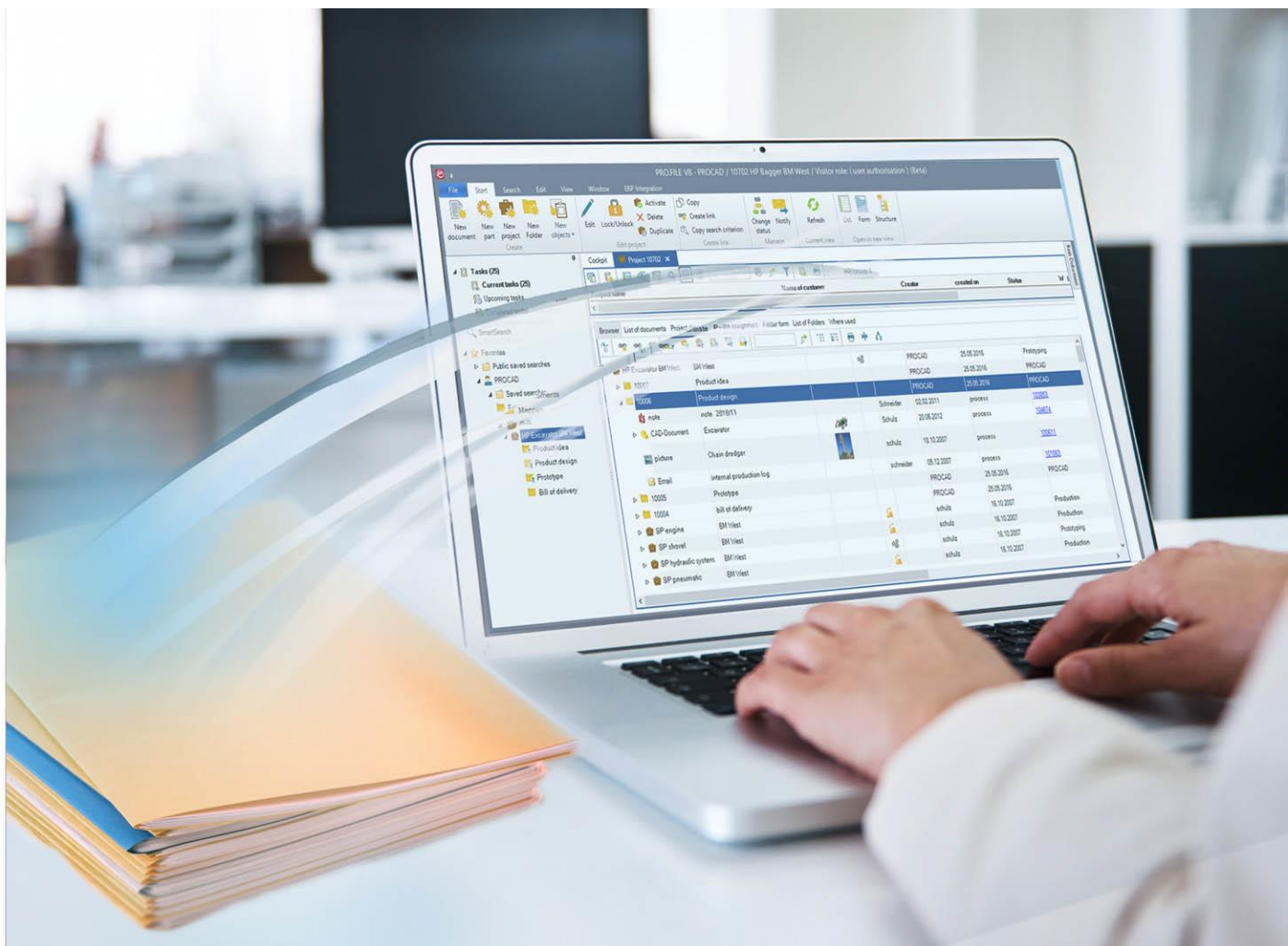


# Functions of the Integration PRO.FILE CATIA V5.NET

PRO.FILE Release 8.7  
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## About this manual

### Step-by-step instructions:

This PRO.FILE manual uses various signs and icons in order to guarantee a good readability and comfortable handling.

For quicker finding within the manual, step-by-step instructions are marked with a margin heading.

#### Menu sequences and function calls

Menu sequences and function calls explained in this manual are marked in bold and in quotation marks.

Example:

"File" => "New" => "Document description"

#### Buttons and keys

Keys and buttons are highlighted by angle brackets.

Example:

"Confirm with <OK>."

#### Notes and warnings

To highlight special information the following icons are used:



#### Function call:

"PRO.FILE" => "Extras" => "Options" => "Performance"



#### Example:

Boxes marked with this icon give subject-relevant examples for the usage of command lines, configuration strings and other software-relevant entries.



#### Note:

Boxes marked with this icon contain useful hints on the operation, configuration or installation of the PRO.FILE software.



#### Attention:

All information given in these boxes is very important and should be read carefully! Non-observance of these hints may lead to wrong functioning, display problems or other negative consequences.



#### Important notes:

The "stop sign" warns you of possible entry or operation errors, which may lead to loss of data!



#### Attention – Undo not possible:

All entries and configurations described in these boxes have to be made carefully, because they cannot be undone!

# 1 The integration PRO.FILE CATIA V5.NET

PRO.FILE PLM speaks the language of design departments and offers functions needed by design engineers.

PRO.FILE is an established PDM system for the administration of data and documents for the technical office. It works fully integrated into the CAD System CATIA V5. Drawings and CAD models can be loaded from or saved to PRO.FILE directly from CATIA V5.

When assemblies are saved, PRO.FILE automatically generates bills of materials and proofs of usage. These can be included in the drawing title block along with the product data.

Interfaces allow the transfer of product data (part master data, bills of material and CAD documents) specified during the design process with CATIA V5 to ERP systems.

## 1.1 The contents of this manual

The following chapters describe the operation of PRO.FILE within the CAD system CATIA V5.

The descriptions assume that the functions of the PRO.FILE basic software are known or can be looked up in the corresponding manual.

This documentation describes the interface between PRO.FILE and CATIA V5. The following topics will be addressed:

- Operation of PRO.FILE from within CATIA V5
- Data representation of structures/references in PRO.FILE
- Integration of PRO.FILE into the CATIA V5 environment

The configuration of the integration is described in the configuration manual of the integration.



### Note: Manual "CAD design supported by PRO.FILE"

When using the integration PRO.FILE CATIA V5.NET, please also note the manual "CAD design supported by PRO.FILE", which describes the basic procedures and related issues from the designer's point of view.

## 2 Let's get started: First steps with the PRO.FILE integration

Via the loading, saving and information functions of the PRO.FILE integration, the user can access information in and functions of PRO.FILE directly from CATIA V5.

The basic functions of the integration are explained in the following chapters:

- [Please note: Special aspects of the integration PRO.FILE CATIA V5.NET](#)
- [Only upon first start: Set up the local work folder](#)
- [Where to find the functions of the PRO.FILE integration?](#)
- [Log in to PRO.FILE](#)
- [A brief overview: The functions of the integration](#)
- [Close the menu of the PRO.FILE Integration](#)

### 2.1 Please note: Special aspects of the integration PRO.FILE CATIA V5.NET

When working with the integration PRO.FILE CATIA V5.NET, some CAD-specific aspects have to be noted. These are described in the following:

- [Operation of the PRO.FILE integration with different CATIA V5 versions](#)
- [Multiple sessions of CATIA V5](#)
- [External references CATIA V5](#)
- [Structures of assemblies within a file](#)
- [Selection](#)
- [Part families/Catalogues](#)
- [DL Names](#)
- [Balloons](#)

#### 2.1.1 Operation of the PRO.FILE integration with different CATIA V5 versions

By default, it is not possible to use the integration with different CATIA V5 versions within one PRO.FILE installation.

If this is required however, there is the possibility of configuring such an inter-versional operation.

For this purpose, a start script has to be created in order to select the CATIA V5 version to be started in connection with the integration.

This can either be done manually via the command line or via a batch file.



The content of the file has to be as follows:

"Path to the cnext of the version to be started" /regserver



#### Example for the script:

An example for a script with CATIA V5 R25 and the corresponding default installation path:

```
"C:\Program Files\Dassault Systemes\  
B25\win_b64\code\bin\cnext.exe" /regserver
```



#### Note:

Whenever you want to switch to a different CATIA V5 versions, this script command has to be run (with the script adjusted to the corresponding CATIA V5 version) before you start CATIA.

Without the usage of such an adjusted script, it is not possible to use the integration PRO.FILE CATIA V5.NET with another CATIA V5 version.

## 2.1.2

### Multiple sessions of CATIA V5

The starting of several CATIA V5 sessions is possible, but only one session can be active with a working integration PRO.FILE CATIA V5.NET at a time. As soon as another CATIA V5 session is started, the functions of the integration are no longer available in any of the sessions.



#### Note: Multiple sessions of CATIA V5

When an additional session is started, the integration displays a corresponding message. This message prompts the user to close one of the CATIA V5 sessions before being able to use the functions of the integration PRO.FILE CATIA V5.NET.

## 2.1.3

### External references CATIA V5

The integration PRO.FILE CATIA V5.NET only supports those references which were generated within the same assembly. Within PRO.FILE only connections between assemblies and sub-assemblies and single parts respectively will be saved and managed. External references that are build up between single parts as well as other types of references will not be supported.

Nevertheless it is possible to use external references in connection with PRO.FILE. The problem occurs when saving the file for the very first time because part A will be renamed during this action. As part A is referenced by another part B, part B will still be referencing the original part A outside PRO.FILE after saving because the integration PRO.FILE CATIA V5.NET cannot identify the external reference and thus it cannot substitute the external reference in an equivalent way. By using an appropriate mode of operation it is possible to evade this problem:

- When using external references it must be worked with PRO.FILE parts only. That means that the reference parts (part A) have to be saved in PRO.FILE first and can then be referenced by other parts (part B). Afterwards the reference will contain the correct file.
- If both part A and part B are parts of the same assembly the entire assembly will be saved in PRO.FILE. When renaming the parts within the assembly the references will be turned around automatically. One has to consider that due to the mode of operation of CATIA V5 a reference will only be turned around if part A is replaced BEFORE part B within the tree structure of the assembly. If part A is replaced BEHIND part B, part B will still reference the old part A after saving. In an interactive interrelation CATIA V5 works just as well.
- If documents have been renamed during the saving process it is important to check the links afterwards.
- It is also possible to substitute the reference manually within CATIA V5.

#### 2.1.4

#### Structures of assemblies within a file

With CATIA V5 it is possible to insert components into a product that do not refer to any file. Thus an assembly structure is generated which uses the basic product concerning savings within the file. To accommodate this specific characteristic such structures will be treated particularly inside the integration PRO.FILE CATIA V5.NET.

When saving such a product to PRO.FILE a metadata will be built for the basic product as well as for every single component. Only the metadata of the basic product contains a file. Within the structure of the document the metadata of the components reference the document of the basic product (reverse structure). This ensures that when selecting a component in PRO.FILE the file of the basic product will be loaded.

The part structure in contrast will be transmitted directly, i.e. the metadata of the components reference the metadata of the basic product. Thus it is possible to map more complex part structures appropriately.

Concerning the components it must be pointed out that your Properties cannot be allocated automatically by values out of PRO.FILE.

### 2.1.5 Selection

The initial point for every command in the integration PRO.FILE CATIA V5.NET is the active single part and the active assembly respectively in the current CATIA V5 session. An active document is the just selected document inside the structure tree of the product. That assures the possibility to query e.g. information about a single part belonging to a complex assembly without opening the part in a separate window.

### 2.1.6 Part families/Catalogues

The integration supports the connection between a CATIA V5 document and the construction table in the form of an Excel file in which the configurations are defined. It is possible that different files use the same table of construction. This fact has to be considered when creating a new version of the Excel table because when versioning the entire table all dependent parts will be versioned automatically.

The integration does not support CATIA V5 catalogues.

### 2.1.7 DL Names

It is now also possible to work with DL names only. In order to do so, the "DL names" option has to be set to "current". If "not allowed" was configured for the local search paths, the working directory of the integration must be defined as a DL name.

### 2.1.8 Balloons

The integration PRO.FILE CATIA V5.NET supports the automatic compilation of balloons within drawings. This function is based on the CATIA V5 function to construct balloons. It only works if the CATIA V5 language is set English. Adjacent these balloons will be filled up with information out of the PRO.FILE bill of material.

This function only uses the first level of the illustrated assembly.

## 2.2 Only upon first start: Set up the local work folder

CAD drawings are loaded directly from PRO.FILE in CATIA V5, and also saved and versioned from CATIA V5 directly to PRO.FILE. For this, the drawings are saved intermediately on the user computer in a "work folder".

The local saving of the CAD drawings makes sure that all required parts and documents required for working with the CAD drawing are available on the user computer.



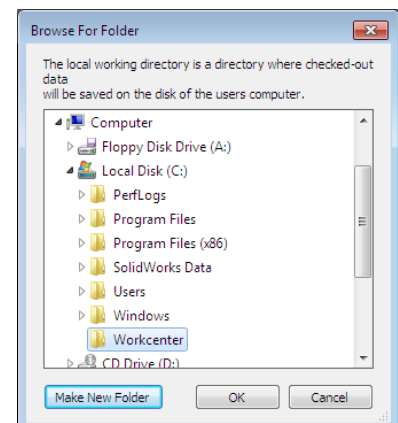
### Note: Local work folder is always required

The CATIA V5 features require the availability of all related data. Without defining a local work folder, it is not possible to work with the integration PRO.FILE CATIA V5.NET.

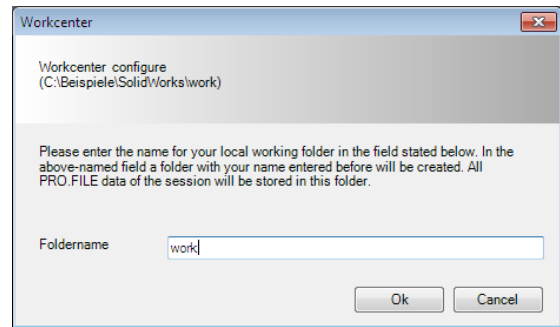
You can freely choose the local work folder at the first start of the integration.

Proceed as follows

1. If no local work folder is defined when the integration is started, an input screen will prompt you to define this folder.
2. You now have to specify a "root folder". The root folder is the superior folder of the local data storage. In this folder you can later create several work folders, which are then supervised by the "Workcenter".
  - The "root folder" can be selected - or created via the button **<Make new folder>**.
  - Once you have selected the desired root folder, confirm with **<OK>**.
3. In the second step, "work folders" are now created in this root folder, which will then be used by the integration. Consequently, you are now prompted to specify a work folder within the root folder:



- ⇒ Please specify a name for the work folder.
- ⇒ Confirm your entry with <OK>.
- ⇒ The configuration of the Workcenter is now finished.



This work folder and other local work folder can be created and managed by the user via the Workcenter. The Workcenter can be accessed via the PRO.FILE menu in CATIA V5 under "Extra" => "Workcenter".

Detailed information can be found in the chapter "[Workcenter](#)".

## 2.3

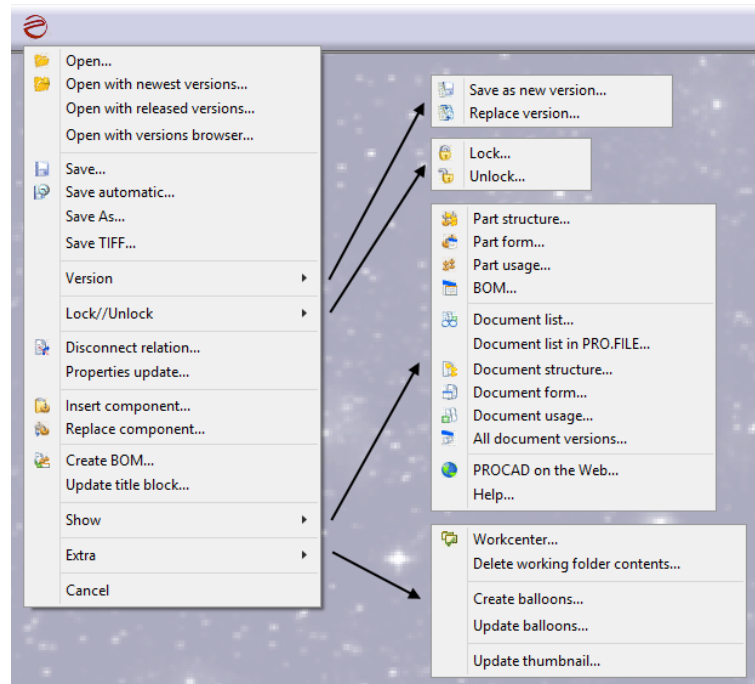
### Where to find the functions of the PRO.FILE integration?

Via the menu "PRO.FILE" in the CATIA V5 menu bar all functions and features of the integration can be accessed.

1. Start CATIA V5.
2. Go the menu bar to the section "PRO.FILE".
3. Select the desired integration function from the menu.

Contents of this PRO.FILE menu depend on the CAD object that is currently active in the foreground.

The integration provides the corresponding functionalities, depending on the active CAD document type – or disables them.



Whenever one of these menu functions is used, PRO.FILE is activated. According to the function used, the required windows in PRO.FILE are opened automatically.

The user can thus access the whole range of PRO.FILE database commands with all possibilities for data management.



#### Note: PRO.FILE macro needs to be activated

After the installation of the integration the PRO.FILE macro for the startup of the integration has to be configured. If the PRO.FILE toolbar is not displayed, please activate it as for the initial configuration of the integration.

For more details, please see the configuration manual of the integration or contact your administrator.

## 2.4

### Log in to PRO.FILE

If you access a PRO.FILE function for the first time within an CATIA V5 session, you have to log in to PRO.FILE.

Via the logon, the user is now identified by his PRO.FILE user name and password. Based on this logon, the user rights, start statuses and function access rights for the logged-on user are activated.

In the login screen, please enter:

- Your PRO.FILE user name
- Your PRO.FILE password

Confirm with <OK>.

The PRO.FILE home screen is now displayed.



**Note: No login required in case of "Autologin"**

This login is not requested, if the PRO.FILE autologin function is activated.

## 2.5

### A brief overview: The functions of the integration

This chapter gives you a brief overview of the possibilities offered by the menu functions. Detailed information on each function can be found later in this manual.

#### Functions to Open CAD objects

- **Open:**  
This function opens PRO.FILE and provides the user the option to choose a CAD document and open it in CATIA V5 by confirming the choice.
- **Open with newest versions:**  
This open function automatically reads the newest version of the references of the CAD objects from PRO.FILE in CATIA V5.
- **Open with released versions:**  
This open function automatically reads the newest released version of the references of the CAD objects from PRO.FILE in CATIA V5.
- **Open with version browser:**  
This function is almost identical to the "Open" function with the exception that the version selection will appear when a document is selected.

#### Saving Functions

These functions allow you to check in newly created CAD objects to PRO.FILE or to save data that is already stored in PRO.FILE and that was retrieved for modification back to PRO.FILE. When saving back, the data in PRO.FILE will be automatically overwritten. The process of saving and modifying also depends on the user's authorizations. This ensures that only authorized users can create or modify documents in PRO.FILE.

- **Save**  
This function saves the entire object with all of its contained elements. For those elements within the object that have not yet been saved, you will be asked to provide information for the document master or, if need be, for the part master.

- **Save automatic**  
This function saves the complete object with all its contained elements.  
The document master and, if need be, the part master for the object will be filled in manually. A document master and, if need be, a part master will be automatically created in PRO.FILE for those elements within the object that have not yet been saved. The name of the object file is given as a name.
- **Save As**  
With this function you can use entire assemblies as a template in order to create a new project with it. The original documents are not referenced by the resulting documents, which are simply copies of them.  
This function saves the entire object with all of its contained elements as a new object in PRO.FILE. The document masters and, if need be, the part masters of the originals are used as a template for the new document/part masters. You can either manually or automatically create the new elements (see "Save" and "Save automatic").  
During this process, the drawings that belong to each element will also be identified. These will also be included in PRO.FILE as new elements and linked to the corresponding products/parts.
- **Save TIFF**  
This function creates a CAD drawing in the neutral TIFF format and saves it as a document. This TIFF document is automatically attached to the body of parts of the drawing.

#### Functions for Versioning

- **Save as new version:**  
Saves the currently active CAD-object as a new version in PRO.FILE. If this function is used on a part that is built into an assembly, the references of the assembly will still be referred to the old version of the part after the new version management of PRO.FILE.
- **Replace version:**  
The command "Replace version", enables an existing built version of a CAD-object to replace an object in all assemblies in which the previous version is built into.

#### Functions to Lock/Unlock CAD objects from PRO.FILE while working in CATIA V5

- **Lock:**  
CAD objects which were read from PRO.FILE in CATIA V5 are not automatically locked for the user. To be able to modify a CAD object, the order "Lock" must be called up beforehand. By this function call, the rights of the user are checked, the topicality of the CAD object is checked and the data is locked in favor of the user, so that no other user can carry out changes.
- **Unlock:**  
This function unlocks the PRO.FILE objects which were locked for processing in CATIA V5. Other users can again carry out changes to the object. Changes to an unlocked object are not automatically stored in PRO.FILE so the storage process must be carried out separately.



## PRO.FILE Database functions

- **Disconnect relation**  
This function deletes the database link of a PRO.FILE part, a PRO.FILE drawing or an entire PRO.FILE assembly including choice of objects contained therein. The CAD objects are then treated as purely locally-saved CAD objects.
- **Properties update**  
If you have linked PRO.FILE metadata to properties in a product or part, you can use this function to update them.

## Working with PRO.FILE components

- **Insert component**  
You can use this function to open the CAD models stored in PRO.FILE within an assembly. To activate the assembly into which the component is to be included, simply double-click on it in CATIA V5. Next, the integration command is executed and the component is selected in PRO.FILE. The positioning is carried out automatically by CATIA V5.
- **Replace component**  
Replace an existing component with one that is already stored in PRO.FILE.

## Functions for bill of materials and title block

- **Create BOM**  
This function calls up PRO.FILE, and creates a new bill of material for the active assembly. If there is already a PRO.FILE BOM, this is updated.
- **Update title block**  
This function allows information on bill of material, modification list and title to be filled in on a drawing upon opening. This requires that lists and fields are pre-configured for the template to be used.

## Viewing and information functions

- **Part structure**  
Shows the structure overview of the displayed/selected object.
- **Part form**  
Changes to material master form of the displayed/selected object..
- **Part usage**  
Changes the proof of usage list of the displayed/selected object.
- **Bill of material**  
Shows the bill of material view of the displayed/selected object.
- **Document list**  
This function calls up the document special list in PRO.FILE, and shows the user the information that has been configured for the actual part, drawing or assembly, and all attached parts.
- **Document structure**  
Changes the document structure list of the displayed/selected object.
- **Document form**  
Changes the document description in the list presentation of the displayed/selected object. If there is no Object specially selected within an assembly, then the form of the displayed assembly will be shown.

- **Document usage**  
Changes the proof of usage list of the displayed/selected object.
- **All document versions**  
Shows all document's versions saved in PRO.FILE.
- **PROCAD on the WEB**  
This menu point opens the PROCAD homepage, as long as the user has internet access.
- **Cancel**  
Closes the PRO.FILE integration menu.

#### Extra: Additional functions

- **Workcenter:**  
All files loaded or saved via the integration PRO.FILE CATIA V5.NET are automatically saved locally in the Workcenter folder. With this function you can manage these files or create additional work folders.
- **Delete working folder contents**  
The locally stored files can be deleted with this function.
- **Create balloons**  
With this function you can insert balloons for bill of materials positions from PRO.FILE in the drawing, when working with drawings and assemblies.
- **Update balloons**  
This function updates existing balloons on a CATIA V5 drawing with values from the bill of materials in PRO.FILE.

## 2.6

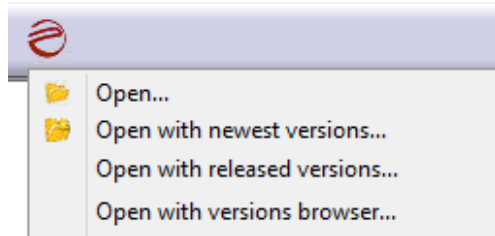
### Close the menu of the PRO.FILE Integration

The menu of the PRO.FILE integration can be closed by clicking on the function "Cancel".

This function can be found at the bottom of the menu.

### 3 Opening CAD Documents from PRO.FILE in CATIA V5?

PRO.FILE manages CAD data centrally and makes them available to users with the corresponding access permissions via the command "Open".



This chapter explains the functions and possibilities in the context of opening documents:

- [Open: Opening CAD Documents from PRO.FILE](#)
- [Open with newest and released versions of the components](#)
- [Open with version browser](#)
- [Attention: Opening of locally existing files](#)



#### Attention:

The data loaded from PRO.FILE in CATIA V5 are not automatically locked when opened in CATIA V5. The user has to lock the objects manually via the function "Lock".

After the object has been edited and saved back to PRO.FILE, it can be unlocked again, so that it is available to other users. For detailed information see the chapter "[Lock/Unlock: Who can change when?](#)"



#### Note: PRO.FILE checks permissions

When the function "Open" is used for documents from PRO.FILE, the corresponding access rights of the user are checked. These permissions depend on the user access rights as well as on the status-dependent permissions of the document.

### 3.1 Open: Opening CAD Documents from PRO.FILE

If you want to access a document from PRO.FILE, use the function "Open" of the integration PRO.FILE CATIA V5.NET.

This function starts the PRO.FILE Checkout wizard, in which you can select the desired document for loading in CATIA V5.

To open a CATIA V5 document from PRO.FILE proceed in the following steps:



Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Open"

#### Step 1

#### Using the PRO.FILE function "Open"

1. Select the function "Open" from the "PRO.FILE" menu.

- "Open" loads the documents they were recently saved.
- (The functions "...newest versions" and "...released versions" relate to CAD data with linked elements from PRO.FILE. In this case you can load specific versions of the linked elements from PRO.FILE. See chapter "[Open with newest and released versions of the components](#)").

⇒ The Checkout wizard for the selection of the desired document is displayed.

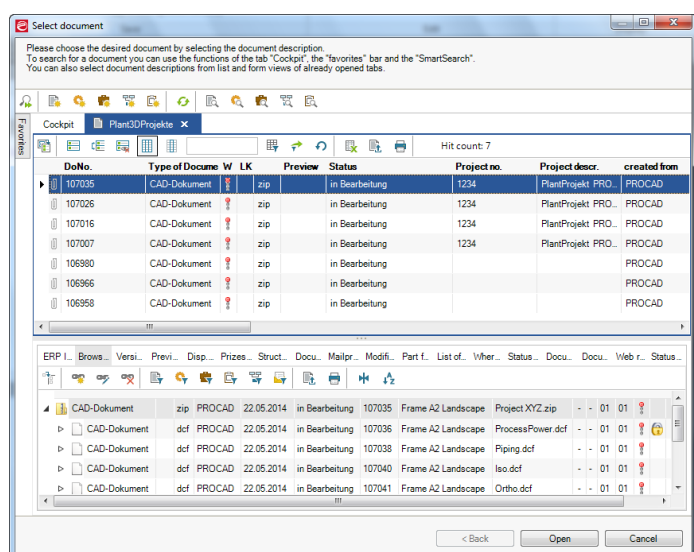
#### Step 2

#### Selecting the desired document in the Checkout wizard

The Checkout wizard displays the PRO.FILE GUI as it was used the last time.

2. If the desired document is not yet displayed in a list or form view, you can start a selection via the following functions:

- Via the tab "Cockpit".
- Via the search function in the icon bar.
- Via favorites, SmartSearch or task assignment



3. If the desired document is displayed in a list view, you can select it. (If the desired document is displayed in a form view, it is already selected. Click <Open>.
- ⇒ The Checkout wizard closes and the dialog screen for the loading type is displayed.

Detailed Information on the Checkout wizard can be found in the following chapter "[Working with the Checkout wizard to search for CAD documents](#)".

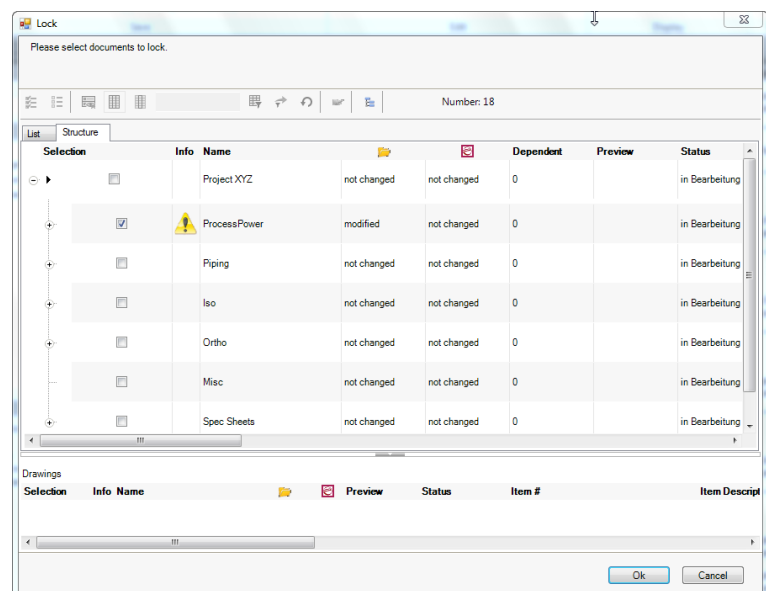
### Step 3

#### You want to edit? Then you must lock the document(s)

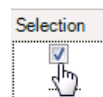
Until now, the selected CAD data are not locked in PRO.FILE and can be modified by other users in PRO.FILE.

This means: If you want to edit the CAD document, you have to lock it. The dialog "Lock" supports you in this process:

- ⇒ If you click on "Lock" the dialog for locking CAD documents is displayed. (Information on the functions and status indicators can be found in the chapter "[Data overview: The document list](#)")



4. Select all documents you want to lock with the corresponding check boxes.
5. Confirm your selection with <OK>.



- ⇒ The selected document is now opened with its components in CATIA V5. The "Open" procedure is thus finished.

Detailed information on the locking of document can be found in the following chapter: "[Lock/Unlock: Who can change when?](#)".

**Note: Why can I not lock a document?**

You want to open a document for editing, but in the "Lock" dialog, you cannot activate the corresponding checkbox?

This may have two reasons:

- The document is already locked by a different user. You can see who the locking user is by selecting document in PRO.FILE and looking at the dependent tab "Status information".
- The document is in a workflow status, in which you are not allowed to edit the document. This is typically the case for "released" documents. If the drawing of a part is in the status "released", the part has already been manufactured and delivered and thus can no longer be changed.

For detailed information on the "Open" process, please see the following chapter:

[Working with the Checkout wizard to search for CAD documents](#)

### 3.1.1

#### Checking the project environment when opening

If you are working with several environments in CATIA V5, it is possible to assign documents to a specific environment when saving. You can specify whether to check the CATIA V5 environment when you open a document.

If the CATIA V5 environment corresponds to the environment that was stored for a document, the document will be loaded. If not, you can specify whether to warn the user or to prevent CATIA V5 from being opened at all.

### 3.1.2

#### Working with the Checkout wizard to search for CAD documents

If you use the function "Open" from the PRO.FILE integration, you have to select the document to be opened in the Checkout wizard.

The aim of this procedure is:

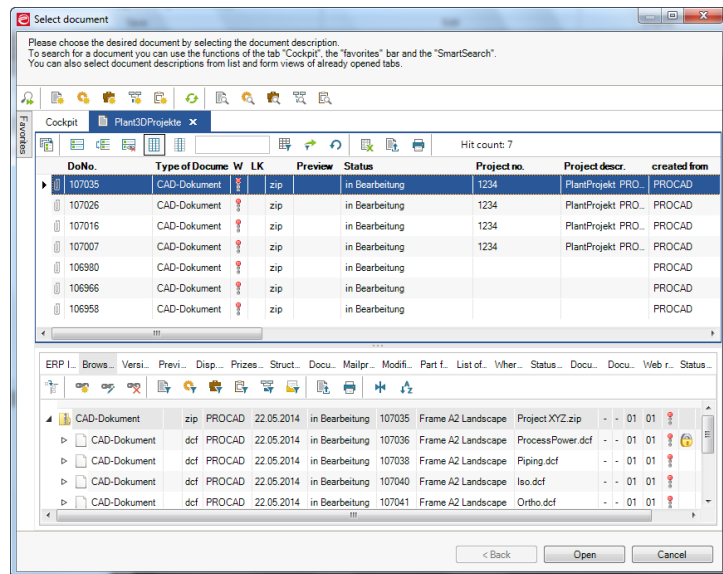
- For PRO.FILE to know which document is to be opened, the desired document description must
  - either be selected in a list view or a structure browser
  - or be displayed in a form view
- Then the button **<Open>** at the right bottom of the Checkout wizard has to be clicked.

Prerequisite for the selection/activation of a document in PRO.FILE is that the document is displayed in a list or form view.

When the Checkout wizard is opened, the PRO.FILE GUI is displayed as it has been used the last time:

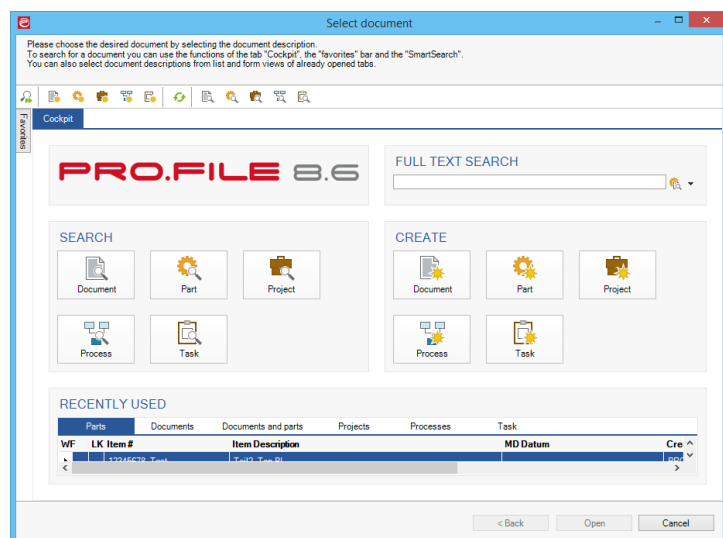
- If search results had previously been opened in a list or form view in a separate tab, you can directly access the displayed documents.

If the desired document is displayed on a tab, select it and click **<Open>**.



- If no search had been made previously, or if the desired document is not displayed on one of the existing tabs, you can now search for it.

For this, several functions, which are explained in the following, are available:



### Attention: Double-click in the Checkout wizard




Documents are selected and then opened via the **<Open>** button. A document cannot be loaded via a double click!

Because a double click means: Open document for viewing!

The checkout will remain active in the background waiting for your selection. But only if the checkout wizard is closed, the document can be used for editing in PRO.FILE.

## Searching for data records in the Checkout Wizard

To search for data records in the Checkout Wizard, several options are available:

- **Searching via the tab "Cockpit"**  
The same icons as in the icon bar can be found on the tab "Cockpit": "Search document", "Full-text search", "Search part", "Search project" have the same function as the icons in the icon bar.  
You can always go back to the tab "Cockpit".
- **Search via the functions of the favorites bar**  
The favorites bar also offers several ways of searching for a document:
  - Via the "Favorites" memorized searches or data lists can be accessed with a double click.
  - With the "SmartSearch" you can create individual search forms.
  - If you are working with PRO.FILE processes and tasks, you can access the documents linked to a task or process via the task or process structure.
- **Search via the icon bar**  
In the superior icon bar you can start a search via the following buttons:
  -  :Search for document descriptions to be displayed in a list.
  -  : Search for parts to be displayed in a list. Documents linked to the part can be displayed in the dependent tabs "Structure" or "Document list".
  -  :Search for projects to be displayed in a list. Documents linked to the project can be displayed in the dependent tabs "Structure" or "Document list".

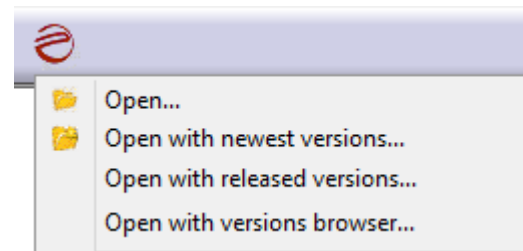
Detailed information on the selection of data in PRO.FILE can be found in the manual "Operation PRO.FILE for Beginners".



## 3.2 Open with newest and released versions of the components

To open a document from PRO.FILE, the user can choose from three options:

- "Open"
- Open with "Released versions"
- Open with "Newest versions"



The two version options do not refer to the document selected for opening in PRO.FILE.

They only refer to the objects linked to the document to be opened from PRO.FILE.

⇒ As a user, you can decide with which version status you want to open the components linked to the PRO.FILE – CAD document.

This means:

- **"Open"**  
The selected document is opened from PRO.FILE as it was saved the last time. Linked CAD documents are loaded with the version status, as they were saved the last time via the PRO.FILE integration.
- **"Released versions"**  
The selected document is loaded with the newest, released versions of the linked CAD documents. The newest versions that are in a release status are loaded. When the function "Released versions" is used for opening an assembly, PRO.FILE checks, whether the assembly contains components for which versions in a release status exist. If this is the case, the newest visible version in a release status of such a document is loaded in the Inventor session.
- **"Newest versions"**  
If the CAD document contains links to other CAD documents in PRO.FILE, the newest versions of these linked CAD documents are loaded. When the function "New versions" is used for opening an assembly, PRO.FILE checks, whether the assembly contains components for which versions exist. If this is the case, the newest visible version of such a document is loaded in the AutoCAD session.

The actual process of opening the document is identical for all three of these options. For more information see the previous chapter "[Open: Opening CAD Documents from PRO.FILE](#)".



### Note:

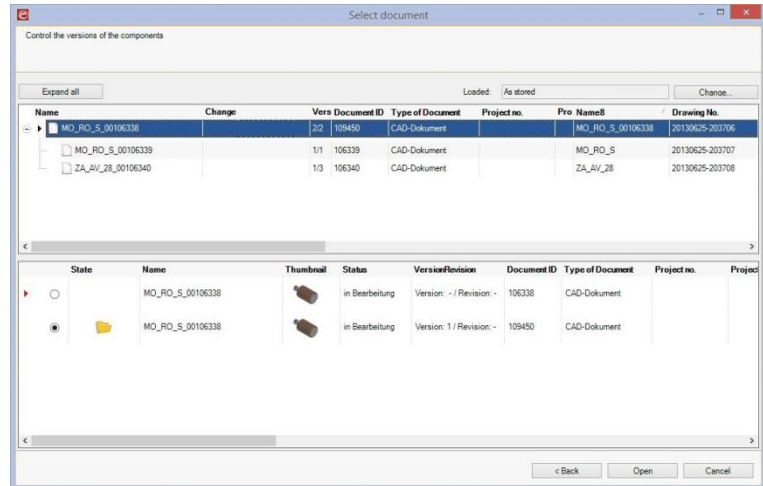
Only the versions, for which the user has the viewing permission can be displayed. If the most recent version is not "visible" for you, you will get the newest version visible for you.

### 3.3 Open with version browser

With the version browser you can open assemblies in dynamic constellations.

You can define via the version browser with which version an assembly and its parts is opened.

The function "Versions choice" works similar to the function "Open" – with the difference that the version browser is displayed after the checkout wizard:



The version browser is divided into two areas:

#### The document structure (top):

- In the upper structure windows the selected CAD document is displayed with all attached components.
- Via the button <Expand all> you can display the entire structure of the part to be opened.
- The field "Loaded" shows the current opening type of the CAD elements displayed in the structure window – without manual version selection. The opening type affects the display of these elements:  
Via the button <Change...> you can choose between the four options for opening:
  - Open "as stored"
  - Open "latest version" of the components
  - Open "latest released version" of the components.
  - Open "latest release version or latest version" of the components, depending on their availability.

#### The version window (bottom):

- In the lower window the different versions of a component are listed.
- You can select the version of the component that you want to open in CATIA V5.




Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Open..." => "Open with version browser"

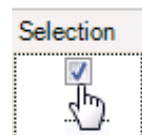
Proceed as follows

1. Select the "PRO.FILE" menu from the menu bar in CATIA V5.
2. Select the function "Open..." => "Open with version browser".  
=> The Checkout wizard is displayed.
3. Select the desired CAD document and click on the <Open> button.  
=> The Checkout wizard closes.  
Detailed information on the Checkout wizard can be found in the following chapter ["Working with the Checkout wizard to search for CAD documents"](#).
- => The screen "Select document" is displayed.
4. Select the component, for which you want to make a version selection, in the document structure.  
=> The lower version window now displays all corresponding versions.
5. By toggling the radio button in the first column of the version window you can activate the desired version of a CAD element:




State	Name	Thumbnail	Status	VersionRevision	Document ID	Type of Document
<input checked="" type="radio"/>	Hydr_Zylinder_101663.ipt		in Bearbeitung	Version: - / Revision: -	101663	CAD-Dokument





Having activated all desired versions, you can leave the version browser by clicking <Open> in order to continue the loading process.

- => The window "Lock" is displayed.
- => At this moment, the selected CAD data is not yet locked in PRO.FILE and still available for other users. This means: If you want to edit the CAD data, you have to lock it.
- => Select all documents you want to lock by setting the checkmark for it in the first column.  
Detailed information on the locking of documents can be found in the following chapter ["Lock/Unlock: Who can change when?"](#).
- => Confirm your selections with <OK>.
- => The selected CAD components are opened in CATIA V5. The process of opening with the version browser is now finished.



In the following table you can find the meaning of the different icons displayed within the version browser:

Icon	Meaning
	Indicates that this version status is the currently saved one.
	Indicates an object, the version of which has been exchanged.
	Shows a version conflict. This can occur, e.g. if a part is used in two assemblies in different versions.

Icon	Meaning
	Icon of CATIA V5 assemblies
	Icon of CATIA V5 parts
	Indicates a softlink.
	Versions reference each other causing a version cycle.

### 3.4

## Attention: Opening of locally existing files

When a CAD document is opened, all required elements and components are loaded into the current work folder.

If the work folder already contains a file of the same name, you will get a list of the elements that are to be overwritten. This also applies for newer or older versions of a CAD documents, which can now be overwritten.



### Attention: Risk of data loss

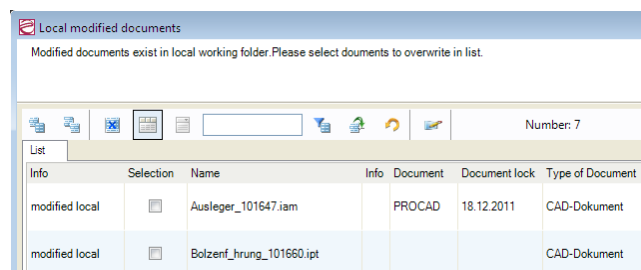
If locally changed files are overwritten with files from PRO.FILE, these local files and all changes to them are irretrievably lost!

If versions are overwritten, the locally existing assembly now point to the newly loaded version which has replaced the original version! You should therefore be careful when deciding to overwrite a locally existing version

A message screen informs you that locally existing files have been found. You can now select, which of the locally existing files is to be loaded anew from PRO.FILE, and thus overwritten.

The list shows which of the files have been changed locally and no longer match the status saved in PRO.FILE

Different versions are also indicated.



You have three options of proceeding:

- Overwrite with status from PRO.FILE: Activate the checkbox in column "Selection" for the list entries, the local status of which is to be overwritten with the status from PRO.FILE. If you confirms this action with <OK>, all files are copied from PRO.FILE to your workstation.

- Do not overwrite: Leave the checkbox unchecked.
- Load data in a different Workcenter folder: You can switch to a different working folder via the command "PRO.FILE" => "Extra" => "Workcenter" => "Activate", to avoid the overwriting of data. (See chapter "[Workcenter functions](#)")

**Note:**

Due to this behavior it is absolutely necessary, that the file names in PRO.FILE are unique. Otherwise, it may happen that a "screw" version M5x16 is overwritten with another variant M12x40 also named "screw".

## 4 Lock/Unlock: Who can change when?

If you are editing a CAD document and want to save the changes back to PRO.FILE, this document has to be locked for other users from the moment the changes begin.

- Only by using the function "Lock" you can make sure, that other users are not making changes to the same document at the same time.
- With the function "Unlock" the CAD document is made available again to other users for editing.

For detailed information see the following sub-chapters:

- [Starting your changes: "Lock" the CAD document](#)
- [The "Unlocking" of CAD documents](#)

The **locking** of a CAD document makes sure that the CAD document is not modified by other users in the meantime:

- A locked CAD document can be opened by other users via the function "open". However, these other users cannot save back any changes to the locked CAD document to PRO.FILE.
- If the CAD document has been opened and not locked, other users may make changes to the document in the meantime and save those changes back to PRO.FILE. In this case, it will no longer be possible for you to save your own changes back to PRO.FILE.

For detailed information please see the manual "CAD design supported by PRO.FILE".

### 4.1 Starting your changes: "Lock" the CAD document

If a document is to be modified, it has to be locked by the user!



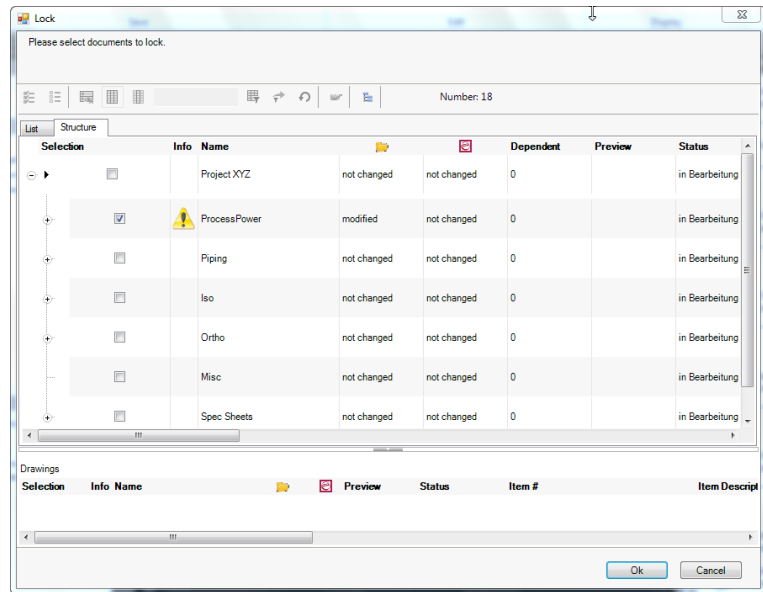
Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Lock/Unlock" => "Lock"

To lock a CAD document manually proceed as follows:

1. Make sure that the CAD document to be locked is displayed in CATIA V5.
2. Select the menu "PRO.FILE" from the CATIA V5 menu bar.
3. Select the function "Lock/Unlock" => "Lock".

- ⇒ The dialog for locking the loaded CAD documents is displayed. (Information on the functions and status information can be found in the chapter "[Data overview: The document list](#)").



- ⇒ With the display of status information in this list PRO.FILE checks:
- whether the user has the permission to edit the document.
  - whether the active documents are up to date and have not been modified by a different user since their opening.
  - whether the active documents does not already have a lock flag.
- ⇒ If any of these checks returns a negative result, the document cannot be locked!

4. Select all document you wish to lock by setting the checkmark in the first column.
5. Confirm your selections with <OK>.



- ⇒ By the command "Lock", the access permission to edit the document is checked for the current user, and the document is locked for all other users in the database.

Once the CAD document is locked, it can be modified. The changes are then saved back to PRO.FILE via the function "Save".



#### Attention: Changes in the team

It is recommended to lock document you want to edit directly after opening.

## 4.2 The "Unlocking" of CAD documents

In analogy to the function "Lock" you can unlock documents that have been locked by you by using the function "Unlock".



### Note:

You can only unlock documents that have been locked by you. The right to unlock documents that have been locked by other users can only be given to administrators.



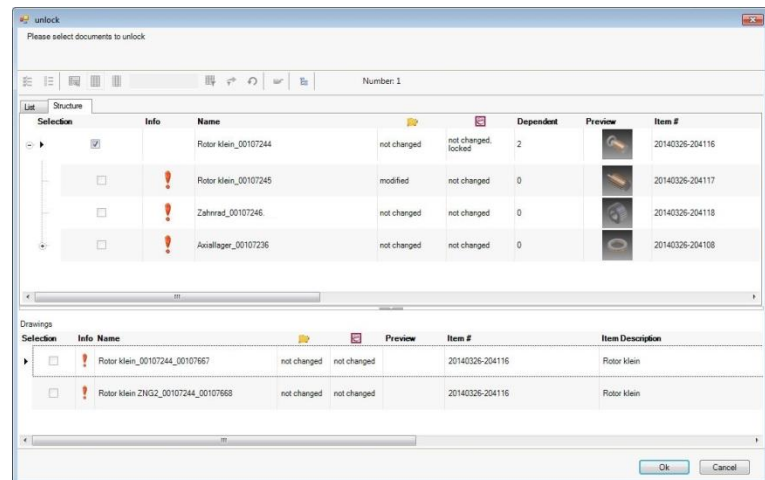
### Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Lock/Unlock" => "Unlock"

Proceed as follows

1. Make sure that the CAD document to be unlocked is displayed in CATIA V5.
2. Select the menu "PRO.FILE" from the CATIA V5 menu bar.
3. Select the function "Lock/Unlock" => "Unlock".

4. The dialog for unlocking the loaded CAD documents is displayed. (Information on the functions and status information can be found in the chapter "[Data overview: The document list](#)").



5. To make the CAD documents saved in PRO.FILE available for other users, select the documents to be unlocked in the list.
6. Confirm your selections with <OK>.



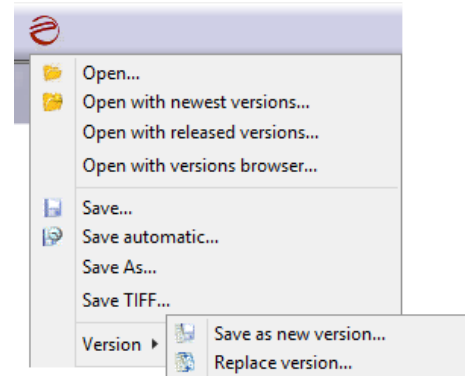
⇒ The lock flag for the selected CAD document is now removed.



## 5 Save: How to save CAD data and changes to PRO.FILE?

For the first-time saving of CAD documents, as well as for the saving of changes, from the local Workcenter folder to PRO.FILE the following functions are available:

- "Save" with the options:
  - [Saving CAD objects for the first time](#)
  - [Save: Saving changed CAD documents](#)
- [Save automatic](#)
- [Save as](#)
- [Only for drawings: Save Tiff](#)



Furthermore, the menu "Version" contains the function

- [Save as new version](#)
- The basic procedure for saving differs depending on whether the CAD data are saved in PRO.FILE for the first time or whether documents from PRO.FILE are saved back after changes have been made to them in Inventor.

Therefore, the description is divided into two chapters:

- [Saving CAD objects for the first time](#)
- [Save: Saving changed CAD documents](#)

The descriptions of the proceeding and possibilities can be found in the following chapters.

Further information on the saving and usage of the Checkin wizard can be found in the manual for "Operation PRO.FILE advanced".



### Note: Manual "CAD design supported by PRO.FILE"

Before using the integration PRO.FILE CATIA V5.NET please also note the manual "CAD design supported by PRO.FILE". This manual describes additional proceedings and related issues from the designer's point of view.



### Note:

To be able to save a CAD document in PRO.FILE, it must have been saved locally before.

## 5.1 Saving CAD objects for the first time

The process of saving is carried out in a number of stages. The outcome of each individual stage determines the dialog that will appear for the next stage.

The following procedure is prerequisite for saving:

- Firstly you must make a local save of your newly created object . This prerequisite is given by CATIA V5.
- Then you can save the object to PRO.FILE.

If you want to save CAD documents from CATIA V5 to PRO.FILE, use the menu entry "Save" from the "PRO.FILE" menu.



### Note:

The description of the processes in connection with PRO.FILE may vary from your actual business situation. This is due to the fact that actions, which are executed after the execution of a command, can be configured differently in PRO.FILE. This particularly applies to the PRO.FILE areas of status administration, part and project assignment, change management and change history



### Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Save"

1. Select the menu "PRO.FILE" from the menu bar.
2. Click on the "Save..." button.
3. The Checkin wizard is displayed supporting you in the saving process.

Saving of new objects in PRO.FILE takes place in three Steps:

- [Checkin wizard Step 1: Creating or assigning a part master record in PRO.FILE](#)
- [Checkin wizard Step 2: Creation of the document description in PRO.FILE](#)
- [Checkin wizard Step 3: Assignment of the created objects to a PRO.FILE project](#)

Furthermore:

- [Notes on first-time saving of assemblies](#)

These steps are described in the following sub-chapters.

## 5.1.1

## Checkin wizard Step 1: Creating or assigning a part master record in PRO.FILE

By default, every CAD document in PRO.FILE is linked to a part master record. The part master record consists of attributes and is used for the creation of bills of materials, for the display of data in the drawing title block, for transfer to an ERP/PPC system, etc.

**Note: Usage of PRO.FILE parts**

If you are not using PRO.FILE parts but only PRO.FILE documents, you can skip this step with the button "Document without part".

In the first step, the assignment of the CAD document to be saved to a PRO.FILE part master record has to be made.

Note: If several CAD documents are being saved, the title bar of the Checkin wizard displays the documents that is currently being handled.

The Checkin wizard offers different options, which can be accessed via the operations bar of the wizard screen:

**Create new**

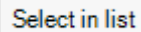
Usage:

- A new part description is to be created for the new document.
- The document to be saved is automatically linked to the new part description afterwards.

Proceeding:

1. Fill in the attributes (fields) for the description of the part master.
2. After entering all required part data, confirm the creation of the part master record in PRO.FILE with <Next>. The new part master record is saved.

## Select in list

A rectangular button with a thin border containing the text "Select in list".

Usage:

- The document to be saved is not to be linked to a new part master record but to an existing part master record.
- It is possible to link several documents to one and the same part master record.

Proceeding:

1. Click the option <Select in list> to select the desired part description.  
⇒ The wizard displays the PRO.FILE surface, as it was opened the last time.
2. If the part master record desired for assignment is not yet displayed in a list or form view, you can use the search functions of the home page, the icon bar or favorites and SmartSearch to start a selection.
3. If the part master record desired for assignment is displayed in a list view, you can now select it. (If the desired part master record is displayed in form view, it is already selected automatically).
4. Confirm your selection with <Next>.

## Search

A rectangular button with a thin border containing the text "Search".

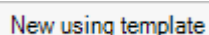
Usage:

- The document to be saved is not to be linked to a new part master record but to an existing part master record.
- The desired part master record is not yet displayed in PRO.FILE and has to be searched for before assignment of the document.

Proceeding:

1. Click the option <Search> to select the desired part description.
2. Enter the search criteria into the displayed search form and click <Search>.
3. Select the desired part master record from the list of results.
4. Confirm your selection with <OK>.

## New using template

A rectangular button with a thin border containing the text "New using template".

Usage:

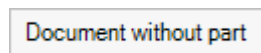
- A new part description is to be created for the new document.

- To make the creation of a new part master record easier, an existing part master record can be used as template with pre-filled fields, which only need to be adjusted.
- Example of usage: You are creating several records for the same screws, only with different lengths. If you do not want to enter the same data over and over again, you can use the function "New using template" and only have to adjust the field "Screw length".
- The document to be saved is automatically linked to the new part description afterwards.

#### Proceeding:

1. Click the option **<New using template>** to select the desired part description.  
⇒ The wizard displays the PRO.FILE surface, as it was opened the last time.
2. If the part master record desired as template is not yet displayed in a list or form view, you can use the search functions of the home page, the icon bar or favorites and SmartSearch to start a selection.
3. If the part master record desired for assignment is displayed in a list view, you can now select it. (If the desired part master record is displayed in form view, it is already selected automatically).
4. Confirm the selection of the part description with **<Next>**.  
⇒ The input form for the creation of the part master record is pre-filled with the data from the selected part master record.
5. Make the necessary adjustments to the pre-filled data.
6. Once all required part data is entered, confirm the creation of the new part master record in PRO.FILE with **<Next>**. The new part master record is saved.

#### Document without part



#### Usage:

- For special usage purposes it may be necessary to create a document description without the link to a part master record.
- You can therefore use this option to skip the creation or selection of the part master record and to proceed directly with the saving of the document description.

#### Proceeding:

1. Click the option **<Document without part>**.  
⇒ The Checkin wizard for parts is skipped. The Checkin wizard for the document description is displayed.

**Attention:**

If the creation of a part master record is skipped and only a document is created, the saved CAD document will not be available for bills of materials and no information is transferred to ERP systems.

## 5.1.2

**Checkin wizard Step 2: Creation of the document description in PRO.FILE**

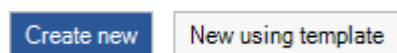
All files saved in PRO.FILE are generally stored under the object type "Document".

In order to save the CAD file now to PRO.FILE, the corresponding document description must be entered to describe and classify the CAD document and to make it available for further usage.

For this, the Checkin wizard for the document description is available:

Note: If several CAD documents are being saved, the title bar of the Checkin wizard displays the documents that is currently being handled.

Here, too, the Checkin wizard offers different options that can be accessed via the operations bar:



- Create new
- New using template

Usage and proceeding for these options are the same as for the assignment of the part master record, only that these functions here relate to the document description.

For detailed information see the previous chapter "[Checkin wizard Step 1: Creating or assigning a part master record in PRO.FILE](#)".

- After the finalization of your entries confirm the saving of the CAD document and the assignment to the desired part master record with <Next>.

- The CAD document is now saved in PRO.FILE.

The Checkin wizard now continues with the options of assigning the newly created objects to a PRO.FILE project.

### 5.1.3

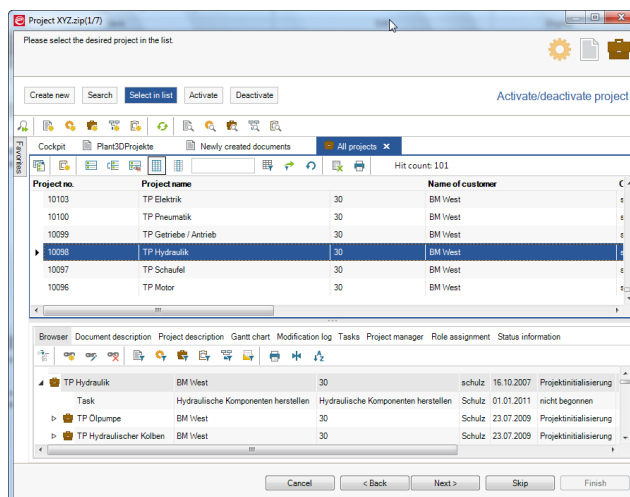
## Checkin wizard Step 3: Assignment of the created objects to a PRO.FILE project

In this step the CAD data just saved can be assigned to a specific PRO.FILE project.



### Note: Usage of PRO.FILE projects

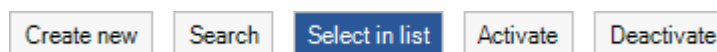
The third step in the Checkin wizard is intended for the use of PRO.FILE projects. If you are not using PRO.FILE projects, you can skip this step with the option <Skip>.



For this project assignment of the newly created document description (and, if created, the new part master record) an existing project must be selected, or a new project must be created.

Note: If several CAD documents are being saved, the title bar of the Checkin wizard displays the documents that is currently being handled.

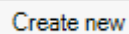
Here, too, the Checkin wizard offers different options that can be accessed via the operations bar:



### Attention: Project must be activated

In order for a CAD document to be linked to a project, this project must be ACTIVATED. This means that for all of the following options, you have to select the option "Activate" afterwards

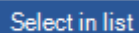
The proceeding for these options is the same as for the first two steps of the Checkin wizard:

**Create new**A rectangular button with a light gray border and the text "Create new" in a blue font.

A new project is created in PRO.FILE. The part master record and document description created in steps 1 and 2 are assigned to this new project.

**Search**A rectangular button with a light gray border and the text "Search" in a blue font.

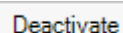
The part master record and document description created in steps 1 and 2 are to be assigned to an existing project. This project is now searched via the search form and selected.

**Select in list**A rectangular button with a dark blue background and the text "Select in list" in a white font.

The part master record and document description created in steps 1 and 2 are to be assigned to an existing project. This project is already displayed in a PRO.FILE list and only has to be selected and confirmed.

**Activate**A rectangular button with a light gray border and the text "Activate" in a blue font.

If a project is activated, all new parts and documents in PRO.FILE are automatically assigned to this project. If no project is currently activated, and you want to do so, you can use this function to activate a project.

**Deactivate**A rectangular button with a light gray border and the text "Deactivate" in a blue font.

Again: If a project is activated, all new parts and documents in PRO.FILE are automatically assigned to this project. If this assignment is not to be made for the current document, you can deactivate the project before finalizing the saving process.

Note: If a project is activated, this is displayed in the title bar of the active PRO.FILE window.

**Proceeding:**

For the assignment of new CAD to a project via the Checkin wizard proceed as follows:

1. Select an existing project or create a new one.
  2. Select this project in the list view (project in form view are automatically selected).
  3. You now must select "**Activate**". Only if the selected project has been activated, the assignment to the project is made after confirmation.
  4. Confirm your proceeding with <**Finish**>.
- ⇒ The saving of the CAD data in PRO.FILE is now finished.



## 5.2 Notes on first-time saving of assemblies

When a new assembly is saved in PRO.FILE, the following points need to be noted:

- **Saving a new assembly**

If you want to save a just created assembly with also just created parts to PRO.FILE you have got different ways available. Either you first save every single part and subsequent the assembly or you choose the command <Save> out of the assembly. Thereby the initial point concerning the command will be the knot within the CATIA V5 product structure which is currently active.



### Attention:

It is not possible to open a part from out of an assembly in CATIA V5, save it separately to PRO.FILE and close it afterwards. The connection between the part and the assembly will be lost in such case! Therefore you should save parts in the context of those assemblies they belong to!

- **Saving assemblies that contain unloaded or non-existent files**

It is possible to save and/or modify assemblies with unloaded or non-existent files to PRO.FILE. Such procedures must be explicitly approved; this is carried out in the relevant configuration parameters in the PRO.FILE Management Console.

If the user tries to save an assembly with an unloaded or non-existent file, a list containing the relevant files will be displayed.

## 5.3 Save: Saving changed CAD documents

After changes have been made to a CAD document opened from PRO.FILE you can use the function "Save" to save your changes back to PRO.FILE.

If you use "Save" for objects already existing in PRO.FILE, the object in PRO.FILE is changed.

Before the saving process is initiated, PRO.FILE checks whether the user has the right to change the corresponding object in PRO.FILE. Furthermore, the program checks whether the used copy of the PRO.FILE object is up to date.

You can choose between the following functions to save changes to a CAD document to PRO.FILE:

- [Save: Saving changed CAD documents](#)
- [Save automatic](#)
- [Save as](#)
- [Only for drawings: Save Tiff](#)
- [Save as new version](#)



### Attention: Only documents that have been locked can be saved

PRO.FILE blocks concurring changes during the work with the CAD system. It is therefore important to make sure that the objects are locked for other users. For this, the function "Lock" is available, offering the user exclusive access to the document and allowing the user to save back his/her changes.

If the document has not been locked and has been modified by a different user in the meantime, who has changed back his/her changes to PRO.FILE, your changes cannot be saved back to PRO.FILE. See chapter "[Lock/Unlock: Who can change when?](#)".



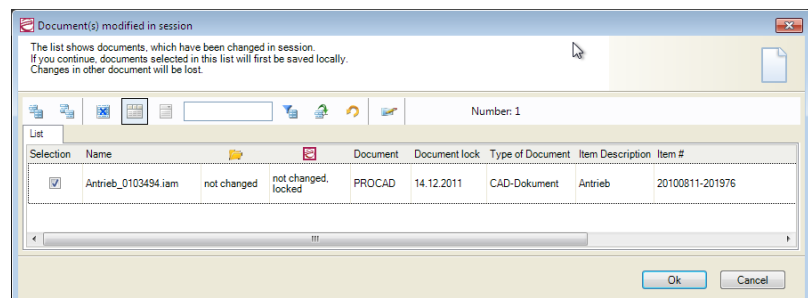
### Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Save"

Proceed as follows

1. Go to the integration menu "PRO.FILE" in CATIA V5.
  2. Select the function "Save".
- ⇒ PRO.FILE recognizes the CAD document as a PRO.FILE object and automatically goes into change mode.

⇒ The dialog for the selection of CAD documents to be saved is displayed.

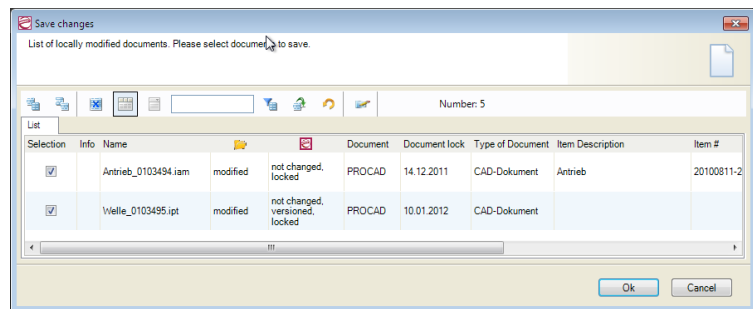


- ⇒ The dialog displays a list with all changed CAD documents from the current CATIA V5 session. (Information on the functions and status information can be found in the chapter "[Data overview: The document list](#)").
- ⇒ For assemblies, the structure is analyzed for changed CAD documents and the list of all documents of this assembly is preselected.
- ⇒ For this list the access permissions for saving the changes of the user are checked. (If the CAD document had been locked before for editing, this prerequisite is fulfilled.)
3. Select all documents you want to save in PRO.FILE. To do so, activate the checkboxes for the desired documents.
  4. Confirm your selection with <OK>.



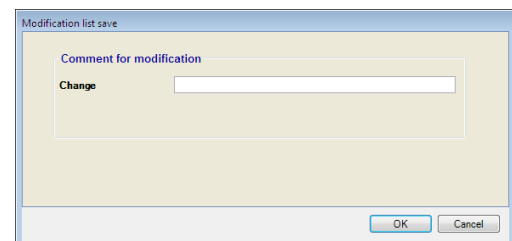
PRO.FILE now checks whether the structure to be saved contains documents that have been changed locally and have not yet been saved to PRO.FILE. If locally changed documents are found, an additional query is displayed.

5. Select all locally changed components you want to save to PRO.FILE.



⇒ The changes are then saved. The CAD document previously saved in PRO.FILE is overwritten.

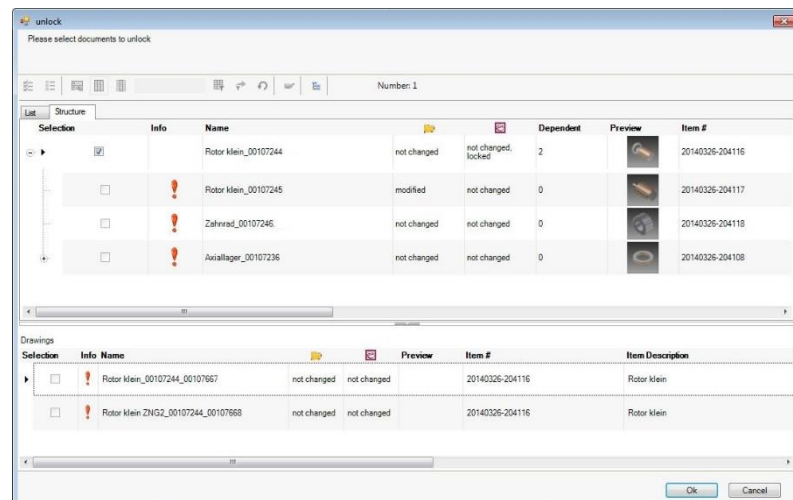
6. Depending on the configuration and PRO.FILE status, you now have to enter a modification comment. Enter the comment information into the fields on the dialog screen.



7. Confirm your modification comment with <OK>.

⇒ The modification comment screen is closed; your modification comment can now be found in the "Modification list" in PRO.FILE.

8. The dialog for documents to be unlocked after saving is displayed. (Information on the functions and status information can be found in the chapter "[Data overview: The document list](#)").



⇒ If documents from PRO.FILE had been locked for editing in CATIA V5, this lock is not automatically removed after saving. The documents remain locked and cannot be changed by other users.

⇒ If you are finished with your changes to the CAD document, you can now unlock the document to make it available for other users.

⇒ To make this process easier, the PRO.FILE CAD documents that are still locked are displayed in the list.

9. To make the documents available again for other users, select the documents in the list. To do so, activate the checkboxes for the desired documents.



10. Confirm your selection with <OK>.

⇒ The lock flag for the selected documents is now removed.

⇒ The saving of your changes to PRO.FIL is now finished.

## 5.4 Save automatic

Apart from the already described menu function <Save> the integration offers the function <**Save automatic**>, which is a very comfortable way of saving documents to PRO.FILE:

"**Save automatic**" allows the automatic creation of documents and parts in PRO.FILE without additional queries.

"**Save automatic**" for documents that are newly saved to PRO.FILE:

- The classification via the Checkin wizard is only made for the first part and document description in PRO.FILE.
- For all further CAD documents to be saved no Checkin wizard is displayed. Document and part descriptions are saved automatically in PRO.FILE.
- Without further query means: The document and part descriptions are not filled in manually. The data record contains only the information that have been pre-configured in the saving form or that are automatically handed over from the CAD system to the saving form for documents that have been opened from PRO.FILE for editing:
- If documents have been opened from PRO.FILE for editing, the data in PRO.FILE is without query overwritten with the modified status of the data. For changed PRO.FILE documents "Save automatic" is identical to the proceeding for the saving of changed documents.

### "Save automatic" for complete assemblies

When an assembly is opened within the CATIA V5 session, and all components of this assembly are to be saved in PRO.FILE, the entire assembly can be saved in PRO.FILE with the function "**Save automatic**".

If this assembly contains parts that are not yet saved in PRO.FILE, a part master record is created automatically and without query for each part.



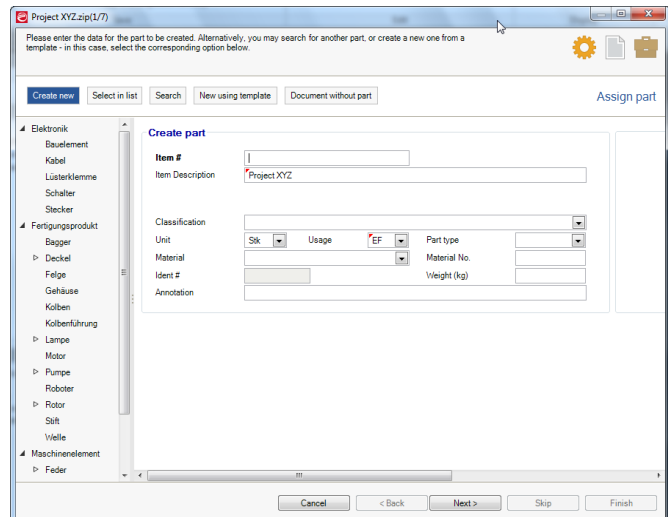
### Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Save..." => "Save automatic"

Proceed as follows

1. Select the "PRO.FILE" menu from the menu bar in CATIA V5.
  2. Select the function "Save automatic" from the menu area "Save...".
- ⇒ For the first document that is unknown to PRO.FILE, the normal saving process is started. The following is displayed:

- The Checkin wizard to create the part description in PRO.FILE.
- Then (depending on the configuration) the Checkin wizard to create the document description in PRO.FILE.
- Finally, (depending on the configuration) the Checkin wizard for the project assignment in PRO.FILE.



3. Go through all steps of the Checkin wizard for the first new CAD document. Detailed information on this can be found in the previous chapter "[Saving CAD objects for the first time](#)".
- ⇒ All further CAD documents are now saved automatically to PRO.FILE, without having to fill in the document and part descriptions. A project assignment is only made if a project is already activated in PRO.FILE.
4. For modified components already known in PRO.FILE:  
If the assembly also contains components that are already saved in PRO.FILE and that have been changed in the session or locally, these components are also offered for saving.  
Select the components to be saved and confirm your selection.  
Detailed information on the saving of changes to PRO.FILE can be found in the previous chapter "[Save: Saving changed CAD documents](#)".



### Note:

"Save automatic" is identical to "Save", with the difference that the metadata is only queried for the first document and part.

- ⇒ As result of the action <**Save automatic**>, a part and document description is created for each CAD document in PRO.FILE, including the correct structure of the assembly and the bill of materials
- ⇒ The process "**Save automatic**" is now finished.

**Attention:**

When CAD documents (sub-assemblies, parts, drawings) are saved automatically, no values are entered manually in the Checkin wizard. The fields remain empty.

This also applies to fields that are configured as required fields. As a consequence, all elements saved with "Save automatic" have to be classified at a later point in time, especially if these fields are required by other systems (ERP interface)

**Attention: ERP interface and "Save automatic"**

When using the function "Save automatic", it may happen that fields, that are required by your ERP interface, are not filled! This may lead to problems during the forwarding of documents/parts to your ERP system. Please check the fields in the different forms for completeness.

It is possible to have specific fields filled automatically by the system. With this you can make sure that important fields are automatically provided the required information. For detailed information see the configuration manual for the Integration PRO.FILE – CATIA V5.

## 5.5

### Save as

With this function you can use entire assemblies as a template in order to create a new project with it. The original documents are not referenced by the resulting documents, which are simply copies of them:

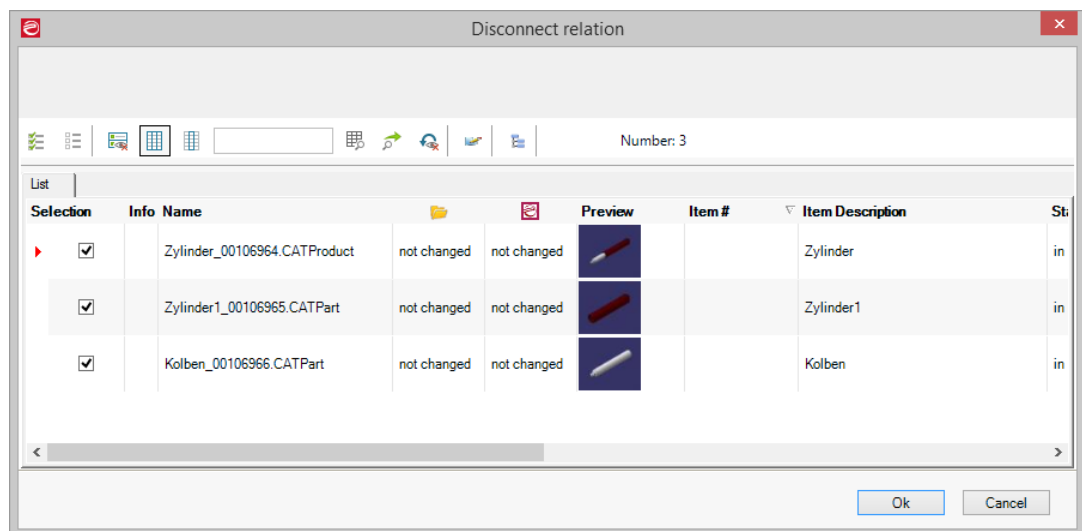
- This function saves the entire object with all of its contained elements as a new object in PRO.FILE.
- The document descriptions and, if need be, the part masters of the originals are used as a template for the new document/part masters.
- You can either manually or automatically create the new elements (see "Save" and "Save automatically").
- During this process, the drawings that belong to each element will also be identified. These will also be included in PRO.FILE as new elements and linked to the corresponding products/parts.

**Note:**

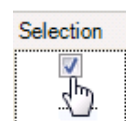
It is possible to change the part number and the instance names of the CATIA V5 files using the Save As function. The configuration in the PRO.FILE Management Console must be taken into account for this.

Proceed as follows:

1. Select the "PRO.FILE" menu in CATIA V5.
  2. Select the function "Save as".
- ⇒ The integration now checks the currently loaded file in CATIA V5.
- ⇒ A check is made for all documents belonging to this structure whether drawings have been saved for them in PRO.FILE. If this is the case, these drawings are taken into account in the structure and the further proceeding.
- ⇒ PRO.FILE displays a list, based on the structure detected in the previous step. Here you can select the documents, you want to continue using:



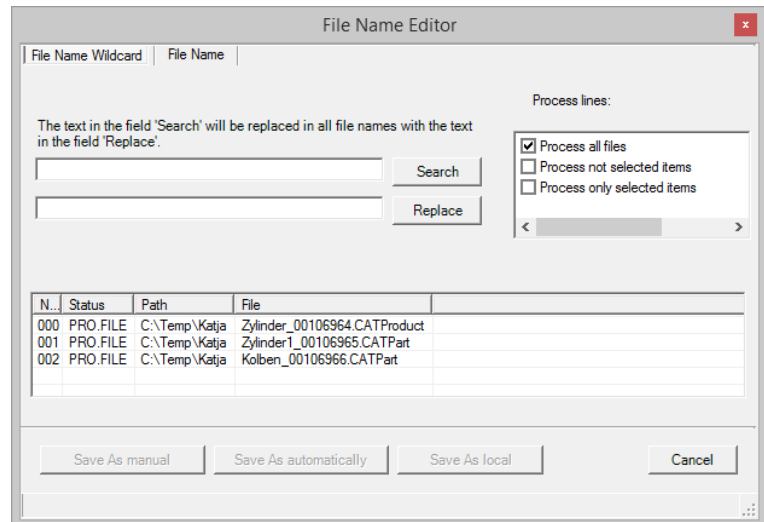
3. Select the desired documents you want to include into the "Save as" procedure.
4. Confirm with <OK>.



⇒ The selected documents are displayed in an editor.

5. You can allocate new file names from here.

These lists also contain the drawings for the documents; this enables the user to edit the names accordingly.



- ⇒ **Attention:** Only those documents that have had the names edited will be considered when working from now on. These documents are marked in a different color.
6. There are 3 possible variations that you can choose from in the editor:
- **Save As manual:** All selected documents are saved first locally, and then saved to PRO.FILE. The save to PRO.FILE is carried out analogous to the procedure in the menu Save, this means that all of the document- and part-master data is requested, when doing this the fields will be filled with the values of the original documents from PRO.FILE.
  - **Save As automatic:** All selected documents are saved first locally, and then saved to PRO.FILE. The save to PRO.FILE is carried out analogous to the procedure in the menu save automatic, this means that no document- and part-master data is requested. The fields will be filled with the values of the original documents from PRO.FILE.
  - **Save As local:** All selected documents are only saved locally.
7. Click on the desired option to start the function.
- ⇒ During the save procedure, the active document is saved first followed by all drawings one by one.
- ⇒ The procedure for "Save as" is thus finished.



## 5.6 Only for drawings: Save Tiff

The integration PRO.FILE CATIA V5.NET offers the user the option to convert a CATIA V5 drawing into a neutral data format (NDF), e.g. TIFF and to save it as a document. By using the function "Save TIFF" a neutral format document is created.

This NDF document is then automatically linked to the document description of the drawing.

3333	Fahrerhaus	St33	PROCAD	23.09.2011	in Bearbeitung	203742
CAD-Dokument	106295	Fahrerhaus	Lockomotive	mi	PROCAD	23.09.2011
NDF-2D	106308		tif	PROCAD	28.09.2011	freigegeben
NDF-2D	106311		pdf	PROCAD	28.09.2011	Produktion

This function is only available for drawings, so the menu entry is only displayed in drawing mode.



### Note: Save NDF only with "Format generators"

The function "Save TIFF" is not a standard feature in PRO.FILE. This means that you need a license to use this function. Furthermore, specific installations (converter) and configurations may be required on your system. See the manual "PRO.FILE Format Generators".



### Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Save TIFF..."

To create a neutral data format document proceed as follows:

1. You have opened a drawing and wish to document the current drawing status.
2. Select the "PRO.FILE" menu from the menu bar in Inventor.
3. Select the function "Save TIFF".
  - ⇒ A dialog is displayed asking, whether you wish to create the neutral format.
4. Confirm with <Yes>.
  - ⇒ The neutral format is now generated.
  - ⇒ Depending on the configuration the action is performed without further interaction.
  - ⇒ The created NDF document is automatically saved in PRO.FILE and linked with the document description of the drawing.
  - ⇒ The creation of the neutral target format for the drawing is now finished.

**Note: Client-side NDF and server-side NDF**

If, apart from the manual creation of NDF documents, you also want to use the automatic server-side creation of NDF documents via workflow status, please make sure to configure the NDF documents according to your requirements (overwrite or version).

**Change management via NDF generation**

With the configuration of the NDF generation via the format generators you can control the behavior of PRO.FILE regarding the handling of TIFF documents. You can configure that an existing TIFF document is not overwritten but versioned when the function "Save TIFF" is used. This way, changes can actively be documented by the design engineer.

## 5.7

**Save as new version**

With the PRO.FILE- CATIA V5.NET Integration it is possible to create different versions during saving of CAD objects.

**Note:**

A version can always be created from the newest version in the version list. Whether the creation of a version from older version is allowed as well depends on the configuration of the parameter "Allow creating a version from an old version" in the PRO.FILE Management Console.

If the function "**Save as new Version**", is called up a copy will be created of the PRO.FILE CAD object, and this new version will increase the version/ revision counter.

- Only the document active in the CAD session is versioned.
- The old version remains saved in PRO.FILE.
- The new version is saved with a new document ID in PRO.FILE and displayed in CATIA V5.
- If a part is versioned in this way using PRO.FILE- CATIA V5.NET Integration, the new version of the part is always saved "before" the most current version. The references of assemblies in higher hierarchies will continue to indicate the older version – until the assembly is saved in PRO.FIL. The assembly structure is then also updated in PRO.FILE.
- If an assembly is versioned using the "Save as new Version" function, the tree structure of the assembly will be built using the currently loaded parts. Multi-layered assemblies must also be versioned layer by layer from bottom to top.

**Function call from the PRO.FILE menu in CATIA V5:**

"PRO.FILE" => "Version" => "Save as new version"

Proceed as follows:

1. Select the "PRO.FILE" menu from the menu bar in CATIA V5.
2. Select the function "Version" => "Save as new version".  
⇒ A list with all documents, of which a new version will be created, is displayed.
3. Confirm with <OK>.  
⇒ A new version of the active CAD documents is now created in PRO.FILE.  
⇒ A message box confirms the successful creation of the version.  
⇒ The new version is displayed in CATIA V5.

**Attention: New version is not locked**

The new version created with the function "Save as new version" is not locked in PRO.FILE. To lock the document, please use the function "[Lock/Unlock: Who can change when?](#)".

The document list always displays the most recent version. To display older versions/revisions of a document, you can use the function "PRO.FILE" => "Show" => "all document versions" from the integration menu.

**Note: Versions of drawings**

It has to be noted that the drawing of a part, that has been versioned with the function "Save as a new version", is not automatically versioned in PRO.FILE.

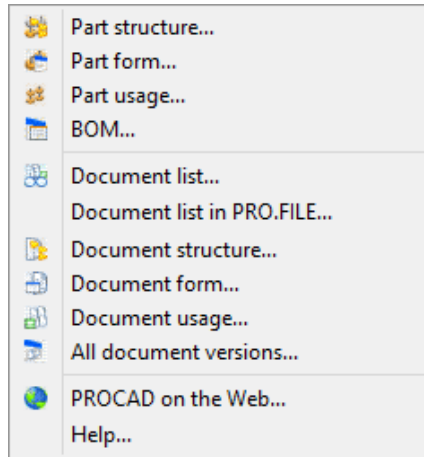
Example: You have created a part and a drawing of the part in CATIA V5. You load the part and save it as a new version. For the drawing of the part no new version is created!

**Note: manual "CAD design supported by PRO.FILE"**

For details on the version concept of the integration, please see the manual "CAD design supported by PRO.FILE".

## 6 Show: PRO.FILE Information at a glance

The area "Show" of the integration offers different functions that give you direct access to PRO.FILE information on the current CAD drawing.



Depending on which details you are interested in, you can use the different menu entries to access the information.

This way, you can immediately access information stored in PRO.FILE on properties and classifications of the active CAD document as well as structure and usage information.

The following information can be found in the area "Show":

- [Data overview: The document list](#)
- [Show: Information on a CAD document in PRO.FILE](#)
- [Direct information in the dialog screens](#)
  - [More comfort: search and list functions in the dialog screens](#)
  - [Up to date or not: Display of status information](#)

Detailed information can be found in the following sub-chapters.

## 6.1 Data overview: The document list

The document list displays the PRO.FILE information on the currently active CAD data. With the function "Document list" you can also see which documents (part drawings) are currently used in your (main) drawing.



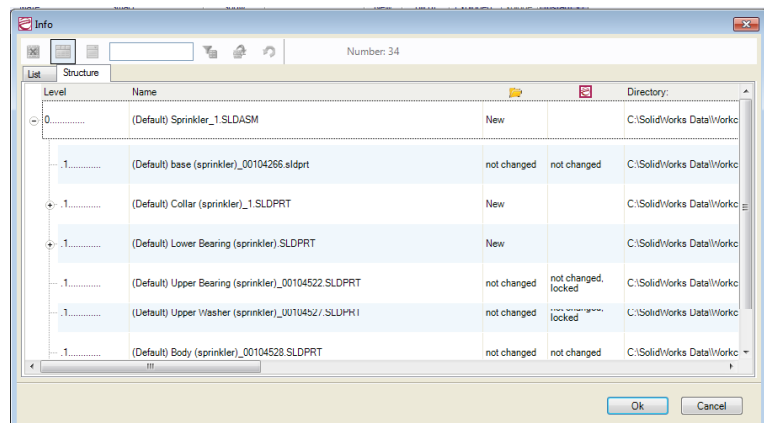
Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Show" => "List of documents"

When the function is used, the document list is displayed:

You find the following information:

- The data from the PRO.FILE document description..



- Information regarding the status of the currently active CAD document. If you have not locked all CAD documents directly after opening, it is recommended to view the status information via the document list before making your changes. If the document is no longer marked "unchanged" it will not be possible without problems to save your changes back to PRO.FILE.

The document list also contains – as all other dialog screens of the integration do – different search and list functions.

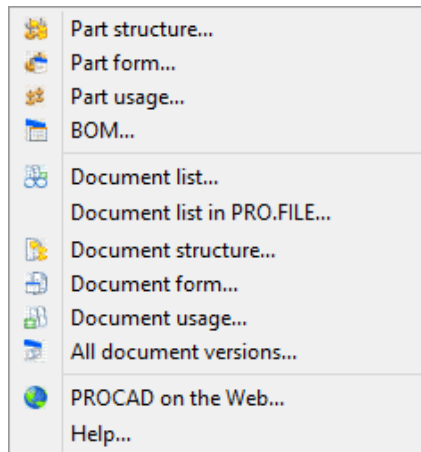
Detailed information can be found in the following chapters:

- [Direct information in the dialog screens](#)
- [Up to date or not: Display of status information](#)

## 6.2

## Show: Information on a CAD document in PRO.FILE

The area "Show" of the PRO.FILE integration contains different functions for the display of information on part master data and document descriptions in PRO.FILE.



- These menu entries access information on the CAD document currently active in CATIA V5.
- The various menu entries allow a targeted access to frequently needed information, without having to navigate in PRO.FILE.
- After a function is selected, PRO.FILE opens and displays the desired information.
- Within the displayed lists and forms, you can perform all actions available in PRO.FILE.



Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Show" => "..."



**Note:**

Please note that you can only use these display functions if the CAD document is already saved in PRO.FILE.

If you have made changes to your CAD document and have not yet saved these changes back to PRO.FILE, these changes are ignored by the display functions.

The following display options are available:

**Part structure**

With the function "Part structure" PRO.FILE displays the part the current CAD document is attached to and other parts used within the CAD structure.

**Part form**

The function "Part form" displays the part master record form of the part the current CAD document is attached to in PRO.FILE.

**Part usage**

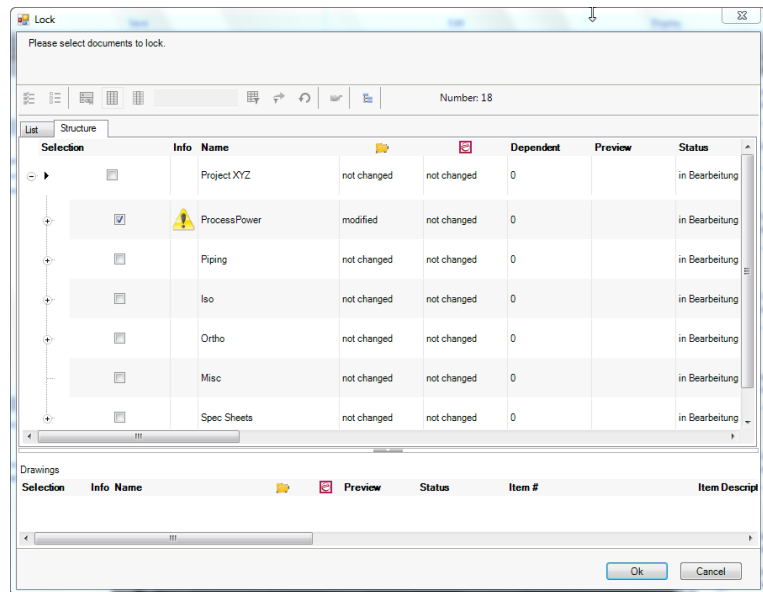
With the function "Part usage" you can see whether your current CAD document is used by other assemblies.  
The usage list displays the "upward" structure.

<b>Bill of materials</b>	The function "Bill of materials" displays the PRO.FILE bill of materials for the active drawing.
<b>Document list</b>	The document list shows an overview of PRO.FILE information on the currently active CAD data. Detailed information on this can be found in the previous chapter " <a href="#">Data overview: The document list</a> ".
<b>Document list in PRO.FILE</b>	With the function "Document list in PRO.FILE" PRO.FILE is started and displays all CAD data currently loaded in CATIA V5 in a list. Contrary to the display option "Document list", no separate window is started in PRO.FILE, but the default list view.
<b>Document structure</b>	With the function "Document structure" you can see which documents (= part drawings) are used in your drawing (= main drawing).
<b>Document form</b>	The function "Document form" displays the document description of your current CAD document in the PRO.FILE form view. Here you can find the specification of the document-describing data for this CAD document.
<b>Document usage</b>	With the function "Document usage" you can see whether the document description of your active CAD document is used in other document or part descriptions.
<b>All document versions</b>	The function "all document versions" displays all visible current and old versions of your CAD document.
<b>PROCAD on the WEB</b>	Via this menu entry you can access the PROCAD web site.
<b>Help</b>	This menu entry opens the PRO.FILE online help.

## 6.3 Direct information in the dialog screens

For the functions "Lock", "Managed Copy", and "Disconnect relation" dialog screens are displayed.

These offer the following possibilities:



- You can switch between list and structure view.
- Via the buttons of the list functions you can make searches and filter the results (see the following sub-chapter "[More comfort: search and list functions in the dialog screens](#)").
- If objects are listed in the assembly tree above the actual node (e.g. drawings), these objects are displayed in a dependent window.
- The lists contain status information for each of the listed objects (see the following sub-chapter "[Up to date or not: Display of status information](#)").
















### 6.3.1 More comfort: search and list functions in the dialog screens

The dialog screens of the PRO.FILE integration contain different search and list functions, as known from the PRO.FILE GUI:



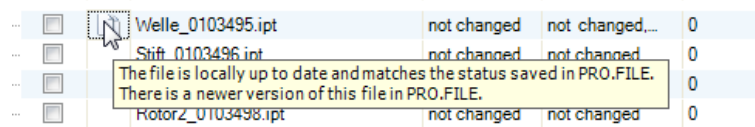
Via these buttons, the following functions are available:



-  **Select all rows:**  
With this button, all rows of a list are highlighted.
-  **Invert selection:**  
With the <Shift> key pressed down, it is possible to select whole areas of a list, with the <Ctrl> key pressed down, you can select several individual rows. The button "Invert selection" can be used to select everything that is not selected and unselect everything that was selected.
-  **Hide selected rows:**  
If several rows of a list are selected, these rows can be hidden from the list with this button.
-   **Search in all columns / Search in active columns:**  
In order to be able to perform a targeted search for terms in the list, the user first has to select whether the search is to be carried out across all columns in the list or only for a specific column in the list.
  - : The search is performed across all columns in the list.
  -  : The search is performed for the active column only. A column is activated by clicking the respective column header.
-  **Define Filter pattern / Filter:** A character string can be entered into the entry field located within the icon bar. Here you can use the already described wildcards/meta characters.  
The search for the entered character string is started using the  icon.  
If the search pattern is found, all matching data records are highlighted.
-  **Next found pattern:** This icon is used to once again compare the entered filter pattern with the columns that are to be searched. The next data record found is highlighted.
-  **Show hidden rows:**  
If rows of a list have been hidden, this button can be used to display them again.
-  **PRO.FILE list selection:**  
The entries of the selected rows are selected and opened in a list in PRO.FILE. This way you can immediately view the stored information without further selection.

### 6.3.2 Up to date or not: Display of status information









The document list - as all other dialog screen of the integration - contain three columns for displaying the status of the CAD data:




- Info: Shows an icon for the data status. If you hover over the icon with the mouse pointer, a tool tip with more information is displayed.



- This information relates to the lock/unlock option. It is only displayed in windows with a relation to the locking/unlocking.
- : Displays the status of the CAD data in the local work folder of the Workcenter.
- : Displays the status of the CAD data in PRO.FILE.

These columns may contain the following:

Info	Local 	PRO.FILE 	Description
	new	unknown	The file is new and unknown in PRO.FILE.
	unchanged	unknown	The file is locally unchanged but comes from a different instance of PRO.FILE and can therefore not be saved back to the current instance.
	changed	unknown	The file is locally changed but comes from a different instance of PRO.FILE and can therefore not be saved back to the current instance.
	unchanged	unchanged locked	The file is locally unchanged and exists in the same form in PRO.FILE. The file is locked by a different user and can therefore not be saved back.
	unchanged	unchanged versioned	The file is locally unchanged and exists in the same form in PRO.FILE. There is a newer version of this file.
	unchanged	unchanged locked versioned	The file is locally unchanged and exists in the same form in PRO.FILE. There is a newer version of this file. The file is locked by a different user and can therefore not be saved back.

Info	Local 	PRO.FILE 	Description
	unchanged	changed	The file is locally unchanged but has been modified in PRO.FILE after it has been copied locally and can therefore not be saved back.
	unchanged	changed locked	The file is locally unchanged but has been modified in PRO.FILE after it has been copied locally and can therefore not be saved back.
	unchanged	changed versioned	The file is locally unchanged but has been modified in PRO.FILE after it has been copied locally and can therefore not be saved back. There is a newer version of this file in PRO.FILE.
	unchanged	changed locked versioned	The file is locally unchanged but has been modified in PRO.FILE after it has been copied locally and can therefore not be saved back. There is a newer version of this file in PRO.FILE.
	changed	unchanged	The file is locally changed but has not yet been saved back to PRO.FILE.
	changed	unchanged locked	The file is locally changed. It is locked by a different user and can therefore not be saved back. Local changes may get lost.
	changed	unchanged versioned	The file is locally changed but has not yet been saved back to PRO.FILE. There is a newer version of this file in PRO.FILE.
	changed	unchanged locked versioned	The file is locally changed. It is locked by a different user and can therefore not be saved back. Local changes may get lost. There is a newer version of this file in PRO.FILE.
	changed	changed	The file has been modified both locally and in PRO.FILE. The local changes cannot be saved back.
	changed	changed locked	The file has been modified both locally and in PRO.FILE. The local changes cannot be saved back.
	changed	changed versioned	The file has been modified both locally and in PRO.FILE. The local changes cannot be saved back. There is a newer version of this file in PRO.FILE.
	changed	changed locked versioned	The file has been modified both locally and in PRO.FILE. The local changes cannot be saved back. There is a newer version of this file in PRO.FILE.

## 7 Functions for the version administration

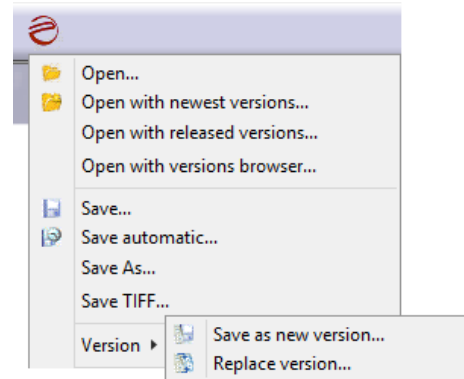
The integration PRO.FILE CATIA V5.NET offers several functions for opening and saving when working with versions:

- [Open with newest and released versions of the components](#)
- [Open with version browser](#)
- [Save as new version](#)

These functions have already been described in previous chapters.

Furthermore, there is the function:

- [Replace version](#)



Information on this can be found in the following sub-chapter.



### Note: manual "CAD design supported by PRO.FILE"

For details on the version concept of the integration, please see the manual "CAD design supported by PRO.FILE".

### 7.1 Replace version



#### Attention: Undo not possible!

By using the function "Replace version" the current structure of the concerned objects is changed. It is not possible to restore the objects in the previous state!

The command "Replace Version" allows an existing, built-in version of a CAD object to be replaced by a new version for all assemblies in which it is used.

In the design process an object may be locked due to feedback from the testing department. This object is no longer to be used. You as a designer now have to adjust all designs.

For this, you can use the function "Replace version".

- Via the function "Replace version" all assemblies are searched, in which the predecessor version of the current part is used (referenced). The reference is then changed to point to the new version of the part.

- PRO.FILE then creates a special document list, in which all documents are listed that are referencing to the old version of the part. You can now select, which assemblies are to be updated. The CAD info "used x times" indicates how often this part is used in other assemblies.
- In all selected assemblies the dependencies are replaced by a reference to the currently active object.
- Before a component is replaced in an assembly, PRO.FILE checks, whether the user has the permission to change this assembly.



#### Function call from the PRO.FILE menu in CATIA V5:

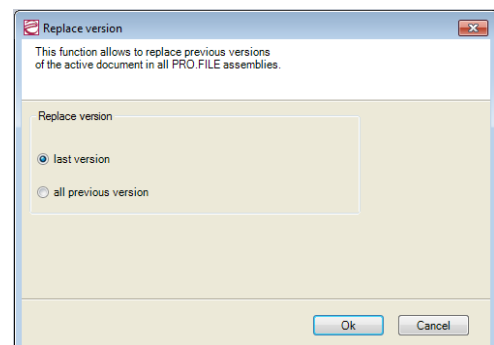
"PRO.FILE" => "Version" => "Replace version"

When the function "Replace version" is used, a check is made based on the PRO.FILE IF of the active CAD document regarding its usage: this check detects in which assemblies and drawings the active document is used.

You can then replace the old version of the active PRO.FILE document in all corresponding assemblies and drawings with the new version.

Proceed as follows

1. Load the new version of the document, with which you want to make the replacement, from PRO.FILE in CATIA V5 (open the replacing document, not the document to be replaced).
2. Select the function "PRO.FILE" => "Version" => "Replace version".
3. Select now, which of the predecessor versions is to be replaced by the new version:
  - Only the direct predecessor version, wherever it is used.
  - All predecessor versions, wherever they are used.



- ⇒ You now get a list of how often and where the predecessor version(s) of the document is/are used.
- ⇒ Select all records, for which a replacement is to be made.
- ⇒ Confirm your selection with <OK>.
- ⇒ The version is now replaced: The currently loaded version is then used by all selected assemblies/drawings.
- ⇒ You thus have cleaned all concerned objects.

- ⇒ If you have not modified all object, you can repeat this action. You then receive a list of all objects using the old version of the component (minus the objects already modified).

**Attention:**

If a part is used in different assemblies, each assembly has to be updated with this function. If the part is used in many complex assemblies and in different versions, this may lead to a certain amount of work to be done.

## 8 Additional Functions to edit Drawings and Assemblies

The integration of PRO.FILE CATIA V5.NET offers the user various functions which may only follow in connection with certain CAD objects.

For CAD object the following functions are available:

- [Disconnect relation](#)
- [Properties update](#)

When an assembly is active, you can also access the following functions:

- [Insert component](#)
- [Replace component](#)
- [Create BOM](#)

When a drawing is active, you can also access the following function:

- [Update title block](#)

### 8.1 Disconnect relation

By the classification of CAD objects with PRO.FILE you have an excellent tool for the administration of your designed models and documents. This "Knowledge base" can be used for new designs. With correct classification you can find similar objects for new designs in PRO.FILE and use them for new designs or modifications.

You thus avoid tedious redesigning of the object. In order for the existing object to become independent from the objects in PRO.FILE, you can use the function "Disconnect relation".

This function removes the database connection of a part, drawing or assembly from the PRO.FILE database. The CAD objects are then treated as local CAD objects and behave like newly designed objects.



Function call from the PRO.FILE menu in CATIA V5:

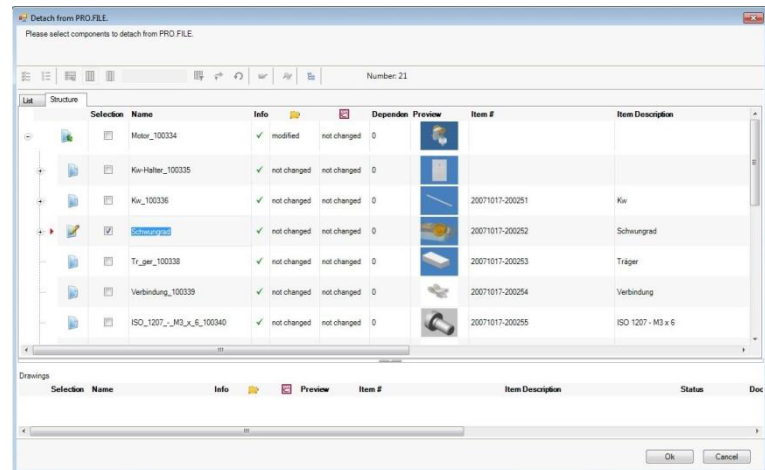
"PRO.FILE" => "Disconnect relation"

Proceed as follows:

1. Select the "PRO.FILE" menu in CATIA V5.
2. Select the function "Disconnect relation".

- ⇒ The dialog for the selection of documents to be disconnected is displayed.

(Information on the functions and status entries can be found in the chapter "[Data overview: The document list](#)").

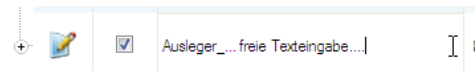


3. Select all CAD documents, the PRO.FILE connection of which you want to dissolve by using the checkboxes.



- ⇒ The previous file name without ID, or with the suffix "\_Index" is suggested as new file name, e.g. "assembly\_1".

4. Give the document to be disconnected a name under which it is to be saved locally. You can edit this name freely in the "Name" column.



5. Once all files are selected and renamed, confirm with <OK>.

- ⇒ The PRO.FILE connection for all selected CAD components is now removed. The existing objects are locally removed from the CAD session and the new objects are loaded.

- ⇒ The disconnection of the objects is thus finished. A check via the function "File" => "Properties" shows that all objects are displayed as if newly designed.

- ⇒ The selected CAD files are now saved locally and no longer have a PRO.FILE connection. Changes to these documents are not saved to PRO.FILE!

- ⇒ To save these CAD files again to PRO.FILE see chapter "[Save: How to save CAD data and changes to PRO.FILE?](#)".



#### Note:

If the database connection of an assembly is dissolved, the document relation of parts contained in this assembly remain intact. You only have disconnected the assembly description.



## 8.2 Properties update

When metadata in PRO.FILE is linked to properties in a product or part, this information can be updated via this function.



Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Properties update"

## 8.3 Insert component

Via the function "Insert component" parts or assemblies saved in PRO.FILE are inserted into the assembly in CATIA V5.

With the PRO.FILE Checkout wizard you select the CAD document, which is then placed in the assembly.

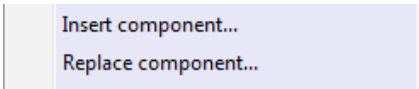


Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Insert component"

Proceed as follows:

1. Activate the assembly, into which the component is to be inserted, with a double click in CATIA V5.
2. Select the "PRO.FILE" menu in CATIA V5.
3. Select the function "Insert component".



Insert component...  
Replace component...

⇒ You are then prompted in the Checkout wizard to select the component in PRO.FILE, which you want to insert into the assembly.

4. Select the desired CAD document and click <Open>.

⇒ Detailed information on the Checkout wizard can be found in the chapter ["Working with the Checkout wizard to search for CAD documents"](#).

5. The positioning is made automatically by CATIA V5.

⇒ The component from PRO.FILE is now inserted into your assembly.



**Note: Only available for assemblies**

This menu entry is only available if an assembly is active.

## 8.4 Replace component

Via the function "Replace component" the components used in the CATIA V5 assembly are replaced by a component from PRO.FILE. After the selection of the replacing PRO.FILE CAD document, you can select the component to be replaced in CATIA V5.

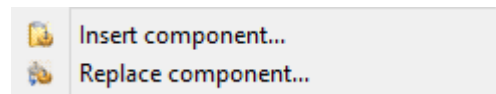


Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Replace component"

Proceed as follows:

1. Select the "PRO.FILE" menu in CATIA V5.
2. Select the function "Insert component".



- ⇒ You are then prompted in the Checkout wizard to select the component in PRO.FILE, which you want to insert into the assembly to replace an existing component.
3. Select the desired CAD document and click <Open>.
  4. Detailed information on the Checkout wizard can be found in the chapter "[Working with the Checkout wizard to search for CAD documents](#)".
  5. Select then the component in your CATIA V5 assembly that is to be replaced by the component from PRO.FILE.
- ⇒ The selected component is now replaced by the component from PRO.FILE.



**Note: Only available for assemblies**

This menu entry is only available if an assembly is active.

## 8.5 Create BOM

With the function "Create BOM" a bill of materials based on the CAD structure of the active document in CATIA V5 is created, which can be viewed in PRO.FILE.

If a bill of materials list structure already exists in PRO.FILE for the assembly body of parts in question, the structure will be updated. If the CATIA V5 geometry contains parts that are not yet contained in the bill of materials, these will automatically be used in the PRO.FILE bill of material structure.



### Note:

Please note the following requirements for the creation of the BOM in PRO.FILE:  
Bills of materials can only be created for assemblies.  
The CAD documents must be linked to a part master in PRO.FILE.



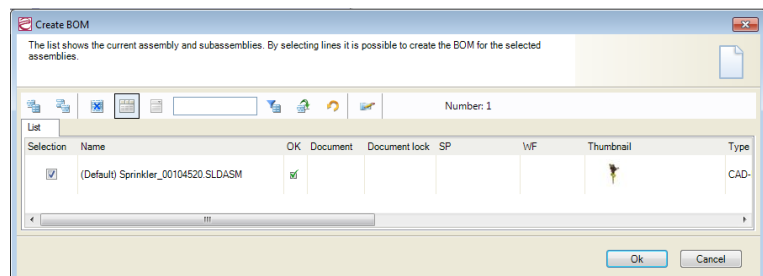
### Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Create BOM"

Proceed as follows:

1. Select the "PRO.FILE" menu in CATIA V5.
2. Select the function "Create BOM".

⇒ A list displays all currently loaded assemblies and sub-assemblies.



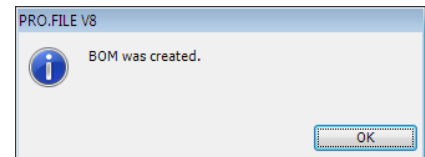
3. Select all CAD documents for which you want to create or update the bill of materials by using the checkboxes.



### Note: Display of conflicts

If the creation of a BOM is not possible for an assembly, the column "OK" shows a red "Attention" symbol. The corresponding tooltip shows the cause of the problem.

4. Confirm your selection with <OK>.
- ⇒ If positions are contained in the PRO.FILE bill of materials list which do not occur in the geometry, a query will occur, in which the user can determine whether the bill of materials list positions should be deleted in PRO.FILE.
5. Select the rows, for which the corresponding BOM position is to be deleted in PRO.FILE
6. Confirm your selection with <OK>.
7. The bill of materials for the selected CAD assemblies is created/updated in PRO.FILE.
- ⇒ The successful creation/update of the bill of materials is confirmed by a message.



To view the bill of materials in PRO.FILE, select the function "Show" => "Bill of materials" as described in the chapter "[Show: Information on a CAD document in PRO.FILE](#)".

**Note:**

Please note that norm parts or auxiliary materials not displayed in the drawing (e.g. water or oil) is not included in the bill of materials by the function "Create BOM". The description of the functions for editing a bill of materials in PRO.FILE can be found in the PRO.FILE manual "Working with structures and bills of materials".

**Note:**

This function can be activated automatically for each saving procedure. Please see the configuration manual of the integration for further details.

## 8.6 Update title block

With this function the title block of the current CATIA V5 drawing is filled with current data from PRO.FILE.

This function can only be used if the current CATIA V5 documents contains a drawing saved in PRO.FILE. Furthermore, the title block already has to be configured.



Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Update title block"

After confirmation of a query mask the title block, modification list and bill of materials are filled automatically, depending on the configured template.

- The modification list always lists the most recent entries.
- If the modification list contains more entries than rows in the configured template, older entries are not displayed.

The following has to be noted

Title block:

- If the values in the title block are not set they will be filled in automatically as long the drawing template is configured accordingly.
- If the values in the title block are set they will be updated automatically.
- If you do not need a certain value in the title block any more you can either delete the according textbox from the background sheet of the drawing or adapt the configuration within PRO.FILE. In order to bring this value onto the drawing once again you will have to create and place the textbox anew.

Bill of materials:

- If the bill of material is not available in the drawing it will automatically be created as long as the drawing template is configured accordingly.
- If the bill of material is already available the values will be update d.
- If you do not want to have a bill of material you will have to delete it from the background sheet of the drawing. In order to bring it onto the drawing once again you will have to create and place the template anew.

**List of modifications:**

- If the values in the list of modifications are not set they will be filled in automatically as long as the drawing template is configured accordingly.
- If the values in the list of modifications are set they will be updated automatically.
- If you do not need a certain value in the list of modifications anymore you can either delete the according textbox from the background sheet of the drawing or adapt the configuration within PRO.FILE. In order to bring this value onto the drawing once again you will have to create and place the textbox anew.

**Balloons:**

- The values inside the balloons will be updated. The balloons can only be created in the menu item **"Extra" => "Create balloons"**.

Within the list of modifications only the current entries will be listed. If there is no space left in the configured list of modifications the oldest modifications will be deleted.

## 9

## Extra

The menu section "Extra" contains the following functions:

- [Workcenter](#)
- [Creating and changing an active working directory](#)
- [Create balloons](#)
- [Update balloons](#)

In order to transfer positions of bills of materials from PRO.FILE to the drawing automatically please follow the instructions below:

1. Click that view you want to create the positions.
2. Activate the command "create position" from the PRO.FILE menu in CATIA V5.  
The positions of the bills of materials are now displayed in the CATIA V5 drawing.

## 9.1

## Workcenter

The Workcenter supports you in the administration of components loaded from PRO.FILE and saved locally.

**Starting the Workcenter from the PRO.FILE menu in CATIA V5:**

"PRO.FILE" => "Extras" => "Workcenter"

You can specify a work folder for each project, display it in Microsoft Explorer, lock and unlock components, get detailed information on parts, documents and bills of materials or delete individual CAD document from your work folder.

Further information can be found in the manual "CAD design supported by PRO.FILE".

**Attention when working with several work folders:**

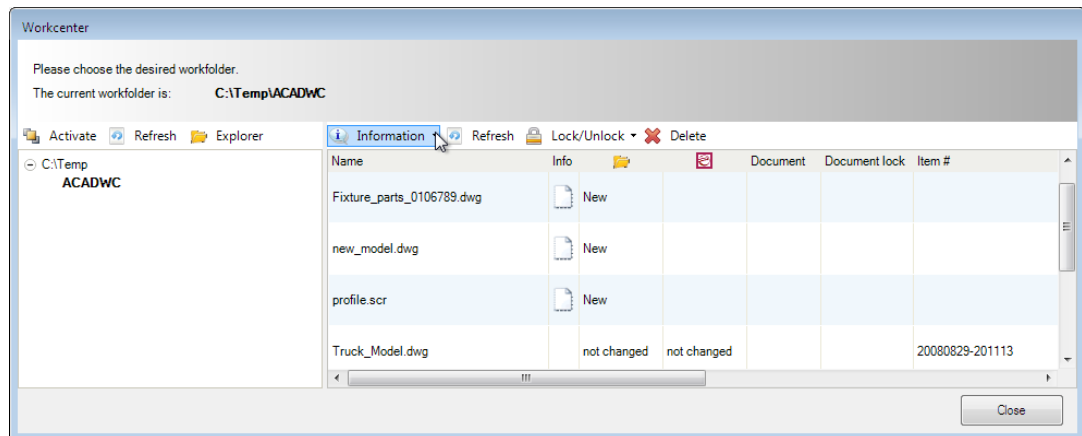
Please be careful when working with several work folders. It may happen that the loaded CAD document has loaded components from work folder A as well as from work folder B. When deleting files from one work folder, make sure that these components are not referenced by documents in a different work folder. To prevent the loss of data, you should only delete CAD documents that are saved in PRO.FILE.

## 9.1.1

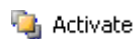
## Workcenter functions

The Workcenter is divided into two areas

- on the left hand side you can find the directory structure of the Workcenter and its commands.
- on the right hand side you can find the commands for all parts or other files currently retrieved from PRO.FILE that can be found in the working directory. You can also find here the status information as described in the chapter "[Up to date or not: Display of status information](#)".

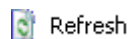


The functions for the directory structure:



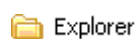
Activate

The selected folder will be used as the new working directory. The current working directory is marked in bold.



Refresh

The view of the directory structure is updated.



Explorer

The selected folder is opened in the Windows Explorer. This gives you the possibility to use the usual Windows functions in order to delete, create or copy a working directory.

The functions for the working directory:



Information ▾

Using the drop-down menu, you can retrieve the following information for marked objects:

Structure of the parts

Part form

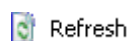
Usage of parts

Bill of materials

Document structure

Document form






Document usage



Refresh

The contents of the marked rows are read again from PRO.FILE and then displayed.



 Lock/Unlock ▼	The respective document is – depending on the user's authorizations – locked or unlocked.
 Delete	The marked documents are deleted from the directory.  If the local status of at least one of the selected files is more recent than the one stored in PRO.FILE, a warning message will be displayed.
 Clear workspace	Starting from the selected work folder, all files that have been saved to PRO.FILE and that have not been modified locally since are deleted – including files in sub-folders.
 Filter	The display filter for the document list can be adjusted via this icon. This can be used to facilitate the finding of objects in large folders.
 Update version	Selected files can be replace by a newer PRO.FILE version (of the same file name). If version conflicts arise, the PRO.FILE dialog for the version selection is displayed.
Open with double click in the CAD system	Double-clicking a file in the Workcenter opens the file in in the CAD system (if it is not already opened).

### 9.1.2

#### Creating and changing an active working directory

When working with the integration PRO.FILE CATIA V5.NET, you can use several parallel working directories (provided that this is configured).

To manually create, activate or change a working directory the function "Workcenter" is used.



Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Extra" => "Workcenter"

#### The basic possibilities in the Workcenter

- To create a new folder, you can use the "Explorer" button. In the displayed Windows explorer you can create and manage subfolders and directory structures.
- The currently active working directory is displayed in bold. In this folder all CAD data from PRO.FILE and CATIA V5 is currently saved.
- To activate a working folder select the desired folder and click the button "Activate".

**Note:**

The first level of the working directory is assigned via the configuration in the PRO.FILE Management Console. The creation of new working folders can only be made below this superior folder.

## 9.2

**Create balloons**

When working with drawings and assemblies the function "create balloons" allows to display positions of bills of materials from PRO.FILE in drawings. Therefore the CATIA V5 function for the automatic positioning of balloons is used.

These balloons will be filled up with information out of the PRO.FILE bill of material. Thereby only the first level within the displayed assembly will be analyzed and a balloons will only be created automatically for parts and sub-assemblies that belong to this level.

**Function call from the PRO.FILE menu in CATIA V5:**

"PRO.FILE" => "Extra" => "Create balloons"

**Requirements for the creation of a position**

To be able to create a position within a drawing certain requirements have to be fulfilled:

- The drawing is saved to PRO.FILE.
- A bill of material has been created in PRO.FILE.
- Within the PRO.FILE Management Console the output formats have to be configured. (See the configuration manual)
- The settings in CATIA V5 concerning the automatic generation of balloons have to be configured correctly. (See the configuration manual)

These requirements have to be fulfilled intending inserting positions of bills of materials successfully.

**Attention:**

The function "create balloons" only works if the language is set to English!

**How to create positions automatically**

In order to transfer positions of bills of materials from PRO.FILE to the drawing automatically please follow the instructions below:

1. Click that view you want to create the positions.

2. Activate the command "create position" from the PRO.FILE menu in CATIA V5.  
The positions of the bills of materials are now displayed in the CATIA V5 drawing.  
The location of the Position-number can be changed per drag&drop.

## 9.3

### Update balloons

The function "update balloons" allows updating already existing balloons in a CATIA V5 drawing with current values from the PRO.FILE bill of material. It is a precondition that the related balloons were also created with the help of the function "update balloons".



Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Extra" => "Update balloons"

## 9.4

### Update thumbnail

If the PRO.FILE document list uses thumbnails, this function can be used to update the thumbnail images.



Function call from the PRO.FILE menu in CATIA V5:

"PRO.FILE" => "Extra" => "Update thumbnail"

## 10 Workplace-specific configurations

The integration PRO.FILE CATIA V5.NET makes enterprise-wide, global configurations of the mode of operation and environment possible - as well as also the local, workplace specific determination of user adjusting.

### Global Configuration

The global configuration applies for all users. It can be configured by using the PRO.FILE Management Console.

With the help of the adequate parameters it is possible to configure the company-wide behavior of the integration PRO.FILE CATIA V5.NET.

### Local Configuration

The user can carry out individual settings when required within the CATIA V5.NET Integration. In order to do the local configuration the PRO.FILE Management Console has to be used. For these purposes the respective parameters will be defined user-specific.

The settings for the following options can be found here:

- Document list
- Original name reference
- Title block
- capacity (update characteristics and title block)
- general parameters

Further information can be found in the configuration manual of the integration.

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