3D PRINTER
Quick Guide *S3 Edition

Set Up:

1. Create an S3 printer profile. From Cura click on settings –> printer –> add printer then click "add non-network printer" –> Ultimaker S3

2. Load your 3D design into Ultimaker Cura on your device.

3. Check that your Cura Settings are correct. **Material: Generic CPE, infill: 20%,** bed temp: 80, print temp: 250

4. Decide if your design needs support or a brim, and adjust settings accordingly

Use:

*On the online form the name should be written as "MY-S3 1"

1. Begin filling out an online form* for your 3D print job: https://go.unc.edu/Rn83P (don't submit until you press 'Print')

2. Look for a 3D printer that is not currently in use, and **verify that it has enough filament for your print.**

3. Take the USB from your chosen printer and load your gcode file from Cura onto it (NOT your original design file.)

4. Use the touch screen to press 'Print' on your file, then submit your online print job form. **REMEMBER: if you do not submit your form, the BeAM staff will cancel your print.**

Clean Up:

1. You must stay in the makerspace until after **the first layer of your print is complete or for 20 minutes** (whichever is longer.)

2. Once your print finishes and the bed has returned to room temperature, it should lift easily from the bed. **DO NOT PRY IT OFF** - if you have to use force, the bed is still too warm.

Want more detail with photos of the process?
https://beam.unc.edu/files/2019/01/3DPRINT_EBOOK5.pdf