

Manual

SolidConnect

1.3.6

July 24, 2025

www.enginia.nl

Enginia B.V.
Kvk 06067120
BTW/VAT NL815834615B01
Rabobank: NL13 RAB00179103822

Enginia Oost
Demmersweg 5-13
7556 BN Hengelo
T +31 541 768420

Enginia Midden
Modemweg 37
3821 BS Amersfoort
T +31 33 4573322

Enginia West
Zilverstraat 69
2718 RP Zoetermeer
T +31 15 3807500

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1 Preface

1.1 Introduction

Enginia's SolidConnect is intended to be the ultimate tool to work optimally with data and to streamline work planning processes. It is a combination of all the knowledge acquired developing separate solutions such as SE2ERP, Prepared Documents and SE2RiQ, into one application.

SolidConnect is an application tool that does the following:

- **Extract data** – A part or whole assembly can be loaded from an active SolidEdge application or PlmXml.
- **Presentation** – It presents the extracted data in a simple and configurable way, allowing you to quickly filter and check your data.
- **Validation** – After loading, SolidConnect validates all data using customizable rules. This is done using regex, a powerful existing framework for checking text. We have built a simple interface on top for most use cases.
- **Exporting data to ERP via files** – With all data loaded and validated, you can export this data to xml or csv, configured exactly how you need. You should be able to configure the export so that you can directly import it into your ERP system.

SolidConnect has been developed with you, as a user, in mind. We wanted the tool to be as accessible as possible, with a user-friendly interface and a minimum amount of 'clicking'. At the same time, we wanted the process, from extraction to exporting of the data, to be as clear as possible. If something went wrong or you need to be warned, a message box will pop-up with a message concerning the issue.

In terms of product knowledge, we recommend the user to have:

- A basic knowledge of Solid Edge, its properties and data that's going to be exported.
- Technical knowhow of manufacturing industry

1.2 Development

SolidConnect is still early in development with many future plans such as:

- **Publishing documents** – Creating files such as pdf's or step files for many parts automatically. Includes many ways to customize what should and should not be exported and in which formats.

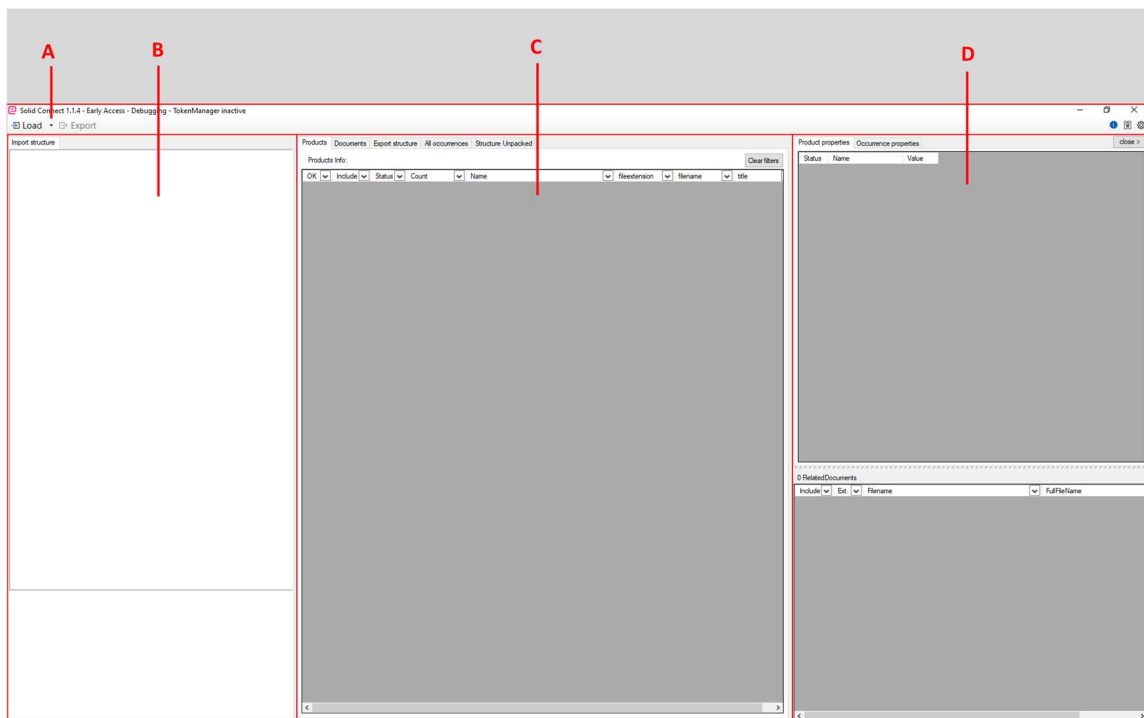
2 Documentation

2.1 Main/home screen

2.1.1 Layout

The first thing you'll see when opening SolidConnect is a splash screen followed by the main screen. The main window consists of the following layout:

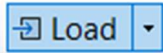
- A. Action bar - with buttons
- B. Import structure, with a snapshot of the selected file
- C. Products, Documents and Export structure tabs
- D. File and occurrence properties



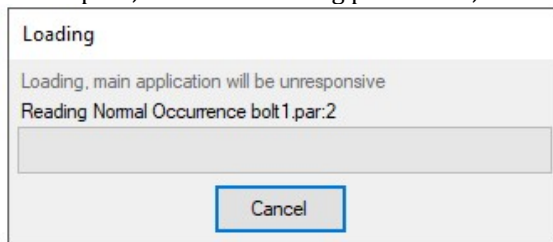
2.1.2 Action bar

At the top of the application on the left and right side are some of the most important buttons you will need when using SolidConnect.

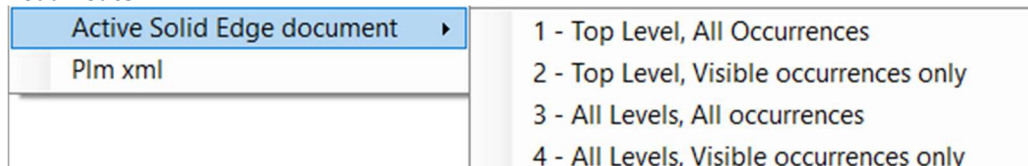
Load button



Press the load button to load data into SolidConnect, by default this loads the active SolidEdge document with all levels and all occurrences. You can also click on the arrow on the right side inside the button to choose a specific load mode. You can customize what the default load mode is when pressing the button normally in the [setup](#). When loading a Solid Edge document, depending on the amount of parts to be loaded this can take a short while. If you make adjustments to the setup or main screen of SolidConnect, you might need to extract data from the Solid Edge document again, otherwise, adjustments won't be processed. After a click on the load button, a loading window will always appear. If you want to stop the loading process, you can cancel the loading with the cancel button. Note that some processes cannot be interrupted, such as activating parts in SE, in this case you will need to wait for it to be cancelled.



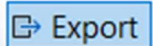
LoadModes



The drop down list shows you two primary options:

- **Active Solid Edge document:** loads the active document from an already active SolidEdge application. There are four modes, these are further in [Usage](#).
- **Plm xml:** will ask you to select a plmxml file to load. (plm xml is a file format exported from TeamCenter)

Export button



The export button will start the export of the products using your settings, these are explained in [Export Settings](#). After exporting, you might get a popup if any warnings have been written to the log, please always check the logs if this is the case because it will likely mean the export will not be exactly as configured.

Quick Refresh

When you press the quick refresh button  , SolidConnect searches again for related documents and re-analyzes properties. This is also automatically done after running flows and changing settings.

Open/Close logs (e)

At the top right is a small button with any of these icons    , while working with the tool, when there is an error or you need to be warned, a message box will pop up. However, not all warnings/errors result in a message pop-up. For this, you can always rely on the logs in which all major processes are noted. The icon shows the highest logged severity (info/warning/error) of the most recent process (before loading and exporting this is reset, so that only a warning/error during that process will be shown).

Click the button to open or close the log within the home screen. The following type of information can be found in the log:

- **Warning**
- Information
- **Errors**

Click on the button again if you want to close the log window and return to the home screen.

Logs do not show messages from previous usage if you have since closed the application, however everything is also written to a file, clicking on the log button while holding the control key will show the location of the log-file in the file explorer.

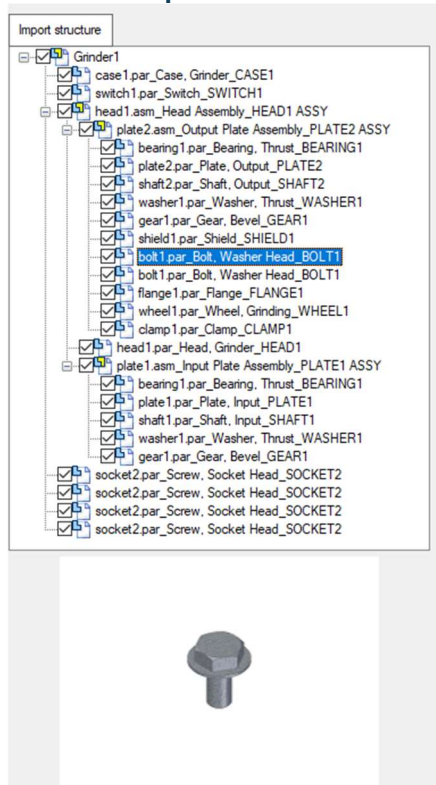
License / Token Manager

Also at the top right is the  button, click this to manage your license or tokens. [more about licenses and tokens](#)

Setup

The small gear button  at the top right will open an interface that contains all settings to customize the application, how your assembly is read and exported and more. [More about setup](#)

2.1.3 Import structure



The import structure has a visual hierarchy (tree-view) representing your active assembly in Solid Edge. The tree view shows all parts (and sub-assemblies depending on the structure level you've chosen) of the assembly comparable with the tree of the Pathfinder in Solid Edge. The name shown for each part is configurable in setting. The checkmark allows you to manually change whether a part will be exported, settings to do this automatically will be discussed later. Ctrl + click to check/uncheck a part in the tree including all sub-parts. There are some differences with Solid Edge such as patterned items are not grouped inside a 'patterns' object like SE but are placed separately with the other parts but these have no functional impact.

2.1.4 Products

Products Documents Export structure								
20 Products (20 for export)								
OK	Include	Status	Draft	Count	Name	document number	revision	material
✓	✓	🔒	📄	1	Grinder1	GRINDER1		
✓	✓	🔒	📄	1	case1	CASE1	0	PVC
✓	✓	🔒	📄	1	switch1	SWITCH1	0	plastic
✓	✓	🔒	📄	1	head1	HEAD1 ASSY	0	
✓	✓	🔒	📄	1	plate2	PLATE2 ASSY	0	
✓	✓	🔒	📄	2	bearing1	BEARING1	0	Bronze, 90%
✓	✓	🔒	📄	1	plate2	PLATE2	0	410 Stainless Steel
✓	✓	🔒	📄	1	shaft2	SHAFT2	0	4140 Steel
✓	✓	🔒	📄	2	washer1	WASHER1	0	510 Copper
✓	✓	🔒	📄	2	gear1	GEAR1	0	8620 Steel
✓	✓	🔒	📄	1	shield1	SHIELD1	0	1020 Steel
✓	✓	🔒	📄	2	bolt1	BOLT1	0	Steel
✓	✓	🔒	📄	1	flange1	FLANGE1	0	4340 Steel
✓	✓	🔒	📄	1	wheel1	WHEEL1	0	Composite
✓	✓	🔒	📄	1	clamp1	CLAMP1	0	4340 Steel
✓	✓	🔒	📄	1	head1	HEAD1	0	A380 Aluminum
✓	✓	🔒	📄	1	plate1	PLATE1 ASSY	0	
✓	✓	🔒	📄	1	plate1	PLATE1	0	410 Stainless Steel
✓	✓	🔒	📄	1	shaft1	SHAFT1	0	4140 Steel
✓	✓	🔒	📄	4	socket2	SOCKET2	0	

The products tab at the center of the application is an overview of all your products, you can see the count and properties of each product. You can use the right-click context menu for various actions (see [Context Menu](#))

The table has a few columns by default:

- OK - displays whether the file has any errors (not loaded) or warnings (bad properties)
- Include - displays whether this file will be included for export or other further processing.
- Status - displays the file status from Solid Edge Data Management (SEDM). For this, you need to have a Status file property defined in the property analysis settings (See [configuration](#)).
- Draft - displays whether an associated draft file has been found. Notice, that only a draft document with the same filename will be checked.
- Count - displays the amount of items in the assembly and will be included to the export.
- Name - displays the name of the item.

The content table can be customized in the following ways:

- Each column has a button to sort and filter rows. Note that multiple sort and filters can be activated at once.
- Reorder columns by dragging the column header.
- Resize columns by dragging the edges between column headers.
- Show/Hide/Edit columns by right-clicking on any column header and (un)checking the column names. Here you can also click Edit to add any property you have defined in the settings (See [configuration](#)) as a column. Lastly you can press Reset to reset back to the default layout.

The rows can any have the following colors:

White: default
Red: error loading file, check logs

Yellow: some properties have not passed the analysis, check details ([configuration](#))

Gray: product is excluded

Some products might appear in multiple rows when they are parts that have unique properties such as frames with differing lengths. At the top right is a clear filters button to clear all filters applied to any of the document columns (does not reset sorting)

2.1.5 Documents

Products Documents Export structure

43 Documents (43 for export)

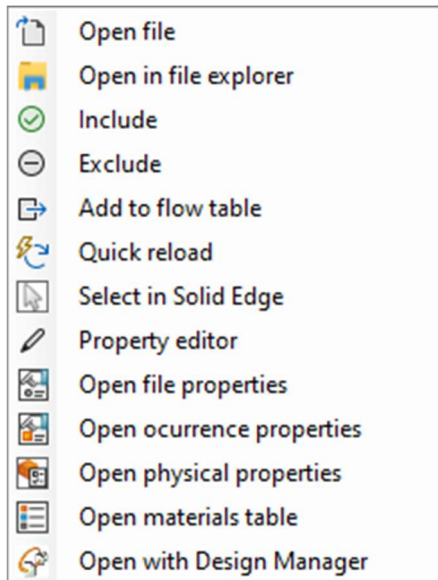
 Clear filters

Include	Ext.	Filename	FullFileName
		bearing1.par	C:\Program Files\Siemens\Solid Edge 2024\Training\bearing1.par
		bolt1.par	C:\Program Files\Siemens\Solid Edge 2024\Training\bolt1.par
		case1.par	C:\Program Files\Siemens\Solid Edge 2024\Training\case1.par
		clamp1.par	C:\Program Files\Siemens\Solid Edge 2024\Training\clamp1.par
		flange1.par	C:\Program Files\Siemens\Solid Edge 2024\Training\flange1.par
		gear1.par	C:\Program Files\Siemens\Solid Edge 2024\Training\gear1.par
		Grinder1.asm	C:\Program Files\Siemens\Solid Edge 2024\Training\Grinder1.asm
		Grinder1.cfg	C:\Program Files\Siemens\Solid Edge 2024\Training\Grinder1.cfg
		Grinder1.dft	C:\Program Files\Siemens\Solid Edge 2024\Training\Grinder1.dft
		Grinder1.dxf	C:\Program Files\Siemens\Solid Edge 2024\Training\Grinder1.dxf
		Grinder1.emf	C:\Program Files\Siemens\Solid Edge 2024\Training\Grinder1.emf
		Grinder1.jpg	C:\Program Files\Siemens\Solid Edge 2024\Training\Grinder1.jpg
		Grinder1.log	C:\Program Files\Siemens\Solid Edge 2024\Training\Grinder1.log
		Grinder1.obj	C:\Program Files\Siemens\Solid Edge 2024\Training\Grinder1.obj
		Grinder1.pdf	C:\Program Files\Siemens\Solid Edge 2024\Training\Grinder1.pdf
		Grinder1_3D.pdf	C:\Program Files\Siemens\Solid Edge 2024\Training\Grinder1_3D.pdf
		head1.asm	C:\Program Files\Siemens\Solid Edge 2024\Training\head1.asm

The documents tab shows all loaded/found documents. SolidConnect automatically searches for similarly named files in the same folder of the files it has loaded. You can use the right-click context menu for various actions such as opening a file (see [Context Menu](#)).

At the top right you can press 'Clear filters' to quickly clear all active column filters (does not clear ordering) You can press the button with the zip icon to save a zip file of the included documents that are currently shown. Only included (green checkmark) documents are added to the zip, also only documents that are visible (because of filters) are added. Note that files that are open in another program might not be successfully added to the zip. For SolidEdge files, SolidConnect will ask to save and close them for you.

2.1.6 Context Menu



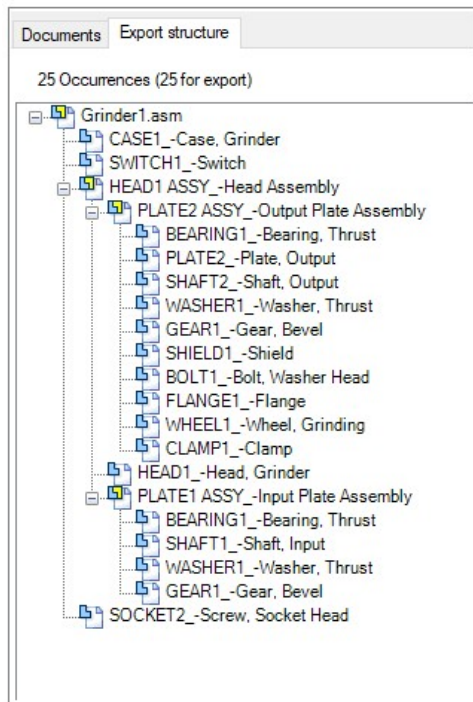
When you right-click on any item in the import structure or in the products/documents tables a context menu pops up that allows you to do a few actions related to the selected item.

- Open file: opens selected file or the primary associated file of the item in the appropriate program
- Open in file explorer: opens selected file or the primary associated file of the item in a new file explorer window with the file selected
- Include/Exclude: Manually includes/excludes an occurrence of a product when selected in the import structure; includes/excludes all occurrences of a product when selected in the product list. includes/excludes a specific file when selected in the document list. Also note that if all occurrences of a product are excluded, the product itself is automatically excluded. All documents of a product are also excluded when the product is excluded.
- Open with Design Manager: opens selected file or the primary associated file of the item in the Design Manager
- Property editor: open the selected file or the primary associated file of the item in the Property editor
- Add to flow table: add the selected file or all of the associated files of the item to the flow table in the flow tab

The following options are only available when you have loaded from SolidEdge:

- Quick reload: reload/update file properties and variables of a single selected part.
- Select in Solid Edge: Select the part(s) in Solid Edge
- Open File Properties: opens the file properties window in Solid Edge of the selected part/document.
- Open Occurrence Properties: opens the occurrence properties window in Solid Edge of the selected part/document.
- Open Physical Properties: opens the physical properties window in Solid Edge of the selected part/document.
- Open Material List: opens the material list window in Solid Edge of the selected part/document.

2.1.7 Export Structure



The export Structure tab shows a modified structure ready for export, here parts that are excluded are removed. Multiple of the same product are combined into one item (see [Uniqueness \(c\)](#)). When an assembly is set to not be exported but sub-parts are, the assembly is removed and the sub-parts are moved one level higher.

2.1.8 Details

File propertiesOccurrence propertiesclose >

Status	Name	Value
✓	Title	Grinder Assembly
✓	Author	jmrobin
✓	Comments	
✓	Status	0 (Available)
✓	Document Number	GRINDER1
✓	Revision	
✓	FullFileName	C:\Program Files\Siemens\Solid Edge
✓	FileName	Grinder1.asm
✓	FileNameNoExtension	Grinder1
✓	FileExtension	.asm
✓	FilePath	C:\Program Files\Siemens\Solid Edge
✓	HasDraft	True

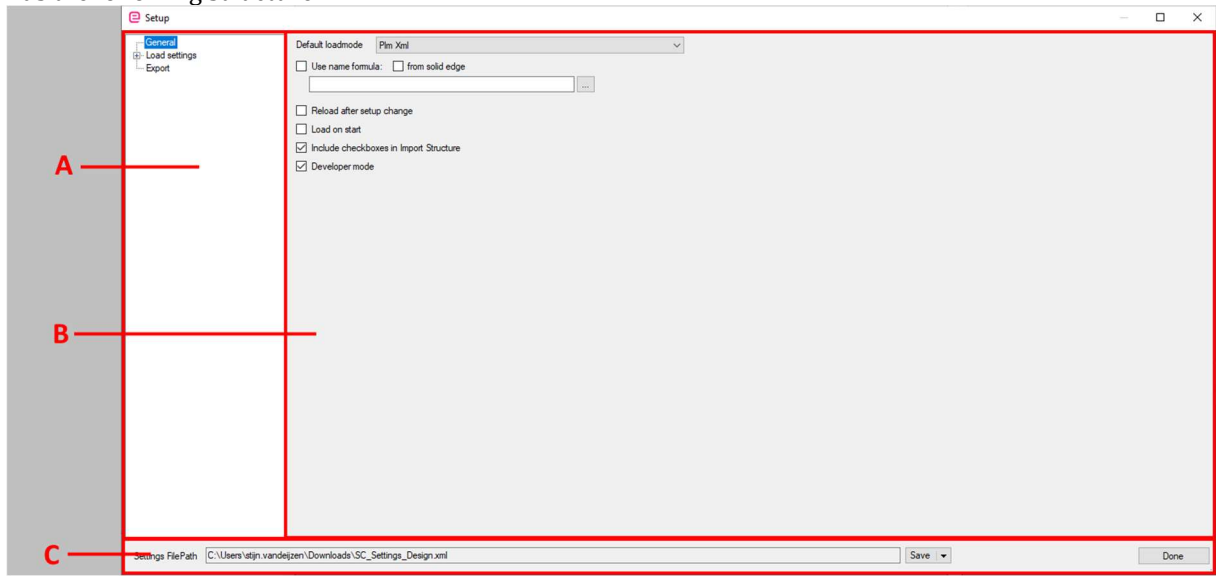
10 related documents

Include	Ext.	Filename
✓		Grinder1.asm
✓		Grinder1.cfg
✓		Grinder1.dft
✓		Grinder1.dxf
✓		Grinder1.emf
✓		Grinder1.jpg
✓		Grinder1.log
✓		Grinder1.obj
✓		Grinder1.pdf
✓		Grinder1_3D.pdf

The details panel show more information about the selected item. This panel can be expanded and closed via the button at the top right. At the top are the file and occurrence properties, these show all properties SolidConnect was able to import of the selected object. Note that these views are contextual; when you select a document in the documents list, this will not show any occurrence properties. When selecting an item in either the tree-view on the Import or export structure tap it will show both file and occurrence properties. The properties also indicate whether they have passed the validation/analysis. When a property has not passed, they are marked with red and have an X icon status. Hovering over this icon shows a tooltip providing some more context why the property failed. At the bottom are related documents, these are documents related to this product, read [Documents](#) for what that means.

2.2 Setup

You can use the setup to structure your way of working; it makes your work more efficient and effective, because you have control on what you want to see and how run certain processes. Get access to the setup window by clicking on the setup button located on the action bar of the main window. The setup layout has the following structure:



- Setting pages/categories - The settings span across different pages and categories, categories can be expanded with the + button
- Setting page - The main window shown the current settings page
- Location,Closing,Saving - At the bottom the settings file location is shown, here you can save your settings or press the arrow on the save button to save as or load a different settings file. The done button will check for changes and prompt if you want to save if there are any.

2.2.1 Preferences

i Preferences are user-specific (independent of settings file) and always editable

Product double click PropertyEditor ▼

Document double click OpenFile ▼

☒ Load on start

☐ Reload after setup change

☐ Show checkboxes in import structure

☐ Load selected part in property editor

Flows

☐ Add products to flow tab on load

☐ Run flows when exporting

Save SolidEdge documents before running flows Ask ▼

The Preference settings are user specific settings and are independent of the settings file, these are always editable, even if the settings file is readonly. The preference settings consists of:

- Default action for double clicking a product.
- Default action for double clicking a document.
- Load on start: when the application has just started it will automatically try to load the active Solid Edge file.
- Reload after setup change: when turned on automatically press the load button after saving the setup.
- Show checkboxes in import structure: this will show or not show the checkboxes next to the nodes in the import structure.
- Load selected part in property editor: instead of showing the active file in the property editor, this will show the selected part.
- Add products to flow tab on load: This will automatically move all products to the flow tab, after everything is loaded in. Useful for quickly running flows on all products
- Run flows when exporting: this will run the default flow selection on all products, when the export is started.
- Save SolidEdge documents before running flows: here you have a choice to always save, never save or always ask. This setting will decide what happens with the open SolidEdge documents.

2.2.2 General settings

Default loadmode Active Solid Edge document ▼

Default name formula [Filename]_[Title]_[Document Number] ...

The General settings consists of:

- Default load mode – choose the load mode that is used when pressing load/import without using the dropdown
- Formula for naming occurrences - Here you can build up a formula with text and properties of occurrences that will be used as the display name. This is the default setting and can be overruled

when using the name formula retrieved from Solid Edge. Remember to make sure the properties you want to use are defined in the property settings.

2.2.3 SolidEdge settings

General

Default loadmode 4 - All Levels, Visible occurrences only

- ☒ Get name formula from solid edge
- ☒ Separate frames by cut length
- ☒ Separate frames by miter
- ☐ Get mass

The default load mode defines how the active document in SolidEdge should be loaded, the options are:

SolidEdge has a name formula system to customize the name of occurrences in the pathfinder, by setting 'get name formula from solid edge' SolidConnect will retrieve this setting from SolidEdge, this will replace the name formula setting in the General settings of SolidConnect.

There are many unique types in Solid Edge that have the same underlying part file but are actually different. The best example is a frame, it is based on the cross section of the frame but each occurrence can have unique length and cuts. These settings enable checking specifically for these unique objects and will extract those unique properties and then regard them as separate parts.

- Separate frames by cut length: when frames are found, the cut length will be retrieved and added as a property. When two frames are the same base file but different lengths, they will be added as separate documents.

Count	FileName	cutlength
5	RectangularTubing 50x30x2.6.par	0,229999999999...
5	RectangularTubing 50x30x2.6.par	0,215

- Separate frames by miter: when frames are found, all cut angles are retrieved and added as a property. When two frames are the same base file but different angles, they will be added as separate documents. Note that there are 6 values to describe these angles.

Count	FileName	cutlength	mitercut1	mitercut2	endface1endar	endface2endar	sideface1endar	sideface2endar
1	SquareTubing 50x50x4.par	0,450000000000...	45	45	45	45	45	45
1	SquareTubing 50x50x4.par	0,425	0	45	0	45	0	45
1	SquareTubing 50x50x4.par	0,449999999999...	45	45	45	45	45	135
1	SquareTubing 50x50x4.par	0,435355339059...	45	67,5	45	22,5	45	67,5
1	SquareTubing 50x50x4.par	0,410355339059...	0	67,5	0	22,5	0	67,5

Get mass: This setting enables retrieving the mass, because mass is a unique property that must be handled differently (especially for frames). This is slower and requires parts to be activated in SolidEdge. This will give better results than reading Mass from a variable.

Import settings

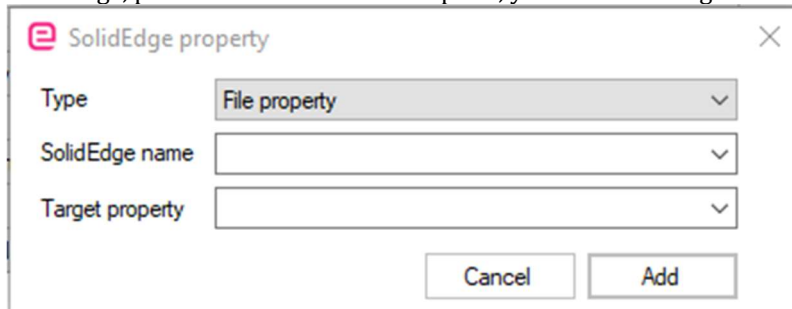
These settings are used to configure how to data should be imported from SolidEdge

Type	SolidEdgeName	Target Property
File property	Title	Title
File property	Subject	Subject
File property	Author	Author
File property	Keywords	Keywords
File property	Comments	Comments
File property	Category	Category
File property	Manager	Manager
File property	Company	Company
File property	Status	Status
File property	Document number	Document number
File property	Revision	Revision
File property	Project Name	Project Name

At the top you have the following buttons:

- Add
- Remove
- Edit
- Duplicate
- Add default TeamCenter X Essentials settings

You should see already see some default settings that are greyed out. To add a new property to read from solid edge, press the + button at the top left, you should then get the following popup:



The dialog box titled "SolidEdge property" contains three dropdown menus: "Type" (set to "File property"), "SolidEdge name", and "Target property". At the bottom are "Cancel" and "Add" buttons.

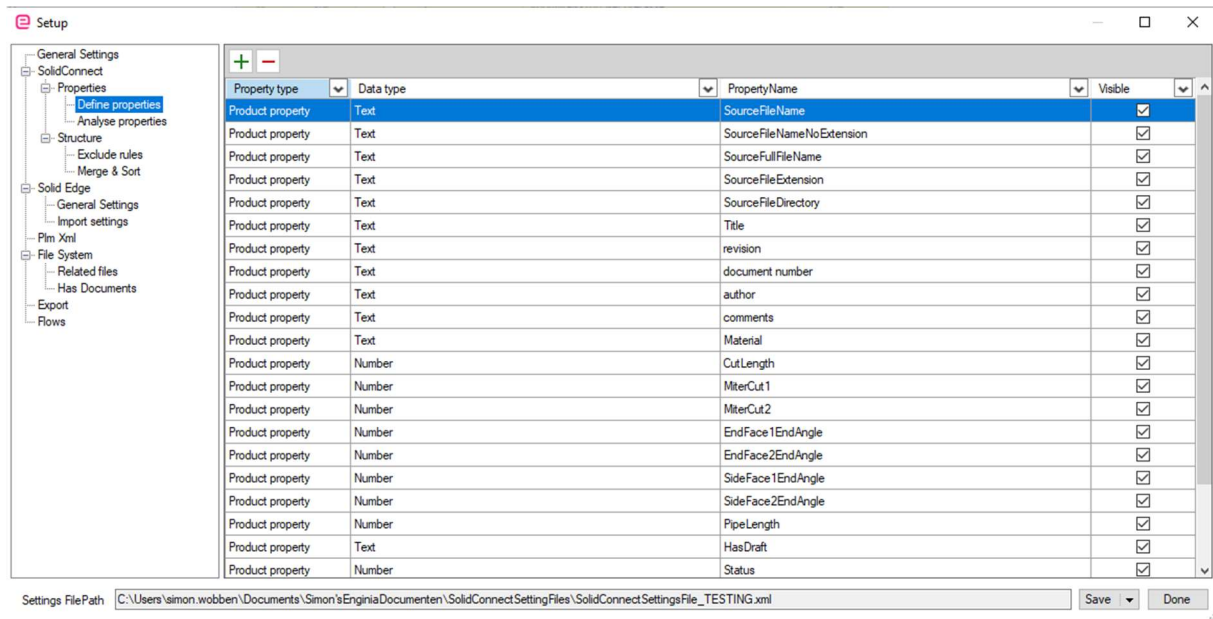
First choose whether the property you want to import is a:

- File property: properties stored directly in the files. Can be opened in SE via Data Management>File Properties. When an assembly has already been loaded in SolidConnect you can use a dropdown to see all fileproperties that have been found.
- Occurrence property: properties that can be different between two instances of the same part. Can be opened in SE via the right-click menu in the pathfinder
- Variable: any variable/dimension from the variables table. Can be opened in Solid Edge via tools>variables.

and enter the name that is used in SolidEdge. The target property field sets under what name the property is added within SolidConnect, this is automatically the same as the SolidEdge name but can be changed to any other name. The target property needs to be defined in the property definition settings, if the entered target property is not already defined it will automatically create it, which is indicated by the ✨ that appears.

2.2.4 Define property settings

In the define property settings you define the properties you want on your products. Properties will need to be defined here before they can be filled with data. When adding import settings and entering a new property name the definition is automatically created for you.



Property definitions have two types: Product property and Occurrence Property

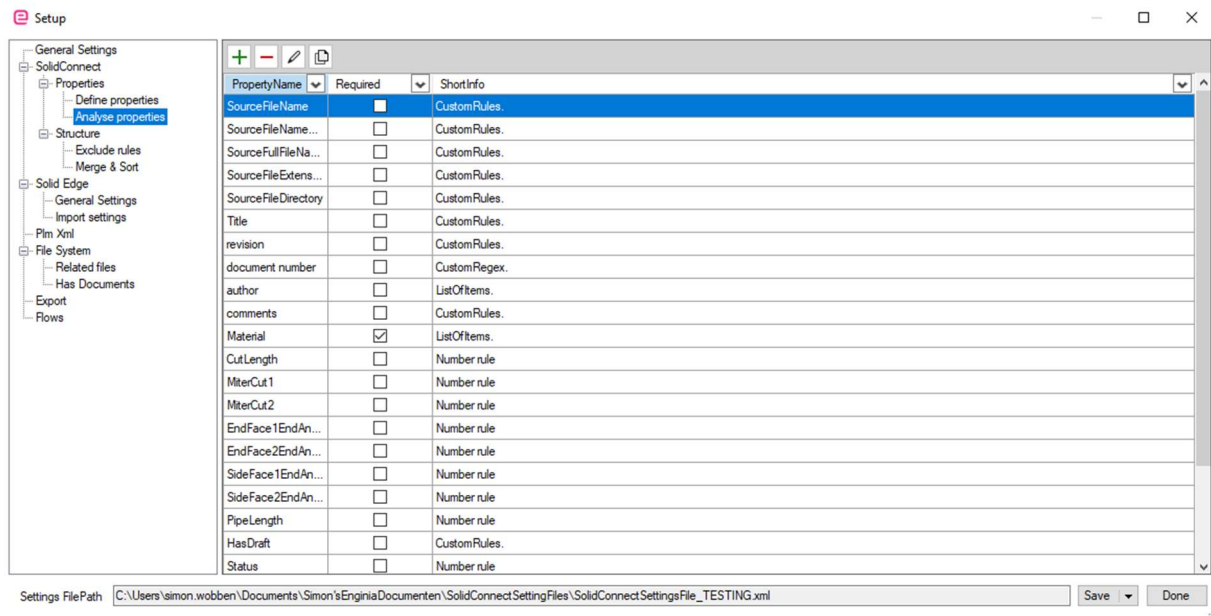
The data type defines what type of data is expected in this property,

- Text
- Numbers
- Yes or no
- Date

The property name is the name that will be shown throughout the application. In the visible column you can click to change if the property should be visible.

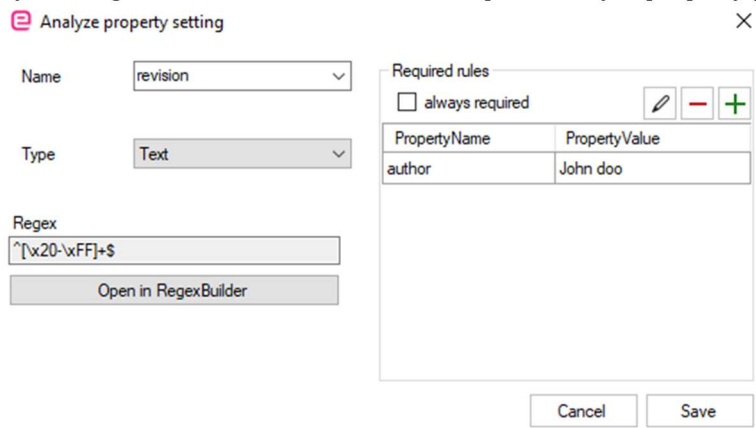
2.2.5 Analyze property settings

In the analyze property settings you can define how a property should be analyzed.



These rules will decide how the property should be analyzed based on the regex rule in the property, this can be a custom rule, a list of items, a number rule (only applies to number properties) and a custom regex. Not every property needs an analyze setting, multiple analyze settings may apply to one property.

By clicking the add or edit button on the top, the analyze property pop-up will pop up.



Here you can select or enter the name of the property you wish to analyze. Next you can select a type of data you want to analyze.

- Text properties are properties containing text, they are analyzed with a regex, this uses a complex syntax but is commonly used in programming. The regex rule is what actually does the analyzing, in the regex builder you can specify the actual rule to analyze the text. With the 'Open in Regex builder' button you can open a user friendly interface to allow you to create a regex. (see [Regex builder](#))

- Number properties are properties containing a numeric value. They are analyzed by a minimum and maximum value. You can set the maximum value to 0 if you do not want a maximum. Number properties do not store a unit. Number properties optionally can be converted from Text, this will try to parse the retrieved value into a valid number. Parsing follows specific rules: only reads characters that are a number, comma or period with no space in between. This means you can have a value like '1,2 kg' and it will become the number '1,2'. Note that this conversion uses window's region settings (might need restart of Solid Connect when changed) for the decimal separator.
- YesOrNo: this property contains a binary value, a yes or no, a true or false. This is currently not very integrated feature.
- Date: this property contains a date value. This is currently not very integrated feature.

The required rules specifies if this current property HAS to be present on the product or occurrence if the value of the referenced property equals the value of the required rule value.

Example: the property revision has to be present on a product if the product contains the property "author" and if that property's value is equal to "John doo"

2.2.6 Logic property settings

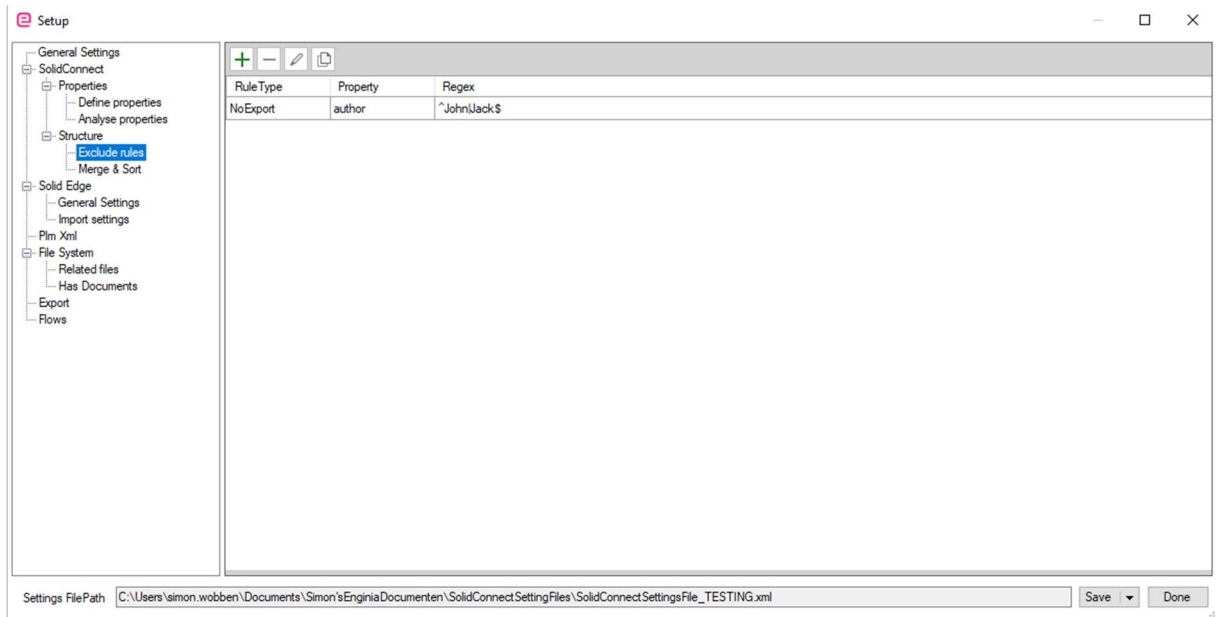
In the Logic property settings you can add logic for properties, which currently includes just "Replace text". Which means that you could replace every "." With a "," for example if your mass properties are formatted like "10.0kg" this would change it to "10,0kg". This won't overwrite the actual property in SolidEdge, but this will be the value shown, used, and exported within SolidConnect.

+ - ✎ 📄	
Type	Property
Replace text	Mass

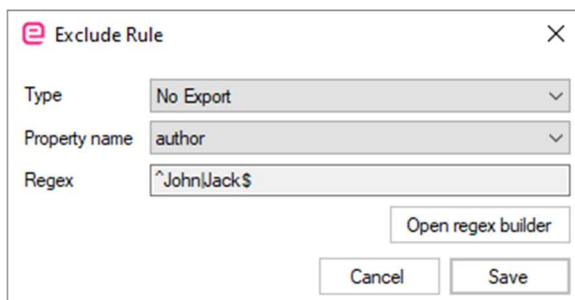
2.2.7 Exclude rules settings

When parts or assemblies should not be exported or only partially, these rules help you automate these decisions based on properties. When loading the assembly, these rules check for any property fulfilling the defined condition. This condition is a regex, which you can create using the [Regex builder](#). The property to check must be defined in the [property analysis settings](#). You have the following options for what the rule does:

1. No Export: do NOT export this assembly, do NOT export sub occurrences, Example: walls
2. Only children: do NOT export this assembly, DO export sub occurrences, Example: help/dummy assemblies for placing multiple parts at once: bolt + washer + nut
3. No children: DO export this assembly, do NOT export sub occurrences, Example: bearing



In this example we do not export parts or assemblies if they were created by Jack or John.



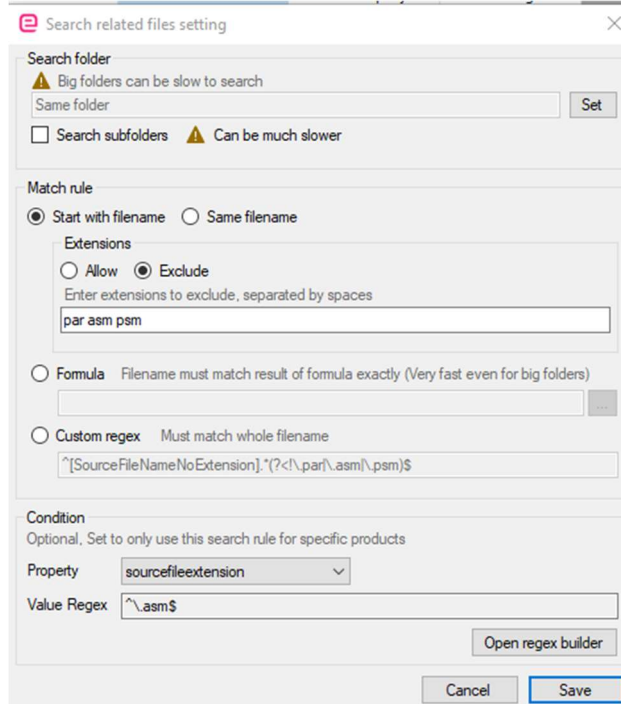
2.2.8 Related files settings

Basic

SolidConnect can search for related files of your products, these settings configure how is searched for these files. While settings allow complex logic, lets first explain the basics. Each row in this table is a rule by which SolidConnect will try to check for related files. The first column contains a checkbox for whether the rule is active. You should see some grey lines already defined, these are the default options, you cannot edit these but you can disable them.

Active	Folder	Subfolders	Match rule	Extensions	ConditionProperty	ConditionRegex
☒	Same folder	☐	MatchBeginFilename	Exclude: par asm psm	fileextension	^\.asm\$
☒	Same folder	☐	MatchBeginFilename	Exclude: par asm psm cfg	fileextension	^\.par\$\.psm\$

The other columns will make more sense when we look into the edit menu:



Search related files setting

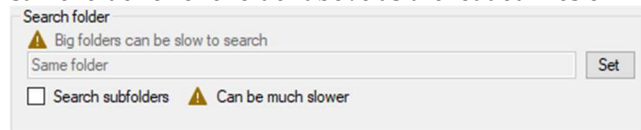
Search folder
 ⚠ Big folders can be slow to search
 Same folder Set
☐ Search subfolders ⚠ Can be much slower

Match rule
☒ Start with filename ☐ Same filename
 Extensions
☐ Allow ☒ Exclude
 Enter extensions to exclude, separated by spaces
 par asm psm
☐ Formula Filename must match result of formula exactly (Very fast even for big folders)
☐ Custom regex Must match whole filename
 ^[SourceFileNameNoExtension].*(?<\.par\.\asm\.\psm)\$

Condition
 Optional. Set to only use this search rule for specific products
 Property: sourcefileextension
 Value Regex: ^\.asm\$ Open regex builder

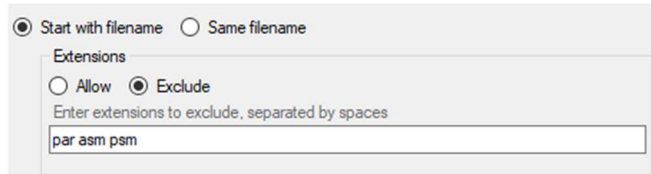
Cancel Save

The folder setting specifies where to search for files, you can use the set button to choose to search in the same folder or one folder above as the loaded files or in a specific folder:



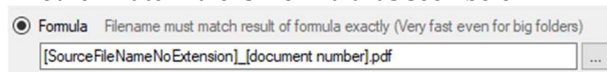
Search folder
 ⚠ Big folders can be slow to search
 Same folder Set
☐ Search subfolders ⚠ Can be much slower

The match rule defines what files are considered related, there are some basic options to cover common use-cases:



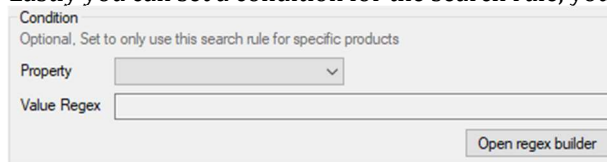
If you choose 'exact filename' it means that all files with the exact same filename and only a different extension are considered related files. If you choose 'Start with filename' it allows some extra text after the filename it is searching for, so for the file 'Grinder.asm', the file 'Grinder Manual.pdf' would also be considered related. Below those settings you can set a list of extensions that are allowed or excluded, When you select the 'Allow' option, only those extensions are allowed, if you select 'Exclude', any extension but those is allowed. You need to type all extensions after each other, separated by spaces. Placing the dot before the extension is optional.

Another match rule is 'Formula' as seen below



This allows you to enter a formula which will check for a specific file without the need to search matching files in a folder which makes it much faster. In this example using the *Grinder from the training folder* as the product to search files for it would search for '*Grinder1_GRINDER1.pdf*' in the selected folder

Lastly you can set a condition for the search rule, you can leave it empty to always execute the rule.



The property you choose and the regex (see [RegexBuilder](#)) are checked, when a product has the chosen property and the regex matches its value, only then will this rule be used.

The two default settings have a condition for the filename, this makes sure that .par and .psm files will not match .cfg files, which are files related to .asm in SolidEdge.

You should try to avoid rules which can find the same file twice, as it will produce warnings and will not add the same file to a second product.

Advanced

If you have a good understanding of regex and your filesystem you can use this to make more complex rules which SolidConnect will use to check for related files.

Firstly the folder setting supports absolute paths and relative paths and also allows for a [formula](#).

Examples:

- 'C:\Test' or '\\Server\Test' are absolute paths
- 'C:\Test\[project name]' path with formula
- '\Test' is relative folder to the drive 'C:\Test'
- '.' In the same folder as the source of the product.
- '\Test' or 'Test' is relative folder to the source of the product.
- '../' One folder above as the folder of the source of the product.

Next you can also use a custom regex to define what files are considered related.

Because the regex can get complex for these settings, you could first use the basic settings and then switch to custom to customize it further. It is built from these parts:

- ^ start
- **[FileNameNoExtension]** the filename must begin with the same filename (without extension)
- .* the filename can have any character after the filename
- \. The literal dot character in the file extension
- **(?!par\$|asm\$|psm\$|cfg\$)** these extensions are excluded (separate each extension with a pipe, make sure to add the \$ after each one)
- .* After the dot, any character is allowed (except those previously excluded)
- \$ end

This regex may contain a [property formula](#), such as the '**[FileNameNoExtension]**' in the example.

2.2.9 Occurrences Merge & Sort

☒ Separate occurrences by properties

Separate occurrences by properties: Two of the same part can have different (custom) occurrence properties, for example when one part has to be maintained more often. If two occurrences have (custom) occurrence properties that differ, they will be separated in the export structure if this setting is on. Parts are only separated in the export structure but are still considered the same part/document, this is an important distinction for how exporting or further processing is handled.

Sort parts (by property)

1st	title	Ascending
2nd		Ascending
3rd		Ascending

When you want your parts and assemblies to be sorted before exporting, you can select by which property should be sorted. You can set by which property will be sorted first, second and third. You can set which way each property is sorted with the checkbox, when checked properties are sorted in ascending order, when unchecked in descending order. With these settings you should be able to match the sorting of the parts list in Solid Edge. The sorting of numbers and text is different, make sure the property definition setting is the correct type.

2.2.10 Has document settings

Logic properties allow you to add extra properties within SolidConnect based on some logic.

The 'Has document' logic property allows you to define a regex which will be used to check if any of the [related files](#) match your criteria, then it will add a property to the product with the specified name with a true or false value based on this check. This can then be displayed in the properties table in details or added as a column in the products list.

Name	hasdraft	hasdxf	haspdf	has3dpdf
Grinder1	✓	✓	✓	✓
case1	⊖	⊖	✓	⊖
switch1	⊖	⊖	⊖	⊖
head1	✓	⊖	⊖	⊖

The basic way to configure this regex would be typing a specific file extension in the 'must end with' field to check if such a file was found.

● Custom Rules

Allowed characters: Any

Length: Min. 0, Max. 0

Must start with:

Must include:

Must end with: .dft

You can optionally even use a property formula in the regex to get more specific. However the Regexbuilder currently does not support building such a regex and this requires a good understanding of regex. An example regex would be: `^.*\\[fileNameNoExtension]_PartsList\\.pdf$`
 This example requires a pdf to have the exact name of the product plus '_PartsList'. Note that the regex is matched against the full path of each related file.

2.2.11 Export settings

Current Export Template

Example_XML_PartsOnly ▾

XML ▾

Save As ▾

⚠ Save as to edit example

The export settings contain multiple templates at once, with examples. You can select and edit these at the top. In the first dropdown you can select a template as the active one. At the right you press SaveAs or the arrow to Rename, Delete, Add from file, Save from file. Not that you cannot edit examples, you will have to Save As first. You do not need to save the template itself after editing.

File Settings

Export folder formula

C:\Temp

...

Select folder

Export filename formula

[Title]

...

At the bottom are two important settings for the folder and filename to export to. You can use the select folder button to open a folder select dialog. Note that you cannot edit these on an example template. Both can be a formula with properties of the loaded part/assembly, these properties will then be replaced with their value just like the name formula in the general settings and as in Solid Edge. Example: C:\SolidConnect\Export\[Title] would place the export in a folder matching the title of the loaded part/assembly. Press the button with the three dots to open an editor for easier editing of the formula. Both settings also have [ExportDate] and [ExportDateTime] as options for properties which will be replaced with the data or date and time of the export. You can also use [MyDocuments] in the export folder formula.

Other than this setting, your settings will vary depending on which export mode you select, each is described below.

CSV export mode

Export categories

Type	FilterProperty	FilterPropertyValue
Part		
BOMLine		

Duplicate

Delete

Edit

Add

Columns

Name	Formula
Part_Number	[Document Number]
Revision	[Revision]
Description	[Title]
File_Name	[FullFileName]
Material	[Material]

Delete

Edit

↑

↓

Add

In Csv mode a table like list of values are exported, each column is separated by a defined character, and rows are placed on separate lines, this is a common format that is used to transfer tables of data, and is easily imported in Excel for example.

The left side of the settings are export categories, here you can define categories for your export, each category is either for a part or a bomline and can have a different export format. You can define filters for these categories allowing you to use a different format based on some property, when a property with the exact same name and value is found, that category will be used. One default is needed that does not have a set filter. There should always be one Part and one BOMLine category with NO filter set, this will be used as the default category when no other category filters match. At the right side you define the format for the selected category, each entry represents a column in your csv file, you give it a name and define the formula for the output value. You can build a [formula](#) for how the value gets exported, combining text with properties you have defined in the analysis settings.

For BOMLines you can also:

1. get a property from a parent part written like [Parent.PropertyName]
2. get the quantity (in context of the parent) written like "[Quantity]"
3. use "[BomLineNumber]", this will be a number starting at 1, incrementing each line.

Note that numbers will be exported using the decimal separator based on Windows's settings.

Delimiter

☒ , (comma)

☐ : (semi-colon)

☐ | (pipe)

☐ tab

☐ Custom

Parts

☒ Export
 ☒ Add column headers
 ☐ As separate files
 File extension

_parts.csv

BOM

☒ Export
 ☒ Add column headers
 ☐ As separate files
 File extension

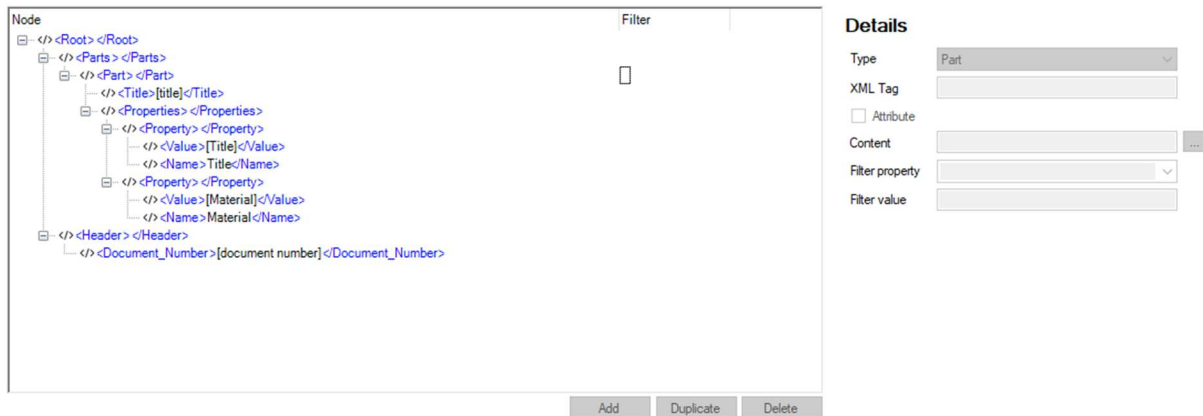
_bom.csv

☒ Enclose values in double-quotes

i Double quotes are used to ensure characters in your data is not mistaken for the delimiter. (RFC 4180)
 If you disable this you need to ensure no data contains the delimiter.

The above settings change how the csv file will look. At the left you can choose a delimiter, the character used to separate the values into columns. Next to that you can find the 'Enclose values in double-quotes' settings, this setting ensures that any character in your data is not mistaken for the delimiter, however some systems do not support this format, in that case you can disable it, however please be aware that you then need to ensure none of your data contains the delimiter. Above that are some settings for Parts/Boms. First you can choose to not export it at all, then you can choose if the column names should appear in the csv as the first line. Then you can choose if you want separate files for each Part/Bom, by default everything is written to a single file for each type. If you export as separate files for Parts, each part creates a separate file with one line of its data. If you export as separate files for Boms, a file is created for each bill of material separately, containing only direct sub-occurrences. The last setting customizes the file extension of each type, note that when these are the same, one will overwrite the other, to avoid this you can also add extra text in front as is visible in the example above.

Xml export mode



In Xml export mode you can flexibly configure how the exported Xml file should be structured. This can be done with the interface as seen above. You define the structure in this hierarchical / tree-like structure which should resemble the desired output. Each node represents an xml element when exported, the displayed text is an approximation of the actual output.

Each node has extra settings visible in the details on the right, many settings have restrictions based on the context. Every node has the XML Tag setting, this setting is always enabled and allows you to change the name/text of the xml tag, this is also shown in the node text. Every node has a type that changes how the node behaves when exported, based on the type the available settings change. When adding a node, it is created with the first possible type, when more than one type is available you can choose a different type in the details panel. When first configuring the interface is probably empty, you can start with adding a new node, this node is the root node, everything else must be contained within this node. The root node is always a static node.

You can drag and drop nodes, this is not always allowed based on the context, the interface indicates when it is not allowed with a . The three buttons: Add, Duplicate and Delete have the keyboard shortcuts ctrl-N, ctrl-D and the delete key respectively.

Node Types:

Static: a static node is the most basic node with no special behavior. This node just creates an xml element with the set tag and content.

AllParts/ AllOccurrences: these two types are container elements for Part / Occurrence nodes respectively. One default node is also automatically created.

Part: this node represents a unique part (document), within this node you can use its properties. If you have multiple Part nodes inside the AllParts node, you can set a filter property and value. When a property with the exact same name and value is found, that Part node will be used to structure the XML. One default is needed that does not have a set filter.

Occurrence: this node represents a unique occurrence (an item in the export structure). Within this node you can use its properties. If you have multiple Occurrence nodes inside the AllOccurrences node, you can set a filter property and value. When a property with the exact same name and value is found, that Occurrence node will be used to structure the XML. One default is needed that does not have a set filter.

Property: the most important node, this is similar to the static node in that it just creates an xml element. This node however allows you to use properties of the Occurrence / Part node this is contained in. This is done using a formula, this is the same as for CSV export and document name formula. Based on the context different properties are available, it is recommended to use the formula builder via the button next to the formula textbox.

Attribute: property and static nodes can be changed into an attribute with the checkbox in details. This will not add a new xml element as a child but instead add an xml attribute with the same name and content to the node this is contained in. The approximate result is shown in the node's text.

TopAssembly: this node counts the same as an Occurrence node but only represents the loaded assembly itself.

SubOccurrences: this node is similar to AllOccurrences, instead of creating elements for each occurrence, it creates an element for each suboccurrence (child occurrence of the current occurrence)

PartOccurrences: this node is similar to AllOccurrences, instead of creating elements for each occurrence, it searches all occurrences of this part.

SubOccurrencesRecursive: this node is the same as the suboccurrences node except that it cannot have any nodes inside it. Instead it uses the Occurrence node it is inside of as the node to use for each suboccurrence (self-repeating).

2.2.12 Flow settings

In the flow settings you can load, delete, disable, enable and set defaults for flows.

Flows (or flow files) are essential to the batch handler and are “blueprints” for the process of execution.

+	-		
Include	Flow name	Flow file	
☒	SaveAsDxf	.\Services\FlowHandler\DefaultFlows\Translate file\SaveA...	☐
☐	SaveAsFBX	.\Services\FlowHandler\DefaultFlows\Translate file\SaveA...	☐
☐	SaveAsJT	.\Services\FlowHandler\DefaultFlows\Translate file\SaveA...	☐
☒	SaveAsPDF	.\Services\FlowHandler\DefaultFlows\Translate file\SaveA...	☒
☒	SaveAsSTP	.\Services\FlowHandler\DefaultFlows\Translate file\SaveA...	☒
☐	SaveAsSTL	.\Services\FlowHandler\DefaultFlows\Translate file\SaveA...	☐

Here we can see all the default flows and their settings. By checking and unchecking the include box we can enable/disable the flow altogether, it will then be shown or hidden in the flow tab (see [Running the batch handler](#)). Each flow is a saved in a file and are only referenced by SolidConnect, by default these default flows are provided with the software and are located in the programs folder. These flows are however greyed out and cannot be edited, you need to duplicate the flow file to a custom location. It is recommended to save all flow files on a network location if your SolidConnect settings are also on a network to ensure the flow files can be used by everyone.

☒	SaveAsPDF	C:\tmp\Flows\SaveAsPDF.flow	☒	☒	☒	☒
☒	SaveAsPDF	C:\tmp\OtherFlows\SaveAsPDF.flow	☒	☒	☒	☒
☒	SaveAsSTL	C:\tmp\Flows\SaveAsSTL.flow	☒	☒	☒	☒
☒	SaveAsSTL	C:\tmp\Flows\SaveAsSTL.flow	☒	☒	☒	☒

These colors represent that flows are referenced multiple times. Or that a flow name is found multiple times, the red color can also represent that the flow file isn't found. In this case it will completely ignore the flow. You can see why exactly the flow is highlighted by hovering over the name or path.

On the top side of the screen you can click the plus button to add a new flow, this will show two options:

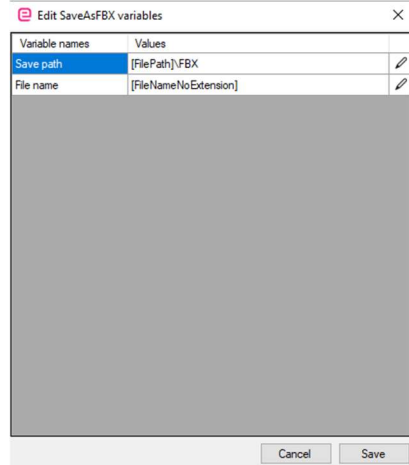
+

Add from Flow Library
Add from File

You can choose to a from the Flow Library, this will open an interface with all flows provided by the application, with a description. When adding a flow from the flow library, you will always be asked where to save it as it will always be a copy. If you already have a flow file, you can add it with the second option. Press the minus button to remove selected flows from your settings, the files will persists. Here you will also find the edit and duplicate buttons, the duplicate button will duplicate the flow file that is selected in the table. This will show a pop-up asking where you want to save the duplicated file. You can also change the name of the file, because the duplicated file is not one of our default flows you are allowed to edit these values (read below on how to do so).

By selecting one flow and clicking the edit button you can edit variables of the selected flow.

It is worth noting that relative and absolute paths follow the basic [Path rules](#).



Here we can see every variable the selected flow contains, by clicking on the value cell you can edit the text in this cell. Because variables can be [formulas](#), you can also click the edit button to use the formula editor.

The columns with extensions in the table defines if the flow should, by default be checked or not for any file with that extension. This will make selecting flows easier as you can have a default selection per file type. Example:

(Settings for SaveAsPDF flow)

Include	Flow name	Flow file	.par	.psm	.asm	.dft
☐	Save3dAsJpg	.\Services\FlowHandler\DefaultFlows\Save3dAsJpg.flow	☐	☐	☐	☐
☐	SaveAs3dPdf	.\Services\FlowHandler\DefaultFlows\SaveAs3dPdf.flow	☐	☐	☐	☐
☐	SaveAsDxf	.\Services\FlowHandler\DefaultFlows\SaveAsDxf.flow	☐	☐	☐	☐
☐	SaveAsFBX	.\Services\FlowHandler\DefaultFlows\SaveAsFBX.flow	☐	☐	☐	☐
☐	SaveAsJT	.\Services\FlowHandler\DefaultFlows\SaveAsJT.flow	☐	☐	☐	☐
☒	SaveAsPDF	.\Services\FlowHandler\DefaultFlows\SaveAsPDF.flow	☐	☐	☐	☒
☐	SaveAsSTEP	.\Services\FlowHandler\DefaultFlows\SaveAsSTEP.flow	☐	☐	☐	☐
☐	SaveAsSTL	.\Services\FlowHandler\DefaultFlows\SaveAsSTL.flow	☐	☐	☐	☐

(Default selection in flow table)

Ext.	Filename	SaveAsPDF
	Stoomwals-3001-Samenstelling-A.asm	☐
	Stoomwals-3001-Samenstelling-A.dft	☒
	Stoomwals-1003-Stuurbeugel-A.dft	☒
	Stoomwals-1008-Achterwiel-A.dft	☒
	Stoomwals-1008-Achterwiel-A.par	☐
	Stoomwals-1003-Stuurbeugel-A.psm	☐

2.2.13 About

On the about page you can find general information like version number, build type, email, support portal, etc. [More info about support.](#)

SolidConnect 1.3.4 - Debug Build
Developed by Enginia B.V. <https://enginia.nl>
© 2025 Enginia B.V. All rights reserved.

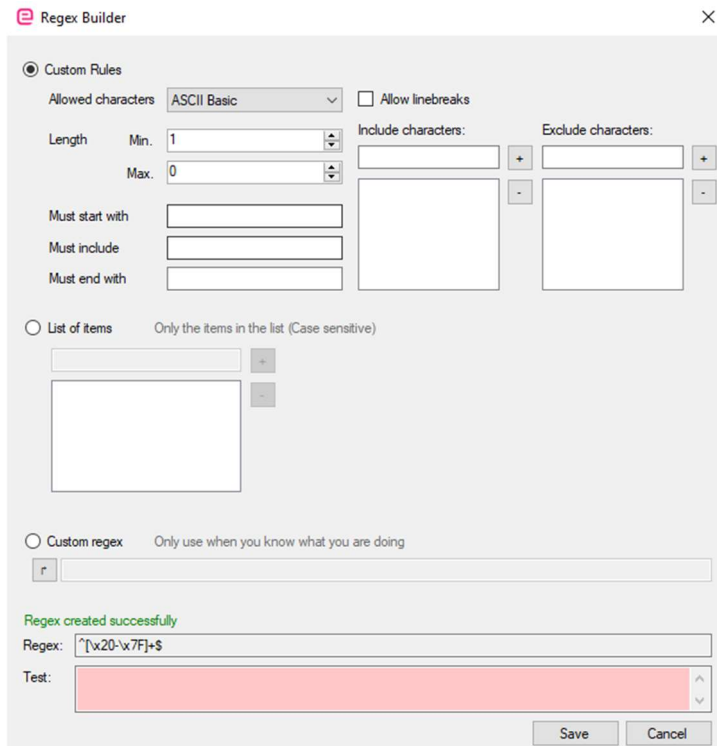
Contact
Email: support@enginia.nl
Phone: +31 85 4891717
Support portal: <https://mijn.enginia.nl>

Export debug package

2.2.14 Permissions

As a system administrator you might want to prevent users to edit settings or attempts to use different settings. SolidConnect currently does not have advanced administrator settings, but we recommend to place your settings file on a network drive and configure write access according to your needs. When a user without write access opens the settings, all settings are disabled and they cannot overwrite the settings file. They would need to explicitly save a copy or load another settings file. There unfortunately is currently no option to prevent these.

2.3 Regex builder



Regex Builder

☒ Custom Rules

Allowed characters: ASCII Basic ☐ Allow linebreaks

Length: Min. 1 Max. 0

Must start with:

Must include:

Must end with:

Include characters: + +

Exclude characters: - -

☐ List of items Only the items in the list (Case sensitive)

+

-

☐ Custom regex Only use when you know what you are doing

[]

Regex created successfully

Regex: ^[x20-x7F]+\$

Test:

Save Cancel

With the regex builder interface you can create simple regexes using basic rules:

Custom Rules: Using basic rules you specify what is and isn't allowed. You can define a minimum and maximum character length, when maximum is 0, it is ignored. You can define text your property value must either start with, contain anywhere or end with. If you wish to add multiple options you can separate them with the pipe character (|). If you need to use the pipe character itself, add a backlash \ before the pipe character. You can define whether line breaks are allowed. Allowed characters are a few options of pre-defined sets of characters that will be allowed. ASCII Basic covers the most basic characters (character code 32-127). ASCII Extended covers extra characters such as éèê and more (character code 32-255). Go to <https://www.ascii-code.com/> to see which characters exactly. Filename Safe allows all characters except those that are not allowed in filenames. Alphanumeric and space is very limited and only allows the basic letters a-z (both cases) the digits 0-9 and the space character. Additionally you can add specific characters to the include or exclude character lists. (you can add multiple characters at once in the textbox)

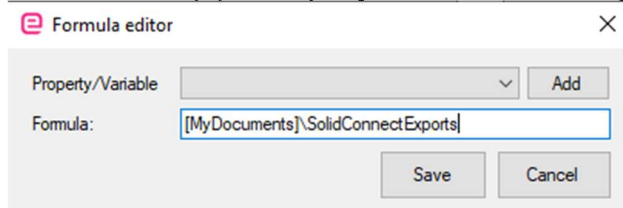
List of Items: The property's value must exactly match one of the values (case sensitive) you define in a list. (you can add multiple options at once by separating them with a comma in the textbox)

Custom Regex: If you understand regex you can write your own. Note that it must match the whole input value to pass so using ^\$ is enforced.

At the bottom is a test box, in here you can test if any input passes the regex, the background will color green or red accordingly.

2.4 Formulas

Solid Connect uses the concept of formulas on many places, especially for exporting. Formulas are built from normal text and properties. Example: '[Title]_[Product number]'. In this example the title of a product and Product number are combined with an underscore in between. This method is also found in SolidEdge at Settings>Helpers>Name formula. At most places you can use a formula there will be a button with three dots (...), when you press this, the following window will pop up:



Here you can easily append a property to the existing formula by first selecting it from the dropdown, then pressing Add.

2.5 Path rules

Anytime you can work with formulas within SolidConnect to create paths, you can use relative paths and absolute paths. Relative paths will always be relative to their context, for related files this is the source file of the product, for batch related actions this is relative to the processed document.

Examples:

- 'C:\Test' or '\\Server\Test' are absolute paths
- 'C:\Test\[project name]' path with formula
- '\Test' is relative folder to the drive 'C:\Test'
- '.' In the same folder
- '\Test' or 'Test' is relative folder
- '../' One folder above

2.6 License

SolidConnect has a few different license types:

- Trial: you can request a trial directly from the application. Only one trial can be requested per computer
- Subscription: When you have a paid subscription to SolidConnect you have access to tokens, tokens allow you to unlock features of SolidConnect. Each feature requires some amount of tokens, but you can activate/deactivate these features after a delay, this way you can choose/test/change different features or move them to a different user of the same license.
- Beta test license: For specific users we give out a beta test license, this license is the only license that works on our beta version, this same license also allows you to use the release version on the same computer.
- Offline license: For specific users an offline license is possible, this is only for users who cannot give internet access to SolidConnect. The activation for this license is described below.

The license system of SolidConnect still has some performance improvements to be had, please be patient while it is busy.

2.6.1 Request a trial

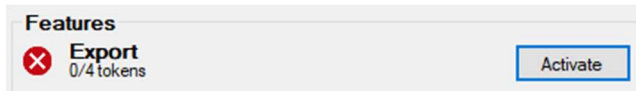
1. Open the license manager with this button  at the top right
2. A window will popup, press the big button that says 'Get a free trial' at the top. If this button says 'Trial already used' you probably have already requested a trial before on this computer, if you believe this is wrong, contact enginia.
3. Another window will popup asking for some basic information such as your name, company and email address. Once you are done entering your information, press Confirm.
4. Please wait while your trial is being requested. If the loading window disappears without an error, your trials should be active.

2.6.2 Activate a license

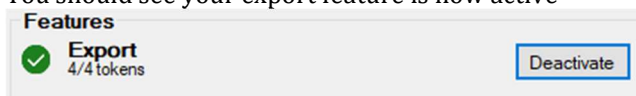
1. Open the license manager with this button  at the top right
2. A window will popup, press the second big button saying 'activate your key'. If this button says 'activate a different key' you likely already have a license, you can press it if you want to activate a different key. You can always enter your current key again, it might help resolve issues.
3. If the loading window disappears without an error, your license should be active.

2.6.3 Using tokens

1. If you have an valid license you should see activate/deactivate buttons besides each feature as shown below:



2. Every feature of SolidConnect costs some tokens, the Export feature shown above costs 4 and is currently not active
3. You can press activate to spend 4 tokens to activate this feature. You can see how many tokens are available in the License info section: **Tokens: 8/36 available** Please wait patiently while your tokens are being checked and activated.
4. You should see your export feature is now active



5. You can press deactivate, to deactivate the feature again and free those token for other features or computers. This however has a delay, you cannot immediately deactivate a feature after activating it. Please wait patiently while your tokens are being checked and deactivated.

2.6.4 Offline license activation

1. Install SolidConnect on the destination computer
2. Go to the application files and run 'Activate Offline License Step 1'
3. A window will popup with a unique code for that computer
4. Send this code (along with you activation key) to us, we will then send back a 'computer key' which you need to finish the activation
5. When you have received your computer key, go to the application files and run 'Activate Offline License Step 2'
6. A window will popup, enter your activation key, press ok
7. A second window will popup, enter the computer key you received from us, press ok
8. If no warning pops up, your license should now be active, you can run SolidConnect normally to verify.

If you want to deactivate this license, you will need to go to the application files and run 'Deactivate Offline License' then send this verification code to Enginia (valid for 1 day).

3 Usage / Guide

3.1 Installation

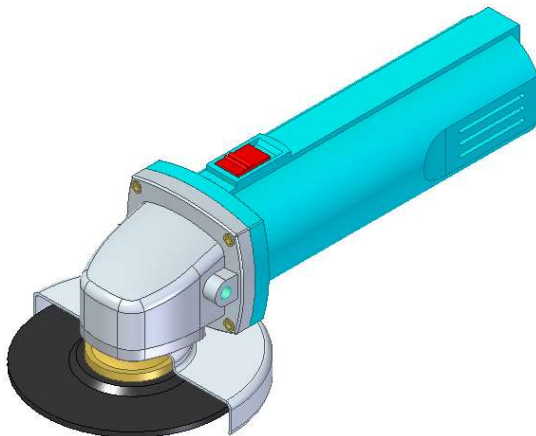
SolidConnect provides an installer with which it can be installed. After installation you can request a trial to get access to all features, you will be prompted the first time you start SolidConnect. You can also press the [License Manager](#) button. Read more about licenses and trials [here](#).

3.2 Sample case

To make the explanation more easy and more specific, an example of an assembly document will be used. For this, the 'Grinder-assembly' from the training folder of Solid Edge is used. You can find this document at this location:

C:\Program Files\Siemens\Solid Edge 202x\Training\Grinder.asm

Do notice that the 'x' in the path depends on your Solid Edge version. Throughout the next chapters, when the sample case is mentioned, it will be this sample case.



3.3 Loading

3.3.1 Loading active document from Solid Edge

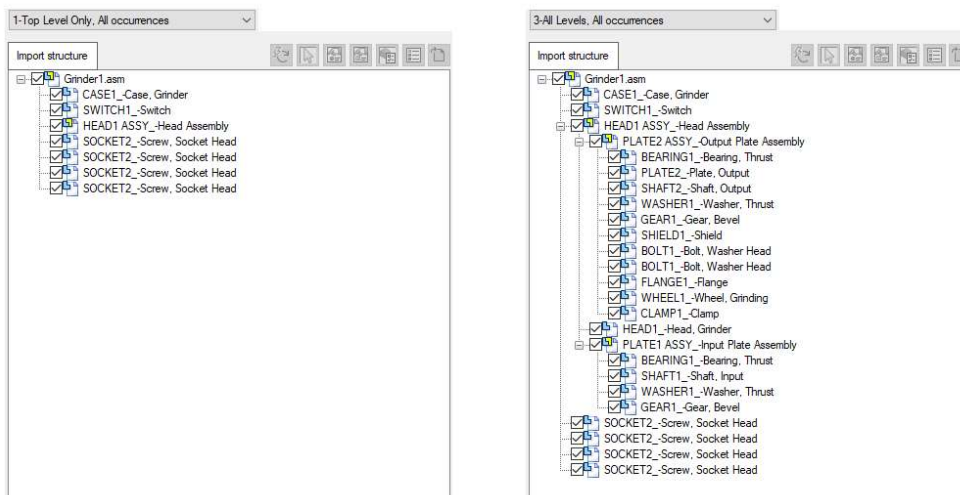
The first step you need to take, is to connect SolidConnect with an assembly document in Solid Edge.

Do the following:

1. Open Solid Edge.
2. Open the assembly of the sample case.
3. Click the Load-button in SolidConnect.

Loading from SolidEdge can be done with one of four modes. When using a 'Top Level Only' option, only parts that are NOT inside another assembly are loaded. The 'All Levels' option will load everything contained in the loaded assembly. When using a 'Visible Only' option, any hidden parts are not loaded. When using 'All Occurrences', all occurrences are unhidden in Solid Edge and will be loaded. Note that loading can take a while depending on the size of your assembly. Based on your configuration, all parts might need to be activated in Solid Edge, this is done automatically but can take a while and cannot be cancelled. When you edit a sub-assembly in-place, only that assembly will be loaded. When you have opened a part used in your assembly in another tab, this can cause problems loaded that part, it is recommended to close them. Additionally, it is recommended to close menu's such as file properties and save your assembly before loading as this can also cause issues reading that part.

The difference between Top Level (left) and All Level (right) is shown in the tree of the import structure:



3.3.2 Loading plmxml

SolidConnect has some basic capabilities for reading a plm xml file exported from TeamCenter. When pressing the load button you are prompted to select a plm xml file to load. Because plm xml is quite flexible, not everything will be supported. The following data from the xml will be supported:

- Design/ Product nodes, and any UserValues in it's UserData
- DesignRevision / ProductRevision nodes, and any UserValues in it's UserData
- Occurrence nodes in a ProductView node and any UserValues in it's UserData
- Classification properties in 'ICO' nodes that refer to a 'UserClass' nodes

These properties however need to be configured differently in setup:

- Any UserValues need to be added to the property analysis tab. For UserValues of Design, Product, DesignRevision, ProductRevision need to go in the file properties list, for Occurrence in the occurrence properties list.
- For Classification properties, you need to add them to the file properties list as 'classification.name' (name of the property of the classification with 'classification.' before it)

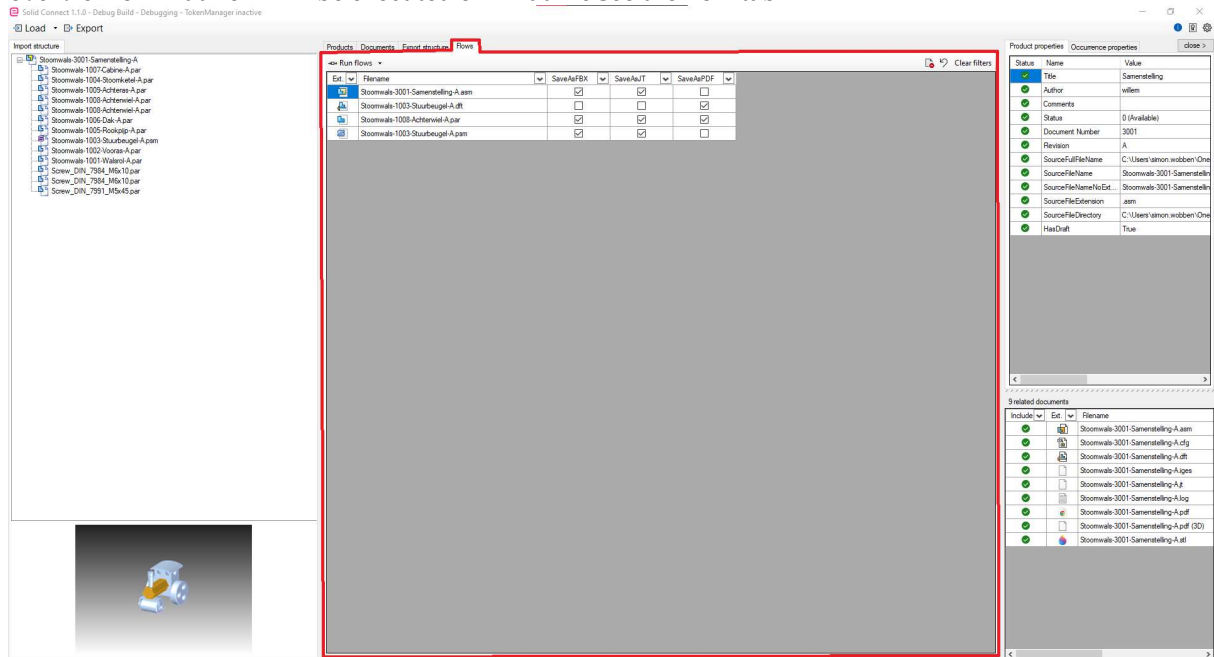
3.4 Including/excluding parts

SolidConnect has several ways you can customize what occurrences will be included for further actions such as exporting and publishing. You can do this manually by (un)checking each occurrence in the import structure view or by selecting documents from the list and using the button in the top right to automatically exclude/include all occurrences of that document. However, it is recommended to use the following automated data-driven options:

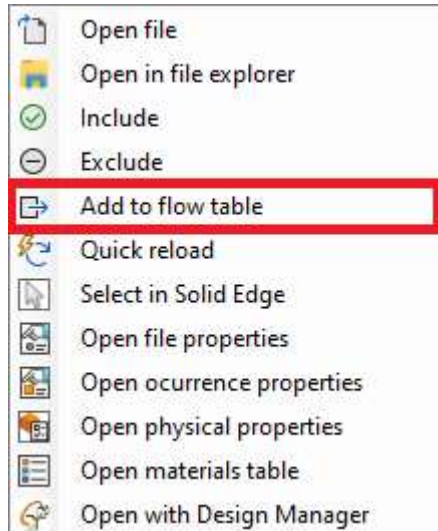
- Setting the Occurrence property 'Reports/Part List' to No. This will automatically exclude that occurrence and any sub-occurrences.
- Creating custom file properties in combination with an exclude rule. (See [Setup](#))

3.5 Running the batch handler

The batch handler is a service that can execute hundreds of tasks on hundreds of files, in a logic based process. These processes are saved in flow files and can be added to your Flow settings. To see an overview of what flow will be executed on what file see the flow tab.

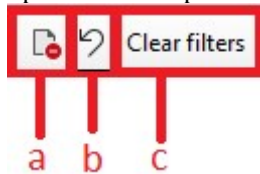


To add files to this flow tab use the context menu in the documents tab or the products tab.



The products tab also contains the  button to add all products to the flow tab.

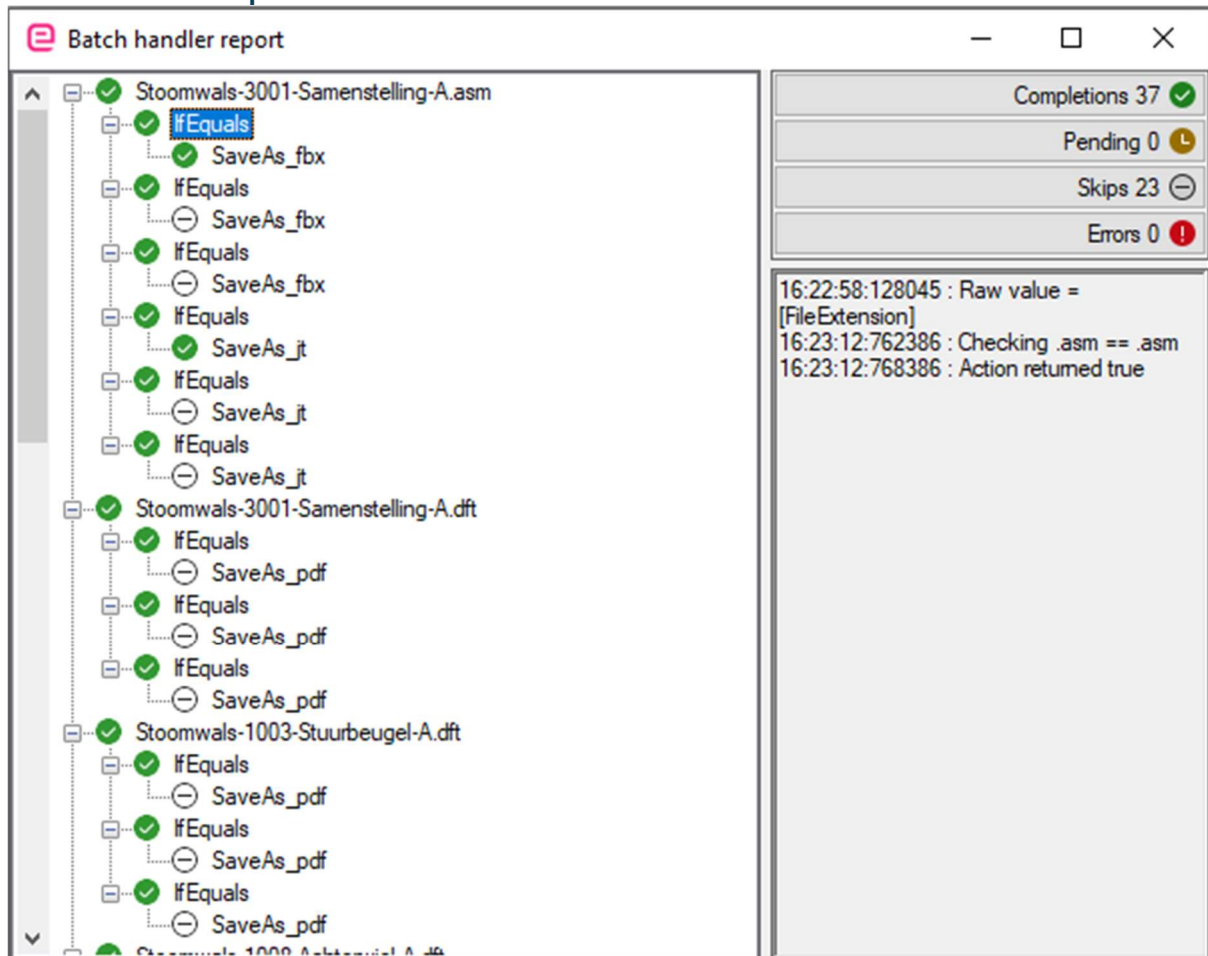
In the documents tab the selected documents will be added individually. But in the products tab the selected product will add all related documents from the product. This action will also pre filter all usable files, meaning that any file that we have no actions for will not be added to the flow tab. Once your files are added to the flow tab we can see them with the default selected flows in the table. By clicking the “Run batch” button all checked flows will be executed upon their accompanying files. When this process is done a report screen opens which will show all actions and their states, along with all their logs. Inside the “Run batch” button you can also press the arrow to continue the last batch (after cancelled or crashed) or to open the last report screen again.



These buttons can be found on the top right of the flow tab.

- a. Will remove all selected rows from the flow tab
- b. Will revert the flow selections to the default defined selections
- c. Will clear all filters from the table

3.5.1 Batch report



In this screen we can see that 37 actions completed successfully and 23 actions were skipped. If an action is skipped this is most likely due to a parent action returning either false or true, depending on what outcome the action was expecting.

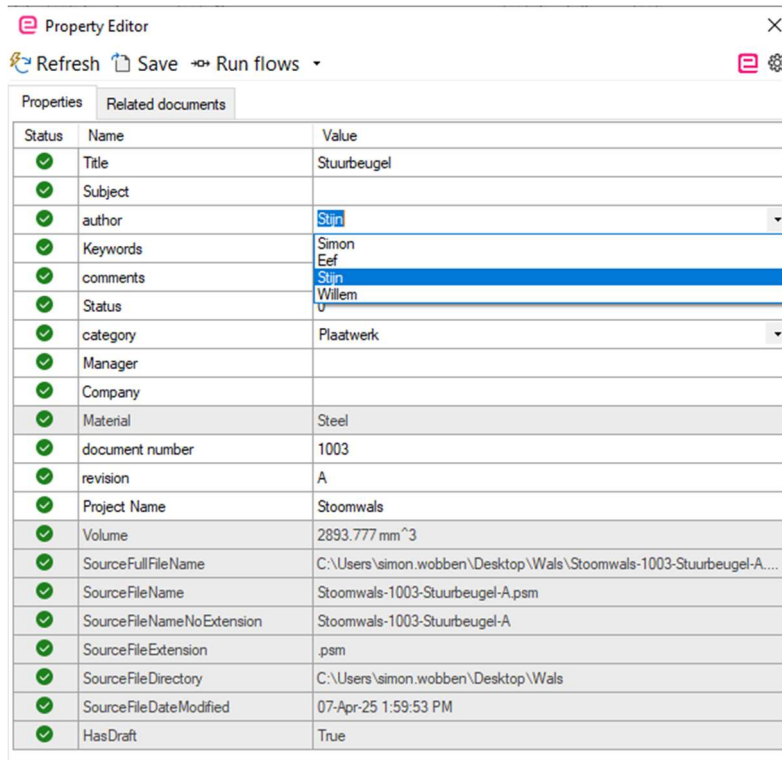
The buttons on the top right are filter buttons, clicking one of these buttons will update the list to show files where that state is found. To go back to no filter click the button of the currently selected filter. This will revert to the original and full list. Clicking a file in this list will show you all logs from all actions of this file. By selecting a single action you will be shown only those logs, as shown in the image above.

If there are still pending actions this means the batch handler did not finish its process, you can continue this process by closing the report screen and clicking the “continue” button which is located in the dropdown menu of the “run batch” button.

If you encounter any errors please contact Enginia support and, share the action logs.

3.6 Using the property editor

The property editor is a separate tool that can be used to validate and change values of file properties. The property editor can be started from the context menu in SolidConnect or can be run separately.



Status	Name	Value
✓	Title	Stuurbeugel
✓	Subject	
✓	author	Stijn
✓	Keywords	Simon Eef
✓	comments	Stijn Willem
✓	Status	U
✓	category	Plaatwerk
✓	Manager	
✓	Company	
✓	Material	Steel
✓	document number	1003
✓	revision	A
✓	Project Name	Stoomwals
✓	Volume	2893.777 mm^3
✓	SourceFullName	C:\Users\simon.wobben\Desktop\Wals\Stoomwals-1003-Stuurbeugel-A...
✓	SourceFileName	Stoomwals-1003-Stuurbeugel-A.psm
✓	SourceFileNameNoExtension	Stoomwals-1003-Stuurbeugel-A
✓	SourceFileExtension	.psm
✓	SourceFileDirectory	C:\Users\simon.wobben\Desktop\Wals
✓	SourceFileDateModified	07-Apr-25 1:59:53 PM
✓	HasDraft	True

The property editor shows all properties and its values, which you can edit unless greyed out. When you press the save button or ctrl-S the properties will be save to SolidEdge. The property editor can suggest a list of values based on the property analysis rules if you add an analyze rule to a property and in the regex builder choose the list of values option.

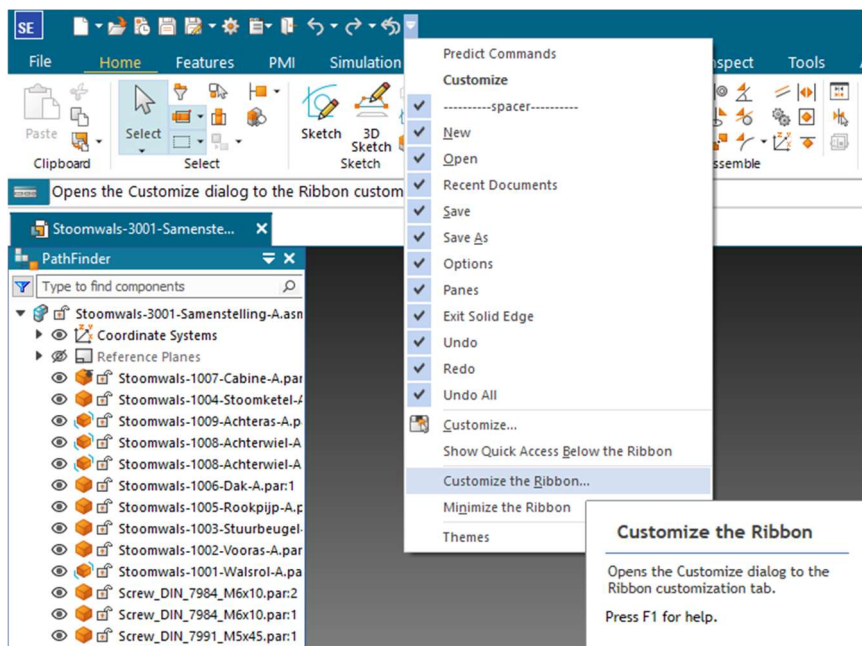
Only properties that have a import setting for SolidEdge can be changed, otherwise the fields will be read only. This automatically makes properties that are created by Solid Connect like “HasDraft” or “SourceFileName” readonly. Material is a specific exception which can never be changed because SolidEdge handles material differently.

The property editor also has a tab for related documents, based on the [related files setting](#).

You can also run the default selection of flows (defined in your settings file) on the current file. Only the flows that are on by default for this file extension will be executed. The current file being the file that is open in the property editor at this time. By clicking the expand button on the “Run flows” button you can also run a singular flow on the file, the list of flows that is shown is based on the included flows in your settings file. See [flow settings](#) and [running the batch handler](#) for more info about flows.

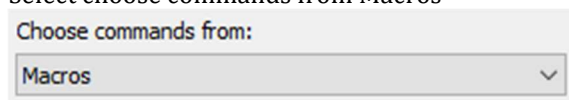
3.7 Adding as macro to SolidEdge

To add SolidConnect or the property editor to SolidEdge, hover over the dropdown icon in your ribbon, and click “Customize the Ribbon”.

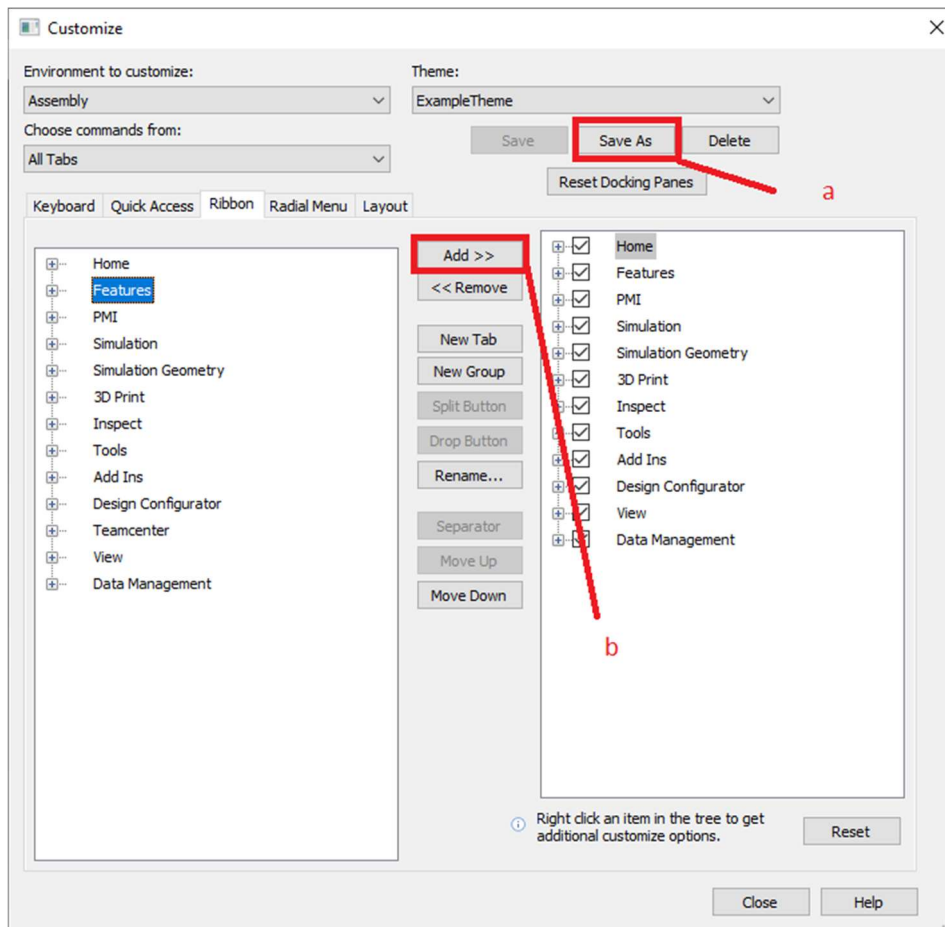


Then, in the customize screen click save as to duplicate the default theme, or select the theme you want to add the button to.

Select choose commands from Macros



Then press Browse at the bottom and navigate to the folder where SolidConnect is installed and press select folder. Then you should see a list of files that are in the folder. Here you can either choose SolidConnect.exe or PropertyEditor.lnk to add as a macro.



Technical details

If in Solid edge 'don't report' is set to Yes, the occurrence will be excluded automatically, overriding this to export manually with the checkmark in Solid Connect can cause incorrect counts.

Solid Edge has many unique object types that are programmatically very different and support for them may vary:

- Patterns: supported, but assemblies cannot be patterned as of now (undefined behavior)
- Mirrors: partially supported, but assemblies cannot be mirrored as of now (undefined behavior), mirroring a part that is already mirrored is not fully supported and will be loaded as if it is a normal part, it will lose unique properties (such as pipe length)
- Frames: supported, if defined so in Setup, their cut length and cut angles can be read and treaded as different parts.
- Pipes: unsupported
- Electrical routing: unsupported
- Tubes: unsupported
- Simplified Assemblies: sub parts will not be read when used as simplified. You can load all assemblies as designed from the top level assembly > right-click > simplified/adjustable > ctrl-click use designed assembly
- Family of assembly: partially supported, take caution

4 Troubleshooting

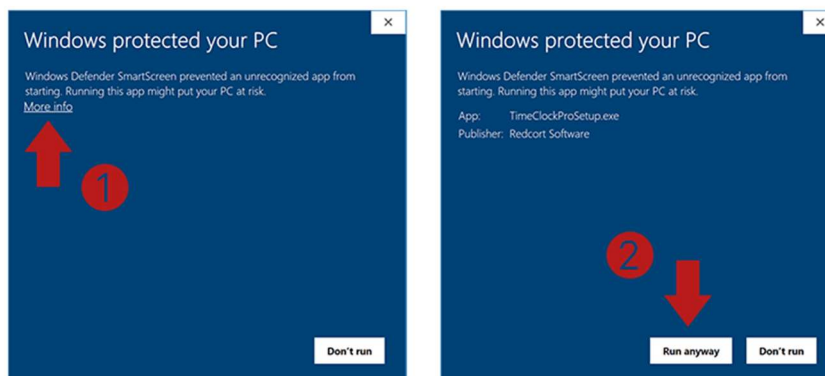
4.1 Contacting support

If you encounter any issue you cannot resolve yourself or with help of this manual you can contact support via our support portal: <https://mijn.enginia.nl>, email: support@enginia.nl or phone: +31 85 4891717.

When contacting support, please try to provide detailed info about your issue and your steps before encountering your issue, additionally please also provide a debugging package which can be exported from setup>about. This will make a zip file of all your recent logs, batch handler files, setting file and other debugging necessary files.

4.2 Windows protected your pc

Because SolidConnect is still a relatively new application, Windows Defender SmartScreen might prevent the application or even the installer to launch. The following blue window will pop up, try the following steps:

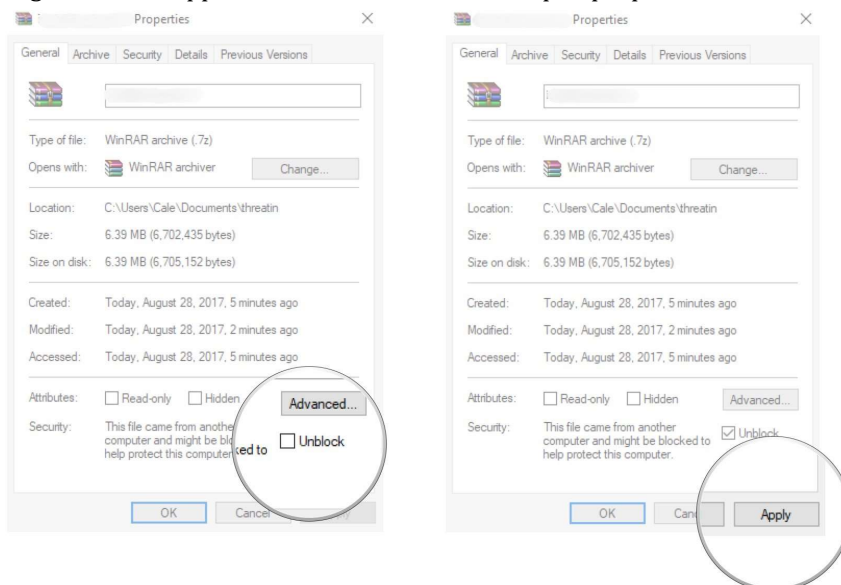


1. Press 'More info'
2. Press 'Run anyway'

If the 'Run anyway' button does not appear, press 'Don't run' and try an alternative solution.

Unblock in file properties

1. Right click the application or installer file and open properties.



2. If there is a 'Unblock' checkbox, check it and press apply
3. Try running the application again.

Disable Windows Defender SmartScreen

Although not recommended, SmartScreen can be disabled through Windows Defender. If you must completely disable SmartScreen, we recommend re-enabling it soon after.

1. Launch 'Windows Security' from your Start menu, desktop, or taskbar
2. Navigate to the 'App and browser control' tab
3. Turn Off the Check apps and files setting. (this might be disabled by your administrator)