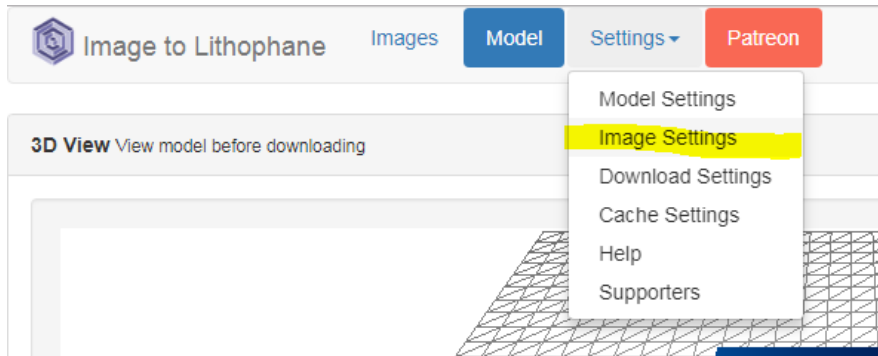


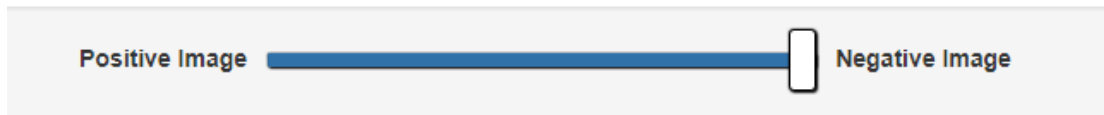
Creating and Printing a Lithophane

A helpful video on creating a lithophane using <http://3dp.rocks/lithophane/> can be found here: <https://www.youtube.com/watch?v=y63sVpeViXo> (relevant portion of video 2:30 – 4:30 minutes)

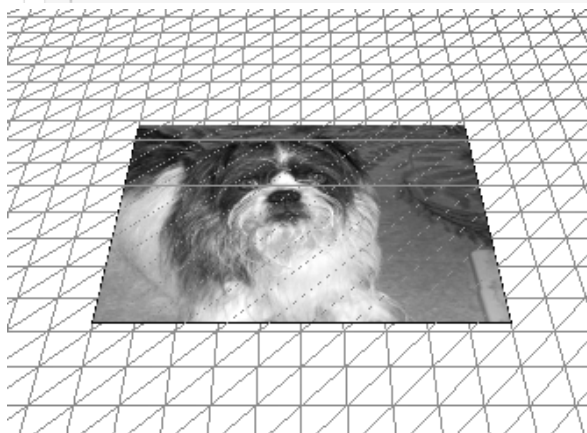
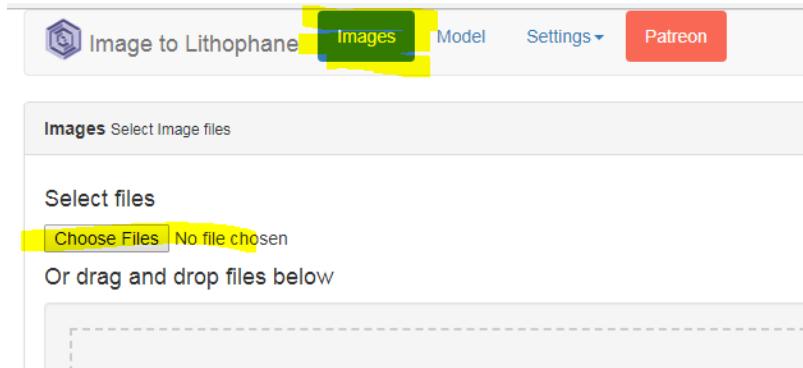
1. Using Google Chrome, open <http://3dp.rocks/lithophane/>
2. Click on Image Settings under the Settings dropdown



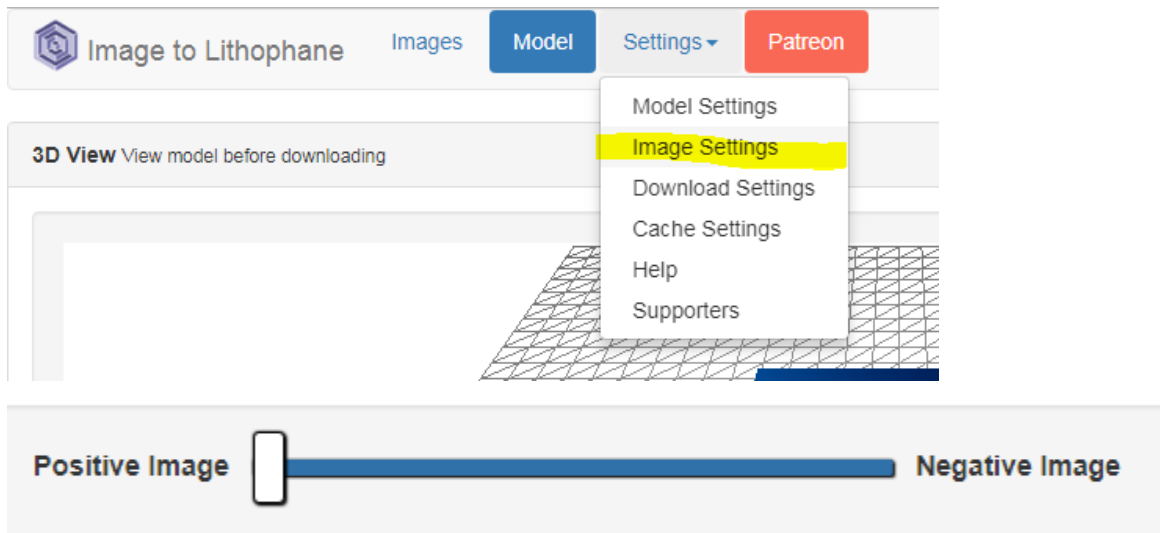
3. Check that the slide bar is set to Negative Image



4. Click Images and upload your file.
 - a. Your file will open as a positive, black and white image on the screen.

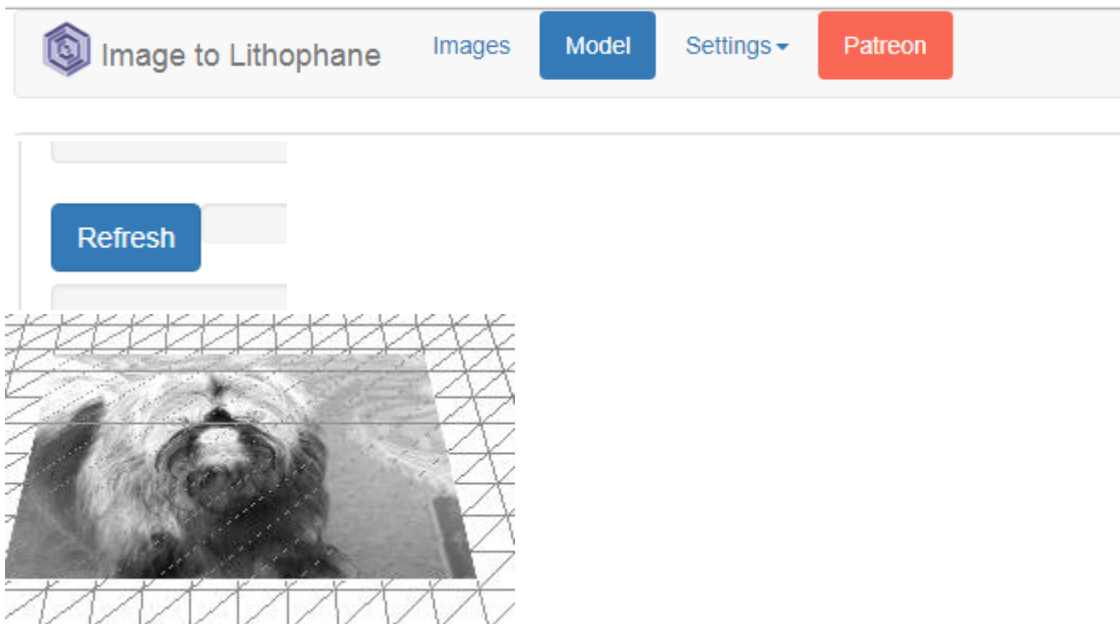


5. Click on Image Settings under the Settings dropdown and move the slide bar to Positive Image

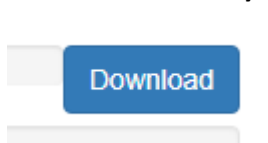


6. Click Model at the top of the screen and then click Refresh.

- a. Your image will now look like a negative, black and white image
- b. If it does not look like a negative image, go back to the Settings dropdown and slide the bar to Positive Image. Your final image needs to be in a “negative” state to print as a positive image.



7. Click Download to save your file



8. To print your lithophane, ask a Makerspace staff member for assistance

- a. Open the file in Makerbot Print
- b. Use white filament
- c. Set the Infill Density to 100%
- d. Orient the item to print “upright”
- e. Print

