

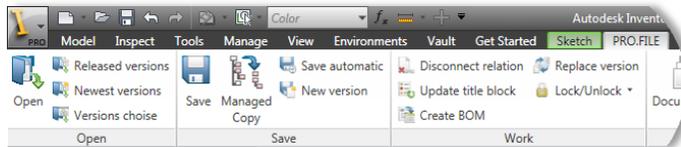
Quick guide

PRO.FILE 8.4
Integration Inventor



This introduction is to give you an overview of the integration functions. For detailed information please read the operation manual.

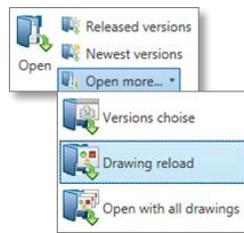
Function overview



Via the menu "PRO.FILE" the user can directly access the PRO.FILE functions for document management from Inventor. This menu offers the functions listed below. The contents of the menu may vary, depending on whether you have opened a part, an assembly or a drawing.

Functions for opening CAD objects

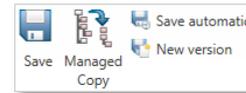
- **Open:**
This function opens PRO.FILE and prompts the user to select a CAD document for loading in Inventor.
- **Open with released versions:**
The selected document is opened from PRO.FILE. If other CAD documents are linked to this document, the newest released versions of these CAD components are loaded.



- **Open with newest versions:**
The selected document is opened from PRO.FILE. If other CAD documents are linked to this document, the newest versions of these CAD components are loaded.
- **Version choice:**
Via the version selection the user can decide which versions of an assembly and its components is to be loaded.
- **Drawing reload:**
If a CAD object (part, assembly) is loaded from PRO.FILE in Inventor, this function can be used to load the corresponding drawing from PRO.FILE as well.
- **Open with all drawings:**
If an assembly is loaded, this function can be used to load all drawings from PRO.FILE that are available within the assembly structure.

Saving functions

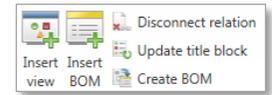
- **Save:**
Via this function newly created CAD documents are checked in to PRO.FILE, or documents already saved in PRO.FILE and checked out for editing are saved back to PRO.FILE.
- **Managed Copy** (only for parts and assemblies):
Managed Copy organizes the data management of complex models in changed constructions. Entire machines can be cloned, including all referenced data and drawings. Assemblies and parts that are to remain in the new structure are taken over. Existing references remain intact.
- **Managed copy automatic:**
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 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 all automatic:**
This function enhances the function "Save automatic": All CAD document loaded in the CAD session are saved to PRO.FILE. No query for data input in PRO.FILE is made, all records are created automatically.



- **New version:**
Saves the currently active CAD document as a new version in PRO.FILE.
- **Save NDF** (only for drawings):
With this function, a neutral data format (e.g. tiff, pdf) is created from the CAD drawing and saved as new document in PRO.FILE. This NDF document is automatically linked to the part master record of the drawing.

Functions for editing CAD objects

- **Insert view** (for drawings):
With this function, Inventor components from PRO.FILE are inserted into the active CAD drawing.
- **Insert BOM** (for drawings):
If you want to display a bill of materials created in PRO.FILE on your CAD drawing, you can use this function to insert and place it. If a PRO.FILE bill of materials is already placed on the drawing, it is updated.
- **Disconnect relation:**
This function deletes the database connection of a CAD document to PRO.FILE. The CAD objects are then treated as a locally saved CAD document and no longer have a PRO.FILE relation.
- **Update title block:**
Via this function the iProperties of the active CAD document and its structure (if exists) are updated. If these iProperties are mapped to text fields of the title block or the modification list in a drawing, these are updated as well. The prerequisite for this is that these lists and fields are already configured for the used drawing template.
- **Create BOM:**
This function creates a bill of materials, based on the current CAD structure, and saves it to PRO.FILE. If a PRO.FILE bill of materials exists already, it is updated.



- **Replace version:**

This function allows to replace a version of a CAD part in all assemblies, in which it is used, with a newer version.



- **Lock:**

CAD documents loaded from PRO.FILE in Inventor are not automatically locked for other users. If you want to edit a CAD document, you have to use the command "Lock" before making the changes. This prevents the document from being changed by a different user in the meantime.

- **Unlock:**

With this function CAD documents, which were locked for editing in Inventor, can be unlocked. Other users can now also edit the object.

- **Place component** (for assemblies):

This function inserts a component from PRO.FILE into the active assembly.

- **Replace component** (for assemblies):

This function replaces a component selected in the active Inventor assembly by a component from PRO.FILE.

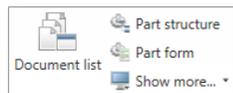
- **Replace all components** (for assemblies): This function replaces all copies of a component selected in the active Inventor assembly.

Information and display functions

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

- **Document list:**

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



- **Part form:**

Switches to PRO.FILE and displays the description form for the active part.

- **Part structure:**

Switches to PRO.FILE and displays the structure overview for the active part.

- **Show more:**

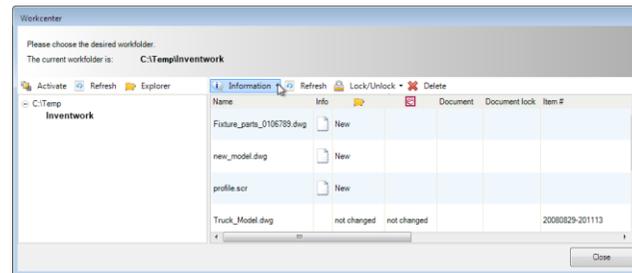
Via the functions in this sub-menu, the following additional information can be accessed in PRO.FILE for the active object:

- **Part usage**
- **BOM**
- **Part position**
- **Document structure**
- **Document form**
- **Document usage**
- **All document versions**

Extras

- **Workcenter:**

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



- **Help:**

This function opens the PRO.FILE online help.

Saving CAD data to PRO.FILE

Via the function "Save" you can save parts, assemblies and drawings you have created in Inventor 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 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. For this, the Checkin Wizard offers different options:



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 options:



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.

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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.

