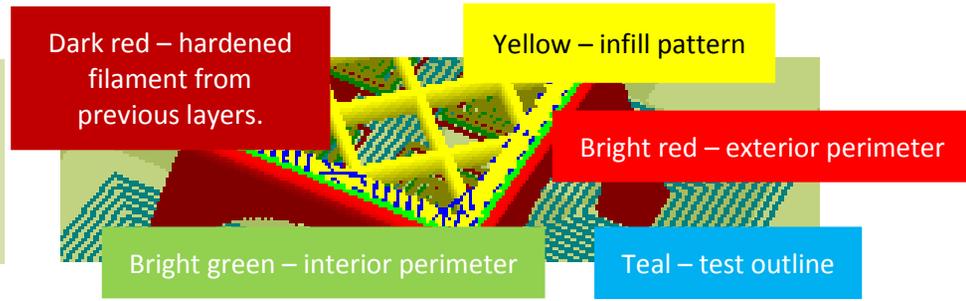
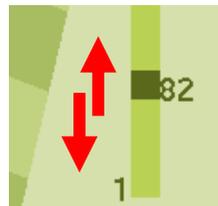


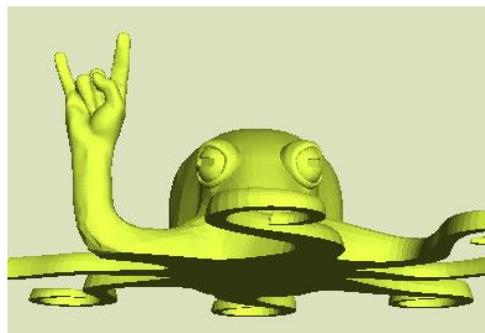
3D Print Preview

Before we print your objects, we take a look at them in the program that controls our 3D printer – LulzBot Cura. You can download and install that software (here: <https://www.lulzbot.com/cura>) and preview your prints before even submitting them to us. Once you’ve downloaded and installed it, open the program up. The first time it will ask you what kind of 3D printer you have. The library has LulzBot Mini 3D printers. Click **LOAD MODEL** and then open up your file. Give it a second to load. Then, click **VIEW MODE**, and then **LAYERS** to see the 3D print preview. Drag the slider up/down to see different layers, make sure your object is water tight, etc.

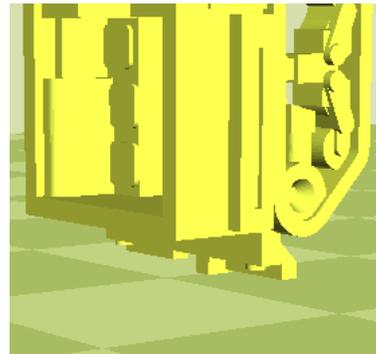


Foundation – 3D printers can’t print in thin air!

The best 3D models have a solid foundation for the rest of the print to build upon. Structures like arches can also work well. And objects shaped like Jedi Master Yoda work well too, with parts toward the top hanging over thin air with some support from previous layers. Objects submitted without a strong foundation may have to be printed with supports, which can be difficult to remove.



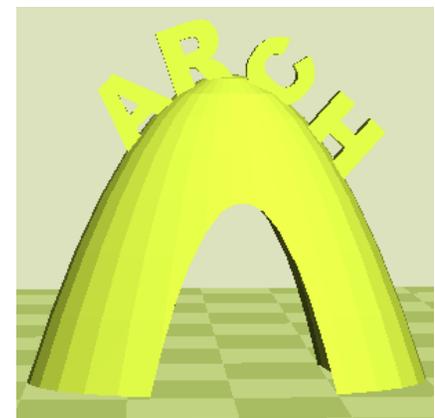
GOOD



BAD



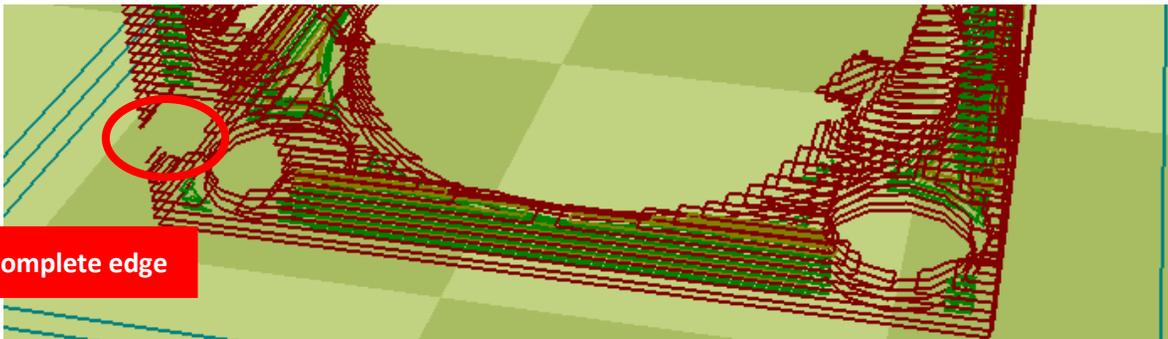
OK – self-supporting overhangs



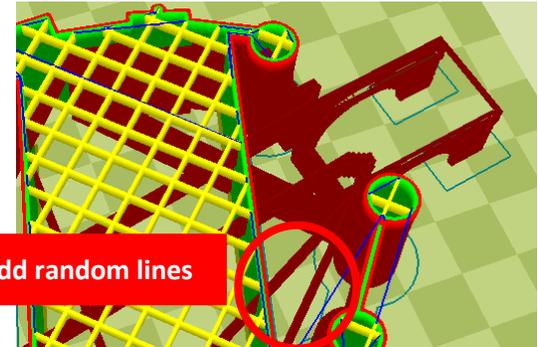
OK – middle will sag a bit

Slicing

The part of LulzBot Cura that turns your model into instructions for the 3D printer to move around is an algorithm called a slicer. Sometimes this slicing algorithm is not perfect. We have occasionally seen it ignore holes designed into models. And we've seen the slicer not connect parts of water tight shapes. The slicer also occasionally mis-designs internal structures of objects. Check to make sure Cura is slicing your object successfully.



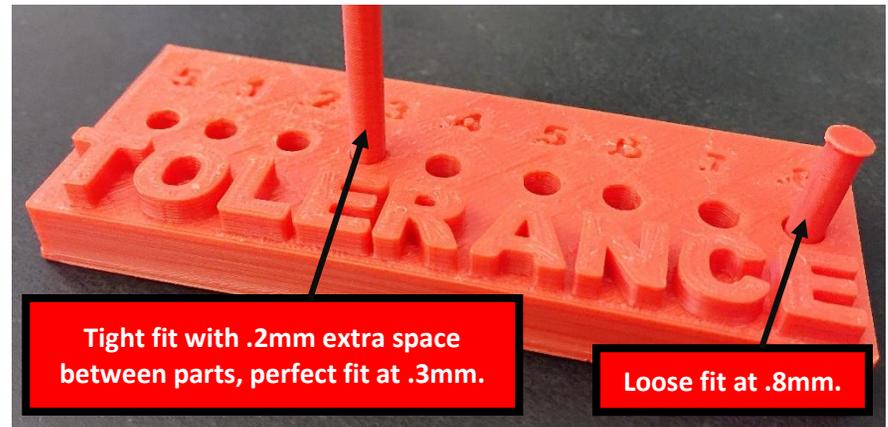
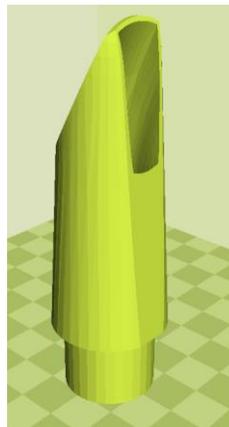
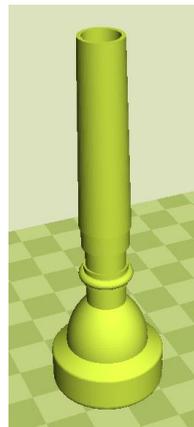
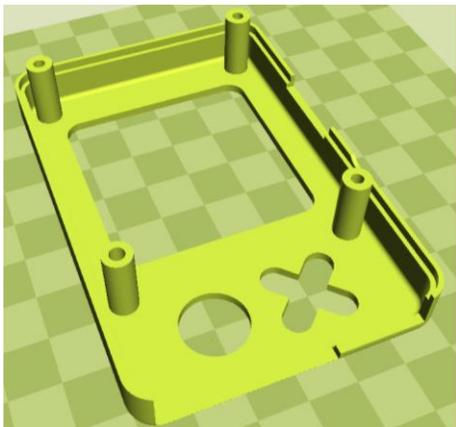
Incomplete edge



Odd random lines

Tolerance – we've measured our LulzBot Mini's tolerance at .3mm

If you are designing a case for a phone, tablet, Raspberry Pi etc., making holes and other interlocking parts is difficult. Our 3D printer tends to make holes a bit tight – you might want to increase the size of holes so parts can fit together.

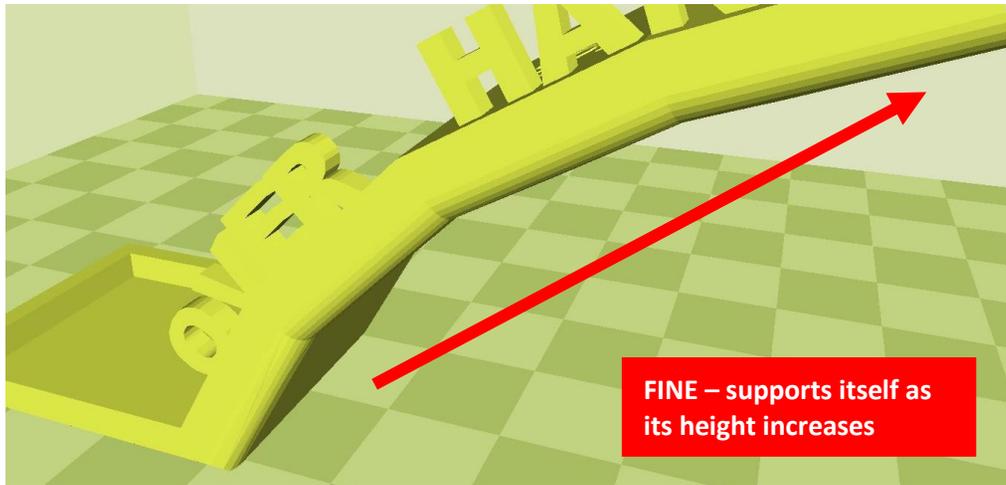


Tight fit with .2mm extra space between parts, perfect fit at .3mm.

Loose fit at .8mm.

Overhang

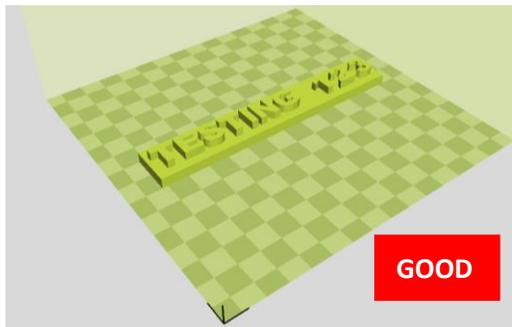
For highest quality printing, each part of your object needs something below it supporting upper elements all the way from the bottom build plate. This is because 3D printers print layer by layer, from bottom to top.



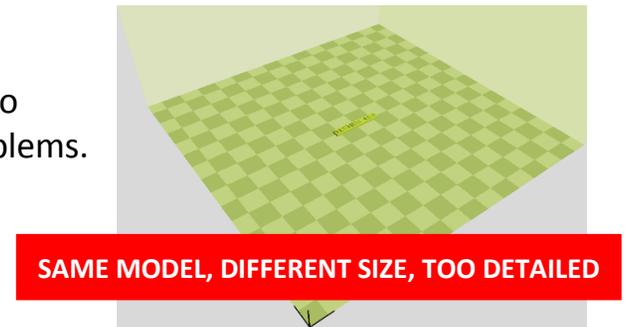
Size & Scaling

There's a limit to how detailed our 3D printer can print. If you downloaded Cura, load up a model and give it the squint test. If you have to squint hard or zoom in a ton, elements like letters of fine details may be too fine to print.

Unfortunately, some 3D design programs output objects without units. These objects tend to be incredibly tiny or sometimes incredibly large and require scaling up or down.



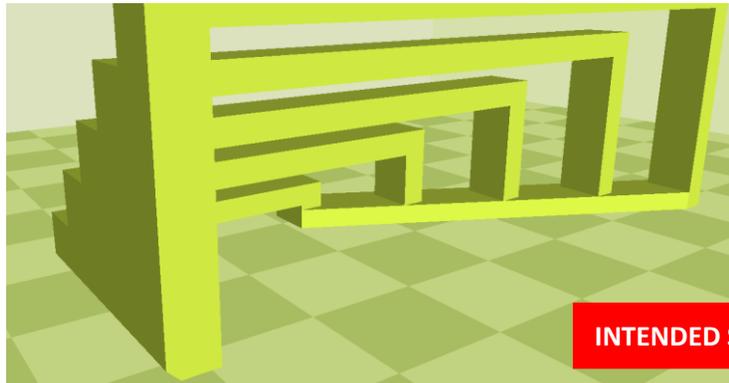
Know the dimensions you intend your objects to print at so we can work with you to fix any problems.



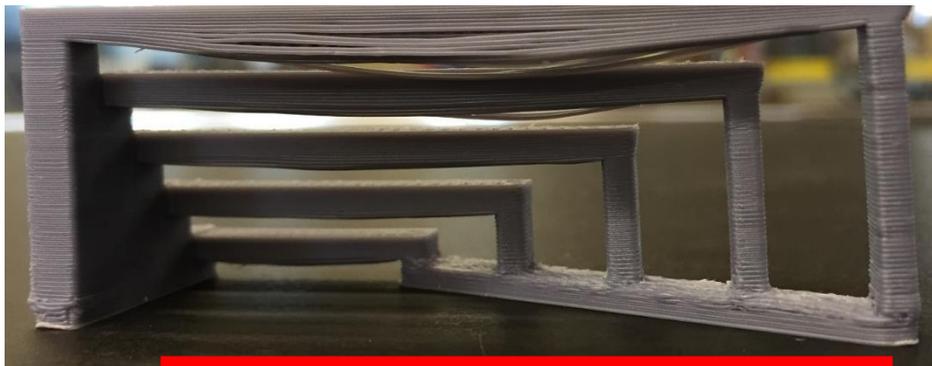
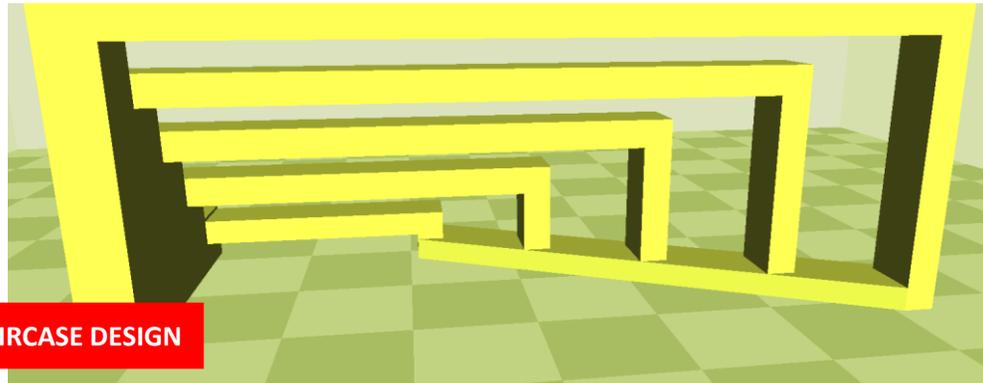
Supports - *and why you don't want us to have to use them*

If you submit objects that don't have a strong foundation, that have slicing errors, have free floating elements, or that have significant unsupported overhangs, we may be forced to print your object with supports. Supports are like a scaffold for 3D printing. They are plastic parts printed to help hold up certain parts of an object while it is printing. They are disposable, designed to be ripped off after printing has been completed successfully. However, we've found that unless supports are removed almost immediately, they can be very difficult to remove. We do not immediately remove them for you because we do not want to risk breaking your object. However, after completed objects have cooled down and hardened more, the difficulty in removing supports can also put your object at risk.

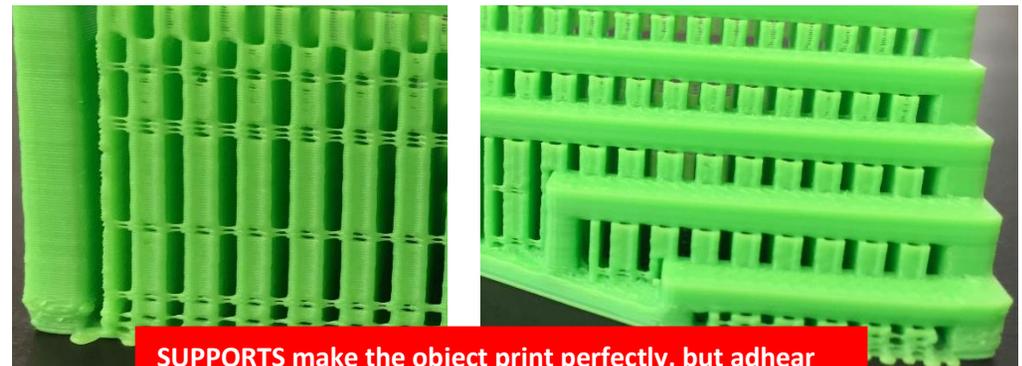
To sum it up: you really don't want us to have to print your object with supports if you can avoid it. Try to check your model in Cura or design to avoid the need for supports.



INTENDED STAIRCASE DESIGN



WITHOUT SUPPORTS it bows down, so it needs supports



SUPPORTS make the object print perfectly, but adhere to all surfaces, thus making them difficult to remove