Electronic Hardware Design

Josh Johnson

BSides Canberra 2021

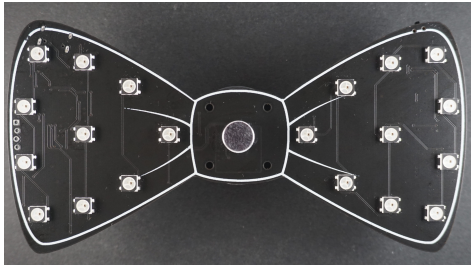
Overview

Workflow

- Requirements
- Locating Resources / Reference Designs
- Schematic Capture
- Parts of a PCB
- Printed Circuit Board Layout
- Ordering PCBs / Components
- Assembly

Examples - Sound Reactive LEDs

- Breadboard
- Through Hole PCB
- Fully Integrated Design



Requirements

Depending on the objective of your design, there are a number of requirements / considerations which need to be taken into account as they will greatly impact your finished product.

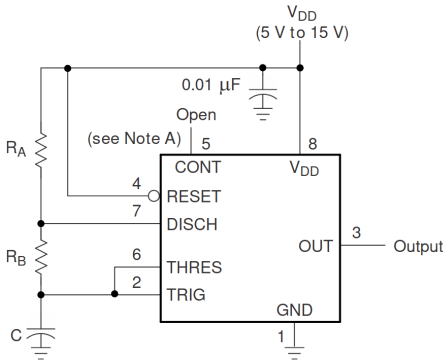
- Physical dimensions / form factor
- Interacting with design (buttons, sensors, displays, LEDs)
- Connectivity (USB, Bluetooth, WiFi)
- Programming (bootloader, on board / external programmer, Arduino / C / Python)
- Power supply / battery
- Complexity (component size, COTS modules, part count)
- Assembly (through hole, surface mount, single vs double sided)
- Cost (PCB features, component selection)

Locating Resources / Reference Designs

With requirements in hand, we can now start searching for suitable parts. Evalulation boards are a great place to start, as they provide working hardware, documentation, and example code if applicable.

- Adafruit / Sparkfun
- Open Source Hardware (GitHub, Hackaday.io)
- Manufacturer Evaluation Boards
- Application Notes
- Datasheet

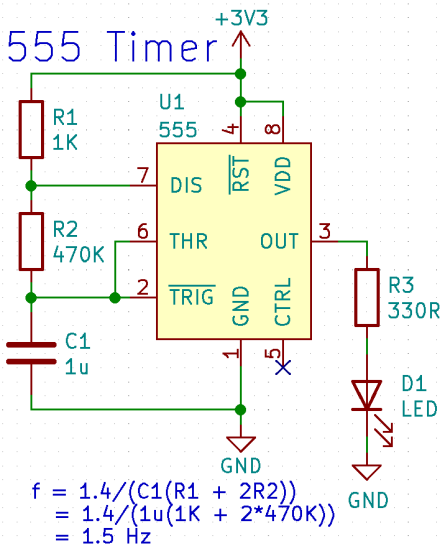
All of the above will provide you with schematics and part numbers to kick start your design.



Schematic Capture

What: Abstract representation of circuit / components.
Why: Communicates purpose and documents design.
How:

- Symbol creation
- Symbol placement
- Connect everything with wires
- Add notes to your design
- Run electrical rules checks (ERC)
- Footprint association (may be done in symbol placement)
- Bill of Materials (BOM) generation



Parts of a PCB

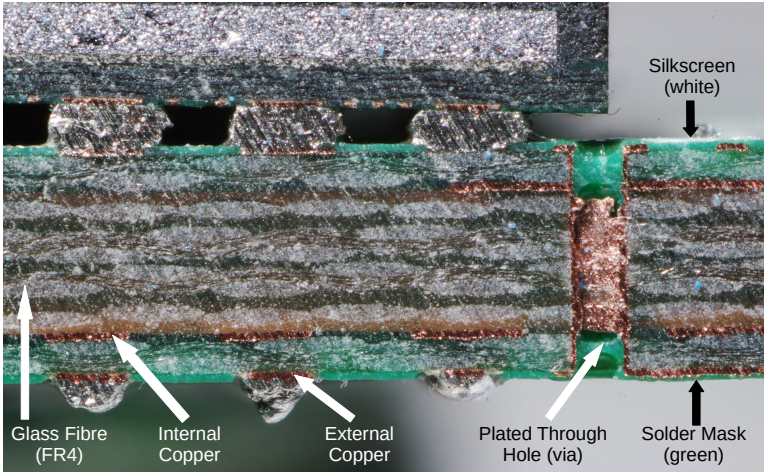
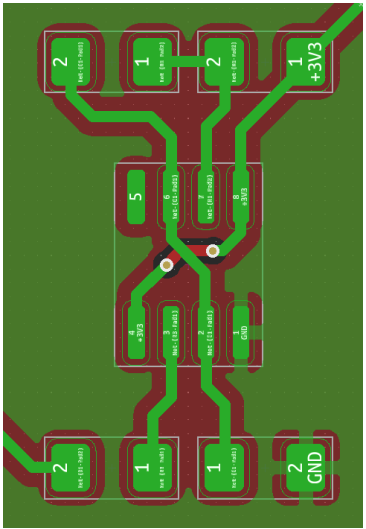


Image: Rainer Knäpper

PCB Layout

What: Physical representation of circuit / components.
Why: Ensures electrical and mechanical function.
How:

- Configure design rules per manufacturer guidelines
- Draw board outline
- Place connectors and mounting holes
- Place electronic components
- Route critical nets, power, then everything else
- Run design rule checks (DRC)
- Add decorative features
- Review in 3D viewer / check mechanical fit
- Export Gerbers



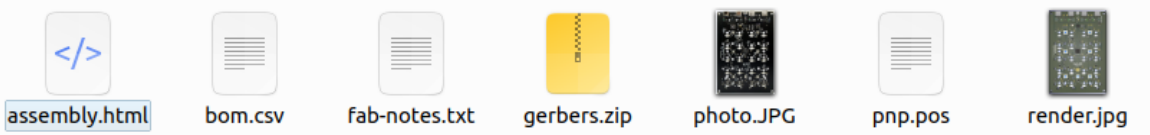
Ordering PCBs / Components

PCB

- Check Gerbers were exported correctly
- Zip up Gerbers
- Upload to manufacturer
- Choose PCB options (quantity, colour, surface finish)

Components

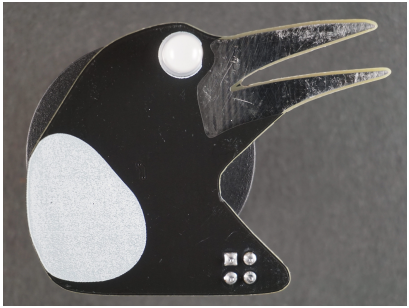
- Export BOM from Schematic
- Upload to supplier
- Confirm package size is correct
- Confirm availability before ordering boards!

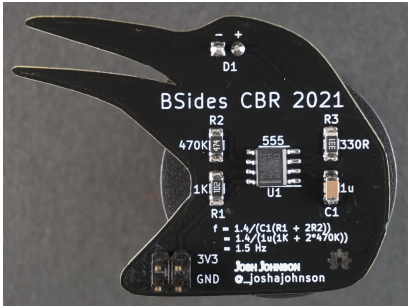


Assembly

You have all the parts, but how to put them together?

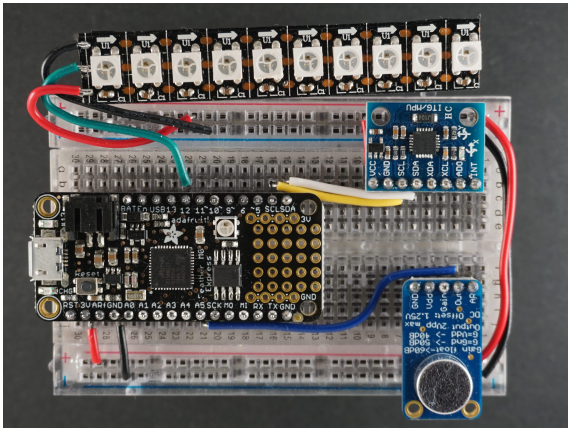
- Development Boards - point to point or wired through a breadboard
- Through Hole - solder with a soldering iron
- Surface Mount - reflow with a stencil and solder paste, or solder with an iron
- All - ask your PCB manufacturer to do it for you (even in small volumes)





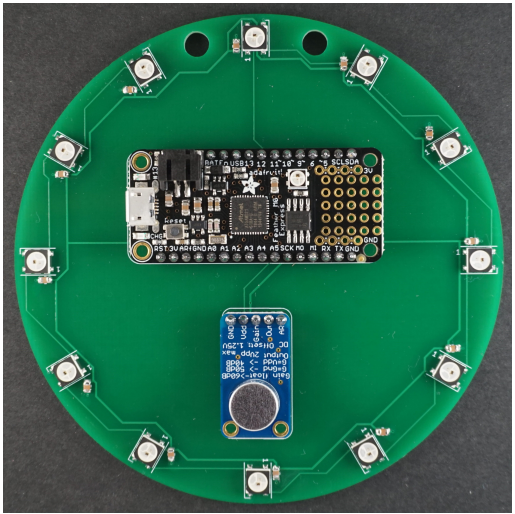
Example - Sound Reactive LEDs - Breadboard Prototype

- Ideal for: Confirming compenents will work as required for your project, or one off builds.
- Wired components following a guide from Adafruit.
- Adafruit M0 Feather, Microphone, WS2812 (Neopixel) strip, MPU6050 IMU.
- If things don't work (like the IMU), easy to remove.
- Pros: Great for prototyping.
- Cons: Challenging to meet mechanical requirements, fiddly to assemble.



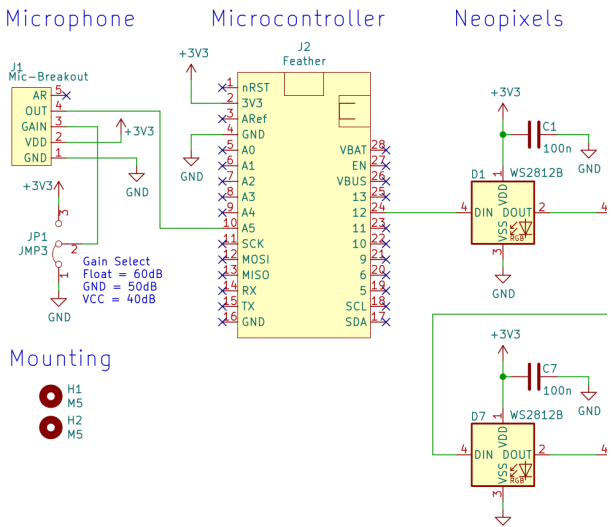
Example - Sound Reactive LEDs - Through Hole PCB

- Ideal for: Non space constrained / low volume designs / prototyping.
- Replaced wires on breadboard with traces on PCB.
- Identical wiring and firmware to breadboard prototype.
- Four unique parts, low risk of PCB design errors.
- Pros: Quick and easy to assemble and bring up, low part count, low complexity.
- Cons: Expensive to manufacture in volume, challenging to make small.



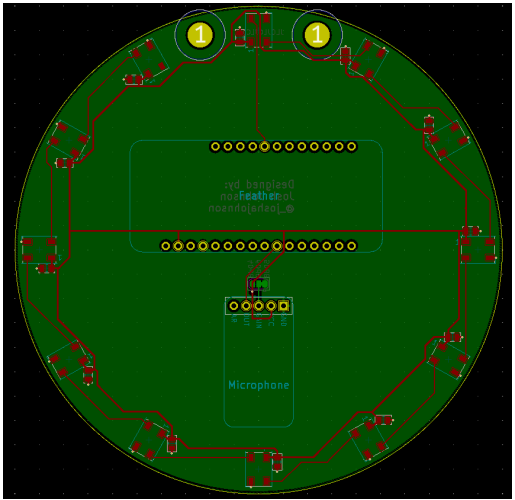
Through Hole PCB - Schematic

- Replicating breadboard wiring in schematic.
- All but microphone in KiCad Library, can use a simple 1x5 connector.
- Added a PCB jumper to change gain if required.
- Known good circuit, so low risk of issues.
- Four unique parts, 26 parts total.



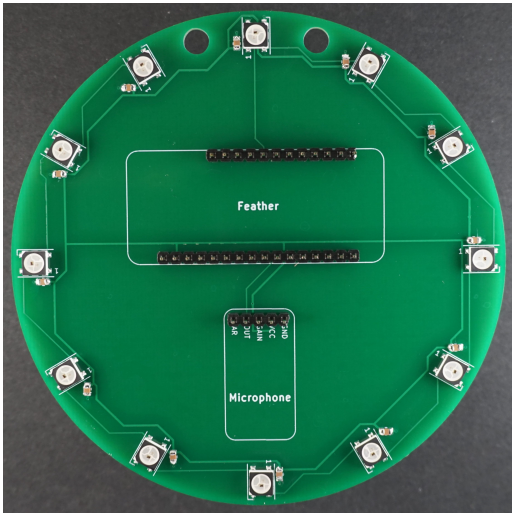
Through Hole PCB - Layout

- Two layer board, ground pour on bottom, signals and power on top.
- Through hole or large surface mount devices (SMD).
- Simple board outline drawn using the circle tool.
- Mounting points added to attach lanyard to.



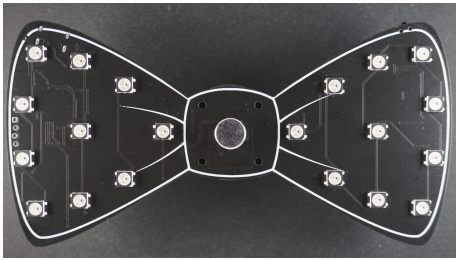
Through Hole PCB - Assembly

- Hand assembled in an hour.
- Soldering iron only, no hot air gun.
- Firmware was loaded over USB just like breadboard.
- One mistake was made - microphone outline is 180° out and fixed after this photo.



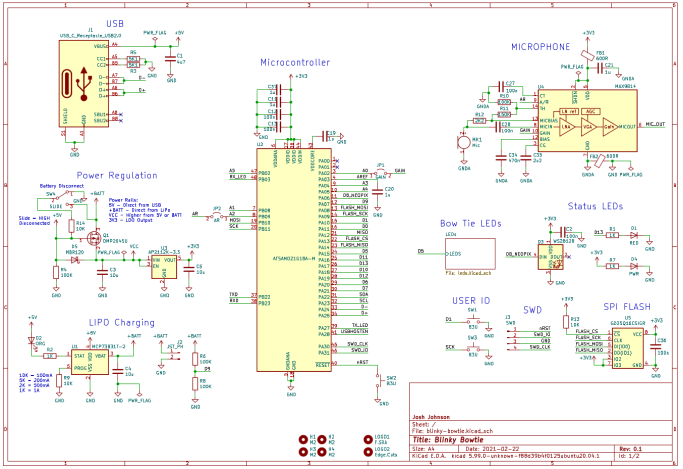
Example - Sound Reactive LEDs - Integrated Design

- Ideal for: Mechanically constrained / high volume designs.
- Ground up schematic capture and layout.
- If designed correctly, compatiable with prototype firmware.
- Higher risk of design errors, especially if designed after midnight...
- Pros: Guaranteed to suit mechanical requirements, easier / cheaper to produce in volume.
- Cons: More time required to design, assemble and bring up prototypes, harder to troubleshoot issues.



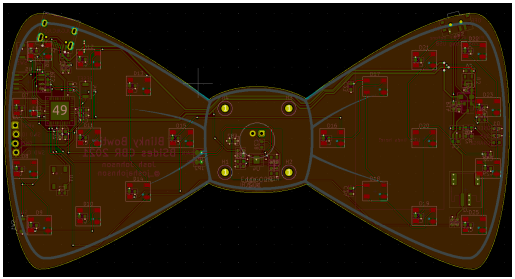
Integrated Design - Schematic

- Replicating Feather M0 and microphone breakout in addition to connections.
- Numerous symbols had to be designed from scratch.
- Numerous new features including power control, buttons.
- Risk of transcription errors from module schematics and higher chance of miswiring components.
- 30 unique parts, 92 parts total.



Integrated Design - Layout

- Two layer board, signals and power pour on bottom, ground pour on top.
- Primarily 0603 passives and leadless IC's.
- Double sided assembly - LEDs and microphone on front, remainder on back.
- Complex board outline and artwork imported from Inkscape.
- Fully 3D modeled, allowing design of the mounting mechanism and identification of part interference.



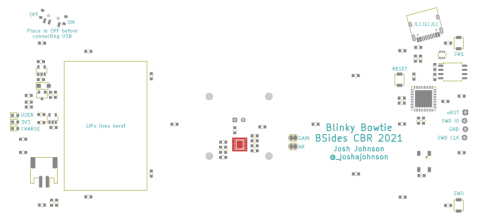
Integrated Design - Assembly

- Double sided reflow assembly using a solder paste stencil.
- Required reflow oven, stencil, solder paste, hot air, microscope, and soldering iron.
- Used the "Interactive HTML BOM" tool to help identify part locations.
- Bootloader flashed using external programmer, then firmware via USB.
- Numerous hardware issues requiring flying wires, along with bootloader issues made initial bringup take three days.
- Feature additions and fine tuning took another day.

Blinky Bowtie Rev...
Josh Johnson 2021-02-...

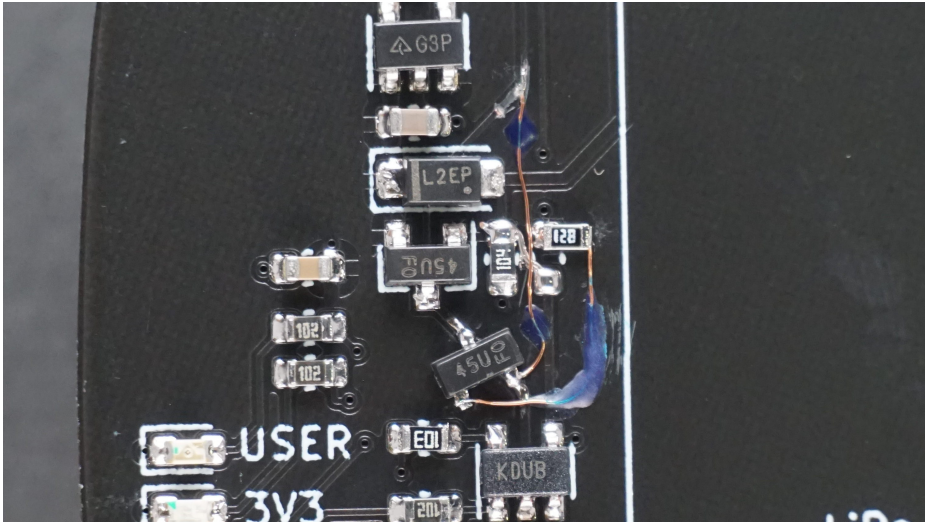
F FB B

17		D3	WS2812B-B	Worldsemi	WS2812B	LED_WS2812B_PLCC4_5.0x5.0mm_P3.2mm	1
18		U3	AP2112K-3.3TRG1	Diodes Incorporated	AP2112K-3.3	SOT-23-5	1
19		U2	ATSAMD21G18A-MF	Microchip Tech	ATSAMD21G18A-M	QFN-48-1EP_7x7mm_P0.5mm_EP5.15x5.15mm	1
20		U5	GD25Q16CSIGR	Gigadevices	W25Q32JVZPIQ	SOP-8_5.28x5.23mm_P1.27mm	1
21		U4	MAX9814ETD+T	Maxim	MAX9814	DFN-14-1EP_3x3mm_P0.4mm_EP1.78x2.35mm	1
22		U1	MCP73831T-2ACI/OT	Microchip Technology	MCP73831T-2	SOT-23-5	1
23		U6	SN74LVC1T45DCKT	Texas Instruments	SN74LVC1T45DCK	SOT-363_SC-70-6	1
24		SW1, SW2, SW3	B3U-1000P	Omron Electronics	B3U	SW_SPST_B3U-1000P	3
25		SW4	PCM12SMTR	C&K	SLIDE	SW_SPDT_PCM12	1

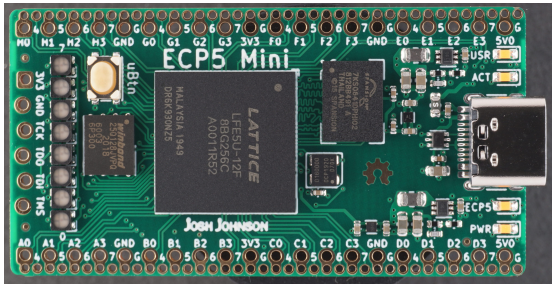


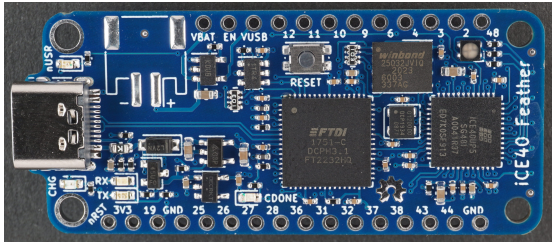
Flying Wire: Microphone Amp to MCU

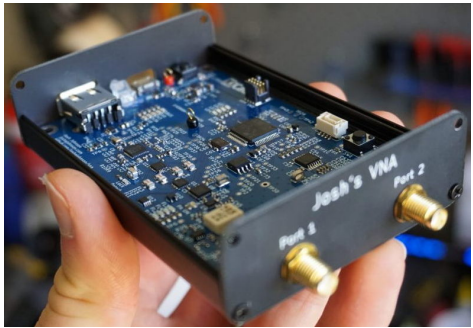
Bodge: Power Selection

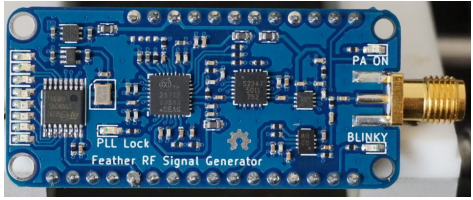


Other Projects









Questions?

Slides and supporting documentation: `github.com/joshajohnson/bsidescbr2021`

Repo contains all KiCad source files, images, and links if you want to learn more.

Please say hello!

Twitter: @_joshajohnson
BSidesCBR Slack: josh
Email: josh@joshajohnson.com