Lithophanes

using Cura and Prusa 3D Printer

1. Find an image

- Search Google Images
- Black and white with high contrast works best

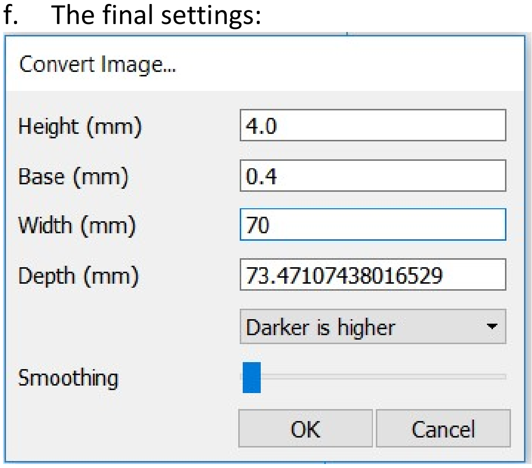
2. Save the image

- Right-click on the image and select “Save picture as”
- Note: The image default will save it to the “Downloads” folder

3. Open the image in Cura

- Under “Settings – Printer” make sure Cura is configured to “Original Prusa i3 MK3”
- Under “File” select “Open File(s)”
- In the dialog box under “Downloads” find and open the saved file. This will bring up the “Convert Image” pop-up screen
- Make the following changes:

  a. Change “Height” to 4 (this sets the thickness of the thickest region)
  b. Change “Base” to 0.4 (this sets the thickness of the thinnest region)
  c. Change “Width” and “Depth” to desired size in millimeters
  d. Change “Lighter is higher” to “Darker is higher”
  e. If the original picture is grainy, a slight amount of “Smoothing” can help
  f. The final settings:

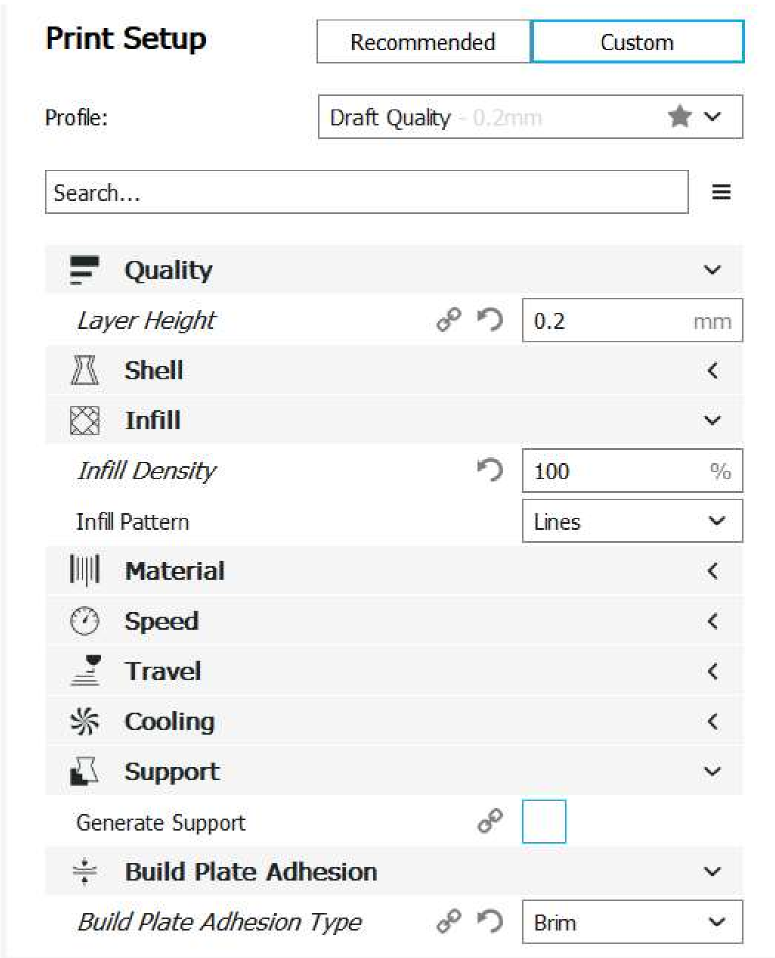


4. Rotate the image

- Select the image and choose the “Rotate” tool on the left-hand side of the screen
- Use the red hoop to rotate the image 90° so it stands on its edge

5. Adjust the print settings

- Choose “Custom” rather than “Recommended” Print Setup on the right-hand side of the screen
- Make the following changes:
  a. Under “Quality” set “Layer Height” to 0.2mm
  b. Under “Infill” set “Infill Density” to 100%
  c. Under “Support” uncheck “Generate Support”
  d. Set “Build Plate Adhesion” to either “Brim” or “Raft”
  e. The final settings:



6. Print the lithophane

- Take note of the print-time at the bottom right
- Save the file to a Prusa SD card
- The settings on this handout work best using a slightly translucent white filament, but feel free to experiment

To create more-advanced lithophanes, check out http://3dp.rocks/lithophane/