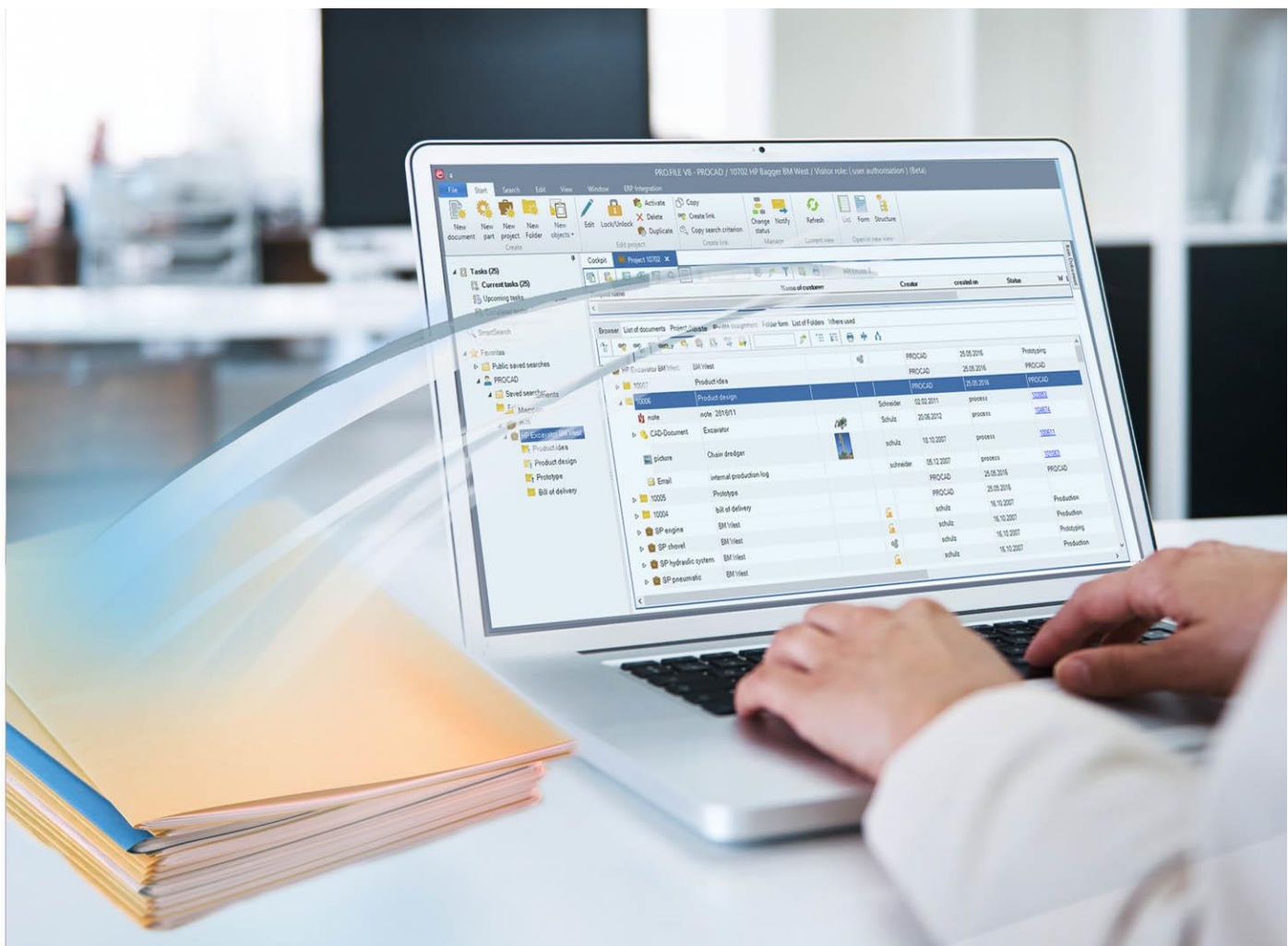


Functions of the Integration PRO.FILE PTC Creo Parametric

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Table of contents

Table of contents	3
About this manual	7
1 The integration PRO.FILE – Creo Parametric	8
1.1 The contents of this manual	8
1.2 Definitions and object dependencies.....	9
2 Let's get started: First steps with the PRO.FILE integration.....	10
2.1 Only upon first start: Setting up the local work folder	10
2.2 Where to find the functions of the integration?	12
2.3 How to log in to PRO.FILE?	13
2.4 A brief overview: The functions of the integration.....	14
3 Opening CAD Documents from PRO.FILE in Creo Parametric	15
3.1 Open: Opening CAD Documents from PRO.FILE.....	16
3.1.1 Working with the Checkout wizard to search for CAD documents.....	21
3.2 Open drawing	23
3.3 Open with all drawings	24
3.4 Browse versions.....	25
3.5 Open CAD documents from PRO.FILE for editing.....	27
3.5.1 Open via drag & drop	28
3.6 Attention: Opening of locally existing files	29
4 Lock/Unlock: Who can change when?	30
4.1 Starting your changes: "Lock" the CAD document	31
4.2 The "Unlocking" of CAD documents.....	33
4.3 Lock selection: Select inactive documents for locking	34
4.4 Unlock Selection: Select inactive documents for unlocking.....	34
5 Save: How to save CAD data and changes to PRO.FILE?	35
5.1 Saving CAD objects for the first time	36
5.1.1 Checkin wizard Step 1: Creating or assigning a part master record in PRO.FILE.....	37
5.1.2 Checkin wizard Step 2: Creation of the document description in PRO.FILE	41
5.1.3 Checkin wizard Step 3: Assignment of the created objects to a PRO.FILE project.....	42

5.2	Save: Saving changed CAD documents	44
5.3	Managed Copy	47
5.3.1	Exchanged or not: What must be observed strictly?	47
5.3.2	Requirement 1: Create an independent copy of a model	48
5.3.3	Requirement 2: Exchange a model in an higher-level assembly using "Managed Copy"	50
5.3.4	How is the function "Managed Copy" executed?	57
5.3.5	Search and replace with Managed Copy	59
5.4	Managed Copy automatic	60
5.5	Update BOM	60
5.6	Save with Phantoms	62
5.6.1	Save an assembly as a phantom	63
5.6.2	Usage of phantom parts from a phantom assembly	63
5.6.3	Externally used phantoms – please cut database relation first	64
5.6.4	Mixed design: Phantom assemblies and PRO.FILE objects	64
5.7	Save automatic	65
5.8	Save all automatic	68
5.9	Save select	69
5.10	Save: All instances	69
5.11	Save: All instances automatic	70
5.12	Neutral Data format: Save plotfile	71
5.13	Append local file	72
5.14	Add Document	74
5.15	Detach	75
6	Functions for the version administration	76
6.1	Save as new version	76
6.2	Replace previous version: "direct" or "all"	78
6.3	Managed Version	80
6.3.1	The proceeding for "Managed Version"	81
6.4	Version Top Down	83
7	DB-Relation: The relation of CAD objects to the PRO.FILE database	84
7.1	Update Parameters	84
7.2	Break up tree	85
7.3	Info Feature	85

7.4	Info selected feature	86
7.5	Modify part feature relations	86
7.6	Break up feature bar relations	86
7.7	Update Parameters select	86
7.8	Get Softlinks	86
8	Drawing: Insert and update bill of materials	87
8.1	Create new BOM	87
8.2	Update BOM	88
9	Show: Displaying PRO.FILE information on the active object	89
9.1	Data overview: The document list	89
9.2	Show: Information on a CAD document in PRO.FILE	91
9.2.1	Show Workcenter	91
9.2.2	Document list	91
9.2.3	Document form	92
9.2.4	Document browser	92
9.2.5	Document usage	92
9.2.6	Document versions	92
9.2.7	Part form	92
9.2.8	Part browser	92
9.2.9	Part usage	92
9.2.10	Show BOM	92
9.3	Direct information in the dialog screens	93
9.3.1	More comfort: search and list functions in the dialog screens	93
9.3.2	Up to date or not: Display of status information	95
10	Extra: Additional functions	97
10.1	Workcenter	97
10.1.1	Workcenter functions	98
10.1.2	Creating and changing an active work folder	99
10.2	Delete local	100
10.3	Startmodel	100
10.4	Managed Rename: Renaming in the structure	101
11	Info selection: Additional selection functions of the integration	105

11.1 Object type and Object selection 105

11.2 Select object with mouse click 106

12 Tips and tricks on specific aspects 107

12.1 Placing with help levels and help coordinates 107

12.2 Handling of drawing frames in PRO.FILE 107

12.2.1 Drawing frames as "phantom Objects" 108

12.2.2 Administration of drawing frames referred with PRO.FILE..... 108

12.2.3 Do not save drawing frames in PRO.FILE..... 108

13 Index..... 109

About this manual

Step-by-step instructions:

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

For quicker finding within the manual, step-by-step instructions are marked with a margin heading.

Menu sequences and function calls

Menu sequences and function calls explained in this manual are marked in bold and in quotation marks.

Example:

"File" => "New" =>

"Document description"

Buttons and keys

Keys and buttons are highlighted by angle brackets.

Example:

"Confirm with <OK>."

Notes and warnings

To highlight special information the following icons are used:



Function call:

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



Example:

Boxes marked with this icon give subject-relevant examples for the usage of command lines, configuration strings and other software-relevant entries.



Note:

Boxes marked with this icon contain useful hints on the operation, configuration or installation of the PRO.FILE software.



Attention:

All information given in these boxes is very important and should be read carefully! Non-observance of these hints may lead to wrong functioning, display problems or other negative consequences.



Important notes:

The "stop sign" warns you of possible entry or operation errors, which may lead to loss of data!



Attention – Undo not possible:

All entries and configurations described in these boxes have to be made carefully, because they cannot be undone!

1 The integration PRO.FILE – Creo Parametric

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

When assemblies are saved, PRO.FILE automatically generates bills of materials and proofs of usage. These can be included in the drawing title block along with the product data.

Interfaces allow the transfer of product data (part master data, bills of material and CAD documents) specified during the design process with Creo Parametric to ERP systems.

1.1 The contents of this manual

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

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 Creo Parametric. The following topics will be addressed:

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



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

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

1.2 Definitions and object dependencies

For the description of the functions, the following terms are used in this manual:

- **Simple objects** are objects that **do not contain any external references** i.e. that do not depend on other objects.
- An **assembly** is dependent on its built-in individual parts. The individual parts are **components** of the assembly.
- A **reflected object** is dependent on its original. The original is treated as a **component** reflection of the object.
- A **drawing** is dependent on the displayed geometric models that it contains. The geometric models are **components** of the drawing.
- An **Instance** of a part family is dependent on its related mother variant. The mother variant is treated as a **component** of the part family.

Object dependence

The CAD-system Creo Parametric forms the basis of a parametric system. Then what follows is the building of complex hierarchies and complex referencing, formed by the modeling of individual parts of an assembly. It results in object dependence, whose observation for the construction success, have a strong influence.

Examples of object dependence can be:

- Assembly dependent on its built-in individual parts
- Reference reflected parts dependent on the original
- Drawings dependent on their contained displayed geometry model
- Instance of a part family dependent on the mother variant ("**generic**")

These object dependences are usable in many ways, and are regulated by the planned use of the assembly part (e.g.: reusability, assembly box construction etc.).

The Integration PRO.FILE – Creo Parametric analyses the dependence structure of Creo Parametric objects, as it lays them down in PRO.FILE, and also carries out their administration. The object dependence can be made transparent for the user. For this, the following PRO.FILE-functions can be used:

- Document form
- Document usage
- Structure Browser
- Document / part Usage
- Show BOM

These functions simplify the construction with Creo Parametric and the analysis of the effects of a change to a CAD-object.

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 Creo Parametric.

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

- [Only upon first start: Setting up the local work folder](#)
- [Where to find the functions of the 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 Creo Parametric, and also saved and versioned from Creo Parametric 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

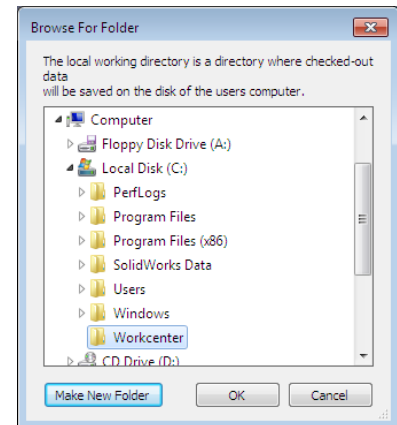
The Creo Parametric features require the availability of all related data. Without defining a local work folder, it is not possible to work with the integration PRO.FILE – Creo Parametric.

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

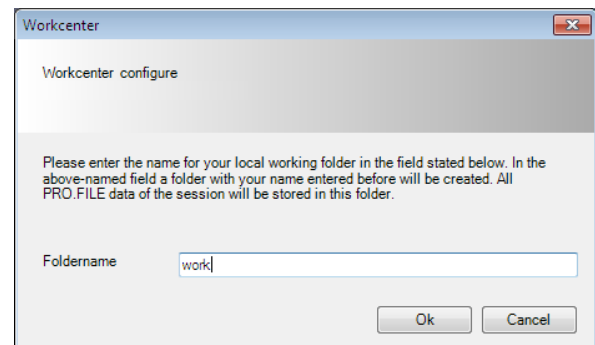
Proceed as follows:

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 Creo Parametric under "Extra" => "Workcenter".

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

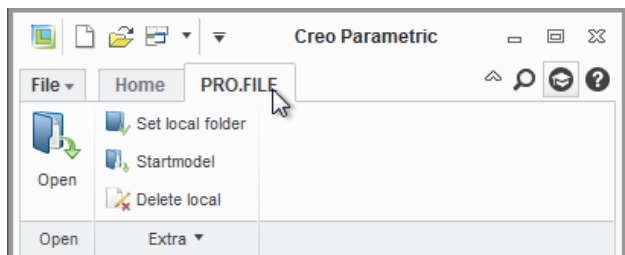
2.2 Where to find the functions of the integration?

Access to the functions of the integration is possible via the menu "PRO.FILE" in the Creo Parametric menu bar:

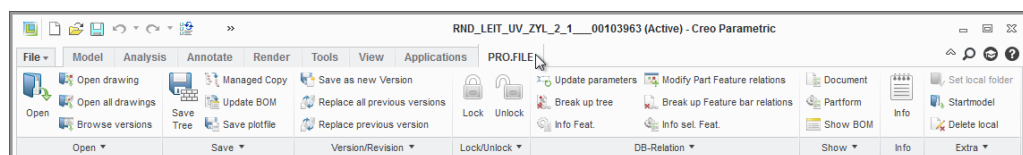
1. Start **Creo Parametric**.
2. Select the "PRO.FILE" tab.
3. Select the desired integration function.

The menu of the PRO.FILE integration is context-dependent on the loaded Creo Parametric file:

- When Creo Parametric is started without a loaded CAD object, the start menu of the integration is displayed.



- If a CAD document is loaded, all basic functions of the integration are displayed in the PRO.FILE menu:



PRO.FILE is activated by any of these functions. Depending on the selected function via the integration, the desired PRO.FILE window is opened automatically.

The user can thus use all relevant PRO.FILE functions of data management via the menu in Creo Parametric.

2.3 How to log in to PRO.FILE?

If you access a PRO.FILE function for the first time within an Creo Parametric session, you have to log in to PRO.FILE.

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

1. In the login screen, please enter:

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

2. Confirm with <LOGIN>.

⇒ The PRO.FILE home screen is now displayed.



Note: No login required in case of "Autologin"

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

2.4

A brief overview: The functions of the integration

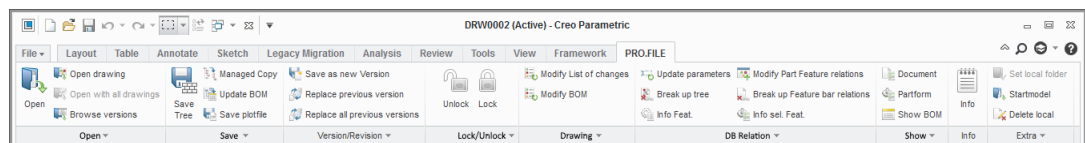
The Integration PRO.FILE – Creo Parametric allows you to utilize many different functions. You can access these functions using the PRO.FILE menu in Creo Parametric.



Function call from the PRO.FILE menu in Creo Parametric:

Creo Parametric menu: "PRO.FILE"

The following chapters describe the different integration functions that are available via this menu:



- [Open: Opening CAD Documents from PRO.FILE](#)

Select CAD objects with the Checkout wizard in PRO.FILE and load them in Creo Parametric. During this step you can directly reserve the CAD components to be loaded for you and thus lock them for other users.

- [Save: How to save CAD data and changes to PRO.FILE?](#)

Save new CAD objects to PRO.FILE or save changes to existing documents.

- [Functions for the version administration](#)

Create and edit versions in PRO.FILE

- [Lock/Unlock: Who can change when?](#)

Lock and unlock PRO.FILE objects for editing.

- [Drawing: Insert and update bill of materials](#)

Transfer PRO.FILE data from the bill of materials into the drawing.

- [DB-Relation: The relation of CAD objects to the PRO.FILE database](#)

Changing the PRO.FILE database connection of the CAD objects active in Creo Parametric.

- [Show: Displaying PRO.FILE information on the active object](#)

Direct access to the selected information from PRO.FILE on the active CAD object.

- [Extra: Additional functions](#)

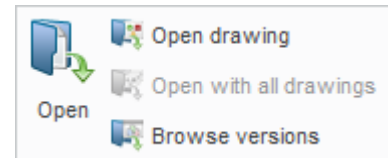
Selecting Work directories in the Workcenter or start models; using Managed Rename.

3 Opening CAD Documents from PRO.FILE in Creo Parametric

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: Opening CAD Documents from PRO.FILE](#)
- [Open drawing](#)
- [Open with all drawings](#)
- [Browse versions](#)
- [Attention: Opening of locally existing files](#)



Attention:

The data loaded from PRO.FILE in Creo Parametric are not automatically locked when opened in Creo Parametric. The user has to lock the objects manually via the function "Lock".

After the object has been edited and saved back to PRO.FILE, it can be unlocked again, so that it is available to other users. For detailed information see the chapter "[Lock/Unlock: Who can change when?](#)".



Note: PRO.FILE checks permissions

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

3.1 Open: Opening CAD Documents from PRO.FILE

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

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

The opening process takes place in several steps

Step 1

Using the PRO.FILE function "Open"



Function call from the PRO.FILE menu in Creo Parametric:

"PRO.FILE" => "Open"

1. Go into the menu bar of Creo Parametric into the menu "PRO.FILE".
 2. Select the menu entry "Open".
- ⇒ "Open" loads documents and its components as it is defined in the parameter "Version load options dialog" in the PRO.FILE Management Console.
- ⇒ The Checkout wizard for the selection of documents is displayed.

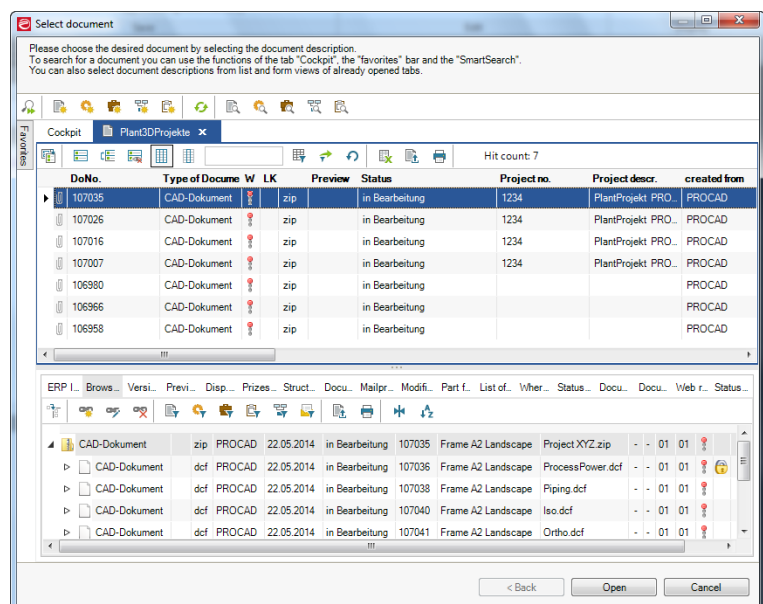
Step 2

Selecting the desired document in the Checkout wizard

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

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

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



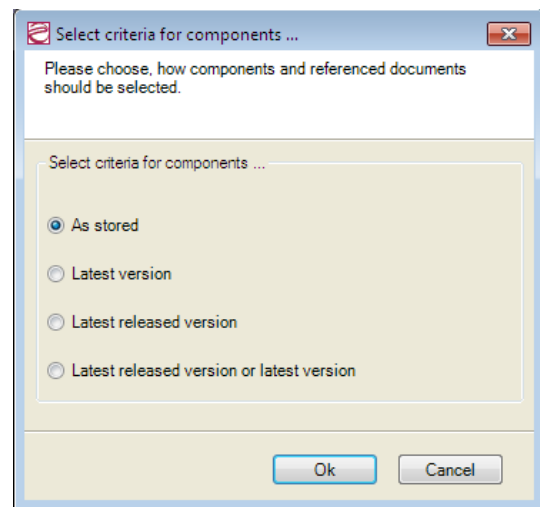
4. If the desired document is displayed in a list view, you can **select** it. (If the desired document is displayed in a form view, it is already selected).
 5. Click **<Open>**.
- ⇒ The Checkout wizard closes and the dialog screen for the loading type is displayed.

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

Step 3

Versions in the structure: How are the components to be opened?

- ⇒ When a CAD object with structure (assembly, drawing, etc.) is loaded via the "Open" function, PRO.FILE checks whether the CAD document contains components for which versions exist.
- ⇒ If this is the case, you can then decide how PRO.FILE is to load the assembly/drawing in question.



6. Select the desired method and confirm with **<OK>**.
 - **As stored:**
the assembly is loaded with the components it was **recently** saved with. Changes to parts that resulted in **new** version of the parts are ignored.
 - **Latest version:**
PRO.FILE replaces all CAD documents for which it finds a newer version and loads the assembly/drawing with the newer object versions. You thus get an updated version of the assembly/drawing.



Note:

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

- **Latest released version:**

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

**Attention:**

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

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

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

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

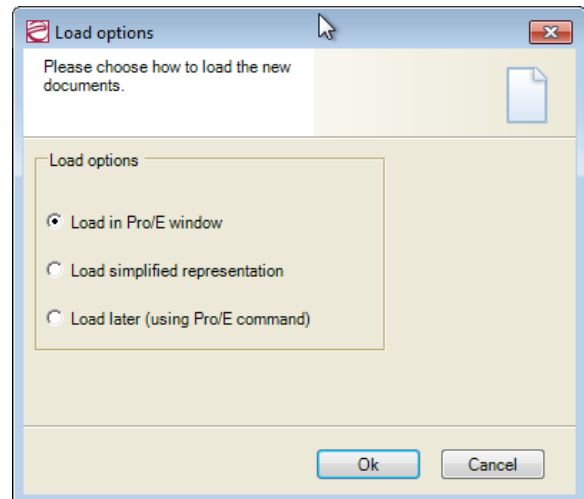
**Note: Open with version browser**

For the targeted selection of the desired version you can use the function "[Browse versions](#)". With the version browser you can open assemblies in dynamic compositions – the user can decide for each component, which version of this component is to be loaded from PRO.FILE.

Step 4**Which loading option is to be used?**

A window then opens, asking you want to load the object immediately in a new window or later:

- ⇒ If you select "In separate window", the selected files are loaded in a separate window in Creo Parametric.
- ⇒ If you select "Load simplified representation" the list of simplified views is offered. The user can then choose which representation is to be opened in Creo Parametric.



- ⇒ If "Load later (using Pro/E command)" is selected, the selected object with all linked elements is loaded into the "proework" directory of the Workcenter. From there you can load the checked-out document via the Creo Parametric function "read" into the current session.

7. Confirm your selection with <OK>.

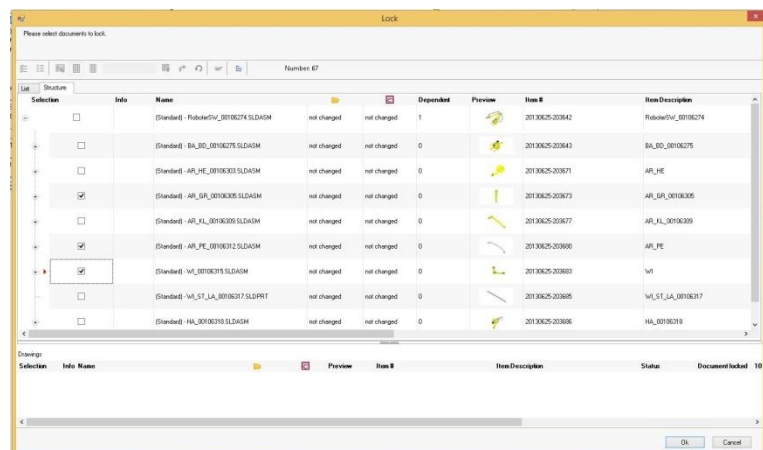
Step 5: You want to edit? Then you must lock the document(s)

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

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

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

(Information on the functions and status indicators can be found in the chapter "[Data overview: The document list](#)")



- 8. Select all documents you want to lock with the corresponding check boxes.
- 9. Confirm your selection with <OK>.



⇒ The selected document is now opened with its components in the CAD system. The "Open" procedure is thus finished.

Detailed information on the locking of document can be found in the following chapter: ["Lock/Unlock: Who can change when?"](#).



Note: Why can I not lock a document?

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

This may have two **reasons**:

- The document is already locked by a different user. You can see who the locking user is by selecting document in PRO.FILE and looking at the dependent tab "Status information".
- The document is in a workflow status, in which you are not allowed to edit the document. This is typically the case for "released" documents.

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

3.1.1

Working with the Checkout wizard to search for CAD documents

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

The aim of this procedure is:

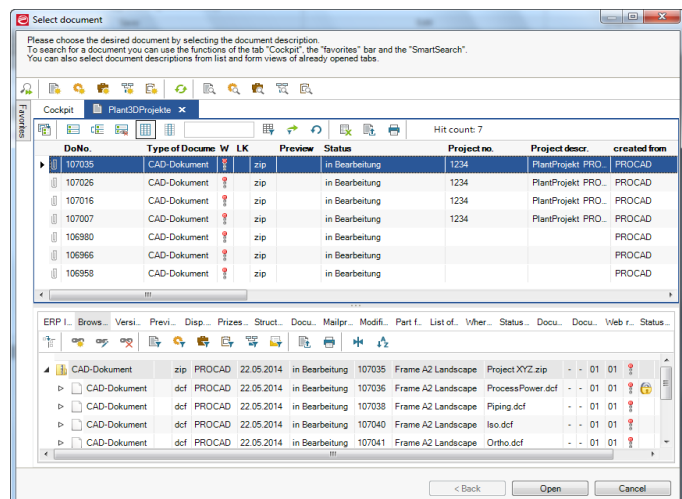
- For PRO.FILE to know which document is to be opened, the desired document description must
 - either be selected in a list view or a structure browser
 - or be displayed in a form view
- Then the button <Open> at the right bottom of the Checkout wizard has to be clicked.

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:

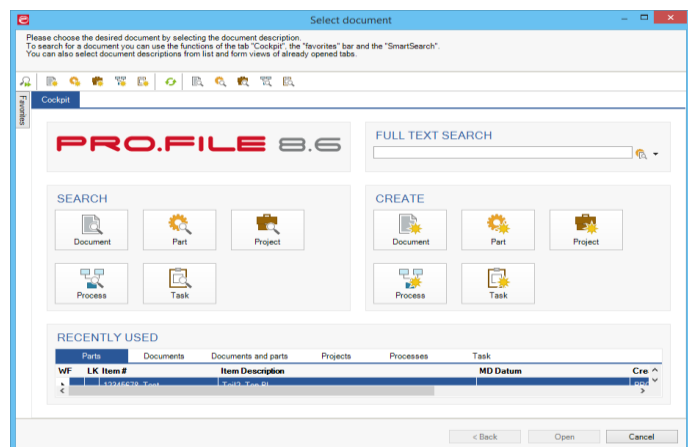
- If search results had previously been opened in a list or form view in a separate tab, you can directly access the displayed documents.

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".

3.2 Open drawing

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

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



Note:

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



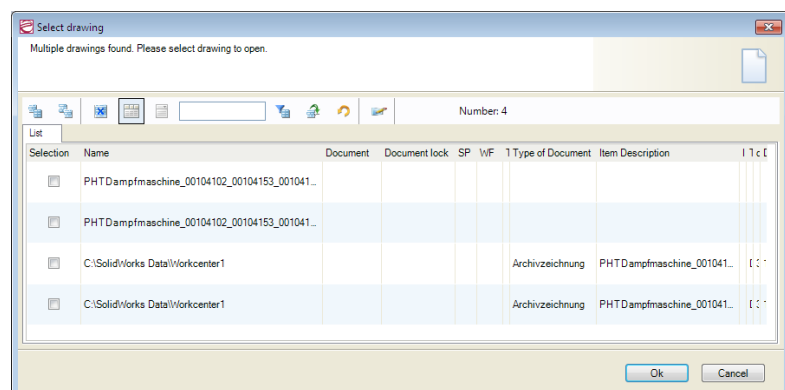
Function call from the PRO.FILE menu in Creo Parametric:

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

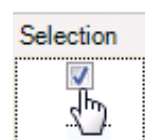
Proceed as follows:

1. Make sure that the CAD document, for which the drawing is to be opened, is displayed in Creo Parametric.
2. Select the "PRO.FILE" menu in Creo Parametric.
3. Select the function "Open..." => "Open drawing".

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



4. Select all drawings you want to open in Creo Parametric with the checkboxes in the first column.
5. Confirm your selection with <OK>.



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

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

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

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



8. Confirm your selection with <OK>.

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

Additional functions for the opening of drawings can be found in the following chapters:

- [Open with all drawings](#)



Attention:

If locally changed files are overwritten with data from PRO.FILE, the local files and changes made therein are lost!

If versions are overwritten, the locally existing assemblies now point to the newly loaded version that has replaced the original version! Please keep that in mind when overwriting local files with data from PRO.FILE.

3.3

Open with all drawings

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



Note: Drawings for the components in PRO.FILE

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



Function call from the PRO.FILE menu in Creo Parametric:

"PRO.FILE" => "Open..." => "Open with all drawings"

If a drawing is already saved in PRO.FILE for the objects (and the user has the appropriate access right to open the drawings, the drawings are opened in Creo Parametric for display.

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

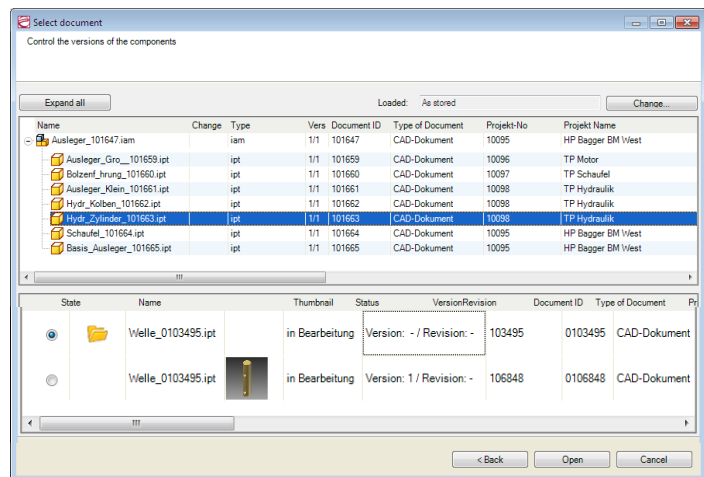
3.4

Browse versions

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

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

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



The version browser is divided into two areas:

The document structure (top):

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

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

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

The version window (bottom):

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

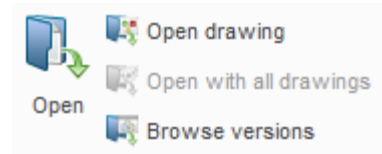


Function call from the PRO.FILE menu in Creo Parametric:

"PRO.FILE" => "Open..." => "Browse versions"

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

1. Select the "PRO.FILE" menu from the menu bar in Creo Parametric.
 2. Select the function "Open..." => "Browse versions".
- ⇒ The Checkout wizard is displayed.
3. Select the desired CAD document and click on the <Open> button.
- ⇒ The Checkout wizard closes.
- Detailed information on the Checkout wizard can be found in the following chapter ["Working with the Checkout wizard to search for CAD documents"](#).
- ⇒ The screen "Select document" is displayed.
4. Select the component, for which you want to make a version selection, in the document structure.
- ⇒ The lower version window now displays all corresponding versions.
5. By toggling the radio button in the first column of the version window you can activate the desired version of a CAD element:










Thumbnail	Name	Status	Version/Revision	Dokument ID
	Bracket_100381	in Bearbeitung	Version: A / Revision: 1	100381

6. Having activated all desired versions, you can leave the version browser by clicking <Open> in order to continue the loading process.
- ⇒ The window "Lock" is displayed.
- ⇒ At this moment, the selected CAD data is not yet locked in PRO.FILE and still available for other users. This means: If you want to edit the CAD data, you have to lock it.
7. Select all documents you want to lock by setting the checkmark for it in the first column.
- Detailed information on the locking of documents can be found in the following chapter ["Lock/Unlock: Who can change when?"](#).
 8. Confirm your selections with <OK>.



- ⇒ The selected CAD components are opened in Creo Parametric.
The process of opening with the version browser is now finished.

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

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

3.5

Open CAD documents from PRO.FILE for editing

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

- Double-click on the file in list or form display.
- Select the document and open it via the menu ribbon "Edit file".
- Select the document and open it via the context menu function "Edit file" => "Edit document".
- Select the file and move it to the CAD GUI via drag & drop.

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

3.5.1

Open via drag & drop

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

**Note:**

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

Proceed as follows

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

**Note:**

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

3.6 Attention: Opening of locally existing files

When a CAD document is opened, all required elements and components are loaded into the current work folder. If the work folder already contains a file of the same name, you will get a list of the elements that are to be overwritten. This also applies for newer or older versions of a CAD documents, which can now be overwritten.



Attention: Risk of data loss

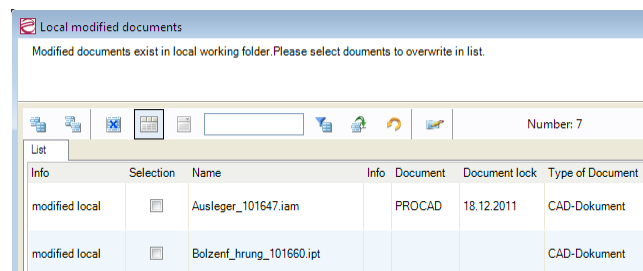
If locally changed files are overwritten with files from PRO.FILE, these local files and all changes to them are irretrievably lost!

If versions are overwritten, the locally existing assembly now point to the newly loaded version which has replaced the original version! You should therefore be careful when deciding to overwrite a locally existing version.

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

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

Different versions are also indicated.



You have the following options of proceeding:

- **Overwrite with status from PRO.FILE:** Activate the checkbox in column "Selection" for the list entries, the local status of which is to be overwritten with the status from PRO.FILE. If you confirms this action with <OK>, all files are copied from PRO.FILE to your workstation.
- **Do not overwrite:** Leave the checkbox unchecked.
- **Load data in a different Workcenter folder:** You can switch to a different working folder via the command "PRO.FILE" => "Extra" => "Workcenter" => "Activate", to avoid the overwriting of data. (See chapter "[Workcenter functions](#)").



Note:

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

4 Lock/Unlock: Who can change when?

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

- Only by using the function "Lock" you can make sure, that other users are not making changes to the same document at the same time.
- With the function "Unlock" the CAD document is made available again to other users for editing.

For detailed information see the following sub-chapters:

- [Starting your changes: "Lock" the CAD document](#)
- [The "Unlocking" of CAD documents](#)
- [Lock selection: Select inactive documents for locking](#)
- [Unlock Selection: Select inactive documents for unlocking](#)



The **locking** of a CAD document makes sure that the CAD document is not modified by other users in the meantime:

- A locked CAD document can be opened by other users via the function "open". However, these other users cannot save back any changes to the locked CAD document to PRO.FILE.
- If the CAD document has been opened and not locked, other users may make changes to the document in the meantime and save those changes back to PRO.FILE. In this case, it will no longer be possible for you to save your own changes back to PRO.FILE.

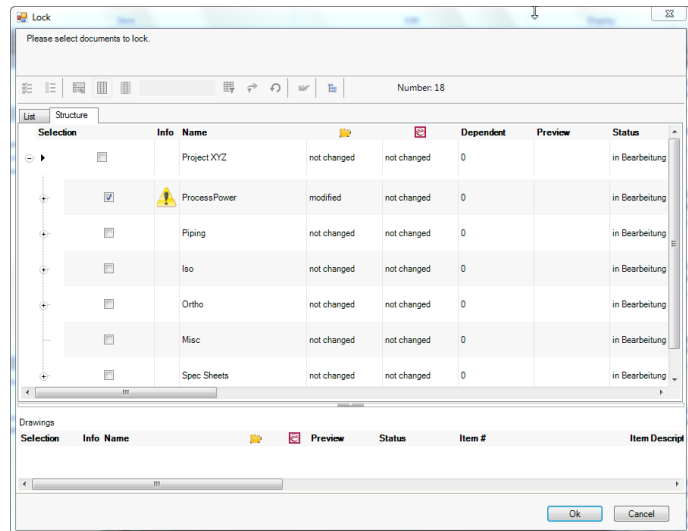
For detailed information please see the manual "CAD design supported by PRO.FILE".

Dynamic lock dialog

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

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

1. Here you can now select the CAD data to be locked by setting the checkmark.
2. Confirm your lock with **<OK>**.



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

4.1

Starting your changes: "Lock" the CAD document

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



Function call from the PRO.FILE menu in Creo Parametric:

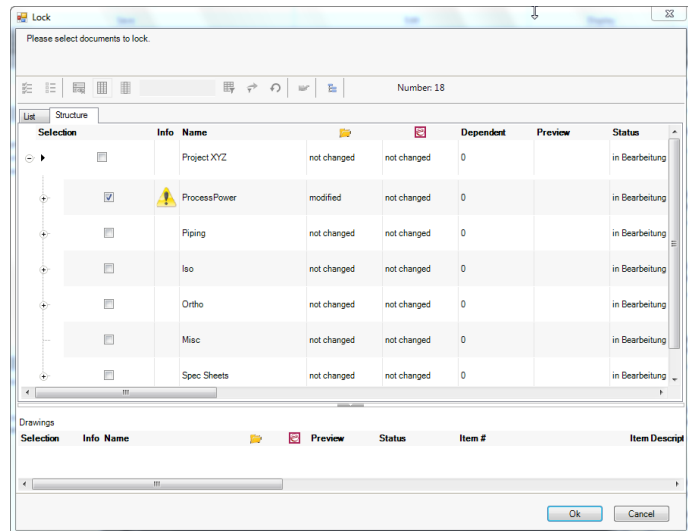
"PRO.FILE" => "Lock/Unlock" => "Lock"

To lock a CAD document manually proceed as follows:

1. Make sure that the CAD document to be locked is displayed in Creo Parametric.
2. Select the menu "PRO.FILE" from the Creo Parametric menu bar.
3. Select the function "Lock/Unlock" => "Lock".

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

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



With the display of status information in this list PRO.FILE checks:

- whether the user has the permission to edit the document.
- whether the active documents are up to date and have not been modified by a different user since their opening.
- whether the active documents does not already have a lock flag.

⇒ If any of these checks returns a negative result, the document cannot be locked!

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

5. Confirm your selections with <OK>.



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

⇒ Once the CAD document is locked, it can be modified. The changes are then saved back to PRO.FILE via the function "Save".



Attention: Changes in the team

It is recommended to lock document you want to edit directly after opening.

4.2 The "Unlocking" of CAD documents

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



Note:

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



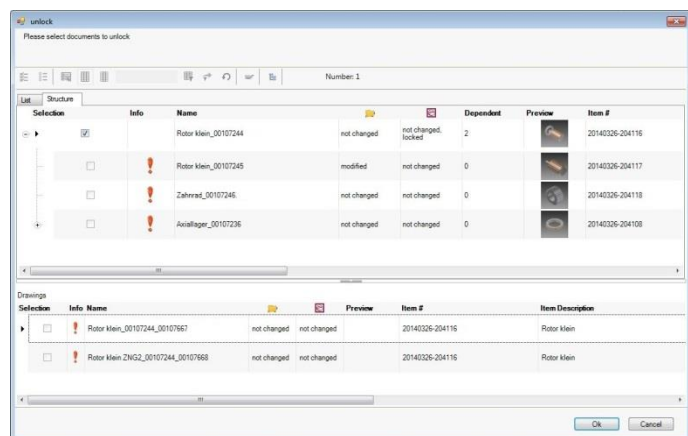
Function call from the PRO.FILE menu in Creo Parametric:

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

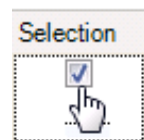
To unlock a document proceed as follows:

1. Make sure that the CAD document to be unlocked is displayed in Creo Parametric.
2. Select the menu "PRO.FILE" from the Creo Parametric menu bar.
3. Select the function "Lock/Unlock" => "Unlock".

⇒ The dialog for unlocking the loaded CAD documents is displayed.
(Information on the functions and status information can be found in the chapter ["Data overview: The document list"](#)).



4. To make the CAD documents saved in PRO.FILE available for other users, select the documents to be unlocked in the list.
5. Confirm your selections with <OK>.



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

4.3 Lock selection: Select inactive documents for locking

If you want to lock an object after it has been opened, you can use the function "Lock".

If you want to lock an object that is loaded in the session, but not activated, then you can use the "Lock selection". You can then select the object of your choice, to which the function "Lock" is applied.

The proceeding for the function "Selection" via the integration menu is described in the chapter "[Info selection: Additional selection functions of the integration](#)".

4.4 Unlock Selection: Select inactive documents for unlocking

If you want to release the reservation on an object, then you can use the function "Unlock".

If you want to release the reservation of an object that is loaded in this session, but not active, you can use the function "Unlock selection".

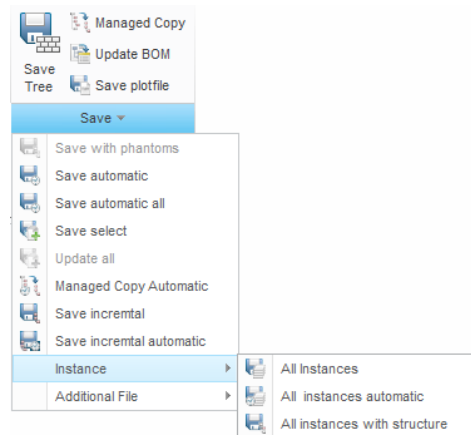
You can then select the desired object, to which the function "Unlock" is applied.

The proceeding for the function "Selection" via the integration menu is described in the chapter "[Info selection: Additional selection functions of the integration](#)".

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

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

- "Save" with the options:
 - ["Saving CAD objects for the first time"](#) and
 - ["Save: Saving changed CAD documents"](#).
- [Managed Copy](#)
- [Update BOM](#)
- [Neutral Data format: Save plotfile](#)
- [Save automatic](#)
- [Save all automatic](#)
- [Save select](#)
- [Managed Copy automatic](#)
- [Save with Phantoms](#)
- [Save: All instances](#)
- [Save: All instances automatic](#)
- [Append local file](#)
- [Add Document](#)
- [Detach](#)



In the menu "Version" you also find the function:

- [Save as new version](#)

This variety of functions is based on one fundamental behavior:

- All functions first check for new, unknown documents. If such documents exist, they are saved to PRO.FILE. Then, locally changed documents are offered for saving, if such documents exist.
- The options "... automatically" and "... phantom" only decide on new, unknown documents to be created.

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

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

Therefore, the description is divided into two chapters:

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



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

Before using the integration PRO.FILE – Creo Parametric please also note the manual "CAD design supported by PRO.FILE". This manual describes additional proceedings and related issues from the designer's point of view.

5.1 Saving CAD objects for the first time

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

The process of saving is carried out in a number of stages. The outcome of each individual stage determines the dialog that will appear for the next stage.

The following procedure is prerequisite for saving:

- Firstly you must make a **local** save of your newly created object. This prerequisite is given by Creo Parametric.
- Then you can save the object to PRO.FILE.

If you want to save CAD documents from Creo Parametric to PRO.FILE, use the menu entry "**Save**" from the "PRO.FILE" menu.



Note:

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



Function call from the PRO.FILE menu in Creo Parametric:

"PRO.FILE" => "Save"

Proceed as follows:

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

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

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

These steps are described in the following sub-chapters.

5.1.1

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:

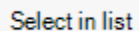
Create new

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:A rectangular button with a thin border and the text "Select in list" in a blue, sans-serif font.**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:

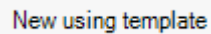
1. Click the option **<Select in list>** to select the desired part description.
- ⇒ The wizard displays the PRO.FILE surface, as it was opened the last time.
2. If the part master record desired for assignment is not yet displayed in a list or form view, you can use the search functions of the home 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:A rectangular button with a thin border and the text "Search" in a blue, sans-serif font.**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:

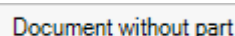
1. Click the option **<Search>** to select the desired part description.
2. Enter the search criteria into the displayed search form and click **<Search>**.
3. Select the desired part master record from the list of results.
4. Confirm your selection with **<OK>**.

New using template:A rectangular button with a light gray border and a light gray background. The text "New using template" is written in a sans-serif font, with "New" in blue, "using" in red, and "template" in black.**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:A rectangular button with a light gray border and a light gray background. The text "Document without part" is written in a sans-serif font, with "Document" in blue, "without" in red, and "part" in black.

Usage:

- For special usage purposes it may be necessary to create a document description without the link to a part master record.
- You can therefore use this option to skip the creation or selection of the part master record and to proceed directly with the saving of the document description.

Proceeding:

1. Click the option **<Document without part>**.
⇒ The Checkin wizard for parts is skipped. The Checkin wizard for the document description is displayed.

**Attention:**

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

5.1.2

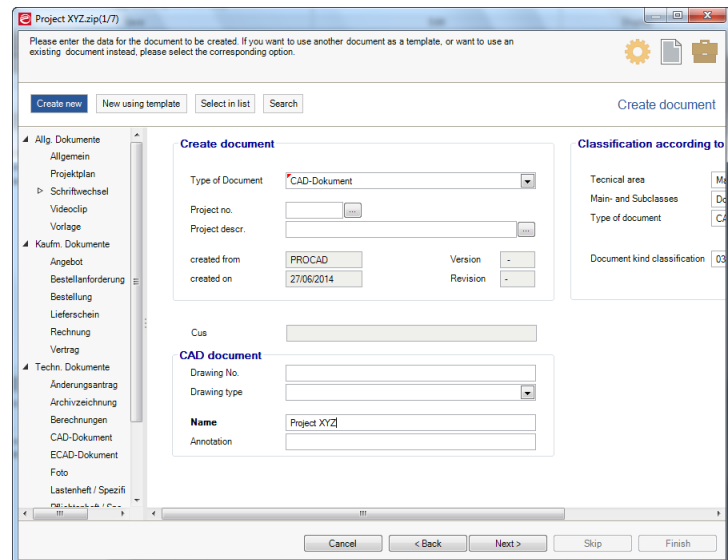
Checkin wizard Step 2: Creation of the document description in PRO.FILE

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

1. 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](#)".

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

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

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

5.1.3

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

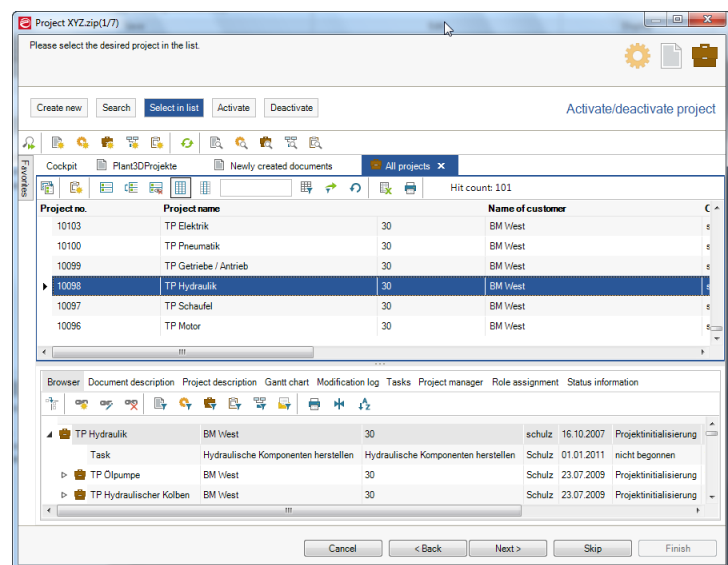
In this step the CAD data just saved can be assigned to a specific PRO.FILE project.

**Note: Usage of PRO.FILE projects**

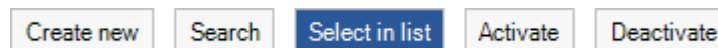
The third step in the Checkin wizard is intended for the use of PRO.FILE projects. If you are not using PRO.FILE projects, you can skip this step with the option **<Skip>**.

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

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

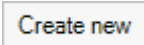
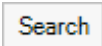
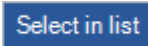

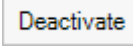


Here, too, the Checkin wizard offers different options that can be accessed via the operations bar:

**Attention: Project must be activated**

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

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

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

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

Proceeding:

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

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

5.2 Save: Saving changed CAD documents

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

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

Before the saving process is started, PRO.FILE checks, whether the user has the permission to change the object in question. Furthermore, the program checks whether the active copy of the CAD object is up to date.



Attention: Only documents that have been locked can be saved

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

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

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

- [Save: Saving changed CAD documents](#)
- [Save automatic](#)
- [Save select](#)
- [Managed Copy](#)
- [Managed Copy automatic](#)
- [Save: All instances](#)
- [Save: All instances automatic](#)
- [Neutral Data format: Save plotfile](#)

The area "Version/Revisions" offers the function:

- [Save as new version](#)

This chapter describes the proceeding for saving changed CAD documents.



Function call from the PRO.FILE menu in Creo Parametric:

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

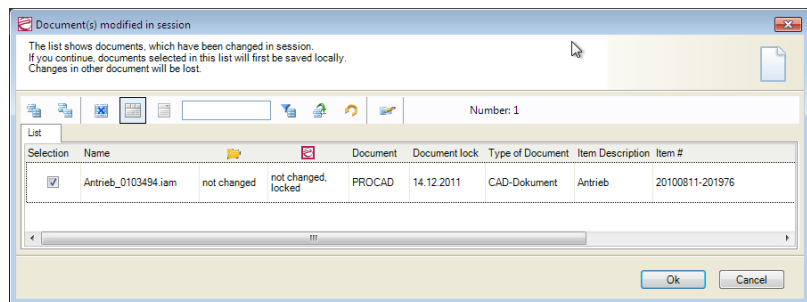
Proceed as follows:

1. Go to the integration menu "PRO.FILE" in Creo Parametric.
2. Select the function "Save" from the section "Save".

- ⇒ PRO.FILE recognizes the CAD document as a PRO.FILE object and automatically goes into change mode.

Selection of the documents to be saved

- ⇒ The dialog for the selection of CAD documents to be saved is displayed.



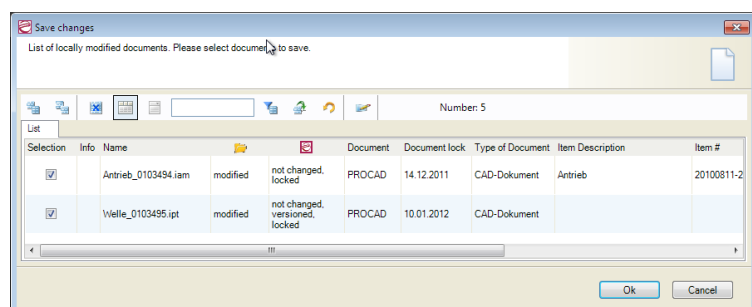
- ⇒ The dialog displays a list with all changed CAD documents from the current Creo Parametric session. (Information on the functions and status information can be found in the chapter "[Data overview: The document list](#)".)
- ⇒ For assemblies, the structure is analyzed for changed CAD documents and the list of all documents of this assembly is preselected.
- ⇒ For this list the access permissions for saving the changes of the user are checked. (If the CAD document had been locked before for editing, this prerequisite is fulfilled.)
3. Select all documents you want to save in PRO.FILE. To do so, activate the checkboxes for the desired documents.
 4. Confirm your selection with <OK>.



Locally changed documents in the structure:

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

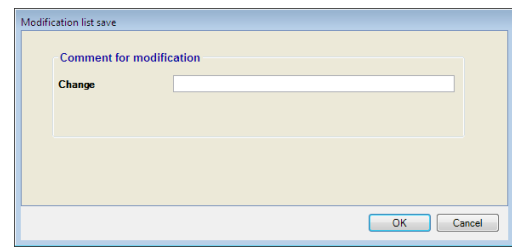
5. Select all locally changed components you want to save to PRO.FILE.



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

Optional: Enter modification comment

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



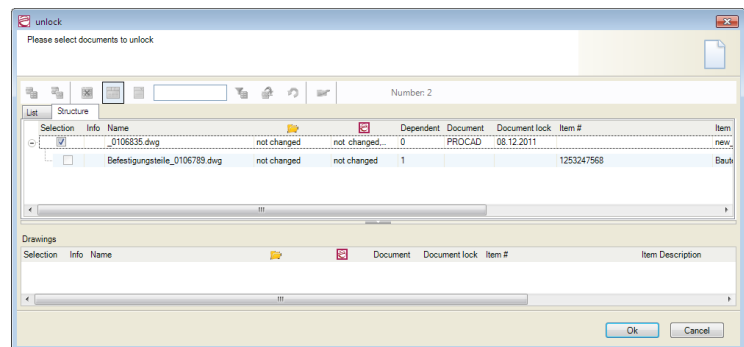
7. Confirm your modification comment with <OK>.

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

Locking/unlocking the saved documents

- ⇒ The dialog for documents to be unlocked after saving is displayed.

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



- ⇒ If documents from PRO.FILE had been locked for editing in Creo Parametric, this lock is not automatically removed after saving. The documents remain locked and cannot be changed by other users.
- ⇒ If you are finished with your changes to the CAD document, you can now unlock the document to make it available for other users.
- ⇒ To make this process easier, the PRO.FILE CAD documents that are still locked are displayed in the list.

8. To make the documents available again for other users, select the documents in the list. To do so, activate the checkboxes for the desired documents.



9. Confirm your selection with <OK>.

- ⇒ The lock flag for the selected documents is now removed.
- ⇒ The saving of your changes to PRO.FIL is now finished.

5.3 Managed Copy

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

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

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

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

5.3.1 Exchanged or not: What must be observed strictly?

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

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



Attention: Result of Managed Copy

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

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

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

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

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

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

5.3.2

Requirement 1: Create an independent copy of a model

The requirement is:

- You want to create a copy of an existing model (assembly, subassembly, CAD part).
- The reference of the higher-level assembly should furthermore refer to the original model, not to the created copy.
- Is there a reference from the model you want to copy to a higher-level assembly, the references should not be exchanged but furthermore refer to the original model.
- The created copy of the assembly should be saved independently in PRO.FILE.

Approach 1A**Only the model you want to copy is loaded in Creo Parametric**

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

**Attention: higher-level assemblies are not be opened**

Using this approach, all higher-level assemblies have to be closed! If higher-level assemblies are opened in the Creo Parametric session, a model copied via "Managed Copy" is exchanged in the higher-level assemblies.

3. Activate the model to copy in the Creo Parametric session.
 4. Call the PRO.FILE function "Managed Copy" up, as described in the following chapter ["How is the function "Managed Copy" executed?"](#).
- ⇒ The created copy of the model is saved in PRO.FILE.
- ⇒ The created copy of the model is referenced in no higher-level assembly.

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

1. Open the drawing of the subassembly to copy via the **"Managed Copy"** in the Creo Parametric session.
- ⇒ Higher-level assemblies in which the model to copy is referenced, can remain open.
2. Activate the drawing of the subassembly to copy in the Creo Parametric session.
 3. Call the PRO.FILE function "Managed Copy" up, as described in the following chapter ["How is the function "Managed Copy" executed?"](#).
 4. Select both the drawing and model to copy in the wizard of **"Managed Copy"** as well as the subassemblies and CAD parts in the structure of the model you want to copy.

- ⇒ The created copy of the drawing as well as the copy of the subassembly are saved in PRO.FILE.
- ⇒ The created copy of the model is referenced in no higher-level assembly.

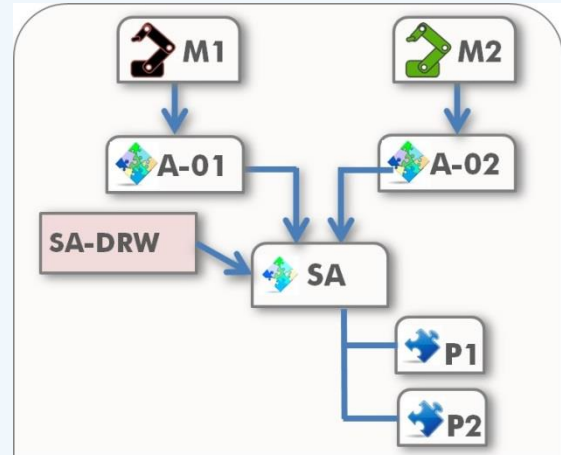


Note: In the structure the drawing of a model is listed above the model

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

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

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



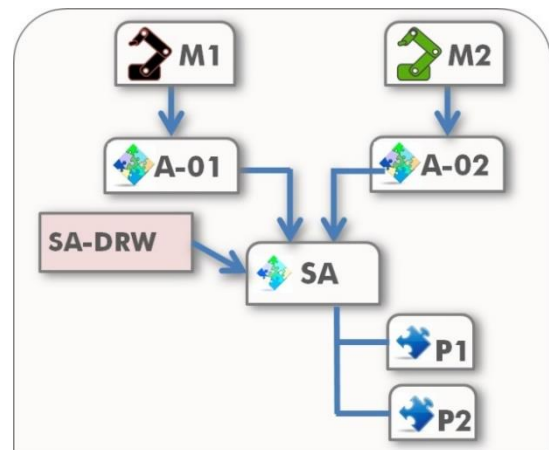
Case study for approach 1B

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

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

Situation:

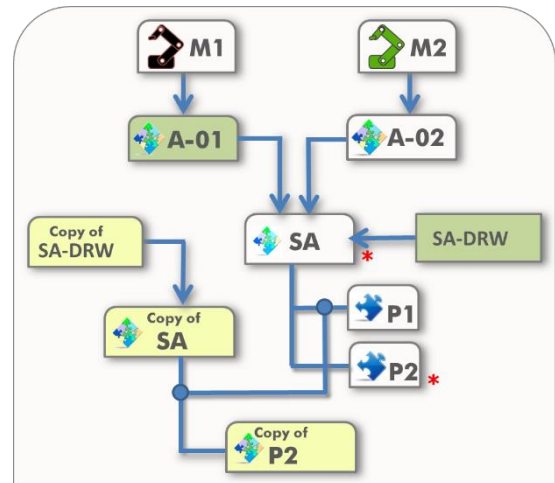
- 2 assemblies ("A-01" and A-02") are loaded in Creo Parametric.
- Assembly "A-01" is part of the machine "M1", assembly "A-02" is part of the machine "M2".
- The subassembly "SA" is installed in both assemblies.
- There is the drawing "SA-DRW" for the subassembly.



- The subassembly "SA" is active in the Creo Parametric session.
- The function "Managed Copy" is called up for "SA-DRW"!
- The sub-assembly "SA" and the drawing "SA-DR" are selected for "Managed Copy".
- Part "P2" is selected for "Managed Copy", Part "P1" is not.

Result:

- The subassembly "SA" is copied via "Managed Copy".
- The references are explicitly exchanged in the higher-level drawing "SA-DRW" by the Integration PRO.FILE Creo Parametric.
- Due to the fact that "Managed Copy" has been executed via the drawing, the assemblies "A-01" and "A-02" furthermore use the "original" subassembly "SA".
- A copy of CAD part "P2" is created, to which the "Copy of SA" refers.
- Like "SA", "Copy of SA" refers to the not copied CAD part "P1".



5.3.3

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

The requirement is:

- You want to create a copy of an existing model (assembly, subassembly, CAD part).
- The copy of the model should exchange the original model.
- The references in the higher-level assemblies should be exchanged and refer to the copied model.

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

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

1. Open all higher-level assemblies in which you want to exchange the model to copy in the Creo Parametric session.
2. Open the model to copy via the "Managed Copy" function in the Creo Parametric session.

**Attention: higher-level assembly opened**

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

3. Activate the model to copy in the Creo Parametric session.
 4. Call the PRO.FILE function "Managed Copy" up, as described in the following chapter [How is the function "Managed Copy" executed?](#).
- ⇒ The created copy of the model is saved in PRO.FILE.
- ⇒ In all higher-level assemblies, which are loaded in a Creo Parametric session, the references are updated and refer to the copied model.
- ⇒ Higher-level assemblies are not automatically saved to PRO.FILE.



Attention: Higher-level assemblies are not saved automatically

The references in higher level assemblies are only updated and point to the copied model after these higher-level assemblies are loaded in Creo Parametric and explicitly saved via the function "Save" of the integration.

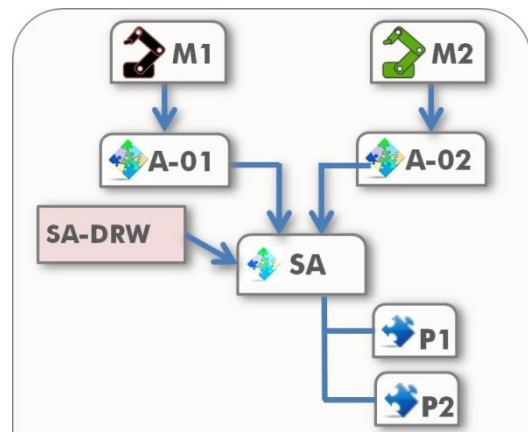
Case study for approach 2A

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

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

Situation:

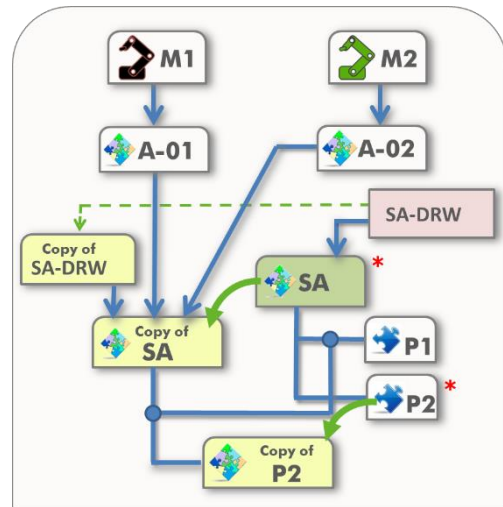
- 2 assemblies ("A-01" and "A-02") are loaded in Creo Parametric.
- Assembly "A-01" is part of the machine "M1", assembly "A-02" is part of the machine "M2".
- The subassembly "SA" is installed in both assemblies.
- There is the drawing "SA-DRW" for the subassembly.



- The subassembly "SA" is active in the Creo Parametric session.
- The function "Managed Copy" is called up for the subassembly "SA".
- The subassembly "SA" itself is selected for "Managed Copy".
- CAD part "P2" is selected for "Managed Copy", CAD part "P1" is not.

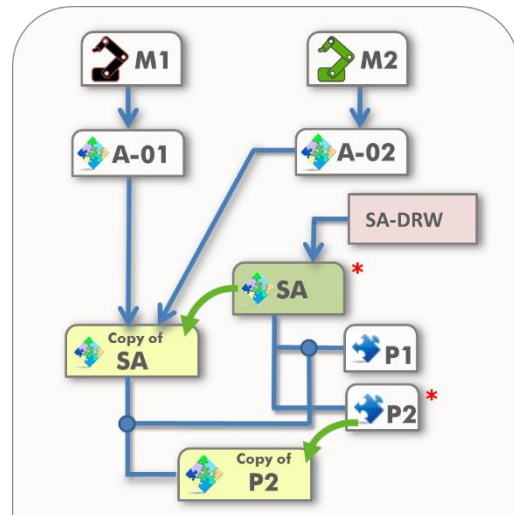
Result IF the drawing was also selected for Managed Copy:

- Due to the fact that "SA" as an active document has been selected to copy via "Managed Copy" and both assemblies "A-01" and "A-02" are loaded in Creo Parametric, the references in both assemblies are exchanged by Creo Parametric.
- In both assemblies the copied subassembly "Copy of SA" is installed.
- A copy is created of the drawing SA-DRW. "Copy of SA-DRW" refers to "Copy of SA".
- A copy of CAD part "P2" is created, to which the "Copy of SA" refers.
- Like "SA", "Copy of SA" refers to the not copied CAD part "P1".



Result IF the drawing was NOT selected for Managed Copy and NOT loaded in Creo Parametric:

- Due to the fact that "SA" as an active document has been selected to copy via "Managed Copy" and both assemblies "A-01" and "A-02" are loaded in Creo Parametric, the references in both assemblies are exchanged by Creo Parametric.
- In both assemblies the copied subassembly "Copy of SA" is installed.
- No copy is created of the drawing SA-DRW. The drawing SA-DRW still refers to "SA".
- A copy of CAD part "P2" is created, to which the "Copy of SA" refers.
- Like "SA", "Copy of SA" refers to the not copied CAD part "P1".





Behavior of the drawing

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

- If the drawing "SA-DRW" were loaded in the Creo Parametric session, it would refer to "Copy of SA", too.
- If the drawing "SA-DRW" is not loaded, but already added to the PRO.FILE structure, the reference is explicitly exchanged and updated by the Integration PRO.FILE-Creo Parametric.

Approach 2B

Exchange a model in a specific assembly

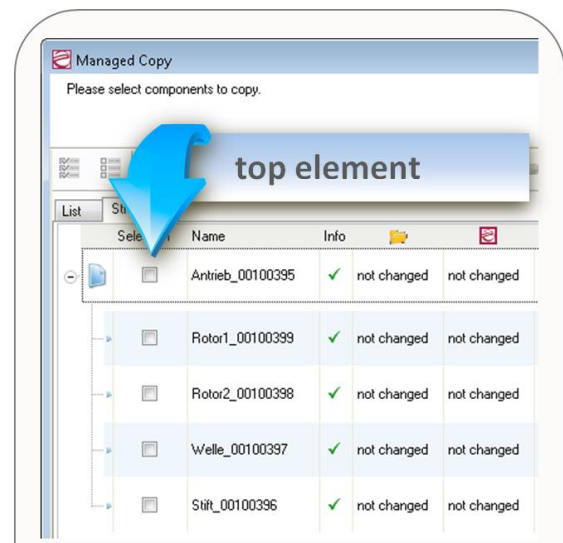
1. Open and activate the higher-level assembly, in which the model (subassembly/CAD part) should be exchanged via **"Managed Copy"** in the Creo Parametric session.

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

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

3. Select in the wizard of "Managed Copy"

- Not the higher-level assembly, which is shown as the top element (top element in a structure).
- Only the model to copy - as well as the subassemblies and CAD parts in the structure of the model.



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

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



Note: Using this approach, only the references are exchanged

The approach avoids the automatic exchange of models copied with "Managed Copy" in opened, higher-level assemblies.

The automatic exchange via "Managed Copy" would only access the assembly chosen as the top element, but not the lower level, in which the model to copy is located.

Due to this reason, using this approach the copied model is exchanged only in the assembly selected for "Managed Copy", but not in the other opened assemblies, in which it is installed.



Note: drawings do not have to be explicitly loaded in the session

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

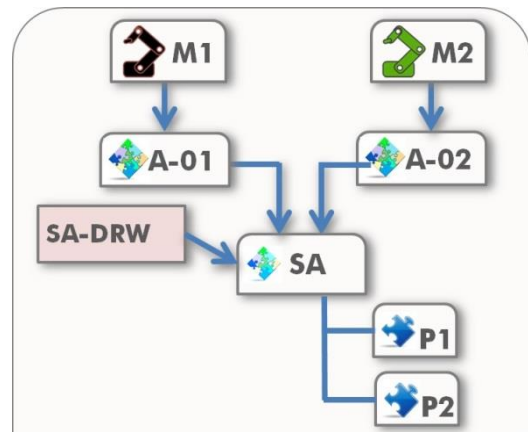
Case study for approach 2B

Replace "SA" only in "A-02"

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

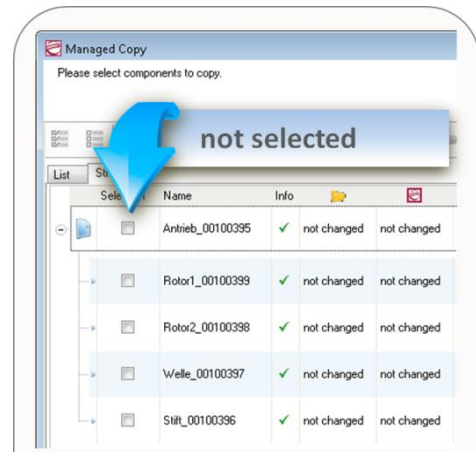
Situation:

- 2 assemblies ("A-01" and A-02") are loaded in Creo Parametric.
- Assembly "A-01" is part of the machine "M1", assembly "A-02" is part of the machine "M2".
- The subassembly "SA" is installed in both assemblies.
- There is the drawing "SA-DRW" for the subassembly.



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

- The function "Managed Copy" is called up for the assembly "A-01".
- In the wizard the subassembly "AS" is selected for "Managed Copy", the assembly "A-01" itself is not selected.
- CAD part "P2" is selected for "Managed Copy", CAD part "P1" is not.



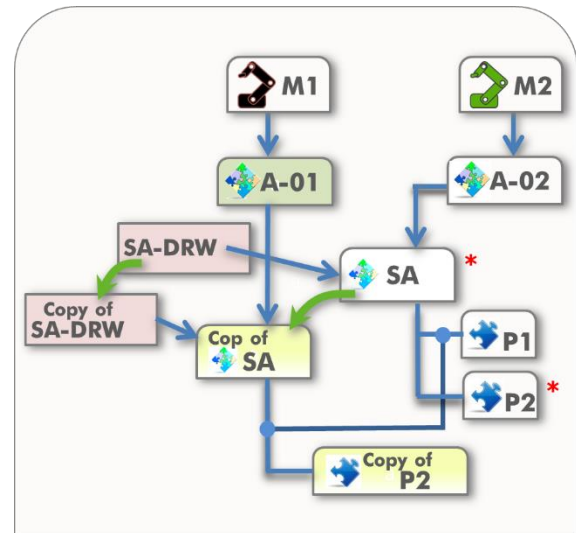
When the sub-assembly is selected, the drawing is automatically selected as well

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

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

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

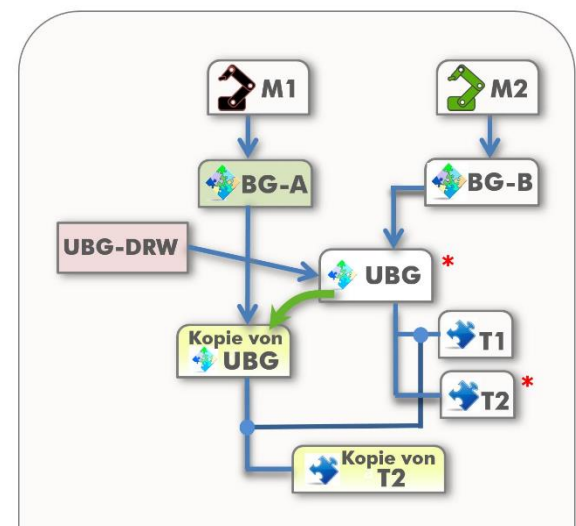
- The sub-assembly "SA" is copied with "Managed Copy".
- The integration explicitly exchanges the references in assembly A-01, so that these references point to the copied sub-assembly "Copy of SA".



- The drawing "SA-DRW" is copied. The drawing "SA-DRW" refers to the sub-assembly SA, the drawing "Copy of SA-DRW" refers to the copy of "SA".
- The assembly "A-02" still refers to sub-assembly "SA".
- A copy is created of "P2", which is referenced by "Copy of SA".
- Like "SA", "Copy of SA" refers to the not copied part "P1".

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

- The sub-assembly "SA" is copied with "Managed Copy".
- The integration explicitly exchanges the references in assembly A-01, so that these references point to the copied sub-assembly "Copy of SA".
- The assembly "A-02" still refers to sub-assembly "SA".



- The drawing "SA-DRW" refers to "Copy of SA"
- A copy is created of "P2", which is referenced by "Copy of SA".
- Like "SA", "Copy of SA" refers to the not copied part "P1".

5.3.4 How is the function "Managed Copy" executed?



Attention: Result of Managed Copy

The result of "Managed Copy" depends on the CAD documents opened in the Creo Parametric session and which CAD document is selected for "Managed Copy"!

See the previous chapter: ["Exchanged or not: What must be observed strictly?"](#).



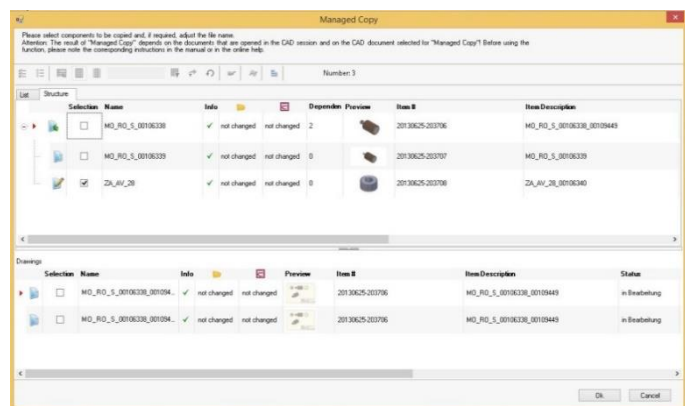
Function call out of the PRO.FILE menu in Creo Parametric:


"PRO.FILE" => "Managed Copy"

Proceed as follows

1. Select the menu item "PRO.FILE" in the menu bar of Creo Parametric.
 2. Click on "**Managed Copy**" in the menu bar of the PRO.FILE Integration.
- ⇒ The wizard of Managed Copy starts and supports you in your further approach.

⇒ The Integration PRO.FILE-Creo Parametric always determines the substructure based on the active CAD document. The substructure of the CAD document depends on the CAD system.



- ⇒ In a second step the substructure is expanded by the related drawings. This "special provision" is required because the drawings are listed above the model depending on references.
- ⇒ In the wizard of "Managed Copy" the tree structure, determined and expanded by drawings, is shown, so that the documents to copy with their dependent data (nodes) can be selected.
- ⇒ The top node and the first step are already folded out. Further steps can be folded out by a click on the structure symbol .
- ⇒ The column "Info" contains further information, e.g. when a part cannot be copied.
- ⇒ The "status" columns shows the current processing status of an object in the working directory and in PRO.FILE (see chapter: ["Up to date or not: Display of status information"](#)).

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



Note: Exchange of components in assemblies

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

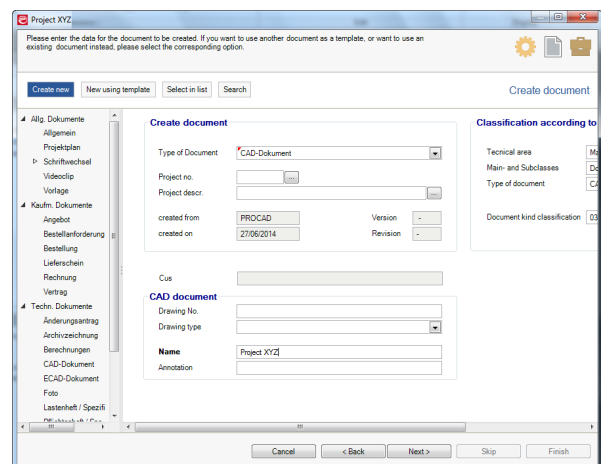
4. **Edit file name:** You can edit and adjust the file names directly in the list, by a click on the column name.



5. Execute this selection and editing of file names for all components to copy.
6. Confirm with <OK>.
- ⇒ If you click on <OK>, the PRO.FILE database reference for all selected objects is deleted. Afterwards the thus created local copies are checked into PRO.FILE. For all not selected components only the references are copied.
7. To complete the process "Managed Copy", all selected components have to be added to the newly created part and document descriptions.

⇒ Therefore appears:

- The check-in wizard to add the part description in PRO.FILE
- then (depending on the configuration) the check-in wizard to add the document description in PRO.FILE
- finally (depending on the configuration) the check-in wizard for the project assignment in PRO.FILE
- The information is requested for each selected component.



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

**Note:**

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

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

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


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

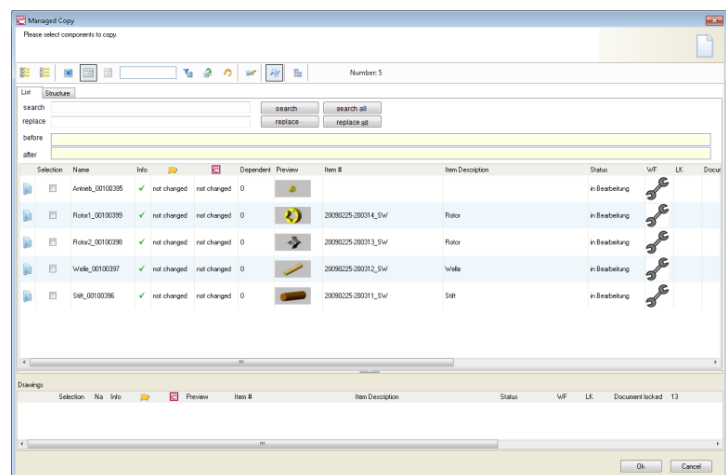
5.3.5

Search and replace with Managed Copy

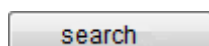
The file names of assemblies and parts can be edited during the execution of "Managed Copy". This is also possible via the function "search and exchange".

Proceed as follows

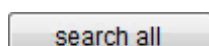
1. Select the list view in the "**Managed Copy**" window.
2. Activate the display "**search and exchange**" by a click on the button .
3. Enter a string to search for in the field "**search**".
4. Enter a string in the field "**exchange**" with which the string to search should be exchanged.



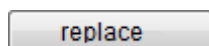
Now you have different possibilities, to execute "search and exchange".



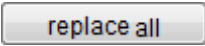
searches and selects the next hits in the list



searches and selects all different hits in the list



replace the next hit



replace all hits in the list

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

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

5.4

Managed Copy automatic

The function "Managed Copy automatic" combines the functions

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

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

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

You will find detailed information for this process in chapter ["Save automatic"](#).



Function call out of the PRO.FILE menu in Creo Parametric:

"PRO.FILE" => Area "Save" => "Managed Copy automatic"

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



Note:

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

5.5

Update BOM

The function "Update BOM" enables you to save a bill of materials to PRO.FILE or update it at a later point in time.

**Note:**

Whether a bill of materials is created or updated during the saving process depends on the setting of the parameter "Update BOM during save" in the PRO.FILE Management Console. There, you also specify whether suppressed components are to be included in the bill of materials.

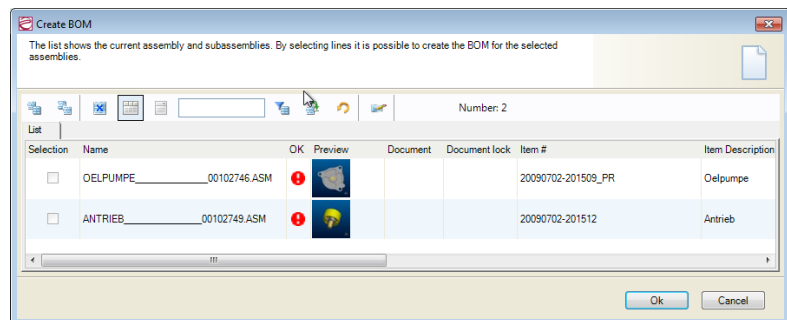
**Function call from the PRO.FILE menu in Creo Parametric :**

"PRO.FILE" => Area "Save" => "Update BOM"

Proceed as follows:

1. If you want to update a bill of materials of an assembly, you must activate the required assembly in Creo Parametric, and then utilize the function "Save..." => "Update BOM".

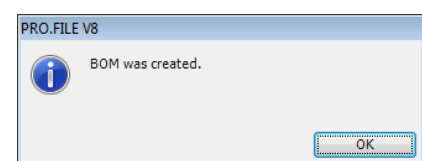
⇒ You receive list of active assemblies and sub-assemblies.



2. From this list you can select, for which assemblies and sub-assemblies you want to update the BOM. Confirm your selection with <OK>.



⇒ The successful creation of the bill of materials is confirmed by a message from PRO.FILE.

**Behavior when updating the bill of materials**

- If the Creo Parametric assembly contains components that are **not** contained in the PRO.FILE-bill of materials, then they will be **automatically** entered into the PRO.FILE-bill of materials.
- If the Creo Parametric assembly contains more components of a specific position than the PRO.FILE-bill of materials, the quantity indication will be corrected.
- If the PRO.FILE-bill of materials positions, that are **not** contained in the Creo Parametric assembly, then the message will appear, asking whether these positions should be deleted from the PRO.FILE-bill of materials.

- If the Creo Parametric assembly contains less components of a specific position, than the PRO.FILE-bill of materials, then the message will appear, asking whether the number in the PRO.FILE-bill of materials, should be corrected.

5.6 Save with Phantoms

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

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

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

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

Definition

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



Note: When to use this function?

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

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

For phantom parts the following applies:

- Only CAD documents that are not yet saved in PRO.FILE can be saved as phantom parts.
- PRO.FILE treats this assembly afterwards like a single part.
- It is possible to change phantom assemblies or parts.
- Phantom parts may not be used in other phantoms.
- Phantom assemblies result in a position within a bill of materials.
- The change of these models can only be made in the context.

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

**Note:**

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

**Function call from the PRO.FILE menu in Creo Parametric:**

"PRO.FILE" => "Save..." => "Save with Phantoms"

5.6.1

Save an assembly as a phantom

1. You are in the active assembly with CAD documents that are not yet saved in PRO.FILE.
2. Select the function "PRO.FILE" => "Save..." => "Save with Phantoms".
 - ⇒ The PRO.FILE Checkin wizard displays a list of all CAD documents to be created in PRO.FILE.
3. Confirm this list with <OK>.
4. Create the part and document description in the Checkin wizard for the phantom assembly. Confirm these steps with <Next>.
 - ⇒ The assembly is saved under one single part master. In the structure you can see that the assembly and **all parts** are summarized under one common part master. The phantom assembly **does not** have a bill of materials.

**Note: Phantom assemblies in the bill of materials**

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

5.6.2

Usage of phantom parts from a phantom assembly

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

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

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

the CAD system, you can open the phantom part directly from the phantom assembly.

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

5.6.3

Externally used phantoms – please cut database relation first

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



Function call from the PRO.FILE menu in Creo Parametric:

"PRO.FILE" => "Break up tree"

Proceed as follows

1. Select the "PRO.FILE" menu from the menu bar in Creo Parametric.
2. Select the function **"Break up tree"** from the area "DB-Relation".
 - ⇒ The dialog for the selection of documents to be disconnected is displayed. (Information on the function and status information can be found in the chapter ["Data overview: The document list"](#)).
 - ⇒ The structure of the phantom part and the assembly, the phantom part is used in, is displayed.
3. Select only the phantom part by using the checkbox in the first column.
4. Confirm your selection with <OK>.
 - ⇒ The connection of the part to the PRO.FILE database is dissolved and the phantom part is transformed into a separate part, which can be used as a new object.



Attention: Usage of phantom parts

Phantom parts cannot be referenced. You need to disconnect a phantom part from the phantom assembly in order to use it as a separate object.

5.6.4

Mixed design: Phantom assemblies and PRO.FILE objects

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

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



Note: Phantom parts/PRO.FILE parts in mixed assemblies

Only new objects (that are not known in PRO.FILE) within a mixed assembly are saved as phantom parts.

Example:

You have designed an assembly, into which you have inserted parts from PRO.FILE, e.g. four times the part "bol1_00019310".

You save this assembly via "**Save phantom**" and then create a bill of materials: Contrary to a normal phantom assembly, the bill of materials contains the four bolts.

You can use the created phantom assembly like a separate object in other designs.

5.7

Save automatic

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

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

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

- The classification via the Checkin wizard is only made for the first part and document description in PRO.FILE.
- For all further CAD documents to be saved **no** Checkin wizard is displayed. Document and part descriptions are saved automatically in PRO.FILE.
- Without further query means: The document and part descriptions are not filled in manually. The data record contains only the information that have been pre-configured in the saving form or that are automatically handed over from the CAD system to the saving form.

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

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

"Save all automatic" for complete assemblies

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

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



Function call from the PRO.FILE menu in Creo Parametric :

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

Proceed as follows:

1. Select the "PRO.FILE" menu from the menu bar in Creo Parametric.
 2. Select the function "Save automatic" from the menu area "Save".
- ⇒ For the first document that is unknown to PRO.FILE, the normal saving process is started. The following is displayed:

- The Checkin wizard to create the part description in PRO.FILE.
- Then (depending on the configuration) the Checkin wizard to create the document description in PRO.FILE.
- Finally, (depending on the configuration) the Checkin wizard for the project assignment in PRO.FILE.

3. Go through all steps of the Checkin wizard for the first new CAD document. Detailed information on this can be found in the previous chapter "[Saving CAD objects for the first time](#)".
- ⇒ All further CAD documents are now saved automatically to PRO.FILE, without having to fill in the document and part descriptions. A project assignment is only made if a project is already activated in PRO.FILE.
4. **For modified components already known in PRO.FILE:**
If the assembly also contains components that are already saved in PRO.FILE and that have been changed in the session or locally, these components are also offered for saving.
Select the components to be saved and confirm your selection.
Detailed information on the saving of changes to PRO.FILE can be found in the previous chapter "[Save: Saving changed CAD documents](#)".

**Note:**

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

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

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

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

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

**Attention: ERP interface and "Save automatic"**

When using the function "Save automatic", it may happen that fields, that are required by your ERP interface, are not filled! This may lead to problems during the forwarding of documents/parts to your ERP system. Please check the fields in the different forms for completeness.

It is possible to have specific fields filled automatically by the system. With this you can make sure that important fields are automatically provided the required information. For detailed information see the configuration manual for the Integration PRO.FILE – Creo Parametric.

5.8 Save all automatic

The function "Save all automatic" enhances the function "Save automatic" by an important step:

- When "Save automatic" is used, only the first document and part description have to be created manually – all others are created automatically.
- When "Save all automatic" is used, all document and part descriptions are created automatically – no user interaction is required.

All records that are thus saved in PRO.FILE get the information that is pre-configured on the saving form or automatically transferred from the CAD system to PRO.FILE.



Function call from the PRO.FILE menu in Creo Parametric :

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



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

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

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



Attention: ERP interface and "Save all automatic"

When using the function "Save all automatic", it may happen that fields, that are required by your ERP interface, are not filled! This may lead to problems during the forwarding of documents/parts to your ERP system. Please check the fields in the different forms for completeness.

It is possible to have specific fields filled automatically by the system. With this you can make sure that important fields are automatically provided the required information. For detailed information see the configuration manual for the Integration PRO.FILE – Creo Parametric .

5.9 Save select

If you want to save an object, then you use the function "Save".

- If you want to change an object, that has been loaded in the session, but is not active, then instead, you can use the function "Save select".



Function call from the PRO.FILE menu in Creo Parametric :

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

You then select the desired object, and finally use the function "Save".

How you select the desired object is described in chapter ["Info selection: Additional selection functions of the integration"](#).

5.10 Save: All instances

The function "All instances" allows you to save instances of a part family.

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



Function call from the PRO.FILE menu in Creo Parametric:

"PRO.FILE" => "Save..." => "All instances"

Proceed as follows:

1. If you want to save all instances of a part family for the first time in PRO.FILE, you have to save the mother part (generic) in PRO.FILE first.
 2. You can then use the function "Save..." => "Save all instances".
 3. When the function is used, a query is displayed, whether new part master records are to be created for the instances.
 - 3.1. If the question is answered with "No", all instances are assigned to the part master record of the generic.
 - 3.2. If the question is answered with "Yes", new part master records are created for all instances.
- ⇒ The further proceeding is identical to the first-time saving of CAD documents to PRO.FILE, as described in the chapter ["Saving CAD objects for the first time"](#).

**Attention:**

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

5.11

Save: All instances automatic

The function "Save instances automatically" combines the functions

- [Save: All instances](#) and
- [Save automatic](#)

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



Function call from the PRO.FILE menu in Creo Parametric:

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

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

**Note:**

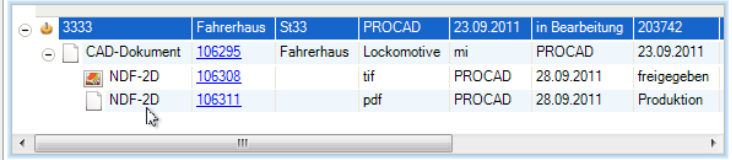
The difference between "All instances automatic" and "All instances" is that the metadata for saving in PRO.FILE is not queried for each object.

5.12 Neutral Data format: Save plotfile

The integration offers the possibility to convert a Creo Parametric drawing into a neutral data format (e.g. Tiff) file (NDF) and to save it in PRO.FILE.

If this function is used, an NDF file is generated from the CAD drawing.

This NDF file is then automatically attached to the document master of the drawing.



3333	Fahrerhaus	St33	PROCAD	23.09.2011	in Bearbeitung	203742
CAD-Dokument	106295	Fahrerhaus	Lockomotive	mi	PROCAD	23.09.2011
NDF-2D	106308		tif	PROCAD	28.09.2011	freigegeben
NDF-2D	106311		pdf	PROCAD	28.09.2011	Produktion



Note: Save TIFF only with "Format Generator"

The function "Save plotfile" is only available in connection with the PRO.FILE Format Generator. Furthermore, additional configurations and installations have to be made to your system. See the manual for the PRO.FILE Format Generators.



Function call from the PRO.FILE menu in Creo Parametric :

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

To generate a neutral format document proceed as follows:

1. You have opened a drawing and you want to document this development status.
2. Please select "Save..." => "Save plotfile" from the PRO.FILE menu.
 - ⇒ A query appears asking you whether you want to create a neutral document.
3. Confirm with <Yes>.
 - ⇒ The neutral data format is now created.
 - ⇒ Depending on the configuration, the action can be continued without further queries.
 - ⇒ The NDF document is then automatically saved in PRO.FILE and linked to the document description of the drawing.
 - ⇒ The creation of the neutral data format for the drawing is thus finished.



Note: Client-side NDF creation and server-side NDF creation

If you are using the NDF creation via the integration as well as the automatic creation of NDF files via status changes, please make sure that the NDF objects are configured according to your requirements (overwrite or version).

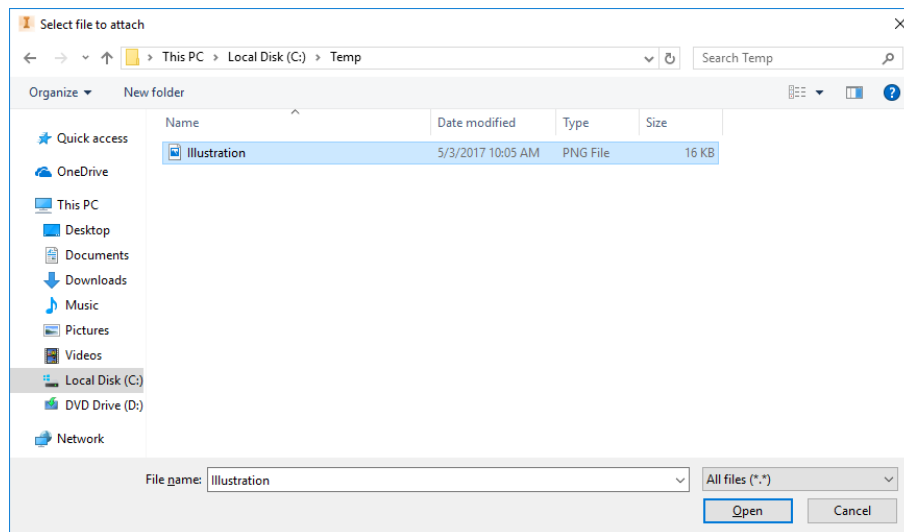
Change management via "Save plotfile"

Via configuration of the NDF generation via the Format Generators you can configure the behavior of PRO.FILE regarding the handling of NDF documents. You can configure that an existing NDF object is not overwritten but versioned when the function "Save NDF" is used. This can be used to document different development /design stages.

5.13 Append local file

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

1. First, load an Inventor object that has been saved in PRO.FILE into your CAD session.
2. Select the function "Save" => "Additional file" => "Append local file".
⇒ An Explorer windows opens.



3. Select the file to be added and confirm your selection with <Open>.
⇒ The part master record of your Inventor object is displayed in PRO.FILE.

C:\Temp\Illustration.png(1/1)

If you want to use the displayed part, simply press <Next>, the document will then be associated with this part.

Accept this Create new Select in list Search New using template Assign part

Show part [X]

Item # [] Created by PROCAD
 Item Description Oelpumpe Creation Date 03/05/2017

Classification [] Part type []
 Material [] Version -
 Material No. [] Weight (kg) 0.076 Revision -
 Unit Stk Usage EF
 Annotation []
 Status in Bearbeitung Ident # 204773
 Check date []

Cancel < Back Next > Skip Finish

4. Confirm the assignment with <Next>.

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

C:\Temp\Illustration.png(1/1)

Please enter the data for the document to be created. If you want to use another document as a template, please select the corresponding option.

Create new New using template Create document

Create document

Type of Document [Additional file]
 Project no. []
 Project descr. []
 created from PROCAD Version -
 created on 03/05/2017 Revision -

Classification according to DIN

Technical area Maschine
 Main- and Subclasses Dokument
 Type of document Zusatzdaten
 Document kind classification 00

Cancel < Back Next > Skip Finish

5. Enter the information for the new document record and confirm your input with <Finish>.

PRO.FILE V8

Document has been saved.

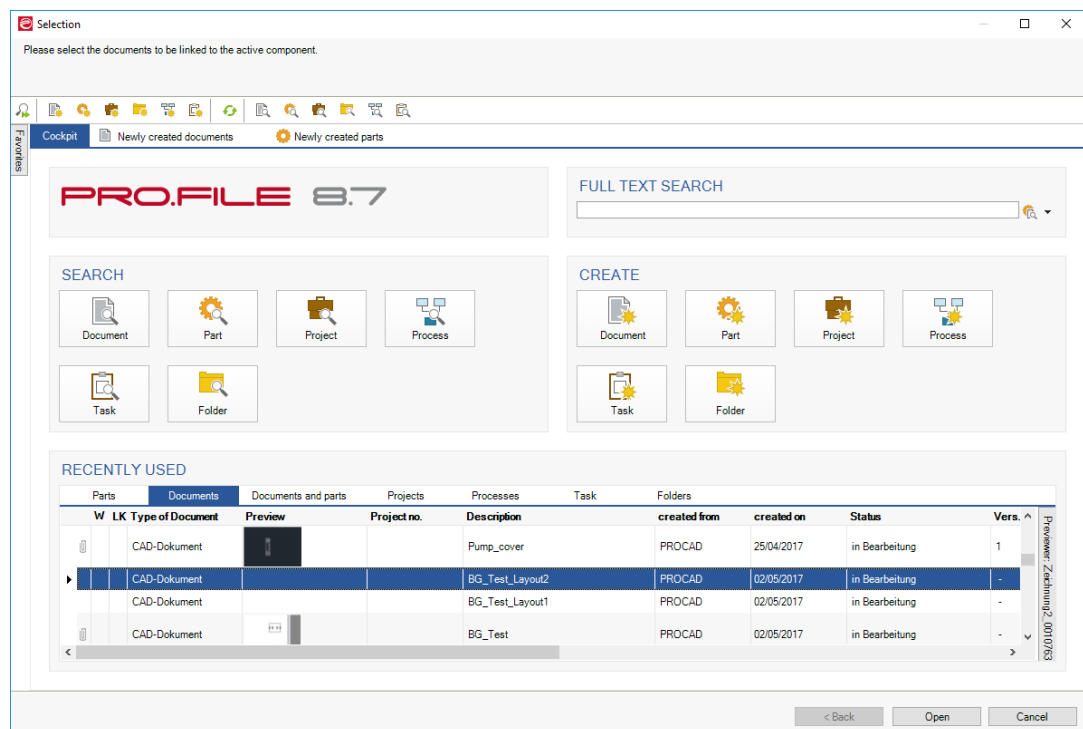
OK

- ⇒ The additional file is saved in PRO.FILE. It is linked below the document description of the CAD object. If possible, a preview file is created for the additional file.
- ⇒ By adding it to the Inventor structure, the additional file is automatically copied into the Workcenter folder.

5.14 Add Document

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

1. First, load an Inventor object that has been saved in PRO.FILE into your CAD session.
 2. Select the function "Save" => "Additional file" => "Add Document".
- ⇒ The PRO.FILE Checkout wizard opens.



3. Select the document record of the file to be added and confirm your selection with <Open>.
- ⇒ The document record with the additional file is linked below the document description of the CAD object.

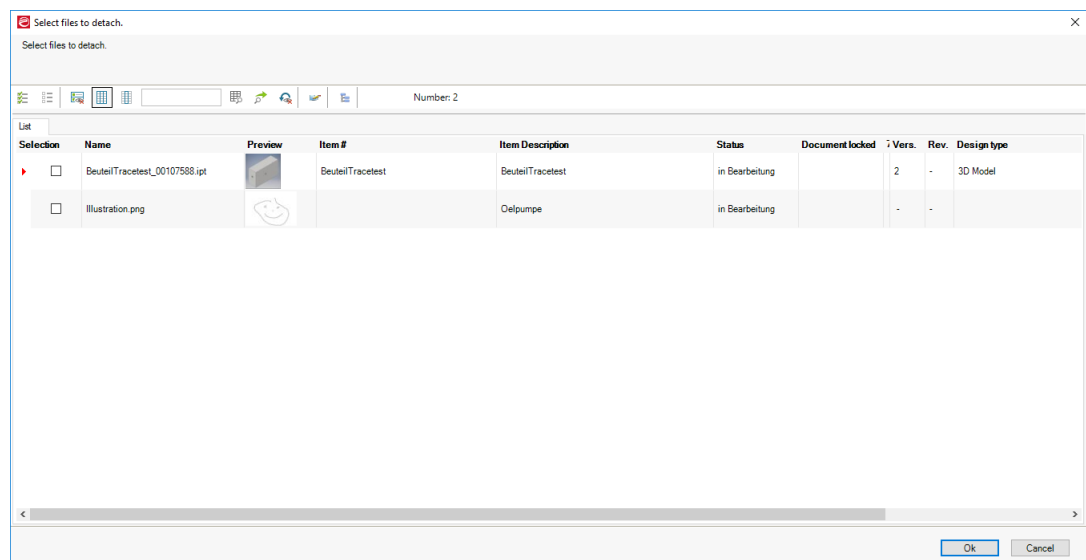
5.15

Detach

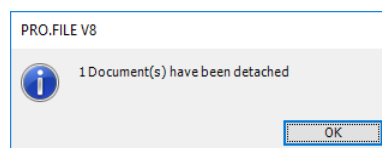
PRO.FILE prevents the deleting of documents as long as the documents are still in use – this also applies for additional files.

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

1. First, load a CAD object that has been saved in PRO.FILE (and that contains the additional file) into your CAD session.
 2. Select the function "Save" => "Additional file" => "Detach".
- ⇒ The dialog for the selection of additional files to be detached is displayed.



3. From the displayed list, select the additional file you want to detach by activating the corresponding checkbox in the column "Selection".
4. Confirm your selection with <OK>.



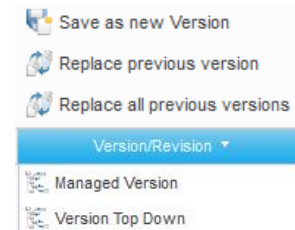
⇒ The selected document is removed from the CAD object structure.

6 Functions for the version administration

The function "**Version**" enables to update a version of Creo Parametric objects, that are already saved in PRO.FILE.

When you use the function "**Version**", you will get an extended PRO.FILE-menu containing the following additional functions:

- [Save as new version](#)
- [Version Top Down](#)
- [Managed Version](#)
- [Replace previous version: "direct" or "all"](#)



Note:

If you want to create a new Creo Parametric object in PRO.FILE, you must first use the function "**Save**".

A detailed description of the version concept of PRO.FILE can be found in the manual "**CAD design supported by PRO.FILE**".

6.1 Save as new version

With the PRO.FILE-Creo Parametric Integration it is possible to create different versions during saving of CAD objects.



Note:

A version can always be created from the newest version in the version list. Whether the creation of a version from older version is allowed as well depends on the configuration of the parameter "Allow creating a version from an old version" in the PRO.FILE Management Console.

If the function "**Save as new Version**", is called up a copy will be created of the PRO.FILE CAD object, and this new version will increase the version/ revision counter.

- Only the document active in the CAD session is versioned.
- The old version remains saved in PRO.FILE.
- The new version is saved with a new document ID in PRO.FILE and displayed in Creo Parametric.

- If a part is versioned in this way using PRO.FILE-Creo Parametric Integration, the new version of the part is always saved "before" the most current version. The references of assemblies in higher hierarchies will continue to indicate the older version – until the assembly is saved in PRO.FIL. The assembly structure is then also updated in PRO.FILE.
- If an assembly is versioned using the "Save as new Version" function, the tree structure of the assembly will be built using the currently loaded parts. A related drawing will also be versioned and a new structure created. Multi-layered assemblies must also be versioned layer by layer from bottom to top.

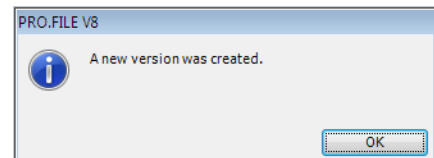


Function call from the PRO.FILE menu in Creo Parametric:

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

Proceed as follows:

1. Select the "PRO.FILE" menu from the menu bar in Creo Parametric.
2. Select the function "Version" => "Save as new version".
 - ⇒ A list with all documents, of which a new version will be created, is displayed.
3. Confirm with <OK>.
4. If you have not opened the newest version from the version chain but an older version instead, it depends on setting of the parameter "Ask for confirmation when creating a version from an old version" in the PRO.FILE Management Console whether a dialog is displayed.
 - ⇒ A new version of the active CAD documents is now created in PRO.FILE.
 - ⇒ A message box confirms the successful creation of the version.
 - ⇒ The new version is displayed in Creo Parametric.



Attention: New version is not locked

The new version created with the function "Save as new version" is not locked in PRO.FILE. To lock the document, please use the function "[Lock/Unlock: Who can change when?](#)".

The document list always displays the most recent version. To display older versions/revisions of a document, you can use the function "PRO.FILE" => "Show" => "Document versions" from the integration menu.

**Note: Versions of complex objects**

If the active Creo Parametric object is a complex object, only the complex object is versioned when the function **"Save as new version"** is used. This means that all changes made to the complex object are included in the new version. Changes made to components of the complex object are not included in the new version of the complex object.

If you want to include the changes to the components in the version as well, you first have to create new versions of the changed components ("Save as new version") and then create the new version of the complex object with **"Version Top Down"**.

**Note: Versions of drawings**

It has to be noted that the drawing of a part, that has been versioned with the function **"Save as a new version"**, is not automatically versioned in PRO.FILE.

Example: You have created a part and a drawing of the part in Creo Parametric. You load the part and save it as a new version. For the drawing of the part no new version is created!

**Note: manual "CAD design supported by PRO.FILE"**

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

6.2

Replace previous version: "direct" or "all"

The commands **"Replace previous version"** and **"Replace all previous versions"** allow an existing, built-in version of a CAD object to be replaced by a new version for all assemblies in which it is used.

In the design process an object may be locked due to feedback from the testing department. This object is no longer to be used. You as a designer now have to adjust all designs.

For this, you can use the function **"Replace version"**.

- Via the function **"Replace previous version"/"Replace all previous versions"** all assemblies are searched, in which the predecessor version of the current part is used (referenced). The reference is then changed to point to the new version of the part.
- PRO.FILE then creates a special document list, in which all documents are listed that are referencing to the old version of the part. You can now select, **which** assemblies are to be updated. The CAD info "used x times" indicates how often this part is used in **other** assemblies.
- In all **selected** assemblies the dependencies are replaced by a reference to the currently active object.

- Before a component is replaced in an assembly, PRO.FILE checks, whether the user has the permission to change **this** assembly.



Attention: Undo not possible!

By using the function "**Replace version**" the current structure of the concerned objects is changed. It is not possible to restore the objects in the previous state!



Function call from the PRO.FILE menu in Creo Parametric:

"PRO.FILE" => "Version" => "Replace previous version"
 => "Replace all previous versions"

When the function "**Replace previous version**"/"**Replace all previous versions**" is used, a check is made based on the PRO.FILE IF of the active CAD document regarding its usage: this check detects in which assemblies and drawings the active document is used.

You can then replace the old version of the **active** PRO.FILE document in all corresponding assemblies and drawings with the new version.

Proceed as follows:

1. Load the new version of the document, with which you want to make the replacement, from PRO.FILE in Creo Parametric (open the replacing document, not the document to be replaced).
 2. Select the function
 - "PRO.FILE" => "Version" => "Replace previous version" to replace only the direct previous version, wherever it is used.
 - "PRO.FILE" => "Version" => "Replace all previous versions" to replace all previous version, wherever they are used.
- ⇒ You now get a list of how often and where the predecessor version(s) of the document is/are used.
3. Select all records, for which a replacement is to be made.
 4. Confirm your selection with <OK>.
- ⇒ The version is now replaced: The currently loaded version is then used by all selected assemblies/drawings.
- ⇒ You thus have cleaned all concerned objects.
- ⇒ If you have not modified all object, you can repeat this action. You then receive a list of all objects using the old version of the component (minus the objects already modified).

**Attention:**

If a part is used in different assemblies, **each** assembly has to be updated with this function. If the part is used in many complex assemblies and in different versions, this may lead to a certain amount of work to be done.

6.3

Managed Version

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

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

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

- When a component is selected for "**Managed Version**", all instances in the assembly are selected. All versions of a component have the same file name by definition.
- All instances of a part family are treated equally and are thus versioned as well.
- The versionability of the components is checked at the beginning of the "**Managed Version**" process. If a component cannot be versioned (e.g. because it is already released), it cannot be selected.

**Function call from the PRO.FILE menu in Creo Parametric:**

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

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

- If no assembly is opened in Creo Parametric, an assembly can be selected via "**Managed Version**". In this case, the PRO.FILE Checkout Wizard is displayed at the beginning of the process.
- If an assembly is already opened in Creo Parametric, the assembly structure can be versioned and the new versions of assembly components can be used immediately.

For further details, see the following sub-chapter:

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

6.3.1

The proceeding for "Managed Version"

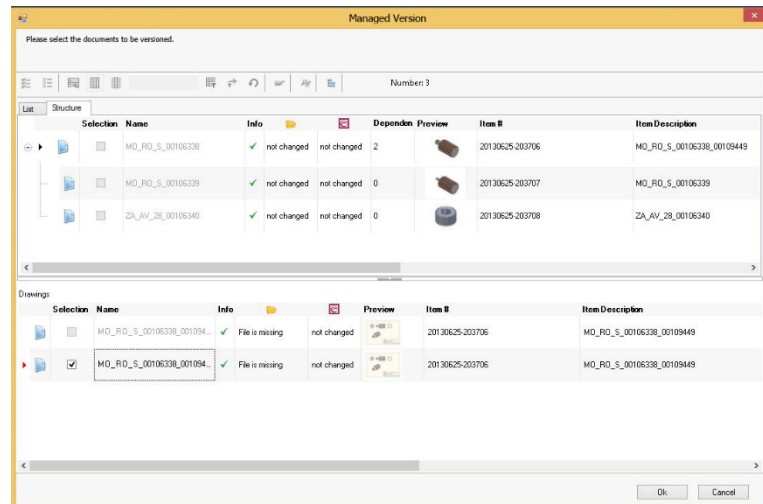
Proceed as follows:

1. Select the menu entry "PRO.FILE" from the menu bar in Creo Parametric.
2. Select the function "Version" => "Managed Version"

⇒ The Managed Version wizard is started.

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

⇒ In the second step, the sub-structure is enhanced by the related drawings.



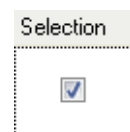
⇒ In the wizard of "Managed Version" the tree structure, determined and expanded by drawings, is shown, so that the documents to version with their dependent data (nodes) can be selected.

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

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

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

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

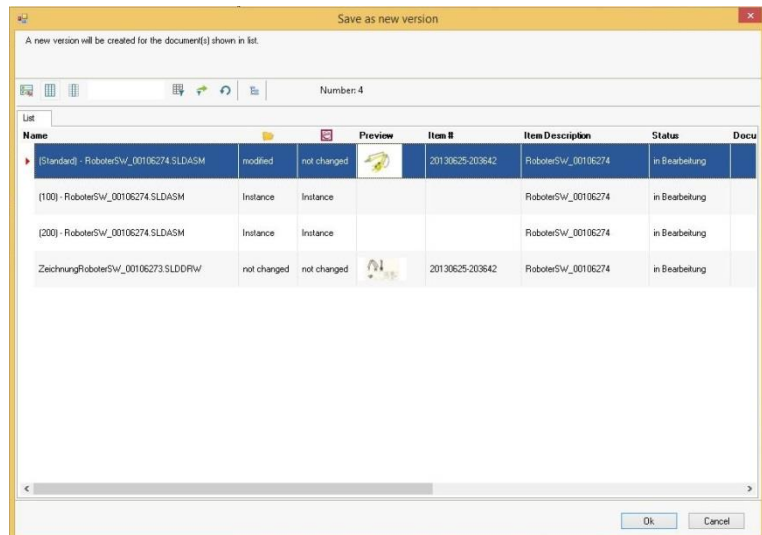


4. Confirm your selection with <OK>.

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

5. Confirm with <OK>.

⇒ The selection components are now versioned.

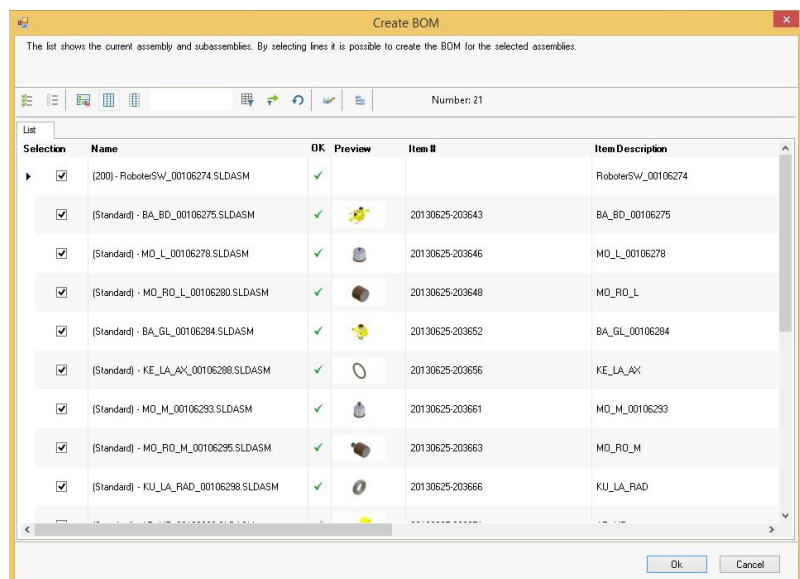


⇒ The successful completion of the process is confirmed by a message.

6. Confirm with <OK>.



⇒ The subsequent list shows the saved assemblies



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

8. Confirm your selection with <OK>.

⇒ The process "Managed Version" is thus finished.

6.4

Version Top Down

**Note:**

If your complex object contains a component of which a more recent version exists in PRO.FILE, you will only get the original version of the document by selecting "as stored" upon loading.

Note the following situation:

- New versions are to be saved of one or more components of complex objects.
- Then a new version of the complex object is to be created, that refers to the new versions of the components.
- To do this, use the function "Version" => "Version Top Down".

The complex object, on which you want to use the function "Version Top Down", does not have to be loaded or active in Creo Parametric when the function is applied.

**Function call from the PRO.FILE menu in Creo Parametric:**

"PRO.FILE" => "Version" => "Version Top Down"

Proceed as follows:

1. You will then be requested to select in PRO.FILE, the Creo Parametric complex object that the function "Version Top Down" is to be applied to.
 - ⇒ After you have selected the desired complex object, you will receive a selection screen.
2. You can now select the criterion that you want to be used for the Top-down-versioning.
 - ⇒ The possible criteria ("latest visible version", "latest released version", "latest released / visible version") are described in chapter ["Open: Opening CAD Documents from PRO.FILE"](#).
3. Confirm your selection with the <OK> button.
 - ⇒ The function "Version Top Down" will then search all components for versions, that meet the criterion requested. If the function finds components with the appropriate versions in the structure, then every superior assembly will be versioned by these criterion.
 - ⇒ You can now take the complex object, or each sub-assembly of the complex objects from PRO.FILE with "as stored", and you will receive the assemblies with references to the newest versions with the desired criterion.

7 DB-Relation: The relation of CAD objects to the PRO.FILE database

The function "DB-Relation" enables you to control the relation of Creo Parametric objects to PRO.FILE.

When you use the function "DB-Relation", you will first be presented with an extended PRO.FILE-menu containing the following additional functions:

- [Update Parameters](#)

To update the drawing title block.

- [Break up tree](#)

Break the relation between a Creo Parametric object tree and the PRO.FILE database.

- [Info Feature](#)

Delivers information of linked parameters and measurements.

- [Info selected feature](#)

Delivers information of linked parameters and measurements using a selection function.

- [Modify part feature relations](#)

Establish a link to the item characteristics which enables you to modify a part master record after it was saved.

- [Break up feature bar relations](#)

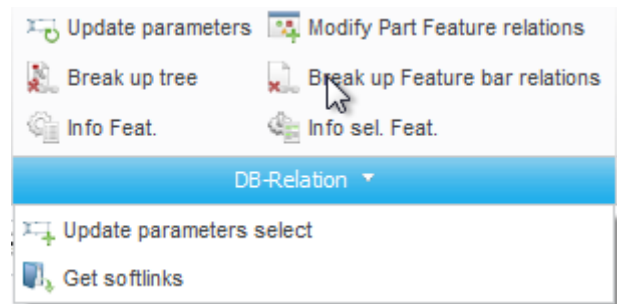
To dissolve the links of parameters and measurements to PRO.FILE-fields.

- [Update Parameters select](#)

To update the drawing title block using a selection function.

- [Get Softlinks](#)

It allows, subsequently still to load, Softlinks that during the opening of the object not were opened.



7.1 Update Parameters

With the function "Update Parameter" you can update the drawing title block of an active object.

If you want to update the fields of your drawing title block, you must first activate the desired drawings and then use the function "DB-Relation" => "Update Parameter".

7.2 Break up tree

The function "**break up tree**" enables you to dissolve the relation of the active Creo Parametric object to the PRO.FILE database.

1. If you want to dissolve the relation between a Creo Parametric object, and a PRO.FILE-database, then you must activate the object in Creo Parametric, and then use the function "**DB-Relation**" => "**Break up tree**".
⇒ If you have a complex object active in Creo Parametric, you will be presented with the special list of documents, in which the activated object and its components are listed.
2. You will then be requested to mark the objects whose database relation to PRO.FILE is to be dissolved.
3. To mark an object that you want to release, you move the cursor to the object of your choice, and then use a click of the mouse. You can mark several objects, by holding down the <Ctrl> button, whilst making your selections. You can mark a whole section, by marking the start point of the section, and then while holding down the <Shift> button, you click on the end of the section.
4. After you have marked the desired objects, you must confirm your choice with the button "**Accept**".
5. A new name must be given to all marked objects, with which they shall now be known in Creo Parametric:
6. PRO.FILE now suggests to you new Creo Parametric names. Enter the names of your choice, or accept the suggestions (they are the original names, with a number attached, to ensure they are unique!) and confirm with <OK>.
⇒ You have now dissolved the relation to the database, and return to Creo Parametric.



Note:

You may now change the dissolved objects, and resave them in PRO.FILE. The "Originals" that were stored in PRO.FILE, remain unaffected by these changes.

7.3 Info Feature

The function "**Info Feature**" shows you information on the linking of parameters and measurements with PRO.FILE-fields.

If you want to display this information, you must activate the desired object in Creo Parametric, and use the function "**DB-Relation**" => "**Info Feature**".

You receive an overview of all SML-links of the active object.

7.4 Info selected feature

If you want to display information about the linking of parameters and measurements with PRO.FILE-fields, then you can use the function **"Info Feature"**. If you want to get information about an object that is loaded in this session, but not activated, then you must use **"Info selected feature"** instead. First select the desired object, to which the function **"Info Feature"** is then applied.

7.5 Modify part feature relations

Using this function you can modify the data of a part via the selection of item characteristics.

7.6 Break up feature bar relations

Using the function **"Break up feature bar relations"**, you can lift the link of parameters and measurements to PRO.FILE-fields.

If you want to lift the link between a Creo Parametric object, and a PRO.FILE-field, you must first activate the desired object in Creo Parametric, and then use the function **"DB-Relation" => "Break up feature bar relations"**.

You will then get a selection menu containing all SML-links. You can now mark the links to be lifted. After confirming your selections, the chosen links will be lifted.

7.7 Update Parameters select

Using the function **"Update parameters"** you can update the drawing title block of an active object. If you only want to update the drawing title blocks of components of active objects, then you can use the function **"Update Parameter select"**.

Before the update takes place, you get a document list in PRO.FILE, from which you can select the documents that are to be updated.

Information on how to select the desired object can be found in the chapter ["Info selection: Additional selection functions of the integration"](#).

7.8 Get Softlinks

If softlinks have not been loaded during the first loading of the CAD object, they can be loaded subsequently with this function.

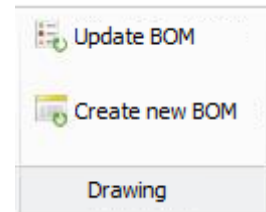
The procedure for this corresponds to the description in chapter ["Open: Opening CAD Documents from PRO.FILE"](#).

8 Drawing: Insert and update bill of materials

The function "Drawing" enables the display of information from PRO.FILE, on a Creo Parametric drawing.

When you use the function "Drawing", you will get the extended PRO.FILE-menu containing the following additional functions:

- [Create new BOM](#)
Inserts the bill of materials created in PRO.FILE into the drawing.
- [Update BOM](#)
Updates an existing bill of materials with data from PRO.FILE.



Note:

The functions "Drawing" => "Create new BOM" and "Drawing" => "Update BOM" can only be used if a drawing is activated in the Creo Parametric session.

8.1 Create new BOM

If you want to display a bill of materials created in PRO.FILE on a Creo Parametric drawing, you must use first activate the desired drawing in Creo Parametric

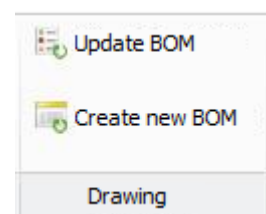


Function call from the PRO.FILE menu in Creo Parametric:

"PRO.FILE" => "Drawing" => "Create new BOM"

Then proceed as follows:

1. Select the function "Drawing" => "Create new BOM" from the integration menu.
⇒ A new table with BOM information from PRO.FILE is inserted into the drawing.
2. The table with the BOM information now has to be placed on the drawing. Follow the instruction to select a point of placement within the drawing in Creo Parametric.
3. The bill of materials is filled in automatically. The format of the bill of materials can be configured in the file "stkli.dat". (This file is only evaluated for the function "New table").



- ⇒ By the creation of a new table, the BOM saved in PRO.FILE is transferred to your drawing.
- 4. If the BOM in your 3D model has changed, you will have to update it in PRO.FILE before, by using "Save..." => "Update BOM".

**Note:**

A bill of materials consists of part information from PRO.FILE. If you have not saved your drawing in PRO.FILE so far, you will have to create or select the desired part master in PRO.FILE when using this function.

8.2

Update BOM

If you want to display a bill of materials created in PRO.FILE on a Creo Parametric drawing, you must use first activate the desired drawing in Creo Parametric

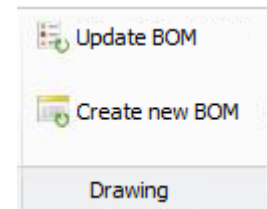


Function call from the PRO.FILE menu in Creo Parametric:

"PRO.FILE" => "Drawing" => "Update BOM"

Then proceed as follows:

1. Select the function "Drawing" => "Update BOM" from the integration menu.
- ⇒ The drawing already contains a table with BOM information from PRO.FILE, which is now updated.



2. If you have decided to update an existing table, the BOM to be updated has to be selected within the drawing.
3. By the update of the table, the BOM saved in PRO.FILE is updated on your drawing. If the BOM in your 3D model has changed, you will have to update it in PRO.FILE before, by using "Save..." => "Update BOM".

**Note:**

A bill of materials consists of part information from PRO.FILE. If you have not saved your drawing in PRO.FILE so far, you will have to create or select the desired part master in PRO.FILE when using this function.

9 Show: Displaying PRO.FILE information on the active object

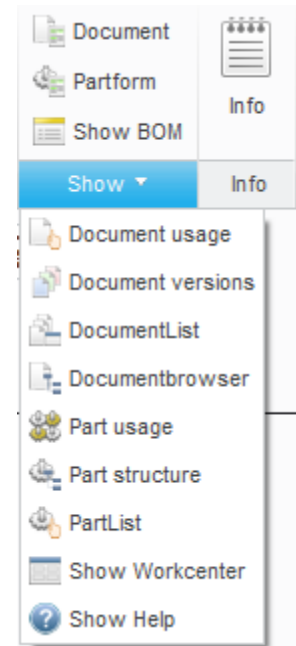
The "Show" and "Info" menu give you direct access to the elements offered in PRO.FILE. This allows for the targeted retrieval of the desired information in PRO.FILE. The corresponding information will always be provided for the currently active object in the Creo Parametric session.

The function "Info" displays the document list.

If you choose the "Show" function, an extended PRO.FILE menu with the following additional functions will be displayed:

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

For more information see the following sub-chapters.



9.1 Data overview: The document list

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



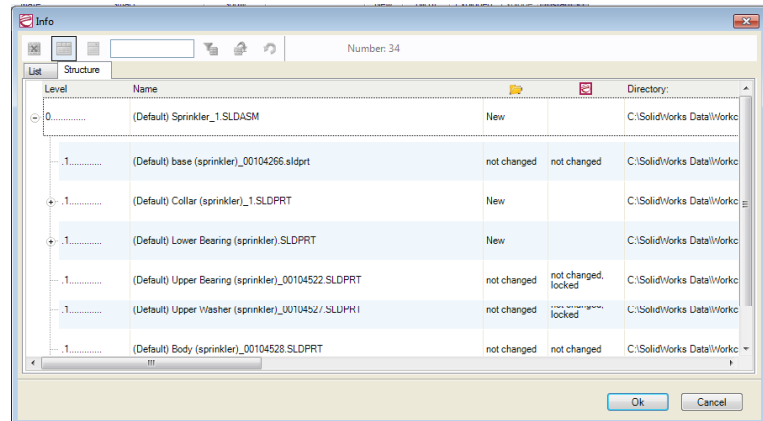
Function call from the PRO.FILE menu in Creo Parametric:

"PRO.FILE" => "Info" => "Info"

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

You find the following information:

- The data from the PRO.FILE document description.



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

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

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

Detailed information can be found in the following chapters:

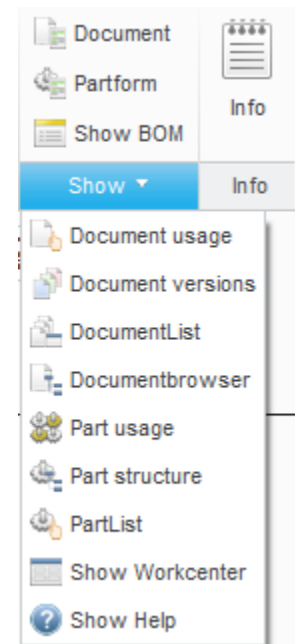
- [Direct information in the dialog screens](#)
- [Up to date or not: Display of status information](#)

9.2

Show: Information on a CAD document in PRO.FILE

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

- These menu entries access information on the CAD document currently active in Creo Parametric.
- The various menu entries allow a targeted access to frequently needed information, without having to navigate in PRO.FILE.
- After a function is selected, PRO.FILE opens and displayed the desired information.
- Within the displayed lists and forms, you can perform all actions available in PRO.FILE.



Function call from the PRO.FILE menu in Creo Parametric:

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



Note:

Please note that you can only use these display functions if the CAD document is already saved in PRO.FILE.

If you have made changes to your CAD document and have not yet saved these changes back to PRO.FILE, these changes are ignored by the display functions.

The following display options are available:

9.2.1

Show Workcenter

Open the Workcenter with the current working directory. Detailed information in the chapter "[Workcenter](#)".

9.2.2

Document list

The document list shows an overview of PRO.FILE information on the currently active CAD data. Detailed information on this can be found in the previous chapter "[Data overview: The document list](#)".

9.2.3 Document form

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

9.2.4 Document browser

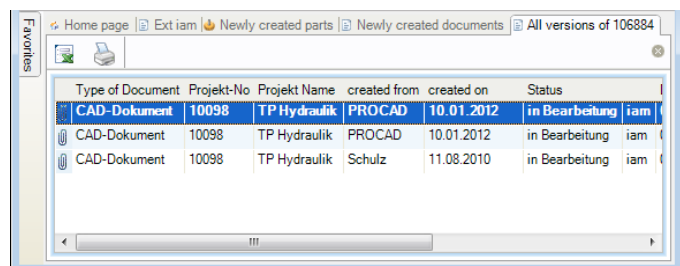
With the function "**Document browser**" you can see which documents (= part drawings) are used in your drawing (= main drawing).

9.2.5 Document usage

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

9.2.6 Document versions

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



Type of Document	Projekt-No	Projekt Name	created from	created on	Status
CAD-Dokument	10098	TP Hydraulik	PROCAD	10.01.2012	in Bearbeitung iam
CAD-Dokument	10098	TP Hydraulik	PROCAD	10.01.2012	in Bearbeitung iam
CAD-Dokument	10098	TP Hydraulik	Schulz	11.08.2010	in Bearbeitung iam

9.2.7 Part form

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

9.2.8 Part browser

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

9.2.9 Part usage

With the function "**Part usage**" you can see whether your current CAD document is used by other assemblies. The usage list displays the "upward" structure.

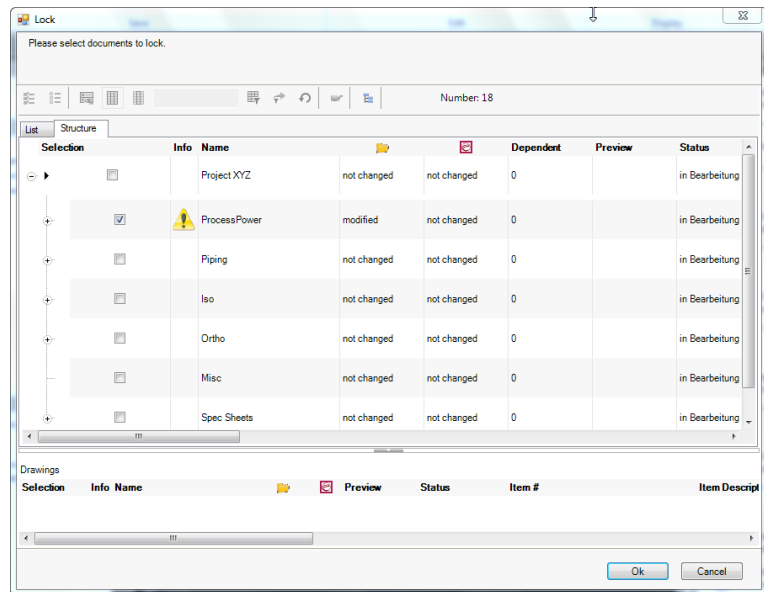
9.2.10 Show BOM

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

9.3 Direct information in the dialog screens

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

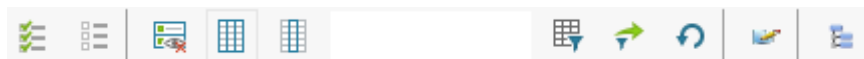
These offer the following possibilities:





- You can switch between list and structure view.
- Via the buttons of the list functions you can make searches and filter the results (see the following sub-chapter "[More comfort: search and list functions in the dialog screens](#)").
- If objects are listed in the assembly tree above the actual node (e.g. drawings), these objects are displayed in a dependent window.
- The lists contain status information for each of the listed objects (see the following sub-chapter "[Up to date or not: Display of status information](#)").

9.3.1 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.












9.3.2 Up to date or not: Display of status information

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

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

These columns may contain the following:

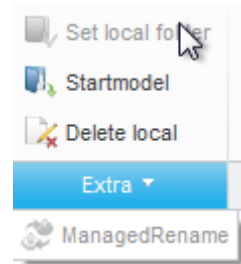
Info	Local 	PRO.FILE 	Description
	new	unknown	The file is new and unknown in PRO.FILE.
	unchanged	unknown	The file is locally unchanged but comes from a different instance of PRO.FILE and can therefore not be saved back to the current instance.
	changed	unknown	The file is locally changed but comes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

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

10 Extra: Additional functions

If you activate the function "Extra", you receive at first an expanded PRO.FILE-menu with following additional functions:

- [Delete local](#)
enables you to delete files in your local work directory.
- Set local folder
Open the Workcenter and manage the working directories (see chapter: [Workcenter](#)).
- [Startmodel](#)
enables you to work with start models.
- [Managed Rename: Renaming in the structure](#)



10.1 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 Creo Parametric:

"PRO.FILE" => "Extra" => "Set local folder"

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

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



Attention when working with several work folders:

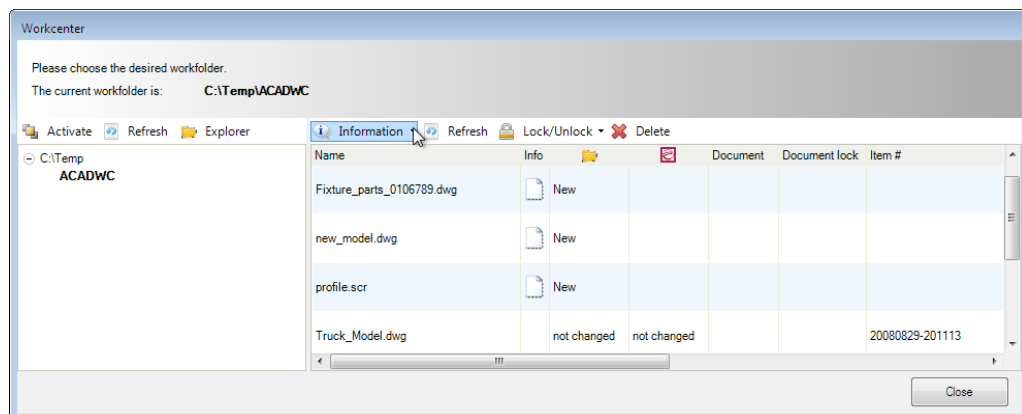
Please be careful when working with several work folders. It may happen that the loaded CAD document has loaded components from work folder A as well as from work folder B. When deleting files from one work folder, make sure that these components are not referenced by documents in a different work folder.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

10.1.1

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](#)".

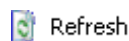


The functions for the directory structure:



Activate

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



Refresh

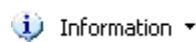
The view of the directory structure is updated.



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:

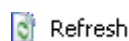


Information ▼

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

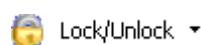
Structure of the parts
Part form
Usage of parts
Bill of materials

Document structure
Document form
Document usage



Refresh

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



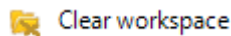
Lock/Unlock ▼

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



Delete

The marked documents are deleted from the directory.

**Clear workspace**

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

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

**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.

Open with double click in the CAD system

Double-clicking a file in the Workcenter opens the file in the CAD system (if it is not already opened).

10.1.2

Creating and changing an active work folder

When working with the Creo Parametric integration, you can use several work folders in parallel (provided that this is configured).

For the manual creation, activation or modification of a work folder, you can use the function "Set local folder".



Function call from the PRO.FILE menu in Creo Parametric:

"PRO.FILE" => "Extra" => "Set local folder"

The basic options in the Workcenter are:

- To create a new folder, use the "Explorer" button. In the displayed Windows Explorer you can create folder and subfolder, as required.
- The currently active work folder is displayed in bold. In this folder, all CAD data from PRO.FILE and Creo Parametric is currently saved.
- For the activation of a different folder, select the desired folder and click "Activate".



Note:

The topmost level of the work folder is defined via the configuration of the integration in the PRO.FILE Management Console. The creation of new folders can only be made within this superior folder.

10.2 Delete local



Important: Risk of data loss!

The function "Delete local" deletes all files in the current work folder. Data that has not yet been saved to PRO.FILE is thus irretrievably lost!

CAD documents that have been saved to PRO.FILE with the most recent editing state can be deleted from the local work folder. To quickly and thoroughly clean up your work folder, you can use the function "Delete local".



Function call from the PRO.FILE menu in Creo Parametric:

"PRO.FILE" => "Extra" => "Delete local"

The function "Delete local" performs the following steps:

- Files in your active work folder of the Workcenter are deleted.
- Files in other work folders remain untouched.
- CAD document currently loaded in the session are not deleted, either – even if they are saved in the current work folder.

Further information on the selection of the current work folder can be found in the chapter "[Workcenter](#)".

10.3 Startmodel

The integration PRO.FILE - Creo Parametric enables you to work with start models.

- A Startmodel is a model template stored in PRO.FILE.



Function call from the PRO.FILE menu in Creo Parametric:

"PRO.FILE" => "Extra" => "Startmodel"

You can create your new start models, by creating them in Creo Parametric and then storing them in PRO.FILE.

If you activate the function "**Startmodel**" now, you receive a selection of the defined templates.

The function "**select**" is always available. PRO.FILE opens and calls you for the selection of a document description.

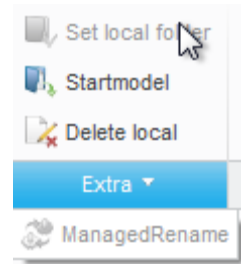
When you select the startmodels configured by you (for example. "**template (Part)**"), so the defined document description is selected automatic.

After the choice of a template the start model is withdrawn from PRO.FILE, the relation to PRO.FILE is solved and stored in PRO.FILE then again. ("Save ").

Then you are able to work with the start model in Creo Parametric.

You can employ also complex objects as start models, for example a drawing with attached part. By retrieving all objects are detached and saved back.

You can configure the modalities of the storage separately for start models (see configuration manual of the PRO.FILE – Creo Parametric integration).



10.4

Managed Rename: Renaming in the structure

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

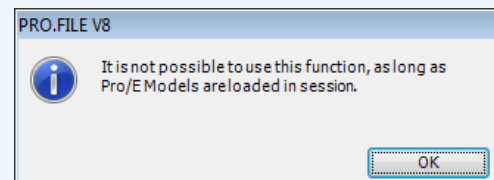
This function is used e.g. for the following cases:

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



Note: Only available without active CAD document

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



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

Please note the following for this process:

- The renaming can only be within one assembly structure. The components to be renamed must not be used in other structures.
- The part to be renamed must not have any versions, since, by definition, all version must have the same file name.
- Instances must not be renamed.
- Both the models to be renamed and the assemblies/drawings containing these models must be savable in PRO.FILE (access rights).
- CAD documents referenced in an assembly that are locked by a different user cannot be renamed.



Attention: "Rename" is a modification

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

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

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



Function call from the PRO.FILE menu in Creo Parametric:

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

Proceed as follows

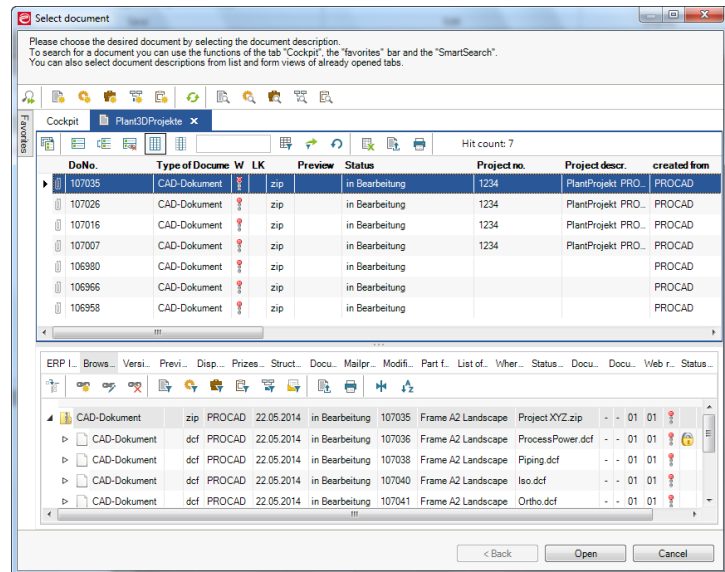
1. Select the "PRO.FILE" menu in Creo Parametric.
 2. Select the function "Extras" => "Managed Rename".
- ⇒ The Checkout wizard to select the CAD document to be renamed is displayed.

Select the desired document in the Checkout wizard

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

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

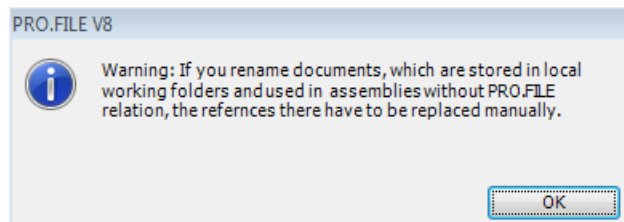
- Via the tab "Cockpit"
- Via the search functions of the icon bar.
- Via favorites, SmartSearch or task assignments.



4. If the desired document is displayed in a list view, **select** it. (If the document is displayed in a form view, it is already selected).
 5. Click **<Open>**.
- ⇒ The Checkout wizard closes and a warning message is displayed.

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

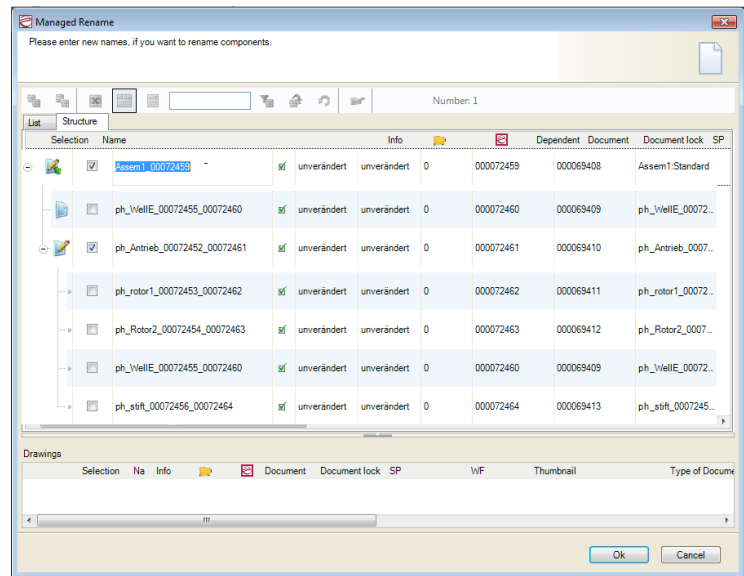
- ⇒ A warning message informs you that all references recognized by PRO.FILE will automatically be exchanged after the renaming.



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

Start the renaming in the structure

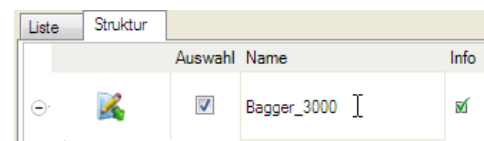
⇒ The window "Managed Rename" is displayed. In this window you can rename the files:



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



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



9. Make the changes for all desired components.
10. Once you have renamed all desired components confirm your changes with <OK>.
- ⇒ The integration now saves the changed file names back to PRO.FILE and updates the references.
- ⇒ The renaming in the structure is now finished.

11 Info selection: Additional selection functions of the integration

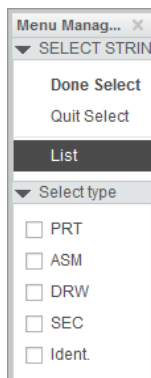
The following functions are also in the PRO.FILE-menu of Creo Parametric as selection functions:

- "Save" => "Save select"
- "Lock/Unlock" => "Lock" ("Lock selection")
- "Lock/Unlock" => "Unlock" ("Unlock selection")
- "DB-Relation" => "Info feature" ("Info selected feature")

All functions carried out here, allow the selection of specific Creo Parametric objects. The selection process is the same for all functions. After the selection has been made, the corresponding function will be carried out on each object. i.e. after the selection of the object "Lock selection", the function "Lock" will be called up.

11.1 Object type and Object selection

When you call up a selection function, the PRO.FILE-menu expands in the following way:



In the section "**Select type**", you can select the object type that the appointed object has. The object types are described by their file endings. The menu point "**Ident.**" is described in the next section. Finally the section "**Select type**" will be replaced by the section "**in use**". In this section, all objects appear, that are of the selected type and are loaded in the session. You can now select the desired object from these objects. And now the corresponding function will be carried out on this object.

11.2 Select object with mouse click

You can also select objects, by using a mouse click to select a component of an active model. You can also call up the selection function in the section "**Select Type**" of the menu point "**Ident.**". Finally you can select your component with the mouse. The rest of the selection process is the same as that of the other standard functions in Creo Parametric.

12 Tips and tricks on specific aspects

This chapter describes the specific characteristics of the PRO.FILE – Creo Parametric integration and gives you tips and tricks that will make it easier for you to work with the system.

These hints aim at the following fields of usage of the integration:

- [Placing with help levels and help coordinates](#)
- [Handling of drawing frames in PRO.FILE](#)

12.1 Placing with help levels and help coordinates

When you lay down a Creo Parametric object in PRO.FILE you should provide this object with sufficient help coordinates, and help levels, so that later you, or other users have no problems in using up those objects.

Refer always to added help levels, so that it is easier to delete Creo Parametric "parts" from an assembly.

You will have problems, if you have not the changing authorities for objects, when using up objects, because the objects are e.g. in a release state. In this case you cannot add in any help coordinates and/or help levels, to make the using up easier.

12.2 Handling of drawing frames in PRO.FILE

Creo Parametric uses drawing frames (Creo Parametric objects of the type "FRM") particularly:

If a drawing frame is loaded in a drawing, then the contained tables, will be copied into the drawing, and the remaining elements of the drawing frame, will be recorded in the drawing as a reference.

If the drawing frame is renamed during a session, then the drawing will still refer to the drawing frame using its original name. In contrast to other dependences, here the renaming is not effective for all objects loaded in the session.

If Creo Parametric is run without PRO.FILE, then normally a "format directory" is created for the drawing frames. The drawing frames that are laid down there, can be called up by direct use of their names. If a drawing frame is loaded into a drawing from this "format directory", then from now on a reference of the drawing to the drawing frame that is on the "format directory" exists.

Problems for PRO.FILE

Due to the renaming of Creo Parametric objects when working in PRO.FILE, there can be several consequences that can affect the behavior of Creo Parametric when handling drawing frames.

12.2.1 Drawing frames as "phantom Objects"

If a drawing is saved in PRO.FILE, that contains a drawing frame that has not been loaded from PRO.FILE, then this drawing frame will be renamed, and saved to this drawing as a "Phantom object".

When loading the drawing, the drawing refers to a drawing frame with its original name (the drawing frame that is saved in the "format directory") because of the special handling of drawing frames in Creo Parametric. The "Phantom object", the saved drawing frame in PRO.FILE, is loaded in the session but not used in the drawing.

If you don't want to make references on the local drawing frame any more, e.g. because the local drawing frame doesn't exist, or should be changed, then you can fix the reference on the drawing frame that is saved in PRO.FILE as "Phantom object" with the interactive function "replace frame". After you have saved your drawings back into PRO.FILE, the drawing refers now always to the "Phantom object". Because it is a "Phantom object", the changes on the drawing frame are not taken on in the "format directory".

12.2.2 Administration of drawing frames referred with PRO.FILE

You still can administer referring drawing frames in PRO.FILE. That means, that a drawing frame is just the same as any other component of a drawing. You always have to save the drawing frame first in PRO.FILE and then when creating a drawing integrate the drawing frame from PRO.FILE into your drawing.

12.2.3 Do not save drawing frames in PRO.FILE

If you only want to refer drawing frames in the "format directory" of Creo Parametric, you are able to prevent drawing frames from being saved into PRO.FILE

The variable DBZEFORMAT is fully described in the configuration manual of Integration PRO.FILE – Creo Parametric. Using this variable you can suppress the saving of drawing frames in PRO.FILE.

If the variable is set to **IGNORE**, the drawing frames will not be saved in PRO.FILE. That means that with "Save" no "phantom object" will be created for the drawing frame.

It means that when drawing frames from the "format directory" of Creo Parametric are being read from PRO.FILE, the corresponding drawing frame will be read from the "format directory".

If you change drawing frames from the "format directory", these changes will appear in all drawings from PRO.FILE that use these drawing frames.



Note:

If you set the variable DBZEFORMAT to IGNORE, Then you must guarantee that all used drawing frames, are available in the "format directory" of Creo Parametric.

13

Index

A

- add PRO.FILE document..... 81
- additional file
 - add..... 79
- additional functions..... 104
- assign created object to PRO.FILE project..... 46

B

- bill of materials..... 94
- Break up feature bar relations 93
- Break up tree..... 92

C

- Checkout wizard
 - search for CAD documents 21
- contents..... 8
- create independent copy of a model 54
- Create new BOM..... 94

D

- DB-Relation..... 91
- Delete local..... 107
- detach document..... 82
- dialog screens 100
- Document browser..... 99
- document description
 - create..... 45
- Document form 99
- document list..... 96
 - search and list functions 100
 - status information 102
- Document list 98
- Document usage 99
- Document versions..... 99
- drawing frames..... 114, 115
 - do not save in PRO.FILE 115
- Drawing frames as "phantom Objects"..... 115

E

- exchange model in higher-level assembly 56
- exchanged 53

F

- first steps..... 10

G

- Get Softlinks 93

H

- help levels and help coordinates 114

I

- Info Feature 92
- Info selected feature..... 93
- integration functions 12
 - overview 14
- integration PRO.FILE Creo Parametric..... 8

L

- local work folder 10
- lock 34, 35
- lock selection 38

M

- Managed Copy 63
- Managed Copy..... 53
 - automatic..... 66
 - search and replace..... 65
- Managed Rename 108
- Managed Version 87
- Modify part feature relations 93

N

- neutral data format..... 78

O

- object dependencies 9
- Object selection 112
- Object type..... 112
- open 15, 16, 30
 - all drawings 26
 - browse versions..... 27
 - drawing for active object 24
 - locally existing files 32

P

- Part browser..... 99
- Part form 99
- part master record
 - create or assign..... 41

Part usage.....	99
PRO.FILE information	96
PRO.FILE Login.....	13
proceeding for "Managed Version"	88

R

Renaming in the structure.....	108
Replace previous version.....	85

S

save	39
all instances	76
all instances automatic	77
as new version.....	83
changed CAD documents.....	48
first time	40
with phantoms.....	68
Save all automatic	75
Save automatic.....	71
Save NDF	78
Save selection.....	76
Select object with mouse click	113
selection functions.....	112
Show BOM	99
show information	98

Show Workcenter	98
Startmodel	107

T

table of contents.....	3
tips and tricks	114

U

unlock.....	34, 37
unlock selection.....	38
Update BOM	66, 95
Update parameters	91
Update Parameters select	93

V

version administration	83
Version Top Down	90

W

work folder	
create	106
Workcenter	10, 104
Workcenter functions	105