FRITZING FOR CIRCUITS AND PCBS

PAT FLEMING

EMAIL: PATFLEMINGHTC@GMAIL.COM

PRESENTATION: HTTPS://GITHUB.COM/PATFLEMING/PRESENTATIONS/FRITZING-FLASHER_CIRCUIT.ZIP

ACKNOWLEDGEMENTS

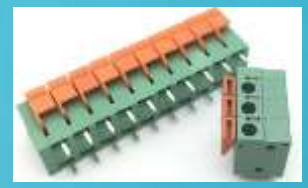
- Robert Heller
  - Developed Model Railroad System (Deepwoods Software)
  - Contributed to Layout Command Control (LLC)
    - Layout device schematics and LLC software
  - Inspiration and working basis of hardware and software for layout automation
- Bob ??
  - Much needed companion for helping with efficiency of this ‘technical’ hobby within a ‘model railroad’ hobby
  - Presentation assistance by proof reading and editing speaker notes
  - Research assistance of software programming, electronics, schematics, etc.
  - Willing to help anyone… ?

AGENDA

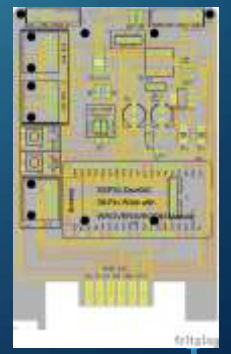
- Why Create Custom PCBs
- What is a PCB?
- What is Fritzing?
- From Design to Production and Test
- Explore Fritzing
- Flasher Circuit Example
- Order a PCB
- Backup

WHY CREATE CUSTOM PCBS

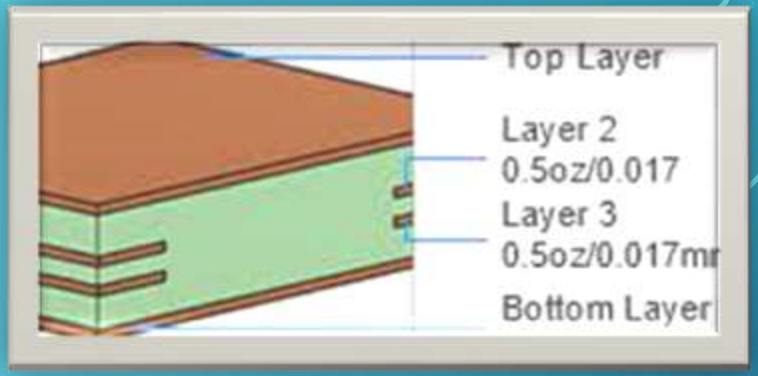
- Circuit not commercially available
- Reduced costs
- Customization needed to align with layout requirements
- Usability enhancements (DIP Switches, Tactile Buttons)
- Add functionality (e.g. Integration with Arduino)
- Custom size and shape
- Make mountable







WHAT IS A PCB



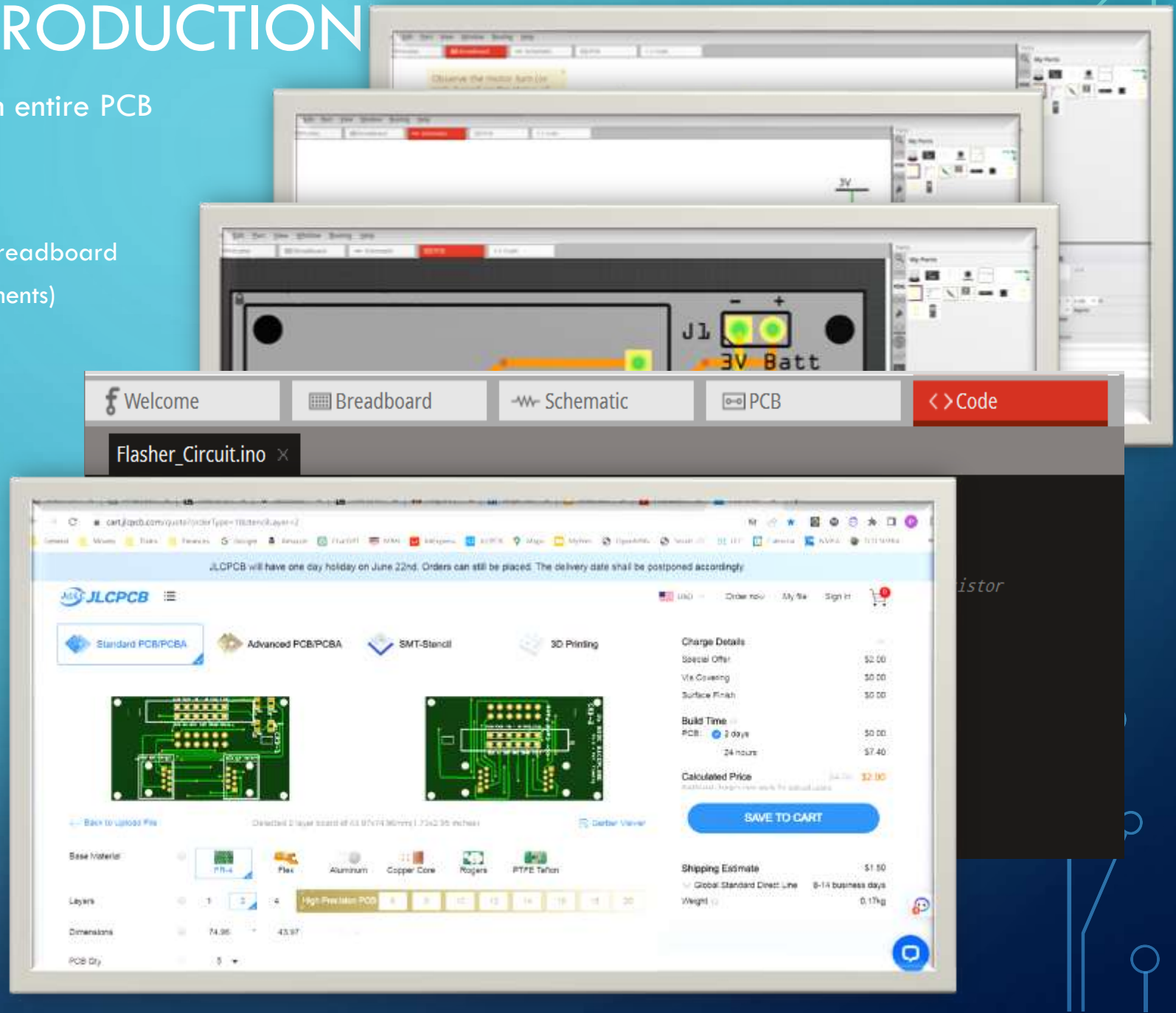
- Printed Circuit Board (PCB)
- Provides electrical conductivity between components via copper ‘traces’
- Can have 1, 2, 4 or more layers of copper
  - Fritzing designs 1 and 2 layer PCBs (2 is most common)
  - JLCPCB fabricates 1 to 20 layers for rigid PCBs
- Can be designed in various sizes and thicknesses
  - MAX Size: 400mm x 500mm (JLCPCB)
  - Thickness: 0.4mm – 2.5mm (JLCPCB)
- Can be produced in various shapes, colors and materials
  - Materials: FR-4 ($, most common), Aluminum ($$), Flex ($$) , Teflon ($$)

WHAT IS FRITZING

- PCB Design Tool and more…
- Open Source Hardware Initiative
- Cost: $9.29 for v1.0.0 (as of June 19, 2023)
  - For years the beta version was free, until v1.0.0 was released
  - Make sure your Windows scale is set to 100% or 200% (Settings -> Systems -> Display -> Scale)
- Runs on Windows, Mac, and Linux (download)
- Developed for hobbyists and amateurs to create electronic hardware
- Useful in creation of PCBs for use with Arduino based projects
- Large user community
- Large number of parts
- Fritzing GitHub Repositories (docs, parts, etc.)
- Wikipedia

FROM DESIGN TO PRODUCTION

- Fritzing provides integrated tools for an entire PCB development process as follows:
- Breadboard View
  - Experimentation of circuit's design on a breadboard
  - Circuit Simulation (limited to basic components)
- Schematic View
  - Capture an existing circuit design
  - Modify existing circuit designs
  - Create a new circuit designs
- PCB View
  - Placement of components on PCB
  - Layout traces between components
  - Design Rules Check (DRC) tool
- Code View
  - Design software for the Arduino
  - Compile/Upload firmware to Arduino
- Production
  - Creation of files for use for production



CIRCUIT DESIGNS
WHICH ONES

- PCB design is required for PCB fabrication.
- Schematic and breadboard designs are optional.
- Schematic design is useful when:
  - There is a need to document the design's circuit as a diagram.
  - Sharing the design with others for validation or open source.
- Breadboard design is useful when:
  - Experimenting with the circuit before fabrication, e.g., trying different component values or types.
  - Validating modifications to existing circuits.
  - Utilizing it as a learning experience for yourself or others.
- Code design within Fritzing is useful when:
  - Shared as an Arduino code example for the circuit

CIRCUIT DESIGNS
ORDER OF DESIGN CREATION FOR PCB FABRICATION

- Fritzing doesn't require all circuit design methods to be implemented
- Start with what you have, if anything
- Create only the designs that are needed
- You can always revisit Fritzing and complete other designs. For example,
  - Create a schematic design of the circuit to share with others
  - Create a breadboard design to test a failing PCB design

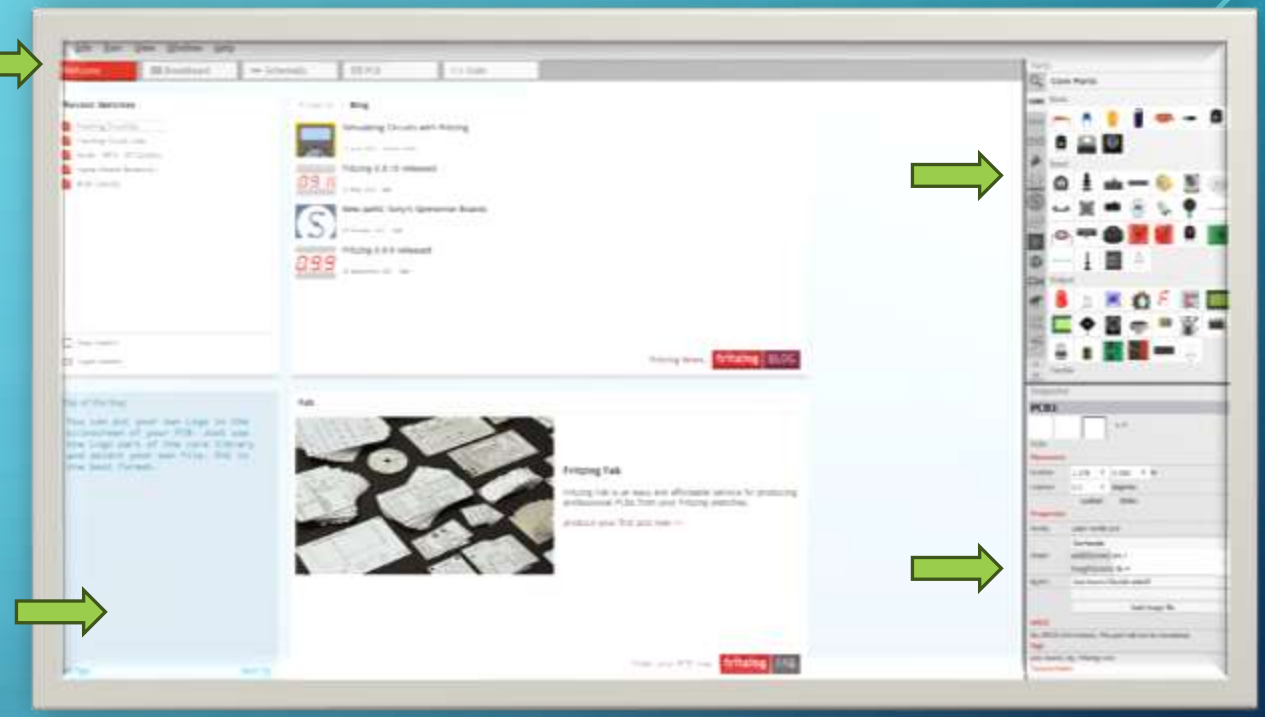
Scenario	Flow of Design Creation	Comments
Only a PCB is needed from a working schematic	PCB	
Modifying existing schematic	Schematic -> Breadboard -> PCB	Modify existing schematic, validate with breadboard, then create PCB
Need to document circuit	Schematic -> PCB	
Creating new design, without sharing	Breadboard -> PCB	
Simple circuit design	PCB	No IC, just connections to terminals and basic components (resistors, capacitors). E.g. LED connection PCB with current limiting resistors.

EXPLORE FRITZING

FRITZING

TABS, PALETTES, PARTS

Tab	Use	Comment
Welcome	Access recent sketches	Recent sketches
Breadboard	Create Breadboard example	Optional, experiment / test design
Schematic	Create Circuit Diagram	Optional, document design
PCB	Create PCB design	Required for PCB fabrication
Code	Develop code for Arduino boards	Use Arduino IDE instead

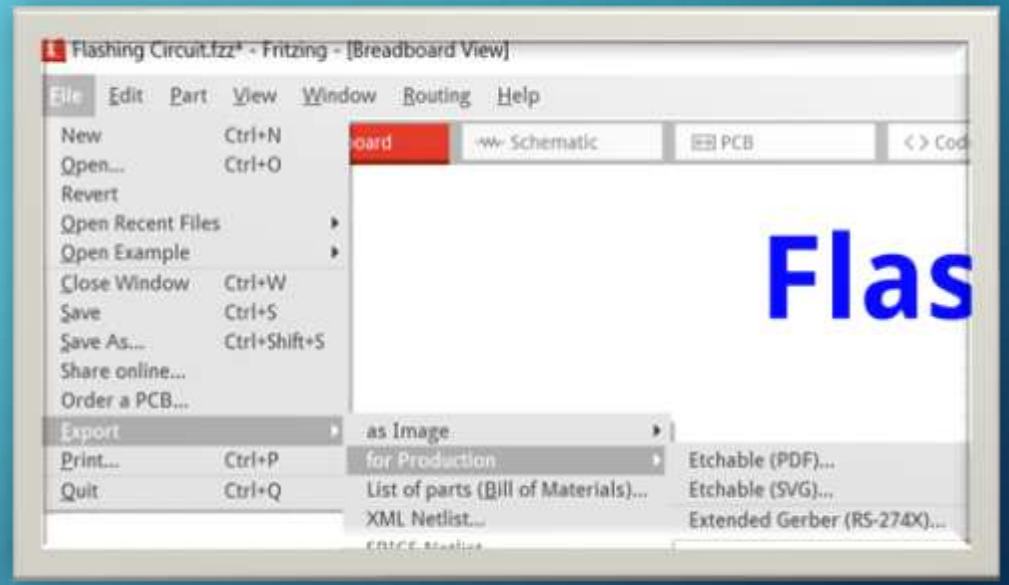


Window Palette	Use	Comment
Main	Circuit design workspace	Edit design999
Parts	Select parts	Parts grouped in bins, search option
Inspector	Review and change part information	Select part variants, change values

FRITZING
FILE MENU

File menu provides basic functions of New, Open and Save

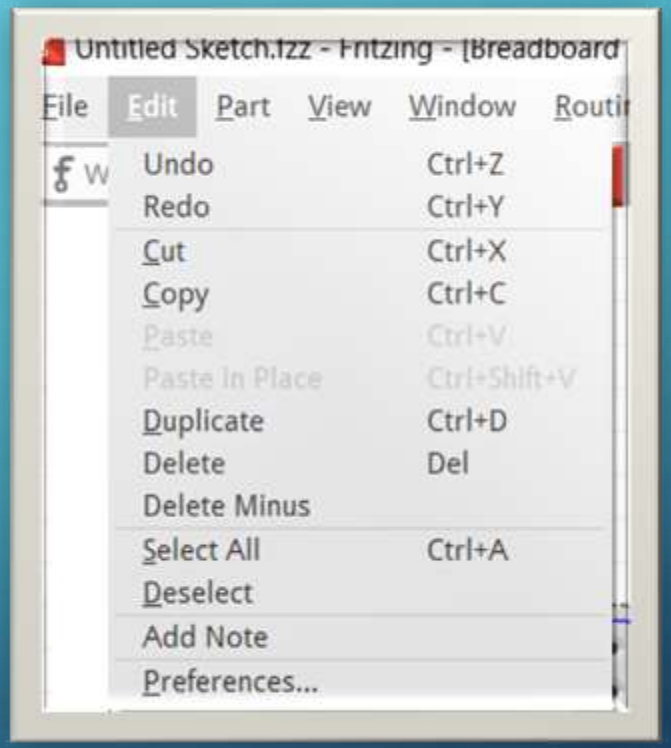
Function Selection	Use	Comment
Revert	Revert back to last saved version	
Order a PCB…	Order PCB from AISLER	$$$ Germany
Export: Production	Create Gerber files	Required for PCB fabrication
Export: As Image	Create image files of circuit designs	Breadboard, Schematic, and PCB
List of Parts (BOM)	List of parts and details	HTML web page

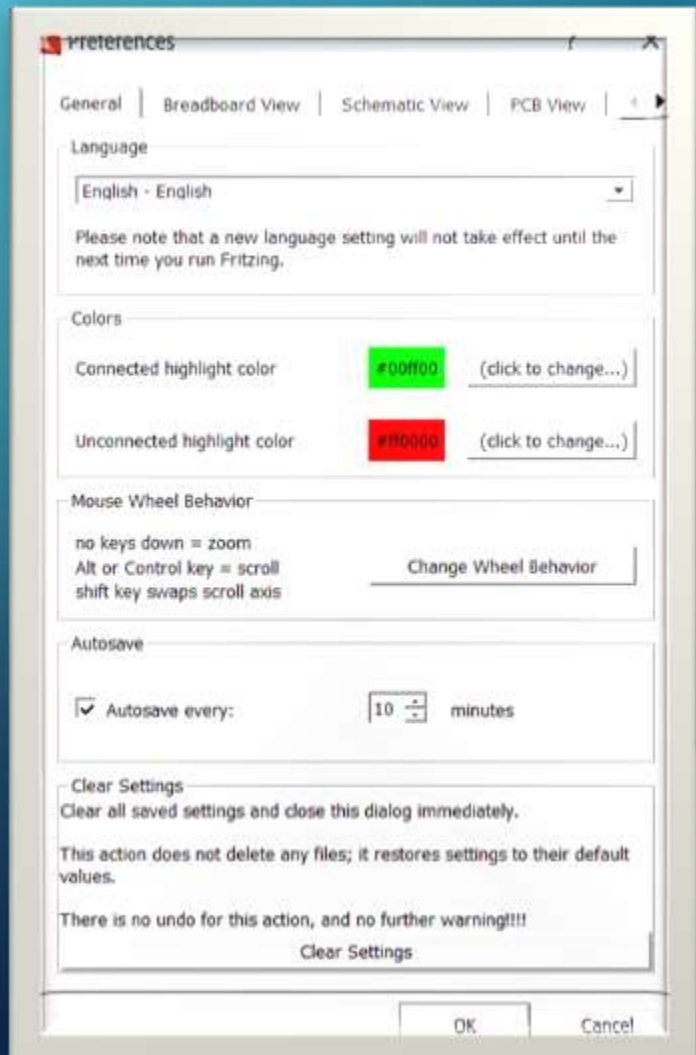


FRITZING
EDIT MENU

Edit menu provides basic editing functions a selected component part or wire

Function Selection	Use	Comment
Add Note	Provide notes for documentation	Use Breadboard and Schematic, not PCB
Preferences: Set curvy lines	Allows lines with curves	Tip: use only with breadboard
Preferences: Autosave	Provides recovery	Important since Fritzing can crash

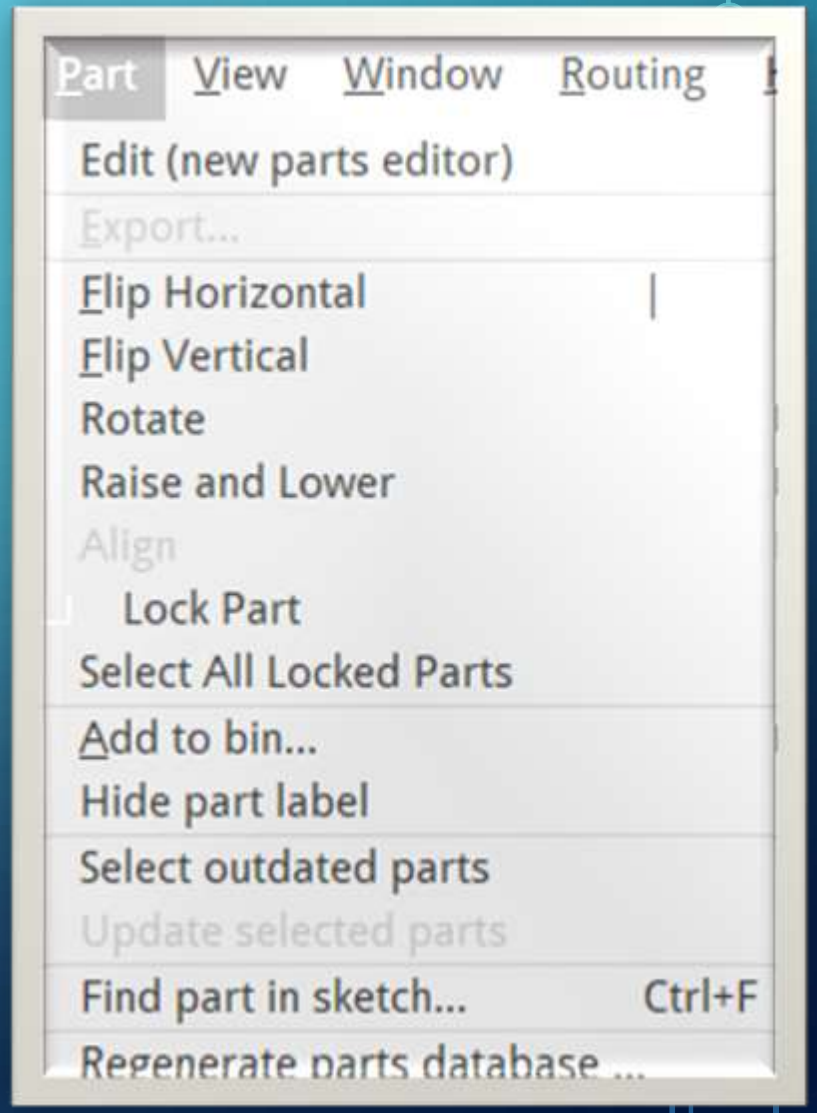




FRITZING
PART MENU

Part menu provides basic functions for working with components and wires

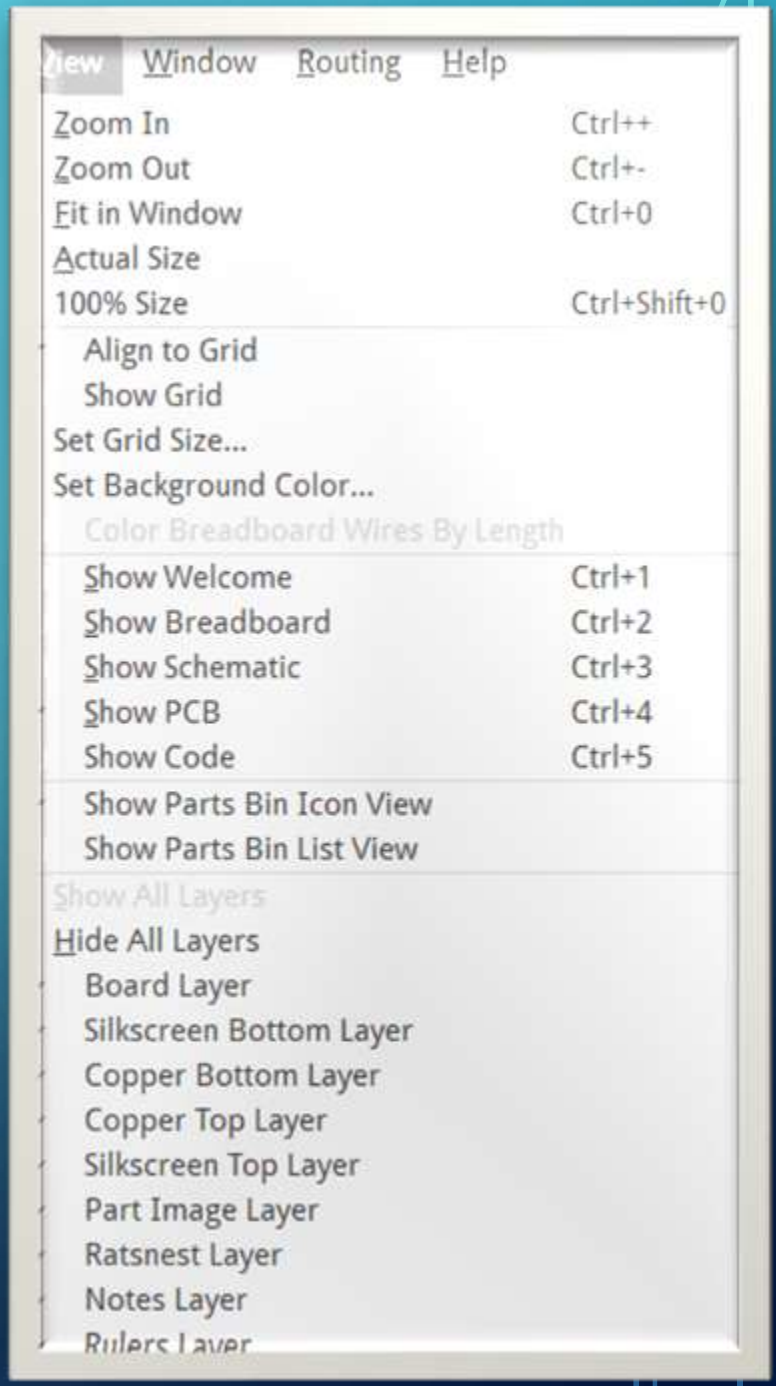
Function Selection	Use	Comment
Edit (new parts editor	Modify existing parts	Advanced, not used often
Flip Horizontal / Flip Vertical	Flip part in place	Works with only some parts
Rotate	Rotate selection in place	Used frequently, same as icon in lower left of main window
Raise and Lower	changes selection's view layer	Works with only some parts
Lock Part	Lock a part or background from being moved or edited	Very useful for background, notes, PCB image
Select All Locked Parts		Used before unlocking all parts
Add to bin…	Adds selected part to MINE bin	Use for quick reuse of common parts
Hide part label	Remove label from view	Useful to gain access to connection points



FRITZING
VIEW MENU

View menu is used to select windows to be displayed and change them

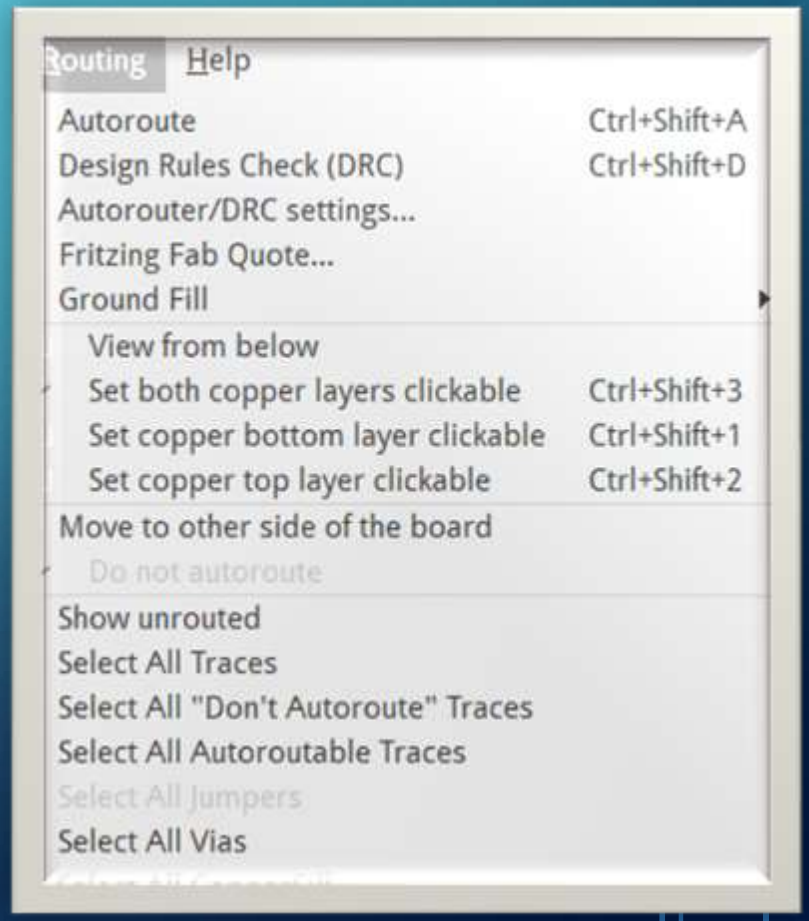
Function Selection	Use	Comment
Zoom In/Out	Zoom	Position cursor, then zoom in/out. Also, use mouse scroll wheel
Fit in Window	Fits everything to window	Very useful
Actual Size	Shows PCB size	Printed PCB is also actual size
Align to Grid	Snap parts, connections to grid points	Use in most cases, unless very small changes are needed
Show Grid		Not very useful, just snap to grid
Set Grid Size		0.1” common for most parts. Use 0.05” or 0.025 for traces on dense PCB
Show xxxx	Change window view	Same as clicking on tabs



FRITZING
ROUTING MENU

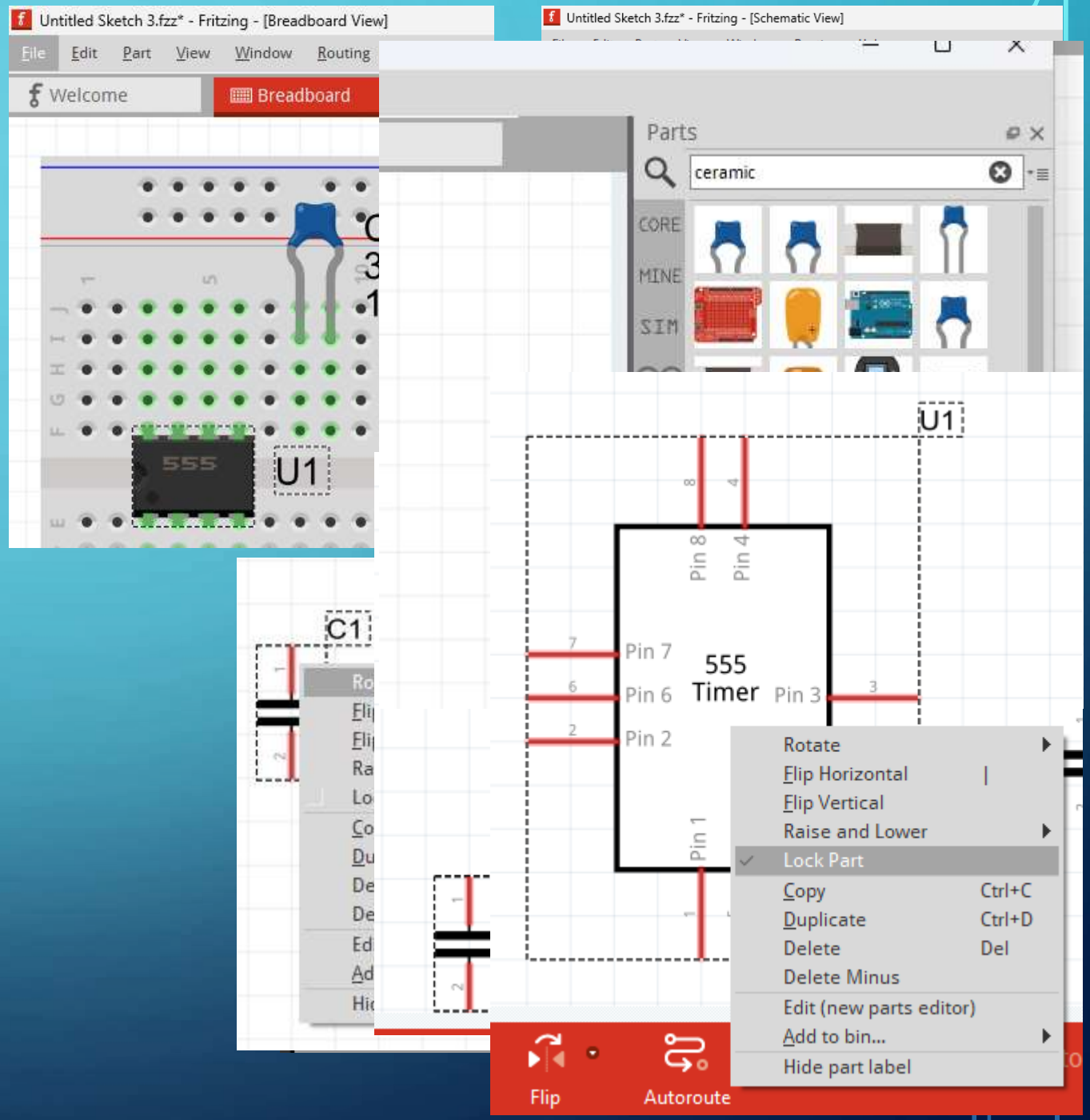
Routing menu is used for working with wires

Function Selection	Use	Comment
Autoroute	Routes (creates) wires from Ratsnest lines	Useful to initially create a wire(s) after placing parts
Design Rules Check (DRC)	Use to valid PCB design	Required before fabrication
Autorouter/DRC settings	Set to Professional	
Fritzing Fab Quote	Getting fabrication quote	Not used
Ground Fill	Create ground copper plane to all GND connections	Not used, difficult to verify if connections are made
View from Below / Top	Change user view to top or bottom of PCB	Use to work with top / bottom of PCB
Set 'xxx layer' clickable	Affects which layer you can click on	Typically set to 'both', change if having problems selecting connections
Select All Traces	Selects all PCB traces	Use to change all trace widths



FRITZING
WORKING WITH PARTS

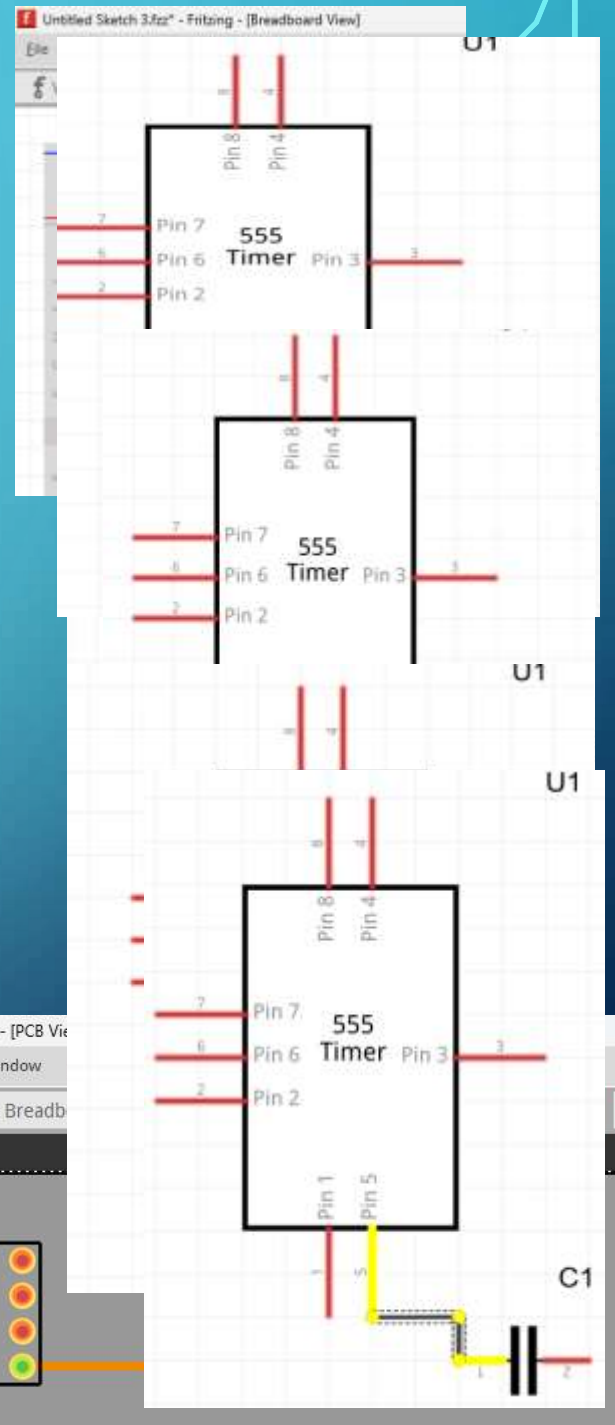
- Working with parts is similar for Fritzing Breadboard, Schematic and PCB designs
- After searching for a part in a bin in the Part Palette window, they can be dragged from the bin onto the design
- To provide easier connections, parts can be rotated using the icon at bottom of the main window or right clicking the part and selecting from the menu
- Part properties can be update in the Inspection Palette
- Part labels can moved, rotated and sized with selected part information by right clicking the label and selecting from the menu
- Parts can be locked to prevent changes
- Video



FRITZING

WORKING WITH WIRES

- Working with parts is similar for Fritzing Breadboard, Schematic and PCB designs
- Wires can be created by left-clicking on a pin and dragging a wire to another object (pin, bend point)
- Connecting a wire changes the color of connections
- Moving a part causes the wire to remain connected and change length or direction
- Hovering over a wire causes the wire to change color, indicating that a left-click will select the wire
- Right-click a selected wire and select Add Bendpoint to allow for a wire to be bent later by dragging the bend point
- Delete a selected wire by pressing the Delete key, or right-click and select Delete
- Disconnect and move a wire endpoint by hovering over the endpoint until it turns blue, then drag away from connection



FLASHER CIRCUIT - EXAMPLE

FLASHER CIRCUIT EXAMPLE
DESIGN CONSIDERATIONS

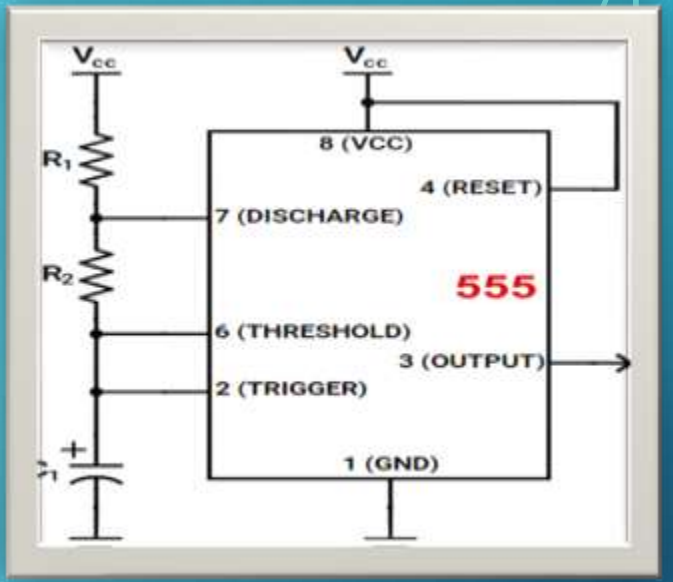
- Cost
  - PCB Production costs
    - determined by complexity, size and weight
    - Generally low cost
  - Component costs
    - Consider using components that are readily available for awhile and mass produced
      - i.e. available in AliExpress.com (China based)
  - Multiple circuits per board to lower costs, reduce space and ease of wiring
- Size / Shape
- Mounting
  - How will the PCB be mounted?
  - Affects location of holes or board dimensions
    - i.e. foam tape, screws, DIN rail, PCB housing
- Terminal connections
  - What type of connection; solder pad, solder hole, screw/spring terminal, CAT cable, or header
  - What size of wire will be used
  - Placement of connection points to ease access of wiring

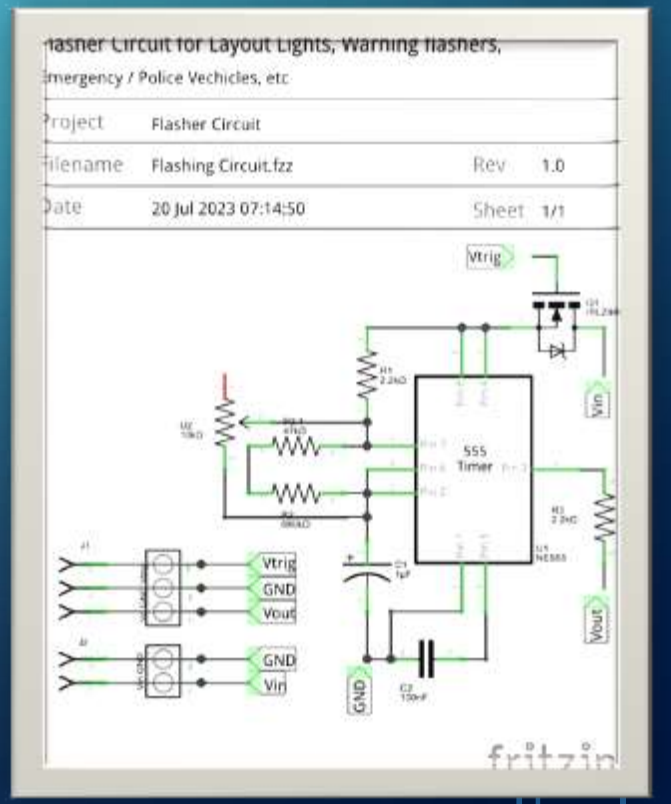
FLASHER CIRCUIT EXAMPLE
REQUIREMENTS

- Functional
  - Standalone flasher circuit to be used with train layout
  - Controlled (start/stop) by switch, button or Arduino (GPIO pin)
  - 500 msec flash rate
  - Power by layout's accessory bus (12v)
  - Flash 1 or 2 LED(s) independently
  - Current-limiting resistors (protect LED)
  - Screw/spring connectors or input/outputs
- Circuit
  - Based on 555 Timer (Wikipedia definition)
  - Astable mode (Free running) – continuous rectangular wave output signal
  - Circuit configuration based on 555 Timer Calculator
- PCB
  - Ease of Implementation (design / build)
    - Designed using Fritzing
    - Use 555 Timer IC
    - Use PTH components for ease of soldering
    - Screw (or spring) terminal connections (solid wire) & solder pads (stranded wire)
  - Low Cost (parts / fabrication)
    - 1 or 2 layers (note: Fritzing only supports 1 or 2 layers)
    - Minimal in size (low cost)

FLASHER CIRCUIT EXAMPLE
SCHEMATIC MODIFICATIONS

- Schematic source: https://www.digikey.com/en/resources/conversion-calculators/conversion-calculator-555-timer
- Astable Mode: continuous rectangular wave form
- Modifications:
  - Added missing pin 5 -> GND
  - Based on 500 msec on/off per sec flash rate (ref 555 Timer Calculator)
    - R1 = 680 ohm
    - R2 = 727k ohm (680k ohm + 47k ohm) OR Trimpot (0-2M ohm)
    - C1 = 1uF (creates 1 sec cycles)
  - Added transistor to allow low voltage switching
  - Added connectors; header holes, pads, terminals
    - Power (Vin & GND) to layout accessory bus (5-16V)
    - Trigger (Vtrigger): to 5-16V (Arduino or button)
    - Output (Vout & GND): LED /w 2.2K resistor



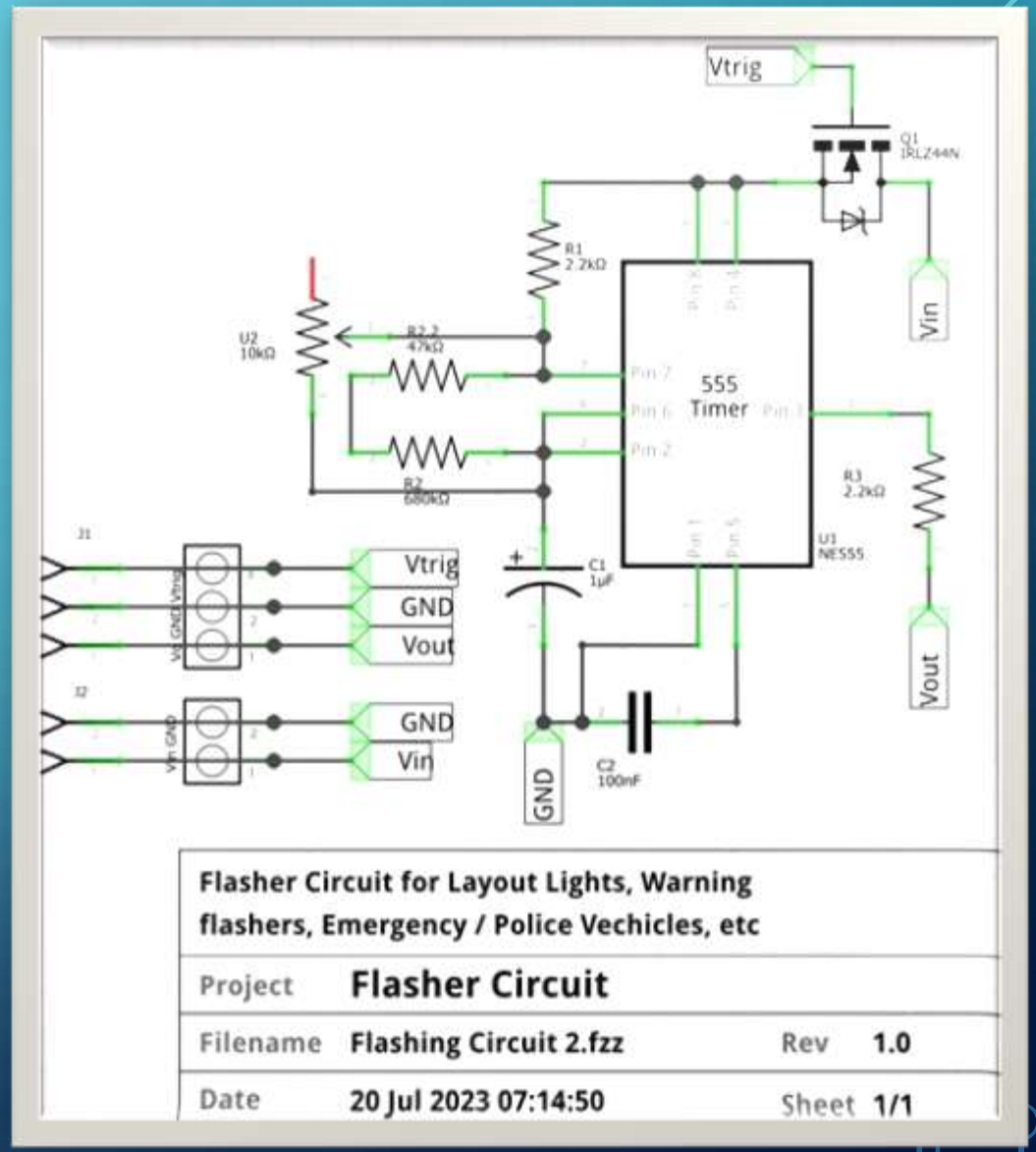


FLASHER CIRCUIT EXAMPLE
COMPONENT SELECTION

Component	Cost (/w link)	Comments
Resistor – 680, 680k, 2.2k, 47k	*Low (<$1.00)*	100 pieces, or kits of assorted types
Transistor – IRLZ44N	*Low (<$0.35)*	10 pieces
Capacitor – 1uF 50V	*Low (<$0.07)*	50 pieces
Capacitor – 100nF	*Low (<$0.03)*	100 pieces
Timer IC – NE555	*Low (<$0.20)*	20 pieces
Terminal – Spring	*Low (<$0.40)*	Sets of 2 or 3 connectors
Potentiometer (Trimpot)	*Low (<$0.10)*	
PCB	*JLCPCB ~ $1.00*	Minimal lot of 5

FLASHER CIRCUIT EXAMPLE
SCHEMATIC VIEW

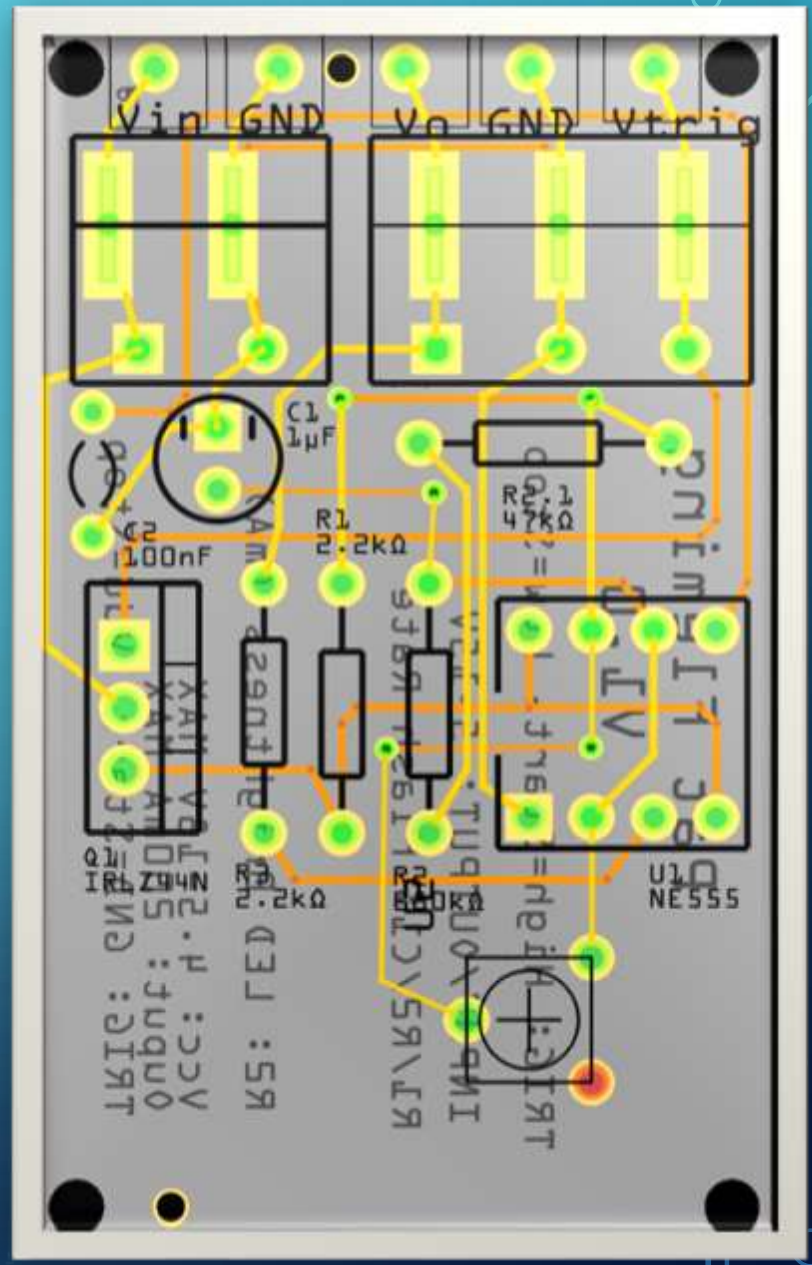
Object	Comments
Schematic Frame	Project information, outline
Net Label	Virtual connections between wires
Timer IC	NE555 part from catalog
Capacitors (C1-C2)	Parts from catalog
Resistors (R1-R3) / Trimpot (U2)	Parts from catalog
Terminal Connectors	Parts from catalog
Pin Header (J1, J2)	Parts from catalog
Transistor (Q1)	Part from catalog
Connections between parts	Generates Ratsnest lines



FLASHER CIRCUIT EXAMPLE

PCB VIEW

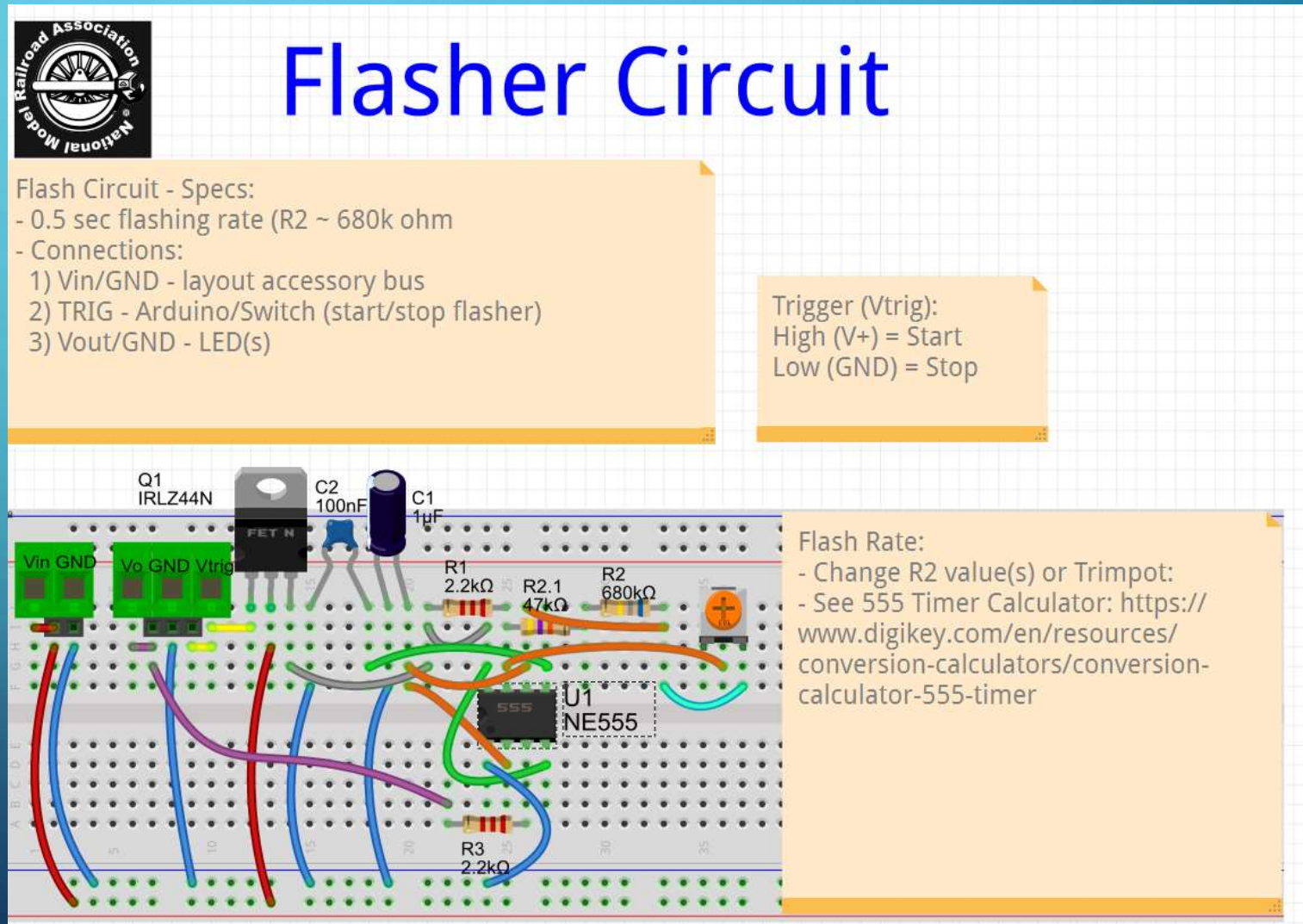
Object	Comments
Hole	Drill holes for screws
Components (Resistors, Capacitors, Transistor, Trimpot, NE555 Timer IC	From parts catalog
Wires	Converted Ratsnest Lines, Rerouted
Via	Hole with copper to connect top wire with bottom wire
Logo	Text on top/bottom silkscreens



FLASHER CIRCUIT EXAMPLE

BREADBOARD VIEW

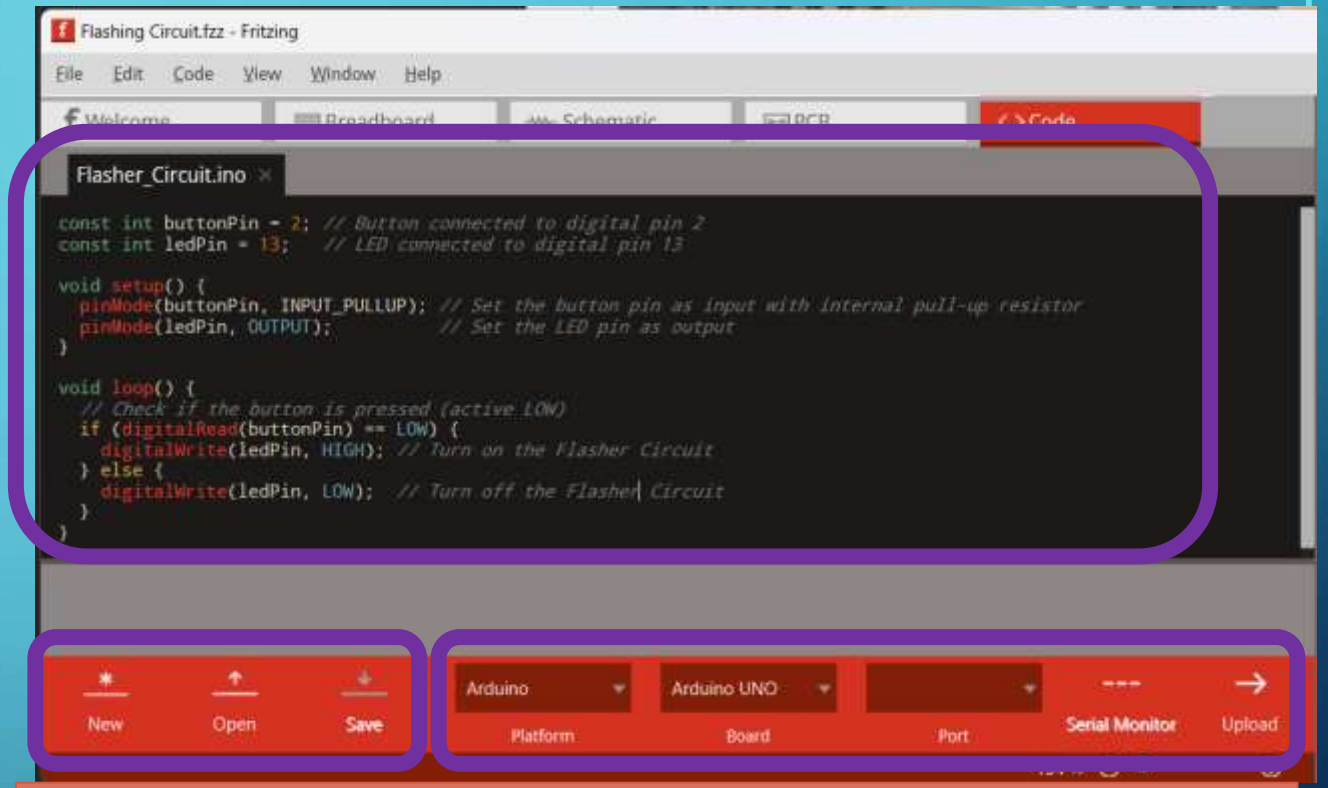
Component	Usage
Breadboard	Place to put parts, different sizes
Logo (Image setting)	Custom Images
Notes	Documentation
Parts	Electronic parts
Wire	Connections



FLASHER CIRCUIT EXAMPLE
CODE VIEW

- Code tab provides a basic Arduino IDE
- Buttons
  - New / Open - create Arduino code (sketch)
  - Save - save and name *.ino file

- Arduino Options
  - Platform
  - Board
  - Port
  - Serial Monitor
    - click to view Arduino serial monitor input/output
  - Upload
    - Compile/Upload firmware to Arduino



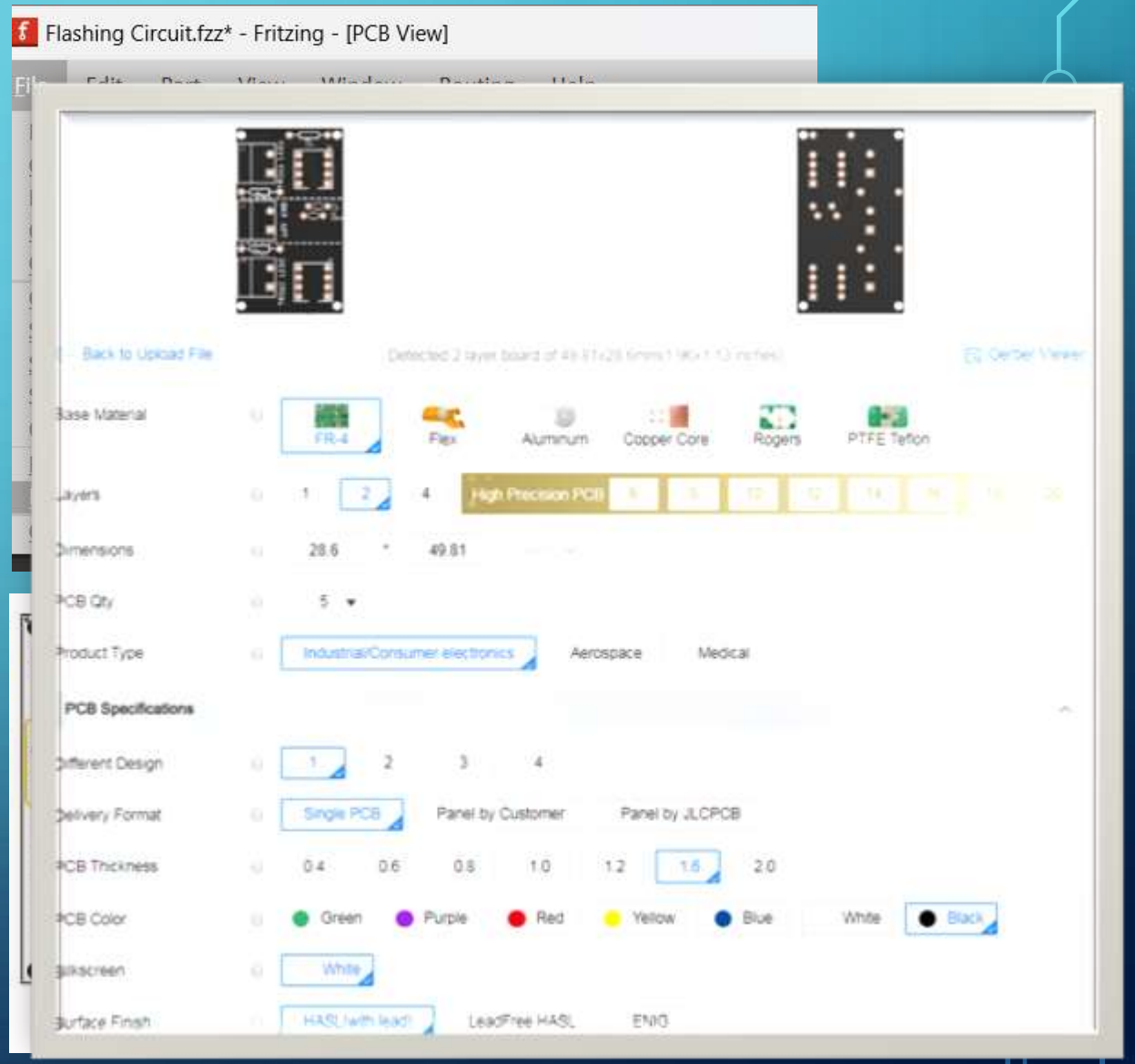
Example shown:

- Sketch for Flasher Circuit
- Defines input pin for a button
- Defines output pin connected to trigger
- setup()
  - sets input / output mode of pins
- loop()
  - Checks for button being pressed
  - Set output pin High while pressed

ORDERING A PCB

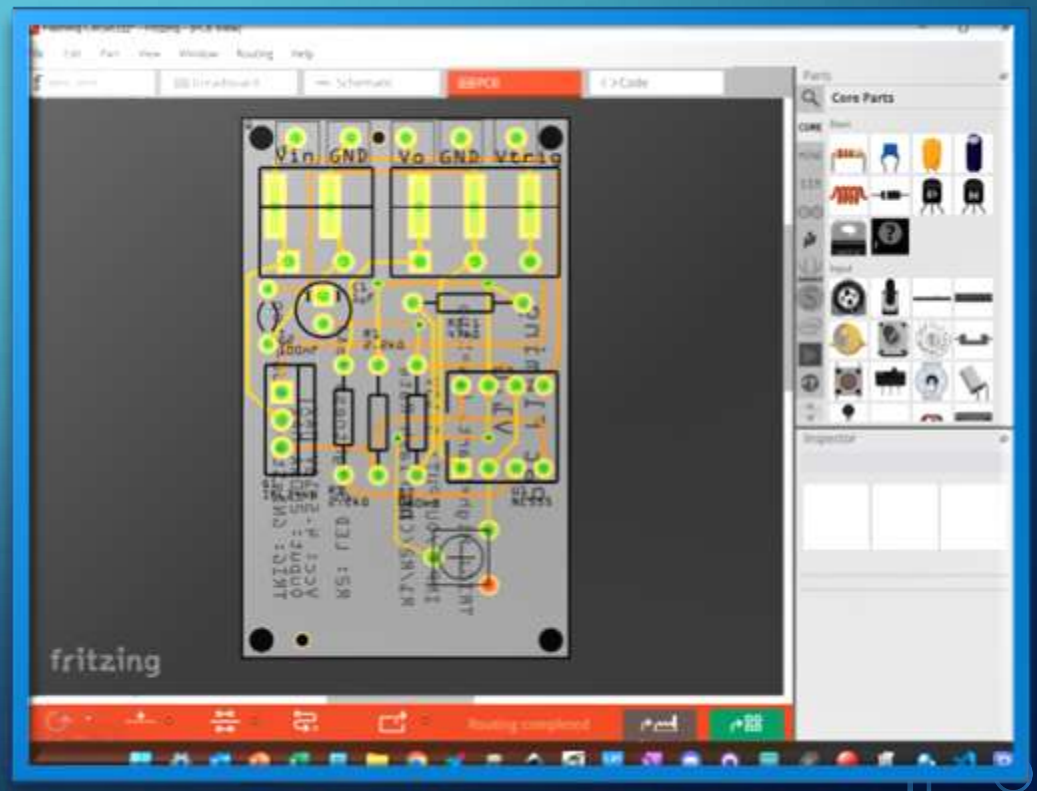
ORDERING A PCB
VERIFICATION

- Before ordering
  - Print PCB (from Fritzing menu: File -> Print)
  - Prints to actual size
  - Check component placement (clearances)
  - Include screws in holes, terminals, etc.
- Select a fabricator
  - Multiple options for PCB fabrication: JLCPCB, PCBWay, etc..
  - JLCPCB provides high quality and low cost fabrication of PCBs
    - Cost vs time: 2-3 week delivery



ORDERING A PCB
CREATE GERBER FILES

- What is a Gerber File:
  - Fabricators require a PCB design packaged as a ‘Gerber’ file
  - Gerber file is a zip file containing multiple files that make up the PCB design
  - Each ‘text’ file describes one aspect of the PCB
    - top copper, bottom copper, top silkscreen, drill holes, etc.
- How to create a Gerber File for Fabrication
  - From Fritzing Top Menu:
    1. Select File
    2. Choose Export
    3. Click on Production
    4. Then, select Extended Gerber (RS-274X)
    5. Select a Folder for output, click Select Folder
       - Tip: match Folder name to Fritzing sketch file name
  - From Windows File Explorer
    - Open the folder with the Gerber Files
    - Select all of the files (alt-a)
    - Right click and choose Compress to ZIP file
    - Tip: Rename to ZIP file to match Fritzing sketch name



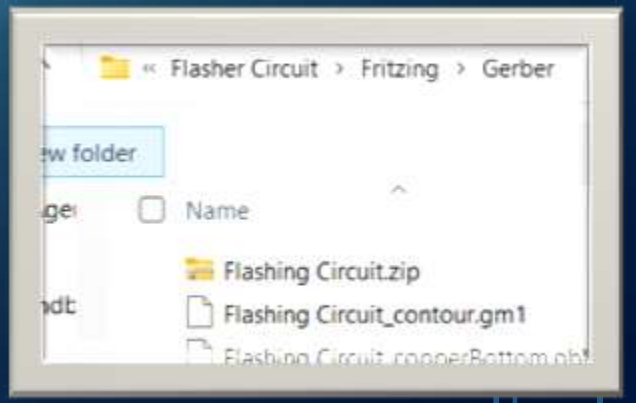
ORDERING A PCB
PCB COSTS (JLCPCB)

Quantity	Board	Shipping	$/PCB
5	$2	$1.44 (1-14 days)	$0.70
5	$2	$16.76 (2-4 days)	$3.80
10	$5	$1.44 (1-14 days)	$0.70

- Production time
  - normally 2 days for Green PCB
  - other colors add 1-2 days to production
- PCB shipping costs
  - based on weight (and volume), thinner boards reduce weight and costs
  - Global Standard Direct shipping (8-14 days) is lowest cost
  - Consider thinner boards when ordering large quantiles or when order faster delivery times
- Stencils – reduce shipping costs by reducing size of the stencil (vs default size)

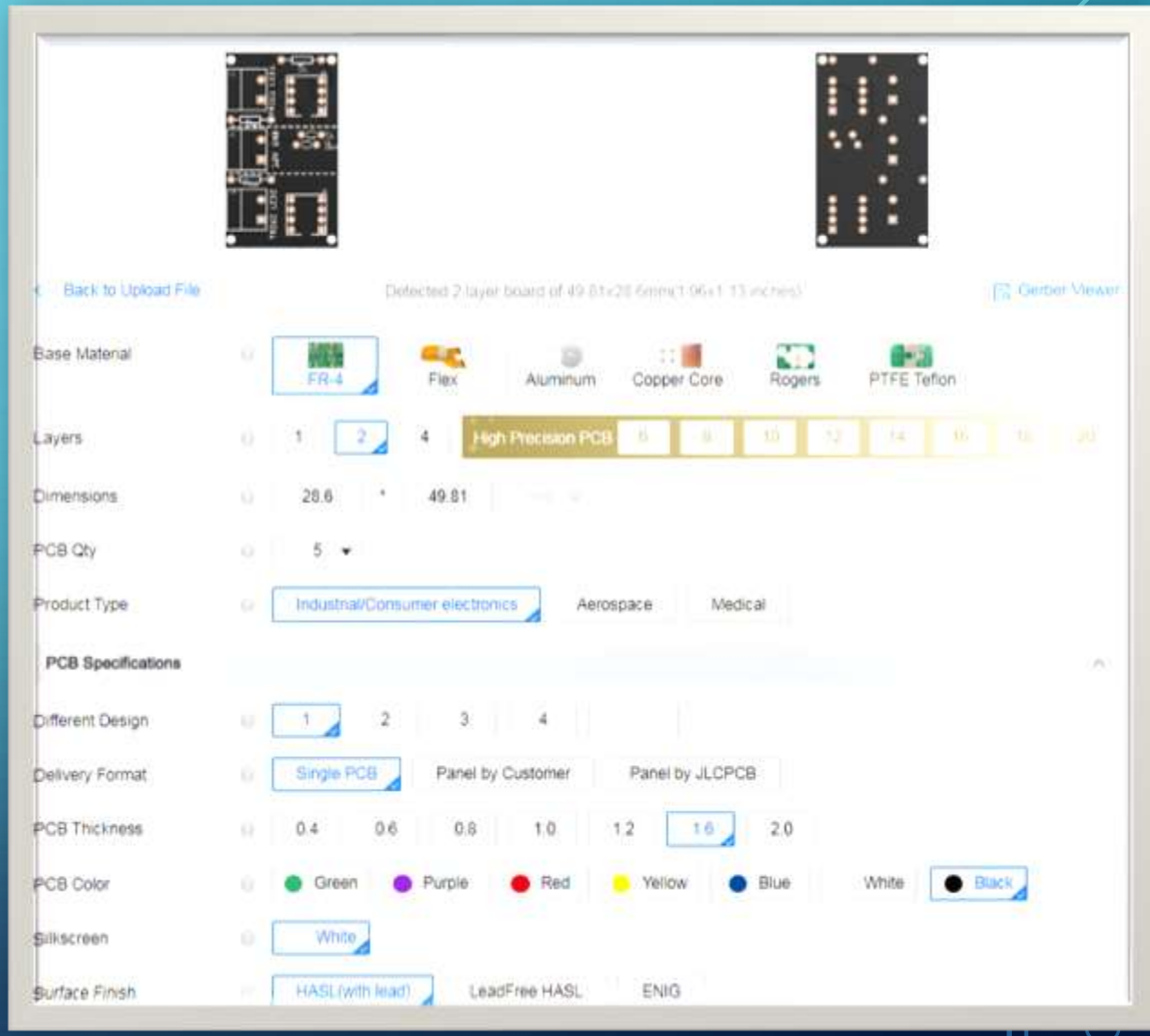
ORDERING A PCB
GETTING STARTED WITH JLCPCB

- Visit JLCPCB - https://jlcpcb.com/
- Create an account
  - Go to Sign In icon, then select Start Here
  - Tip: use Google account for fastest sign-in
- Start a new quote by selecting:
  - From top menu, click on Order now
  - From bottom right, click Instant Quote button
  - From bottom left, click Add Gerber file (goes directly to a quote)
- Click the Add Gerber File button
  - Upload Gerber (zip) file previously created from Fritzing production file



ORDERING A PCB
FILLING OUT A QUOTE

- Help text available for each order option (click icon left of selection)
- Use default values for most options
- Most common selection changes include:
- PCB Qty (multiples of 5)
- PCB Color: non-green increases production time by 1-2 days
- Remove Order Number – cost ~$1 per order
- Stencil: select “Yes” when using SMD parts to increase accuracy and save time applying solder paste
- Tips to reduce shipping costs when ordering faster service, larger boards, or larger quantities
  - Reduce PCB Thickness to 0.8mm
  - Stencil: use Custom Size to minimize stencil to reduce shipping package size



FLASHER PCB ASSEMBLY - MANUAL

- Solder NE555
  - Orientation by aligning indent on IC with notch in silkscreen and on IC so pin 1 (GND) is located in lower left corner
  - Hint: seat the IC by first soldering one leg, then push down IC while reheating to seat flat against PCB. Finish soldering remaining 7 legs.
- Solder IRLZ44N Transistor
  - Orientation based on IC facing forward with head sink in back
  - Hint: seat the IC by soldering one leg, seating IC flat to PCB and then solder remaining legs.
  - Trim legs on back of board
- Solder 1uF Capacitor
  - Orientation based on white stripe on side matched to GND (square) hole.
  - Hint: seat the capacitor first inserting both leads, bent on back to hold in seated position. Solder and trim leads.
- Solder 100nF Capacitor
  - Orientation – none, bidirectional.
  - Hint: seat the capacitor first inserting both leads, bent on back to hold in seated position. Solder and trim leads.
- Solder (4) Resistors
  - Orientation – none, bidirectional.
  - Hint: seat the capacitor first inserting both leads, bent on back to hold in seated position. Solder and trim leads.
- Solder Terminals
  - Orientation – wire holes point outward
  - Hint: seat the terminal by soldering one leg, seat terminal flat to PCB and solder remaining legs.

PCB – ASSEMBLY (BY JLCJCB)

- JLCPCB Parts Catalog
- Details on ordering
- Bill of Materials(BOM) File
  - Provides list of required parts, matched to designators
  - Uploaded to JLCPCB in .csv, .xls, or xlsx format
  - JLCPCB Instructions
  - Flasher Circuit – BOM file
- Pick & Place File
  - Provides component placement on circuit board
  - JLCPCB Instructions
  - Obtain location info from PCB view Inspector palette window
    - Set location field unit of measure to 'mm' (not in)
    - Use positive orientation, not negative (i.e. 270 not -90, 180 not -180)
  - Uploaded to JLCPCB in .csv, .xls, or xlsx format
  - Flash Circuit – CPL File
- Cost + Shipping
  - $38 (5) PCB = $8 each

SUMMARY	
Merchandise Total	$21.60
Shipping	$16.76
Grand Total:	$38.36

Comment	Designator	Footprint	JLCPCB Part # (optional)
IRLZ44n	Q1	TO220	C38774
680Ω	R1	THT-400 mil	C119866
680kΩ	R2	THT-400 mil	C70785
47kΩ	R2.1	THT-400 mil	C61302
2.2kΩ	R3	THT-400 mil	C120063
1uF	C1	Plugin,D6.3xL11mm	C46084
100nF	C2	Plugin,P=5.08mm	C2761731
EN555	U1	DIP8	C114801

Designator	Mid X	Mid Y	Layer	Rotation
Q1	-0.254mm	24.384mm	T	90
R1	6.667mm	26.848mm	T	90
R2	6.215mm	23.851mm	T	90
R2.1	17.920mm	12.065mm	T	0
R3	7.480mm	29.845mm	T	90
C1	4.953mm	34.798mm	T	0
C2	1.524mm	15.113mm	T	0
U1	18.796mm	23.241mm	T	270

PCB ASSEMBLY (BY JLCJCB) – BILL OF MATERIALS

- JLCPCB Assembly Summary
- Quantities reflect requirements for purchase, not what is actually used

Top Side Total 8 parts detected 8 Parts confirmed

Upload BOM/CPL

Uploaded BOM Data

Review Matched Parts

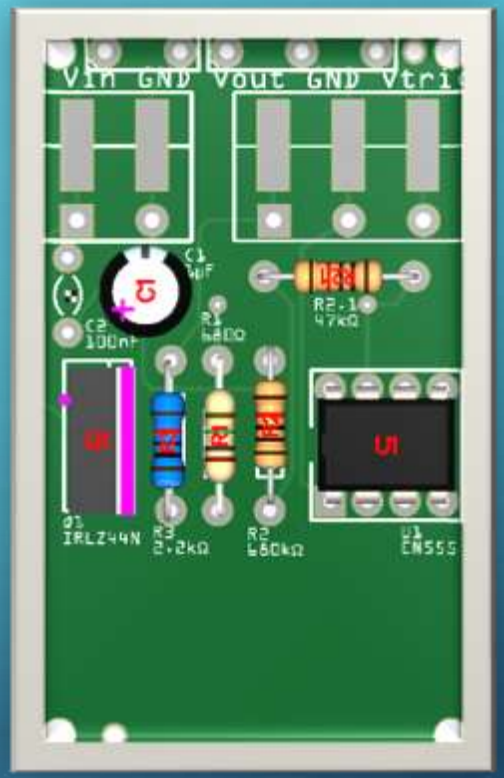
Top Designator	Comment	Footprint	Matched Part Detail		Qty	Source	Lib Type	Total Cost	Select
R3	2.2kΩ	THT-400mil	CF1/4W-2.2KΩ±5% T Carbon Resister 2.2kΩ 250mW ±350ppm/℃ ±5...	C120063	6	JLCPCB	Extended	$0.0324	☑
C2	100nF	CeramicCapaci...	DCS104Z26Y5VF6FJ5BQ -20%~+80% 100nF Y5V 50V Plugin,P=5mm C...	C2896072	7	JLCPCB	Extended	$0.1729	☑
R1	680Ω	THT-400mil	CR1/8W-680Ω±5%-ST52 Carbon Resister 680Ω 125mW ±350ppm/℃ ±5...	C2897091	7	JLCPCB	Extended	$0.0154	☑
Q1	IRLZ44n	TO220	IRLZ44NPBF 55V 47A 22mΩ@10V,25A N Channel TO-220...	C38774	5	JLCPCB	Extended	$3.7715	☑
R2.1	47kΩ	THT-400mil	CFR0W4J0473A50 Carbon Resister 47kΩ 250mW ±450ppm/℃ ±5...	C61302	6	JLCPCB	Extended	$0.0504	☑
U1	EN555	DIP8	NE555DR SOP-8 Timers / Clock Oscillators ROHS	C695838	5	JLCPCB	Extended	$0.2745	☑
C1	1uF	ElectrolyticCap...	ERT105M2GE11RR 1uF 400V ±20% Plugin,D6.3xL11mm Aluminum...	C697501	6	JLCPCB	Extended	$0.4278	☑
R2	680kΩ	THT-400mil	CFR0W4J0684A50 Carbon Resister 680kΩ 250mW -700ppm/℃~0...	C70785	6	JLCPCB	Extended	$0.0252	☑

Please carefully check the packages of selected parts before proceeding.

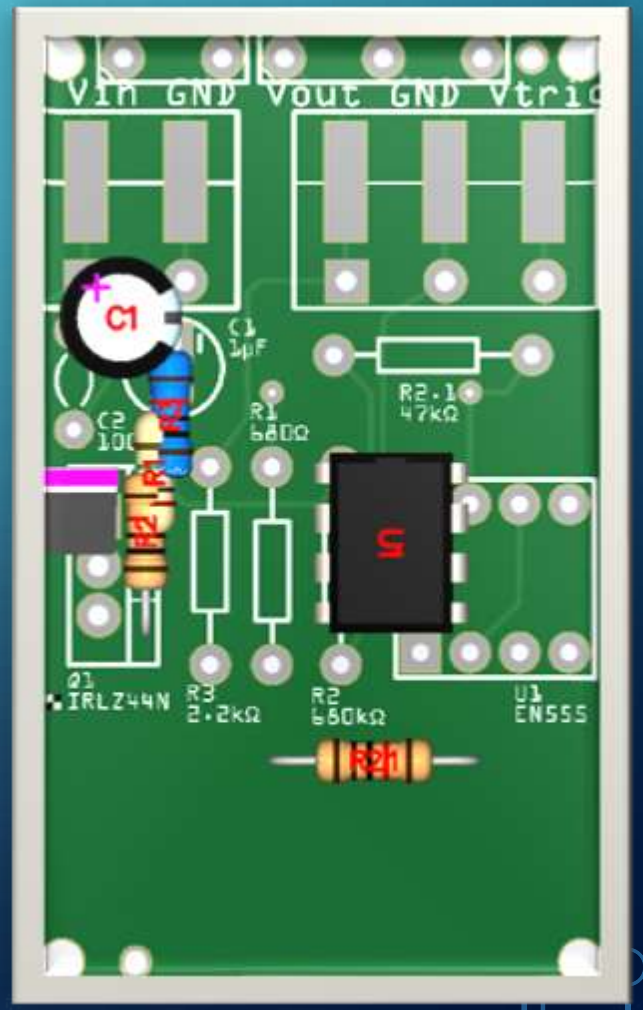
PCB ASSEMBLY COMPONENT PLACEMENT

- JLCPCB Component Placement is not accurate
- Either correct yourself or have JLCPCB engineers place
- Manual placement by
  - Selecting component and repositioning
  - Right click to rotate
- Not all components have a ‘footprint’ (icon)

Default Placement



Manual Placement



WHAT'S NEXT FOR YOU

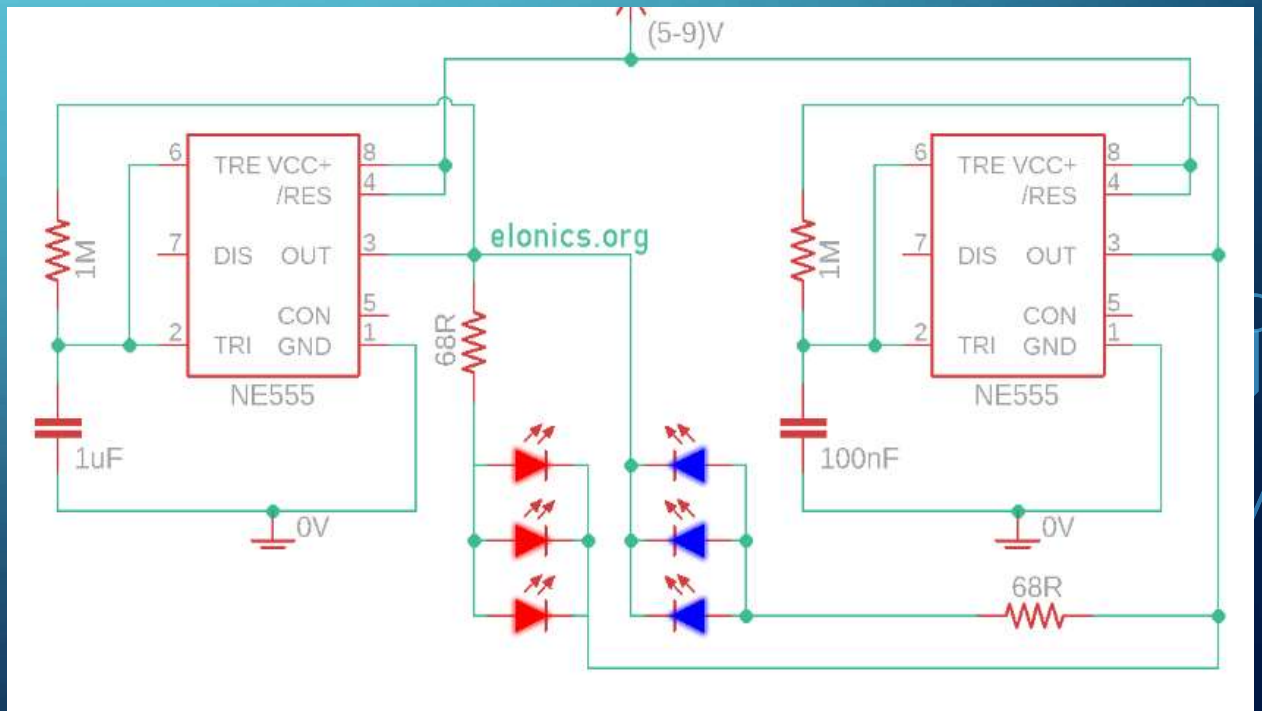
MORE 555 TIMER BASED CIRCUITS

- TALKING ELECTRONICS – 50 CIRCUITS USING 555 TIMER IC (BY COLIN MITCHELL)

- LARGE COLLECTION OF CIRCUITS BASED ON 555 TIMER IC
  - Police Flashers / Siren
  - Crossing Lights
  - Relays, Servos
  - Switches, Debounce
  - Delay
  - LEDs: Fade in/out, up/down

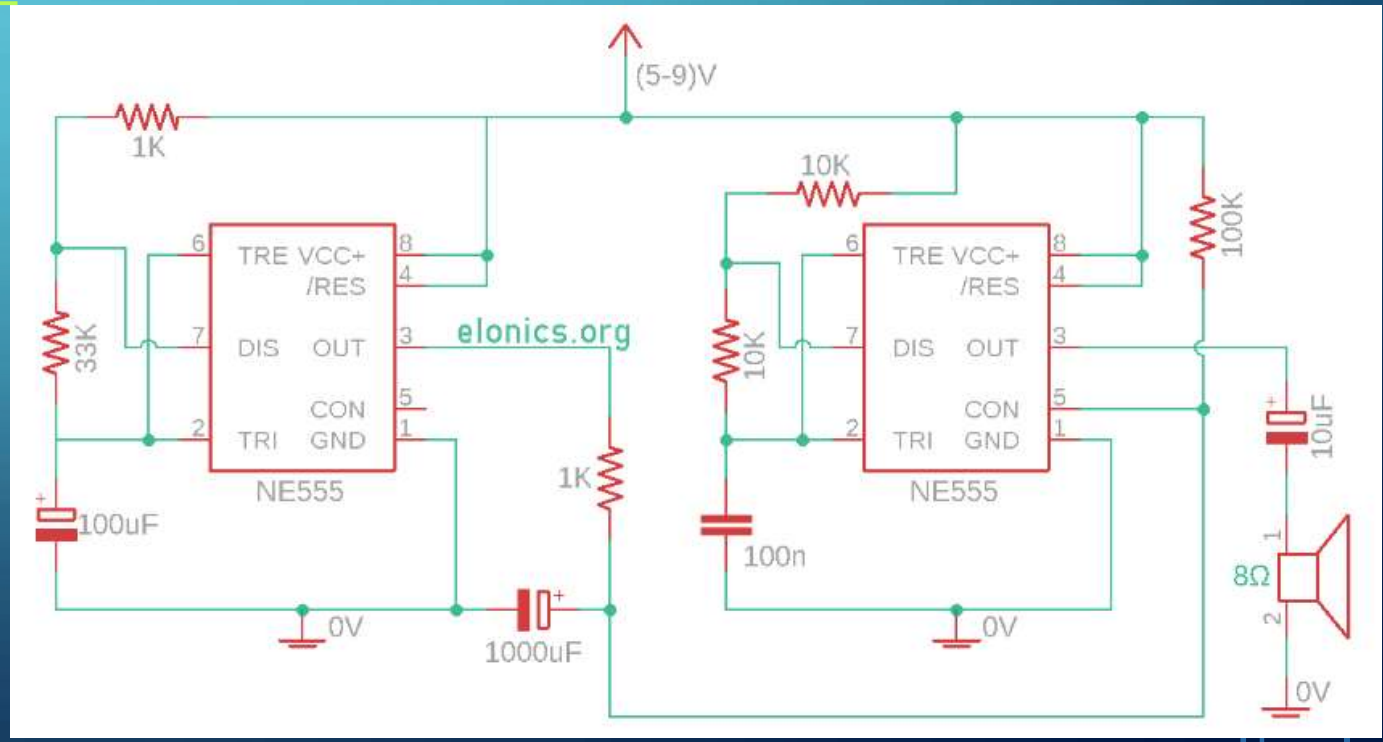
POLICE LIGHTS THEMED FLASHING LED CIRCUIT USING 555 IC

- Uses 2x NE555 Timer IC
- Alternating red/blue LED's flashing
- Article / Source: https://elonics.org/police-lights-themed-flashing-led-circuit-using-555-ic/
- Video: https://youtu.be/XAi5aJ-89WY
- Parts:
  - 2 x 555 Timer ICs
  - LED's: 3 x Red, 3 x Blue
  - Resistors: 2 x 1M, 2 x 68R (Resistors for LED's)
  - Capacitors: 1uF, 100nF
  - Breadboard
  - Few Breadboard Connectors
  - (5-12)V Power Supply



POLICE SIREN CIRCUIT USING 555 IC

- Uses 2x NE555 Timer IC
- 2 different tones/frequencies, similar to the sound emitted from police cars
- Article / Source: https://elonics.org/police-siren-circuit-using-555-ic/
- Video: https://youtu.be/ca4KMl4gui4
- Parts:
  - 2 x 555 Timer IC
  - 8Ω Speaker
  - Resistors: 100K, 33K, 2 x 1K, 2 x 10K
  - Capacitors: 1000uF, 100uF, 10uF, 100nF
  - (5-9)V Power Supply



PROBLEM SOLVING

EXAMPLES: PCB FOR DWARF SIGNAL

- Requirements:
  - Leverage low-cost housing (HO)
  - Mount 2x LEDs into housing straight
  - Hide LED wiring on back of housing
  - Easy wiring to LED with color and markings
  - Use of JST wire connections for easy under layout connections

- Solution:
  - Circuit board /w 2x LEDs
  - 3 labeled solder pads and holes (2 color, 1 common)

- Assembly:
  1. Solder 3mm LEDs
  2. Grind LED tops flat (diffuse, Lens appearance)
  3. Grind PCB back smooth (flatten solder connections)
  4. Spray with flat black (cover lens)
  5. Add heat shrink tube (insertion into layout board)

EXAMPLES: PCB FOR SIGNAL MAST

- Requirements:
  - Leverage low-cost housing (HO)
  - Mount 3x LEDs into housing straight
  - Hide LED wiring on back of housing
  - Easy wiring to LED with color and markings
  - Easy mount of housing to mask

- Solution:
  - Circuit board /w 3x LEDs
  - 4x labeled solder pads (3 color, 1 common)

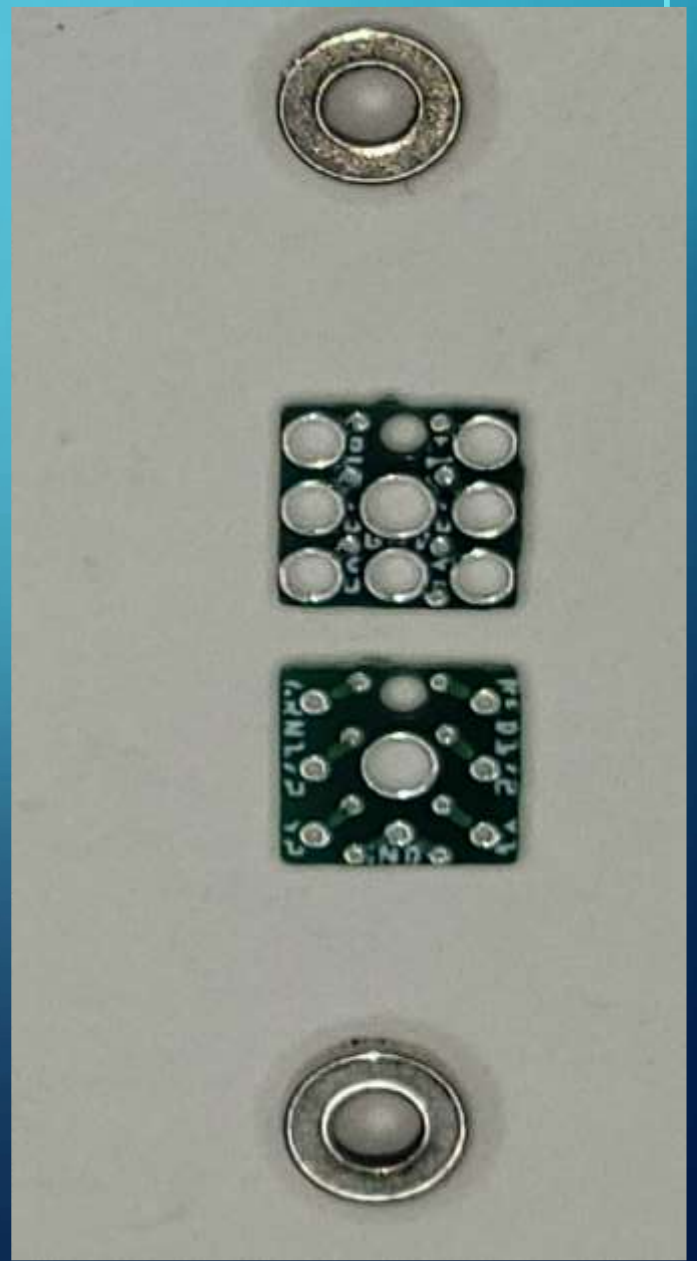
- Assembly:
  1. Solder 3mm LEDs
  2. Grind LED tops flat (diffuse, Lens appearance)
  3. Grind PCB back smooth (flatten solder connections)
  4. Glue to mask (straw, 2.5mm copper tube, ??)
  5. Spray with flat black (cover lens)

EXAMPLES: PCB FOR SIGNAL MAST BASE MOUNTING

- Requirements:
  - Break away signal mast
  - Easy placement and removal (modular setup, working on scenery, etc.)
  - Cover signal mast base with small signal box (0.5” square)

- Solution:
  - Pair PCBs with 6 connections for (2) 3-lamp heads
  - Use Pogo pins (spring loaded) to form (6) constant connection between boards
  - PCB form a plug and socket pair
    - Plug PCB uses ‘pogo’ pins and round donut (rare-earth) magnet
    - Socket PCB uses JST socket connectors on back
  - Magnets hold PCBs together, with Pogo pins forming (6) connections
  - Hold thru center of magnets and PCB for mounting 2.5mm mast

- Assembly:
  1. Solder Pogo pins (drop in plug (top) PCB holes, very little solder on back)
  2. Mount mast (slot at bottom to allow wires out of tube into (6) holes, solder
     - Use 30 awg Mag wire
  3. Solder (2) JST XH 2.54mm SMD sockets on socket (bottom) PCB

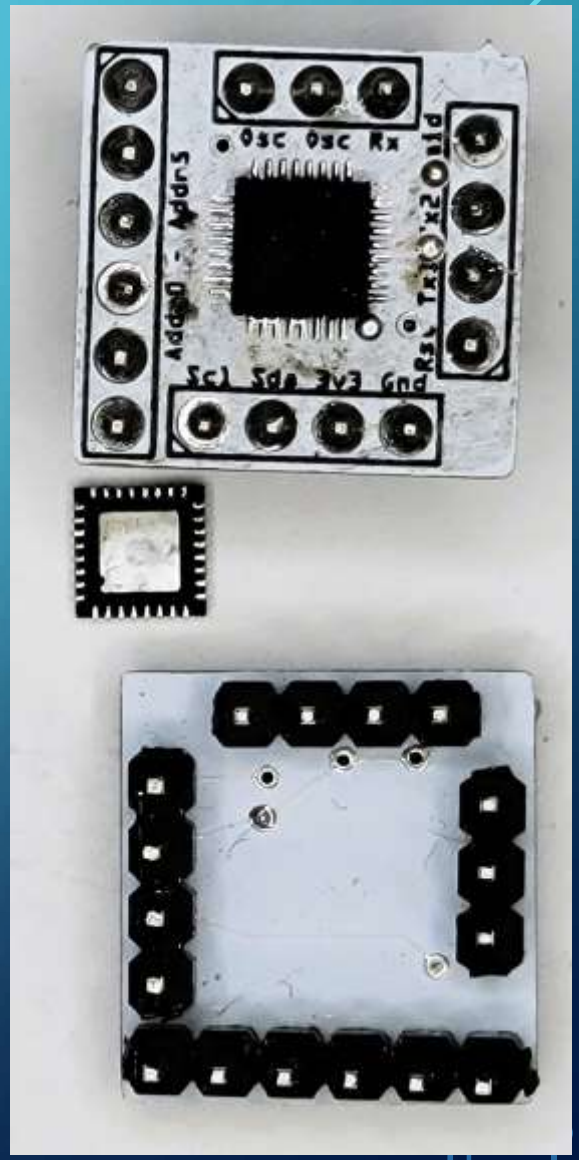


EXAMPLES: BREAKOUT PCB FOR SMALL IC

- Requirements:
  - Solder very small IC (QFN32 Package /w 32 solder pads, 5mm x 5mm)
  - Provide option for JLCPCB assembly
  - Provide for easy testing and rework if/when needed

- Solution:
  - Small breakout board, specifically for QFN32 package
  - Unique pin arrange for correct seating into main board
  - Extend QFN32 solder pads using Fritzing Parts Editor & Inscape (SVG editor)

- Assembly:
  1. Reflow
     1. Use PCB stencil to apply paste (difficult)
     2. Reflow in counter top oven
  2. Soldering Iron
     1. Use low temp solder with Oblique Horseshoe soldering tip
     2. Apply solder by stoking tip straight out from solder pad
        1. Solder flows from tip to solder pad, connecting to bottom of IC
     3. Solder one corner pin and verify alignment of IC pads to PCB pads
        1. Requires magnification and looking at side of mounted IC
     4. Solder opposite corner and verify alignment
     5. Solder remaining pins

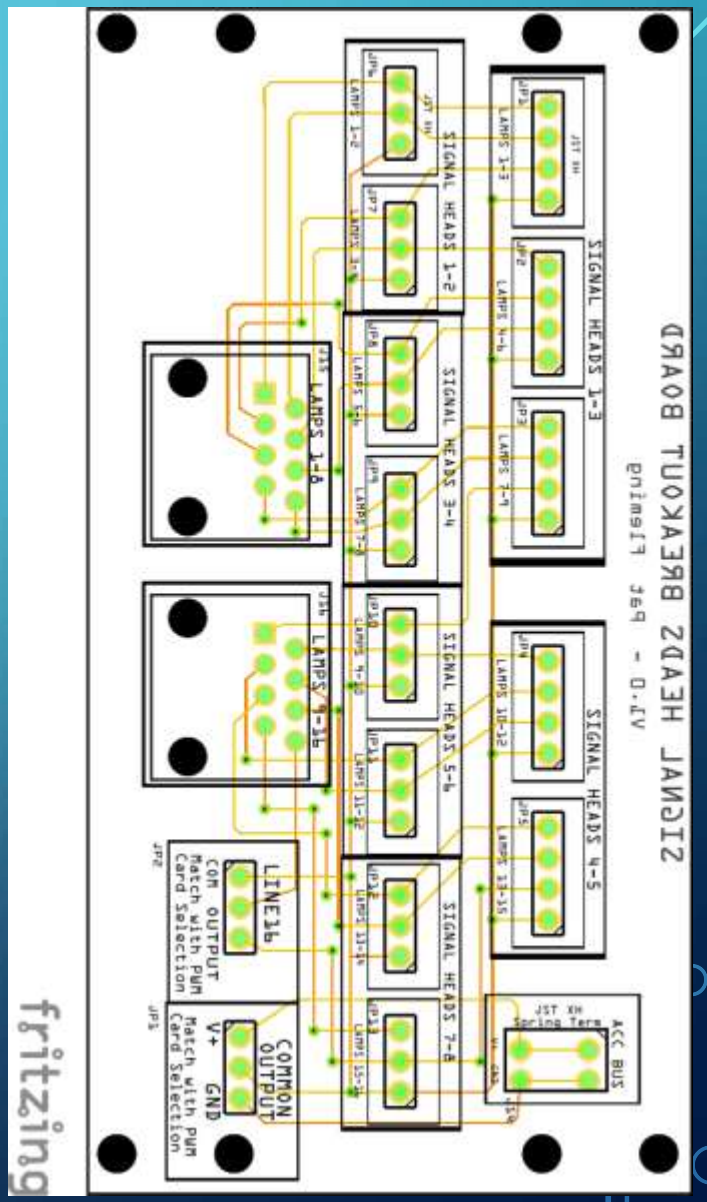


EXAMPLES: PCB FOR CONNECTION CONVERSION

- Requirements:
  - Simplify under layout wiring and connections for signal masts
  - Provide labeled connections for plugging in 2-3 wire plugs (JST XH 2.54mm)
  - Convert from (2) CAT cables(16 wires) to 5-6 JST sockets (5-6 signal heads)
  - Connections for layout accessory power
  - Multiple mounting options

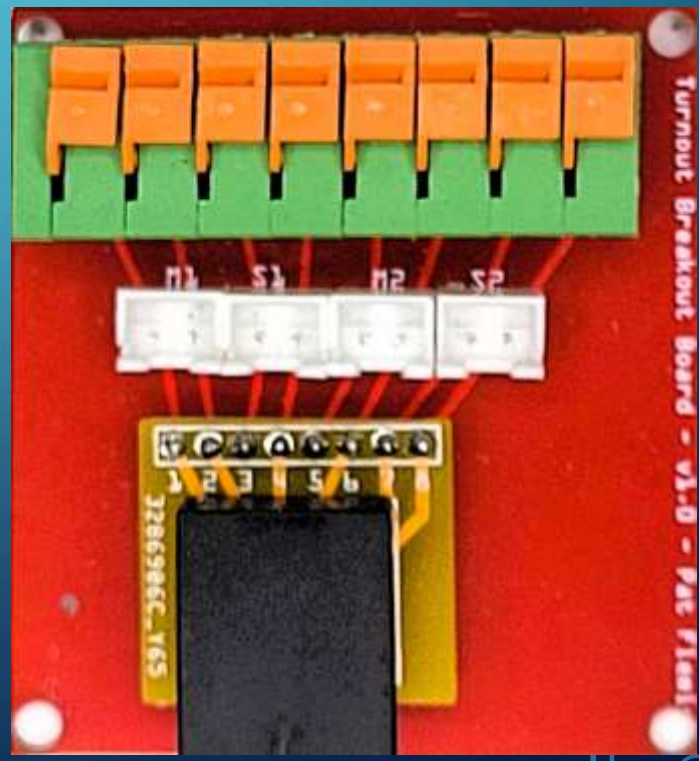
- Solution:
  - (2) RJ45 sockets for CAT cable
  - (6) 3-wire JST sockets for (6) 2-lamp mast heads
  - (5) 4-wire JST sockets for (5) 3-lamp mast heads
  - (1) Molex connector for layout accessory connection
  - Jumpers for setting LED common wire (anode/cathode)
  - 50mm height for PCB housing mount
  - Drill holes for screw mounting or DIN mounting brackets

- Assembly:
  1. Set socket into holes
  2. Hold socket in place with tack putty
  3. Solder one pin from back
  4. Reheat soldered pin and push socket flat to PCB
  5. Solder remaining socket pins



EXAMPLES: PCB FOR CONNECTION OPTIONS

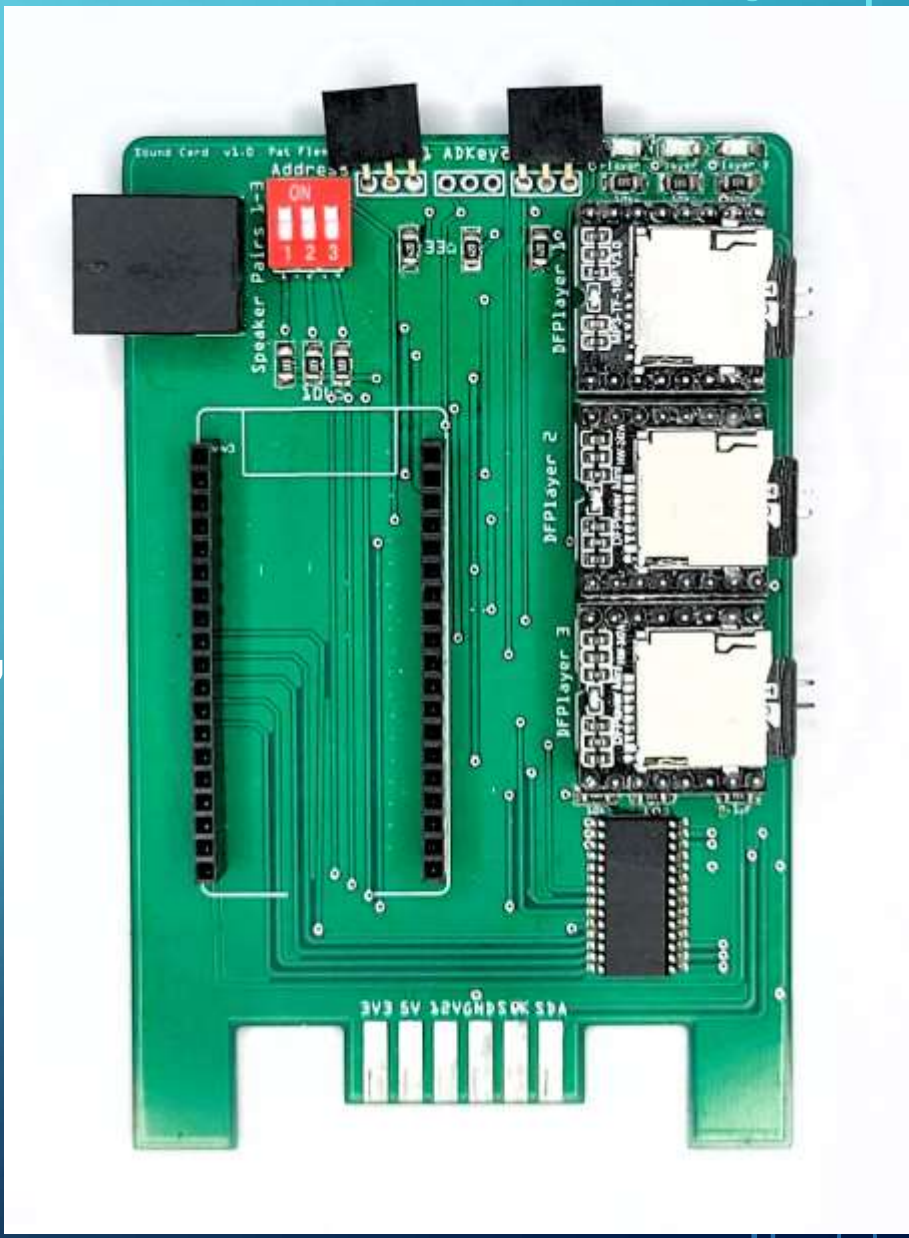
- Requirements:
  - Provide breakout board for multiple connection options
  - Input:
    - 2.54mm header for breakout board (RJ45)
    - 2.54mm terminal (screw or spring)
  - Output:
    - 2.54mm terminal (screw or spring)
    - JST XH 2.54mm socket
- Solution
  - Secondary breakout board for RJ45 socket, with 2.54mm pins
  - (4) JST XH sockets
  - 2.54mm holes for terminal connector
  - Drill holes for mounting



EXAMPLES: PCB FOR MODULES

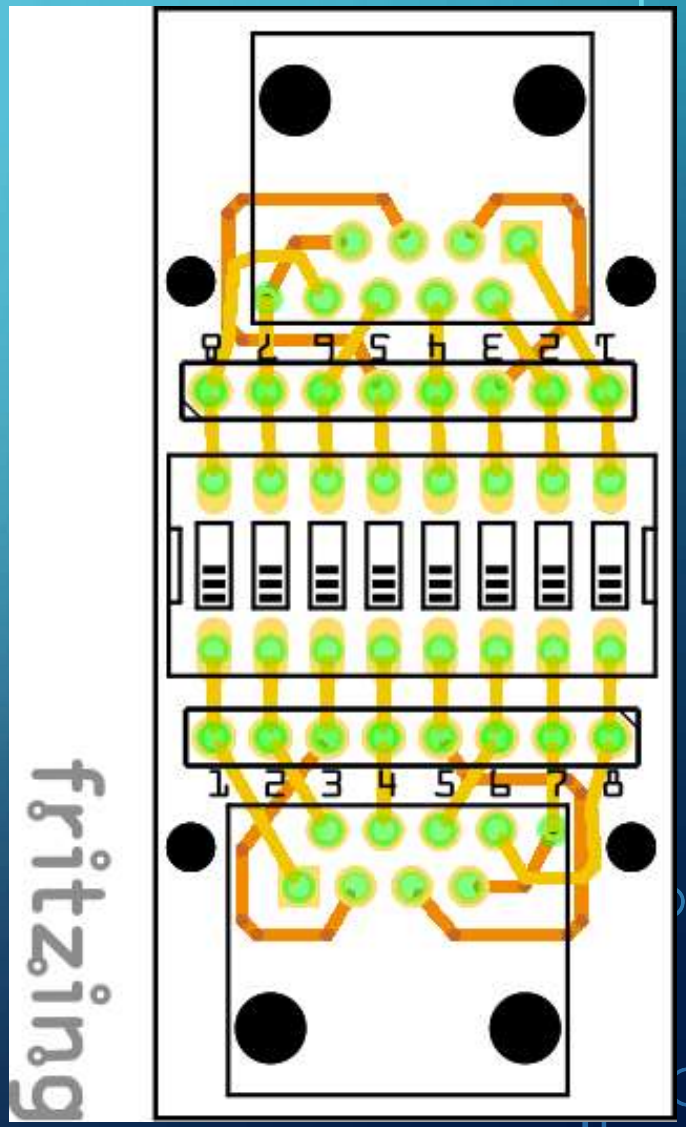
- Requirements:
  - Leverage commercial ‘modules’
  - Example:
    - MP3 player “DFPlayer Mini” (by DFRobot), with Mico-SDcard
    - ESP32 (Espressif)

- Solution
  - (3) sets of 8-pin female headers to mount DFPlayer Mini modu
  - (1) set of 19-pin female header to mount ESP32
  - 3-DIP switch for setting I2C address
  - (1) RJ45 socket for CAT cable to wire (3) speakers



PCB: REROUTING CAT CABLE WIRES

- Requirements
  - Patch panel concept; using switches and jumper wires (no electronics)
  - Custom wire routing between CAT cables
  - Input/Output – CAT cables
  - Simple, intuitive, documentable
  - DIN Rail Mountable
  - Expandable
- Solution
  - PCB with RJ45 sockets
  - Female Headers
  - Use 4” Dupont jumper wires for interconnections
    - male-male colored wires
    - Jumper wires can be connected between PCB boards
  - 8-DIP Switch for direct wire pass thru
    - ex. Connect input pin1 to output pin8
  - Label CAT cables to document wires (spreadsheet)



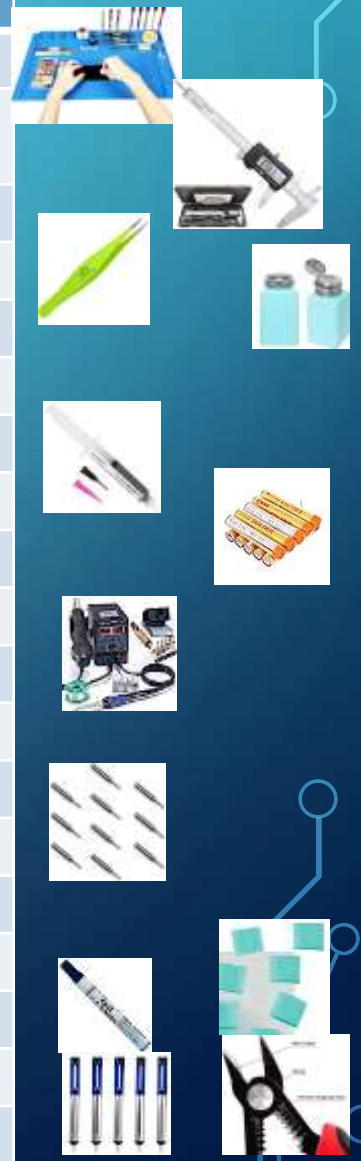
BACKUP

CIRCUIT REFERENCES

- Specifications:
  - Digikey has links to all IC specifications
  - E.g. Google “NE 555 Digikey”, then click on PDF icon next to part’s information
- Geoff Bunza’s blog (MRH)
- Model Railroad System (Deepwoods Software) by Robert Heller
- Model Railroad & Misc. Electronics by R. Paisley
- Talking Circuits – 4 free books of circuits for Model Railroads
- Fritzing projects and examples: Fritzing -> Files -> Open Examples…

PCB TOOLS

Tool	Purpose	Considerations	Purchase / Cost
Silicone Repair Mat	Mat to protect/organize	Invaluable during PCB work, heat resistant, easy to clean	Amazon - $22
Digital Caliper	Measurement of PCB / Components	Pin and component spacing, thickness	Amazon - $22
Tweezer	Place Components	Fine point, Non-Magnetic	Amazon - $10
Alcohol Dispenser	Cleaning PCB	Convenient pump	Amazon - $12 (2)
Scraper / Scraper	SMD: Spread solder paste	Stiff but flexible	Use old credit card / $0
Magnets	SMD: Hold PCB and Stencil	Rare Earth – 15mm x 3mm round	Amazon - $10
Counter Top Oven	SMD: "Reflow" solder	Digital (accuracy) , convection (airflow)	Amazon - $90
Solder Paste	SMD: Soldering Components	Low melting point (278F)	Amazon - $10
Component Organizer	Store small SMD Components	Small, multiple containers, clear, cap, reorganize	Hobby Lobby - $4
Solder Wire	PTH: Soldering Components	Low temp, thin wire, convenient to hold	Amazon - $10 (10)
Solder Station	Soldering	'Rework' & solder both PTH and SMD, control temp	Amazon - $76
Soldering Iron	Soldering		
Soldering Iron Cleaner	Soldering Iron Clean/Holder	Cleans better than sponge	Amazon - $16
Soldering Iron Tip	PTH: Soldering	Oblique Horseshoe – holds solder better	Amazon – $10 (10)
Soldering Flux	Improves soldering	Pen – more convenient	Amazon - $16
Desoldering	Removes solder	Suction works well	Amazon - $8 (3)
Digital Multimeter	Measure AC/DC, Ohms, Amp	Probes that attach to leads, test clips, etc.	Amazon - $13
Tack Putty	Hold components for soldering	Place over PTH component while soldering opposite side	Amazon - $10
Flush Wire Cutter	Cut PTH wires on PCB bottom	Flush cutting, optional wire stripper	Amazon - $8



WHERE TO BUY PCB COMPONENTS

- AliExpress - https://www.aliexpress.com (China)
  - Lowest cost
  - 4-6 week delivery time
  - Large selection of common parts
- Digikey
- Mouser
- Amazon
- eBay

PCB MOUNTING COMPONENTS

Component	Purpose	Considerations	Purchase
Screws	PCB Mounting	Use M3x8 /w 3.2 mm drill hole	AliExpress - $2.25 (50)
Standoff	PCB mounting spacers	Card stacking	AliExpress - $1.63 (50)
DIN Rail			
DIN Bracket	PCB to DIN mounting	1-2 per PCB board	AliExpress - $5 (5 pr)
DIN Rail Clamp	Mount DIN rail to surface	Quick removal of DIN rail	AliExpress - $10 (10)
PCB Housing	Holds 50mm height PCB	Optional end caps	AliExpress - $3/meter

PCB FABRICATORS

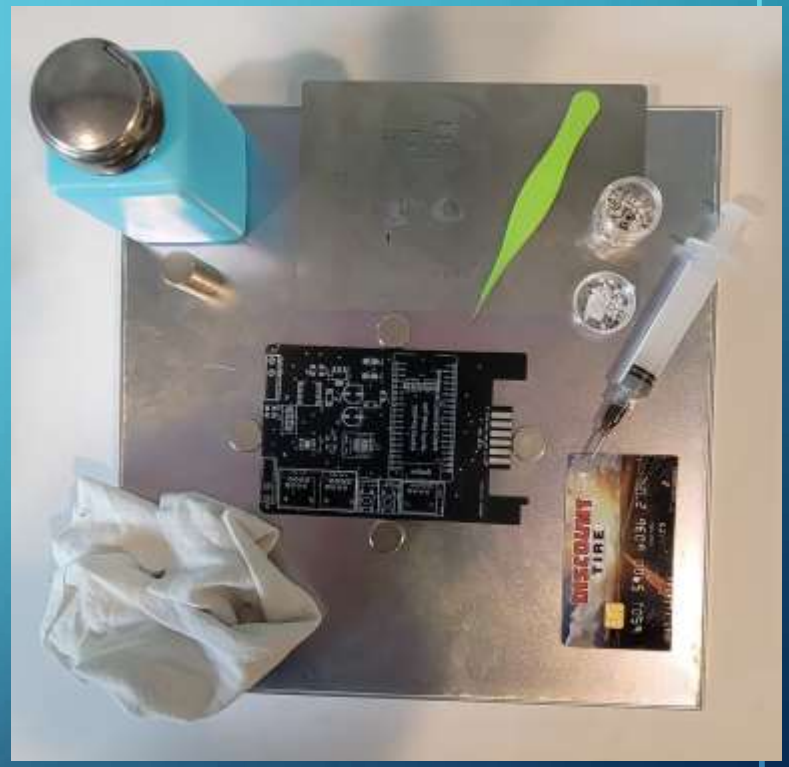
- JLCPCB ($) - https://jlcpcb.com/
- PCBWay ($$) - https://www.pcbway.com/
- OSH Park ($$) - https://oshpark.com/
- Others ? ($$$)

SMD SOLDERING
REFLOW OVERVIEW

- Solder Reflow
  - Process of using solder paste to solder SMD components on a PCB
  - PCB is heated until solder melts, bonding SMD to board
  - Fast, accurate and efficient when using an oven to solder multiple SMD components (to one side of PCB)
  - Use Hot Air Rework Station to solder individual components, including bottom of PCB
- PCB
  - Use 1206 SMD variant of resistors and capacitors (largest, easy placement)
  - Run traces straight out from pad (avoids clearance issue between pads)
- Stencil
  - Fastest and most accurate method of applying solder paste to PCB
  - PCB Stencil is flat sheet metal with cutouts matching solder pads for SMD components
  - Order Stencil with PCB
    - Costs: $7 + $8 shipping (JLCPCB)
    - Note: customize size to minimize shipping costs
  - Stencil insures correct amount of solder applied accurately on solder pad
  - Lay stencil on PCB, spread solder paste, remove stencil

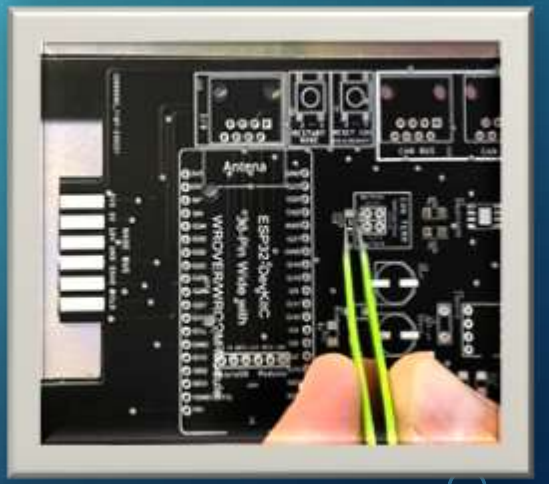
SMD SOLDERING
REFLOW SUPPLIES

- Cleaning PCB and Stencil
  - Alcohol dispenser and cloth
- PCB and stencil placement
  - Metal work area (for use with magnets)
  - Rare earth magnets (15mm x 3mm) (Amazon - $10)
  - Magnets used hold PCB and stencil in place
- Component Placement
  - Solder paste – low temp syringe (Amazon - $12)
  - Fine point non-magnetic tweezer
  - Component Organizer (Hobby Lobby - $4)
  - Old credit card to spread paste



SMD SOLDERING
REFLOW PREP

1. Place stencil, align and hold with magnets
   - Round 15mm x 3mm work best, easy to hold and strong
   - Stack PCB(s) to get correct height (~ 3mm)
   - Magnets will flip into place, aligned with magnet beneath
   - Tip: slide magnet to edge of metal plate to grab and remove
2. Place paste near each of the stencil's SMD cutouts
3. Spread paste into cutouts and fill void
4. Remove stencil and clean up with alcohol
5. Place components and tap into paste
   - Use fine point, non-magnetic tweezer
6. Place into counter top over to reflow
   - Use convection setting, 4 minutes @ 300
   - Note: temp is based on solder being used
   - PCB board may darken from heat
7. Let PCB(s) cool in oven with door open



FRITZING – LICENSING OF DESIGNS

- PCB, schematic, and breadboard designs are made up of parts (components)
  - These parts are available under the Creative Commons Attribution-ShareAlike 3.0 Unported License (CC BY-SA 3.0).
  - This includes parts you modify (via Fritzing Parts Editor)
- To share your designs, this license requires you to:
  1. Provide attribution:
     1. Give appropriate credit to the original creator of the work
     2. Provide a link to the license
     3. Indicate if changes were made
     4. Example from Adafruit's Fritzing-Library README.md:
        1. Subparts Art Attribution
        2. MicroSD Card holder, ESP-12E module, and Bluefruit LE (MDBT40) art is from PigHixx's lovely diagrams! The pighixx site may not exist anymore but we still want to give attribution!
  2. Distribute your derivative work under the same CC BY-SA 3.0 license as the original work
     1. Example: Adafruit Fritzing-Library license.txt file for Fritzing parts distributed via GitHub for use within Fritzing
  3. Include a link to the CC BY-SA 3.0 license with the distribution of your design (link
- Parts that you create and use, are included in the sketch's *.fzz file
- Published circuits and diagrams generated by Fritzing requires providing 'credit' to Fritzing.
  - Example: "this image was created with Fritzing."
- No Additional Restrictions: You cannot apply legal terms or technological measures that legally restrict others from doing anything the license permits.
- No Endorsement: You must not imply that the original creator of the work endorses you or your use of the work.
- No Warranty: The licensor offers no warranties regarding the licensed material and disclaims all liability for damages resulting from its use.

FRITZING PARTS EDITOR

- Select a part 'similar' to the part you want
  - For an IC, select a part with the same 'package' (i.e. form factor)
  - Search on 'Generic' to see a list of existing 'generic' parts
    - E.g. search shows SO16 package that can be edited for an IC using a SO16 package
  - Create the new version of the generic part
    - Right click on an existing part, select Edit Part (new parts editor)
    - On Connectors tab, change connector names / descriptions
    - On Metadata tab, change Title, etc.
    - Save the part
      - Select File -> Save as new part
      - Enter name for the part
  - Using the new part
    - Select MINE parts bin
    - New parts show at end of the parts bin
    - Drag and drop onto PCB
- To change the # or shape of connectors
  - From Parts Editor, select PCB view and import new SVG file
  - Use SVG editor, such as Inkscape to edit existing SVG file(s) for the part
- Parts that you create and use, are included in the sketch's *.fzz file

IC PACKAGE TYPES

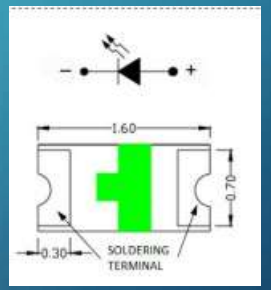
PCB TRACES – WIDTH VS AMPS

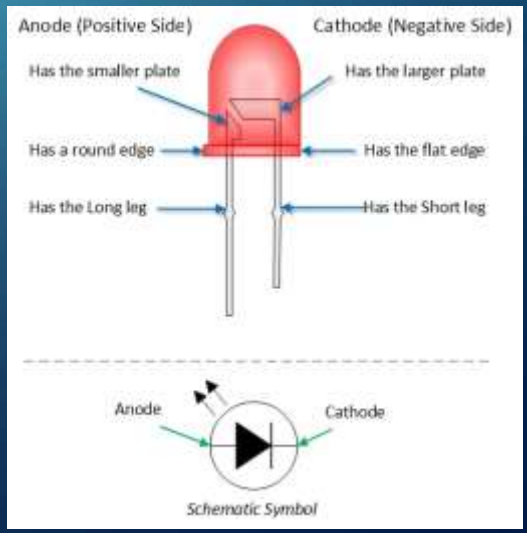
- PCB Trace Width Calculator
  - Conversion – 1 mil = 1/1000” = 0.001”
  - Trace Thickness = $1oz/ft^2$
  - Trace Length: doesn’t affect current, just V)

Trace Width (mil)	Fritzing Description	MAX Current (Amp)
8	Super Fine	0.75
12	Extra Thin	1.0
16	Fine	1.25
24	Standard	1.70
32	Thick	2.0
48	Extra Thick	2.75

CONTROLLING LED CURRENT VS BRIGHTNESS

- LED Resistor Calculator
- 20mA is recommended MAX current for an LED
- Red/Yellow/Green LEDs have similar characteristics
  - based forward voltage being similar (2V / 2.1V / 2.1)\
- White LED much different
  - 3.6V forward voltage
- Brightness (MCD) is linear below 20mA
  - For example, 10mA = 50% brightness
- Connecting LEDs in 'series' will result in a voltage drop
  - For example, (2) Red LEDs in series:
    - 1V of forward voltage
    - For 100% brightness (20ma), reduces resistor from 150 ohm to 50 ohm
- Connecting LEDs in 'parallel' will result in no voltage change, but increase in current (additive)
  - For example, (2) Red LEDs in parallel:
    - 150 ohm resistor would be 50% brightness, using 40mA.





Volts	LED Color	Resistor (ohm)	Brightness (%)	Current (mA)
5	Red/Yellow/Green	1500 ->150	10 -> 100	2 -> 20
5	White	700 -> 70	10 -> 100	2 -> 20
12	Red/Yellow/Green	10,000 -> 1,000	10 -> 100	2 -> 20
12	White	4200 -> 420	10 -> 100	2 -> 20

SMD SIZES

Standard Size	Metric	Inches
2920	7.4 x 5.1	0.29 x 0.20
2725	6.9 x 6.3	0.27 x 0.25
2512	6.3 x 3.2	0.25 x 0.125
2010	5.0 x 2.5	0.20 x 0.10
1825	4.5 x 6.4	0.18 x 0.25
1812	4.6 x 3.0	0.18 x 0.125
1806	4.5 x 1.6	0.18 x 0.06
1210	3.2 x 2.5	0.125 x 0.10
1206	3.0 x 1.5	0.12 x 0.06
1008	2.5 x 2.0	0.10 x 0.08
0805	2.0 x 1.3	0.08 x 0.05
0603	1.5 x 0.8	0.06 x 0.03
0402	1.0 x 0.5	0.04 x 0.02
0201	0.6 x 0.3	0.02 x 0.01
01005	0.4 x 0.2	0.016 x 0.008