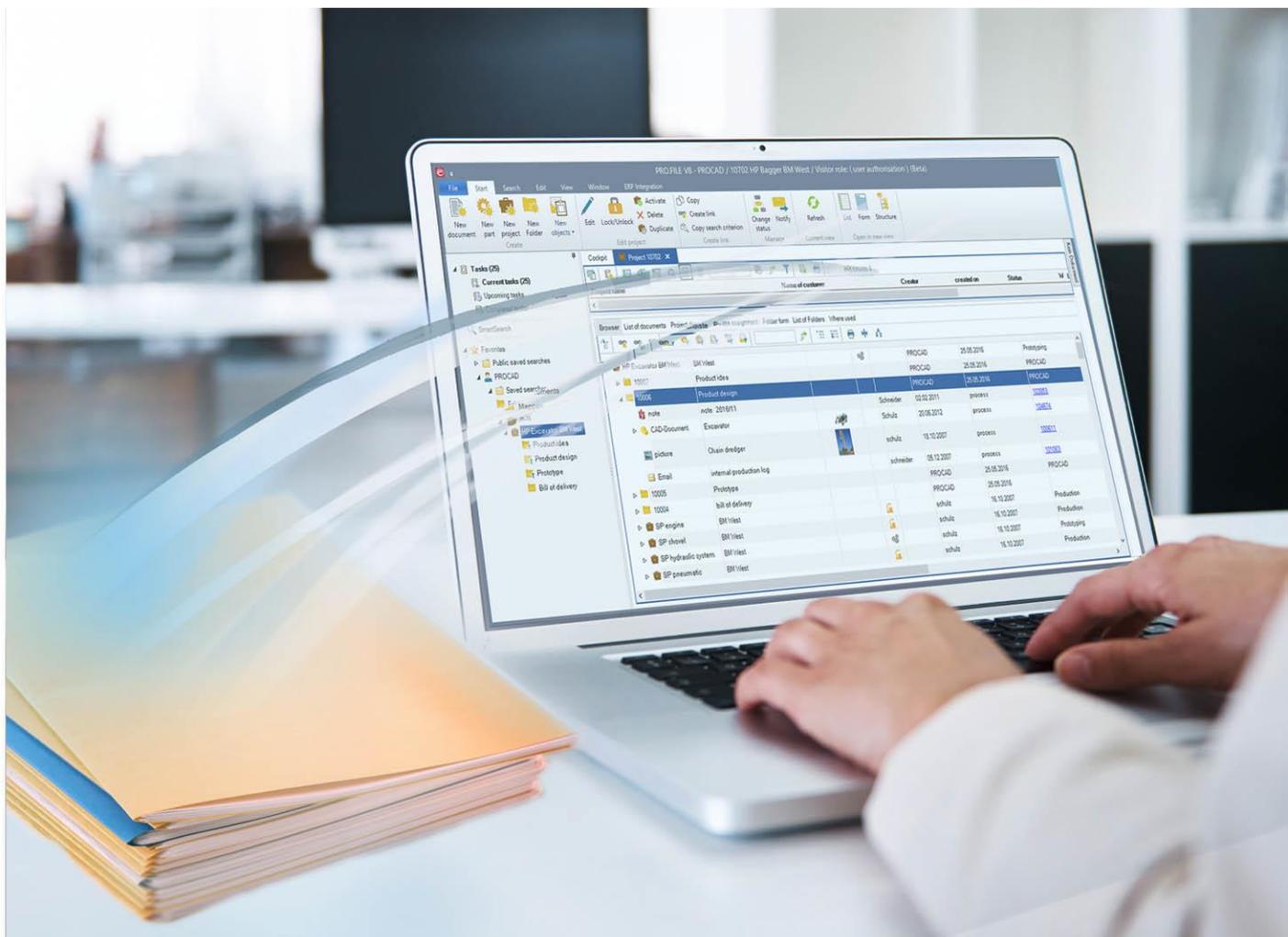


Functions of the integration PRO.FILE Microstation

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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 – MicroStation

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 in Microstation. CAD data and CAD documents are directly loaded from and saved back to PRO.FILE.

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 MicroStation to ERP systems.

1.1 The contents of this manual

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

The descriptions assume that the functions of the PRO.FILE basic software are known or can be looked up in the corresponding manual.

This description documents the interface between PRO.FILE and the CAD system Microstation. The following topics will be addressed:

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



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

When using the integration PRO.FILE – Microstation, 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 MicroStation.

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

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

2.1 How to log in to PRO.FILE?

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

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

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



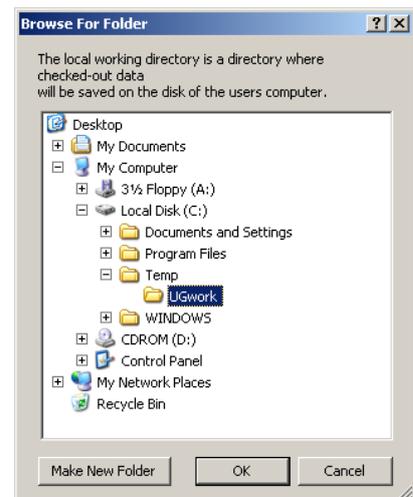
Note: Local work folder is always required

The MicroStation 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 – MicroStation.

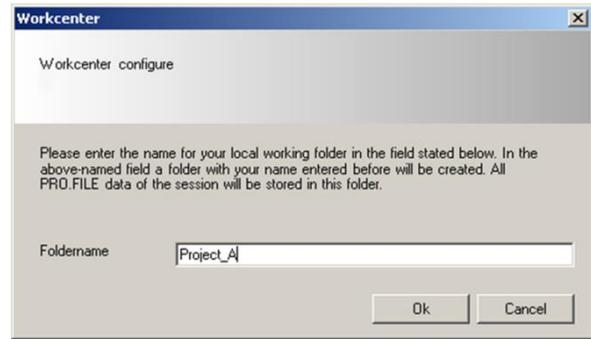
You can choose the work folder for the local storage of the CAD data upon the first start of the integration.

Proceed as follows:

1. If no local work folder is defined when the integration is started, an input screen will prompt you to define this folder..
2. You now have to specify a "root folder". The root folder is the superior folder of the local data storage. In this folder you can later create several work folders, which are then supervised by the "Workcenter".
 - The "root folder" can be selected - or created via the button <Make new folder>..
 - Once you have selected the desired root folder, confirm with <OK>.
3. In the second step, "work folders" are now created In this root folder, which will then be used by the integration. Consequently, you are now prompted to specify a work folder within the root folder:



- Please specify a name for the work folder.
 - Confirm your entry with <OK>.
- ⇒ The configuration of the Workcenter is now finished



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

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

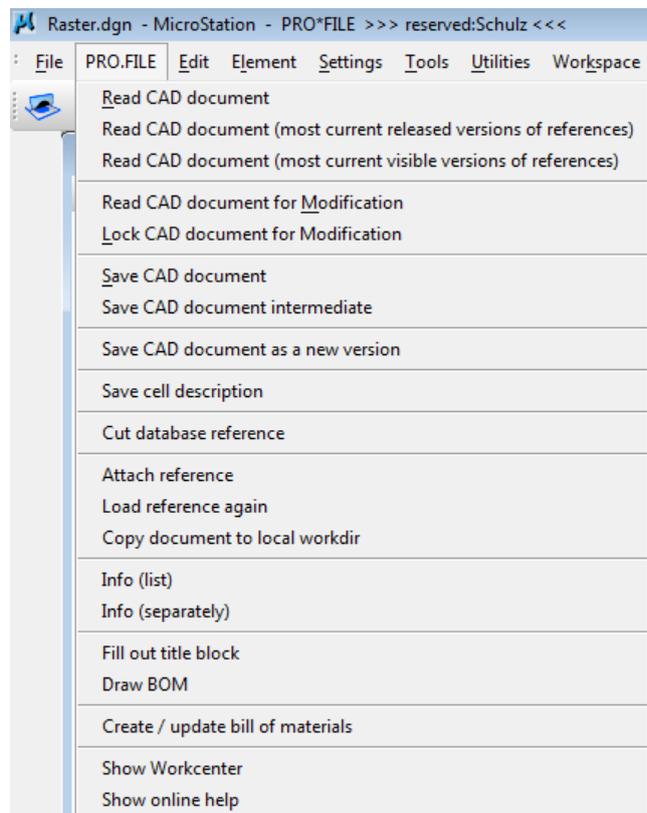
2.3 Where can I find the functions of the PRO.FILE integration?

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

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

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

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



2.4 A brief overview: The functions of the integration

This menu offers you the following functions. Detailed information on each function can be found in the following chapters of this manual.

- The function "**Read CAD document**" is used for retrieving drawings that have not been assigned a lock attribute.
- To display your module assembly only with the most recently released references, use the "**Read CAD document (most current released versions for references)**" function.
- The "**Read CAD document (most current visible versions of references)**" function allows you to show your module assemblies with the latest references.
- The "**Read CAD document for Modification**" function allows you to retrieve drawings that have been assigned a lock attribute.
- If you want to make changes to your drawing and want to stop other users from editing your document in the meantime, you can use the function "**Reserve CAD document.**"
- Use the "**Save CAD documents**" function to save your drawings.
- The "**Save CAD document intermediate**" function is used for temporarily storing drawings for immediate processing.
- "**Save CAD document as a new version**" allows you to save a drawing as a new version.
- The "**Save cell description**" function allows you to save cell information in PRO.FILE.
- Use the "**Cut database reference**" function to remove the link of a drawing with the PRO.FILE database.
- The "**Attach reference**" function allows you to attach reference files.
- Use the "**Load reference again**" function to reload reference files.
- The "**Copy document to local workdir**" function allows you to copy the document into a local working directory.
- The "**Info (list)**" function includes information on all the references stored in this drawing.
- Use the "**Info (separately)**" function to obtain information on a special reference.
- The "**Fill out title block**" function allows you to fill the title block of a drawing with information from the database.
- The function "**Draw BOM**" can be used to position a dynamic bill of materials on the drawing.
- Use the "**Create / update bill of materials**" function to create a new parts list or update an existing one.
- The "**Show Workcenter**" function opens the working directory.
- The function "**Online help**" starts the PRO.FILE help folder.

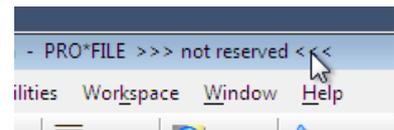
3 Read functions: How to load CAD documents from PRO.FILE in MicroStation?

PRO.FILE administrates CAD data centrally and makes it available to users with the corresponding access permissions via the command "Read CAD document".

If you want to load a drawing from PRO.FILE to display it in MicroStation, the PRO.FILE menu in MicroStation offers four functions. These functions basically differ in the option whether the drawing is to be reserved for editing or not:

- **Read CAD document**
 The selected document is loaded from PRO.FILE as it has been saved the last time. Linked CAD documents are loaded in the version status, in which they were saved last time via the PRO.FILE integration.
- **Read CAD document (most current released versions of references)**
 The selected CAD document is loaded from PRO.FILE with the newest, released versions of the referenced CAD documents. The newest version that is in a released status is loaded.
 When the assembly is loaded with the function "Read CAD document (most current released versions of references)", PRO.FILE first checks, whether the assembly contains parts with versions in released statuses. If this is the case, the part version that was most recently transferred into the released status is loaded.
- **Read CAD document (most current visible versions of references)**
 If further CAD documents are linked to the document to be opened, the newest versions of these linked CAD documents are loaded.
 When the assembly is loaded with the function "Read CAD document (most current visible versions of references)", PRO.FILE first checks, whether the assembly contains parts with versions. If this is the case, the newest visible version of the part, for which the user has viewing permissions, is loaded in MicroStation.

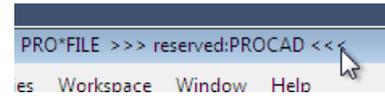
⇒ After the loading, the CAD document is not reserved for you. Changes cannot be saved back to PRO.FILE. (See chapter "[Lock CAD document for Modification: Read-only first, editing later](#)")



Attention:

The files loaded via the three "Read" functions above are not locked in the PRO.FILE database and can be accessed and edited by other users. A document must be actively locked by the user – directly after opening (see below) or later via the function "Lock CAD document".

- **Read CAD document for Modification**
After the loading the CAD document is reserved for you – and locked for other users for modifications. Changes can now be saved back to PRO.FILE directly.



Further information on the read functions can be found in the chapters:

- [Lock CAD document for Modification: Read-only first, editing later](#)
- [Attention: Opening of locally existing files](#)



Note: PRO.FILE checks permissions

When using the new functions "Read CAD document (most current released versions of reference)" and "Read CAD document (most current visible versions of references)" the drawing is not locked in PRO.FILE.

The new functions only make the "visible" versions available. In other words, only versions are displayed for which you have read access rights. If the latest version is not "visible" to you, you will only be shown the latest version that is visible for you.

3.1 Read: Opening documents from PRO.FILE in MicroStation

To open a MicroStation document from PRO.FILE proceed in 2 steps:



Funktionsaufruf call from the PRO.FILE menu in MicroStation:
 "PRO.FILE" => "Read CAD document"

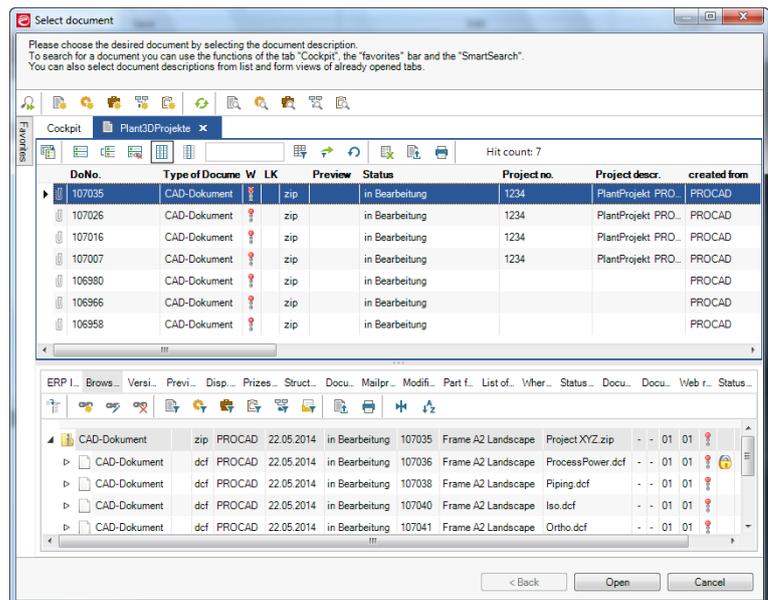
Step 1: Select the PRO.FILE function "Read CAD document"

1. Go into the menu bar of Microstation into the menu "PRO.FILE".
 2. Select the menu entry "Read CAD documents".
- ⇒ "Read CAD documents" loads documents as they were saved the last time in PRO.FILE.
- ⇒ The Checkout wizard for the selection of documents is displayed.

Step 2: Select the desired document in the Checkout wizard

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

3. If the desired document is not yet displayed in a list or form view, you can start a selection via the following functions:
 - Via the tab "Cockpit".
 - Via the search function in the icon bar.
 - Via favorites, SmartSearch or task assignment



4. 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: Lock the document

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.

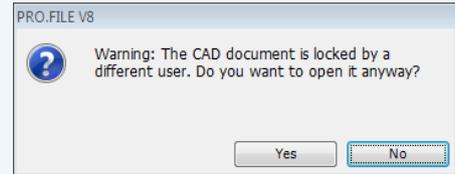
- To do so, select the function "[Lock CAD document for Modification](#)", as described in the following chapter.



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

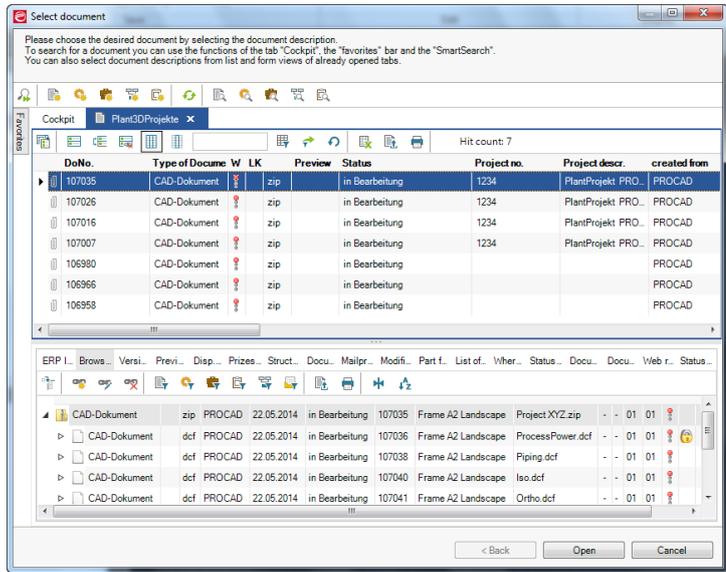
- For PRO.FILE to know which document is to be opened, the desired document description must
 - either be selected in a list view or a structure browser
 - or be displayed in a form view
- Then the button <Open> at the right bottom of the Checkout wizard has to be clicked.

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:

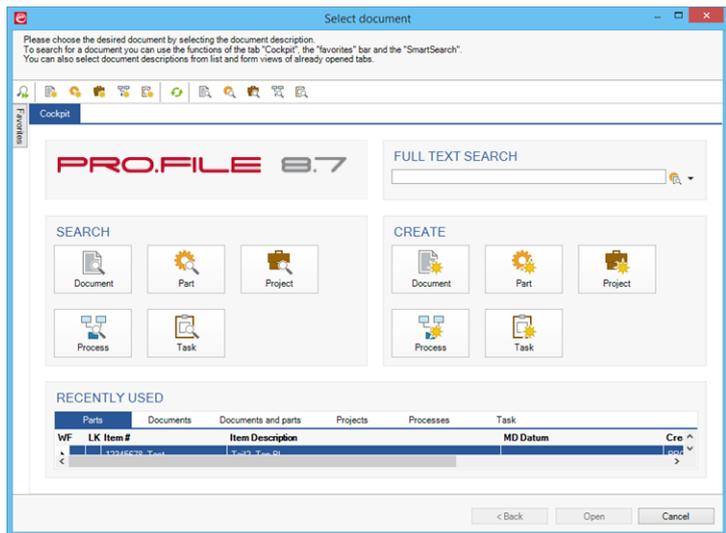
- If search results had previously been opened in a list or form view in a separate tab, you can directly access the displayed documents.

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](#)".

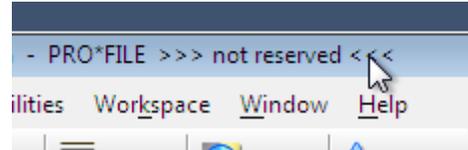
3.2 Lock CAD document for Modification: Read-only first, editing later

If you have loaded a CAD document from PRO.FILE with the functions

- "Read CAD document",
- "Read CAD document (most current released versions of references)" or
- "Read CAD document (most current visible versions of references)"

in MicroStation, this CAD document is not reserved for you.

The title bar of MicroStation displays a corresponding entry:



To avoid concurrent changes, only the user who has reserved the CAD document can save changes back to PRO.FILE.

- If a document is reserved for you, it is marked with a lock flag in PRO.FILE – other users cannot change the document, as long as the lock flag is set in PRO.FILE.



Note:

A locked drawing can be opened by other users for viewing but not for editing.

- If the document has not been locked and has been modified by another user in the meantime, you will not be able to save your own concurrent changes back to PRO.FILE, because, due to the changes of the other user, your own loaded version of the CAD document is no longer up to date.

In order to save changes to a CAD document back to PRO.FILE, you have to lock the document.



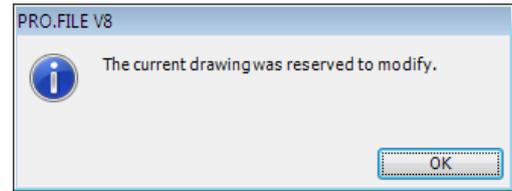
Function call:

"PRO.FILE" => "Lock CAD document for Modification"

Proceed as follows

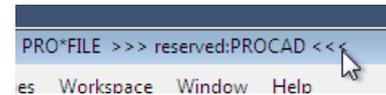
1. Select the menu "PRO.FILE " from the menu bar in MicroStation.
2. Select the function "Lock CAD document for Modification".

⇒ If it is still possible to lock the CAD document, a corresponding message is displayed:



3. Confirm this message with <OK>.

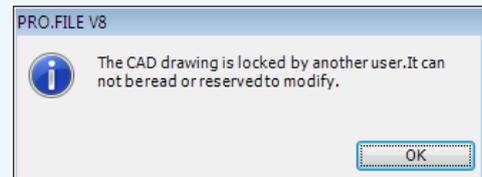
⇒ The CAD document is reserved for you and can be modified. This is displayed in the title bar of the MicroStation window.



Note:

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.

3.3 Read CAD document for Modification

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 "Read CAD document for Modification".

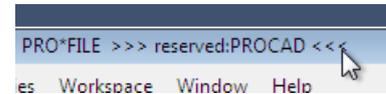


Function call:

"PRO.FILE" => "Read CAD document for Modification"

The procedure is identical to the previously described function "[Read CAD document](#)":

1. Select the "PRO.FILE" menu from the menu bar in MicroStation.
2. Select the function "Read CAD document for Modification".
 - ⇒ 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 MicroStation.
 - ⇒ The CAD document is reserved for you and can be modified. This is displayed in the title bar of MicroStation.



With the function "Save CAD document" the reservation of a CAD document in PRO.FILE is removed.

3.4 Attention: Opening of locally existing files

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

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



Attention:

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

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

A drawing exists already in the work folder?

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

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

You now have to options:

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



Note:

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

4 Saving 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 MicroStation:

- "Save CAD document" with the different situations:
 - [Saving CAD objects for the first time](#)
The CAD drawing from MicroStation is saved in PRO.FILE with a new document description.
 - [Resaving CAD objects in PRO.FILE](#)
A CAD document loaded from PRO.FILE in MicroStation is saved back to PRO.FILE after modification.
- Intermediate saving of changed CAD documents:
If you use the function "Save CAD document intermediate", the drawing remains open in Microstation after the saving process and also remains reserved in PRO.FILE.
- ["Save CAD document as a new version"](#)
If you use the function "Save CAD document as a new version" your drawing is saved as a new version in PRO.FILE. The original drawing thus remains in PRO.FILE as old version.
- ["Save cell description"](#):
PRO.FILE offers the possibility of saving cells. These are assigned a document description in PRO.FILE upon saving and can be inserted into a drawing with the function "Read CAD document".

This chapter also contains a description of the handling of references:

- ["Saving drawings with unknown references"](#).
- 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:
 - ["Read CAD document for Modification"](#): The document is reserved automatically after being loaded from PRO.FILE. Other users can no longer change the document.
 - ["Lock CAD document for Modification: Read-only first, editing later"](#): 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 saving changes to CAD documents to PRO.FILE, the following has to be noted:

- 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 MicroStation and modified, the status in the MicroStation 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 MicroStation.

- 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 CAD document as a new version](#)" keeps the original drawing as old version.
 - With the function "[Cut database reference](#)" you can cut the link of your current drawing to the PRO.FILE database. If you save the current drawing again to PRO.FILE, it is saved as a new, independent document with a new document description in PRO.FILE.

4.1 Saving CAD objects for the first time

With the function "Save CAD document" you can save parts, assemblies and drawings created in MicroStation to PRO.FILE.



Function call:
 "PRO.FILE" => "Save CAD document"

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.

Proceed as follows:

1. Select the menu "PRO.FILE" from the menu bar.
2. Select the function "Save CAD document".

⇒ 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.

4.1.1

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.

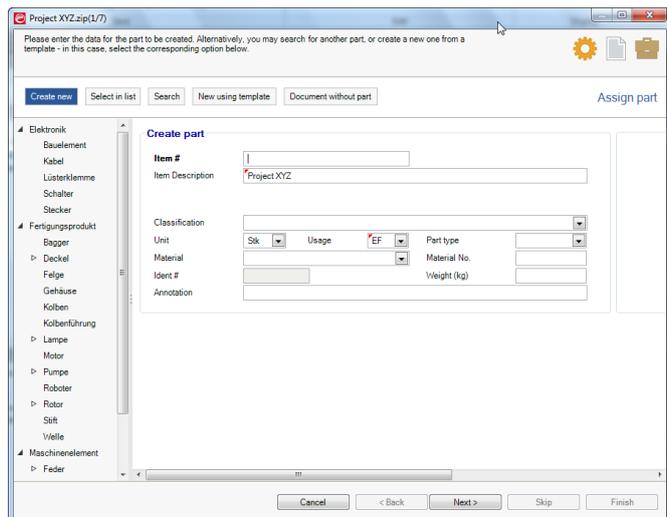


Note: Usage of PRO.FILE parts

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:

A blue rectangular button with the text "Create new" in white.

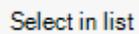
Create new

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.

A light blue rectangular button with the text "Select in list" in dark blue.

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 cockpit, the icon bar or favorites and SmartSearch to start a **selection**.
3. If the part master record desired for assignment is displayed in a list view, you can now select it. (If the desired part master record is displayed in form view, it is already selected automatically).
4. Confirm your selection with <Next>.

Search**Search**

Usage:

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

Proceeding:

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

New using template**New using template**

Usage:

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

Proceeding:

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

- ⇒ The input form for the creation of the part master record is pre-filled with the data from the selected part master record.
- 5. Make the necessary adjustments to the pre-filled data.
- 6. Once all required part data is entered, confirm the creation of the new part master record in PRO.FILE with <Next>. The new part master record is saved.

Document without part

Document without part

Usage:

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

Proceeding:

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



Attention:

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

4.1.2

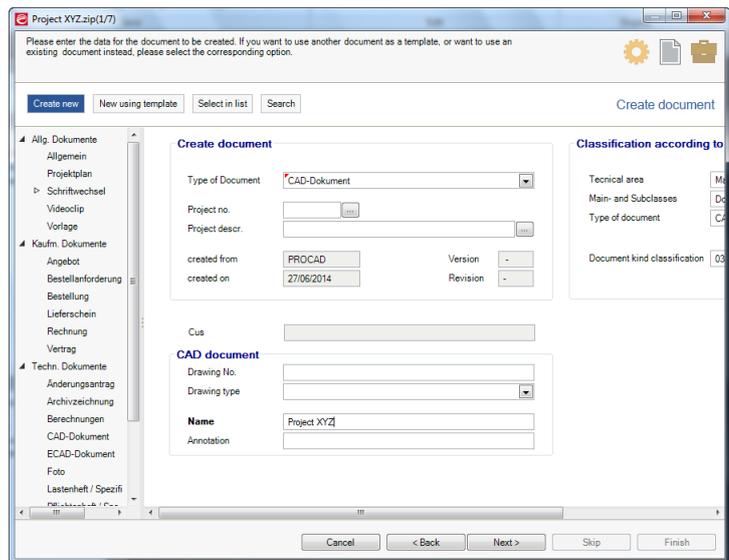
Checkin wizard Step 2: Creation of the document description in PRO.FILE

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

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

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

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



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



- Create new
- New using template

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

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

- After the finalization of your entries confirm the saving of the CAD document and the assignment to the desired part master record with <Next>.
- The CAD document is now saved in PRO.FILE.
- The Checkin wizard now continues with the options of assigning the newly created objects to a PRO.FILE project.

4.1.3

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

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

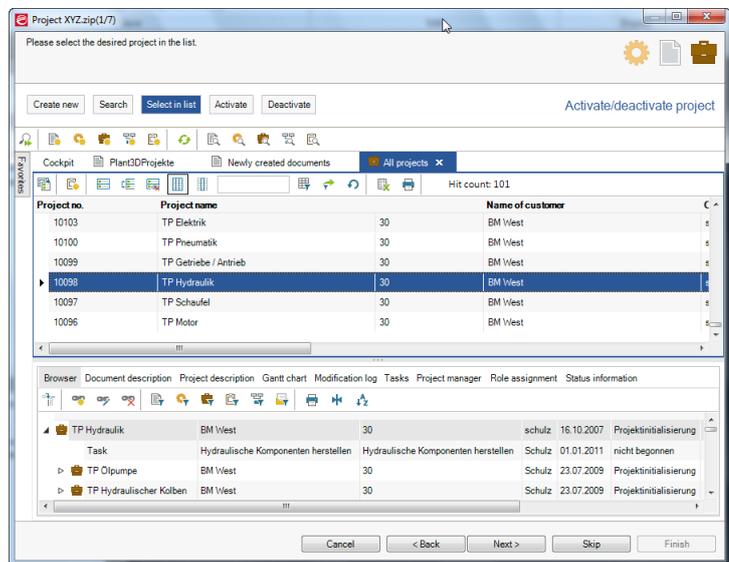


Note: Usage of PRO.FILE projects

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

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

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



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



Attention: Project must be activated

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

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

-  **Create new:**
A new project is created in PRO.FILE. The part master record and document description created in steps 1 and 2 are assigned to this new project.
-  **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.

4.2

Resaving CAD objects in PRO.FILE

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

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



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.

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

For this the function "Lock CAD document for Modification" is available, granting the user exclusive access for editing and saving. See chapter "[Lock CAD document for Modification: Read-only first, editing later](#)".



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

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

- ["Save CAD document intermediate"](#)
- ["Save CAD document as a new version"](#)

An overview of the differences between these saving functions can be found in the previous chapter "[Saving functions: Saving CAD data and changes to PRO.FILE](#)".

This chapter describes the procedure "Save CAD document" 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.
- Depending on the configuration (see configuration manual) the document is closed after saving.

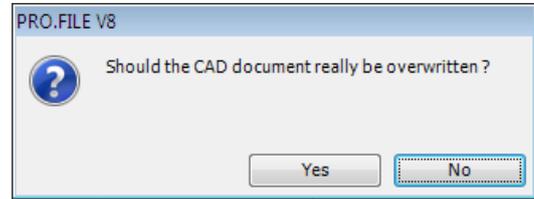


Function call:

"PRO.FILE" => "Save CAD document"

Proceed as follows:

1. Select the menu "PRO.FILE" from the menu bar in MicroStation.
 2. Select the function "Save CAD document".
- ⇒ 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.

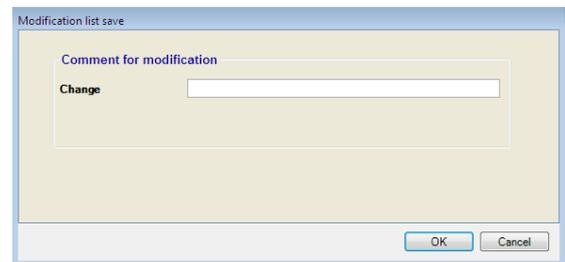


Attention:

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.



Note: Further changes after saving

Please note that you have to reserve (lock) the document again (as described in chapter "[Lock CAD document for Modification: Read-only first, editing later](#)") if you want to make further changes after having saved the document.

If you want to save the status of a CAD document in PRO.FILE without removing the reservation, you can use the function "[Save CAD document intermediate](#)" which is described in the following chapter.

4.3 Save CAD document intermediate

The function "Save CAD document intermediate" is almost identical to the function described in the chapter "[Resaving CAD objects in PRO.FILE](#)" – 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.
- You can immediately continue editing the CAD document.

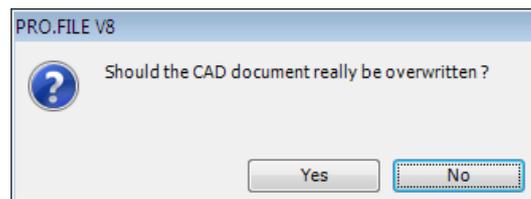


Function call:

"PRO.FILE" => "Save CAD document intermediate"

Proceed as follows:

1. Go to the integration menu "PRO.FILE" in MicroStation.
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.
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

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



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.

4.4 Save CAD document as a new version

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

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

- Only the document active in the 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 MicroStation.
- Within an assembly the reference is switched to the new version.

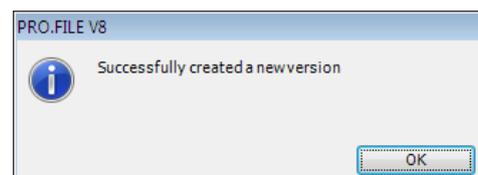


Function call from the PRO.FILE menu in MicroStation:

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

Proceed as follows:

1. Go to the integration menu "PRO.FILE" in MicroStation.
 2. Select the function "**Save as new version**" from the area "Save".
- ⇒ 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 MicroStation.





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 "[Lock CAD document for Modification: Read-only first, editing later](#)".

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



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

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

4.5 Save cell description

MicroStation allows working with cells:

- Cells can be inserted one or several times into drawings.
- Cells are not referenced, i.e. a later change to a cell has no effects on the drawing.

In order for the PRO.FILE search functions to find cells as well, cell descriptions can be saved in PRO.FILE. The cell data itself is not saved in PRO.FILE, but in the local cell library.



Note: Toolbox "Cells" in MicroStation

In order to work with cells, you need the toolbox "Cells" in MicroStation. This toolbox can be displayed under "Tools" => "Toolboxes" => "Cells".

Creating a cell in PRO.FILE

PRO.FILE gives you the option of saving cells. These are attached to a selected parts form and can be inserted in a drawing via the "Read CAD document" function. PRO.FILE only deals with the management of the cells. It does not save the geometries. The cell libraries must therefore be created so that all users have access to them.

Attention: PRO.FILE only manages cells. It does not save the geometries. The cell libraries therefore have to be deposited in such way, that all users have reading access to them.



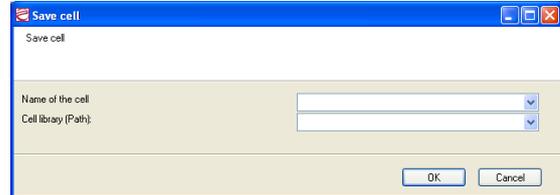
Function call:

"PRO.FILE" => "Save cell description"

Proceed as follows:

1. Create a cell in MicroStation.
2. Select the menu "PRO.FILE" from the menu bar in MicroStation.
3. Select the function "Save cell description".

⇒ The screen "Save cell" for the selection of the cell name and cell library is displayed.



4. Enter the name of the cell (e.g. "Cell1") and the corresponding cell library (including the path without extension, e.g. "C:\temp\frames"). The length of the name must not exceed 40 characters. The path only has to be entered, if it has not been set in the search paths of MicroStation.
5. Confirm with <OK>.
 - If the entry is correct, the Checkin wizard for the saving of new CAD data in PRO.FILE is started. The further proceeding is described in the chapter "[Saving CAD objects for the first time](#)", as it is identical for the saving of cell descriptions.
 - If the cell is not found by PRO.FILE, the function is cancelled and the program returns to MicroStation.

4.6 Saving drawings with unknown references

PRO.FILE offers a number of options to effectively save in PRO.FILE drawings that contain references not yet known in PRO.FILE:

- Create document interactively
- Create document automatically without confirmation
- Create document automatically and assign it to the part master of the main drawing

When saving a drawing that does not yet have references in PRO.FILE a prompt will first ask whether you wish to save the drawing anyway. If you answer with 'Yes', you will be asked to create the parts description for your main drawing.

After the references have been saved, you can make modifications or additions as necessary to the generated CAD document descriptions.



Note:

Drawings that are included into a parent drawing as references cannot be saved under a different name. That is why the parameter settings for the assignment of original names (see PRO.FILE Management Console => "Configuration" => "Parameter" => "CAD" => "Integration" => "Microstation" => "Others" => "_18_CAD_ORGNAME") are ignored for references. If you wish to assign a corresponding name to such drawings, you will need to save them in PRO.FILE prior to including them into another drawing (the reference will then be included with the correct name).

Ask for part assignment

If you use the "Ask for part assignment" method, for every unknown reference you will be asked to create a new parts form or to select an existing form. A CAD document description will then be created for every parts form.

Create part automatic

The "Create part automatic" method creates parts forms for the unknown references automatically instead of manually. The appropriate CAD document descriptions are created in the same way as with the "Use part of main drawing" method, except for the fact that each description is linked with its own parts form.



Note:

The REF_COMMENT_CAD_FIELD " parameter in the Management Console (MMC) can be used to specify which field of the parts description is to accept the comment of the reference.

Use part of main drawing

If you wish to attach all references directly to the parts form of the main drawing, use the "Use part of main drawing" method. This method causes CAD document descriptions to be created for all unknown references and to be linked directly with the parts form of the main drawing.

**Note:**

Different variables in the Management Console (MMC) can be used to specify which fields of the CAD document description are to accept the file names and the logical names of the reference (see PRO.FILE Management Console => "Configuration" => "Parameter" => "CAD" => "Integration" => "Microstation" => "Others" => "REF_COMMENT_CAD_FIELD" , "REF_FILENAME_CAD_FIELD")

Ignore references

Use the "Ignore references" function if you do not wish to save unknown references in PRO.FILE.

5 Important to note: Working with raster references

The integration PRO.FILE – MicroStation supports the work with raster references. Due to some limitations of MicroStation, certain aspects have to be observed.

Therefore, you should note the following sections before working with raster references to make sure that the used raster references are correctly saved in PRO.FILE.



Note:

MicroStation only allows the inclusion of raster references, if "<Foldername> + <Filename>" does not exceed 64 characters.

5.1 Saving of drawings with raster references

...with unknown raster references

When saving to PRO.FILE, unknown raster references are recognized. For each raster reference a document description is created and included into the document structure of the corresponding drawing.

...with already known raster references

You can include raster references already known in PRO.FILE by opening the raster reference in PRO.FILE via the commands "Edit" => "File edit" => "Check out document" on the PRO.FILE GUI.

The raster reference is then copied into the folder "%PROTMP%\<DocID>\". If you include the raster reference from this folder into your drawing, PRO.FILE will recognize the known raster reference upon saving and will not create a new part and document description for it.



Note:

If you do not open the raster reference with "Check out document" but with "Display document", the file is also copied into a sub-folder of "%PROTMP%". The sub-folder is then, however, not only named after the document ID, but instead "d+ <DocID>". If you then include this raster reference into your drawing, PRO.FILE will not recognize the raster reference as a known PRO.FILE object and will create a new part and document description.

This proceeding corresponds to the reading with subsequent "Cut database reference".

5.2 Reading of drawings with raster references

When a drawing with raster references is loaded, the embedded raster references are copied into the local work folder:

- However, MicroStation searches the embedded raster references first in the folder, from which the reference was originally embedded.
- Only if the file does not exist in this folder, it is searched in the local work folder.

It is therefore important to note: To make sure that the correct raster references are displayed, you should delete the raster reference from the original folder after the saving of an unknown reference.

5.3 Changing of raster references

When saving a drawing via the integration PRO.FILE – MicroStation, changes to raster references are ignored.

- To change raster references you have to edit the raster reference via default PRO.FILE functions.
- If you then load the drawing again from PRO.FILE, the changed raster references are displayed.



Note:

By opening the raster reference via "Edit" => "File edit" => "Check out document" the raster reference is locked. If you do not overwrite the raster reference, it remains locked in PRO.FILE, even after the saving of a drawing, in which this reference has been embedded, and has to be unlocked manually.

6 Additional functions of the integration

In addition to the basic functions, the "PRO.FILE" menu in MicroStation offers the following functions:

- [Cut database reference](#)
- [Attach reference](#)
- [Reloading reference files](#)
- [Copy document to local workdir](#)
- [Info \(list\)](#)
- [Info \(separately\)](#)
- [Fill out drawing title block](#)
- [Draw BOM](#)
- [Create/update bill of materials](#)
- [Show Workcenter](#)

| |
|-----------------------------------|
| Cut database reference |
| Attach reference |
| Load reference again |
| Copy document to local workdir |
| Info (list) |
| Info (separately) |
| Fill out title block |
| Draw BOM |
| Create / update bill of materials |
| Show Workcenter |
| Show online help |

These functions are described in detail in the following sub-chapters.

6.1 Cut database reference

All drawings loaded from PRO.FILE have a database reference when opened in MicroStation. PRO.FILE monitors these drawings and therefore recognizes them, when they are saved again in PRO.FILE.

This connection of a drawing to PRO.FILE can be cut off. This is done with the function "Cut database relation" in the PRO.FILE menu of MicroStation. This function removes the link of the drawing currently opened in MicroStation and the document description in PRO.FILE. The CAD document. The drawing is no longer treated as a PRO.FILE document but as a purely locally stored CAD file and marked accordingly in the Workcenter.

The current drawing in MicroStation can now be modified and saved as a new document in PRO.FILE with a new name and document ID.

The "old" drawing still exists in PRO.FILE. The function "Cut database reference" does not delete any files – only the local copy of the drawing loaded from PRO.FILE is disconnected from the database.



Function call in MicroStation:
 "PRO.FILE" => "Cut database reference"

Proceed as follows:

Select the "PRO.FILE" menu from the menu bar in MicroStation.

1. Select the function "Cut database reference".
2. The local document file is now disconnected from PRO.FILE.



Attention: "Cut database reference" cannot be undone

The selected CAD files are now only local and no longer have a PRO.FILE connection. Changes to the drawing are not automatically saved to PRO.FILE!

In order to save the CAD data again in PRO.FILE, you can use the saving functions of the integration described in the chapter "[Saving functions: Saving CAD data and changes to PRO.FILE](#)".

6.2

Attach reference

Files already stored in PRO.FILE can be attached as references – as it is common in MicroStation.

It is thus possible to compose a main drawing of several detail drawings and to always include the newest status of these detail drawings.

Each time the main drawing is opened from PRO.FILE, the references are displayed according to the used function.



Function call:

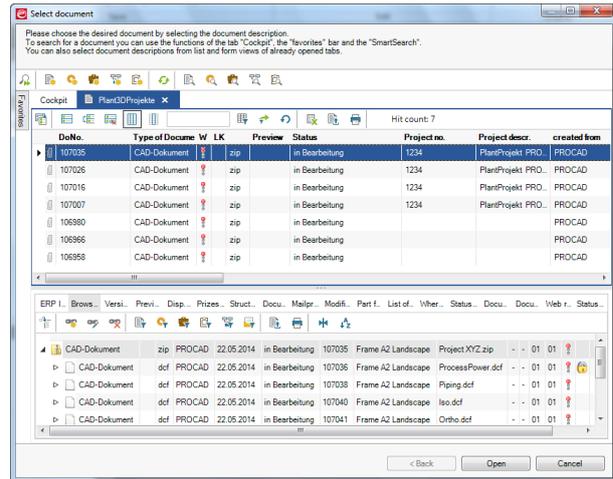
"PRO.FILE" => "Attach reference"

Proceed as follows:

1. Select the "PRO.FILE" menu from the menu bar in MicroStation.
 2. Select the function "Attach reference".
- ⇒ The PRO.FILE Checkout wizard is displayed.

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 Checkin wizard closes. The dialog screen for attaching the reference is displayed.



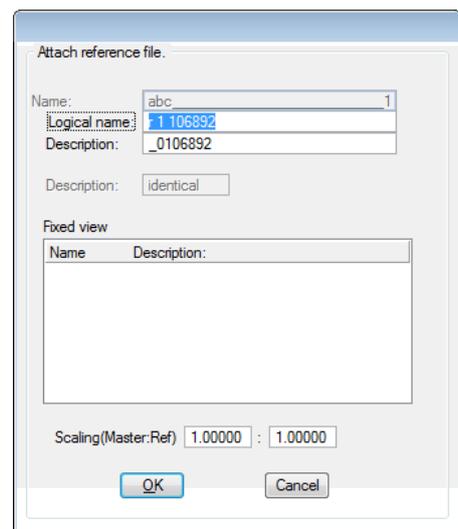
Note: Information on the Checkin wizard

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

6. The dialog "Attach reference file" is displayed. Here you can specify the logical name, the description and the scaling.

Confirm your settings with <OK>.

- ⇒ The reference is then attached to the drawing.



Please note the following in the context of references:

- After the saving of a reference, all reference files attached with "Attach reference" are summarized in a CAD document structure.
- If the drawing is loaded again, all reference files included in the CAD document structure are copied to the local hard disk. This is to make sure that MicroStation can find the reference files.



Note:

The working (moving, copying, etc.) with attached references is still made via the MicroStation menu. With the MicroStation command "Detach Reference" the reference files can be removed from the drawing in the known way.

6.3 Reloading reference files

The latest version of the attached reference files saved in PRO.FILE can be loaded into a drawing. Select the "Load reference again" function to open a window that contains all the attached reference files.



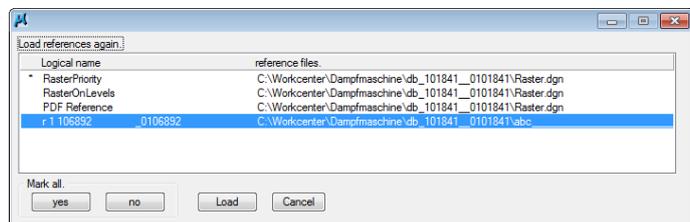
Function call:

"PRO.FILE" => "Load reference again"

Proceed as follows:

1. Select the "PRO.FILE" from the menu bar in MicroStation.
 2. Select the function "Load reference again".
- ⇒ The dialog "Load references again" is displayed, in which you can select all reference files included in the drawing for update.

3. You can select single references with a double click.
 - You can also select or unselect all references at once.



- The selected references are marked with a "*".
4. Once all desired reference files are selected, confirm with "Load".
- ⇒ The selected references are now updated. The newest, visible status of the selected reference files is loaded from PRO.FILE into the main drawing.

6.4 Copy document to local workdir

With this function a document from PRO.FILE is copied into the currently active work folder of the Workcenter.

- This way you can deposit ".jpg" or ".tif" files from PRO.FILE locally in the Workcenter and reference then from there in the MicroStation drawing.
- When the drawing is saved, PRO.FILE recognizes these referenced files as PRO.FILE objects. These links are displayed in the document structure.



Function call:

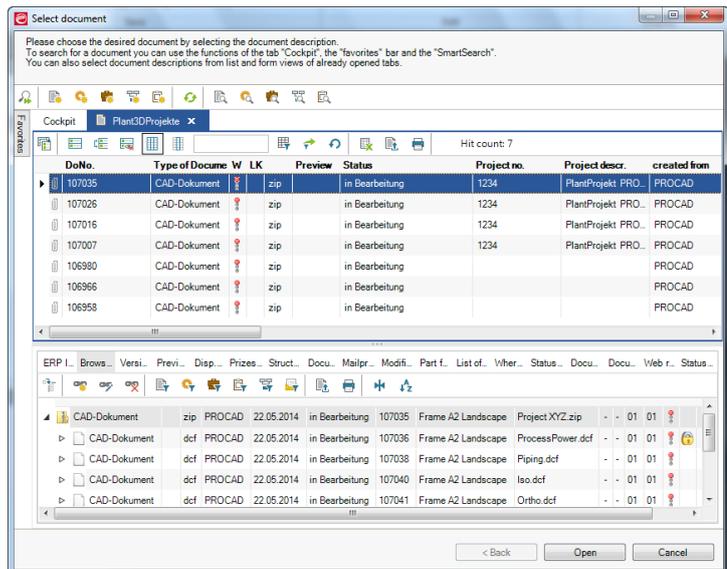
"PRO.FILE" => "Copy document to local workdir"

Proceed as follows:

1. Select the "PRO.FILE" menu from the menu bar in MicroStation.
 2. Select the function "Copy document to local workdir".
- ⇒ The Checkout wizard is displayed.

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 Checkin wizard closes and the message: "Document was copied to local workdir" is displayed for confirmation.
- ⇒ You can now insert this file via the known MicroStation functions from the local work folder into the drawing.

Further information

- On the checkout wizard can be found in the chapter "[Working with the Checkout wizard to search for CAD documents](#)".
- on the Workcenter and the activation of desired work folders can be found in the chapter "[Show Workcenter](#)".

6.5 Info (list)

This function opens the document special list. All the references contained in the opened drawing are listed. The current drawing contains the attribute "MicroStation"; PRO.FILE references contain the attribute "Reference" Those references not from PRO.FILE contain their file name as attributes.



Function call:

"PRO.FILE" => "Info (list)"

After the function has been selected, 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 documents:
 - The current drawing has the attribute "Drawing".
 - PRO.FILE references have the attribute "Reference".
 - References that do not come from PRO.FILE have the attribute "Not in PRO.FILE".

| state | Filename | up to date | reserved | path | Document | Document lock | Type of Document | Item |
|-----------------|---------------|------------|------------|-----------------------------|-----------|---------------|------------------|------|
| Drawing | Raster.dgn | no | Schulz | C:\Workcenter\Dampfma... | Schulz | 07.12.2011 | CAD-Dokument | Grui |
| Reference | Raster.dgn | no | Schulz | C:\Workcenter\Dampfma... | Schulz | 07.12.2011 | CAD-Dokument | Grui |
| Reference | Raster.dgn | no | Schulz | C:\Workcenter\Dampfma... | Schulz | 07.12.2011 | CAD-Dokument | Grui |
| Reference | Raster.dgn | no | Schulz | C:\Workcenter\Dampfma... | Schulz | 07.12.2011 | CAD-Dokument | Grui |
| Reference | abc_... | yes | not locked | C:\Workcenter\Dampfma... | Reference | | CAD-Dokument | |
| Not in PRO FILE | satellit.tif | | | y:\workspace\projects\ex... | | | | |
| Not in PRO FILE | satellit.tif | | | y:\workspace\projects\ex... | | | | |
| Not in PRO FILE | satellit.tif | | | y:\workspace\projects\ex... | | | | |
| Not in PRO FILE | 2DExample.pdf | | | Y:\Workspace\Projects\E... | | | | |
| Not in PRO FILE | satellit.tif | | | y:\workspace\projects\ex... | | | | |
| Not in PRO FILE | satellit.tif | | | y:\workspace\projects\ex... | | | | |
| Not in PRO FILE | satellit.tif | | | y:\workspace\projects\ex... | | | | |

If you have not locked all CAD documents directly upon loading, it is recommended to view information via the document list before making changes.

If the CAD document no longer has the status "unchanged", it will not be possible without problems to save the CAD document back to PRO.FILE.

6.6 Info (separately)

This function retrieves available information on an attached reference from PRO.FILE. After the command has been activated, the user is asked to select an element from the drawing.

Contrary to the function "Info (list)" the list does not display information on all objects loaded in the session, but only on the selected object.



Function call:

"PRO.FILE" => "Info (separately)"

Proceed as follows:

1. Select the "PRO.FILE" menu from the menu bar in MicroStation.
 2. Select the function "Info (separately)".
 3. Select the element of the drawing for which you want to view separate information.
- ⇒ If you have selected a referenced element, the PRO.FILE window is then displayed, showing information from PRO.FILE in the document list.

Here you find the following information:

- The data from the PRO.FILE document description.
- Information regarding the status of the currently active CAD documents:
 - The current drawing has the attribute "Drawing".
 - PRO.FILE references have the attribute "Reference".
 - References that do not come from PRO.FILE have the attribute "Not in PRO.FILE".

6.7 Fill out drawing title block

The "Fill out title block" function in the MicroStation PRO.FILE menu allows you to update the title block of a drawing.



Function call:

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

Proceed as follows

1. Select the "PRO.FILE" menu from the menu bar in MicroStation.
 - "PRO.FILE" => "Fill out title block"Select the function "Fill out title block".
 - "PRO.FILE" => "Fill out title block"Identify the elements the information of which you want to update in the drawing title block.
- ⇒ PRO.FILE updates the title block information on the drawing.

Please note the following:

- If the drawing is not yet saved in PRO.FILE, you cannot transfer data from PRO.FILE to the drawing title block. A message will inform you of this and the function call is aborted.
- If only the frame is inserted but not filled out immediately, it is filled out automatically upon saving.



Note:

To create a title block for a drawing from data in PRO.FILE, see to the configuration manual for the integration PRO.FILE – MicroStation.

The automatic filling of a drawing title block when a drawing is saved is only possible if the update group is correctly set in PRO.FILE (see configuration manual for the integration PRO.FILE – MicroStation).

6.8 Draw BOM

With the function "Draw BOM" you can place the bill of materials created with PRO.FILE on the MicroStation drawing.

In order to insert a bill of materials from PRO.FILE, it has to be created in PRO.FILE first.



Function call:

"PRO.FILE" => "Draw BOM"

Proceed as follows:

1. Select the function "**Draw BOM**" from the menu "**PRO.FILE**" in the menu bar in MicroStation.
 2. By specifying a position on the drawing you can place the bottom left point for the positioning of the bill of materials.
 3. You now have to specify the display width of the bill of materials. To do so, click on a second position right of the previously selected position.
- ⇒ The bill of materials with the information from PRO.FILE is displayed on the drawing.

If a PRO.FILE bill of materials already exists on the drawing, it can be updated with the menu function "Create/update bill of materials" with new values from PRO.FILE.

6.9 Create/update bill of materials

PRO.FILE enables parts lists to be created automatically. These parts lists contain all the elements that are used to build a module assembly.



Function call:

"PRO.FILE" => "Create/update bill of materials"

Proceed as follows

1. Select the function "Create/update bill of materials" from the menu "PRO.FILE" in the menu bar in MicroStation.
- ⇒ Your bill of materials is now created or updated with the values saved in PRO.FILE.

You can view the generated bill of materials in PRO.FILE with the function "Multi-level BOM" via the menu "View" => "Bill of materials". To do so, make sure that you have selected the corresponding part in PRO.FILE.

Furthermore, you can view the part usage and similar information on the part via the menu "View" => "Bill of materials" in PRO.FILE.



Note:

Currently every reference is set to "one" when the parts list is created or generated. The quantity can be changed manually if required. In the Management Console (MMC) (see manual "PRO.FILE Management Console", "MODIFY_EXIST_BOM_ENTRY") you can specify if existing entries are to be overwritten in the parts list or not.

References that belong to the parts description selected for the parts list are not included in the parts list. These kinds of references are created if a module assembly is saved with new references using the "Use part of main drawing" method.

References that were not attached from PRO.FILE are not included in the parts list. Otherwise all elements inserted as references are listed in the parts list.

6.10 Show Workcenter

The Workcenter helps you manage components that were retrieved from PRO.FILE and temporarily stored locally. It allows you to work with several databases at a time and helps you keep track of your work.



Function call: Starting the Workcenter:

"PRO.FILE" => "Show Workcenter"

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

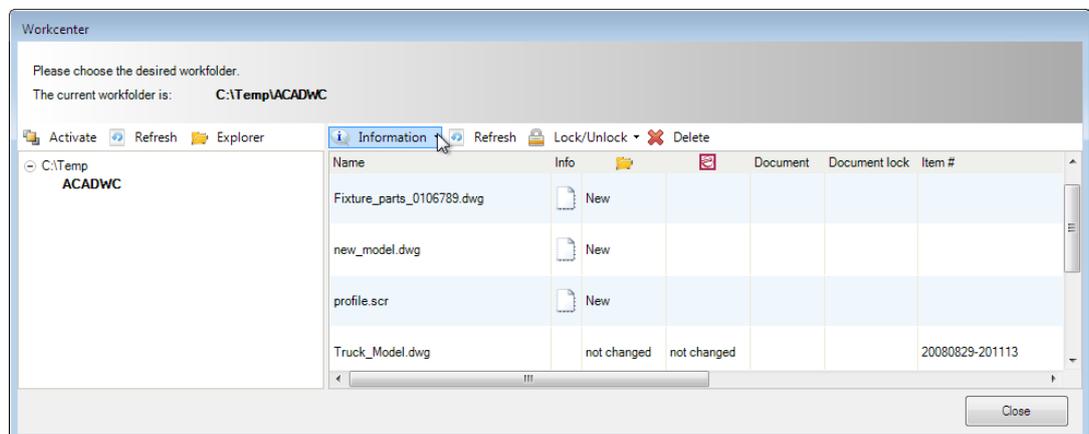


Attention: Attention when working with several work folders

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.

6.10.1 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 "[Only upon first start: Setting up the local work folder](#)".



The functions for the directory structure



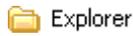
Activate

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



Refresh

The view of the directory structure is updated.



Explorer

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

The functions for the working directory



Information ▾

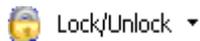
Using the drop-down menu, you can retrieve the following information for marked objects:

| | |
|------------------------|--------------------|
| Structure of the parts | Document structure |
| Part form | Document form |
| Usage of parts | Document usage |
| Bill of materials | |



Refresh

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



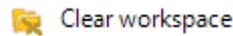
Lock/Unlock ▾

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



Delete

The marked documents are deleted from the directory. If the local status of at least one of the selected files is more recent than the one stored in PRO.FILE, a warning message will be displayed.



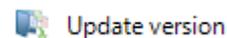
Clear workspace

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



Filter

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



Update version

Selected files can be replaced by a newer PRO.FILE version (of the same file name). If version conflicts arise, the PRO.FILE dialog for the version selection is displayed.

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).

6.10.2 Up to date or not: Display of status information

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

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

These columns may contain the following:

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

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

7 Appendix

Please note the following when working with the integration PRO.FILE – MicroStation:

Load drawings from the local working directory

If drawings are loaded from the local working directory and there are existing references, so the last modification date of the references is checked. If meanwhile the references were changed in PRO.FILE, so the user is pointed out at that, and upon request the current version is provided immediately from PRO.FILE.

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