surface Color

Defines the surface color of the partial surface.

### Edge Type

Defines the edge type of the partial surface's border.

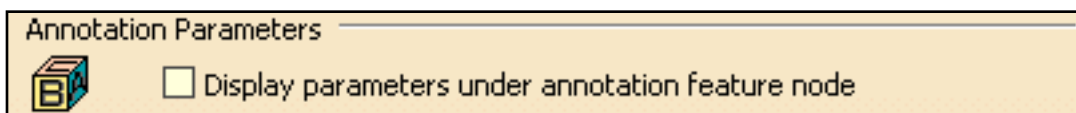
### Edge Thickness

Defines the edge thickness of the partial surface's border.

### Edge Color

Defines the edge color of the partial surface's border.

## Annotation Parameters

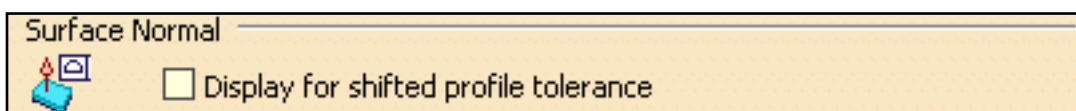


### Display parameters under annotation feature node

Defines that knowledge parameters (such as tolerance values, datum label, etc.) of annotations should be displayed under the annotation feature node in the specification tree; also defines that feature parameters of dimensions (accessible through the **Edit Generative Parameter** command) should be displayed under the dimension feature node in the specification tree.

Note that in order to have the value of the parameters displayed in the specification tree, you need to select the **With value** knowledge setting in **Tools -> Options -> General -> Parameters and Measure -> Knowledge** tab.

## Surface Normal

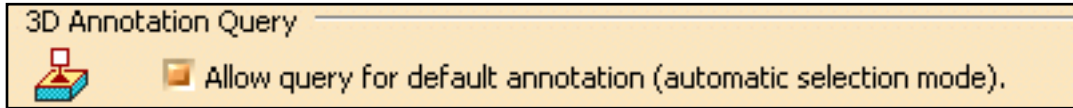


### Display for shifted profile tolerance

Defines whether the normal of all the selected surfaces are displayed, or not, when a shifted profile tolerance is specified or queried.



## 3D Annotation Query




### Allow query for default annotation (automatic selection mode)

Defines whether the query for default annotation is allowed, or not. This option allows you to highlight the related annotations or geometrical elements with the selected annotation or the related annotations with the selected geometrical element.



# Constructed Geometry

 This task will show you how to customize 3D Functional Tolerancing & Annotation tolerances settings.



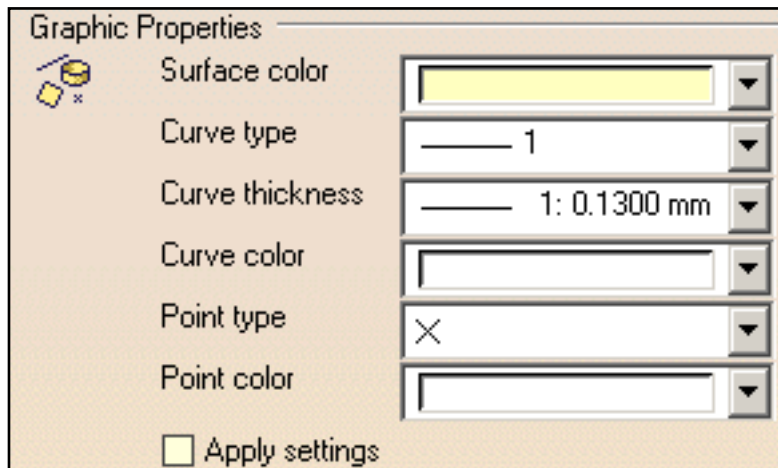
1. Select **Tools** -> **Options...**

The **Options** dialog box appears.

2. Click the **Mechanical Design** category.
3. Click the **3D Functional Tolerancing & Annotation** subcategory.
4. Click the **Constructed Geometry** tab.

The **Constructed Geometry** tab appears, displaying three categories of options:

## Graphic Properties



### Surface color

Defines the constructed geometry surface color for plane and cylinder.

### Curve type

Defines the constructed geometry curve type.

### Curve thickness



Defines the constructed geometry curve thickness.

**Curve color**

Defines the constructed geometry curve color.

**Point type**

Defines the constructed geometry point type.

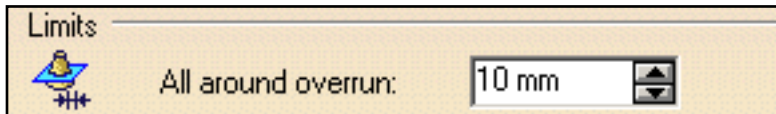
**Point color**

Defines the constructed geometry point color.

**Apply settings**

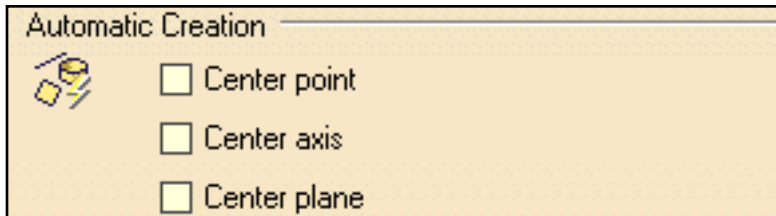
Defines whether all the graphic properties settings are applied or not.

## Limits

**All around overrun**

Defines the minimal limit between the constructed geometry and its related geometry.

## Automatic Creation

**Center point**

Defines whether all the center point's constructed geometry is automatically created or not, for circle center, sphere center.

**Center axis**

Defines whether all the center axis's constructed geometry is automatically created or not, for cylinder, cone.

**Center plane**

Defines whether all the center plane's constructed geometry is automatically created or not, for slot.



# Manipulators

 This task will show you how to customize 3D Functional Tolerancing & Annotation manipulators settings.



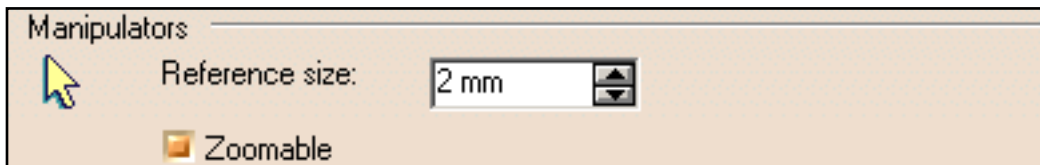
1. Select **Tools** -> **Options...**

The **Options** dialog box appears.

2. Click the **Mechanical Design** category.
3. Click the **3D Functional Tolerancing & Annotation** subcategory.
4. Click the **Manipulators** tab.

The **Manipulators** tab appears, displaying two categories of options:

## Manipulators









### Reference size

Defines the annotation manipulator's size.

### Zoomable

Defines whether the annotation manipulator is zoomable or not.

## Dimension Manipulators

| Dimension Manipulators                                                            |                                                                                                       | Creation                 | Modification                        |
|-----------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------------------|--------------------------|-------------------------------------|
|  | <input type="checkbox"/> Modify overrun:                                                              | <input type="checkbox"/> | <input type="checkbox"/>            |
|                                                                                   | <input type="checkbox"/> Modify blanking:                                                             | <input type="checkbox"/> | <input checked="" type="checkbox"/> |
|                                                                                   |  Insert text before: | <input type="checkbox"/> | <input type="checkbox"/>            |
|                                                                                   |  Insert text after:  | <input type="checkbox"/> | <input type="checkbox"/>            |
|  | Move value:                                                                                           | <input type="checkbox"/> | <input type="checkbox"/>            |
|  | Move dimension line:                                                                                  | <input type="checkbox"/> | <input type="checkbox"/>            |
|  | Move dimension line secondary part:                                                                   | <input type="checkbox"/> | <input type="checkbox"/>            |

**Modify overrun**

Defines whether overrun extension lines can be modified during creation or modification, or not.

**Modify blanking**

Defines whether blanking can be modified during creation or modification, or not.

**Insert text before**

Defines whether a text before can be inserted during creation or modification, or not.

**Insert text after**

Defines whether a text after can be inserted during creation or modification, or not.

**Move value**

Defines whether only the value can be moved during creation or modification, or not.

**Move dimension line**

Defines whether only the dimension line can be moved during creation or modification, or not.

**Move dimension line secondary part**

Defines whether only the dimension line secondary part can be moved during creation or modification, or not.



# Dimension



This task will show you how to customize 3D Functional Tolerancing & Annotation dimension settings.



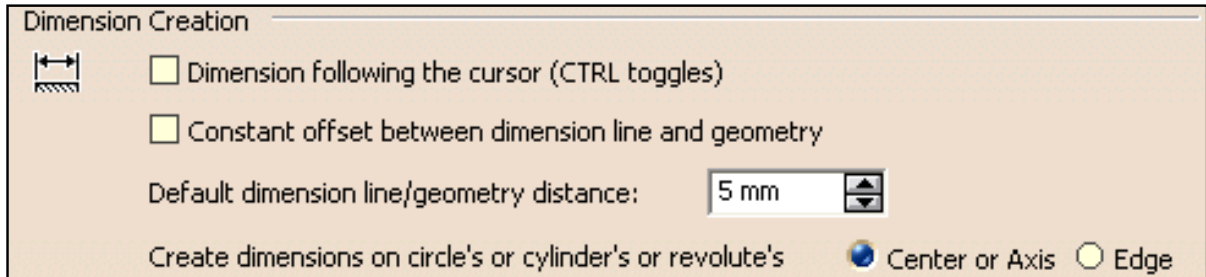
1. Select **Tools** -> **Options...**

The **Options** dialog box appears.

2. Click the **Mechanical Design** category.
3. Click the **3D Functional Tolerancing & Annotation** subcategory.
4. Click the **Dimension** tab.

The **Dimension** tab appears, displaying two categories of options:

## Dimension Creation



### Dimension following the mouse

Defines whether the dimension line is positioned according to the cursor, following it dynamically during the creation process or not.

### Constant offset between dimension line and geometry

Defines whether the distance between the created dimension and the geometry remains the same when you move the geometry or not.

### Default dimension line/geometry distance


Defines the value at which the dimension is created from the geometry. If you create associativity between the dimension and the geometry, you can define the value at which the dimension will remain positioned.

### By default, dimension circles on their

Defines the dimension you will create between a circle and another element will be either on the circle center or on the circle edge.

## Move

Move

  Activate Snapping (SHIFT toggles) [Configure...](#)

Move only selected sub-part

### Activate Snapping

Defines whether the dimension will be snapped on the grid and/or the dimension value will be located at its default position between symbols (it will work only if the cursor is between the symbols) or not.


### Move only selected sub-part

Defines whether only a dimension sub-part (text, line, etc...) will be moved or not.



## Line-Up

Line-Up

 Default offset to reference:

Default offset between dimensions:

Align stacked dimension values

Align cumulated dimension values

Automatically add a funnel

### Default offset to reference

Defines the distance and angular offset between the smallest dimension and the reference element.

### Default offset between dimension

Defines the distance and angular offset between the dimension.

### Align stacked dimension values

Defines whether the values of a group of stacked dimensions are aligned on the value of the smallest dimension of the group or not.

### Align cumulated dimension values

Defines whether the values of a group of cumulated dimensions are aligned on the value of the smallest dimension of the group or not.


### Automatically add a funnel

Defines whether the value of a cumulated dimension requires a funnel added automatically to be displayed correctly or not.



## Dimension related to an origin

Dimension related to an origin


  Use Dimension Origin Symbol

**Use Dimension Origin Symbol**

Defines whether the dimension origin symbol is used or not.



# Annotation

 This task will show you how to customize 3D Functional Tolerancing & Annotation annotation settings.



1. Select **Tools -> Options...**

The **Options** dialog box appears.

2. Click the **Mechanical Design** category.
3. Click the **3D Functional Tolerancing & Annotation** subcategory.
4. Click the **Annotation** tab.

The **Annotation** tab appears, displaying two categories of options:

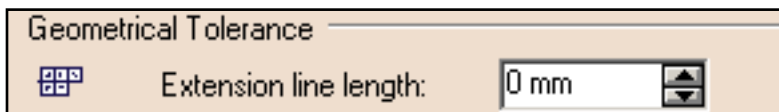
## Annotation Creation



### Annotation following the mouse (Ctrl toggles)

Defines whether the annotation is positioned according to the cursor, following it dynamically during the creation process or not.

## Geometrical Tolerance




### Extension line length

Defines the extension line length between the geometrical frame and the its leader.



# Tolerances

 This task will show you how to customize 3D Functional Tolerancing & Annotation tolerances settings.



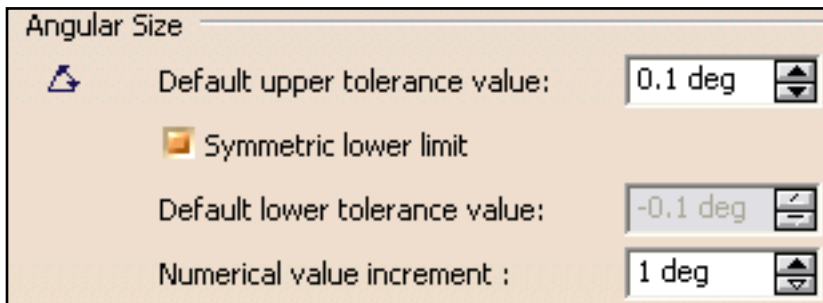
1. Select **Tools** -> **Options...**

The **Options** dialog box appears.


2. Click the **Mechanical Design** category.
3. Click the **3D Functional Tolerancing & Annotation** subcategory.
4. Click the **Tolerances** tab.

The **Tolerances** tab appears, displaying three categories of options:

## Angular Size



The screenshot shows the 'Angular Size' dialog box with the following settings:

|                                                                                     |                                |          |
|-------------------------------------------------------------------------------------|--------------------------------|----------|
|  | Default upper tolerance value: | 0.1 deg  |
| <input checked="" type="checkbox"/>                                                 | Symmetric lower limit          |          |
|                                                                                     | Default lower tolerance value: | -0.1 deg |
|                                                                                     | Numerical value increment :    | 1 deg    |

### Default upper tolerance value

Defines the default upper tolerance value for angular size.

### Symmetric lower limit

Defines whether the default lower tolerance value is symmetric in relation to the default upper tolerance value.

### Default lower tolerance value

Defines the default lower tolerance value for angular size, disable when Symmetric lower limit is checked.


### Numerical value increment



Defines the increment for angular size numerical value.

## Linear Size

Linear Size

 Default upper tolerance value: 0.1 mm

Symmetric lower limit

Default lower tolerance value: -0.1 mm

Numerical value increment: 0.01 mm

Default tabulated value: H7

### Default upper tolerance value

Defines the default upper tolerance value for linear size.

### Symmetric lower limit

Defines whether the default lower tolerance value is symmetric in relation to the default upper tolerance value.

### Default lower tolerance value

Defines the default lower tolerance value for linear size, disable when Symmetric lower limit is checked.

### Numerical value increment


Defines the increment for linear size numerical value.

### Default tabulated value

Defines the default tabulated for linear size.

## Geometrical Tolerance

Geometrical Tolerance

 Default numerical value: 0.1 mm

Numerical value increment: 0.01 mm

Precision: 0.01

Separator: ,

Display trailing zeros

Display leading zero

### Default numerical value

Defines the default numerical for geometrical tolerance.

**Numerical value increment**

Defines the increment for the to geometrical tolerance numerical value.

**Precision**

Defines the geometrical tolerance precision after the numerical separator.

**Separator**

Defines the geometrical tolerance symbol used as numerical separator.

**Display trailing zeros**

Defines whether "0"s complete the number of digit displayed after the separator, according to the precision, or not.

**Display leading zeros**

Defines whether the "0" before the numerical separator, when value is less than 1, is displayed or not.



# View/Annotation Plane



This task will show you how to customize 3D Functional Tolerancing & Annotation view/annotation plane settings.



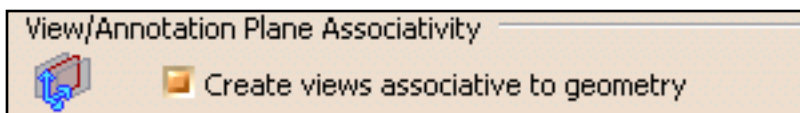
1. Select **Tools** -> **Options...**

The **Options** dialog box appears.

2. Click the **Mechanical Design** category.
3. Click the **3D Functional Tolerancing & Annotation** subcategory.
4. Click the **View/Annotation Plane** tab.

The **View/Annotation Plane** tab appears, displaying the following categories of options:

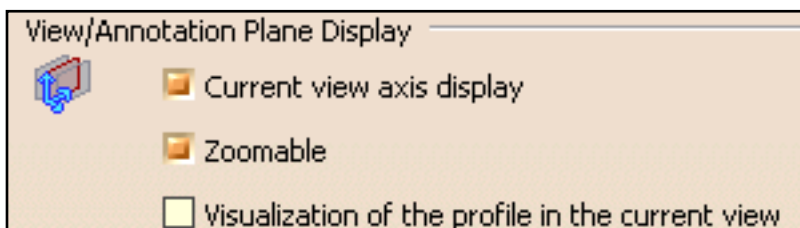
## View/Annotation Plane Associativity



### Create views associative to geometry

Creates views associative to the geometry, so that views and their annotations are automatically updated when the geometry is modified.

## View/Annotation Plane Display



### Current view axis display

Defines whether the active annotation plane axis system is displayed.

**Zoomable**

Defines whether the annotation plane axis is zoomable.

**Visualization of the profile in the current view**

Defines whether the view/annotation plane profile on the part/product is displayed or not.



# Shape

Free Style  
Automotive Body in White Fastening  
Generative Shape Design  
Shape Sculptor

# Customizing For FreeStyle Shaper, Optimizer, and Profiler



This task explains how to customize settings for FreeStyle workbenches.



1. Select the **Tools** -> **Options** menu item.

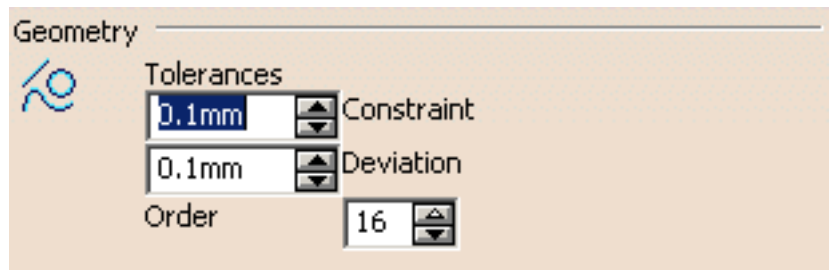
The Options dialog box is displayed.

2. Select the **Shape** category then select the **FreeStyle** subcategory.

The options for FreeStyle settings appear, organized in separate areas depending on the type of setting.

- Geometry
- Auto-detection
- Display
- Tuning

3. In the **Geometry** area, check:



- **Tolerances:** to impose user-defined tolerances to every elements to be created and modified within a FreeStyle workbench.  
You can define:
  - the **constraint tolerance value** used to define the continuity variation. In this case if the continuity is over the set value the element is not created/modified.  
In P1 mode, the **constraint tolerance** value is set and cannot be modified.
  - the **deviation tolerance value**, used when converting elements using the Converter Wizard for example (see tasks "Approximating/Segmenting Procedural Curves" and "Approximating/Segmenting Procedural Surfaces" within the *FreeStyle Shaper, Optimizer, and Profiler User's Guide*). In this case you accept the conversion up to the set tolerance value.
- **Order:** to set the maximum allowed order along the U (and V) directions for each curve segment and surface patch.  
This maximum order value ranges from 5 to 16 included, in a given direction.

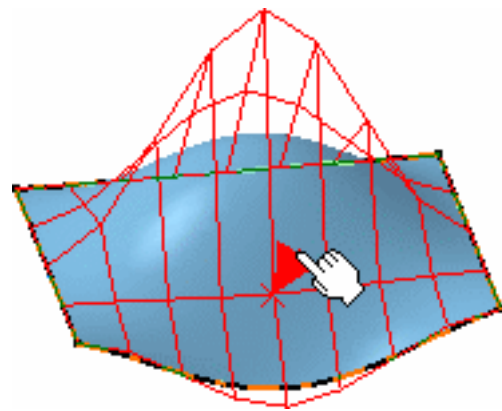
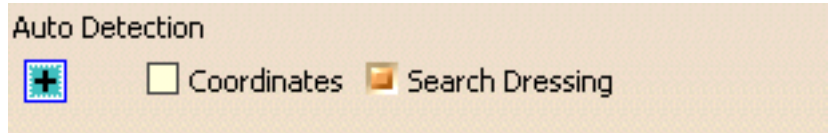


In P1 mode, the **Max order** is set to 16 and cannot be modified.



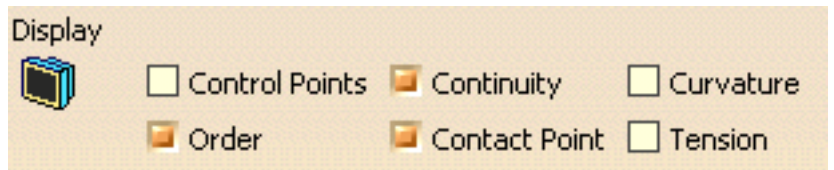
4. In the **Auto-Detection** area, you can choose to display:

- the point coordinates as the pointer moves along the geometry by selecting the **Coordinates** checkbox
- the dressing mode by selecting the **Search dressing** checkbox. This option allows the display of control points on geometry detected with the pointer when using the **Snap on Cpt** (snap on control points) option from the Dashboard.

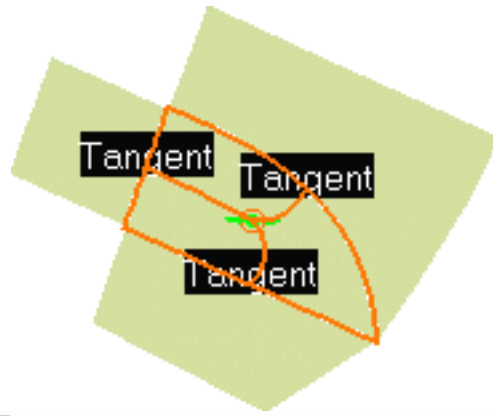


5. In the **Display** area, you set the information to be displayed on the geometry as you manipulate it, that is:

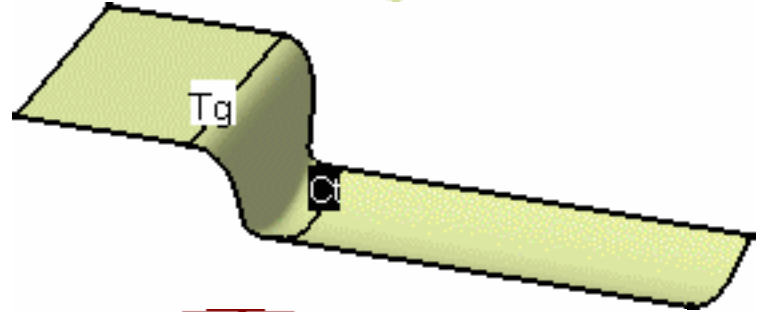
- the **Control Points** on the geometry. See "Displaying Control Points Temporarily" in the *FreeStyle Shaper, Optimizer, and Profiler User's Guide*.



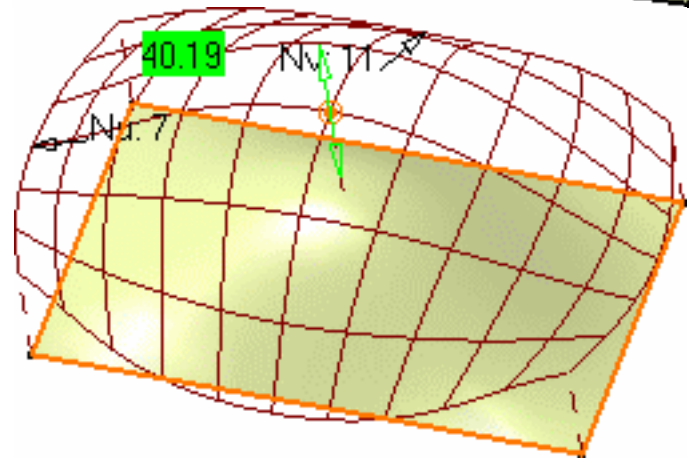
- the **Continuity** at every connection between elements (such as blend and match curves and surfaces, fill surfaces, and so forth)



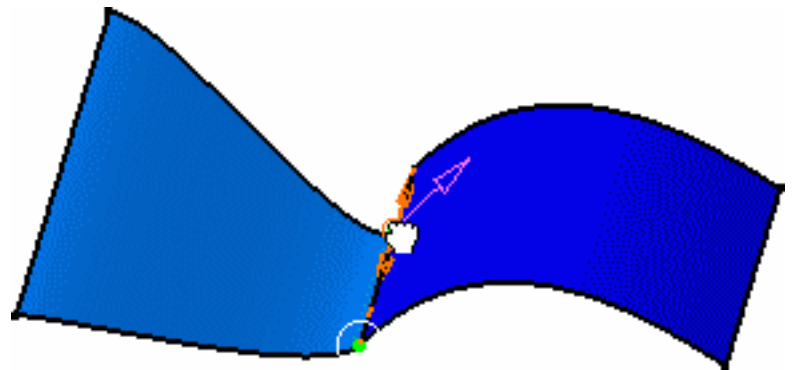
- the **Curvature** texts on an element (such as blend and match curves and surfaces, and so forth)



- the **Order** number along the U (for curves) or U and V directions (for surfaces).



- the **Contact Point** at every connection between connected elements (such as blend and match curves and surfaces, and so forth)



P2

- the **Tension** manipulators at every connection between connecting elements (such as blend and match curves and surfaces, and so forth)

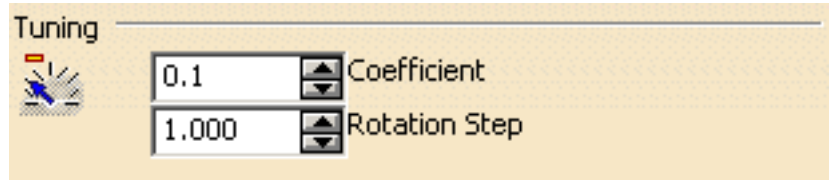






6. In the **Tuning** area, you set the values used when manipulating views or geometry (P2 only).

See "Editing Surfaces Using Control Points", and "Manipulating Views" in the *FreeStyle Shaper, Optimizer, and Profiler User's Guide* for example.



7. Click **OK** to confirm setting these permanent options.



For further information on the Performance tab, refer to the chapter called Performance in the *Infrastructure User's Guide*.



# Customizing For Automotive Body In White Fastening

This section describes how to customize different settings specific to the Automotive Body In White Fastening workbench. The settings described here deal with permanent setting customization.

[General Settings](#)

[BiW Fastening Application Display User Settings](#)

# General Settings



This task explains how to customize Automotive BiW settings.



1. Select the **Tools** -> **Options** menu item.

The Options dialog box appears.

2. Click the **Shape** category then select the **Automotive Body In White** subcategory to display the General tab:

The screenshot shows the 'BiW Fastening Application Parameters Set-Up' dialog box with the 'General' tab selected. The dialog has two tabs: 'General' and 'Display'. The 'General' tab contains the following settings:

- Default Standard File :** E:\DownloadOfCXR13rel\intel\_a\reffiles\GbfStandards\GBF\_STD.xls (with a Browse icon)
- Automatic Set-Up and Standard Import
- Check the Standard File Consistency at Import Time
- Export default directory :** C:\DOCUME~1\spu\LOCALS~1\Temp\ (with a Browse icon)
- Report default directory :** C:\DOCUME~1\spu\LOCALS~1\Temp\ (with a Browse icon)
- Generative Extract Parameters Set-Up :**
  - Fixed Drawing Spot Size: 0mm
  - Angular Tolerance: 90deg
  - Include View Reverse Direction

Default paths are displayed. You can change them by clicking the **Browse** icon



## Setting Up Parameters

- The Default Fasteners Parameters Set-Up Standard File corresponds to the path where the default standards are stored.
  - You can automatically configure the BiW Application and import the standard by clicking on the **Automatic Set-Up and Standard Import** check box (checked by default). If this option is activated, all the icons (except **Export**, **Rename** and **Check**) are accessible in the workbench. Selecting any of these icons automatically configure the BiW Application and import the standard.
  - You can also **Check the Standard File Consistency at Import Time** (unchecked by default) to check the consistency of the standard values when importing the standard file (whose path is mentioned above)  
If the standard file is not valid, it cannot be imported and an error message is issued.

## Defining the Export Directory

The Export default directory is the directory where the export file is stored.

## Defining the Report Directory

The Report default directory is the directory where the report file is stored.



## Setting Up Extract Parameters

- You can define a fixed size for the 2D drawing spots: check the **Fixed Drawing Spot Size** button and define a diameter value.  
As a consequence, all spots in the 2D document will have this diameter, whatever their size in the 3D document.
- You can define a user **Angular Tolerance**.
- You can check the **Include View Reverse Direction** checkbox to visualize 2D reverse spots.



Refer to the chapter Integration With Drafting to have further information.

3. Click **OK** to confirm setting these options.



[ Up ] [ Next ]

# BiW Fastening Application Display User Settings



This task shows you how to control the display of the fasteners in the geometry area.



1. Select the **Tools** -> **Options** command.

The **Options** dialog box is displayed.

2. Click the **Shape** category, then the **Automotive BiW Fastening** subcategory.
3. Select the Display tab.

General Display

3D Fasteners All Display Options

Unset All Set All

Shape

Show

ID

Show

Fastener Parameters

Show

Process Type Symbol Thickness 1

Process Category Symbol Color

None Symbol Mark Thickness 1

Fastener Normal Vector Orientation

Show

Support Contact Zone Material Orientation

Show

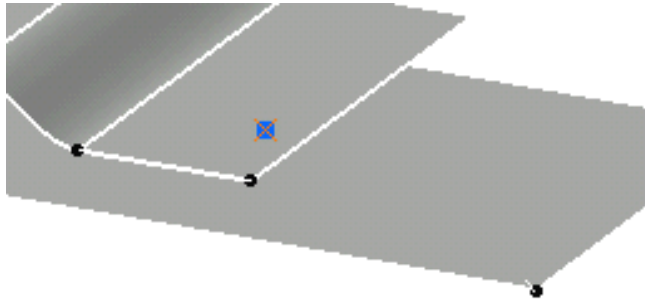
Contact Zones

Show Projected Points

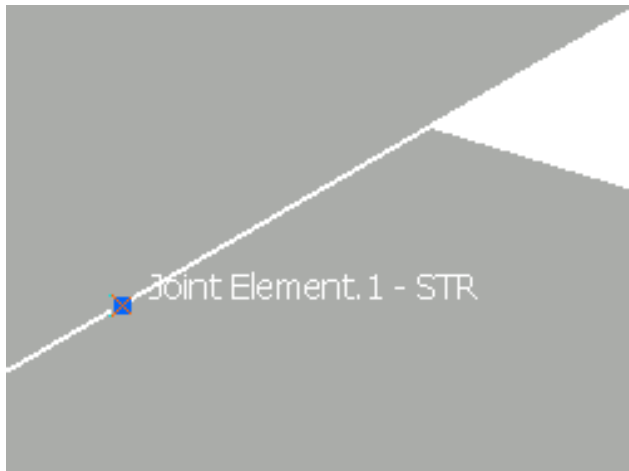
4. Click the **Unset All** button to uncheck all the options, except for the Shape Representation option. Similarly click the **Set All** button to check all options.

5. Click the **Show** buttons to visualize the different options.

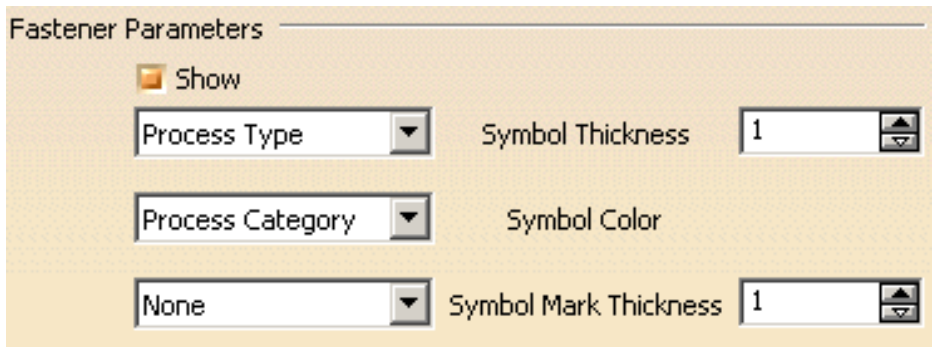
- **Shape Representation**





- **Text ID**



- **Fastener Parameters**



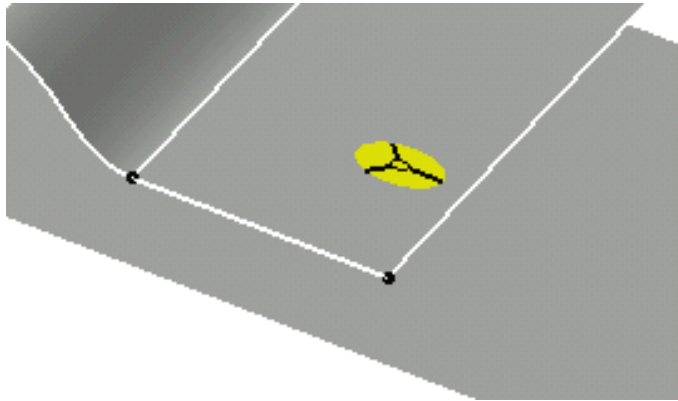
You can associate a symbol with a specification:


- a process type (by default):  
the symbol is displayed according to the symbol code assigned to the Fastener Process Type in the standard file. By default, if no symbol is assigned to the process type in standard, or if there is no imported standard, the symbol is  (101)
- an unspecified type (Unspec):  
the default symbol  is displayed. It can be modified through the **Properties**

contextual command.

- the thickness count: the symbol is displayed according to symbol code assigned to the Thickness Count in the standard file.

By default, if no symbol is assigned to the thickness count in standard, or if there is no imported standard, the symbol is:




 for 1 thicknesses


(1)

 for 2 thicknesses


(2T)

 for 3 thicknesses

(3T)

 for 4 thicknesses

(4T)

 for more than 4 thicknesses







- None: no symbol is visualized.



Once the symbol is associated with a specification, it can not longer be modified when editing the fastener's properties, except for the Unspecified type.

*The example above shows that the graphic symbol is assigned to the Laser process type. The symbol color is assigned to the Welding process category.*

| // Symbol<br>Dependency from<br>Thickness Count |     |                                                                                           |
|-------------------------------------------------|-----|-------------------------------------------------------------------------------------------|
| NTH_2_SYM                                       | 151 | For 1T:  |
| NTH_3_SYM                                       | 113 | For 2T:  |
| NTH_4_SYM                                       | 114 | For 3T:  |
|                                                 |     | For 4T:  |

The example above shows how to define a graphic symbol associated to a thickness count

**You can associate a color with a specification:**

- a process category (by default): the color is displayed according to the color code assigned to the Fastener Process Category in the standard file.



By default, if no color is assigned to the Process Category or no standard is imported, the default color is:

- Yellow for Welding
- Cyan for Adhesive
- Magenta for Sealant
- Green for BiW Mechanical
- White for Unspecified

- an unspecified type (Unspec): a default color is displayed:

- Yellow for Welding
- Cyan for Adhesive
- Magenta for Sealant
- Green for BiW Mechanical
- White for Unspecified

- None: no symbol color is visualized.

-  In this case, the color associated with the process category will be transparent.
-  Once the color is associated with a specification, it can not longer be modified when editing the fastener's properties, except for the Unspecified type.



**You can associate marks with the specification:**

- Robustness
- Regulation
- Finish
- None

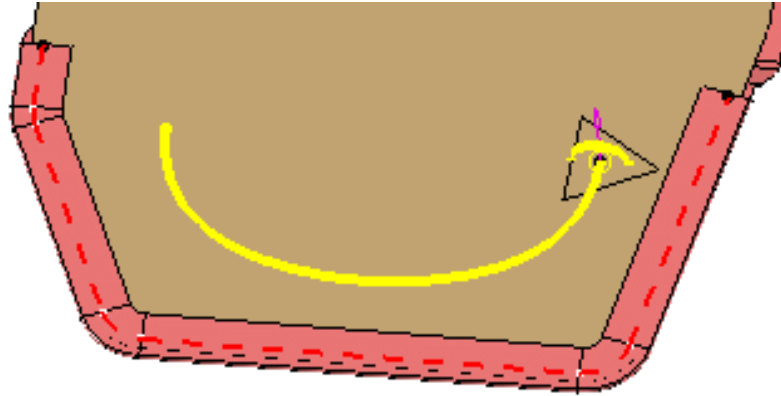
For instance assign the 1 mark code for the Robustness parameter and which value set by default is C:



You will get this Mark:

Note that:

- 1 is for assigning a mark
- 0 is for assigning no mark.



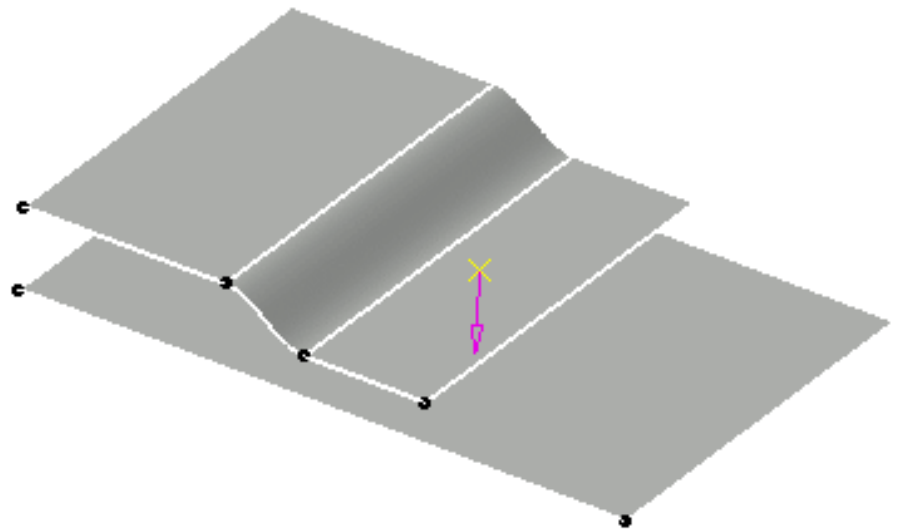
In this case all the Fasteners with the Robustness specification will get the mark symbol.



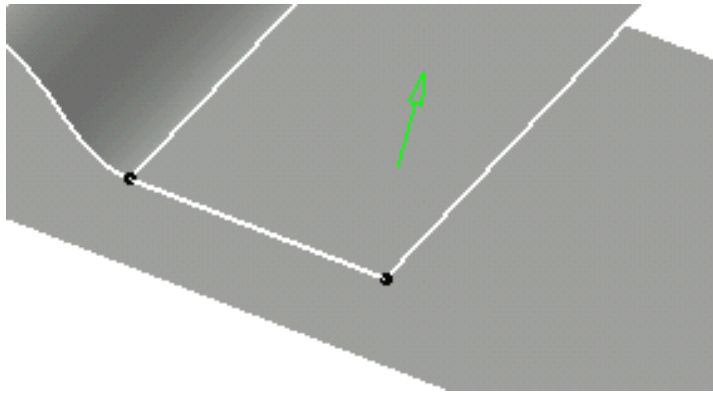
In the Standard file, Marks are assigned by default:

- For Robustness parameter ("Safety") enter the C value.
- For Regulation parameter ("Norm A") enter the A value.
- For Finish parameter ("Class A") enter the A value.

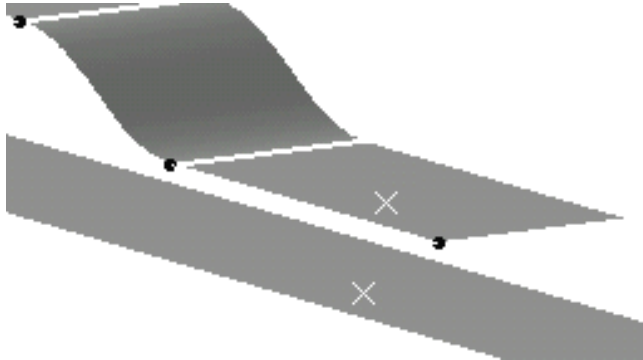
- **Fastener Normal Vector Orientation**



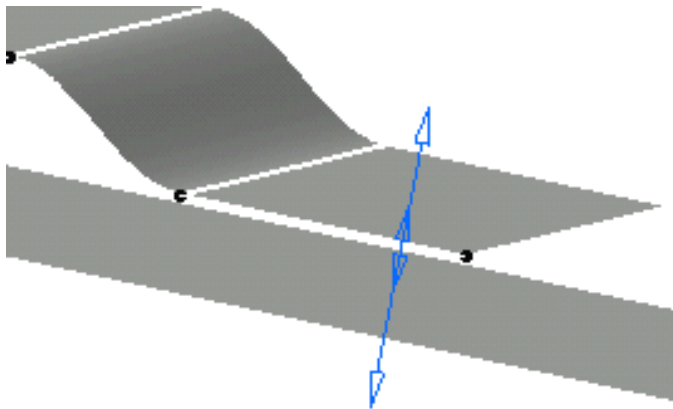
- **Support Contact Zone Material Orientation**



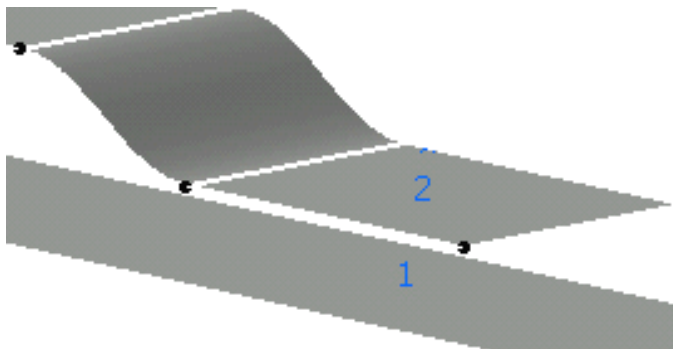
- **Show Projected Points**



- **Show Material Orientation**



- **Show Thickness Stack-Up Order**



6. Click **OK** to confirm setting these options.




- You can define the thickness and the material orientation for a GSD feature using the *Thin Parts Attribute* command in the Generative Shape Design workbench. Refer to the *Applying a Thickness* chapter in the Generative Shape Design documentation.
- The options listed above can also be accessed using the fasteners' properties via the contextual menu. Refer to the *Editing the Fasteners' Properties* chapter.



# Customizing For Generative Shape Design

[General Settings](#)  
[Working with a Support](#)

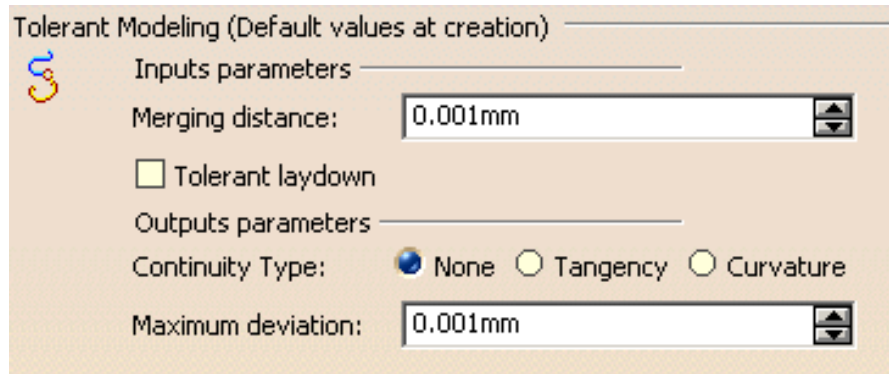
# General Settings

 This task explains how to customize general settings for Generative Shape Design workbenches.

 **1.** Select the **Tools** -> **Options** menu item.

The Options dialog box is displayed.


**2.** Select the **Shape** category then the **Generative Shape Design** subcategory to display the General tab:



 These settings are available with the Projections and Parallel Curves functionalities: next time you open one of them, the Smoothing type and the Maximum deviation will already be set.


## Tolerant Modeling

### Input parameters

 • **Merging distance:** default value defining the distance below which elements are to be joined or healed.

 This option is available with the following commands: Join and Healing.

• **Tolerant laydown:** in case the lay down of an input guide on a surface fails, you can check the **Tolerant laydown** button. When activated, a fixed lay down tolerance of 0.1mm is applied.

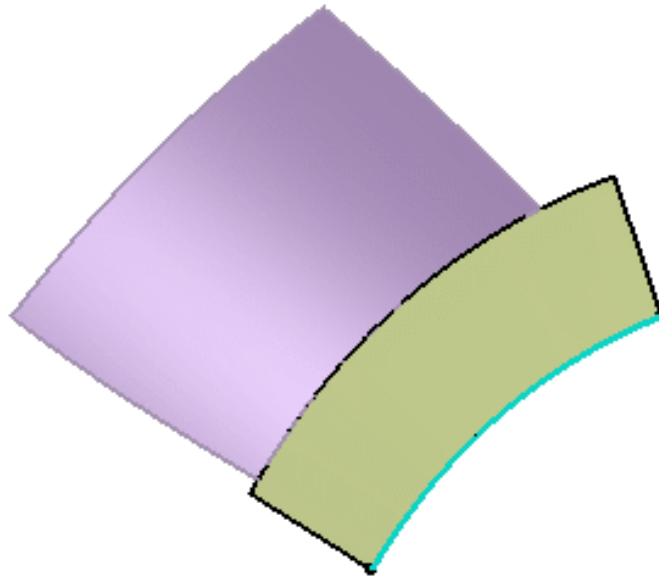
 This option is available with the following commands: Parallel Curve, Sweep, Multi-Sections Surface, Blend, Split, and Curve Smooth.



Here is a scenario with the Sweep command. You can open the [TolerantLaydown.CATPart](#) document.

1. Create a tolerant swept surface and define a **Deviation from guide** of 0.1 mm.

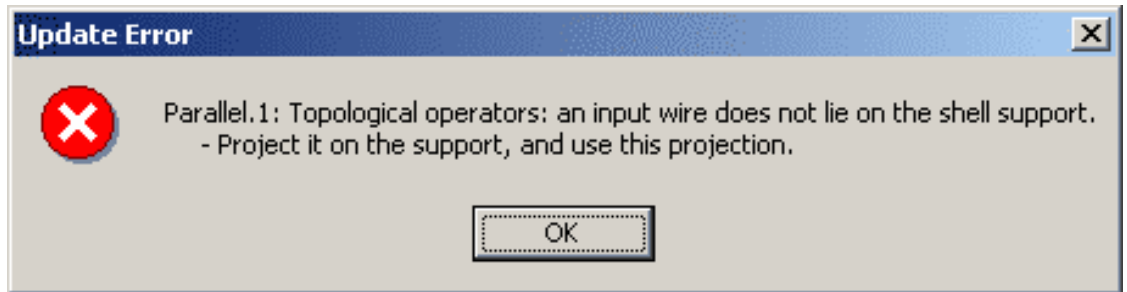
In our scenario we created a swept surface using an implicit linear profile and the "two limits" sub-type. We selected Curve.1 and Curve.2 as the guide curves.



2. Create a parallel curve of one of the guide curves using the swept surface as the support.

The creation of the parallel curve fails.

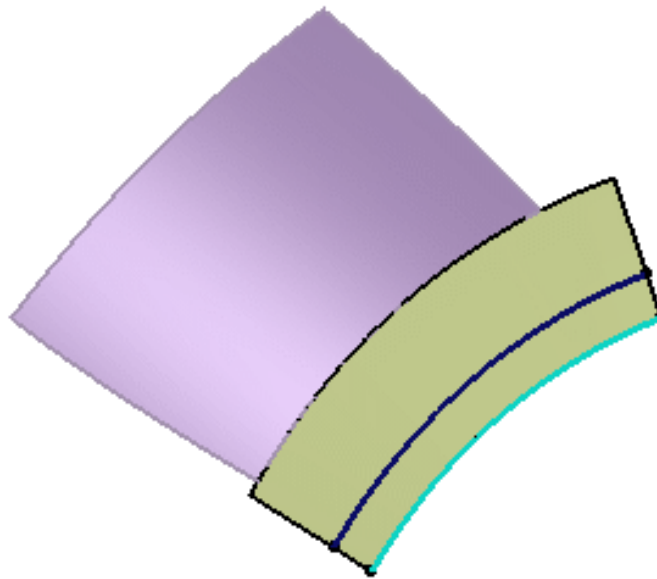
An error message opens informing you that the guide curve does not lie on the swept surface.




3. Check the **Tolerant laydown** option.

4. Perform step 2 again.


The creation of the parallel curve is successful.




 It is advised not to use a wire that lies on the edge of the sweep when working with the **tolerant laydown** option.

### Output parameters

- Choose the Continuity Type:
  - **None**: deactivates the smoothing result
  - **Tangency**: enhances the current continuity to tangent continuity
  - **Curvature**: enhances the current continuity to curvature continuity

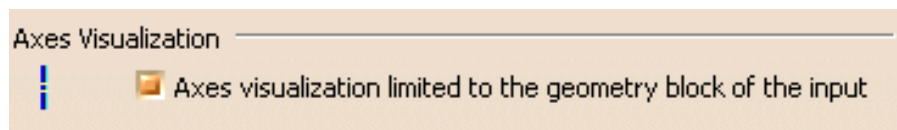
 This option is available with the following commands: Project and Parallel Curve.

- You can specify a **Maximum deviation** to set the allowed deviation between the initial element and the smoothed element by entering a value or using the spinners.

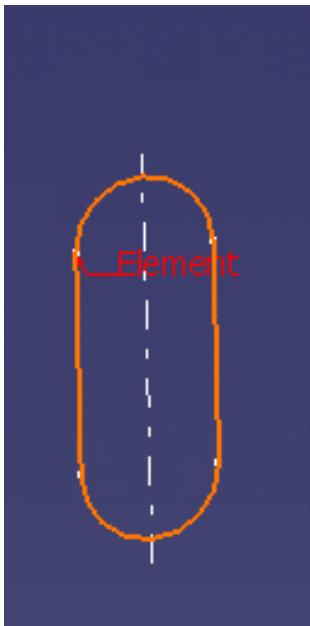
 This option is available with the following commands: Project, Parallel Curve, Sweep, Multi-Sections Surface, and Curve Smooth.

For the Sweep and Multi-Sections Surface commands, only the **Deviation** parameter can be defined from **Tools -> Options** (the **Angular correction** parameter cannot be defined here). The **Deviation** and **Angular correction** will be activated only if the smoothing type is set to **Tangency** or **Curvature**.

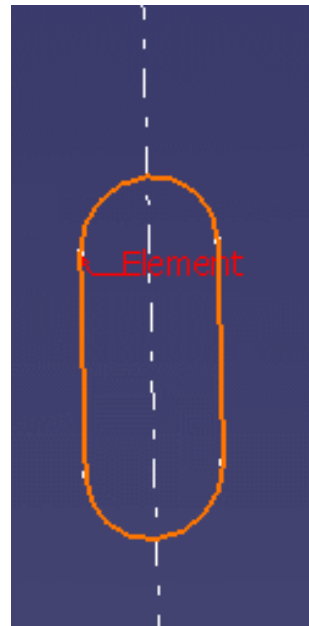
## Axes Visualization



Uncheck the **Axes visualization limited to the geometry block of the input** button to visualize an infinite axis in the 3D geometry.

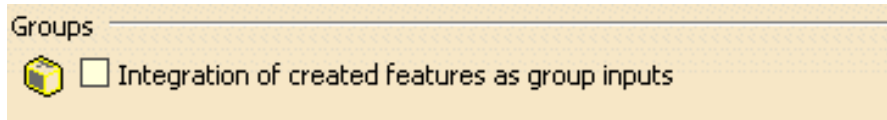


*Option checked*



*Option unchecked*

 **Groups**



- Check the **Integration of created features as group inputs** option if you want each new feature to be included as an input in an existing group and remain visible in the specification tree. If you uncheck this option, created features will not be included in the group and will be hidden in the group tree (expand the group to be able to see it).



- This option is only available when creating a new feature.
- It is only available for features accessible in the Generative Shape Design workbench. All other features will not be included in the group even if the option is checked.


For further information, please refer to the Managing Groups chapter.

3. Click **OK** to confirm setting these permanent options.





# Working with a Support

 This task explains how to customize certain options that are used when creating a support.

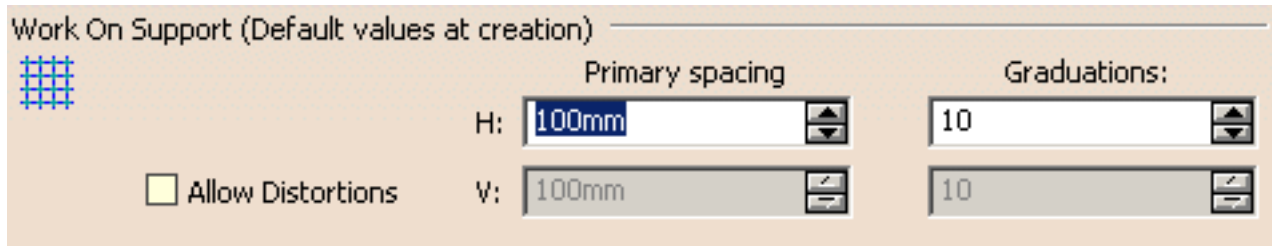


1. Select the **Tools** -> **Options** menu item.

The Options dialog box is displayed.

2. Select the **Shape** category then the **Generative Shape Design** subcategory to display the Work On Support tab:

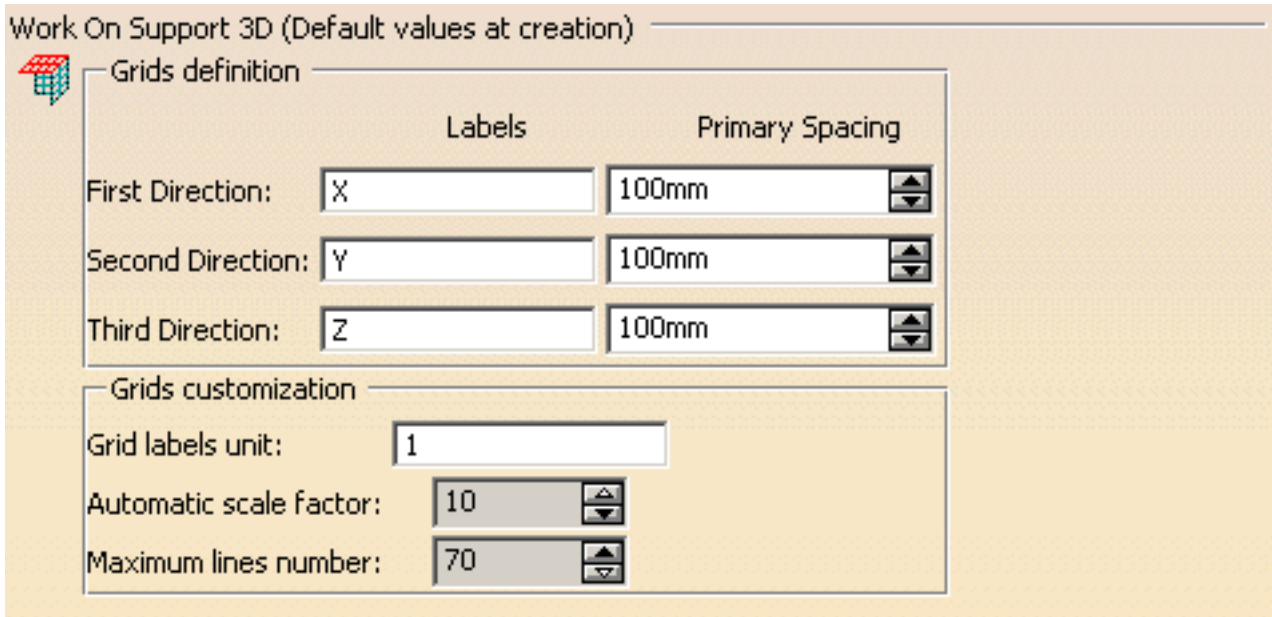
## Work on Support



- Define the First direction scale (H for horizontal), by setting **Primary spacing** and **Graduations** values.
- If you wish, you can define another scale for the **Second direction** scale (V for vertical), thus allowing distortions of the grid. Check the **Allow distortions** option to activate the **Primary spacing** and **Graduations** fields of the second direction.




## Work on Support 3D




- Define the default values for the grids, by defining **Labels** names and **Primary Spacing** values for each direction.
- Define the **Grid labels unit**, that is the coefficient of division for all straight lines units. For example, the grid label unit is set to 1 and the X label for one line is set to 500. If you modify the grid label unit to 10, the X label will be 50.
- Define the **Automatic Scale Factor**, that is the scale to be used when the grid becomes too small. The factor is comprised between 2 to 10.
- Define the **Maximum lines number** to be displayed on the screen, from 70 up to 500 lines.

These values will be used as the default values when creating a support.

 Please refer to the *Working With a Support* and *Working With a 3D Support* chapters in the Generative Shape Design documentation for further information.



# Customizing For Shape Sculptor

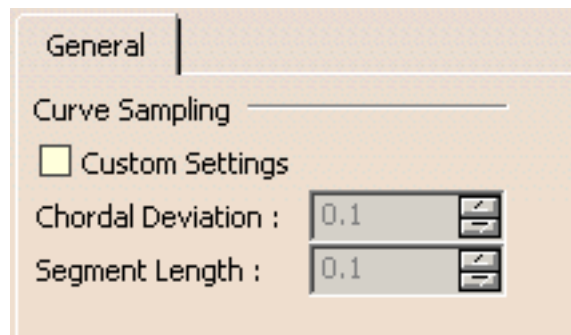
 This task explains how to customize general settings for the Shape Sculptor workbench.



1. Select the **Tools** -> **Options** menu item.

The Options dialog box is displayed.

2. Select the **Shape** category then the **Shape Sculptor** subcategory to display the General tab:



3. Check **Custom Settings** to enable the modification of parameters that control the precision of the curves.
4. Define the **Chordal Deviation** by entering a value or using the spinners.
5. Define the **Segment Length** by entering a value or using the spinners.
6. Click **OK** to confirm setting these options.



# Analysis & Simulation

Tolerance Analysis  
Analysis

# Customizing for Tolerance Analysis of Deformable Assembly

This section describes the different types of setting customization you can perform in Tolerance Analysis of Deformable Assembly using the **Tools -> Options** command.

All tasks described here deal with permanent setting customization.



# General

 This task will show you how to customize Tolerance Analysis of Deformable Assembly general settings.



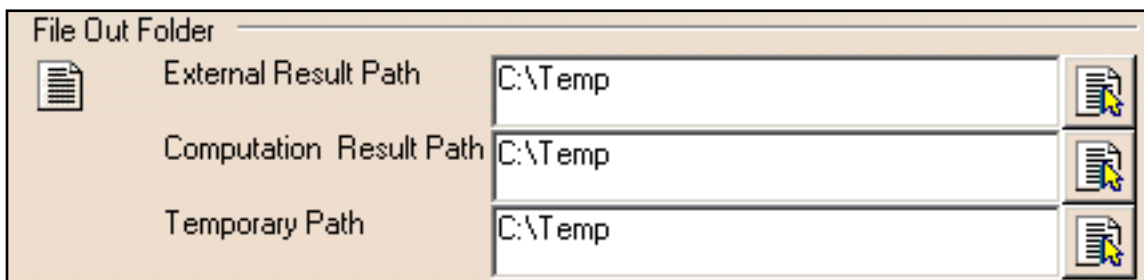
1. Select **Tools -> Options...**

The **Options** dialog box appears.

2. Click the **Analysis & Simulation** category.
3. Click the **Tolerance Analysis** subcategory.
4. Click the **General** tab

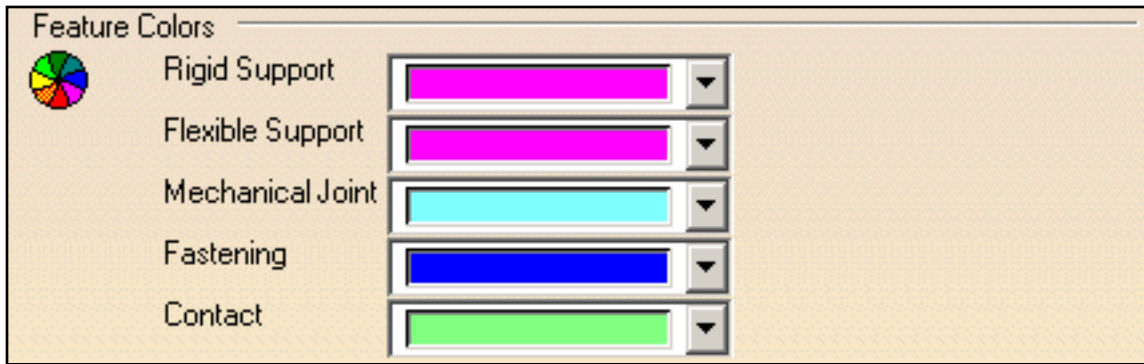
The **General** tab appears, displaying five categories of options:

## File Out Folder



- **External Result Path**
  - Define the folder path where to save file out of tolerance analysis files.
- **Computation Result Path**
  - Define the folder path where to save file out of computation files.
- **Temporary Result Path**
  - Define the folder path where to save file out of temporary files.

## Feature Colors



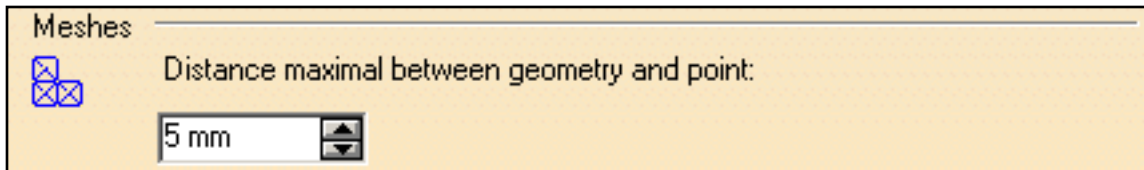
- **Rigid Support**
  - Define the default colors of rigid support.
- **Flexible Support**
  - Define the default colors of flexible support.
- **Mechanical joint**
  - Define the default colors of mechanical joint.
- **Fastening**
  - Define the default colors of Fastening.
- **Contact**
  - Define the default colors of Contact.

## Links Creation Mode



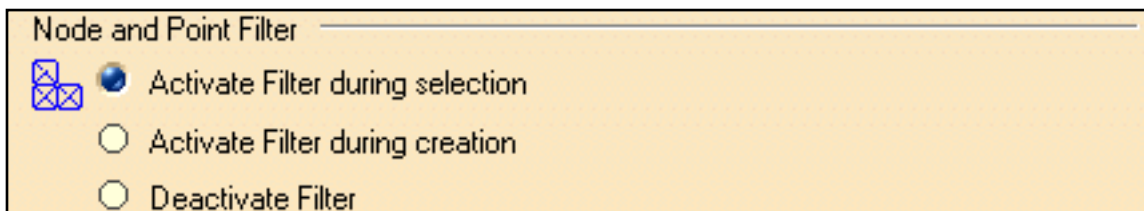
- **Automatic**
  - Define whether the links creation can be automatic or not. With the automatic mode, the application take into account the appropriate contacts around a fastening element, else user must define the contacts take into account the fastening element.

## Meshes



- **Distance maximal between geometry and point**
  - Define the maximal distance allowed between a point defined as node by user and the part geometry.  
Out of this distance the point is not take into account when the mesh is automatically generated.

## Node and Point Filter



- **Activate Filter during selection**
  - Validate node or point according to the tolerance analysis feature context when selecting them.
- **Activate Filter during creation**
  - Validate node or point according to the tolerance analysis feature context when creating the tolerance analysis feature (Clicking **OK** in its dialog box).
- **Deactivate Filter**
  - Validate node or point according to the tolerance analysis feature context when computing the tolerance analysis.





# Fastening

 This task will show you how to customize Tolerance Analysis of Deformable Assembly fastening settings.

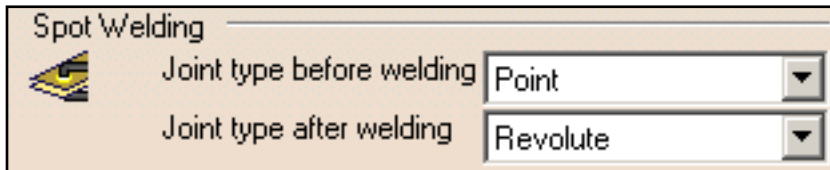
 **1. Select Tools -> Options...**

The **Options** dialog box appears.

- 2. Click the Analysis & Simulation category.**
- 3. Click the Tolerance Analysis subcategory.**
- 4. Click the Fastening tab**

The **Fastening** tab appears, displaying four categories of options:

## Spot Welding

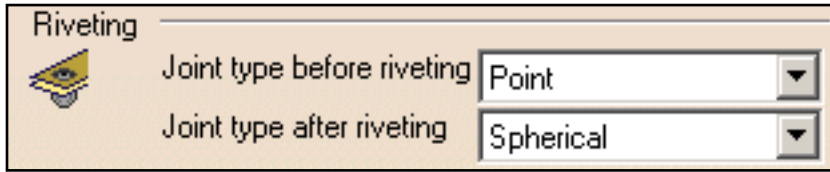


- **Joint type before welding**
  - Define the mechanical joint take into account to represent spot welding before a spot welding activity.
- **Joint type after welding**
  - Define the mechanical joint take into account to represent spot welding after a spot welding activity.

Available settings are:

| Before | After                       |
|--------|-----------------------------|
| Point  | Revolute<br>or<br>Spherical |
| Planar | Revolute                    |

# Riveting

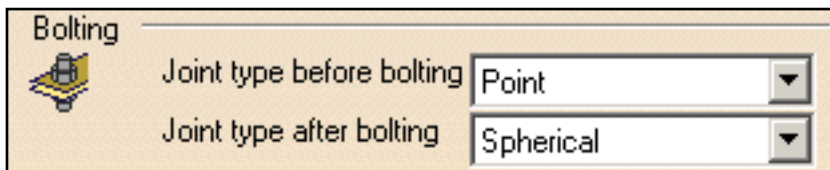


- **Joint type before riveting**
  - Define the mechanical joint take into account to represent spot welding before a riveting activity.
- **Joint type after riveting**
  - Define the mechanical joint take into account to represent spot welding after a riveting activity.

Available settings are:

| Before    | After                 |
|-----------|-----------------------|
| Point     | Revolute or Spherical |
| Planar    | Revolute              |
| Spherical | Revolute or Spherical |
| Revolute  | Revolute              |

# Bolting



- **Joint type before bolting**

- Define the mechanical joint take into account to represent spot welding before a bolting activity.

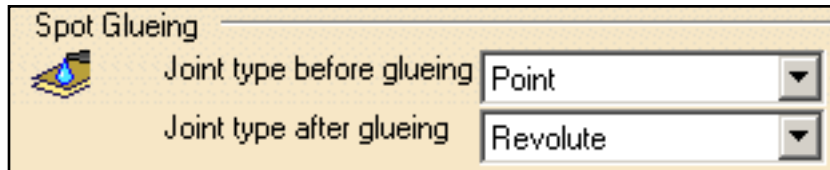
- **Joint type after bolting**

- Define the mechanical joint take into account to represent spot welding after a bolting activity.

Available settings are:

| Before    | After                       |
|-----------|-----------------------------|
| Point     | Revolute<br>or<br>Spherical |
| Planar    | Revolute                    |
| Spherical | Revolute<br>or<br>Spherical |
| Revolute  | Revolute                    |

## Spot Glueing



- **Joint type before glueing**

- Define the mechanical joint take into account to represent spot welding before a spot glueing activity.

- **Joint type after glueing**


- Define the mechanical joint take into account to represent spot welding after a spot glueing activity.

Available settings are:

| <b>Before</b> | <b>After</b>                |
|---------------|-----------------------------|
| Point         | Revolute<br>or<br>Spherical |
| Planar        | Revolute                    |



# Tolerancing

 This task will show you how to customize Tolerance Analysis of Deformable Assembly tolerancing settings.



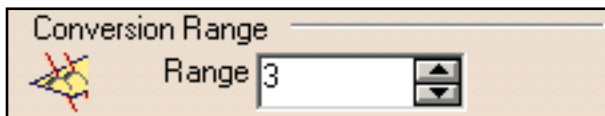
1. Select **Tools -> Options...**

The **Options** dialog box appears.

2. Click the **Analysis & Simulation** category.
3. Click the **Tolerance Analysis** subcategory.
4. Click the **Tolerancing** tab

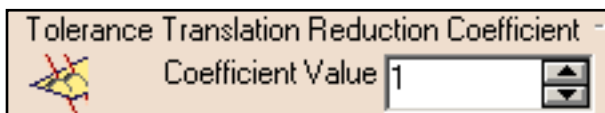
The **Tolerancing** tab appears, displaying two categories of options:

## Conversion Range



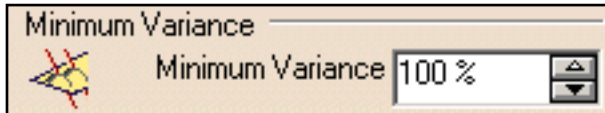
- **Range**
  - Define the range value to switch between tolerance interval and statistic law. Example, with a range value of 3 and applied to a normal law, 99.73% of measures contained in the tolerance interval represent the statistic law field.

## Tolerance Translation Reduction Coefficient



- **Coefficient Value**
  - Define the coefficient value to translate Functional tolerances due to form uncertainty in deviation point grid.

# Minimum Variance



- **Minimum Variance**
  - Define the percentage value of minimum variance to translate Functional tolerances.



# Customizing



This section describes the different type of setting customization you can perform in the Analysis workbenches using the **Tools -> Options...** submenu.

This type of customization is stored in permanent setting files: these settings will not be lost if you end your session.

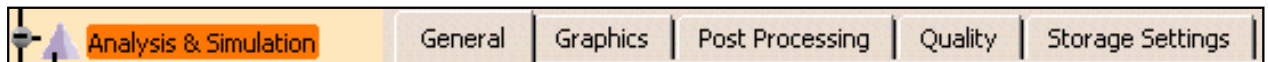


1. Select the **Tools -> Options...** submenu.

The Options dialog box appears.

2. Select the **Analysis & Simulation** category.

The following tab appears:



These tabs lets you define the

- o [general settings](#)
- o [graphic settings](#)
- o [post processing settings](#)
- o [quality settings](#)
- o [storage settings](#)

3. Change the desired parameters.
4. Click **OK** in the Options dialog box when done.



# General

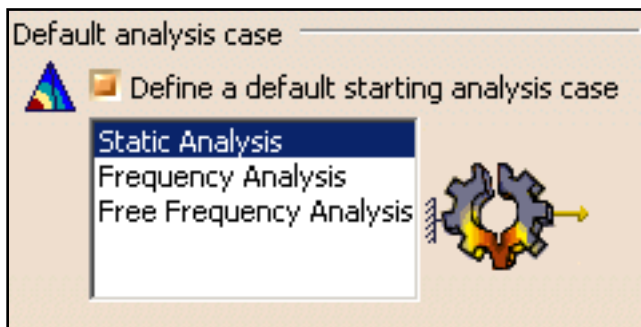
This task explains how to customize Analysis & Simulation general settings.

The General tab deals with the following settings:

- [the default analysis case choice](#)
- [the specification tree display](#)

Click [here](#) to see the parent page.

## Default Analysis Case




### Define a default starting analysis case

This option lets you define a default analysis case that will be inserted each time you enter the Generative Structural Analysis workbench or the Advanced Meshing Tools workbench.


Before defining a default analysis case using **Tools->Options** command, make sure you started the Analysis & Simulation (Generative Structural Analysis or Advanced Meshing Tools) workbench at least once.

The default starting analysis case is Static Analysis. You can decide that the new default case will be:

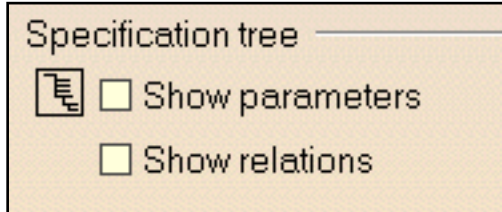
- **Static Analysis**
- **Frequency Analysis**
- **Free Frequency Analysis**

 The cases will only be displayed if an analysis workbench has been loaded at least once because the listed cases are linked to the Analysis workbenches last loaded.




 By default, this option is deactivated.

## Specification Tree




### Show parameters

This option lets you display parameters in the specification tree.

 By default, this option is deactivated.

### Show relations

This option lets you display relations in the specification tree.

 By default, this option is deactivated.

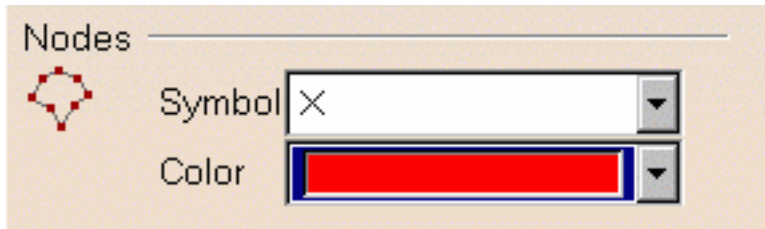
# Graphics

This page deals with the following options:

- [Nodes](#)
- [Elements](#)

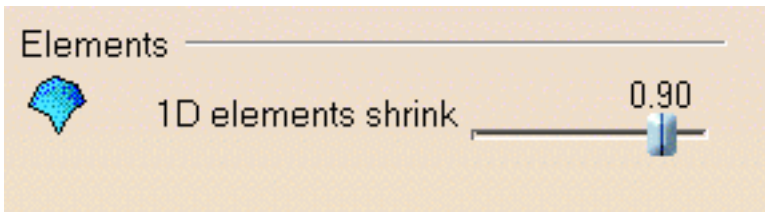
Click [here](#) to see the parent page.

## Nodes



This option lets you select the symbol and color you wish to assign to the nodes.

## Elements



This option lets you define the shrink of 1D elements.

# Post Processing

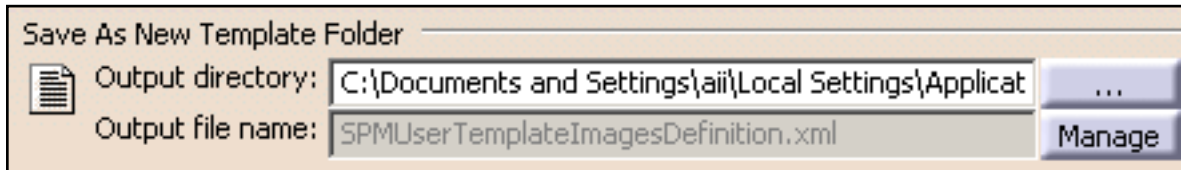
This task explains how to customize Analysis & Simulation post processing image settings.

You can define the location of the **SPMUserTemplateImageDefinition.xml** file or manage this file.

This file contains all the generated images that have been saved with the **Save As New Template** contextual menu.

Click [here](#) to see the parent page.

## Save As New Template Folder



### Output directory

This option lets you choose the directory in which you want to store the **.xml** file.

 By default, this field is empty.

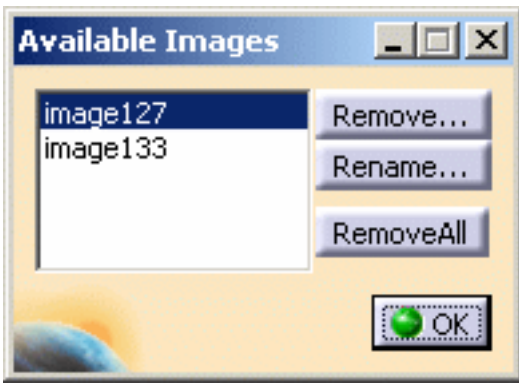
### Output file name

This option indicates the name of the associated **.xml** file. You can rename or remove the stored images.



Images must have been saved with the **Save As New Template** contextual menu and a **.CATAnalysis** document must be launched (in the opposite case, the **Manage** button is not available).

If you click the **Manage** button, the Available Images dialog box appears.



- **Remove...:** lets you remove the selected images (multi-selection is available).
- **Rename...:** lets you rename a selected image.



- **Remove All:** lets remove all the images that are stored in the xml file.



All modifications are updated only if you click **OK** in the Options dialog box.

For example: if you change the path directory after managing images and without clicking **OK** in the Options dialog box, your modifications are not preserved.

# Quality



This page deals with the following options:

- [Quality Criteria](#)
- [Default Standard File](#)
- [Export Default Format](#)



Click [here](#) to see the parent page.

## Quality Criteria



| Quality Criteria                      |                                                                                   |                                                                                   |
|---------------------------------------|-----------------------------------------------------------------------------------|-----------------------------------------------------------------------------------|
| Criteria                              |  |  |
| <input type="checkbox"/> Taper        | 0.5                                                                               | 0.7                                                                               |
| <input type="checkbox"/> Skewness     | 0.7                                                                               | 0.9                                                                               |
| <input type="checkbox"/> Distortion   | 35                                                                                | 45                                                                                |
| <input type="checkbox"/> Jacobian     | 0.2                                                                               | 0                                                                                 |
| <input type="checkbox"/> Warp Factor  | 5                                                                                 | 10                                                                                |
| <input type="checkbox"/> Warp Angle   | 30                                                                                | 60                                                                                |
| <input type="checkbox"/> Skew Angle   | 60                                                                                | 30                                                                                |
| <input type="checkbox"/> Stretch      | 0.3                                                                               | 0.1                                                                               |
| <input type="checkbox"/> Min. Length  | 0.0001                                                                            | 1e-006                                                                            |
| <input type="checkbox"/> Max. Length  | 10000                                                                             | 1e+005                                                                            |
| <input type="checkbox"/> Shape Factor | 0.3                                                                               | 0.1                                                                               |
| <input type="checkbox"/> Length Ratio | 5                                                                                 | 10                                                                                |

This frame lets you visualize the quality criteria that are taken into account and their limit values between:

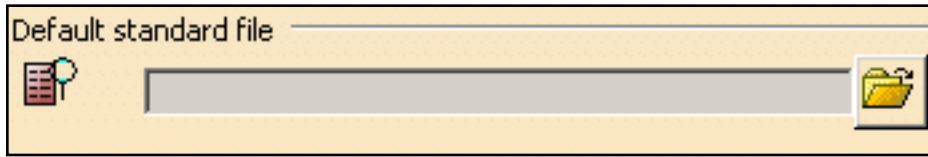
-  good and poor elements
-  poor and bad elements

 By default, all the **Quality Criteria** are taken into account.

The limit values change as you define the **Default Standard File** option.



## Default Standard File

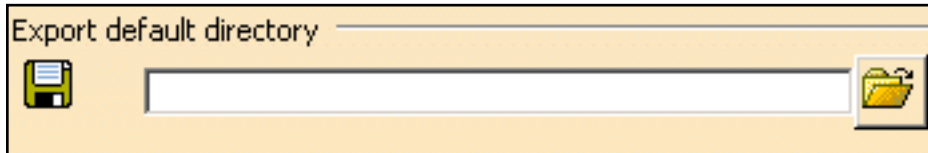


Default standard file

This option lets you define the list of quality criteria that will be used by default.

By default, the **Default Standard File** field is empty and so all the **Quality Criteria** are taken into account.

## Export Default Directory



Export default directory

This option lets you define the default directory in which the criteria configuration have been saved.

By default, the **Export Default Directory** field is empty.

While a default directory is not defined, you cannot use the **Export Criteria** option in the **Quality Analysis** functionality in the Advanced Meshing Tools workbench (for more details, please refer to the *Advanced Meshing Tools User's Guide - Analyzing Element Quality*)

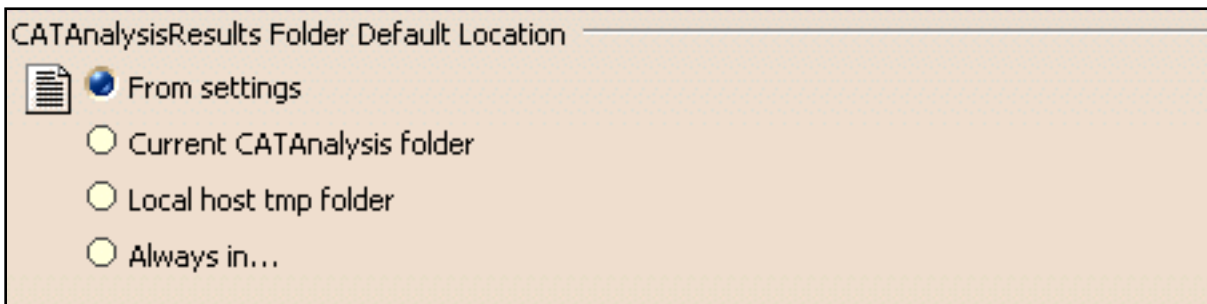
# External Storage

This page deals with the following options:

- [CATAnalysisResults Folder Default Location](#)
- [CATAnalysisComputations Folder Default Location](#)
- [Temporary Data Folder Default Location](#)
- [Clear Computations Mode](#)
- [Renaming Mode](#)

Click [here](#) to see the parent page.

## CATAnalysisResults Folder Default Location

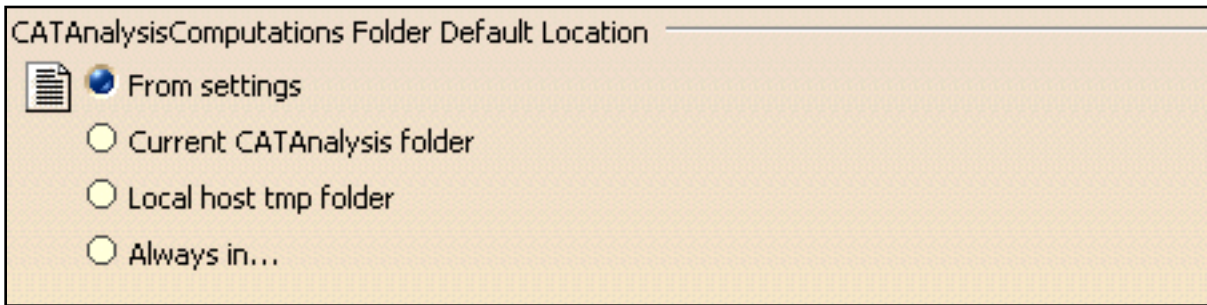


This option lets you change the default directory location of the CATAnalysisResults file.

- **From settings:** lets you choose the last selected default directory location (CATSettings).
- **Current CATAnalysis folder:** lets you choose the same default directory as the current CATAnalysis directory.
- **Local host tmp folder:** lets you choose the temporary directory.
- **Always in...:** lets you define a default directory that will be always the same. You have to define the path directory.

 By default, the **From settings** option is activated.

## CATAnalysisComputations Folder Default Location

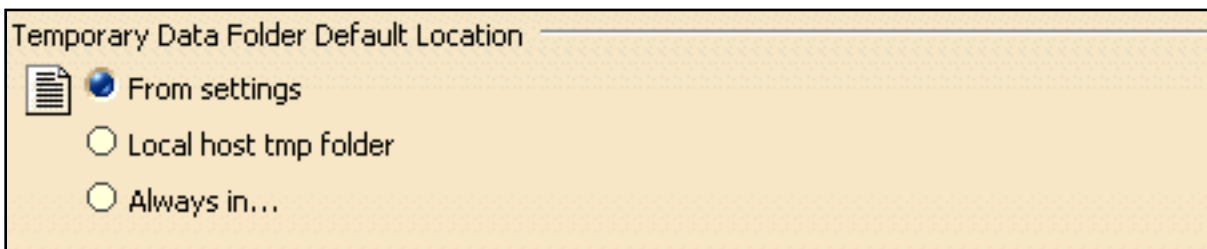


This option lets you change the default directory location of the CATAnalysisComputations file.

- **From settings:** lets you choose the last selected default directory location (CATSettings).
- **Current CATAnalysis folder:** lets you choose the same default directory as the current CATAnalysis directory.
- **Local host tmp folder:** lets you choose the temporary directory.
- **Always in...:** lets you define a default directory that will be always the same. You have to define the path directory.

 By default, the **From settings** option is activated.

## Temporary Data Folder Default Location



This option lets you specify the default directory location of temporary data.

- **From settings:** lets you choose the last selected default directory location (CATSettings).
- **Local host tmp folder:** lets you choose the temporary directory.
- **Always in...:** lets you define a default directory that will be always the same. You have to define the path directory.

 By default, the **From settings** option is activated.



## Clear Computations Mode



### Clear Computations Mode



Automatic clearing of computations data before save.

### Automatic clearing of computations data before save

This option lets you set the automatic clearing of computations data before saving documents.

By default, this option is deactivated.



## Renaming Mode

### Renaming Mode



Automatic renaming of CATAnalysisResults and CATAnalysisComputations files

### Automatic renaming of CATAnalysisResults and CATAnalysisComputations files



This option lets you rename automatically the CATAnalysisResults and CATAnalysisComputations files when you save a CATAnalysis document (using the **Save as...** menu or the **Save Management** menu) with the new name of the associated CATAnalysis document.

By default, this option is deactivated.

# AEC Plant

Refer to the *Version 5 Plant Layout User's Guide*

# Customizing Settings for NC Manufacturing



This section describes how to customize settings for NC Manufacturing.

Before you start your first working session, you can customize the settings to suit your working habits. Your customized settings are stored in permanent setting files: they will not be lost at the end of your session.



1. Select **Tools > Options** from the menu bar: the Options dialog box appears.
2. Select the **NC Manufacturing** category in the tree to the left. The options for NC Manufacturing settings appear, organized in tab pages.



3. Select the tab corresponding to the parameters to be customized.

| Parameters in this tab...     | Allow you to customize...                          |
|-------------------------------|----------------------------------------------------|
| <a href="#">General</a>       | general settings for all NC Manufacturing products |
| <a href="#">Resources</a>     | tooling, feeds&speeds and resource files           |
| <a href="#">Operation</a>     | machining operations                               |
| <a href="#">Output</a>        | PP files and NC data output                        |
| <a href="#">Program</a>       | manufacturing programs (sequencing, and so on)     |
| <a href="#">Photo / Video</a> | material removal simulation                        |

4. Change these options according to your needs.
5. Click **OK** to save the settings and quit the Options dialog box.



# Customize General Settings

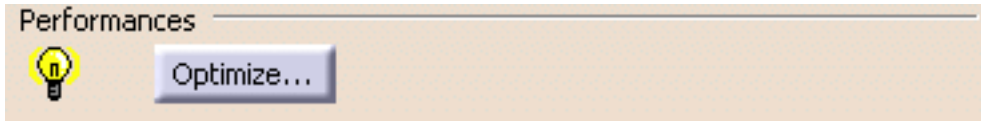
This document explains how to customize General settings for NC Manufacturing products.

These settings are accessed as follows:

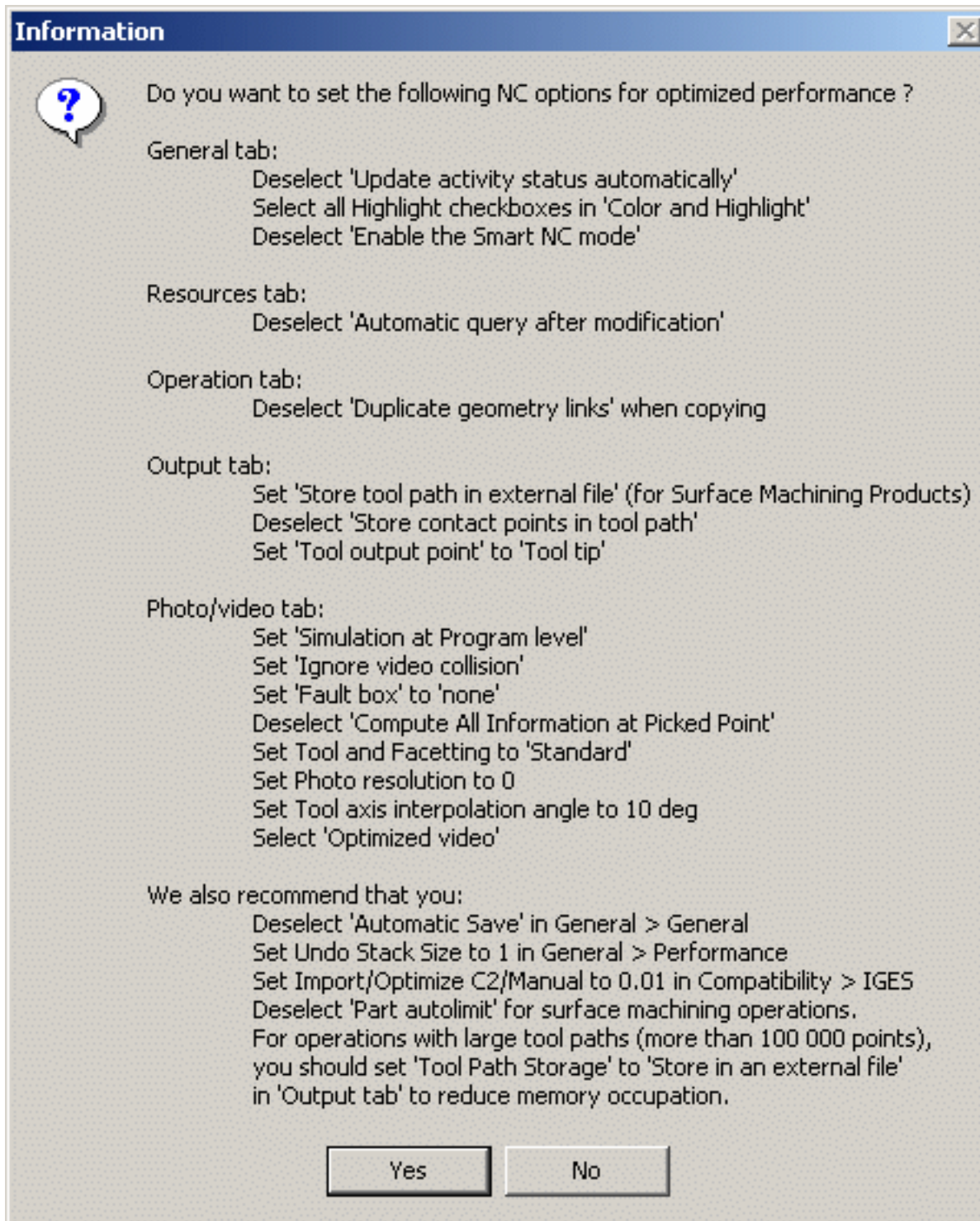
- Select **Tools > Options** from the menu bar.
- Select the **NC Manufacturing** category in the tree to the left. The options for NC Manufacturing settings appear, organized in tab pages.
- Select the **General** tab, which is divided up into areas.

| Parameters in this area...             | Allow you to customize...                                          |
|----------------------------------------|--------------------------------------------------------------------|
| <a href="#">Performance</a>            | settings for optimized performance                                 |
| <a href="#">Trees Display</a>          | display of the specification tree                                  |
| <a href="#">Color and Highlight</a>    | colors of displayed geometry and parameters                        |
| <a href="#">Tool Path Replay</a>       | tool display during tool path replay                               |
| <a href="#">Complementary Geometry</a> | handling of geometry necessary for manufacturing                   |
| <a href="#">Design Changes</a>         | use of the Smart NC mode and enhanced detection of design changes. |

## Performance



- Click the **Optimize** button in order to automatically set a number of the NC Manufacturing options for optimized performance. These options are listed in the Information dialog box that appears:

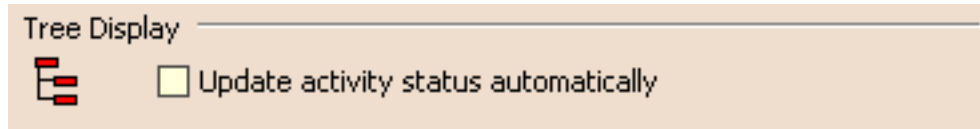




If you click **Yes**, these options will be set as described in the dialog box. Note that, if needed, you may locally reset any of these options.

If you click **No**, the options will remain with their current settings.

The Information box also lists some recommendations for manually setting other options that have an influence on performance.

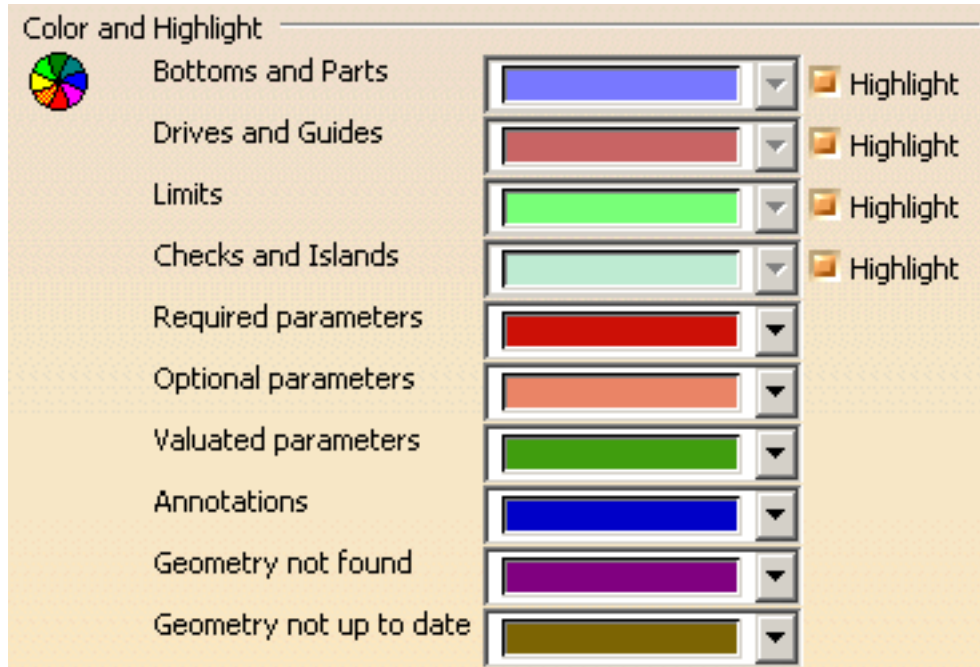
## Tree Display



- Select the checkbox if you want the status of activities in the tree to be updated automatically.
- If this checkbox is not selected:
  - you can update activity status manually in your workbench using the Update Status icon  in the Auxiliary Commands toolbar.
  - the status of the activity after a manual update is masked at the first action on the node (for example, edit, replay, collapse/expand of a parent node). To retrieve the status of the activity you must select the Update Status icon  again.

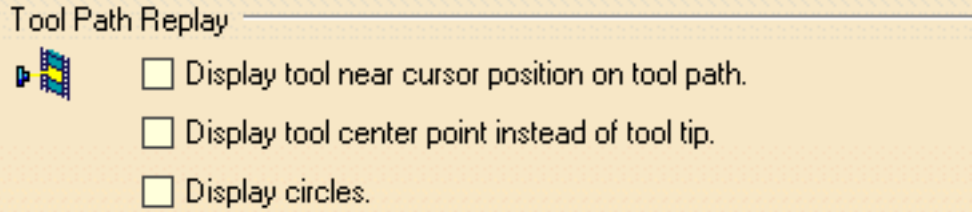
If this checkbox is not selected, performance is improved.

## Color and Highlight



- Select the colors to be used to identify the various manufacturing entities by means of the combos. Note that for Geometry that is not found or not up to date, you can select the colors used to display the valuated parameters in the corresponding Operation or Feature dialog boxes.
- For certain entities, you can select the corresponding checkbox to use highlighting. Performance is improved when all the Highlight checkboxes are selected.

## Tool Path Replay



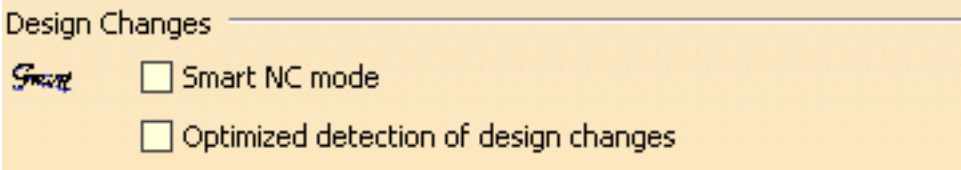
- Select the first checkbox if you want to display the tool near your cursor position on the trajectory during a tool path replay
- Select the second checkbox if you want to display the tool center point instead of the tool tip during a tool path replay
- Select the third checkbox if you want to display each circular trajectory as a circular arc instead of a set of discretization points. The extremities of the circular arc are indicated by means of 'O' symbols. This allows better control of the Point by Point replay mode, where it is necessary to make several interactions to replay a circle (because of its representation by a set of points). With the graphic representation as a circle, only one interaction is necessary to perform the replay.

## Complementary Geometry



- Select the checkbox to create a CATPart dedicated to manufacturing-specific geometry in the Product List of the PPR tree.

## Design Changes



- Select the first checkbox to activate the Smart NC mode. In this mode, an image of the geometry selected in machining operations is kept to allow analysis of design changes. Performance is improved when this checkbox is not selected.
- Select the second check box to enable a geometrical comparison mode in order to more precisely determine the design change status of machining operations.

# Customize Resource Settings

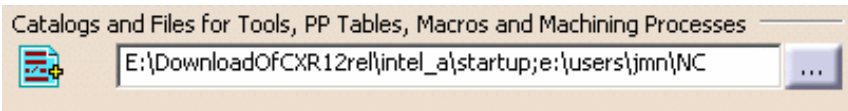
This document explains how to customize Resource settings for NC Manufacturing products.

These settings are accessed as follows:

- Select **Tools > Options** from the menu bar.
- Select the **NC Manufacturing** category in the tree to the left. The options for NC Manufacturing settings appear, organized in tab pages.
- Select the **Resources** tab, which is divided up into areas.

| Parameters in this area...                                           | Allow you to customize...                                |
|----------------------------------------------------------------------|----------------------------------------------------------|
| <a href="#">Catalogs and Files</a>                                   | the path name for resource files                         |
| <a href="#">Tool Selection</a>                                       | the selection of tools                                   |
| <a href="#">Automatic Compute from Tool Feeds and Speeds</a>         | the update of feeds and speeds according to tooling data |
| <a href="#">Tool Query Mode in Machining Processes Instantiation</a> | tool queries in machining processes                      |

## Catalogs and Files



- Enter the path of the folder containing tool catalogs, PP tables, macros, and machining processes. You can choose a folder by clicking the [...] button.

You can concatenate paths using:

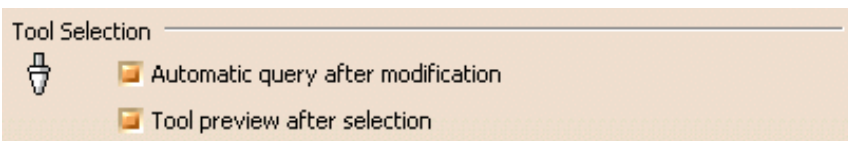
- a semi colon (;) character for Windows NT platforms.
- a colon (:) character for UNIX platform.

For example, if the concatenated folders `E:\DownloadOfCXR12rel\intel_a\startup` and `e:\users\jmn\NC` in the figure above contain PP tables, then those PP tables will be available for selection in the Part Operation's Machine Editor dialog box.

Please note, however, that:

- PP tables must be contained in folders named Manufacturing\PPTables
- tools must be contained in folders named Manufacturing\Tools.

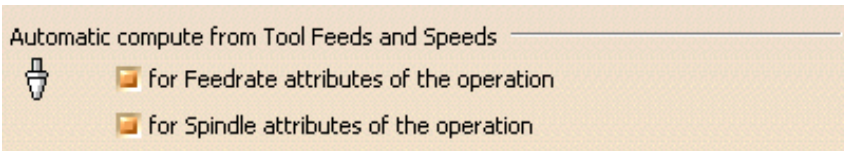
## Tool Selection



- Select the first checkbox if you want to activate an automatic query after each modification of a tool parameter. Performance is improved when this checkbox is not selected.
- Select the second checkbox if you want to preview the tool after selection.

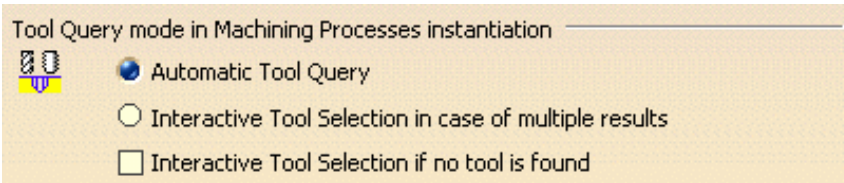


## Automatic Compute from Tool Feeds and Speeds



- Select the first checkbox if you want the Automatic Update of Feedrates option to be set by default in the Feeds and Speeds tab page of machining operations. This option allows feedrates of operations to be automatically updated whenever feedrate information on the tool is modified.
- Select the second checkbox if you want the Automatic Update of Speeds option to be set by default in the Feeds and Speeds tab page of machining operations. This option allows spindle speeds of operations to be automatically updated whenever speed information on the tool is modified.

## Tool Query mode in Machining Processes Instantiation



select the type of Tool Query to be executed when a Machining Process is instantiated:

- automatically computed Tool Query
- interactively defined Tool Selection in case of multiple results
- interactively defined Tool Selection if no tool is found.

Depending on the selected option, the Advanced tab page of the Search Tool dialog box shows the solved Tool Query for each operation in the Machining Process.

**In the example below**, you can choose one of the tools found in the ToolsSampleMP, or use the Look in combo to select a tool from the current document or another tool catalog.

**Search Tool** ? X

Look in:

Simple
Advanced

Search with criteria:

Nominal diameter <= 10mm  
Nominal diameter >= 5mm

Attribute:  Condition:  Value:

| Tool number | Comment    | Name        | Cutting angle | Length | Nominal diameter |
|-------------|------------|-------------|---------------|--------|------------------|
| 10          | Drill Tool | Drill D6    | 120           | 60     | 6                |
| 11          | Drill Tool | Drill D6,5  | 120           | 60     | 6                |
| 12          | Drill Tool | Drill D8    | 120           | 60     | 8                |
| 13          | Drill Tool | Drill D8,5  | 120           | 60     | 8                |
| 14          | Drill Tool | Drill D10   | 120           | 60     | 10               |
| 15          | Drill Tool | Drill D10,5 | 120           | 60     | 10               |

6 tool(s) found

The diagram shows a side view of a drill bit. The cutting diameter is labeled as  $db = 10\text{mm}$ . The total length is  $L = 100\text{mm}$ . The length of the cutting edge is  $l = 60\text{mm}$ . The length of the cutting lip is  $lc = 50\text{mm}$ . The length of the cutting edge from the tip to the start of the cutting lip is  $ld = 2\text{mm}$ . The cutting angle is  $A = 60\text{deg}$ . The diameter of the cutting lip is  $D = 6\text{mm}$ .

# Customize Operation Settings

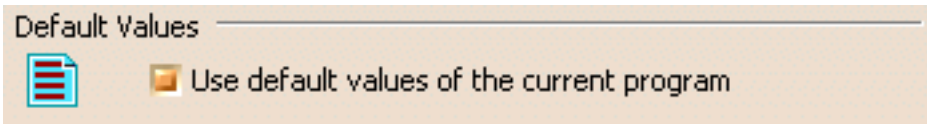
This document explains how to customize Operation settings for NC Manufacturing products.

These settings are accessed as follows:

- Select **Tools > Options** from the menu bar.
- Select the **NC Manufacturing** category in the tree to the left. The options for NC Manufacturing settings appear, organized in tab pages.
- Select the **Operation** tab, which is divided up into areas.

| Parameters in this area...                                             | Allow you to customize...                                               |
|------------------------------------------------------------------------|-------------------------------------------------------------------------|
| <a href="#">Default Values</a>                                         | the use of default values                                               |
| <a href="#">After Creation or Machining process (MP) Instantiation</a> | what happens after creating machining operations or machining processes |
| <a href="#">When Copying</a>                                           | the duplication of geometry links                                       |
| <a href="#">Display</a>                                                | tool path displays of operations                                        |
| <a href="#">User Interface</a>                                         | dialog boxes of 3-axis surface machining operations.                    |

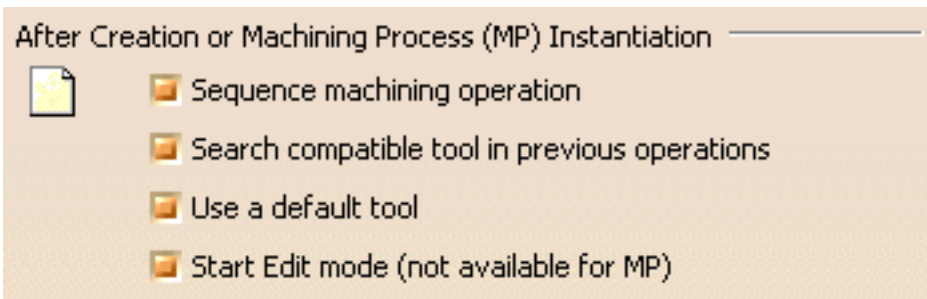
## Default Values



- Select the checkbox if you want operations to be created with the values used in the current program. The values and units of attributes at the creation step of an operation are set to the values and units of the last edited and validated operation whatever its type (that is, exit the operation definition dialog box using OK) .

Otherwise the default settings delivered with the application are used.

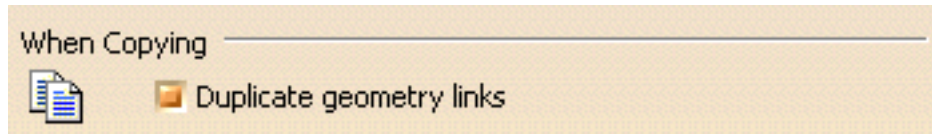
## After Creation or Machining Process (MP) Instantiation



Select the desired checkboxes to specify conditions to be applied when you create machining operations or machining processes.

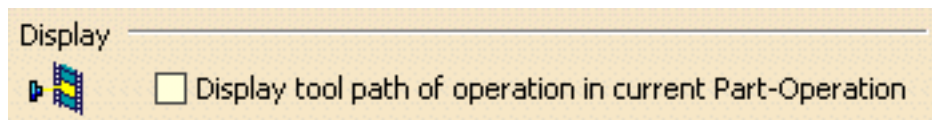
- **Sequence machining operations**  
Machining operations are automatically sequenced in the current program after creation. Otherwise, sequencing can be managed in the feature view.
- **Search compatible tool in previous operations**  
When creating an operation, if a compatible tool exists in a previous operation of the current program, it will be set in the new operation. Otherwise, the operation will be incomplete.
- **Use a default tool**  
When creating an operation, a search is done in the document to find a compatible tool. If no compatible tool exists, a default one is created in the document and set in the created operation. If checkbox is not selected, no tool will be defined on the operation.
- **Start edit mode (for machining operations only, not for machining processes)**  
When creating an operation, Edit mode is automatically started to allow modifying parameters of the created operation. Otherwise, the operation is added to the program but the machining operation editor is not started.

## When Copying



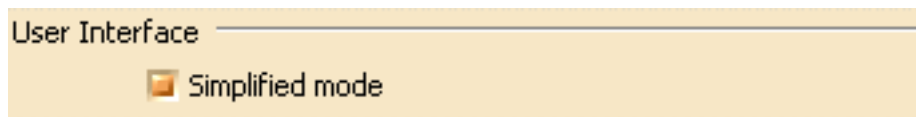
- Select the checkbox if you want geometry links to be duplicated in a copied operation. Otherwise the geometry must be defined for the copied operation. Performance is improved when this checkbox is not selected.

## Display



- Select the checkbox if you want to display tool paths of operations in the current Part Operation.

## User Interface



- Select the checkbox if you want to have the possibility of simplifying the dialog boxes of machining operations (that is, you can display the minimum number of parameters necessary for a correct tool path). This setting is available for 3-axis surface machining operations only.

# Customize Output Settings

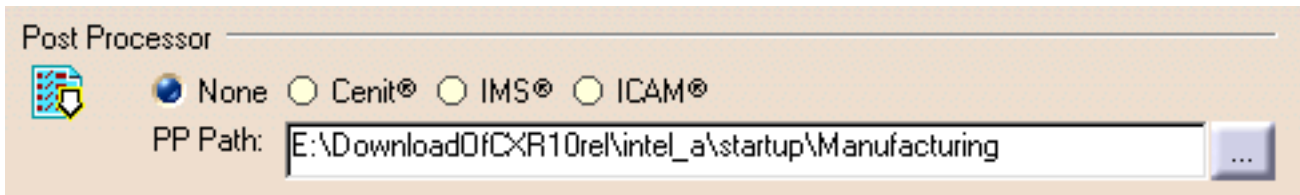
This document explains how to customize Output settings for NC Manufacturing products.

These settings are accessed as follows:

- Select **Tools > Options** from the menu bar.
- Select the **NC Manufacturing** category in the tree to the left. The options for NC Manufacturing settings appear, organized in tab pages.
- Select the **Output** tab, which is divided up into areas.

| Parameters in this area...     | Allow you to customize...                                                                                |
|--------------------------------|----------------------------------------------------------------------------------------------------------|
| Post Processor                 | the type of PP files to be used for generating NC code output and the path where these files are located |
| Tool Path Storage              | the tool path storage capability                                                                         |
| Tool Path Edition              | the tool path edition capability                                                                         |
| During Tool Path Computation   | contact point storage                                                                                    |
| Tool Output Point              | type of tool output point                                                                                |
| Tool Output Files ... Location | default paths for NC output files storage.                                                               |

## Post Processor

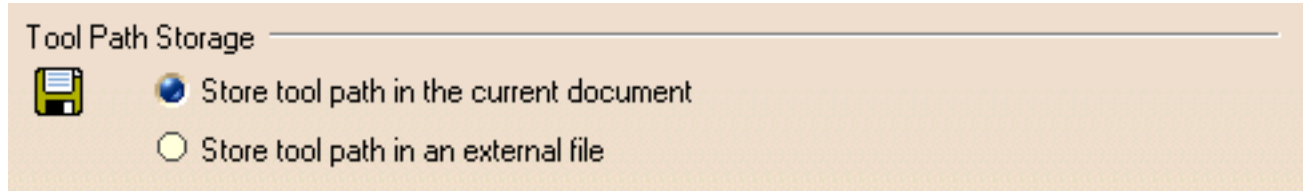


Select the desired Processor option:

- **None:** no Post Processor is defined. NC code output is not possible in this case
- **Cenit:** you can choose from among the Post Processor parameter files proposed by Cenit to generate your NC code
- **IMS:** you can choose from among the Post Processor parameter files proposed by Intelligent Manufacturing Software (IMS) to generate your NC code
- **ICAM:** you can choose from among the Post Processor parameter files proposed by ICAM Technologies Corporation (ICAM) to generate your NC code.

Enter the path of the folder containing Post processors. You can choose a folder by clicking the [...] button. File concatenation is possible.

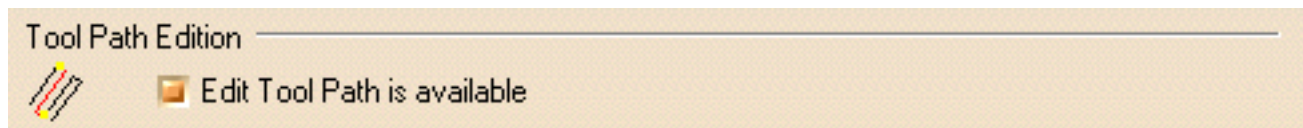
## Tool Path Storage



Select the desired option to store tool path data either in the current document or in an external file (as a tpl file).

For operations with large tool paths (more than 100 000 points), tool path storage in an external file is recommended.

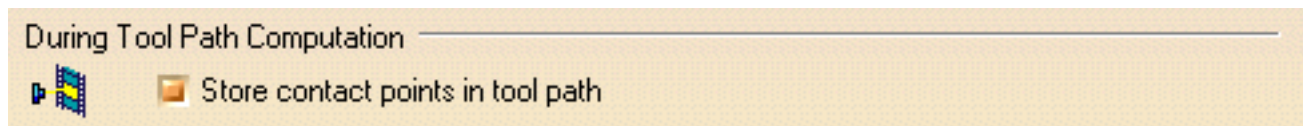
## Tool Path Edition



Select the checkbox if you want to be able to edit tool paths even when the operation is locked.

This capability is available only for activities with a tool path node in the specification tree.

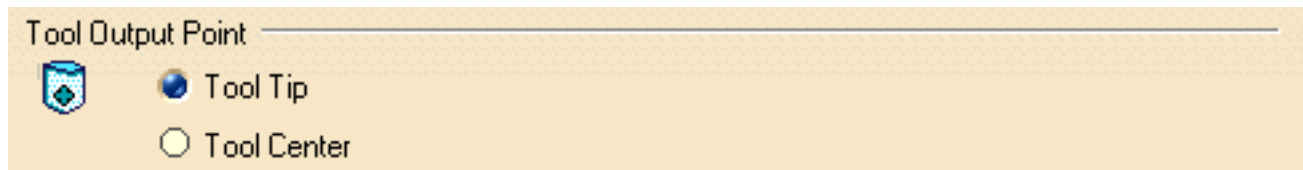
## During Tool Path Computation



Select the checkbox if you want to store contact points in the tool path.

Performance is improved when this checkbox is not selected.

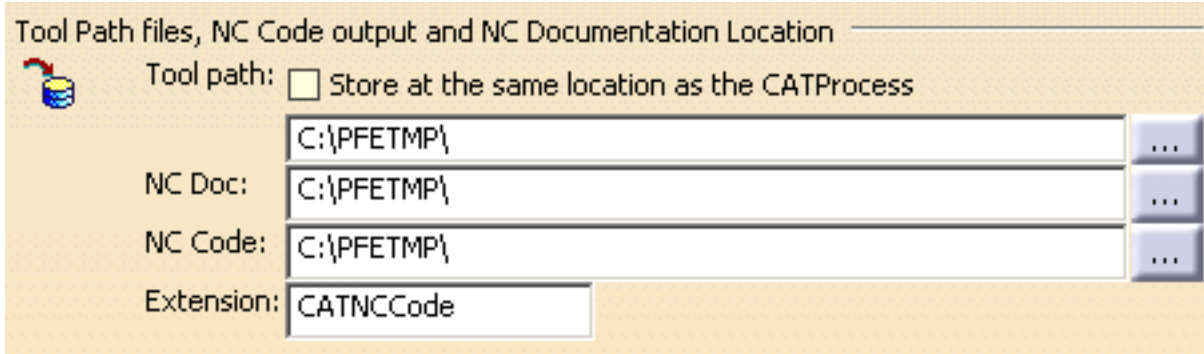
## Tool Output Point



Select the desired option to select either the tool tip or tool center point as output point.

Performance is better when the Tool Tip option is selected.

## Default File Locations



Tool Path files, NC Code output and NC Documentation Location

Tool path:  Store at the same location as the CATProcess

NC Doc: C:\PFETMP\

NC Code: C:\PFETMP\

Extension: CATNCCode

Specify default locations for storing **Tool Path files**, **NC Documentation**, and **NC Code output**.

You can store tool paths files (tpl files) in the same folder as the CATProcess by selecting the checkbox. This allows you to store these files according to your CATProcess context. Otherwise, you can choose another location by clicking the [...] button.

For NC Documentation, and NC Code output you can choose a folder easily by clicking the [...] button.

You can customize the extension to be used for NC Code output (by default, the suffix used is CATNCCode).

# Customize Program Settings

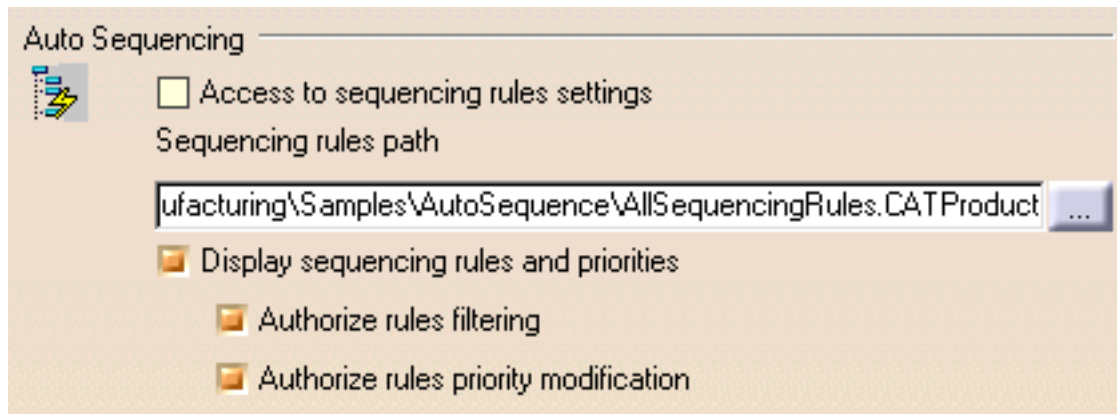
This document explains how to customize Program settings for NC Manufacturing products.

These settings are accessed as follows:

- Select **Tools > Options** from the menu bar.
- Select the **NC Manufacturing** category in the tree to the left. The options for NC Manufacturing settings appear, organized in tab pages.
- Select the **Program** tab to customize program auto-sequencing rules and priorities. These settings are mainly intended for the administrator.

Make sure that the document in the sequencing rules path (AllSequencingRules.CATProduct in the example below) is accessible in Read/Write.

## Auto Sequencing



Select the **Access to sequencing rules settings** checkbox to authorize user access to sequencing rules.

You can then specify the path for the rules base  
You can choose a rules base easily by clicking the [...] button.

Select the **Display sequencing rules and priorities** checkbox to authorize the display of sequencing rules and priorities in the user's view. In this case two more checkboxes can be selected in order to:

- allow the user to filter rules
- allow the user to modify rule priorities.



# Customize Photo and Video Settings

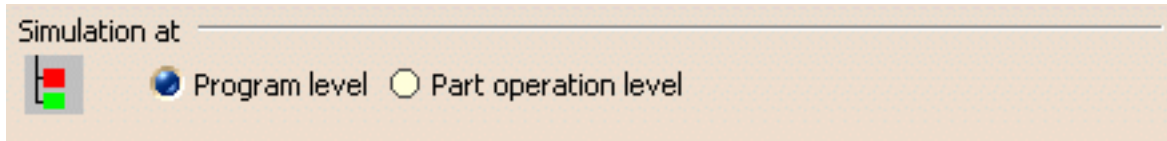
This document explains how to customize Photo / Video material removal simulation settings for NC Manufacturing products.

To access these settings:

- Select **Tools > Options** from the menu bar.
- Select the **NC Manufacturing** category in the tree to the left. The options for NC Manufacturing settings appear, organized in tab pages.
- Select the **Photo /Video** tab, which is divided up into areas.

| Parameters in this area...       | Allow you to customize...                                      |
|----------------------------------|----------------------------------------------------------------|
| <a href="#">Simulation at</a>    | material removal simulation at program of Part Operation level |
| <a href="#">Video</a>            | Video material removal simulation options                      |
| <a href="#">Photo</a>            | Photo material removal simulation options                      |
| <a href="#">Performance</a>      | settings that influence performance                            |
| <a href="#">Color</a>            | color during material removal simulation                       |
| <a href="#">Positioning Move</a> | allowed tool axis variation between two operations             |

## Simulation at



- Select the desired option to perform material removal simulation at either Program or Part Operation level.  
Depending on the selected level, simulation begins either from the start of the manufacturing program or from the start of the Part Operation.  
Best performance is obtained with Program level.

## Video

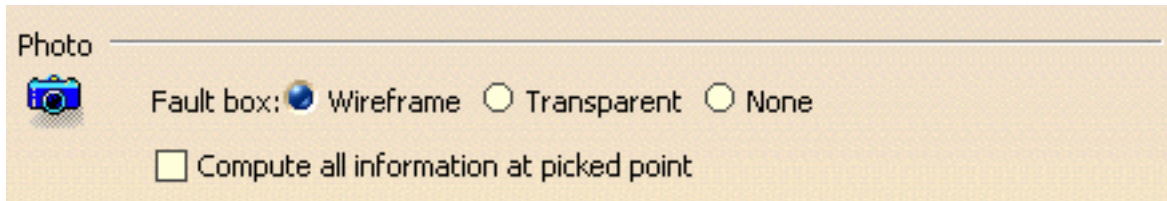


- Select the **Stop at tool change** checkbox if you want the Video simulation to stop each time a tool change is encountered in the program.
- Select the desired **Collisions detection** option to:
  - ignore collisions during the Video simulation
  - stop the Video simulation at the first collision
  - continue the Video simulation even when collisions are detected. In this case, you can consult the list of collisions at any time during the simulation.

Best performance is obtained when collisions are ignored.

- Select the **Touch is collision** checkbox if you want touch (or contact) type of collision to be detected.

## Photo

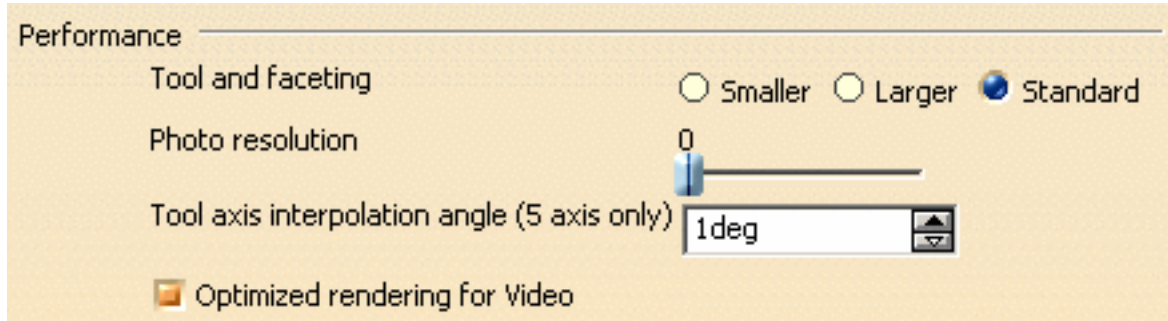


- Select the desired **Fault box** type for examining remaining material or gouges:
  - Transparent: to display a transparent bounding box
  - Wireframe: to display a wireframe bounding box
  - None: if no bounding box is required.

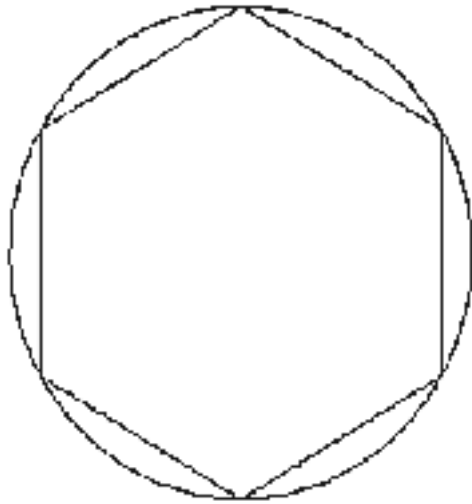
Best performance is obtained when no bounding box is required and the checkbox is not selected.

- Select the checkbox to compute all information at picked point.

## Performance

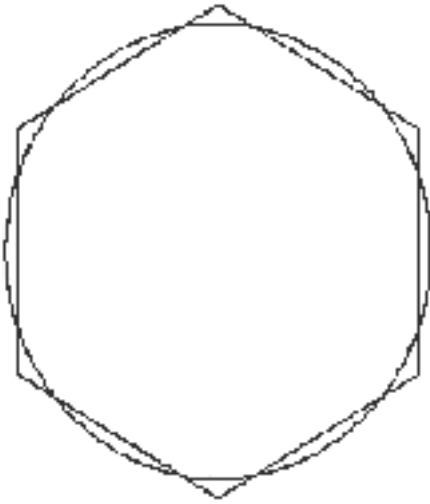


- There are three methods of tool faceting used in Video simulation: Standard, Smaller and Larger. The number of facets for a tool representation is determined by the chord deviation that is set for the tool diameter (0.005% of the tool diameter).
  - **Smaller:** The picture shows a rough approximation of a tool with six facets. Note that the chord deviation is always inside the actual circle, and that the points are always on the circle (accurate).



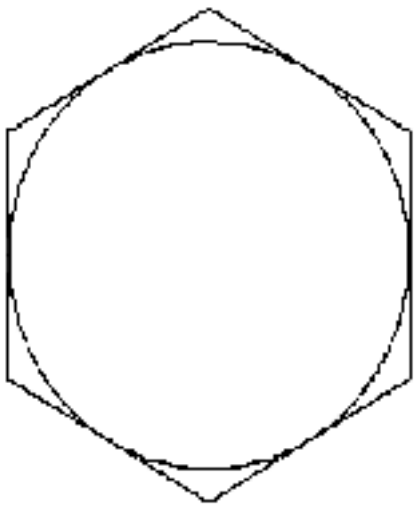
This is the most accurate method for the Arc through Three Points command.

- **Standard:** The picture shows a rough approximation of a tool with six facets. Note that the chord deviation is partly inside and partly outside the actual circle, and that the points are not always on the circle.



This is the best method for material removal simulation.  
However, this is not suitable for the Arc through Three Points command.

- **Larger:** The picture shows a rough approximation of a tool with six facets. Note that the chord deviation is outside the actual circle, and that the points are not on the circle.



This is not suitable for the Arc through Three Points command.  
However, it can be useful for gouge detection.

- Set the resolution for Photo simulation.  
Best performance is obtained when the resolution is set to 0. In this case, a detailed simulation of a portion of the part can be obtained using the **Closeup** command.  
Increasing the resolution improves machining accuracy and gives a very detailed simulation. However, this requires increased memory and computation time.
- Specify the maximum angle that the tool axis is allowed to vary between two consecutive points.  
Best performance is obtained for an angle of 10 degrees. Decreasing the angle improves the precision of the simulation. However, this requires increased memory and computation time.
- Set the **Optimized rendering for Video** checkbox to obtain an optimized rendering to improve Video simulation performance.  
Otherwise, more realistic colors are obtained with a slightly degraded performance.

## Color

Color

Tool and machined area:  Same  Last tool different  All different

Tool1

Tool holders

Parts

Fixtures

- Set the tool (and associated machined area) color to be the same as or different from the last tool, or have different colors for all tools. Best performance is obtained with same colored tools.
- Assign colors to the different tools using the associated color combo.
- Assign colors to tool holders, parts, and fixtures using the associated color combos.

## Positioning Move

Positioning Move

Maximum tool axis variation

- Set the **Maximum tool axis variation** that is to be allowed between the end point of an operation and the start point of the next operation. If the tool axis varies by an amount greater than the specified value, then the tool is positioned at the start of the following operation.

# Digital Mockup

DMU Navigator

DMU Space Analysis

DMU Fitting

DMU Immersive Review

DMU Optimizer

DMU Space Engineering

# Customizing for DMU Navigator

[Customizing General DMU Settings](#)  
[Customizing DMU Navigator Settings](#)  
[Customizing Product Visualization Settings](#)  
[Customizing Multi-Process Settings](#)  
[Customizing Cache Settings](#)  
[Customizing Visualization Settings](#)  
[Customizing 2D Workshop Visualization Settings](#)  
[Customizing Product Structure Settings](#)  
[Customizing Navigation Settings](#)  
[Customizing External Format Import](#)

# Customizing General DMU Settings



This task explains how to customize general DMU settings.

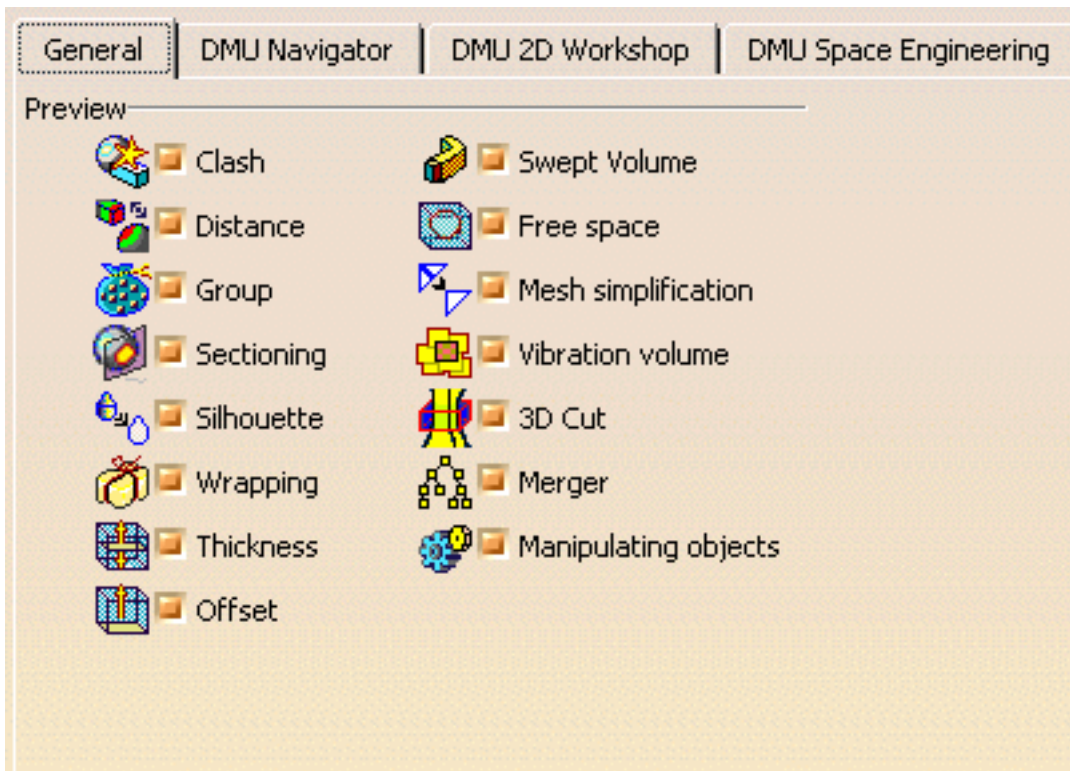


1. Select **Tools** -> **Options** from the menu bar:

The Options dialog box appears

2. Click the **Digital Mockup** category.

The **General** tab is active.



The General tab lets you customize:

- Automatic display of preview windows

3. Click preview checkboxes as appropriate to change the automatic display setting of preview windows when creating cameras, manipulating objects, etc.

By default, preview windows are automatically displayed.

4. Click **OK** in the dialog box when done.





# Customizing DMU Navigator Settings



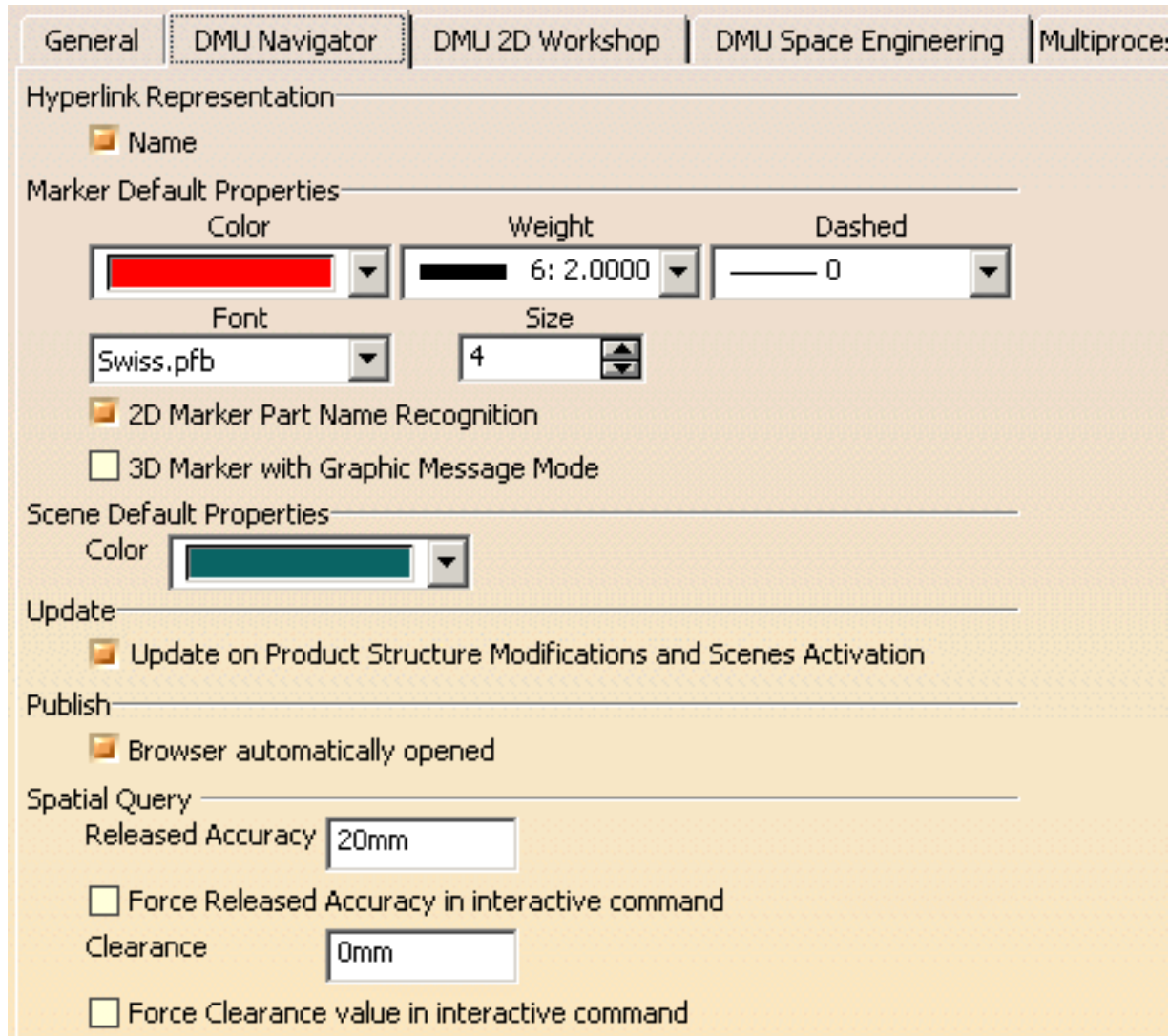
This task explains how to customize DMU Navigator settings.

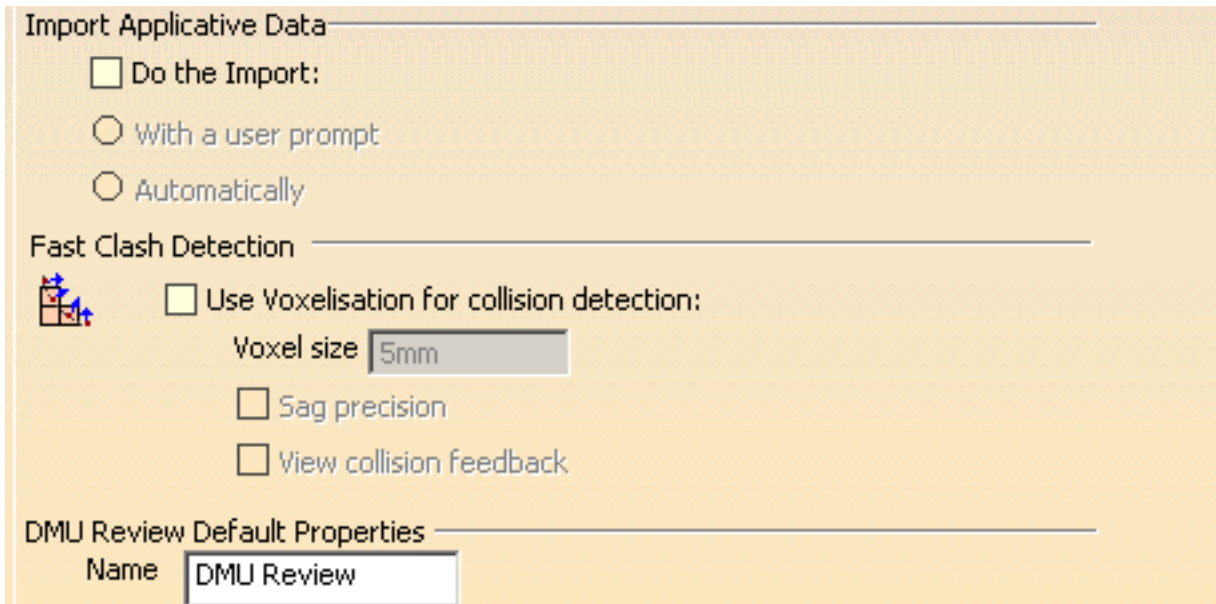


1. In the menu bar, select **Tools** -> **Options**.

The Options dialog box appears.

2. Click the **Digital Mockup** category, then click the **DMU Navigator** tab.





The screenshot shows a dialog box with three main sections:

- Import Applicative Data:** A checkbox labeled "Do the Import:" is unchecked. Below it are two radio buttons: "With a user prompt" (selected) and "Automatically".
- Fast Clash Detection:** A small icon of a 3D model is on the left. To its right is a checkbox labeled "Use Voxelisation for collision detection:" which is unchecked. Below this checkbox is a text input field for "Voxel size" containing the value "5mm". Further down are two more unchecked checkboxes: "Sag precision" and "View collision feedback".
- DMU Review Default Properties:** A text input field labeled "Name" contains the text "DMU Review".

The **DMU Navigator** tab enables you to customize:

- Hyperlink representation (name)
- Marker Default Properties
- Scene Default Properties
- 3D annotation update (when moving the product)
- Spatial Query parameters
- Import of applicative data



You can add hyperlinks to your document and then use them to jump to a variety of locations, for example, to a marketing presentation, a Microsoft Excel spreadsheet or a HTML page on the intranet.

3. In the **Hyperlink Representation** area, check the **Name** selection box.

By default, hyperlink names are displayed : all hyperlink will be textual. The name you give the link in the Manage Hyperlink dialog box when you create it will appear when using go to hyperlink.

4. Click preview checkboxes as appropriate to change the automatic display setting of preview windows when creating cameras, manipulating objects, etc.

By default, preview windows are automatically displayed.

5. Change the color, font type and font size of 2D markers.

Note: Defining the Marker default properties in the **Tools->Options->DMU Navigator** sets the selected properties as default properties and changes how new annotations will look when you create them.

6. Click the **2D Marker Product Name Recognition** checkbox in order to activate the option to automatically use a Part's Name as the default for the creation of text annotations.

7. Click the **3D Marker with Graphic Message Mode** checkbox in order to activate the mechanism that enables you to transform temporary markers into persistent 3D annotations.

8. Change the Scene background color as desired.

9. Uncheck the **Update on Product Structure Modifications and Scenes Activation** option if necessary. This option manages a publish/subscribe mechanism which will manage the automatic updates of certain events:

- move
- shape activation
- insert
- replace product
- delete product
- UI-activation modification
- graphic attributes modification
- Enhanced Scene activation

If the option is unchecked, some applicative data will not be automatically updated upon the above events. However, if the option is unchecked, performance will be faster.

10. To activate the automatic launching of Publish results in a browser upon the stopping of the Publish process, in the **Publish** area, check the **Browser automatically opened** checkbox .

## 11. Spatial Query Released Accuracy

The default value for released accuracy is 20mm.  
This value can also be defined by the administrator.

How do you, as administrator, define the released accuracy value?

- you **set** a precise and required value in the **Released Accuracy** field.
- you then **run** the CATDMUtility batch process without defining the `-vox` option, the released accuracy value is taken into account. The data is pre-tessellated.
- you **check** the **Force Released Accuracy in interactive command** checkbox if you wish to force users of the **Spatial Query** command to use the defined **Released Accuracy**.

## 12. Spatial Query Clearance

The default value for clearance is 0mm.  
This value can also be defined by the administrator.

How do you, as administrator, define the clearance value?

- you **set** a precise and required value in the **Clearance** field.
- you **check** the **Force Clearance in interactive command** checkbox if you wish to force users of the Spatial Query command to use the defined Clearance value.

## 13. Import Applicative Data

- Do the Import:
  - activated: the applicative data will be inserted
  - not activated: the applicative data will not be inserted
- with user prompt: a dialog box will appear enabling you to select any subset of applicative data to be inserted
- automatically: all applicative data will be inserted, no dialog box will appear
  - activated: the applicative data will be inserted

## 14. Fast Clash Detection

- If you select a voxel size and check the View Collision Feedback button, then you obtain a clash detection with a voxel precision of the size selected, with the clashing parts highlighted, and the center of the voxel points appearing in red.

- If the View Collision Feedback button is unchecked, then you will obtain a clash detection with only the clashing parts highlighted.
- If you select sag precision and check the View Collision Feedback button, then you obtain a clash detection with the precision of the sag. The clashing parts are highlighted and the intersection curve of the clashing parts appears in red.
- If the View Collision Feedback button is unchecked, then you obtain clash detection with with the clashing parts highlighted.



### 15. DMU Review Default Properties

In the **Name** text-entry field, enter the desired default name for DMU Reviews.

16. Note that you need to exit and restart to see the effects of customization, so click **OK** to confirm.



# Customizing Product Visualization Settings



This task explains how to customize Product Visualization settings.

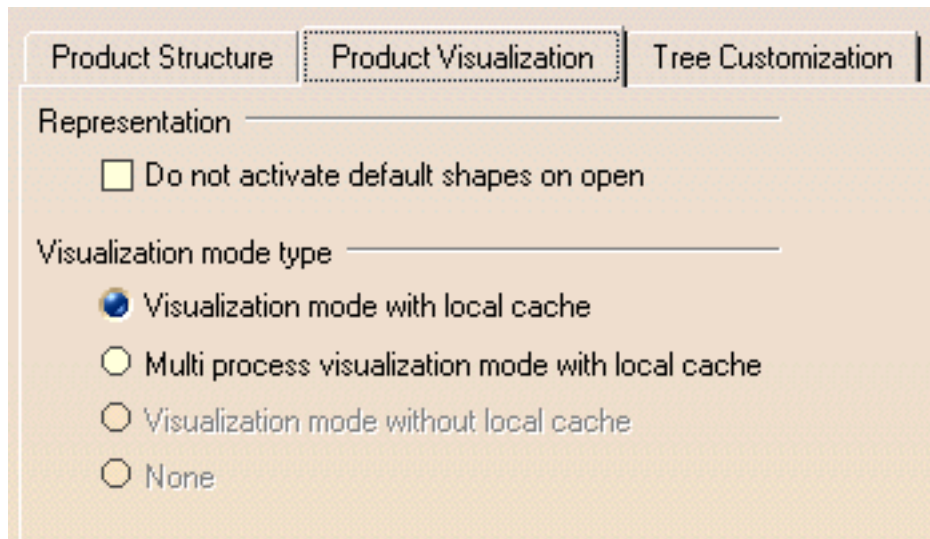


1. Select **Tools** -> **Options** from the menu bar:

The Options dialog box appears

2. Click the **Infrastructure** -> **Product Structure** category.

3. Click the **Product Visualization** tab.



The **Product Visualization** tab enables you to customize:

- Default shape activation on open
- Visualization mode type

4. In the **Representation** area, check the **Do not activate default shapes on open** selection box.

5. In the Visualization mode type area, check one of the following visualization mode types:

- Visualization mode with local cache
- Multi-process visualization mode with local cache

Note that the last two visualization mode types are currently grayed out and are not available.

By default, preview windows are automatically displayed.



# Customizing Multi-Process Settings



This task explains how to customize Multi-Process settings. These multi-processes concern the CATDMUUtility processes that are created as child processes for the tessellation of documents when performing a **File -> Open** of a CATProduct or when running the CATDMUUtility batch process using the **-mp** option. This new feature enables the management of possible coredumps and / or looping problems during tessellation such that the impact is localized to the specific document that cannot be tessellated; the state of a V5 session will no longer be impacted by these problems and a CATDMUUtility batch process will run to conclusion. A log file indicates which documents could not be tessellated.

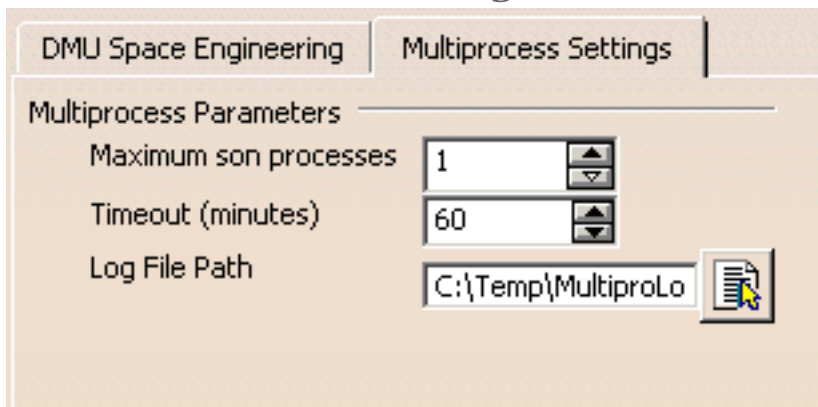


1. Select **Tools -> Options** from the menu bar.

The Options dialog box appears.

2. Click the **Digital Mockup** category.

3. Click the **Multi-Process Settings** tab.



The **Multi-Process Settings** tab enables you to customize:

The maximum number of child processes that will be launched.

The maximum amount of time a process will be allowed to loop before it is killed.

The path of the log file that will report the success or non-success of each tessellation concerned with the opening of a CATProduct.




The number of child processes is limited to 1.

4. In the **Maximum son processes** text-entry field, enter the maximum number of child CATDMUUtility processes to be used during tessellation.

5. In the **Timeout (minutes)** text-entry field, enter the maximum time that a single tessellation process will be allowed to loop before being aborted.

6. In the **Log File Path** text-entry field, enter the path of the log file of the tessellation process or

click the Browse icon  to choose a path via the File Browser.



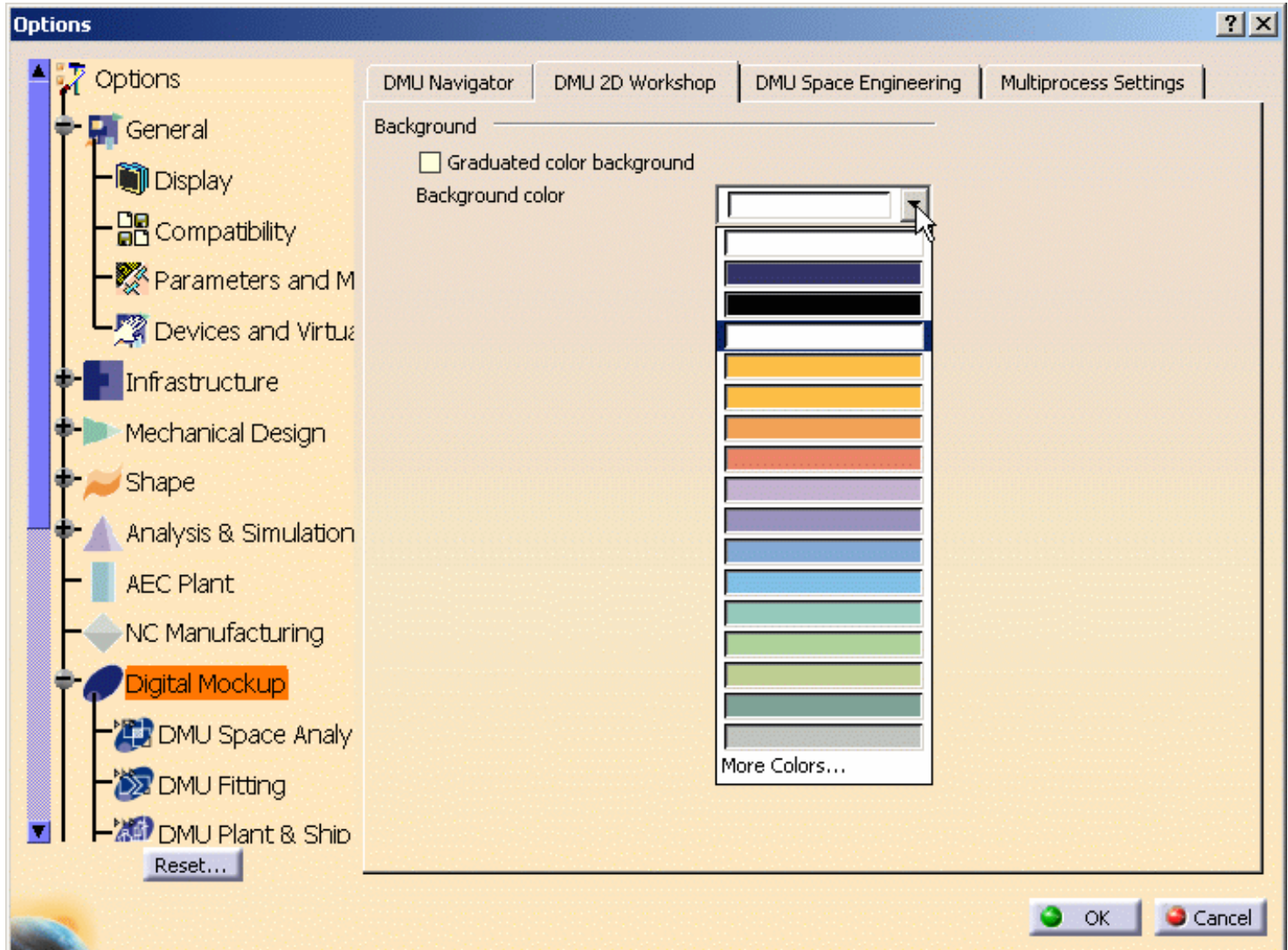


# Customizing 2D Workshop Visualization Settings

This task explains how to customize the background color of your 2D Workshop session.

1. Select **Tools** -> **Options** from the menu bar.

The Options dialog box appears.



2. Select the **Digital Mockup** category.

3. Click the **2D Workshop** tab.

4. Modify the Background color to the color of your choice:

Click on the **Background color** bar .

Enter the different values for Red, Blue, Green or click on the combo-box selection button and choose the desired standard color from the proposed list.

- Click on the **Background color** bar .
- Enter the different values for Red, Blue, Green or click on the combo-box selection button and choose the desired standard color from the proposed list.

5. If you wish the background color to be graduated, click the **Graduated color background** selection box.





# Customizing for DMU Space Analysis

[DMU Clash](#)

[DMU Clash - Detailed Computation](#)

[DMU Clash - Penetration](#)

[DMU Clash - Rule](#)

[DMU Clash - Process](#)

[DMU Sectioning](#)

[DMU Distance](#)

[Measure Tools](#)

# DMU Clash



This task explains how to customize the clash settings of the **Clash** command.

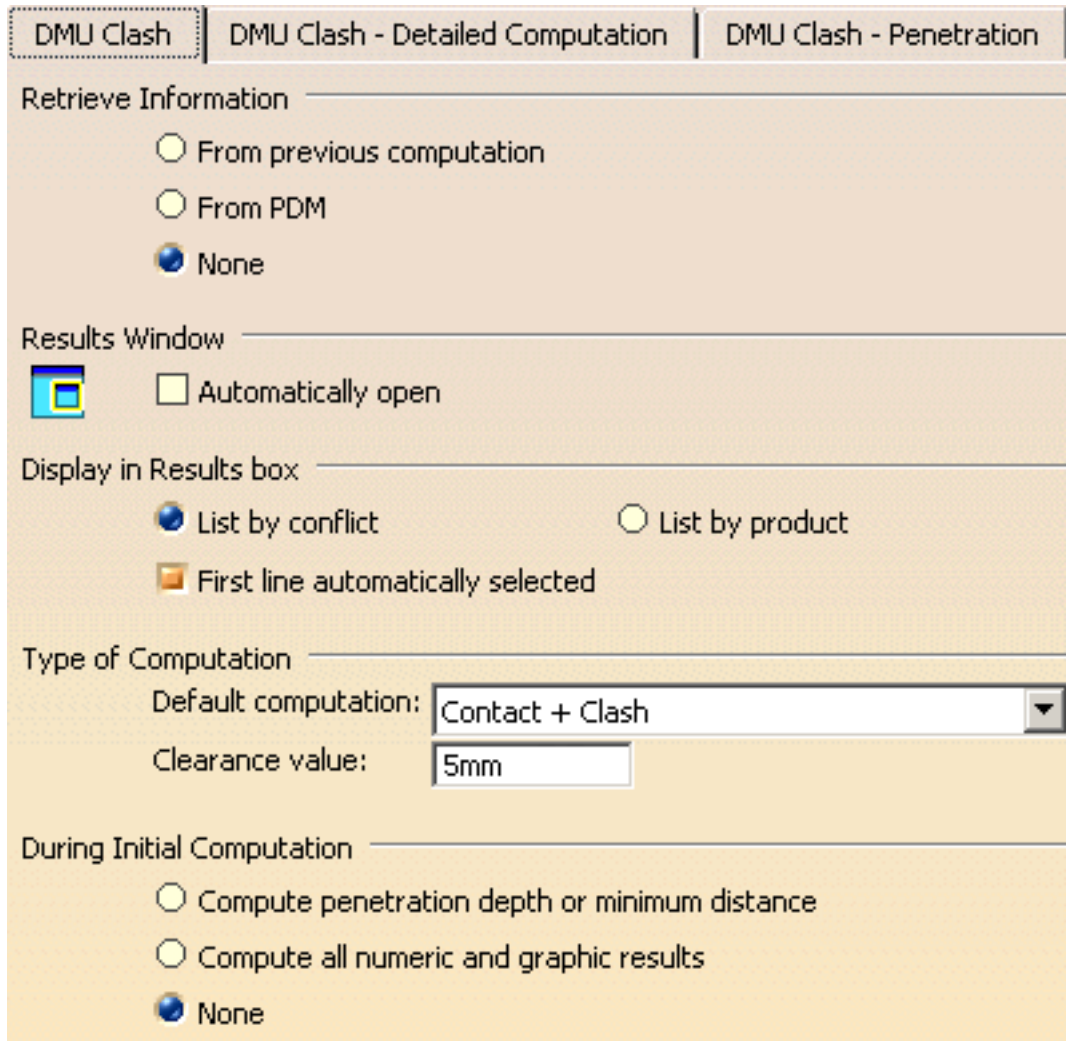


1. Select **Tools** -> **Options** from the menu bar:

The Options dialog box appears.

2. Click **Digital Mockup category**, then the **DMU Space Analysis** subcategory.

3. Click the **DMU Clash** tab.



The DMU Clash tab lets you customize the following clash command settings:

**Retrieve Information:** retrieves clash results for comparison purposes:

- From previous computation

- From PDM: from both Enovia V5 and Enovia VPM.  
Note: This setting cannot be combined with the authorized penetration interference type.
- No comparison (default setting)

**Results Window / Automatically open:** automatically displays the results window when you run a clash command computation.

**Display in Results box:** three options to set the default display in the results box of the Check Clash dialog box:

- List by conflict tab (default setting).
- List by product tab.
- First line automatically selected: automatically selects the first line in the List by Conflict or List by Product tab (default setting).

**Type of Computation:** sets the default computation type and, if necessary, the default clearance value.

**During Initial Computation:** specifies what the system computes in addition to detecting and identifying the different types of interference.

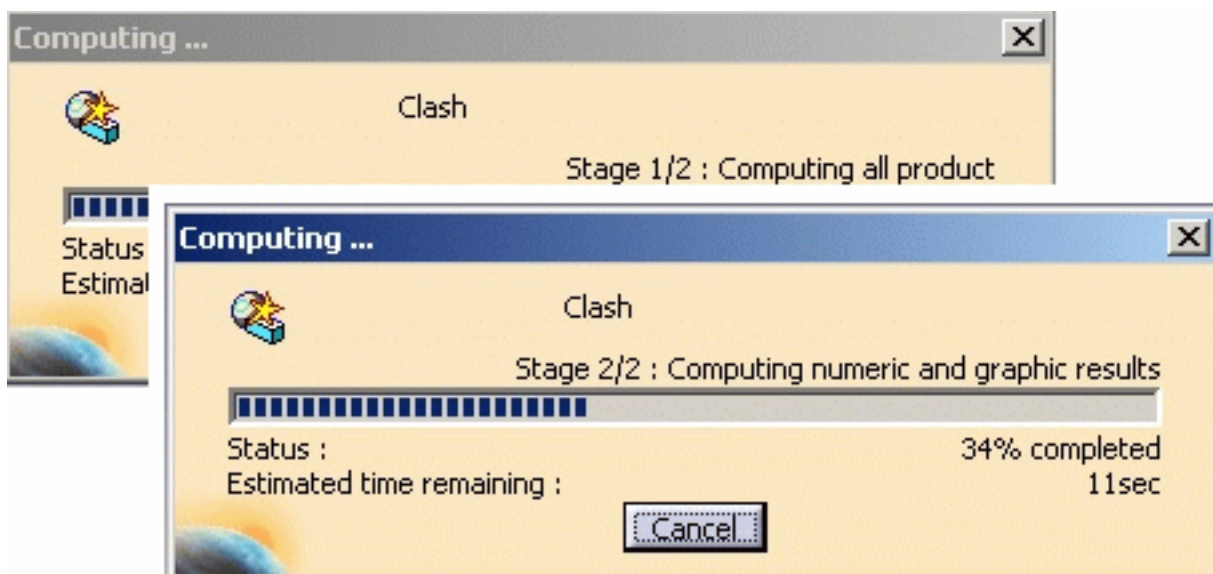
- Compute penetration depth or minimum distance: if selected, automatically computes and displays the penetration depth and minimum distance for all interferences detected.
- Compute all numeric and graphic results: if selected, automatically computes and displays numeric results as well as graphic results as defined in the [Detailed Computation](#) tab.

Note: Computation time can be long.

In this case, the progress bar shows the status of the calculation in two different steps:

Stage 1/2: determines the number of interferences

Stage 2/2: computes appropriate numeric and graphic results.



- None: if selected, the initial computation detects and identifies the different types of interference only.
4. Click **OK** in the dialog box when done



# DMU Clash - Detailed Computation



This task explains how to customize the detailed computation settings of the **Clash** command.

The detailed computation mode defines what is computed when you select a conflict or product in the Check Clash Results box. It can be run at element or product level.

By default, the detailed computation is run at element level. The graphics representation of interferences (triangles for contacts and clearances, curves for clashes) and the penetration depth or minimum distance are computed.

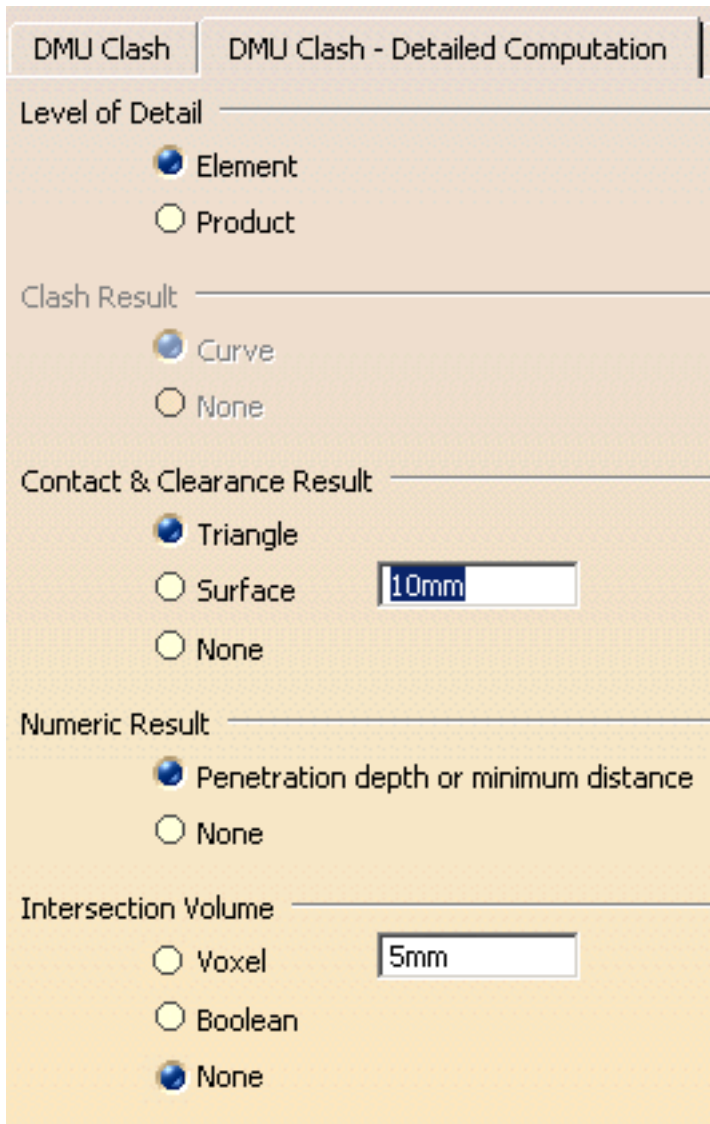


1. Select **Tools** -> **Options** from the menu bar:

The Options dialog box appears.

2. Click **Digital Mockup** -> **DMU Space Analysis** in the left-hand box.

3. Click the **DMU Clash - Detailed Computation** tab.



The DMU Clash - Detailed Computation tab lets you customize the following settings:

**Level of Detail:** sets the level of detail.

- **Element:** lets you work globally at product level while allowing you to pinpoint the elements involved.
- **Product:** gives information at product level only.

Note: Element analysis in Detailed Results and Visualization boxes is only available if the level of detail is set to Element.

**Clash Result:** specifies the graphics representation for clashes.

- **Curve:** Red intersection curves identifying clashing products are computed and displayed.
- **None:** No intersection curves are computed.

**Contact & Clearance Result:** specifies the graphics representation for contacts and clearances.

- **Triangles:** Yellow and green triangles identifying products in contact and products separated by less than the specified clearance distance respectively are computed and displayed.

Note: The graphics display and storage of the triangular representation is costly and may adversely affect performance.

- **Surface:** Yellow and green surfaces identifying products in contact and products separated by less than the specified clearance distance respectively are computed and displayed.

If you select this option, you must set an accuracy. The value entered defines the maximum value for the length of the longest side of a triangular representation of the results. This representation is used to obtain the yellow and green surfaces.

- **None:** No triangles or surfaces are computed.

**Numeric Result:** specifies the numeric result computed and displayed.

- **Penetration depth or minimum distance:** The penetration depth of clashes and the minimum distance for clearances is computed and displayed.
- **None:** No penetration depth or minimum distance is computed.

**Intersection Volume:** specifies a graphics representation for clashes. This representation can be requested in addition to that selected under Clash Result.

**Important:** The intersection volume is only visible in the Results window.

- **Voxel:** Voxels (cubes) identifying clashing products are computed and displayed.

If you check this option, you must set the voxel size. The value entered is the edge-length of the voxel or cube. A lower value will give a more accurate result but will also require a longer computation time and more memory. The minimum value is 1mm (0.03937 inch).

- **Boolean:** Boolean solids identifying clashing products are computed and displayed.
- **None:** No intersection volume is computed.


Notes:

- The intersection volume is also given in the Detailed Results box of the Check Clash dialog box.
- The intersection volume is not stored in the model and cannot be saved in ENOVIA LCA or ENOVIA VPM.

4. Click **OK** in the dialog box when done.



# DMU Clash - Penetration

 This task explains how to customize the penetration mode setting of the **Clash** command.

The penetration mode defines how penetration depth is computed. Two modes are available:

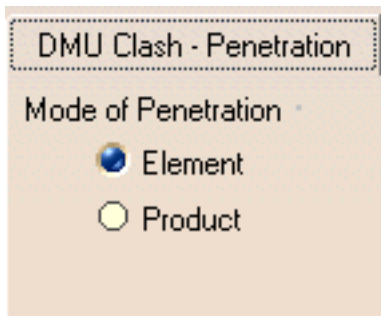
- Element (default mode)
- Product.

 1. Select **Tools** -> **Options** from the menu bar:

The Options dialog box appears.

2. Click **Digital Mockup** -> **DMU Space Analysis** in the left-hand box.

3. Click the **DMU Clash - Penetration** tab.



The DMU Clash - Penetration tab lets you set the penetration mode:

**Mode of Penetration: Element**

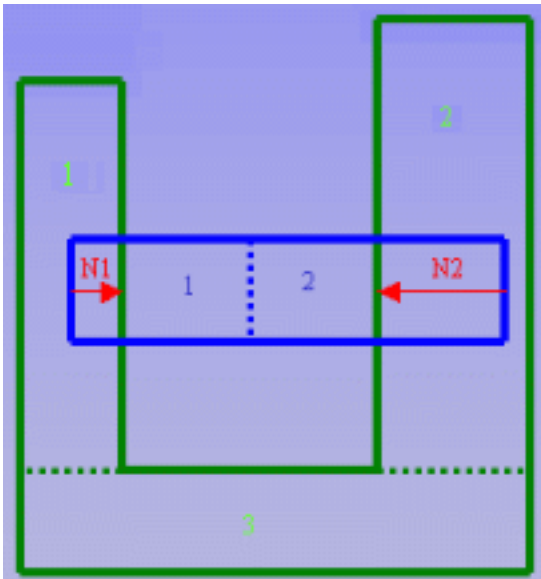
This is the default mode and lets you assess the seriousness of a clash.

In the Element mode, penetration depth corresponds to the depth of the intersection area visualized by the penetration vector. The maximum value is output, in our example, N2.

**Note:** You cannot use this calculation to translate products and avoid clashes.

In our example, the green product comprises



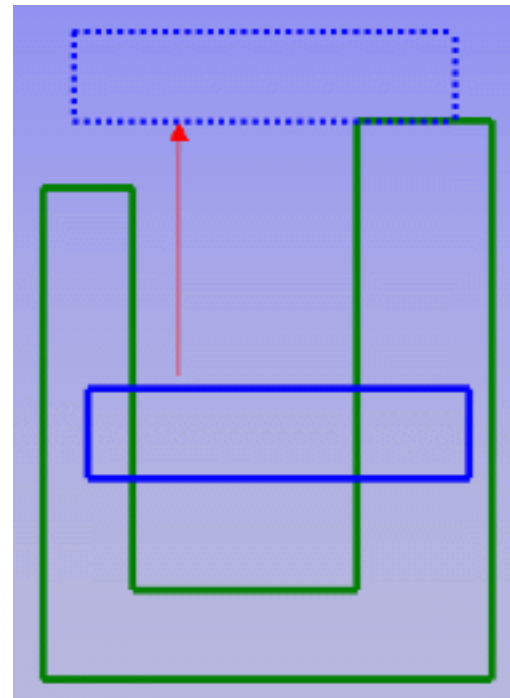


three elements and the blue product two elements.

**Mode of Penetration: Product**

In the Product mode, penetration depth (visualized by a vector) is the minimum distance by which it is necessary to translate a product to avoid a clash.

This mode corresponds to the penetration depth computation prior to Version 5 Release 8.



4. Click **OK** in the dialog box when done



# DMU Clash - Rule



This task explains how to customize clash rule settings.

Clash rules written using knowledgeware capabilities can be used in the **Clash** command.

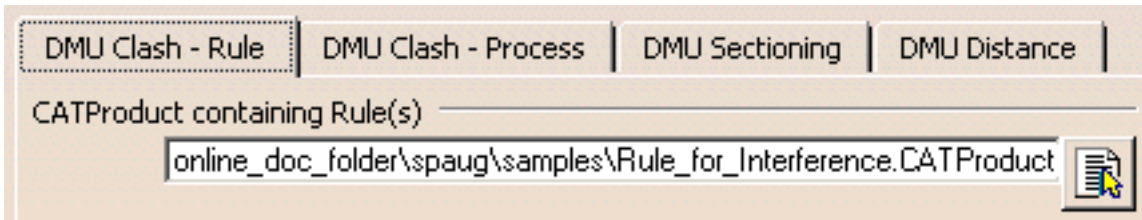


1. Select **Tools -> Options** from the menu bar:

The Options dialog box appears.

2. Click **Digital Mockup -> DMU Space Analysis** in the left-hand box.

3. Click the **DMU Clash - Rule** tab.



The DMU Clash -Rule tab lets you identify the full path to the CATProduct containing knowledgeware clash rules.

4. Click **OK** in the dialog box when done



# DMU Clash - Process



This task explains how to customize process settings of the **Clash** command.

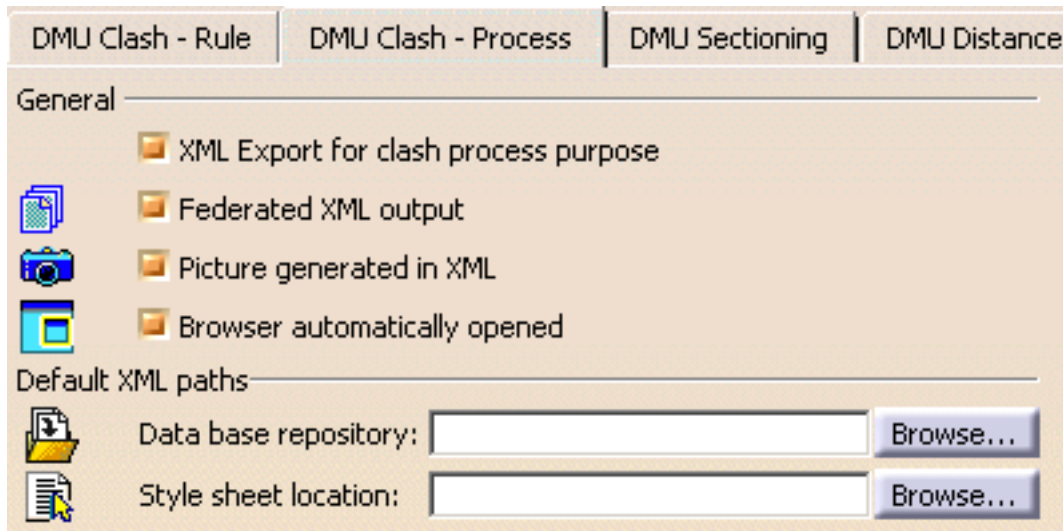


1. Select **Tools** -> **Options** from the menu bar:

The Options dialog box appears.

2. Click **Digital Mockup** -> **DMU Space Analysis** in the left-hand box.

3. Click the **DMU Clash - Process** tab.



The DMU Clash - Process tab lets you customize XML settings:

- **XML export for class process purpose:** exports clash results and geometry to XML files as part of a standalone clash process.
- **Federated XML output:** generates one file per conflict plus one header file summarizing results. If you de-activate this option, a single large XML file containing all conflicts is generated.
- **Picture generated in XML:** generates pictures in png format in a picture folder. This option has an obvious impact on performance.
- **Browser automatically opened:** automatically opens a browser displaying results when clash results are exported.
- **Database repository:** sets the default location in which to save the XML files.
- **Style sheet location:** lets you select and apply your customized style sheet.

4. Click **OK** in the dialog box when done



# DMU Sectioning



This task explains how to customize sectioning settings.



1. Select **Tools** -> **Options** from the menu bar:

The Options dialog box appears

2. Click **Digital Mockup** -> **DMU Space Analysis** in the left-hand box

3. Click the **DMU Sectioning** tab

DMU Clash - Rule | DMU Clash - Process | **DMU Sectioning**

**Section Planes**

Default color:

Normal:  X  Y  Z

Origin:  0,0,0  Selection

Hide the plane

Hide the section results in the No Show space

Automatic computation of the result

Wireframe elements cut

**Section Grid**

| Mode                                      | Style                                  | Steps                                                       |
|-------------------------------------------|----------------------------------------|-------------------------------------------------------------|
| <input checked="" type="radio"/> Absolute | <input checked="" type="radio"/> Lines | <input type="checkbox"/> Automatic filtering                |
| <input type="radio"/> Relative            | <input type="radio"/> Crosses          | Width: <input type="text" value="100.000mm"/>               |
|                                           |                                        | Height: <input type="text" value="100.000mm"/>              |
|                                           |                                        | <input checked="" type="checkbox"/> Automatic grid resizing |

**Results Window**

Automatically open

Always 2D view

Automatically reframe

Section fill

The DMU Sectioning tab lets you customize:

**Section Planes:** sets default section plane settings.

- **Default color:** sets the default color of the section plane.
- **Normal X,Y,Z:** specifies the absolute axis along which you want to orient the normal vector of the section plane (master plane in the case of slices and boxes).
- **Origin:** locates the center of the plane at absolute coordinates 0,0,0 or at the center of the bounding sphere around the products in the selection you defined.
- **Hide the plane:** hides the plane on exiting the command. De-activate this option to keep the plane in the Show space.
- **Hide the section results in the No Show space:** transfers the section results to the No Show place on exiting the command.
- **Automatic computation of the result:** automatically updates sectioning results while manipulating the plane. De-activating this option means that results are computed when you release the mouse button.



- **Wireframe elements cut:** the section plane sections any wireframe elements present. Points represent the intersection of the plane with wireframe elements. De-activating this option means that wireframe elements are not taken into account.

**Section Grid:** sets default grid options.

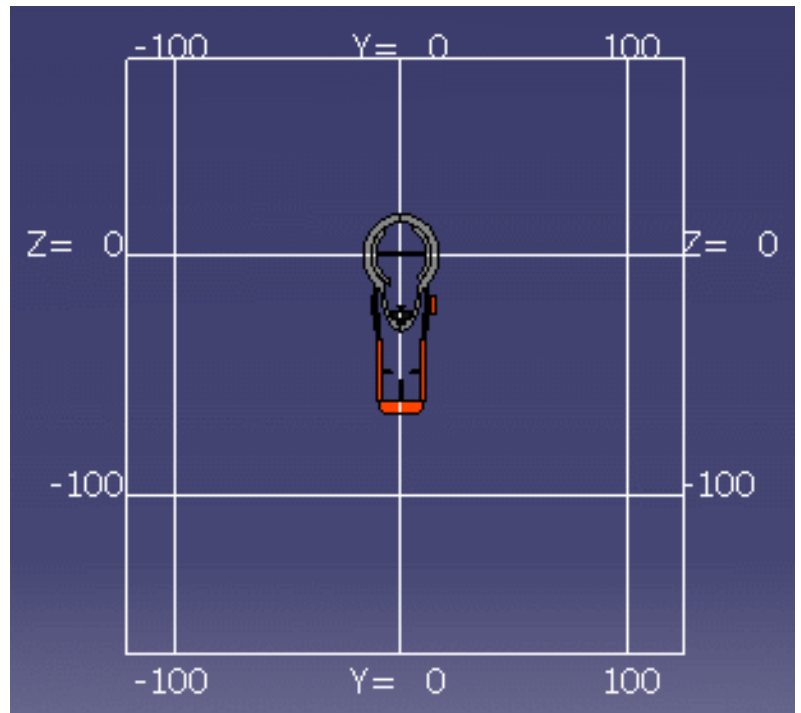
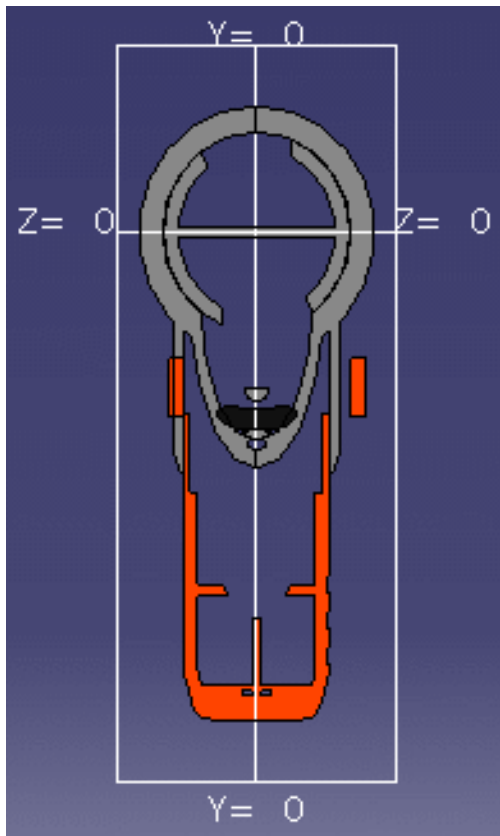
- **Absolute mode:** sets grid coordinates with respect to the absolute axis system of the document.
- **Relative mode:** places the center of the grid on the center of the section plane (master plane in the case of slices and boxes).
- **Style:** sets the grid representation to lines or crosses.
- **Automatic filtering:** if clicked, automatically adjusts the level of detail of the grid display when you zoom in and out.
- **Steps:** specifies the spacing between grid lines. The default value is 100. Units are current units set using **Tools -> Options**.



- **Automatic grid resizing:** automatically re-sizes the grid to section results when moving the section plane. De-activating this option means that the grid has the same dimensions as the section plane.

Automatic grid resizing ON:

Automatic grid resizing OFF:



**Results Window:** sets default Section viewer settings.

- Automatically open: always displays the Section viewer when in the Sectioning command.  
 Note: If cleared, the preview window is displayed.  
 If, in addition, you clear the Sectioning preview option in the Digital Mockup General tab (**Tools -> Options -> Digital Mockup -> General**), then no viewers are displayed.
- Always 2D view: always displays the 2D view in the Section viewer.
- Automatically reframe: automatically fits the results into the available space in both the Section viewer and preview window when manipulating the section plane in the document window.
- Section fill: fills in the section to make a surface for measurement and display.

4. Click **OK** in the dialog box when done



# DMU Distance



This task explains how to customize results window and publish settings of the **Distance and Band Analysis** command.

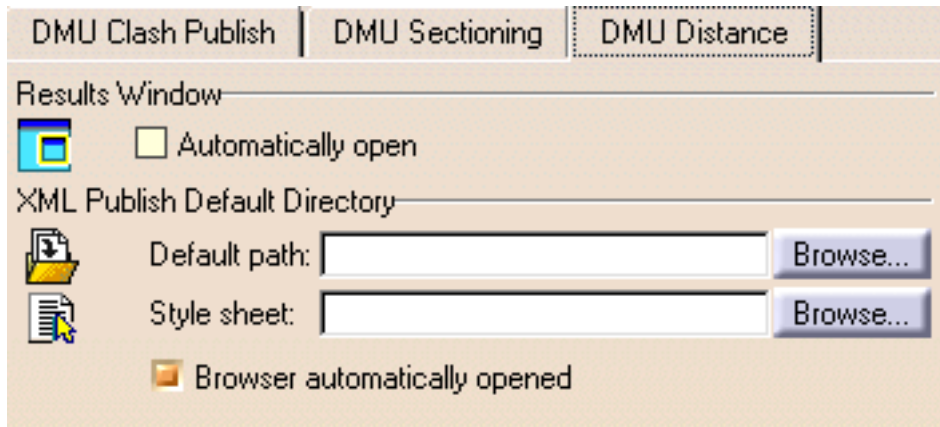


1. Select **Tools** -> **Options** from the menu bar:

The Options dialog box appears.

2. Click **Digital Mockup** -> **DMU Space Analysis** in the left-hand box.

3. Click the **DMU Distance** tab.



The DMU Distance tab lets you customize Distance and Band Analysis command settings.

**Results Window / Automatically open:** automatically displays the results window when you run a distance command computation.

**XML Publish Default Directory** (Band analysis only):

- **Default path:** sets the default location in which to save the XML file.
- **Style sheet:** lets you select and apply your customized style sheet.
- **Browser automatically opened:** automatically opens a browser displaying results when distance results are exported.

4. Click **OK** in the dialog box when done



# Customizing for DMU Fitting Simulator

[DMU Fitting Settings](#)  
[DMU Manipulation Settings](#)



# Customizing DMU Fitting Settings



This task explains how to customize DMU Fitting settings.



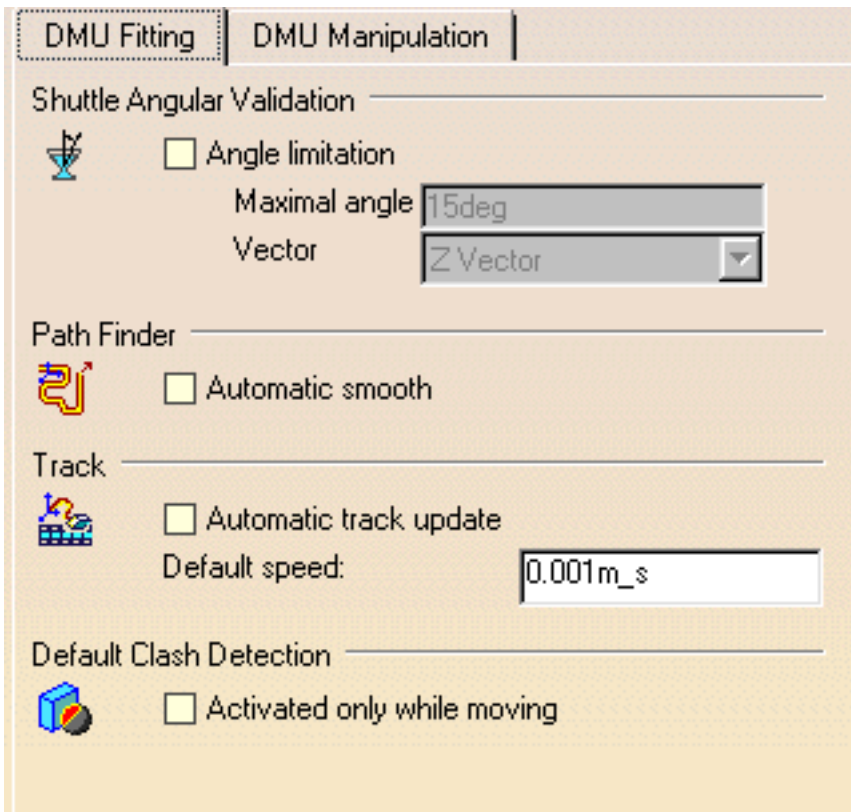
1. Select **Tools** -> **Options** from the menu bar:

The Options dialog box appears.

2. Click Digital Mockup in the left-hand frame, then DMU Fitting.



The DMU Fitting tab is active by default. If it is not on top, select it. The default values are shown below.



## Shuttle Angular Validation

Checking the box enables you set the angular shuttle validation option in the Edit Shuttle dialog box.

You can specify a maximum rotation angle for the shuttle and the vector around which the shuttle will rotate.

## Path Finder

**Automatic Smooth:** If checked, smooth is performed within the pathfinder command once you click the OK button.

## Track

**Automatic track update:** if checked, the track is automatically updated after modification. The manual update is set by default (see the track contextual menu).

Default speed is 0.001m\_s. You can modify this value at any time. When you modify the default speed, the modification applies to tracks created after the modification. To modify the speed on existing tracks, use the Properties dialog box.

## Default Clash Detection

When activating the default clash detection, the clash computation can take several minutes before the simulation dialog boxes are displayed.

Sometimes it is more convenient not to compute this clash detection for the first step.

This step can already be validated by a static clash analysis.

- **Activated only while moving:** if checked, no calculation will be performed when the default clash detection is **activated** (ON, STOP). When running tracks, sequencesm, and simulations, clashes are still calculated with respect to the default clash detection state (OFF, ON, STOP)
  - By default this option is not checked
3. Click **OK** in the dialog box when done.



# Customizing DMU Manipulation Settings



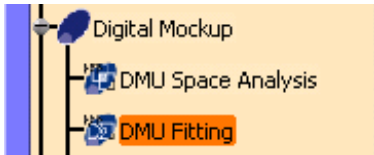
This task explains how to customize DMU Fitting settings.



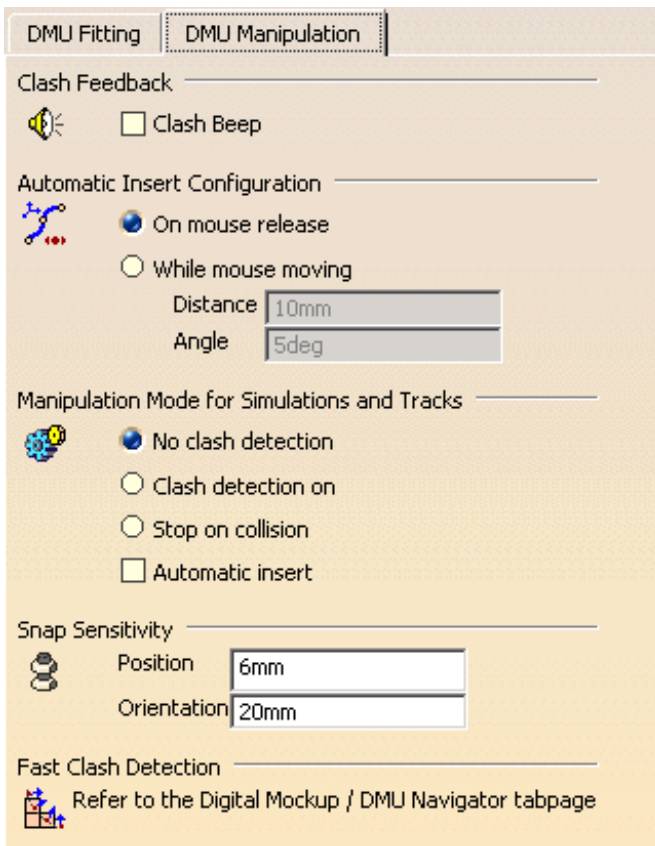
1. Select **Tools -> Options** from the menu bar:

The Options dialog box appears.

2. Click Digital Mockup in the left-hand frame, then DMU Fitting.



3. Select the DMU Manipulation tab. The default values are shown below.



## Clash Feedback

Click the Clash Beep checkbox to activate the Beep during interference analyses, simulation recordings, etc.

By default, the Clash Beep option is unchecked.

## Automatic Insert Configurations

- On mouse release (default): Lets you record positions on mouse release.
- While mouse moving:
  - Distance: Specifies the minimum distance step during simulation recording .
  - Angle: Specifies the maximum rotation angle step during simulation recording.

The Distance and Angle fields are only grayed out when **On mouse release** is checked. Once **While mouse moving** is checked, you can enter the appropriate values to record the the motion.






Regardless of which configuration you have selected, you will not get Automatic Insert unless the Automatic Insert option is checked in Manipulation Mode, described in the section below.

## Manipulation Mode for Simulations and Tracks

Lets you validate the shuttle simulation detecting clashes.

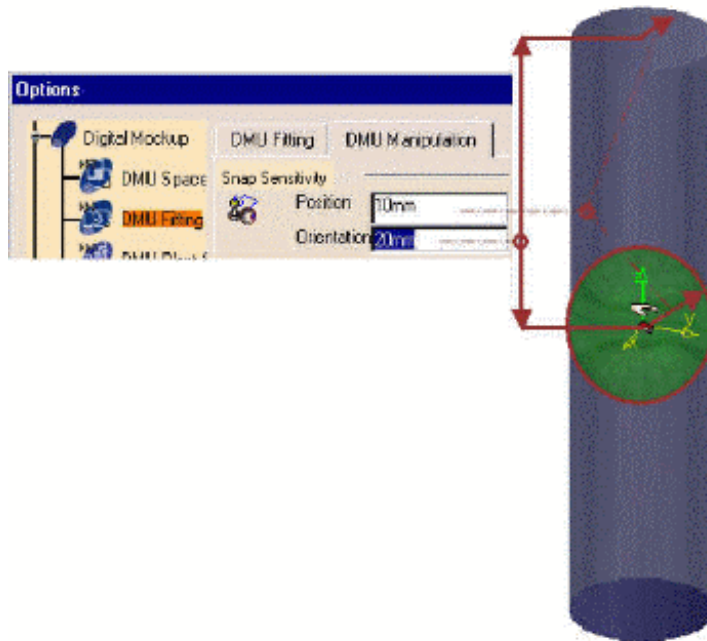
This setting is linked to the Automatic Clash Detection toolbar. The icon changes automatically with respect to the option checked.

- No clash detection (default) : Clash detection will not occur.
- Clash detection on : Activates the clash detection mode (The objects in clash are highlighted in the geometry area while recording the simulation )
- Stop on collision : Activates the stop mode. Consequently, the simulation stops once any objects clash. The objects in clash are highlighted in the geometry area.
- Automatic insert: When checked, it enables automatic insert mode. Automatic insert mode enable users to insert shots in a track without using the Record (Insert) command on the Recorder toolbar. The two configurations available for Automatic Insert appear above.

## Snap Sensitivity

Lets you customize the snap functionality parameters within a sphere and a cylinder:

- Position value: Specifies the sphere radius (first area) where the shuttle is snapped onto the object.
- Orientation value: Specifies the cylinder height (second area) where the shuttle is snapped onto the object.



3. Click **OK** in the dialog box when done.



# Customizing for DMU Immersive Review

Customizing DMU Immersive Review  
Customizing 3D Joystick Buttons

# Customizing DMU Immersive Review Settings



This task will show you how to customize DMU Immersive Review Settings



1. Select **Tools->Options** from the menu bar.

The **Options** dialog box appears

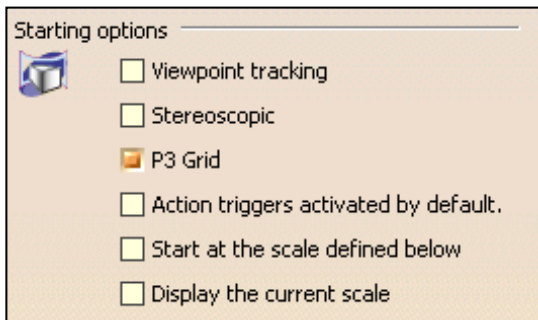
2. Expand the **Digital Mockup** category from the upper-left tree

3. Click the **DMU Immersive** item:

The DMU Immersive Review tab page appears containing three categories of options:

- [Starting Options](#)
- [Scale of the review](#)
- [Default toolbar position](#)

## Starting Options



### Viewpoint tracking

If you select this check box, the viewpoint changes as your head turns if you use a head tracker device. This option is linked to Stereoscopic option which enables or disables 3D visualization of your session. You need to launch the device driver. Please refer to [Customizing Devices Settings](#)

### Stereoscopic

Enables or disables stereoscopic visualization of graphical data with a perception of realistic, three-dimensional images.

### P3 grid

Enables to display the P3 grid when you enter the DMU Immersive Workbench

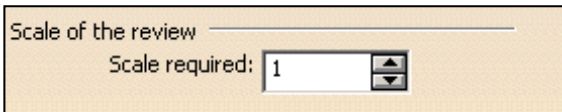
### Action triggers activated by default

If you select this check box a "demo mode" is launched which means each time you select an object which has been assigned an action trigger, the operation is automatically launched (the CATScript macro is triggered). In this case, only the standard toolbar is available, the action triggers command is activated.

If you clear this check box, you need to click the **Action Triggers** icon first and then select the object. All immersive review toolbars are available.

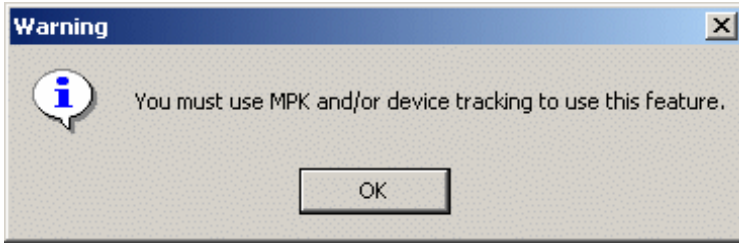
## Scale of the Review





To get the most of this capability, make sure either the Stereoscopic check box is selected (MPK compliant) or both Viewpoint Tracking and Stereoscopic check boxes are selected.

**Note:** when selecting the Start at the scale defined below check box, the following warning message is displayed:



**Start at the scale defined below**

activates the scale command when entering Immersive Review workbench, it is selected by default.

**Display the current scale**

activates the display of the current scale factor (at the bottom left in the review window)

**Scale of the review:**

allows the user to define a specific scale factor (i.e the required scale, applied by the Scale command)

**Default toolbar position**



**Default toolbar position**

You can choose the toolbars location. The default location is at the bottom left (lower left option), but is can be useful to turn on the upper left option depending on the display configuration you need (one or more screens)

4. Click **OK** in the dialog box when done.



# Customizing 3D Joystick Buttons



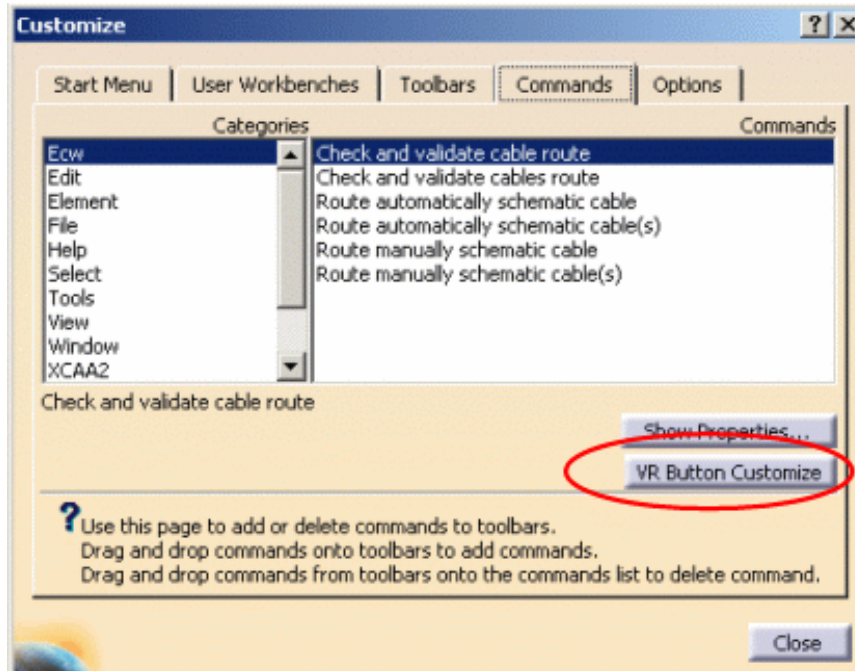
This task explains how to customize 3D Joystick buttons which allows you to assign (i.e. map) any given command or macro a device button.

This functionality is available for any application-supported hardware providing the necessary buttons, e.g. Space Mouse and the Space ball range of 3D input devices.

Please also read [Customizing Buttons](#) in the *Infrastructure User's Guide*



1. Select **Tools->Customize commands**, choose the **Commands** tab then click the **VR Customize button**

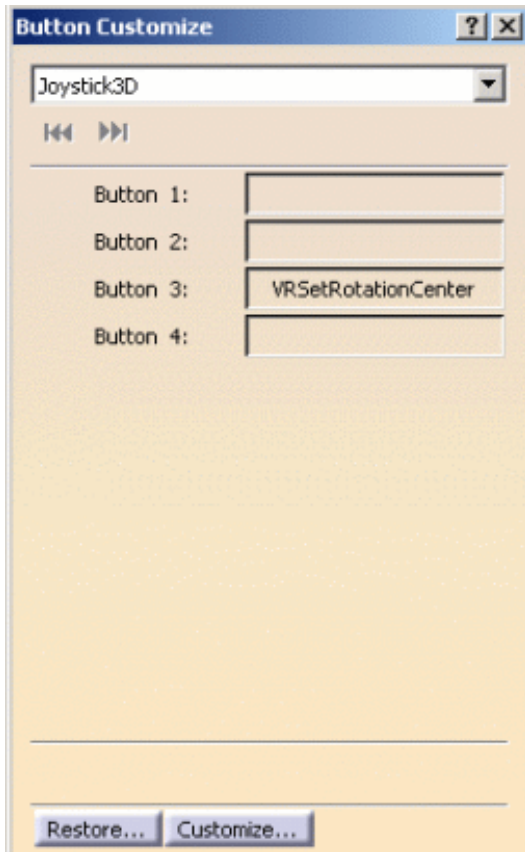


You can also access the Button Customize dialog box using one of the methods below:

1. select the **View->Commands List** commands then select the **VR Button Customize** command
2. type **c:VR Button Customize** or **c:3Dx Button Customize** in the power input field then press **ENTER**
3. press button "1" of your Space Mouse or Spaceball (if you are using original settings).

The **Button Customize** dialog box is displayed:





2. Use the input device name area on top of the dialog box to select the desired device from the drop-down list in order to modify button mappings.

In our example, the device is a Joystick3D.

3. Map the desired device button to a particular command or macro using one of the methods detailed in [Customizing Buttons](#)

For example, using the Drag and Drop capability from the Customize toolbar (still open)

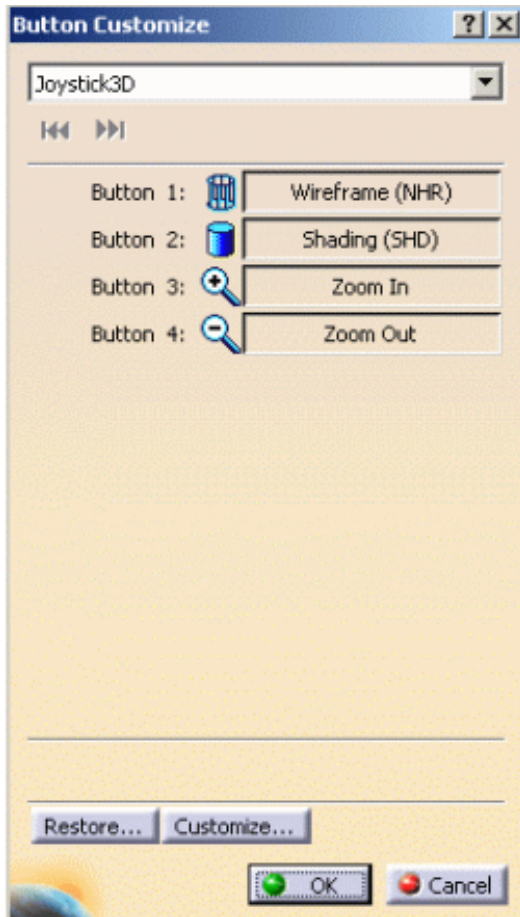
1. In the Categories list, select a category to map a command, or select the "Macros" category to map a macro.
2. Select the command or macro to be mapped from the list of commands.
3. Press and hold down the left-mouse button on top of the command or macro entry.
4. Drag the icon to the top of the button drag-and-drop area.
5. Release the left-mouse button.

The command is assigned to the device button.

4. Repeat step 3. as many times as needed to map your 4 device buttons.

In our example, we assigned the following commands:

| Commands   | Buttons |
|------------|---------|
| Wireframe: | Button1 |
| Shading    | Button2 |
| Zoom In    | Button3 |
| Zoom Out   | Button4 |



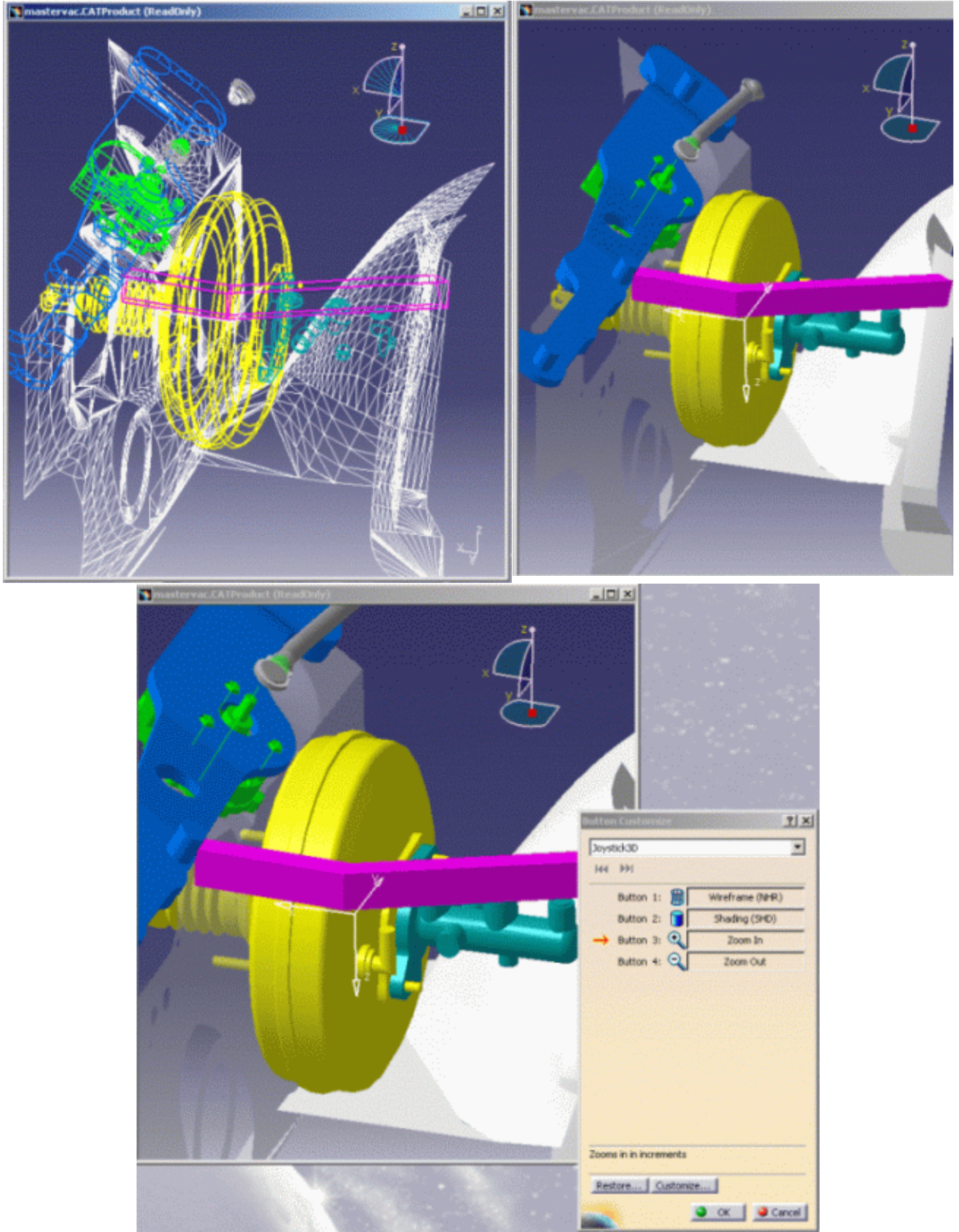
Note that when entering the DMU Immersive Review workbench, the commands assigned to the first two buttons (button1 button2) are automatically deactivated



If you wish to remove a button mapping, simply right-click the desired button drag-and-drop area then select the **Delete** contextual command. This will empty the button drag-and-drop area to indicate that no macro or command is currently assigned to this device button.

5. Press the desired device button to activate the command you assigned to it.

This is what you obtain if you press button1 and button2 successively:



The help area in bottom part of the panel displays a help text for the corresponding mapped command when a device button is pressed (a red arrow identifies in the panel the button you press)



# DMU Optimizer Settings



This task will show you how to manage representations as alternate shapes automatically.



1. Select **Tools->Options** from the menu bar

The **Options** dialog box appears

2. Expand the Digital Mockup category from the upper-left tree

DMU Optimizer

Alternate Shapes Management

|                                           | Shape name       | Activate                            | Default                  | Path For Save        |
|-------------------------------------------|------------------|-------------------------------------|--------------------------|----------------------|
| <input type="checkbox"/> Silhouette       | SILHOUETTE       | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="text"/> |
| <input type="checkbox"/> Wrapping         | WRAPPING         | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="text"/> |
| <input type="checkbox"/> Thickness        | THICKNESS        | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="text"/> |
| <input type="checkbox"/> Offset           | OFFSET           | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="text"/> |
| <input type="checkbox"/> Swept volume     | SWEPTVOLUME      | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="text"/> |
| <input type="checkbox"/> Free space       | FREESPACE        | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="text"/> |
| <input type="checkbox"/> Simplification   | SIMPLIFICATION   | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="text"/> |
| <input type="checkbox"/> Vibration Volume | VIBRATION VOLUME | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="text"/> |
| <input type="checkbox"/> Merger           | MERGER           | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="text"/> |
| <input type="checkbox"/> 3D Cut           | 3D CUT           | <input checked="" type="checkbox"/> | <input type="checkbox"/> | <input type="text"/> |

Wrapping color

Use part color when possible, otherwise...      default color

Vibration volume color

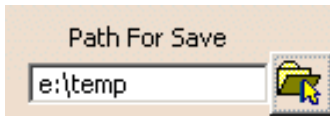
Use part color when possible, otherwise...      default color


New 3D Cut and Merger representations can be managed as alternate shapes

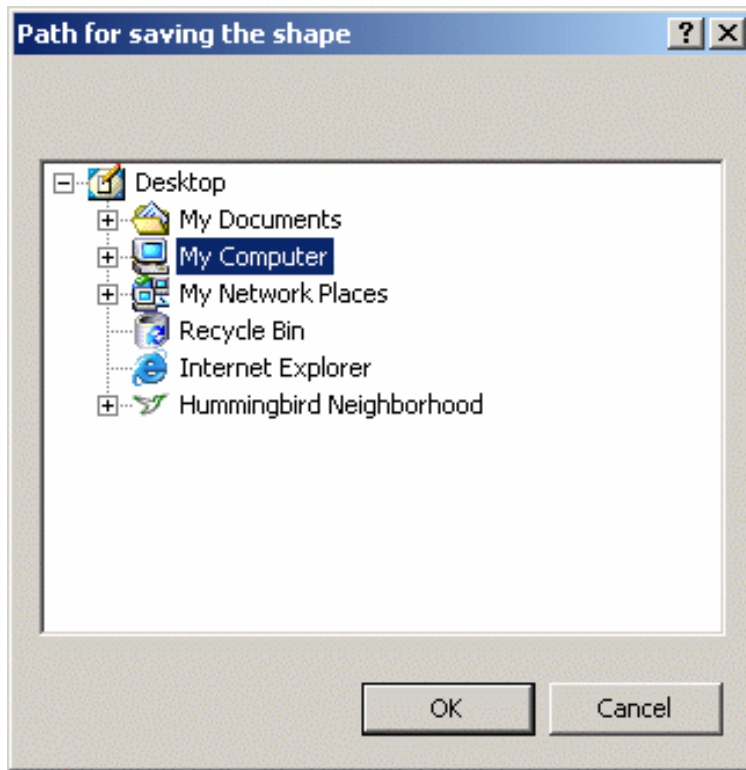
3. Click DMU Optimizer item to display the corresponding tab
4. Select the required representation, for example **Offset**.
5. Customize the representation settings as required using the following options:
  - o **Manage as alternate shape:** if this check box is selected, the resulting offset shape is managed as the offset component alternate shape (i.e. input component)
  - o **Activate Shape:** if this check box is selected, the offset representation becomes the one visualized in the session.
  - o **Default Shape:** if this check box is selected, the offset representation is the default shape, i.e. the one loaded when opening the product.


## Saving Operation...

6. Specify the required path for save:
  - o enter the precise path



- o use the **browse** button  to display the following dialog box:

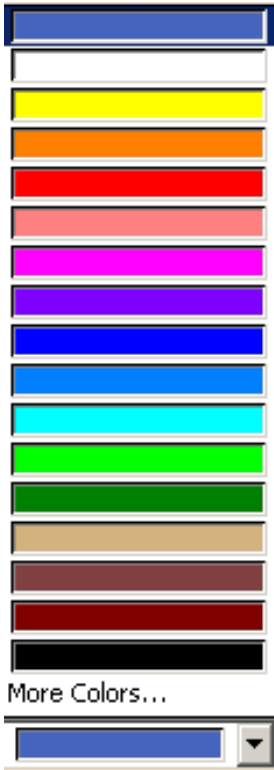


 **Note:** the path/directory you selected for one shape is now the default path specified in the Save as dialog box. You can still change it, if you need to when saving. You can customize a specific path/directory for each alternate shape.

**7.** You can also customize vibration volume and wrapping color

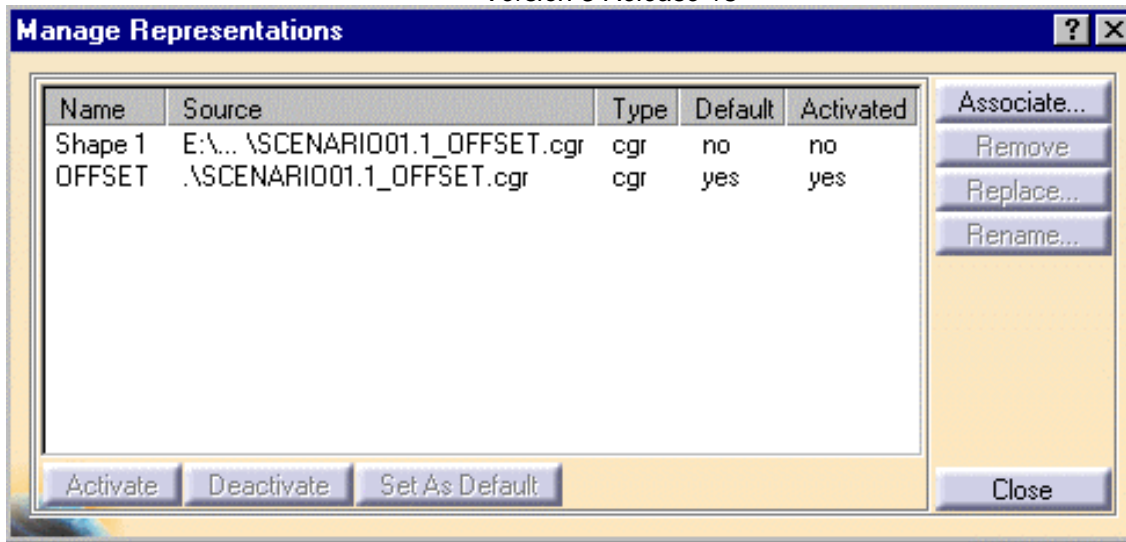
- o selecting the **Use part color when possible, otherwise...** check box: this is useful to swap from the part color to another color when it is appropriate (default color or a color of your choice)
- o selecting a color of your choice from the color palette (using the drop-down list) or keeping the default color

A list of sixteen default colors from the color palette as well as the More Colors option appear



8. Click **Ok** to confirm your operation

After a new offset calculation, if you right-click the product and select Manage representations, the Manage representation dialog box is automatically updated with the offset representation.



In DMU Optimizer, it is impossible to generate various alternate shapes with the same name. Only the last generated alternate shape is taken into account.





# Customizing DMU Space Engineering Settings



This task explains how to customize DMU Space Engineering settings.



1. Select **Tools** -> **Options** from the menu bar:

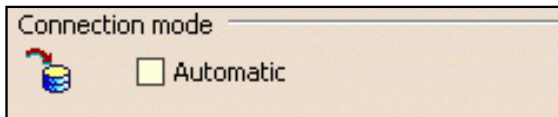
The **Options** dialog box appears

2. Click **Digital Mockup** category.
3. Click the **DMU Space Engineering** tab.

The tab page appears, containing three categories of options:

- o [Connection Mode](#)
- o [Server Connection Parameters](#)
- o [Clash Result Path](#)

## Connection Mode



### Automatic

If you select the Automatic check box, DMU space Engineering Assistant is automatically connected to the http server, you do not

need to click the Connect/Disconnect icon .  
(The connection is established during the Send to CATIA... operation)

If disabled (default mode), you need to connect manually the server clicking the Connect/Disconnect icon in the Spy toolbar.

## Server Connections Parameters

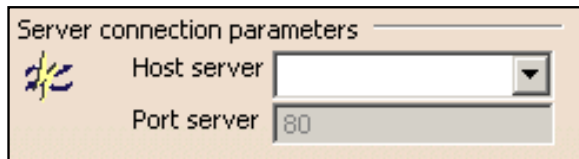


**Host Server**

Specify the host server name (name of the machine hosting the Apache server) using the drop-down list.

**Port Server**

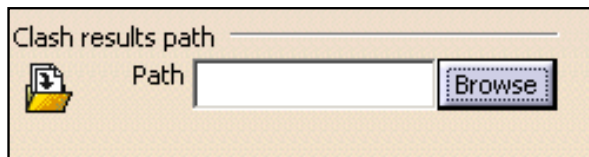
Specify the port server name (it is grayed out by default).



Server connection parameters

Host server

Port server 80

**Clash Result Path**

Clash results path

Path  Browse

**Path**

Click Browse and specify the path directory where the clash results are to be written.

4. Click **OK** in the dialog box when done.



# Equipment and Systems

Understanding Project Resource Management

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Display

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Diagrams

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Design Criteria

Electrical Process Interfacing

Electrical Assembly Design

Electrical Harness Flattening

Electrical Wire Routing

Electrical Harness Installation

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Structure Drawing

Structure Preliminary Layout

Structure Functional Design

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# Understanding Project Resource Management



This task shows you how to manage project resources like catalogs and dictionaries. These changes can only be made at system administrator level.



The Project Resource Management (PRM) file identifies resources (such as line list catalogs, user dictionaries, etc.) to the application. Specifically, the PRM file identifies each resource and its location (directory path). The PRM file also organizes the resources by discipline and application, associating resources to specific applications. Therefore, you get the correct resource, equipment catalog for instance, for the resource you are working in.

It's hierarchical structure allows you to share resources, so that you do not have to place duplicate copies of the same resource in several directories.

The PRM file is also used for certain other purposes, such as setting flags. These are explained below.

A sample file is provided with this application, and it is best to make a copy of it and edit it. The default location is `...intel_a\startup\EquipmentAndSystems\ProjectData` and the file is named **Project.xml**.

The application will function even if the user does not enter information particular to his site, but nothing can be saved. *Therefore, the first task an administrator needs to do is enter information relevant to his site or project.*

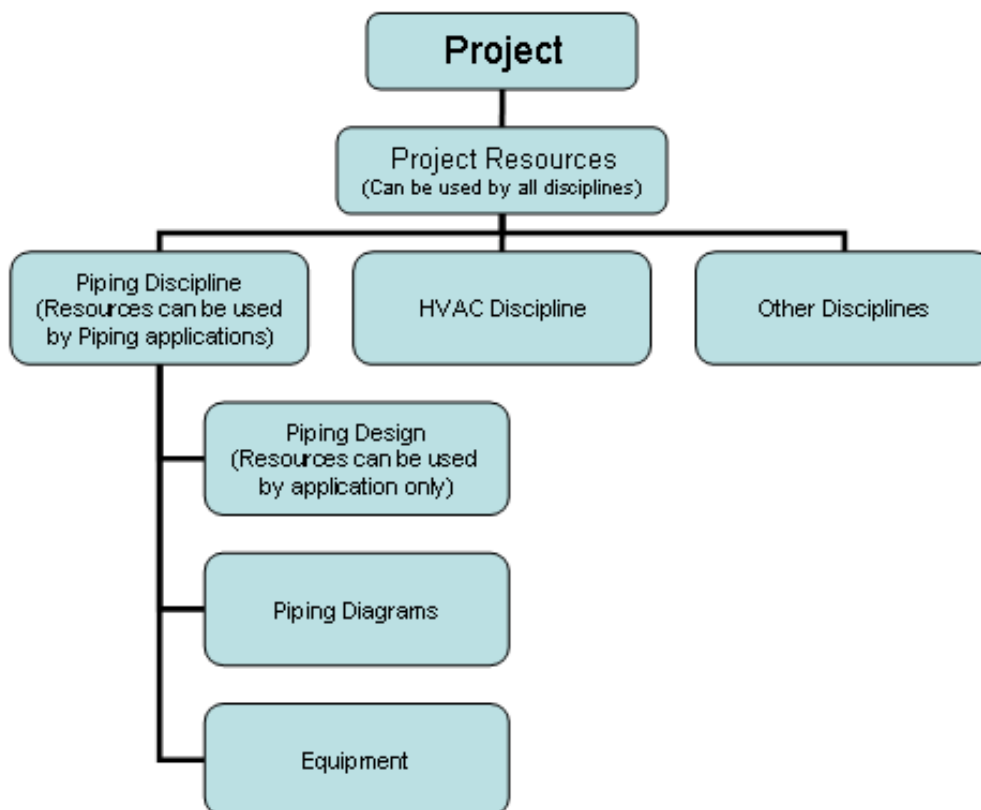


1. If there is more than one project at your site then make a project resource management file for each project. You can name the file anything you want to and change its location too (see below). However, you must set the variable in the Environment Editor. You do this by opening the Environment Editor dialog box and entering against the line **CATDisciplinePath** the directory in which your project resource management files are:

|                          |                                                                           |
|--------------------------|---------------------------------------------------------------------------|
| CATReport                | CSIDL_LOCAL_APPDATA\DassaultSystemes\CATReport                            |
| <b>CATDisciplinePath</b> | <b>g:\V5R8Dev\prjapa8\intel_a\startup\EquipmentAndSystems\ProjectData</b> |
| USER_HOME                | CSIDL_PERSONAL                                                            |

If you have more than one project resource management file then by default the application will start with the file named **Project.xml**, if there is such a file, *or the last file used*. If you wish to select another resource management file then you must open it by clicking **Tools - Project Management**. Go to the **Select** tab, and select the file and discipline you need. This dialog box shows each resource available to you, unless the value of "**Visible**" (see below) against a resource or application is set to No.

2. The resource management file is organized into several sections to make it easier to manage and utilize resources. The image below shows its hierarchical structure, with an exception, which is explained below. "Other disciplines" refers to other disciplines like Tubing, or AEC V4 to V5 Migration Discipline, that are placed at the same level. They are not shown in this image for reasons of space.



The **Project** box refers to the project for which the PRM file has been created, and will frequently be identified by the file name. In this case it is Project.xml.

When you look at the Project.xml file you will see that the resource listings begin with **project resources**. Any resources you place under the heading Project Resources can be used by all applications that are included in the file. In the sample file you will see items like zones, feature dictionary and application-generated object names in this section. All disciplines and applications shown under project resources will be able to access the resources identified under the heading project resources.

The next level is the **discipline** level. All resources placed under a discipline, such as Piping, can be used by all the applications which are listed under it. In the example above, Piping Diagrams, Piping Design and Equipment applications are shown under the Piping Discipline. Resources such as various catalogs and report definitions are listed at the discipline level and can be accessed by all applications that belong to that discipline.

The next level of resource management is the application itself. Any resource referred to at this level can only be used by that application. Resources like catalogs, files that contain connector attributes and resolved parts directories are listed at this level.

Equipment Arrangement and Hanger Design have been placed under several disciplines. These applications are considered multi-discipline because their resources are used by all disciplines. When you are working in these disciplines you may need Equipment or Hanger resources. However, in different disciplines you may not use the same resources from Equipment Arrangement or Hanger Design and this structuring allows you to place different resources under each discipline. For instance, under the Piping Discipline, Equipment Arrangement may have a different catalog (with Piping related equipment only) than under the HVAC Discipline.

Equipment and Hanger applications have *also been placed at the project resource level*, primarily to allow sharing of resources, under the headings "Equipment Application Resources" and "Hanger Design Application Resources".

To explain: If you are working in Piping Design you are in the Piping Discipline. You select the discipline by clicking Tools - Project Management, or simply by opening an application, which will activate the correct discipline. For this to happen your PRM file must be set up correctly.

Now, while in the piping workbench, you want to place equipment in your document, and you launch Equipment Arrangement. When you do so you will only have access to equipment resources that are identified in the PRM file under the piping discipline, for instance the "piping equipment catalog" containing piping related equipment. But let us assume that in your project you have certain types of equipment that are used by all disciplines. You can create a catalog that contains all this common equipment and identify it under "Equipment Application Resources" at the project level (it should **not** be identified under discipline also). When you do this you will have access to

this catalog when you open Equipment Arrangement under any discipline.

Some applications have the entry "Visible = yes". If you set the value to No then this application cannot be used and will not be visible in the **Tools - Project Management** dialog box.

**3.** A resource entry looks like this:

```
<Resource Name= "PipingIDSchema" Description= "Piping ID Schema Directory">
<ID Type= "Path" Driver= "File" Location= "..\..\EquipmentAndSystems\Piping\DataDictionary"/>
</Resource>
```

- o You should not change the Resource Name, even if you replace a resource with a different one.
- o You can change the Description if you want to - this is a brief explanation of the resource.
- o The Type field refers to file type. If the Type field says CATIA, then it refers to a file type unique to CATIA, such as **.catalog**. The type Misc is used for resources which are of a type not unique to CATIA and must be opened in another way. The type Path is similar, except that in the Location field only the directory in which the resource is located is named.
- o If your resources do **not** reside in ENOVIA then enter File in the Driver field. Define the Location field as follows: As shown in the resource example above, the location entry is relative - it is relative to the entry you made in the Environment Editor as shown in Step 1. The entry in the location field will be *added* to the entry you had in the Environment Editor and it is in that location that the application will look for the resource. *Which is why it is preferable to enter absolute paths - including the drive letter - in the location field.*
- o If your resources are **ENOVIA-based** then enter ENOVIAV5 in the Driver field. For ENOVIA-based resources, all you need to enter in the Location field is the file name of the resource without the file extension. For instance, the catalog **PipingParts.catalog** would be entered as **PipingParts**.
- o The Location field is sometimes used to enter a value for a flag or behavior, such as 0 or 1, or True and False.
- o Some resources have the entry "Visible = yes". If you set the value to No then this resource cannot be used and will not be visible in the **Tools - Project Management** dialog box.

**4.** The **Project Resources** listed in the sample Project.xml file are described in the following sections, beginning with the resources referred to under Project Resources:

- o **ID schema resources:** The "location" lists the directory where the rules for naming objects are stored. Use the default location provided in the sample file. You need to have this for every application you use. In addition, there is an entry for "MultiDisciplineIDSchema." This location is used for objects -such as zones - that are used by all disciplines.
- o **User dictionary resources:** In the location field enter the name of the CATfct file for each application. You do not need to enter the location. The CATfct file is used to store all the classes and attributes created by you. The default names for CATfct files in each application are included in the sample project.xml file and you should use these names unless you have created a different CATfct file, or changed the default name. There is also a MultiDisciplineUserDictionary - this CATfct file can be referred to by all applications.
- o **Zones catalog:** The zones that you create need to be stored in a catalog accessible to all users, because they are shared. The default location is CATMidZone.catalog. Even though zones are only created in schematic applications, other applications may use them when documents are moved from schematic to 3D. Enter a different name or location if you change them.
- o **Discrete values:** Many attributes have discrete values and this directory is used to store them.
- o **Schematic driven flag:** This is a flag that needs to be set for schematic driven routing and parts placement. If the value of "Location" is set to 0 then individual users can check or uncheck an option that allows schematic driven 3D design. If the value is set to 1 then the option "schematic driven" is always selected and users cannot uncheck it.
- o **Create Part Flag:** This is a flag that needs to be set if you intend to use Enovia as a database. If you use Enovia as a database then every run you create must be saved as a CATPart. If you set the value of "Location" to 1 then every run you make will be saved as a CATPart. If you set the value to 0 then the parts will not be deleted when you deleted the run.
- o **Graphic representations:** When you create graphic representations for a part you need a file in which to store the categories (single, double, etc.). That file is created within an application, as you will see later. There is also a file under Project Resources because the categories must be available to all applications. If you add a new category you must include it in this file too. Enter a new location and file name if you want to change them.
- o **Penetration openings catalog:** The profiles of the holes you may want to make through walls and partitions to pass pipes and ducts are noted in this catalog. If you make a new profile you must enter it here too. Enter a new location and file name if you want to change them.

- **Penetration cutout catalog:** This contains the rules for how much clearance to allow for the pipes and ducts you want to put through walls. Enter a new location and file name if you want to change them.
- **ID Sequence Number directory:** The IDSequenceNumbers directory contains the last sequence number that was generated for an object. You should specify a location for it.
- **Importer CATfacts:** This lists the location of your CATfct files, and is used when migrating V4 models to V5. If you change the location for the CATfct files you must enter the new location in the "Locatoin" field. You do not need to do anything if you continue to use the default location.
- **Reference Grid System:** Location of the CATPart that contains the reference grid definition used by applications. You need to modify this entry if you change the location of the CATPart or rename it.
- **Discipline super class:** This is a text file that lists the object classes that will be visible in a class browser. You can have files at project, discipline and application level.

5. The next level, as explained above, is the **Discipline** level. Resources placed under the **Discipline** category can be used by all the applications in that discipline.

The sample Project.xml file places the following resources at the Discipline level. Each entry names the file and gives its default location. If you intend to use different resources, which is likely, then you must enter the new file name and location, as appropriate. The following resources are referenced, but not all disciplines will have all of these resources.

- Specifications catalog
- Insulation specifications catalog
- Material specifications catalog
- Standards catalog
- Design rules: You need to use the default location and file name
- Parts catalog name and location: contains parametric parts
- Piping lines shared catalog: The file where shared piping lines are stored and its location.
- Sample data directory: This is the location where the reports you run will be stored.
- Report definitions directory: The formats (definitions) you create for running reports are stored in this directory.

6. In addition to resources placed at the Project and Discipline levels, resources are also placed at the **application** level and are only available to the application under which they are placed.

For information about resources placed under **2-D applications** and **3-D applications**, click [here](#).

For information about the **AEC Migration Discipline**, click [here](#). The AEC Migration Discipline refers to a product that enables you to migrate V4 models to V5. It has Piping Design, Piping and Instrumentation Diagrams, Equipment Arrangement, Tubing Design and HVAC Design resources under it.

For information about resources in the **Structure Discipline**, see appropriate application user's guides.

7. Make sure to save your changes.



# Using the PRM Command



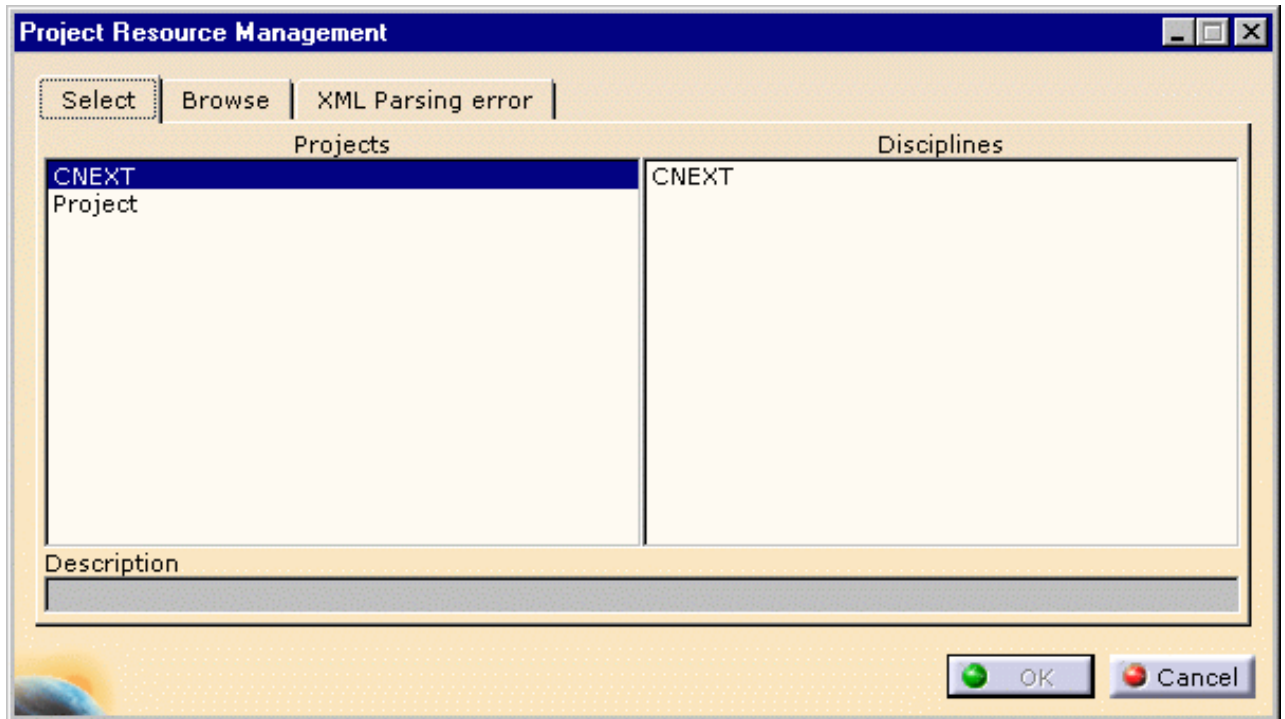
This task explains how to select a project and a discipline.



1. From the appropriate CATIA workbench, launch the PRM command by selecting **Tools -> Project Management Resource** from the menu bar.



The Project Resource Management dialog box is displayed:



The **Select** tab lists all the XML projects setup found in the directory defined by the environment variable **CATDisciplinePath**. This list begins with a CNEXT project which contains only the applications with their resources as defined and delivered with CNEXT.

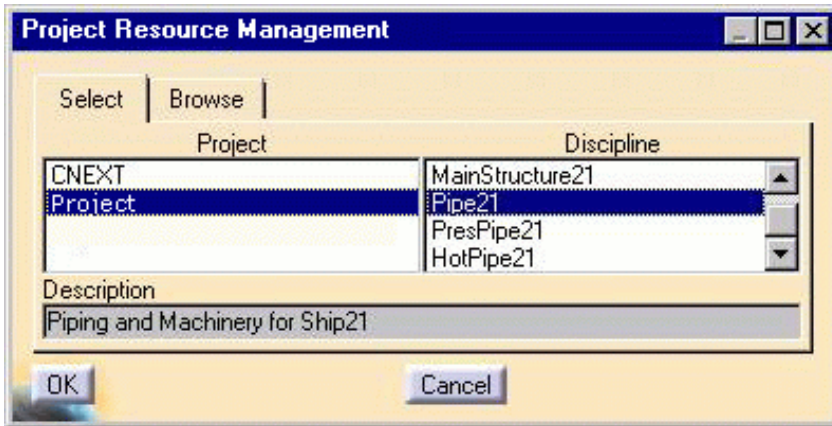
2. Select the project of your choice, here Project.

The XML is read and the disciplines it contains are listed. Similarly, all projects start with a CNEXT discipline, that is to say applications as delivered with CNEXT.



3. Select the corresponding discipline, here Pipe21.

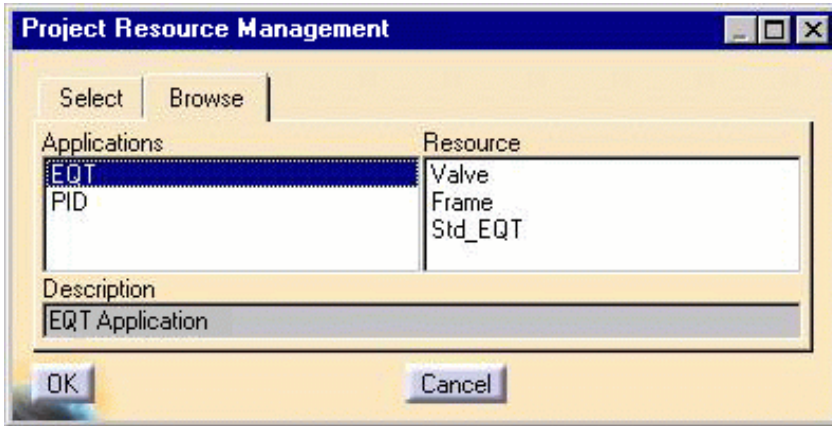
Its description is displayed, and the Browse page is populated. This discipline becomes the active discipline for the PRM system.



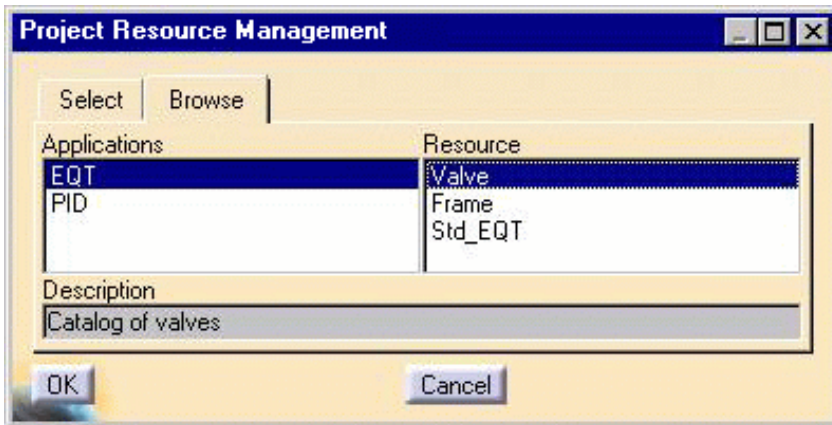
4. Select the **Browse** tab.

Once a project and a discipline have been selected, the **Browse** window lists all the applications defined for the chosen discipline, the description of the selected application, and the list of resource of that application.

5. Select the application of your choice (here EQT) to display its description and list its resources:



6. Select a resource (here Valve) to display its description.



Note that the description does not declare as active the application to the PRM.



# General Settings



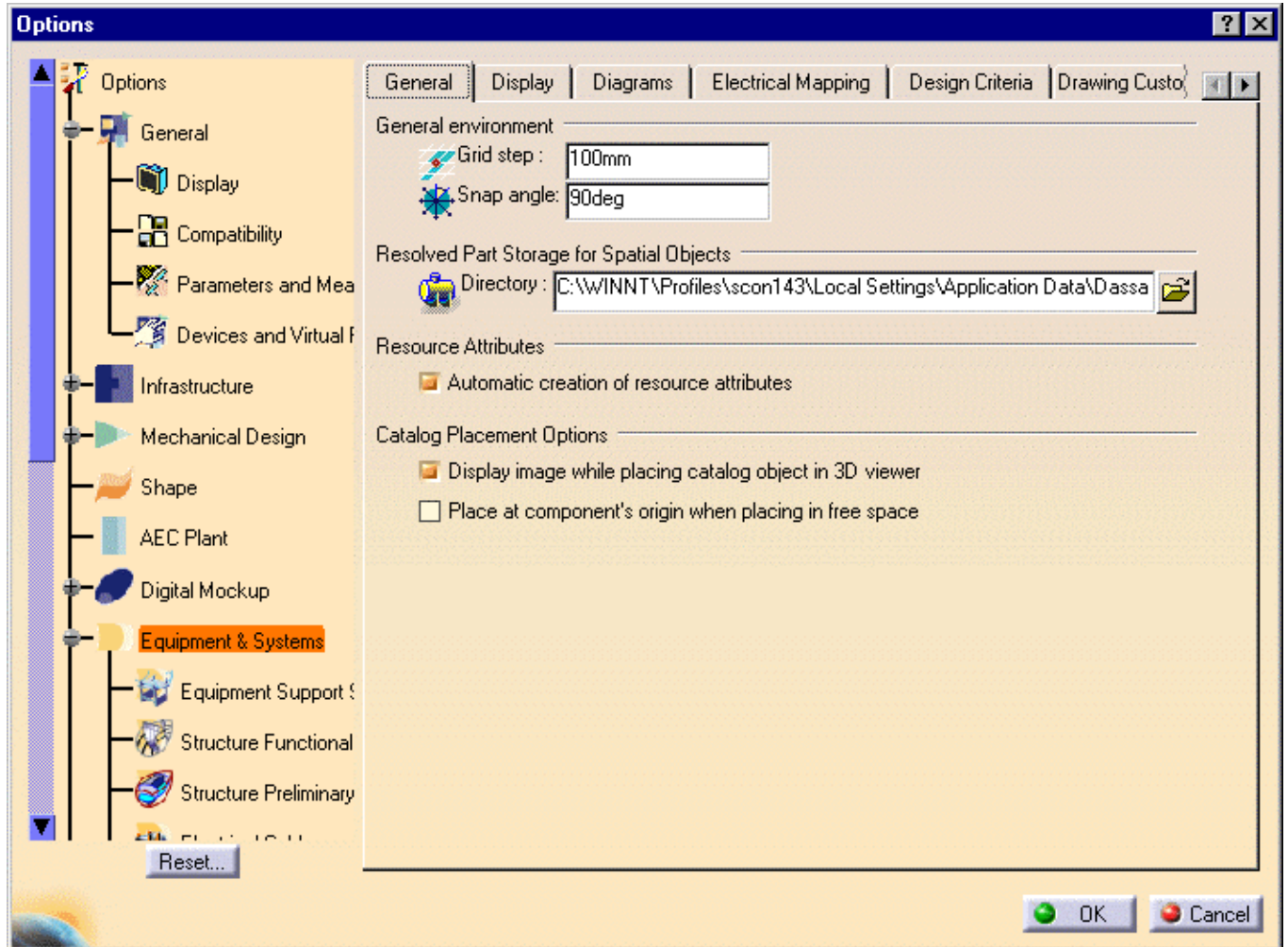
This task is to inform you about the settings in the **General** tab.



The General tab provides settings for General Environment, Resolved Part Storage path, Resource Attributes and Catalog Placement Options.



1. Click **Tools - Options - Equipment & Systems** and select the **General** tab.



2. General Environment:

- **Grid Step.** This is the default grid step setting that displays in the General Environment toolbar at the bottom of the 3-D viewer. Even if you change the value in the toolbar during your session, the value entered in the Grid Step field will re-appear when you open a new session. Enter your preference for the Grid Step. Note: The Units (unit of measure) is set in the Units tab under Tools - Options - General - Parameters and Measure. See also Set Correct Working Units and Grid.
- The **Snap Angle** sets the degree of rotation of the plane manipulator. See Using the Plane Manipulator for more information.

3. Resolved Part Storage for Spatial Objects:

- **Directory.** This is the location where resolved *Spatial* objects *unique to the Plant Layout, Systems Routing and Systems Space Reservations products* are stored. Click the Open file button and navigate in the Resolved Part Storage Directory Browser window to set or change the location for the directory.

4. Resource Attributes:

- **Automatic creation of resource attributes.** Certain attributes accompany *Spatial* objects (those created in Plant Layout, Systems Routing and Systems Space Reservations); objects such as item reservations, areas, runs, etc. These attributes, or properties, are generated specifically for the Delmia product line and cannot be assigned or changed in other product lines.

Although you will not be making use of these resource properties, their creation will have no affect on the product you are working in. If you do not want these properties to be created, uncheck the box **Automatic creation of resource attributes** and click **OK**.

5. Catalog Placement Options:

- *Display image while placing catalog object in 3-D viewer.* You can choose to hide or display a "preview" of the part you are placing. Check or uncheck the box **Display image while placing catalog object in 3D viewer** to obtain the effect you want.
- *Place at component's origin when placing in free space.* By default, when you click (in free space) to place a part, the center of the part will be placed at that location. The application determines the center of the part by creating a box around it and selecting the center of the *base* of the box. You can also choose to place the *origin* of the part at the point where you click. To do this check the box **Place at component's origin when placing in free space**.

6. Click OK when you are done.



# Display Settings



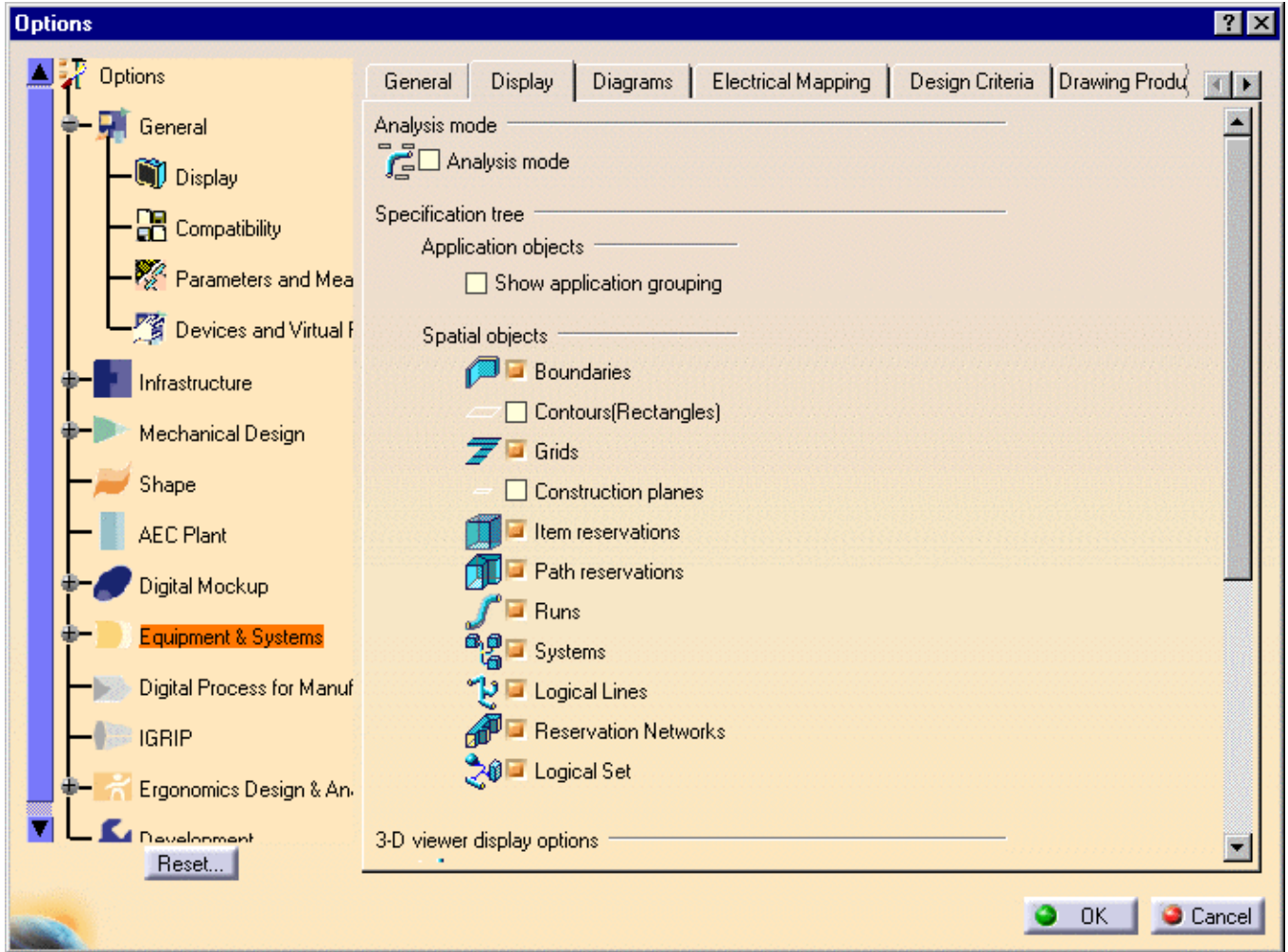
This task is to inform you about the options available in the Display tab.



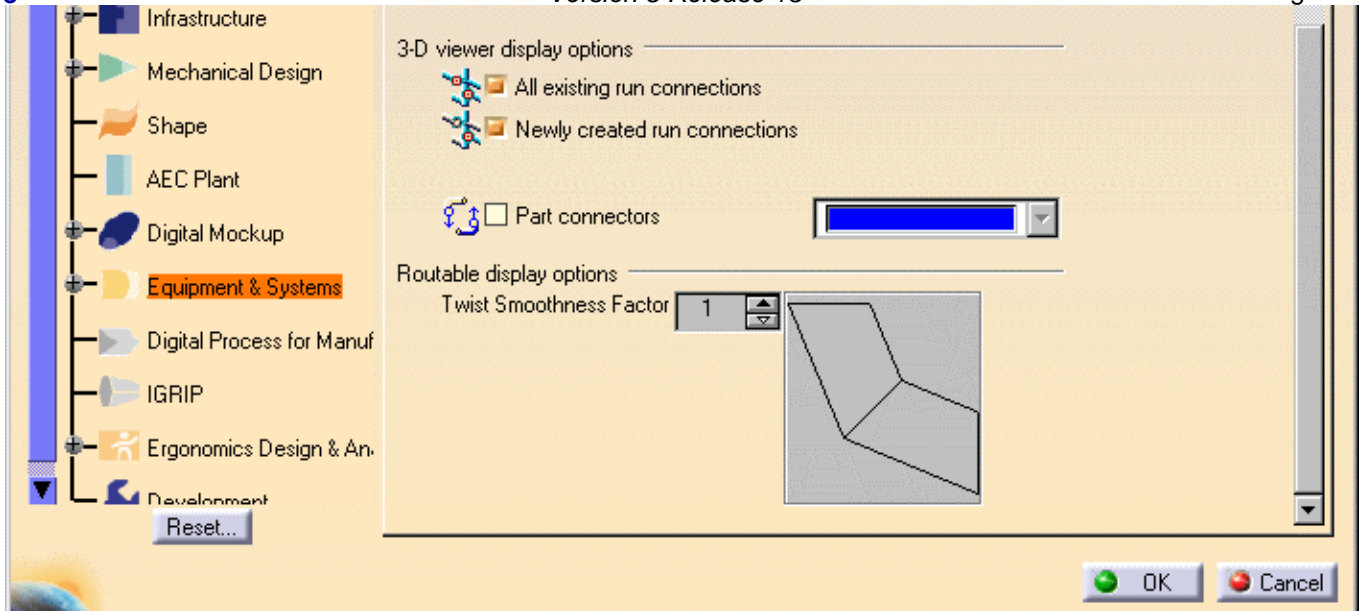
The Display tab provides display options for Analysis Mode, Specifications Tree, 3-D Viewer Display and Routable Display options.



1. Go to **Tools - Options - Equipment & Systems** and click the **Display** tab.




- o Analysis Mode: Check the box in Analysis Mode to display information about routables. It works only when you are performing an action with a routable, such as placing a part, branching a run or creating an offset route.
- o Specification Tree: In *Application objects* check the box **Show application grouping** to organize the objects in the tree by application, e.g. Piping objects, Equipment, etc. Under *Spatial objects* check the boxes of the items you want to appear in the specifications tree. Spatial objects may only be placed using the *Systems Routing*, *Systems Space Reservation* and *Plant Layout* workbenches.




- o 3-D Viewer Display Options: Check or uncheck the boxes to set the connection and connector display options as desired. Set the color of the connection and connector symbols using the drop down color palette.
- o Routable Display Options: Increase the Twist Smoothness Factor to increase the smoothness of a run at a twisting turn.

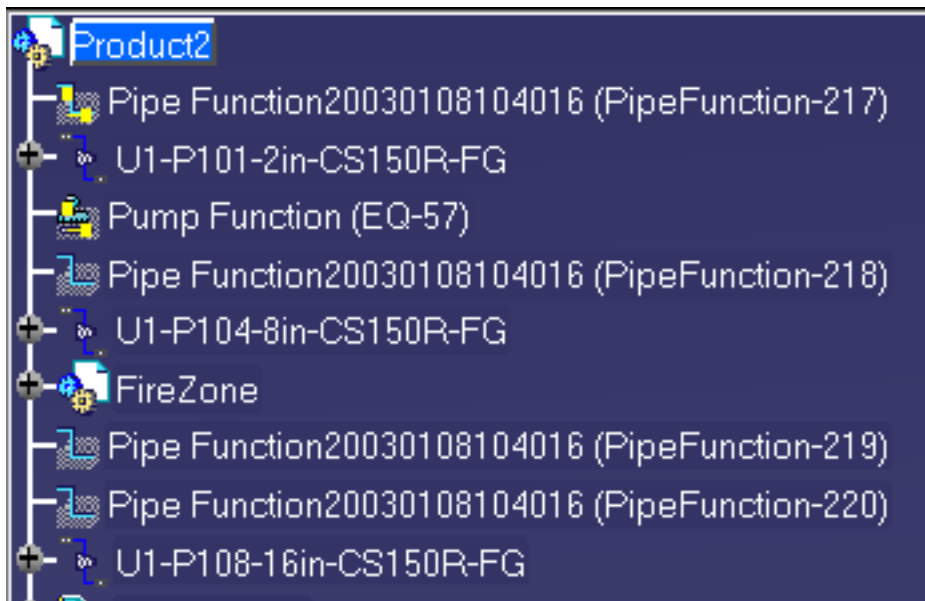



# Specifications Tree Settings

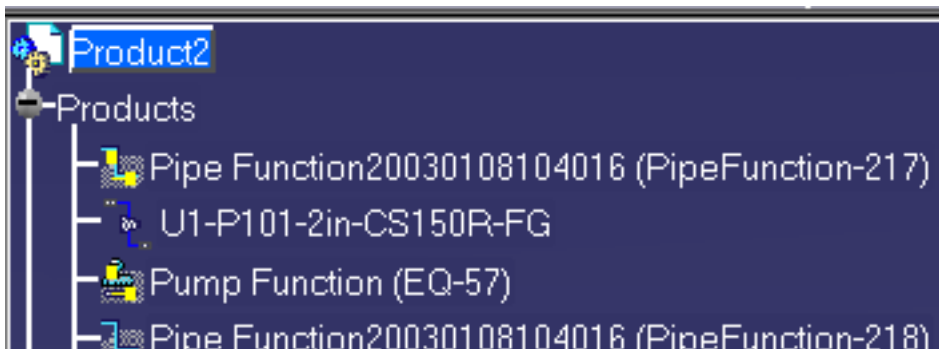
 This task shows you how to change the specifications tree settings so that the objects in the tree display in an organized manner of your choice.

 The default display setting for objects in the specifications tree is single-tier, and the objects appear in the order in which they were placed in the document.

By changing the **Tools** -> **Options** settings they can be organized in two ways, by product and, as an additional step, by application group also. If you do not check the option as described below you will be unable to collapse the specification tree display and it looks as shown below. After you check the **Products** option the objects will be organized under the **Products** node and the tree can be collapsed. The Products node is simply an organizational element. After you check the **Application Grouping** option the objects will be further grouped by type.

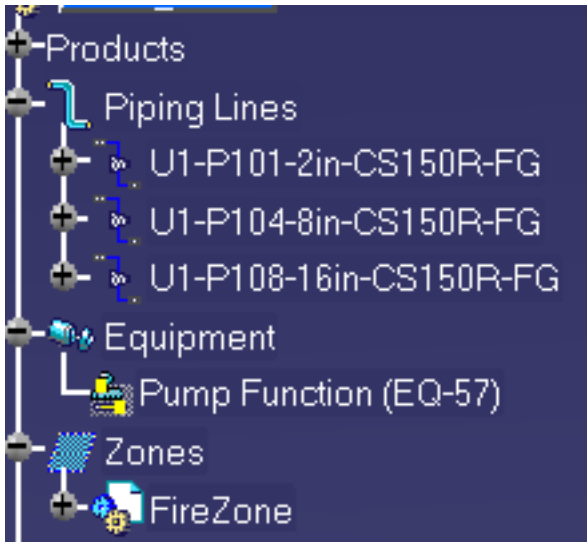


-  1. With your document open, go to **Tools** -> **Options** -> **Infrastructure** -> **Product Structure** and click the **Product Structure** tab. Check to activate the **Products** option. The objects in the specifications tree will be organized under the **Products** node.



2. To organize the specifications tree in application groups go to **Tools** -> **Options** -> **Equipment & Systems** and click the **Display** tab.

Under **Specifications Tree**, activate the **Show application grouping** option: Objects in the specifications tree will be organized under type of object. (You can expand the **Products** node to see the ungrouped elements.)




When working with Design Rules such as **Attribute Filters** and **FunctionPhysicalMapping**, or during parts creation, turn on the **Relations** and **Parameters** options.

3. Go to **Tools -> Options -> Infrastructure -> Product Structure** and click the **Tree Customization** tab.

The Specification Tree Order table displays: You can toggle any of the entries from **Yes** to **No** and back by clicking in the **Activated** column.



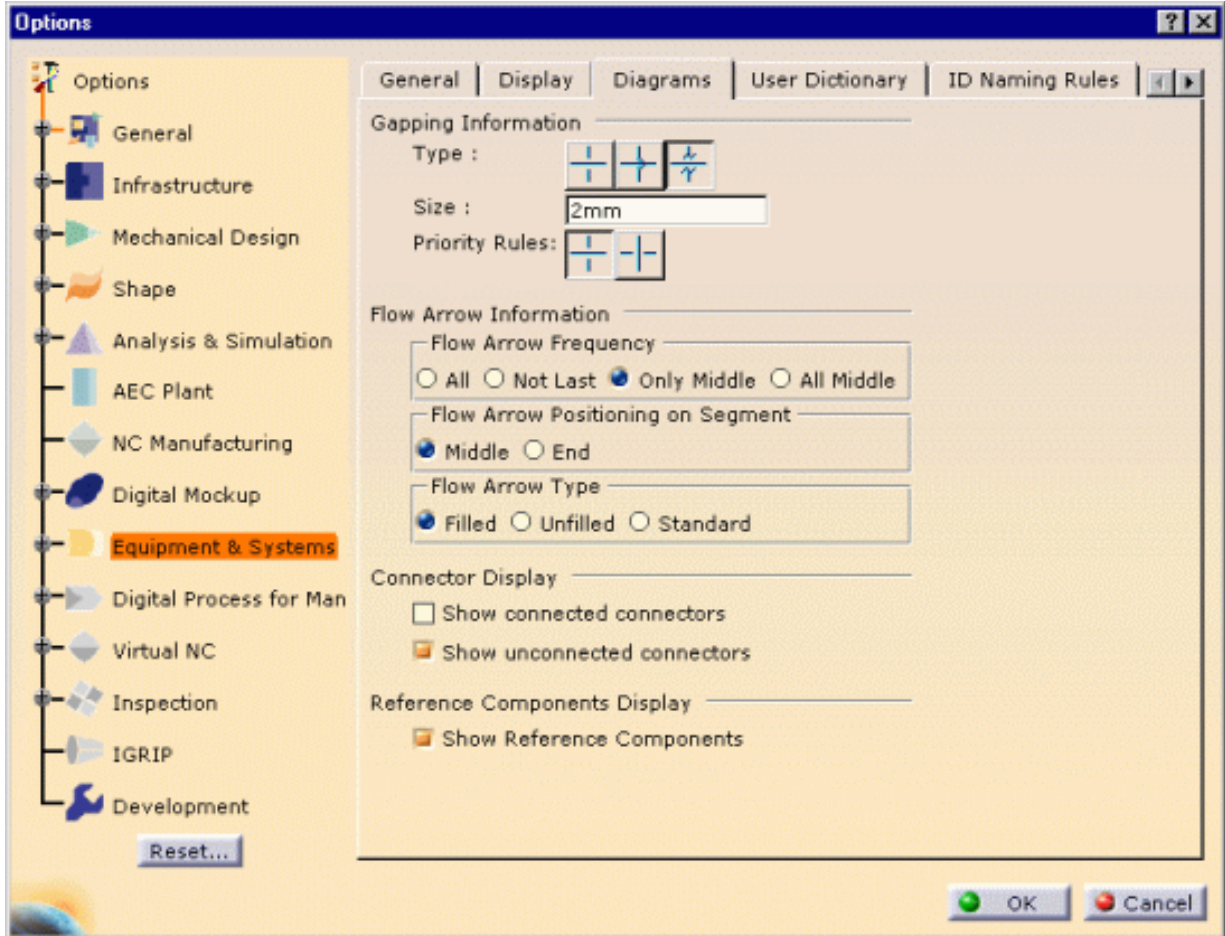
# Diagrams Settings

 This task explains the diagrams settings.

 The Diagrams tab is one of three tabs under the **Options -> Equipment & Systems** menu that is specific to P&ID, HVAC, Electrical and Waveguide Diagrams.

 **1. Click Tools -> Options, Equipment & Systems and select the Diagrams tab.**

The **Options** window displays:



**2. Make your selections from the options.**

Most of them are self explanatory.

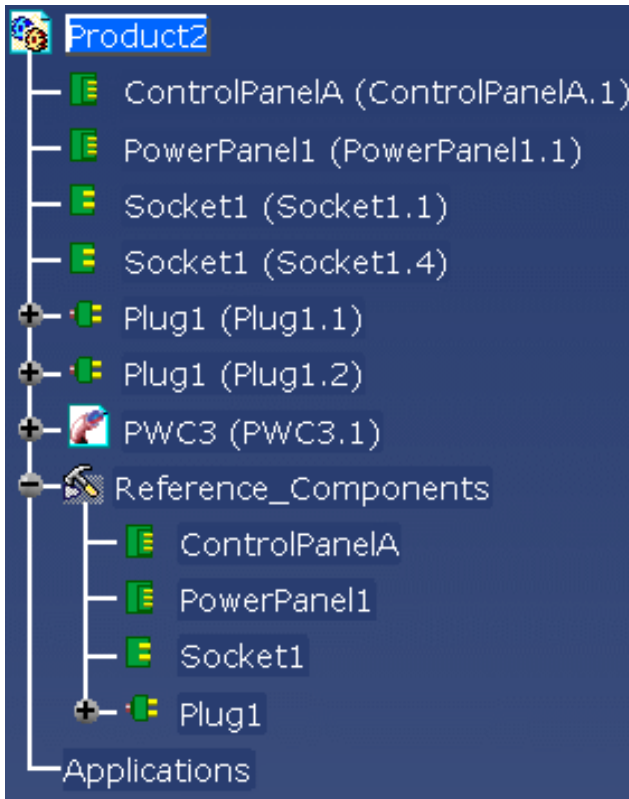
In the Connector Display frame, two options allow you to show or not the connectors in the drawing sheet:

By default,

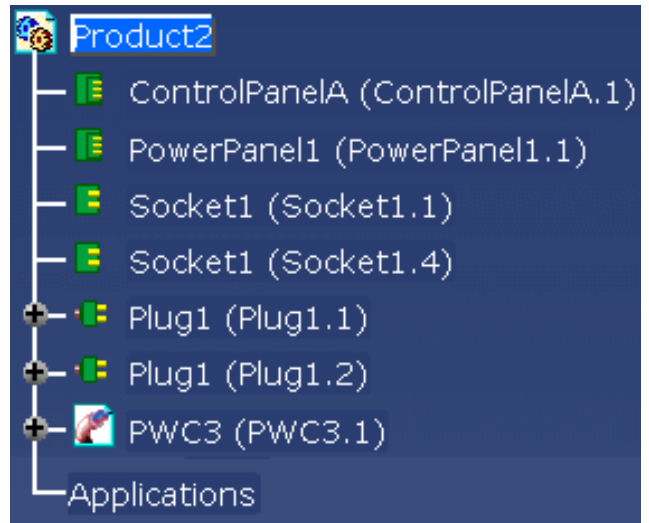
- o **Show connected connectors** is unchecked to lighten the display
- o **Show unconnected connectors** is checked to emphasize the display.

In the Reference Components Display frame, the option allows you to show or not the reference components in the specification tree: by default, the option is checked.





If you uncheck the option, the specification tree looks like this:



3. Click **OK** to validate.



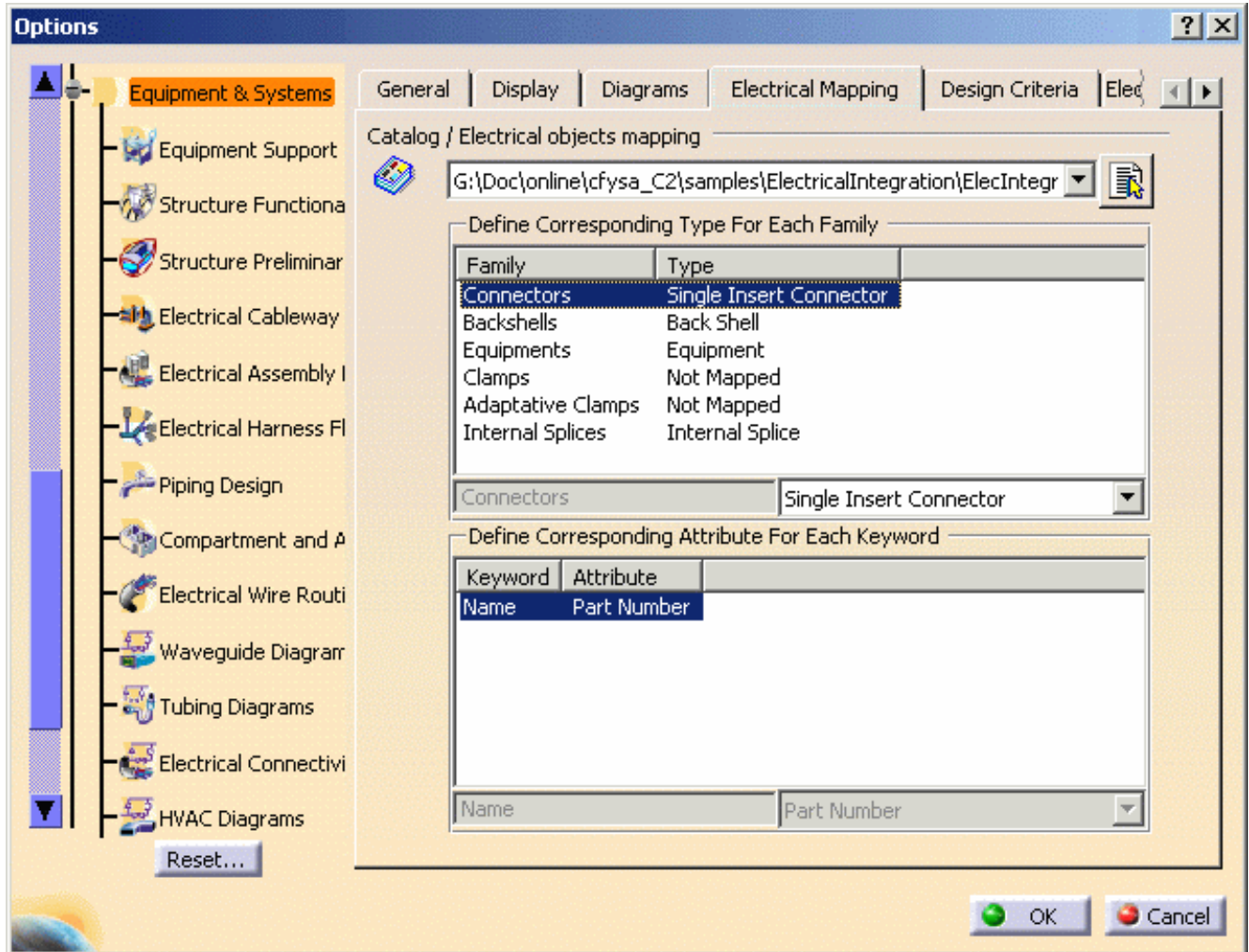
# Electrical Mapping




In order to valuate the keywords automatically when the devices are stored in the catalog with their properties, you need to define a mapping between keywords of each family and the attributes of the component type.



1. Select the **Tools -> Options** command.  
The **Options** dialog box is displayed.
2. Click **Equipments & Systems** in the left-hand box.
3. Click the **Electrical Mapping** tab.

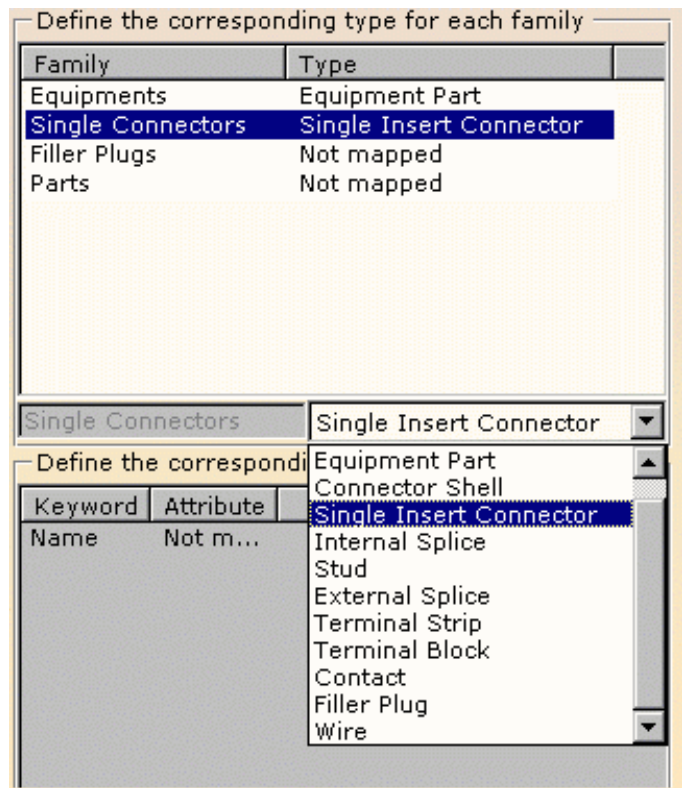


The tab lets you:

- define the current catalog using the drop-down list or the **Browse** button 
- customize the mapping between the catalog and the electrical objects
- define the keyword attributes.

4. Customize the mapping: associate an electrical **type** to the currently selected **Family**.

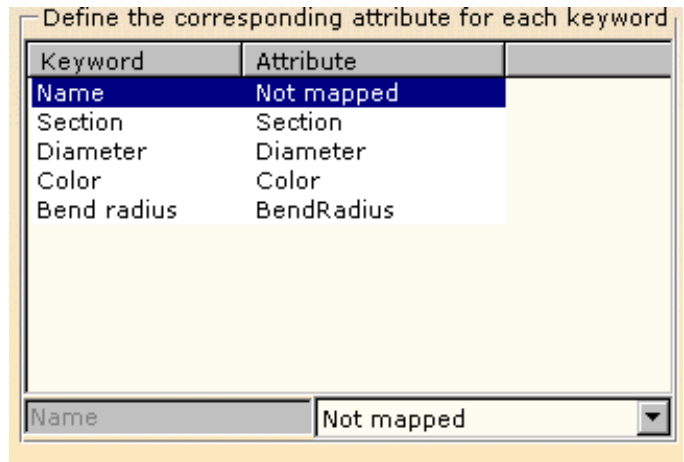
Use the combo list to select the corresponding type.




- 5. Define for each keyword the corresponding reference attribute.

The result looks like this:

Use the combo list to select the corresponding attribute.



- 6. Click **OK** to validate the options.

 The mapping definition can be set by the administrator. In this case it can also be locked.

To get more information about the Administration Mode in CATIA, refer to *CATIA - Infrastructure User's Guide*.



# Design Criteria Settings



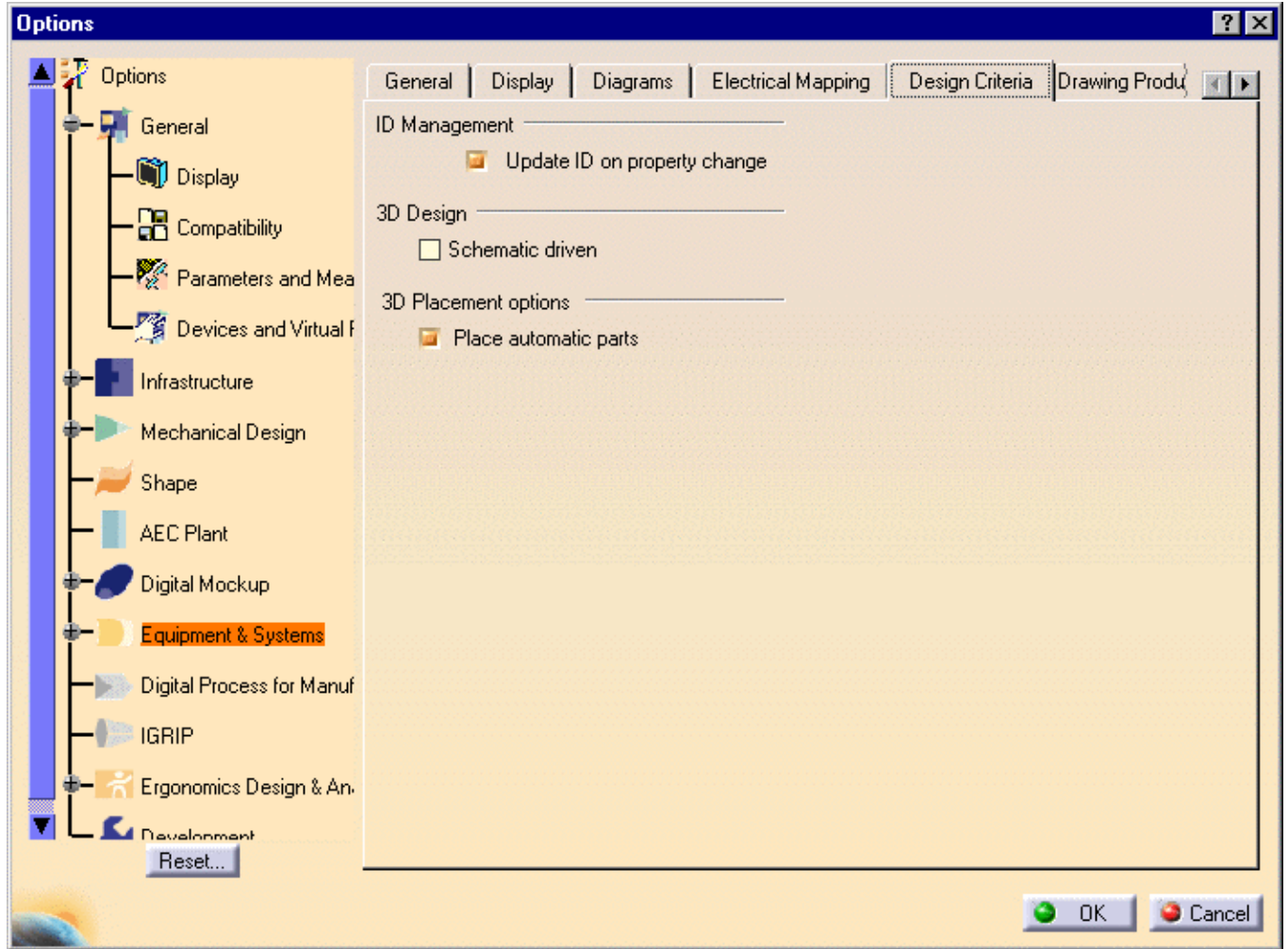
This task is to inform you about the settings in the Design Criteria tab.



The Design Criteria tab allows you to toggle settings for ID Management, 3D Design, and 3D Placement.




1. Click **Tools - Options - Equipment & Systems** and select the Design Criteria tab.



- ID Management: *Update ID on property change*. When the object naming convention (ID schema) includes an attribute, e.g. nominal size, and you change its value using the Properties dialog box, the name of the object will change to reflect the new value of the attribute.
- 3D Design: Check *Schematic Driven* if you are placing parts from a schematic into a 3D design document. Used with Piping Design, Tubing Design, HVAC Design and Waveguide Design applications.
- 3D Placement Options: When the *Place automatic parts* box is checked you can place an object such as a valve and the flanges and gaskets will be placed automatically.



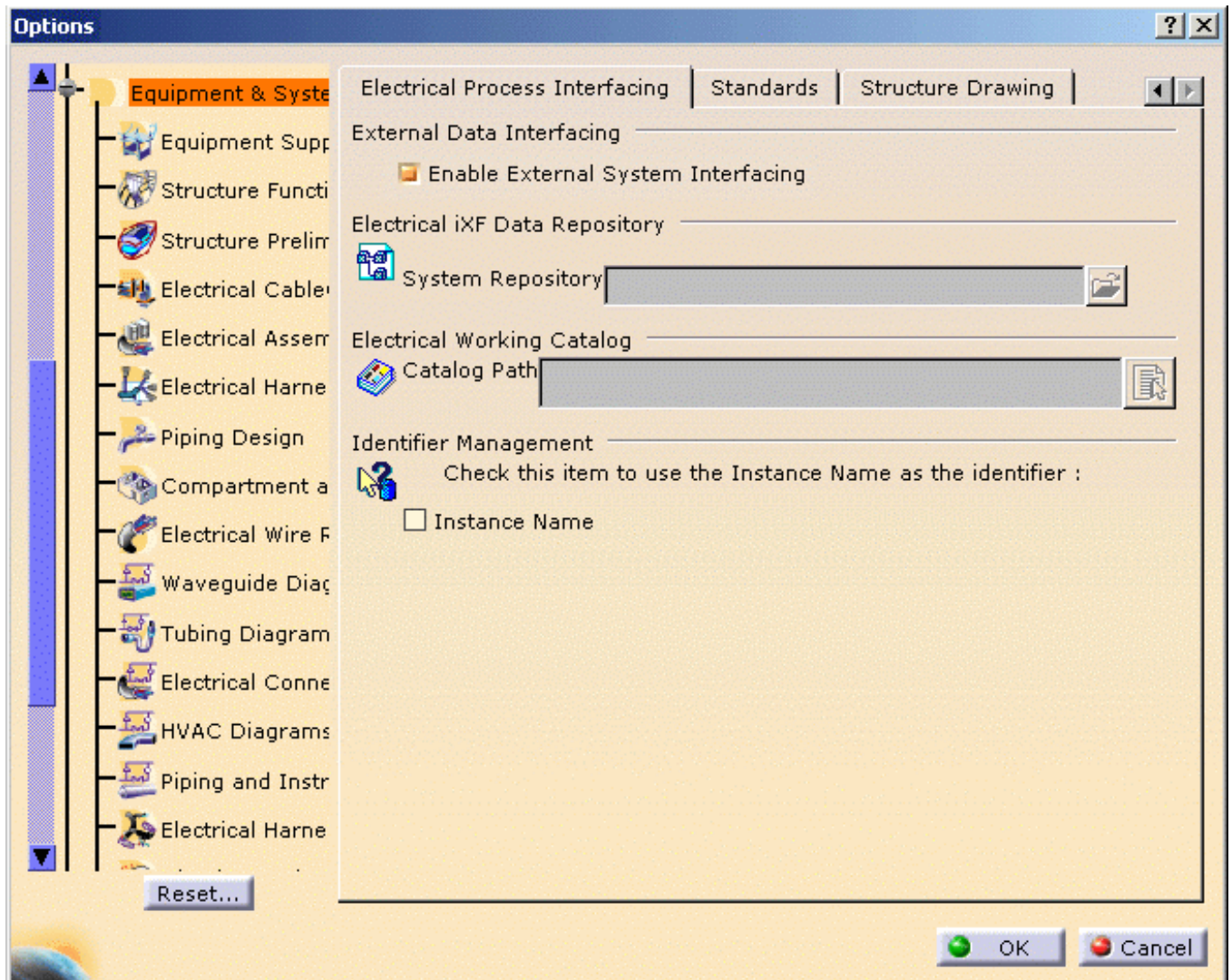
# Electrical Process Interfacing

 This document shows how to set up the options to take advantage of external data from partners through XML files or from CAA V5 APIs to get information from legacy databases.

 **Note** that the default value for External Data Interfacing is enabled:  
**Enable External System Interfacing** is checked.



1. Select the **Tools -> Options...** command.  
The **Options** dialog box is displayed.
2. Click **Equipments & Systems** category.
3. Click the **Electrical Process Interfacing** tab.




The tab lets you define:


- the access to external data
- the electrical iXF repository path
- the electrical working catalog path
- the identifier for mapping management.


4. Make sure **Enable external systems interfacing** is checked.

However, you are required to identify the path:

- of the folder in which the XML files available are stored
- of the working catalog containing the V5 parts which **Part Number** are referred to in the XML files.

5. Use  to locate the iXF systems repository.

6. Use  to indicate the catalog path.
7. Check the **Instance Name** option so that the **Instance Name** will be taken into account instead of the **Reference Designator** to insure the mapping between 3D objects and external system's objects.
8. Click **OK** to validate the options.

 It is also necessary to define the **electrical objects mapping** regarding the working catalog.  
To do so:

- use the **Tools -> Options...** menu.
- then in the **Equipment & Systems** category, select the **Electrical mapping** tab.

Refer to the [Customizing - Equipment & Systems - Electrical mapping](#) documentation.



# Customizing for Electrical Library



Before you start your first working session, you can customize the way you work to suit your habits.

This is done using **Tools** -> **Options** from the menu bar.

This type of customization is stored in permanent setting files. Settings will not be lost if you exit your session.

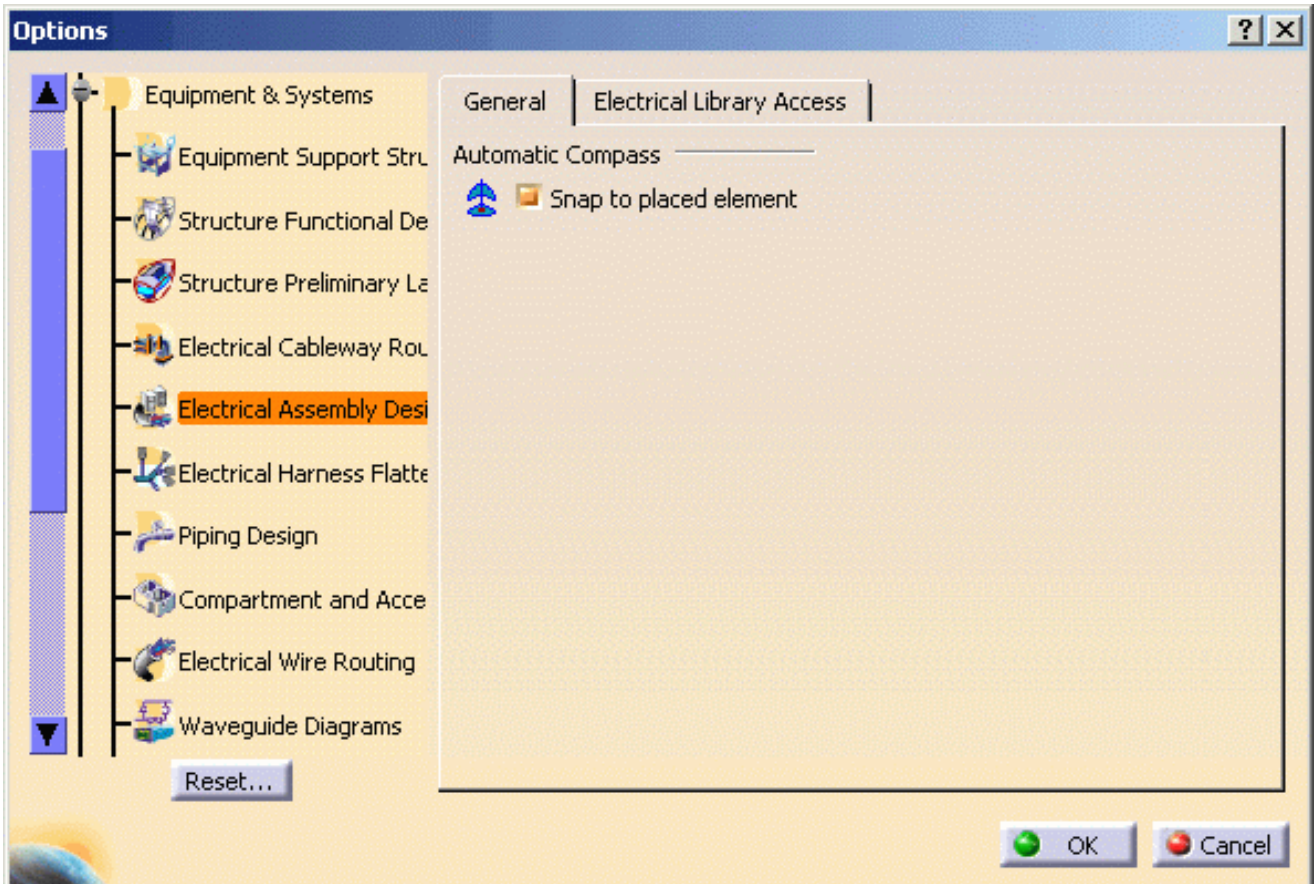
## General

### Electrical Library Access

# General



1. Select the **Tools** -> **Options** menu.  
The **Options** dialog box is displayed.
2. Click **Equipment & Systems** in the left-hand box.
3. Click the **Electrical Assembly Design** workbench.



It lets you define the **Automatic compass** option:

- Check the **Snap to placed element** to take advantage of the compass:  
when you place an element, the compass snaps to this element allowing you to modify the orientation and location.
  - By default this option is checked.
4. Click **OK** when done.





# Electrical Library Access



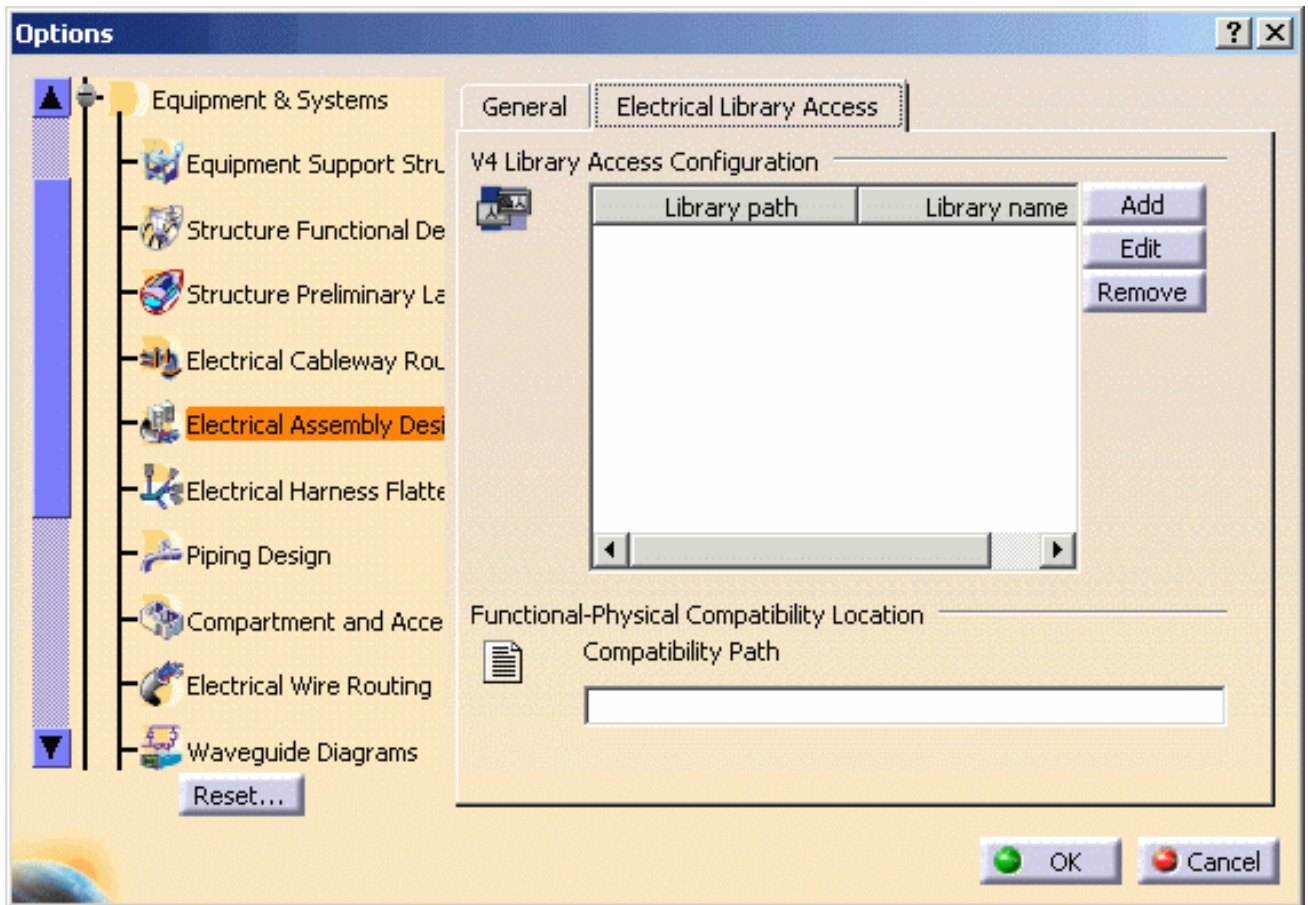
Electrical System Functional Definition takes advantage of Electrical Library for the mapping between functional equipments and connectors to V4 library parts. It consists of the definition of the corresponding physical components for functional equipments or connectors. This is done in EFD.

The V4 libraries access is defined in the **Tools -> Options** menu.

Moreover, a compatibility can be established between functional and physical components in order to reduce the list of physical components available for a functional component. The compatibility sets a relation between the **External Reference** attribute of the functional component and the **Part Number** attribute of the physical component. These relations are stored in a compatibility table, in csv format.



1. Select the **Tools -> Options** menu.  
The **Options** dialog box is displayed.
2. Click **Equipment & Systems** in the left-hand box.
3. Click the **Electrical Assembly Design** workbench.



It lets you define:

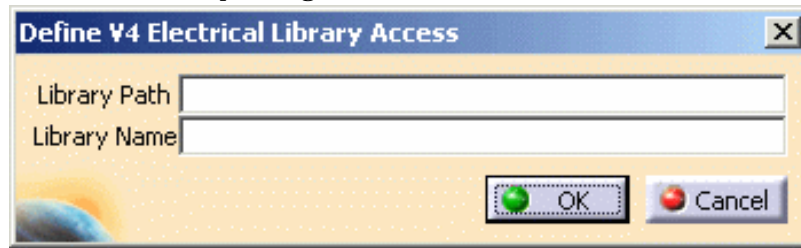
- o the **library names and their locations** in the upper frame
- o the **location of the compatibility table** if it exists  
(for example: **c:\Temp**)

4. Click **OK** when done.

## VA Library Access Configuration

- To add a library in the list, click the corresponding button.

The dialog box opens:



- Enter the library name and path then click **OK** to validate.
- You can delete or modify the library access, using respectively the **Remove** or **Edit** button.

### Functional-Physical Compatibility Location

- To gain access to the compatibility table, enter its path in the **Compatibility Storage Path** field. Let's see its structure:

|   | A        | B | C | D   | E | F |
|---|----------|---|---|-----|---|---|
| 1 | ExtRefEq |   |   | PN1 |   |   |
| 2 | ExtRefEq |   |   | PN2 |   |   |
| 3 | ExtRefEq |   |   | PN3 |   |   |
| 4 |          |   |   |     |   |   |
| 5 |          |   |   |     |   |   |
| 6 |          |   |   |     |   |   |
| 7 |          |   |   |     |   |   |

In this sample, the functional equipment which **External Reference** is ExtRefEq is compatible with three physical equipments which **PartNumber** are PN1, PN2, PN3.

For more information about the interoperability with CATIA - Electrical System Functional Definition, refer to Mapping Functional Components to Physical Parts in this User's Guide.



The Electrical Library Access can be set by the administrator. In this case it can also be locked.

To get more information about the Administration Mode in CATIA, refer to *CATIA - Infrastructure User's Guide*



# Customizing Electrical Harness Flattening



Before you start your first working session, you can customize the way you work to suit your habits.

This type of customization is stored in permanent setting files: these settings will not be lost if you end your session.



For the time being, Electrical Harness Flattening does not function with the **Visualization Mode**.

The entire geometrical bundle must be in **Design mode** for the workbench to work properly.



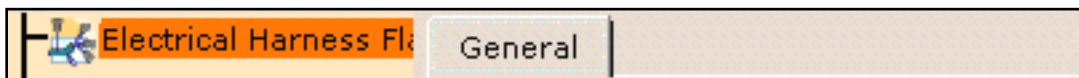
1. Select the **Tools -> Options** command.

The Options dialog box displays.

2. Choose the **Equipment & Systems** category in the left-hand box.

3. Click the **Electrical Harness Flattening** workbench.

The **General** tab displays:

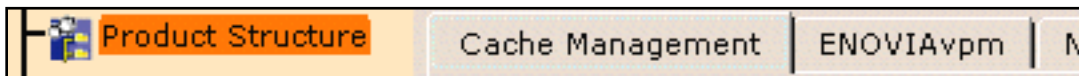


This tab lets you define:

- the synchronization report repository
- the graphic replacement catalog

4. If you want to work with the cache system, select the **Infrastructure -> Product Structure** item.

The **Cache Management** tab displays:



This tab lets you define:

- the cache activation

5. Fill up these options according to your needs.

6. Click **OK** when done.



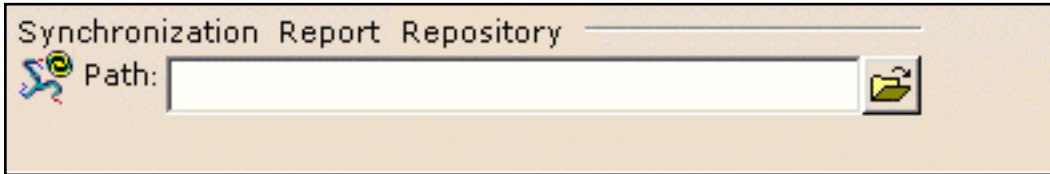
# Electrical Harness Flattening

This tab lets you define:

- the synchronization report repository
- the graphic replacement catalog


Click [here](#) to see the parent page.

## Synchronization Report Repository



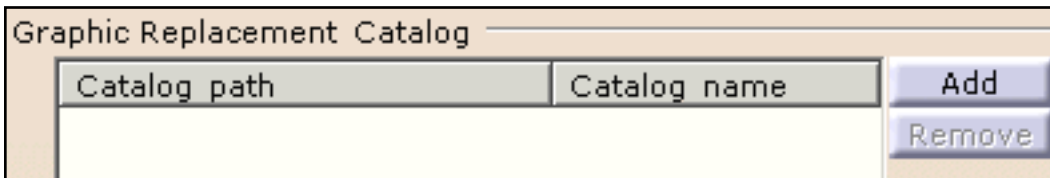
### Path

Indicate the folder path where you want to save the synchronization report html files.

 By default, this field contains: ...\DassaultSystemes\CATTemp.




## Graphic Replacement Catalog



| Catalog path | Catalog name |
|--------------|--------------|
|              |              |

### Catalog path and Catalog name

Indicate the catalog path and name where the drawing 2D details are stored.

 By default, this field is empty.

# Cache Management

Click [here](#) to see the parent page.

## Cache Activation




Cache Activation

Work with the cache system

### Work with the cache system

If this option is checked, it means that a representation of the geometry only is available and stored in the cache.

For more information, refer to the customizing section about [cache management](#).

 By default, this option is not checked.

# Customizing Electrical Wire Routing

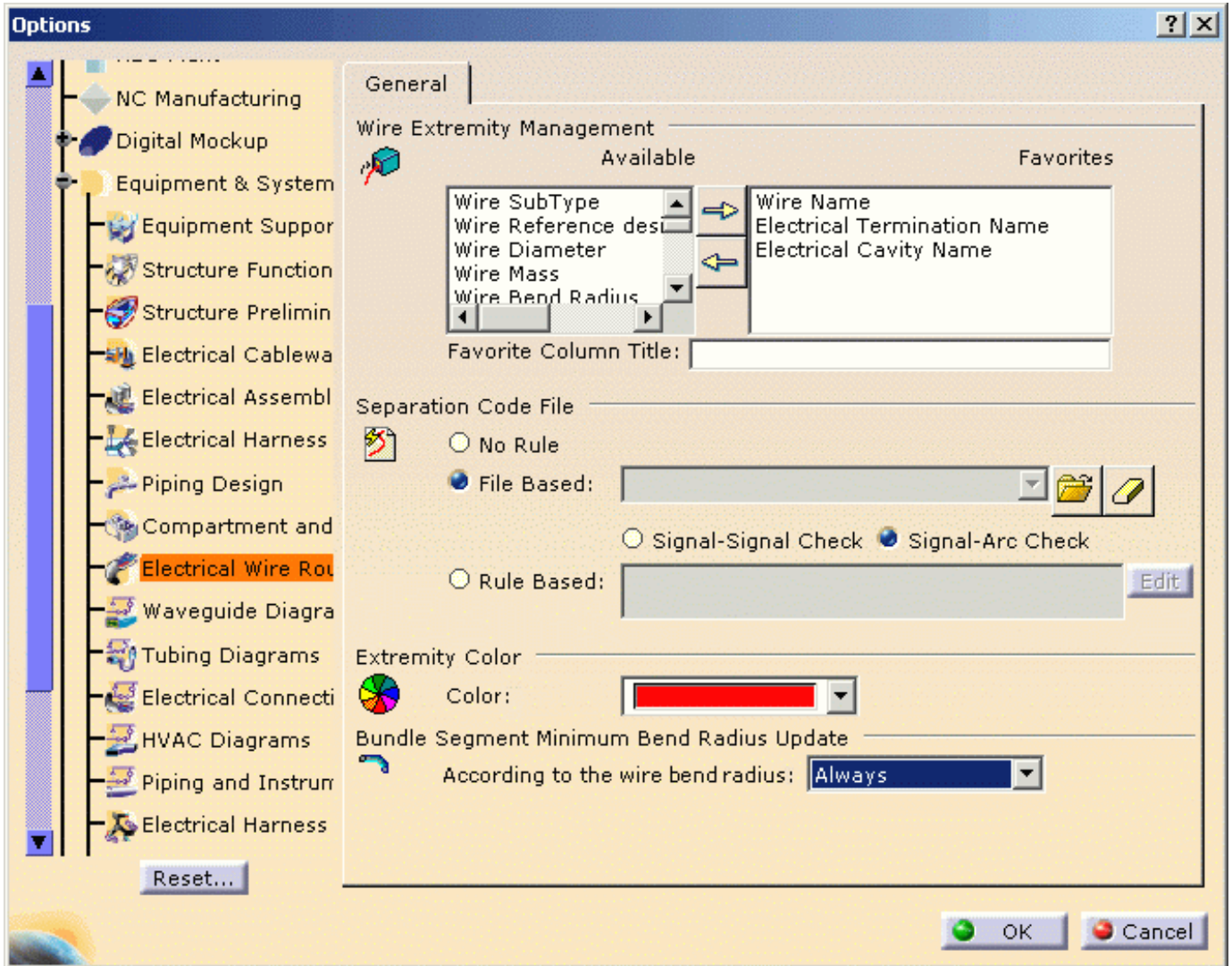


Before you start your first working session, you can customize the way you work to suit your habits. This is done using **Tools -> Options** from the menu bar. This type of customization is stored in permanent setting files. Settings will not be lost if you exit your session.



1. Select the **Tools -> Options** command.  
The **Options** dialog box is displayed.
2. Expand the **Equipments & Systems** in the left-hand box.
3. Click the **Electrical Wire Routing** workbench.

The General tab is displayed:



This tab lets you customize:

- o the [wire extremities management](#)
- o the [property naming](#) to be displayed as a column title
- o the [separation code file](#) access
- o the [extremity color](#)
- o the [bundle segment update](#) mode.

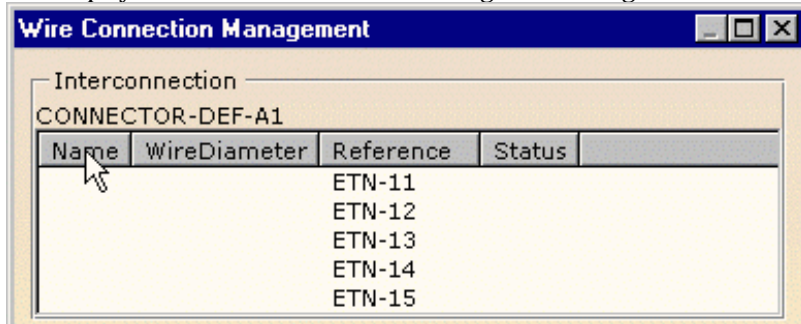
4. Click **OK** in the dialog box when done.

## Wire Extremities Management

- The **Favorites** list contains the properties displayed in the Interconnection frame of the Wire Extremities Management dialog box. To add properties to the list, select the property in the **Available** list and click the ---> arrow.
- To remove properties, do the reverse: select the property from the **Favorites** list and click the <--- arrow.

## Property Naming

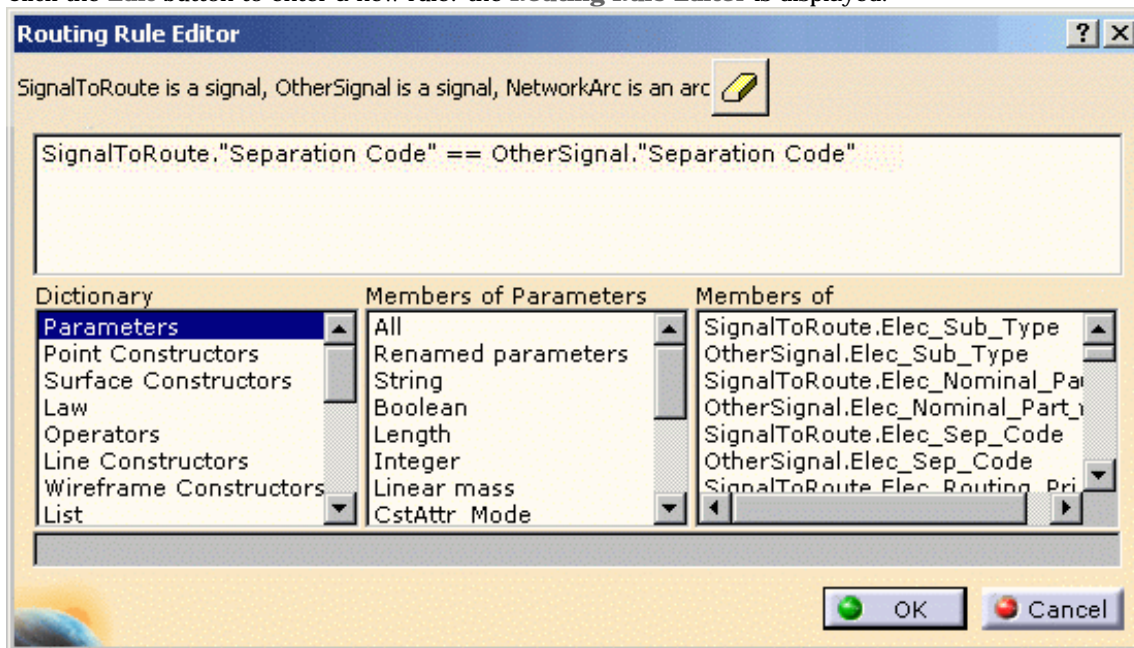
- Select a property in the **Favorites** list and enter a name in the **Favorite column Title** field. It is displayed in the Wire Connection Management dialog box:



## Separation Code File

The **Separation Code File** area is used to define Separation Code rules to optimize the routing.

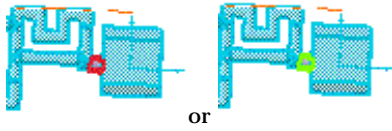
- the separation code is not used. The routing is done according to the shortest route found. It is the default value.
- the separation codes may be **File Based**. In that case, define the path to access the compatibility table by clicking the **Browse** button to choose the separation code file. This file is used during the Automatic Routing and the Routing Simulation. Two options are available:
  - **Signal-Signal** manages compatibility between signals,
  - **Signal-Arc** manages compatibility between signal and pathways.
- the separation codes may also be **Rule Based**, using CATIA Knowledgeware. Only one rule can be implemented at a time, but this rule may combine several conditions. Click the **Edit** button to enter a new rule: the **Routing Rule Editor** is displayed.



The line above the input field is a reminder of the knowledgware syntax. The **Eraser** icon is used to clear the input field.

## Extremity Color

- The **Extremity Color** field is used to change the color of the extremity boxes. Select a color from the combo box.

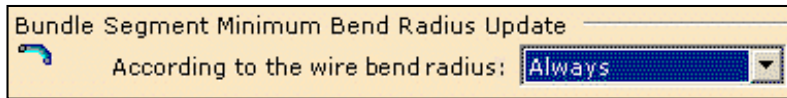


### Bundle Segment Minimum Bend Radius Update

Three modes are available:


- **Never:** no update of the bundle segment minimum bend radius is performed.
- **Always:** the greatest bend radius of the wires routed in the bundle segment determines the segment bend radius to be applied.
- **Conditional:** the wire bend radius is taken into account for update when it is greater than the bundle segment bend radius.

The default mode is **Always**.





# Customizing Electrical Harness Installation

 Before you start your first working session, you can customize the way you work to suit your habits. This type of customization is stored in permanent setting files: these settings will not be lost if you end your session.

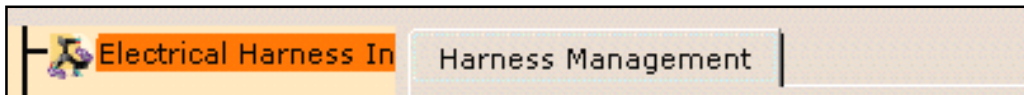


1. Select the **Tools -> Options** command.

The Options dialog box displays.

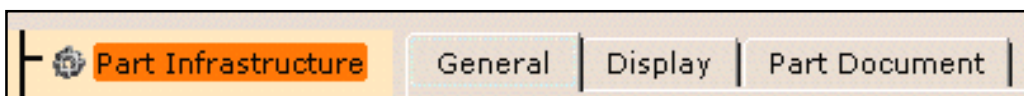
2. Choose the **Equipment & Systems** category in the left-hand box.
3. Click the **Electrical Harness Installation** workbench.

The Harness Management tab displays.

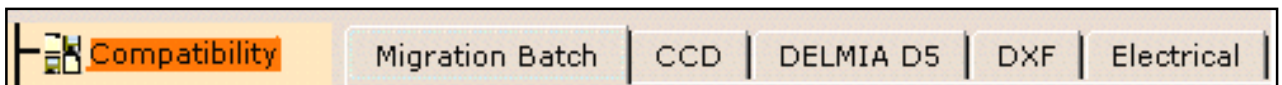


This tab lets you define:

- o the bundle segment creation options
  - o the bundle segment naming rule
  - o the bundle segment connection options
  - o the bundle segment disconnection options
  - o the management of contextual links.
4. Two other tabs, located in the **Infrastructure** category, in the **Part Infrastructure** workbench, also interfere with Electrical Harness Installation:



- o General
  - o Display
5. For **migration** purpose, you may need to customize the **Electrical** and **Migration Batch** tabs located in the **General -> Compatibility** category.



- o Migration Batch
  - o Electrical
6. Change these options according to your needs.
  7. Click **OK** when done.



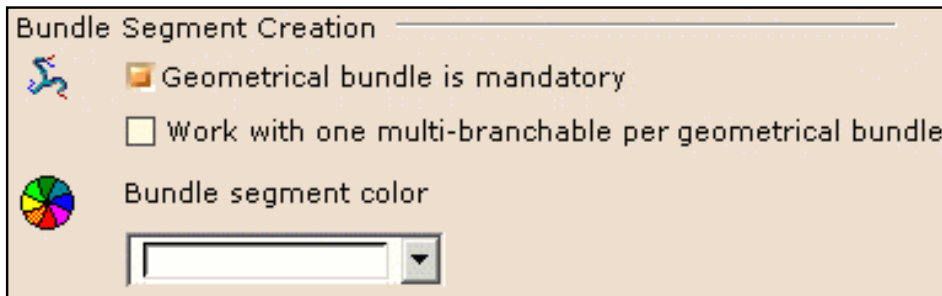
# Electrical Harness Installation

This page deals with the options concerning:

- [the bundle segment creation](#)
- [the bundle segment naming rule](#)
- [the bundle segment connection](#)
- [the bundle segment disconnection](#)
- [the management of contextual links.](#)


Click [here](#) to see the parent page.

## Bundle Segment Creation



### Geometrical bundle is mandatory

If this option is checked, it means that you need to create a geometrical bundle prior to defining a bundle segment.


 By default, this option is checked.

### Work with one multi-branchable per geometrical bundle

If this option is checked, only one multi-branchable bundle segment can be created within a geometrical bundle: a single part contains the full harness, reducing the size of the document, the number of contextual links and consequently the update time.

However to take advantage of the ENOVIA configuration management, this option needs to be unchecked if you want to manage the effectivity at a more detailed granularity level: the branchable bundle segment.

Check this option if the configuration management does not need to be done at the level of each branchable bundle segment, but at the level of a set of branchable bundle segments created in a single document.

 By default, this option is unchecked.

### Bundle segment color

Use the color chooser to define the bundle segment color at creation.

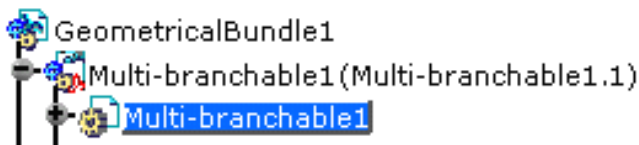
## Bundle Segment Naming Rule

### Bundle Segment Naming Rule

- Bundle segment (#PN#)
- Parent-bundle segment (#PN#)

#### Bundle segment (#PN#)

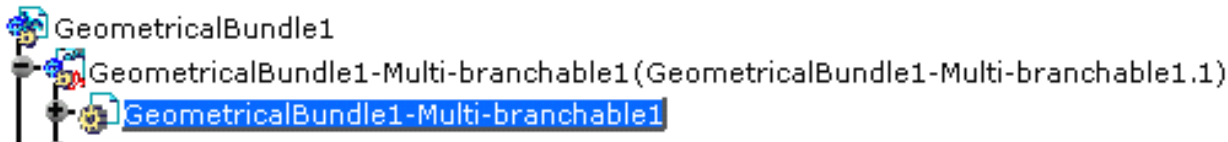
If this option is selected, the bundle segment name shows like this in the specification tree:



 It is the default value.

#### Parent-bundle segment (#PN#)

If this option is selected, the bundle segment name shows like this in the specification tree:




## Bundle Segment Connection

### Bundle Segment Connection

- Geometrical link
- Automatic link to new bundle segment


#### Geometrical link

If this option is checked, when the selection is completed, the bundle segment is extended to the second point selected (which is either a bundle segment extremity or a bundle connection point).

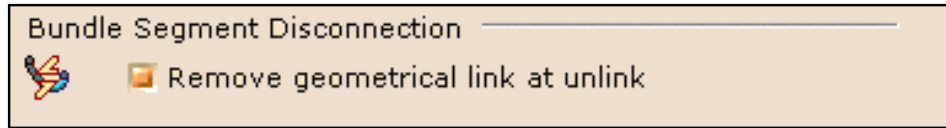
 By default, this option is checked.

#### Automatic link to new bundle segment

If this option is checked, the electrical connection is created at the same time as the bundle segment.


 By default, this option is checked.

## Bundle Segment Disconnection

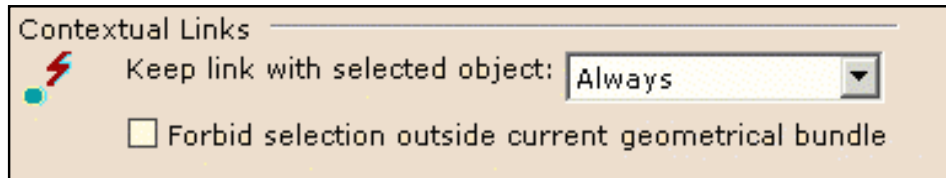


### Remove geometrical link at unlink


If this option is checked, when a bundle segment is disconnected from a connector, the geometrical link is also removed and the bundle segment geometry is modified to its last but one definition point.

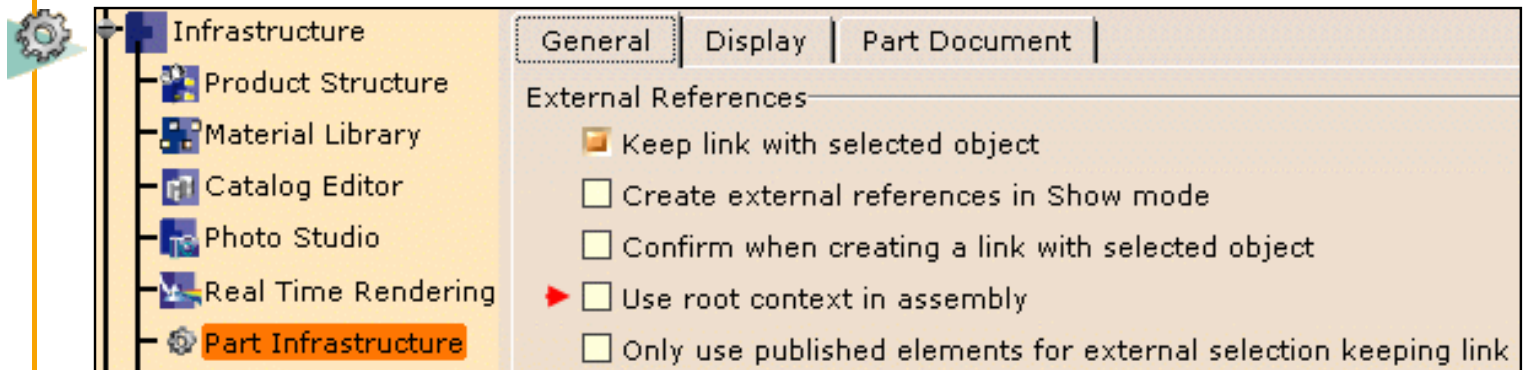
 By default, this option is checked.

## Contextual Links




For this option to be taken into account, you must first uncheck the **Use root context in assembly** option, which is located in the **Part Infrastructure -> General -> External References** frame. If not, the links will be created to the root of the assembly.

 By default, this option is checked.




### Keep link with selected object

The object created from the selection will or not keep the link according to this option. This option can take three values:

- **Always:** always creates the external reference links.  
 It is the default value.
- **In geometrical bundle:** creates external reference links only if the selected object is part of the geometrical bundle of the bundle segment.
- **Never:** always creates a local copy of the selected object with no associativity.

#### **Forbid selection outside current geometrical bundle**

If this option checked, the selection of points, supports or devices which do not belong to the current geometrical bundle for bundle segment routing is forbidden.

 By default, this option is unchecked.

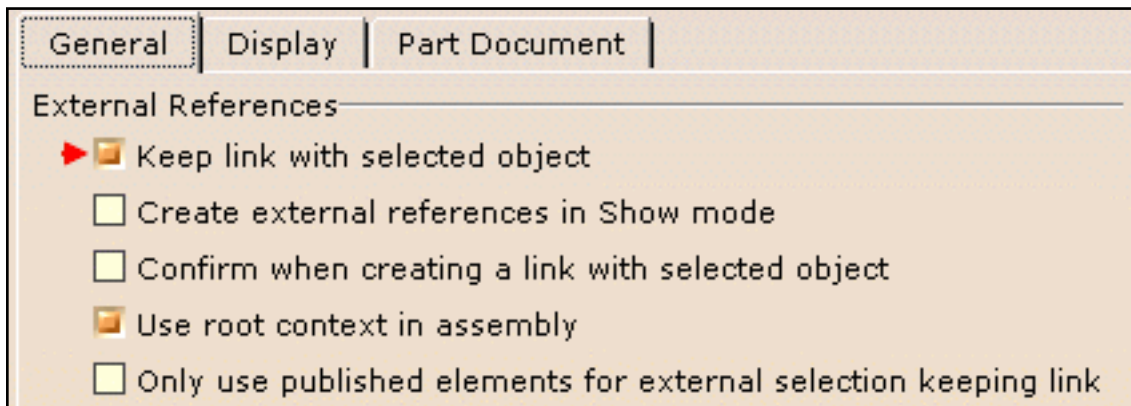
# Part Infrastructure for Electrical Harness Installation

This page deals with the options concerning:

- [the external references: keep link with selected object](#)
- [the specification tree display](#).

Click [here](#) to display the parent page.


## Part Infrastructure General option



Click [here to know more](#) about the Part Infrastructure **General** options.

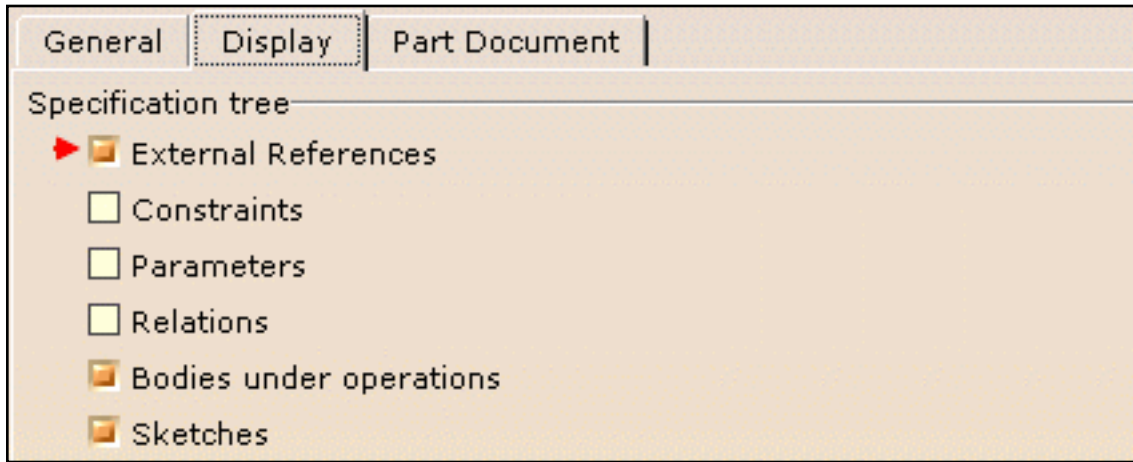
### Keep link with selected object

You need to check this option to take advantage of the associativity between the construction points or part body and the bundle segment.

 By default, this option is unchecked.

## Part Infrastructure Display option




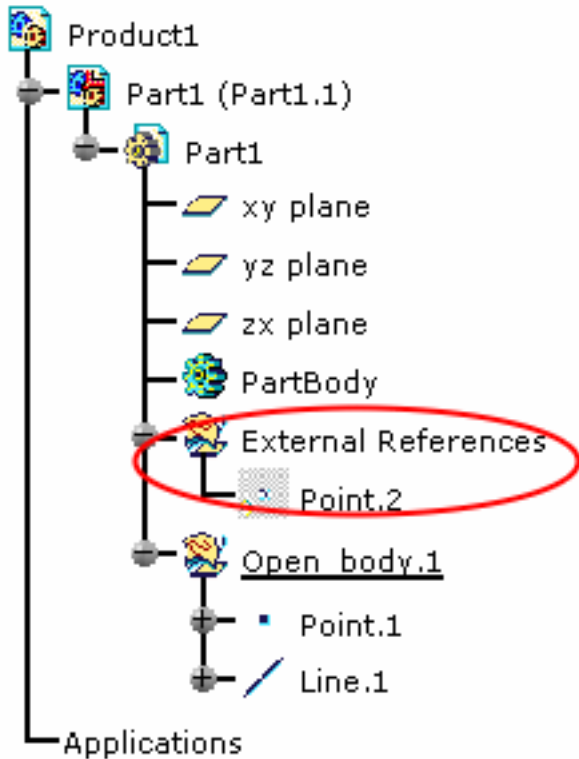


[Click here to know more](#) about the Part Infrastructure **Display** options.

### External References

If this option is checked, the **External References** will display in the specification tree.

 By default, this option is checked.



# Compatibility for Electrical Harness Installation

This page deals with the options concerning:

- [the migration batch](#)
- [the migration options](#).

Click [here](#) to display the parent page.

## Migration Batch



|                                     |
|-------------------------------------|
| Migration Interface                 |
| Interface Name <input type="text"/> |

Click [here to know more](#) about the **Compatibility -> Migration Batch** options.

### Interface Name

You need to enter **CATIE3DMigration** in this field if you want to migrate V4 electrical data to V5.

 By default, this option is empty.

## Electrical




|                                                                                                                                                                                                       |
|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Bundle Segment Migration Mode                                                                                                                                                                         |
| <input checked="" type="radio"/> Bundle segments                                                                                                                                                      |
| <input type="radio"/> Multi-branchable bundle segments                                                                                                                                                |
| Migrate V4 BNSs to V5 bundle segments.<br>Each V5 bundle segment will have its own CATPart document.<br>All these bundle segments will be instantiated in the geometrical bundle CATProduct document. |

Click [here to know all the details](#) about the **Compatibility -> Electrical** options.



## Bundle segments

If this option is checked, each V4 BNS is migrated to a V5 bundle segment (CATPart document).

 By default, this option is checked.

## Multi-branchable bundle segments

If this option is checked, all the BNSs of a V4 GBN are migrated to one multi-branchable bundle segment in a V5 geometrical bundle. All the V5 bundle segments are stored in a single CATPart document, the multi-branchable bundle segment CATPart.

# Customizing Electrical Connectivity Diagrams



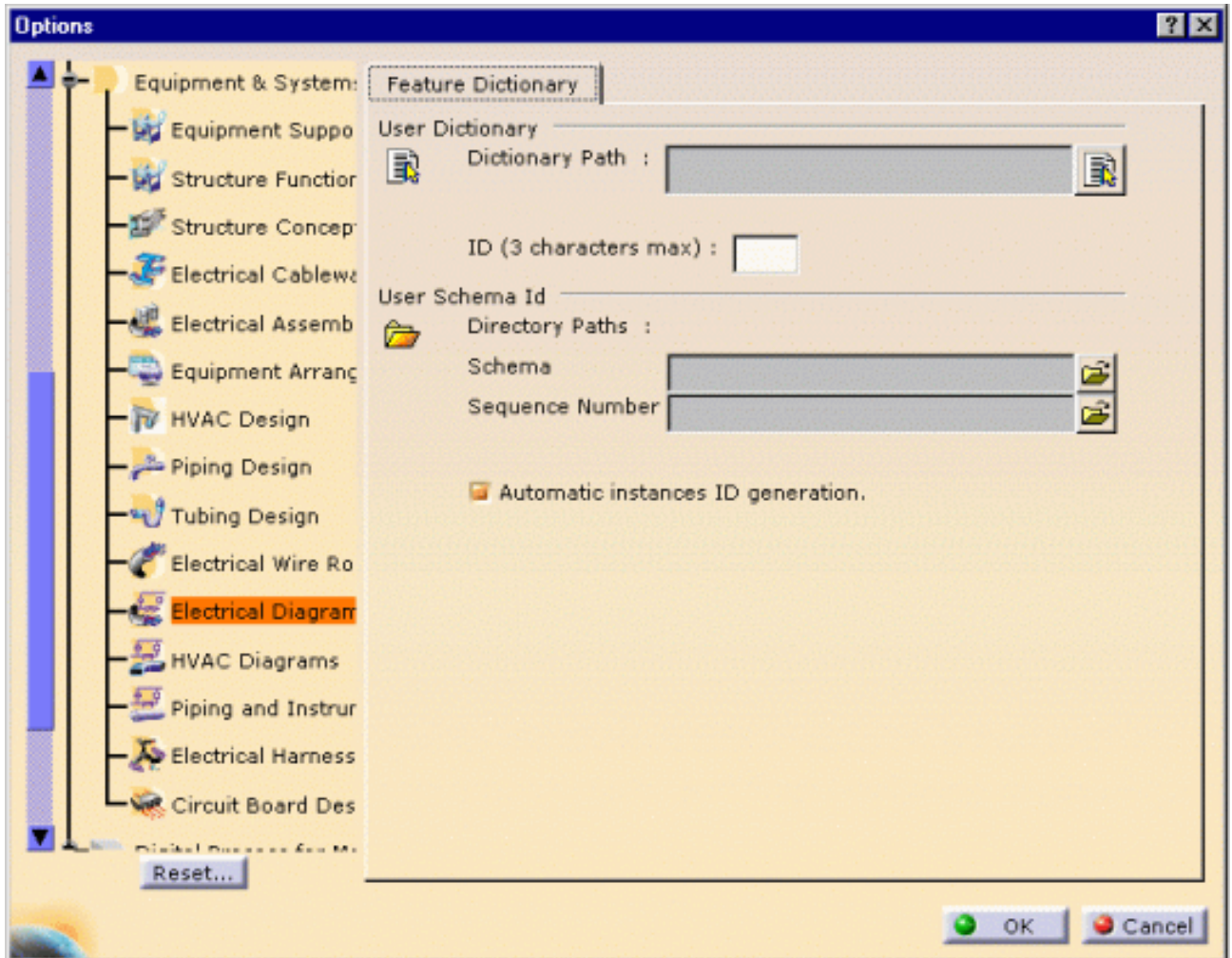
This file explains the settings specific to Electrical Connectivity Diagrams workbench. Before you start your first working session, you can customize the way you work to suit your habits. This is done using **Tools** -> **Options** from the menu bar. This type of customization is stored in permanent setting files. Settings will not be lost if you exit your session.

**Feature Dictionary Settings**  
**From-To List**

# Feature Dictionary Settings

 This file explains how to set the options for the user dictionary and schema Id.

 **1. Click Tools -> Options -> Equipment & Systems -> Electrical Connectivity Diagrams.**



The Feature Dictionary tab offers two frames:

- o User Dictionary
- o User Schema ID

**2.** Fill up the fields according to the information given below.

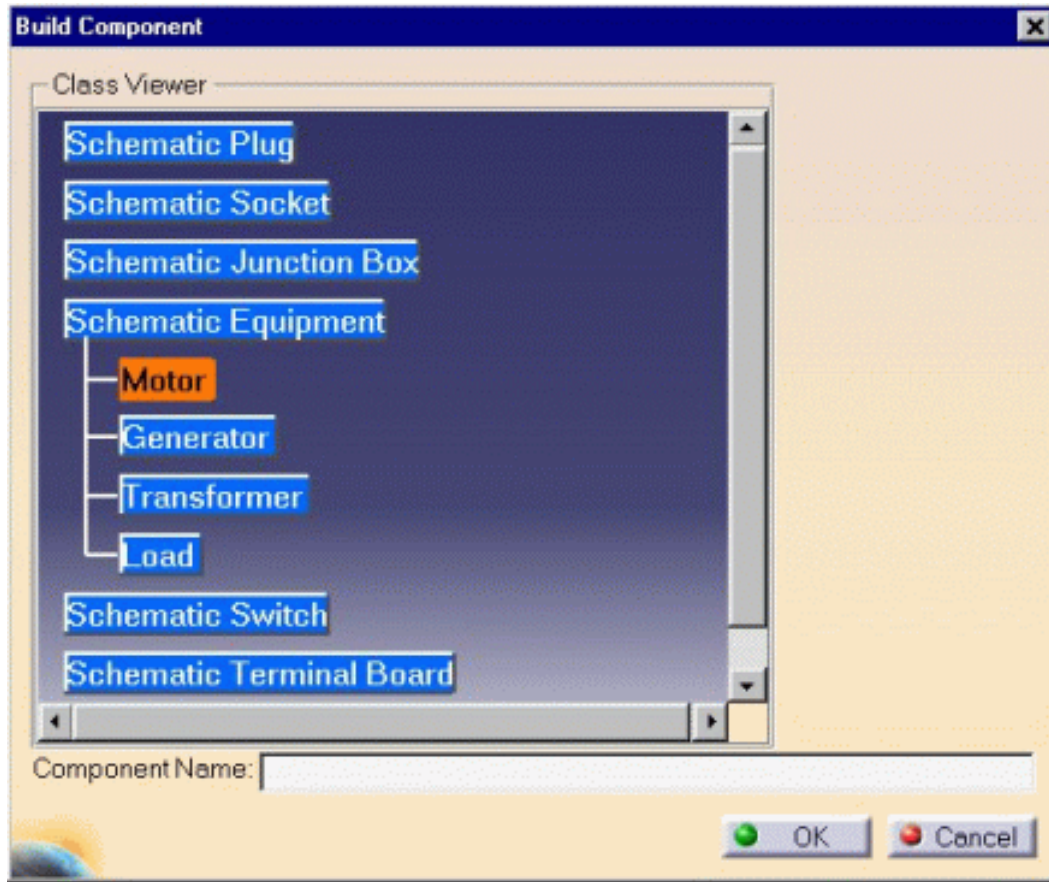
**3.** Click **OK** when you are done.

## User Dictionary



The dictionary contains object basic classes such as: equipments, sockets, plugs, switches, cables, etc. These objects have standard properties: type, part number, etc. According to you needs, you may want to define more properties such as subtype classes, and other specific attributes:

In this example, four subtype classes have been defined for the Schematic Equipment class.

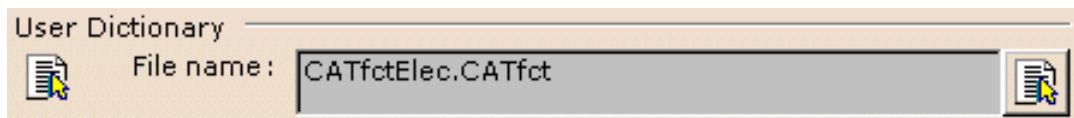


This is done using the [Feature Dictionary Editor](#) , available in the **Start -> Equipment & Systems** menu.

The classes, subclasses and their properties are saved in a document with a CATfct extension. This file must be stored in the following directory according to your operating system:

- on NT: .../intel\_a/resources/graphic/
- on Solaris: .../solaris\_a/resources/graphic/
- on Irix: .../irix\_a/resources/graphic/
- on Aix: .../aix\_a/resources/graphic/
- on Hpux: .../hpux\_a/resources/graphic/

In the **User Dictionary File Name** field, only enter the file name and extension.



You can have more than one .CATfct file, but you will reference only one at a time.

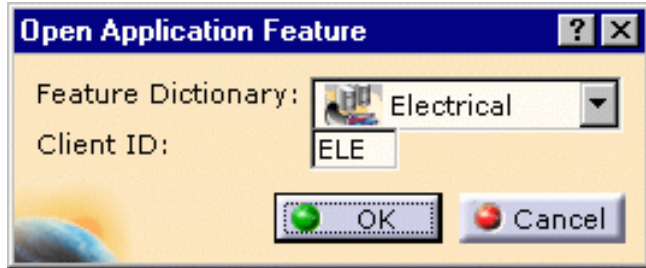


In order to access the user subclasses and properties defined in the .CATfct file, always reference an existing .CATfct file. If a .CATfct file has been used to create data, it must not be deleted to preserve the data consistency.

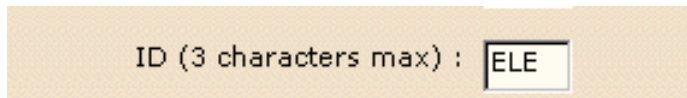
## Client ID



The Client ID is defined in the Feature Dictionary Editor as a protection on the CATfct file, before creating the user dictionary and the subclasses.



In the ID field, enter the same three characters:

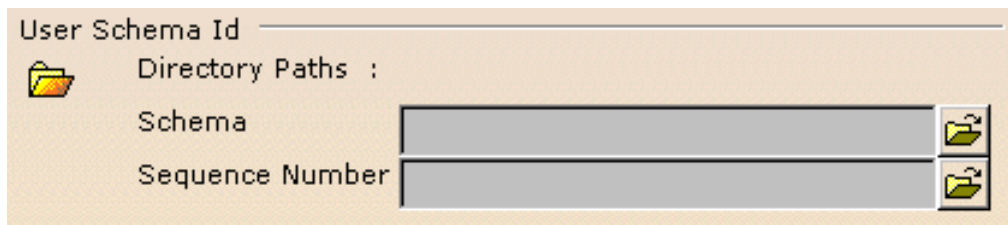


## User Schema ID



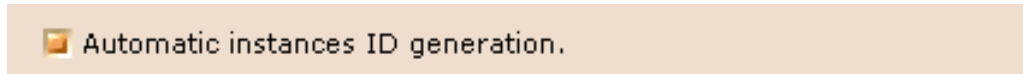
Within a project, it is possible to define naming rules for the schematic objects. They are used to set automatically the Reference Designator attribute on electrical components. These rules (**Schema ID**) are saved in xml files and stored together in specific directory. It is the path to this directory that you indicate in the first field (**Schema**) of the frame below.

The **Sequence Number** is assign automatically provided that the file server is stored in a directory shared between all the users working on the project. Enter the path to this directory in the second field (**Sequence Number**) of the frame below



You can decide then to use the naming rules you have defined or not.

In the first case, check the **Automatic instances ID generation**.



To enter manually the Reference Designator, uncheck this box.



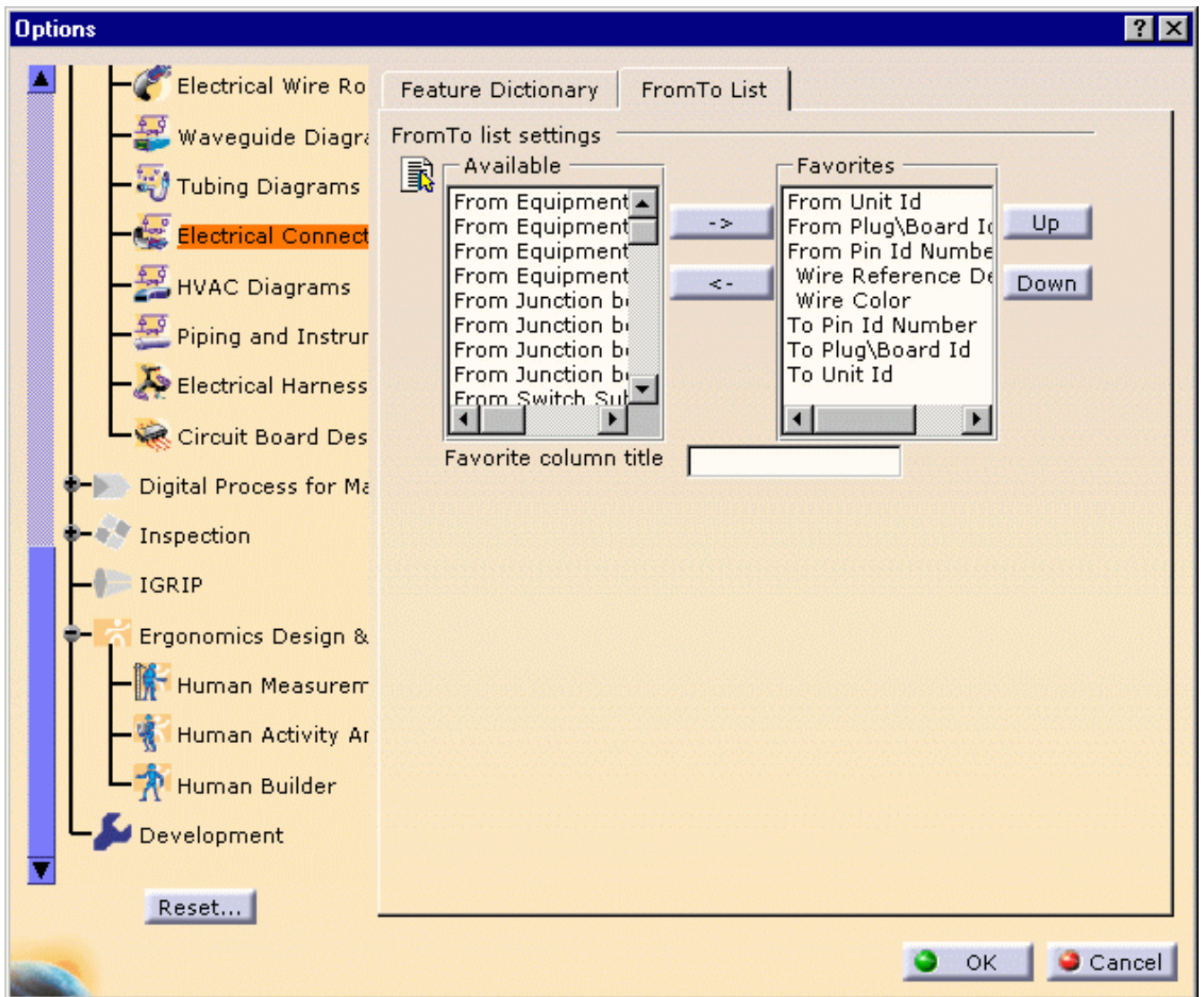
# From-To List Settings



This file explains how to set the options for the from-to list, to display the wires and the components connected to their extremities.



1. Click **Tools** -> **Options** -> **Equipment & Systems** -> **Electrical Connectivity Diagrams**.



The From-To List tab lets you define which attributes will be displayed in the list.

Favorites attributes are selected by default.

If other attributes have been defined using the Feature Dictionary Editor, they will be available in the left column.

2. Select the attributes of interest in the left column and click the right arrow.  
The attributes shown in the right column will be displayed in the list.
3. Use the **Up** and **Down** buttons to reorder the list.
4. Enter a name in the **Favorite column title** to change the title of this column in the list.
5. Click **OK** when you are done.

## Detailing the From-to list



**Wire Management** [?] [X]

Selected Cable: C3

| From Unit Id | From Plug\Board Id | From Pin Id Number | Wire Reference Des... | Wire Color | To Pin Id Number | To Plug\Boar... | To Unit Id |
|--------------|--------------------|--------------------|-----------------------|------------|------------------|-----------------|------------|
| Power1       | PowerSocket2       | Id Number-2        | P1                    | Blue       | Id Number-6      |                 | Fuse1      |
| Power1       | PowerSocket2       | Id Number-3        | P2                    | Red        | Id Number-7      |                 | Fuse1      |
| Power1       | PowerSocket2       | Id Number-4        | P3                    | Green      | Id Number-8      |                 | Fuse1      |

From

- Id Number-2
- Id Number-3
- Id Number-4

Disconnect

To

- Id Number-3
- Id Number-4
- Id Number-5
- Id Number-6
- Id Number-7
- Id Number-8

Disconnect

Close

Each row displays a wire and the components connected to its extremities.

**Unit Id**

It is the reference designator of the component (i.e. equipment, junction box or switch).

**Plug\Board Id**

It is the reference designator of the connector on the board or plug.

**Pin Id Number**

It is the Id Number of the pin.

**Wire Reference Designator**

It is the reference designator of the wire.

**Wire Color**

it is the color code of the wire.



# Customizing Equipment Support Structures



Before you start your first session, you can customize the way you work to suit your habits. This type of customization is stored in permanent setting files: these settings will not be lost if you end your session.



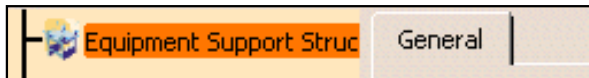
1. Select the **Tools** -> **Options** command.

The Options dialog box displays.

2. Select the **Equipment & Systems** category in the left-hand box.

3. Select the **Equipment Support Structures** sub-category.

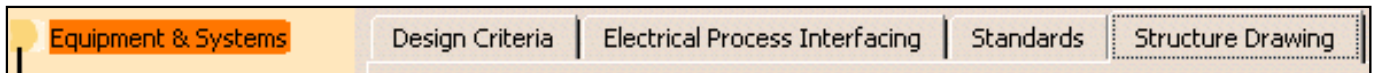
The General tab displays.



This tab lets you define:

- o **Color** options: default color for members and plates
- o **Material** options: default materials for members and plates
- o **User types**: create and manage member and plate user types
- o **Catalogs storage** path: path of the folder containing sample standard catalog sections
- o **Sections storage** path: path of the folder that will contain the list of available sections
- o **Thickness table** path

One other tab, **Structure Drawing**, located in the **Equipment & Systems** category, is also needed for Equipment Support Structures.



- o The **Structural Drawing** tab lets you customize how data extracted from the 3D document is presented on structural drawings.
4. Set options in these tabs according to your needs.
  5. Click **OK** in the Options dialog box when done.





# Equipment Support Structures

The Equipment Support Structures tab contains the following categories of options:


- [Color](#)
- [Material](#)
- [User types](#)
- [Catalogs storage](#)
- [Sections storage](#)
- [Thickness table](#)

## Color

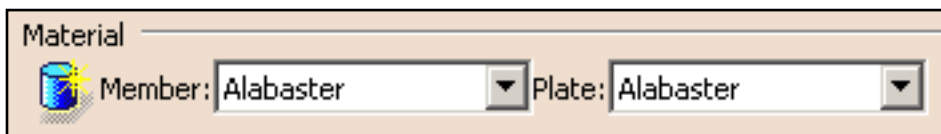


### Member or Plate

Select the default color for members and plates in the appropriate box list.

 By default, colors are as shown above.

## Material

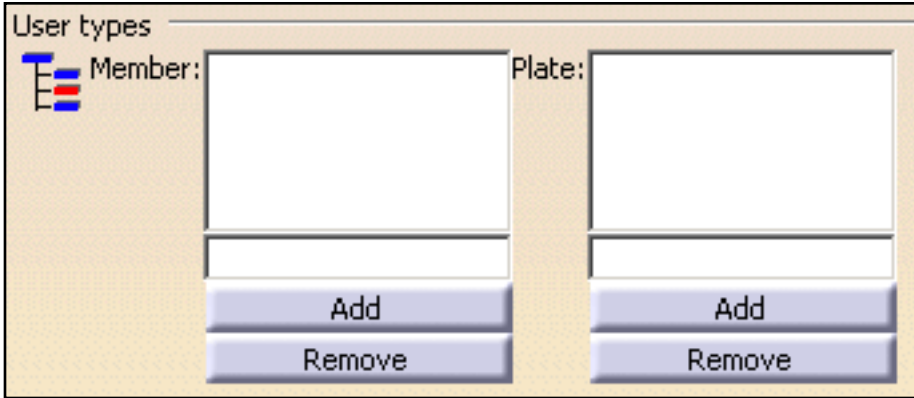


### Member or Plate

Select the default material for members and plates in the appropriate box list.

 By default, materials are as shown above.

## User types



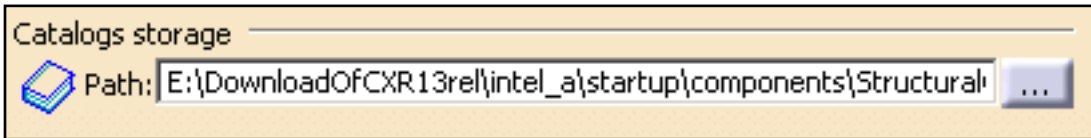
### Member or Plate

Lets you create and manage user types:

- Add: enter the name of the member or plate type then click Add to add user types.
- Remove: select the member or plate type in the list then click Remove to remove user types.

 By default, these options are blank.

## Catalogs storage




### Path

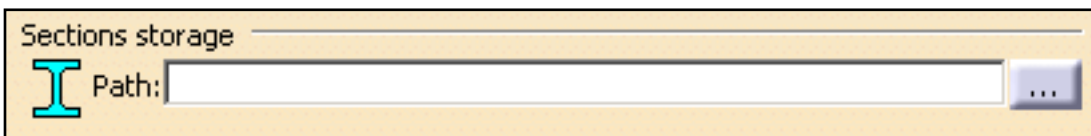
Enter the path of the folder containing sample standard catalog sections supplied with the product:

`downloadirectory/OS/startup/components/structuralcatalogs`

where OS is the operating system, for example intel\_a (Windows).

 By default, this option is left blank.

## Sections storage



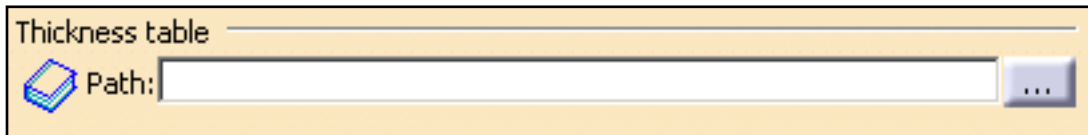
### Path

Enter the full path of the folder that will contain the list of available sections, for example **e:\Section**, or use [...] to locate the folder of interest.

This folder will contain resolved user-defined sections stored here directly and any sample standard or user-defined catalog sections accessed via the Section command or the Other section... option. It is recommended that an empty directory be identified.

 By default, this option is left blank.

## Thickness table




### Path

Enter the path of the thickness table. A sample table, ThicknessListSample.txt, is supplied with the product.

`downloaddirectory/OS/startup/components/StructuralCatalogs/Thickness Tables/ThicknessListSample.txt`

where OS is the operating system, for example intel\_a (Windows).

 By default, this option is left blank.



# Customizing Electrical 3D Design & Documentation



Before you start your first working session, you can customize the way you work to suit your habits. This type of customization is stored in permanent setting files: these settings will not be lost if you end your session.

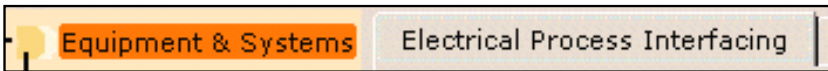


1. Select the **Tools** -> **Options** command.

The Options dialog box displays.

2. Choose the **Equipment & Systems** category in the left-hand box.
3. Click the **Electrical 3D Design** workbench.

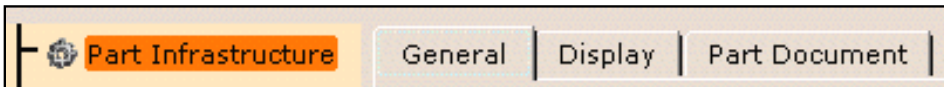
The options for 3D Design settings appear, organized in tab pages.



4. Select the tab corresponding to the parameters to be customized.

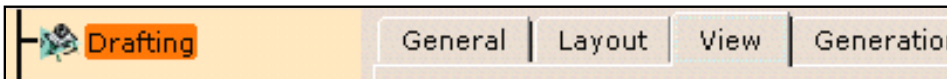
| Parameters in this tab...                      | Allow you to customize...             |
|------------------------------------------------|---------------------------------------|
| <a href="#">General</a>                        | the automatic compass                 |
| <a href="#">Harness Flattening</a>             | the synchronization report repository |
| <a href="#">Harness Management</a>             | the bundle segment creation options   |
| <a href="#">Wire Routing</a>                   | the options to optimize the routing   |
| <a href="#">Electrical Process Interfacing</a> | the external data file repository     |

5. A tab located in the **Infrastructure** category, in the **Part Infrastructure** workbench, also interfere with Electrical 3D Design:



- o [General](#)

6. Another tab located in the **Mechanical Design** category, in the **Drafting** workbench, also interfere with Electrical 3D Design:



- o [View](#)

7. Change these options according to your needs.
8. Click **OK** when done.



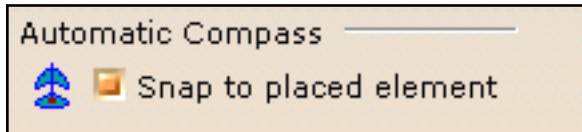
# General

This page deals with the options concerning:

- [the automatic compass](#)

Click [here](#) to see the parent page.


## Automatic Compass



### Snap to placed element

if this option is checked, you take advantage of the compass:

when you place an element, the compass snaps to this element allowing you to modify the orientation and location.

 By default this option is checked.

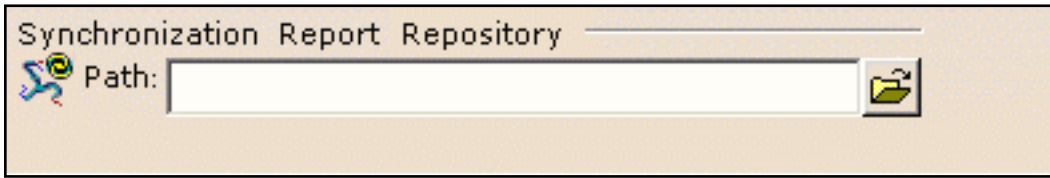
# Harness Flattening

This page deals with the options concerning:

- [the synchronization report repository](#)
- [the graphic replacement catalog](#)


Click [here](#) to display the parent page.

## Synchronization Report Repository

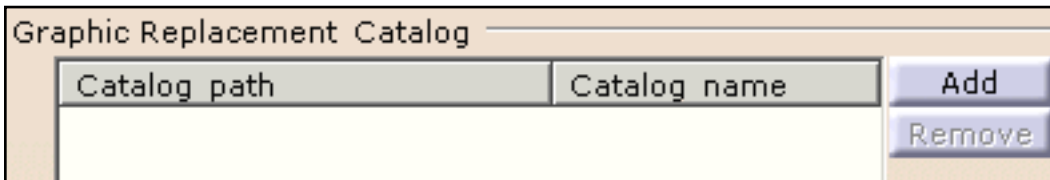


### Path

Indicate the folder path where you want to save the synchronization report html files.

 By default, this field contains: ...\DassaultSystemes\CATTemp.

## Graphic Replacement Catalog



| Catalog path | Catalog name |
|--------------|--------------|
|              |              |

### Catalog path and Catalog name

Indicate the catalog path and name where the drawing 2D details are stored.

 By default, this field is empty.

# Harness Management

This page deals with the options concerning:


- [the bundle segment creation](#)

Click [here](#) to see the parent page.

## Bundle Segment Creation



Bundle Segment Creation

 Bundle segment color

### Bundle segment color

Use the color chooser to define the bundle segment color at creation.

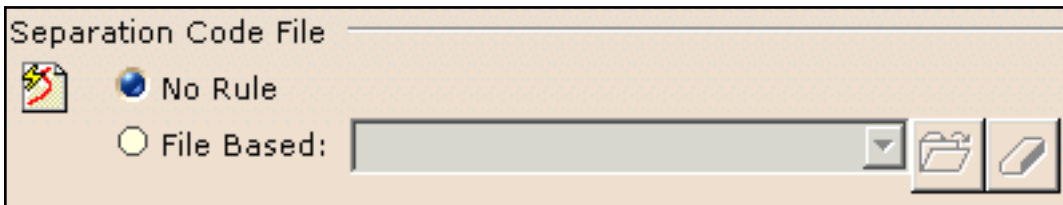
# Wire Routing

This page deals with the options concerning:

- [the separation code file](#)
- [the bundle segment bend radius update](#)

Click [here](#) to see the parent page.


## Separation Code File



This option is used to define separation code rules to optimize the routing.

### No rule

The separation code is not used. The routing is done according to the shortest route found.

 It is the default value.

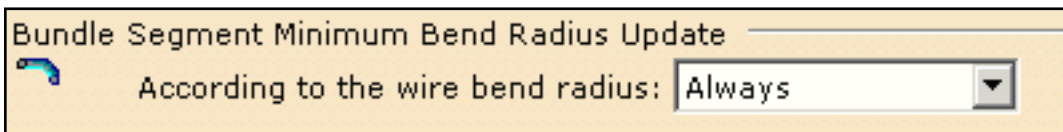
### File based

The separation codes may be **File Based**.

In this case, define the path to access the compatibility table by clicking the **Browse** button to choose the separation code file.

This file is used during the automatic routing.

## Bundle Segment Bend Radius Update




### According to the wire bend radius



Three modes are available:

- **Never:** no update of the bundle segment minimum bend radius is performed.
- **Always:** the greatest bend radius of the wires routed in the bundle segment determines the segment bend radius to be applied.
- **Conditional:** the wire bend radius is taken into account for update when it is greater than the bundle segment bend radius.

 The default mode is **Always**.

# Electrical Process Interfacing

This page deals with the options concerning:

- [the electrical iXF repository path](#)

Click [here](#) to see the parent page.

## Electrical iXF Data Repository



Electrical iXF Data Repository

### System repository

You are required to identify the path of the folder in which the XML files available are stored.

Use to locate the iXF systems repository.

By default the field is empty.



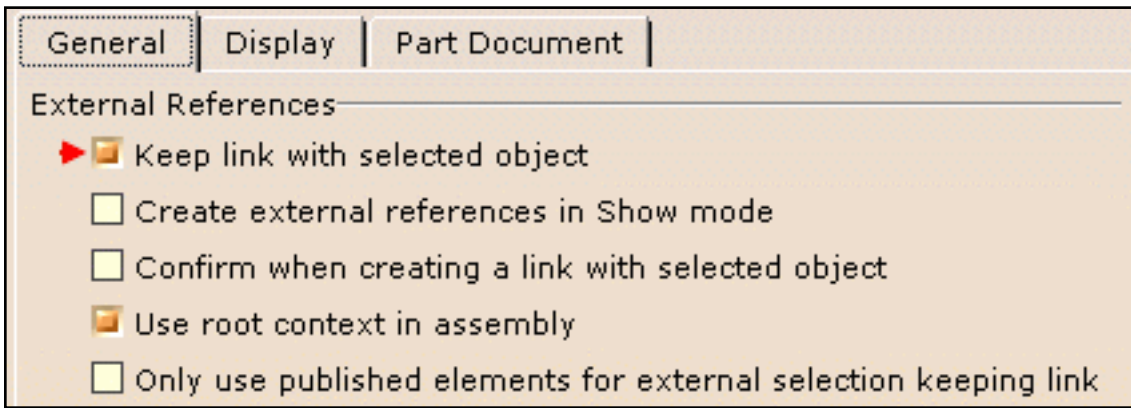
# Part Infrastructure for Electrical 3D Design

This page deals with the options concerning:

- [the external references: keep link with selected object](#)

Click [here](#) to display the parent page.


## Part Infrastructure General option



Click [here to know more](#) about the Part Infrastructure **General** options.

### Keep link with selected object

You need to check this option to take advantage of the associativity between the construction points or part body and the bundle segment.

 By default, this option is unchecked.

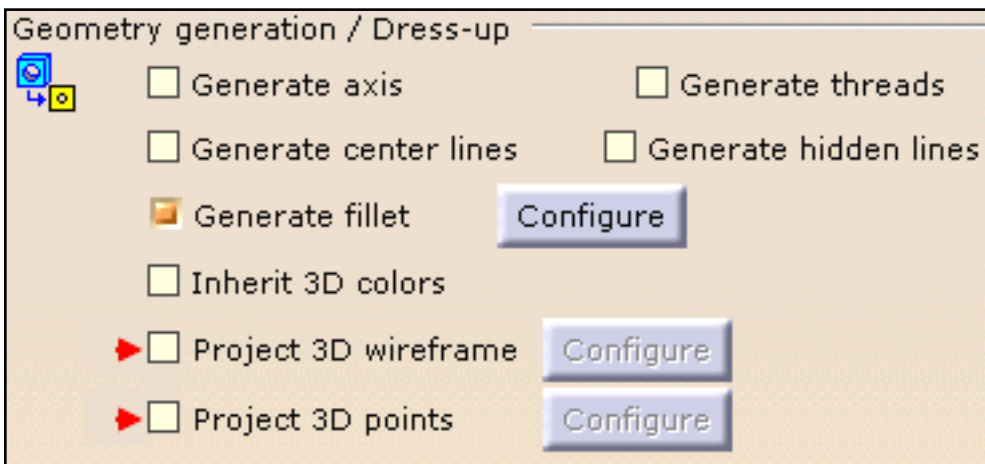
# Drafting for Electrical 3D Design

This page deals with the options concerning:

- [Geometry generation / Dress-up](#)

Click [here](#) to display the parent page.


## Drafting View options



[Click here to know more](#) about the Drafting **View** options.


### Project 3D wireframe

You need to check this option to take advantage of the line type personalization used to display the bundle segments as single line in the drawings you will generate.

 By default, this option is unchecked.

### Project 3D points

You need to check this option to display construction points in the drawings you will generate..

 By default, this option is unchecked.

# Structure Drawing

The Structure Drawing tab contains the following categories of options:


- **Molded forms:** to define how molded forms will be represented on drawings.
- **Members/Stiffeners:** to define how members or stiffeners will be represented on drawings.
- **Plates:** to define how plates will be represented on drawings.
- **Openings:** to define how openings will be represented on drawings
- **Midship designation**
- **Datum line**

## Molded forms




### Linetype

Defines the linetype used to represent molded forms on drawings. Select the desired linetype from the drop-down list.

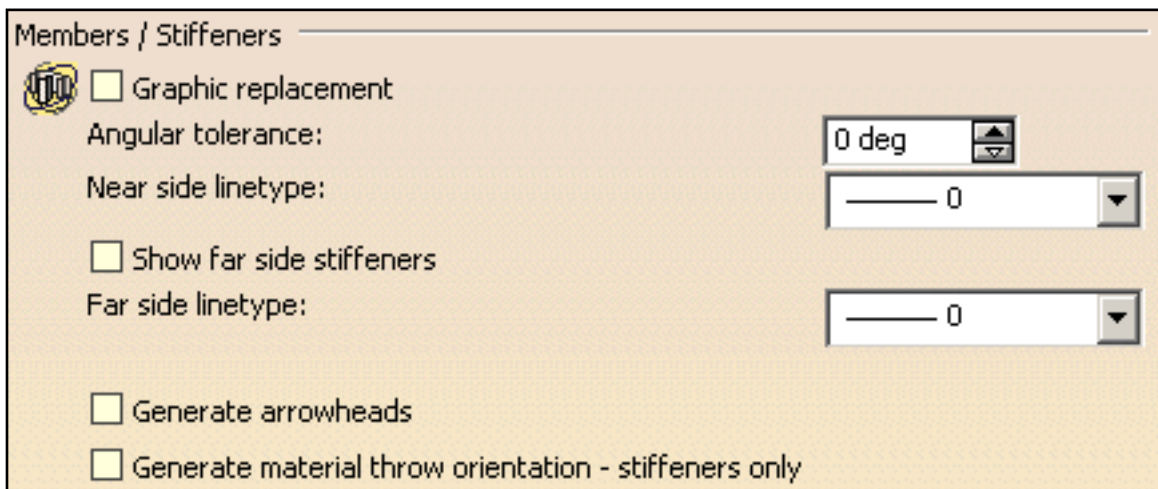
 By default, this option is set to 0.

### Apply linetype on profile view only

If this option is selected, the selected linetype is applied to profile views only.

 By default, this option is cleared.

## Members / Stiffeners



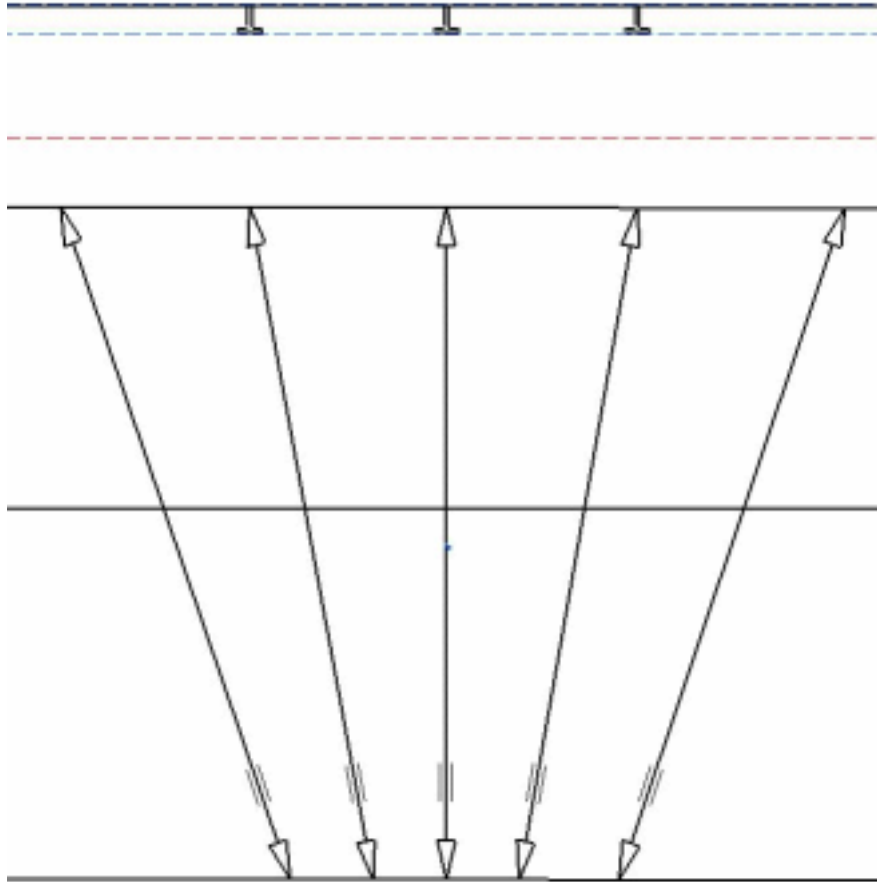
### Graphic replacement

If this option is selected, a line representing the trace of visible members / stiffeners is generated on drawings. The user can specify the linetype (near side linetype) for this line.

 By default, this option is cleared and the 3D representation of members / stiffeners is generated on drawings.

### Angular tolerance

Defines the value above which the member/ stiffener section is not represented. It is the angle between the member/ stiffener section and the projection vector. In the example below, the angular tolerance is 15.




### Near side linetype

Defines the linetype to be used for the trace of visible members / stiffeners if the Graphic replacement option is selected. Select the desired linetype from the drop-down list. A solid line font is recommended.

### Show far side stiffeners

If this option is selected, a line representing the trace of hidden members / stiffeners, below a deck for example, is generated on drawings. The user can specify the linetype (far side linetype) for this line.


 By default, this option is cleared and hidden members / stiffeners are not shown on drawings.

### Far side linetype

Defines the linetype to be used for the trace of hidden members / stiffeners if the Show far side stiffeners option is selected. Select the desired linetype from the drop-down list. A phantom font is recommended.

### Generate arrowheads

If this option is selected, arrows are generated at ends of members / stiffeners.

 By default, this option is cleared and no arrowheads are shown on drawings.

### Generate material throw orientation - stiffeners only

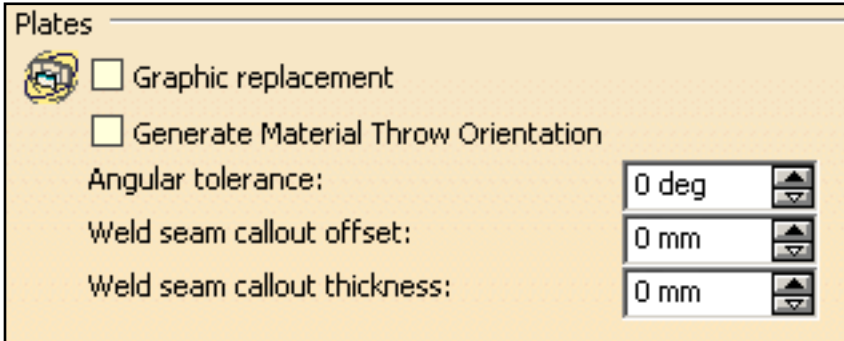
If this option is selected, marks indicating the material throw orientation of stiffeners are generated.



Note: This option is valid for stiffeners only.

By default, this option is cleared and no marks are generated on drawings.

## Plates



### Graphic replacement

If this option is selected, you can define how plates will be represented on drawings.

By default, this option is cleared and the 3D representation of plates is generated.

### Generate Material Throw Orientation

If this option is selected, marks indicating the material throw orientation of plates are shown on drawings.

By default, this option is cleared.

### Angular tolerance

Specifies an angular tolerance. If the angle between plates is less than the specified value, weld seams are generated; if the angle is greater than or equal to the specified value, knuckle lines are generated.

By default, this option is set to 0 degrees.

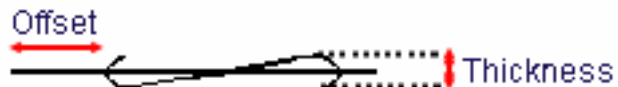
### Weld seam callout offset

Specifies the offset for weld seam callouts.

By default, this option is set to 0 mm.

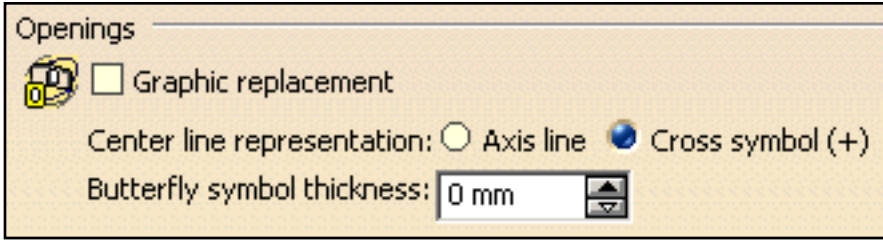
### Weld seam callout thickness

Specifies the thickness for weld seam callouts.



By default, this option is set to 0 mm.

## Openings



### Graphic replacement

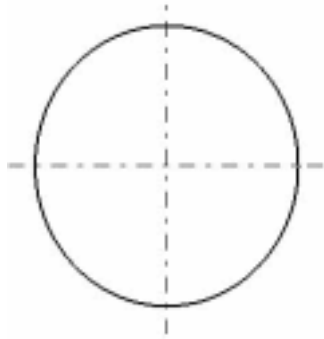
If this option is selected, the user can define the symbols to use to represent openings on drawings.

By default, this option is cleared and the 3D representation of openings is generated.

### Center line representation

Defines the symbol used for the centerline on front views if the Graphic replacement option is selected. Two options are available:

- Axis line



- Cross symbol: the centerline is represented by a cross.

By default, the axis line is used.

### Butterfly symbol thickness

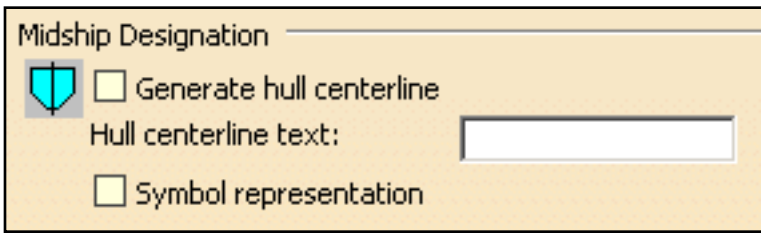
Defines the thickness of the butterfly symbol in mm. Applies to profile views.



By default, this option is set to 0 mm.

## Midship Designation





Midship Designation

Generate hull centerline

Hull centerline text:

Symbol representation

### Generate hull centerline

If this option is selected, the hull centerline is generated on drawings.

 By default, this option is cleared.

### Hull centerline text


Defines the text to be placed at both ends of the centerline.

 By default, this field is blank.

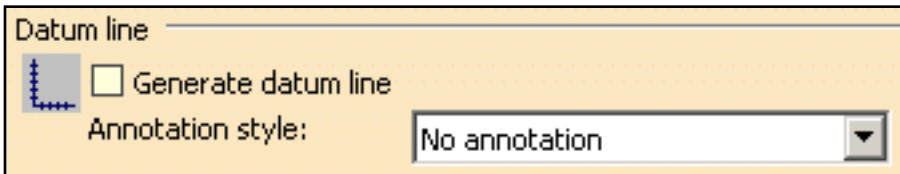
### Symbol representation

If this option is selected, the midship symbol is generated on drawings.

Note that dimensions of this symbol do not change.

 By default, this option is cleared.

## Datum Line




Datum line

Generate datum line

Annotation style:

### Generate datum line


If this option is selected, a datum line is generated on drawings.

 By default, this option is cleared.

### Annotation style

Defines the annotation style to use for the datum line. Options available are:

- No annotation
- Names from 3D: specifies frame names
- Position values: specifies the offset from the origin (units are indicated)
- Position values without units: specifies the offset from the origin.

 By default, this option is set to No annotation.



# Customizing Structure Preliminary Layout



Before you start your first session, you can customize the way you work to suit your habits. This type of customization is stored in permanent setting files: these settings will not be lost if you end your session.



1. Select the **Tools -> Options** command.

The Options dialog box displays.

2. Select the **Equipment & Systems** category in the left-hand box.
3. Click the **Structure Preliminary Layout** sub-category.

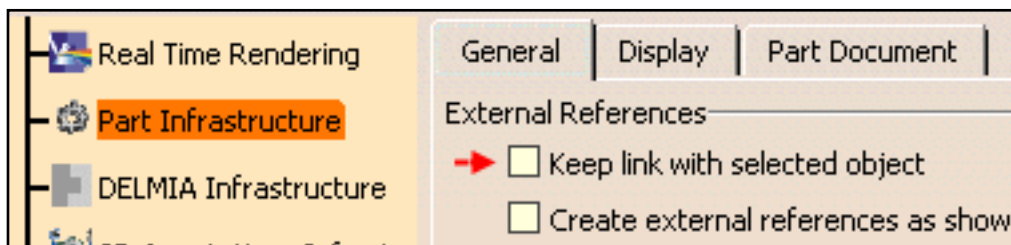
The Design, Drafting and Offset Data tabs display.



- o The **Design** tab lets you set default colors for main items.
- o The **Drafting** tab lets you customize lines drawings.
- o The **Offset Data** tab lets you manage the output formats for your offset data.

One other setting, **Keep link with selected object**, is also needed to ensure associativity between the items you create and entities selected to create them.

4. Select **Infrastructure -> Part Infrastructure -> General**, then select **Keep link with selected object**.



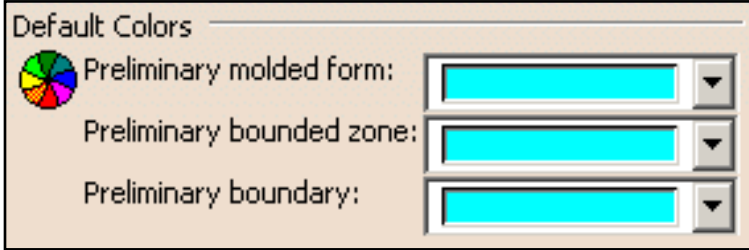
5. Set options in these tabs according to your needs.
6. Click **OK** in the Options dialog box when done.




# Design Settings

The Design tab contains one category of options: Default colors.

## Default Colors



Use the color chooser to define the default color for **molded forms**, **bounded zones** (compartments) and **boundaries**.

 By default, colors are as shown above.

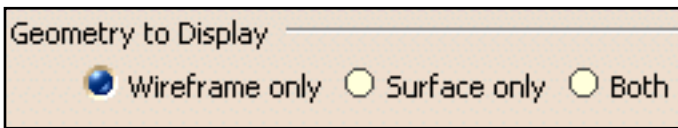


# Drafting

The Drafting tab contains five categories of options letting you customize your lines drawings:

- [Geometry to display](#)
- [Waterlines view](#)
- [Waterline position](#)
- [Body plan](#)
- [Midship position](#)

## Geometry to Display



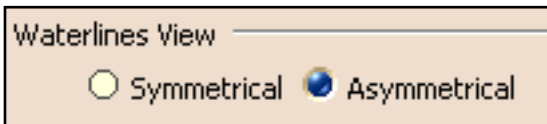
Select the option defining how the shape of the ship is shown on lines drawings: **Wireframe only** or **Surface only**, or **Both**.

 By default, the option is Wireframe only.



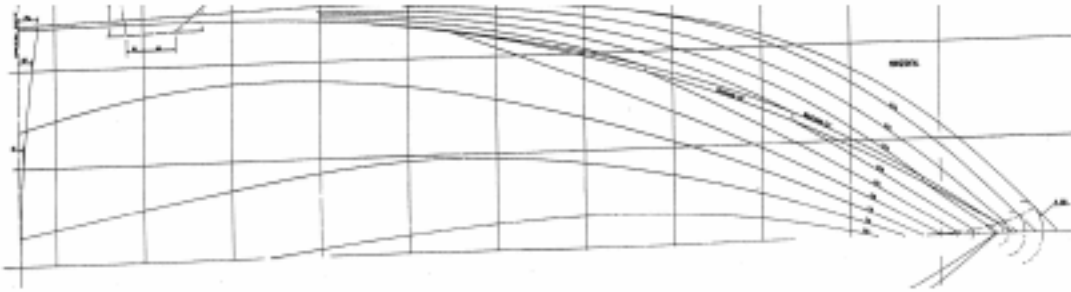
For a body plan, ensure Wireframe only is checked.

## Waterlines View



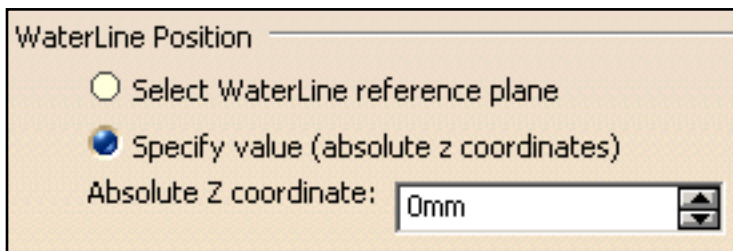
Select the option defining the view looking at the ship from above: **Symmetrical** or **Asymmetrical** depending on whether or not the ship is symmetrical about the centerline. This generates a half-breadth or full-breadth view respectively.

Typical half-breadth view:



 By default, the option is Asymmetrical.

## Waterline Position



### Select WaterLine reference plane

If this option is selected, the waterline is located at the selected reference plane in the 3D document.

### Specify value (absolute z coordinates)

If this option is selected, the waterline is located along the Z direction at the absolute coordinate entered in the Absolute Z coordinate box.

 By default, the option is Specify value.

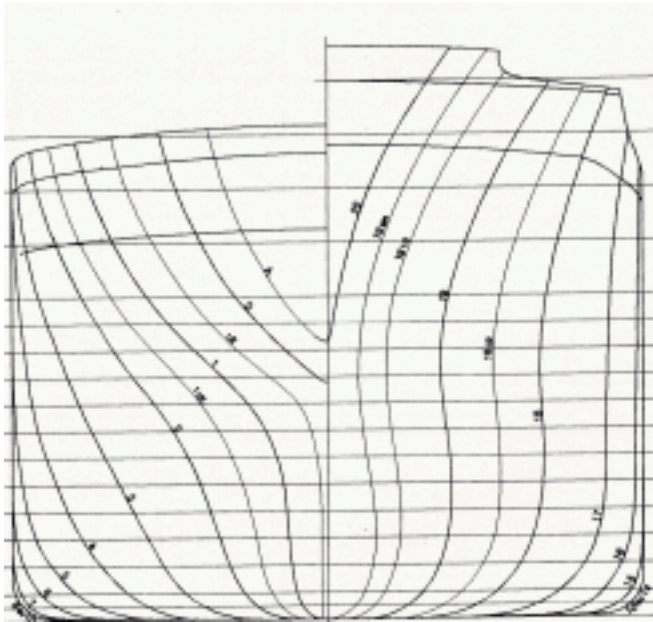
## Body Plan



Select the option defining the view showing the shapes of the frame lines: **Symmetrical** or **Asymmetrical** depending on whether or not the ship is symmetrical about the centerline.

The generated view is made of two parts. If the ship is symmetrical, only one side of the ship is shown (usually the portside) and the right-hand part looks aft at the forward portside of the ship, while the left-hand part looks forward at the after half of the portside. If the ship is asymmetrical, both sides of the ship are shown.

Typical body plan:



To generate this drawing, you need to define the midship position (see below).

By default, the option is Asymmetrical.

## Midship Position

Midship Position

Select middle station

Specify value (absolute x coordinates)

Absolute X coordinate:

### Select middle station

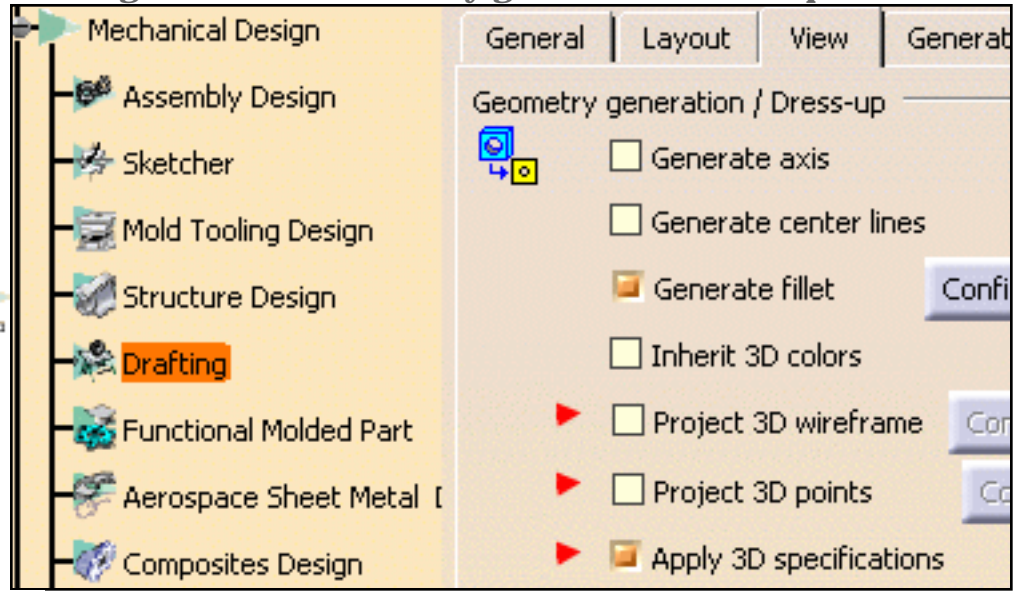
If this option is selected, the midship is located at the selected reference plane in the 3D document.

### Specify value (absolute xcoordinates)

If this option is selected, the midship is located along the X direction at the absolute coordinate entered in the Absolute X coordinate box.

By default, the option is Specify value.

For the options in this tab page to be taken into account, you need to select **Project 3D wireframe**, **Project 3D points** and **Apply 3D specifications** located in **Mechanical Design -> Drafting -> View -> Geometry generation/Dress-up frame**.



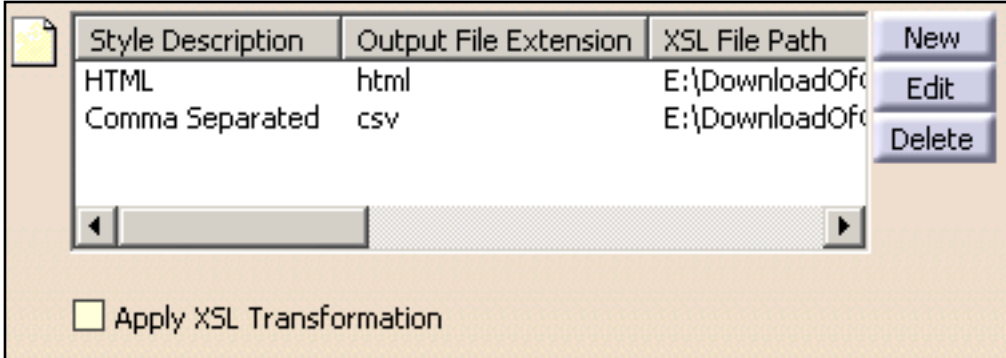
By default, **Project 3D wireframe** and **Project 3D points** are cleared, and **Apply 3D specifications** is selected.



# Offset Data

The Offset Data tab contains one category of options to manage output formats, for example HTML pages or CSV files, for your offset data: Global.

## Global




### New / Edit / Delete

Use **New**, **Edit** or **Delete** to add new stylesheets, and edit or delete existing ones.

Sample stylesheets (OffsetDataAsHTML.xls and OffsetDataAsCSV.xls) for two formats, HTML and CSV respectively, are provided. By default, they are located in the installation folder under [OS/startup/EquipmentAndSystems/StructurePreliminaryLayout/StyleSheet](#).

### Apply XSL Transformation

Select this option to take all stylesheets into account and choose the format of your output data when interactively generating your offset data.

 By default, this option is cleared and only XML files will be generated.





# Customizing Structure Functional Design



Before you start your first session, you can customize the way you work to suit your habits. This type of customization is stored in permanent setting files: these settings will not be lost if you end your session.



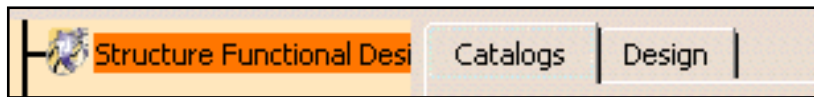
1. Select the **Tools** -> **Options** command.

The Options dialog box displays.

2. Select the **Equipment & Systems** category in the left-hand box.

3. Click the **Structure Functional Design** sub-category.

The Catalogs and Design tabs display.



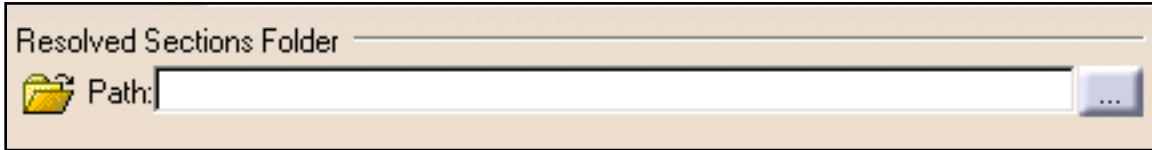
- The **Catalogs** tab lets you identify the path to the resolved sections folder.
  - The **Design** tab lets you define default colors and a default material-grade combination.
4. Set options in these tabs according to your needs.
  5. Click **OK** in the Options dialog box when done.



# Catalog Settings

The Catalogs tab contains only one category of options: Resolved Sections Folder.


## Resolved Sections Folder



### Path

Enter the path of the folder in which the list of available sections will be stored, for example **e:\Section**, or use [...] to locate the folder of interest.

This directory will contain resolved user-defined sections which are stored here directly as well as any standard or user-defined catalog sections selected via the Section icon or the Other section... option. It is recommended that you identify an empty directory.

 By default, this option is left blank.

Note: Other catalogs, such as the standard section catalog, thickness table and opening catalog are project resources and can be browsed via the **Project Management** command (**Tools** menu).

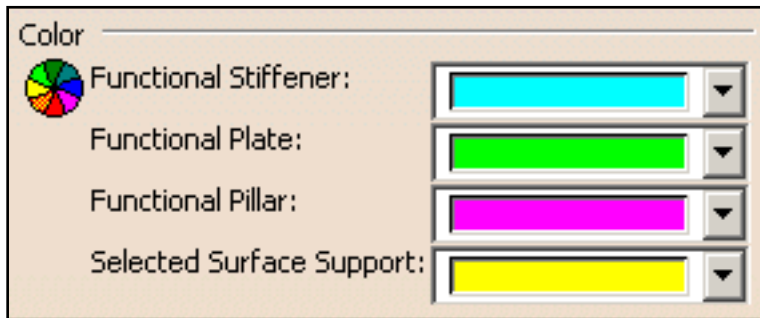


# Design Settings


The Design tab contains the following categories of options:

- [Color](#)
- [Material](#)

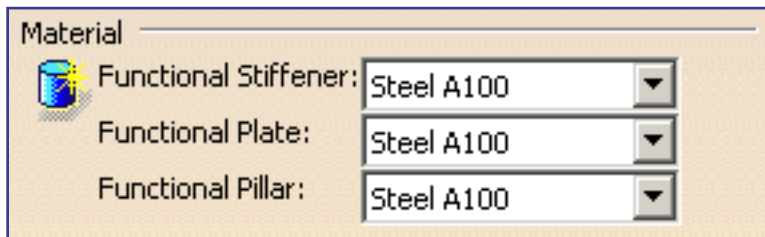
## Color



Use the color chooser to define the default color for **functional stiffeners**, **functional plates**, **functional pillars** and the **selected support surface**.

 By default, colors are as shown above.

## Material



Select the default material-grade combination for **functional stiffeners**, **functional plates** and **functional pillars** from the corresponding list.

This is the combination proposed when creating functional plates, insert plates, stiffeners, twisted profiles and pillars if no default value has been specified in the Feature dictionary.

 By default, the material-grade combination is Steel A100.



# Customizing Circuit Board Design



This task shows you how to change the user settings. These are permanent settings.



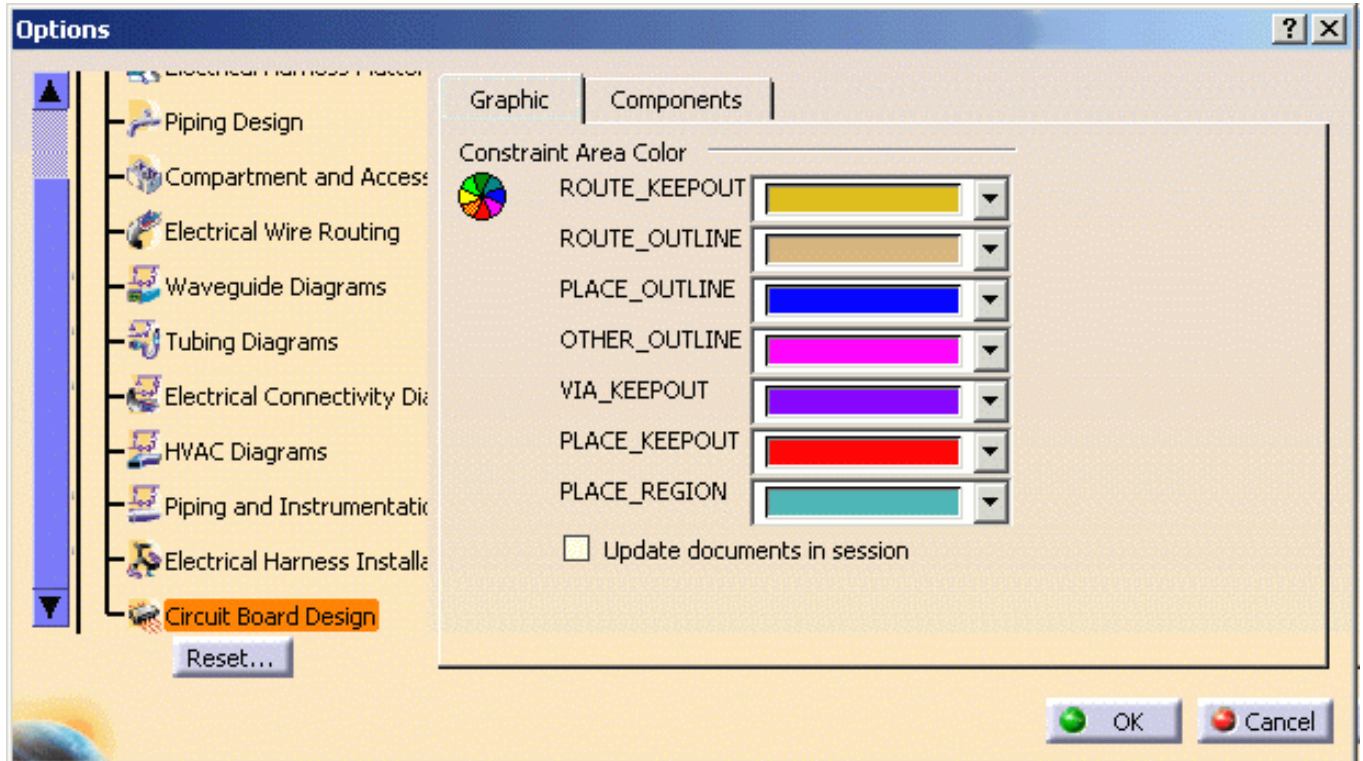
1. Select the **Tools -> Options** menu.

The Options dialog box opens.

2. Click **Equipment & Systems** in the left-hand box.

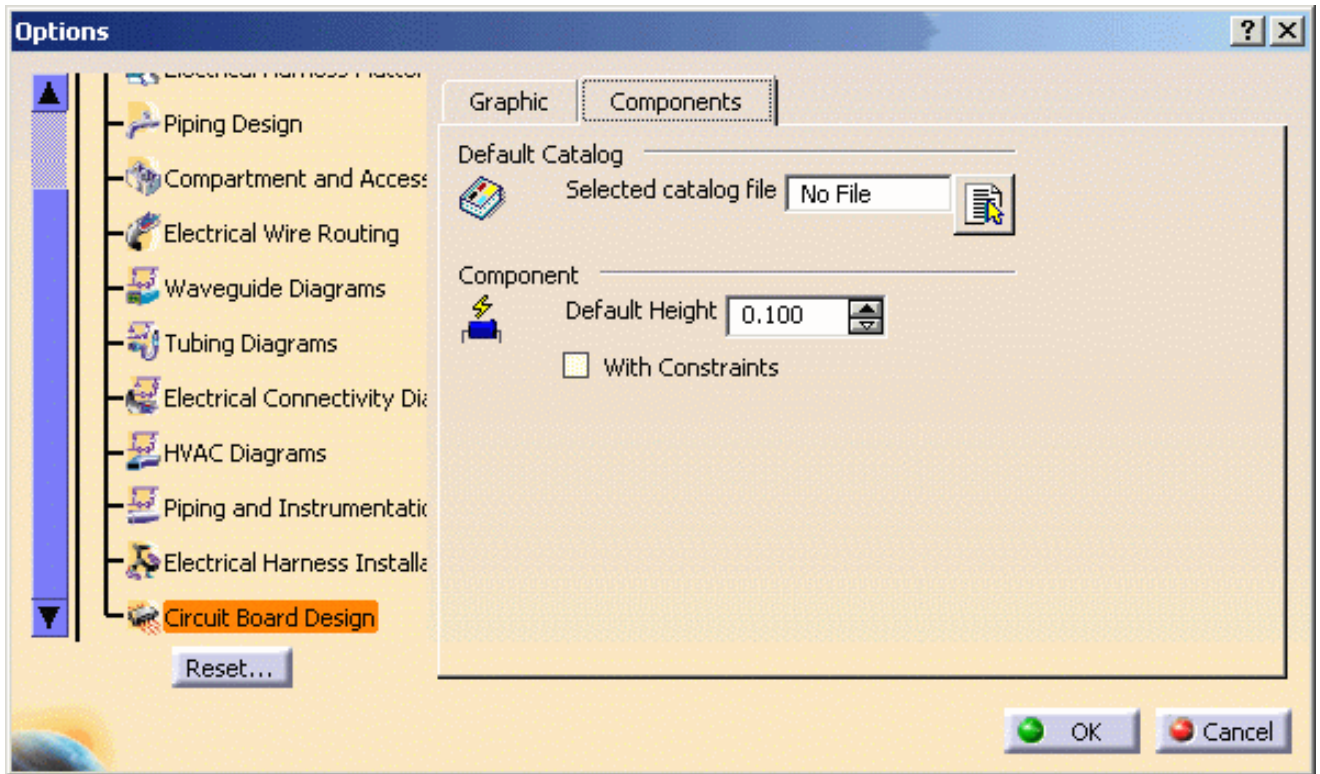
3. Click the **Circuit Board Design** workbench:

The **Circuit Board Design** workbench displays with two tabs:



The **Graphic** tab lets you set the color of your choice to the different areas.

If you check the **Update documents in session** box, your change will be applied to the current document.



The Components tab allows you to:

- browse to specify the default catalog,
- set a default height for the component used when importing data:  
the .lib file contains the component footprint.  
If its height is null, the **Default height** will be used as the component pad height.
- check the **With constraints** box to generate the mechanical constraints according to the component status:
  - the component is fixed: **FixTogether** constraint
  - the component is placed: **Offset** constraint.

4. Click **OK** to validate.



# Customizing for Knowledgeware

This section describes the ways in which you can customize the Knowledgeware workbenches. See also the Parameters and Measure customizing section.

[Knowledge Tab](#)

[Language Tab](#)

[Report Generation Tab](#)

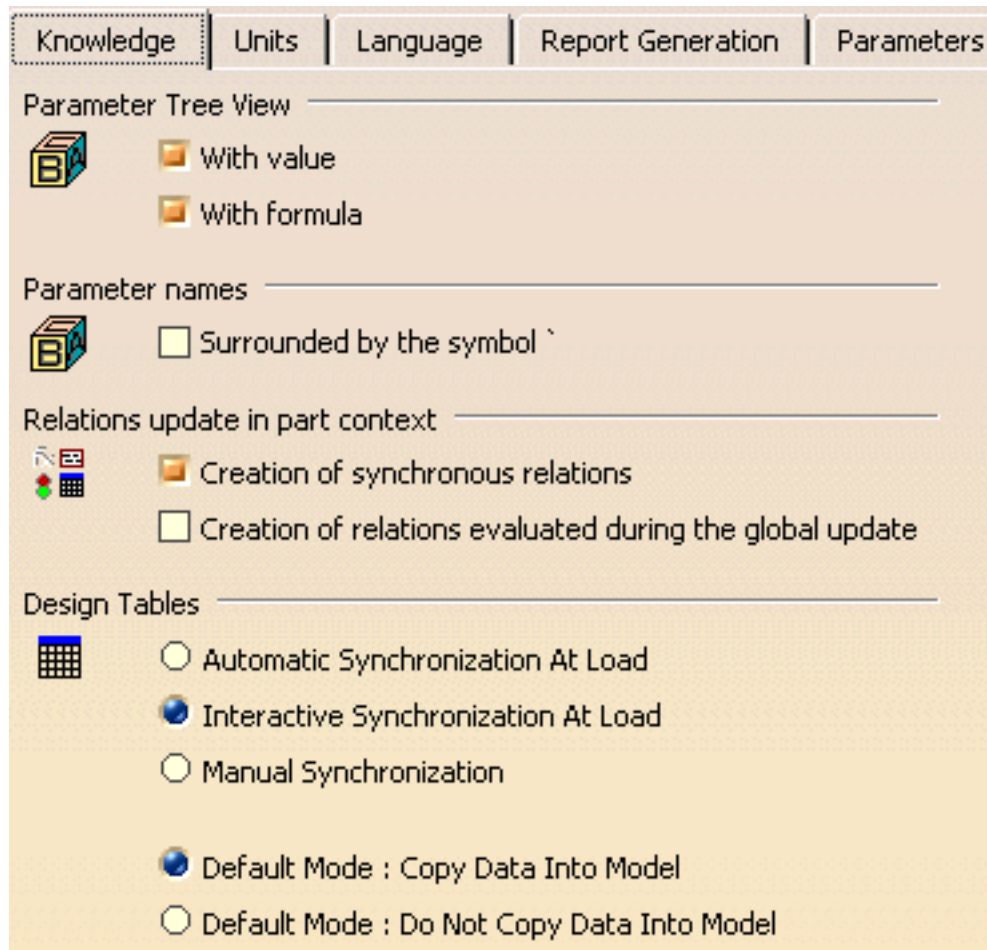
# Knowledge Tab



This task explains how to specify the options when working with Knowledgeware relations, parameters and design tables. Refer to [Using Knowledgeware Capabilities](#) for more information.



1. Select the **Tools->Options...** command. The **Options** dialog box displays.
2. Select **General->Parameters and Measure** and click the **Knowledge** tab. This is what you can see onscreen:



## Parameter Tree View Field

- Check **Tools->Options...->General->Parameters and Measure->Knowledge->Parameter Tree View->With value** to display the parameter values in the specification tree.
- Check the **Tools->Options...->General->Parameters and Measure->Knowledge->Parameter Tree View->With formula** to display the formulas constraining the parameter in the specification tree.

## Parameter names Field

- Check the **Tools->Options...->General->Parameters and Measure->Knowledge->Parameter Names->Surrounded by the symbol'** option if you work with non-Latin characters. If this option is unchecked, parameter names should have to be renamed in Latin characters when used in formulas.

## Relations update in part context

Before V5R12, Knowledge relations (formulas, rules, checks, design tables, and sets of equations) used to execute as soon as one of their inputs was modified.

The user can now choose, when creating the relation, if it will be synchronous (i.e. the evaluation will be launched as soon as one of its parameters is modified) or asynchronous (i.e. the evaluation will be launched when the Part is updated). Each relation can therefore be synchronous or asynchronous.

The 2 following options enable the user to create synchronous or asynchronous relations.

### Creation of synchronous relations

Enables the user to create synchronous relations, that is to say relations that will be immediately updated if one of their parameters/inputs is modified. Relations based on parameters are the only one that can be synchronous.

### Creation of relations evaluated during the global update

Enables the user to associate the evaluation of asynchronous relations with the global update. The relations can be asynchronous for 2 reasons:

- The user wants the relations to be asynchronous
- The relation contains measures.

- Relations based on parameters: These relations can be synchronous or asynchronous.
- Relations based on geometry: These relations can only be asynchronous.
- Relations based on parameters and on geometry: For the part of the relations containing parameters, the user decides if he wants the update to be synchronous or not. For the other part of the relations, the update occurs when the global update is launched.



Note that the user can also decide if already existing relations are synchronous or asynchronous. To know more, see Controlling Relations Update in the Infrastructure User's Guide.



## Design Tables Field

|                                           |                                                                                                                                                                                                                                                                                                                                                                                                                        |
|-------------------------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| Automatic Synchronization at Load         | When loading a model containing user design tables, if the design table files have been modified and the external file data is contained in the model, the design table will be synchronized automatically if this radio button is checked.                                                                                                                                                                            |
| Interactive Synchronization at Load       | When loading a model containing user design tables whose external source file was deleted, this option enables the user to select a new source file or to save the data contained in the design tables in a new file.                                                                                                                                                                                                  |
| Manual Synchronization                    | When loading a model containing user design tables, if the design table files have been modified and the external file data is contained in the model, the design table will be synchronized if this radio button is checked. To synchronize both files, right-click the design table in the specification tree and select the <b>DesignTable object-&gt;Synchronize</b> command or the <b>Edit-&gt;Links</b> command. |
| Default Mode: Copy Data Into Model        | If checked, the data contained in the external source file will be copied into the model.                                                                                                                                                                                                                                                                                                                              |
| Default Mode: Do Not Copy Data Into Model | If checked, the data contained in the external source file will not be copied into the model.                                                                                                                                                                                                                                                                                                                          |

3. Click **OK** when done.

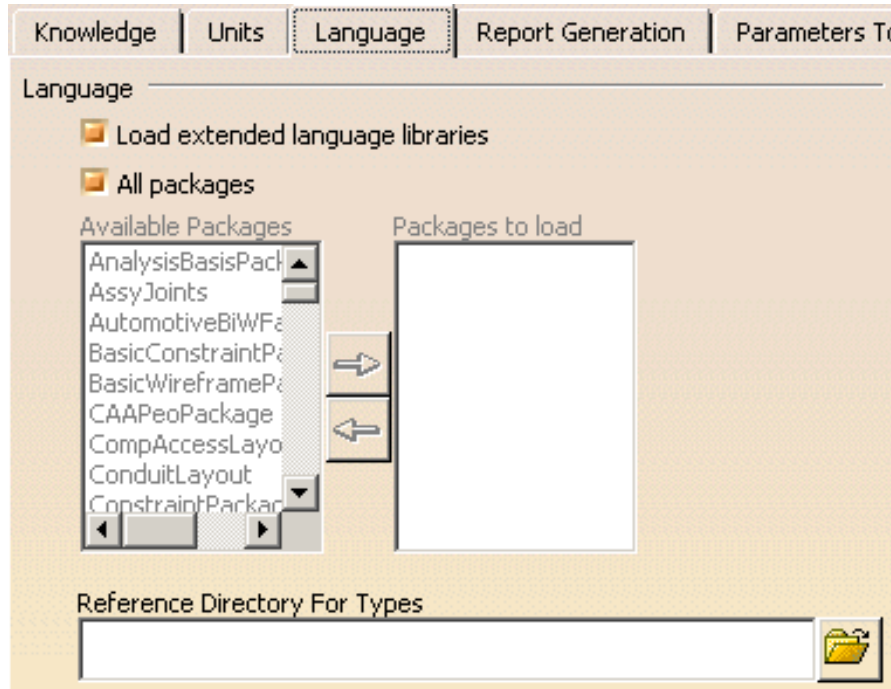
# Language Tab



This task explains how to specify the options you may need to check when working with Knowledge browsers.



1. Select the **Tools->Options...** command. The **Options** dialog box is displayed.
2. Select **General->Parameters and Measure** and click the **Language** tab. This is what you can see onscreen:



## Language Field

This field is to be used when using *measures* in relations or *user functions*. Measures are specific functions to be used in formulas and rules.

The *Knowledge Advisor User's Guide* provides you with tasks explaining how to use measures. For how to create and use user functions, see the *CATIA Application Architecture* documentation.

- Check **Tools->Options...->General->Parameters and Measure->Knowledge->Load extended language libraries** and select the packages you want to load under Packages to load if you want to load a limited number of packages.



This option is particularly useful for the administrator to limit the number of packages used by the user. It is also very useful to improve performances since only the required libraries are loaded.



- When you open a document and some relations are broken, you might need to load all libraries to solve the error, which may take quite a long time.
- It is strongly recommended to identify the packages you will need and to select them.

- **Check Tools->Options...->General->Parameters and Measure->Knowledge ->Load extended language libraries and select All packages to load all the packages displayed under Packages to load.**

### ***Reference Directory For Types field***

If you want to reuse the generated type in another *CATIA* session in the PKT workbench, proceed as follows:

- save the CATGScript file in the Directory indicated in the Reference Directory for Types field (see **Tools->Options->Parameters and Measure-> Language tab**)
- check the **Load extended language libraries** check box and select the package containing the type you created.

**3.** Click **OK** when done.

# Report Generation Tab



This task explains how to customize the reports generated by the Global Check Analysis tool in the Knowledge Advisor and Knowledge Expert workbenches.

You can choose to display a xml or a html report.

## Displaying a HTML report

To generate a html report when performing the check analysis, go to **Tools->Options->General->Parameters and Measure** and select the **Report customization** tab. Select **Html** in the Select **Report Visualization** area.

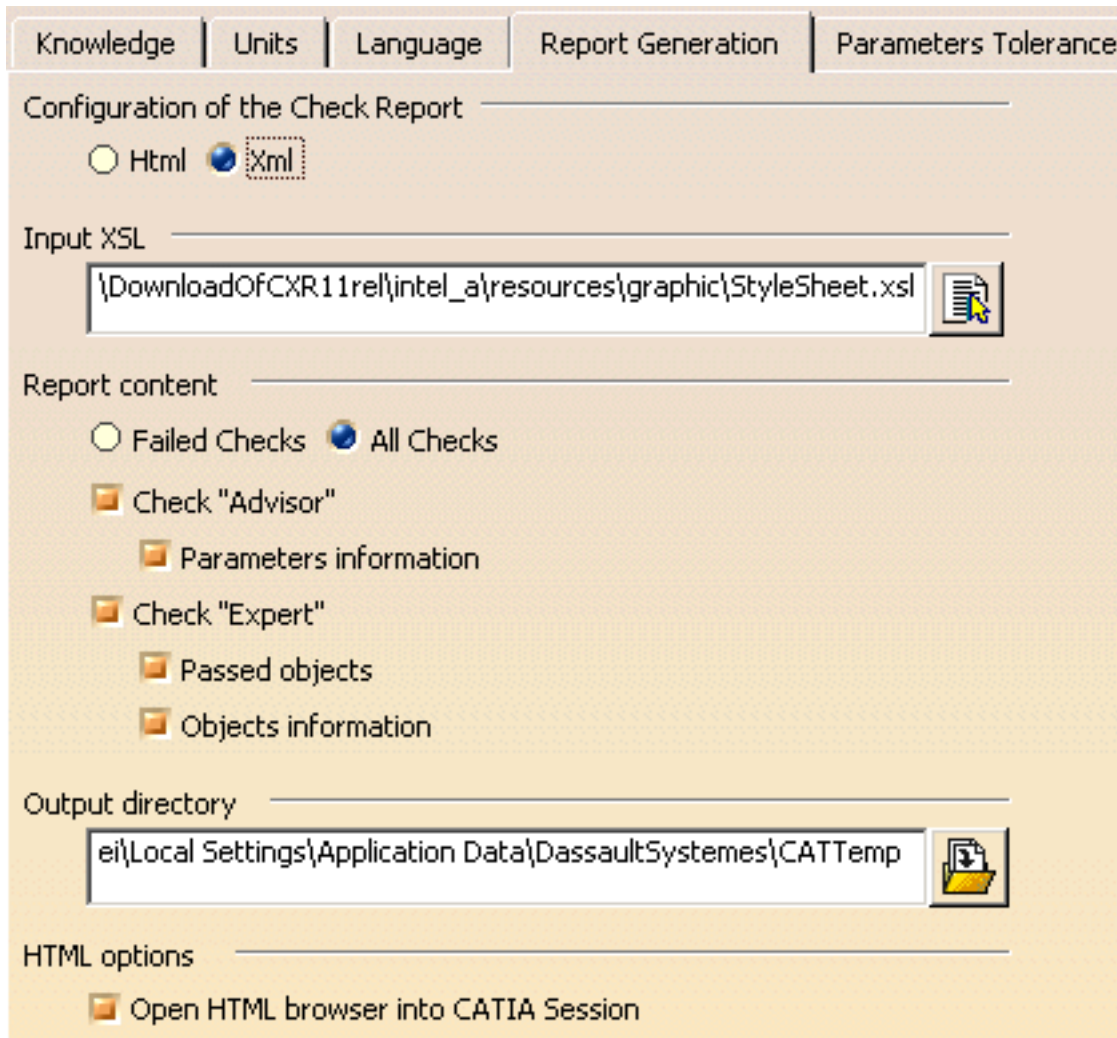
In this case, only the **Check Advisor**, the **Check expert** and the **Passed objects** options are available in the Report content area. You can specify the output directory containing the generated HTML report in the **Select output directory** field.



Select **Html** if you use a Netscape browser.

## Displaying a XML report

To display a XML report when performing the check analysis, go to **Tools->Options->General->Parameters and Measure** and select the **Report customization** tab. Select **Xml** in the Select **Report Visualization** area. The following window opens:



The **Report Customization** tab is made up of 4 different fields: The **Select Input XSL**, the **Report Content**, the **Select output directory**, and the **HTML options** fields.

### Select Input XSL field

This field enables the user to select the XSL style sheet that will be applied to the generated XML report. The StyleSheet.xml file is the default XSL file, but you can use your own template.

### Report content field

|                      |                                                                                                                             |
|----------------------|-----------------------------------------------------------------------------------------------------------------------------|
| <b>Failed Checks</b> | If checked, the generated report will contain information about the failed checks only.                                     |
| <b>All Checks</b>    | If checked, the generated report will contain information about all the checks contained in the document.                   |
| <b>Check advisor</b> | If checked, the generated report will contain information about all the Knowledge Advisor checks contained in the document. |

|                     |                              |                                                                                                                                                                                                           |
|---------------------|------------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
|                     | <b>Parameter information</b> | If checked, the generated report will contain information about the parameters of the Advisor checks.                                                                                                     |
| <b>Check expert</b> |                              | If checked, the generated report will contain information about all the Knowledge Expert checks contained in the document.                                                                                |
|                     | <b>Passed objects</b>        | If checked, the generated report will contain information about the objects that passed the Expert checks as well as information about the parameters of these objects (diameter, depth, pitch,...).      |
|                     | <b>Objects information</b>   | If checked, the generated report will contain information about all the objects contained in the Expert checks as well as information about the parameters of these objects (diameter, depth, pitch,...). |

## Select output directory field

This field enables the user to select the output directory containing the generated XML report.

## HTML options field

This option is available for Windows only. It enables the user to define if the report will be opened in a *CATIA* session (in this case, the check box should be checked) or if it will be opened in an Internet Explorer session (in this case, the check box should remain unchecked.)



Note that it is highly recommended not to use this report as a basis for macros or for other applications. It is only provided for information purposes.



# Digital Process for Manufacturing

## **DPM Process for Manufacturing**

(refer to the *DPM Process Planner User's Guide*)

## **DPM Body In White**

(refer to the *DPM Body-in-White Process Planner User's Guide*)

## **DPM Machining Process Planner**

(refer to the *DPM Machining Process Planner User's Guide*)

# Ergonomics Design & Analysis

## **Human Measurements Editor**

(refer to the *Human Measurements Editor User's Guide*)

## **Human Task Simulation**

(refer to the *Human Task Simulation User's Guide*)

## **Human Activity Analysis**

(refer to the *Human Activity Analysis User's Guide*)

## **Human Builder**

(refer to the *Human Builder User's Guide*)



# Glossary



## A



- access key** The key that corresponds to an underlines letter on a menu or control (also referred to as a mnemonic access key).
- active end** The ending point for a selected range of objects. It is usually established at the object logically nearest the hot spot of the pointer when a user releases a mouse button. Compare [anchor point](#).
- 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. Compare [inactive window](#).
- album** Collection of images you generate with the **Tools** -> **Capture** command.
- anchor point** The starting point for a selected range of objects. An anchor point is usually established at the object logically nearest the hot spot of the pointer when a user presses a mouse button. Compare [active end](#).
- application window** Window containing the Version 5 application.
- apply** To commit a set of changes or pending transactions made in a secondary window, typically without closing that window.
- auto-repeat** An event or interaction that is automatically repeated. Auto-repeat events usually occur when a user holds down a keyboard key or presses and holds a special control (for example, scroll bar buttons).

## B



**background** Area in document windows to which you can apply a background color.

**box edit** A standard Microsoft Windows interface control that provides a discrete area for entering each character. A user can also edit text within the control.

## C



**cancel** To halt an operation or process and return to the state before it was invoked. Compare [stop](#).

**cascading menu** A menu that is a submenu of a menu item (also referred to as a hierarchical menu, child menu, or submenu).

**check box** A standard Windows control that displays a setting, either checked (set) or unchecked (not set). Compare [option button](#).

**child menu** See [cascading menu](#).

**click** (v.) To position the pointer over an object and then press and release a mouse button. See also [press](#).  
(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.

**clipping** Depth effect for viewing parts or all of a 3D object between two planes.

**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**

**combo box** A standard Windows control that combines a text box and interdependent list box.

**command button** A standard Windows control that initiates a command or sets an option (also referred to as a push button).

**context-sensitive Help** Information about an object and its current condition. It answers the questions "What is this" and "Why would I want to use it?" Compare [reference Help](#) and [task-oriented Help](#).

|                             |                                                                                                                                                                                                                                                                       |
|-----------------------------|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>contextual menu</b>      | A menu that is displayed at the location of a selected object (also referred to as a shortcut menu). The menu contains the commands that are contextually relevant to the selection, and the most frequently used. You display the contextual menu by right-clicking. |
| <b>contiguous selection</b> | A selection that consists of a set of objects that are logically sequential or adjacent to each other (also referred to as range selection).                                                                                                                          |
| <b>current</b>              | State of an object when selected.                                                                                                                                                                                                                                     |
| <b>cursor</b>               | A generic term for the visible indication of where a user's interaction will occur. See also input focus, <a href="#">insertion point</a> , and <a href="#">pointer</a> .                                                                                             |

## D



|                        |                                                                                                                                                                                                                                                                           |
|------------------------|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>default</b>         | An operation or value that the system or application assumes, unless a user makes an explicit choice.                                                                                                                                                                     |
| <b>desktop</b>         | The visual work area that fills the display. The desktop is also a container and can be used as a convenient location to place objects stored in the file system.                                                                                                         |
| <b>dialog box</b>      | 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. |
| <b>document</b>        | 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.                                                                                          |
| <b>document window</b> | A window that provides a primary view of a document (typically its content).                                                                                                                                                                                              |
| <b>double-click</b>    | (v.) To press and release a mouse button twice in rapid succession.<br>(n.) The act of double-clicking.                                                                                                                                                                   |
| <b>duplicate</b>       | Performs mirroring or pattern operations on an active object.                                                                                                                                                                                                             |

## E



- edit field** See [text box](#).
- Edit menu** A common drop-down menu that includes general purpose commands for editing the current object, such as Cut, Copy, and Paste.
- ellipsis** The "..." suffix added to a menu item or button label to indicate that the command requires additional information to be completed. When a user chooses the command, a dialog box is usually displayed for the user input of this additional information.
- embedded object** See [OLE embedded object](#).
- enter** (v.) To type a character from the keyboard.  
(n.) A classification of an object based on its characteristics, behavior, and attributes.
- extended selection** A section technique that is optimized for the selection of a single object or single range using contiguous selection techniques (that is, canceling any existing selection when a new selection is made). However, it also supports modifying an existing selection using disjoint selection techniques.

## F



- filter** Tool for organizing elements of V4 model document into layers.
- File menu** A common drop-down menu that includes commands for file operations, such as Open, Save, and Print.
- font** A set of attributes for text characters.
- font size** The size of a font, typically represented in points.

## G



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

**glyph** A generic term used to refer to any graphic or pictorial image that can be used on a button or in a message box. Compare [icon](#).

## H



**handle** An interface element added to an object that provides a control point for moving, sizing, reshaping, or other operations pertaining to that object.

**Help menu** A common drop-down menu that includes commands that provide access to Help information or other forms of user assistance. See also [context-sensitive Help](#), and [task-oriented Help](#).

**hierarchical menu** See [cascading menu](#).

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

**hot spot** The specific portion of the pointer (or pointing device) that defines the exact location, or object, to which a user is pointing.

## I



**icon** A pictorial representation of an object. Compare [glyph](#).

**inactive window** 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. Compare [active window](#).

**in-place activation** The ability to edit an OLE embedded object in place, without opening it into its own window.

**insertion point** 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.

**interoperability**

- Ability to exchange data between CATIA Version 4 and Version 5.
- Ability to exchange Version 5 data between Version 5 workshops
- Ability to exchange data between Version 5 and OLE-compliant applications.

**J****jump**

A special form of a link that navigates to another location (also referred to as a hyperlink).

**L****label**

The text (or graphic) that identifies a control (also referred to as a caption).

**landscape**

An orientation where the long dimension of a rectangular area (for example, screen or paper) is horizontal.

**link**

(v.) To form a connection between two objects.

(n) A reference to an object that is linked to another object. See also [OLE linked object](#).

**link path**

The descriptive form of referring to the location of a link source (also referred to as a moniker).

**list box**

A standard Windows control that displays a list of choices.

**M****marquee**

See [region selection](#).

**maximize**

To make a window its largest size. See also [minimize](#).

**menu**

A list of textual or graphical choices from which a user can choose. See also [contextual menu](#).

|                           |                                                                                                                                                                                                                                     |
|---------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <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>minimize</b>           | To minimize the size of a window: in some cases, this means to hide the window. See also <a href="#">maximize</a> .                                                                                                                 |
| <b>mirror</b>             | An operation creating a 3D object by duplicating an initial object. The duplication is defined by symmetry.                                                                                                                         |
| <b>model</b>              | CATIA Version 4 model.                                                                                                                                                                                                              |
| <b>model document</b>     | Document containing a CATIA Version 4 model.                                                                                                                                                                                        |
| <b>most recently used</b> | List of most recently used files (MRU) located in the File menu for easy access.                                                                                                                                                    |
| <b>mouse</b>              | 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 head pointers). |

## O



|                            |                                                                                                                                                                |
|----------------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>object</b>              | An entity or component identifiable by a user that can be distinguished by its properties, operations, and relationships.                                      |
| <b>OLE</b>                 | Object Linking and Embedding. The name that describes the technology and interface for implementing support for object interaction.                            |
| <b>OLE embedded object</b> | A data object that retains the original editing and operating functionality of the application that created it, while physically residing in another document. |

- OLE linked object** An object that represents or provides an access point to another object that resides at another location in the same container or a different, separate container. See also [link](#).
- 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). Compare [check box](#).

## P



- package** An OLE encapsulation of a file so that it can be embedded in an OLE container.
- 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).
- portrait** An orientation where the long dimension of a rectangular area (for example, screen or paper) is vertical.
- press** To press and release a keyboard key. See also [click](#).
- preview** Tool allowing you to view a document or album image prior to printing.
- progress indicator** Any form of feedback that provides the user with information about the state of a process.
- property** Attribute or characteristic of an object that define its state, appearance, or value.
- push button** See [command button](#).

## R





|                         |                                                                                                                                                                                    |
|-------------------------|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>radio button</b>     | See <a href="#">option button</a> .                                                                                                                                                |
| <b>range selection</b>  | See <a href="#">contiguous selection</a> .                                                                                                                                         |
| <b>redo</b>             | Repeat the last operation.                                                                                                                                                         |
| <b>reference Help</b>   | A form of online Help information that can contain conceptual and explanatory information. Compare <a href="#">task-oriented Help</a> and <a href="#">context-sensitive Help</a> . |
| <b>region selection</b> | A selection technique that involves dragging out a bounding outline (also referred to as a marquee) to define the selected objects.                                                |
| <b>right-click</b>      | Click using the right mouse button (to display contextual menu).                                                                                                                   |

## S



|                            |                                                                                                                                                                                                                                                                |
|----------------------------|----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>scale</b>               | Operation that resizes the contents of document prior to printing or previewing.                                                                                                                                                                               |
| <b>scroll</b>              | To move the view of an object or information to make a different portion visible.                                                                                                                                                                              |
| <b>scroll arrow button</b> | A component of a scroll bar that allows the information to be scrolled by defined increments when the user clicks it. The direction of the arrow indicates the direction in which the information scrolls.                                                     |
| <b>scroll bar</b>          | A standard Windows control that supports scrolling.                                                                                                                                                                                                            |
| <b>scroll box</b>          | A component of a scroll bar that indicates the relative position (and optionally the proportion) of the visible information relative to the entire amount of information. The user can drag the scroll box to view areas of information not currently visible. |
| <b>section view</b>        | 3D view of a cross-section generated using a plane.                                                                                                                                                                                                            |
| <b>select</b>              | To identify one or more objects upon which an operation can be performed.                                                                                                                                                                                      |
| <b>selection</b>           | An object or set of objects that have been selected.                                                                                                                                                                                                           |
| <b>selection handle</b>    | A graphical control point of an object that provides direct manipulation support for operations of that object, such as moving, sizing, or scaling.                                                                                                            |
| <b>selection set</b>       | A group of selected objects that you can store and retrieve.                                                                                                                                                                                                   |

|                            |                                                                                                                                                                   |
|----------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>Send To</b>             | Tool used for sharing document files with other users.                                                                                                            |
| <b>settings</b>            | Set of setup parameters and user preferences stored in non-editable files.                                                                                        |
| <b>shell</b>               | A generic term that refers to the interface that allows the user control over the system.                                                                         |
| <b>shortcut</b>            | A generic term that refers to an action or technique that invokes a particular command or performs an operation with less interaction than its usual method.      |
| <b>shortcut icon</b>       | A link presented as an icon that provides a user with access to another object.                                                                                   |
| <b>shortcut key</b>        | A keyboard key or key combination that invokes a particular command (also referred to as an accelerator key).                                                     |
| <b>shortcut menu</b>       | See <a href="#">contextual menu</a> .                                                                                                                             |
| <b>slider</b>              | A standard Windows control that displays and sets a value from a continuous range of possible values, such as brightness or volume.                               |
| <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>spin box</b>            | A control composed of a text box and increment and decrement buttons that allow a user to adjust a values from a limited range of possible values.                |
| <b>split</b>               | An operation allowing splitting 3D objects using a plane.                                                                                                         |
| <b>standard view</b>       | Typical view available in 3D workshops: top, bottom, front, back, left, right, isometric.                                                                         |
| <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. Compare <a href="#">cancel</a> .                                    |
| <b>STRIM/STYLER models</b> | Model files generated by STRIM or STYLER. These files have the same ".tdg" extensions. They are read and converted to Version 5 documents for further processing. |
| <b>submenu</b>             | See <a href="#">cascading menu</a> .                                                                                                                              |

## T



|                           |                                                                                                                                                                               |
|---------------------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|
| <b>taskbar</b>            | A special toolbar that docks on an edge of the desktop supplied by the system. The taskbar includes the Start button, buttons for each open primary window, and a status bar. |
| <b>task-oriented Help</b> | Information about the steps involved in carrying out a particular task. Compare <a href="#">context-sensitive Help</a> and <a href="#">reference Help</a> .                   |
| <b>template</b>           | An object that automates the creation of new objects of a particular type.                                                                                                    |
| <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>toolbar</b>            | A frame or special area that contains a set of other controls.                                                                                                                |
| <b>tooltip</b>            | A standard Windows control that provides a small pop-up window that provides descriptive text such as a label, for a control or graphic object.                               |

## 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.                                                                             |
| <b>user-defined view</b> | 3D view an end user can generate by customizing view parameters. The view can be named. Also referred to as named view.      |

## V



|                      |                                                                   |
|----------------------|-------------------------------------------------------------------|
| <b>View toolbar</b>  | Toolbar containing viewing tools.                                 |
| <b>viewing tools</b> | Tools for viewing contents of current document in different ways. |

**VRML** Virtual Reality Markup Language. A vector-based language for modeling three-dimensional environments. It sends ASCII text files over the Internet, which are translated by the VRML viewing engine at the other end. VRML complements HTML. This format is useful for viewing Version 5 data using a Web browser.

## W



**window** A standard Windows object that displays information. A window is a separately controllable area of the screen that typically has a rectangular border.

**workbench** Set of tools for completing specific tasks. Each type of document can be edited with a document-specific set of tools.

**workspace** A window or task management technique that consists of a container holding a set of objects, where the windows of the contained objects are constrained to a parent window. Similar to the multiple document interface, except that the window displayed within the parent window are of objects that are also contained in the workspace.

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





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