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CUSTOMTOOLS
GETTING STARTED
GUIDE

CUSTOMTOOLS

CUSTOMTOOLS

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Chapter 1: Introduction

Read this first

CUSTOMTOOLS offers a powerful set of tools that helps the SOLIDWORKS users to optimize the management of their SOLDIWORKS data. CUSTOMTOOLS allows its users to automate certain routines when it comes to managing data. The software can easily be configured based on the specific needs of its users.

About this manual

The main aim of this manual is to give a brief insight into how the main tools of CUSTOMTOOLS can be configured. This manual contains 13 chapters:

CHAPTER 1: INTRODUCTION

Provides information about this manual.

CHAPTER 2: ADMINISTRATION TOOL

Manage the CUSTOMTOOLS database, profile, users, user groups and link to external data source.

CHAPTER 3: STARTING CUSTOMTOOLS IN SOLIDWORKS

Activate the CUSTOMTOOLS Add-in in SOLIDWORKS.

CHAPTER 4: IMPORTING OLD DESIGN DATA INTO THE CUSTOMTOOLS DATABASE

Import the file references of the SOLIDWORKS documents that were designed before using CUSTOMTOOLS in the CUSTOMTOOLS database.

CHAPTER 5: CUSTOMTOOLS OPTIONS

Configure the CUSTOMTOOLS profile, manage your dictionaries and user preferences.

CHAPTER 6: BATCH PRINTING YOUR SOLIDWORKS DRAWINGS

Automate the printing of your SOLIDWORKS assembly so that drawings are always printed to the right printer, size and orientation.

CHAPTER 7: BATCH CONVERTING YOUR SOLIDWORKS DOCUMENTS

Automate the conversion of SOLIDWORKS assembly in all file formats available in SOLIDWORKS.

CHAPTER 8: PROPERTY MANAGEMENT

Optimize the management of your property.

CHAPTER 9: SEARCHING DOCUMENTS WITH CUSTOMTOOLS

Search documents based on the file type, project or property value.

CHAPTER 10: EXCEL REPORTING

Generate an Excel report from the top assembly containing custom properties and preview image.

CHAPTER 11: AUTOMATIC FILE NAMING

Automate your own file naming conventions.

CHAPTER 12: MANAGING YOUR PROJECTS

Manage your file based on projects or sub-projects. Load default property value or defined specific destination folder where to save the SOLIDWORKS documents for each project.

CHAPTER 13: COPYING AND RENAMING ASSEMBLIES

Copy and rename SOLIDWORKS assemblies by using defined file naming conventions and modifying the property values of the document to copy.

Intended Audience

This document is designed for the Administrator and new users of CUSTOMTOOLS.

Late Changes

This document might not include all the enhancements of CUSTOMTOOLS 2015.

Using this Document

This document has been written by following the SOLIDWORKS conventions.

Conventions used in this document

Convention	Meaning
Bold	Any CUSTOMTOOLS command or menu item.
Italic	Refers to example
	Tip, Note

CUSTOMTOOLS

Chapter 2: Administration tool

Introduction

What does it do?

The **CUSTOMTOOLS Administration** tool is used to manage the CUSTOMTOOLS database, profile, users, user groups and link to external data source.

How does it work?

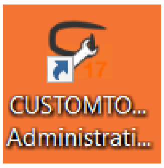
The **CUSTOMTOOLS Administration** tools does not require a SOLIDWORKS license. It is designed to be used by the administrator of CUSTOMTOOLS. Only one license of the **Administration tool** is needed in order to run the **SOLIDWORKS Add-in of CUSTOMTOOLS** in a single/multi user(s) environment.

How can you use it?

The **Administration Tool** can be accessed from:

- The Desktop

If the **Administration Tool** has been selected during the installation, then CUSTOMTOOLS will automatically create a Desktop icon.



- The Program files folder

The **Administration Tool** can also be accessed from the following path:
C:\Program Files\ATR Soft\CUSTOMTOOLS 2015\CUSTOMTOOLS Administration.exe

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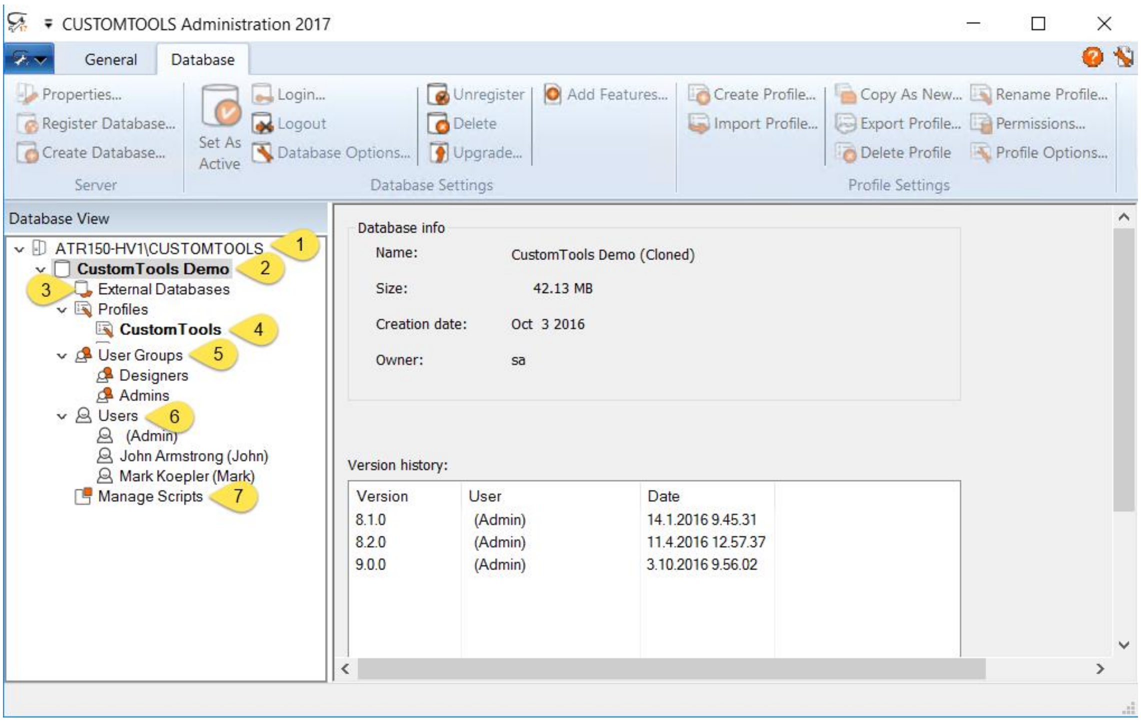

1. Server where the CUSTOMTOOLS database is/will be installed.
2. CUSTOMTOOLS database where the files references of the SOLIDWORKS files are stored.
3. **External Databases**: Create a link to an external data source.
4. **Profiles**: Where all the settings and rules are defined and stored (*e.g. custom properties, print and conversion profiles…*).
5. **User Groups**: Group of CUSTOMTOOLS users.
6. **Users**: Individual CUSTOMTOOLS users.
7. **Manage Scripts:** Manage the CUSTOMTOOLS script *(e.g. Excel report…)*.

CUSTOMTOOLS’ Server

The CUSTOMTOOLS server is used to store the CUSTOMTOOLS database. CUSTOMTOOLS uses an SQL Server to store its database. The **Microsoft SQL Server Express Edition** is included in the CUSTOMTOOLS installation but CUSTOMTOOLS can also use an existing SQL Server.

The server where the CUSTOMTOOLS database will be installed should be accessible for all the different CUSTOMTOOLS’ users.

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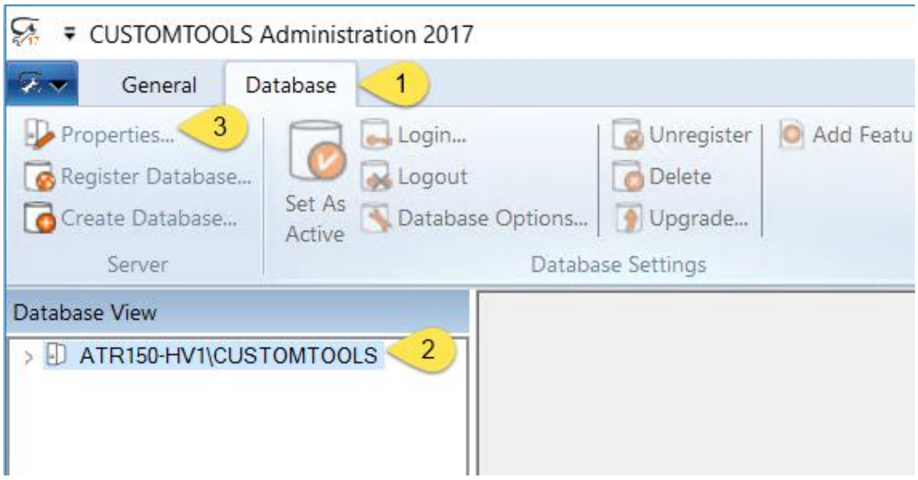
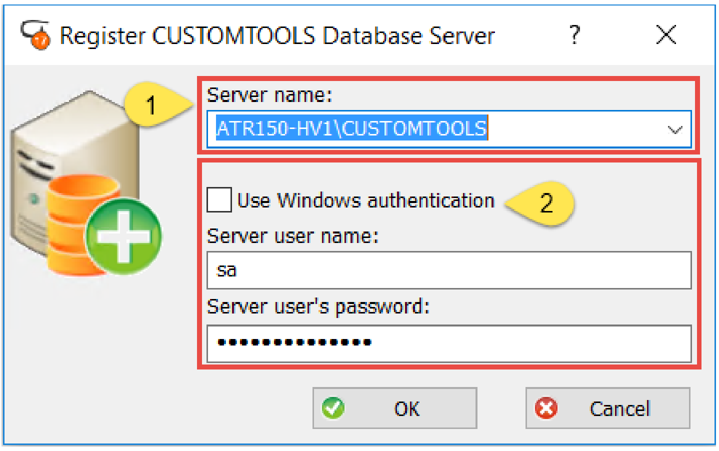


To open the **CUSTOMTOOLS Database Server** dialog where the server to install the **CUSTOMTOOLS Database** can be selected:

1. Activate the **Database** tab,
2. Defined **Server** where the CUSTOMTOOLS database will be installed,
3. Click on **Properties**... to select a different server.



1. From the **Server name** select the server where the CUSTOMTOOLS Database will be installed.

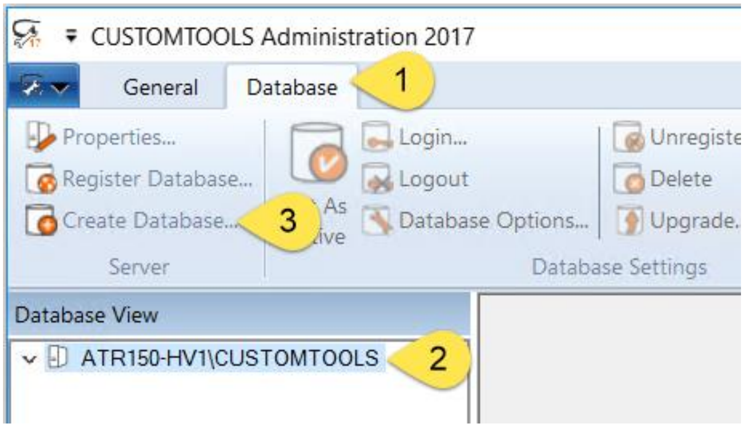
   By clicking on the ⌄ key then the list of available servers that can be accessed from the current machine appears.
2. To connect to the selected Server, select the **Use Windows authentication** check box or the SQL sa user credential. If the SQL Server was installed with CUSTOMTOOLS, then the credentials for the user sa were defined during the installation. If you are connected to an existing SQL Server then request the SQL sa user from your IT Admin.

CUSTOMTOOLS Database

The CUSTOMTOOLS database is used to store the SOLIDWORKS Documents file references. The file references are automatically imported into the CUSTOMTOOLS database on save operations for documents that are created with CUSTOMTOOLS. The SOLIDWORKS Documents that were created before using CUSTOMTOOLS can be imported into the CUSTOMTOOLS database, by using the **Import** functionality of CUSTOMTOOLS.

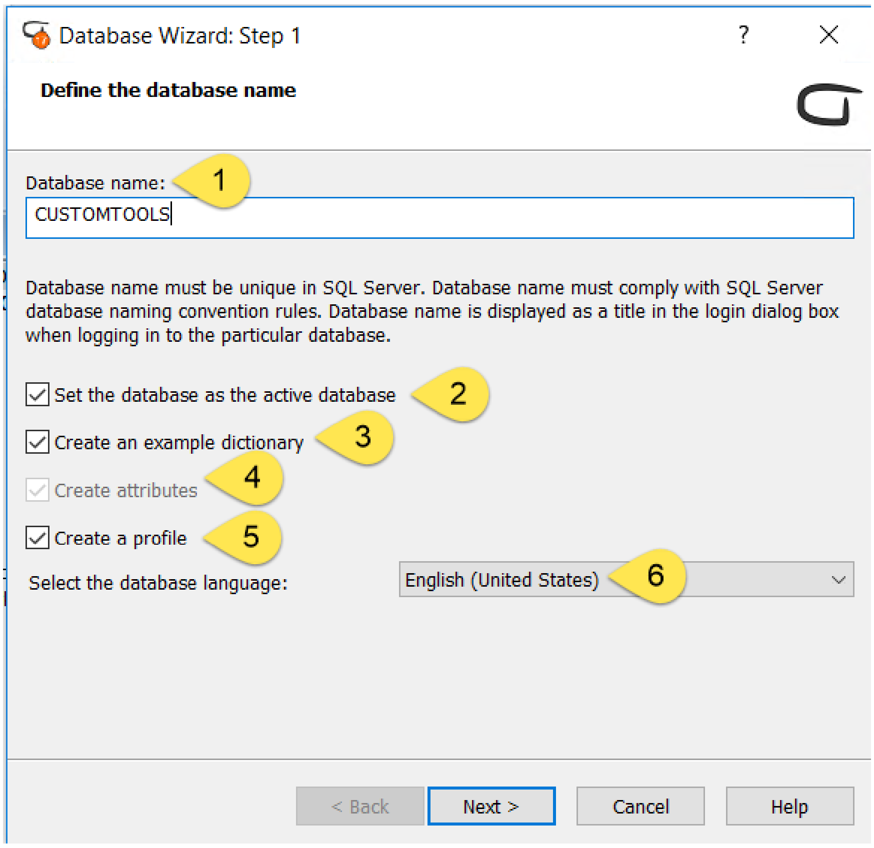
NOTE: Only one database is needed for a multi/single user(s) environment, as all the users are connecting to the same database.

Create a new CUSTOMTOOLS database

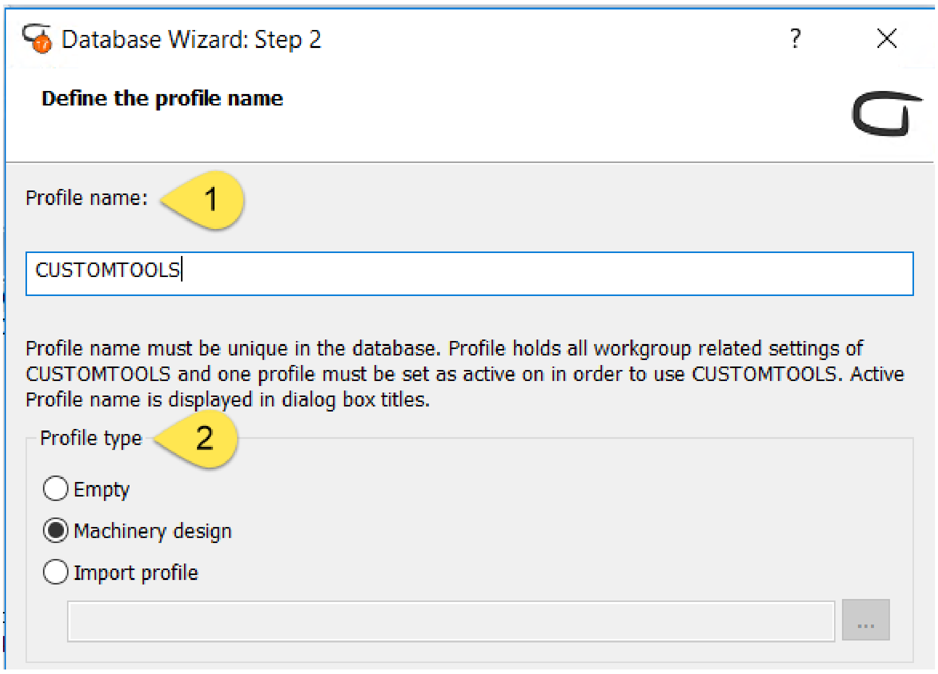


To create a new database,

1. Select the **Database** tab,
2. Select the **Server** where to install the CUSTOMTOOLS database,
3. Click **Create Database...** Then follow the wizard.



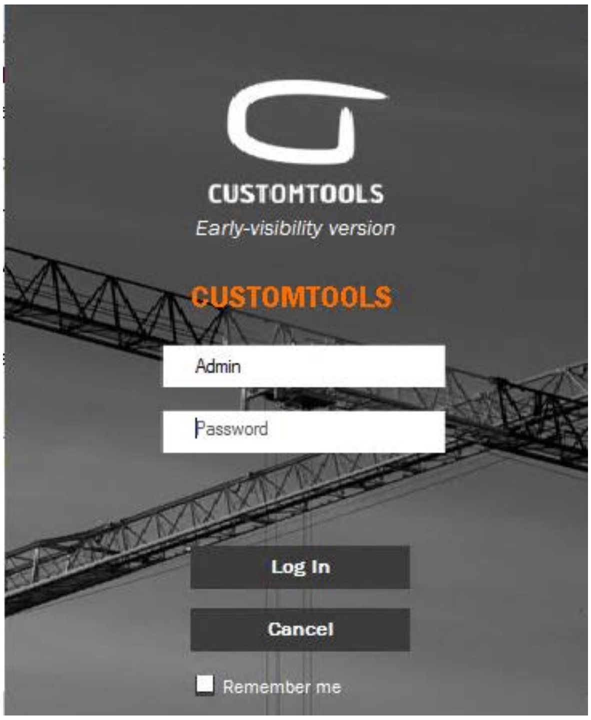
1. **Database name**: Define the name of the CUSTOMTOOLS database.
2. **Set the database as the active database**: The new database will be automatically selected.
3. **Create an example dictionary**: Dictionary can be used to translate custom properties during the printing and conversion operations.
4. **Create attributes**: Attributes are the link between the CUSTOMTOOLS Database and the custom properties of the SOLIDWORKS Documents.
5. **Create a Profile**: The profile is used to store the settings and configurations and is shared between the different users.
6. **Select the database language**: Select in which language the profile will be created.



1. **Profile name:** Define the name of the profile (*e.g. Company name*).
2. **Profile type:** CUSTOMTOOLS offers different possibilities to create a profile:
    a. **Empty:** The profile does not contain any settings and needs to be fully defined.
    b. **Machinery design:** Sample profile that contains pre-defined settings.
    c. **Import profile:** Import a profile from CUSTOMTOOLS (*.Ctprof*), CT3 or Property Manager.

TIP: It is recommended to use the **Machinery design** profile as a base and then modify the profile based on the requirements.

Log in into the CUSTOMTOOLS database



The database **Log in** dialog will appear once the CUSTOMTOOLS database has been created. By default, CUSTOMTOOLS will offer to log in with the CUSTOMTOOLS **Admin** User. Press **Log in** without defining any value for the **Password**.

NOTE: By default, the Admin user does not have any password. A password can be later added by editing the Admin user.

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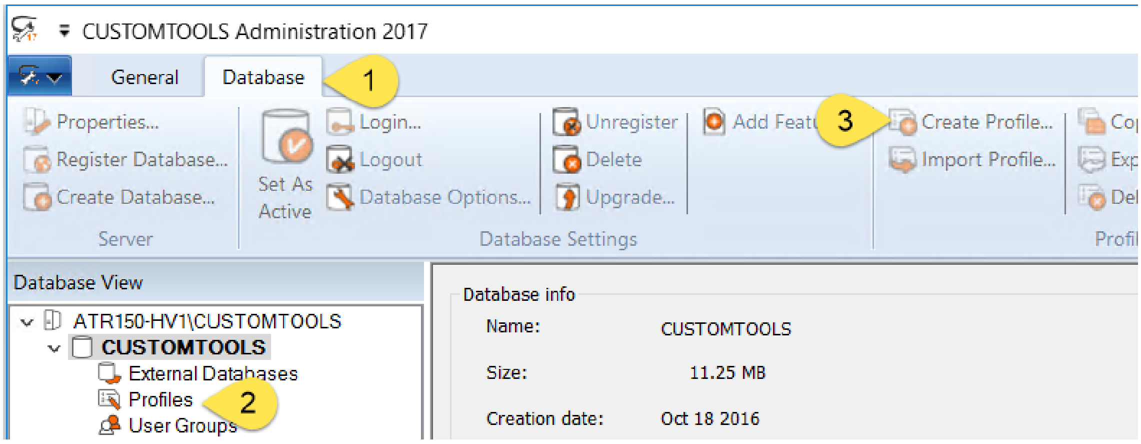
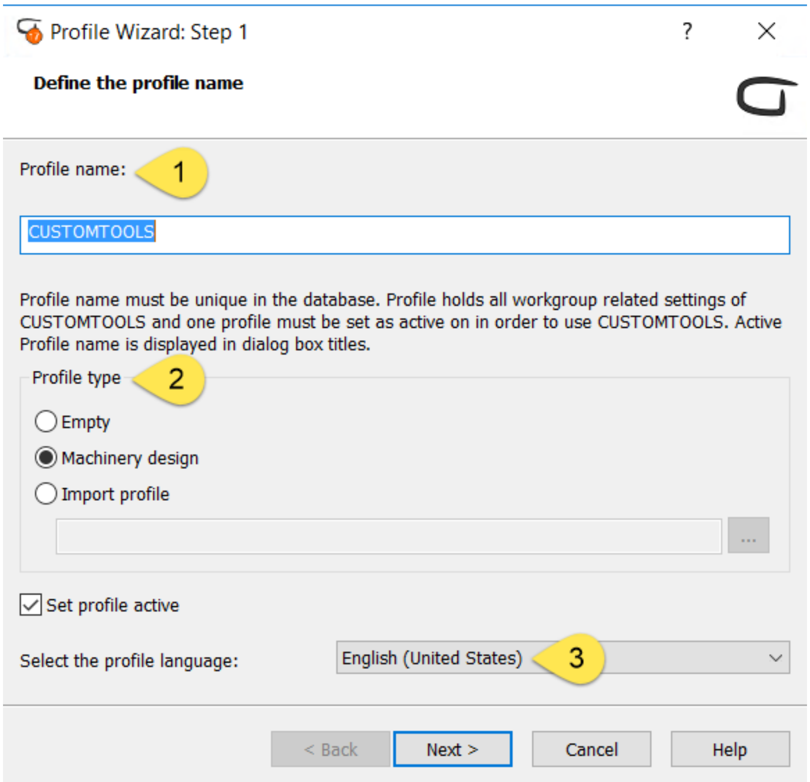
CUSTOMTOOLS profile

The CUSTOMTOOLS profile is used to store all your settings (*e.g. Custom properties, printing and conversion profiles, file naming rules and convention…*). The same profile is shared by multiple users.



To create a new profile, log in to the **CUSTOMTOOLS database**,

1. Activate the **Database** tab,
2. Select **Profiles**,
3. Click on **Create Profiles…** then follow the wizard.



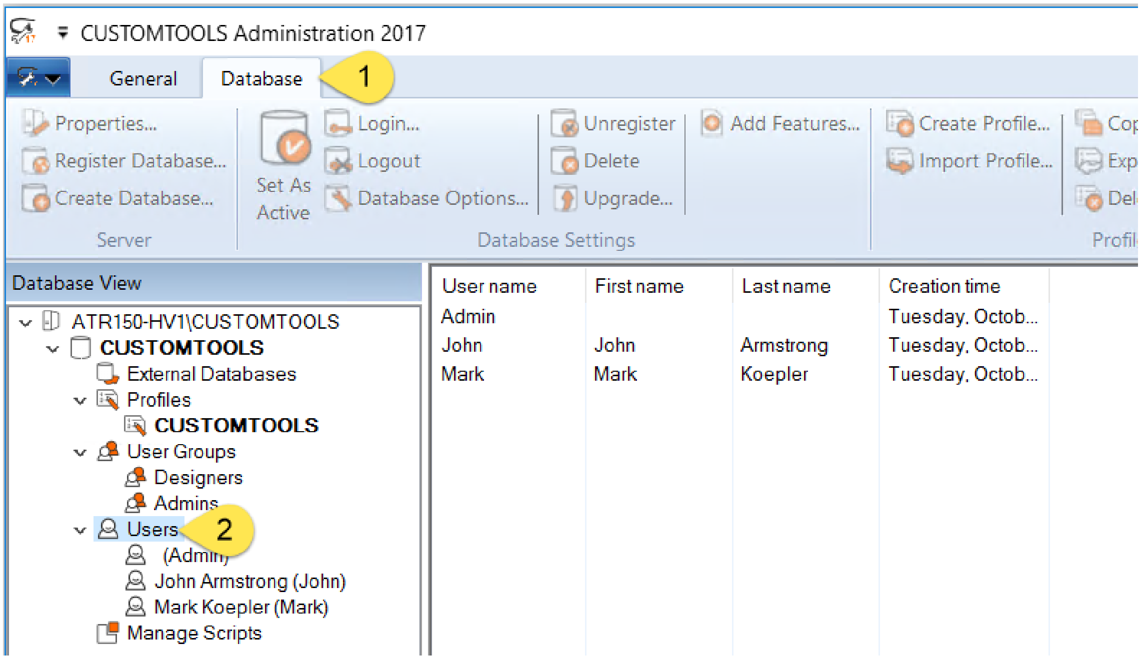
1. **Profile name:** Define the name of the profile.
2. **Profile type:** CUSTOMTOOLS offers different possibilities to create a profile:
    a. **Empty:** The profile does not contain any settings and needs to be fully defined.
    b. **Machinery design:** Sample profile that contains pre-defined settings.
    c. **Import profile:** Import a profile from CT3 or Property Manager.
3. **Select the database language:** Select in which language the profile will be created.

TIP: It is recommended to use the **Machinery design** profile as a base then modify the profile based on the requirements.

Managing Users and User Groups

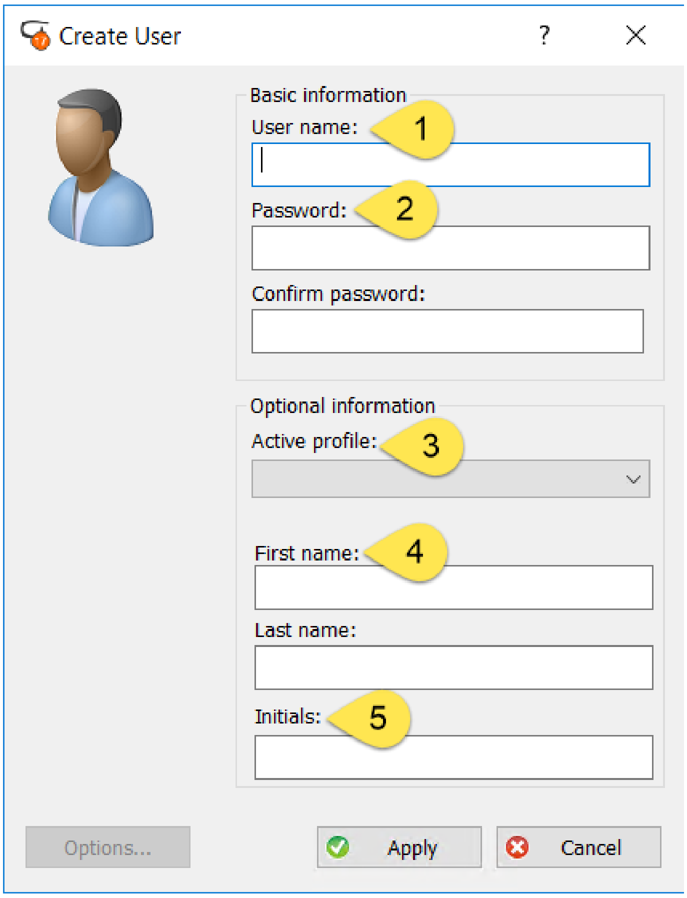
Create a new CUSTOMTOOLS user

A CUSTOMTOOLS user must be defined for every SOLIDWORKS or non-SOLIDWORKS users who are going to use CUSTOMTOOLS in order for them to access the CUSTOMTOOLS database.



To create a new user,

1. Activate the **Database** tab,
2. Right click on **User,**
3. Click **Create New...**



1. **User name**: Define the user name that will be used by the CUSTOMTOOLS user to log in into the **CUSTOMTOOLS Log in** dialog.
2. **Password**: The password that will be used by the CUSTOMTOOLS user to log into the CUSTOMTOOLS database in the **CUSTOMTOOLS Log in.**

Note: It is not compulsory to define a password. The **Password** and **Confirm password** fields can be left empty.

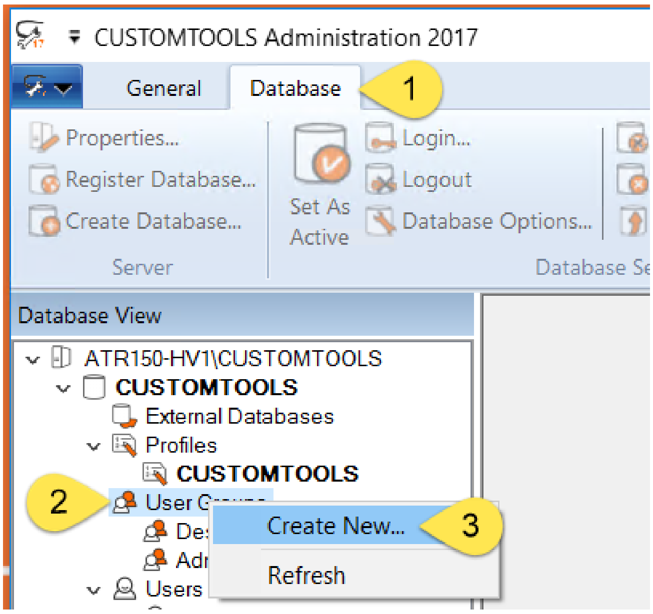
Optional information:

3. **Active profile**: Select the CUSTOMTOOLS profile to be used by the user.
4. **First name/Last name**: Define the first and last name of the user.
5. **Initials**: The initials defined for the user can be linked to a property.

Note: The initials can be loaded automatically when the current logged in user creates a new document in SOLIDWORKS.

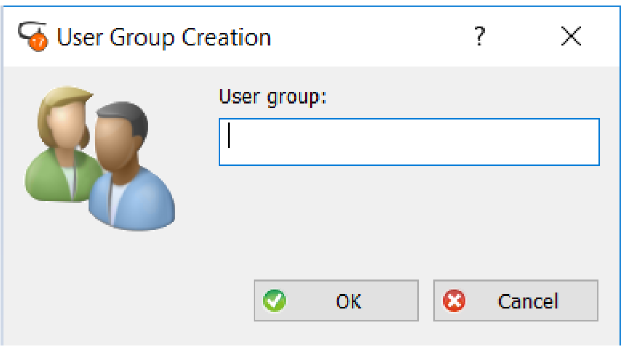
Add a new user group

User groups are used to manage users based on groups. Different rights and permissions to use and/or modify the Profile or database settings can be assigned to different CUSTOMTOOLS **Users** belonging to the same **User Group**.



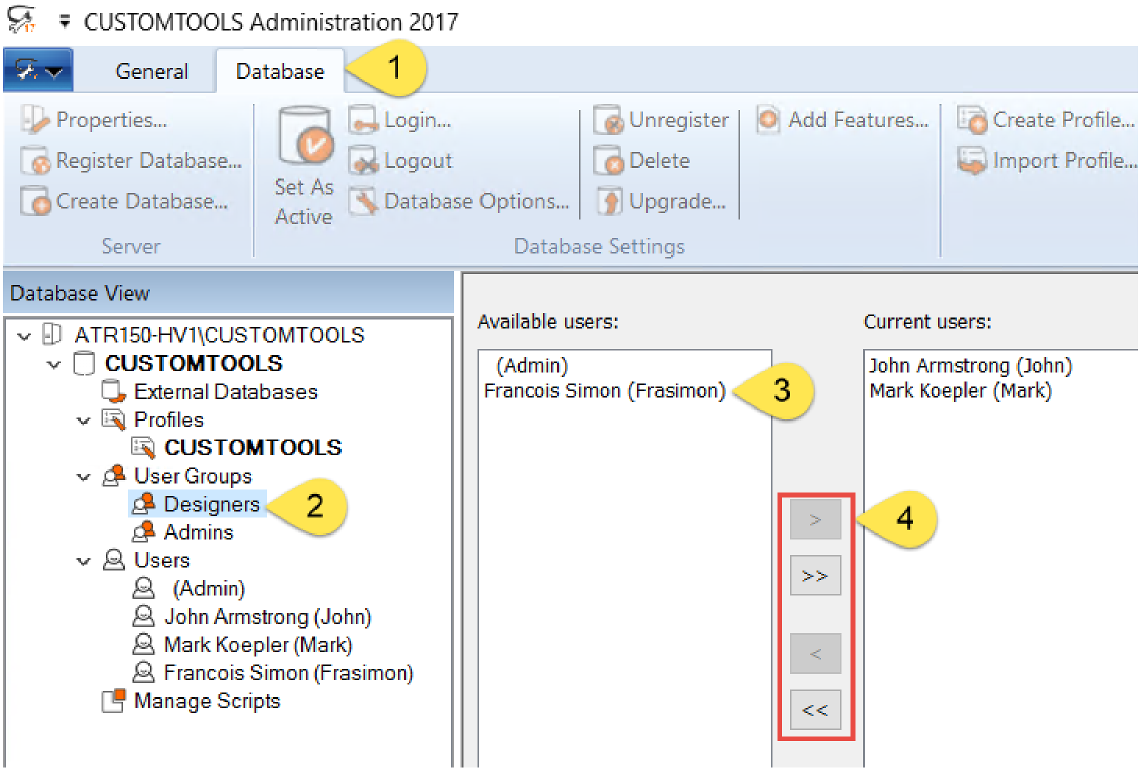
From the **Administration tool**,

1. Activate the **Database** tab,
2. Right click on the **User group**,
3. Click **Create New...**



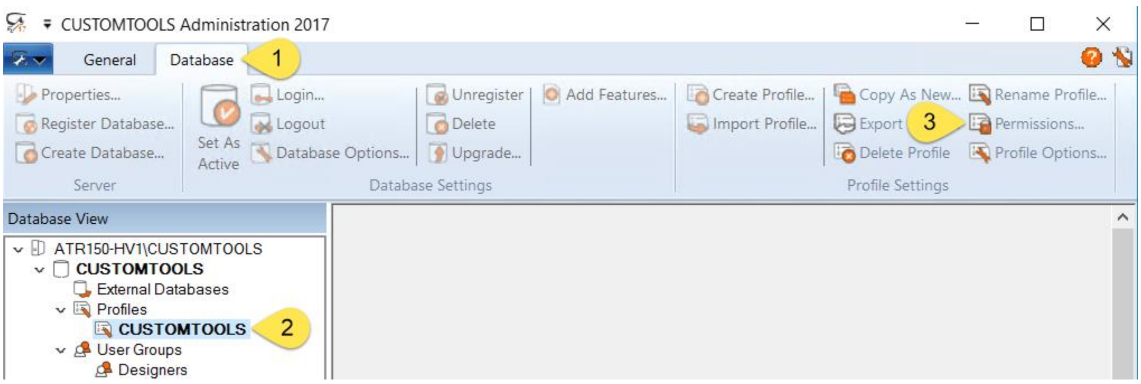
User group: Define the name of the User group.

Assigning users to a User group



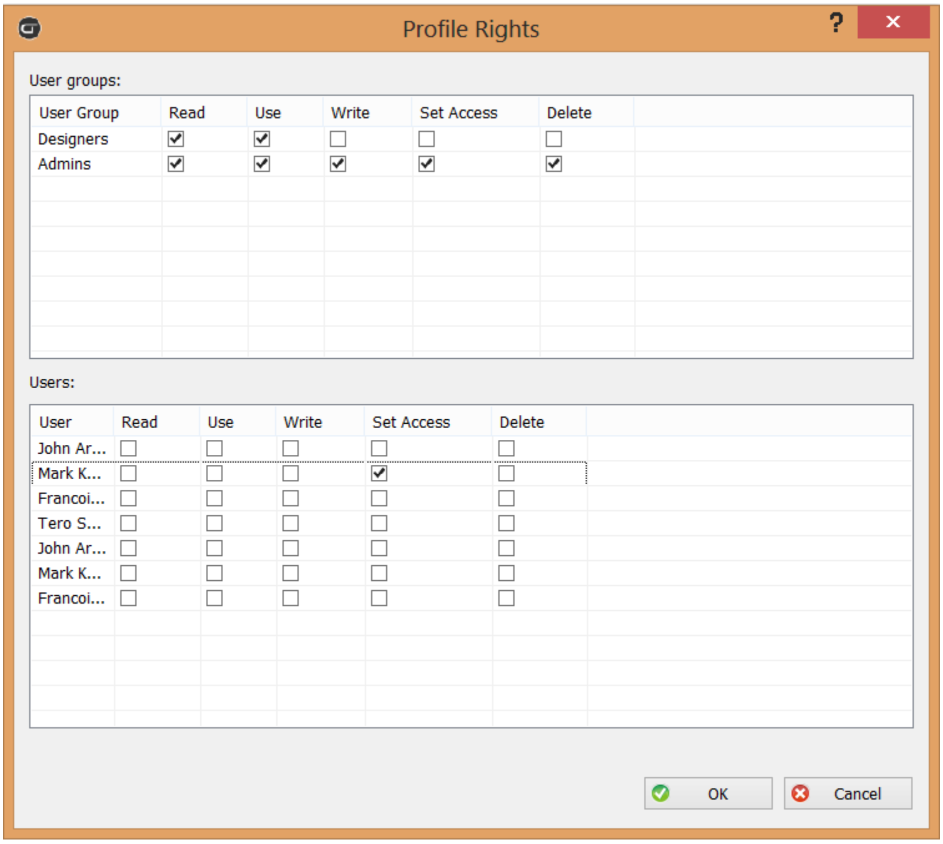
1. Activate the **Database** tab,
2. Select the **User group** (*e.g. Designers*) where to add/remove user,
3. Select the user(s) to be added/removed (*e.g. Francois Simon (FSI)*) from the group,
4. Click on **>**, to add the selected user(s), or **>>**, to add all the available users at once.

Defining the profile rights of the users/user groups



Different rights and permissions can be defined for each **User group** or **User** to manage or access the CUSTOMTOOLS profile. To access the **Profile rights**,

1. Activate the **Database** tab,
2. Select the desired profile,
3. Click **Permissions…** or right click on the desired profile, and select **Permissions**.



Rights and permissions can be defined at the **User group** level or individually for each **User**.

- **Read**: The **user/user group** is allowed to access the profile settings in the CUSTOMTOOLS options but cannot modify them.
- **Use**: The **user/user group** is allowed to use the profile in SOLIDWORKS where all the settings have been defined.
- **Write**: The **user/user group** is allowed to modify the profile settings in the CUSTOMTOOLS options.
- **Set Access**: The **user/user group** is allowed to grant or restrict the profile access rights to other users.
- **Delete**: The **user/user group** is allowed to delete the profile from the **Administration Tool**.

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Chapter 3: Starting CUSTOMTOOLS in SOLIDWORKS

Introduction

What does it do?

Activating the CUSTOMTOOLS add-ins in SOLIDWORKS launches the CUSTOMTOOLS add-ins and allows the user to access the different tools available in SOLIDWORKS (e.g. Batch printing & Conversion, property…).

How does it work?

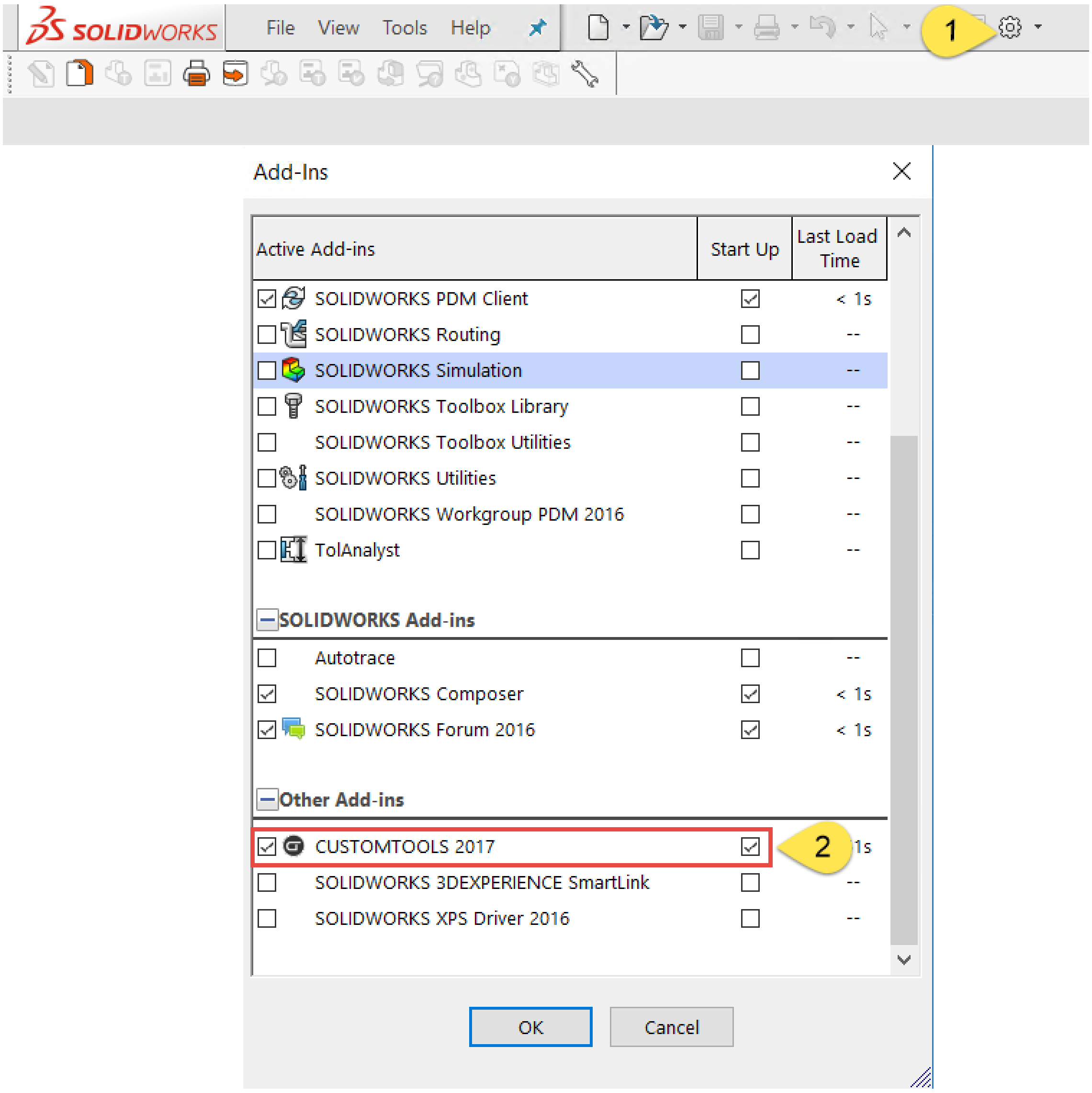
All the different tools available in CUSTOMTOOLS will be installed and accessible from SOLIDWORKS. CUSTOMTOOLS comes in different packages (Basic, Express, Standard and Professional). The Professional version of CUSTOMTOOLS is always installed and then the limitations for each package are applied after the license has been activated. Functionalities that are not included in certain packages can be accessed but the usage will be restricted.

How can you use it?

Once the CUSTOMTOOLS add-in is available in SOLIDWORKS, it can be accessed from the **Command manager** of SOLIDWORKS through the CUSTOMTOOLS menu, or from the SOLIDWORKS Menu (Click **CUSTOMTOOLS** if using earlier version of SOLIDWORKS 2015 or **Tools** -> **CUSTOMTOOLS** if using SOLIDWORKS 2015).

CUSTOMTOOLS

Activating the CUSTOMTOOLS add in



When starting **SOLIDWORKS**,

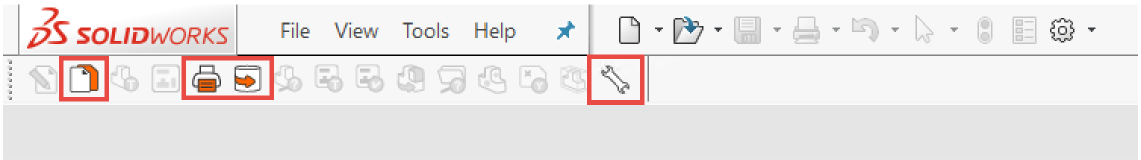
1. Open the SOLIDWORKS Add-**ins** dialog,
2. Select **CUSTOMTOOLS 2015.**

NOTE: The SOLIDWORKS **Add-ins** dialog can also be accessed by clicking **Tools**, from the SOLIDWORKS menu and selecting **Add-ins** (at the bottom of the menu).

Using the CUSTOMTOOLS add-ins

When no documents are opened in SOLIDWORKS

Certain functionalities of CUSTOMTOOLS can be accessed if no documents are opened in SOLIDWORKS.

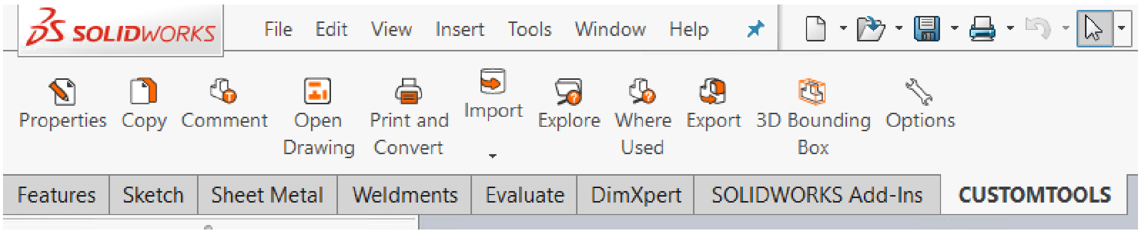


The following functionalities of CUSTOMTOOLS can be accessed:

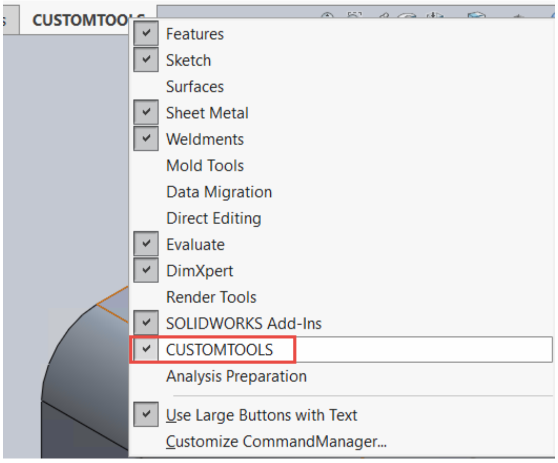
- **Copy**
- **Print & Convert**
- **Import**
- **Options**

When documents are opened in SOLIDWORKS

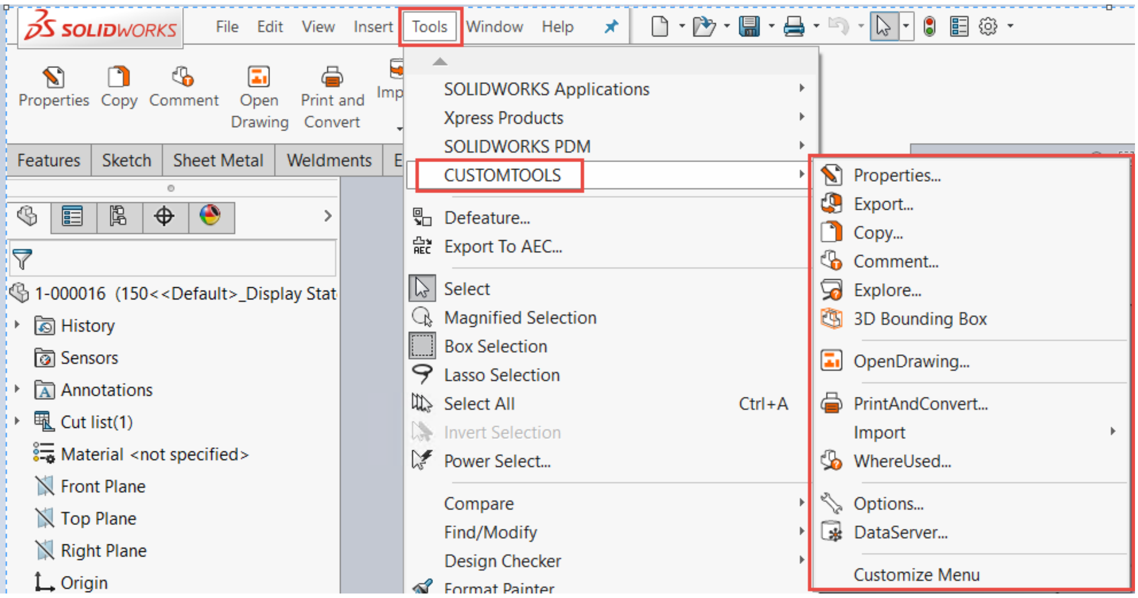
From the CUSTOMTOOLS menu in the Command Manager



If the CUSTOMTOOLS menu is not available in the Command Manager of SOLIDWORKS, right click on the command manager menu and select **CUSTOMTOOLS**.



From the SOLIDWORKS Menu



Click **Tools**, **CUSTOMTOOLS**, and then the CUSTOMTOOLS menu appears in SOLIDWORKS 2015 or Click CUSTOMTOOLS if using earlier release of SOLIDWORKS 2015.

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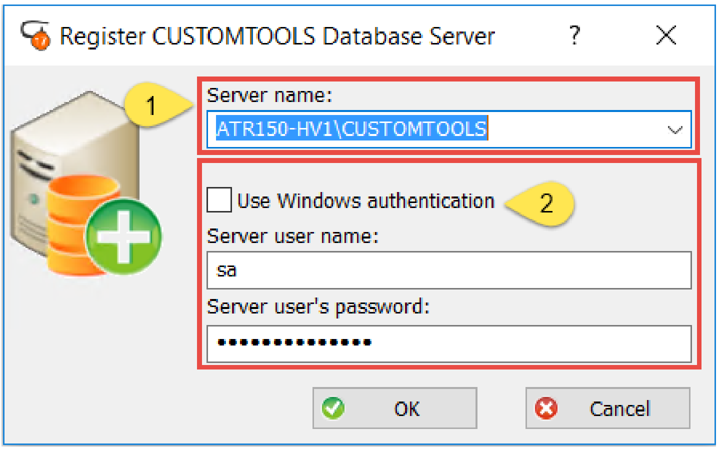
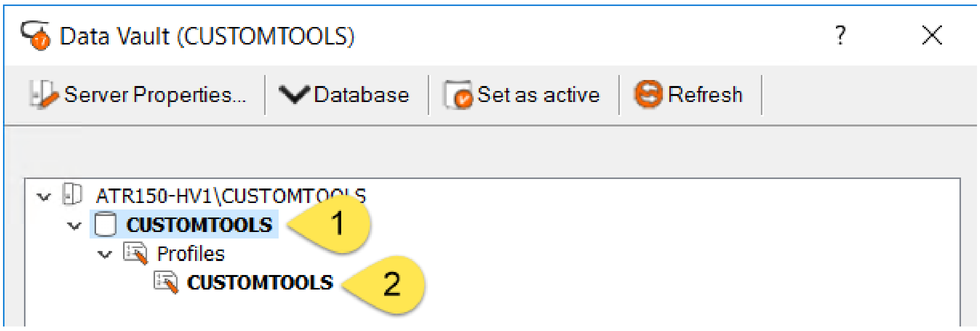
Selecting the server where the CUSTOMTOOLS database has been installed



Once the CUSTOMTOOLS Add-ins has been activated the **Register CUSTOMTOOLS Database Server** dialog appears automatically.

1. Select the Server where the CUSTOMTOOLS database has been created (in the **CUSTOMTOOLS Administration** Tool) from the **Server name**,
2. Use the defined credential to access it.



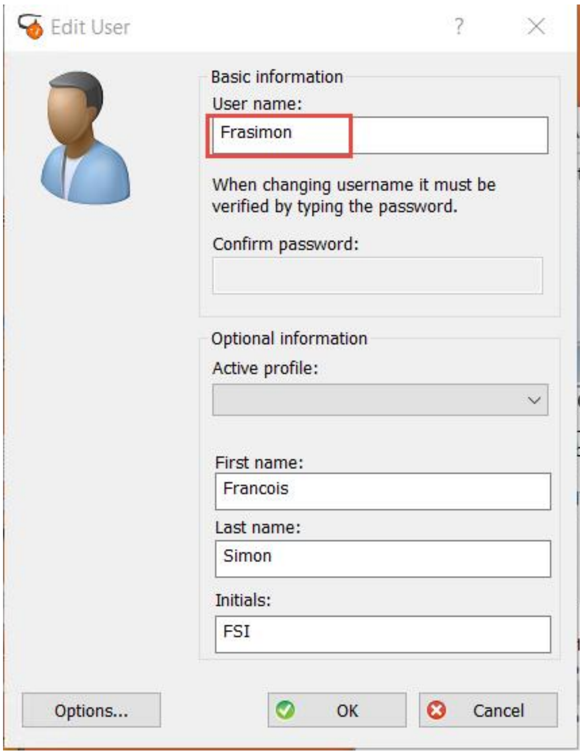
Once the Server has been selected:

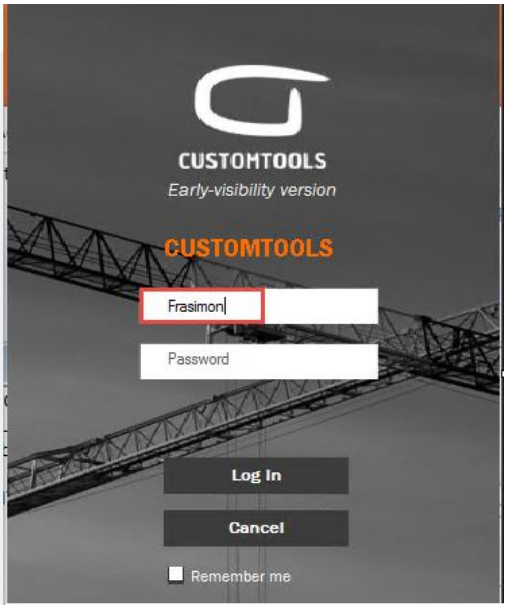
1. Select the CUSTOMTOOLS database *(e.g. CUSTOMTOOLS Demo),*
2. Select the profile to be used can be selected *(e.g. CUSTOMTOOLS Demo).*

Log in into the CUSTOMTOOLS database

Once the CUSTOMTOOLS server and database have been selected, the user will be prompted with the **CUSTOMTOOLS Log-in** dialog where he/she needs to log in with his/her user credentials defined in the **Administration Tool**. Click Log in to **Log in** to the CUSTOMTOOLS database.

TIP: Remember the user by selecting the **Remember me** check box in the CUSTOMTOOLS login.





Chapter 4: Importing old design data into the CUSTOMTOOLS database

Introduction

What does it do?

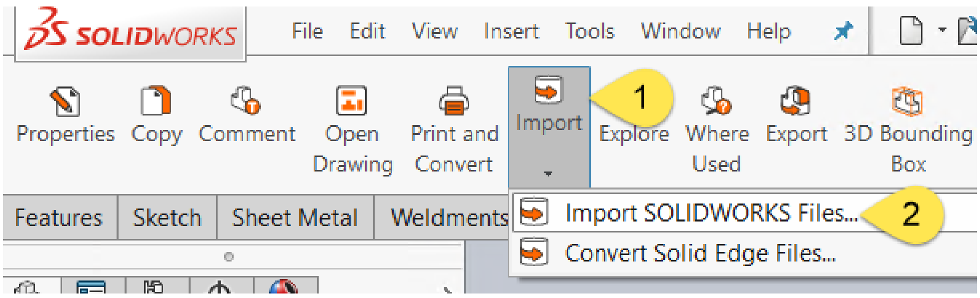
The import tool can be used to import SOLIDWORKS file references that were designed prior to using CUSTOMTOOLS. The main benefit of importing old design files into the CUSTOMTOOLS database is that it will allow the user to use the CUSTOMTOOLS **Search** functionality.

NOTE: In order to batch print or convert old designed files where the child components or drawings are stored into different folders/different name than the refering assembly, the SOLIDWORKS files need to be imported into the CUSTOMTOOLS database in order to be found by CUSTOMTOOLS.

How does it work?

CUSTOMTOOLS will import all the references of the SOLIDWORKS Files that are added to the list of **Files to import**. CUSTOMTOOLS also offers the possibility to search and import the drawings of the listed 3D model.

How can you use it?



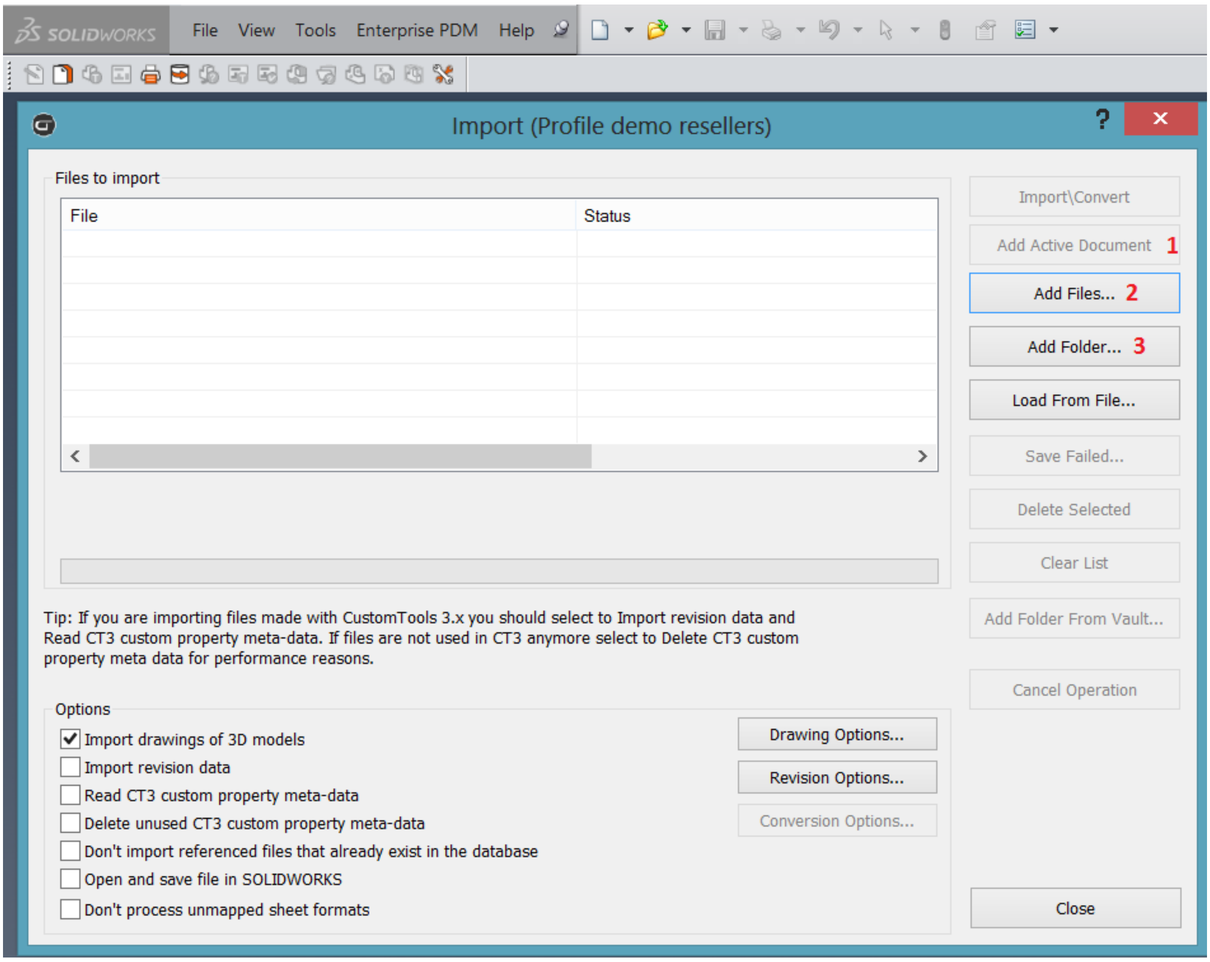
To open the Import dialog,

1. Click **Import,**
2. Select **Import SOLIDWORKS Files....**

SOLIDWORKS Files to be imported to the CUSTOMTOOLS database can then be added to the list of **Files to import**.

Using the Import functionality

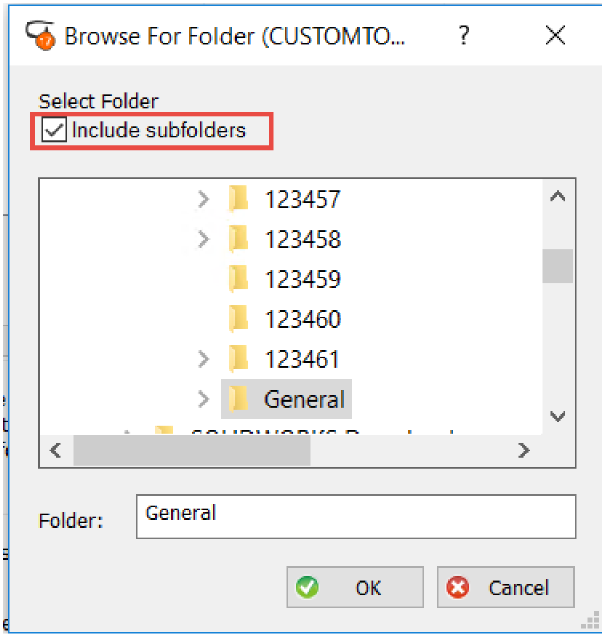
Selecting the files to be imported to the CUSTOMTOOLS database



1. **Add Active Document:** Adds the assembly or part currently active in SOLIDWORKS and all sub-components child to the list of **Files to import**.
2. **Add files…**: Opens the **Windows Explorer** where the files to be added to the list of **Files to import** can be manually selected.
3. **Add folder…**: Opens the **Browse for folder** dialog where all the SOLIDWORKS files stored in the selected folder will be added to the list of Files to import.
4. **Import/Convert:** Click import and convert to import the files into the CUSTOMTOOLS database.

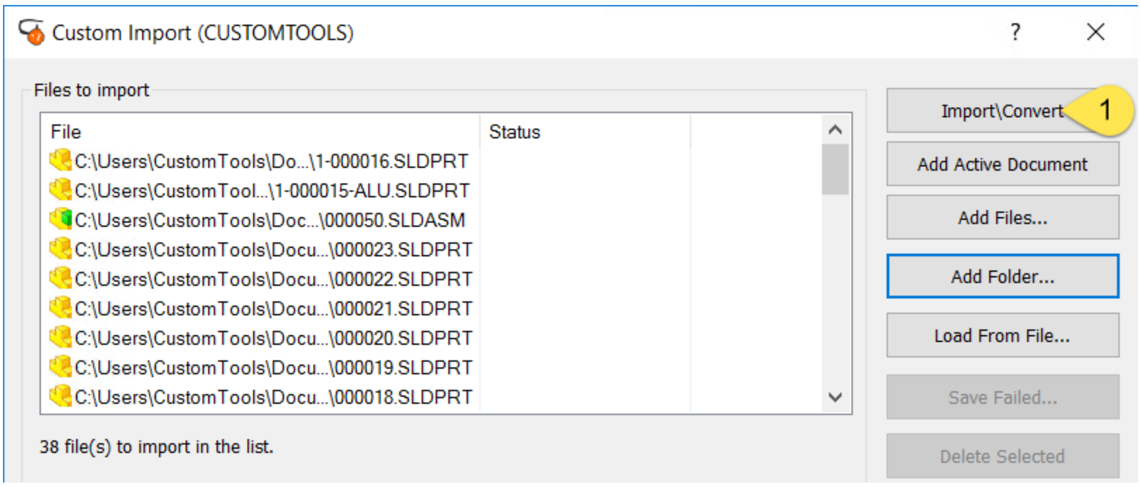
NOTE: Sub-folders can also be included.

Importing files from the sub-folders



If the **Add folder...** option has been selected from the **Import** dialog, then the **Browse for folder** dialog opens. Select the **Include sub-folder** check box to add SOLIDWORKS files stored in the sub-folders to the list of **Files to import**.

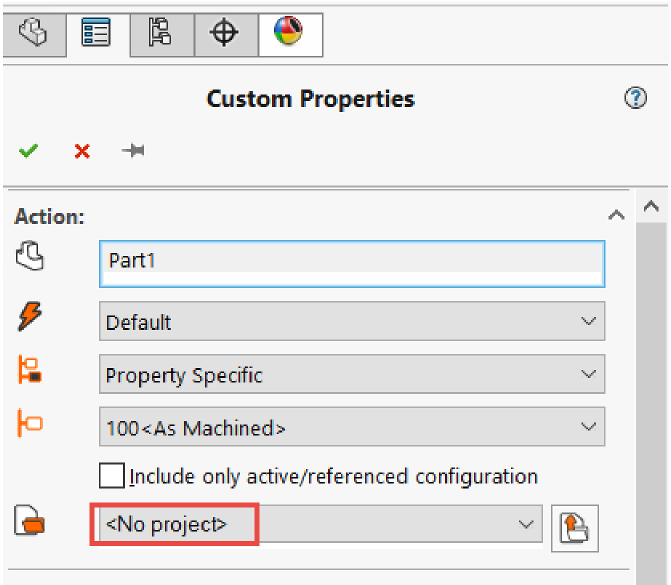
Importing files into the database



Once the list of files to import has been defined click on the **Import\Convert** button.

No Project information

Files that have been designed prior to CUSTOMTOOLS and that are imported in the CUSTOMTOOLS will have the **<No project>** information. The **<No project>** option is only available to files that are imported into the CUSTOMTOOLS project and cannot be selected from the list of projects.



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Chapter 5: CUSTOMTOOLS Options

Introduction

What does it do?

The CUSTOMTOOLS Options can be used to configure the CUSTOMTOOLS Profile settings (*E.g. custom properties, batch printing & file conversion profile, file naming rules,…*). It can also be used to manage the Database **Options** (*e.g. Manage dictionaries…*) and the User preferences.

How does it work?

The CUSTOMTOOLS options consists of three different tab:

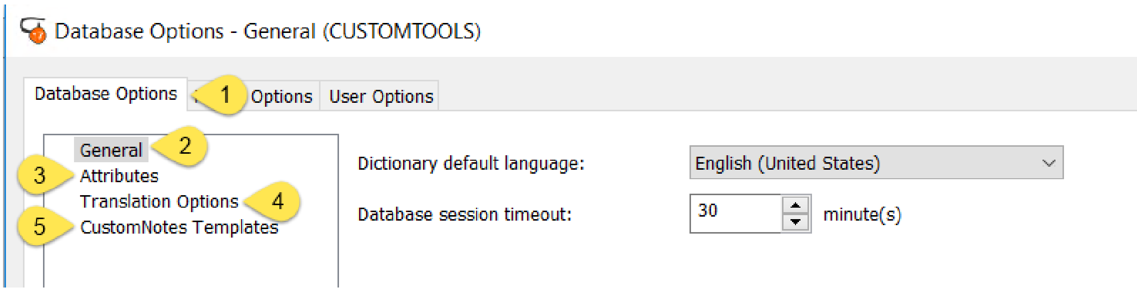
- **Database Options**
- **Profile Options**
- **User Options**

The access to the Options can be restricted based on the users or user groups rights and permissions.

How can you use it?

The Options can be accessed from **SOLIDWORKS** or from the **CUSTOMTOOLS Administration Tool**. The access to the Options can be restricted based on the user. The rights and permissions to access the **Database Options** or **Profile Options** can be defined in the **Administration Tool** for every user or user group.

Database Options



To access the Database options, open the Options:

1. Activate the **Database Options** tab,

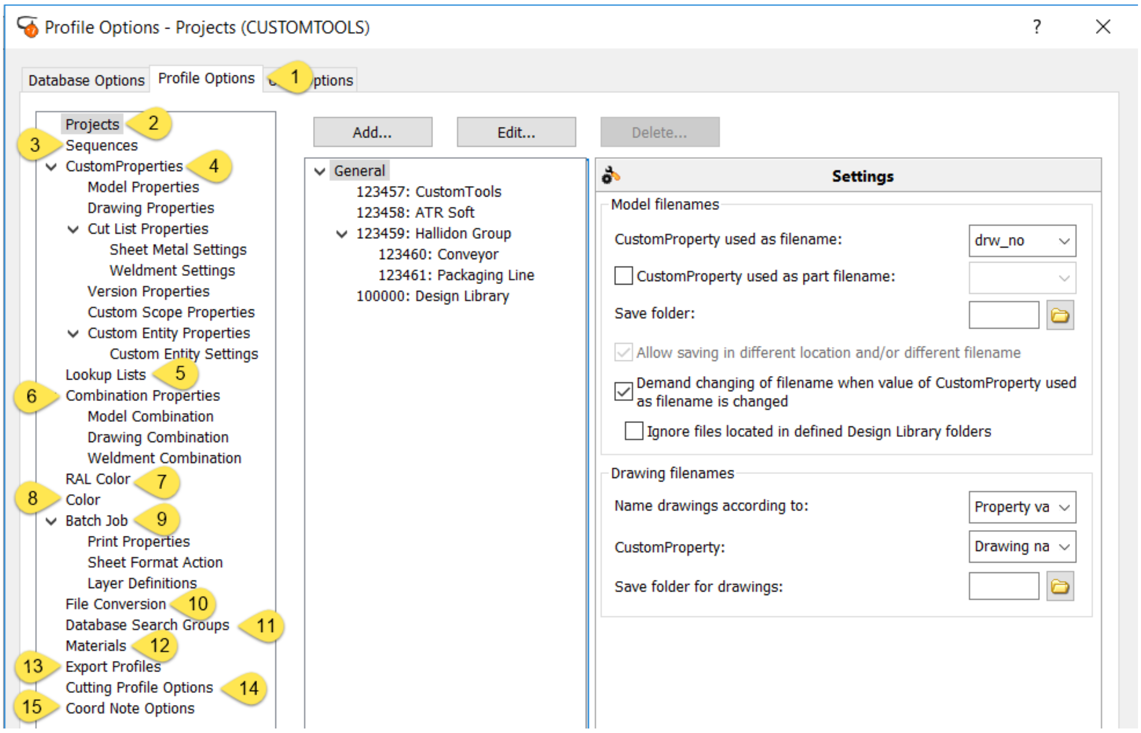
The **Database Options** tab contains the following items:

2. **General**: Select in which language custom properties retrieved from the language dictionaries are saved into the CUSTOMTOOLS database. Define the timeout for the database.
3. **Attributes**: An attribute is the link between the value stored in the CUSTOMTOOLS database and a property. Attributes must be unique.

   NOTE: Attributes can be created directly from the Property wizard.
4. **Translation Options**: Manages the dictionaries and languages available to translate custom properties during the batch printing and file format conversion.
5. **CustomNotes Templates**: Manages the property notes to be inserted in the DXF/DWG files.

Profile Options

The Profile Options can be used to configure the different functionalities available in CUSTOMTOOLS. If modifications are applied to the Profile, then they will be immediately available to the users.



To access the Profile options, open the Options:

1. The **Profile Options** tab is always activated when opening the Options.

The **Profile Options** tab contains the following items:

2. **Projects**: Manages the projects and the file naming and storing rules.
3. **Sequences:** Defines the sequences to be used by a property.
4. **Properties**: Defines the custom properties for:
    a. **Model Properties**: Configure the Properties pane for Parts & Assemblies.
    b. **Drawing Properties**: Configure the Properties pane for Drawings.
    c. **Cut List properties**: Configure the Properties pane for Cut list items.
    d. **Version properties**: Configure the custom properties available in the revision table.
5. **Lookup Lists:** Defines the different look up lists available for custom properties with the type defined as: Combobox, Editable Combobox or Hierarchical combo.
6. **Combination Properties:** Combines multiple custom properties to create a new property (*E.g. to be used as a property to generate a file name*).
7. **RAL Color:** Manages the RAL color available with a property that retrieve a RAL Color.
8. **Color:** Manages the color available with a property that retrieve a Color.
9. **Batch Job:**

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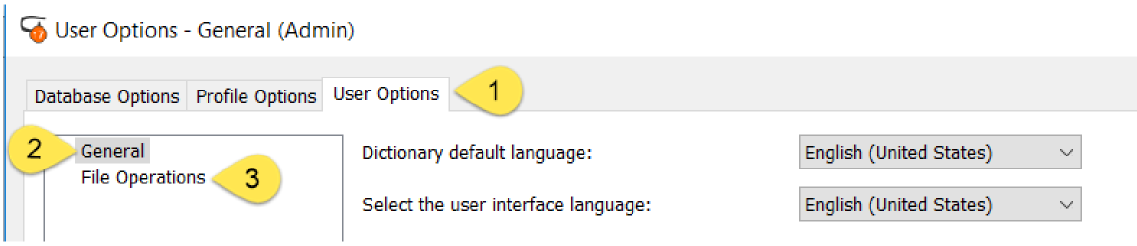
   a. **Print Properties:** Defines the batch printing profiles available in the Print & Conversion.
   b. **Sheet Format Action**: Defines the profiles to reload or convert the drawing sheet format.
   c. **Layer Definition**: Defines the profiles that hide or show certain layers of the drawings during the batch printing or file conversion.

10. **File conversion:** Defines the file conversion profiles available in the Print & Conversion.
11. **Database Search Groups:** Creates a search in a table of an external database.
12. **Material:** Manages custom material.
13. **Export Profiles:** Manages the Excel report profile.
14. **Cutting Profile Options**: Defines the settings used by the Cutting profile to create a new drawing sheet containing a flatten view ready to be converted to DXF/DWG.
15. **Coord Note Options:** Defines the settings used to insert coordinates into the Drawings.

User options

Each CUSTOMTOOLS users defined in the **AdministrationTool** is able to define its own preferences.



To access the User Options, open the **Options**:

1. Activate the **User Options** tab,

The **User Options** tab contains the following items:

2. **General**: Selects the language for the User Interface and Dictionaries of CUSTOMTOOLS. Configure how the Search results will be displayed.
3. **File Operations**: Defines settings when saving a SOLIDWORKS Documents with CUSTOMTOOLS.

Chapter 6: Batch printing your SOLIDWORKS drawings

Introduction

What does it do?

Automates the printing of the SOLIDWORKS drawings.

How does it work?

The batch printing is managed via profiles. Printing settings can be defined based on the Sheet formats of the SOLIDWORKS drawings (*e.g. printer, paper size and orientation*). Multiple printing settings can be defined for different sheet formats of the same **Print profile**.

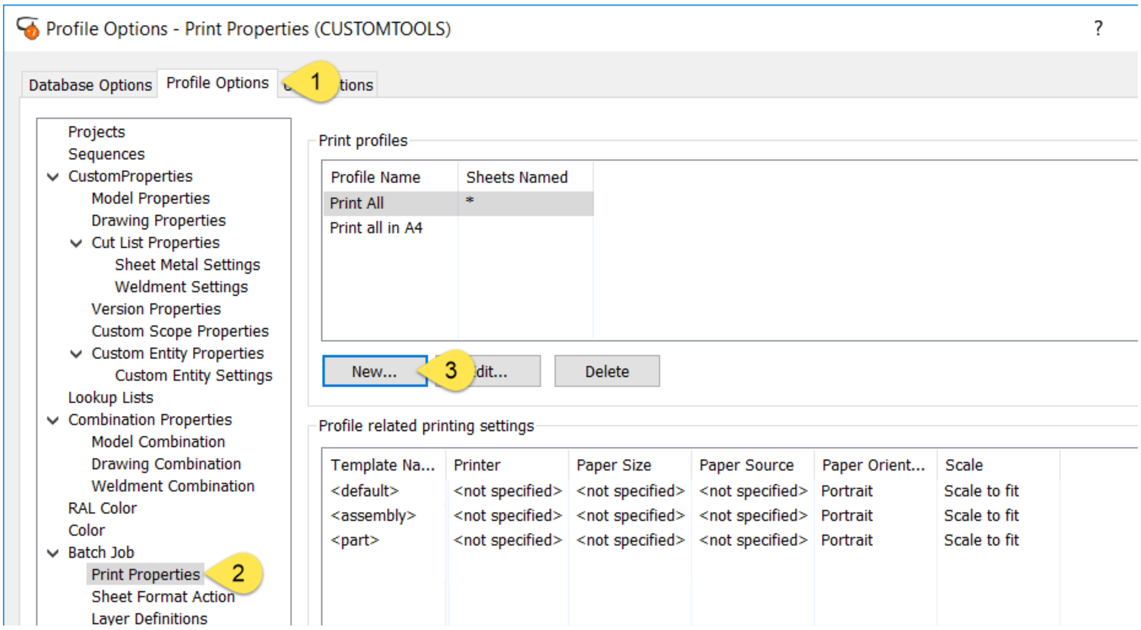
How can you use it?

If an assembly is opened, click on the **Print & Convert** icon. The **Print & Convert** dialog opens, with the **Print profiles** tab active. Multiple print profiles can be selected at once. The SOLIDWORKS files do be printed do not have to be opened in SOLIDWORKS. The **Print & Convert** dialog can also be launched from the **Search results** or from the CUSTOMTOOLS **command bar**, when no SOLIDWORKS files have been opened.

CUSTOMTOOLS

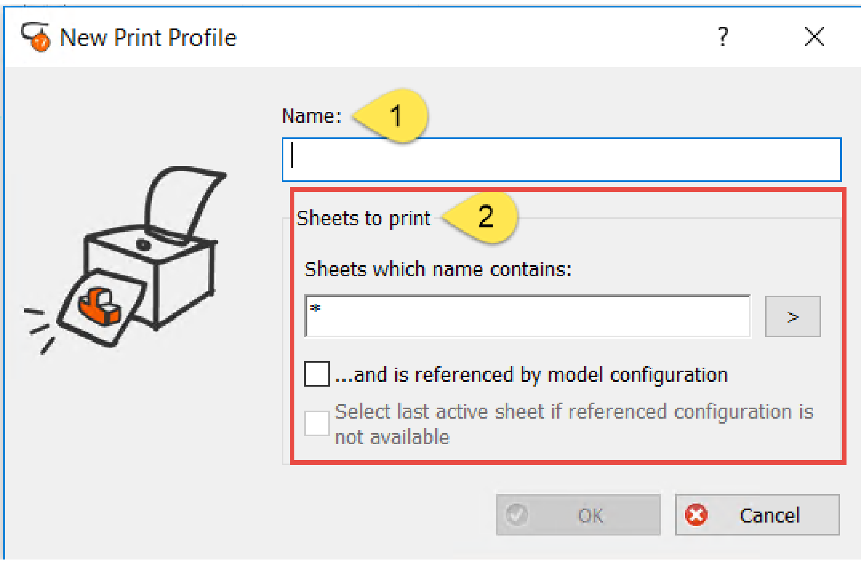
Creating a print profile

Print profiles are used to store the printing settings. Multiple profiles can be defined (e.g. *To print your drawings in a specific size: A4 or to their current size*).



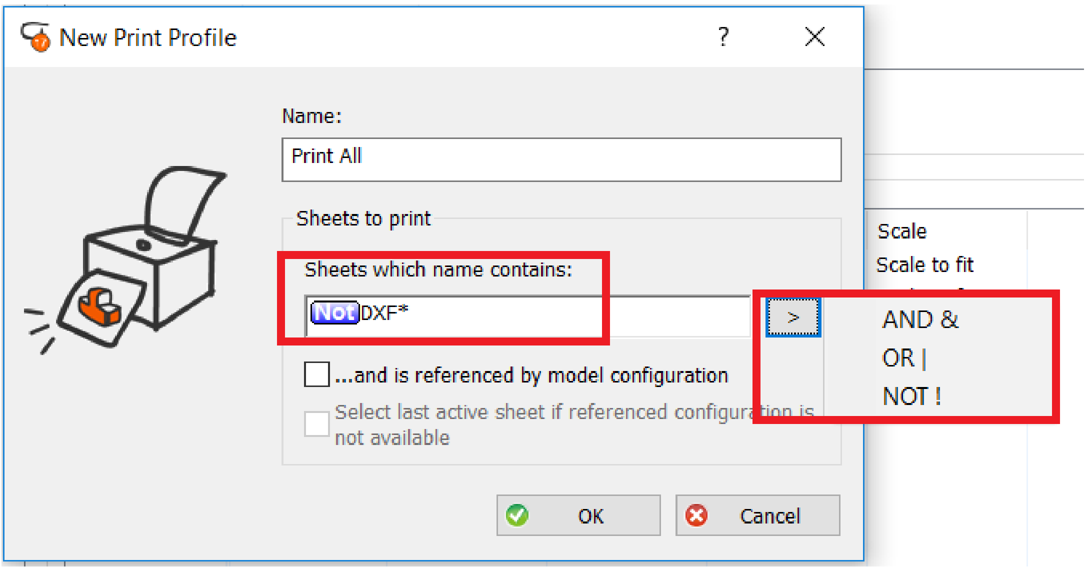
To create a new print profile, open the CUSTOMTOOLS **Options** dialog:

1. Activate the **Profile Options** tab,
2. Select **Print properties** from the tree view,
3. Click on **New** to open the **New Print Profile** dialog.



1. **Name**: Defines the name of the print profile as it will appear in the **Print & Convert** dialog.

2. **Sheets to print**: CUSTOMTOOLS is able to manage the selection of the sheet of the drawings to print:
   - **Sheets which name contains:** Filter the drawings sheets to be selected based on the name of the sheet when the profile is *selected (e.g. NOT DXF -> Exclude sheets called DXF from being selected)*.



By clicking on the **>**

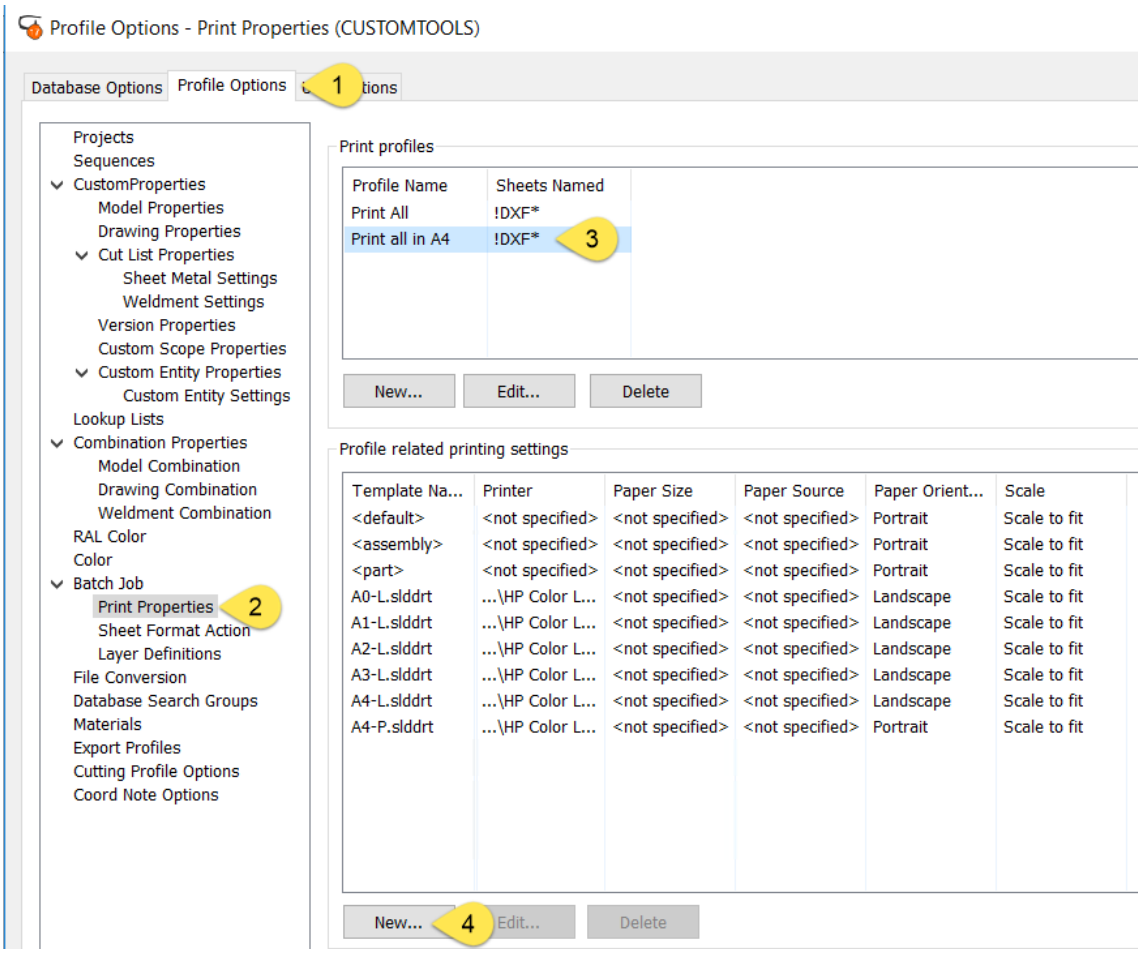
- **AND &**: Selects the sheets to be printed based on multiple criteria that must be contain in the sheet name in order to be selected.
- **OR |**: Selects the sheets to be printed if one of the defined criteria matches.
- **NOT !**: Excludes the sheets from being selected if it matches with the criteria. This can be used to exclude certain sheets from being selected (*E.g.: Sheet called DXF*).

   - **... and is referenced by model configuration:** Prints configuration specific drawing sheets.
   - **Select the last active sheet if referenced configuration is not available:** Selects the sheet that was last active if the specific referenced configuration cannot be found.

TIP: To make your selection more effective, you can search for word variations using the asterisk (*) as a wildcard symbol.

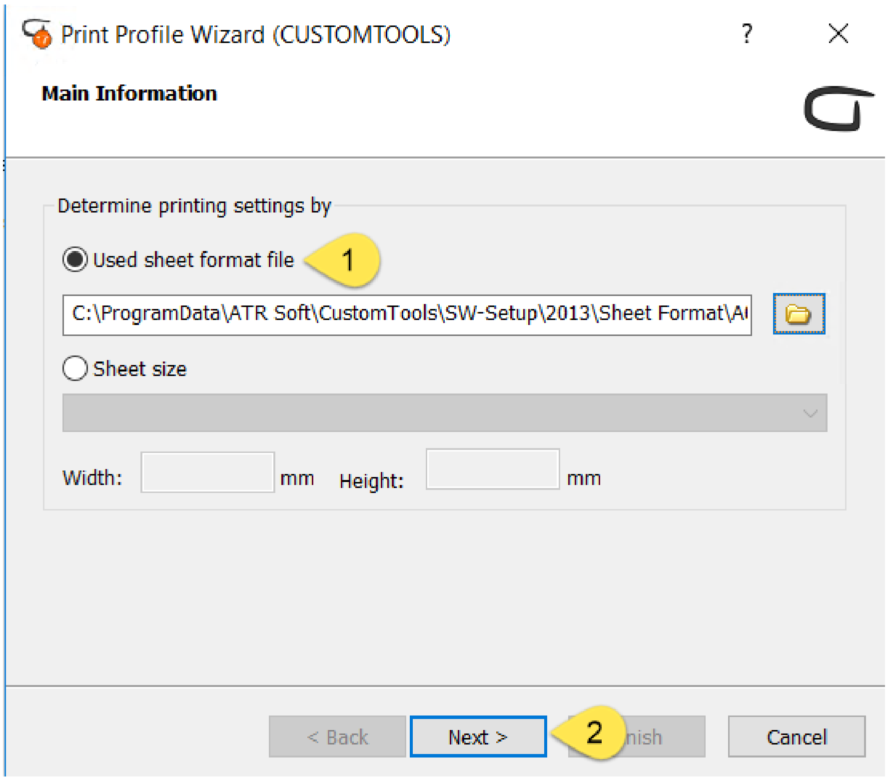
Defining the printing settings of a print profile:

The printing settings in CUSTOMTOOLS are defined based on the sheet format of the drawing. Different sheet formats printing settings can be defined for the same print profile.



To define the print settings of a Print profile, open the CUSTOMTOOLS **Options** dialog,

1. Activate the **Profile Options** tab,
2. Select **Print properties** from the tree view,
3. Select the **Print Profile** for which the printing settings are to be defined,
4. From the **Profile related printing settings**, click **New.**



The **Print Profile Wizard** dialog opens:

1. From the **Used sheet format file**, select the SOLIDWORKS sheet formats for which the printing settings will be defined.
2. Then, click **Next**.

NOTE: Select files with the .slddrt file extension.

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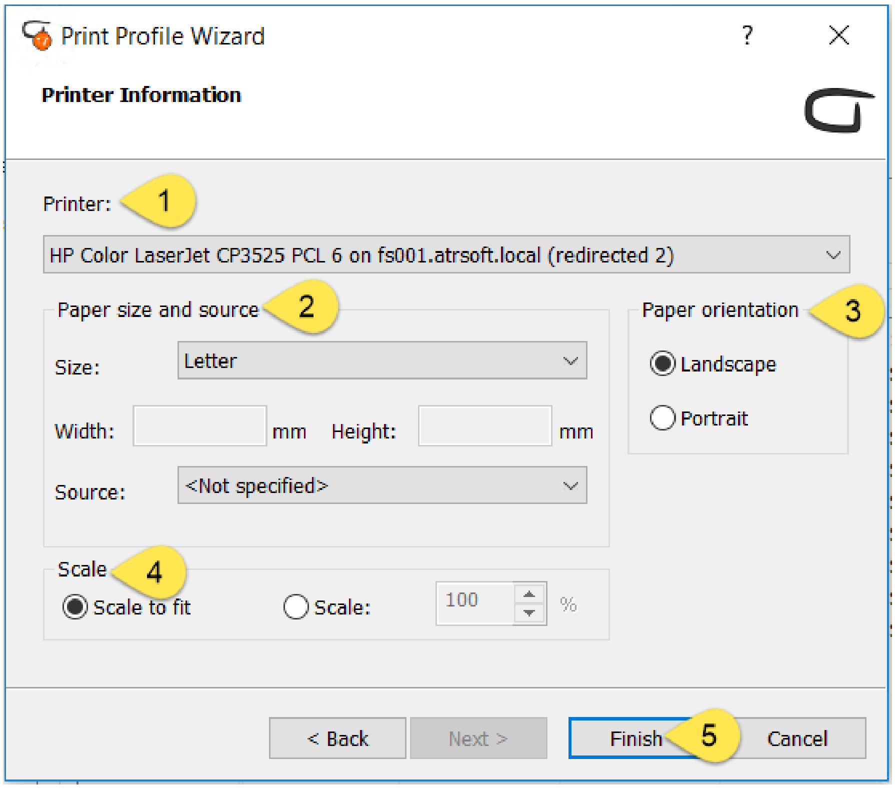

Select the different printing information:

1. **Printer**: Select the printer to be used to print the SOLIDWORKS Drawings.
2. **Paper size and Source**: Select the output paper size and printer tray used by the printer. The list of paper size and the trays available updates based on the selected Printer.
3. **Paper orientation**: Select the orientation of the printed SOLIDWORKS Drawings:
   - **Landscape**
   - **Portrait**
4. **Scale**: Define the scale that will be used to during the batch printing:
   - **Scale to fit**: Scales small pages up and large pages down to fit the paper.
   - **Scale:** Type the percentage to magnify or reduce the print.
5. Click **Finish.**

TIP: Different printers can be used with the same **Print profile**.

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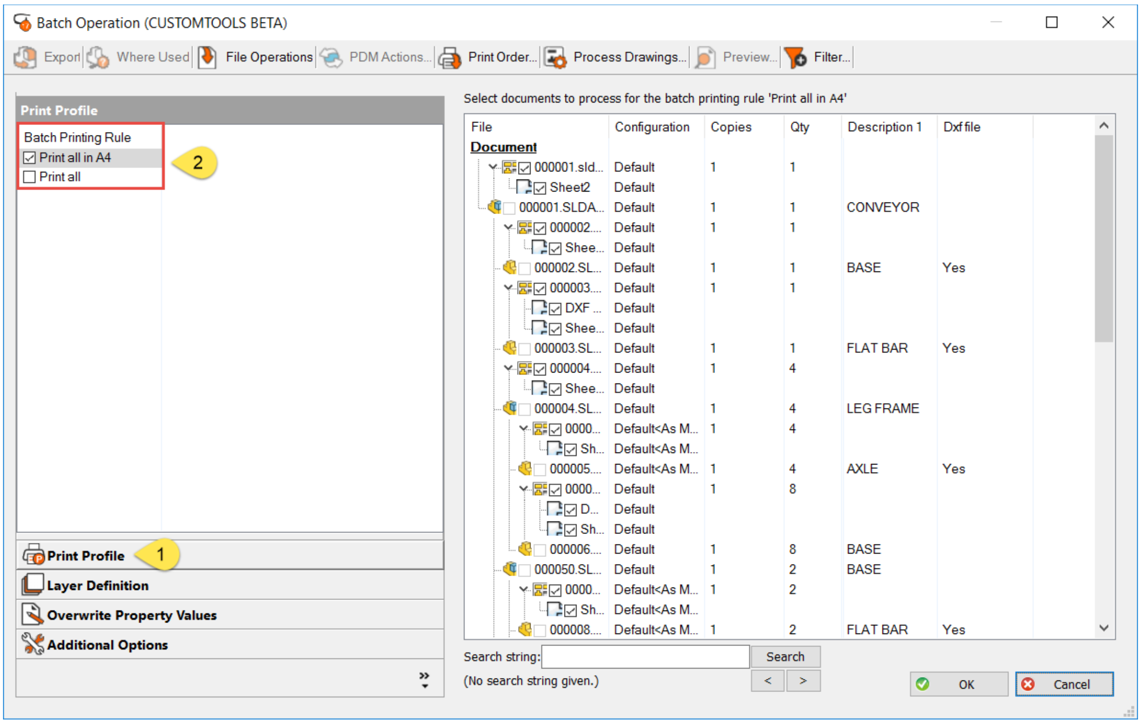
Printing your SOLDIWORKS Drawings

From an opened assembly in SOLIDWORKS

To batch print all the drawings related to an assembly. Open the top level assembly in SOLIDWORKS and click **Print & Convert** to open the **Batch operation** dialog.

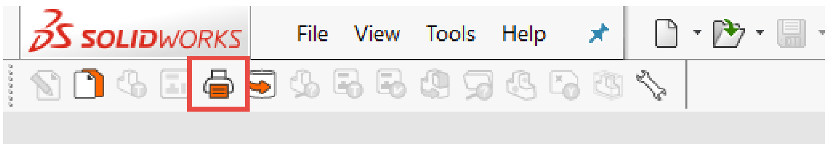


Open the **Print & Conversion** functionality:

1. The **Print Profile** button is active,
2. Select the **Print profiles** to be used from the **Batch Printing Rule**.

The sheets of the listed drawings will be automatically selected based on the **Sheet to print** settings defined for the selected print profile. Click **OK** to start the printing process.

If no documents are opened in SOLIDWORKS

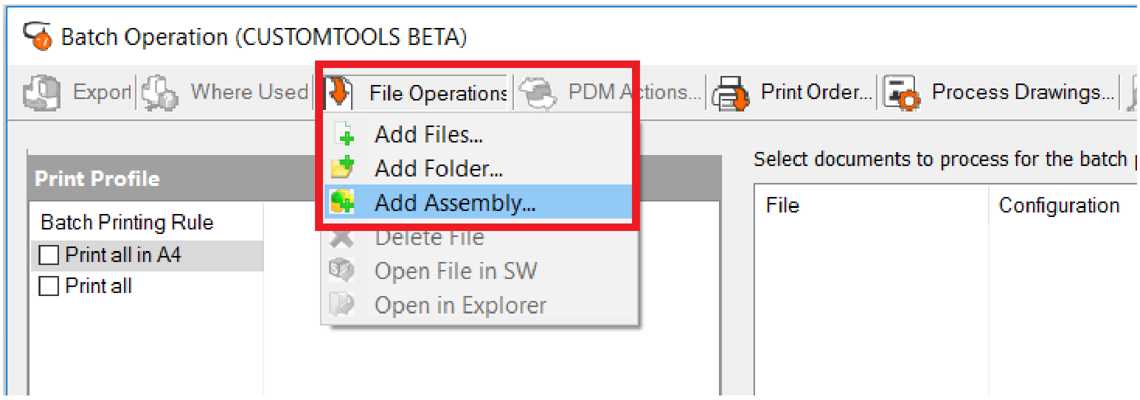


Opening the Batch operation dialog

The **Print & Convert** functionality can also be accessed if no documents are opened in SOLIDWORKS. Click on the **Print and Convert** icons from the **CommandManager** to open the batch operation dialog.

Adding the files to be printed

When the **Batch operation** dialog opens, the list of files to process is empty.



To add files to the list of files to process, click **File Operations**, then CUSTOMTOOLS offers three possibilities to select the files to be added to the list of files to print:

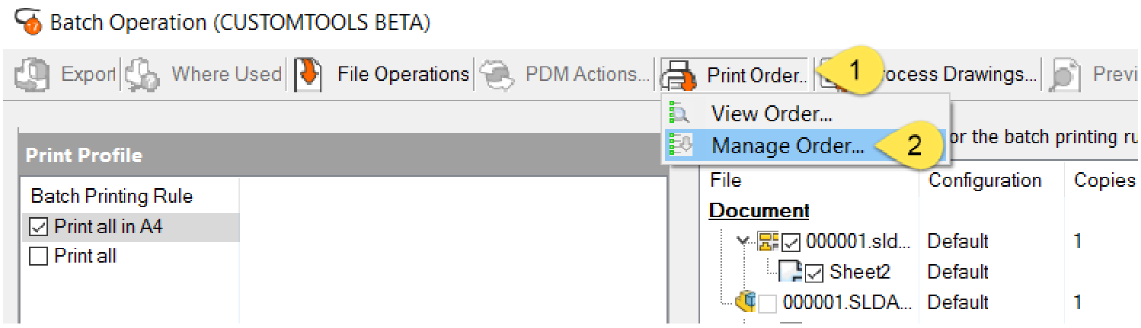
Add files: The files to be added are selected by the user from the Windows Explorer.

Add folder: The user selects the folder where the files to be added are located. If the **Include Subfolders** check box is selected, then files located in sub-folders will also be added.

Add assembly: The user selects an assembly and all child components are automatically loaded.

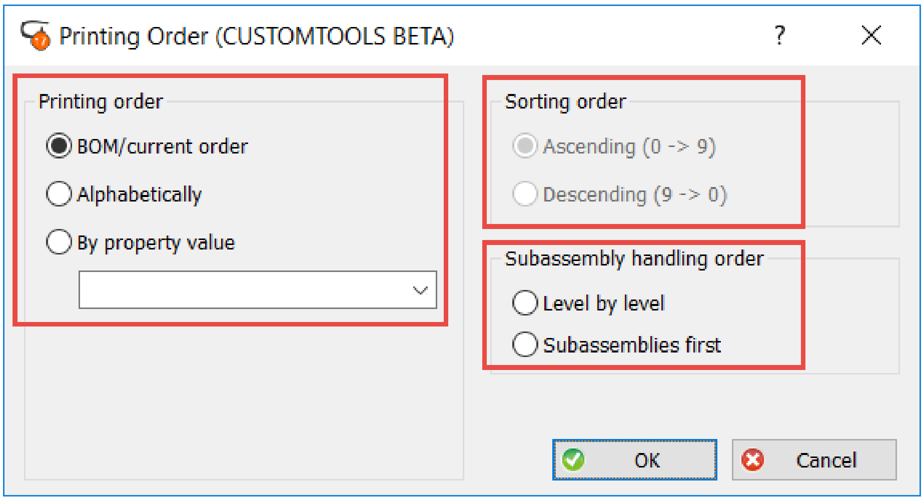
Managing the printing order

The order in which documents are printed can be managed in CUSTOMTOOLS. By default, CUSTOMTOOLS prints the assembly's drawings and all referred documents in the Bill of Material (BOM) order.



To manage the print order:

1. Click **Print order**, from the Batch Operation dialog,
2. Select **Manage Order.**



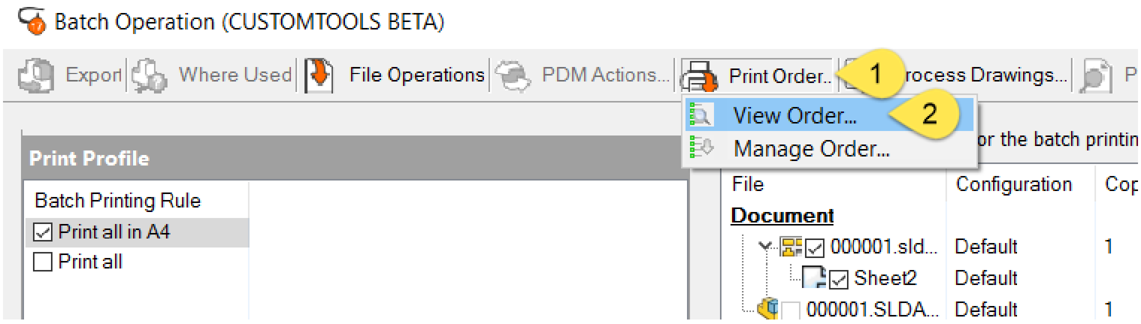
Printing order:

- **BOM/Current order**: Documents are processed in the BOM order.
- **Alphabetically:** Documents are processed alphabetically based on the filename.
- **By property value:** Select a property.

Sorting order: Arranged the printing order from largest to smallest OR smallest to largest.

Subassembly handling order: Defines how sub-assemblies are printed.

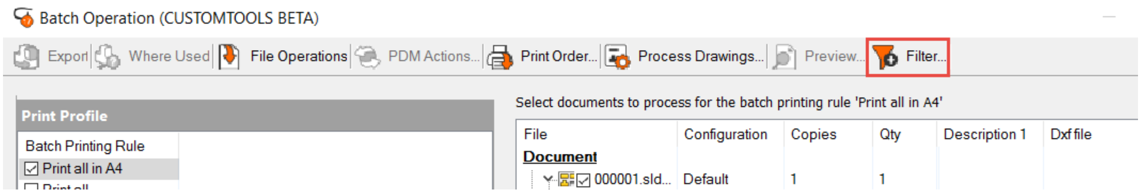
TIP: The printing order can be viewed before the documents are being processed.



. From the **Batch Operation** dialog:

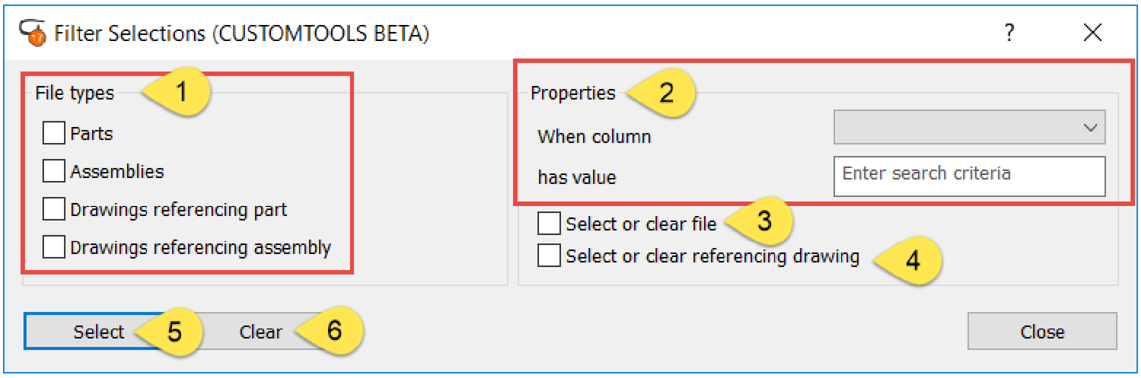
1. Click **Print order**,
2. Select **View Order.**

Filtering the documents to be selected for printing



To open the **Filter Selections** dialog, click **Filter** from the **Batch Operation** dialog.

NOTE: The filters are applied to the files selected by the selected **Print profile.**



1. **Files types:** The filter can be used to select or exclude certain file types from being selected, parts, assemblies, drawings made for part or assembly can be selected.

TIP: The filter can be used to print only the assemblies' drawings, if the **Drawing referencing assembly** check box is selected.

2. **Custom properties**: Documents to print can also be filtered based on a property value. From the **When Column** menu, select the property used to filter the selection and define the selection criteria from the **Has value** text box.

   NOTE: Custom properties used has filter must have the option **Display in Print and convert** selected from the **Property wizard** on the **Additional Options** page.

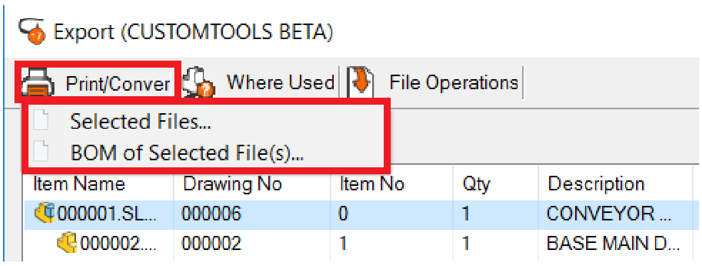
3. **Select or clear**: If the Property is defined at the part or assembly level then parts or assemblies will be selected/cleared.
4. **Select or clear referencing drawing**: If the Property is defined at the part or assembly level then the drawings made for those parts or assembles will be selected/cleared.
5. **Select**: Select the files that matches with the filter criteria defined in the **File type** or **Custom properties.**
6. **Clear:** Clear the files that matches with the filter criteria defined in the **File type** or **Custom properties.**

From other functionalities of CUSTOMTOOLS

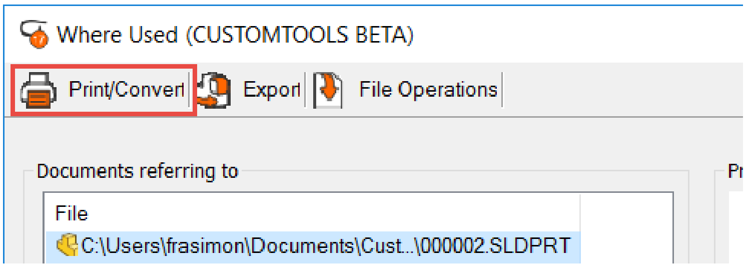
The Print functionality of CUSTOMTOOLS can also be launched from:

- **Search results:**

  Select an assembly from the search result and click **PrintAndConvert**. If an assembly has been selected, click **BOM of Selected File(s)…** to add the assembly and its child component to the list of document to print.



- **Where used:**


Where Used (CUSTOMTOOLS BETA)
Print/Convert
Export
File Operations
Documents referring to
File
C:\Users\frasimon\Documents\Cust...\000002.SLDPRT

CUSTOMTOOLS

Chapter 7: Batch converting your SOLIDWORKS Documents

Introduction

What does it do?

Automates the conversion of your SOLIDWORKS parts, assemblies and drawings. CUSTOMTOOLS supports all 25 file formats available in SOLIDWORKS.

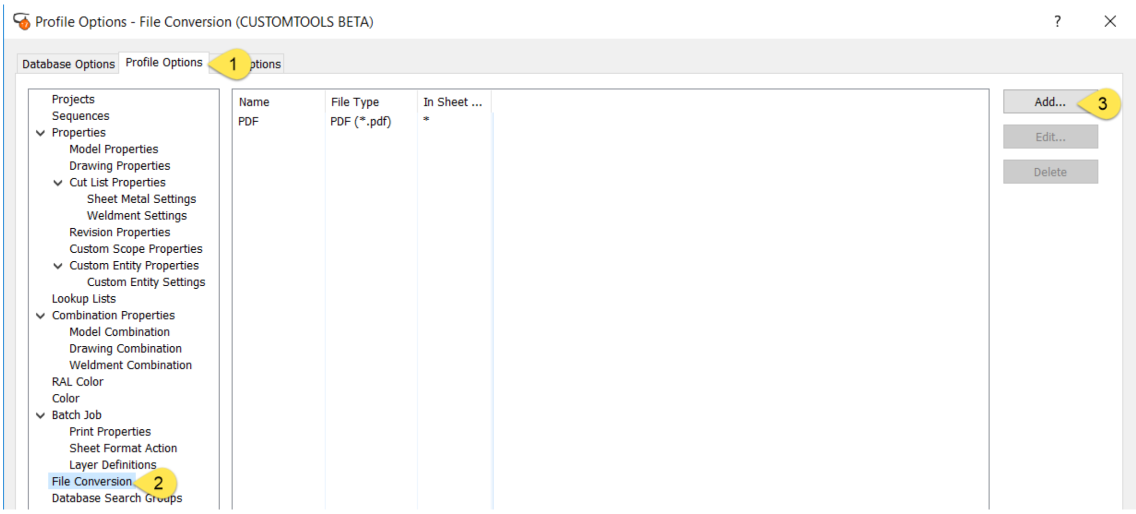
How does it work?

The file conversion is managed via profiles. Profiles can be defined based on the output file format *(e.g. PDF, DXF...).* Rules where the converted documents will be stored and named can also be defined.

How can you use it?

If an assembly is opened in SOLIDWORKS, click on the **Print & Convert** icon. The **Print & Convert** dialog opens, then select the **Batch Conversion** button. The list of defined conversion profiles appears. Multiple conversion profiles can be selected at once. The SOLIDWORKS Files do be converted do not have to be opened in SOLIDWORKS. The Print & Convert dialog can also be accessed from the **CUSTOMTOOLS search results** and from **the CUSTOMTOOLS command bar**, when no SOLIDWORKS files are opened.

Creating a conversion profile



To create a conversion profile, open the CUSTOMTOOLS **Options** dialog,

1. Activate the **Profile Options** tab,
2. Select **File conversion** from the tree view,
3. Click on **Add** to open the **Add batch file conversion rule** dialog.

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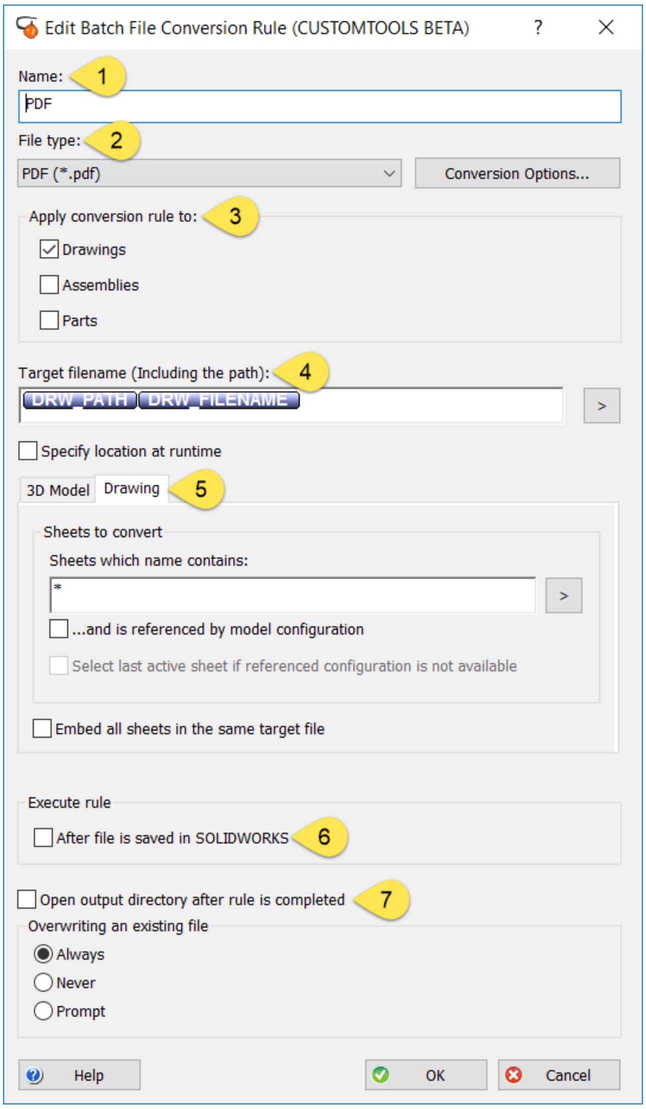


1. **Name:** Define the name of the conversion rule as it will appear in the **Print & Convert** dialog.
2. **File type:** Select the output file format to be used by the conversion rule (*e.g. PDF, DXF,...*).
3. **Apply conversion rule to:** Select the type of SOLIDWORKS documents where the conversion profile will be applied: Drawing, Part or Assembly.
4. **Target file name (Including path):** Define the rules used to store and name the converted documents.
5. **Sheets to convert:** With CUSTOMTOOLS the selection of the drawings' sheet to convert can be managed. The **Sheets to convert** offers three choices:
    a. **All***:* Once the profile is selected, all the sheets available in the list of documents to print will be selected.
    b. **... and is referenced by model configuration:** Prints configuration specific drawing sheets.

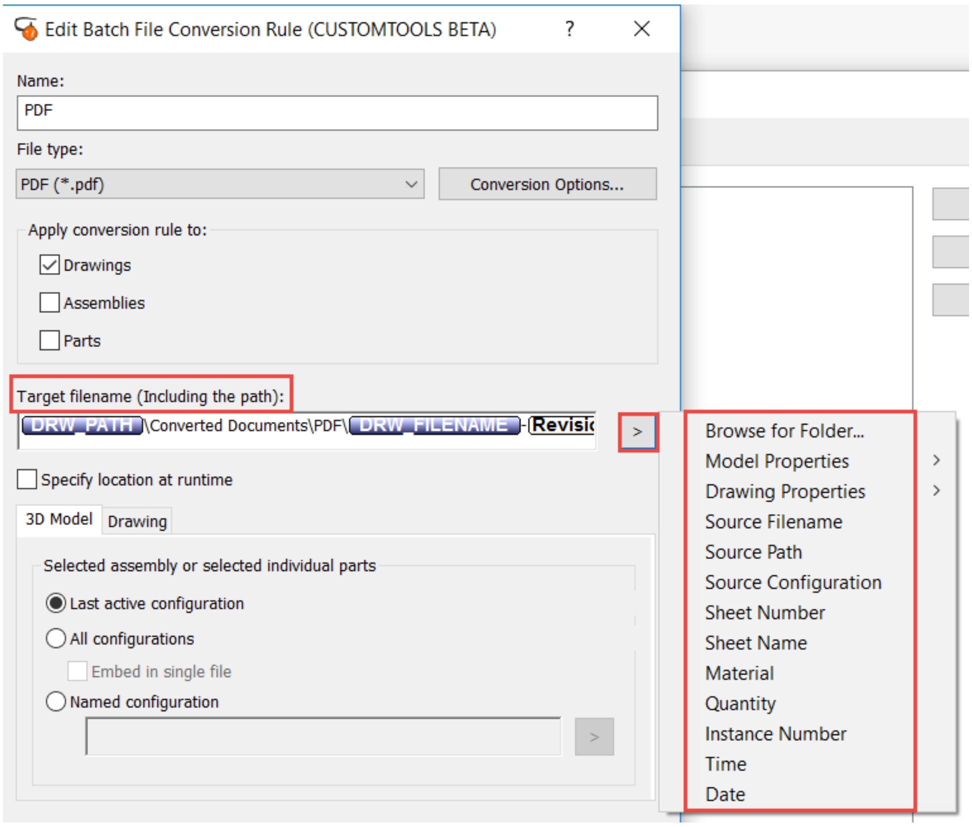
c. Select the last active sheet if referenced configuration is not available: Selects the sheet that was last active if the specific referenced configuration cannot be found By clicking on the **>**

i. **AND &**: Selects the sheets to be converted based on multiple criteria that must be contain in the sheet name in order to be selected.
ii. **OR |**: Selects the sheets to be converted if one of the defined criteria matches.
iii. **NOT !**: Excludes the sheets from being selected if it matches with the criteria. This can be used to exclude certain sheets from being selected (*E.g.: Sheet called DXF*).

6. **Execute rule:** Run the conversion rule after the SOLIDWORKS file is saved into SOLIDWORKS.

Defining the destination and naming rules used by the conversion rule

The path and naming rules used for the converted SOLIDWORKS files can freely be defined.



The storing and naming rules are defined under the **Target filename (Including the path)**. By clicking on **>** then different options are offered to the users.

Defining the location where to save the converted files:

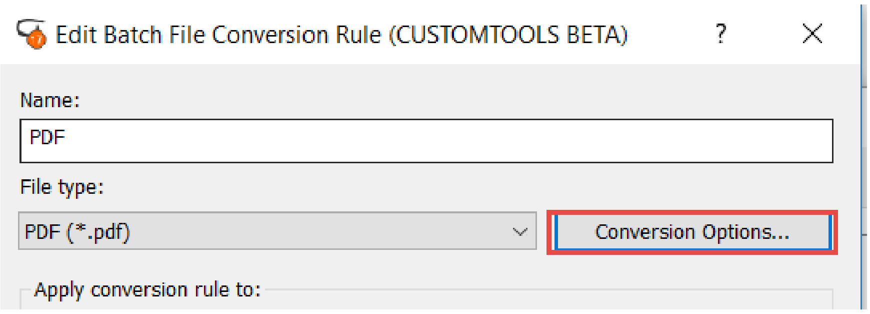
Select the **Source path** to store the converted files in the same folder as the SOLIDWORKS source files. A folder and/or sub-folder structure can also be built. Folders to be created can also be manually typed and created automatically during the conversion (*e.g.* *Converted documents**PDF*\).

Defining the file naming convention of the converted files:

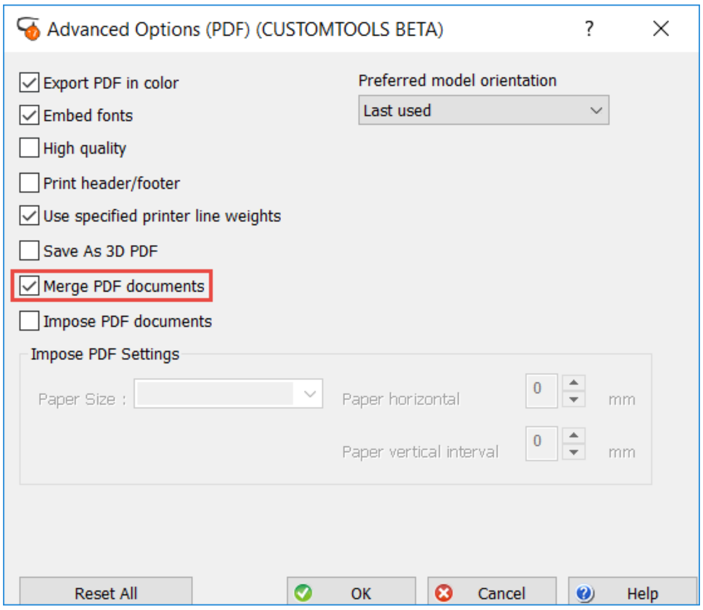
Select the **Source filename** to use the same file name between the SOLIDWORKS source file and the converted file. More advanced file name can be generated by combining different rules (e.g. *property value, source configuration, sheet name project number…*).

Merge PDF Option

All the drawings made for an assembly can be merged into a single PDF file.



From the **Add batch file conversion rule** click on **Conversion Options** to open the **Advanced Options** dialog.



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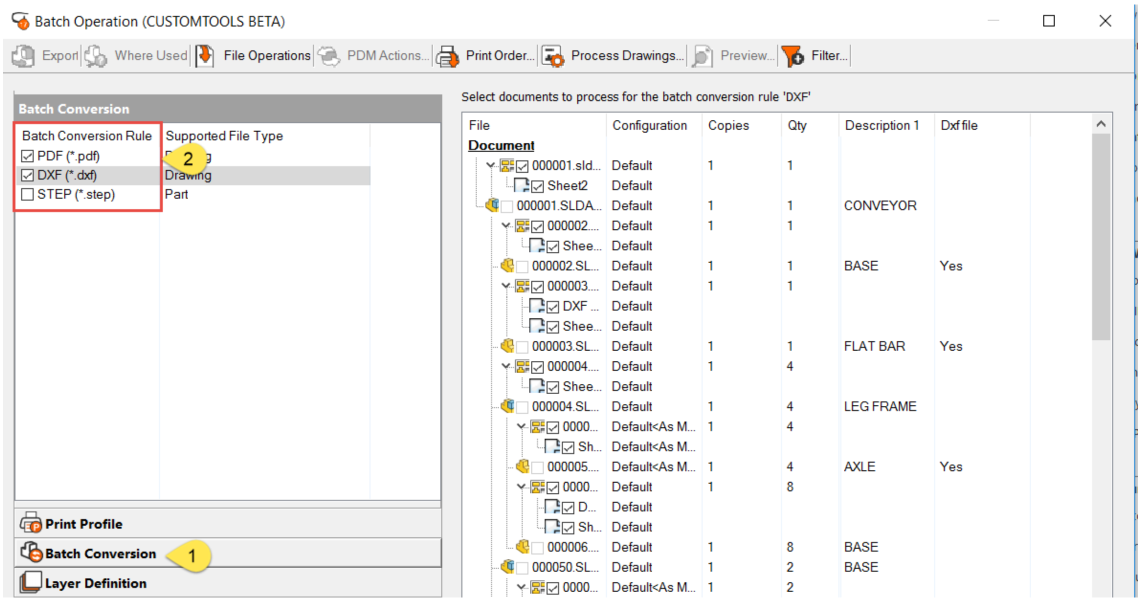
Select the **Merge PDF documents** check box and click on **OK**.

NOTE: The merge PDF is created in the BOM Order.

Converting your SOLDIWORKS Drawings

From an opened assembly in SOLIDWORKS

To batch convert all the drawings related to an assembly. Open the top level assembly in SOLIDWORKS and click **Print & Convert** to open the **Batch operation** dialog.



Open the **Print & Conversion** functionality:

1. Click on the **Batch Conversion** button, to view the **Batch Conversion Rule**:
2. Select the **Batch conversion** rules to be used.

The sheets of the listed drawings will be automatically selected based on the settings defined for the selected file conversion rule in the **Options**. Click **OK** to start the printing process.

If no documents are opened in SOLIDWORKS

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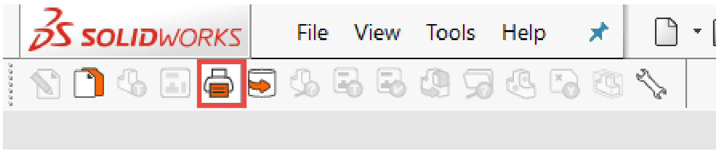
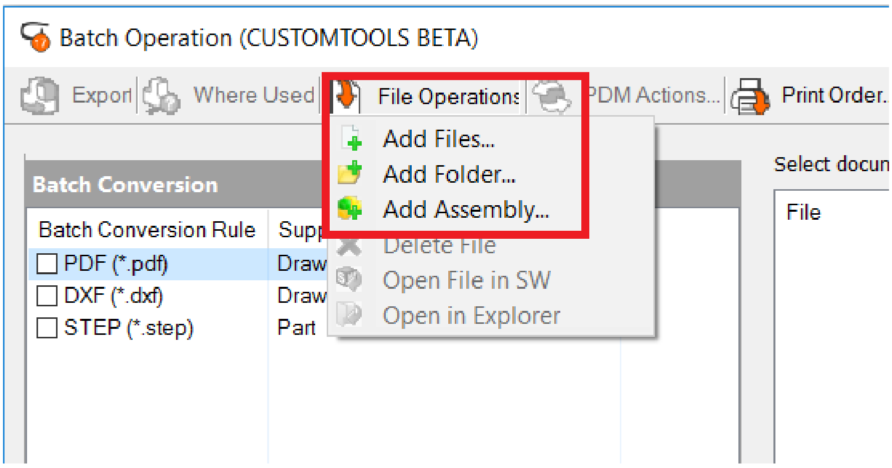


Opening the Batch operation dialog

The **Print & Convert** functionality can also be accessed if no documents are opened in SOLIDWORKS. Click on the **Print and Convert** icons from the **CommandManager** to open the batch operation dialog.

Adding the files to be printed

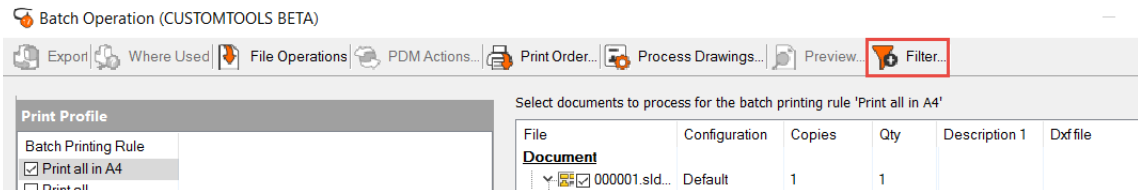
When the **Batch operation** dialog opens, the list of files to process is empty.



To add files to the list of files to process, click **File Operations**, then CUSTOMTOOLS offers three possibilities to select the files to add to the list of files to convert:

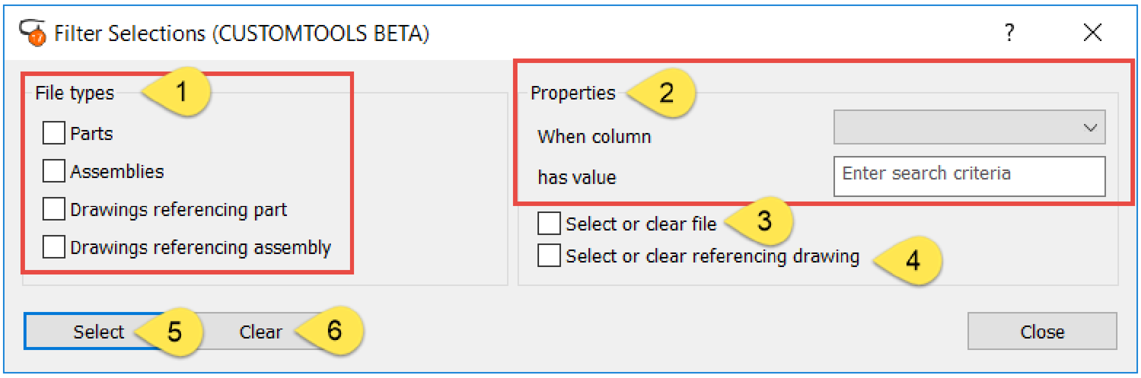
- **Add files**: The files to be added are selected by the user from the Windows Explorer.
- **Add folder**: The user selects the Folder where the files to be added are located. If the **Include Subfolders** check box is selected, then files located in sub-folders will also be added
- **Add assembly**: The user selects an assembly and all child components are automatically loaded.

Filtering the documents to be selected for conversion



To open the **Filter Selections** dialog, click **Filter** from the **Batch Operation** dialog.

NOTE: The filters are applied to the files selected by the selected profile.



1. **Files types**: The filter can be used to select or exclude certain file types from being selected, parts, assemblies, drawings made for part or assemblies can be selected.

   TIP: The filter can be used to convert only the assemblies' drawings, if the **Drawing referencing assembly** check box is selected.

2. **Custom properties**: Documents can also be converted based on a property value. From the **When Column** menu, select the property used to filter the selection and define the selection criteria from the **Has value** text box.

   NOTE: Custom properties used has filter must have the option **Display in Print and convert** selected from the **Property wizard** on the **Additional Options** page.

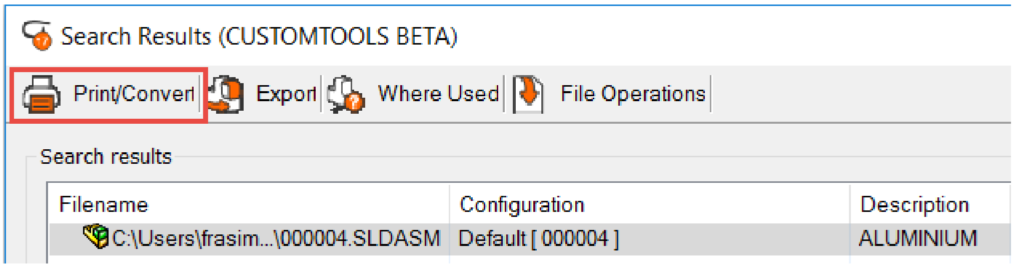
3. **Select or clear**: If the Property is defined at the part or assembly level then parts or assemblies will be selected/cleared.
4. **Select or clear referencing drawing**: If the Property is defined at the part or assembly level then the drawings made for those parts or assembles will be selected/cleared.
5. **Select**: Select the files that matches with the filter criteria defined in the **File type** or **Custom properties.**

6. **Clear:** Clear the files that matches with the filter criteria defined in the **File type** or **Custom properties.**

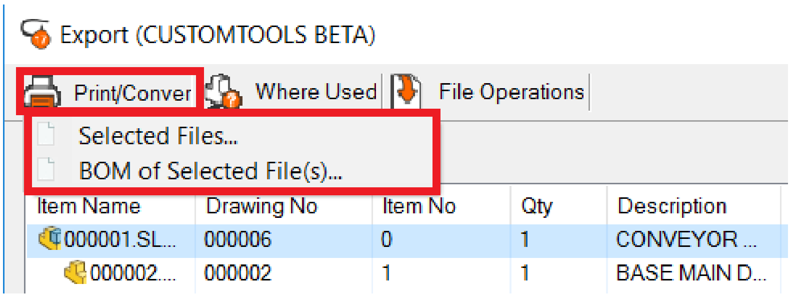
From other functionalities of CUSTOMTOOLS

The batch conversion functionality of CUSTOMTOOLS can also be launched from:

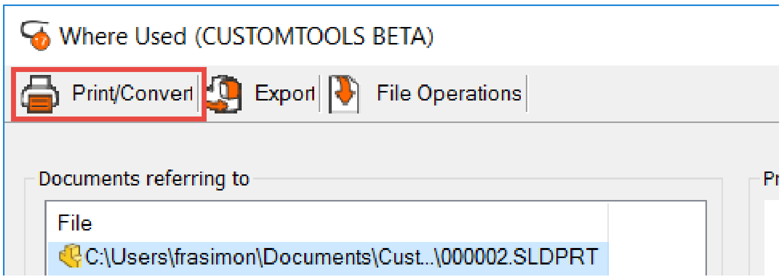
- **The Search results:**
  Select an assembly from the search result and click **PrintAndConvert.** If an assembly has been selected, click **BOM of Selected File(s)...** to add the assembly and its child component to the list of document to process.



- **Export**



- **Where used or Open drawing**



Creating DXF from the SOLIDWORKS Drawings with the Cutting Profile

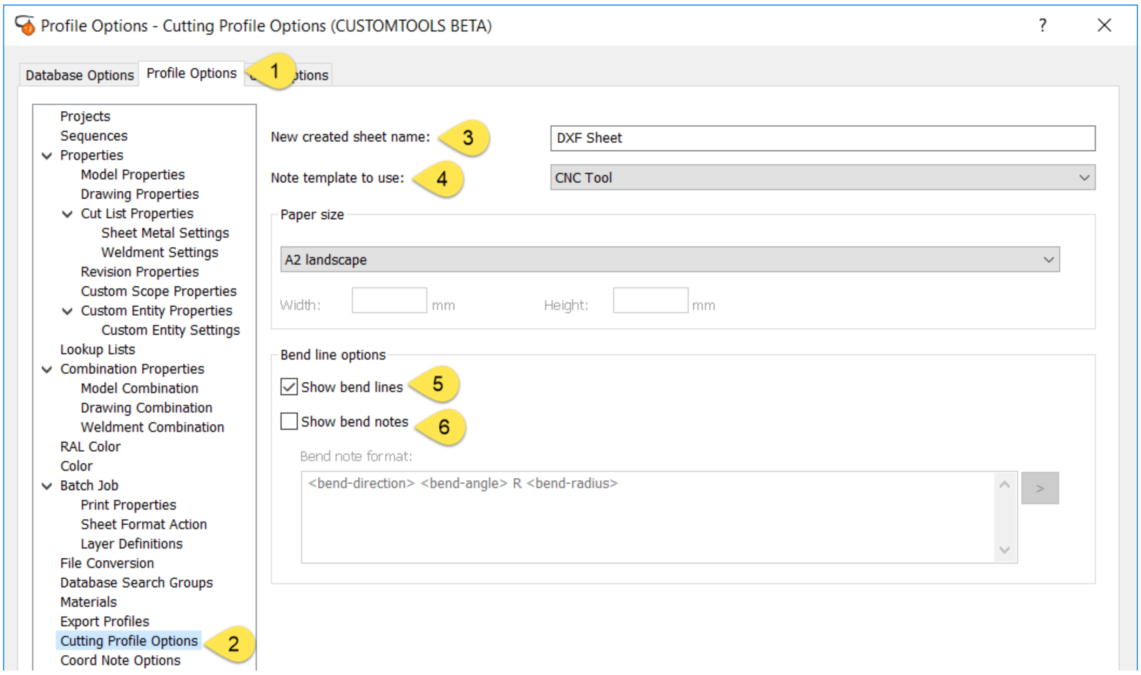
Using the Cutting Profile

Use the **Cutting Profile** functionality (available only from the SOLIDWORKS Drawings) to add a new sheet to the drawing. The sheet that is added is a flat pattern view of the sheet metal, scaled 1:1 and without any annotations.

The **Cutting Profile** can be accessed from the Command manager or from the CUSTOMTOOLS menu.

Defining the name used by the sheet created by the Cutting Profile

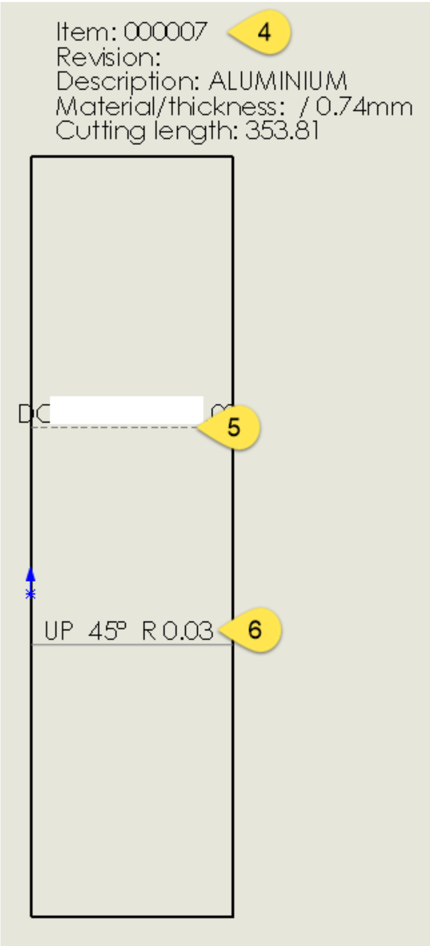
The name of the sheet that will be added by the **Cutting Profile** into the drawing can be customized.



Open the CUSTOMTOOLS **Options**,

1. From the **Profile Options** tab,
2. Select **Cutting Profile Options**, from the tree view,
3. Enter the name of the drawing sheet created by the **Cutting Profile** in **New created sheet name.**

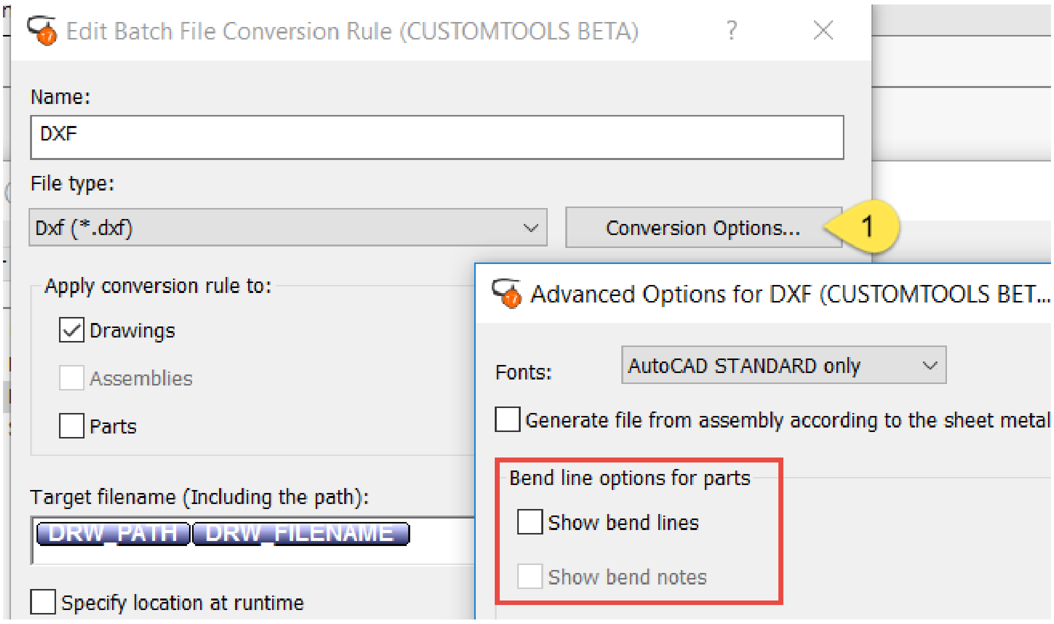
TIP: The drawing sheet created by the **Cutting Profile** can be associated with a conversion profile based on the sheet name. (*E.g. with a DXF file conversion rule*).



4. A **Note template**, defined in the **Database Options, Note Templates** can also be inserted when the view is flattened. The note can be configured with property values.
5. **Show bend lines:** Bend lines can be inserted in the flattened view.
6. **Show bend notes:** Information related to the bend can also be inserted.

1. Select **Show bend notes** to insert the bending information,

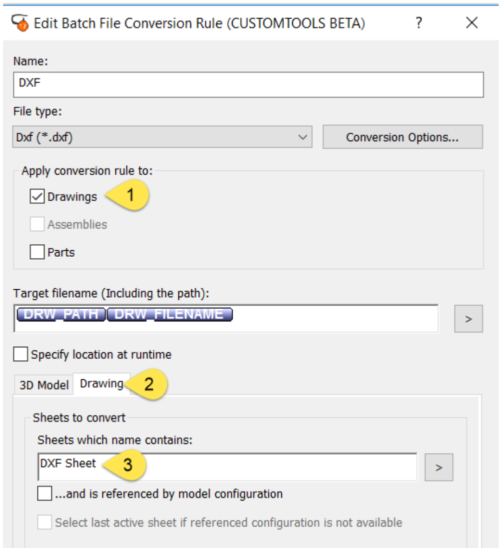
2. **Bend note format**: Configure what attributes will be used in the note and how they will be display,
3. List of attributes available to customize the bend note.

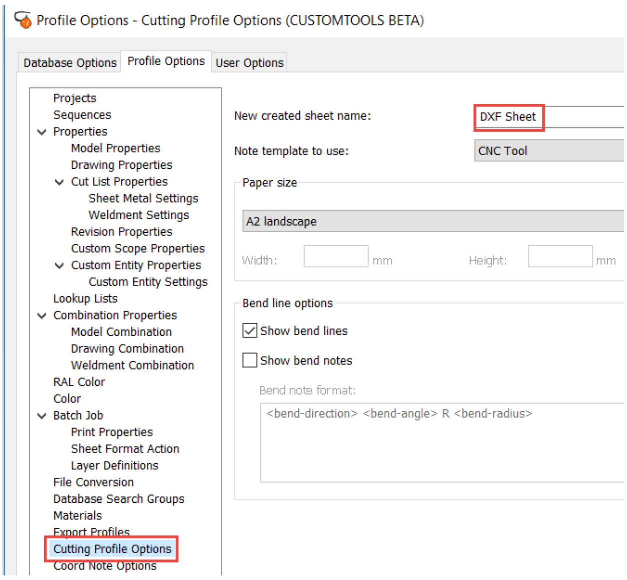


NOTE: The **Show bend lines** and **Show bend notes** can also be used if you are creating DXF directly from the parts. The Bend lines options are available in the **Conversion Options** of the conversion rule.

Configuring the file conversion profile

File Conversion profiles that automates the creation of DXF from drawing can be configured to select only the sheets that were created by the **Cutting Profile**.

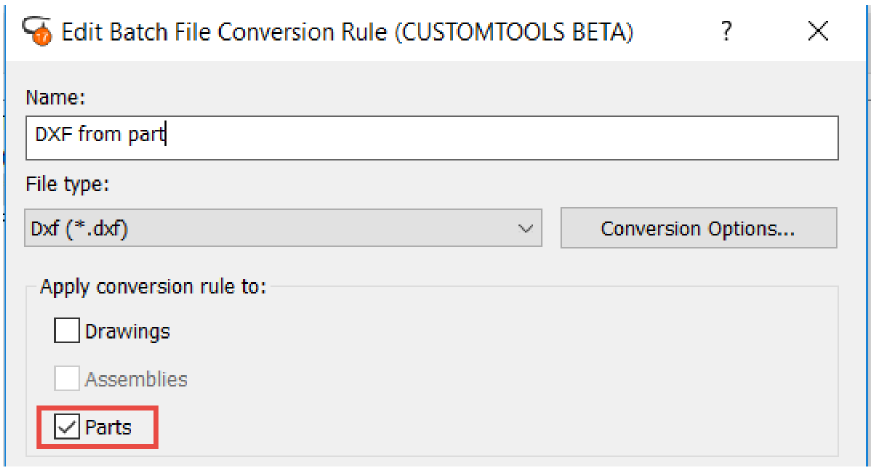




Edit/create a **File conversion rule.** From the **File type**, select **DXF** or **DWG.**

1. Select **Drawings** from **Apply conversion rule** to,
2. Activate the **Drawing** tab,
3. Enter the name of the sheet created by the Cutting Profile in CUSTOMTOOLS (*E.g. DXF Sheet*) in the **Sheet which name contains** text box.

Creating DXF from the SOLIDWORKS parts



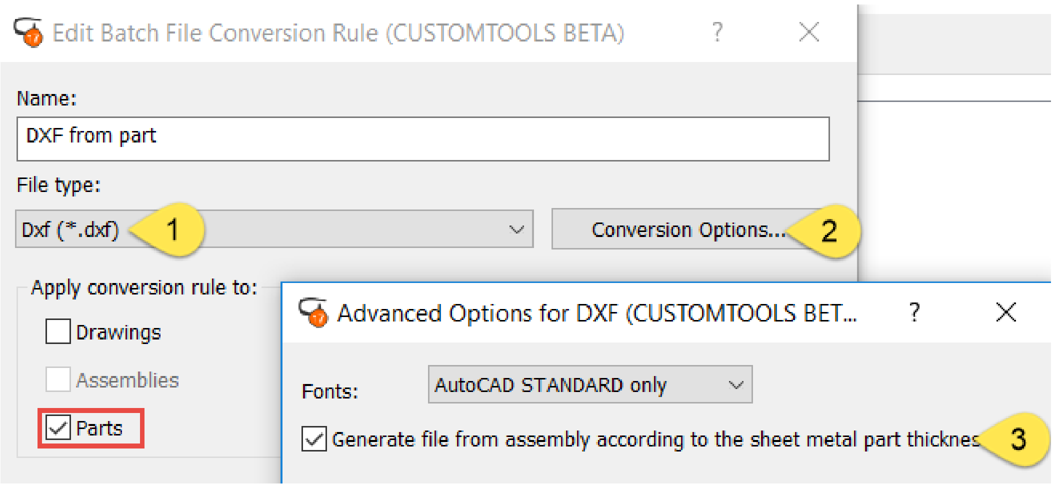
To automate the creation of DXF/DWG file from part, create/edit a file conversion profile.

- From the **File type** select **DXF** or **DWG**. Select **Parts** from **Apply conversion rule to.**

CUSTOMTOOLS

Merging sheet metal parts into the same DXF files

CUSTOMTOOLS can be used to merge sheet metal plates having the same thickness and using the same material into the same DXF/DWG file.



To merge sheet metal into the same DXF, open/edit a **File conversion rule**.

1. Select **Dxf** or **Dwg** from **File type**.
2. Then click on the **Conversion Options** to open the **Advanced Options for DXF** dialog.
3. Select the **Generate file from assembly according to sheet metal part thickness** check box.

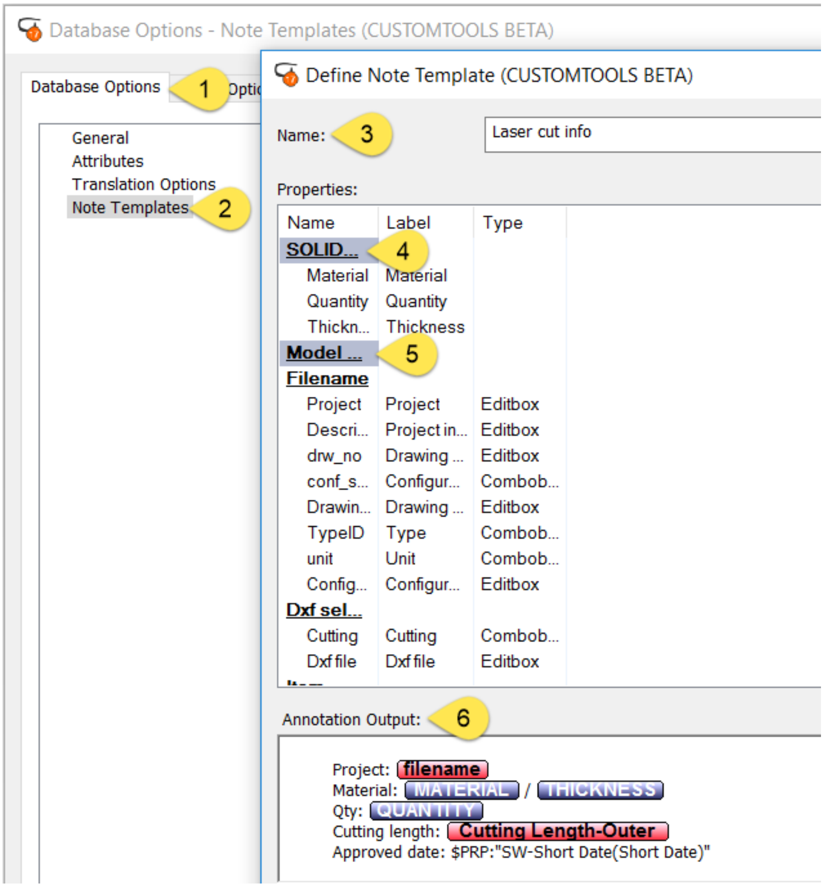
NOTE: Please the merge DXF options is only available when the conversion rule applies to Parts.

Inserting a property note in the converted DXF

Create a note template

DXF or DWG files created by CUSTOMTOOLS can be customized with notes containing property values.

To create a DXF Note template, open the CUSTOMTOOLS **Options**, activate the **Database Options** tab, then select **Notes templates**.

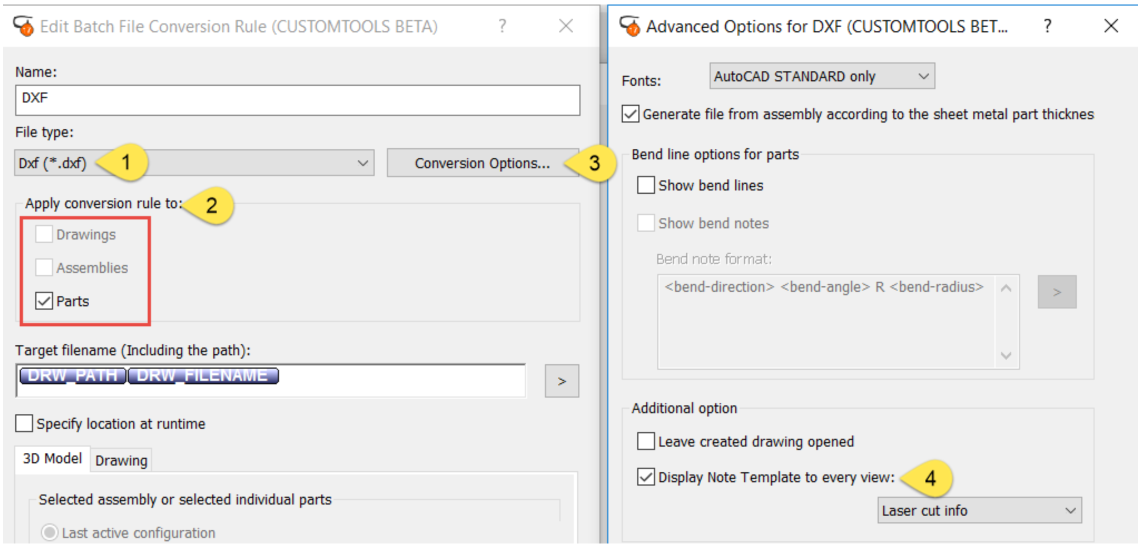


1. Activate the **Database Options** tab,
2. Select **Note Templates,**
3. **Name**: Defines the name of the note template as it will appear in the **Conversion Options** of the **File conversion rule** if a DXF or DWG type has been selected,
4. **SOLIDWORKS**: Inserts a SOLIDWORKS attributes available: Material, Quantity or Thickness,
5. **Model Properties**: Inserts a property defined in CUSTOMTOOLS,
6. **Annotation Output**: Defines how the note will be formatted. Text can be manually typed and custom properties to be added can be dragged and dropped or doubled clicked.

Associate a note template with a file conversion profile.

Once a note template has been added it can be associated with a **File conversion rule**.

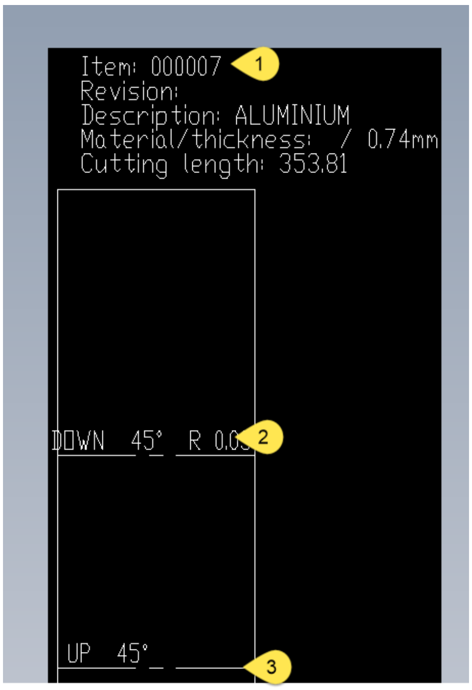
CUSTOMTOOLS



To associate a **CustomNote Template** to a **File conversion rule**, open/edit a file conversion profile.

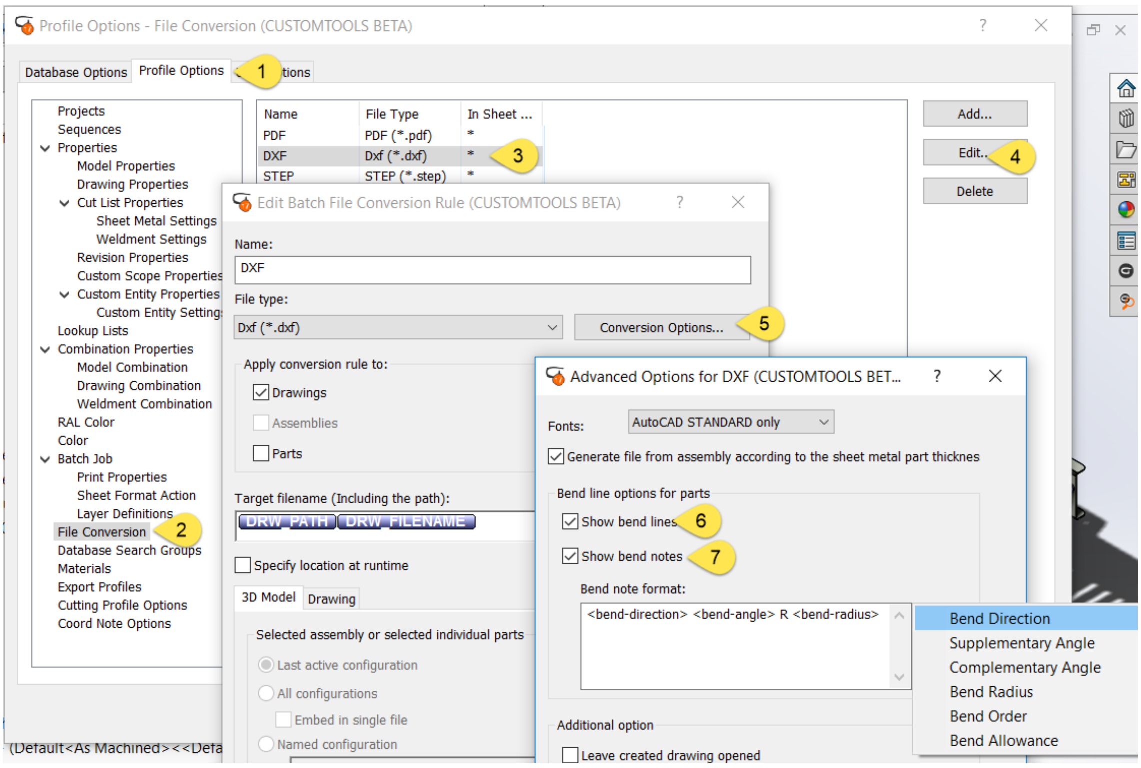
1. Then select **Dxf**/**Dwg** from **File Type**,
2. Select **Drawings** or **Parts** from **Apply conversion rule to,**
3. Then click on **Conversion Options** to open the **Advanced Options for DXF** dialog,
4. Select the **Display CustomNote Template to every view** check box. Then select the **CustomNote template** to be used by the **File conversion rule**.

Inserting bend lines and bend notes



Bend lines and Bend notes can be inserted in the output DXF files created by CUSTOMTOOLS.

1. Notes configured with CUSTOMTOOLS or SOLIDWORKS attributes can be inserted. The note is defined in the **Note template** under the **Database Options**, in the **CUSTOMTOOLS Options,**
2. **Show bend notes:** Information related to the bend can also be inserted,
3. **Show bend lines:** Bend lines can be inserted in the flattened view.

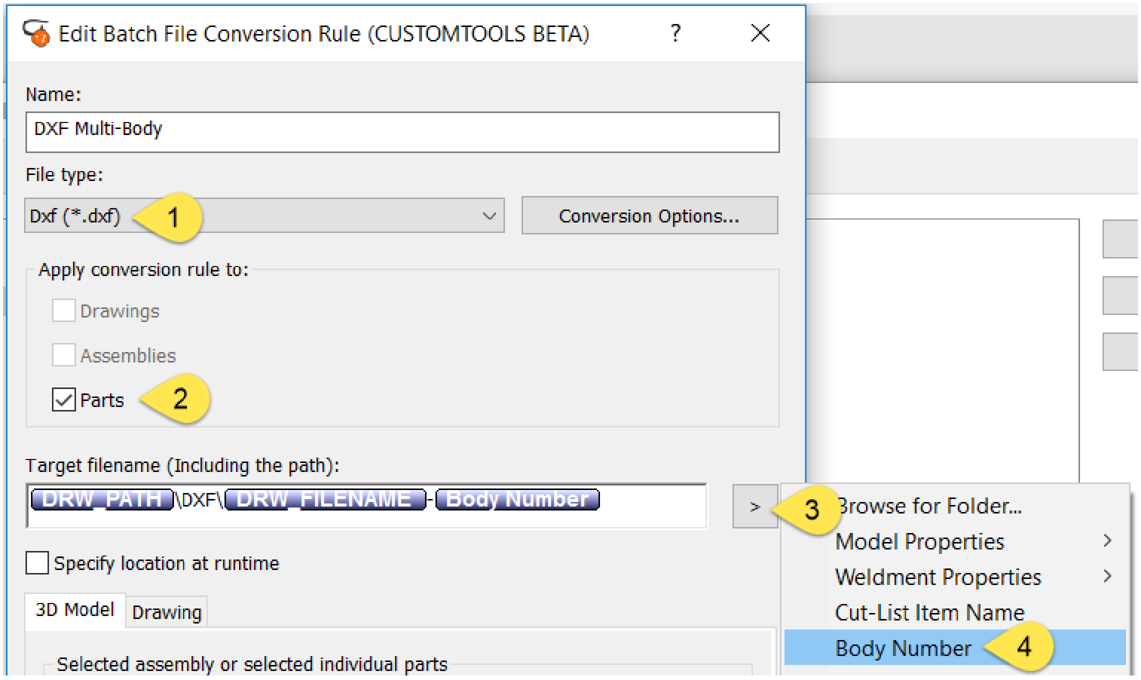


To insert Bend lines and bend notes in DXF created from part

1. From the **Profile Options** tab,
2. Select **File Conversion,**
3. Select a profile,
4. Click **Add,**
5. Click on the **Conversion Options,**
6. Select **Show bend lines** from the **Bend line options** for part to insert bend lines in the created DXF,
7. Select **Show bend notes** to activate the **Bend note format** and configure how the note will be inserted and displayed in the DXF.

Creating DXF for multi-body parts

If working with multi-body parts, it can be defined how CUSTOMTOOLS will create DXF files: one DXF file per body or one DXF file containing all the bodies (Merged).



To create a DXF per sheet metal body, **Add** or **Edit** a File conversion rule.

1. Select **Dxf** or **Dwg** from the **File type,**
2. Select **Parts** from the **Apply conversion rule to:**

   NOTE: The creation of individual Dxf per body can only be ran from parts, NOT from drawings.
3. Click on **>** from the **Target filename** to expand the menu,
4. Select **Body Number** or **Cut-List Item Name.**

NOTE: If **Body number** or **Cut-List Item Name** are not selected then multi-body part will automatically be merged in the same DXF/DWG file.

Chapter 8: Property Management

Introduction

What does it do?

CUSTOMTOOLS allows you to manage the custom properties of your SOLIDWORKS documents. The custom properties defined in CUSTOMTOOLS are automatically saved into the file properties of the SOLIDWORKS documents. The custom properties can also be managed specifically based configurations.

How does it work?

The Properties pane of CUSTOMTOOLS can easily be customized based on your needs. CUSTOMTOOLS offers numerous tools to customize the Properties pane.

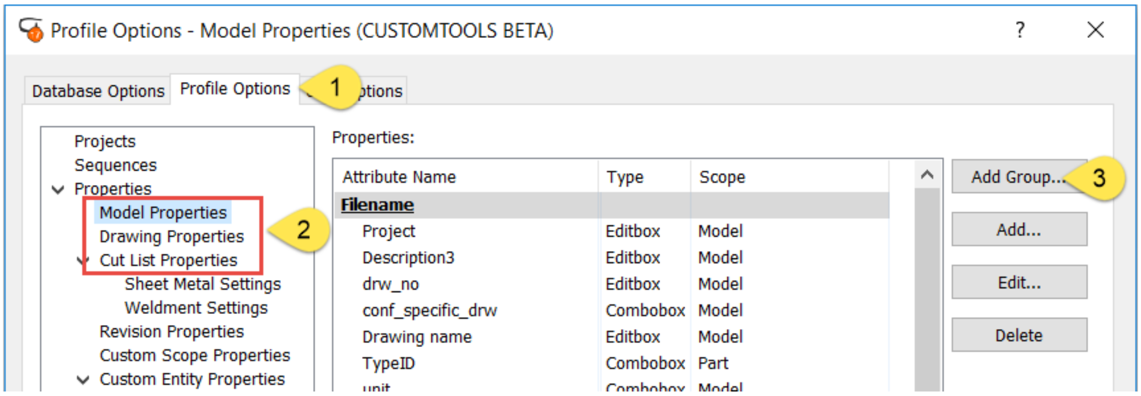
How can you use it?

CUSTOMTOOLS offers a Properties pane fully embedded into SOLIDWORKS. The Properties pane always opens for the active component that is being selected. If an assembly is opened, a part or sub-assembly can be selected and by clicking on Properties, it will open the custom properties of the selected component.

Managing group of properties

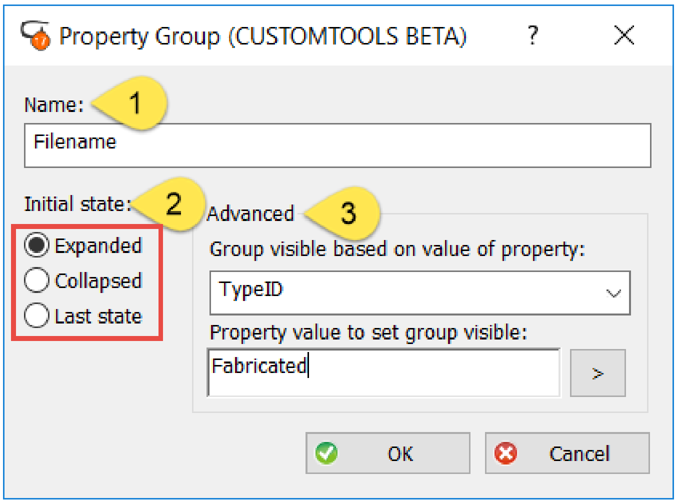
Adding a new group of properties

Custom properties are managed into group of custom properties in CUSTOMTOOLS.



To add a new Property group, open the CUSTOMTOOLS **Options**:

1. From the **Profile Options** tab,
2. Select **Model properties**, to add a group for Parts and Assemblies,
3. Click **Add group.**



1. **Name**: Define the name of the group of properties as it will be displayed in the Properties pane.
2. **Initial state**:
   a. **Expanded:** The property group will always be fully visible when the Properties pane is opened.
   b. **Collapsed:** The property group needs to be extended to display its properties.

c. **Last State**: CUSTOMTOOLS uses the last state defined for the last property group that was created.

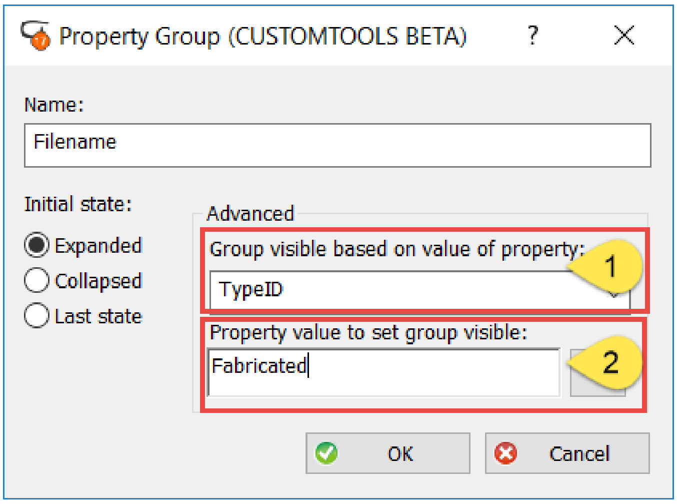
3. **Advanced**: The visibility of the group can be managed based on a property value.

Managing the group visibility

The visibility of a group of properties can be managed based property values used in other group of properties. If a Property value matches with the group visibility criteria, then the groups of properties will automatically appear.

To manage the visibility of a group of properties, open the CUSTOMTOOLS **Options**,

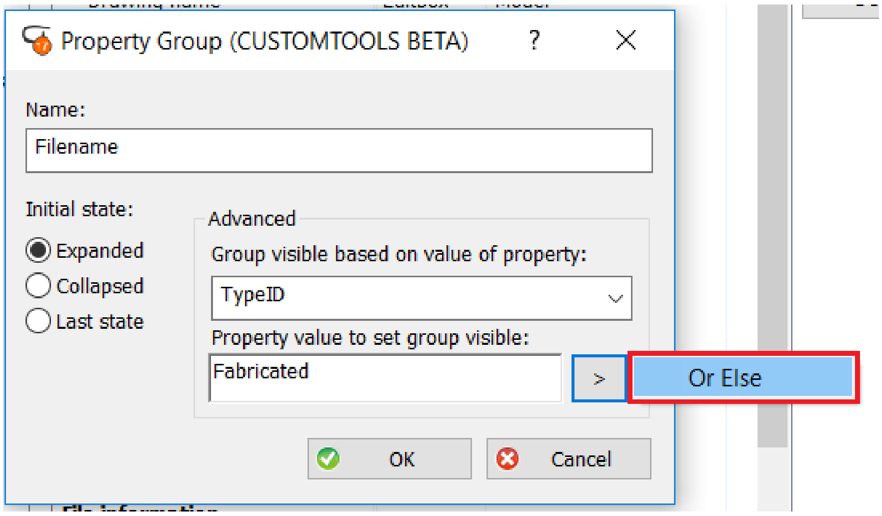
1. From the **Profile Options** tab, select **Model properties**,
2. Click **Add** or **Edit** a Property group.



From the **Advanced** options of the **Property Group** dialog,

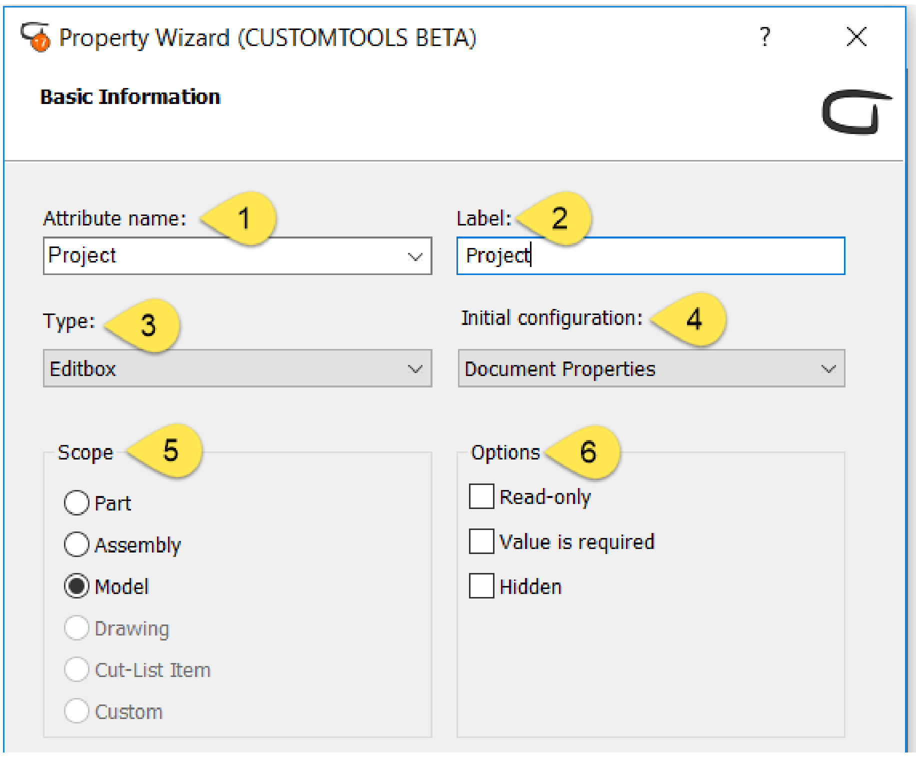
1. Select the property to be used to manage the group visibility. When the **Group visible based on value of property** is extended, a drop down menu containing all defined custom properties appears,
2. From the **Property value to set group visible**, define the property value used to control the property group visibility.
3. Click **OK** to save and close the dialog.

The group visibility can be managed based on multiple Property value. By clicking on the **>** sign you can select **Or Else** to add additional property values criteria.



Adding a new property

Properties basic information



1. **Attribute name:** Corresponds to how the property will be saved in the database. The attribute must be unique.
2. **Label**: Defines how the property name will appear in the Properties pane of CUSTOMTOOLS.
3. **Type**: CUSTOMTOOLS offers different type of properties:

a. **Checkbox:** Check boxes toggle between two predefined values.

b. **Combobox:** Contains a list of pre-defined values.

c. **Date:** Insert a date. The date format can be customized.

d. **Dimensions:** Link a dimensions of your 3D Model to a property.

e. **Editable combobox:** Contains a list of pre-defined values. If a value is not available in the combo, then it can be manually typed.

f. **Editbox:** Offers many different functionalities.

g. **Hierarchical combo:** Contains a list of pre-defined values with multiple levels.

4. **Initial configuration:** If you are using the configurations in SOLIDWORKS, defines to which configuration the property value will be written.
5. **Scope:** Defines to which types of SOLIDWORKS documents the property applies. Please note that Model applies to Part & Assembly.
6. **Options:**

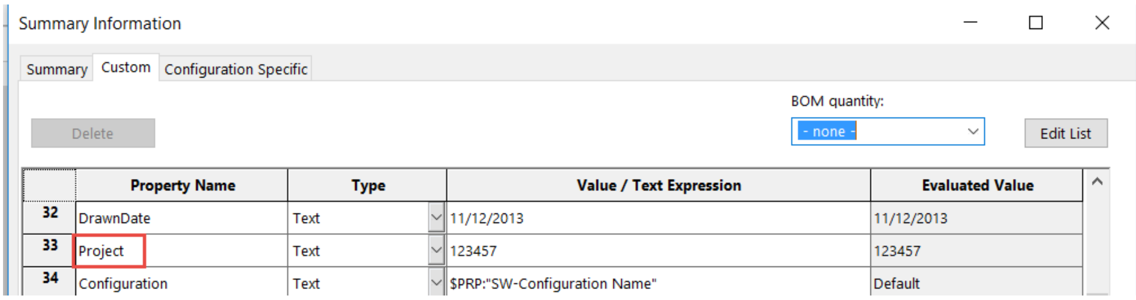
a. **Read-Only**: The property value is displayed but cannot be modified or deleted. It prevents other users from modifying the value of a property.

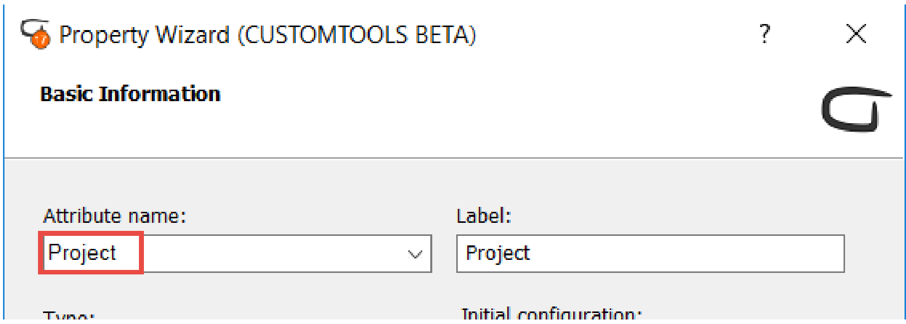
b. **Value is required**: The user is forced to define a property value in order to save the property.

c. **Hidden**: The property value is written to the document properties but hidden from the CUSTOMTOOLS Properties pane. This can be used with Key value pair list for example to hide a key, which can be used in a combination of properties.

Displaying an existing property in the CUSTOMTOOLS Properties pane.

The CUSTOMTOOLS Properties pane can be configured to display properties values that have already been written to the SOLIDWORKS Documents (e.g. in the EPDM data card, in the SOLIDWORKS Tab builder or the file properties).





To add an existing property in the CUSTOMTOOLS Properties pane, open the CUSTOMTOOLS **Options**.

1. From the **Profile Options** tab,
2. Select **Model properties** to add the property for part and assembly or Drawing properties to add the property for the Properties pane displayed for drawings,
3. Click **Add** to add a new property,
4. In the **Attribute name field** of the **Property wizard** use the exact same name as the one used as the property name in the file properties of the SOLIDWORKS document.

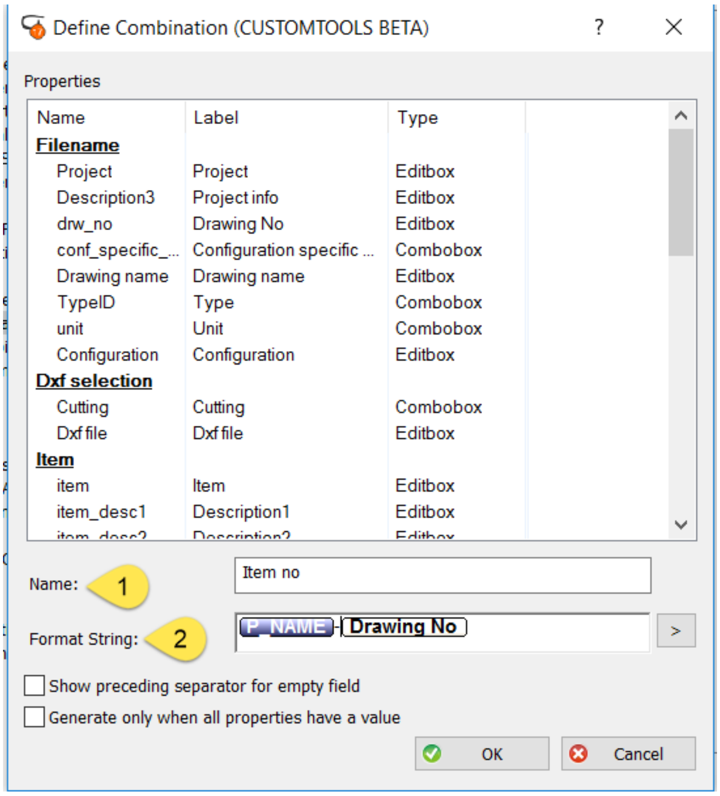
Combining multiple custom properties

Multiple custom properties can be combined together to create a new property. Combination of properties can be used for example to generate a filename.

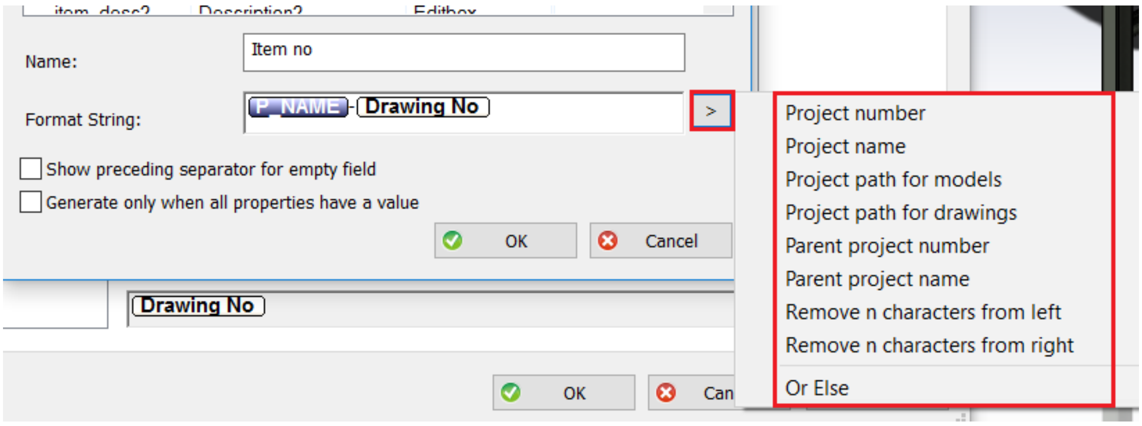
Creating a combination of properties

To create a combination of properties, open the CUSTOMTOOLS **Options**,

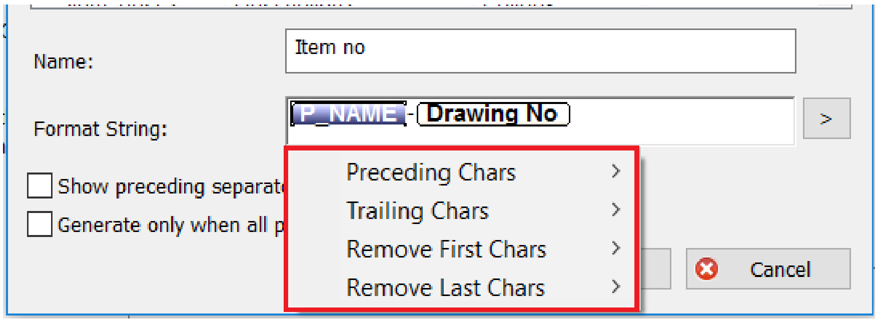
1. From the **Profile Options** tab,
2. Select **Model combination** from **Combination of properties**,
3. Click **Add** to create a new combination of properties,
4. The **Define combination** dialog appears:



1. **Name**: Define the name of the combination as it will appear in the Property wizard.
2. **Format String**: Define which custom properties are used in the combination. Custom Properties can be dragged and dropped or added by double clicking on the property to be added. If you click on the **>** key then a pre-defined list of values can be selected and added to the combination of properties *(e.g. Project number, Project name…).*



If one of the property used in the combination is selected, then the number of characters to be inserted can be controlled. If **Preceding Chars** is selected, then the number of the first characters to be inserted can be defined. (E.g. *property value: Aluminum -> insert the first 3 Characters used in the combination, then Alu will be inserted in the combination*). If **Trailing Chars** is selected, then the number of the characters to be inserted from the end of the selected property value can also be defined (*e.g. property value: Aluminum -> insert the last 3 Characters used in the combination, then num will be inserted in the combination*).



Creating a property that uses a combination of properties

Once the combination of properties has been created, it can be used by a property.

In the Property wizard (Options -> Profile Options -> Properties-> E.g. Model properties).

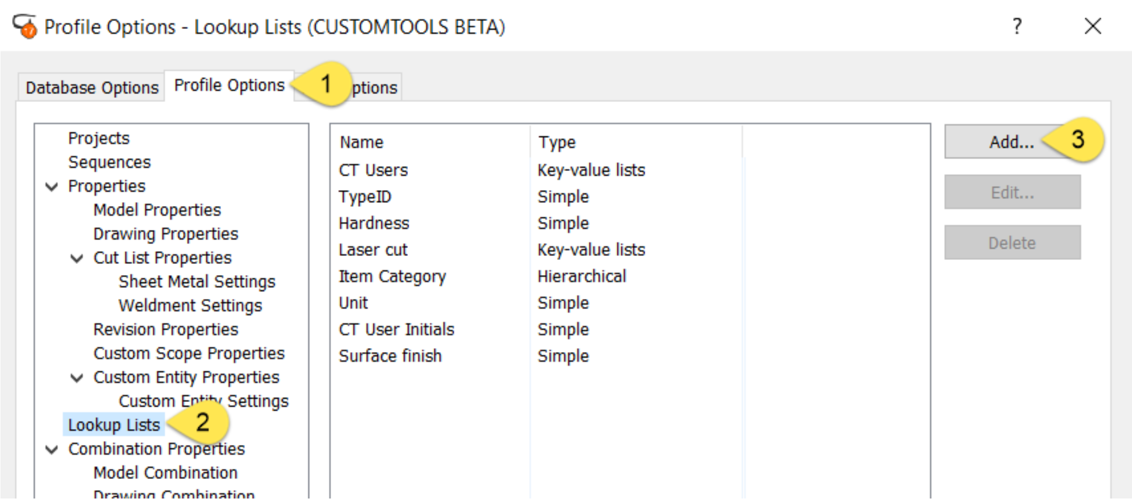
1. Select **Editbox** from the property **Type** in the **Basic Information** page of the Property wizard,
2. Then click **Next** to open the **Functions** page of the **Property wizard**.

1. From the **Data function**, select **GetCombinationValue,**
2. Select the combination to be used from the **Combination** drop down menu (*e.g. File name*).

Managing look up list

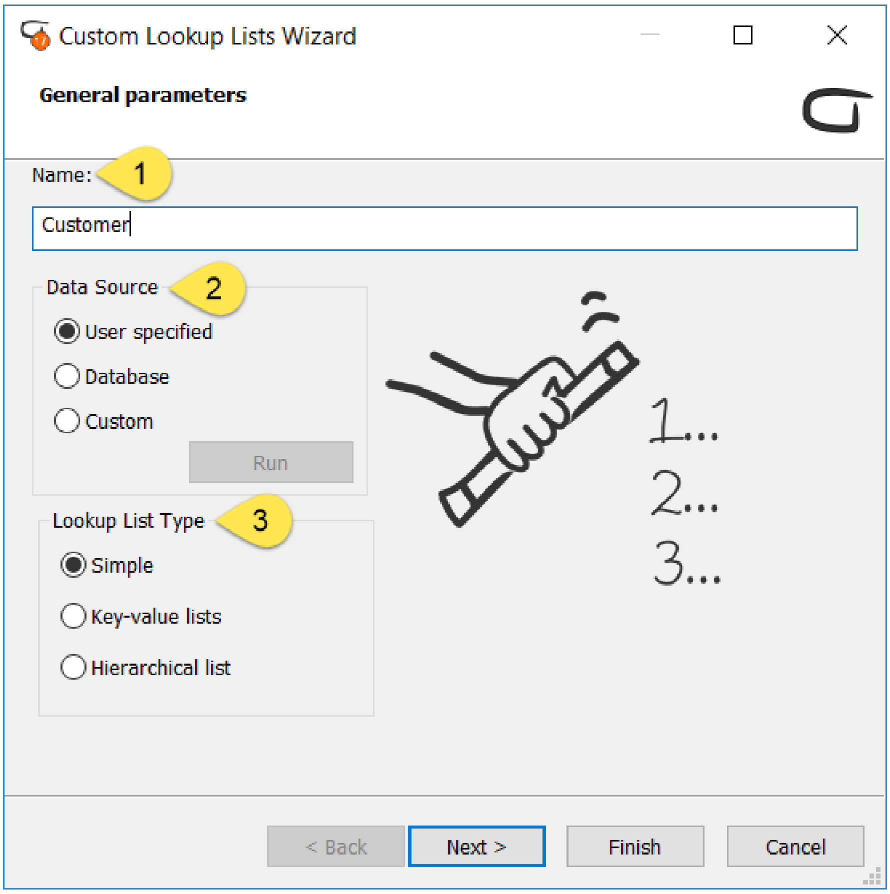
CUSTOMTOOLS offers different types of look up list to manage custom properties.

Creating a lookup list



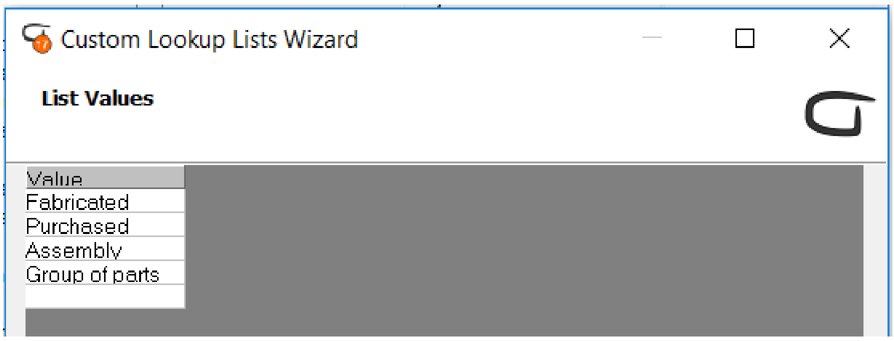
To create a lookup list, open the CUSTOMTOOLS **Options**,

1. From the **Profile Options** tab,
2. Select **Lookup list**,
3. Click **Add** to create a new lookup list. The **Custom Lookup Lists Wizard** opens.

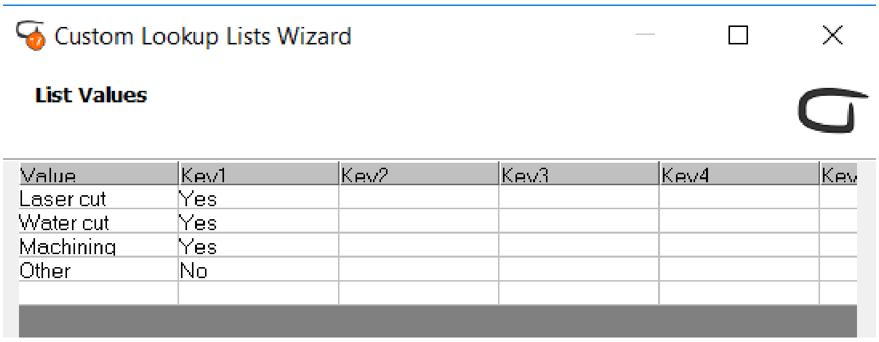


1. **Name:** Defines the name as it will appear in the **Property wizard**.

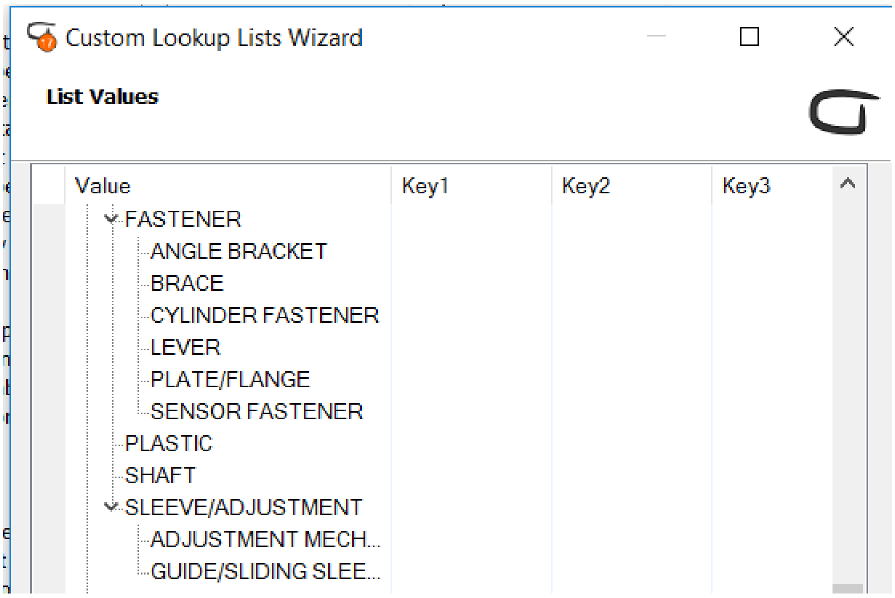
2. **Data source**: CUSTOMTOOLS offers different possibilities to fill the content of the look up list:
    a. **User Specified**: The lookup list content is defined manually by the user.
    b. **Database**: The lookup list content is pulled from an external data source by using an SQL Query.
    c. **Custom**: Calls a specific command.
3. **Lookup List Type**:
    a. **Simple:** The content is defined in a single column.



    b. **Key-value lists:** The content is defined in multiple columns. Multiple keys (up to 15 keys) can be defined for the same value.



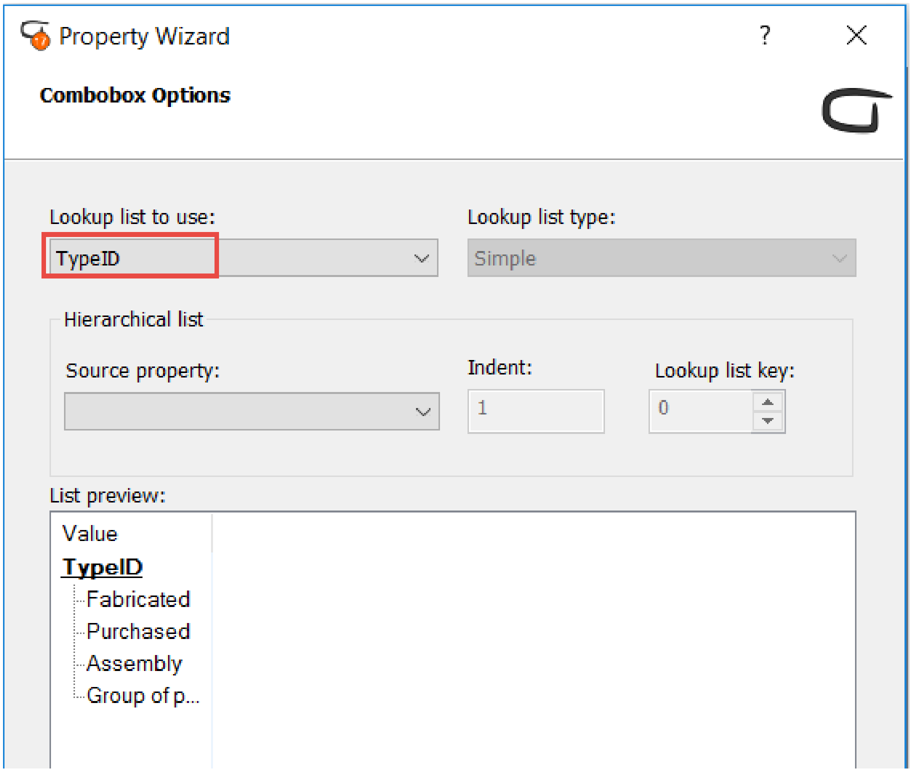
    c. **Hierarchical list:** The content of the lookup list can have multiple levels.



Creating a property that uses a look up list

Once the look list has been created, it can be used by a property.

In the **Property wizard** (Options -> Profile Options -> Properties-> E.g. Model properties). Select **Combobox, Editable Combobox** or **Hierarchical Combo** from the property **Type** in the **Basic Information** page of the Property wizard. Then click **Next** until the **Combobox Options** page of the **Property wizard** appears.

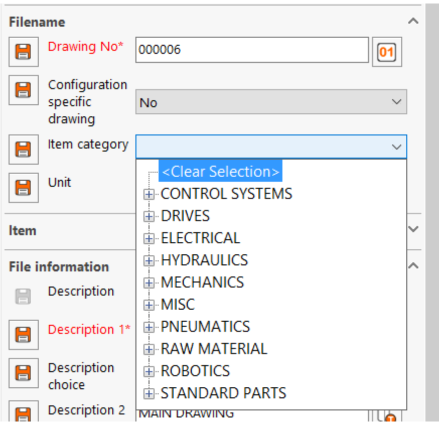


1. Select the look up list to be used by the property from the **Lookup list to use** menu,
2. Then click **Finish**.

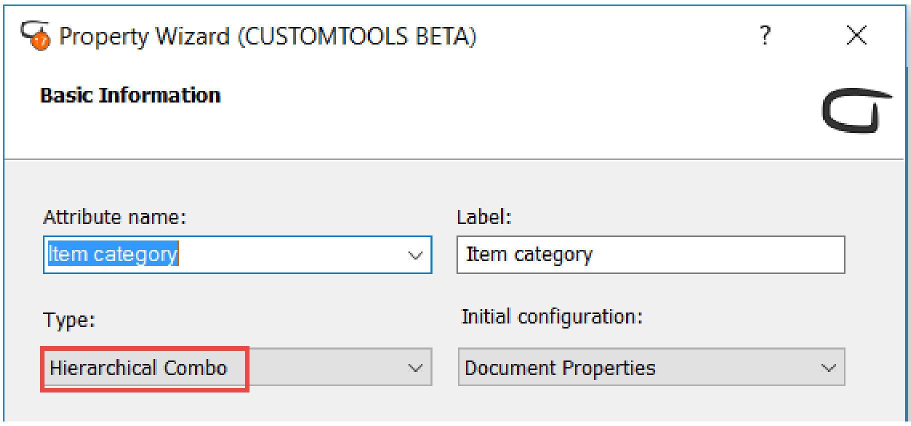
Creating a property that uses a hierarchical look up list

CUSTOMTOOLS offers two different ways to link a hierarchical look up list to a property. The multi-level look up list can be managed in a single property or each level can be managed in a separate property.

Using a single property to manage a hierarchical lookup list

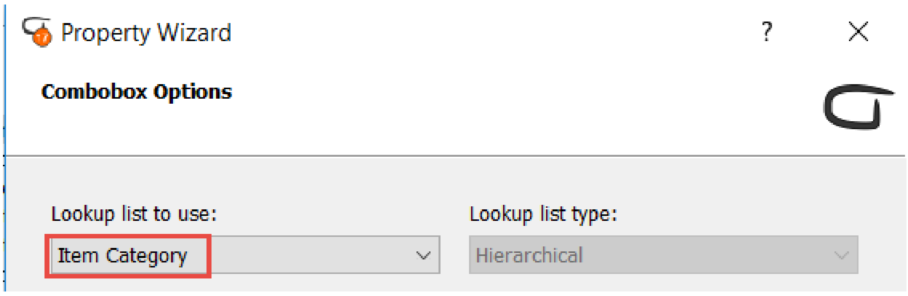


The single property allows the users to view the different levels of the hierarchical look up list within the same property. All the levels can be extended.



In order to link a hierarchical look up list to a property, create a new property (Options -> Profile Options -> Properties-> E.g. Model properties).

1. Select **Hierarchical Combo** from the property **Type** in the **Basic Information** page of the **Property wizard**,
2. Then click **Next** to access the **Combobox Options** page of the **Property wizard** appears,

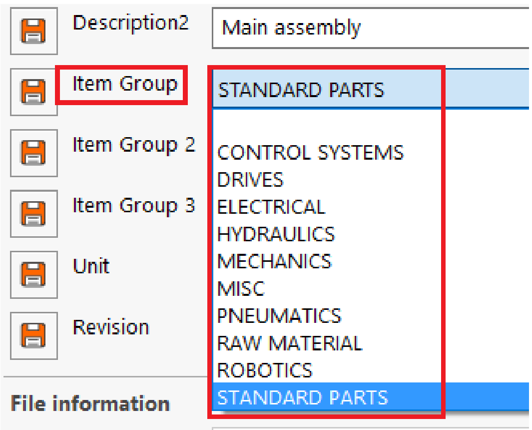


3. From the **Lookup list to use**, select the hierarchical look up list to be used by the property. Please note that only hierarchical look up list that have been created will appear in the selection.

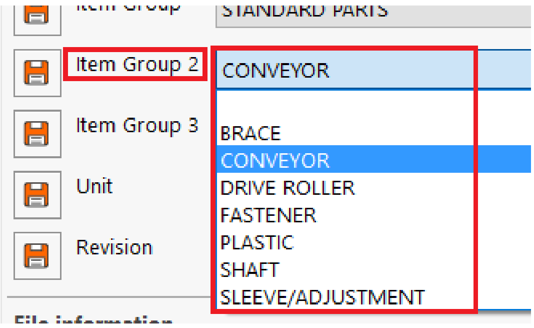
Using multiple custom properties to manage a hierarchical lookup list

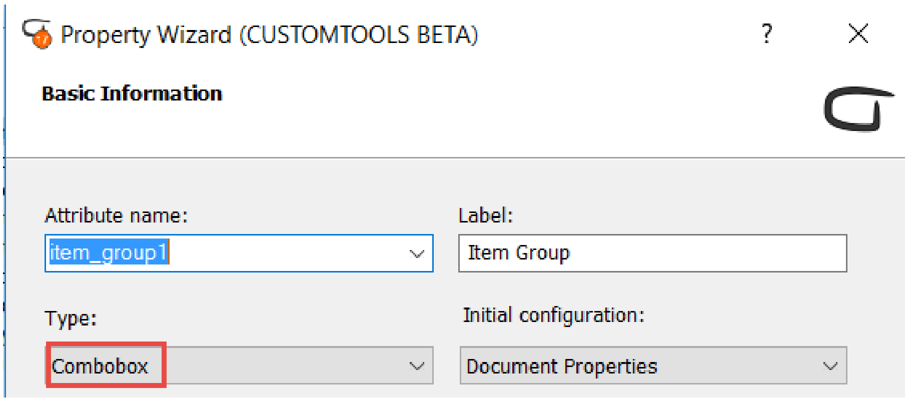
Multiple custom properties can be used to manage the different levels defined in a hierarchical lookup list, where one property is used per level. The content available in the different look up lists of the child updates automatically based on what was selected from the parent look up list.

Selecting a property value from the first level:



Once the value has been selected for the first level (*e.g. Item group: STANDARD PARTS*), the content available for the next property (*e.g. Item group 2*) updates accordingly.

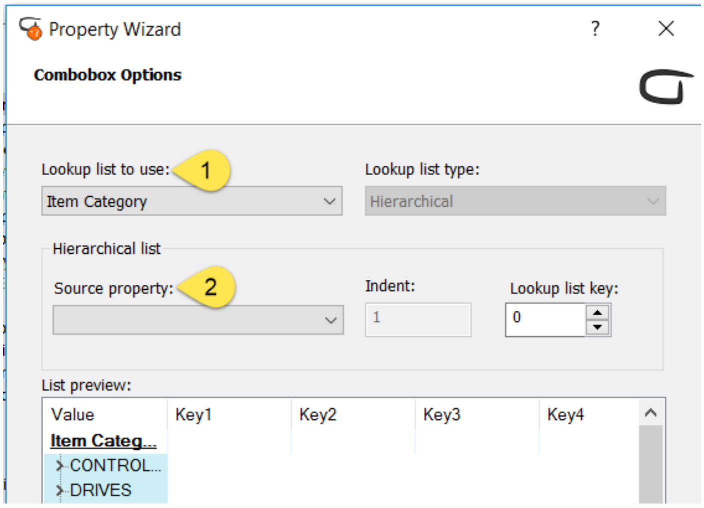




In order to link a hierarchical look up list to multiple custom properties, create one property for each level used in the hierarchical lookup list (Options -> Profile Options -> Properties-> E.g. Model properties).

1. Select **Combobox** from the property **Type** in the **Basic Information** page of the **Property wizard.**
2. Then click **Next** to access the **Combobox Options** page of the **Property wizard** appears.

Defining the first level:

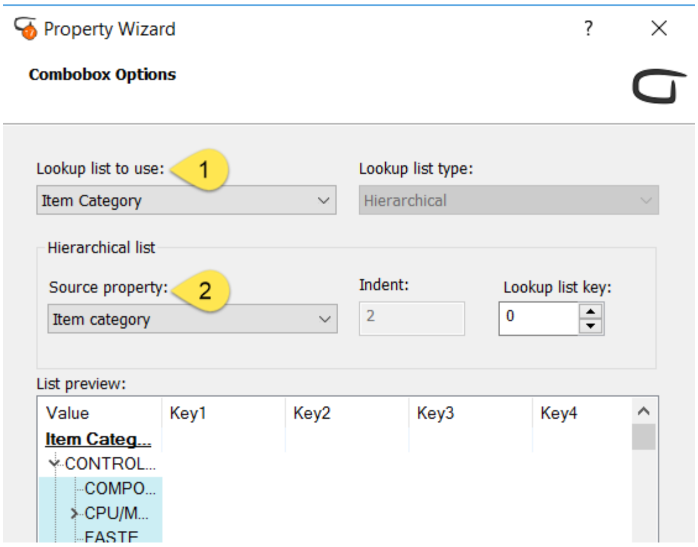


From the **Lookup list to use** menu,

1. Select the hierarchical lookup list to be used. The first level of the hierarchical lookup list is automatically selected.
2. Please note that the **Source property** must be left empty for the first level.

Defining child level:

Create a property for each child level (Options -> Profile Options -> Properties-> E.g. Model properties). Select **Combobox** from the property **Type** in the **Basic Information** page of the **Property wizard**. Then click **Next** to access the **Combobox Options** page of the **Property wizard** appears.

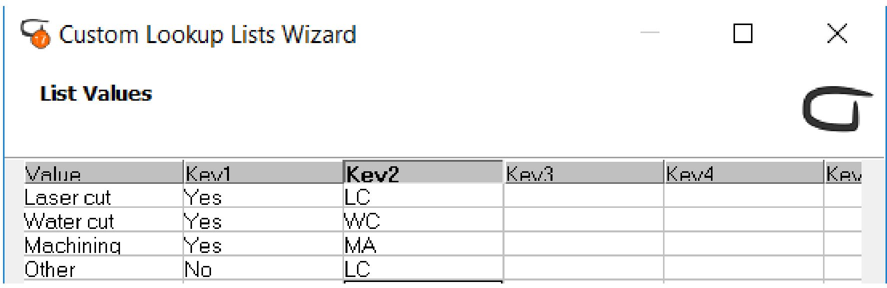


1. From the **Lookup list to use**, select the hierarchical look up list that was used earlier.
2. From the **Source property**, select the second level of the hierarchical lookup list. The second level corresponds to the first value available in the **Source property** menu.

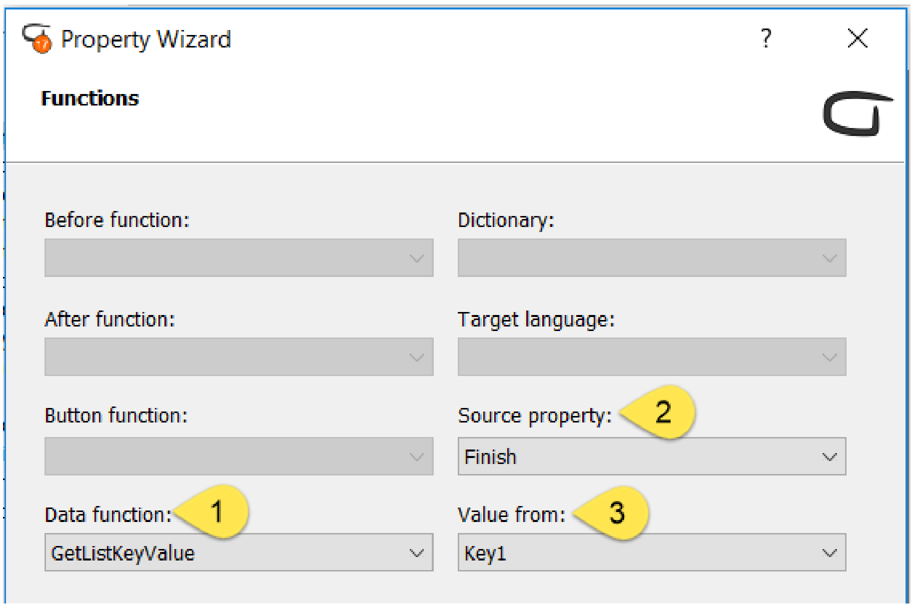
NOTE: Please note that the look up list content selected from the **Source property** can be previewed from the List preview.

Creating a property that uses a key retrieved from a look up list

CUSTOMTOOLS allows you to create look up list that can have up to 15 keys. Lookup list keys can be used in different cases (*e.g. File naming, filtering in the printing…*). Key value pair look up list allows the user to write additional property values to the document.



In order to link a property to a key, the look up list and property that uses that look up list must be created beforehand. In the Property wizard (Options -> Profile Options -> Properties-> E.g. Model properties). Select **Editbox** from the property **Type** in the **Basic Information** page of the **Property wizard.** Then click **Next** to access the **Function** page of the **Property wizard** appears.

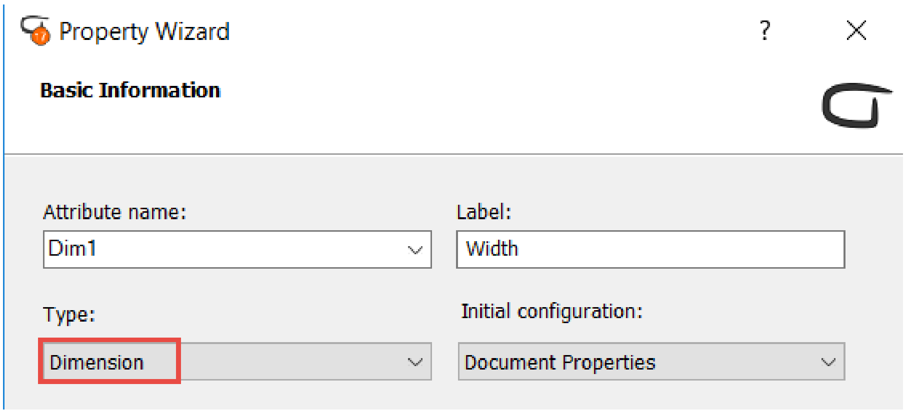


1. From the Data function, select **GetListKeyValue**,
2. From the **Source property** select the property that uses the look up list where the keys should be retrieved from,
3. Select the **Key** to be used from the **Value from** menu,
4. Click **Finish**.

Link a dimensions from the model to a property

Dimensions defined in the sketch can be linked to a property, by clicking on the property that uses a dimension type and selecting the desired dimension from the 3D Model side.

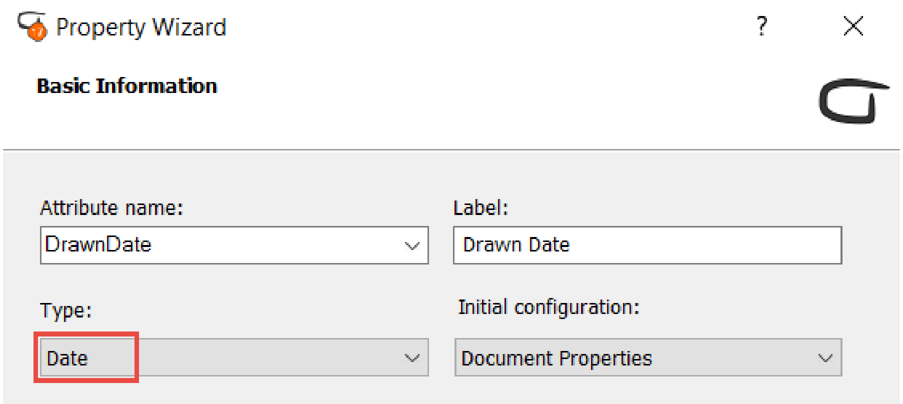
In the **Property wizard** (Options -> Profile Options -> Properties-> E.g. Model properties).



1. Select **Dimensions** from the property **Type** in the **Basic Information** page of the **Property wizard,**
2. Click **Finish**.

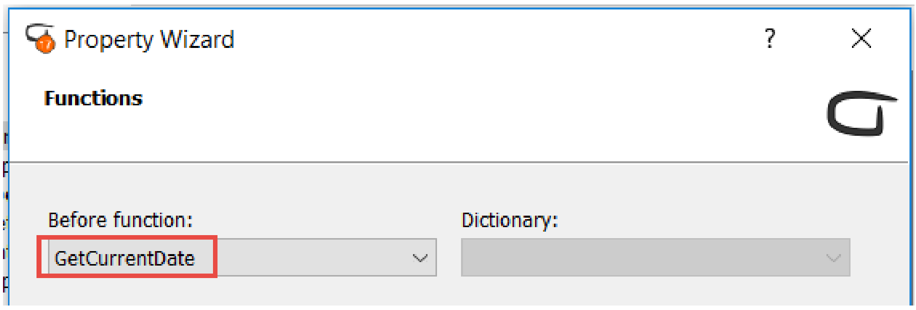
Insert a date in a property

In the **Property wizard** (Options -> Profile Options -> Properties-> E.g. Model properties).



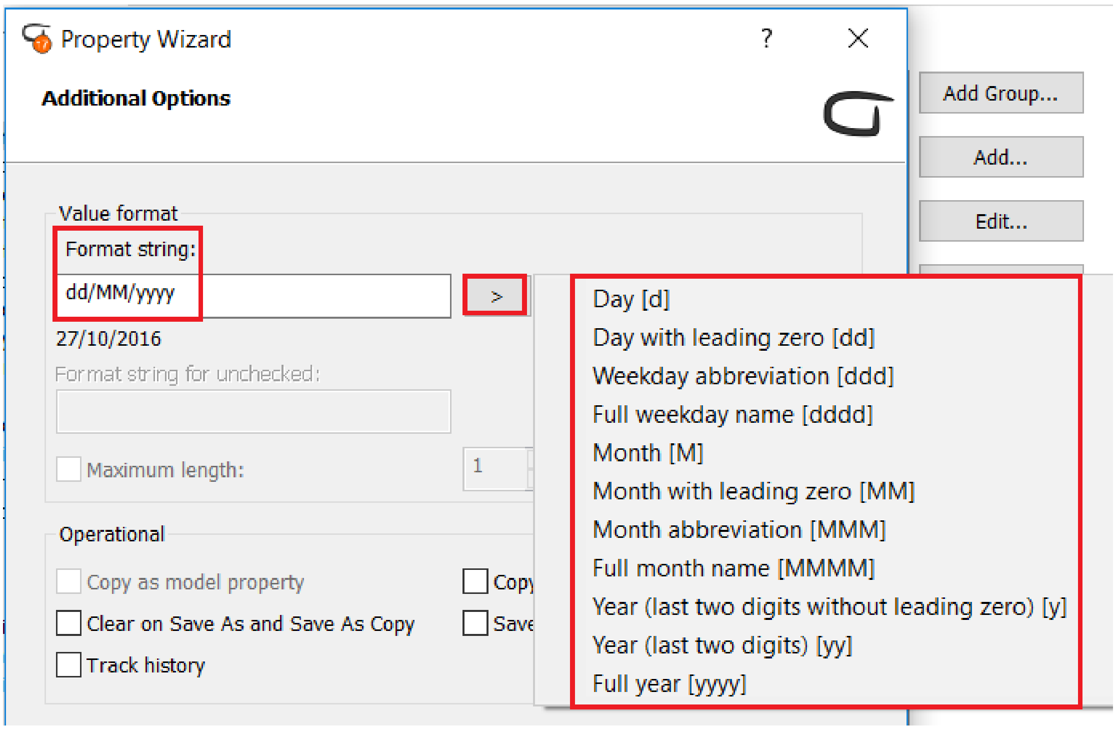
Select **Date** from the property **Type** in the **Basic Information** page of the **Property wizard**, then click **Next**.

The current date can automatically be inserted once the file is created.



From the **Functions** page of the **Property Wizard** select **GetCurrentDate** from the **Before function.**

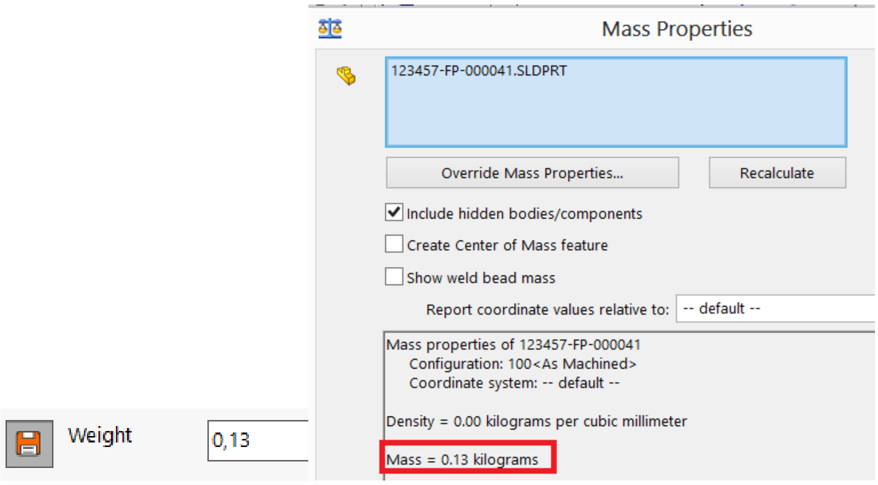
The date format can easily be customized (*e.g. 14.02.2015, 02-14-15,...)*

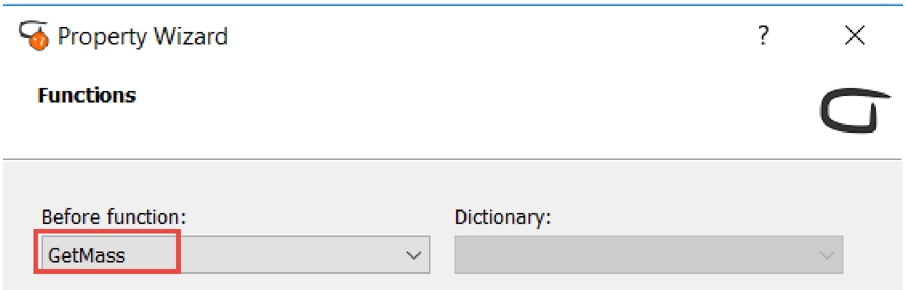


From the **Additional Options** page of the **Property wizard**, click on the **>** key from the **Format string** and select how the date should be formatted.

Insert the mass

The mass of the 3D model can automatically be inserted in the Properties pane.



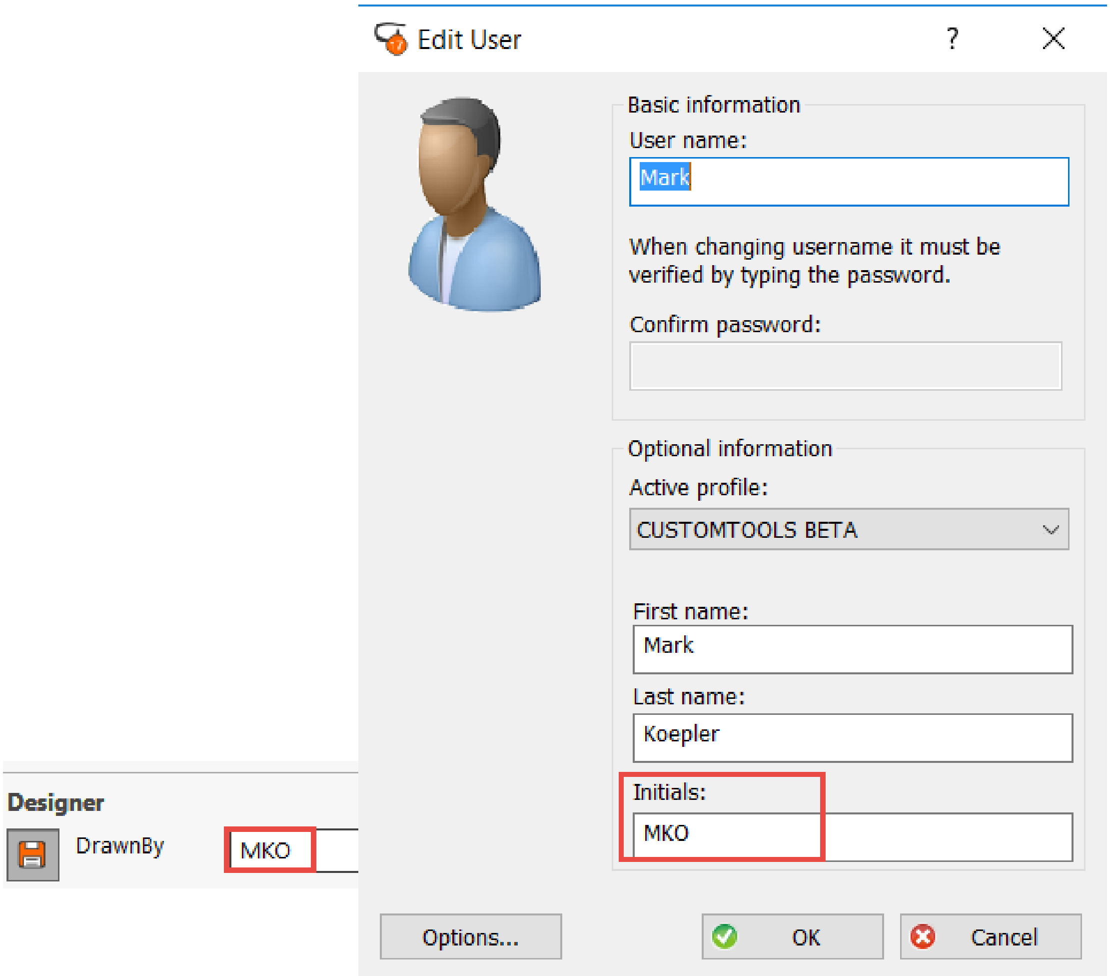


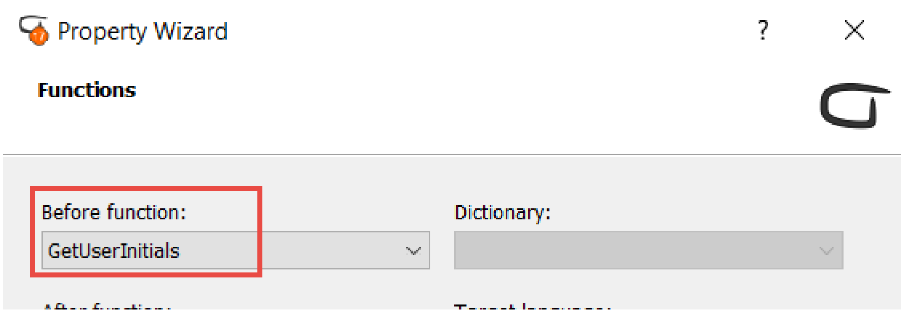
In the **Property wizard** (Options -> Profile Options -> Properties-> E.g. Model properties).

1. Select **Editbox** from the property **Type** in the **Basic Information** page of the **Property wizard**,
2. Then click next, to open the **Functions** page of the **Property wizard**,
3. Then select **GetMass** from the **Before function**.

Retrieve the initials of the CUSTOMTOOLS user

Initials can be defined for every CUSTOMTOOLS users in the **CUSTOMTOOLS Administration.** The initials can automatically be retrieved in the Custom properties in SOLIDWORKS based on the CUSTOMTOOLS users who is currently logged in.





In the **Property wizard** (Options -> Profile Options -> Properties-> E.g. Model properties).

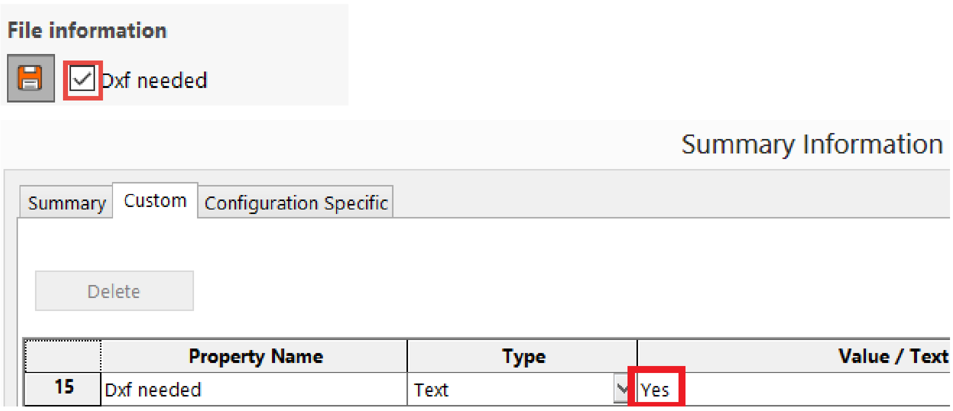
1. Select **Editbox** from the property **Type** in the **Basic Information** page of the **Property wizard**,

2. Then click next, to open the **Functions** page of the **Property wizard**,
3. Then select **GetUserInitials** from the **Before function**.

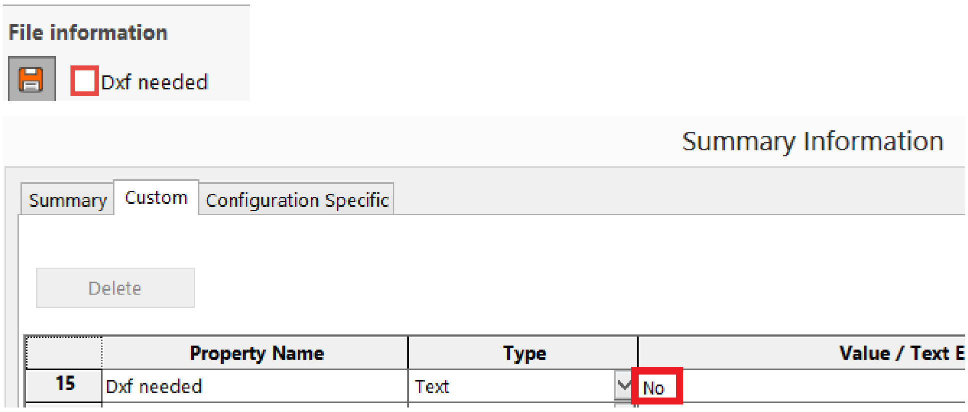
Insert a check box

Property values that will be written to the document property if the Check box is selected or not can be defined.

In the picture below the property called *Dxf needed* is selected, then the value *Yes* is written to the document properties.



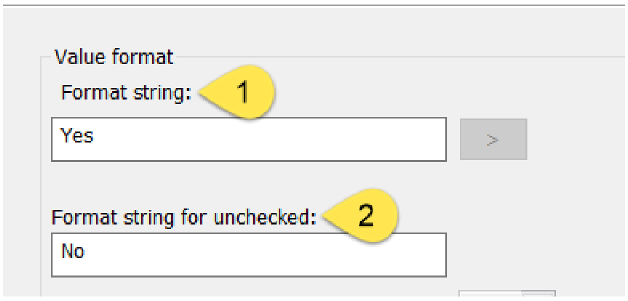
In the picture below the property called *Dxf needed* is not selected, then the value *No* is written to the document properties.



In the **Property wizard** (Options -> Profile Options -> Properties-> E.g. Model properties).

1. Select **Checkbox** from the property **Type** in the **Basic Information** page of the **Property wizard**,
2. Then click next, to open the **Additional Options** page of the **Property wizard**.

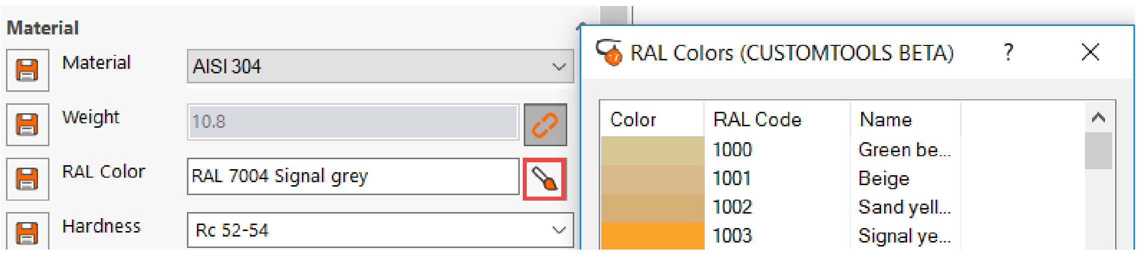
Additional Options



1. From the **Format string**, enter the value that will be written to the document property if the checkbox is selected (e.g. *Yes*).
2. From the **Format string for unchecked**, enter the value that will be written to the document property if the checkbox is not selected (e.g. *No*).

Insert a RAL Color or Color

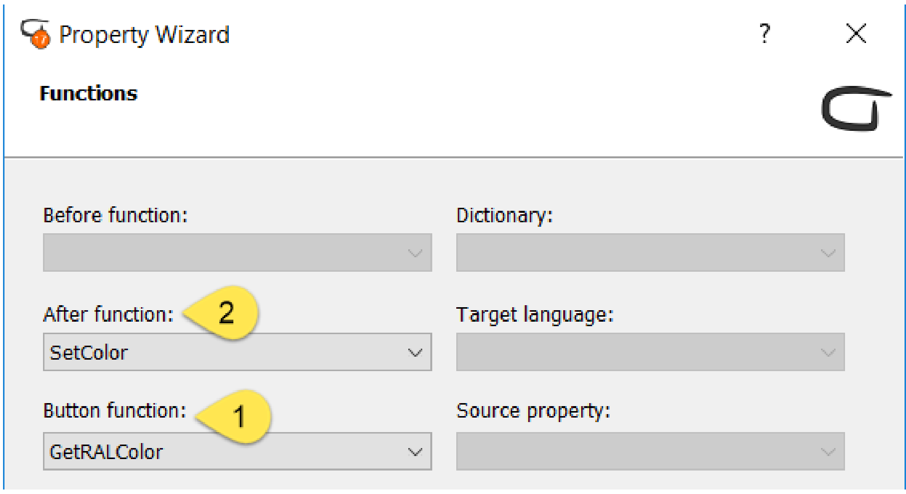
A RAL Color or Color can be selected from a property and apply the selected color to the selected part. The list of available RAL Colors and Colors available can be configured as well.



Create a property that uses a RAL Color or Color

In the **Property wizard** (Options -> Profile Options -> Properties-> E.g. Model properties).

1. Select **Editbox** from the property **Type** in the **Basic Information** page of the **Property wizard**,
2. Then click next, to open the **Functions** page of the **Property wizard**.

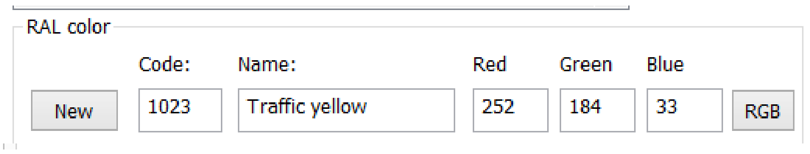


1. From the **Button function**, select **GetRALColor** for RAL Color or **GetColor** for RGB color,
2. From the **After function,** select **GetColor** to apply the selected to the part (3D Model).

Customizing the RAL Color or Color

The list of RAL Colors and Colors that are available from the Properties pane can be customized. New colors can be added or removed.

To add or remove RAL Color or Color open the **Options**, from the **Profile Options** tab, select **RAL Color** or **Color**. Click **New** to add a new color and define the color settings.

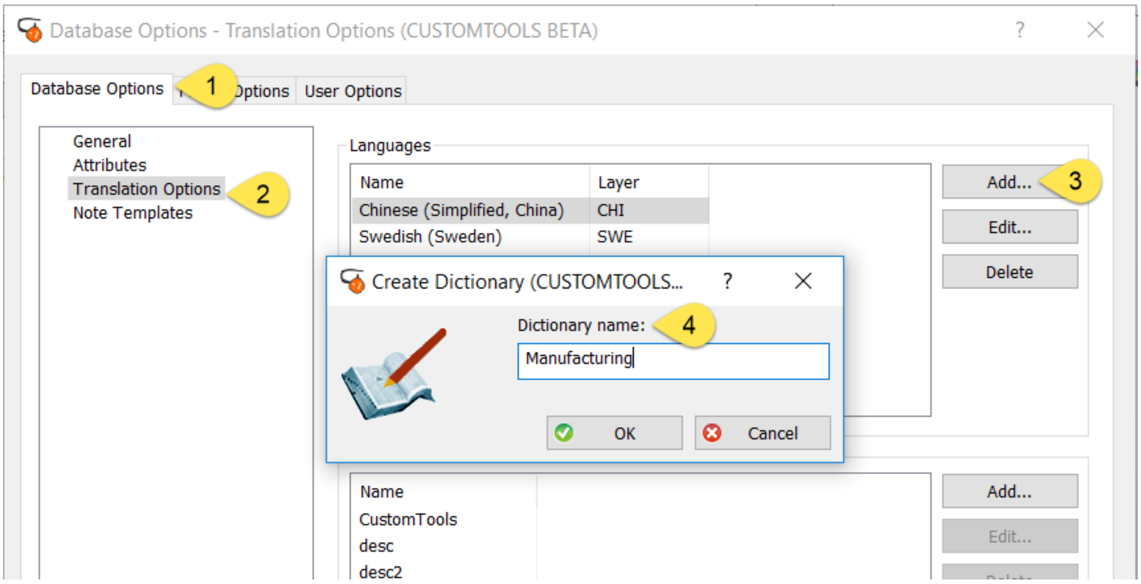


Manage properties via dictionaries

Custom properties can be searched and retrieved from dictionaries. Dictionaries can be used also to translate property values during the printing and conversion of your SOLIDWORKS drawings.

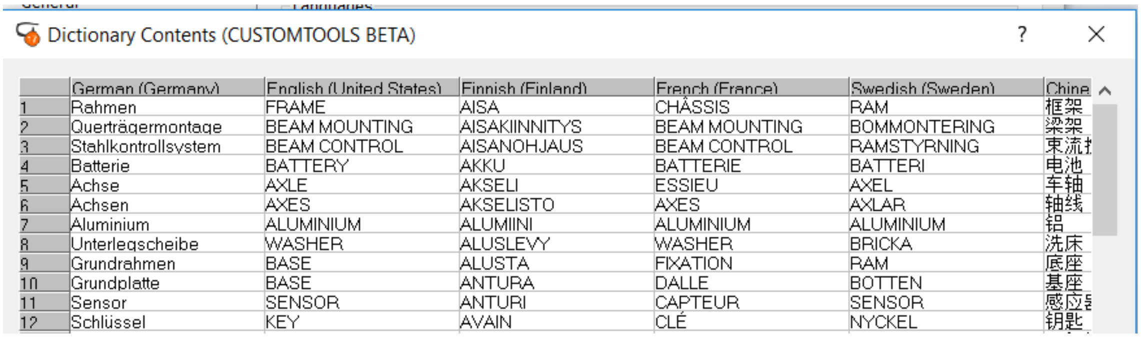
Creating a dictionary

To add a new dictionary, open the CUSTOMTOOLS **Options**,



1. activate the **Database Options** tab,
2. select **Translation Options**,
3. Click **Add** to create a new dictionary,
4. Define the name of the dictionary in the **Dictionary name**.

Once the dictionary has been created, the content can be added or pasted (e.g. from Excel).

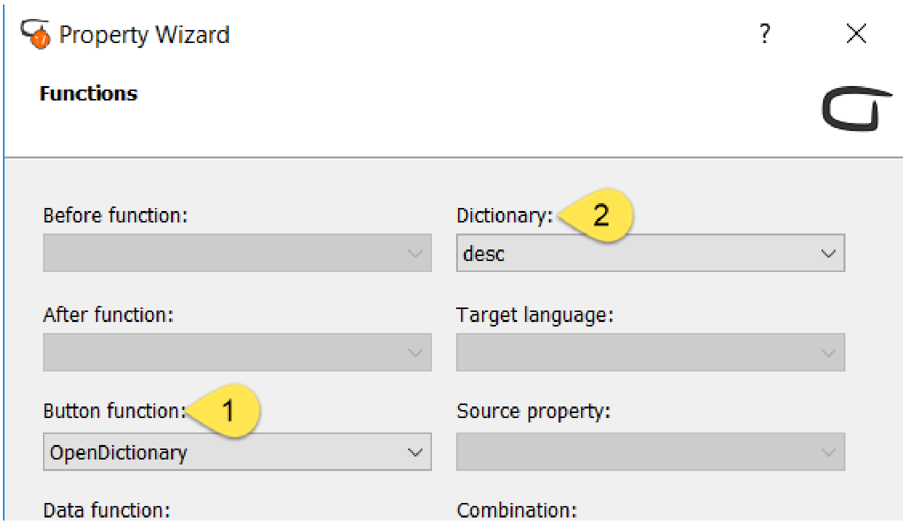


To open the dictionary content, select the dictionary to edit from **Dictionaries** and click on **Contents...** Select the last row and press the **Enter** key to define new rows.

Creating a property that retrieve a property from a dictionary

In the **Property wizard** (Options -> Profile Options -> Properties-> E.g. Model properties).

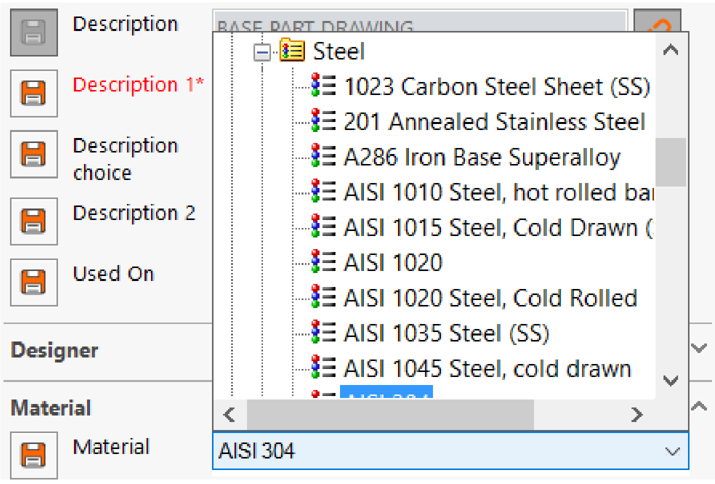
1. Select **Editbox** from the property **Type** in the **Basic Information** page of the **Property wizard**,
2. Then click next, to open the **Functions** page of the **Property wizard**.

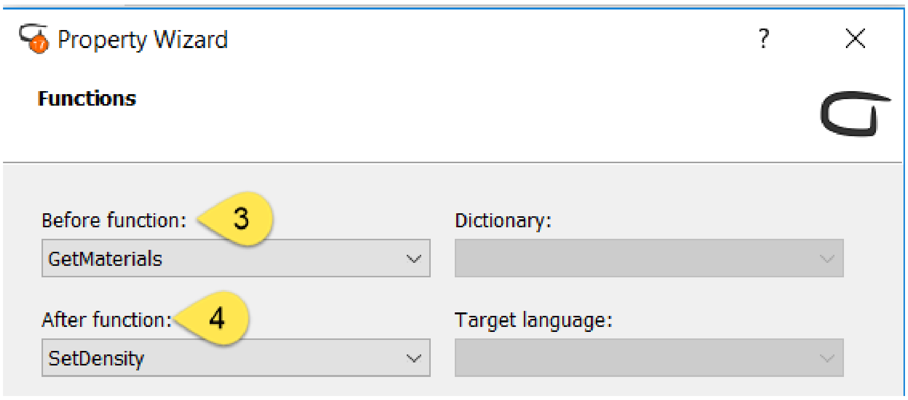


1. From the **Button function**, select **OpenDictionary.**
2. From **Dictionary**, select the dictionary to be opened.

Select a material from the Property

SOLIDWORKS material can be accessed from the Properties pane.



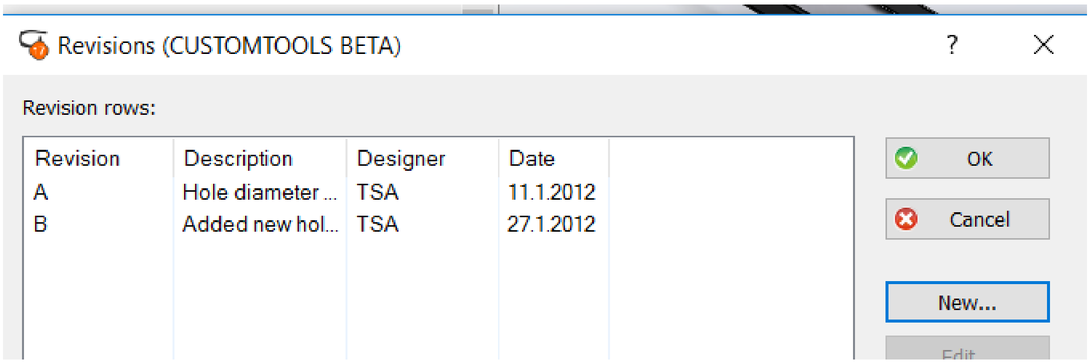


In the **Property wizard** (Options -> Profile Options -> Properties-> E.g. Model properties).

1. Select **Combobox** from the property **Type** in the **Basic Information** page of the **Property wizard**,
2. Then click **Next**, to open the **Functions** page of the **Property wizard**,
3. From the **Before function** select **GetMaterials,**
4. Select **SetDensity** from the After function to calculate the density based on the selected material.

Manage revisions data with CUSTOMTOOLS

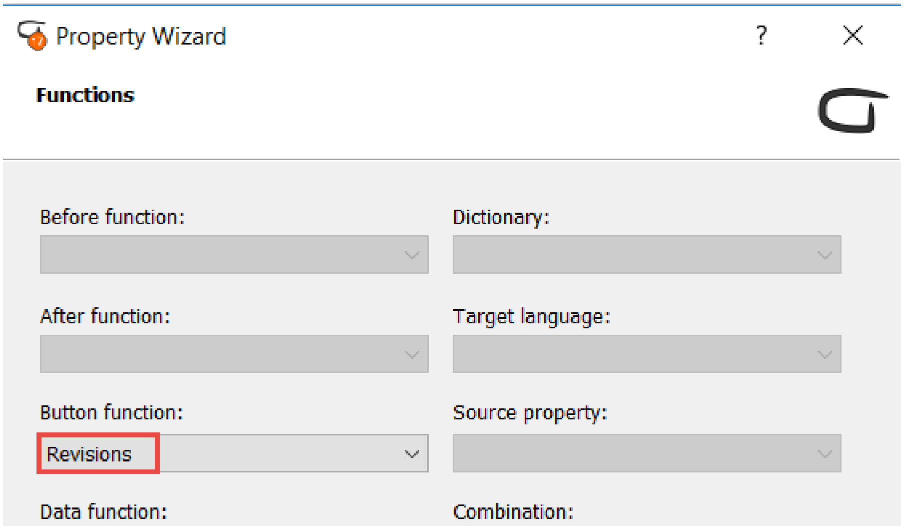
CUSTOMTOOLS can be used to manage the revision data.



Customize the revision table

The custom properties available in the revision table can be customized.

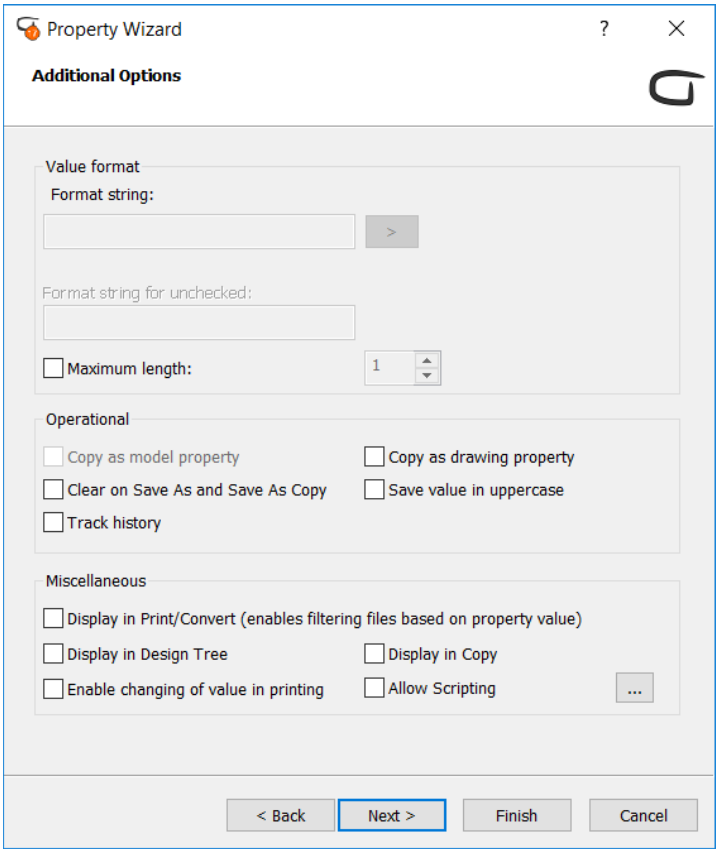
In the **Property wizard** (**Options** -> **Profile Options** -> **Version properties** -> Click **Add** to add a new property to the revision table.



In the **Property wizard** (Options -> Profile Options -> Properties-> E.g. Model properties).

1. Select **Editbox** from the property **Type** in the **Basic Information** page of the **Property wizard**,
2. Then click **Next**, to open the **Functions** page of the **Property wizard**,
3. From the **Button function** select **Revisions.**

Additional Options



Value Format:

- **Format string:** Define how the date will be formatted (*e.g. 14.02.2015*) or a value used if a property that uses a checkbox is selected *(e.g. Yes).*
- **Format string for unchecked**: Define a value used if a property that uses a checkbox type is not selected *(e.g. No)*.
- **Maximum length**: If the checkbox is selected then a number of maximum characters to be used by the property can be defined. *(e.g. 5 characters).*

Operational:

- **Copy as model property**: Allows the user to copy cut list item properties as properties of the model.
- **Copy as drawing property**: Allows the user to copy model properties as properties of the drawing.
- **Clear on Save As or Save As Copy**: The property value will be cleared during certain saving operations.
- **Save value in uppercase**: The property value will be automatically written in capital letters *(e.g. CONVEYOR*).

- **Track history**: Tracks the 10 latest changes in property value.

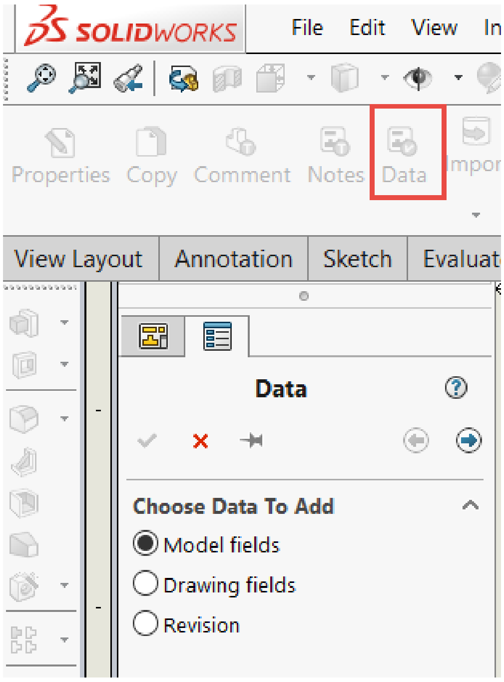
Miscellaneous

- **Display in Print and Convert**: Filter documents to print or convert based on a property value.
- **Display in design tree**: Display the property in the design tree of the CUSTOMTOOLS Viewer.
- **Display in Copy**: Display the property in the list of documents to copy.
- **Enable changing of value in Printing**: Allow to modify the property value.

Customizing your drawing template with your Custom properties

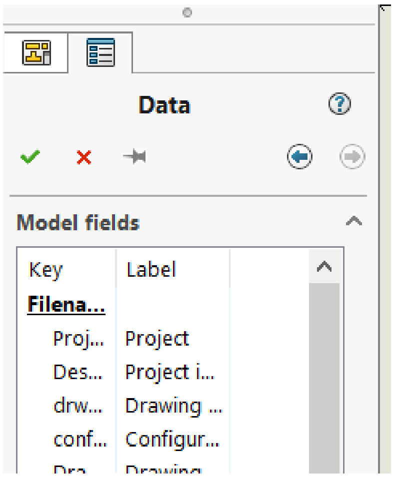
Drawings or drawing templates can easily be customized with the custom properties defined in CUSTOMTOOLS.

Open a drawing or drawing template in SOLIDWORKS, then click **Data** from the CUSTOMTOOLS menu.

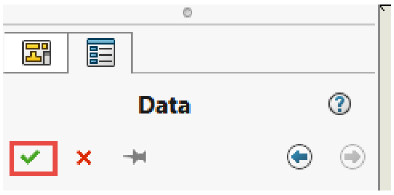


The **Data** pane opens. CUSTOMTOOLS offers three choices to insert data in your drawing:

- **Model fields**: Insert properties defined for the model (Part & Assembly).
- **Drawing fields**: Insert properties defined for the drawing.
- **Revisions**: Insert the revision table.



Select a property to insert in the drawing, move the mouse focus to the drawing and press Enter or click on **OK**.



CUSTOMTOOLS

Chapter 9: Searching documents with CUSTOMTOOLS

Introduction

What does it do?

The CUSTOMTOOLS search is a powerful search engine that allows users to search for SOLIDWORKS files within the CUSTOMTOOLS database.

How does it work?

The user can use different search criteria to search for SOLIDWORKS documents. CUSTOMTOOLS offers a free search, or allows the user to search files based on the type of SOLIDWORKS files, CUSTOMTOOLS project or property value. The user is able to define which custom properties are available in the CUSTOMTOOLS search pane.

How can you use it?

The CUSTOMTOOLS search can be accessed from the SOLIDWORKS task pane. Multiple search criteria can be combined together to run advanced searches. The SOLIDWORKS files can be dragged and dropped from the search results into the 3D Model area.

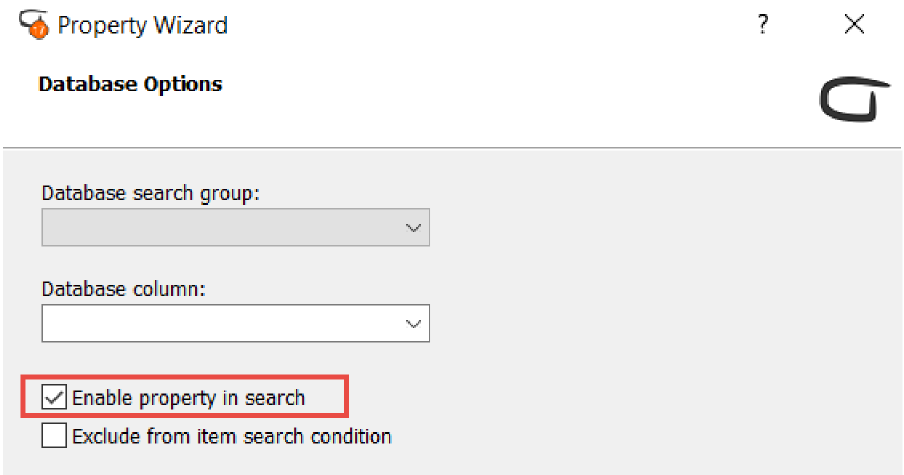
Searching SOLIDWORKS files that were designed before CUSTOMTOOLS

SOLIDWORKS Documents that were designed before CUSTOMTOOLS can also be searched if they have been imported into the CUSTOMTOOLS database. To import the references of SOLIDWORKS files that were designed prior to using CUSTOMTOOLS, please use the CUSTOMTOOLS **Import** functionality (*Check Chapter for more information).*

CUSTOMTOOLS

Adding a property to the CUSTOMTOOLS search pane

The CUSTOMTOOLS search can easily be customized with the Custom properties defined in CUSTOMTOOLS.

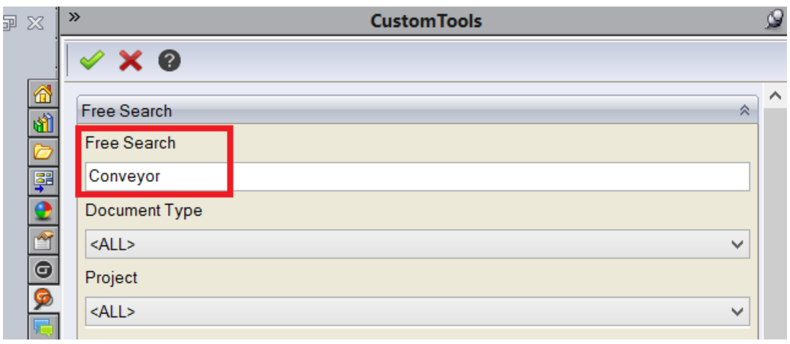


To add a property to the **CUSTOMTOOLS search pane**,

1. Open the CUSTOMTOOLS **Options,**
2. From the **Profile Options** tab, select **Custom properties**, then select **Model properties** or **Drawing properties** from the tree view,
3. Select the property that should be added to the **CUSTOMTOOLS search pane** and click **Edit**,
4. From the **Property wizard** click **Next** twice to open the **Database options** page,
5. Select the **Enable property in search** check box and click **Finish.**

Using the CUSTOMTOOLS search

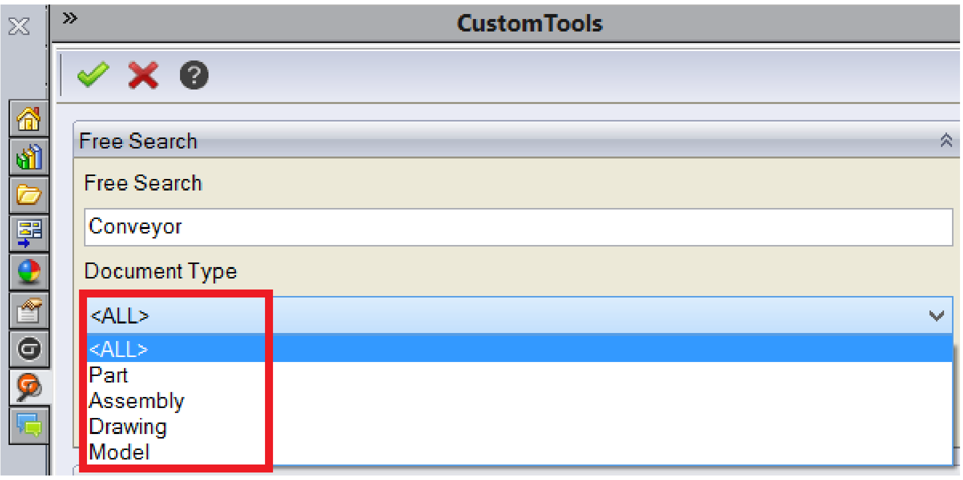
Using the free search



The free search can be used to search documents based on any defined Custom properties.

NOTE: It is not possible to combine property values used in different custom properties in the free search.

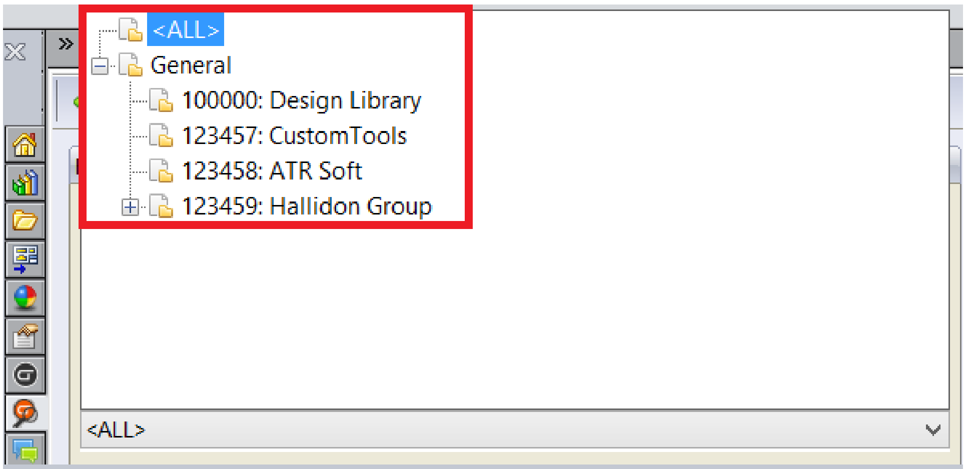
Searching for specific SOLIDWORKS document type



Documents can be searched based on their types. The following types of documents are available:

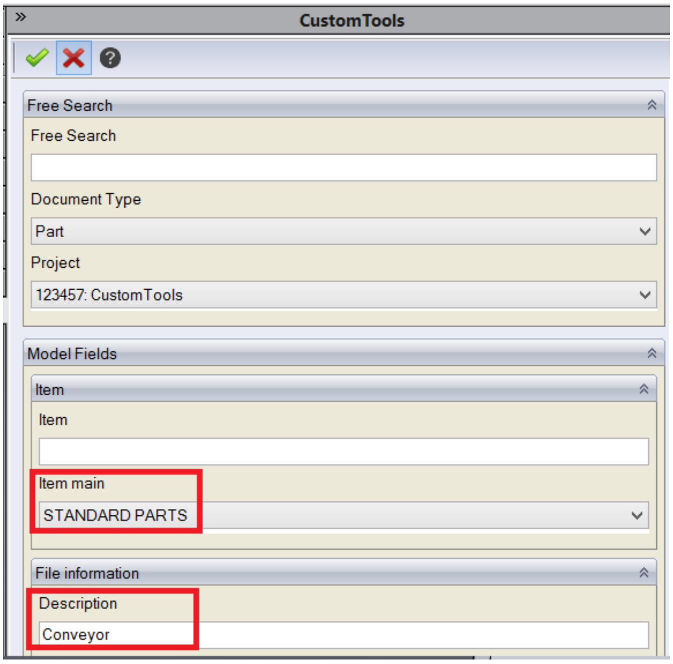
- Part
- Assembly
- Drawing
- Model (Part & Assembly)

Searching files based on the CUSTOMTOOLS project



Files can be searches within a specific CUSTOMTOOLS project.

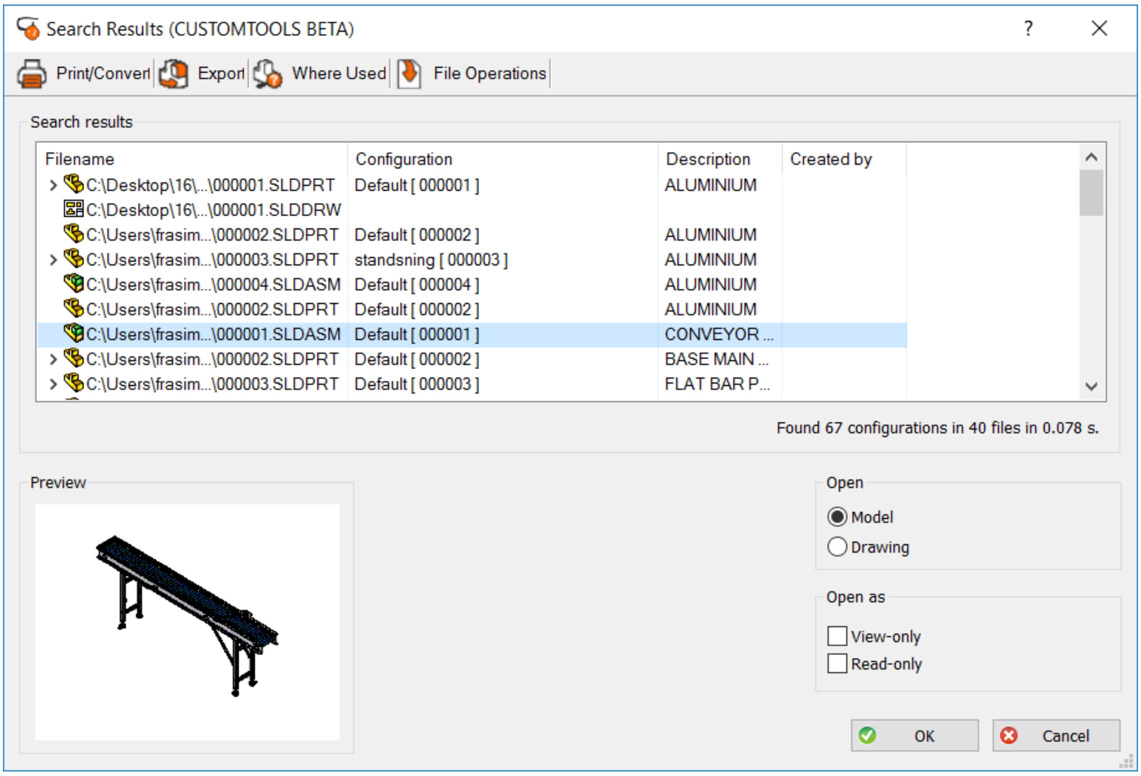
Searching based on a property value



Files can be searches based on property values.

NOTE: Multiple Property values can be combined into the search.

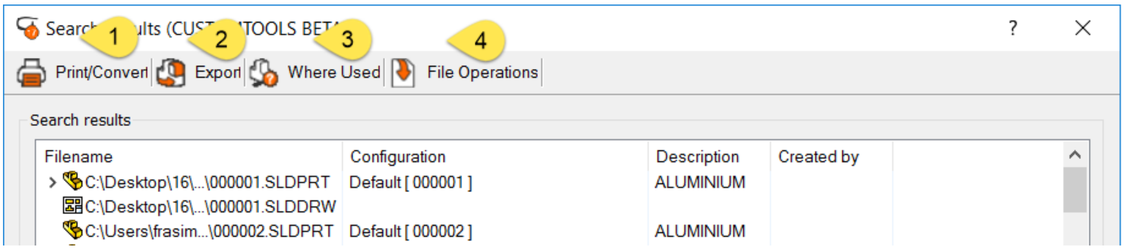
Inserting parts or assemblies from the search result into the active model



- **To insert the selected component from the Search Results into the active 3D Model:**
  Select a part(s) or assembly(ies) and drag it to the 3D Model area.

- **Open the selected component from the Search Results in SOLIDWORKS:**
  Select the file(s) to be open and click **OK.**

Launching other CUSTOMTOOLS functionalities from the search results



The following functionalities of CUSTOMTOOLS can be accessed from the CUSTOMTOOLS search:

1. **Print And Convert**: Opens the **Batch Operation** dialog with the selected assembly and all referred documents or the selected files from the search results.
2. **Export**: Opens the **Export** dialog (*e.g. to generate an Excel report*).

TIP: The assembly to be printed, converted or exported to Excel does not need to be opened in SOLIDWORKS.

3. **WhereUsed**: Locate in which assembly a part/assembly is being used
4. **File Operations**:
    a. **Delete files**: Delete the files from the CUSTOMTOOLS database and/or from the hard drive.
    b. **Open File in SW:** Opens the selected files in SOLIDWORKS.
    c. **Open Drawings:** Opens the drawing(s) of the selected files. If the model has more than one drawing then the **OpenDrawings** dialog opens where all the drawings are listed.
    d. **Open in Explorer:** Locates the selected file in Windows Explorer.

CUSTOMTOOLS

Chapter 10: Excel reporting

Introduction

What does it do?

CUSTOMTOOLS offers an out of the box report template to generate BOM report in Excel. The out of the box report contains the BOM Structure, metadata retrieved from SOLIDWORKS and Property values defined in CUSTOMTOOLS, as well as a preview image that can also be embedded in the report.

Those Excel reports can also be configured and customized based on specific needs and requirements. For more information regarding the customized Excel report, please contact your reseller or ATR Soft at info@CUSTOMTOOLS.info.

How does it work?

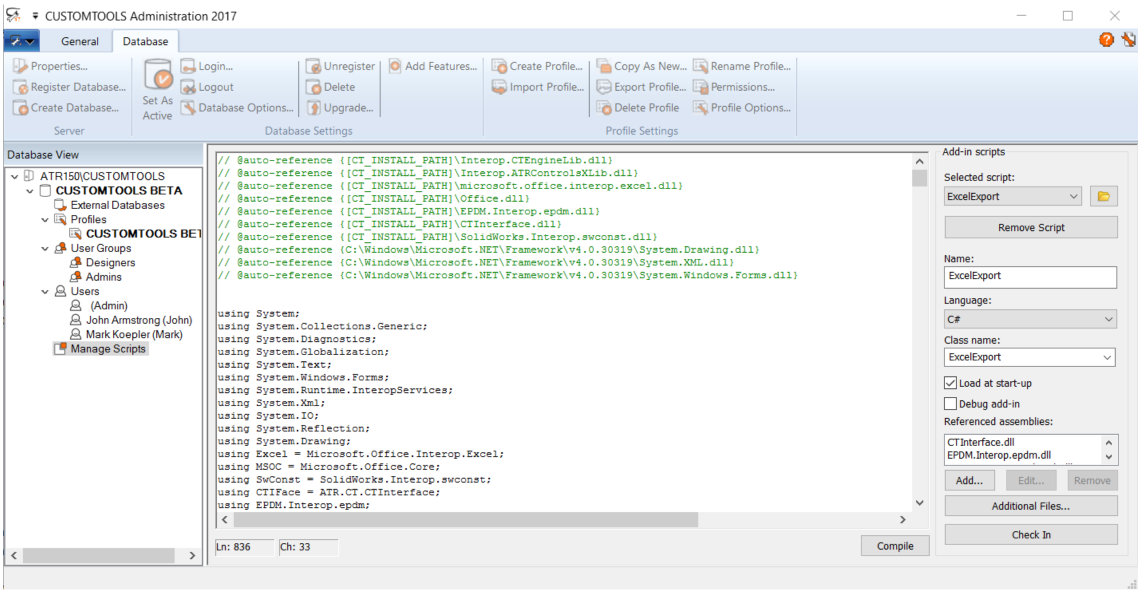
The Excel report is managed via profiles. The out of the box Excel report can easily be configured with SOLIDWORKS properties (Filename, quantity, configuration…) or custom properties.

How can you use it?

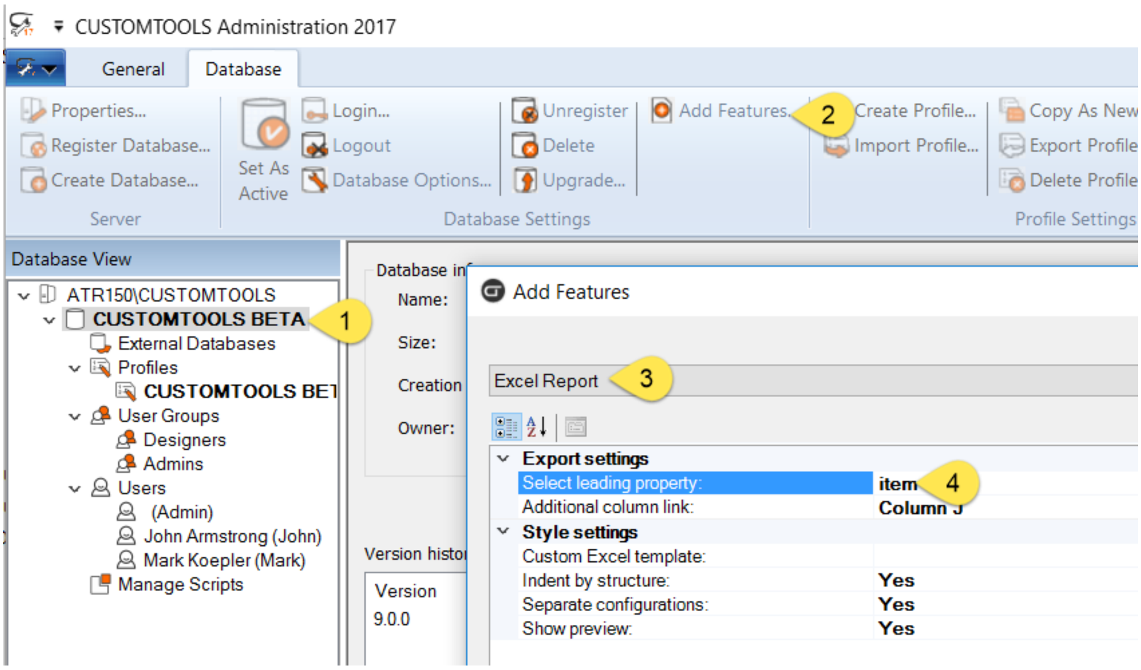
If an assembly is opened, then click on the Export icon. The **Export** dialog opens, then select **Excel profile**. The SOLIDWORKS assembly from which a report is to be generated, does not have to be opened in SOLIDWORKS. The Export dialog can also be accessed from the CUSTOMTOOLS search results.

Adding the Excel script

The out of the box Excel report comes automatically with the Mechanical engineer profile. If the Excel report is not available in the CUSTOMTOOLS profile, then it needs to be added.



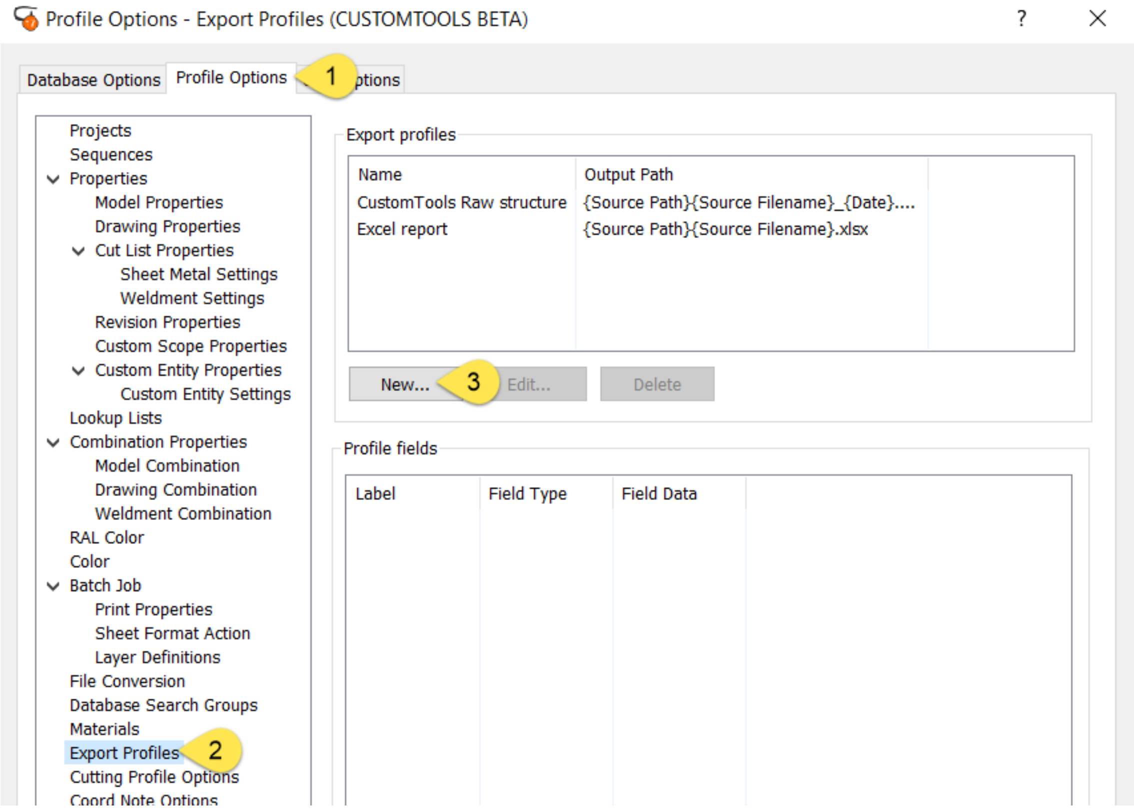
To add the Excel report feature, open the **CUSTOMTOOLS Administration tool,**



1. Select your CUSTOMTOOLS database (e.g. CUSTOMTOOLS Demo),
2. Click on **Add feature...** to open the **Add Features** dialog,
3. Select **Excel Report** from the drop down menu,
4. From the **Select leading property** under **Export settings**, select a custom property that will be used as the first column of the Excel report,
5. Then click **Create Feature** to add the Excel report feature to the active profile.

Adding the Excel report profile

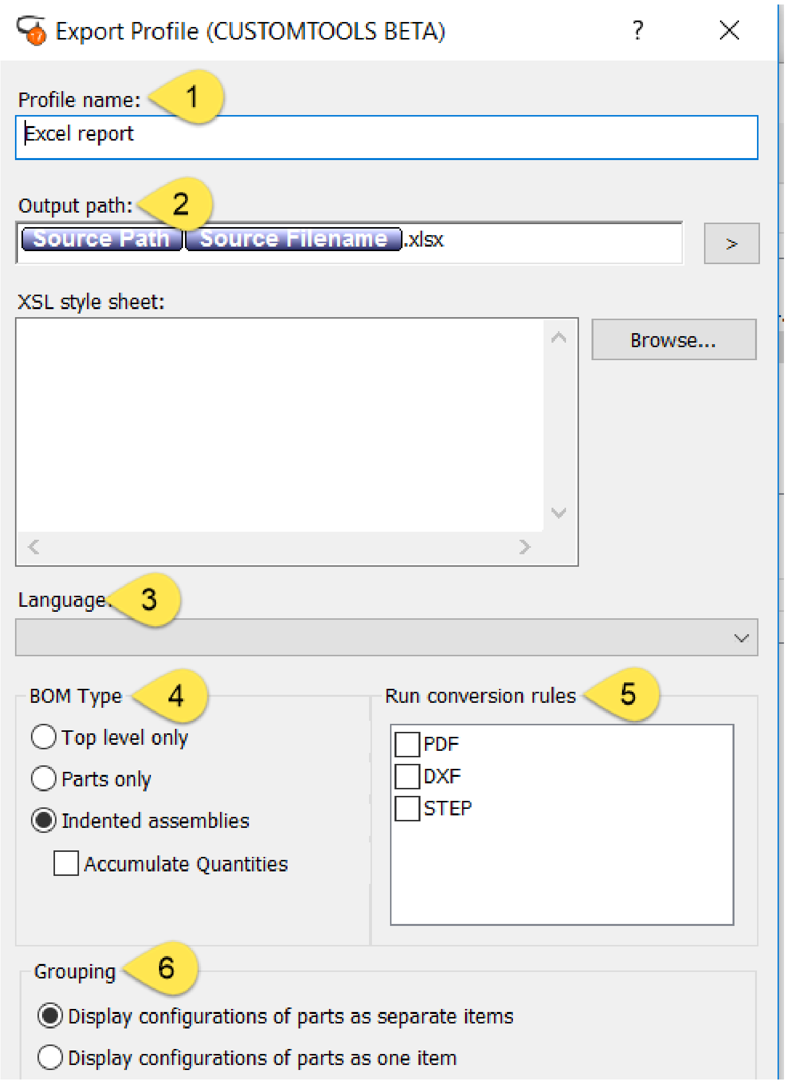
Once the Excel report script has been added in the Administration Tool, an export profile needs to be created in the CUSTOMTOOLS Profile.



To add the Excel report profile, open the CUSTOMTOOLS **Options**,

1. From the **Profile Options** tab,
2. Select **Export Profiles**,
3. Click **New** from **Export Profiles.**

The Export profiles dialog appears.



1. **Profile name:** Defines the name of the export profile as it will appear in the Export dialog.

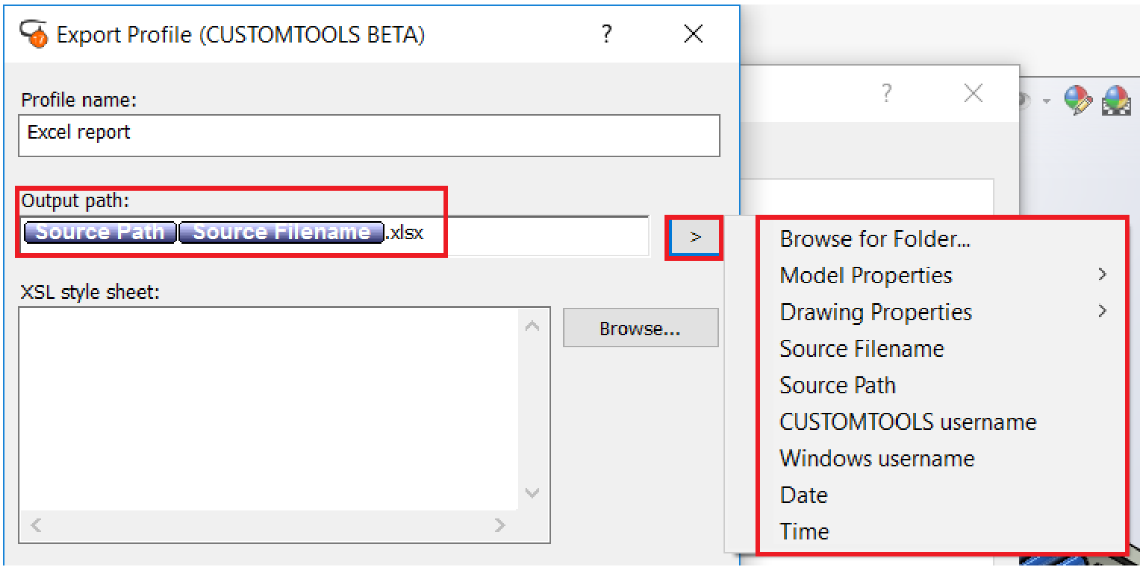
NOTE: In order to use the Excel script, the name of the export profile must start with Excel (e.g. Excel report).

2. **Output path**: Defines the destination and saving rule used to save the Excel report.
3. **Language**: Select the out put language to tranlate properties rertrieved from dictionnaries.
4. **BOM Type**: Defines how parts and assemblies are listed in the Excel report:
    a. **Top level only:** List parts and subassemblies, but not subassembly components.
    b. **Parts only:** Does not list subassemblies. Lists subassembly components as individual items.
    c. **Intented assemblies:** Lists subassemblies. Indents subassembly components below their subassemblies.

5. **Run conversion rules**: Selects the file conversion rules that are used to convert SOLIDWORKS files along with the Excel report.
6. **Grouping**: Defines how configurations are handled in the Excel report:
    a. **Display configurations of parts as separate items**: If a component has multiple configurations, each configuration is listed in the BOM.
    b. **Display configuratione as parts as one item**: If a component has multiple configurations, the component is listed in only one row in the BOM.

Define the destination and naming rules used by the Excel report

In the Export profile dialog, the Output path is used to define the destination folder and rules use to define the file name of the Excel report



Defining the destination path:

Under the **Output path** define the path where the Excel report will be saved. By clicking on the **>** sign, saving options will appear. Select the **Browse for folder** option to specify the destination folder or use the **Source path** to use the same folder as the refering assembly. Folders can also be manually entered and will be created during the export.

Defining the naming rules:

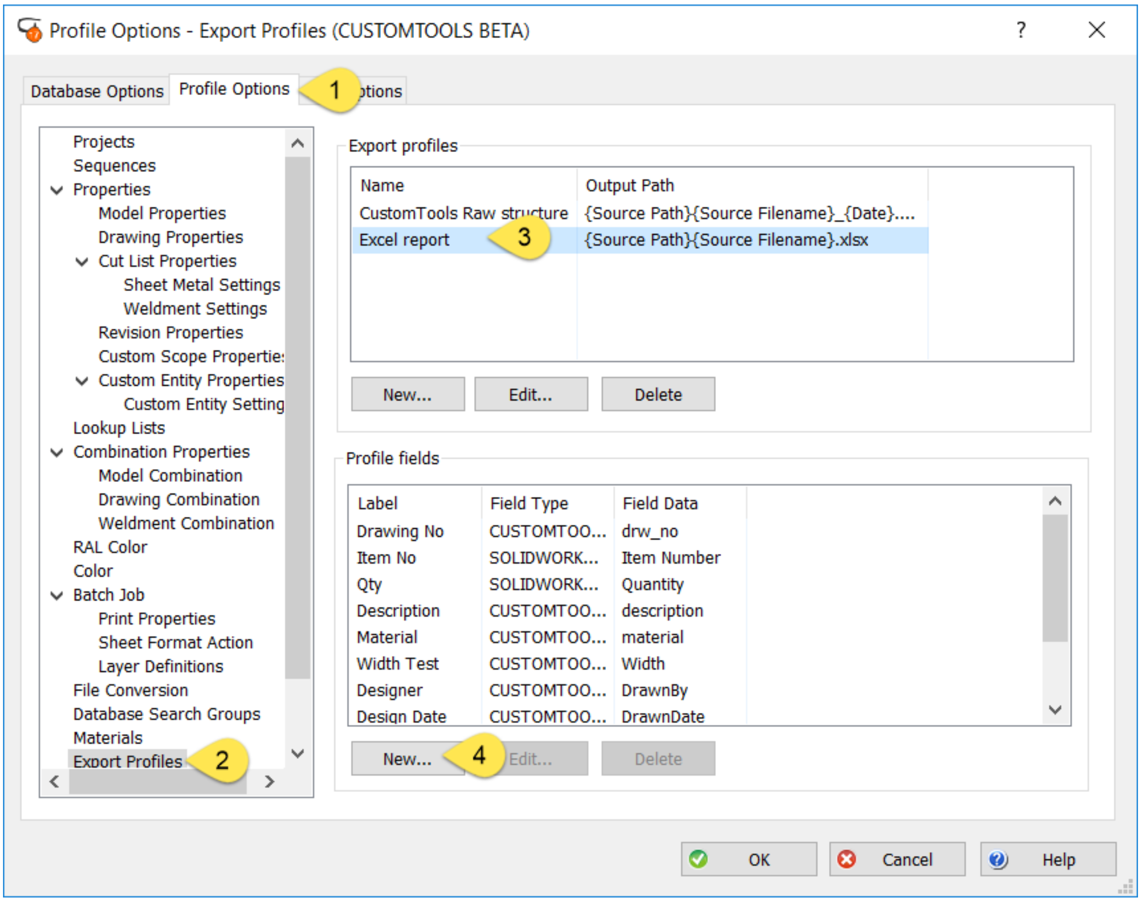
Once the path has been defined, the naming rules can be specified. By clicking on the **>** sign, saving options will appear. Select the **Source filename** to use the same filename as the refering assembly. Custom properties can also be associated with the file name as well as other options.

NOTE: The file extension also needs to be specified in the Output path (*e.g. xlsx*)

Configuring the Excel report

The out of the box Excel report can be configured to add additional columns containing information about your custom properties, SOLIDWORKS property or a preview image.

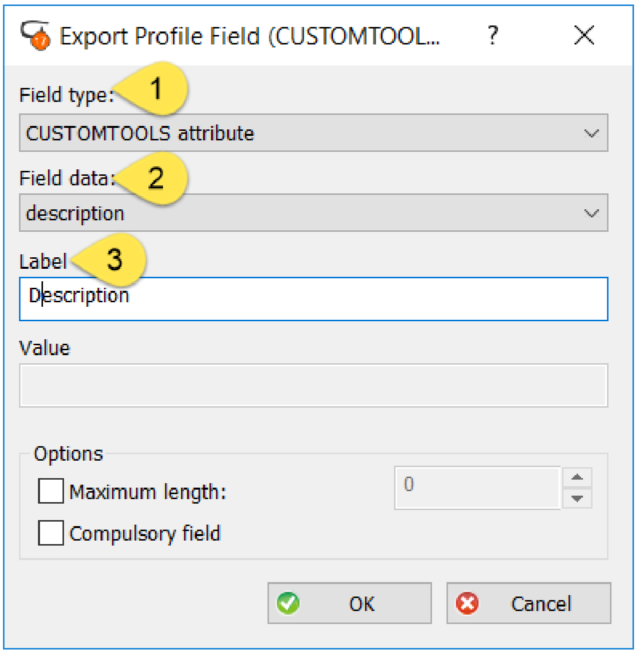
Adding columns to the Excel report



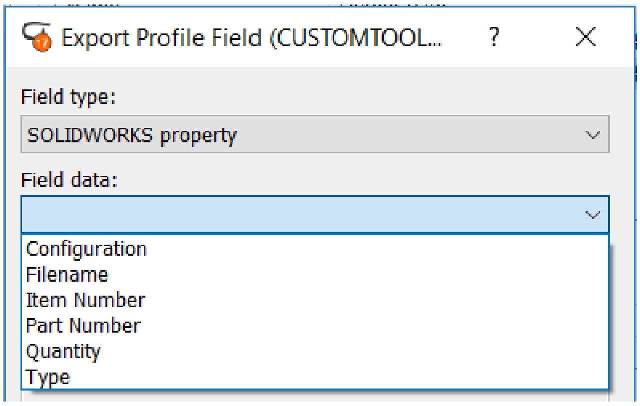
Open the CUSTOMTOOLS **Options** dialog,

1. From the **Profile Options** tab,

2. Select the **Export Profiles** from the tree view,
3. From the **Export profiles**, select the **Excel report**,
4. From the Profile fields, click **New** to open the **Export Profile Field** dialog.



1. **Field type**: Select the source of the field to be added. The following fields are available:
   a. **CUSTOMTOOLS attribute**: Retrieve a property value defined in CUSTOMTOOLS.
   b. **SOLIDWORKS property**: Retrieve a value from a pre-defined list of SOLIDWORKS 'attributes.

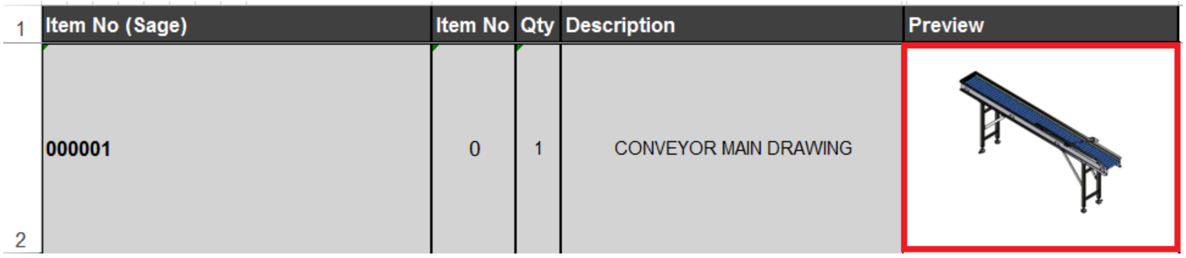


   c. **Value:** Insert a constant value or a preview image.
2. **Field data**: Based on what was selected in the **Field type**, the content will update dynamically.
   a. If a **CUSTOMTOOLS attributes** has been selected, then a property defined in CUSTOMTOOLS can be selected.
   b. If a **SOLIDWORKS property** has been selected, then a pre-defined lists of SOLIDWORKS attributes (*e.g. Item number, quantity…*) can be selected.

NOTE: If **Value** has been selected, then the **Field data** is disabled.

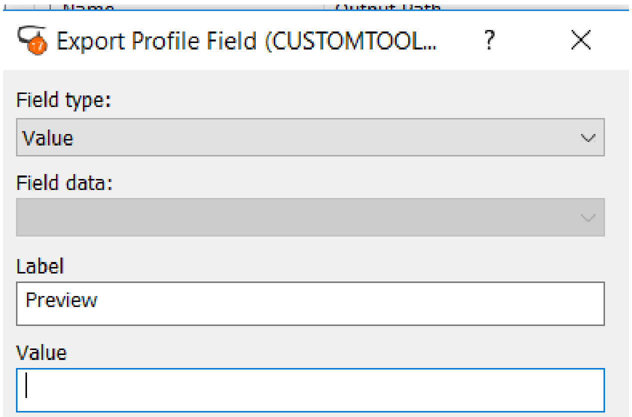
3. **Label**: This corresponds to the name of the column that will be generated in the Excel file. This field is automatically populated based on the value selected in the **Field data**.

Inserting a preview image



Item No (Sage)	Item No	Qty	Description	Preview
000001	0	1	CONVEYOR MAIN DRAWING	

A preview image of the 3D model (Part or Assembly) can be inserted in the Excel report.



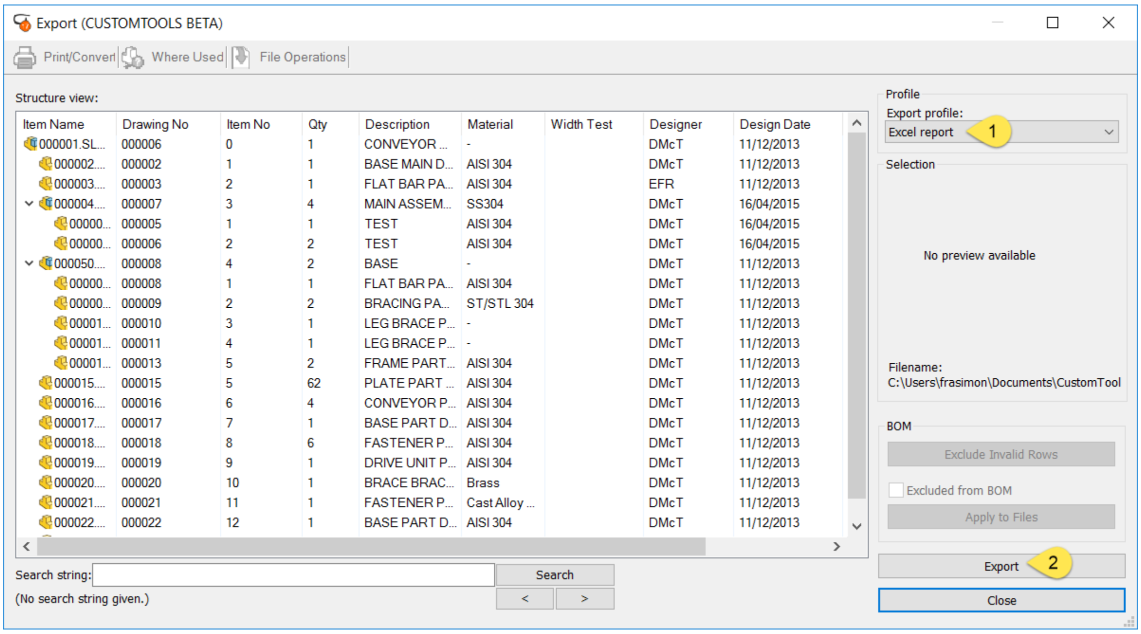
To insert a preview image,

1. Select **Value** from **Field Type**,
2. Type **Preview** in **Label**.

NOTE: The size of the preview image can be modified by editing the Excel script in the Administration Tool and modifying the size of the column where the preview image is used.

Generate an Excel report for your assembly

To generate an Excel report, open the top level assembly in SOLIDWORKS and click **Export**. You can also open the **Export** dialog directly from the CUSTOMTOOLS **Search result** dialog.



From the **Export** dialog,

1. Select the export profile (*e.g. Excel report*),
2. Click on **Export**.

Chapter 11: Automatic file naming

Introduction

What does it do?

Automate the file naming and storing convention used to save SOLIDWORKS documents.

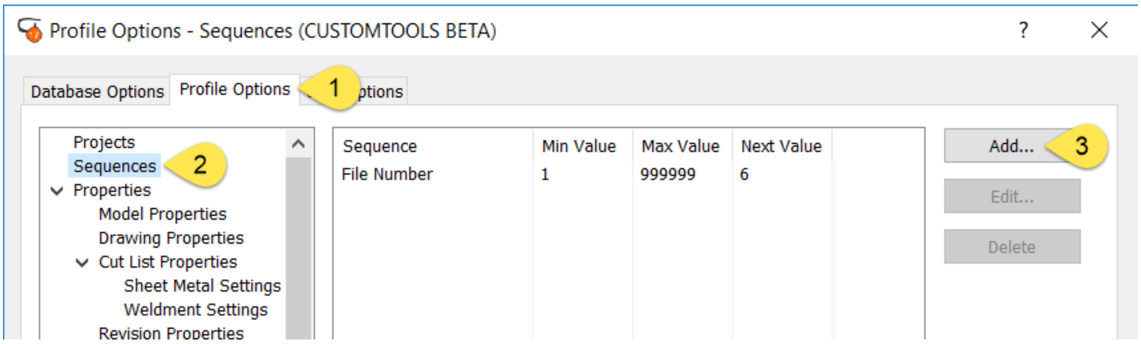
How does it work?

CUSTOMTOOLS uses a property value to generate a file. Multiple custom properties can be combined together to create a new property which can then be used to generate a file name. Also the user is able to generate sequences that are automatically shared between the different CUSTOMTOOLS users.

How can you use it?

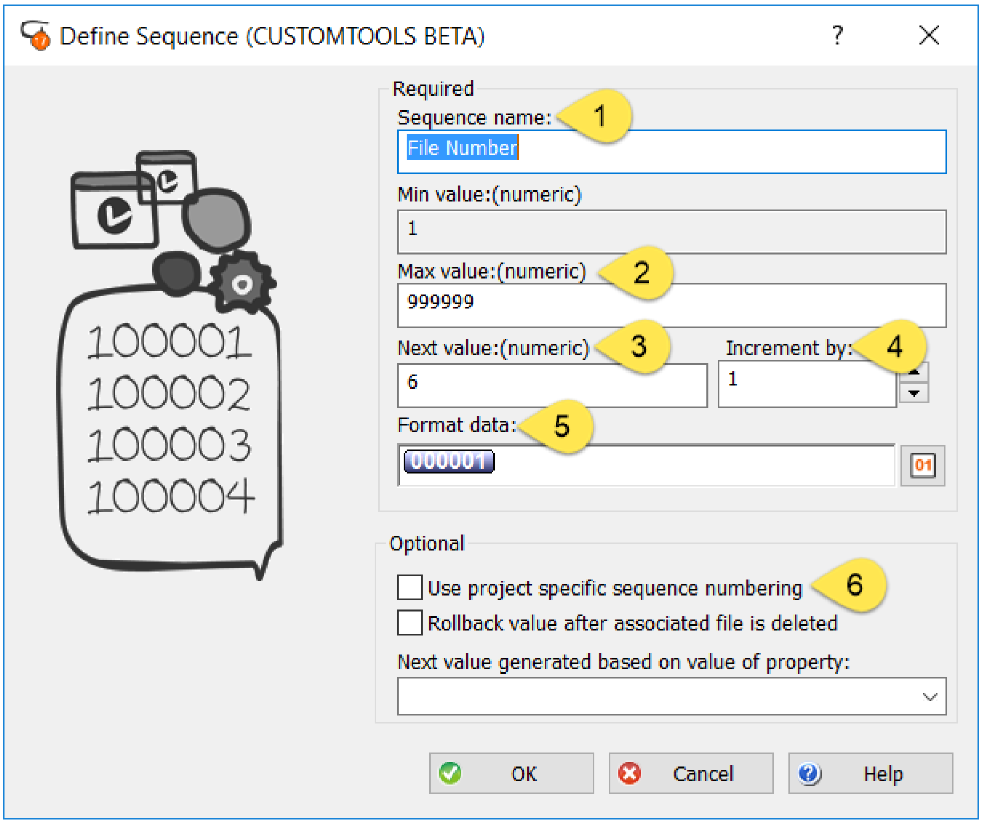
After creating a new model in SOLIDWORKS, open the Properties pane to fill the properties used as file name and save the document.

Creating a sequence

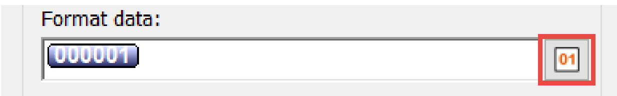


To add a new sequence, open the CUSTOMTOOLS **Options** dialog:

1. From the **Profile Options** tab,
2. Select **Sequences** from the tree view,
3. Click **Add** to open the **Define sequence** dialog.



1. **Sequence name**: Define the name of the sequence as it will appear in the Properties pane.
2. **Max value**: Define the last sequence number that can be generated by the sequence. This is field can also be left empty. Once the sequence reaches the Max value it can be extended.
3. **Next value**: Define the next sequence number that will be generated. This can be used to map the CUSTOMTOOLS sequence to a former sequence.
4. **Increment by**: Define how the numbers generated by the sequence increase (e.g. *Increment by 10 then the sequence is generated as 0, 10, 20, 30…)*.
5. **Format data**: Define how the sequence will be generated. Start/end constant values can be inserted (e.g. *Proto-0001*).

   Click on the **01** button to define how many digits are used by the sequence *(e.g. 001, 0001, 00004,…)*.
6. **Use project specific numbering**: If the CUSTOMTOOLS project functionality is being used, then the sequence can restart from the beginning for each project (e.g. *Project A: 001, 002, 003 – Project B: 001, 002, 003,…)*.

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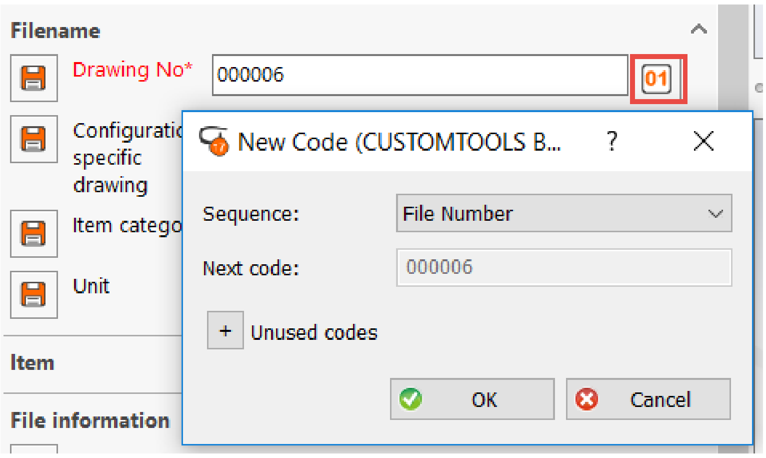
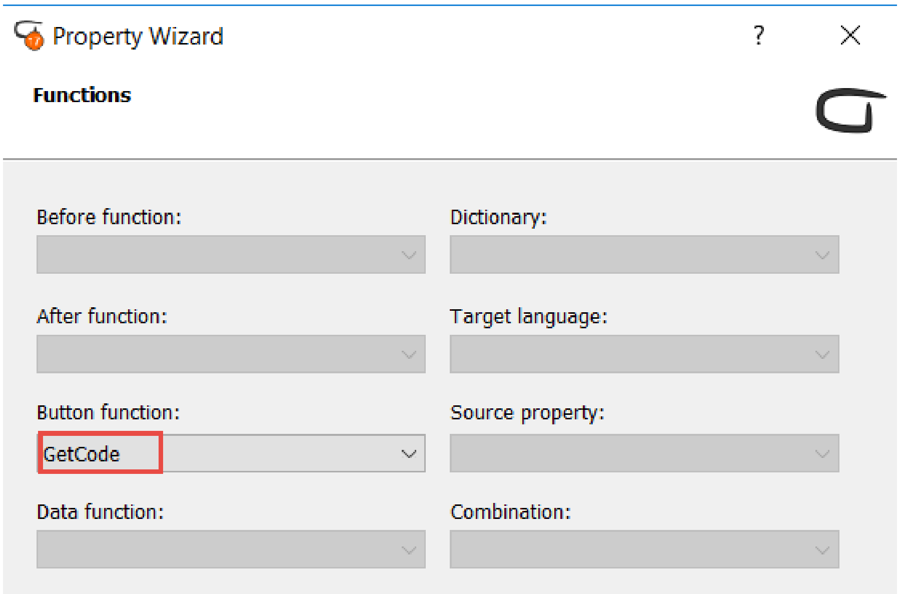
Define a property that generates a sequence code



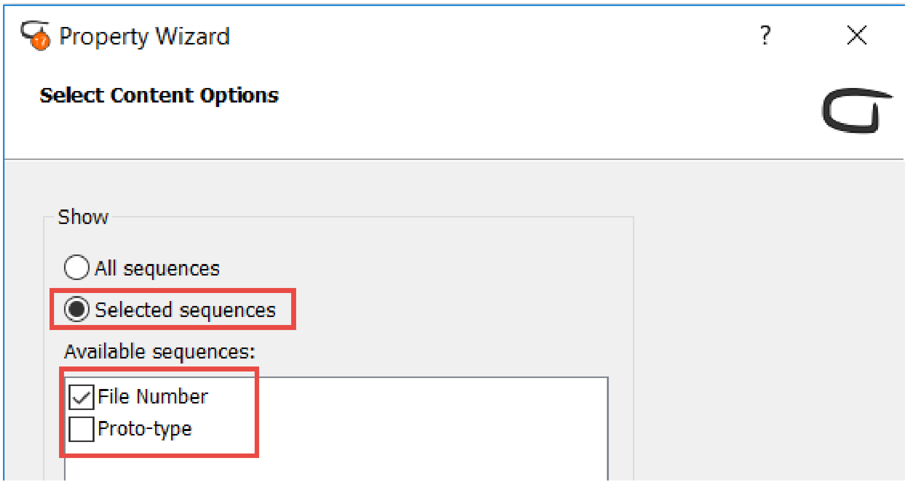
Sequence number are generated from a property defined in CUSTOMTOOLS.



To associate a sequence with a property, open the CUSTOMTOOLS **Options** dialog, from the **Profile Options** tab, select **Model properties/Drawing properties**.

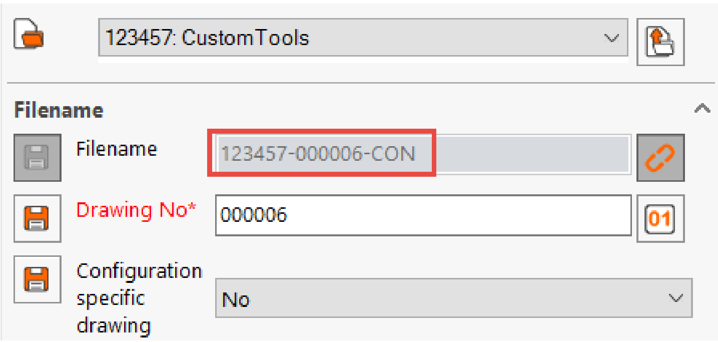
1. Click **Add** to add a new property,
2. From the **Property wizard** dialog, select **Editbox** from the **Type**,
3. Then click **Next** to open the **Functions** page of the **Property wizard**,
4. From the **Button function** box, select **GetCode** and click **Finish**.

CUSTOMTOOLS



NOTE: Specific sequences can be associated with a property. From the **Property wizard** open the **Select Content Options** and select the **Selected Sequences** button to select which sequences are to be used with the property.

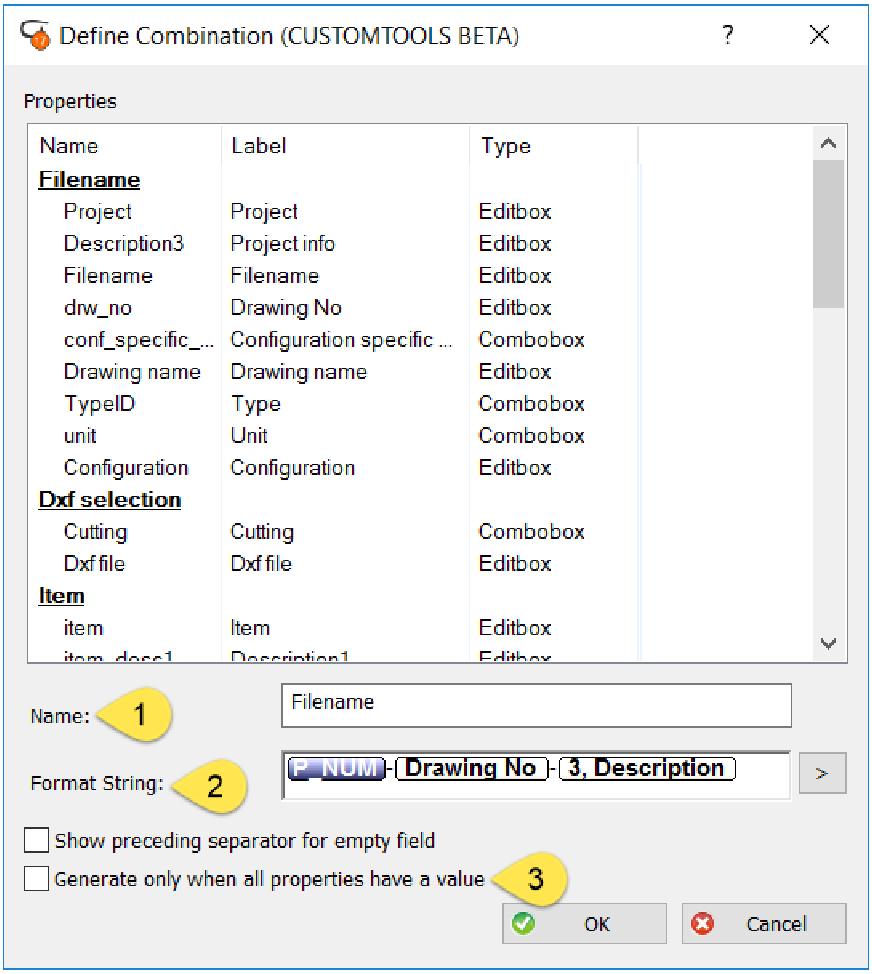
Create a combination of properties to generate a file name



Multiple custom properties can be combined together to create a new property. Combination of properties can be used for example to generate a file name *(e.g. The property File name is used to generate a file name and combine the project number, drawing No and part of the description).*

Create a combination of properties

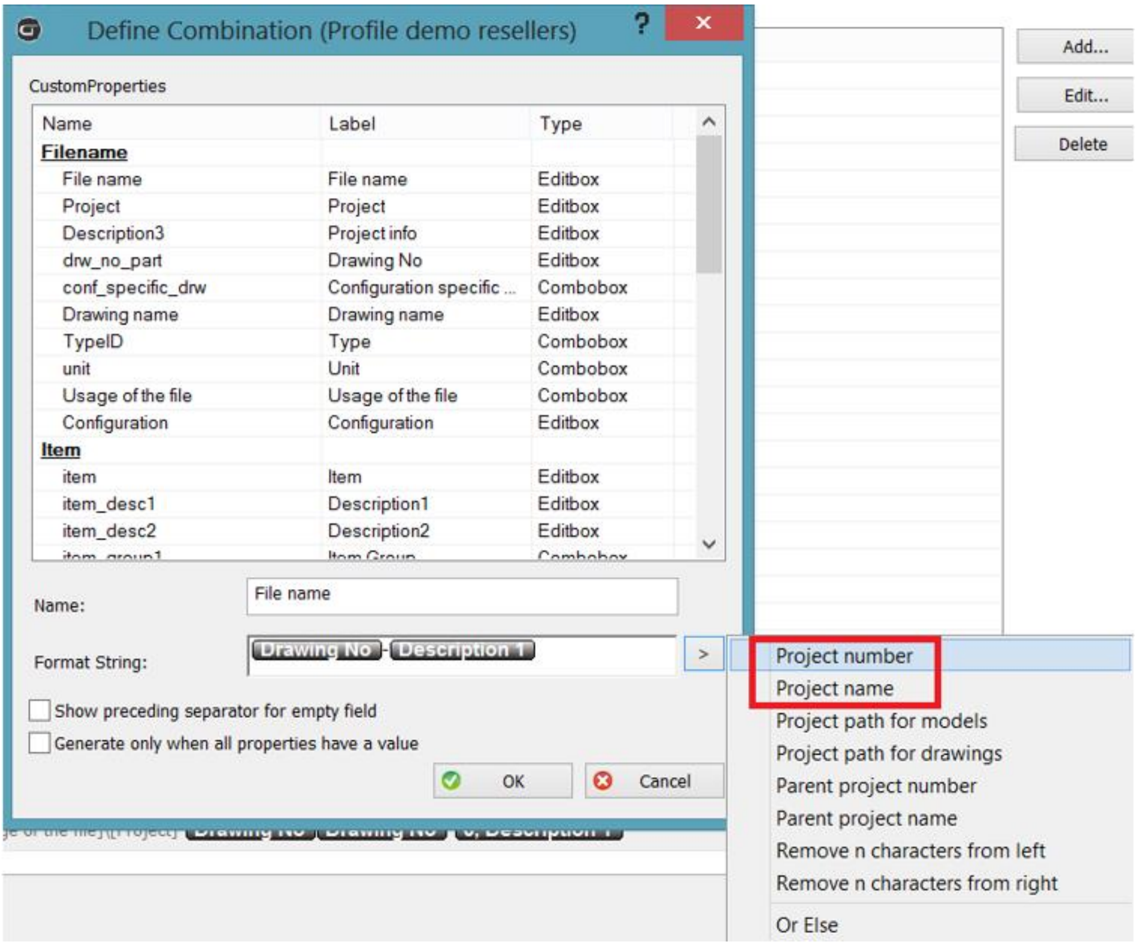
To create a combination of properties, open the CUSTOMTOOLS **Options** dialog, from the **Profile Options** tab, select **Model combination**, under **Combination properties** in the tree view. Click **Add** to open the **Define Combination** dialog.



1. **Name:** Defines the name of combination as it will appear in the **Property wizard**.
2. **Format String**: Defines which properties are to be associated in the combination. To add a property to the **Format String**, select the property and drag & drop it into the **Format String** or double click on the property to add it directly to the **Format String.**
3. **Generate only when all properties have a value:** The combination is generated only if ALL the different custom properties used in the combination have a defined value.

Associate a project Number/name with the combination

If the CUSTOMTOOLS project functionality is used, then the Project name or number can be used into the combination of properties.



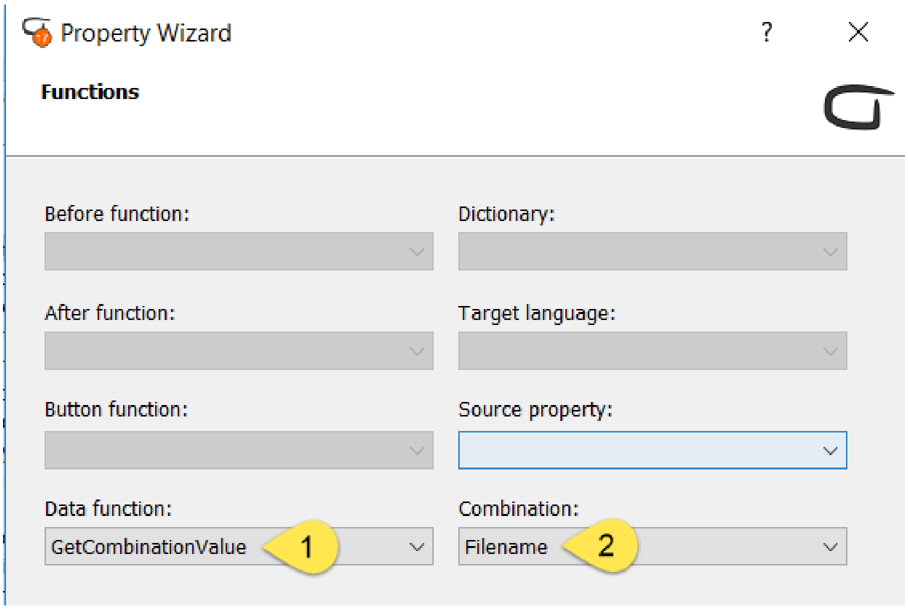
From the **Define Combination** dialog, click on the **>** and select the **Project number** or **Project name** to add it to the **Format String**.

Create a property that uses a combination of properties

Once the combination of properties has been created, it can be associated with a property. From the CUSTOMTOOLS **Option** dialog, select Model properties from the **Profile Options** tab and **Properties**.

1. Click **Add**, to add a new property, the **Property wizard** dialog opens,
2. From the **Basic Information**, select **Editbox** as the **Type** of property,
3. Click Next to open the **Functions.**

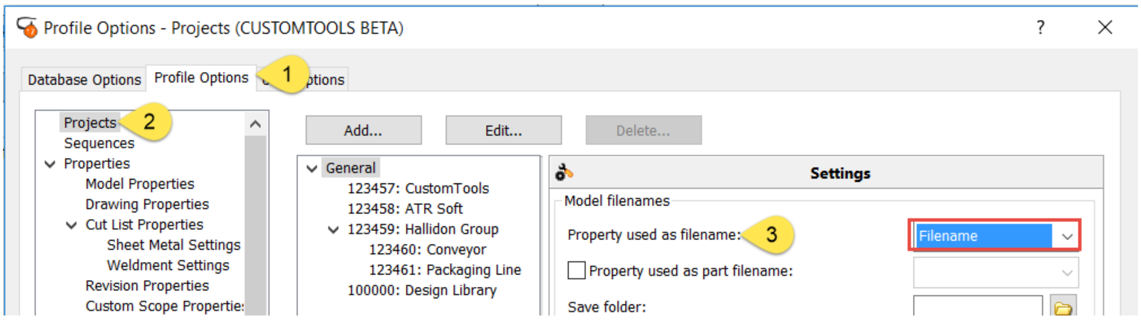
CUSTOMTOOLS



1. Select **GetCombinationValue** from the **Data function**,
2. From **Combination** select the combination of properties to be used.

Select the property to be used as a filename

The same property is used to name parts and assemblies



Open the CUSTOMTOOLS **Options** dialog, from the **Profile Options** tab,

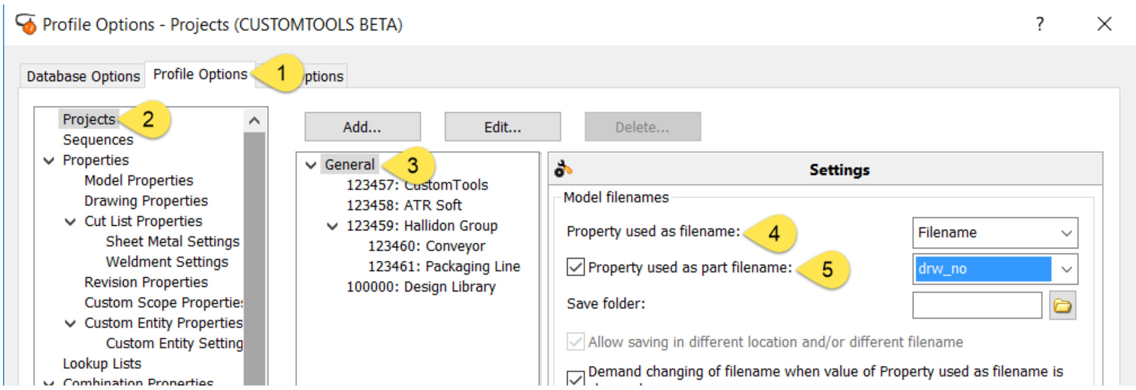
1. Select **Projects** from the tree view,
2. Select the **General** project or other defined projects,
3. From the **Model filenames,** select the property used to generate a file name for assemblies and parts from the **Property used as filename**.

CUSTOMTOOLS

NOTE: If different projects are used then different file naming rules can be defined for each project (e.g. different **Property used as filename** or different **Save folder** can be defined).

Using different property to generate a name for parts and assemblies

If different file naming conventions are used to name parts and assemblies, then different property can be used to generate a filename for parts and assemblies in CUSTOMTOOLS.



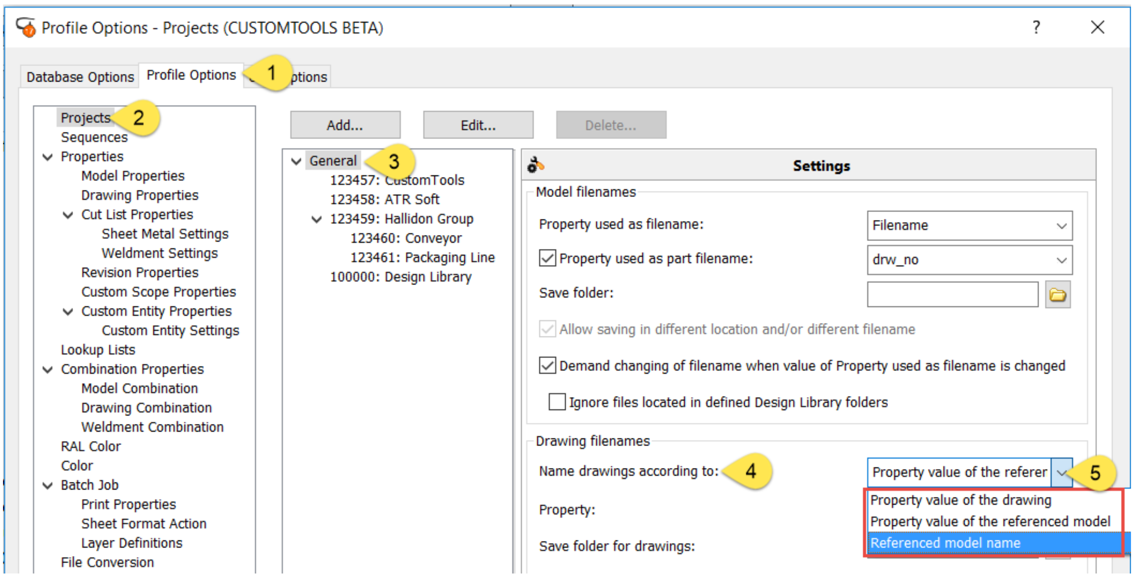
Open the CUSTOMTOOLS **Options** dialog,

1. From the **Profile Options** tab,
2. Select **Projects** from the tree view,
3. Select the **General** project,
4. From the **Model filenames,** select the property used to generate a file for assemblies from the **Property used as filename**,
5. Select the **Property used as part filename** check box and select the property used to generate a name for parts.

Defining file naming rules for your drawings

Different file naming rules can be used to generate a filename for drawings. This can be very useful for users having different names between the referred 3D model and its drawing(s) (e.g. if working with configurations in SOLIDWORKS, then it allows the user to create one drawing per configuration).

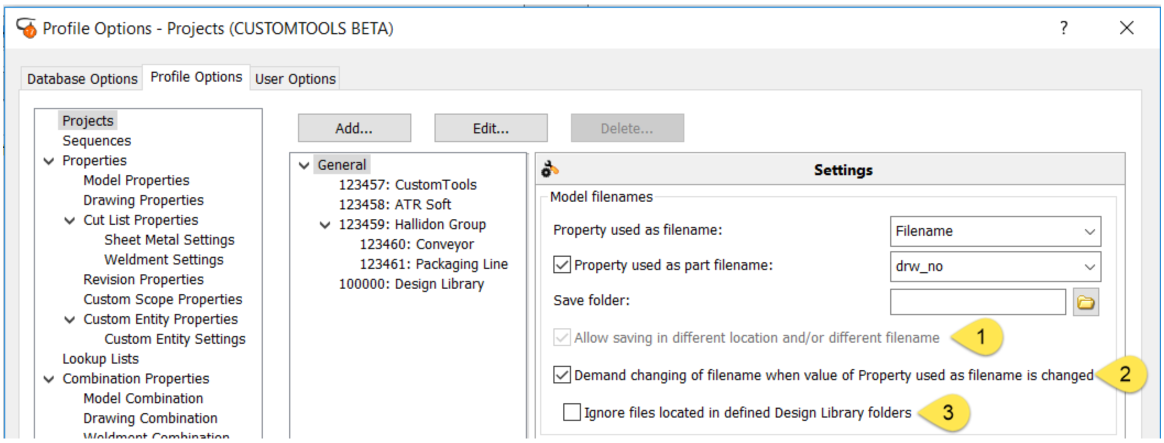
CUSTOMTOOLS



Open the CUSTOMTOOLS **Options** dialog,

1. From the **Profile Options** tab,
2. Select **Projects**,
3. Select the **General** project,
4. From the **Drawing filenames**, select how the drawing filename is generated from the **Name drawings according to** drop down menu. CUSTOMTOOLS offers three possibilities to generate a drawing filename:
   - **Property value of the drawing**: Uses a property value defined in the **Drawing properties.**
   - **Property value of the referenced model**: Uses a property value defined for **the Model properties (Part and Assemblies).**
   - **Referenced model name:** Uses the same filename for the drawing and the model to which it is referring.

Additional file saving options



1. **Allow saving in different location and/or different name**:
    a. If the check box is selected, then SOLIDWORKS Save As dialog will appear while saving a SOLIDWORKS documents for the first time. The user will be able to select a different save folder and modify the file name in the Save As dialog.
    b. If the check box is NOT selected, then the SOLIDWORKS will be saved automatically to the defined folder and name.

    NOTE: The SOLIDWORKS Save As dialog will not appear

2. **Demand changing of filename when value of Property used as filename is changed**: If the value of the property used to generate a file name is modified then CUSTOMTOOLS notifies the users of the change and offers the different saving options to the users (Save as, rename...).

    NOTE: If the 3D model is saved as new file or rename then CUSTOMTOOLS offers the possibility to copy and name drawings accordingly.

3. **Ignore files located in defined Design Library folders**: CUSTOMTOOLS will not offer to rename files that are stored in the Design library folders if the property value used to generate a file name is modified.

    NOTE: The check box will be activated if the **Demand changing of filename when value of Property used as filename is changed** check box is selected.

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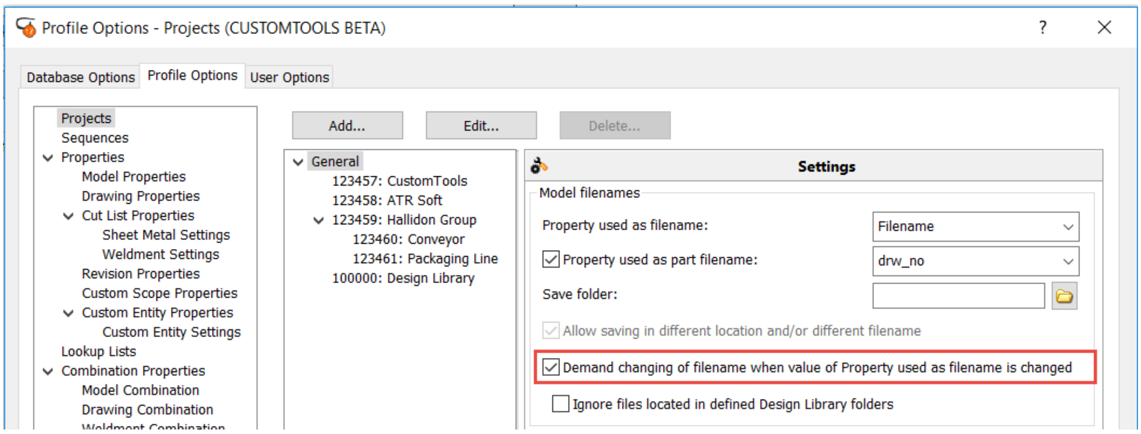
File name changes in CUSTOMTOOLS

If the property used to generate a file name is modified in the Properties pane, then those property value changes can be ignored or CUSTOMTOOLS can offer the possibility to copy or rename the 3D model.

Ignoring the changes of the property values used as file name

If the property value used to generate a file name is modified then CUSTOMTOOLS will not notify the user of the changes. When saving the Properties pane, the file name will remain unchanged.

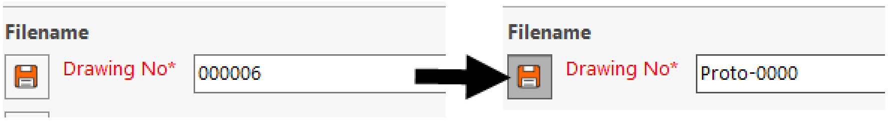


To ignore the property value modification, open the **Options**,

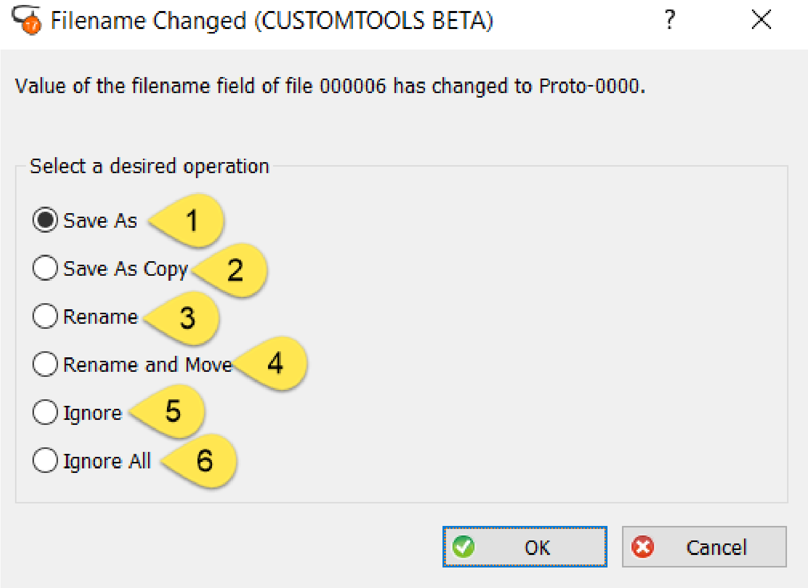
1. Select **Projects** from the **Profile Options** tab,
2. Leave the **Demand changing of filename when value of Property used as filename is changed** check box NOT selected.

Notifying the user of the changes of the property values used as file name

If the property used to generate a file name is modified, then the user will be informed and ask to select an action to be done.

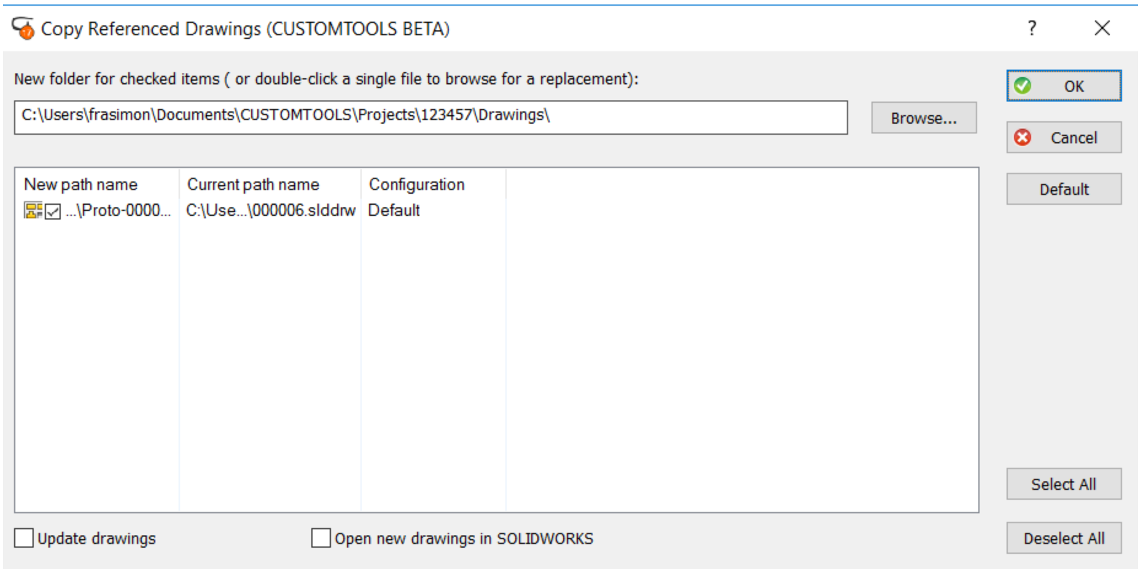


In this example, the property *Drawing No* was modified, then the property called *File name*, used to generate a file name, is updated. When pressing **OK** on the Properties pane, the **Filename Changed** notification appears.



The user is notified and offered to select an action:

1. **Save As**: Creates a new file with the new file name.
2. **Save As Copy**: Creates a new file with the new file name and replaces the old model with the new model in the assembly.
3. **Rename**: Renames the existing document with the new file name.
4. **Rename and Move**: Renames the original file and moves it to the desired new location.
5. **Ignore:** The document is saved without applying any changes to the file naming.
6. **Ignore All:** The document is not saved at all.



CUSTOMTOOLS

If the 3D model has one or multiple drawings, then the user will be prompted with the **Copy Referenced Drawings** dialog. If the referred drawings need to be renamed, then the check box next to the drawing needs to be selected.

NOTE: CUSTOMTOOLS will used the drawing file naming rules to name/rename the drawings.

CUSTOMTOOLS

Chapter 12: Managing your projects

Introduction

What does it do?

The projects can be used to ease the management of the SOLIDWORKS documents. CUSTOMTOOLS projects can be configured so that documents are always saved to the right folder and to the right name. In addition, default property values can be automatically loaded, once the project is selected from the Properties pane.

How does it work?

The user is able to define project structure including sub-projects. Specific file naming and storing conventions can be defined for each project. Default property values can be defined for the assemblies, part and drawing.

How can you use it?

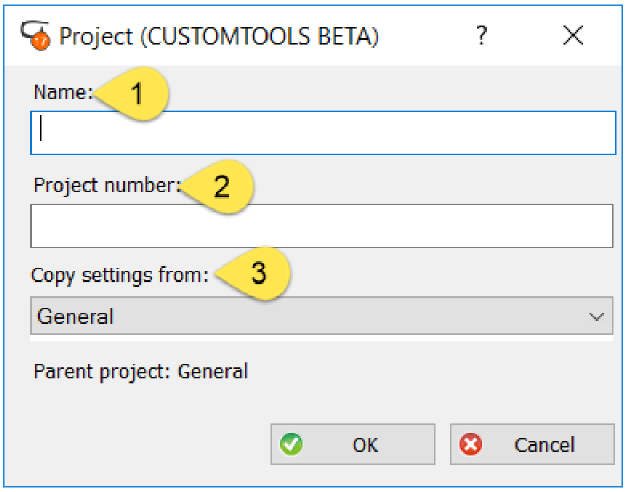
The projects can be accessed from the Properties pane of CUSTOMTOOLS. Once a project is selected, all the default property values assigned to the project will be automatically loaded. Once the user clicks “OK” on the property the file will be saved to the designated folder. The projects can also be accessed from the Copy and Search functionalities of CUSTOMTOOLS.

NOTE: The general project is the default project of CUSTOMTOOLS and cannot be removed. Files that were designed prior to CUSTOMTOOLS will have **No project** assigned by default.

Add a new project

To add a new project, open the CUSTOMTOOLS **Options** dialog,

1. Select **Project** from the **Profile Options** tab,
2. Click **Add**. The **Project** dialog opens,



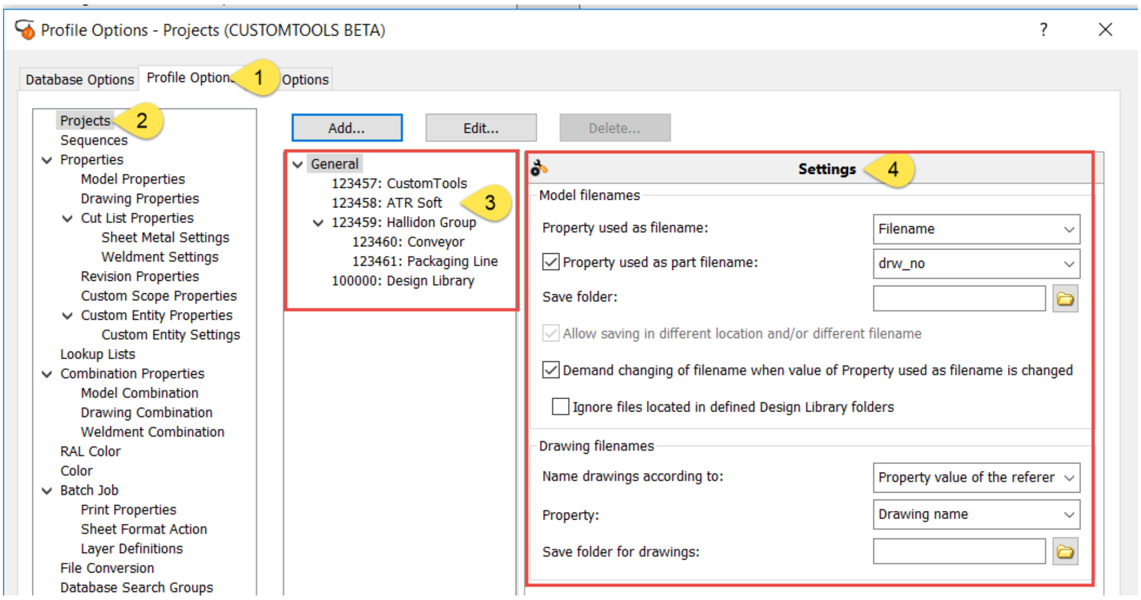
1. **Name**: Defines the name of the project as it will appear in the Properties pane (*e.g. Pamet*).
2. **Project number**: Defines a project number (*e.g. 125460*).

   TIP: Project name/number can be associated in the file name by using a combination of property.
3. **Copy settings from**: Copies the settings of an existing project to the new project. File naming rules, destination folder and default property value will be copied to the new settings of the project.

Define the file saving rules used by a project

Specific **Save folder** and file **Property used as filename** can be assigned for each project. This allows the user to ease the management of SOLIDWORKS files.

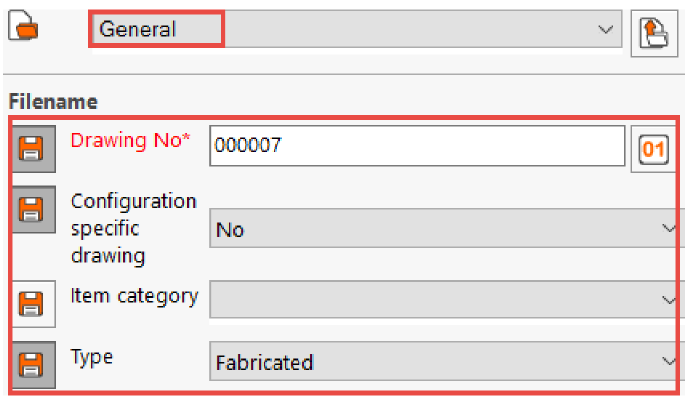
CUSTOMTOOLS



To define the saving conventions used by a project, open the **Options**,

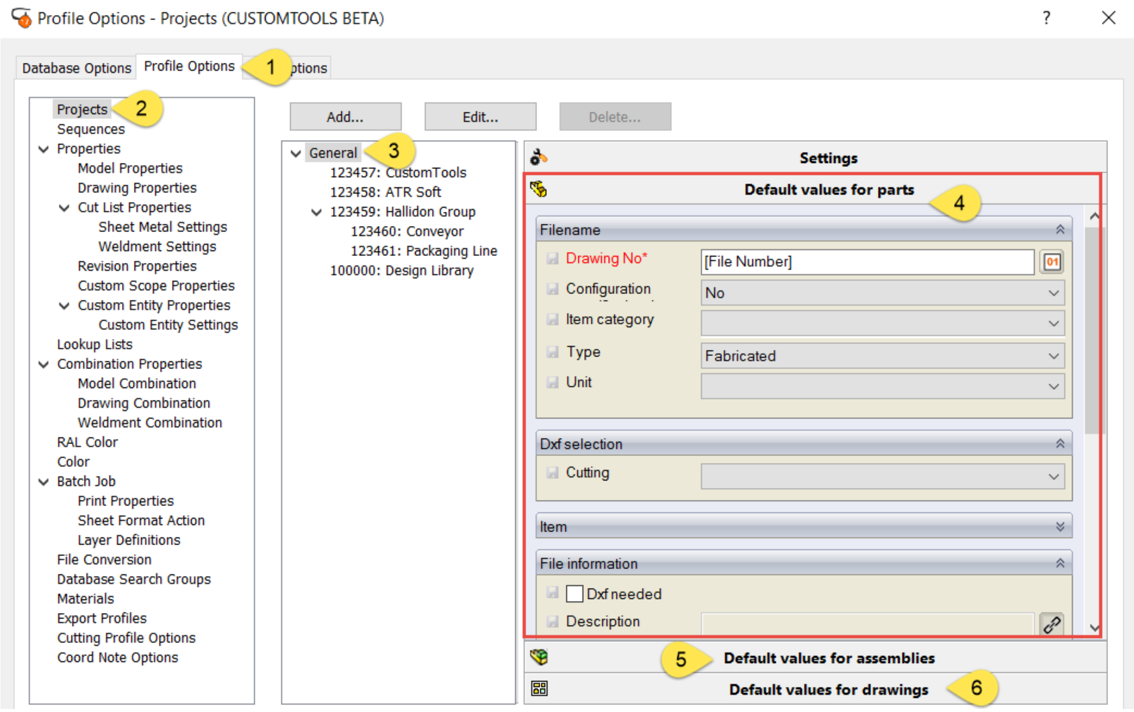
1. From the **Profile Options** tab,
2. Select **Project**, from the tree view.
3. Then select the specific project for which the saving settings are to be defined.
4. Select the **Property used as filename** used to generate a file name. Specify the **Save folder** used by CUSTOMTOOLS to save your Models and Drawings.

Assign default property values for a project



When a project is selected from the Properties pane of CUSTOMTOOLS, certain property values can automatically be loaded.

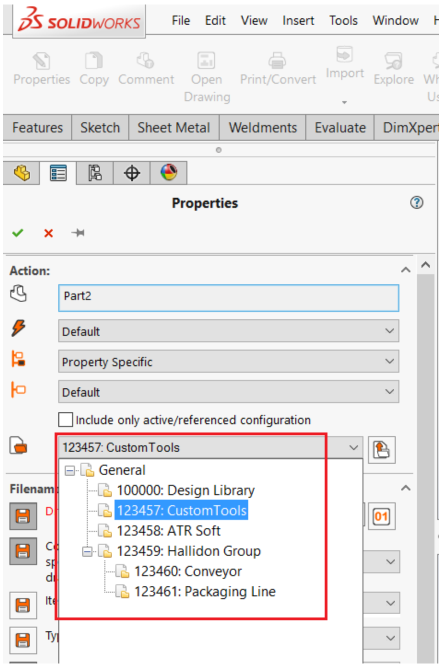
Default property values can be assigned for Parts, Assemblies and Drawings.



To assign default property values to your project, open the CUSTOMTOOLS **Options** dialog:

1. From the **Profile Options** tab,
2. Select **Project** from the tree view.
3. Select the project where the default property value should be assigned.
4. To define the default custom properties for parts, select **Default Values for parts.** Once selected, the Properties pane appears. Custom properties that should be loaded automatically with the project can be defined.
5. Defines default values to be loaded with assemblies.
6. Defines default values to be loaded with drawings.

Selecting a project from the Properties pane



Click on **Properties** to open the **Properties pane** of CUSTOMTOOLS. By default, CUSTOMTOOLS, loads the last used Project for new models created in SOLIDWORKS or selects the projects assigned to the model. If no project is used (e.g. for files that were designed before using CUSTOMTOOLS) then the **No Project** is selected.

By clicking on **General**, then all the Projects defined in CUSTOMTOOLS appears in the menu. The desired project can then be selected.

TIP: Project default values can be unloaded by clicking on the icon.

CUSTOMTOOLS

Chapter 13: Copying and renaming assemblies

Introduction

What does it do?

The copy functionality can be used to copy and rename SOLIDWORKS assembly all referred components. In addition, property value of the documents to copy can be modified. This functionality can also be used to propagate property value to the SOLIDWORKS documents.

How does it work?

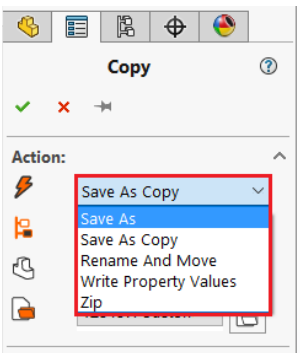
The copy allows the user to easily copy and rename files by using existing file naming and storing rules defined in CUSTOMTOOLS.

How can you use it?

Documents can be added to the list of files to process when no documents are opened in SOLIDWORKS. The documents can then be added from the Copy pane. In order to add the active 3D Model, select the desired part or assembly and click **Copy** from the **CommandManager** or from the CUSTOMTOOLS menu.

NOTE: If an assembly is selected, then all the referenced files used in that assembly will also appear in the list of documents to process.

Action menu



The copy functionality offers different set of actions:

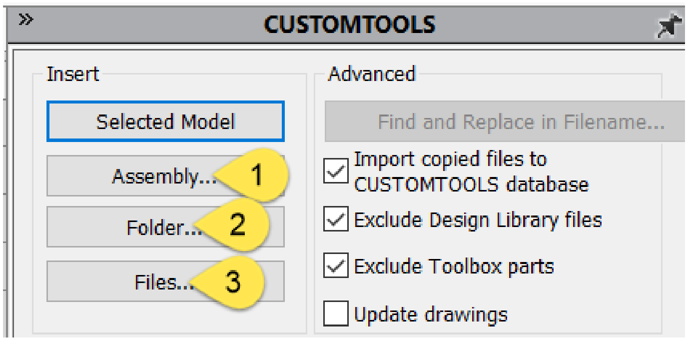
- **Copy**: Copies and renames the SOLIDWORKS documents based on the defined naming and storing rules defined in CUSTOMTOOLS. The copied documents are not replaced in the active assembly.
- **Save As**: Copies, renames and replaced the SOLIDWORKS documents in the assembly currently opened.
- **Rename and Move:** Renames and moves the SOLIDWORKS documents based on the defined naming and storing rules defined in CUSTOMTOOLS.

  NOTE: It is possible to only rename the SOLIDWORKS documents without moving them. They can also be moved without being renamed.
- **Write Property Values:** Propagated property values to the 3D model, part, assembly or drawing.
- **Zip:** Creates a zip file of the listed document.

Adding files to the list of documents to processed

If no documents are opened in SOLIDWORKS

The copy functionality can be accessed from the **Command Manager** or CUSTOMTOOLS menu when no documents are being opened in SOLIDWORKS. The SOLIDWORKS files can then be added to the list of files to process from the Insert:



1. **Assembly**: Prompts the user with the **Open** dialog to select an assembly and inserts all referred components.
2. **Folder**: Prompts the user with the **Browse For Folder** dialog to select all the SOLIDWORKS files located in a folder

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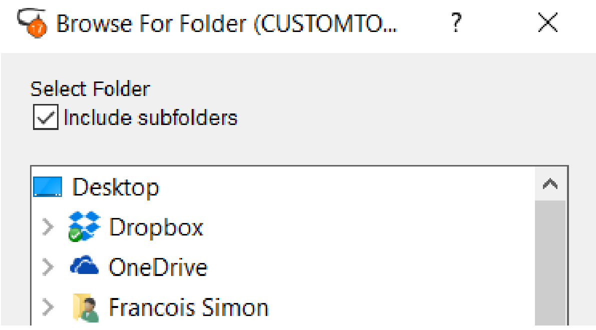


NOTE: Files located in subfolder can also be added, by selecting the **Include subfolders** check box.

3. **Files**: Prompts the user with the **Open** dialog to select the files to be added to the list of documents.

Copying the active assembly

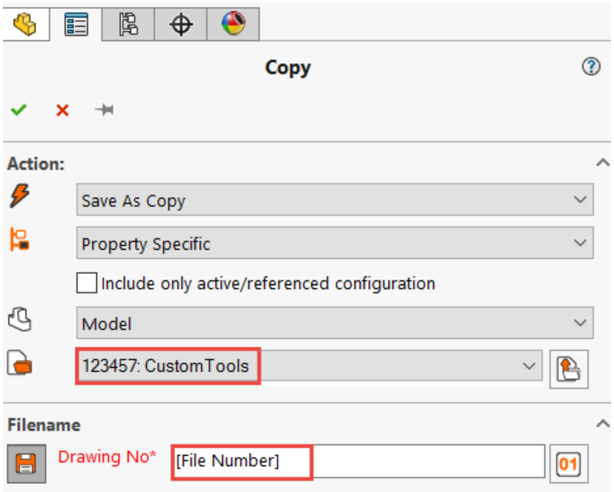
By selecting the top assembly in SOLIDWORKS and clicking **Copy,** from the **Command Manager** or CUSTOMTOOLS menu, CUSTOMTOOLS will automatically load all the SOLIDWORKS Documents referred by the selected component.

Defining the new filename for the files to be copied

CUSTOMTOOLS uses the file naming rules defined in CUSTOMTOOLS to rename the SOLIDWORKS documents.

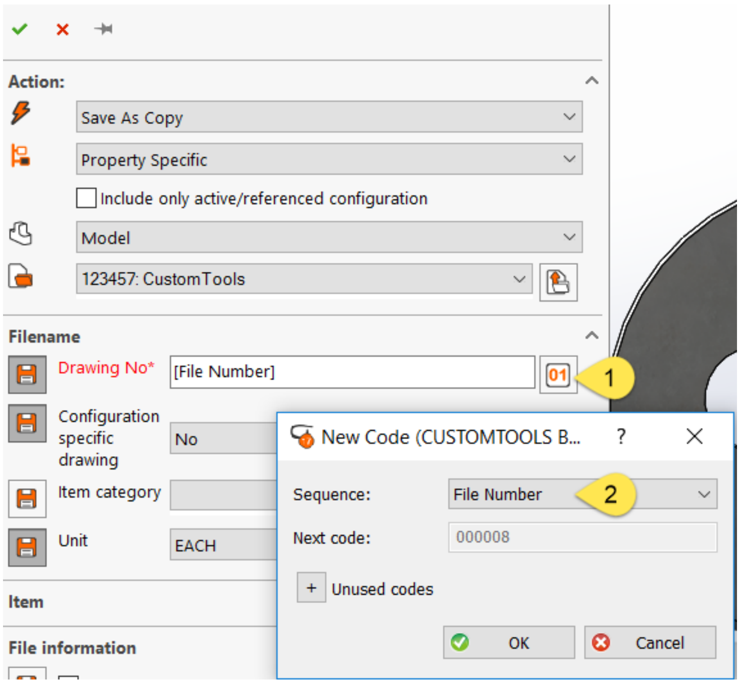
Defining the property used in the filename

Using the project to load a sequence



If a project is selected from the Copy pane (Left pane), then CUSTOMTOOLS will load all the default custom properties automatically, if a sequence is associated with the project, then the name of the sequence will appear in the property.

Selecting a sequence



Defined sequences can also be selected directly from the Properties pane:

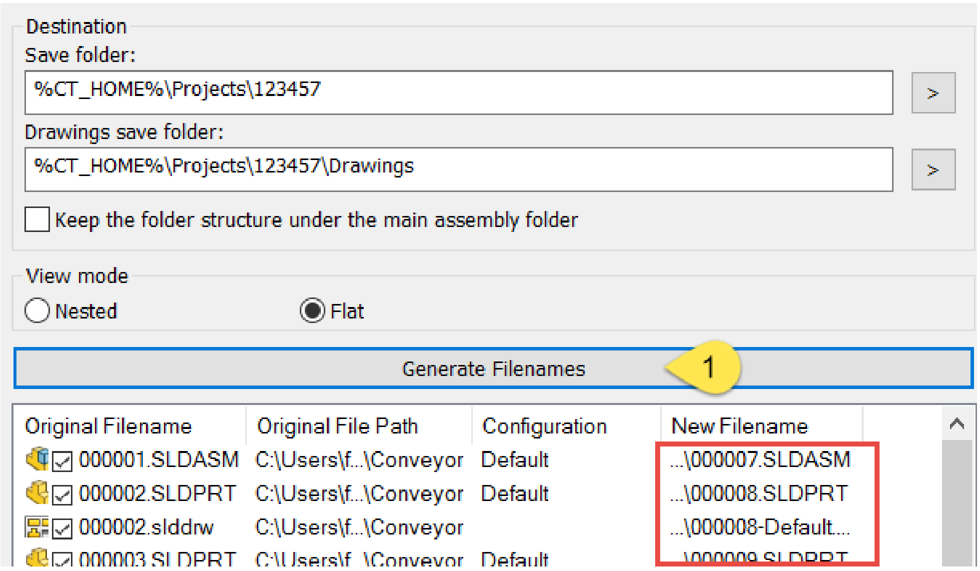
1. By clicking on the **GetSequence** button use to open the **New Code** dialog,
2. Select the sequence to be used.

Modifying property value used in a combination to generate a filename

If a property is used for example in a combination of properties to generate a filename, then the property value can be modified from the Copy pane used in the copy to generate a filename accordingly.

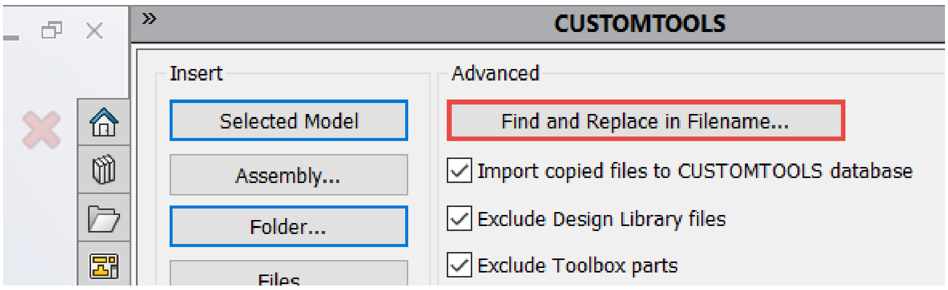
Generate the filename to be used by the copied SOLIDWORKS documents

Once all the property values have been defined in the Copy pane of the Copy, the filename for the documents to copy or rename can be generated.



Click **Generate Filenames** from the CUSTOMTOOLS pane (Left pane) to generate the new filename to be applied to the list of documents process.

Find and replace in Filename



Once the file name has been generated, the user is able to modify part of the filename by clicking on the **Find and Replace in Filename** button from the **Advanced** option in the **Copy.**

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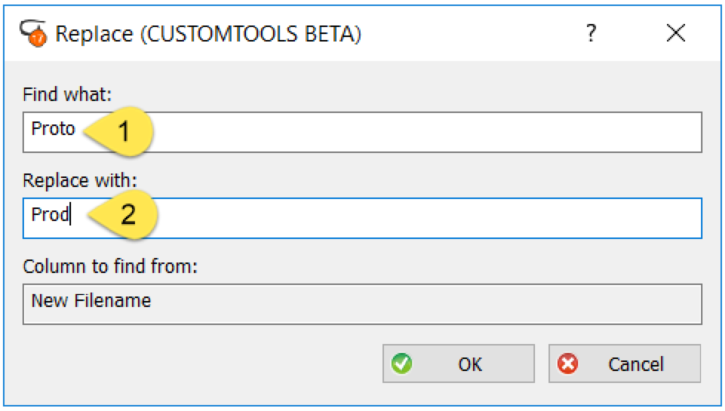
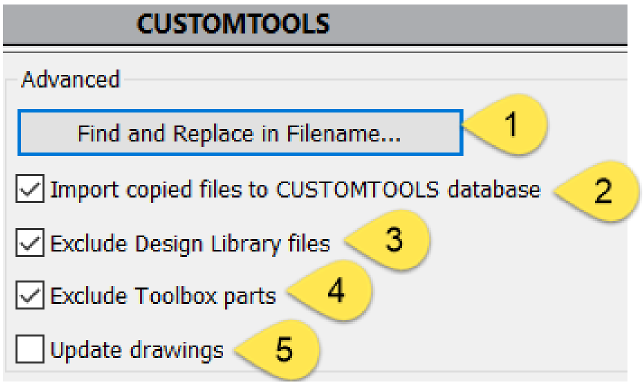


1. **Find what:** Type the text that should be search for and replace from the New Filename column,
2. **Replace with:** Type the replacement text.

Additional options

Advanced options

The Advanced options can be found from the CUSTOMTOOLS pane that appears on the right side of SOLIDWORKS, once the Copy is opened.

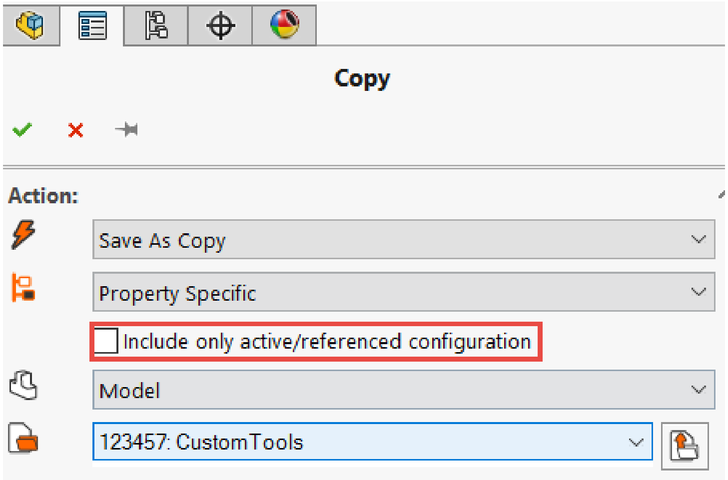


1. **Find and Replace in Filename:** Opens the **Replace** dialog where the user can search and replace specific part of the Filename.

   NOTE: Only works when the **New filename** to be given to the SOLDIWORKS Documents has been generated.
2. **Import copied files to CUSTOMTOOLS database:** The files that are copied will automatically be imported into the CUSTOMTOOLS database.
3. **Exclude Design library files:** Files that are stored in designated Design library folders of SOLIDWORKS will be excluded from the list of files to copy or rename.

4. **Exclude Toolbox parts:** Files that are stored in the Toolbox library folders of SOLIDWORKS will be excluded from the list of files to copy or rename.
5. **Update drawings:** If the custom properties have been modified then CUSTOMTOOLS opens the drawings and updates the custom properties inserted in the drawing.

Handling configurations of the files to be copied



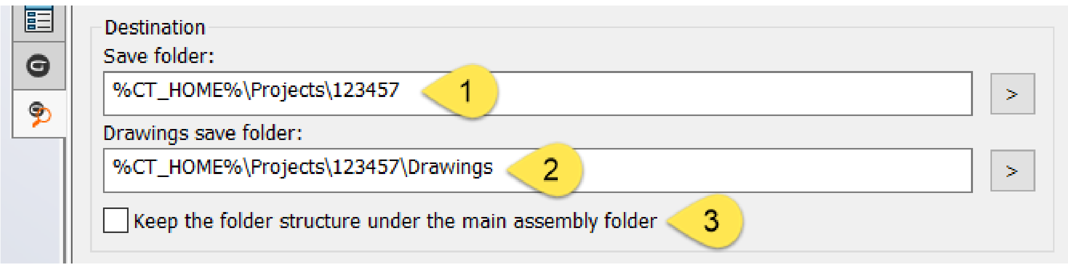
When copying an assembly or part, the user has the possibility to only include the active configuration in the new copied documents or the configurations referenced in the top level assembly.

Select the **Include only active/referenced configuration** check box.

NOTE: This option should NOT be used with the **Rename and Move** action, as all other existing configurations will be removed.

Defining the destination folder for the copied document

Different save folders can be defined where to save 3D models and their referring drawings. In addition, CUSTOMTOOLS offers the possibility to keep the folder structure under the main assembly. The destination folders are defined from the CUSTOMTOOLS pane that appears on the right side of SOLIDWORKS.



1. **Save folder:** Defines the destination folder for parts and assemblies.
2. **Drawing save folder:** Defines the destination folder for drawings.

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3. **Keep the folder structure under the main assembly folder**: The copied documents are maintained in the same folder structure that is built under the main assembly being copied.

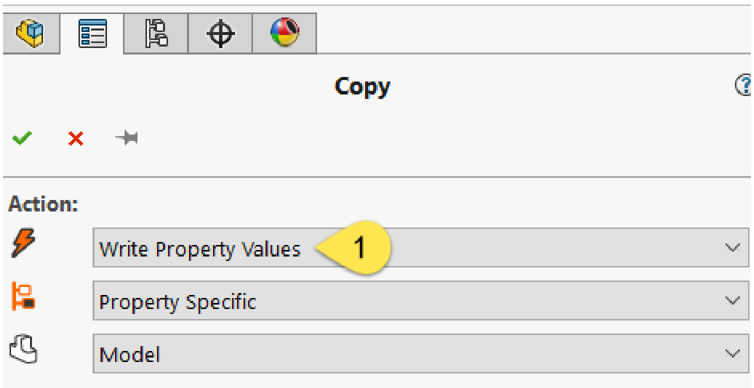
NOTE: If a CUSTOMTOOLS project is selected from the Properties pane (Left pane), then the destination folders for 3D Models and drawing will be updated accordingly based on the settings defined for the project.

Modifying property values to your SOLIDWORKS documents

The custom properties of the SOLIDWORKS documents to be copied can be modified for 3D Model, parts, assemblies or drawing in a similar Properties pane as the one defined in CUSTOMTOOLS.

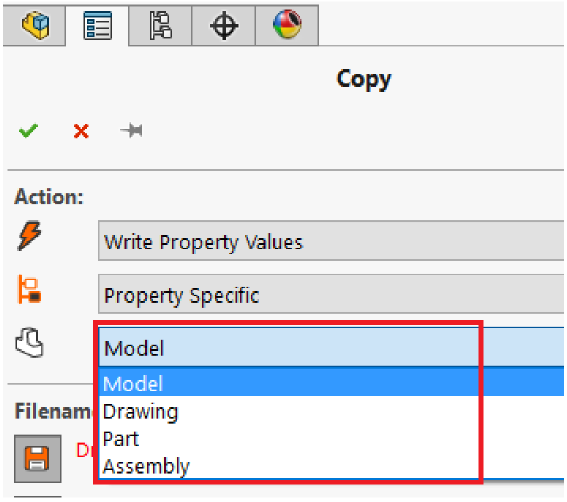
Propagating property value

CUSTOMTOOLS can be used to propagate property value to the SOLIDWORKS Documents without copying or renaming the files.



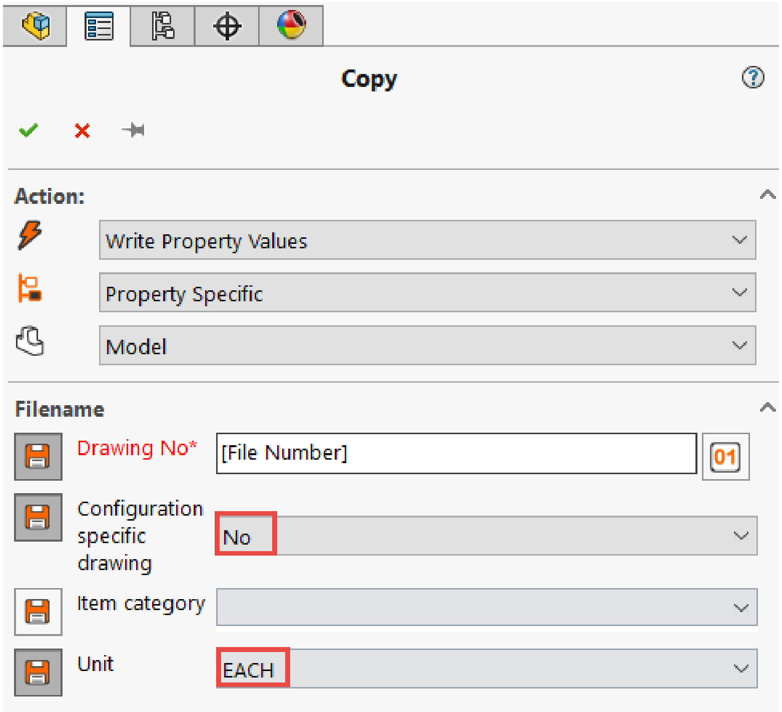
Select **Write Property Values** from the **Action** of the Properties pane opened with the Copy functionality.

Defining for which file type the properties will be modified



Select the file type (Model, Drawing, Part or Assembly) from the Action of the Properties pane opened with the Copy functionality.

Insert new property values



Custom properties can freely be modified in the Properties pane available in the copy.

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NOTE: The property value that is defined will be applied to all the documents selected in the list of documents to process (e.g. The value Fabricated, will be written to all the documents having the property Type.