

Functions of the Integration PRO.FILE Siemens NX (Unigraphics)

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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 – Siemens NX

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

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 Siemens NX to ERP systems.

1.1 The contents of this manual

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

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 Siemens NX. The following topics will be addressed:

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



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

When using the integration PRO.FILE – Siemens NX, 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 Siemens NX.

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

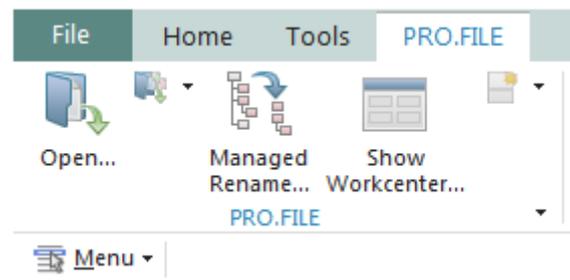
- [How to start the PRO.FILE – SIEMENS NX integration?](#)
- [What to do if the PRO.FILE menu is not displayed in Siemens NX?](#)
- [Only upon first start: Setting up the local work folder](#)
- [Where can I find the functions of the PRO.FILE integration?](#)
- [How to log in to PRO.FILE?](#)
- [A brief overview: The functions of the integration](#)

2.1 How to start the PRO.FILE – SIEMENS NX integration?

By the installation of the integration, the menu "PRO.FILE" is added to the menu bar of Siemens NX.

The image to the right shows the PRO.FILE menu directly after the start of Siemens NX.

In order to get this menu, you must start the integration via the integration icon on your desktop:



After the start of the Siemens NX integration via the  menu, PRO.FILE is displayed automatically.

2.2 What to do if the PRO.FILE menu is not displayed in Siemens NX?

If you cannot access the PRO.DILE menu after starting Siemens NX, this may have two possible reasons:

- You did not start Siemens NX via the integration icon on your desktop, but via the regular Siemens NX icon. The integration can only be started via the integration icon on your desktop:
- The integration PRO.FILE – Siemens NX is not properly installed on your computer. In this case, please contact your administrator.



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

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



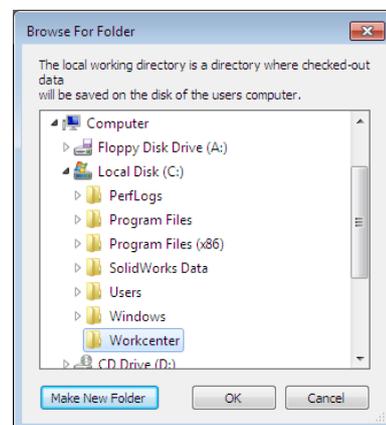
Note: Local work folder is always required

The Siemens NX 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 – Siemens NX.

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

Proceed as follows:

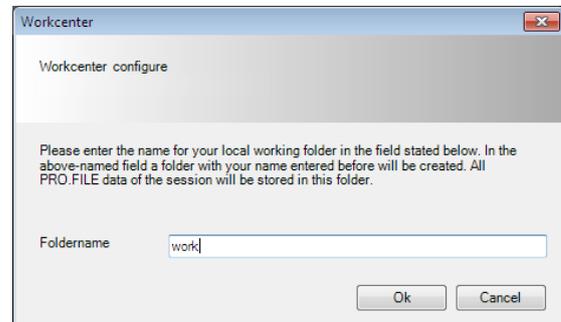
1. If no local work folder is defined when the integration is started, an input screen will prompt you to define this folder.
2. You now have to specify a "root folder". The root folder is the superior folder of the local data storage. In this folder you can later create several work folders, which are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



4. In the second step, "work folders" are now created In this root folder, which will then be used by the integration.

Consequently, you are now prompted to specify a **work folder** within the root folder:

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



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

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

2.4

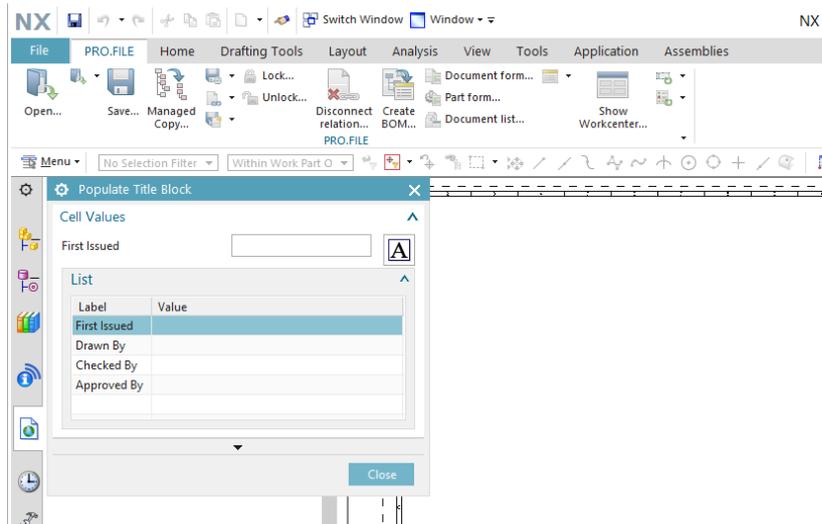
Where can I find the functions of the PRO.FILE integration?

Via the **menu "PRO.FILE"** in the Siemens NX menu bar all functions and features of the integration can be accessed.

1. Start **Siemens NX** via the start icon of the PRO.FILE integration .
2. Go the menu bar to the section "PRO.FILE".
3. Select the desired integration function from the menu.

Contents of this PRO.FILE menu depend on the CAD object that is currently active in the foreground.

The integration provides the corresponding functionalities, depending on the active CAD document type – or disables them.



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: Making the PRO.FILE menu appear in Siemens NX

In order to get this menu, you must start the integration via the integration icon  on your desktop as described in the previous chapter "[How to start the PRO.FILE – SIEMENS NX integration?](#)".

The contents of the PRO.FILE menu is enhanced by additional functions, if Siemens NX is activated in Drafting Mode.



Note:

The functions available in the drafting mode are explained in the chapter: [Additional functions of the integration](#).

2.5 How to log in to PRO.FILE?

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

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

In the login screen, please enter:

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

Confirm with <LOGIN>.

The PRO.FILE home screen is now displayed.



Note: No login required in case of "Autologin"

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

2.6

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.

Functions to open CAD objects

- **Open:**
This function opens PRO.FILE and provides the user the option to choose a CAD document and open it in Siemens NX by confirming the choice. The version constellation of the referenced components is determined by the setting of the parameter "Show Version dialog during opening" in the PRO.FILE Management Console.
See also Chapter: [Opening CAD Documents from PRO.FILE in Siemens NX](#).
- **Open as stored:** The selected document is loaded from PRO.FILE with the constellation of component versions as it as last been saved.
- **Open with latest released or newest versions:**
This open-function automatically loads the newest released versions of references of CAD objects from PRO.FILE. Objects, for which no released version exists, are loaded in the newest available version.
- **Open with newest versions:**
This open-function automatically loads the newest versions of references of CAD objects from PRO.FILE.
- **Open latest released versions:** The selected document is opened from PRO.FILE with the newest released versions of linked CAD components.
- **Open with Versions browser:**

With the version browser you can open assemblies as dynamic compositions, i.e. you can decide, which versions of the components you want to load in Siemens NX. See also Chapter: [Open with version browser](#).

- **New from seed part**

With this function you can open design templates saved in PRO.FILE. More information in the chapter [New from seed part](#).

Functions to work with CAD objects from PRO.FILE

- **Lock:**

CAD objects which were read from PRO.FILE in Siemens NX are not automatically locked for the user. To be able to modify a CAD object, the order "Lock" must be called up beforehand. By this function call, the rights of the user are checked, the topicality of the CAD object is checked and the data is locked in favor of the user, so that no other user can carry out changes. See also Chapter: [Starting your changes: "Lock" the CAD document](#).

- **Unlock:**

This function unlocks the PRO.FILE objects which were locked for processing in Siemens NX. Other users can again carry out changes to the object. Changes to an unlocked object are not automatically stored in PRO.FILE so the storage process must be carried out separately. See also Chapter: [The "Unlocking" of CAD documents](#).

Saving and Storage Functions:

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

- **Save automatic:**

Document and part descriptions for all components are created in PRO.FILE automatically without query. File names and properties can be configured to be transferred automatically into specific PRO.FILE fields. See also Chapter: [Save automatic](#).

- **Save all:**

This function saves the entire assembly with all objects contained. For objects which have not been saved within this assembly a part master will automatically be created in PRO.FILE. The file name of the object will serve as the name.

See also Chapter: [Save all: Saving complete assemblies](#).

- **Save NDF:**

This function creates a CAD drawing in the neutral TIFF format and saves it as a document. This TIFF document is automatically attached to the body of parts of the drawing.

Also see Chapter: [For drawings: Save NDF](#).

- **Managed Copy:**

Save copies of selected elements and create a new bill of materials. Please also refer to chapter: [Managed Copy](#).

- **Managed Rename:**

With this function, it is possible to change the file names of CAD documents from PRO.FILE. The references saved in PRO.FILE are updated accordingly. See also Chapter: [Managed Rename: Renaming in the structure](#).

- **Save incremental:** Via this function, only the currently active level of an assembly and the level immediately below are searched for modified documents to be saved.
- **Save incremental automatic:** This function unites the functions "Save incremental" and "Save automatic". In analogy to "Save automatic", no further user input is required during the save process.

Functions for Versioning

- **Save as new version:**

Saves the currently active CAD-object as a new version in PRO.FILE. If this function is used on a part that is built into an assembly, the references of the assembly will still be referred to the old version of the part after the new version management of PRO.FILE. See also Chapter: [Save as new version](#).

- **Replace version:**

The command "Replace version", enables an existing built version of a CAD-object to replace an object in all assemblies in which the previous version is built into. See also Chapter: [Replace version](#).

- **Managed Version:**

This function is used for the creation of versions within assembly structures. See also Chapter: [Managed Version](#).

Functions for additional files

- **Append local file:**

This function is used to add files to the structure that are not yet saved in PRO.FILE.

- **Add document:**

This function is used to add files to the structure that are already saved in PRO.FILE.

- **Detach:**

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

Information Functions

- **Show...**

Using this menu item PRO.FILE-Information about the current CAD-object can be called up and seen.

Also see Chapter: [Show: PRO.FILE Information at a glance](#).

**Database
Functions**

- **Update title block**

This function allows information on bill of materials, modification list and title to be filled in on a drawing upon opening. This requires that lists and fields are pre-configured for the template to be used.
- **Disconnect relation**

This function deletes the database link of a PRO.FILE part, a PRO.FILE drawing or an entire PRO.FILE assembly including choice of objects contained therein. The CAD objects are then treated as purely locally-saved CAD objects.

Also see Chapter: [Disconnect relation](#).
- **Insert Bill of Materials**

This function opens profile and creates a new bill of materials for the active assembly. Also see Chapter: [Create BOM](#).
- **Refresh part attributes**

With the call of this function the attributes defined in the PRO.FILE title block are updated.
- **Show Workcenter**

Open the Workcenter and manage the working directories. Please also refer to chapter: [Extra: The Workcenter](#).
- **Key characteristic features**

With these functions key characteristic features in PRO.FILE can be coupled with CAD parameters. Also see Chapter: [Extra: Connect key characteristic features in PRO.FILE with CAD – parameters](#).
- **Update thumbnails:**

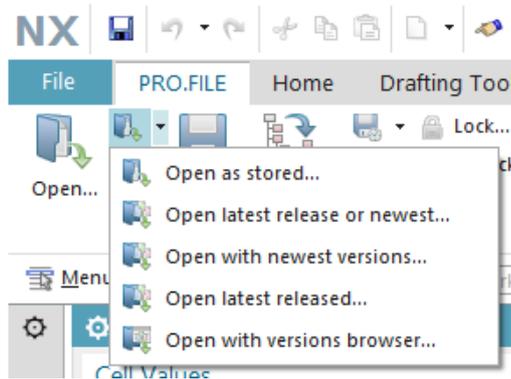
If a thumbnail is used in the PRO.FILE document list, this function can be used to update the thumbnail.

3 Opening CAD Documents from PRO.FILE in Siemens NX

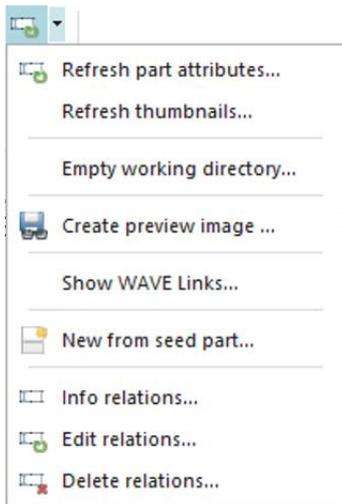
PRO.FILE manages CAD data centrally and makes them available to users with the corresponding access permissions via the command "Open".

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

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



- [Open: Load CAD Documents from PRO.FILE](#)
- [Open CAD documents with linked components](#)
- [Open with version browser](#)



- [New from seed part](#)



Attention:

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

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

**Note: PRO.FILE checks permissions**

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

3.1

Open CAD documents from PRO.FILE for editing

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

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

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

**Note:**

If components are opened from PRO.FILE in the above way, no file properties or title blocks are updated. If you want to update these, you have to use the corresponding integration function afterwards.

3.2 Open: Load CAD Documents from PRO.FILE

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

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

To open a Siemens NX document from PRO.FILE proceed as follows:

Step 1 Select the PRO.FILE function "Open"



Function call from the PRO.FILE menu in Siemens NX:

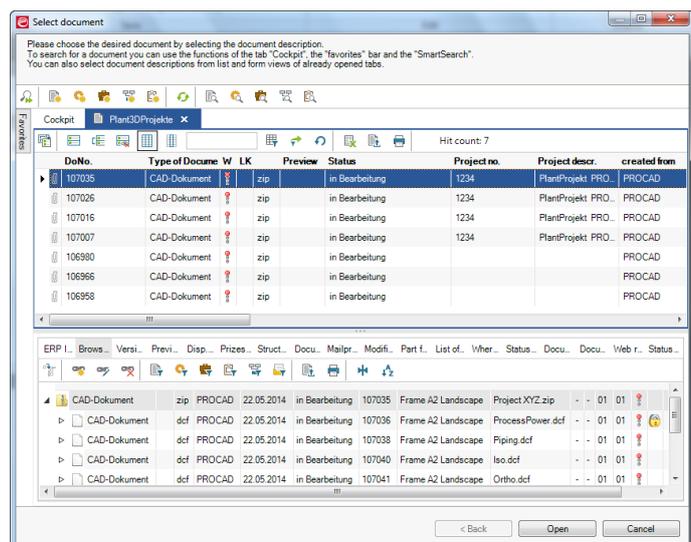
"PRO.FILE" => "Open"

1. Go to the "PRO.FILE" menu in the Siemens NX menu bar.
 2. Select the function "Open...".
- ⇒ "Open" loads the documents as they were last saved in PRO.FILE.
- ⇒ The Checkout wizard for the selection of the desired document 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.)

5. Click <Open>.
- ⇒ The Checkout wizard closes and the dialog screen for the loading type is displayed.

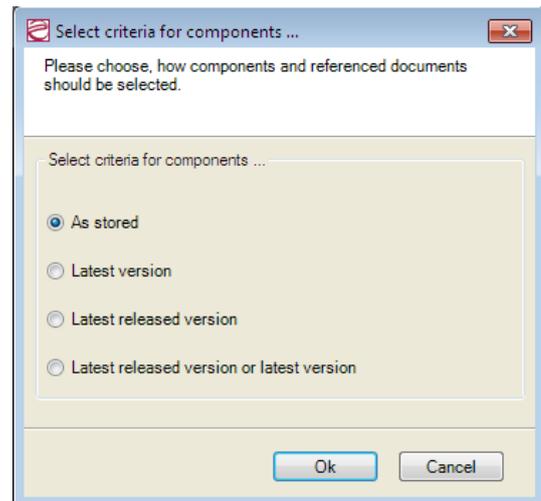
Detailed Information on the Checkout wizard can be found in the following chapter "[Working with the Checkout wizard to search for CAD documents](#)".

Step 3

Versions in the structure: How are the components opened?

When a CAD object with structure (assembly, drawing, etc.) is loaded via the "Open" function, PRO.FILE checks whether the CAD document contains components for which versions exist.

If this is the case, you can then decide how PRO.FILE is to load the assembly/drawing in question.



6. Select the desired method and confirm with <OK>.

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



Note:

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

- **Latest released version:**

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

**Attention:**

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

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

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

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

**Note: Open with version browser**

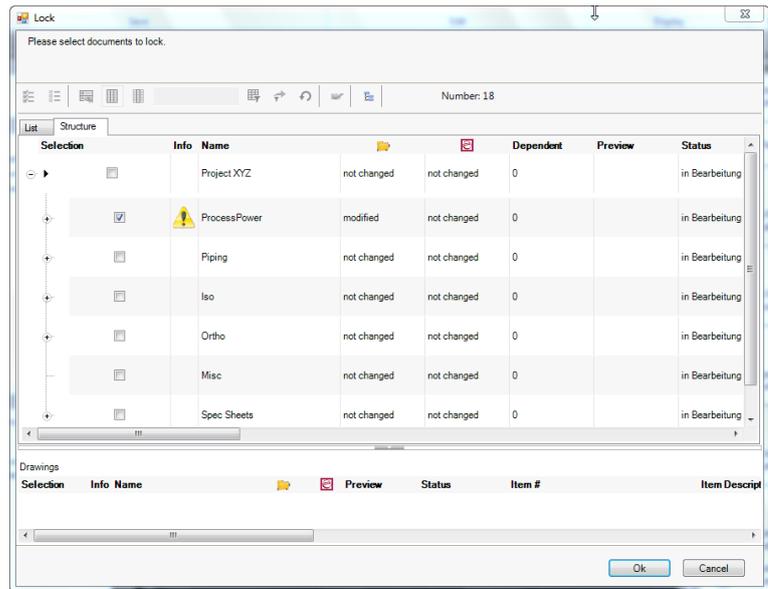
For the targeted selection of the desired version you can use the function "[Open with version browser](#)". With the version browser you can open assemblies in dynamic compositions – the user can decide for each component, which version of this component is to be loaded from PRO.FILE.

Step 4**You want to edit? Then you must lock the document(s)**

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

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

⇒ If you click on "Lock" the dialog for locking CAD documents is displayed. (Information on the functions and status indicators can be found in the chapter ["Direct information in the dialog screens"](#)).



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



⇒ The selected document is now opened with its components in Siemens NX. The "Open" procedure is thus finished.

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



Note: Why can I not lock a document?

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

This may have two reasons:

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

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

3.2.1 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.

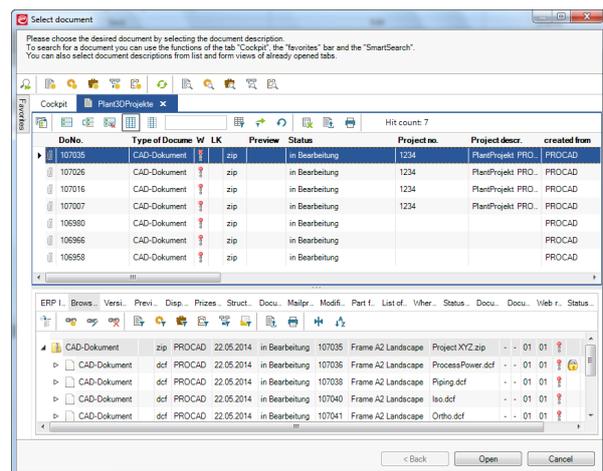
The aim of this procedure is:

- 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.

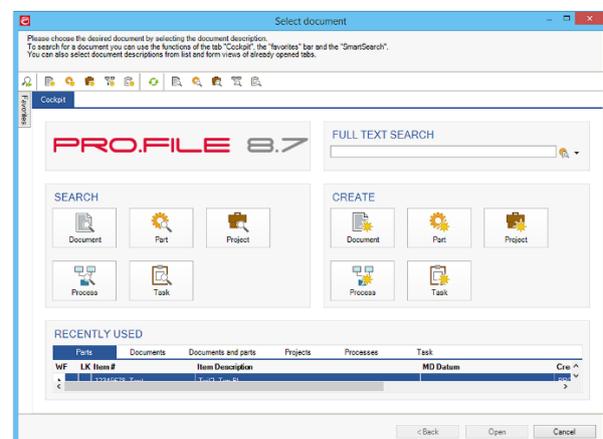
When the Checkout wizard is opened, the PRO.FILE GUI is displayed as it has been used the last time:

- 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.

Search

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

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

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

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

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

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

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

- **Search via the icon bar**

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

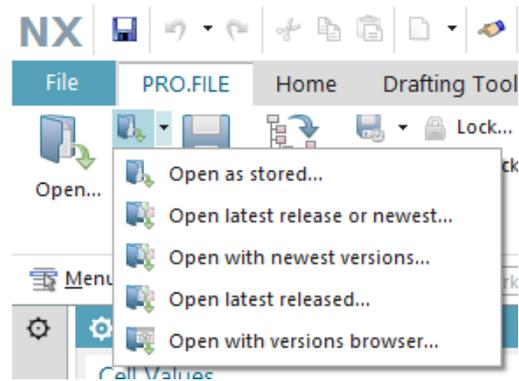
-  : Search for document descriptions to be displayed in a list.
-  : Search for parts to be displayed in a list. Documents linked to the part can be displayed in the dependent tabs "Structure" or "Document list".
-  : Search for projects to be displayed in a list. Documents linked to the project can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

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

3.3 Open CAD documents with linked components

To open files from PRO.FILE, the integration offers three options:

- Open as stored
- Open with latest released or newest versions
- Open with newest versions
- Open with latest released versions
- Open with version browser



The "version" options do not refer to the document selected in PRO.FILE for opening but **only** to the objects from PRO.FILE linked to the document to be opened.

As user you can determine, which version of the CAD documents linked in the main document are to be loaded.

This means:

- **Open as stored**

The selected document is opened from PRO.FILE as it was saved the last time. Linked CAD documents are loaded with the version status, as they were saved the last time via the PRO.FILE integration.

- **Open with latest released or newest versions**

The selected document is loaded with the newest, released versions of the linked CAD documents. The newest versions that are in a release status are loaded.

When the function "Open with newest released versions" is used for opening an assembly, PRO.FILE checks, whether the assembly contains components for which versions in a release status exist. If this is the case, the newest visible version in a release status of such a document is loaded in the session. Components for which no released version exists are loaded in the newest version.

- **Open with newest versions**

If the CAD document contains links to other CAD documents in PRO.FILE, the newest versions of these linked CAD documents are loaded.

When the function "**Open with newest versions**" is used for opening an assembly, PRO.FILE checks, whether the assembly contains components for which versions exist. If this is the case, the newest visible version of such a document is loaded in the CAD session.

- **Open with latest released versions**

The selected document is loaded with the newest released versions of the linked CAD documents. The newest version that is in a released status is loaded.

When an assembly is loaded via the function "Open with released versions", PRO.FILE checks whether the assembly contains components with versions in released statuses. If this is the case, the part version that was last transferred in the released status is loaded.

- **Open with versions browser**

With the version browser the user can decide which version of an assembly and its components is to be loaded.

The further procedure of opening via "Open with released versions" is identical to the "Open" procedure as described in the chapter "[Open: Load CAD Documents from PRO.FILE](#)".



Note: The difference between "Open with ... versions" and "Open"

Contrary to the function "Open" the function "Open with ... versions" does not display a list/dialog indicating that newer versions have been found. Furthermore, the dialog asking which method is to be used for opening is not displayed, as this question is already answered.



Note:

You can only load version for which you have viewing permissions. If the most recent version is not "visible" for you, the **newest visible version is loaded**.

3.3.1

Scenarios for the usage of "Open with released versions"

The following scenarios are to illustrate the function "Open with released versions". The following is assumed:

- You want to open an assembly from PRO.FILE via the function "Open with released versions" in the CAD system.
- Within this assembly, some parts have been versioned.

Let's take a look at different scenarios regarding the released statuses of versions and how the integration reacts in the different cases:

Scenario 1

All newest version in the assembly are in the status "Released"

- You are in the CAD system and load the assembly from PRO.FILE with the function "Version" => "Open with released versions".
- Since all objects match the selection criterion, the assembly is loaded with the newest released versions in the CAD system.

Scenario 2

The assembly contains an older version in the status "Released"

A part in the assembly has been versioned several times. The current version (e.g. number 2) is still in development. The previous version (number 1) is "released".

- You are in The CAD system and load the assembly from PRO.FILE with the function "Version" => "Open with released versions".
- The CAD system loads the assembly with part version number 1 – not number 2. This means that you do not automatically get the newest version when using this function.

Scenario 3

The assembly contains a part with a version that is not in a released status

A part in the assembly has been versioned. The part and the version are not (!) in a released status. All other version within the assembly are in the status "Released".

- You are in The CAD system and load the assembly from PRO.FILE with the function "Version" => "Open with released versions".
- PRO.FILE begins analyzing the assembly and finds this versions. As it does not match the criterion, an error message is displayed saying that no objects have been copied into your Workcenter.



Attention: Open with released versions

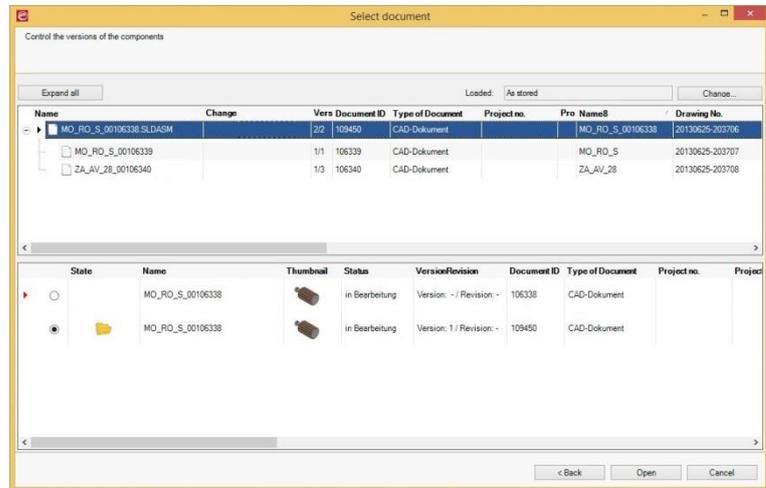
If a CAD document is loaded from PRO.FILE via the function "**Open with released versions**", and the document structure contains a document for which no released version exists, the CAD document is not loaded.

3.4 Open with version browser

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

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

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



The version browser is divided into two areas:

The document structure (top):

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

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

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

The version window (bottom):

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



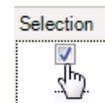
Function call from the PRO.FILE menu in Siemens NX:
 "PRO.FILE" => "Open..." => "Open with version browser"

Open a document with the version browser

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

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

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



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

Icon	Meaning
	Indicates that this version status is the currently saved one.
	Indicates an object, the version of which has been exchanged.

Icon	Meaning
	Shows a version conflict. This can occur, e.g. if a part is used in two assemblies in different versions.
	Icon of Siemens NX assemblies
	Icon of Siemens NX parts
	Indicates a softlink
	Versions reference each other causing a version cycle.

3.5

New from seed part

Via the function "New from seed part" you can create a new CAD document based on a start model saved in PRO.FILE.



Function call from the PRO.FILE menu in Siemens NX:

"PRO.FILE" => " Extra" => "New from seed part"

If you call the function "**New from Seed part**", then PRO.FILE opens a menu with predefined start models (Seed parts) which are stored in PRO.FILE. If you start a model, the document relation is disconnected. In addition, the called model is stored just newly after this, if configured so.

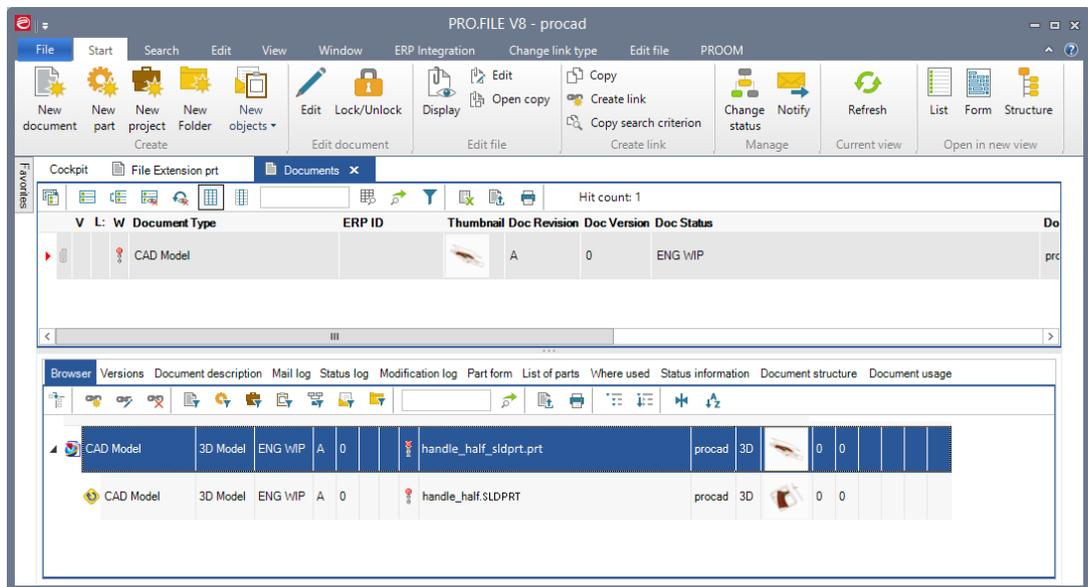
3.6 Working with third-party file formats

Via the integration PRO.FILE Siemens NX, it is possible to use components in third-party file formats in your assemblies, i.e. components from a different CAD system. These components are opened via the regular "Open" function of the integration. During the opening process, the data is then converted.

If the third-party format file is already saved in PRO.FILE, it is linked to the new document description.

Proceed as follows

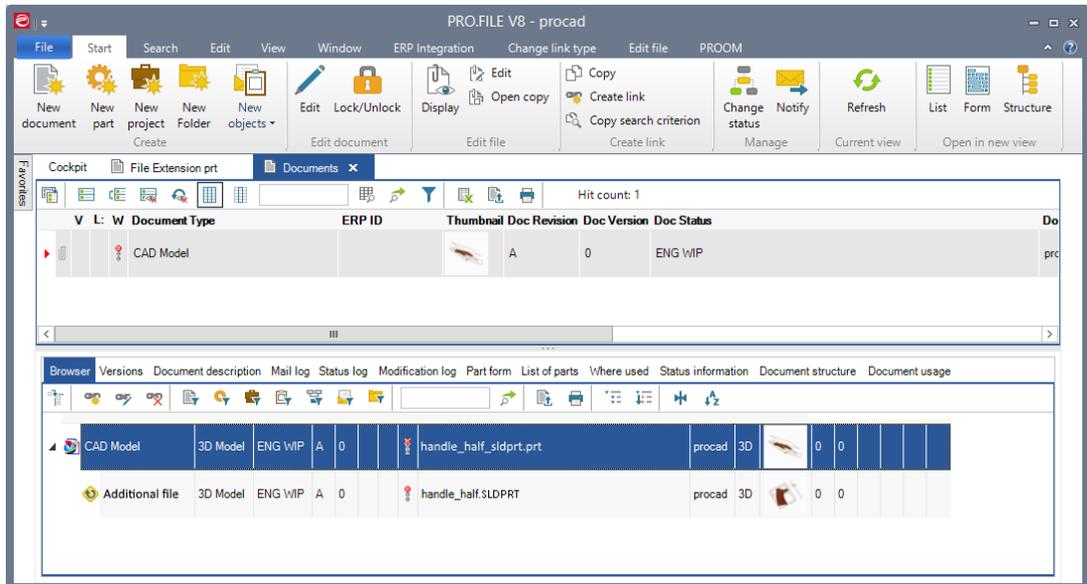
1. Open the third-party format file, e.g. a SolidWorks file, via the "Open" function of the integration.
 - ⇒ The files are then converted. The CAD system generates a file name that contains the original name.
 - ⇒ You can now use the file in your assembly.
2. Save the file locally.
3. Save the file to PRO.FILE via the integration function.
4. The dialog for the creation of a new part description is displayed (see chapter "[Saving CAD objects for the first time](#)").
 - ⇒ A new document description is created in PRO.FILE. The structure shows that the original file s



If the third-party format file only exists locally, it is stored in PRO.FILE as additional file during the saving process.

Proceed as follows

1. Open the third-party format file, e.g. a SolidWorks file, via the opening function of the CAD system.
 - ⇒ The data is then converted. The CAD system generates a file name containing the original name.
 - ⇒ You can now use the file.
2. Save the file locally.
3. Then save the file via the integration function.
4. The dialog for the creation of a new part description is displayed (see chapter "[Saving CAD objects for the first time](#)").
5. In the next step, the dialog for saving a new document description is displayed.
 - ⇒ The new document description is saved in PRO.FILE.
 - ⇒ The original file is saved as additional file.



4 Lock/Unlock: Who can change when?

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

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

For detailed information see the following sub-chapters:

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

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

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

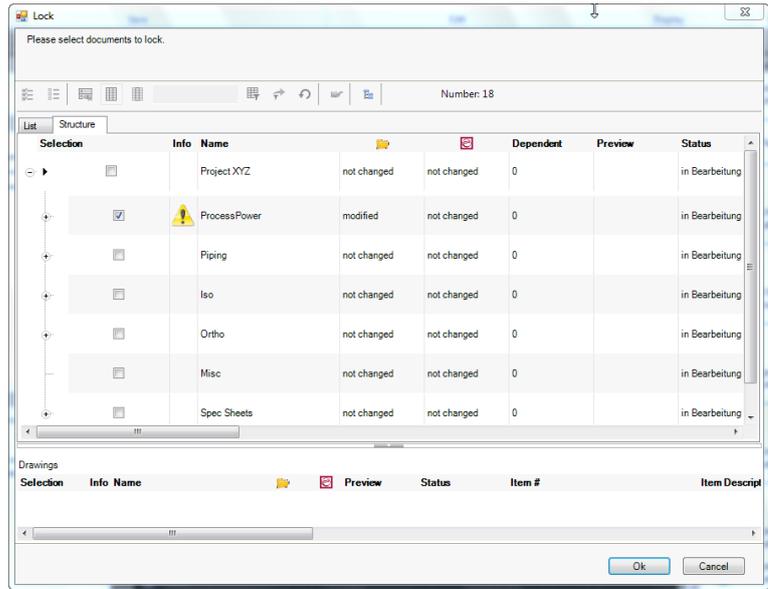
For detailed information please see the manual "CAD design supported by PRO.FILE".

Dynamic lock dialog

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

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

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



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

4.1 Starting your changes: "Lock" the CAD document

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



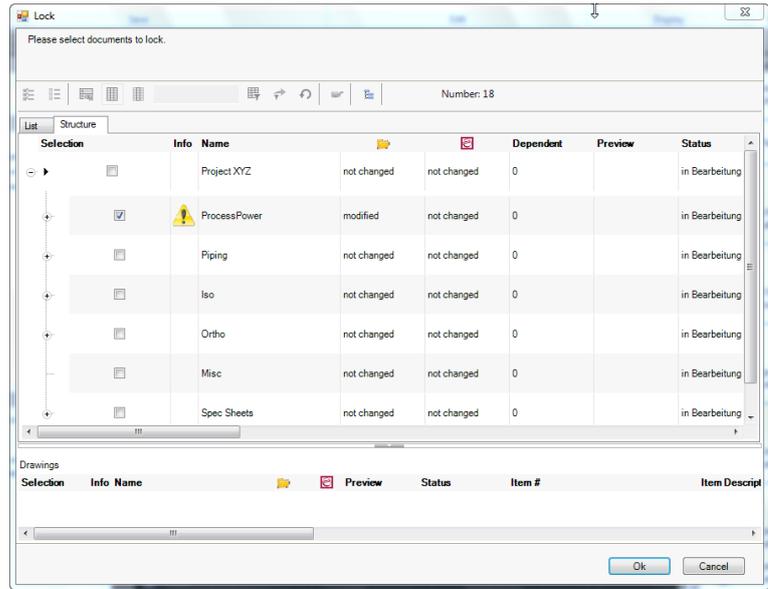
Function call from the PRO.FILE menu in Siemens NX:

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

Lock a CAD document manually

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

⇒ The dialog for locking the loaded CAD documents is displayed. (Information on the functions and status information can be found in the chapter "[Direct information in the dialog screens](#)").



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

- whether the user has the permission to edit the document.
 - whether the active documents are up to date and have not been modified by a different user since their opening.
 - whether the active documents does not already have a lock flag.
- ⇒ If any of these checks returns a negative result, the document cannot be locked!

4. Select all document you wish to lock by setting the checkmark in the first column.
5. Confirm your selections with <OK>.



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

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



Attention: Changes in the team

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

4.2 The "Unlocking" of CAD documents

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



Note:

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



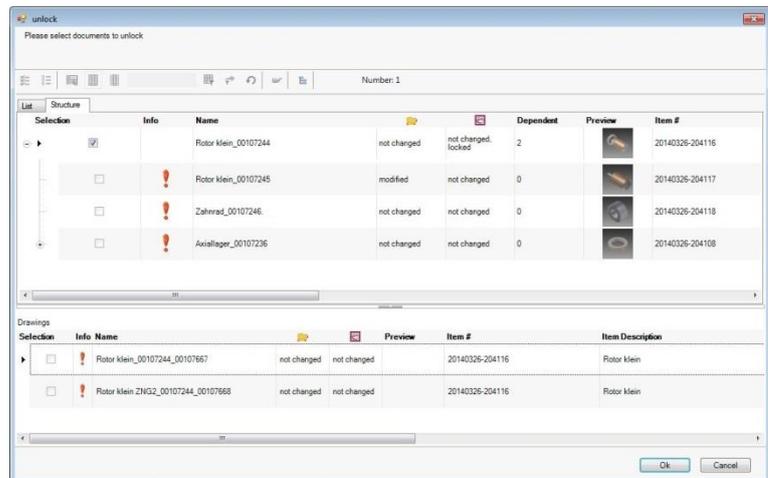
Function call from the PRO.FILE menu in Siemens NX:

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

To unlock a document proceed as follows:

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

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



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



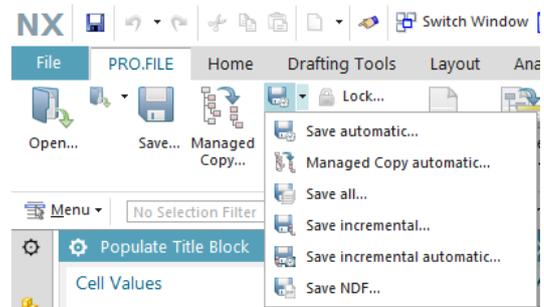
⇒ The lock flag for the selected CAD document is now removed.

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

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

"Save" with the options:

- [Saving CAD objects for the first time](#)
- [Save: Saving changed CAD documents](#)
- [Save automatic](#)
- [Save all: Saving complete assemblies](#)
- [Managed Copy](#)
- [Managed Rename: Renaming in the structure](#)



When a drawing is opened you also find the function:

- [For drawings: Save NDF](#)

The menu version also contains the function:

- [Save as new version](#)

This variety of functions is based on one fundamental behavior:

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

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

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

Therefore, the description is divided into two chapters:

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



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

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

5.1 Saving CAD objects for the first time

With the use of the function "Save ", Siemens NX objects are saved to PRO.FILE.

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

The following procedure is **prerequisite for saving**:

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

If you want to save CAD documents from Siemens NX to PRO.FILE, use the menu entry "Save" from the "PRO.FILE" menu.



Note:

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



Function call from the PRO.FILE menu in Siemens NX:

"PRO.FILE" => "Save"

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

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

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

These steps are described in the following sub-chapters.

5.1.1

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

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

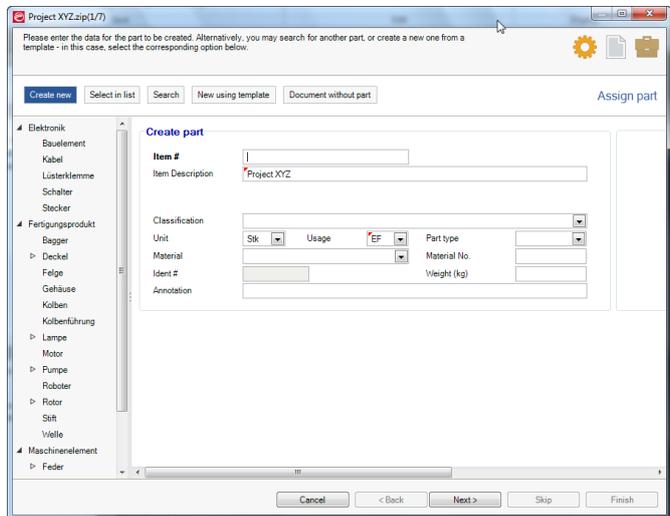


Note: Usage of PRO.FILE parts

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

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

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



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



Create new:

Usage:

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

Proceeding:

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

Select in list Select in list:

Usage:

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

Proceeding:

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

Search Search:

Usage:

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

Proceeding:

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

New using template New using template:

Usage:

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

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

Proceeding:

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

Document without part

Document without part:

Usage:

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

Proceeding:

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



Attention:

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

5.1.2

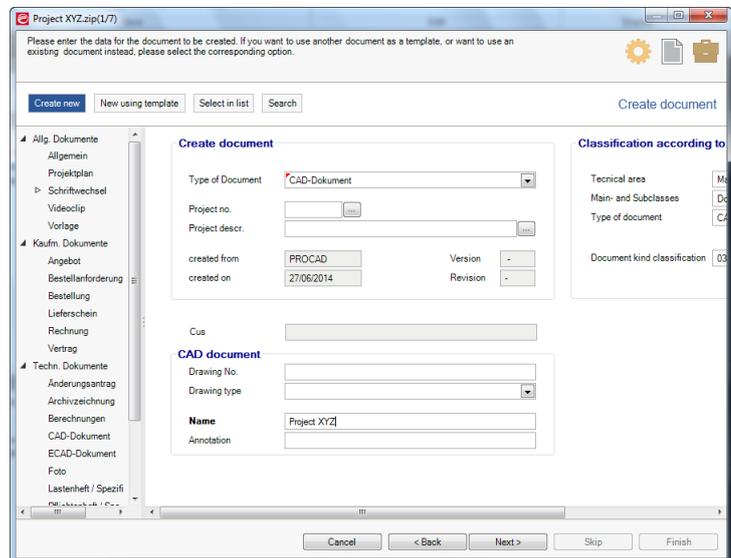
Checkin wizard Step 2: Creation of the document description in PRO.FILE

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

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.

5.1.3

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

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

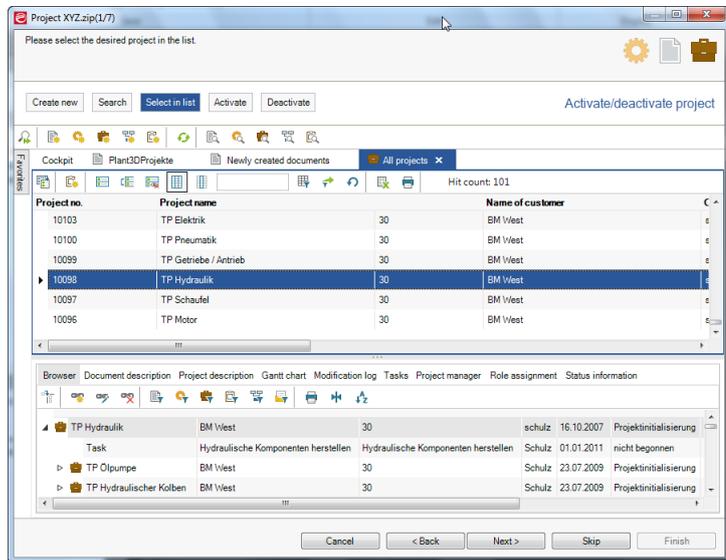


Note: Usage of PRO.FILE projects

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

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

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



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



**Attention: Project must be activated**

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

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

- **Create new** 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.

5.2 Save: Saving changed CAD documents

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

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

Before the saving process is initialized, PRO.FILE checks whether the user is entitled to change the object concerned in PRO.FILE. It is also checked whether the user works with a current copy of the corresponding PRO.FILE Object.



Attention: Only documents that have been locked can be saved

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

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

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

- Save (as described in the following).
- [Save automatic](#)
- [Save all: Saving complete assemblies](#)
- [For drawings: Save NDF](#)
- [Managed Copy](#)
- [Managed Rename: Renaming in the structure](#)
- [Save as new version](#)

This chapter now describes the procedure for saving changed CAD documents



Function call from the PRO.FILE menu in Siemens NX:

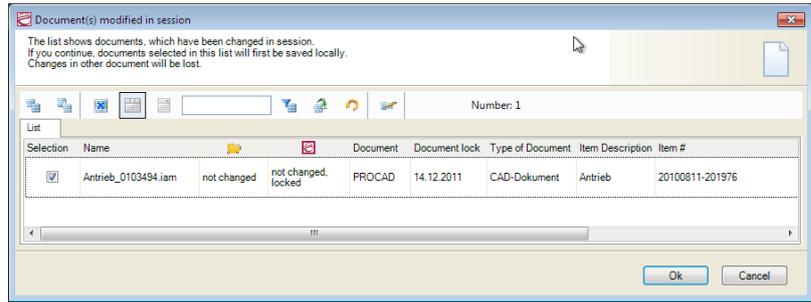
"PRO.FILE" => "Save"

Proceed as follows

1. Go to the integration menu "PRO.FILE" in Siemens NX.
2. Select the function "Save".

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

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



⇒ The dialog displays a list with all changed CAD documents from the current Siemens NX session. (Information on the functions and status information can be found in the chapter "[Direct information in the dialog screens](#)").

⇒ For assemblies, the structure is analyzed for changed CAD documents and the list of all documents of this assembly is preselected.

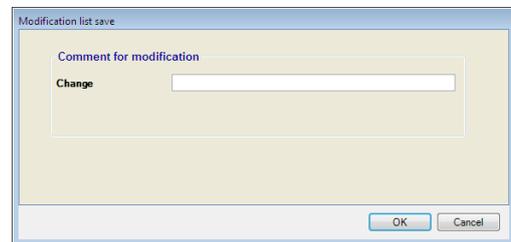
⇒ For this list the access permissions for saving the changes of the user are checked. (If the CAD document had been locked before for editing, this prerequisite is fulfilled.)

3. Select all documents you want to save in PRO.FILE. To do so, activate the checkboxes for the desired documents.



4. Confirm your selection with <OK>.

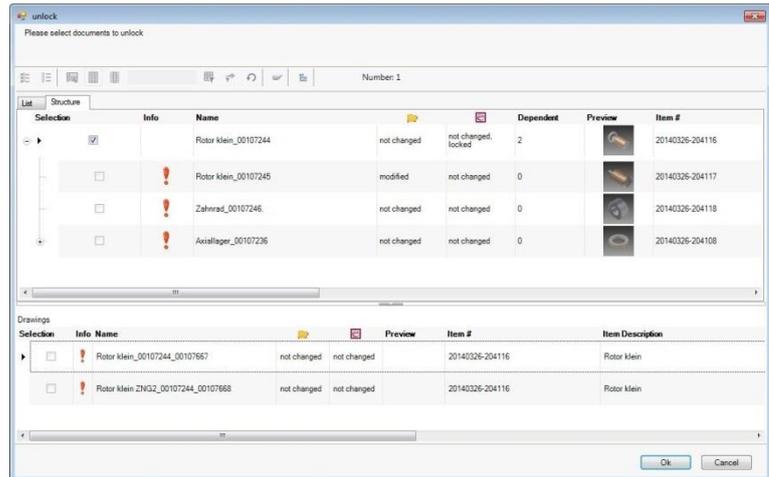
5. Depending on the configuration and PRO.FILE status, you now have to enter a modification comment. Enter the comment information into the fields on the dialog screen.



6. Confirm your modification comment with <OK>.

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

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



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

7. To make the documents available again for other users, select the documents in the list. To do so, activate the checkboxes for the desired documents.
8. Confirm your selection with <OK>.



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

5.3 Save automatic

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

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

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

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

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

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

"Save automatic" for complete assemblies

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

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



Function call from the PRO.FILE menu in Siemens NX:

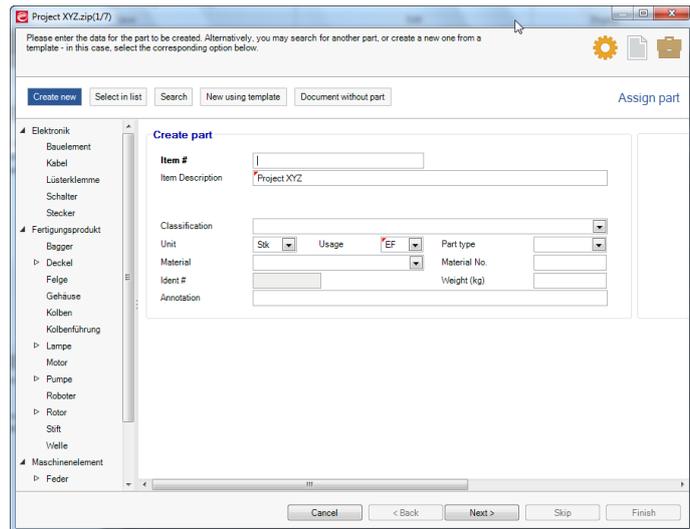
"PRO.FILE" => "Save..." => "Save automatic"

Proceed as follows

1. Select the "PRO.FILE" menu from the menu bar in Siemens NX.
 2. Select the function "Save automatically" from the menu area "Save...".
- ⇒ For the first document that is unknown to PRO.FILE, the normal saving process is started.

The following is displayed:

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



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



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

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

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

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

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

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

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

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

5.4

Save all: Saving complete assemblies

If an assembly is open within a Siemens NX session which should include saving all group parts contained in PRO.FILE, then the entire assembly may be saved in PRO.FILE using the command "**Save All**".

**Function call from the PRO.FILE menu in Siemens NX:**

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

By using the command "**Save All**" the entire assembly will be saved in PRO.FILE, including all parts contained therein.

This function corresponds to a normal saving process for all opened assemblies/parts.

5.5

For drawings: Save NDF

The integration of PRO.FILE-Siemens NX offers the user the option to convert a Siemens NX drawing into the neutral data format, e.g. TIFF, and to save it as a document. By calling up the function "Save NDF" the version of the CAD drawing is created in TIFF.

This neutral document is then automatically linked to the part master record of the drawing.

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

This menu item is only available when the drawing mode (Drafting) is activated.



Note: Save NDF only with "Format generators"

The function "Save NDF" is not a standard feature in PRO.FILE. This means that you need a license to use this function. Furthermore, specific installations (converter) and configurations may be required on your system.



Function call from the PRO.FILE menu in Siemens NX:

"PRO.FILE" => "Save NDF"

Create a neutral data format document

1. You have opened a drawing and wish to document the current drawing status.
2. Select the "PRO.FILE" menu from the menu bar in Siemens NX.
3. Select the function "Save NDF" from the menu area "Save"..
 - ⇒ A dialog is displayed asking, whether you wish to create the neutral format.
4. Confirm with <Yes>.
 - ⇒ The neutral format is now generated.
 - ⇒ Depending on the configuration the action is performed without further interaction.
 - ⇒ The created NDF document is automatically saved in PRO.FILE and linked with the document description of the drawing.
 - ⇒ The creation of the neutral target format for the drawing is now finished.



Note: Client-side NDF and server-side NDF

If, apart from the manual creation of NDF documents, you also want to use the automatic server-side creation of NDF documents via workflow status, please make sure to configure the NDF documents according to your requirements (overwrite or version).

**Attention:**

This menu item is only available when the drawing mode (Drafting) is activated.

Change management via NDF generation

With the configuration of the NDF generation via the format generators you can control the behavior of PRO.FILE regarding the handling of TIFF documents. You can configure that an existing TIFF document is not overwritten but versioned when the function "Save NDF" is used. This way, changes can actively be documented by the design engineer.

5.6 Managed Copy

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

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

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

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

5.6.1 Exchanged or not: What must be observed strictly?

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

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

**Attention: Result of Managed Copy**

The result of "Managed Copy" depends on the CAD documents opened in the Siemens NX session and the CAD document selected for "Managed Copy"! If higher-level assemblies are opened in the Siemens NX session, a subassembly /CAD part, for which the function "Managed Copy" has been selected, is **exchanged** in these assemblies!

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

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

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

For these requirements there are different approaches, which are described in the following.

5.6.2

Requirement 1: Create an independent copy of a model

The **requirement** is:

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

To do this, the following approach **must strictly be observed**:

Approach 1

Only the model you want to copy is loaded in Siemens NX

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



Attention: higher-level assemblies are not be opened

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

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

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

The requirement is:

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

To do this, the following approach **must strictly be observed**:

Approach 2

Exchange the model in several higher-level assemblies

1. Open **all** higher-level assemblies in which you want to exchange the model to copy in the Siemens NX session.
2. Open the model to be copied via the "Managed Copy" function in the Siemens NX session.



Attention: higher-level assembly opened

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

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



Attention: Higher-level assemblies are not saved automatically

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

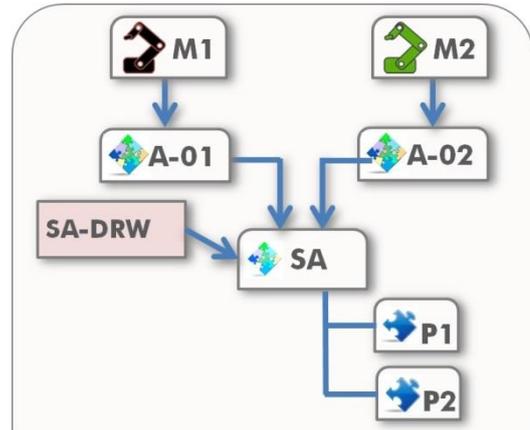
Case study for approach 2A

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

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

Situation:

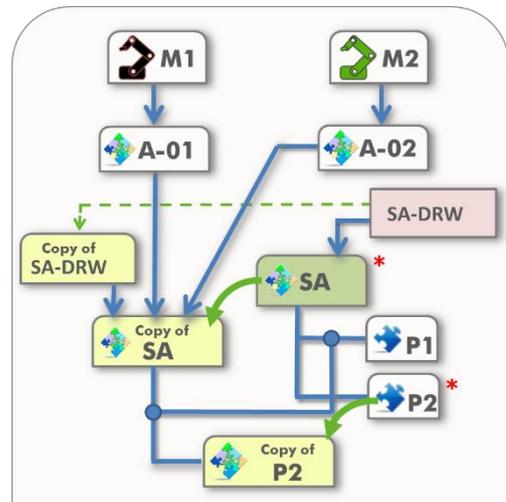
- 2 assemblies ("A-01" and A-02") are loaded in Siemens NX.
- Assembly "A-01" is part of the machine "M1", assembly "A-02" is part of the machine "M2".
- The subassembly "SA" is installed in both assemblies.
- There is the drawing "SA-DRW" for the subassembly.



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

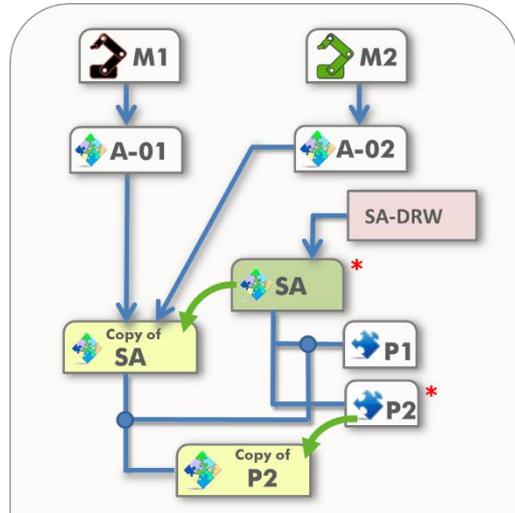
Result IF the drawing was also selected for Managed Copy:

- Due to the fact that "SA" as an active document has been selected to copy via "Managed Copy" and both assemblies "A-01" and "A-02" are loaded in Siemens NX, the references in both assemblies are exchanged by Siemens NX.
- In both assemblies the copied subassembly "Copy of SA" is installed.
- A copy is created of the drawing SA-DRW. "Copy of SA-DRW" refers to "Copy of SA".
- A copy of CAD part "P2" is created, to which the "Copy of SA" refers.
- Like "SA", "Copy of SA" refers to the CAD part "P1".



Result IF the drawing was NOT selected for Managed Copy and NOT loaded in Siemens NX:

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



Behavior of the drawing

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

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

Approach 2B

Exchange a model in a specific assembly

1. Open and activate the higher-level assembly, in which the model (subassembly/CAD part) should be exchanged via "Managed Copy", in the CAD session.
 - ⇒ Additional higher-level assemblies, in which the model to copy is referred but should not be exchanged, can remain open. In higher-level assemblies, references are only exchanged if the Managed Copy function is run from within the subassembly.
2. Call the PRO.FILE function "Managed Copy" up, as described in the following chapter ["Managed Copy"](#).
3. In the wizard of "Managed Copy", select **only** the model to be copied and the contained subassemblies and parts that are to be copied as well.
 - It is important that you do **not** select the higher-level assembly, which is shown as the top element (top element in the structure).
 - ⇒ The created copy of the model is saved in PRO.FILE.
 - ⇒ The created copy of the model is exchanged in the assembly for which the function "Managed Copy" has been called up.

**Note: Using this approach, only the references are exchanged**

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

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

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

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

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

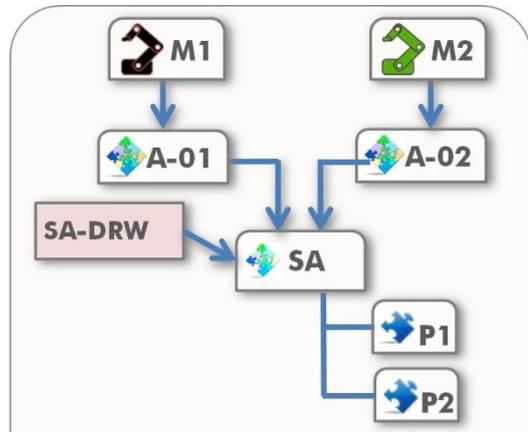
Case study for approach 2B

Replace "SA" only in "A-02"

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

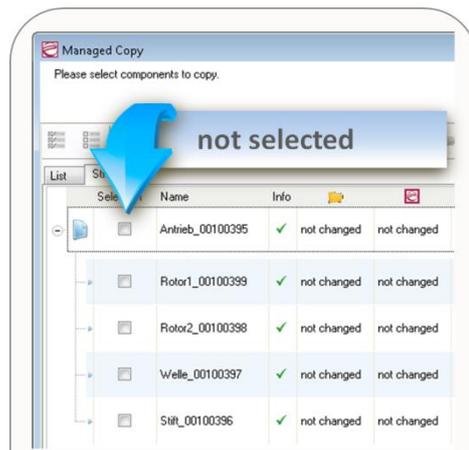
Situation:

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



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

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



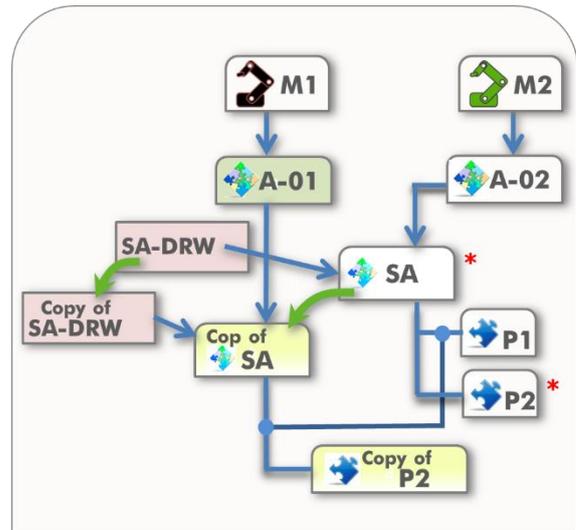
When the sub-assembly is selected, the drawing is automatically selected as well

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

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

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

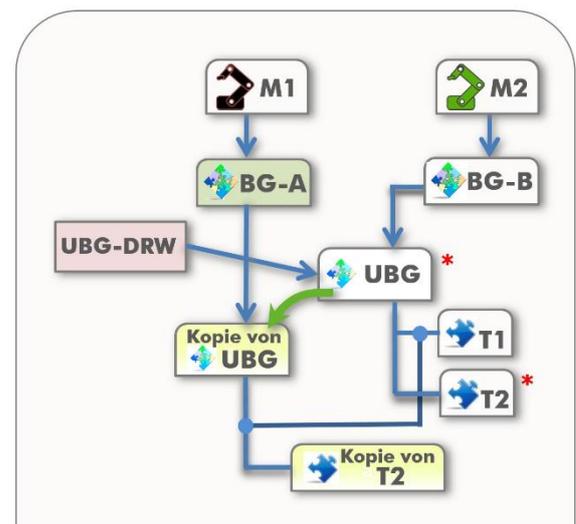
- The sub-assembly "SA" is copied with "Managed Copy".
- The integration explicitly exchanges the references in assembly A-01, so that these references point to the copied sub-assembly "Copy of SA".



- The drawing "SA-DRW" is copied. The drawing "SA-DRW" refers to the sub-assembly SA, the drawing "Copy of SA-DRW" refers to the copy of "SA".
- The assembly "A-02" still refers to sub-assembly "SA".
- A copy is created of "P2", which is referenced by "Copy of SA".
- No copy is created of part "P1". "Copy of SA" refers to the part "P1".

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

- The sub-assembly "SA" is copied with "Managed Copy".
- The integration explicitly exchanges the references in assembly A-01, so that these references point to the copied sub-assembly "Copy of SA".
- The assembly "A-02" still refers to sub-assembly "SA".
- The drawing "SA-DRW" refers to "SA". "Copy of SA" is not referenced by a drawing.



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

5.6.4 How is the function "Managed Copy" executed?



Attention: Result of Managed Copy

The result of "Managed Copy" depends on the CAD documents opened in the Siemens NX session and which CAD document is selected for "Managed Copy"! See the previous chapter: ["Exchanged or not: What must be observed strictly?"](#).



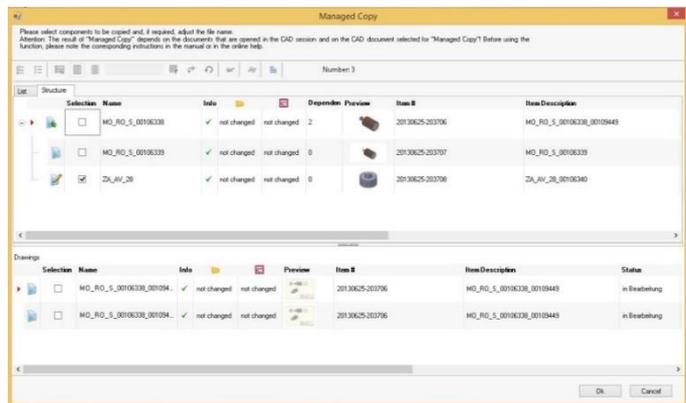
Function call out of the PRO.FILE menu in Siemens NX:

"PRO.FILE" => "Managed Copy..."

Proceed as follows:

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

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



⇒ In a second step the substructure is expanded by the related drawings. This "special provision" is required because the drawings are listed above the model depending on references.

⇒ In the wizard of "Managed Copy" the detected tree structure, enhanced by drawings, is shown, so that the documents to copy with their dependent data (nodes) can be selected.

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

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

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

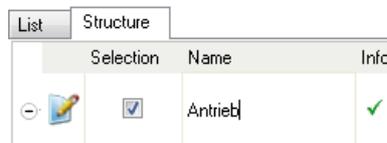
- Select:** Select all components which you want to save as a new copy in PRO.FILE. Therefore activate the checkbox in the listed CAD documents as shown on the right.



Note: Exchange of components in assemblies

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

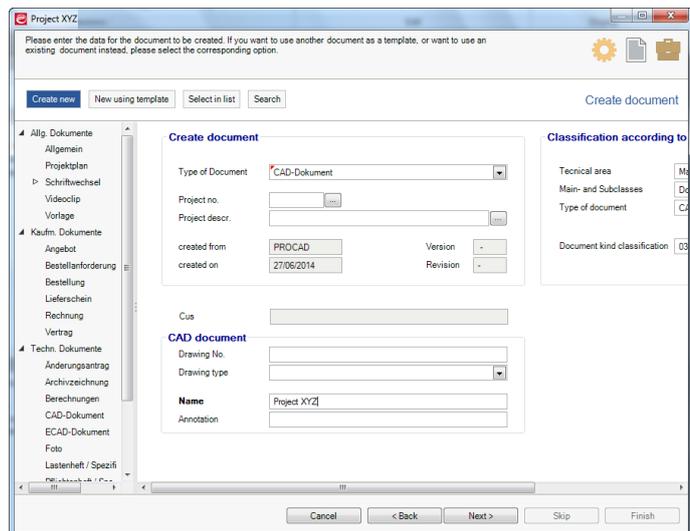
- Edit file name:** You can edit and adjust the file names directly in the list, by a click on the column name.



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

⇒ Therefore appears:

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



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



Note:

Also in assemblies that are not explicitly selected for "Managed Copy" the reference to the subassemblies/CAD parts is exchanged locally (in the working directory and the interface). In a second step, these changes to existing PRO.FILE assemblies are offered to save in PRO.FILE via "saving of changes". Thereby the user can choose which local changes he actually wants to take in PRO.FILE. The integration also checks the user and status authorization during the saving process.

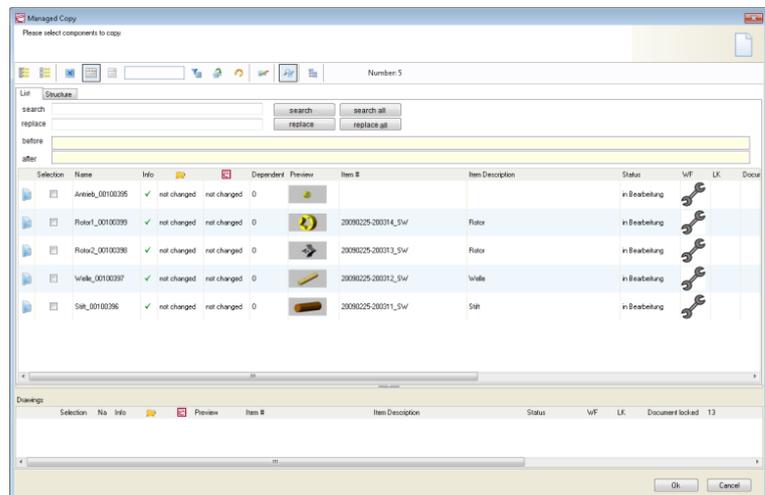
⇒ Finally a bill of materials is derived for the "cloned" assembly. The process "Managed Copy" is thus finished.

5.6.5 Search and replace with Managed Copy

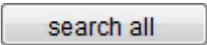
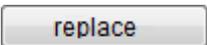
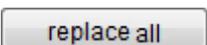
The file names of assemblies and parts can be edited during the execution of "Managed Copy". This is also possible via the function "search and exchange".

To search and exchange, proceed as follows:

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



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

-  searches and selects the next hits in the list
-  searches and selects all different hits in the list
-  replace the next hit
-  replace all hits in the list

A preview for the editing of the file name is shown in the fields "before" and "after" Search and exchanged in finished by executing the function "Managed Copy".

5.7

Managed Rename: Renaming in the structure

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

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

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



Note: Only available without active CAD document

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

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

Please note the following for this process:

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



Attention: "Rename" is a modification

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

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

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



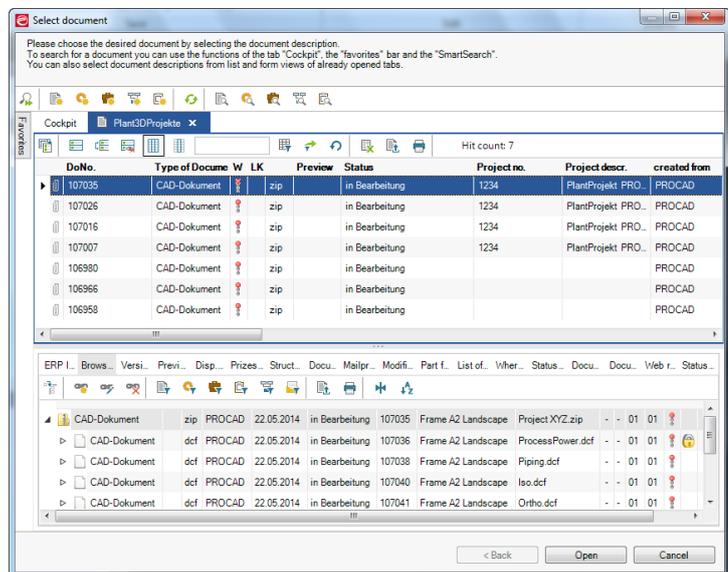
Function call from the PRO.FILE menu in Siemens NX:

"PRO.FILE" => "Managed Rename"

Proceed as follows

1. Select the "PRO.FILE" menu in Siemens NX.
 2. Select the function "Extras" => "Managed Rename".
- ⇒ The Checkout wizard to select the CAD document to be renamed is displayed. The Checkout wizard displays the PRO.FILE surface as it was recently opened.

3. If an assembly with the components to be renamed is not displayed on one of the existing tabs, you can now search for it:
 - Via the tab "Cockpit"
 - Via the search functions of the icon bar.
 - Via favorites, SmartSearch or task assignments.

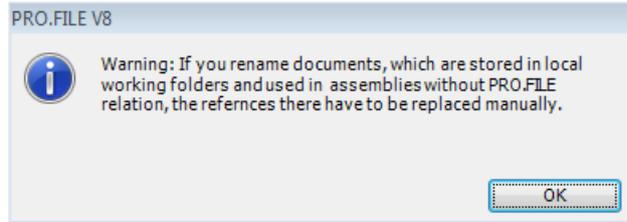


4. If the desired document is displayed on a tab, select it and click <Open>.

⇒ The Checkout wizard closes and a warning message is displayed.

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

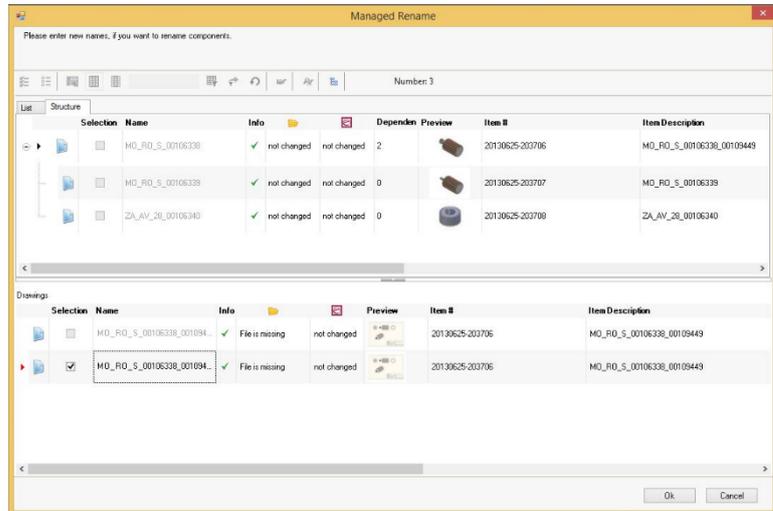
⇒ A warning message informs you that all references recognized by PRO.FILE will automatically be exchanged after the renaming.



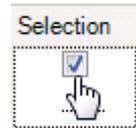
⇒ If the documents selected for renaming are used elsewhere, this cannot be recognized automatically. In such a case, manual post-processing would be necessary.

5. Confirm the warning message with <OK>.

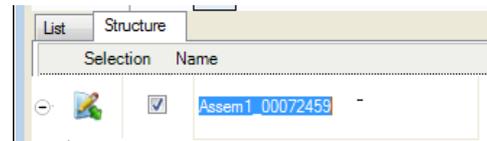
⇒ The window "Managed Rename" is displayed. In this windows you can rename the files:



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



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



8. Make the changes for all desired components.

9. Once you have renamed all desired components confirm your changes with <OK>.

⇒ The integration now saves the changed file names back to PRO.FILE and updates the references.

⇒ The renaming in the structure is now finished.

5.8 Save incremental

The incremental save function of the integration is aimed at saving assemblies and parts faster to PRO.FILE.

To achieve this, the sometimes time-consuming functions of the "standard" saving process are modified: The scope of the search for drawings and sub-assemblies to be saved is reduced. This means that when this function is used, only the active level of an assembly and the level below are searched for new or modified components and these components are offered for saving. If new or modified components are found, these are further searched (sub-levels) until no more new or modified components are found.

The function "**Save incremental**" is different from the function "**Save**" in the following limitations:

- The function "**Save incremental**" does not search the active folder for corresponding drawings.
- The function "**Save incremental**" does not display the list of locally modified documents in the Check-in wizard. It will only save new or modified documents that are part of the active assembly or the next sub-level of the structure.



Function call from the PRO.FILE integration menu:

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

The further proceeding for the function "**Save incremental**" corresponds to the proceeding described in the chapter "[Saving CAD objects for the first time](#)".

5.9 Save incremental automatic

This function "Save incremental automatic" differs from the "Save incremental" function in the fact that document and part master records



Function call:

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

6 Functions for the version administration

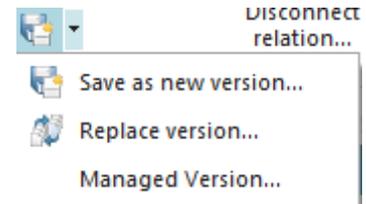
The integration PRO.FILE – Siemens NX offers several functions for opening and saving when working with versions:

- [Open with version browser](#)

This function has already been described in previous chapters.

Furthermore, there are the functions:

- [Save as new version](#)
- [Replace version](#)
- [Managed Version](#)



Information on this can be found in the following sub-chapters.



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

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

6.1 Save as new version

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



Note:

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



Note:

If you want to use several different versions of one component within an assembly, a new file name has to be generated for the component version. For this purpose, the parameter "Create new file name when versioning" has to be configured in the PRO.FILE Management Console, see manual "CAD supported by PRO.FILE".

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

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

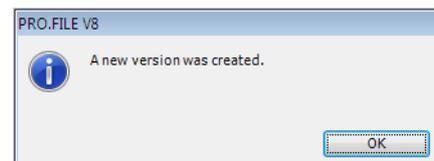


Function call from the PRO.FILE menu in Siemens NX:

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

Proceed as follows

1. Select the "PRO.FILE" menu from the menu bar in Siemens NX.
2. Select the function "Version" => "Save as new version".
 - ⇒ A list with all documents, of which a new version will be created, is displayed.
3. Confirm with <OK>.
 - ⇒ A new version of the active CAD documents is now created in PRO.FILE.
 - ⇒ A message box confirms the successful creation of the version.
 - ⇒ The new version is displayed in Siemens NX.



Attention: New version is not locked

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

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



Note: Versions of drawings

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

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



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

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

6.2

Replace version

The command "Replace Version" allows an existing, built-in version of a CAD object to be replaced by a new version for all assemblies in which it is used.

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

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

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



Attention: Undo not possible!

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



Function call from the PRO.FILE menu in Siemens NX:

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

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

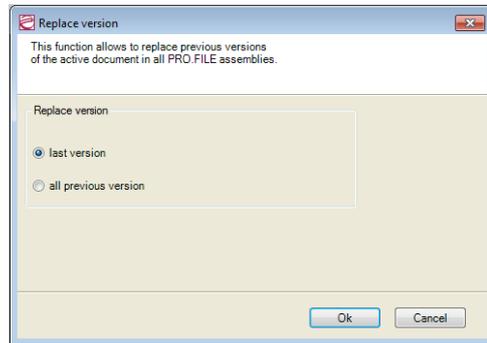
You can then replace the old version of the **active** PRO.FILE document in all corresponding assemblies and drawings with the new version.

Proceed as follows

1. Load the new version of the document, with which you want to make the replacement, from PRO.FILE in Siemens NX (open the replacing document, not the document to be replaced).
2. Select the function "PRO.FILE" => "Version" => "Replace version".
3. Select now, which of the predecessor versions is to be replaced by the new version:

Only the direct predecessor version, wherever it is used.

All predecessor versions, wherever they are used.



4. You now get a list of how often and where the predecessor version(s) of the document is/are used.
5. Select all records, for which a replacement is to be made.
6. Confirm your selection with <OK>.

⇒ The version is now replaced: The currently loaded version is then used by all selected assemblies/drawings.

⇒ You thus have cleaned all concerned objects.

If you have not modified all object, you can repeat this action. You then receive a list of all objects using the old version of the component (minus the objects already modified).



Attention:

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

6.3 Managed Version

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

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

Please note the following for "Managed Version":

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



Function call from the PRO.FILE menu in Siemens NX:

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

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

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

For further details, see the following sub-chapter:

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

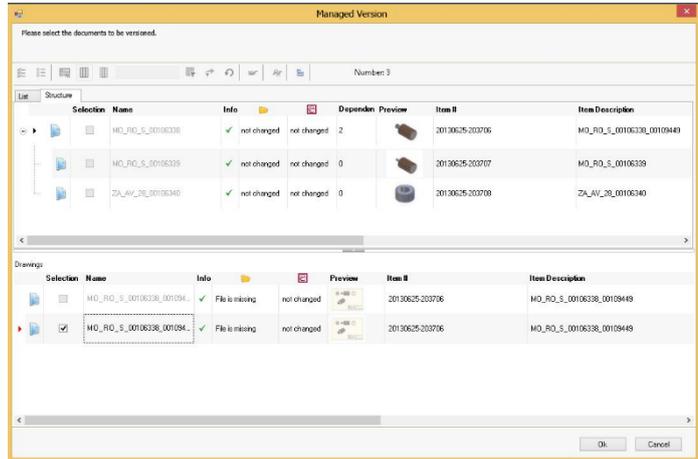
6.3.1 The proceeding for "Managed Version"

Proceed as follows:

1. Select the menu entry "PRO.FILE" from the menu bar in Siemens NX.
 2. Select the function "Version" => "Managed Version"
- ⇒ The Managed Version wizard is started.

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

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



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

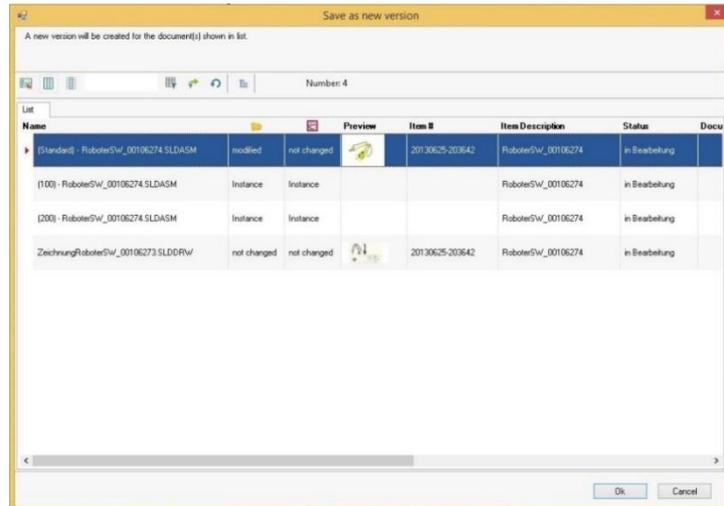
3. **Select:** Select all components which you want to save as a new version in PRO.FILE. Therefore activate the checkbox in the listed CAD documents as shown on the right.
4. Confirm your selection with <OK>.



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

5. Confirm with <OK>.

⇒ The selection components are now versioned.

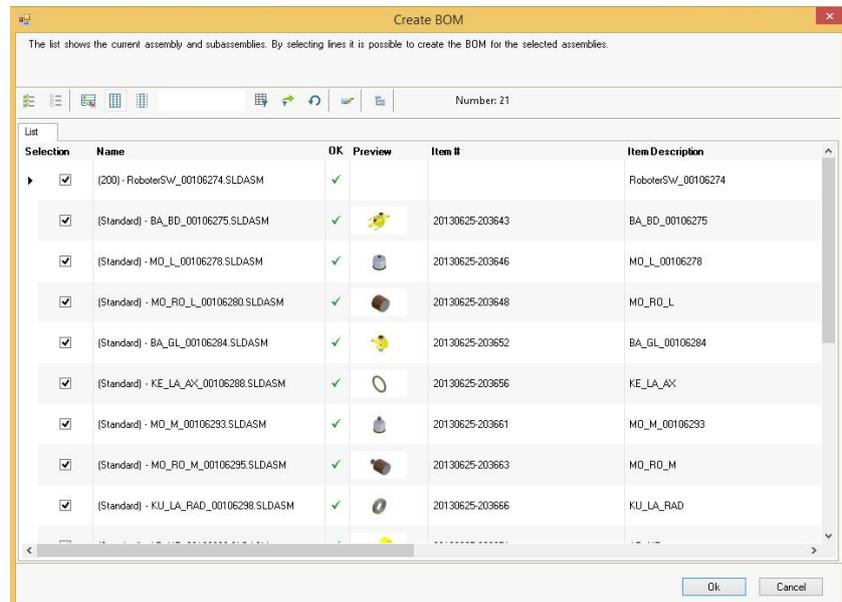


⇒ The successful completion of the process is confirmed by a message.

6. Confirm with <OK>.



⇒ The subsequent list shows the saved assemblies.



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

8. Confirm your selection with <OK>.

⇒ The process "Managed Version" is thus finished.

7 Linking of additional files

It is possible to link additional files to CAD objects. Such additional files may be text or image documents as well as CAD components from other CAD systems



Note:

CAD components from other CAD systems that have been saved with another CAD integration, have to be copied to the Workcenter folder via the function "Supply document" before they can be used.

For additional files, the following functions are available:

- [Append local file](#)
- [Add document](#)
- [Detach](#)

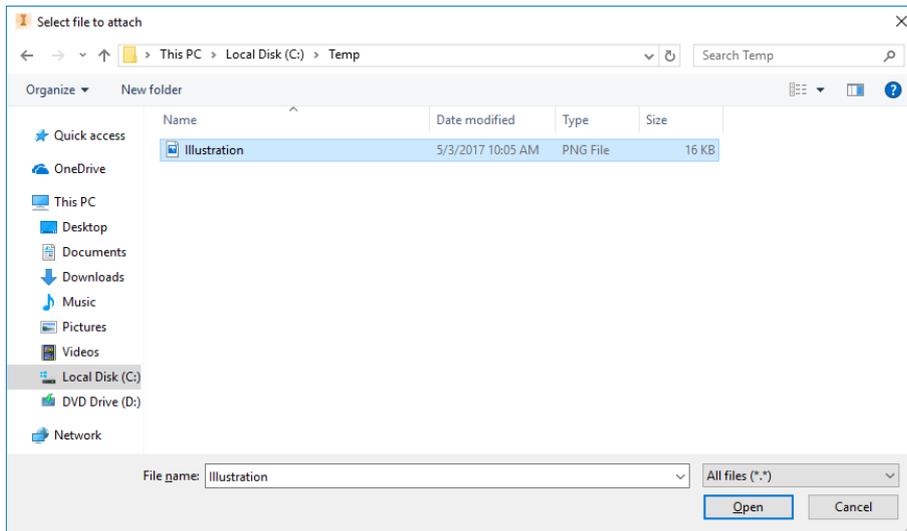
When opening a CAD object, the linked additional files are copied into the Workcenter folder. Later changes to the additional file always have to be made in the local Workcenter folder.

For CAD objects, additional files are displayed in the PRO.FILE tab "Browser", as well as in the selection dialogs for "Managed Copy" and "Disconnect relation". The versioning of bills of materials ignores additional files.

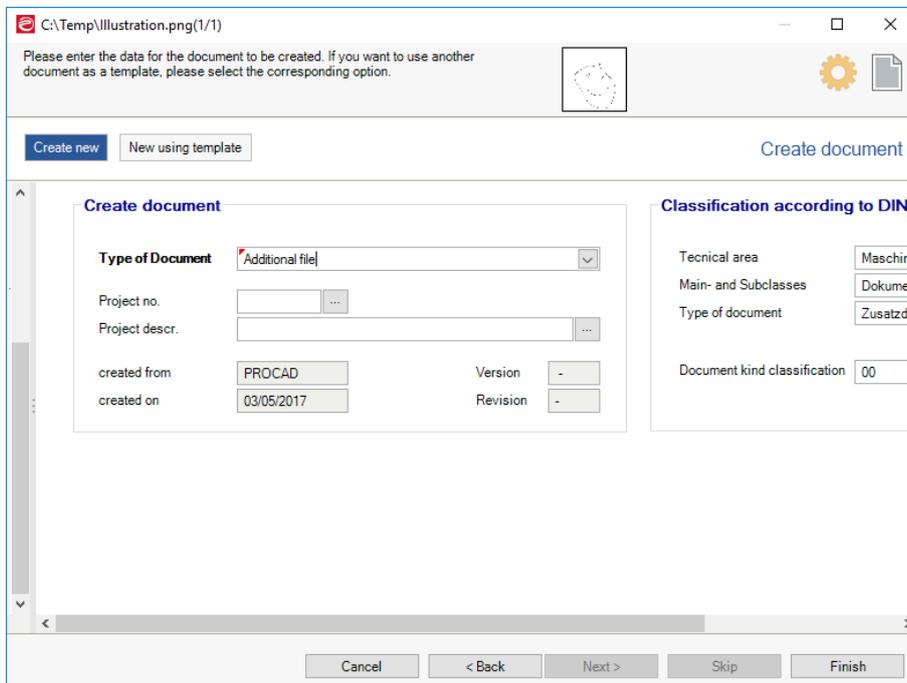
7.1 Append local file

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

1. First, load a CAD object that has been saved in PRO.FILE into your CAD session.
 2. Select the function "Links..." => "Append local file".
- ⇒ An Explorer window opens.

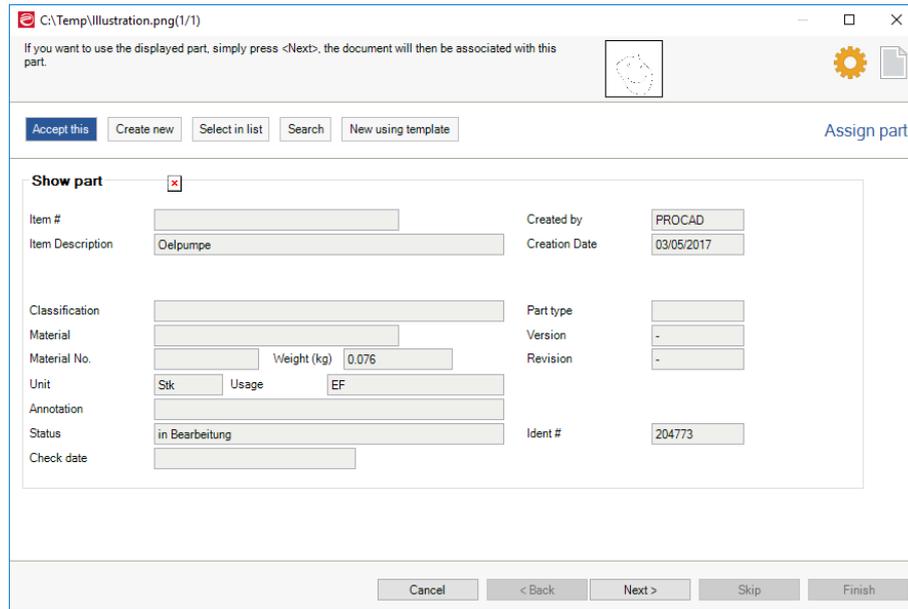


3. Select the file to be added and confirm your selection with <Open>.
- ⇒ The part master record of your CAD object is displayed in PRO.FILE.

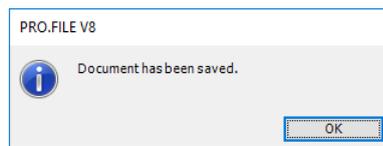


4. Confirm the assignment with <Next>.

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



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

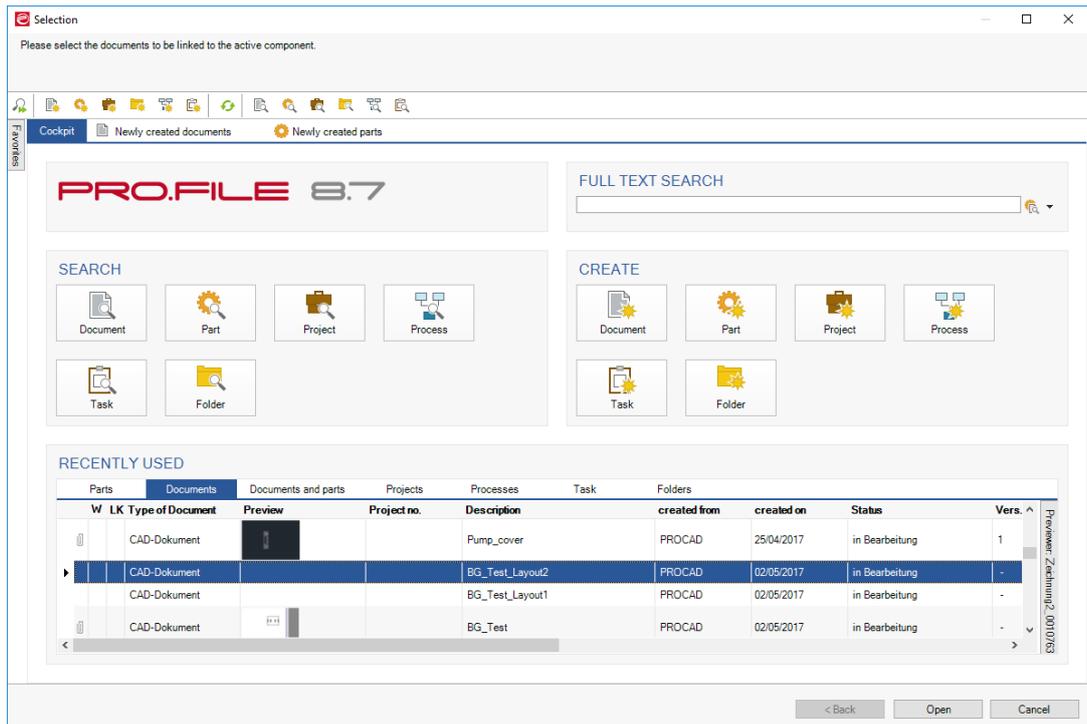


- ⇒ The additional file is saved in PRO.FILE. It is linked below the document description of the CAD object. If possible, a preview file is created for the additional file.
- ⇒ By adding it to the CAD structure, the additional file is automatically copied into the Workcenter folder.

7.2 Add document

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

1. First, load a CAD object that has been saved in PRO.FILE into your CAD session.
 2. Select the function "Links..." => "Add document".
- ⇒ The PRO.FILE Checkout wizard opens.



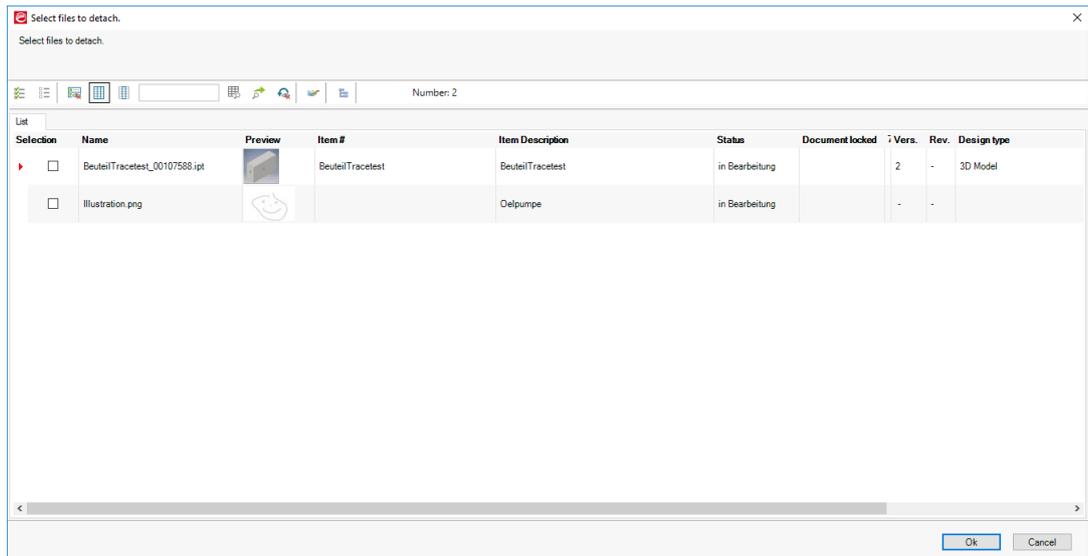
3. Select the document record of the file to be added and confirm your selection with <Open>.
- ⇒ The document record with the additional file is linked below the document description of the CAD object.

7.3 Detach

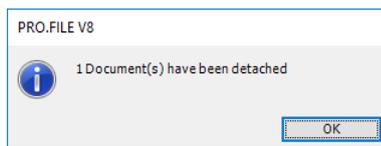
PRO.FILE prevents the deleting of documents as long as the documents are still in use – this also applies for additional files.

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

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



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



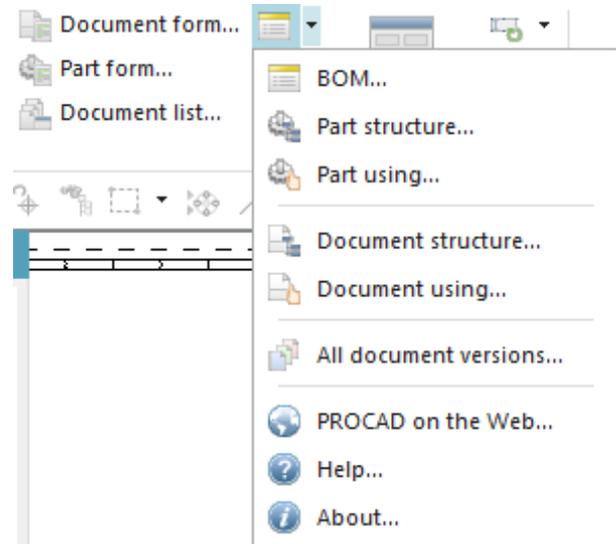
⇒ The selected document is removed from the CAD object structure.

8 Show: PRO.FILE Information at a glance

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

Depending on which details you are interested in, you can use the different menu entries to access the information.

This way, you can immediately access information stored in PRO.FILE on properties and classifications of the active CAD document as well as structure and usage information.



The following information can be found in the area "Show":

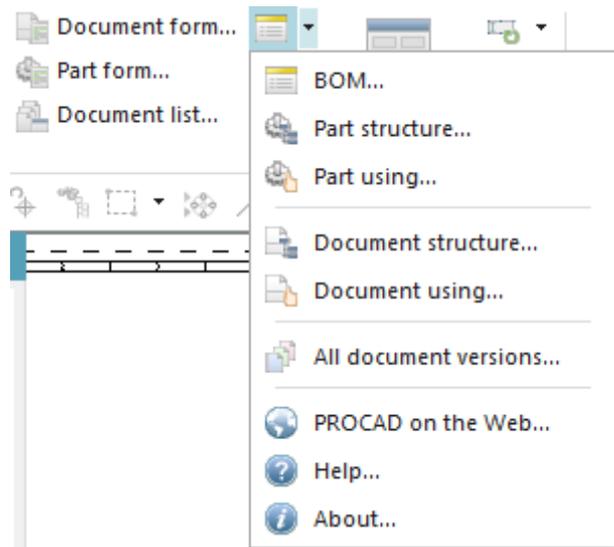
- [Show: Information on a CAD document in PRO.FILE](#)
- [Direct information in the dialog screens](#)
- [More comfort: search and list functions in the dialog screens](#)
- [Up to date or not: Display of status information](#)

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

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

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

- These menu entries access information on the CAD document currently active in Siemens NX.
- The various menu entries allow a targeted access to frequently needed information, without having to navigate in PRO.FILE.



- After a function is selected, PRO.FILE opens and displayed the desired information.
- Within the displayed lists and forms, you can perform all actions available in PRO.FILE.



Function call from the PRO.FILE menu in Siemens NX:

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



Note:

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

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

The following display options are available:

BOM

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

Part structure

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

Part form

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

Part usage

With the function "Part usage" you can see whether you current CAD document is used by other assemblies.

The usage list displays the "upward" structure.

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

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

Document structure With the function "Document structure" PRO.FILE displays the document the current CAD document is attached to and other documents used within the CAD structure.

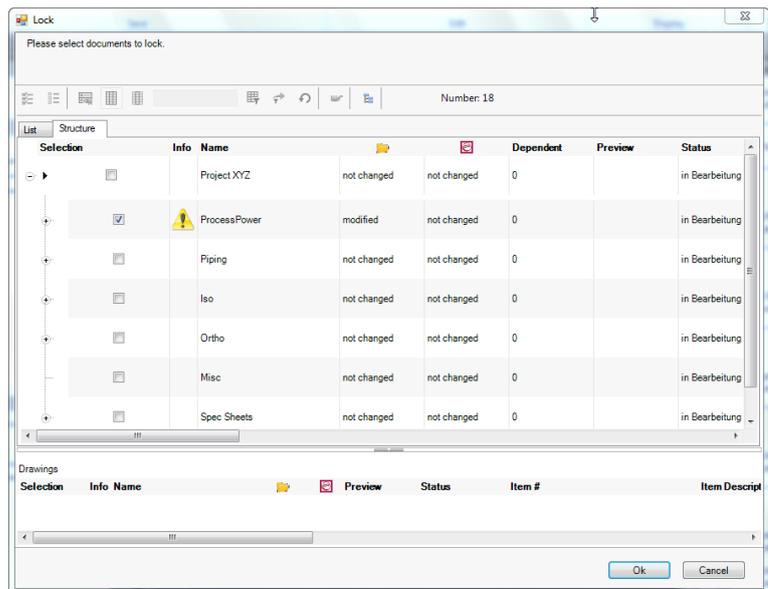
Document usage With the function "Document usage" you can see whether the document description of your active CAD document is used in other document or part descriptions.

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

8.2 Direct information in the dialog screens

For the functions lock, unlock and document list, dialog screens are displayed.

These offer the following functions:



- You can switch between list and structure display.
- Via the buttons of the list functions you can make searches and filter the displayed objects (see the following chapter "[More comfort: search and list functions in the dialog screens](#)").
- The lists contain status information for each of the listed objects (see the following chapter "[Up to date or not: Display of status information](#)").

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

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



Via these buttons, the following functions are available:

-  **Select all rows:**
 With this button, all rows of a list are highlighted.
-  **Invert selection:**
 With the <Shift> key pressed down, it is possible to select whole areas of a list, with the <Ctrl> key pressed down, you can select several individual rows. The button "Invert selection" can be used to select everything that is not selected and unselect everything that was selected.
-  **Hide selected rows :**
 If several rows of a list are selected, these rows can be hidden from the list with this button.
-   **Search in all columns / Search in active columns:**
 In order to be able to perform a targeted search for terms in the list, the user first has to select whether the search is to be carried out across all columns in the list or only for a specific column in the list.

 - : The search is performed across all columns in the list.
 - : The search is performed for the active column only. A column is activated by clicking the respective column header.
 -  **Define Filter pattern / Filter:**
 A character string can be entered into the entry field located within the icon bar. Here you can use the already described wildcards/meta characters.
 The search for the entered character string is started using the  icon.
 If the search pattern is found, all matching data records are highlighted.
-  **Next found pattern:**
 This icon is used to once again compare the entered filter pattern with the columns that are to be searched. The next data record found is highlighted.
-  **Show hidden rows:**
 If rows of a list have been hidden, this button can be used to display them again.

- 
PRO.FILE list selection:
 The entries of the selected rows are selected and opened in a list in PRO.FILE. This way you can immediately view the stored information without further selection.

8.2.2 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.

Info	Local 	PRO.FILE 	Description
	unchanged	changed locked	The file is locally unchanged but has been modified in PRO.FILE after it has been copied locally and can therefore not be saved back.
	unchanged	changed versioned	The file is locally unchanged but has been modified in PRO.FILE after it has been copied locally and can therefore not be saved back. 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.

9 Additional functions of the integration

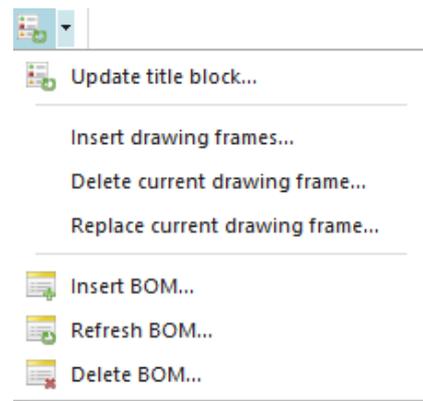
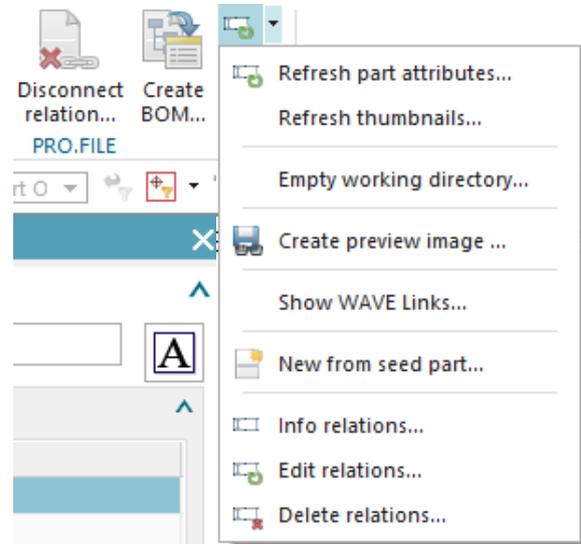
The integration PRO.FILE – Siemens NX offers several functions for opening and saving when working with versions:

- [Disconnect relation](#)
- [Create BOM](#)
- [Extra: Connect key characteristic features in PRO.FILE with CAD – parameters](#)

When a drawing is activated, you also have access to the functions:

- [Update title block](#)
- [Extra: Insert BOM / Refresh BOM / Delete BOM](#)
- [Extra: Insert, delete and replace drawing frame](#)

These functions are described in the following sub-chapters.



Furthermore you can access the following functions via the "Extra" menu:

- **Extra: Show Wave Links**
Via this functions you can view components of a model linked via WAVE link.
- **Extra: Refresh part attributes**
This functions updates the model attributes with current values from PRO.FILE.
- **Update thumbnails:**
If a thumbnail is used in the PRO.FILE document list, this function can be used to update the thumbnail.

9.1 Disconnect relation

By the classification of CAD objects with PRO.FILE you have an excellent tool for the administration of your designed models and documents. This "Knowledge base" can be used for new designs. With correct classification you can find similar objects for new designs in PRO.FILE and use them for new designs or modifications.

You thus avoid tedious redesigning of the object. In order for the existing object to become independent from the objects in PRO.FILE, you can use the function "Disconnect relation".

This function removes the database connection of a part, drawing or assembly from the PRO.FILE database. The CAD objects are then treated as local CAD objects and behave like newly designed objects.



Note:

The PRO.FILE function "Disconnect relation" renames the file.

- If the active document contains renamed CAD elements that are also used in assemblies or drawings, this results in an **immediate** new referencing of the assemblies, drawings and parts **opened in the background** to the new file names. This is a default behavior of Siemens NX and cannot be influenced by PRO.FILE.



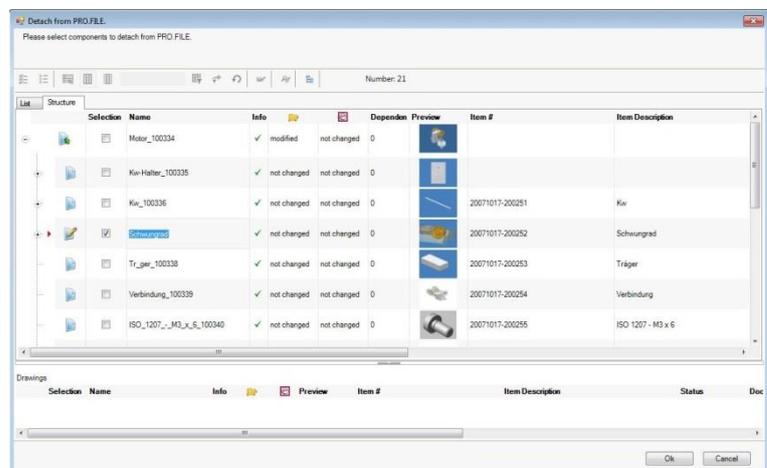
Function call from the PRO.FILE menu in Siemens NX:

"PRO.FILE" => "Disconnect relation"

Proceed as follows:

1. Select the "PRO.FILE" menu in Siemens NX.
2. Select the function "Disconnect relation".

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



3. Select all CAD documents, the PRO.FILE connection of which you want to dissolve by using the checkboxes.



⇒ The previous file name without ID, or with the suffix "_Index" is suggested as new file name, e.g. "assembly_1".

4. Give the document to be disconnected a name under which it is to be saved locally. You can edit this name freely in the "Name" column.



5. Once all files are selected and renamed, confirm with <OK>.

⇒ The PRO.FILE connection for all selected CAD components is now removed. The existing objects are locally removed from the CAD session and the new objects are loaded.

⇒ The disconnection of the objects is thus finished. A check via the function "File" => "Properties" shows that all objects are displayed as if newly designed.

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

To save these CAD files again to PRO.FILE see chapter "[Save: How to save CAD data and changes to PRO.FILE?](#)".



Note:

If the database connection of an assembly is dissolved, the document relation of parts contained in this assembly remain intact. You only have disconnected the local copy of the assembly from the document description.

9.2 Create BOM

With the function "Create BOM" a bill of materials based on the CAD structure of the active document in Siemens NX is created, which can be viewed in PRO.FILE.

If a bill of materials list structure already exists in PRO.FILE for the assembly body of parts in question, the structure will be **updated**. If the Siemens NX geometry contains parts that are not yet contained in the bill of materials, these will automatically be used in the PRO.FILE bill of material structure.



Note:

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

- Bills of materials can only be created for assemblies.
- The CAD documents must be linked to a part master in PRO.FILE.



Note: Exclude suppressed components from the BOM

If you have suppressed components of an assembly within the CAD session and you do not want these components to become part of the BOM, you can specify this via the parameter "Ignore suppressed components in BOM" in the PRO.FILE Management Console. By default, suppressed components are displayed in the BOM.



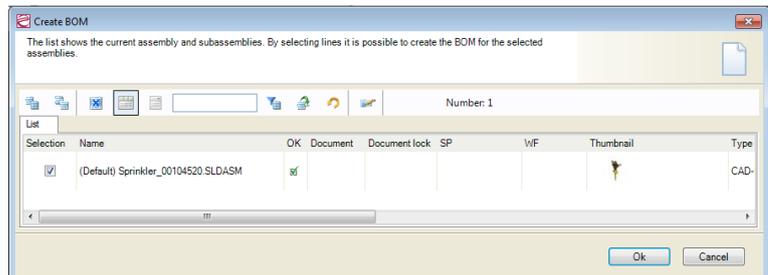
Function call from the PRO.FILE menu in Siemens NX:

"PRO.FILE" => "Create BOM"

Proceed as follows

1. Select the "PRO.FILE" menu in Siemens NX.
2. Select the function "Create BOM".

⇒ A list displays all currently loaded assemblies and sub-assemblies.



3. Select all CAD documents for which you want to create or update the bill of materials by using the checkboxes.

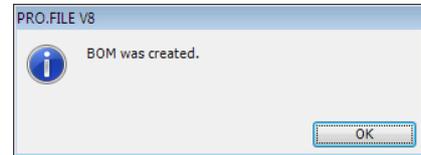


Note: Display of conflicts

If the creation of a BOM is not possible for an assembly, the column "OK" shows a red "Attention" symbol. The corresponding tooltip shows the cause of the problem.

4. Confirm your selection with <OK>.

- ⇒ If positions are contained in the PRO.FILE bill of materials list which do not occur in the geometry, a query will occur, in which the user can determine whether the bill of materials list positions should be deleted in PRO.FILE.
- 5. Confirm your selection with <OK>.
- ⇒ The bill of materials for the selected CAD assemblies is created/updated in PRO.FILE.
- ⇒ The successful creation/update of the bill of materials is confirmed by a message.



To view the bill of materials in PRO.FILE, select the function "Show" => "Bill of materials" as described in the chapter "[Show: Information on a CAD document in PRO.FILE](#)".



Note:

Please note that norm parts or auxiliary materials not displayed in the drawing (e.g. water or oil) is not included in the bill of materials by the function "Create BOM".

The description of the functions for editing a bill of materials in PRO.FILE can be found in the PRO.FILE manual "Operation PRO.FILE advanced".



Note: Remove assemblies from BOM

If an assembly has the string attribute "BomMoveUp" with the value "YES", the components of this assembly are moved one level upwards in the PRO.FILE BOM. As a result, assemblies that are only there for structure purposes do no longer appear in the bill of materials.



Note:

This function can be activated automatically for each saving procedure. Please see the configuration manual of the integration for further details.

9.3 Update title block

With this function the drawing legend of the current Siemens NX drawing is filled with current data from PRO.FILE.

This function can only be used if the current Siemens NX documents contains a drawing saved in PRO.FILE. Furthermore, the drawing legend already has to be configured.



Function call from the PRO.FILE menu in Siemens NX:

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

After confirmation of a query mask the title block, modification list and bill of materials are filled automatically, depending on the configured template.

- The modification list always lists the most recent entries.
- If the modification list contains more entries than rows in the configured template, older entries are not displayed.



Attention:

This menu entry is only available if the drafting mode in Siemens NX is activated.

9.4 Extra: Insert BOM / Refresh BOM / Delete BOM

With this function you can place, update or delete up to five parts lists on the drawing.



Note:

At most five bills of materials can be managed on a drawing.

9.5 Extra: Insert, delete and replace drawing frame

If a drawing is active, you have access to the following drawing frame functions:

- **Insert drawing frames:**
The function inserts a drawing framework matching the sheet size.
- **Delete current drawing frame:**
The active drawing framework is deleted.
- **Replace current drawing frame:**
With this function you can interactively select and insert a drawing framework.

9.6 Extra: Connect key characteristic features in PRO.FILE with CAD – parameters

For more information on how to build key characteristic feature connections and on their menu commands, see the following chapters:

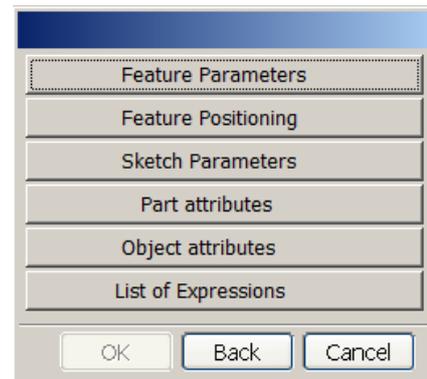
- [Building up key characteristic feature connections](#)
- [Menu items in Siemens NX](#)

9.6.1 Building up key characteristic feature connections

Building up a key characteristic feature connection works between a CAD document and the part master the document is part of. No connection can be built up between a document and a part tribe which aren't connected. This also means that no parameters/attributes of sub-modules/parts of an assembly can be selected. For this the active part is switched over in the Siemens NX window when storing an assembly for the first time, when the part masters are created as well.

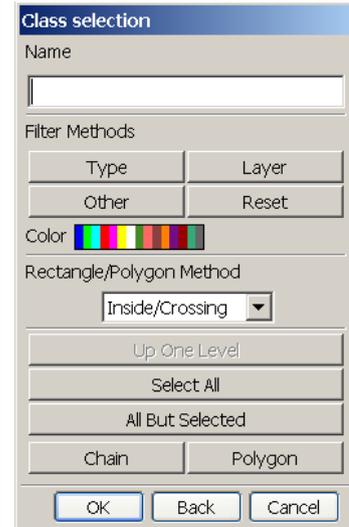
When storing a part to PRO.FILE for the first time you can use the selection button for the fields configured before (see first item). You switch to Siemens NX and a choice dialog appears with four options. With the help of that you can select a parameter or an attribute of the part active in Siemens NX.

- **Feature Parameter:** You can list all form elements in the active part/assembly and then list the parameters in the selected form element. The parameters aren't shown in the graphic window. As a remedy one can have the dimensioning shown on the Construction element navigator for the selected construction element.



- **Feature Positioning:** You can list all form elements in the active part/assembly and then list the positioning parameters for the selected form element. The parameters also are shown on the graphic window and can be selected directly.
- **Sketch Parameter:** You can list all drafts in the active part/assembly and then list the parameters for the selected draft. The parameters also are shown on the graphic window and can be selected directly.
- **Part attributes:** All attributes are listed in the active part/assembly:

- **Object attributes:** In this dialog, at first a mask appears in which one can define different object types to the selection. This mask works analogously to the Siemens NX menu item "Information/Object" (Version NX). All attributes which can be chosen for the selected objects appear in the next dialog after the object was selected. The attributes of objects in the active part are respectively taken into account. When storing the attribute name, in addition to the attribute name also internal information is saved for the object in the parameter name for the key characteristic feature coupling. The different information is separated by the string @@. The attribute name mustn't be longer than 16 signs.



- **List of expressions:** All parameters are listed for the active part/assembly. When activating a parameter the form element/object, to which the parameter is associated, is activated.
- **Numerical values:** The corresponding numerical value is returned for the parameter values. For the attributes the character string is filtered and the corresponding floating value is formed.
- **Character strings:** The string after the sign "=" is transferred.

9.6.2

Menu items in Siemens NX

The following menu items are available to you in SIEMENS NX:

- **Info relations:**

A list is shown with all key characteristic feature relations built up. The list refers to the part master, that you connect with the document active in Siemens NX.
- **Edit relations:**

A list is shown with all key characteristic feature relations built up. The list refers to the part master, that you connect with the document active in Siemens NX.

In the second step the part master for this document is shown. So that you can edit the part master fields and the necessary key characteristic feature relations you must switch to the part tribe editing mode (e.g. right mouse button and change). In this window the selection button for the fields configured before is used, in the same way as when creating key characteristic feature relations (see chapter: [Extra: Connect key characteristic features in PRO.FILE with CAD – parameters](#)). You can edit existing key characteristic feature relations or build up new relations.
- **Delete relations:**

A list is shown with all key characteristic feature relations built up. The list refers to the part master, that you connect with the document active in Siemens NX.

In this list you can select single or all entries and then delete by confirmation.

10 Extra: The Workcenter

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



Starting the Workcenter from the PRO.FILE menu in Siemens NX:

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

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

This chapter described:

- [Workcenter functions](#)
- [Empty working directory](#)

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



Attention when working with several work folders:

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

10.1 Workcenter functions

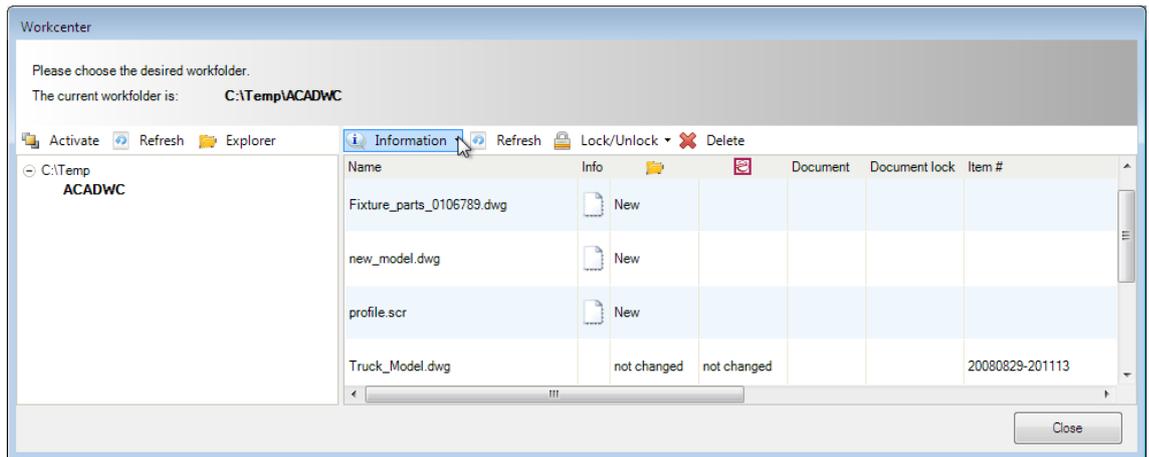


Starting the Workcenter from the PRO.FILE menu in Siemens NX:

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

The Workcenter is divided into two areas

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



The functions for the directory structure:



Activate

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



Refresh

The view of the directory structure is updated.



Explorer

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

The functions for the working directory:



Information ▾

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

Structure of the parts	Document structure
Part form	Document form
Usage of parts	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).

10.2 Empty working directory



Important: Risk of data loss!

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

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



Function call from the PRO.FILE menu in Siemens NX:

"PRO.FILE" => "Extra" => "Empty working directory"

The function "Empty working directory" performs the following steps:

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

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

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