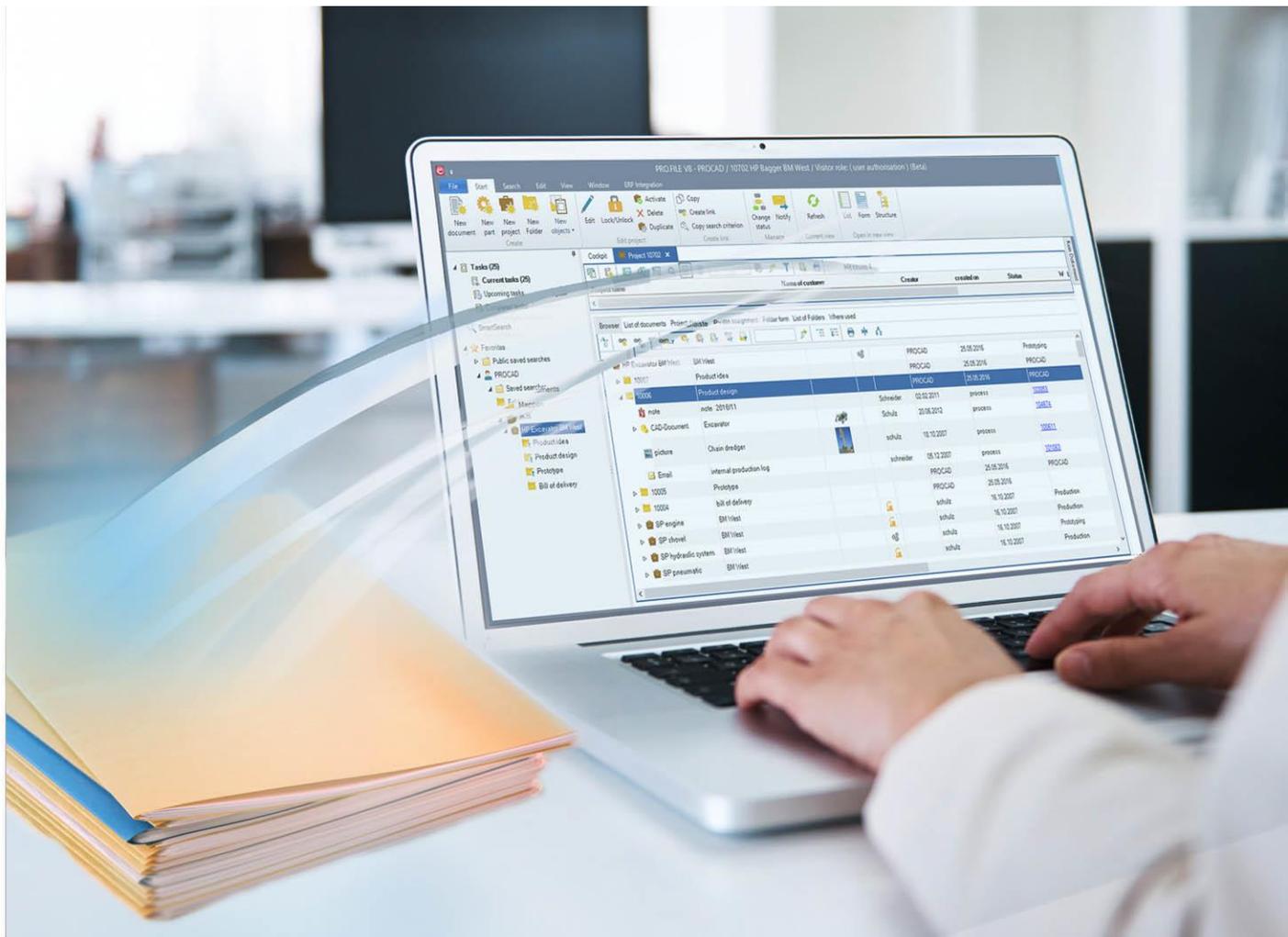


Functions of the Integration PRO.FILE AUTOCAD Plant 3D

PRO.FILE Release 8.7
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Table of contents

Table of contents	3
About this manual.....	5
1 The integration PRO.FILE – AutoCAD Plant 3D.....	6
2 Let's get started: First steps with the PRO.FILE integration.....	7
2.1 Only upon first start: Setting up the local work folder	7
2.2 Where can I find the functions of the PRO.FILE integration?	9
2.3 How to log in to PRO.FILE?.....	9
2.4 A brief overview: The functions of the integration.....	10
3 Functions for opening CAD documents from PRO.FILE in AutoCAD Plant 3D	14
3.1 Open: Loading documents from PRO.FILE into AutoCAD Plant 3D.....	15
3.1.1 What happens at the file level when a plant project is opened?	18
3.1.2 Working with the Checkout wizard to search for CAD documents	18
3.1.3 Open sub-projects and drawings.....	21
3.2 Update	21
3.3 Open with newest and released versions of linked CAD documents	22
3.4 Open with version browser.....	24
3.5 Attention: Opening of locally existing files	26
4 Saving CAD documents to PRO.FILE.....	27
4.1 Save Plant project: Saving CAD objects for the first time.....	29
4.1.1 Checkin wizard Step 1: Creating or assigning a part master record in PRO.FILE	30
4.1.2 Checkin wizard Step 2: Creation of the document description in PRO.FILE	33
4.1.3 Checkin wizard Step 3: Assignment of the created objects to a PRO.FILE project	34
4.2 Save sub-project	37
4.3 Save drawing	37
4.4 Saving changed CAD documents to PRO.FILE.....	38
4.5 Save automatically	41
4.6 Saving of new versions.....	42
4.6.1 Save as new version	42
4.6.2 Managed Version.....	43
4.7 Save NDF (neutral data format).....	46

4.8	Part assign	47
4.8.1	Assignment of PID objects and part master records	47
4.8.2	Further functions for the assignment of PID objects and part master records	50
4.8.3	Collective assignment of PID objects and part master records.....	51
5	Edit: Additional functions of the integration.....	53
5.1	Update properties	53
5.2	Create bill of materials	54
5.3	Lock/Unlock	55
5.3.1	Starting your changes: "Lock" the CAD document	56
5.3.2	"Unlock" the CAD document	57
5.3.3	Lock/Unlock drawing.....	58
5.4	Delete database relation.....	58
5.5	Make pipe specification available.....	59
6	Display: Access to PRO.FILE information	61
6.1	The document list	62
6.2	Display: Information on a CAD document in PRO.FILE	63
6.3	Direct information in the dialog screens	65
6.3.1	More comfort: search and list functions in the dialog screens	65
6.3.2	Up to date or not: Display of status information.....	66
7	Extras: The Workcenter.....	68
7.1.1	Workcenter functions.....	68
8	Index.....	70

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 Plant 3D

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 Plant 3D. Drawings and CAD models can be loaded from or saved to PRO.FILE directly from AutoCAD Plant 3D.

When projects, sub-projects or drawings are saved, PRO.FILE automatically generates document structures 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 and CAD documents) specified during the design process with AutoCAD Plant 3D to ERP systems.

The contents of this manual

The following chapters describe the operation of PRO.FILE within the CAD system AutoCAD Plant 3D.

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 Plant 3D. The following topics will be addressed:

- Operation of PRO.FILE from within AutoCAD Plant 3D
- Data representation of structures/references in PRO.FILE
- Integration of PRO.FILE into the AutoCAD Plant 3D environment



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

When using the integration PRO.FILE – AutoCAD Plant 3D, 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.



Note: Plant projects and sub-projects ≠ PRO.FILE projects

In PRO.FILE it is possible to assign part and document descriptions to PRO.FILE projects. These are not the projects used in AutoCAD Plant 3D but the object type "Project". Projects from AutoCAD Plant 3D are saved via part and document descriptions.

It is, however, still possible to link the document structure resulting from an AutoCAD Plant 3D project to a PRO.FILE project via the default structure functions of PRO.FILE.

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

Via the loading, saving and information functions of the PRO.FILE integration, the user can access information in and functions of PRO.FILE directly from AutoCAD Plant 3D.

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

- [Only upon first start: Setting up the local work folder](#)
- [Where can I find the functions of the PRO.FILE integration?](#)
- [How to log in to PRO.FILE?](#)
- [A brief overview: The functions of the integration](#)

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

CAD drawings are loaded directly from PRO.FILE in AutoCAD Plant 3D, and also saved from AutoCAD Plant 3D 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

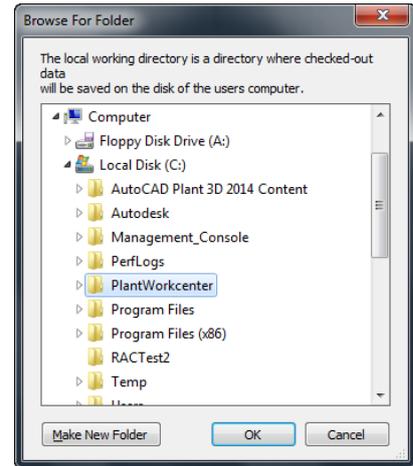
The AutoCAD Plant 3D features require the availability of all related data. Without defining a local work folder, it is not possible to work with the integration PRO.FILE – AutoCAD Plant 3D.

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

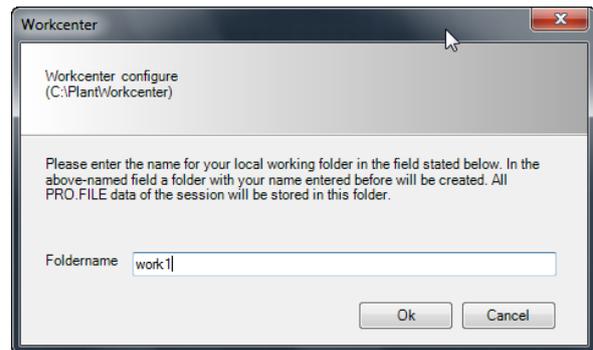
Proceed as follows

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

2. You now have to specify a "root folder". The root folder is the superior folder of the local data storage. In this folder you can later create several work folders, which are then supervised by the "Workcenter".
 - ⇒ The "root folder" can be selected - or created via the button <Make new folder>.
3. Once you have selected the desired root folder, confirm with <OK>.



4. In the second step, "work folders" are now created in this root folder, which will then be used by the integration.
 - ⇒ Consequently, you are now prompted to specify a work folder within the root folder:
5. Please specify a name for the work folder.
6. Confirm your entry with <OK>.
 - ⇒ The configuration of the Workcenter is now finished.



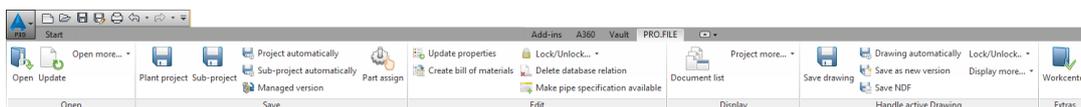
This work folder and other local work folder can be created and managed by the user via the Workcenter. The Workcenter can be accessed via the PRO.FILE menu in AutoCAD Plant 3D under "Extras" => "Workcenter".

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 Plant 3D via the menu "PRO.FILE":

1. Start "AutoCAD Plant 3D"
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 Plant 3D session, you have to log in to PRO.FILE.

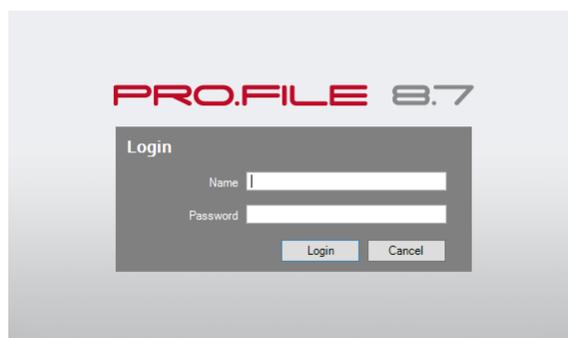
Via the logon, the user is now **identified** by his PRO.FILE user name and password. Based on this logon, the user rights, start statuses and function access rights for the logged-on user are activated.

In the login screen, please enter:

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

Confirm with <LOGIN>.

The PRO.FILE home screen is now displayed.



Note: No login required in case of "Autologin"

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

2.4 A brief overview: The functions of the integration

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

Group "Open"

- **"Open":**

Only entire projects can be opened in Plant3D. It is, however, possible to update sub-projects or drawings at a later point, by loading them into the active project via the function "Open". If a sub-project is selected via the function "Open", its structure is updated in the local project structure, but no drawing is opened. If a drawing is selected via the function "Open", this drawing is opened and activated in AutoCAD Plant 3D.
- **"Update":**

The "Update" function corresponds to the function "Open" with the difference that the user does not have to select a plant project in PRO.FILE (since the project already has to be opened). The data record of the project currently opened in AutoCAD Plant 3D is used.
- **"Open with version browser":**

Via the version browser you can decide for each component of the project structure the version to be loaded.
- **"Open as stored":**

This function opens the Plant project in the constellation of component versions it has last been saved in.
- **"Open with newest versions":**

This function opens a Plant project or sub-project with the newest available versions of the contained components (sub-project, drawings).
- **"Open with newest released versions":**

This functions opens a Plant project or a sub-project with the newest released versions of the contained components (sub-projects, drawings). This function is only available if a released version actually exists for each of the contained components.

Group "Save"

- **"Plant project", "Sub-project"**

Via the functions "Save Plant project" and "Save Sub-project" the currently active project or sub-project is saved to PRO.FILE. If the plant project has not been saved to PRO.FILE before, a document description (and, depending on the configuration also a part description) is then created for the affected objects.
- **"Project automatically", "Sub-project automatically"**

Via these functions the currently active project or sub-project is saved to PRO.FILE. Document and part descriptions for all components are created in PRO.FILE automatically without query. File names and properties can be configured to be transferred automatically into specific PRO.FILE fields.
- **"Managed Version"**

This function allows for the targeted creation of versions of the different components or sub-structures or the entire project.

- **"Part assign"**

Via this function it is possible to assign PID-Objects in P&ID-drawings specific part master records in PRO.FILE. In this context, it is possible to use existing part master records for the assignment or to create new parts in PRO.FILE.

Group "Edit"

- **"Update properties":**

Via this function the properties of the plant project are updated with the metadata of the project and the drawing in PRO.FILE. If the drawing is then opened via the project manager browser, the updated properties are also transferred into the referencing text fields of the drawing. This update is not made automatically if the drawing was already opened.

- **"Create bill of materials":**

With this function all PRO.FILE part master records that have been assigned to PID objects within the plant project are summarized in PRO.FILE in a bill of materials.

- **"Lock/Unlock":**

Via these functions the project, the sub-project and drawings can be locked by one or several users.

After one of the functions has been selected, the lock/unlock screen is displayed with the document structure.

- **"Delete database relation":**

With this function, the relation of the entire Plant project to the PRO.FILE database can be deleted. The project (including all its sub-components) is then treated as a local project that has not yet been saved to PRO.FILE.

- **"Make pipe specification available":**

SpecSheets for pipes can be saved and administrated centrally in PRO.FILE. With the function "Make pipe specification available" it is then possible to select the pipe specification from PRO.FILE, so that it is referenced by the project and does not have to be saved anew for each project.

Group "Display"

Via the functions in this menu group you can display information on the current project and contained documents.

- **"Document list":**

Displays the local project structure, the local status and the PRO.FILE status of the documents.

- **"Project more..."**

- **"Part structure":** Displays the structure of the part descriptions created for the active project in PRO.FILE.

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

- **"Bill of materials":** This function is not available in the current version of the integration.

- "Document list PRO.FILE": Displays the documents contained in the current project structure in list form in PRO.FILE.
 - "Document structure": Displays the document structure of the project in PRO.FILE.
 - "Document form": Displays the document form of the project in PRO.FILE
- Group "Handle active drawing"
- "Save":
Via the function "Save drawing", the active drawing is saved to PRO.FILE. If the drawing has not been saved to PRO.FILE before, a document description (and, depending on the configuration also a part description) is then created for the affected objects.
 - "Save automatically":
Via this function the currently active drawing is saved to PRO.FILE. The document description for the drawing is created in PRO.FILE automatically without query.
 - "Save as new version":
With this function, a new version of the active drawing, which is already saved in PRO.FILE, can be created and saved to PRO.FILE.
ATTENTION: The local project structure is updated by this process, but the project structure in PRO.FILE remains unchanged until the changed sub-project and the Plant project with the updated structure are saved to 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 document description of the drawing.
 - "Lock/Unlock":
Via these functions, the active drawing can be locked or unlocked by using the lock/unlock dialog.
 - "Display more..."
 - "Document list": Displays the structure, the local status and the PRO.FILE status of the current drawing.
 - "Document list PRO.FILE": Displays the document of the active drawing and its structure (XRefs) in a list in PRO.FILE.
 - "Document structure": Displays the document structure of the current drawing in PRO.FILE.
 - "Document form": Displays the document description of the current drawing in PRO.FILE.
 - "Document usage": Displays the document usage of the current drawing in PRO.FILE.

Group "Extras"

- **Workcenter:**

All files loaded or saved via the PRO.FILE integration in AutoCAD Plant 3D are automatically saved locally in the Workcenter folder. With this function you can manage these files or create additional work folders.

Detailed information on the individual functions and their usage can be found in the following chapters.

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

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

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



- [Open: Loading documents from PRO.FILE into AutoCAD Plant 3D](#)
- [Working with the Checkout wizard to search for CAD documents](#)
- [Open with newest and released versions of linked CAD documents](#)
- [Attention: Opening of locally existing files](#)



Attention:

The data loaded from PRO.FILE in AutoCAD Plant 3D are **not automatically locked** when opened in AUTOCAD PLANT 3D. The user has to lock the objects manually via the function "Lock".

After the object has been edited and saved back to PRO.FILE, it can be **unlocked** again, so that it is available to other users.

For detailed information see the chapter "[Lock/Unlock](#)".



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: Loading documents from PRO.FILE into AutoCAD Plant 3D

If you want to access a project, sub-project or a drawing from PRO.FILE, you have to open the entire project first. Use the function "Open" of the PRO.FILE – AutoCAD Plant 3D integration menu for this.



Note:

The PRO.FILE menu in AutoCAD Plant 3D is only available if at least one drawing is opened. If required, create a new empty drawing.

The **open** functions start the PRO.FILE Checkout wizard, in which you can select the desired document description of the project for loading in AutoCAD Plant 3D.

Step 1 Select the PRO.FILE function "Open"



Function call from the PRO.FILE menu in AutoCAD Plant 3D:

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

1. Go into the menu bar of AutoCAD Plant 3D into the menu "PRO.FILE".
 2. Select the menu entry "Open".
- ⇒ "Open" loads documents as they were saved the last time in PRO.FILE.



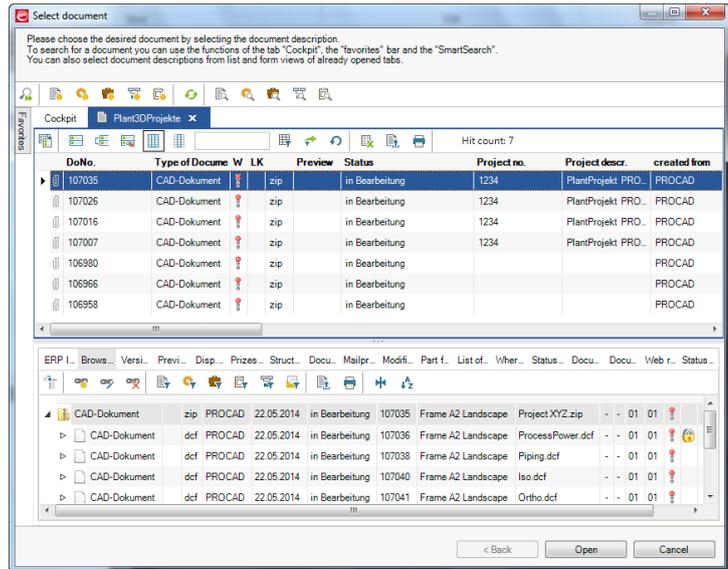
⇒ The Checkout wizard for the selection of documents is displayed.

Step 2 Select the desired document in the Checkout wizard

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

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

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



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

5. Click **<Open>**.

⇒ The Checkout wizard closes and the project is opened in AutoCAD Plant 3D.

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

Step 3

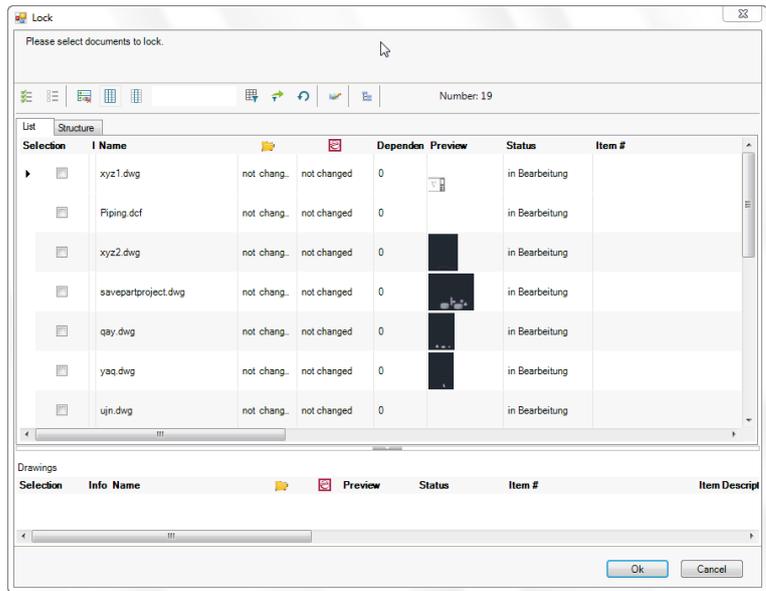
You want to edit? Then you must lock the document(s)

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

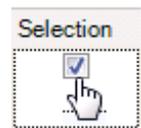
This means: If you want to edit the CAD document, you have to lock it.

The "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](#)")



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



⇒ The selected documents and its components are now opened in AutoCAD Plant 3D. 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 to search for CAD documents](#)

3.1.1 What happens at the file level when a plant project is opened?

A plant project is saved in PRO.FILE as a zip file to the document description of the project. This zip file contains the complete folder of the project including the sub-folders and all non-referencing files.

To the document descriptions of the sub-projects, the database files (.dcf, .dcfx) referenced in the plant project are saved. The drawings referenced in the sub-projects are saved to the document descriptions of the drawings.

When opening a plant project from PRO.FILE the folders contained in the zip file are created locally in the working folder and their contents are extracted.

Then, the remaining referenced files of the sub-projects and drawings are copied to the project folder. A collision check is then made. If there are locally changed files, they are displayed in a list and can either be overwritten or kept.

3.1.2 Working with the Checkout wizard to search for CAD documents

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

The **aim** of this procedure is:

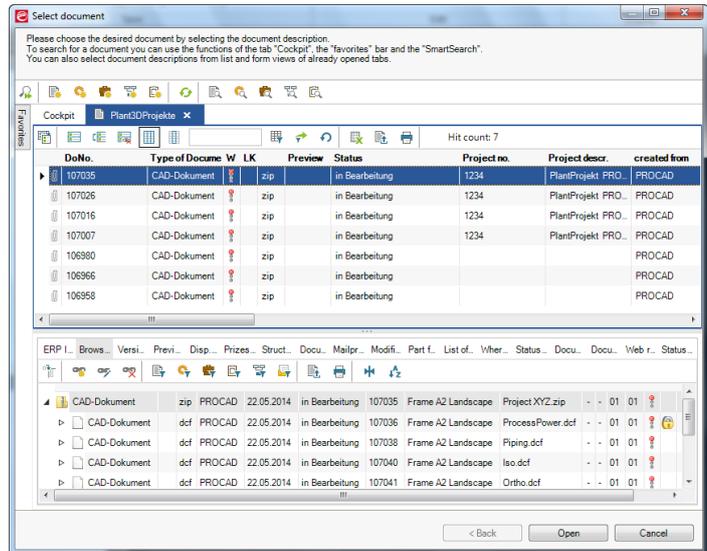
- For PRO.FILE to know which document is opened, the desired document description must
 - either be selected in a list view or a structure browser
 - or be displayed in a form view
- Then the button <Open> at the right bottom of the Checkout wizard has to be clicked.

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

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

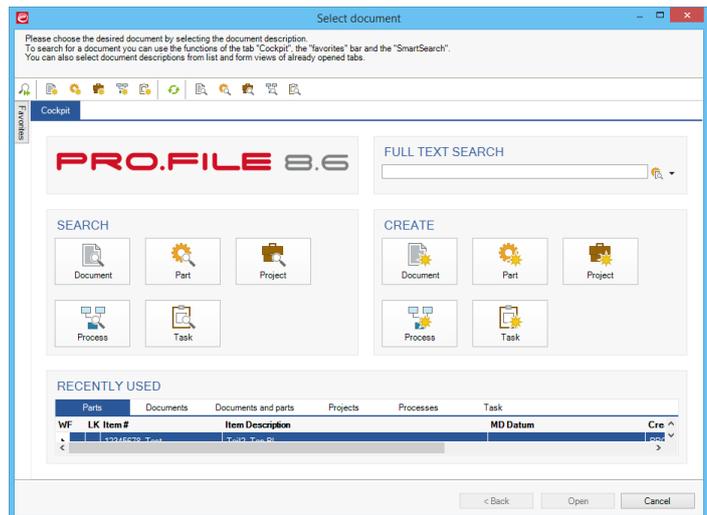
If search results had previously been opened in a list or form view in a separate tab, you can directly access the displayed documents.

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



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

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



Attention: Double-click in the Checkout wizard

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

Because a double click means: Open document for viewing!

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

Searching for data records in the Checkout Wizard

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

- **Searching via the tab "Cockpit"**

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

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

- **Search via the functions of the favorites bar**

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

- Via the "Favorites" memorized searches or data lists can be accessed with a double click.
- With the "SmartSearch" you can create individual search forms.
- If you are working with PRO.FILE processes and tasks, you can access the documents linked to a task or process via the task or process structure.

- **Search via the icon bar**

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

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

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

3.1.3 Open sub-projects and drawings

The opening of a sub-projects or drawings is actually an update, since this is only possible when the entire project is already in the Workcenter folder and opened in AutoCAD Plant 3D.

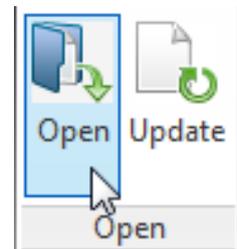
If a drawing has an offline change, a corresponding message is displayed during the open process and the user can then update the corresponding sub-project.

Before the loading in the Workcenter folder, a collision check is made, showing a dialog where you can select local documents to be overwritten with the status from PRO.FILE.

Sub-projects or drawings can be opened from PRO.FILE.

Proceed as follows

1. Go into the menu bar of AutoCAD Plant 3D 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. The further proceeding for the opening of sub-projects/drawings is identical to the proceeding for the opening of plant projects, as described in the chapter "[Open: Loading documents from PRO.FILE into AutoCAD Plant 3D](#)". The difference, however, is that instead of a document description of a plant project, you select the document description of a sub-project or drawing.

3.2 Update

The update function corresponds to the function "Open" for projects with the following differences:

- You do not have to select a plant project in PRO.FILE. The record of the currently opened plant project is used.
- In the dialog for the collision check, which displays the documents to be copied from PRO.FILE, all documents locked by the current user are not selected and cannot be selected for copying.

This way, you can update your local status with data from PRO.FILE without losing changes to documents locked by you.



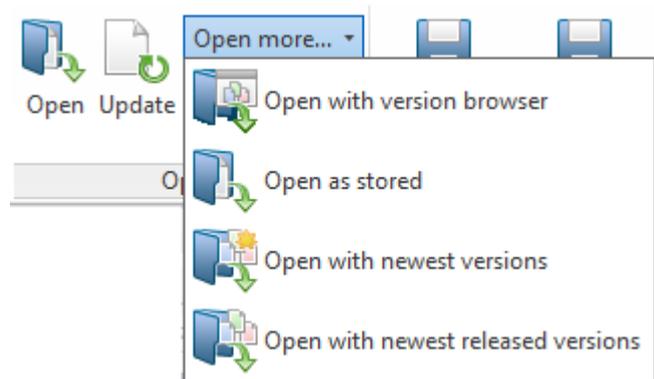
Attention:

If the function "Open" is used instead, e.g. to load (i.e. to update) a sub-project, locked documents can be overwritten as well.

3.3 Open with newest and released versions of linked CAD documents

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

- Open as stored
- Open with newest versions
- Open with newest released versions



Note:

The latter two options do **not** refer to the Plant project to be opened from PRO.FILE. They refer to the objects contained in the project (sub-projects and drawings) from PRO.FILE. As a user, you can decide with which version status you want to open the components linked to the project

The different options are:

- "Open as stored"

The selected plant project (or sub-project if a plant project is already open) 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 Plant project or sub-project contains links to other CAD documents (sub-projects or drawings) in PRO.FILE, the newest versions of these linked CAD documents are loaded.

When the function "Open with newest versions" is used for opening a project or sub-project, PRO.FILE checks, whether the project contains components for which versions exist. If this is the case, the newest visible version of such a component is loaded in the AutoCAD Plant 3D session.
- "Open with newest released versions"

The selected project or sub-project is loaded with the newest, released versions of the linked sub-projects or drawings. The newest versions that are in a release status are loaded.

When the function "Open with newest released versions" is used for opening a project or sub-project, PRO.FILE checks, whether the project 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 component is loaded in the AutoCAD Plant 3D session.



Attention:

The function "Open with newest released versions" can only be used, if a released version actually exists for each of the contained components (sub-projects, drawings). If this is not the case, the open process is aborted with a message, saying that no released versions were found.

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 Plant 3D](#)".



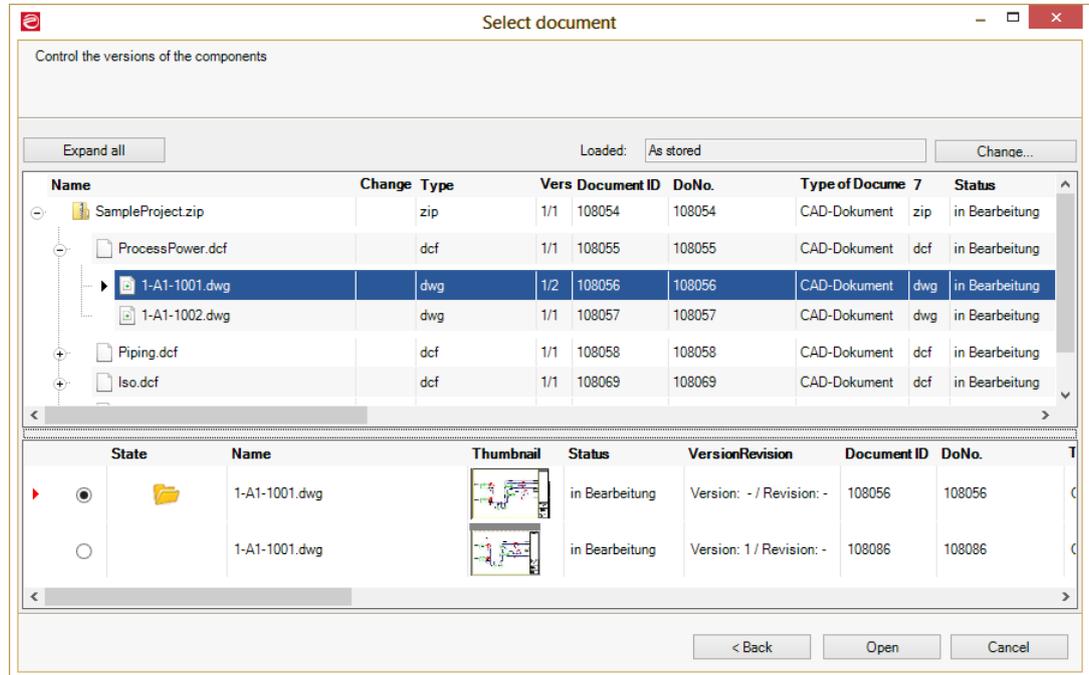
Note:

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 projects in dynamic constellations. You can define via the version browser with which version a project and its components are 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 project 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 "newest version" of the components
- Open "newest released version" of the components.
- Open "newest released version or newest 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 Plant 3D.
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.
 - ⇒ At this moment, the selected CAD data is not yet locked in PRO.FILE and still available for other users. This means: If you want to edit the CAD data, you have to lock it.
7. Select all documents you want to lock by setting the checkmark for it in the first column. Detailed information on the locking of documents can be found in the following chapter "[Lock/Unlock](#)".
8. Confirm your selections with <OK>.
 - ⇒ The selected CAD components are opened in Inventor. The process of opening with the version browser is now finished.

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" => "Extras" => "Workcenter" => "Activate" in order to prevent the overwriting of the files.



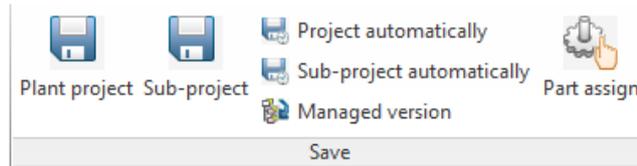
Note:

Due to this behavior it is absolutely necessary, that the file names in PRO.FILE are unique. Otherwise, it may happen that a "plant" 1234 is overwritten with another variant 5678 also named "plant".

4 Saving CAD documents to PRO.FILE

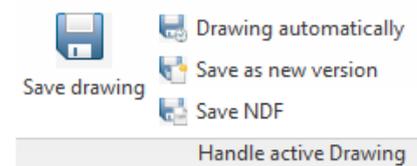
For the first-time saving of CAD documents, as well as for the saving of changes from the local Workcenter to PRO.FILE, the following functions are available:

In the ribbon category "Save"



- Save Plant project
- Save Sub-project
- Save Project automatically
- Save Sub-project automatically
- Managed Version
- Part assign

In the ribbon category "Handle active drawing"



- Save Drawing
- Save Drawing automatically
- Save as new version
- Save NDF

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

The description is therefore divided into three chapters:

- [Save Plant project: Saving CAD objects for the first time](#)
- [Saving changed CAD documents to PRO.FILE](#)
- [Saving of new versions](#)

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 "Operation PRO.FILE for Advanced".



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

When using the integration PRO.FILE – AutoCAD Plant 3D, 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.

- "Save":

Via the functions "Save Project", "Save Sub-project", "Save Drawing", the opened project, the active sub-project or the active drawing is saved to PRO.FILE.

First, all open drawings are saved locally. The open but not activated drawings are closed.

The project folder, or parts of the project folder are then packed to zip files and copied to PRO.FILE.

The drawings referenced in the sub-projects are directly saved to the document descriptions of the drawings and excluded from the zip files.

The corresponding document structure is built.

If a project, sub-project or drawing already has a project relation, it is a change to be saved. It is then checked whether you have the permission to save the affected object.

A document list is displayed, in which all savable documents are listed. All non-savable documents cannot be selected in this list, and thus cannot be saved.

**Note:**

For new projects, the entire project has to be saved to PRO.FILE first, since the project folder is then copied to the Workcenter folder (if it is not already there) and a consistent structure is built in PRO.FILE.

To ensure this, a check is made when a sub-project / drawing is to be saved, whether the superior plant project is already known in PRO.FILE. If this is not the case, an error message is displayed.

4.1 Save Plant project: Saving CAD objects for the first time

With the use of the function "Save Plant project", you can save the entire AutoCAD Plant 3D project, with all sub-projects, drawings and related files to PRO.FILE.



Note:

For new projects, the entire project has to be saved to PRO.FILE first, since the project folder is then copied to the Workcenter folder (if it is not already there) and a consistent structure is built in PRO.FILE.

To ensure this, a check is made when a sub-project / drawing is to be saved, whether the superior plant project is already known in PRO.FILE. If this is not the case, an error message is displayed.

The action "Save" takes place in several steps, with different dialogs being displayed, depending on the results of the previous step



Note:

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

To save a plant project from AutoCAD Plant 3D to PRO.FILE for the first time open the function "Save Plant project".



Function call:

"PRO.FILE" => "Save" => "Save Plant project"

Proceed as follows

1. Select the menu "PRO.FILE" from the menu bar.
2. Click on the "Save..." button.



Plant project :

⇒ The Checkin wizard is displayed, which will support you in the proper saving of your plant project.

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

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

These steps are described in the following sub-chapters.

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.

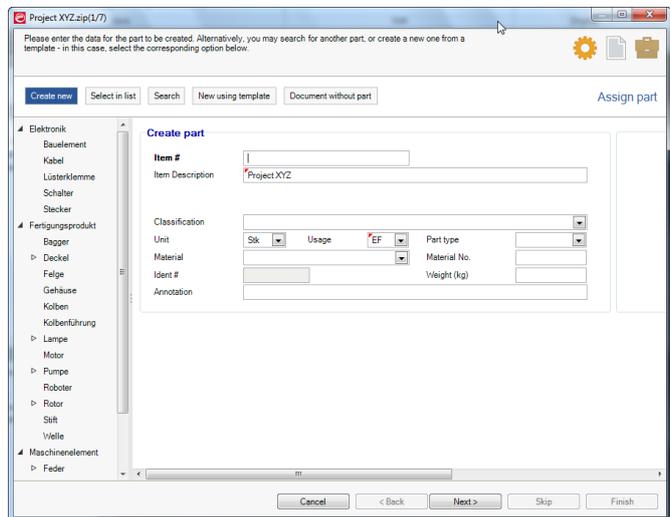


Note: Usage of PRO.FILE parts

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:

Create new

Usage:

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

Proceeding:

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

Select in list:

Select in list

Usage:

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

Proceeding:

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

Search:

Search

Usage:

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

Proceeding:

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

New using
template:

New using template

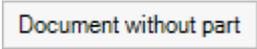
Usage:

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

Proceeding:

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

Document without part:



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.

4.1.2

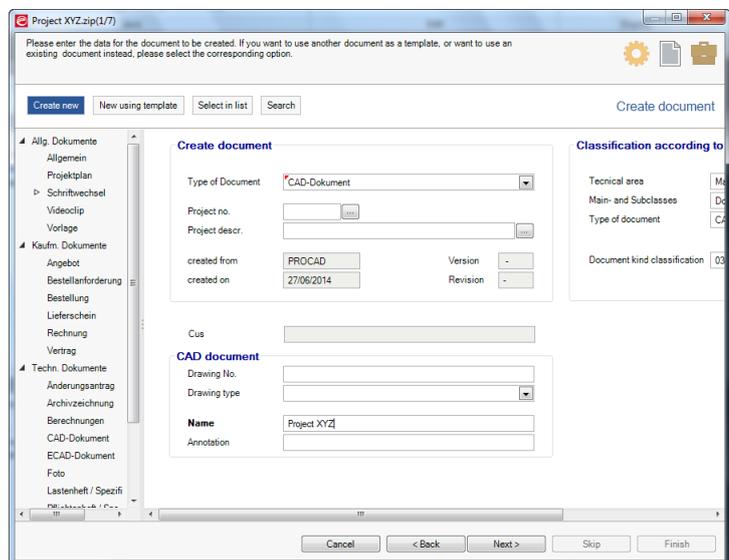
Checkin wizard Step 2: Creation of the document description in PRO.FILE

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.

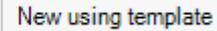
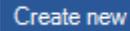
For this, the Checkin wizard for the document description is available:

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.

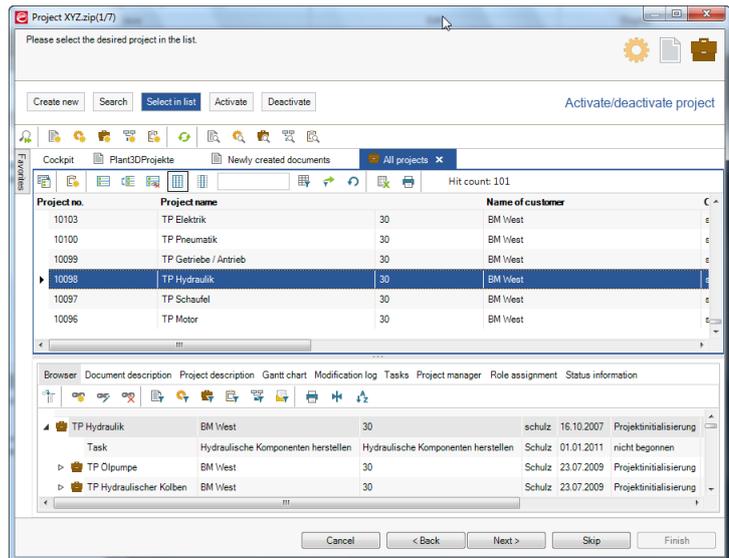


Note: Usage of PRO.FILE projects

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.

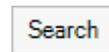
The proceeding for these options is the same as for the first two steps of the Checkin wizard:

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.

Search:

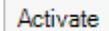


The part master record and document description created in steps 1 and 2 are to be assigned to an existing project. This project is now searched via the search form and selected.

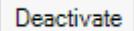
Select in list:



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

Activate:A rectangular button with a thin border and the text "Activate" centered inside.

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

Deactivate:A rectangular button with a thin border and the text "Deactivate" centered inside.

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

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

Proceeding:

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

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

4.2 Save sub-project

Via the function "Save Sub-project" you can save a sub-project with all contained drawings and related files to PRO.FILE.



Note:

For new projects, the entire project has to be saved to PRO.FILE first, since the project folder is then copied to the Workcenter folder (if it is not already there) and a consistent structure is built in PRO.FILE.

To ensure this, a check is made when a sub-project / drawing is to be saved, whether the superior plant project is already known in PRO.FILE. If this is not the case, an error message is displayed.

Consequently, the function "Save Sub-project" is only reasonable and available for the saving of changes to a sub-project (adding of drawings, saving of drawing changes). Further information can be found in the chapter "[Saving changed CAD documents to PRO.FILE](#)".

4.3 Save drawing

Via the function "Save Drawing" you can save a drawing with all referenced drawings and related files to PRO.FILE.



Note:

For new projects, the entire project has to be saved to PRO.FILE first, since the project folder is then copied to the Workcenter folder (if it is not already there) and a consistent structure is built in PRO.FILE.

To ensure this, a check is made when a sub-project / drawing is to be saved, whether the superior plant project is already known in PRO.FILE. If this is not the case, an error message is displayed.

Consequently, the function "Save Drawing" is only reasonable for the saving of changes to a sub-project (adding of drawings, saving of drawing changes). Further information can be found in the chapter "[Saving changed CAD documents to PRO.FILE](#)".

If the function "Save Drawing" is applied to a drawing newly created within the sub-project, the drawing is saved to PRO.FILE but not assigned to the sub-project in PRO.FILE. To make this assignment later, the sub-project has to be saved as well.

4.4 Saving changed CAD documents to PRO.FILE

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

If you use one of the "Save" functions 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:

- ["Saving changed CAD documents to PRO.FILE"](#) =(Modify)
- ["Save automatically"](#) (see following chapter).

This chapter describes the proceeding for the saving of changes.



Function call from the PRO.FILE menu in AutoCAD Plant 3D:

"PRO.FILE" => "Save" => "Plant project"

"PRO.FILE" => "Save" => "Sub-project"

"PRO.FILE" => "Handle active drawing" => "Save Drawing"

- When "Save" => "Plant project" is used, all changes to the entire project, including sub-projects and drawings, are saved.
- When "Save" => "Sub-project" is used, all changes to the active sub-project, including drawings, are saved.
- When "Handle active drawing" => "Drawing" is used, only changes to the active drawing are saved.

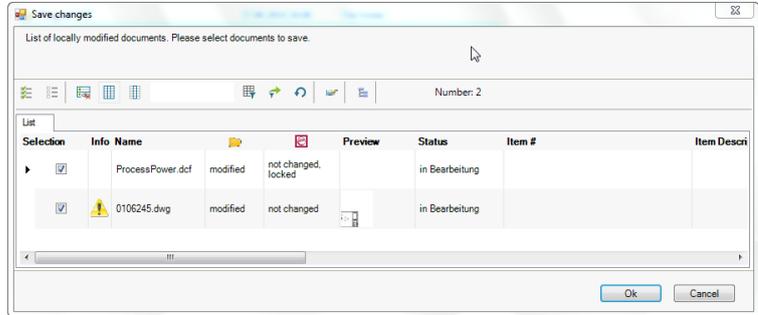
Proceed as follows

1. Go to the integration menu "PRO.FILE" in AutoCAD Plant 3D.

2. Select the desired "Save" function from the area "Save".
- ⇒ PRO.FILE recognizes the CAD document as a PRO.FILE object and automatically goes into change mode.

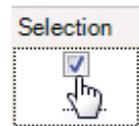
Selection of the documents to be saved

- ⇒ The dialog for the selection of CAD documents to be saved is displayed. (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 Plant 3D 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.)

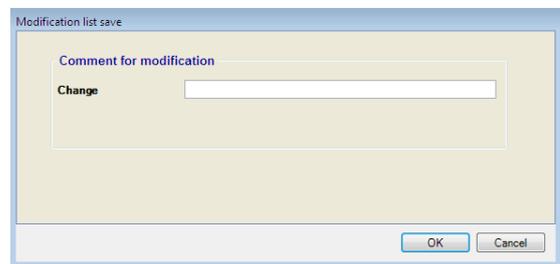
3. Select all documents you want to save in PRO.FILE. To do so, activate the checkboxes for the desired documents.
4. Confirm your selection with <OK>.



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

Optional: Enter modification comment

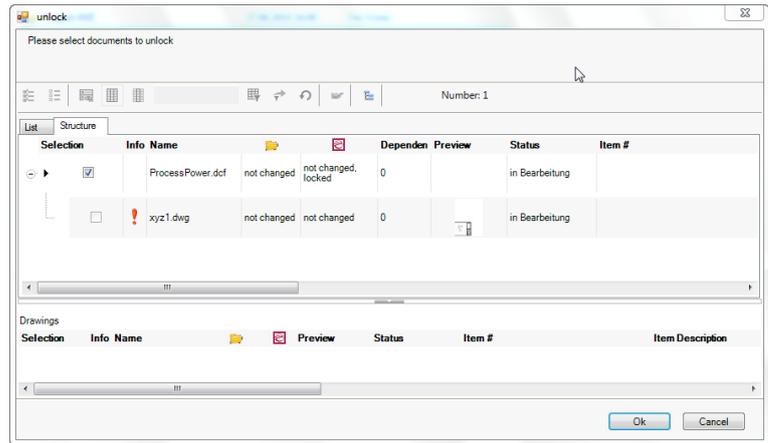
5. Depending on the configuration and PRO.FILE status, you now have to enter a modification comment. Enter the comment information into the fields on the dialog screen.



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.

Locking/unlocking the saved documents

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



- ⇒ If documents from PRO.FILE had been locked for editing in AutoCAD Plant 3D, 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.
- 7. To make this process easier, the PRO.FILE CAD documents that are still locked are displayed in the list. 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.
- 8. Confirm your selection with <OK>.
- ⇒ The lock flag for the selected documents is now removed.
- ⇒ The saving of your changes to PRO.FILE is now finished.



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](#)".

4.5 Save automatically

Apart from the already described menu functions <Save> the integration offers the function <Save Project automatically>, <Save Sub-project automatically> and <Save Drawing automatically>, 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 Plant 3D:

"PRO.FILE" => "Save" => "Project automatically"

"PRO.FILE" => "Save" => "Sub-project automatically"

"PRO.FILE" => "Handle active drawing" => "Drawing automatically"



Note:

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

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



Attention: "Required fields" and <Save automatically>

For all automatically created CAD documents (sub-projects, drawings), the values that are to be entered via the Checkin wizard do NOT exist. This is also the case for fields defined as required fields. As a result, all documents created automatically have to be classified in detail at a later point in time, especially if field information is required for the transfer to other systems (ERP integrations)!

4.6 Saving of new versions

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

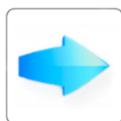
- With the function "**Save as new version**" in the category "Handle active drawing" a version can only be created from the active drawing. The versioning of projects and sub-projects is not possible via this function.
- Via the function "**Managed Version**" you can create versions of any desired number of components of a project (including the project itself).

Both functions require that the object to be versioned is already saved in PRO.FILE. The detailed proceeding is described in the following sub-chapters.

4.6.1 Save as new version

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

- Only this active drawing 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 Plant 3D.
- Within a project/sub-project the local reference is switched to the new version of the drawing.
- A new version can only be created of the newest version of a drawing. Example: A new version of a drawing "Version 2" cannot be created if a "Version 3" of this drawing already exists. Instead, you would have to open and activate "Version 3" to create a new version (provided that version 3 is the newest version).



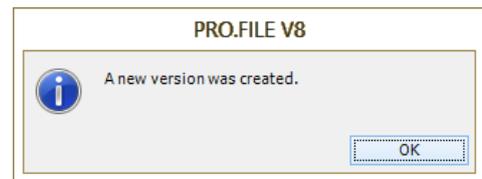
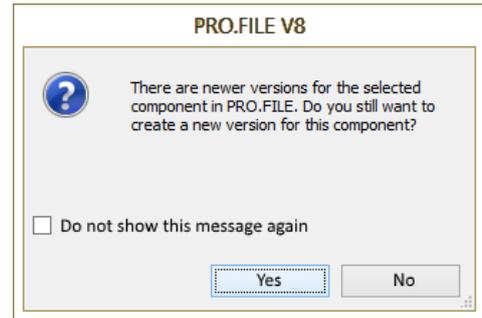
Function call from the PRO.FILE menu in AutoCAD Plant 3D:

"PRO.FILE" => "Handle active drawing" => "Save as new version"

Proceed as follows

1. Make sure that the drawing, of which you want to create a version, is opened and active in AutoCAD Plant 3D.
2. Go to the integration menu "PRO.FILE" in AutoCAD Plant 3D.
3. Select the function "**Save as new version**" from the area "**Handle active drawing**".

4. If you have not opened the newest version of this drawing, but want to create a new version from an older version (provided that this is allowed for you), the setting of the parameter "Ask when creating version from an older version" determines whether a dialog is now displayed.
 5. If you want to create this version from an older version, confirm the dialog 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 Plant 3D.



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 dependent tab "Versions" of the document description in PRO.FILE.

4.6.2

Managed Version

The function "**Managed Version**" is used for the creation of versions within project structures. These functions support the following requirements:

- Contained components (sub-projects, drawings) are included in the process
- The file name of the versioned project components remain unchanged.

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

- By selecting a component for "**Managed Version**", this component is included in the process of version creation. Sub-components of this selected component (sub-projects and drawings) are not automatically versioned as well – they need to be selected individually. Per definition, all versions of a component have the same file name.

- Components that are not selected are not versions. If, for example, a sub-project is versioned but not its drawings, the new version of the sub-project will still refer to the old version of the not selected drawings.
- The versionability of the components is checked at the beginning of the "Managed version" process. If a component cannot be versioned (e.g. already released or a newer version exists already), it cannot be selected. In such a case, the version dialog displays an icon and a tooltip on the cause of the problem.

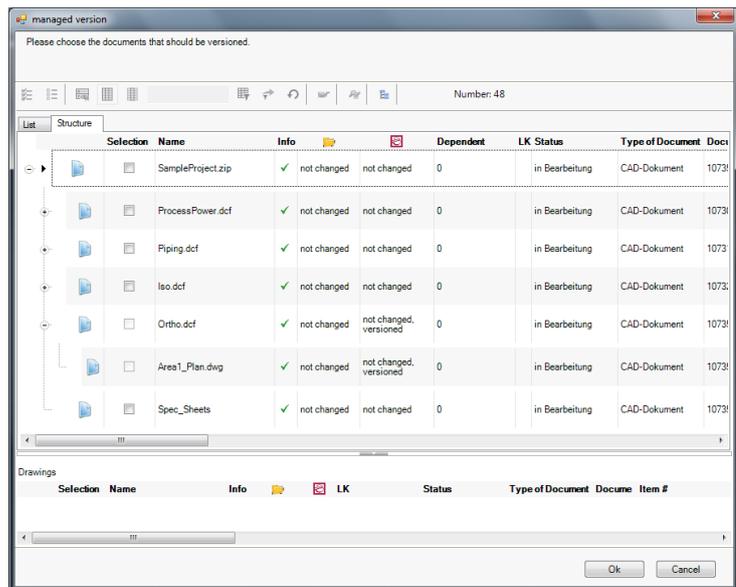


Function call from the PRO.FILE menu in AutoCAD Plant 3D:
 "PRO.FILE" => "Save" => "Managed Version"

Proceed as follows

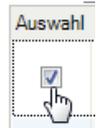
1. Open the project, the structure of which you want to use for "Managed Version" via the project managed.
 2. Select the function "Save" => "Managed Version" from the "PRO.FILE" menu of the integration.
- ⇒ The Managed Version wizard is displayed.

⇒ The integration detects the structure of the project based on the current CAD document.



- ⇒ The wizard displays the detected project structure, so that you can select the documents you want to create a version of.
- ⇒ The top node and the first level are expanded. Further levels can be expanded via the corresponding structure icon.
- ⇒ The column "Info" contains additional information, e.g. if a component cannot be versioned. By moving the mouse pointer over the error icon, a tooltip is displayed explaining the cause.
- ⇒ The "Status" columns show the current editing status of an object in the workcenter folder and in PRO.FILE (see chapter: [Up to date or not: Display of status information](#))

3. **Select:** Select all components you want to create a version in PRO.FILE of. To do so, activate the checkbox next to the component.

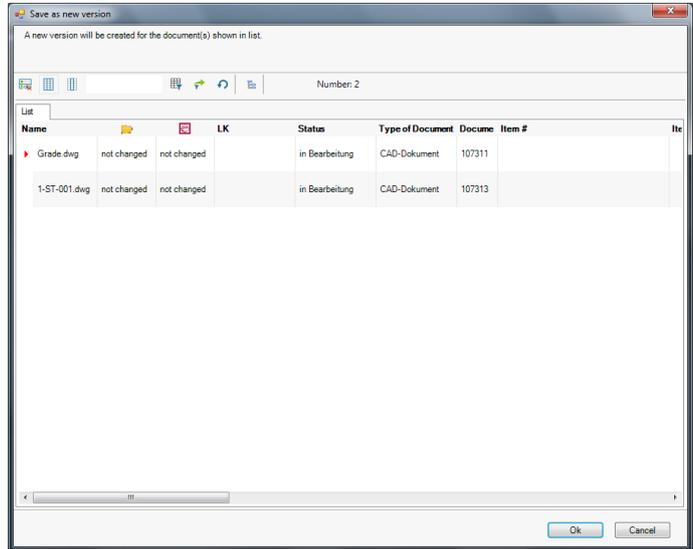


4. Once you are done, confirm with <OK>.

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

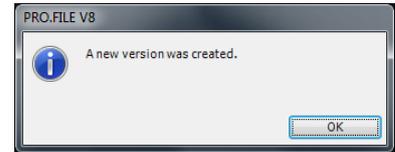
5. Confirm this list with <OK>.

⇒ New versions are now created of all selected components.

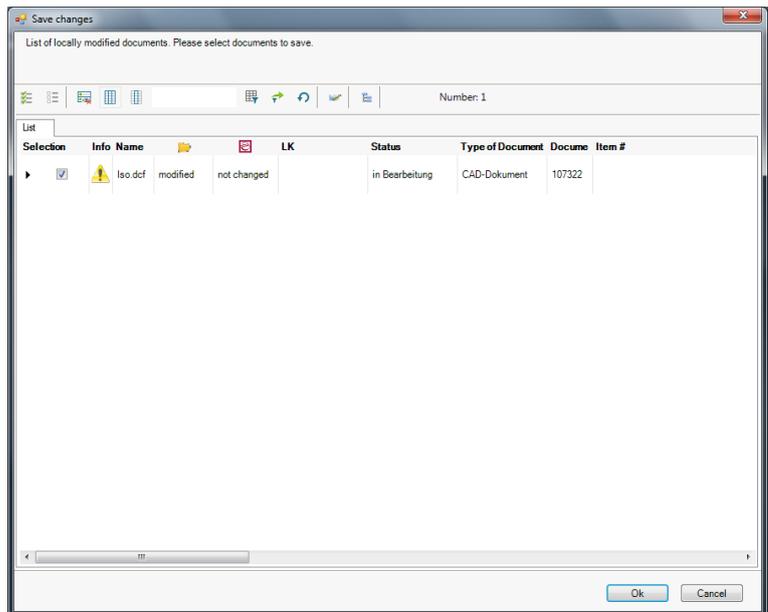


⇒ The successful creation of the new version(s) is confirmed by a message.

6. Confirm this message with <OK>.



⇒ A final list shows the document (sub-projects), that have been changed locally as a consequence of the version creation.



7. In this list please select all documents to be saved by activating the corresponding checkbox.

8. Confirm your selection with <OK>.
- ⇒ The process "Managed Copy" is thus finished.

4.7 Save NDF (neutral data format)



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 Plant 3D offers the possibility to convert an AutoCAD Plant 3D 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 Plant 3D:

"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.

Struktur 107050										
CAD-Dokument	4711	dwg	PROCAD	26.05.2014	Freigabe	107050	4711.dwg	-	-	01 01
Archivzeichnung	4711	tif	PROCAD	27.05.2014	Freigabe	107056	4711.tif	-	-	01 01



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.

4.8 Part assign

The function "Part assign" facilitates the administration of the relation between PID objects in the drawing to part master records in PRO.FILE. This relation is then also displayed in the Data Manager of AutoCAD Plant 3D.

PnPID	Part Family Long Desc	Compatible Standard	Manufacturer	Material	Material Code
479	PIPE, PE, ASME ...	ASME B36.10		CS	ASTM A106 Gr.
502	Gate Valve, Dou...	ASME B16.10		CS	ASTM A 351 Gr

4.8.1 Assignment of PID objects and part master records

Part master records from PRO.FILE can be assigned to an object in your P&ID drawing.

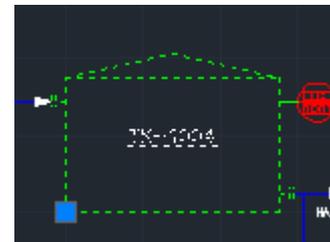


Function call from the PRO.FILE-menu in AUTOCAD Plant 3D:

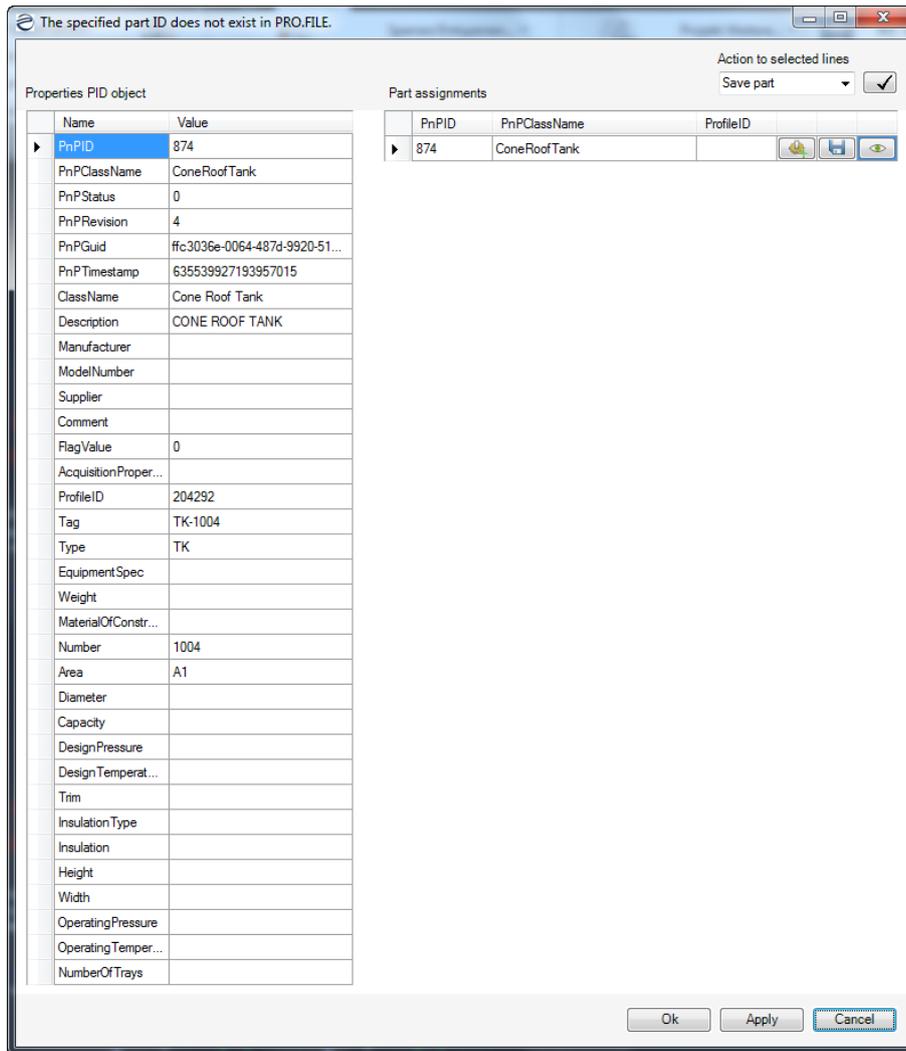
"PRO.FILE" => "Save" => "Part assign"

Proceed as follows

1. In your drawing, select one or more PID objects, for which you want to make a part master record assignment.



2. From the area "Save" of the integration menu select the function "Part assign".
Note: If you did not select an object in the first step, you are now prompted by AutoCAD Plant 3D to select objects.
 => The screen for the assignment of PRO.FILE part master records is displayed.

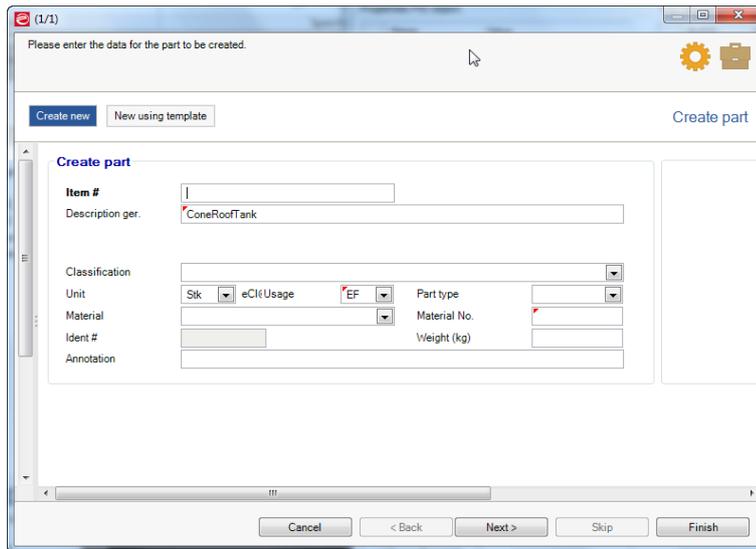


- ⇒ The left-hand area of this window lists the properties of the active PID object. The right-hand area lists the PID objects selected in the first step. The column ProfileID is empty, provided that no assignment has been made so far.
- 3. You can now make the assignment as follows:

Option 1

If no part master record exists in PRO.FILE for this PID object so far

1. Click on the button "Save part" .
- ⇒ The Checkin Wizard for the creation of part master records is displayed in PRO.FILE.

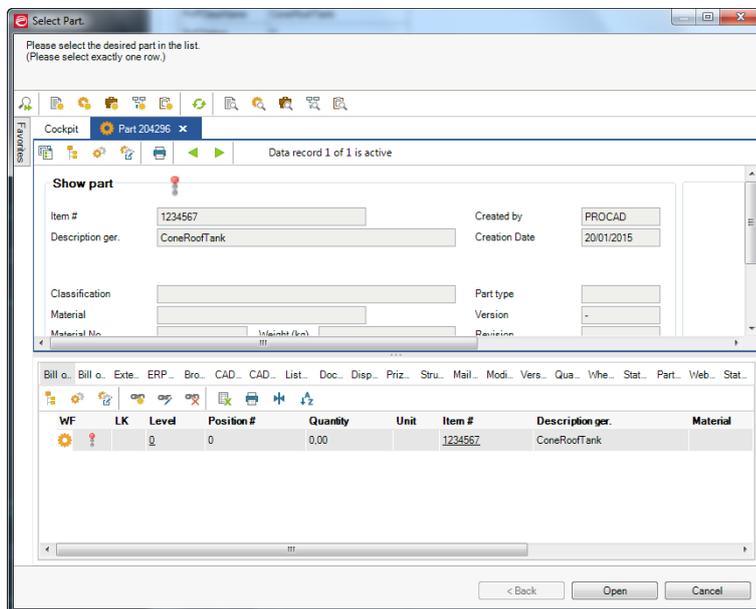


2. Here you can enter the describing data for the PRO.FILE part.
3. Confirm your entries with <Finish>.

Option 2

If there is already a part master record in PRO.FILE for this PID object

1. Click on the button "Assign part" .
- ⇒ The PRO.FILE wizard for the selection of an existing part master record is displayed.



2. Here you can select the desired part master record from a form/list display (if the desired part master record is not yet displayed in a list/form, you can start a search for it via the tab "Cockpit").
3. Confirm your selection with <Open>.

⇒ You will receive the following result for both possibilities: In the window for the assignment of part master records, the column "ProfileID" now shows the ID of the PRO.FILE part master record.

Part assignments			
PnPID	PnPClassName	ProfileID	
8743	ConeRoofTank	204292	  

4. If you want to make further assignments for other PID objects, please repeat the steps above accordingly. Otherwise, you can confirm your assignments with <OK>.

⇒ The selected PID objects are now assigned part master records from PRO.FILE. This information is also displayed in the data manager.



Attention: Manual assignment without wizard

You can also make the assignment of a PID object to a part master record manually without the wizard – by entering the ID of the part master record into the field "ProfileID" – but then you have to make sure that this part master record ID actually exists and belongs to the correct part.



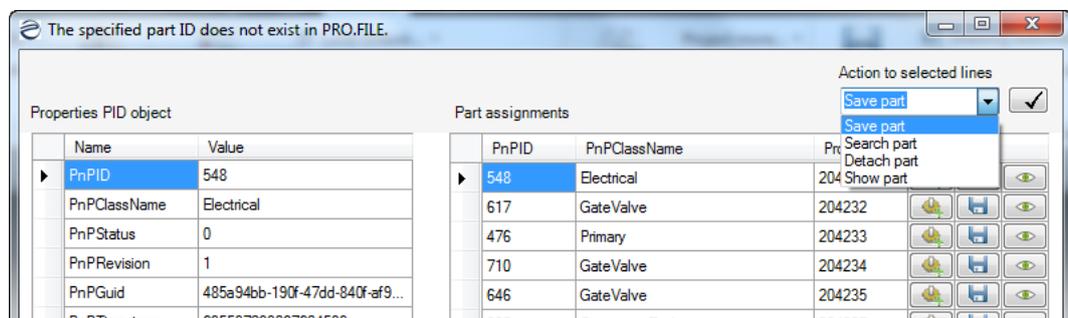
Note:

To undo the assignment of a single part master record to a PID object, it is sufficient to delete the part master record ID from the field "ProfileID". As soon, as you close the window with <OK>, the change is applied.

4.8.2

Further functions for the assignment of PID objects and part master records

To make the assignment of PID objects and PRO.FILE part master records easier, the window for the assignment offers further functions:



These functions have the following effects;

- "Save part" in the dropdown box "Action to selected lines":

If several objects are selected in the list of PID objects, this function can be used to create part master records for all selected objects in PRO.FILE and assign them.

Further information can be found in the chapter "[Collective assignment of PID objects and part master records](#)".

- "Search part" in the dropdown box "Action to selected lines":
If several objects are selected in the list of PID objects, this function can be used to automatically select part master records for all selected objects in PRO.FILE and assign them. Further information can be found in the chapter "[Collective assignment of PID objects and part master records](#)".
- "Detach part":
Via this function, the assignment of all PRO.FILE part master records is undone for all selected PID objects.
- "Show part":
If this function is applied to a single selected PID object (to which a PRO.FILE part is already assigned), PRO.FILE opens and displays the corresponding part in a form display. If the function is applied to several selected objects via the dropdown box "Action to selected lines", PRO.FILE opens and displays the corresponding parts in a list display.

All actions you want to perform via the dropdown box "Action to selected lines" have to be confirmed afterwards via the button .

4.8.3

Collective assignment of PID objects and part master records

To make the assignment of PRO.FILE parts to PID objects easier, the assignment can also be made as a collective process via the dropdown box "Action to selected lines".

Automatically create part master records in PRO.FILE for PID objects

Proceed as follows

1. In the list of PID objects, select all objects for which you want to create parts in PRO.FILE.
 2. From the dropdown box "Action to selected lines", select the function "Save part" and confirm with the button .
- ⇒ The parts are created in PRO.FILE and assigned to the PID objects.



Attention: Automatic creation of part master records

Since the part master records are created without further user interaction, it has to be made sure that all required information (name, etc.) is transferred automatically to the part master record form. This may be necessary because:

- the part master records would otherwise require post-editing, or
- the part master record form may contain required fields.

In the latter case, an automated creation of part master records without automatic transfer of required information is not possible.

For the automated transfer of information to the PRO.FILE part master record a corresponding configuration of the parameter "Copy parameters from CAD to PRO.FILE" (internal name `_38_AUTOPAR_CFG`) is required in the PRO.FILE Management Console. Details can be found in the configuration manual of the integration.

If necessary, contact your administrator.

Proceed as follows

Automatically assign existing parts in PRO.FILE to the corresponding PID objects

1. In the list of PID objects, select all objects for which you want to make part assignments in PRO.FILE.
 2. From the dropdown box "Action to selected lines", select the function "Search part" and confirm with the button .
- ⇒ PRO.FILE now searches for parts matching the defined criteria (see warning below) of the PID objects. If such parts are found, they are automatically assigned to the corresponding PID objects.



Attention: Automatic assignment of part master records

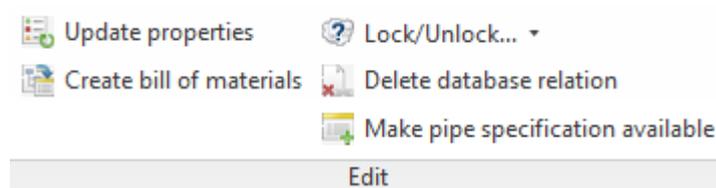
Since the part master records are to be assigned without further user interaction, it has to be made sure that the corresponding parts are found in PRO.FILE based on specified criteria.

For this, the correct configuration of the parameter "Automatic assignment to existing part master record" (internal name `_38_PART_MATCH_FIELDS`) is required in the PRO.FILE Management Console. Details can be found in the configuration manual of the integration.

If necessary, contact your administrator.

5 Edit: 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:

- [Update properties](#)
- [Create bill of materials](#)
- [Lock/Unlock](#)
- [Delete database relation](#)
- [Make pipe specification available](#)

5.1 Update properties

In AutoCAD Plant 3D, certain text fields in the drawing reference to the properties of the plant project.

The synchronization of these text fields is made indirectly via the update of the properties of the plant project.

Via the function "Update properties" the properties of the plant project are updated with the metadata of the project and the drawing in PRO.FILE. If the drawing is then opened in the project manager, the updated properties are also transferred to the referencing text fields of the drawing. This update of the drawing is not made automatically if the drawing was already opened.

Furthermore, there is the possibility to synchronize field contents with the PRO.FILE document description of the drawing by attribute blocks. For this, the function "Update properties" must be used while the drawing is opened and active.



Function call from the PRO.FILE menu in AutoCAD Plant 3D:

"PRO.FILE" => "Edit" => "Update properties"

After this function is used, all contained elements of the drawing legend (title block, modification list) 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 Create bill of materials

In the P&ID drawings and 3D drawings you can assign the various PID objects part master records from PRO.FILE – in which case you can either use existing part master records or create new ones. Furthermore, the integration allows the creation of a bill of materials from all part master records that have been assigned to PID objects in the above way.

During the creation of the bill of materials, the integration checks the number of occurrences of the various objects: if an object occurs several times within a drawing and is always assigned to the same part master record, the object is only listed once in the bill of material (one position) but with an accordingly increased number of units (e.g. pieces).

As BOM header, the PRO.FILE part master record the document description of the plant project is linked to is used.



Function call from the PRO.FILE menu in AUTOCAD Plant 3D:

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

Creation of a bill of materials for a plant project:

Proceed as follows

1. Open the plant project from PRO.FILE.
 2. Make sure that the plant project (i.e its document description in PRO.FILE) is linked to a part master record (see chapter "[Checkin wizard Step 1: Creating or assigning a part master record in PRO.FILE](#)").
 3. Make sure that the PID objects that are to be part of the bill of materials are linked in PRO.FILE to the corresponding part master records (see chapter "[Part assign](#)").
 4. In the menu bar of the PRO.FILE integration, in the area "Edit", select the function "Create bill of materials".
- ⇒ The bill of materials is now built from all PID objects that have been assigned a PRO.FILE part master record and is saved in PRO.FILE. For this process, please note the following:
- For the creation of the bill of materials, all plant class objects that have a filled field "ProfileID" and are either part of the P&ID or the Plant 3D sub-project.
 - If several plant class objects have the same PRO.FILE part master record ID and the same plant tag, they are only listed once in the bill of materials.
 - If several plant class objects in the Plant 3D sub-project have the same PRO.FILE part master record ID but different plant tags, they are summarized under one position with a quantity > 1 (according to the number of their occurrences).

- If several plant class objects in the P&ID sub-project have the same PRO.FILE part master records ID but different plant tags and do not exist in the Plant 3D sub-project they are summarized under one position with a quantity > 1 (according to the number of their occurrences).

⇒ The bill of materials is now saved in PRO.FILE.

5.3 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](#)



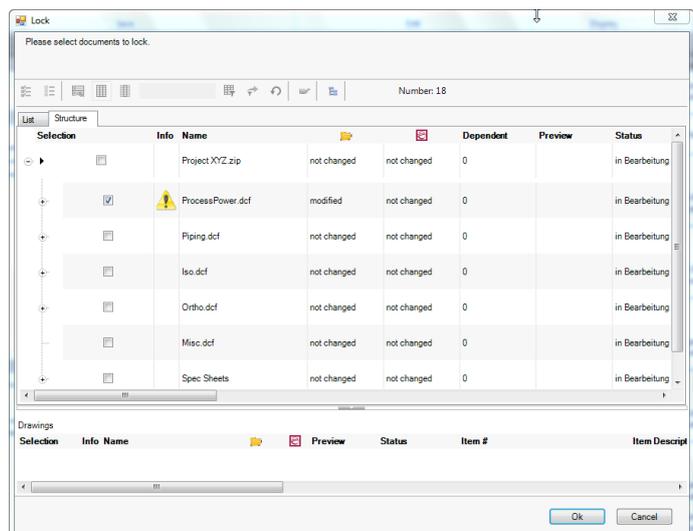
For further information please see the manual "CAD design supported by PRO.FILE".

Dynamic lock dialog with PRO.FILE

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 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.



5.3.1 Starting your changes: "Lock" the CAD document

If a plant project, sub-project or drawing is to be edited, it has to be locked by the user!



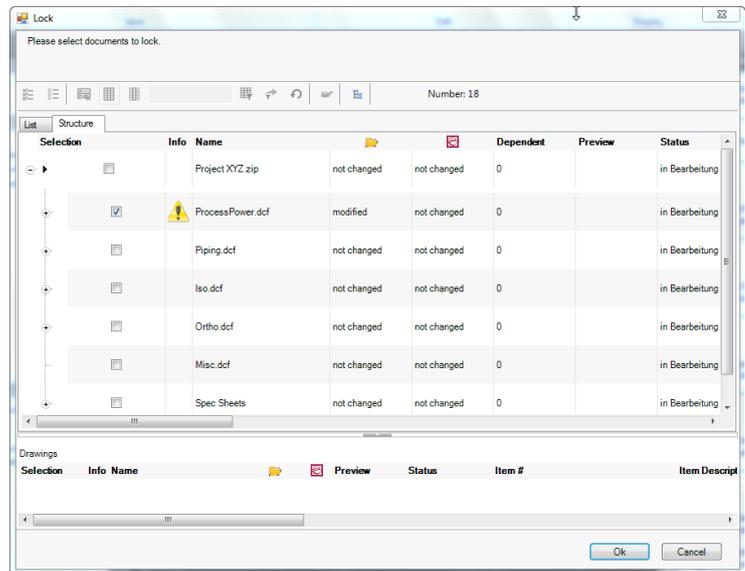
Function call:

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

Proceed as follows

1. Go to the "PRO.FILE" menu in AutoCAD Plant 3D.
2. 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](#)").



3. Select all documents you want to lock. To do so, activate the checkboxes for the desired documents.
4. 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.

5.3.2 "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.



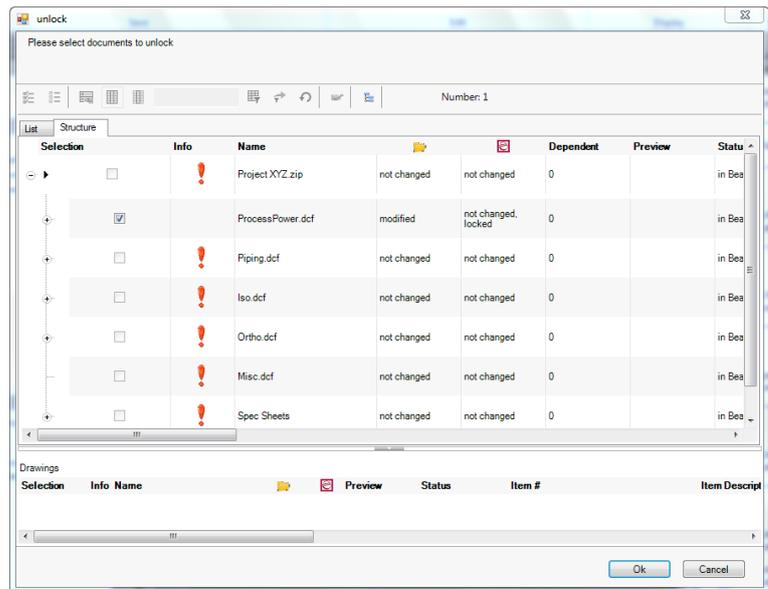
Function call:

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

Proceed as follows

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

⇒ 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](#)").



3. 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.
4. Confirm your selection with <OK>.



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

5.3.3 Lock/Unlock drawing

In addition to the function "Lock/Unlock" from the category "Edit" of the integration menu, which relates to all components of the project, the area "Handle active drawing" also contains the function "Lock/Unlock". This function only relates to the active drawing within the current project.

The behavior of the function for locking/unlocking drawings is identical to the function of the same name from the "Edit" category", with the difference that the lock/unlock dialog for this function only offers the currently active drawing for selection.

5.4 Delete database relation

This function dissolves the connection between the current Plant project and the document description in PRO.FILE. The project thus no longer has a PRO.FILE connection. The project is now treated as a locally saved Plant project and marked accordingly in the Workcenter.

The current project can be changed in AutoCAD Plant 3D and saved again in PRO.FILE with a new name and document ID.

The "old" project still exists in PRO.FILE. The function "Delete database relation" does not delete any documents.



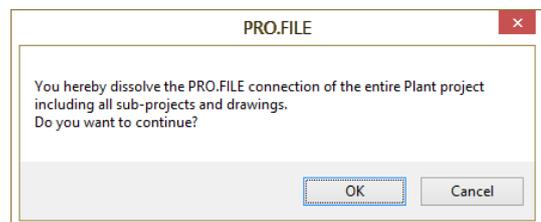
Function call from the PRO.FILE menu in AutoCAD:

"PRO.FILE" => "Edit" => "Delete database relation"

Proceed as follows

1. Go to the "PRO.FILE" menu in AutoCAD Plant 3D.
2. Select "Edit" => "Delete database relation".

⇒ The message, that this will remove the database relation of the entire project (including all its components) is displayed.



3. Confirm this message with <OK>.

⇒ The PRO.FILE connection for the entire project 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 Plant project: Saving CAD objects for the first time](#)".

5.5 Make pipe specification available

With the integration PRO.FILE – AutoCAD Plant 3D, it is possible to save the pipe specifications to be used centrally in PRO.FILE. The advantage is that the usage of such specifications can then be referenced by the various projects, so that these do not have to be saved anew for each project. For this, the administration of pipe specifications in PRO.FILE has to be activated.

If the pipe specifications are managed centrally in PRO.FILE, you can use the function "Make pipe specification available" within a AutoCAD Plant 3D to use it within your project. The desired specification is then selected in PRO.FILE and copied to the project.



Note:

To be able to use the function "Make pipe specification available", the administration of pipe specifications in PRO.FILE has to be activated. Furthermore, the SpecSheets have to be imported to PRO.FILE. Please contact your administrator for further information.



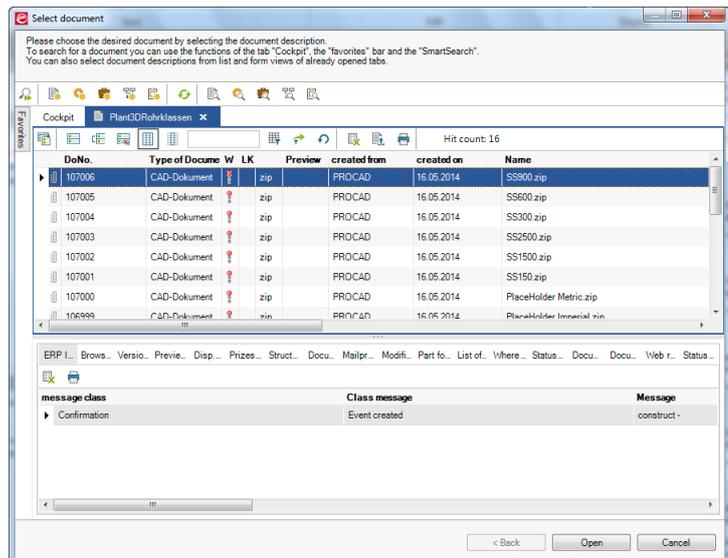
Function call:

"PRO.FILE" => "Edit" => "Make pipe specification available"

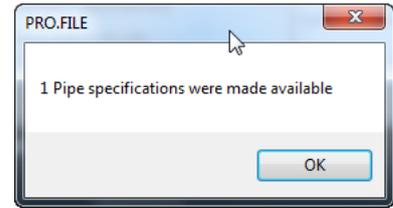
Proceed as follows

1. Go to the "PRO.FILE" menu in AutoCAD Plant 3D.
2. Select "Edit" => "Make pipe specification available".

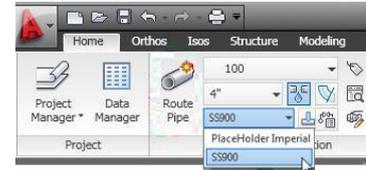
⇒ The Checkout wizard displays the PRO.FILE GUI as it was used the last time. The document of the desired pipe specification (SpecSheets are packed in zip files) can now be selected to be opened in AutoCAD Plant 3D. The behavior of the wizard corresponds to the normal "Open" process (see chapter ["Open: Loading documents from PRO.FILE into AutoCAD Plant 3D"](#)).



3. Confirm your selection via the <Open> button.
- ⇒ The transfer of the pipe specification to your project is confirmed.



- ⇒ The pipe specification can then be used for the creation of pipe routes on a drawing.

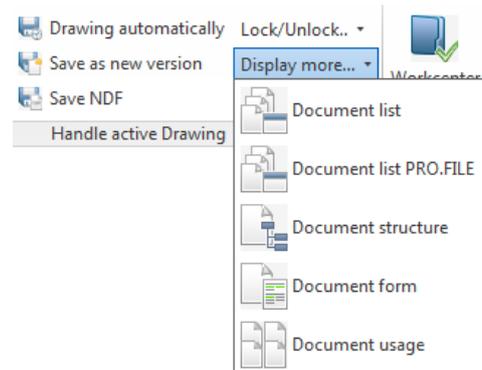
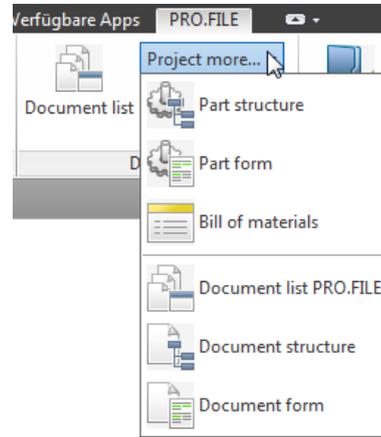


6 Display: Access to PRO.FILE information

The areas "Display" and "Handle active drawing" of the integration offer different functions that give you direct access to PRO.FILE information on the current CAD drawing or the project/sub-project.

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 "Display":

- [The document list](#)
- [Display: 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

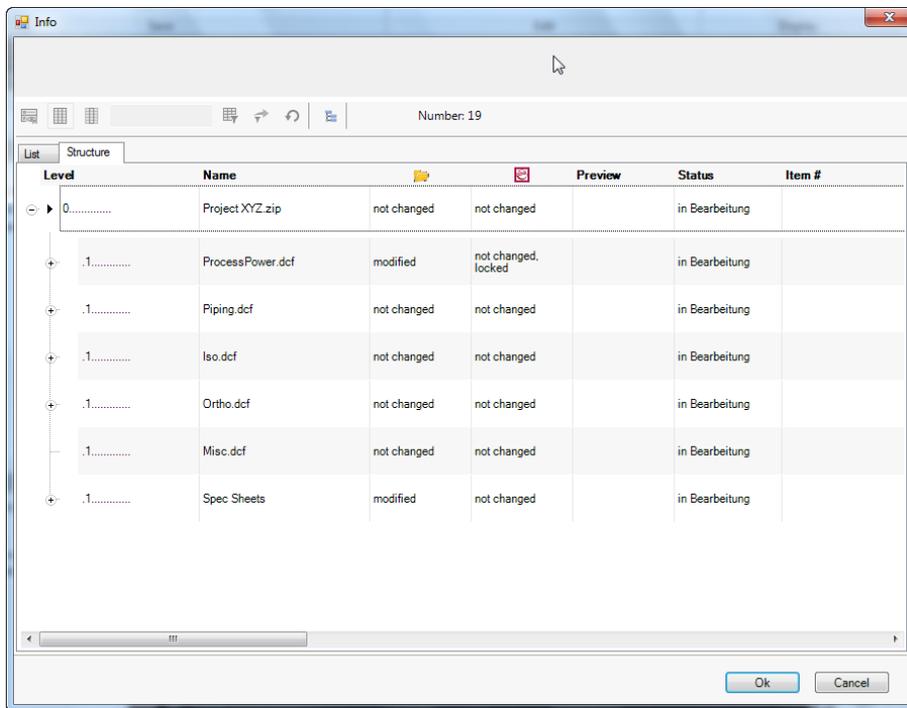
The document list displays the local and PRO.FILE information on the currently active project. With the function "Document list" you can also see which documents (sub-projects, drawings) are currently used in your plant project.



Function call from the PRO.FILE menu in AutoCAD Plant 3D:

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

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



You find the following information:

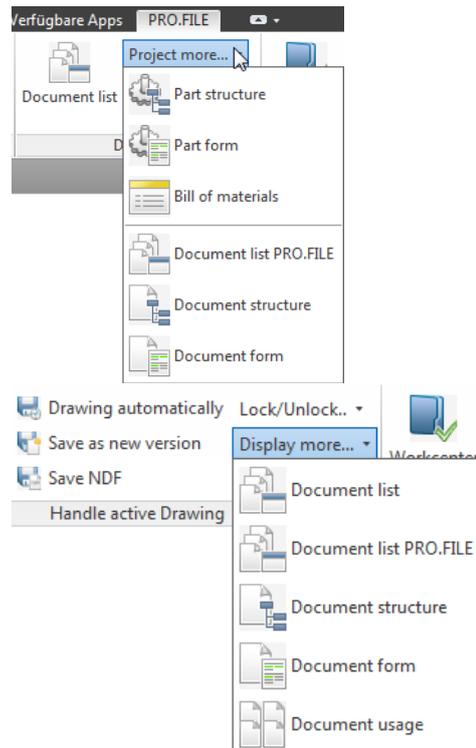
- The documents contained in the project that are in your local working folder.
- Information relating to the status of the CAD documents – both local as well as the PRO.FILE status.

The document list also contains – as all other dialog screens of the integration do – different search and list functions (see chapter "[More comfort: search and list functions in the dialog screens](#)").

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

The areas "Display" and "Handle active drawing" of the integration offer 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 Plant 3D.
- 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 Plant 3D:

"PRO.FILE" => "Display" => "..."

"PRO.FILE" => "Handle active drawing" => "..."



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 are not yet visible in PRO.FILE.

The following views are available for **project documents**:

Part structure

Displays the structure overview of the part to which the current project is attached in PRO.FILE.

Part form

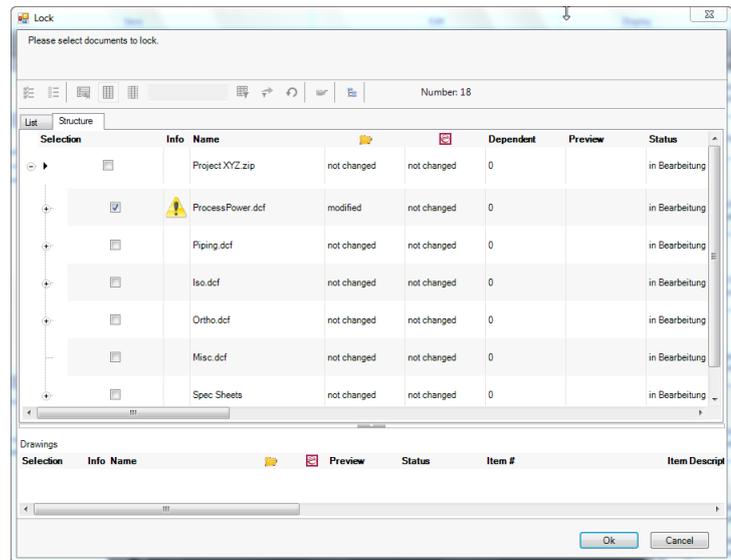
Displays the part master record form of the current project in PRO.FILE.

Bill of materials	The functionality for bills of materials is not available in the current version of the integration.
Document list PRO.FILE	With this function you can display the list of documents belonging to the active project in PRO.FILE.
Document structure	With this function you can see which documents (sub-projects, drawings) are used within your project.
Document form	Displays the document description form of the current project in PRO.FILE.
	The following views are available for drawings :
Document list	With this function you can display the list of documents belonging to the active project to see the current local status as well as the PRO.FILE status of the documents.
Document list PRO.FILE	With this function you can display the list of documents belonging to the active drawing in PRO.FILE.
Document structure	With this function you can see which documents (part drawings) are used within your drawing.
Document form	Displays the document description form of the current drawing in PRO.FILE.
Document usageFehler! Textmarke nicht definiert.	Shows a list of objects, in which the current drawing is being used.

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 Plant 3D integration contain different search and list functions, as known from the PRO.FILE GUI:



Via these buttons, the following functions are available:

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

-  **Hide selected rows:**

If several rows of a list are selected, these rows can be hidden from the list with this button.
-   **Search in all columns / Search in active columns:**

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

 - : The search is performed across all columns in the list.
 - : The search is performed for the active column only. A column is activated by clicking the respective column header.
-  **Define Filter pattern / Filter:** A character string can be entered into the entry field located within the icon bar. Here you can use the already described wildcards/meta characters.

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

If the search pattern is found, all matching data records are highlighted.
-  **Next found pattern:**

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

If rows of a list have been hidden, this button can be used to display them again.
-  **PRO.FILE list selection:**

The entries of the selected rows are selected and opened in a list in PRO.FILE. This 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	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.
	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	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.

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 Plant 3D:
 "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".

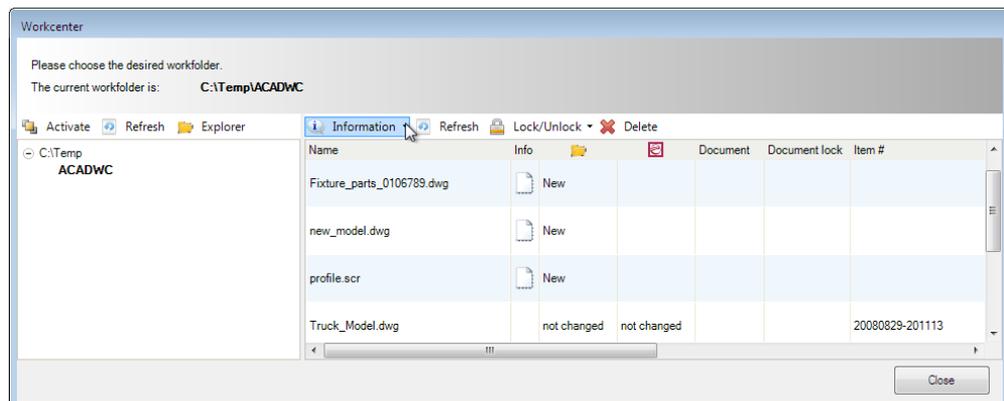


Attention when working with several work folders:
 Please be careful when working with several work folders. It may happen that the loaded CAD document has loaded components from work folder A as well as from work folder B. When deleting files from one work folder, make sure that these components are 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.

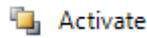
7.1.1 Workcenter functions

The Workcenter is divided into two areas

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



The functions for the directory structure



Activate

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



Refresh

The view of the directory structure is updated.



Explorer

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

The functions for the working directory



Information ▾

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

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



Refresh

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



Lock/Unlock ▾

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



Delete

The marked documents are deleted from the directory.

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



Clear workspace

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



Filter

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



Update version

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.

8 Index

A	
assign created object to PRO.FILE project	37
assign part master records	53
Assignment of PID objects	53
C	
Checkout wizard	19
collective assignment	57
collision check	28
contents	6
Create bill of materials	61
D	
delete database relation	65
display	69
display information	71
document description	
create	36
document list	70, 74
search and list functions	74
status information	75
E	
editing functions	60
F	
file level	19
first steps	7
function overview	10
further functions	56
I	
integration menu	9
integration PRO.FILE AutoCAD Plant 3D	6
L	
lock	62, 63
lock drawing	65
M	
make pipe specification available	67
Managed Version	49
O	
open	15
drawing	22
sub-project	22
with newest versions	23
with released versions	23
open documents from PRO.FILE	14
P	
Part assign	53
part master record	32
PRO.FILE login	9
S	
save	29
automatically	47
changed documents	41
drawing	40
NDF	52
new version	48
sub-project	40
save as new version	48
save plant project	
first time	31
search	19
T	
table of contents	3
U	
unlock	62, 64
unlock drawing	65
Update	22
update properties	60
W	
Workcenter	7, 77
functions	77