

Quick guide

PRO.FILE 8.5 Integration CATIA V5



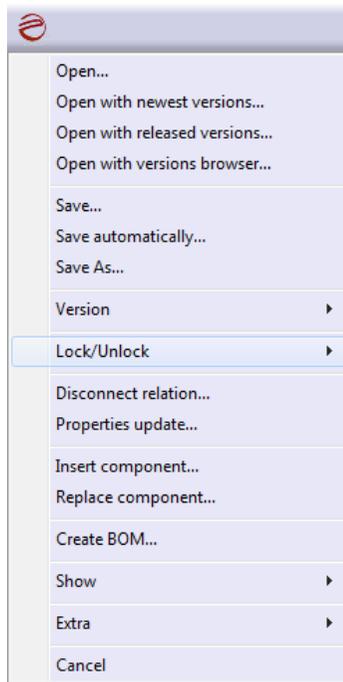
This quick guide is to give you an overview of the integration functions. For detailed information please see the operation manual.

An overview of the integration functions

Via the menu "PRO.FILE" the user can access the PRO.FILE functions for document management directly from CATIA V5. This menu offers the functions listed in the following.

Functions for the opening of CAD objects

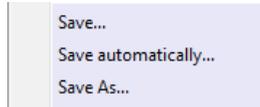
- **Open:**
This function opens PRO.FILE and provides the user the option to choose a CAD document and open it in CATIA V5.
- ... with newest versions:
Specifies that the structure of a document is analyzed when the document is opened from PRO.FILE, and that the newest available version of each component is loaded.
- ... with released versions:
Specifies that the newest released version (if available) of each component is loaded.



- ... 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 CATIA V5.

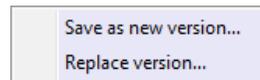
Saving functions

- **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 as:**
This function can be helpful to use entire existing assemblies as template for starting a new design project. The resulting "cloned" documents do not have a relation to the originals but are only a copy thereof.
- **Save NDF:**
With this function, a neutral data format (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

- **Save as new version:**
Saves the currently active CAD object as a new version in PRO.FILE. If the function is applied to a part that is used in an assembly, the references of the assembly still point to the old version, following the new version management of PRO.FILE.
- **Replace version:**
This function allows an existing built version of a CAD object to be replaced by a newer version of the object in all assemblies in which the previous version is used.



Functions to Lock/Unlock CAD Objects

- **Lock:**
CAD objects which were read from PRO.FILE in CATIA V5 are not automatically locked for the user. To be able to modify a CAD object, the function "Lock" must be called up beforehand. By this function, 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 CATIA V5. 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.



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.
- **Properties update:**
If metadata from PRO.FILE has been linked to properties of a product or part, these properties can be updated (refreshed) with this function.



Working with components from PRO.FILE

- **Insert component:**
With this function you can open the CAD models saved in PRO.FILE within an assembly. The assembly, into which the component is to be inserted, is activated in CATIA V5 with a double click. You can then use the integration function and select the component in PRO.FILE. The positioning is made automatically by CATIA V5.
- **Replace component:**
This function allows the exchange of an existing component in an assembly with another component saved in PRO.FILE.





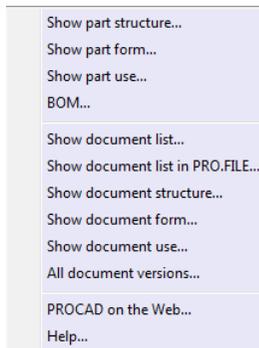
Functions for bill of materials and drawing legend

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

Information and display functions

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

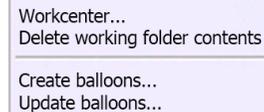
- **Part structure:**
Shows the structure overview of the displayed/selected object.
- **Part form:**
Displays the part master record form of the displayed/selected object.
- **Part usage:**
Displays the proof of usage list of the displayed/selected object.
- **Bill of material:**
Displays the bill of material view of the displayed/selected object.
- **Document list:**
This function calls up the document special list in PRO.FILE, and shows the user the information that has been configured for the current part, drawing or assembly, and all attached parts.
- **Document structure:**
Displays the document structure list of the displayed/selected object.
- **Document form:**
Displays the document description in form view of the displayed/selected object. If there is no object specifically selected within an assembly, the form of the displayed assembly is displayed.



- **Document usage:**
Displays the proof of usage list of the displayed/selected object.
- **All document versions:**
Shows all versions of the document saved in PRO.FILE.

Extras and Configuration options

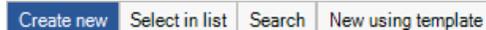
- **Workcenter:**
All files that are read or saved by CATIA V5 while working with the integration of PRO.FILE are automatically stored locally on the user computer. By using the function "Working directory activate" it is possible to select and modify the local storage path. If several sub-directories have been created in a directory, it is possible to set the path to be used for data storage by activating it.
- **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 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 CATIA V5 menu 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:

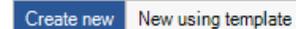


Create new: A new part master record is to be created for the new document.
Select in list / Search: The document to be saved is not to be linked

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.

PROCAD GmbH & Co. KG
Vincenz-Prießnitz-Straße 3
D-76131 Karlsruhe
Tel +49 (721) 96 56-5
Fax +49 (721) 9656-650


PROCAD
<http://www.procad.de>
Mail: info@procad.de

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.

