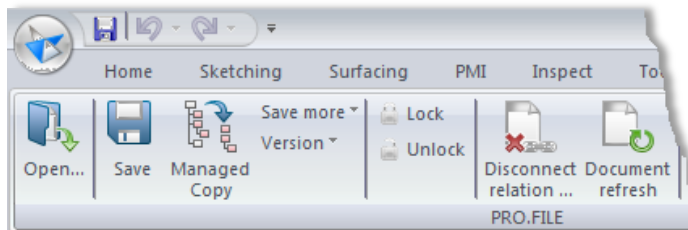**Q u i c k   g u i d e**

PRO.FILE 8.4
Integration Solid Edge

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 Solid Edge. This menu offers the functions listed in the following.

Functions for opening CAD objects

- **Open:**
  This function opens PRO.FILE and prompts the user to select a CAD document for loading in Solid Edge.
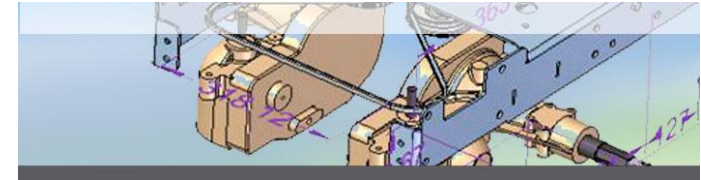
Functions for saving objects to PRO.FILE

- **Save**
  Via this function newly created CAD documents are checked in to PRO.FILE, or documents already saved in PRO.FILE and checked out for editing are saved back to PRO.FILE.

- **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.

- **Save Phantom**
  By calling up this function, it is possible to lay down a complete assembly under a single part master in PRO.FILE, and to save all objects that are contained in this assembly under the part master. From then on, this assembly will be treated as an individual part by PRO.FILE. The objects contained in the assembly are laid down as phantom objects, and cannot be called up explicitly from PRO.FILE.

- **Managed Copy**
  Managed Copy organizes the data management of complex models in the change design. Entire machines are cloned, including all referenced data and workshop drawings. Assemblies and components that are to remain in the new design are taken over. Existing references remain intact.

- **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.
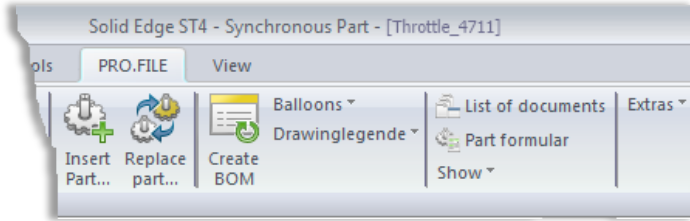
Functions for Versioning in PRO.FILE

- **Open with released versions**
  This function automatically loads the newest released versions of references of CAD objects from PRO.FILE to Solid Edge.

- **Open selection of versions**
  With the version selection you can load assemblies in dynamic compositions to be defined by the user. The user can decide which version of an assembly and of its components is to be opened.

- **Save as new version**
  Saves the actual 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.

- **Replace version**
  This function allows an existing used version of a CAD object to be replaced by a different object in all assemblies in which the previous version is used.

- **Append version**
  This command allows to save the currently active object as a new version to a same object (Part to part, assembly to assembly, etc.). It will be "attached" as new version to the version chain. If you open one of these objects later on, you will be informed that a newer version has been found.

PRO.FILE Database functions

- **Lock**
  CAD objects which were read from PRO.FILE in Solid Edge 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.

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

- **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.

- **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 to modify CAD objects from PRO.FILE

- **Insert part**
  With this function you can select a part saved in PRO.FILE and insert it into the assembly in Solid Edge.

- **Replace part**
  This function allows replacing a selected part in the assembly by a part saved in PRO.FILE.

Functions for assemblies and drawing content

- **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.

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

- **Drawing legend**
  This menu contains functions for updating the drawing legend, for deleting elements from the drawing legend or for copying the drawing legend from an existing document into the currently loaded document.

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**
  opens the special documents list and displays the configured information on the current part, the drawing or the assembly, including all linked objects.

- **Structure of the parts**
  switches to PRO.FILE and displays the structure overview of the current part.

- **Part form**
  switches to PRO.FILE and displays the description form for the active part.

- **Use of parts**
  displays the proof of use list of the displayed/selected part in PRO.FILE.

- **Bill of materials**
  displays the bill of materials view of the displayed/selected object in PRO.FILE.

- **Document structure**
  switches to PRO.FILE and displays the structure overview of the current document.

- **Document form**
  switches to PRO.FILE and displays the description form for the active document.

- **Use of documents**
  displays the proof of use list of the displayed/selected document in PRO.FILE.

- **Drawing** – The corresponding drawing is opened for a model. At first, a check is performed, whether this is saved locally. If this is not the case, the drawing is searched in the PRO.FILE database and opened afterwards

- **All document versions**
  displays the version list of the selected object in PRO.FILE.
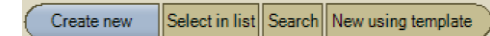
Extras and configuration options

- **Working directory activate**
  All files that are read or saved by Solid Edge while working with the Solid Edge integration of PRO.FILE are automatically stored locally on the user computer. By using the function "Working directory activate" it is possible to select and modify the local storage path. If several sub-directories have been created in a directory, it is possible to set the path to be used for data storage by activating it.

- **Options**
  In this menu point you can make a large number of different settings for your active Solid Edge integration. If required, contact your administrator.

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

Saving CAD data to PRO.FILE

Via the function "**Save**" you can save parts, assemblies and drawings you have created in Solid Edge 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 Solid Edge 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:
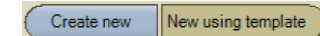

  Create new: A new part master record is to be created for the new document.

  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:

  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.