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SOLIDWORKS FOR MULTI-MATERIAL BUILDS—BEST PRACTICES

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INTRO

Stratasys Connex multi-material 3D printing gives you the ability to create models with multiple materials in a single build. This document outlines how to prepare SolidWorks CAD files for multi-material PolyJet prints.

FILE PREREQUSITES

SEPARATE SOLID BODIES

Each region of a part that will be 3D printed in a separate specified material must be represented by a separate solid body in a SolidWorks part file, or by a separate part in a SolidWorks assembly file.

NO OVERLAPPING GEOMETRY

Regions of a part or assembly that are going to be 3D printed in different materials must not have any overlapping geometries.

STARTING WITH MULTIBODY PART IN SOLIDWORKS

SKIP THIS STEP IF YOU HAVE EACH MATERIAL REPRESENTED BY A SEPARATE PART IN A SOLIDWORKS ASSEMBLY.

If starting with more than one solid body in a single SolidWorks part file, create an assembly from your multi-body part file by following these steps:

1. SAVE SOLID BODIES

Right click Solid Bodies folder in the feature tree and select Save Bodies.
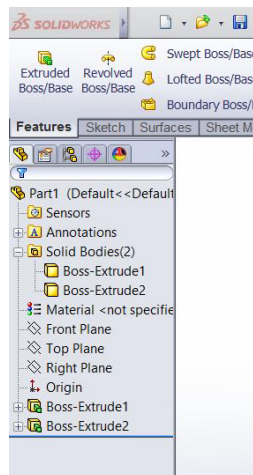
2. NAME PARTS

Fig. 1

Assign names to each of the solid bodies. You can do this manually for each part by double clicking each file in the Resulting Parts list shown in Fig. 2 and entering the file name in the resulting save as pop-up window. Alternatively, SolidWorks can automatically assign a name to each part by clicking the Auto-assign Names button as shown in Fig. 2.

3. CREATE AN ASSEMBLY

Click the Browse button in the Create Assembly section shown in Fig. 2 to select a file directory for your new assembly file and enter the file name in the pop-up window just as you would normally save an assembly. Click the Save button when you have chosen a location and named the new assembly.

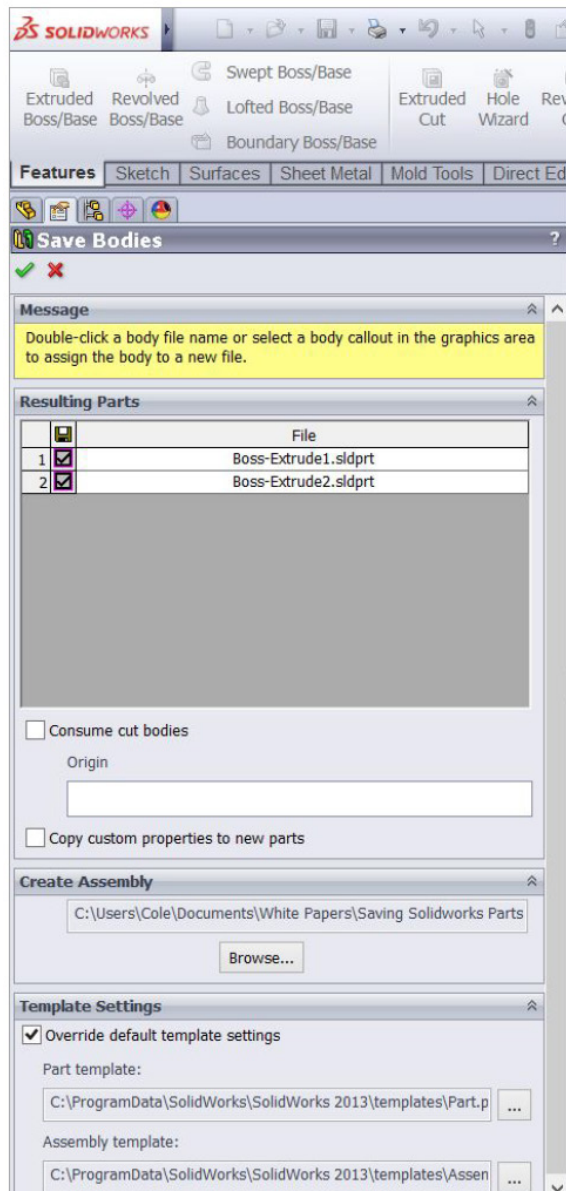4. Press the green checkmark to create the assembly and complete the task.

Fig. 2

SAVING STL FILES FROM ASSEMBLY

1. Open your assembly file.

2. SAVE AS STL

Select File > Save As. Select STL (*.stl) from the Save As type dropdown menu as shown in Fig. 3.

3. EXPORT OPTIONS

Click the Options button to open the Export Options pop-up window shown in Fig 4. Populate the fields with the options and values in Table 1. These are general guidelines and may be varied to suit your needs.

4. Click OK to save selected options and return to Save As window.

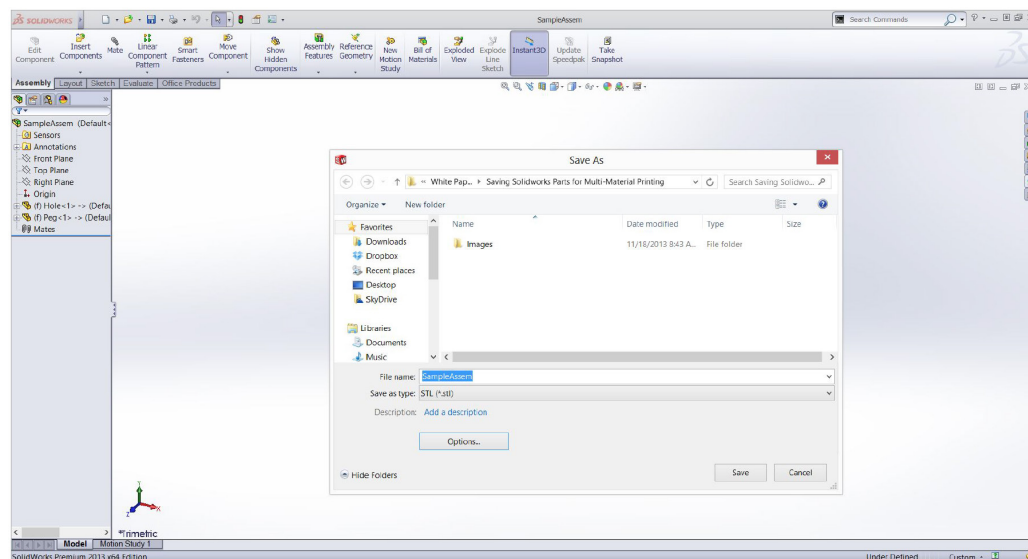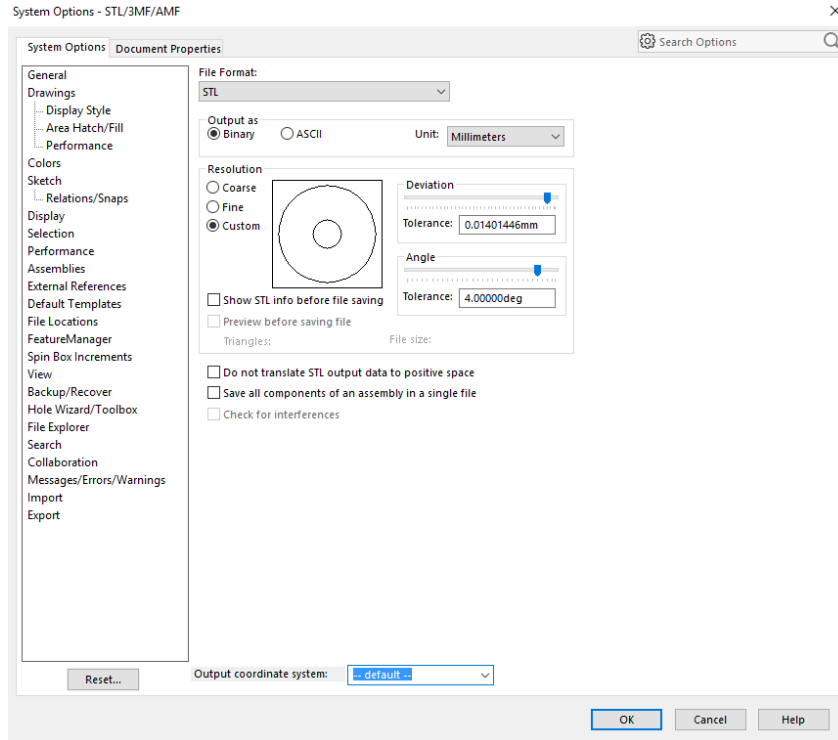| FIELD	VALUE
Output As:	Binary
Resolution:	Custom
Deviation Tolerance:	0.014 mm (this can change depending on part geometry)
Angle Tolerance:	4°
Save all components of an assembly in a single file.	Deselected

Table 1

Fig. 3

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Fig. 4

3D PRINTING WITH GAPS

When 3D printing a part with bodies that you intend to remain separate for motion, linkage, or other purposes, there must be a gap of at least 0.020 inches between the separate solid bodies to ensure that removable support material is 3D printed between the two bodies and the two separate bodies don't fuse together during the build. This number can be reduced iteratively by trial and error, but it's a reliable starting point.