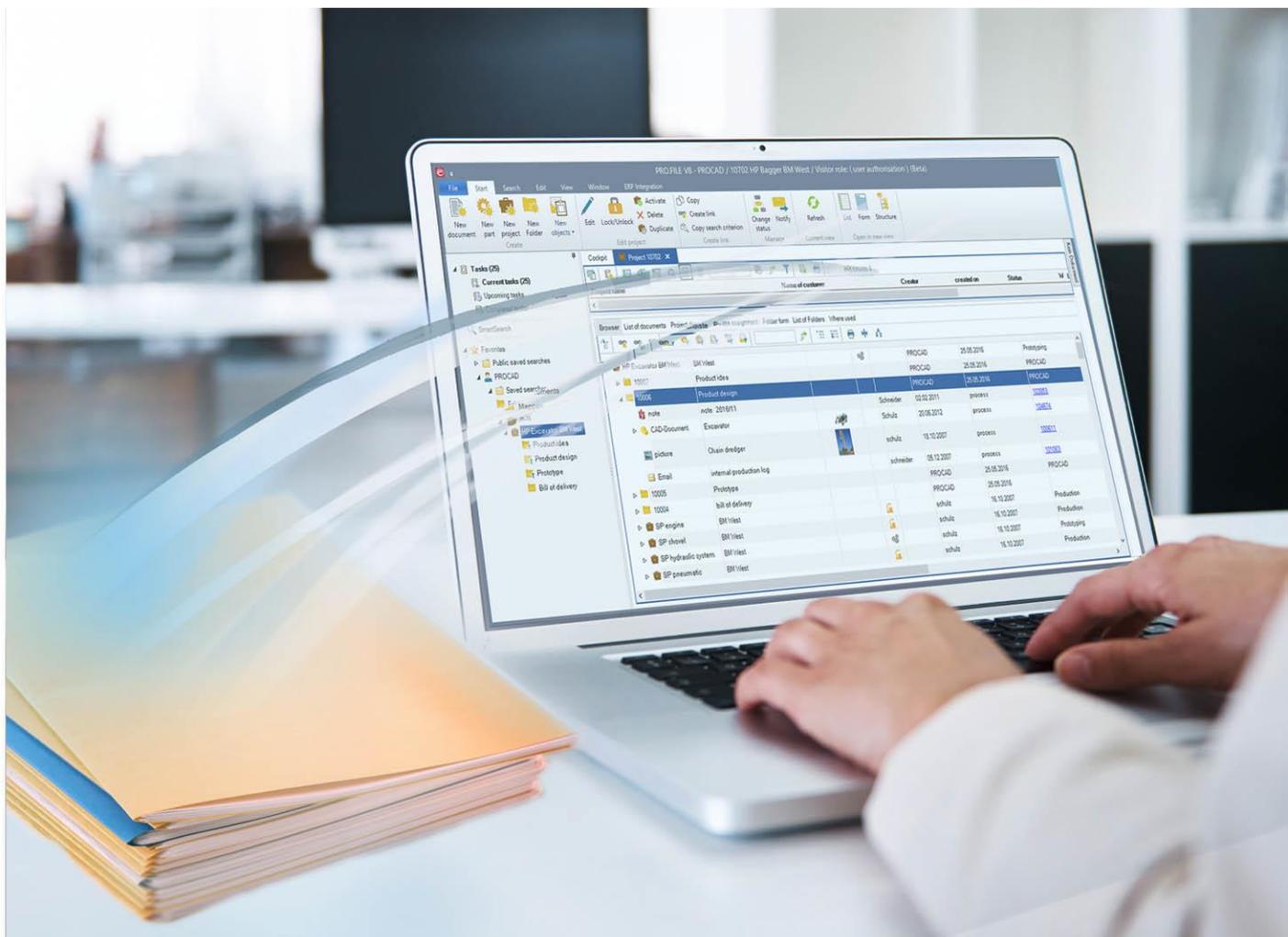


# Functions of the integration PRO.FILE Creo Elements Direct Drafting (ME10)

PRO.FILE Release 8.7  
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Vincenz-Prießnitz-Straße 3 • 76131 Karlsruhe • info@procad.de • www.procad.de

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# About this manual

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

**Step-by-step instructions:**

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

**Menu sequences and function calls**

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

Example:

"File" => "New" => "Document description"

**Buttons and keys**

Keys and buttons are highlighted by angle brackets.

Example:

"Confirm with <OK>."

**Notes and warnings**

To highlight special information the following icons are used:



**Function call:**

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



**Example:**

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



**Note:**

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



**Attention:**

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



**Important notes:**

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



**Attention – Undo not possible:**

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

# 1 The integration PRO.FILE – Creo Elements / Direct Drafting

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 Element / Direct Drafting. Drawings and CAD models can be loaded from or saved to PRO.FILE directly from Creo Elements / Direct Drafting.

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 Elements / Direct Drafting to ERP systems.

## 1.1 The contents of this manual

The following chapters describe the operation of PRO.FILE within the CAD system Creo Elements / Direct Drafting.

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 Elements / Direct Drafting. The following topics will be addressed:

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



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

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

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

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 Elements / Direct Drafting.

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

- [How to start the integration?](#)
- [Where can I find the functions of the PRO.FILE integration?](#)
- [How to log in to PRO.FILE?](#)
- [A brief overview: The functions of the integration](#)

### 2.1 How to start the integration?

After the installation of the integration PRO.FILE – Creo Elements / Direct Drafting an icon to start the integration is available on your desktop

Use this icon to start the integration – and thus start Creo Elements / Direct Drafting and PRO.FILE.



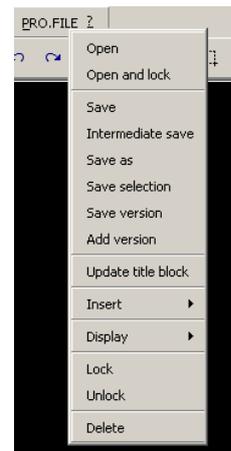
**Attention:** Always start the integration via the corresponding icon

Only when Creo Elements / Direct Drafting is started via the icon of the PRO.FILE integration, the direct connection between the CAD system and PRO.FILE is available.

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

The functions of the PRO.FILE integration can be accessed directly in Creo Elements / Direct Drafting via the menu "PRO.FILE":

1. Start "Creo Elements / Direct Drafting. "
2. Go the menu bar to the section "PRO.FILE".
3. Select the desired integration function from the menu.



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

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



### Note:

If you start the integration for the first time, you may possibly have to configure the required menu manually. Detailed information can be found in the configuration manual of the integration.

## 2.3 How to log in to PRO.FILE?

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

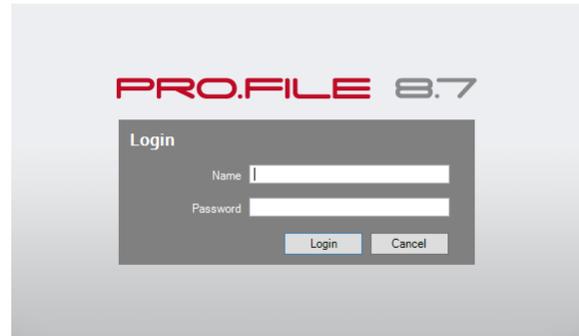
Via the logon, the user is now identified by his PRO.FILE user name and password. Based on this logon, the user rights, start statuses and function access rights for the logged-on user are activated.

In the login screen, please enter:

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

Confirm with <OK>.

The PRO.FILE home screen is now displayed.



**Note: No login required in case of "Autologin"**

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

## 2.4 A brief overview: The functions of the integration

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

- **"Open":**  
This function opens PRO.FILE and prompts the user to select a CAD document for loading in Creo Elements / Direct Drafting.
- **"Open and lock":**  
This functions also opens a drawing from PRO.FILE in Creo Elements / Direct Drafting. After the loading, the drawing is reserved for you – and locked for other users for editing. You can directly save your changes back to PRO.FILE.
- **"Save":**  
Via this function newly created CAD documents are checked in to PRO.FILE, or documents already saved in PRO.FILE and checked out for editing are saved back to PRO.FILE. When changes are saved back, the existing document in PRO.FILE is overwritten automatically.
- **"Intermediate save":**  
Saving of a drawing known in PRO.FILE, without removing the lock and without closing the file or clearing the screen.
- **"Save as":**  
First-time saving of a drawing not yet known in PRO.FILE.

- **"Save selection"**  
Saving of a part from a part structure of the active drawing as new document in PRO.FILE.
- **"Save version"**  
Saving of a drawing already known in PRO.FILE as a new version.
- **"Add version"**  
With this function you save a new drawing in PRO.FILE as a version of a drawing already existing in PRO.FILE.
- **"Update title block":**  
Via this function the title block of a drawing can be filled with a call. Prerequisite for this is that these lists and fields have already been configured for the used drawing template.
- **"Insert":**  
This menu entry allows you to insert elements from PRO.FILE into the active drawing. Via the sub-menus you can insert a part, bill of materials, position number or frame in different sizes or even a customized frame from PRO.FILE.
- **"Display":**  
Via each of these functions PRO.FILE is opened and the part form, the part structure, the part usage, the bill of materials, the document form, the document structure, the document usage and all document versions are displayed. You can also display information on the installed integration and the PRO.FILE online help.
- **"Lock":**  
CAD documents loaded from PRO.FILE in Creo Elements / Direct Drafting are not automatically locked for other users. If you want to edit a CAD document, you have to use the command "Lock" before making the changes. This prevents the document from being changed by a different user in the meantime.
- **"Unlock":**  
With this function CAD documents, which were locked for editing in Creo Elements / Direct Drafting, can be unlocked, so that other users can also edit the document.
- **"Delete":**  
With this function only the current screen in Creo Elements / Direct Drafting is cleared.

### 3 Functions for opening CAD documents from PRO.FILE

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

If you want to load a drawing from PRO.FILE in Creo Elements / Direct Drafting, the PRO.FILE menu in Creo Elements / Direct Drafting offers two functions.

These functions basically differ in the option whether the drawing is to be reserved for editing or not:

- ["Open": Load drawing from PRO.FILE for viewing](#):  
The selected drawing is loaded from PRO.FILE as it was saved the last time.

PRO.FILE	
STATUS	
Part No.:	203961
Doc. No.:	106896
Status:	unlocked

- ["Open and lock": Open drawings for editing](#)  
After the loading, the drawing is reserved for you and locked for changes by other users in PRO.FILE. Changes can be saved back directly to PRO.FILE.

PRO.FILE	
STATUS	
Part No.:	203961
Doc. No.:	106896
Status:	locked

When the drawing is opened, the status window displays important information:

- "Part No." shows the PRO.FILE ID number of the corresponding part description.
- "Doc No." shows the ID number of the corresponding document description.
- "Status" informs you of whether your drawing is locked or not.



#### Attention:

Files loaded from PRO.FILE in Creo Elements / Direct Drafting with the function "Open" are not locked and can still be edited by other users. In this case, a lock has to be manually initiated – via the function ["Starting your changes: Lock the CAD document"](#).



#### Note: PRO.FILE checks permissions

When the function "Open" is used for documents from PRO.FILE, the corresponding access rights of the user are checked:

- Is the allowed to see the document?
- Is the user allowed to edit the document?

These access permissions depend on both the user permissions and the status of the document. A document in a "released" status, usually must not be edited any more.

### 3.1 "Open": Load drawing from PRO.FILE for viewing

To open Creo Elements / Direct Drafting data from PRO.FILE proceed in two steps:



Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:

"PRO.FILE" => "Open"

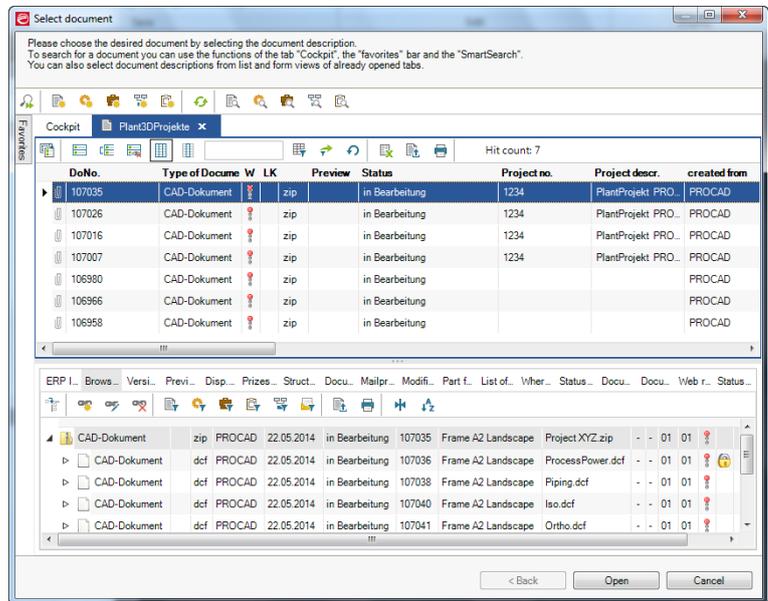
Step 1: Using the PRO.FILE function "Open"

1. Go into the menu bar of Creo Elements / Direct Drafting into the menu "PRO.FILE".
  2. Select the menu entry "Open".
- ⇒ The Checkout wizard for the selection of the desired document 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. the desired document is displayed in a list view, you can select it. (If the desired document is displayed in a form view, it is already selected. Click <Open>.
- ⇒ The Checkout wizard closes and the dialog screen for the loading type is displayed.

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

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

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

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

- To do so, use the function "Starting your changes: "Lock" the CAD document", as described in the following chapter.



#### Attention: Document locked by other user

If you want to open a document, that is locked by a different PRO.FILE user, a message will inform you of this.

You can still open the document.

However, you cannot lock the document and save any changes back to PRO.FILE. This is only possible after the other user has finished his modification, unlocked the document and you have loaded the most recent state from PRO.FILE again.

### 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:

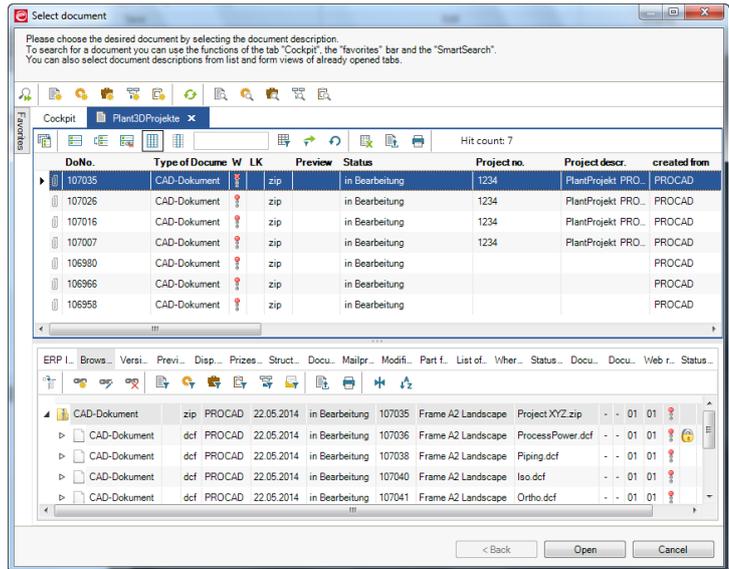
- For PRO.FILE to know which document is to be opened, the desired document description must
  - either be selected in a list view or a structure browser
  - or be displayed in a form view
- Then the button <Open> at the right bottom of the Checkout wizard has to be clicked.

**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:

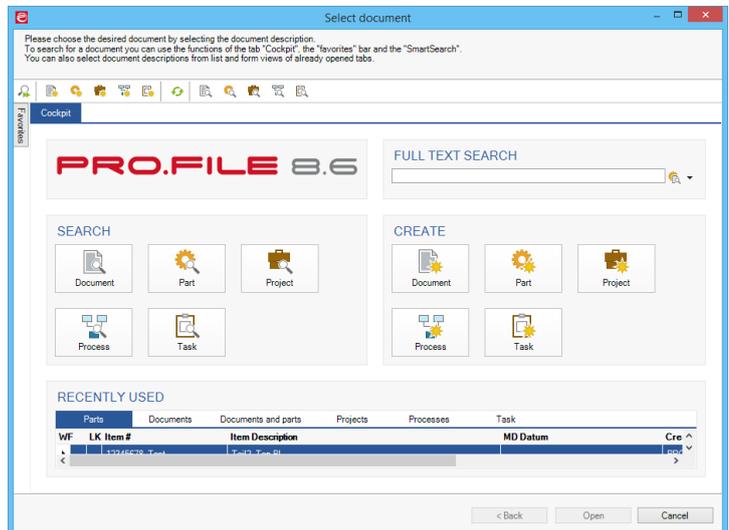
- If search results had previously been opened in a list or form view in a separate tab, you can directly access the displayed documents.

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 and lock": Open drawings for editing

If you want to load a CAD document from MicroStation in order to edit it, this is only possible if the CAD document is reserved for you and thus locked for other users.

To achieve this when opening the CAD document, you can use the function "Open and lock".

Always open a drawing with the function "Open and Lock" if you intend to modify the drawing. This is to avoid conflicts with other users who may want to modify the same drawing. The lock makes sure that your changes can be saved back to PRO.FILE.



Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:

"PRO.FILE" => "Open and lock"

The procedure is identical to the previously described function ["Open": Load drawing from PRO.FILE for viewing](#):

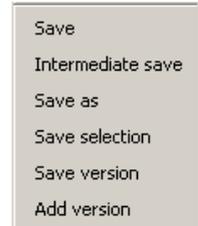
1. Select the "PRO.FILE" menu from the menu bar in Creo Elements / Direct Drafting.
2. Select the function "Open and lock".
  - ⇒ The Checkout wizard for the selection of the desired document is displayed.
  - ⇒ The Checkout wizard displays the PRO.FILE GUI as it was when last used by you.
3. In the Checkout wizard select the desired CAD document and click <Open>.
  - ⇒ The selected CAD documents with its components is opened from PRO.FILE in Creo Elements / Direct Drafting.
  - ⇒ The CAD document is reserved for you and can be modified.

With the function "[Save](#)": Resaving CAD objects in PRO.FILE" the reservation of a CAD document in PRO.FILE is removed and the drawing is closed.

## 4 Saving functions: Saving CAD data and changes to PRO.FILE

To save CAD documents to PRO.FILE the following functions are available in the PRO.FILE menu in Creo Elements / Direct Drafting:

- ["Save"](#):  
Saving of a drawing already known in PRO.FILE. The drawing is closed after saving.
- ["Intermediate save"](#):  
Saving of a drawing already known in PRO.FILE without removing the lock and closing the file.
- ["Save as"](#):  
Saving of a new drawing yet unknown in PRO.FILE.
- ["Save selection"](#):  
Saving of a part from the part structure of the active drawing.
- ["Save version"](#):  
Saving of a drawing (already known in PRO.FILE) as a new version.
- ["Add version"](#):  
With this function a new Creo Elements / Direct Drafting drawing can be saved as a version of an already existing in PRO.FILE.



When saving changes to CAD documents to PRO.FILE, the following has to be noted:

- You can only save documents that have been reserved:  
PRO.FILE forbids concurrent changes when working with the CAD system. You therefore have to make sure that no other user is editing the document at the same time than you are.  
The only way to guarantee this is to set a lock flag in PRO.FILE, stopping other users from opening the CAD document for editing. This can be achieved via the following functions:
  - ["Open and lock": Open drawings for editing"](#): The document is reserved automatically after being loaded from PRO.FILE. Other users can no longer change the document.
  - ["Starting your changes: Lock the CAD document"](#): A CAD document opened only for viewing can be reserved at a later point in time. By this reservation you get exclusive modification access to the document and therefore can save your changes back to PRO.FILE.

- When changes are saved back to PRO.FILE, the previous status of the drawing is overwritten:  
When a drawing is loaded from PRO.FILE in Creo Elements / Direct Drafting and modified, the status in the Creo Elements / Direct Drafting session is the most recent one and newer than the status saved in PRO.FILE.  
When the CAD document is saved back to PRO.FILE, the status saved in PRO.FILE is overwritten with the newer status from Creo Elements / Direct Drafting.
- Keeping the status of a drawing saved in PRO.FILE, without overwriting  
You have two possibilities of saving changes to a drawing to PRO.FILE without overwriting the original drawing:
  - The function ["Save version"](#) keeps the original drawing as old version.
  - With the function ["Save as": Saving CAD objects for the first time"](#) you can save the current drawing again to PRO.FILE as a new, independent document with a new document description.

## 4.1 "Save": Resaving CAD objects in PRO.FILE

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



**Note:** "Save" only for drawings already known in PRO.FILE

If you want to save a new drawing in PRO.FILE, you have to use the function ["Save as": Saving CAD objects for the first time"](#).

After a drawing has been loaded in Creo Elements / Direct Drafting with the command "Open", it can be locked, modified and saved back.

If you select "Save", the drawing saved in PRO.FILE is modified.



**Attention:** Only documents that have been locked can be saved again

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. See chapter ["Starting your changes: Lock" the CAD document"](#).

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

This means: Without "Lock" the changes to a CAD document cannot be saved back to PRO.FILE

This chapter describes the procedure "Save" for changed documents:

- The changes made to the CAD document are saved to PRO.FILE.
- The status saved in PRO.FILE until then is overwritten with the changed status.
- The reservation of the CAD document is removed. Other users can access the document for editing.
- The document is closed after saving. (If you want to continue editing your drawing, you should use the function "[Intermediate save: Save and continue](#)" instead.



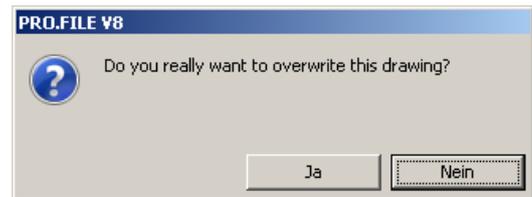
Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:

"PRO.FILE" => "Save"

Proceed as follows

1. Select the menu "PRO.FILE" from the menu bar in Creo Elements / Direct Drafting.
  2. Select the function "Save".
- ⇒ PRO.FILE recognizes the CAD document as a PRO.FILE object and goes into change mode.

A dialog appears asking you whether the CAD document in PRO.FILE really is to be overwritten with your changes.



3. If you want to overwrite the status in PRO.FILE, confirm with <Yes>. If you click <No>, the saving process is cancelled.
- ⇒ If you confirmed with <Yes>, your changes are saved in PRO.FILE and the document in PRO.FILE is overwritten.

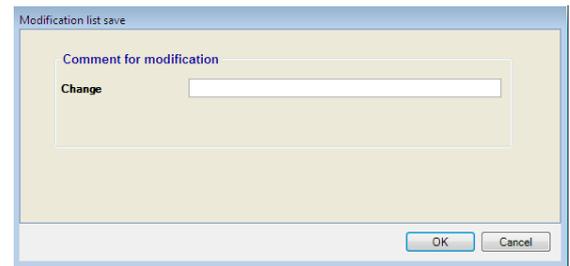


Attention:

By overwriting the document in PRO.FILE, the old status of the drawing is irretrievably lost

### Optional: Enter modification comment

4. 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.



5. 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.
  - ⇒ Your reservation of the document is removed. The document can now be edited by other users.
  - ⇒ The saving process is finished.

If you want to save the status of a CAD document in PRO.FILE without closing the file in Creo Elements / Direct Drafting, you can use the function ["Intermediate save": Save and continue](#) which is described in the following chapter.

## 4.2 "Intermediate save": Save and continue

The function "Intermediate save" is almost identical to the function "Save" – with the important difference that your reservation of the document is not removed after the saving process. The document remains locked for other users:

- The changes made to the CAD document are saved to PRO.FILE.
- The status saved in PRO.FILE until then is overwritten with the changed status.
- The reservation of the CAD document is not removed. Other users still cannot modify this document.
- No modification comment is queried.
- The fields of the drawing title block are updated.
- You can immediately continue editing the CAD document.



Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:

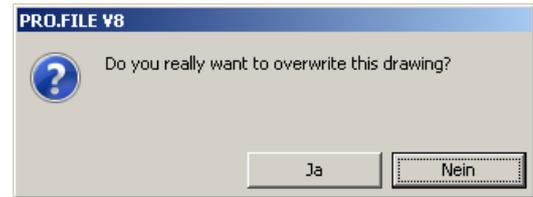
"PRO.FILE" => "Intermediate save"

Proceed as follows

1. Go to the integration menu "PRO.FILE" in Creo Elements / Direct Drafting.
2. Select the function "Save CAD document intermediate".

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

⇒ A dialog appears asking you whether the CAD document in PRO.FILE really is to be overwritten with your changes.



⇒ If you want to overwrite the status in PRO.FILE, confirm with <Yes>. If you click <No>, the saving process is cancelled.

⇒ If you confirmed with <Yes>, your changes are saved in PRO.FILE and the document in PRO.FILE is overwritten.



#### Attention:

By overwriting the document in PRO.FILE, the old status of the drawing is irretrievably lost!

⇒ Your reservation of the document is not removed after the saving process. The document remains locked for other users.



#### Note: No modification comment during intermediate saving

Even if the option of a modification comment upon saving is configured for your document types, no modification comment is requested during intermediate saving.

## 4.3

### "Save as": Saving CAD objects for the first time

With the function "Save as" you can save drawings created in Creo Elements / Direct Drafting to PRO.FILE.

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



#### Note:

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

To save a document from AutoCAD to PRO.FILE for the first time proceed as follows:



Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:

"PRO.FILE" => "Save as"

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

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

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

These steps are described in the following sub-chapters.

#### 4.3.1

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

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

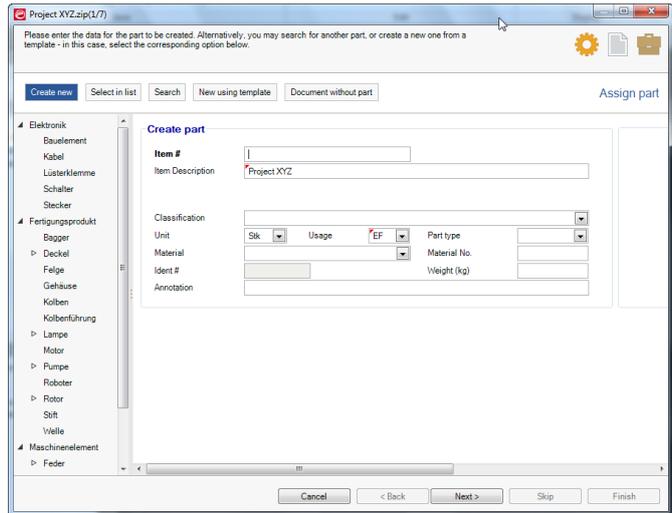


#### Note: Usage of PRO.FILE parts

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

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

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



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



Create new:

Usage:

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

Proceeding:

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



Select in list:

Usage:

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

Proceeding:

1. Click the option <Select in list> to select the desired part description.
- ⇒ The wizard displays the PRO.FILE surface, as it was opened the last time.

2. If the part master record desired for assignment is not yet displayed in a list or form view, you can use the search functions of the home page, the icon bar or favorites and SmartSearch to start a selection.
3. If the part master record desired for assignment is displayed in a list view, you can now select it. (If the desired part master record is displayed in form view, it is already selected automatically).
4. Confirm your selection with <Next>.

**Search**

Search:

Usage:

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

Proceeding:

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

**New using template**

New using template:

Usage:

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

Proceeding:

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

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

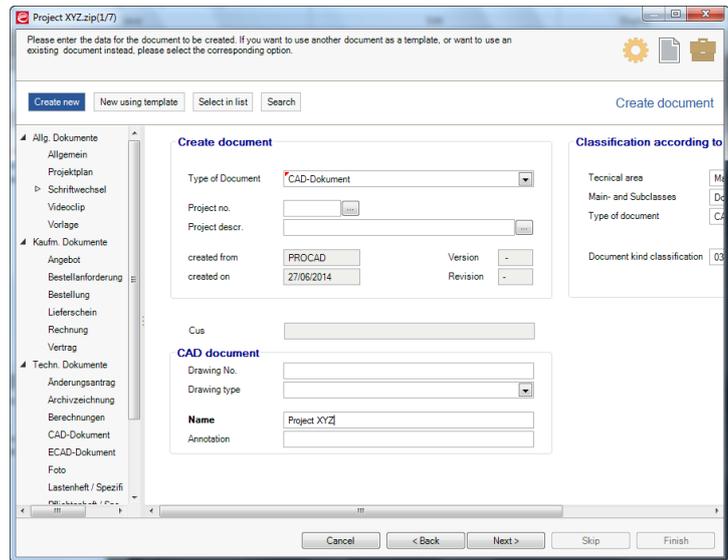
### 4.3.2 Checkin wizard Step 2: Creation of the document description in PRO.FILE

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

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

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

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



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



- Create new
- New using template

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

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

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

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

### 4.3.3 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.

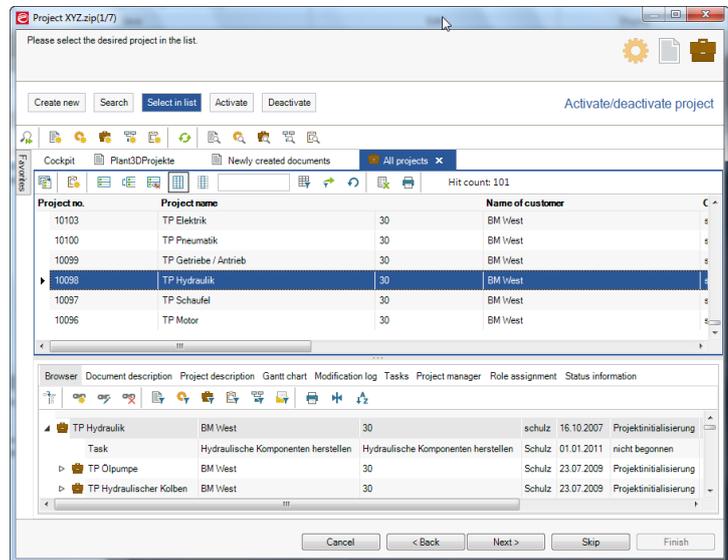


#### Note: Usage of PRO.FILE projects

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** Create new:  
A new project is created in PRO.FILE. The part master record and document description created in steps 1 and 2 are assigned to this new project.
- **Search** Search:  
The part master record and document description created in steps 1 and 2 are to be assigned to an existing project. This project is now searched via the search form and selected.
- **Select in list** Select in list:  
The part master record and document description created in steps 1 and 2 are to be assigned to an existing project. This project is already displayed in a PRO.FILE list and only has to be selected and confirmed.
- **Activate** Activate:  
If a project is activated, all new parts and documents in PRO.FILE are automatically assigned to this project. If no project is currently activated, and you want to do so, you can use this function to activate a project.
- **Deactivate** Deactivate:  
Again: If a project is activated, all new parts and documents in PRO.FILE are automatically assigned to this project. If this assignment is not to be made for the current document, you can deactivate the project before finalizing the saving process.  
Note: If a project is activated, this is displayed in the title bar of the active PRO.FILE window.

#### Proceeding:

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

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

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

## 4.4 "Save selection"

The function "**Save selection**" allows the saving of a specific part from the part list of the active drawing. This part can be selected in the part structure editor and saved separately to PRO.FILE via the Checkin wizard.



Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:

"PRO.FILE" => "Save selection"

Proceed as follows

1. Select the "PRO.FILE" menu from the menu bar in Creo Elements / Direct Drafting.
2. Select the function "Save selection".
  - ⇒ The part structure editor of Creo Elements / Direct Drafting is displayed.
3. Select the part to be saved from the Creo Elements / Direct Drafting part list.
  - ⇒ The PRO.FILE Checkin wizard is displayed.
4. You can now save the selected element in PRO.FILE.

The proceeding corresponds to the proceeding for the function "["Save as": Saving CAD objects for the first time](#)". You can find a detailed description in the previous chapter.

## 4.5

### "Save version"

The function "Save version" can be used to save a drawing already known in PRO.FILE without losing the status saved in PRO.FILE. The drawing is saved under a new version number in PRO.FILE.

- Only the document active in the CAD session is versioned.
- The old version remains in PRO.FILE.
- The new version is saved with a new document ID in PRO.FILE and displayed in Creo Elements / Direct Drafting.
- Creo Elements / Direct Drafting- drawings, into which the told version of a drawing had been inserted, are not changed or updated. (Drawings are inserted, not referenced).
- You can create as many versions of a drawing as is required.



Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:

"PRO.FILE" => "Save version"

Proceed as follows

1. Go to the integration menu "PRO.FILE" in Creo Elements / Direct Drafting.
2. Select the function "Save version".
  - ⇒ A new version of the active CAD document is now saved in PRO.FILE.
  - ⇒ A message will inform you of the successful creation of the version.

⇒ The new version is displayed in Creo Elements / Direct Drafting.



#### Attention: New version is not locked

The new version that has just been saved in PRO.FILE is not locked. To lock the document please use the function described in the chapter "Starting your changes: "Lock" the CAD document".

The document list always displays the most recent version. To display older versions/revisions of a document you can use the function "PRO.FILE" => "Display" => "All document versions" in the integration.



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

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

## 4.6

### "Add version"

With this function it is possible to save a new Creo Elements / Direct Drafting drawing as version of a drawing already existing in PRO.FILE.

- Only the document active in the CAD session is versioned.
- The old version remains in PRO.FILE.
- The new version is attached to an existing document description in PRO.FILE. If this document description already contains a file, the existing file becomes an older version within the version chain.
- Creo Elements / Direct Drafting- drawings, into which the told version of a drawing had been inserted, are not changed or updated. (Drawings are inserted, not referenced).
- You can create as many versions of a drawing as is required.



#### Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:

"PRO.FILE" => "Add version"

Proceed as follows

1. Go to the integration menu "PRO.FILE" in Creo Elements / Direct Drafting.
  2. Select the function "Add version".
- ⇒ The Checkout wizard to select the desired document is displayed.

- ⇒ Select the document description to which the active Creo Elements / Direct Drafting drawing is to be attached as new version. Information on the Checkout wizard can be found in the previous chapter "[Open](#)": [Load drawing from PRO.FILE for viewing](#)".
3. Confirm your selection with <Open>.
- ⇒ The active drawing is now saved as newest version of an existing document in PRO.FILE.
- ⇒ The versioned drawing remains open in Creo Elements / Direct Drafting and is locked in PRO.FILE. You can continue editing the drawing.

**Attention:**

If the selected document description is used in other documents, i.e. the drawing is referenced in another document (e.g. from a different CAD system), a message will inform you of this.

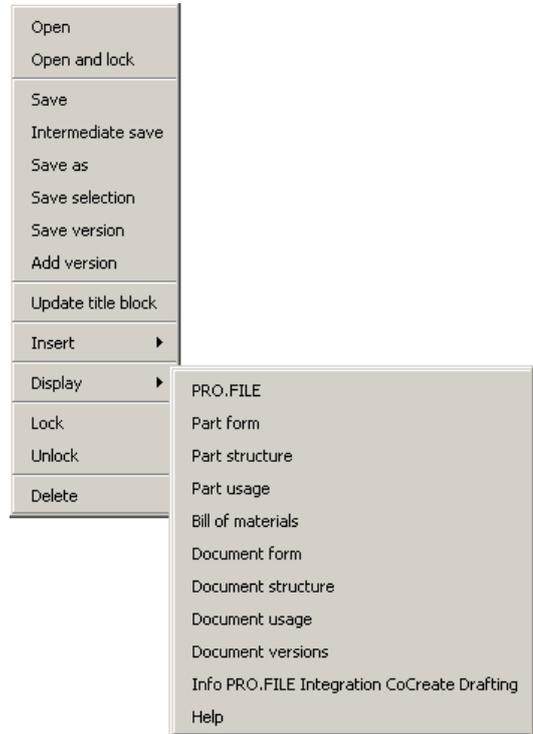
If a new version is "plonked" in front of this referenced document, the referencing documents can no longer be loaded with the newest versions!

- In this case, it is advisable to select <No>. The saving process is thus canceled.
- With <Yes> you confirm that the new version is to be created and the existing references are to point to this new version. This is not recommended.

# 5 Display: Information on a drawing from PRO.FILE

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

- These menu entries access the information directly on the drawing active in Creo Elements / Direct Drafting.
- The various menu entries thus allow direct access to required information without having to navigate in PRO.FILE.
- Once an entry is selected, pf opens and displays the selected view.
- Within the displayed lists or forms you can perform all actions available in PRO.FILE.



Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:  
 "PRO.FILE" => "Display" => "..."



**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 views are available:

- **PRO.FILE:**  
 Via "Display" => "PRO.FILE" the PRO.FILE window is opened. You can now perform all basic actions, e.g. modify document or part descriptions or make workflow status changes.

- **Part form:**  
 Via "Display" => "Part form" you get the form view of the part description connected to the active drawing in PRO.FILE.
- **Part structure:**  
 Via "Display" => "Part structure" you start the structure browser of the part description connected to the active drawing in PRO.FILE.
- **Part usage:**  
 Via "Display" => "Part usage" you start the part usage browser of the part description connected to the active drawing.
- **Bill of materials:**  
 Via "Display" => "Bill of materials" you get – if available – the bill of materials for the active drawing.



**Attention:**

Please consider the effects of the BOM-Versioning – if activated in PRO.FILE. For further information see the manual "Working with bills of materials and structures in PRO.FILE".

- **Document form:**  
 Via "Display" => "Document form" you get the form view of the document description connected to the active drawing.
- **Document structure:**  
 Via "Display" => "Document structure" you get the structure browser of the document description connected to the active drawing.
- **Document usage:**  
 Via "Display" => "Document usage" you receive the document usage browser of the document description connected to the active drawing.
- **Document versions:**  
 Via "Display" => "Document versions" you receive the version list of the document descriptions connected to the active drawing.
- **Info PRO.FILE Integration Creo Elements/Direct Drafting:**  
 Via "Display" => "Info PRO.FILE Integration Creo Elements / Direct Drafting" you are informed about the version number of the installed PRO.FILE – Creo Elements / Direct Drafting Integration. With this function you can check whether you work with the current version of the integration.

- **Help:**  
Via this menu entry, the online help for the integration PRO.FILE – Creo Elements / Direct Drafting is started. By default, this help is opened in its most recent version from the PROCAD help portal in HTML format.

## 6 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.

The status window of the integration m10 displays the status (locked/unlocked) of the loaded drawing.

PRO.FILE	
STATUS	
Part No.:	203962
Doc. No.:	106897
Status:	unlocked

For further information see the following sub-chapters:

- [Starting your changes: "Lock" the CAD document](#)
- [The "Unlocking" of CAD documents](#)



### Attention: You cannot save without locking before

The saving functions "Save", "Save selection" and "Save version" can only be used, if the drawing has been locked before.

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".

## 6.1 Starting your changes: "Lock" the CAD document

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



Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:

"PRO.FILE" => " Lock"

Proceed as follows

1. Select the menu "PRO.FILE " from the menu bar in Creo Elements / Direct Drafting.
  2. Select the function "Lock".
- ⇒ By the command "Lock" the user permissions to edit the document are checked, and the drawing is locked for all other users in the database.
- ⇒ If the locking was successful, a corresponding message is displayed.
3. Confirm this message with <OK>.
- ⇒ The CAD document is reserved for you and can be modified. Changes can be saved back directly to PRO.FILE via the function "Save".



### Note: Why can I not lock a document?

You want to lock a document after opening it, but PRO.FILE does not allow this? This can have two reasons:

- The document is already locked by a different PRO.FILE user. You can see the name of this user in PRO.FILE by selecting the document and looking at the dependent tab "Status information"
- The document is in a status, in which you have no permission to change the file. This is, for example, the case with a released status.



### Attention: Modifications in the team

It is advisable to lock documents opened for editing immediately after opening.

If you have already opened a drawing from PRO.FILE in Creo Elements / Direct Drafting and want to make changes to it, you must lock the drawing to make sure that your changes can be saved back to PRO.FILE.

## 6.2 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".

It is then possible for other users again to open and edit the drawing.



### 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 Elements / Direct Drafting:

"PRO.FILE" => "Unlock"

Proceed as follows

1. Make sure that the CAD document to be unlocked is displayed in Creo Elements / Direct Drafting.
  2. Select the menu "PRO.FILE" from the menu bar.
  3. Click on the function "Unlock".
- ⇒ The lock flag for the active drawing is removed in PRO.FILE.
- ⇒ A message will confirm this.



4. Close the message with <OK>.

## 7 Additional functions of the integration

In addition to the basic functions, the "PRO.FILE" menu in Creo Elements / Direct Drafting offers the following functions:

- ["Update title block"](#)
- ["Insert": Place elements from PRO.FILE on the drawing](#)
- ["Delete": Close the drawing, clear screen](#)

### 7.1 "Update title block"

This function allows you to update the title block of a drawing in Creo Elements / Direct Drafting with up to data information from PRO.FILE.



Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:

"PRO.FILE" => "Update title block"

Proceed as follows

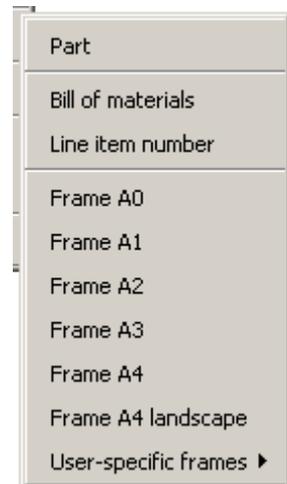
1. Select the "PRO.FILE" menu from the menu bar in Creo Elements / Direct Drafting.
  2. Select the function "Update title block".
- ⇒ PRO.FILE updates the title block information on the drawing.

### 7.2 "Insert": Place elements from PRO.FILE on the drawing

Via the sub-menu "Insert" of the integration PRO.FILE – Creo Elements / Direct Drafting you can place different elements from PRO.FILE on the active drawing:

- [Insert: "Part"](#)
- [Insert: "Bill of materials"](#)
- [Insert: "Line item number"](#)
- [Insert: "Frame"](#)
- [Insert: "User-specific frames"](#)

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



## 7.2.1

### Insert: "Part"

If you want to insert an existing drawing from PRO.FILE into a new drawing in Creo Elements / Direct Drafting, then you can use the function "Insert" => "Part".

#### Please note:

- The part is inserted as a copy. Later changes to the part saved in PRO.FILE do not affect the drawing in which the part has been inserted.
- For this reason, PRO.FILE does not build up a document structure for inserted parts. However, you can create a bill of materials in PRO.FILE via the function "Insert" => "Bill of materials". This bill of materials is created based on the inserted parts.



Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:

"PRO.FILE" => "Insert" => "Part"

#### Proceed as follows

1. Select the "PRO.FILE" menu from the menu bar in Creo Elements / Direct Drafting.
2. Select the function "Insert" => "Part".  
⇒ The Checkout wizard for the selection of the desired document is displayed.
3. Select the desired part drawing in the Checkout wizard. Detailed information can be found in the previous chapter "["Open": Load drawing from PRO.FILE for viewing](#)".  
⇒ After confirmation via the button <Open> you can place the selected drawing in your active drawing.
4. To do so, select the placing position for the component in Creo Elements / Direct Drafting.

The insertion process is now finished. You can now update the bill of materials.



#### Attention:

The part is inserted as a copy. Later changes to the part saved in PRO.FILE do not affect the drawing in which the part has been inserted.

## 7.2.2 Insert: "Bill of materials"

If you have inserted parts into a drawing, you can create a bill of materials in PRO.FILE via the command "Insert" => "Bill of materials".

### Please note:

- The bill of materials is only produced for the top level, this means for the parts which were inserted directly. If components were inserted in the inserted components also, then a bill of materials must be produced for these components separately using "Insert" => "Bill of materials".
- All inserted parts must already be stored in PRO.FILE, otherwise the bill of materials cannot be produced correctly.

You can display the produced parts list e.g. using "Display" => "Bill of materials".



Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:

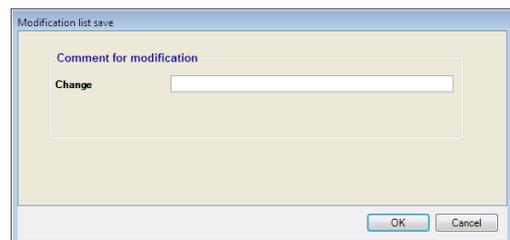
"PRO.FILE" => "Insert" => "Bill of materials"

### Proceed as follows

1. Select the "PRO.FILE" menu from the menu bar in Creo Elements / Direct Drafting.
2. Select the function "Insert" => "Bill of materials".  
=> If differences between the status in PRO.FILE and the CAD system occur during the insertion, a window with the positions in question is displayed.
3. In this case select the positions to be either removed from or added to the bill of materials in the window "Compare quantities of BOM positions".

### Optional: Enter modification comment

4. 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.



5. Confirm your modification comment with <OK>  
=> The modification comment screen is closed.  
=> The bill of materials is now created in PRO.FILE.

You can display the created bill of materials, e.g. via "Display" => "Bill of materials".



**Attention:**

Please consider the effects of the BOM-Versioning – if activated in PRO.FILE. For further information see the manual "**Working with bills of materials and structures in PRO.FILE**".

### 7.2.3

#### Insert: "Line item number"

You can insert the position numbers of the bill of materials from PRO.FILE in your drawing via this function.



**Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:**

"PRO.FILE" => "Insert" => "Line item number"

Proceed as follows

1. Select the "PRO.FILE" menu from the menu bar in Creo Elements / Direct Drafting.
  2. Select the function "Insert" => "Line item number".
  3. You are then prompted to identify the object for which you want to insert the position number.
  4. You can then specify the start, folding and end points of the position number by mouse click.
- ⇒ The correct position number is automatically entered on the line between folding point and end point.
- ⇒ If you wish, you can now directly continue with the identification of the next object for the insertion of the next position number.

### 7.2.4

#### Insert: "Frame"

The integration Creo Elements / Direct Drafting makes default drawing frames available. Via the function "Insert" => "Frame ..." you can insert drawing frames in the formats "A0", "A1", "A2", "A3", "A4" and "A4 landscape" in your drawing.



**Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:**

"PRO.FILE" => "Insert" => "Frame ..."

Proceed as follows

1. Select the "PRO.FILE" menu from the menu bar in Creo Elements / Direct Drafting.
  2. Select the function "Insert" => "Frame ...".
- ⇒ The frame is now displayed as outline. You can now freely place the frame.

3. Select the place for the frame and confirm by mouse click.

The frame is now inserted into your drawing.

### 7.2.5 Insert: "User-specific frames"

You can also insert user-specific frames.



Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:

"PRO.FILE" => "Insert" => "User-specific frames" => "..."

The proceeding is identical to the proceeding for the function "Insert" => "Frame ...", which is described in the previous sub-chapter.

### 7.3 "Delete": Close the drawing, clear screen

With the function "Delete" only the active Creo Elements / Direct Drafting screen is cleared. The drawings saved in PRO.FILE are not deleted.

You can use this function if you want to close a drawing after editing, but still want to work in Creo Elements / Direct Drafting.



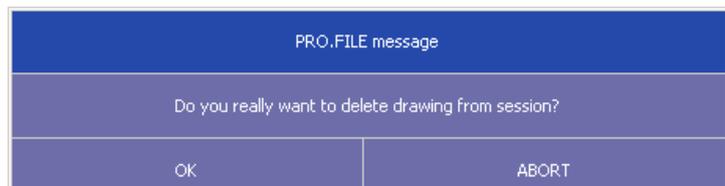
Function call from the PRO.FILE menu in Creo Elements / Direct Drafting:

"PRO.FILE" => "Delete"

Proceed as follows

1. Select the "PRO.FILE" menu from the menu bar in Creo Elements / Direct Drafting.
2. Select the function "Delete".

⇒ The following message is displayed:



3. Consider your choices:
  - <OK>: The active drawing is closed. Changes not yet saved to PRO.FILE are lost!

- <Abort>: You can continue working. Changes not yet saved to PRO.FILE can now be saved.
- ⇒ The drawing is closed. Creo Elements / Direct Drafting displays the empty start screen.



**Attention: Risk of data loss!**

With the function "PRO.FILE" => "Delete" the active drawing is closed. Changes not yet saved to PRO.FILE are lost!

## 8 Important notes

When working with the integration PRO.FILE – Creo Elements / Direct Drafting, please note the following:

- [Multiple sessions](#)
- [Creating a new drawing: Risk of data loss!](#)
- [Mandatory: Assignment of a document description to a part description](#)

### 8.1 Multiple sessions

With Creo Elements/Direct Drafting it is possible to open multiple sessions. This way it is possible to work on several projects simultaneously. The PRO.FILE menu is displayed in every session and can be used normally.



#### Attention:

It is not possible to run several PRO.FILE commands in different sessions at the same time. A new command can be called only if no more else is active in PRO.FILE.

Example: While in a session a drawing with PRO.FILE is opened just now, it is not possible to save a drawing in another session at the same time. Only if the first drawing is opened completely and therefore the command is completed, the next drawing can be saved.



#### Note:

If Creo Elements / Direct Drafting was not duly shut down e.g. because of a crash, while you were active in PRO.FILE, this can cause an error at the next start with the following message : "PRO.FILE connection with other Creo Elements / Direct Drafting window is active! This must be ended first."

To resolve this problem, please run the following macro in Creo Elements / Direct Drafting:

```
pc_unset_lock
```

Make sure that only one Creo Elements / Direct Drafting session is active in PRO.FILE when you execute the macro.

## 8.2 Creating a new drawing: Risk of data loss!

If you want to create a new drawing after loading a drawing from PRO.FILE, you should not use the Creo Elements / Direct Drafting function "File" > "New".

- The newly created drawing would be considered by the integration as the drawing recently opened from PRO.FILE.
- When saving, the new drawing would overwrite the previous drawing.
- Always use the function "Delete" from the PRO.FILE menu of Creo Elements / Direct Drafting instead in order to clear your screen contents.



### Note: Important

If you have opened and locked a drawing from PRO.FILE in Creo Elements / Direct Drafting, all changes to this drawing are saved back to PRO.FILE when the function "PRO.FILE" => "Save" is used.

If the Creo Elements / Direct Drafting function "File" => "New" is used, PRO.FILE does not recognize the result of this action as the beginning of work on a new drawing (unknown in PRO.FILE) but rather as **changes to the drawing loaded from PRO.FILE!**

If the "Save" function of the integration were now called, this would save these "changes" back to PRO.FILE, thus overwriting the status in PRO.FILE with a new, empty drawing.

Therefore, to finish working on a drawing from PRO.FILE, it is important that you use:

- the function "Save", to save your changes back to PRO.FILE and to close the document afterwards, or
- the function "Delete" to close the function without saving your changes back to PRO.FILE.

## 8.3 Mandatory: Assignment of a document description to a part description

The integration PRO.FILE – Creo Elements/Direct Drafting can manage only documents which are assigned to at least one part description.

When saving a drawing to PRO.FILE for the first time, it is therefore necessary that a new part description is created or that an existing part description is selected for assignment.

For detailed information on the usage of part and document descriptions in PRO.FILE see the manual "CAD design supported by PRO.FILE".

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