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**CUSTOMTOOLS
EXCEL REPORTING**



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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?

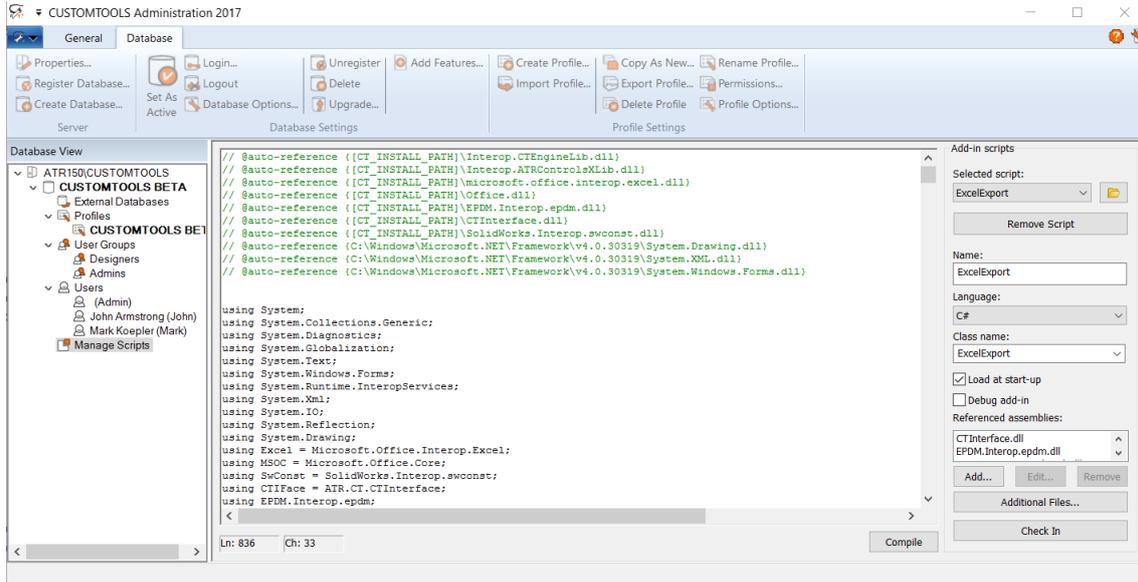
If an assembly is opened, then click on the Export icon. The **Export** dialog opens, then select **Excel profile**. The SOLIDWORKS assembly from which a report is to be generated, does not have to be opened in SOLIDWORKS. The Export dialog can also be accessed from the CUSTOMTOOLS search results.

Adding the Excel script

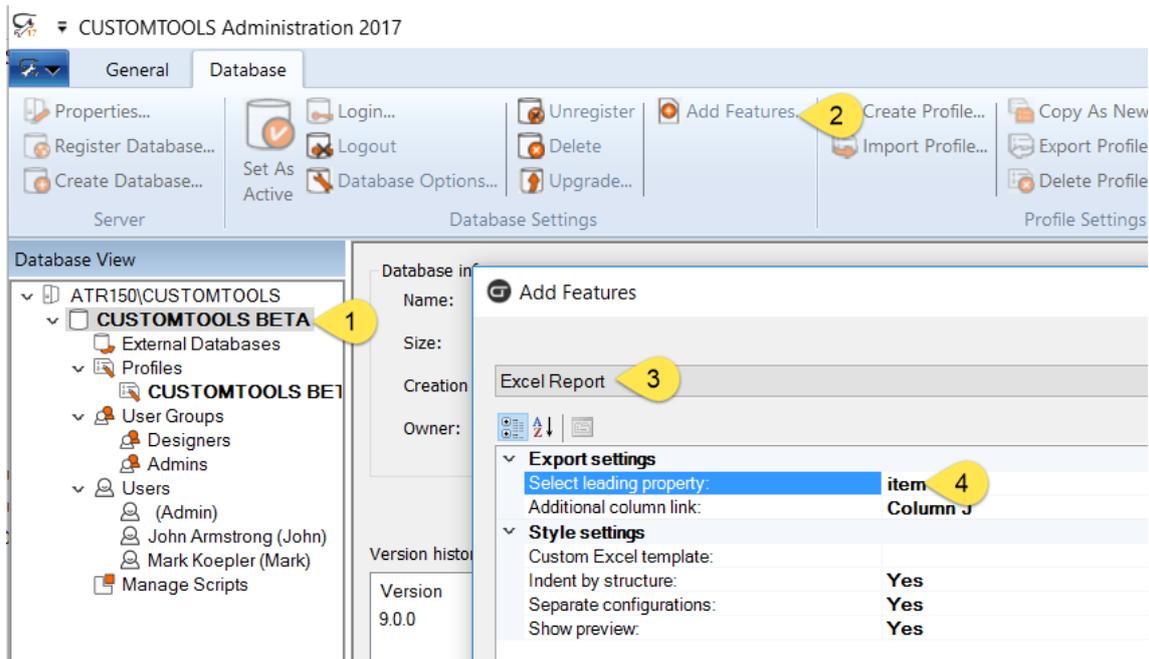
The out of the box Excel report comes automatically with the Mechanical engineer profile. If the Excel report is not available in the CUSTOMTOOLS profile, then it needs to be added.



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



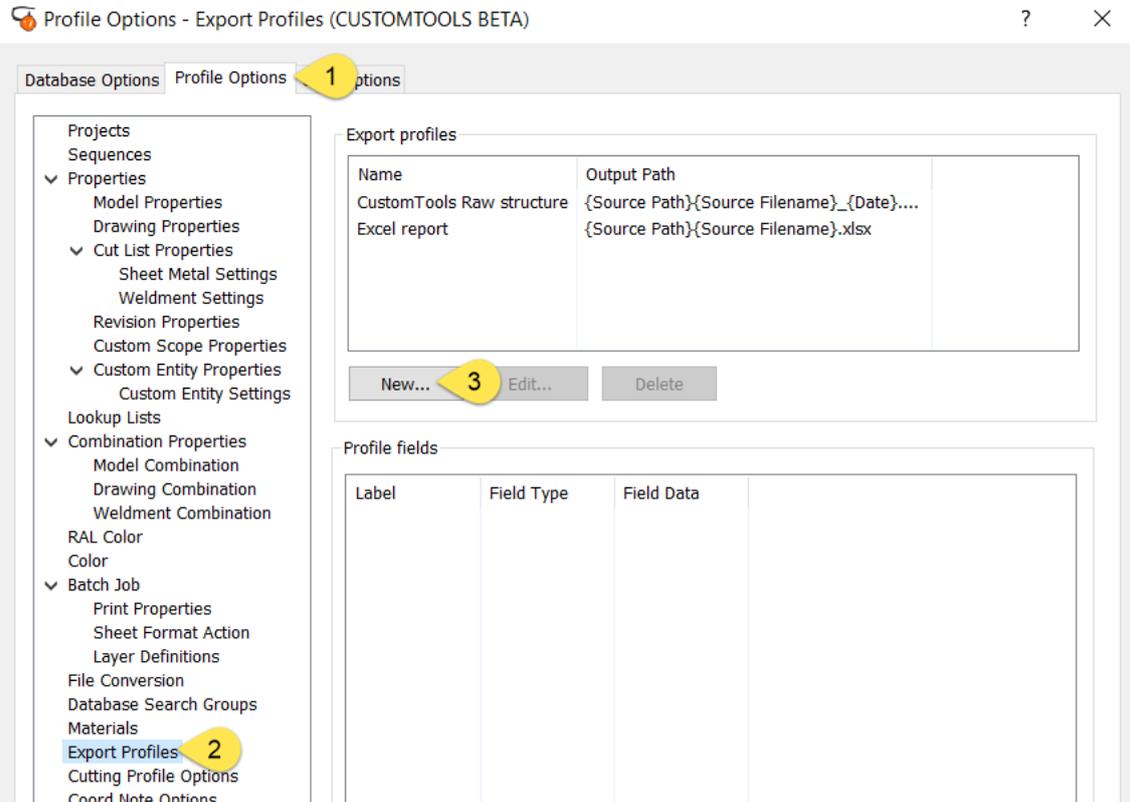
1. Select your CUSTOMTOOLS database (e.g. CUSTOMTOOLS Demo),
2. Click on **Add feature...** to open the **Add Features** dialog,
3. Select **Excel Report** from the drop down menu,
4. From the **Select leading property** under **Export settings**, select a custom property that will be used as the first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



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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.



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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.

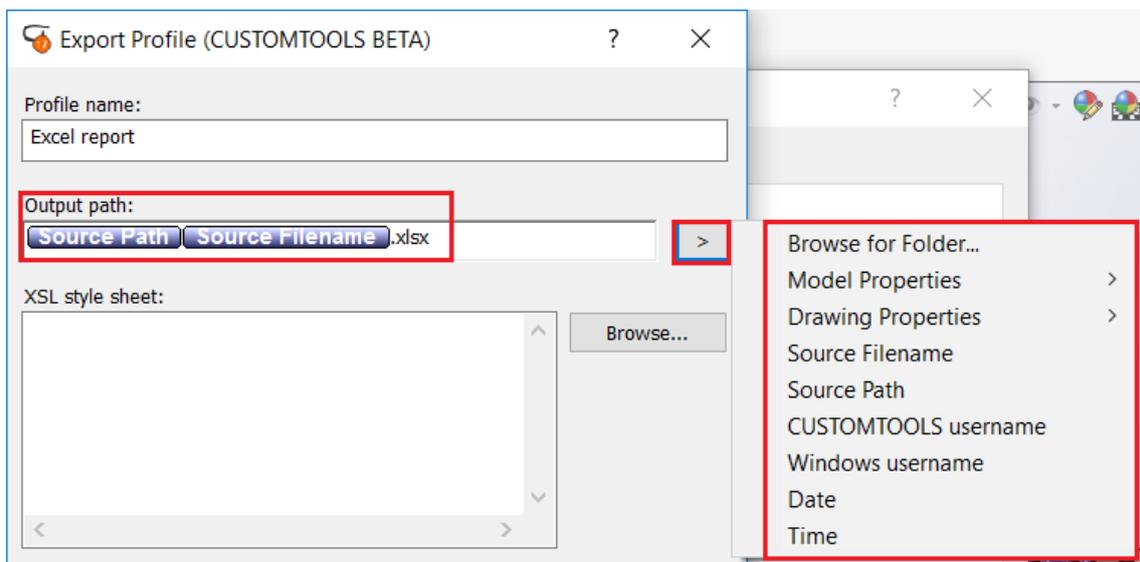


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5. **Run conversion rules:** Selects the file conversion rules that are used to convert SOLIDWORKS files along with the Excel report.
6. **Grouping:** Defines how configurations are handled in the Excel report:
 - a. **Display configurations of parts 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 referring assembly. Folders can also be manually entered and will be created during the export.



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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 referring assembly. Custom properties can also be associated with the file name as well as other options.

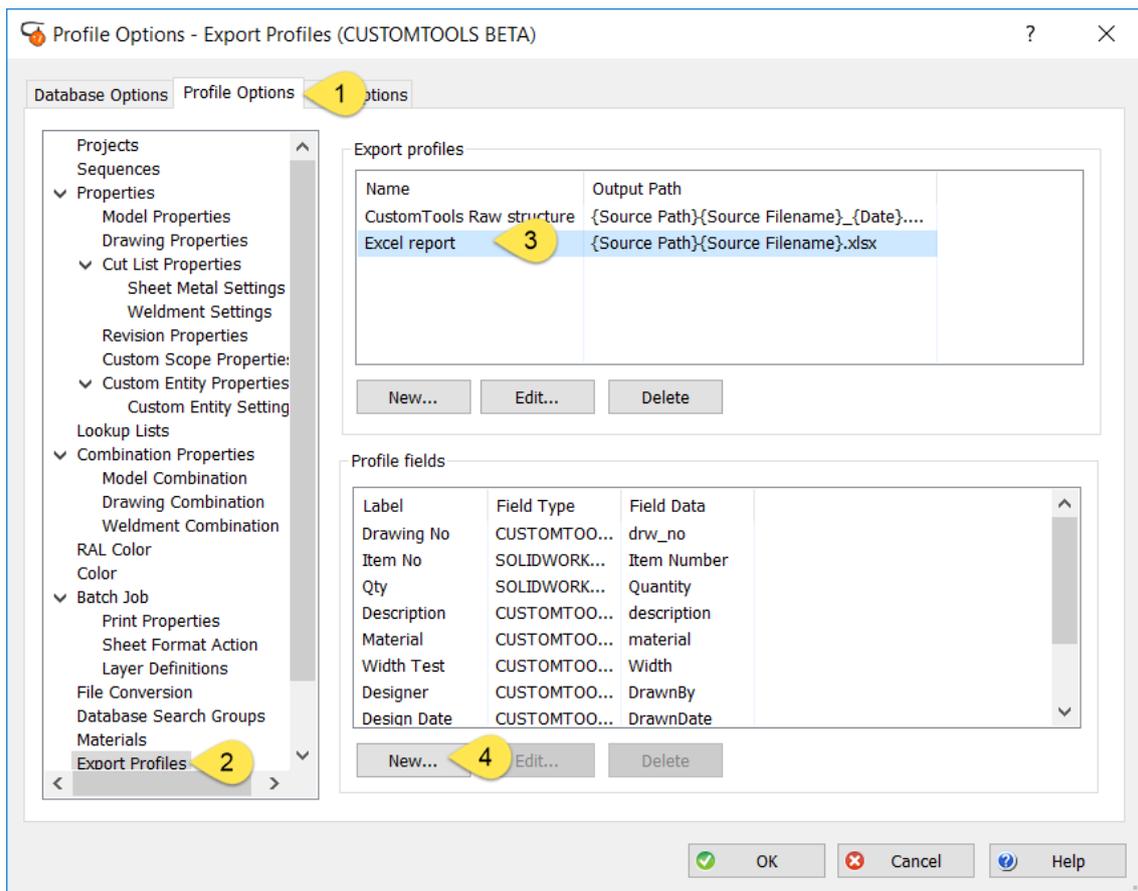


NOTE: The file extension also needs to be specified in the Output path (e.g. xlsx)

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,



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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.

Export Profile Field (CUSTOMTOOL... ? X

Field type: 1
CUSTOMTOOLS attribute

Field data: 2
description

Label 3
Description

Value

Options

Maximum length: 0

Compulsory field

OK Cancel

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.

Export Profile Field (CUSTOMTOOL... ? X

Field type:
SOLIDWORKS property

Field data:
Configuration
Filename
Item Number
Part Number
Quantity
Type

- 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.



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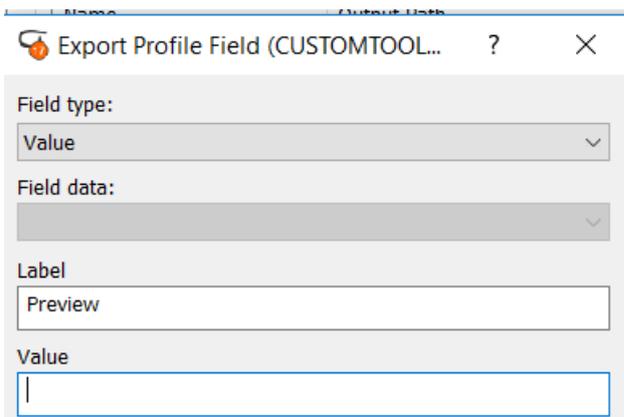
NOTE: If **Value** has been selected, then the **Field data** is disabled.

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

1	Item No (Sage)	Item No	Qty	Description	Preview
2	000001	0	1	CONVEYOR MAIN DRAWING	

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



Export Profile Field (CUSTOMTOOL... ? X

Field type:
Value

Field data:
[Disabled]

Label
Preview

Value
[Empty]

To insert a preview image,

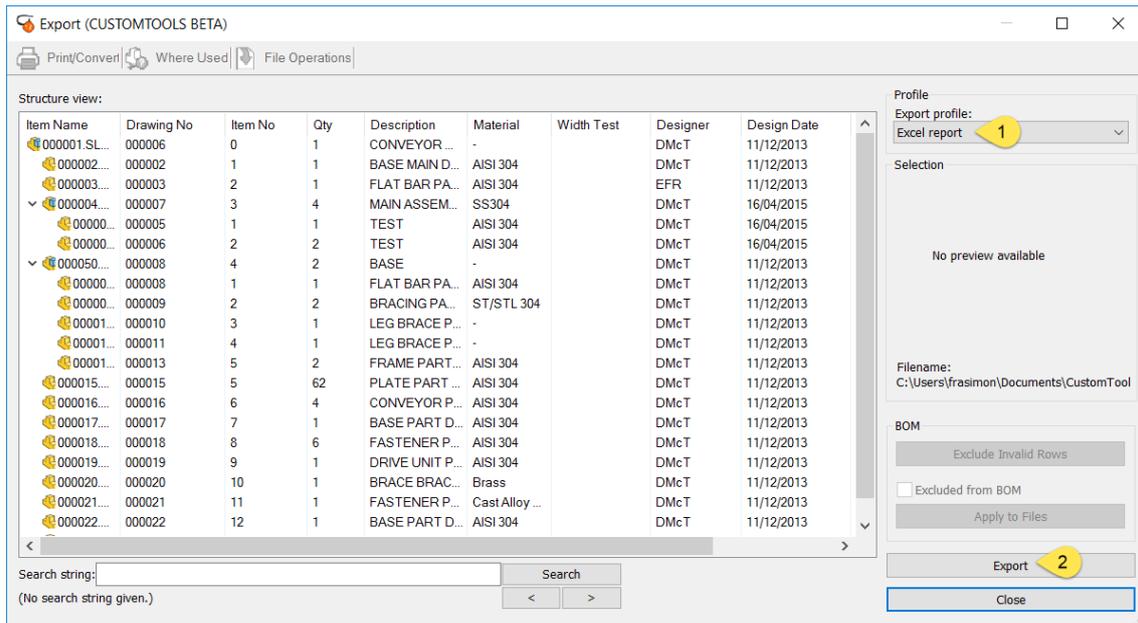
1. Select **Value** from **Field Type**,
2. Type **Preview** in **Label**.



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**.