Custom Materials in SOLIDWORKS

Although SOLIDWORKS has an extensive material library, users often need to create custom materials. The ability to add custom materials enables users to calculate an accurate mass or simulation of a part or assembly. The following steps show the processes to create a new material in SOLIDWORKS.

Step 1: How to change the material in a part or simulation.

In a part, materials can be changed by going to the design tree, right clicking Material, and selecting Edit Material.

In a simulation, materials can be changed without opening the individual part. Simply right click on the part in the design tree and select Edit Material.
Step 2: Create a new library.

Once the Martial dialogue box is open, scroll to the bottom of the material tree. Right click on custom material and select New Library, then create a New Category within the Library. This example uses a library called Custom Plastics and a category called Non-Linear Plastics. Click Apply.
Step 3: Create a new material.

New materials can be created by copying and pasting a similar material, or entering the material properties manually. Existing SOLIDWORKS materials cannot be edited. To copy a material, simply click on a material in the existing SOLIDWORKS Library, right click and select copy. Right click in the new category and select Paste. Click Apply.

Step 4: Set the new material properties.

Select the model type, and enter the values by double clicking the value box. Click Apply.
Congratulations! You have now successfully created a new material in SOLIDWORKS. If you have any further questions, contact GoEngineer Tech Support at 888-559-3242 or support@goengineer.com.