PDM Workbench

PDM Workbench
Release 3.9
for Aras Innovator

User Manual

Version 1

Copyright

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This edition obsoletes all previous editions.

Your Comments are Welcome

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Preface

About this Manual

This manual describes the main functionality delivered by the PDM Workbench.

The functionality of the PDM Workbench as described in this manual uses Aras Innovator as backend PDM system for CATIA V5.

Other PDM systems might support the functionality of the PDM Workbench with their own types of objects. Layout of dialogs as well as object and relation types or classes might vary accordingly.

This manual is intended for end users of the PDM Workbench.

It assumes that the reader is familiar with the CATIA application and with Aras Innovator.

Related Documents

The following manuals contain information about installation, administration, usage, and customization of the PDM Workbench:

Manual Title	Version
PDM Workbench Installation & Administration Manual	3.9
PDM Workbench User Manual	3.9

Organization

This manual contains the following chapters:

Chapter 1 provides basic information about the PDM Workbench and describes some features of this application.

Chapter 2 describes the supported data models.

Chapter 3 describes the functionality which is implemented in the CATIA V5 workshop.

Glossary contains the PDM Workbench terminology.

Trademarks

CATIA is a registered trademark of Dassault Systèmes.

Aras is a registered trademark of Aras Corporation.

Names of other products mentioned in this manual are used for identification purpose only and might be trademarks of their companies.

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CHAPTER 1

Overview

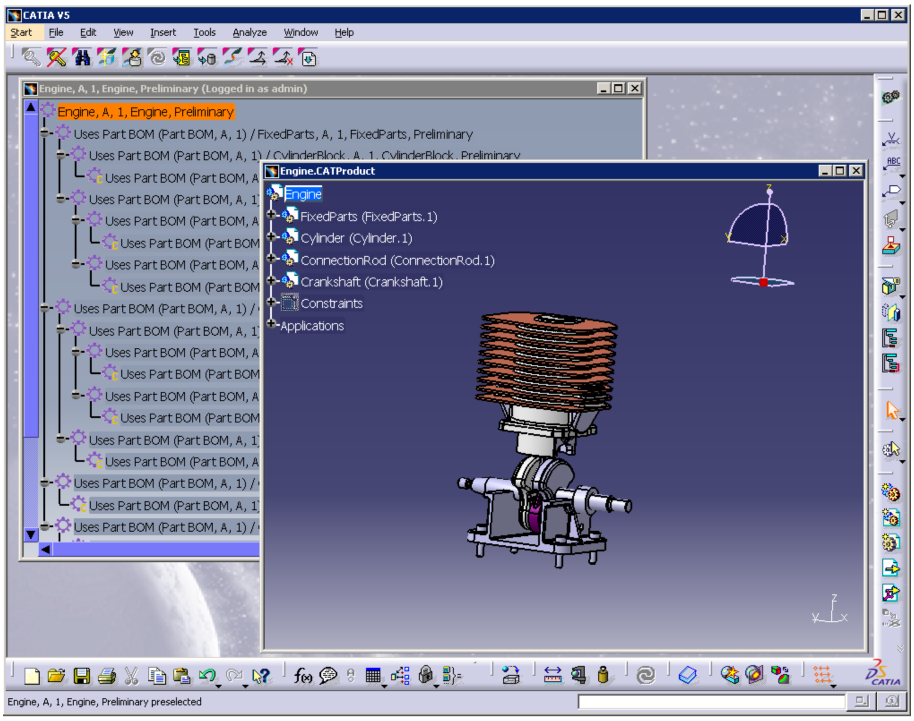
This chapter provides basic information about the *PDM Workbench* and lists some features of this application.

Introducing PDM Workbench

The *PDM Workbench* is developed by T-Systems as a high-end integration between the CAD system *CATIA V5* and several PDM systems. Those PDM systems might be customized or virtual (a combination of several PDM systems). No matter which type they are the PDM Workbench will integrate them into CATIA V5. How those PDM object types are to present their dialogs and functionalities within the PDM Workbench is defined in a configuration file (called *PDM Workbench Schema File*).

The PDM Workbench workshop works with two different windows. The first window type presents the results of your queries. There you also might expand your product structure. Or you can create new objects in this window.

You can load this product structure into a CATIA V5 native window, and you can modify the content of this CATIA V5 native window (see *Picture 1: PDM Workbench workshop in CATIA V5*).



Picture 1: PDM Workbench workshop in CATIA V5

CHAPTER 2

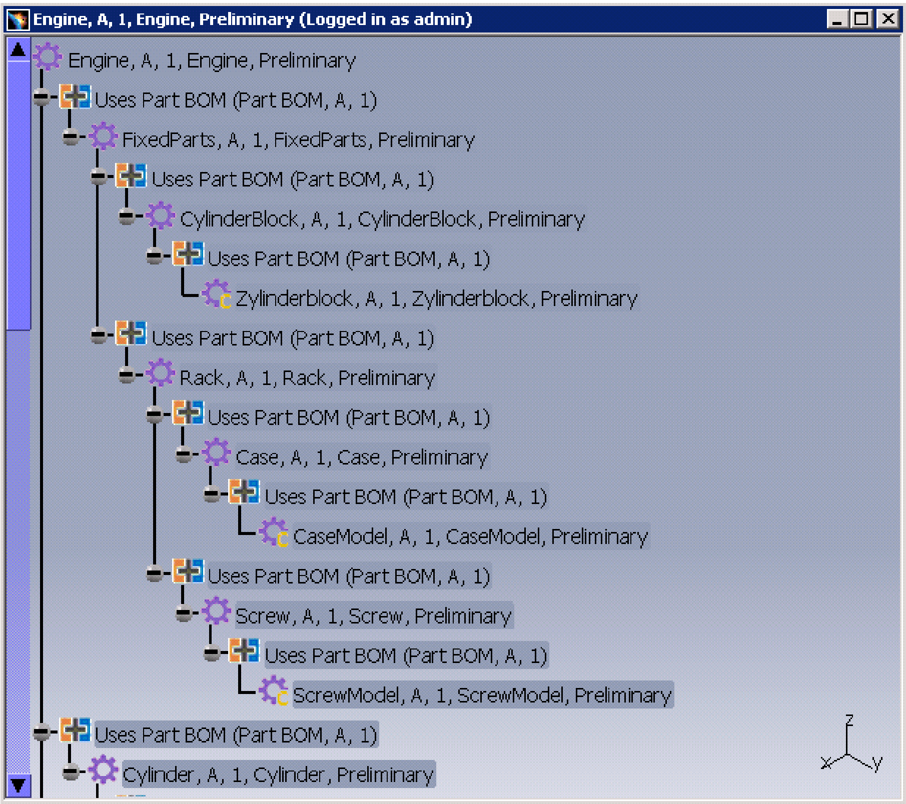
Supported Data Models

The PDM Workbench supports two different data models.

BOM Part Structure data model

In the BOM Part Structure data model the PDM structure is represented by Parts (Assembly or Component). The relation "Part BOM" is used.

Each Part is described by a CAD Document which includes the CATIA file for a CATProduct, CATPart, or CATDrawing (see *Picture 2: Structure in the BOM Part Structure data model*).

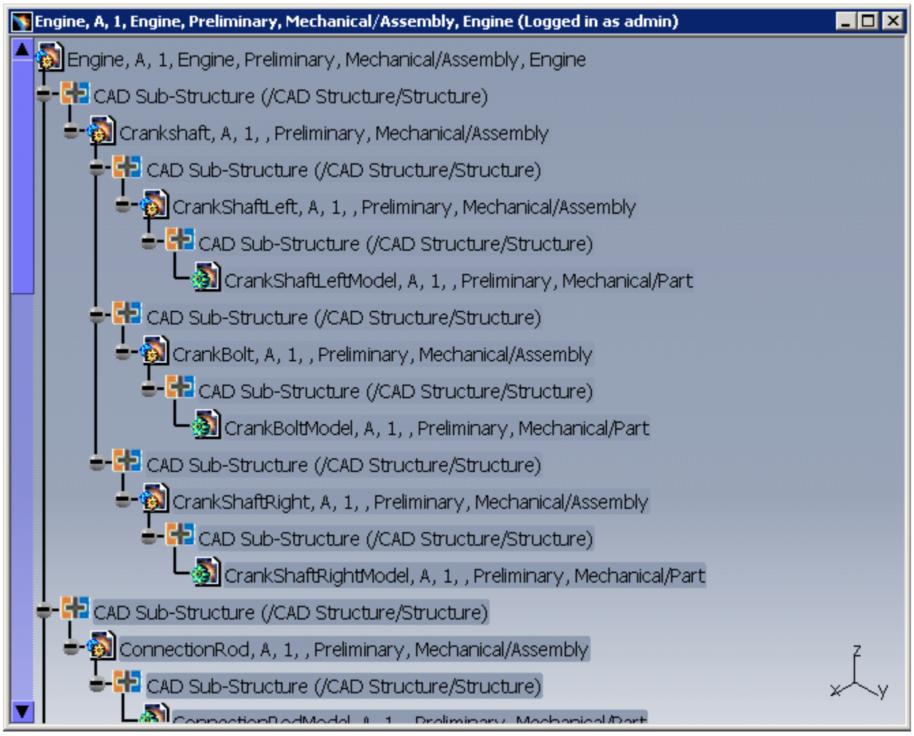


Picture 2: Structure in the BOM Part Structure data model

Document data model

In the Document data model the PDM structure is represented by CAD Documents. The relation "CAD Structure" is used.

Each CAD Document includes the CATIA file for a CATProduct, CATPart, or CATDrawing (see *Picture 3: Structure in the Document data model*).



Picture 3: Structure in the Document data model

CHAPTER 3

Getting Started

This chapter describes the functionality of the PDM Workbench which is implemented in the CATIA V5 workshop.

We suppose that you have installed CATIA V5 with the PDM Workbench workshop on your computer. All configurations for the PDM Workbench (including the configurations for the PDM system) are done properly.

Note: All user actions described below are based on the corresponding configuration of the data model and the actions to be provided by the PDM system.

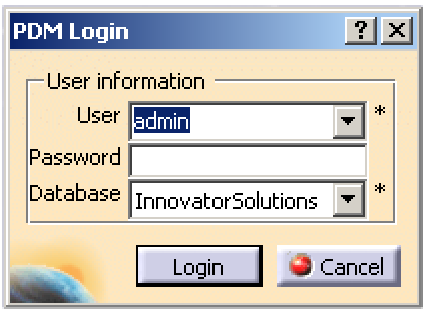
Login

In order to access the PDM Workbench functionality you must log in into the PDM system.

You select the “Login” icon  within the PDM Workbench toolbar (see *Picture 4: The PDM Workbench toolbar before the login*) in CATIA V5. The other PDM Workbench icons remain deactivated.

Picture 4: The PDM Workbench toolbar before the login

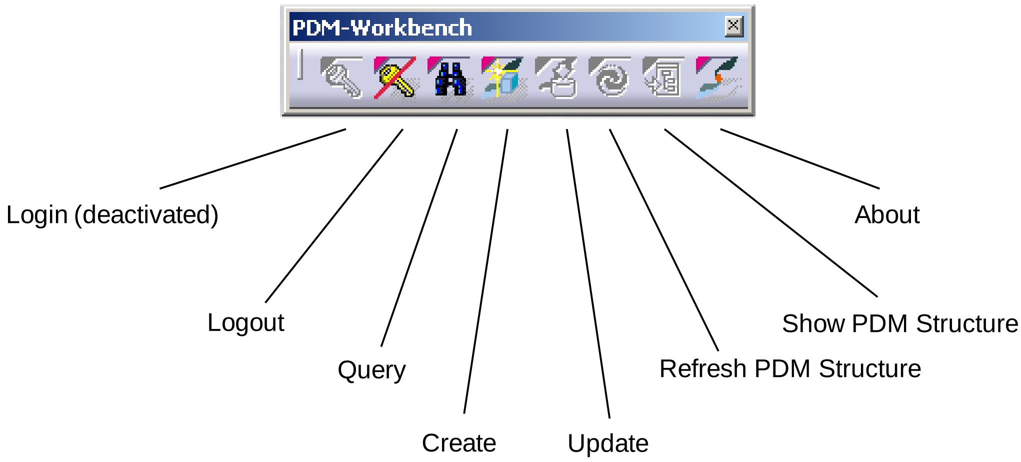
The following dialog (see *Picture 5: “Login” dialog*) prompts you for all information necessary to identify yourself in the PDM system. In our example you are asked to enter your “User”, “Password”, and “Database”. The identification items marked with an asterisk are defined as necessary for Login in the PDM Workbench configuration file.



Picture 5: “Login” dialog

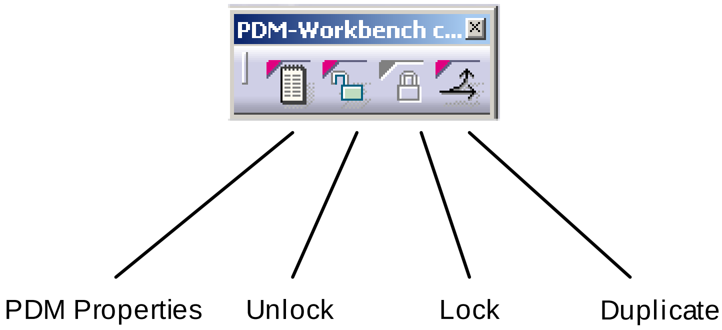
Click the “Login” button.

Once the login action was successful most of the icons in the PDM Workbench toolbar will turn active while some will remain inactive as they depend on further user actions to be done to get them available (see *Picture 6: The PDM Workbench toolbar after the login*).



Picture 6: The PDM Workbench toolbar after the login

For CATPart and CATDrawing as top level object of the CATIA V5 window you have to use the actions “PDM Properties”, “Unlock”, “Lock”, and "Duplicate" of the “PDM Workbench context commands” toolbar (see *Picture 7: The PDM Workbench context commands toolbar*). The icons in this toolbar are only repainted (e.g. switch from “Lock” to “Unlock”) when you newly activate the CATIA V5 window.

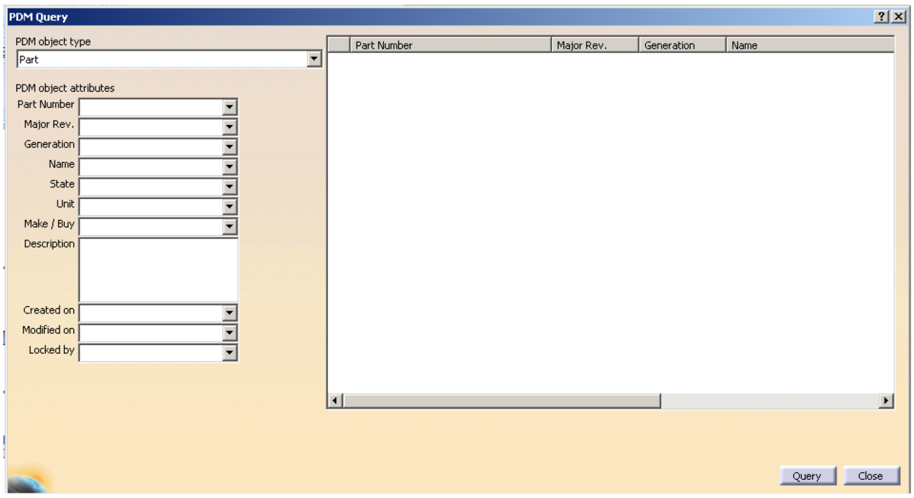


Picture 7: The PDM Workbench context commands toolbar

Query

You can query for any object stored in the PDM system.

Once you click the “Query” icon  within the PDM Workbench toolbar the query dialog opens. On the left side of the dialog you can define the query criteria. On the right side you will see the query results (see *Picture 8: “PDM Query” dialog*).



Picture 8: "PDM Query" dialog

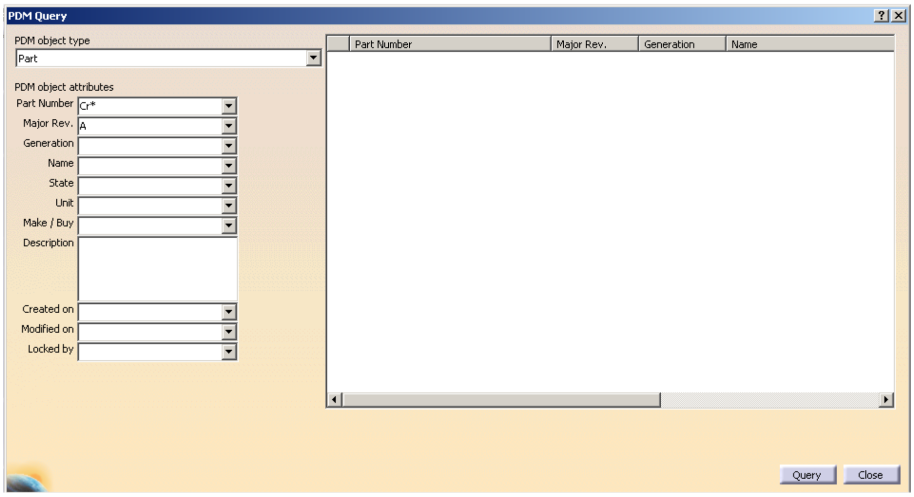
The query criteria are defined by the type and the attributes for this type. A default type is selected and its attributes are shown.

You can select a type in the single selector list in the first line. The attributes will be shown automatically.

Specify your selection criteria in the lines below.

You can use the wildcard "*" in this dialog. All attributes visible in this dialog are attributes of the PDM system for the selected object type. (These attributes as well as their adherence to the "Query" dialog of this type are defined in the PDM Workbench configuration file).

Some attribute values can be keyed in as free text while for others a value might be selected from a list.

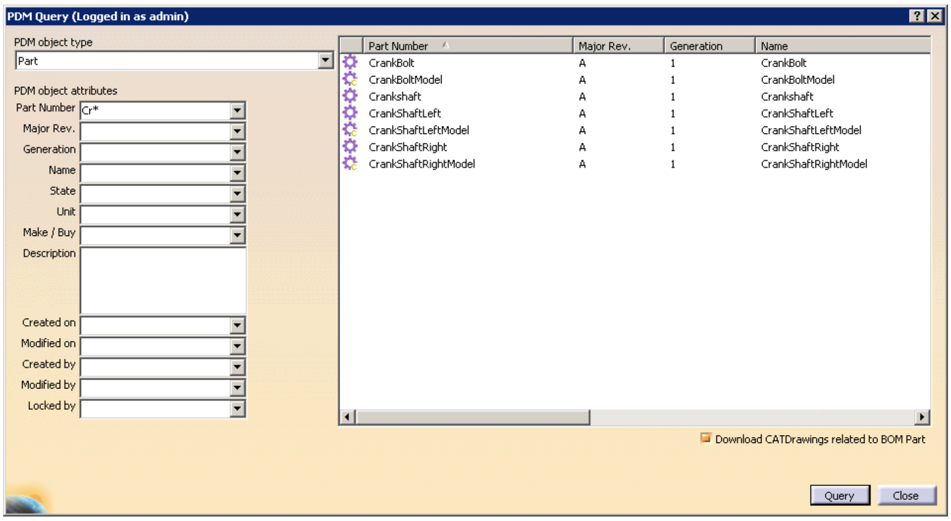


Picture 9: "PDM Query" dialog – enter query criteria

When you have specified the selection criteria you can start the query for PDM objects with a click on the "Query" button.

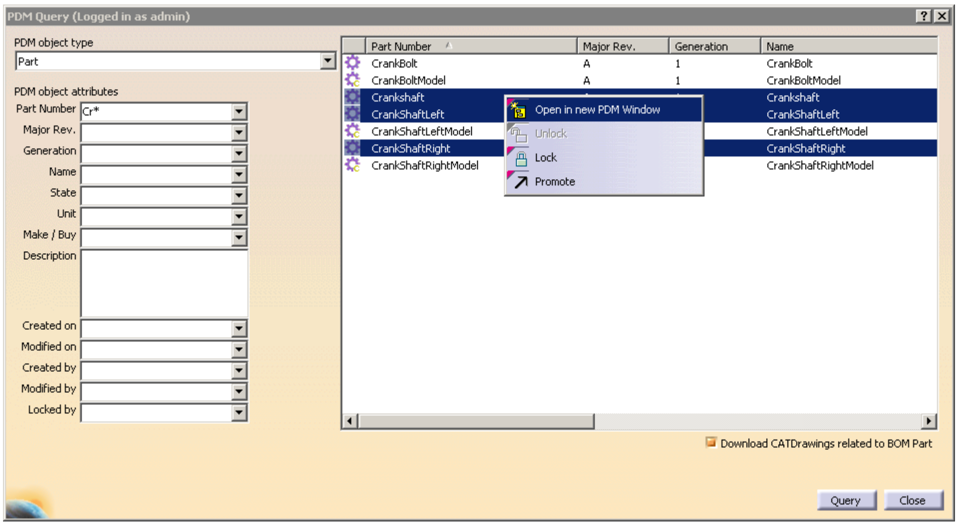
In the settings of the PDM Workbench you can define the columns for the query result on the right side of the dialog (see chapter *Options* - *Query Mode*).

The found objects will be presented on the right side of the dialog. By clicking on the column name you can sort the query result (see *Picture 10: "PDM Query" dialog – found objects*).



Picture 10: “PDM Query” dialog – found objects

There you can select the object or the objects to be opened in a new PDM Workbench window. Click on the right mouse button and select “Open in New PDM Window” (see *Picture 11: “PDM Query” dialog – Action "Open in New PDM Window"*).

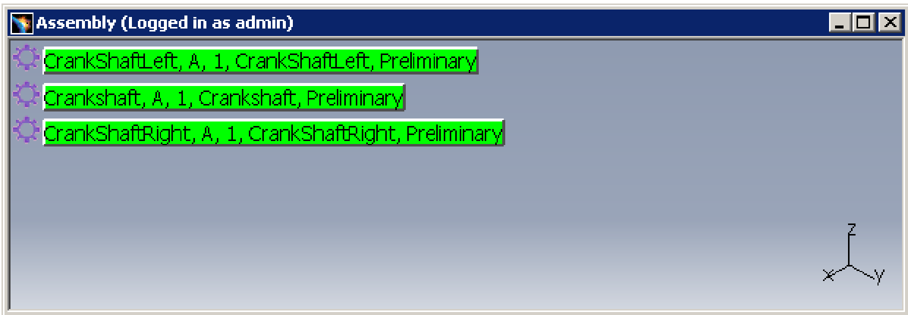


Picture 11: “PDM Query” dialog – Action "Open in New PDM Window"

A double click on a single object opens it in a new PDM Workbench window.

Now you can close the query dialog by clicking on the “Close” button.

The found objects are opened in the PDM Workbench window now (see *Picture 12: Query result in PDM Structure window*).



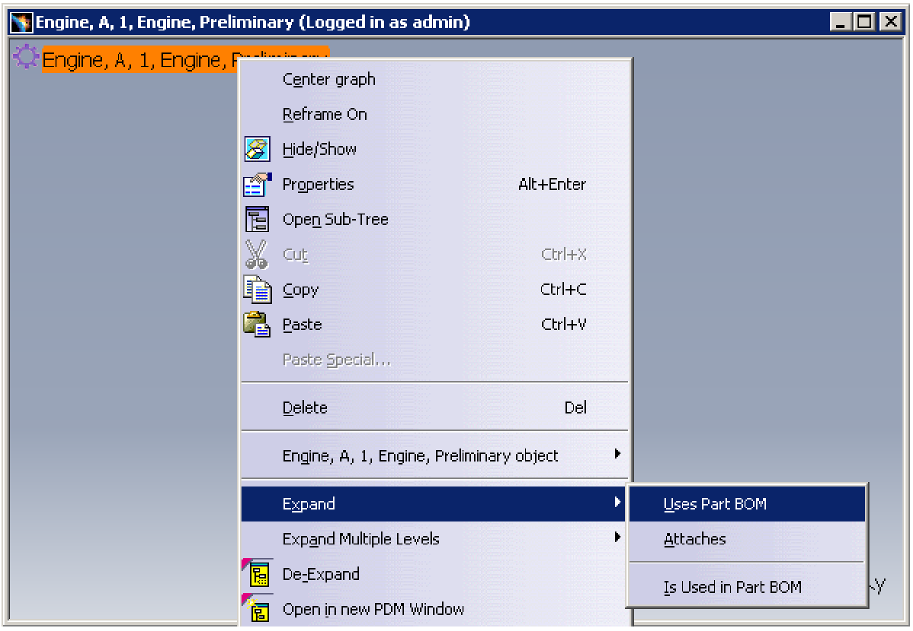
Picture 12: Query result in PDM Structure window

Expand Single Level

You can expand from the selected object to other related objects via several relations in two directions.

Select the object from which you want to expand and open the context menu by clicking on the right mouse button. Select the context action “Expand”. The “Expand” sub menu offers the possible relation directions for expansion from the selected object.

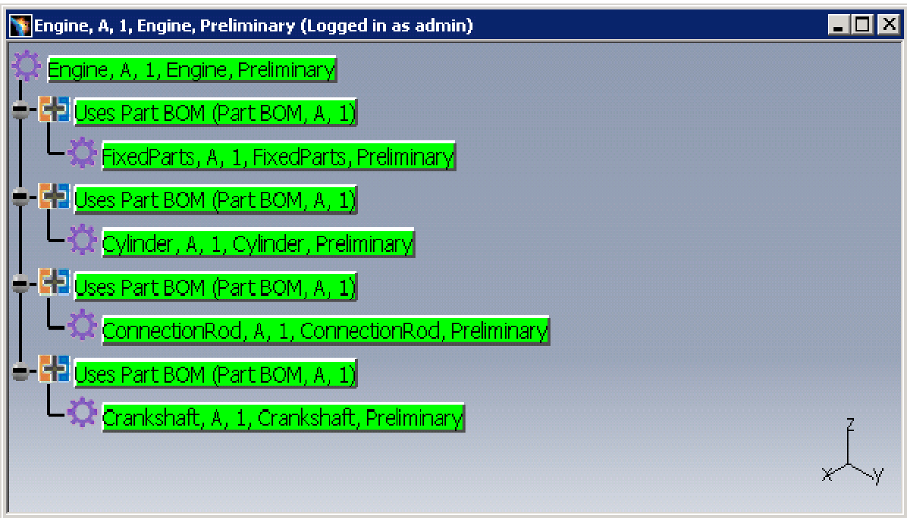
In the example in *Picture 13: Action “Expand Single Level”* you select the direction “Uses Part BOM” for the selected “Assembly” object “Engine”.



Picture 13: Action “Expand Single Level”

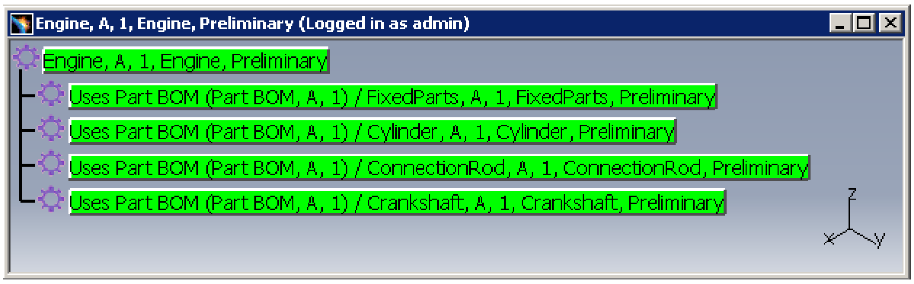
The related objects for the selected “Assembly” object “Engine” are shown in *Picture 14: Result of expand single level*.

In the product structure tree you can see the objects and relations. The type of the relation is displayed in braces in the line of the relation.



Picture 14: Result of expand single level

When you have switched off the display of the relations in the options (see *Options - PDM Relations*) then the expand tree looks like displayed in *Picture 15: Result of expand single level without relations*.



Picture 15: Result of expand single level without relations

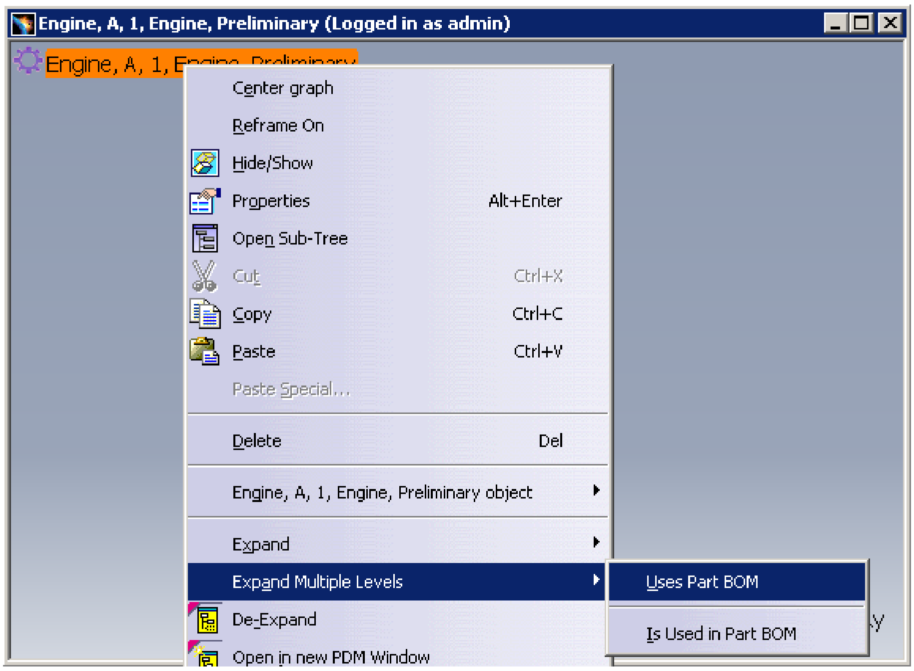
The type of the relation, the revision, and the generation of the relation are displayed in the braces in the line of the relation. The attributes can be configured in the PDM Workbench configuration file.

Expand Multiple Levels

You also might expand a relation direction in multiple levels from the selected object.

Select the object from which you want to expand and open the context menu by clicking on the right mouse button. Select the context action “Expand Multiple Levels”. The “Expand Multiple Levels” sub menu offers the possible relation directions for expansion from the selected object.

In the example in *Picture 16: Action “Expand Multiple Levels”* you select the direction “Uses Part BOM” for the selected “Assembly” object “Engine”.

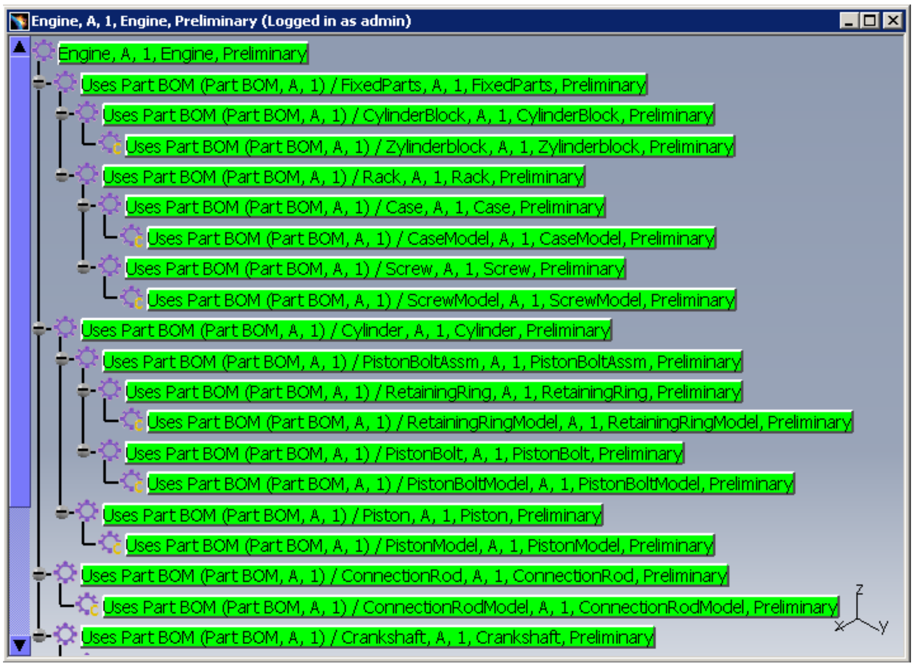


Picture 16: Action “Expand Multiple Levels”

The related objects in multiple levels are shown for the selected “Assembly” object “Engine” in *Picture 17: Result of expand multiple levels*.

In the product structure tree you can see the objects and relations. The type of the relation, the revision, and the generation are displayed in braces in the line of the relation.

When you have switched off the display of the relations in the options (see *Options - PDM Relations*) then the relation objects will not be displayed in the expanded product structure tree.

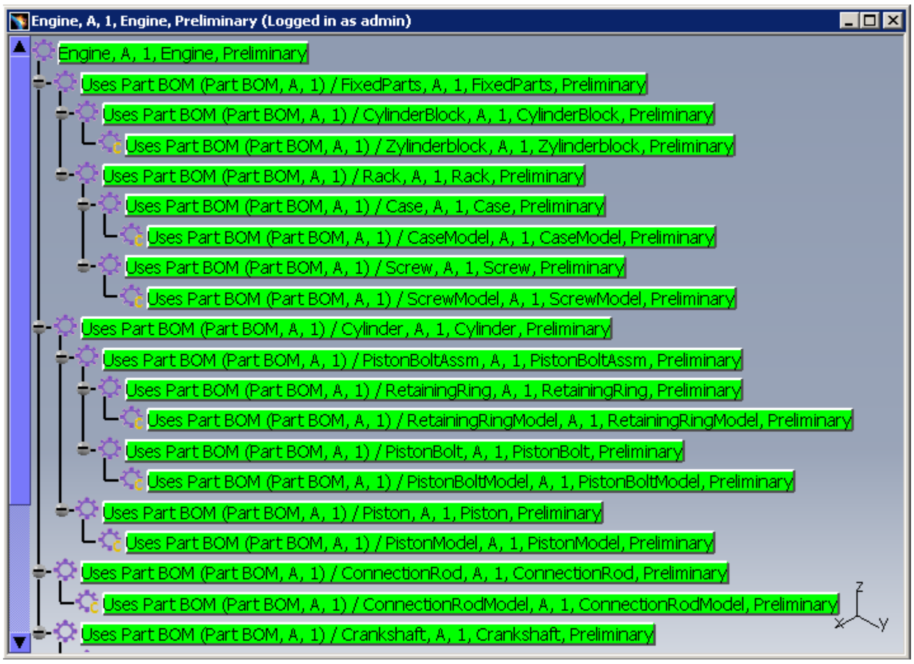


Picture 17: Result of expand multiple levels

De-Expand

When you want to reduce the displayed PDM structure then you can de-expand parts of the structure.

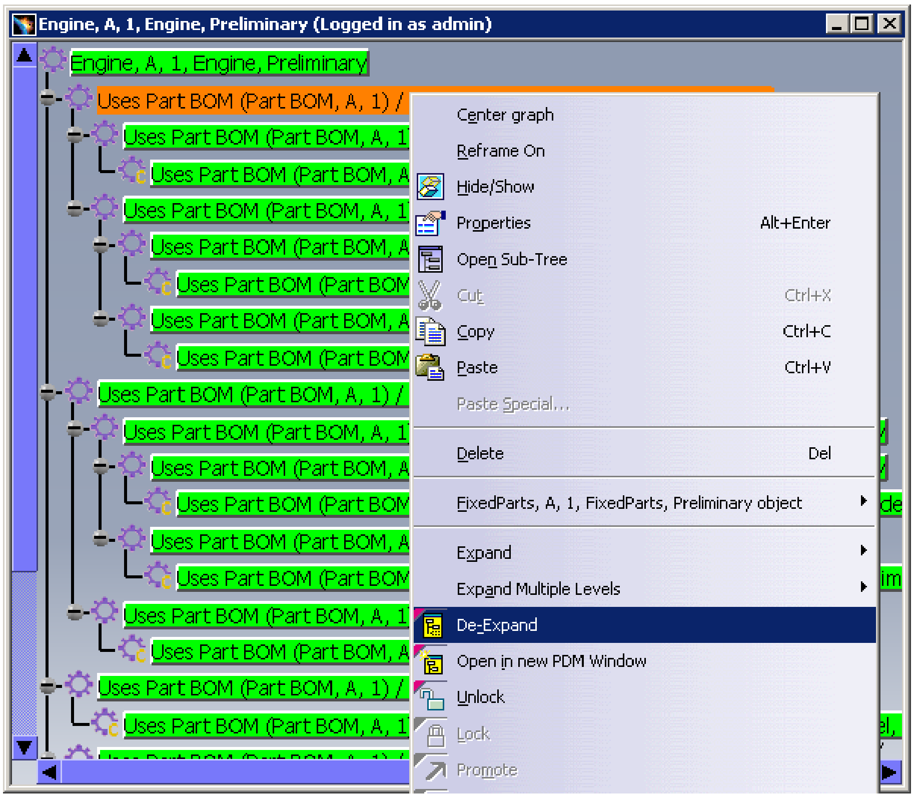
In *Picture 18: PDM structure before the De-Expand* you can see the PDM structure before the de-expand.



Picture 18: PDM structure before the De-Expand

First you select the root element of the sub structure which you want to de-expand. You also might de-expand a single object. Then you open the context menu by clicking the right mouse button and you select the action “De-Expand”.

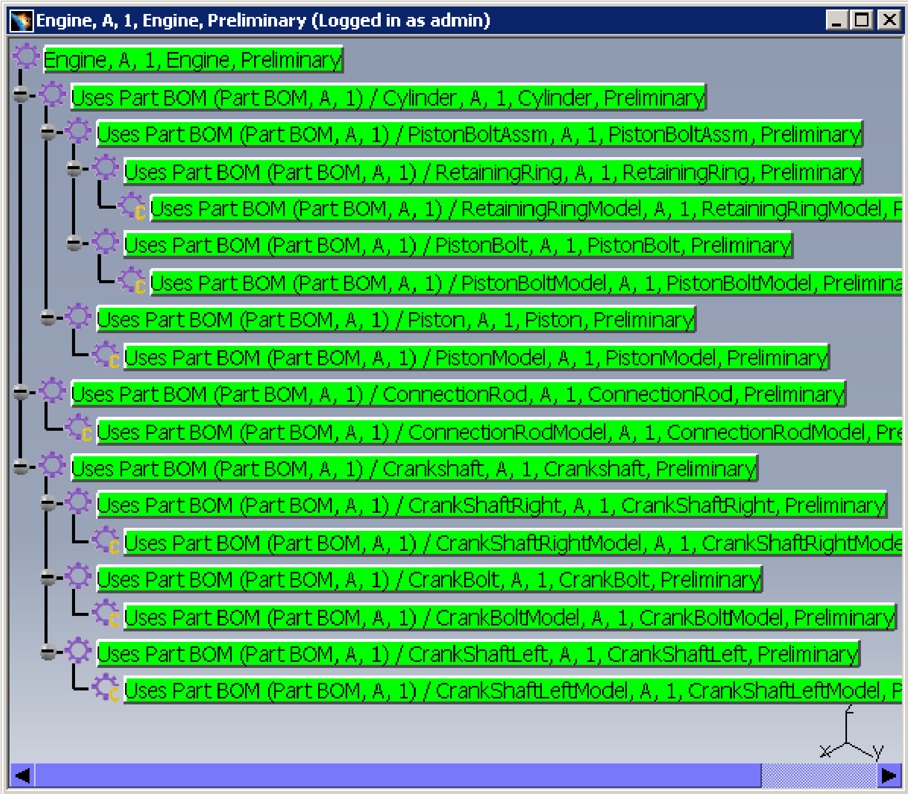
In the example in *Picture 19: Action “De-Expand”* you select the object “FixedParts_Assm” as root of the sub structure to be de-expanded.



Picture 19: Action “De-Expand”

In *Picture 20: PDM structure after the De-Expand* you see that the selected sub structure with the root element “FixedParts_Assm” is de-expanded as it is no longer displayed in the PDM structure.

But remember that the PDM structure will not be modified by this de-expand. The sub structure will not be deleted from the PDM structure.

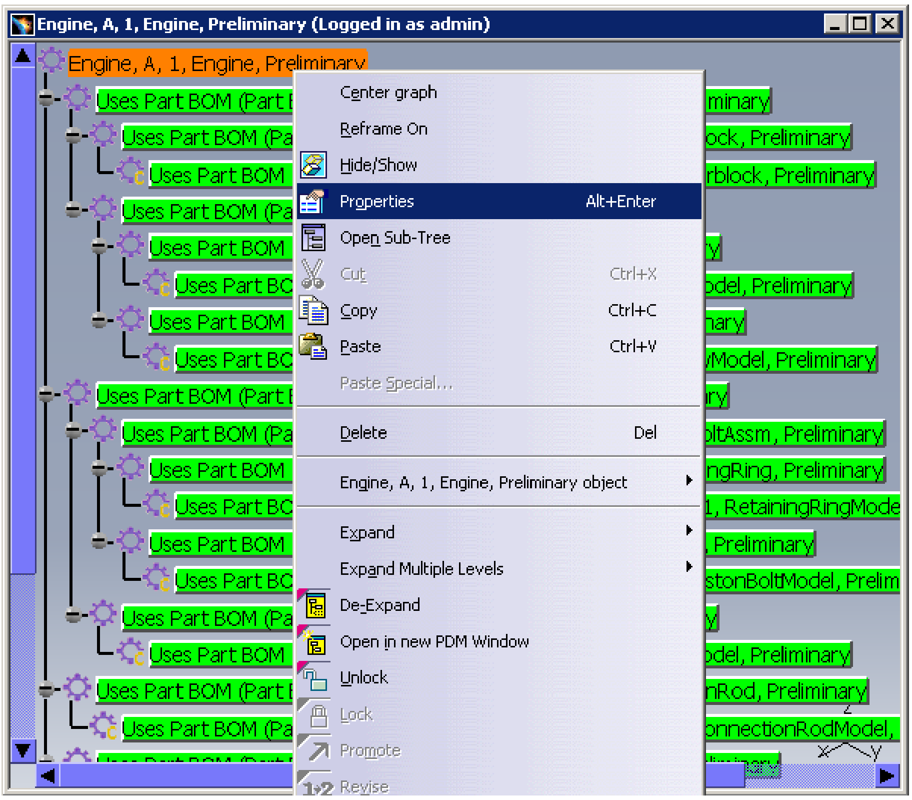


Picture 20: PDM structure after the De-Expand

Properties

The PDM properties of the objects in the PDM Workbench window are transferred from the PDM system when the objects are displayed in the PDM Workbench window for the first time, e.g. when they are shown as a result of a query or an expand action.

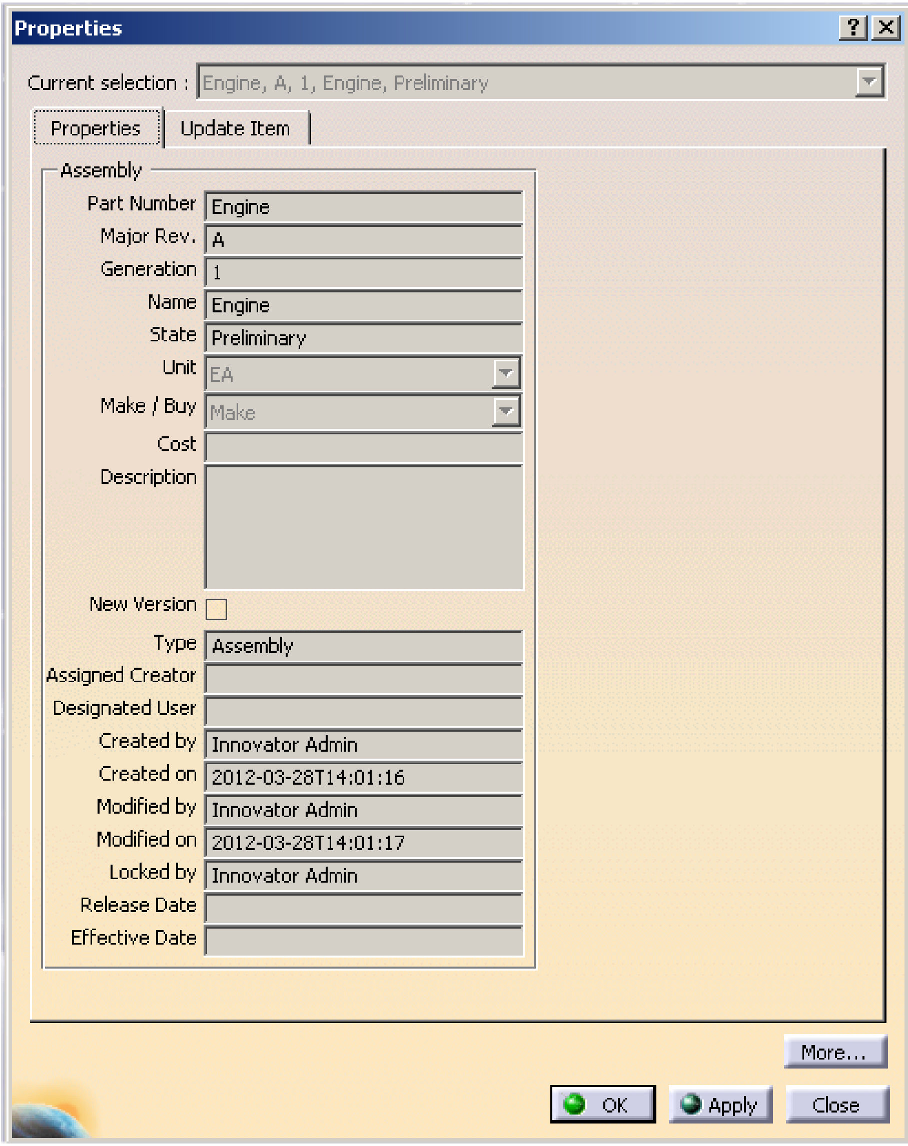
You take a look at those properties and/or change these properties when opening the “Properties” dialog. Therefore click on the right mouse button. Now the context menu opens and you select the action “Properties” (see *Picture 21: Action “Properties”*).



Picture 21: Action "Properties"

The "Properties" dialog will be opened (see *Picture 22: "Properties" dialog – tab "Properties"*). It consists of two tabs.

In the tab "Properties" the values of the attributes are shown on a grey background. This indicates that the values cannot be changed. All properties that are shown within this dialog can be specified within the PDM Workbench configuration file.



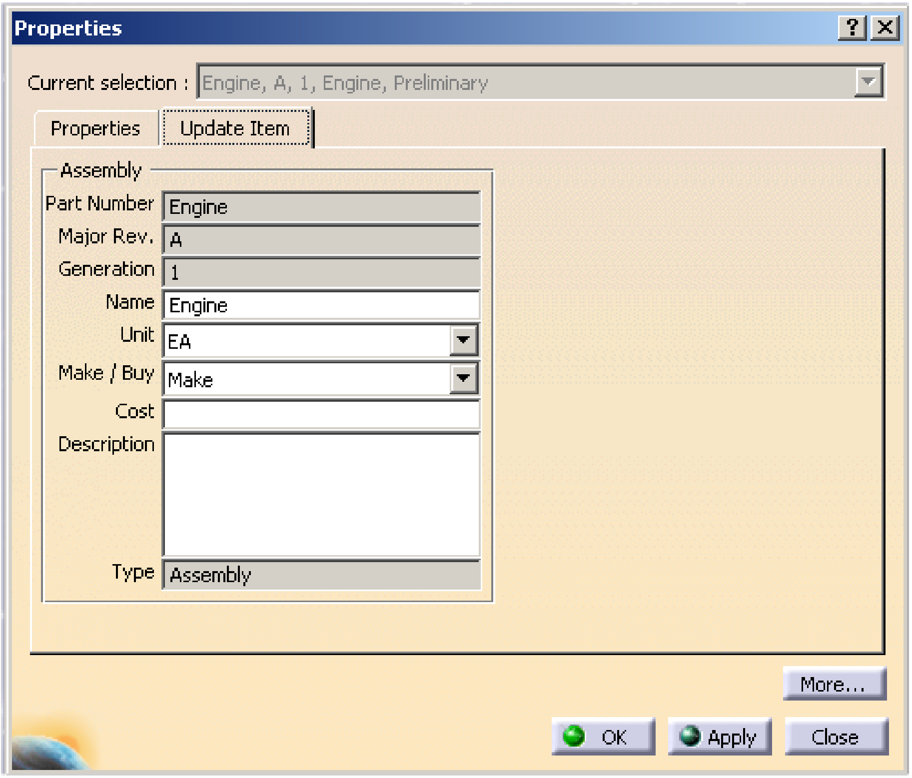
Picture 22: "Properties" dialog – tab "Properties"

In the tab "Update Item" those attribute values shown on a grey background cannot be changed. Those presented with the white background can be changed (see *Picture 23: "Properties" dialog – tab "Update Item"*).

Required attributes are marked with an asterisk (*) on the right side of the input field.

There are several types of attributes presented on corresponding widgets:

- Free text can be inserted.
- One value can be selected in a single selector list or combo box for instance or any other widget.
- Multiple values can be selected in a multi selector list or other widget types supporting this feature.
- The value can be marked or unmarked in a check box for instance.



Picture 23: "Properties" dialog – tab "Update Item"

Please use the "OK" button to close the "Properties" dialog.

In the tab "Update Item" the required attributes get checked for being satisfied. If not you will get an error window that describes your fault.

The object gets updated in the PDM system database according to the Update dialog values. When the update is not possible in the PDM system (because of a wrong value of an attribute or different reasons) then you will get an error window.

The PDM system properties of a CATPart and CATDrawing as top level object of the CATIA V5 window cannot be inspected with the context menu. You have to use the "PDM Properties" button  within the "PDM Workbench context commands" toolbar. The values of the attributes are in read-only mode. You are not able to make changes on the PDM system attributes in this dialog.

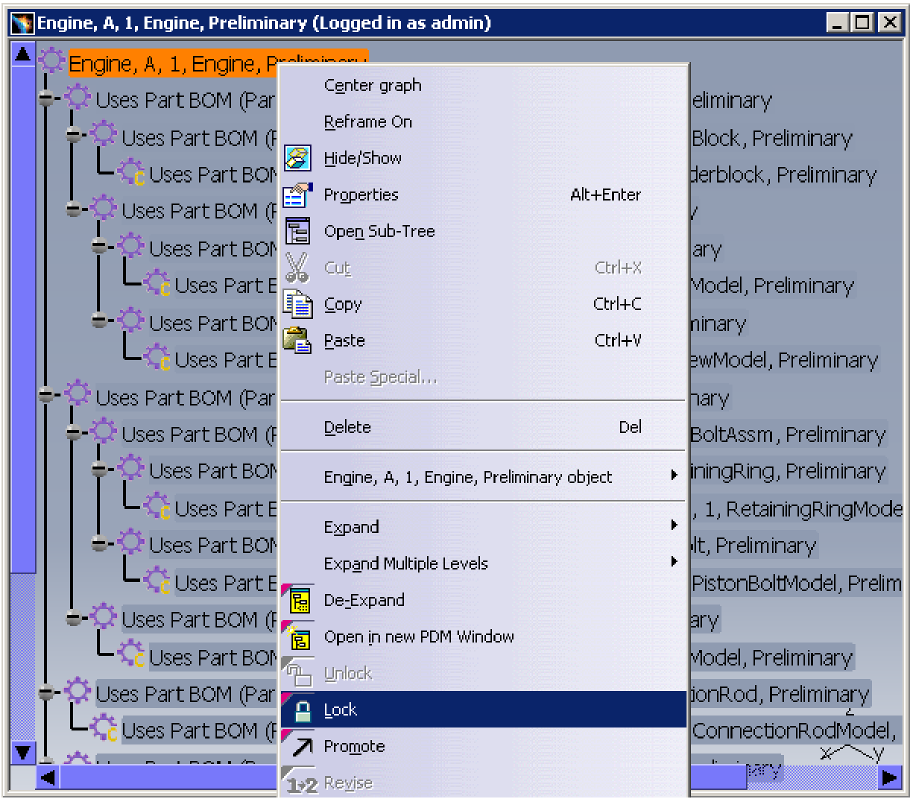
Lock

Lock of the Part in the PDM Workbench window

You have to lock the PDM objects prior to changing and updating them.

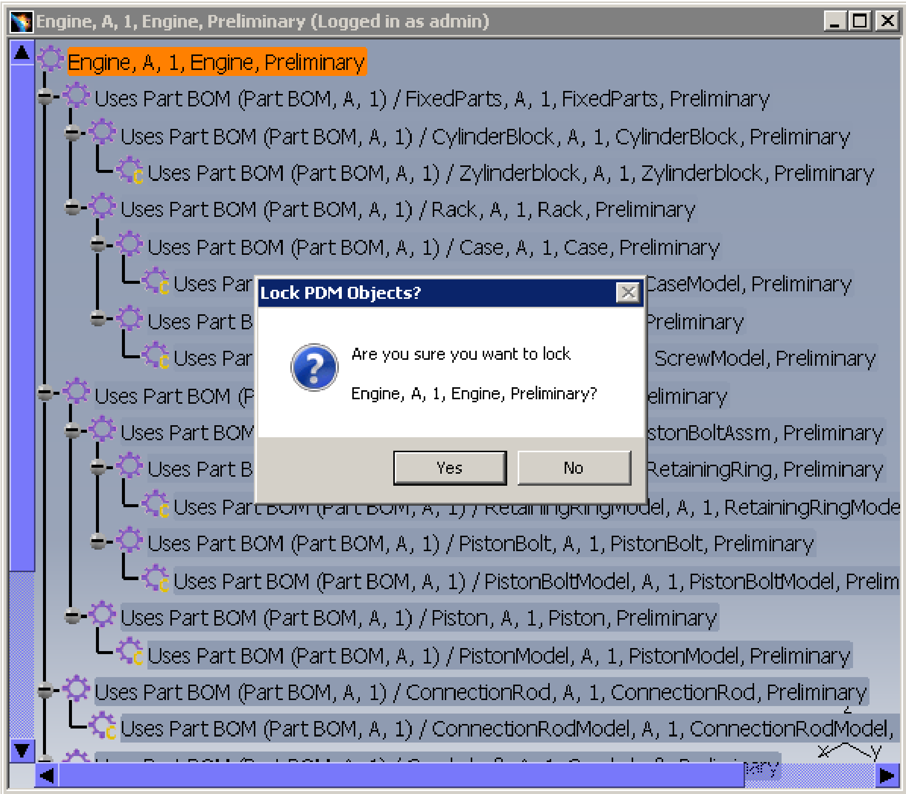
So you select the object in the PDM Workbench window and click the right mouse button. In the context menu you select the context action "Lock" (see *Picture 24: Action "Lock"*). This action is only active in the context menu when it is possible for the selected object. Otherwise it will be deactivated in the context menu.

A multi-select of objects is also supported.



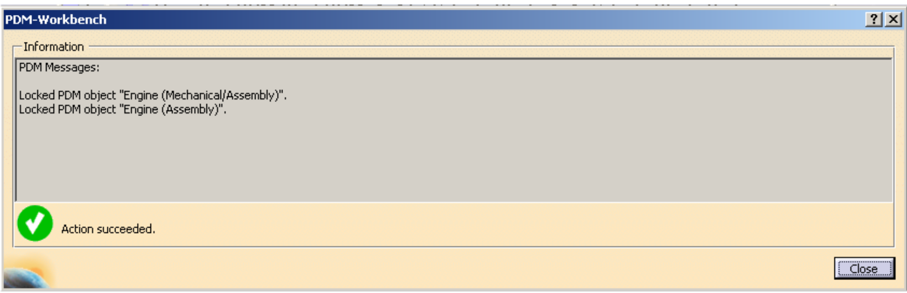
Picture 24: Action “Lock”

The confirmation message for the Lock opens (see *Picture 25: Confirm the Lock of the part*). If it is the correct object that you want to lock then just press the “Yes” button.



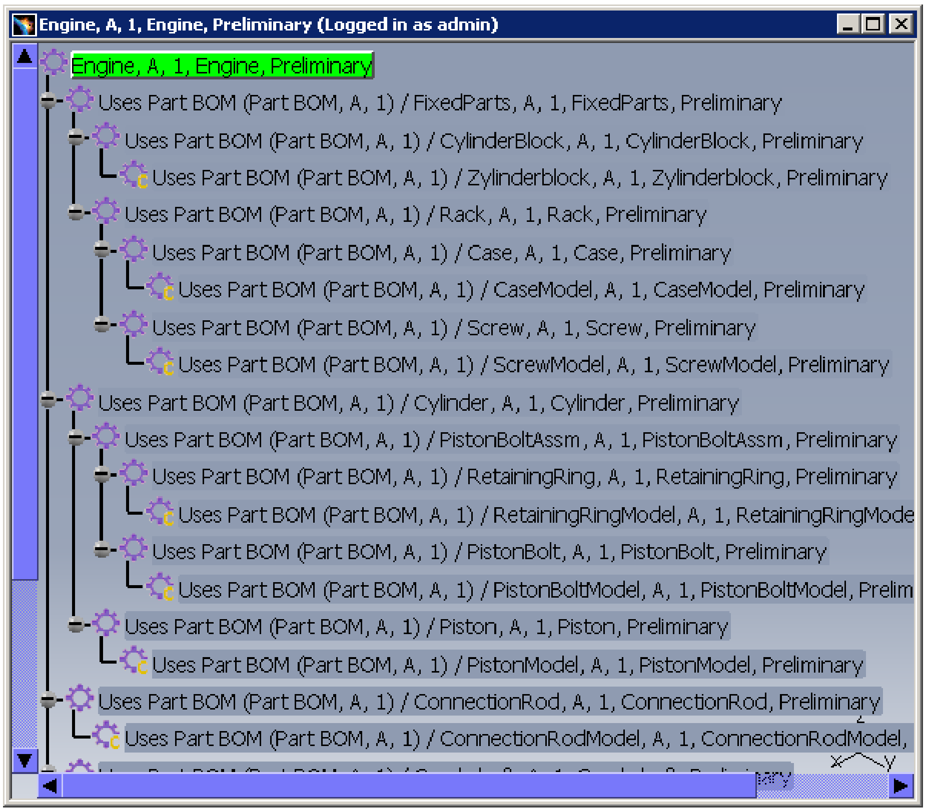
Picture 25: Confirm the Lock of the part

The selected object and the attached documents will be locked by the user (see *Picture 26: Object is locked*).



Picture 26: Object is locked

The background color of the locked object changed to green in the PDM Workbench window (see *Picture 27: Locked object*).



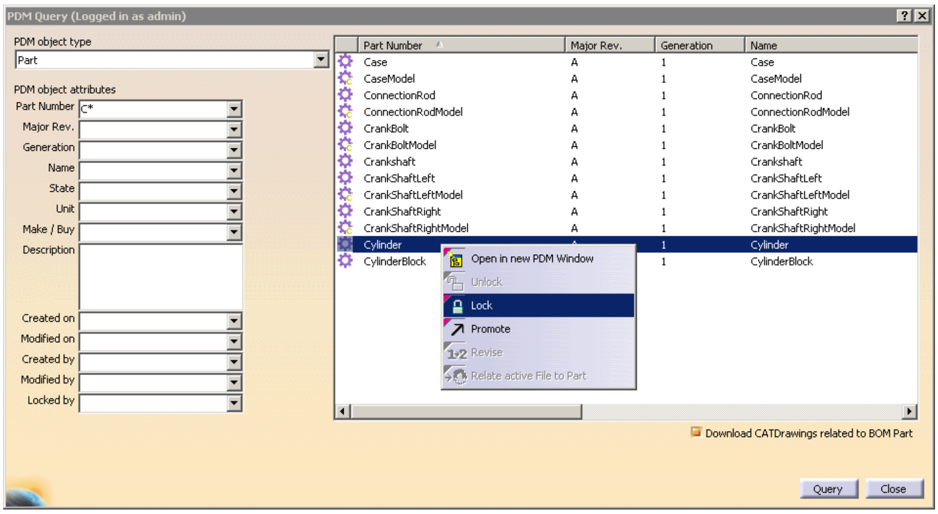
Picture 27: Locked object

The green background color of the locked object indicates that the object in the PDM system can be changed by the user.

Lock of the object in the Query dialog

It is also possible to lock the object from the query result list of the "Query" dialog. In the example in *Picture 28: Action "Lock" in the query result list* you select the object and click on "Lock" in the context menu. This action is only active in the context menu when it is possible for the selected object. Otherwise it will be deactivated in the context menu.

A multi-select of objects is also supported.

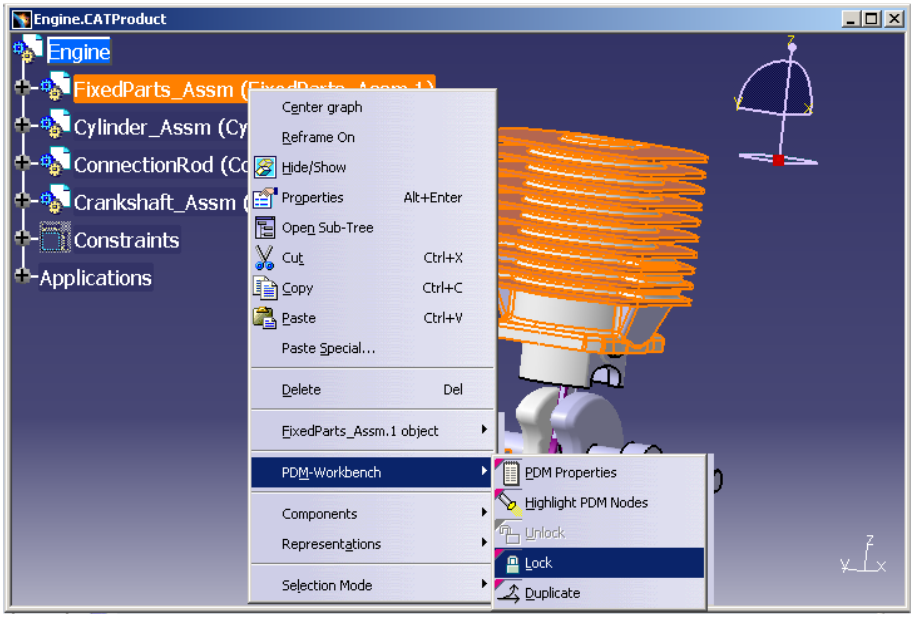


Picture 28: Action "Lock" in the query result list

Lock of the Document in the CATIA V5 window

You have to lock the PDM objects prior to changing and updating them.

You select the object in the CATIA V5 window and click the right mouse button. In the context menu you select the context action "PDM Workbench→Lock" (see *Picture 29: Action "Lock" in the CATIA V5 window*).



Picture 29: Action "Lock" in the CATIA V5 window

For further details of the lock dialogs please refer to the chapter *Lock of the Part in the PDM Workbench window*.

The lock of a CATPart and CATDrawing as top level object of the CATIA V5 window cannot be done with the context menu. You have to use the "Lock" button of the "PDM Workbench context commands" toolbar. The icons in this toolbar are only repainted (e.g. switch from "Lock" to "Unlock") when you newly activate the CATIA V5 window.

A multi-select of objects is also supported.

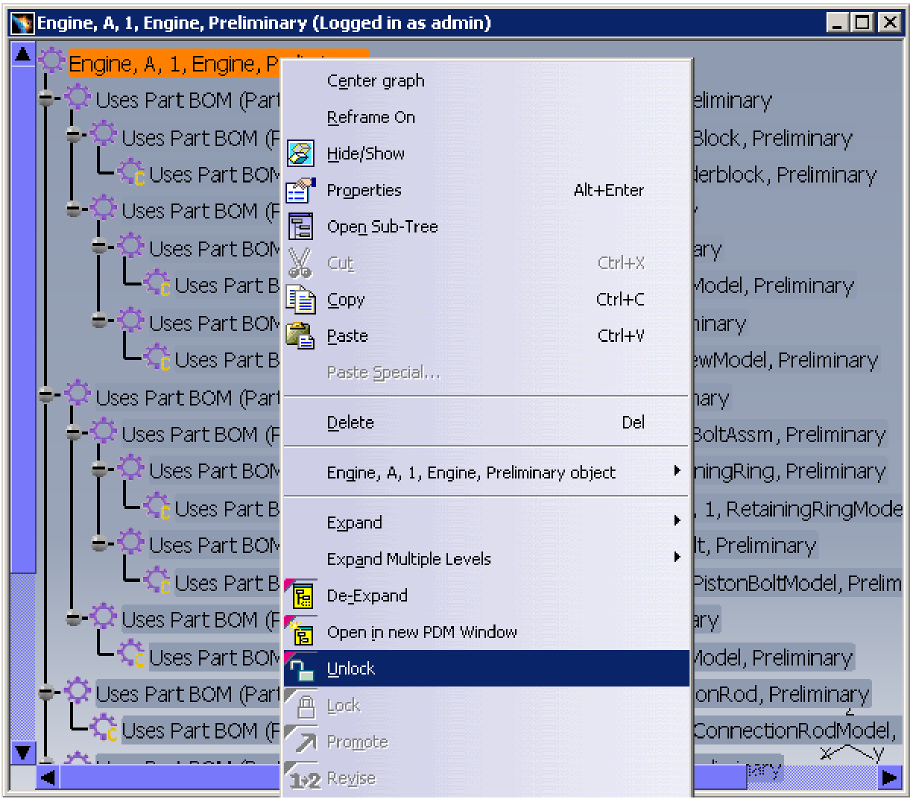
Unlock

Unlock of the Part in the PDM Workbench window

When an object is locked by you then you have to unlock it in the PDM system to make it available for all other users.

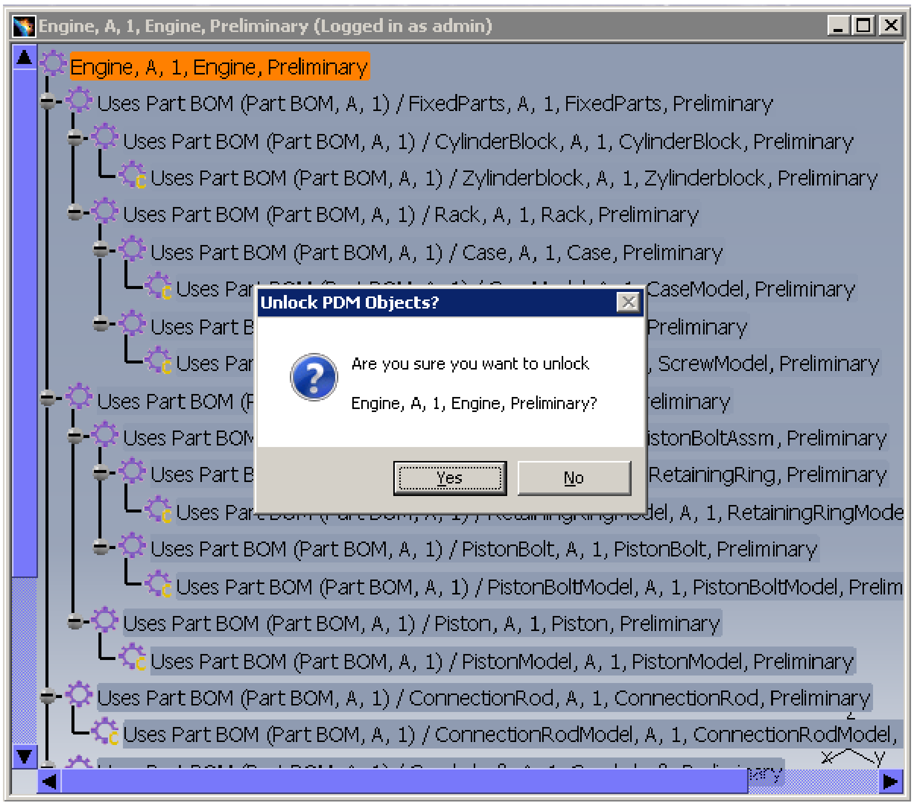
You select the object in the PDM Workbench window and click the right mouse button. In the context menu you select the “Unlock” context action (see *Picture 30: Action “Unlock”*). This action is only active in the context menu when it is possible for the selected object. Otherwise it will be deactivated in the context menu.

A multi-select of objects is also supported.



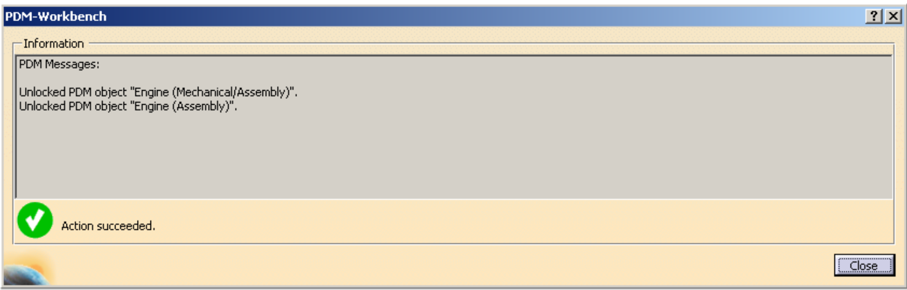
Picture 30: Action “Unlock”

The confirmation message for the “Unlock” opens (see *Picture 31: Confirm the Unlock of the part*). If it is the correct object that you want to unlock then just select the “Yes” button.



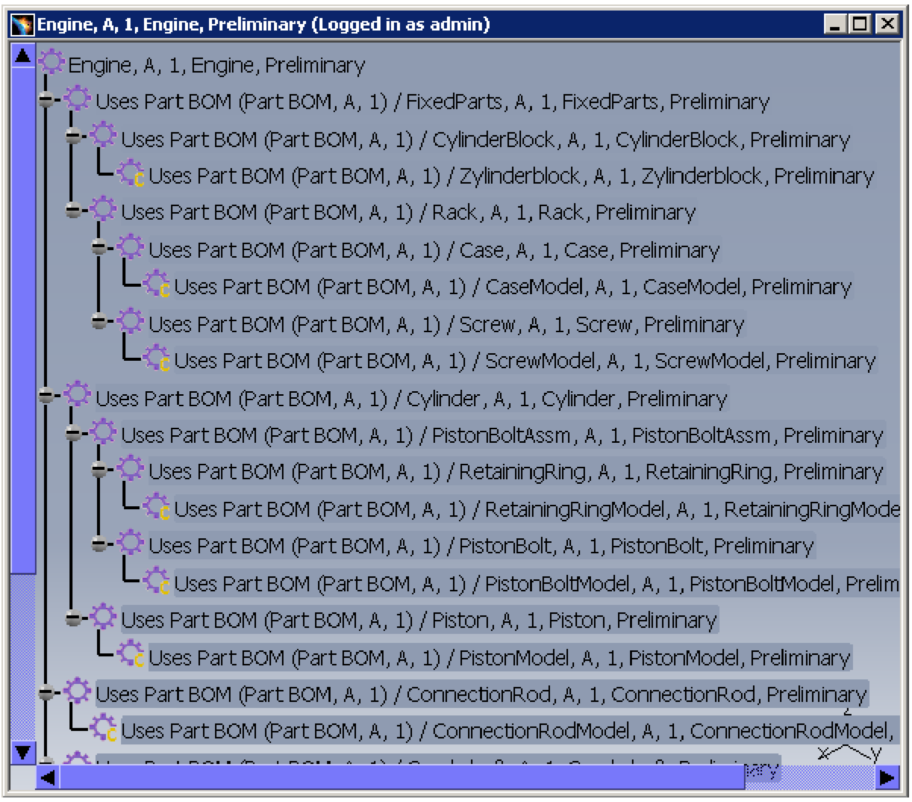
Picture 31: Confirm the Unlock of the part

The selected object and the attached documents will be unlocked by the user (see *Picture 32: Object is unlocked*).



Picture 32: Object is unlocked

The background color of the unlocked object changed to blank in the PDM Workbench window (see *Picture 33: Unlocked object*).



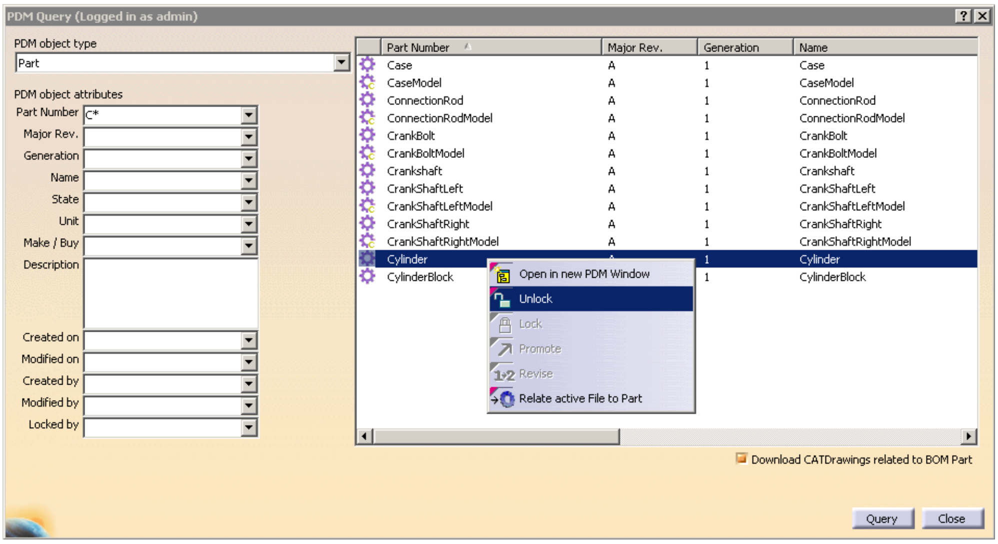
Picture 33: Unlocked object

The blank background color of the unlocked object indicates that the object in the PDM system cannot be changed by the user.

Unlock of the object in the Query dialog

It is also possible to unlock the object from the query result list of the "Query" dialog. In the example in *Picture 34: Action “Unlock” in the query result list* you select the object and click on "Unlock" in the context menu. This action is only active in the context menu when it is possible for the selected object. Otherwise it will be deactivated in the context menu.

A multi-select of objects is also supported.

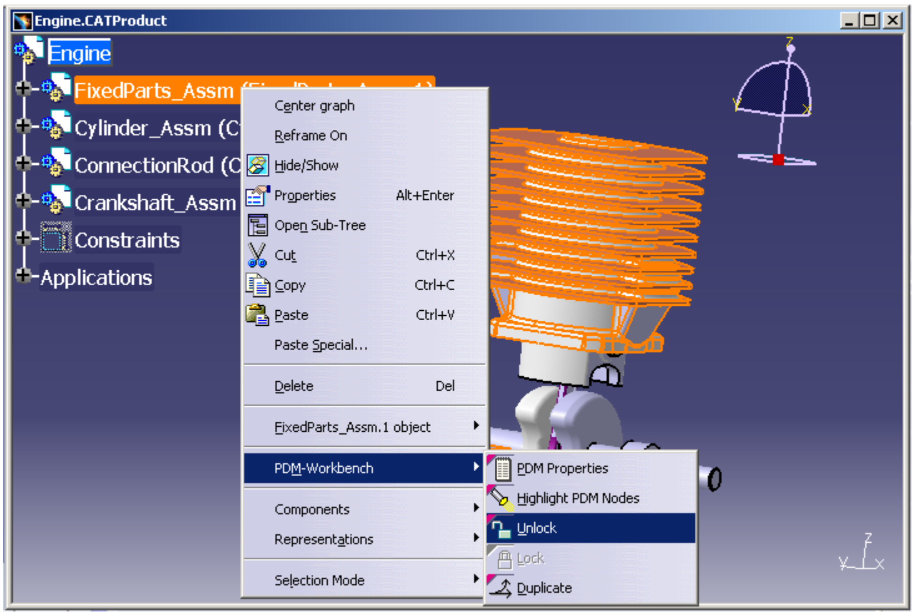


Picture 34: Action “Unlock” in the query result list

Unlock of the Document in the CATIA V5 window

When an object is locked by you then you can unlock it in the PDM system to make it available for all other users.

You select the object in the CATIA V5 window and click the right mouse button. In the context menu you select the context action “PDM Workbench→Unlock” (see *Picture 35: Action “Unlock” in the CATIA V5 window*).



Picture 35: Action “Unlock” in the CATIA V5 window

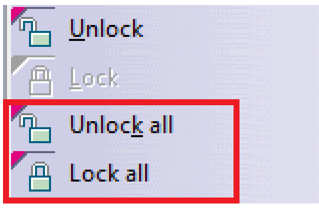
For further details of the “Unlock” dialogs please refer to the chapter *Unlock of the Part in the PDM Workbench window*.

The unlock of a CATPart and CATDrawing as top level object of the CATIA V5 window cannot be done with the context menu. You have to use the “Unlock” button within the “PDM Workbench context commands” toolbar. The icons in this toolbar are only repainted (e.g. switch from “Lock” to “Unlock”) when you newly activate the CATIA V5 window.

A multi-select of objects is also supported.

“Lock All” / “Unlock All”

It is possible lock or unlock all the items in a structure.



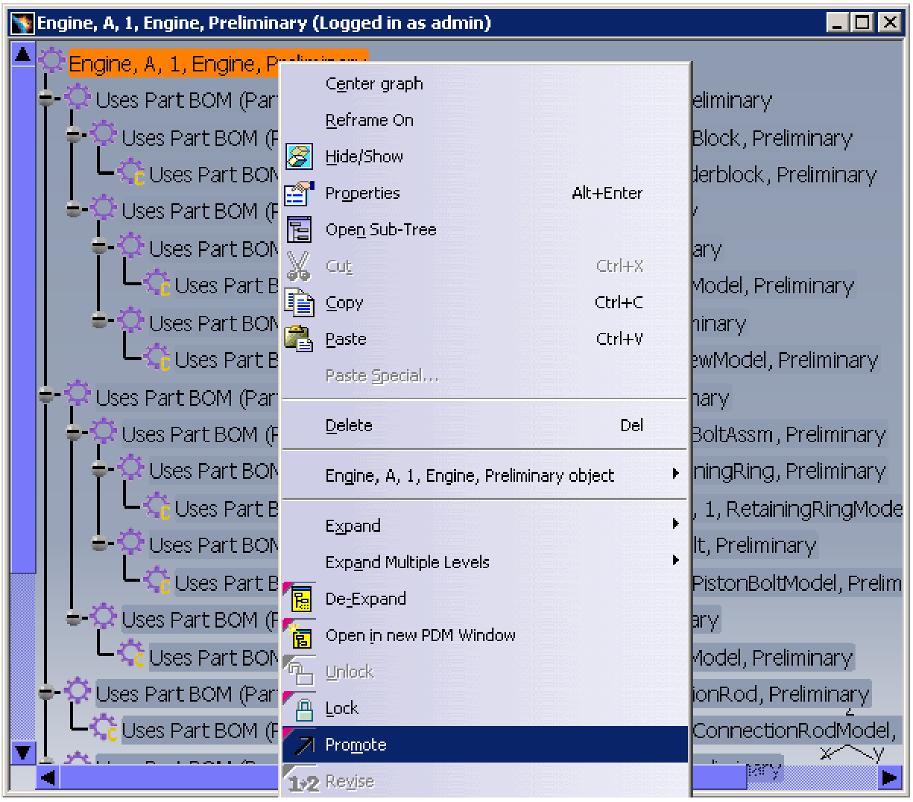
Picture 36: “Unlock all” / “Lock all” context menu items

Promote

The PDM objects can be promoted.

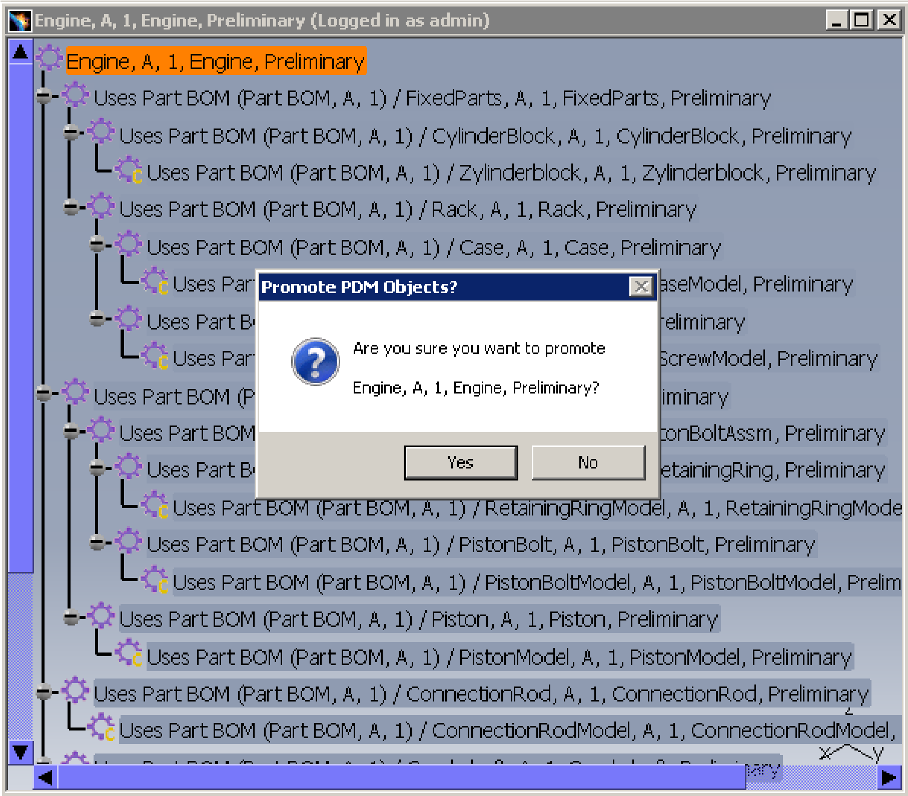
You select the object in the PDM Workbench window and click the right mouse button. In the context menu you select the “Promote” context action (see *Picture 37: Action “Promote”*).

The part and the document have to be unlocked in order to be promoted.



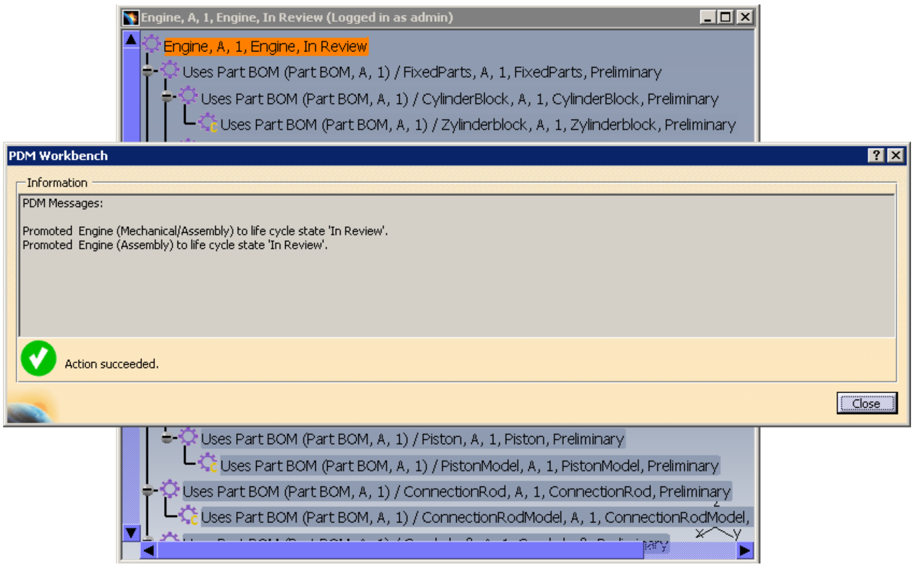
Picture 37: Action “Promote”

You will be asked if you really want to promote the objects. Please confirm with "Yes" (see *Picture 38: Confirm the “Promote" action*).



Picture 38: Confirm the "Promote" action

The selected object and the attached documents will be promoted. In this example from the life cycle state "Preliminary" to the state "In Review" (see *Picture 39: Object is promoted*).



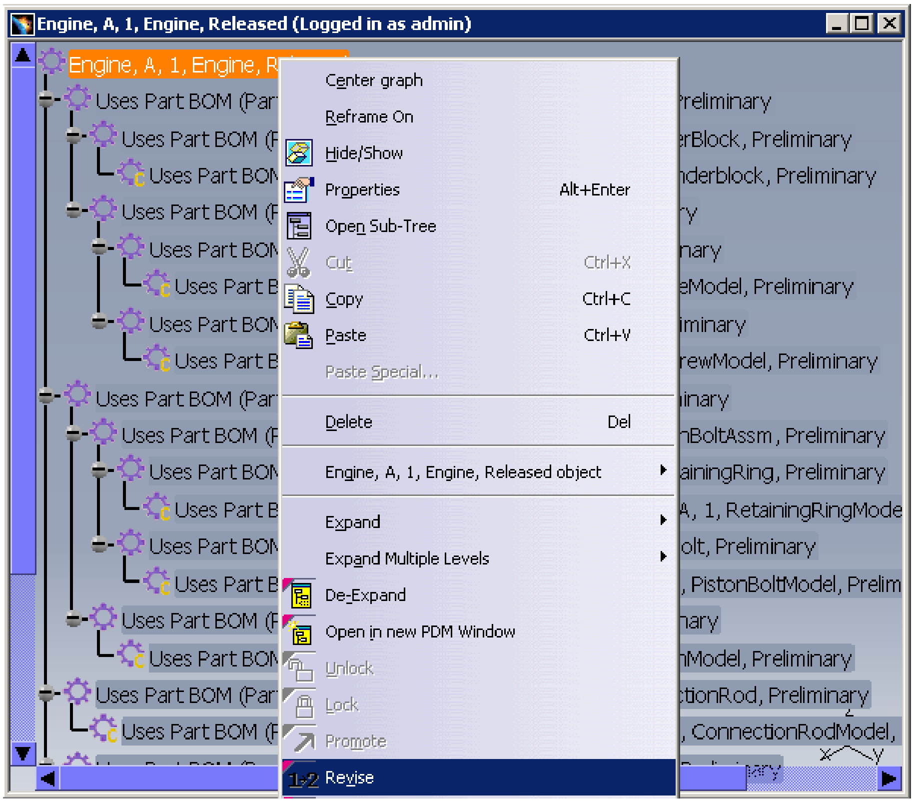
Picture 39: Object is promoted

Revise

The PDM objects can be revised if the item is in released mode.

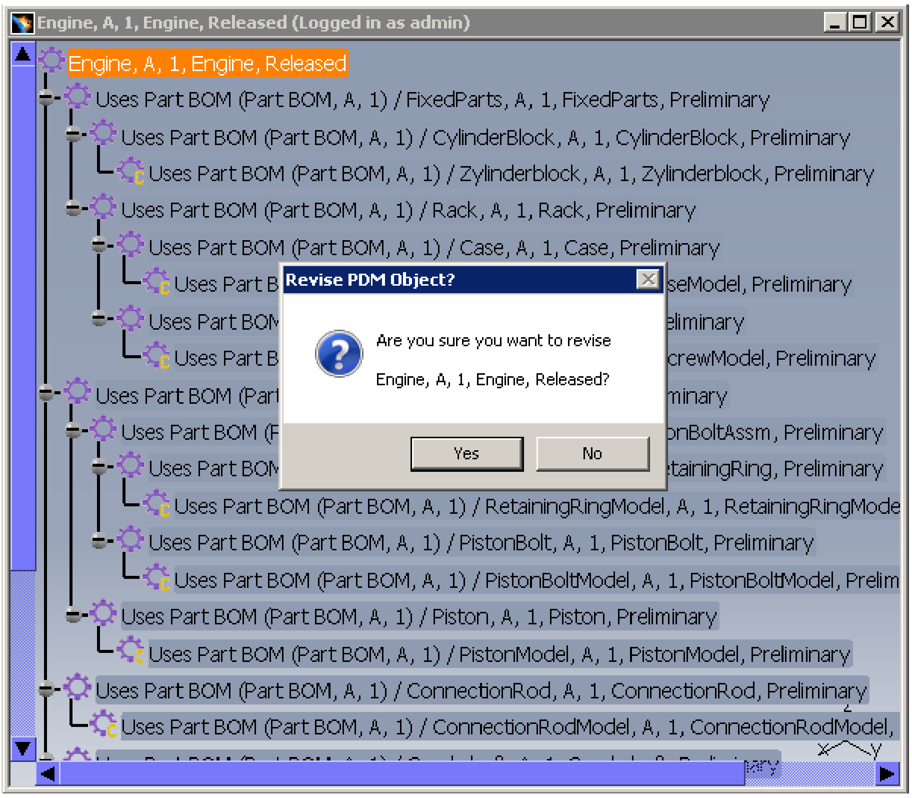
You select the object in the PDM Workbench window and click the right mouse button. In the context menu you select the "Revise" context action (see *Picture 40: Action "Revise"*).

The part and the document have to be released in order to be revised.



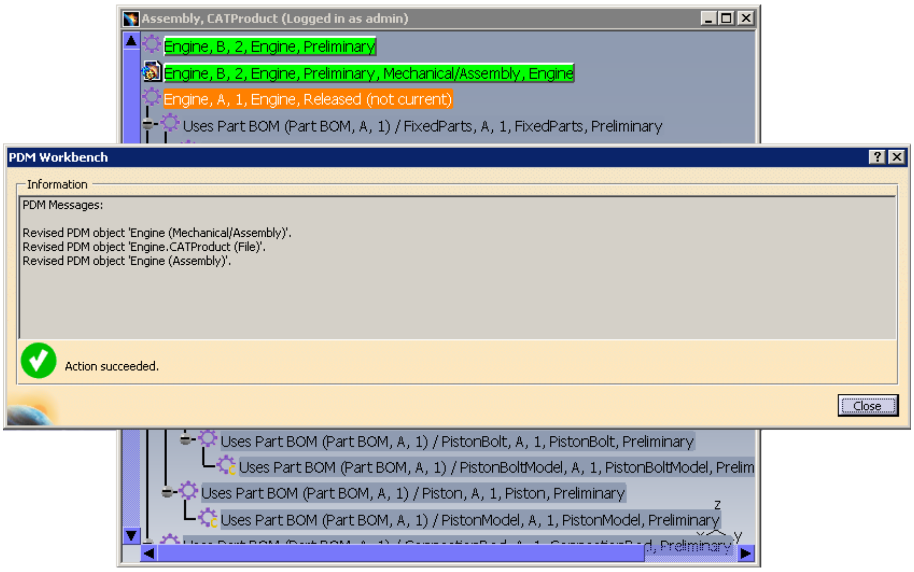
Picture 40: Action "Revise"

You will be asked if you really want to revise the object. Please confirm with "Yes" (see *Picture 41: Confirm the "Revise" action*).



Picture 41: Confirm the "Revise" action

The selected object will be revised (see *Picture 42: Object is revised*).



Picture 42: Object is revised

The new object was added in the window on the top.

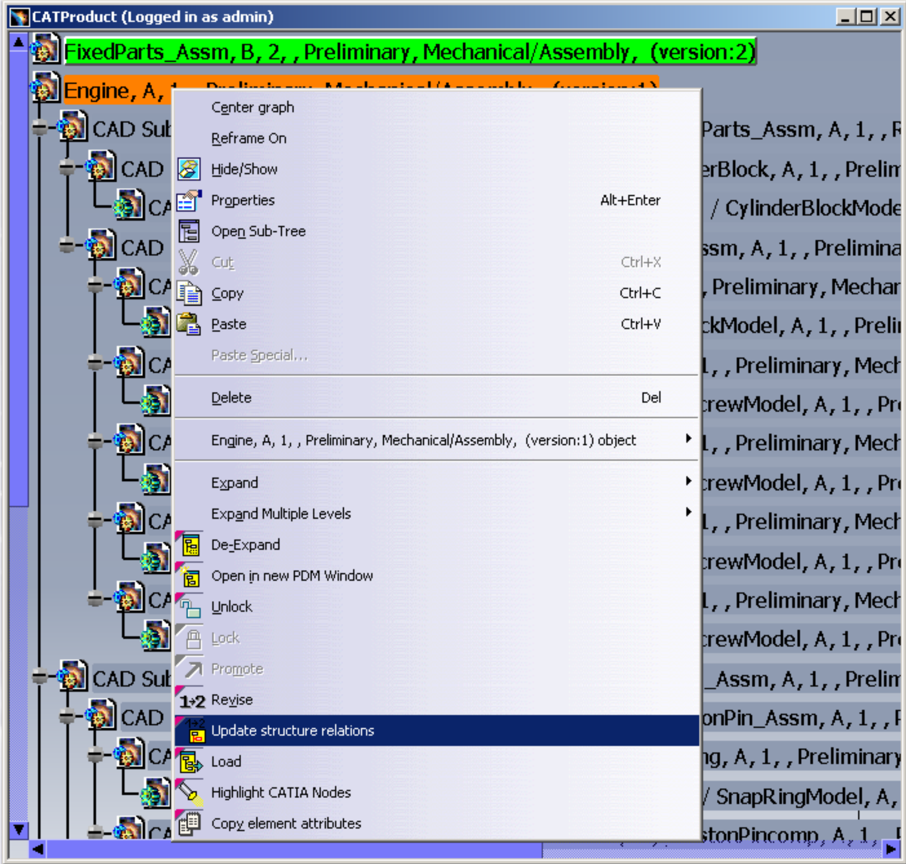
Update structure relations

This functionality is only available for the document data model.

You have the possibility to update an object with a new revision of an already used object.

In this example the document "FixedParts_Assm" has been revised from "A" to "B". The revision "A" was already used by the document "Engine".

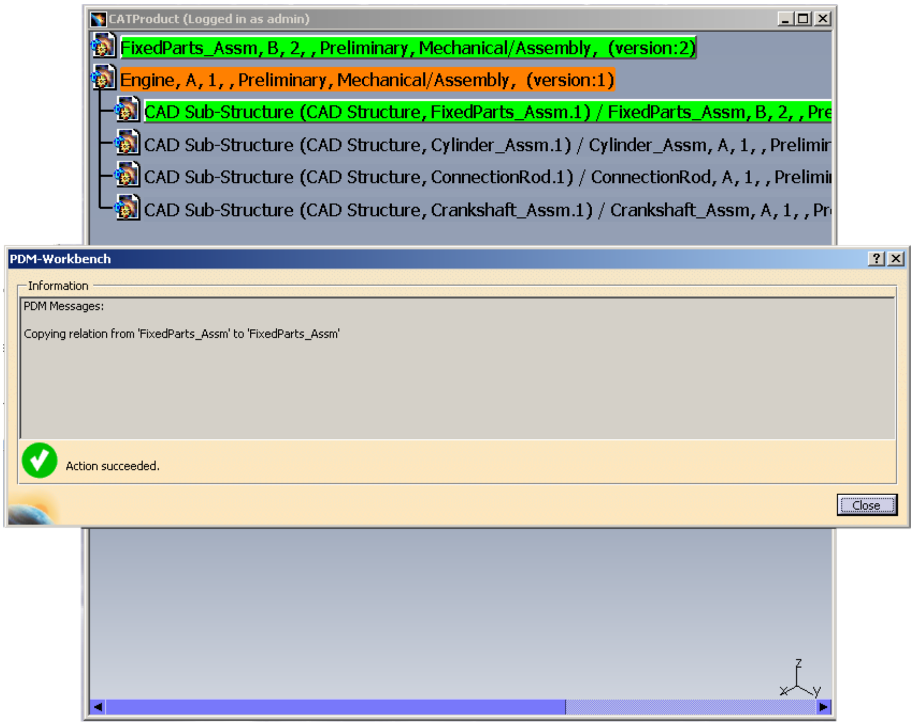
You have to select the CATIA Product "Engine" and choose “Update structure relations” (see *Picture 43: Action “Update structure relations”*).



Picture 43: Action “Update structure relations”

Now the new revisions of the used objects are related to this object and the relations to the old revisions are deleted.

You can see that the revision "B" of the "FixedParts_Assm" is used by the "Engine" now (see *Picture 44: Updated structure relations*).



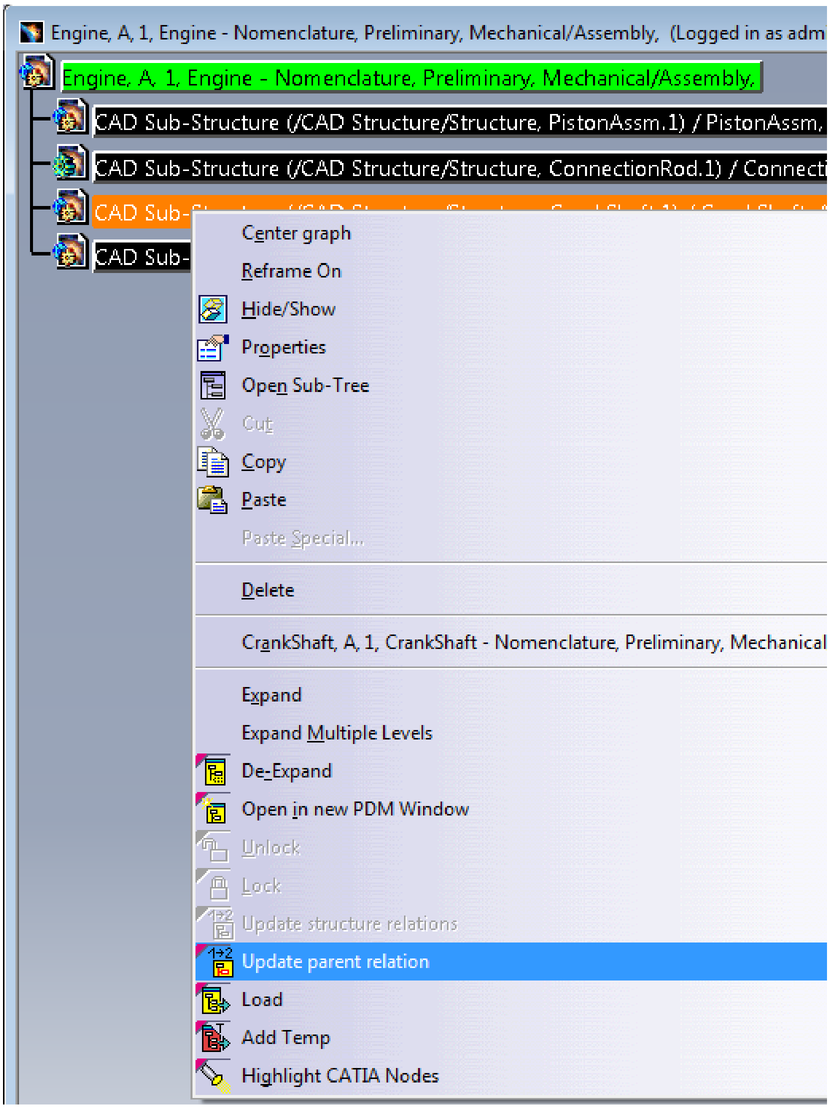
Picture 44: Updated structure relations

Update parent relation

This functionality is only available for the document data model.

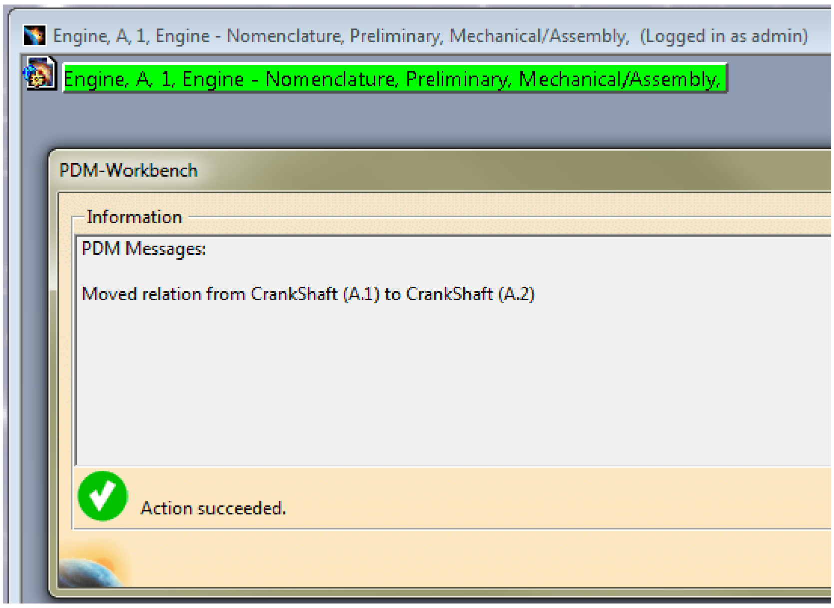
Like “Update structure relations”, this functionality updates the structure relations of a used document to the latest version of that document. The difference is that in this case, only the selected child node (all instances of the document) is updated, not all the direct child nodes of the parent document.

The context action is only available for child nodes in a structure, not for the root node.



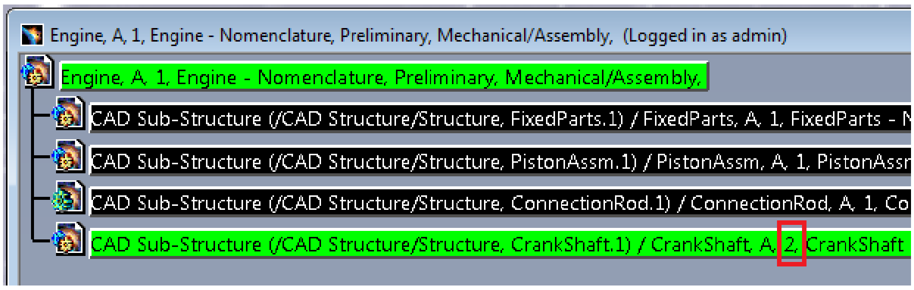
Picture 45: Action “Update parent relation”

The relation has been updated (see *Picture 47: Updated structure relation*).



Picture 46: Updated relation in PDM

The structure relations will have to be expanded again to show the current status (see *Picture 47: Updated structure relation*):

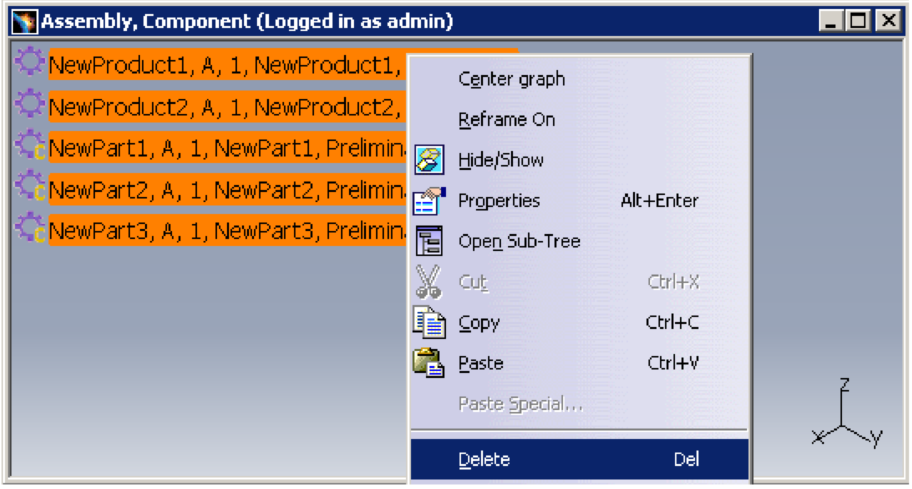


Picture 47: Updated structure relation

Delete

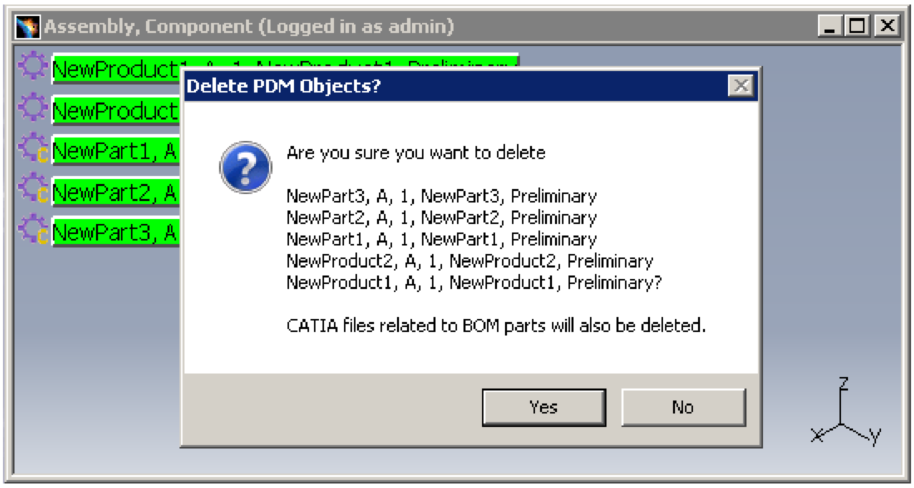
Existing PDM objects can be deleted from the PDM Workbench. Delete is a set based operation, which means multiple objects can be deleted in one action.

Select objects in the PDM Workbench window and from the contextual menu choose "Delete" (see *Picture 48: Action "Delete"*).



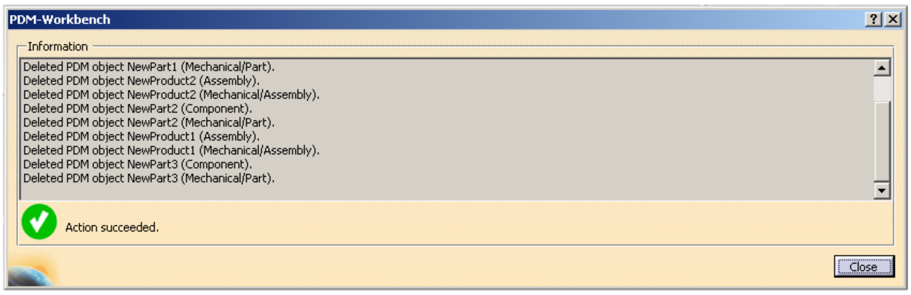
Picture 48: Action "Delete"

A confirmation message box is displayed listing the objects which will be deleted. When you confirm this dialog with "Yes" the objects will be deleted.



Picture 49: Confirm Delete objects

The operation result dialog is displayed containing error or success messages.



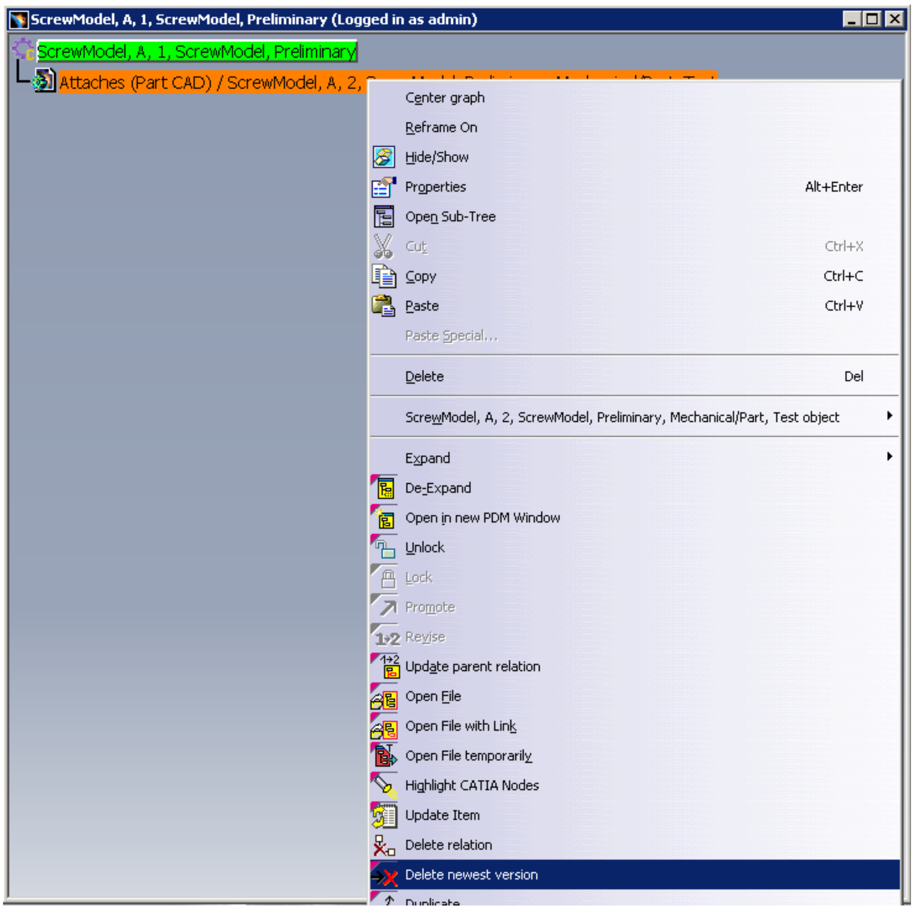
Picture 50: Delete result window

When you delete a part object the attached documents will be deleted, too.

Delete newest version

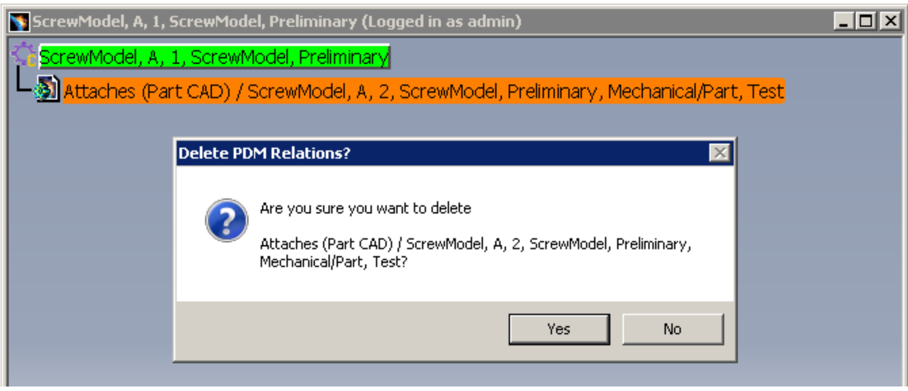
You can delete the newest version of the file when there exist more than one version for the file and you see that you do not need this version anymore because you want to design the geometry a different way.

You have to select the last version of the document and click on the right mouse button. The context menu will be opened. There you select "Delete newest version" (see *Picture 51: Action "Delete newest version"*).



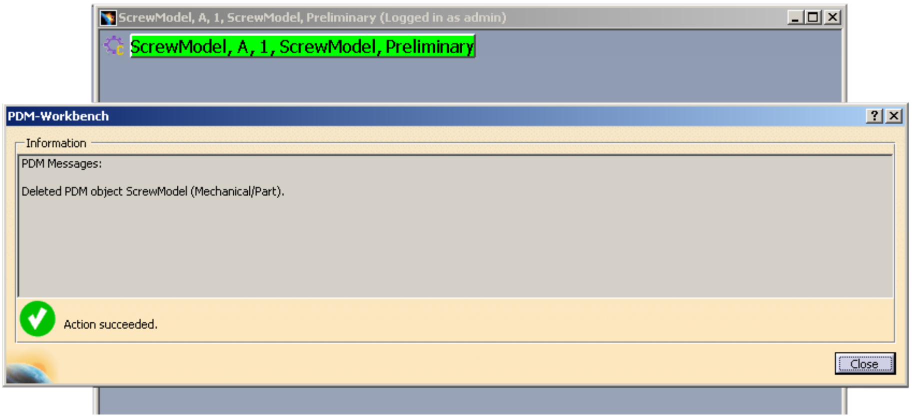
Picture 51: Action "Delete newest version"

Then you are asked to confirm the delete of the newest version. You have to click the "Yes" button (see *Picture 52: Confirm Delete newest version*).



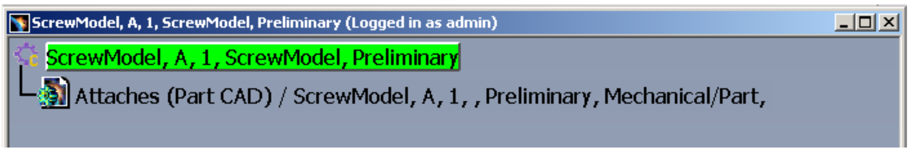
Picture 52: Confirm Delete newest version

The newest version will be deleted. The document will be removed from the window (see *Picture 53: Delete newest version result window*).



Picture 53: Delete newest version result window

You have to re-expand the document in order to see the document that is attached to the component (see *Picture 54: Re-Expand of the document*).



Picture 54: Re-Expand of the document

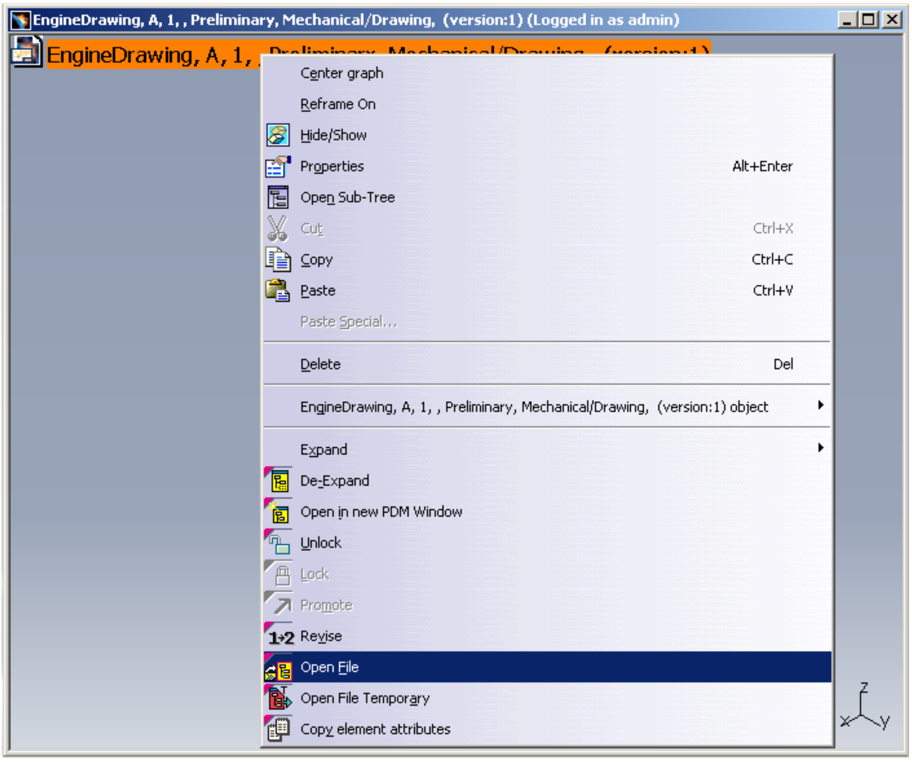
Open File

This functionality is only available for the document data model.

You can open a single CATIA V5 Drawing file existing in the PDM data base with the PDM Workbench in the native CATIA V5 window.

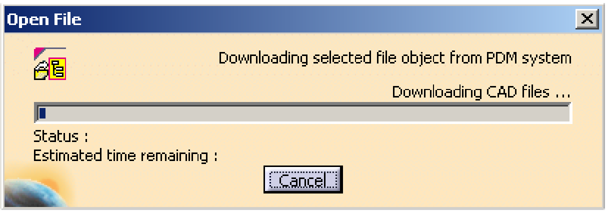
You can also open a single CATIA V5 Part or CATIA V5 Product file existing in the PDM data base in the native CATIA V5 window.

To open the file in CATIA V5 you select the PDM file object in the PDM Workbench window and click the right mouse button to open the context menu. There you select the context action “Open File” (see *Picture 55: Action “Open File”*).



Picture 55: Action "Open File"

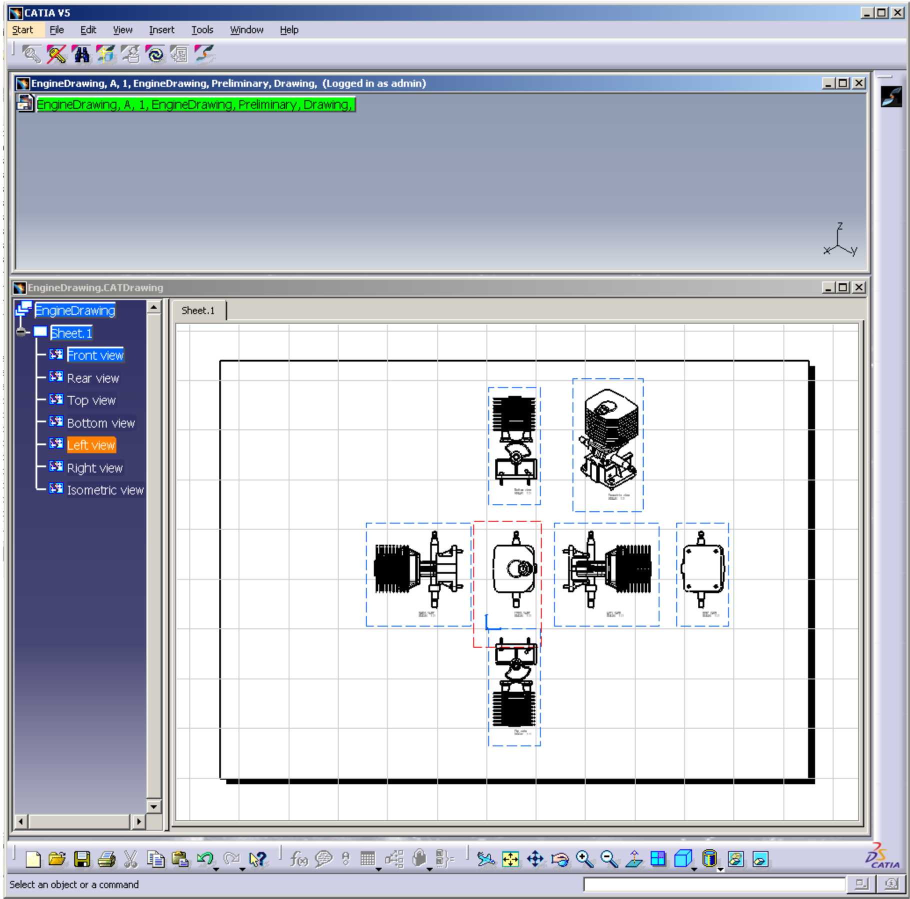
The PDM Workbench downloads the required CAD file to the client's special PDM Workbench exchange map. You can watch the download progress on the "Open File" progress bar (see *Picture 56: Open File - progress bar*).



Picture 56: Open File - progress bar

The geometry opens in its corresponding CATIA V5 native window (see *Picture 57: Split window after Open File – PDM Workbench node and CATIA drawing*).

In the above window (PDM Workbench window) you see the selected PDM file object. The window on the bottom shows the loaded CATDrawing.



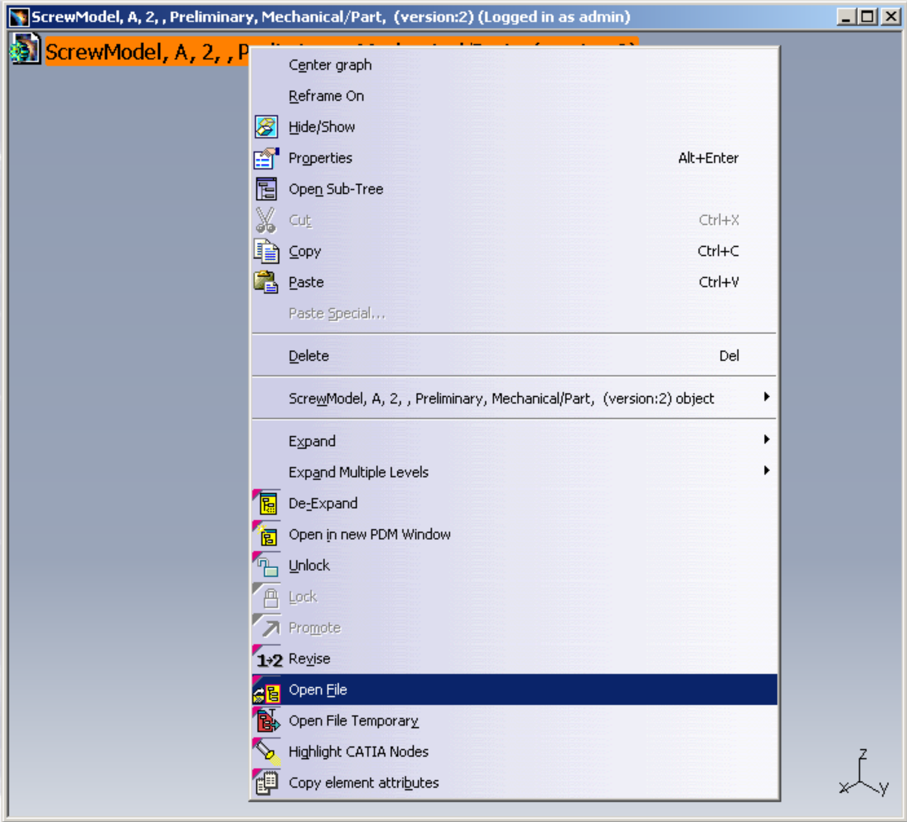
Picture 57: Split window after Open File – PDM Workbench node and CATIA drawing

Open File Temporary

The action "Open File Temporary" allows the user to visualize a temporary file of a different version together with the working version.

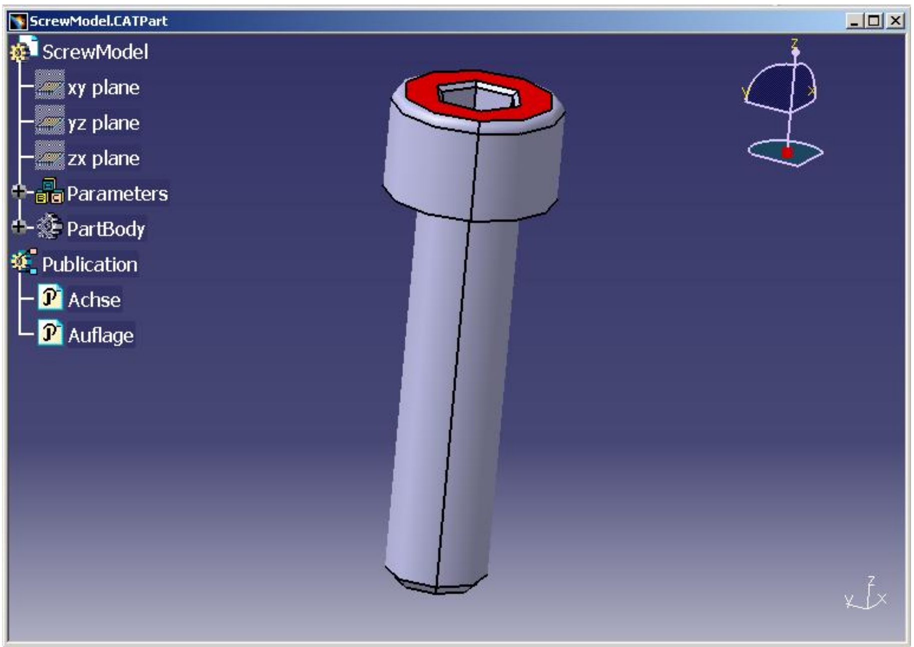
In the first step you load the current version of the file. In this example you open the version "2" of the "ScrewModel".

You select the object and click on the right mouse button. In the context menu you select the action "Open File" (see *Picture 58: Action "Open File"*).



Picture 58: Action “Open File”

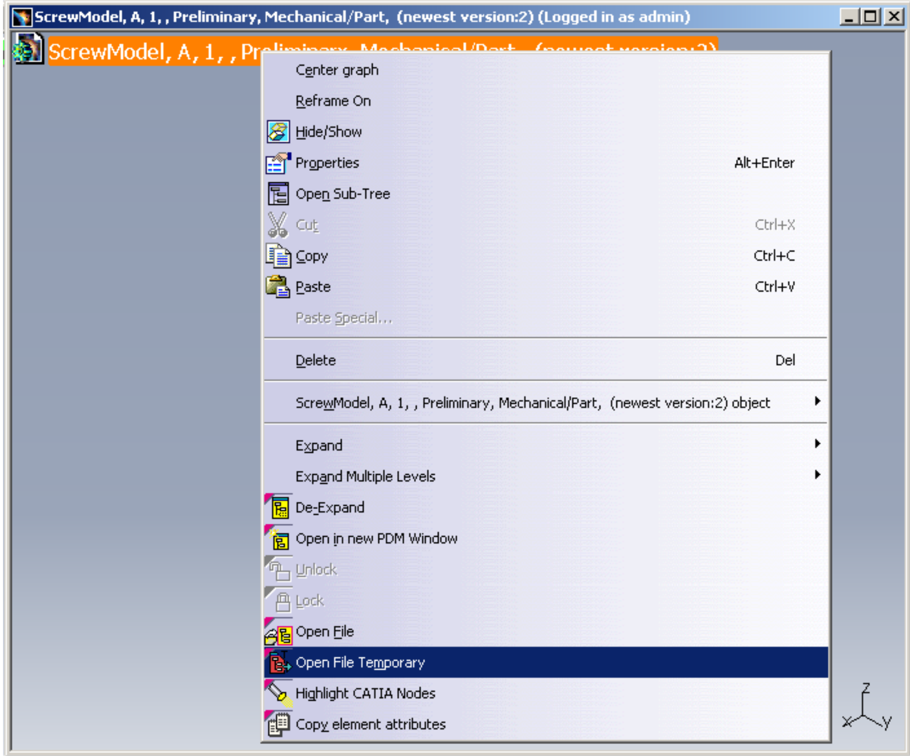
The current version of the file is loaded in CATIA V5, now (see *Picture 59: Current file*).



Picture 59: Current file

Then you query for a different version (in this case version "1") and open the file temporary.

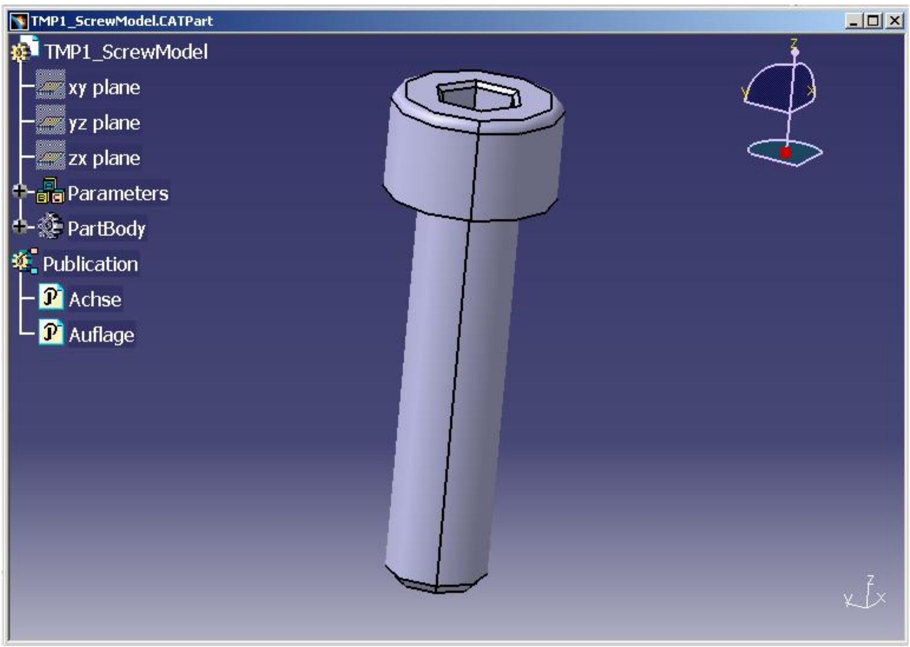
You select the object and click on the right mouse button. In the context menu you select the action "Open File Temporary" (see*Picture 60: Action “Open File Temporary”*).



Picture 60: Action "Open File Temporary"

The version "1" of the file is opened temporary in CATIA V5 (see *Picture 61: Temporary opened file*).

The Part Number and the File Name of the temporarily opened geometry are prefixed with "TMP#_", where "#" is a counter in CATIA V5, beginning with 1. Every action "Open File Temporary" will increase the counter. This prefix is customisable by the customer. For details please refer to the *PDM Workbench Installation & Administration Manual.*



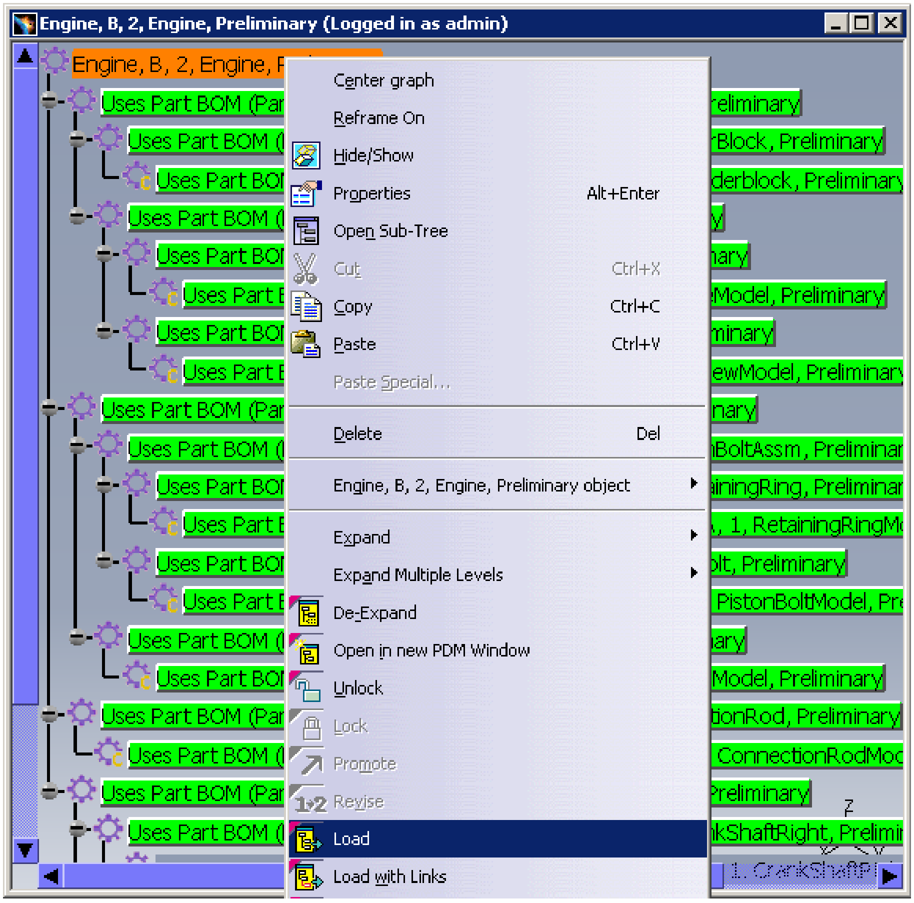
Picture 61: Temporary opened file

Now you can compare the both versions of the file.

Load

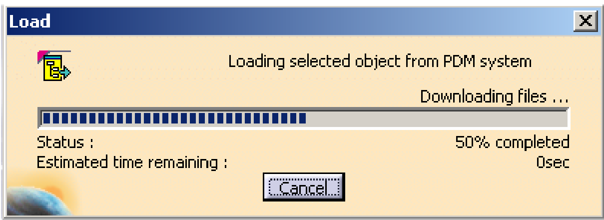
It is possible to load geometry corresponding to an expanded PDM product structure into a native CATIA V5 window in order to work on it, e.g. perform geometric transformations, geometry changes and so on.

To load the geometry in CATIA V5 you select the root PDM object wherefrom downward you want to get the geometry and click the right mouse button to open the context menu and you select the context action “Load” (see *Picture 62: Action “Load”*).



Picture 62: Action “Load”

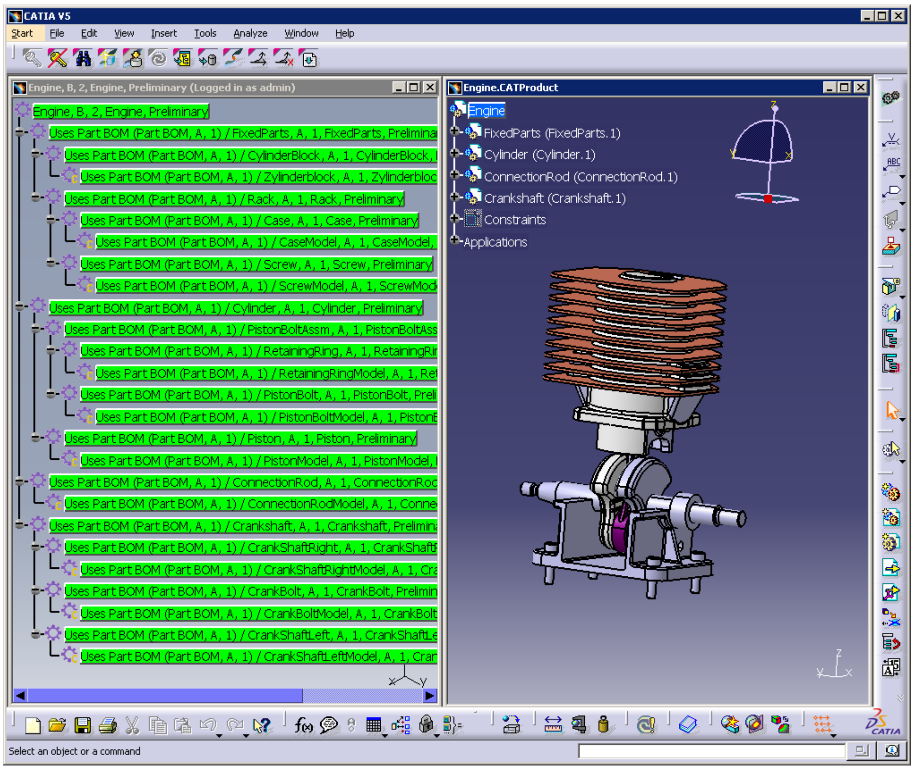
The PDM Workbench downloads the corresponding CAD files (CATParts, CATProducts, and CATDrawings) to the client’s PDM Workbench exchange map. The download progress is indicated by the “Load” progress bar (see *Picture 63: Load - progress bar*).



Picture 63: Load - progress bar

The geometry downloaded opens in a CATIA V5 native window (see *Picture 64: Split window after Load – PDM Workbench and CATIA V5 nodes*).

In the left window (PDM Workbench window) you see the expanded product structure wherefrom you opened the CATIA V5 native window presenting the geometry on the right. In the right window you see the loaded geometry.



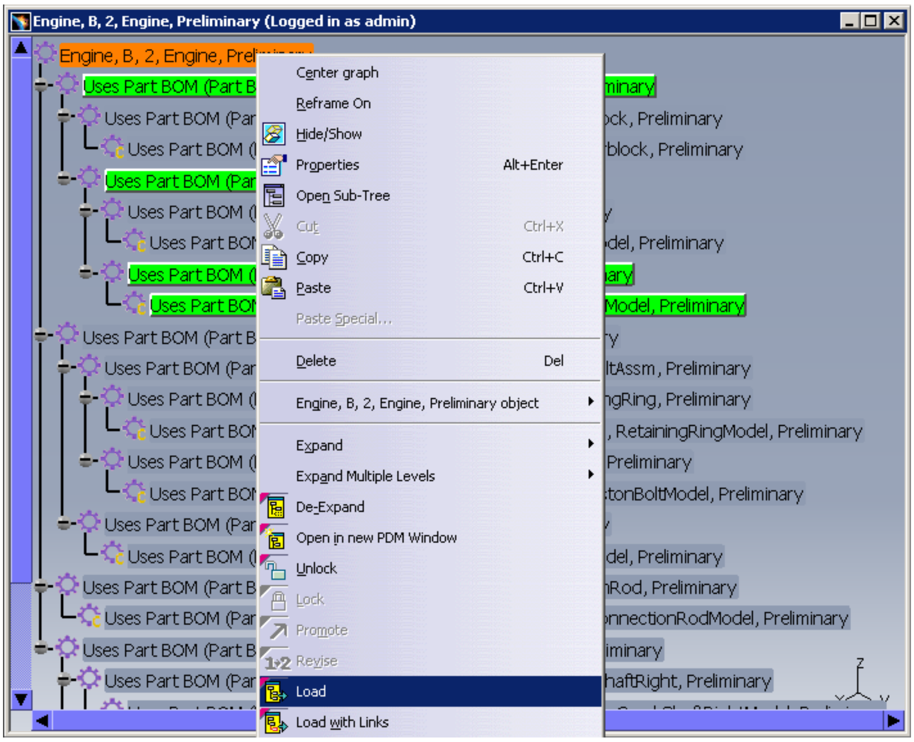
Picture 64: Split window after Load – PDM Workbench and CATIA V5 nodes

Add Temp

The action "Add Temp" allows the user to visualize a temporary structure together with the working one.

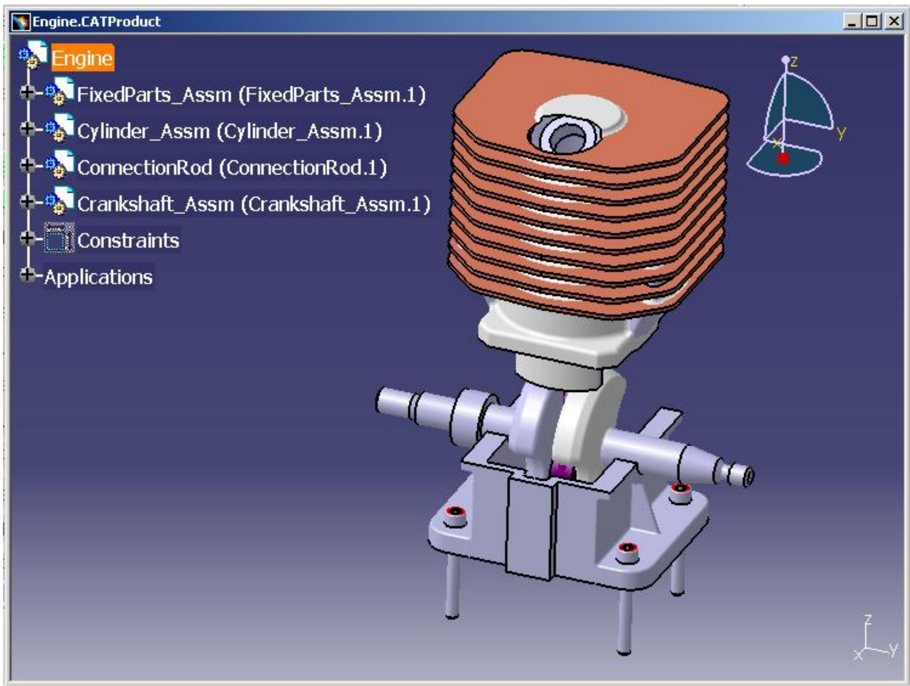
In the first step you load the current version of the structure. In this example you open the revision "B" of the "Engine".

You select the object and click on the right mouse button. In the context menu you select the action "Load" (see *Picture 65: Action "Load"*).



Picture 65: Action “Load”

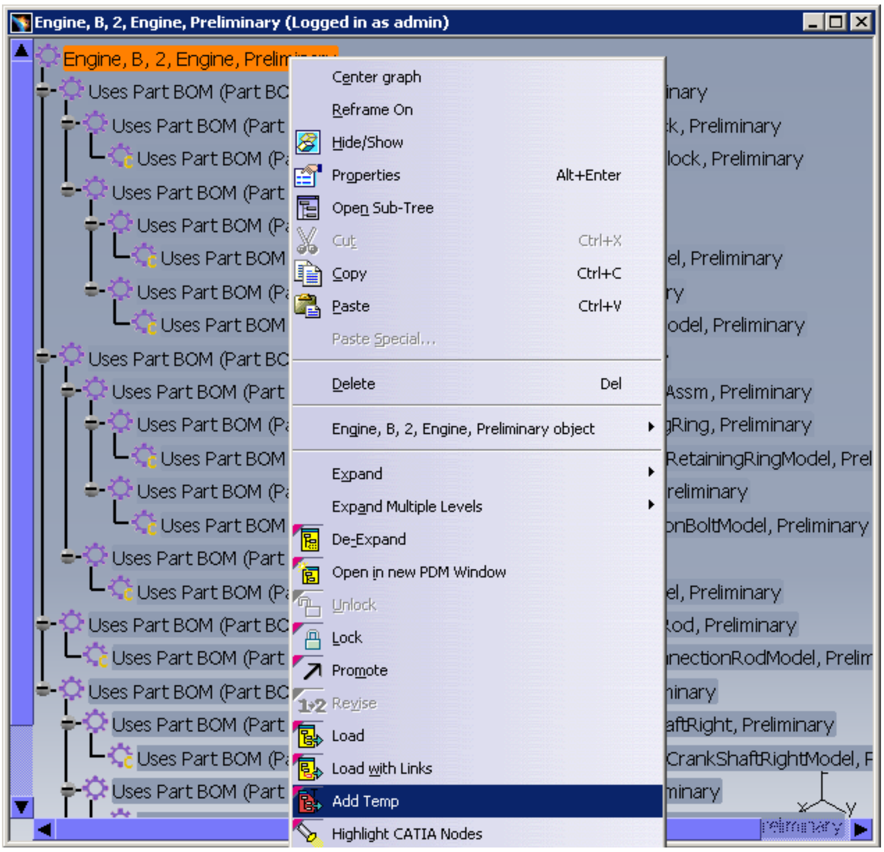
The current version of the structure is loaded in CATIA V5, now (see *Picture 66: Loaded geometry for revision "B"*).



Picture 66: Loaded geometry for revision "B"

Then you query for a different version (in this case revision "A") and load the structure temporary.

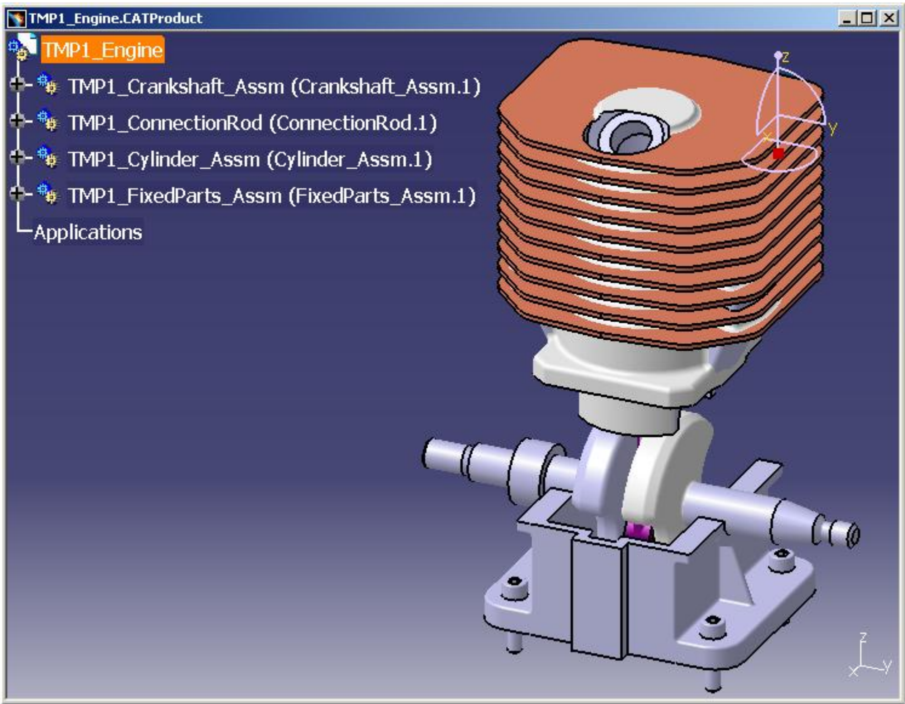
You select the object and click on the right mouse button. In the context menu you select the action "Add Temp" (see *Picture 67: Action “Add Temp”*).



Picture 67: Action “Add Temp”

The CATProducts will not be loaded. Only the CATParts are loaded and positioned correctly (see *Picture 68: Loaded geometry for revision "A"*).

All Part Numbers and File Names in the temporarily added structure are prefixed with “TMP#_”, where “#” is a counter in CATIA V5, beginning with 1. Every action "Add Temp" will increase the counter. This prefix is customisable by the customer. For details please refer to the *PDM Workbench Installation & Administration Manual.*



Picture 68: Loaded geometry for revision "A"

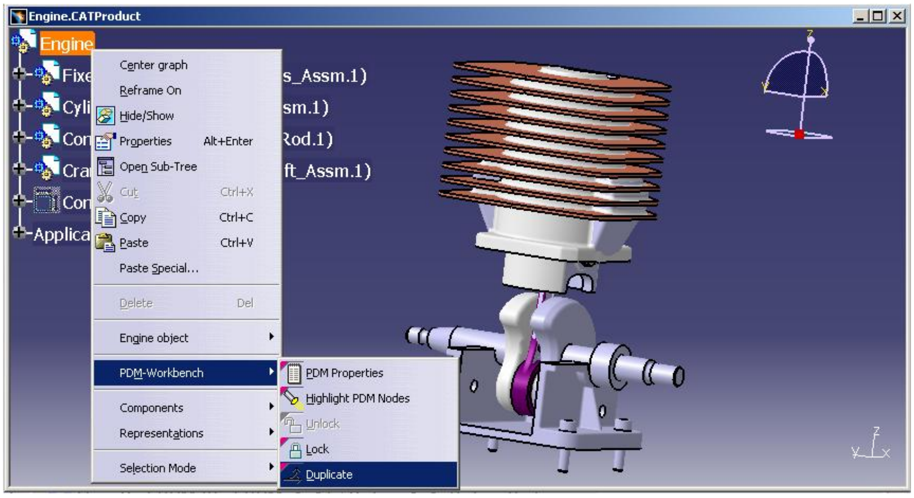
Now you can compare both geometry structures.

Duplicate

It is possible to duplicate existing CATProduct and CATPart objects.

You have to open the objects in CATIA V5 and click on the right mouse button in order to open the context menu. There you have to select the action "Duplicate" (see *Picture 69: Action "Duplicate"*).

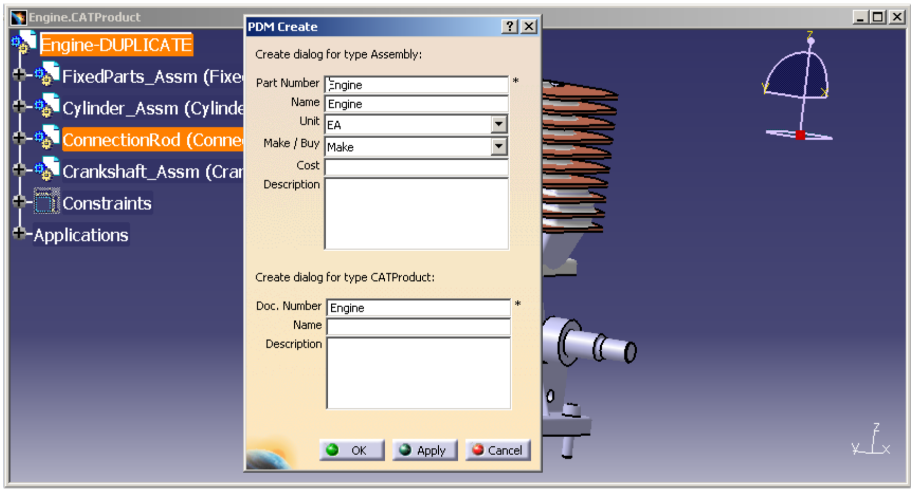
For CATParts you have to use the "Duplicate" action in the "PDM Workbench context commands" toolbar.



Picture 69: Action "Duplicate"

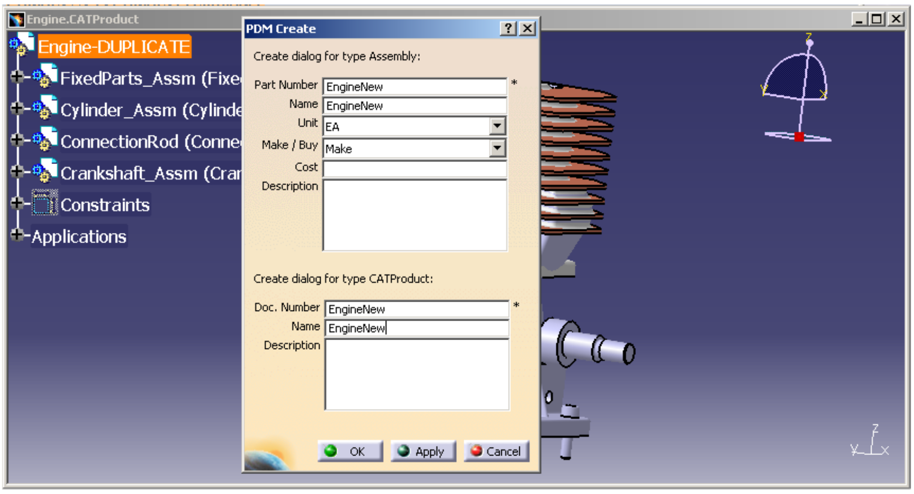
The "PDM Create" dialog will be opened. The correct type is already selected. The identifying name of the document is filled, too.

You have to fill or change the attributes (see *Picture 70: "PDM Create" dialog for duplicate*). The CATIA object has the temporary suffix "-DUPLICATE".

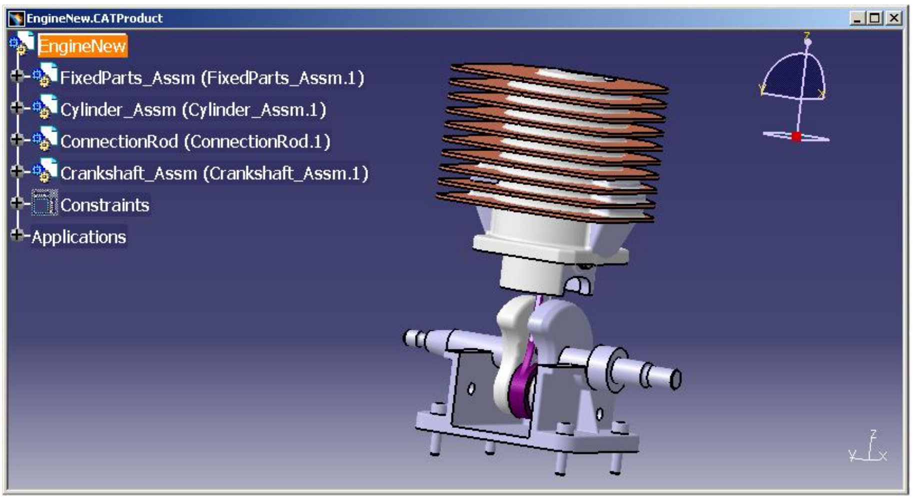


Picture 70: "PDM Create" dialog for duplicate

When you close the dialog with "OK" (see *Picture 71: Filled "PDM Create" dialog for duplicate*) the CATIA document will be renamed to the new part number and created in the PDM system (see *Picture 72: Duplicated CATProduct object*).



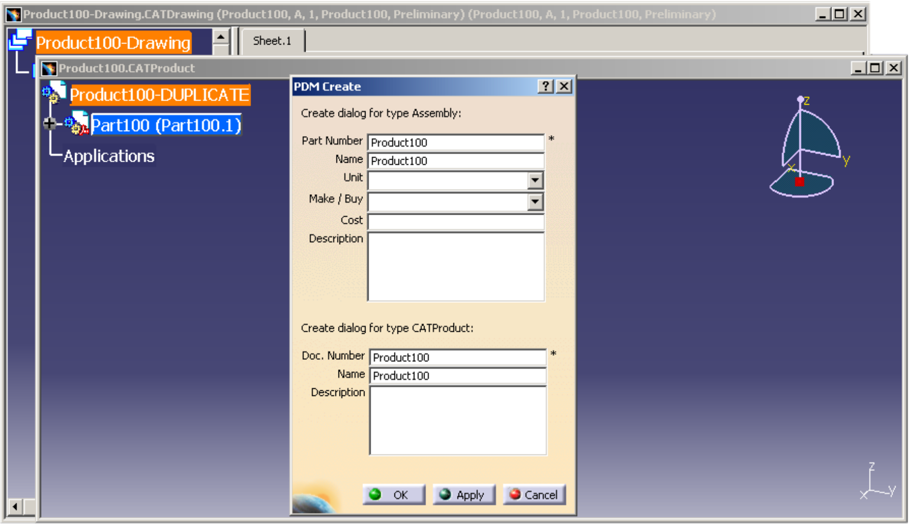
Picture 71: Filled "PDM Create" dialog for duplicate



Picture 72: Duplicated CATProduct object

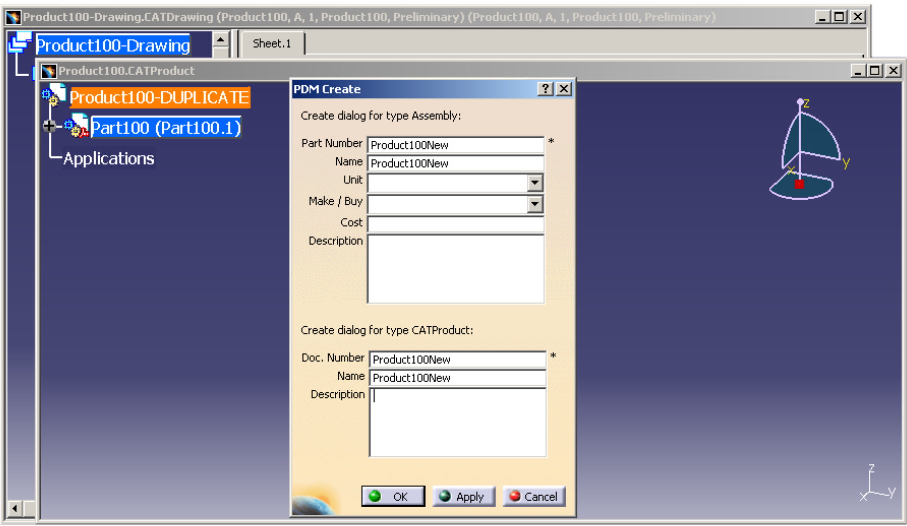
When a CATDrawing object is related to the CATProduct and the file name and the document name of the CATDrawing ("Product-No-100-Drawing.CATDrawing") includes the part number of the CATProduct ("Product-No-100") as prefix then the CATDrawing can be duplicated and related to the CATProduct.

You load the product structure with the drawing. They are opened in two CATIA windows. Then you select the top product and open the context menu and select the action "Duplicate". The product will be renamed with the suffix "-DUPLICATE" and the "PDM Create" dialog will be opened. The document name is filled (see *Picture 73: "PDM Create" dialog for Assembly and CATProduct*).



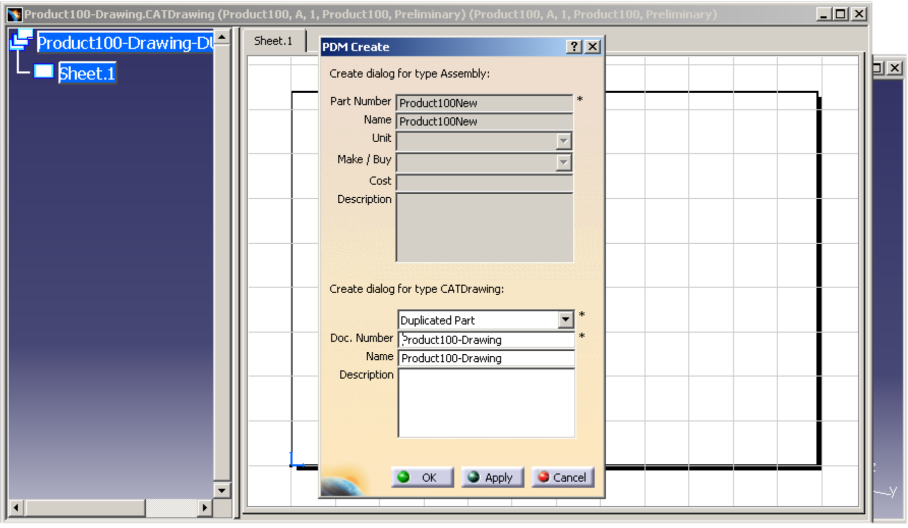
Picture 73: "PDM Create" dialog for Assembly and CATProduct

You change and fill the dialog and select the "OK" button (see *Picture 74: Filled "PDM Create" dialog for Assembly and CATProduct*).



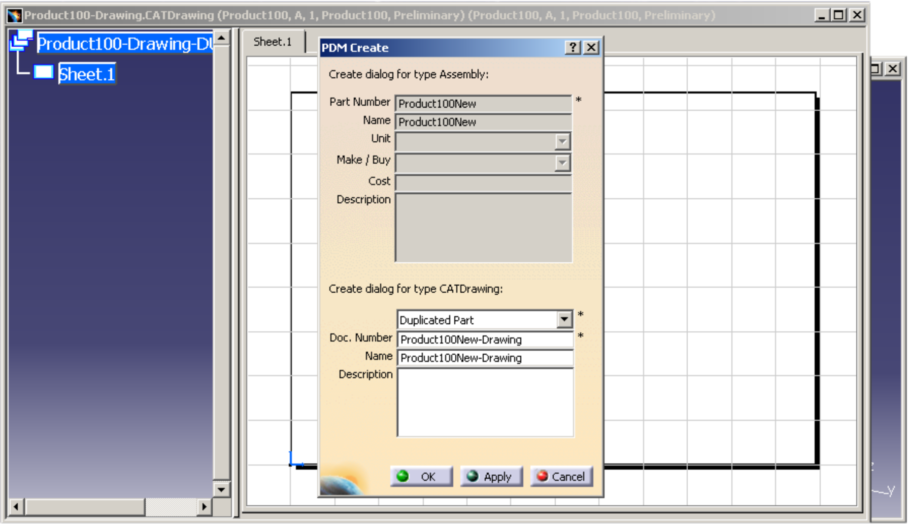
Picture 74: Filled "PDM Create" dialog for Assembly and CATProduct

The CATProduct will be renamed to the desired name and created in the PDM system. The "PDM Create" dialog for the CATDrawing will be opened. The part number and the name of the assembly are filled. The action is set to "Duplicate Part" (see *Picture 75: "PDM Create" dialog for Assembly and CATDrawing*).



Picture 75: "PDM Create" dialog for Assembly and CATDrawing

You change and fill the dialog. The Document Name should have the Part Number of the Assembly as prefix. Then you start the process with the "OK" button (see *Picture 76: Duplicated CATProduct object*).



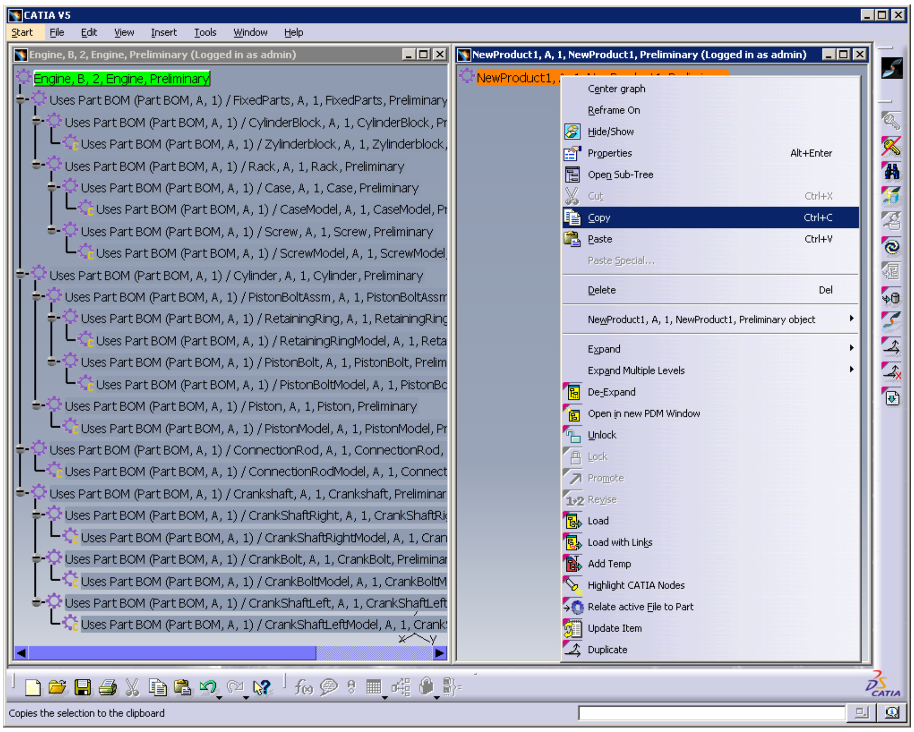
Picture 76: Duplicated CATProduct object

The drawing document will be created in the PDM system and related to the assembly object. Analogous you can duplicate a CATPart with its CATDrawing.

Create Relation between windows

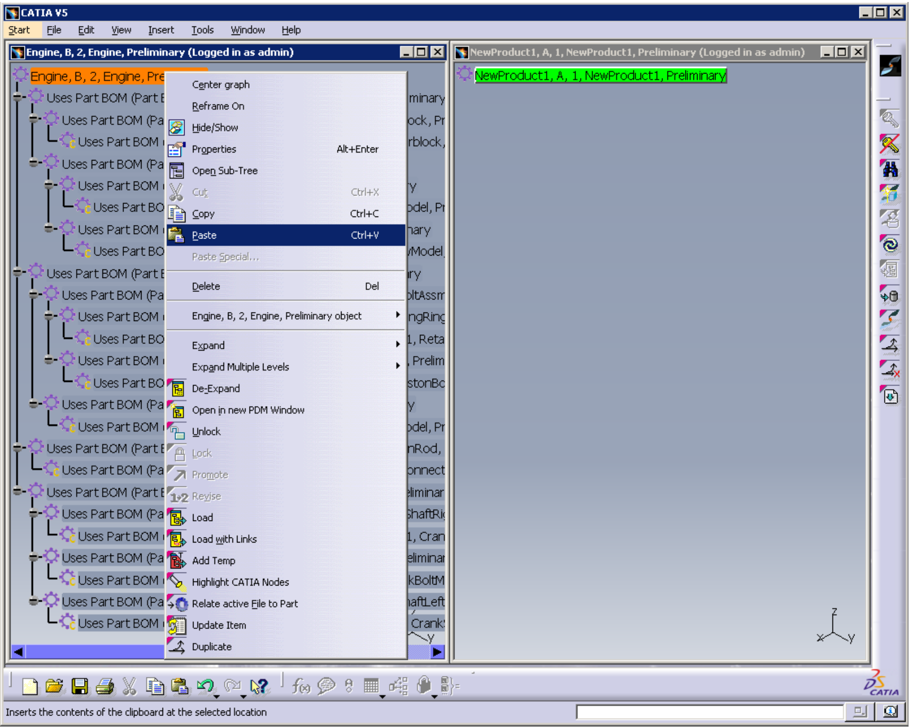
You might modify the product structure by adding existing objects from several PDM Workbench windows to the structure in another PDM Workbench window.

You select the object you want to copy and click the right mouse button to get the context menu. Then you select the context action “Copy” (see *Picture 77: Action “Copy” between windows*). Of course you also can use the short cut “CTRL+C”.



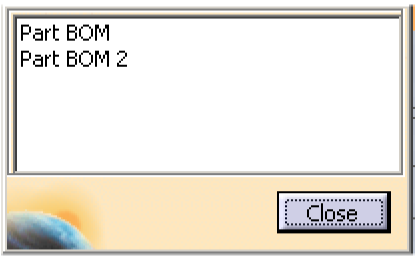
Picture 77: Action “Copy” between windows

Then you select the object where you want to add the copied object to and click the right mouse button to open the context menu. You select the context action “Paste” (see *Picture 78: Action “Paste” between windows*). Of course you also can use the short cut “CTRL+V”.



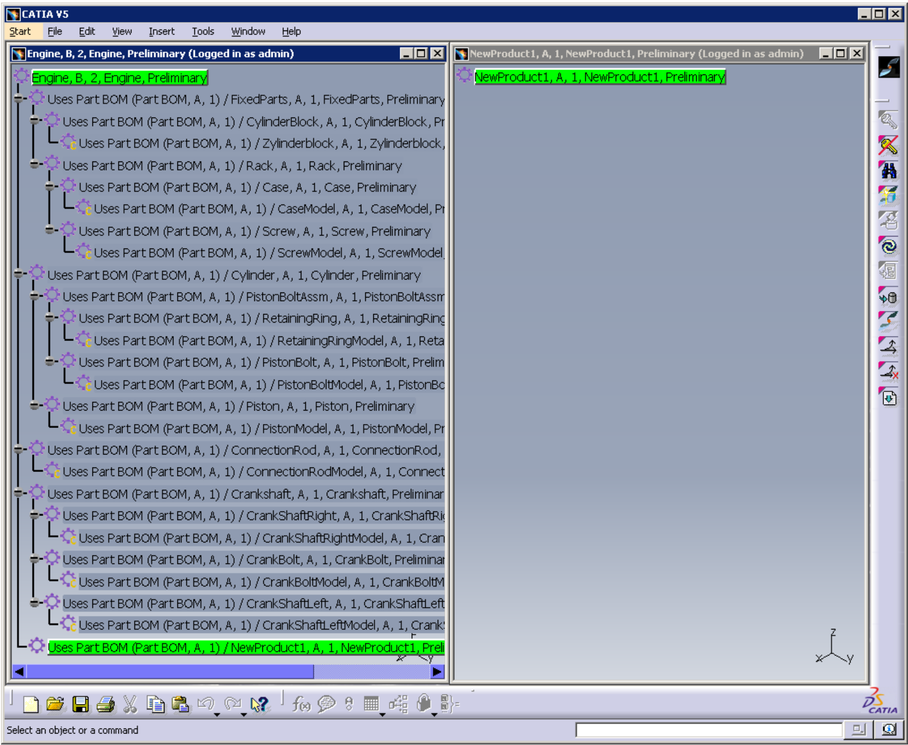
Picture 78: Action “Paste” between windows

You specify the relation you want to create in the structure between the two objects (see *Picture 79: Select the new relation*) once there is more than one relation type available. In the dialog window you see all relations possible between the two objects. In case you want to add the Assembly object in the structure to another Assembly object then you might choose the “Part BOM” relation for example. The dialog will not appear if there is only one relation type available.



Picture 79: Select the new relation

The instance object gets inserted into the existing product structure tree and the new relation gets created in the PDM system (see *Picture 80: Product Structure with inserted object*).



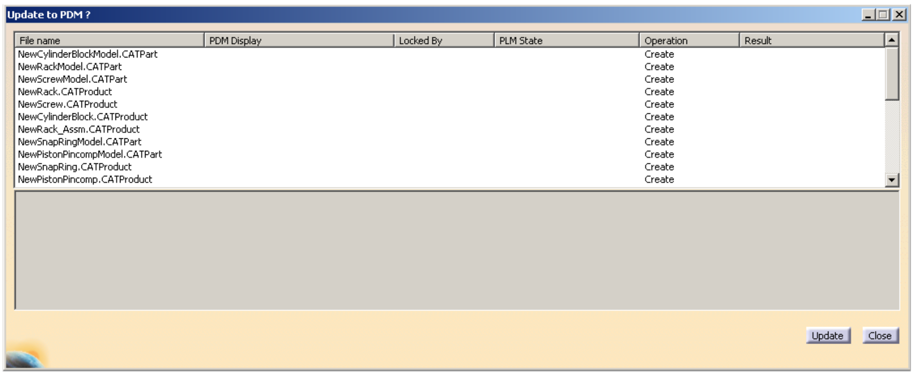
Picture 80: Product Structure with inserted object

Update

The Update functionality can be used to create, to complete, or to update the product structure in the PDM system based on the geometry in the CATIA V5 window.

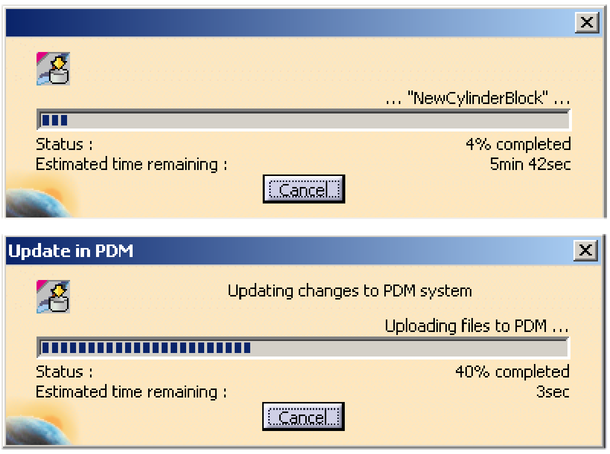
You can start the Update process by clicking on the “Update” icon .

A dialog opens and asks to confirm the described actions. In this example the CATIA documents will be created (see *Picture 81: Confirm the Update (with Create) action*).



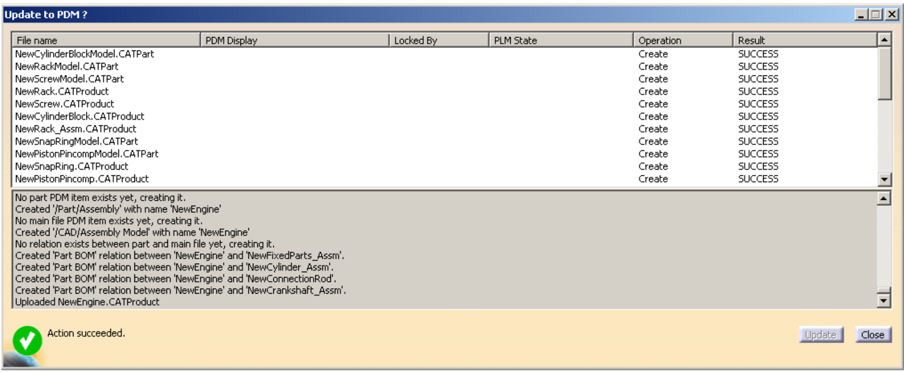
Picture 81: Confirm the Update (with Create) action

The progress of the Update will be shown with the progress bars (see *Picture 82: Progress bars for Update action*).



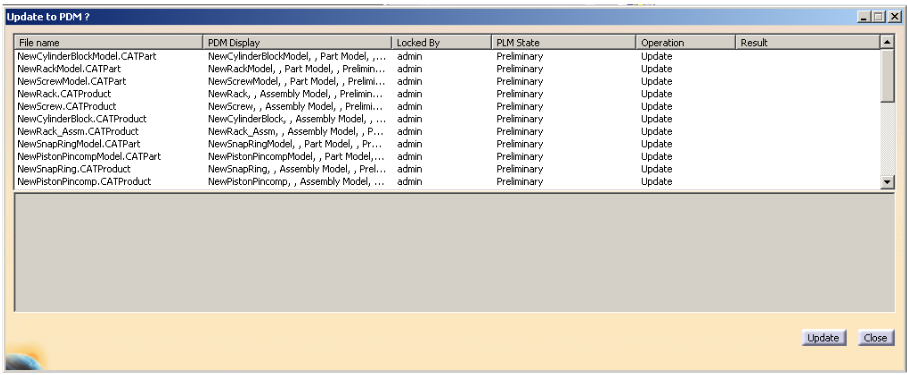
Picture 82: Progress bars for Update action

When the Update process has finished you are informed about the actions that have been performed. The related instances and the created objects are reported in the information window (see *Picture 83: Information window when Update (with Create) is finished*).



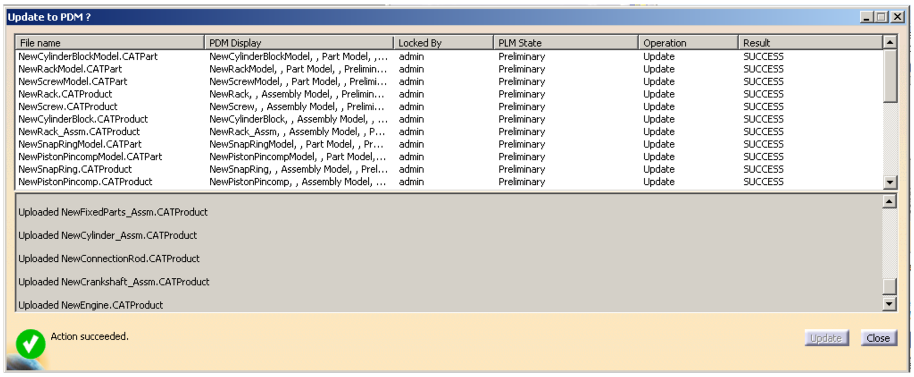
Picture 83: Information window when Update (with Create) is finished

If there are no objects to be created then it will only be asked if you want to update (see *Picture 84: Confirm the Update action*).



Picture 84: Confirm the Update action

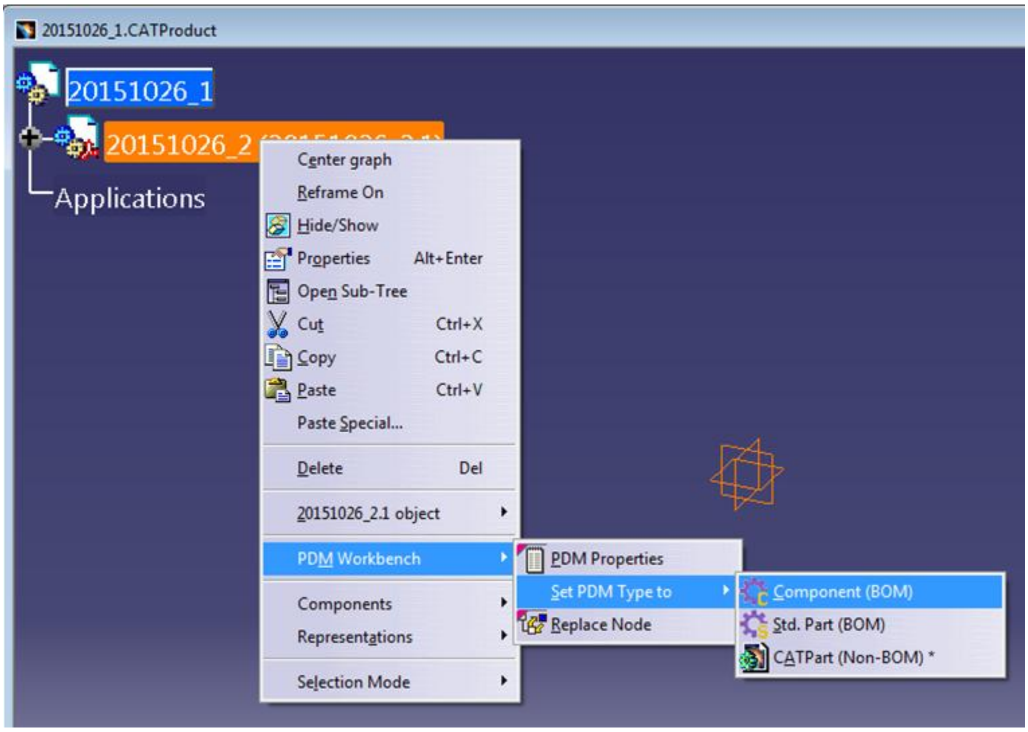
When the Update process has finished you are informed about the actions that have been performed (see *Picture 85: Information window when Update is finished*).



Picture 85: Information window when Update is finished

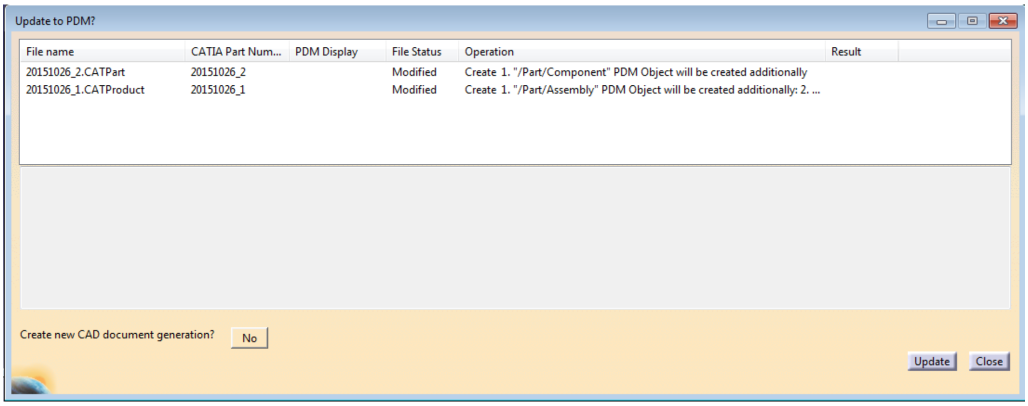
Select Type of additional Parts in Document mode

If PDM Workbench is configured to work in the CAD document structure mode it is possible to create an additional part item in Aras Innovator while creating a CAD document during the PDM update. By default there is a configured part type that will be created in this case. The new functionality allows you to select a specific part type to be created:



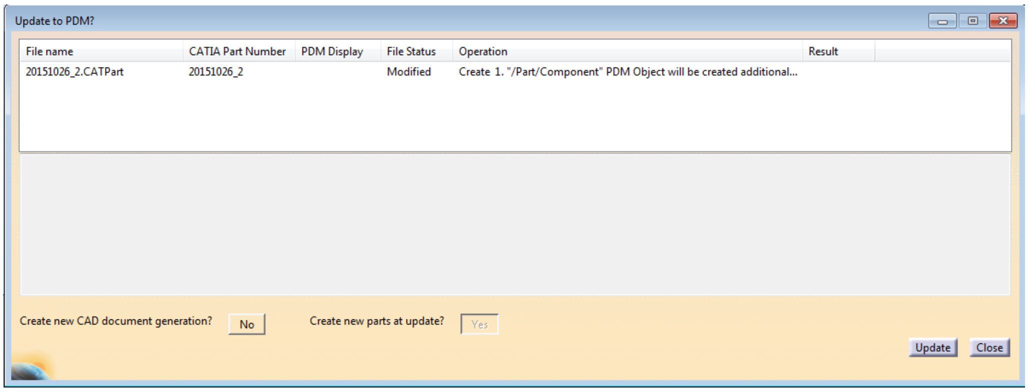
Picture 86: "Set PDM Type" context menu

When updating a product structure the type of the additional part will be shown in the "Operation" column of the update dialog. The "Create Parts at Update" button will be hidden.



Picture 87: Update dialog for CATProduct structure

When updating a single CATPart the "Create Parts at Update" button will be shown. If a BOM type was selected for the CATPart, the button will be deactivated, otherwise the "Create new Parts at Update" button will be active. In this case the default part type will be created for the CATPart when setting the button to "YES"



Picture 88: Update dialog for CATPart document

Constraints:

- It is only possible to select a BOM type if the parent Product is marked to have an additional BOM Part or if the parent CATProduct was loaded from PDM.
- If you switch the type of a CATProduct from BOM to NON-BOM all children will be switched to NON-BOM.

As there is no context menu for a single CATPart you have either to use the default part type for single loaded CATParts, or you have to put the CATPart in a temporary CATProduct and set the type of the additional PDM part item before updating the single CATPart.

Reconnect at Update

This functionality can be used for an initial import of existing CAD data into Aras.

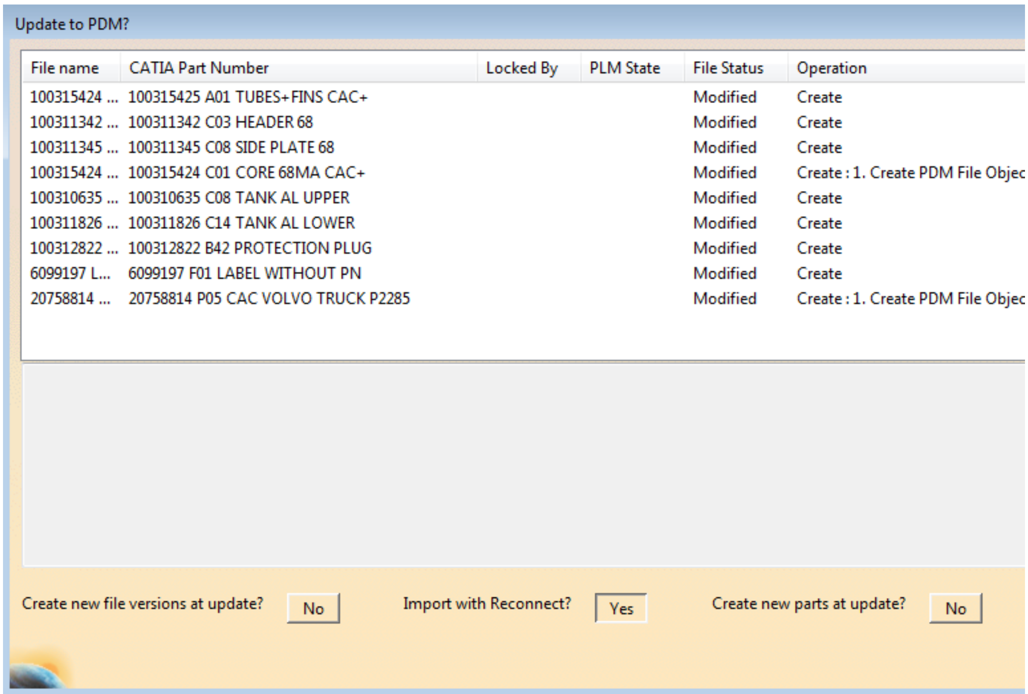
It is possible to reconnect CATParts and CATProducts inside a structure loaded from disc to already existing CADDocuments in Aras even if the CATIA files in Aras were renamed (rule based) during the first import. When a CATPart / CATProduct is reconnected, the external file is not saved to Aras.

Before using this functionality it may be necessary to clean up the existing CATIA data in that way, that the files can be renamed during import using a rule. T-Systems can provide a tool that checks if all file names, part numbers, … fit to the naming convention of your company.

Usage

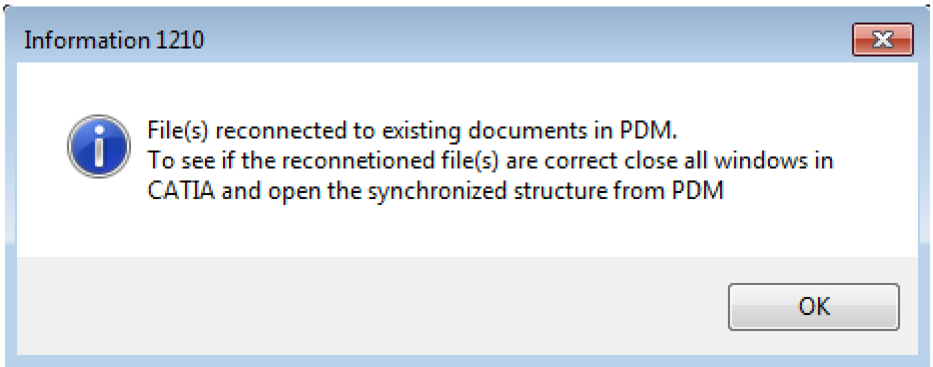
1. **Import Product Structure**

Open a cleaned structure from disc and use PWB Update to save the structure into Aras. If some of the files may be already stored in Aras, select “Import with Reconnect” before performing the update:

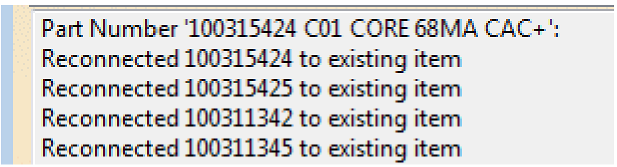


Picture 89: Update dialog with “Import with Reconnect” button

During update the files are renamed according to the installed rule and reconnected if already found in Aras:



Picture 90: “Reconnect” prompt



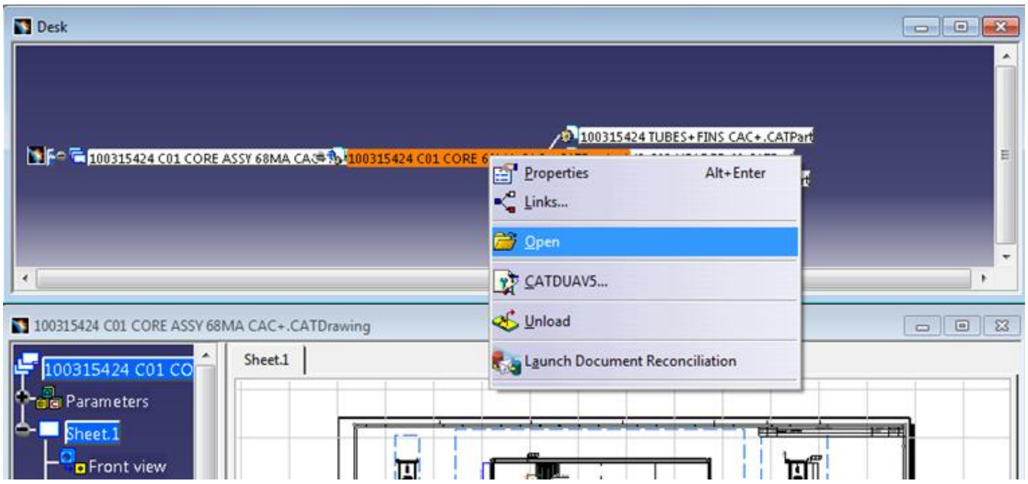
Picture 91: Messages about reconnected items

To check if the reconnected Documents in Aras use the same content like the imported files on disk, the user has to close all windows in CATIA and open the structure from Aras.

2. Import CATDrawing

CATDrawings itself are not reconnected in Aras, but the referenced CATProducts / CATParts may be renamed during import. Therfore the following procedure can be used to import a CATDrawing without braking the links.

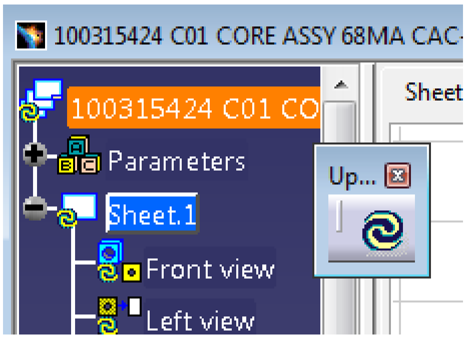
Open the CATDrawing and use File → Desk to open the directly referenced CATPart(s) / CATProduct(s)



Picture 92: Opening referenced 3D geometry files

Update the opened structure like described in “Import Product Structure”. Do not close the renamed Product structure after update.

After save / reconnect the structure in Aras the CATDrawing activates the “Update current Sheet” button that indicats that the CATDrawing need an update:



Picture 93: Updating the current sheet

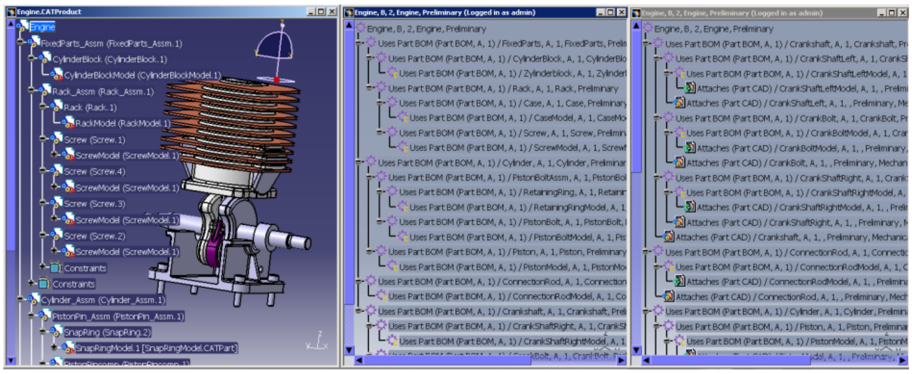
Use the “Update current Sheet” command to make sure the drawing is clean.

Use the PWB Update function to save the Drawing in Aras. To save a CATDrawing in PDM do not select “Import with Reconnect”.

Show PDM Structure

When you have opened the geometry in CATIA V5 you have the possibility to show the corresponding PDM product structure.

For this you have to click on the “PDM Structure” icon . Then a PDM Workbench window with the PDM product structure will be opened (see *Picture 94: PDM Structure for geometry*).



Picture 94: PDM Structure for geometry

Please note that with BOM-part structures, the expanded part structure (window in the middle) usually does not contain the related CATIA files, but the structure displayed by "Show PDM Structure" does (right window).

Refresh PDM Structure

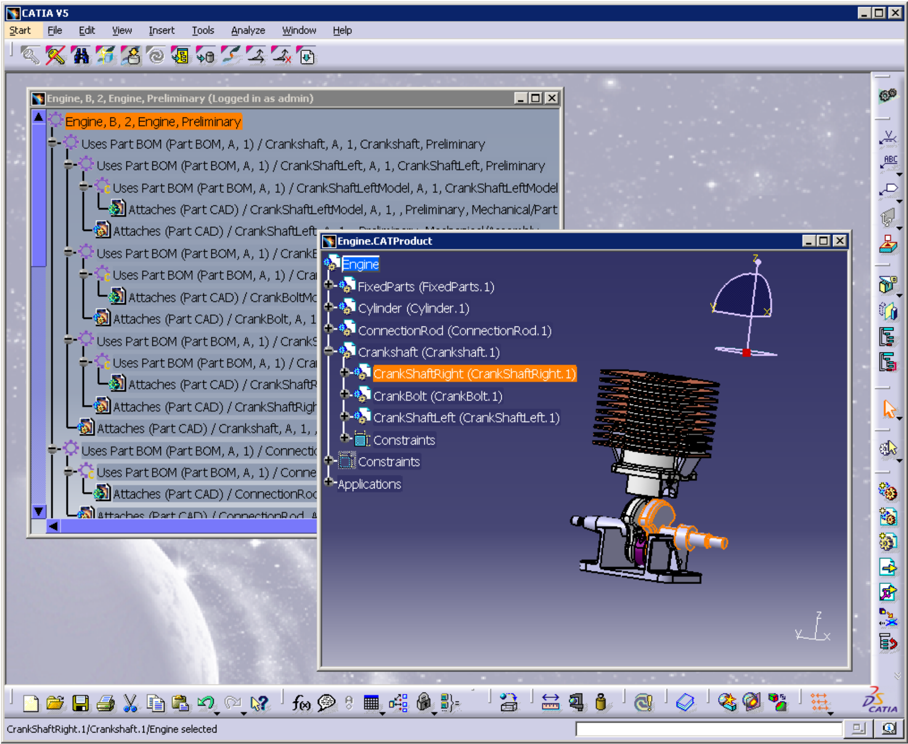
When you have made some actions on the geometry that have an impact on the status of the objects in the PDM product structure, that are not displayed automatically, then you have to update the status display manually.

After the load of the geometry the status of the objects of the PDM product structure is in the default state.

There are the following possible states:

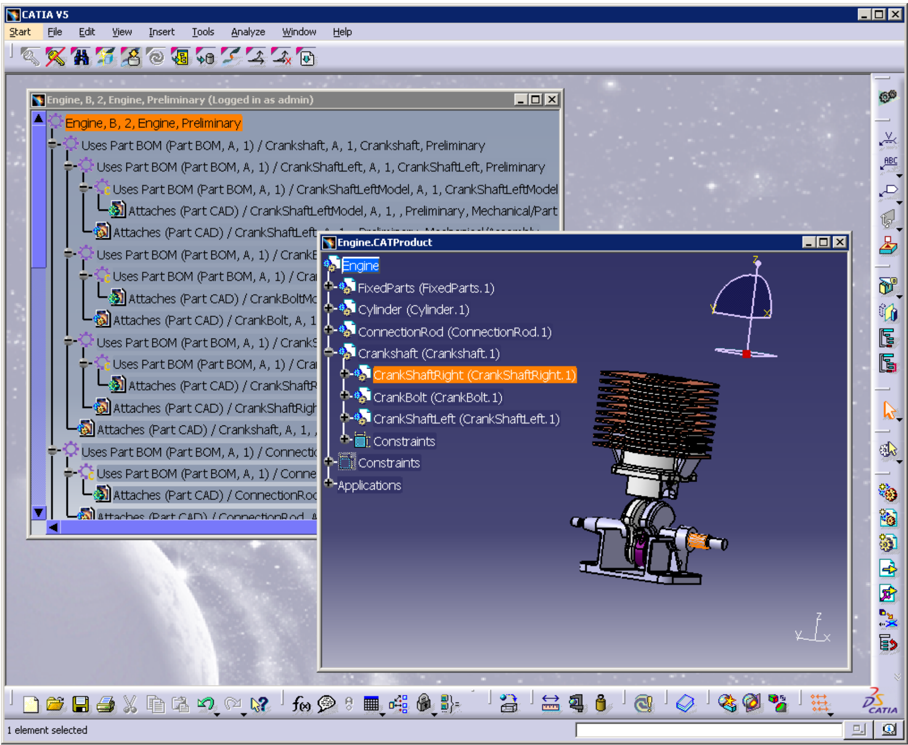
- No background color: not owned by session user, no changes
- Green: owned by session user, no changes
- Yellow: owned by session user, dirty because of changes
- Red: not owned by session user, dirty because of changes
- Black: the object is finalized in the PDM (for example: released state)
- White: the object does not have the file in the PDM

The CATIA Part is owned by the session user and no changes. It is marked in green (see *Picture 95: PDM Structure and geometry in CATIA V5*).

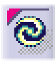


Picture 95: PDM Structure and geometry in CATIA V5

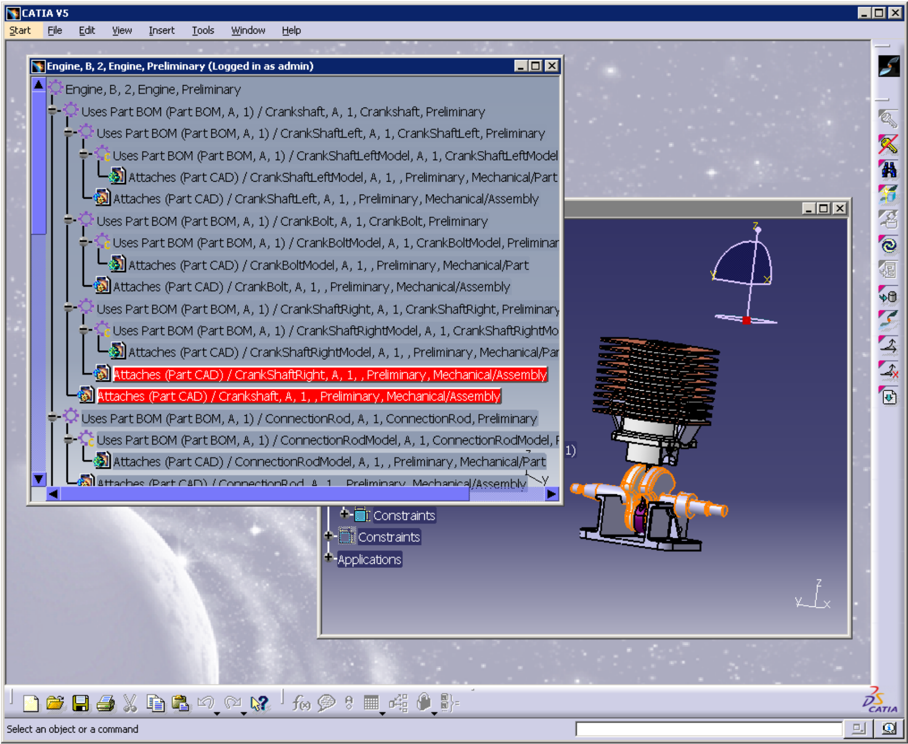
In the CATIA V5 you can make some changes in the geometry (see *Picture 96: Making changes in the geometry*) that make the objects in the PDM product structure dirty. This state change will not be displayed automatically.



Picture 96: Making changes in the geometry

In order to display the changes you have to click on the “Refresh PDM Structure” icon . Then the status of the changed objects in the PDM product structure will be updated.

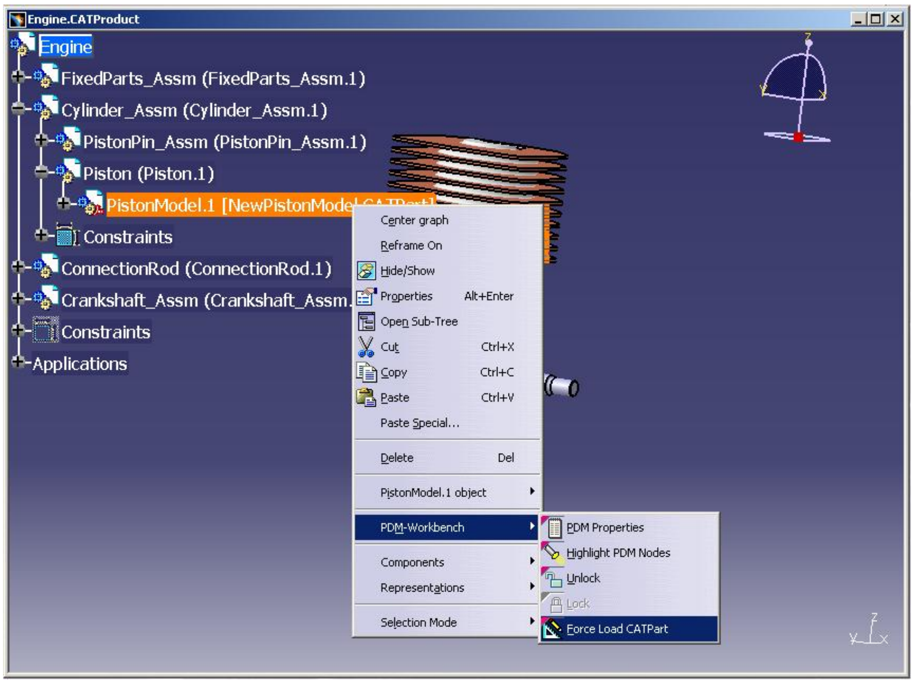
Now e.g. the dirty object owned by the session user will be displayed in yellow in the PDM product structure. The dirty objects on the way to the root are displayed in red because they are not owned by the session user (see *Picture 97: Refreshed PDM Structure*).



Picture 97: Refreshed PDM Structure

Force Load CATPart

When you have loaded the CATIA node into the structure via the Desk command because the file could not be loaded then you can load the CATIA data with the required PDM Workbench information. For this you have to select the CATIA node and open the context menu. Then choose “PDM Workbench→Force Load CATPart” (see *Picture 98: Action "Force Load CATPart"*).

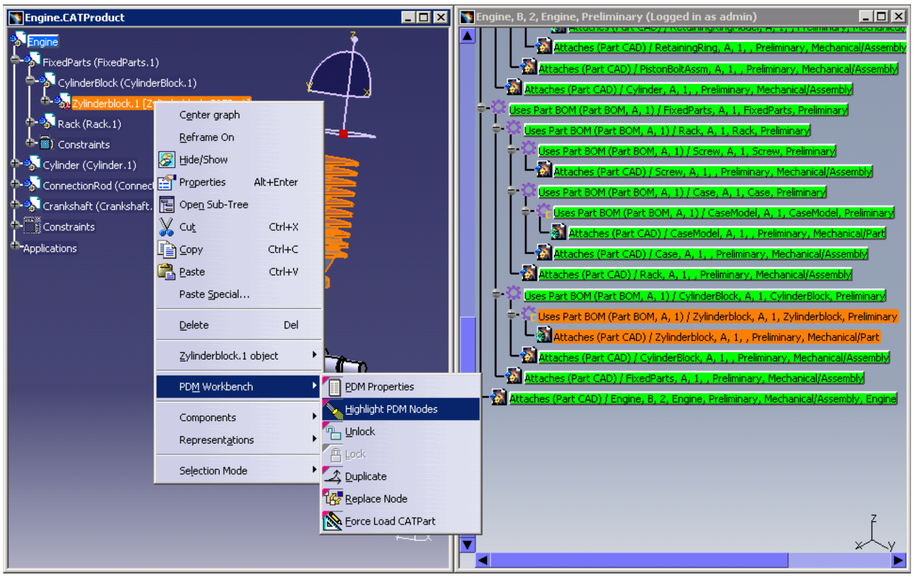


Picture 98: Action "Force Load CATPart"

Highlight PDM Nodes

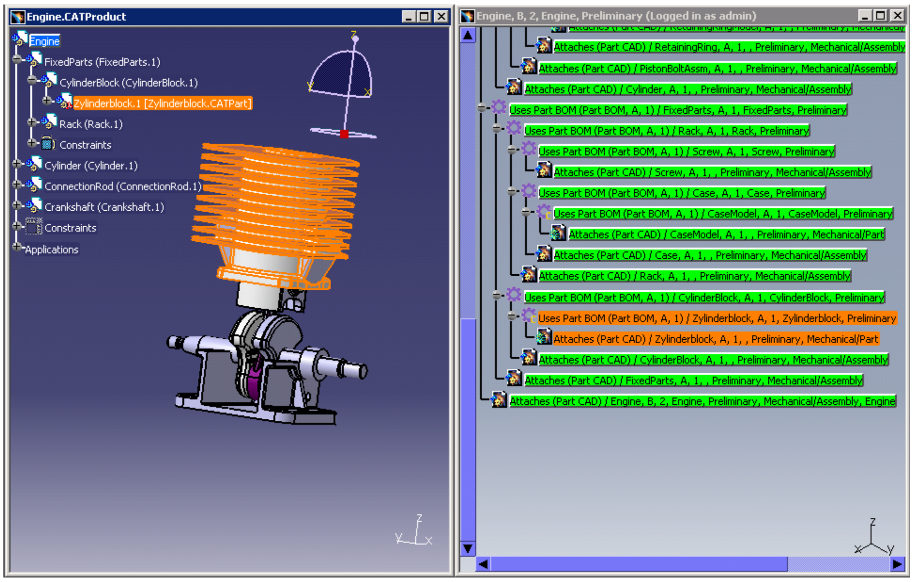
Sometimes it is important to find a PDM node when you are working on the corresponding object in the geometry, e.g. in order to lock the PDM object. Or you have selected an object in the PDM product structure and want to see the corresponding object in the geometry.

For this you can select an object in the geometry and click on the right mouse button. The context menu opens and you have to choose “PDM Workbench→Highlight PDM Nodes” (see *Picture 99: Action “Highlight PDM Nodes”*).



Picture 99: Action “Highlight PDM Nodes”

The PDM Workbench window will be displayed in the foreground and the objects that correspond to the selection in the geometry will be highlighted (marked) (see *Picture 100: Highlighted nodes in PDM Structure*). If there is no PDM Workbench window opened you will get a warning that you have to open the PDM structure first. It is important that you have only one PDM Structure window for this Part Number.



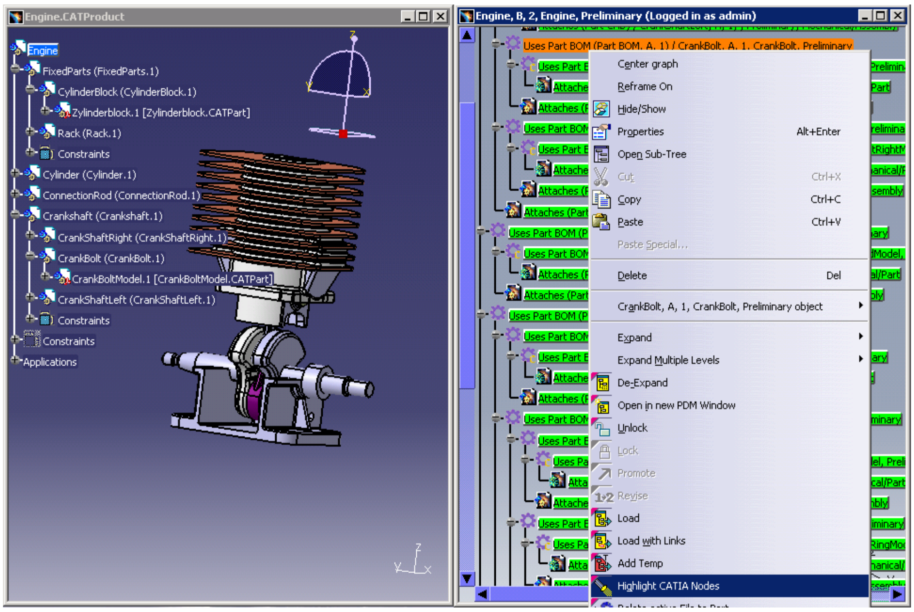
Picture 100: Highlighted nodes in PDM Structure

This works in the opposite direction, too (see *Highlight CATIA Nodes*).

Highlight CATIA Nodes

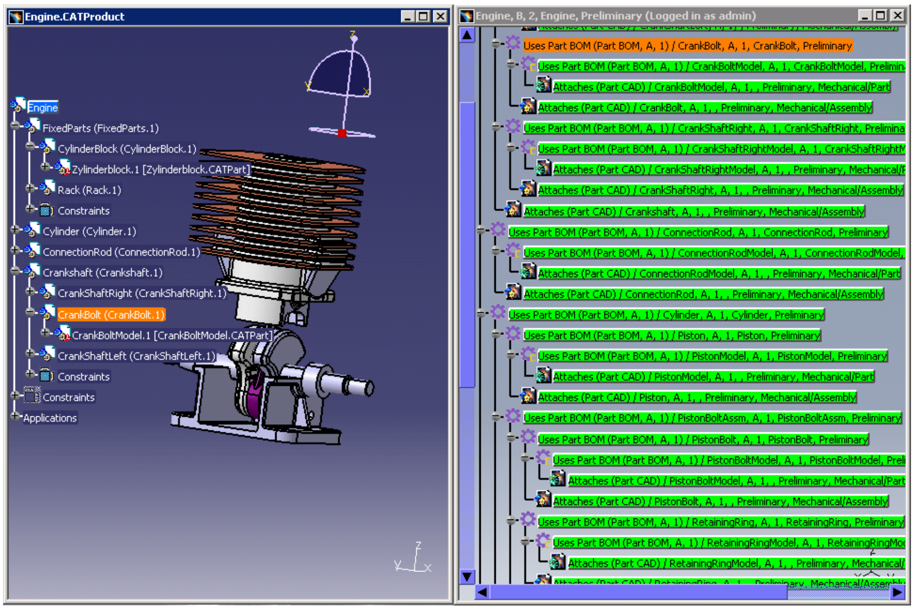
Sometimes it is important to find a CATIA node when you are working on the corresponding object in the product structure, e.g. in order to lock the PDM object. Or you have selected an object in the geometry and want to see the corresponding object in the PDM product structure.

For this you can select an object in the PDM product structure and click on the right mouse button. The context menu opens and you have to choose “PDM Workbench→Highlight CATIA Nodes” (see *Picture 101: Action “Highlight CATIA Nodes”*).



Picture 101: Action “Highlight CATIA Nodes”

The CATIA V5 geometry window will be displayed in the foreground and the objects that correspond to the selection in the PDM product structure will be highlighted (marked) (see *Picture 102: Highlighted nodes in CATIA geometry*). If there is no CATIA V5 geometry window opened you will get a warning that you have to open the CATIA V5 geometry window.



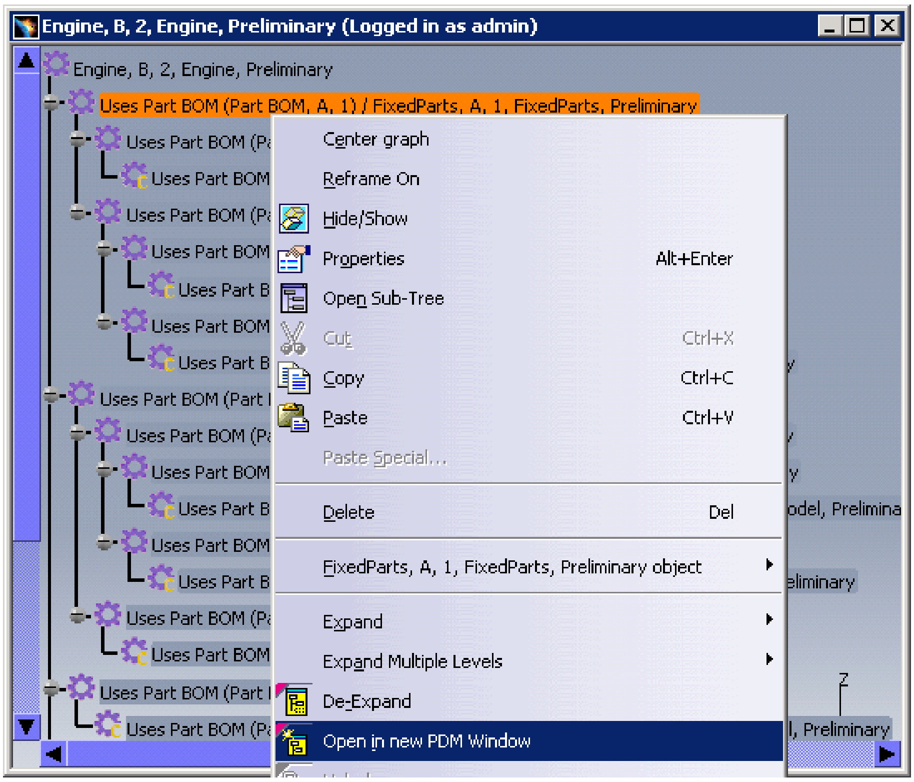
Picture 102: Highlighted nodes in CATIA geometry

This works in the opposite direction, too (see *Highlight PDM Nodes*).

Open in New Window

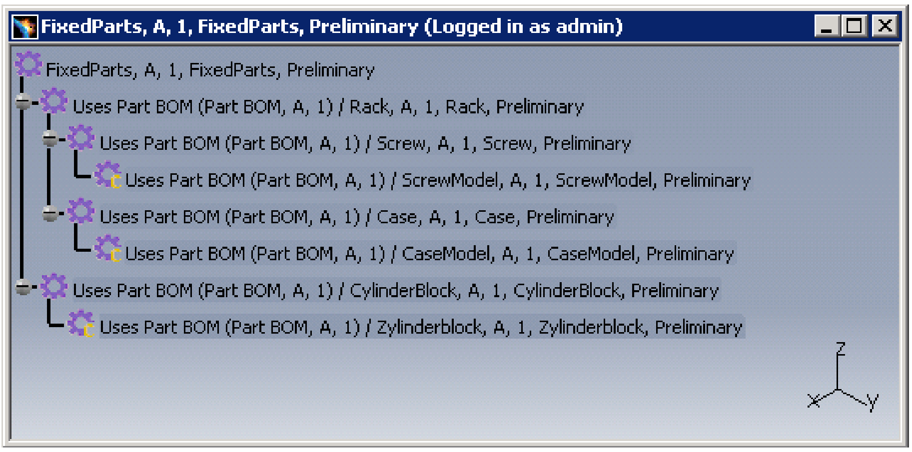
When you have a lot of objects in one window, e.g. received by query or expand then it can be necessary to open a subset of them in a new window.

You select the objects you want to open in a new window. In the context menu you select the action “Open in New Window” (see *Picture 103: Action “Open in New Window”*).



Picture 103: Action "Open in New Window"

The selected objects will be opened in a new window (see *Picture 104: The selected objects in the new window*).

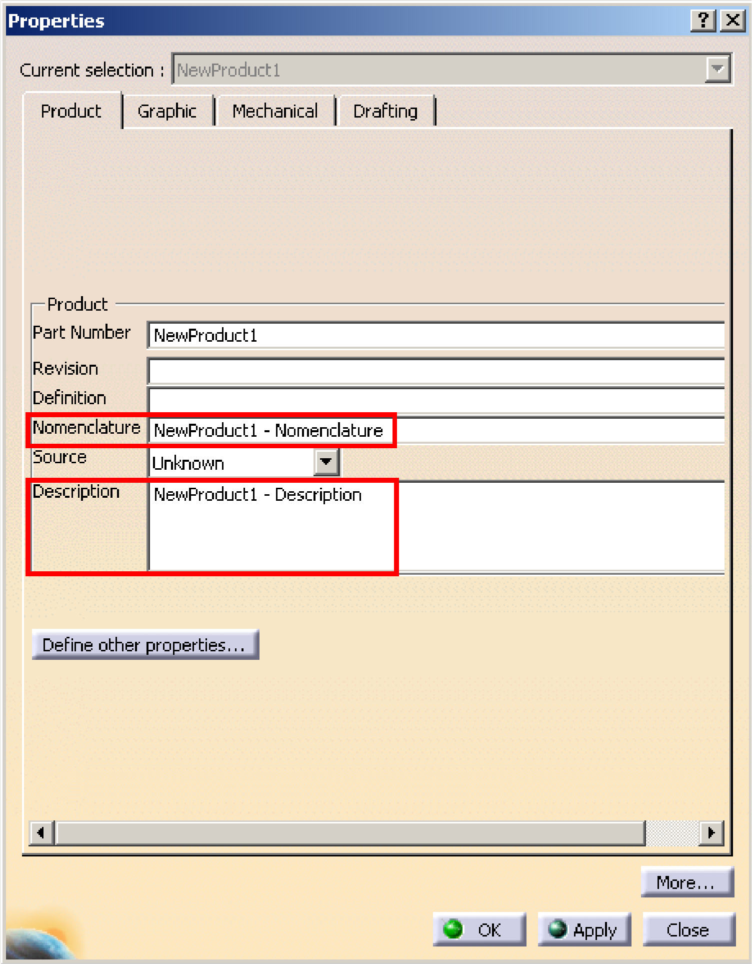


Picture 104: The selected objects in the new window

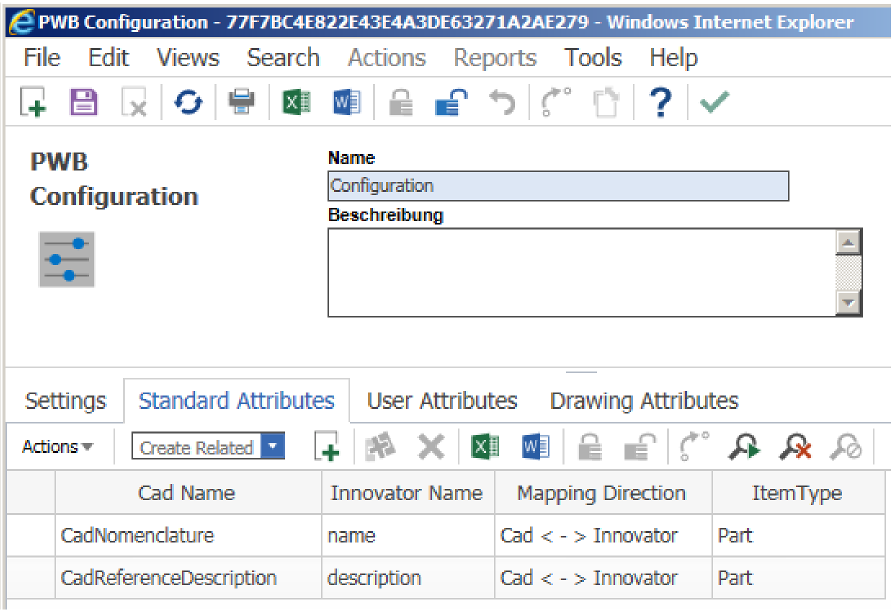
Attribute mapping functionality

CATIA standard and user-defined properties can be mapped to PDM attributes.

In the following example the standard CATIA attributes "Nomenclature" and "Description" are mapped to the attributes "name" and "description" of the Aras Innovator part object (see *Picture 105: Standard attributes in the "Properties" dialog* and *Picture 106: Configuration of standard attributes in Aras Innovator*).

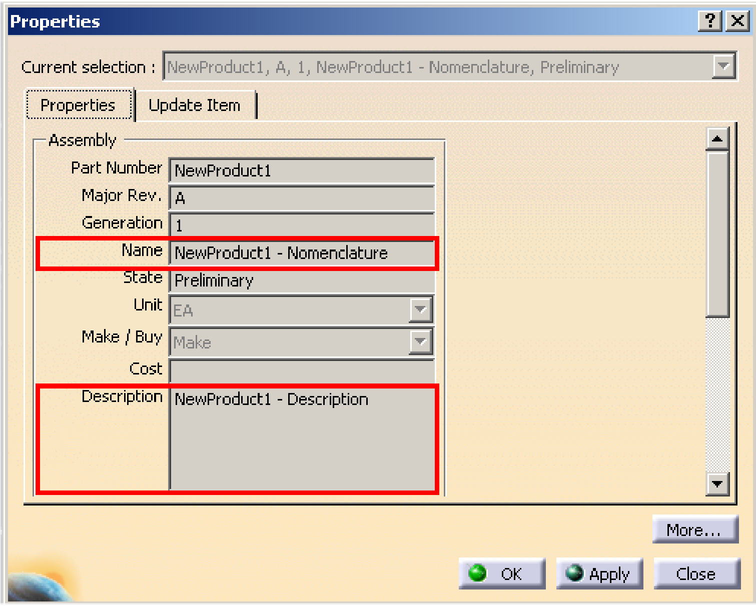


Picture 105: Standard attributes in the "Properties" dialog

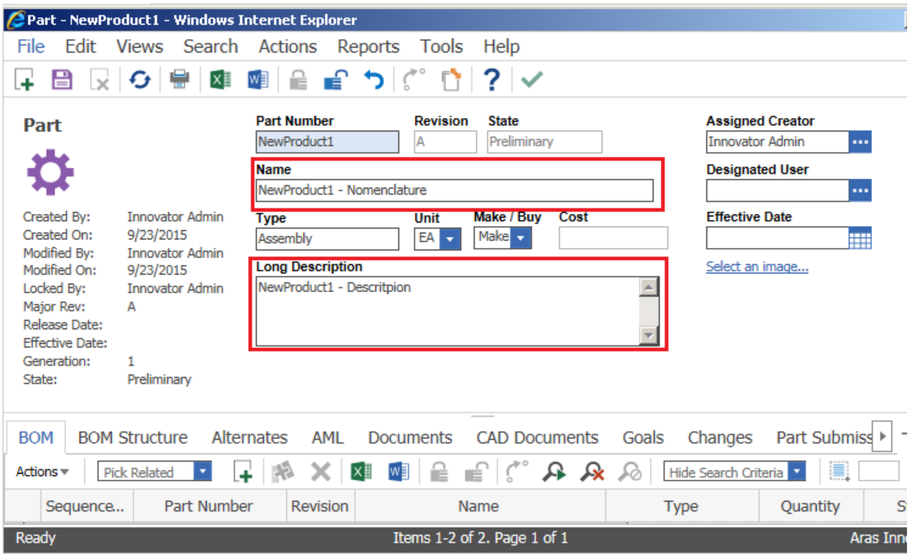


Picture 106: Configuration of standard attributes in Aras Innovator

After creating the part with Update the defined CATIA attribute values have been written to the PDM part object (see *Picture 107: Standard attributes in the "Properties" dialog of the PDM node* and *Picture 108: Standard attributes in Aras Innovator window*).

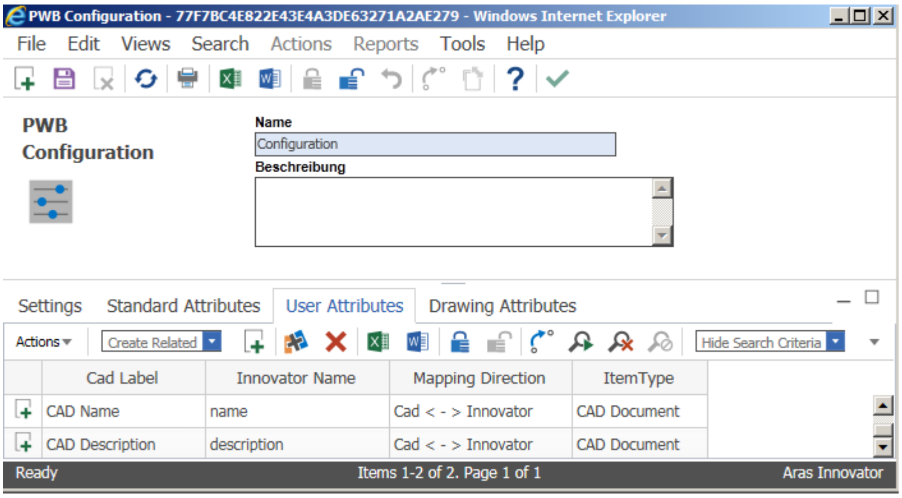


Picture 107: Standard attributes in the "Properties" dialog of the PDM node



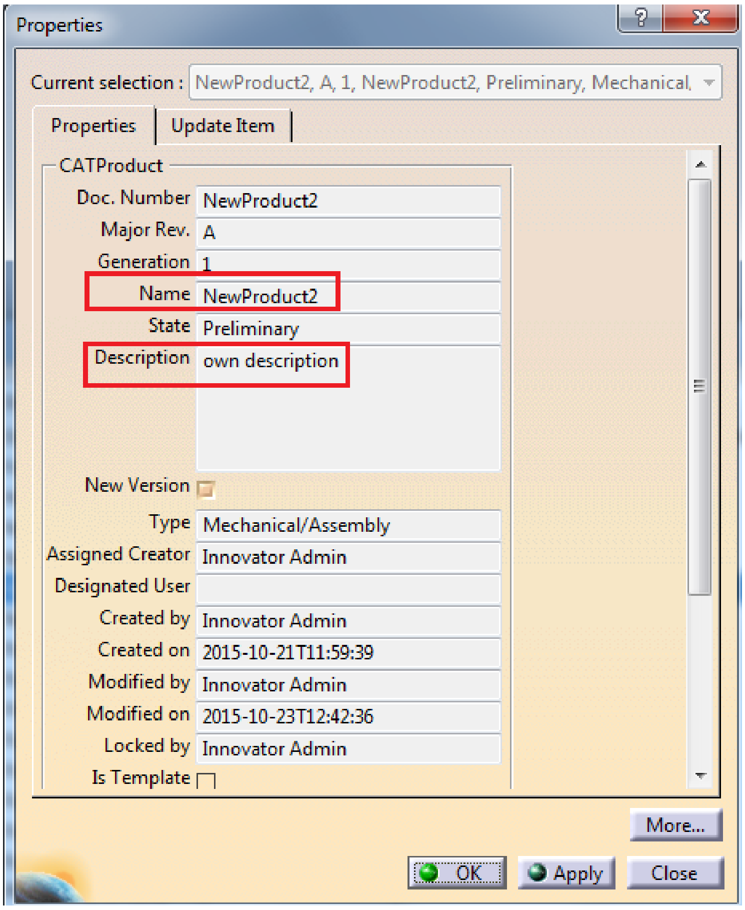
Picture 108: Standard attributes in Aras Innovator window

User-defined CATIA properties can also be mapped (see *Picture 109: Configuration of user-defined attributes in Aras Innovator*).

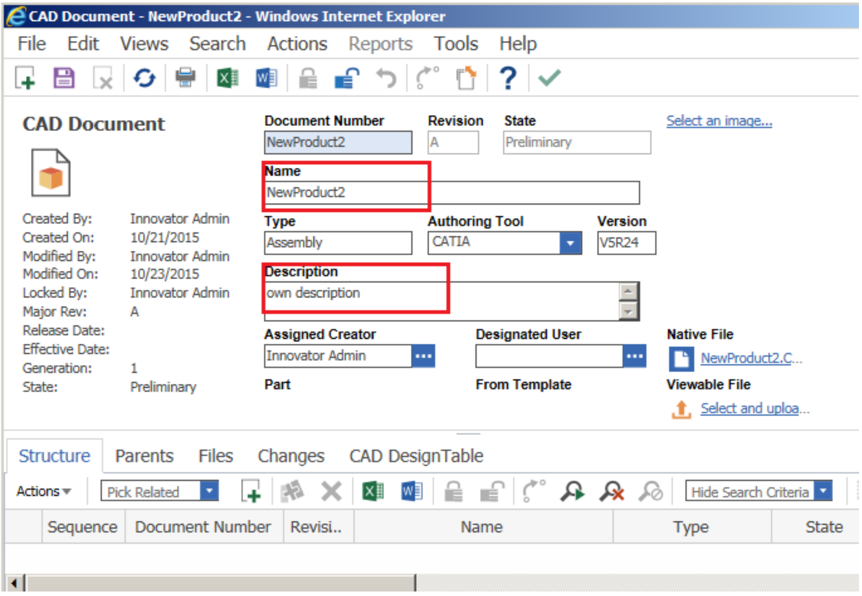


Picture 109: Configuration of user-defined attributes in Aras Innovator

While the structure is imported the values are written to the defined attributes of the Aras Innovator CAD document object (see *Picture 110: User-defined attributes in the "Properties" dialog of the PDM node* and *Picture 111: User-defined attributes in Aras Innovator window*).

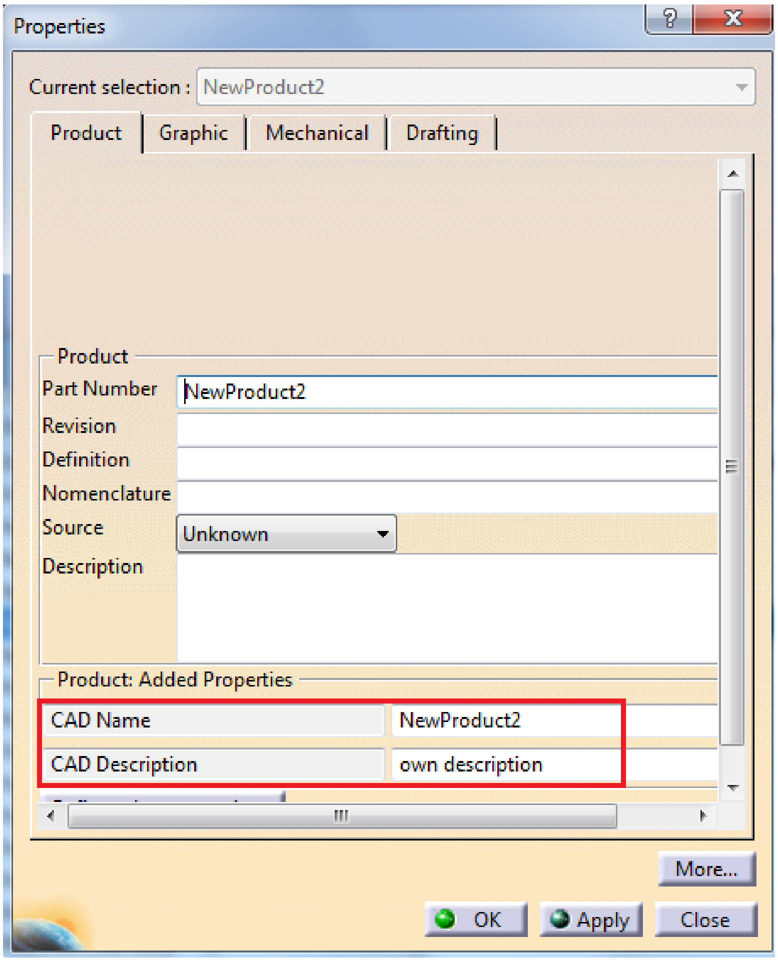


Picture 110: User-defined attributes in the "Properties" dialog of the PDM node



Picture 111: User-defined attributes in Aras Innovator window

After the import or after loading the structure it can be shown that the values are written from the PDM attributes into the CATIA files (see *Picture 112: User-defined attributes in the "Properties" dialog*).

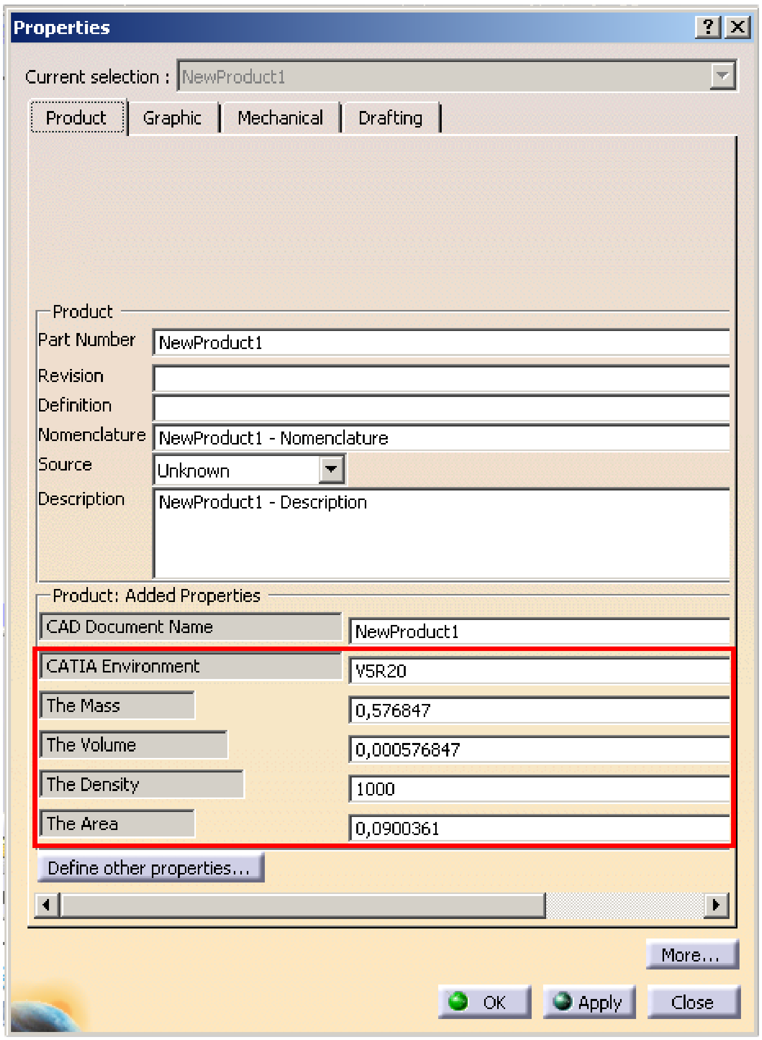


Picture 112: User-defined attributes in the "Properties" dialog.

Because of the mapping direction of the properties it is also possible to change the values in the Catia properties. After the update of the structure the values are written from the CATIA files into the PDM attributes.

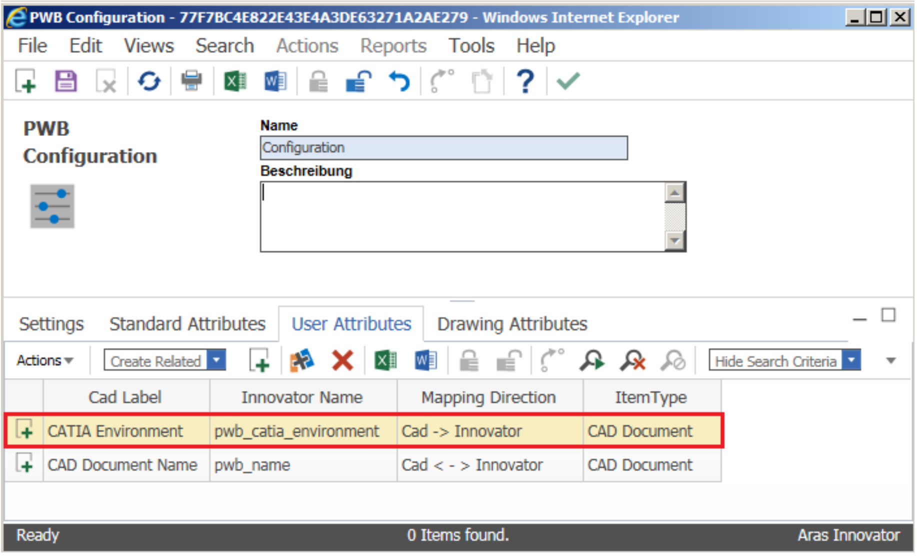
Internal CATIA information can be written to user-defined CATIA properties

If they are defined this way then the corresponding current values are written into the user-defined CATIA properties (see *Picture 113: User-defined attributes with internal CATIA information in the "Properties" dialog*).



Picture 113: User-defined attributes with internal CATIA information in the "Properties" dialog

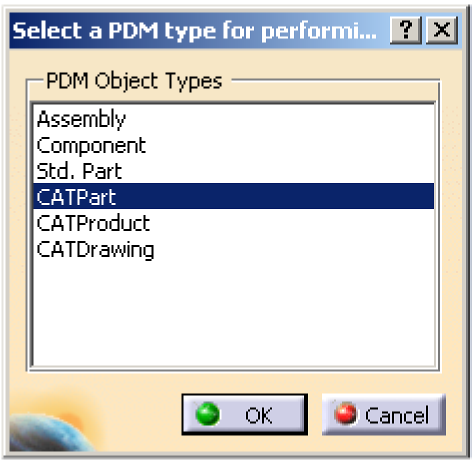
If these values are configured correspondingly, like for instance "CATIA Environment" in the picture below, then the values are automatically written to the mapped attributes of the PDM object (see *Picture 114: Configuration of user-defined attributes in Aras Innovator*).



Picture 114: Configuration of user-defined attributes in Aras Innovator

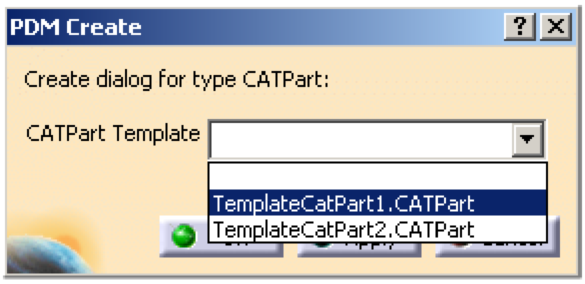
Template-based CAD document and part creation

In the dialog which appears after the user clicks on the toolbar action “Create”  the type of the new object can be selected (see *Picture 115: Select a PDM type for the “Create” dialog*).



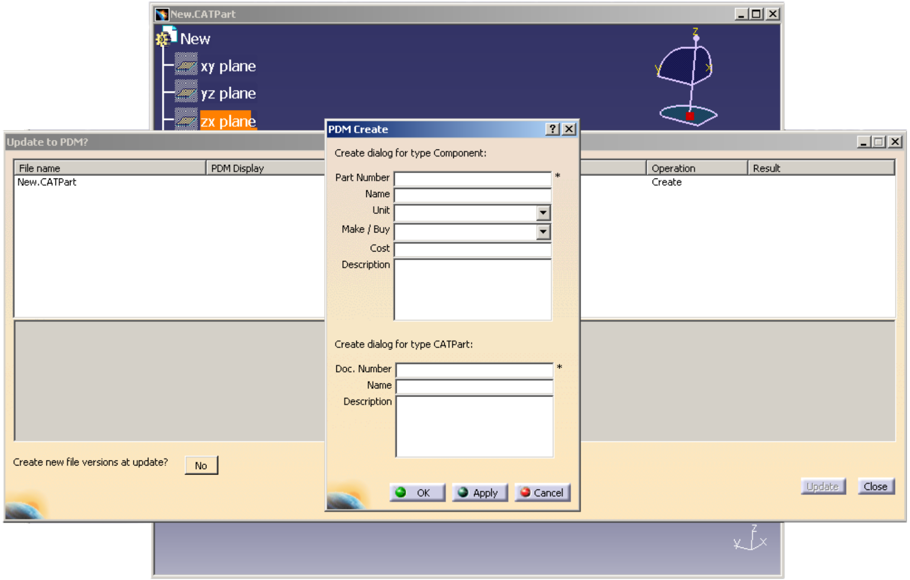
Picture 115: Select a PDM type for the “Create” dialog

If one of those types is selected then the next dialog opens. You have to select the template for the new object from a dropdown list. The template names are defined in the PDM Workbench configuration file. If you do not select a template an empty CATIA file is opened (see *Picture 116: “Create” dialog for CATPart – Select Template*).

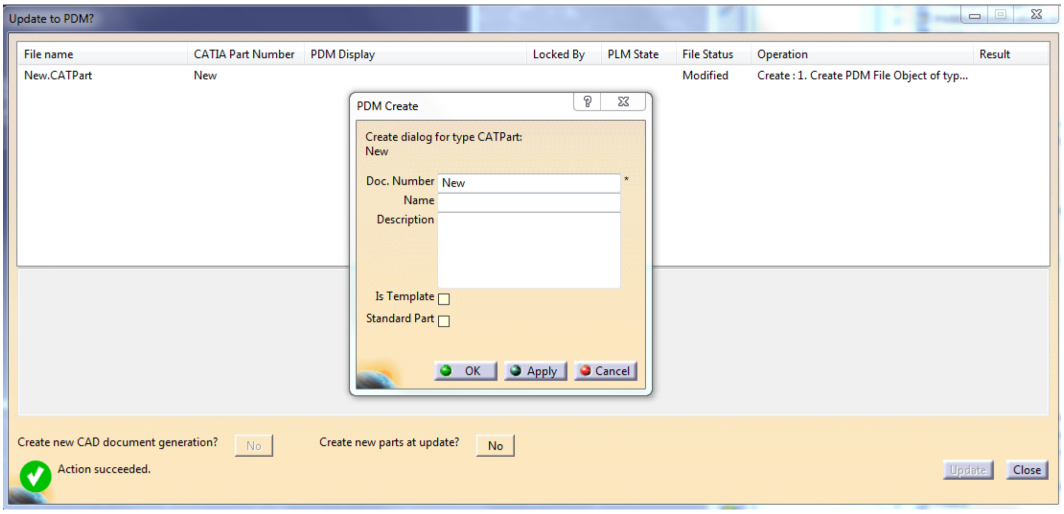


Picture 116: “Create” dialog for CATPart – Select Template

When you confirm the dialog with "OK" the template file or the empty file is opened in CATIA and the "Update to PDM?" dialog is opened. You have to type in the attribute values of the item object and the document object to be created (see *Picture 117: “Create” dialog for CATPart in BOM Part Structure*).

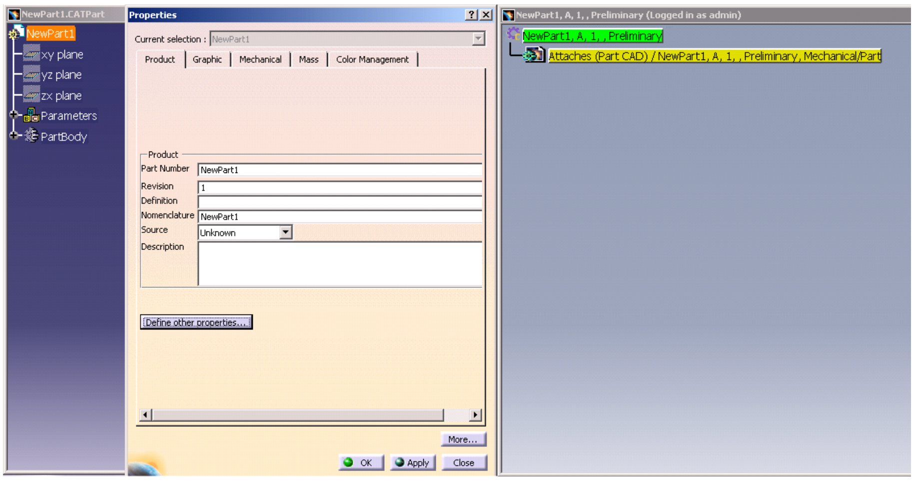


Picture 117: “Create” dialog for CATPart in BOM Part Structure



Picture 118: “Create” dialog for CATPart in Document Structure

Then you have to change the name and click on the “OK” button. A normal update is performed, which creates a part with the corresponding CAD document in the PDM system and uploads the file (see *Picture 119: Created Part*).



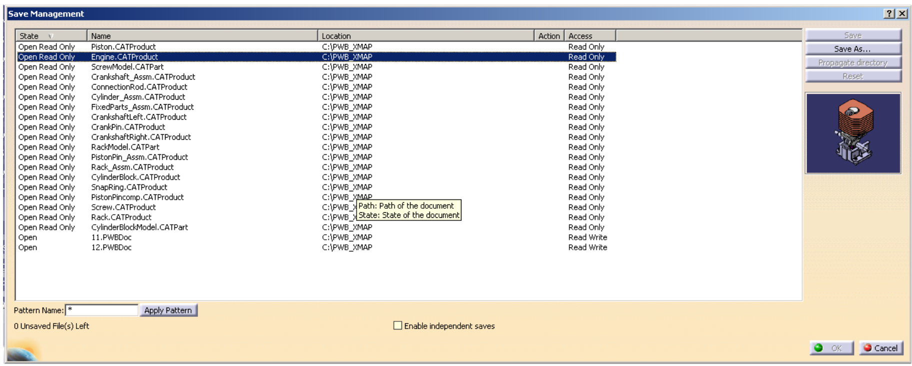
Picture 119: Created Part

Using standard parts

When CATParts which are defined as corresponding to a standard part are added to a CATProduct structure the update process will not try to create the corresponding part and CAD document objects in PDM. Instead the standard part object which has the same part number as the CATPart's CATIA part number will be queried and added to the part structure instead. Using this functionality standard part geometry can be added to a structure without having to first load the standard parts into the CATIA session.

CATIA documents are set to read-only if corresponding PDM node is not modifiable

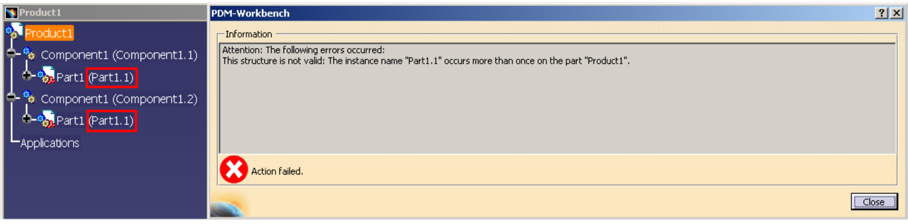
Loading a structure sets the corresponding CATIA files to read only (see *Picture 120: Save Management*).



Picture 120: Save Management

Added check whether CATIA structure is valid before update

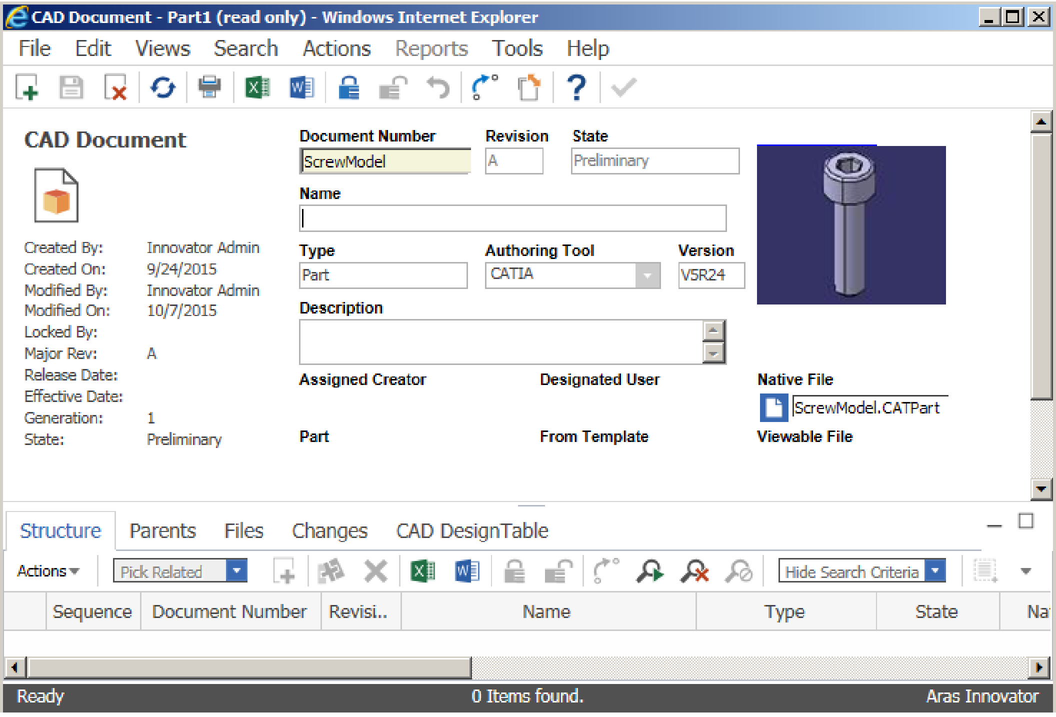
Before the structure will be created in the PDM system it will be checked if it is valid (see *Picture 121: Check if CATIA structure is valid*).



Picture 121: Check if CATIA structure is valid

Thumbnails

In the properties dialog for part and drawing documents a thumbnail will be shown (see *Picture 122: CAD Document properties in Aras Innovator*).

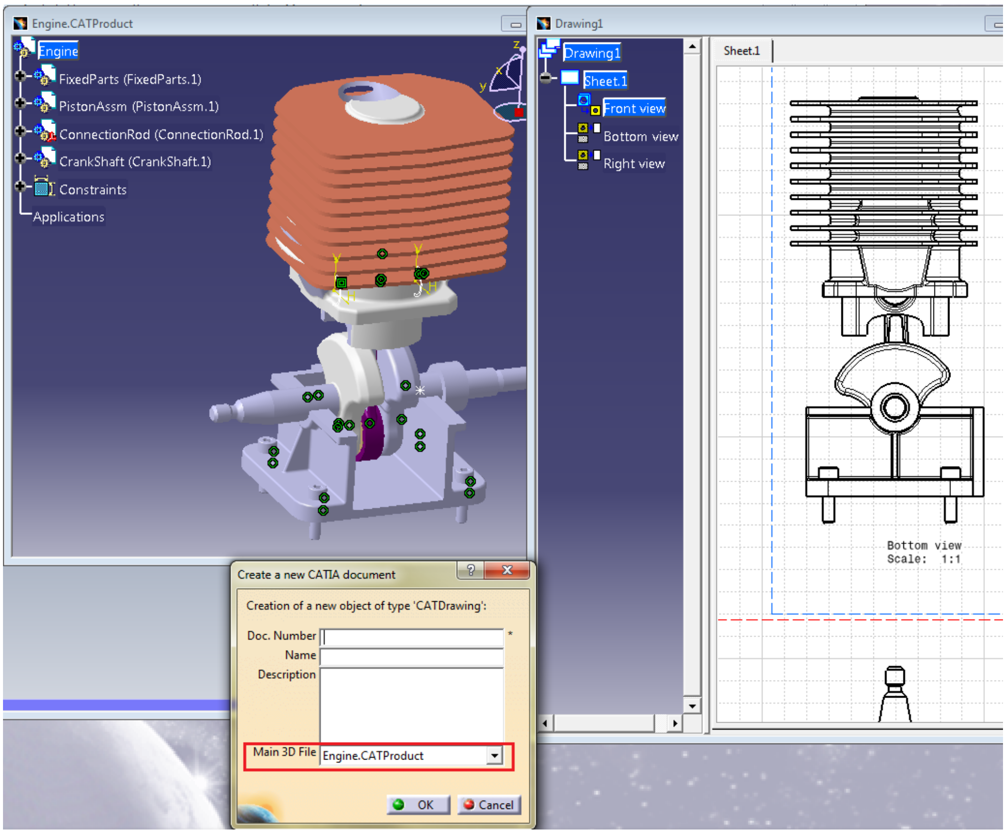


Picture 122: CAD Document properties in Aras Innovator

Basic Drawing Link Support

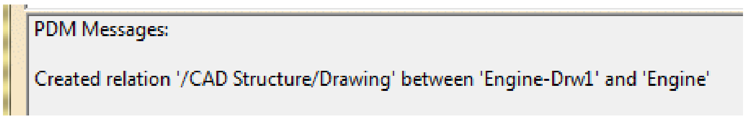
When a CATDrawing CAD document is created the user can decide which of the related 3D geometry documents is defined to be the main source geometry document. A link of the type “/CAD Structure/Drawing” is created from the CATDrawing to that 3D document when the drawing document is created.

If the source 3D geometry documents of the drawing are loaded from PDM and the PDM Workbench session contains information about the corresponding CAD documents in PDM then the linked 3D geometry document will be selected in the “Main 3D File” combo box if a single 3D document is linked (see *Picture 123: Creating a CATDrawing document with a link to 3D geometry*).



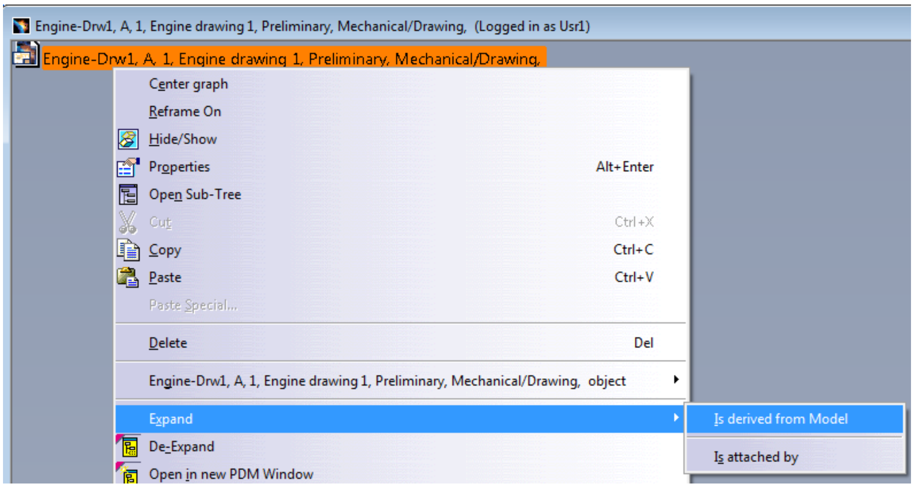
Picture 123: Creating a CATDrawing document with a link to 3D geometry

When the user creates the drawing CAD document a PDM relation of the type “/CAD Structure/Drawing” will be created in the PDM system after the CATDrawing PDM document has been created (see *Picture 124: PDM message about created drawing link*).



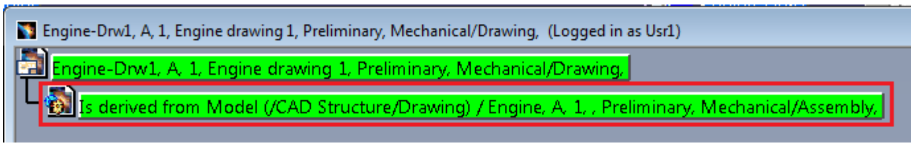
Picture 124: PDM message about created drawing link

You can expand to this document by selecting the CATDrawing object. Then click the right mouse button and select "Expand→Is derived from Model" (see *Picture 125: Expanding newly created drawing link*).



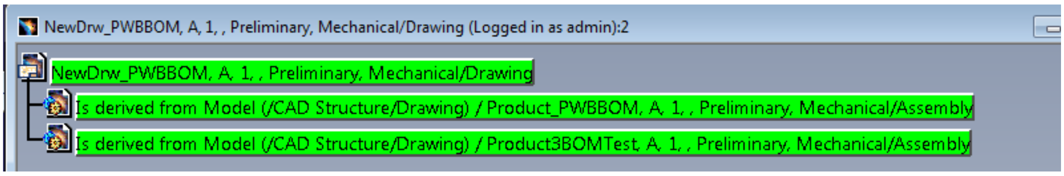
Picture 125: Expanding newly created drawing link

The document will be displayed in the window (see *Picture 126: Displaying newly created drawing link*).



Picture 126: Displaying newly created drawing link

If the user creates a drawing with links to more than one 3D geometry file then both linked documents will be displayed in the window (see *Picture 127:*).



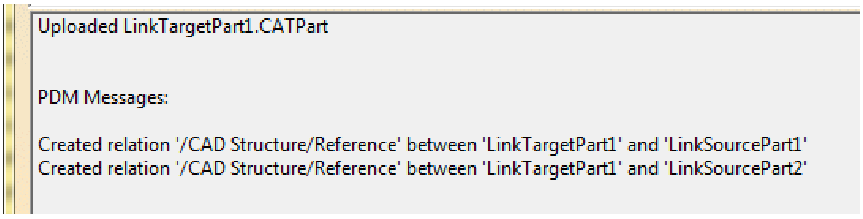
Picture 127: Displaying all created drawing links.

Drawing links are not updated or deleted when a CATDrawing is updated, even if links are created or removed in CATIA. The related primary 3D document is not supposed to change during the lifetime of the CATDrawing.

Basic Multi-Model Link Support

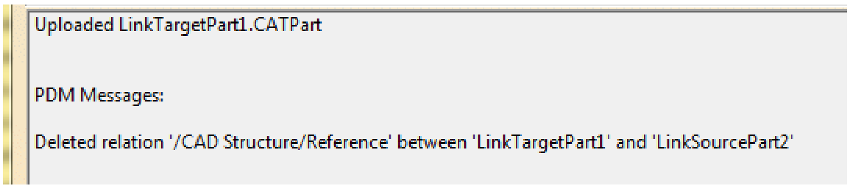
When a CATPart CAD document is created or updated the geometry links of imported 3D geometry will be updated as PDM links of the type “/CAD Structure/Reference”. Both reference links and instance links are supported.

If the functionality is switched on, when a CATPart contains geometry links, PDM relations of the type “/CAD Structure/Reference” which correspond to these links are created (see *Picture 128: Information when reference links are created*).



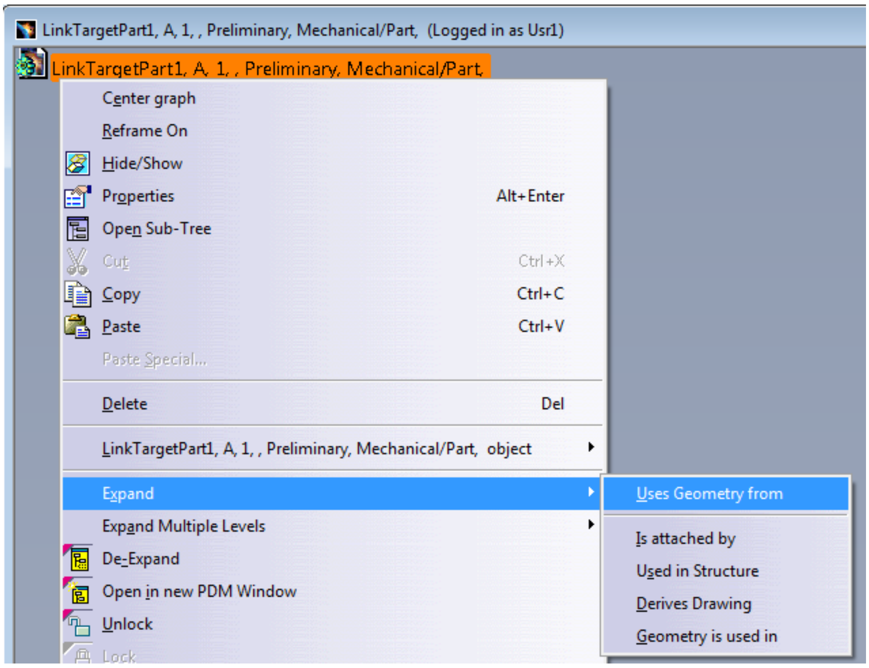
Picture 128: Information when reference links are created

If the links are removed from the CATIA file then the corresponding PDM relations are deleted (see *Picture 129: Information when reference links are deleted*).



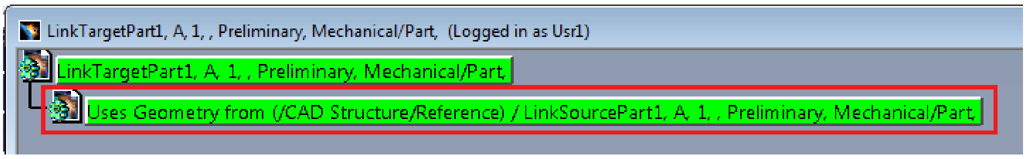
Picture 129: Information when reference links are deleted

The created links can be expanded. Select the document and click the right mouse button. Select "Expand→Uses Geometry from" (see *Picture 130: Expanding geometry links*).



Picture 130: Expanding geometry links

The document will be displayed in the window (see *Picture 131: Geometry link expansion result*).



Picture 131: Geometry link expansion result

Management of CATIA templates in Innovator

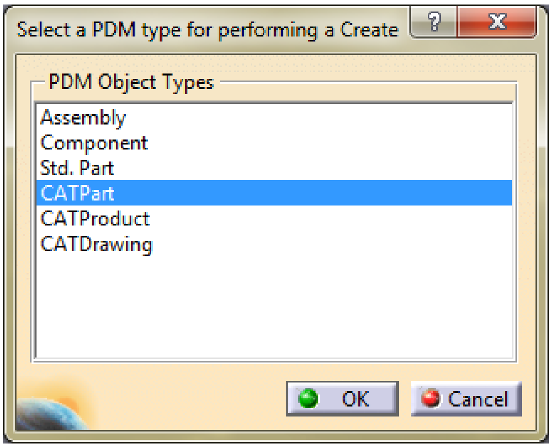
The existing template file functionality, where template CATIA files are stored on a local directory which is accessible from CATIA V5, is extended such that the file templates can be stored in PDM.

The template file functionality can be used in two ways:

1. When new CATIA document objects are created from the 'Create' command in the PDM Workbench toolbar.

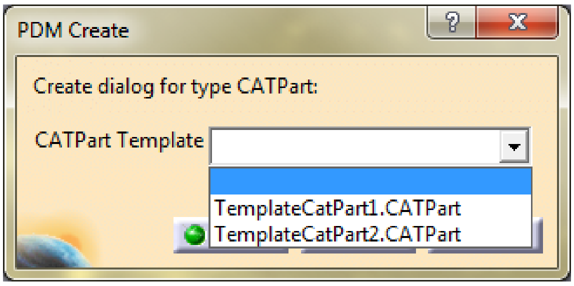
Example:

Create CATPart (see *Picture 132: Template file functionality: Creating a CATPart*).



Picture 132: Template file functionality: Creating a CATPart

If templates are configured then the user gets to choose a template file name (see *Picture 133: Template file functionality: Selecting a template file*).



Picture 133: Template file functionality: Selecting a template file

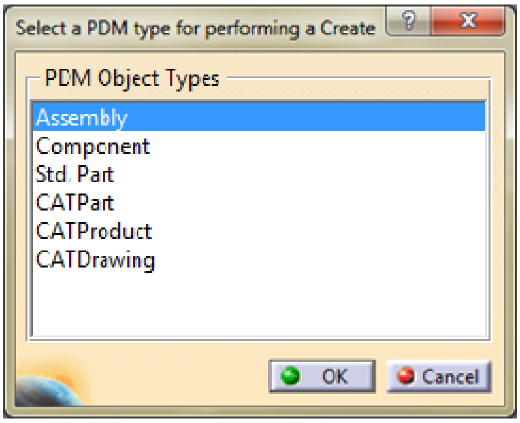
If no template file is chosen a new CATIA CATPart file will be created.

2. When a part structure is loaded where one or more parts do not have a related CATIA file.

This use case is applicable when using BOM part structures ("UseBomPartStructure" is set to "true").

Example:

Create an Assembly (see *Picture 134: Template file functionality: Creating an assembly*).



Picture 134: Template file functionality: Creating an assembly

Fill all the necessary Assembly information on the Create dialog and click OK.

The created Assembly is opened in a new PDM window.

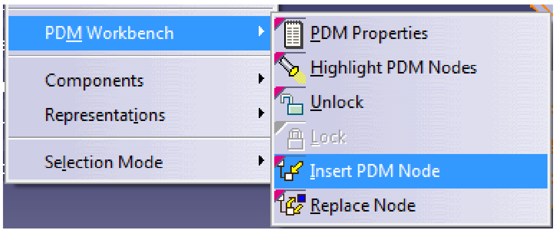
Right click on the Assembly node and chose "Load" from the context menu.

In this case the corresponding CATIA structure nodes are created on the fly using the first template file in the list, which is defined as the default template.

Insert from PDM

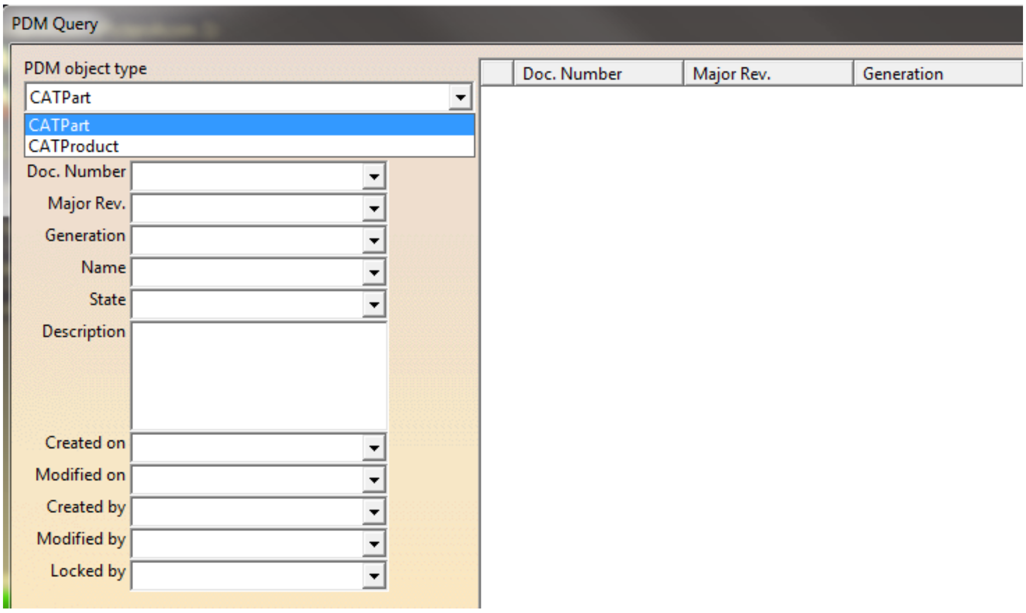
The CATIA file which corresponds to an existing Innovator item can be inserted directly into the CATIA structure.

The user right-clicks on an existing CATProduct node in a CATIA structure window and selects “Insert PDM Node” (see *Picture 135: “Insert PDM Node” context menu*).



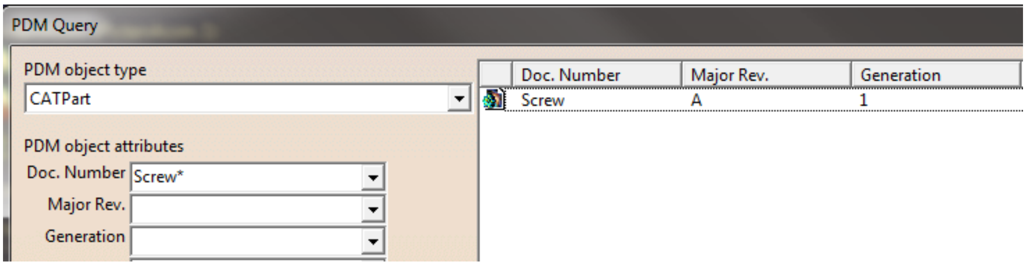
Picture 135: “Insert PDM Node” context menu

A restricted query window opens. In CAD structure mode CATPart and CATProduct items can be queried, in BOM structure mode Part items can be selected (see *Picture 136: “Insert PDM Node” query dialog type selection*).



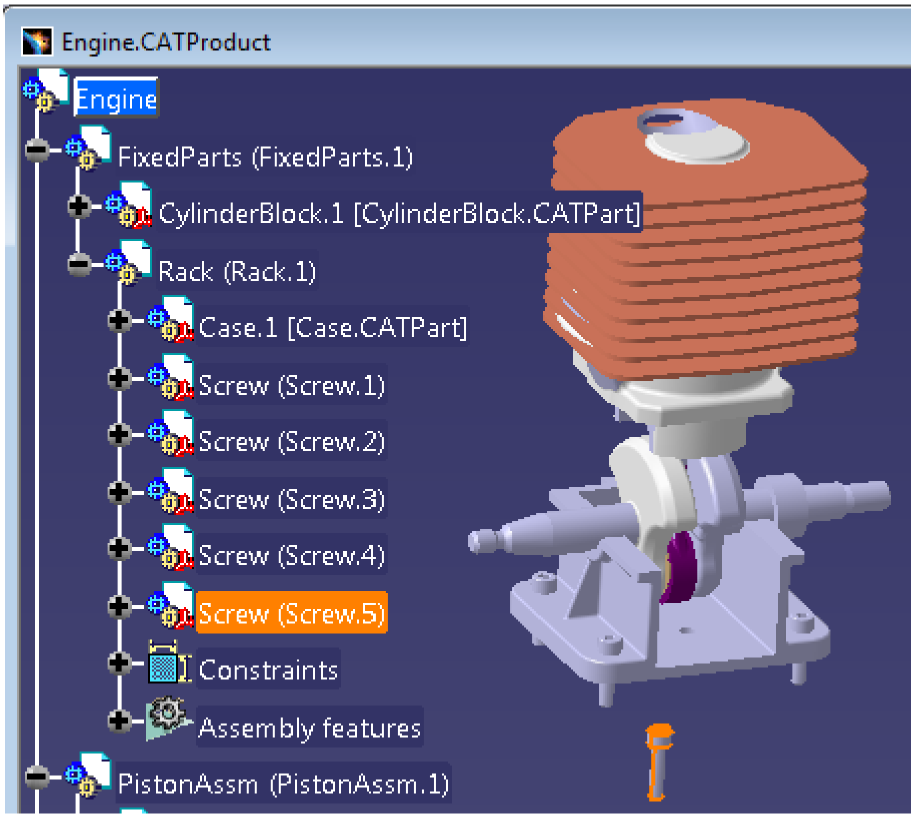
Picture 136: “Insert PDM Node” query dialog type selection

After the query is performed all resulting items are displayed in the result list, like in the regular query dialog (see *Picture 137: “Insert PDM Node” query result*).



Picture 137: "Insert PDM Node" query result

Double-clicking on one of the result items in the list causes its corresponding file to be downloaded and inserted into the selected CATProduct node in the CATIA structure:



Picture 138: Item inserted in existing structure

It is possible to query for items which are already contained in the CATIA structure, as well as for items which do not exist in the structure yet.

The newly inserted CATIA node is not updated to PDM yet, the next Update process will create the corresponding structure relation.

Replace from PDM

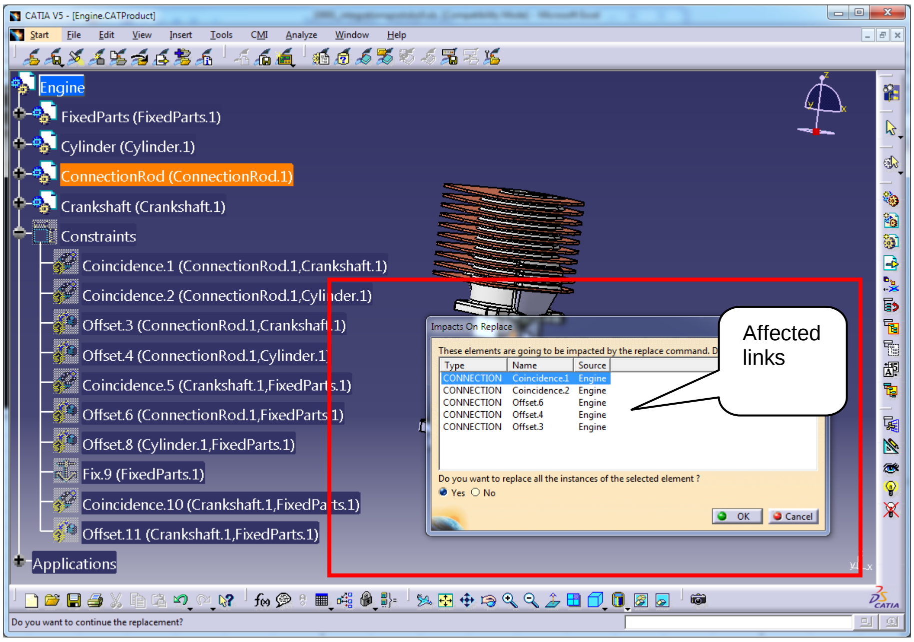
The node selected in the CATIA structure can be replaced by the CATIA file which corresponds to an existing Innovator item.

Warning: This functionality can create broken links in the CATIA structure.

In order for CATIA links to be preserved, the link conditions of the replaced geometry and the replacing geometry has to be compatible.

If the new CATIA document is not compatible with the link conditions of the product structure the CATIA V5 Replace functionality presents a warning dialog.

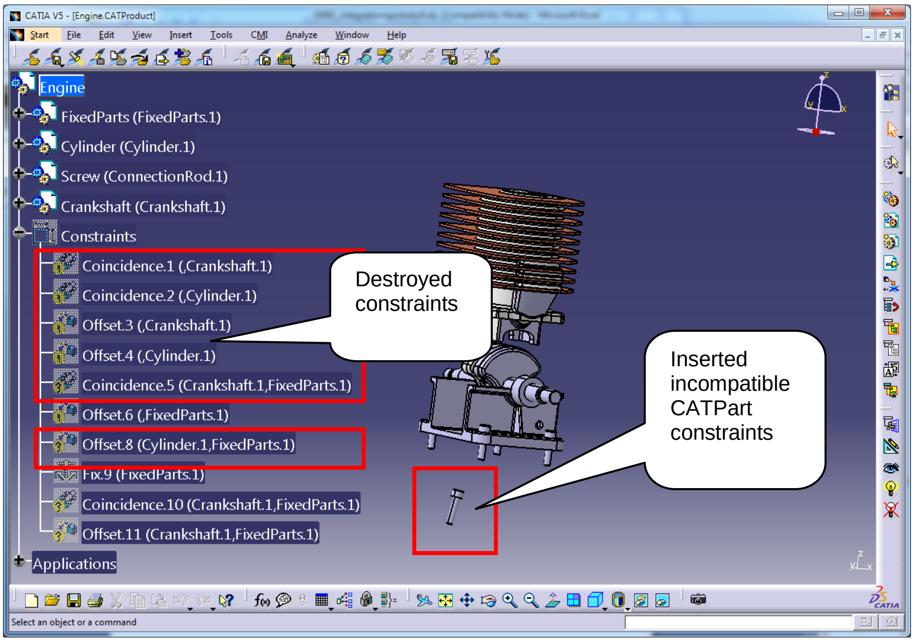
This is the “Impacts on Replace” dialog as presented by the standard CATIA V5 Replace functionality (see *Picture 139: The “Impacts on Replace” standard CATIA dialog*).



Picture 139: The “Impacts on Replace” standard CATIA dialog

This “Impacts on Replace” dialog cannot be implemented by T-Systems due to lack of sufficient APIs.

If the inserted CATIA document is not compatible linkage information in the product structure gets lost. In this case some constraints are broken (see *Picture 140: Constraints destroyed by “Replace” operation*).

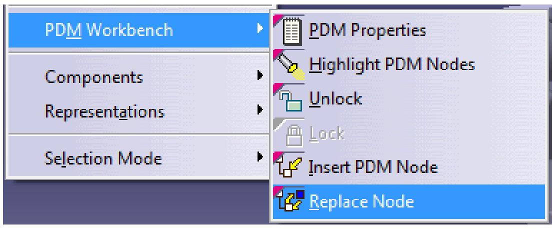


Picture 140: Constraints destroyed by “Replace” operation

This possibility of breaking CATIA links needs to be taken into account when this functionality is used. If in doubt please load the geometry which is supposed to replace existing geometry in the structure with “Load” and use the regular CATIA replace operation.

The functionality is used as follows:

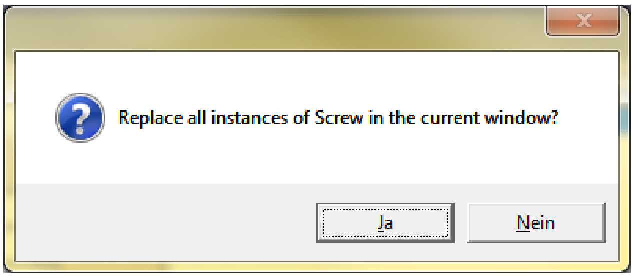
The user right-clicks on an existing CATProduct or CATPart node in a CATIA structure window and selects “Replace Node” (see *Picture 141: “Replace Node” context menu*).



Picture 141: “Replace Node” context menu

As in the “Insert PDM Node” case, a restricted query window opens. The user can query for existing items and double-click on one of the found items to select the one which should replace the selected CATIA structure node.

The user then gets to decide whether only the selected node or all instances of the document should be replaced (see *Picture 142: “Replace all instances” prompt*).



Picture 142: “Replace all instances” prompt

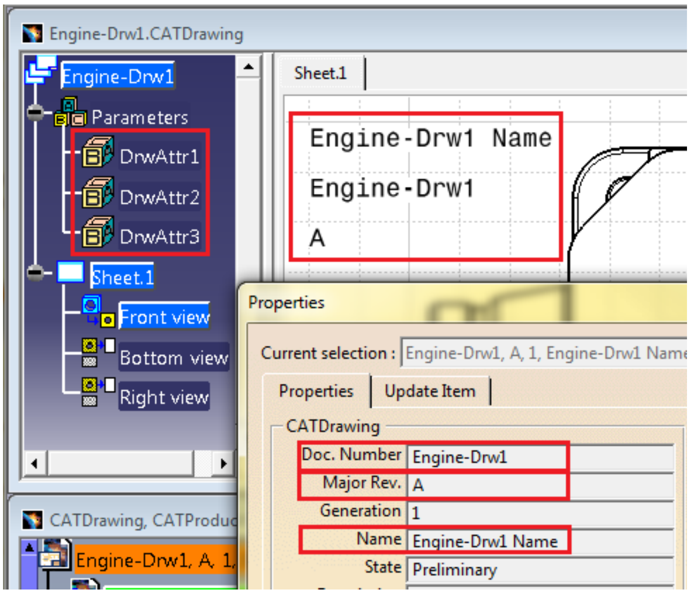
After that the selected instance or all of the instances of the selected CATIA node will be replaced by the CATIA file related to the queried and selected PDM item.

It is not possible to replace CATIA documents in a CATIA session when the new file has the same filename as the file to be replaced, because both files would be located in the same directory (PWB exchange map).

Replacing CATIA nodes does not change the instance names of the replaced nodes. Only nodes in the structure in the active window are affected by the replace operation.

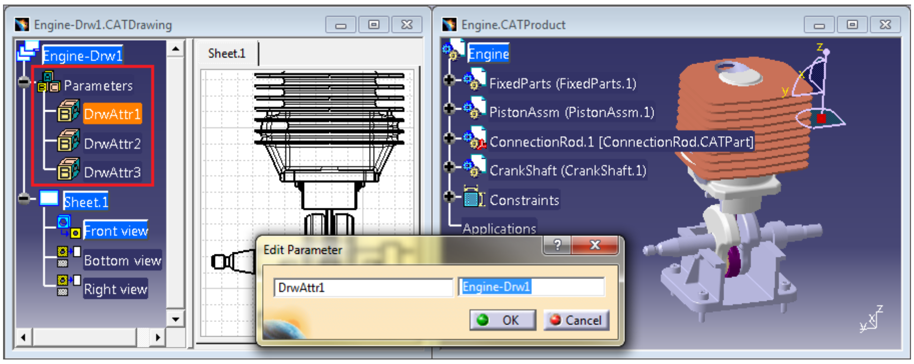
CATDrawing attribute mapping

PDM attribute values of the drawing CAD document item can be mapped to CATDrawing attributes (see *Picture 143: CATDrawing attribute mapping*).



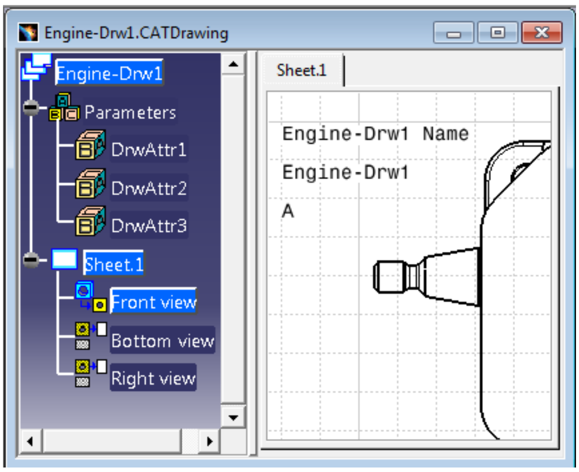
Picture 143: CATDrawing attribute mapping

After the drawing is created in PDM the CATDrawing file contains the attributes defined in the configuration attribute "CatiaDrawingCadFileAttributes", which in turn contain the values of the PDM attributes on the CATDrawing document defined in "CatiaDrawingPdmDocAttributes" (see *Picture 144: Derived CATDrawing containing attributes*).



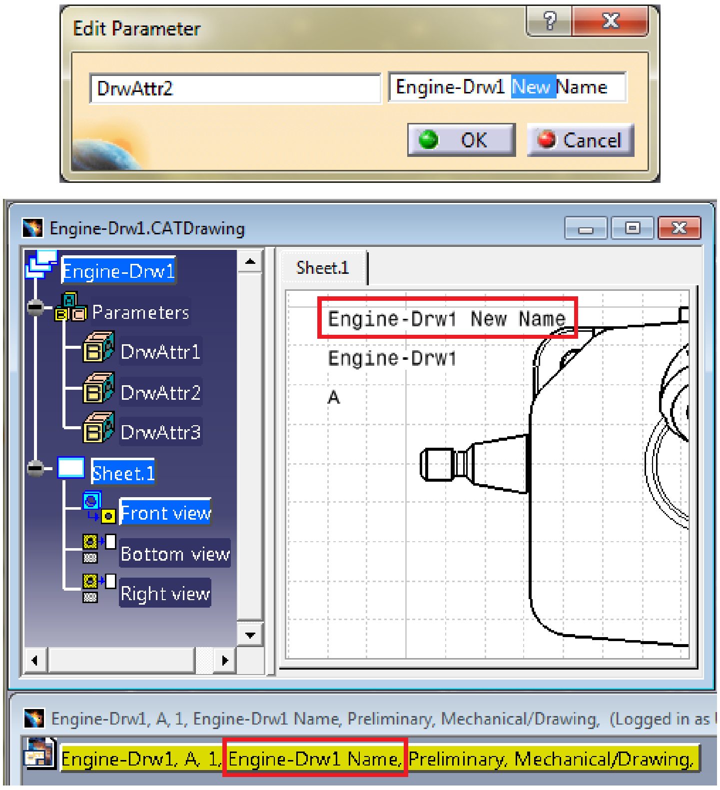
Picture 144: Derived CATDrawing containing attributes

These drawing attributes' values can be assigned to text fields in the 2D drawing (see *Picture 145: Drawing attributes' values assigned to text fields*).



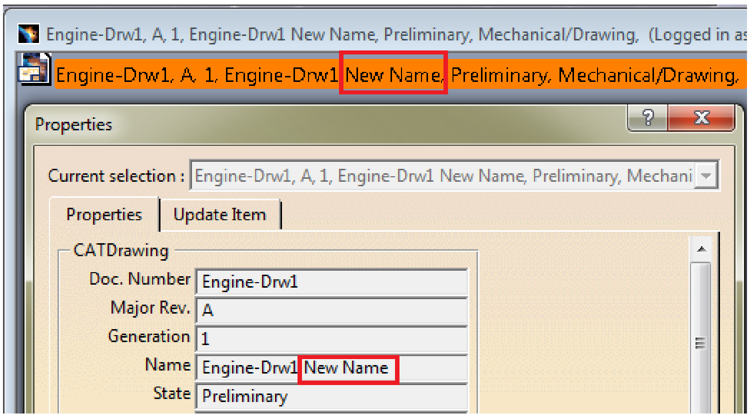
Picture 145: Drawing attributes' values assigned to text fields

The user can modify one of the drawing attributes (see *Picture 146: Modified drawing attribute value*).



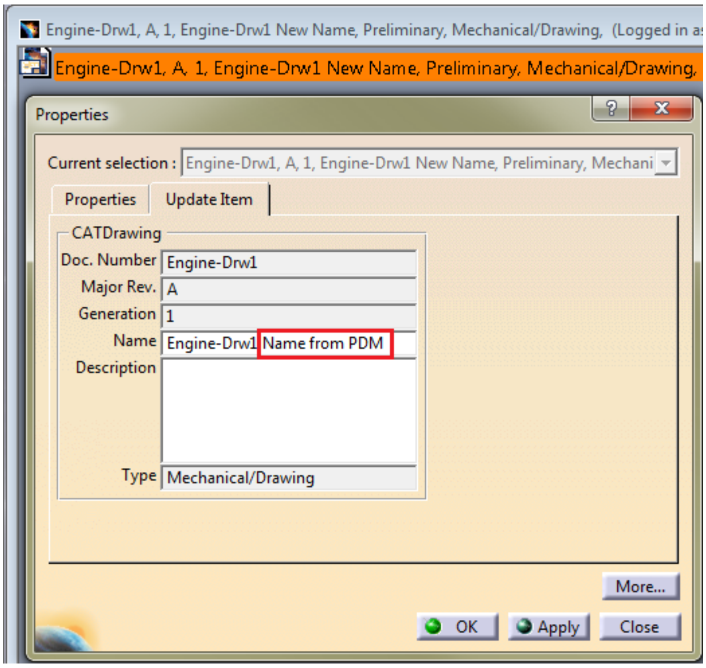
Picture 146: Modified drawing attribute value

After a PDM update, the new attribute value is written into its corresponding PDM attribute (see *Picture 147: Modified PDM attribute value*).



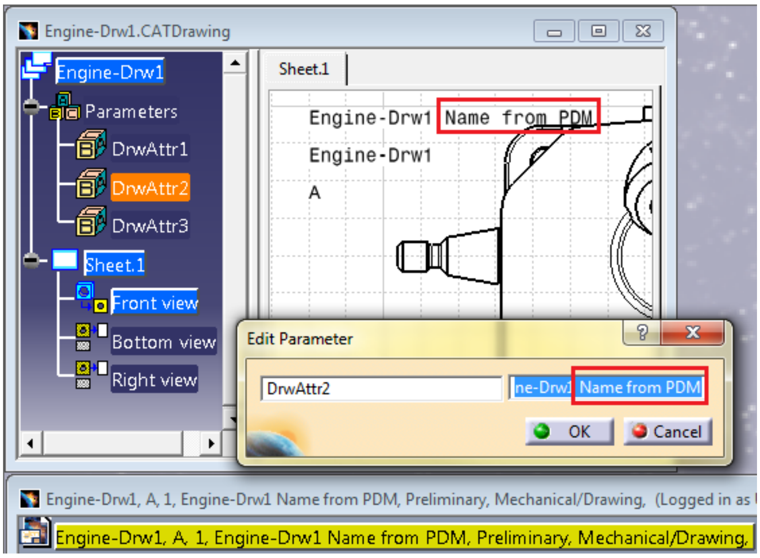
Picture 147: Modified PDM attribute value

If, on the other hand, the user changes one of the mapped PDM attributes, the new value will be written into the CATDrawing attribute the next time the CATDrawing is loaded to CATIA (see *Picture 148: PDM attribute value modified from Innovator*).



Picture 148: PDM attribute value modified from Innovator

You can see the PDM attributes in the parameters of the CATDrawing (see *Picture 149: Drawing attribute value changed to PDM attribute value*).



Picture 149: Drawing attribute value changed to PDM attribute value

Create new version

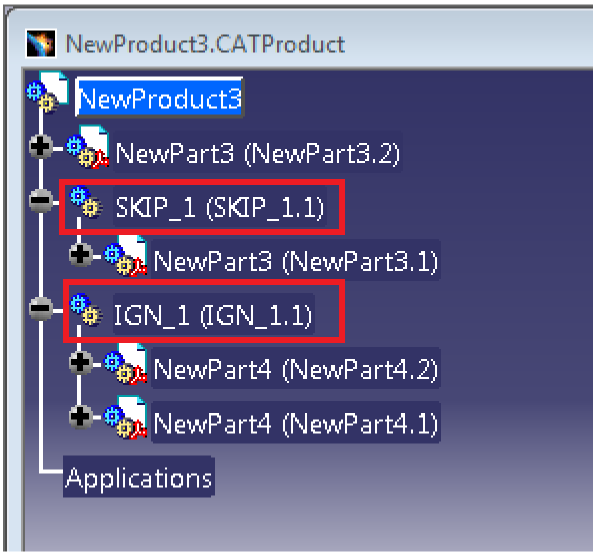
A new generation of a CAD document can be created by clicking on the “Create new version” context menu in the PDM window (see *Picture 150: “Create new version” PDM context menu*).

Picture 150: “Create new version” PDM context menu

Configurable CATIA components support

It is possible to load and update CATProduct structures which contain embedded CATIA components. Depending on the part number prefix the embedded component nodes can either be “skipped”, that is, the node is treated as if it does not exist, but its child nodes are processed, or they can be “ignored”, that is, the node and all its child nodes are treated as if they do not exist.

In the following example the two instance nodes of the CATPart “NewPart3” are treated as if they were both directly under the CATProduct “NewProduct3”, and the two instance nodes of the CATPart “NewPart4” are completely ignored, that is, the structure is treated as if they do not exist (see *Picture 151: Embedded CATIA component nodes*).



Picture 151: Embedded CATIA component nodes

Support Electrical / Tubing

With this functionality it is possible to use functions like “Electrical Harness”, “Electrical Wire Routing”, “Piping Design”, “Tubing Design”, … of the CATIA “Equipment & Systems Engineering” section.

The functions of the “Equipment & Systems Engineering” section often create embedded leaf components (without files) of special types like “ElecWireLight”.

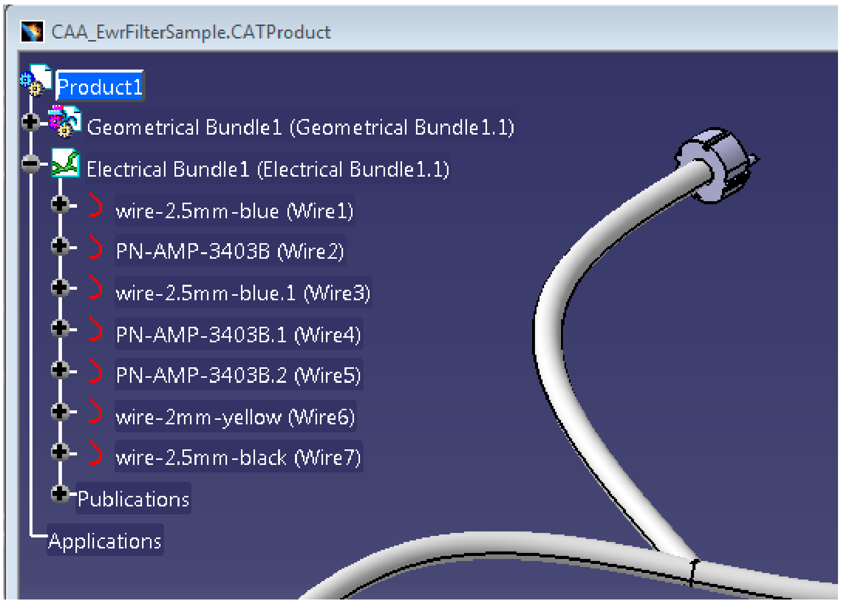
By default PDM Workbench does not support embedded components in the CATIA structure, because an embedded component does not have an own file, but is stored in the parent CATProduct. Therefore such a component cannot be reused under a different parent.

There is no need to map these leaf components to PDM documents / parts. The parent CATProduct of the embedded leaf components holds all information of the embedded leaf components.

This functionality allows two ways to use such embedded leaf components:

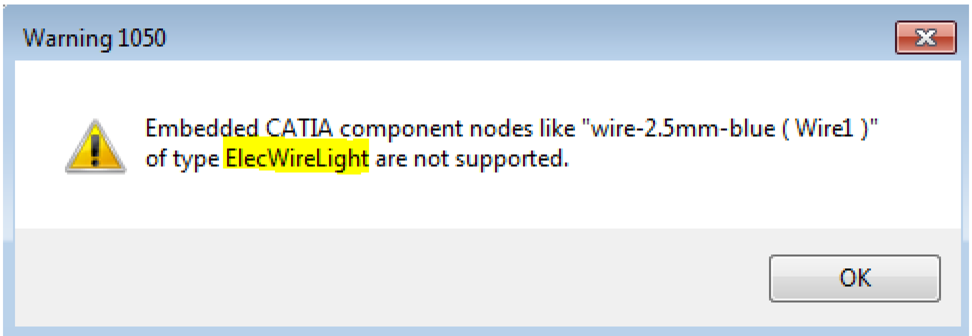
a) Allow leaf components of any type in the CATIA structure. Leaf components are not mapped to PDM documents / parts

b) Configure special types of like “ElecWireLight” to be allowed in the CATIA structure. Components of the configured types are not mapped to PDM documents / parts



Picture 152: Example document containing electrical components

To get the type of a component just use the PWB Update functionality. If there is an unsupported component, a message box shows the type of the first unsupported component:

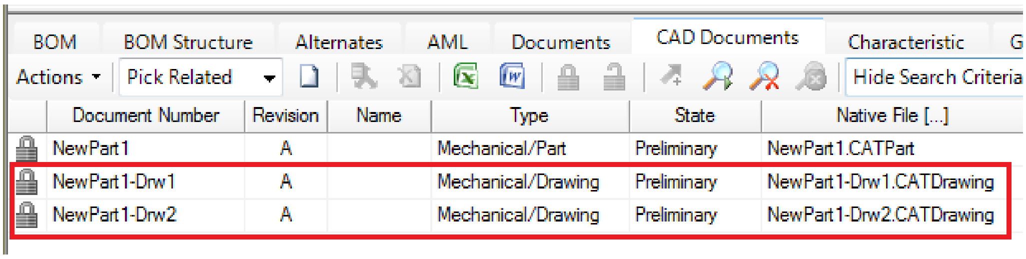


Picture 153: Warning about unsupported CATIA component node

Download Drawing Option in Query Dialog

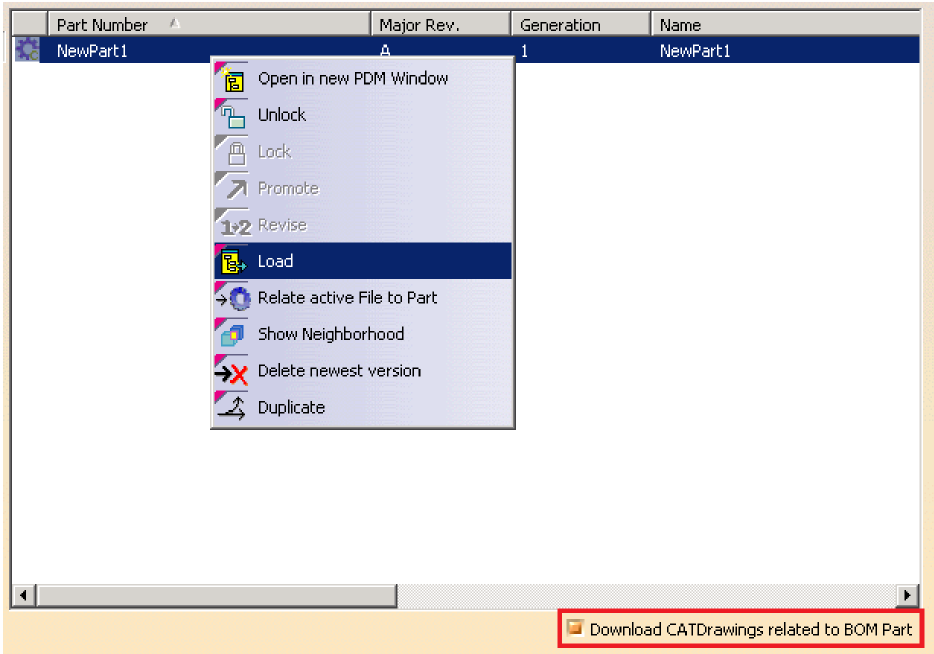
In the part structure mode the query result dialog contains a new check box with which the user can define whether CATDrawings related to the part being loaded should also be downloaded.

In this example two CATDrawing documents are related to the part item, in addition to the CATPart document:



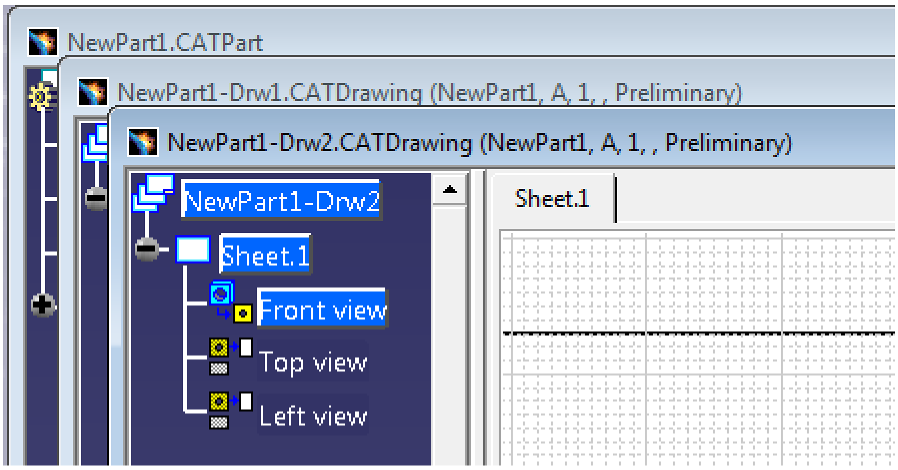
Picture 154: CATDrawing documents related to part item

When the check box is checked then the related CATDrawings are downloaded and opened in the CATIA session when the part is loaded.



Picture 155: Query result dialog with drawing download check box

➔

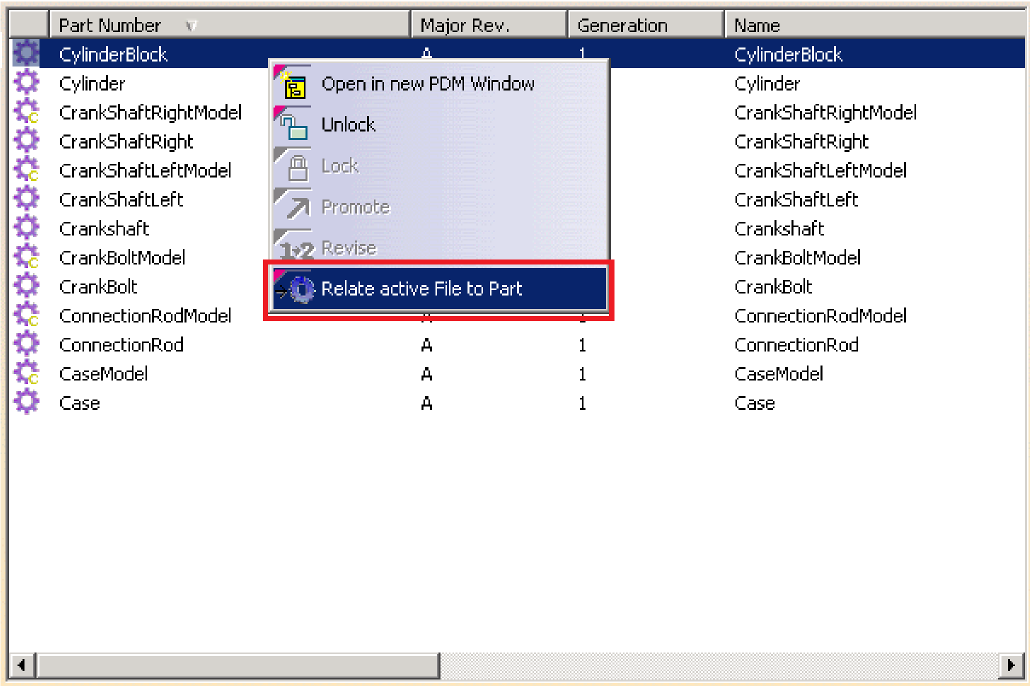


Picture 156: CATDrawings opened in CATIA session

Support for relating a new CATIA file to an existing Part

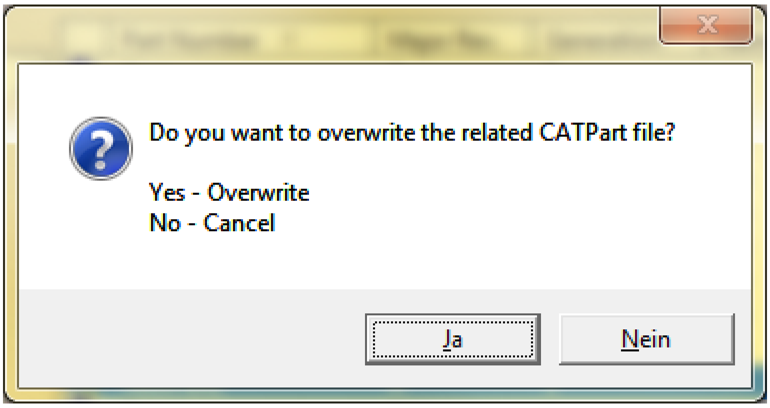
The currently active CATIA document (only CATParts or CATDrawings) can be related to an existing BOM part item. If there is already a corresponding CAD document related to the part the document's file can be overwritten.

The CATPart or CATDrawing file which is the currently active document in the CATIA session can be related to a part in the query result list by a new context menu action:



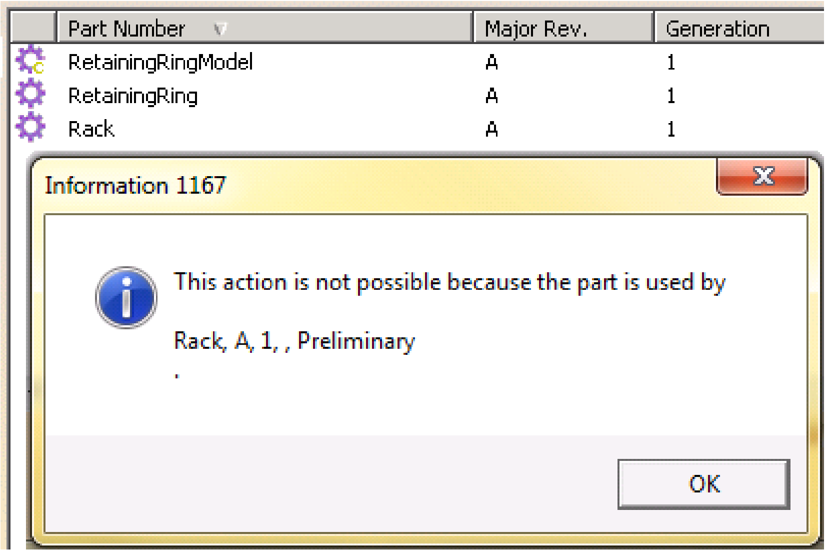
Picture 157: "Relate active File to Part" context menu action

If there is already a CAD document related to the part the user is asked whether he wants to overwrite the corresponding file:



Picture 158: Overwrite prompt for "Relate active File to Part" context menu action

The file can only be overwritten if the BOM part item is not used in a Part BOM structure:

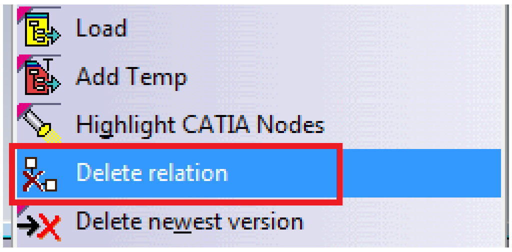


Picture 159: Information prompt for "Relate active File to Part" context menu action

"Delete relation" context menu action in the PDM structure window

PDM relations can be deleted in the PDM structure window with a single context menu action now, even if the PDM relations are not displayed in the structure.

The "Delete relation" action deletes the expanded parent relation of the selected PDM structure node in the PDM structure window (see *Picture 160: "Delete relation" context menu action*).

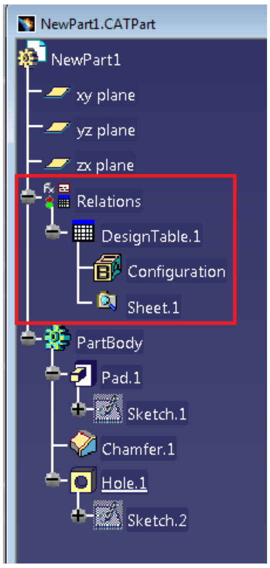


Picture 160: "Delete relation" context menu action

Support for CATIA Design Tables

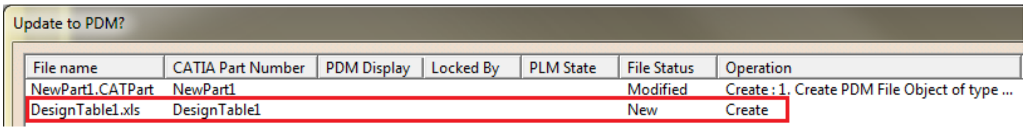
It is possible to load and update text files or Microsoft Excel files which contain CATIA design table information.

The user can create a design table for a CATPart or a CATProduct.



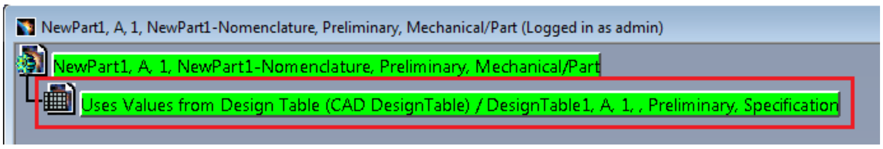
Picture 161: CATPart with design table

Updating to PDM will create a Document item for the design table and upload the file:



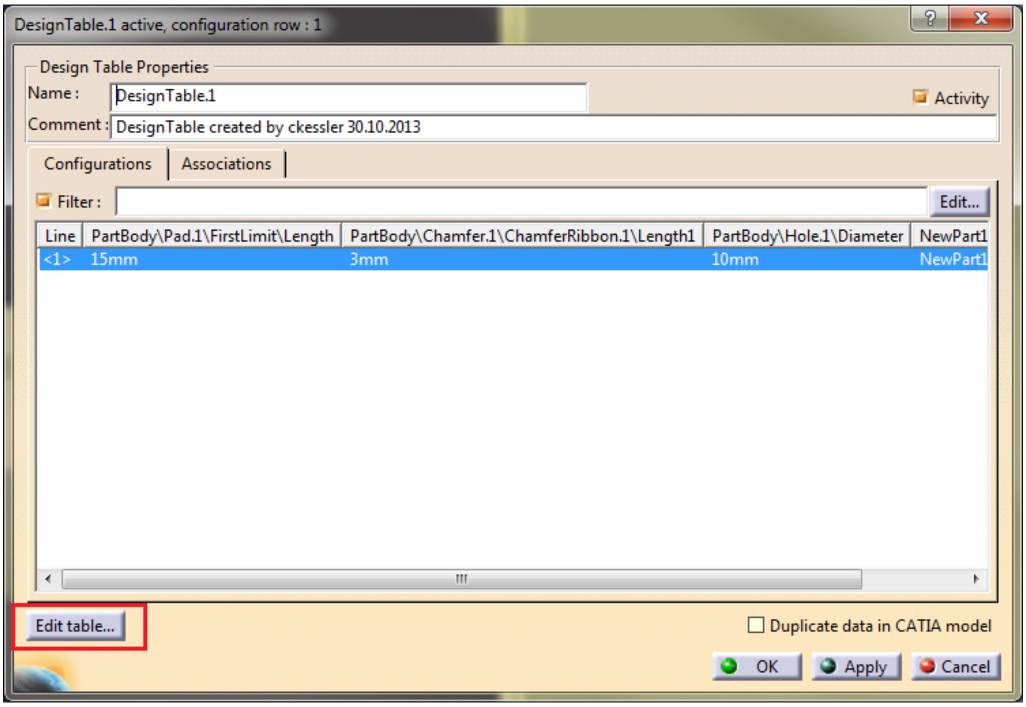
Picture 162: Update dialog containing a design table

After the update the design table is related to the CAD document:

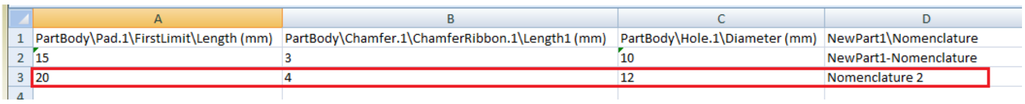


Picture 163: Design table document related to CAD document

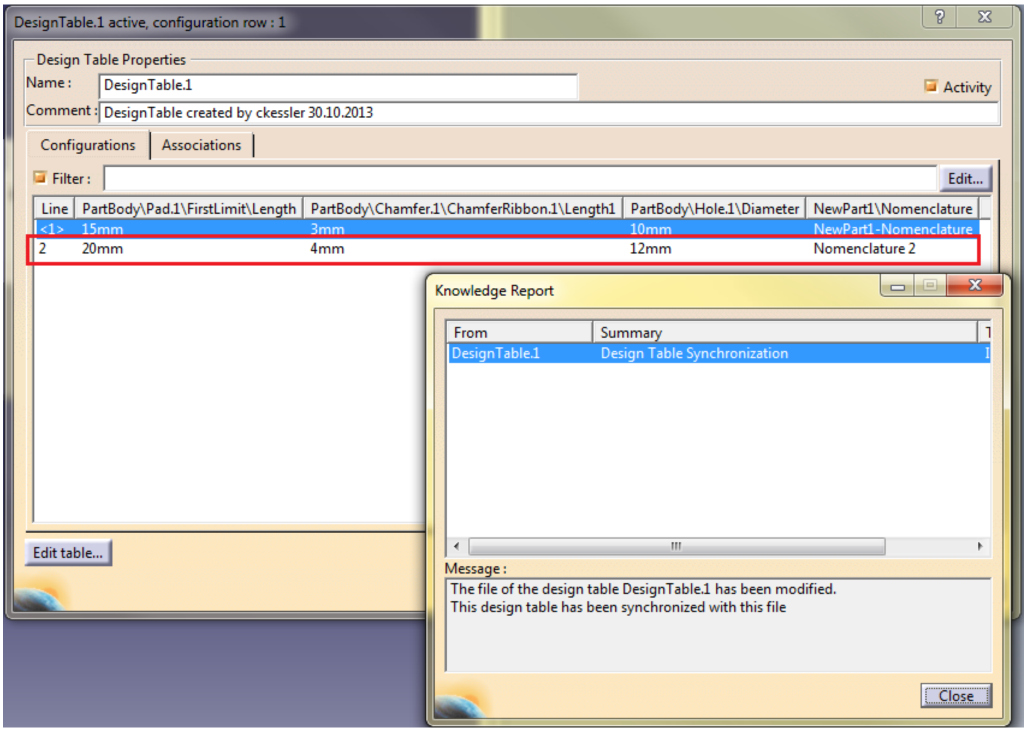
The design table file can be modified and uploaded to PDM again:



Picture 164: Editing a design table

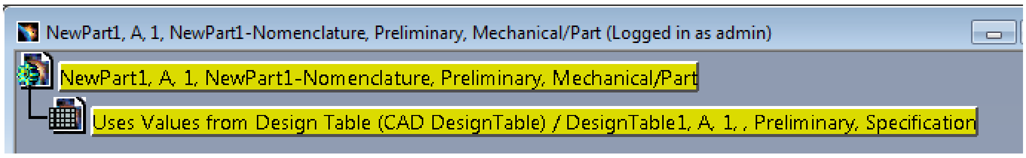


Picture 165: Adding a line to the design table excel sheet



Picture 166: The design table is updated in the CATIA session

Refreshing the PDM structure window shows that both the CATIA document and the design table are modified:



Picture 167: Refreshed PDM structure window containing the design table

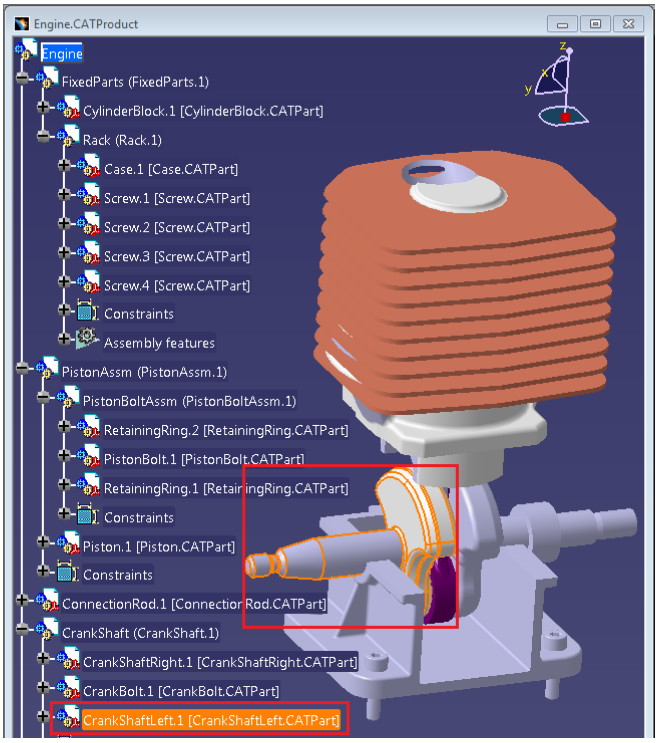
PDM update uploads both changed files.

When the design table functionality is switched on the design table files that are related to CAD documents are also downloaded when the CATIA files are downloaded.

Bounding Box Management / "Show Neighbor" functionality

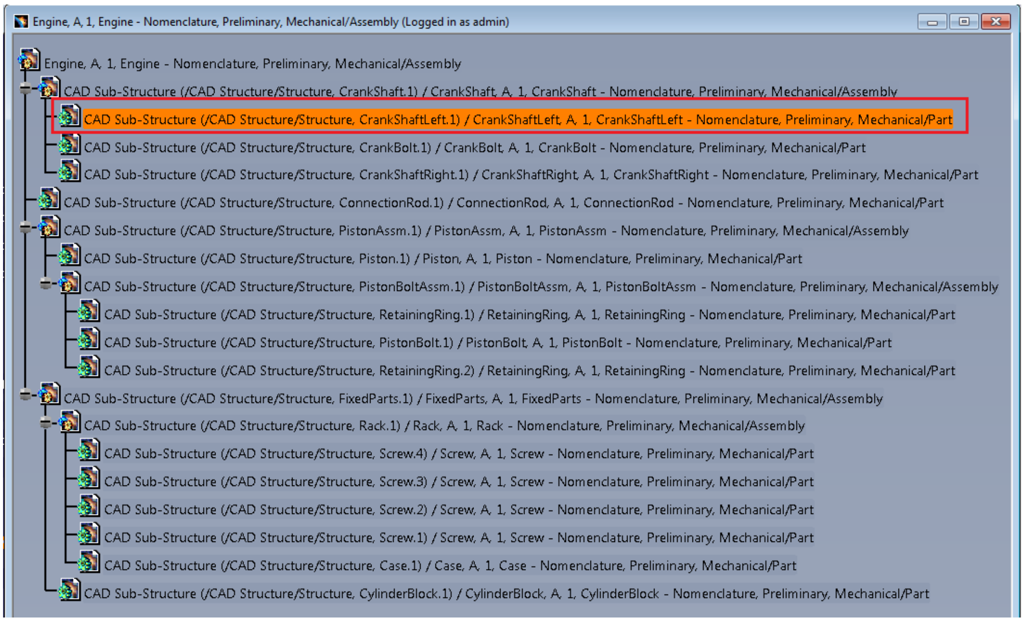
PDM Workbench can be set up such that the bounding box values of the updated CATParts are saved in the PDM CAD document items. If that is done it is possible to use these values to find the neighbor geometry documents whose bounding boxes overlap with the bounding box of the selected CATPart.

As an example, the user wants to find out which bounding boxes of other CATParts in the structure "Engine.CATProduct" overlap with the bounding box of the CATPart "CrankShaftLeft.CATPart":



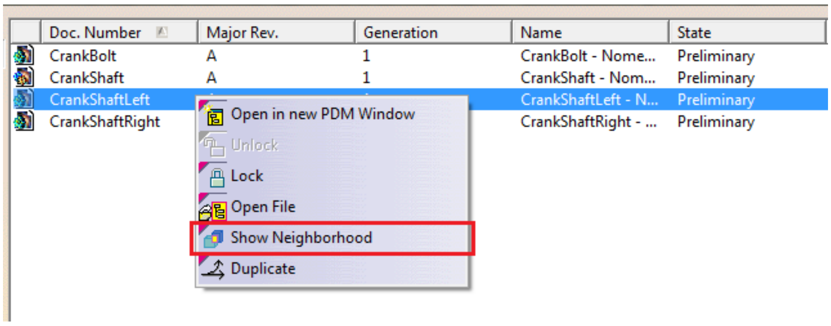
Picture 168: CATPart geometry in the context of a CATProduct structure

This is the corresponding CAD structure in PDM:



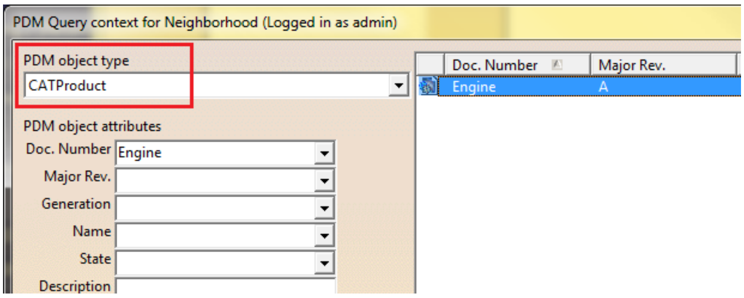
Picture 169: CATPart document in CAD structure

The user first queries for the CATPart document which he wants the neighborhood of. Then he clicks on the context menu “Show Neighborhood”:



Picture 170: “Show Neighborhood” context action

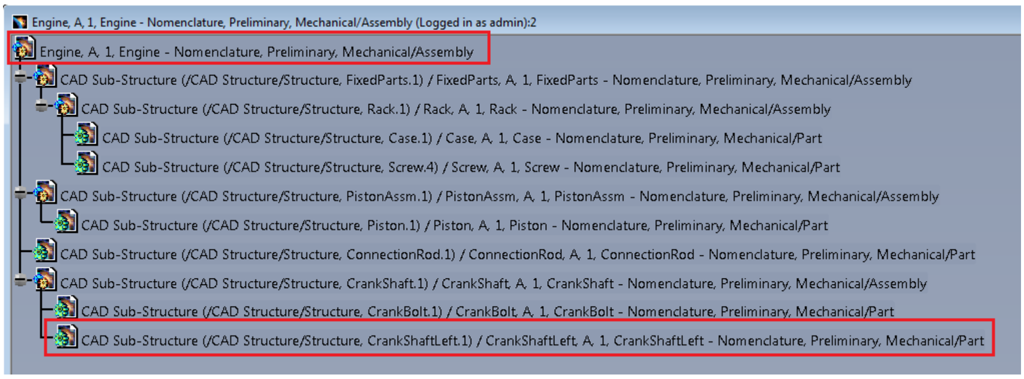
A query window appears where the user can search for a CATProduct document (or an assembly part in the BOM part structure mode):



Picture 171: Query dialog for context assembly node

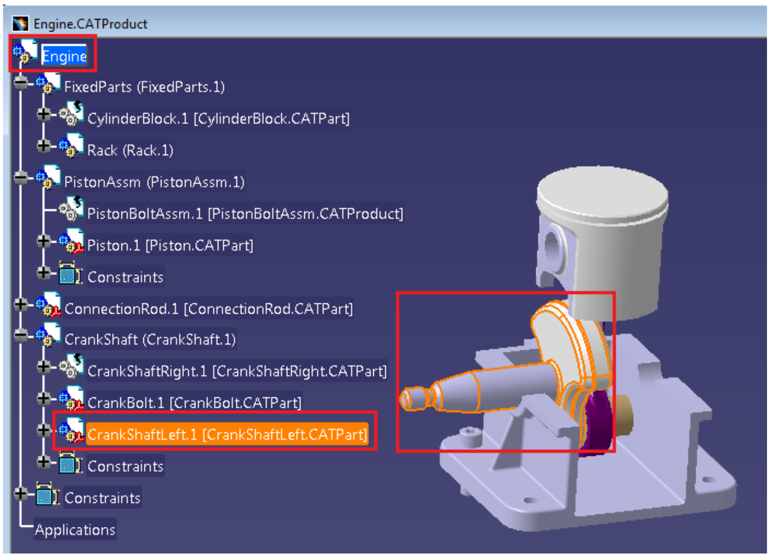
The structure, of which the selected CATProduct is the root document, has to contain the previously selected CATPart.

When the user double-clicks on the selected CATProduct a specific multi-level structure expand is performed which only returns the parts of the structure where the CATPart’s bounding boxes overlap with the bounding box of the previously selected CATPart. This is a sub-set of the complete structure:



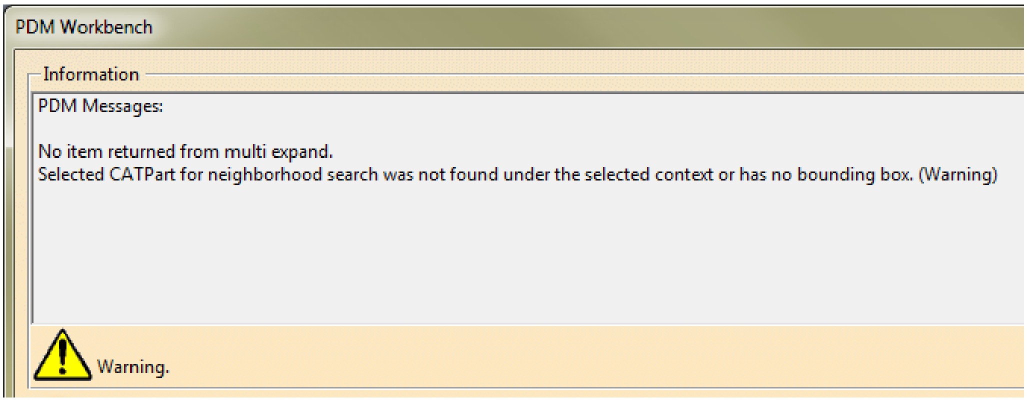
Picture 172: Reduced structure containing only neighbor models

When this structure is loaded to CATIA the user can see the geometry where the bounding boxes overlap with the originally selected CATPart's bounding box:



Picture 173: Reduced structure loaded to CATIA

If the selected structure does not contain the selected CATPart the user will receive a warning message:

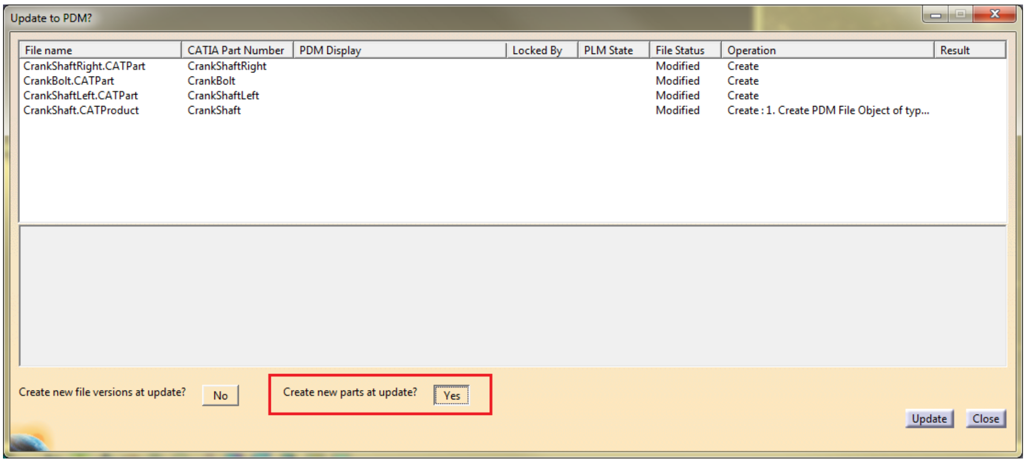


Picture 174: The selected structure does not contain the selected CATPart

Automatic Part Creation in CAD Structure Mode

It is possible to automatically create BOM part items when a new CAD document is created.

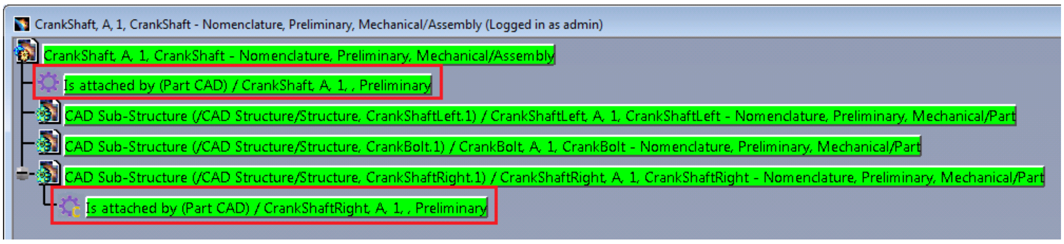
The user can define whether for new CATIA files, where new CAD document items will be created in the Update process, new BOM part items should also be created:



Picture 175: “Create new parts at Update” check box

After the update process has completed the part items are created in PDM, and the corresponding CAD documents are related to the parts with the “Part CAD” relation.

Expanding the Part CAD relation shows the part items in the PDM structure window:

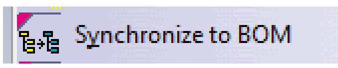


Picture 176: CAD structure with related Part items

Synchronize CAD structure to BOM

It is possible to perform a synchronization of the CAD structure information to the corresponding Part BOM structure if every CAD document in the structure has a corresponding Part item (see “Automatic Part Creation in CAD Structure Mode”).

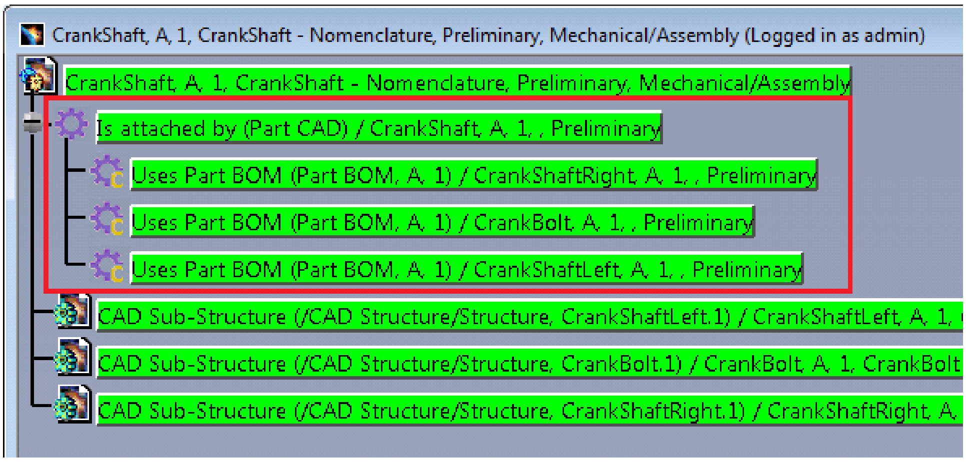
The user clicks on the “Synchronize to BOM” context menu on the CATProduct document:



Picture 177: “Synchronize to BOM” context action

If all CAD documents in the structure are related to a part, and if all the parts are locked by the user, then the CAD structure instance information (instance name, instance description, and transformation matrix) is applied to the part structure.

The resulting expand structure can be expanded in the PDM structure window:



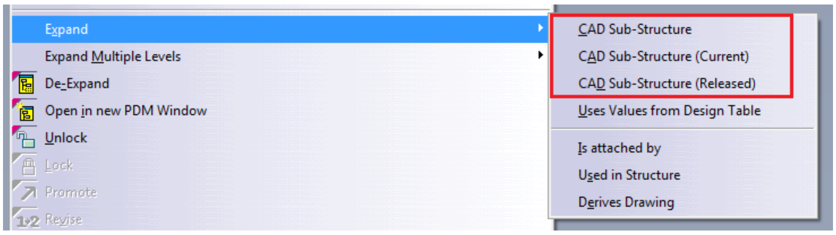
Picture 178: Created or updated part structure

"Current" and "Released" Expand Modes for "CAD Structure"

In addition to the default expand mode for the CAD structure ("As Saved") the modes "Current" and "Released" are supported.

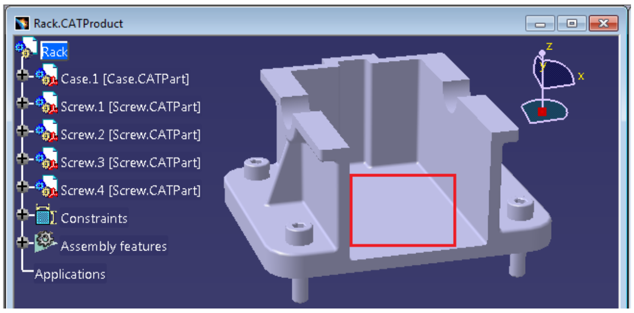
Two new CAD structure expand modes are available, Current and Released.

The existing default mode is the AsSaved mode.



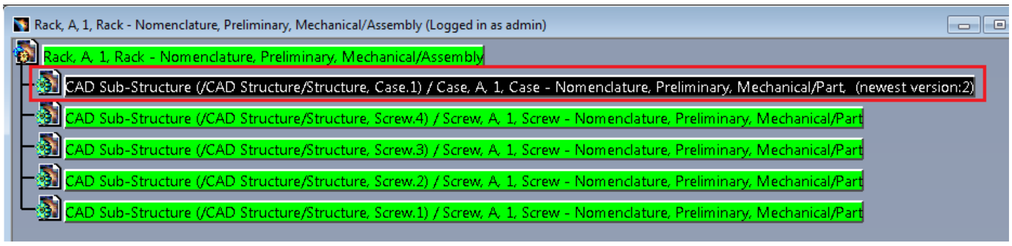
Picture 179: Three CAD Structure expand modes

In an example, a CATPart in a structure exists in two generations, generation 1 and generation 2:



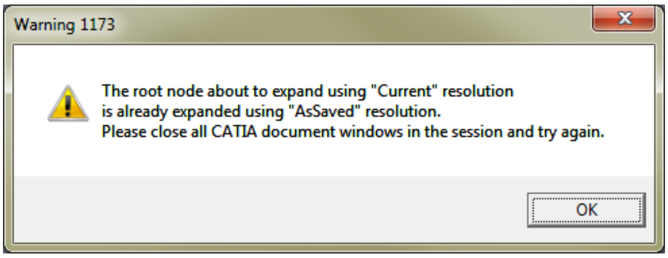
Picture 180: Generation 1 of CATPart

The structure in PDM uses the generation 1 of the CAD document:



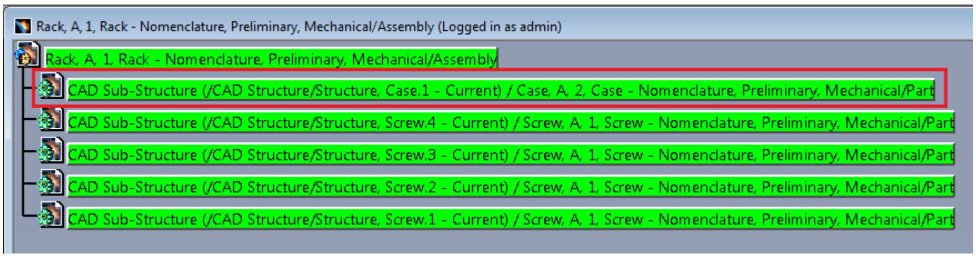
Picture 181: CAD structure containing generation 1 of CATPart

When a CATIA structure is loaded in one expand resolution it is not possible to expand the structure using a different expand mode. If the user attempts that he gets a warning:



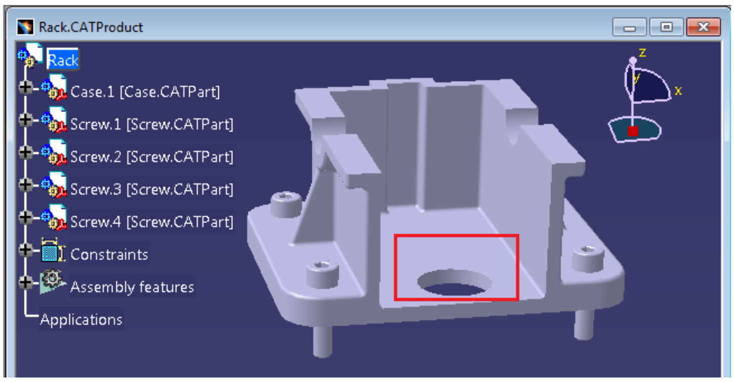
Picture 182: Warning about different expand resolution

When all CATPart and CATProduct windows are closed it is possible to expand the structure using a different expand resolution, for instance “Current”:



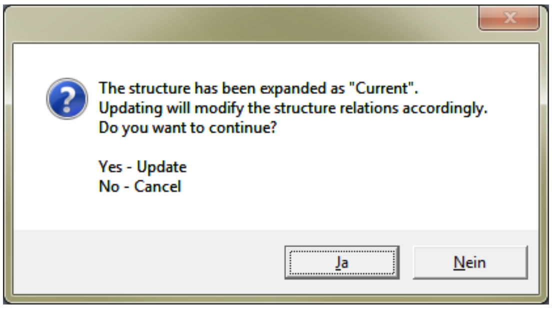
Picture 183: CAD structure expanded as “Current”

The “Current” structure, which contains the latest versions of all CAD documents, can be loaded into the CATIA session:



Picture 184: CATIA structure containing the latest generations of the CATIA documents

When the user updates a CATIA structure that was not expanded as saved a confirmation dialog appears. If the user continues the update all the CAD structure relations in the loaded structure will be updated to the current generations of the documents.



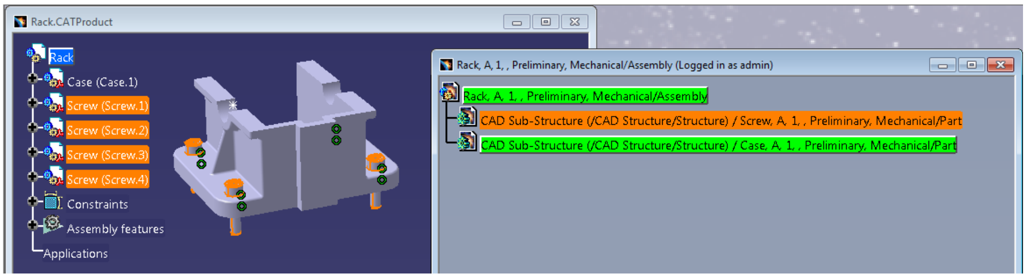
Picture 185: Confirmation dialog at Update

Support for the new CAD structure instance handling introduced in Innovator 9.4 and 10.0

A new relation with the name “CAD Instance” has been introduced, which contains instance information for “CAD Structure” relations.

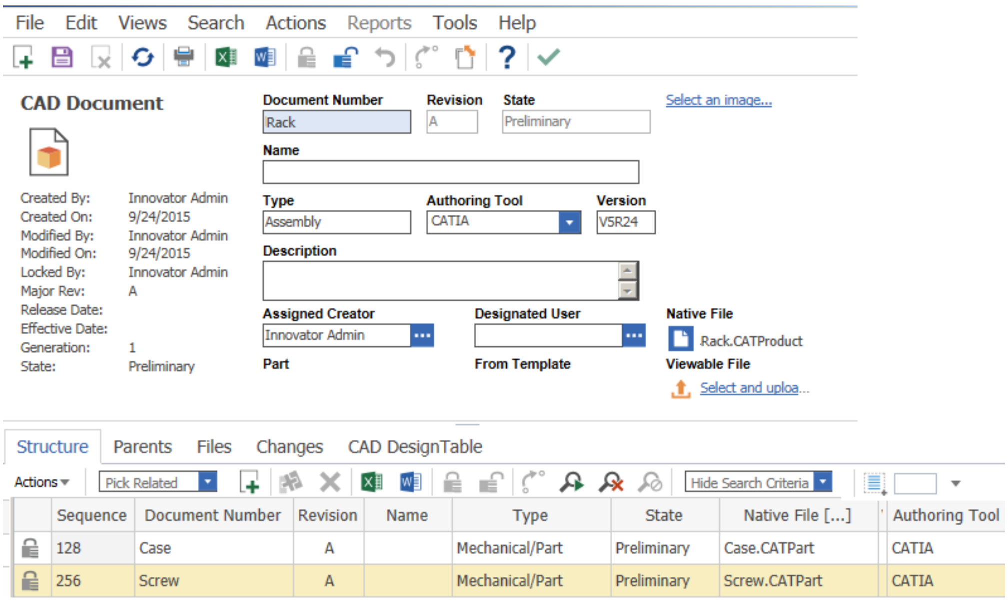
One visible difference in the usage is that the CAD Structure relations become multi-quantity relations, that is, there is only one relation for any number instances on a CATProduct.

Here is an example with four instances of a child part:



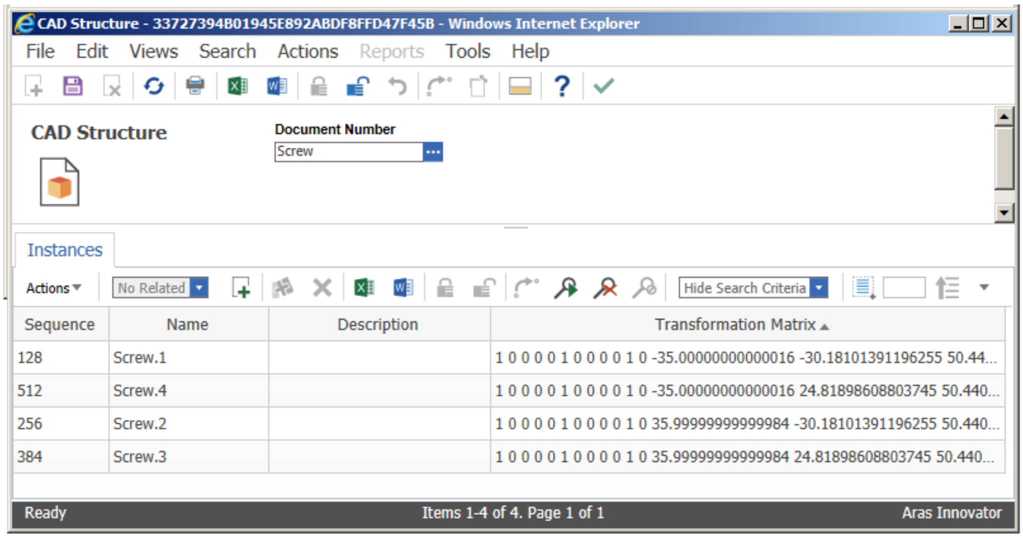
Picture 186: Structure with four instances

The four instances are stored in one CAD Structure relation which contains four CAD Instance relations:



Picture 187: One CAD Structure relation for each used CAD document

The CATIA instance information is stored in the CAD Instance relations:

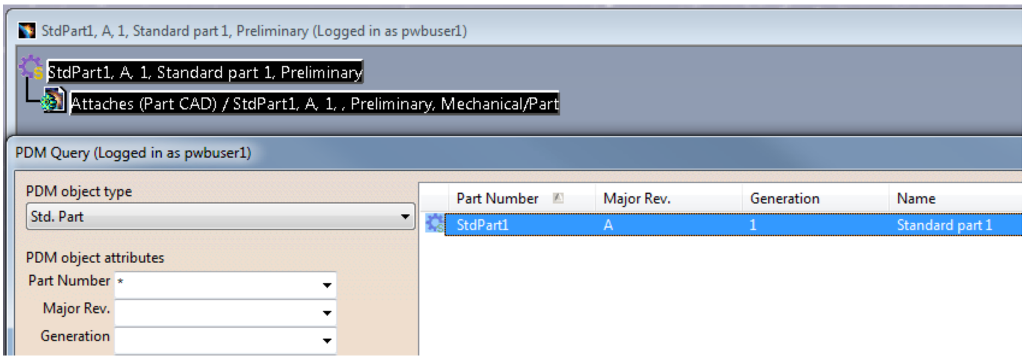


Picture 188: CAD Instance information

Standard Part Functionality

In part structure mode, it is possible to define part items and their corresponding CAD document items as standard parts. Standard parts are supposed to be parts which are used in a wide variety of different contexts and which are generally not modified by the designer, only used in the product structures that the designer works on.

The user can query for a standard part explicitly. Please note that regular users can not lock and modify standard parts, they can only use them in their structures:



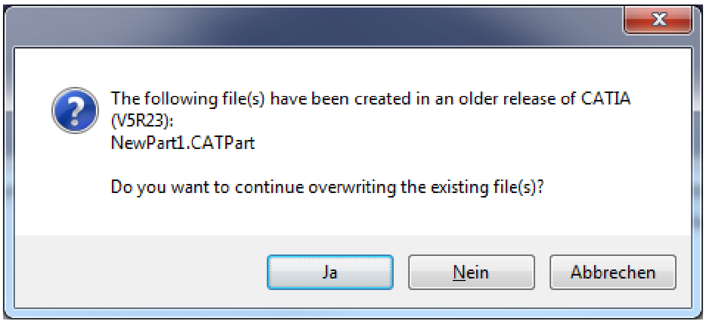
Picture 189: Querying for a standard part

Standard parts can be used like regular parts. The exceptions are that regular users can not create or update standard parts, and it is possible to import CATProduct structures which contain standard parts which are already defined in PDM. In that case the existing standard part items are used for that structure.

Check for CAD document CATIA release at PDM update

A new functionality optionally asks the user before overwriting a file which has been created with a lower release of CATIA V5.

If the user is about to overwrite a file which has been created with a lower release of CATIA V5 he is asked whether he wants to continue:

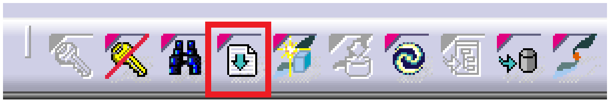


Picture 190: Asking the user whether to continue the update process

Local Workspace Information

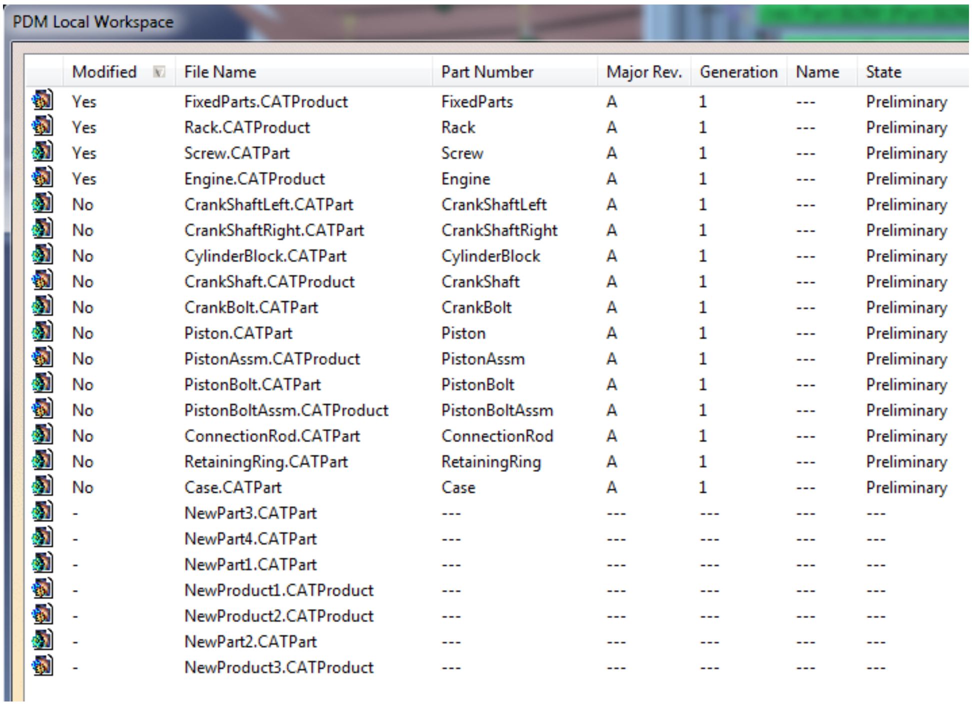
It is possible to check the status of the CATIA documents which are downloaded to the local working directory (PWB_XMAP). A list displays the local files and information about their corresponding CAD documents in PDM if they exist.

When the user clicks on the "Local Workspace" icon a window containing a list of CATIA files appears:



Picture 191: "Local Workspace" icon

➔



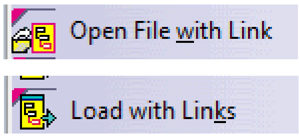
Modified	File Name	Part Number	Major Rev.	Generation	Name	State
Yes	FixedParts.CATProduct	FixedParts	A	1	---	Preliminary
Yes	Rack.CATProduct	Rack	A	1	---	Preliminary
Yes	Screw.CATPart	Screw	A	1	---	Preliminary
Yes	Engine.CATProduct	Engine	A	1	---	Preliminary
No	CrankShaftLeft.CATPart	CrankShaftLeft	A	1	---	Preliminary
No	CrankShaftRight.CATPart	CrankShaftRight	A	1	---	Preliminary
No	CylinderBlock.CATPart	CylinderBlock	A	1	---	Preliminary
No	CrankShaft.CATProduct	CrankShaft	A	1	---	Preliminary
No	CrankBolt.CATPart	CrankBolt	A	1	---	Preliminary
No	Piston.CATPart	Piston	A	1	---	Preliminary
No	PistonAssm.CATProduct	PistonAssm	A	1	---	Preliminary
No	PistonBolt.CATPart	PistonBolt	A	1	---	Preliminary
No	PistonBoltAssm.CATProduct	PistonBoltAssm	A	1	---	Preliminary
No	ConnectionRod.CATPart	ConnectionRod	A	1	---	Preliminary
No	RetainingRing.CATPart	RetainingRing	A	1	---	Preliminary
No	Case.CATPart	Case	A	1	---	Preliminary
-	NewPart3.CATPart	---	---	---	---	---
-	NewPart4.CATPart	---	---	---	---	---
-	NewPart1.CATPart	---	---	---	---	---
-	NewProduct1.CATProduct	---	---	---	---	---
-	NewProduct2.CATProduct	---	---	---	---	---
-	NewPart2.CATPart	---	---	---	---	---
-	NewProduct3.CATProduct	---	---	---	---	---

Picture 192: "Local Workspace" window

Optional Load of linked CATPart Files

In the CAD structure mode it is possible to load a single CATPart file or a CATProduct structure with the CATPart files which are linked with the ‘CAD Structure/Reference’ relation (CATIA multi-model links).

Clicking on “Load with Links” for CATProduct documents or on “Open File with Link” for CATPart documents downloads and opens the selected files, plus the CATPart files which are related by the ‘CAD Structure / Reference’ links in PDM.

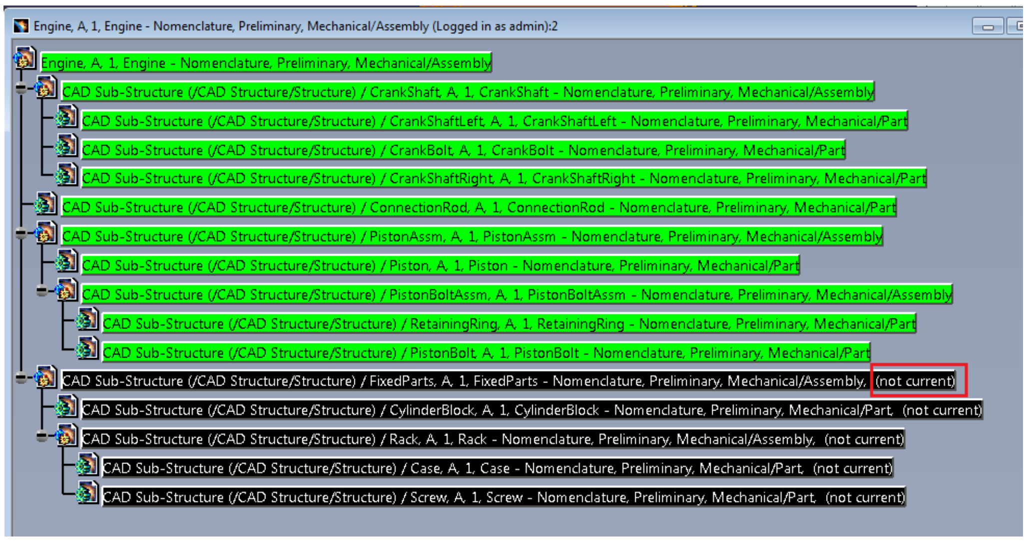


Picture 193: “Open / Load files with links” context menu action

Newest Version Info Context Menu

For performance reasons not all version data is retrieved for CAD documents in a structure which are not current. To retrieve additional information (which the newest version is, and which user has locked the newest version) a new context menu action has been added.

In the following example a sub-structure of a CAD document tree is not current. By default the information on the PDM tree node is “not current”.

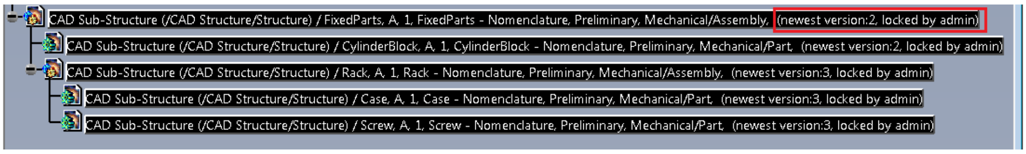


Picture 194: Default information “not current”

Click on the “Newest Version Info” context menu …

Picture 195: “Newest Version Info” context menu action

… retrieves additional information about the latest version of the documents.

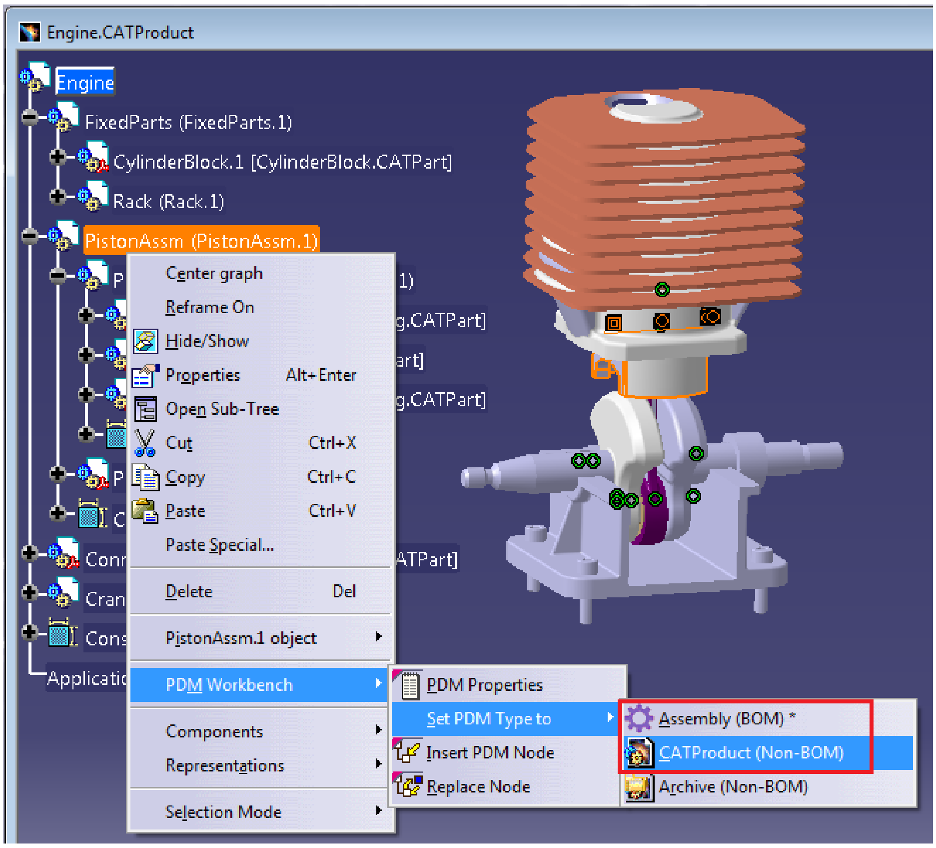


Picture 196: Additional version information

Non-BOM CATParts and CATProducts

In the part structure mode it is now possible to define CATParts and CATProducts in the CATIA structure to be defined as not BOM-relevant. In this case no corresponding part items will be created in PDM.

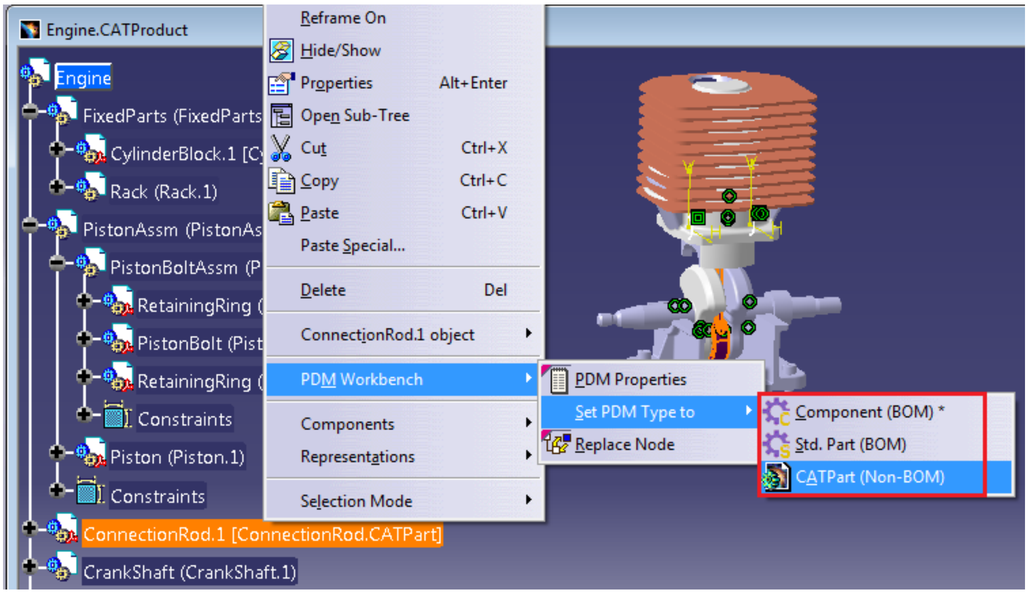
Before a CATProduct structure is created in PDM it is possible to change the wanted PDM type from a part type like Assembly or Component to the CATIA file type:



Picture 197: Setting a CATProduct to the non-BOM type

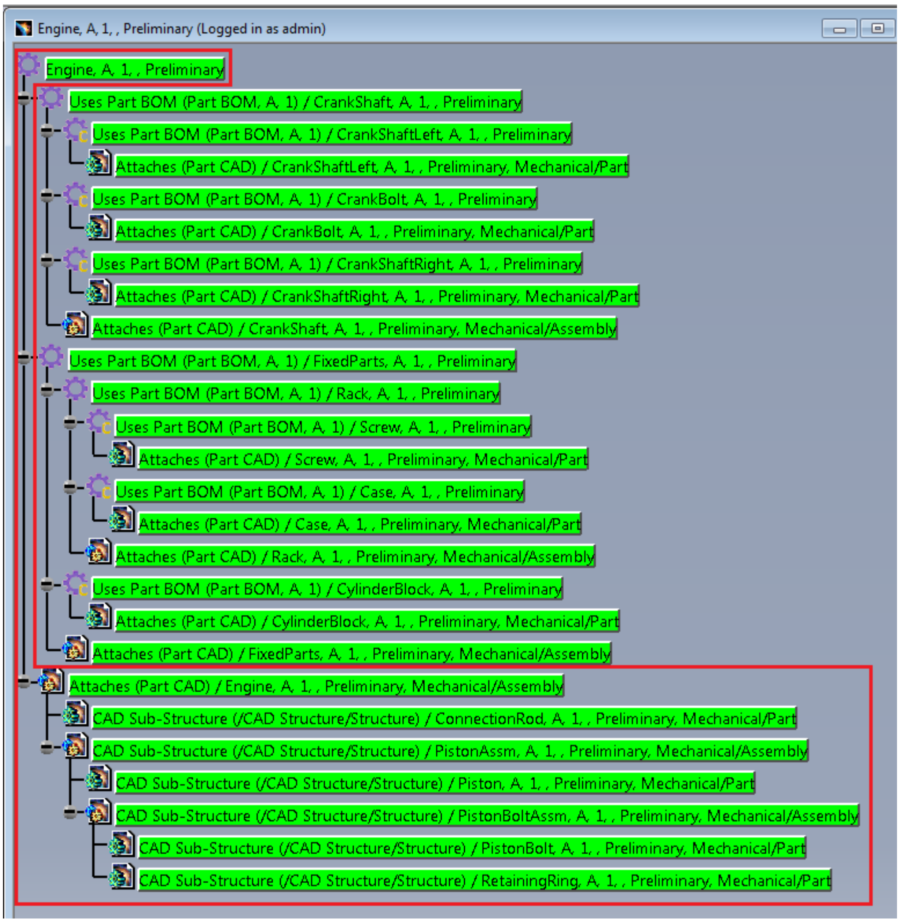
In that case all sub-nodes of the CATProduct also become non-BOM.

CATParts also can be changed to the non-BOM type:



Picture 198: Setting a CATPart to the non-BOM type

The result is a structure in PDM which contains both part structures and CATIA document structures:



Picture 199: Resulting PDM Structure

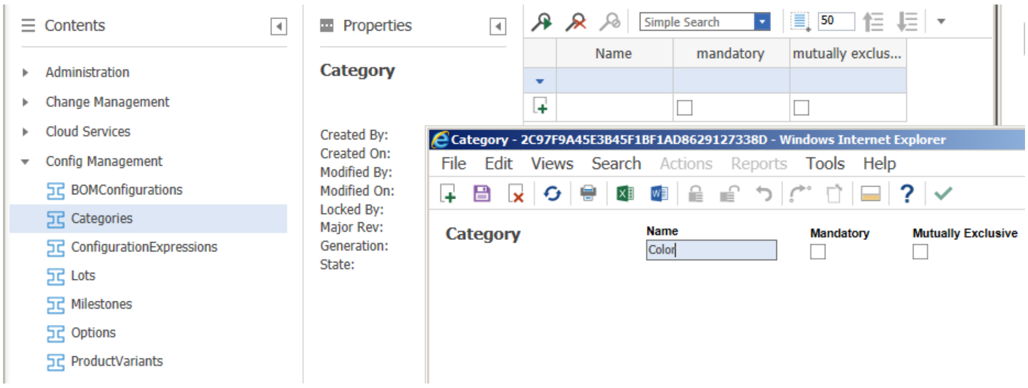
When nodes are added to or removed from the non-BOM CAD document structure "Update" synchronizes the changes in the CATProduct structure to the non-BOM CAD structure, just like to the BOM part structure.

Configuration of BOM Part Structure

In the BOM part structure mode it is possible to create product configurations where, depending of the currently set configuration context, only a sub-set of the product structure is expanded and loaded. With this functionality it is possible to create and to work on different configurations of the same product.

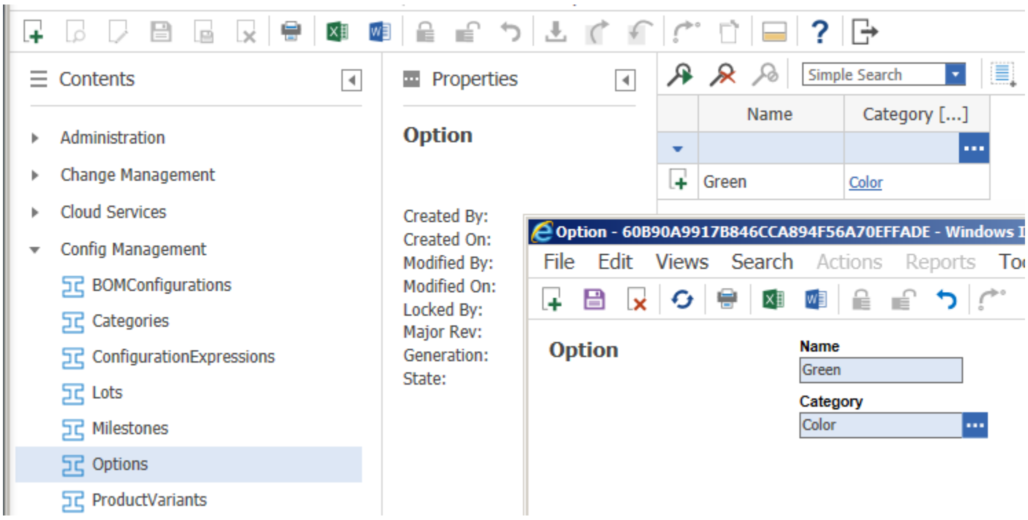
This is a small example of the configuration functionality which shows the configuration management with options:

First a category, in this example named “Color”, has to be created.



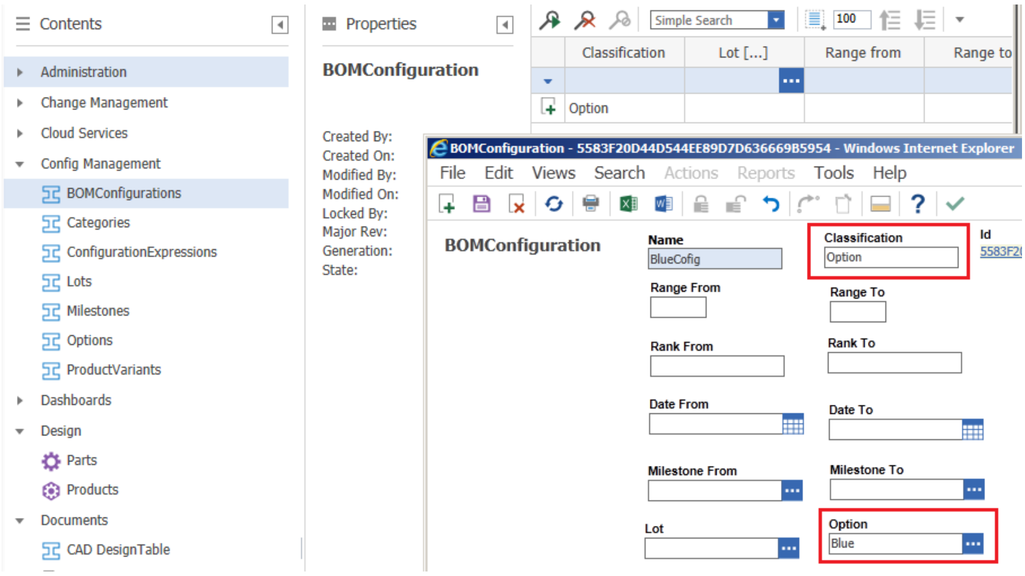
Picture 200: Creating a new category “Color”

Then option items which refer to the category “Color” are created, in this case named “Blue”, “Green”, and “Yellow”.



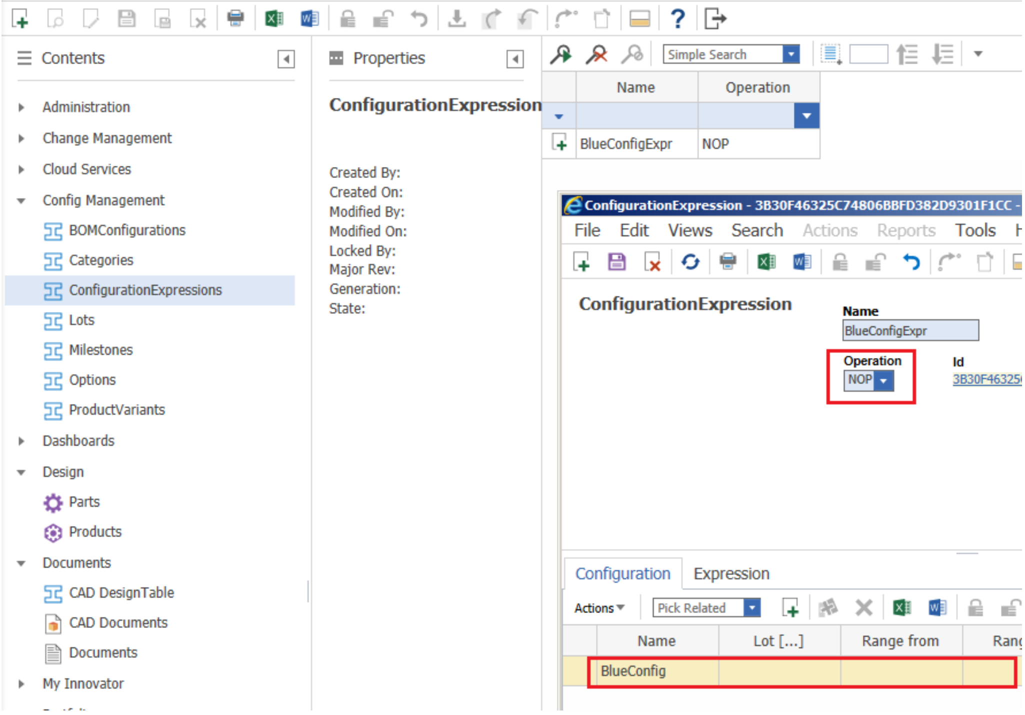
Picture 201: Creating the options “Blue”, “Green”, and “Yellow”

Then BOMConfiguration items are created which refer to these color options. The names are “BlueConfig”, “GreenConfig”, and “YellowConfig”.



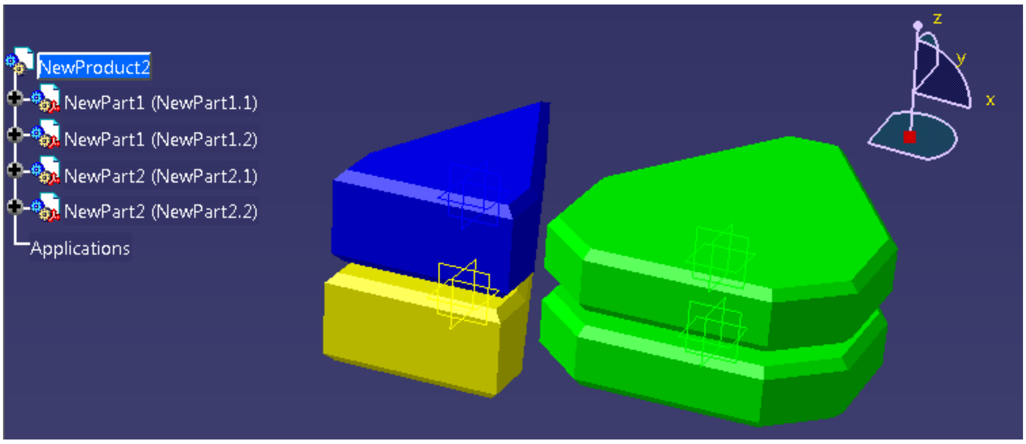
Picture 202: Creating BOMConfiguration items

The next step is to create configuration expressions (“BlueConfigExpr”, “GreenConfigExpr”, “YellowConfigExpr”). Configuration expressions can be combined using the logical operators AND, OR, and NOT.



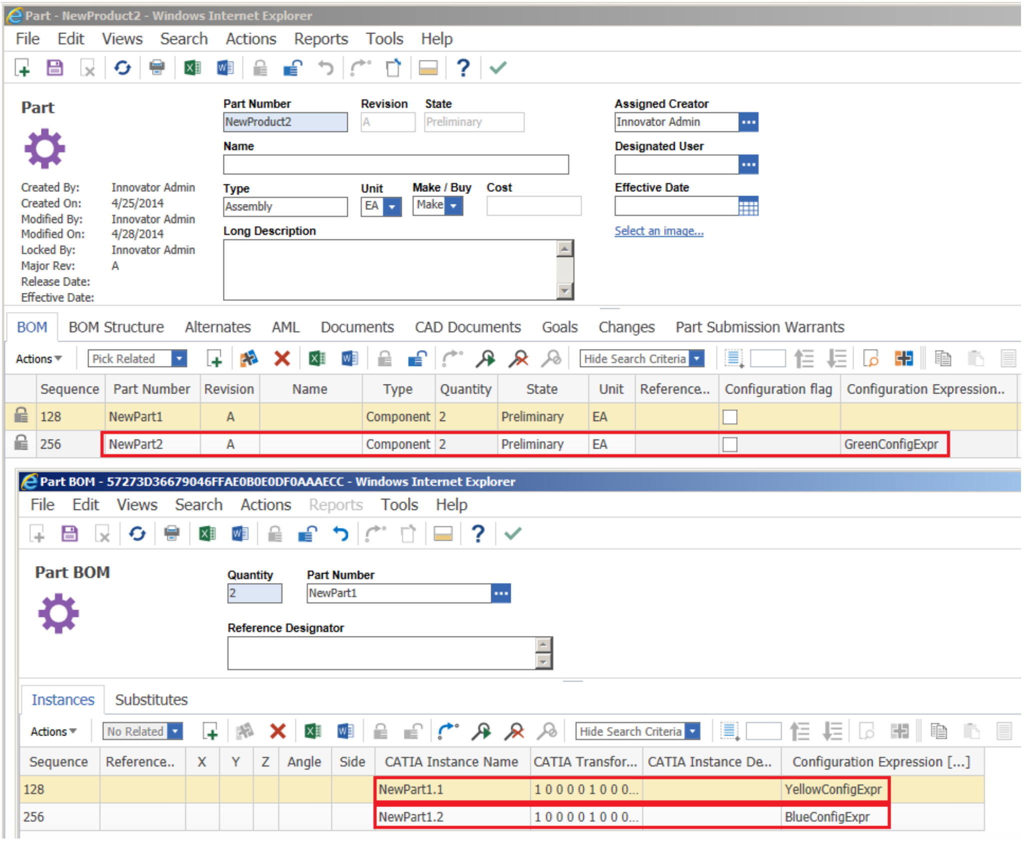
Picture 203: Creating Configuration Expression items

Then a sample CATIA product structure is imported, creating a part BOM structure in Innovator.



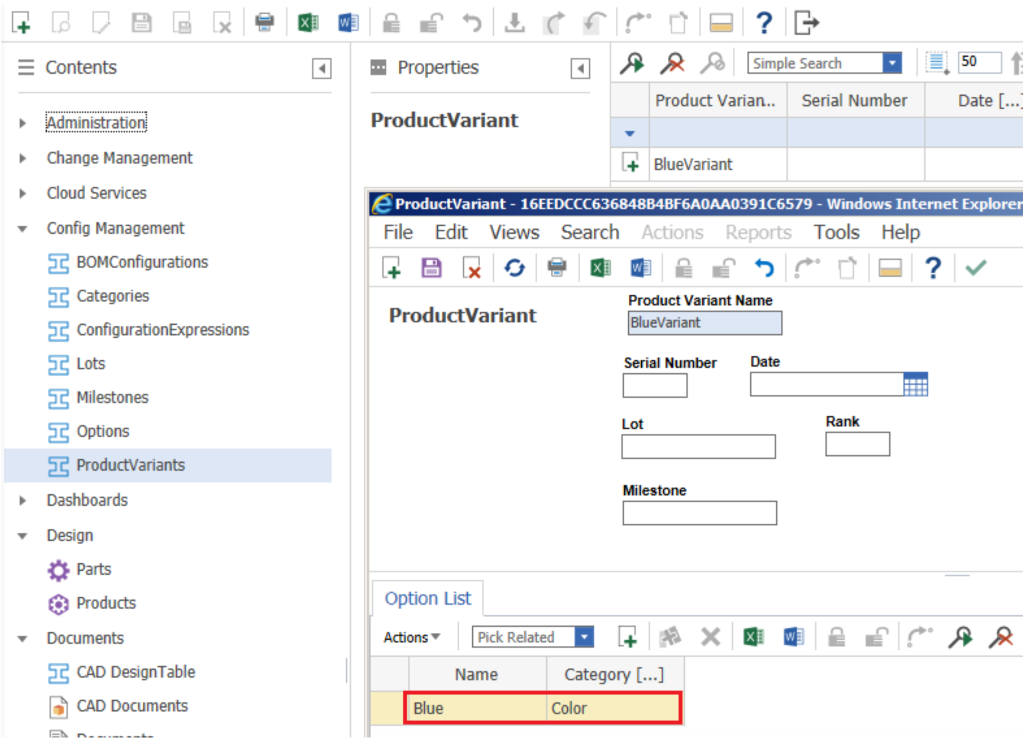
Picture 204: Sample CATIA product structure

The previously created ConfigurationExpression items can be related to either "Part BOM" or to "BOM Instance" relation items.



Picture 205: Relating configuration expressions to PLM relations

In order to be able to set the configuration context ProductVariant items have to be created ("BlueVariant", "GreenVariant", and "YellowVariant").

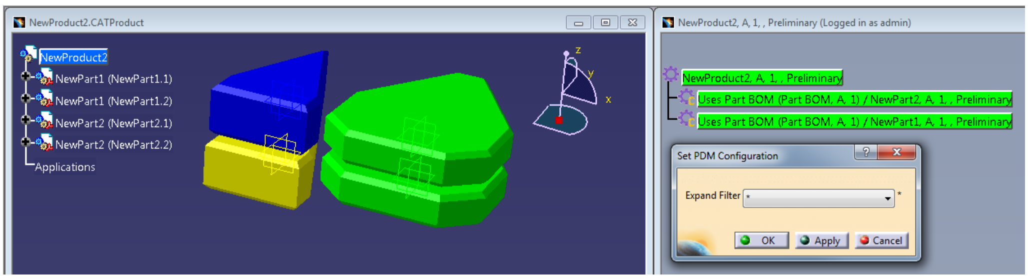


Picture 206: Creating Product Variant items

Now the previously created part structure can be expanded and loaded in different configurations.

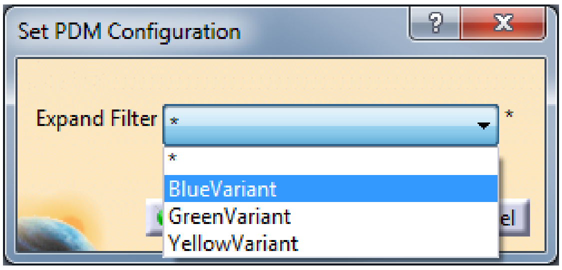
Picture 207: Setting a product variant for the part BOM expansion

First, if no configuration is set, the complete structure is expanded and loaded.

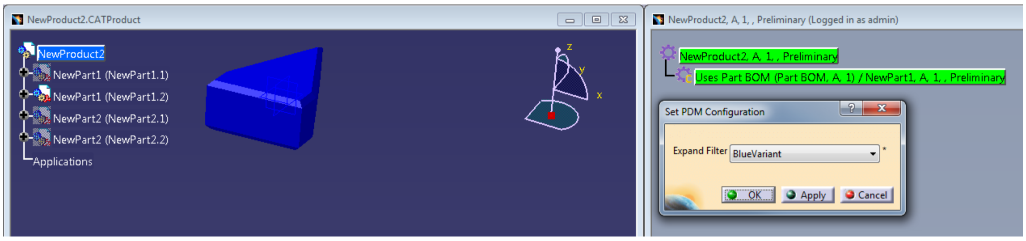


Picture 208: Expanding and loading the complete structure

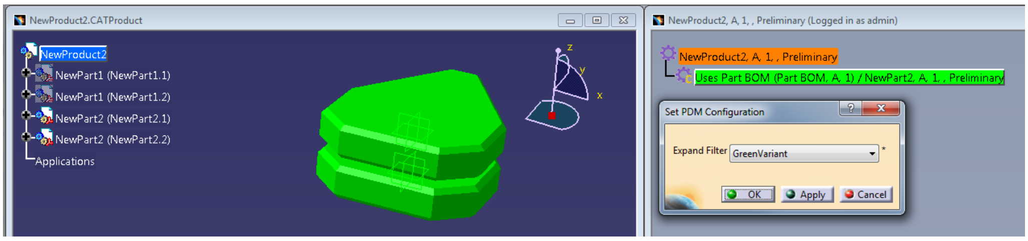
Then, if a particular product variant is set, expanded, and loaded, then only the configured parts are expanded and loaded.



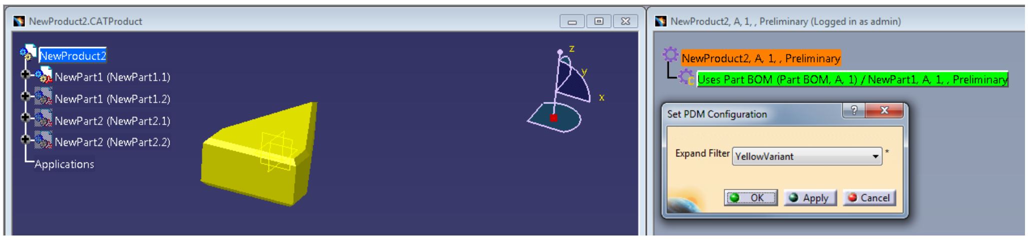
Picture 209: Setting different product variant expand filters



Picture 210: Loaded the "Blue" variant (one BOM Instance)



Picture 211: Loaded the "Green" variant (one Part BOM with all instances)

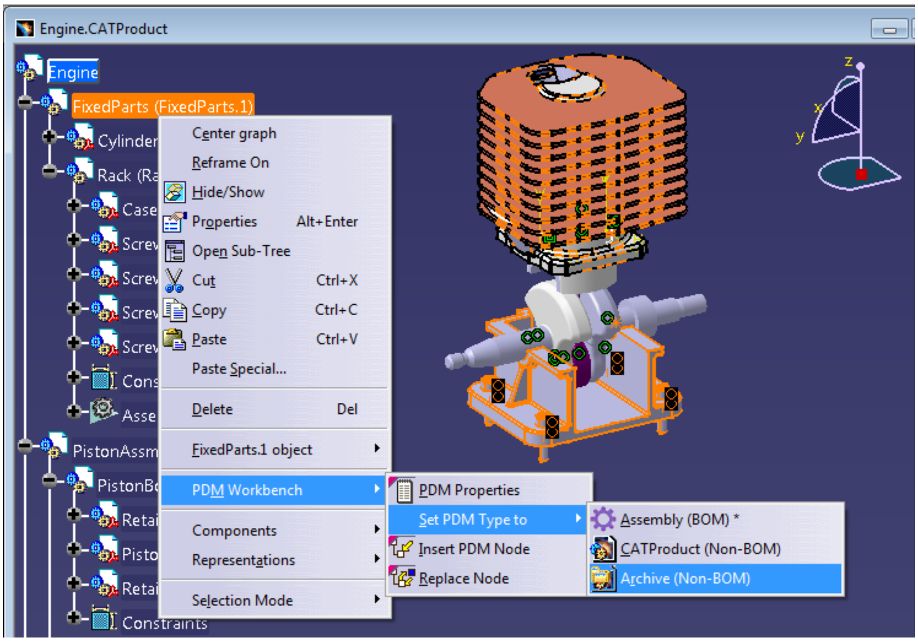


Picture 212: Loaded the "Yellow" variant (one BOM Instance)

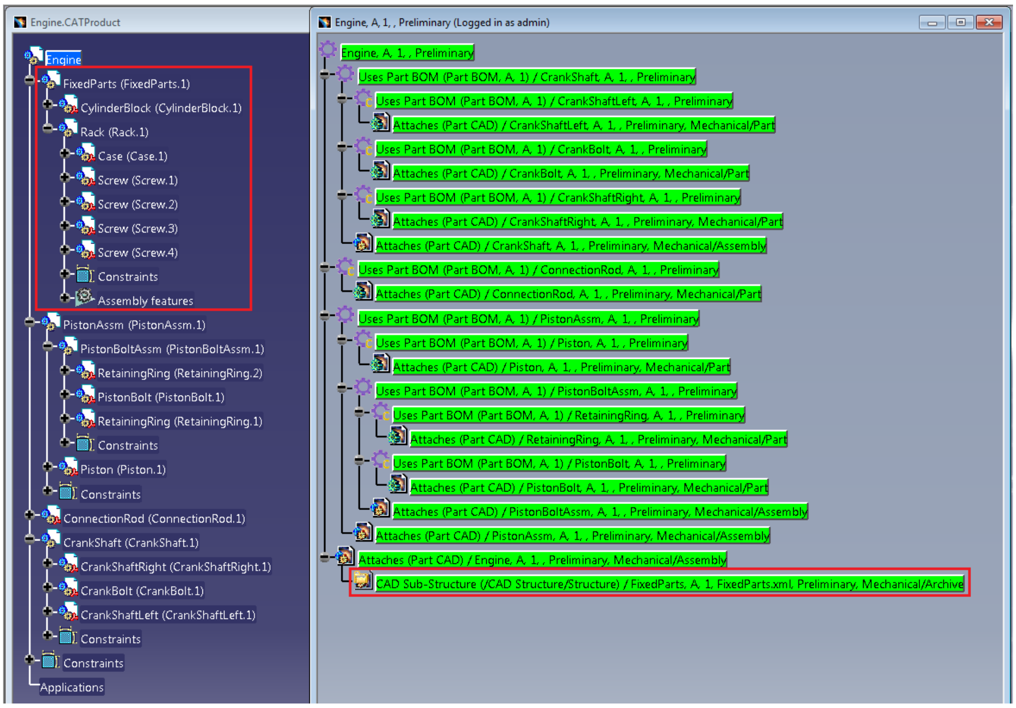
Archives

It is possible to compress a complete CATProduct sub-structure into one Zip file and to manage this compressed file in PDM. This makes it possible to hide a complicated CATProduct structure in one CAD document if it is not necessary to manage the structure information in PDM.

Any CATProduct sub-structure which has not been created in PDM can be defined as an archive. If this is done the subsequent Update process compresses this CATProduct structure into one single ZIP file and manages this ZIP file as a CAD document in PDM instead of the normal CATProduct structure.



Picture 213: Defining a CATProduct structure as an archive



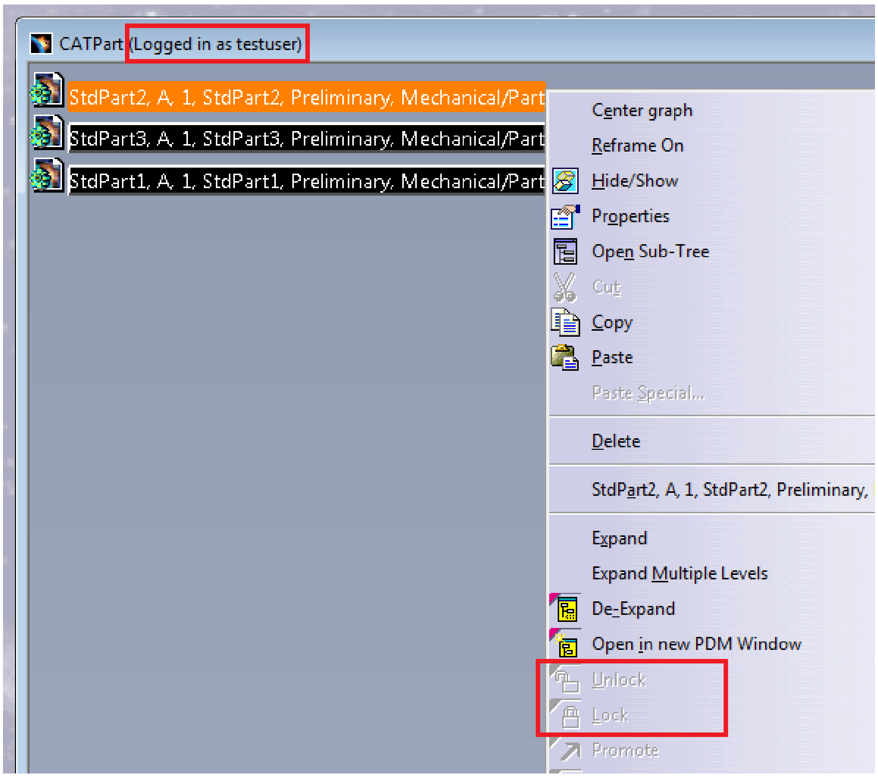
Picture 214: The resulting archive CAD document in PDM

Standard Part functionality for CAD structure mode

The standard part functionality has been extended to work with CAD document structures.

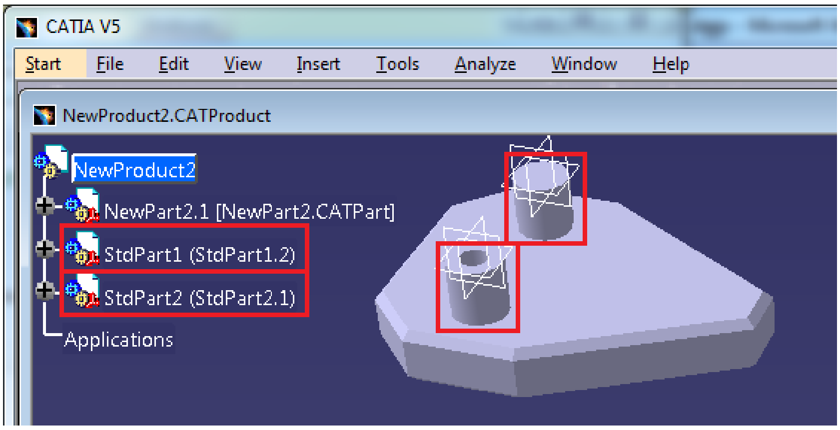
After the standard part CAD document items have been created any regular CAD user can query for them, by checking the “Standard Part” check box in the query dialog.

Regular users can not lock or otherwise modify standard part CAD documents:



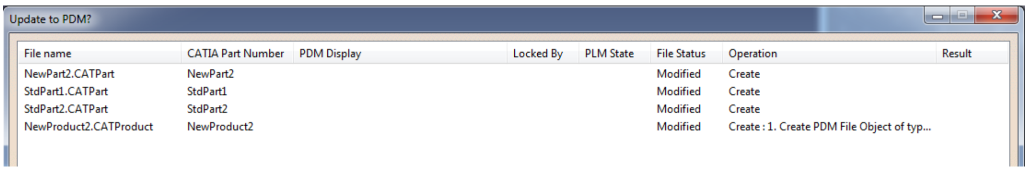
Picture 215: Using standard parts as a regular user

They can use standard parts in the CATIA structures that they work on:



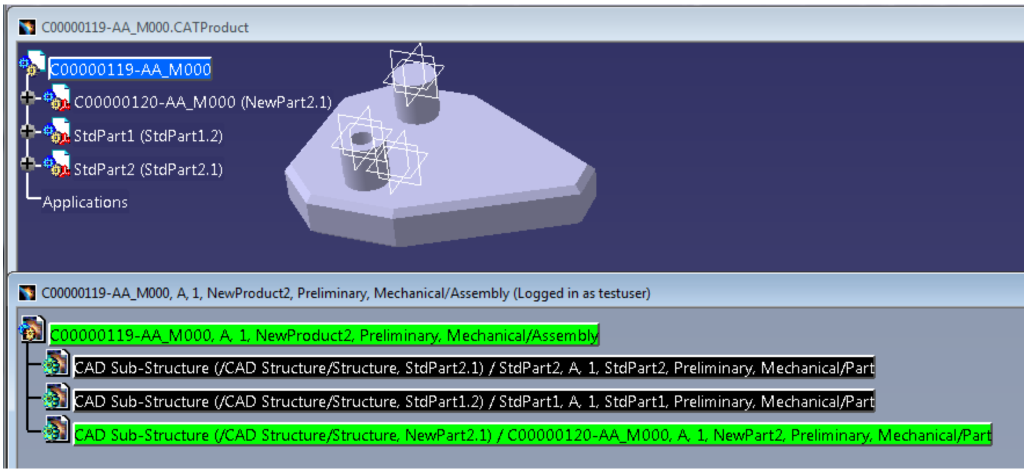
Picture 216: Using standard parts in CATIA structures

Adding standard parts to an existing structure at first does not seem different from adding other CATPart nodes …



Picture 217: Update dialog with standard parts

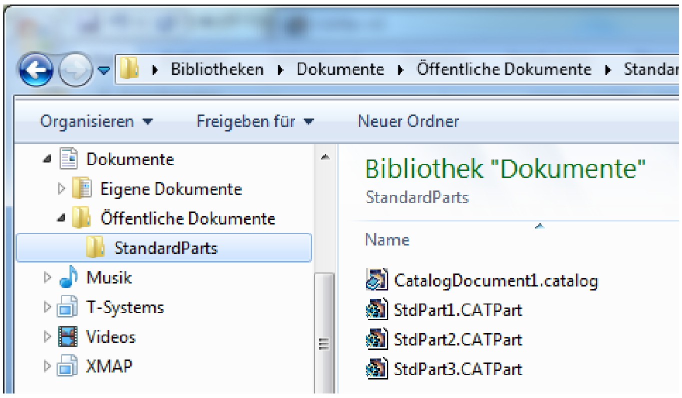
… but the standard parts are not created by the update process, but the existing ones, which have been created by the standard part administrator, are used:



Picture 218: Existing standard parts being used in a new structure

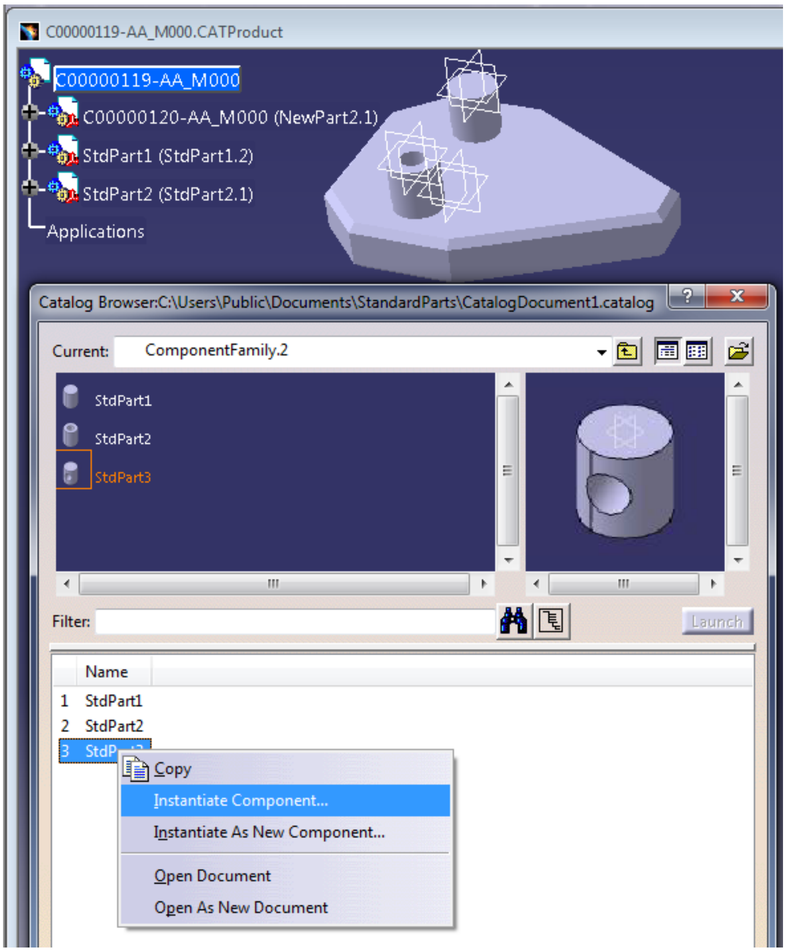
Standard part CATParts can be inserted to a CATProduct structure manually or by the CATIA catalog functionality. For this the standard CATParts have to be added to a CATIA catalog file first.

The catalog file can reside in any client directory which is accessible to CATIA V5. It can be a network drive.



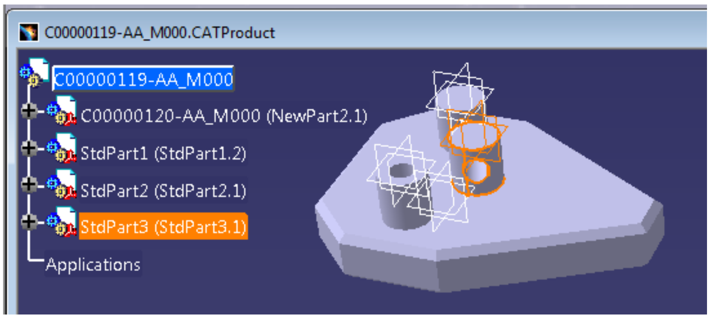
Picture 219: CATIA catalog containing standard part CATParts

Then the standard parts can be inserted to a CATProduct structure:



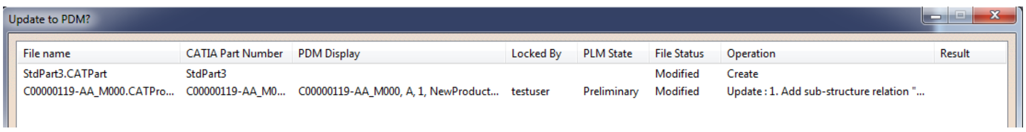
Picture 220: Standard part CATParts created from a catalog

➔



Picture 221: Inserted standard parts

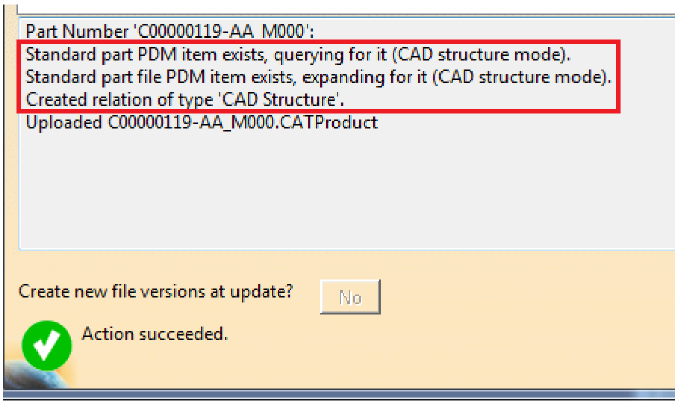
In the update process the standard part item from the database is taken.



File name	CATIA Part Number	PDM Display	Locked By	PLM State	File Status	Operation	Result
StdPart3.CATPart	StdPart3				Modified	Create	
C00000119-AA_M000.CATPro...	C00000119-AA_M0...	C00000119-AA_M000, A, 1, NewProduct...	testuser	Preliminary	Modified	Update : 1. Add sub-structure relation "...	

Picture 222: Update dialog with standard parts

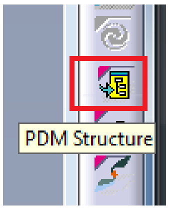
➔



Picture 223: Update result

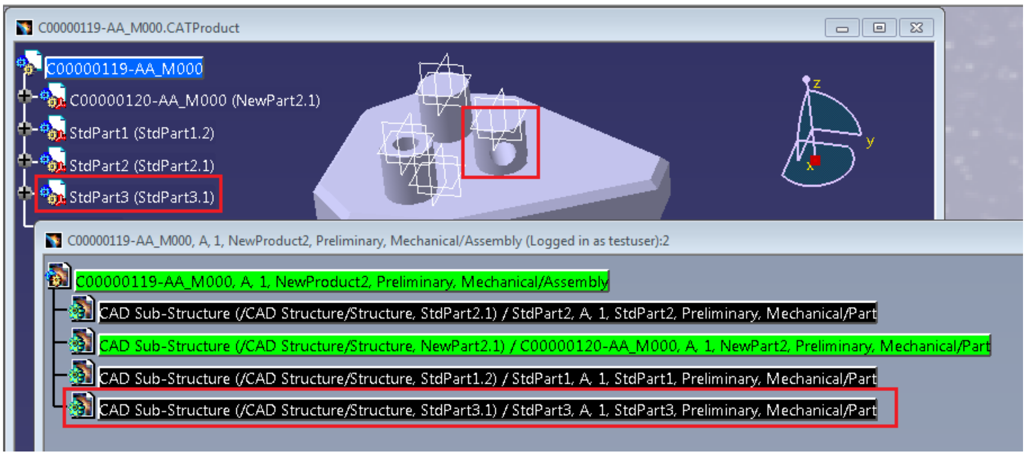
It is important to make sure that the standard CATPart files in the Innovator vault and in the local directory are exactly the same.

As with regular CATParts, the new standard CATPart node is added to the CAD document structure:



Picture 224: "Show PDM Structure" icon

➔

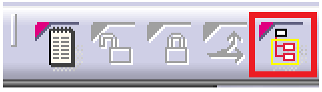


Picture 225: CAD Document structure containing standard parts

Check CAD Links

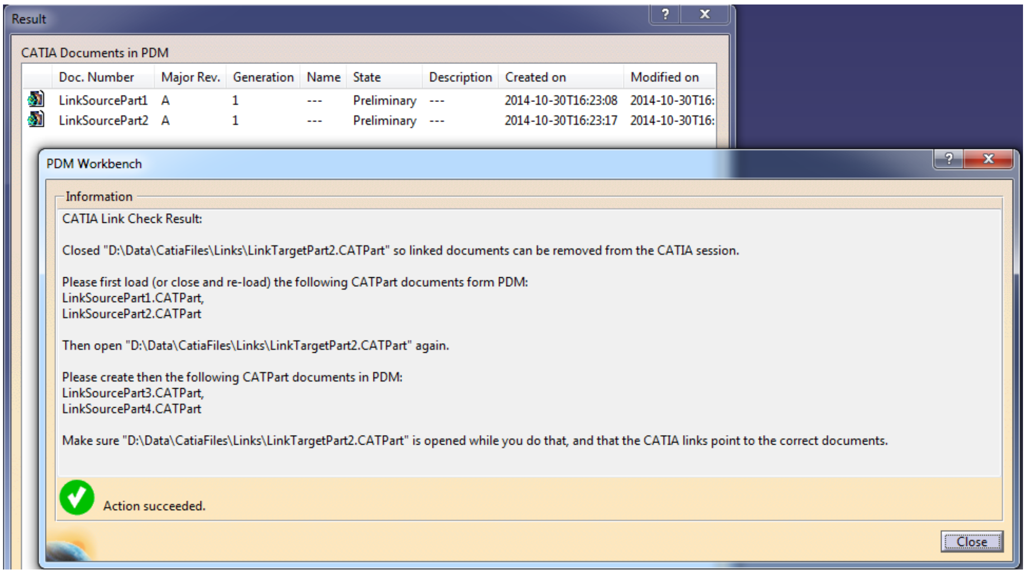
When CATIA documents with 3D links need to be imported this functionality helps the user to determine which documents have to be imported in which order, and which documents have to be in the CATIA session so the links are created correctly.

When a CATPart or a CATDrawing is the active CATIA document the user can click on the "Check CAD Links" icon to get information about which of the linked CATPart documents already exist in PDM, and which still have to be created:



Picture 226: “Check CAD Links” icon

In addition to this the functionality also opens a window containing all the CAD document items in PDM which should be opened in the CATIA session before the current CATIA document is imported to PDM.

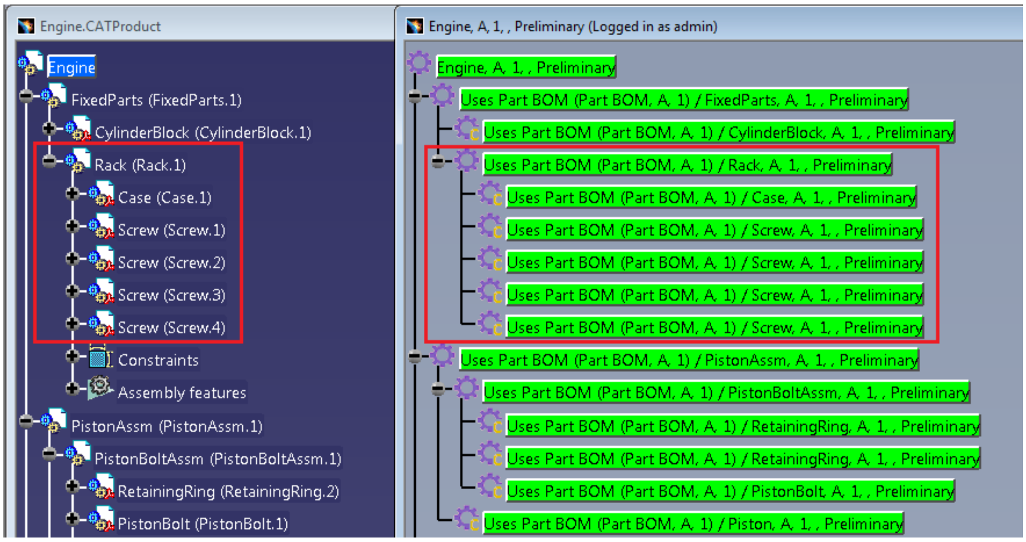


Picture 227: Result of “Check CAD Links” action

Displaying part structure instances as separate nodes

The display of the part structure in the PDM structure window can be changed such that every part instance is shown as a separate node.

When this functionality is switched on and a part structure containing several instances of the same part is expanded then all the instances are shown as separate nodes:



Picture 228: Part structure showing every instance as a separate node.

Saving PDM Session Information

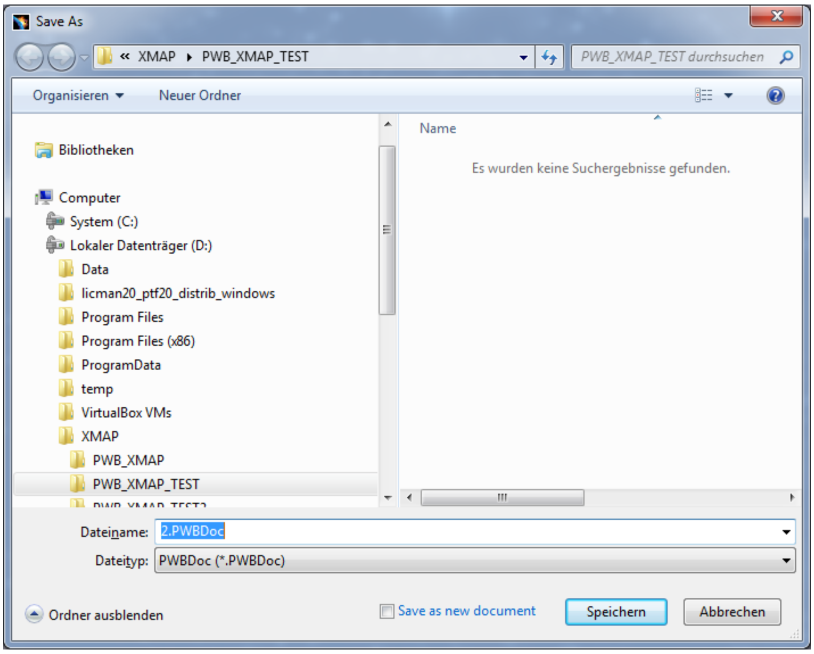
It is possible to save the content of a PDM structure window in a PWBDoc file, and to re-load the content of that window later from that file. The window created by opening the PWBDoc file has the same properties as a PDM structure window opened by querying and expanding PDM nodes, except that the content may be out of date with the actual server database for a longer time.

Usage

The content of any PDM structure window can be saved to a PWBDoc file:

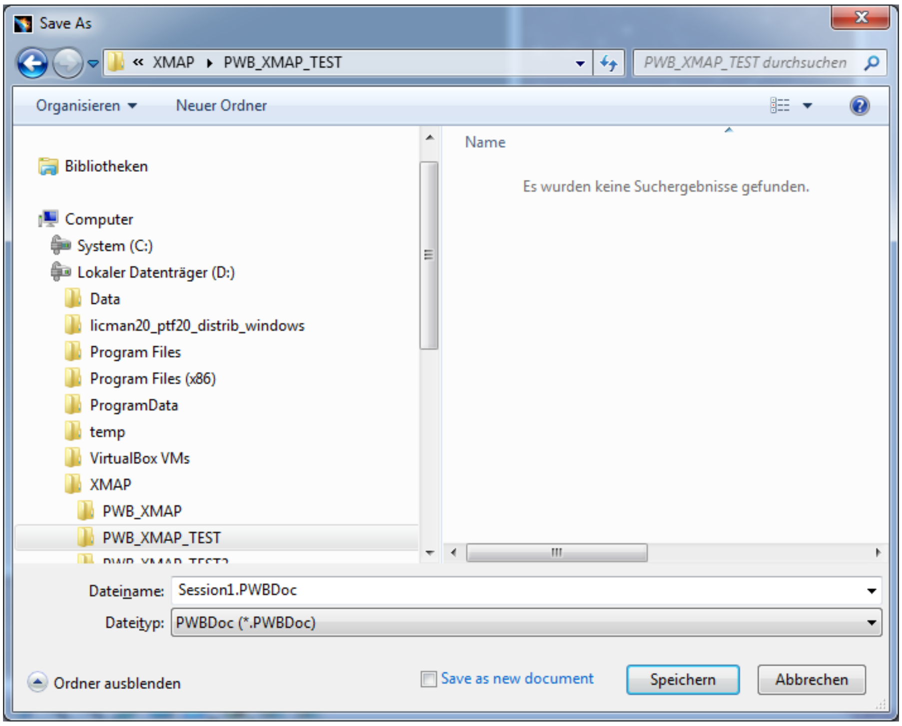
Picture 229: Example content of a PDM structure window

The content of this window can be saved by selecting “File / Save As” from the menu:



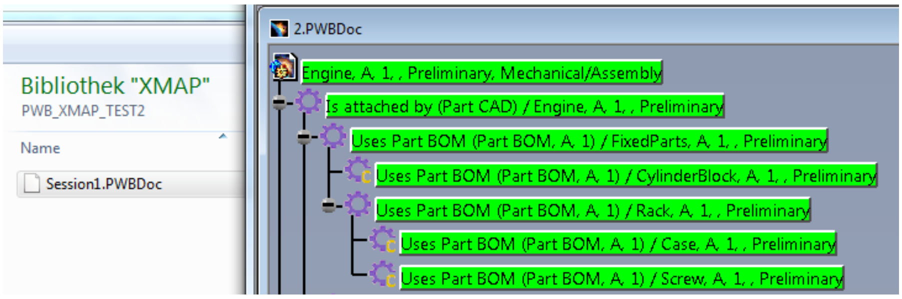
Picture 230: PWBDoc save dialog

➔



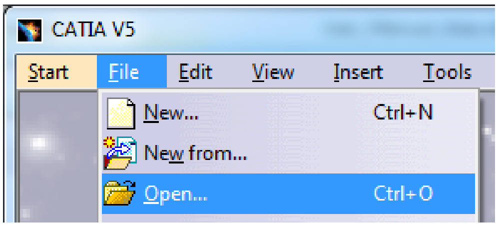
Picture 231: Saving the window content under a specific name

After saving, the new PWBDoc file can be seen in the Windows Explorer:



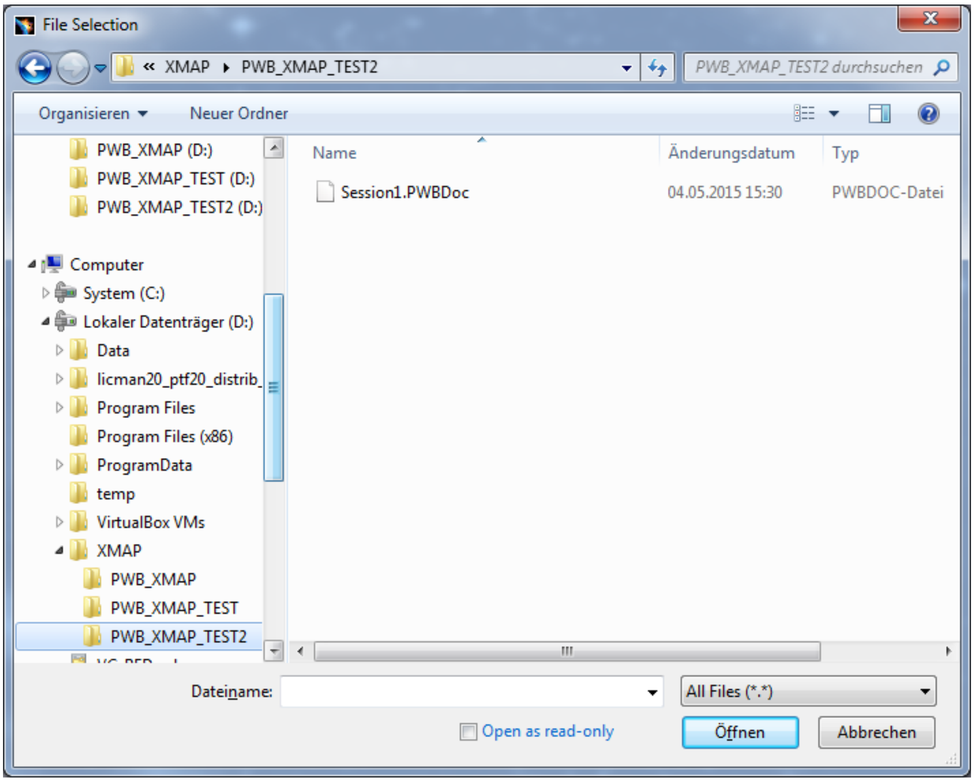
Picture 232: Newly created PWBDoc file

In the same session, or in a later session, this file can be opened again:



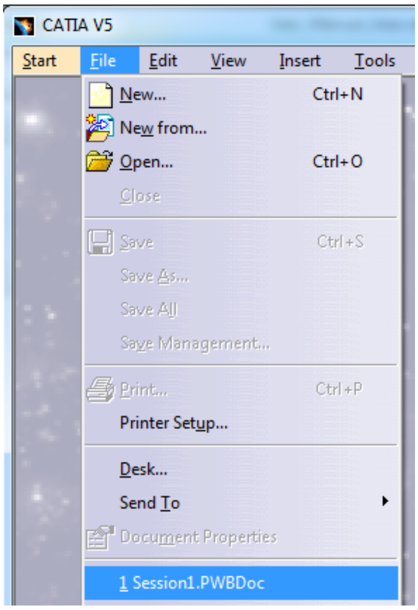
Picture 233: Opening a PWBDoc file (1/2)

➔



Picture 234: Opening a PWBDoc file (2/2)

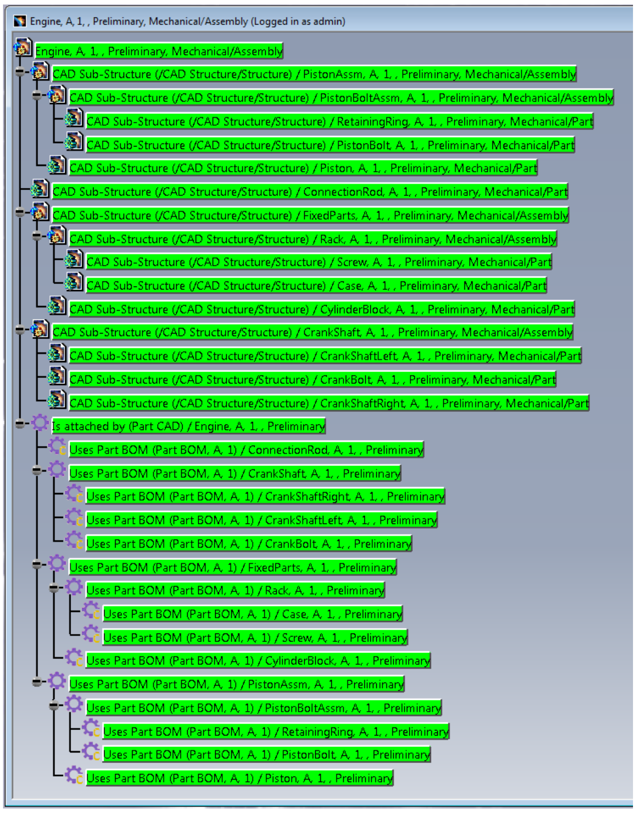
The file can also directly be opened from the most recently file list:



Picture 235: Opening a PWBDoc file from the most recently used file list

The user has to be logged on to save or to load a PWBDoc file.

After the file is opened the PDM structure window can be used like any other opened PDM structure window.

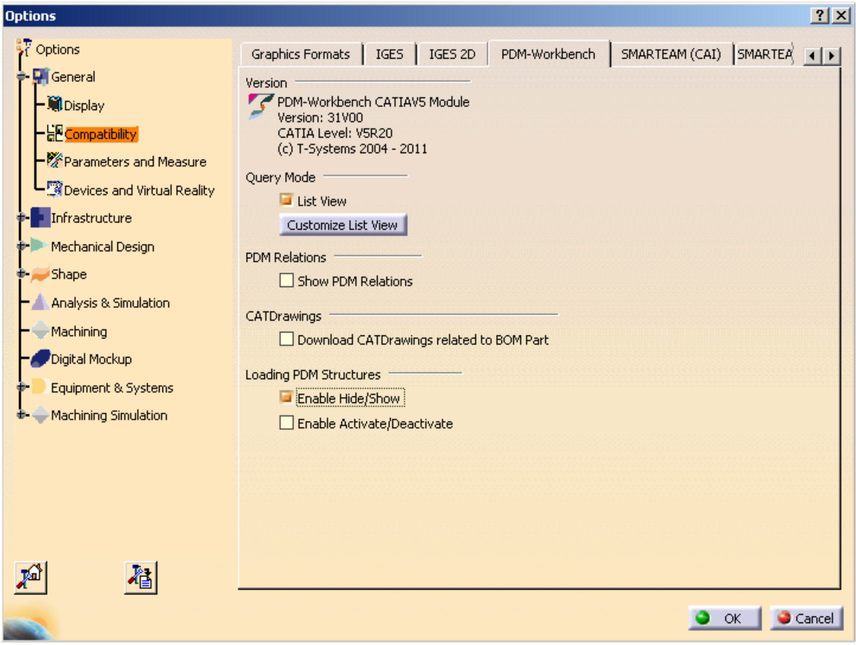


Picture 236: PDM structure window opened from PWBDoc file

Options

Once you are logged in into the PDM Workbench you can set some options for the PDM Workbench.

You open the options dialog with *Tools→Options* in CATIA V5. In the slider "PDM Workbench" you can set the options for the PDM Workbench (see *Picture 237: The PDM Workbench options*).



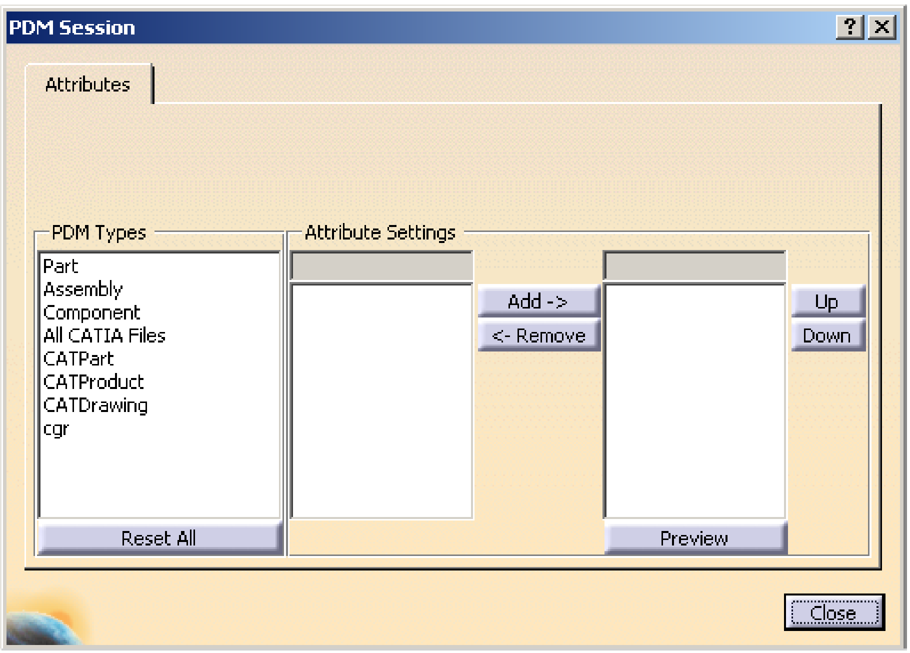
Picture 237: The PDM Workbench options

Query Mode

When you set "List View" for the Query Mode the query result will be opened in a list view window.

You can customize the columns to be used in the list view window.

Please click "Customize List View" to open the "Customize List View" dialog (see *Picture 238: "Customize List View" dialog*).

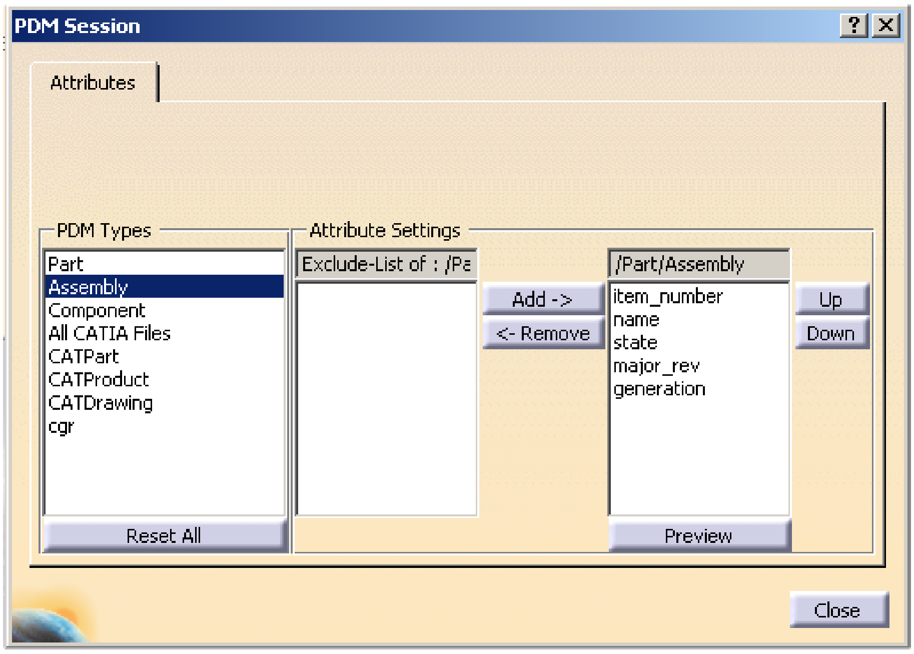


Picture 238: "Customize List View" dialog

When you are using a new PDM Workbench configuration file then you have to reset the column settings by clicking on "Reset All".

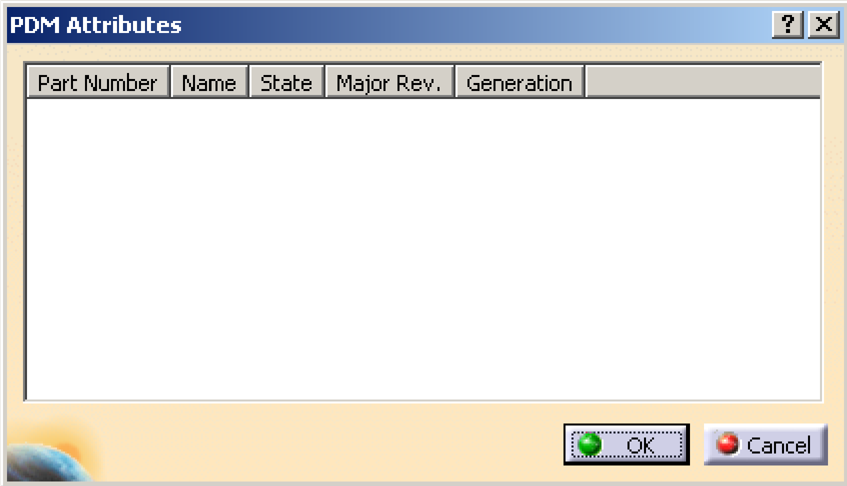
In order to customize the columns for an object type you have to select the object type. In the example in *Picture 239: "Customize List View" dialog for "Assembly"* the object type "Assembly" has been selected. In the right part of the dialog the attributes to be shown as columns are displayed. In the middle part of the dialog the attributes not to be shown as columns are displayed. In this case no attribute is hidden.

You can select an attribute on the right and remove it by clicking on the "Remove" button. Or you can select an attribute in the middle and by clicking on the "Add" button you can add it to the columns to be shown.



Picture 239: "Customize List View" dialog for "Assembly"

By clicking on the "Preview" button you can see a preview of the list view (see *Picture 240: Preview of the "List View" dialog*).



Picture 240: Preview of the "List View" dialog

PDM Relations

You have the possibility to hide or to show the PDM relations in the PDM product structure.

CATDrawings

CATDrawing files that are related to the root part of a part structure can be automatically downloaded when the part structure is loaded to CATIA.

Loading PDM Structures

When a PDM structure is loaded to CATIA sub-structures in the CATProduct tree can be hidden or deactivated if their corresponding PDM structures are not expanded.

PDM Session Configuration

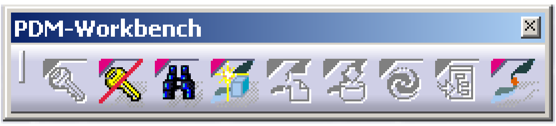
No configuration has to be set currently from CATIA for the Aras integration.

The configuration has to be made directly in the Aras Innovator application. For details please refer to the *PDM Workbench Installation & Administration Manual.*

Logout

Once you finished your work in PDM Workbench you do a Logout from the PDM system.

You select the "Logout" icon  within the PDM Workbench toolbar (see *Picture 241: The PDM Workbench toolbar after the login*) in CATIA V5 …



Picture 241: The PDM Workbench toolbar after the login

… and the session in the PDM system will be closed.

All PDM Workbench windows get closed. Please consider that CATIA native windows resulting from a “Load” or “Open File” PDM Workbench context action remain opened but that they are now out of synchronization with the PDM system. So we recommend you to close them, too.

CHAPTER 4

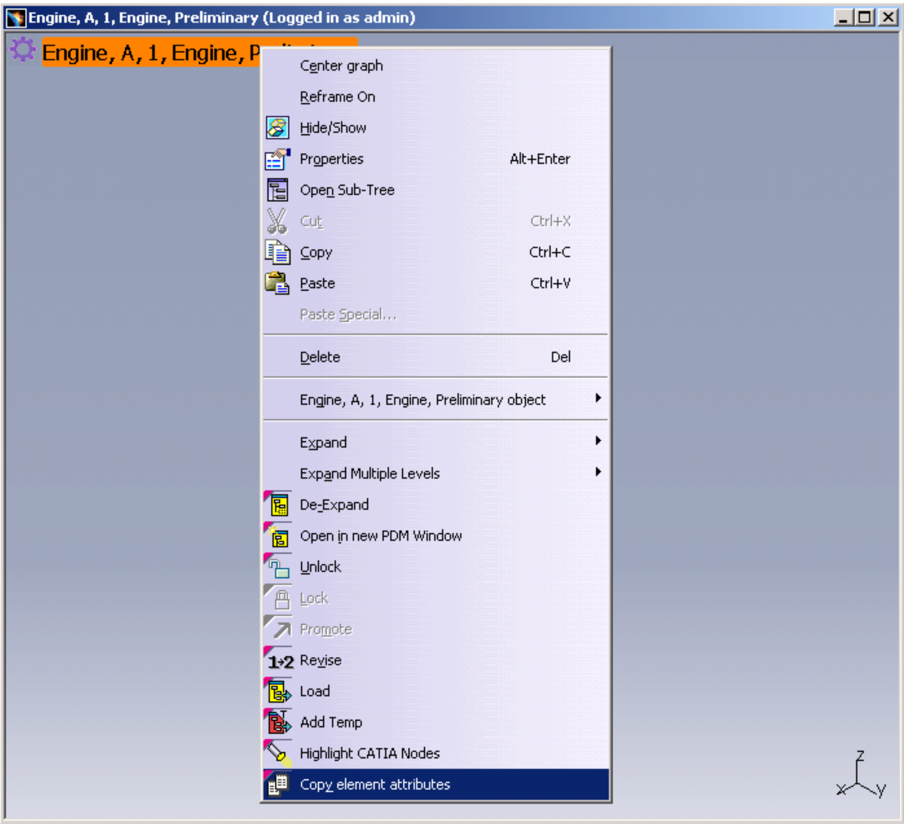
Additional optional functionality

This chapter describes optional functionalities of the PDM Workbench which are able to add in the CATIA V5 workshop.

Copy element attributes

It is possible to copy the attributes from a PDM object in order to use them in a create dialog.

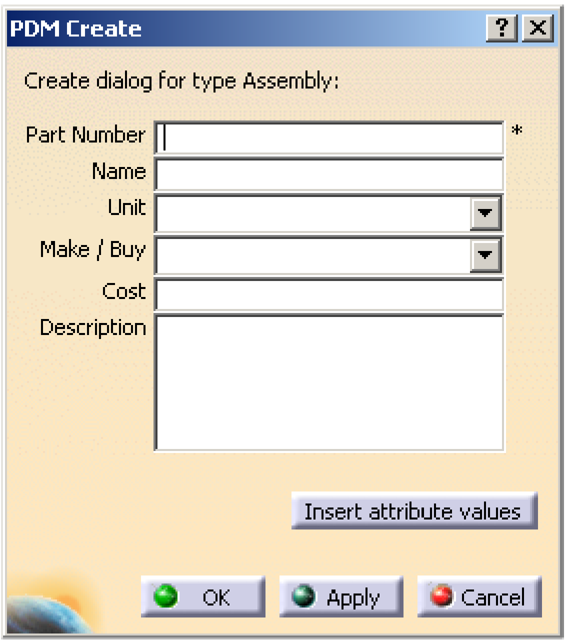
You can select a PDM object in the PDM window and click on the right mouse button. Then you select the action "Copy element attributes" (see ***Fehler! Verweisquelle konnte nicht gefunden werden.***).



Picture 242: Action “Copy element attributes”

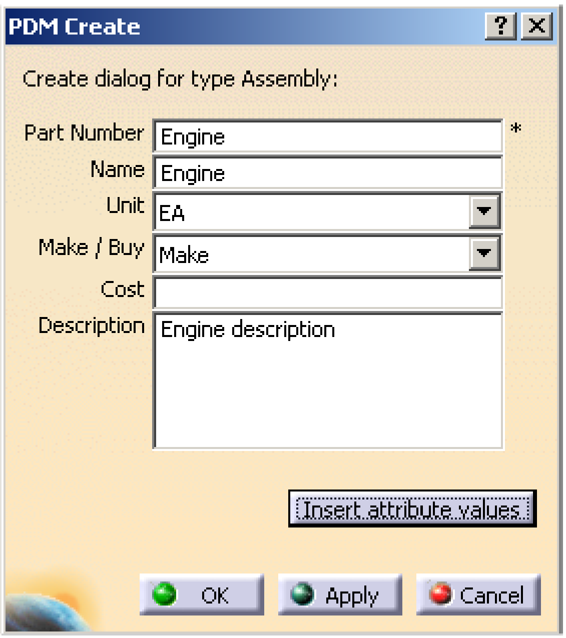
The attributes will be copied to the clipboard.

In the next step you select the action "Create" from the toolbar and select the corresponding class for the object to be created, in this case "Assembly" for the copied attributes of the "Engine". The create dialog will be opened. It has the button "Insert attribute values" (see ***Fehler! Verweisquelle konnte nicht gefunden werden.***).



Picture 243: “Create” dialog for Assembly

When you click on the button "Insert attribute values" the attributes of the dialog will be filled (see ***Fehler! Verweisquelle konnte nicht gefunden werden.***).

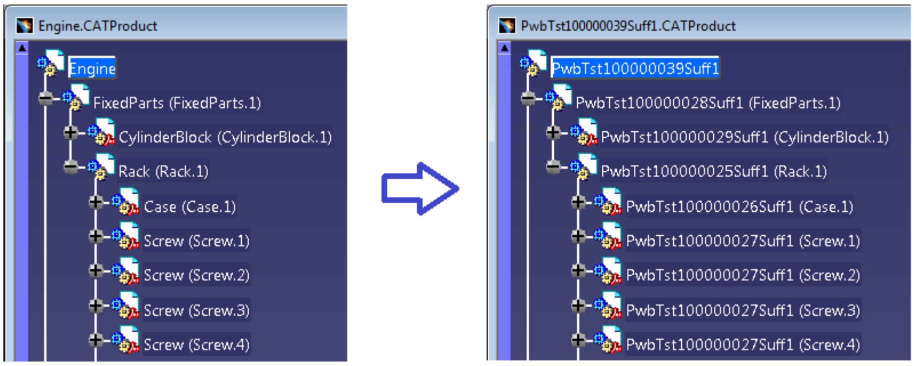


Picture 244: “Create” dialog for Assembly - Inserted attribute values

You can change the attribute values and start the create process by clicking on the "OK" button.

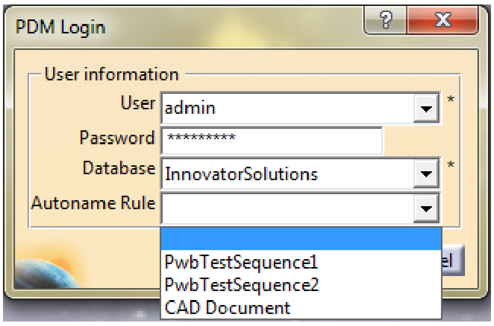
Autoname Support using Innovator Sequence Items

It is possible to optionally use Innovator sequence items to rename CATIA structures or single CATIA documents when they are created (see ***Fehler! Verweisquelle konnte nicht gefunden werden.***).



Picture 245: CATIA structure before and after import to PDM

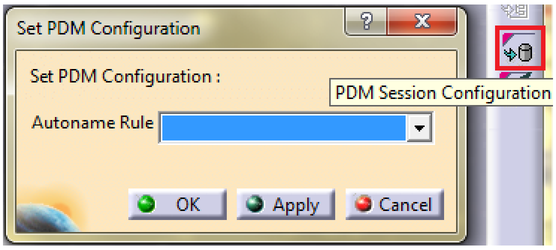
In the login dialog the user can select one of the autoname rule (Innovator sequence item) names (see ***Fehler! Verweisquelle konnte nicht gefunden werden.***).



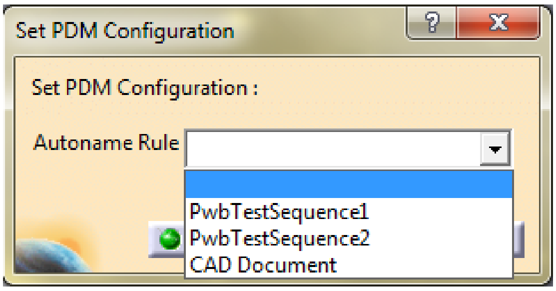
Picture 246: "Login" dialog with autoname rule

If none of the names are selected then the autoname functionality is not used.

Later in the session the user can change the selected autoname rule by clicking on the "PDM Session Configuration" icon in the PDM Workbench toolbar and selecting one of the sequence item names. This dialog can also be used to switch off the autoname functionality by selecting the entry containing the empty string (see ***Fehler! Verweisquelle konnte nicht gefunden werden.*** and ***Fehler! Verweisquelle konnte nicht gefunden werden.***).

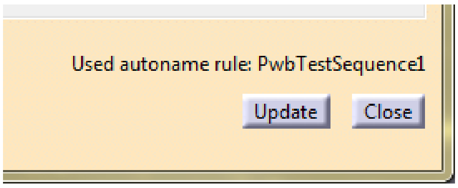


Picture 247: PDM session configuration dialog



Picture 248: Autoname rule combo box in PDM session configuration dialog

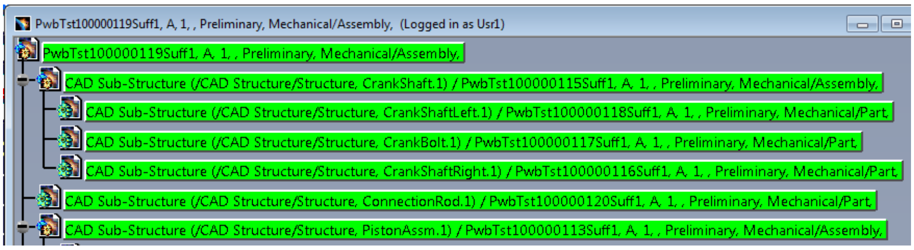
If an autoname rule is selected the update dialog will contain the information which autoname rule is selected (see ***Fehler! Verweisquelle konnte nicht gefunden werden.***).



Picture 249: Selected autoname rule displayed in update dialog

After creating new PDM items which correspond to the new CATIA documents the CATIA files will be renamed. The CATIA instance names will not change.

The corresponding PDM items will also have the names created by the selected sequence item (see ***Fehler! Verweisquelle konnte nicht gefunden werden.***).



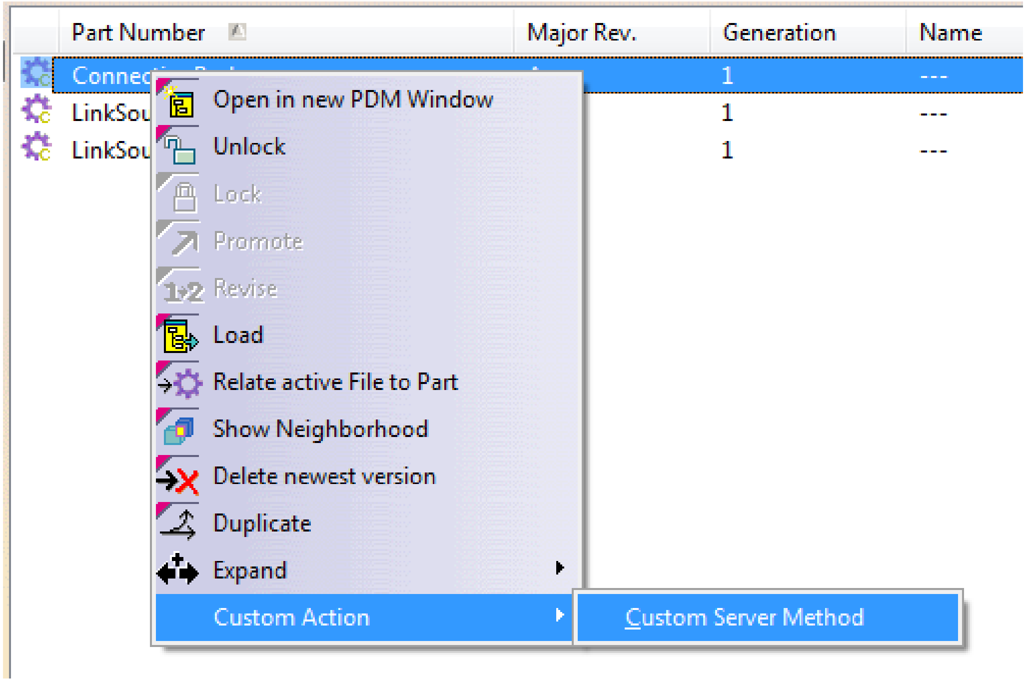
Picture 250: PDM structure named by sequence item

Further updates will not affect the names of the CAD documents and PDM items.

Possibility to call a server method for a PDM item

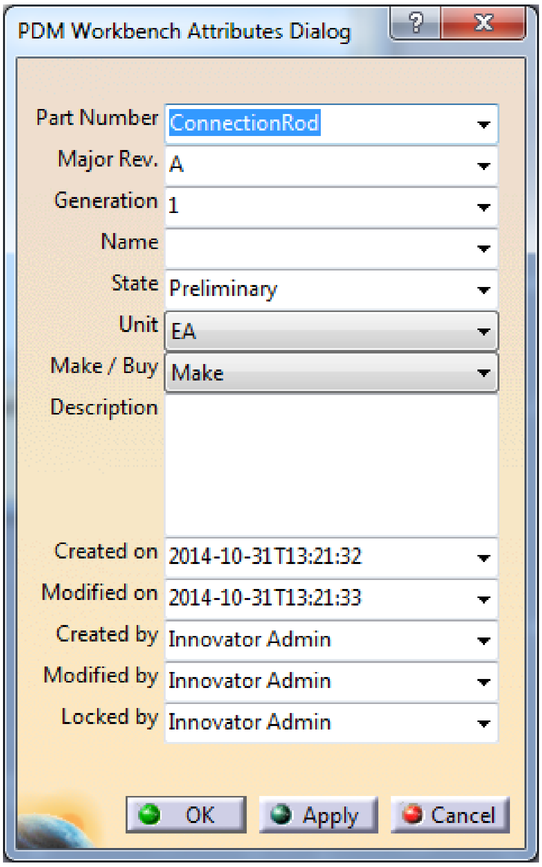
It is possible to call custom server methods with a PDM item and optionally with a dialog as input.

The user can right-click on a PDM item and select one of the custom server methods.



Picture 251: Selecting a custom method on a part item

If a dialog is configured it appears pre-filled with the attributes of the item:



Picture 252: Dialog with pre-filled attributes

Glossary

Unlock

Action withdrawing the right to update a work item. Normally this corresponds with publishing the work item to a larger number of people getting read access on this object.

Lock

Action giving the user the exclusive right to update a work item.

Context Menu

The menu that appears when the user selects an *icon* and holds the right mouse button pressed.

Dialog Window

Window in which the user enters information.

GII

Graphics Interactive Interface. The GII is a powerful programming tool, which completes the Open System Access to the CATIA environment.

Icon

Graphical representation of an *object*.

Object

An item or a relationship.

Query

To search the database for *objects* that match specific criteria.