Functions of the Integration PRO.FILE Solid Edge

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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 – Solid Edge

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

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 Solid Edge to ERP systems.

1.1   The contents of this manual

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

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 Solid Edge. The following topics will be addressed:

- Operation of PRO.FILE from within Solid Edge

- Data representation of structures/references in PRO.FILE

- Integration of PRO.FILE into the Solid Edge environment

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

1.2   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 and functions in PRO.FILE directly from Solid Edge.

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

- Setting up the connection to Solid Edge
- 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

1.3   Setting up the connection to Solid Edge

Contradictory settings within the Solid Edge environment can cause serious connection problems when working with the PRO.FILE Solid Edge integration, as described in the configuration manual for the PRO.FILE Solid Edge Integration.

It is therefore necessary that you check all Solid Edge work stations for complete and correct settings before you start working with the integration, and before errors occur.

All information on the setup of the connection can be found in the **configuration manual of the integration PRO.FILE – Solid Edge**.

!	Attention:
	These settings of the Solid Edge environment should be made before the first start of the integration. It is harder to remove errors later, once they have occurred.

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

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

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**>.
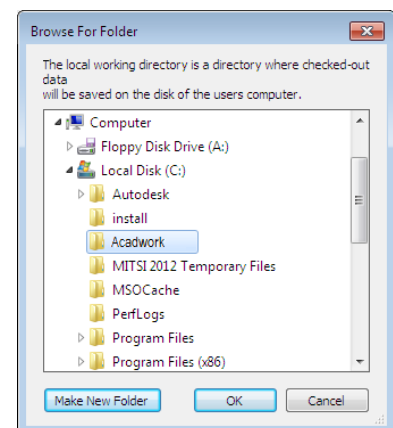
4. In the second step, "**work folders**" are now created In this root folder, which will then be used by the integration. Consequently, you are now prompted to specify a **work folder** within the **root folder**:

5. Please specify a name for the work folder.

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

⇒ The configuration of the Workcenter is now finished.
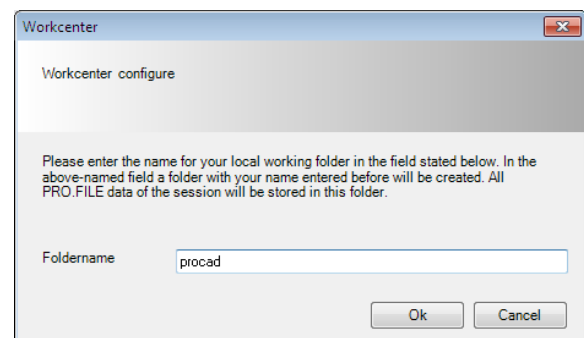
This work folder and other local work folder can be created and managed by the user via the Workcenter. The Workcenter can be accessed via the PRO.FILE menu in Solid Edge under "**Extra**" => "**Workcenter**". Detailed information can be found in the chapter "[Extras: The Workcenter](#)".

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

By setting up the integration, the menu entry PRO.FILE is added to the menu bar of Solid Edge. The functions of the integration can thus be directly accessed via the "**PRO.FILE**" **menu** in Solid Edge:

1. Start "**Solid Edge**".

2. Go the menu bar to the section "**PRO.FILE**".

3. Select the desired integration function from the menu.

⇒ The menu of the PRO.FILE integration depends on the loaded Solid Edge file:

⇒ When Solid Edge is started – without a loaded CAD object – the start menu of the PRO.FILE integration is displayed:
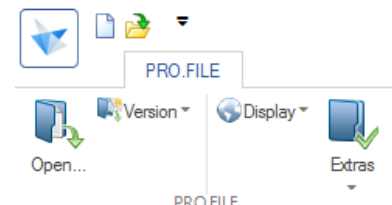
When a **part** is loaded, all basic functions of the integration are offered in the PRO.FILE menu:
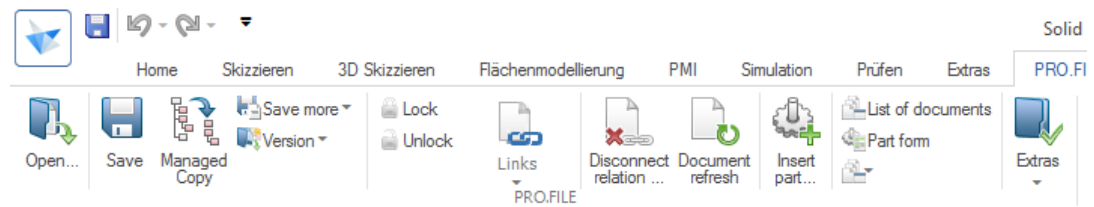
When a **drawing** is loaded, the PRO.FILE menu is enhanced by the following functions:

- Create BOM

- Create balloon

- Drawing legend
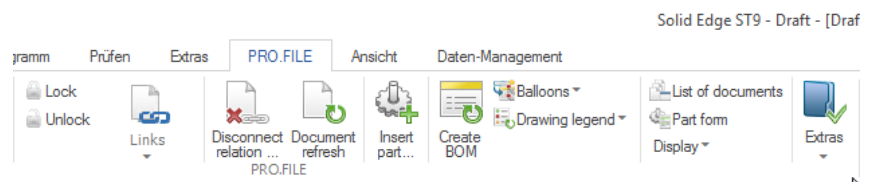
When an **assembly** is loaded, the PRO.FILE menu contains the following functions:

- Replace part
- Create BOM
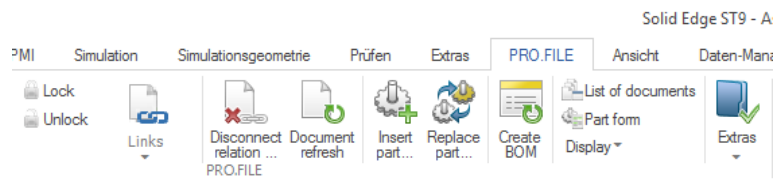
⇒ Whenever one of these menu functions is used, PRO.FILE is activated. According to the function used, the required windows in PRO.FILE are opened automatically.

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

1.6 How to log in to PRO.FILE?

If you access a PRO.FILE function for the first time within a Solid Edge 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.

1. In the login screen, please enter:
   - Your PRO.FILE user name
   - Your PRO.FILE password.
2. Confirm with <**Login**>.

⇒ The PRO.FILE home screen is now displayed.
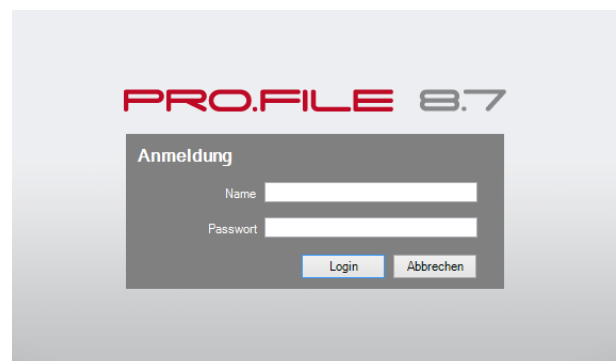
(i)	**Note: No login required in case of "Autologin"**
	This login is not requested, if the PRO.FILE autologin function is activated.

1.7   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.

These menu entries are not sorted according to their occurrence in the menu but according to subject areas:

Functions for opening CAD objects from PRO.FILE

- Open

  This function opens PRO.FILE and prompts the user to select a CAD document for loading in Solid Edge.

  When a document with linked CAD components is opened, the opening method is determined by the setting of the parameter "Version dialog during open" in the PRO.FILE Management Console.

Functions for saving objects to PRO.FILE

- 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

  only for parts and assemblies): Managed Copy organizes the data management of complex models in the change design. Entire machines are cloned, including all referenced data and workshop drawings. Assemblies and components that are to remain in the new design, are taken over. Existing references remain intact.

- Save NDF

  With this function, a **n**eutral **d**ata **f**ormat (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.

- 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.

- Save Phantom

  By calling up this function, it is possible to lay down a complete assembly under a single part master in PRO.FILE, and to save all objects that are contained in this assembly under the part master. From then on, this assembly will be treated as an individual part by PRO.FILE. The objects contained in the assembly are laid down as phantom objects, and can't be called up from PRO.FILE.

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

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

Functions for Versioning in PRO.FILE

- **Open with latest (released) versions**

  This function automatically loads the newest (released) versions of references of CAD objects from PRO.FILE to Solid Edge. This function is used in the context of versioning and is therefore listed in the integration under the menu entry "Version".
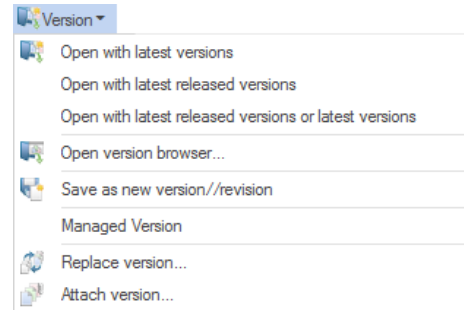
- **Open version browser**

  With the version selection you can load assemblies in dynamic compositions to be defined by the user. The user can decide which version of an assembly and of its components is to be opened.

- **Save as new version/revision**

  Saves the actual 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

- **Managed Version**

  The function "**Managed Version**" can be used to create versions within assembly structures.

- **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.

- **Attach version**

  This command allows to save the currently active object as a new version to a same object (Part to part, assembly to assembly, etc.). It will be "**attached**" as new version to the version chain. If you open one of these objects later on, you will be informed that a newer version has been found.

Functions for the linking of additional files

- **Add additional file**

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

- **Add PRO.FILE document**

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

- **Detach document**

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

Database
functions

- Lock/Unlock

  This function enables the locking and releasing for other users in PRO.FILE, of actual loaded objects in Solid Edge.

- Disconnect relation

  This function deletes the database relationship of a PRO.FILE part, a PRO.FILE drawing, or a complete PRO.FILE assembly with the selection of the contained object. The CAD objects are then treated purely as locally saved CAD objects.

- Document refresh

  Using this function enables metadata to be updated to read PRO.FILE objects with the actual stand of the information in PRO.FILE.

Functions to
modify CAD
objects from
PRO.FILE

- Insert part

  This function is for pasting an object from PRO.FILE.

- Replace part (Assembly/Weldment)

  This function is for replacing an object in PRO.FILE.

Modification of
drawing contents
via PRO.FILE

- Create Bill of Materials

  This function opens PRO.FILE and creates a new Bill of Materials for the currently active assembly.

- Balloons

  Adds balloons from PRO.FILE to a drawing.

- Drawing legend

  This menu contains functions for updating the drawing legend, for deleting elements from the drawing legend or for copying the drawing legend from an existing document into the currently loaded document.

Information functions in the menu "Show"

- **List of documents**

  This functions opens the special documents list and displays the configured information on the current part, the drawing or the assembly, including all linked objects.

- **Structure of the parts**

  Switches to PRO.FILE and displays the structure overview of the current part.

- **Part form**

  Switches to PRO.FILE and displays the description form for the active part.

- **Use of parts**

  Displays the proof of use list of the displayed/selected object.

- **Bill of materials**

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

- **Document structure**

  Changes the document structure list of the displayed/selected object.

- **Document form**

  Changes the document description in the list presentation of the displayed/selected object. If there is no Object specially selected within an assembly, then the form of the displayed assembly will be shown.

- **Use of documents**

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

- **Drawing**

  The corresponding drawing is opened for a model. At first, a check is performed, whether these are saved locally. If this is not the case, the drawing is searched in the PRO.FILE database and opened afterwards.

- **All document versions**

  Changes the version list of the displayed/selected object.

- **PROCAD on the WEB**

  This menu point opens the PROCAD homepage, as long as the user has internet access.

- **Readme show**

  Via the readme file you can view the newest system information on the integration PRO.FILE – Solid Edge.

- **Help**

  Opens the PRO.FILE online help.
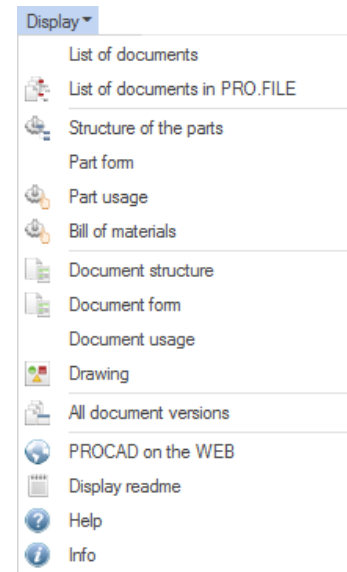
- Info

    Shows the PROCAD logo as well as the Solid Edge integration version number that is being used. For detailed information, you can select the menu point options, and a list of all-important information and the version number will be shown in a list.

Extras
- Activate work folder

    All files that are read or saved by Solid Edge while working with the PRO.FILE Solid Edge integration of PRO.FILE are automatically stored locally on the user computer. By using the function "Activate work folder" it is possible to select and modify the local storage path. If several sub-directories have been created in a directory, it is possible to set the path to be used for data storage by activating it.

- Options

    In this menu point you can make a large number of different settings for your active Solid Edge integration. Please also refer to the configuration manual of the Integration PRO.FILE – Solid Edge

- Update Thumbnail

- If a thumbail preview is used in the PRO.FILE document list, this function is used to update the thumbnail image.

2 Functions for opening CAD documents from PRO.FILE in Solid Edge

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 from within PRO.FILE:

- Open CAD documents from PRO.FILE for editing

Opening from within the integration:

- Open: Loading CAD data from PRO.FILE
- Open CAD documents with linked components
- Open version browser
- Attention: Opening of locally existing files
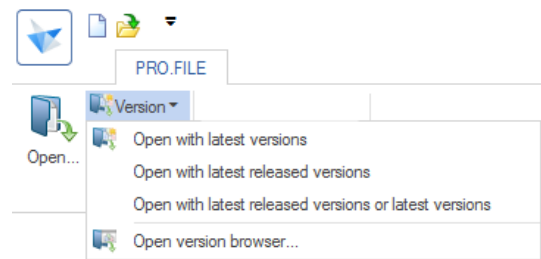
Attention:

The data loaded from PRO.FILE in Solid Edge are **not automatically locked** when opened in Solid Edge. 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.

2.1   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.

2.1.1   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.FIL into the CAD 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.

2.2   Open: Loading CAD data from PRO.FILE

If you want to open a document from PRO.FILE in Solid Edge, you can use the function "**Open**" in the PRO.FILE – Solid Edge integration.

The function "Open" starts the PRO.FILE Checkout wizard, where you can select the desired document to be loaded in Solid Edge. Proceed as follows

Step 1   Use the PRO.FILE function "Open"

Function call from the PRO.FILE menu in Solid Edge:

"PRO.FILE" => "Open"

1. Go into the menu bar of Solid Edge 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   Select 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
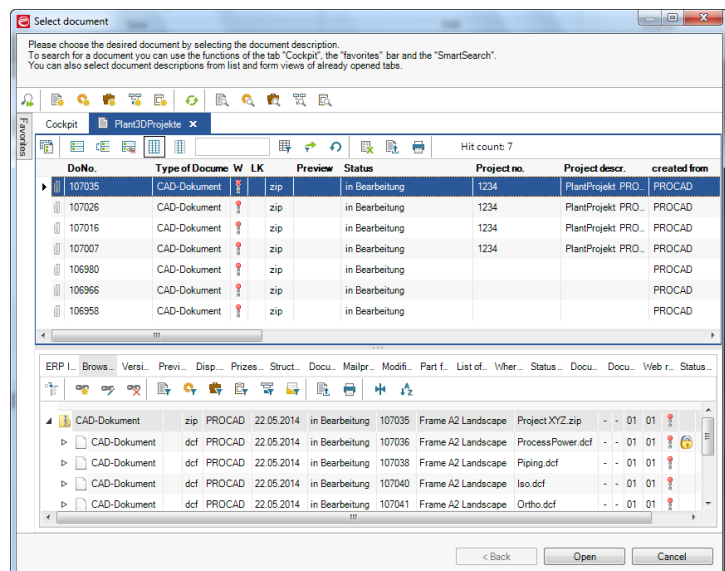
4. If the desired document is displayed in a list view, you can select it. (If the desired document is displayed in a form view, it is already selected.)

5. Click <**Open**>.

⇒   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".

Step 3

Versions in the structure: How to handle the components?

When a CAD object with structure (assembly/drawing) is opened, PRO.FILE checks whether other versions exists for the components.

If this is the case, you can now decide how PRO.FILE is to load the assembly. Whether this dialog is displayed or whether the opening method is already preconfigured depends on the setting of the parameter "Version dialog during opening" in the PRO.FILE Management Console.
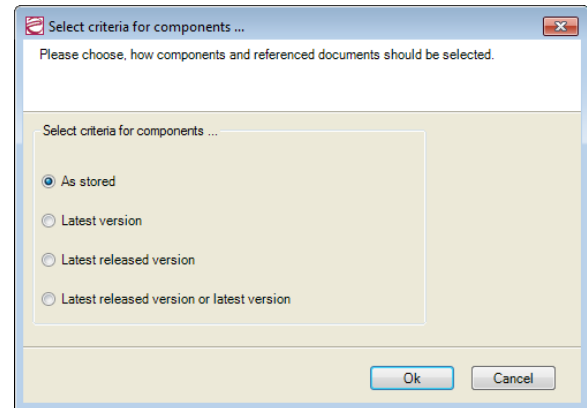
6. Select the desired criterion and confirm with <OK>.

- **As stored**:

    The assembly is loaded with the component versions it was last saved with. Changes to parts that have been made in newer versions are thus ignored.

- **Latest version**:

    PRO.FILE replaces all CAD documents, for which newer versions are found and loads the assembly with the latest version of these components. You thus get a newer version of the assembly/drawing.

Note:

You can only see the versions for which you have viewing permission. If the most recent version is not "visible" for you, you get the newest visible version.

- **Latest released version**:

    PRO.FILE replaces all CAD documents, for which newer versions in a released status are found and loads the assembly with these newer version of these components. According to the above note, the software checks whether you have permission to see these versions.

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

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

  This option is important if an assembly consists both of released and unreleased components. PRO.FILE replaces all CAD documents, for which newer versions in a released status are found. If no version in a released status can be found for a component, the newest version is loaded. PRO.FILE opens the assembly with all available objects in a released status – all other components are loaded in their newest available version.

> **Note: Open version selection**
>
> For the targeted selection of the desired versions you can use the function "Open version browser". Via the version browser you can load assemblies in dynamic compositions to be defined by the user. The user can decide which version of an assembly and of its components is to be opened.

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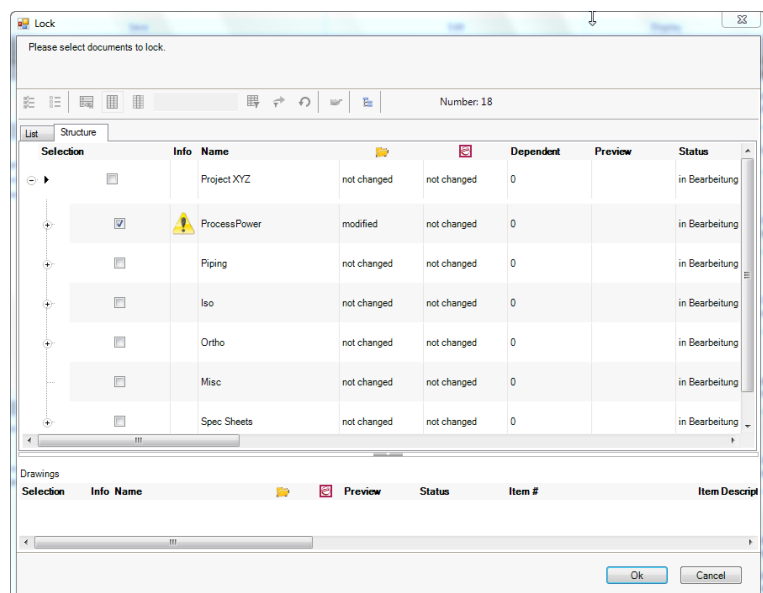
7. Select all documents you want to lock with the corresponding check boxes.

8. Confirm your selection with <OK>.

⇒ The selected document is now opened with its components in Solid Edge. 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

2.2.1   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**>.
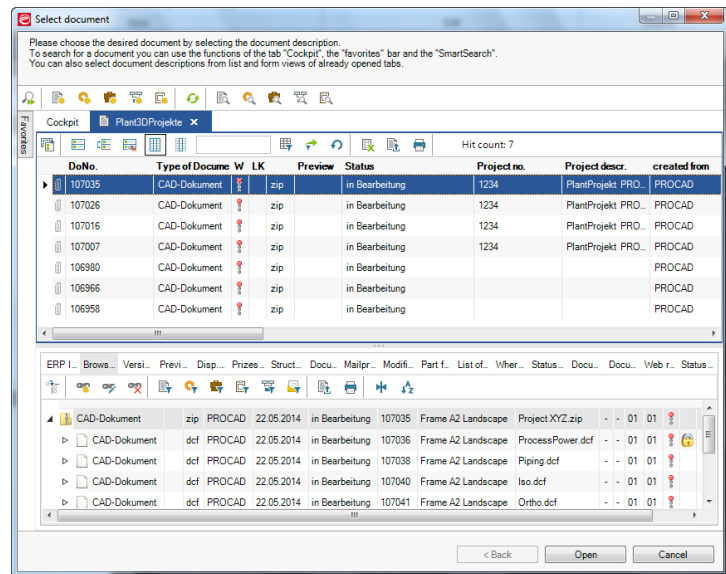
- If no search had been made previously, or if the desired document is not displayed on one of the existing tabs, you can now search for it.
  For this, several functions, which are explained in the following, are available:
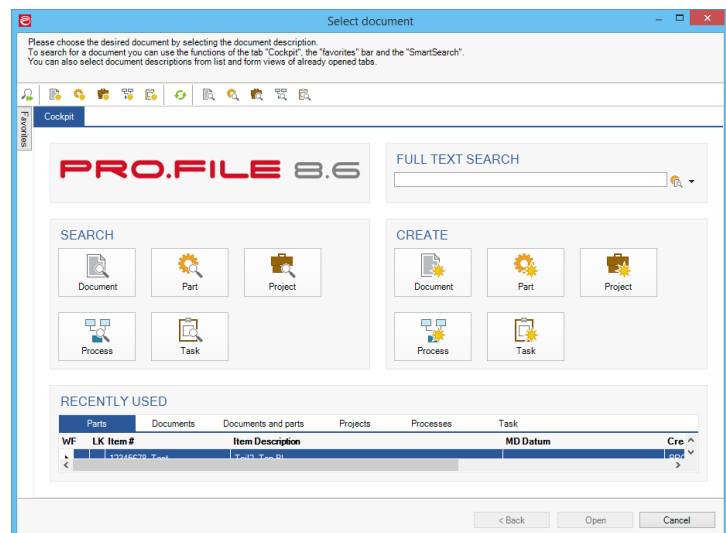
⚠ **Attention: Double-click in the Checkout wizard**

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 for data records in the Checkout Wizard

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

- Searching via the tab "Cockpit"

    The same icons as in the icon bar can be found on the tab "Cockpit": "Search document", "Full-text search", "Search part", "Search project" have the same function as the icons in the icon bar.
    You can always go back to the tab "Cockpit".

- Search via the functions of the favorites bar

    The favorites bar also offers several ways of searching for a document:

    - Via the "Favorites" memorized searches or data lists can be accessed with a double click.

    - With the "SmartSearch" you can create individual search forms.

    - If you are working with PRO.FILE processes and tasks, you can access the documents linked to a task or process via the task or process structure.

- Search via the icon bar

    In the superior icon bar you can start a search via the following buttons:

    - :Search for document descriptions to be displayed in a list.

    - :Search for parts to be displayed in a list. Documents linked to the part can be displayed in the dependent tabs "Structure" or "Document list".

    - :Search for projects to be displayed in a list. Documents linked to the project can be displayed in the dependent tabs "Structure" or "Document list".

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

2.3 Open CAD documents with linked components

To open a document with linked components from PRO.FILE, the user can choose from several options:

- "Open with latest versions"

- "Open with latest released versions"

- "Open with latest released versions or latest versions"

- "Open version browser"
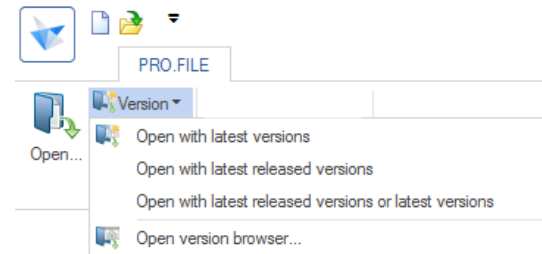
Note:

These 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 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 Inventor session.

- "**Open with latest released versions or latest versions**"

  The assembly/drawing is opened with the released version of all components for which a version in a released status exists. All other components are opened in the latest available version.

- "**Open version browser**"

  Via the version browser, the user can decide for each component, which version is to be loaded.

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

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**.

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

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

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

- 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 Solid Edge and load the assembly from PRO.FILE with the function "**Version**" => "**Open with latest released versions**".

- Since all objects match the selection criterion, the assemblies loaded with the newest released versions in Solid Edge.

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 Solid Edge and load the assembly from PRO.FILE with the function "**Version**" => "**Open with released versions**".

- Solid Edge 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 Solid Edge 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 latest released versions**", and the document structure contains a document for which no released version exists, the CAD document is not loaded.

2.4 Open 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 "**Open selection of versions**" works similar to the function "**Open**" – with the difference that the version browser is displayed after the checkout wizard:
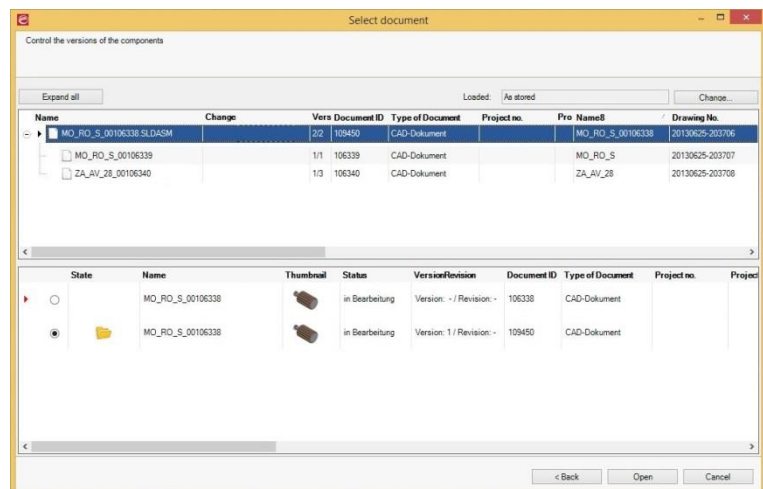
The version browser is divided into two areas:

| 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 Solid Edge.

Function call from the PRO.FILE menu in Solid Edge:
"**PRO.FILE**" => "**Version**" => "**Open selection of versions**"

Proceed as follows

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

2. Select the function "**Open selection of versions**" from the area "**Open**".

⇒ 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.par		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 screen "**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 Solid Edge. The process of opening with the version browser is now finished.

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

Icon	Meaning
💾	Indicates that this version status is the currently saved one.
⚠️	Indicates an object, the version of which has been exchanged.
ⓘ	Shows a version conflict. This can occur, e.g. if a part is used in two assemblies in different versions.
🔷	Icon of Solid Edge assemblies.
🔶	Icon of Solid Edge parts.
🔗	Indicates a softlink.
🔄	Versions reference each other causing a version cycle.

2.5   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.
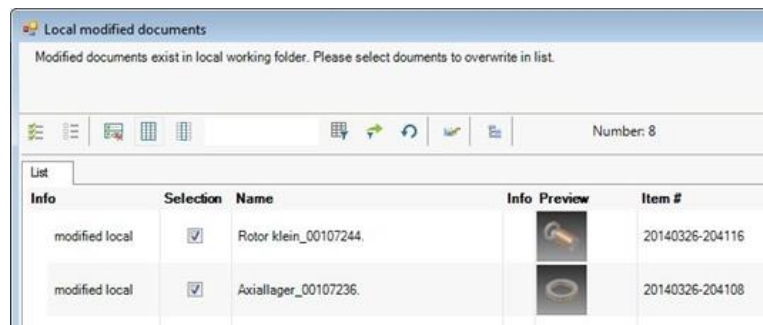
You have the following options of proceeding:

- **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**" => "**Activate work folder**", 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".

3 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

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:

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

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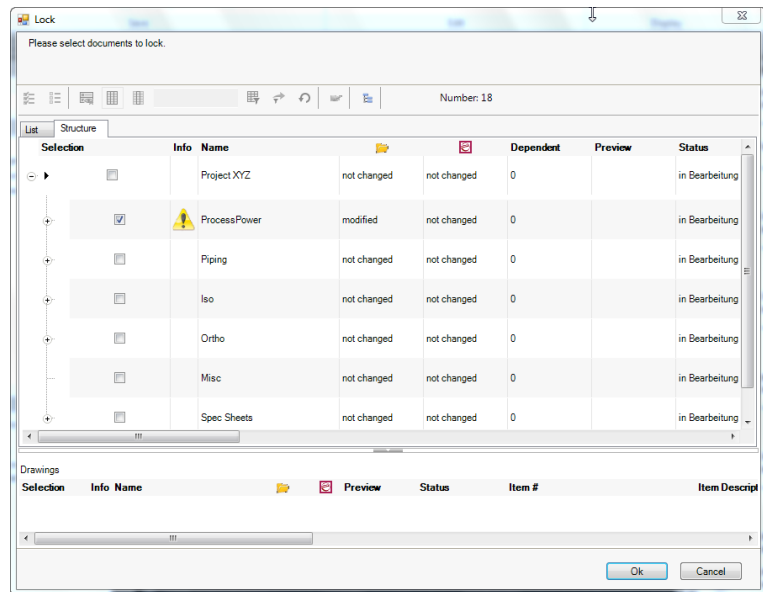
⇒ Displayed entries that are not selected for locking, will not be offered again for locking during the active CAD session.

3.1 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 Solid Edge:

"PRO.FILE" => "Lock"

Proceed as follows

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

2. Select the menu "**PRO.FILE**" from the Solid Edge menu bar.

3. Click on the function "**Lock**.

⇒ The dialog for locking the loaded CAD documents is displayed. (Information on the functions and status information can be found in the chapter "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.
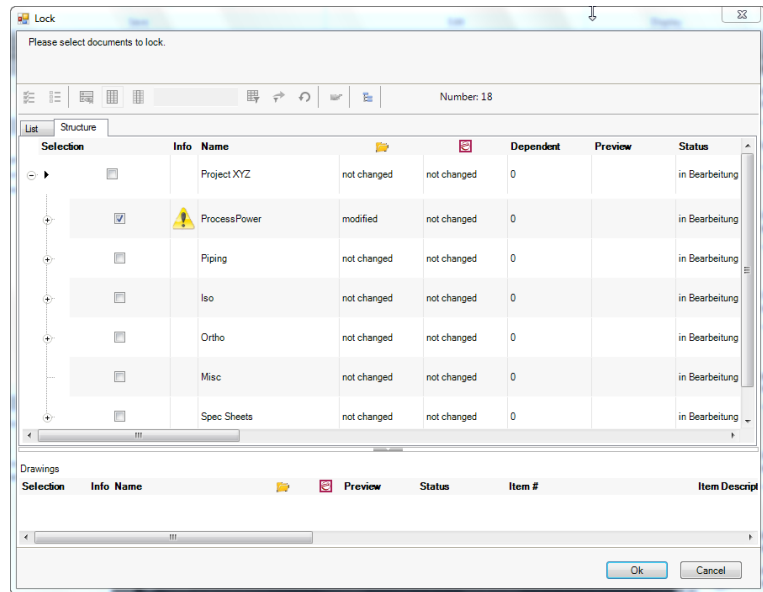
- whether the active documents are up to date and have not been modified by a different user since their opening.

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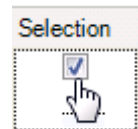
⇒ By the command "Lock", the access permission to edit the document is checked for the current user, and the document is locked for all other users in the database.

⇒ 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.

3.2   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 Solid Edge:

"**PRO.FILE**" => "**Unlock**"

To unlock a document Proceed as follows

1. Make sure that the CAD document to be unlocked is displayed in Solid Edge. Select the menu "**PRO.FILE**" from the Solid Edge menu bar.

2. Click on the function "**Unlock**".

⇒ The dialog for unlocking the loaded CAD documents is displayed. (Information on the functions and status information can be found in the chapter "The document list").
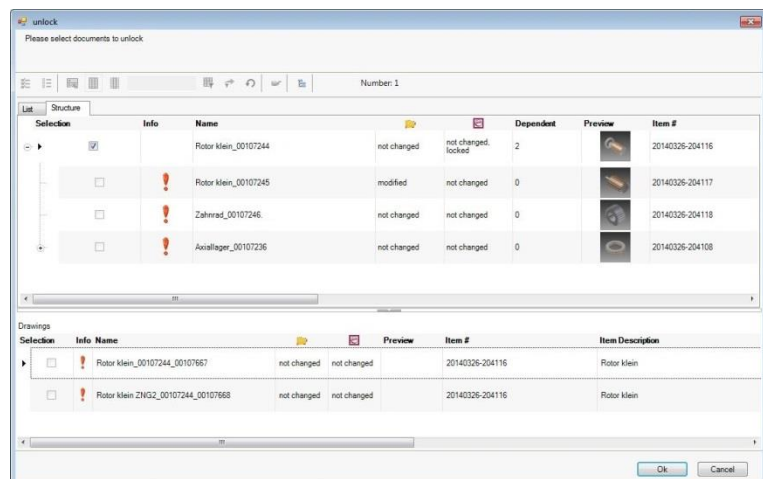
3. Select all document you wish to lock by setting the checkmark in the first column.

4. Confirm your selections with <**OK**>.
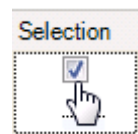
⇒ The lock flag for the selected CAD document is now removed.

4   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:
  - Save: Saving CAD objects for the first time
  - Save: Saving changed CAD documents
- Managed Copy
- Managed Copy automati
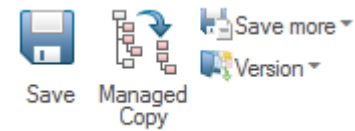
In the menu "**Version**":

- Save as new version/revision

In the menu "**Save more**":

- Save NDF
- Save automatically
- Save Phantom
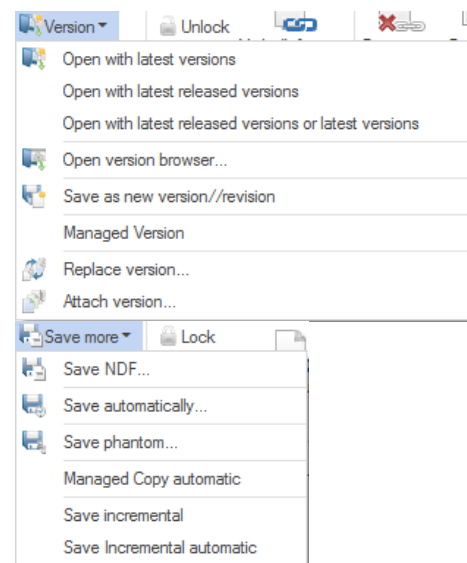
The basic procedure for saving differs depending on whether the CAD data are saved in PRO.FILE for the first time or whether documents from PRO.FILE are saved back after changes have been made to them in Solid Edge.

Therefore, the description is divided into two chapters:

- Save: Saving CAD objects for the first time
- Save: Saving changed CAD documents

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

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

Note:

The PRO.FILE functions "**Save**", "**Managed Copy**", "**Save automatically**" and "**Disconnect relation**" result in a renaming of the files.

If the active document contains renamed CAD elements which are also used in other assemblies, drawings or parts, this results in an **immediate** re-referencing of the assemblies, drawings and parts **opened in the background**. This is a standard behavior of Solid Edge and can therefore not be altered via configuration.

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

Before using the integration PRO.FILE – Solid Edge 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.

4.1 Save: Saving CAD objects for the first time

Via the function "**Save**" you can save parts, assemblies and drawings that you have created in Solid Edge in the PRO.FILE database.

The basic procedure is this:

1. First, you must save your new object locally. This is required by Solid Edge.
2. You can then save your objects to PRO.FILE.
⇒ The action "**Save**" takes places in several steps, with different dialogs being displayed depending on the results of the previous step.

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 Solid Edge:

"PRO.FILE" => "Save"

Proceed as follows

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

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

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

These steps are described in the following sub-chapters.

4.1.1 Check-in 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 Check-in wizard displays the documents that is currently being handled.
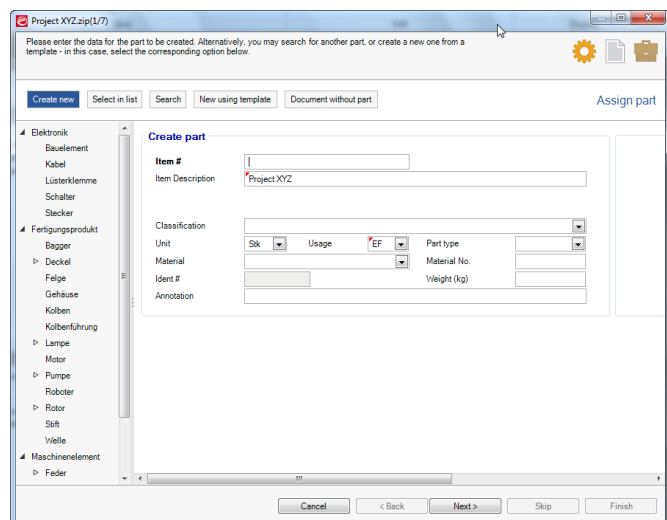
The Check-in wizard offers different options, which can be accessed via the operations bar of the wizard screen:

Create new:

Usage:

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

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

Proceeding:

1. Fill in the attributes (fields) for the description of the part master.

2. After entering all required part data, confirm the creation of the part master record in PRO.FILE with <**Next**>. The new part master record is saved.

Select in list:

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.

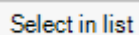⇒ The wizard displays the PRO.FILE surface, as it was opened the last time.

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>.
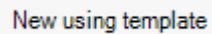
⇒ The input form for the creation of the part master record is pre-filled with the data from the selected part master record.

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 Check-in wizard for parts is skipped. The Check-in 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.

## 4.1.2 Check-in 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 Check-in wizard for the document description is available:

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

Here, too, the Check-in wizard offers different options that can be accessed via the operations bar:
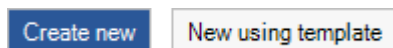
- Create new

- New using template

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

For detailed information see the previous chapter "Check-in 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 Check-in wizard now continues with the options of assigning the newly created objects to a PRO.FILE project.

4.1.3 Check-in 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 Check-in 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 Check-in wizard displays the documents that is currently being handled.
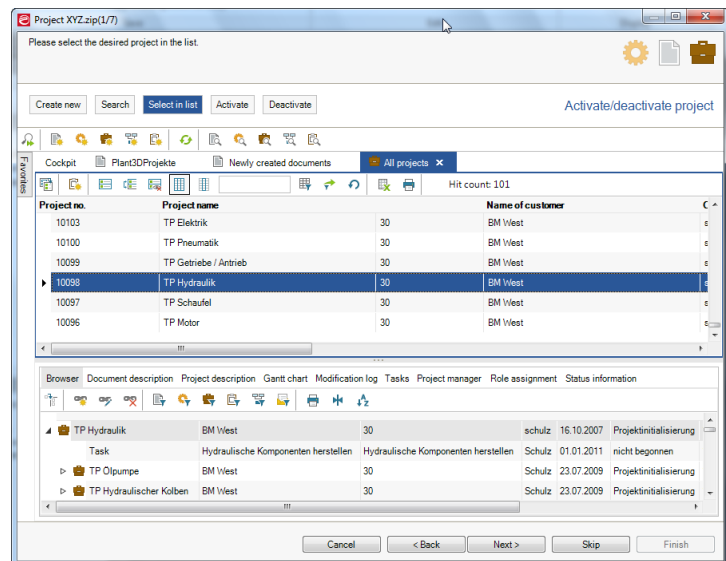
Here, too, the Check-in wizard offers different options that can be accessed via the operations bar:
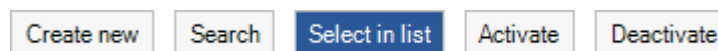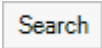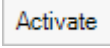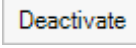
Attention: Project must be activated

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 Check-in wizard:

• 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.

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.

4.2 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.

> **!**
>
> ### 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"
* "Managed Copy" (see following chapter).
* "Save as new version/revision" (see following chapter).

This chapter describes the proceeding for saving changed CAD documents.
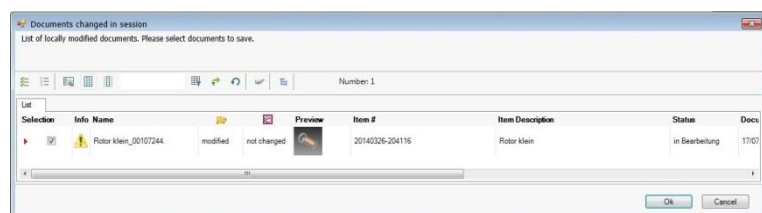
> ➡
>
> ### Function call from the PRO.FILE menu in Solid Edge:
>
> "PRO.FILE" => "Save"

Proceed as follows

1. Go to the integration menu "**PRO.FILE**" in Solid Edge.

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 Solid Edge session. (Information on the functions and status information can be found in the chapter "The document list").

⇒ For assemblies, the structure is analyzed for changed CAD documents and the list of all documents of this assembly is preselected.

⇒ For this list the access permissions for saving the changes of the user are checked. (If the CAD document had been locked before for editing, this prerequisite is fulfilled.)

3. Select all documents you want to save in PRO.FILE. To do so, activate the checkboxes for the desired documents.

4. Confirm your selection with <OK>.
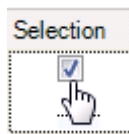
PRO.FILE now checks whether the structure to be saved contains documents that have been changed locally and have not yet been saved to PRO.FILE. If locally changed documents are found, an additional query is displayed.

5. Select all locally changed components you want to save to PRO.FILE. To do so, activate the checkboxes for all desired CAD documents.

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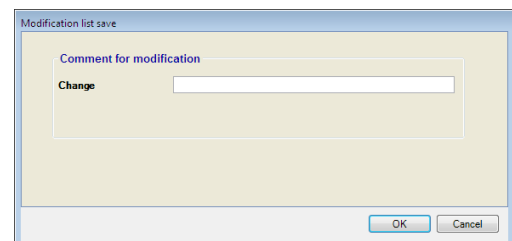
7. Confirm your modification comment with <OK>.

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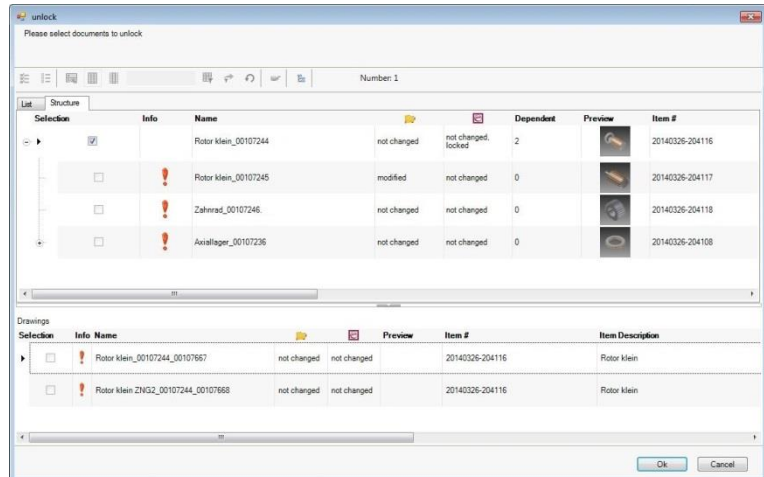
⇒ If documents from PRO.FILE had been locked for editing in Solid Edge, this lock is not automatically removed after saving. The documents remain locked and cannot be changed by other users.

⇒ 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.FIL is now finished.
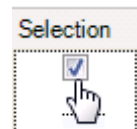
4.3   Managed Copy

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

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

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.

4.3.2   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 Solid Edge

1. Close all higher-level assemblies with references to the model to copy in the Solid Edge session.
2. Open the model to copy via the "**Managed Copy**" function in the Solid Edge session.

!	**Attention: 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 Solid Edge session, a model copied via "Managed Copy" is exchanged in the higher-level assemblies.

3. Activate the model to copy in the Solid Edge 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 Solid Edge 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 Solid Edge 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, is 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.
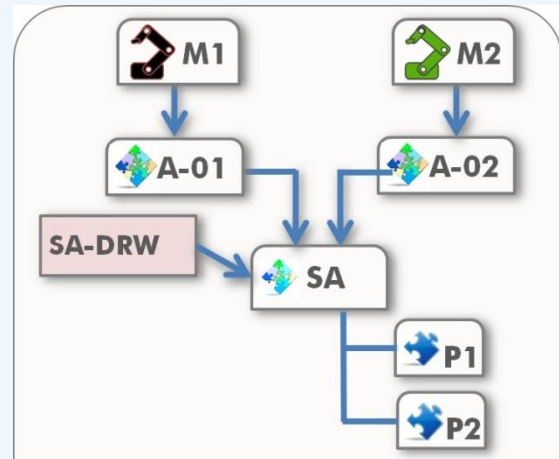
Case study for approach 1B

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 Solid Edge session and the activated CAD documents.

Situation:

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

- 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 Solid Edge session.

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

- The sub-assembly "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 Solid Edge.

- 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".
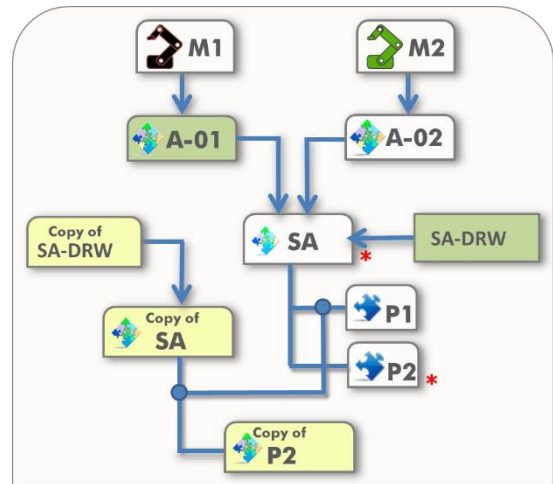

4.3.3   Requirement 2: Exchange a model in an higher-level assembly using "Managed Copy"

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 Solid Edge session.

2. Open the model to copy via the "**Managed Copy**" function in the Solid Edge 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 Solid Edge 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 Solid Edge 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 Solid Edge 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 Solid Edge session and the activated CAD documents.

Situation:

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

- 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 Solid Edge 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 Solid Edge, the references in both assemblies are exchanged by Solid Edge.

- 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".
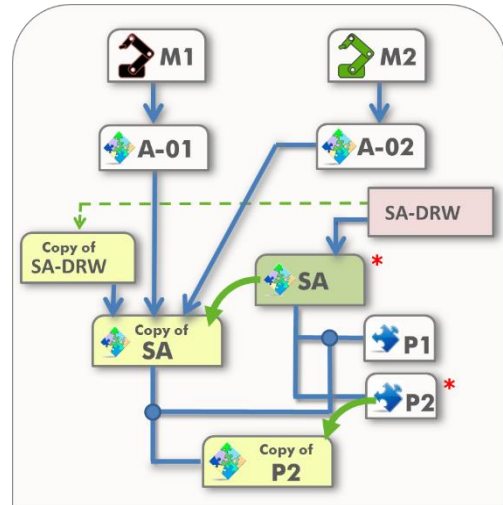
Result IF the drawing was NOT selected for Managed Copy and NOT loaded in Solid Edge:

- 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 Solid Edge, the references in both assemblies are exchanged by Solid Edge.

- 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".
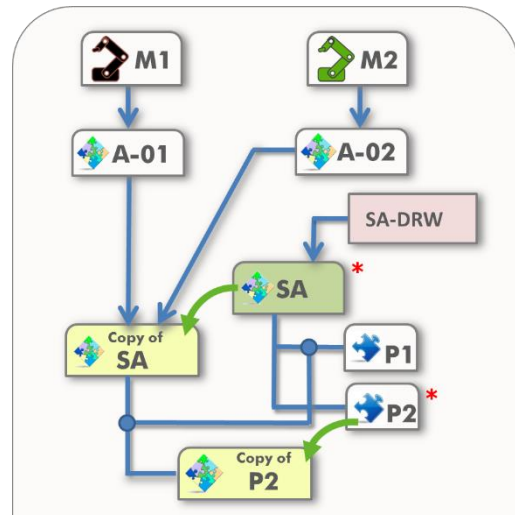
|	**Behavior of the drawing**
	Concerning the dependency of the references, drawings are listed above the model!
	• If the drawing "SA-DRW" were loaded in the Solid Edge session, it would refer to "Copy of SA", too.
	• If the drawing "SA-DRW" is not loaded, but already added to the PRO.FILE structure, the reference is explicitly exchanged and updated by the Integration PRO.FILE-Solid Edge.

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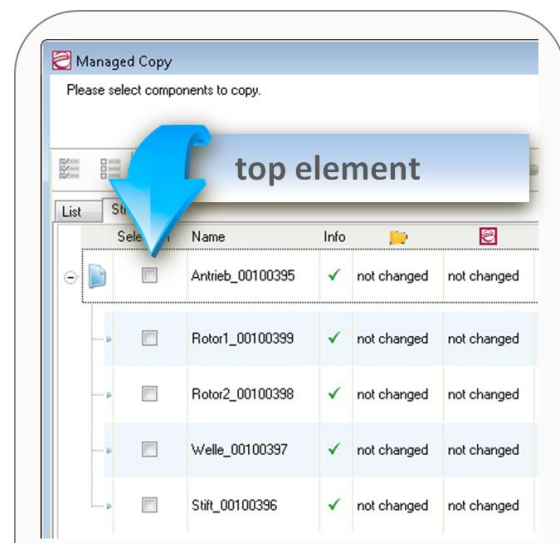
⇒ The created copy of the model is saved in PRO.FILE.

⇒ 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 Solid Edge session and the activated CAD documents.

Situation:

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

- 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.
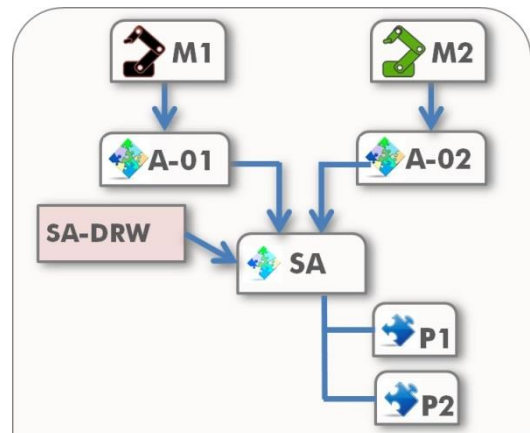
Exchange "AS" via "Managed Copy" only in "A-01"

- 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**
>
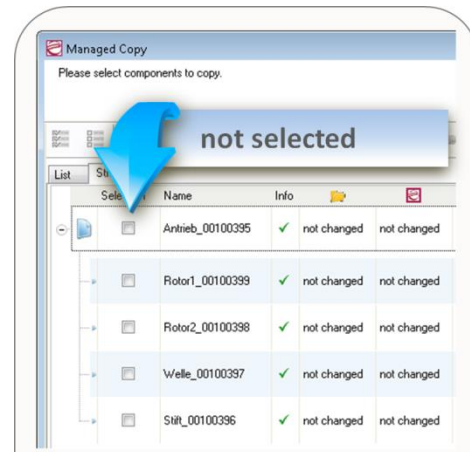> When a sub-assembly is selected in the dialog of Managed Copy, the drawing linked to this sub-assembly in PRO.FILE is also activated for Managed Copy and thus copied. This affects the result of Managed Copy as the following examples show.
>
> For the cases, in which the drawing is not to be copied, 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".
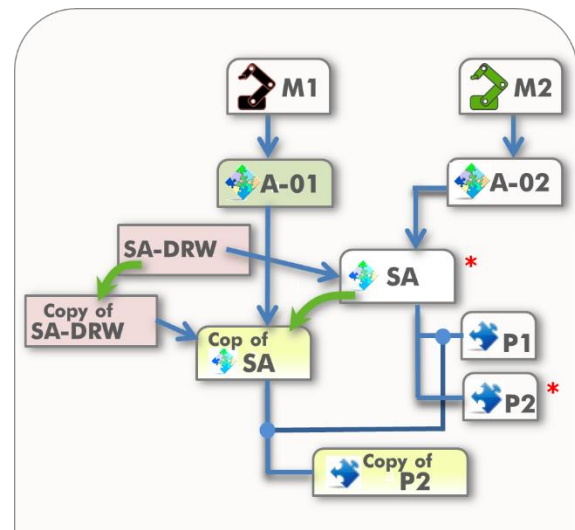
- The drawing "SA-DRW" is copied. The drawing "SA-DRW" refers to the sub-assembly SA, the drawing "Copy of SA-DRW" refers to the copy of "SA".

- 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 Solid Edge session:

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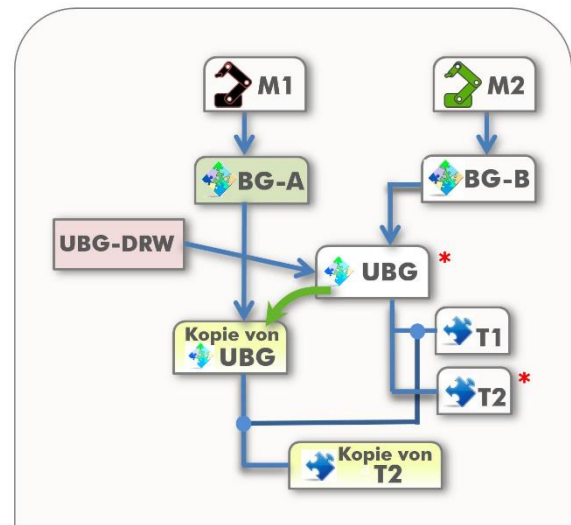
- The integration explicitly exchanges the references in assembly A-01, so that these references point to the copied sub-assembly "Copy of SA".

- 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".

4.3.4 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 Solid Edge 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 Solid Edge:**
	"PRO.FILE" => "Managed Copy..."

Proceed as follows

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

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 for Solid Edge always determines the substructure **based on the active CAD document**. The substructure of the CAD document depends on the CAD system.
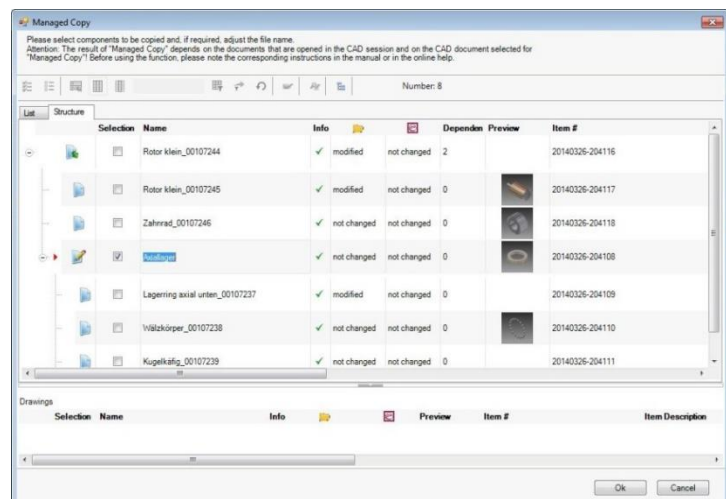
⇒ In a second step the substructure is expanded by the related drawings. This "special provision" is required because the drawings are listed **above** the model depending on references.

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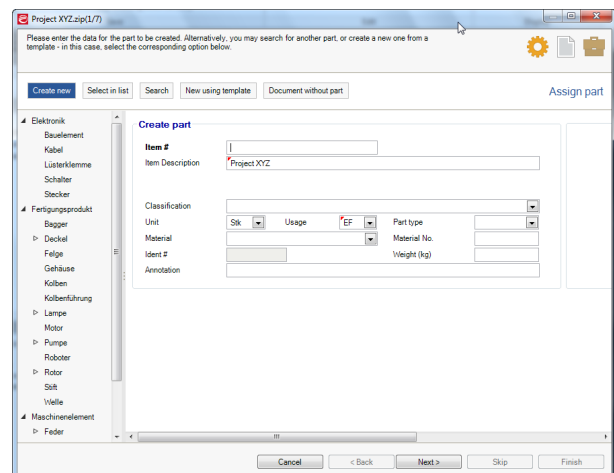
You will find Information on how to use the check-in wizard in the previous chapter "Save: How to save CAD data and changes to PRO.FILE?".

> **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.

4.3.5 How is the proceeding in "Managed Copy" concerning drawings?

Due to the fact that drawings are listed in the CAD structure above the models, there is no 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.

4.3.6   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.
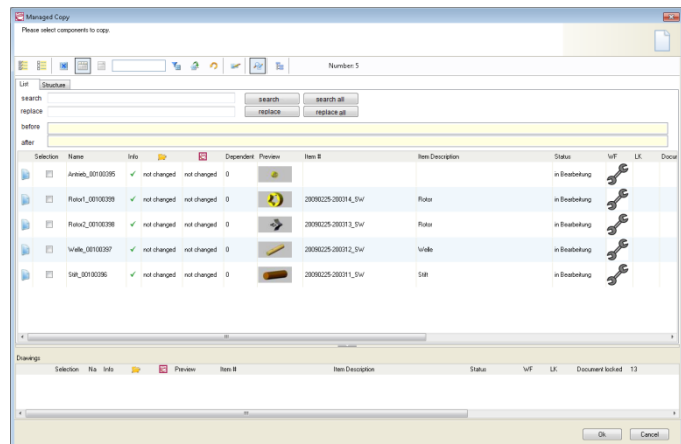
⇒ Now you have different possibilities, to execute "search and exchange".
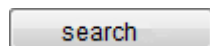
search	searches and selects the next hits in the list
search all	searches and selects all different hits in the list
replace	replace the next hit
replace all	replace all hits in the list
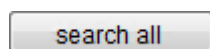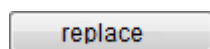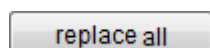
⇒ A preview for the editing of the file name is shown in the fields "before" and "after"

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

4.4 Managed Copy automatic

The function "**Managed Copy automatic**" 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 Solid Edge session and the CAD document selected for "Managed Copy"! If higher-level assemblies are opened in the Solid Edge 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 Solid Edge 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 Solid Edge:
"PRO.FILE" => "save other" => "**Managed Copy automatic**"

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

4.5   Save as new version/revision

With the PRO.FILE Solid Edge 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/revision**", 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 Solid Edge.

- If a part is versioned in this way using the 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.FILE. The assembly structure is then also updated in PRO.FILE.

- If an assembly is versioned using the "Save as new version/revision" 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 Solid Edge:

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

Proceed as follows	1. Select the "**PRO.FILE**" menu from the menu bar in Solid Edge.
	2. Select the function "**Version**" => "**Save as new version/revision**".
	3. If you have not opened the newest version from the version chain but an older version instead, it depends on setting of the parameter "Ask for confirmation when creating a version from an old version" in the PRO.FILE Management Console whether a dialog is displayed.
	4. If the dialog is displayed, confirm it with <**Yes**>.
	⇒ A list with all documents, of which a new version will be created, is displayed.
	5. 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 Solid Edge.
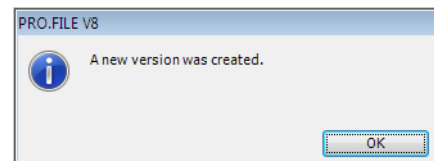
Attention: New version is not locked

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 "<u>Lock/Unlock: Who can change when?</u>".

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

Attention:

Saving parts with published configurations as a version is not possible. Only unpublished part families can be versioned.

|	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**".

4.6   Save NDF

The integration PRO.FILE Solid Edge offers the possibility to convert a Solid Edge 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.
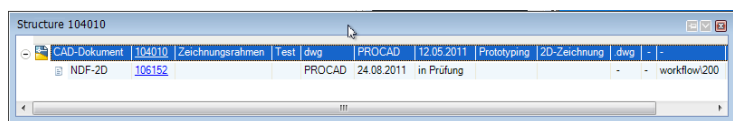
This function is only available for drawings, so the menu entry is only displayed in drawing mode.

	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 Solid Edge:
	"**PRO.FILE**" => "**Save more**" => "**Save NDF…**"

Proceed as follows

1. You have opened a drawing and wish to document the current drawing status.
2. Select the "**PRO.FILE**" menu from the menu bar in Solid Edge.
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.

$\Rightarrow$ 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.

4.7 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 Check-in wizard is only made for the first part and document description in PRO.FILE.

- For all further CAD documents to be saved **no** Check-in 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.

Note:

The PRO.FILE functions "**Save**", "**Managed Copy**", "**Save automatically**" and "**Disconnect relation**" result in a renaming of the files.

If the active document contains renamed CAD elements which are also used in other assemblies, drawings or parts, this results in an **immediate** re-referencing of the assemblies, drawings and parts **opened in the background**. This is a standard behavior of Solid Edge and can therefore not be altered via configuration.

"Save automatically" for complete assemblies

When an assembly is opened within the Solid Edge 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 Solid Edge:

"**PRO.FILE**" => "**Save more**" => "**Save automatically**"

Proceed as follows

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

2. Select the function "**Save automatically**".

⇒ For the first document that is unknown to PRO.FILE, the normal saving process is started.

⇒ The following is displayed:

- The Check-in wizard to create the part description in PRO.FILE.

- Then (depending on the configuration) the Check-in wizard to create the document description in PRO.FILE.

- Finally, (depending on the configuration) the Check-in wizard for the project assignment in PRO.FILE.

3. Go through all steps of the Check-in wizard for the first new CAD document. Detailed information on this can be found in the previous chapter "[Save: 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.
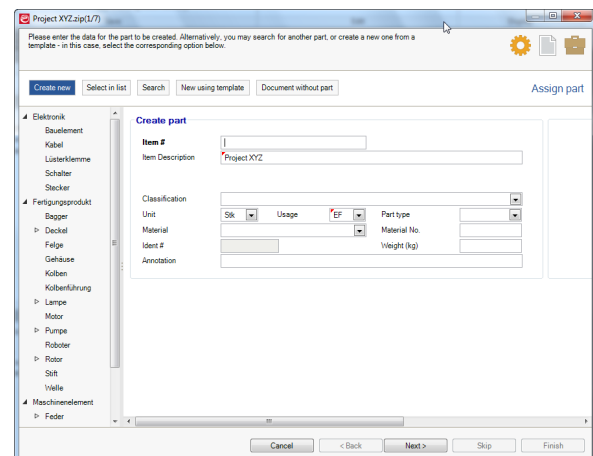
4. For modified components already known in PRO.FILE:
   If the assembly also contains components that are already saved in PRO.FILE and that have been changed in the session or locally, these components are also offered for saving.
   Select the components to be saved and confirm your selection.

⇒ 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 automatically**" 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 Check-in 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 automatically", 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 – Solid Edge.

4.8   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-Solid Edge 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 Solid Edge. 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 Solid Edge 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).

	Function call from the PRO.FILE menu in Solid Edge:
	"PRO.FILE" => "Save more=> "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 Check-in 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 Check-in 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 – Solid Edge.

4.8.1   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 Solid Edge. If the phantom assembly is still opened in Solid Edge, 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.

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 Solid Edge:
	"PRO.FILE" => "Disconnect relation"

Proceed as follows

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

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 "[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.

4.8.2 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 "bolt1_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.

4.9   Save incremental

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 "**Save incremental**" does not search the active folder for corresponding drawings.
- The function "**Save incremental**" 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 Solid Edge:
	"PRO.FILE" => "Save" => "Save incremental"

The further proceeding for the function "**Incremental save**" corresponds to the proceeding described in the chapter "**Fehler! Verweisquelle konnte nicht gefunden werden.**".

4.10   Save incremental automatic

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" => "Save incremental automatic"

5   Linking of additional files

It is possible to link additional files to Solid Edge 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:

- [Add additional file](#)
- [Add PRO.FILE document](#)
- [Detach document](#)

When opening a Solid Edge 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 Solid Edge 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.
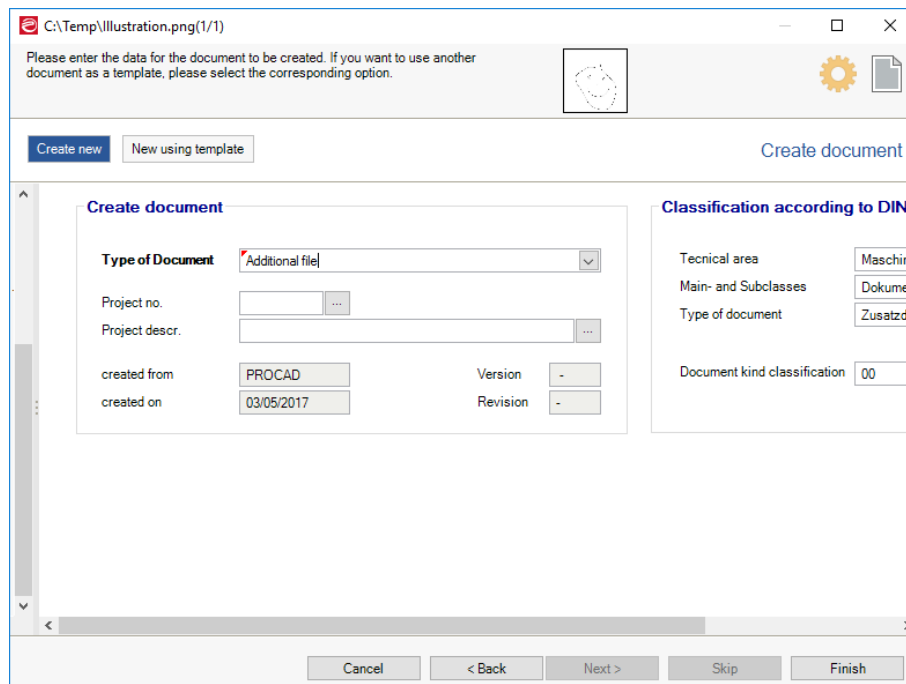
5.1   Add additional file

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

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

2.   Select the function "**Save…**" => "**Add additional file**".

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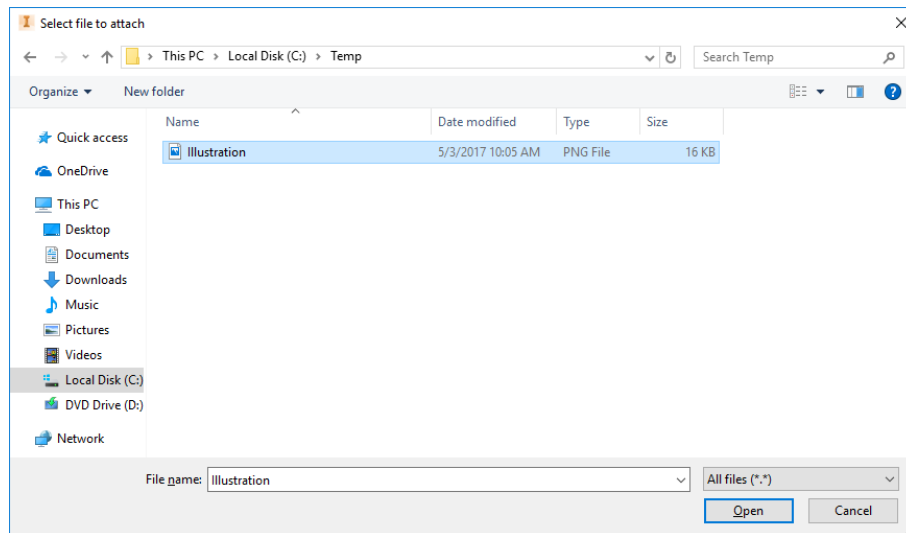
3.   Select the file to be added and confirm your selection with <**Open**>.

⇒   The part master record of your Solid Edge object is displayed in PRO.FILE.

4.   Confirm the assignment with <**Next**>.

$\Rightarrow$ 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".

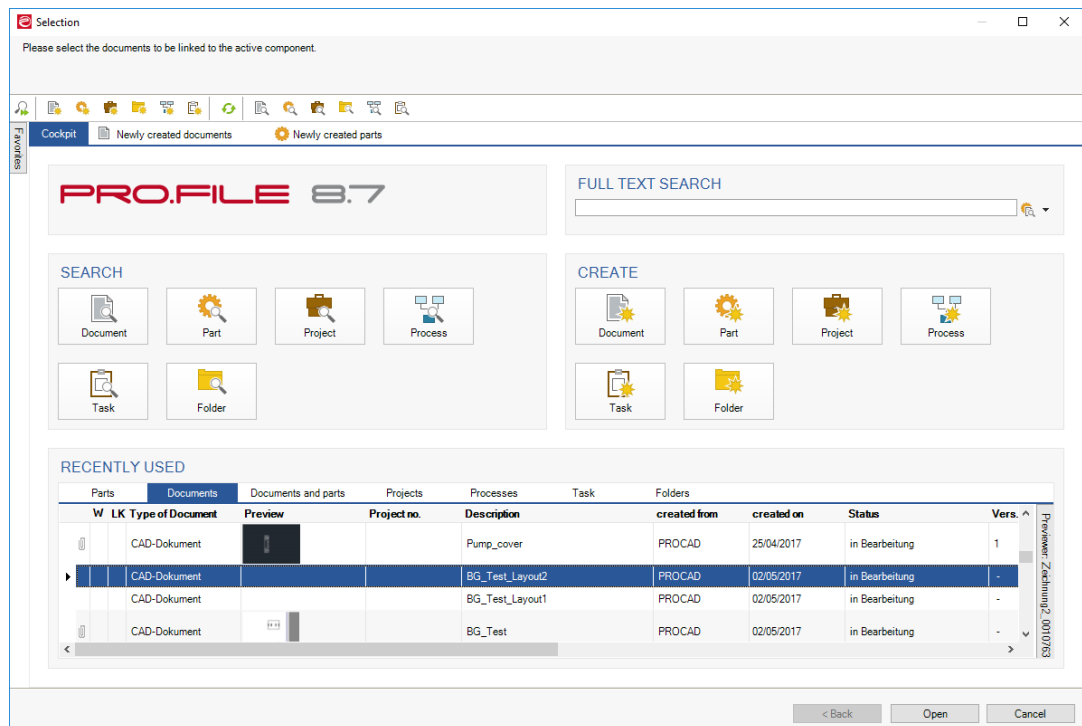5. Enter the information for the new document record and confirm your input with <Finish>.
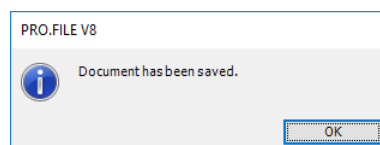
$\Rightarrow$ The additional file is saved in PRO.FILE. It is linked below the document description of the Solid Edge object. If possible, a preview file is created for the additional file.

$\Rightarrow$ By adding it to the Solid Edge structure, the additional file is automatically copied into the Workcenter folder.

5.2 Add PRO.FILE document

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

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

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

$\Rightarrow$ 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 Solid Edge object.
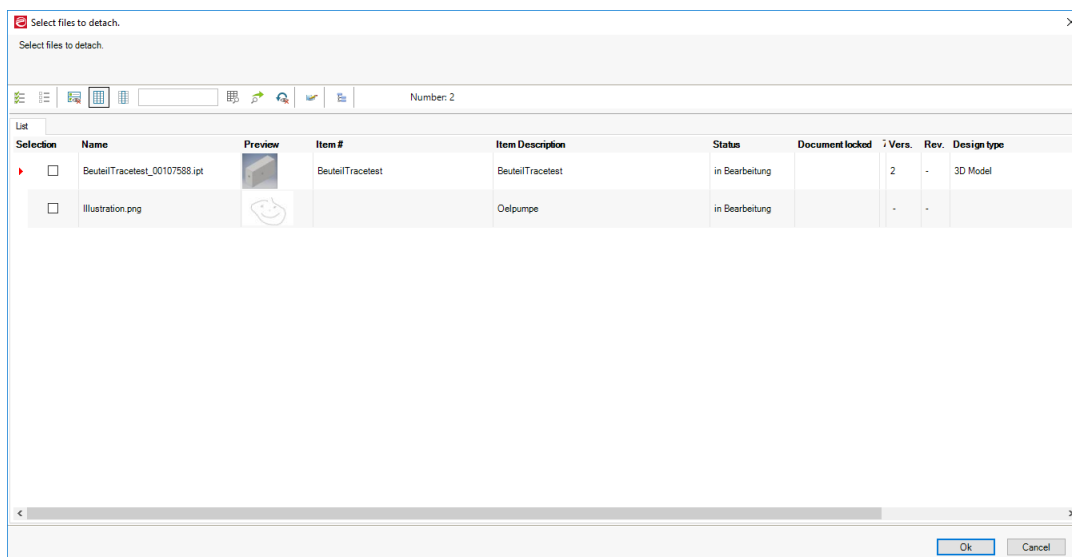
5.3 Detach document

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 Solid Edge object that has been saved in PRO.FILE (and that contains the additional file) into your CAD session.

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

⇒ 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 detach by activating the corresponding checkbox in the column "Selection".
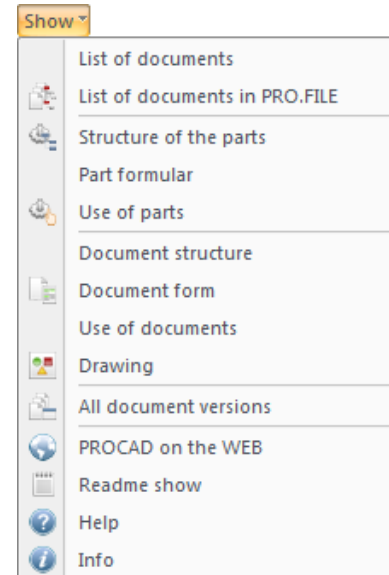
4. Confirm your selection with <OK>.
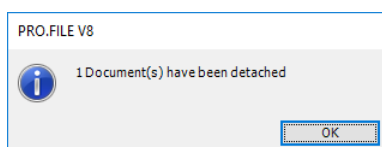
⇒ The selected document is removed from the Solid Edge object structure.

6   Show: PRO.FILE Information at a glance

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

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

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

Show ᵛ
List of documents
List of documents in PRO.FILE
Structure of the parts
Part formular
Use of parts
Document structure
Document form
Use of documents
Drawing
All document versions
PROCAD on the WEB
Readme show
Help
Info

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

- The document list

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

- Direct information in the dialog screens

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

  - Up to date or not: Display of status information

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

6.1 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.
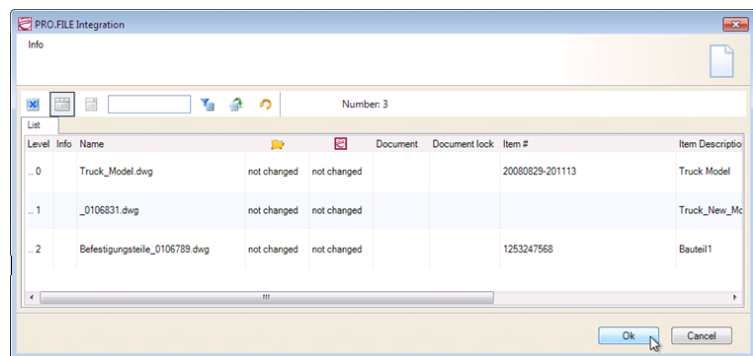
Function call from the PRO.FILE menu in Solid Edge:

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

- "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.

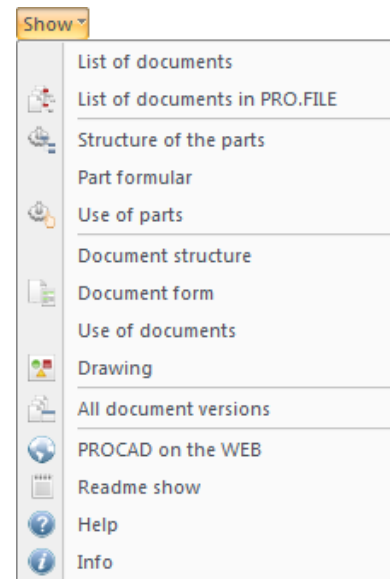Detailed information can be found in the following chapters:

- [Direct information in the dialog screens](#)

- [More comfort: search and list functions in the dialog screens](#)

- [Up to date or not: Display of status information](#)

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

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

- These menu entries access information on the CAD document currently active in Solid Edge.

- 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.

Show ⌄
List of documents
List of documents in PRO.FILE
Structure of the parts
Part formular
Use of parts
Document structure
Document form
Use of documents
Drawing
All document versions
PROCAD on the WEB
Readme show
Help
Info

Function call from the PRO.FILE menu in Solid Edge:

"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:

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 "The document list".

List of documents in PRO.FILE

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

Part structure

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

Part form	The function "**Part form**" displays the part master record form of the part the current CAD document is attached to in PRO.FILE.
Part usage	With the function "**Part usage**" you can see whether you current CAD document is used by other assemblies. The usage list displays the "upward" structure.
Bill of materials	The function "**Bill of materials**" displays the PRO.FILE bill of materials for the active drawing.
Document structure	With the function "**Document structure**" you can see which documents (= part drawings) are used in your drawing (= main drawing).
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.
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.
Drawing	If a drawing is saved in PRO.FILE for the active assembly or part, this drawing can be selected in PRO.FILE and viewed via the function "**Show**" => "**Drawing**".

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.	
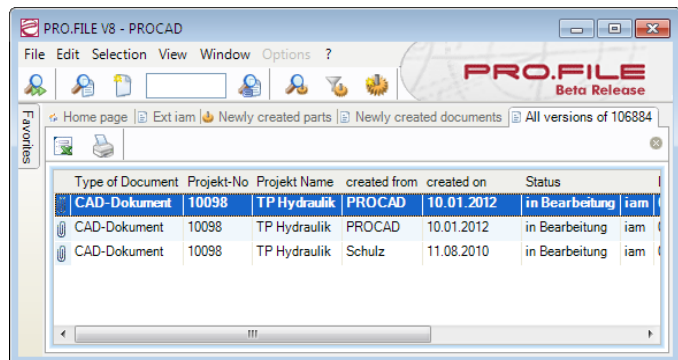
PROCAD on the WEB	This menu point opens the PROCAD homepage, as long as the user has internet access.
Readme show	Via the readme file you can view the newest system information on the integration PRO.FILE – Solid Edge.
Help	Opens the PRO.FILE online help.
Info	Shows the Solid Edge integration version number that is being used.

6.3   Direct information in the dialog screens

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

These offer the
following possibilities:
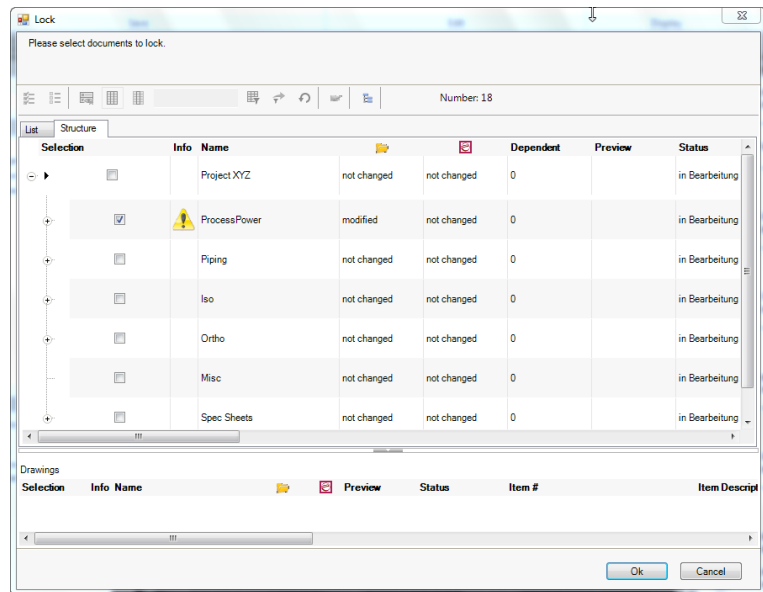
- You can switch between list and structure view.

- Via the buttons of the list functions you can make searches and filter the results (see the following sub-chapter "More comfort: search and list functions in the dialog screens").

- If objects are listed in the assembly tree above the actual node (e.g. drawings), these objects are displayed in a dependent window.

- The lists contain status information for each of the listed objects (see the following sub-chapter "Up to date or not: Display of status information").

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

The dialog screens of the PRO.FILE integration contain different search and list functions, as known from the PRO.FILE GUI:
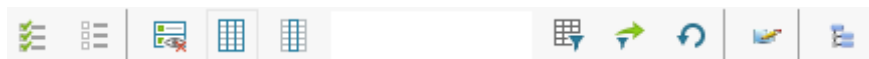
Via these buttons, the following functions are available:

- **Select all rows:**

  With this button, all rows of a list are highlighted.

- ▤ Invert selection:

  With the <Shift> key pressed down, it is possible to select whole areas of a list, with the <Ctrl> key pressed down, you can select several individual rows. The button "Invert selection" can be used to select everything that is not selected and unselect everything that was selected.

- ▤ Hide selected rows:

  If several rows of a list are selected, these rows can be hidden from the list with this button.

- ▦ ▦ Search in all columns / Search in active columns:

  In order to be able to perform a targeted search for terms in the list, the user first has to select whether the search is to be carried out across all columns in the list or only for a specific column in the list.

  - ▦ : The search is performed across all columns in the list.

  - ▦ ▤ : The search is performed for the active column only. A column is activated by clicking the respective column header.

- ▦Define Filter pattern / Filter:

  A character string can be entered into the entry field located within the icon bar. Here you can use the already described wildcards/meta characters.

  The search for the entered character string is started using the ▤ icon.

  If the search pattern is found, all matching data records are highlighted.

- ⤴ Next found pattern:

  This icon is used to once again compare the entered filter pattern with the columns that are to be searched. The next data record found is highlighted.

- ↻ Show hidden rows:

  If rows of a list have been hidden, this button can be used to display them again.

- ▥ PRO.FILE list selection:

  The entries of the selected rows are selected and opened in a list in PRO.FILE. This ways you can immediately view the stored information without further selection.

6.3.2   Up to date or not: Display of status information

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

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

- : 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.

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

# 7   Version administration with the Integration

The integration PRO.FILE – Solid Edge offers several functions for opening and saving when working with versions:

- [Open CAD documents with linked components](#)
- [Open version browser](#)
- [Save as new version/revision](#)

These three functions have already been described in previous chapters.

Furthermore, there are the functions:

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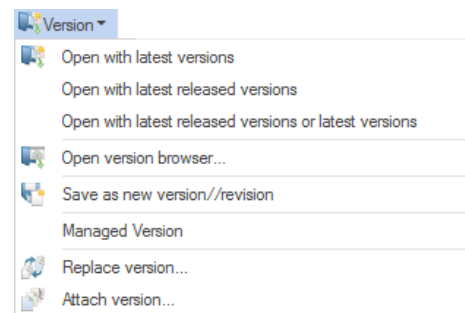
Information on this can be found in the following sub-chapters.

7.1   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 Solid Edge:

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

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

- If no assembly is opened in Solid Edge, 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 Solid Edge, 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"](#)

## 7.1.1 The proceeding for "Managed Version"

Proceed as follows

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

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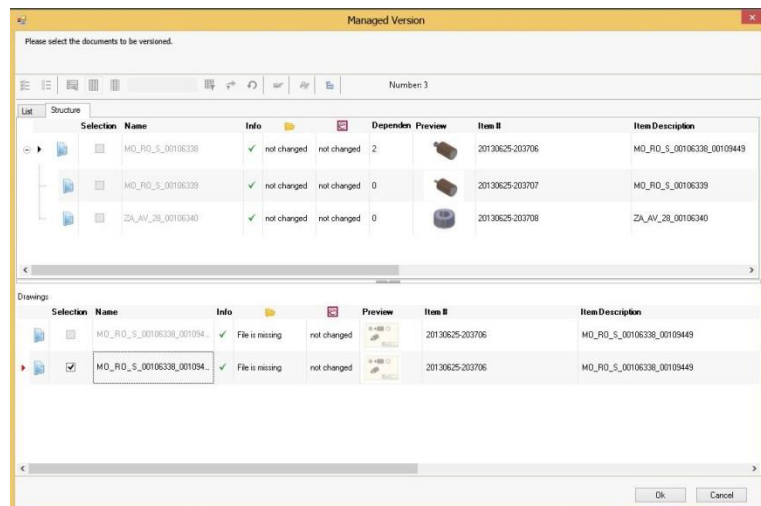
⇒ In the wizard of "Managed Version" the tree structure, determined and expanded by drawings, is shown, so that the documents to version with their dependent data (nodes) can be selected.

⇒ 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.
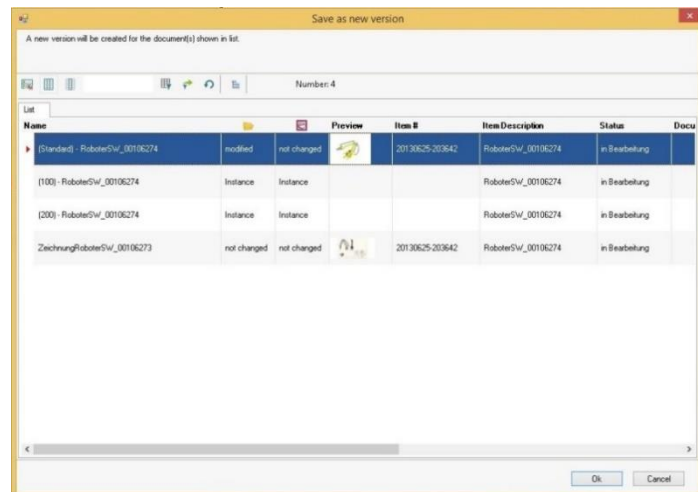

4.   Confirm your selection with <OK>.

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

5.   Confirm with <OK>.

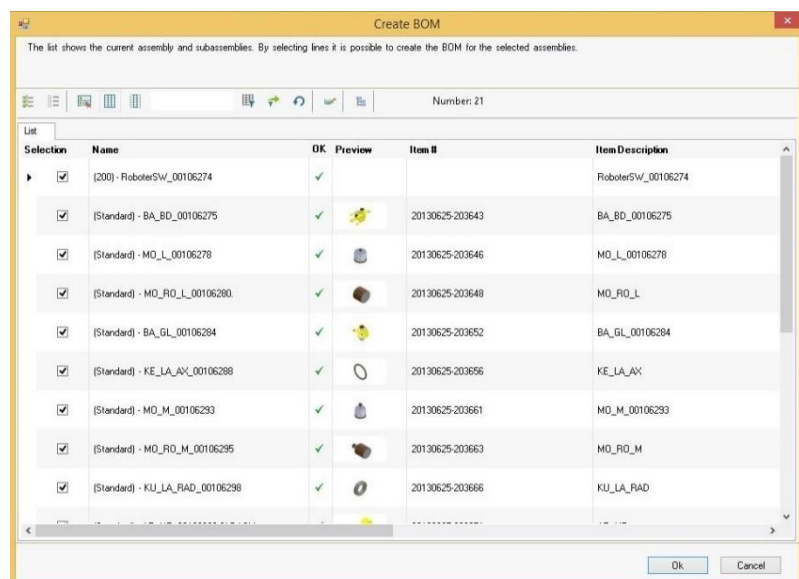⇒   The selection components are now versioned.

⇒   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.

7.2   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 Solid Edge:

"**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 Solid Edge (open the replacing document, not the document to be replaced).

2. Select the function "**PRO.FILE**" => "**Version**" => "**Replace version**".

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

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

4. 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).

	Note:
	It is not possible to synchronize a version chain with this function!
	You can **only replace the respective predecessor version step by step**.

	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.

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

7.3 Attach version

With the function "**Append version**" the active CAD document is attached to an existing document in PRO.FILE as a new version.

The difference to the function "**Replace version**" is the following:

- The function "Replace version" modifies the active assembly so that the previous version cannot be restored.

- If you want to keep the previous version state, you can use the function "Attach version".

Note: Prerequisite for the function "Append version"

The function "**Append version**" can only be used for PRO.FILE objects. The CAD document itself does not necessarily have to be a version. The target object must be of the same type like the object you want to attach: A part cannot be attached to the document description of an assembly.

You opened a part in Solid Edge and you want to attach this part to another CAD document in PRO.FILE as a new version.

Function call from the PRO.FILE menu in Solid Edge:

"**PRO.FILE**" => "**Version**" => "**Attach version**"

Proceed as follows

1. Select the function "**PRO.FILE**" => "**Version**" => "**Attach version**".

⇒ PRO.FILE opens and prompts you to select a document description, the current object is to be appended to as a new version. The wizard displays the PRO.FILE GUI as it was last used.

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

   - Via the tab "Cockpit".

   - Via the search functions of the icon bar.

   - Via favorites, SmartSearch or task assignments.

3. If the desired document is displayed in a list view, **select** it. (If the desired document is displayed in a form view, it is already selected).

4. Click <**Confirm**> in the wizard.

⇒ The active document is now saved as a new version of the selected document description in PRO.FILE.

⇒ The appending process is thus finished.

8   Additional functions of the integration

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

These functions are described in the following sub-chapters:

*   [Managed Rename: Renaming in the structure](#)
*   [Disconnect relation](#)
*   [Document refresh](#)
*   [Insert part](#)
*   [For assemblies: Replace part](#)
*   [For assemblies and drawings: Create BOM](#)
*   [For drawings: Create balloon](#)
*   [For drawings: Drawing legend](#)

8.1   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 Solid Edge. 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 Solid Edge:

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

Proceed as follows

1. Select the "PRO.FILE" menu in Solid Edge.

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

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

Select the desired document in the Checkout wizard

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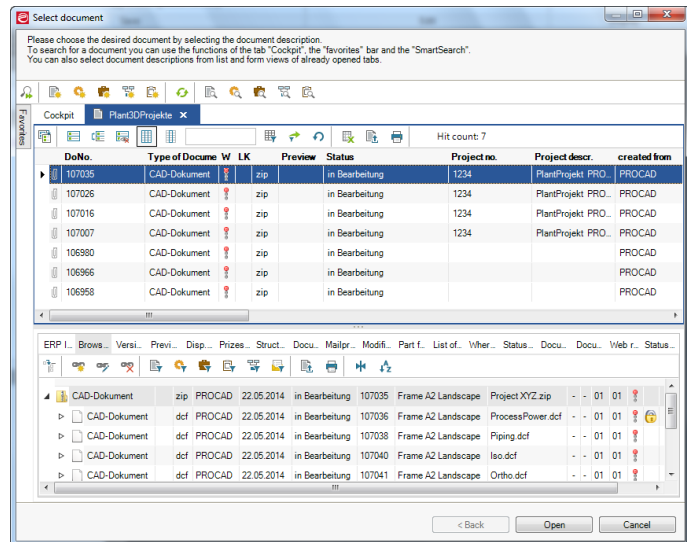
4. If the desired document is displayed on a tab, select it and click <**Open**>.

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

Detailed information on the Checkout wizard can be found in the chapter "Open: Loading CAD data from PRO.FILE".

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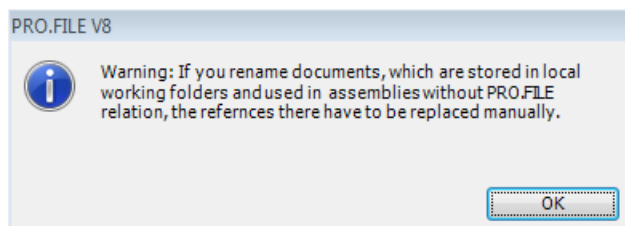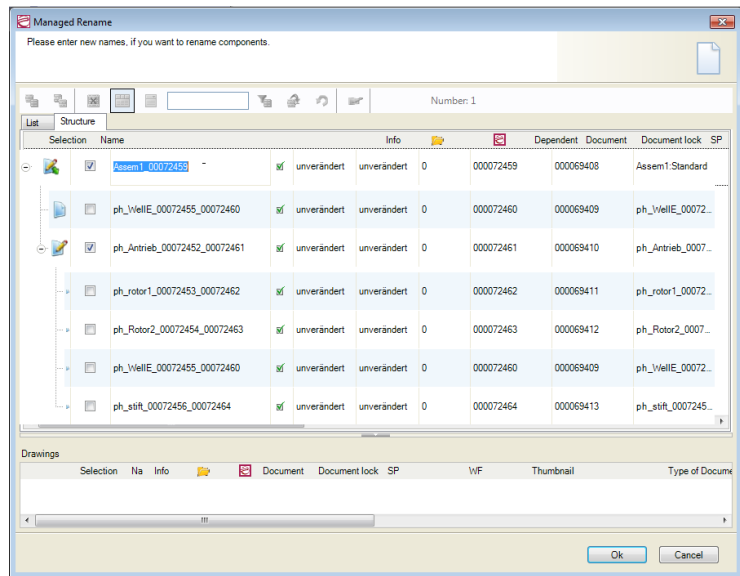

⇒   If the documents selected for renaming are used elsewhere, this cannot be recognized automatically. In such a case, manual post-processing would be necessary.

5. Confirm the warning message with <**OK**>.
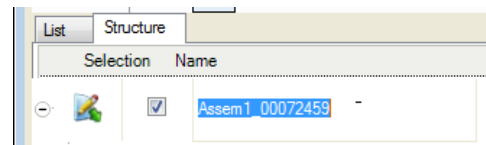
Start the renaming in the structure

⇒ The window
    "Managed Rename"
    is displayed.
    In this windows you
    can rename the
    files:

6. **Select**: Select all components for which you want to change the file
    name by using the checkboxes.

7. **Change file names**: You can edit the
    file names directly in the list.

8. Make the changes for all desired components.

9. 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.

8.2   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 Solid Edge and cannot be influenced by PRO.FILE.

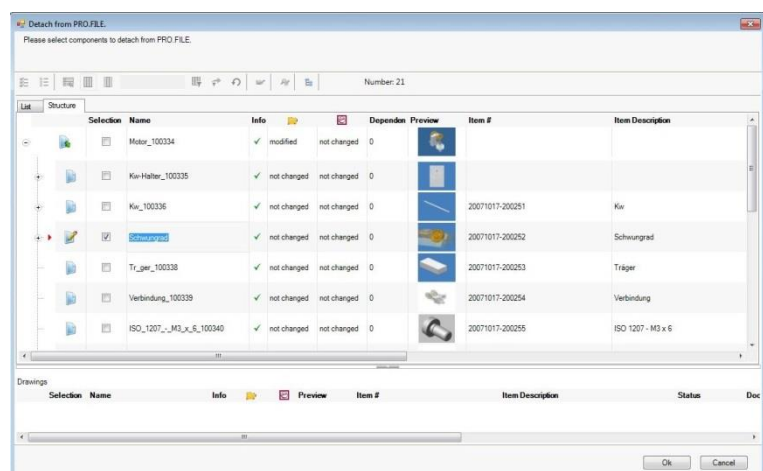**Function call from the PRO.FILE menu in Solid Edge:**

"PRO.FILE" => "Disconnect relation"

Proceed as follows

1. Select the "**PRO.FILE**" menu in Solid Edge.

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 "[The document list](#)").
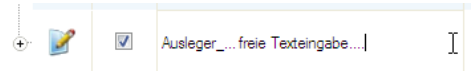
3. Select all CAD documents, the PRO.FILE connection of which you want to dissolve by using the checkboxes.
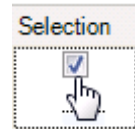
⇒ The previous file name without ID, or with the suffix "_Index" is suggested as new file name, e.g. "**assembly_1**".

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

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

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

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

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

To save these CAD files again to PRO.FILE see chapter "Save: How to save CAD data and changes to PRO.FILE?".

Note:

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

8.3 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.

Function call from the PRO.FILE menu in Solid Edge:

"PRO.FILE" => "Document refresh"

Proceed as follows

1. Select the "**PRO.FILE**" menu in Solid Edge.

2. Select the function "**Document refresh**".

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

The following options are available:

- Documents that have been modified in PRO.FILE are updated in the Solid Edge session.

- The file properties are updated.

- The drawing title block is updated (only available if a drawing is active).

- The draft views are updated (only available if a drawing is active).

- The used version difference is displayed in the EdgeBar (not displayed for documents).
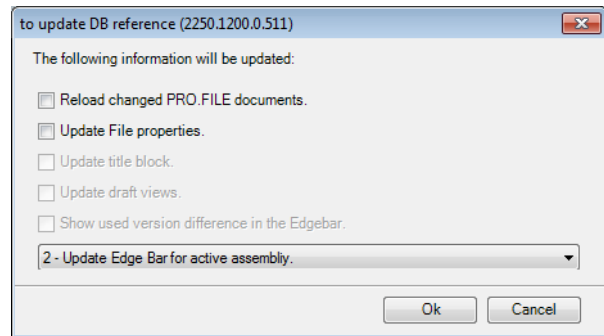
⇒ If the checkbox "**Show used version difference in the EdgeBar**" is activated, you can choose between five different configuration options in the drop-down menu:

- **0 – do not update EdgeBar**:

  The Edgebar is not updated.

- **1 – Update EdgeBar recursive**:

  All parts listed in the EdgeBar are updated. This may lead to performance problems for large assemblies.
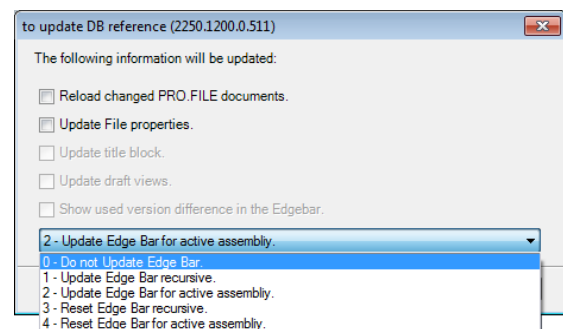
- **2 – Update EdgeBar for active assembly**:

  Only the first level of the list is updated.

- **3 – Reset EdgeBar recursive**:

  With this setting you can undo an update. All updated parts are reset.

- **4 – Reset EdgeBar for active assembly**:

  With this setting you can reset the updated data of the assembly on the first level.

3. Confirm your selection with <OK>.

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 and 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!

8.4   Insert part

Via the function "**Insert part**" assemblies or parts saved in PRO.FILE can be inserted into the currently active Solid Edge document.

The CAD document is selected via the PRO.FILE Checkout wizard and then placed with the Solid Edge functions.

Function call from the PRO.FILE menu in Solid Edge:

"PRO.FILE" => "Insert part"

Proceed as follows

1.   Select the "**PRO.FILE**" menu in Solid Edge.

2.   Select the function "**Insert part**".

⇒   The Checkout wizard now prompts you to select the component in PRO.FILE that you want to insert into the assembly.

3.   Select the desired CAD document and click <**Open**>.

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

4.   Select now the Solid Edge placing data for the component in your CAD document.

⇒ After you have defined all levels/coordinates, the component from PRO.FILE is inserted into your CAD document.

Note: Insert part several times

The function "**Insert part**" can be used several times in an assembly.

Not for parts: It is not possible to insert several objects from PRO.FILE in a part. After you have inserted one object from PRO.FILE into your part, you cannot insert an additional object.

8.5   For assemblies: Replace part

Via the function "**Replace part**" it is possible to replace an element **on the first level** of an assembly with an object from PRO.FILE.

Function call from the PRO.FILE menu in Solid Edge:

"**PRO.FILE**" => "**Replace part**"

You have loaded an assembly from PRO.FILE that contains one or more elements. You want to replace an object (e.g. a screw) with a different object (different screw), which is also saved in PRO.FILE. In this case it is possible to overwrite local versions.

Proceed as follows

1. Select the component in your Solid Edge assembly that you want to replace with the component from PRO.FILE.

2. Select the function "**PRO.FILE**" => "**Replace part**"

⇒ You are now prompted to select the component in PRO.FILE, which is to replace the component in your assembly, in the Checkout wizard.

3. Select the desired CAD document and click <**Open**>.
   (Detailed information on the Checkout wizard can be found in the chapter "Working with the Checkout wizard to search for CAD documents".)

⇒ The selected component in Solid Edge is replaced by the selected component from PRO.FILE.

Note: Only available for assemblies

This menu entry is only available when an assembly is displayed.

Note: Display of the object name in the EdgeBar

After the replacing of the object, the name of the replaced object is still displayed in the EdgeBar. The display can be refreshed with the function "Document refresh".

Replace part in complex assemblies

The function "**Replace part**" is only available for objects on the first level in Solid Edge. If you use this function in a complex assembly and have selected an element that belongs to a sub-assembly, a corresponding message is displayed.

Note:

When using the function "Replace part", note the structure of the EdgeBar. If, for example, an assembly has been loaded into an ASM template, the template is considered the first level. Here, you can only replace the entire assembly!

8.6   For assemblies and drawings: Create BOM

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

Note:

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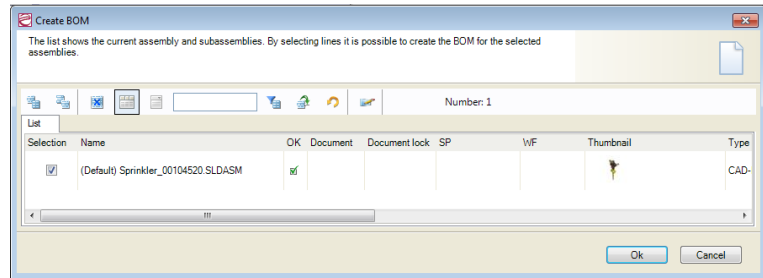- The CAD documents must be linked to a part master in PRO.FILE.

Function call from the PRO.FILE menu in Solid Edge:

"PRO.FILE" => "Create BOM"

Proceed as follows

1. Select the "**PRO.FILE**" menu in Solid Edge.

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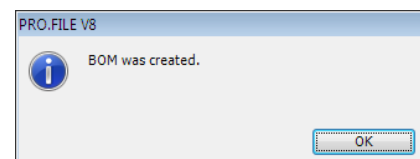
To view the bill of materials in PRO.FILE, select the function "Show" => "Bill of materials" as described in the chapter "Show: Information on a CAD document in PRO.FILE".

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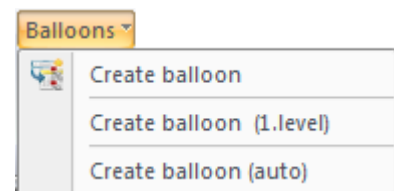
> **Note:**
>
> This function can be activated automatically for each saving procedure. Please see the configuration manual of the integration for further details.

8.7   For drawings: Create balloon

The function "**Create balloon**" allows bill of materials positions to be displayed in the drawing.

Balloons ▾

- Create balloon
- Create balloon (1.level)
- Create balloon (auto)

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.

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

➡ **Function call from the PRO.FILE menu in Solid Edge:**

"PRO.FILE" => "Balloons" => "Create balloon"

Proceed as follows

1. Load a drawing from PRO.FILE in Solid Edge and lock it.

2. Select the function "**Balloons**" => "**Create balloon …**" from the "**PRO.FILE**" menu. You can choose between three options:

    - "**Create balloon**" – for all selected components

    - "**Create balloon (1. Level)**" – only for the first level according to SE default

- "**Create balloon (auto)**" – one level for all parts.

3. Select the parts and placing points for the balloons on the drawing in Solid Edge. Depending on the selected function, this has to be repeated until all desired balloons are inserted.

⇒ The inserting of balloons is thus finished.

8.8   For drawings: Drawing legend

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

- The function can only be accessed, if the active Solid Edge document is a drawing saved in PRO.FILE.

- Furthermore, a drawing legend must already be configured via the menu "**Extras**" => "**Options**" => "**Drawing legend**".

The menu "**Drawing legend**" contains the following functions:

- Update drawing legend
- Delete single elements of the drawing legend with
  - Delete title block
  - Delete BOM
  - Delete list of changes
  - Delete balloon
- Copy drawing legend
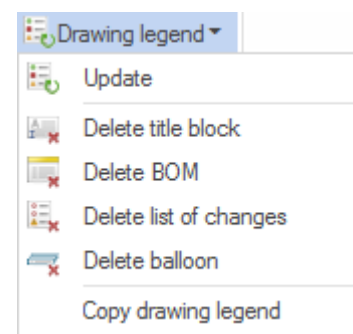
For these functions see the following sub-chapters.

8.8.1   Update drawing legend

	Function call from the PRO.FILE menu in Solid Edge:
	"PRO.FILE" => "Drawing legend" => "Update"

After selection of this function, 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.

8.8.2 Delete single elements of the drawing legend

The following elements of the drawing legend can be deleted separately:

* Title block
* Bill of materials

* List of changes
* All balloons

Function call from the PRO.FILE menu in Solid Edge:

"PRO.FILE" => "Drawing legend" => "Delete <element>"

Proceed as follows

1. Select the function "**Drawing legend**" => "**Delete** <**element**>" (depending on the element you want to delete) from the "PRO.FILE" menu.

2. A warning message will ask for your confirmation of the deletion action. If you are sure you want to delete the element, confirm with <**Yes**>.

⇒ Depending on the selected function, the title block, the bill of materials, the list of changes or the balloons are removed from your drawing legend.

Attention: Undo not possible – Risk of data loss

The functions "Delete title block", "Delete BOM", "Delete list of changes" and "Delete balloon" cannot be undone. Once the element is deleted and can only be reinserted via the configuration of the drawing legend (or via the function "Copy drawing legend".

8.8.3 Copy drawing legend

Via this function, the drawing legend of an existing drawing can be loaded into the current drawing in Solid Edge.

Function call from the PRO.FILE menu in Solid Edge:

"PRO.FILE" => "Drawing legend" => "Copy drawing legend"

Proceed as follows

When the function is selected, you can select a drawing with and existing in the Explorer. The program then searches a format in this drawing that matches the active drawing, then the drawing legend is copied into the active drawing.

9   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 Solid Edge:
	"PRO.FILE" => "Extras" => "Activate work folder"

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.

Further information on the Workcenter can be found in the following chapters:

- [Workcenter functions](#)
- [Activate work folder](#)

9.1   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](#)".
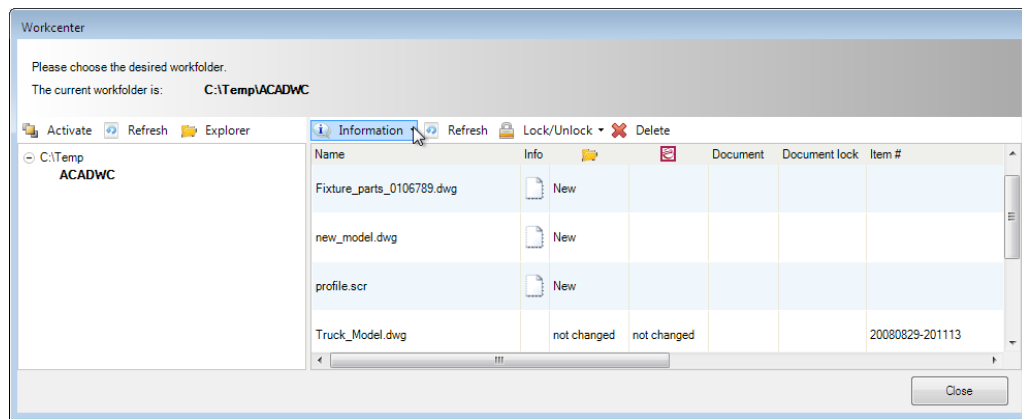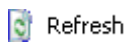
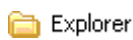**The functions for the directory structure:**

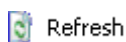| Icon	Description
Activate	The selected folder will be used as the new working directory. The current working directory is marked in bold.
Refresh	The view of the directory structure is updated.
Explorer	The selected folder is opened in the Windows Explorer. This gives you the possibility to use the usual Windows functions in order to delete, create or copy a working directory.

The functions for the work folder:

| Icon	Description
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 Document usage 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 facilitate the finding of objects in large folders.
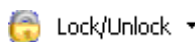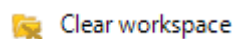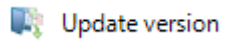
![] Update version	Selected files can be replace by a newer PRO.FILE version (of the same file name). If version conflicts arise, the PRO.FILE dialog for the version selection is displayed.
Open with double click in the CAD system	Double-clicking a file in the Workcenter opens the file in in the CAD system (if it is not already opened).

9.1.1   Activate work folder

Function call from the PRO.FILE menu in Solid Edge:

"PRO.FILE" => "Extras" => "Activate work folder

It is possible to work with several parallel working directories when working with the Solid Edge integration (as long as this is configured using the PRO.FILE Management Console). The additional working directories must be first created in Windows explorer, or another file manager within the basic directory. Now that several working directories have been defined, the selection of which working directory is to be used is made using the menu points "PRO.FILE" => "Extras" => "Activate work folder".

Using this menu entry, you can also list all objects that are present in the selected working directory. If you select this menu entry, you will be presented with an overview of all of your defined working directories.

By selecting the appropriate directory, you will receive a list of the available elements. You can use this to create an overview of where which object is stored, or to load specific objects in Solid Edge from the local working directory.

10   Extras: Options for the integration

For each CAD workstation you can make user-specific local settings. This is made in the integration via the menu entry "**Extras**" => "**Options**".

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

Settings can be made for the following three areas:

- [Options: Document list](#)
- [Options: Performance](#)
- [Options: Drawing legend](#)

For detailed information see the following sub-chapters.

	Note:
	For further information on the configuration of the integration, please see the configuration manual of the integration.

10.1   Options: Document list

In the menu section "Document list" you can configure the user-specific display of the document list. Different columns of the document list can be displayed or hidden.

The settings only apply for the current user and can thus be adjusted individually.

	Attention: only possible in case of correct parameter settings
	The displayed settings for the document list can only be adjusted if the validity range of the parameter "Settings in the info list" – DOCLIST_SHOW under "**Configuration**" => "**Parameter**" => "**CAD**" => "**Integration**" => "**Solid Edge**" => "**Others**" in the PRO.FILE Management Console is set to "**Individual user**".
	Only then it is possible that every user makes individual settings at his/her workstation.

Function call from the PRO.FILE menu in Solid Edge:

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

For the configuration of the display of the document list you can activate the columns to be displayed.

Seven entries are available, which you can activate via the checkboxes.
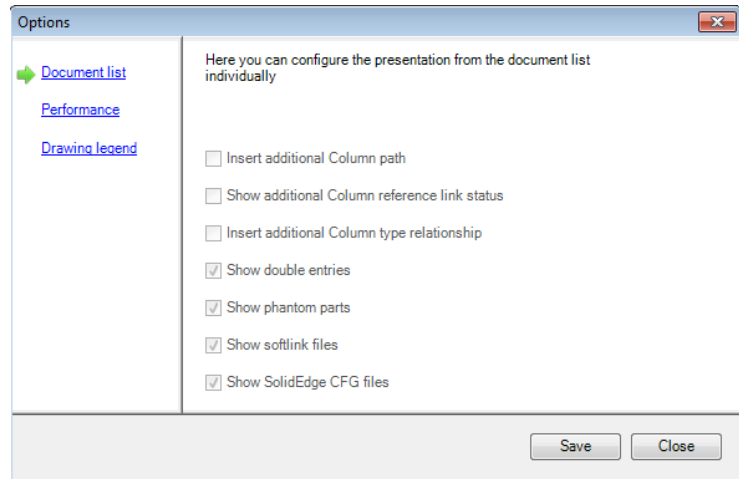
| Insert additional column "Path" | If this option is activated, the document list also contains a column with the path at which the CAD document is stored locally. |

| Show additional column reference link status | Here you can display the link status. Possible displays: |

=>   Normal link

*-=>   Softlink (cycle)

| End of softlink (cycle)

Insert additional column type relationship

This column displays, among others, the program (Solid Edge or RevisionManager) the references are loaded with:

CFG - File	Configuration file of the assembly
ASM. – Instance	Instance of an assembly family
OLE - OBJEKT	OLE – Document link (e.g. Office)
Rev.Manager Solid Edge	Link found with Revision Manager
Solid Edge	Link found with Solid Edge

Show double entries

If this checkbox is **deactivated** parts that are built in the assembly several times (e.g. screws) are only displayed once.

Show phantom parts	Phantom parts are "usually" not displayed in the document list, unless this checkbox is activated.
Show softlink files	Only if this checkbox is activated, softlink files (cycle) are displayed in the document list.
Show Solid Edge CFG files	Configuration files of the assemblies are only displayed in the document list if this checkbox is activated.

10.2 Options: Performance

Via the options in the section "**Performance**" you can make performance-relevant settings, e.g. switch of undesired messages and dialogs.

The settings only apply for the current user and can thus be adjusted individually.

Function call from the PRO.FILE menu in Solid Edge:

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

You can make the following settings:
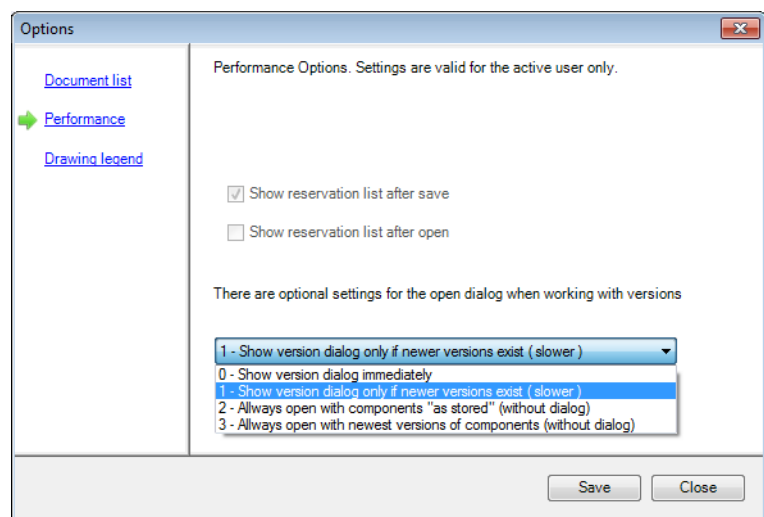
- Show reservation list

    Via these checkboxes you can specify whether the reservation list is to be built and displayed after saving or after opening.

There are optional settings for the open dialog when working with versions:

- **0 – Show version dialogue immediately**

  Here you can decide individually whether versions are to be loaded. This dialogue will always appear if no new versions are available.

- **1 – Show version dialogue only if newer versions exist (Slower)**

  If this option is selected, the version dialogue will only appear if a new version does exist.

- **2 – Always open with components "as stored" (without dialogue)**

  No dialogue is displayed. The parts are loaded as they have been saved.

- **3 – Always open with newest version of components (without dialogue)**

  No dialogue is displayed. The newest version is always loaded.

10.3   Options: Drawing legend

Function call from the PRO.FILE menu in Solid Edge:

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

For the configuration of the drawing legend, different settings in the PRO.FILE Management Console and in the menu displayed on the right have to be made.
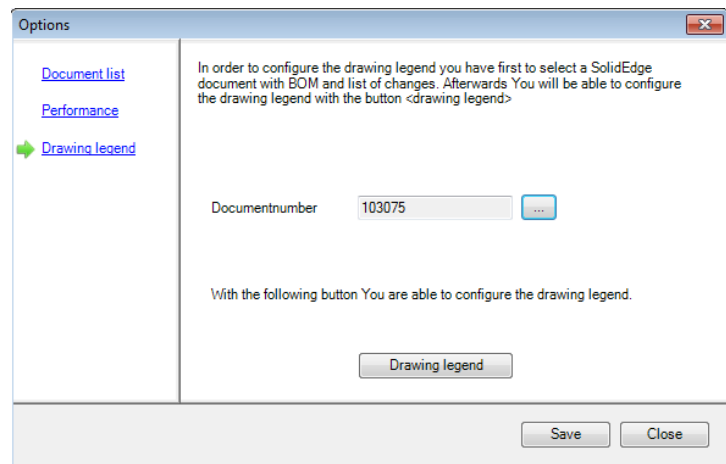
Due to the necessity to access the PRO.FILE Management Console, the further proceeding for this configuration section is described in the manual "**Configuration of the Integration PRO.FILE – Solid Edge**".

11 Tips and Workarounds

This chapter describes tips and workarounds for known problems.

11.1 PRO.FILE does not recognize my documents any more

You have loaded documents from PRO.FILE and saved them locally. Now you would like to check in those documents again. But PRO.FILE does not recognize them and therefore asks whether you want to created new parts for all documents.

This may be due to the fact that, upon check-out of documents from PRO.FILE, xml files are created in the sub-directory _profinfo_. These connection files are required to save the status of the Solid Edge documents.

If Solid Edge documents are moved, the integration cannot find the connection files and PRO.FILE treats the documents like new ones.

No Solid Edge documents must be moved or renamed.

11.2 BOM cannot be created

You have inserted an assembly into another assembly and used identical part numbers. The inserted assembly is not to be included in the BOM, but the "top" assembly should be included. The message "**not allowed**" is displayed.

You can solve this problem by using **phantom parts**.

Save the assembly as a phantom in PRO.FILE and insert this phantom assembly into your new assembly afterwards.

Deactivate the BOM events for this position and create the BOM.

11.3 The directory for variable name is not renamed

If no ring connections exist, these documents are always to be edited in the context of the top document.

If in Solid Edge you open a document with variables not within the assembly context, the variables are not adapted or updated by Solid Edge.

If you open the document within the context of the assembly, the paths of the variables are adapted by Solid Edge.

Please open these documents in Solid Edge always within the context of the assembly.

11.4   Problems when copying the drawing legend

For each format (background sheet) a sheet has to be present in the selected template (source). The drawing legend is loaded on this sheet and copied into the active drawing.

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