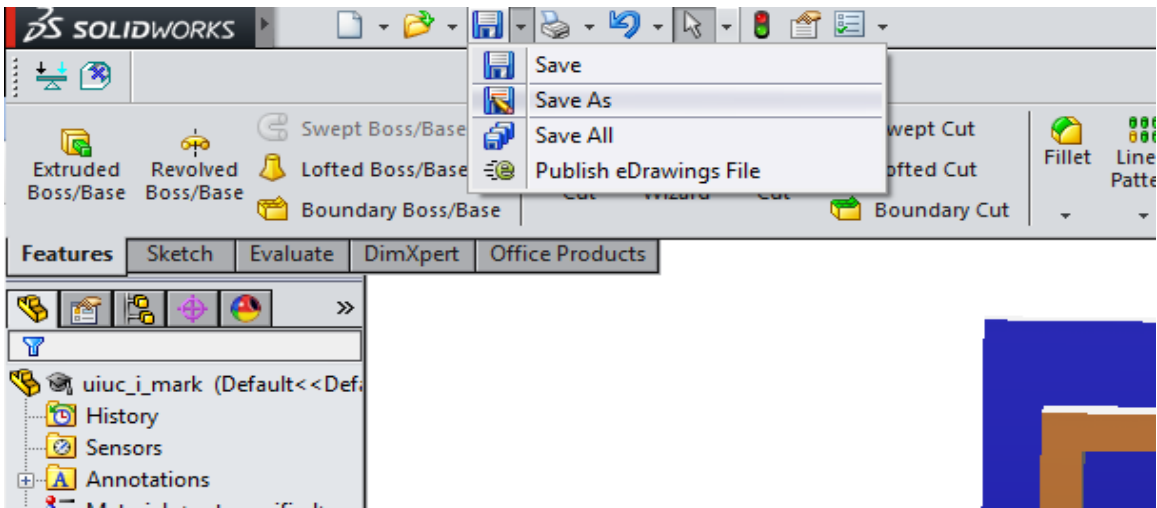

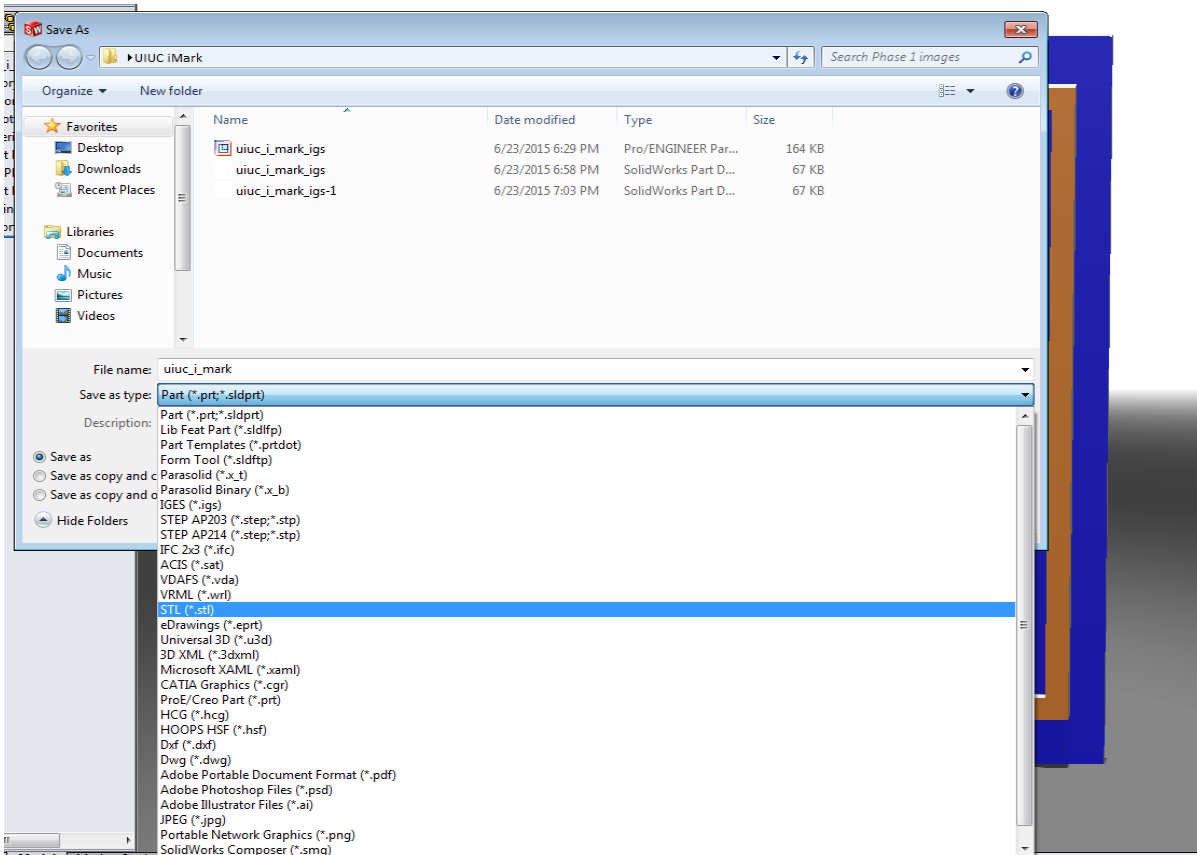


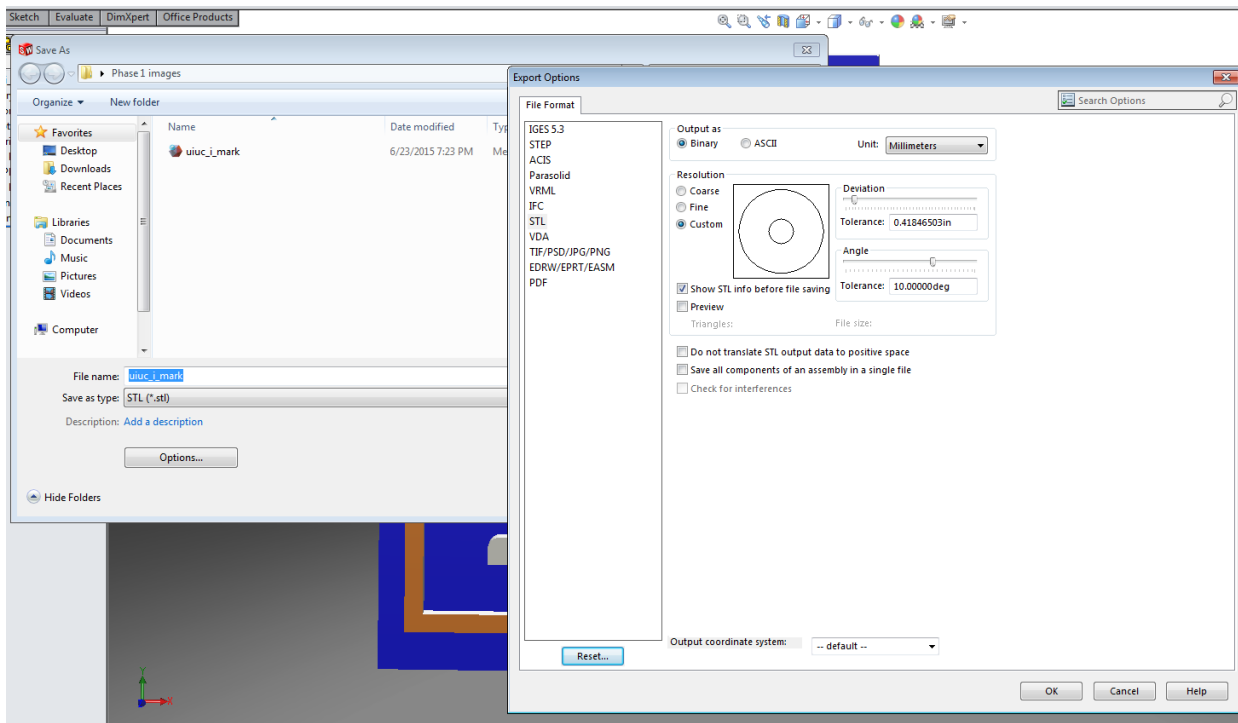
SOLIDWORKS



Open the **Save Options Menu** by selecting . Select **Save As**.

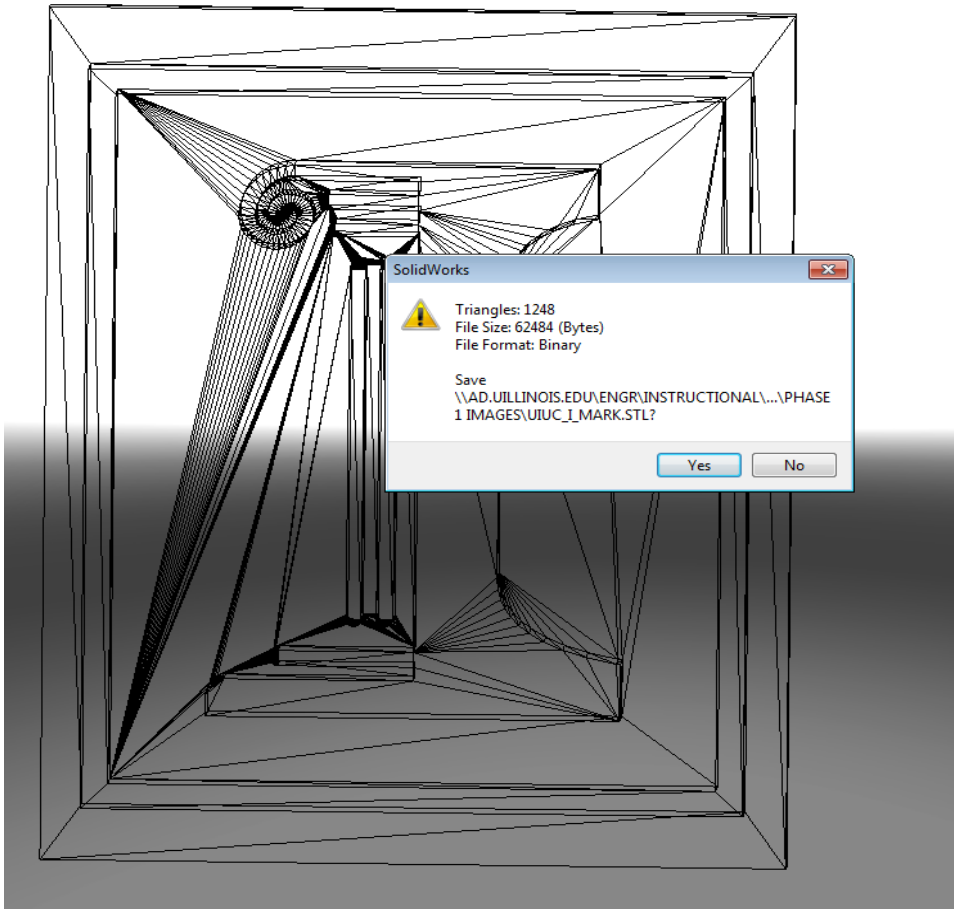


The **Save As** dialogue box will appear. In the **Save as type** dropdown menu, choose **STL (*.stl)**.



The **Export Options** dialogue box will appear. Under **Output as**, select **Binary** and the appropriate units in the **Unit** dropdown menu. Under **Resolution**, select **Fine**. Check **Preview** and **Show STL info before file saving** (This provides a convenient way to inspect your saved geometry and file size before you export the STL).

If you are saving an Assembly, the default behavior is to save each component into a separate STL file. If you wish to print your assembly en-bloc, select **Save all components of an assembly in a single file**.



Inspect your saved geometry and your file properties. Note that maximum uploaded file size is 25 Mb. Pick **Yes** to save your file.

STL files are unit-less. Please note the units used in designing this part and make the appropriate selection when creating a Ticket for your part.

Advanced Instructions

- **Deviation Tolerance** is the maximum deviation between the designed part and the STL representation of the part. Since an STL file approximates your part's surface geometry using tessellated triangles, there is a tradeoff between accurate representation of curves and the number of triangles used. The larger the number of triangles, the larger the file size.
- **Angle Tolerance** controls the maximum angle between the normal vectors of each triangle. The smaller this value, the more dense the tessellation in curved surfaces. Increases both resolution and file size.
- If you are printing an **Assembly** in SOLIDWORKS, please ensure that the configuration of your assembly is according to your design intent. Interfering components will be printed conjoined into one piece and moving components require the appropriate allowance with other.