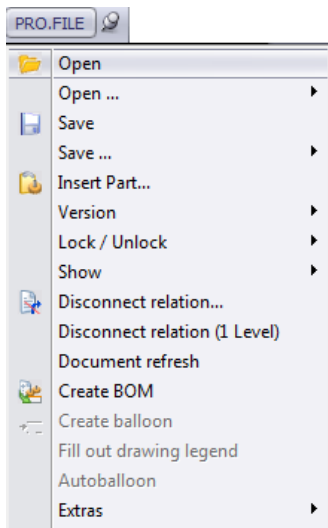**Quick guide**

PRO.FILE 8.4
Integration SolidWorks

This quick guide is to give you an overview of the integration functions. For detailed information please see the operation manual.

An overview of the integration functions

Via the menu "**PRO.FILE**" the user can access the PRO.FILE functions for document management directly from SolidWorks. This menu offers the functions listed in the following.

Functions for the opening of CAD objects

- **Open:**
This function opens PRO.FILE and provides the user the option to choose a CAD document and open it in SolidWorks.

- **Open with Version 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 SolidWorks.

- **Open drawing:**
With this function, you can directly load the drawing saved in PRO.FILE for the object active in SolidWorks.

- **Copy only:**
This function can be used if files saved in PRO.FILE are only to be loaded into the local work folder, but, due to time reasons, not to be loaded in SolidWorks.

- **Open with all drawings:**
If an assembly is opened, you can use this function to open all drawings available in PRO.FILE for the assembly structure in SolidWorks.

Saving 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.

- **Save automatically:**
Document and part descriptions for all sub-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.

- **Save all instances:**
If you have created several instances of one object, this function allows you to save all objects without having to save each instance separately.

- **Save instances automatically:**
This feature combines the functions for saving all instances and automatic saving. With this function, when you are saving an object, you do not have to enter part and document information for each instance.

- **Save phantom:**
This function is used to save an entire assembly under an individual part master record in PRO.FILE and to save all objects contained in the assembly under this part master record. As a result, this assembly is treated like a single part in PRO.FILE. The objects contained in the assembly are saved as phantom objects and cannot be opened explicitly in PRO.FILE

Note: Only the objects not known in PRO.FILE yet will be saved as phantom objects. For all objects already known in PRO.FILE "Save Phantom" will have the same effect as "Save".

- **Managed Copy:**
Managed Copy organizes the data management of complex 3D models in changed constructions. Entire machines can be cloned, including all referenced data and drawings. New numbers of the cloned machine and the new structures are automatically updated in the dependent drawings. Assemblies and parts that are to remain in the new structure are taken over. Existing references remain intact.

- **Managed Copy automatically:**
This feature combines "Managed Copy" with the function "Save automatically". When copied components are saved anew, you do not have to enter part and document information for each part.

- **Save NDF:**
With this function, a **n**eutral **d**ata **f**ormat (e.g. tiff, pdf) is created from the CAD document and saved as new document in PRO.FILE. This NDF document is automatically linked to the part master record of the drawing.

Functions for versioning

- **Open with released versions:**
This open function automatically reads the newest released version of the references of the CAD objects from PRO.FILE in Solid Works.

- **Save as new version:**
Saves the currently active CAD object as a new version in PRO.FILE.

- **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.

Functions to Lock/Unlock CAD Objects

- **Lock:**
CAD objects which were read from PRO.FILE in SolidWorks 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.

- **Lock (1 level):**
By calling this function only CAD objects of the first sub-level of an assembly are listed. This results in a limited list of objects to be locked, but this list loads much faster and is easier to handle.

- **Unlock:**
This function unlocks the PRO.FILE objects which were locked for processing in SolidWorks. 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.

- **Unlock (1 level):**
By calling this function only CAD objects of the first sub-level of an assembly are listed. This results in a limited list of objects to be unlocked, but this list loads much faster and is easier to handle.

PRO.FILE database functions

- **Disconnect relation:**
  This function deletes the database link of a PRO.FILE part, a PRO.FILE drawing or an entire PRO.FILE assembly including a selection of objects contained therein. The CAD objects are then treated as purely locally-saved CAD objects without reference to PRO.FILE.

- **Disconnect relation (1 level):**
  The function "Delete Database Link (1 level)" lists only the first sub-level of an assembly. This results in a limited but very fast list of CAD objects to be unlocked, the database links of which are to be deleted.

- **Document refresh:**
  This function allows all parts of an assembly to be compared with the latest version in PRO.FILE, and be re-read into the assembly.

Functions for assemblies

- **Insert part:**
  With this function you can select a part saved in PRO.FILE and insert it into the assembly in SolidWorks via the SolidWorks "Insert part" wizard.

Functions for bill of materials and drawing legend

- **Create BOM:**
  This function calls up PRO.FILE and creates a new bill of materials for the active assembly. If there is already a PRO.FILE BOM, it will be updated.

- **Create balloon:**
  This function allows you to display the bill of material positions when working with drawings and assemblies from PRO.FILE.

- **Fill out drawing legend:**
  This function allows information on bill of materials, modification lists and titles, to be filled in on a drawing upon opening. This requires the lists and fields to be pre-configured for the template that is to be used.

- **Autoballoon:**
  This function reads the position numbers of parts from PRO.FILE and transfers them to the drawing in SolidWorks.

Information and display functions

The sub-menu "Show" offers several functions for the display of PRO.FILE information on the active CAD document:

- **List of documents / List of documents (1 level)**
- **Document structure**
- **Document form**
- **Use of documents**
- **All document versions**
- **Structure of the parts**
- **Part form**
- **Use of parts**
- **Bill of materials**
- **Configurations (Instances)**
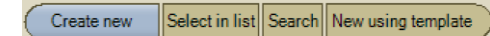
Extras and Configuration options

- **Workcenter:**
  All files loaded or saved via the PRO.FILE integration in SolidWorks are automatically saved locally in the Workcenter folder. With this function you can manage these files or create additional work folders.

- **Managed Rename:** This function allows the later renaming of CAD documents from PRO.FILE. The references known to PRO.FILE are then updated accordingly.

- **Drawing plot:**
  All, or certain drawings of a BOM, can be plotted with this function.

- **Options:**
  The local integration can be configured using this menu item. It includes the possibility to establish document lists, to show or hide messages, to configure original name references, title blocks and performance setting. If required, contact your administrator.

Saving CAD data to PRO.FILE

Via the function "**Save**" you can save parts, assemblies and drawings you have created in SolidWorks to the PRO.FILE database.

- First, you have to save your newly created objects locally. After that, you can save the objects to PRO.FILE. To do so, activate the PRO.FILE menu in the SolidWorks menu bar and select the function "**Save**".

  The **Checkin Wizard** is displayed. In the first step, the assignment of the CAD documents to be saved to a PRO.FILE part master record has to be made. The wizard offers different functions:
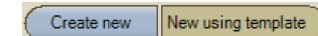
Create new	Select in list	Search	New using template

  Create new: A new part master record is to be created for the new document.
  Select in list / Search: The document to be saved is not to be linked to a new part master record, but to an existing part master record.
  New using template: A new part master record is to be created for the new document. An existing part master record is to be used as template for the new one.
  - Once you have made the desired entries for the classification of the part, confirm the saving of the part master record with <**Next**>.

- In the second step, the document description is created. Here too, the Checkin Wizard offers different functions:

Create new	New using template

  Usage of and proceeding for these options is identical to the usage and proceeding of the options for the part master record, with the difference that the options here refer to the document description.
  - Once you have made the desired entries for the classification of the document, confirm the saving of the document description with <**Next**>, if you also want to make a project assignment, otherwise with <**Finish**>.

- If you also want to make an assignment to a project, the third part of the Checkin Wizard is displayed. Here you can select or, if required, create a project. In order for the new CAD documents to be assigned to the selected Project, you need to use the option "**Activate**" before finalizing the wizard screen.

Quick guides for operation manuals

To make the usage of PRO.FILE and its modules as easy as possible, the basic functions of the module are explained in this leaflet.

- If you want to use this guide as a pamphlet, you can print it double-sided (duplex: Flip on Short Edge) and fold the sheet along the dotted lines, so that page "1" faces upwards (see image below).

- Alternatively you can use the sheet without folding, e.g. as laminated desktop pad.