

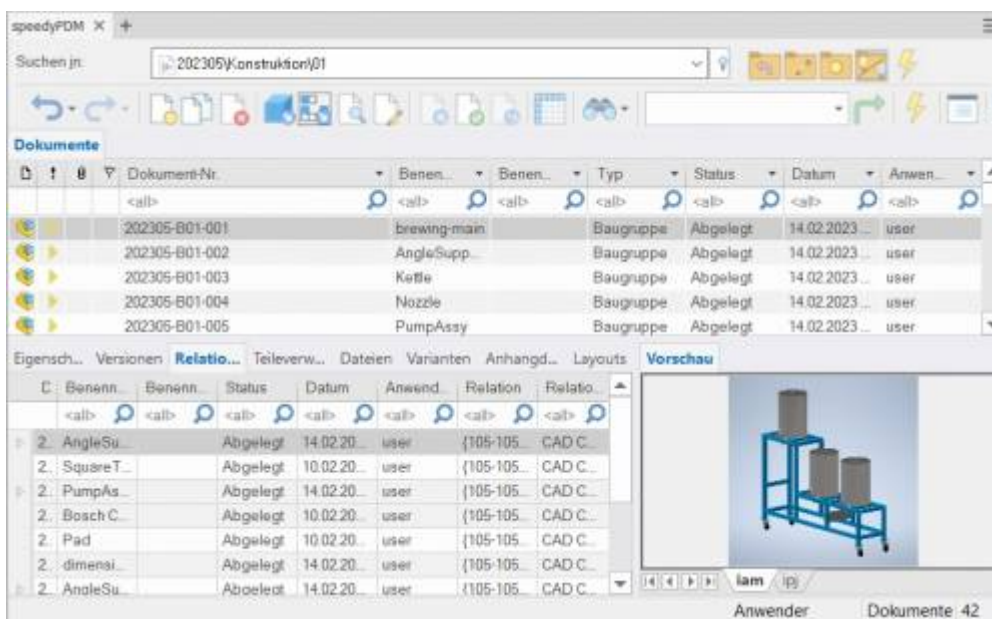
Solid Edge

The integration of speedyPDM in Solid Edge ® takes place as an addin.

The speedyPDM Solid Edge addin provides a roll-up window in addition to the usual menus and toolboxes.

Rollup

The rollup window is positioned above the title bar of SolidEdge and opens automatically when you move the mouse pointer over the title bar.



Rollup window

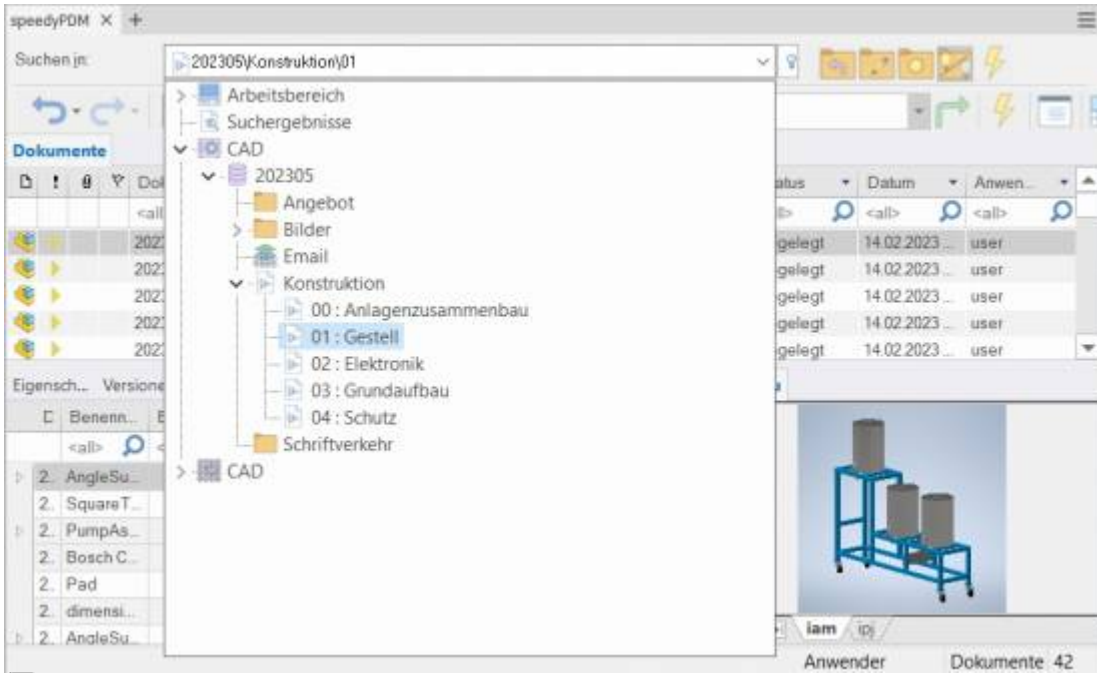
The rollup window closes automatically when you work in SolidEdge.

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.

Alternatively, the roll-up window can also be displayed as a SolidEdge window that can be pinned. In this case, the window can also be positioned freely or pinned to a main window page.

Navigation and project/folder management in the rollup window

The navigation area is located in the upper part of the roll-up window in the form of a drop-down bar. 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 folder 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 folder. If the locker is active in the navigation, a new project folder is created. If a project folder or another folder is active, a new subfolder is created.
	Edit folder properties
	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
	File document
	Share document
	Document properties
	View document
	Delete document
	Refresh view
	Search document
	User options

Create new document

There are several options to choose from:

SolidEdge:

- You create a new file with the SolidEdge command File „ New.
- You create the model or assembly using the usual SolidEdge methods.
- You select the SolidEdge 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 file

There are several options to choose from:

SolidEdge:

- Select the SolidEdge 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

Ribbon speedy “ Open drawing Context menu „ Open drawing

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.

Selecting a component in an assembly opens the drawing of the selected component.

Open component

Ribbon speedy “ Open model Context menu assembly „ Open model

If you have opened an assembly and want to open the model of a component or a subassembly for editing, you must use the speedy command **Open Model** command. Parts and sub-assemblies are only loaded in read-only mode when opening an assembly to prevent unnecessary extraction from the speedy database and accidental modification. So when you use the SolidEdge command to open parts, they are opened in read-only mode.

Insert component

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

SolidEdge:

- Select the SolidEdge command Insert “ Place Component „.
- The speedy dialogue Open file 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 into the drawing area of SolidEdge(Drag & Drop) with the left mouse button pressed.
- The part or assembly linked to the document is inserted into the current assembly and „hangs“ on the mouse pointer.

Extract

Extracts the current file or selected component for editing.

If you have opened a file only for viewing, you can use the command **Remove** to put the current file to work.

A selected subcomponent of an assembly is put to work and opened by the command.

Item Properties

Edits the item properties of the current item or the selected component.

After closing the properties dialogue, the iProperties are updated according to the configuration.

If the command is executed in a drawing, the title block is updated.

Save

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

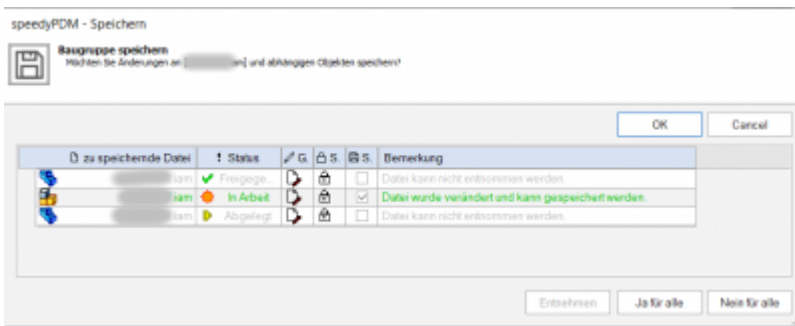
The message can only be confirmed with OK. However, the file is saved in any case. Use the command Rename Component to rename parts that do not belong to the project.

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



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

If a referenced file has been changed in an assembly, an extended dialogue appears with a list of all referenced files:



The list shows the following information:

Column	Heading	Description
1	File to be saved	File name of the referenced file.
2	Status	Current document status (Filed, In progress, Released, ...).
3	Changed	A tooltip explains the type of change in the file.
4	Write-protected	The symbol indicates whether the file is currently write-protected (closed lock) or writable (open lock).
5	Save	Check box for marking the files to be saved. Only files that are also writable can be saved.
6	Comment	Note text or error message.

- By double-clicking on the status symbol in a line, the document is taken in process and can also be saved.
- The command [Remove] takes the document of the selected line into work.
- The command [Yes for all] marks all files for saving that can also be saved (taken by the current user).
- The command [No for all] removes the marking for saving.

If an error occurs with a file during saving, this is logged in the comments column. The dialogue closes automatically after a successful save.

Saving and closing

To close a file managed in speedy, you should select the command Save+Close from the speedymenu.

Ribbon speedy “ Save+Close

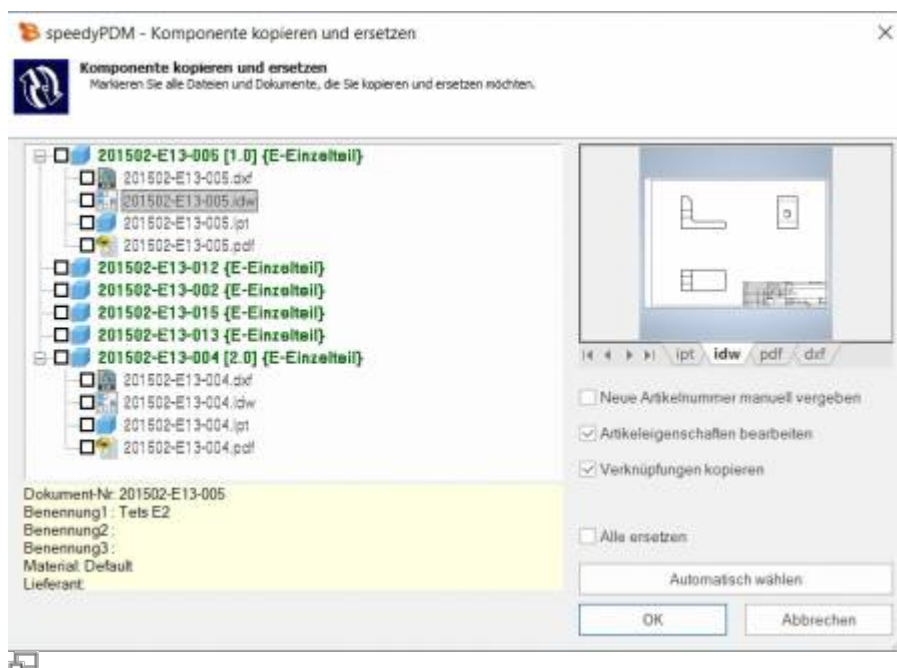
The model is zoomed into the window and an isometric view is set to provide a meaningful file preview, then the file is saved and closed.

Copy and Replace Components

The command „Copy and Replace Components“ is used to rename components that are not part of the project or to add files that are not yet managed.

Ribbon speedy „ Replace component Context menu assembly “ Replace component

The following dialogue appears:



The dialogue shows the complete assembly structure with all referenced files. Select all the components you want to replace.

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.

By pressing the „Select automatically“ button, all non-project components are automatically selected. By confirming the OK button, the marked components are renamed one after the other. Depending on the option „Assign new item number manually“ or „Edit item properties“, further dialogues appear for each new item.

Update component

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.

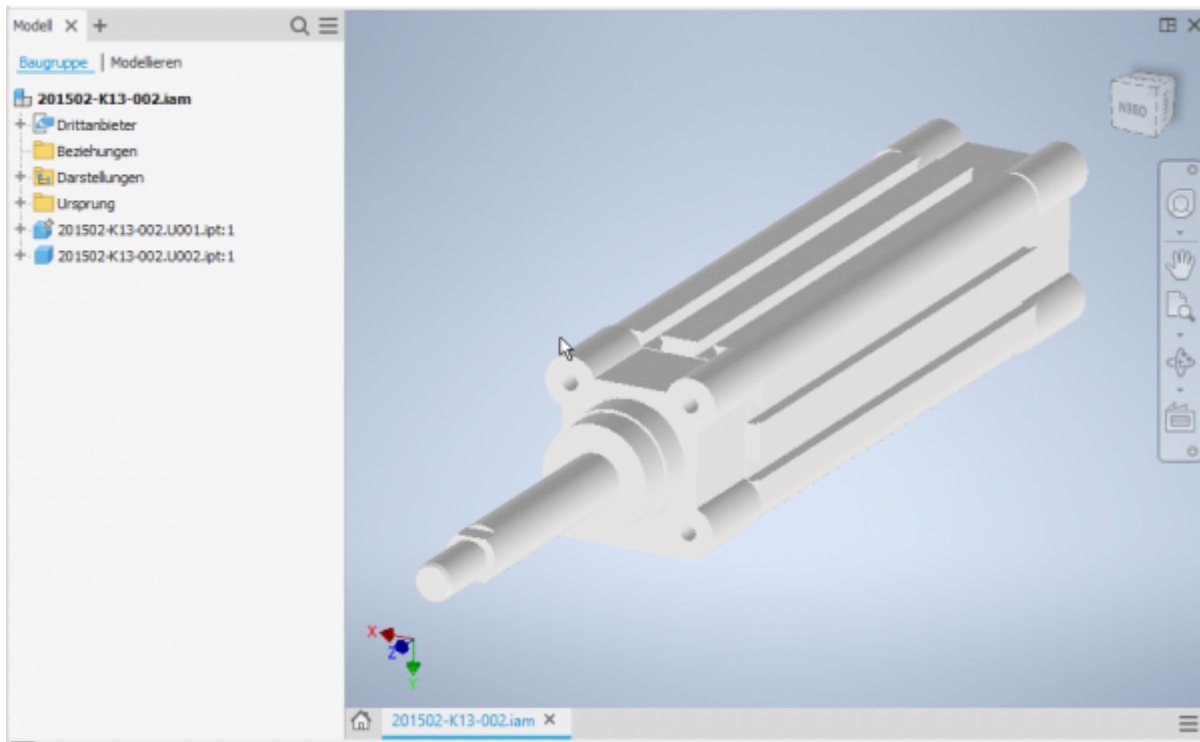
Reference Manager

Opens the reference manager for the current assembly file (see also [Reference Manager](#)).

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 [solidedge.partrename.partdoctypes].

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

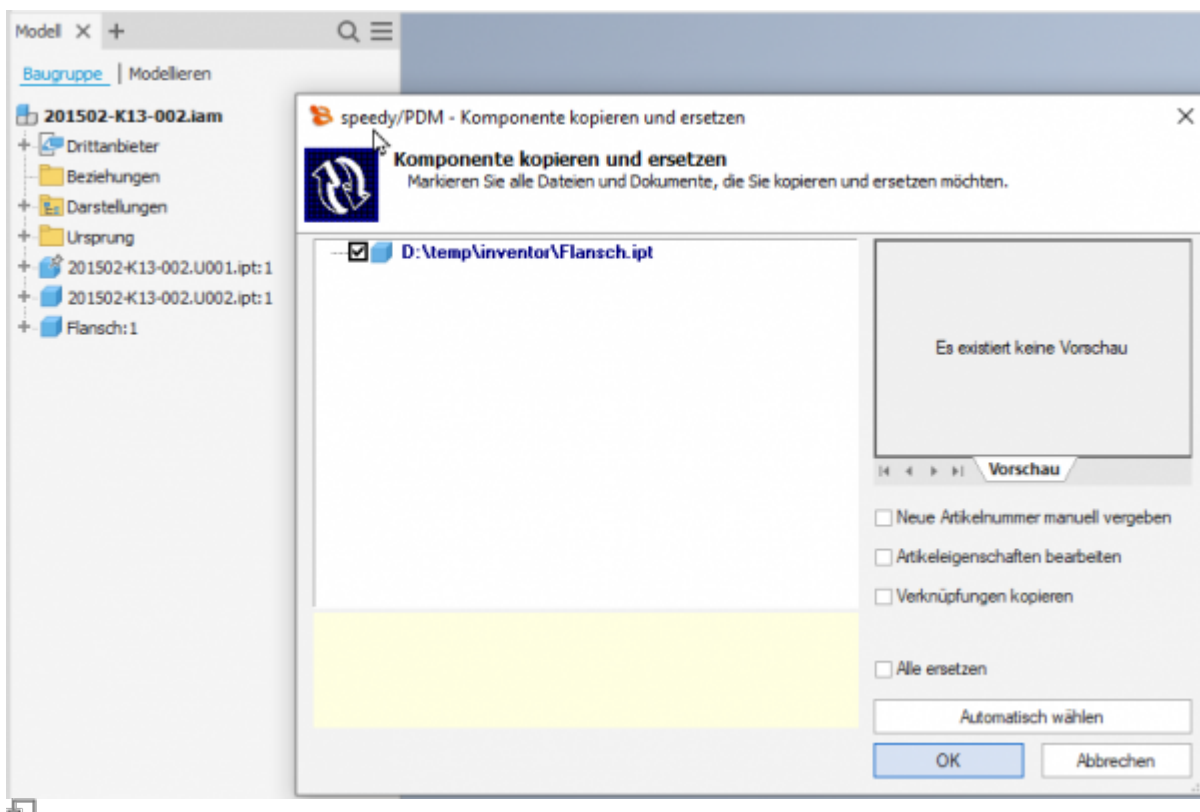


Purchased part cylinder with U-parts

Add more U-parts

If you want to add more U-parts to an existing purchased part, add the component to the SolidEdge assembly using the usual means.

Now use the speedy „Copy and Replace Component“ command.



Select the component and do not select anything else in the dialogue.

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 on an existing purchased part, open the SolidEdge assembly for machining.

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.

Edit parts list

Ribbon speedy „ Edit parts list

Context menu assembly “ 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.

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



If you have opened an assembly in SolidEdge, you should only edit the parts list using the parts list command in SolidEdge.

This is the only way to ensure that changes to the parts list are immediately applied to the open assembly.

Assembly and drawing opened:

If you have opened the assembly and the associated 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.

Save as

Saves the active file in an additional file format.

The additional file is saved with the same file name in the same directory as the current file.

Depending on the opened file type (.par, .asm, .dft), different storage formats are available.

User Options

Opens the dialogue for editing the user options.

Reset messages

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.

Installation

System requirements

An addin is available for the following SolidEdge versions:

SolidEdge Version	speedy Addin
SolidEdge 2022	seSpeedy2022.dll

Register Addin

If you have used the installer for the client workstation the SolidEdge addin will be registered automatically. Manually, the SolidEdge addin can be installed using the batch file register_solidedge.cmd. Start the batch file register_solidedge.cmd from the respective platform subfolder (bin64) of the speedy directory. The batch file always registers the latest SolidEdge addin. If you want to install a specific addin version, you can also register it via command line. To do this, open a command prompt and enter the following command:

```
regsvr32.exe \\<servername>\speedy_bin$\bin64\seSpeedy2022.dll
```

In SolidEdge you can use the Addin Manager to check whether speedy has been registered correctly and to control the loading behaviour.

Client Installation/Update

The installation steps mentioned above are carried out automatically by the client setup programme (speedy-client.exe). Alternatively, you can also call the batch file register_solidedge.cmd after a SolidEdge update or upgrade, for example.

```
x:\speedy\bin64\register_solidedge.cmd
```

Administration

File Properties

The assignment of speedyPDM document properties to file properties is done with the help of a configuration and assignment file (see also [Konfiguration](#)).

The configuration files are located in the subdirectory „\config“.

Assignment of document properties to file properties

The assignment of the document properties to the file properties is done in the configuration file **seSpeedy.ini**.

SolidEdge divides the file properties into several sections:

Name	DisplayName
SummaryInformation	
ExtendedSummaryInformation	
DocumentSummaryInformation	
ProjectInformation	
MechanicalModeling	
Custom	user-specific properties

The user specific properties are mapped normally by attribute mapping. The SolidEdge specific file properties are addressed via the range:

DisplayName	Name
	Title@SummaryInformation
	Subject@SummaryInformation
	Author@SummaryInformation
	Keywords@SummaryInformation
	Comments@SummaryInformation
	Revision number@SummaryInformation
	Last author@SummaryInformation
	Editing time@SummaryInformation
	Last printing date@SummaryInformation
	Date of origin@SummaryInformation
	Last saved@SummaryInformation
	Page number@SummaryInformation
	Word count@SummaryInformation
	Character count@SummaryInformation

DisplayName	Name
	Application name@SummaryInformation
	Data security@SummaryInformation
	Name of Saving Application@ExtendedSummaryInformation
	DocumentID@ExtendedSummaryInformation
	Status@ExtendedSummaryInformation
	Username@ExtendedSummaryInformation
	CreationLocale@ExtendedSummaryInformation
	Status Change Date@ExtendedSummaryInformation
	User Display Name@ExtendedSummaryInformation
	Category@DocumentSummaryInformation
	Presentation Format@DocumentSummaryInformation
	Number of bytes@DocumentSummaryInformation
	Lines@DocumentSummaryInformation
	Paragraphs@DocumentSummaryInformation
	Slides@DocumentSummaryInformation
	Notes@DocumentSummaryInformation
	Hidden Objects@DocumentSummaryInformation
	Multimedia clips@DocumentSummaryInformation
	Manager@DocumentSummaryInformation
	Company@DocumentSummaryInformation
	Document Number@ProjectInformation
	Revision@ProjectInformation
	Project Name@ProjectInformation
Material	Material@MechanicalModeling

Advanced customisation

With the help of the script file „aisSpeedy.vbs“ further adjustments can be made in Autodesk SolidEdge. The script file is located in the speedy subdirectory „\script“ and is also loaded when the addin is loaded.

In the script, it is possible to react to various events in Autodesk SolidEdge and the speedy addin. The following event functions are available:

Event routine	Description
OnInit	Called after the addin has been initialised.
OnFileSave	Called when a file is saved.
OnFileSaveAs	Called when a file is saved.
OnFileCheckIn	Called when saving and closing a document.
OnFileRelease	Called when a document is released.
OnReleaseRecreate_SaveAs	Called when releasing in speedy-Explorer and additional files (e.g.: .pdf) need to be renewed. The function is called separately for each file to be created.
OnSaveAsDWG	Called from the speedy command „Save as DWG“.
OnSaveAsDXF	Called from the speedy command „Save as DXF“.
OnSaveAsBMP	Invoked by the speedy command „Save as BMP“.

Event routine	Description
OnSaveAsJPG	Invoked by the speedy command „Save as JPG“.
OnSaveAsPDF	Invoked by the speedy command „Save as PDF“.
OnSaveAs3DPDF	Invoked by the speedy command „Save as 3D PDF“.
OnSaveAsPNG	Invoked by the speedy command „Save as PDF“.
OnSaveAsX_B	Invoked by the speedy command „Save as X_B“.
OnSaveAsX_T	Invoked by the speedy command „Save as X_T“.
OnSaveAsSAT	Called by the speedy command „Save as SAT“.
OnSaveAsSTEP	Called by the speedy command „Save as STEP“.
OnSaveAsTIF	Invoked by the speedy command „Save as TIF“.

Configuration parameters

solidedge.

Key	Description
solidedge.autoset.material	Defines whether the material should be set automatically when opening.
solidedge.bomtable.autocreate	Defines whether speedy should automatically create a parts list when saving.
solidedge.bomtable.autosave	All changes are saved automatically.
solidedge.bomtable.autosorted	The parts list is always sorted automatically.
solidedge.bomtable.autofreepos	Number of free positions between the individual sort criteria.
solidedge.bomtable.basepartignore	Ignore items from base item folders.
solidedge.bomtable.fixed	The configuration parameter controls the use of item numbers of the SolidEdge parts list. 0 := Item numbers can be changed in speedy depending on the lock in SolidEdge. 1 := All position numbers are interpreted as fixed, i.e. the position number from SolidEdge is always taken over. Position number editing is locked in speedy.
solidedge.bomtable.filterunhandled	Exclude unmanaged files from the parts list.
solidedge.bomtable.pathignore	Defines whether the components (see solidedge.bomtable.pathnames) are ignored when determining the parts list.
solidedge.bomtable.pathnames	Components that originate from one of the specified directories are not included in the parts list.
solidedge.bomtable.summarize	Sum up BOM items when reading the SolidEdge BOM. Default value:=0 SolidEdge already sums up identical parts on its own.
solidedge.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 [solidedge.variable.quantity].
solidedge.checkin.forcesave	Always save when „Save & Close“.

Key	Description
solidedge.checkoutondrag	The document is checked out when it is inserted into an assembly by drag & drop.
solidedge.component.attachinvisible	
solidedge.configuration.forceupdate	Defines whether the update of the variant list is forced when saving.
solidedge.configuration.purgereleased	Delete variants that are no longer needed for previously released items.
solidedge.document.openfiletype	File extensions of the model files (*.asm;*.par)
solidedge.document.openfiletype2	File endings for drawing files (*.dft)
solidedge.documentlist.coldescname	Default column description for document list in rollup
solidedge.dropcreatenew.editobject	
solidedge.extensions	All
solidedge.material.synchronize	Writing material back to the item properties can be controlled. 1 := Material is written back (default) 0 := Material is not written back.
solidedge.multipleuse.onsavecheck	Check for multiple use when saving: 0 := do not check 1 := Warn and consult user 2 := Automatically save as new part/assembly 3 := Warn and ask user, if the answer is „save anyway“ do nothing, otherwise save automatically as a new part/assembly.
solidedge.multipleuse.savesilent	
solidedge.partrename.autoondrop	Automatically renumber non-project parts when dragging and dropping.
solidedge.partrename.completestruct	Defines whether the entire subassembly structure should be processed when renumbering components.
solidedge.partrename.oninsert	Defines whether the determination of non-project parts is to be carried out each time a component is inserted.
solidedge.partrename.partdoctypes	
solidedge.partrename.pathignore	Defines whether components in defined directories (see solidedge.partrename.pathnames) are to be ignored when determining non-project parts.
solidedge.partrename.pathnames	Defines the directories that are to be ignored when determining non-project parts. Enumeration of directories separated by semicolons.
solidedge.propertyview.autoswitch	Defines that the Attachment Parts tab is activated if the document has attachment parts.
solidedge.propertyview.configurations	
solidedge.propertyview.defaulttab	The system remembers the last active tab for the next programme call.
solidedge.propertyview.document	Defines whether the Document tab is displayed (in the speedy rollup window).
solidedge.propertyview.files	Defines whether the Files tab is displayed.
solidedge.propertyview.library	Defines whether the Attachments tab is displayed.
solidedge.propertyview.properties	Defines whether the Properties tab is displayed.
solidedge.propertyview.relations	Defines whether the Relations tab is displayed.
solidedge.propertyview.revisions	Defines whether the Versions tab is displayed.
solidedge.propertyview.whereused	

Key	Description
solidedge.relation.forceupdate	Defines whether the relations should always be updated when saving an assembly.
solidedge.revision.count	Number of revision entries in the revision table.
solidedge.revision.first	First revision to be displayed. A value of 1 means the revision table starts with the 1st release. Default value := 1
solidedge.revision.firstline	Write only the 1st line of the revision comment to the version table.
solidedge.revision.revlabel	Format the revision label in the revision index. see also [session.revlabel]
solidedge.rollup.autorefresh	Defines whether the document list should be refreshed automatically when the rollup window is opened.
solidedge.rollup.delay	Defines the delay time for closing the rollup window in 250ms steps.
solidedge.rollup.delta	
solidedge.rollup.deltaremember	
solidedge.rollup.dockable	Show the speedy window either as a dockable SolidEdge window or as a rollup window.
solidedge.rollup.unrolltimer	Defines the delay time for opening the rollup window in 250ms steps.
solidedge.rollup.show	Show rollup window yes/no.
solidedge.save.dependent	When saving or saving&closing, check whether referenced files have been changed. If referenced files have been changed, a dialogue with further options appears.
solidedge.script.shutdown	Defines an executable file to be executed on shutdown (e.g. .batch, .exe).
solidedge.script.startup	Defines an executable file to be executed at startup (e.g. .batch, .exe).

From:
<https://wiki.speedy-pdm.de/> - **speedyPDM - Wiki**

Permanent link:
https://wiki.speedy-pdm.de/doku.php?id=en:speedy:20_cad:solidedge&rev=1674628574

Last update: **2023/01/25 07:36**

