



FILER VIEW

Getting started

Short instructions for using Filer

Selection of the configuration
Select a part of the files in the BOM folder to view.

Search
Search for part from the part tree

Design tree
The structure of the assembly is displayed according to the SolidWorks structure.

Show component properties
Opens and closes a window that displays the selected section's information and related files.

Settings
Open and close the settings tab.

Download the part
The files related to the part are copied to the selected folder

Download part and subparts
The files related to the part and its subparts are copied to the selected folder.

Print
Print a PDF file of the selected section.

Open PDF
Open the PDF file of the selected part with the default PDF viewer.

Notifications
Tells if a PDF file cannot be found for a part or if the revision of the PDF file differs from the revision given in the BOM file.

Id	Nimi	Materiaali	Mitat	Kpl	Määrä
15	M132861	KÄSIJOHDE		1	1.15
14	M132860	KÄSIJOHDE		1	0.71
13	M132859	LÄSI		1	38.17
12	M132858	LÄSI		2	31.06
11	M132857	PEITELEVY		1	40.43
10	M132856	PEITELEVY		1	24.04
9	M132855	MUNO		1	38.99
8	M132842	TUKITOPPA		1	18.17
7	M132837	MUNO		1	43.32
6	M132834	MUNO		1	88.27
5	M132828	SIVURUNKO		1	90.82
4	M132827	LANKKU		1	8.48
3	M132826	LANKKU		1	4.82
2	M132825	LANKKU		1	3.02
1	M132824	LANKKU		12	3.33



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Settings

Save and load settings

PDF Directory

Set the path to the folder where the PDF and other assembly-related files are stored.

BOM File Directory

Set the path to the folder where the BOM files are saved

Save Settings Load Settings

Create new BOM file

Favorite BOM list

Change Edit

PDF Directory

D:\Malliva Oy Dropbox\7 ASIAKKAAT\Malliva\Filer_test_files\kulmaporras_PDF

Change

BOM File Directory

D:\Malliva Oy Dropbox\7 ASIAKKAAT\Malliva\Filer_test_files\kulmaporras_BOM

Change

Create new BOM file

Create a new BOM text file. For example, you can create a new top-level assembly for the order, in which you select the parts belonging to the order as parts.

Favorite BOM list

In the Favorites text file, you can set parts that appear first in the assembly selection drop-down menu. The lines of the text file must have the format "filename"; "caption". (semicolon is used as separator) Filename is the name of the BOM file without the file extension. The legend is the text displayed in the menu. The legend can be left blank, in which case the file name will be displayed in the menu.



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Create BOM files

The structure is transferred from SolidWorks to Filer using BOM files. Each assembly and part has its own text file

BOM files are generated with the Export-for-filer macro supplied with the program. Open the desired assembly in Solidworks. Run the structure Export-for-filer macro from the menu tools -> macro -> run. The macro asks if you want to create the transfer files only for the selected assembly or also for the subassemblies. The macro can be found in the program's installation folder C:\Users\"username"\AppData\Local\Programs\Filer Project 2022\Export_macro and can be copied elsewhere if desired. By default, BOM files are saved in the folder c:\Temp\FILER_BOM, which is created automatically when the program is installed.

The save folder can be changed by editing the Export-for-filer.ini file. The export macro also needs a folder for temporary files. This defaults to c:\Temp\export_temp

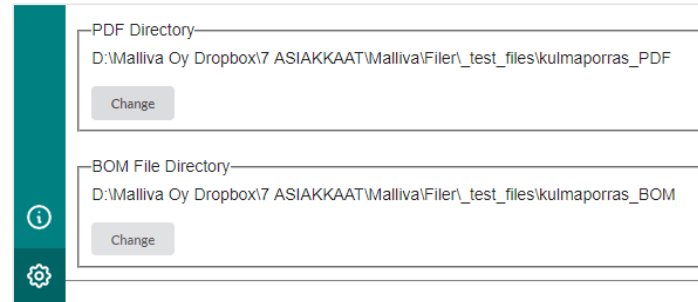
Editing macro settings is presented in more detail in the section "Export macro settings".

```
' folder for the result file
SaveToFolder=c:\Temp\FILER_BOM\
' folder to save the temp-file
TempFolder=c:\Temp\export_temp\
```

Setting file locations in Filer

Filer reads files from two folders. BOM files are saved in the desired folder and PDF, DXF, STEP or any other files related to the model are saved in another folder. Parts related to the same assembly must be saved in the same folder, but the same folder can contain files from several assemblies.

Folder paths are set in Filer under "PDF Directory" and "BOM File Directory". By default, these are c:\Temp\FILER_BOM and c:\Temp\FILER_PDF-DXF and do not necessarily need to be changed.





Select the structure to view

Select an assembly or part from the drop-down menu.

You can search for a part based on the file name from the drop-down menu. Favorites can also be assigned to it, which appear at the beginning of the list. Favorites are defined by a text file that can be defined and edited in the settings.



Updating the structure and files

In order for the changes made to the structure on the SolidWorks side to be updated in Filer, a new transfer file must be run for the changed parts and moved to the BOM folder used by Filer. Since each part has its own transfer file, there is no need to run the macro for the entire structure.

The structure must also be read again in Filer. Filer stores the given folder information for parts whose file path in the structure remain the same.

Correspondingly changed PDF images and other files are transferred to Filer's PDF folder. Filer always uses the latest revision of files, so there is no need to delete old drawings. Revision should appear at the end of the file name so that different revisions can be distinguished from each other.



Export macro settings

```
' does macro ask if the user wants to export
Userquestion= TRUE

' ass default, does macro export whole structure (TRUE) or just open model (FALSE)
StructureExport= FALSE

' folder for the result file
SaveToFolder=c:\Temp\FILER_BOM\

' folder to save the temp-file
TempFolder=c:\Temp\export_temp\

' parameters of the master model in order (*xx* area special values - statics)
' parameters can be combined to same field by using "+"
MasterRow=*Type*,*Filename*,*Config*,Item,desc1,version,*PCS*,desc2+standardi+koko,material+materialf,dimensions+koko,weight

' definition texts for the abow defined parameters
MasterRowHeader=Tyyppi,Tiedostonimi,Konfiguraatio,Nimike,Nimitys 1,Revisio,KPL,Nimitys 2,Materiaali,Mitat,Massa

' BOM template = Exportstructure.sldbomtbt whole structure or just open model
```

The format of the BOM files and the operation of the export macro can be modified by changing the Export-for-filer.ini file and the Export-for-filer.sldbomtbt BOM template.

- **Userquestion:** When running the macro, is the user asked if they want to make a BOM only for the open part or also for sub-parts. "TRUE" if you want to be asked. Otherwise "FALSE"
- **StructureExport:** "TRUE" or "FALSE". If false is selected, the macro will by default make only the open part of the transfer file. If true is selected, the migration file will also be created from all sub-files.
- **SaveToFolder:** File path to the folder where the macro writes the transfer files. The folder must exist, and the macro will not create it itself if it is missing.
- **TempFolder:** The file path to the folder where the macro writes temporary files.

- **MasterRow:** Solidworks custom property names for the data that the macro writes to the BOM file. The information given here can be seen in the Filer and on the quantity lists written by the Filer. The first seven of these must be in order: part type, file name, configuration, part number, title, revision and quantity. Those marked with a star do not need to be changed. Others must be given custom Property values where the information can be found. After these mandatory first seven, the desired number of fields can be added. These optional fields don't have to be in a specific order and can be given any custom properties or combinations of these using the "+" sign.
- **MasterRowHeader:** Header fields for the information given above. Free-form text can be entered in the title fields. These are used in Filer as data field names and in quantity lists as header fields. This row must provide as much information as the MasterRow row.
- **BOM pohja:** If you want to make changes to what information is written to the BOM files, the changes must also be made to the Exportstructure.sldbomtbt BOM base. It must have the same data in columns as specified in MasterRow in the same order. The BOM template can be modified in SolidWorks just like any BOM template in SolidWorks.