

Wai

Your Manufacturer Is Stupid

Help Them

Wai

Chris Denney

CTO - Worthington Assembly



CircuitHub

Chris Denney



CircuitHub

Chris Denney

Jerk Who Tells You There's A Problem

Wai

Your Manufacturer Is Stupid

Have to act stupid

Wai

Your Manufacturer Is Stupid

Play dumb on new products



Information about your design is unclear

Phone calls needs to be made

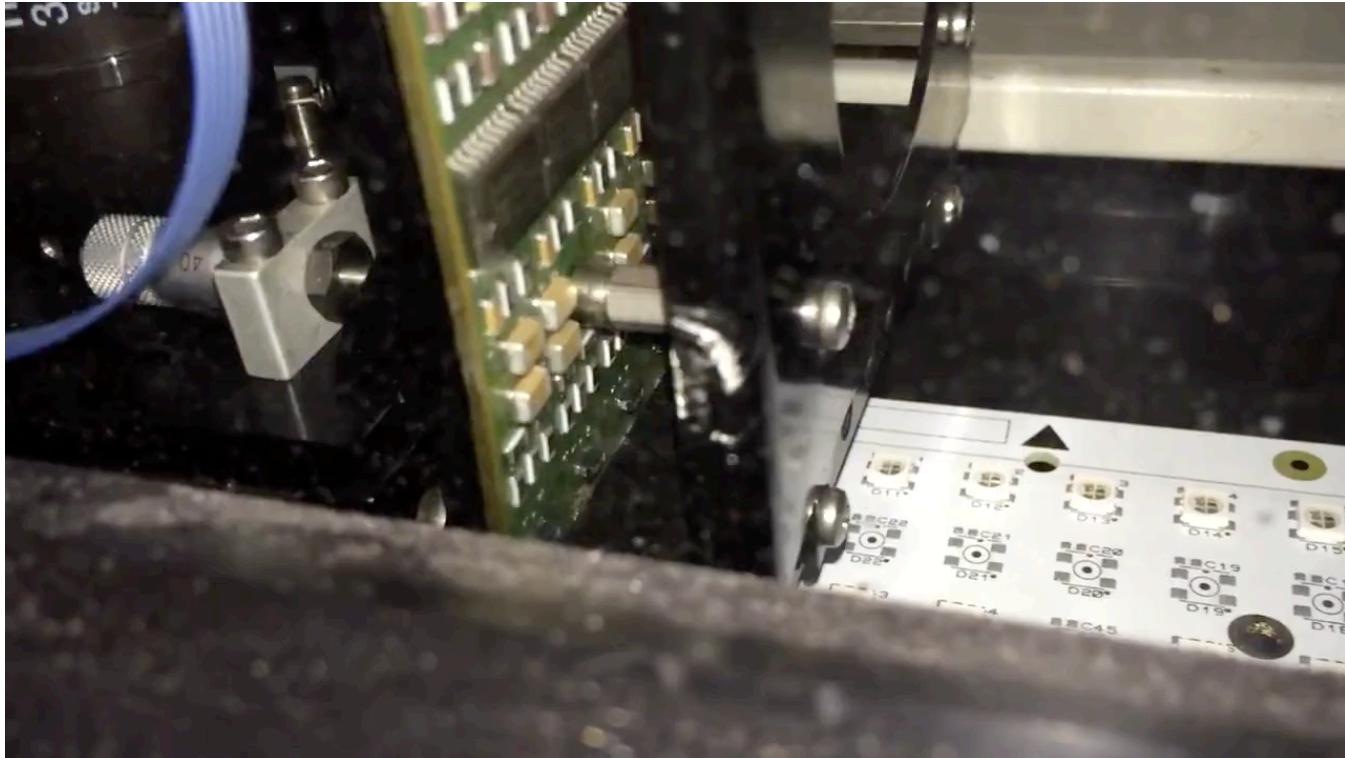


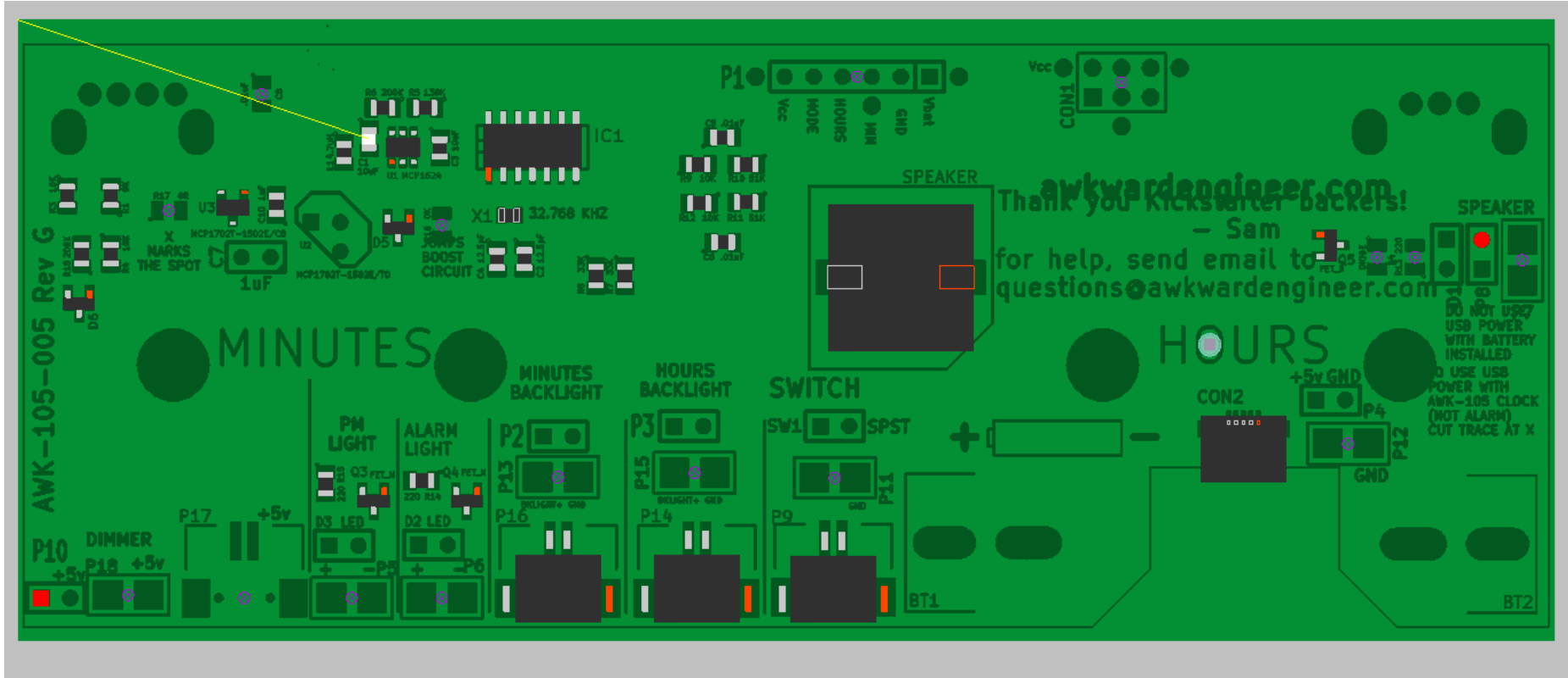
I hate people calling me.
I hate calling people.

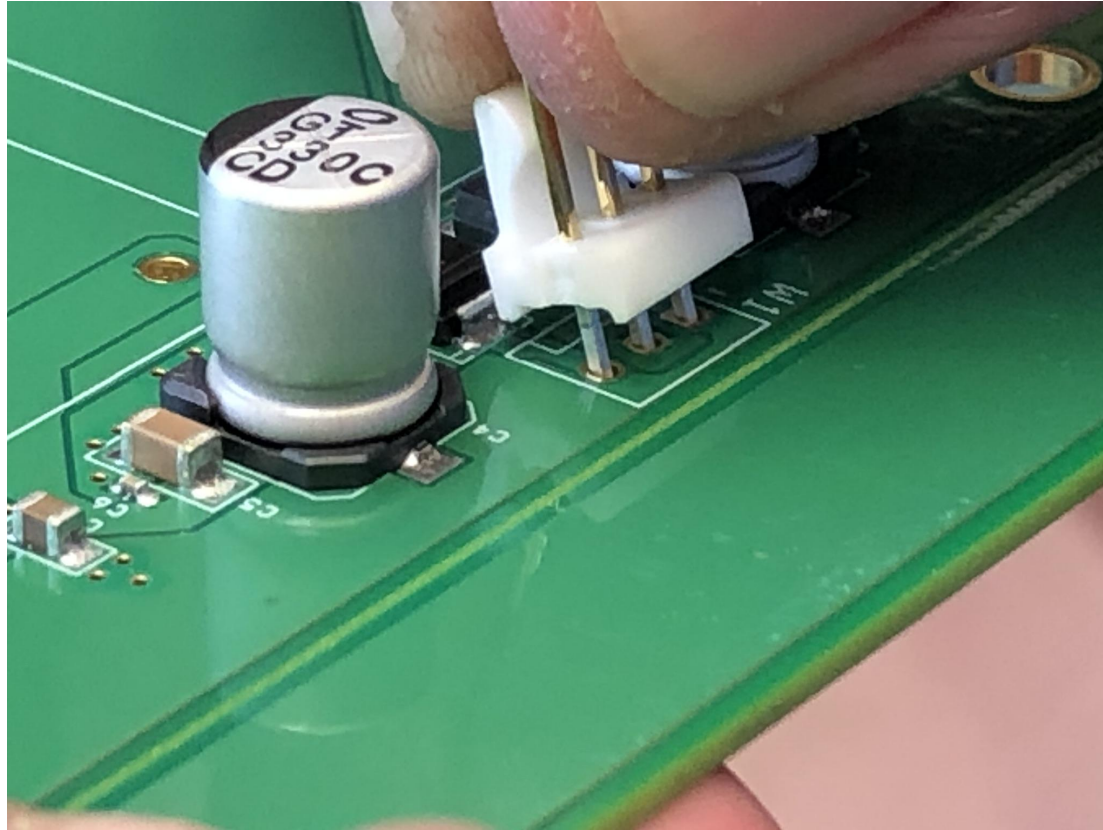




But machines build things!











1. Identifying polarity of components
2. Silkscreen legibility
3. Panelization
4. PCB properties
5. Specific manufacturer's part numbers



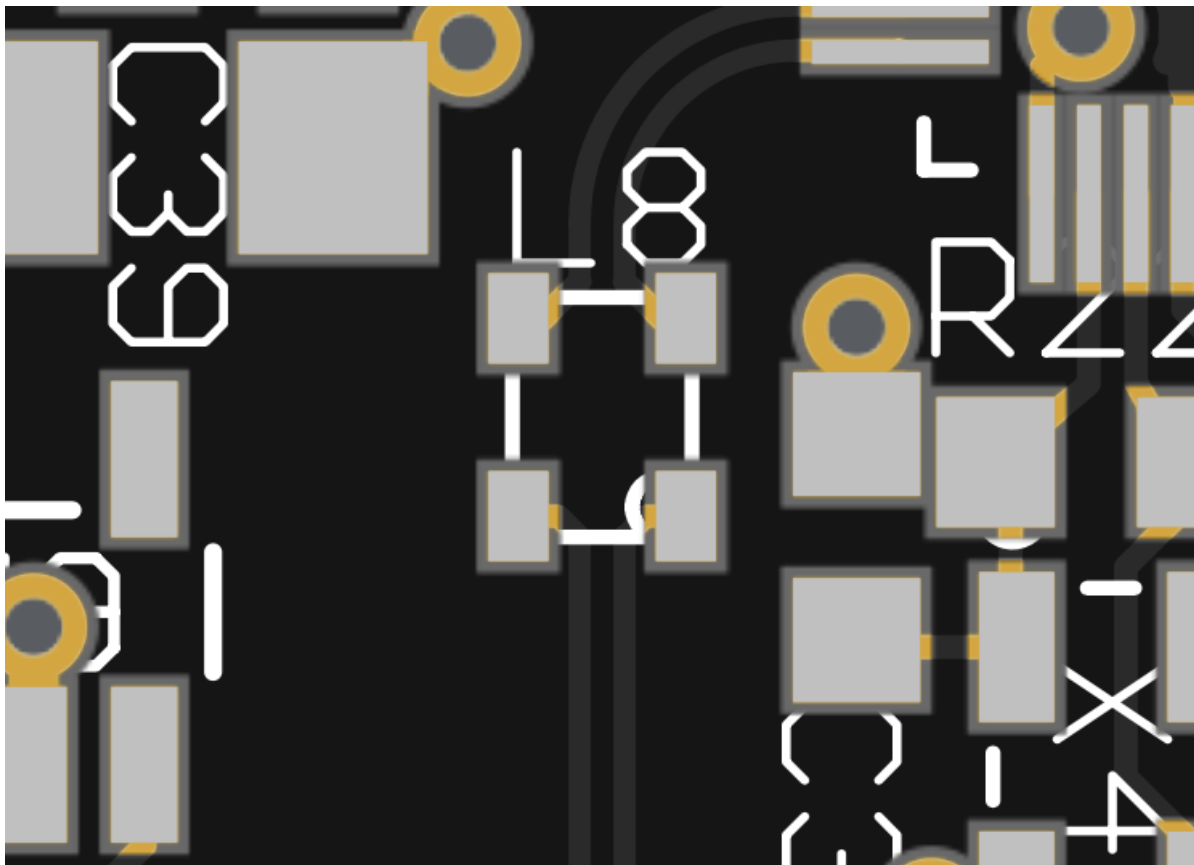
1. Identifying polarity of components
2. Silkscreen legibility
3. Panelization
4. PCB properties
5. Specific manufacturer's part numbers

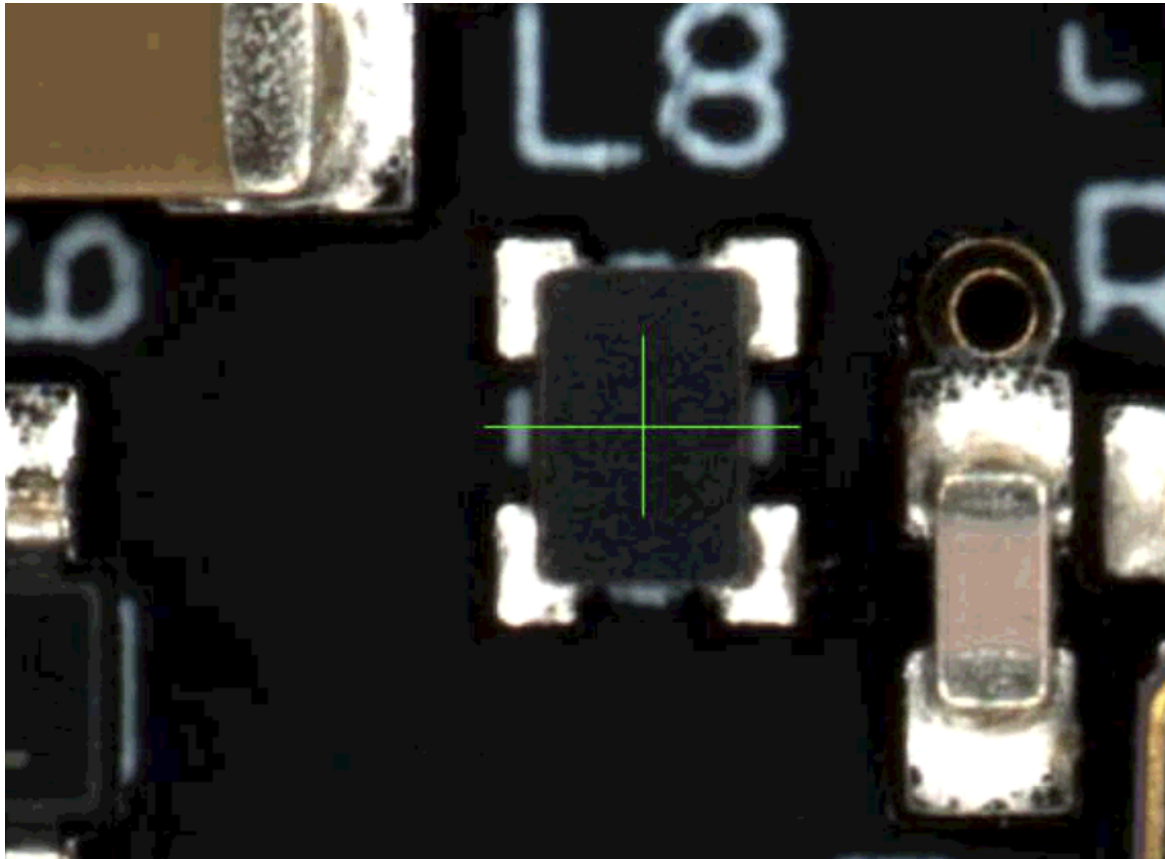


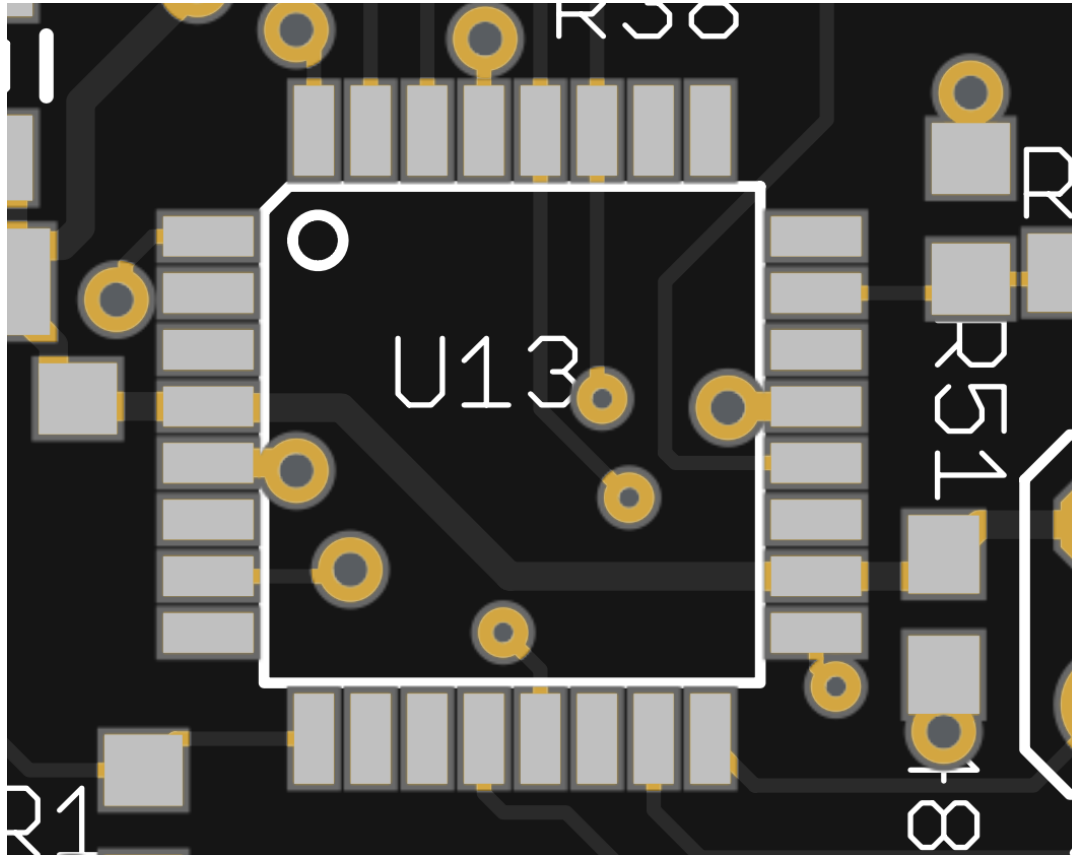
The number 1
problem

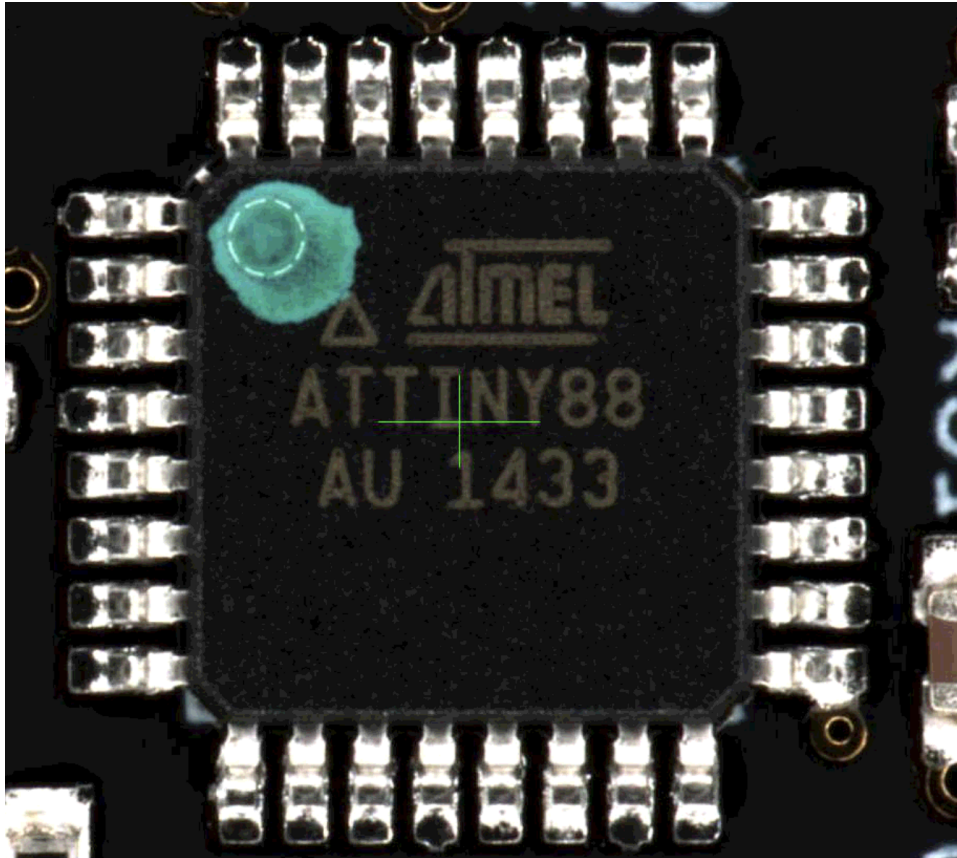


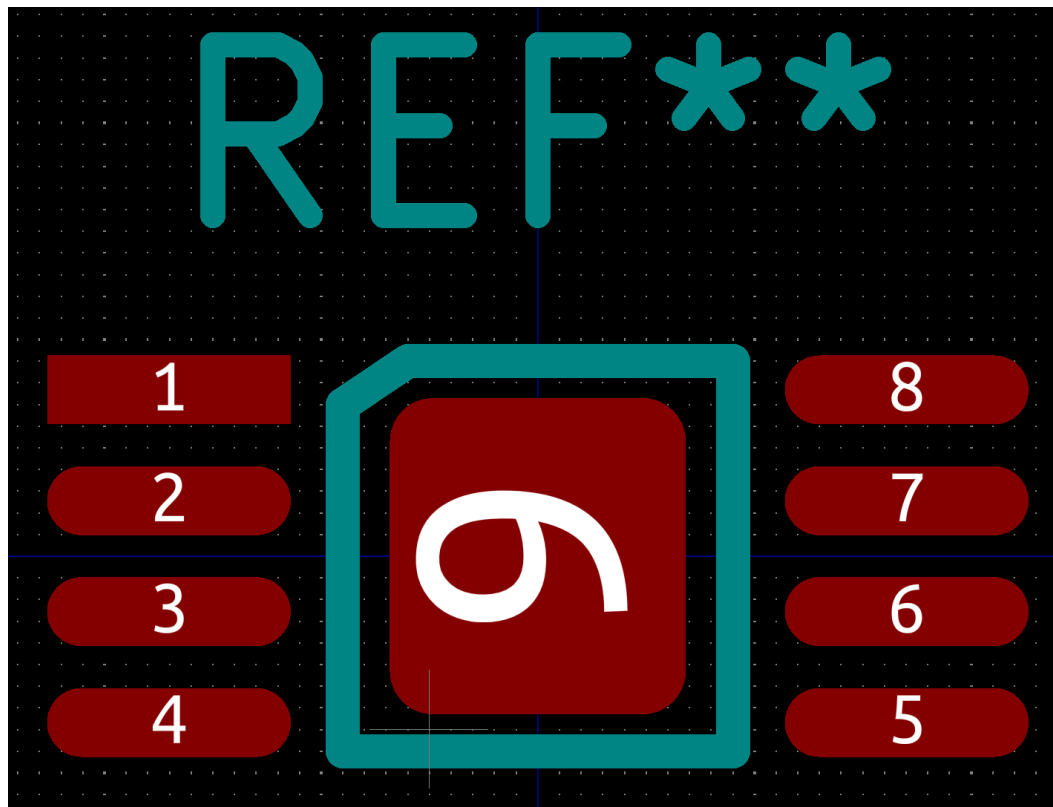
- Polarity of all IC's
 - What is this IC's reference designator?
 - Which pad is Pin 1
- Polarity of all diodes
- Polarity of all LED's
 - This LED here, what reference designator is it?
 - Which pad is the cathode?

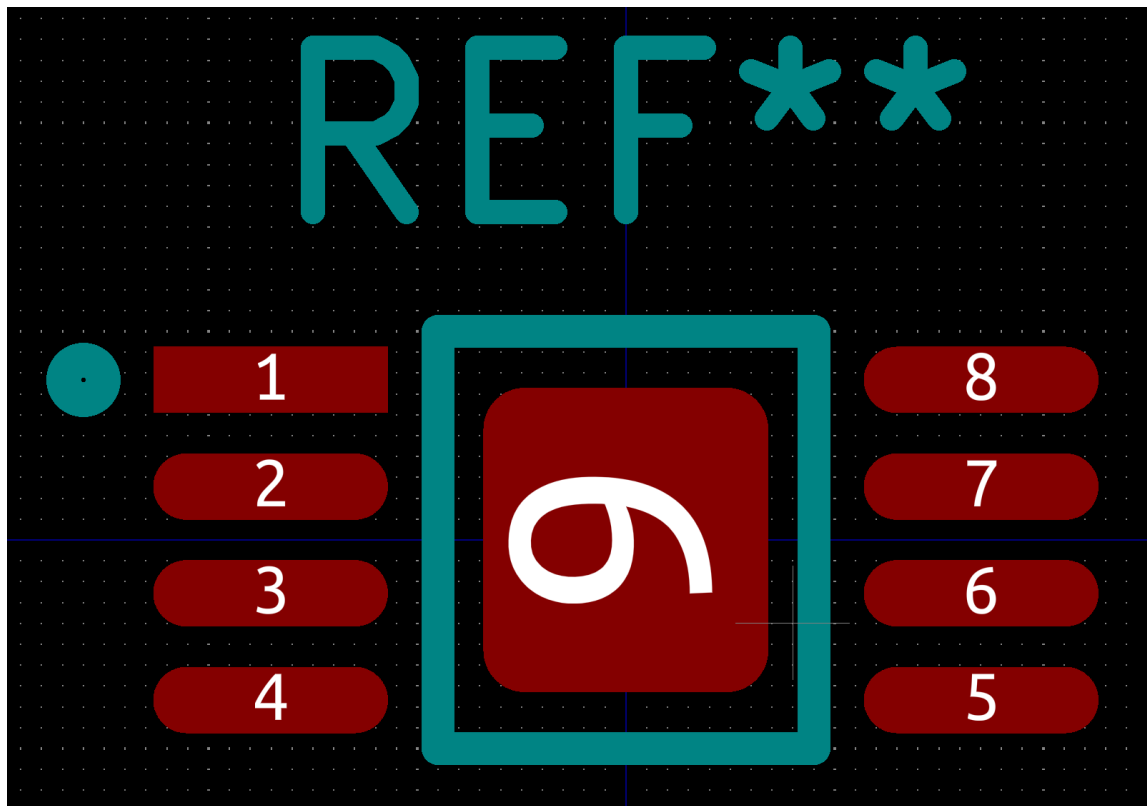


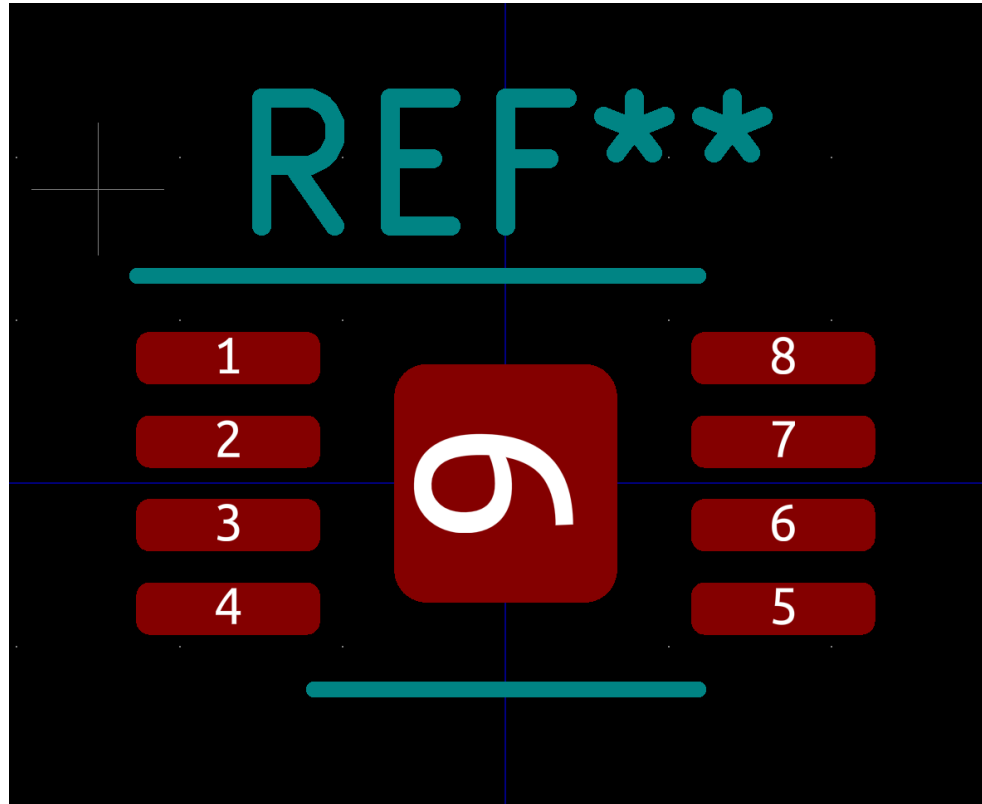


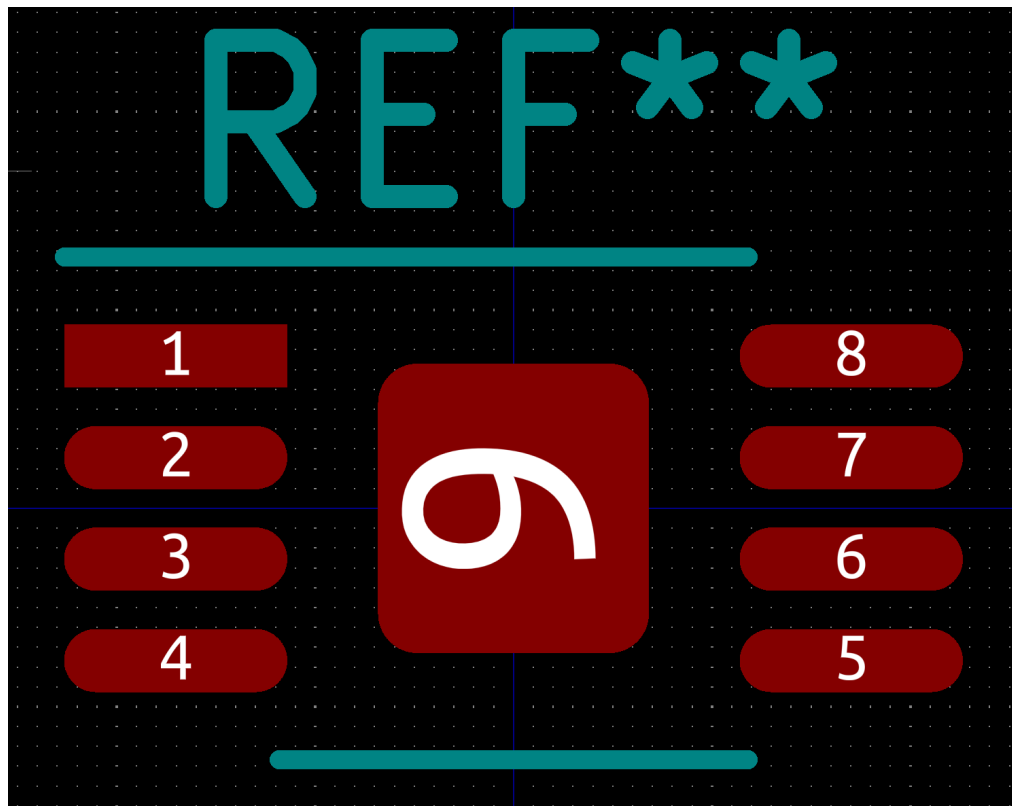


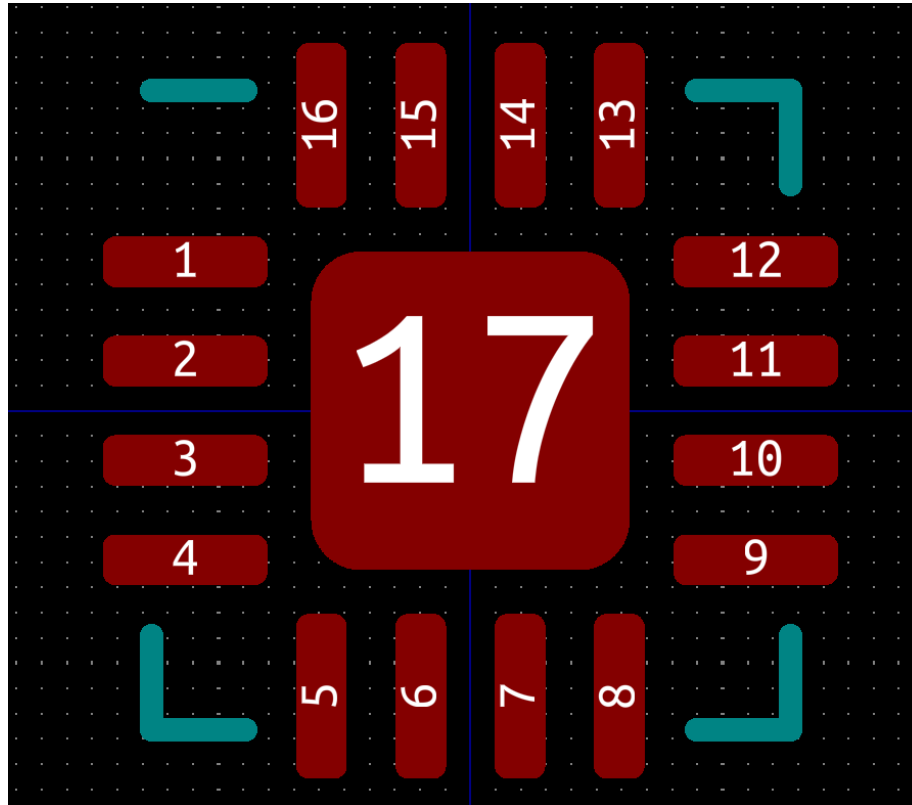


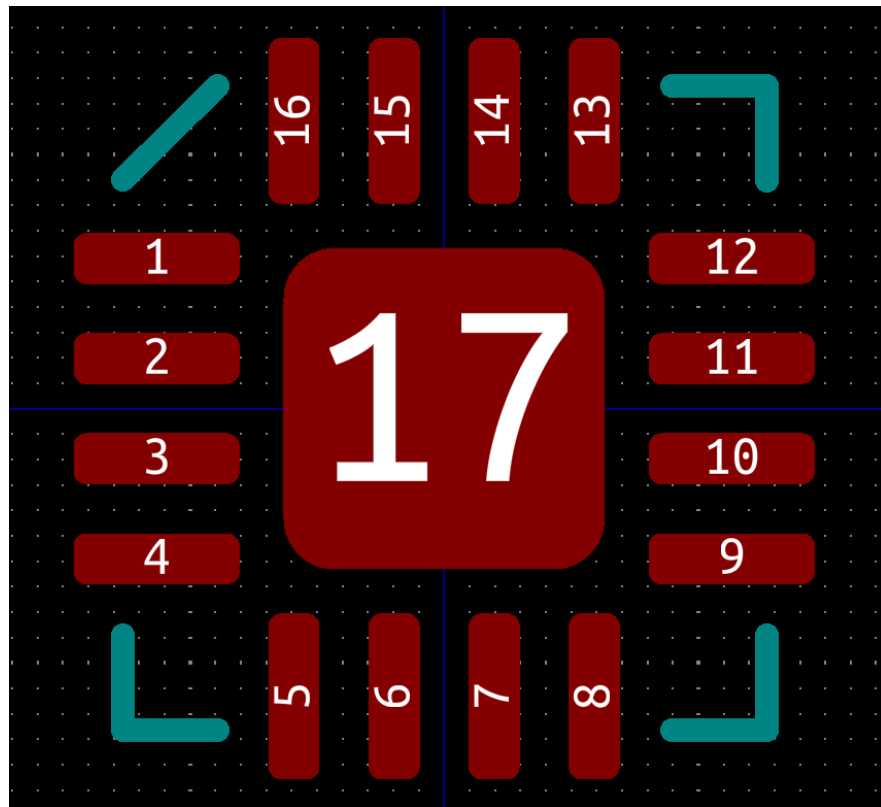


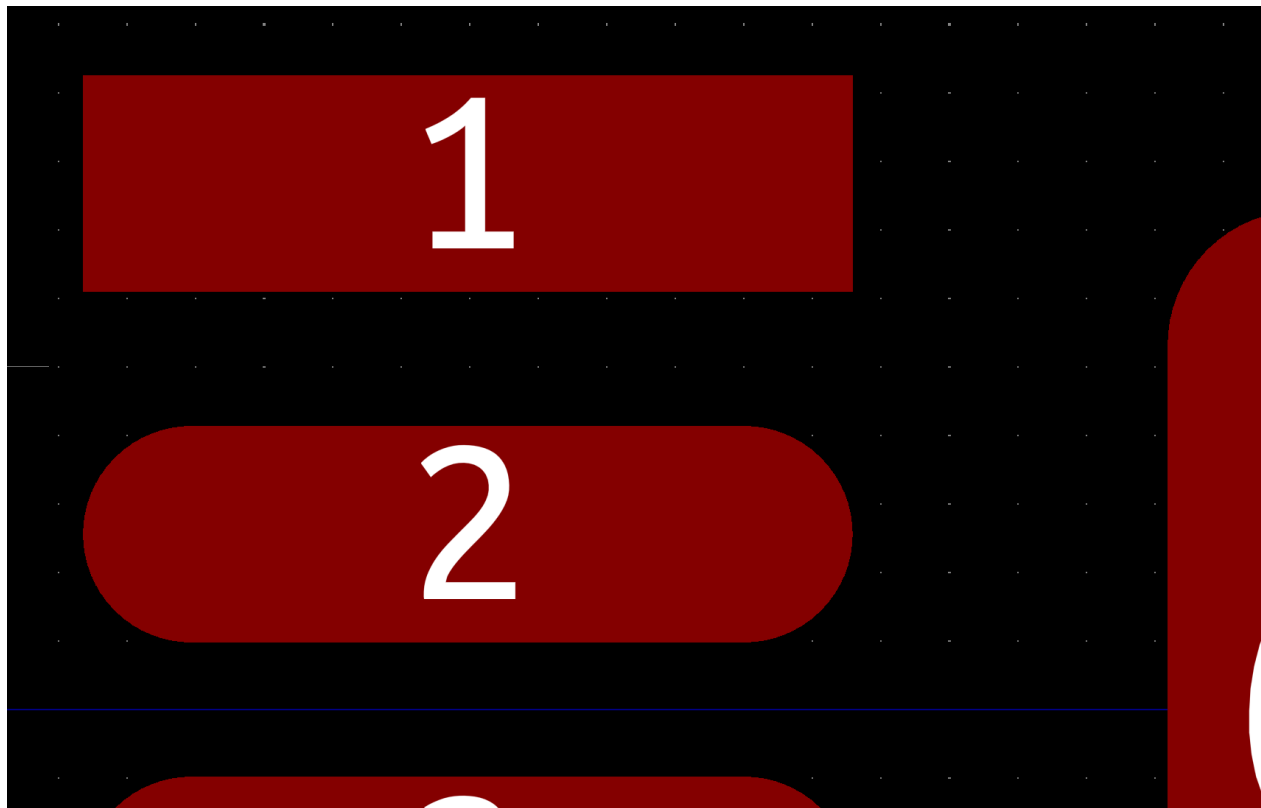


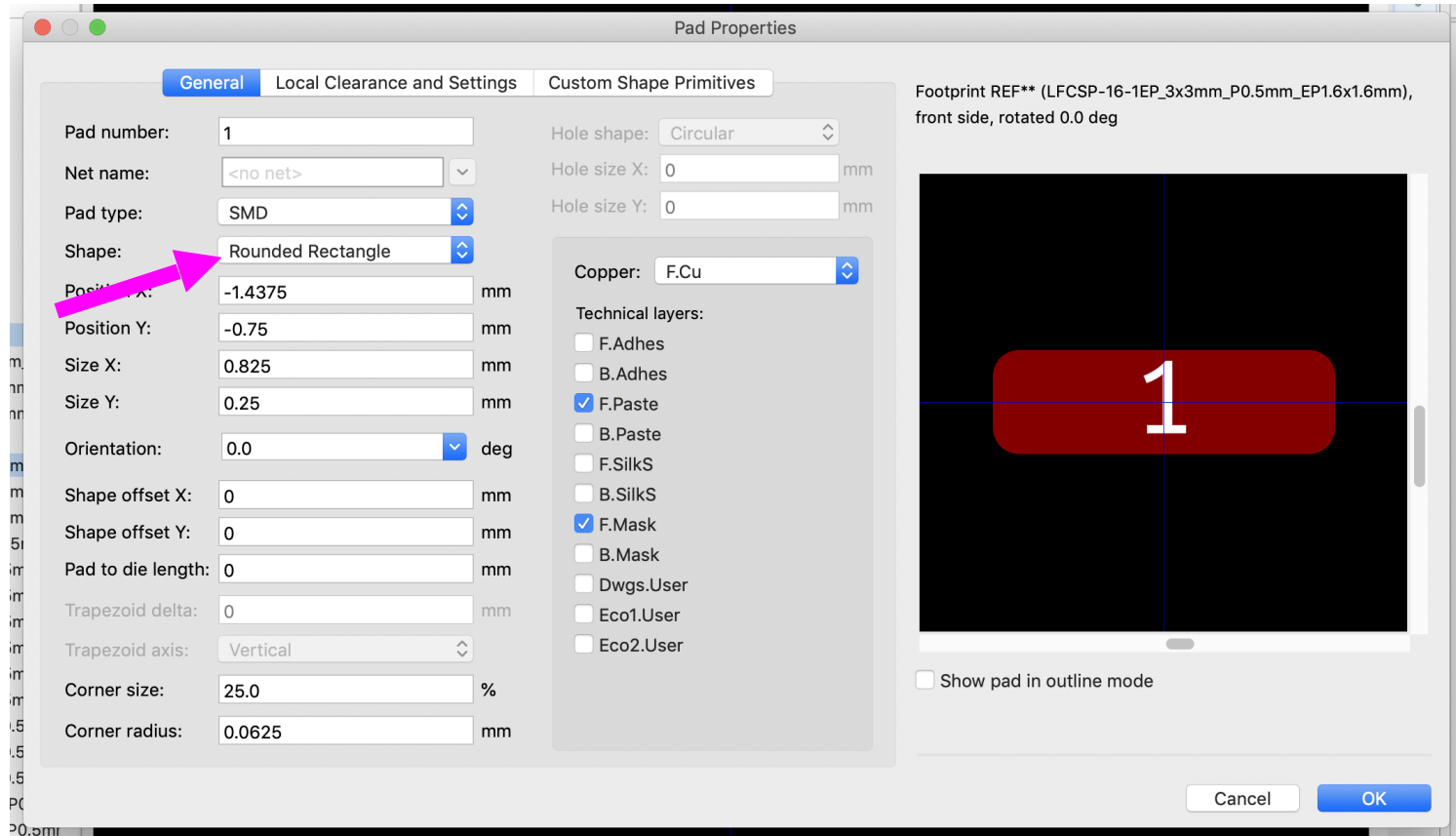


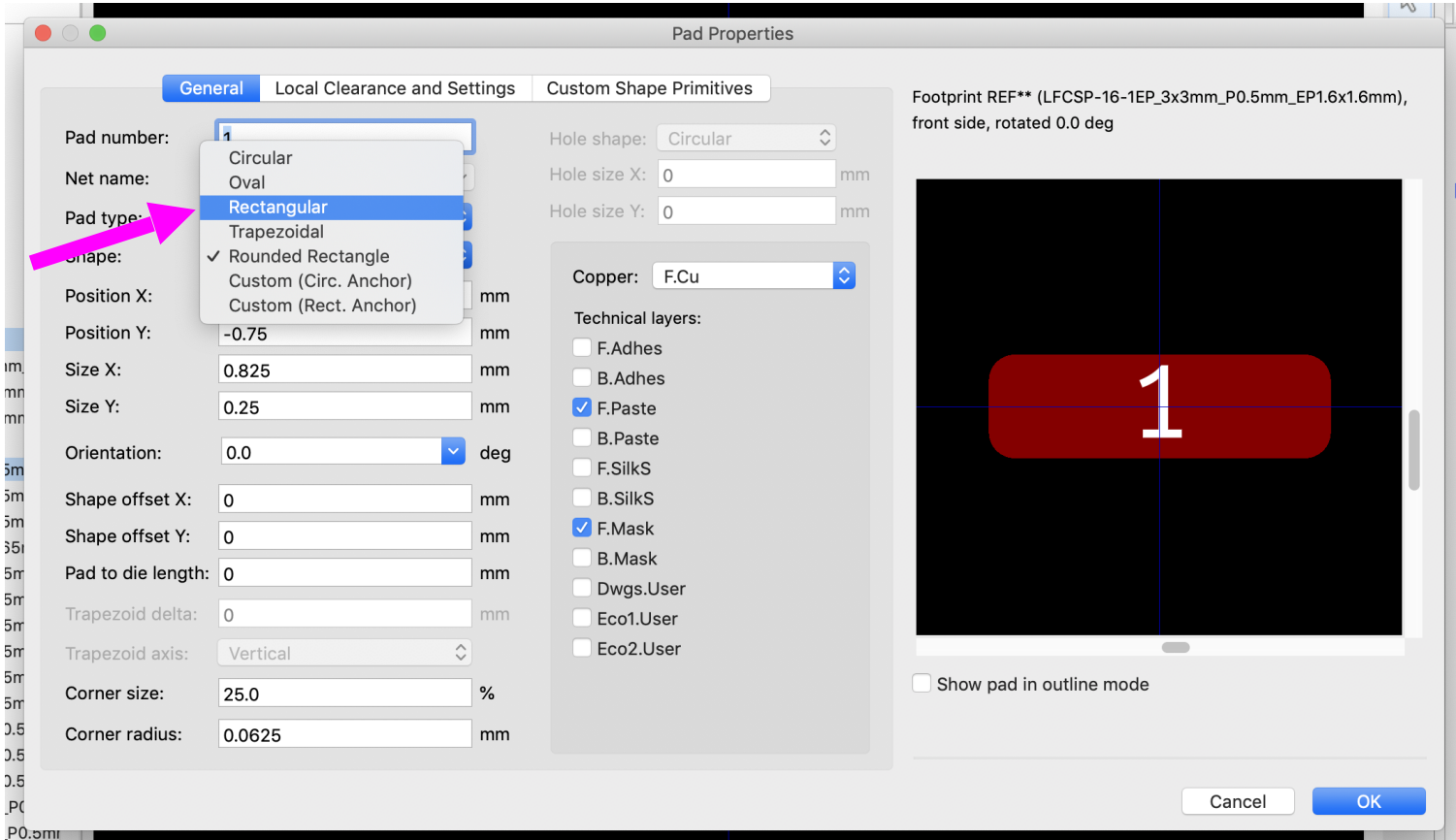


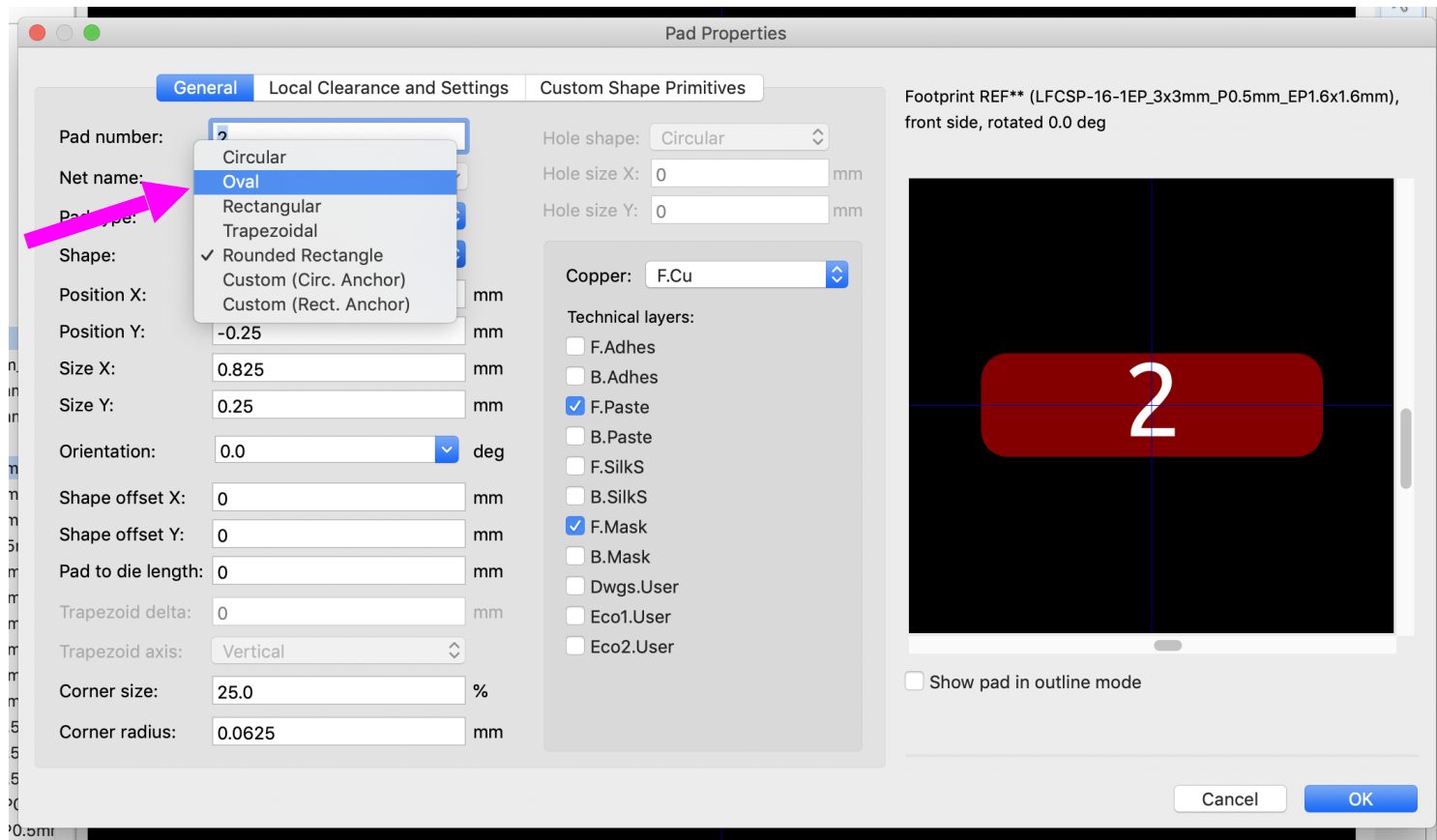


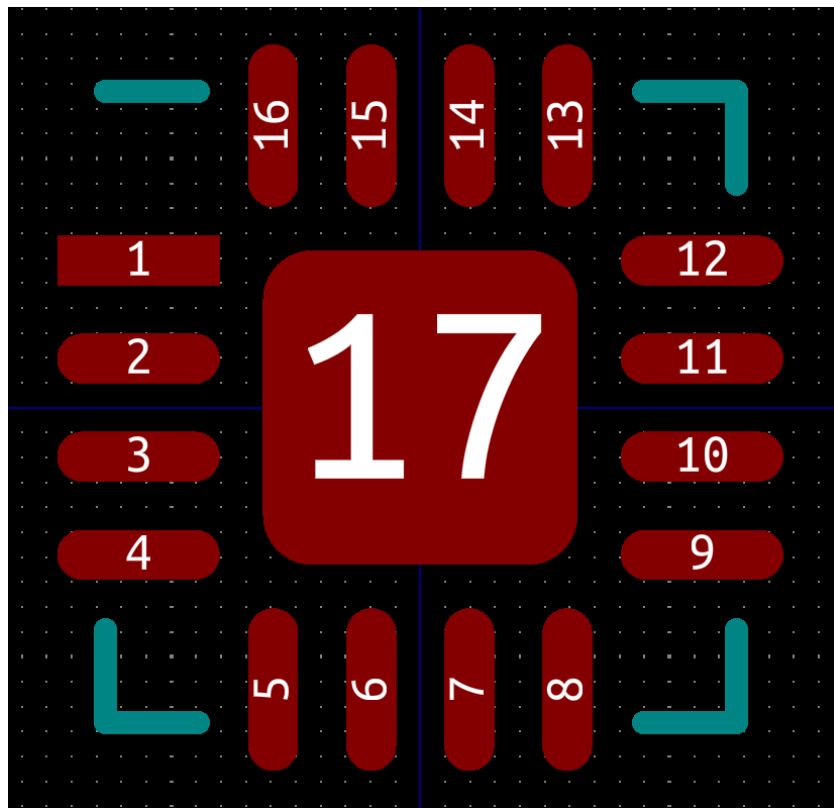


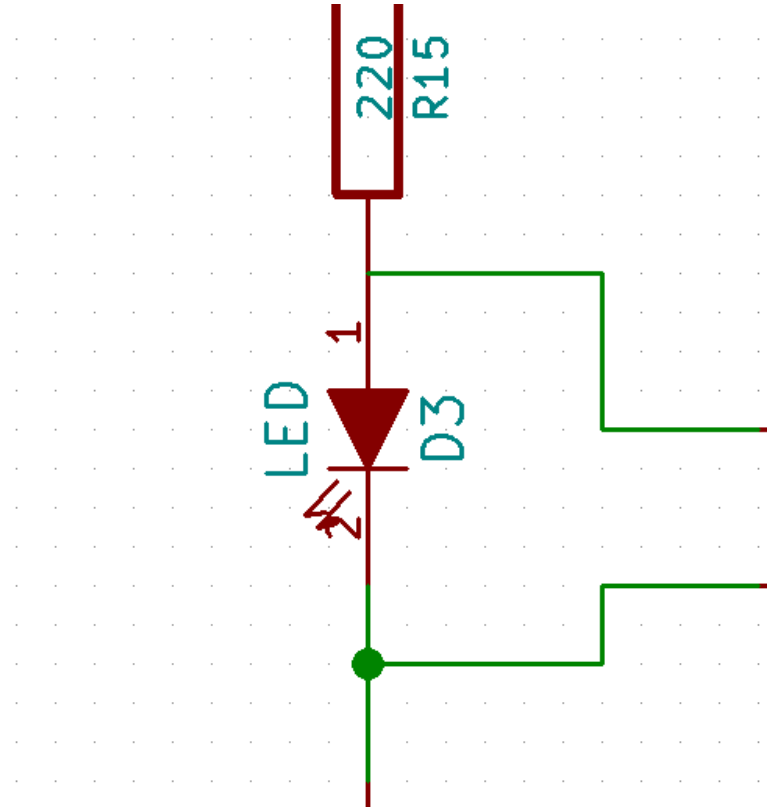


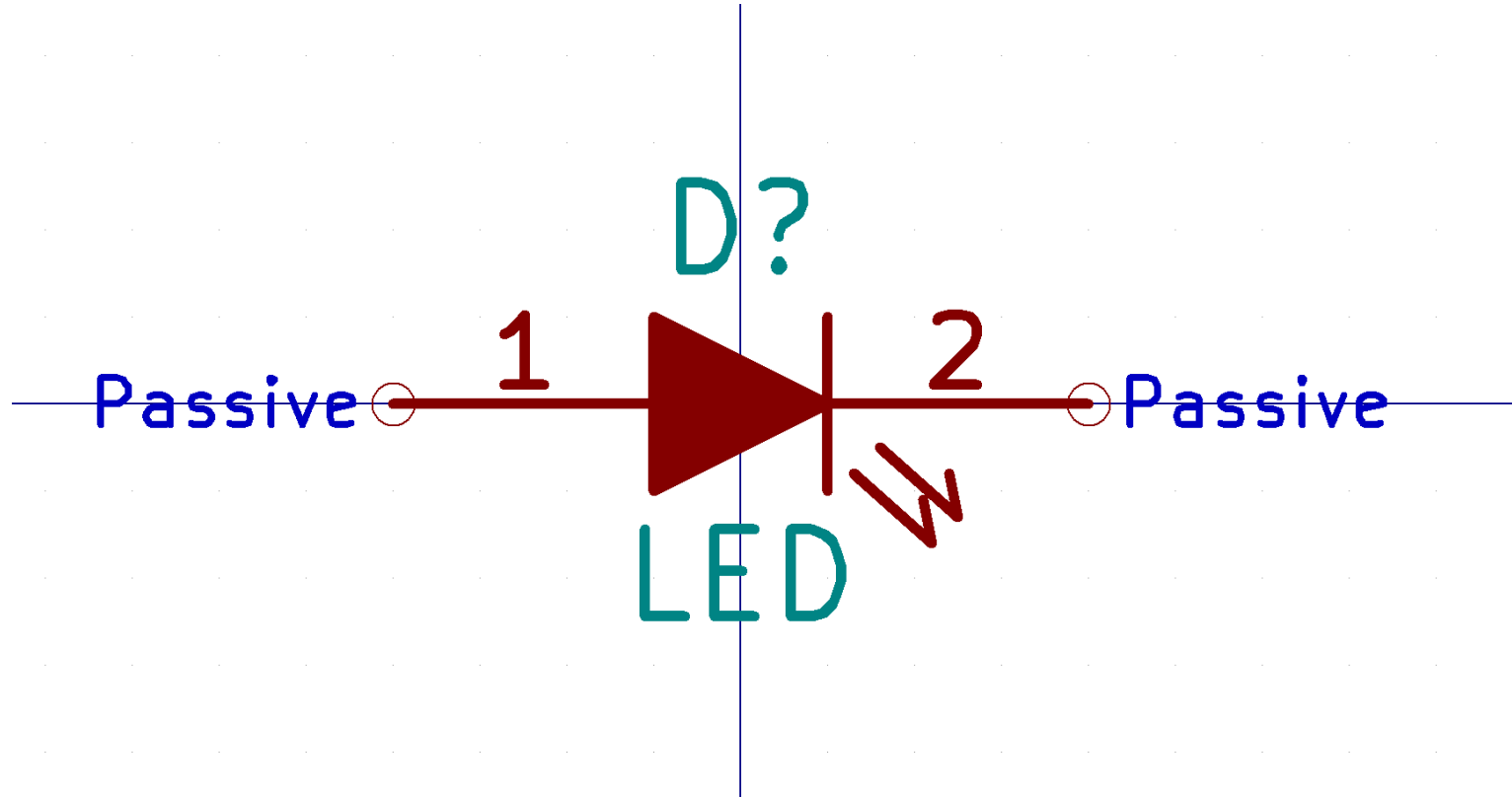


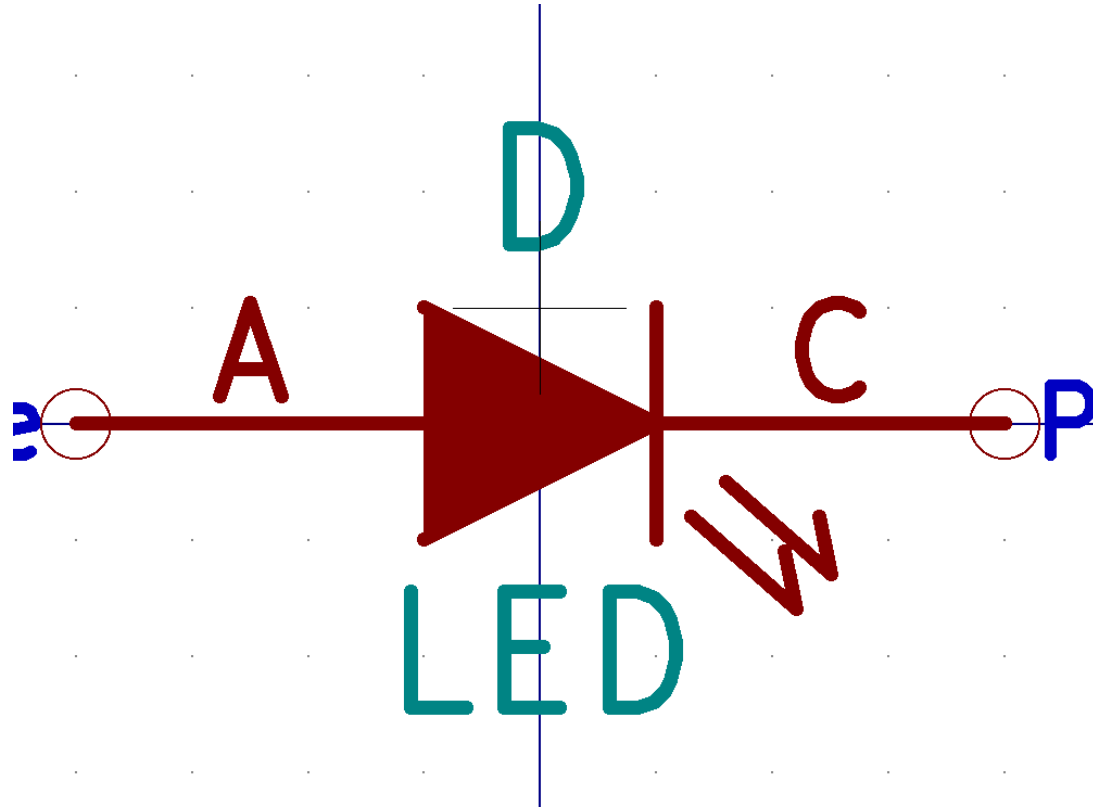














Pin Properties

Pin name:

Pin number:

Electrical type:

Graphic style:

X position: mm

Y position: mm

Orientation:

Pin length: mm

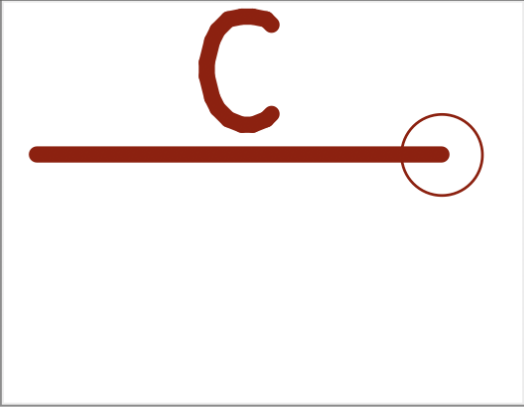
Name text size: mm

Number text size: mm

☐ Common to all units in symbol

☐ Common to all body styles (DeMorgan)

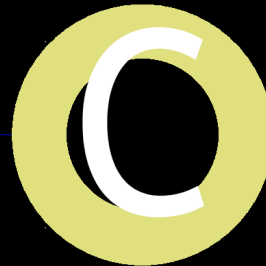
☒ Visible

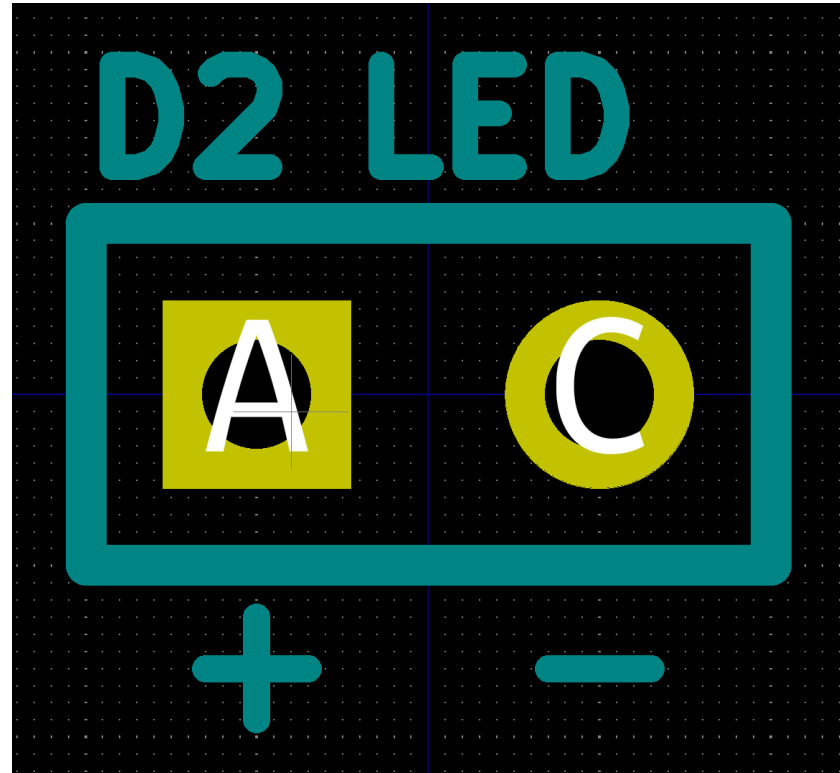


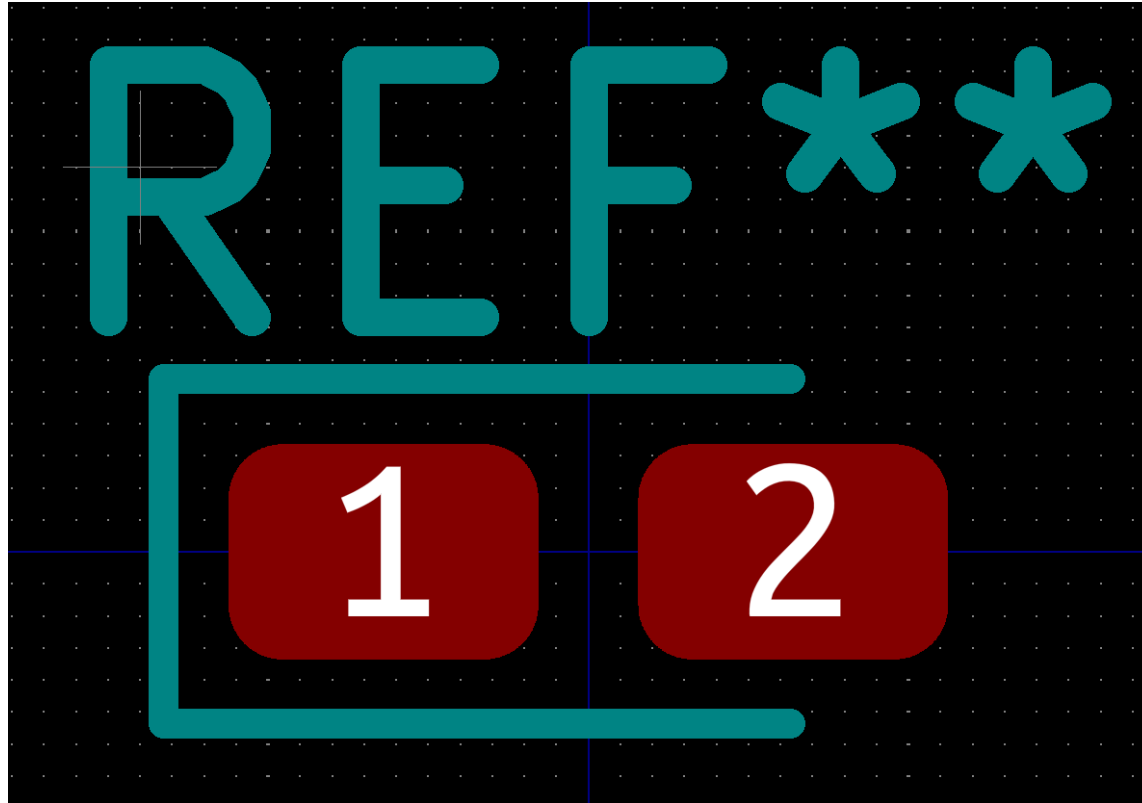
Cancel OK

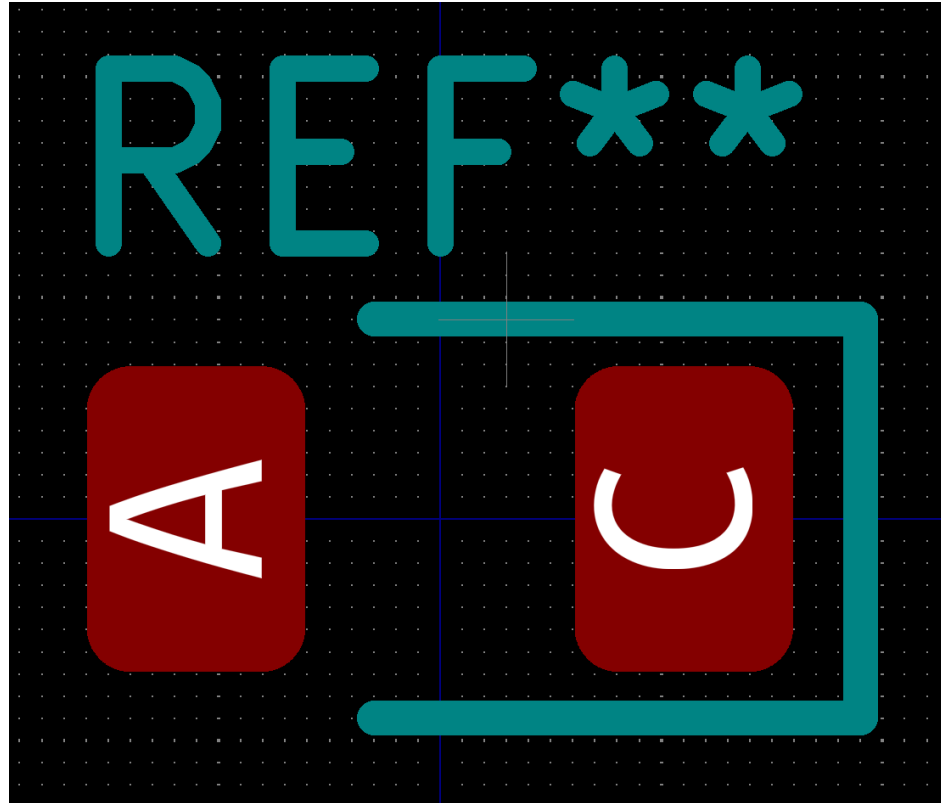


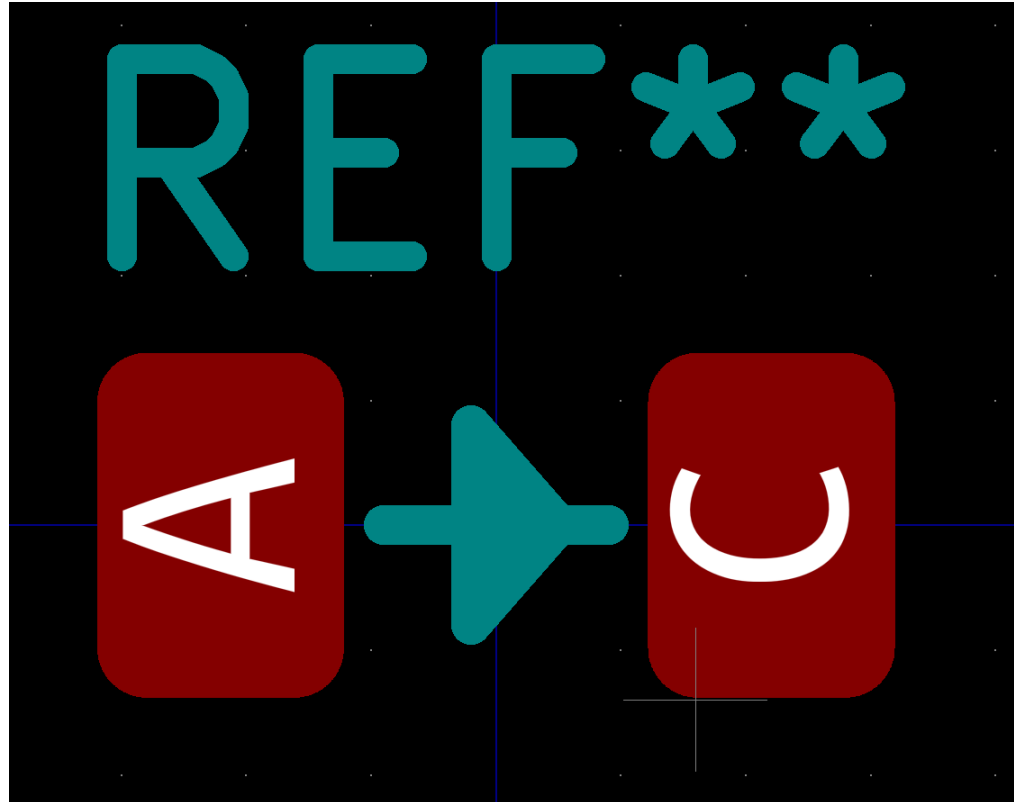
D2 LED

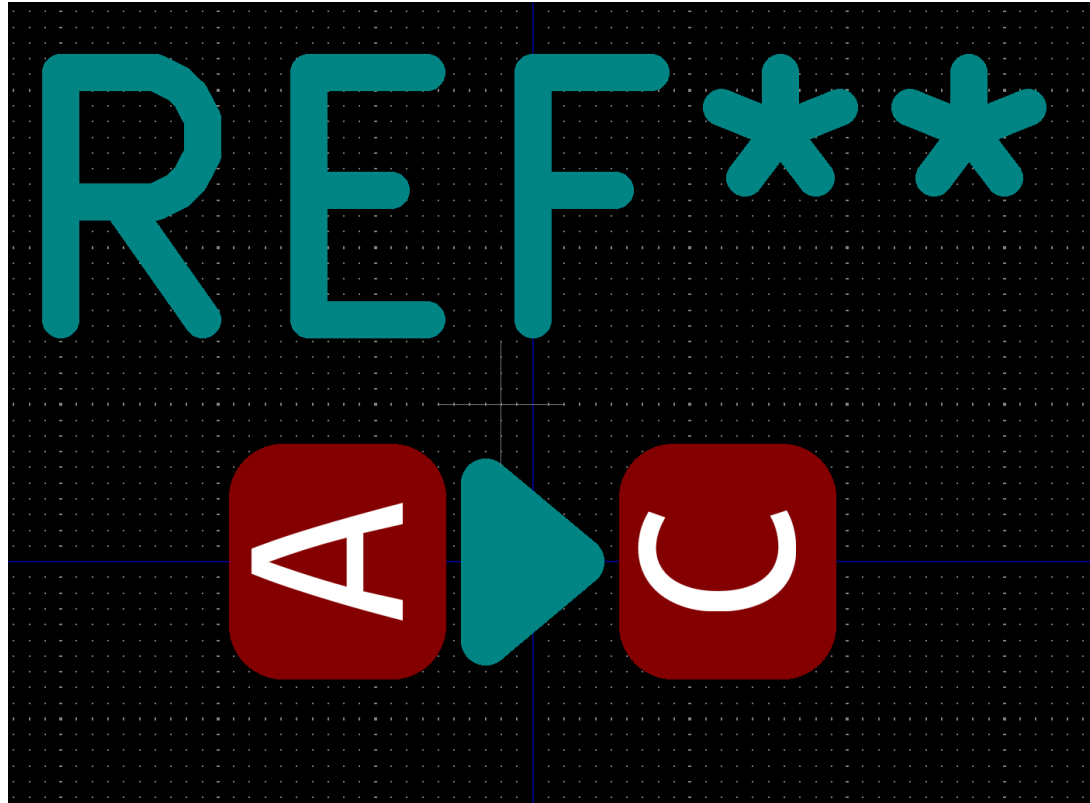










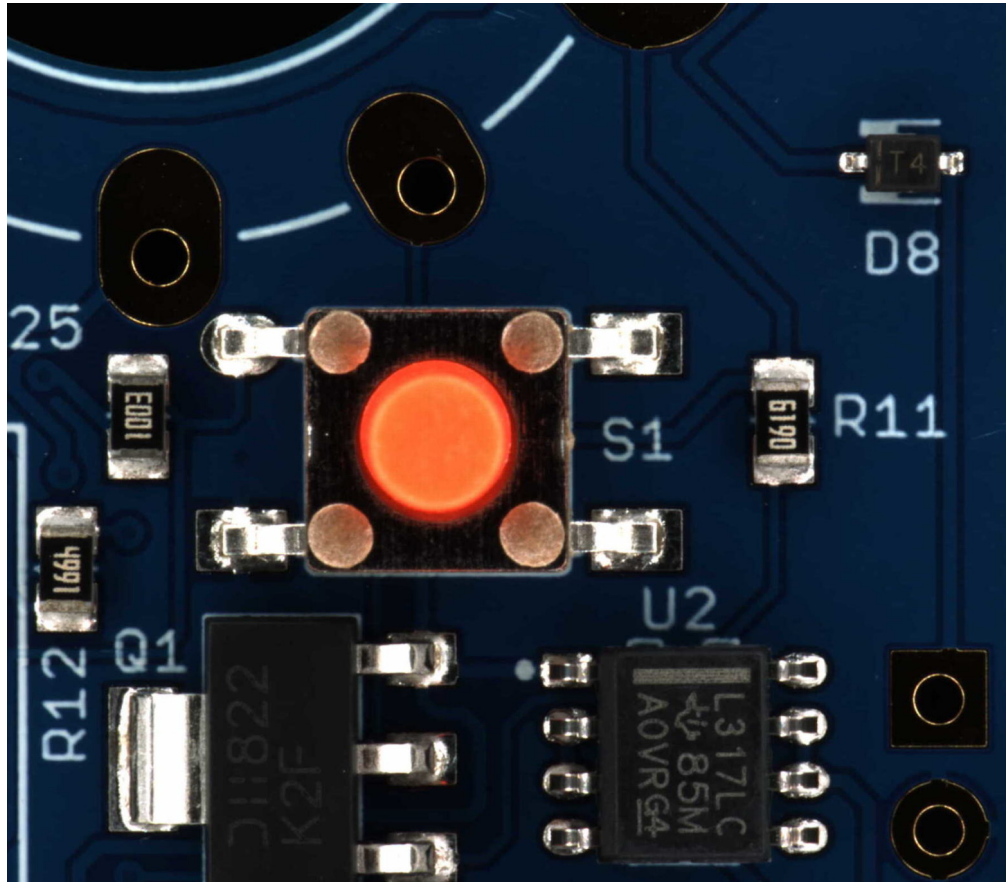




1. Silkscreen pin 1 indicators
 - a. Move them out from underneath the placed part
 - b. Square pad for Pin 1 on IC's
2. LED cathode indicators
 - a. Use a C in pin naming scheme
 - b. Use symbol in silkscreen
 - c. Use + and - symbols in silkscreen
 - d. Thru-hole: Square pad for Anode. Circle for Cathode



1. ~~Identifying polarity of components~~
2. Silkscreen legibility
3. Panelization
4. PCB properties
5. Specific manufacturer's part numbers





Not minimums



Not maximum



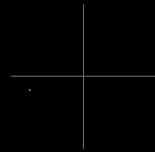
Cozy





R2  Cozy Text Height 1mm

The diagram shows the text 'R2' followed by a double-headed vertical arrow indicating the height of the text. The arrow is positioned between two horizontal lines that span the width of the 'R2' text. The text 'Cozy Text Height 1mm' is to the right of the arrow.

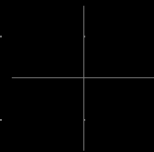




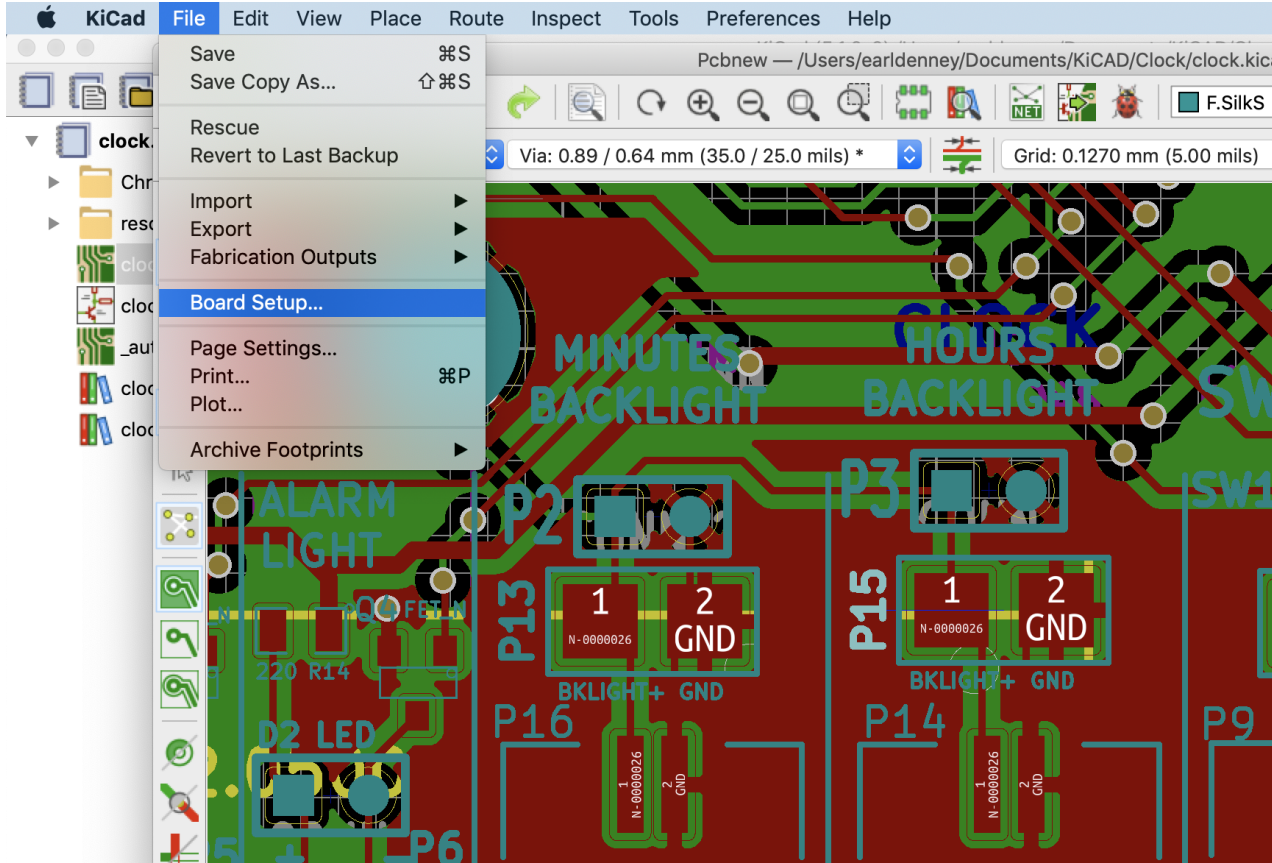
R2



Cozy Text Width 1mm









Board Setup

▼ Layers

- Text & Graphics
- ▼ Design Rules
 - Net Classes
 - Tracks & Vias
 - Solder Mask/Paste

Default properties for new graphic items:

	Line Thickness	Text Width	Text Height	Text Thickness	Italic	Keep Upright
Silk Layers	0.18 mm	1 mm	1 mm	0.18 mm	<input type="checkbox"/>	<input checked="" type="checkbox"/>
Copper Layers	0.2 mm	1.5 mm	1.5 mm	0.3 mm	<input type="checkbox"/>	<input checked="" type="checkbox"/>
Edge Cuts	0.1 mm					
Courtyards	0.05 mm					
Other Layers	0.15 mm	1 mm	1 mm	0.15 mm	<input type="checkbox"/>	<input checked="" type="checkbox"/>

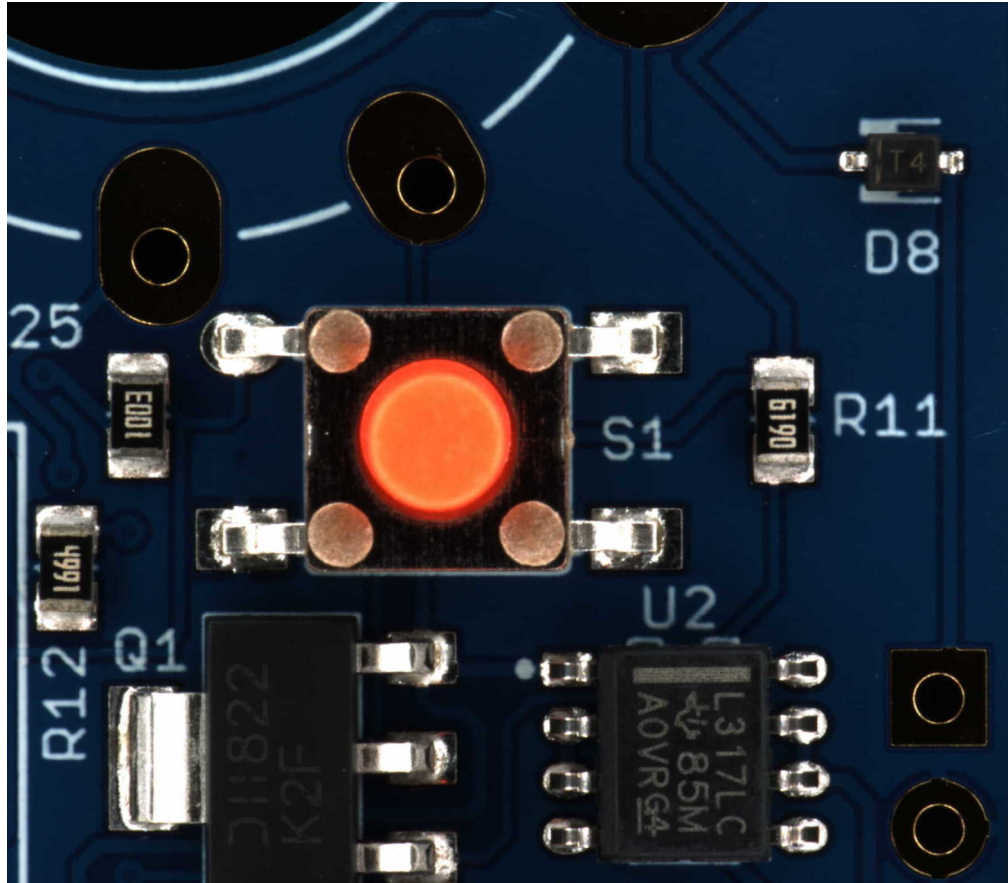
Import Settings...

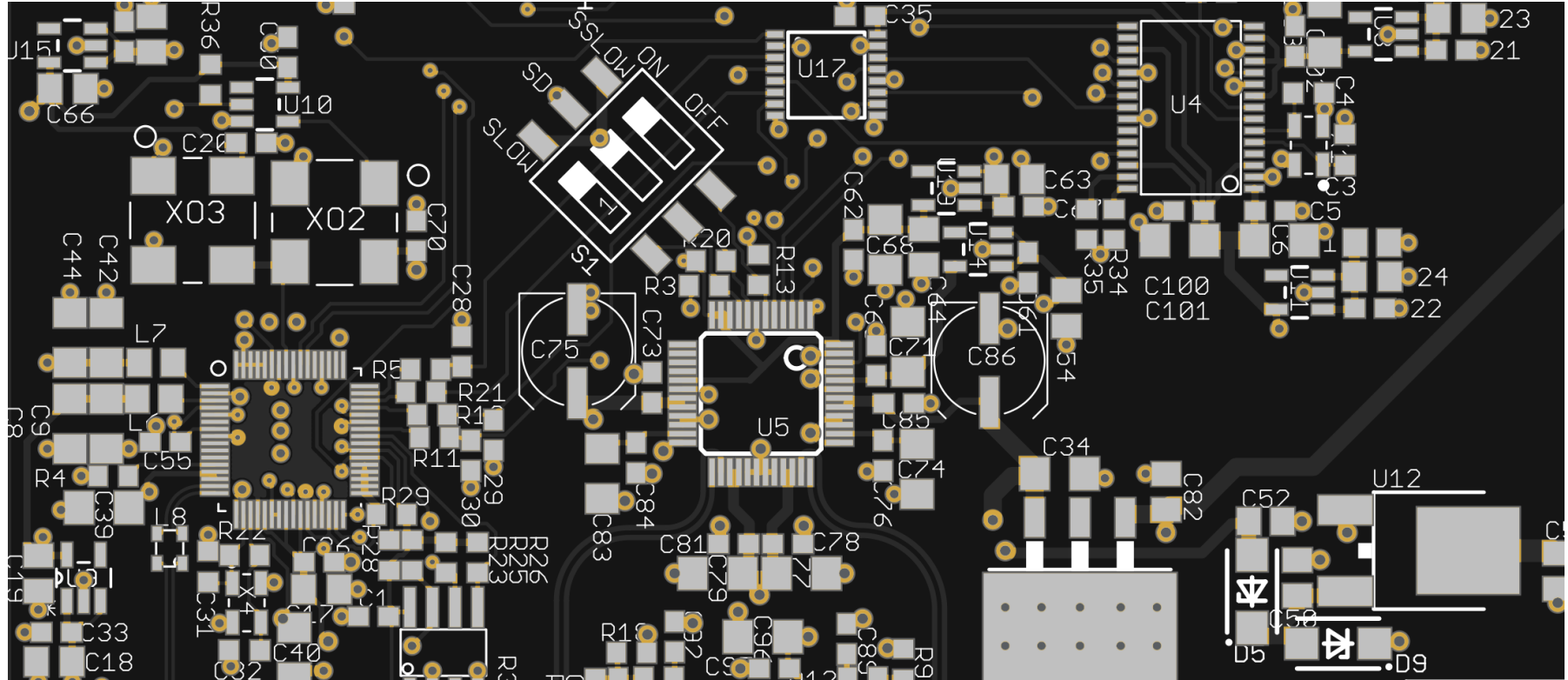
Cancel OK

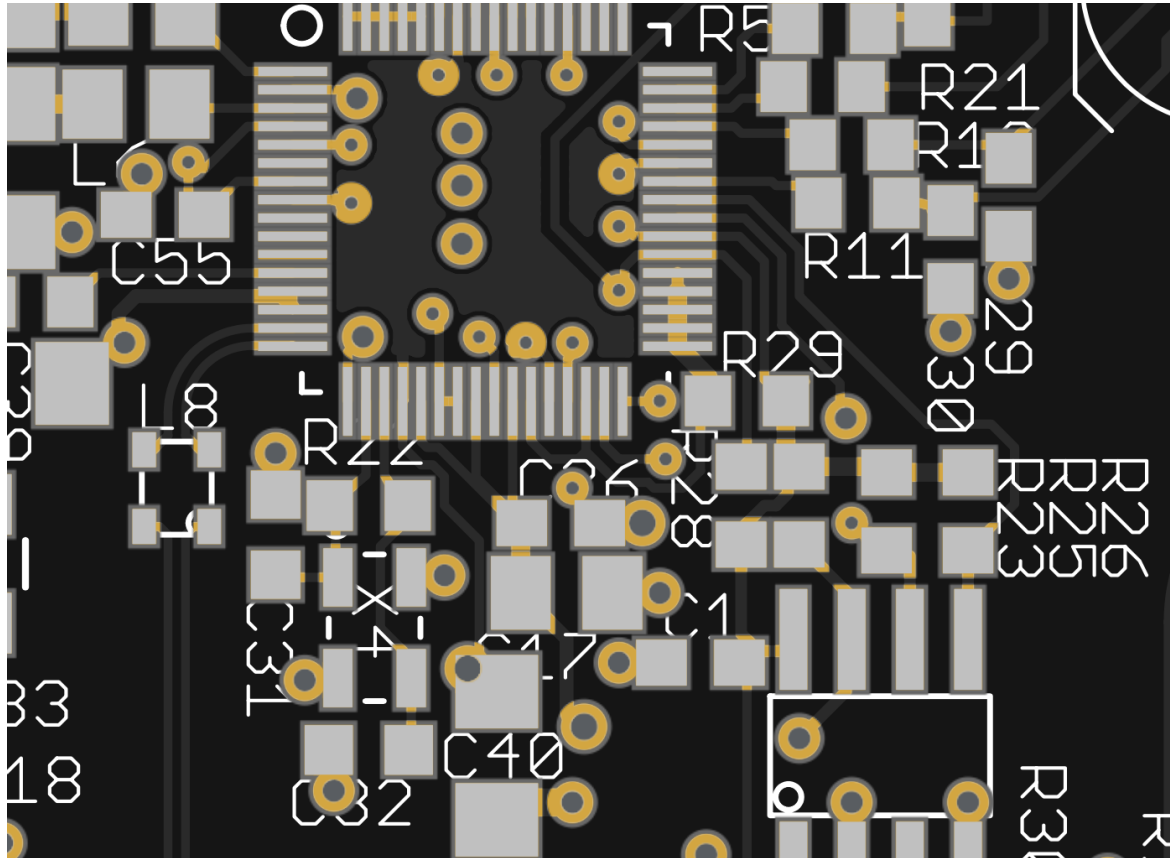


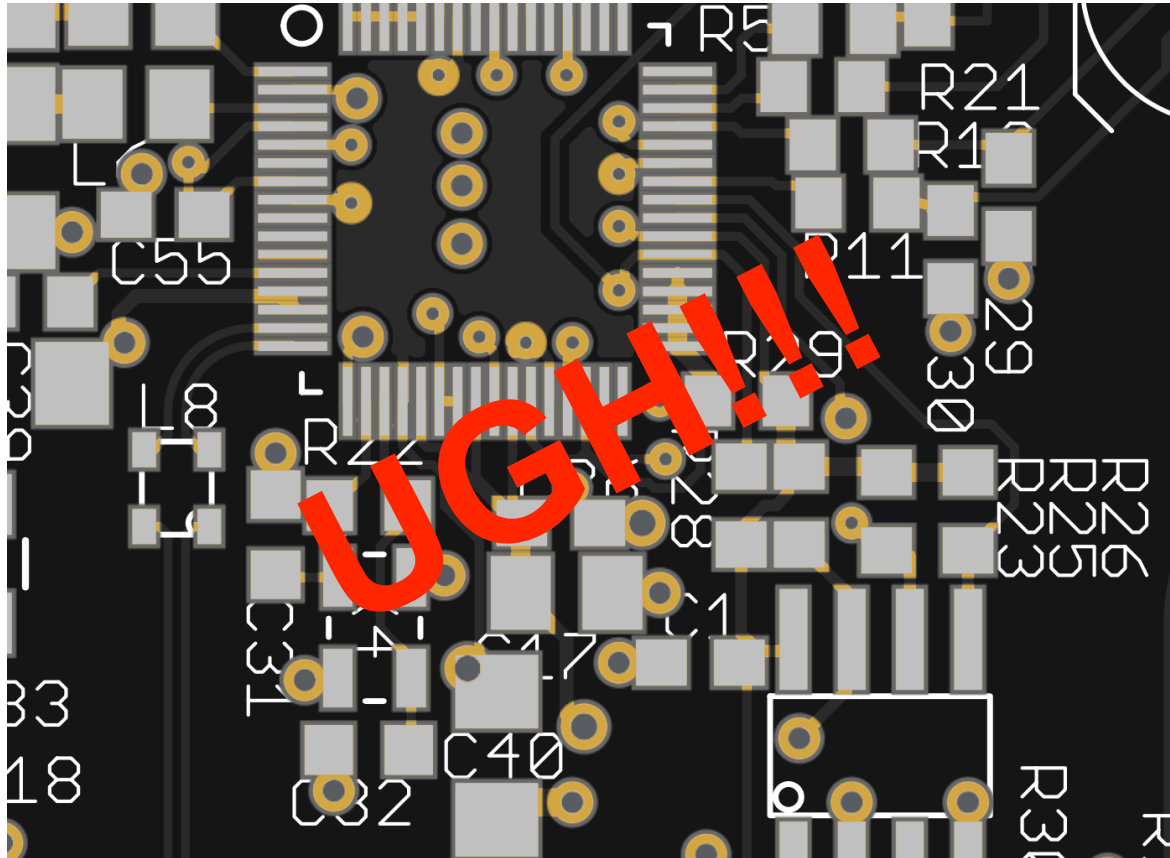
Default properties for new graphic items:

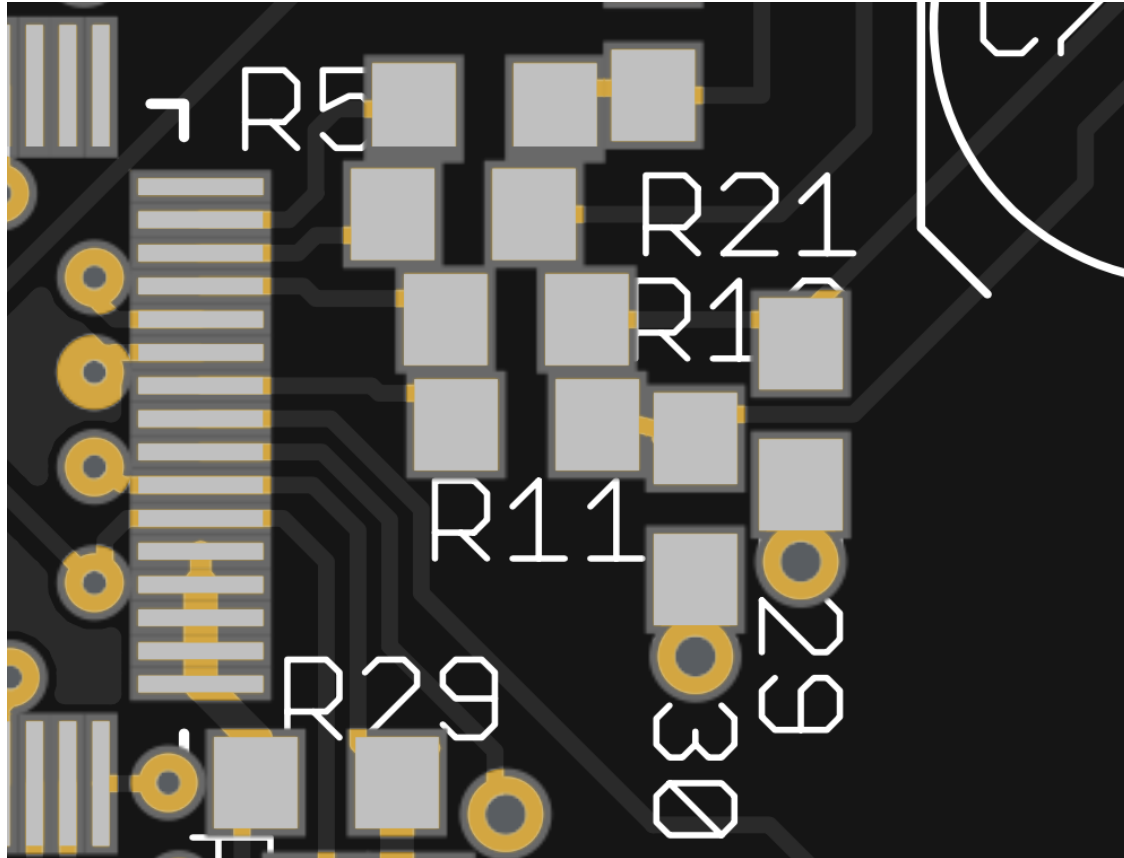
	Line Thickness	Text Width	Text Height	Text Thickness	Italic	Keep Upright
Silk Layers	0.18 mm	1 mm	1 mm	0.18 mm	<input type="checkbox"/>	<input checked="" type="checkbox"/>
Copper Layers	0.2 mm	1.5 mm	1.5 mm	0.3 mm	<input type="checkbox"/>	<input checked="" type="checkbox"/>

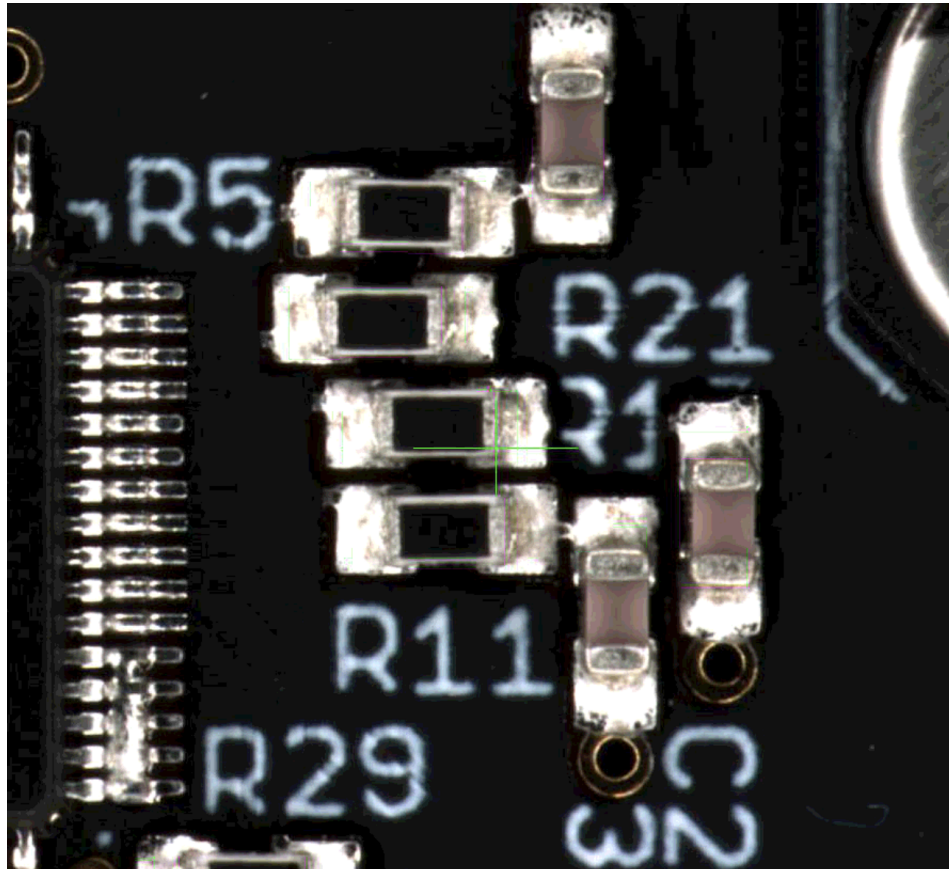














Plot

Plot format: Gerber Output directory: Gerber/

Included Layers

- ☒ F.Cu
- ☒ B.Cu
- ☒ F.Adhes
- ☒ B.Adhes
- ☒ F.Paste
- ☒ B.Paste
- ☒ F.SilkS
- ☒ B.SilkS
- ☒ F.Mask
- ☒ B.Mask
- ☒ Dwgs.User
- ☒ Cmts.User
- ☒ Eco1.User
- ☒ Eco2.User
- ☒ Edge.Cuts

General Options

- ☐ Plot border and title block
- ☒ Plot footprint values
- ☒ Plot footprint references
- ☐ Force plotting of invisible values / refs
- ☒ Exclude PCB edge layer from other layers
- ☒ Exclude pads from silkscreen
- ☐ Do not tent vias
- ☐ Use auxiliary axis as origin
- Drill marks: None
- Scaling: 1:1
- Plot mode: Filled
- Default line width: 0.15 mm
- ☐ Mirrored plot
- ☐ Negative plot
- ☒ Check zone fills before plotting

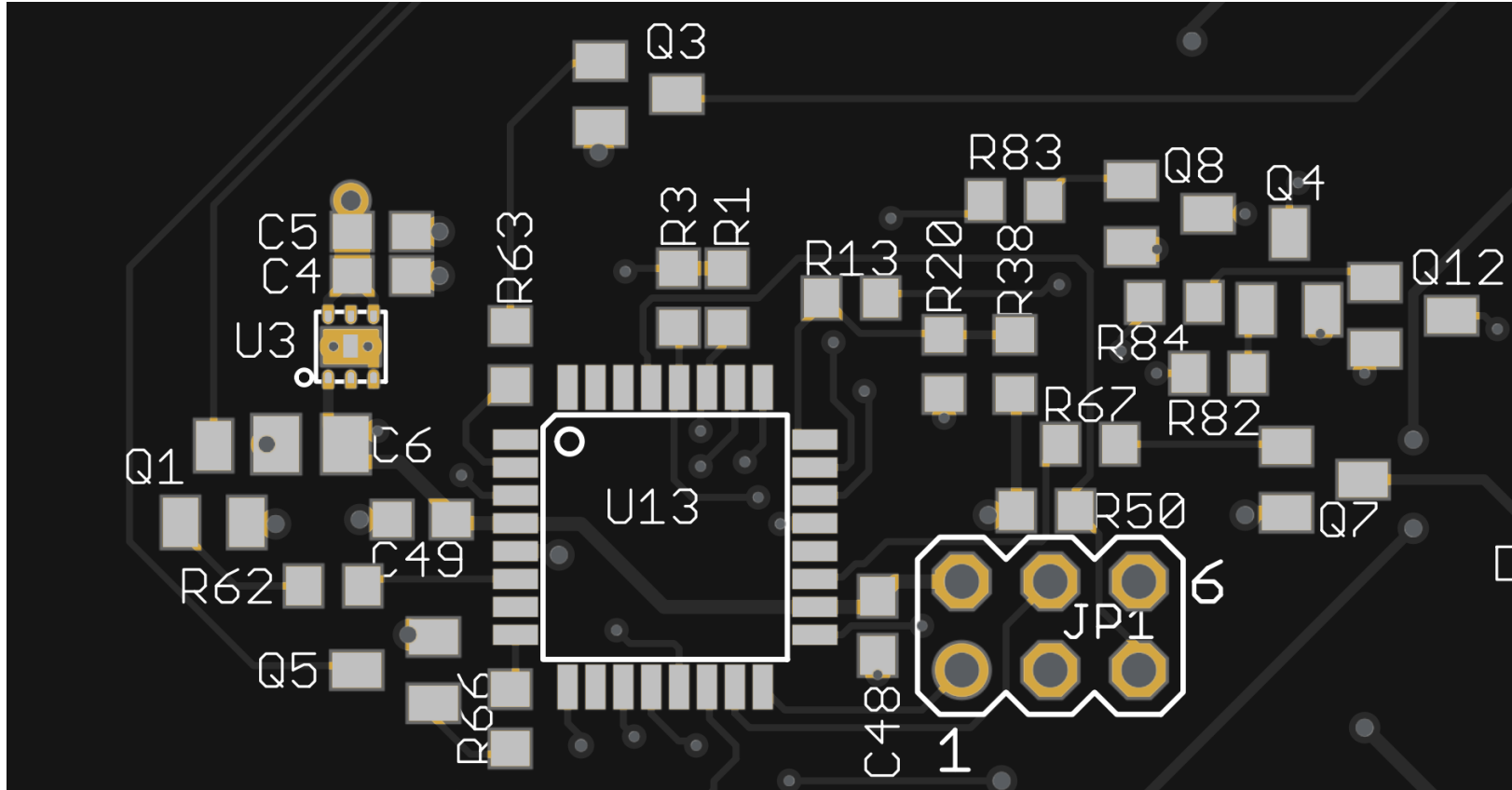
Gerber Options

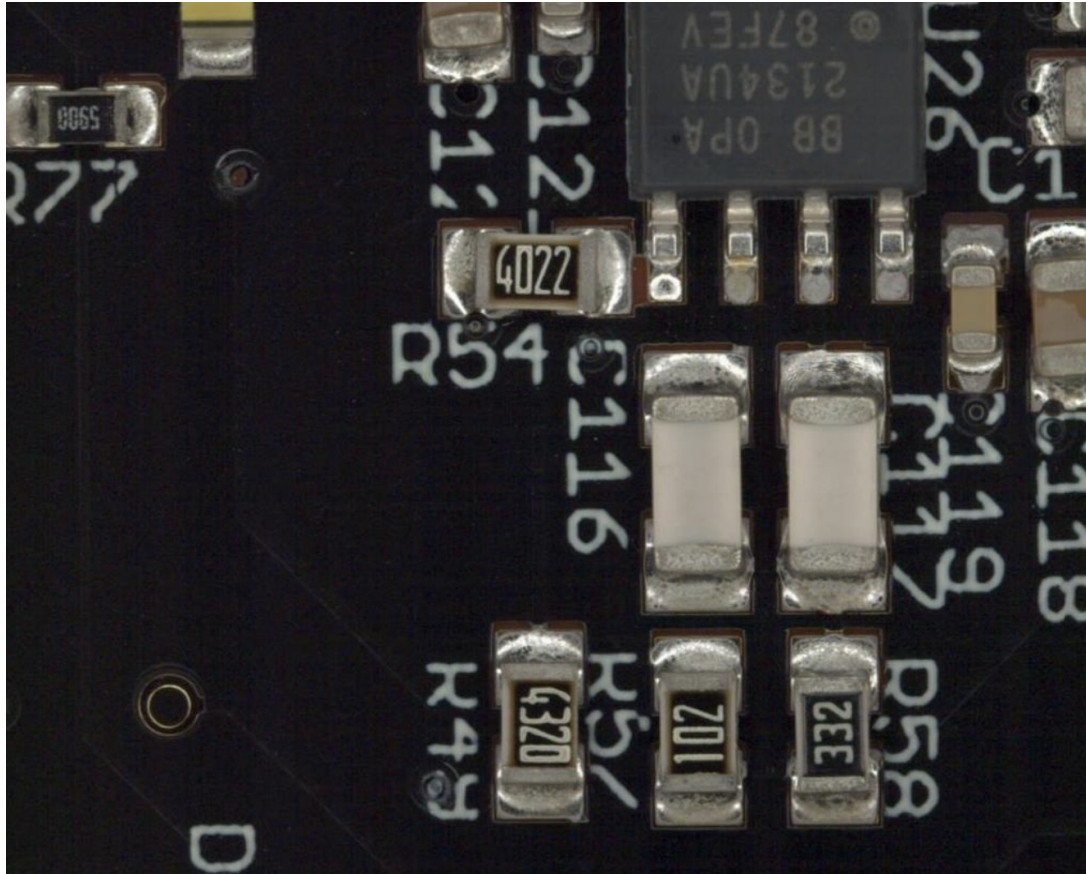
- ☒ Use Protel filename extensions
- ☐ Generate Gerber job file
- ☒ Subtract soldermask from silkscreen
- Coordinate format: 4.6, unit mm
- ☐ Use extended X2 format
- ☐ Include netlist attributes

Output Messages

Show: ☐ All ☒ Errors ☒ Warnings ☒ Actions ☒ Infos

Run DRC... Close Generate Drill Files... Plot





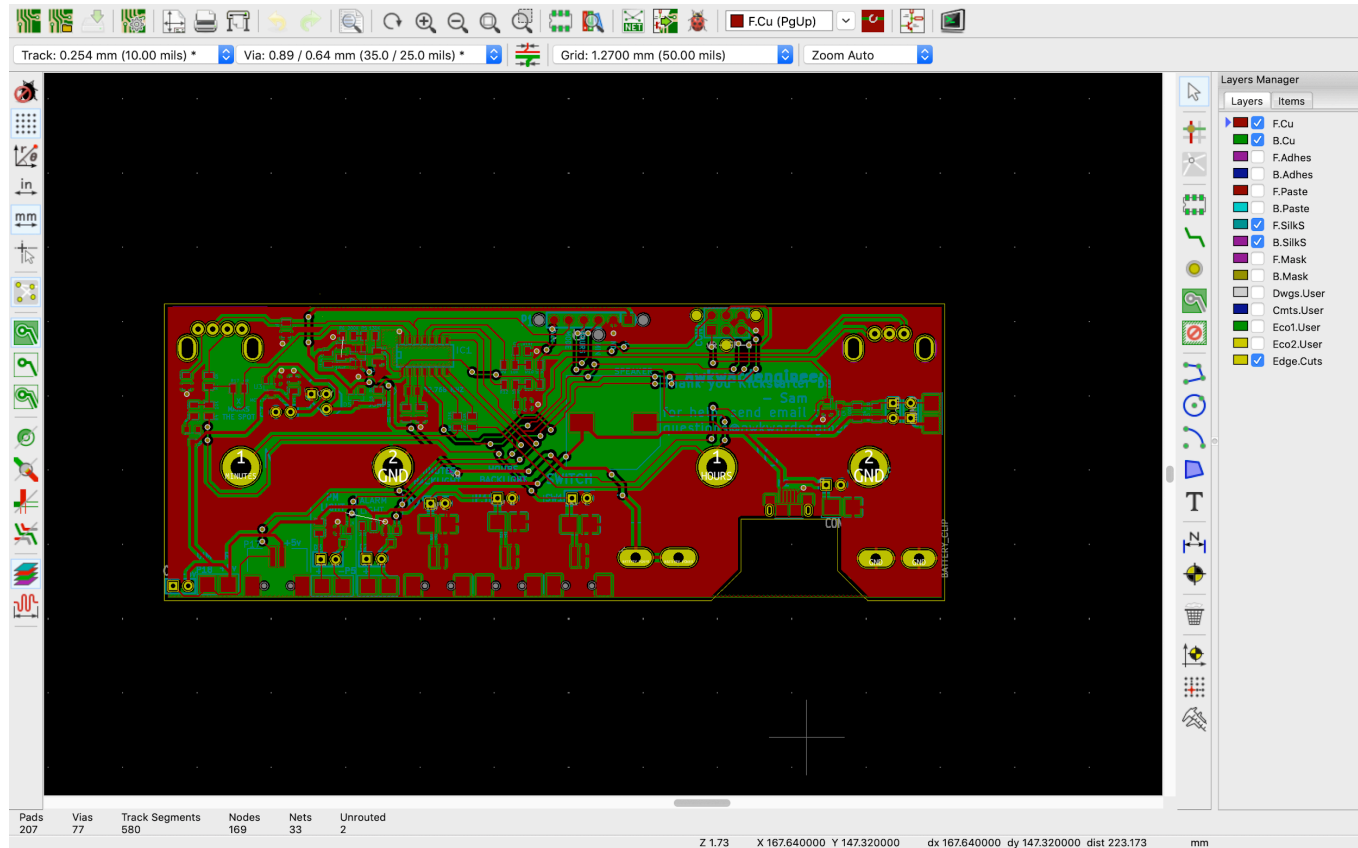


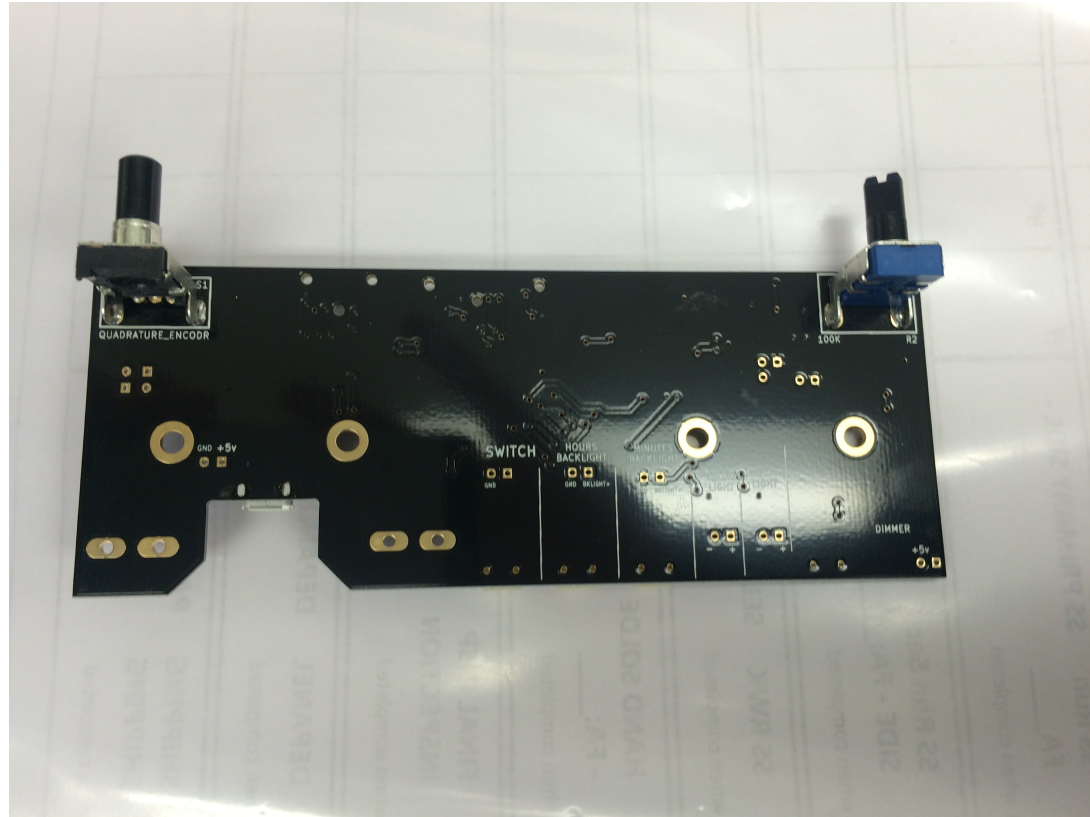


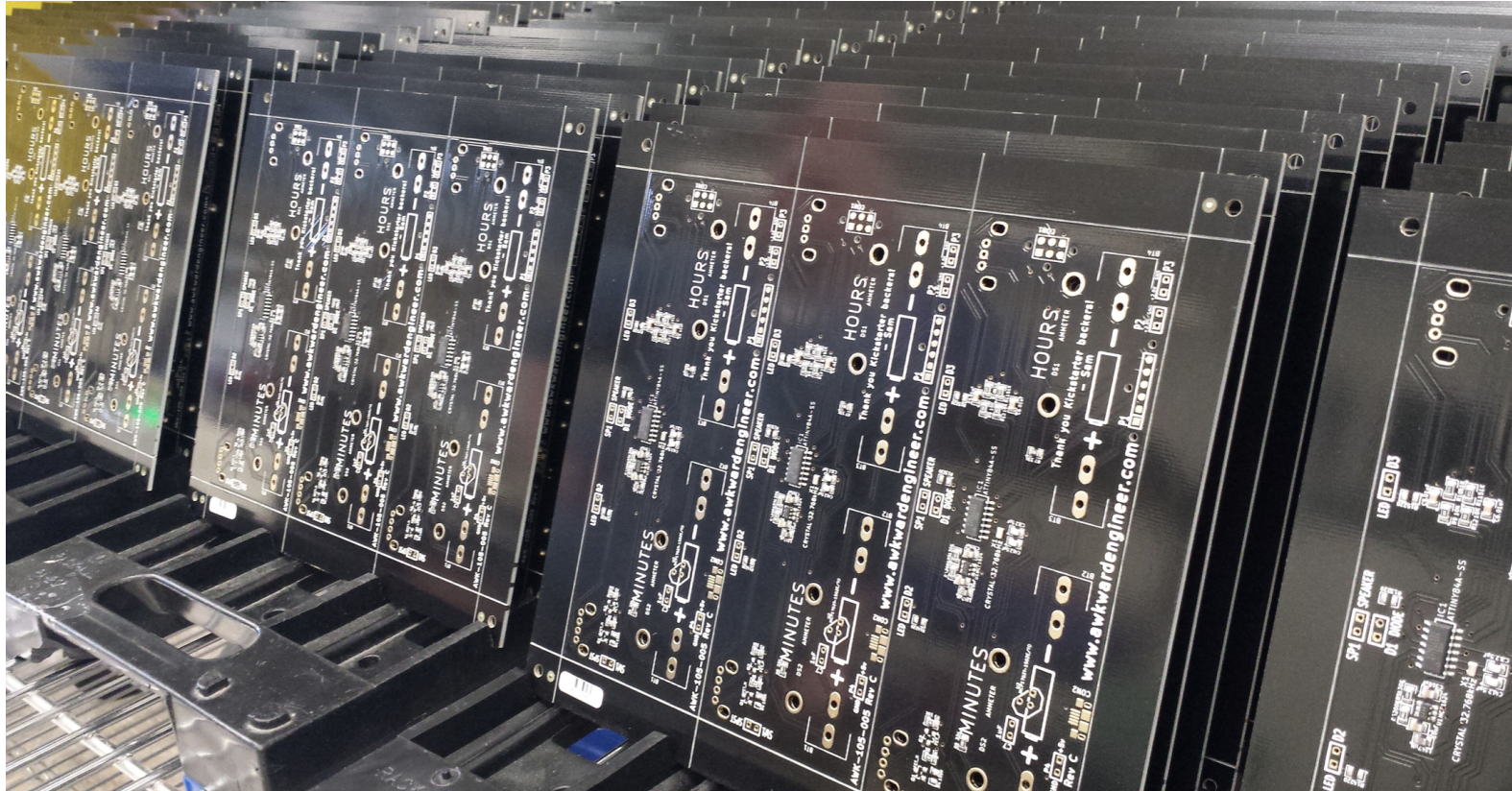
1. Cozy Text: 1x1mm with 0.18mm line width
2. Move away from pads, holes, and vias
3. Tent vias to print on top of vias - but only if you must

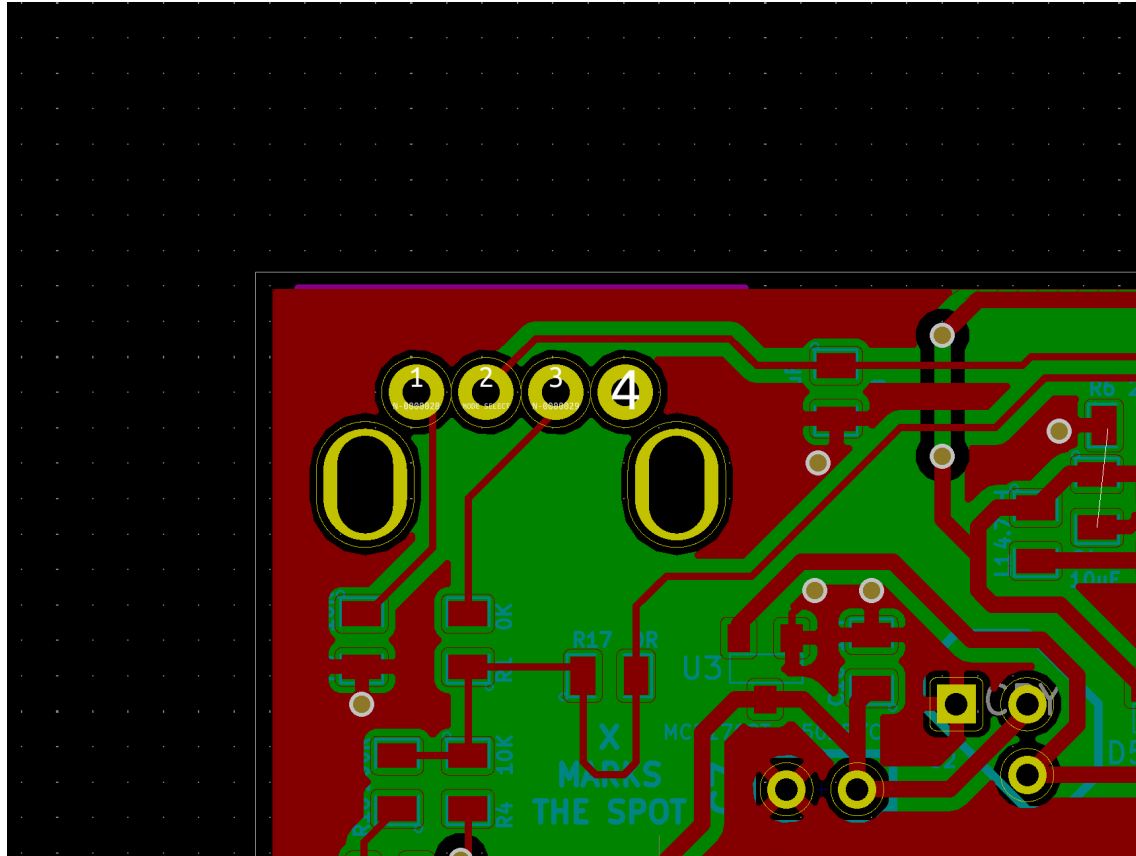


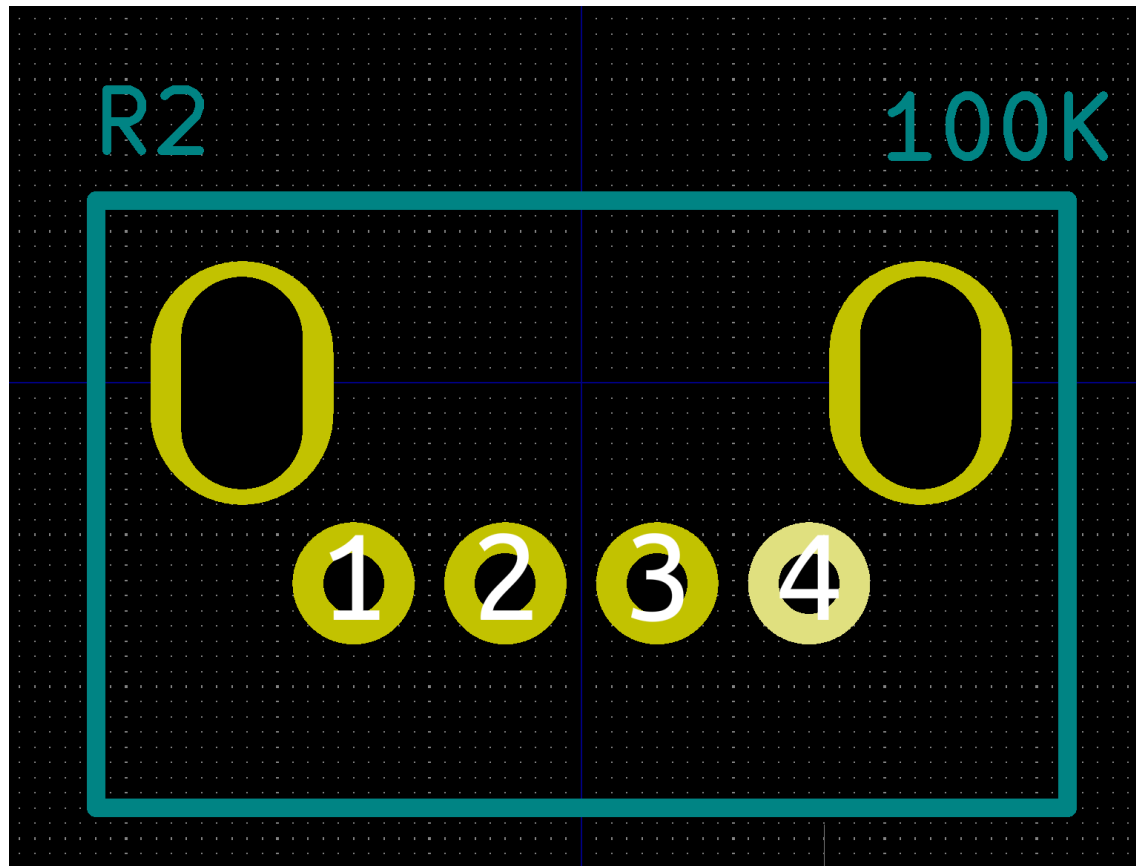
1. ~~Identifying polarity of components~~
2. ~~Silkscreen legibility~~
3. **Panelization**
4. PCB properties
5. Specific manufacturer's part numbers





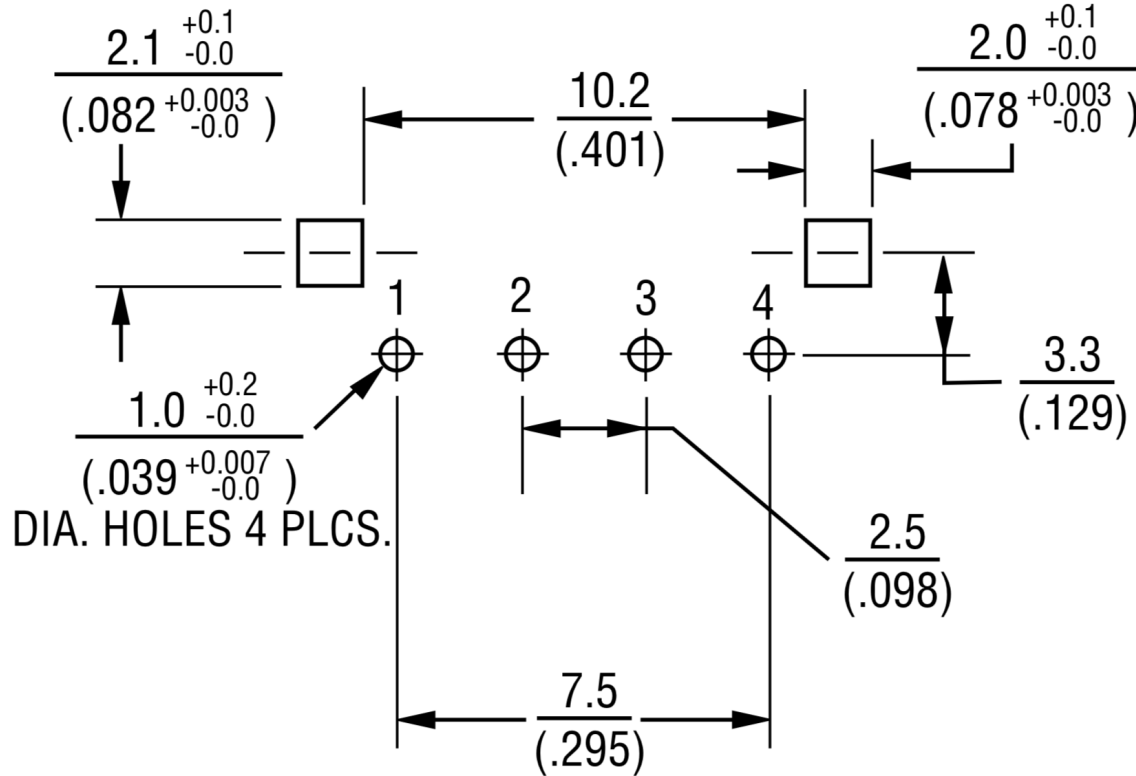






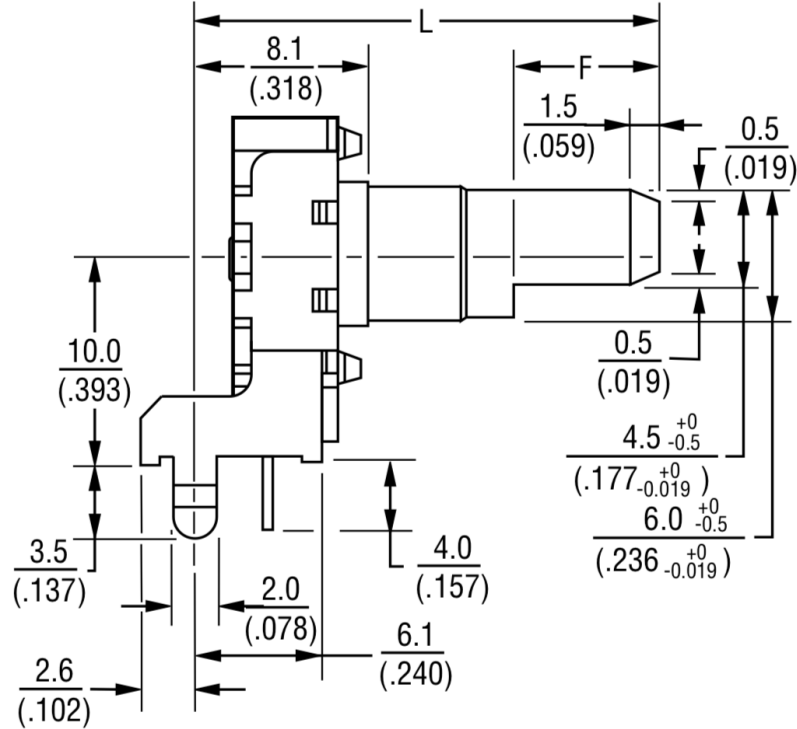


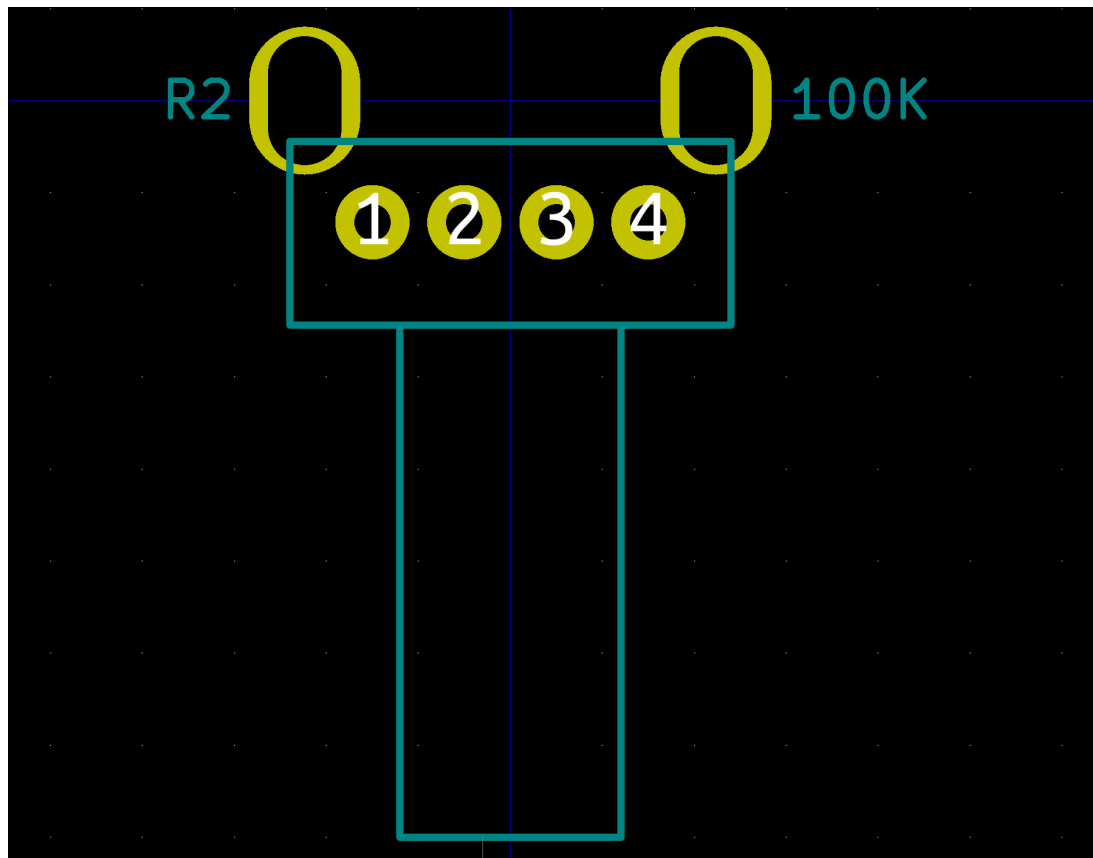
RECOMMENDED PCB LAYOUT

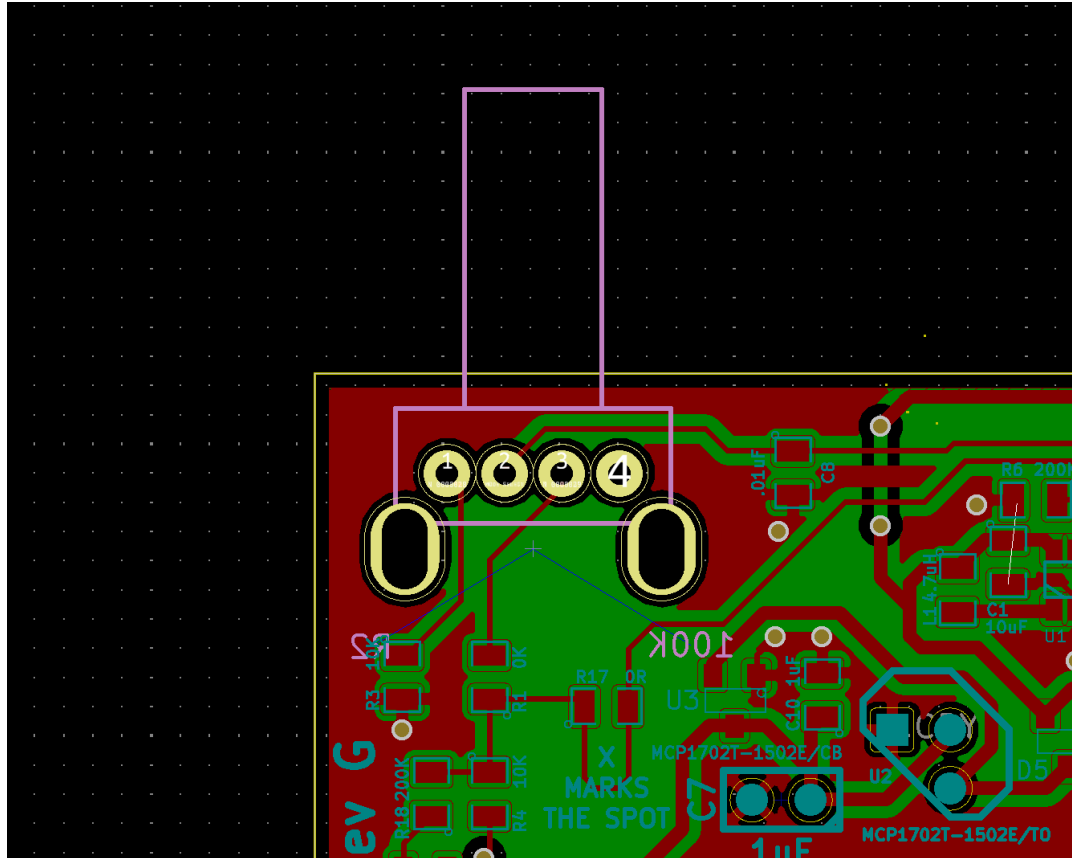


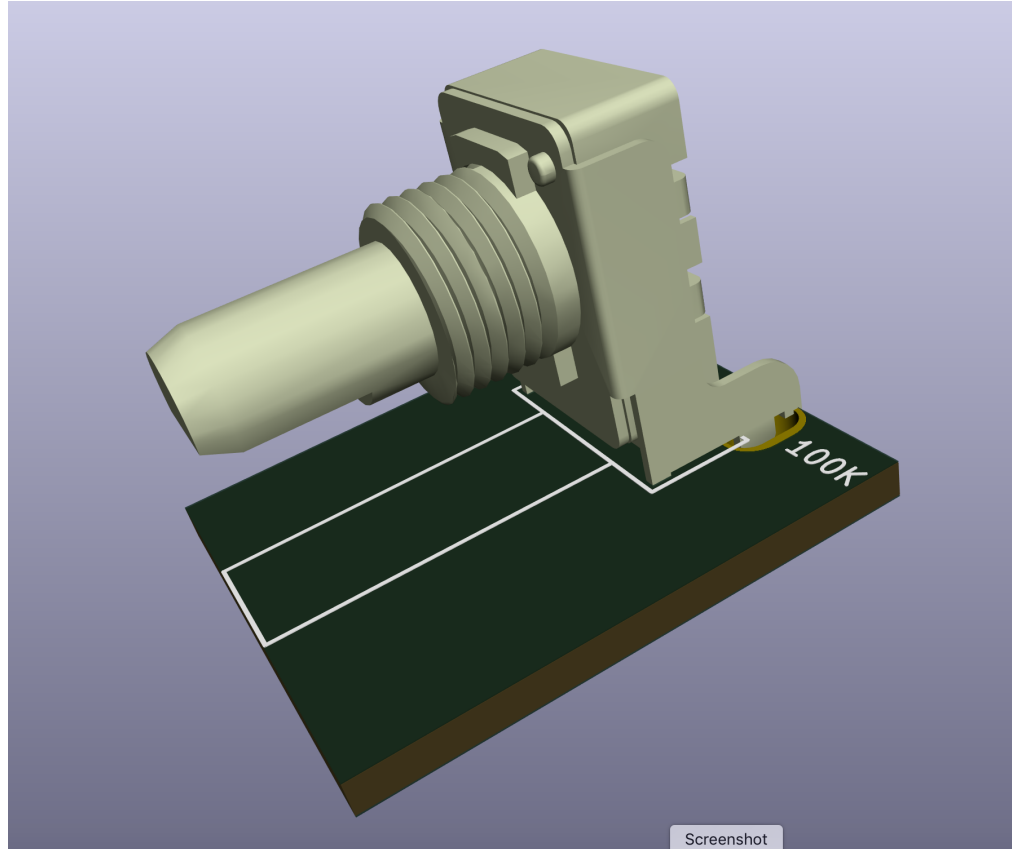


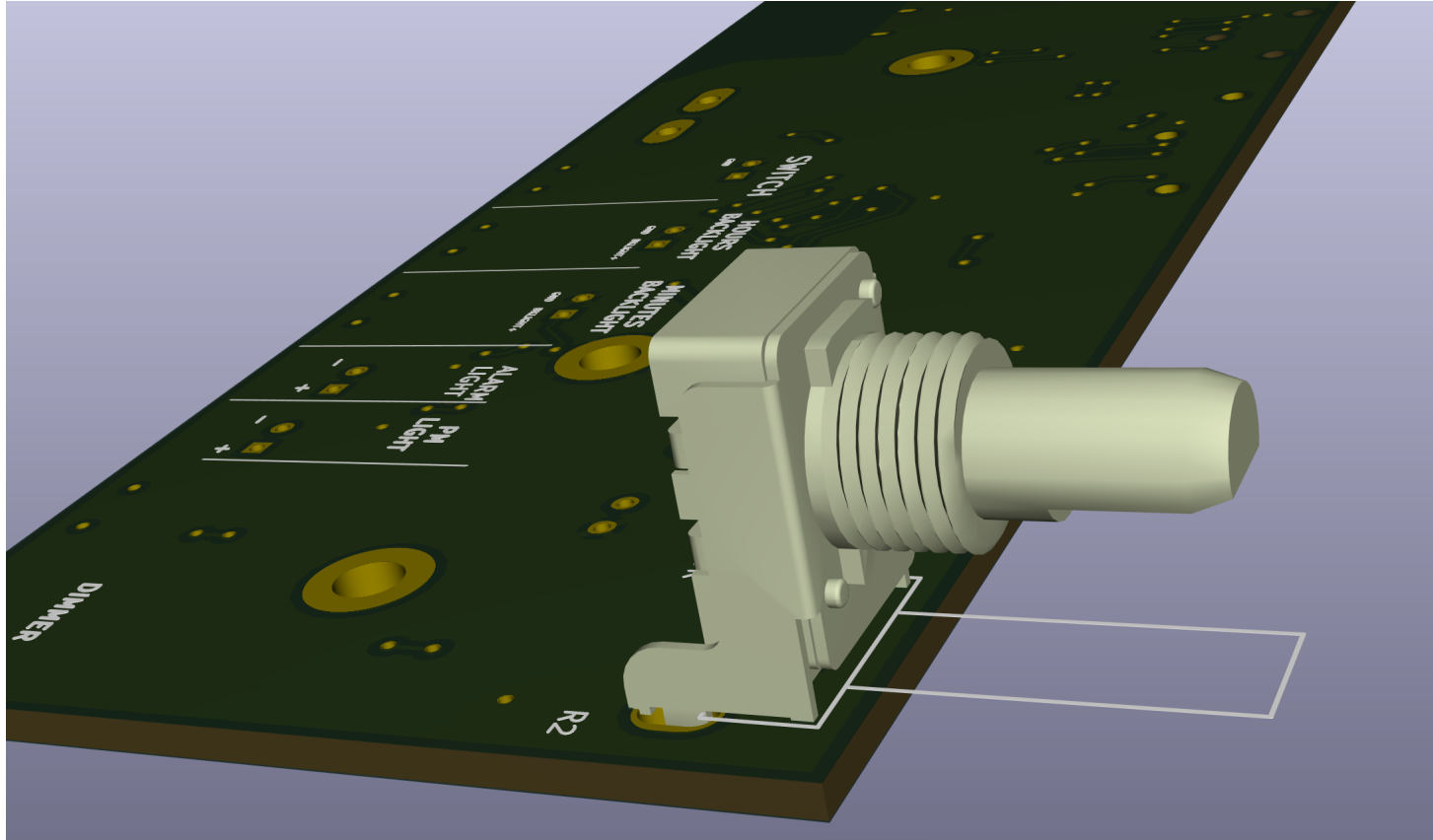
PTV111-2













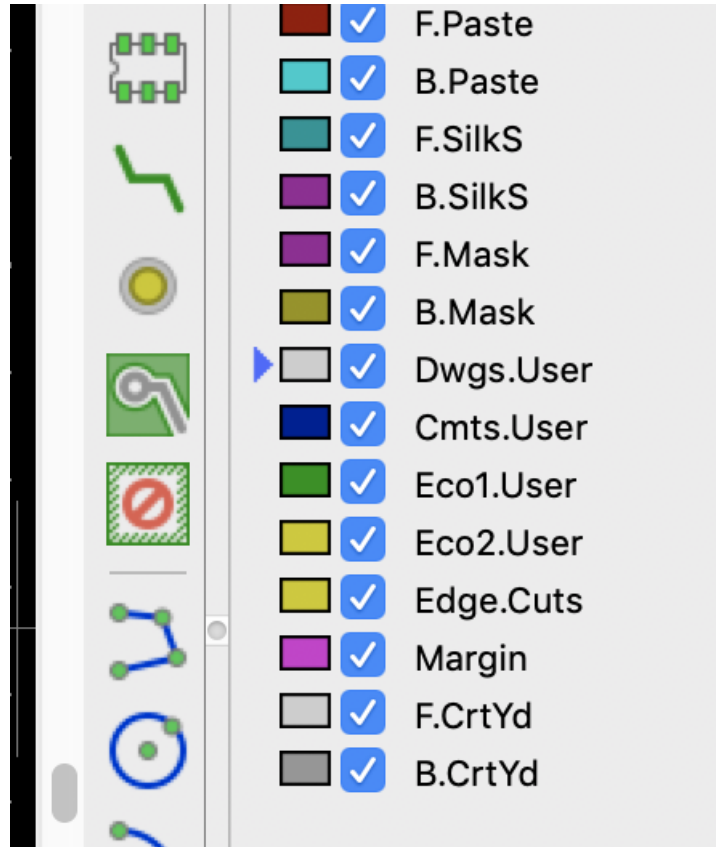
1. Add silkscreen representing body of component.
 - 1.1. Doesn't have to be elaborate, just accurate.
 - 1.2. Don't worry, silkscreen that falls outside of the Outline will be trimmed.
2. Add 3D Model if you can



1. ~~Identifying polarity of components~~
2. ~~Silkscreen legibility~~
3. ~~Panelization~~
4. **PCB properties**
5. Specific manufacturer's part numbers



1. Number of layers
2. Overall PCB thickness
3. TG rating
4. Inner copper weight
5. Outer copper weight
6. Silkscreen color
7. Soldermask color
8. Via-in-pad
9. Impedance control
10. E-Test Required





Text Properties

Text:

1. Number of Layers: 4
2. PCB Thickness: 1.6mm
3. TG Rating: TG170
4. Inner Copper Weight: 0.5oz
5. Outer Copper Weight: 1oz
6. Silkscreen Color: White
7. Soldermask Color: Green
8. Via-in-Pad: Fill and Plate
9. Impedance Control: Top Layer 23.00mil single-ended Coplanar Waveguide 50 ohm (+-8%)

Layer: ☐ Dwgs.User

Width: mm

Height: mm

Thickness: mm

Position X: mm

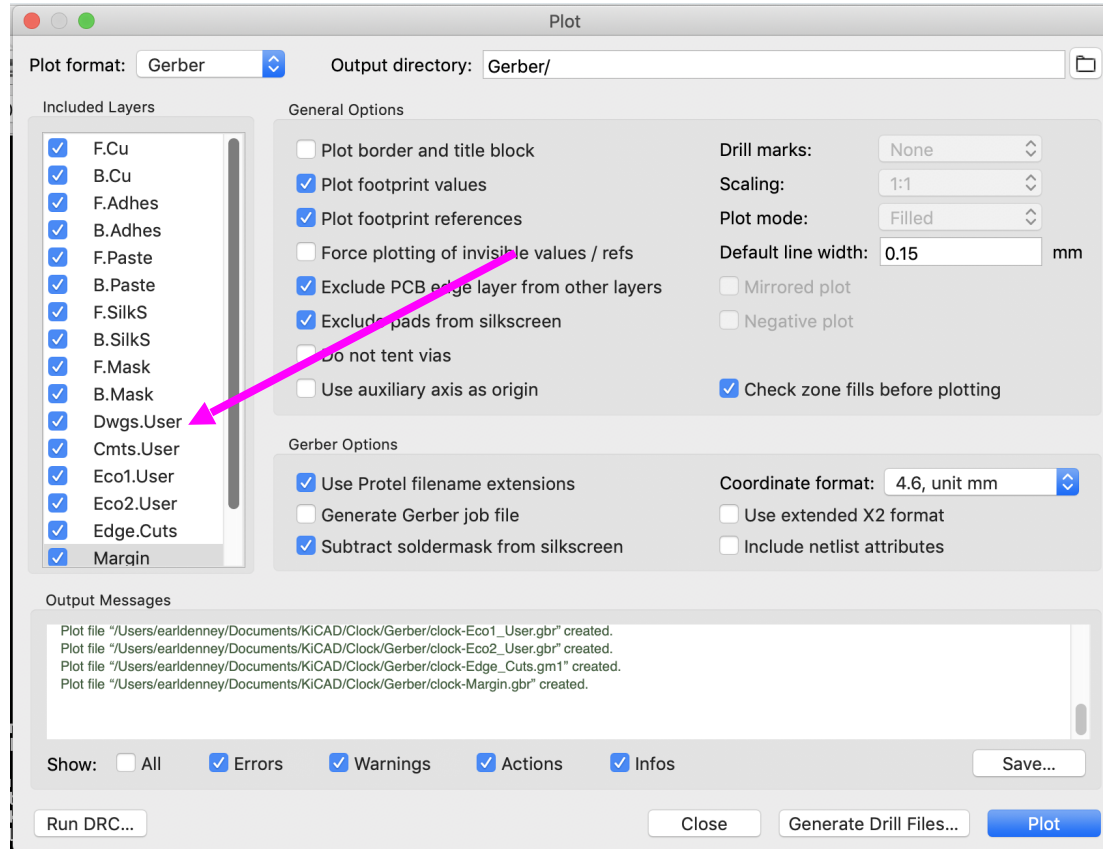
Position Y: mm

☐ Italic

Justification:

Orientation:

☐ Mirrored





Plot

Plot format: PDF

Output directory: Gerber/

Included Layers

☐ B.Silks
 ☐ F.Mask
 ☐ B.Mask
 ☒ Dwgs.User
 ☐ Cmts.User
 ☐ Eco1.User
 ☐ Eco2.User
 ☐ Edge.Cuts
 ☐ Margin
 ☐ F.CrtYd
 ☐ B.CrtYd

General Options

☒ Plot border and title block
 ☒ Plot footprint values
 ☒ Plot footprint references
 ☐ Force plotting of invisible values / refs
 ☒ Exclude PCB edge layer from other layers
 ☒ Exclude pads from silkscreen
 ☐ Do not tent vias
 ☐ Use auxiliary axis as origin

Drill marks: None

 Scaling: 1:1

 Plot mode: Filled

 Default line width: 0.15 mm

☐ Mirrored plot

☐ Negative plot

☒ Check zone fills before plotting

Output Messages

Plot file "/Users/earlidenney/Documents/KiCAD/Clock/Gerber/clock-B_Pastes.gba" created.
 Plot file "/Users/earlidenney/Documents/KiCAD/Clock/Gerber/clock-F_Paste.gtp" created.
 Plot file "/Users/earlidenney/Documents/KiCAD/Clock/Gerber/clock-B_Paste.gbp" created.
 Plot file "/Users/earlidenney/Documents/KiCAD/Clock/Gerber/clock-F_Silks.gto" created.
 Plot file "/Users/earlidenney/Documents/KiCAD/Clock/Gerber/clock-B_Silks.gbo" created.
 Plot file "/Users/earlidenney/Documents/KiCAD/Clock/Gerber/clock-F_Mask.gts" created.
 Plot file "/Users/earlidenney/Documents/KiCAD/Clock/Gerber/clock-B_Mask.gbs" created.

Show:
 ☐ All
 ☒ Errors
 ☒ Warnings
 ☒ Actions
 ☒ Infos

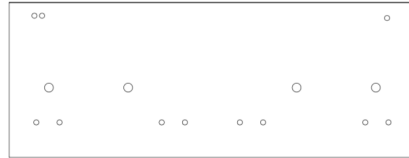
Save...

Run DRC...

Close

Generate Drill Files...

Plot



1. Number of Layers: 4
2. PCB Thickness: 1.6mm
3. TG Rating: TG170
4. Inner Copper Weight: 0.5oz
5. Outer Copper Weight: 1oz
6. Silkscreen Color: White
7. Soldermask Color: Green
8. Via-in-Pad: Fill and Plate
9. Impedance Control: Top Layer 23.00mil single-ended Coplanar Waveguide 50 ohm (+/-8%)

sam@wavidengineer.com
 Layout Design: Sam Feller
 Schematic Design: Sam Feller
 Please direct all questions to Sam Feller
The Awkward Engineer
 Sheet:
 File: clock.kicad_pcb
Title: Clock
 Size: A3 Date: 2019-09-09 Rev: A
 KICAD E.D.A. kicad (5.1.0-0) 10: 1/1



- | | | | | | | |
|---|--|---|---|---|---|---|
| E | 1. Number of Layers: 4 | | | | | |
| | 2. PCB Thickness: 1.6mm | | | | | |
| | 3. TG Rating: TG170 | | | | | |
| | 4. Inner Copper Weight: 0.5oz | | | | | |
| | 5. Outer Copper Weight: 1oz | | | | | |
| | 6. Silkscreen Color: White | | | | | |
| | 7. Soldermask Color: Green | | | | | |
| | 8. Via-in-Pad: Fill and Plate | | | | | |
| F | 9. Impedance Control: Top Layer 23.00mil single-ended Coplanar Waveguide 50 ohm (+-8%) | 1 | 2 | 3 | 4 | 5 |



sam@awkwardengineer.com
Layout Design: Sam Feller
Schematic Design: Sam Feller
Please direct all questions to Sam Feller
The Awkward Engineer

Sheet:
File: clock.kicad_pcb

Title: Clock

Size: A3 Date: 2019-04-09

Rev: A

KiCad E.D.A. kicad (5.1.0-0)

Id: 1/1



1. Number of layers
2. Overall PCB thickness
3. TG rating
4. Inner copper weight
5. Outer copper weight
6. Silkscreen color
7. Soldermask color
8. Via-in-pad
9. Impedance control
10. E-Test Required



1. ~~Identifying polarity of components~~
2. ~~Silkscreen legibility~~
3. ~~Panelization~~
4. ~~PCB properties~~
5. **Specific manufacturer's part numbers**



Symbol Properties

Fields

Name	Value	Show	H Align	V Align	Italic	Bold	Text Size
Reference	C7	<input checked="" type="checkbox"/>	Left	Center	<input type="checkbox"/>	<input type="checkbox"/>	1.016 mm
Value	1uF	<input checked="" type="checkbox"/>	Left	Center	<input type="checkbox"/>	<input type="checkbox"/>	1.016 mm
Footprint	C1	<input checked="" type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>	0.762 mm
Datasheet	~	<input checked="" type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>	1.524 mm
MPN	CL21B105KAFNNNE	<input type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>	1.270 mm
Manufacturer	Samsung	<input type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>	1.270 mm

+

↑

↓

Update Fields from Library...

Symbol

Library Reference: clock-rescue:C

Unit:

☐ Alternate symbol (DeMorgan)

Orientation

☒ 0

☐ +90

☐ +180

☐ -90

Aspect

☒ Default

☐ Mirror around X axis

☐ Mirror around Y axis

Unique ID: 54A89CEC

Edit Spice Model...

Cancel

OK



Symbol Properties

Fields

Name	Value	Show	H Align	V Align	Italic	Bold	Text Size
Reference	C7	<input checked="" type="checkbox"/>	Left	Center	<input type="checkbox"/>	<input type="checkbox"/>	1.016 mm
Value	1uF	<input checked="" type="checkbox"/>	Left	Center	<input type="checkbox"/>	<input type="checkbox"/>	1.116 mm
Footprint	C1	<input checked="" type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>	0.762 mm
Datasheet	~	<input checked="" type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>	1.524 mm
MPN	CL21B105KAFNNNE	<input type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>	1.270 mm
Manufacturer	Samsung	<input type="checkbox"/>	Center	Center	<input type="checkbox"/>	<input type="checkbox"/>	1.270 mm

+

↑

↓

Update Fields from Library...

Symbol

Library Reference: clock-rescue:C

Unit:

☐ Alternate symbol (DeMorgan)

Orientation

☒ 0

☐ +90

☐ +180

☐ -90

Aspect

☒ Default

☐ Mirror around X axis

☐ Mirror around Y axis

Unique ID: 54A89CEC

Edit Spice Model...

Cancel

OK



☒ Group symbols

Field

Show

Group By

Reference

Value

Footprint

Datasheet

MPN

Manufacturer

☒
☒
☒
☒
☒
☒
☐
☐

Add Field...

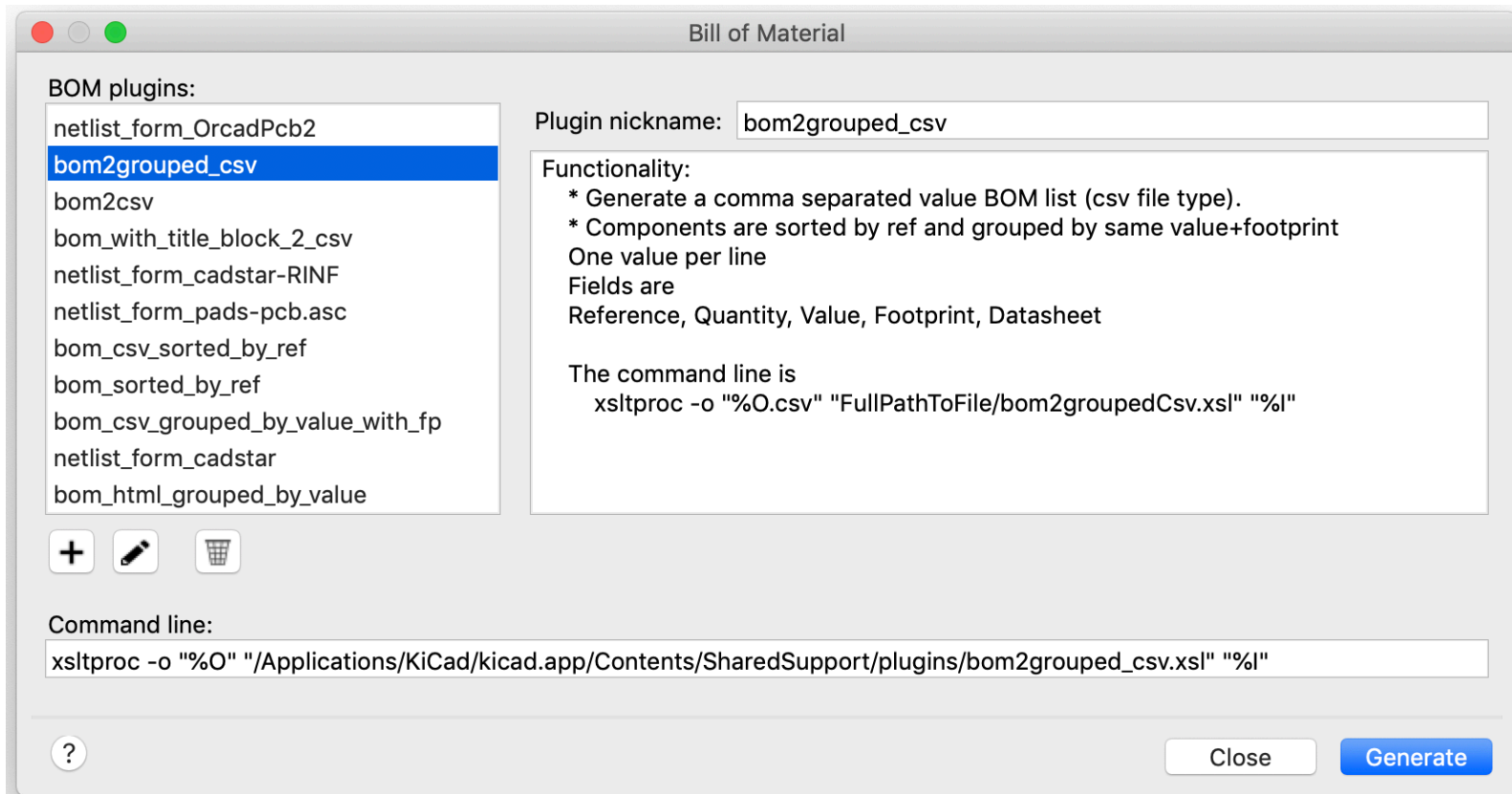
Symbol Fields

Reference	Value	Footprint	atasheet	MPN	Manufacturer	Qty
R17	0R	SM0805	~	DNP		1
> R1, R16	0K	SM0805	~	ERJ-6GEY0R00V	Panasonic	2
> C2, C4	12.5pF	SM0805	~	CL21C120JB61PNC	Samsung	2
> C5, C6, C8	.01uF	SM0805	~	CL21B103KBANNNC	Samsung	3
C7	1uF	C1	~	CL21B105KAFNNNE	Samsung	1
C10	1uF	SM0805	~	CL21B105KAFNNNE	Samsung	1
L1	4.7uH	SM0805	~	LQM21NN4R7K10L	Murata	1
> C1, C3	10uF	SM0805	~	CL21B106KOQNNNE	Samsung	2
> R13-R15	220	SM0805	~	ERJ-6GEYJ221V	Panasonic	3
> R3, R4, R9, R12	10K	SM0805	~	RMCF0805FT10K0	Stackpole	4
> R7, R8	33K	SM0805	~	ERJ-6ENF3302V	Panasonic	2
> R10, R11	51K	SM0805	~	ERJ-6GEYJ513V	Panasonic	2
R2	100K	Chris_Denney_Library:bourns_knob	~	PTV111	Bourns	1
R5	130K	SM0805	~	ERJ-6ENF1303V	Panasonic	1
> R6, R18	200K	SM0805	~	ERJ-6ENF2003V	Panasonic	2
> DS1, DS2	AMMETER	ammeter		DNP		2
IC1	ATTINY84A-SS	SO14E		ATTINY84A-SSU	Microchip	1
CON1	AVR-ISP-6	ISPheader		DNP		1
> BT1, BT2	BATTERY_CLIP	AA_Battery_Clip_THRU_HOLE_BK-92	92		Keystone	2
> P2-P4, P8, P10	CONN_2	SIL-2		DNP		5

Apply, Save Schematic & Continue

Cancel

OK





CircuitHub

ExplorePartsInternal

HelpBlog

EChrisDenney
Acting as EChrisDenney

admin guest

Drag files here

- or -

Choose files

To receive a quote, please upload your original EDA design files.[Why?](#)

By default all your information is private.

CircuitHub accepts **Eagle**, **KiCad** and **Altium** files.

CircuitHub does not accept gerbers, as they don't provide all the information required for manufacture and assembly.

- For **Altium** projects, upload your **.pcbdoc** and **all** the relevant **.schdoc** files.
- For **Eagle** projects, upload your **.brd** file and one **.sch** file.
- For **KiCad** projects, upload your **.kicad_pcb**, **-cache.lib** and **.sch** files.

Display a menu

A blue circular icon with a white speech bubble inside, indicating a chat or support feature.



No website or product link provided

Revision #12

New Revision

Fork 3

Download assets



Parts

Schematics

Board

Firmware

Settings

ORDER BY References

Part					Quantity	Quantity	Price
BATTERY_CLIP	AA_Battery_Clip_T...	92 Keystone	BATTERY CONTACT CLIP A/AA PC PIN		1,100	\$0.1287	\$141.5700
					100 for attrition		
10uF	SM0805	C0805C104K5RACTU Kemet	CAP CER 0.1UF 50V X7R 0805		1,050	\$0.0294	\$30.8385

Assembled Boards

500 boards

Lead Time

Ships on May 27 (20 business days) [9 pcb, 10 assembly]

	Quantity	Unit Price	Total
PCB	500	\$2.01	\$1,004.16
Parts	500	\$4.01	\$2,007.38
Assembly	500	\$8.91	\$4,452.77
Total	500	\$14.93	\$7,464.31

[Hide details](#)

Display a menu

Order





No website or product link provided

Revision #12

New Revision

Fork 3

Download assets



Parts

Schematics

Board

Firmware

Settings

ORDER BY References

Part					Quantity	Quantity	Price
BATTERY_CLIP	AA_Battery_Clip_T...	92 Keystone	BATTERY CONTACT CLIP A/AA PC PIN		1,100	\$0.1287	\$141.5700
					100 for attrition		
10uF	SM0805	C0805C104K5RACTU Kemet	CAP CER 0.1UF 50V X7R 0805		1,050	\$0.0294	\$30.8385

Assembled Boards

500 boards

Lead Time

Ships on May 27 (20 business days) [9 pcb, 10 assembly]

	Quantity	Unit Price	Total
PCB	500	\$2.01	\$1,004.16
Parts	500	\$4.01	\$2,007.38
Assembly	500	\$8.91	\$4,452.77
Total	500	\$14.93	\$7,464.31 Hide details

Order



Display a menu



Assembled Boards



500 boards

Lead Time



Ships on May 27 (20 business days) [9 pcb, 10 assembly]



Parts



Schematics



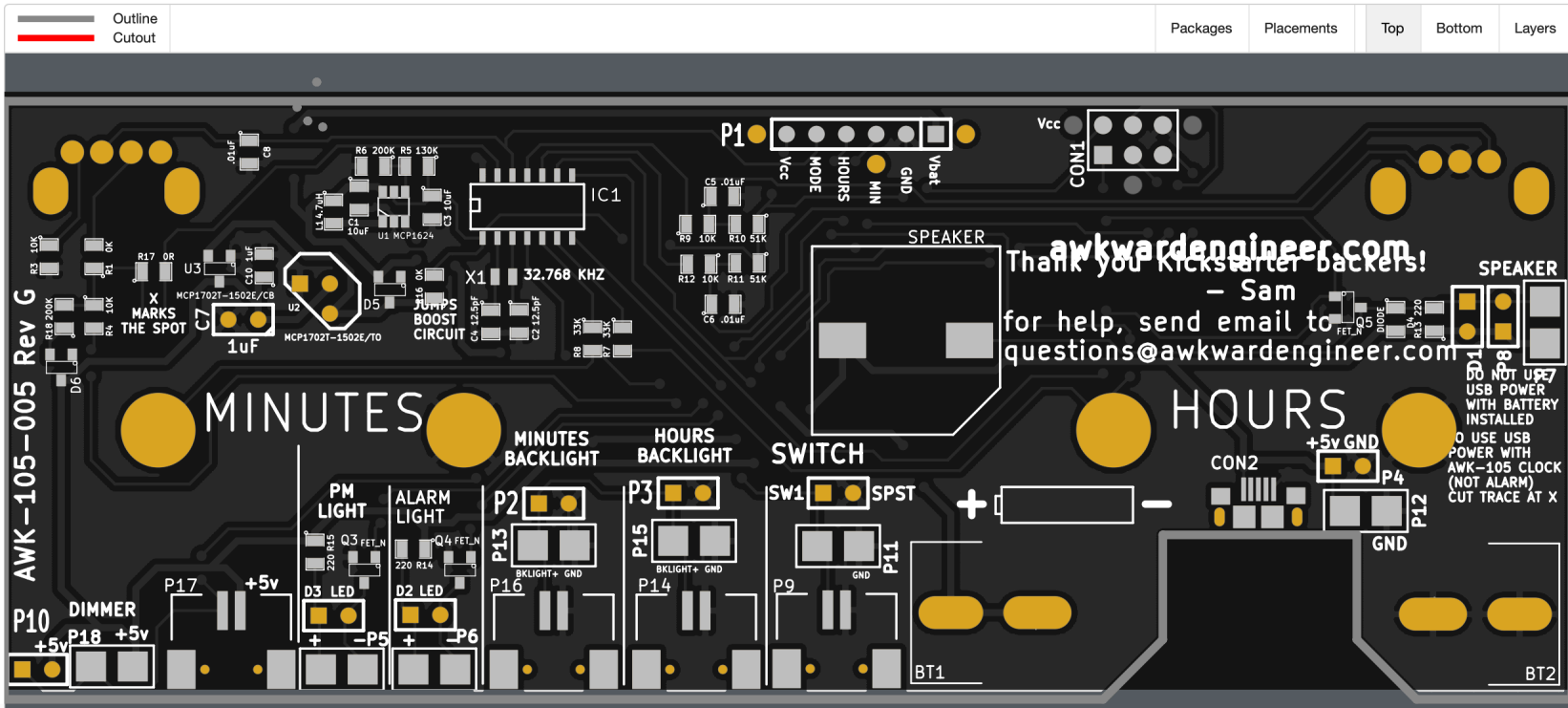
Board



Firmware



Settings





Color

Soldermask color

Black

Silkscreen color

White

Edit

Miscellaneous

Cutouts

Board does not contain cutouts

Via fill

No via fill required

Add via fill locations

Via in pad

No via in pad fill required

Add locations which require filling and plating

Castellated edges

Does not require castellated edges

Blind/Buried layer sets

0

Stackup

Board thickness

1.6

Copper weight

Outer1

InnerNot specified

Material

☒ FR4 TG 130-135 ☐ FR4 High TG

Custom stackup

Add custom stackup

Edit

Specifications

Width	133.35	mm
Length	50.81	mm
Number of layers	2	



1. Include Manufacturer
2. Include MPN
3. Do this when you assign footprint
4. Real legitimate quotes in seconds - not days - using CircuitHub



1. Identifying polarity of components
2. Silkscreen legibility
3. Panelization
4. PCB properties
5. Specific manufacturer's part numbers



1. Use square pads or clear silkscreen.
2. 1x1mm character size. 0.18mm lines.
3. Draw overhanging parts
4. Include PCB properties in dwgs. User
5. Add Manufacturer and MPN to symbols



Bonus Tip



General

Local Clearance and Settings

Custom Shape Primitives

Pad number:

17

Net name:

<no net>

Pad type:

SMD

Shape:

Rounded Rectangle

Position X:

0

mm

Position Y:

0

mm

Size X:

1.6

mm

Size Y:

1.6

mm

Orientation:

0.0

deg

Shape offset X:

0

mm

Shape offset Y:

0

mm

Pad to die length:

0

mm

Trapezoid delta:

0

mm

Trapezoid axis:

Vertical

Corner size:

15.6

%

Corner radius:

0.2496

mm

Hole shape:

Circular

Hole size X:

0

mm

Hole size Y:

0

mm

Copper:

F.Cu

Technical layers:

☐ F.Adhes

☐ B.Adhes

☐ F.Paste

☐ B.Paste

☐ F.Silks

☐ B.Silks

☒ F.Mask

☐ B.Mask

☐ Dwgs.User

☐ Eco1.User

☐ Eco2.User

Footprint REF** (LFCSP-16-1EP_3x3mm_P0.5mm_EP1.6x1.6mm), front side, rotated 0.0 deg

☐ Show pad in outline mode



General

Local Clearance and Settings

Custom Shape Primitives

Pad number:

17

Net name:

<no net>

Pad type:

SMD

Shape:

Rounded Rectangle

Position X:

0

mm

Position Y:

0

mm

Size X:

1.6

mm

Size Y:

1.6

mm

Orientation:

0.0

deg

Shape offset X:

0

mm

Shape offset Y:

0

mm

Pad to die length:

0

mm

Trapezoid delta:

0

mm

Trapezoid axis:

Vertical

Corner size:

15.6

%

Corner radius:

0.2496

mm

Hole shape:

Circular

Hole size X:

0

mm

Hole size Y:

0

mm

Copper:

F.Cu

Technical layers:

☐ F.Adhes

☐ B.Adhes

☒ F.Paste

☐ B.Paste

☐ F.SilkS

☐ B.SilkS

☒ F.Mask


☐ B.Mask

☐ Dwgs.User

☐ Eco1.User

☐ Eco2.User

Footprint REF** (LFCSP-16-1EP_3x3mm_P0.5mm_EP1.6x1.6mm),
front side, rotated 0.0 deg



☐ Show pad in outline mode



- Gang soldermask problems
- Plated hole sizes
- Fiducials
- Smallest component sizes
- Single sided vs double sided
- SMT parts being too close to thru-hole
- Cutouts
- Wrong footprints or part numbers
- Your 0402 footprint is probably awful
- Solder paste 1:1 with copper
- Overhanging microUSB ports
- Paste on hybrid components (SMT and Thru-hole in one part)



More Info

worthingtonassembly.com/best-practices

echrisdenney.com



Chris Denney - CTO - Worthington Assembly Inc.

Chris Denney - Jerk That Tells You There's a Problem - CircuitHub Inc.

`cdenney@worthingtonassembly.com`

@WAssembly on Twitter

Thank to Sam Feller (aka The Awkward Engineer) for letting me use his project in this presentation <https://www.awkwardengineer.com/>