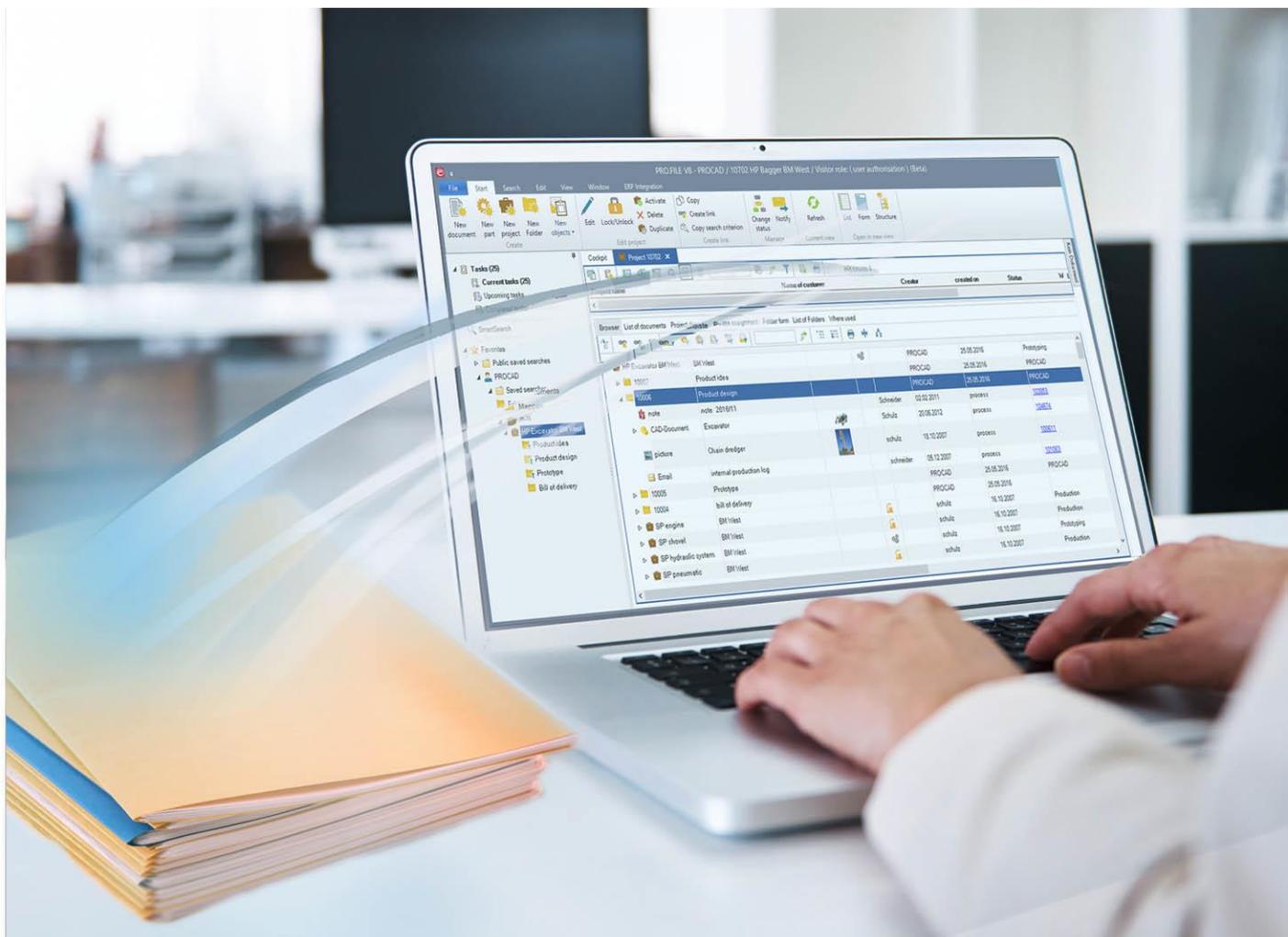


Functions of the Integration PRO.FILE AutoCAD

PRO.FILE Release 8.7
June 2017



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About this manual

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

Step-by-step instructions:

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 – AutoCAD

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

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 AutoCAD to ERP systems.

1.1 The contents of this manual

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

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 AutoCAD. The following topics will be addressed:

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



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

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

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

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

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 AutoCAD.

- [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 AutoCAD, and also saved and versioned from AutoCAD 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 AutoCAD 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 – AutoCAD.

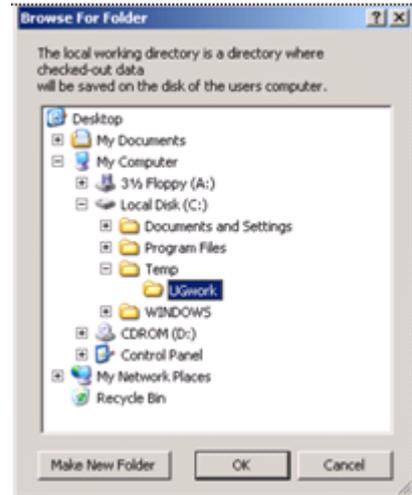
You can freely choose the local work folder at the first start of the integration.

Proceed as follows:

1. If no local work folder is defined when the integration is started, an input screen will prompt you to define this folder.

2. You now have to specify a "root folder". The root folder is the superior folder of the local data storage. In this folder you can later create several work folders, which are then supervised by the "Workcenter".

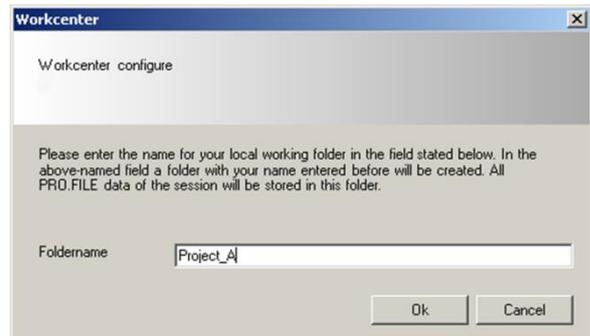
- The "root folder" can be selected - or created via the button <Make new folder>.
- Once you have selected the desired root folder, confirm with <OK>.



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

- Please specify a name for the work folder.
- Confirm your entry with <OK>.

⇒ The configuration of the Workcenter is now finished.



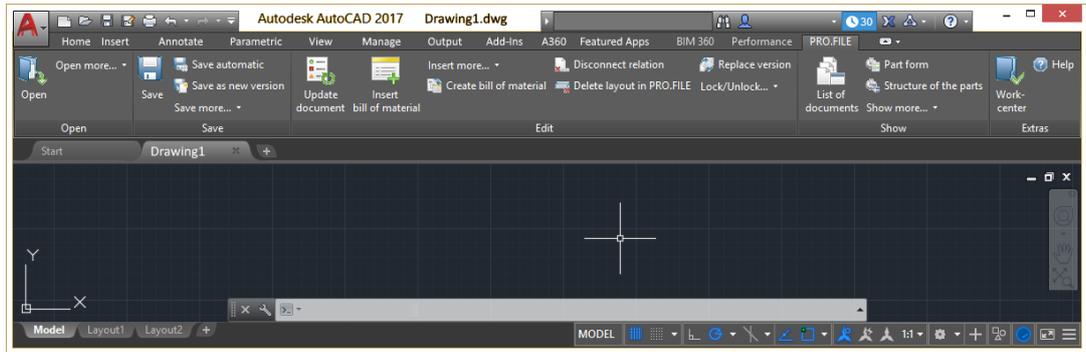
This work folder and other local work folder can be created and managed by the user via the Workcenter. The Workcenter can be accessed via the PRO.FILE menu in AutoCAD under "Extra" => "Workcenter".

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

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

The functions of the PRO.FILE integration can be accessed directly in AutoCAD via the menu "PRO.FILE":

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



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 AutoCAD session, you have to log in to PRO.FILE.

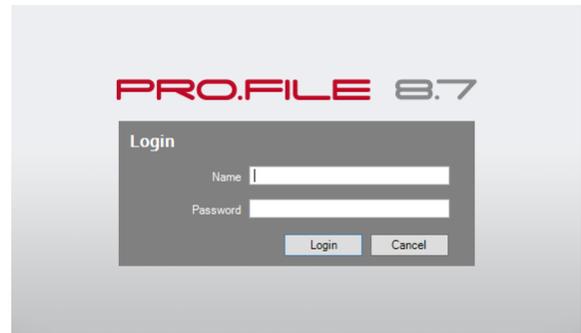
Via the logon, the user is now identified by his PRO.FILE user name and password. Based on this logon, the user rights, start statuses and function access rights for the logged-on user are activated.

In the login screen, please enter:

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

Confirm with <OK>.

The PRO.FILE home screen is now displayed.





Note: No login required in case of "Autologin"

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

2.4

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

"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 automatic:**
Document and part descriptions for all components are created in PRO.FILE automatically without query. File names and properties can be configured to be transferred automatically into specific PRO.FILE fields.
- **Save as new version:**
Saves the currently active CAD document as a new version in PRO.FILE.
- **Save NDF:**
With this function, a neutral data format (e.g. tiff, pdf) is created from the CAD document and saved as new document in PRO.FILE. This NDF document is automatically linked to the part master record of the drawing.
- **Save as Xref:**
Saves the currently active CAD document as Xref in PRO.FILE.
- **Save as block:**
Saves a block in PRO.FILE.

- **Version structure:**
When this function is used all structure elements, starting from the top element are versioned and switched to the new version.

"Edit":

- **Update document:**
Via this function, the bill of materials, the modification list and the title block on the drawing are updated. This requires the lists and fields to be properly configured for the used drawing template.
- **Insert bill of materials:**
If you want to display the bill of materials saved in PRO.FILE on the CAD drawing, you can use this function.
- **Insert more...:**
Below this menu entry you find the following commands to insert AutoCAD elements from PRO.FILE into the current CAD document:
 - Insert Xref (M1:1)
 - Insert Xref (other scale)
 - Reload Xref
 - Insert Block (M1:1)
 - Insert Block (other scale)
 - Insert raster image
- **Create bill of materials:**
This function creates a bill of materials, which is saved in PRO.FILE, based on the current CAD structure. If a bill of materials exists already, it is updated.
- **Disconnect relation:**
This function deletes the database relation of a CAD document to PRO.FILE. The CAD data are then treated as a locally saved CAD document and no longer have a PRO.FILE relation.
- **Delete layout in PRO.FILE:**
If layouts are saved in PRO.FILE for an AutoCAD model, these layouts can be deleted from PRO.FILE.
- **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 AutoCAD 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 AutoCAD, can be unlocked, so that other users can also edit the document.

"Show":

- **List of documents:**
This functions opens the special documents list and displays the configured information on the current part, the drawing or the assembly, including all linked objects.
- **Part form:**
Switches to PRO.FILE and displays the description form for the active part.
- **Structure of the parts:**
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 bill of materials, the part position, the documents structure, the document form, the document usage or all document versions for the active part.

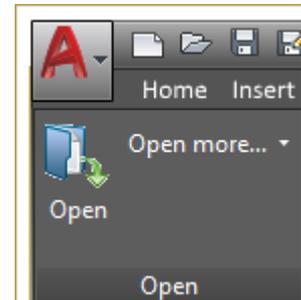
"Extras":

- **Workcenter:**
All files loaded or saved via the PRO.FILE integration in AutoCAD 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.

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

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:



From PRO.FILE:

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

From within the integration:

- [Open: Loading documents from PRO.FILE into AutoCAD](#)
- [Working with the Checkout wizard](#)
- [Open with released and newest versions of linked CAD documents](#)
- [Attention: Opening of locally existing files](#)



Attention:

The data loaded from PRO.FILE in AutoCAD are not automatically locked when opened in AUTOCAD. 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 ["Unlock" the CAD document](#)



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 data from PRO.FILE for editing

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

- Double-click on a file in a list or browser display
- Select the file and open it via the "Edit" icon in the menu tab "Edit file".
- Select the file and open the function "Edit file" => "Edit document" from the context menu.

If a CAD document is opened in one of the above ways, the setting of the parameter "Version load options dialog" determines the further procedure. If the document is opened via the "Modify" icon or via the context menu function, the CAD file is automatically locked and cannot be edited by another user.

3.2 Open: Loading documents from PRO.FILE into AutoCAD

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

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

To open an AutoCAD document from PRO.FILE proceed in 3 steps:

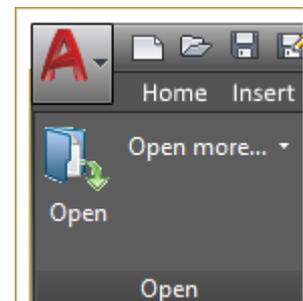
Step 1

Use the PRO.FILE function "Open"



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

1. Go into the menu bar of AutoCAD into the menu "PRO.FILE".
2. Select the menu entry "Open".
 - "Open" loads documents as they were saved the last time in PRO.FILE.



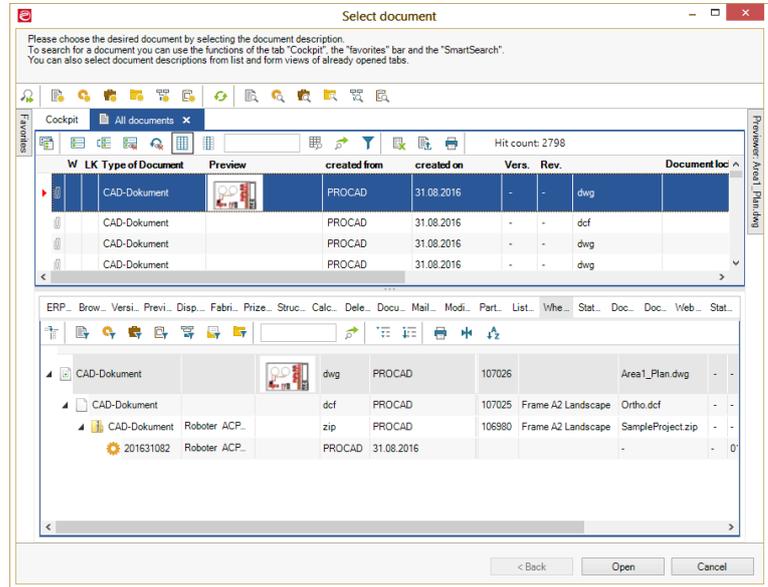
⇒ The Checkout wizard for the selection of documents is displayed.

Step 2

You now have to 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. 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](#)".

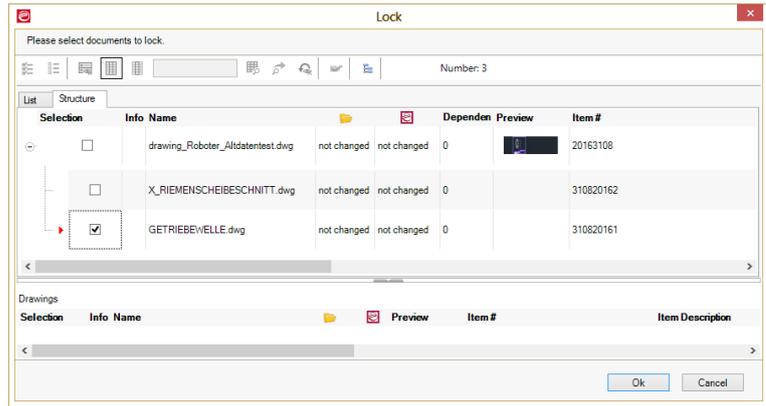
Step 3

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.

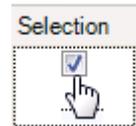
This means: If you want to edit the CAD document, you have to lock it.

The "Lock" dialog 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](#)")



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



⇒ The selected documents and its components are now opened in AutoCAD. The process of opening a document is now finished.

Detailed information on the locking of documents can be found in the chapter "[Lock/Unlock](#)".



Note: Why can you 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](#)

3.2.1 Working with the Checkout wizard

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

The aim of this procedure is:

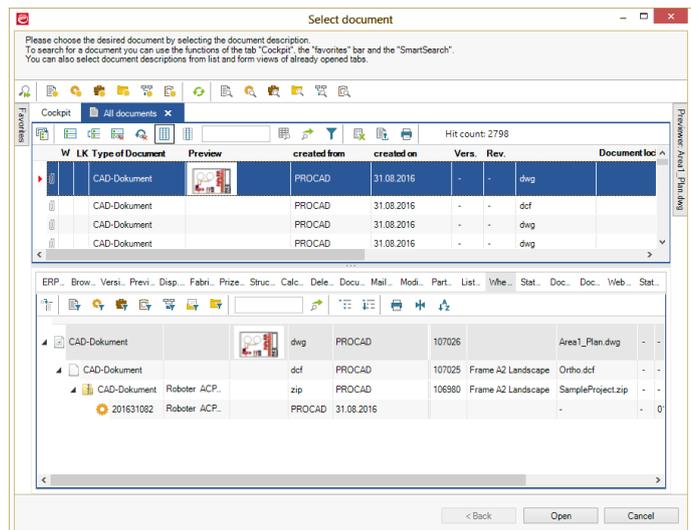
- For PRO.FILE to know which document is to be opened, the desired document description must
 - either be selected in a list view or a structure browser
 - or be displayed in a form view
- Then the button **<Open>** at the right bottom of the Checkout wizard has to be clicked.

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

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

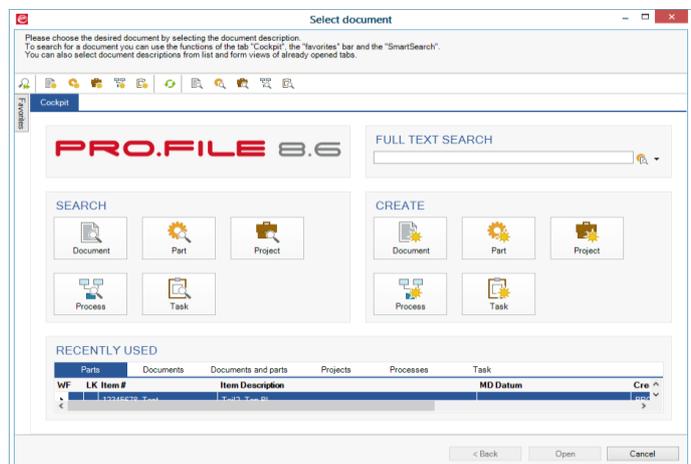
- If search results had previously been opened in a list or form view in a separate tab, you can directly access the displayed documents.

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



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

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



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

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

Because a double click means: Open document for viewing!

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

Searching

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

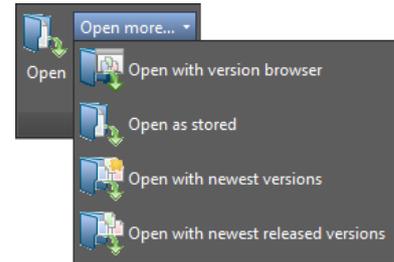
- **Searching via the tab "Cockpit"**
The same icons as in the icon bar can be found on the tab "Cockpit": "Search document", "Full-text search", "Search part", "Search project" have the same function as the icons in the icon bar.
You can always go back to the tab "Cockpit".
- **Search via the functions 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 newest versions
- Open with newest released versions



Note:

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

This means:

- **Open as 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 versions**
If the CAD document contains links to other CAD documents in PRO.FILE, the newest versions of these linked CAD documents are loaded.
When the function "New versions" is used for opening an assembly, PRO.FILE checks, whether the assembly contains components for which versions exist. If this is the case, the newest visible version of such a document is loaded in the AutoCAD session.
- **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 "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 AutoCAD session.

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



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 version browser

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

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

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

The version browser is divided into two areas:

The document structure (top)

- In the upper structure windows the selected CAD document is displayed with all attached components.
- Via the button **<Expand all>** you can display the entire structure of the part to be opened.
- The field **"Loaded"** shows the current opening type of the CAD elements displayed in the structure window – without manual version selection. The opening type affects the display of these elements:

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

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

The version window (bottom):

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



Function call:

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

Proceed as follows:

1. Select the "PRO.FILE" menu from the menu bar in AutoCAD.
2. Select the function "Open with version browser".
=> The Checkout wizard is displayed.
3. Select the desired CAD document and click on the **<Open>** button.

- ⇒ The Checkout wizard closes.
- ⇒ 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.
- 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.
- ⇒ Until now, 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](#)".
- 8. Confirm your selections with **<OK>**.
- ⇒ The selected CAD components are opened in AutoCAD. The process of opening with the version browser is now finished.

3.5

Attention: Opening of locally existing files

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

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



Attention:

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 dialog will inform you that "**Locally existing files**" were found. You can now select, which of the locally existing files are to be loaded anew from PRO.FILE and thus be overwritten.

The list also shows whether files have been changed locally and thus no longer match the document loaded from PRO.FILE. The list also shows version differences.

You now have to options:

- By selecting the corresponding files and confirming the action, all files are copied to your work folder.
- You can activate a different work folder via the command "PRO.FILE" => "Extra" => "Workcenter" => "Activate" in order to prevent the overwriting of the files.



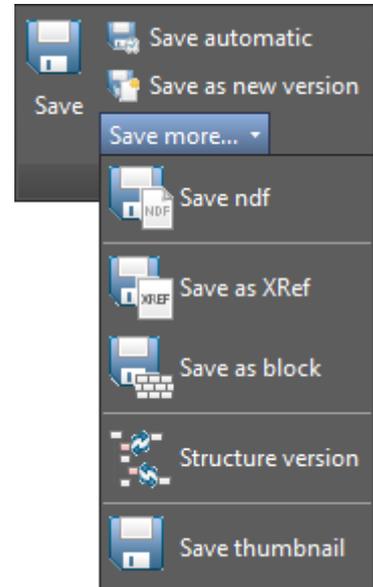
Note:

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 Saving CAD documents to PRO.FILE

The following functions are available for saving CAD documents for the first time or from the Workcenter to PRO.FILE:

- [Save: Saving CAD objects for the first time](#)
- [Save automatic](#)
- [Save as new version](#)
- [Save NDF](#)
- [Save as XRef / Save as Block](#)
- [Structure version](#)



The basic procedure of saving differs according to whether the CAD data are saved in PRO.FILE for the first time, or whether existing documents are opened for editing and saved back to PRO.FILE.

The description is therefore divided into two chapters:

- [Save: Saving CAD objects for the first time](#)
- [Resaving CAD objects in PRO.FILE](#)

The description of the procedure and the possibilities can be found in the following chapters.

Additional information on the saving of files and the usage of the Checkin wizard can be found in the manual "PRO.FILE for Advanced – Create, manage and change objects".



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

When using the integration PRO.FILE – AutoCAD, 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.

4.1 Save: Saving CAD objects for the first time

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

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



Note:

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

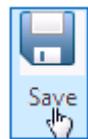


Function call:

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

Proceed as follows

1. Select the menu "PRO.FILE" from the menu bar.
 2. Click on the "Save..." button.
- ⇒ The Checkin wizard is displayed, which will support you in the proper saving of your document.



Saving of new objects in PRO.FILE takes place in several steps:

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

These steps are described in the following sub-chapters.

4.1.1

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

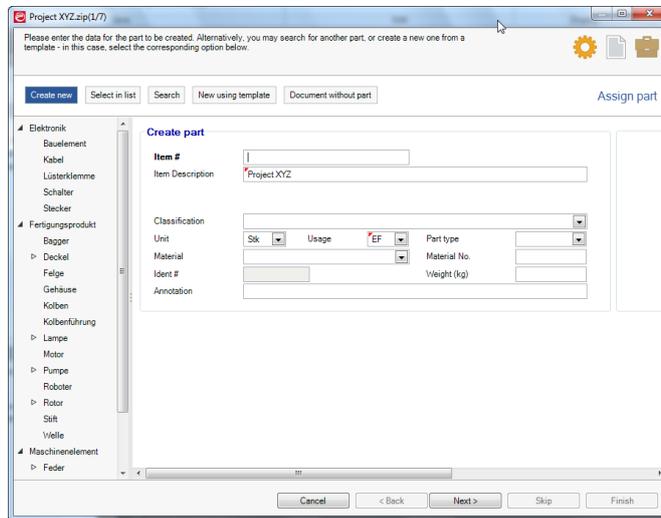
By default, every CAD document in PRO.FILE is linked to a part master record. The part master record consists of attributes and is used for the creation of bills of materials, for the display of data in the drawing title block, for transfer to an ERP/PPC system, etc.



Note: Usage of PRO.FILE parts

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

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



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

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



Create new

Usage:

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

Proceeding:

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

Select in list



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



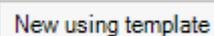
Usage:

- 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



Usage:

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

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

Proceeding:

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

Document without part

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.

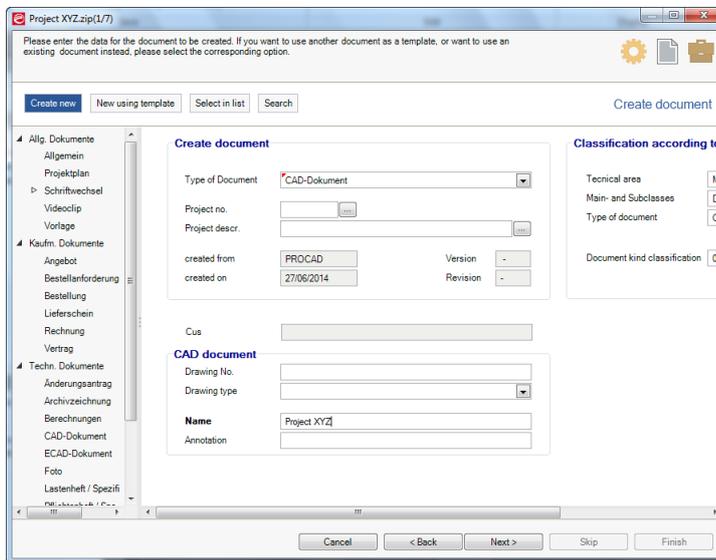
4.1.2

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

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

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

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



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

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



- Create new
- New using template

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

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

- After the finalization of your entries confirm the saving of the CAD document and the assignment to the desired part master record with <Next>.
- The CAD document is now saved in PRO.FILE.

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

4.1.3

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

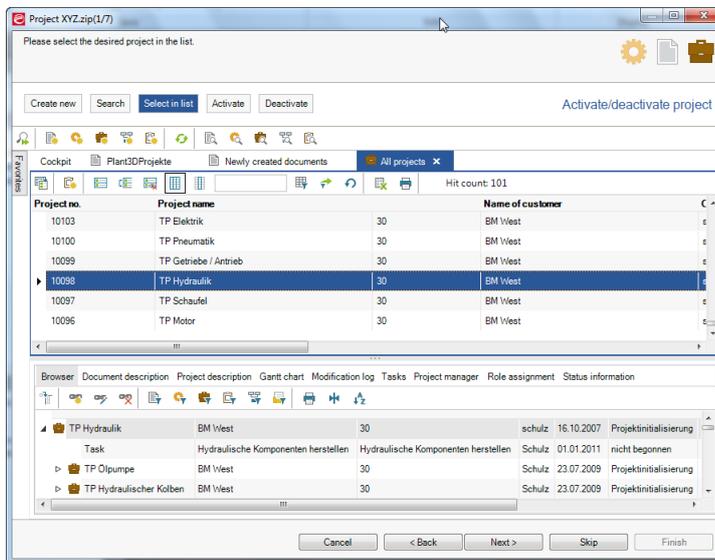
In this step the CAD data just saved can be assigned to a specific PRO.FILE project.



Note: Usage of PRO.FILE projects

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

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



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

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



**Attention: Project must be activated**

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

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

Create new:

A 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.

[Create new](#)**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.

[Search](#)**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.

[Select in list](#)**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.

[Activate](#)**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.

[Deactivate](#)

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.

4.2 Resaving CAD objects in PRO.FILE

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:

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

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

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

- "Save" =(Modify)
- "[Save as new version](#)" (see following chapter)
- "[Save automatic](#)" (see following chapter).



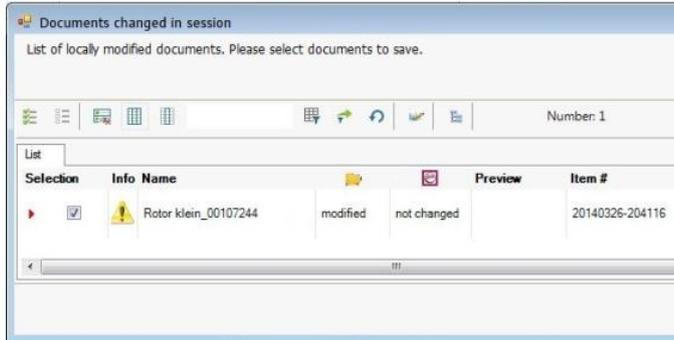
Function call from the PRO.FILE menu in AutoCAD:

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

Proceed as follows:

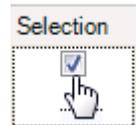
1. Go to the integration menu "PRO.FILE" in AutoCAD.
 2. Select the function "Save" from the area "Save".
- ⇒ PRO.FILE recognizes the CAD document as a PRO.FILE object and automatically goes into change mode.

- ⇒ The dialog for the selection of CAD documents to be saved is displayed. (Information on the functions and status information can be found in the chapter "[The document list](#)").



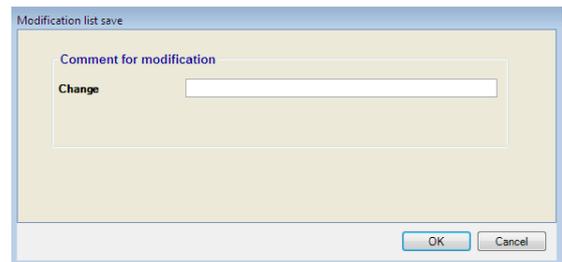
- ⇒ The dialog displays a list with all changed CAD documents from the current AutoCAD session.
- ⇒ For assemblies, the structure is analyzed for changed CAD documents and the list of all documents of this assembly is preselected.
- ⇒ For this list the access permissions for saving the changes of the 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>.



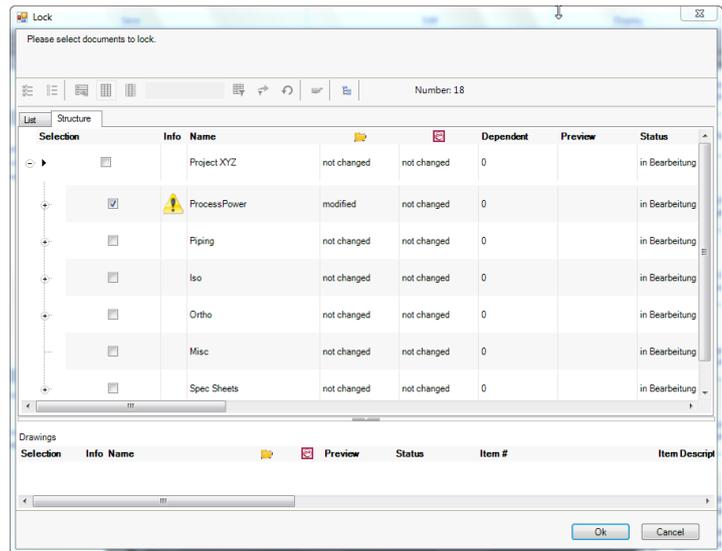
- ⇒ The changes are then saved. The CAD document in PRO.FILE is thus overwritten.

5. 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.



6. 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.

7. 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 AutoCAD, 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.



Note: Further changes after unlocking

Please note that if you want to make further changes to your CAD document, you have to lock them again, as described in the chapter "[Starting your changes: "Lock" the CAD document](#)".

4.3 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:

- This function allows the automatic creation of documents and parts in PRO.FILE.
- Descriptions (metadata, classifications) have to be entered for the first document and part in PRO.FILE. All sub-documents and sub-parts are then created automatically in PRO.FILE.
- The difference to the function "Save" is that no query for part and document information is made after the first document and part.



Function call from the PRO.FILE menu in AutoCAD:

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



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.



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).

4.4 Save as new version

With the PRO.FILE AutoCAD 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 AutoCAD 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 AutoCAD.
- Within an assembly the reference is switched to the new version.

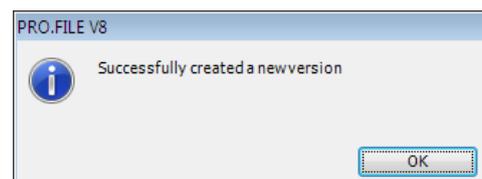


Function call from the PRO.FILE menu in AutoCAD:

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

Proceed as follows:

1. Go to the integration menu "PRO.FILE" in AutoCAD.
 2. Select the function "Save as 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 AutoCAD.



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/Unlock](#)".

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.



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

For detailed information on the versioning concept of the integration, please see the manual "[CAD design supported by PRO.FILE](#)".

4.5 Save NDF



Note:

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.

The integration PRO.FILE AutoCAD offers the possibility to convert an AutoCAD 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.



Function call from the PRO.FILE menu in AutoCAD:

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

- By using the function "Save NDF" a neutral format document is created from the CAD drawing.
- For this NDF document, a document description is created in PRO.FILE and automatically linked to the document description and/or part description of the drawing.

CAD-Dokument	104010	Zeichnungsrahmen	Test	dwg	PROCAD	12.05.2011	Prototyping	2D-Zeichnung	dwg	-	-
NDF-2D	106152				PROCAD	24.08.2011	in Prüfung			-	- workflow200



Attention: Function only available with PRO.FILE Format Generators

The function <Save NDF> is only available in connection with the PRO.FILE Format Generators. Furthermore, specific installations (converter) and configurations may be required on your system.

4.6 Save as XRef / Save as Block

The functions "Save as XRef" and "Save as Block" of the integration offer the possibility to define Xrefs and blocks, which can also be used in other drawings after the saving in PRO.FILE.



- "Save as Block" (comparable to the AutoCAD function wblock)
- "Save as Xref" (comparable to the AutoCAD function wblock)

Proceed as follows:

1. Select the "PRO.FILE" menu in AutoCAD.
2. Select the function "Save as XRef" or "Save as Block" from the function area "Save"
- ⇒ The PRO.FILE Checkin wizard is displayed.
3. Save the document description and, if required, also the part description in PRO.FILE, as described in the previous chapter ["Save: Saving CAD objects for the first time"](#).
4. In AutoCAD you can now specify the insertion point of the Xref / block.
5. For blocks, a window is now displayed, in which all elements that are to belong to this block can be selected. Please note that only those elements are accepted, that are fully within the window.

The Xref / the block is stored in the database and can be loaded from PRO.FILE with the function "Insert XRef" or "Insert Block".

- The saved block in PRO.FILE is "cleaned", i.e. all unused layers, line types, etc. are removed.
- An Xref/block saved in this ways remains in the active drawing.

4.7 Structure version

When the function "Structure version" is used, Xrefs in the main drawing are checked for newer versions and updated. After that, the new constellation of the main drawing is saved as a new version.



Function call from the PRO.FILE menu in AutoCAD:

"PRO.FILE" => "Save" => "Structure version"



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

For detailed information on the versioning concept of the integration, please see the manual "[CAD design supported by PRO.FILE](#)".

4.8 Save thumbnail

If a thumbnail preview image is to be used in PRO.FILE document lists, this thumbnail can be updated with the function "Save thumbnail".

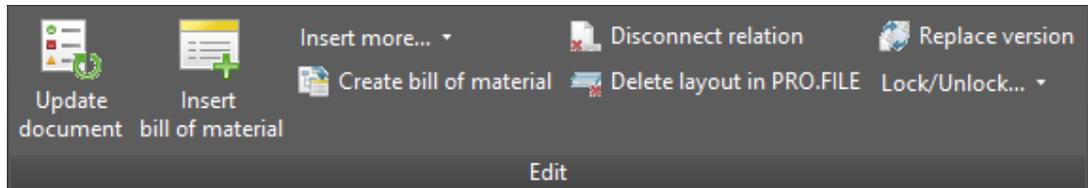


Function call:

"PRO.FILE" => "Save" => "Save more..." => "Save thumbnail"

5 Additional functions of the integration

The integration menu "Edit" offers various different functions for the editing of your documents.



These functions are described in the following sub-chapters:

- [Fill out title block](#)
- [Insert bill of materials](#)
- [Insert more: XREF, Block, Raster image](#)
- [Create bill of materials](#)
- [Disconnect relation](#)
- [Delete Layout in PRO.FILE](#)
- [Replace version](#)
- [Lock/Unlock](#)

Furthermore, you have access to optional commands that can be included into the menu:

- [Optional functions](#)
 - [Automatic Lock / Unlock](#)
 - [Open / Save and automatic Lock / Unlock](#)

5.1 Fill out title block

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



Function call from the PRO.FILE menu in AutoCAD:

```
"PRO.FILE" => "Edit" => "Fill out title block"
```

After this function is used, all contained elements of the drawing legend (title block, modification list, bill of materials) is filled with data from PRO.FILE.

Within the modification list, always the most recent entries are listed.

This function does not require any further interaction from the user.

5.2 Insert bill of materials

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

To be able to insert a bill of materials from PRO.FILE, the following requirements have to be met:

- The drawing must be created with Xrefs or blocks in order to create the bill of materials automatically.
- The external references and blocks must be inserted from PRO.FILE.
- The bill of materials must be created in PRO.FILE.



Function call from the PRO.FILE menu in AutoCAD:

```
"PRO.FILE" => "Edit" => "Insert bill of materials"
```

Proceed as follows:

1. Select the function "Insert bill of materials" from the "Edit" area of the PRO.FILE menu in AutoCAD.
2. You can now place the bill of materials on your drawing by entering a position. If a PRO.FILE bill of materials already exists on the drawing, this function updates the bill of materials.

5.3 Insert more: XREF, Block, Raster image

With the menu entry "Insert more..." AutoCAD-specific elements Xref, blocks and raster images can be inserted from PRO.FILE into your AutoCAD drawing.



Function call from the PRO.FILE menu in AutoCAD:

```
"PRO.FILE" => "Edit" => "Insert more..." => ...
```

For detailed information please see the following sub-chapters:

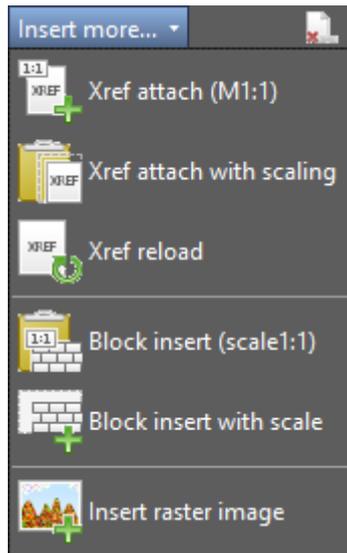
- [Xref attach...](#)
- [Blocks...](#)
- [Insert raster image](#)

5.3.1 Xref attach...

If a drawing is composed from several other part drawings that are subject to change, you can link these part drawings as external references. Therefore you can use the function "PRO.FILE" -> "Xref attach...".

If the main drawing is loaded once more later with "open", all detail drawings also are loaded newly. If changes were carried out at a detail drawing, then these are also visible in the main drawing..

The user has two possibilities of inserting Xrefs in an existing drawing:



- "Xref attach (M1:1)" -> in the original scale
- "Attach with scaling" with scaling factors for the X and Y direction
- "Reload" The external references in the drawing that are no longer up to date, are extracted from PRO.FILE, and re-loaded. Before this is carried out, the drawings that are no longer up to date will be listed so that they can be selected.



Note:

The usage of the function "Xref attach with scaling" allows the entering of scaling factors X, Y and Z in the AutoCAD command line.

After saving a drawing, all PRO.FILE blocks inserted with the XRef -> Attach 1:1 or the Attach Other Scale function are compiled into a document structure.

Upon reloading the drawing, all files in the document structure are copied to the local hard drive. This assures that AutoCAD will be able to find referenced blocks and that the drawing can be edited as desired.

The directory to which the files are copied is determined by the TEMPDIR CLICONFIG variable. This directory will include subdirectories for each drawing. You can edit and work with references (i.e. move copy, etc.) in the same way you usually do in AutoCAD. However, they may not be modified.

The AutoCAD XRef Bind command allows for including externally referenced blocks in the drawing. They then aren't listed in the structure anymore!

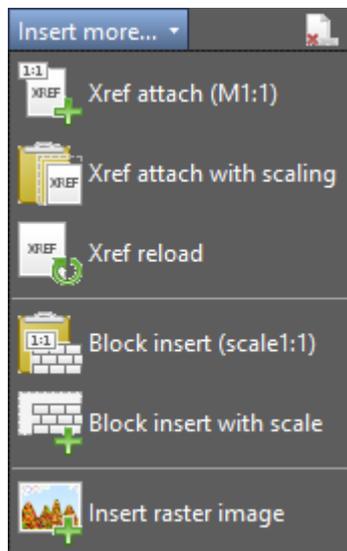


Note:

Modifications performed on XRefs must be stored to PRO.FILE.
 If XRefs or parts from PRO.FILE have been assigned to/inserted into the drawing, PRO.FILE creates a document structure when saving the drawing.
 If an XRef or part has been embedded or removed again and the AutoCAD command "purge" has been used to remove all unused XRefs/parts, the drawing will show the correct document structure after the next saving process, since the document structure is always checked and refreshed during saving processes. The document structure can be viewed by the user (select document; PRO.FILE menu "Show" => "Document structure").

5.3.2 Blocks...

The integration PRO.FILE – AutoCAD offers the possibility of working with blocks (mainly used in 2D) and objects (in 3D).



The user has two options of inserting a block into an existing drawing::

- "Block insert (scale 1:1)" -> original scale
- "Block insert with scale" with scaling factors for the X and Y direction



Function call from the PRO.FILE menu in AutoCAD:

"PRO.FILE" => "Edit" => "Insert more" => "Block insert..."

Proceed as follows:

1. Go to the PRO.FILE menu in AutoCAD and select the function "Insert more..." => "Block insert...".
 => The Checkout wizard is displayed.
2. Select the block to be inserted into the drawing. It is not relevant whether this part has previously been saved as block or as drawing.
 (For detailed information on the Checkout wizard see chapter "[Working with the Checkout wizard](#)").

3. After selecting the block to be inserted you are prompted to enter an insertion point (via keyboard or mouse) and to specify a rotation angle.

⇒ The block is inserted into the drawing.

For blocks inserted this way, the insertion point lies on the root of the block, unless it has been pre-defined during the creation of the drawing with the AutoCAD command "basis".



Note:

The usage of the function "Block insert with scaling" allows the entering of scaling factors X, Y and Z in the AutoCAD command line.

5.3.3

Insert raster image

In AutoCAD it is possible to include raster images in a drawing. When a drawing with raster images is saved, the images are included in the data structure of the document, similar to referenced drawings. The raster images are then copied into the work folder with their original names and are then visible in the AutoCAD drawing.

- These raster images can be managed in PRO.FILE.
- With the function "Insert raster image" it is possible to include images (e.g. .gif, .tif or .bmp files) that are stored in PRO.FILE into the CAD drawing.

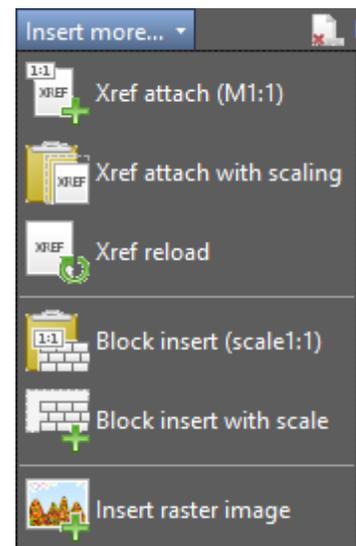


Function call from the PRO.FILE menu in AutoCAD:

"PRO.FILE" => "Edit" => "Insert more..." => "Insert raster image"

Proceed as follows:

1. Go to the "PRO.FILE" menu in AutoCAD.
 2. Select "Edit" => "Insert more..." => "Insert raster image".
- ⇒ The PRO.FILE Checkout wizard is displayed.
3. First, select the drawing, into which the raster image from PRO.FILE is to be inserted. (For detailed information on the Checkout wizard see chapter "[Working with the Checkout wizard](#)").



4. When the desired drawing is selected in the Checkout wizard, confirm this with **<Open>**.
- ⇒ The Checkout wizard is displayed again.
5. Select now the raster image to be inserted, e.g. in .gif, .tif or .bmp format.
6. When the desired raster image is selected in the Checkout wizard, confirm this with **<Open>**.
- ⇒ You get back to your AutoCAD session. The image can now be placed.



Note

It has to be taken into account that pixel pictures can be printed only on corresponding printers. HPGL plotters are not able to do this. This also has to be taken into when creating a TIFF-Document using the function "Save NDF" or the Generic Job Server.

5.4 Create bill of materials

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



Note:

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

- The drawing must be created with Xrefs or blocks :
- The external references and blocks must be inserted from PRO.FILE.

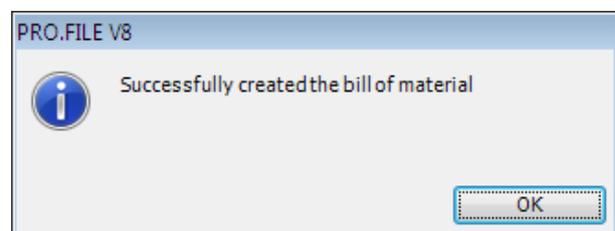


Function call from the PRO.FILE menu in AutoCAD:

"PRO.FILE" => "Edit" => "Create bill of materials"

Proceed as follows:

1. Go to the "PRO.FILE" menu in AutoCAD.
 2. Select "Edit" => "Create bill of materials".
- ⇒ The bill of materials is created from the currently active AutoCAD document and saved in PRO.FILE.
- ⇒ A message confirms the successful creation 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:

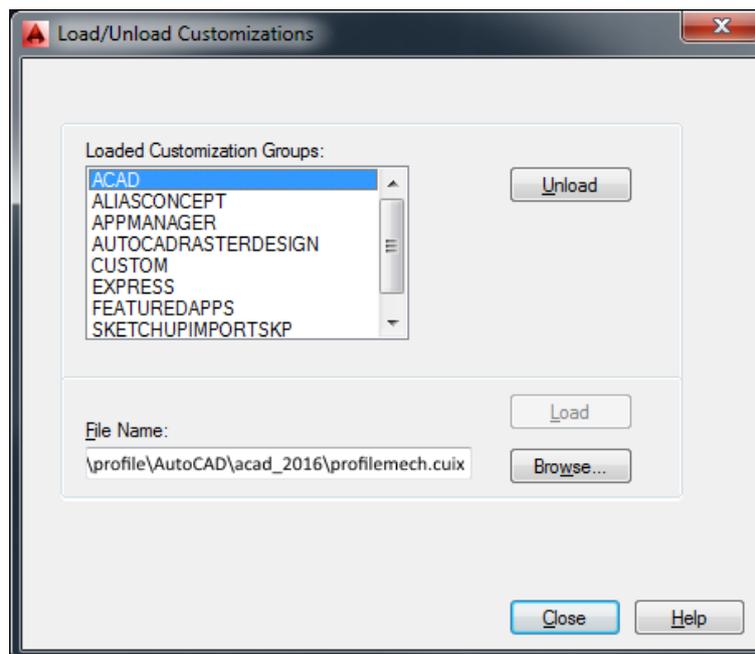
Please note that norm parts or auxiliary materials (e.g. water or oil) that are not displayed on the drawing are not transferred into the bill of materials, when the function "Create bill of materials" is used.

You can add these parts to the bill of materials in PRO.FILE manually.

5.5 Create bill of materials (AutoCAD Mechanical)

If you want to create a bill of materials from AutoCAD Mechanical and run the corresponding functions, this has to be configured by your administrator before (see manual "Configuration Integration PRO.FILE AutoCAD"). Once this is done, you have to load the specific AutoCAD Mechanical menu of the integration:

1. Open AutoCAD. The integration is loaded as well.
2. Enter the command "_cuiload".
3. Load the file "profilemech.cuix" from the installation folder of the AutoCAD integration.

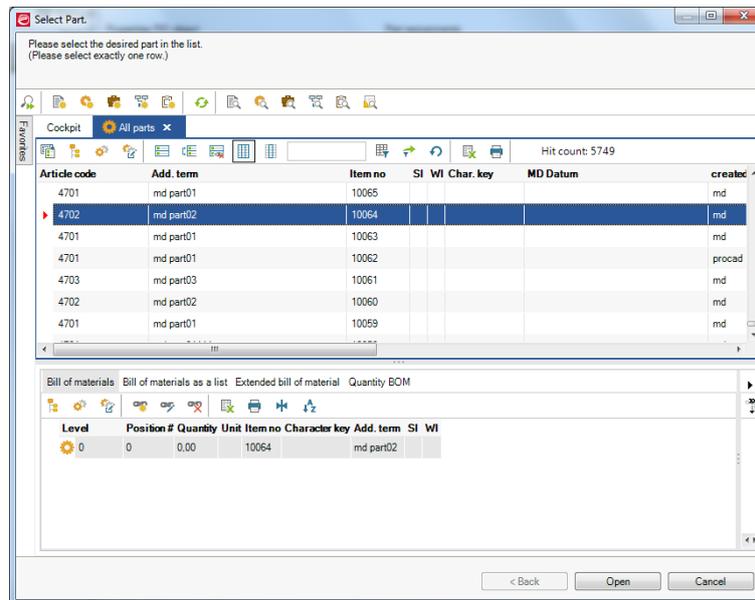


⇒ The menu "Part reference" is available in the PRO.FILE tab of the integration.

5.5.1 Create part reference

Via the function "Part reference" => "Create", an AutoCAD part reference is created with an assignment to the PRO.FILE part master record. After the selection of the function proceed as follows:

1. Select the corresponding position in the CAD system.
2. Select a PRO.FILE part master record to be assigned.



- ⇒ An AutoCAD part reference is created and the key value of the PRO.FILE part master record is entered in the index field of the AutoCAD part reference (according to the configuration – see manual "[Configuration Integration PRO.FILE AutoCAD](#)").
- ⇒ Furthermore, additional parameter can be loaded via the "AutoCAD title block" configuration.

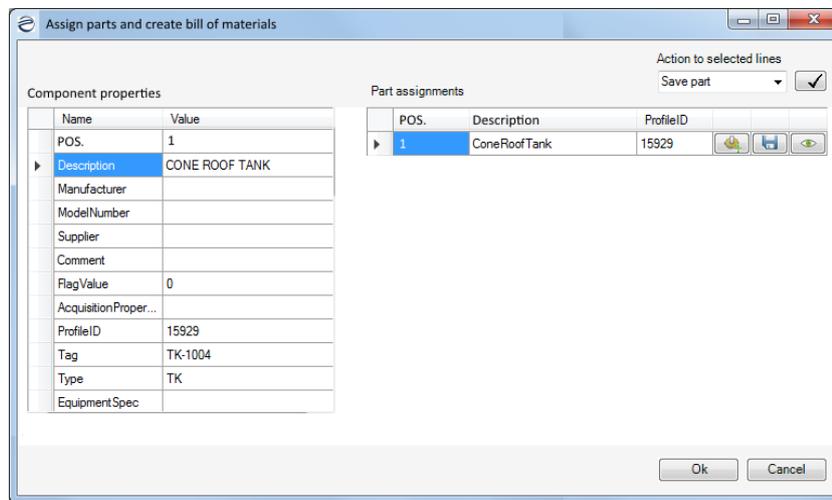
5.5.2 Assign part reference

Via the function "Part reference" => "Assign", existing AutoCAD part references can be assigned to PRO.FILE part master records. After the selection of the function proceed as follows:

1. Select an AutoCAD part reference.
 2. Select a PRO.FILE part master record.
- ⇒ The key value of the PRO.FILE part master record is entered in the index field of the AutoCAD part reference (according to the configuration – see manual "[Configuration Integration PRO.FILE AutoCAD](#)").
 - ⇒ Furthermore, additional parameter can be loaded via the "AutoCAD title block" configuration.

5.5.3 Create bill of materials

Via the function "Create bill of materials", the AutoCAD Mechanical bill of materials (not the parts list) is loaded and displayed in the following form:



The integration searches for existing part master records in PRO.FILE and – if successful – shows the corresponding PRO.FILE part record ID (ProfileID).

The second column contains the AutoCAD part reference value.

Via the buttons "Assign part" and "Save part", the assignments can be made (if this has not already been done).

The following actions are allowed for selected positions:

- Save part
- Detach part
- Show part

Once all part assignments have been made, the bill of materials can be created via the <OK> button.

5.6 Disconnect relation

This function dissolves the connection of the selected CAD drawing and the document description in PRO.FILE. The drawing thus no longer has a PRO.FILE connection. The CAD document is now treated as a locally saved CAD file and marked accordingly in the Workcenter.

- The current drawing can be changed in AutoCAD and saved again in PRO.FILE with a new name and document ID.
- The "old" drawing still exists in PRO.FILE. The function "Disconnect relation" does not delete any documents.



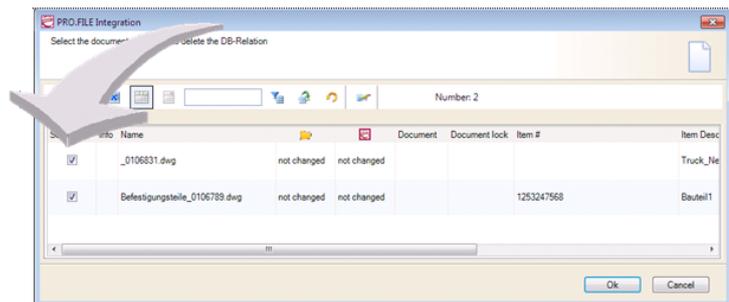
Function call from the PRO.FILE menu in AutoCAD:

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

Proceed as follows:

1. Go to the "PRO.FILE" menu in AutoCAD.
2. Select "Edit" => "Disconnect relation".

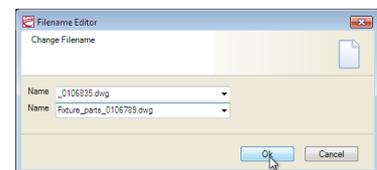
⇒ The dialog for the selection of documents to be disconnected is displayed. (Information on the functions and status information can be found in the chapter "[The document list](#)").



3. Select all CAD documents you want to disconnect from PRO.FILE. To do so, activate the checkboxes for the desired documents. Confirm your selection with <OK>.



⇒ The file name editor is displayed. By the disconnection of the document from PRO.FILE, the CAD documents are saved locally in your work folder and therefore require new file names. You can assign these names manually and save the CAD documents with the new names.



4. Enter the desired file names and confirm with <OK>.

⇒ The document connection to PRO.FILE is now successfully dissolved.

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

To save the CAD data again in PRO.FILE, use the save function of the integration as described in the chapter "[Save: Saving CAD objects for the first time](#)".

5.7 Delete Layout in PRO.FILE

If layouts are saved for a CAD model in PRO.FILE, you can use this function to delete them from the PRO.FILE database.



Attention: Risk of data loss

Deleting of a layout in PRO.FILE cannot be undone!

Proceed as follows:

1. Go to the "PRO.FILE" menu in AutoCAD.
2. Select "Edit" => "Delete layout in PRO.FILE".
⇒ After selection of this command, a list of layouts known in PRO.FILE is displayed.
3. Select the layouts you want to delete. To do so, activate the checkboxes for the desired documents.
If a layout record cannot be deleted in PRO.FILE – e.g. because it is in a released status – the layout cannot be selected in the list.
4. Confirm your selection with <OK>.

The selected layout records are now deleted in PRO.FILE.



Note:

To delete layouts in PRO.FILE, the parameter "Support layouts" has to be activated in the PRO.FILE Management Console. In this case, please contact your system administrator.

5.8 Replace version

The function "Replace version" allows to replace an existing version of a CAD document with another version of this document in all assemblies, in which it is used.



Function call from the PRO.FILE menu in AutoCAD:

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

The function "**Replace version**" detects where the currently active PRO.FILE CAD document is used, based on its document ID in PRO.FILE.

After this, it is possible to replace the old version of the active PRO.FILE document with the new version in all corresponding assemblies and drawings.

Proceed as follows:

1. Open a new version of a document from PRO.FILE.
 2. Select the function "**Replace version**" of the "Edit" are of the PRO.FILE menu in AutoCAD.
 3. You get a list of where and how often the predecessor versions of this document is being used.
 4. Select the data record, for which you want to apply the loaded version.
 5. Confirm your selection with <OK>.
- ⇒ The current version is then used by the selected assemblies/drawings.



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

For detailed information on the versioning concept of the integration, please see the manual "[CAD design supported by PRO.FILE](#)".

5.9 Lock/Unlock

If you want to edit a CAD document and save your changes back to PRO.FILE, the document has to be locked for other users before you start your changes.

- To "Lock" the document is the only way to make sure that a document is opened for editing by several users at the same time, thus making concurring changes.
- With "Unlock" the CAD drawing is made available to others again for editing.

For detailed information see the following sub-chapters:

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



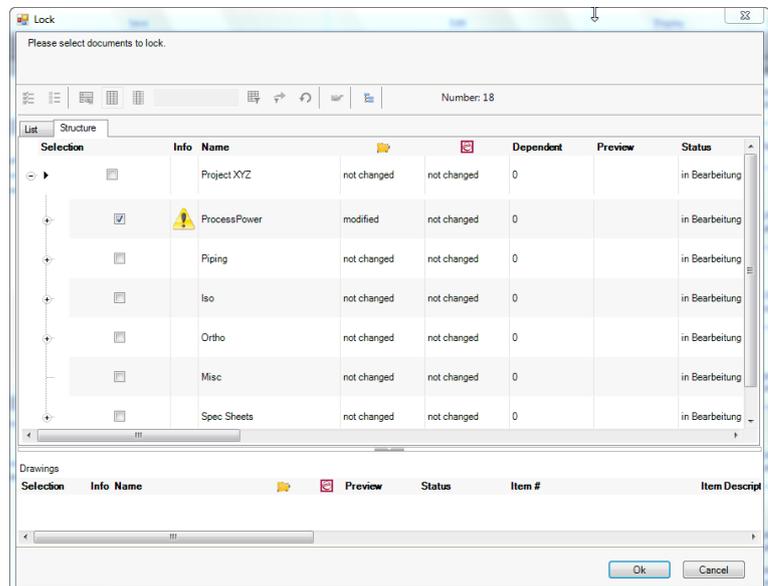
For further information please see the manual ["CAD design supported by PRO.FILE"](#).

Dynamic lock dialog

Until now, the designer was able to make local changes, without being actively or directly informed about possible conflicts with CAD documents saved in PRO.FILE.

PRO.FILE 8.6 now recognizes local changes. The integration evaluates the saving processes of the CAD system and displays a dynamic lock dialog:

- You can select the CAD documents to be locked via the checkboxes.
- With <OK> the locking is confirmed.



Displayed entries that are not selected for locking here, are afterwards not offered again for locking during the CAD session.

5.9.1 Starting your changes: "Lock" the CAD document

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



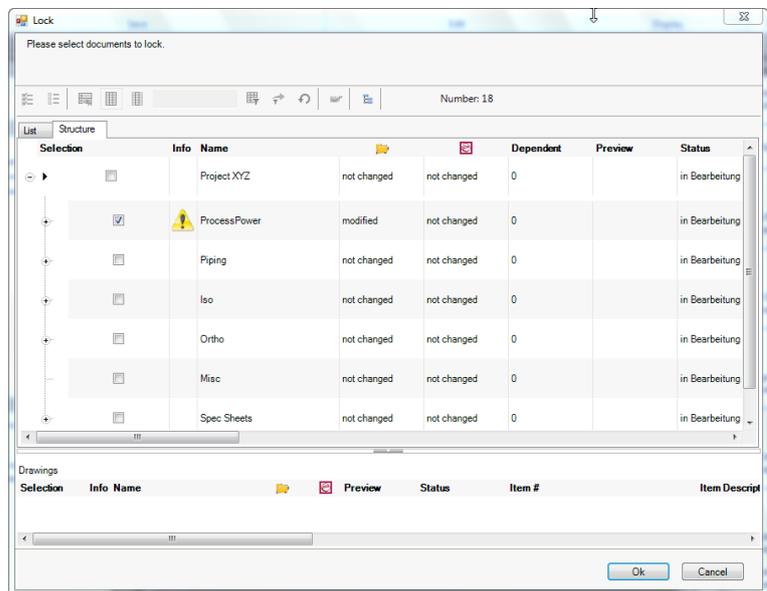
Function call:

"PRO.FILE" => "Edit" => "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 AutoCAD.
2. Go to the "PRO.FILE" menu in AutoCAD.
3. Select "Edit" => "Lock/Unlock" => "Lock".

⇒ The dialog screen for locking 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 documents you want to lock. To do so, activate the checkboxes for the desired documents.
5. Confirm your selection with <OK>.



⇒ With this "Lock" command, the access permissions of the user to edit the document are checked, and the document is locked so that other users cannot edit it.

Once the CAD document is locked, it can be edited. The changes can be saved back to PRO.FILE via the function "Save" of the integration.



Attention: Editing in the team

It is recommended to lock documents, which you open with the intention of editing, directly after opening. You must lock your CAD documents to make sure that your changes can later be saved back to PRO.FILE.

5.9.2 "Unlock" the CAD document

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



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



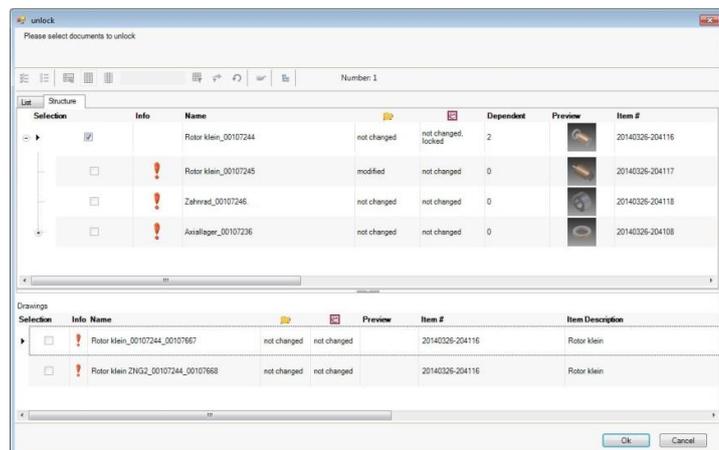
Function call
"PRO.FILE" => "Lock/Unlock" => "Unlock"

Proceed as follows:

1. Make sure that the CAD document to be unlocked is displayed in AutoCAD.
2. Go to the "PRO.FILE" menu in AutoCAD.
3. Select "Edit" => "Lock/Unlock" => "Unlock".

4. The dialog screen for unlocking the loaded CAD documents is displayed.

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



5. To make the CAD documents saved in PRO.FILE available for other users again, select the documents that you want to unlock. To do so, activate the checkboxes for the desired documents.
6. Confirm your selection with <OK>.



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

5.10 Optional functions

In addition to the commands contained in the PRO.FILE menu some further functions are at your disposal. These new commands correspond essentially to functions from the Classic Version. These aren't registered in the enclosed menu file. To use these functions, you must adapt your menu file correspondingly.

You can add the following AutoCAD commands to your menu file:

- [Automatic Lock / Unlock](#)
- [Open / Save and automatic Lock / Unlock](#)

5.10.1 Automatic Lock / Unlock

The following commands are available for the automatic locking/unlocking of documents:

- **ReserveTop**
Executes an automatic lock of the open AutoCAD drawing without query. If Xrefs are loaded, only the opened assembly drawing is locked nevertheless. If the drawing cannot be locked, a message is shown.
- **UnReserveTop**
Executes an automatic unlock of the open AutoCAD drawing without query. If Xrefs are loaded, only the opened assembly drawing is unlocked nevertheless. If the drawing cannot be unlocked, a message is shown.

5.10.2 Open / Save and automatic Lock / Unlock

To connect the functions "Open" and "Save" in PRO.FILE automatically with the lock/unlock functions, the following commands are available:

- **Checkout**
The drawing is loaded and locked automatically (Retrieve to modify).
- **CheckinTop**
The current, open drawing becomes stored and automatically unlocked

5.10.3 Save all automatic

The function "Save all automatic" is an enhancement of <Save automatic> (see chapter: "[Save automatic](#)"). For the activation of the function enter the following command

- **ProSaveAutoAll**

This function facilitates the automatic creation of documents and parts in PRO.FILE without further interaction. Only an overview of all documents opened in the session that are to be saved is displayed. This overview is for information purposes only. Assignment to a project is only made if a project is already activated in PRO.FILE.

When using this function, two cases can be distinguished:

1. "Save all automatic" for new documents that have to be saved to PRO.FILE:
 - For these documents to be saved, no Check-In wizard is displayed – not even for the first document (as with the function "Save automatic"). Document and part descriptions are created in PRO.FILE without interaction.
 - Without interaction means: The manual filling of the document or part description is not possible. The data record therefore only gets the information that is pre-filled in the "Save" form or that is automatically handed over to the form by the CAD system.
2. For documents that have been opened from PRO.FILE for editing:
 - If documents have been loaded from PRO.FILE for editing, the function "Save all automatic" will overwrite all affected files in PRO.FILE with the changed files from the CAD session.

⇒ As result of the function <Save all automatic> a part description and document description is created in PRO.FILE for each new document. After that, the structure of the assembly is created and the bill of materials is derived.



Attention: "Required fields" and <Save all automatic>

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

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

5.10.4 Save without layout support

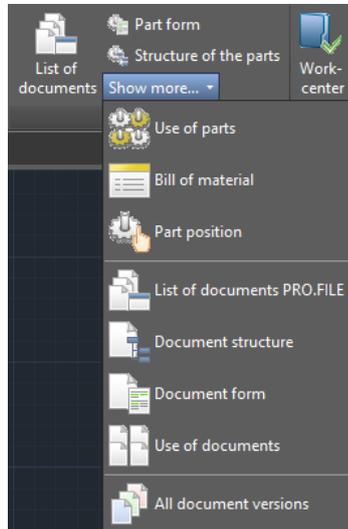
The following functions have the effect that no layout documents are created in PRO.FILE during the saving process. This is regardless of whether the MMC parameter "_5_USE_LAYOUT" is activated or not.

Apart from that these functions are not different from the standard functions "Save" and "Save automatic":

- **ProSaveNotUseLayout**
Corresponds to the function "Save" (see chapter "[Save: Saving CAD objects for the first time](#)") but with the difference that no layouts are saved.
- **ProSaveAutoNotUseLayout**
Corresponds to the function "Save automatic" (see chapter "[Save automatic](#)") but with the difference that no layouts are saved.
- **ProSaveAutoAllNotUseLayout**
Corresponds to the function "ProSaveAutoAll" (see chapter: [Save all automatic](#)), but with the difference that no layouts are saved.

6 Show: Access to PRO.FILE information

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.

6.1 The document list

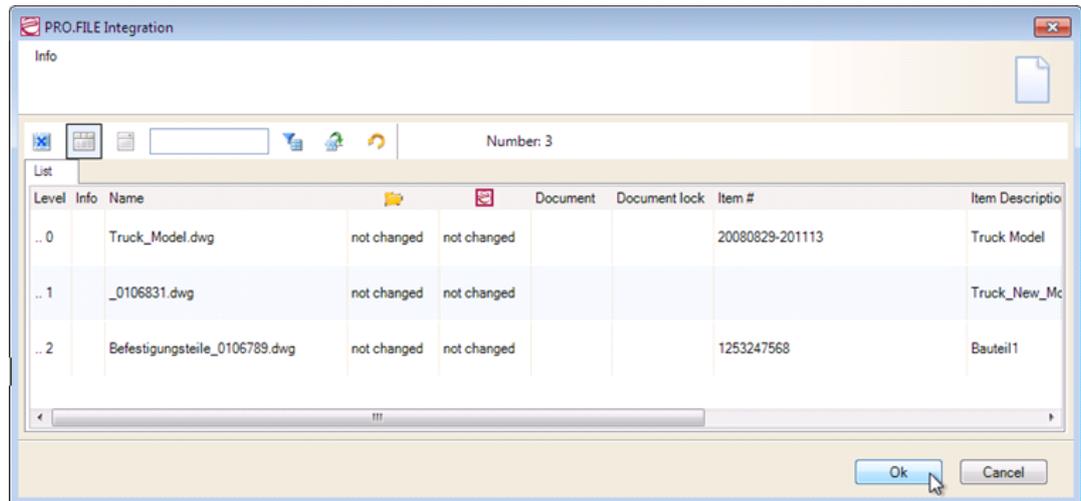
The document list displays the PRO.FILE information on the currently active CAD data. With the function "Document list" you can also see which documents (part drawings) are currently used in your (main) drawing.



Function call from the PRO.FILE menu in AutoCAD:

"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.

Detailed information can be found in the following chapters:

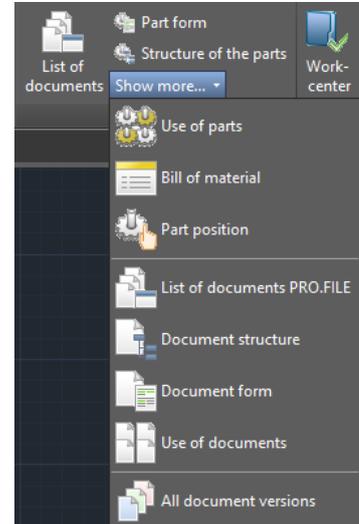
The document list also contains – as all other dialog screens of the integration do – different search and list functions.

- [More comfort: search and list functions in the dialog screens](#)
- [Up to date or not: Display of status information](#)

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

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

- These menu entries directly access the information on the CAD document currently active in AutoCAD.
- The individual menu entries allow a targeted access to the required information without having to search in PRO.FILE.
- After a function is used, PRO.FILE opens and displays the selected view.
- Within the displayed lists and forms, you can make use of all available PRO.FILE functions.



Function call from the PRO.FILE menu in AutoCAD:

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



Note:

Please note that these display functions can only be used if the CAD drawing is already known (i.e. saved) in PRO.FILE.

If you have made changes to your file locally and have not saved these changes back to PRO.FILE, information resulting from these changes is not yet visible in PRO.FILE.

The following views are available:

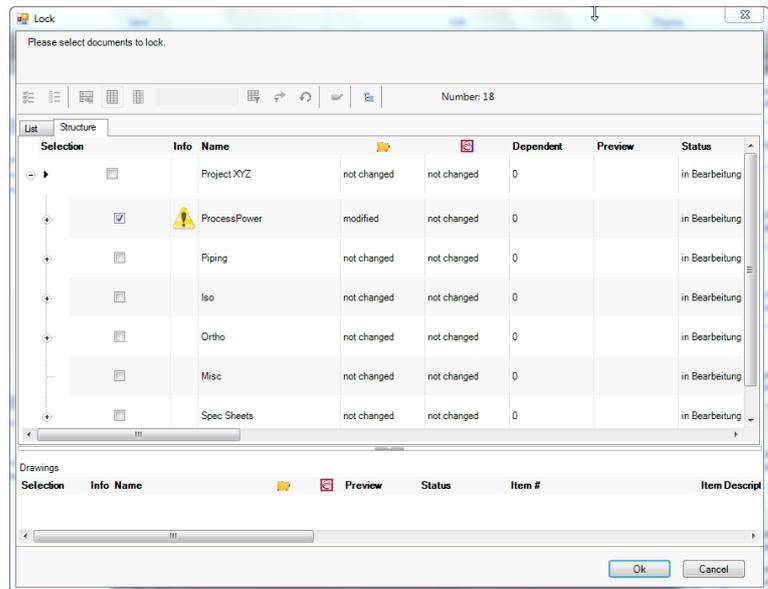
- Part form
Displays the part master record form of the current object.
- Part structure
Displays the structure overview of the current part.
- Part usage
Shows a list of objects, in which the current object is being used.
- BOM
Displays the bill of materials of the current object.

- Part position
With the function "part position" you can display the PRO.FILE part master record by clicking on a position number.
After you have selected this function, you will be prompted to select a balloon. Once you have done so, the referencing part master record is displayed in PRO.FILE.
- Document structure
Using the function "Document structure" you can see which documents (part drawings) are used within your main drawing.
- Document form
Displays the document master record form of the current object.
- Document usage
Shows a list of objects, in which the current object is being used.
- All document versions
The function "All document versions" lists the current visible and all older visible version of your CAD drawing.

6.3 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](#)").

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

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



Via these buttons, the following functions are available:

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

-  **Hide selected rows :**
If several rows of a list are selected, these rows can be hidden from the list with this button.
-   **Search in all columns / Search in active columns:**
In order to be able to perform a targeted search for terms in the list, the user first has to select whether the search is to be carried out across all columns in the list or only for a specific column in the list.
 -  : The search is performed across all columns in the list.
 -  : The search is performed for the active column only. A column is activated by clicking the respective column header.
-   **Define Filter pattern / Filter:** A character string can be entered into the entry field located within the icon bar. Here you can use the already described wildcards/meta characters.
The search for the entered character string is started using the  icon.
If the search pattern is found, all matching data records are highlighted.
-  **Next found pattern:** This icon is used to once again compare the entered filter pattern with the columns that are to be searched. The next data record found is highlighted.
-  **Show hidden rows:**
If rows of a list have been hidden, this button can be used to display them again.
-  **PRO.FILE list selection:**
The entries of the selected rows are selected and opened in a list in PRO.FILE. This way you can immediately view the stored information without further selection.

6.3.2 Up to date or not: Display of status information

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

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

These columns may contain the following:

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

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

7 Extras: The Workcenter

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



Starting the Workcenter from the PRO.FILE menu in AutoCAD:

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

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

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



Attention when working with several work folders:

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

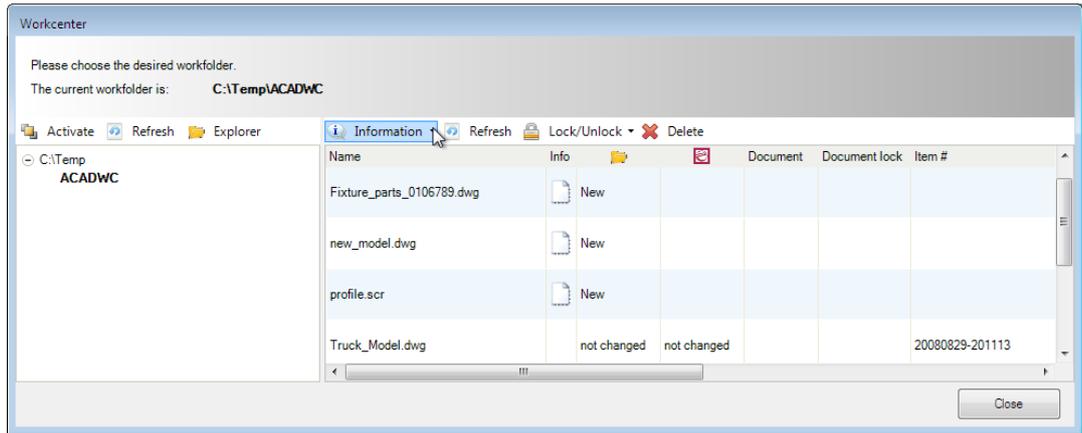
Handling of additional files

The integration PRO.FILE AutoCAD handles additional files, particularly those created by CAD add-ins for the administration of structures. These additional files are visible in the Workcenter subfolders.

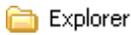
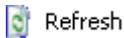
7.1 Workcenter functions

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:

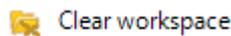
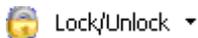


The selected folder will be used as the new working directory. The current working directory is marked in bold.

The view of the directory structure is updated.

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 Workcenter folders:



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 | |

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

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

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.

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.

The display filter for the document list can be adjusted via this icon. This can be used to facilitate the finding of objects in large folders.



Selected files can be replaced 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 the CAD system (if it is not already opened).

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