

Creating Artistic PCBs

How to hack the layers of a PCB to make beautiful things. Circuits can be functional *and* elegant.
This is art for nerds.



TECH AS ART

DarcyNeal.com
SporkLogic.com

Two main ways to create PCB art

1) Hack a PCB by assigning graphics to different layers

- Doesn't add to cost
- More complex to design (multi layer)
- Color palette limited to 6 PCB finishes
- Can't do gradients

2) Import multicolor layer to print on your PCB

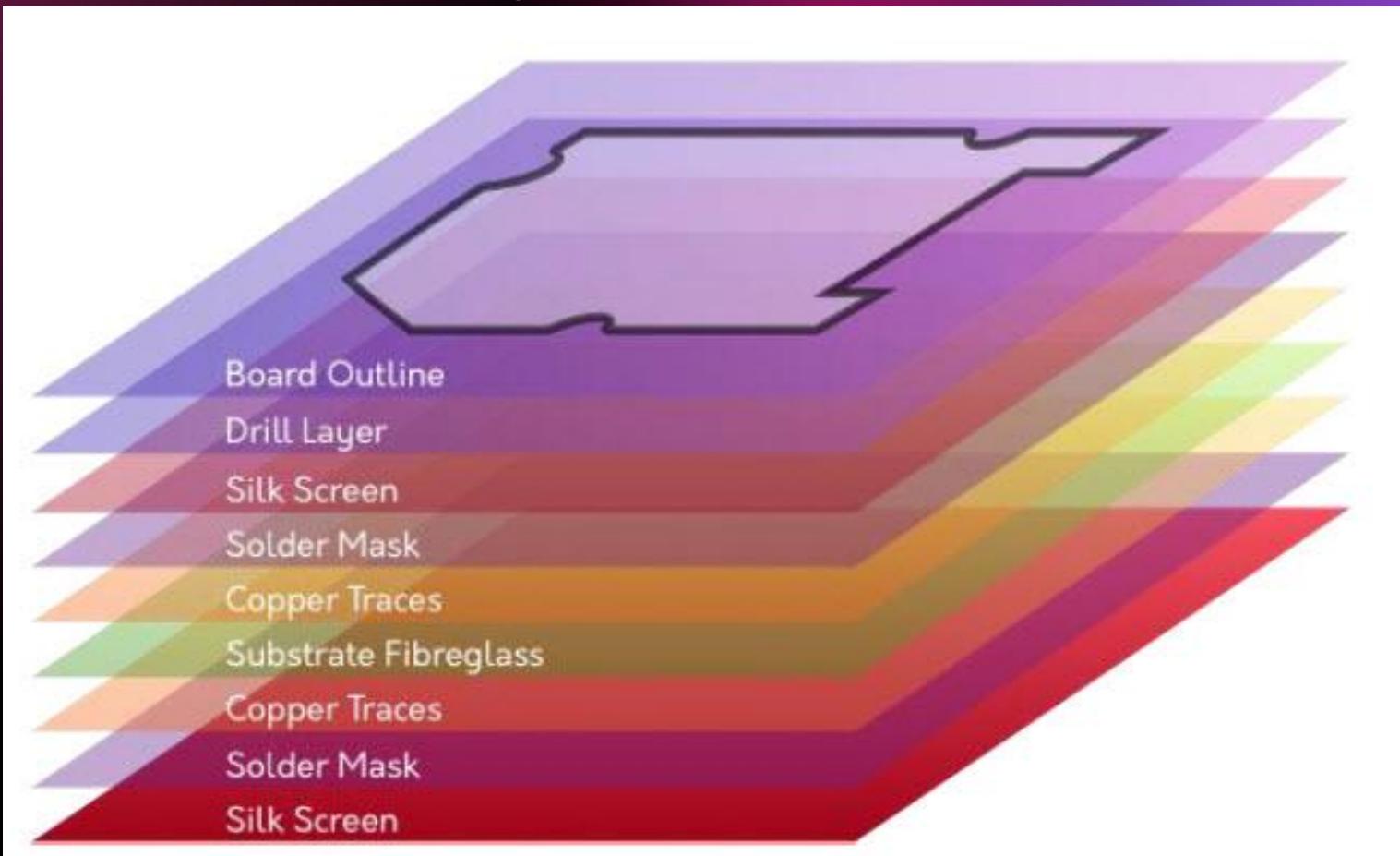
- Can be more expensive
- Typically easier to design for
- Multicolor!

There's other ways to make PCBs fun and interesting to look at. Experiment! Try new things.

How to hack layers of a PCB for art



Layers of a PCB



PCB layer sample combinations



- < Solder mask with no copper underneath
- < Copper plate/traces with solder mask
- < Copper plate/traces no solder mask
- < No solder mask (one side)
- < No Solder mask (either side)
- < Silk screen

Ordering a PCB with bare copper finish (seen here) is called ENIG. You can get this layer plated with a silver color by specifying HASL for the surface finish.

Layers represented in HEX color codes

0x010101	0xDCB950	0xDAC465	0xFDE7BF	0x614228	0xD1E1F2	0xD1DFF4	Solder Mask
0xEAF6F5	0xC99357	0xDAC465	0xFDE7BF	0x614228	0xFF9D07	0xDFA100	Solder Mask + Copper
0x010101	0xD8B24C	0xDAC465	0xFDE7BF	0x614228	0x007A59	0x006F48	Copper
0xEAF6F5	0xA6B66C	0xDAC465	0xFDE7BF	0x614228	0x002678	0x00245B	Copper (Reflective)
0xEAF6F5	0x9EAC6D	0xDAC465	0xFDE7BF	0x614228	0x812A84	0x553967	No Solder Mask (Both Sides)
0xEAF6F5	0xA7A76E	0xDAC465	0xFDE7BF	0x614228	0x1E2732	0x18222E	No Solder Mask (Front Side)
0xEAF6F5	0xA1AC70	0xDAC465	0xFDE7BF	0x614228	0x18222E	0x18222E	Silkscreen

Use these color palettes when designing your PCB art

Art PCB color examples

Purple example uses all features with an ENIG finish (bare copper)

Black examples use all features with a lead-free HASL finish (tin plating over copper)

Solder mask with no copper underneath



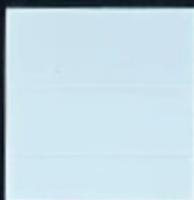
Silk screen

No Solder
mask (either
side)

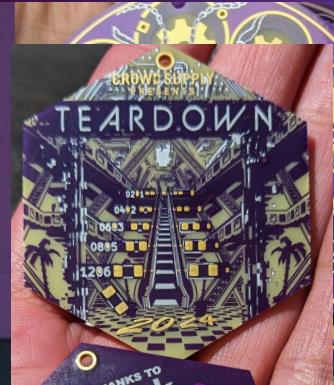
No solder
mask (one
side)

Copper plate
and no
solder mask
(ENIG finish)

Copper plate
with solder
mask



Solder mask with no copper underneath



Design software settings to use

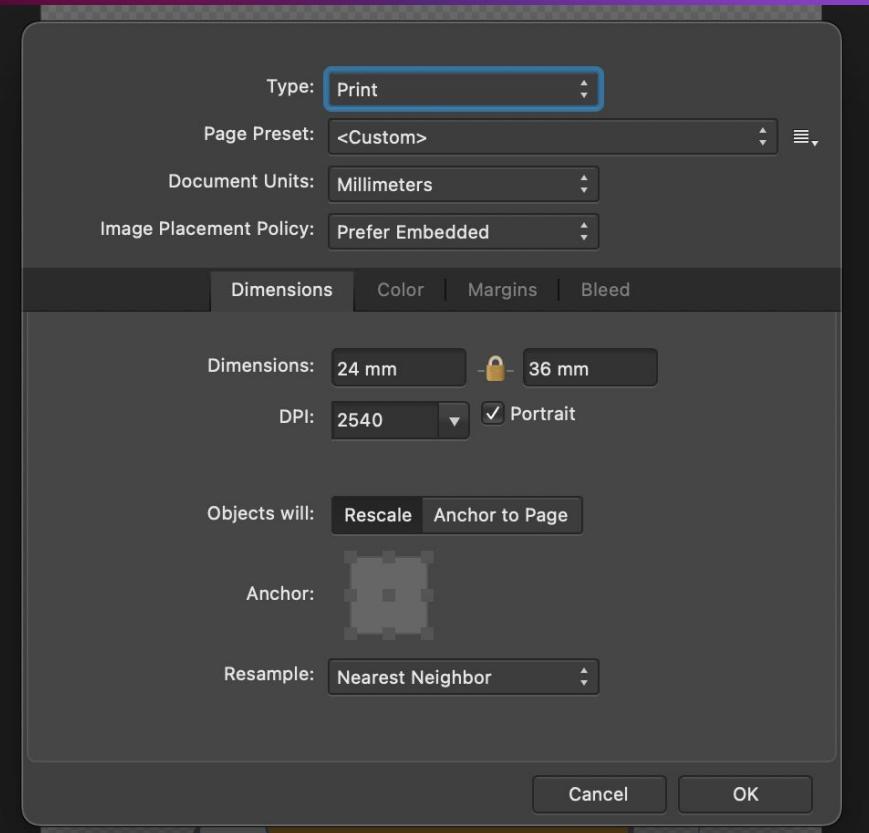
Document settings in Affinity

It's **highly recommended** to change your page settings to use millimeters and 2540 DPI, as shown here>>

2540 DPI happens to be 1000 dots per mm, which helpfully avoids rounding issues when exporting the design from your vector editor and when converting the outline and drills.

Set the dimensions to your PCB size, or be sure to export your files at the same size of your PCB.

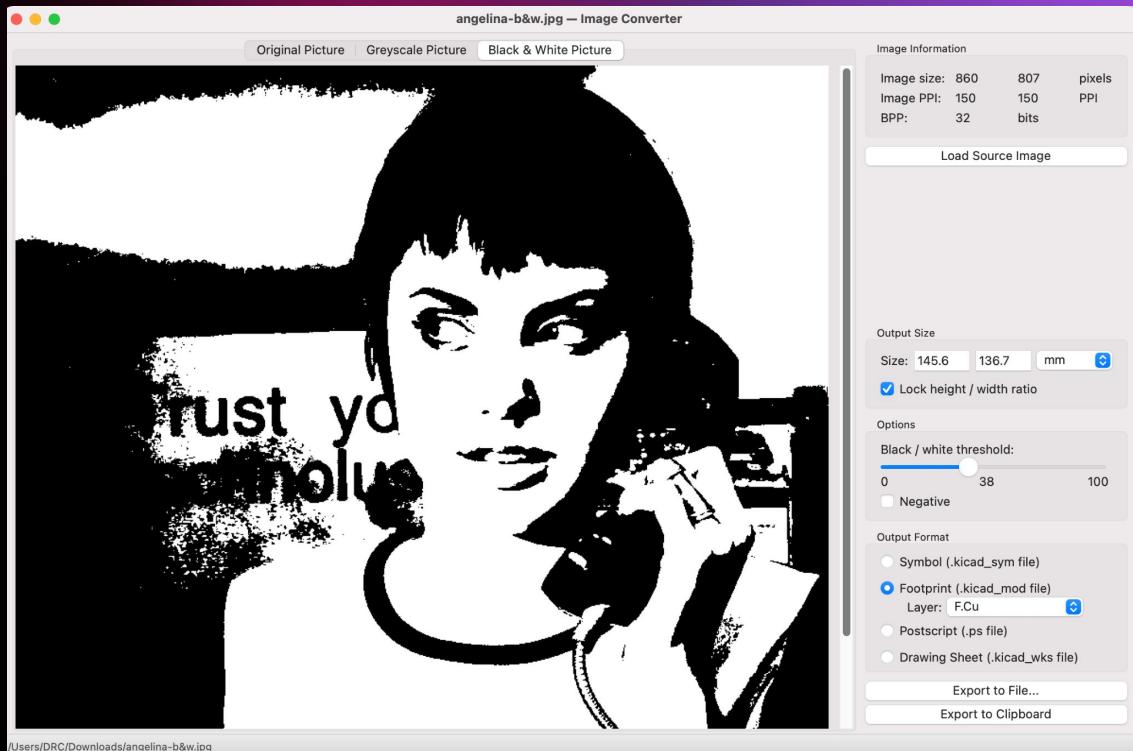
These settings were pulled from [important if you use gingerbread.wntr.dev](https://importantifyouusegingerbread.wntr.dev) for importing your design files.



How to import graphics to PCB layers

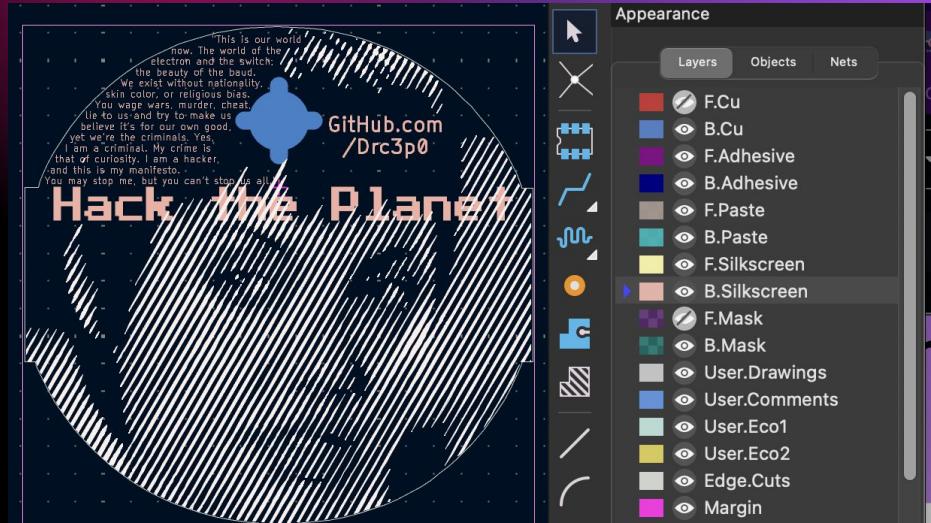
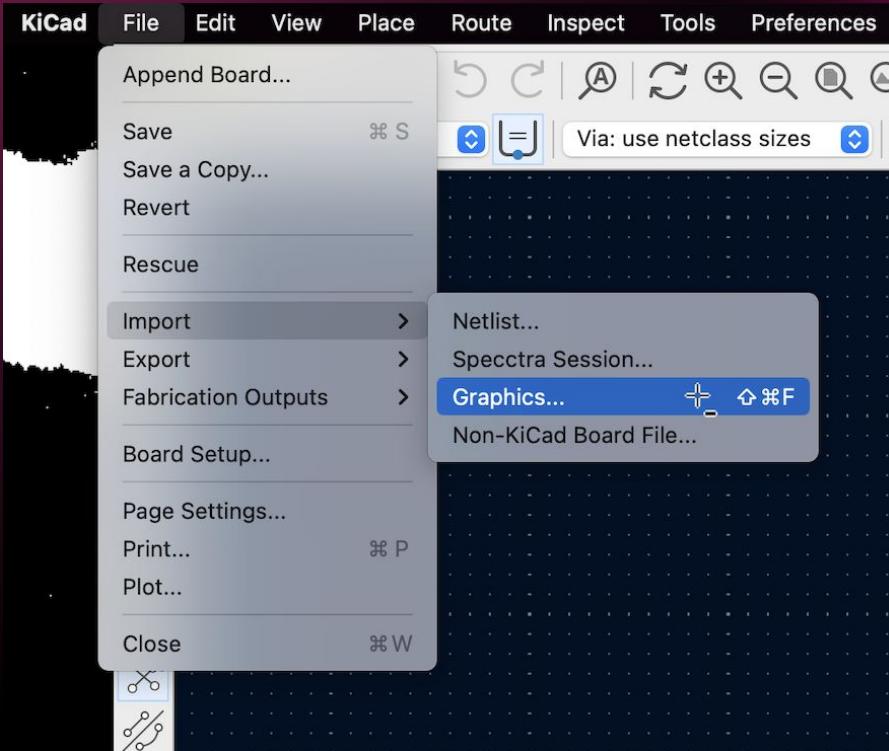
Option A:
Image Converter
in Kicad's project
launcher window

(works with JPEGs
and PNGs)



^ Solid color only. Use halftone/newsprint technique to replicate gradients

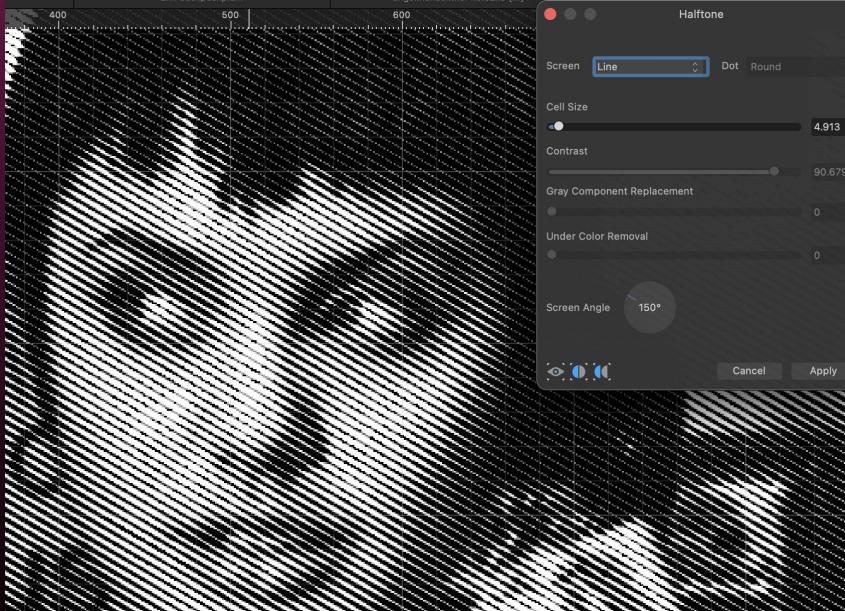
How to import graphics to PCB layers



**Option B:
Import vectors**

Kicad PCB editor: File > Import > Graphics
(must be SVG format)

Tips for adding shading to PCB layers



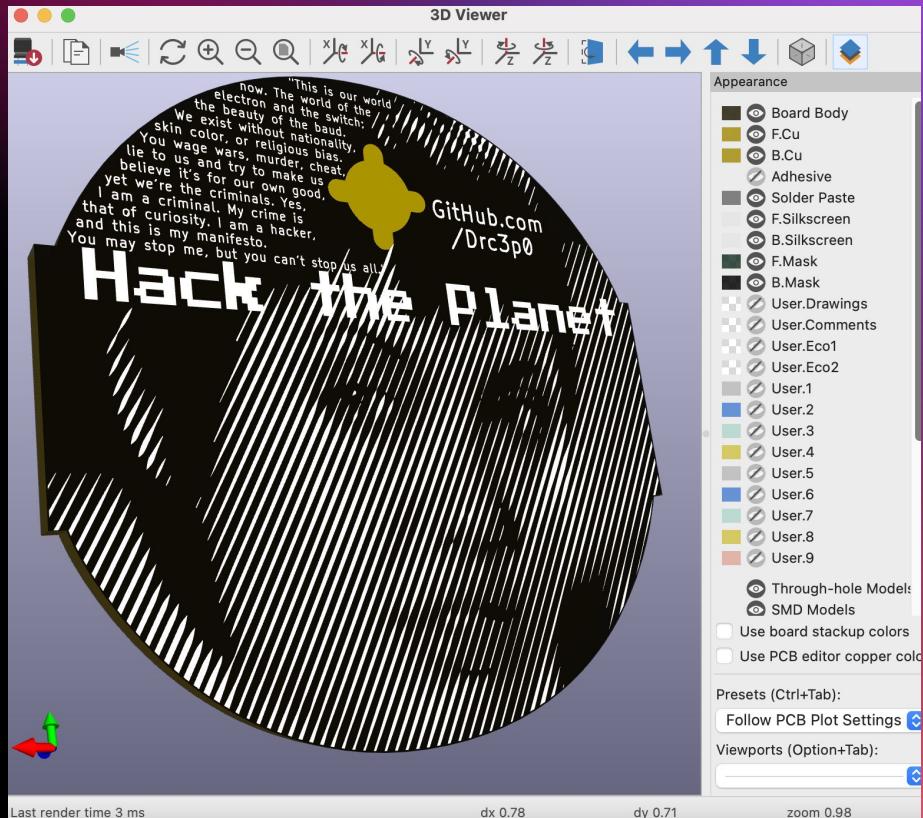
<< Affinity Photo
Filter>Colors>Halftone



Results on a PCB >>
(larger image is ~38 mm wide)

Export files for PCB ordering

1. Use 3D viewer to preview your design before exporting.
2. Create Gerbers
3. Upload to the fab house of your choice.
 - a. Choose soldermask and silkscreen color
 - b. Choose finish (ENIG or HASL)
 - c. Choose "remove mark from PCB"



How to add multicolor printing to PCBs



Prepare your graphics

- 1200 DPI minimum, mm
- Set canvas size to PCB (optional)
- Capture image of PCB to use as a design reference
 - Place PCB image in bottom layer, scaled to PCB dimensions
 - Place multicolor graphics on next layer.
 - Use full bleed off the edge of the PCB
 - Place all images, text, notes, etc.
 - When ready, export as JPG or PNG

Multicolor UV printing: JLCPCB vs PCBWay

They're both good in different ways. It's up to you which one you prefer!

PCBWay

- Flexible ordering process
- €€€ small production runs

JLCPCB

- Must use their EasyEDA software to order multicolor boards.
- €€ small production runs

	JLCPCB	PCBWay
Board qty / size	50 pcs / 100 x 150 mm	50 pcs / 100 x 150 mm
Regular Silkscreen (No UV color)	\$63.70 5-6 days	\$118.95 1-4 days
Single Sided color	N/A	\$140.71 6-7 days
Double Sided Color	\$69.95 6-7 days	\$152.87 6-7 days
ENIG	Default choice when ordering UV color	\$203.30
Soldermask color	White only	Can choose any color (white used when full coverage multicolor)

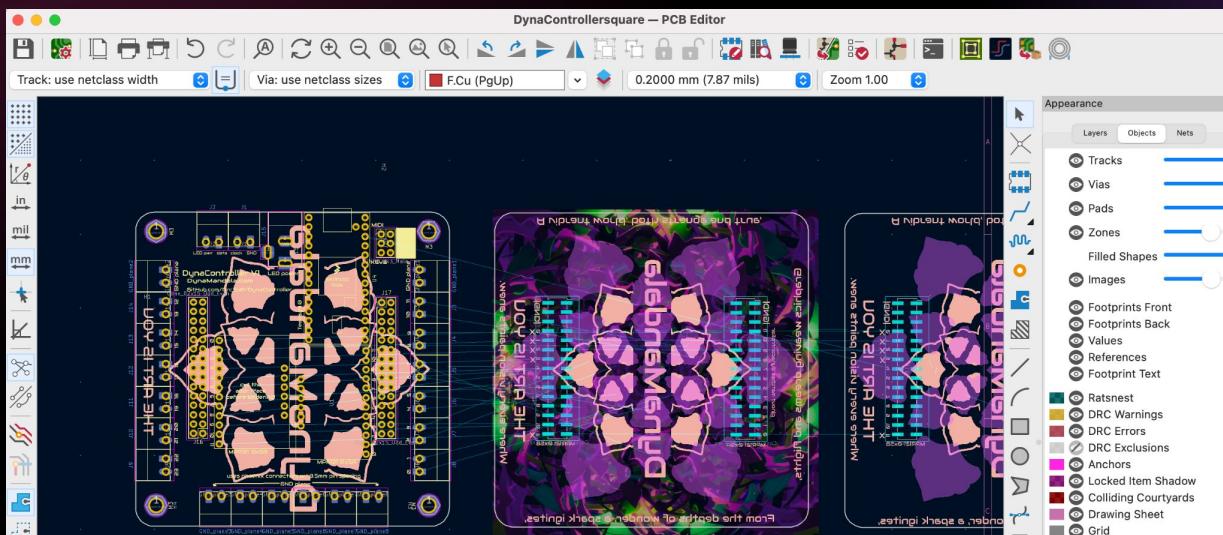
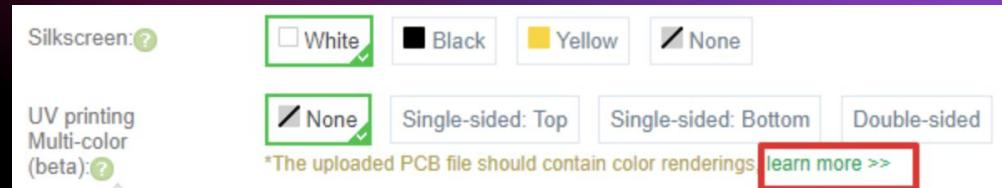
(These \$'s came from Joseph Long's Hackaday 2024 Supercon talk!)

<https://hackaday.com/2025/03/26/supercon-2024-a-new-world-of-full-color-pcb/>

Techniques for creating multicolor layers for PCBWay

Design workflow:

- Create Gerber files as usual
- Create multicolor files (top, bottom, and positioning image)
- .zip gerber files and image files together
- Order as usual, using the options seen here >>



Design tip:
Bring graphics into Kicad to preview (optional)

- Place > Place Reference Images
- Change opacity
- (this reference image will not be included when exporting gerbers)

Techniques for creating multicolor layers for PCBWay

Detailed guide
to ordering
multicolor with
PCBWay



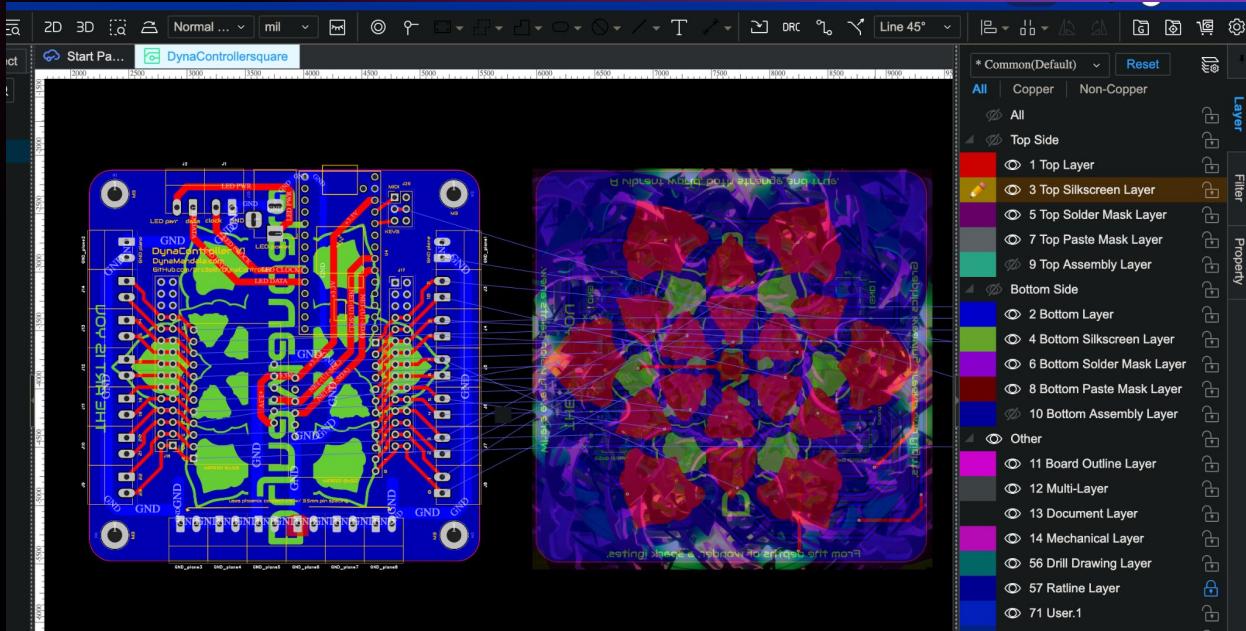
Creating multicolor UV layers for JLCPCB

Files **have** to be exported from EasyEDA Pro. (pro.easyeda.com/editor)

You *can* import Kicad files, (file>import>Kicad) but there may be issues when importing that need clean up in EasyEDA.

In EasyEDA

- Import image(s) to silkscreen layer(s)
- Check box for “appy jlcpcb colorful silkscreen”
- View 2D or 3D to verify results
 - (set silkscreen view type to “colorful”)
- Export > PCB Fabrication File (Gerber)
- Select “multicolor silkscreen” when ordering



Creating multicolor UV layers for JLCPCB



Detailed guide
to ordering
multicolor with
JLCPCB

Helpful things for artistic PCB creation

Tools for viewing PCB layers:

Gingerbread.wntr.dev for viewing affinity/inkscape files, and importing as a single file into Kicad.

Kicanvas.org for viewing Kicad PCB files from GitHub links

Design software for creating art:

Inkscape - free vector design tool

Affinity Photo/Designer - competitor to Adobe Photoshop/Illustrator. One time payment model <3

Ordering PCBs:

Aisler (EU based) (green PCBs only, no multicolor options available)

OshPark (US based) (purple PCBs with gold finish only) (no multicolor options available)

JLCPCB or **PCBWay** for ordering prototypes/production runs and multicolor boards (China)

Awesome guides n talks about creating PCB art:



Supercon talk
about multicolor
PCBs



[hackster.io](https://hackster.io/PCBWay/multicolor-guide)
PCBWay
multicolor guide

Please donate to support this workshop

Your support makes these workshops sustainable. <3

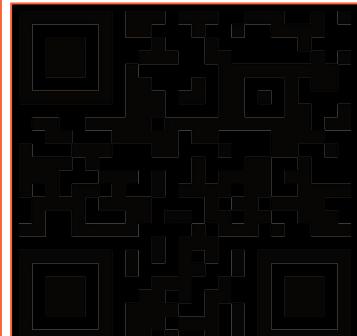
Call HSNL
(4765) to reach
the village for
donations after
this workshop.
Ask for Darcy
(or come by!)



Donate with Stripe



Donate with PayPal



Donate with Ko-fi

Year



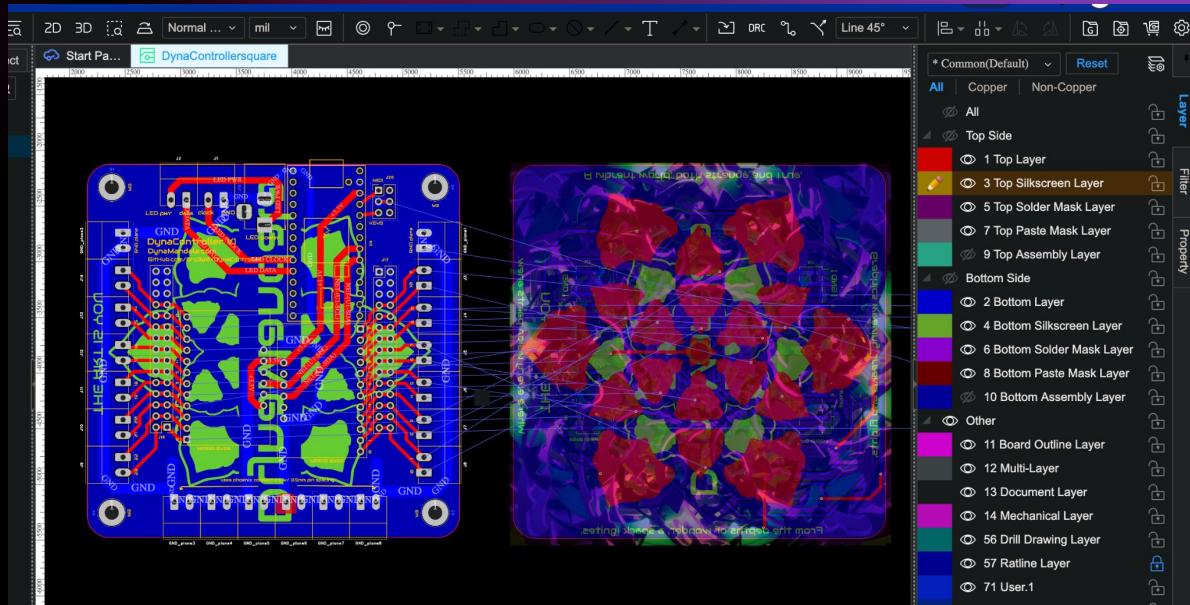
Creating multicolor UV layers for JLCPCB

Files **have** to be exported from EasyEDA Pro. (pro.eeasyeda.com/editor)

You *can* design your board in Kicad, then import to their software, (file>import>Kicad) but there may be issues when importing that you'll have to clean up in EasyEDA.

In EasyEDA

- Import image(s) to silkscreen layer(s)
- Check box for “appy jlpcpcb colorful silkscreen”
- Adjust the image within EasyEDA
- View 2D or 3D to verify results
 - (set silkscreen view type to “colorful”)
- Export > PCB Fabrication File (Gerber)
 - Check the multicolor silkscreen option
 - Soldermask will be white



Detailed guide:

<https://jlpcpcb.com/help/article/How-to-design-multi-color-silkscreen-using-EasyEDA>

Techniques for creating UV layers for PCBWay

Design workflow:

- Create PCB (almost always KiCad around here)
- Generate Gerber files as usual
- Bring scaled image of PCB into design software.
 - Max size per PCB: 270 x 470 mm
 - Don't mirror backside image
 - Avoid placing ink in areas with soldermask openings
 - Indicate position with alignment holes or a reference placement image
 - Accepted image formats: AI, PDF, JPEG, PNG, or TIFF
- Create Image files for PCBWay (top, bottom, and positioning image)
- Compress gerber files and image files into a zip file
- Upload .zip file to PCB order page
- Select "UV Printing Multi-Color"
- You can choose the soldermask color, but white is usually best for contrast

Bring graphics into Kicad for a preview (optional)

Place > Place Reference Images

- Import reference image
- (this reference image will not be included when exporting gerbers)

<https://www.pcbway.com/blog/News/>

[Unlock Color PCB Printing with PCBWay 0939d559.html](Unlock%20Color%20PCB%20Printing%20with%20PCBWay%200939d559.html)

