FOCUS ON REAL DESIGN AUTOMATE THE REST

CUSTOMTOOLS

AUTOMATIC

FILENAMING

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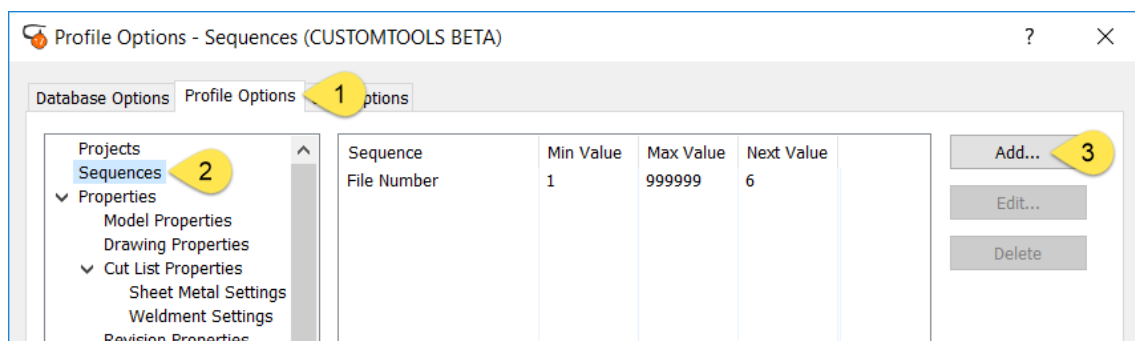
To add a new sequence, open the CUSTOMTOOLS **Options** dialog:

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*).
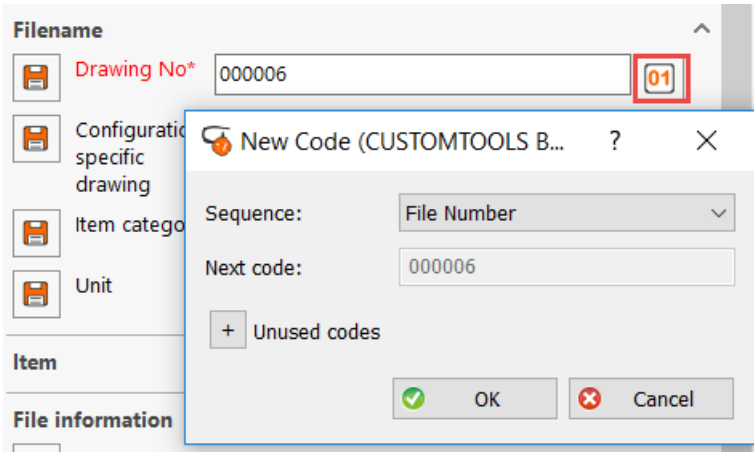

Click on the **01** button to define how many digits are used by the sequence *(e.g. 001, 0001, 00004,...)*.

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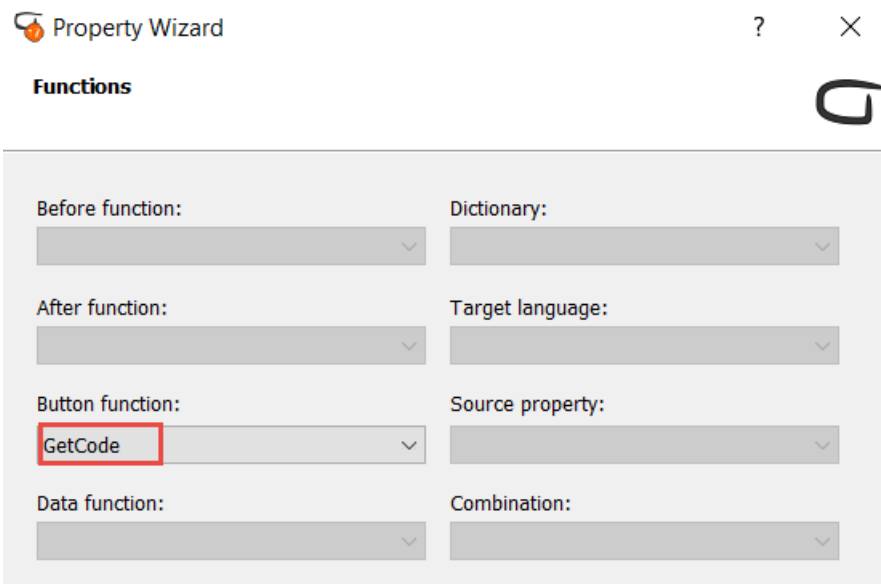
To associate a sequence with a property, open the CUSTOMTOOLS **Options** dialog, from the **Profile Options** tab, select **Model properties/Drawing properties**.

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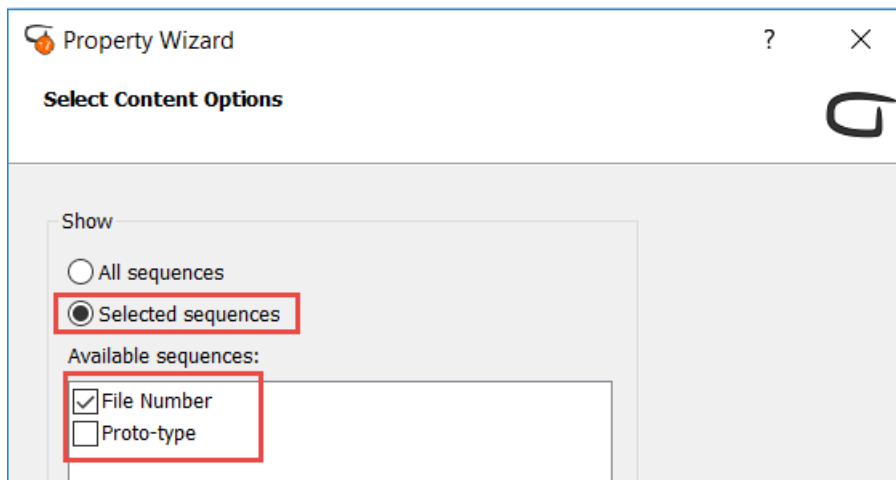
🔆 **NOTE:** Specific sequences can be associated with a property. From the **Property wizard** open the **Select Content Options** and select the **Selected Sequences** button to select which sequences are to be used with the property.

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).*
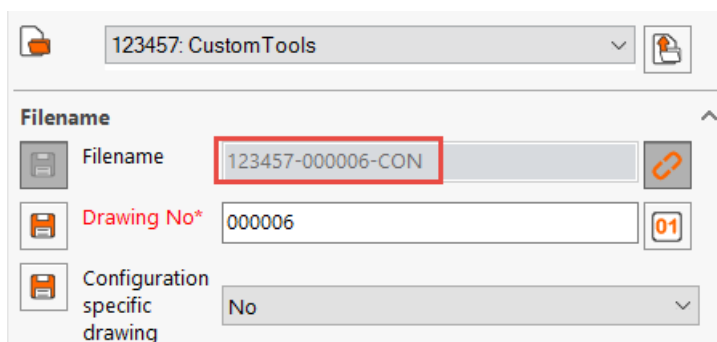
Create a combination of properties

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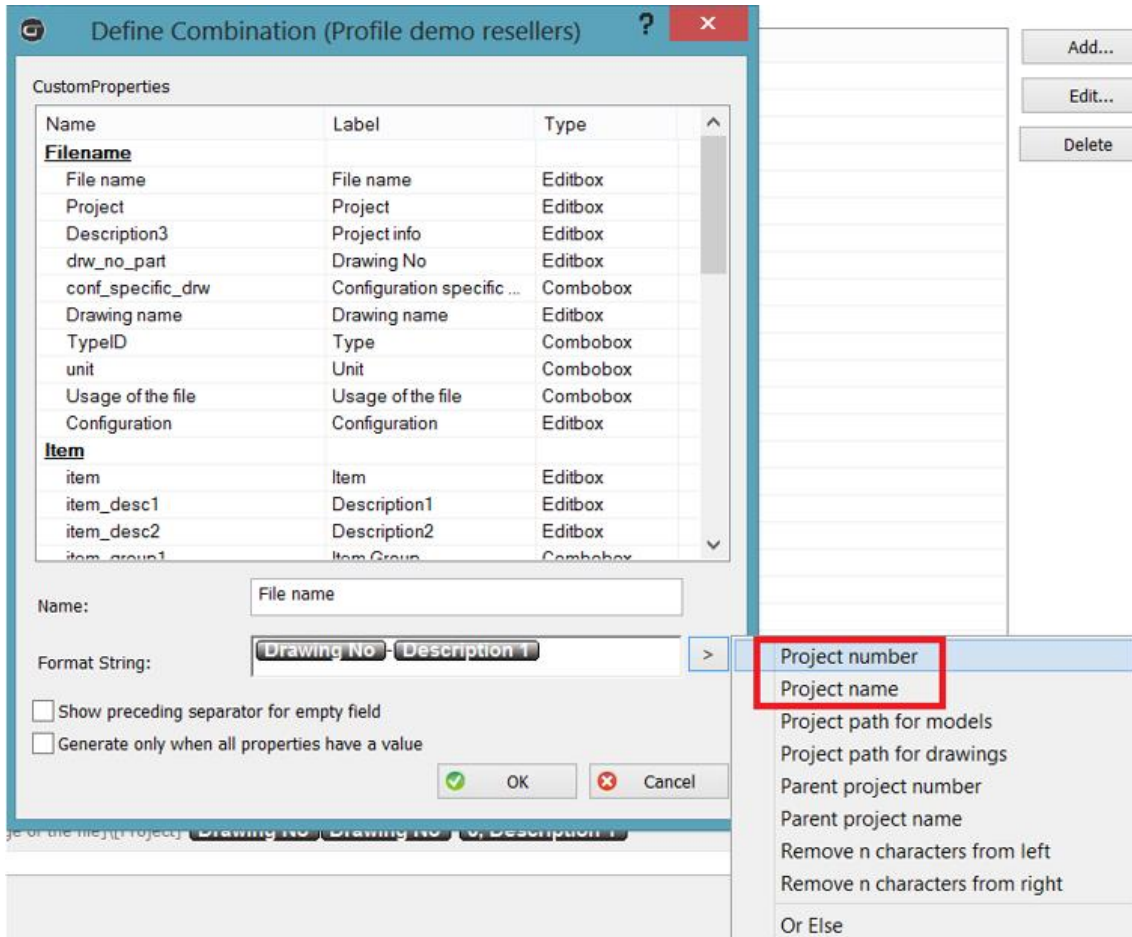
From the **Define Combination** dialog, click on the **>** and select the **Project number** or **Project name** to add it to the **Format String**.

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.**

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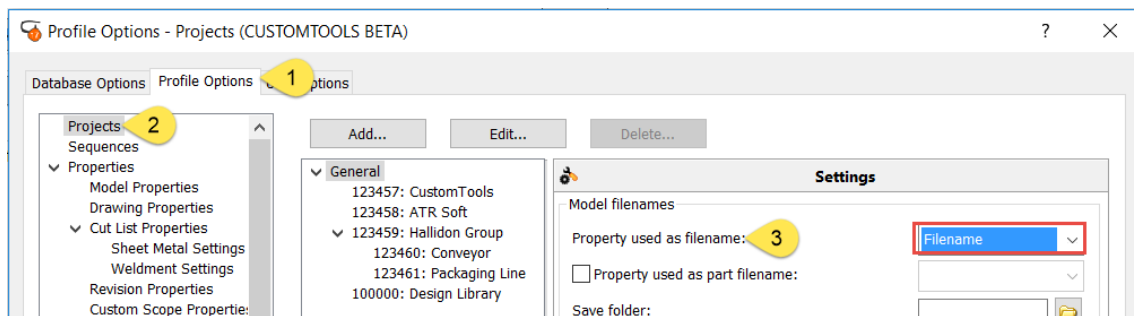
Open the CUSTOMTOOLS **Options** dialog, from the **Profile Options** tab,

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.**

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💡 **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).

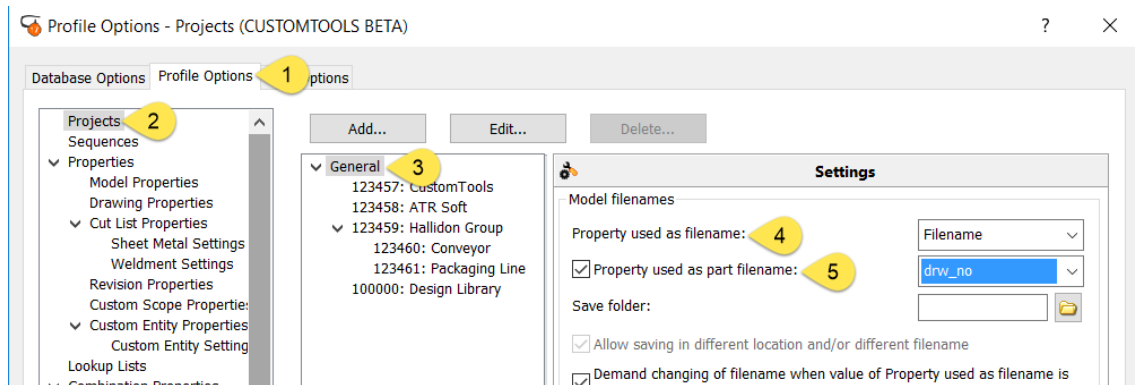## Using different property to generate a name for parts and assemblies

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).
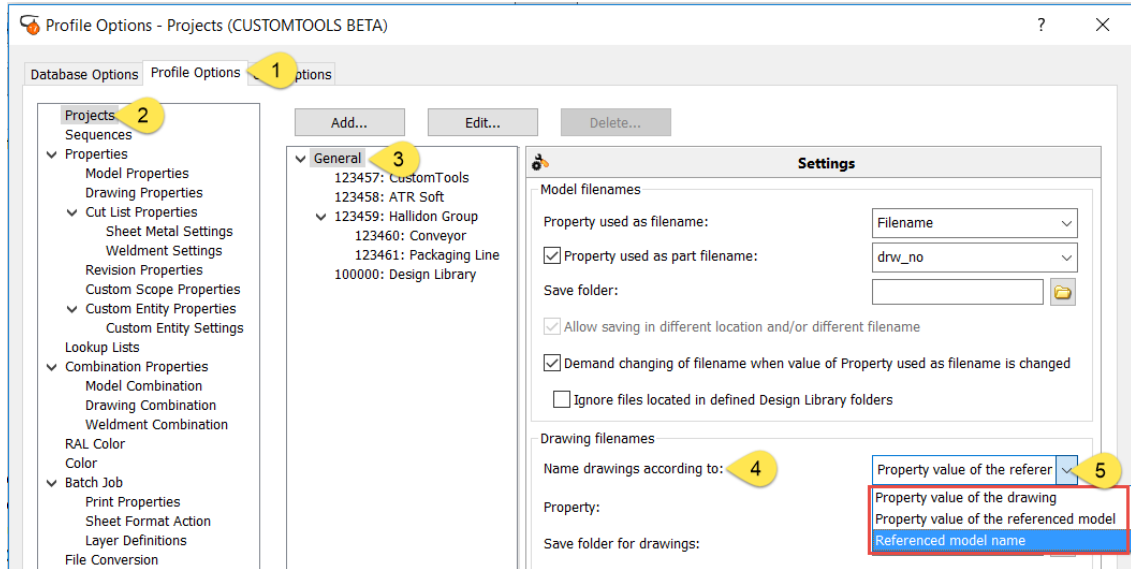
Open the CUSTOMTOOLS **Options** dialog,

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:

   o **Property value of the drawing**: Uses a property value defined in the **Drawing properties.**

   o **Property value of the referenced model**: Uses a property value defined for **the Model properties (Part and Assemblies).**

   o **Referenced model name:** Uses the same filename for the drawing and the model to which it is referring.

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Additional file saving options
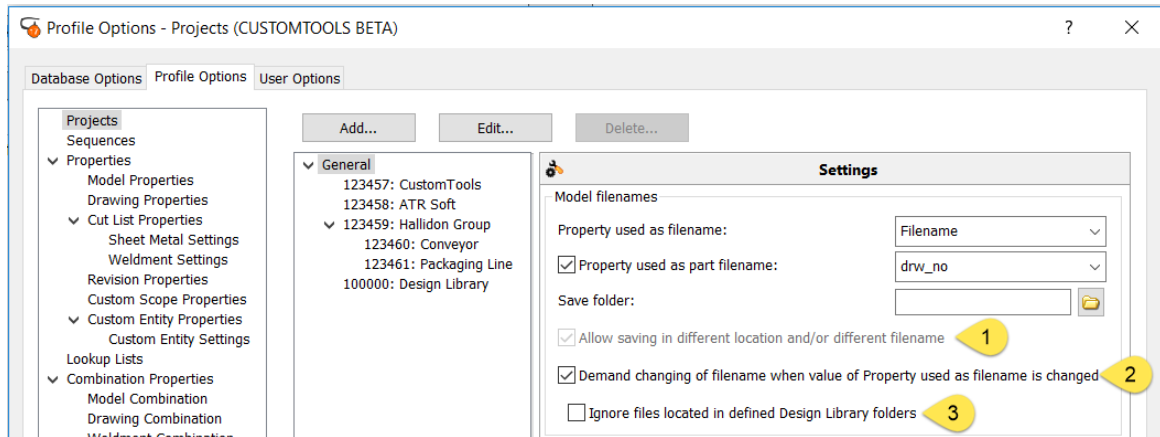
1. **Allow saving in different location and/or different name**:
   a. If the check box is selected, then SOLIDWORKS Save As dialog will appear while saving a SOLIDWORKS documents for the first time. The user will be able to select a different save folder and modify the file name in the Save As dialog.
   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.

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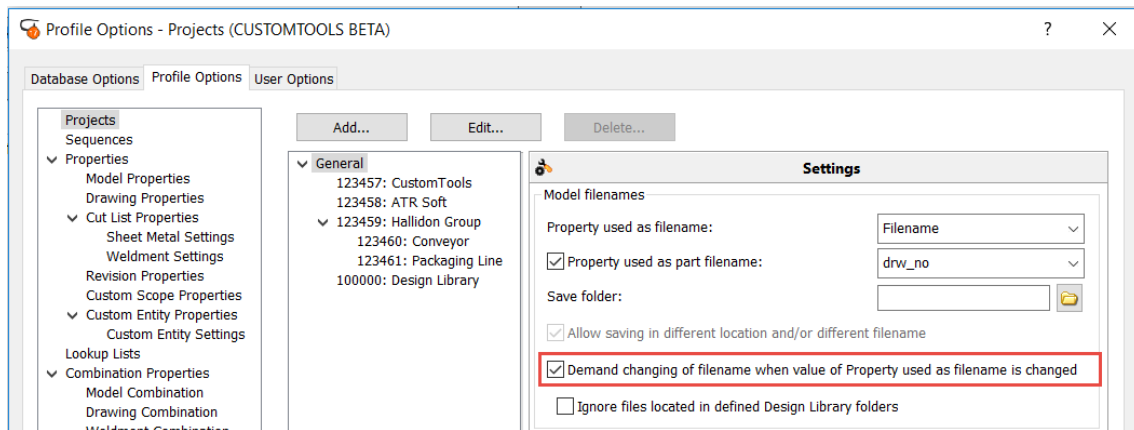
To ignore the property value modification, open the **Options**,

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.
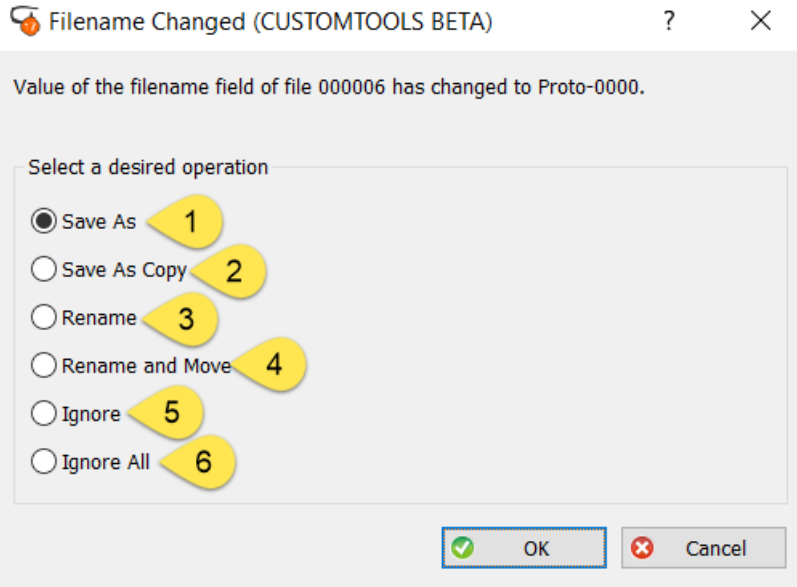
The user is notified and offered to select an action:

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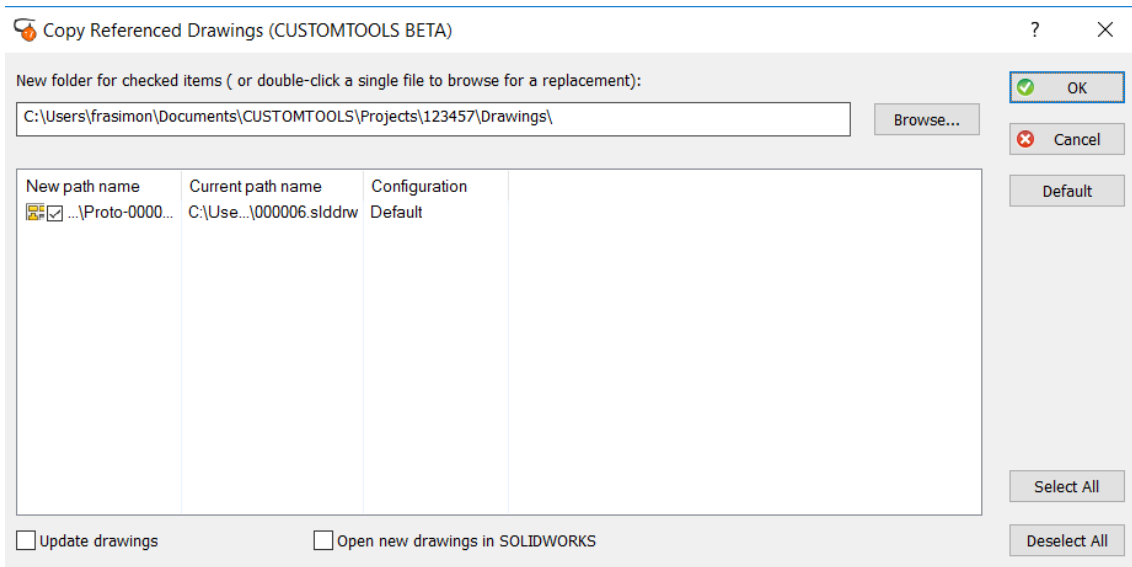
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If the 3D model has one or multiple drawings, then the user will be prompted with the **Copy Referenced Drawings** dialog. If the referred drawings need to be renamed, then the check box next to the drawing needs to be selected.

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