

Zortrax // Z-Suite - Quick Start Guide

Before you begin: Work locally. Always start by moving your file to the desktop. Opening and editing files from a USB (or other) drive can create problems.

Instructions

1 The software that prepares models for 3D printing on the Zortrax printers is called Z-Suite.

You may use any 3D modeling software that you are comfortable using to create your file, as long as you are able to output it as an .stl.

2 Open your .stl file in Z-Suite, by navigating to the "+" symbol on the left navigation bar, and then to your saved .stl. (Note: large .stl files may take several minutes to load, see the [3D Scan Cleanup Guide](#) for details on how to reduce the size of your .stl file)

3 Additional models may be added by clicking the "+" button.

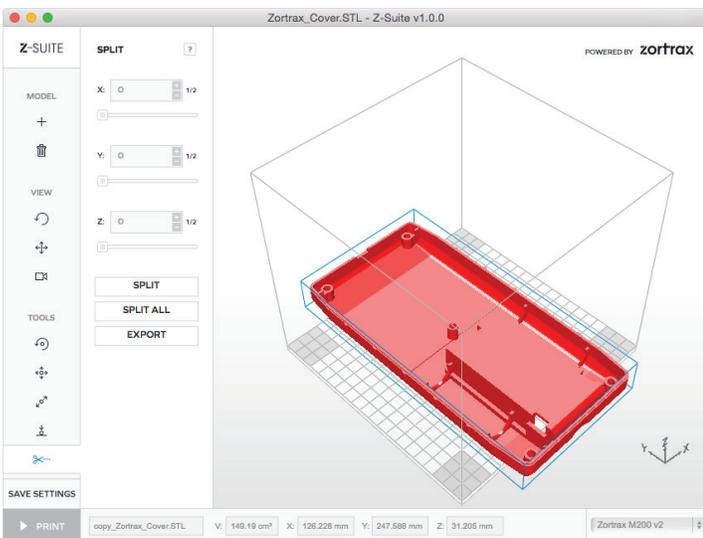


Fig 5 - Splitting the Model

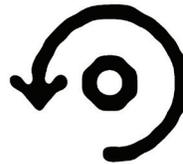


Fig 1 - Rotate

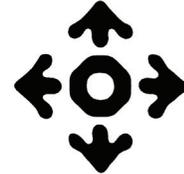


Fig 2 - Move



Fig 3 - Scale



Fig 4 - Auto-Arrange

4 Orient and size your model for printing. Generally, printing in or near the middle of the plate is best. Also, be sure to orient your model to minimize overhangs and for the easiest removal of support material. Select your model by right-clicking on it. Then select the appropriate tool to modify your print.

The Move button allows you to shift the position on the plate, the Rotate button rotates the model, and the Scale button changes the size of the model.

Models that overlap or fall off the edge of the build plate will be highlighted in red. Models too large for the build plate may be split into multiple pieces using the Split Tool

5 When your object is oriented and ready to print, click the Print button.

Instructions

6 The default settings work well for most models, but there are some settings that - depending on your model - you may want to adjust.

Profile: Should ALWAYS be set to Z-ABS

Layer Thickness: Selecting a smaller layer thickness will give you more detail, but will take significantly longer to print. Larger layer thickness will print faster but with less detail.

Speed: Should ALWAYS be set to Normal for best results.

Infill: Determines how dense the interior mesh of the model is. Full infill will take the longest to print, but yield a more sturdy and solid model.

Support: The higher the angle of the support, the more support will print. Increase this angle if your model has areas that are sagging due to being unsupported. A setting of 0% will print with no support material. The Support Lite check box will make supports easier to break off of your model, but more likely to break off and fail while printing.

Print Cooling: Normally, auto fan speed works best. If your print is having trouble with warping, set the fan speed to 0%. If fine details are drooping, increase the fan speed.

7 When your print is ready, select Prepare to Print. Note: this process may take several minutes depending on the size of your print.

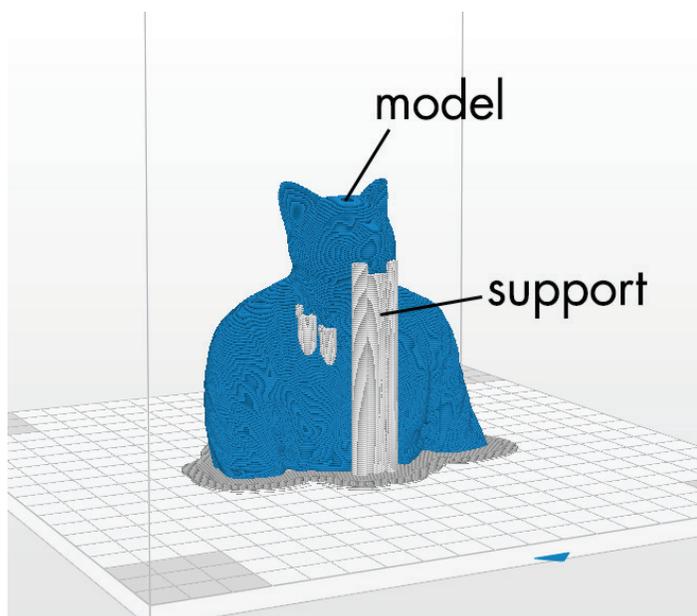


Fig 7 - Print Preview

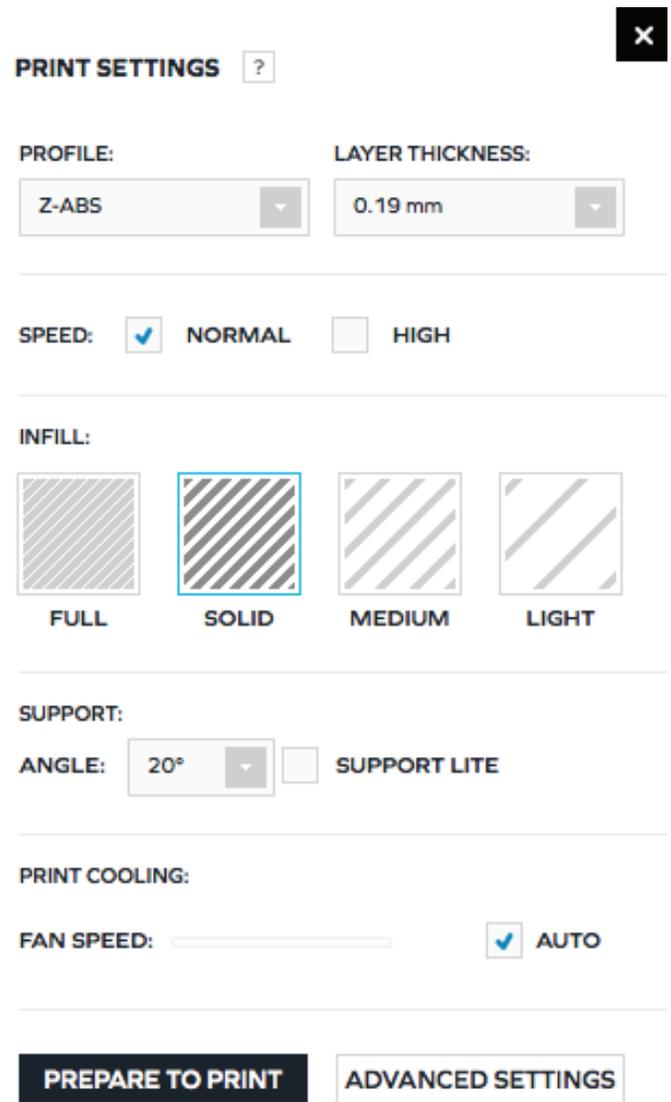


Fig 6 - Default Print Settings

8 When your print is ready Z-Suite will show you a preview. The blue is your model and white is support material. Across the bottom of your screen you will see a summary of your settings as well as an estimated print time.

If something does not look correct, click Back and re-adjust your print settings.

If everything looks good, click Save to Print to save your .zcode file.

9 Copy your .zcode file to a USB stick or memory card and bring it to the Digital Output Support Assistant in 235A to add your print to the queue!