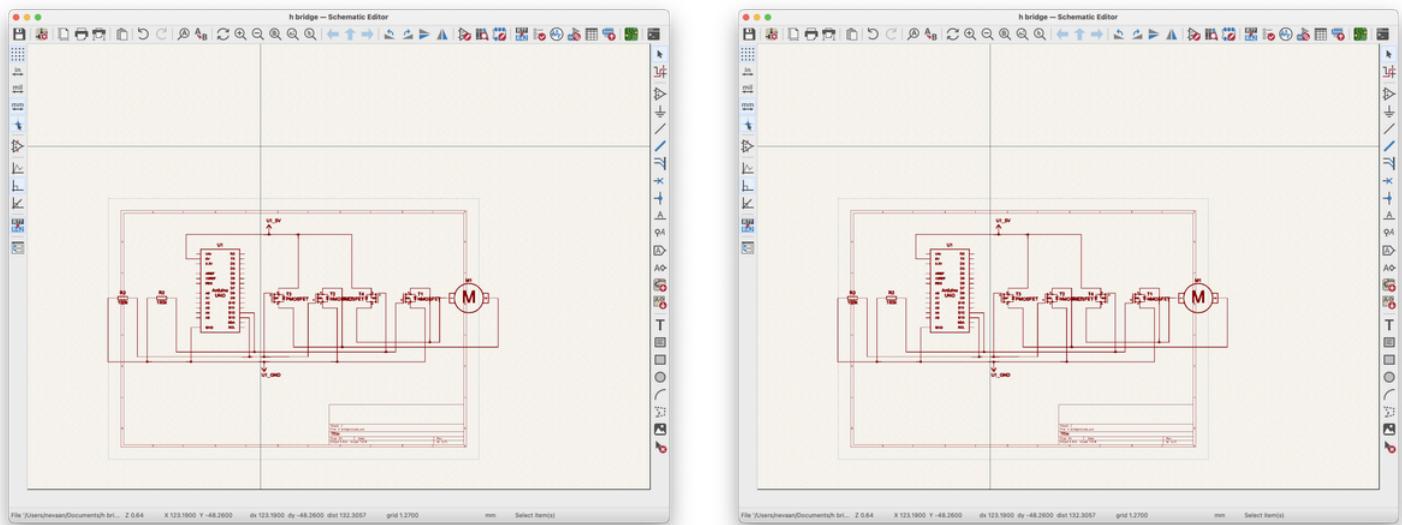


Kicad: Convert a Pdf to a Schematic

By [nevaang08](#) in [CircuitsComputers](#)



Introduction: Kicad: Convert a Pdf to a Schematic

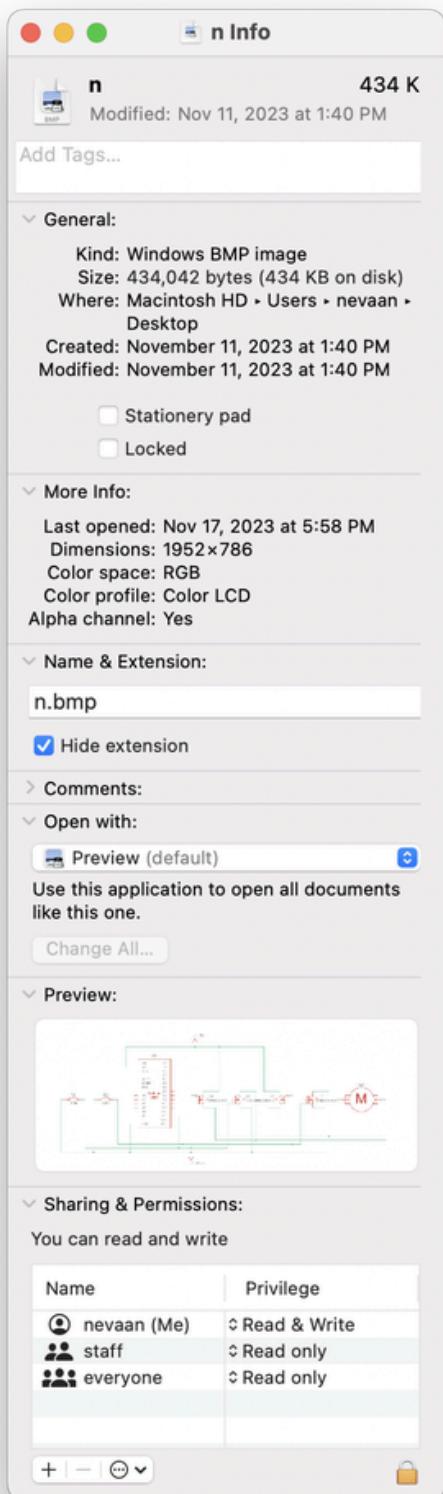


Today I will show you how to make a template of old printed schematic in cicada

Step 1: Scan Schematic

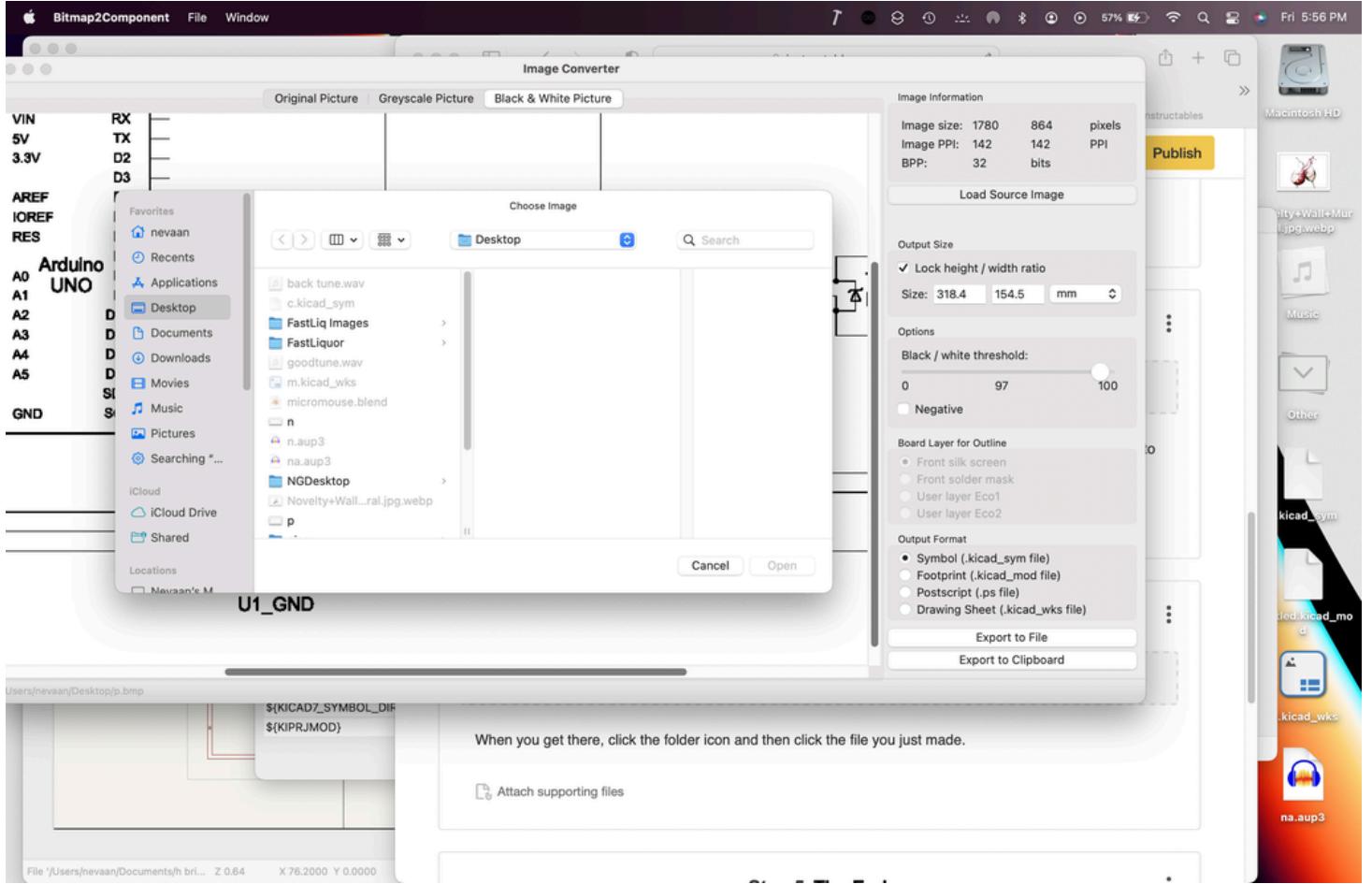
Scan/ take picture the printed schematic

Step 2: Convert the Scanned Schematic to Bitmap



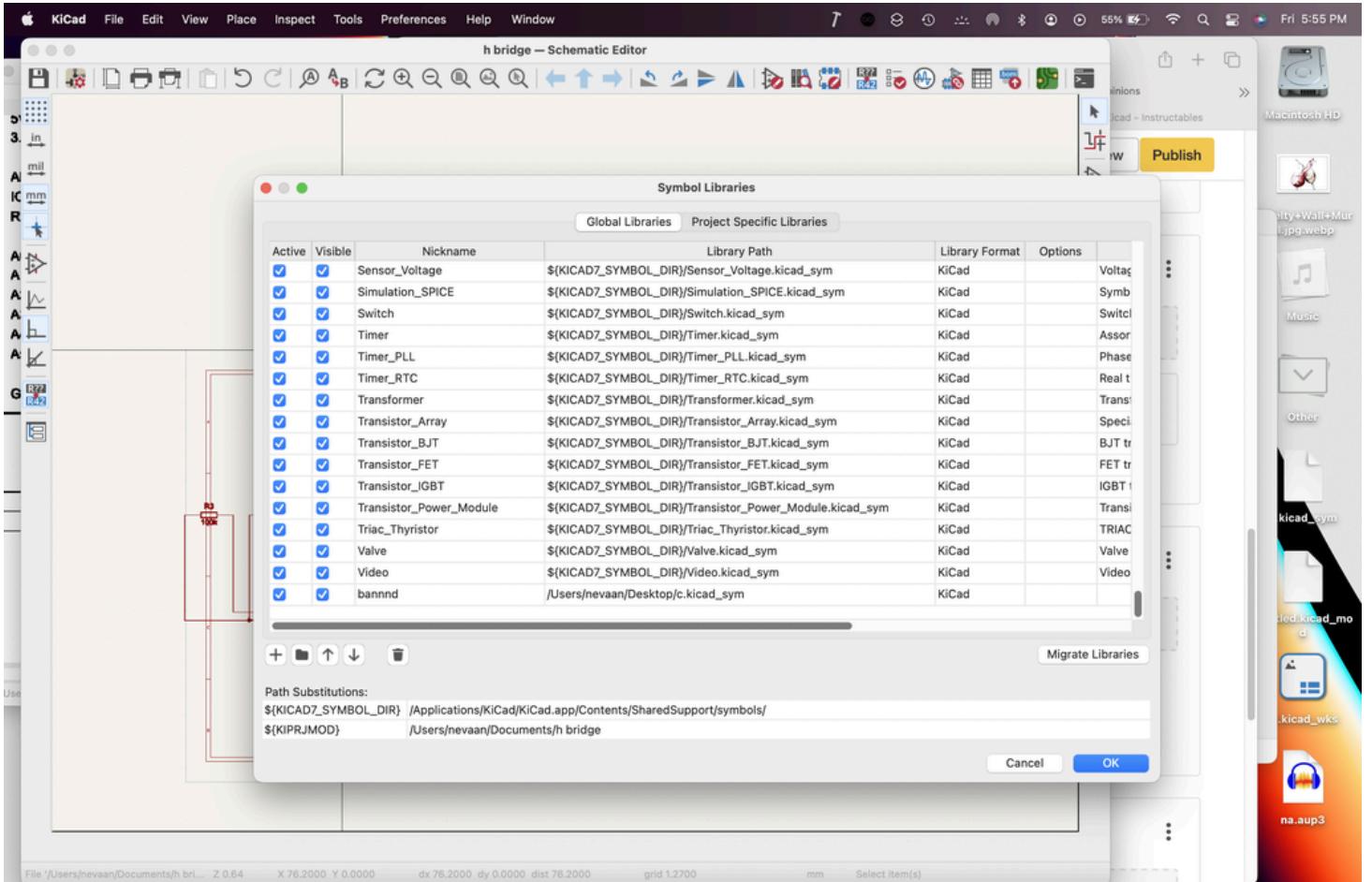
Convert The Scanned Schematic To Bitmap

Step 3: Go to Bitmap2component



Bitmap2component comes with Kicad, so load schematic image into it and click symbol, then click export to file

Step 4: Go to Preferences.Manage Symbol Libraries



When you get there, click the folder icon and then click the file you just made.

Step 5: The End

Now you can add a template of your printed schematic by searching the symbol you made a minute ago