

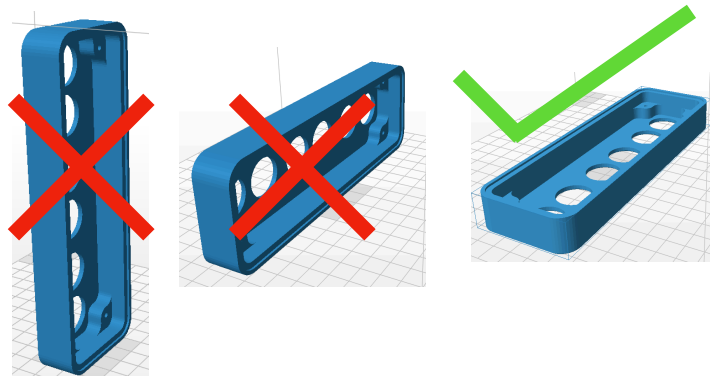
# zortrax

Simon Carlsson

GUIDE

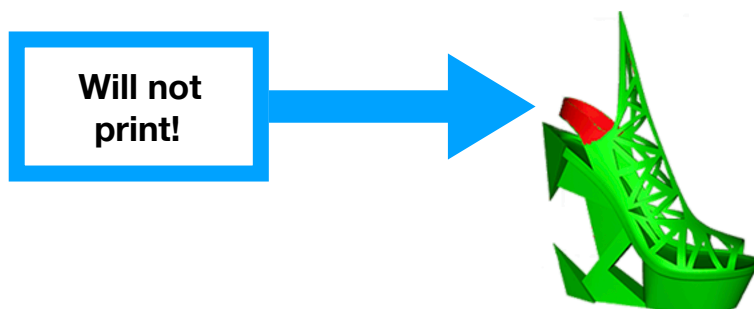
*Note that the guide is based on Z-suite version 2.7.4 and the steps described here might be slightly different from any newer versions. However, they will be basically the same.*

1. Download Z-suite from Zortrax's website.
2. When prompted type in the serial key: **Z846FOC66**
3. Select the correct printer: **Zortrax M200**
4. You need a .stl-file to print! This can be obtained from your favorite CAD-software or by downloading one you like, for example from [thingiverse.com](http://thingiverse.com)
5. If required, the model can be moved around or even resized. When placing the model keep this in mind:
  - Avoid step overhangs whenever possible. Gradual or small overhangs are fine.
  - Try to keep the "center of mass" as low as possible.



- When you're happy with the placement, press **Analysis**

6. In the **analysis** section you can view how well the details of the models are going to be reproduced. Details that are too small for the printer will show up as red. If you have too much red in preview, you might want to consider enlarging these details. When this looks good, move on to **Print Settings**



7. Select the material that you wish to use, like **Z-ABS** or **Z-ULTRAT**. **You have to set this to the material that is in the printer when you print! Otherwise the printer may get damaged! Z-ABS** will be a good material for most prints.

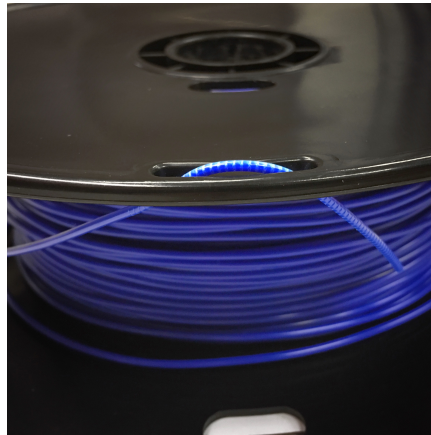


8. The printer can not print in mid air, therefore, a thing called **Support** exists. This is sacrificial material that holds up plastic that would otherwise not be able to be supported. **Support** is recommended if you have any large overhangs. If you have selected **Automatic** then support will be added automatically where the slope angle is greater than the specified **ANGLE**. The **EDITABLE** option will let you manually specify where to place the supports in the next screen. By selecting this option and skipping the next **Support** screen, you will not generate any support at all (which is also fine in some cases).



9. Select the **quality** of the print. This can be done in two ways:
- Choosing a **Profile** will automatically change all the settings according to that profile. Higher quality profiles takes longer to print.
  - Changing the **layer thickness** will also directly impact the quality. Lowering the layer thickness will result in better quality prints at the cost of longer print time. The thickness can be adjusted between **0.09 mm** and **0.39 mm**, however a layer thickness of **0.19 mm** is recommended for most prints.
  - **Infill** is always recommended, this will fill your model with a zigzag pattern, helping to increase its strength and printability. **Medium (30 %)** is recommended for most prints. Higher percentage of infill will usually yield a stronger part.
10. Preview and save your file to the Zortrax SD-Card, which is located right above the front LCD-screen. A SD-Card to USB adapter is located in the drawer under the printers.

11. Turn on the machine and navigate to **Model**. Find your model and open it, verify that the material name on screen matches the material in the printer. If not, the material can easily be **swapped**.
  - Navigate to **Material**. Choose **Unload filament** (this takes some time), when the printer is done the filament can be unloaded by rotating the spool at the back, **Clockwise**.
  - **When storing the filament, put the end of the filament in the spool according to the image!**



- Take the new filament and using a pair side cutters, cut off approx. 4 cm of the end. This ensures that the printer can grab the material.
  - Put the material on the spool holder and feed the material manually through the plastic tube until you can't feed anymore. Press **Load filament** (this can take some time). Observe if the new filament extrudes. If it does not go in all the way, you can remove the clear plastic guide tube from the extruder and push it from there and then press **Load filament**. If the filament is not loading properly, (ie it is making a clicking noise), then the extruder is clogged and it needs cleaning.
12. Turn on the **fume extractor** at the back of the printer.
  13. Go back to **model** and press **print!**  
**NOTE (VERY IMPORTANT): You need to watch the printer start the first layer. If it does not home correctly, then turn it off IMMEDIATELY!**
  14. When the print is done it needs to be removed from the build plate.
    - The build plate can be removed. This is done by lifting the build plate carefully (it's fixated via magnets). **Make sure to unplug the two wires at the back before removing the build plate completely. Otherwise the build plate can be damaged.**
    - Use The yellow plastic spudger or the metal spudger to remove the prints. Be careful not to damage the print bed or your model.
    - All Zortrax prints use rafts that dig into the build plate, this makes them difficult to remove, so take your time!
    - When the model is released, cleanup can be done if necessary with a pair of side cutters.
  15. Remove any remaining filament on the build plate and reattach it and the cables to the printer.
  16. Pat yourself on the back! You've just 3D-printed something!!