

Creating and 3D Printing Lithophanes

March 2025 – www.FredricRaab.com

A **lithophane** is a three-dimensional image created from a flat surface that becomes visible when backlit.

Lithophane Maker is web-based application that enables you to easily convert images into lithophane .STL files for 3D printing. In addition to flat panels, it contains options to create boxes, circled panels, night lights, lamps, and globes.

American Filament sells filaments especially designed to create lithophanes, but in general any white filament will work.

Creating lithophanes is a trial and error process. But there are many YouTube videos to help you that can be found by searching “lithophane 3D print”. A good introduction is by “3D Printing Zone”.

Not all images make good lithophanes. Pick an image highlighting something closeup without too much detail that has good lighting and strong contrasts (bright lights and dark shadows). Simpler backgrounds are better, blurred backgrounds are ideal. If necessary, you can adjust the brightness, contrast and shadows of an image in a photo editor such as Photoshop.

To create the .STL file:

1. In a browser, go to www.lithophanemaker.com
2. Select the desired project type. Start with the Flat Lithophane Maker.
3. Note: Scrolling down reveals instructions and with a video. Watch the video if this is your first time.
4. Click the *Choose File* button and open the desired image file. Your image with a rectangle showing the currently set frame size. If you want to use the full image, uncheck the *Crop?* checkbox.
5. If you are going to crop the image, do so by changing the values of the *X-Shift*, *Y-Shift* and *Rectangle Scale* boxes.
6. Select the desired *Frame Option*. Note some parameter boxes will be grayed out depending on the selected frame options.
7. The *Width* and *Height* values determine the size of the final lithophane.
Note: It's best to start small (*30-50 mm*) with *No Border* to save time and material.
8. The *Maximum* and *Minimum Thickness* values control the lithophane's brightness and contrast.
9. Enter your email address as requested. Note: An estimate of the run time (in hours) and file size are provided.
10. Click the *Create .stl* button and wait for the file to download.
11. Open the download folder and extract the file. The Settings text file documents the settings used to create the lithophane.

To Slice the file using the Bambu Slicer program:

1. Open the Bambu Slicer.
2. Click the *Prepare tab* and *Add* the .STL file.
3. If necessary, change the filament *color* to white.
4. In the *Process section pull-down*, select the desired *Lithophane* setting. This will set most of the default parameters.
5. Under *Wall generator*, verify *Classic* is set for flat lithophanes or *Arachne* is set for other types of lithophanes.
6. In the Others Tab, verify that the *Bed adhesion* parameters are set to *Outer and inner brim* with a 5mm width.
7. Click the *Slice plate* button.
8. In Preview, look at the print layer by layer. You want to see a good variety of thick and thin areas as you look through the layers. If they are all thick, you may need to lower your maximum thickness in Lithophane Maker. If all too thin, increase your maximum thickness.
9. Send the print to the desired printer.
10. Make sure the brim and first couple of layers go down smoothly.
11. You may wish to come back an hour later and shine a light through the print to see if you like the way it looks.

Refer to American Filament's *Beginner's Guide to Printing Lithophanes* for details on advanced slicer parameters.