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**CUSTOMTOOLS
BATCH CONVERTING
YOUR SOLIDWORKS
FILES**



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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.

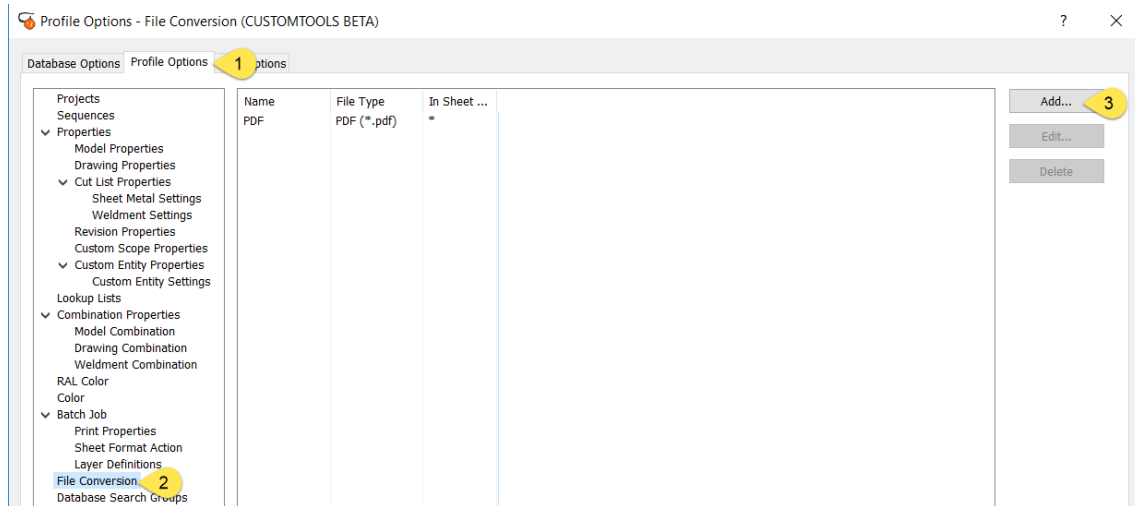
How can you use it?

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.



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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.



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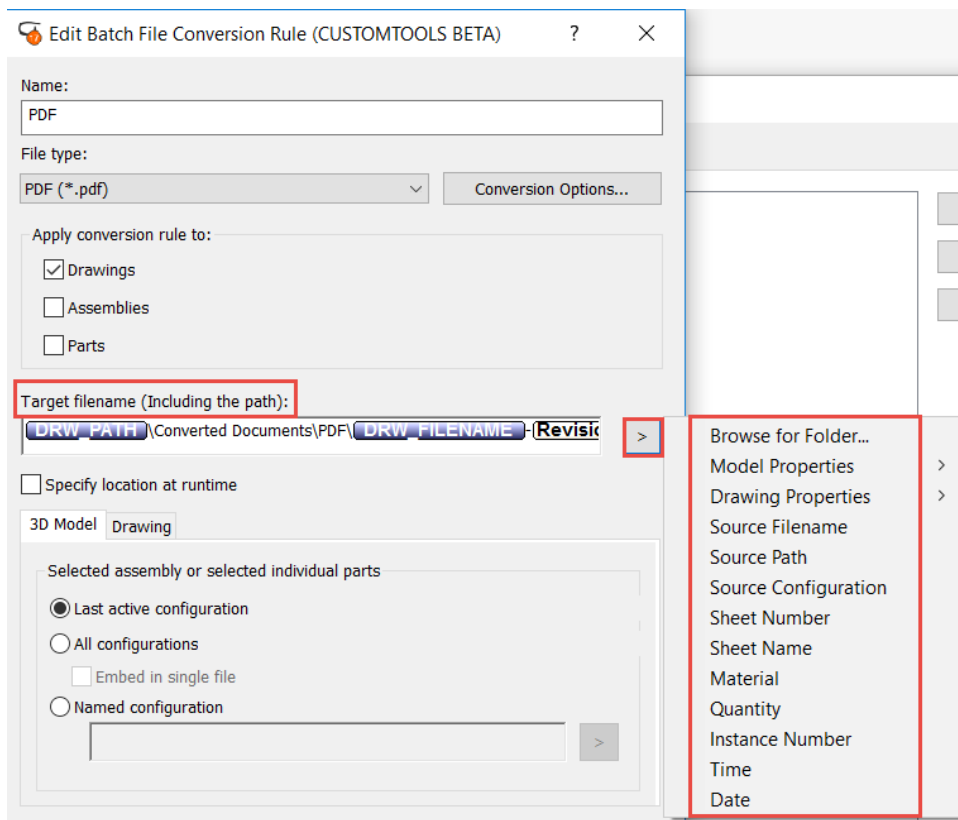
c. **Select the last active sheet if referenced configuration is not available:** Selects the sheet that was last active if the specific referenced configuration cannot be found. By clicking on the >

- 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:



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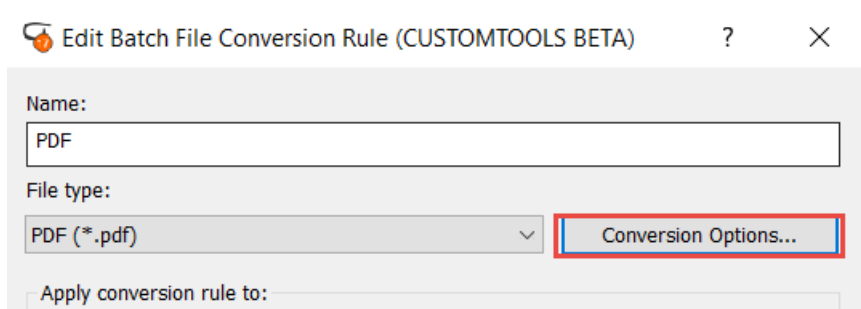
Select the **Source path** to store the converted files in the same folder as the SOLIDWORKS source files. A folder and/or sub-folder structure can also be built. Folders to be created can also be manually typed and created automatically during the conversion (e.g. `\Converted documents\PDF\`).

Defining the file naming convention of the converted files:

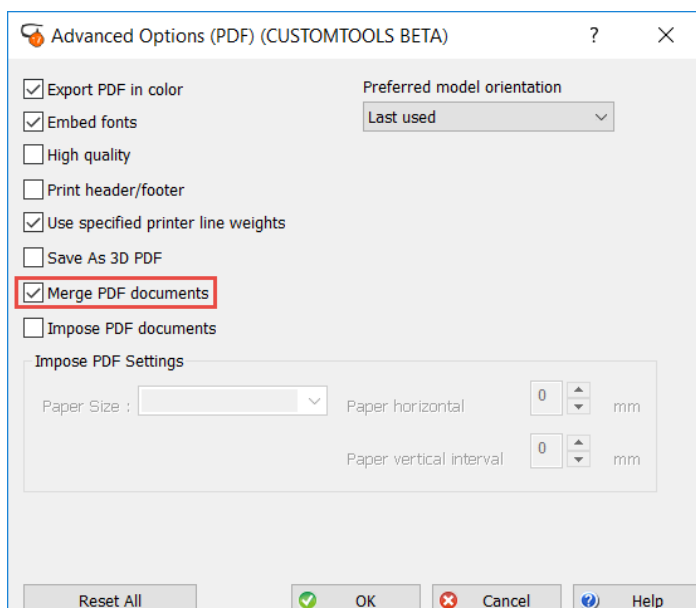
Select the **Source filename** to use the same file name between the SOLIDWORKS source file and the converted file. More advanced file name can be generated by combining different rules (e.g. *property value, source configuration, sheet name project number...*).

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**.

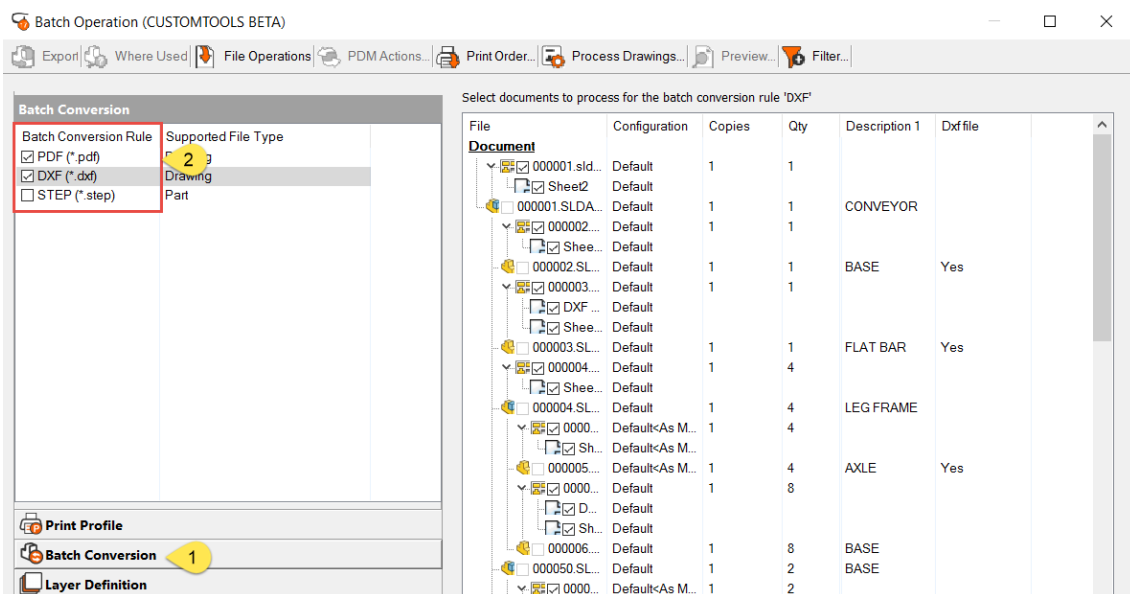


NOTE: The merge PDF is created in the BOM Order.

Converting your SOLIDWORKS 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:

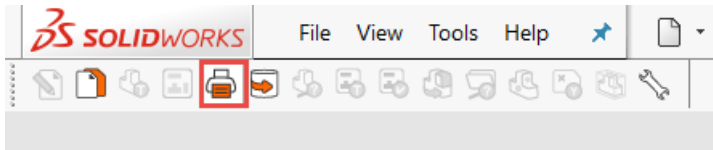
1. Click on the **Batch Conversion** button, to view the **Batch Conversion Rule**:
2. Select the **Batch conversion** rules to be used.

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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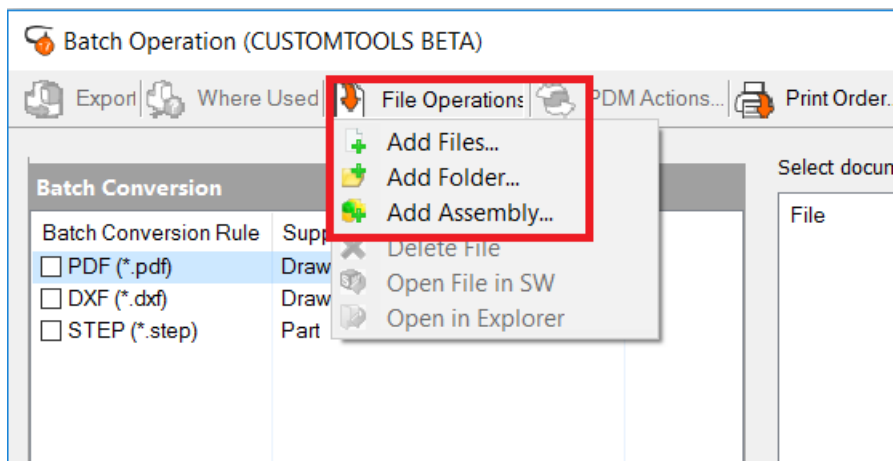


Opening the Batch operation dialog

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.



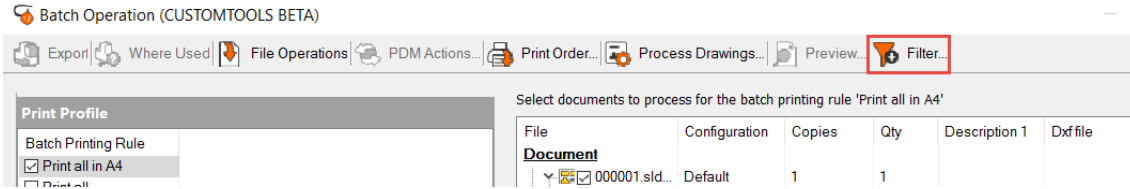
To add files to the list of files to process, click **File Operations**, then CUSTOMTOOLS offers three possibilities to select the files to add to the list of files to convert:

- **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.



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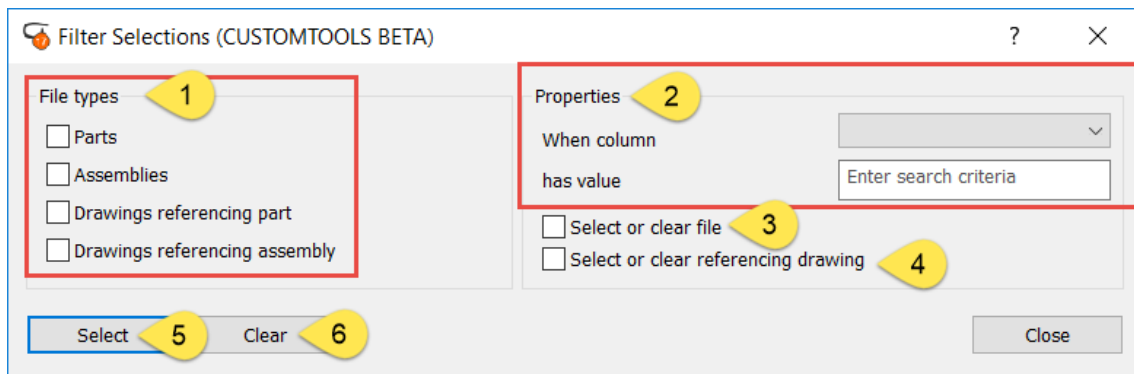
Filtering the documents to be selected for conversion



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 assemblies will be selected/cleared.
5. **Select:** Select the files that matches with the filter criteria defined in the **File type** or **Custom properties**.



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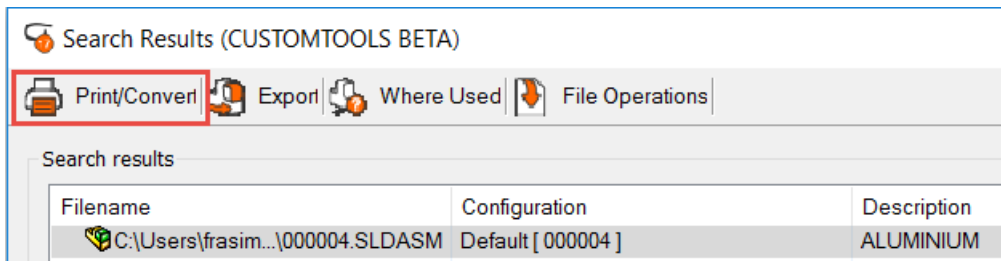
6. **Clear:** Clear the files that matches with the filter criteria defined in the **File type** or **Custom properties**.

From other functionalities of CUSTOMTOOLS

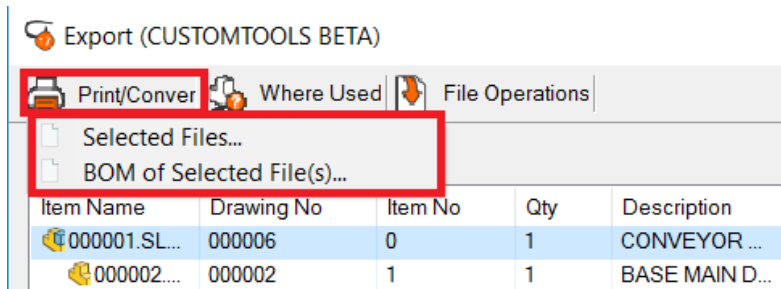
The batch conversion functionality of CUSTOMTOOLS can also be launched from:

- **The Search results:**

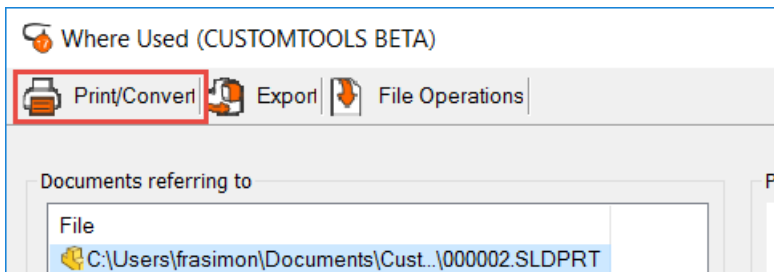
Select an assembly from the search result and click **PrintAndConvert**. If an assembly has been selected, click **BOM of Selected File(s)...** to add the assembly and its child component to the list of document to process.



- **Export**



- **Where used or Open drawing**





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Creating DXF/DWG files

Creating DXF from the SOLIDWORKS Drawings with the Cutting Profile

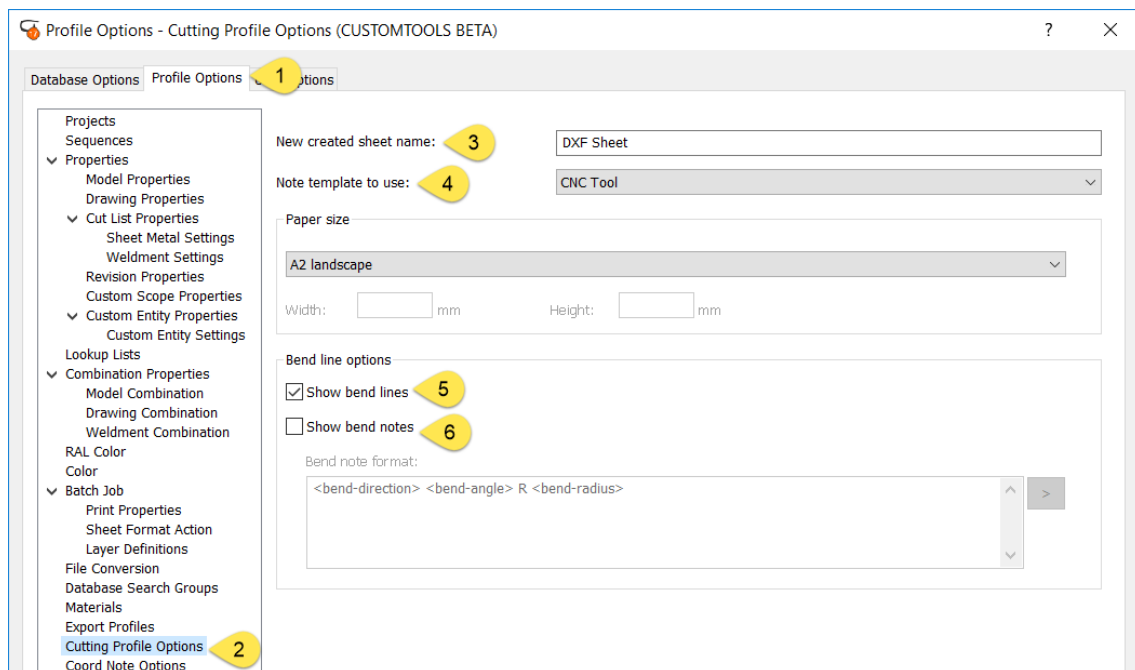
Using the Cutting Profile

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,

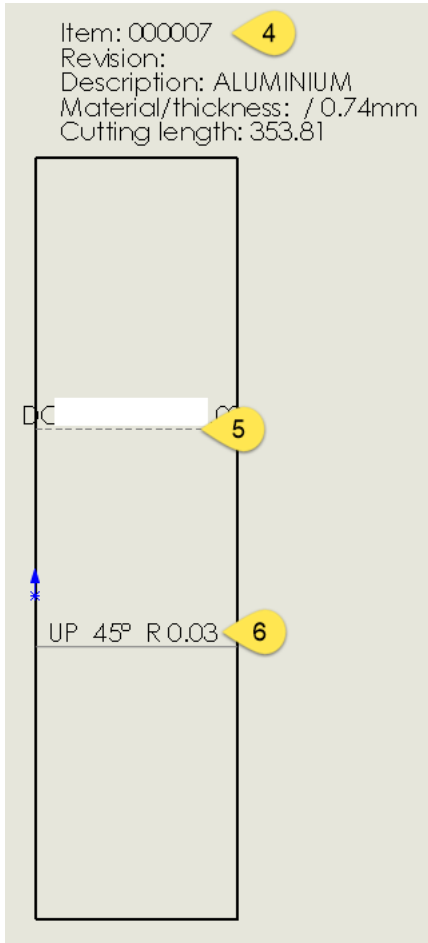
1. From the **Profile Options** tab,
2. Select **Cutting Profile Options**, from the tree view,
3. Enter the name of the drawing sheet created by the **Cutting Profile** in **New created sheet name**.



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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.

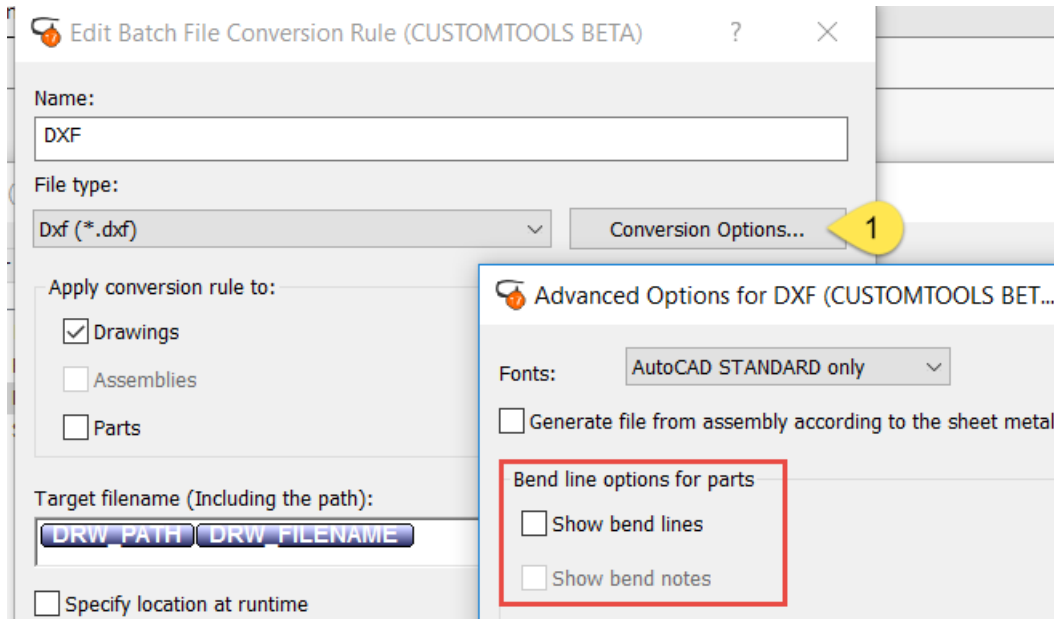


1. Select **Show bend notes** to insert the bending information,



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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.



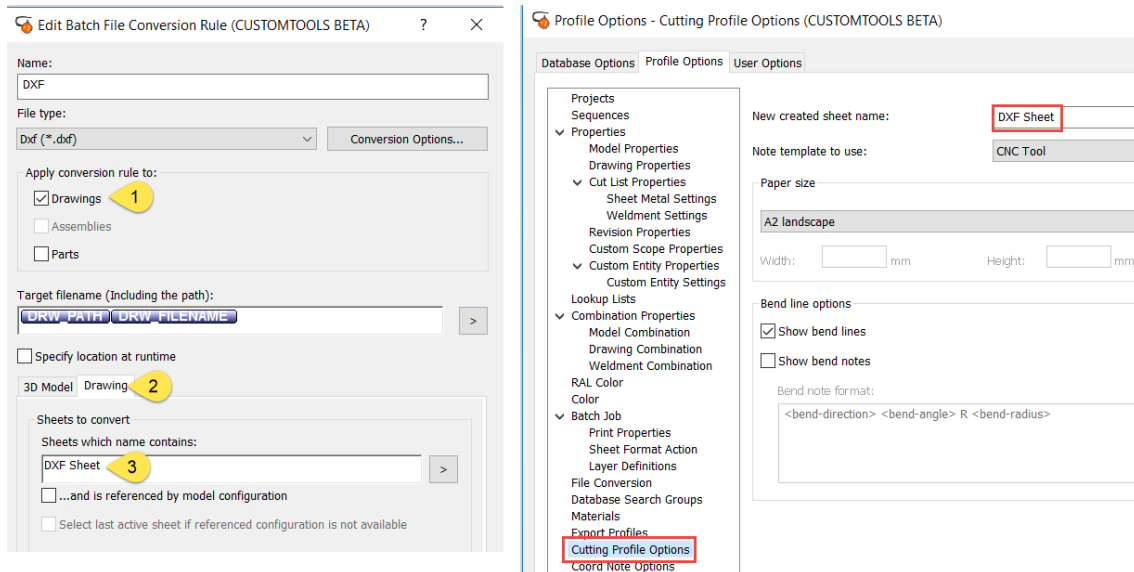
NOTE: The **Show bend lines** and **Show bend notes** can also be used if you are creating DXF directly from the parts. The Bend lines options are available in the **Conversion Options** of the conversion rule.

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**.



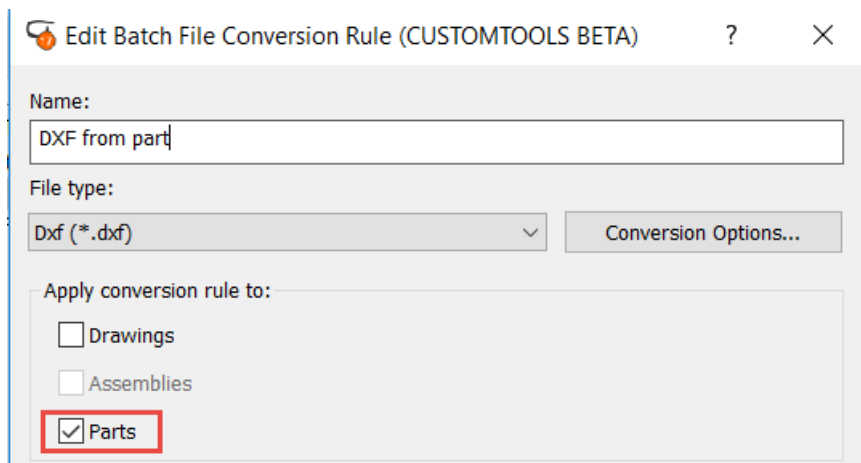
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Edit/create a **File conversion rule**. From the **File type**, select **DXF** or **DWG**.

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**.

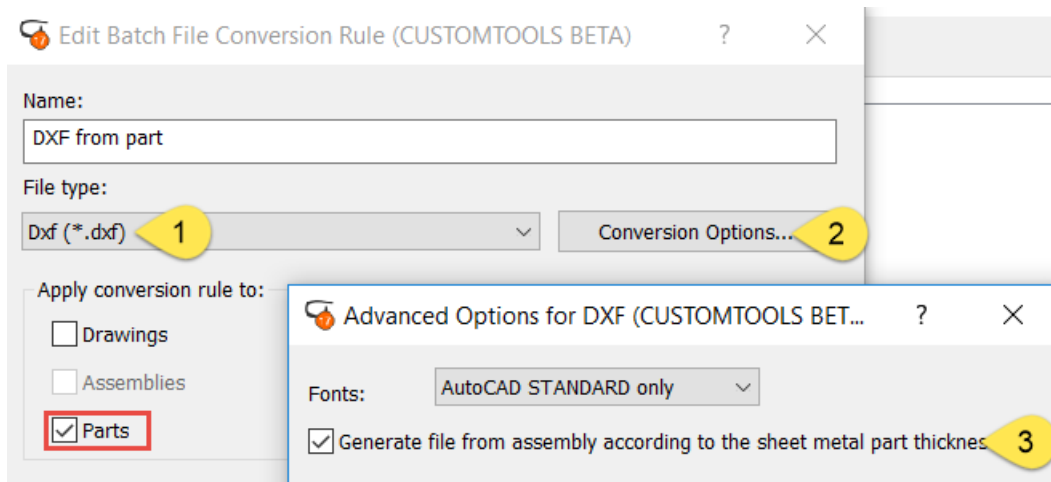


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Merging sheet metal parts into the same DXF files



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

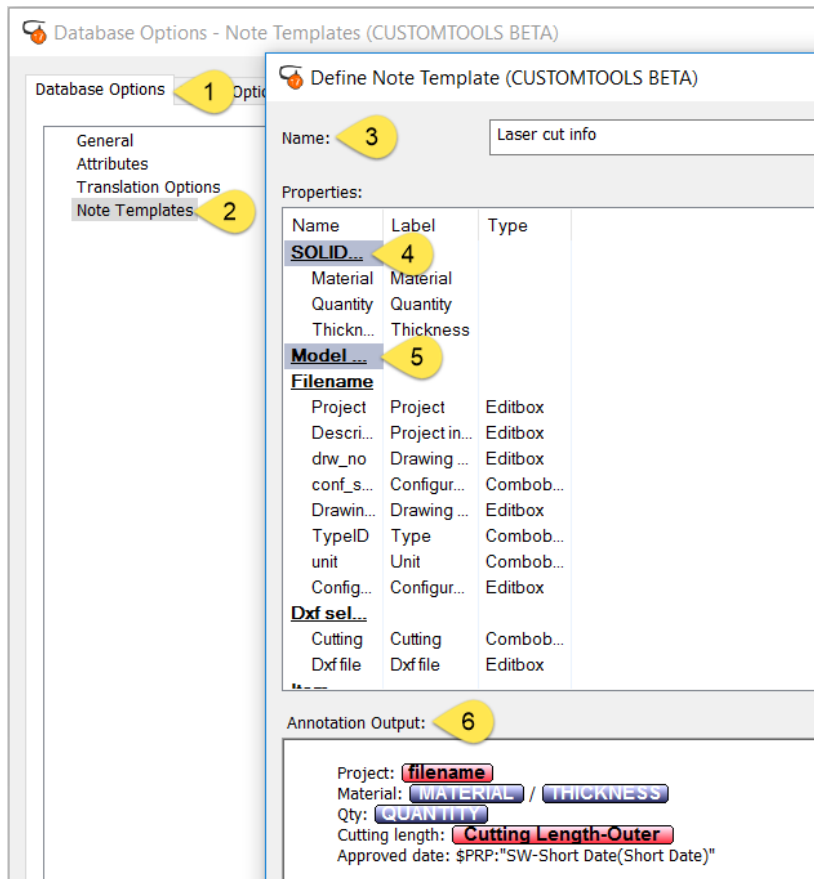




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DXF or DWG files created by CUSTOMTOOLS can be customized with notes containing property values.

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



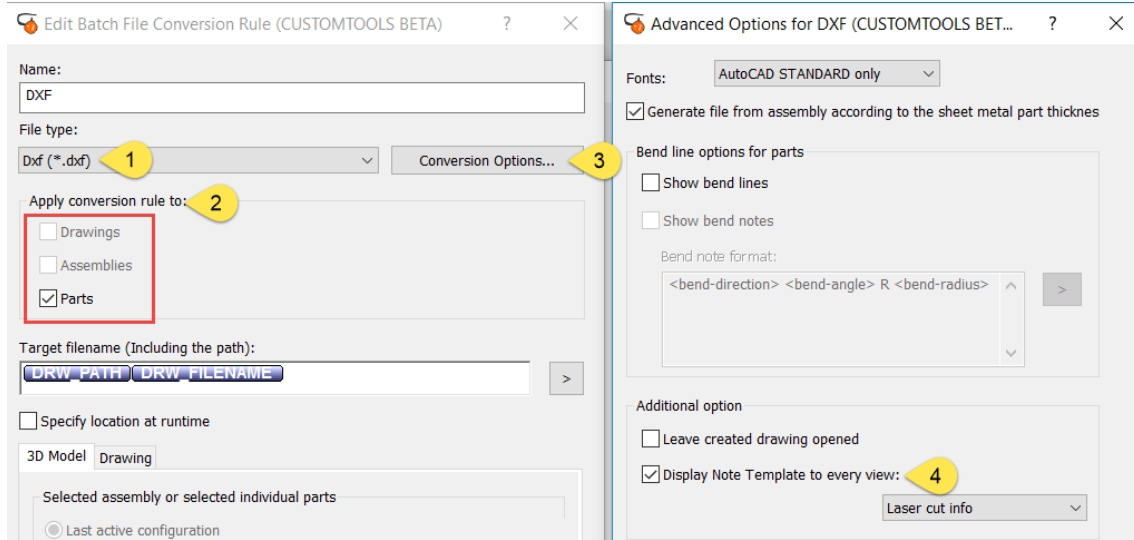
1. Activate the **Database Options** tab,
2. Select **Note Templates**,
3. **Name:** Defines the name of the note template as it will appear in the **Conversion Options** of the **File conversion rule** if a DXF or DWG type has been selected,
4. **SOLIDWORKS:** Inserts a SOLIDWORKS attributes available: 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**.



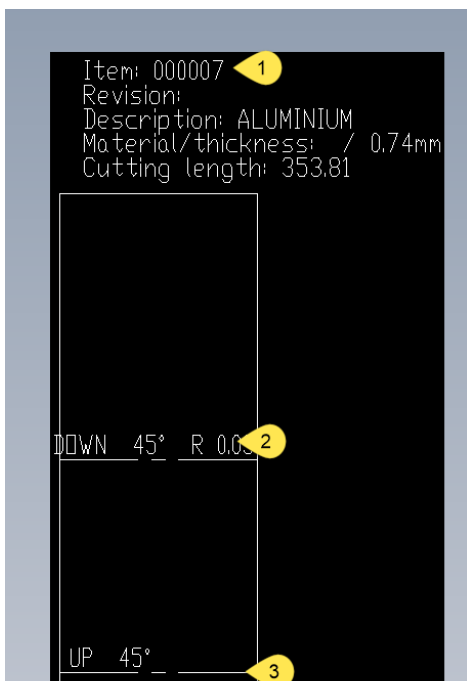
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To associate a **CustomNote Template** to a **File conversion rule**, open/edit a file conversion profile.

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

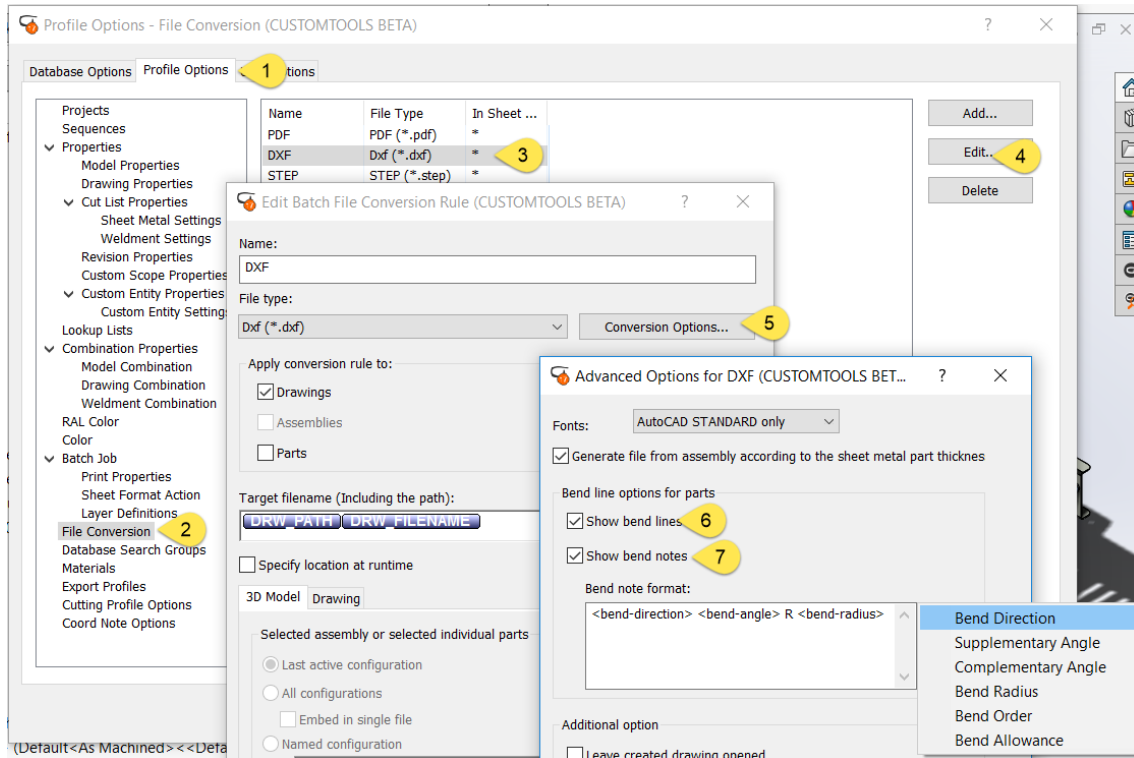




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Bend lines and Bend notes can be inserted in the output DXF files created by CUSTOMTOOLS.

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

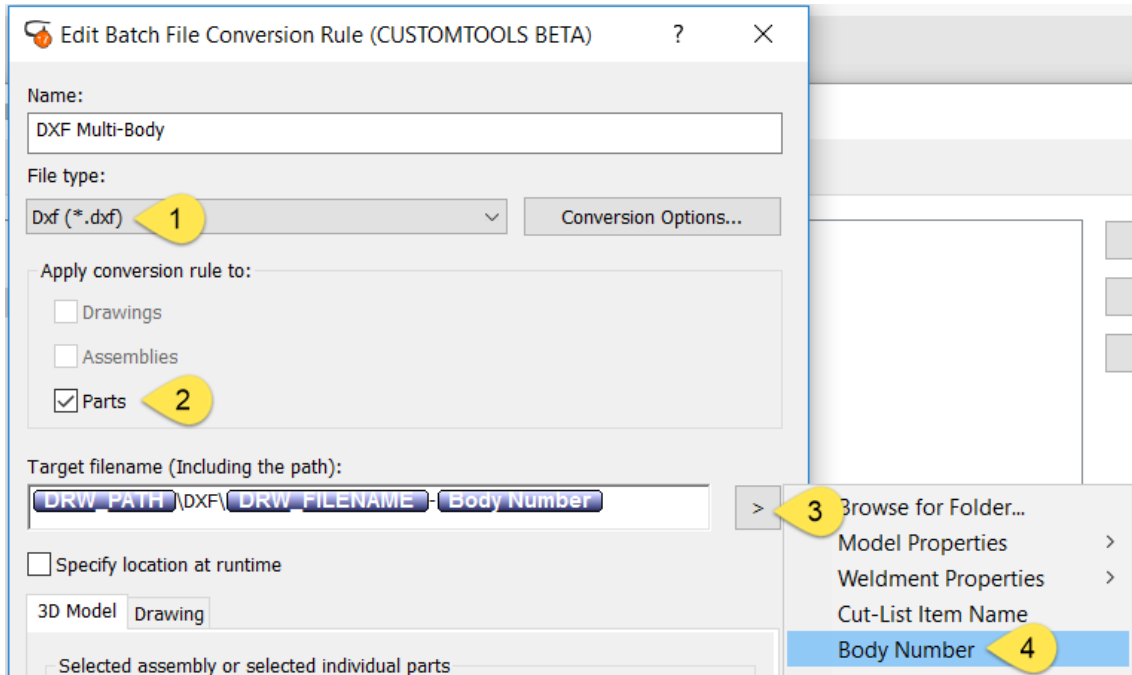
1. From the **Profile Options** tab,
2. Select **File Conversion**,
3. Select a profile,
4. Click **Add**,
5. Click on the **Conversion Options**,
6. Select **Show bend lines** from the **Bend line options** for part to insert bend lines in the created DXF,
7. Select **Show bend notes** to activate the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



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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.