

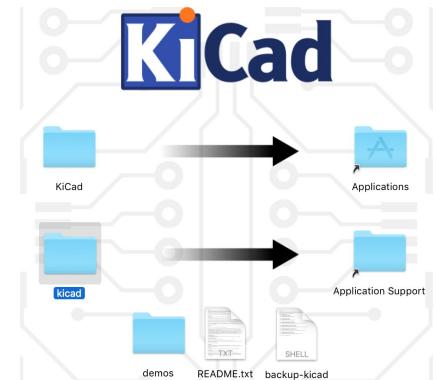


Electronic Product

workshop series

Download KiCAD

1. Ensure you have 4 GB of free space on your system.
2. [Download KiCAD](#).
3. Install it with the default instructions for your system.
 - o You don't need to install the libraries for this workshop.
4. Download the [aero-1 project](#) folder and open *aero-1.pro* with KiCAD



Introductory Information

PCB: Printed circuit board

KiCAD: Schematic and PCB layout software. Alternatives: Altium, EAGLE.

Schematic Symbol: an electrical representation of a component

Footprint: physical representation of a component

Workshop Outline

1. Documentation and up-front management

- Required documentation for a successful project.

2. Schematic capture, electronics design, and footprints

- Designing the electronics of the project.

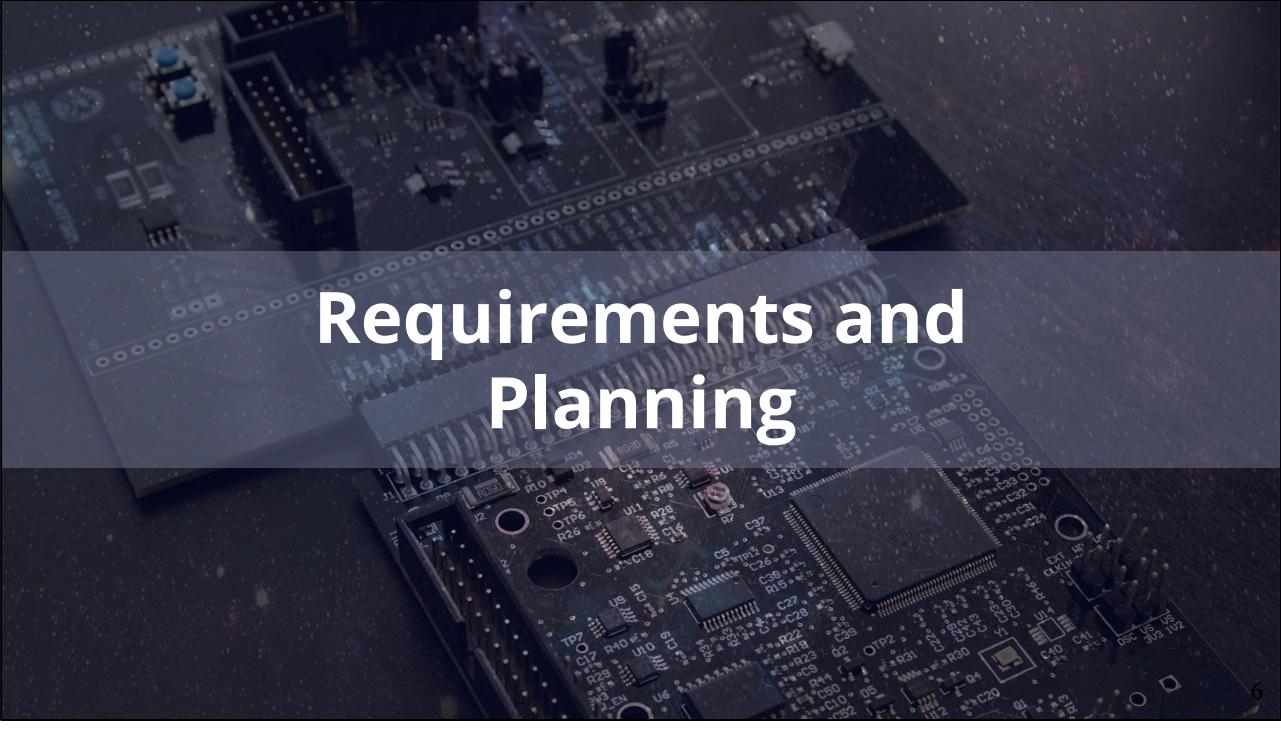
3. PCB layout, review, and manufacturing

- How to lay out boards, review them, and get them professionally manufactured.

Performance Objectives

After this workshop, you will be able to:

- Create a schematic symbol and a footprint from a datasheet.
- Use and extend an existing symbol/footprint library.
- Create schematic sheets and use the following features:
 - Various kinds of labels
 - Hierarchical sheets
- Demonstrate basic PCB layout skills on a 4-layer board.
- Explain the PCB development flow and considerations related to manufacturing.
- Review a PCB design and a schematic.
- Explain how to manage details associated with an actual project such as BoMs, pin mapping, and requirements.



Requirements and Planning

Challenge: Knowing What to Design

Develop good requirements and a feature list for the product before you start.

- **Think about testability and usage.**
 - Do you want to put it in a breadboard?
 - Does it need to plug into anything else?
 - Will it always be used with a bench supply? What if someone wants to take it home?
- **Seek out app notes and reference designs.**
 - SparkFun PCBs are often simple and open. These are a great way to learn.
 - Manufacturer development boards are a good example if using a microcontroller.
 - Example: copy the LaunchPad design since you know it works.
 - For every large chip, there will be application notes with recommended support components.

Product Requirements/Feature List

- [See the example](#)
- GitHub/GitLab issues are a great way to track progress.
 - An issue per main component can have these tasks:
 - Find/make symbol and footprint
 - Add support components
 - Make schematic section or sheet
 - Get reviewed

Flip SD card

#8 · opened 4 months ago by Richard Arthurs · [Next revision](#)

Testing headers

0 of 1 task completed

#7 · opened 4 months ago by Richard Arthurs · [Next revision](#)

FPGA power

1 of 1 task completed

#6 · opened 4 months ago by Richard Arthurs · [Next revision](#)

LEDs

#5 · opened 4 months ago by Richard Arthurs · [Next revision](#)

Power Topology

2 of 2 tasks completed

#4 · opened 4 months ago by Richard Arthurs · [Next revision](#)

D2 Pin 1 Indicator

#3 · opened 4 months ago by Richard Arthurs · [Next revision](#)

DNP labels

#2 · opened 4 months ago by Richard Arthurs · [Next revision](#)

Silkscreen Labels

2 of 3 tasks completed

#1 · opened 4 months ago by Richard Arthurs · [Next revision](#)

Challenges With PCB Design

It will take longer than you expect, and rushing will lead to problems.

- ORCASat OBC:
 - Component selection took 2 months because we were rigorous and documented everything. This effort was worth it!
 - Layout began in December, assembly in March.
 - Low productivity over winter break.
 - Found a critical schematic flaw the day we were going to order.
- Assume 2 weeks for PCB manufacture. One weekend for assembly. One month for bringup if you're busy.
 - SFUSat OBC v1 didn't work for two months. Root cause: a mixed up pin on a reset button.

PCB Timeline for Design Teams

Component Research	1 week/component/person
Component Review	1 week/component/person
Schematic Creation	2 weeks +
Board Layout	2 weeks (2-layer, first time)
Board Review	2 weeks (include 1 week of not looking at the board)
Board Ordering	2 weeks
Board Assembly	1 week
Bringup	1 hour - 3 months

Pin Mapping

- Create a pin mapping spreadsheet if you're using a microcontroller.
 - Add all microcontroller pins from its datasheet into a spreadsheet.
 - Create a list of your signals. These should be the net/hierarchical label names in the schematic.
 - *FLASH.SPI.MISO, ACCEL.SPI.CS*
 - Add signal names to each microcontroller pin for your own functionality.
 - You can use data validation in sheets/Excel to detect duplicates and do other fancy things like mapped/unmapped colouring.

Signal Designator	Status	In Altium
OBC.CAN1.RXD	Mapped	y
OBC.CAN1.TXD	Mapped	y
GPS.RX	Unmapped	on backplane
GPS.TX	Unmapped	on backplane

Functional Group	MCU Name	Altium Unit	MCU Number	Signal Map	Default Value	Input/Output	Pull-up/Pull-down	Backplane Pin	Notes
GIO	GIOA[0]	G	2	RTCB.CS0	1	Out	Up	OB	
GIO	GIOA[2]/N2HET2[0]/EQEPII	G	9	FLASH.CS0	1	Out	Up	OB	
GIO	GIOA[5]/EXTCLKIN1/EPWM1A /N2HET1_PIN_nDIS	D	14	OB.CEXTCLKIN				OB	Not a GPIO, this is a clock signal.

BoM Generation

The bill of materials is a list of all components for the board.

Description	Reference Designator	Subcircuit	Footprint	Manufacturer	Manufacturer Part No.	Mouser	Digikey	Price per unit (CAD)	Units Per Board	Quantity to purchase
TMS570LS0714 MCU	U13	MCU	144-LQFP	TI	TMS5700714APGEQQ1R	595-S57007		\$27.19	1	5
CAN Transceiver	U17	CAN	8-SOIC	TI	TCAN334DR	595-TCAN3296-44139		\$3.14	1	5
1V2 Voltage Supervisor & Watchdog	U16	WD	10-VDFN	TI	TPS3850G12QDRCRQ1	595-PS3851296-45516		\$3.14	1	5
Capacitors										
4.7nF Capacitor, 6.3V	C48	CAN	0403	Murata	GCM155R71H472KA37D	81-GCM155490-49151		\$0.07	1	10
16pF Capacitor, 6.3V	C42,C43	XTAL	0402	Murata	GJM1555C1H160GB01D	81-GJM155490-80811		\$0.23	2	10
Miscellaneous										
128Mb ECC NOR Flash		FLASH_EV	8-SOIC	Cypress Semic	S25FL127SABMFI100	727-S25FL428-39761		\$3.76	0	2
26-Pin Cable Connector		TH	-	3M	M3AAA-2606J	M3AAA-26		\$3.81	1	5
J-Link EDU	-	-	-	Segger	J-Link EDU	943-8.08.90 899-10081		\$84.42	0	5
Anti-Static Bags	-	-	-	SCS	300810		SCP315-NC	\$0.42	0	10

BoM Generation

1. Start the BoM early in the project. Be sure to add non-schematic items like ribbon cables, anti-static bags, and jumpers.
2. Through schematic generation, record any part numbers you decide upon.
3. When layout is done, export a BoM from KiCAD and manually group components.
 - a. Group by function (IC, resistor, etc.)
 - b. Consolidate like-valued components to generate a line for generic 0.1 μF capacitors, for example.
4. Go shopping for parts you don't have yet. Typically passives.
5. Fill in the supplier links, prices, quantity to order.
6. Mark line items at several stages: reviewed, added to order, received.

BoM and Ordering Tips

- Include the footprint.
- Include all reference designators in the BoM
- Quantity to order:
 - 1-2 extra ICs, fewer if ICs are large and easy to place.
 - 20% more of frequently-used passives (0.1 µF, 10 kΩ), 2 extra for rare small passives.
- Mouser tends to be cheaper for ICs.
- Make 3 copies minimum. It's not that much more expensive.
- Manually review the BoM and mark each item when it's reviewed.

Customer Reference Field

The customer reference field on DigiKey and Mouser will print your text on the component bag.

Not having to sort components during assembly saves a lot of time.

QUANTITY

MY_COMPONENT

Add to Cart

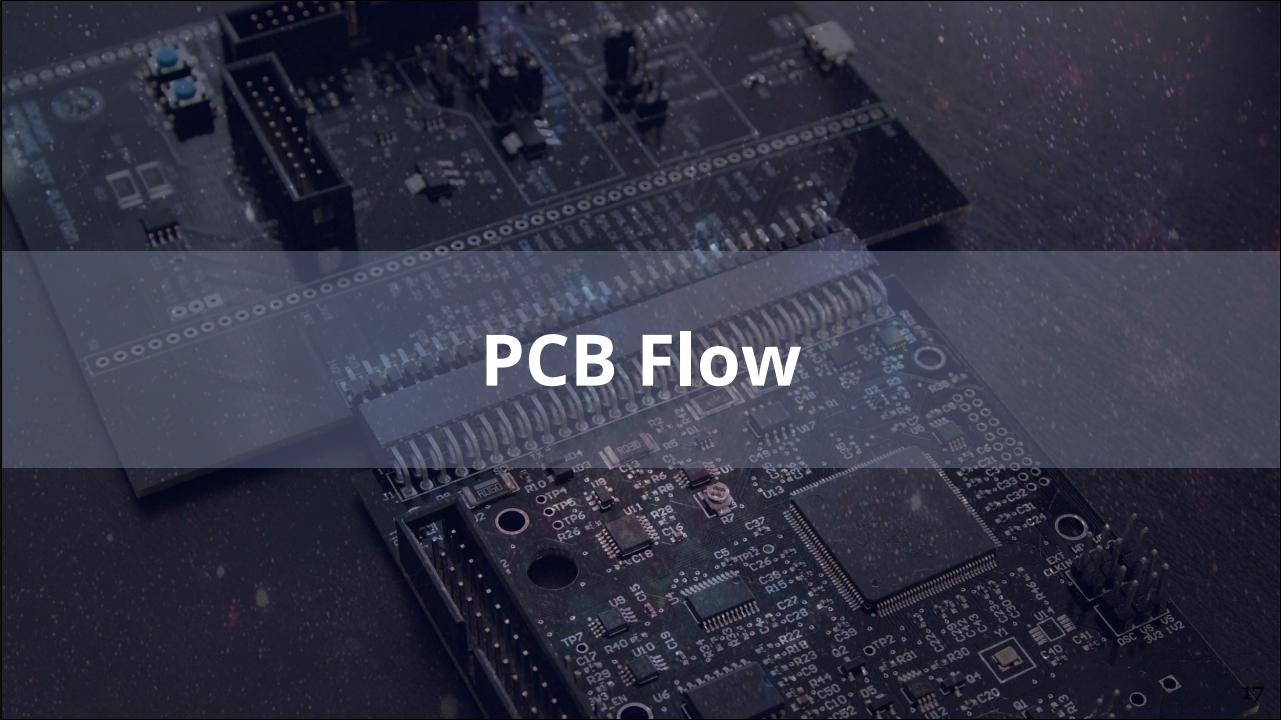
All prices are in CAD.

PRICE BREAK	UNIT PRICE	EXTENDED PRICE
1	131,145.93000	\$131,145.93

BoM Reduction

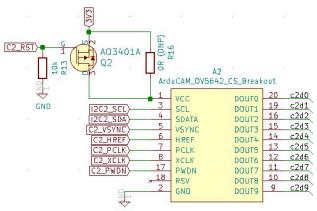
Try to design with as few unique components as possible. This makes ordering and assembly easier.

- Use a single MOSFET through the entire design, for example.
 - Useful as a generic LED driver and for switching components (DMN2075-U).
- Select the same footprint for all passives, consolidate values.
 - Decoupling caps can often all be the same value (0.1 µF).
 - All pullups/pulldowns can often be the same value (10 kΩ).
- Pin headers: buy breakaway headers in bulk instead of individual 2-pin and 7-pin headers, for example.
 - Connectors are much cheaper on EBay, order many at the start of a project.



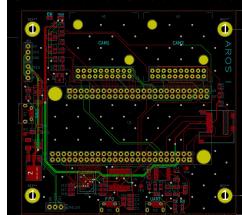
PCB Flow

PCB Process Flow



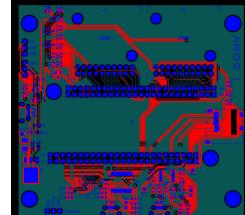
Schematic

The electrical representation of your circuit.



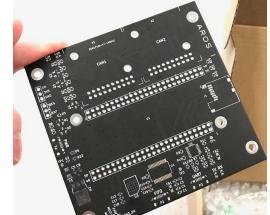
Board Layout

Digital version of the completed PCB.



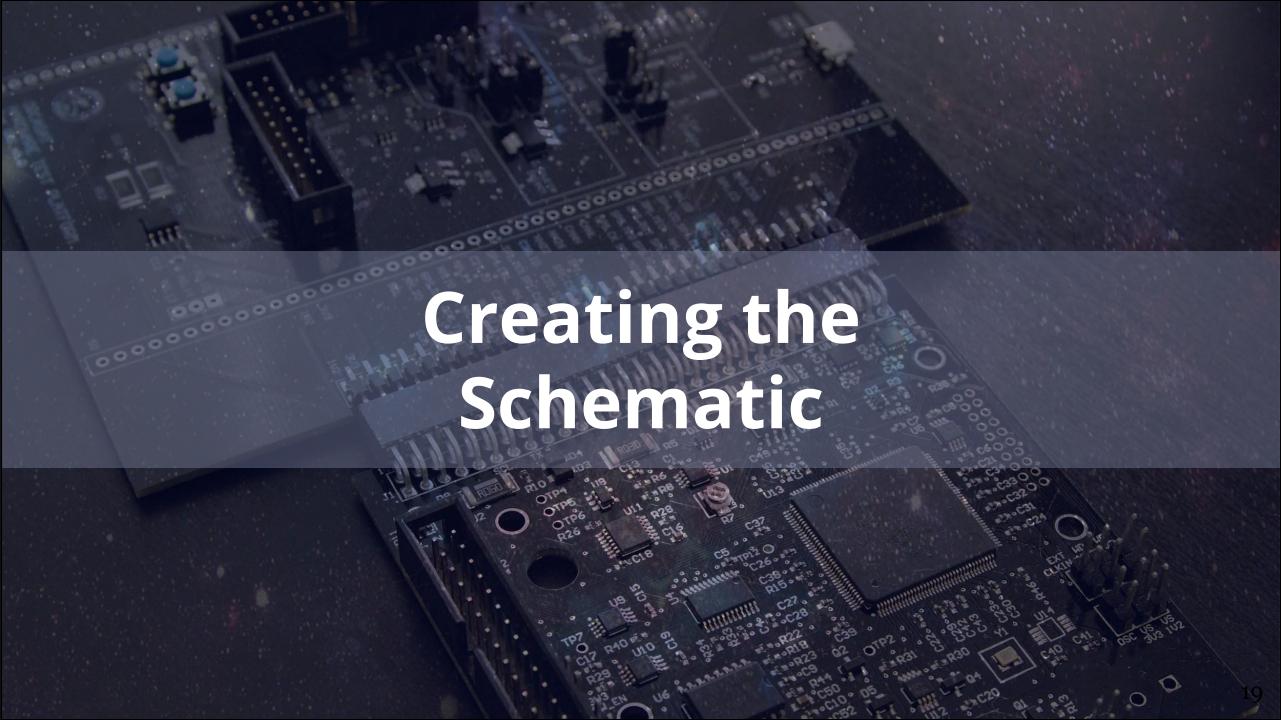
Gerber Files

Files the board house will use to manufacture the board.



PCB

The end result!



Creating the Schematic

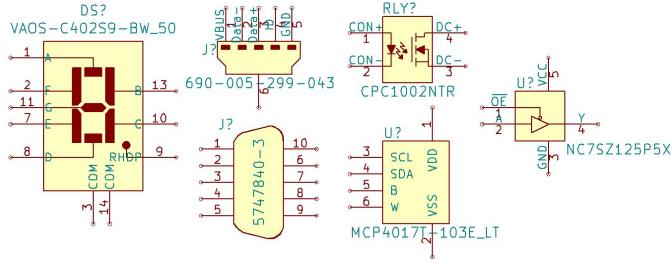
Schematic Creation Steps

Designing the actual electronics for your system.

1. Know your requirements and features.
2. Select major parts.
3. Create an initial schematic hierarchy.
4. Populate sections of the hierarchy, making symbols as you go.
5. Generate netlist and send to PCB layout software.

Schematic Symbols

Symbols are the drawings of a component that show the pins.



- Symbols are placed in libraries.
 - Each team should maintain a library of common parts.
 - You can also maintain libraries alongside the project (as in the example).

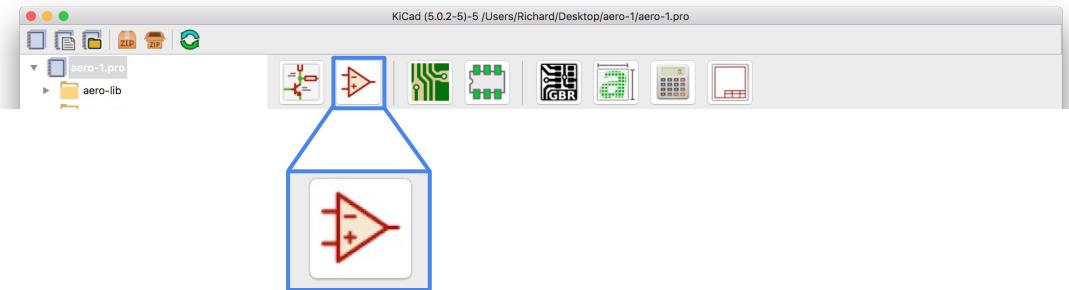
Creating a Symbol

We will make a symbol and a footprint at the same time. The general steps are as follows:

1. Get the datasheet.
2. Make the symbol.
3. Determine if you need to make a footprint.
 - You often will, and hunting down the footprint can be difficult.
 - For parts like standard resistors and capacitors, use a common footprint.
 - Review every footprint, even if it's from the manufacturer. They often have errors.

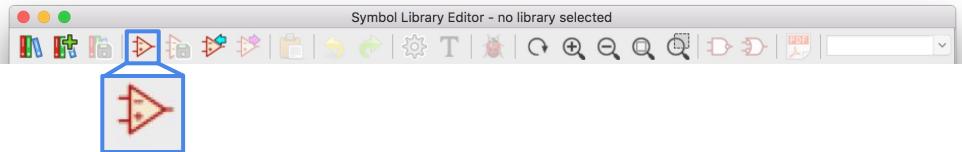
Symbol Creation

1. Open the Symbol Editor

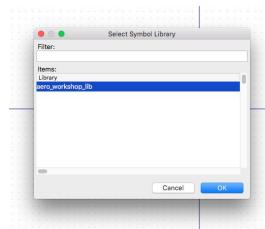


Symbol Creation

2. Create new symbol

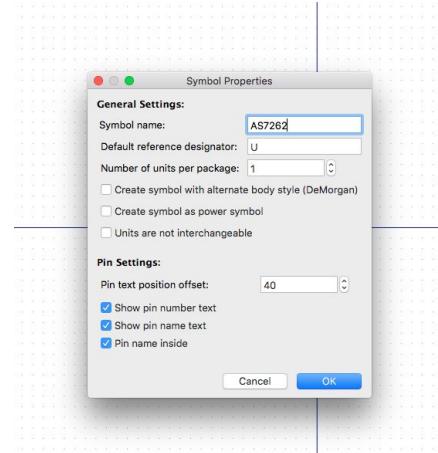


3. Select library



Symbol Creation: Step 1,2

- If the component has a footprint-specific pinout, add the footprint name to the symbol name.
 - Ex: TMS570LS0714 PGE is 144 pins, others have 100 pins.
 - Ex: AS7262_LQFP
- Select the correct reference designator:
 - U = Chip
 - R = Resistor
 - Y = Crystal
 - [Full List](#)



Pinout

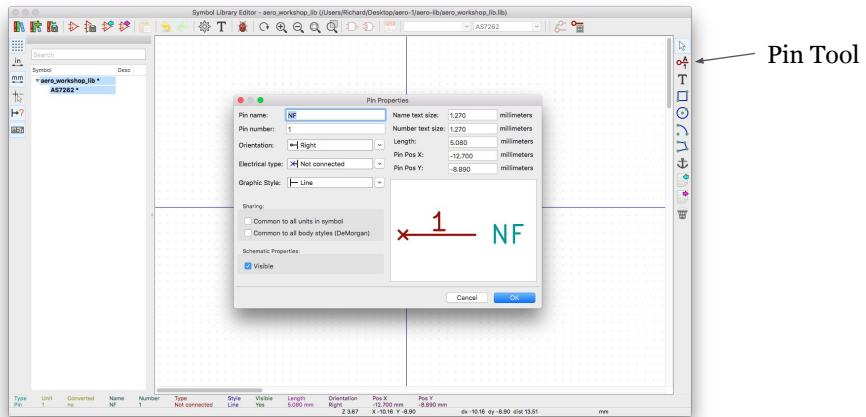
The pinout can be found in the datasheet. It's often near the top.

[Datasheet page 3](#)

Make sure to use the pinout for the correct package, if you have a choice.

Pin Number	Pin Name	Description
1	NF	Not Functional. Do not connect.
2	RESN	Reset, Active LOW
3	SCK	SPI Serial Clock
4	MOSI	SPI Master Out Slave In
5	MISO	SPI Master In Slave Out
6	CSN_EE	Chip Select for external serial Flash memory, Active LOW
7	CSN_SD	Chip Select for SD Card Interface, Active LOW
8	I ² C_ENB	Select UART (Low) or I ² C (High) Operation
9	NF	Not Functional. Do not connect.
10	NF	Not Functional. Do not connect.
11	RX/SCL_S	RX (UART) or SCL_S (I ² C Slave) Depending on I ² C_ENB
12	TX/SDA_S	TX (UART) or SDA_S (I ² C Slave) Depending on I ² C_ENB
13	INT	Interrupt, Active LOW
14	VDD2	Voltage Supply

Create Pins



Pin Labelling Tips

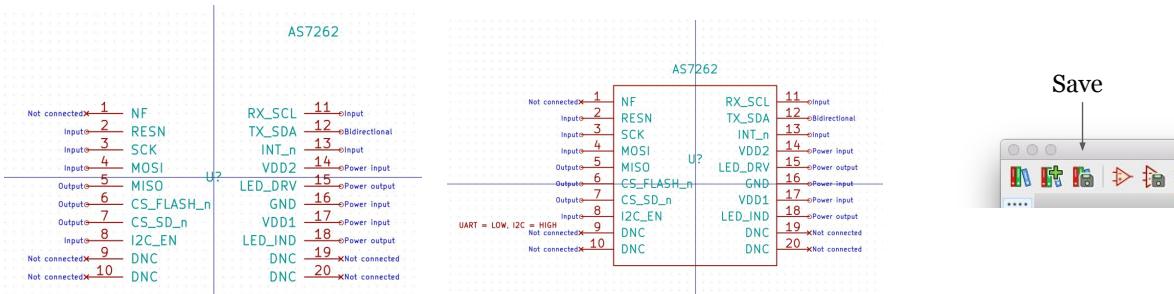
1. Active LOW pins get a **_n** suffix.
2. If a pin name is more reasonable to you, use that name.
 - o **CSN = CS_n**
 - o **I2C_ENB = I2C_EN** (ENB is long, I always use EN)
 - o **NF = DNC = NC** (Non-functional to do not connect)
3. Add text to indicate any weird things about a pin.
4. The electrical type is not critical, but it's nice to have.
 - o I don't find electrical rule check very useful, which is where this usually comes into play. I often end up doing weird things with pins that ERC doesn't like, so I spend more time waiving errors than I save by manually checking things.

KiCAD Mechanics and Shortcuts

Click, release, keyboard shortcut, drag, click again.

- Shortcuts
 - **c = copy**
 - **m = move**
 - **e = edit properties**

Finishing the Symbol



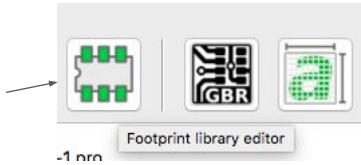
Pin labelling complete.

Add a box, center the text, add a note about pin 8.

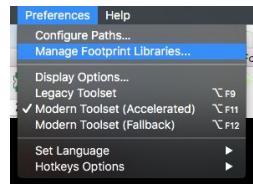


Creating a Footprint

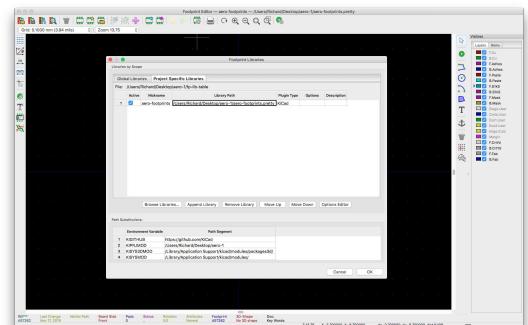
1. Open the Footprint Editor



2. Go to the footprint library.

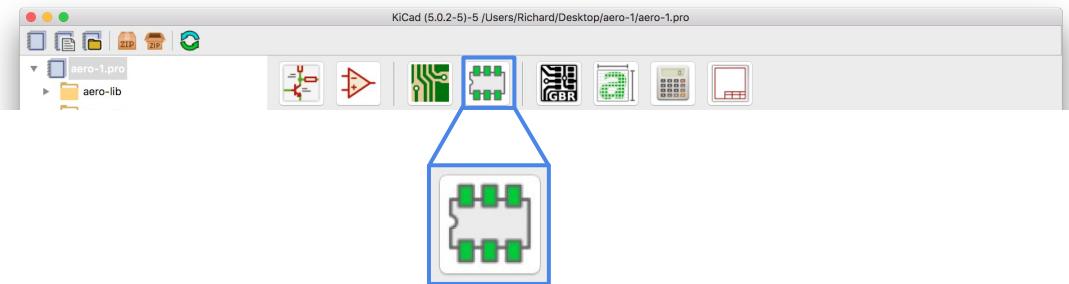


3. Select library.



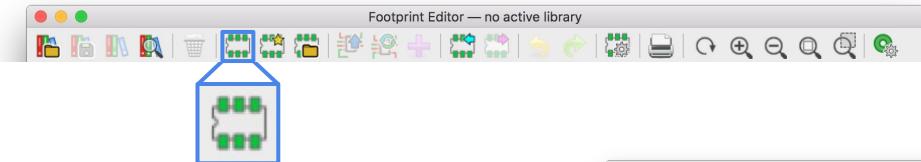
Footprint Creation

1. Open the Footprint Editor

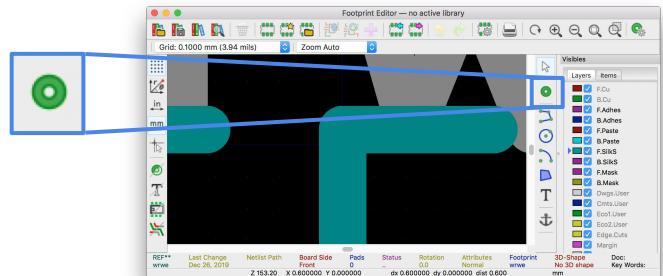


Footprint Creation

2. Create a new footprint

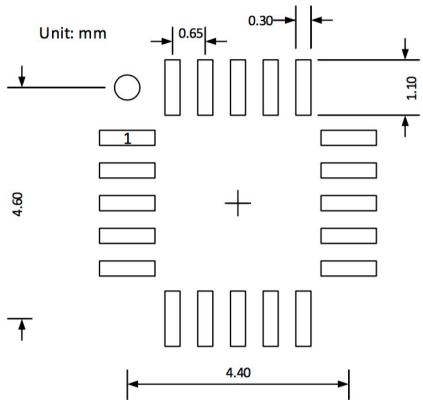
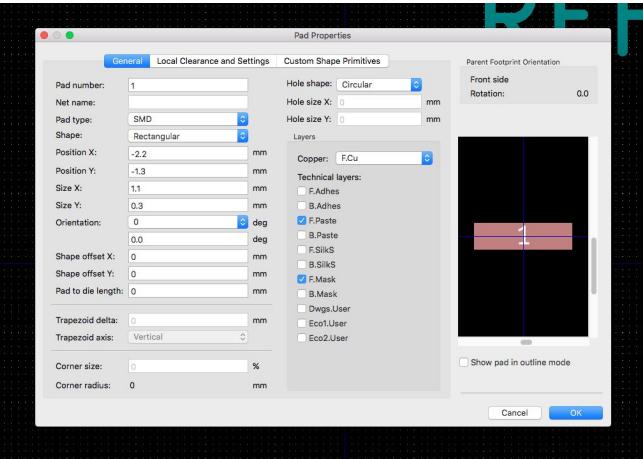


3. Add pads



Pad Creation

On this screen, you define the size of a pad or a through-hole.



[Datasheet page 36](#)

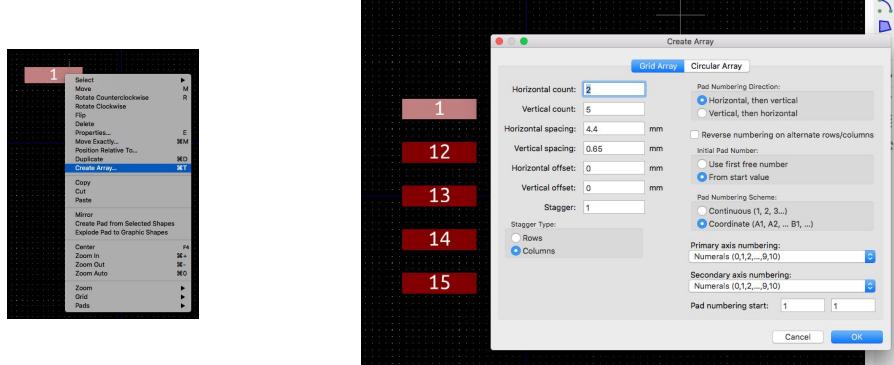
Footprint and Pad Tips

1. Pad properties input boxes can do math:
 - o Position X: $0.65*4$
 - o Position Y: $0.65*4$
 - o This is useful since footprint drawings never have useful dimensions.
2. Take care with your reference point. Drawings usually use centres of pads and components.
3. -Y is up for some reason :/
4. Use the array feature
5. Always put an indicator for pin 1. Make sure it's not ambiguous.
6. Add outlines on the silkscreen layer for connectors.
 - o Connector bodies extend way beyond the pad area, and you don't want to accidentally place components there.

Array

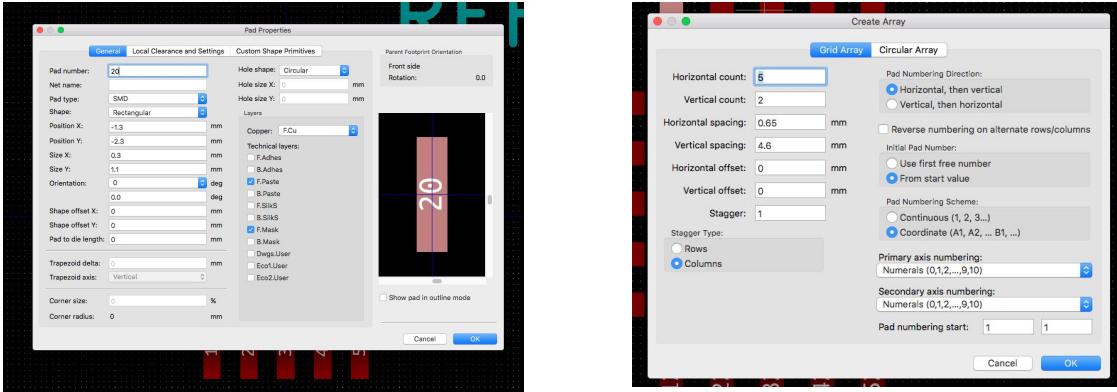
Use the array feature to quickly replicate pins with a desired spacing.

- Right-click the pin, *Create Array...*.
- Enter the settings shown. This will create two columns of pins with correct spacing.

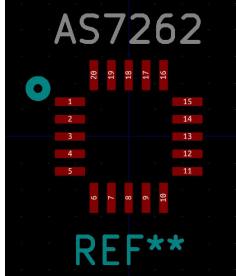


Create Top and Bottom Pins

Create the top left pin (pin 20) and use an array to create two rows of pins.



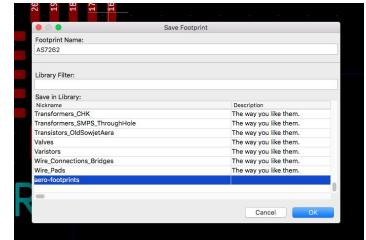
Add Pin 1 Indicator, Save



Use the circle tool to add a pin 1 indicator



Save the footprint



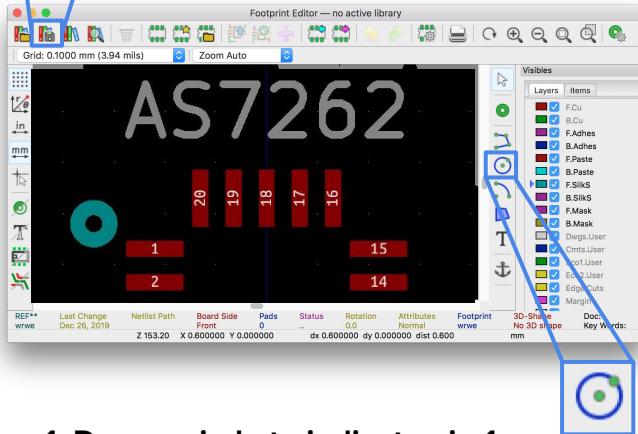
Set the name



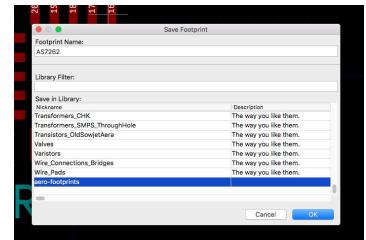
Add Pin 1 Indicator, Save



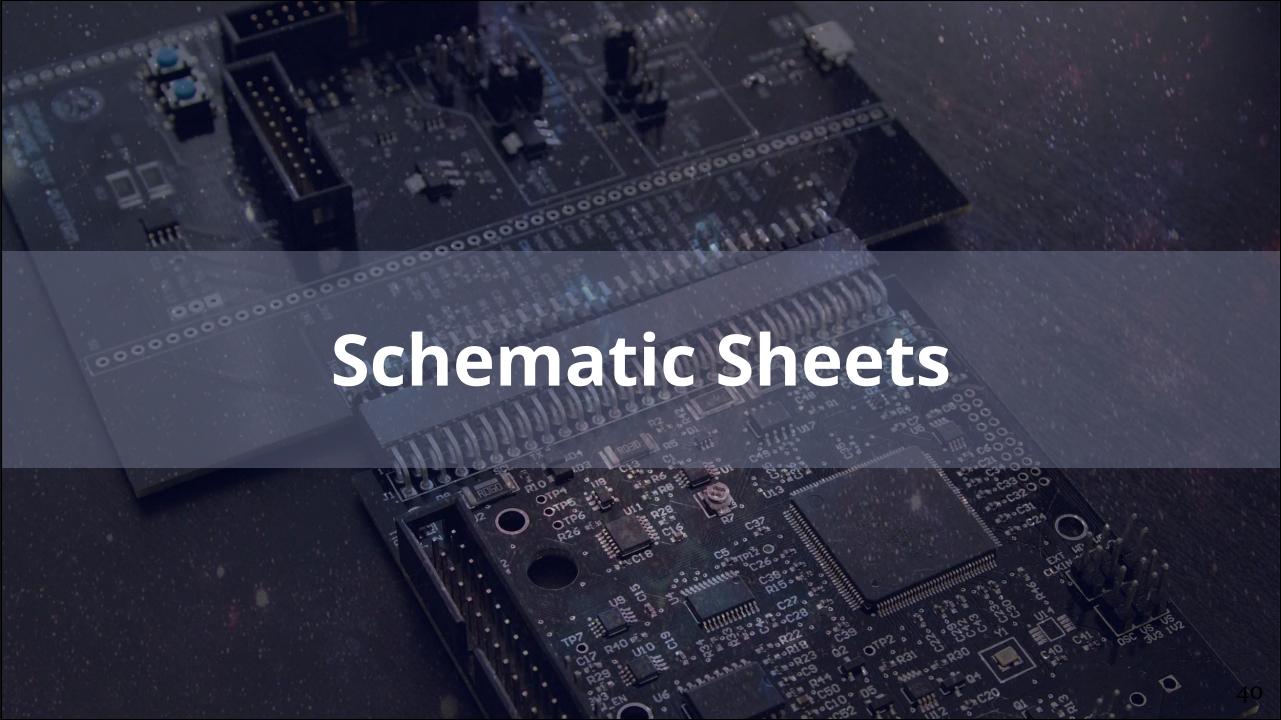
2. Save the footprint



1. Draw a circle to indicate pin 1



3. Set the name



Schematic Sheets

Hierarchical Schematics

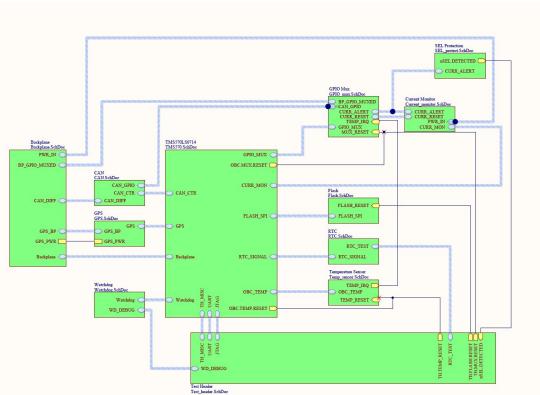
A schematic hierarchy splits the schematic into multiple "sheets."

- It is also possible to create a single-sheet schematic.
 - This is fine with a small project.
- Each sheet should have a section of the functionality.
 - Ex: power, microcontroller, misc. chips
- Sheets are connected together with labels.

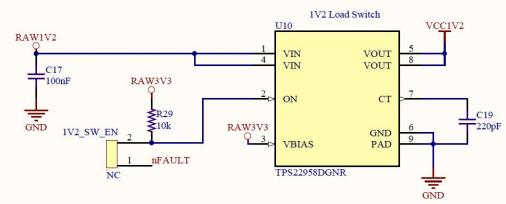
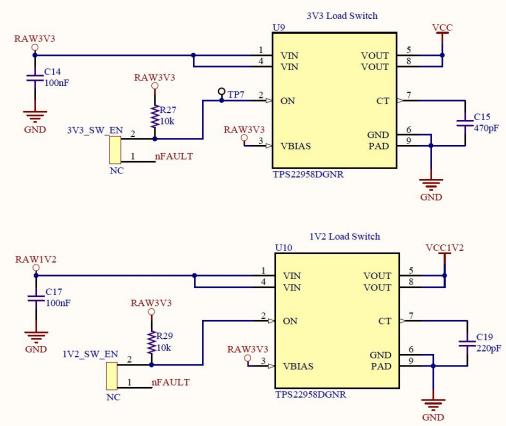
I like to lay my top-level schematics out like a block diagram for the system.

Schematic Flowdown

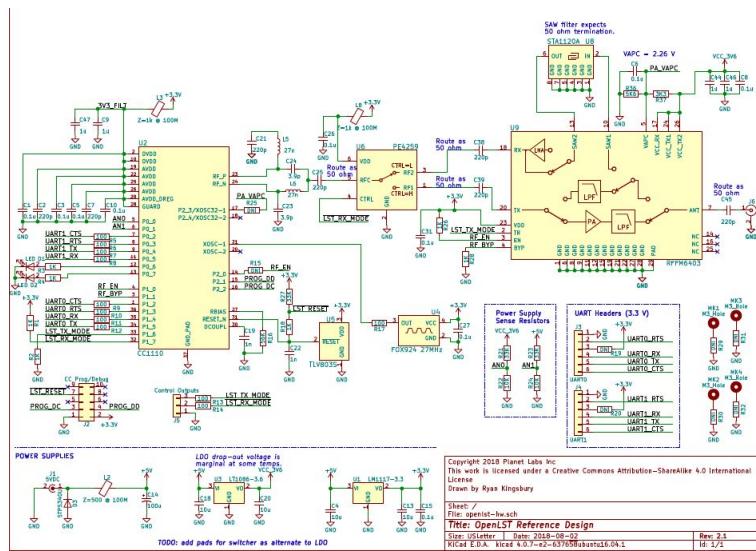
Top-level sheet.
The overview.



Second level of hierarchy.
The details.



A Schematic That Needs Hierarchical Sheets



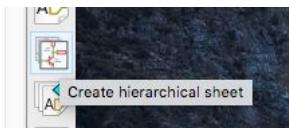
Electronic Product Workshop | richardarthurs.com

43

<https://github.com/OpenLST/openlst-hw/blob/master/pdf/openlst-hw.pdf>

Creating Hierarchical Sheets

1. Create the sheet symbol.



2. Give it a name.



Editing Schematics

Add components from libraries and wire them up.

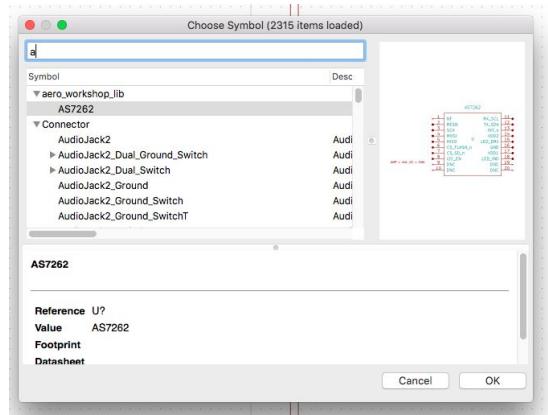
- Add extra documentation text everywhere!
 - Example: describing jumper settings or notes from a datasheet.
 - Labelling what various pieces are for (ex: power-on indicator LED)
- Try to minimize crossed-over wires.
- Use net labels frequently to clean things up.
- Put in component values as soon as you know them.
- Maintain a TODO list on a top-level sheet.

Add the Sensor

1. Add Component



2. Search for the component in the library.



Electronic Product Workshop | richardarthurs.com

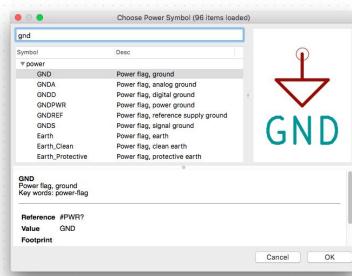
Add Power Connections

Power connections are special: they are shared across sheets automatically.

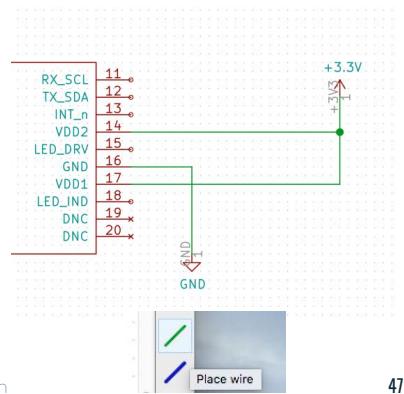
1. Click "Place power port"



2. Search for 3v3 and GND. Add them.



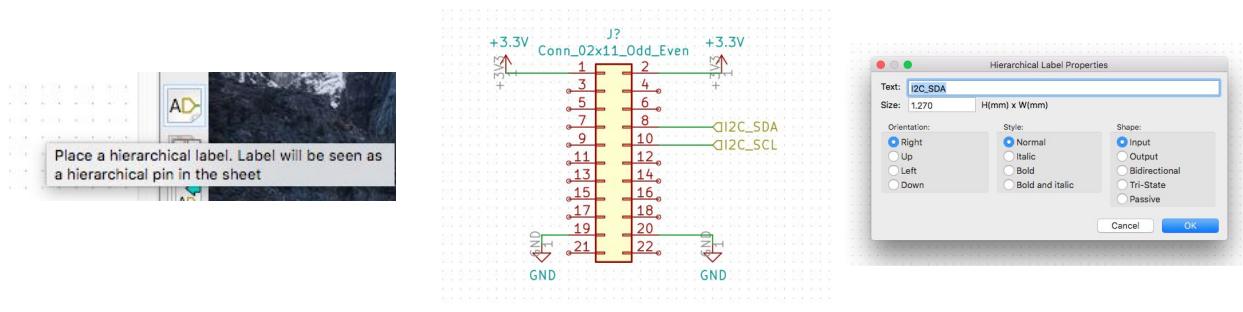
3. Wire them up.



Add Hierarchical Labels

These labels allow you to move signals between sheets.

- We'll connect I2C between the following sheets: *backplane, mcu, sensor*

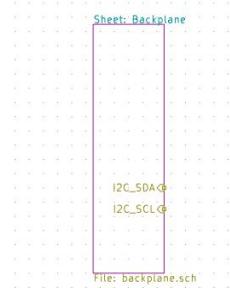


Connecting Hierarchical Pins

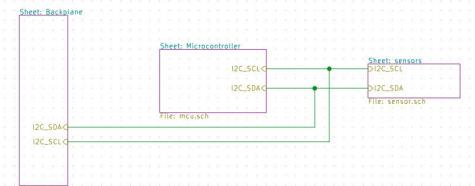
1. Click "Place hierarchical pin..."



2. Click on the backplane sheet symbol twice.



3. Wire up I2C_SCL and I2C_SDA.



Or right click > Import sheet pins



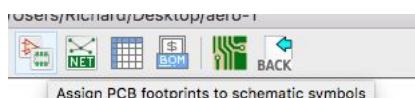
Annotate and Assign Footprints

Annotation assigns the unique reference designators to each component.

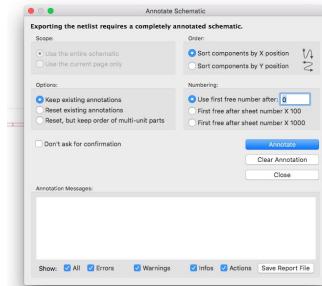
- R1, R2, U1, etc...

Each symbol in the schematic needs a footprint associated with it too.

1. Attempt to assign footprints.



2. You may be asked to annotate. Keep the defaults and continue.



Footprint Selection

The footprint selection for each component is important.

- *Resistors and capacitors*
 - 0402 is the smallest that's easily placed by hand
 - 0603 is larger and easier
 - Stick to a single size for all passive components if possible.
- *Chips*
 - Avoid BGA. They are difficult to place, hard to inspect, and are often too small to route on inexpensive PCB processes.
 - Wider pitch is larger but easier to place, solder, and rework.
- *Connectors and through-hole*
 - Surface mount connectors should be epoxied down.
 - Prefer connectors that are entirely through-hole or include through-hole support pins.

SMD Packages



Electronic Product Workshop | richardarthurs.com

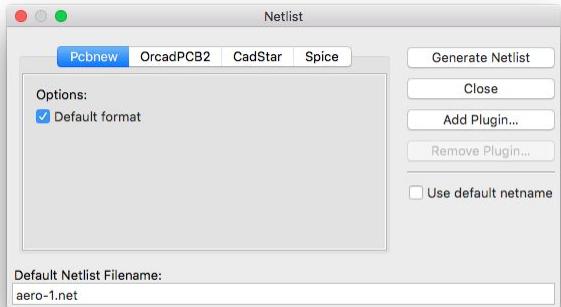
52

[bga-and-qfn-failure-mitigation-underfilling-edge-bonds-and-corner-staking-physics-of-failure-package.jpg](#)

Export Netlist

The netlist is the description of pin-to-pin connections.

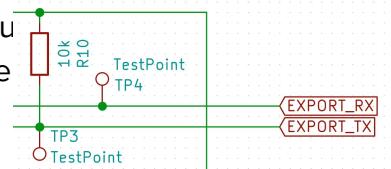
- The netlist is loaded into PCBNew
- Accept if prompted to overwrite th



Test Points

Test points should be inserted on signals you may want to probe.

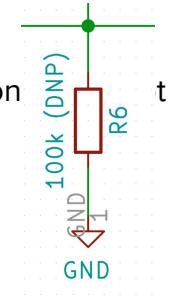
- List conversion factors on the PCB.
 - Ex: 150 mV per mA
- For interfaces (ex: SPI), put a 0.1" header instead of mu
- A through-hole is much easier to probe than a surface
- Test points should be labelled on the schematic.
- Bonus: line up all test points on the board.
 - This is difficult to accomplish, but it's nice!



DNP's

DNP: do not populate.

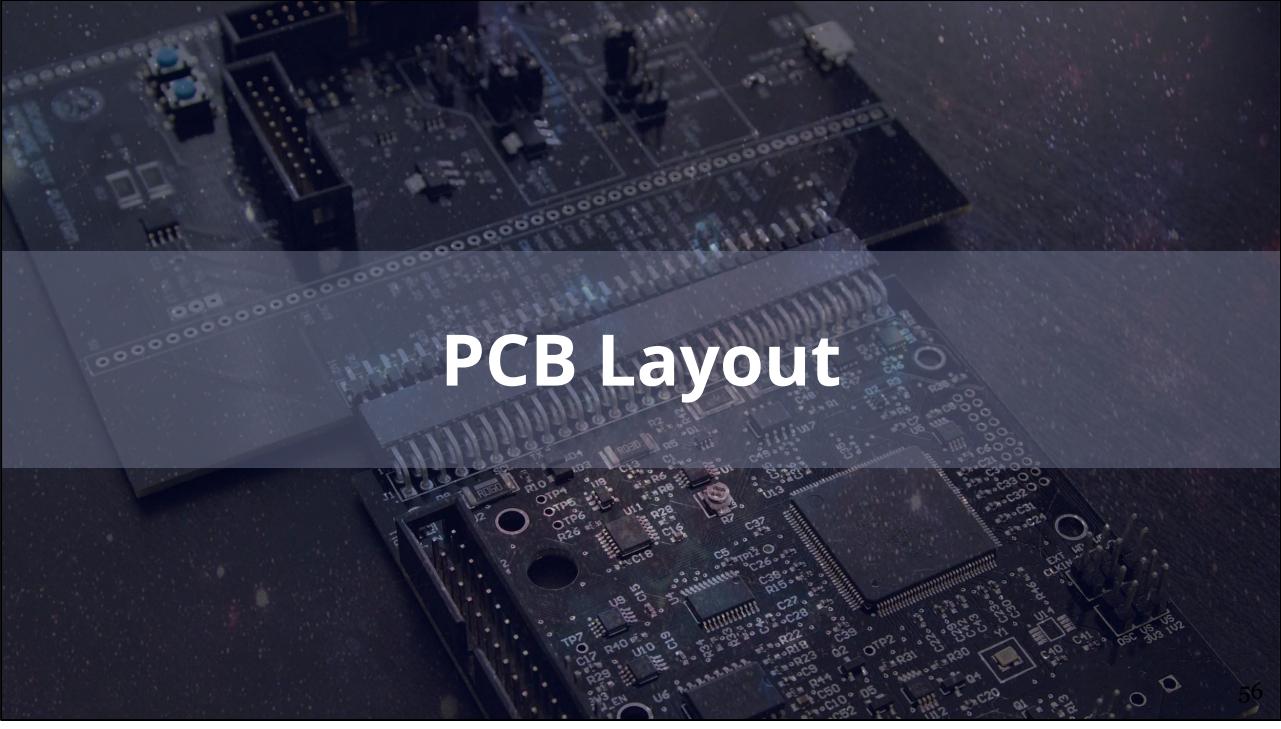
- Sometimes you have components that are just for experimentation ordinarily used.
 - 0Ω jumper resistors for configuration
 - Experimental chips
 - Series/parallel components to change the value of something
 - Things you're not sure about



RECOMMENDATION

Indicate the DNP with a * in silkscreen.





PCB Layout

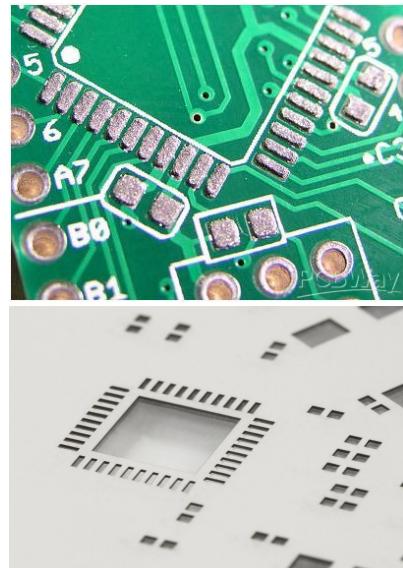
PCB Layout

We are laying out how the PCB will actually look.

1. Decide on a PCB process
2. Set up design rules
3. Create the board outline
4. Add plane layers
5. Import the netlist and lay out the parts.
6. Draw traces
7. Add silkscreen
8. Inspect and review the board

PCB Layers

- **Solder mask:** coloured insulating material
- **Copper:** makes electrical connections
- **Solder paste:** used to make stencil holes
- **Silkscreen:** text



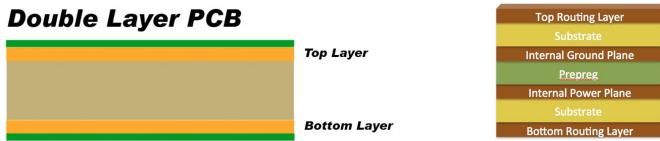
Electronic Product Workshop | richardarthurs.com

58

[6365914697701680086648.jpg](#)
[SS Stencil_1_Cropped_1200_x_800.jpg](#)

Picking a PCB Process

- Generally the selection is between 2 or 4 layers.
 - Two layer is cheaper.
 - Much easier to lay out if it's your first time.
 - Four layer is good for more complex designs.
 - Select 4 layer if you have a larger microcontroller (64+ pins), high speed signals (>20 MHz).
 - Four layer can be easier if you have many parts and are in a rush.
- Decide on a PCB manufacturer early on.
 - You need their constraints (design rules) set before beginning to lay out the board.
 - See backup slides for standard settings.



59

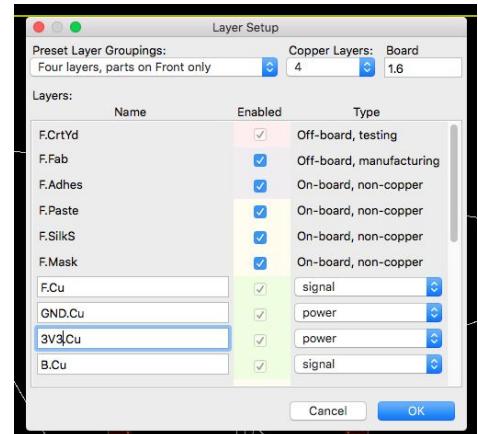
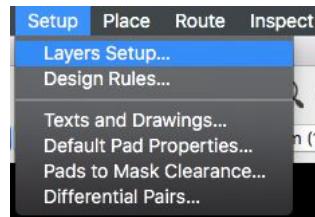
Images:

<http://www.circuitbasics.com/make-custom-pcb/>

http://www.bitweenie.com/wp-content/uploads/2013/02/PCB_Fabrication.png

Setting up Layers

- Based off "Four layers, parts on Front only preset".
 - Named the middle layers **3V3.Cu** and **GND.Cu**
 - These are the power and ground planes
 - Scrolled down and enabled **B.SilkS**, **B.Mask**
 - Bottom silkscreen (text)
 - Bottom solder mask (colour)



Steps 1 and 2: DRC

Design Rules: constraints placed on the PCB layout by the manufacturer. They are minimum/maximum sizes and spacings for traces, holes, masks, and more.

1. Find the design rules for your PCB house.
2. Open *PCBnew* and set the design rules.

Important Design Rules

PCBWay Design Rules

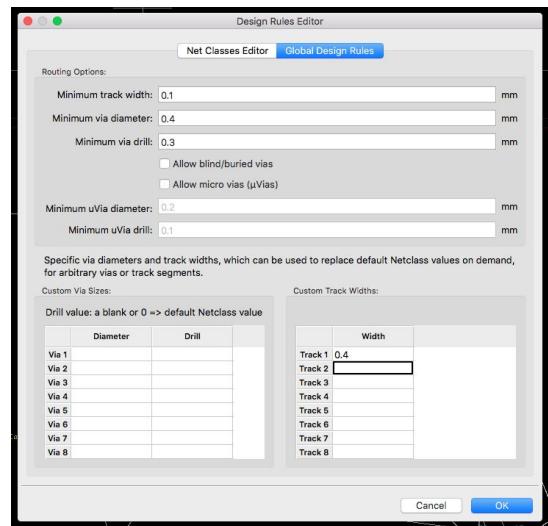
Min Trace		0.1mm/4mil	Min manufacturable trace is 4mil(0.1mm), strongly suggest to design trace above 6mil(0.15mm) to save cost.
Min Spacing			Min manufacturable spacing is 4mil(0.1mm), strongly suggest to design spacing above 6mil(0.15mm) to save cost.
Min Width of Annular Ring		0.15mm(6mil)	For pads with vias in the middle, Min width for Annular Ring is 0.15mm(6mil).
Finished Hole Diameter (CNC)		0.2mm-6.2mm	The finished hole diameter will be smaller than size of drill bits because of copper plating in the hole barrels
Minimum Character Height (Legend)	-	0.8mm	Characters of less than 0.8mm high will be too small to be recognizable.

Set Design Rules

Setup > Design Rules...

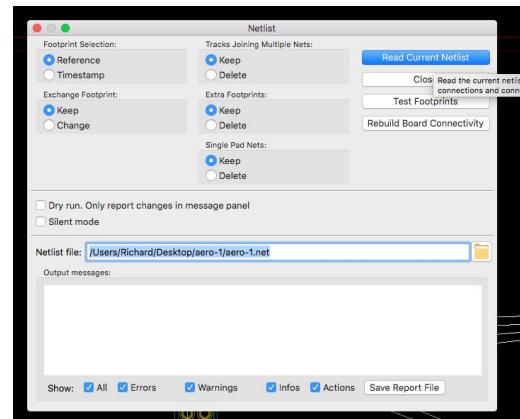
0.25 mm is a good signal trace width.

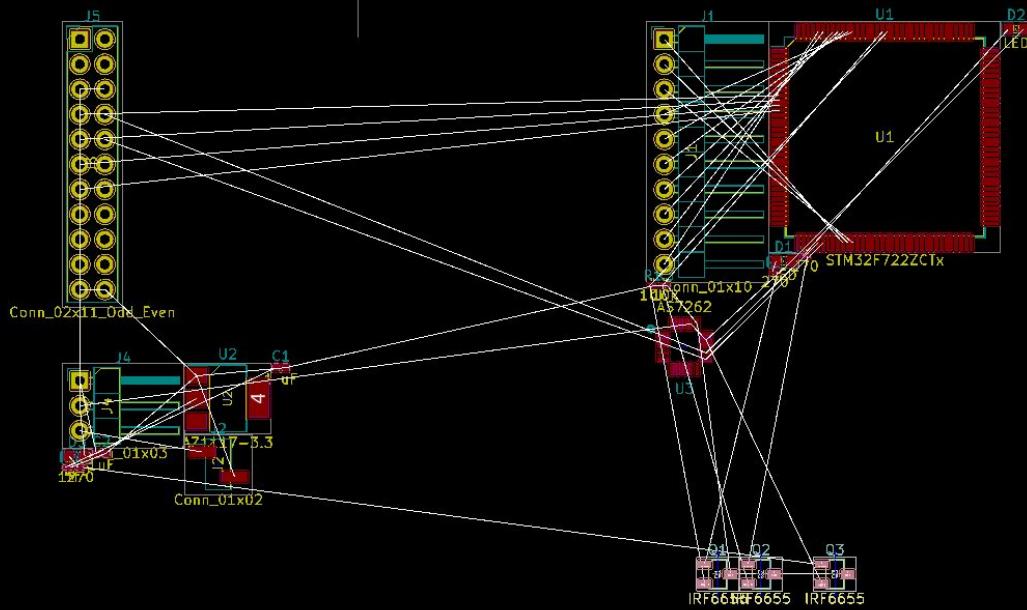
- Use thicker traces for power if high currents are required.
 - You can set certain nets to be larger widths automatically.
 - You can find calculators online to determine the width.



Read Netlist

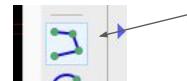
Reading the netlist will bring in the components and connections from the schematic.



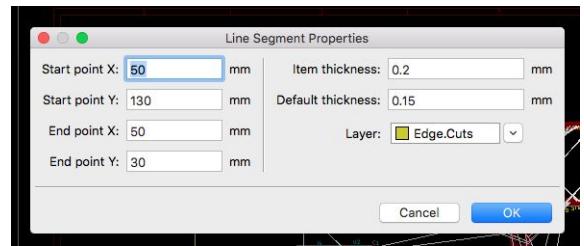


Define Board Outline

Edge.Cuts is the board outline layer.



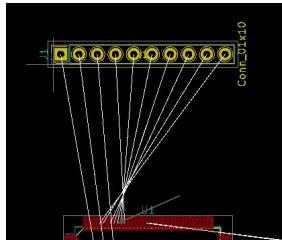
- Add lines, set their start/end points, and set the layer to Edge.Cuts.



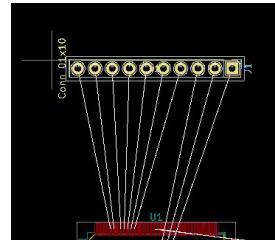
Place Components

Start with a rough placement of components. The more time you spend doing this, the easier the layout will be.

- Drag parts around into regions.
 - Group by functionality.
 - Rotate components to minimize crossovers.



Bad: crossover

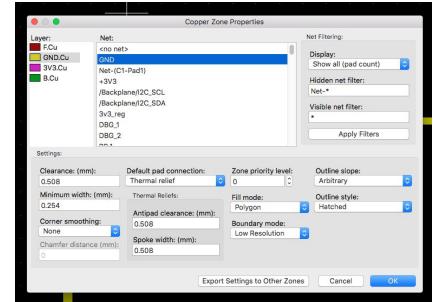


Good: parallel

Add Ground and Power Planes

Place > Zone

- Planes are large filled areas that connect to power or ground.
 - Four layer PCBs usually have a power and a GND plane.
 - Planes significantly reduce the amount of routing that needs to be done.
 - They can also reduce electrical noise.



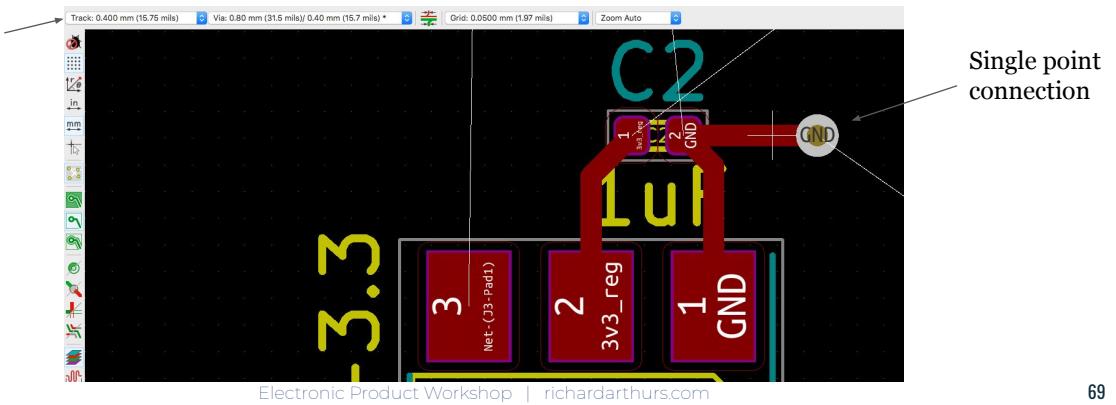
Add planes (zones) for GND and 3V3.Cu Nets/Layers

- Zones should not extend all the way to the edge of the board.

Routing Power Sections

We'll use the 0.4 mm trace for power regulator sections.

- Use a single point connection into the ground plane.



Vias

Vias allow a signal to jump between layers.

- In general, minimize the number of vias.
- Don't place vias in pads (usually).

When already in the Wire tool, press V to create a via.

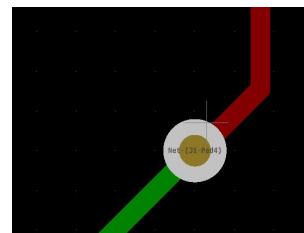
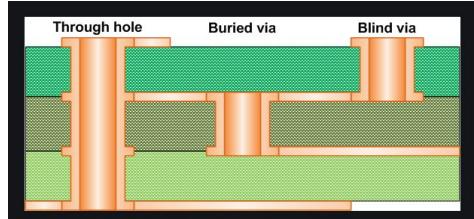
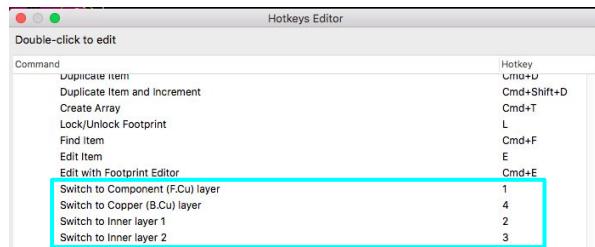


Image: [170420307.png](#)

Routing Shortcuts

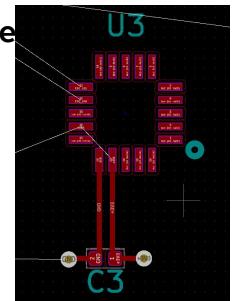
- **I** button will select an entire trace
- **Delete** button will delete a trace that is selected
- **V** button will add a via
- Change the **1-4** buttons to switch between layers
 - Preferences > Hotkeys > Edit Hotkeys



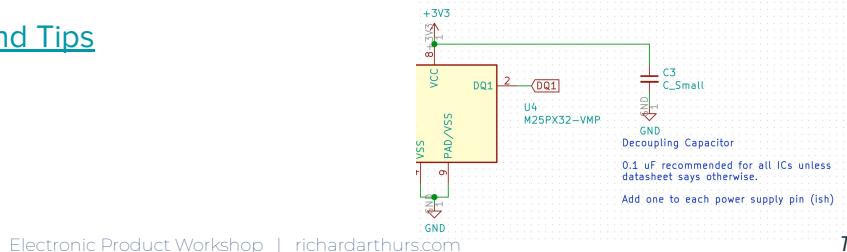
Decoupling Capacitors

Decoupling capacitors isolate chip power supplies from high frequency noise.

- 0.1 μF (typical) between chip power and GND
- Add one for each power supply pin
 - Pin 12 in the picture should have a decoupling capacitor too (3v3)
 - In parallel with the load
- Place close to chip power lines
 - Connect GND directly, then use a via into the ground plane

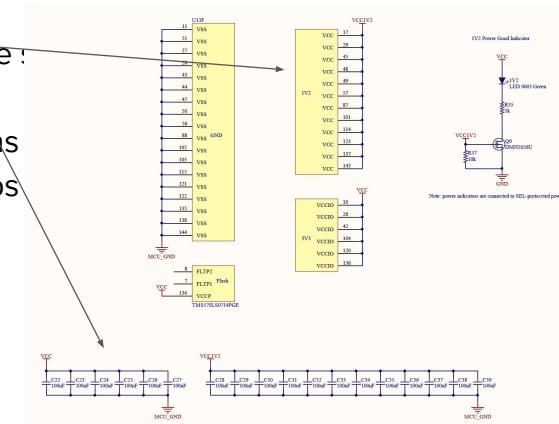


Decoupling vs. Bypass and Tips



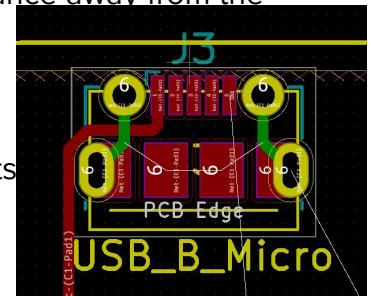
Decoupling for Many Power Supplies

1. Connect supply directly to power nets
 2. Place a bunch of decoupling caps on the
 3. Place the chip on the board.
 4. Place the decoupling caps on power pins
 5. Delete any unnecessary decoupling caps
(close).



Connector Placement

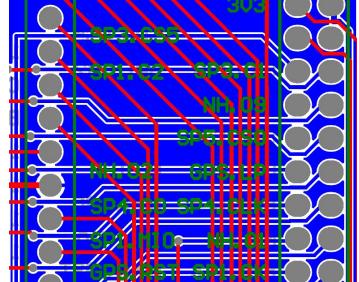
- Make sure connectors are placed an appropriate distance away from the board edge.
- Make sure connectors aren't flipped.
- Don't place connectors too close to each other.
- Give lots of space around connectors. The plastic parts



Backwards USB connector

Layout Recommendations

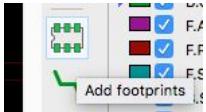
- In general, keep horizontal traces on one layer, vertical on another.
 - This makes it easier to cross over.
- L-shaped traces are good.
 - Go as far East/West, then as far North/South as you can.
- Avoid 90° corners
 - They're ugly and the trace width is 1.41x as wide in the corner (changes the impedance)
- Rip up a section and redo it if it's going badly.
- Look at the 3D viewer to get an idea of how the components are spaced.



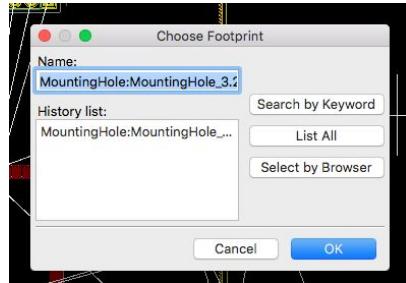
Add Mounting Holes

Mounting holes are added using footprints.

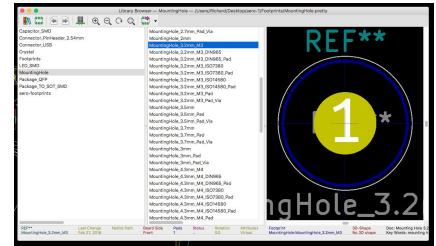
1. Add Footprints



2. Select by Browser



3. Pick library and hole size.



More Layout Recommendations

- If you have time, lay out the board twice.
 - The second version will be better (cleaner, fewer vias, etc.)
- Once you're done laying out the board, don't look at it for a week. Come back and you will notice things to fix.
- Print out the board on paper to get a sense of scale.
 - Useful for checking footprints if you have components already.
- Keep all of the power components in one area.
- Fiddle with the grid setting as you are placing components initially.
- Line things up nicely in rows or columns.
 - This is great to do with indicator LEDs.
- Rotate and move label text so it's all facing one direction.
- Add a date to the board.

More Layout Recommendations

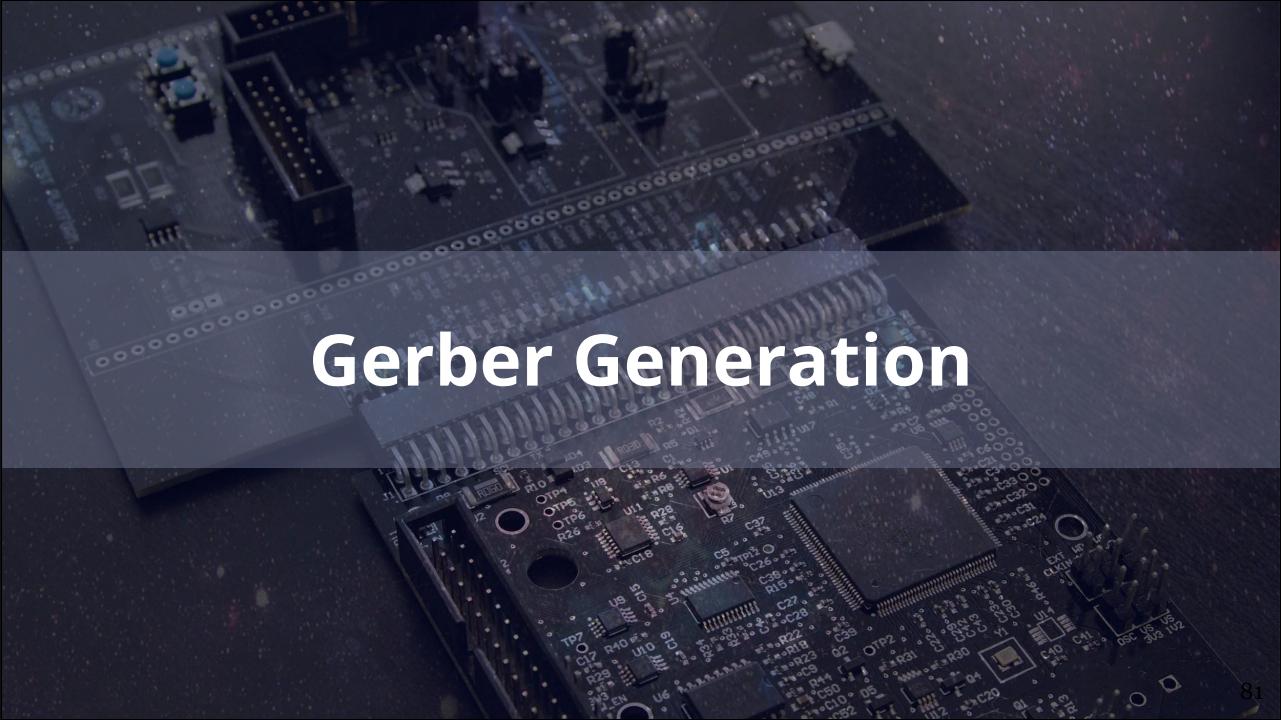
- Label everything possible. Labelling is annoying but it's worth it.
 - +/- indicators
 - All pins on connectors.
 - Label on bottom and top. Bottom labels are useful when there's no space on the top.
 - Do you really want to open up the board file while you're debugging?
- Put the month and year on the board.
- Look for silkscreen on top of pads. You don't want any of this!
- Put lines describing jumper settings.

KiCAD Tips

- For 4-layer boards, set the keys 1-4 to switch between these layers.
 - This makes it *much* easier to jump between layers when making vias.
- Open up the board and schematic at the same time. Clicking a component on the board will open up the corresponding schematic section.
- Use a mouse.

Review Checklist

1. Component spacing is reasonable.
 - a. Set the grid to 1 mm and look again. Remember how small 1 mm is! Everything looks larger and widely spaced on screen.
2. Mounting holes and other mechanical considerations such as connector orientation.
3. Labels are present on both sides, and absolutely everything that can be labelled is labelled.
4. Traces are cleanly routed.
5. DNPs are clearly marked.



Gerber Generation

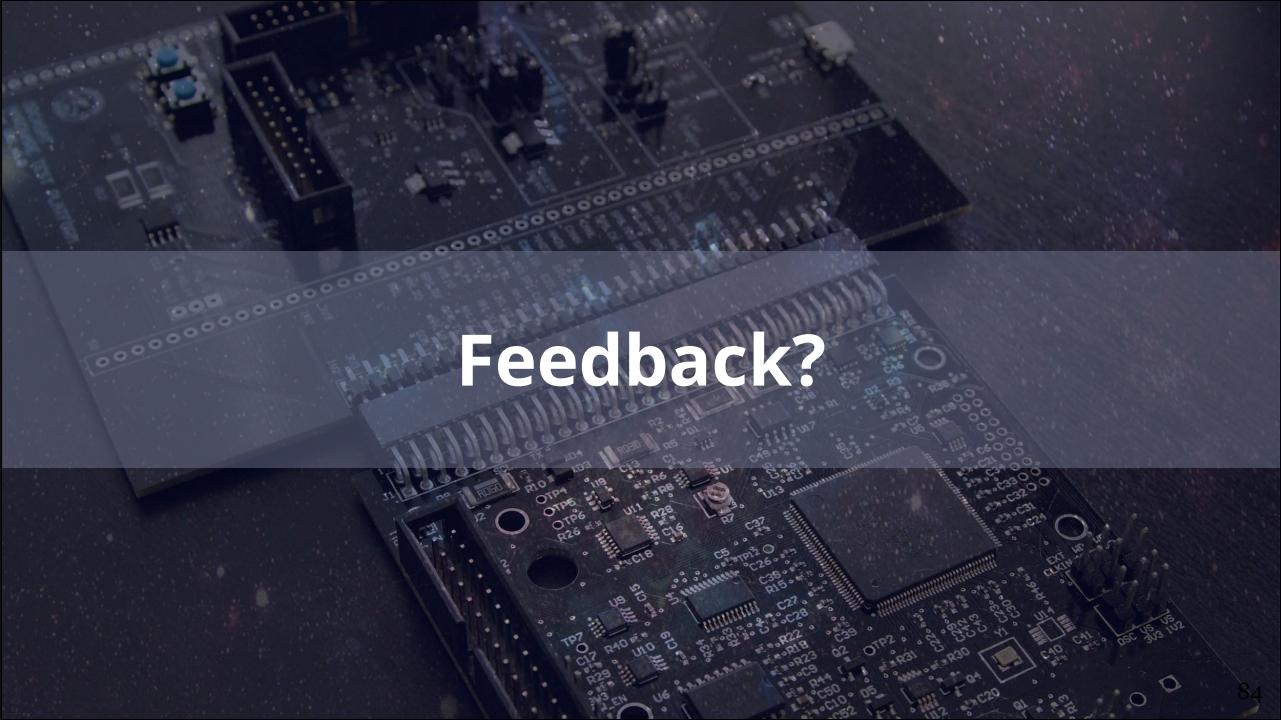
Gerber Generation

We are generating the files that the board house will use to make the board.

1. Double check that DRC is set correctly.
2. Run DRC, ensure there are no errors.
3. Zero unrouted traces.
4. Ensure planes are filled.
5. Set up gerber generation.
6. Export and inspect the gerbers.

Gerber Inspection

- Open the gerbers in a gerber viewer app.
 - Cycle through the layers quickly and check that things line up.
 - It's much easier to see silkscreen on top of pads with a dedicated gerber viewer.
 - Look at layers individually.
 - Right angle traces will be obvious when doing this.
- The board house will also review your gerbers against their design rules.
 - You can upload your gerbers as if doing a PCB order, get their review, then cancel the order.



Feedback?



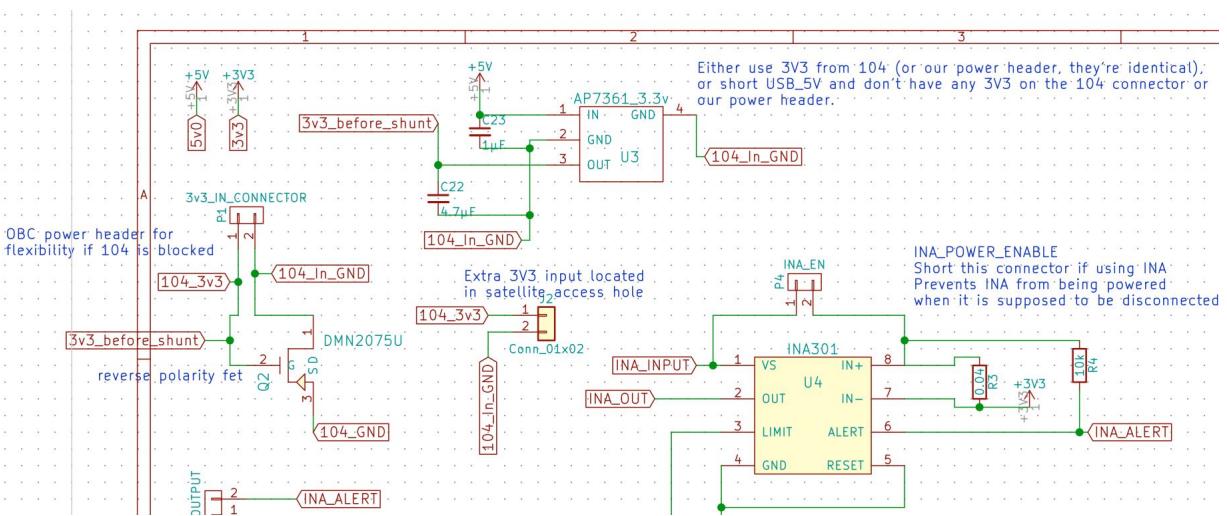
Case Studies

Case Studies

Let's look at some layouts and schematics and review them.

- Review against the pointers in this presentation.
- Be critical! That's the point!
- Come up with some good and bad aspects.

Case Study: OBC v0.5 Schematic



Case Study: OBC v0.5

Sheet: PC104_Connector
File: PC104_Connector.sch

Sheet: SPI
File: SPI.sch

Sheet: MCU_Support_Reset
File: MCU_Support_Reset.sch

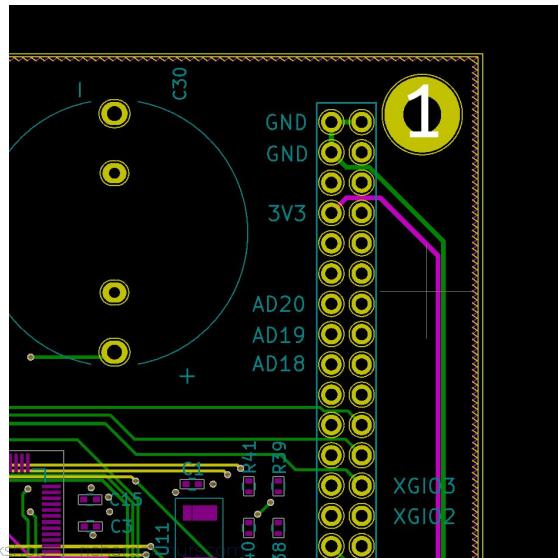
Clock, Reset and Watchdog, JTAG

Sheet: ADC
File: ADC.sch

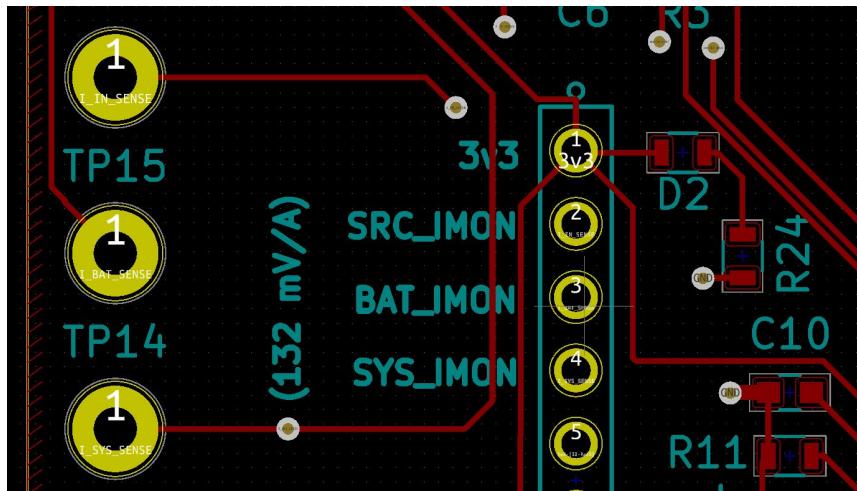
Sheet: NHEI
File: NHEI.sch

Sheet: UART
File: UART.sch

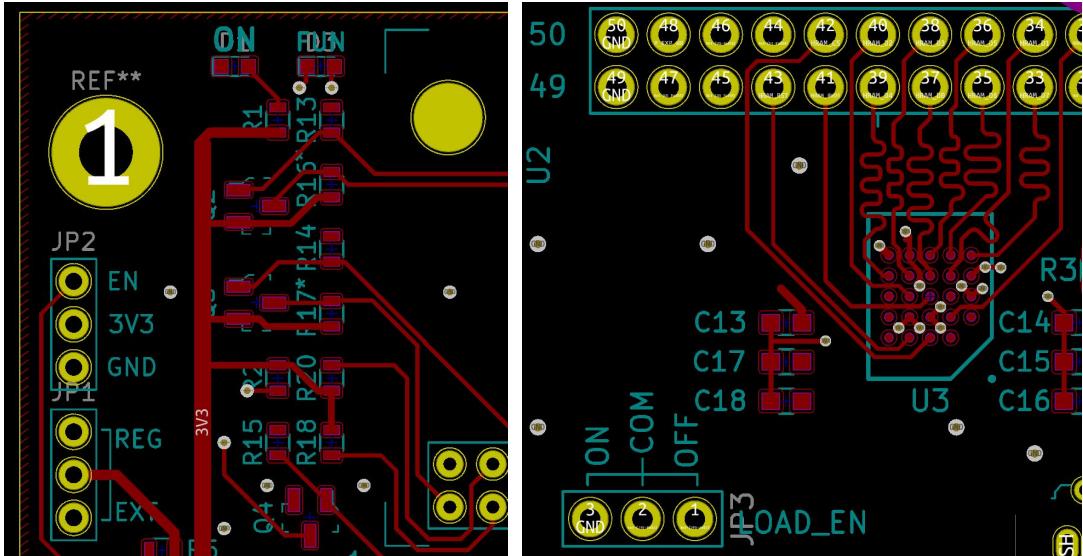
Electronic Product Work



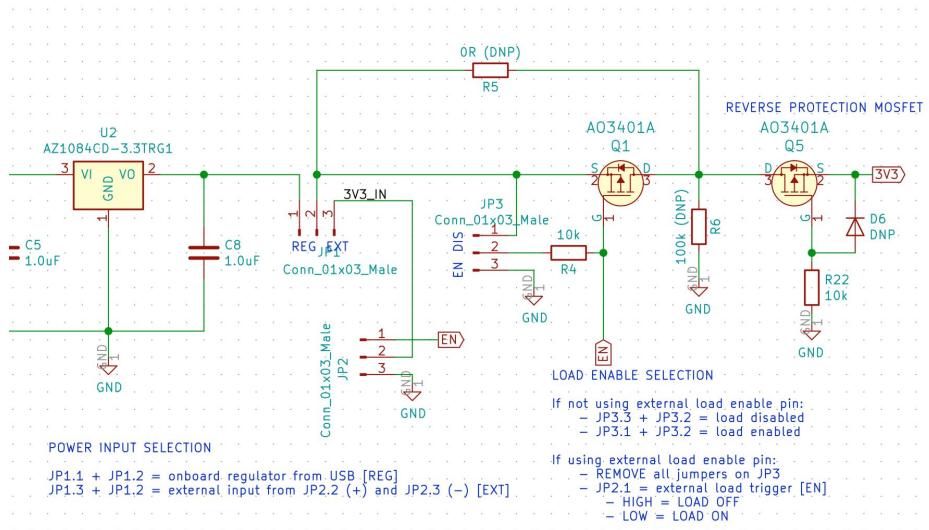
SFUSat BMSS

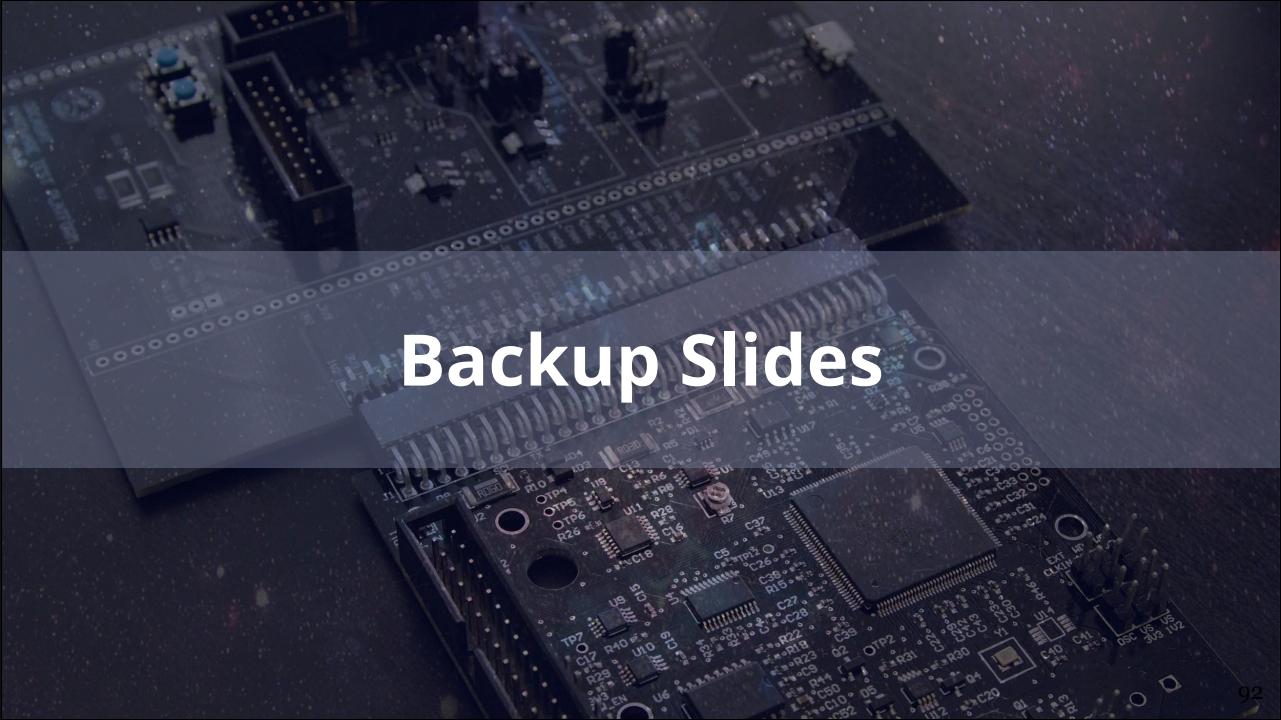


AROS-1



AROS-1





Backup Slides

Backup: Copying Default Libraries

Copied libs from:

Macintosh HD/Library/Application Support/kicad/library

Backup: Standard PCB Settings

Standard settings for a 2 or 4 layer board from PCBWay or any inexpensive manufacturer:

- Thickness: *1.6 mm (RF boards may be an exception)*
- Trace/space: *6 mil*
- Min hole size: *0.3 mm*
- Solder mask: *yes, and matte black looks coolest*
- Gold fingers: *no*
- Finish: *HASL (hot air surface levelling) with lead*
 - This should match your solder paste selection. Lead-free solder is difficult to work with.
 - ENIG: gold plated. This is fine regardless of solder choice. Sometimes they will give you ENIG for free if there is spare room in the production batch.
- 1 Oz copper