# Cura Assignment Sheet

As the final component of the DFAM training/study, you will prepare your part for 3D printing. For homework due at the beginning of your next studio, you will turn in **a) the 3D object file generated by Cura and b) a screenshot of the Cura results**, as described later in this document. First, model your final design (from the end of the redesign phase in studio) in Solidworks. Do not make any changes to your design as you take it from a sketch to the CAD model. Then, export your Solidworks part as an STL and open it in Cura. Below are the instructions for setting up the Cura inputs.

1. Open the Cura software and select the Ultimaker 2+ as the printer (you may have to select "Add Printer" and then "Add a non-networked printer")
2. Choose PLA as your material and a 0.4 mm nozzle size
3. Under the Recommended settings, select 0.1 mm profile, 20% infill, check the Support Box, and uncheck the Adhesion box.
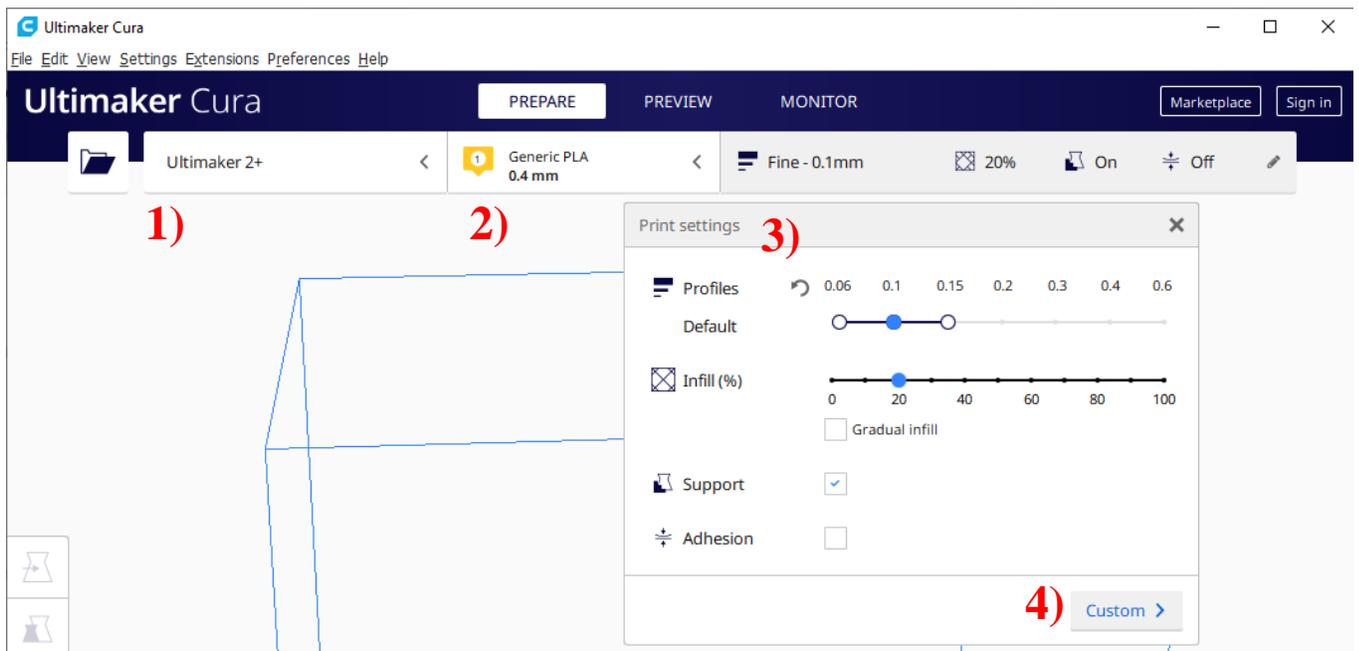
4. Expand the Custom settings.
5. You will keep all of the Custom settings as the default except for the ones listed below. The figure on the next page shows what the default settings should be.
   a. Infill: Infill Density = 20%, Infill Pattern = Grid
   b. Support: Generate Support, Support Placement = Everywhere, Support Overhang Angle = 45°
   c. Build Plate Adhesion: Type = None

Print settings   ✕

Profile   Fine · 0.1mm   ★ ⌄

🔍 Search settings   ≡

Quality   ⌄
Layer Height   🔗   0.1   mm

Shell   ⌄
Wall Thickness   1.05   mm
  Wall Line Count   3
Top/Bottom Thickness   0.8   mm
  Top Thickness   0.8   mm
    Top Layers   8
  Bottom Thickness   0.8   mm
    Bottom Layers   8
Horizontal Expansion   0   mm

5a) **Infill**   ⌄
Infill Density   20   %
Infill Pattern   Grid   ⌄

Material   ⌄
Enable Retraction   ✓

Speed   ⌄
Print Speed   50   mm/s

Travel   ⌄
Z Hop When Retracted   ☐

Cooling   ⌄
Enable Print Cooling   ✓
Fan Speed   100.0   %

5b) **Support**   ⌄
Generate Support   🔗 ↺   ✓
Support Placement   🔗   Everywhere   ⌄
Support Overhang Angle   🔗 ↺   45   °

5c) **Build Plate Adhesion**   ⌄
Build Plate Adhesion Type   🔗 ↺   None   ⌄

Dual Extrusion   ‹

‹ Recommended

6. Align the part on the build plate in the way that you feel is most appropriate, given what you have learned in class.
7. When you are done orienting the part, press "Slice" in the bottom right corner of the screen. Go to the tab at the top titled "Preview."
8. Mouse over the Info button in the window in the bottom right. While this window is open, take a screenshot of the page (you may have to use the Print Screen key so that the info window does not close). Be sure to include a clear view of the part on the print bed as well (you may have to zoom and rotate the view). An example is shown on the next page.
9. Finally, go to File > Save in the Cura toolbar. Save the project file, then submit it to Canvas along with your screenshot. The file type should be "3D Object."
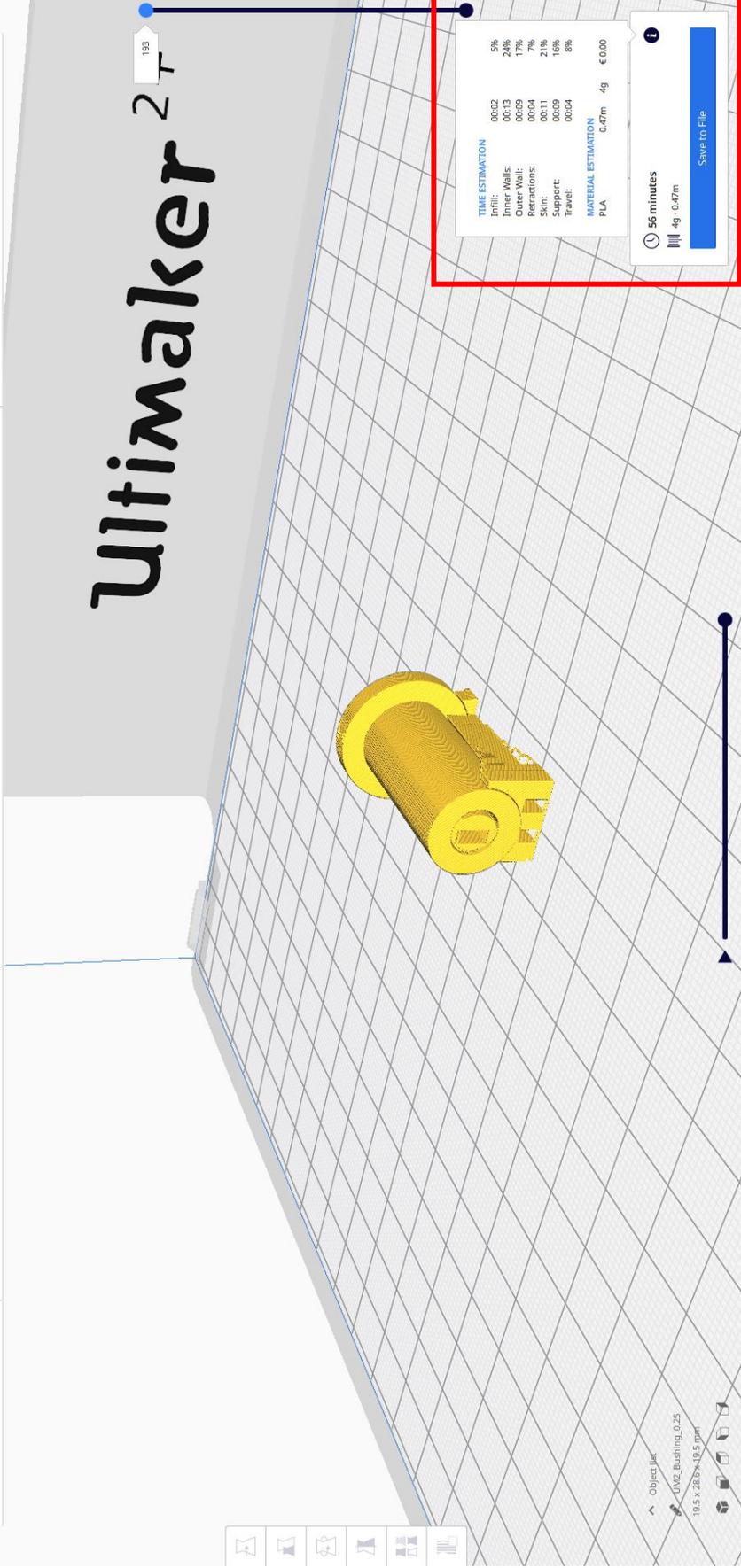
7)

8)

TIME ESTIMATION
Infill: 00:02   5%
Inner Walls: 00:13   24%
Outer Wall: 00:09   17%
Retractions: 00:04   7%
Skin: 00:11   21%
Support: 00:09   16%
Travel: 00:04   8%

MATERIAL ESTIMATION
PLA   0.47m   4g   € 0.00

56 minutes
4g · 0.47m

Save to File