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Functions of the Integration PRO.FILE SolidWorks

PRO.FILE Release 8.7
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PROCAD

Registered Trademarks:

Responsible for Content:

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About this manual

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 – SolidWorks

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 SolidWorks. Drawings and CAD models can be loaded from or saved to PRO.FILE directly from SolidWorks.

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 SolidWorks to ERP systems.

	Note: Support of weldments
	The integration PRO.FILE SolidWorks as of Release 8.6 supports the administration of weldments. The corresponding functions are available for SolidWorks 2015 or higher. These functions are not supported by the integration PRO.FILE SolidWorks 2014.

1.1   The contents of this manual

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

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 SolidWorks. The following topics will be addressed:

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

	Note: Manual "CAD design supported by PRO.FILE"
	When using the integration PRO.FILE – SolidWorks, 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 SolidWorks.

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

- Attention - Special Reference: Connections to SolidWorks
- Only upon first start: Setting up the local work folder
- Where can I find the functions of the PRO.FILE integration?
- How to log in to PRO.FILE?
- A brief overview: The functions of the integration

! Attention: Configure references before start

Please note that the references in the SolidWorks environment have to be configured before the first start of the integration. For this, see the following chapter: Attention - Special Reference: Connections to SolidWorks.

2.1 Attention - Special Reference: Connections to SolidWorks

Certain settings within the SolidWorks-environment may lead to serious connection problems when working with PRO.FILE-SolidWorks Integration as already explained in detail in the configuration manual for PRO.FILE - SolidWorks Integration.

The settings described in this chapter have to be made on each user workstation on which the integration is installed.

We strongly recommend that you check all SolidWorks workstations for complete and correct settings **before** you start working with the integration, and **before** errors occur.

! Attention:

This configuration of the SolidWorks-environment has to be made **before** the first start of the integration.

Configure the external references in SolidWorks

1. You can access the index menu by calling up "**Options**", from the, "**Tools**", menu within the SolidWorks menu bar.

2. Please select the list point "**external references**", from the list of system options on the left hand side.

3. Please deactivate the option "**Automatically generate names for referenced geometry**", as shown below.

4. Confirm your settings with <**OK**>.

5. Please make this setting for all SolidWorks work stations.
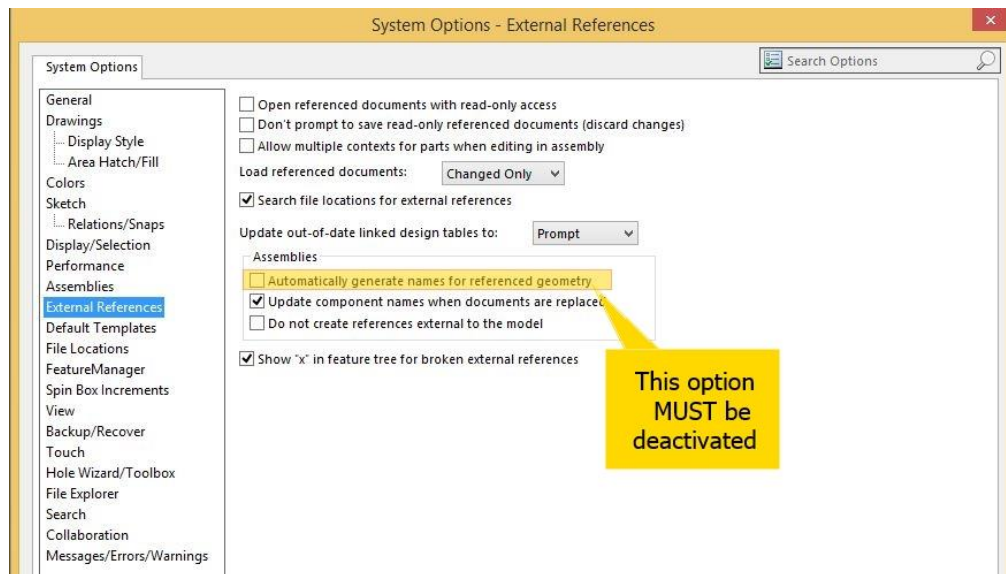
⚠	**Attention:**
	Please note that the use of the option switch "**Automatically generate names for referenced geometry**" can lead to connection problems when working with the integration.

2.2   Only upon first start: Setting up the local work folder

CAD drawings are loaded directly from PRO.FILE in SolidWorks, and also saved and versioned from SolidWorks 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 SolidWorks 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 – SolidWorks.

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**>.

3. Once you have selected the desired root folder, confirm with <**OK**>.
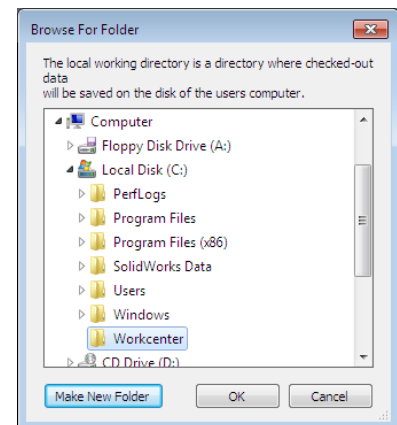
4. In the second step, "work folders" are now created In this root folder, which will then be used by the integration.
Consequently, you are now prompted to specify a work folder within the root folder:

5. Please specify a name for the work folder.

6. Confirm your entry with <**OK**>.

⇒ The configuration of the Workcenter is now finished.
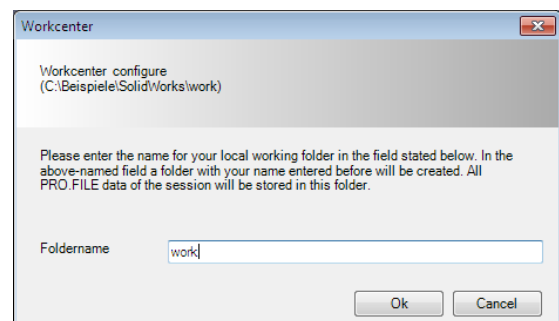
This work folder and other local work folder can be created and managed by the user via the Workcenter. The Workcenter can be accessed via the PRO.FILE menu in SolidWorks under "**Extra**" => "**Workcenter**".

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

2.3 Where can I find the functions of the PRO.FILE integration?

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

1. Start SolidWorks.

2. In the startup view of SolidWorks, the PRO.FILE functions are displayed in this icon bar:
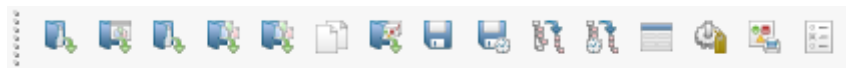
3. Select the desired integration function from the icon bar. This will start PRO.FILE and gives you access to the required data saved in PRO.FILE.

4. Once a document is opened in SolidWorks, the menu ribbon also contains a PRO.FILE menu containing the available integration functions.
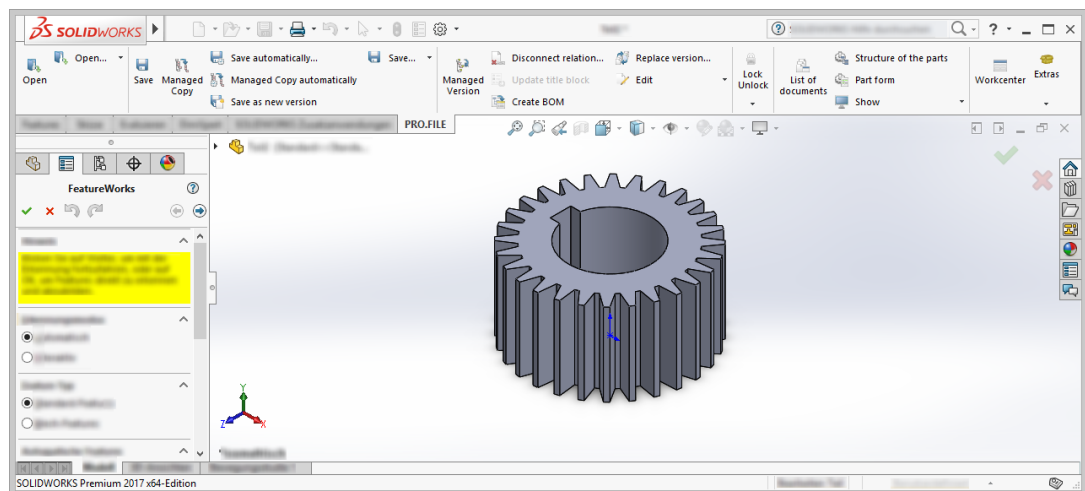
Contents of this PRO.FILE menu depend on the CAD object that is currently active in the foreground.

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: Making the PRO.FILE menu appear in SolidWorks**
>
> If the PRO.FILE menu is not shown, activate it using the SolidWorks functions "**Tools**" => "**Add Ins**" from the SolidWorks Menu Bar.

2.4   How to log in to PRO.FILE?

If you access a PRO.FILE function for the first time within a SolidWorks 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.
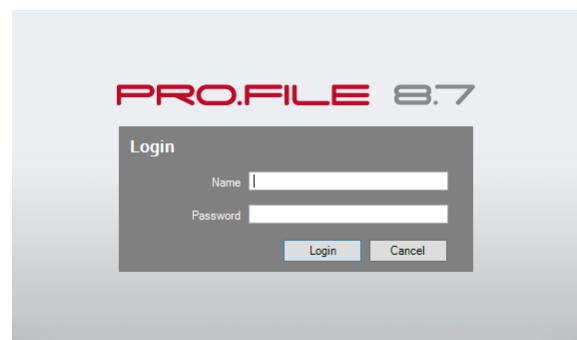
> **Note: No login required in case of "Autologin"**
>
> This login is not requested, if the PRO.FILE autologin function is activated.

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 SolidWorks by confirming the choice. The subsequent method for opening depends on the settings of the parameter "Version load options dialog" in the PRO.FILE Management Console.

- **Open with version browser:**

  With the version browser you can open assemblies as dynamic compositions, i.e. you can decide, which versions of the components you want to load in the CAD session.

- **Open as saved:**

  The selected document is loaded from PRO.FILE with the constellation of component versions as it as last been saved.

- **Open with latest versions:**

  The selected document is opened from PRO.FILE. If other CAD documents are linked with this document, the newest versions of these CAD components are loaded.

- **Open with latest released versions:**

  The selected document is opened from PRO.FILE with the newest released versions of linked CAD components.

- **Open drawing:**

  With this function, you can directly load the drawing saved in PRO.FILE for the object active in SolidWorks.

- **Make document available:**

  This function can be used if files saved in PRO.FILE are only to be loaded into the local work folder, but, due to time reasons, not to be loaded in SolidWorks.

- **Open with all drawings:**

  If an assembly is opened, you can use this function to open all drawings available in PRO.FILE for the assembly structure in SolidWorks.

Saving and
Storage
Functions:

- "Save":

  Via this function newly created CAD documents are checked in to PRO.FILE, or documents already saved in PRO.FILE and checked out for editing are saved back to PRO.FILE. When changes are saved back, the existing document in PRO.FILE is overwritten automatically.

- Managed Copy:

  Managed Copy organizes the data management of complex 3D models in changed constructions. Entire machines can be cloned, including all referenced data and drawings. New numbers of the cloned machine and the new structures are automatically updated in the dependent drawings. Assemblies and parts that are to remain in the new structure are taken over. Existing references remain intact.

- Save automatically:

  Document and part descriptions for all components are created in PRO.FILE automatically without query. File names and properties can be configured to be transferred automatically into specific PRO.FILE fields.

- Managed Copy automatically:

  This feature combines "Managed Copy" with the function "Save automatically". When copied components are saved anew, you do not have to enter part and document information for each part. The parts are created in PRO.FILE automatically. File name and file properties are automatically transferred to specific PRO.FILE fields, depending on the configuration.

- Incremental save:

  Via this function, only the currently active level of an assembly and the level immediately below are searched for modified documents to be saved.

- Incremental save automatically:

  This function unites the functions "Save incremental" and "Save automatically". In analogy to "Save automatic", no further user input is required during the save process.

- Save all instances:

  If you have created several instances of one object, this function allows you to save all objects without having to save each instance separately.

- Save instances automatically:

  This feature combines the functions for saving all instances and automatic saving. With this function, when you are saving an object, you do not have to enter part and document information for each instance. The instances are created in PRO.FILE automatically. File name and file properties are automatically transferred to specific PRO.FILE fields, depending on the configuration.

- Save Phantom:

  This function is used to save an entire assembly under an individual body of parts in PRO.FILE and to save all objects contained in the assembly under this body of parts. As a result, this assembly is treated like a single part in PRO.FILE. The objects contained in the assembly are saved as phantom objects and cannot be opened explicitly in PRO.FILE.

Note: Only the objects not known in PRO.FILE yet will be saved as phantom objects. For all objects already known in PRO.FILE "Save Phantom" will have the same effect as "Save".

- Save NDF:

  With this function, a neutral data format (e.g. tiff, pdf) is created from the CAD document and saved as new document in PRO.FILE. This NDF document is automatically linked to the part master record of the drawing.

- Link local file:

  This function is used to add additional files to the structure that are not yet saved in PRO.FILE.

- Link document:

  This function is used to add files to the structure that are already saved in PRO.FILE.

- Dissolve document link:

  This function is used to remove the link of the additional file to the CAD object.

Functions for assemblies

- Insert part:

  With this function you can select a part saved in PRO.FILE and insert it into the assembly in SolidWorks via the SolidWorks "Insert part" wizard.

Functions for Versioning

- Open with released versions:

  This open function automatically reads the newest released version of the references of the CAD-objects from PRO.FILE in Solid Works.

- 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

- Lock:

  CAD objects which were read from PRO.FILE in SolidWorks 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.

- Lock (1 level):

  By calling the function "Lock (1 level)" only CAD objects of the first sub-level of an assembly are listed. This results in a limited list of objects to be locked, but this list loads much faster and is easier to handle.

- Unlock:

  This function unlocks the PRO.FILE objects which were locked for processing in SolidWorks. 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.

- Unlock (1 level):

  By calling the function "Unlock (1 level)" only CAD objects of the first sub-level of an assembly are listed. This results in a limited list of objects to be unlocked, but this list loads much faster and is easier to handle.

Display and information functions in the menu "Show"

- List of documents:

  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.

- List of documents (1 level):

  Shows the special list of documents, but limited to the first sub-level. The resulting list loads faster and is easier to handle.

- Document structure:

  Switches to the display of the PRO.FILE 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, the form of the displayed assembly will be shown.

- Use of documents:

  Switches to the display of the proof of usage list of the displayed/selected object.

- All document versions:

  Switches to the display of the version list of the displayed/selected object.

- Structure of the parts:

  Shows the structure overview of the displayed/selected object.

- Part form:

  Changes to material master form of the displayed/selected object.

- Use of parts:

  Changes the proof of usage list of the displayed/selected object.

- Bill of materials:

  Shows the bill of materials view of the displayed/selected object.

- Configurations (Instances):

  This view assists the user in the administration of the different configurations of a part directly from the integration.

- PROCAD on the WEB:

  This menu point opens the PROCAD homepage, provided that the user has internet access.

- Help:

  This menu entry opens the PRO.FILE online help for the operation of the integration. A prerequisite for this is the installation of the online help setup on this computer.

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 a selection of objects contained therein. The CAD objects are then treated as purely locally-saved CAD objects without reference to PRO.FILE.

- Disconnect relation (1 level):

  The function "Disconnect relation (1 level)" lists only the first sub-level of an assembly. This results in a limited but very fast list of CAD objects to be unlocked, the database links of which are to be deleted.

- Document refresh:

  This function allows all parts of an assembly to be compared with the latest version in PRO.FILE, and be re-read into the assembly.

Functions for bill
of materials and
title block
(drawing legend)

- Create BOM:

  This function calls up PRO.FILE and creates a new bill of materials for the active assembly. If there is already a PRO.FILE BOM, it will be updated.

- Create balloon:

  This function allows you to display the bill of material positions when working with drawings and assemblies from PRO.FILE.

- Update title block:

  This function allows information on bill of materials, modification lists and titles, to be filled in on a drawing upon opening. This requires the lists and fields to be pre-configured for the template that is to be used.

- Autoballoon:

  This function reads the position numbers of parts from PRO.FILE and transfers them to the drawing in SolidWorks.

Extra: Additional functions for the configuration and the plotting of drawings

- Workcenter:

  All files loaded or saved via the PRO.FILE integration in SolidWorks are automatically saved locally in the Workcenter folder. With this function you can manage these files or create additional work folders.

- Managed Rename:

  This function allows the later renaming of CAD documents from PRO.FILE. The references known to PRO.FILE are then updated accordingly.

- Drawing plot:

  All, or certain drawings of a BOM, can be plotted with this function.

- Options:

  The local integration can be configured using this menu item. It includes the possibility to establish document lists, to show or hide messages, to configure original name references, title blocks and performance setting.

- Configuration title block, Configuration modification list, configuration bill of materials, configuration balloons:

  These functions are described in the manual "Configuration of the integration PRO.FILE – SolidWorks" due to the required access to the PRO.FILE Management Console.

- Update thumbnail:

  If a preview thumbnail is used in the PRO.FILE document list, you can use this function to update the thumbnail picture.

- **Create preview file**: Creates a PDF or STEP file of the current document, which is then displayed in the preview tab in PRO.FILE.

3   Opening CAD Documents from PRO.FILE in SolidWorks

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:

Opening documents from within PRO.FILE:

- Open CAD documents from PRO.FILE for editing

Opening from within the integration:

- Open: Opening CAD Documents from PRO.FILE

In the menu "**Open…**":

- Open with version browser
- Opening CAD documents with linked components
- Open drawing
- Make document available
- Open with all drawings
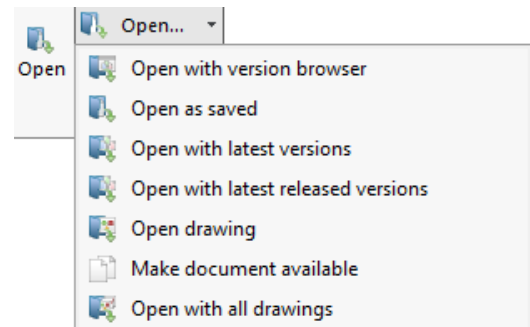
Attention:

The data loaded from PRO.FILE in SolidWorks are **not automatically locked** when opened in SolidWorks. 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 CAD documents from PRO.FILE for editing

Apart from opening a document from within the integration, you can also open CAD files directly from PRO.FILE. The following options are available:

- Double-click on the file in list or form display.

- Select the document and open it via the menu ribbon "Edit file".

- Select the document and open it via the context menu function "Edit file" => "Edit document".

- Select the fila and move it to the CAD GUI via drag & drop.

The subsequent method for opening depends on the settings of the parameter "Version load options dialog" in the PRO.FILE Management Console. When a document is opened via the "Edit file" ribbon or via the context menu, the CAD file is automatically locked and cannot be edited by another user.

3.1.1 Open via drag & drop

You can open CAD objects from PRO.FILE via drag & drop and use them in your assemblies. To do so, drag the desired CAD component from PRO.FILE into the CAD system GUI.

Note:

If components are opened via drag & drop from PRO.FILE, no file properties or title blocks are updated. If you want to update these, you have to use the corresponding integration function afterwards or the update during the saving to PRO.FILE has to be activated.

Proceed as follows

1. Select the desired CAD document in PRO.FILE in a list display.
2. Hold down the CTRL-key and grab the paper clip icon.
3. Drag the icon into the CAD GUI and drop it there.
⇒ The file is copied into the Workcenter folder and is opened.

Note:

If version conflicts occur during the copying of the file, the process is cancelled. A message is displayed, indicating the problem. In such a case, you can only open the document via the integration functions.

3.2   Open: Opening CAD Documents from PRO.FILE

If you want to access a document from PRO.FILE, use the function "**Open**" of the PRO.FILE – SolidWorks integration.

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

To open a SolidWorks document from PRO.FILE proceed in 4 steps:

Step 1   Using the PRO.FILE function "Open"

	Function call from the PRO.FILE menu in SolidWorks:
	"PRO.FILE" =>"Open"

1. Go into the menu bar of SolidWorks into the menu "**PRO.FILE**".

2. Select the menu entry "**Open**".

⇒ "Open" loads documents as they were saved the last time in PRO.FILE.

⇒ The Checkout wizard for the selection of documents 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.

3. 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
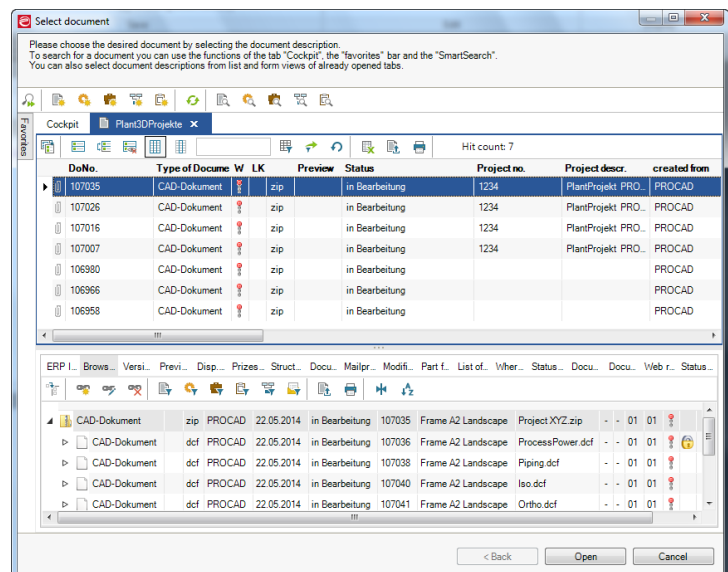
4. The desired document is displayed in a list view, you can select it. (If the desired document is displayed in a form view, it is already selected.

5. 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

Versions in the structure: How are the components opened?

When a CAD object with structure (assembly, drawing, etc.) is loaded via the "Open" function, PRO.FILE checks whether the CAD document contains components for which versions exist.

If this is the case, you can then decide how PRO.FILE is to load the assembly/drawing in question.
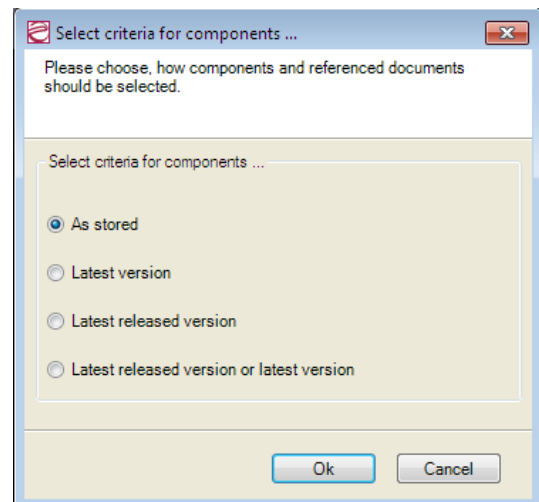
6. Select the desired method and confirm with <**OK**>.

• **As stored**:

the assembly is loaded with the components it was recently saved with. Changes to parts that resulted in new version of the parts are ignored.

• **Latest version**:

PRO.FILE replaces all CAD documents for which it finds a newer version and loads the assembly/drawing with the newer object versions. You thus get an updated version of the assembly/drawing.

Note:

You can only load version for which you have reading access in PRO.FILE. If the most recent version is not "visible" for you, you will only be displayed the newest visible version.

• **Latest released version**:

PRO.FILE replaces all CAD documents for which it finds a newer version in a released status and loads the assembly/drawing with the released object versions. In analogy to the previous method, you can only load objects, for which you have viewing permissions.

> ⚠ **Attention:**
>
> If no version in a released status can be found for a CAD document, the assembly/drawing is not loaded.

- **Latest released version or latest version:**

  This option is important if an assembly consists of both released and unreleased components. PRO.FILE replaces all CAD documents, for which it finds a newer version in a released status. If no version can be found in a released status, the newest visible version is loaded.

  PRO.FILE opens the assembly/drawing with all available objects in a released status – all other objects are loaded in the newest visible version.

> ℹ **Note: Open with version browser**
>
> For the targeted selection of the desired version you can use the function "Open with version browser". With the version browser you can open assemblies in dynamic compositions – the user can decide for each component, which version of this component is to be loaded from PRO.FILE.

Step 4

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")
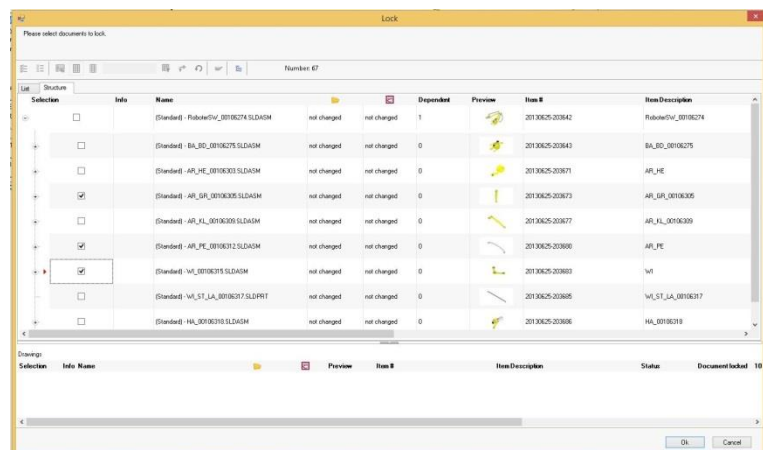
7. Select all documents you want to lock with the corresponding check boxes.

8. Confirm your selection with <OK>.
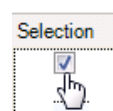
⇒ The selected document is now opened with its components in SolidWorks. The "Open" procedure is thus finished.

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.

For detailed information on the "Open" process, please see the following chapter: Working with the Checkout wizard to search for CAD documents.

3.2.1 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 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**>.
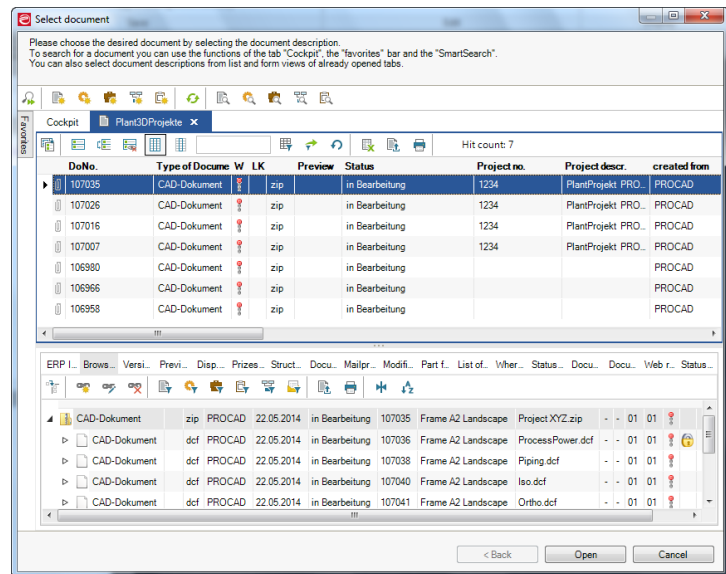
- If no search had been made previously, or if the desired document is not displayed on one of the existing tabs, you can now search for it.
  For this, several functions, which are explained in the following, are available:

!
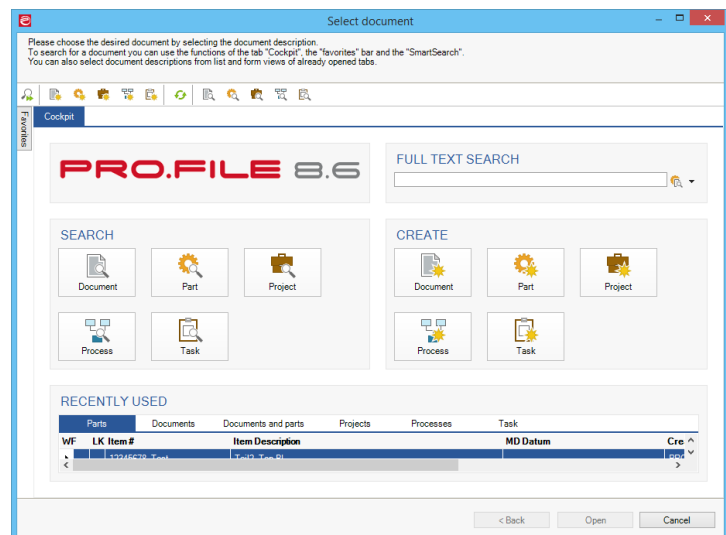
Attention: Double-click in the Checkout wizard

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

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 ".

3.3   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:
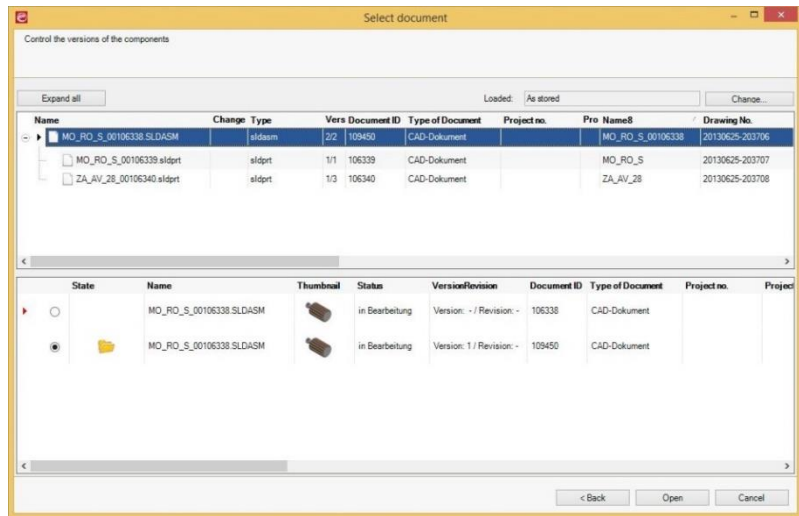
The version browser is divided into two areas:

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 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 SolidWorks.

Function call from the PRO.FILE menu in SolidWorks:

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

| Proceed as follows | 1. | Select the "**PRO.FILE**" menu from the menu bar in SolidWorks. |

1. Select the "**PRO.FILE**" menu from the menu bar in SolidWorks.

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
◉	📁 Hydr_Zylinder_101663.ipt		in Bearbeitung	Version: - / Revision: -	101663	CAD-Dokument

6. 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.

7. Select all documents you want to lock by setting the checkmark for it in the first column.
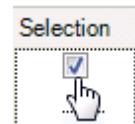
Detailed information on the locking of documents can be found in the following chapter "Lock/Unlock: Who can change when?".

8. Confirm your selections with <**OK**>.

⇒ The selected CAD components are opened in SolidWorks. 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.
ℹ️	Shows a version conflict. This can occur, e.g. if a part is used in two assemblies in different versions.
🔷	Icon of SolidWorks assemblies

⬜ Icon of SolidWorks parts

⬜ Indicates a softlink.

◈ Versions reference each other causing a version cycle.

3.4   Open drawing

If a CAD object (part, assembly) that is known in PRO.FILE is loaded in SolidWorks, this function can be used to directly load the corresponding drawing from PRO.FILE without having to search for the document.

If several drawings exist for this CAD object, all of these drawings are opened

Note:

In order for this function to work correctly, the corresponding drawing of course has to be saved already in PRO.FILE. If no drawing is saved for the active CAD object, this function has no effect.

Function call from the PRO.FILE menu in SolidWorks:

"PRO.FILE" => "Open…" => "Open drawing"

Proceed as follows:

1. Make sure that the CAD document, for which the drawing is to be opened, is displayed in SolidWorks.

2. Select the "**PRO.FILE**" menu in SolidWorks.

3. Select the function "**Open…**" => "**Open drawing**".

⇒ If several drawings are available for a CAD object, a dialog to select the drawing is displayed.
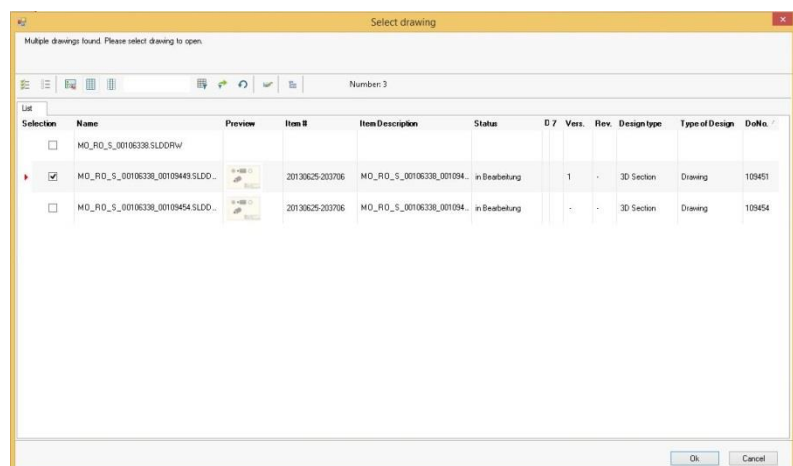
4. Select all drawings you want to open in SolidWorks with the checkboxes in the first column.

5. Confirm your selection with <OK>.

⇒ In analogy to the procedure for opening documents from PRO.FILE as described in the chapter " Open: Opening CAD Documents from PRO.FILE", the list of locally changed files is displayed (see chapter "Attention: Opening of locally existing files").

6. Select whether you want to overwrite the files located in the Workcenter or not. Confirm your selection with <OK>.

⇒ The dialog for the locking of loaded CAD documents is displayed.

7. Select all documents you want to lock with the checkboxes in the first column.
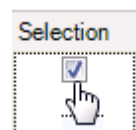
8. Confirm your selection with <OK>.

⇒ After confirmation of the selection window with <OK>, the drawing for the active object is opened in SolidWorks.

3.5 Make document available

When opening CAD data the "normal" way from PRO.FILE the data is automatically loaded in the SolidWorks session.

Depending of the size of the part or assembly, this can take some time.

If the SolidWorks files from PRO.FILE are only to be copied into the local Workcenter but not to be loaded in the SolidWorks session, the function "**Open**" => "**Copy only**" can be used.

	Function call from the PRO.FILE menu in SolidWorks: "PRO.FILE" => "Open…" => "Copy only"

- The procedure is the same as for " Open: Opening CAD Documents from PRO.FILE".
- The only difference is that, after the dialogs have been confirmed, the files are not loaded in SolidWorks but only copied into the active work folder.
- From there, the files can be opened at a later point in time.

3.6 Open with all drawings

If an assembly that is known in PRO.FILE is loaded in SolidWorks, the function "Open with all drawings" can be used to directly open and display all corresponding drawings within the assembly structure from PRO.FILE

	Note: Drawings for the components in PRO.FILE In order for this function to work correctly, the corresponding drawings of course have to be saved already in PRO.FILE. If no drawing is saved for the active CAD objects, this function has no effect.

	Function call from the PRO.FILE menu in SolidWorks: "PRO.FILE" => "Open…" => "Open with all drawings"

The further proceeding (listing of locally changed documents, question whether to overwrite local files) is identical to the function "Open drawing" and described in the chapter "Open drawing".

3.7   Opening CAD documents with linked components

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

- Open as saved

- Open with latest versions

- Open with latest released versions
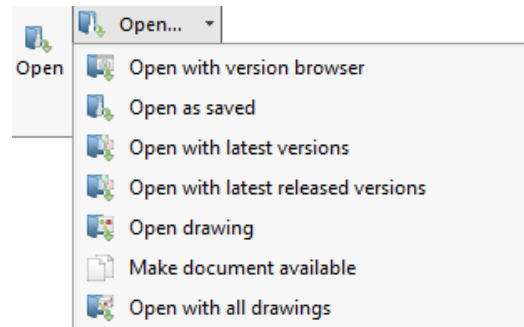
Note:

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 as saved"

  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.

- "Open with latest 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 "Open with latest 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 CAD session.

- "Open with latest 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 "Open with latest 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 CAD session.

The actual process of opening the document is identical for all three of these options. For more information see the previous chapter "Open CAD documents from PRO.FILE for editing".

	Note: The difference between "Open with released versions" and "Open"
	Contrary to the function "Open" the function "Open with released version" does not display a list/dialog indicating that newer versions have been found. Furthermore, the dialog asking which method is to be used for opening is not displayed, as this question is already answered.

	Note:
	You can only load version for which you have viewing permissions. If the most recent version is not "visible" for you, the **newest visible version is loaded**.

3.7.1   Scenarios for the usage of "Open with latest released versions"

The following scenarios are to illustrate the function "Open with latest released versions". The following is assumed:

- You want to open an assembly from PRO.FILE via the function "Open with released versions" in SolidWorks.

- Within this assembly, some parts have been versioned.

Let's take a look at different scenarios regarding the released statuses of versions and how the integration reacts in the different cases:

Scenario 1   **All newest version in the assembly are in the status "Released"**

- You are in SolidWorks and load the assembly from PRO.FILE with the function "**Version**" => "**Open with released versions**".

- Since all objects match the selection criterion, the assembly is loaded with the newest released versions in SolidWorks.

Scenario 2   **The assembly contains an older version in the status "Released"**

A part in the assembly has been versioned several times. The current version (e.g. number 2) is still in development. The previous version (number 1) is "released".

- You are in SolidWorks and load the assembly from PRO.FILE with the function "**Version**" => "Open with released versions".

- SolidWorks loads the assembly with part version number 1 – not number 2. This means that you do not automatically get the newest version when using this function.

Scenario 3

The assembly contains a part with a version that is not in a released status

A part in the assembly has been versioned. The part and the version are not (!) in a released status. All other version within the assembly are in the status "Released".

- You are in SolidWorks and load the assembly from PRO.FILE with the function "Version" => "Open with latest released versions".

- PRO.FILE begins analyzing the assembly and finds this versions. As it does not match the criterion, an error message is displayed saying that no objects have been copied into your Workcenter.

> **!** **Attention: Open with released versions**
>
> If a CAD document is loaded from PRO.FILE via the function "**Open with released versions**", and the document structure contains a document for which no released version exists, the CAD document is not loaded.

3.8 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.
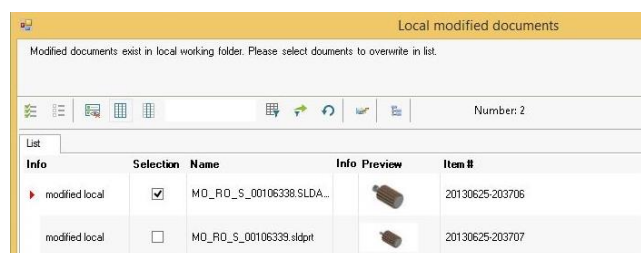
You have three options of proceeding:

- **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".

Dynamic lock dialog with PRO.FILE

Up to now, the designer could make local changes without being actively and directly notified about a possible conflict with the CAD documents saved in PRO.FILE.

With PRO.FILE 8.6, local changes are now recognized. The integration evaluates the saving activities of the CAD system and displays the dynamic lock dialog:

- Here you can now select the CAD data to be locked by setting the checkmark.

- Confirm your lock with <OK>.
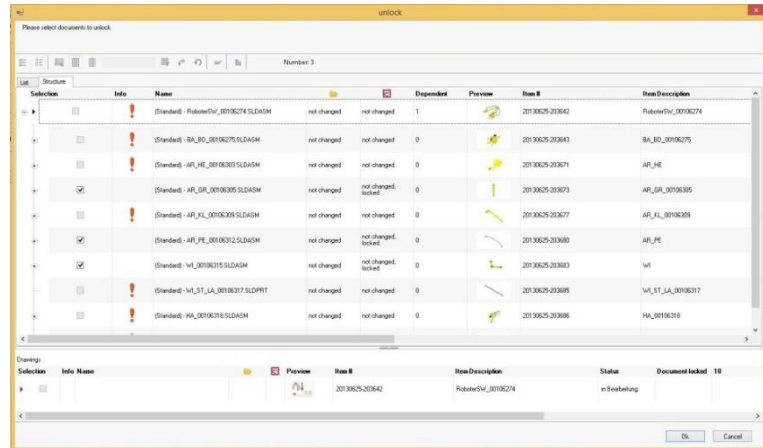
Displayed entries that are not selected for locking, will not be offered again for locking during the active CAD session.

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 SolidWorks:

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

Lock a CAD document manually

1.   Make sure that the CAD document to be locked is displayed in SolidWorks.

2.   Select the menu "PRO.FILE" from the SolidWorks 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]").
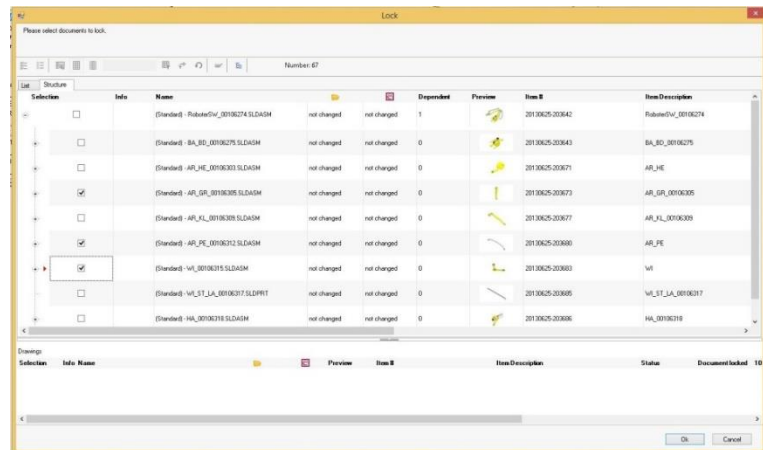
⇒ With the display of status information in this list PRO.FILE checks:

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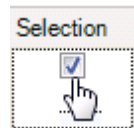
⇒ By the command "Lock", the access permission to edit the document is checked for the current user, and the document is locked for all other users in the database.

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

To unlock a document proceed as follows:

1. Make sure that the CAD document to be unlocked is displayed in SolidWorks.

2. Select the menu "**PRO.FILE**" from the SolidWorks menu bar.

3. Select the function "**Lock/Unlock**" => "**Unlock**".

⇒ 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").
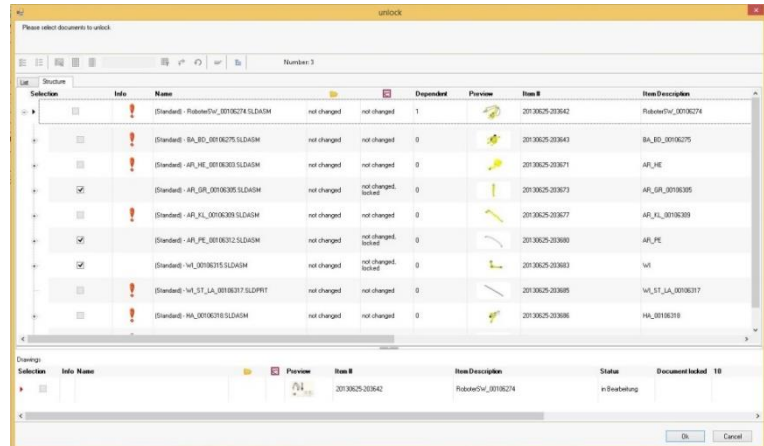
4. To make the CAD documents saved in PRO.FILE available for other users, select the documents to be unlocked in the list.

5. Confirm your selections with <**OK**>.

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

4.3   Lock and unlock CAD objects (1 level)

When regularly locking or unlocking PRO.FILE objects loaded in SolidWorks, a list of all active objects from PRO.FILE is always shown by the integration.

With complex assemblies, this list can become quite large and may take significant amounts of time to load.

*   By using the functions "Lock (1 level)" and "Unlock (1 level)" only CAD objects of the first sub-level of an assembly are listed.

*   This results in a limited list of objects to be locked or unlocked, but this list loads faster and is easier to handle.

	Function call: "PRO.FILE" => "Lock/unlock" => "Lock (1 level)" => "Unlock (1 level)"

The further proceeding is the same as with the regular saving function and is described in detail in previous chapters.

> **Note:**
>
> The difference between "Lock – Unlock" and "Lock – Unlock (1 level)" lies only within the size of the offered list of active CAD objects to be locked/unlocked.

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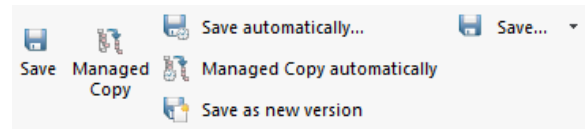
    - [Saving CAD objects for the first time](#)

    - [Save: Saving changed CAD documents](#)

    - [Managed Copy](#)

    - [Managed Copy automatically](#)

    - [Save as new version](#)

- The menu "**Save…**" contains the functions:

    - [Save automatically](#)

    - [Save all instances](#)

    - [Save instances automatically](#)

    - [Save Phantom](#)

    - [Save NDF (Neutral data format)](#)
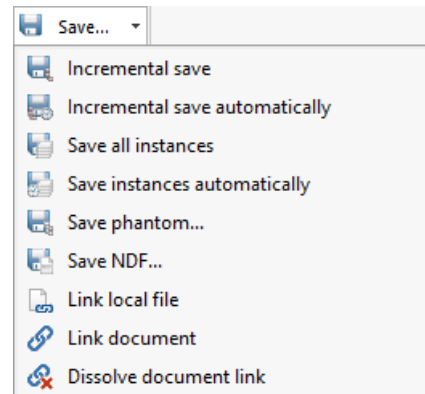
This variety of functions is based on one fundamental behavior:

- All functions first check for new, unknown documents. If such documents exist, they are saved to PRO.FILE. Then, locally changed documents are offered for saving, if such documents exist.

- The options "**… automatically**" and "**… phantom**" only decide on new, unknown documents to be created.

- The options "**Incremental…**" decide on depth of the structure that is used for searching for locally changed files offered for saving.

Solid Works objects need to be saved locally before being saved to PRO.FILE. If this is not the case, this local saving is enforced by the integration.

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 SolidWorks:

* [Saving CAD objects for the first time](#)

* [Save: Saving changed CAD documents](#)

Additional saving functions for weldments are described in the chapter:

* [Saving of weldments](#)

	Note: Manual "CAD design supported by PRO.FILE"
	Before using the integration PRO.FILE – SolidWorks 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.

5.1 Saving CAD objects for the first time

With the use of the function "**Save** ", SolidWorks objects are saved to PRO.FILE.

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 have to save the new CAD object locally. This is required by SolidWorks.

* Then you can save the object to PRO.FILE.

If you want to save CAD documents from SolidWorks 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 SolidWorks:
	"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 several 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](#)

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

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 **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).

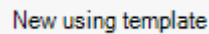4. Confirm your selection with <**Next**>.

Search **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   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 filed "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 | 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

- 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.
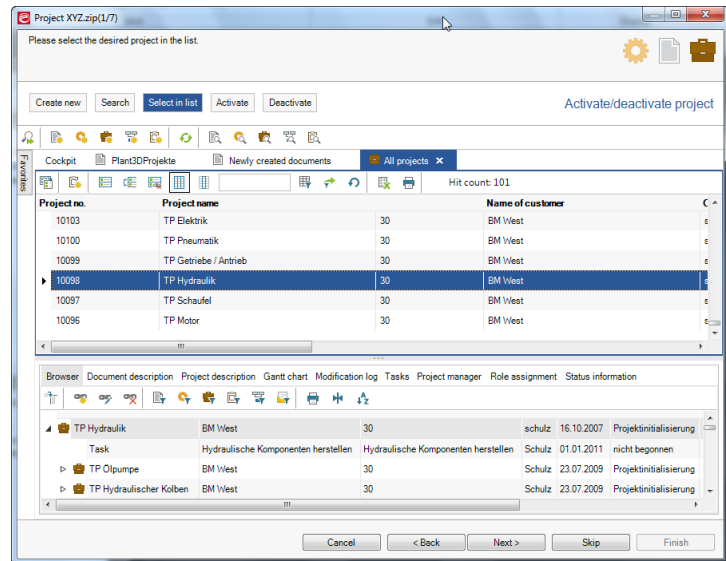
Here, too, the Checkin wizard offers different options that can be accessed via the operations bar:
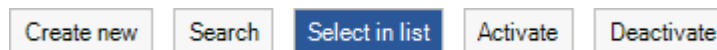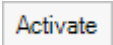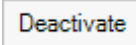
⚠️ **Attention: Project must be activated**

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** **Create new:**

  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** **Search:**

  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** **Select in list:**

  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: Activate:

  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: Deactivate:

  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.

5.1.4   Saving parts of an assembly

When working on an assembly or an element of an assembly, you must make sure that when you do a save the references remain. It is possible to open a part within an assembly (out of the assembly) and save it into PRO.FILE. When you save the assembly the reference will remain as long as the part stays open during this session.

When you have the assembly open you have all of the necessary parts for this session, loaded in the window. If you open a part from this assembly from SolidWorks using <**Open**> from the working directory of your hard drive, SolidWorks cannot recognize the reference. If you change this part, and save it to PRO.FILE, the reference to the other part will remain for the assembly.

5.2   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 started, PRO.FILE checks whether the user has the permission to change the corresponding object in PRO.FILE. The software also checks whether the copy of the object that is used in the CAD system is up to date.

> **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?".

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

- Save: Saving changed CAD documents
- Save automatically
- Incremental save
- Incremental save automatically

- Save all instances
- Save Phantom
- Managed Copy
- Managed Copy automatically

This chapter describes the proceeding for saving changed CAD documents.

> **Function call from the PRO.FILE menu in SolidWorks:**
>
> "**PRO.FILE**" => "**Save**"

Proceed as follows:

1. Go to the integration menu "**PRO.FILE**" in SolidWorks.
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
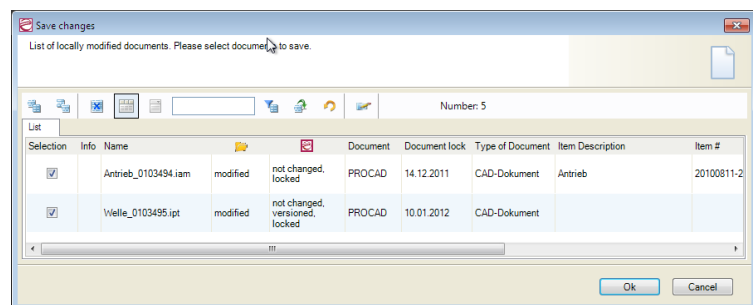   documents
    to be
    saved is
    displayed.
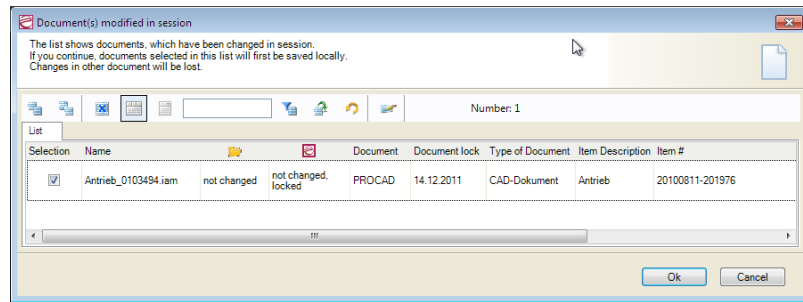
⇒ The dialog displays a list with all changed CAD documents from the current SolidWorks session. (Information on the functions and status information can be found in the chapter "Data overview: The document list").

⇒ 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>.

Locally changed documents in the structure

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.
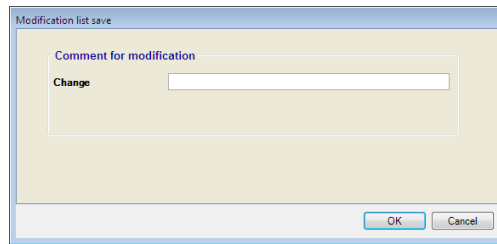
7. Confirm your modification comment with <OK>.

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

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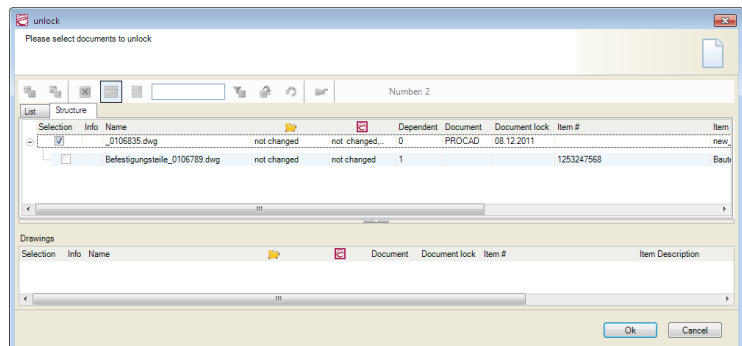
⇒ If documents from PRO.FILE had been locked for editing in SolidWorks, this lock is not automatically removed after saving. The documents remain locked and cannot be changed by other users.

⇒ 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.

8. 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.
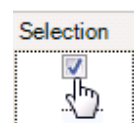
9. Confirm your selection with <OK>.

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

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

5.3   Save automatically

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

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

"**Save automatically**" 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 automatically" for complete assemblies

When an assembly is opened within the SolidWorks 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 automatically**".

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 SolidWorks:

"PRO.FILE" => "Save automatically"

Proceed as follows:

1. Select the "**PRO.FILE**" menu from the menu bar in SolidWorks.

2. Select the function "**Save automatically**" from the menu ribbon.

⇒ 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.
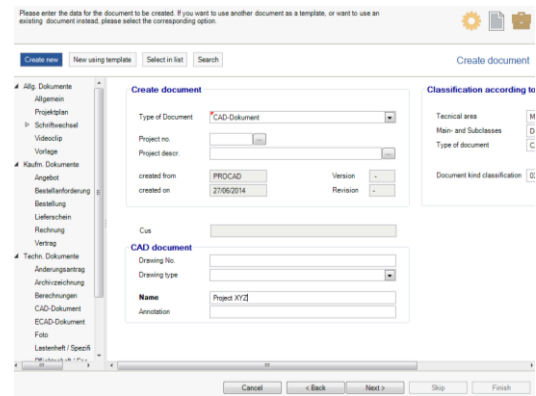
3.   Go through all steps of the Checkin wizard for the first new CAD document. Detailed information on this can be found in the previous chapter "Saving CAD objects for the first time".

⇒   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 automatically**" 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 automatically**>, 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: "Required fields" and <Save automatically>

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 automatically"
	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 – SolidWorks.

5.4   Incremental save

The incremental save function of the integration is aimed at saving assemblies and parts faster to PRO.FILE.

To achieve this, the sometimes time-consuming functions of the "standard" saving process are modified: The scope of the search for drawings and sub-assemblies to be saved is reduced. This means that when this function is used, only the active level of an assembly and the level below are searched for new or modified components and these components are offered for saving. If new or modified components are found, these are further searched (sub-levels) until no more new or modified components are found.

The function "**Save incremental**" is different from the function "**Save**" in the following limitations:

- The function "**Incremental save**" does not search the active folder for corresponding drawings.
- The function "**Incremental save**" does not display the list of locally modified documents in the Check-in wizard. It will only save new or modified documents that are part of the active assembly or the next sub-level of the structure.

	Function call from the PRO.FILE menu in SolidWorks:
	"PRO.FILE" => "Save" => "Incremental save"

The further proceeding for the function "**Incremental save**" corresponds to the proceeding described in the chapter "Saving CAD objects for the first time".

5.5   Incremental save automatically

This function "Incremental save automatically" differs from the "Incremental save" function in the fact that document and part master records

Function call:
"PRO.FILE" => "Save more" => "Incremental save automatically"

5.6   Save all instances

The function "**Save all instances**" allows you to save instances of a part family.

If you have generated several instances of one object, this function allows you to save all objects, without having to save every instance separately.

Function call from the PRO.FILE menu in SolidWorks:
"PRO.FILE" => "Save…" => "Save all instances"

Proceed as follows:

1. If you want to save all instances of a part family for the first time in PRO.FILE, you have to save the mother part (generic) in PRO.FILE first.

2. You can then use the function "**Save…**" => "**Save all instances**".

⇒ When the function is used, a query is displayed, whether new part master records are to be created for the instances.

⇒ If the question is answered with "**No**", all instances are assigned to the part master record of the generic.

⇒ If the question is answered with "**Yes**", new part master records are created for all instances.

The further proceeding is identical to the first-time saving of CAD documents to PRO.FILE, as described in the chapter "Saving CAD objects for the first time".

Attention:
Instances (SolidWorks configurations with PRO.FILE reference) must under no circumstances be deleted from the SolidWorks file, since otherwise the PRO.FILE documents and superior assemblies will then be without reference.

5.7   Save instances automatically

The function "Save instances automatically" combines the functions

- [Save all instances]() and
- [Save automatically]()

If this function is used, the user does not have to enter part master record information. The instances are saved to PRO.FILE without user interaction.

	Function call from the PRO.FILE menu in SolidWorks:
	"PRO.FILE" => "Save…" => "Save instances automatically"

The further proceeding is identical to the automatic saving of CAD documents as described in the chapter "[Save automatically]()".

5.8   Save Phantom

The function "**Save Phantom**" is available for assemblies and drawings.

Assemblies can be saved in PRO.FILE under one single parts master with all assembly parts by using PRO.FILE-SolidWorks Integration. Using the function "**Save Phantom**" all objects contained in an assembly will be saved under this parts master.

All components not known yet in PRO.FILE are stored then as "**phantom parts**".

- The elements of the assembly can thus no longer be loaded separately from PRO.FILE.
- As a consequence, this phantom part is treated in PRO.FILE like a single part even if it consists of several parts in SolidWorks. The objects contained in the assembly are listed as phantom parts and cannot be explicitly opened from PRO.FILE.

Definition
- Phantom assembly: An assembly consisting of phantom parts. The assembly is treated in PRO.FILE like a single part.
- Phantom part: The parts within a phantom assembly. These parts cannot be used on their own, as they are "invisible".

Note: When to use this function?

You need the model of a purchase part to be used in a SolidWorks design, because you want to check its assembling conditions, or because fixture holes in your design depend on the purchase part.

For this purpose you may receive a simplified assembly from your supplier, or you make this design yourself.

For phantom parts the following applies:

- Only CAD documents that are not yet saved in PRO.FILE can be saved as phantom parts.

- PRO.FILE treats this assembly afterwards like a single part.

- It is possible to change phantom assemblies or parts.

- Phantom parts may not be used in other phantoms.

- Phantom assemblies result in a position within a bill of materials.

- The change of these models can only be made in the context.

- A versioning/revisioning of the phantom (assembly/part) is possible (is also generally made in the context).

Note:

Documents already known in PRO.FILE can only be saved as phantom parts, if the DB relation is dissolved before (Function "Disconnect relation") and if the file name has been changed for external configurations.

Function call from the PRO.FILE menu in SolidWorks:

"PRO.FILE" => "Save…" => "Save phantom"

Proceed as follows

1. You are in the active assembly with CAD documents that are not yet saved in PRO.FILE.

2. Select the function "**PRO.FILE**" => "**Save…**" => "**Save phantom**".

⇒ The PRO.FILE Checkin wizard displays a list of all CAD documents to be created in PRO.FILE.

3. Confirm this list with <**OK**>.

4. Create the part and document description in the Checkin wizard for the phantom assembly. Confirm these steps with <**Next**>.

⇒ The assembly is saved under one single part master. In the structure you can see that the assembly and **all parts** are summarized under one common part master. The phantom assembly does **not** have a bill of materials.

Note: Phantom assemblies in the bill of materials

The display of the phantom assembly in the bill of materials depends on the configuration. For further information see the configuration manual for the integration PRO.FILE – SolidWorks.

Usage of phantom parts from a phantom assembly

If you want to use elements from a phantom assembly in other designs, you have to detach the element from the phantom assembly.

Since you cannot open the phantom element directly from PRO.FILE, the proceeding is as follows:

Open the phantom assembly from PRO.FILE. Elements of the phantom assembly are copied into your work folder. From here you can open the required element (phantom element) in SolidWorks. If the phantom assembly is still opened in SolidWorks, you can open the phantom part directly from the phantom assembly.

If you now want to save the phantom element via "**Save**", a message will inform you that you have to cut the database relation first.

Note: Externally used phantoms – please cut database relation first

You have to "unlink" the document from PRO.FILE by using the function "**Disconnect relation**" from the integration menu.

Function call from the PRO.FILE menu in SolidWorks:

"PRO.FILE" => "Disconnect relation"

Proceed as follows

1. Select the "**PRO.FILE**" menu from the menu bar in SolidWorks.

2. Select the function "**Disconnect relation**".

⇒ The dialog for the selection of documents to be disconnected is displayed. (Information on the function and status information can be found in the chapter "Data overview: The document list"

⇒ The structure of the phantom part and the assembly, the phantom part is used in, is displayed.

3. Select only the phantom part by using the checkbox in the first column.

4. Confirm your selection with <**OK**>.

The connection of the part to the PRO.FILE database is dissolved and the phantom part is transformed into a **separate** part, which can be used as a new object.

> **Attention: Usage of phantom parts**
>
> Phantom parts **cannot** be referenced. You need to disconnect a phantom part from the phantom assembly in order to use it as a separate object.

5.8.1 Mixed design: Phantom assemblies and PRO.FILE objects

If you are using objects that are already known in PRO.FILE within an assembly, the property of these objects remains intact after using the function "**Save phantom**".

This means that not all parts of an assembly/drawing/weldment, which is saved as a phantom in PRO.FILE, are necessarily transformed into phantom parts.

> **Note: Phantom parts/PRO.FILE parts in mixed assemblies**
>
> Only new objects (that are not known in PRO.FILE) within a mixed assembly are save das phantom parts.

> **Example:**
>
> You have designed an assembly, into which you have inserted parts from PRO.FILE, e.g. four times the part "bol1_00019310".
>
> You save this assembly via "**Save phantom**" and then create a bill of materials: Contrary to a normal phantom assembly, the bill of materials contains the four bolts.
>
> You can use the created phantom assembly like a separate object in other designs.

5.9 Saving of weldments

The integration PRO.FILE SolidWorks (PRO.FILE Release 8.6 in combination with SolidWorks 2015) supports the administration of weldments.

The corresponding cut list elements may be structure parts, caps, gusset plates, etc.

The combination of structure parts into on PRO.FILE part master record is preconfigured in the file properties of the SolidWorks profile file. When a structure part is saved to PRO.FILE, the cut list element is automatically assigned to the corresponding part master record.

The automatic assignment can only be made for structure parts. For gusset plates, end caps, etc., this is not possible, since there are no profile files or similar that may hold the properties for a later automatic assignment. The assignment to a PRO.FILE part master record is made via the PRO.FILE Check-In wizard.

Note: Configuration of structure parts

In order to automatically assign structure parts to the corresponding PRO.FILE part master record in PRO.FILE, the profile file in SolidWorks has to be configured accordingly. For details, see the configuration manual of the integration PRO.FILE Solid Works.

Proceed as follows

1. The saving of weldments in PRO.FILE is made via the Check-In wizard and the function "Save" of the integration (see chapter "Saving CAD objects for the first time")

⇒ When saving with the Check-In wizard, new or modified parts/assemblies/drawings are first saved locally and then in PRO.FILE. For new parts/assemblies/drawings, the required part and document master records are created in PRO.FILE.

⇒ Then the bill of materials is created in PRO.FILE. In this step, the assignment of elements from the cut list to the corresponding part master records is made.

- The cut list elements that are SolidWorks structure parts are automatically assigned.

- Other cut list elements cannot be assigned automatically. This assignment has to be made via a dialog.

⇒ The dialog "Cut list: Assign PRO.FILE parts" is displayed. This dialog lists:

- The ID numbers of the PRO.FILE part maste record, the cut list element has automatically been assigned to.

- The list of elements, for which no automatic assignement could be made.

2. In the dialog list, select the cut list element you want to assign a PRO.FILE part master record to.

3. Click on <**Assign cut list element**>.

⇒ The PRO.FILE wizard is displayed. Via "Select in list" you can select an existing part master record and confirm you selection with <**Open**> (see "Working with the Checkout wizard to search for CAD documents").

4. Once all cut list elements are assigned, confirm the dialog with <**OK**>.

Note: Assignment of cut list elements

The dialog window cannot be closed with <**OK**> before all cut list elements are assigned- If the assignment is aborted via <**Cancel**>, the cut list is not transferred to PRO.FILE. Other functions of the "Save" command are not affected by this.

To save the cut list of a single SolidWorks part to PRO.FILE after cancelling the dialog, the integration command "**Create BOM**" can be used. The assignment dialog is then displayed again.

For larger assemblies containing new parts with assemblies (and thus cut lists), it is recommended to save the parts to PRO.FILE individually before saving the assembly. This reduces the complexity of the saving process.

⇒ After the creation of the BOM that contains SolidWorks parts with weldments, the elements of the cut list are displayed in the BOM.

5.9.1 Save cut list element files as PRO.FILE documents

Cut list elements of weldments are saved in PRO.FILE as parts. The elements of the cut list are then displayed in the PRO.FILE bill of materials. Via the Check-In wizard of PRO.FILE, the files of the SolidWorks profiles are not saved in PRO.FILE, since these are not required for the bill of materials.

To assign these PRO.FILE parts the SolidWorks profile files as PRO.FILE document as well, the elements of the cut list can be saved as separate parts in SolidWorks first and then transferred to PRO.FILE.

To do so, use the SolidWorks command "**Insert into new part**". Via this command, you can save SolidWorks parts of the single cut list elements.

Note:

Please note that it is possible to save several elements in one SolidWorks part. However, it is then not possible to assign these joint elements separate part master records in PRO.FILE.

Proceed as follows

1. When a weldment is designed, the various features are displayed in the SolidWorks cut list.

2. Via the context menu, select the function "**Insert into New Part…**".
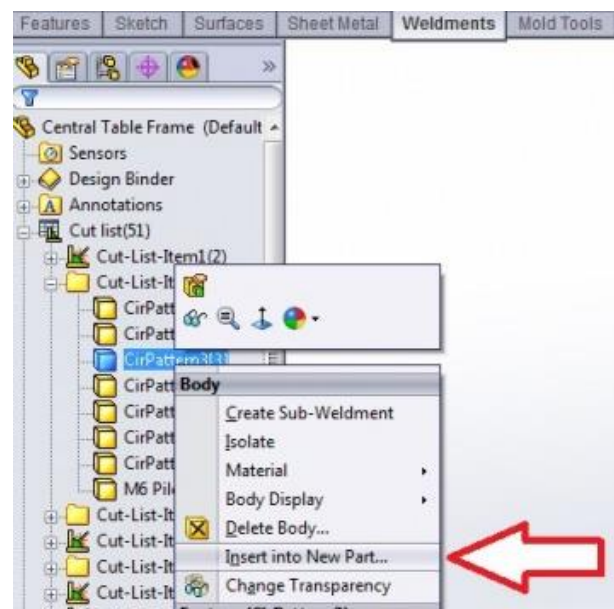
3. Once the property dialog is confirmed with the green checkmark, the "save" dialog of SolidWorks is displayed.

4. Via this dialog, the cut list element can be saved as a local part in SolidWorks.

⇒ The FeatureManager of the cut list element as a part then shows a reference of the cut list element part to the basic part, i.e. the weldment.

> **Note: Changes to the part of the cut list element**
>
> If changes are to be made to the part of the cut list element, these changes have to be made in the weldment. Changes in the weldment are transferred to the part of the cut list element after it is opened in SolidWorks. If the part of the cut list element is open in the background in SolidWorks while the changes are made to the weldment, the changes are immediately

5. Now, select the function "**Save**" from the integration menu.

⇒ Via the PRO.FILE Check-in wizard, the weldment and the part of the cut list element are offered for saving.

6. For the part of the cut list element, a PRO.FILE part has already been created during the initial saving of the weldment to PRO.FILE. In order for the document of SolidWorks part to be linked to the existing part master record in PRO.FILE, this part master record has to be selected via the option "**Select in list**".

7. Confirm your selection with <**Finish**>.

⇒ After the part has been successfully saved in PRO.FILE, it is also displayed in the structure browser of the weldment in PRO.FILE:
The SolidWorks part of the cut list element is linked to the PRO.FILE document record below the part master record. The weldment is displayed as an external reference.

⇒ The saving of the cut list element file in PRO.FILE is thus finished.

5.10   Managed Copy

Managed Copy helps the designer engineer working in design modification (assemblies, subassemblies, parts) with the specific copy of models within an assembly structure. Entire machines can be cloned, including all referenced data and drawings.

Managed Copy therefore enables a specific selection of the models to copy within an assembly structure. It is up to the user which elements within an assembly structure are saved as the new copy.

Simultaneously the title of the copied components can be adjusted. Finally a bill of materials in PRO.FILE is derived.

To get the desired result of Managed Copy, there are specific prerequisites and approaches that must be **observed strictly**. See the following chapters for more information.

5.10.1   Exchanged or not: What must be observed strictly?

The function "Managed Copy" enables to copy whole assembly structures and select for each model (assembly, subassembly, CAD part) within a structure whether the model itself or only the reference to this model is copied.

To get the desired result, the basic connections are must **strictly** be observed.

> ⚠️ **Attention: Result of Managed Copy**
>
> The result of "Managed Copy" depends on the CAD documents opened in the SolidWorks session and the CAD document selected for "Managed Copy"!
> If higher-level assemblies are opened in the SolidWorks session, a subassembly /CAD part, for which the function "Managed Copy" has been selected, is **exchanged** in these assemblies!
>
> **If you want to make sure that no accidental exchange takes place in other assemblies, do not load additional assemblies in the SolidWorks session.**

The approach of Managed Copy for models (assemblies, subassemblies and parts) is determined by the following requirements:

- **Requirement 1**: You want to create an independent copy of a model?
- **Requirement 2**: You want to exchange a subassembly/CAD part within one or several assemblies by a copy created with "Managed Copy"?

For each of these requirements there are two possible approaches, which are described in the following.

Requirement 1   **Create an independent copy of a model**

The requirement is:

- You want to create a copy of an existing model (assembly, subassembly, CAD part).

- The reference of the higher-level assembly should furthermore refer to the original model, **not** to the created copy.

- Is there a reference from the model you want to copy to a higher-level assembly, the references should not be exchanged but furthermore refer to the original model.

- The created copy of the assembly should be saved independently in PRO.FILE.

To do this, the two possible **approaches must strictly** be observed:

Approach 1A   Only the model you want to copy is loaded in SolidWorks

1. Close **all** higher-level assemblies with references to the model to copy in the SolidWorks session.

2. Open the model to copy via the "Managed Copy" function in the SolidWorks session.

> **Note: higher-level assemblies are not be opened**
>
> Using this approach, all higher-level assemblies have to be closed! If higher-level assemblies are opened in the SolidWorks session, a model copied via "Managed Copy" is exchanged in the higher-level assemblies.

3. Activate the model to copy in the SolidWorks session.

4. Call the PRO.FILE function "Managed Copy" up, as described in the following chapter "How is the function "Managed Copy" executed?".

⇒ The created copy of the model is saved in PRO.FILE.

⇒ The created copy of the model is referenced in no higher-level assembly.

Approach 1B   "Managed Copy" is executed via the drawing of the subassembly you want to copy

1. Open the **drawing** of the subassembly to copy via the "Managed Copy" in the SolidWorks session.

⇒ Higher-level assemblies in which the model to copy is referenced, can remain open.

2. Activate the drawing of the subassembly to copy in the SolidWorks session.

3. Call the PRO.FILE function "Managed Copy" up, as described in the following chapter "How is the function "Managed Copy" executed?".

4. Select both the drawing and model to copy in the wizard of "Managed Copy" as well as the subassemblies and CAD parts in the structure of the model you want to copy.

⇒ The created copy of the drawing as well as the copy of the subassembly are saved in PRO.FILE.

⇒   The created copy of the model is referenced in no higher-level assembly.


Note: In the structure the drawing of a model is listed above the model

Concerning the dependency of the references, drawings are listed above the model!

The higher-level assemblies therefore do not refer to the drawing of the subassembly.

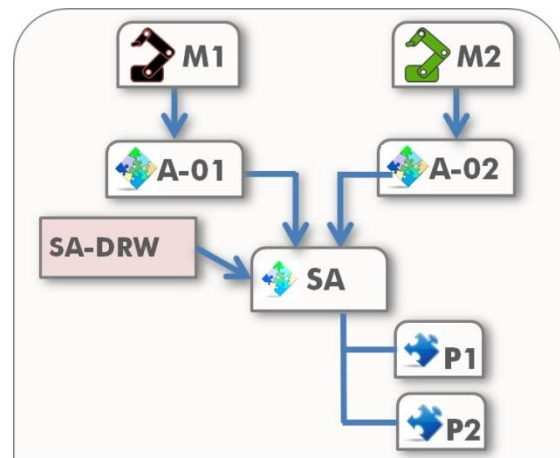Due to this reason, using the approach 1B, the references in the opened, higher-level assemblies are not automatically updated to the copy of the subassembly.
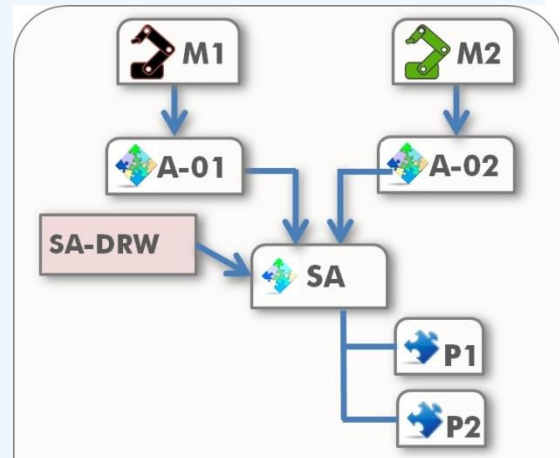
Case study for approach 1B

Do not exchange "SA", neither in "A-01" nor in "A-02"

The following case study explains which results "Managed Copy" provides in dependence of the loaded SolidWorks session and the activated CAD documents.

Situation:

- 2 assemblies ("A-01" and A-02") are loaded in SolidWorks.

- Assembly "A-01" is part of the machine "M1", assembly "A-02" is part of the ma-chine "M2".

- The subassembly "SA" is installed in both assemblies.

- There is the drawing "SA-DRW" for the subassembly.

- The subassembly "SA" is active in the SolidWorks session.

- **The function "Managed Copy" is called up for "SA-DRW"!**

- The sub-assemby "SA" and the drawing "SA-DR" are selected for "Managed Copy".

- Part "P2" is selected for "Managed Copy", Part "P1" is not.

Result:

- The subassembly "SA" is copied via "Managed Copy".

- The references are explicitly exchanged in the higher-level drawing "SA-DRW" by the Integration PRO.FILE SolidWorks.

- Due to the fact that "Managed Copy" has been executed via the drawing, the assemblies "A-01" and "A-02" furthermore use the "original" subassembly "SA".

- A copy of CAD part "P2" is created, to which the "Copy of "SA" refers.

- Like "SA", "Copy of SA" refers to the not copied CAD part "P1"
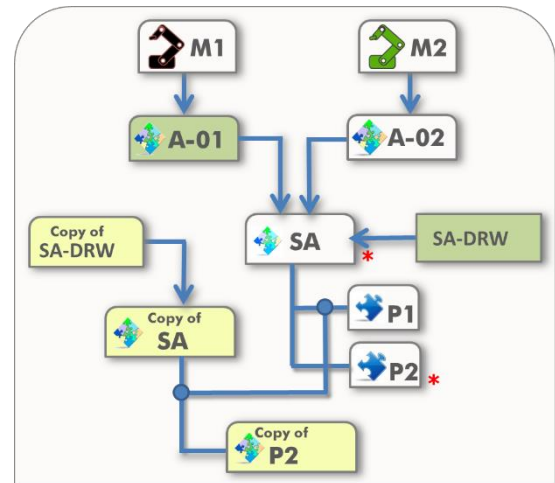
Requirement 2   **Exchange a model in an higher-level assembly using "Managed Copy"**

The **requirement** is:

- You want to create a copy of an existing model (assembly, subassembly, CAD part).

- The copy of the model **should** exchange the original model.

- The references in the higher-level assemblies should be exchanged and refer to the copied model.

To do this, the two possible **approaches must strictly** be observed:

Approach 2A   **Exchange the model in several higher-level assemblies**

1. Open **all** higher-level assemblies in which you want to exchange the model to copy in the SolidWorks session.

2. Open the model to copy via the "Managed Copy" function in the SolidWorks session.

> **Attention: higher-level assembly opened**
>
> Using this approach, all higher-level assemblies, in which the copied model should be exchanged, have to be opened.

3. Activate the model to copy in the SolidWorks session.

---

4. Call the PRO.FILE function "Managed Copy" up, as described in the following chapter [How is the function "Managed Copy" executed?](#).

⇒   The created copy of the model is saved in PRO.FILE.

⇒   In all higher-level assemblies, which are loaded in a SolidWorks session, the references are updated and refer to the copied model.

⇒   Higher-level assemblies are not automatically saved to PRO.FILE.

> **!**   **Attention: Higher-level assemblies are not saved automatically**
>
> The references in higher level assemblies are only updated and point to the copied model after these higher-level assemblies are loaded in SolidWorks and explicitly saved via the function "Save" of the integration.

Case study for approach 2A

Exchange "SA" in "A-01" and "A-02"

The following case study explains which results "Managed Copy" provides in dependence of the loaded SolidWorks session and the activated CAD documents.

Situation:

*   2 assemblies ("A-01" and A-02") are loaded in SolidWorks.

*   Assembly "A-01" is part of the machine "M1", assembly "A-02" is part of the machine "M2".

*   The subassembly "SA" is installed in both assemblies.

*   There is the drawing "SA-DRW" for the subassembly.

*   The subassembly "SA" is active in the SolidWorks session.

*   The function "Managed Copy" is called up for the subassembly "SA".

*   The subassembly "SA" itself is selected for "Managed Copy".

*   CAD part "P2" is selected for "Managed Copy", CAD part "P1" is not.

Result IF the drawing was also selected for Managed Copy:

- Due to the fact that "SA" as an active document has been selected to copy via "Managed Copy" and both assemblies "A-01" and "A-02" are loaded in SolidWorks, the references in both assemblies are exchanged by SolidWorks.

- In both assemblies the copied subassembly "Copy of SA" is installed.

- A copy is created of the drawing SA-DRW. "Copy of SA-DRW" refers to "Copy of SA".

- A copy of CAD part "P2" is created, to which the "Copy of SA" refers.

- Like "SA", "Copy of SA" refers to the not copied CAD part "P1".
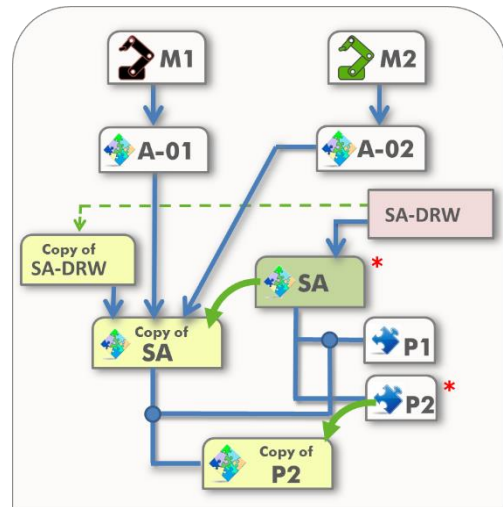
Result IF the drawing was NOT selected for Managed Copy and NOT loaded in SolidWorks:

- Due to the fact that "SA" as an active document has been selected to copy via "Managed Copy" and both assemblies "A-01" and "A-02" are loaded in SolidWorks, the references in both assemblies are exchanged by SolidWorks.

- In both assemblies the copied subassembly "Copy of SA" is installed.

- No copy is created of the drawing SA-DRW. The drawing SA-DRW still refers to "SA".

- A copy of CAD part "P2" is created, to which the "Copy of SA" refers.

- Like "SA", "Copy of SA" refers to the not copied CAD part "P1".
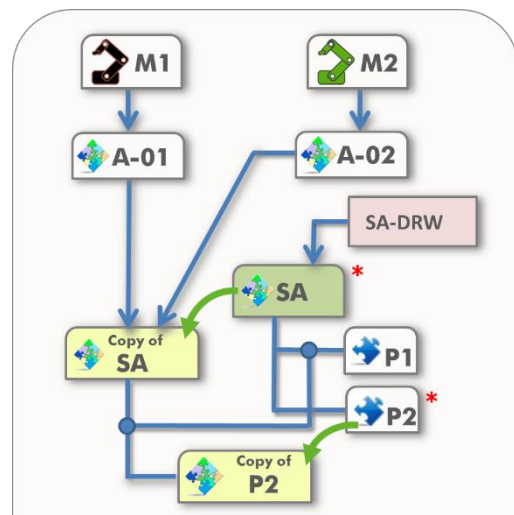
> **Behavior of the drawing**
>
> Concerning the dependency of the references, drawings are listed above the model!
>
> - If the drawing "SA-DRW" were loaded in the SolidWorks session, it would refer to "Copy of SA", too.
> - If the drawing "SA-DRW" is not loaded, but already added to the PRO.FILE structure, the reference is explicitly exchanged and updated by the Integration PRO.FILE-SolidWorks.

Approach 2B

Exchange a model in a specific assembly

1. Open and activate the higher-level assembly, in which the model (subassembly/CAD part) should be exchanged via "Managed Copy" in the SolidWorks session.

   ⇒ Additional higher-level assemblies, in which the model to copy is referred but should not be exchanged, can remain open.

2. Call the PRO.FILE function "Managed Copy" up, as described in the following chapter "How is the function "Managed Copy" executed?".

3. Select in the wizard of "Managed Copy"

   - **Not** the higher-level assembly, which is shown as the top element (top element in a structure).

   - **Only** the model to copy - as well as the subassemblies and CAD parts in the structure of the model.
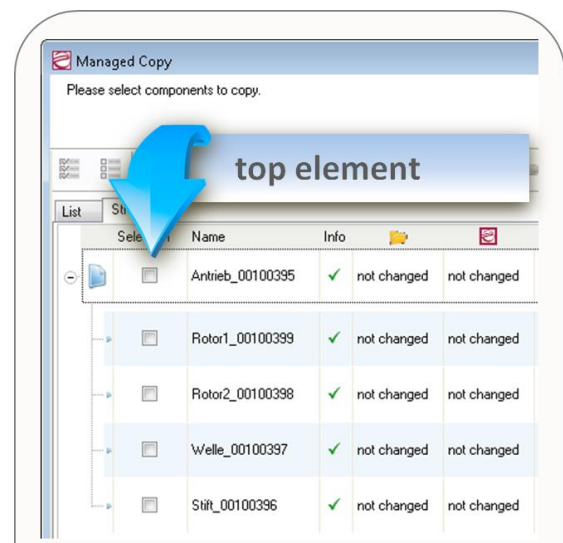
⇒ The created copy of the model is saved in PRO.FILE.

⇒ The created copy of the model is exchanged in the assembly for which the function "Managed Copy" has been called up.

	Note: Using this approach, only the references are exchanged
	The approach avoids the automatic exchange of models copied with "Managed Copy" in opened, higher-level assemblies.
	The automatic exchange via "Managed Copy" would only access the assembly chosen as the top element, but not the lower level, in which the model to copy is located.
	Due to this reason, using this approach the copied model is exchanged only in the assembly selected for "Managed Copy", but not in the other opened assemblies, in which it is installed.

	Note: drawings do not have to be explicitly loaded in the session
	To include the drawings, they do not have to be loaded explicitly in the session! Even due to reasons of performance and maybe unintended effects to the automatic exchange of models this is not recommended.

Case study for approach 2B

Replace "SA" only in "A-02"

The following case study explains which results "Managed Copy" provides in dependence of the loaded SolidWorks session and the activated CAD documents.

Situation:

- 2 assemblies ("A-01" and "A-02") are loaded in SolidWorks.

- Assembly "A-01" is part of the machine "M1", assembly "A-02" is part of the machine "M2".

- The subassembly "SA" is installed in both assemblies.

- There is the drawing "SA-DRW" for the subassembly.
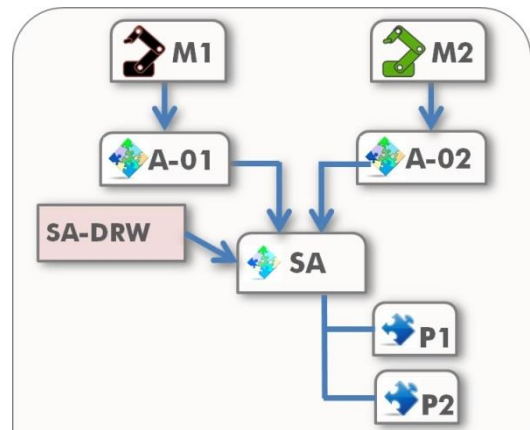
Exchange "AS" via "Managed Copy" only in "A-01"

- The function "Managed Copy" is called up for the assembly "A-01".

- In the wizard the subassembly "AS" is selected for "Managed Copy", the assembly "A-01" itself is not **selected**.

- CAD part "P2" is selected for "Managed Copy", CAD part "P1" is **not**.

!
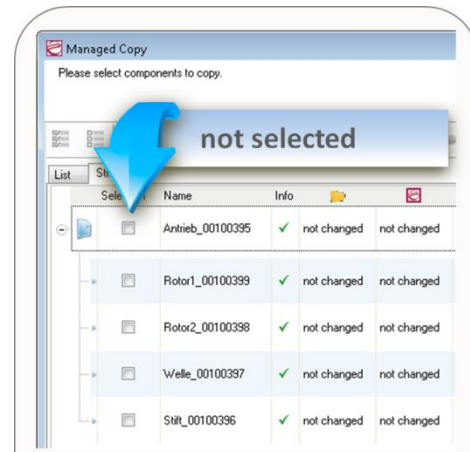
When the sub-assembly is selected, the drawing is automatically selected as well

When a sub-assembly is selected in the dialog of Managed Copy, the drawing linked to this sub-assembly in PRO.FILE is also activated for Managed Copy and thus copied. This affects the result of Managed Copy as the following examples show.

For the cases, in which the drawing is not to be copied, the drawing has to be deactivated in the Managed Copy dialog.

Result, IF the drawing is also selected for Managed Copy:

- The sub-assembly "SA" is copied with "Managed Copy".

- The integration explicitly exchanges the references in assembly A-01, so that these references point to the copied sub-assembly "Copy of SA".
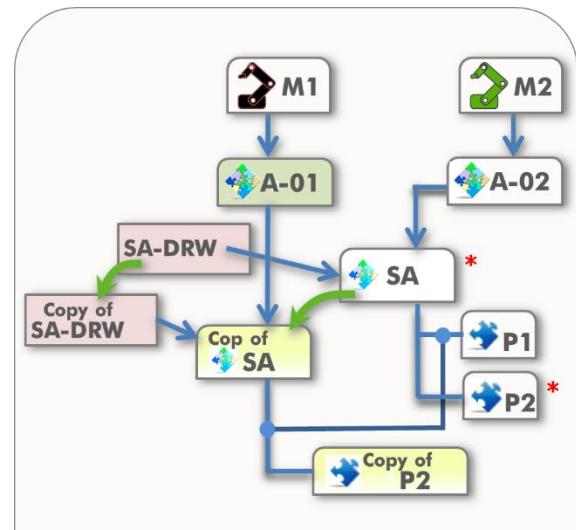
- The drawing "SA-DRW" is copied. The drawing "SA-DRW" refers to the sub-assemblly SA, the drawing "Copy of SA-DRW" refers to the copy of "SA".

- The assembly "A-02" still refers to sub-assembly "SA".

- A copy is created of "P2", which is referenced by "Copy of SA".

- Like "SA", "Copy of SA" refers to the not copied part "P1".

Result, IF the drawing is NOT selected for Managed Copy and was not opened in the SolidWorks session:

- The sub-assembly "SA" is copied with "Managed Copy".

- The integration explicitly exchanges the references in assembly A-01, so that these references point to the copied sub-assembly "Copy of SA".

- The assembly "A-02" still refers to sub-assembly "SA".
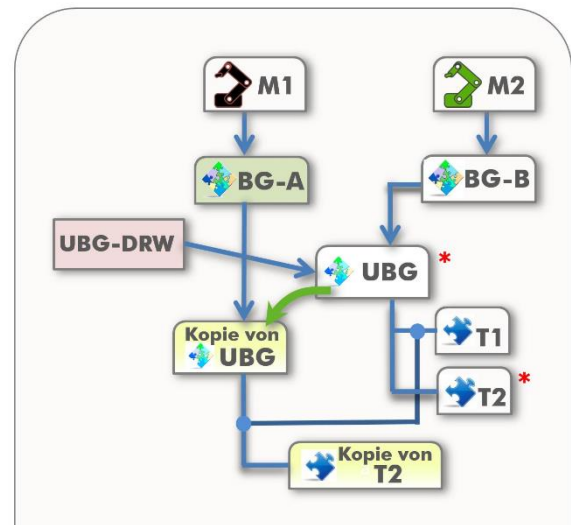
- The drawing "SA-DRW" refers to "Copy of SA"

- A copy is created of "P2", which is referenced by "Copy of SA".

- Like "SA", "Copy of SA" refers to the not copied part "P1".

5.10.2   How is the function "Managed Copy" executed?

> **Attention: Result of Managed Copy**
>
> The result of "Managed Copy" depends on the CAD documents opened in the SolidWorks session and which CAD document is selected for "Managed Copy"!
>
> See the previous chapter: "[Exchanged or not: What must be observed strictly?](#)".

> **Function call out of the PRO.FILE menu in SolidWorks:**
>
> "PRO.FILE" => "Managed Copy..."

Proceed as follows:

1. Select the menu item "**PRO.FILE**" in the menu bar of SolidWorks.

2. Click on "**Managed Copy**" in the menu bar of the PRO.FILE Integration.

⇒ The wizard of Managed Copy starts and supports you in your further approach.

⇒ The Integration PRO.FILE-Solid Works always determines the substructure **based on the active CAD document**. The substructure of the CAD document depends on the CAD system.

⇒ In a second step the substructure is expanded by the related drawings. This "special provision" is required because the drawings are listed **above the model depending on references**.

⇒ In the wizard of "Managed Copy" the tree structure, determined and expanded by drawings, is shown, so that the documents to copy with their dependent data (nodes) can be selected.

⇒ The top node and the first step are already folded out. Further steps can be folded out by a click on the structure symbol ⊕.
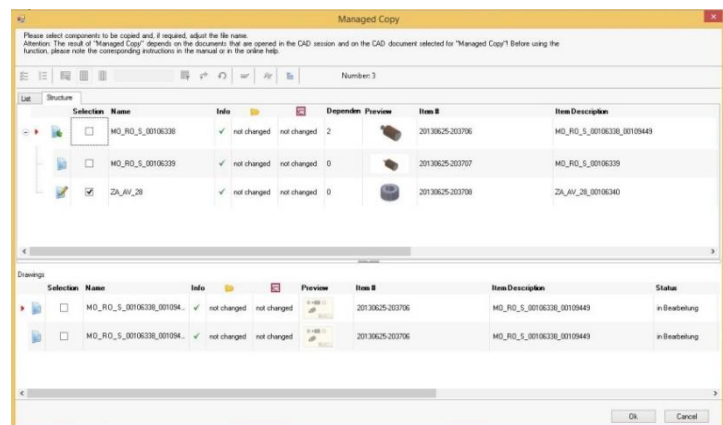
⇒ The column "Info" contains further information, e.g. when a part cannot be copied.

⇒ The "status" columns shoes the current processing status of an object in the working directory and in PRO.FILE (see chapter: "Up to date or not: Display of status information").

3. **Select**: Select all components which you want to save as a new copy in PRO.FILE. Therefore activate the checkbox in the listed CAD documents as shown on the right.

Note: Exchange of components in assemblies

If components in an assembly are selected for "Managed Copy", but not the assembly as top element, the components will be exchanged by the created copy. Thus the assembly in PRO.FILE is changed!

4. **Edit file name**: You can edit and adjust the file names directly in the list, by a click on the column name.

5. Execute this selection and editing of file names for all components to copy.

6. Confirm with <OK>.

⇒ If you click on <OK>, the PRO.FILE database reference for all selected objects is deleted. Afterwards the thus created local copies are checked into PRO.FILE. For all not selected components only the references are copied.

7. To complete the process "Managed Copy", all selected components have to be added to the newly created part and document descriptions.

⇒ Therefore appears:

• The check-in wizard to add the part description in PRO.FILE

• then (depending on the configuration) the check-in wizard to add the document description in PRO.FILE

• finally (depending on the configuration) the check-in wizard for the project assignment in PRO.FILE

- The information is requested for each selected component.

⇒ You will find Information on how to use the check-in wizard in the previous chapter "[Save: How to save CAD data and changes to PRO.FILE?](#)".

Note:

Also in assemblies that are not explicitly selected for "Managed Copy" the reference to the subassemblies/CAD parts is exchanged locally (in the working directory and the interface).

In a second step, these changes to existing PRO.FILE assemblies are offered to save in PRO.FILE via "saving of changes".

Thereby the user can choose which local changes he actually wants to take in PRO.FILE. The integration also checks the user and status authorization during the saving process.

⇒ Finally a bill of materials is derived for the "cloned" assembly. The process "Managed Copy" is thus finished.

5.10.3 How is the proceeding in "Managed Copy" for drawings?

Due to the fact that drawings are listed in the CAD structure above the models, there is not direct method in the CAD systems itself to determine related drawings.

- For the models and drawings managed in PRO.FILE this reference can be determined via the PRO.FILE document usage.

When CAD models without PRO.FILE relation are saved, the detection of related drawings is only possible with limitations:

- The integration can scan the work folder for drawings and check whether these reference the model.
- **Attention**: Drawings that are not in the same work folder than the current model, for which the function "Managed Copy" is used, cannot be found by the integration.

If drawings are found for the model, they are offered in a list. Via this list, the user can select the drawings to be included in the "Managed Copy" process.

Note: drawings don't have to be loaded in the session

To include the drawings, they don't have to be loaded explicitly in the interface!

Even due to reasons of performance and maybe unintended effects to the automatic exchange of models this is not recommended.
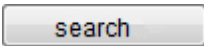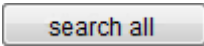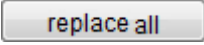

5.10.4 Search and replace with Managed Copy

The file names of assemblies and parts can be edited during the execution of "Managed Copy". This is also possible via the function "search and exchange".

Proceed as follows:

1. Select the list view in the "Managed Copy" window.

2. Activate the display "search and exchange" by a click on the button

3. Enter a string to search for in the field "search".

4. Enter a string in the field "exchange" with which the string to search should be exchanged.
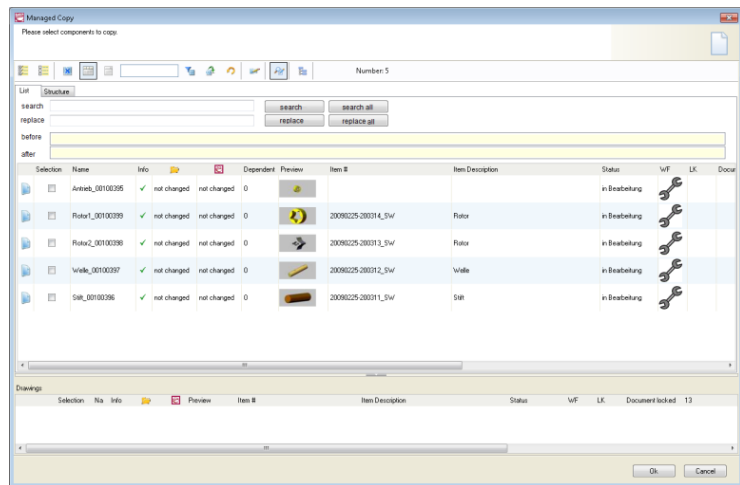
Now you have different possibilities, to execute "search and exchange".

search	searches and selects the next hits in the list
search all	searches and selects all different hits in the list
replace	replace the next hit
replace all	replace all hits in the list

A preview for the editing of the file name is shown in the fields "before" and "after"

Search and exchanged in finished by executing the function "Managed Copy".

5.11 Managed Copy automatically

The function "**Managed Copy automatically**" combines the functions

- [Managed Copy](#) and
- [Save automatically](#)

Attention: Result of von Managed Copy

The result of "Managed Copy" depends on the CAD documents opened in the SolidWorks session and the CAD document selected for "Managed Copy"!

If higher-level assemblies are opened in the SolidWorks session, a subassembly /CAD part, for which the function "Managed Copy" has been selected, is **exchanged** in these assemblies!

If you want to make sure that no accidental exchange takes place in other assemblies, do not load additional assemblies in the SolidWorks session.

The selection of components to copy takes place like with "Managed Copy".

At the creation of these selected components – according to "save automatically" – no further input from the user is required during the creation of the part and document description. The object(s) are saved to PRO.FILE automatically without any possibility to interfere.

You will find detailed information for this process in chapter "Save automatically".

Function call out of the PRO.FILE menu in SolidWorks:

"PRO.FILE" => "save other" => "**Managed Copy automatically**"

- The selection for "Managed Copy automatically" corresponds to the approach described in the chapter "Managed Copy".
- The further steps correspond to those of the automatic saving of CAD documents in PRO.FILE, as you can learn from the chapter "Save automatically".

Note:

"**Managed Copy automatically**" distinguishes from "**Managed Copy**" by the fact that the meta data for the filling in PRO.FILE are not required individually.

5.12 Save NDF (Neutral data format)

The integration PRO.FILE SolidWorks offers the possibility to convert a SolidWorks drawing into a neutral format (e.g. PDF, Tiff, …) and to save this document in PRO.FILE.

By using the function "Save NDF" a neutral format document is created and then attached automatically to the part master record of the drawing.

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

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 NDF**" 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.

Function call from the PRO.FILE menu in SolidWorks:

"PRO.FILE" => "Save" => "Save NDF…"

Create a neutral data format document

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 SolidWorks.
3. Select the function "**Save NDF**" from the menu area "**Save..**"
⇒ 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 NDF" is used. This way, changes can actively be documented by the design engineer.

5.13   Save as new version

With the PRO.FILE-SolidWorks 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.

Note:

If you want to use several different versions of one component within an assembly, a new file name has to be generated for the component version. For this purpose, the parameter "Create new file name when versioning" has to be configured in the PRO.FILE Management Console, see manual "CAD supported by PRO.FILE".

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 SolidWorks.

- If a part is versioned in this way using PRO.FILE-SolidWorks 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. A related drawing will also be versioned and a new structure created. Multi-layered assemblies must also be versioned layer by layer from bottom to top.

	Function call from the PRO.FILE menu in SolidWorks:
	"PRO.FILE" => "Version" => "Save as new version"

Proceed as follows:

1. Select the "PRO.FILE" menu from the menu bar in SolidWorks.

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 SolidWorks.
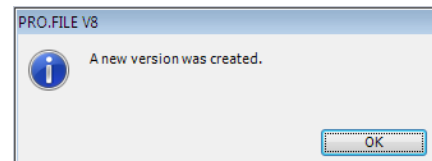
Attention: New version is not locked

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 SolidWorks. You load the part and save it as a new version. For the drawing of the part no new version is created!

---

6   Linking of additional files

It is possible to link additional files to SolidWorks objects. Such additional files may be text or image documents as well as CAD components from other CAD systems

Note:

CAD components from other CAD systems that have been saved with another CAD integration, have to be copied to the Workcenter folder via the function "Supply document" before they can be used.

For additional files, the following functions are available:

- [Link local file](#)
- [Link document](#)
- [Dissolve document link](#)

When opening a SolidWorks object, the linked additional files are copied into the Workcenter folder. Later changes to the additional file always have to be made in the local Workcenter folder.

For SolidWorks objects, additional files are displayed in the PRO.FILE tab "Browser", as well as in the selection dialogs for "Managed Copy" and "Disconnect relation". The versioning of bills of materials ignores additional files.

6.1   Link local file

The function "**Link local file**" is used to add files to the structure that are not yet saved in PRO.FILE.

1.   First, load a SolidWorks object that has been saved in PRO.FILE into your CAD session.

2.   Select the function "**Save…**" => "**Link local file**".

⇒   An Explorer window opens.
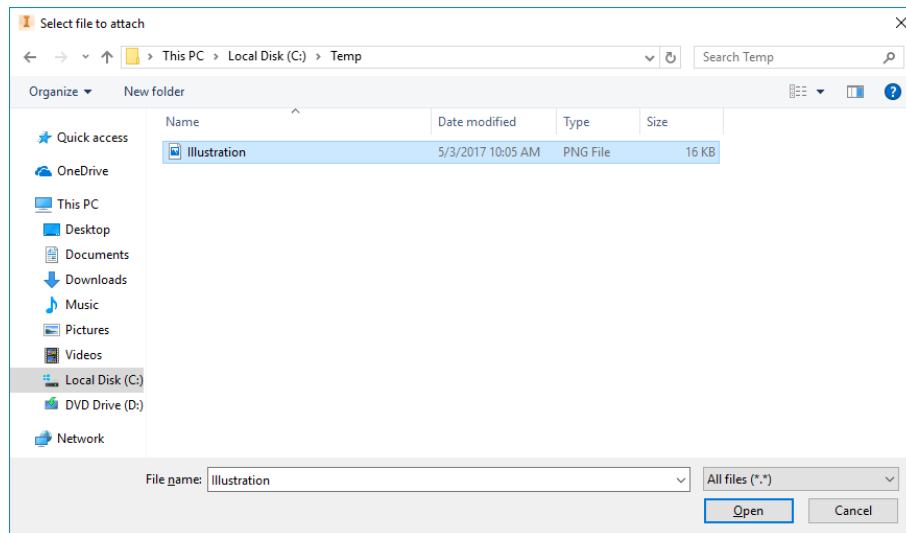
3.   Select the file to be added and confirm your selection with <**Open**>.

⇒   The part master record of your SolidWorks object is displayed in PRO.FILE.
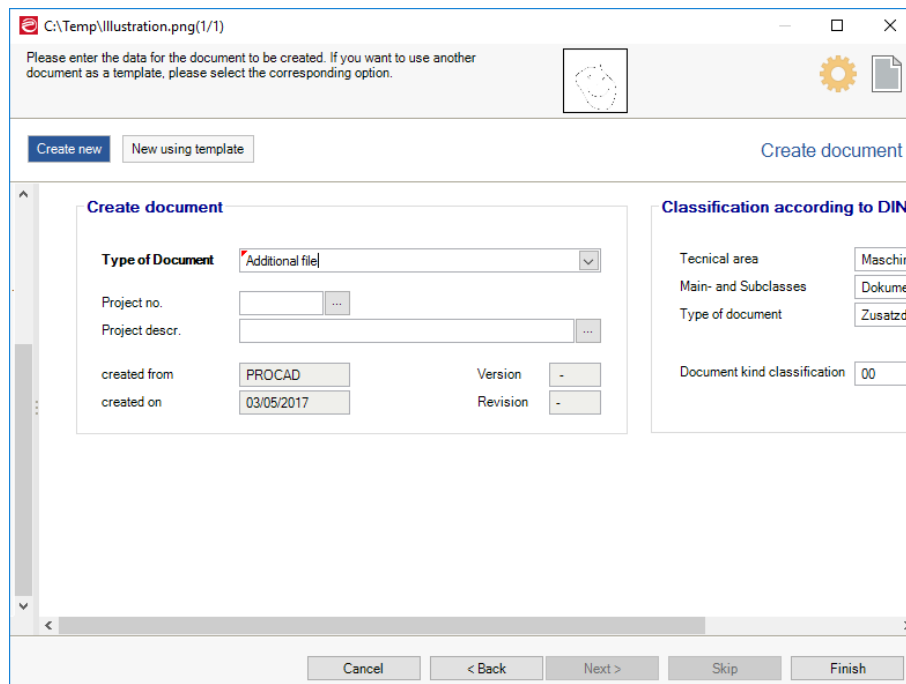
4.   Confirm the assignment with <**Next**>.

⇒ The dialog for the creation of a document master record for the additional file is displayed. By default, the document type is set to "Additional file".

5. Enter the information for the new document record and confirm your input with <Finish>.
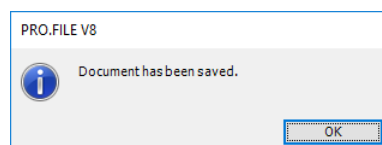
⇒ The additional file is saved in PRO.FILE. It is linked below the document description of the SolidWorks object. If possible, a preview file is created for the additional file.

⇒ By adding it to the SolidWorks structure, the additional file is automatically copied into the Workcenter folder.

6.2   Link document

The function "**Link document**" is used to add files to the structure that are already saved in PRO.FILE.

1. First, load a SolidWorks object that has been saved in PRO.FILE into your CAD session.

2. Select the function "**Save…**" => "**Link document**".

⇒ The PRO.FILE Checkout wizard opens.
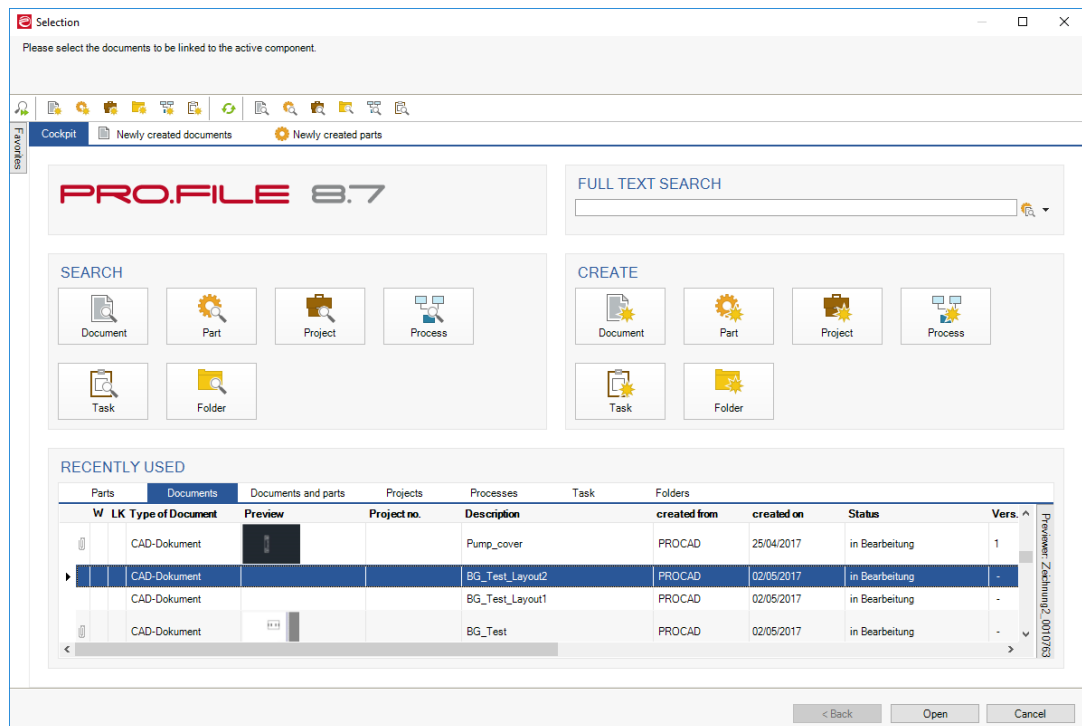
3. Select the document record of the file to be added and confirm your selection with <Open>.

⇒   The document record with the additional file is linked below the document description of the SolidWorks object.

6.3   Dissolve document link

PRO.FILE prevents the deleting of documents as long as the documents are still in use – this also applies for additional files.

The function "Detach document" is used to remove the link of the additional file to the CAD object.

1. First, load a SolidWorks object that has been saved in PRO.FILE (and that contains the additional file) into your CAD session.

2. Select the function "**Save**" => "**Link…**" => "**Dissolve document link**".

⇒   The dialog for the selection of additional files to be detached is displayed.
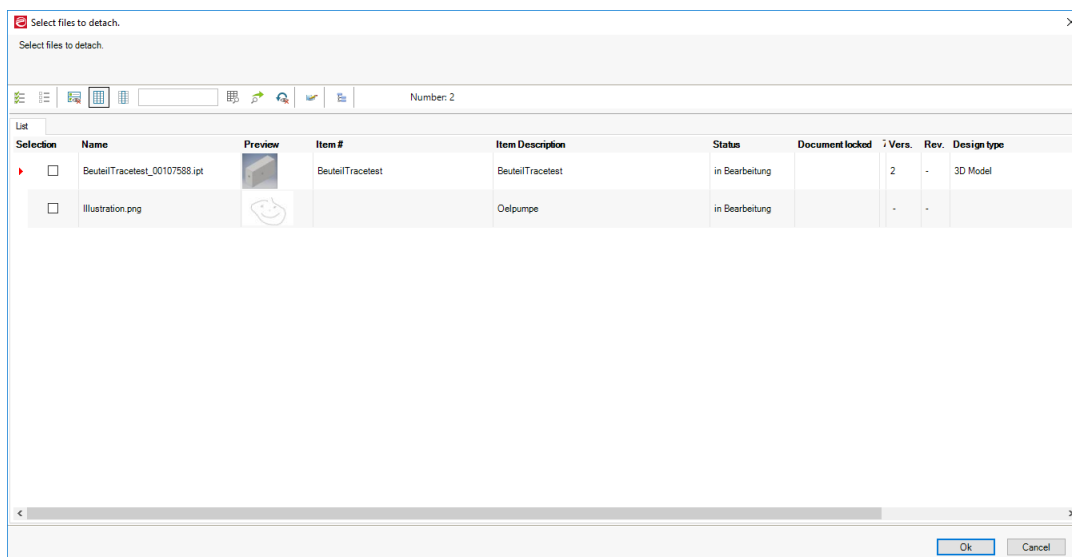
3. From the displayed list, select the additional file you want to detech by activating the corresponding checkbox in the column "Selection".

4. Confirm your selection with <OK>.
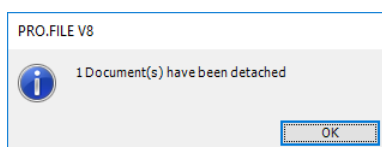
⇒   The selected document is removed from the SolidWorks object structure.

7   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.
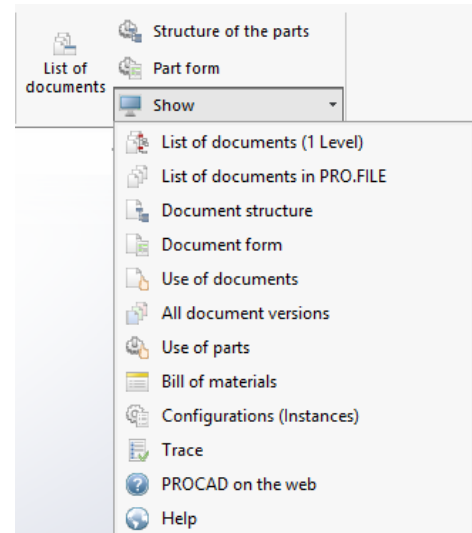
The following information can be found in the area "**Show**":

- [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.

7.1   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 SolidWorks:

"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.
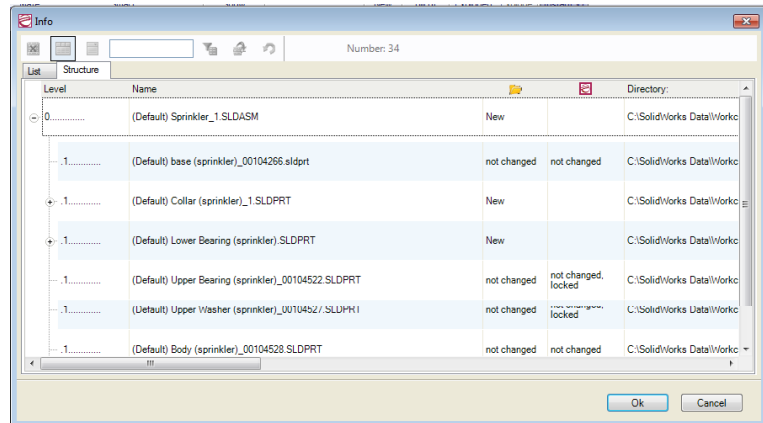
- Information regarding the status of the currently active CAD document.

  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](#)

7.2   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 SolidWorks.

- 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 displayed the desired information.

- Within the displayed lists and forms, you can perform all actions available in PRO.FILE.
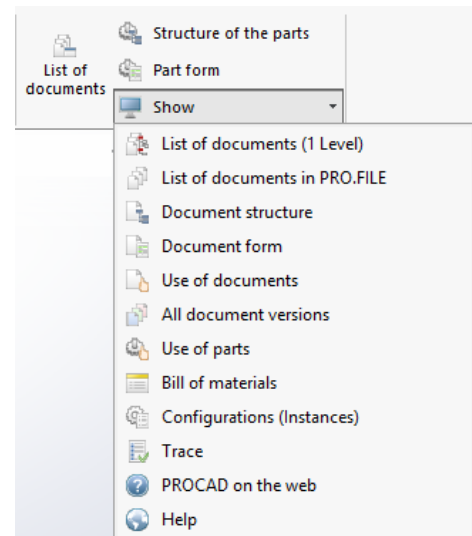
	Function call from the PRO.FILE menu in SolidWorks:
	"PRO.FILE" => "Show" => "…"

	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.

7.2.1   List of documents

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 "Data overview: The document list".

7.2.2   List of documents (1 level)

For complex assemblies the document list can be quite large and therefore may require some time to be displayed.

The display "**Document list (1 level)**" only lists the CAD objects of the first sub-level of an assembly. Apart from this, the display is identical to the "[List of documents]".

7.2.3   List of documents in PRO.FILE

With the function "**Document list in PRO.FILE**" PRO.FILE is started and displays all CAD data currently loaded in SolidWorks in a list. Contrary to the display option "Document list", no separate window is started in PRO.FILE, but the default list view.

7.2.4   Document structure

With the function "**Document structure**" you can see which documents (= part drawings) are used in your drawing (= main drawing).

7.2.5   Document form

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.
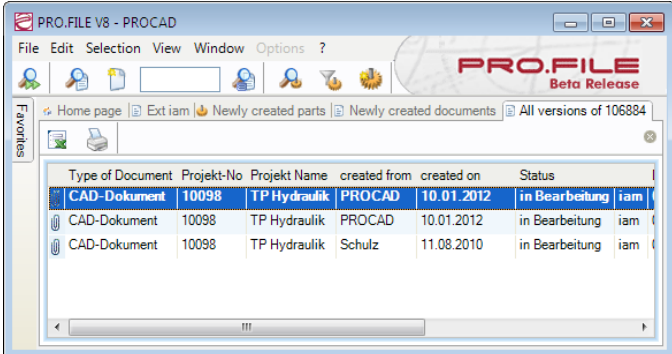
7.2.6   Document usage

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.

7.2.7   All document versions

The function "**all document versions**" displays al visible current and old versions of your CAD document.

The marker "*****" indicates the current version.

7.2.8   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.

7.2.9   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.

7.2.10   Part usage

With the function "**Part usage**" you can see whether you current CAD document is used by other assemblies.

The usage list displays the "upward" structure.

7.2.11   Bill of materials

The function "**Bill of materials**" displays the PRO.FILE bill of materials for the active drawing.

7.2.12   Configurations (instances)

The integration PRO.FILE – SolidWorks offers the possibility of assigning configurations to a part master in different ways.

Three basic types are available:

- Several configurations are assigned to a part via a single document record.
- Configurations are assigned to the part via separate document records.
- Each configuration is assigned to a separate part.

The function "**Show**" => "**Configurations**" is available to manage the various configurations of a part directly via integration.

Function call from the PRO.FILE menu in SolidWorks:

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

After the call up of this function the PRO.FILE information window for the configurations appears:
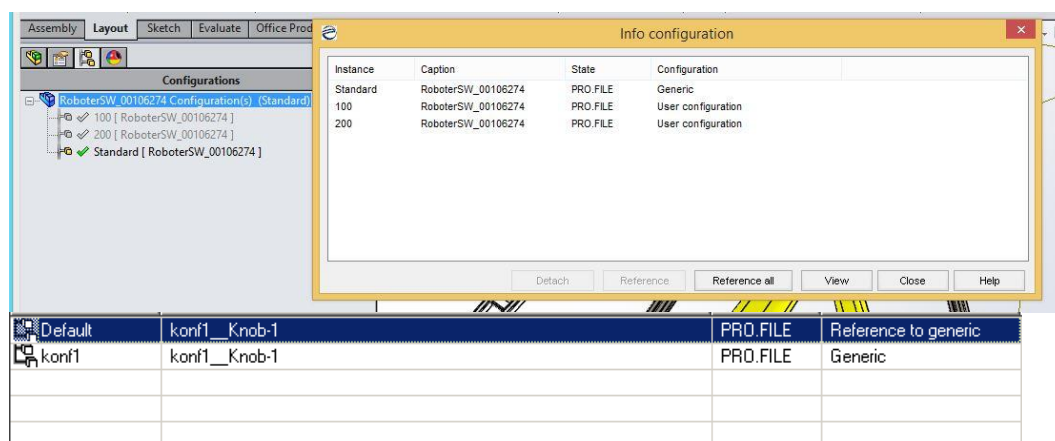
This info list provides the details on the following:

- **Instance:**

  Provides the name of the instance.

- **Designation:**

  Provides the file name of the instance.

- **Status:**

  Provides the status of the instance. The entry "PRO.FILE" shows that the instance is already saved in PRO.FILE. The entry "CAD", only describes a locally-saved instance.

- **Type of Configuration:**

  There are several settings for the configuration types:

  - **Generic:**

    The Instance stored in PRO.FILE first is described as a "Generic". With this Generic the geometry is also stored in PRO.FILE. All further instances that are stored in PRO.FILE refer to this original configuration.

  - **Reference to Generic:**

    The instance is referenced to the Generic.

  - **User Instance:**

    By the indication "user instance" it is expressed that the geometry of this configuration is stored in PRO.FILE in another document record as the configuration itself.

  - **Reference to Instance:**

    The configuration is referenced to another instance.

Functions in the list "Info Configuration"

The following functions are available using the information list "Configurations":

- **Detach:**

  Removes the referencing of an instance.

- **Reference:**
  After the call up of this function the PRO.FILE operating screen appears. You can now select the document part master, to which you want to reference the Instance. You then confirm the selection.

- **Reference all:**

  Using this function, all configurations without a PRO.FILE-Database-Link can be referenced to a certain document master. After the call up of the function the PRO.FILE operating screen appears. You can now select the document master to which you want to reference the instances. Confirm your choice.

- **View:**

  With the function "displays" the instance marked in the list is retrieved and shown in SolidWorks. This function can also be executed by a double mouse click on a list entry.

- Close:

    Closes the info list configuration and the user returns to SolidWorks

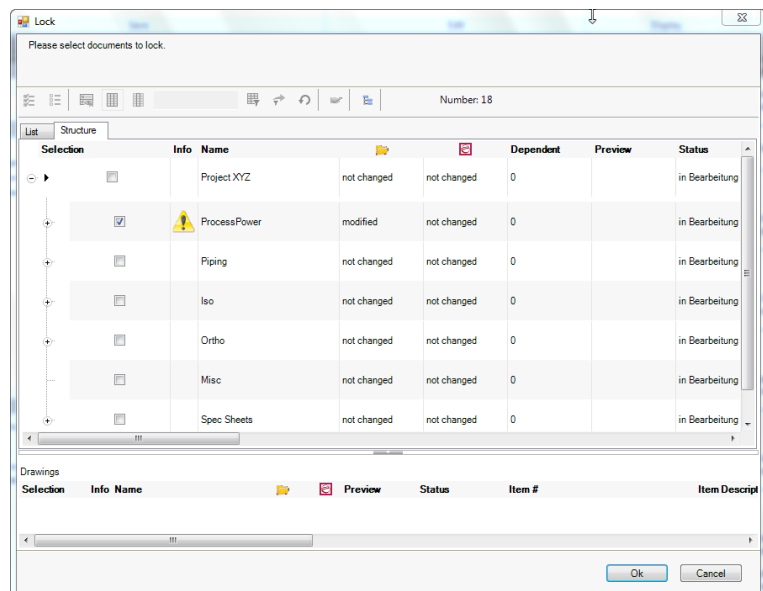- Help:

    Option to use online help

> **Note:**
>
> Changes to the configurations only take effect after saving in PRO.FILE.

7.3 Direct information in the dialog screens

For the functions lock, unlock and document list, dialog screens are displayed.
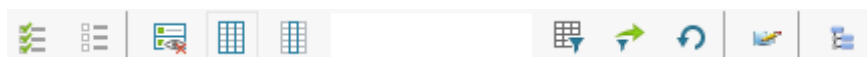
These offer the following functions:

- You can switch between list and structure display.
- Via the buttons of the list functions you can make searches and filter the displayed objects (see the following chapter "More comfort: search and list functions in the dialog screens").
- The lists contain status information for each of the listed objects (see the following chapter "Up to date or not: Display of status information").

7.3.1 More comfort: search and list functions in the dialog screens

The dialog screens of the PRO.FILE AutoCAD Plant 3D 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 ways you can immediately view the stored information without further selection.

7.3.2   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.

- : 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.
⚠️	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.

Info	Local 📂	PRO.FILE 🗃	Description
❗	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.

8   Functions for the version administration

The integration PRO.FILE – SolidWorks offers several functions for working with versions:

- [Managed Version](#)
- [Replace version](#)
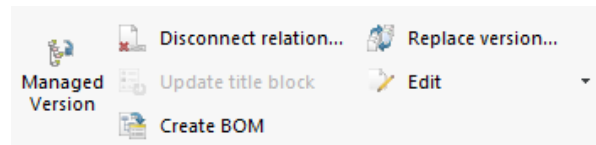
Information on this can be found in the following sub-chapter

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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".

8.1   Replace version

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.

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!

Function call from the PRO.FILE menu in SolidWorks:

"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 SolidWorks (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.
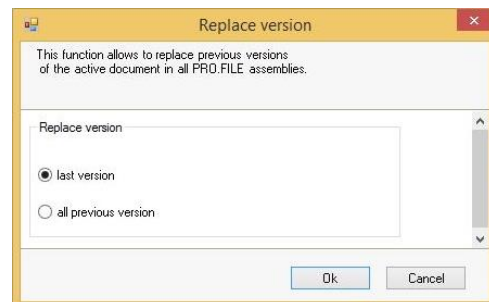
    ⇒ You now get a list of how often and where the predecessor version(s) of the document is/are used.

4. Select all records, for which a replacement is to be made.

5. 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.

8.2   Managed Version

The function "Managed Version" is used for the creation of versions within assembly structures. This function supports the following requirements:

- Inclusion of related drawings

- The file names of the versioned assembly components remain the same.

Please note the following for "**Managed Version**":

- When a component is selected for "**Managed Version**", all instances in the assembly are selected. All versions of a component have the same file name by definition.

- All instances of a part family are treated equally and are thus versioned as well.

- The versionability of the components is checked at the beginning of the "**Managed Version**" process. If a component cannot be versioned (e.g. because it is already released), it cannot be selected.

Function call from the PRO.FILE menu in SolidWorks:

"PRO.FILE" => "Version" => "Managed Version"

The usage of "Managed Version" can be made in two ways:

- If no assembly is opened in SolidWorks, an assembly can be selected via "Managed Version". In this case, the PRO.FILE Checkout Wizard is displayed at the beginning of the process.

- If an assembly is already opened in SolidWorks, the assembly structure can be versioned and the new versions of assembly components can be used immediately.

For further details, see the following sub-chapter:

- [The proceeding for "Managed Version"](#)

## 8.2.1   The proceeding for "Managed Version"

Proceed as follows:

1. Select the menu entry "**PRO.FILE**" from the menu bar in SolidWorks.

2. Select the function "**Version**" => "**Managed Version**"

⇒ The Managed Version wizard is started.

⇒ Starting from the active CAD document, the integration checks the sub-structure according to references in the CAD system.

⇒ In the second step, the sub-structure is enhanced by the related drawings.
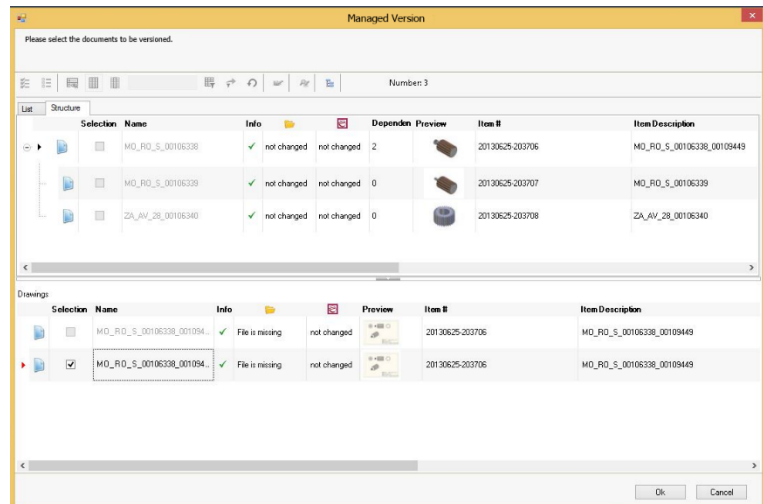
⇒ In the wizard of "Managed Version" the tree structure, determined and expanded by drawings, is shown, so that the documents to version with their dependent data (nodes) can be selected.

⇒ The top node and the first step are already folded out. Further steps can be folded out by a click on the structure symbol ⊕.

⇒ The column "Info" contains further information, e.g. when a part cannot be copied.

⇒ The "status" columns shoes the current processing status of an object in the working directory and in PRO.FILE (see chapter: "Up to date or not: Display of status information").

3. **Select**: Select all components which you want to save as a new version in PRO.FILE. Therefore activate the checkbox in the listed CAD documents as shown on the right.
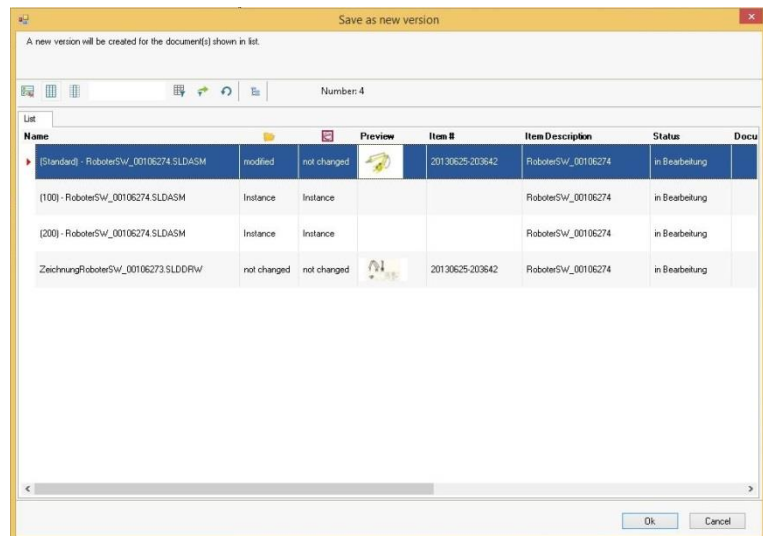
4. Confirm your selection with <OK>.

⇒ An overview of all documents selected for versionining is displayed.

5. Confirm with <OK>.

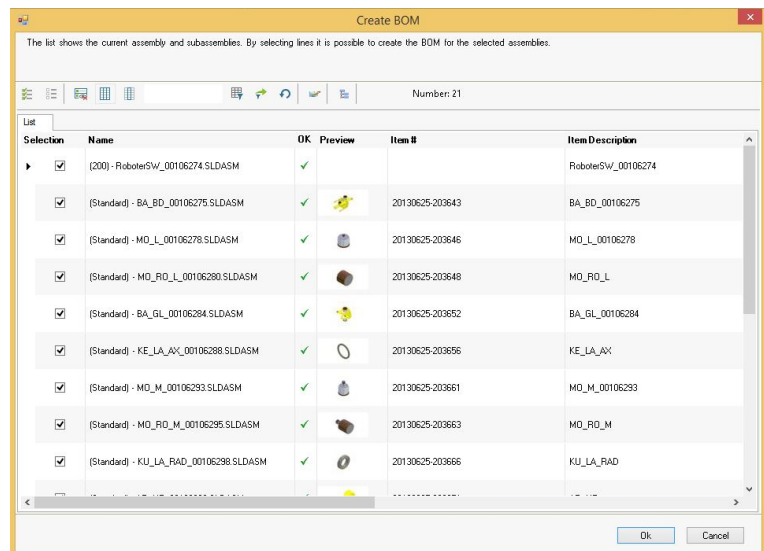⇒ The selection components are now versioned.

⇒ The successful completion of the process is confirmed by a message.

6. Confirm with <OK>.

⇒ The subsequent list shows the saved assemblies.

7. In this list, you can select all assemblies, for which the bill of materials is to be updated.

8. Confirm your selection with <OK>.

⇒ The process "**Managed Version**" is thus finished.

9   Additional Functions to Edit Drawings and Assemblies

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

For active assemblies, the following additional function may be accessed:

- [Managed Rename: Renaming in the structure](#)
- [Insert Part](#)
- [Disconnect relation](#)
- [Disconnect relation (1 level)](#)
- [Document refresh](#)
- [Create BOM](#)

For an active drawing the following additional functions may be accessed:

- [Create balloon](#)
- [Update title block](#)
- [Autoballoon](#)
- [Drawing plot](#)

These functions are described in the following sub-chapters.

9.1   Managed Rename: Renaming in the structure

With the function "Managed Rename" it is possible to change the file name of CAD models already saved in PRO.FILE, while regarding and updating the references to this file name.

This function is used e.g. for the following cases:

- The file name is to describing but the information required for this file name are not yet available (e.g. article number).
- For performance reasons, the file name is to be displayed in the CAD browser. It should therefore also contain PRO.FILE metadata.

> **Note: Only available without active CAD document**
>
> The function "Managed Rename" is only available, if no CAD document is opened in SolidWorks. This is the only way to make sure that the files to be renamed can be processed by the integration without errors.

When the function "**Managed Rename**" is used, CAD data is selected for renaming by the user in PRO.FILE and then put into the local work folder. You can then confirm the file names in the CAD structure according to your needs in an overview window. The PRO.FILE integration then writes the changed file names back to PRO.FILE and updates the references of the documents.

Please note the following for this process:

- The renaming can only be within one assembly structure. The components to be renamed must not be used in other structures.

- The part to be renamed must not have any versions, since, by definition, all version must have the same file name.

- Instances must not be renamed.

- Both the models to be renamed and the assemblies/drawings containing these models must be savable in PRO.FILE (access rights).

- CAD documents referenced in an assembly that are locked by a different user cannot be renamed.

Attention: "Rename" is a modification

The function "Managed Rename" has the same effect on the concerned data like a modification. The renamed components in PRO.FILE correspond to the newest editing status. All local states of these component are thus outdated – and can no longer be saved back to PRO.FILE.

If a user has loaded a component of the renamed assembly locally in his Workcenter, but has not locked it, he/she cannot save any changes back to PRO.FILE. The local stat no longer matches the newest state in PRO.FILE.

This particularly applies for the assembly containing the renamed component: A renaming of the component is a modification of the assembly.

**Function call from the PRO.FILE menu in SolidWorks:**

"PRO.FILE" => "Extras" => "Managed Rename"

Proceed as follows:

1. Select the "**PRO.FILE**" menu in SolidWorks.

2. Select the function "**Extras**" => "**Managed Rename**".

⇒ The Checkout wizard to select the CAD document to be renamed is displayed.

⇒ The Checkout wizard displays the PRO.FILE surface as it was recently opened.

3. If an assembly with the components to be renamed is not displayed on one of the existing tabs, you can now **search** for it:

- Via the tab "Cockpit"

- Via the search functions of the icon bar.

- Via favorites, SmartSearch or task assignments.
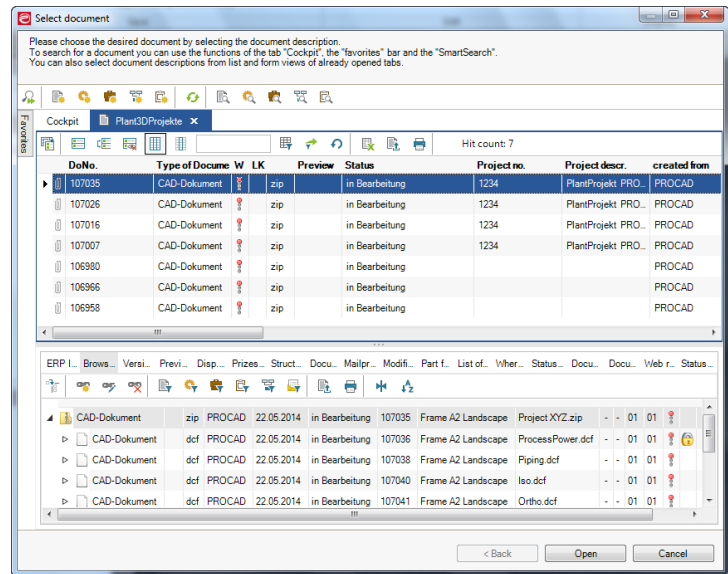
4. If the desired document is displayed on a tab, select it and click <**Open**>.

⇒ The Checkout wizard closes and a warning message is displayed.

5. A warning message informs you that all references recognized by PRO.FILE will automatically be exchanged after the renaming.
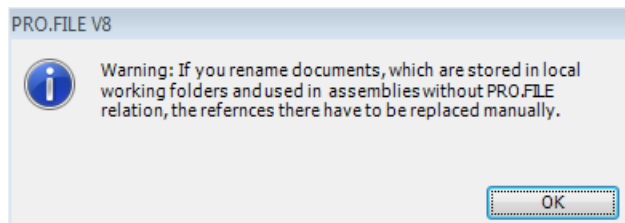

⇒ If the documents selected for renaming are used elsewhere, this cannot be recognized automatically. In such a case, manual post-processing would be necessary.
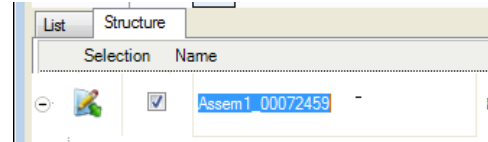
6. Confirm the warning message with <**OK**>.

⇒ The window "Managed Rename" is displayed. In this windows you can rename the files:
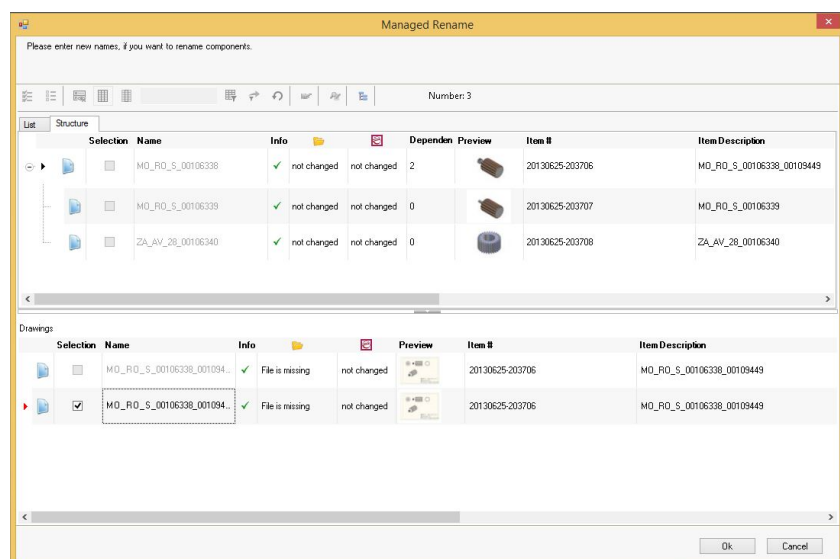
7. **Select**: Select all components for which you want to change the file name by using the checkboxes.
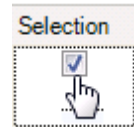
8. **Change file names**: You can edit the file names directly in the list.

9. Make the changes for all desired components.

10. Once you have renamed all desired components confirm your changes with <OK>.

⇒ The integration now saves the changed file names back to PRO.FILE and updates the references.

⇒ The renaming in the structure is now finished.

9.2   Insert Part

This function is used to insert a part from PRO.FILE into the SolidWorks session.

After selecting the function "**Open**" => "**Insert Part**" the desired part is selected in PRO.FILE and loaded in SolidWorks.

The procedure for this is described in the previous chapter "Working with the Checkout wizard to search for CAD documents".

Once all steps for the opening of the part from PRO.FILE are done, the SolidWorks functionality for the insertion of parts is started.

Once the part has been successfully inserted into the assembly, it can also be found in the assembly structure in PRO.FILE.
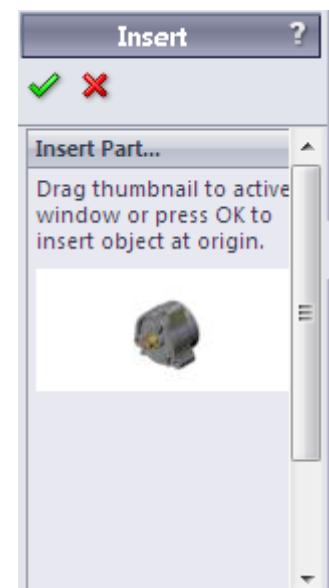
9.3   Disconnect relation

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.

Note:

The PRO.FILE function "**Disconnect relation**" renames the file.

=> If the active document contains renamed CAD elements that are also used in assemblies or drawings, this results in in an immediate new referencing of the assemblies, drawings and parts **opened in the background** to the new file names. This is a default behavior of SolidWorks and cannot be influenced by PRO.FILE.

**Function call from the PRO.FILE menu in SolidWorks:**

"**PRO.FILE**" => "**Disconnect relation**"

Proceed as follows:

1. Select the "**PRO.FILE**" menu in SolidWorks.

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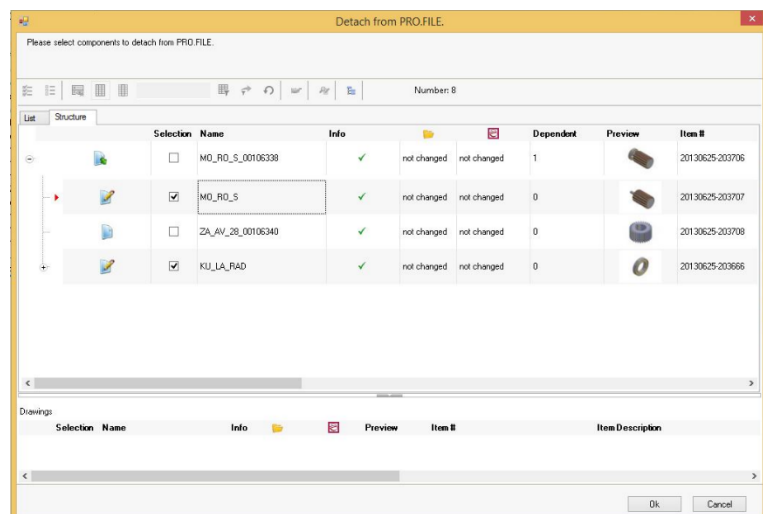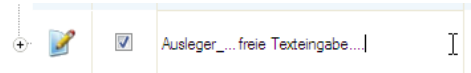
3. Select all CAD documents, the PRO.FILE connection of which you want to dissolve by using the checkboxes.

⇒ 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.

9.4   Disconnect relation (1 level)

The function "**Disconnect relation**", as described in the previous chapter" lists all objects currently active in SolidWorks. This list may be very long in case of complex assemblies and may take a certain time to be displayed.

- The function "**Disconnect relation (1 level)**" only lists the first sub-level of an assembly.

- This gives you a reduced but very quick selection of CAD objects you may want to disconnect.

	Function call from the PRO.FILE menu in SolidWorks:
	"PRO.FILE" => "Disconnect relation (1 level)"

The further proceeding corresponds to the function "Disconnect relation".

9.5　　　　Document refresh

By using the function "Document refresh" it is possible to synchronize parts, drawings and assemblies with the current status of the data in PRO.FILE.

	Attention: Undo not possible: Data loss!
	It should be noted that during updating with newer data from PRO.FILE, all windows in SolidWorks will be closed. Modifications made locally that are not saved will be lost during the updating of the database!

|	Function call from the PRO.FILE menu in SolidWorks:
	"PRO.FILE" => "Document refresh"

Proceed as follows:

1. Select the "PRO.FILE" menu in SolidWorks.

2. Select the function "Document refresh".

⇒ The dialog for the selection of the update type is displayed.

The following options are available:

- **Load latest document data for currently used version**: Documents that were changed in PRO.FILE in the meantime are reloaded in SolidWorks. Used versions are updated – not exchanged (if newer versions exist).

- **Load latest version of document from PRO.FILE**: If there are newer versions of the components to be updated, these are loaded (only versions the user has permission to see).

- Update title block data

- Update metadata in custom properties
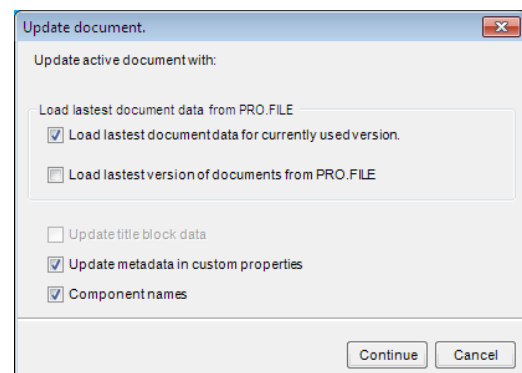
- Update component names

3. Make your selection and confirm with <**Continue**>.

Note:

The more information is selected for the update, the longer this update may take, especially in the case of large and complex assemblies.

⇒ The integration now checks for the **active** object, whether all existing references to the contained elements match the current state in the database.

⇒ If documents are found that are more recent in PRO.FILE than in your local work folder and that should therefore be refreshed. These documents are listed in a window.

4. Select all the CAD documents to be refreshed.

5. Confirm your selection with <**OK**>.

⇒ The security question that is now displayed once again points out the necessity to save your locally changed files, before they are updated und thus overwritten from PRO.FILE.

⇒ The locally stored files are now updated from PRO.FILE and replaced. After that, the recently active window is opened again.

Attention: – Undo not possible:

If the software discovers that a locally stored object is no longer up to date, it is displayed in a special list. With this list you are asked, whether these local files are to be overwritten with the state from PRO.FILE.

If you confirm this selection, the local files are irretrievable overwritten with the newer files loaded from PRO.FILE!

9.6 Create BOM

With the function "Create BOM" a bill of materials based on the CAD structure of the active document in SolidWorks 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 SolidWorks 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.

For this, like in the CAD system, the SolidWorks bill of materials attributes of the model data is evaluated:

- Phantom and reference objects are suppressed.

- Parts of the phantom assembly are put one level higher.

When the bill of materials is derived from cut list elements, an additional dialog may appear for the assignment of these cut list elements to PRO.FILE part master records. For details see "Saving of weldments".

Note:

Please note the following requirements for the creation of the BOM in PRO.FILE:

- Bills of materials can only be created for assemblies.
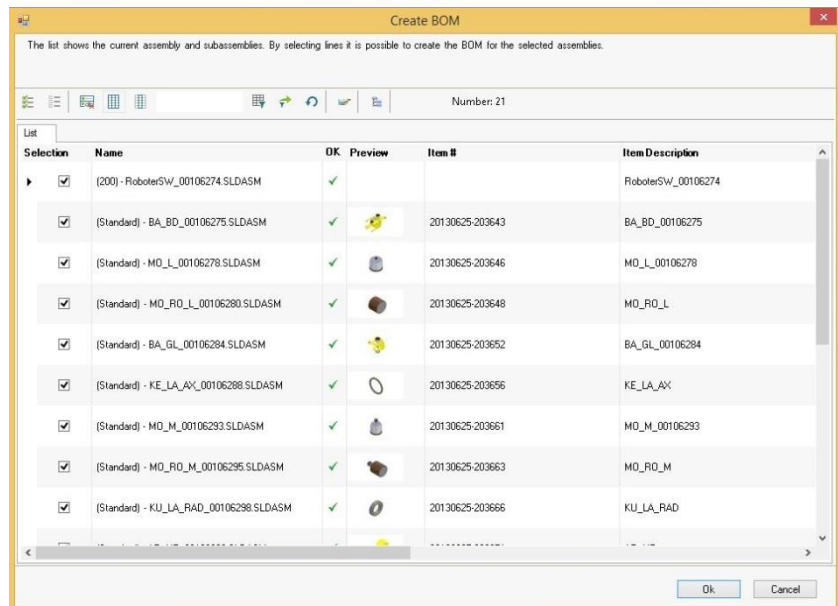- The CAD documents must be linked to a part master in PRO.FILE.

Function call from the PRO.FILE menu in SolidWorks:

"PRO.FILE" => "Create BOM"

Proceed as follows:

1. Select the "PRO.FILE" menu in SolidWorks.
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.
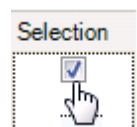
> **Note: Display of conflicts**
>
> If the creation of a BOM is not possible for an assembly, the column "OK" shows a red "Attention" symbol. The corresponding tooltip shows the cause of the problem.

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. Confirm your selection with <**OK**>.

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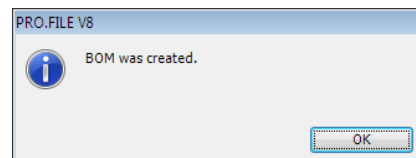
To view the bill of materials in PRO.FILE, select the function "Show" => "Bill of materials" as described in the chapter "Show: Information on a CAD document in PRO.FILE".

> **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.

9.7   Create balloon

The function "Create balloon" allows bill of materials positions

Requirements for Creating a balloon

Certain requirements must be fulfilled to enable the creation of a position within a drawing or assembly:

- The assembly or drawing must be saved in PRO.FILE.

- A bill of material must be created in PRO.FILE.

- The corresponding print format must be configured within the PRO.FILE Management Console.

- The positioning arrow must be configured in the drawing legend.

Only when these prerequisites are fulfilled can the bill of material positions be successfully inserted.

Function call from the PRO.FILE menu in SolidWorks:

"PRO.FILE" => "Create balloon"

To create a BOM position from PRO.FILE for display in the drawing or assembly proceed as follows:

1. Select a part for which the BOM position is to be inserted. The selected click point is the position for the balloon.

2. Select the function "**Create balloon**" from the PRO.FILE menu in SolidWorks (if no part has been selected, a corresponding message is displayed).

⇒ The BOM position is now displayed within the SolidWorks document.

⇒ The position of the balloon can be changed via drag&drop.

Note: Shortcut

You can select the function "Create balloon" from the context menu (right mouse click) in SolidWorks.

If no part has been selected, a corresponding message is displayed.

9.8   Update title block

With this function the drawing legend of the current SolidWorks drawing is filled with current data from PRO.FILE.

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

Function call from the PRO.FILE menu in SolidWorks:

"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.

9.9   Autoballoon

This function corresponds to the "Autoballoon" function in SolidWorks – with the difference that the position numbers are loaded from PRO.FILE and inserted in the drawing if the integration function "Autoballoon" is used.

Function call from the PRO.FILE menu in SolidWorks:

"PRO.FILE" => "Autoballoon"

To use Autoballoon proceed as follows:

1. Open the drawing for a part or assembly from PRO.FILE in SolidWorks.
2. Select a view within the drawing.
3. Select the function "**Autoballoon**" from the "**PRO.FILE**" menu in SolidWorks.
⇒ The PRO.FILE information for all components in the selected view is now loaded and the available position numbers are included in the drawing view.

Note: Confusion with SolidWorks texts

If the PRO.FILE position number is configured as a pure numeric index (like the SolidWorks positions), the integration adds a blank before and after the PRO.FILE position texts. This is to avoid confusion with SolidWorks texts.

9.10   Drawing plot

With this function you can plot all or specific drawings of an assembly. For this function, the drawings and the assemblies must be saved in PRO.FILE.

Note:

To use this function, certain configurations have to be made:

- The Display type "Bill of materials on drawing" must be configured via the form designer of the PRO.FILE Management Console.

- In the last column of the display type "Bill of materials on drawing" the part master ID must be configured. The part master ID must be at the last position of all fields,

- The corresponding plotting profiles must be configured in the PRO.FILE Management Console.

For detailed information see the configuration manual of the integration.

Function call from the PRO.FILE menu in SolidWorks:

"PRO.FILE" => "Extras" => "Drawing plot"

Plot drawings

1. Select the function "Drawing plot" from the integration menu. A message on the further proceeding is displayed. Confirm this message with <**OK**>.

2. You then have to select a part master in PRO.FILE, to which the bill of materials with the drawings to be plotted is assigned.

⇒   All drawings found for the part master and bill of materials in PRO.FILE are now displayed in a new window.

3. In this window you can select all drawings you want to plot. Confirm your selection with <OK>.
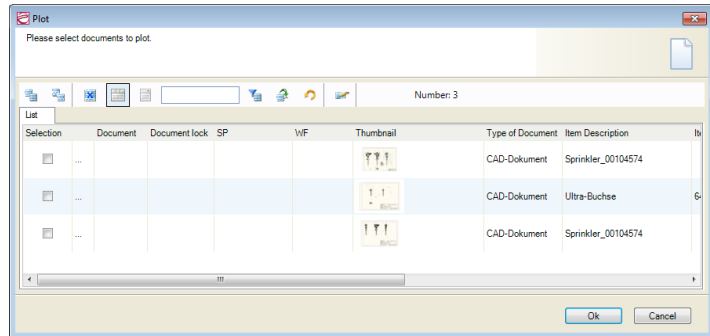
4. You can now select the printer profile for the plotting process. The dialog lists all printer profiles defined in the PRO.FILE Management Console.

5. Confirm your selection with <OK>.

⇒ The plotting is started and all selected drawings are printed out.
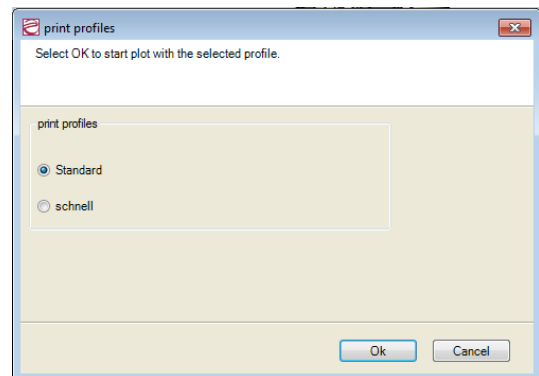
10   Extras: The Workcenter

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 SolidWorks:
	"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.

10.1   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".
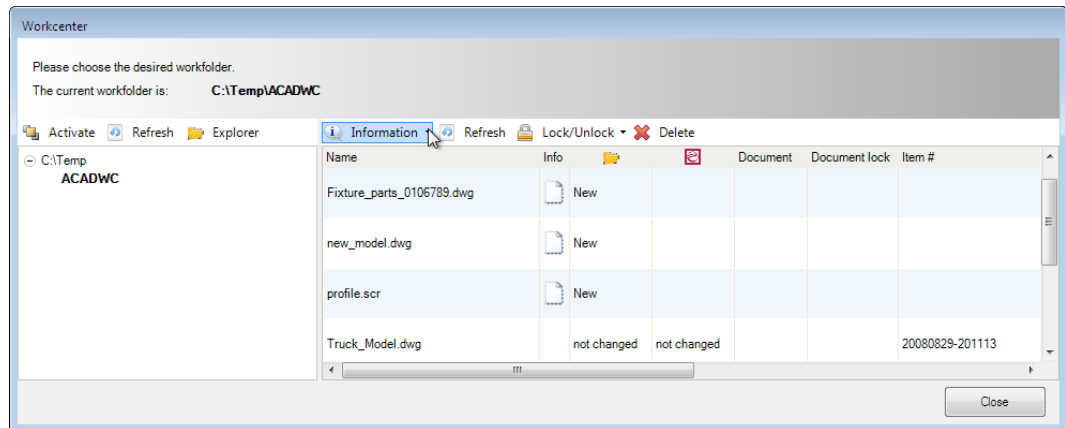
The functions for the directory structure:

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	Document structure
Part form	Document form
Usage of parts	Usage of documents
Bill of materials	

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.
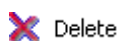
Delete

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 facilitat 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).

11   Extras: Workplace-specific configurations

> **Note:**
>
> The functions "**PRO.FILE**" => "**Extras**" =>
>
> - "Configuration Title Block"   - "Configuration Bill of materials"
> - "Configuration list of changes"   - "Configuration Balloon"
>
> Are described in the manual "**Configuration of the Integration PRO.FILE – SolidWorks**" because these configurations require access to the PRO.FILE Management Console.

The integration PRO.FILE – SolidWorks, makes global configurations of the mode of operation and environment, company-wide possible - as well as local, workplace specific determination of user settings.

- **Global Configuration:** The global configuration applies for all users, and is carried out using the PRO.FILE Management Console. You can configure the company-wide behavior of the SolidWorks Integration using the parameters of this administration tool for the SolidWorks Integration. Detailed information can be found in the manual "**Configuration of the integration PRO.FILE – SolidWorks**".

- **Local Configuration**: When required, the user can carry out individual settings within the SolidWorks Integration.

The local configuration is accessed via "**PRO.FILE**" => "**Extras**" => "**Options**".

> **Note:**
>
> Local configuration can only be carried out if the range of validity of corresponding parameters in the PRO.FILE Management Console has been set to "Individual user" or "PC parameter". If this is not the case the functions are greyed out.

> **Function call from the PRO.FILE menu in SolidWorks:**
>
> "**PRO.FILE**" => "**Extras**" => "**Options**"

To make local configurations of the integration PRO.FILE – SolidWorks on your computer proceed as follows:

1. Select the "**PRO.FILE**" menu in SolidWorks.
2. Select the menu entry "**Extras**" => "**Options**".

⇒ The dialog for the configuration options is displayed. It contains four tabs:

- [Options: Document list](#)
- [Options: System options for the performance optimization](#)
- [Options: Messages](#)
- [Settings for the original name reference](#)

11.1 Options: Document list

Different functions of the integration that access PRO.FILE display the document list for the display and selection of CAD documents.

This document list contains all relevant information on a CAD document.

Via the options menu of the integration you can adjust the display of this document list to your specific needs.

	Function call from the PRO.FILE menu in SolidWorks:
	"PRO.FILE" => "Extras" => "Options" => "Document list"

You can now make different settings in the window that appears. These settings determine the data and information that is shown in the PRO.FILE **document list** on your work station.
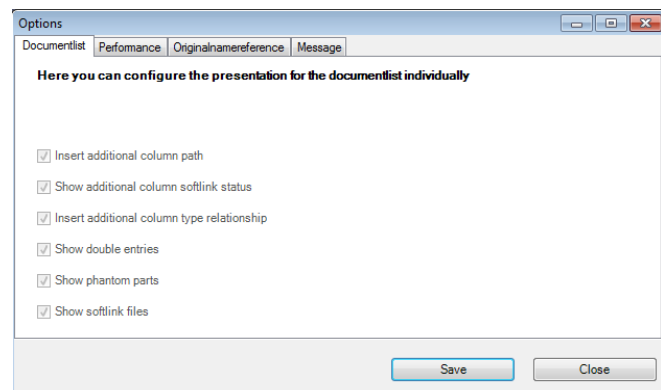
The following, additional information can be defined here for the document list:

- Insert additional column path
- Show additional column softlink status
- Show double entries
- Show phantom parts
- Show softlink files

Select the required options by activating the corresponding check box.

Save your settings using the <**Save**> button.

11.2   Options: System options for the performance optimization

Settings can be user specified for the PRO.FILE – SolidWorks integration to optimize performance.

The settings for the performance optimization reduce the quantity of data that can be loaded or transported.

Function call from the PRO.FILE menu in SolidWorks:

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

These settings can be carried out individually by each user. These individual settings do not affect the settings for other users.

The following options can be selected for the performance optimization:

- On open: For overwrite, automatically close all open documents.

- Automatically show list to lock objects after open.

- Automatically show list to unlock objects after save.
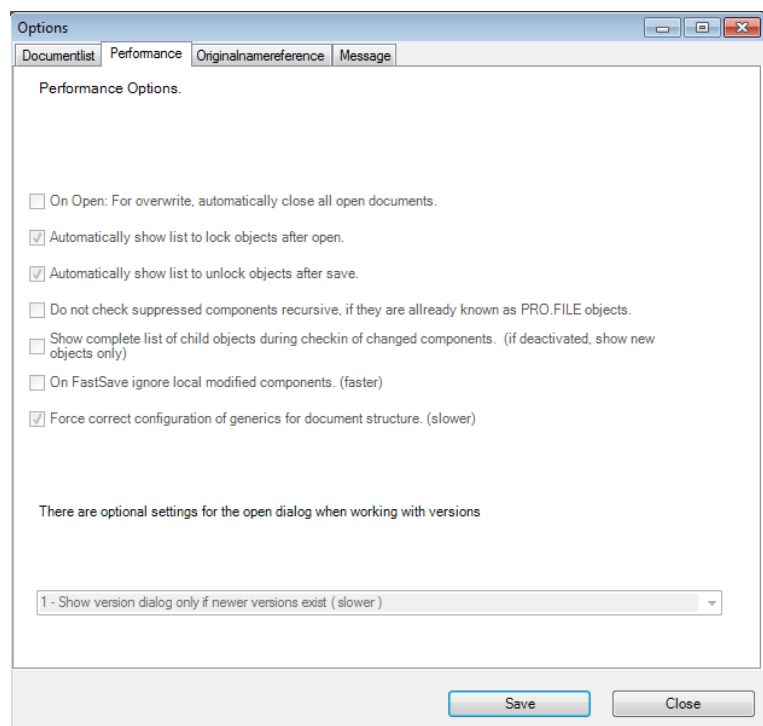
- Do not check suppressed components recursive, if they are already known as PRO.FILE objects

- Show complete list of child objects during check-in of changed components.
If this box is checked, all objects will be offered in a list upon new saving. If the box is deactivated, not all loaded documents but only new documents are offered for saving.

- Force correct configuration of generics for document structure.

- On FastSave ignore local modified components.

When working with versions, you have the possibility of selecting the method of extraction of data from PRO.FILE (from the dialog window) that is to be used for opening CAD objects.

You can influence this dialog by using the selection menu within the performance options. The following choices are available:

- Show version dialog, always.

- Show version dialog only if newer versions exist. (Due to the required check of the data by PRO.FILE this option is more performance intensive and therefore slower).

- Opening the CAD documents without a user query from the dialog window, "always", exactly how they were saved.

- Opening the CAD documents without a user query from the dialog window with latest versions "always".

11.3   Options: Messages

When working with the integration of PRO.FILE – SolidWorks, user support is provided by so called message windows, which give the user information about the called up functions.

These message windows can be deactivated one by one by selecting the field <**Do not show this message again**>. This message will then no longer be displayed.

In PRO.FILE 8.6 message texts are often integrated into the corresponding dialogs and are not displayed as "preceding message box".

To display these message windows at a later date, you can reactivate them using the integration function "**Message**".

	Function call from the PRO.FILE menu in SolidWorks:
	"PRO.FILE" => "Extras" => "Options" => "Message"

Display hidden messages again

1. Select the menu entry "**PRO.FILE**" => "**Extras**" => "**Options**" => "**Message**" in SolidWorks.

⇒   A window that lists all
    hidden messages is
    displayed.

2. Every message that is to
    be displayed again can
    be selected with the
    corresponding checkbox.

3. Confirm your selection
    with <**Enable**>.
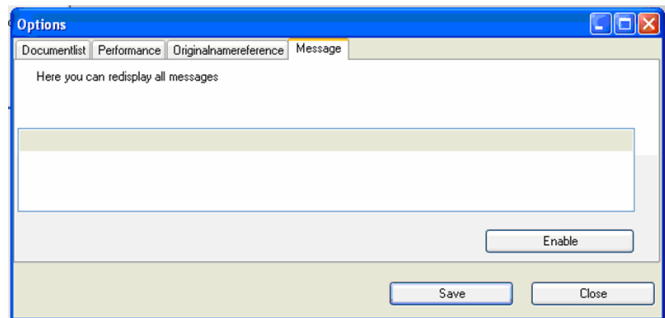
⇒   The selected messages are now displayed again, once the corresponding function
    is used.

These settings only refer to the workstation on which they were made.


11.4   Settings for the original name reference

The local settings are carried out using the menu "options" in the PRO.FILE menu of the
integration.

Function call from the PRO.FILE menu in SolidWorks:

"PRO.FILE" => "Extras" => "Options" => "Original name reference"

In the mask that then appears, you can activate the corresponding options "analyze
Oldref logfile", and "write Oldref logfile" in the "Originalnamereference" folder.
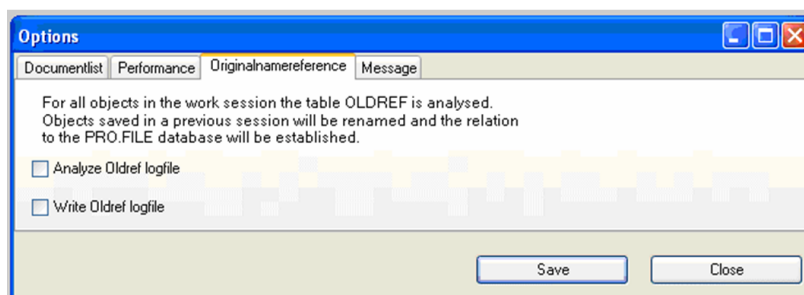
This possibility of user configuration is predefined by the SolidWorks administration.

•   Either The OLDREF protocol is set explicitly to "on", or "off", so that the display in
    this mask remains blank.

•   It is only possible to activate a user specified setting here, when the appropriate
    setting has been made in "freely configurable for users" in the SolidWorks
    administration.

After changes to the SolidWorks administration, SolidWorks and the integration have to
be restarted in order for the local setting to become effective.

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

- [Write Oldref Logfile](#)
- [Analyze OLDREF Logfile](#)

11.4.1　　Write Oldref Logfile

Function call from the PRO.FILE menu in SolidWorks:

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

The Option "Write Oldref logfile" can be activated in the section "Originalnamereference", in the mask that now appears.

This button must be set if data that is not related to PRO.FILE is to be taken over and **analyzed**, as in the description above. User cases include; e.g. the taking on of small amounts of old data, the reading of CAD-objects by other companies, or departments that do not use PRO.FILE.

When the button "Write Oldref logfile" is set, a protocol is written for every CAD-object (PART, DRAFT, ASM ..) that is saved, and which is then assigned a CAD-file name (Name of the original file = Filename incl. extension) and a document identity number.

OBJ_ID	OLD_NAME	TYP
13998	MOTOR_FORM	PRT
13997	MOTOR_CASE_	PRT
13999	END_CAP_	PRT
14000	WORM_GEAR_	PRT
14001	MOTOR_SHAFT_	PRT
13996	MOTOR_	ASM
14003	CD_MECH_SHELL_	PRT
14004	LARGE_GEAR_	PRT
14005	TRAVEL_BAR_	PRT
14007	LASER_BASE_	PRT
14008	SLIDER_	PRT
14009	CIRCUIT_BOARD	PRT
14010	L_PLUG_	PRT
14011	S_PLUG_	PRT
14012	S_SCREW_	PRT
14006	LASER_BASE_	ASM
14014	SPINDLE_MOTOR_SHAFT	PRT

Note:

The filling in of the Oldref table by setting the switch <**Write oldref logfile**> can be set while working with norm parts. If then all norm parts are saved in PRO.FILE, the setting <**Analyze Oldref logfile**> can be set.

11.4.2   Analyze OLDREF Logfile

Function call from the PRO.FILE menu in SolidWorks:
"PRO.FILE" => "Extras" => "Options"

The Option "Analyze Oldref logfile" can be activated in the section "**Originalnamereference**", in the mask that will now appear.

By setting this switch, you activate the surveillance function, so that every time a CAD-object is saved, a check is made to see whether the original file name is present in the Oldref table. There are two possible outcomes:

New object saved

An object is loaded in the SolidWorks session. The parameters are set. The command <**Save**> is used.

The integration recognizes the original name of the file that is to be saved, and searches through the Oldref table. As the object is new, no entries are found, and the object is saved to PRO.FILE.

If the parameter <**Write Oldref Logfile**> is activated, the original name of the object that has just been saved will be entered into the Oldref table.

If the parameter is not set, no entry will be made in the table. This means that the next object that has the same name will also be saved and laid down in PRO.FILE.

An object with the same original name is saved to PRO.FILE

An object is loaded in the SolidWorks session. The parameters are set. The command <**Save**> is used.

The integration recognizes the original name of the file that is to be saved, and searches through the Oldref table. The name is found in the Oldref table.

You will then receive a list in PRO.FILE, which will contain the items available.

This object is not saved in PRO.FILE, but a reference to the object that is already in PRO.FILE is created.

11.4.3   Variation of the Oldref parameters and their effect on your CAD-Objects

Variation of the parameters and their influence on extremely complicated processes can take place when working in the integration. In the following chapter a few situations and their possible consequences are described. The aim is, to help you to judge the behavior of the Oldref function in various situations, so that you are in the position to assess the results of a planned process.

> ### Attention:
>
> To take on data with the help of the Oldref function, the **basic data** that is to be saved to PRO.FILE must be unique! If two objects that share the same file name are present, the second object will not be saved to PRO.FILE during the save process.
>
> If you do not want this to happen, you must change the parameter for each user case.

11.4.4   Limitations when using Oldref

The functions described here, stretch the technical possibilities within certain CAD-objects to their limits. This applies among others to:

- Phantom parts
- Standard parts
- Versions

Versions

All versions of an object have the same name in PRO.FILE. Therefore the versions of an object cannot be differentiated from one another. It is therefore not possible to sort the versions of an object with the help of the Oldref function within a version chain.

No version chains are formed.

- **Newer version in PRO.FILE**

  The newer version is built into the actually saved object.

- **New extern version**

  The new version is saved to PRO.FILE. In the loaded assembly or drawing, the old version is referenced when saved to PRO.FILE.

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