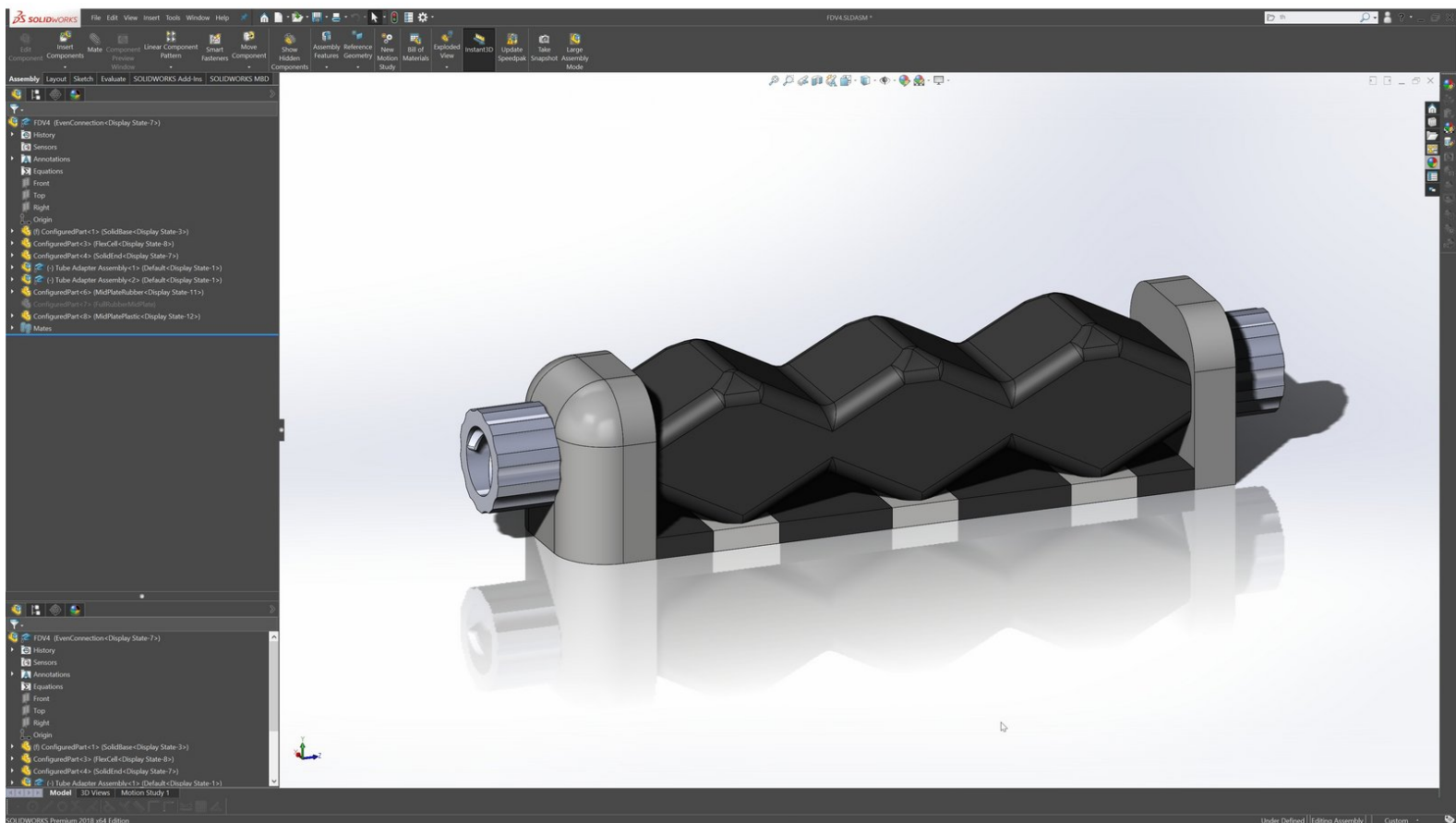# Multi-material .stl creation - Solidworks

This guide will go through the process of exporting multi-material .stl files for 3D printing.
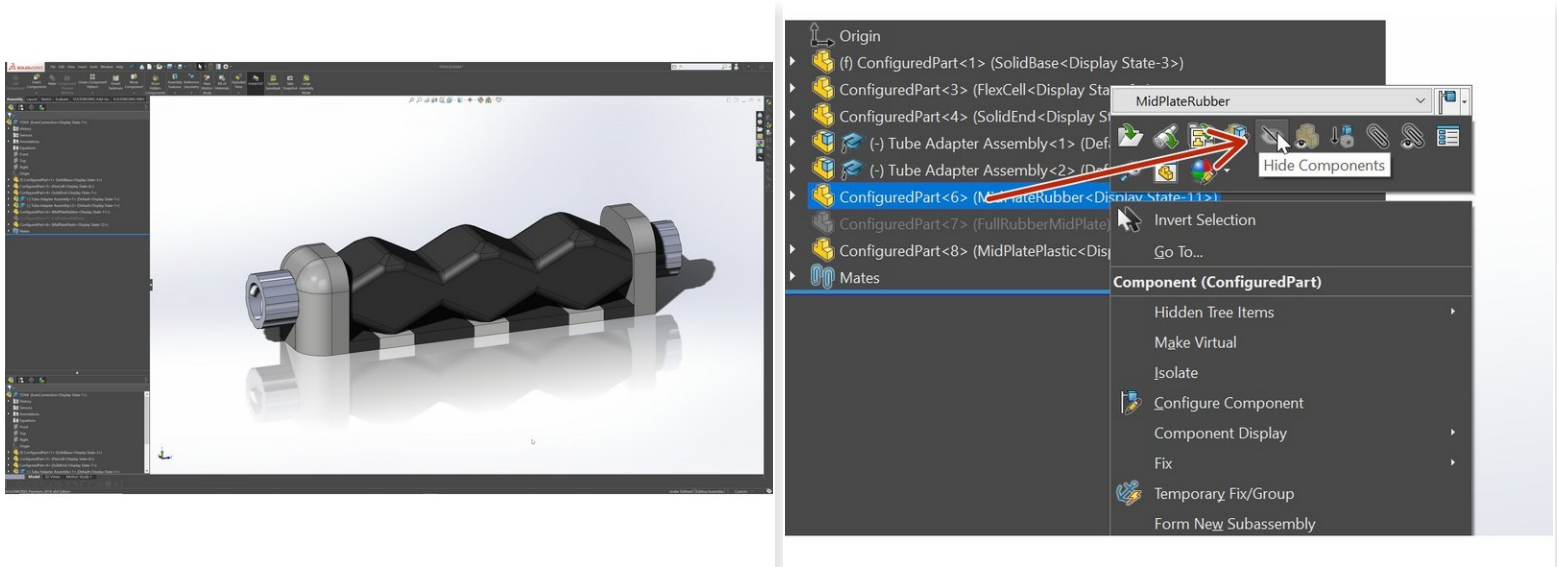
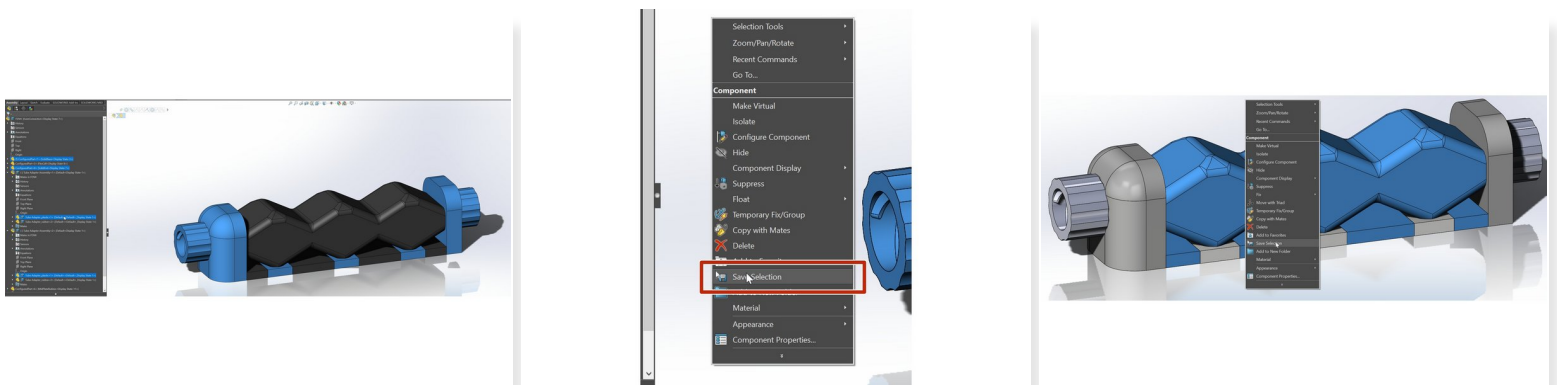**Written By: Jacob Andrew Garnett**

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Introduction

This guide will go through the process of exporting multi-material .stl files for 3D printing.

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Step 1 — Assemble printed parts

- Create a Solidworks Assembly that only has the parts you want to print.

- If a non-printed part is needed to align or setup the assembly then hide the part before continuing
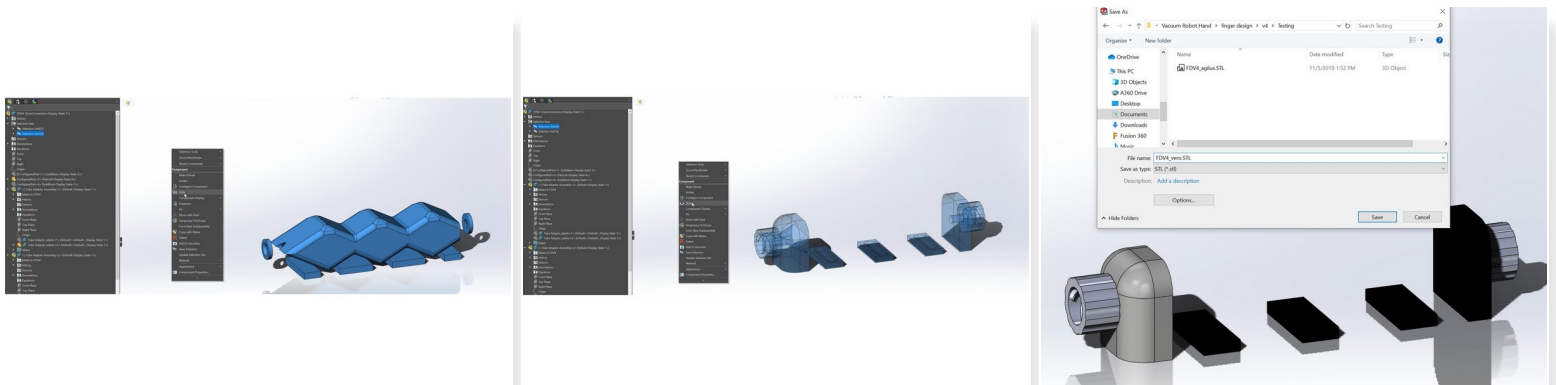
Step 2 — Create Selection Sets

- Select all parts which will be the same material.

- Save the currently selected parts

- Select and then save the selection for each additional material you will be printing in.

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Step 3 — Isolate and Export the 1st Material

- Open up selection sets and then select and hide all of them except for one.

- "Save As" the assembly.

- Select .stl file type and include the material name as a suffix to the file name.

- Open Options and make sure "Do not translate..." and "Save ... in a single file" are both checked.

Step 4 — Isolate and Export Additional Materials

- Hide the material selection set that was just exported.

- Show the next material selection set to be exported.

- Repeat the "Save As" process.

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