

**FOCUS
ON REAL
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THE REST**

**CUSTOMTOOLS
COPYING AND
RENAMING
ASSEMBLIES**



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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.

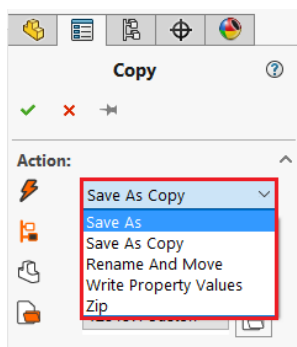
How can you use it?

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





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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.

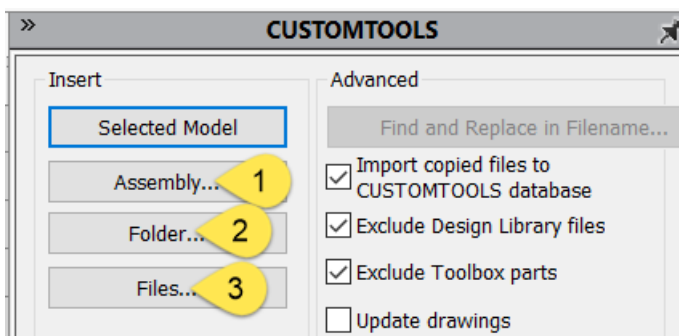
They can also be moved without being renamed.

- **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

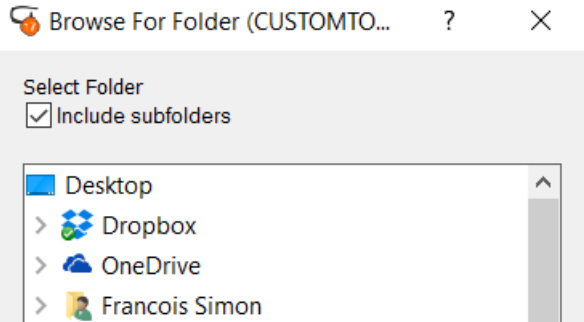
The copy functionality can be accessed from the **Command Manager** or CUSTOMTOOLS menu when no documents are being opened in SOLIDWORKS. The SOLIDWORKS files can then be added to the list of files to process from the Insert:



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

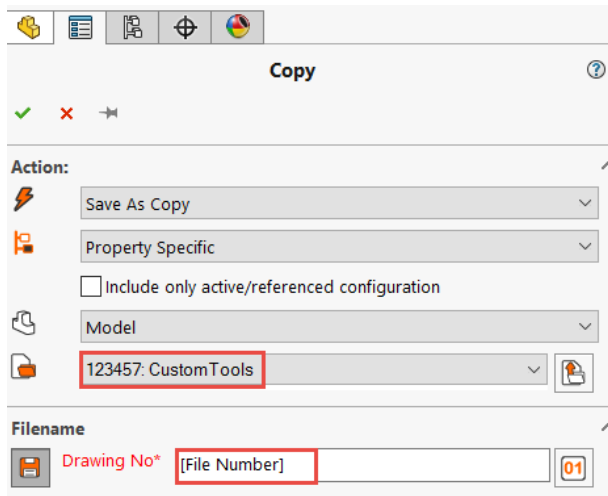
CUSTOMTOOLS uses the file naming rules defined in CUSTOMTOOLS to rename the SOLIDWORKS documents.

Defining the property used in the filename

Using the project to load a sequence

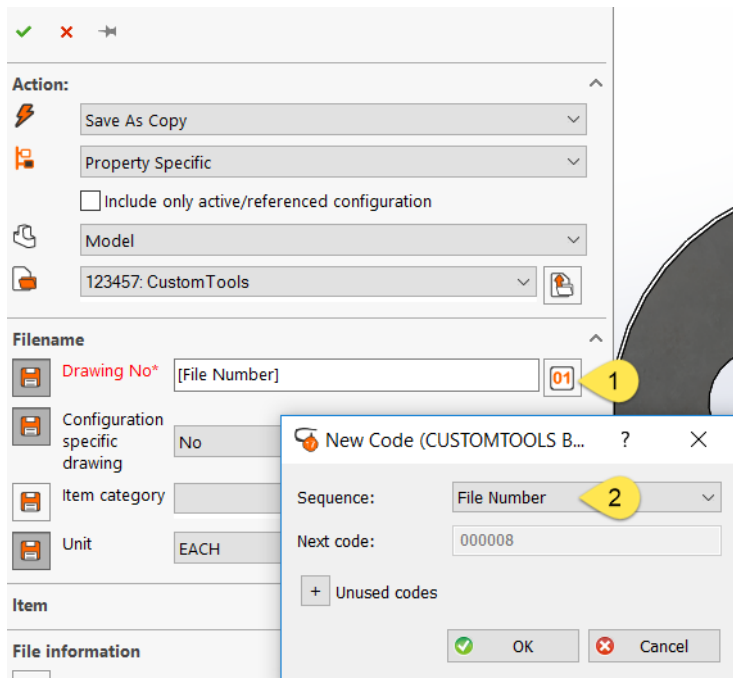


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If a project is selected from the Copy pane (Left pane), then CUSTOMTOOLS will load all the default custom properties automatically, if a sequence is associated with the project, then the name of the sequence will appear in the property.

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.



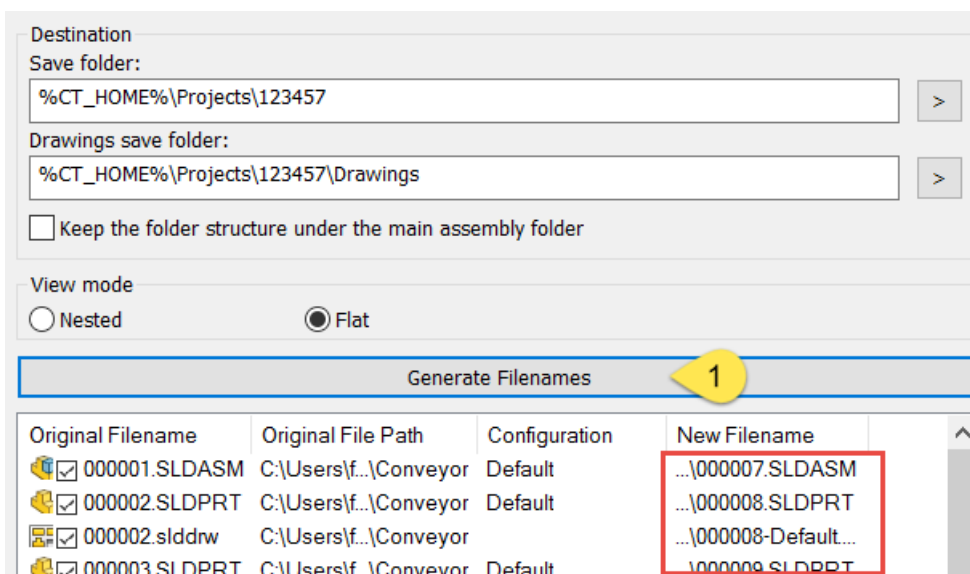
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Modifying property value used in a combination to generate a filename

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.

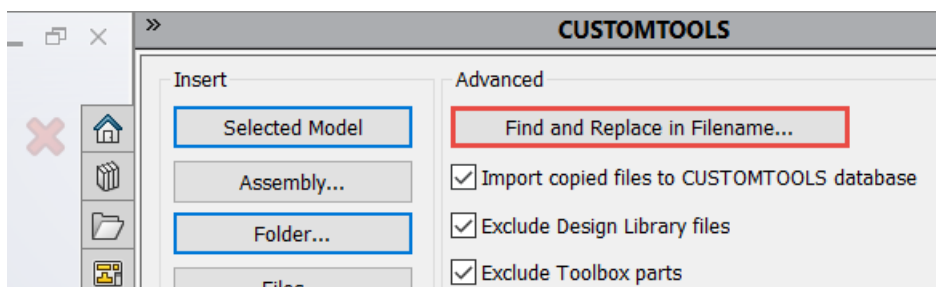
Generate the filename to be used by the copied SOLIDWORKS documents

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.

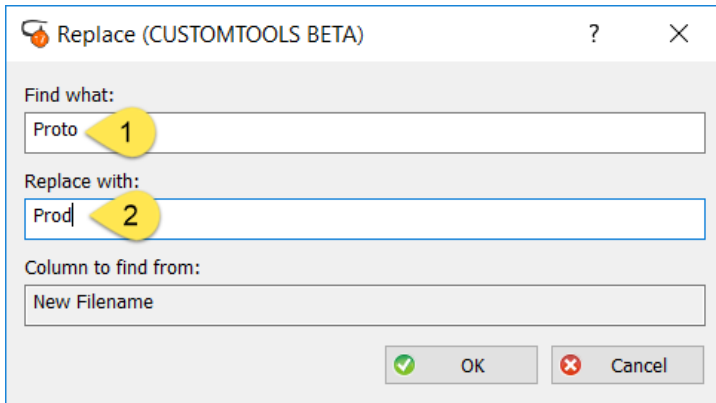
Find and replace in Filename



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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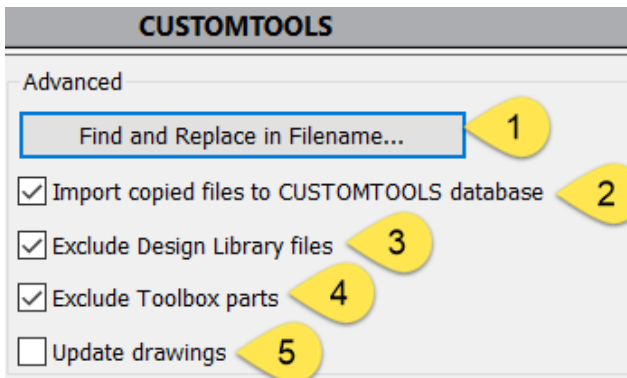


1. **Find what:** Type the text that should be search for and replace from the New Filename column,
2. **Replace with:** Type the replacement text.

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 SOLIDWORKS Documents has been generated.

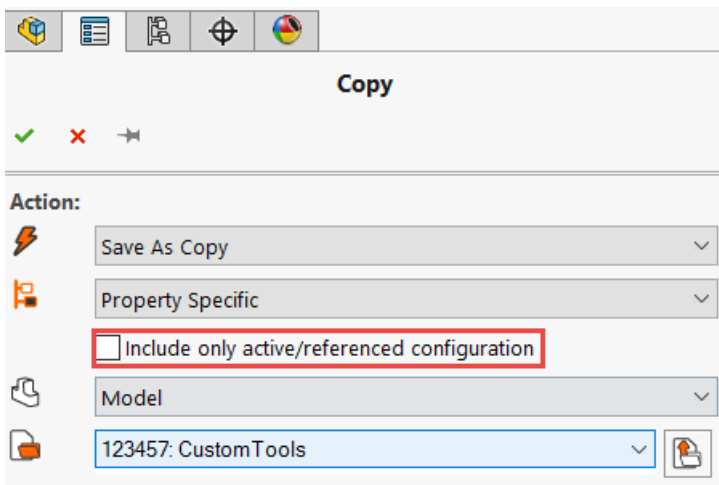
2. **Import copied files to CUSTOMTOOLS database:** The files that are copied will automatically be imported into the CUSTOMTOOLS database.
3. **Exclude Design library files:** Files that are stored in designated Design library folders of SOLIDWORKS will be excluded from the list of files to copy or rename.



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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.



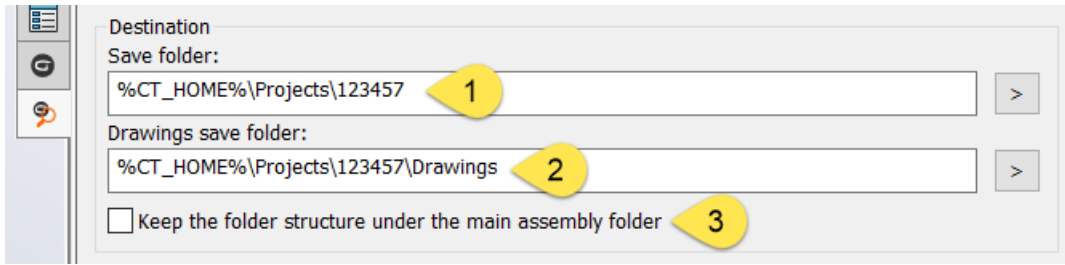
NOTE: This option should NOT be used with the **Rename and Move** action, as all other existing configurations will be removed.

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.



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



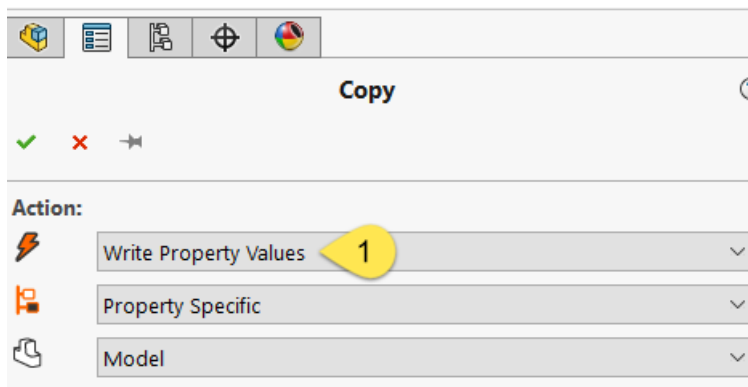
NOTE: If a CUSTOMTOOLS project is selected from the Properties pane (Left pane), then the destination folders for 3D Models and drawing will be updated accordingly based on the settings defined for the project.

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.

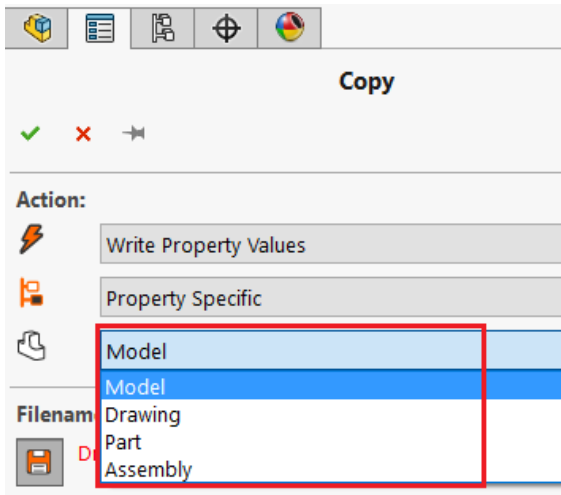




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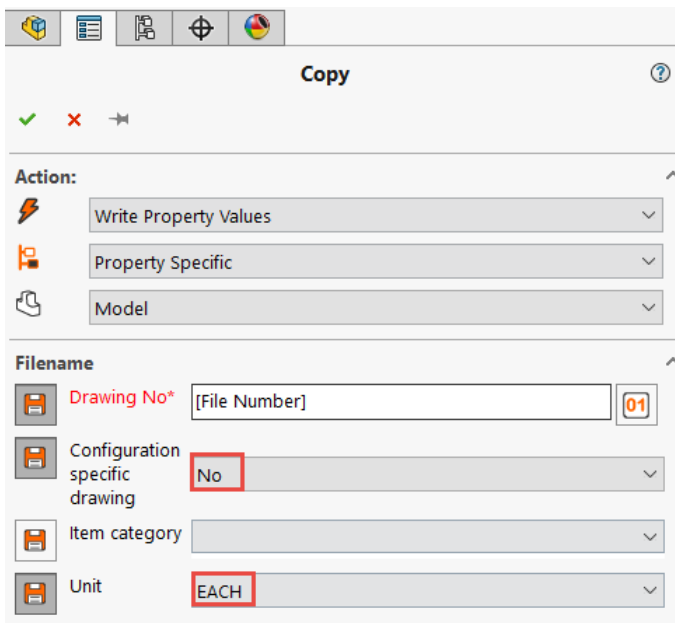
Select **Write Property Values** from the **Action** of the Properties pane opened with the Copy functionality.

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





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



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).