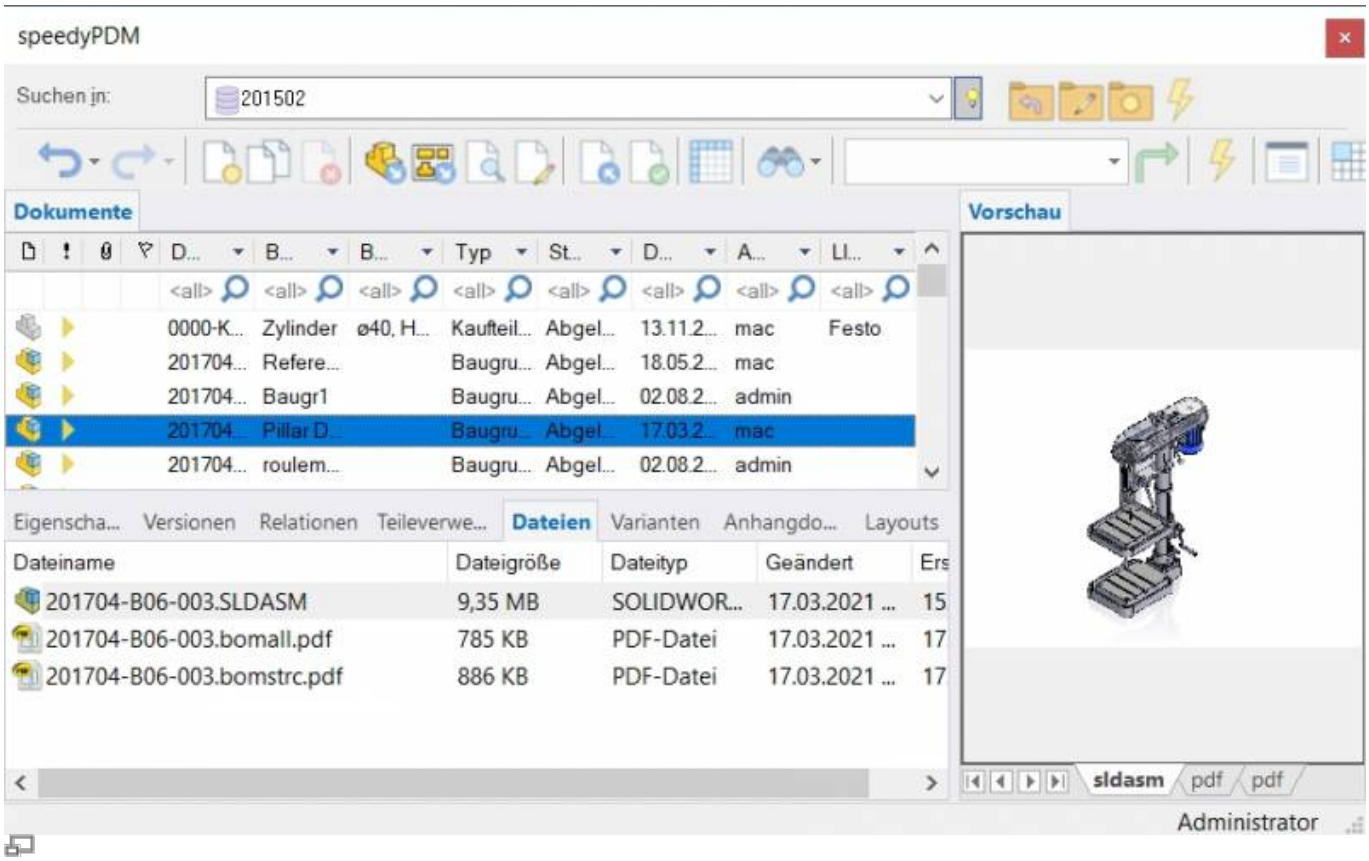
SOLIDWORKS

The integration of speedyPDM in SOLIDWORKS ® takes place as an additional application (addin). The speedyPDM Solidworks addin provides, in addition to the usual menus and toolboxes, a rollup window or alternatively a task window.

Rollup

The rollup window is positioned above the title bar of Solidworks and opens automatically when you move the mouse pointer over the title bar.
Alternatively, the rollup window can also be displayed in the task window area. The configuration parameter [solidworks.rollup.dockable] controls the behaviour.



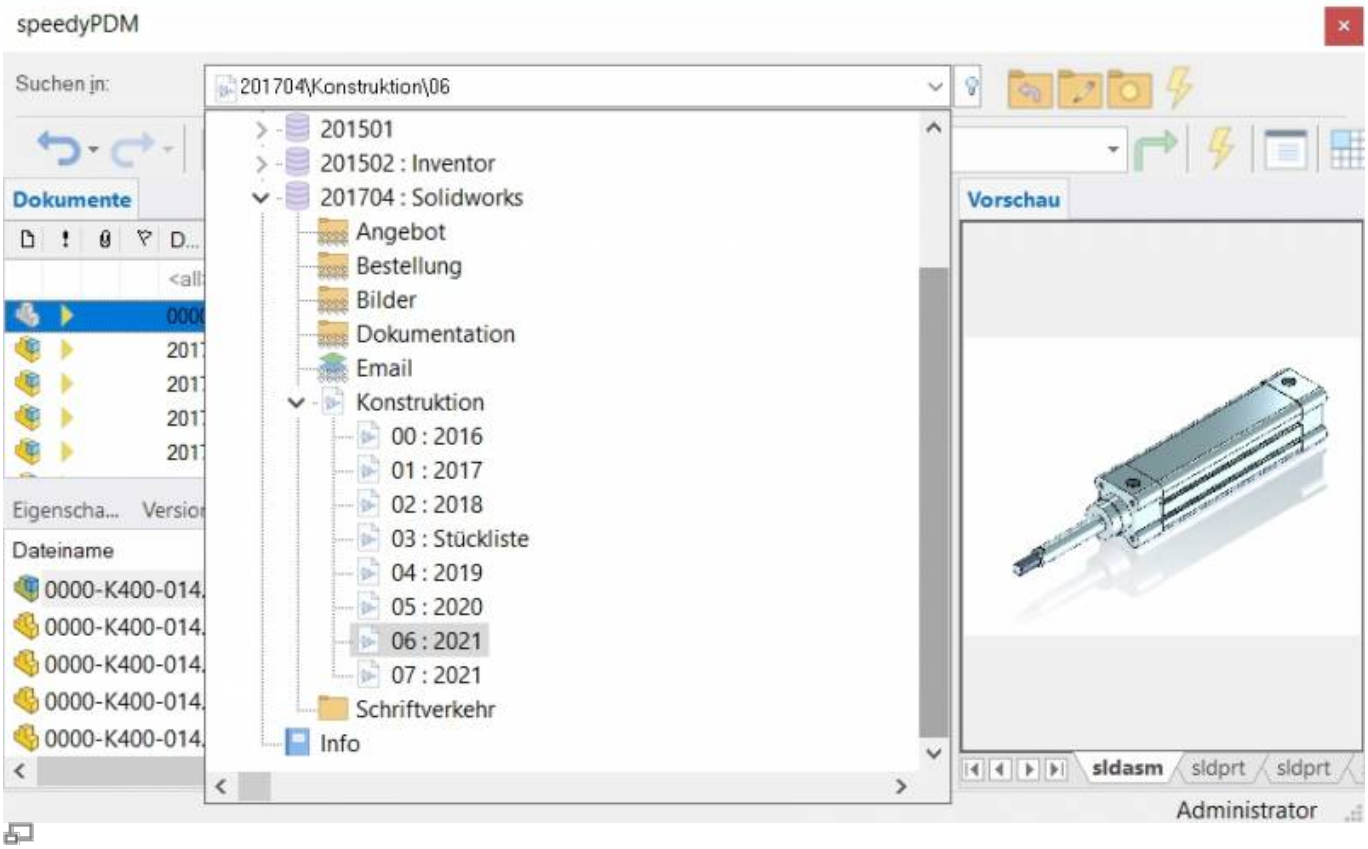
If the rollup window is not placed in the task window area, it automatically closes again when you work in Solidworks.

The rollup window has a similar structure to the speedyexplorer with the exception of the navigation bar, which has been implemented in the form of a drop-down list to save space.

The position of the individual sections in the rollup window can be adjusted manually and are saved by user.

Navigation and project/folder management in the roll-up window

The navigation area is located in the upper part of the roll-up window in the form of a drop-down list. The operation is similar to the navigation in speedy-Explorer.



Navigation in the rollup window

Click on the [-] or [+] symbol to open or close the corresponding tree.
The same is achieved by double-clicking on the container symbol.
Double-clicking on a directory entry sets the working directory and closes the navigation window.

After selecting a path, the documents are already displayed in the background.

To the right of the navigation bar is a toolbox for project and folder management:

Symbol	Meaning
	Move up one level in the folder structure
	Create a new container. If the vault is active in the navigation, a new project folder is created. If a project folder or another container is active, a new subfolder is created.
	Edit folder properties

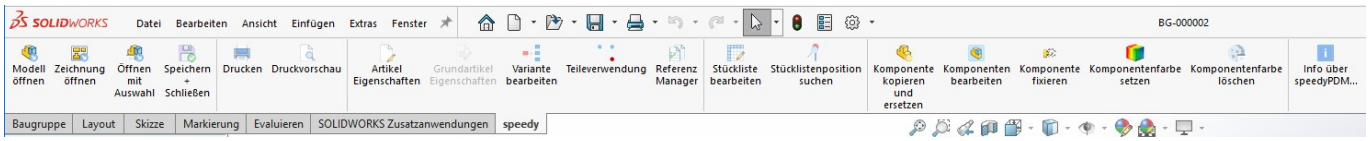
Symbol	Meaning
	Update structure view
	Project visibility on/off.

Toolbar

Symbol	Meaning
	Navigation history back
	Navigation history forward
	New document
	Open document (model or assembly)
	Open drawing
	Check-in document
	Share document
	Document properties
	View document
	Delete document
	Refresh view

Symbol	Meaning
	Search document
	User options

The speedy Ribbon Bar



Create new document

There are several options to choose from:

Solidworks:

- You create a new file with the Solidworks command File „ New.
- You create the model or assembly using the usual Solidworks methods.
- You select the Solidworks command File “ Save.
- The speedy dialogue „New Document“ appears.
  Enter your details for the new item.

Rollup window:

- Select the command „New Document“.
- The speedy New Document dialogue appears.
  Enter your details for the new item.
  speedy creates a new document and creates an empty document file based on the template.
- The new article is marked in the document list.
- Select the „Open Document“ command.
- The empty document file opens.

Open document

There are several options to choose from:

Solidworks:

- Select the Solidworks command File „ Open.
- The speedy dialogue Open file is displayed.
- Select the desired document and confirm the dialogue with Open.

Rollup window:

- Search for the document to be opened.
- Select the command „Open document“.

Open drawings

Menu speedy “ Open drawing
Context menu assembly „ Open drawing
Open drawing toolbox

Opens the drawing for the currently opened model (part/assembly).
If no drawing exists yet, a drawing is automatically created and opened based on the templates defined in the document type.
If several drawing templates have been defined, a selection is made from the list of possible templates.

If the drawing template already contains predefined views, these are automatically filled with the model.

Open component

Menu speedy “ Open part/assembly
Context menu assembly „ Open part/assembly
Open component toolbox

If you have opened an assembly and want to open a component or subassembly for editing, you must use the speedy command Open Part/Assembly. Parts and sub-assemblies are only loaded in read-only mode when opening an assembly to prevent unnecessary check-outs from the speedy database and unintentional changes. Therefore, if you use the Solidworks command to open parts, they will be opened in read-only mode.

Open component drawing

Menu speedy “ Open component drawing
Context menu assembly „ Open component drawing

Opens the drawing file of the selected component.

Insert component

You have opened an assembly and want to insert an existing part or another assembly. You have various options to choose from:

Solidworks:

- Select the Solidworks command Insert “ Component „ Existing Part/Assembly.
- Select the SEARCH button.
- The speedy Open File dialogue is displayed.
- Select the desired document and confirm the dialogue with OPEN.

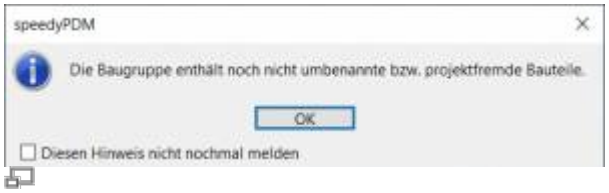
Rollup window:

- Locate the document to be opened.
- Select the document in the document list.
- Drag and drop into the Solidworks drawing area while holding down the left mouse button.
- The part or assembly linked to the document is inserted into the current assembly and „hangs“ on the mouse pointer.

If the document contains configurations or is a variant part, the Edit Variant dialogue appears after insertion.

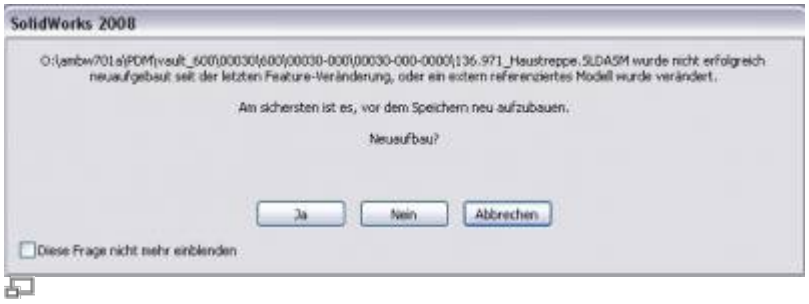
Save

When saving assemblies, the system checks whether the assembly still contains parts or subassemblies that do not belong to the current project.



The message can only be confirmed with OK. The file is saved in any case. Use the command Rename Component to rename parts that do not belong to the project.

If the current file is used several times, a message window also appears when saving:



You now have the option of checking-in the file under a new item number or saving it under the same name.

Saving and closing

To close a document managed in speedy you should select the command Save+Close from the speedymenu or the speedytoolbox.

Menu speedy “ Save+Close
Ribbon speedy „ Save+Close

The model is zoomed into the window and an isometric view is set to provide a meaningful file preview, then the file is saved and closed.

Copy and replace component

Menu speedy “ Copy and replace component
Context menu assembly „ Copy and replace component

Used to replace components that are not part of the project.

Meaning of the input fields

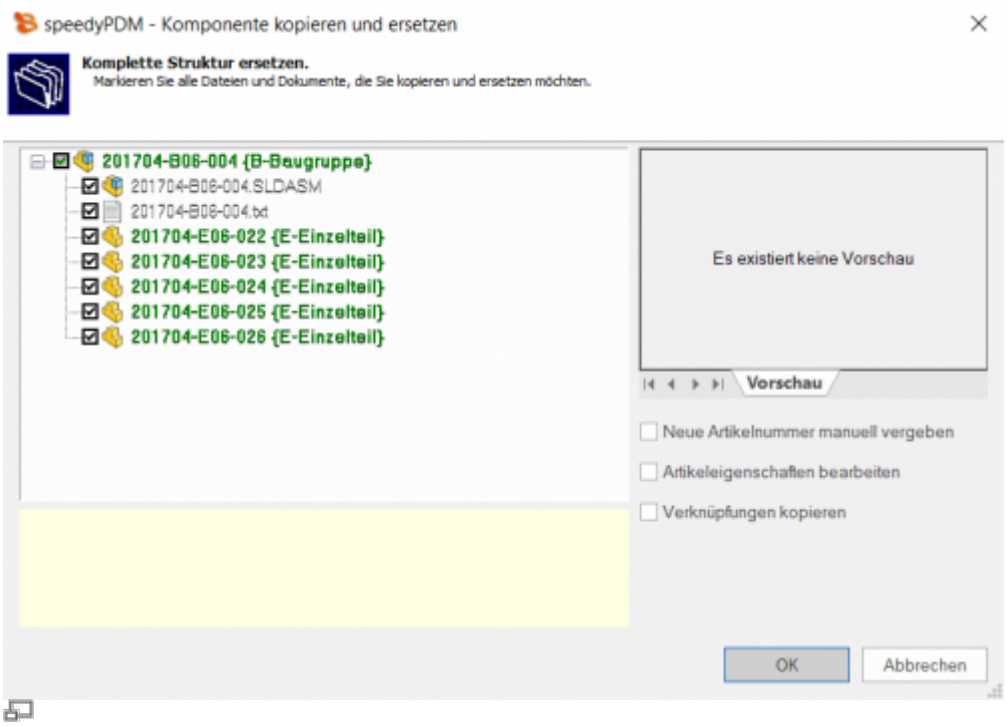
Components	List of selected components
Select automatically	Searches for all components in the 1st level that need to be replaced.
Replace selected components only	Replaces only the selected components.
Replace complete structure	Replaces the selected components and their subassemblies and subcomponents.

When you confirm the properties page with OK, speedy works through the selected components in sequence and then copies and replaces them. To do this, the speedydialogue [New Document] appears for each component, in which you define the document name. Before the dialogue appears, speedy checks the Solidworks file for defined properties and, depending on the degree of recognition, fills in the document type, the name and the supplier.

If the option „Replace selected components only“ was selected and it is an already managed speedy document, a new number is automatically assigned. The new document is created in the container of the current assembly. In this case, no new dialogue appears.

Replace complete structure:

If you have selected the option Replace complete structure, the following dialogue appears:



The dialogue shows the complete assembly structure and all associated files.

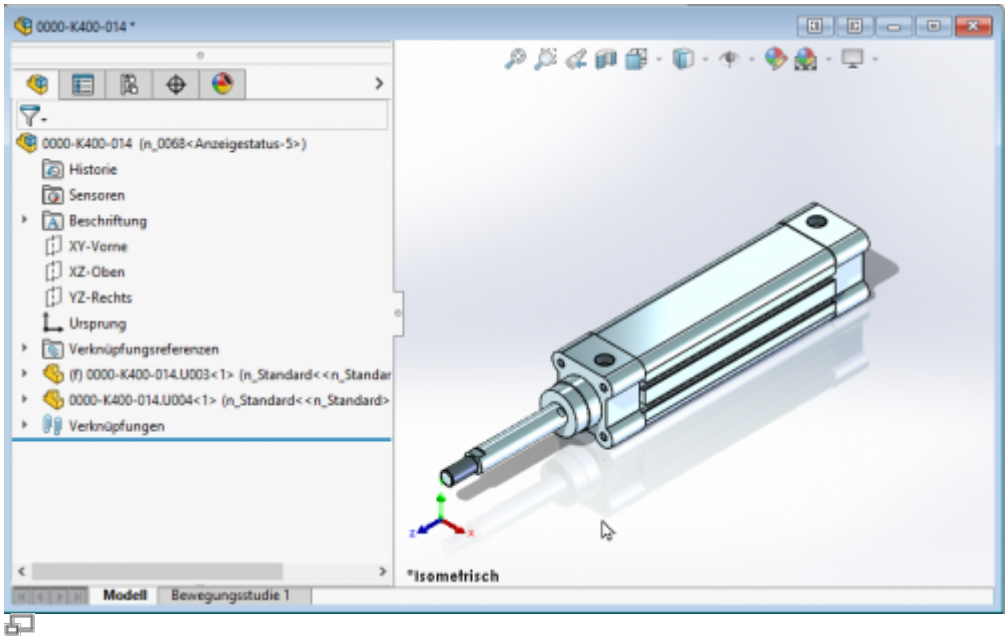
The tree structure distinguishes the following features in the display:

- Entries in green font represent an already known speedy document.
- Entries in blue font represent an unknown local file.
- Entries in red font represent a file that does not exist.
- Entries in grey font represent an additional file with the same name as a main entry.
- **Bold** Entries written in bold are main entries for which there may be additional files with the same name.

Purchased parts/U-Parts

Frequently, purchased, standard or customer parts must be described with the help of an assembly file and several parts files. A typical example is a pneumatic cylinder. In order to be able to use movement or different installation states, an assembly file with corresponding part files must be constructed. However, the administration is only interested in the cylinder as a whole.
For this purpose, either all document types that do not have a parts list are treated as a „parts“ document type or the document types are enumerated with the configuration parameter [inventor.partrename.partdoctypes].

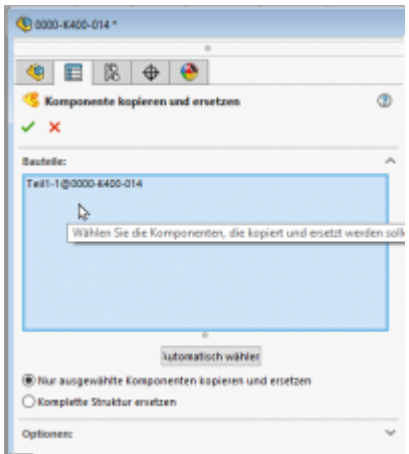
If an assembly file is now stored in SOLIDWORKS in speedy and a „parts“ document type is selected in the New dialogue, all components are automatically checked-in as U-parts:



Purchased part cylinder with U-parts

Add more U-parts

If you want to add more U-parts to an existing purchased part, add the component to the SOLIDWORKS assembly using the usual means. Now use the speedy „Copy and Replace Component“ command.



Select the component and check „Copy and replace selected components only“.
The component is now automatically added to the speedy document as a U-part and renamed accordingly.

Replace U-parts

If you need to replace certain U-parts in an existing purchased part, open the SOLIDWORKS assembly for editing.
Now replace the desired U-parts with the new components.
Now use the speedy command „Copy and replace component“.
Select the component and do not mark anything else in the dialogue.
The component is now automatically added to the speedy document as a U-part and renamed accordingly.

Convert assembly to purchased part

To convert an assembly with managed components into a purchased part with U-parts, carry out the following steps:

- Open the assembly in SOLIDWORKS.
- Save as command.
- In the New dialogue, select a document type „K - Purchase Part“, for example.
- The new purchased part item is created.
- The components are converted into U-parts.

Convert purchased part to assembly

To convert a purchased part with U-parts into an assembly with managed components perform the following steps:

- Open the purchased part (assembly with U-parts).
- Command „Save as“.
- In the New dialogue select e.g. the document type „B - Assembly“.
- The assembly part is created.
- Command „Copy and replace component“.
- Select the U-parts.
- Activate the option „Copy and replace selected components only“.
- The new dialogue appears for each U-part.

Replace component with current version

Replaces the selected component with the latest version of the item.

Part Usage

Displays the part usage of the current file or the selected component.

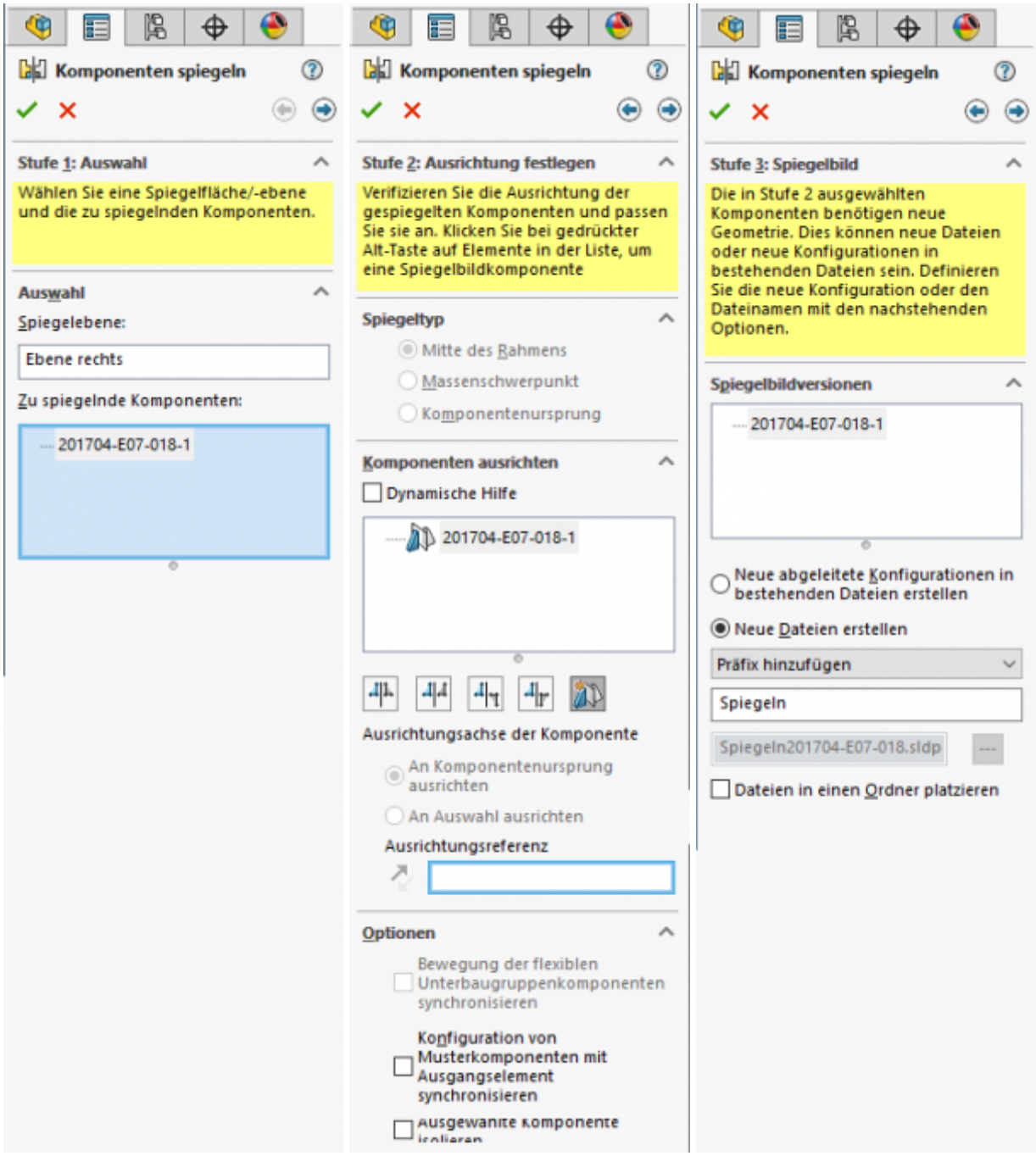
Reference Manager

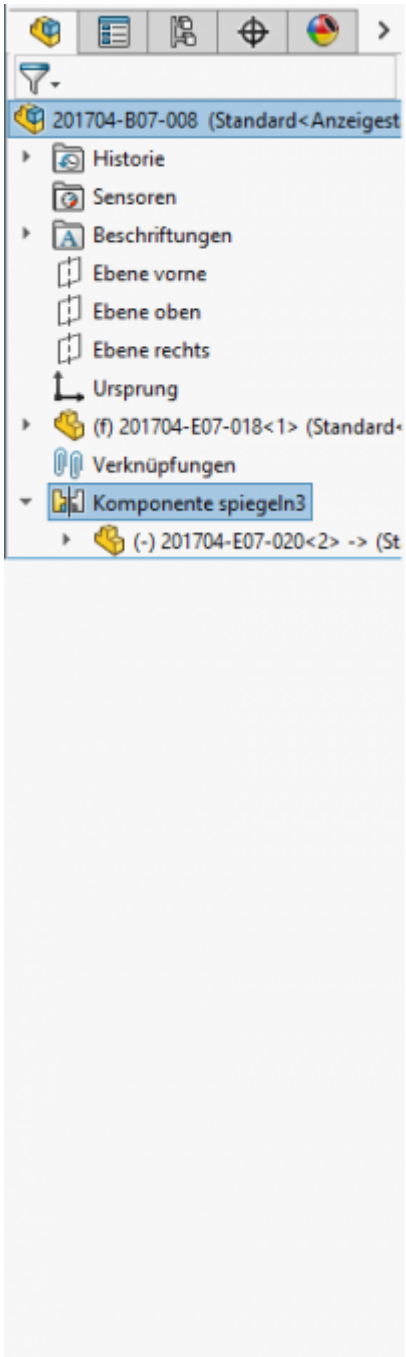
Opens the reference manager for the current assembly file (see also Reference Manager)

Mirror

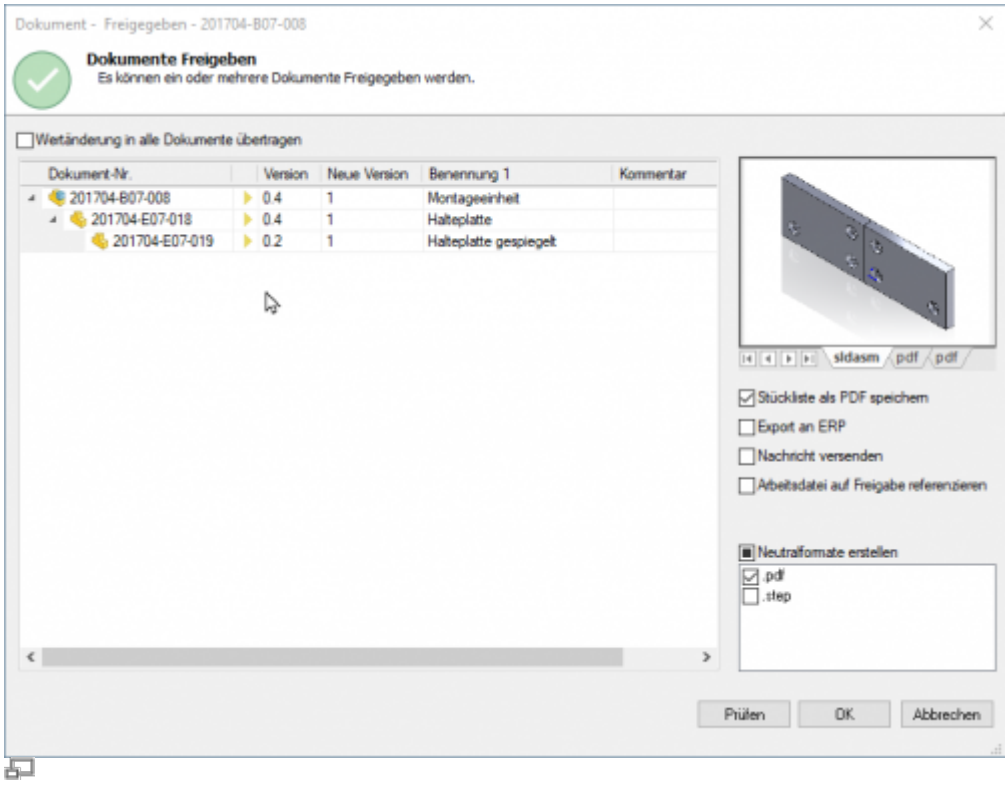
speedyPDM supports mirroring of individual parts as well as assemblies. speedyPDM intervenes in the Solidworks save dialogue.

Use the known mechanisms for mirroring parts or assemblies in Solidworks:





An article is also created for the mirrored part in speedy.
The mirrored part is linked to the original part in speedy (relation MIRRORED).
This ensures that when the original part is released, the mirrored part is also released.



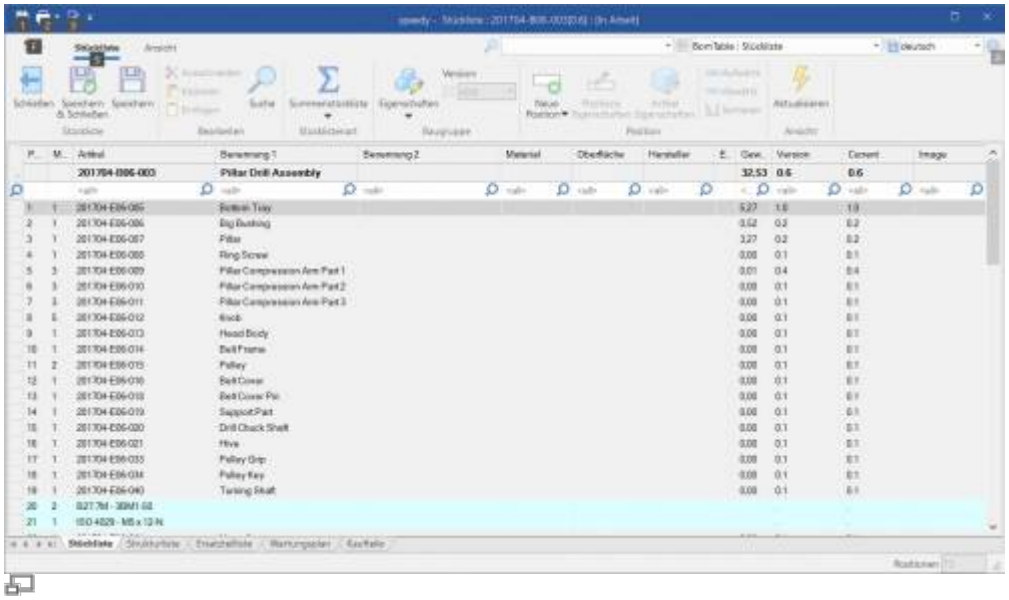
Release of mirrored parts

Please note:
When an original part is changed, the mirrored part is also taken into work.
To ensure that the changes are also applied to the mirrored part, you should also open the mirrored part.

Releasing a mirrored part does not release the original part.
Mirrored parts should not be released directly. Always release only the original part.

Edit parts list

Opens the parts list dialogue with the current assembly parts list.
If there is no BOM yet, the BOM is created automatically.
Menu speedy “ Edit parts list
Edit Bill of Materials Toolbox
Context menu assembly „ Edit parts list



To update the speedy parts list, the Solidworks assembly is analysed.

The following Solidworks settings are taken into account:

- Suppressed components are not included in the parts list.
- Components that are located in a defined feature container are not included in the parts list. Administratively, names can be defined for feature containers that are excluded from the BOM.
- Component property „Exclude from BOM“.
- Configuration property „Do not show child components in BOMs if used as subassembly.

Assembly and drawing opened:
If you have opened the assembly and the corresponding drawing, you must save changes in the assembly before switching to the drawing.
When you save, the speedy parts list is updated. This ensures that BOM changes are immediately visible in the drawing.

Search parts list item

Search for an item by item number or document number.

Insert text

Inserts a text into a drawing, which is replaced according to the translation when the language is switched.

Switch language

Switches the language in the drawing according to the set project language.
The texts in the title block and manually inserted texts are switched.

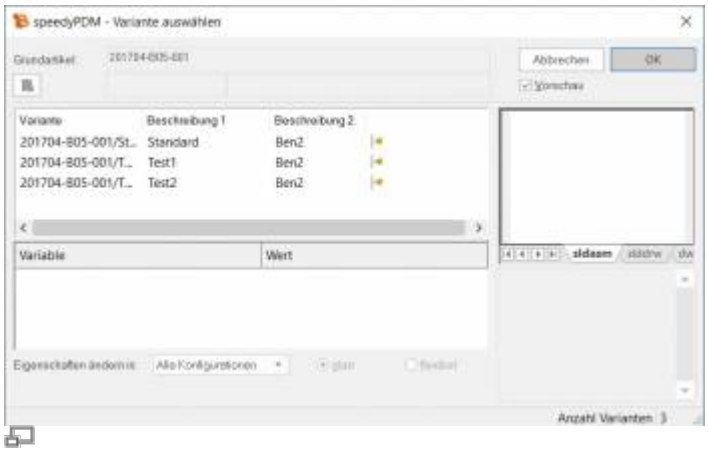
Customer-specific title blocks defined in the project settings are also switched.

Fix component

Moves the selected component to the assembly origin and fixes it there.

Select/Edit Variant

In the Select Variant dialogue you can select an existing Solidworks configuration or, if the item is a variant item, create a new variant.
Menu speedy “ Edit Variant
Edit Variant Toolbox
Context menu part/assembly “ Edit variant



Create new variant:
To create a new variant, select the entry „New variant“ in the list or click the button. In the variable list, all variant variables in the Value column are now filled with „???“. Define the respective variable values. If you have entered the last variable, speedy searches whether a variant with the entered values already exists and marks it; otherwise, a new variant with the values is created.

Save as

Saves the active file in an additional file format.
The additional file is checked-in with the same file name in the same directory as the current file.
Depending on the opened file type (sldprt, sldasm, smddrw), different storage formats are available.

User Options

Opens the dialogue for editing the user options.

Reset Messages/Errors/Warnings

Some warnings, error messages and user queries can be deactivated by selecting an option „Do not ask this question again“ or „Do not report this notice again“ in the message dialogue. These deactivated messages can be reactivated using this command. A dialogue with all deactivated messages appears. Select all messages that are to be reactivated, i.e. that are to reappear the next time a message is displayed, and confirm the dialogue with OK.

Material / Mass properties

Material can be exchanged bidirectionally from speedy.
The mass properties (weight, volume, …) are normally only read by speedy. The weight is an exception.

Case 1:

- The mass properties are determined automatically by Solidworks, depending on the material or its density.
- In speedy the item property WEIGHT is read-only.
- ⇒ The weight is read in from the CAD model and saved in the article master.

Case 2:

- The item property WEIGHT is not write-protected.
- No material is defined in the article master.
- ⇒ The weight is transferred from speedy to the model and the property „Overwrite mass properties“ is set.

Case 3:

- The item property WEIGHT is not write-protected.
- A material is defined in the item master.
- ⇒ The weight is read from the model into the item master.
- ⇒ If the weight is changed manually in speedy, this is transferred to the model. The property „Overwrite mass properties“ is set.
- ⇒ If the material in speedy changes, the property „Overwrite mass properties“ is deactivated again and Solidworks calculates automatically again.

Installation

System requirements

An addin is available for the following SOLIDWORKS versions:

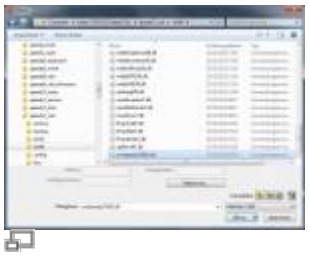
SOLIDWORKS version	speedy Addin
SOLIDWORKS 2016	swxSpeedy72016.dll (x64) (Support ends in 2020)

SOLIDWORKS version	speedy Addin
SOLIDWORKS 2017	swxSpeedy72017.dll (x64) (support ends 2021)
SOLIDWORKS 2018	swxSpeedy72018.dll (x64) (support ends 2022)
SOLIDWORKS 2019	swxSpeedy72019.dll (x64) (support ends 2023)
SOLIDWORKS 2020	swxSpeedy72020.dll (x64)
SOLIDWORKS 2021	swxSpeedy72021.dll (x64)
SOLIDWORKS 2022	swxSpeedy72022.dll (x64)
SOLIDWORKS 2023	swxSpeedy72023.dll (x64)

Register additional application

You need administrative rights to register the add-on application in Solidworks.
Since Windows 7, you should note that when executing „As Administrator“ commands, the linked network drives are normally not available. References to files must always be made via UNC specification (\\servername\release\...) .

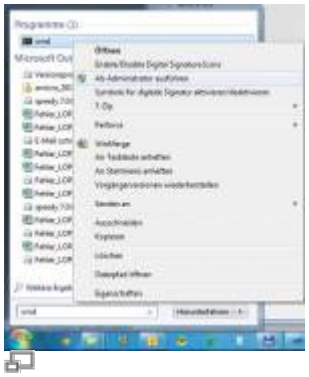
Variant 1: Within Solidworks



You have administrative rights on the workstation. Start Solidworks and select the command „Open file“. In the open dialogue, set the file type filter „Add-Ins (*.dll)“ and select the file swxSpeedy72023.dll in the speedy directory.

Note that for 32 bit Solidworks the add-in file has to be selected from the speedy directory bin32 and for 64 bit Solidworks from the bin64 directory.
If the registration was successful, either speedy prompts you to log in directly or you have to activate it in the list of additional applications.

Variant 2: Manual



If all the above variants do not work, the registration should be done via console. For this purpose, a console window „As Administrator“ must be executed.

In the console window, carry out the registration by transferring the add-in module.

```
regsvr32.exe  \\<server>\speedy_bin$\bin64\swxSpeedy72023.dll
```

Variant 3: Batch file

Depending on the operating system, start the batch file register_solidworks.cmd in the speedy programme directory (bin32 or bin64). The Solidworks Add-In is registered with the operating system. From Windows 7 onwards, you should run the batch file „as administrator“.

Administration

File Properties

The assignment of speedy/PDM document properties to file properties is done with the help of a configuration and assignment file (see also Konfiguration).

Assignment of document properties to file properties

Assignment of document properties to file properties is done in the configuration file **swxSpeedy.ini**.

Toolbox parts in parts lists

Assignment of file properties of Toolbox parts to BOM properties is done in the configuration file **swxToolbox.ini**. If toolbox parts are not managed in speedy, the file properties must be assigned to speedy properties so that the toolbox parts can be entered in the speedy parts list. In order for a toolbox part to be transferred from Solidworks to speedy, the item must either already be a managed document or a property must be assigned as a document number (dm_docno).

The following special properties are available:

swx.partnumber	Either the recognised document number (dm_docno) because the component is already managed or file name of the component.
swx.component.name	Name of the component.
swx.component.title	File name of the component
swx.component.reference	Reference value of the component
swx.configuration.name	Configuration name of the component
swx.configuration.comment	Comment of the configuration of the component
swx.configuration.alternatename	Alternative name of the configuration
swx.configuration.usealternatename	Use alternative name Yes/No

```
[ATTRIBUTE_MAP]
dm_docno  = "swx.configuration.name"
BEN1      = "swx.component.title"
```

Cutting list as parts list

The assignment of properties from the cutting list of weldments to parts list properties is done in the file **swxWeldment.ini**.

```
[ATTRIBUTE_MAP]
dm_docno          = "BESCHREIBUNG"
BEN1              = "Beschreibung"
LÄNGE             = "LÄNGE"
WINKEL1           = "WINKEL1"
WINKEL2           = "WINKEL2"
MATERIAL          = "MATERIAL"
```

Configuration parameters

solidworks.

Key	Description
solidworks.autoset.colour	The colour set on the material (class) is set to the SWX body.
solidworks.autoset.hatch	The hatch set on the material is assigned to the part.
solidworks.autoset.material	Defines whether the material should be set automatically when opening.
solidworks.bomposition.deletenothandled	Defines whether parts list symbols should be deleted from undefined positions. There is a user query before deletion.
solidworks.bomposition.autodeletenothandled	Defines whether balloons that are not in the BOM are automatically deleted.
solidworks.bomtable.autocreate	Defines whether speedy should automatically create a BOM when saving.
solidworks.bomtable.autoinsert	Defines whether a (invisible) BOM is automatically inserted into Solidworks drawings.
solidworks.bomtable.autoupdate	Defines whether a (invisible) parts list is automatically updated in Solidworks drawings.
solidworks.bomtable.autosave	All changes are saved automatically.
solidworks.bomtable.autosorted	The parts list is always sorted automatically.
solidworks.bomtable.autofreepos	Number of free positions between the individual sort criteria.
solidworks.bomtable.comparereference	When collecting the components, also use the component reference to find the same positions. Default value:=0

Key	Description
solidworks.bomtable.configignore	Defines whether parts lists are updated if the first configuration is not active or a derived configuration is active for variant assemblies. 0 := BOMs are only updated if the first configuration is active. It is not updated if a derived configuration is active. 1 := BOMs are always updated.
solidworks.bomtable.featuresorted	Sorting of the parts list corresponds to the order in the feature tree of the assembly. There is no automatic sorting. **Do not activate this parameter if BOM items have received additional item properties (extension of table [dm_bpos_prop]).**
solidworks.bomtable.hidden	Defines whether a BOM should be invisible in SolidWorks or not.
solidworks.bomtable.bomtype	Part list type of the Solidworks part list 1 := Add only single parts to the BOM 2 := Only the top level 3 := Parts list with indentation see also [solidworks.bomtable.showchildcomponents].
solidworks.bomtable.indentednumberingtype	Type of numbering for indented BOMs
solidworks.bomtable.detailedcutlist	Displays detailed cut lists for indented BOMs.
solidworks.bomtable.anchortype	BOM anchor point 1 := Top left 2 := Top right 3 := Left bottom 4 := Right bottom
solidworks.bomtable.useranchorpoint	Defines whether an anchor point in the drawing frame should be used or the specified insertion coordinates.
solidworks.bomtable.insertpointx	Insertion point for „invisible“ parts list.
solidworks.bomtable.insertpointy	Insertion point for „invisible“ BOM.
solidworks.bomtable.includehidden	Invisible parts in SolidWorks are also inserted into the BOM; if not defined bom.includehidden is used.
solidworks.bomtable.includesuppressed	Include suppressed parts in SolidWorks in the BOM; if not defined bom.includesuppressed is used.
solidworks.bomtable.folderignore	Defines whether the components (see solidworks.bomtable.foldernames) are ignored in the BOM determination.
solidworks.bomtable.foldernames	Components that have been assigned to a feature folder are not included in the BOM if the folder name has been defined in the list. Enumeration of folder names separated by semicolons.
solidworks.bomtable.pathnames	Components that originate from one of the specified directories are not included in the parts list.
solidworks.bomtable.pathignore	Defines whether the components (see solidworks.bomtable.pathnames) are ignored in the BOM determination.

Key	Description
solidworks.bomtable.showchildcomponents	Specifies whether the child components should be considered at all when determining the SolidWorks BOM. If the parameter is set, the parts list table in the drawing is always created with the option „Parts only“. see also [solidworks.bomtable.bomtype].
solidworks.bomtable.updateconfigurations	Defines whether when updating the parts list it is updated for all configurations or only for the current one.
solidworks.bomtable.unitcheck	When adding up BOM items (components), note the unit of the respective item. If the unit assigned to the article is not activated for summation, the BOM items are not summed up either, i.e. each component receives its own item in the BOM. See also [document.property.unit] or [solidworks.variable.quantity].
solidworks.bomtable.summarize	Sum up BOM items when reading in components. Default value:=1
solidworks.bomtable.template	Template of the invisible Solidworks BOM.
solidworks.bomtable.traverschildcomponents	Defines whether subassemblies are included in the parts list. In the subassembly in the configuration „One level up“ must be activated or the configuration parameter [solidworks.bomtable.showchildcomponents] must be activated.
solidworks.bomtable.useranchorpoint	Defines whether an anchor point in the drawing frame should be used or the specified insertion coordinates.
solidworks.checkin.firstsheet	Always activate the 1st sheet when checking in a drawing.
solidworks.checkin.forcesave	Specifies that when saving + closing a save is forced, regardless of whether something has been changed or not.
solidworks.checkin.shownamedview	Default views when saving 1 := Front side 2 := Back side 3 := Left 4 := Right 5 := Top 6 := Bottom 7 := Isometric 8 := Trimetric 9 := Dimetric
solidworks.checkin.zoomtofit	Zoom on save&close view.
solidworks.clipboard.extension	**-**
solidworks.clipboard.previewsize	**-**
solidworks.combinebom.headheight	??
solidworks.combinebom.rowheight	??

Key	Description
solidworks.component.attachinvisble	Controls the behaviour of component files that do not currently have their own editing window. If the configuration parameter is activated: - The addin already connects to components of assemblies loaded in the background when it is opened. - After editing subparts/assemblies, they are not automatically checked in again. With deactivated configuration parameter: - After editing subparts/assemblies, they are automatically checked in again. In the event that the child documents are also to be checked out when parent documents are checked out (relation type: COMPONENT), the parameter is automatically active. Default value:= 0
solidworks.component.handledimchange	Defines whether to check out after a dimension change.
solidworks.component.handlestatechange	Monitoring the status change from reduced to complete or vice versa causes a check-out or check-in.
solidworks.configignore	Defines whether file properties, mass properties and dimensions are updated if the first configuration is not active or a derived configuration is active for variant assemblies. 0 := Properties are only updated if the first configuration is active. It is not updated if a derived configuration is active. 1 := Properties are always updated.
solidworks.configuration.alldimension	Defines whether all dimensions of a configuration should be read in or only the configuration-specific dimensions.
solidworks.configuration.createchilds	Defines whether derived configurations should be created automatically when creating a new configuration.
solidworks.configuration.dlgline1	Defines the content of the 1st line of the configuration wizard.
solidworks.configuration.dlgline2	Defines the content of the 2nd line of the configuration wizard. ✖
solidworks.configuration.forceupdate	Defines whether the update of the variant list is forced when saving.
solidworks.configuration.purgereleased	Delete variants that are no longer needed for previously released items.

Key	Description
solidworks.configuration.options	Configuration options when creating a new configuration. 1 := Use alternative names 2 := Do not show parts in the parts list 4 := Suppressed by default 8 := Invisible by default 16 := Minimum Feature Manager 32 := Inherit features 64 := Link to Parents see SolidWorks Add-Ins API Help (swConfigurationOptions2_e)
solidworks.configuration.onfileopen	Defines whether the configuration dialog should appear when opening variant articles.
solidworks.configuration.prefix	Prefix that is put in front of the configuration name when creating or must be put in front of an uncontrolled variant so that it is taken over by speedy.
solidworks.configuration.preview	File extension for the standard preview in the configuration dialogue.
solidworks.configuration.property.delete	Defines whether the configuration-specific properties are deleted for non-variant parts.
solidworks.configuration.property.import	Also read in configuration-specific properties of the active configuration when importing.
solidworks.configuration.released	Defines whether a new variant can be created for released variants without a new release. If the configuration parameter is activated, the write protection of the release is removed in the background, the variant is created and then the write protection is set again. If the configuration parameter is deactivated, the item is checked out, the variant is created and then the release is granted again.
solidworks.configuration.useoptions	In the „Select Variant“ dialogue, offer the selection option to define whether changes only affect this or all configurations.
solidworks.documentlist.coldescname	Default column description for document list in rollup.
solidworks.docmgr.lickey	License key for Solidworks Document Manager Library.
solidworks.disolvesubassembly	An assembly is automatically disolved after drag & drop if the part number ends with the defined name (suffix).
solidworks.drawingreferencecheck	Checks whether the model references on the 1st sheet of a drawing refer to the model with the same name.
solidworks.dropcreatenew.classification	
solidworks.dropcreatenew.deleteconfiguration	If you specify with drag and drop that the selected variant is created as a new document then the switch says that all other configurations are deleted.

Key	Description
solidworks.dropcreatenew.doctype	Default document type when a new document is created during drag and drop.
solidworks.dropcreatenew.editobject	Defines whether document properties should be edited after creation.
solidworks.dropcreatenew.renameconfiguration	Defines if remaining or active configurations should be renamed when a new document is created during drag and drop.
solidworks.dropcreatenew.specialpart	Defines if the checkbox 'Insert as design part' is active when a part is inserted into an open assembly by drag and drop.
solidworks.dropcreatenew.manual	
solidworks.extensions	List of valid SolidWorks file extensions Optional.
solidworks.filedrop.react	Enables/disables the reaction of speedy to the insertion of components into an assembly.Default value:=1
solidworks.filedrop.useversion	Behaviour when inserting a released model into an assembly: 0 := Inquire (default) 1 := Always insert latest version. 2 := Insert released version of the document.
solidworks.filedrop.ExcludeFromBOM	If „Do not insert into BOM“ is defined for the document type of the inserted item, the component is also excluded from the BOM in Solidworks. This means that the component will also no longer appear in the Solidworks BOM.
solidworks.import.cadenas	
solidworks.langtext.balloonsize	Balloon size in the drawings 0 := Narrowest 1 := 1 character 2 := 2 characters 3 := 3 characters 4 := 4 characters 5 := 5 characters
solidworks.langtext.balloonstyle	Balloon Style 0 := Nothing 1 := Circular 2 := Triangle 3 := Hexagon 4 := Box 5 := Diamond 6 := Circular gap 7 := Pentagon 8 := Flags Pentagon 9 := Flags Triangle 10 := Underlined
solidworks.material.database	File name of the SolidWorks material database (incl. path)
solidworks.material.database.ApplyAppearance	Defines whether the appearance of material should be applied. Default value:=1
solidworks.material.database.ApplyMaterialColorToPart	Defines whether the part colour matches the material colour. Default value:=1

Key	Description
solidworks.material.database.ApplyMaterialHatchToSection	Defines whether the drawing section views use the material hatching. Default value:=1
solidworks.material.synchronize	Writing material back to the item properties can be controlled. 1 := Material is written back (default) 0 := Material is not written back.
solidworks.mirrorname	List of names that SolidWorks uses when mirroring parts. (e.g. mirror;mirror)
solidworks.multipleuse.onsavecheck	Check for multiple use when saving: 0 := do not check 1 := warn and ask user 2 := Automatically save as new part/assembly 3 := Warn and ask user, if answer „save anyway“ do nothing else automatically save as new part/assembly
solidworks.multipleuse.savesilent	???
solidworks.partrename.oninsert	Defines if renumbering should be done when inserting a non-project part into an assembly.
solidworks.partrename.autoondrop	Automatically renumber non-project parts during drag & drop.
solidworks.partrename.completestruct	Defines whether the entire structure of the subassemblies should be processed when renumbering components.
solidworks.partrename.folderignore	Defines whether components that have been assigned to feature folders (see solidworks.partrename.foldernames) should be ignored when determining non-project parts.
solidworks.partrename.foldernames	Components that have been assigned to a feature folder are ignored when determining non-project parts. Enumeration of folder names separated by semicolons.
solidworks.partrename.oninsert	Defines whether the determination of non-project parts should be carried out each time a component is inserted.
solidworks.partrename.partdoctypes	Enumeration of document types to be interpreted as individual parts when deciding whether to create U-parts. If the parameter is empty, all document types for which no BOM has been defined are used as part document types.
solidworks.partrename.pathignore	Defines whether components in defined directories (see solidworks.partrename.pathnames) should be ignored when determining non-project parts.
solidworks.partrename.pathnames	Defines the directories that should be ignored when determining non-project parts. Enumeration of directories separated by semicolons.
solidworks.partrename.silent	??

Key	Description
solidworks.path.currentdirectory	Defines the path that is set when the user chooses save as or open local (default path for SolidWorks files). ATTENTION: This has no effect on the current path when „saving as“ files already managed in speedy (system dependent).
solidworks.path.sheetformat	Search path of the SolidWorks sheet formats. If this path is set, the Solidworks search path for the sheet formats is automatically changed to this path when Solidworks is started.
solidworks.preference.extrefnopromptorsave	See also Solidworks API Help. „Specifies whether to close read-only reference documents without prompting or attempting to save changes made to them when saving or closing the parent document.“
solidworks.preference.extrefopenreadonly	See also Solidworks API Help. „Specifies whether to open referenced documents as read-only.“ For this to work, the property format must be set to **Check out parent document requires check out child component** must not be set.
solidworks.preference.extrefupdatecompnames	See also Solidworks API Help. „Specifies whether to mate to read-only parts using internal face IDs of the parts or to replace components using surface identifiers to write-access parts.“
solidworks.propertyview.autoswitch	Defines that the Attachment Parts tab is switched active if the document has attachment parts. If there are variants, the Variants tab is switched active (SolidWorks Rollup).
solidworks.propertyview.configurations	
solidworks.propertyview.defaulttab	The system remembers the last active tab for the next programme call.
solidworks.propertyview.document	Defines if the tab Document is displayed (In the speedy rollup window in SolidWorks).
solidworks.propertyview.files	Defines whether the Files tab is displayed.
solidworks.propertyview.library	Defines whether the Attachments tab is displayed.
solidworks.propertyview.properties	Defines whether the Properties tab is displayed.
solidworks.propertyview.relations	Defines if the tab Relations is displayed.
solidworks.propertyview.revisions	Defines whether the tab Versions is displayed.
solidworks.relation.forceupdate	Defines if the relations should always be updated when saving a SolidWorks assembly.
solidworks.relation.includesuppressed	Save relations of suppressed components.
solidworks.relation.includeenvelope	Store relations to envelope components.

Key	Description
solidworks.revision.count	Number of revision entries in the revision table.
solidworks.revision.first	First revision to display. A value of 1 means the revision table starts with the 1st release. Default value := 1
solidworks.revision.firstline	Write only 1st line of the revision comment into the revision table.
solidworks.revision.revlabel	Format the revision label in the revision index. see also [session.revlabel]
solidworks.revisiontable	Defines whether a Solidworks revision table is filled in/created.
solidworks.revisiontable.column.approved	Column name „Approved“ for version table
solidworks.revisiontable.column.date	Column name „Date“ for version table
solidworks.revisiontable.column.description	Column name „Description“ for version table
solidworks.revisiontable.column.revision	Column name „Revision“ for version table
solidworks.revisiontable.column.zone	Column name „Zone“ for version table
solidworks.revisiontable.hidden	Revision table invisible yes/no
solidworks.revisiontable.anchortype	see Solidworks API ISheet::InsertRevisionTable2.
solidworks.revisiontable.autoupdatezoomcells	
solidworks.revisiontable.insertpointx	
solidworks.revisiontable.insertpointy	
solidworks.revisiontable.symbolshape	
solidworks.revisiontable.useranchorpoint	
solidworks.revisiontable.template	
solidworks.rollup.autorefresh	Defines whether the document list should be refreshed automatically when the rollup window is opened.
solidworks.rollup.delay	Defines the delay time for closing the rollup window in 250ms steps.
solidworks.rollup.delta	XY coordinate offset when positioning the rollup window with respect to the upper right corner of Solidworks.
solidworks.rollup.dockable	Show the speedy window either as a dockable task window or as a rollup window.
solidworks.rollup.unrolltimer	Defines the delay time for opening the rollup window in 250ms steps.
solidworks.rollup.show	Show rollup window yes/no.
solidworks.route.assembly.subfile	Create routing assemblies as U-parts.
solidworks.route.parts.subfile	Create routing parts as U-parts.
solidworks.savebmp.autosave	
solidworks.savebmp.overwrite	Defines whether an existing .bmp file should be overwritten.
solidworks.savedxf.overwrite	Defines whether an existing .dxf file should be overwritten.
solidworks.savedxf.autosave	Defines whether a DXF file is automatically created when „Save + Close“ is selected.
solidworks.savedxf.autosave.slddrw	Defines whether a DXF file is automatically created/updated for a drawing when „Save + Close“ is selected.

Key	Description
solidworks.savedxf.autosave.sldprt	Defines whether a DXF file is automatically created/updated for a part when „Save + Close“ is selected.
solidworks.savedxf.modelviews	An enumeration of model view names to be exported when saving a part as DXF/DWG. If the parameter is not defined the default views (swFrontView, swLeftView, swTopView) are exported. The enumeration may contain numbers for the predefined model views: - swFrontView = 1, - swBackView = 2, - swLeftView = 3, - swRightView = 4, - swTopView = 5, - swBottomView = 6, - swIsometricView = 7, - swTrimetricView = 8, - swDimetricView = 9 (e.g.: „1;4;5;7“ ⇒ The views swFrontView, swRightView,swTopView and swIsometricView are exported). Furthermore, the enumeration can contain names of model views to be defined by the user. The names can be described with wildcard characters. (e.g.: „Export*;Lasern“) ⇒ All views are exported that begin with „Export“ or are called „Lasern“. This parameter does NOT work for sheet metal parts.
solidworks.savedxf.persheet	Defines if all sheets of a drawing are saved in a DXF/DWG or if one DXF/DWG is saved per sheet.
solidworks.savedxf.sheetmetaloptions	Defines which lines/elements are saved additionally when saving unfoldings. 1 := to export flat-pattern geometry 2 := to include hidden edges 4 := to export bend lines 8 := to include sketches 13 := to include bend lines, flat-pattern geometry and sketches 16 := to merge coplanar faces 32 := to export library features 64 := to export forming tools 2048 := to export bounding box See also Solidworks API „ExportToDWG2“ → „SheetMetalOptions“. Default value := 1
solidworks.saveedrawing.overwrite	Defines if an already existing .edrawing file should be overwritten.
solidworks.saveedrawing.autosave	Defines whether a new .edrawing file is automatically created/updated for the document when „Save + Close“ is selected.

Key	Description
solidworks.saveedrawing.autosave.sldprt	Defines whether an .edrawing file is automatically created/updated when individual parts are „saved + closed“.
solidworks.saveedrawing.autosave.sldasm	Defines whether an .edrawing file is automatically created/updated when „saving + closing“ assemblies.
solidworks.saveedrawing.autosave.slddrw	Defines whether an .edrawing file is automatically created/updated when „saving + closing“ drawings.
solidworks.saveigs.overwrite	Defines whether an already existing .igs file should be overwritten.
solidworks.saveigs.autosave	Defines whether a new .igs file is automatically created/updated for the document when „Save + Close“ is selected.
solidworks.saveigs.autosave.sldasm	Defines whether a .igs file is automatically created/updated when assemblies are „saved + closed“.
solidworks.saveigs.autosave.sldprt	Defines whether an .igs file is automatically created/updated when „saving + closing“ individual parts.
solidworks.savejpg.overwrite	Defines whether an existing .jpg file should be overwritten.
solidworks.savejpg.autosave	Defines whether a new .jpg file is automatically created for the drawing or an older version is overwritten when saving + closing.
solidworks.savepdf.autosave	Defines whether a PDF file is automatically created/updated for the drawing when „save + close“.
solidworks.savepdf.overwrite	Defines whether an existing .pdf file should be overwritten.
solidworks.savesat.overwrite	Defines whether an existing .sat file should be overwritten.
solidworks.savesat.autosave.sldasm	Defines whether a .sat file is automatically created/updated when assemblies are „saved + closed“.
solidworks.savesat.autosave.sldprt	Defines whether a .sat file is automatically created/updated when „saving + closing“ individual parts.
solidworks.savestep.overwrite	Defines whether an existing .step file should be overwritten.
solidworks.savestep.autosave.sldasm	Defines whether a .step file is automatically created/updated when „saving + closing“ assemblies.
solidworks.savestep.autosave.sldprt	Defines whether a .step file is automatically created/updated when „saving + closing“ individual parts.
solidworks.savestep.version	Defines the step version for saving (either 203 or 214).
solidworks.savestl.overwrite	Defines whether an existing .stl file should be overwritten.

Key	Description
solidworks.savestl.autosave.sldasm	Defines whether a .stl file is automatically created/updated when „saving + closing“ assemblies.
solidworks.savestl.autosave.sldprt	Defines whether a .stl file is automatically created/updated when „saving + closing“ individual parts.
solidworks.savetif.autosave	
solidworks.savetif.capture	Tiff screen or print size 0 := Screen size 1 := Print size
solidworks.savetif.compressionscheme	Compression Scheme 0 := Tiff Uncompressed 1 := Tiff Packbits Compression 2 := Tiff Group 4 Fax Compression
solidworks.savetif.imagetype	Tiff Image Type 0 := Tiff Black / White Image 1 := Tiff RGB image
solidworks.savetif.overwrite	Defines whether an existing .tif file should be overwritten.
solidworks.savetif.papersize	swTiffPrintPaperSize Tiff paper size Special values for drawing paper sizes 0 := Paper: Size A 1 := Paper: Size A Vertical 2 := Paper: Size B 3 := Paper: Size C 4 := Paper: Size D 5 := Paper: Size E 6 := paper: size A4 7 := Paper: Size A4 Vertical 8 := Paper: Size A3 9 := Paper: Size A2\\10 := Paper: Size A1 11 := Paper: Size A0 12 := Custom Paper Size
solidworks.savetif.resolution	Tiff Print DPI 0 := Drawing only 1 :=Tiff screen or print size must be set to 1.
solidworks.savetif.scalefactor	Tiff print scale factor 0 := Scale factor: Integer value
solidworks.savetif.scaletofit	See also Solidworks API Help. swTiffPrintScaleToFit „For drawings only; specifies whether to scale drawing to fit page. „
solidworks.script.shutdown	Defines an executable file to run on shutdown (e.g. .batch, .exe)
solidworks.script.startup	Defines an executable file to run at startup (e.g. .batch, .exe)
solidworks.teamwork	Defines whether the teamwork function is active.
solidworks.template.dina0	**-**
solidworks.template.dina1	**-**
solidworks.template.dina2	**-**
solidworks.template.dina3	**-**

Key	Description
solidworks.template.dina4h	**-**
solidworks.template.dina4q	**-**
solidworks.template.npart_block	**-**
solidworks.template.vstl_head	**-**
solidworks.template.vstl_pos	**-**
solidworks.toolboxpartfolder	Defines the base directory of the toolbox parts.
solidworks.variable.dynamic	Prefix for dynamic variables in variant construction. With the help of the formula „swx_dynamic“ dynamic variables can be calculated depending on the other variables when creating a new variant.
solidworks.variable.needprefix	Variants are only created if dimension variables with a prefix are required and also exist accordingly. I.e. no variants are created from models without corresponding dimension variables. Default value:=0
solidworks.variable.prefix	Defines the prefix that must be placed in front of variables in order for them to be taken into account by speedy. Default value:=var_
solidworks.variable.prefix1	**-**
solidworks.variable.quantity	Defines a quantity variable that is used to determine the quantity of a variant within an assembly. Default value:=var_quantity
solidworks.variant.filetype	– sonplas VIS –
solidworks.variant.table.fixedcolumns	Enumeration of fixed columns Column header1=Document property; Column heading2=document property; The enumerations are separated by a semicolon.
solidworks.variant.table.title	Defines the title of the variant table (e.g. variant table).
solidworks.variant.table.anchortype	Anchor point of the variant table 1 := Top left 2 := Top right 3 := Left bottom 4 := Right bottom
solidworks.variant.table.useranchorpoint	Defines whether either the coordinates or the anchor point in the variant table should be used.
solidworks.variant.table.insertpointx	Insert coordinates
solidworks.variant.table.insertpointy	Insert coordinates
solidworks.view.internal	Defines that SolidWorks files are opened for viewing in SolidWorks regardless of which viewer is defined in speedy.

Key	Description
document.default.container.solidworks	This parameter defines if the working directory of the SolidWorks plugin is stored separately so that it is independent from the other modules. **This parameter must be added manually by the administrator.**

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