



Z-suite: Slicing

This guide will explain the basics of slicing within the proprietary Zortrax software, Z-Suite.

Written By: Nico Willem Lagendyk

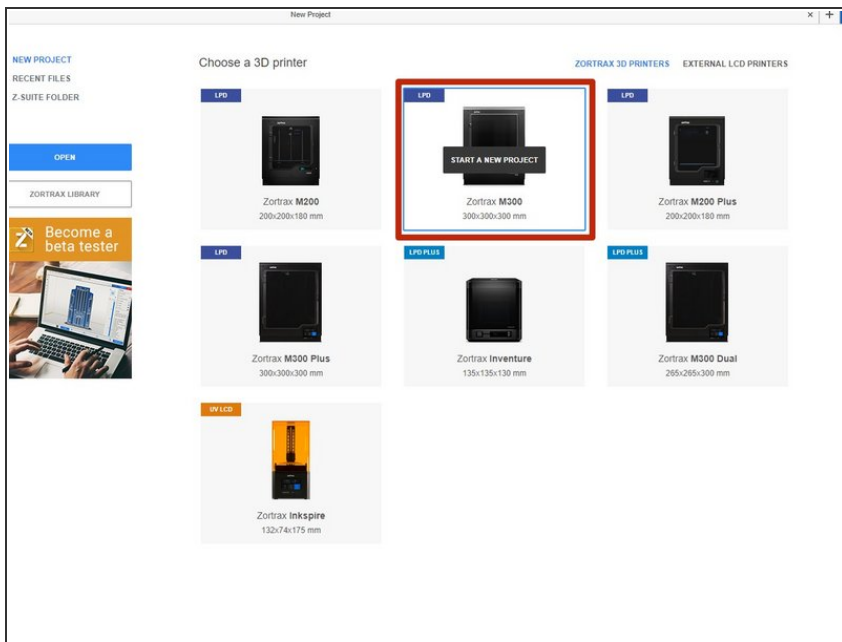


Introduction

This guide will explain the basics of slicing in Zortrax's proprietary software, Z-Suite, for generating .zcode to be used with the M200 and M300 printers.

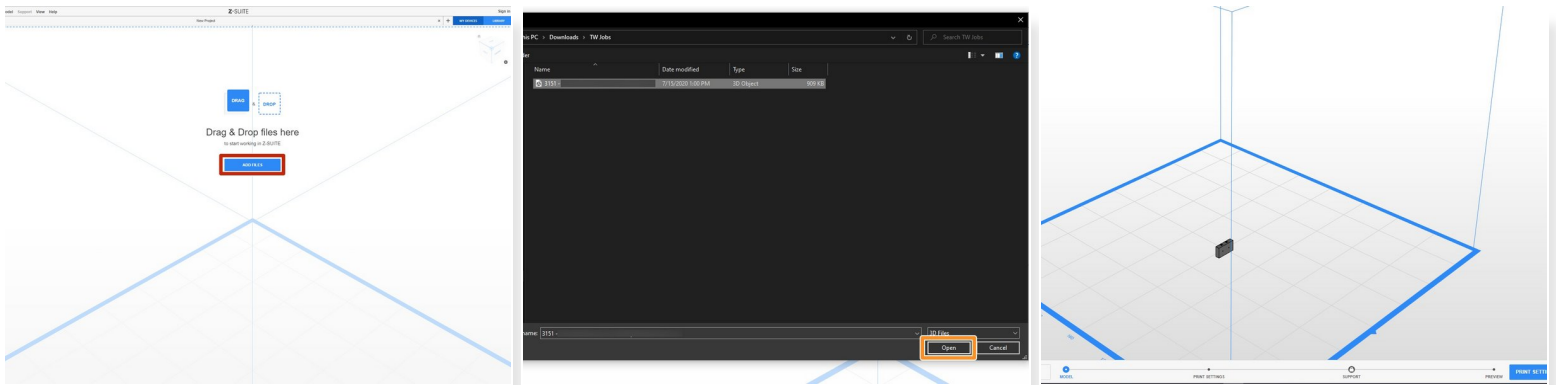
Images By: Nico Legendyk

Step 1 — Opening Z-Suite



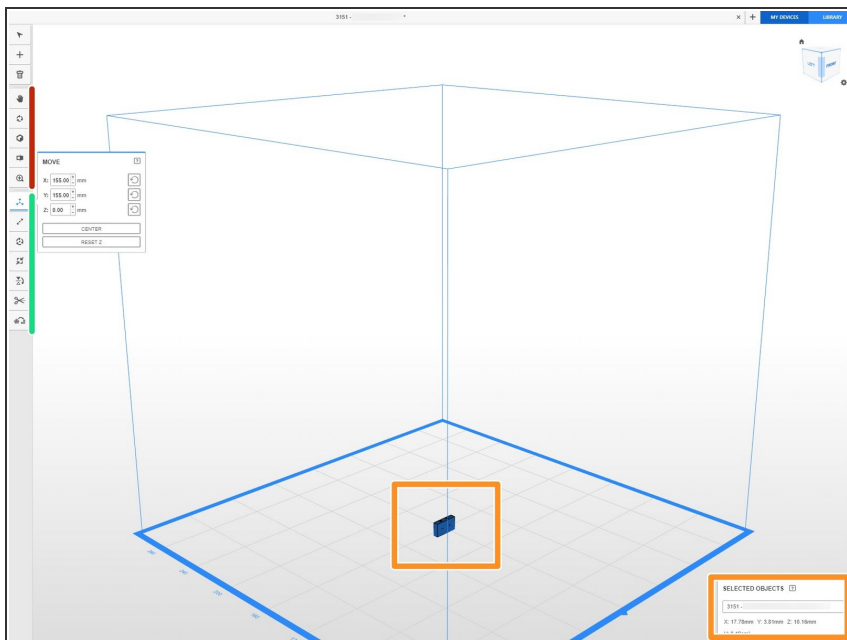
- When you first open Z-Suite you'll be presented with an option to choose a device to print from.
- Select the appropriate device you need to use for the job
- i For this guide we'll be using the M300

Step 2 — Importing a File



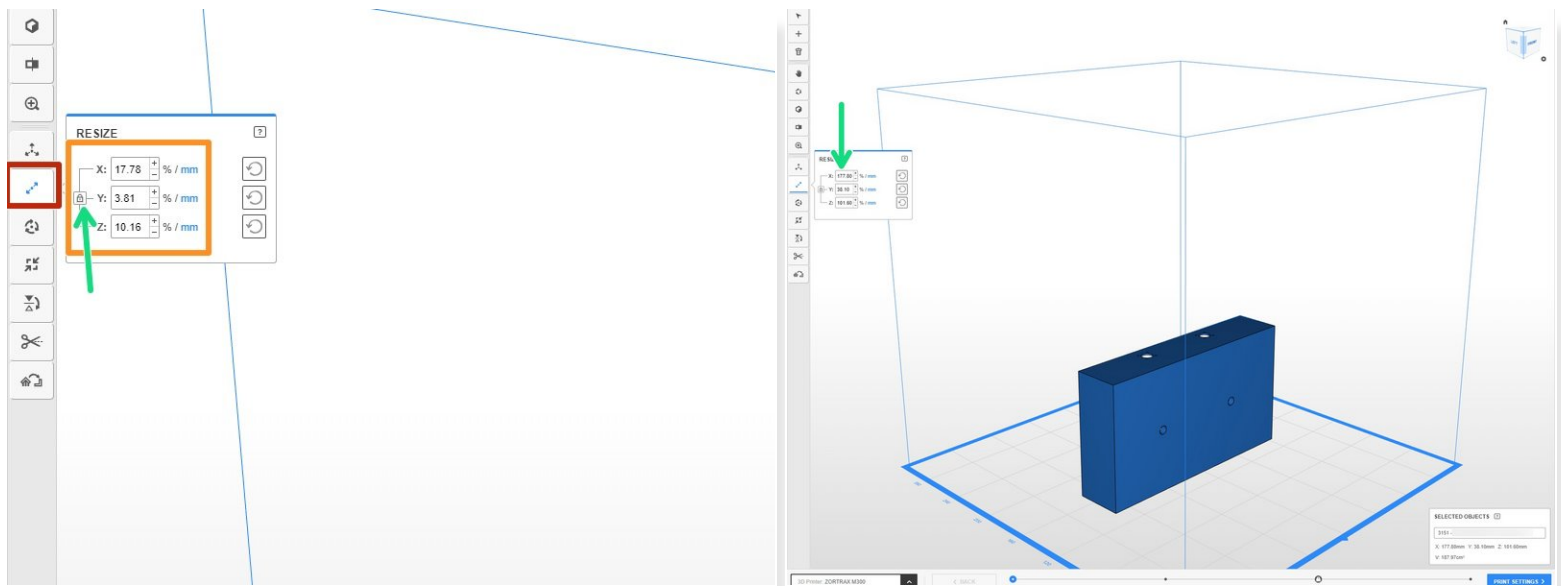
- Drag and drop your files into the software or browse for files by clicking "Add Files"
- Clicking add files brings up your file explorer where you can select the 3d object you want to print
- ★ Make sure your file is in the appropriate file type of .stl
- Select your file, then click open
- Your file is then imported onto a plate virtually representing the build plate

Step 3 — Navigation



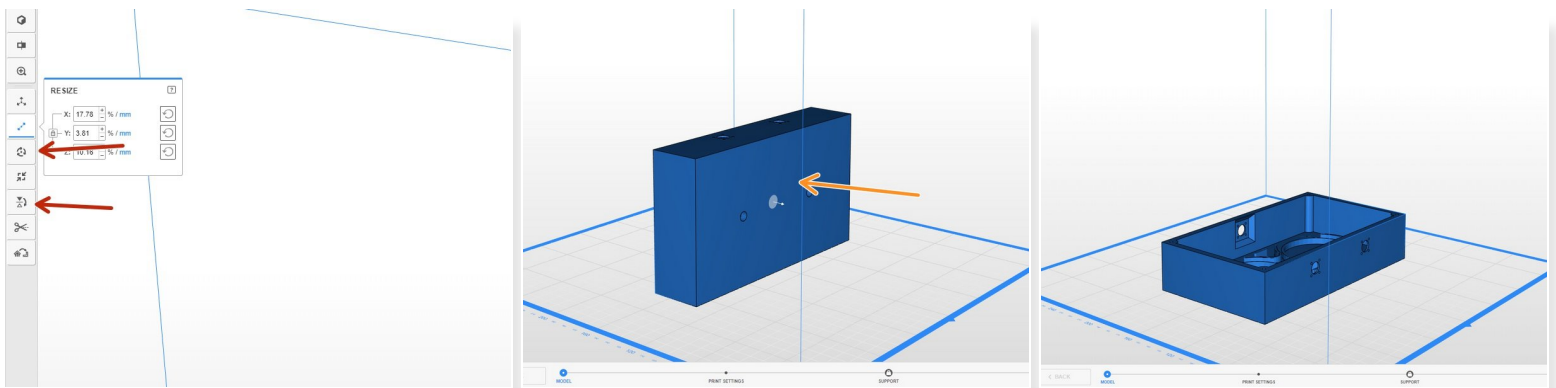
- Navigating is pretty easy and similar to other 3D software
- Scrolling changes the zoom and holding down the left button of your mouse allows you to rotate the plate
- All these basic tools have associated buttons on the left toolbar
- Selecting a model on the plate allows access to a new set of tools on the toolbar
- This includes moving, scaling, rotating, placing by face, and splitting the model

Step 4 — Resizing a Model



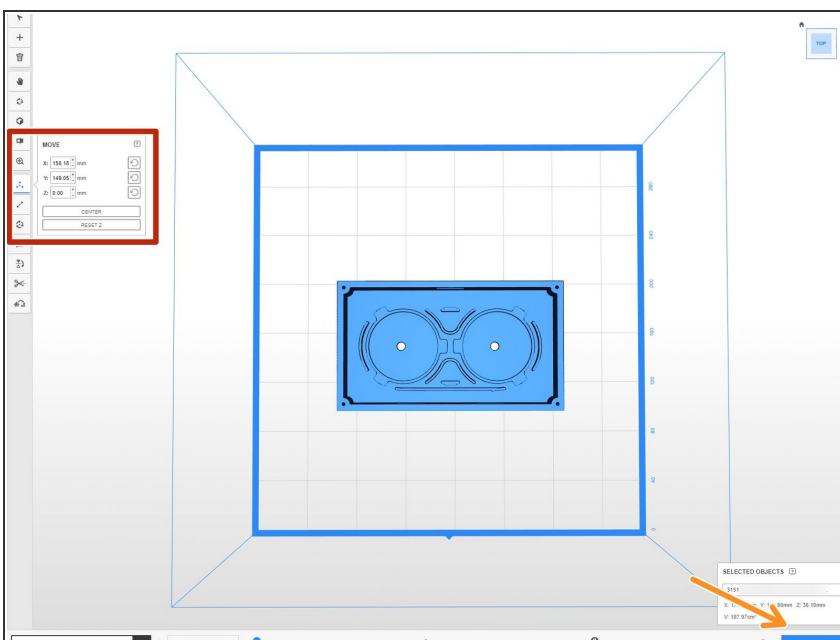
- Access the resizing tool by clicking on the button with a line going through a point
- ❗ Files can be exported in one unit system and then be imported in another so a part becomes either too small or too large
- Retrieve the bounding box dimensions to fix this issue
- This model should be 7" x 4" x 1.5", but as you can see, it has been scaled 0.1x in mm and is clearly too small
- Convert one of your bounding box dimensions to mm and make sure that the little lock next to the dimensions is locked to preserve the scaling
- ❗ Once you change the dimension your model will look like the appropriate size

Step 5 — Correcting Orientation for Printing



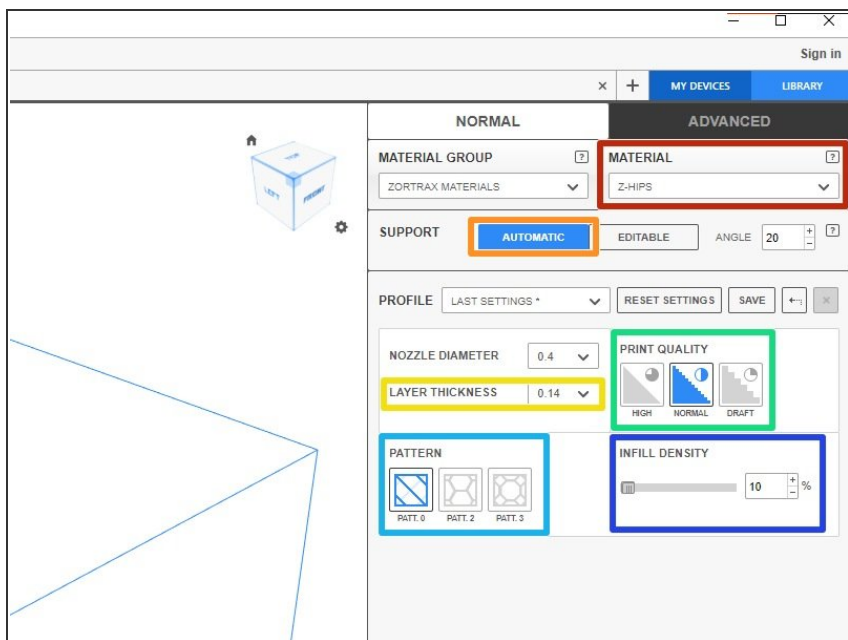
- Orient your model in the best way to be printed
 - Minimize overhangs
 - Optimize for the least amount of support
- For this model it is pretty straight forward that we want the large flat face on the plate
- Use the rotate model or place by face tool to achieve this action

Step 6 — Moving the Model



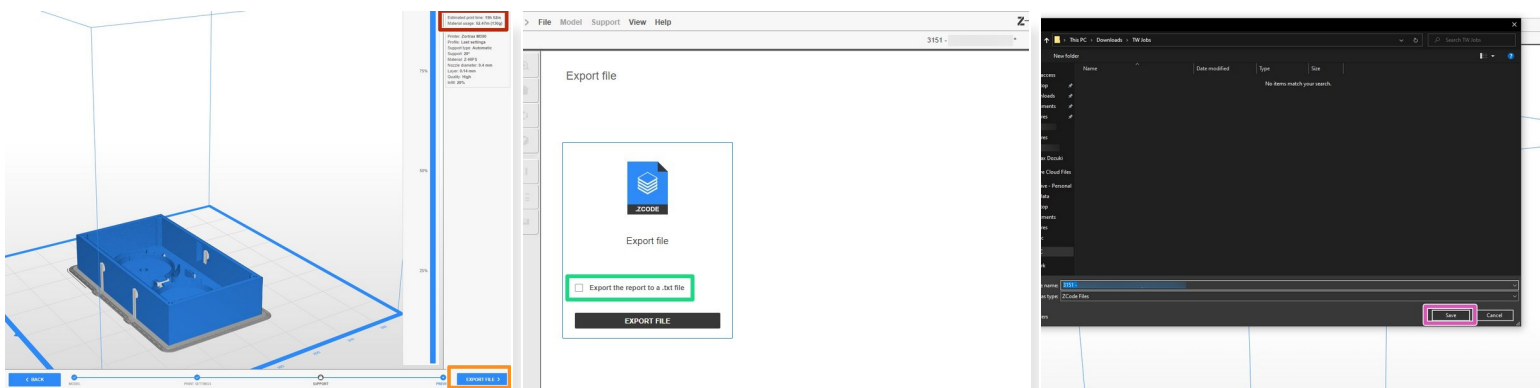
- Move the model towards the center of the build plate and ensure that it fits properly
- Change values in the move function box or drag the model when the tool is selected
- Once the model is correctly scaled and moved you can move on to the next page by clicking "Print Settings"

Step 7 — Printer Settings



- Select the correct material from the drop down menu
- Click on "Automatic" for support
- Use 0.14mm layer thickness for normal parts, but make it smaller for finer parts and larger for coarser parts
- Set print quality to "Normal" in most cases
- Use rectilinear infill pattern unless customer states otherwise
- Use 5-15% infill unless customer states otherwise
- Click "Preview" down at the bottom when finished

Step 8 — Previewing



- i** When you click preview, the software will go through and add support where its necessary and generate the .zcode for the printer to read
- Note the time and amount of material needed
 - Export the file to use on the printer
 - Uncheck the "Export the report to a .txt file" unless you want that information
 - Save your .zcode file in your desired location