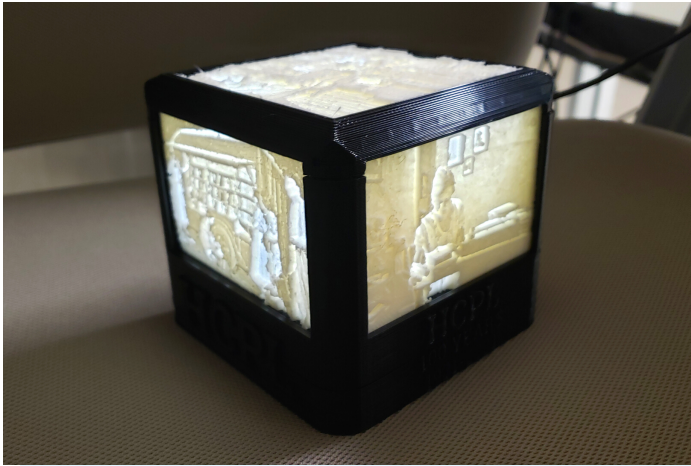


3D-printed Lithophane



You'll need:

- A computer
- 3D Slicing software
- 3D printer
- White Filament

Instructions:

1. Select your image and use a photo editing application to change the saturation of your image to zero, making it completely grayscale.
2. In a 3D slicing software like Cura, click File>Open and select your image. A dialog box may pop up where you can define settings of your lithophane, like thickness or smoothness of your image. 2-4mm thickness should give a good amount of depth for your photo.
3. Click OK, and your 3D lithophane will generate. To get the clearest view of your lithophane in the software, rotate the view until you are looking straight down on the 3D model.
4. Click File>Save and save your 3D model as an STL. You can use an STL to 3D-print, edit in 3D design software, and more.
5. If you have access to a 3D printer, make sure that you are using white filament, since this is the easiest for light to shine through. Then, you want to use high-detail settings, like changing your layer height to 0.1 mm. And lastly, make sure your infill is set to 100%, making your print completely solid.
6. Once you have changed the settings to your liking, slice your file and send to your printer. In a matter of hours, your lithophane is complete!