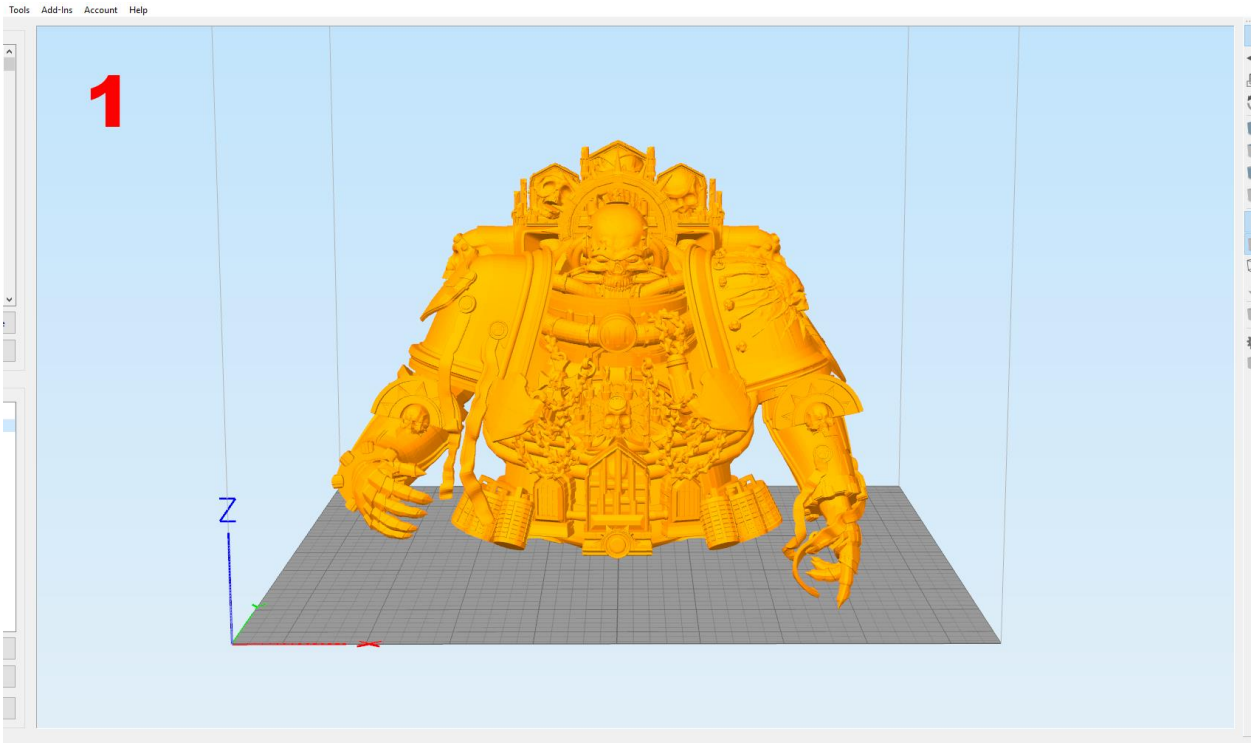
Simplify3D User Guidelines

These guidelines will help you configure correctly your Simplify3D slicing program. We will walk you through all the steps.

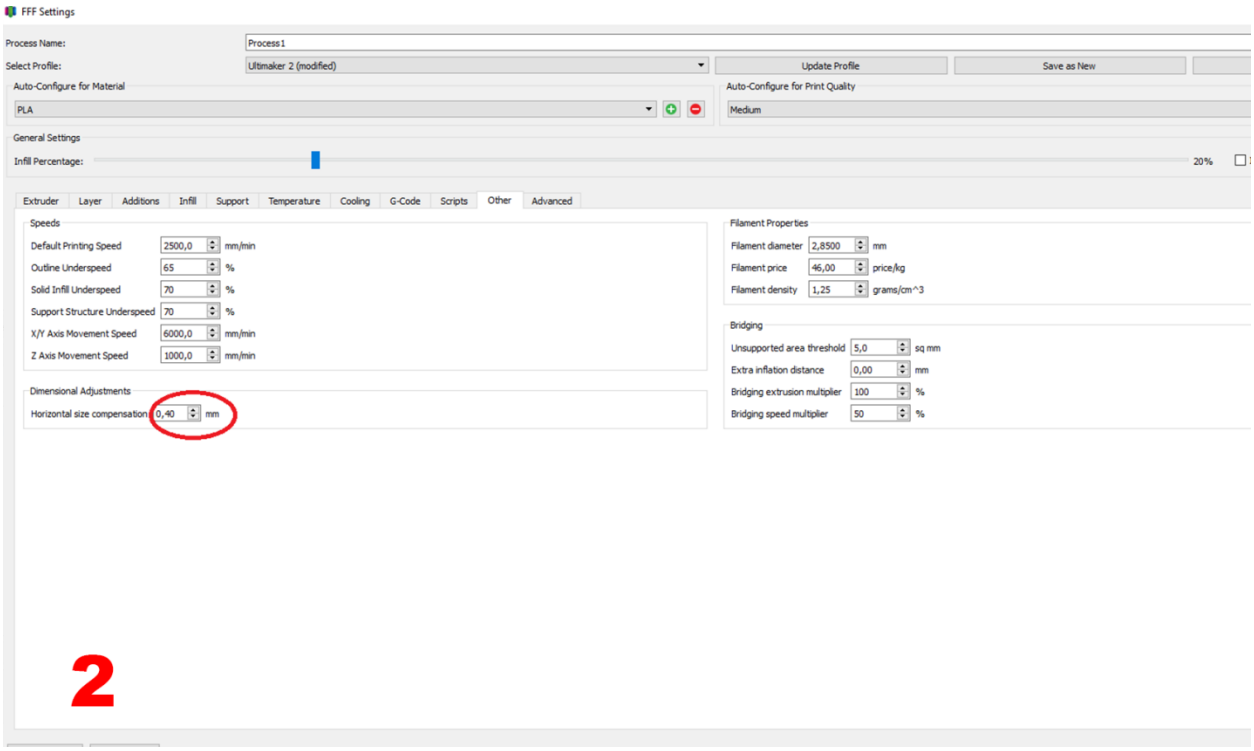
We have recently started receiving a lot of messages and emails concerning the printing of a 3D model using Simplify3D software.

These guidelines will assist you in configuring your slicing program correctly and will ensure an enjoyable 3D printing experience.

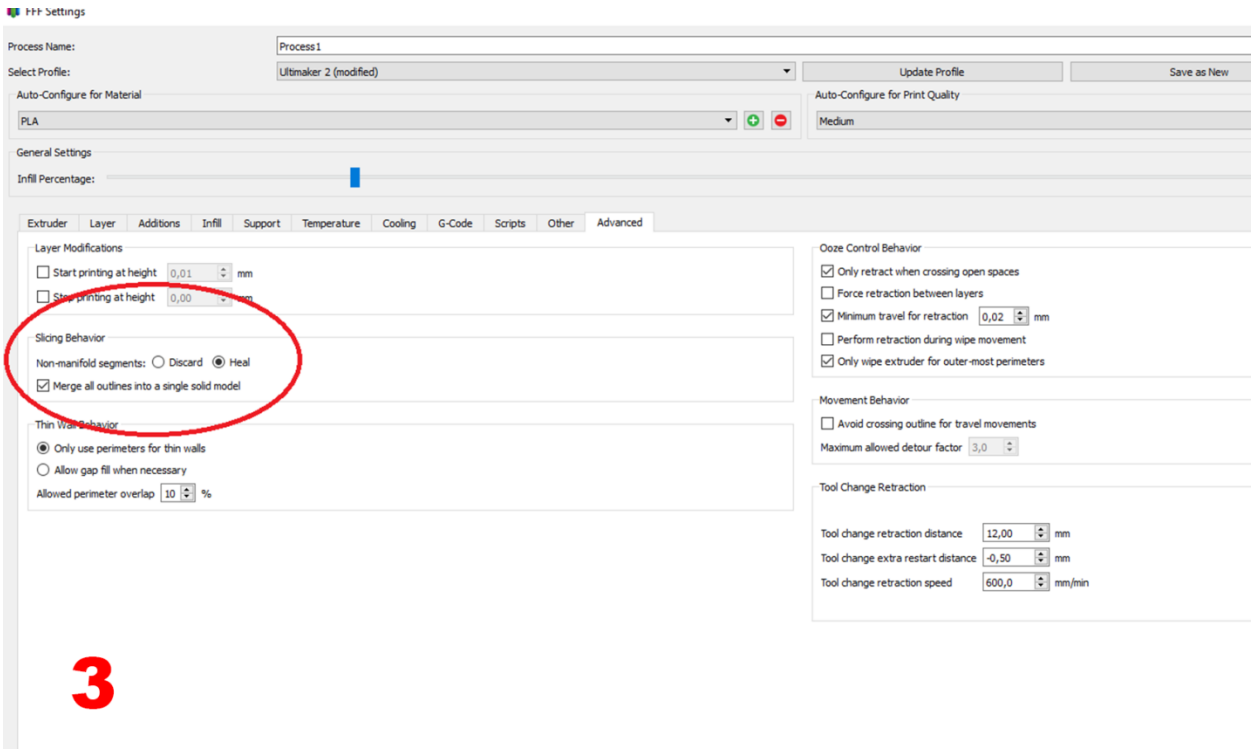
Step 1. Upload or drag your 3D model into the slicing program.



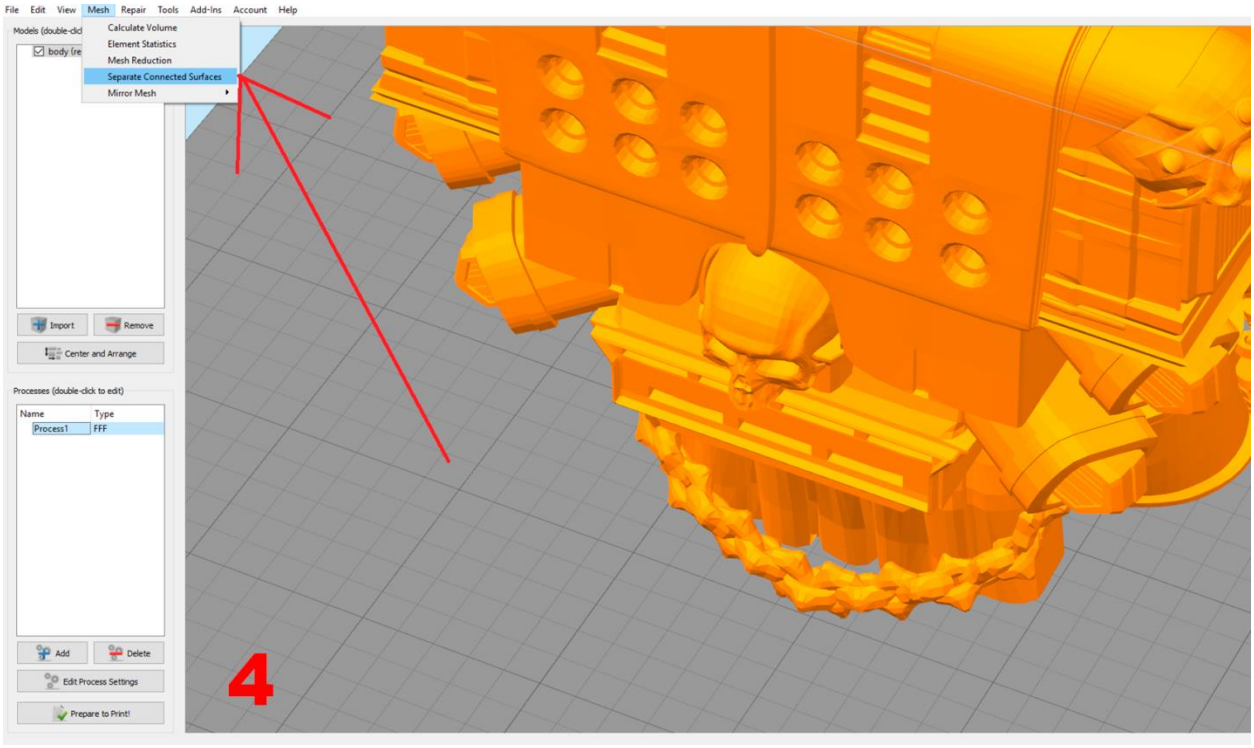
Step 2. Set *Horizontal size compensation* at **0.40mm**.



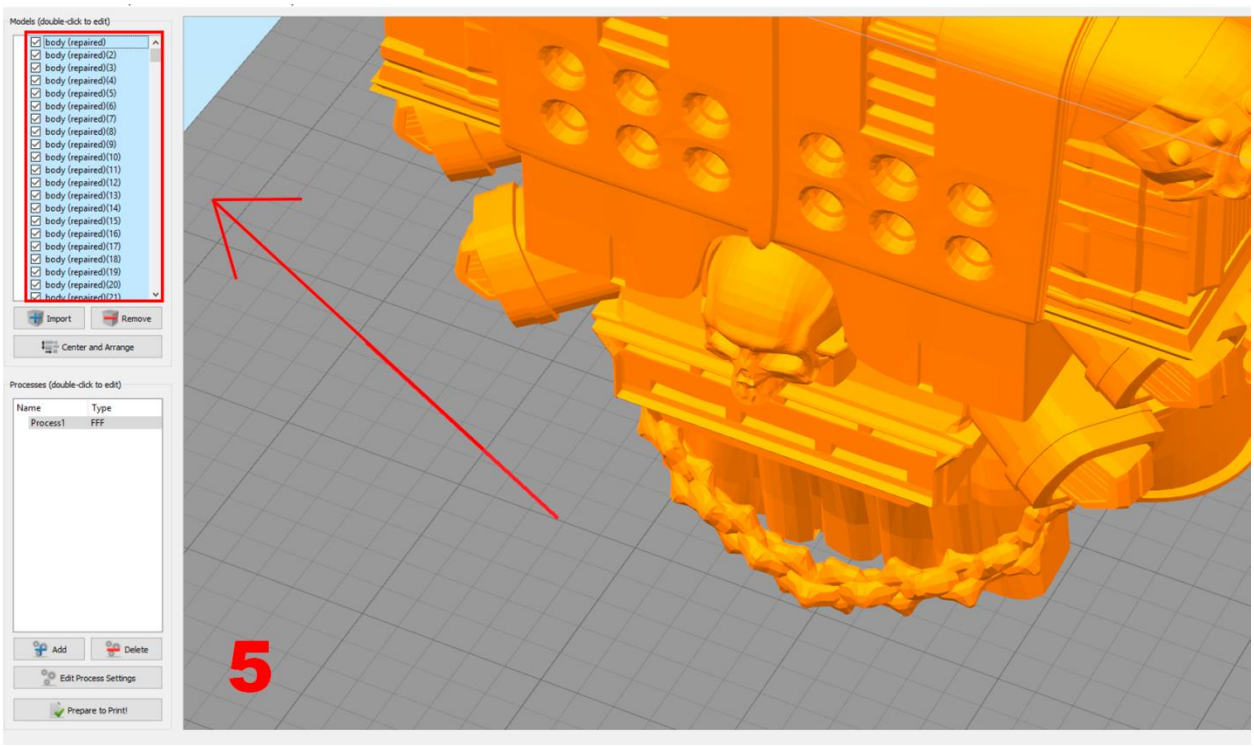
*** Step 3.** Click *Edit Process Settings*, choose *Advanced*, and put the check mark in the box next to *Merge all outlines into a single solid model* feature. Press *OK*.



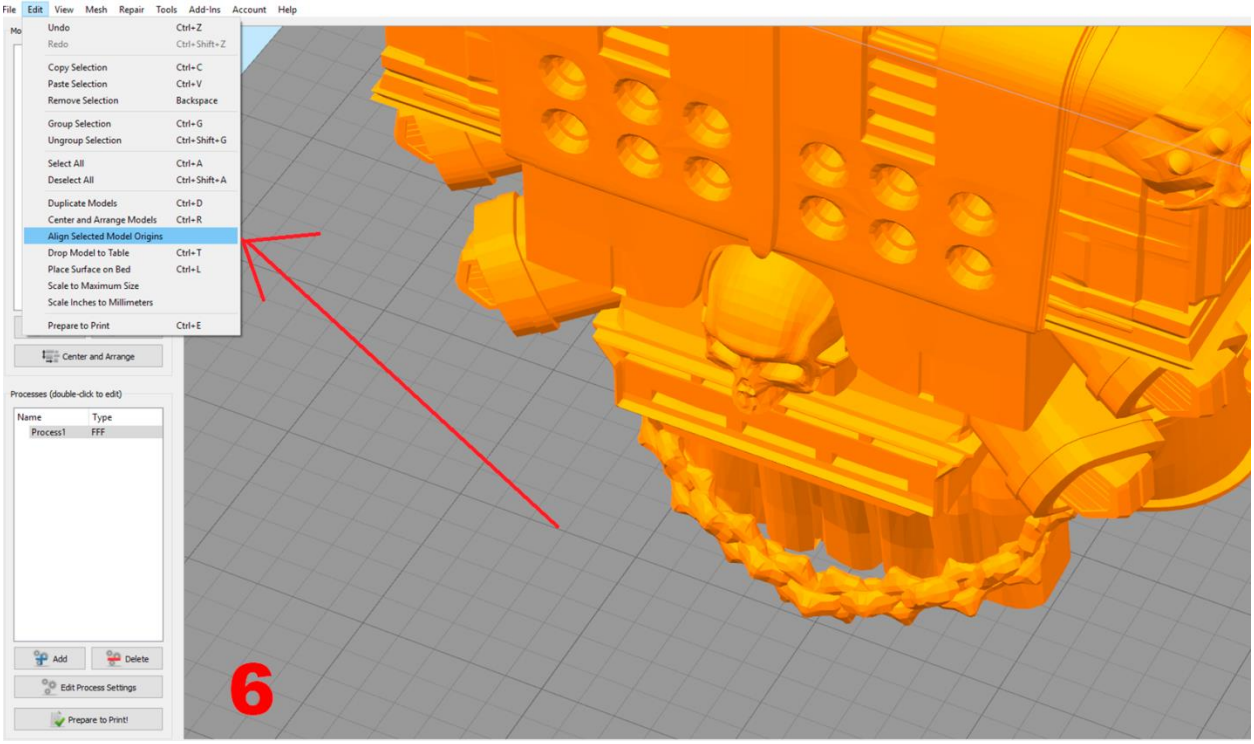
Step 4. Select *Mesh > Separate Connected Surfaces*



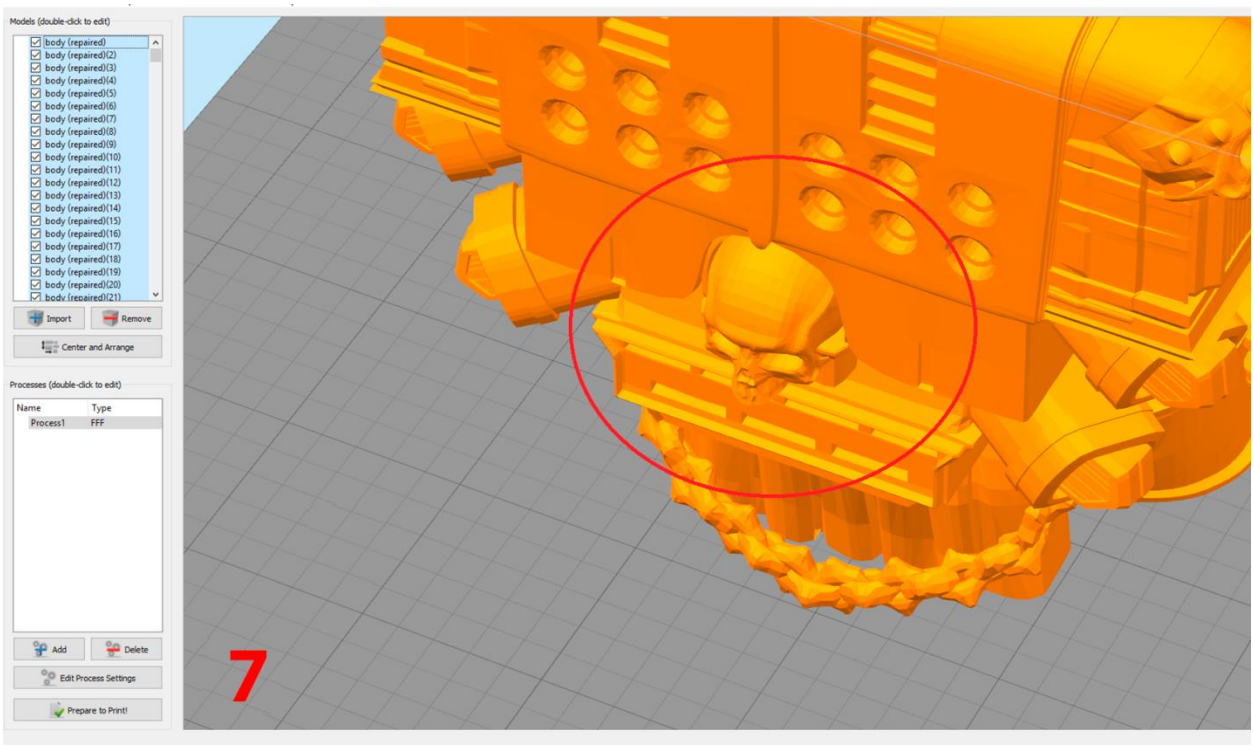
Step 5. Select all edits from the first through the last.



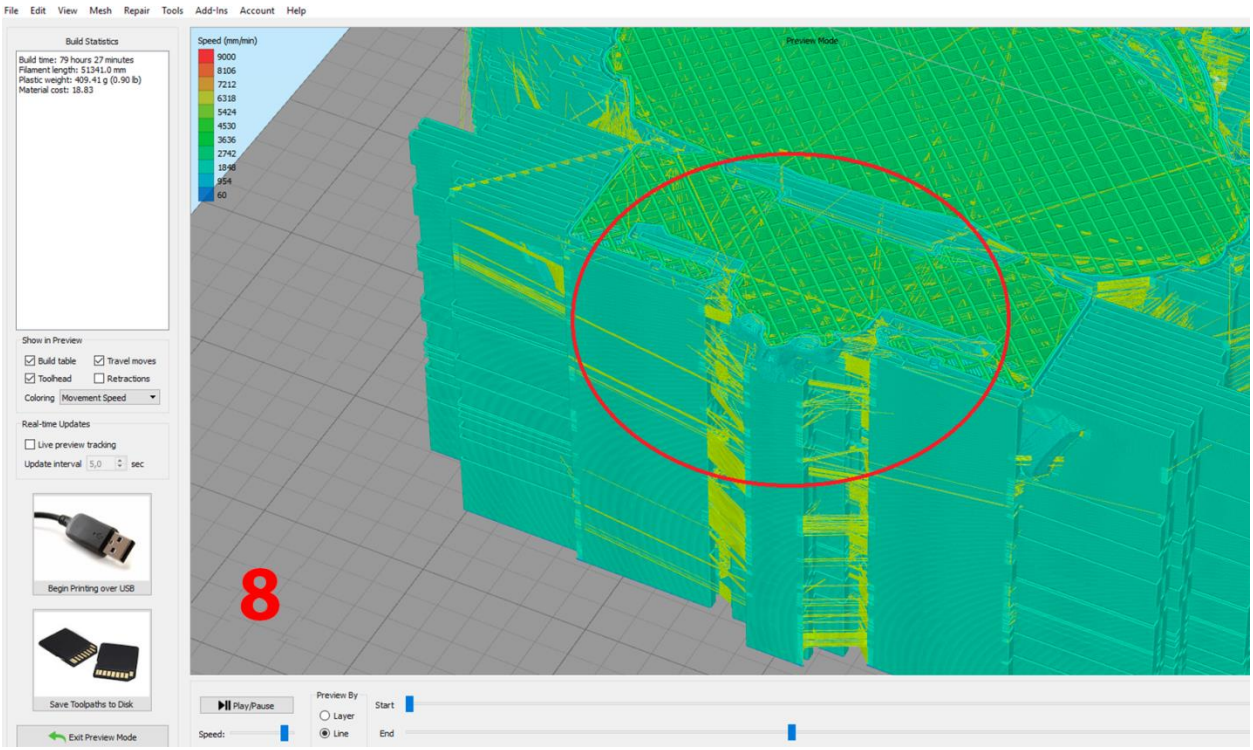
Step 6. Go to *Edit > Align Selected Model Origins*



Step 7. Result



Step 8. Result



* For some 3D printing models, avoid putting the check mark in the box next to the *Merge all outlines into a single solid model*. To know exactly for which 3D models the box must be checked or not, please test both with and without the check mark before sending the 3D model to the printer.