Functions of the integration PRO.FILE Inventor

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Registered Trademarks:

Responsible for Content:

Table of contents

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 – Inventor

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

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 Inventor to ERP systems.

1.1   The contents of this manual

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

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 Inventor. The following topics will be addressed:

- Operation of PRO.FILE from within Inventor

- Data representation of structures/references in PRO.FILE

- Integration of PRO.FILE into the Inventor environment

> **Note: Manual "CAD design supported by PRO.FILE"**
>
> When using the integration PRO.FILE – Inventor, please also note the manual "CAD design supported by PRO.FILE", which describes the basic procedures and related issues from the designer's point of view.

2   First steps with the PRO.FILE integration

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

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

- [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](#)

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

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

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".

3. The "**root folder**" can be selected - or created via the button <**Make new folder**>.

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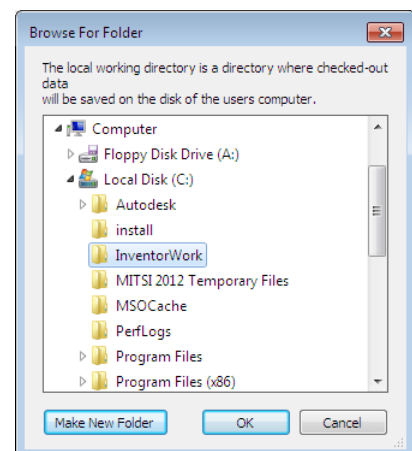
5. In the second step, "**work folders**" are now created In this **root folder**, which will then be used by the integration.
Consequently, you are now prompted to specify a work folder within the **root folder**:

6. Please specify a name for the work folder.

7. Confirm your entry with <OK>.

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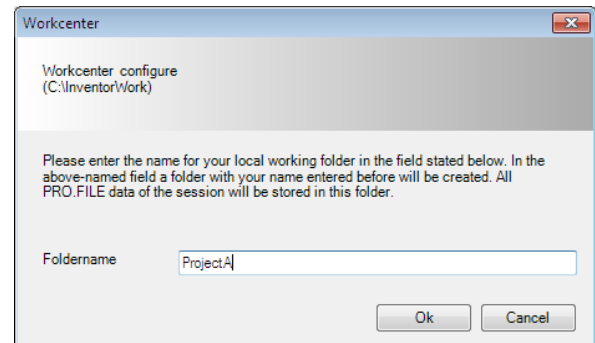
This work folder and other local work folder can be created and managed by the user via the Workcenter. The Workcenter can be accessed via the PRO.FILE menu in Inventor under "**Extra**" => "**Workcenter**".

Detailed information can be found in the chapter "<u>Extra: Additional functions of the integration</u>".

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

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

1.   Start "**Inventor**".

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 Inventor file:

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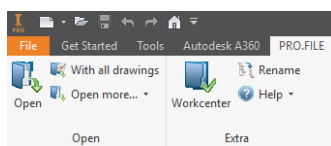
When a part is loaded, all basic functions of the integration are offered in the PRO.FILE menu:
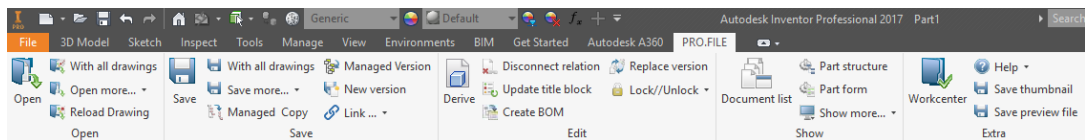
When a drawing is loaded, the PRO.FILE menu is enhanced by the following functions:

•   Save NDF

•   Insert view

•   Insert BOM
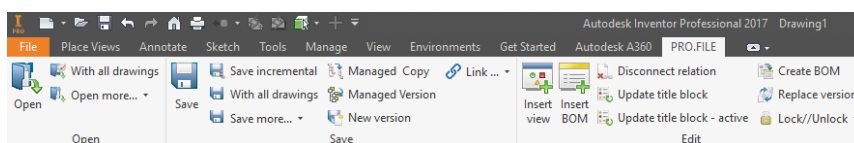
When an assembly is loaded, the PRO.FILE menu contains the following functions:

•   Insert component

•   Replace component

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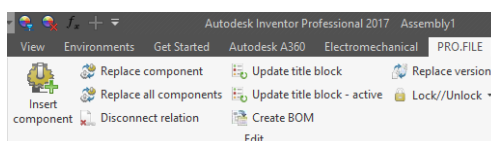
Whenever one of these menu functions is used, PRO.FILE is activated. According to the function used, the required windows in PRO.FILE are opened automatically.

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

2.3 How to log in to PRO.FILE?

If you access a PRO.FILE function for the first time within an Inventor 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.

Note: No login required in case of "Autologin"

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

2.4 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.

"Open"
- **Open**: This function opens PRO.FILE and prompts the user to select a CAD document for loading in Inventor.

- **With all drawings**: If an assembly is opened, it is possible to open all drawings available within the assembly structure in Inventor via this function.

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

- **Open with versions browser**: With the version browser the user can decide, in which version an assembly and its components are to be opened.

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

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

- **Supply document**: The selected CAD document is loaded into the Workcenter folder.

- **Reload drawing**: If a PRO.FILE object is loaded in Inventor, the function "**Reload drawing**" can be used to load the corresponding drawing from PRO.FILE in Inventor.

"Save"

- **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.

- **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 automatic". In analogy to "Save automatic", no further user input is required during the save process.

- **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.

- **Managed Copy automatic**: The function "Managed Copy automatic" unites the functions "Managed Copy" and "Save automatic". In analogy to "Save automatic", no further user input is required for the creation of the part and document descriptions when the function "Managed Copy automatic" is used.

- **Save automatic**: Document and part descriptions for all components are created in PRO.FILE automatically – the data only has to be entered for the first record. File names and properties can be configured to be transferred automatically into specific PRO.FILE fields.

- **Save all automatic**: This function enhances the function "Save automatic": All CAD documents loaded in the CAD session are saved in PRO.FILE. No data input is required, all data records are created automatically.

- **New version**: Saves the currently active CAD document as a new version in PRO.FILE. If this function is applied to a part that is used in an assembly, the references of the assembly still point to the old version of this part, according to the new version management of PRO.FILE.

- **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.

- **Managed Version**: This function can be used for the creation of versions within assembly structures.

"Edit":

- **Disconnect relation**: This function deletes the database connection of a CAD document to PRO.FILE. The CAD data are thus treated as purely locally stored CAD documents and no longer have a PRO.FILE connection.

- **Update title block**: Via this function the iProperties of the active CAD document and its structure (if exists) are updated. If these iProperties are mapped to text fields of the title block or the modification list in a drawing, these are updated as well. A prerequisite for this function to work is that these lists and fields are configured for the used drawing.

- **Create BOM**: This function creates a bill of materials in PRO.FILE based on the active CAD structure. If a bill of materials exists already in PRO.FILE, it is updated.

- **Replace version**: This function allows to replace a version of a CAD part in all assemblies, in which it is used, with a newer version.

- **Lock**: CAD documents loaded from PRO.FILE in Inventor are not automatically locked for other users. If you want to edit a CAD document, you have to use the command "Lock" before making the changes. This prevents the document from being changed by a different user in the meantime.

- **Unlock**: With this function CAD documents, which were locked for editing in Inventor, can be unlocked, so that other users can also edit the document.

- **Insert view (for drawings)**: With this function, Inventor components from PRO.FILE are inserted into the active CAD drawing.

- **Insert BOM (for drawings)**: If you want to display the bill of materials from PRO.FILE on the CAD drawing, you can use this function to insert and place it. If a bill of materials is already placed on the drawing, it is updated.

- **Insert component (for assemblies)**: This function inserts a component from PRO.FILE into the active assembly.

- **Replace component (for assemblies)**: This function replaces a component selected in the active Inventor assembly by a component from PRO.FILE.

- **Replace all components (for assemblies)**: This function replaces all copies of a component selected in the active Inventor assembly.

"Show":

- **Document list**: 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.

- **Part form**: Displays the description form for the active part in PRO.FILE.

- **Part structure**: Switches to PRO.FILE and displays the structure overview of the current part.

- **Show more**: Each of these functions switches to PRO.FILE and displays either the part usage, the BOM, the document structure, the document form, the document usage or all document versions for the active CAD- part.

"Extra":

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

- **Help**: Opens the PRO.FILE online help.

- **Save thumbnail**: If a preview thumbnail is used in the PRO.FILE document list, you can use this function to uptdate the thumbnail picture.

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

Additional functions

The following functions are configured via parameters in the PRO.FILE Management Console:

- **Management of OLE references**:

  If the OLE management is activated, OLE references are read out and saved to PRO.FILE when a CAD document is saved.

- **Management of content center**: If the norm part management is activated, all parts generated from the content center (norm parts) are only created once in PRO.FILE.

3 Functions for opening CAD documents from PRO.FILE in Inventor

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

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

Open from PRO.FILE:

- [Open CAD documents from PRO.FILE for editing](#)

Open from within the integration:

- [Open with released and newest versions of linked CAD documents](#)
- [Open with versions browser](#)
- [Open with all drawings](#)
- [Supply document](#)
- [Attention: Opening of locally existing files](#)
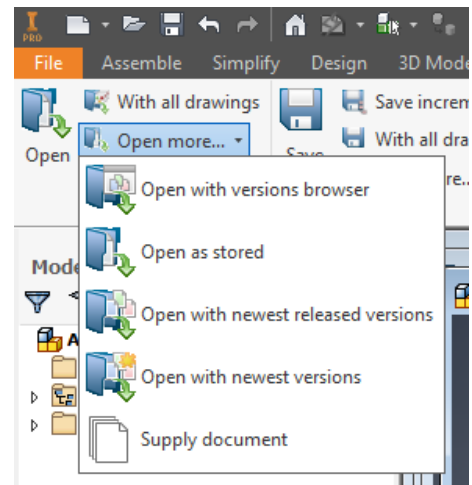
If a CAD object is loaded from PRO.FILE in Inventor, the following function is available:

- [Reload drawing](#)

!	**Attention:**
	The data loaded from PRO.FILE in Inventor are **not automatically locked** when opened in the CAD system. 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?](#)".

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

3.1   Open CAD documents from PRO.FILE for editing

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

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

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

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

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

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

3.1.1   Open via drag & drop

You can open CAD objects from PRO.FILE via drag & drop and use them in your assemblies. To do so, drag the desired CAD component from PRO.FIL into the Inventor 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 Inventor GUI and drop it there.
⇒ The file is copied into the Workcenter folder and is opened.

Note:

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

3.2   Open: Load CAD files from PRO.FILE

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

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

Step 1:   Use the PRO.FILE function "**Open**"

Function call from the PRO.FILE menu in Inventor:

"PRO.FILE" => "Open" => "Open"

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

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

⇒   "**Open**" loads documents and its components as it is defined in the parameter "Version load options dialog" in the PRO.FILE Management Console.
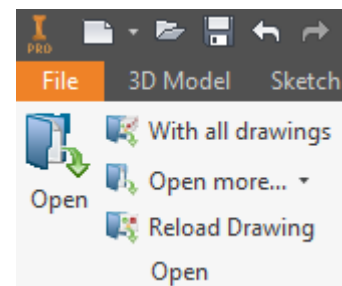
⇒   The Checkout wizard for the selection of documents is displayed.

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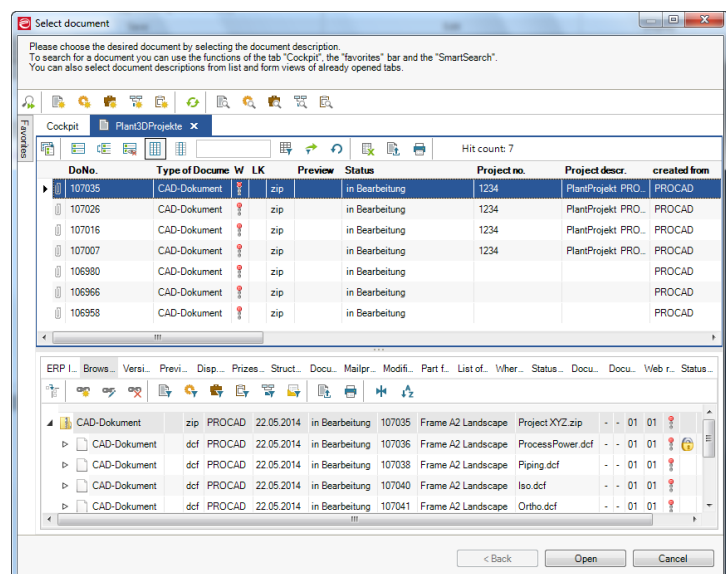
4. If the desired document is displayed in a list view, you can **select** it. (If the desired document is displayed in a form view, it is already selected.

5. Click <**Open**>.

⇒ The Checkout wizard closes and the dialog screen for the loading type is displayed.

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

Step 3   You want to edit? Then you must lock the document(s)

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

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

⇒ If you click on "Lock" the dialog for locking CAD documents is displayed.

(Information on the functions and status indicators can be found in the chapter "The document list")
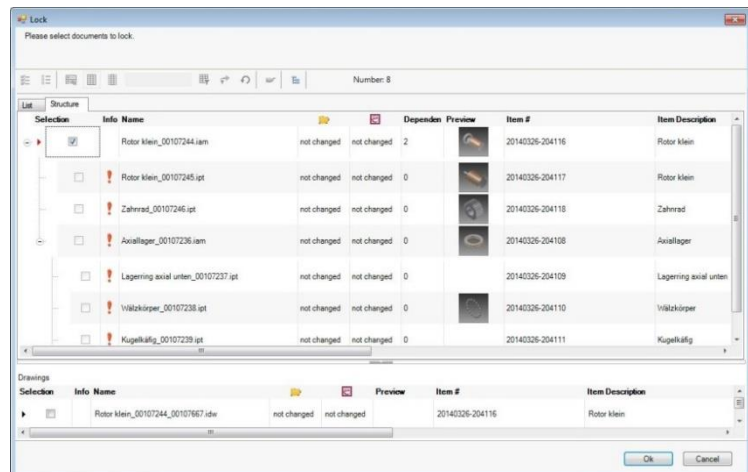
6. Select all documents you want to lock with the corresponding check boxes.

7. Confirm your selection with <**OK**>.
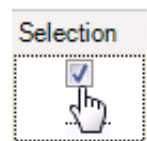
⇒ The selected documents and its components are now opened in Inventor. The process of opening a document is now finished.

Detailed information on the locking of documents can be found in the chapter "Lock/Unlock: Who can change when?".

Note: Why can I not lock a document?

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

This may have two **reasons**:

- The document is already locked by a different user. You can see who the locking user is by selecting document in PRO.FILE and looking at the dependent tab "Status information".

- The document is in a workflow status, in which you are not allowed to edit the document. This is typically the case for "released" documents.

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

3.2.1 Working with the Checkout wizard to search for CAD documents

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

The **aim** of this procedure is:

- For PRO.FILE to know which document is to be opened, the desired document description must

  - either be selected in a list view or a structure browser

  - or be displayed in a form view

- Then the button <**Open**> at the right bottom of the Checkout wizard has to clicked.

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

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

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

  If the desired document is displayed on a tab, select it and click <**Open**>.
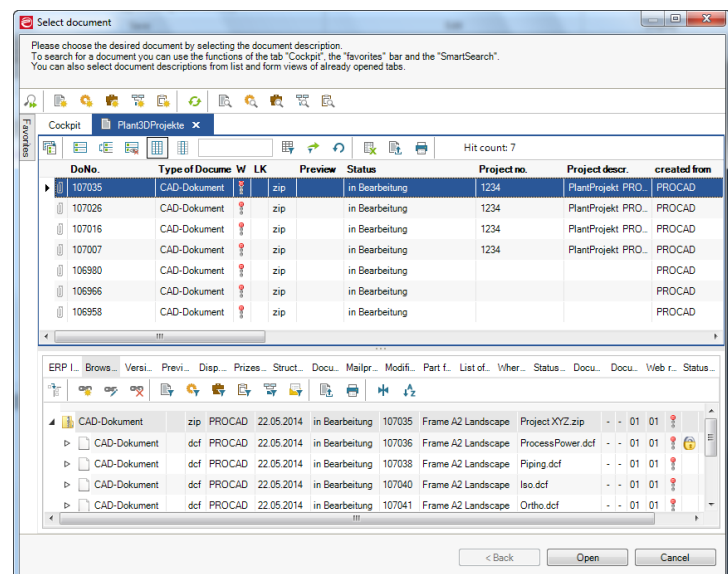
- If no search had
  been made
  previously, or if the
  desired document is
  not displayed on one
  of the existing tabs,
  you can now search
  for it.

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

!	**Attention: Double-click in the Checkout wizard**
	Documents are selected and then opened via the <**Open**> button. A document **cannot be loaded via a double click**!
	Because a double click means: Open document for viewing!
	The checkout will remain active in the background waiting for your selection. But only if the checkout wizard is closed, the document can be used for editing in PRO.FILE.

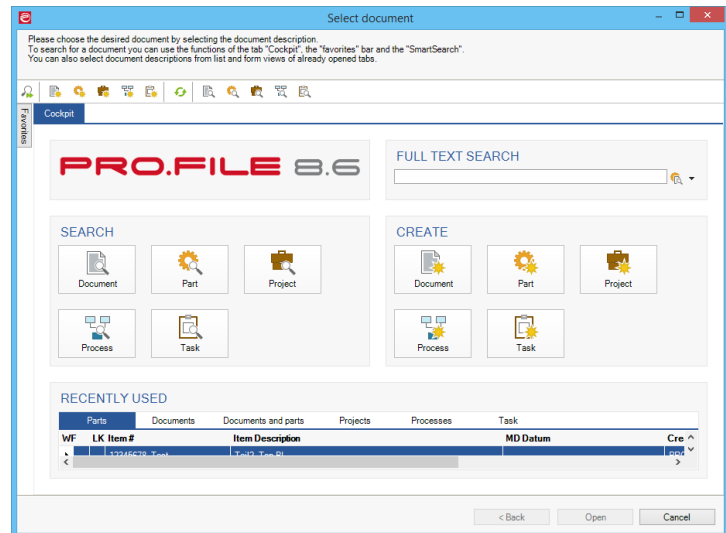**Searching for data records in the Checkout Wizard**

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

- Searching via the tab "Cockpit"

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

  You can always go back to the tab "Cockpit".

- Search via the functions of the favorites bar

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

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

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

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

- Search via the icon bar

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

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

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

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

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

3.3   Open with released and newest versions of linked CAD documents

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

- "Open as stored"
- Open with "Released versions"
- Open with "Newest versions"
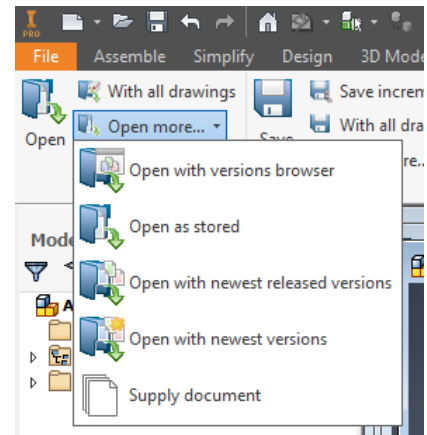
Note:

The two version options do not refer to the document selected for opening in PRO.FILE. They only refer to the objects linked to the document to be opened from PRO.FILE. As a user, you can decide with which version status you want to open the components linked to the PRO.FILE – CAD document.

This means:

- "**Open as stored**"

  The selected document is opened from PRO.FILE as it was saved the last time. Linked CAD documents are loaded with the version status, as they were saved the last time via the PRO.FILE integration.

- "**Open with newest 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 newest 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 newest versions**"

  If the CAD document contains links to other CAD documents in PRO.FILE, the newest versions of these linked CAD documents are loaded.

  When the function "**Open with newest 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.

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

Note:

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

3.4 Open with versions browser

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

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

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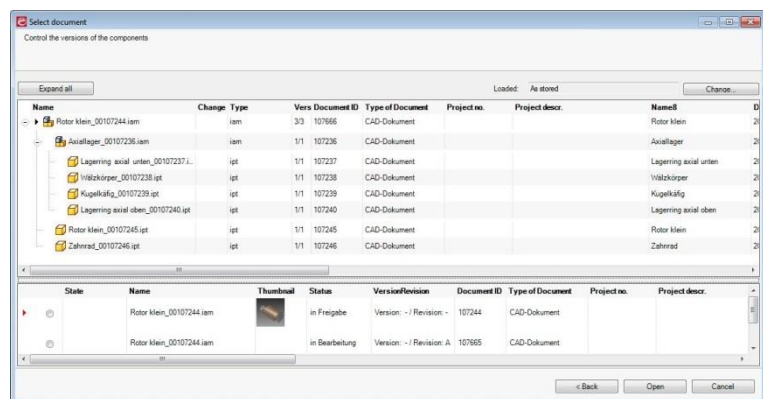
The version browser is divided into two areas:

The document structure (top)

- In the upper structure windows the selected CAD document is displayed with all attached components.

- Via the button <Expand all> you can you can display the entire structure of the part to be opened.

- The field "Loaded" shows the current opening type of the CAD elements displayed in the structure window – without manual version selection. The opening type affects the display of these elements:

  Via the button <Change…> you can choose between the four options for opening:

  - Open "as stored"

  - Open "latest version" of the components

  - Open "latest released version" of the components.

  - Open "latest release version or latest version" of the components, depending on their availability.

The version window (bottom):

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

Function call from the PRO.FILE menu in Inventor:
"PRO.FILE" => "Open" => "Versions browser"

To open a document with the version browser proceed as follows:

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

2. Select the function "**Open with versions browser**".

⇒ The Checkout wizard is displayed.

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

⇒ The Checkout wizard closes.

⇒ Detailed information on the Checkout wizard can be found in the following chapter "Working with the Checkout wizard to search for CAD documents".
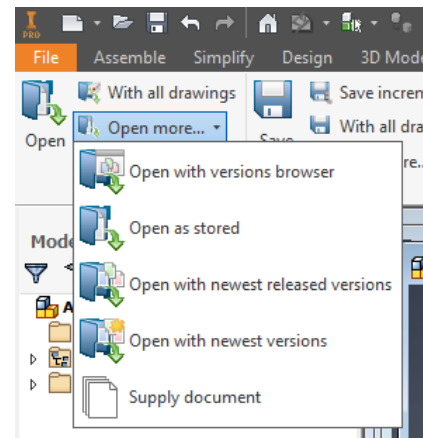
⇒ The screen "Select document" is displayed.

4. Select the component, for which you want to make a version selection, in the document structure.

⇒ The lower version window now displays all corresponding versions.

5. By toggling the radio button in the first column of the version window you can activate the desired version of a CAD element:

State	Name	Thumbnail	Status	VersionRevision	Document ID	Type of Document
⊙	📁 Hydr_Zylinder_101663.ipt		in Bearbeitung	Version: - / Revision: -	101663	CAD-Dokument

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

⇒ The 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.
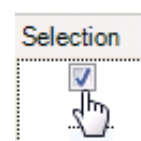
⇒ Detailed information on the locking of documents can be found in the following chapter "Lock/Unlock: Who can change when?".

8. Confirm your selections with <OK>.

⇒ The selected CAD components are opened in Inventor. 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 Inventor assemblies
	Icon of Inventor parts
	Indicates a softlink.
	Versions reference each other causing a version cycle.

3.5 Open with all drawings

Via the function "Open with all drawings" all drawings belonging to a CAD document are offered directly for opening from PRO.FILE when the CAD document is opened from PRO.FILE.

	Note: Drawings for components in PRO.FILE
	A prerequisite for this function to work is, of course, that the drawings for the parts or the assembly are already saved in PRO.FILE.

	Function call from the PRO.FILE menu in Inventor:
	"PRO.FILE" =>"Open" => "With all drawings"

All drawings linked to the CAD document to be opened from PRO.FILE are then displayed in a selection window.

Tick the checkboxes for the desired drawings you want to open along with the CAD document.

The proceeding for this function corresponds to the proceeding for the function "Drawing reload".

Detailed information on the proceeding for this function can therefore be found in the chapter "Reload drawing".

3.6   Reload drawing

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

If several drawings are available for a CAD object, a dialog to select the drawing is displayed.
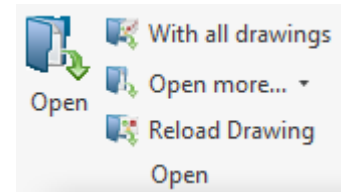
Note:

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

Function call from the PRO.FILE menu in Inventor:

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

Proceed as follows

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

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

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

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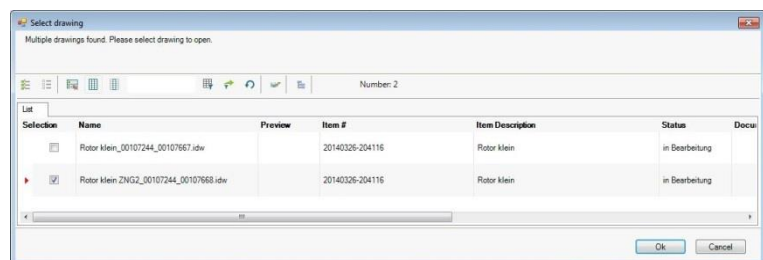
4. Select all drawings you want to open in Inventor with the checkboxes in the first column.

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

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

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

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

7. Select all documents you want to lock with the checkboxes in the first column.

8. Confirm your selection with <OK>.

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

3.7   Supply document

'The Inventor integration supports the usage of external components, i.e the usage of components from other CAD systems. For this purpose, these components are copied into the Workcenter folder.

Via the function "Supply document", the selected CAD file, the sub-structure and the corresponding XML files are copied into the Workcenter folder.

Note:

Assemblies used in Inventor in the way described below, cannot be administrated in PRO.FILE, since PRO.FILE cannot read the file names in the structure. Administration of imported assemblies can be made via the function "Add additional file".

Function call from the PRO.FILE menu in Inventor:

"PRO.FILE" => "Open more…" => "Supply document"

1. Select the function "**Open more…**" => "**Supply document**".

⇒ The PRO.FILE Checkout wizard is displayed.

2. Select the desired document and confirm your selection by clicking on <**Open**>.

⇒ The file is copied into the Workcenter folder and can then be used in Inventor.

> **Note:**
>
> For external CAD components, the original file names are kept when the assembly is saved to PRO.FILE. This is done regardless of the setting of the parameter "Keep CAD file name during PRO.FILE save" in the PRO.FILE Management Console.

3.8   Attention: Opening of locally existing files

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

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

> **Attention: Risk of data loss**
>
> If locally changed files are overwritten with files from PRO.FILE, these local files and all changes to them are irretrievably lost!
>
> If versions are overwritten, the locally existing assembly now point to the newly loaded version which has replaced the original version! You should therefore be careful when deciding to overwrite a locally existing version

A drawing exists already in the work folder?

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

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

Different versions are also indicated.

You have three options of proceeding:

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

- **Do not overwrite**: Leave the checkbox unchecked.

- **Load data in a different Workcenter folder**: You can switch to a different working folder via the command "**PRO.FILE**" => "**Extra**" => "**Workcenter**" => "**Activate**", to avoid the overwriting of data. (See chapter "Workcenter functions").

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

4   Lock/Unlock: Who can change when?

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

- Only by using the function "**Lock**" you can make sure, that other users are not making changes to the same document at the same time.

- With the function "**Unlock**" the CAD document is made available again to other users for editing.

For detailed information see the following sub-chapters:

- [Start your changes: "Lock" the CAD document](#)

- [The "Unlocking" of CAD documents](#)
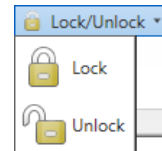
The locking of a CAD document makes sure that the CAD document is not modified by other users in the meantime:

- A **locked** CAD document can be opened by other users via the function "open". However, these other users cannot save back any changes to the locked CAD document to PRO.FILE.

- If the CAD document has been opened and **not locked**, other users may make changes to the document in the meantime and save those changes back to PRO.FILE. In this case, it will no longer be possible for you to save your own changes back to PRO.FILE.

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

Dynamic lock
dialog with
PRO.FILE 8.6

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>.
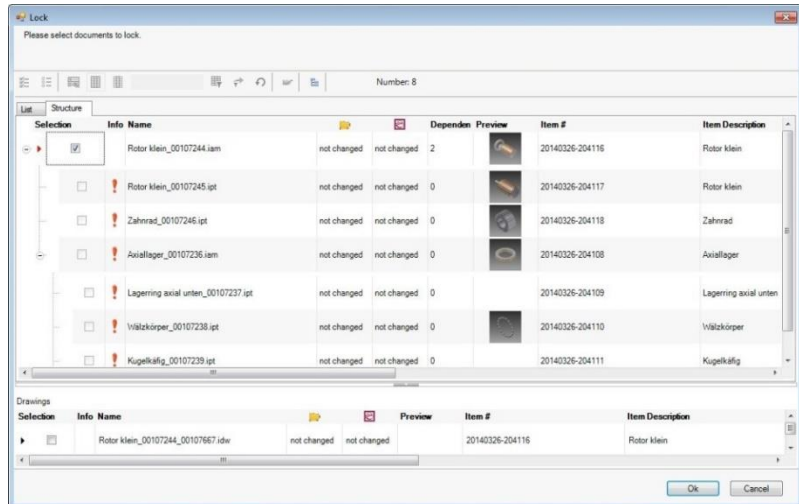
⇒ Displayed entries that are not selected for locking, will not be offered again for locking during the active CAD session.

4.1 Start your changes: "Lock" the CAD document

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

Function call:

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

To lock a CAD document manually proceed as follows:

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

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

3. Click on the function "**Lock/Unlock**" => "**Lock**" in the PRO.FILE menu area "**Work**".

⇒ 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]").
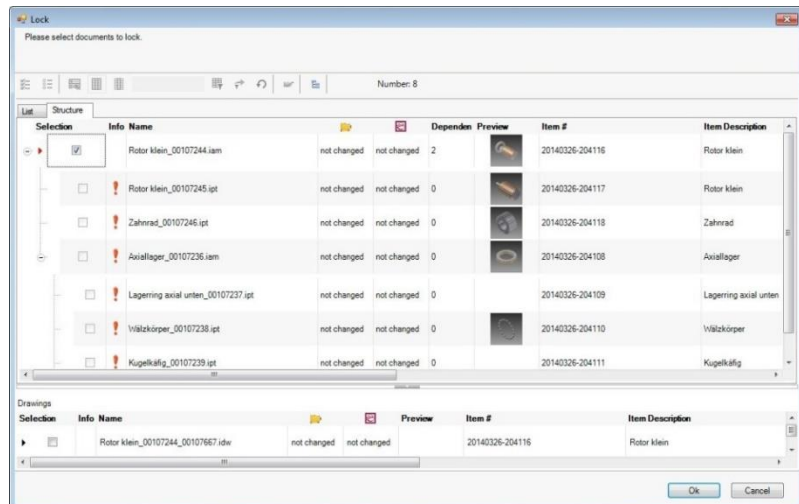
With the display of status information in this list PRO.FILE checks:

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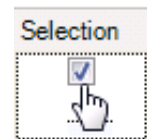
⇒ By the command "**Lock**", the access permission to edit the document is checked for the current user, and the document is locked for all other users in the database.

⇒ 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.

4.2   The "Unlocking" of CAD documents

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

Note:

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

Function call:

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

To unlock a document proceed as follows:

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

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

3. Click on the function "**Lock/Unlock**" => "**Unlock**" in the PRO.FILE menu area "**Work**".

⇒ 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").
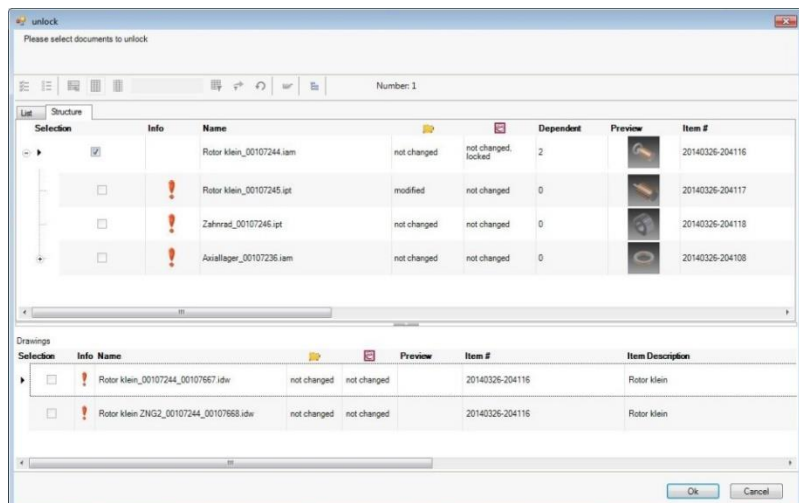

4. Select all document you wish to lock by setting the checkmark in the first column.

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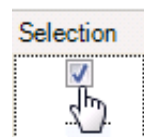
⇒ The lock flag for the selected CAD document is now removed.

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

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

"**Save**" with the options:

- Saving CAD objects for the first time

- Save: Saving changed CAD documents

- Save automatic

- Save all automatic

- New version

In the menu for drawings you find:

- For drawings: Save NDF (Neutral data format)
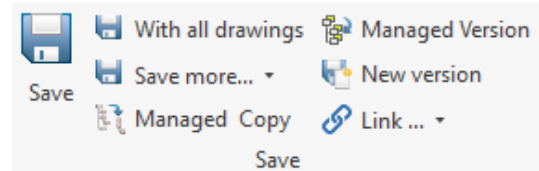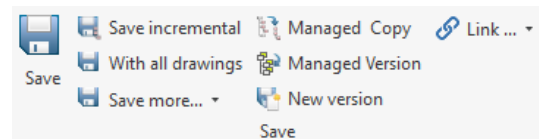
And in the menu for assemblies and parts you find:

- Managed Copy

- Managed Copy automatic

The basic procedure for saving differs depending on whether the CAD data are saved in PRO.FILE for the first time or whether documents from PRO.FILE are saved back after changes have been made to them in Inventor.

Therefore, the description is divided into two chapters:

- Saving CAD objects for the first time

- Save: Saving changed CAD documents

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

Please note the following chapter:

- Special aspects when saving Inventor objects

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

> **Note: Manual "CAD design supported by PRO.FILE"**
>
> Before using the integration PRO.FILE – Inventor please also note the manual "CAD design supported by PRO.FILE". This manual describes additional proceedings and related issues from the designer's point of view.

5.1 Special aspects when saving Inventor objects

The following sections describe special aspects to be noted when saving Inventor objects.

Saving of drawings and presentations

When drawings and presentations from Inventor are saved in PRO.FILE, it is common to use the same part master record for the document description of the drawing and the document description of the contained view.

- First, save the document description of the view.

- When saving the drawing or presentation, the part master record of the view is offered for usage with this drawing.

- If you select this part master record, both document descriptions are linked to one and the same part master record.

Saving of part families

When saving iParts or iAssemblies, these are recognized as such and marked with the corresponding geometry type in PRO.FILE.

Instances of iParts and iAssemblies should always be inserted in an assembly via PRO.FILE. If the instance does not yet exist in PRO.FILE, the part family has to be selected. The desired variant is then selected in the following table dialog.

5.2   Saving CAD objects for the first time

With the use of the function "**Save**", CAD objects are saved into PRO.FILE.

	Function call:
	"**PRO.FILE**" => "**Save**"=> "**Save**"

The process of saving takes place in several steps. Different dialogues appear depending on the results.

	Note: Saving order is a matter of the configuration
	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.

Proceed as follows

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

2. Click on the "**Save…**" button.

⇒ The Checkin wizard is displayed supporting you in the saving process.
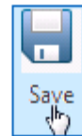
Saving of new objects in PRO.FILE takes place in three Steps:

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

These steps are described in the following sub-chapters.

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

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

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

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

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

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

Create new:　　　　Usage:

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

Proceeding:

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

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

Select in list:

> Select in list

Usage:

- The document to be saved is not to be linked to a new part master record but to an existing part master record.

- It is possible to link several documents to one and the same part master record.

Proceeding:

1. Click the option <Select in list> to select the desired part description.

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

2. If the part master record desired for assignment is not yet displayed in a list or form view, you can use the search functions of the home page, the icon bar or favorites and SmartSearch to start a **selection**.

3. If the part master record desired for assignment is displayed in a list view, you can now select it. (If the desired part master record is displayed in form view, it is already selected automatically).

4. Confirm your selection with <Next>.

Search:

> Search

Usage:

- The document to be saved is not to be linked to a new part master record but to an existing part master record.

- The desired part master record is not yet displayed in PRO.FILE and has to be searched for before assignment of the document.

Proceeding:

1. Click the option <Search> to select the desired part description.

2. Enter the search criteria into the displayed search form and click <Search>.

3. Select the desired part master record from the list of results.

4. Confirm your selection with <OK>.

New using template:

New using template

Usage:

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

- To make the creation of a new part master record easier, an existing part master record can be used as template with pre-filled fields, which only need to be adjusted.

- Example of usage: You are creating several records for the same screws, only with different lengths. If you do not want to enter the same data over and over again, you can use the function "New using template" and only have to adjust the filed "Screw length".

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

Proceeding:

1. Click the option <**New using template**> to select the desired part description.

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

2. If the part master record desired as template is not yet displayed in a list or form view, you can use the search functions of the home page, the icon bar or favorites and SmartSearch to start a selection.

3. If the part master record desired for assignment is displayed in a list view, you can now select it. (If the desired part master record is displayed in form view, it is already selected automatically).

4. Confirm the selection of the part description with <**Next**>.

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

5. Make the necessary adjustments to the pre-filled data.

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

Document without part:

Document without part

Usage:

- For special usage purposes it may be necessary to create a document description without the link to a part master record.

- You can therefore use this option to skip the creation or selection of the part master record and to proceed directly with the saving of the document description.

Proceeding:

1. Click the option <**Document without part**>.

⇒ The Checkin wizard for parts is skipped. The Checkin wizard for the document description is displayed.

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

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

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

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

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

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

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

Create new   New using template

- Create new

- New using template

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

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

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

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

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

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

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

Note: Usage of PRO.FILE projects

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

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

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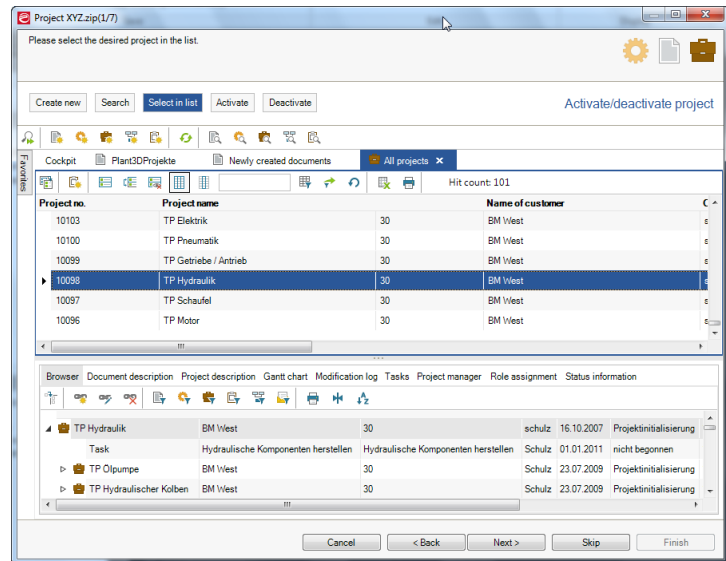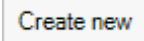

Here, too, the Checkin wizard offers different options that can be accessed via the operations bar:
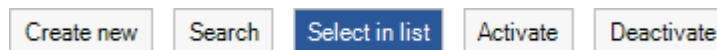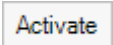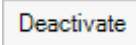

Attention: Project must be activated

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

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

- **Create new**:

  A 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**:

  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**:

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

- **Activate** Activate:

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

- **Deactivate** Deactivate:

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

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

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

1. Select an existing project or create a new one.

2. Select this project in the list view (project in form view are automatically selected).

3. You now **must** select "**Activate**". Only if the selected project has been activated, the assignment to the project is made after confirmation.

4. Confirm your proceeding with <**Finish**>.

⇒   The saving of the CAD data in PRO.FILE is now finished.

5.3   Save: Saving changed CAD documents

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

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

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

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

- "<u>Save: Saving changed CAD documents</u>" =(Change)
- "<u>Save automatic</u>" (see following chapter).
- "<u>New version</u>" (see following chapter).

This chapter describes the proceeding for saving changed CAD documents.

|	
➡	**Function call from the PRO.FILE menu in Inventor:**
	"PRO.FILE" => "Save" => "Save"

Proceed as follows:

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

Selection of the documents to be saved

⇒ The dialog for the selection of CAD documents to be saved is displayed.
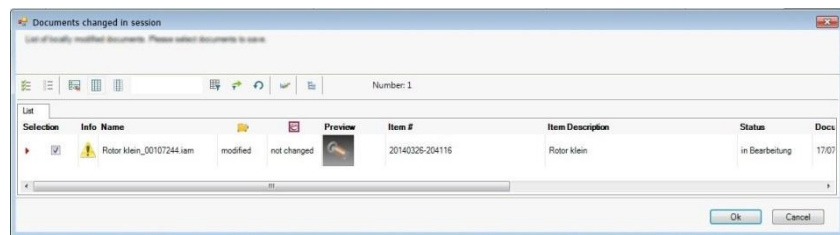
⇒ The dialog displays a list with all changed CAD documents from the current Inventor session. (Information on the functions and status information can be found in the chapter "The document list").

⇒ 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>.

Locally changed documents in the structure:

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

5. Select all locally changed components you want to save to PRO.FILE. To do so, activate the checkboxes for all desired CAD documents.
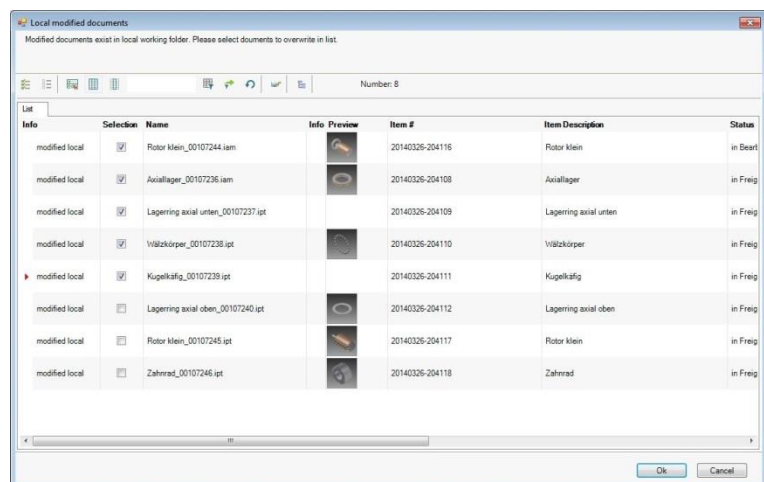
⇒ The changes are then saved. The CAD document previously saved in PRO.FILE is overwritten.

Optional: Enter modification comment

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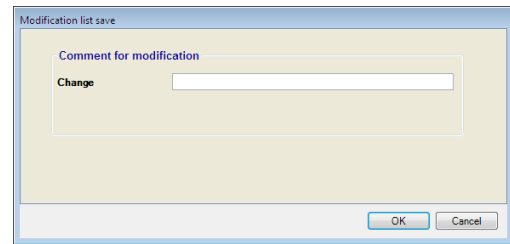
7.   Confirm your modification comment with <OK>.

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

Locking/unlocking the saved documents

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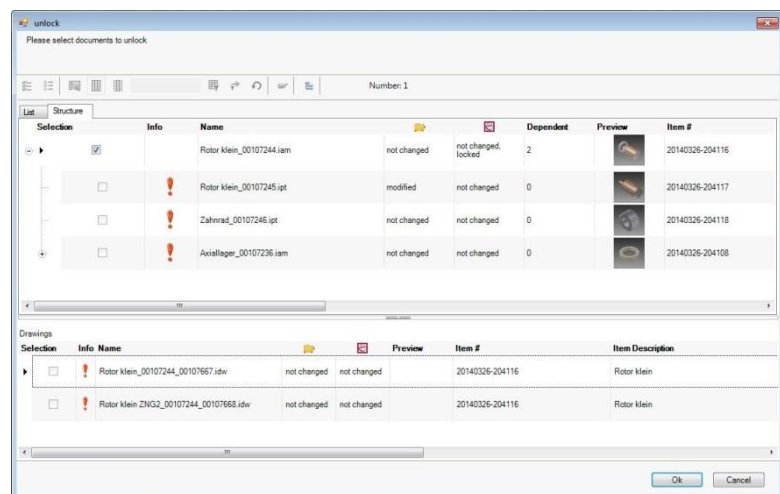
⇒   If documents from PRO.FILE had been locked for editing in the CAD system, this
     lock is not automatically removed after saving. The documents remain locked and
     cannot be changed by other users.

⇒   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.

5.4   Save automatic

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

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

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

* For the CAD documents to be saved **no** Checkin wizard is displayed. Document and part descriptions are saved automatically in PRO.FILE.

* Without further query means: The document and part descriptions are not filled in manually. The data record contains only the information that have been pre-configured in the saving form or that are automatically handed over from the CAD system to the saving form.

for documents that have been opened from PRO.FILE **for editing**:

* If documents have been opened from PRO.FILE **for editing**, the data in PRO.FILE is **without query** overwritten with the modified status of the data. For changed PRO.FILE documents "Save automatic" is identical to the proceeding for the saving of changed documents.

"Save automatic" for complete assemblies

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

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

	Function call from the PRO.FILE menu in Inventor:
	"PRO.FILE" => "Save more…" => "Save automatic"

Proceed as follows:

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

2. Select the function "**Save automatic**" from the menu area "**Save**".

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

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

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

- Finally, (depending on the configuration) the Checkin wizard for the project assignment in PRO.FILE.
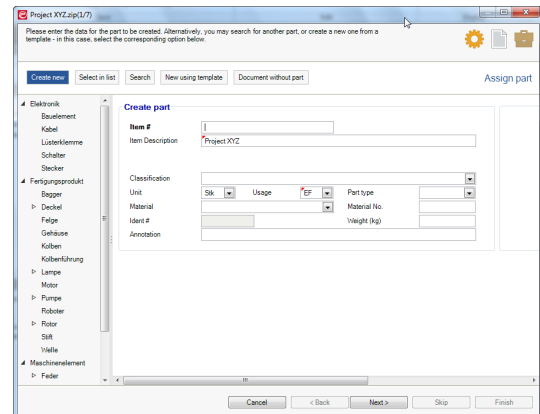
3. Go through all steps of the Checkin wizard for the first new CAD document. Detailed information on this can be found in the previous chapter "Saving CAD objects for the first time".

⇒ All further CAD documents are now saved automatically to PRO.FILE, without having to fill in the document and part descriptions. A project assignment is only made if a project is already activated in PRO.FILE.

4. For modified components already known in PRO.FILE:
   If the assembly also contains components that are already saved in PRO.FILE and that have been changed in the session or locally, these components are also offered for saving.
   Select the components to be saved and confirm your selection.
   Detailed information on the saving of changes to PRO.FILE can be found in the previous chapter "Save: Saving changed CAD documents".

Note:

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

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

⇒ The process "**Save automatic**" is now finished.

Attention: "Required fields" and <Save automatic>

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

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

> ⚠️ **Attention: ERP interface and "Save automatic"**
>
> When using the function "Save automatic", it may happen that fields, that are required by your ERP interface, are not filled! This may lead to problems during the forwarding of documents/parts to your ERP system. Please check the fields in the different forms for completeness.
>
> It is possible to have specific fields filled automatically by the system. With this you can make sure that important fields are automatically provided the required information. For detailed information see the configuration manual for the Integration PRO.FILE – Inventor.

5.5 Save all automatic

The menu function "**Save all automatic**" is an enhancement of the function "**Save automatic**".

"**Save all automatic**" allows the automatic creation of documents and parts in PRO.FILE without any interaction.

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

- For the CAD documents to be saved **no** Checkin wizard is displayed. Document and part descriptions are saved automatically in PRO.FILE.

- Without further query means: The document and part descriptions are not filled in manually. The data record contains only the information that have been pre-configured in the saving form or that are automatically handed over from the CAD system to the saving form.

For documents that have been opened from PRO.FILE **for editing**:

- If documents have been opened from PRO.FILE for editing, the data in PRO.FILE is **without query** overwritten with the modified status of the data. For changed PRO.FILE documents "Save automatic" is identical to the proceeding for the saving of changed documents..

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

"Save all automatic" for complete assemblies

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

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

	Function call from the PRO.FILE menu in Inventor:
	"PRO.FILE" => "Save more…" => "Save all automatic"

Proceed as follows:

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

2. Select the function "**Save all automatic**" from the menu area "**Save more…**".

⇒ An overview of all documents loaded in the CAD session to be saved us displayed. This overview is only for information purposes.

3. Confirm this list of all documents to be now saved automatically.

⇒ 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.

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

⇒ The process "**Save all automatic**" is now finished.

	Attention: "Required fields" and <Save all automatic>
	When CAD documents (sub-assemblies, parts, drawings) are saved all automatically, no values are entered manually in the Checkin wizard. The fields remain empty.
	This also applies to fields that are configured as required fields. As a consequence, all elements saved with "Save automatic" have to be classified at a later point in time, especially if these fields are required by other systems (ERP interface.

	Attention: ERP interface and "Save all automatic"
	When using the function "Save automatic", it may happen that fields, that are required by your ERP interface, are not filled! This may lead to problems during the forwarding of documents/parts to your ERP system. Please check the fields in the different forms for completeness.
	It is possible to have specific fields filled automatically by the system. With this you can make sure that important fields are automatically provided the required information. For detailed information see the configuration manual for the Integration PRO.FILE – Inventor.

5.6   New version

With the PRO.FILE Inventor integration it is possible to create new versions when saving CAD documents.

If the function "Save as new version" is used, PRO.FILE creates a new version of the CAD document and increases the version/revision counter for this document accordingly.

- Only the document active in the Inventor session is versioned.

- The old version remains in PRO.FILE.

- The new version is saved with a new document ID in PRO.FILE and displayed in Inventor.

- Within an assembly the reference is switched to the new version.

	Function call from the PRO.FILE menu in Inventor:
	"PRO.FILE" => "Save" => "New version"

Proceed as follows:

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

2. Select the function "**New version**" from the area "**Save**".

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 new version of the active CAD document is now saved in PRO.FILE.

⇒ A message will inform you of the successful creation of the version.

⇒ The new version is displayed in Inventor.
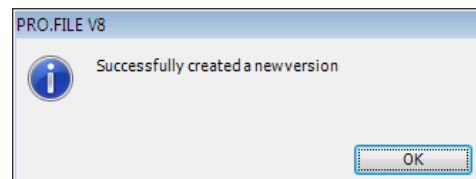
	Attention: New version is not locked
	The new version that has just been saved in PRO.FILE is not locked. To lock the document please use the function "Lock".

⇒ 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**" in the integration.

5.7   Save incremental

This function oft he integration PRO.FILE Inventor is intended for the incremental saving of assemblies and drawings in PRO.FILE. 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 Inventor:
	"PRO.FILE" => "Save" => "Save incremental"

The further proceding for the function "**Save incremental**" corresponds to the proceeding described in the chapter "Saving CAD objects for the first time".

	Parameter: "Save incremental: Ignore modified components"
	This MMC parameter influences the behavior of the function "**Save incremental**": Here you can specify whether only new components or new and modified components are to be handled.

5.8   Save incremental automatic

The function "**Save incremental automatic**" is different from the function "**Save incremental**" in the aspect that document and part master records are created silently (i.e. without user interaction, where possible.

	Function call from the PRO.FILE menu in Inventor:
	"PRO.FILE" => "Save" => "Save incremental automatic"

5.9   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 Inventor:

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

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

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

5.9.1 The proceeding for "Managed Version"

Proceed as follows:

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

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

⇒ The Managed Version wizard is started.

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

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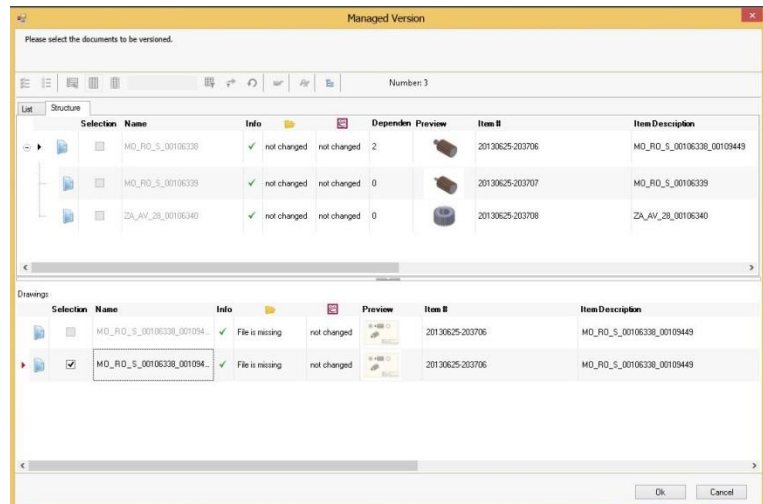
⇒ In the wizard of "Managed Version" the tree structure, determined and expanded by drawings, is shown, so that the documents to version with their dependent data (nodes) can be selected.

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

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

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

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

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

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

5. Confirm with <OK>.

⇒ The selection components are now versioned.
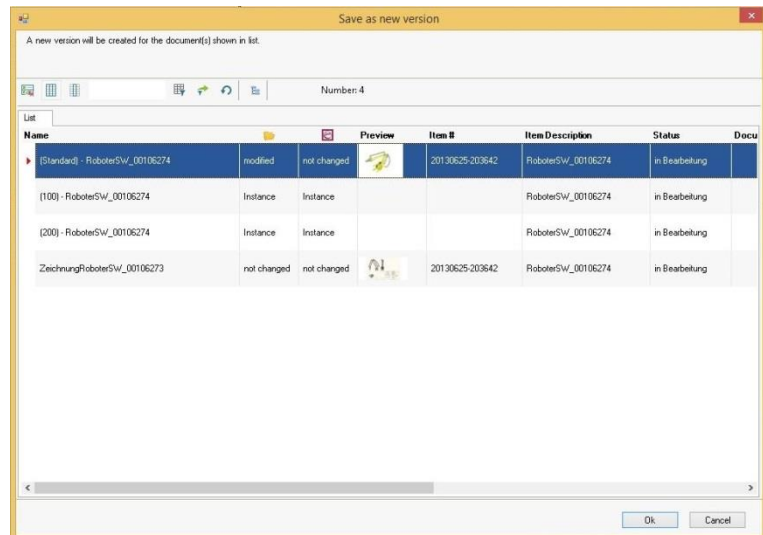
⇒ The successful completion of the process is confirmed by a message.

6. Confirm with <OK>.

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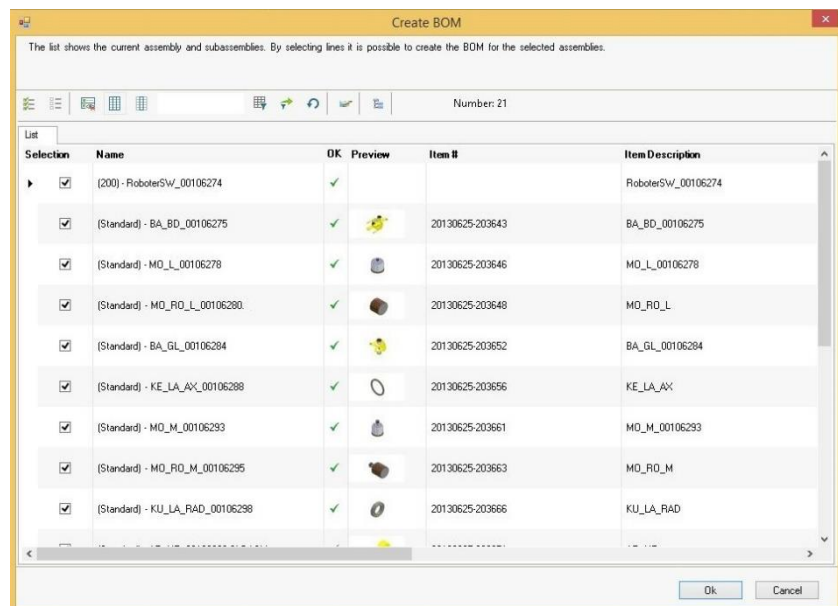
7. In this list, you can select all assemblies, for which the bill of materials is to be updated.

8. Confirm your selection with <OK>.

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

5.10   For drawings: Save NDF (Neutral data format)

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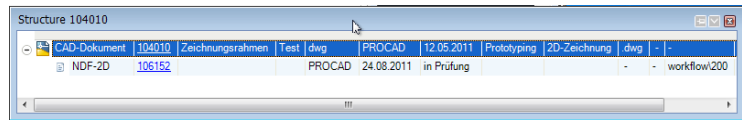
This function is only available for drawings, so the menu entry is only displayed in drawing mode.

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 Inventor:

"PRO.FILE" => "Save" => "Save NDF"

Create a neutral data format document

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

Note: Client-side NDF and server-side NDF

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

Change management via NDF generation

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

5.11 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.

- Assemblies and components to remain in the new design, are copied. New components are newly designed.

- New numbers of the cloned machine and new statuses in the dependent drawings are automatically updated.

- For all components not selected for Managed Copy, only the references are copied. Existing references thus remain intact.

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

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

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

5.11.1 Exchanged or not: What must be observed strictly?

The result of "Managed Copy" depends on the CAD documents opened in the CAD session and the CAD document selected for "Managed Copy"!
If higher-level assemblies are opened in the CAD session, a subassembly /CAD part, for which the function "Managed Copy" has been selected, is **exchanged** in these assemblies!

> **Attention: Result of Managed Copy**
>
> In Inventor, always all referencing assemblies and drawing loaded in the current session are switched to the copy created via Managed Copy.
>
> If you want to make sure that no accidental exchange takes place in other assemblies, do not load additional assemblies in the CAD session.

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

- **Requirement 1**: Create an independent copy of a model
- **Requirement 2**: 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.

5.11.2 Requirement 1: Create an independent copy of a model

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

- 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.

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

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

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

3. Activate the model to copy in the CAD session.
4. Call up the PRO.FILE function "Managed Copy", as described in the following chapter **Usage of the function "Managed Copy"**".
⇒ The created copy of the model is saved in PRO.FILE.
⇒ The created copy of the model is referenced in no higher-level assembly.

5.11.3 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.

Approach 2

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 Inventor session.
2. Open the model to copy via the "Managed Copy" function in the Inventor 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 Inventor session.
4. Call the PRO.FILE function "Managed Copy" up, as described in the following chapter [Usage of the function "Managed Copy"](#).
⇒ The created copy of the model is saved in PRO.FILE.
⇒ In all higher-level assemblies, which are loaded in a Inventor 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 Inventor and explicitly saved via the function "Save" of the integration.

Case study

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

Situation

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

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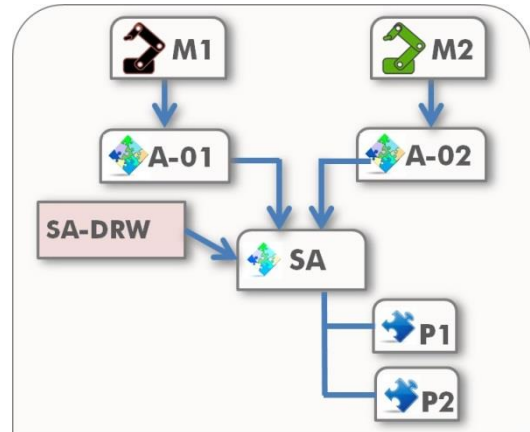
- The subassembly "SA" is active in the Inventor session.

- 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 1　　　　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 Inventor, the references in both assemblies are exchanged by Inventor.

- 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".
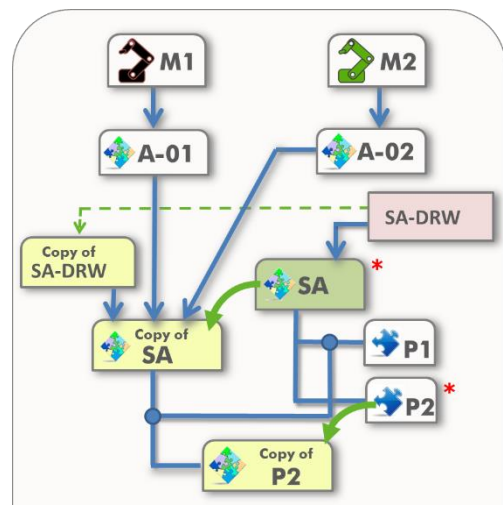
Result 2

IF the drawing was NOT selected for Managed Copy and NOT loaded in Inventor:

- 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 Inventor, the references in both assemblies are exchanged by Inventor.

- 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".
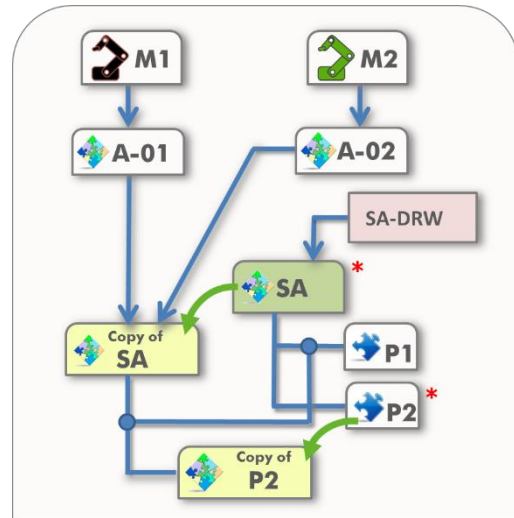
Behavior of the drawing

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

- If the drawing "SA-DRW" were loaded in the Inventor 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-Inventor.

5.11.4   Usage of the function "Managed Copy"

> **Attention: Result of Managed Copy**
>
> The result of "Managed Copy" depends on the CAD documents opened in the Inventor 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 Inventor:**
>
> "PRO.FILE" => area "Save" => "Managed Copy..."

Proceed as follows:

1. Select the menu item "PRO.FILE" in the menu bar of Autodesk Inventor.

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.

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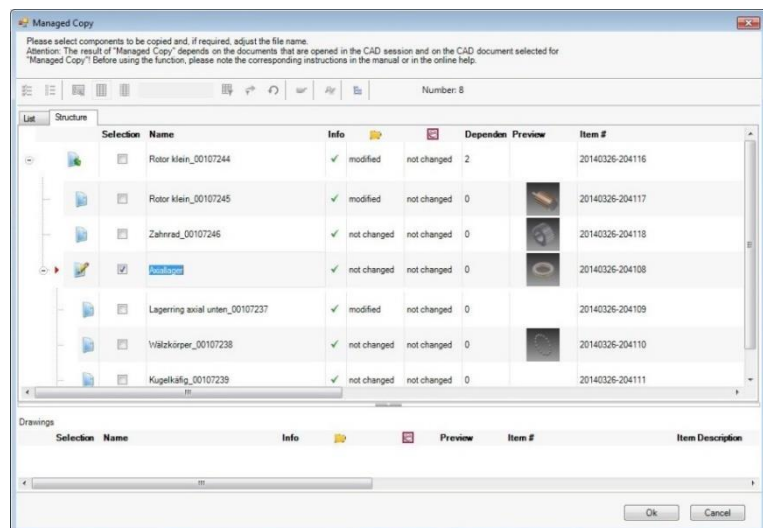
The top node and the first step are already folded out. Further steps can be folded out by a click on the structure symbol ⊕.

⇒   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
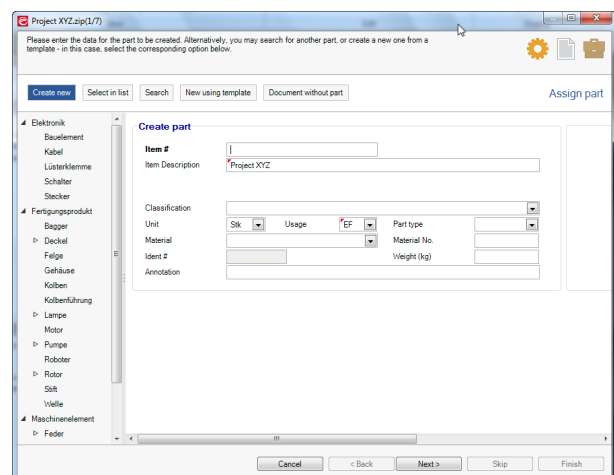
⇒   The information is requested for each selected component.

⇒   You will find Information on how to use the check-in wizard in the previous chapter "Saving CAD objects for the first time".

5.11.5　　　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 not direct method in the CAD systems itself to determine related drawings.

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

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

- The integration can scan the work folder for drawings and check whether these reference the model.

- **Attention**: Drawings that are not in the same work folder than the current model, for which the function "Managed Copy" is used, cannot be found by the integration.

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

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

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

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

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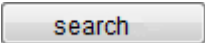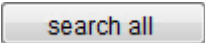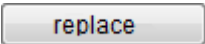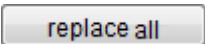
The file names of assemblies and parts can be edited during the execution of "Managed Copy". This is also possible via the function "**search and exchange**".

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.
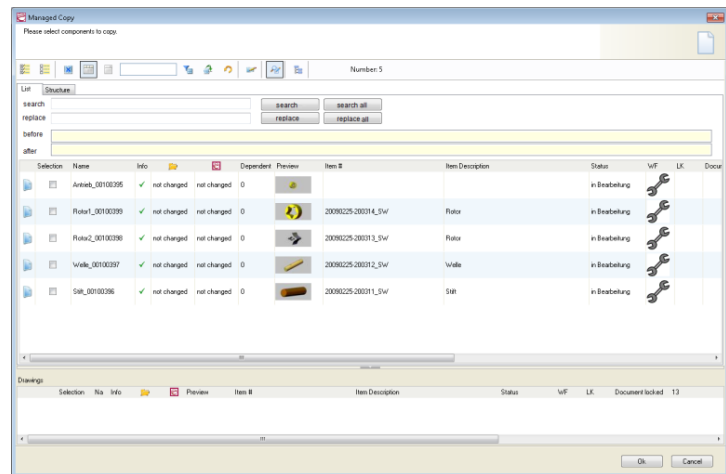
Now you have different possibilities, to execute "search and exchange".

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".

5.12   Managed Copy automatic

The function "**Managed Copy**" automatically combines the functions "**Managed Copy**" and "**save automatically**".

The function "**Save instances automatically**" combines the two functions

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

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

At the creation of these selected components – according to "save automatic" – 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 automatic](#)"

➡	Function call from the PRO.FILE menu in Inventor: "**PRO.FILE**" => area "**Save**" => "**Managed Copy automatic**"

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

6   Linking of additional files

It is possible to link additional files to Inventor 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 an Inventor 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 Inventor objects, additional files are displayed in the PRO.FILE tab "Browser", as well as in the selection dialogs for "Manged Copy" and "Disconnect relation". The versioning of bills of materials ignores additional files.

6.1   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 an Inventor object that has been saved in PRO.FILE into your CAD session.

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

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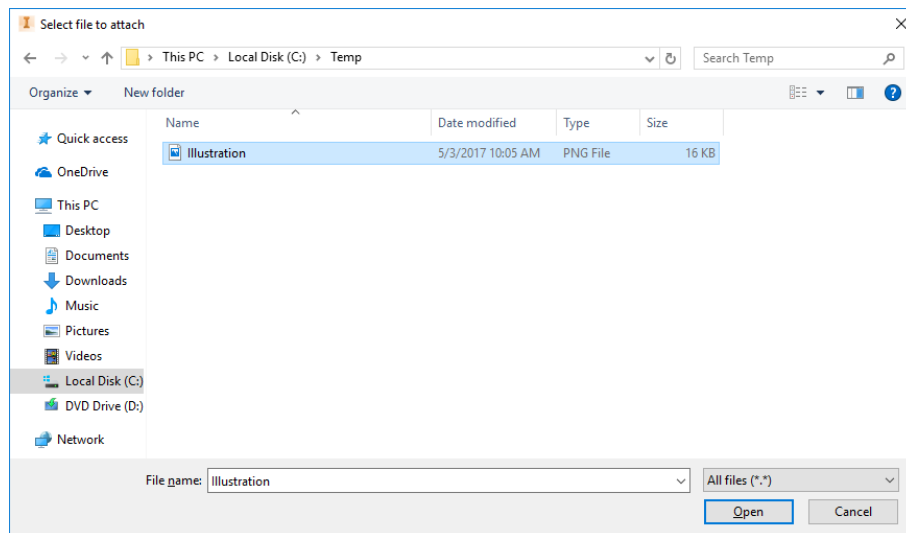
3. Select the file to be added and confirm your selection with <**Open**>.

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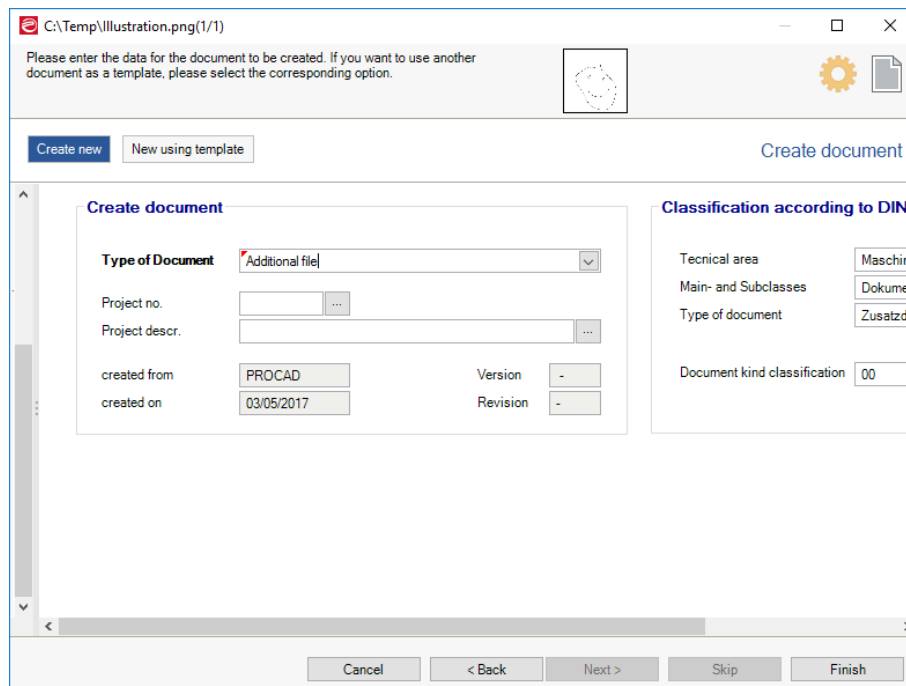
4. Confirm the assignment with <**Next**>.

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

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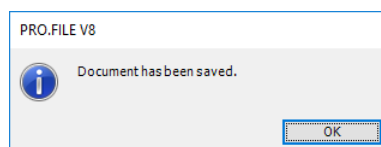
⇒ The additional file is saved in PRO.FILE. It is linked below the document description of the Inventor object. If possible, a preview file is created for the additional file.

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

6.2   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 an Inventor object that has been saved in PRO.FILE into your CAD session.

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

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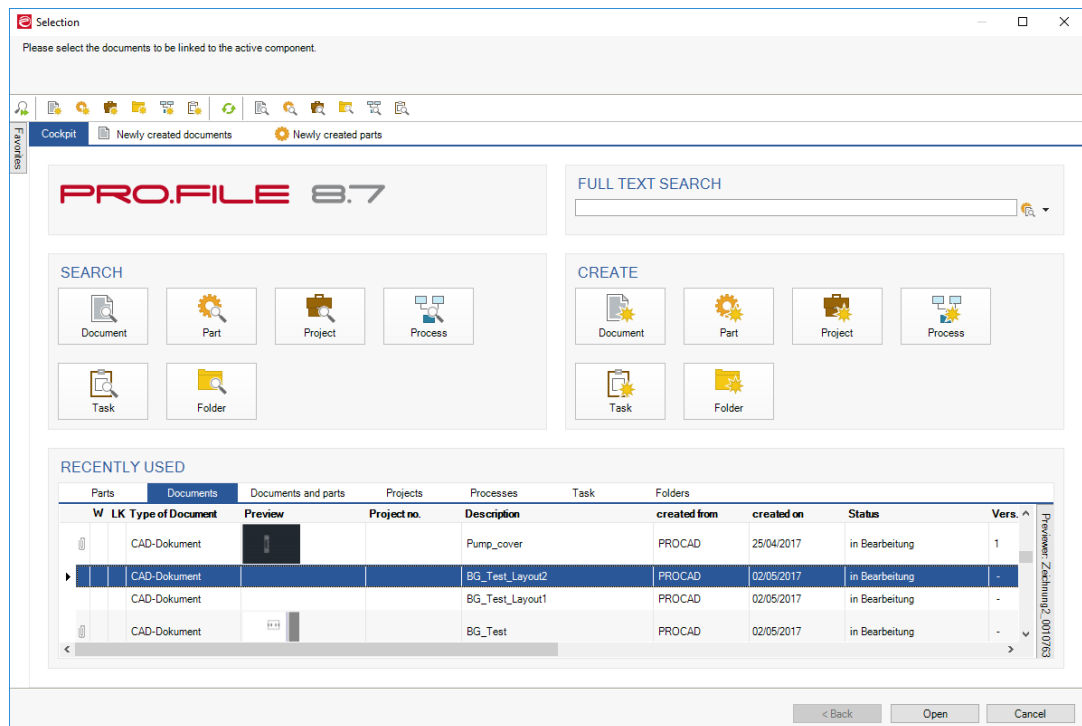
3.   Select the document record of the file to be added and confirm your selection with
     <**Open**>.

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

6.3   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 an Inventor 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.
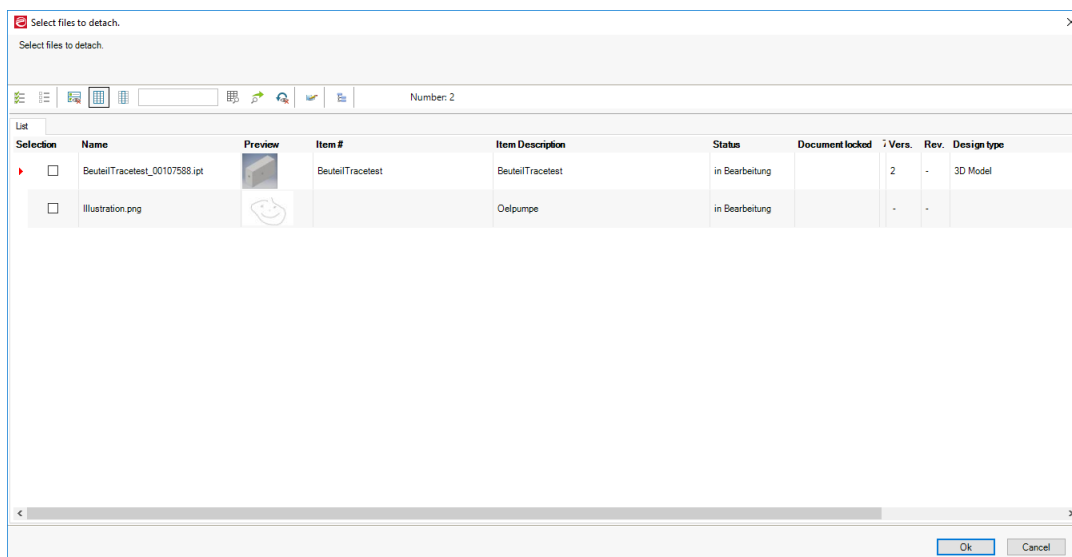
3. From the displayed list, select the additional file you want to detech by activating the corresponding checkbox in the column "Selection".

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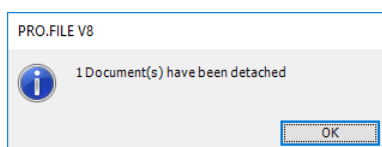
⇒ The selected document is removed from the Inventor object structure.

7   Show: PRO.FILE Information at a glance

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

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

This way, you can immediately access information stored in PRO.FILE on properties and classifications of the active CAD document as well as structure and usage information.
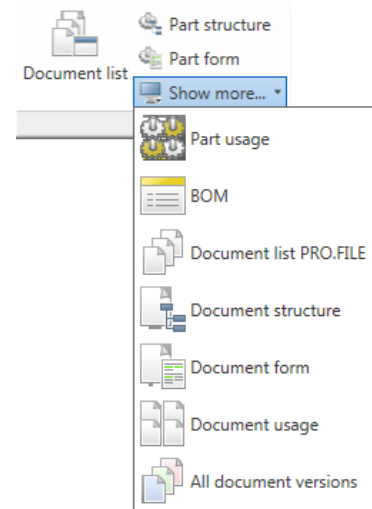
The following information can be found in the area "**Show**":

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

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

7.1   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 Inventor:

"PRO.FILE" => "Show" => "Document list"

When the function is used, the document list is displayed:
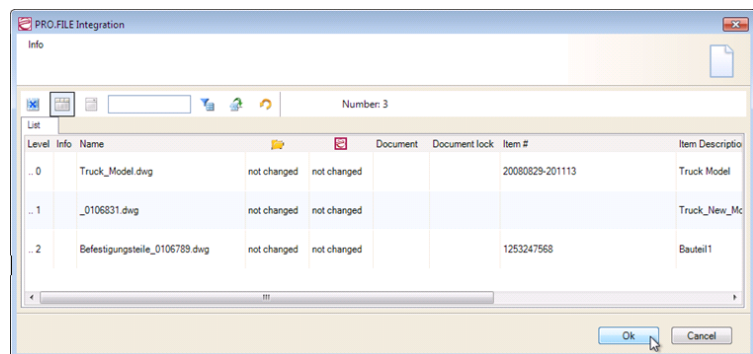
You find the following information:

- The data from the PRO.FILE document description.

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

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

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

Detailed information can be found in the following chapters:

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

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

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

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

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

- The various menu entries allow a targeted access to frequently needed information, without having to navigate in PRO.FILE.

- After a function is selected, PRO.FILE opens and displayed the desired information.

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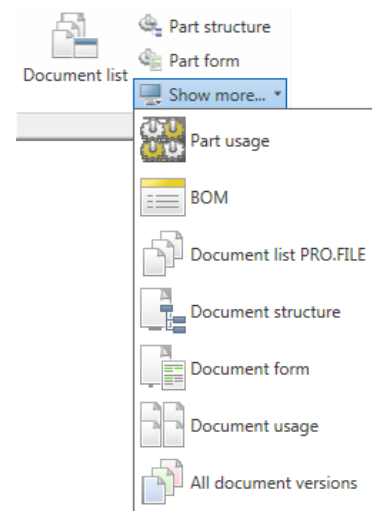
	Function call from the PRO.FILE menu in Inventor: "PRO.FILE" => "Show" => "…"

	Note: Please note that you can only use these display functions if the CAD document is already saved in PRO.FILE. If you have made changes to your CAD document and have not yet saved these changes back to PRO.FILE, these changes are ignored by the display functions.

The following display options are available:

7.2.1   Part structure

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

7.2.2   Part form

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

7.2.3   Part usage

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

The usage list displays the "upward" structure.

7.2.4   Bill of materials

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

7.2.5   Document list incremental

This function searches the active level of the assembly and the level below for new and modified components and displays them in a list. If new and/or modified documents are found, the structures of these are further searched until no more new or modified documents are found. This does not apply for phantom assemblies – phantom assemblies are always searched.

The behavior of this function is influenced by the parameter "Save incremental: Ignore modified components". Here you can specify whether only new components or new and modified components are to be handled.

7.2.6   Document list in PRO.FILE

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

7.2.7   Document structure

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

7.2.8   Document form

The function "**Document form**" displays the document description of your current CAD document in the PRO.FILE form view. Here you can find the specification of the document-describing data for this CAD document.

7.2.9   Document usage

With the function "**Document usage**" you can see whether the document description of your active CAD document is used in other document or part descriptions.

7.2.10   All document versions

The function "**all document versions**" displays al visible current and old versions of your CAD document.
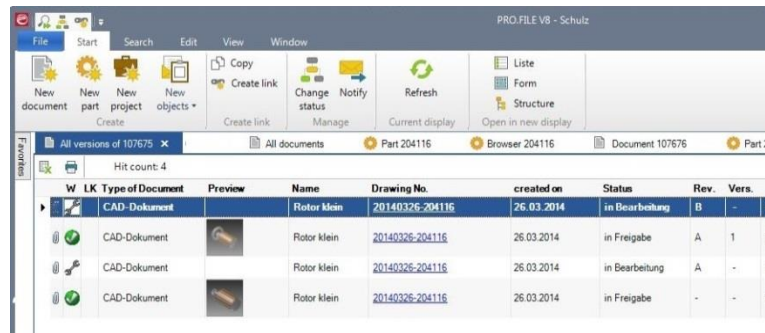
7.3   Direct information in the dialog screens

For the functions lock, unlock and document list, dialog screens are displayed.

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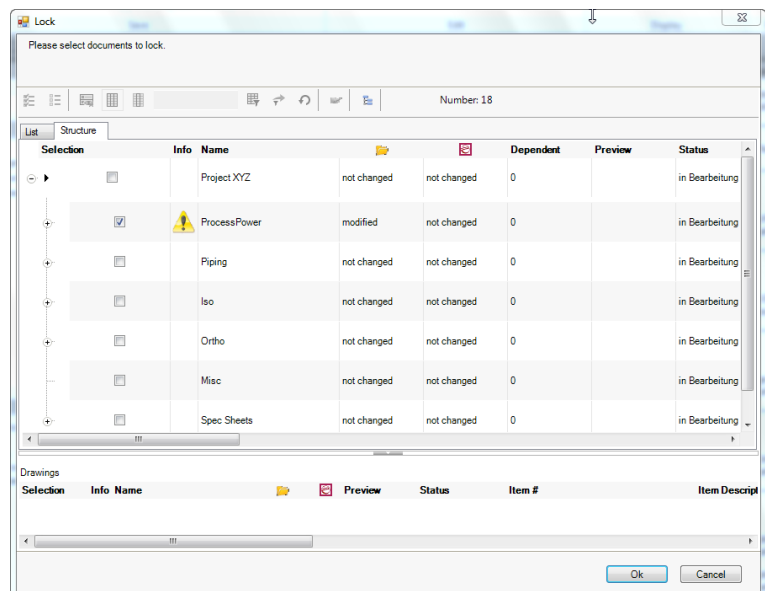
- You can switch between list and structure display.
- Via the buttons of the list functions you can make searches and filter the displayed objects (see the following chapter "More comfort: search and list functions in the dialog screens").
- The lists contain status information for each of the listed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

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

The dialog screens of the PRO.FILE CAD 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.

- Change row height:

  Reduces the row height by removing thumbnails from the rows. As a result, more rows can be displayed on the screen.

7.3.2   Up to date or not: Display of status information

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

- Info: Shows an icon for the data status. If you hover over the icon with the mouse pointer, a tool tip with more information is displayed.
- : Displays the status of the CAD data in the local work folder of the Workcenter.
- : Displays the status of the CAD data in PRO.FILE.

These columns may contain the following:

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

Info	Local 📁	PRO.FILE 🗃	Description
			different user and can therefore not be saved back.
⚠️	unchanged	changed	The file is locally unchanged but has been modified in PRO.FILE after it has been copied locally and can therefore not be saved back.
⚠️	unchanged	changed locked	The file is locally unchanged but has been modified in PRO.FILE after it has been copied locally and can therefore not be saved back.
📄⚠️	unchanged	changed versioned	The file is locally unchanged but has been modified in PRO.FILE after it has been copied locally and can therefore not be saved back. There is a newer version of this file in PRO.FILE.
📄⚠️	unchanged	changed locked versioned	The file is locally unchanged but has been modified in PRO.FILE after it has been copied locally and can therefore not be saved back. There is a newer version of this file in PRO.FILE.
⚠️	changed	unchanged	The file is locally changed but has not yet been saved back to PRO.FILE.
❗	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

Info	Local 📁	PRO.FILE 🔴	Description
			back. There is a newer version of this file in PRO.FILE.

8 Editing: What other functions does the integration offer?

Via the PRO.FILE menu "**Work**" in Inventor you can access additional functions available via the integration.
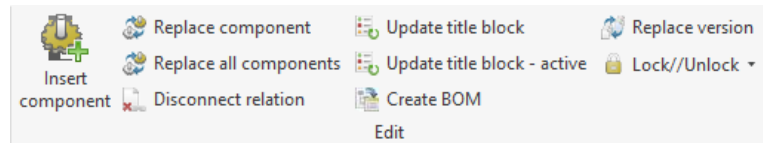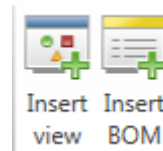
These functions are described in the following chapters:

- [Disconnect relation](#)
- [Update title block](#)
- [Update title block – active](#)

- [Create](#)
- [Replace version](#)

When a drawing is active, you can also find the functions:

- [For drawings: "Insert view"](#)
- [For drawings: "Insert BOM"](#)

For active assemblies you have access to the functions:

- [For assemblies: "Insert component"](#)
- [For assemblies: "Replace component"](#)
- [For assemblies: "Replace all components"](#)
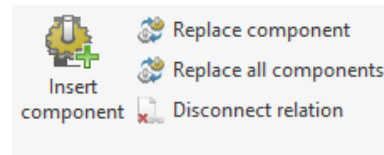
For detailed information see the following sub-chapters.

8.1 Disconnect relation

This function dissolves the connection between the selected CAD document and the PRO.FILE document description. The document then no longer has a PRO.FILE connection. The CAD documents are thus treated as purely locally stored CAD data and marked accordingly in the Workcenter.

- The current document in Inventor can now be changed and saved as a new document in PRO.FILE, with a different name and ID number.

- The "**old**" document still exists in PRO.FILE. With the function "**Disconnect relation**" no data gets lost, only the local copy of the CAD document is disconnected from the document description in PRO.FILE.

Function call from the PRO.FILE menu in Inventor:

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

Proceed as follows:

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

2. Select the function "**Disconnect relation**" from the area "**Edit**".

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

   (Information on the functions and status information can be found in the chapter "The document list").
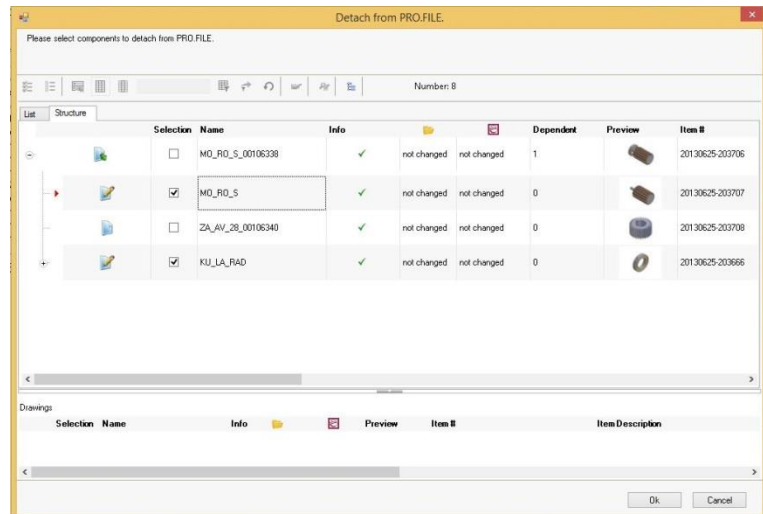
3. Select all CAD documents, the connection to PRO.FILE of which you want to dissolve. To do so, activate the checkbox next to the listed documents.

⇒   As new file name the current name without ID or with the suffix "_Index" is suggested, e.g. "**assembly_1**".

4. Assign the file names for the files to be disconnected so that the files can be saved locally. You can edit the file names freely in the column "Names".
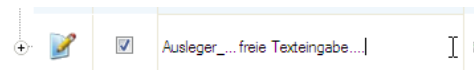
5. Once all file names are specified, confirm with <**OK**>.

⇒   The PRO.FILE connection for all selected CAD components is now dissolved.

⇒   The dissolving of database connection for CAD documents is now finished.

⇒   The selected CAD data is now stored locally and no longer has a PRO.FILE connection. Changes to these components are not saved to PRO.FILE!

To save these CAD data again in PRO.FILE you can use the saving functions described in the chapter "Save: How to save CAD data and changes to PRO.FILE?".

8.2   Update title block

With the function "**Update title block**" the file attributes of the drawing legend of the current Inventor drawing are filled with current data from PRO.FILE.

The iProperties of the active CAD document and its structure (if such structure exists) are updated. If these iProperties are mapped to text fields of the title block or the modification list in a drawing, these are updated as well.

	Function call from the PRO.FILE menu in Inventor:
	"PRO.FILE" => "Edit" => "Update title block"

The update of the title block does not require any user action apart from the actual function call:

- When the function is selected, all elements contained in the drawing legend (title block, modification list, bill of materials) are automatically filled with current data from PRO.FILE.

- The modification list always contains the most recent entries.

	Note:
	A prerequisite for this function to work is that these lists and fields are configured for the used drawing. The configuration of the drawing legend is described in the configuration manual for the Inventor integration.
	The update of the drawing legend is also made upon the saving and loading of CAD documents.

8.3   Update title block – active

In contrast to the function "**Update title block**", this function only update the iProperties of the active CAD document and the levels bolow it.

	Function call from the PRO.FILE menu in Inventor:
	"PRO.FILE" => "Edit" => "Update title block – active"

8.4   Create BOM

 With the function "Create BOM" a bill of materials is created in PRO.FILE based on the CAD structure of the active document.

If a bill of materials structure already exists in PRO.FILE for the active CAD structure, it is updated. If the assembly contains parts that are not yet contained in the bill of materials, they are automatically included in the bill of materials structure.

For this, the Inventor bill of materials structure attributes of the model data are evaluated, as with the CAD system:

- Phantom and reference objects are suppressed.

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

|	Note:
	Please note the following requirements for the creation of the bill of materials in PRO.FILE: • Bills of materials can only be created for assemblies. • The CAD documents must be connected to a part master record in PRO.FILE.

	Function call:
	"PRO.FILE" => "Edit" => "Create BOM"

Proceed as follows:

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

2. Select the function "**Create BOM**" from the area "**Edit**".

⇒ A list shows all currently loaded assemblies and sub-assemblies.
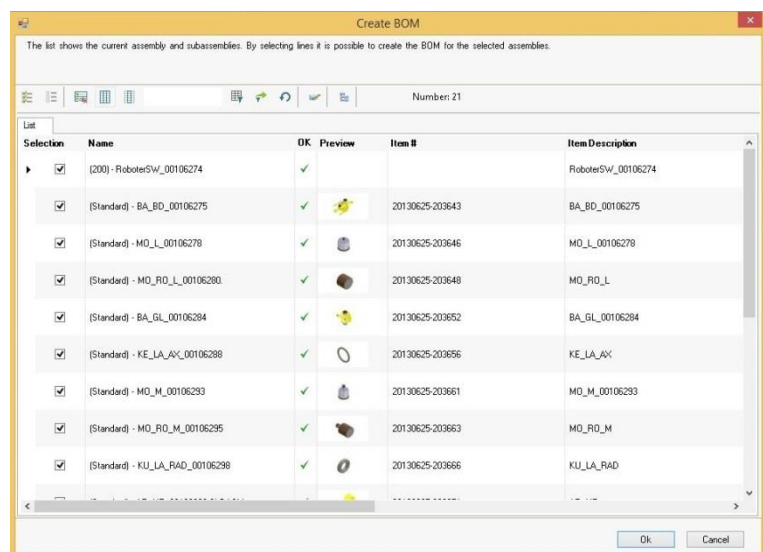
3. Select all CAD documents, for which you want to create or update the bill of materials. To do so, activate the checkboxes next to the listed CAD documents.

Note: Display of conflicts

If the bill of materials cannot be created for an assembly / sub-assembly, the column "OK" displays an "Attention" icon. The corresponding tool tip shows the problem in detail.

4. Confirm your selection with <OK>.

⇒ If, during the update, the PRO.FILE bill of materials contains positions, that do not occur in the geometry, a corresponding query is displayed.
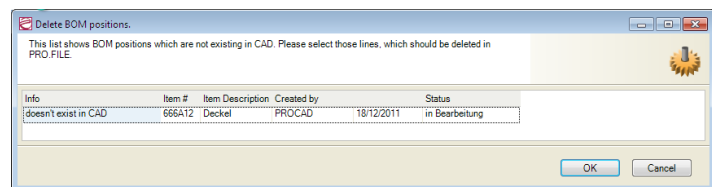
⇒ Since it is possible to edit the bill of materials structure manually in PRO.FILE, the bill of materials may contain positions that are not displayed in the geometry.

5. Select the lines, the bill of materials position in PRO.FILE of which is to be deleted.

6. Confirm your selection with <OK>.

⇒ The bill of materials is created in PRO.FILE for the selected CAD assemblies.

⇒ A message confirms the successful creation/update of the bill of materials in PRO.FILE.
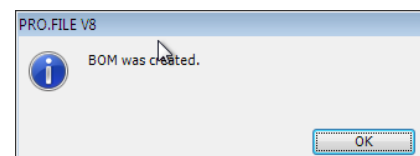
⇒ To view the bill of materials in PRO.FILE, select the function "**Show**" => "**Show more…**" => "**Bill of materials**", as described in the chapter "Show: Information on a CAD document in PRO.FILE".

Note:

Please note that norm parts or auxiliary materials (e.g. water or oil) that are not displayed on the drawing are not transferred into the bill of materials, when the function "**Create bill of materials**" is used.

You can add these parts to the bill of materials in PRO.FILE manually. The description of the functions for editing a bill of materials in PRO.FILE can be found in the PRO.FILE operation manual "**Structures and bills of materials**".

Note:

This function can be activated automatically after each saving process. For detailed information see the configuration manual for the integration PRO.FILE – Inventor.

8.5 Replace version

The function "**Replace version**" allows to replace an existing version of a CAD document in all assemblies, in which it is used, by a newer version.

Function call from the PRO.FILE menu in Inventor:

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

When the function "**Replace version**" is called, a usage check is made for the currently active CAD document based on its PRO.FILE ID. The program checks in which assemblies and drawings the active document is being used.

After that the old version of the active PRO.FILE document can be replaced in all corresponding assemblies and drawings by the new version.

To replace a version proceed as follows:

1. Open a new version of a document from PRO.FILE.

2. Select the function "**Replace version**" from the integration menu area "**Work**" in Inventor.

⇒ You now get a list of how often and where the previous version of the document is used.

3. Select the data records, for which the loaded version is to be used.

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

⇒ The current version is now used by the selected assemblies/drawings.

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

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

8.6   For assemblies: "Insert component"

With the menu function "**Insert component**" assemblies or parts saved in PRO.FILE can be inserted into the assembly loaded in Inventor.

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

Function call:

"PRO.FILE" => "Edit" => "Insert component"

Proceed as follows:

1. Select the "**PRO.FILE**" menu from the menu bar in Inventor.
2. Select the function "**Insert component**" in the "**Edit**" are of the menu.
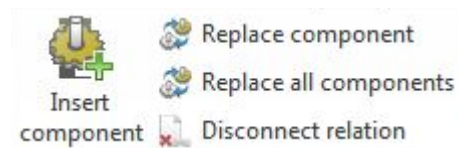
⇒ You are now prompted to select the component in PRO.FILE, which is to be inserted into the assembly.

⇒ The Checkout wizards supports you in this process.

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

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

4. Then select the insertion point for the component in your Inventor assembly.

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

Note: Only available for assemblies

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

8.7   For assemblies: "Replace component"

Via the function "**Replace component**" components used in the Inventor assembly are replaced by a component from PRO.FILE. After selection of the replacing PRO.FILE CAD document you can determine the component to be replaced in Inventor by mouse click.

Function call:

"PRO.FILE" => "Edit" => "Replace component"

Proceed as follows:

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

2. Select the function "**Replace component**" from the area "**Edit**".
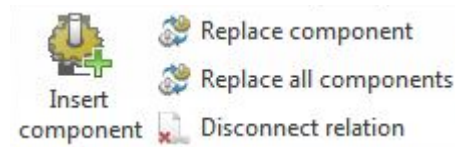
⇒ You are now prompted to select the component in PRO.FILE that is to replace a component used in the assembly.

⇒ The Checkout wizards supports you in this process.

3. In the Checkout wizard select the desired CAD document and click <**Open**>.

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

4. You can then select the component in Inventor, that is to be replaced by the component from PRO.FILE, by mouse click.

⇒ The selected component is now replaced by the component from PRO.FILE.

Note: Only available for assemblies

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

8.8   For assemblies: "Replace all components"

If a component is used several times in an Inventor assembly, the function "Replace all components" can be used to replace this component with a component from PRO.FILE at all places in the assembly.

Function call:

"PRO.FILE" => "Edit" => "Replace all components"

The proceeding corresponds to the proceeding for the function "For assemblies: "Replace component"".

The difference is in the scope of replacement: When the component to be replaced is selected, it is not only replaced at the selected location but in all location it is used within the assembly.

Note: Only available for assemblies

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

8.9   For drawings: "Insert view"

With the function "**Insert view**" assemblies or parts saved in PRO.FILE are inserted into the drawing.

With the PRO.FILE Checkout wizard you can select a document, which can then be placed in the drawing.

Function call:

"**PRO.FILE**" => "**Edit**" => "**Insert view**"

Proceed as follows:

1.   Select the menu "**PRO.FILE**" in the menu bar in Inventor.

2.   Select the function "**Insert view**" from the area "**Edit**".
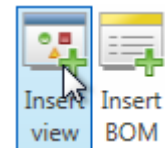
⇒   You are now prompted to select the component in PRO.FILE, which is to be inserted into the drawing.

⇒   The Checkout wizards supports you in this process.

3.   In the Checkout wizard select the desired CAD document and click <**Open**>.

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

⇒   The Inventor wizard for the insertion of a component is displayed.

4.   Select the desired Inventor insertion option and click <**OK**>.

5.   You can then select the insertion point for the view in your Inventor drawing.
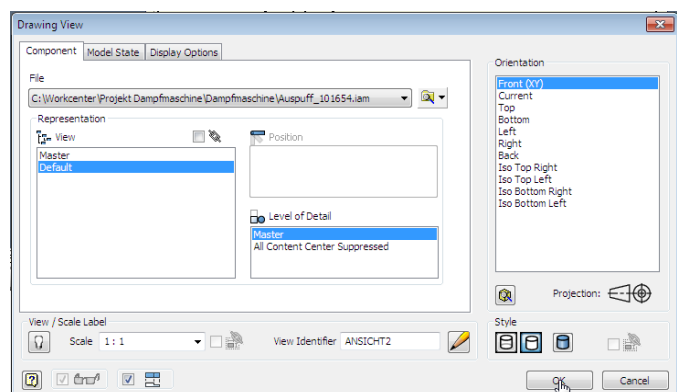
⇒   The component view from PRO.FILE is now inserted in your drawing.

	Note: Only available in drawing mode
	This menu entry is only available when Inventor is in drawing mode.

8.10   For drawings: "Insert BOM"

You can use this function to display the bill of materials for your CAD data from PRO.FILE on the drawing.

To insert a bill of materials from PRO.FILE the following requirements must be met:

- The components of the assembly must be saved in PRO.FILE.

- The bill of materials must be created in PRO.FILE (see function "Create ").

	Function call:
	"PRO.FILE" => "Edit" => "Insert BOM…"

Proceed as follows:

1. Select the function "**Insert BOM**" from the area "**Edit**" of the PRO.FILE menu in Inventor.
2. You can now place the bill of materials by entering a position on the drawing.
⇒ If a PRO.FILE bill of materials already exists on the drawing, the function "**Insert BOM**" will update it.

	Note: Single-level bills of materials
	To avoid display problems it is recommended to display only the first level of the bill of materials on the drawing. This behavior can be configured via the MMC parameter "Max. nesting level of BOM on drawing".

	Note: Only available in drawing mode
	This menu entry is only available when Inventor is in drawing mode.

8.11   Management of OLE references

Note: Prerequisites

The management of FEA OLE references is supported by Inventor 2012 and higher.

To use this feature, the parameter "Manage OLE references" has to be activated. For details, please contact your PRO.FILE project manager.

OLE references are references to non-CAD-model documents. This may include images or Office documents.

If the OLE management is activated, OLE references in CAD documents are read out and also saved to PRO.FILE when the CAD document is saved to PRO.FILE.

- Pure OLE references are saved as additional files (without geometry type) in PRO.FILE (only OLE references, NOT FEA OLE references).

- FEA references (Finite Element OLE references) are saved with the geometry type "Analysis" in PRO.FILE. FEA references are not renamed.

The following scenarios are to illustrate the management of OLE references:

Scenario 1

Saving a drawing with OLE link to an excel file

You are saving an Inventor drawing containing an OLE link to a Microsoft Excel file.

The result is the following:

- The OLE reference to the Excel file is saved in PRO.FILE and displayed in the document structure in parallel to the CAD references.
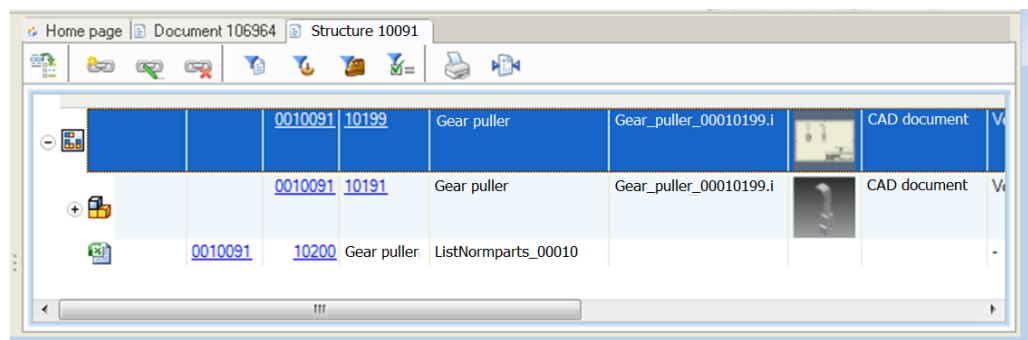
Scenario 2

Saving a drawing with FEA OLE Reference

You are saving an assembly with FEA analysis (finite elements).

The result is the following:

- The FEA OLE reference is saved in PRO.FILE and displayed in the document structure in parallel to the CAD references.
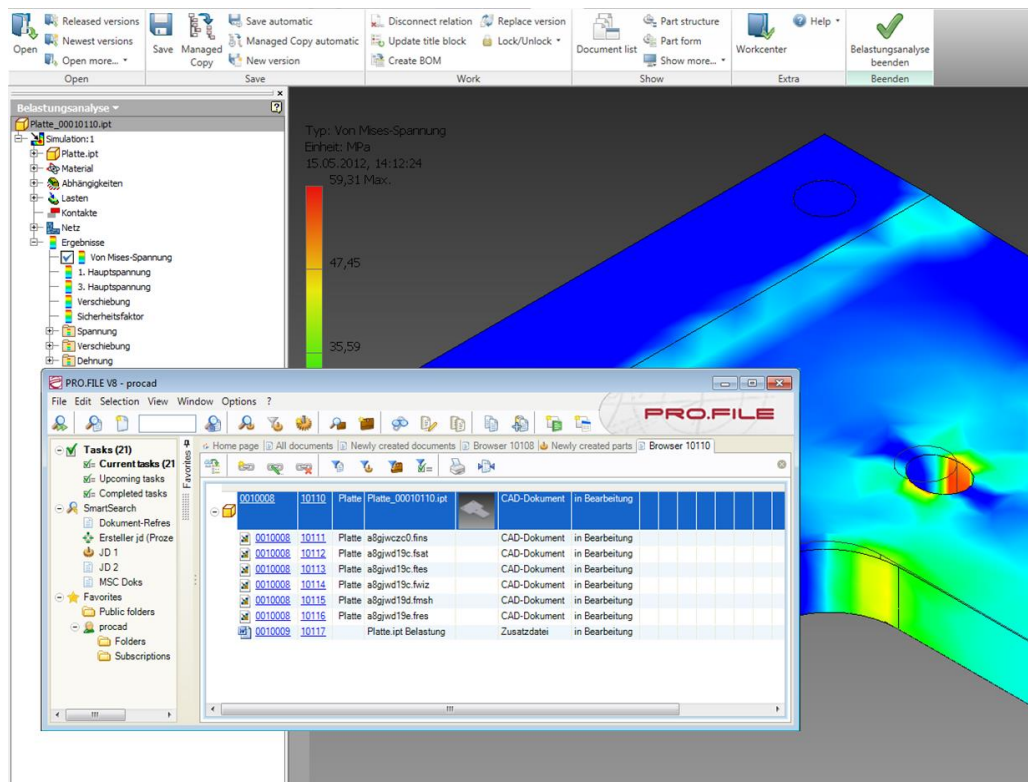
8.12   Management of content center norm parts

Note: Prerequisites

The management of content center norm parts is supported by Inventor 2012 and higher.

To use this feature, the parameter "Manage content center standard parts" has to be activated. For details, please contact your PRO.FILE project manager.

If the norm part administration in PRO.FILE is activated, all parts generated from the content center (norm parts) are only stored once in PRO.FILE.

This means: If a norm part that is already saved in PRO.FILE is used again from the content center, this is recognized by PRO.FILE upon saving and replaced by the PRO.FILE part.

The following scenario is to illustrate the management of content center norm parts:

Scenario: Saving an assembly with screw joints from the content center

Step 1: You are saving an assembly with a screw joint from the content center to PRO.FILE. The result is the following:

- New document descriptions are created in PRO.FILE for all existing components.

- The result is displayed in the PRO.FILE document structure.
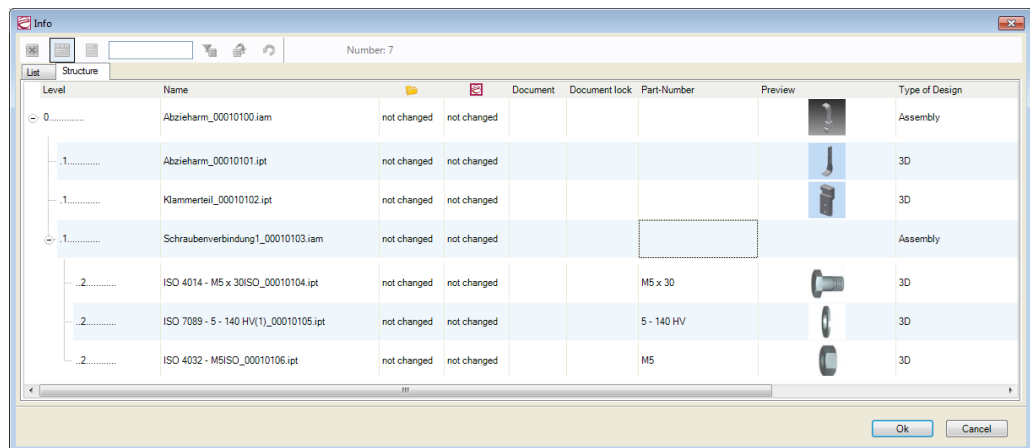
Step 2: A second, identical screw joint is built into the assembly and saved to PRO.FILE. The result is the following:

- The three norm parts contained in the screw joint 2 are recognized as parts known in PRO.FILE and displayed in the list. During the following saving process, these parts are replaced by the parts already saved in PRO.FILE.
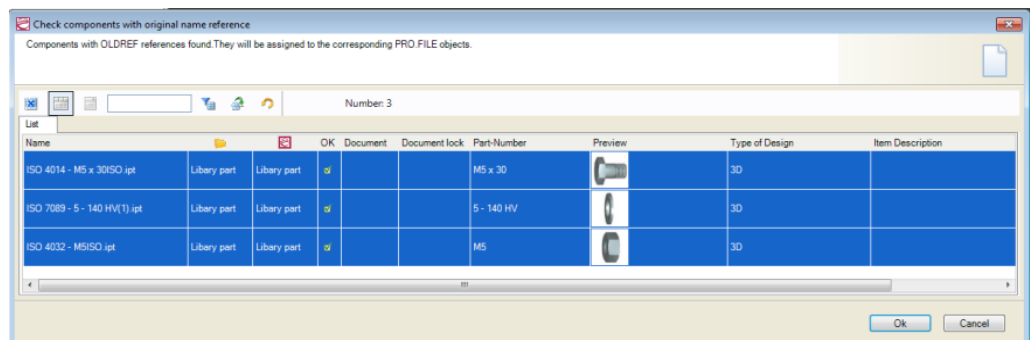
- Only the document of the assembly "Screw joint1" is newly created in PRO.FILE.
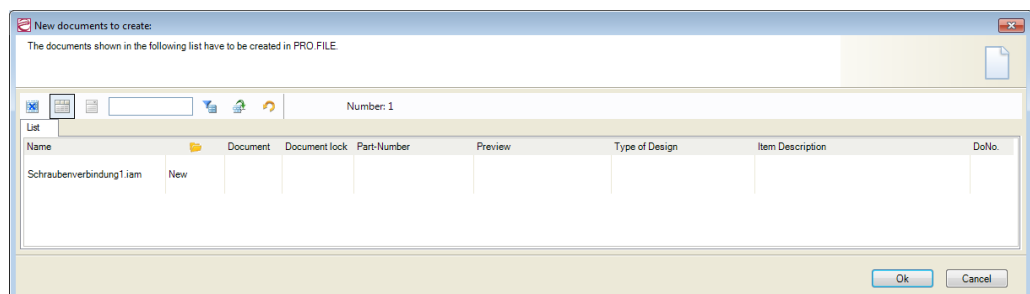
- This result is now also visible in the document structure. Identical norm part components are used (=referenced) in both screw joints.

- This result is also visible in the PRO.FILE bill of materials:

	Note: Technical background

Please note the following conditions:

- The content center for norm parts is identified by the iPropertySet "Content Center".

As key for the document identification in PRO.FILE the internal name of the CAD document (GUID) is used. This means, especially for documents using the same GUID:

- If a document with a GUID is saved to PRO.FILE and there is already a document with this GUID in PRO.FILE, it is **replaced**!

9 Extra: Additional functions of the integration

The menu "Extra" contains functions for the local work folder, the online help and for the saving of thumbnails and preview files.

9.1 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 Inventor:
	"PRO.FILE" => "Extra" => "Workcenter"

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

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

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

9.1.1 Workcenter functions

The Workcenter is divided into two areas

- on the left hand side you can find the directory structure of the Workcenter and its commands.

- on the right hand side you can find the commands for all parts or other files currently retrieved from PRO.FILE that can be found in the working directory. You can also find here the status information as described in the chapter "Up to date or not: Display of status information".
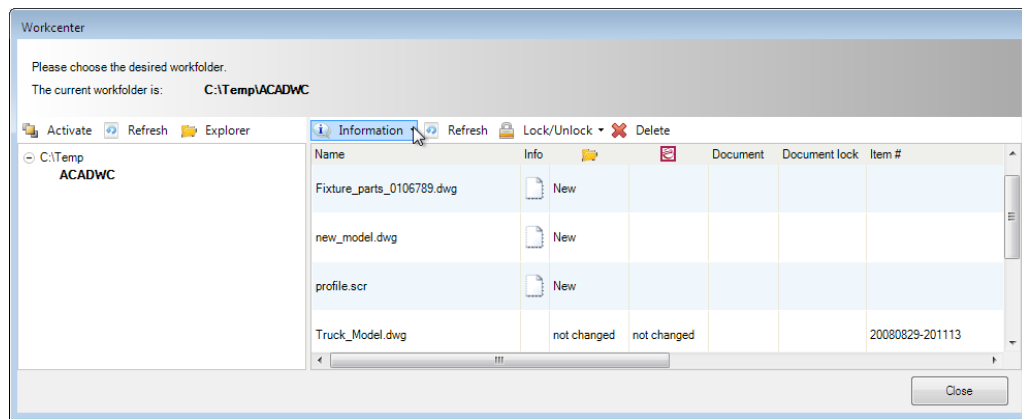
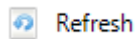**The functions for the directory structure**

 Activate

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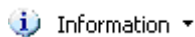
 Refresh

The view of the directory structure is updated.

 Explorer

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

The functions for the working directory

 Information ▾

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

Structure of the parts	Document structure
Part form	Document form
Usage of parts	Usage of documents
Bill of materials	
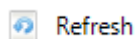
 Refresh

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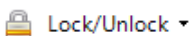
 Lock/Unlock ▾

The respective document is – depending on the user's authorizations – locked or unlocked.

 Delete

The marked documents are deleted from the directory.

If the local status of at least one of the selected files is more recent than the one stored in PRO.FILE, a warning message will be displayed.
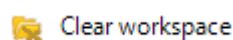
 Clear workspace

Starting from the selected work folder, all files that have been saved to PRO.FILE and that have not been modified locally since are deleted – including files in sub-folders.

Filter	The display filter for the document list can be adjusted via this icon. This can be used to facilitat the finding of objects in large folders.
Update version	Selected files can be replace by a newer PRO.FILE version (of the same file name). If version conflicts arise, the PRO.FILE dialog for the version selection is displayed.
Open with double click in the CAD system	Double-clicking a file in the Workcenter opens the file in in the CAD system (if it is not already opened).

9.2   Save thumbnail

If the document lists of PRO.FILE also display thumbnails, the function "**Save thumbnail**" is used to update the thumbnail picture.

> Function call from the PRO.FILE menu in Inventor:
>
> "**PRO.FILE**" => area "**Extra**" => "**Save thumbnail**"

9.3   Save preview file

For documents of drawings or 3D parts/assemblies, a preview can be displayed in a separate tab in Pro.FILE. The display in this preview tab is facilitated by a viewer (e.g. Oracle AutoVue).

However, if the viewer does not support the current version of your CAD software, a specific preview file can be created via the function "**Save preview file**". This is a STEP or PDF file which can be handled by the viewer without problems.

The function "**Save preview file**" has to be configured by the administrator. The proceeding is described in the manual "**Administration PRO.FILE**".

> Function call from the PRO.FILE menu in Inventor:
>
> "**PRO.FILE**" => area "**Extra**" => "**Save preview file**"

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