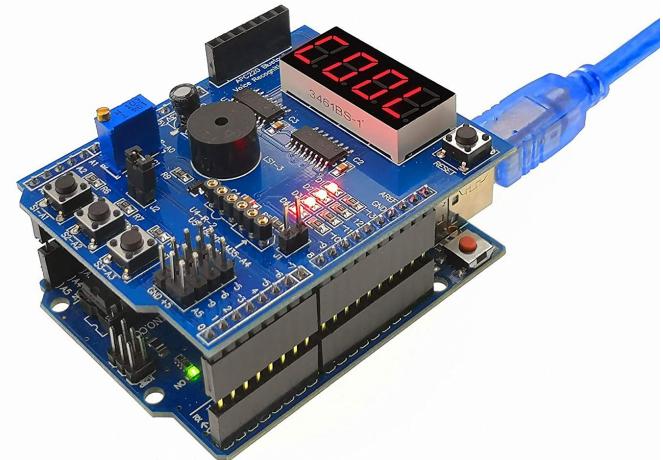
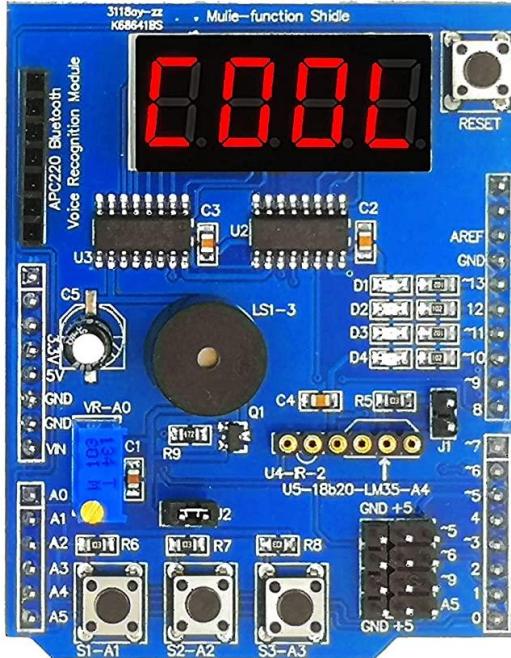
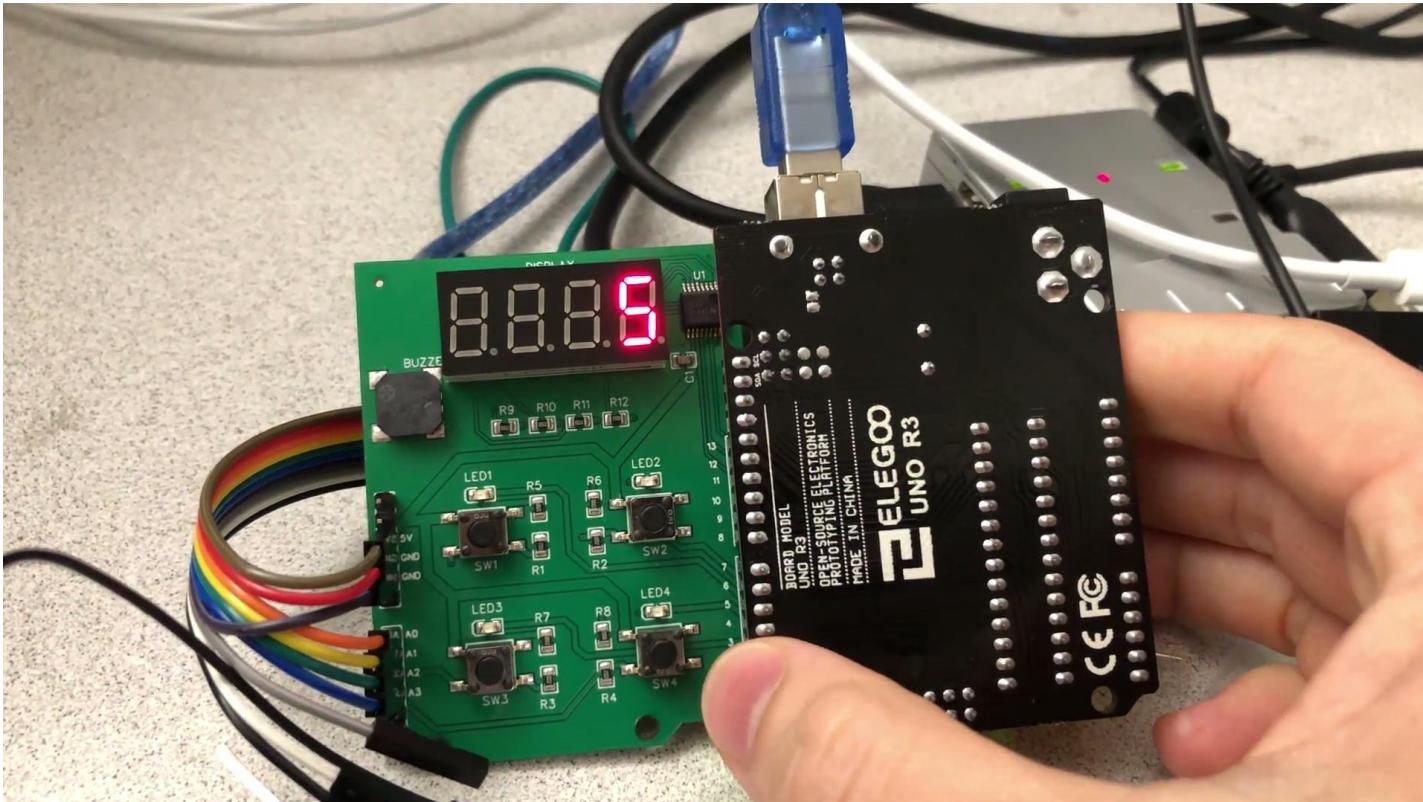


Extending Arduino Functionality via Printed Circuit Board (PCB)



EEEEE UNO R3 &
Multi-Function Shield Starter
kit on Amazon [here](#)

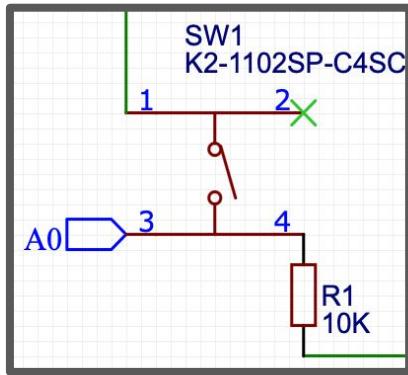
Joe's PCB: Whack-a-mole video



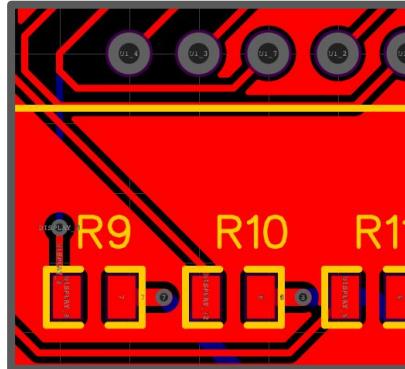
PCB Design Process



Idea



Schematic



Layout



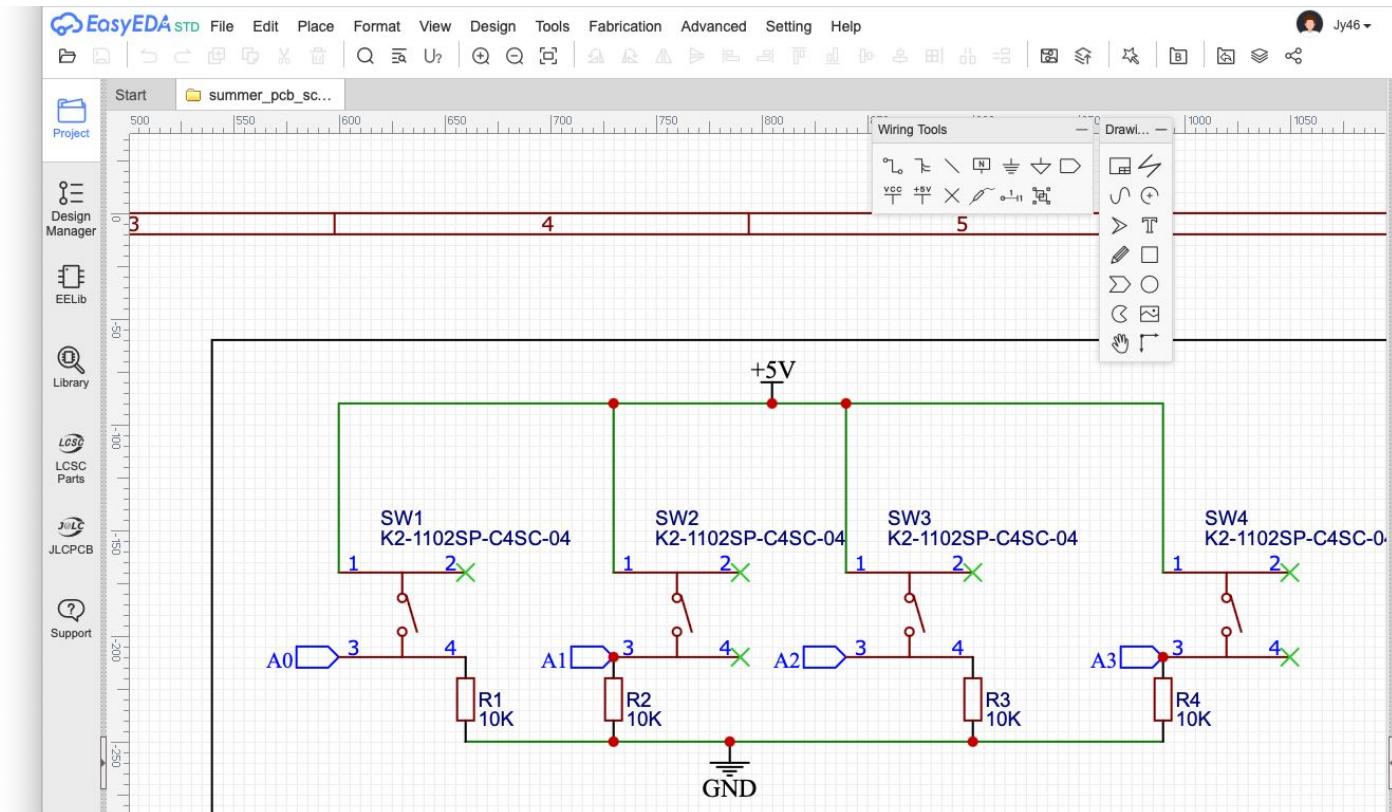
Manufacture

PCB Design Programs

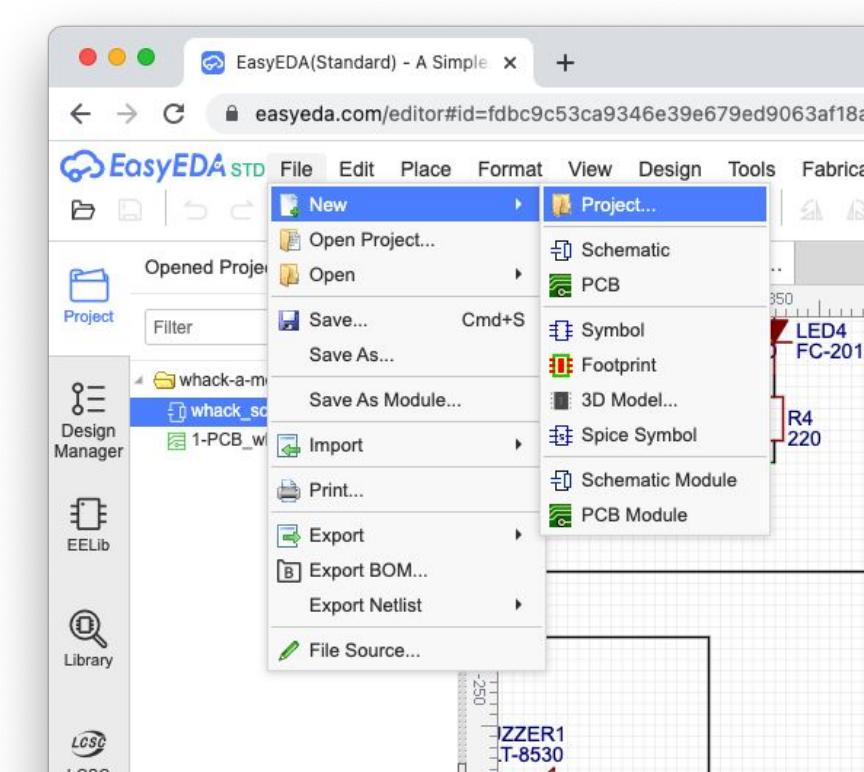
- EAGLE (free student license, part of AutoDesk)
- MultiSim/UltiBoard (National Instruments)
- Allegro (Professional grade)
- KiCad (free)
- **EasyEDA (free, linked to a manufacturer)**
- Altium (Professional grade)

Can simulate schematic

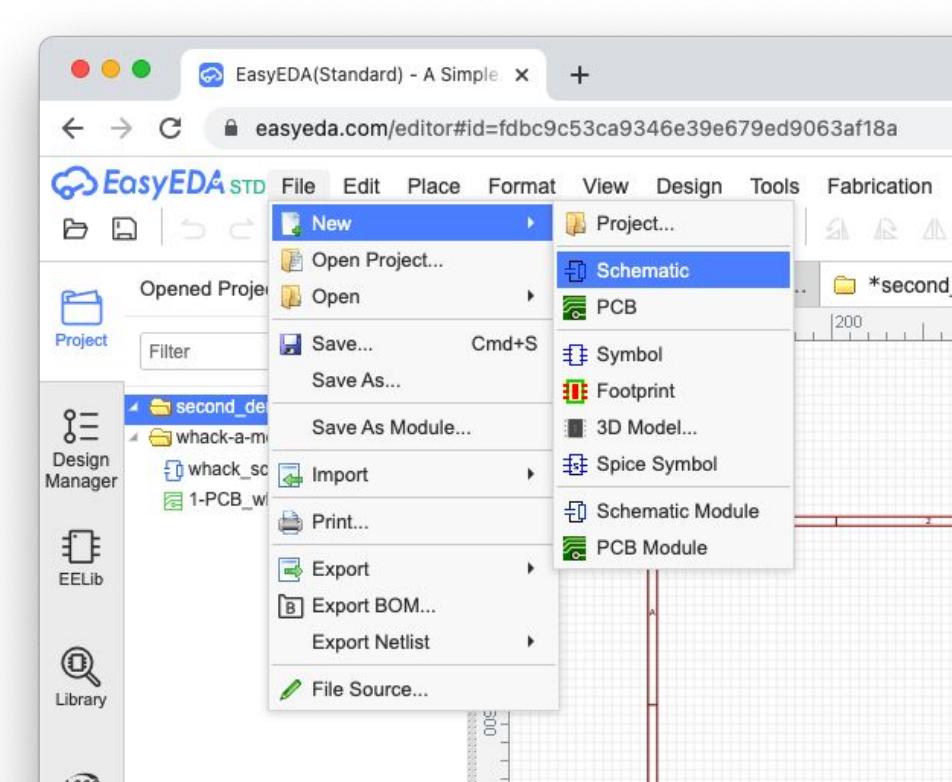
Graphical Schematic Editing



Make New Project in EasyEDA

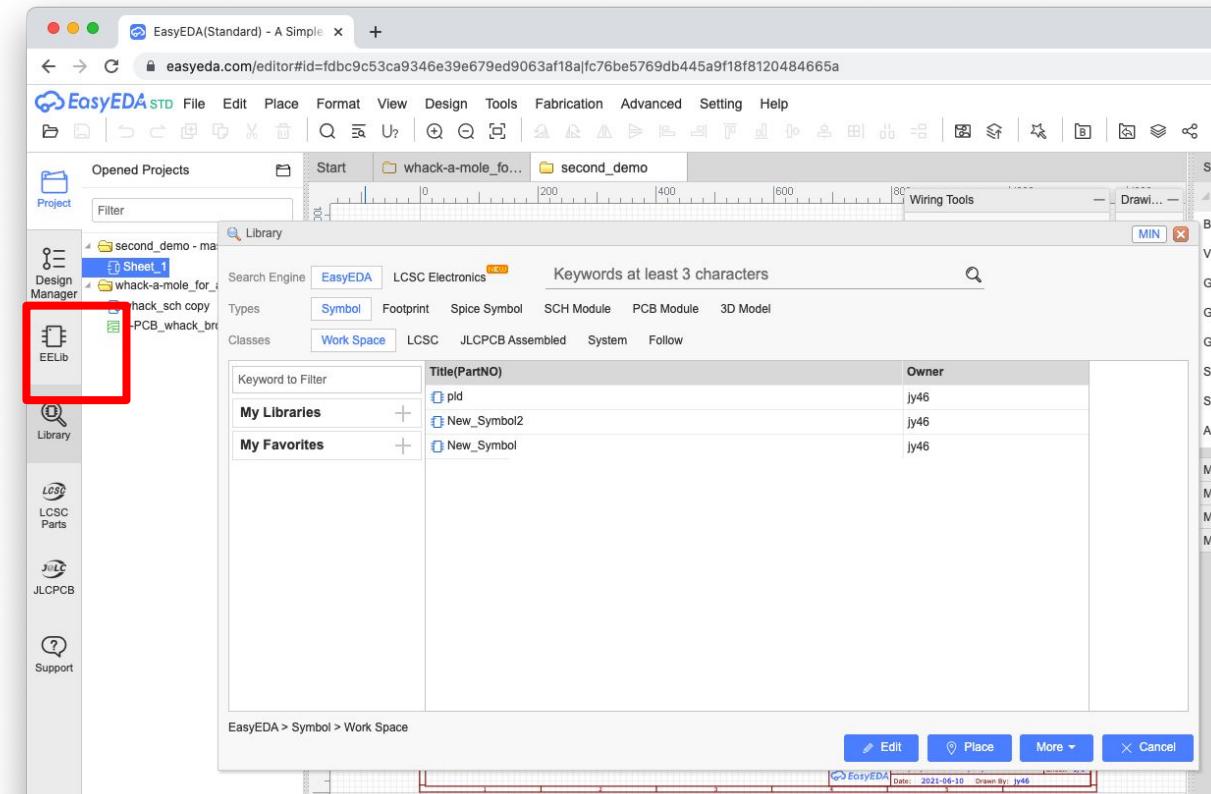


Make New Schematic in EasyEDA



Be sure to save it as whatever you like :)

Add the Parts We Need: Start by clicking Library



Type in Keyword for Button Model: K2-1102SP-C4SC-04

The screenshot shows the EasyEDA Library search interface. The search bar at the top contains the text "K2-1102SP-C4SC-04", which is highlighted with a red box. Below the search bar, there are tabs for "Search Engine" (EasyEDA), "LCSC Electronics" (NEW), "Symbol", "Footprint", "Spice Symbol", "SCH Module", "PCB Module", and "3D Model". The "Symbol" tab is selected. There are also tabs for "Classes", "Work Space(0)", "LCSC(1)", and "JLCPCB Assembled(1)" (highlighted with a red box). Below these tabs, there is a "Keyword to Filter" input field and a sidebar with categories: Amplifiers, Analog ICs, Audio Products/M..., Capacitors, and Connectors. The main results table has columns: Title(PartNO), Footprint, JLCPCB Part Class, and Manufacturer. One result is listed: K2-1102SP-C4SC-04, Footprint: KEY-SMD_4P-L6.0-W6.0-P4.50-LS9.5-BL, JLCPCB Part Class: Extended Part, Manufacturer: HRO.

Title(PartNO)	Footprint	JLCPCB Part Class	Manufacturer
K2-1102SP-C4SC-04	SMT KEY-SMD_4P-L6.0-W6.0-P4.50-LS9.5-BL	Extended Part	HRO

Make Sure JLCPCB Assembled Selected

The screenshot shows the EasyEDA Library search interface. At the top, there is a search bar with the text "K2-1102SP-C4SC-04". Below the search bar, there are tabs for "Search Engine" (EasyEDA), "LCSC Electronics" (NEW), "Symbol", "Footprint", "Spice Symbol", "SCH Module", "PCB Module", and "3D Model". A red box highlights the "JLCPCB Assembled(1)" tab, which is currently selected. Below these tabs, there are buttons for "Classes", "Work Space(0)", "LCSC(1)", "System(0)", "Follow(0)", and "User Contributed(0)". On the left side, there is a sidebar with a "Keyword to Filter" input field and a list of categories: Amplifiers, Analog ICs, Audio Products/M..., Capacitors, and Connectors, each with a plus sign next to it. The main search results table has columns for "Title(PartNO)", "Footprint", "JLCPCB Part Class", and "Manufacture". The first result listed is "K2-1102SP-C4SC-04" with a footprint of "KEY-SMD_4P-L6.0-W6.0-P4.50-LS9.5-BL", categorized as "Extended Part" by JLCPCB, and manufactured by HRO.

Title(PartNO)	Footprint	JLCPCB Part Class	Manufacture
K2-1102SP-C4SC-04	SMT KEY-SMD_4P-L6.0-W6.0-P4.50-LS9.5-BL	Extended Part	HRO

Select Part

Screenshot of the EasyEDA library search interface showing the selection of a tactile switch part.

The search results for "K2-1102SP-C4SC-04" are displayed. The first result, "KEY-SMD_4P-L6.0-W6.0-P4.50-LS9.5-BL", is highlighted with a red box.

Title(PartNO)	Footprint	JLCPCB Part Class	Manufacturer
K2-1102SP-C4SC-04	KEY-SMD_4P-L6.0-W6.0-P4.50-LS9.5-BL	Extended Part	HRO

Details for the selected part:

- Price: \$0.0479
- LCSC Part#: C127509
- Stock: 313480 (275099 for JLCPCB SMT Service)
- Minimum: 10
- Distributor: LCSC

Buttons at the bottom: Edit, Place, More, Cancel.

Bottom right corner: Date: 2021-06-10 Drawn By: jy46

Click Place

Screenshot of the EasyEDA library search interface showing the "Place" button highlighted.

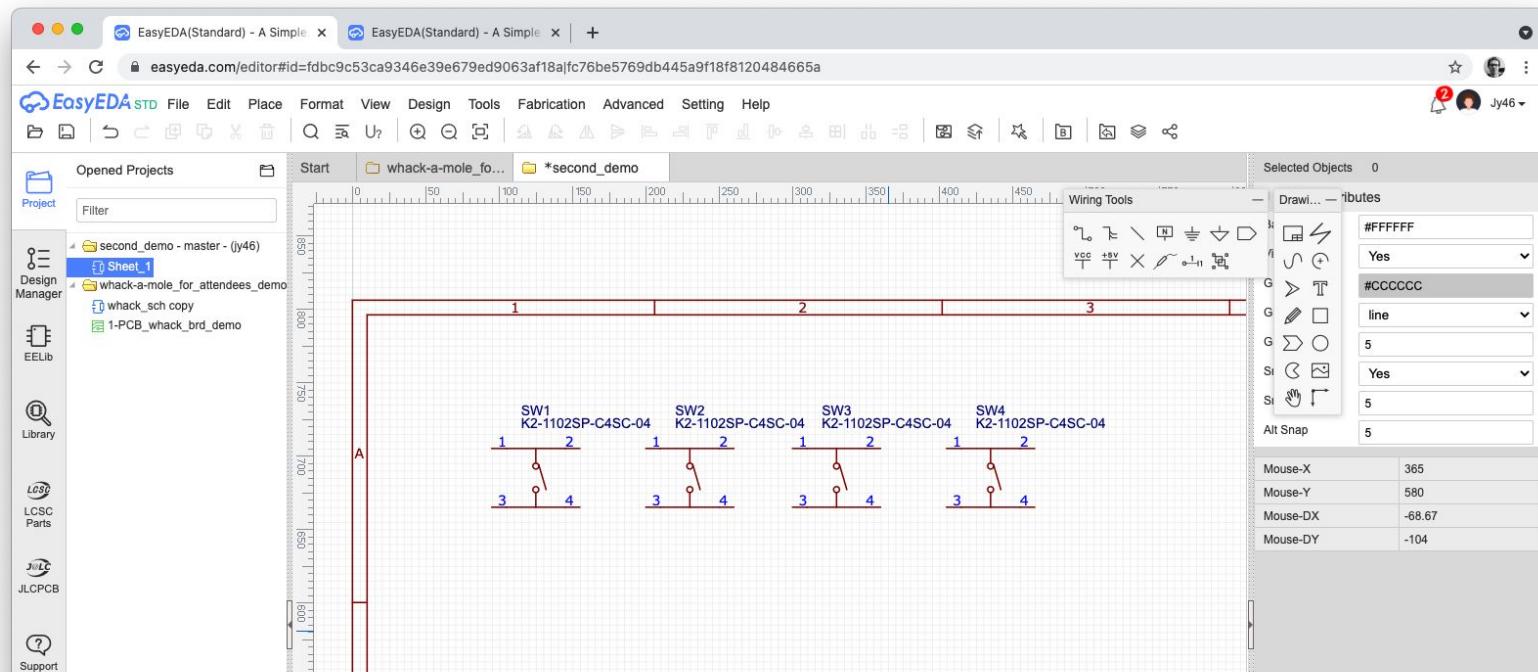
The search results for part K2-1102SP-C4SC-04 are displayed. The part is listed under the "JLCPCB Assembled(1)" category. The "Place" button at the bottom of the search results is highlighted with a red box.

Key details from the screenshot:

- Search Engine:** EasyEDA
- Part Number:** K2-1102SP-C4SC-04
- Footprint:** KEY-SMD_4P-L6.0-W6.0-P4.50-LS9.5-BL
- JLCPCB Part Class:** Extended Part
- Manufacturer:** HRO
- Symbol:** Tactile Switch symbol showing four pins labeled 1 through 4.
- Image:** 3D model of the tactile switch component.
- Price:** \$0.0479
- Stock:** 313480 (275099 for JLCPCB SMT Service)
- Minimum:** 10
- Distributor:** LCSC

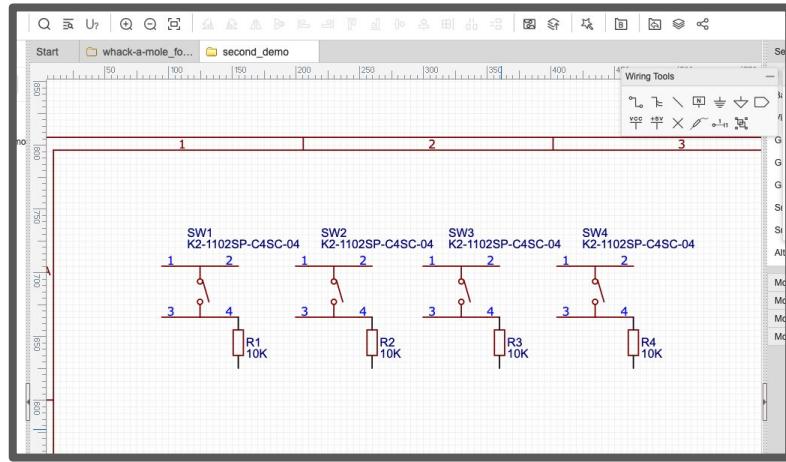
Bottom navigation bar buttons: Edit, Place (highlighted), More, Cancel.

Place 4 by Left Clicking 4 Times & Then Stop by Right Clicking



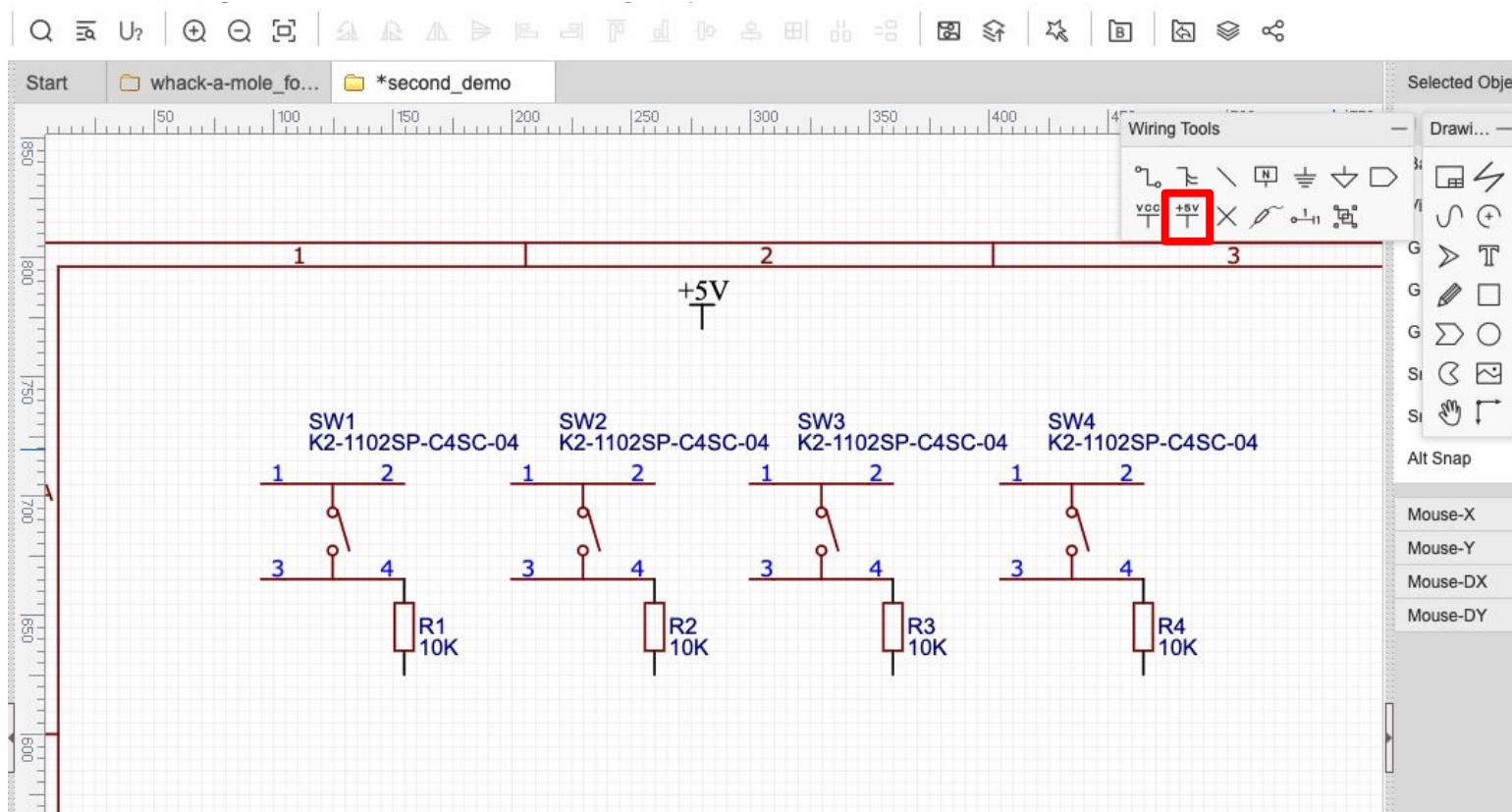
Click Library, Search & Place the “0805W8F1002T5E” Resistor

The screenshot shows the EasyEDA software interface. The main window displays a search result for the component "0805W8F1002T5E". The search results table includes columns for Title(PartNO), Footprint, JLCPBC Part Class, Resistance, and Manufacturer. The first result is "0805W8F1002T5E" with a footprint of R0805, basic part class, 10K resistance, and UniOhm manufacturer. The second result is "0805W8F1002T5E_C38522" with the same footprint and resistance but a different manufacturer. The left sidebar shows the Design Manager with a project named "Sheet_1" selected. The bottom status bar provides information about the component: "EasyEDA > Symbol > JLCPBC Assembled > Chip Resistor - Surface Mount > 0805W8F1002T5E", price "\$0.0030", part number "C17414", stock "1042200 (3441780 for JLCPBC SMT Service)", minimum quantity "100", and distributor "LCSC".

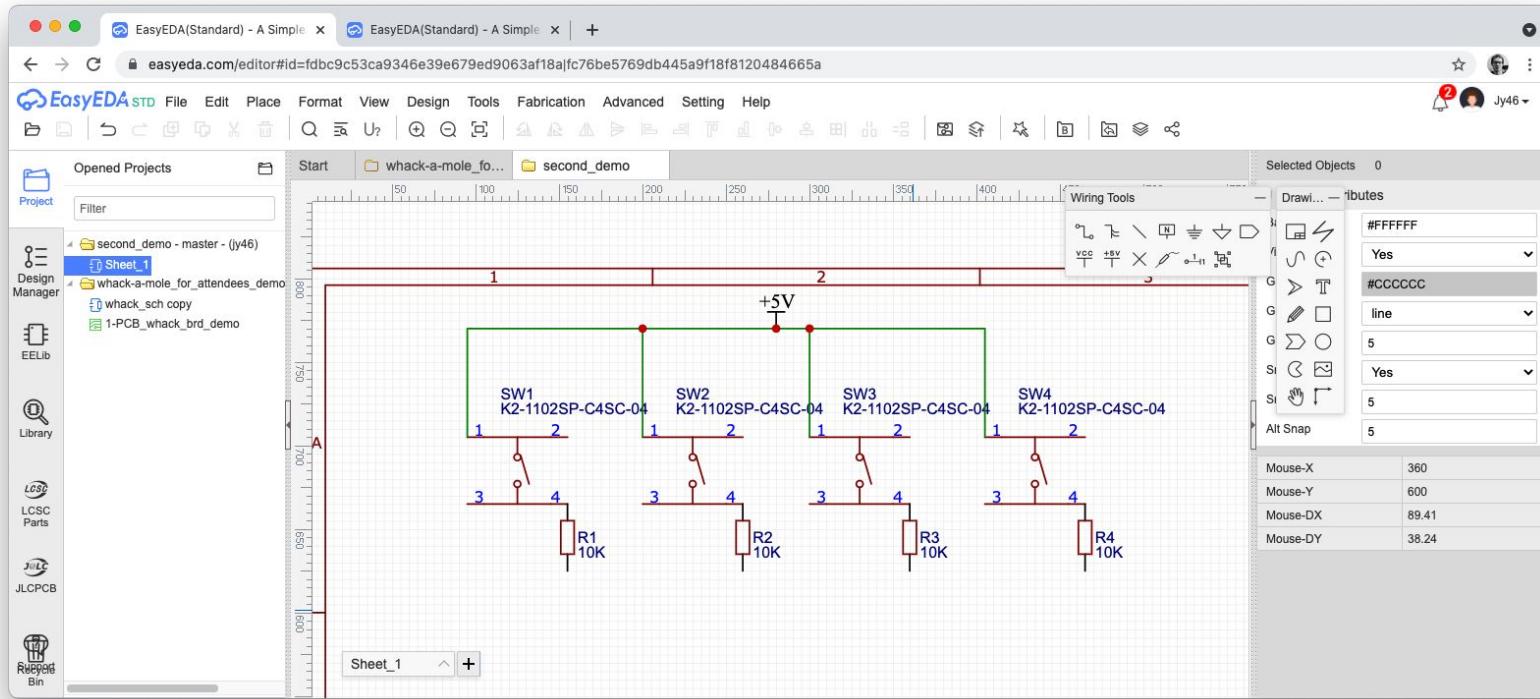


Type R to rotate the 10k pull-down resistor

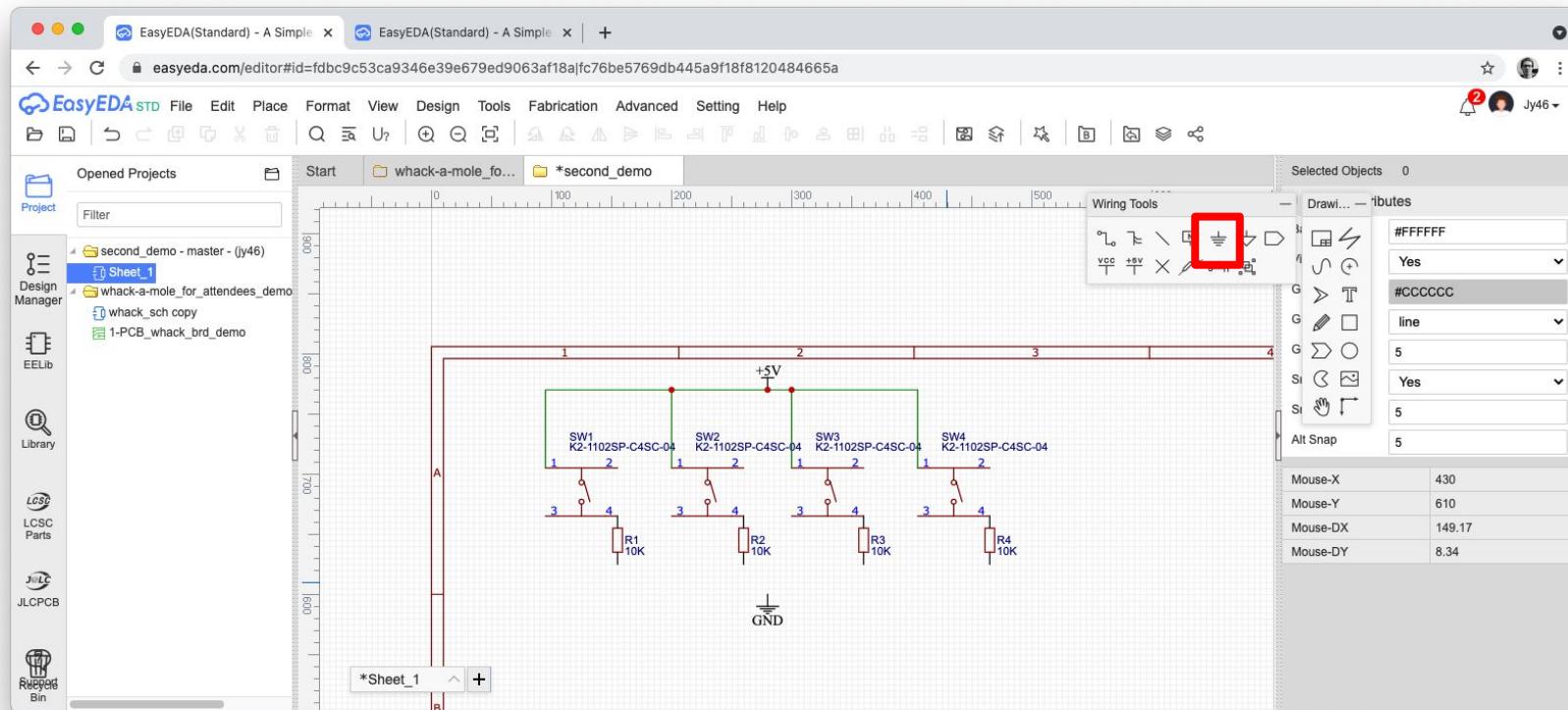
Place +5 V “Net Flag”



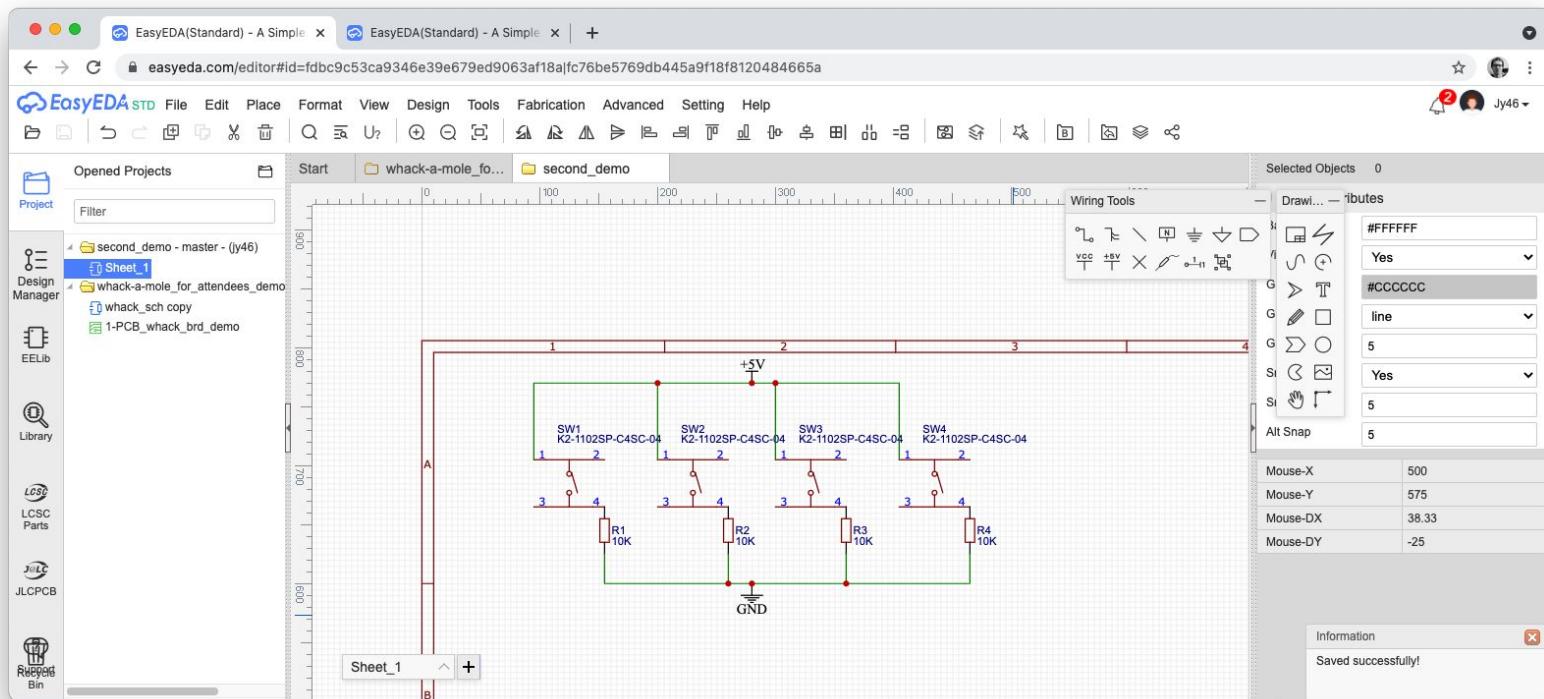
Connect Switches to +5 V via Wires (Type W or click wire symbol)



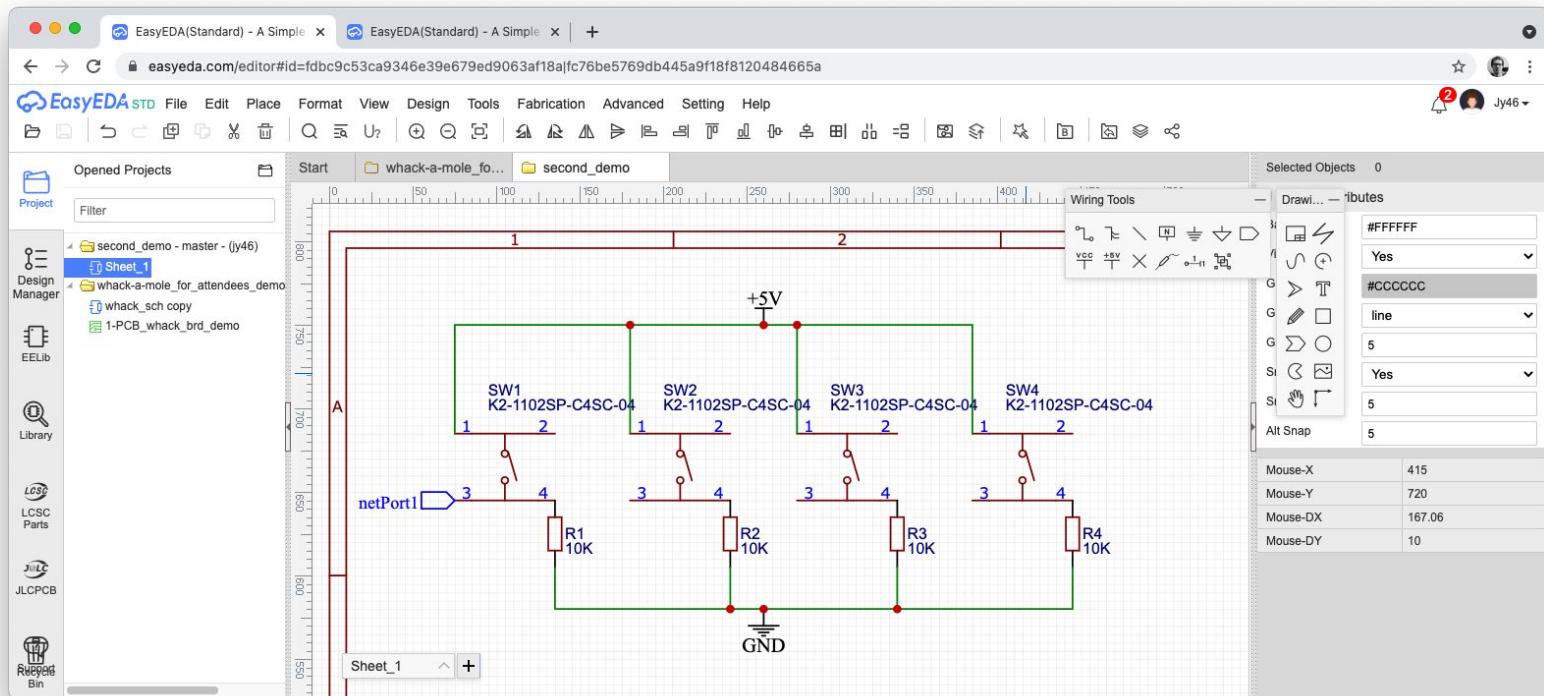
Place GND (ground) “Net Flag”



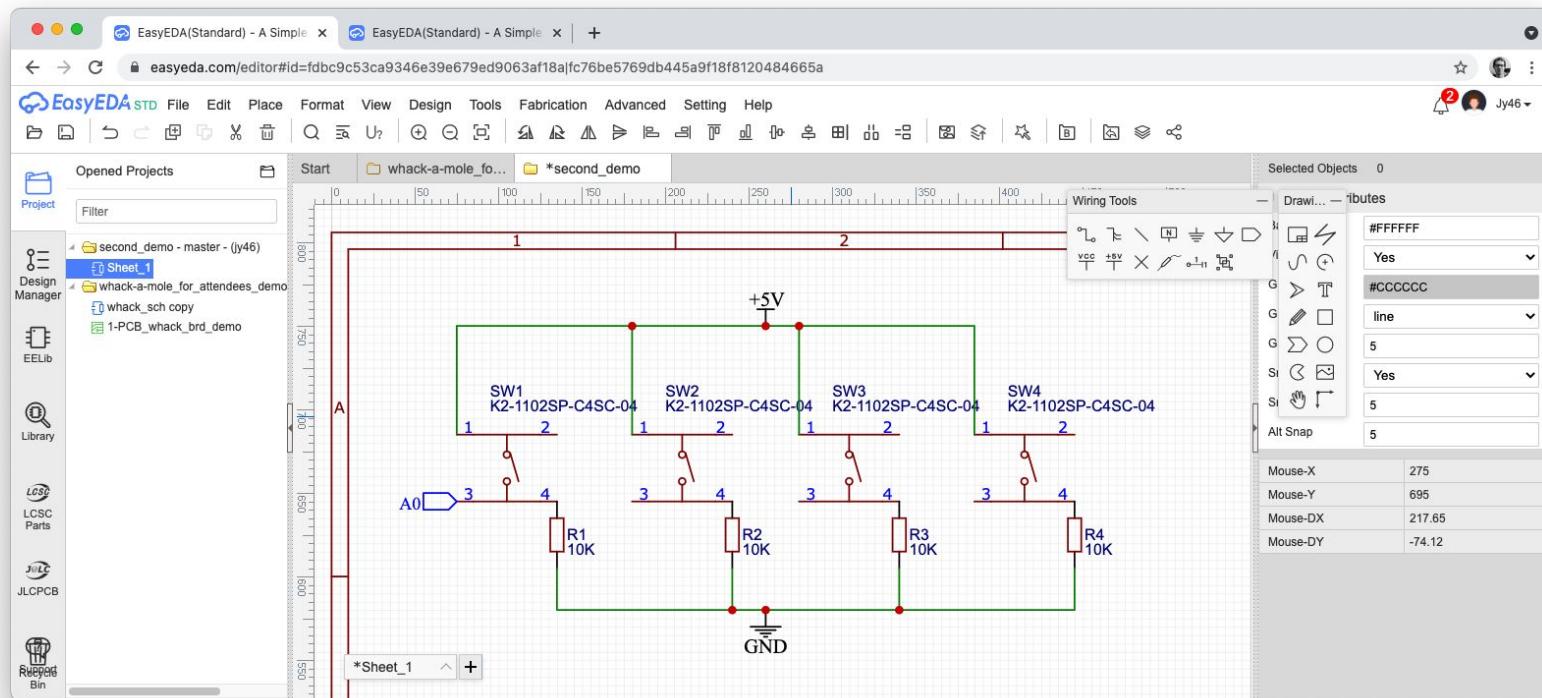
Connect Switches to GND via Wires



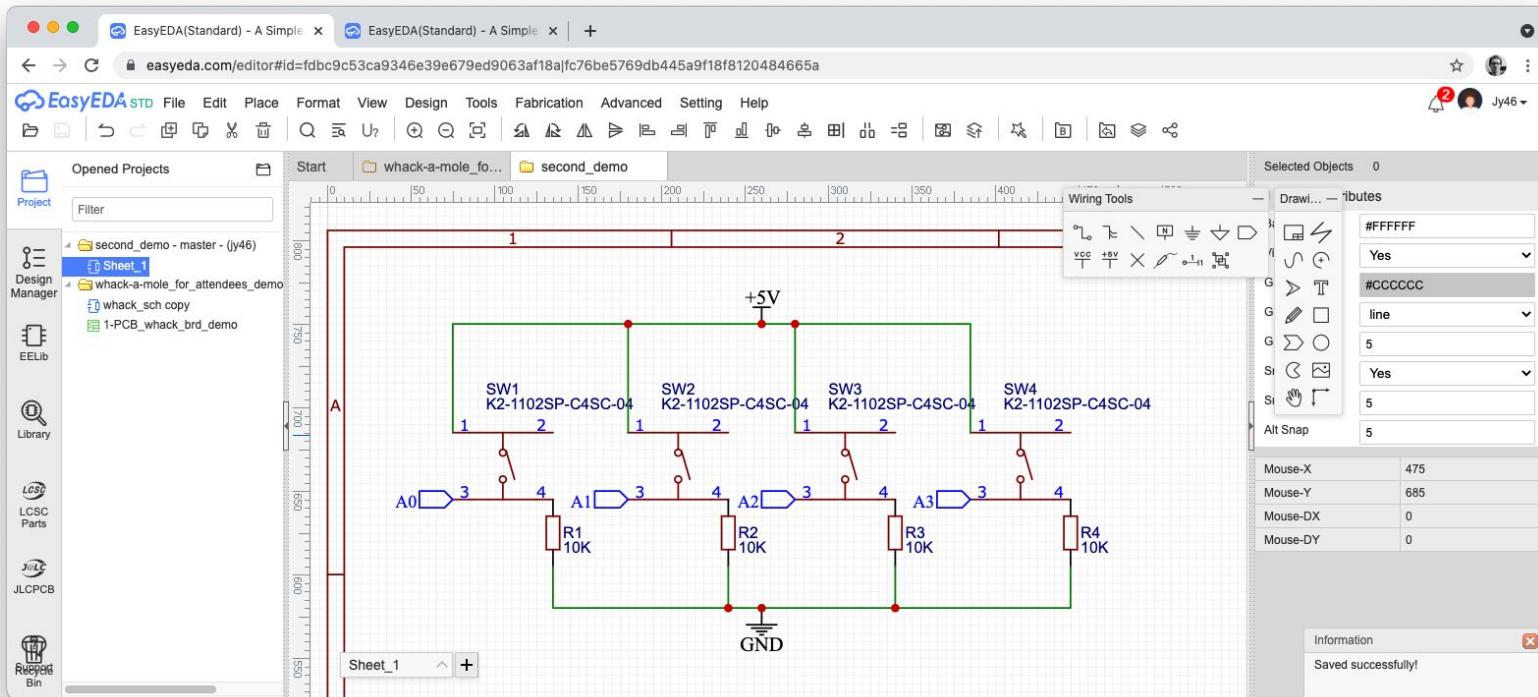
Add Net Port To Switch



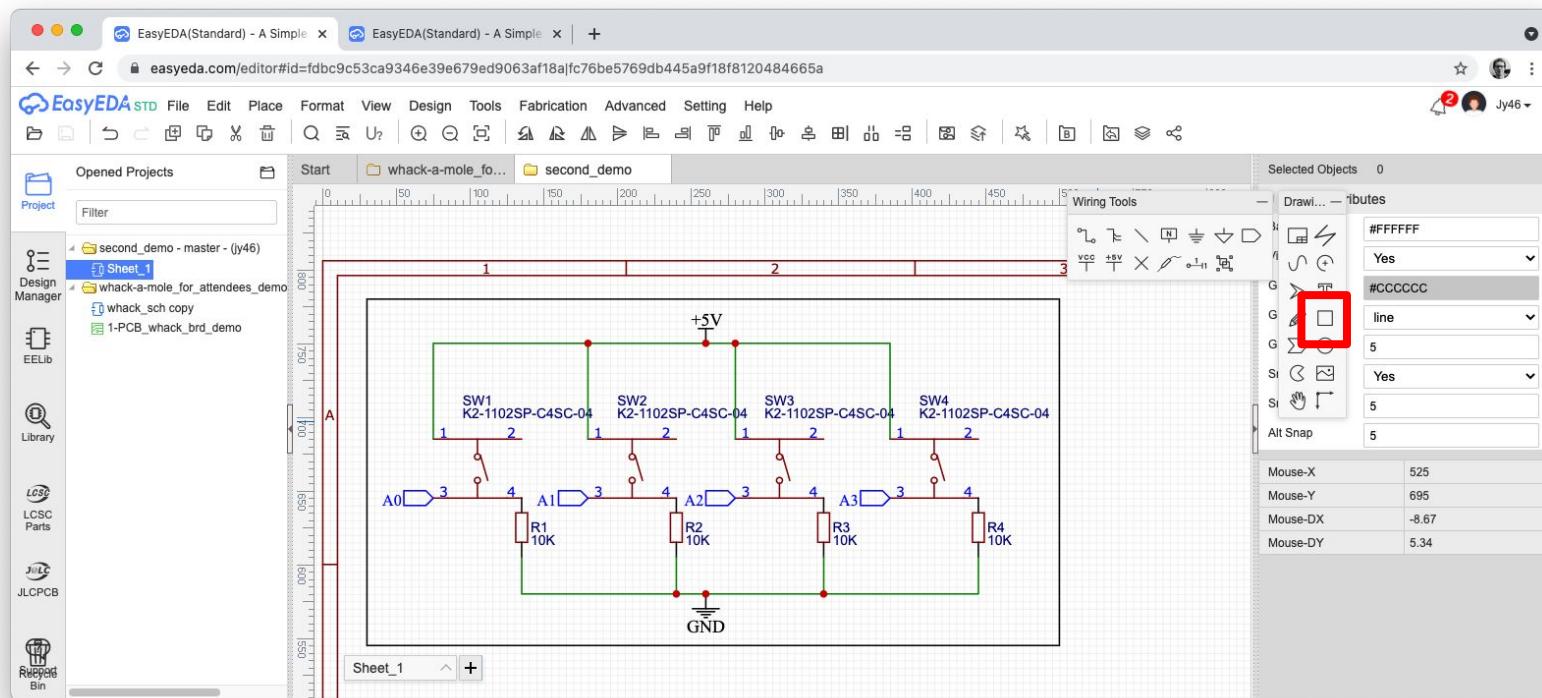
Rename Net Port to A0



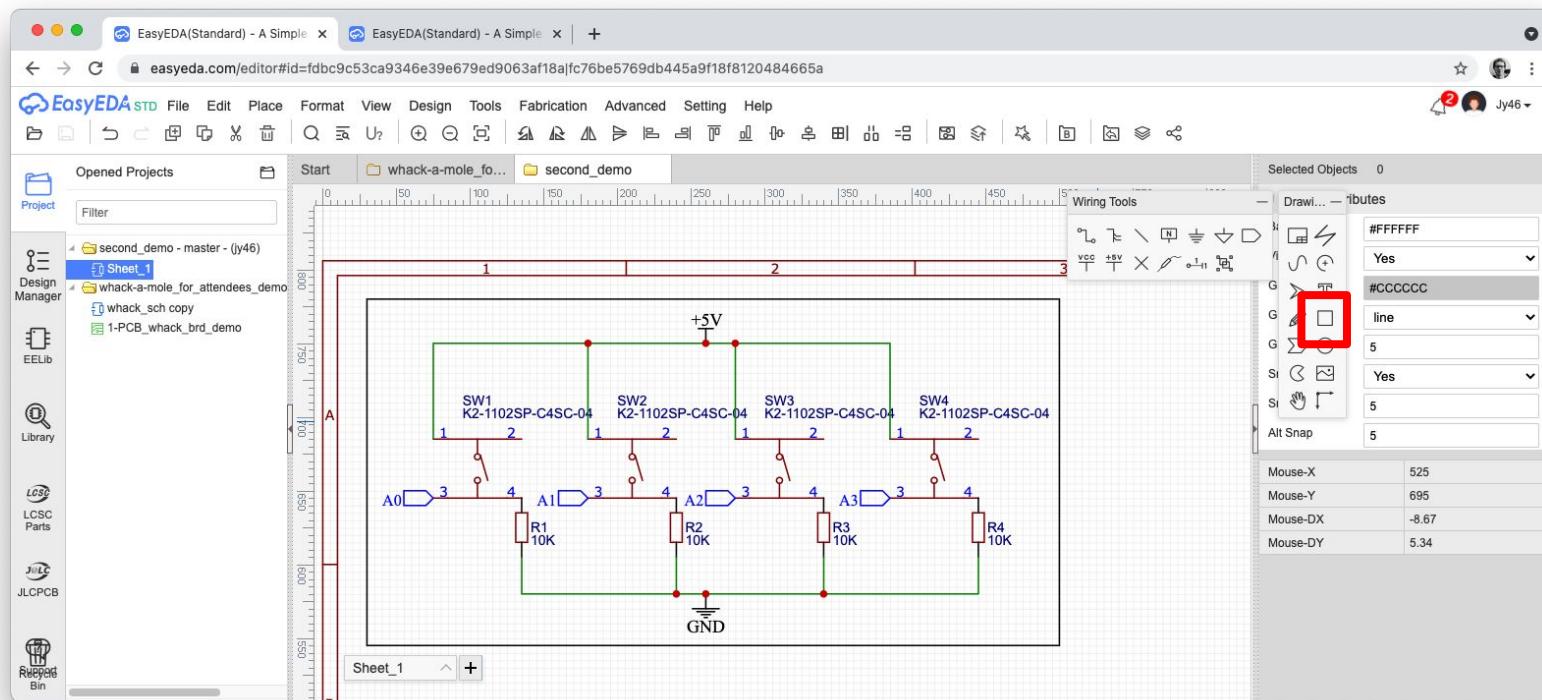
Copy & Paste Net Port to Other Switches - Rename to A1, A2, A3



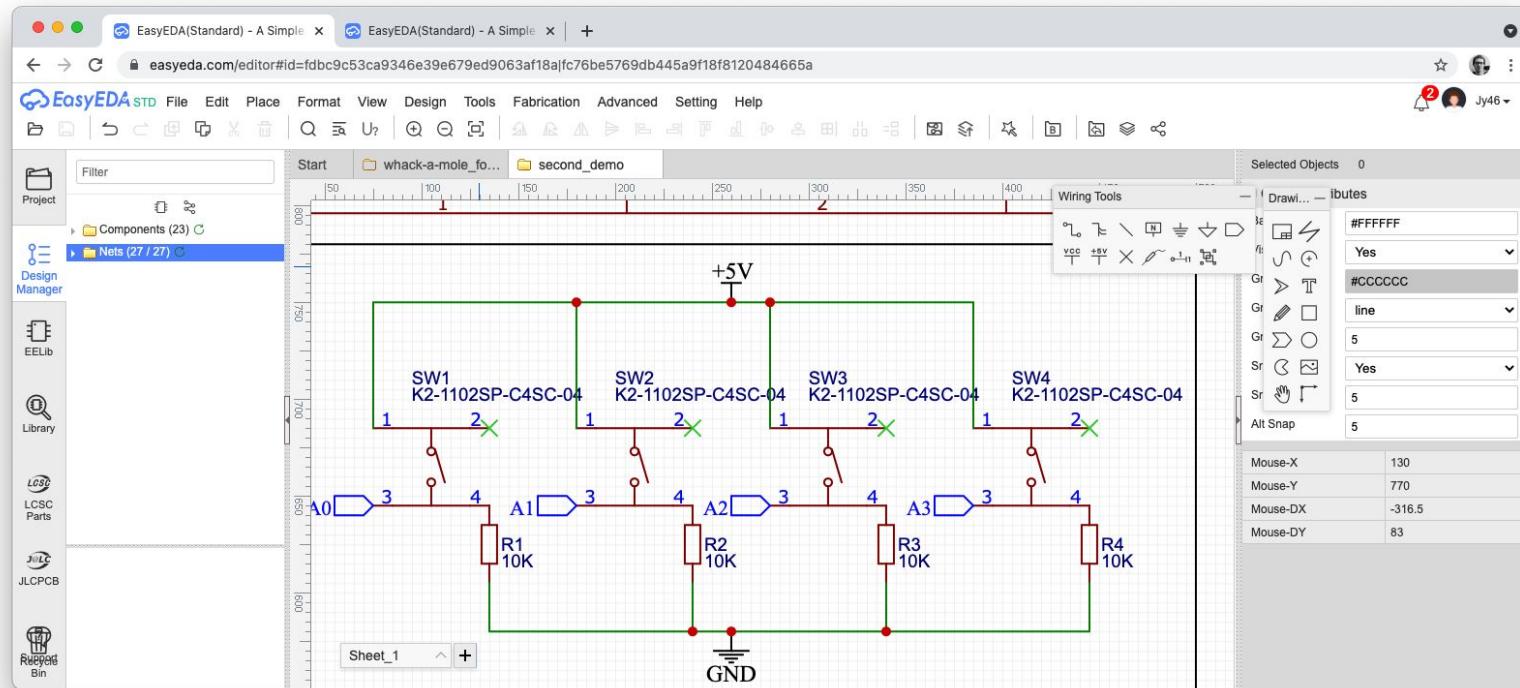
Draw Rectangle Around Diagram



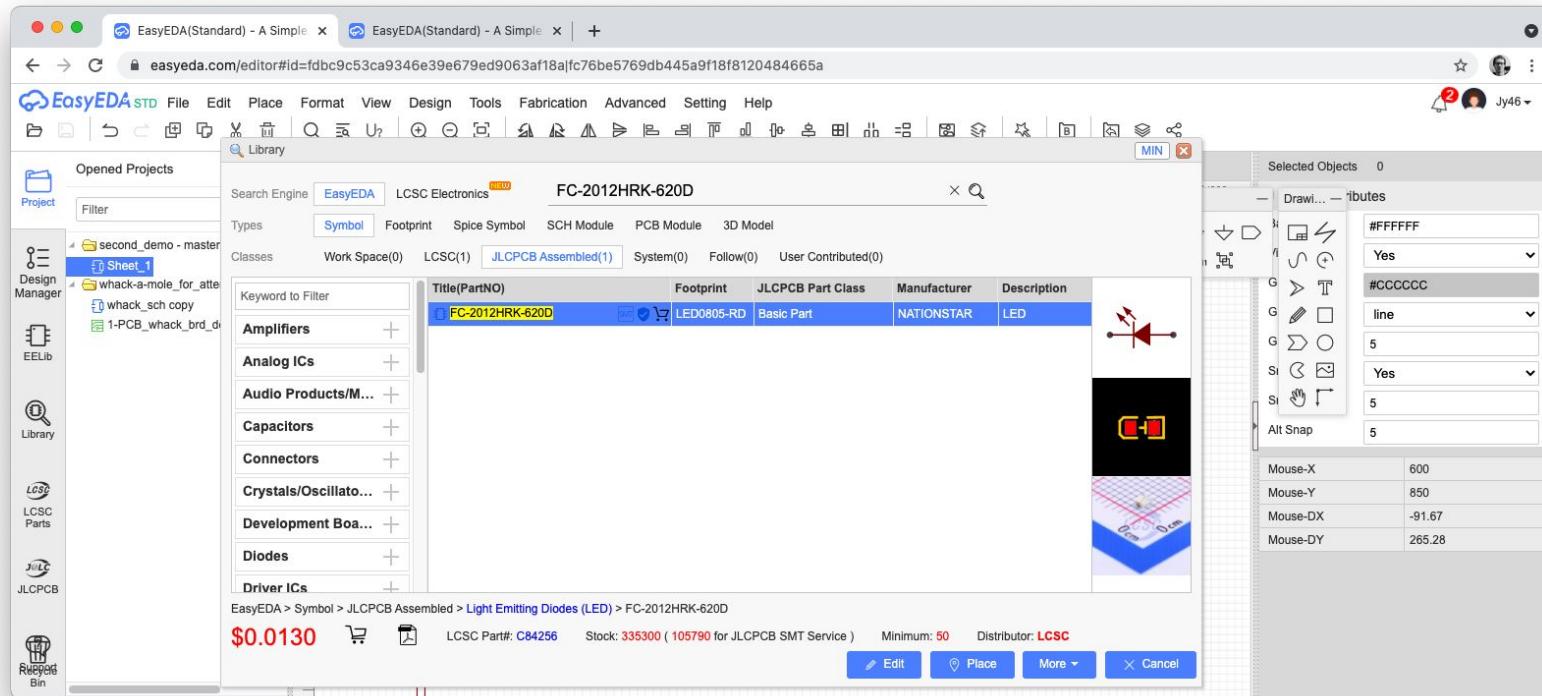
Draw Rectangle Around Diagram



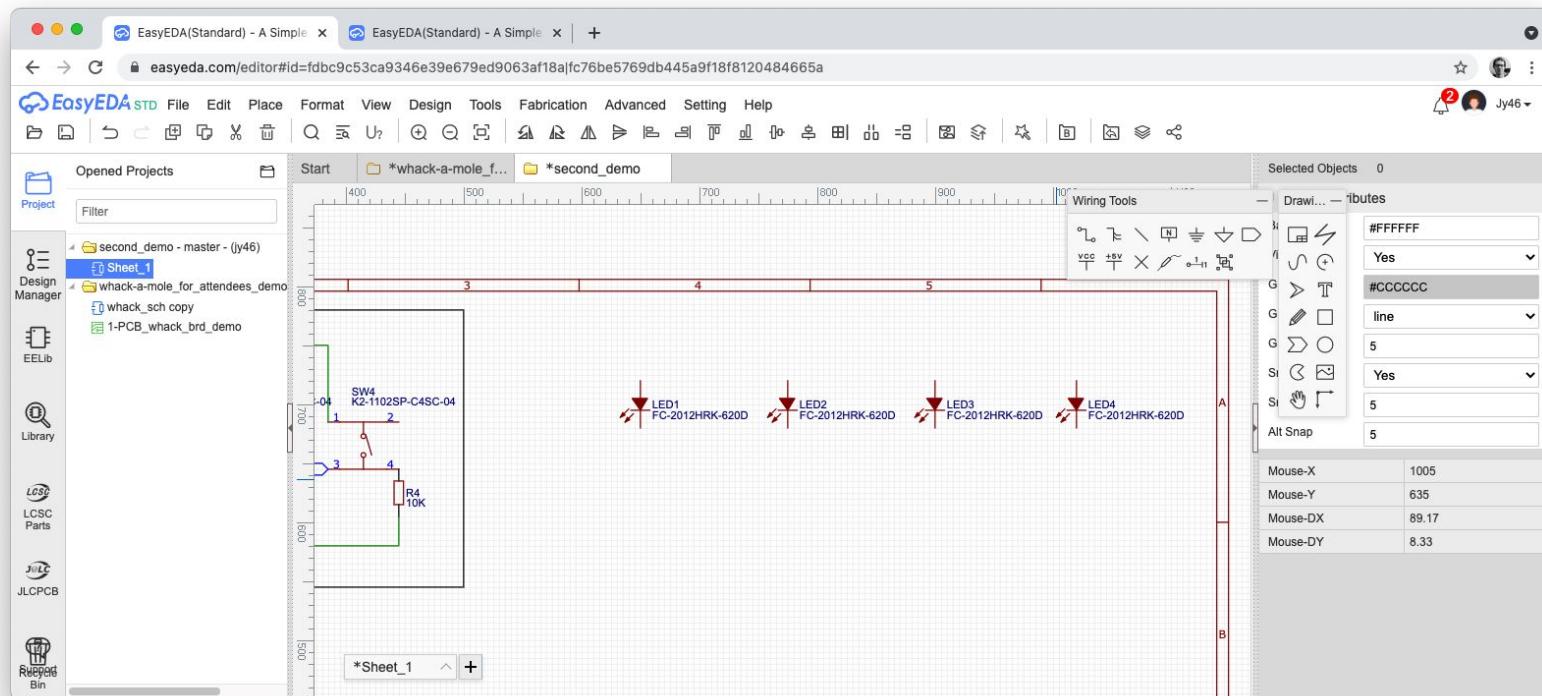
Also Add No Connect Flags to Switches



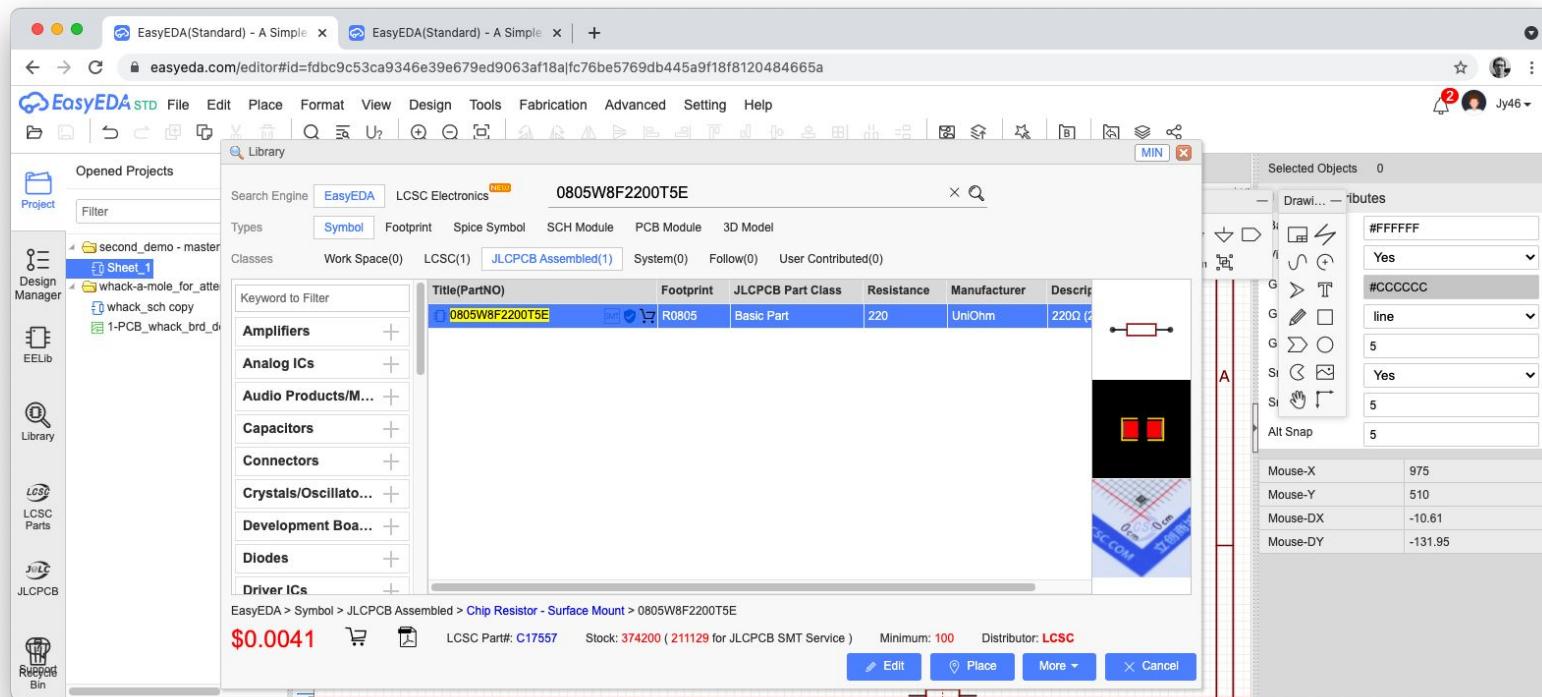
Go to Library and Grab LEDs: “FC-2012HRK-620D”



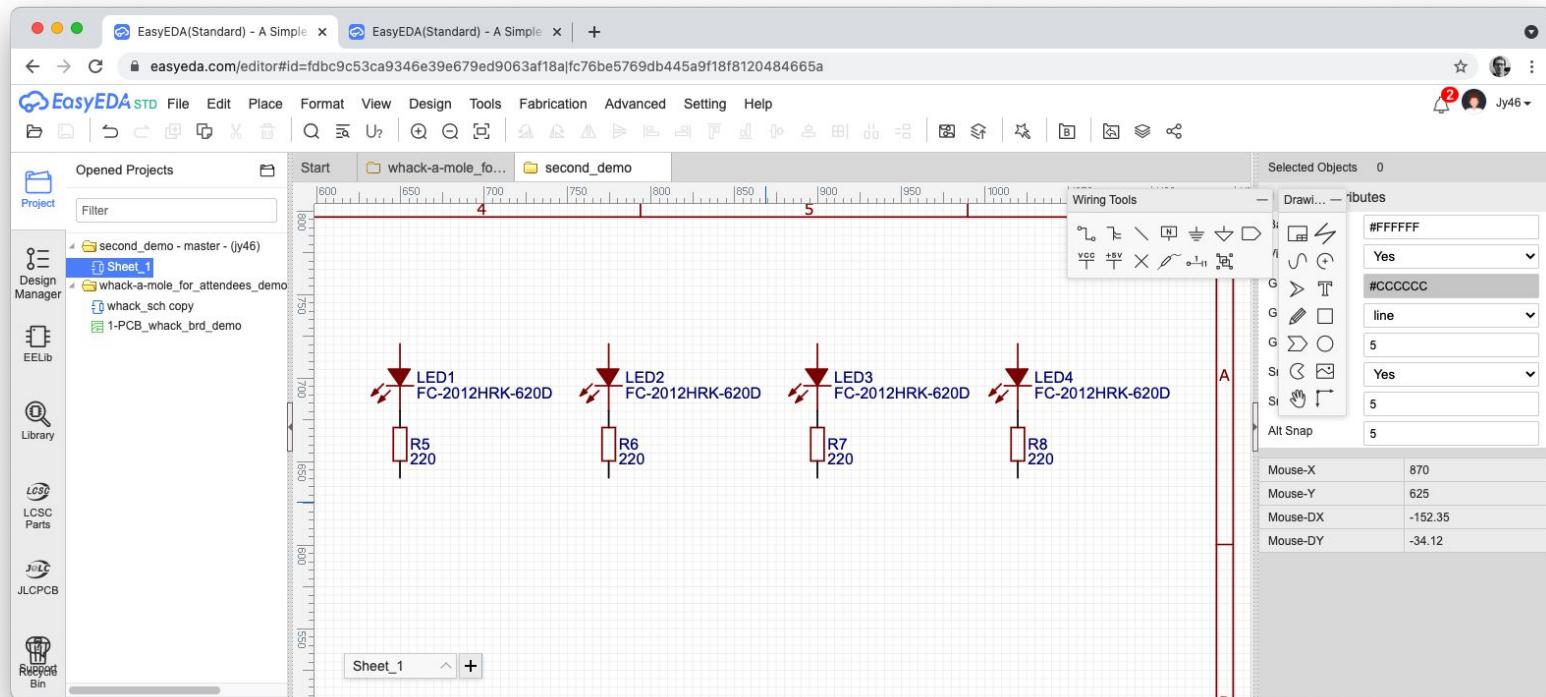
Place 4 LEDs



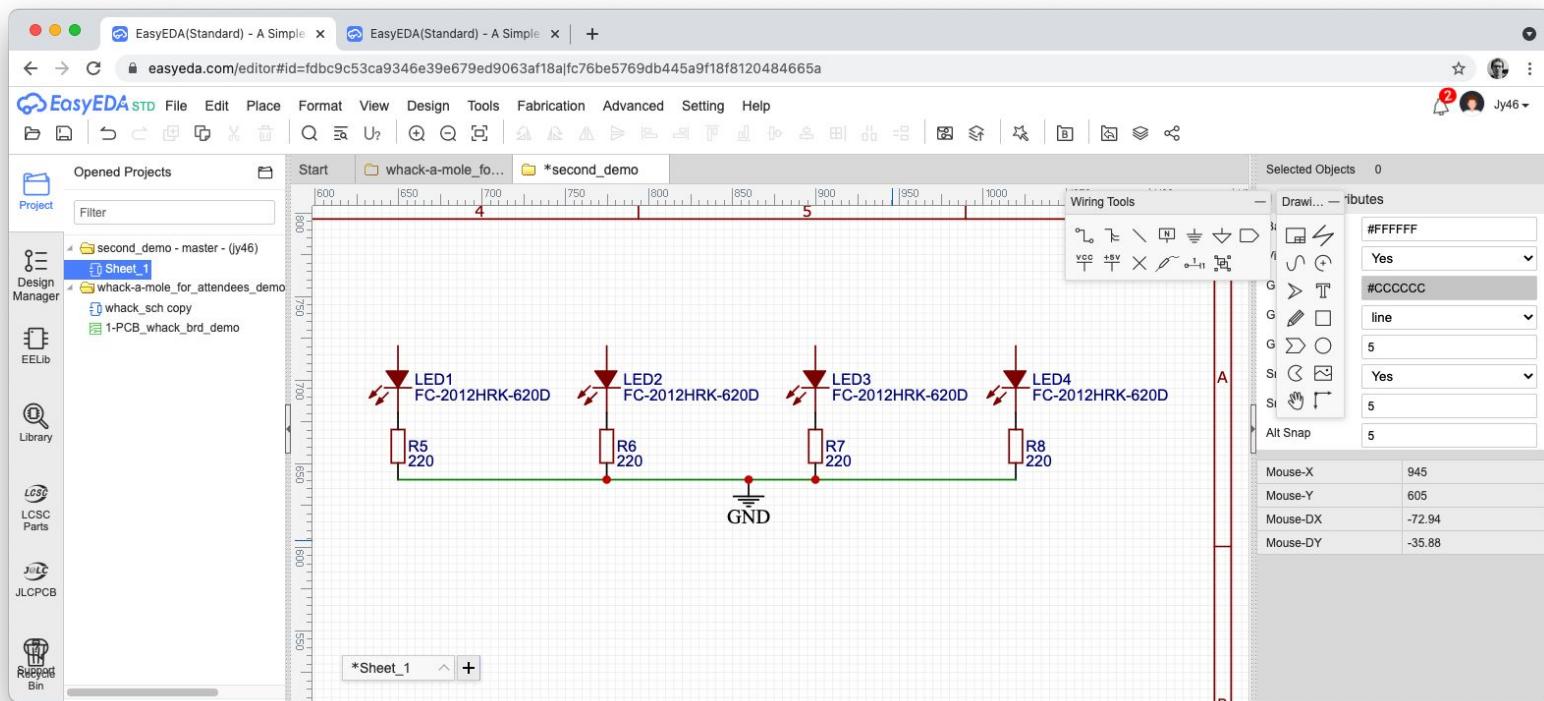
Go to Library & Grab 220 Ohm Resistors: “0805W8F2200T5E”



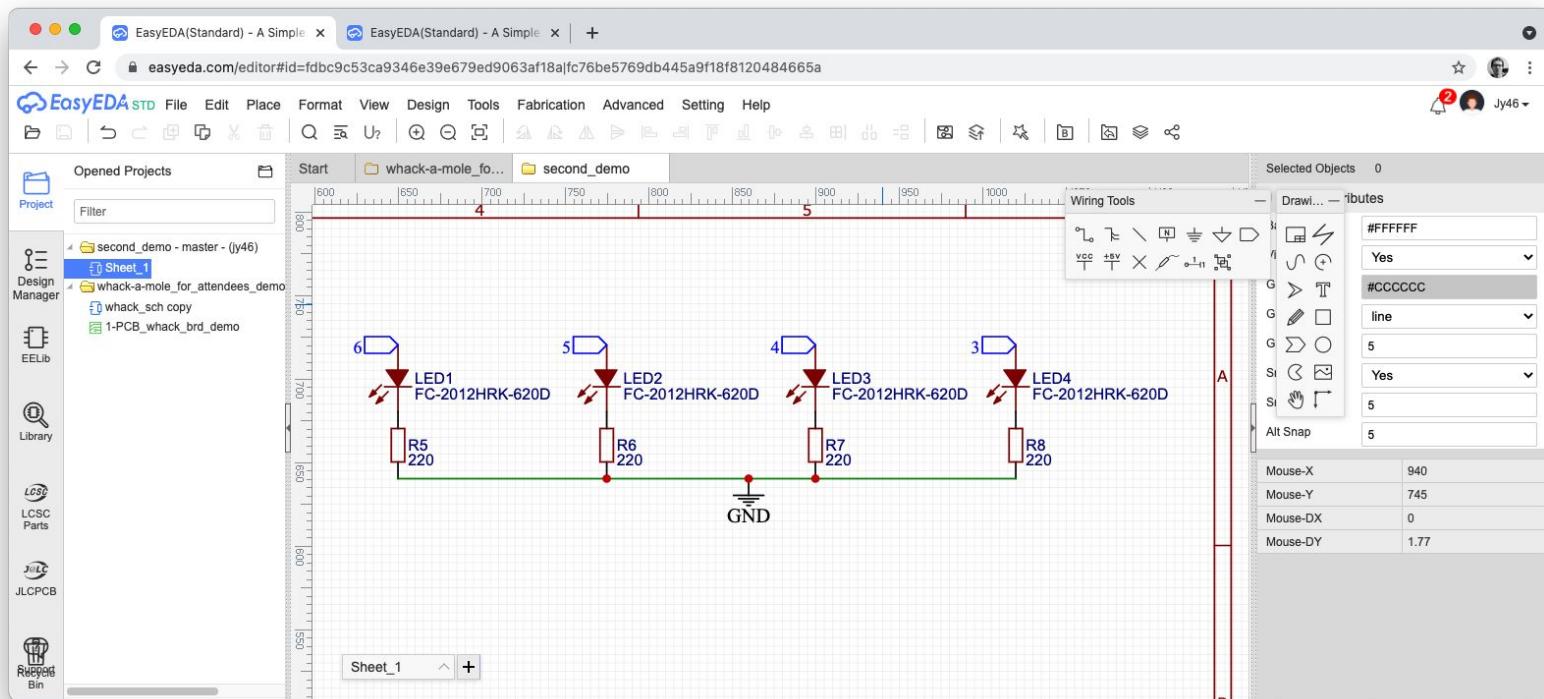
Place 4 of the 220 Ohm Resistors



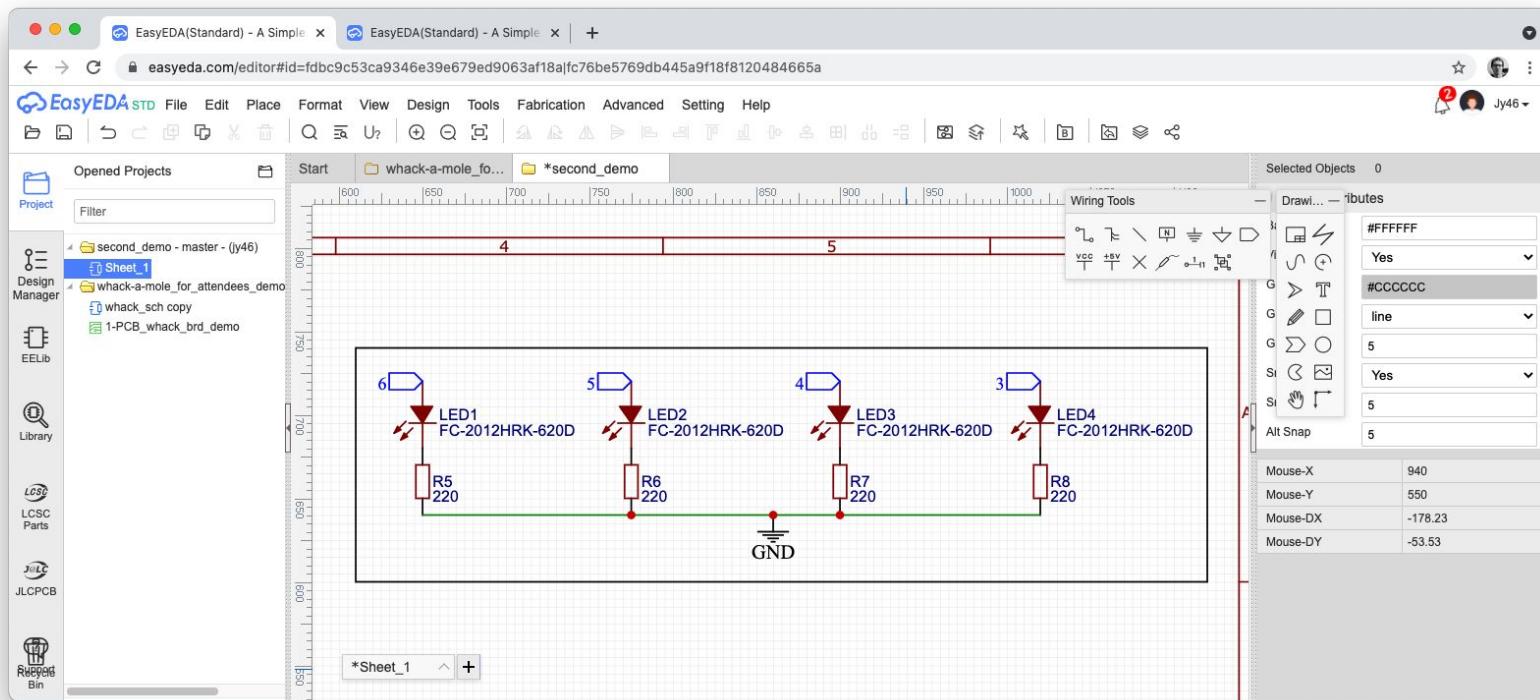
Place GND & Connect Resistors to GND



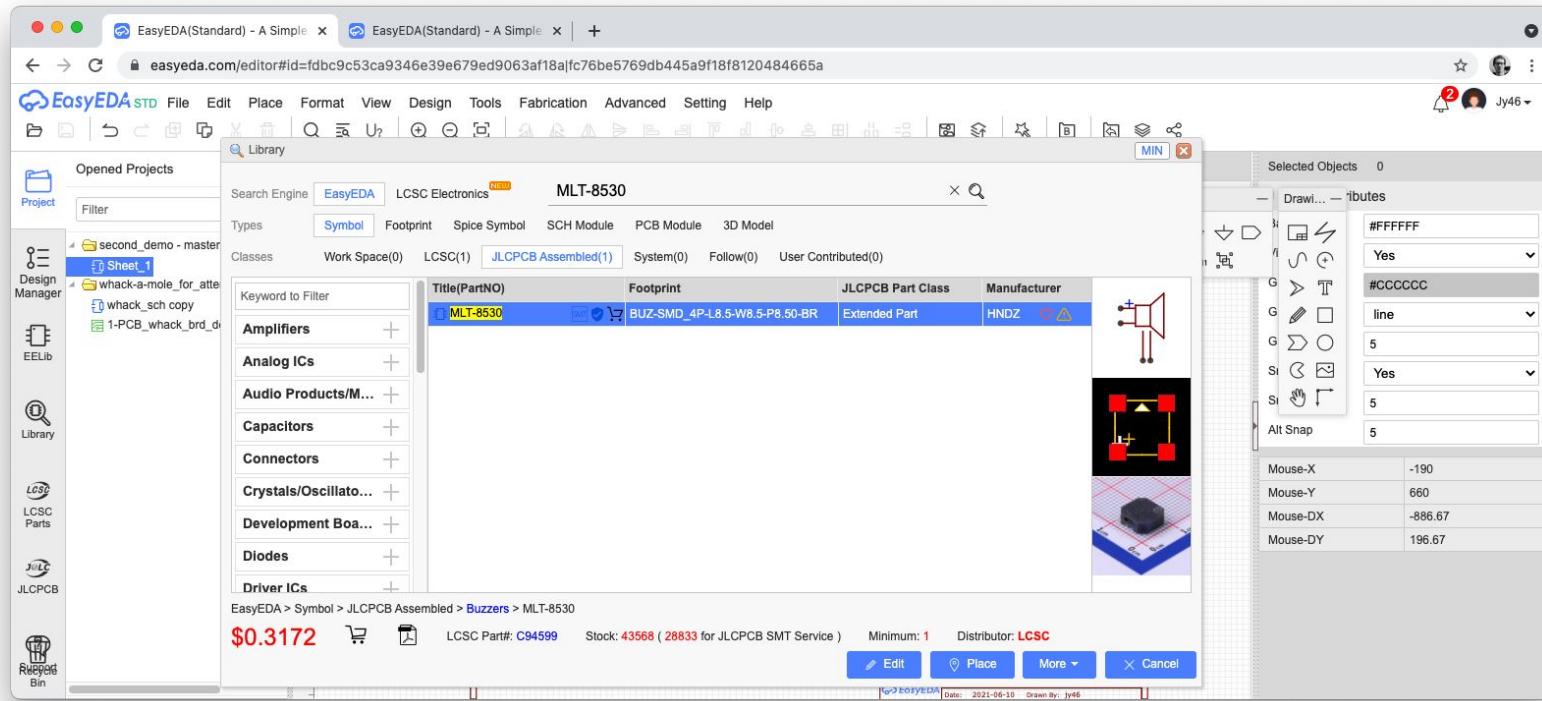
Add Net Ports 6, 5, 4, 3 to LEDs



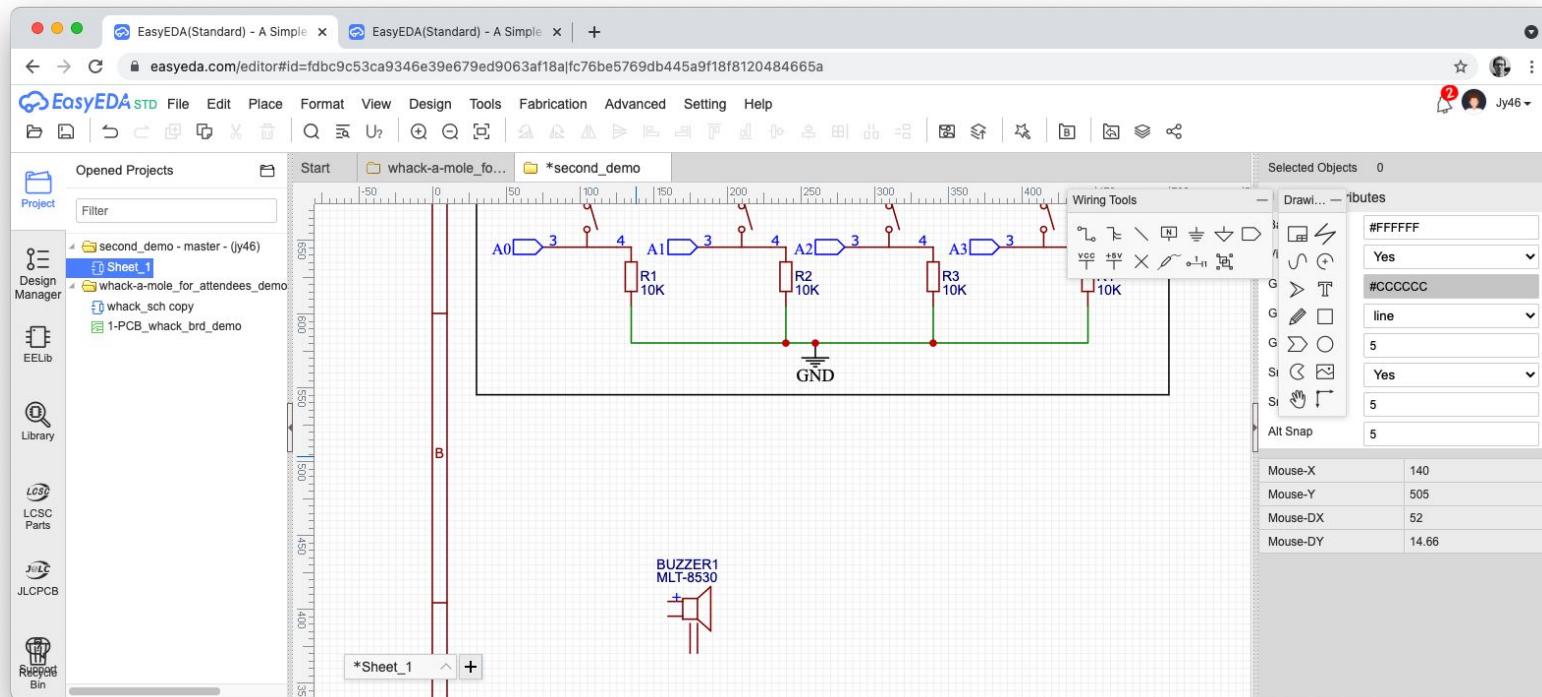
Draw Box Around This



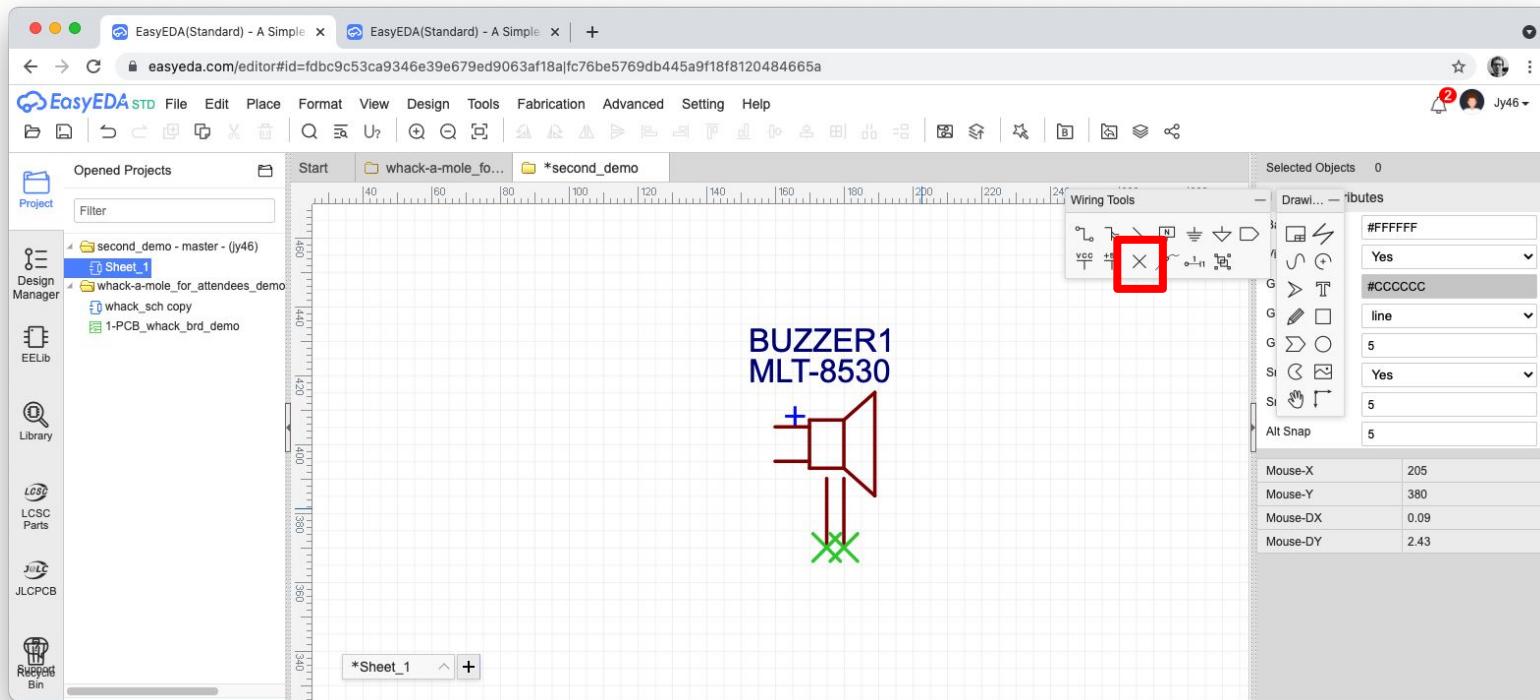
Grab Buzzer from Library: “MLT-8530”



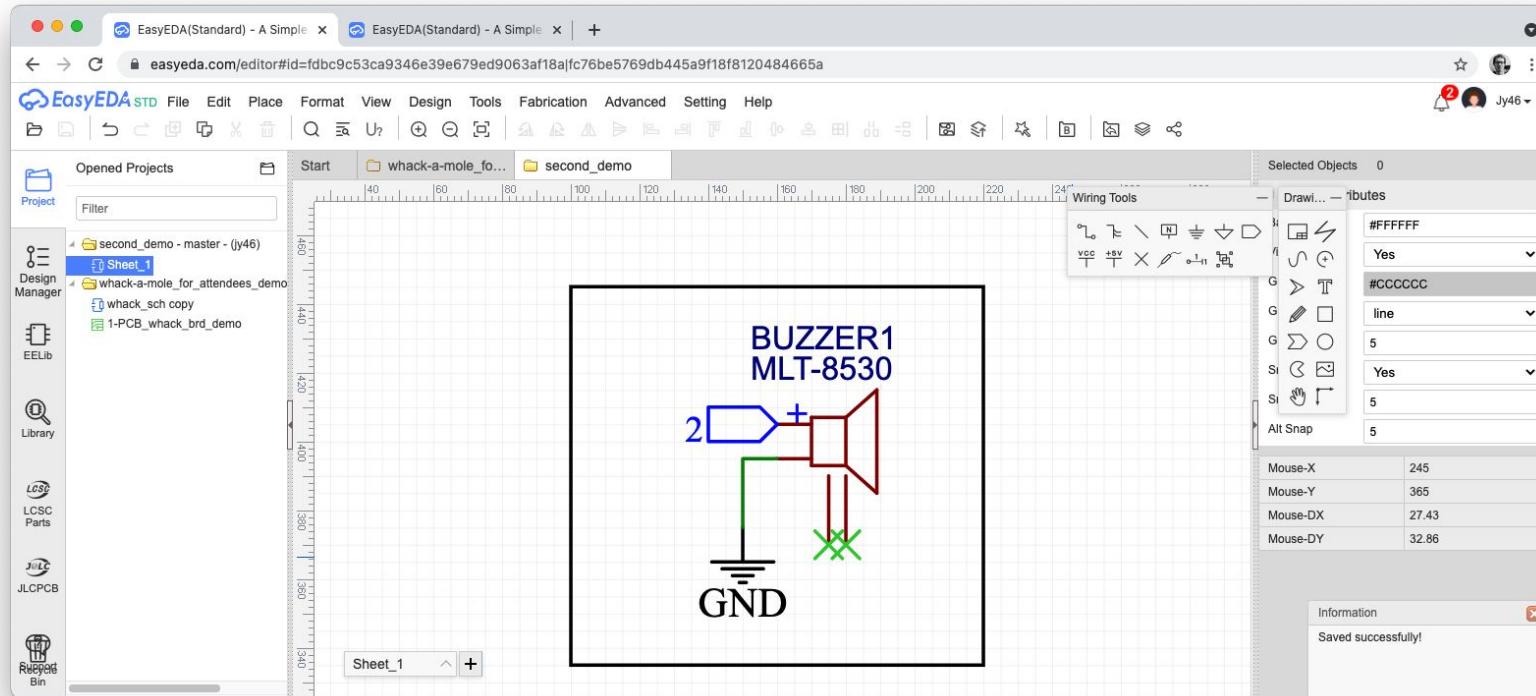
Place Buzzer



We Don't Use the Bottom Terminals: Place No Connect Flags

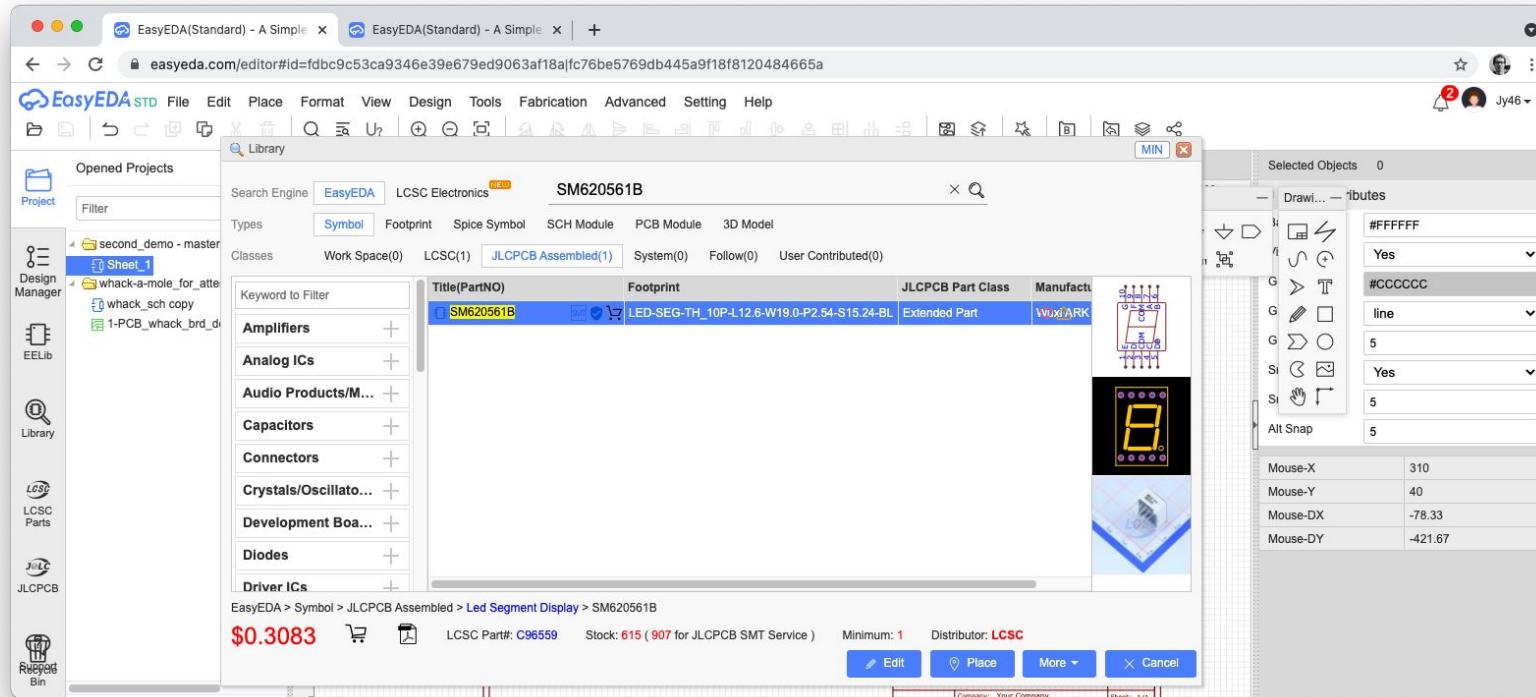


Connect Other Terminals to GND & Net Port 2

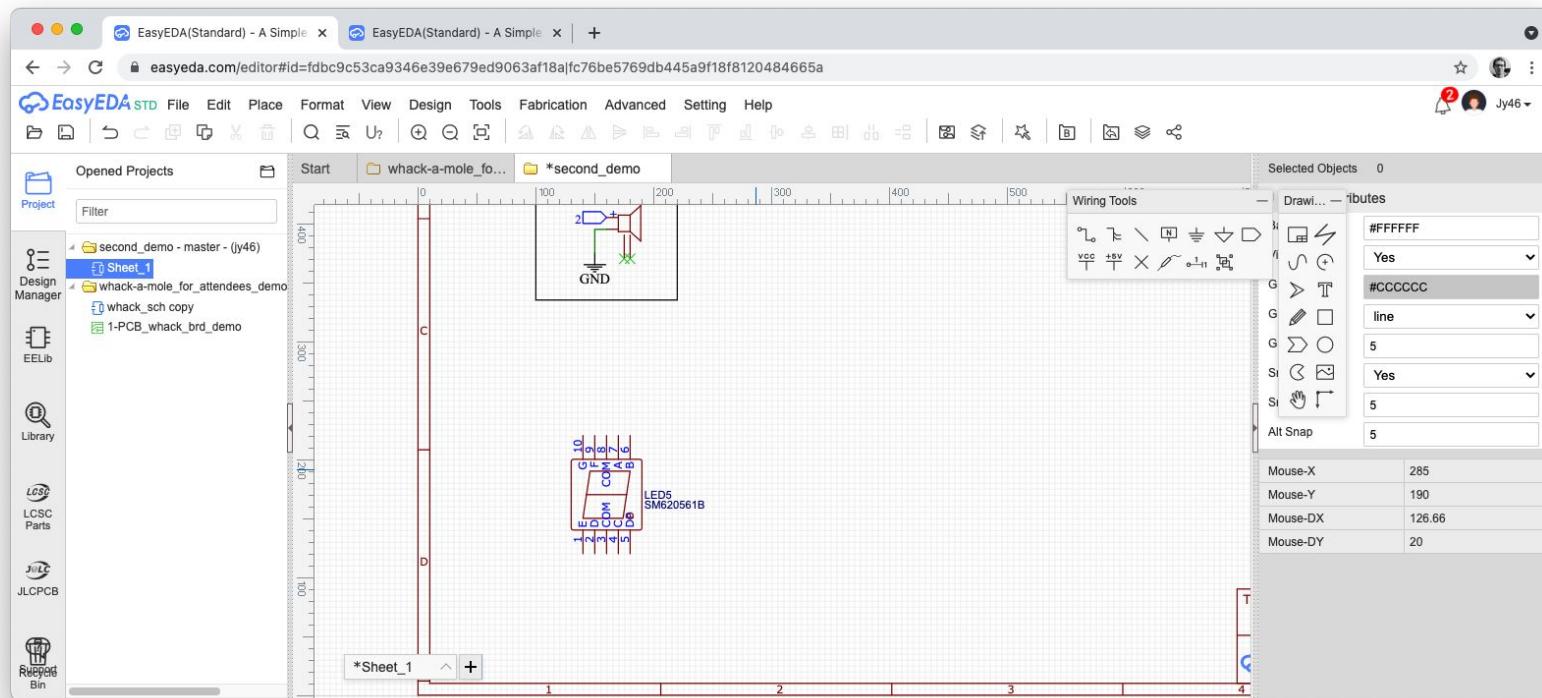


Also add
a box!

Grab 7 Segment Display from Library: “SM620561B”

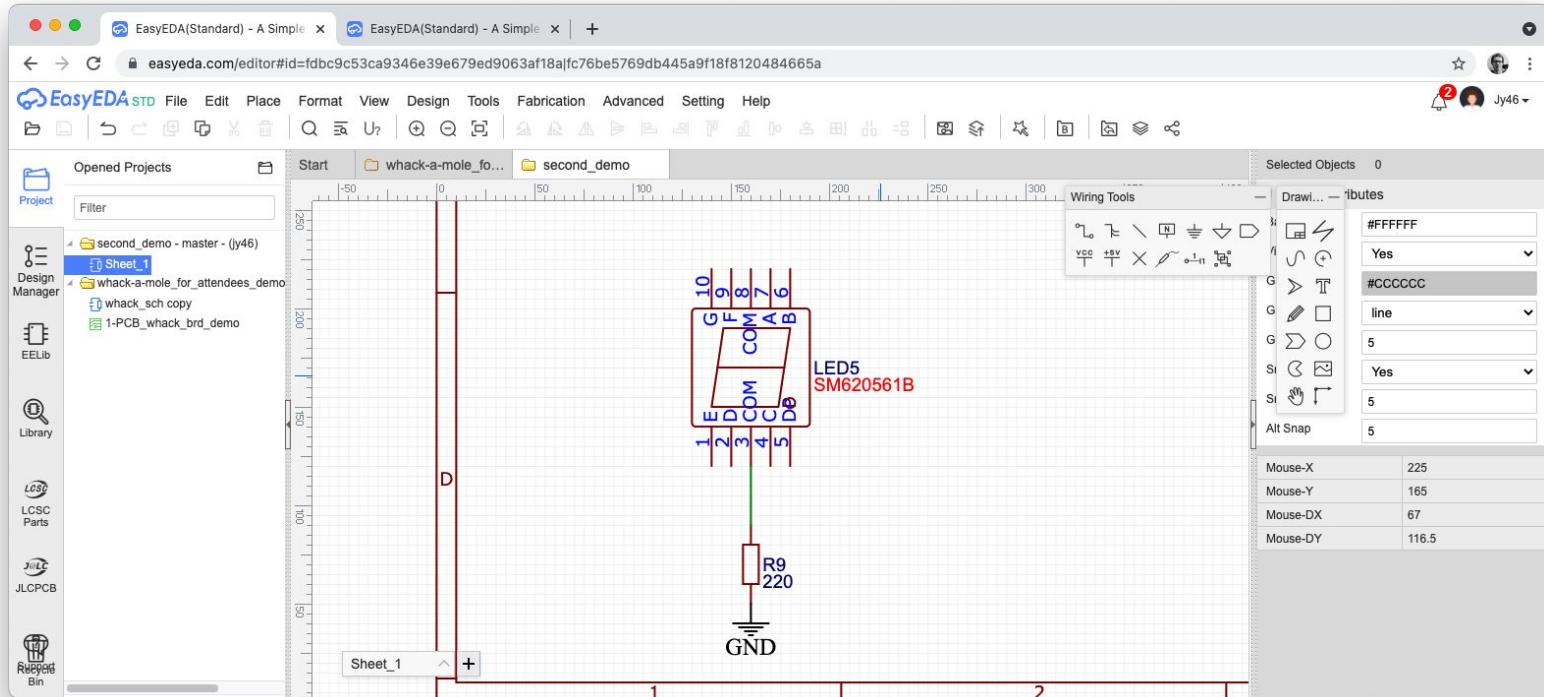


Place Display

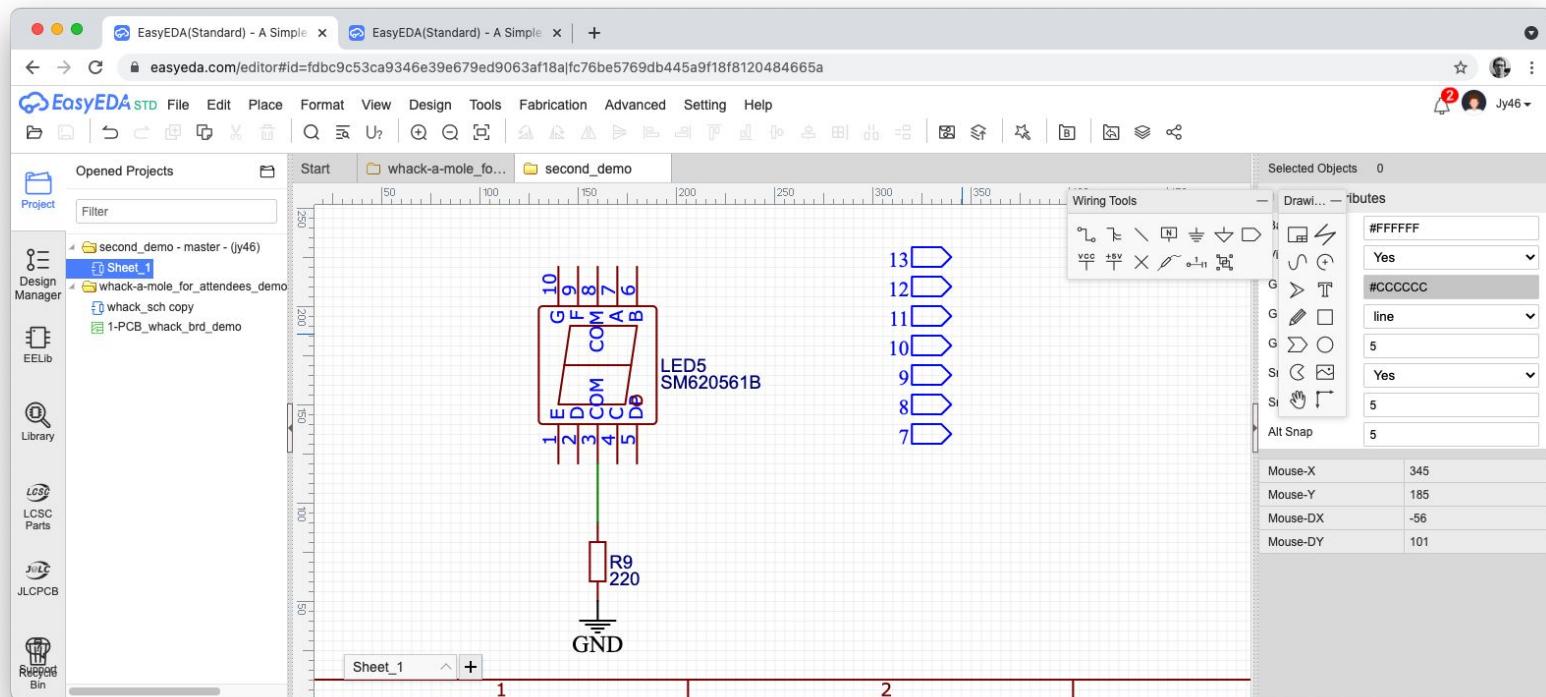


Copy and Paste Resistor Used with LEDs

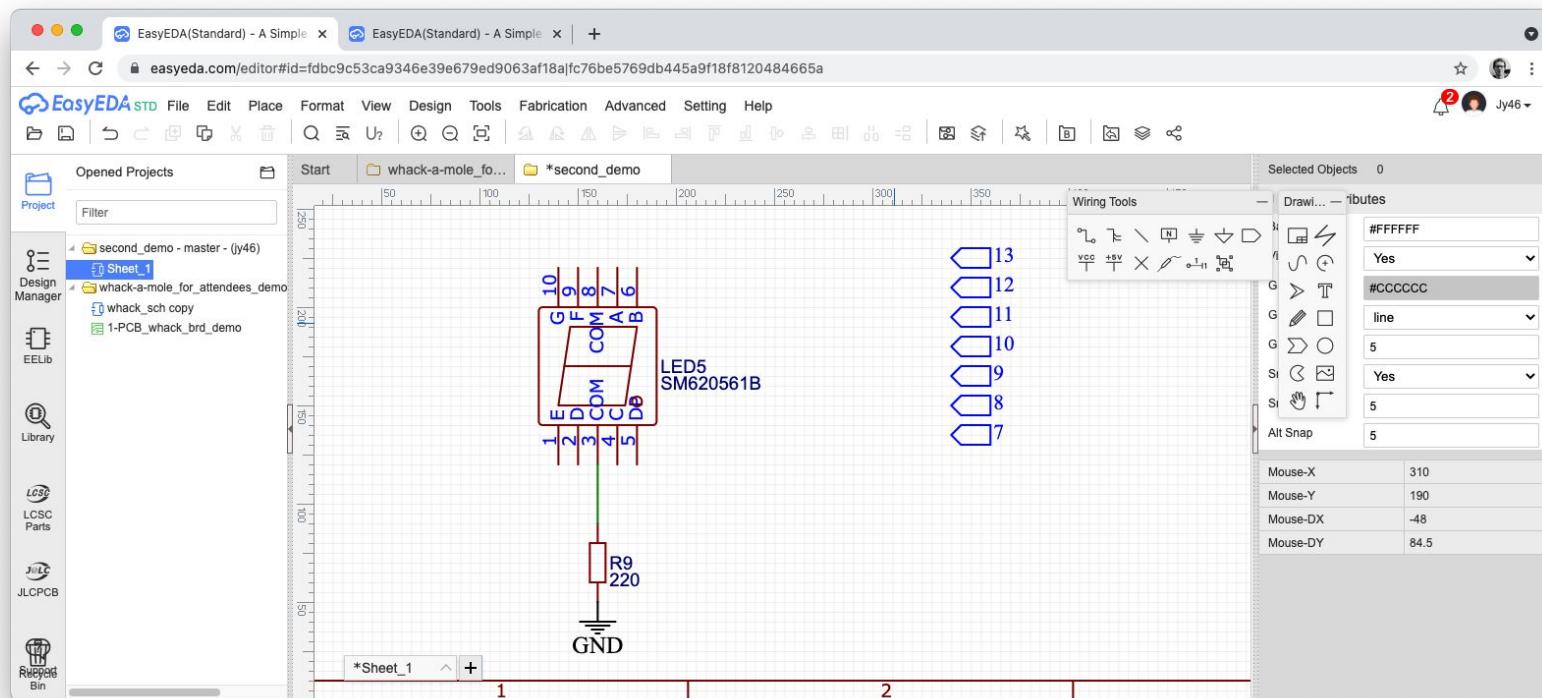
Connect the Resistor to GND & Pin 3



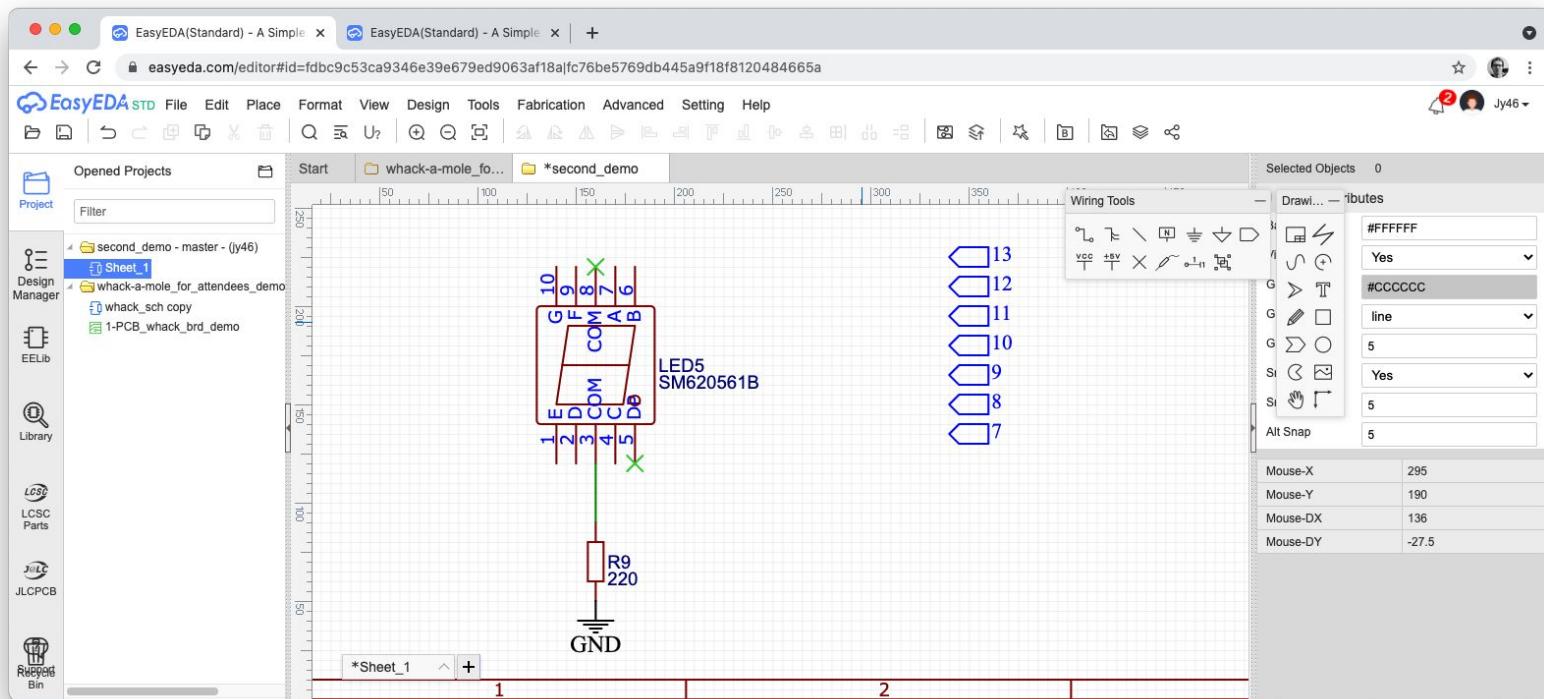
Add Net Ports 7 to 13



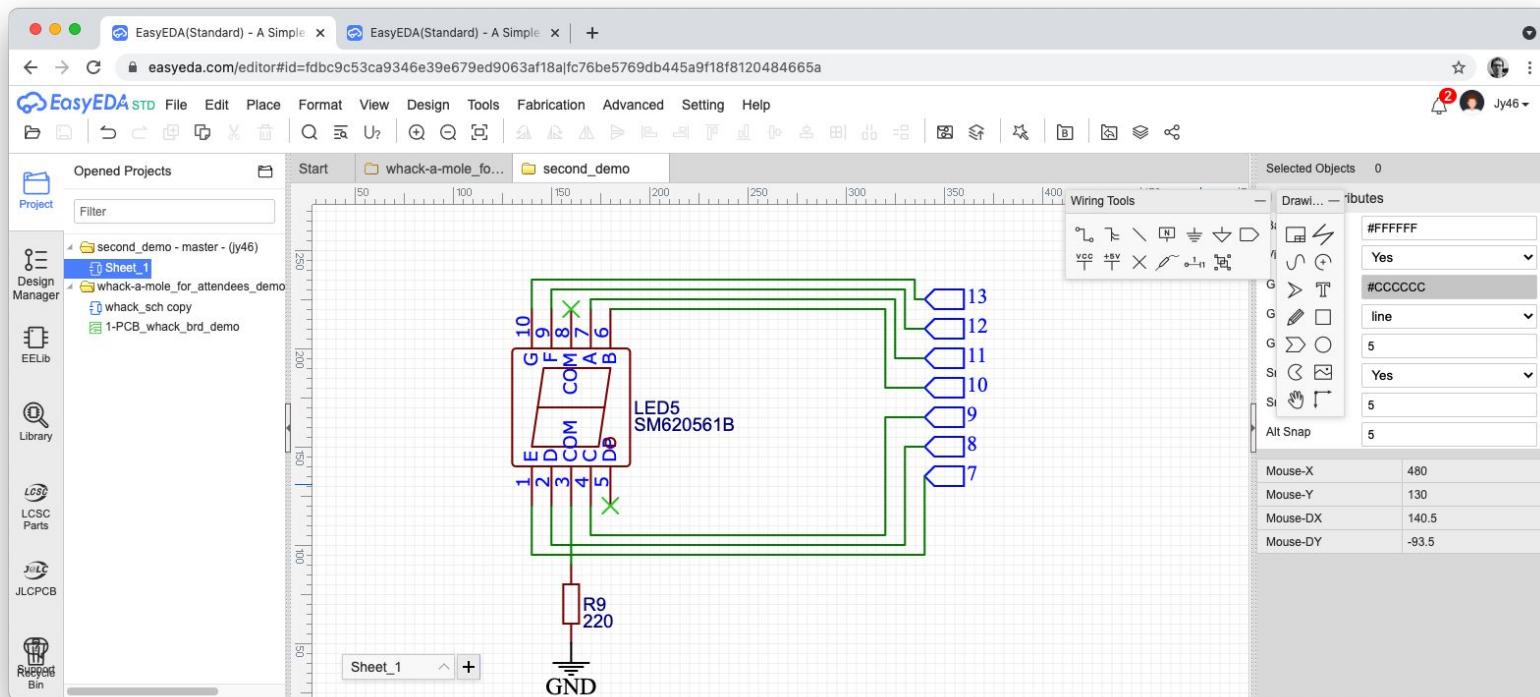
Rotate Net Ports



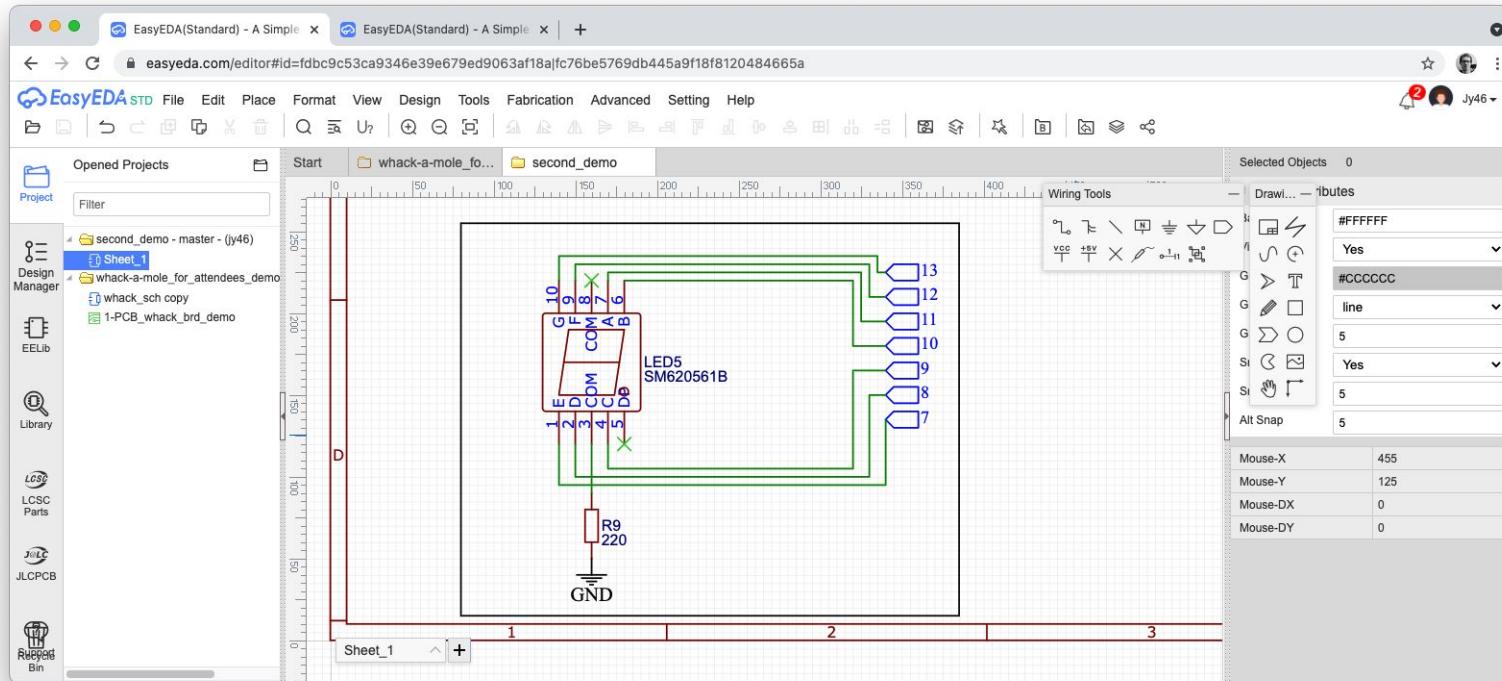
Add No Connect Flags to Pins 5 and 8



Wire Other Pins to Net Ports (counter-clockwise)

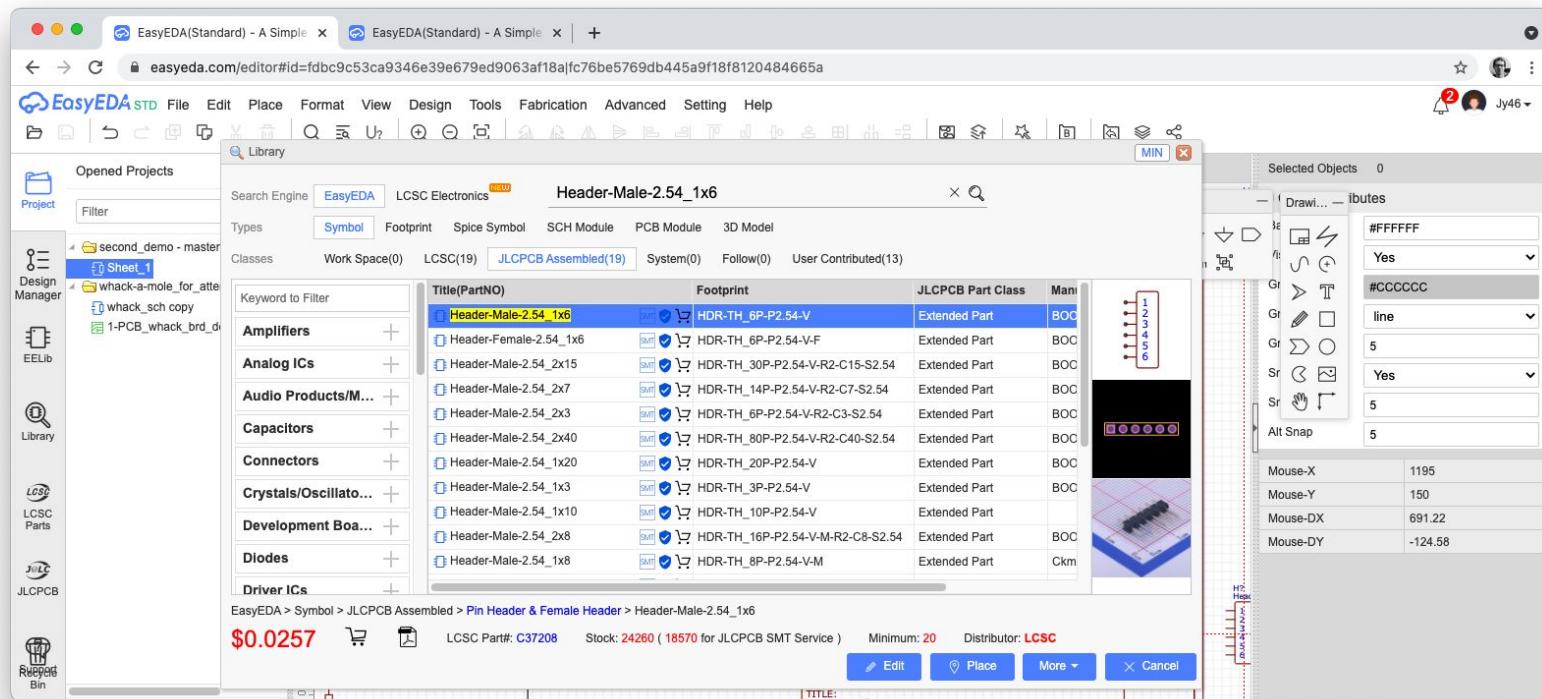


Wire Other Pins to Net Ports (counter-clockwise)

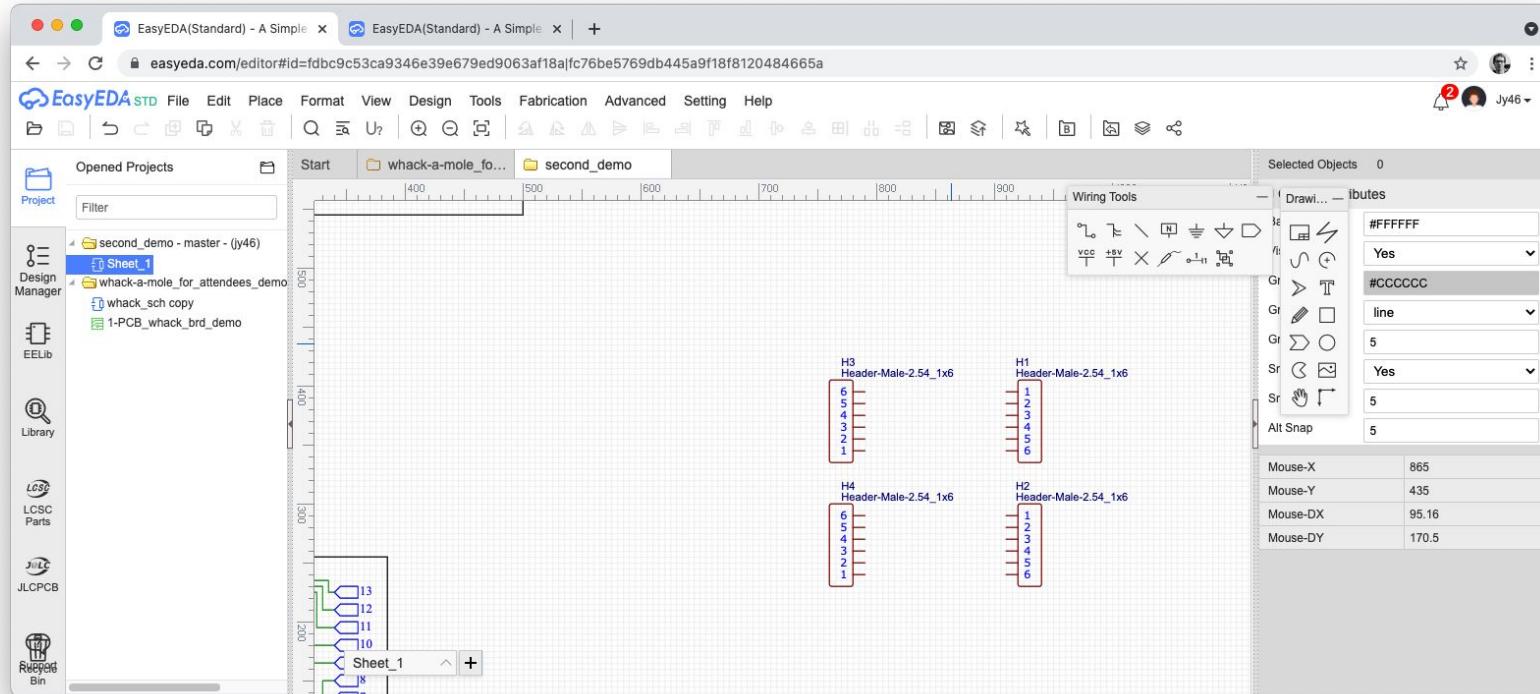


Also add
a box!

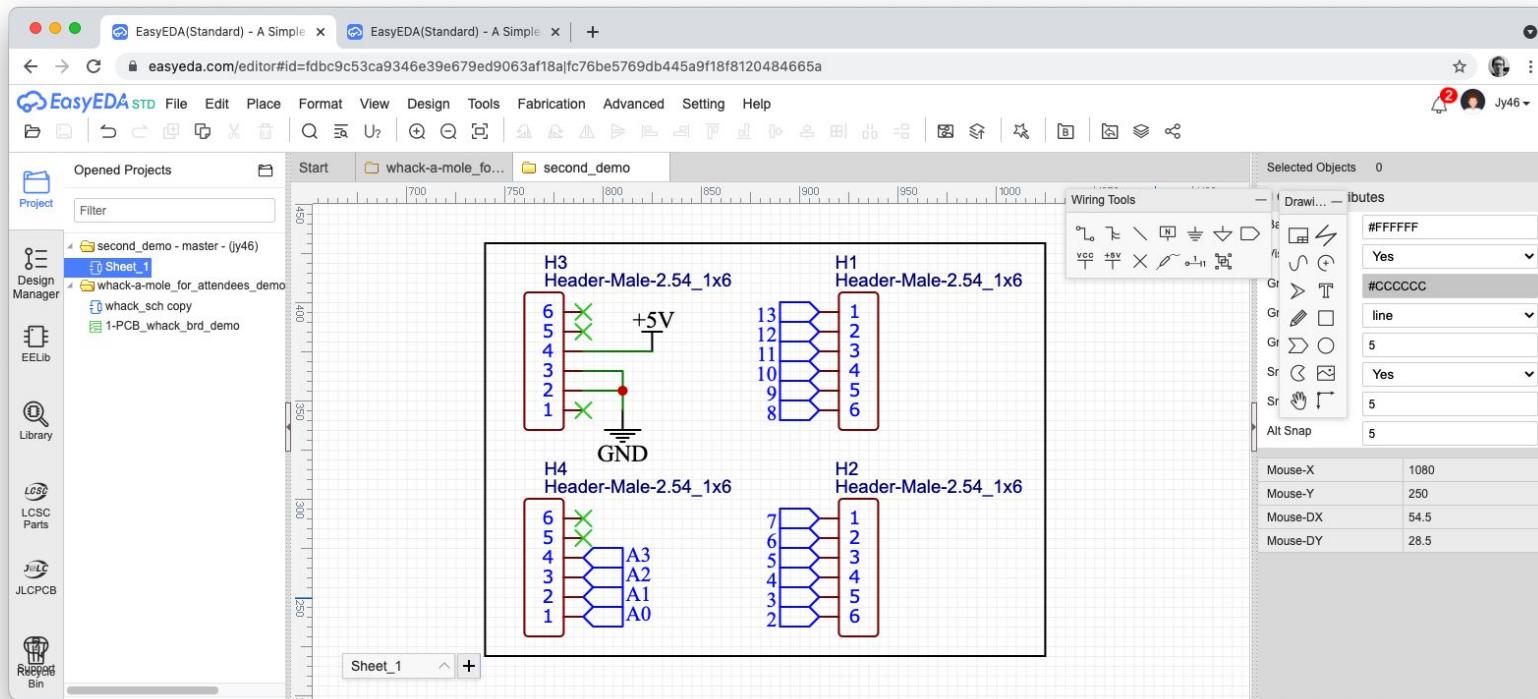
Now Let's Grab Header from the Library: "Header-Male-2.54_1x6"



Place 4 Headers

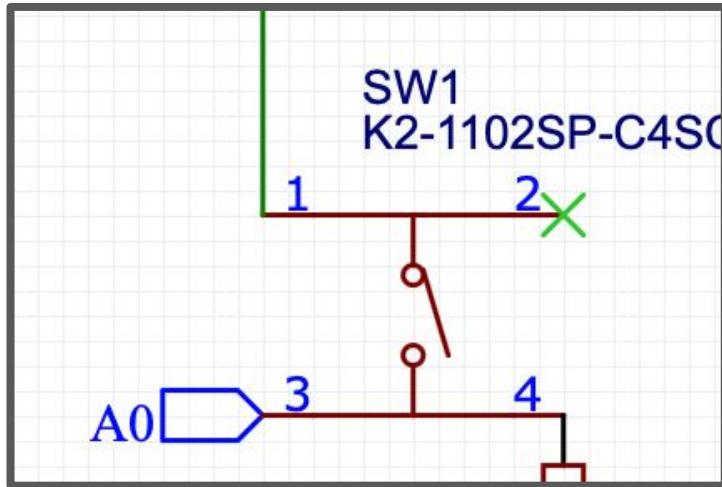


Add Net Ports, No Connect Flags, +5 V, and GND

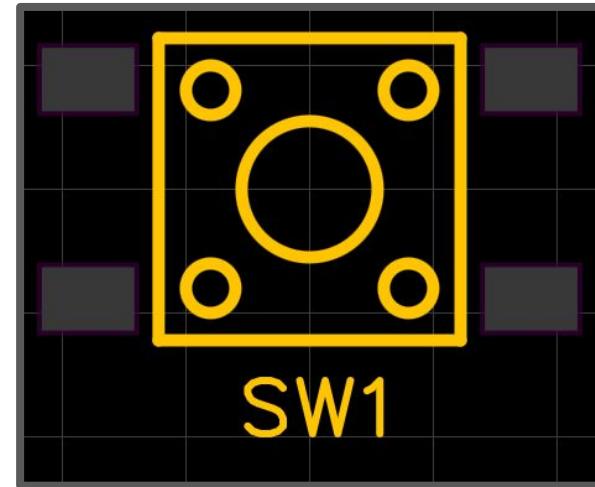


Also add
a box!

Schematic Components Associated with PCB Footprints



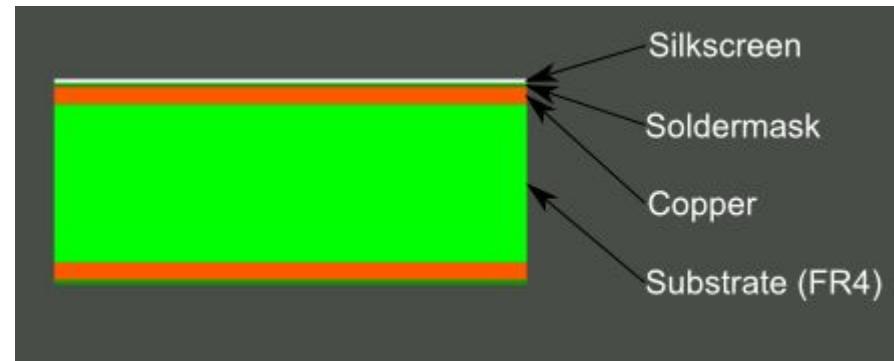
Schematic



PCB

Information Needed by PCB Program: Board size+layers

- Board size
- # layers of copper
 - 2 or 4 common
 - More layers means higher cost
 - Motherboards could have 16 layers
 - Other layers exist by default
 - Solder mask
 - Silkscreen

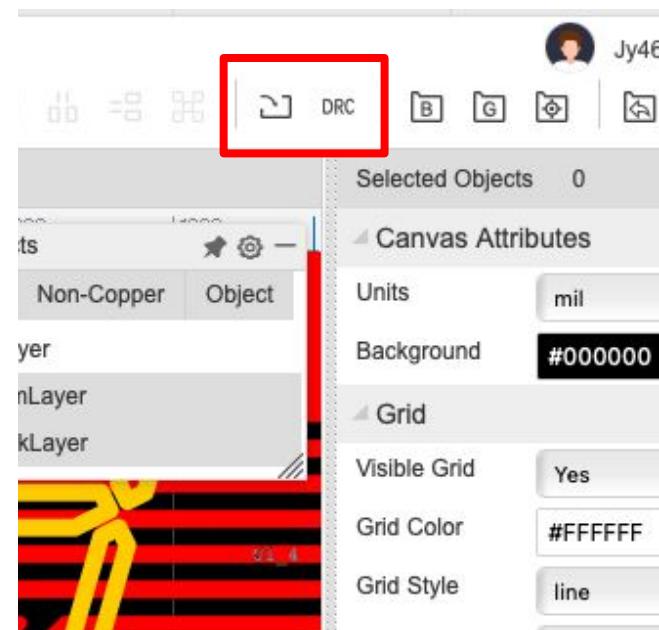


<https://learn.sparkfun.com/tutorials/pcb-basics/all>

Information Needed by PCB Program: Design rules

- Design rules
 - Specified by manufacturer (fabrication house)
 - For example: minimum metal line size or spacing

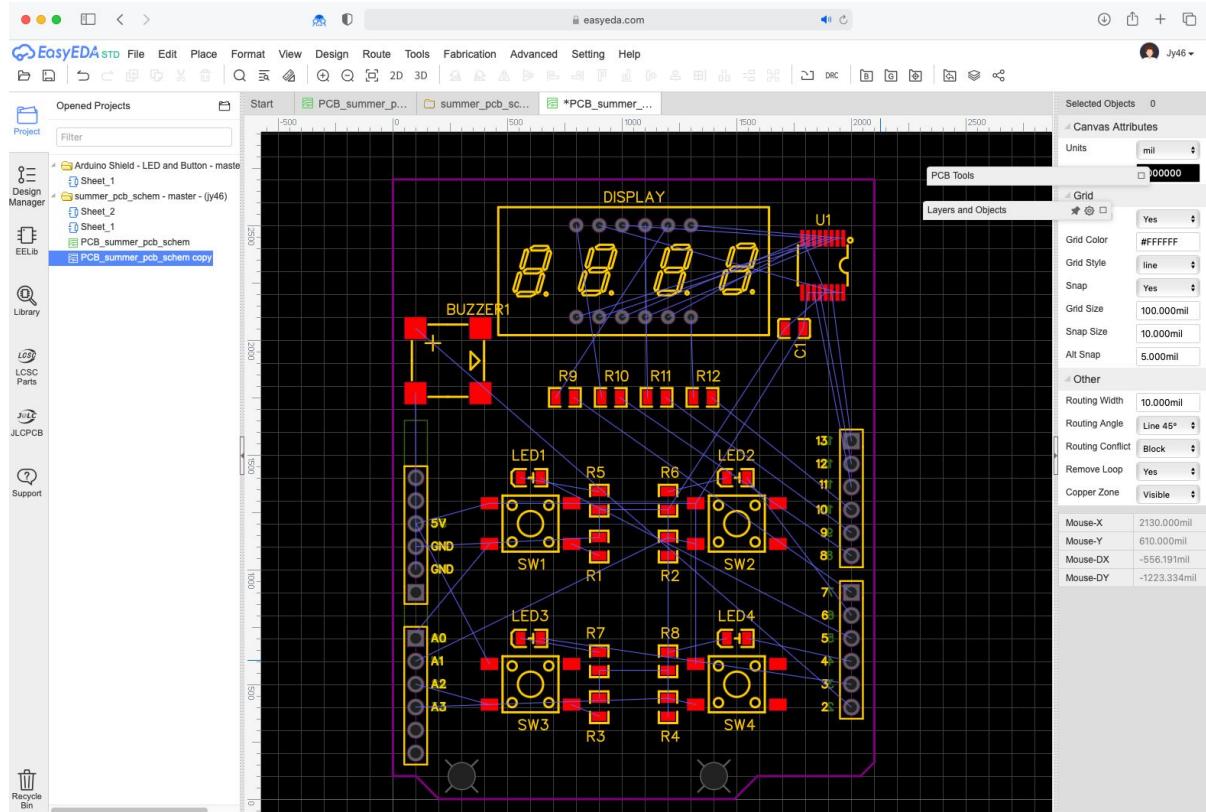
Design Rule					
Rule	Track Width	Clearance	Via Diameter	Via Drill Diameter	Track Length
Default	10	6	24	12	



You've Finished a Schematic, What's Next?

Move parts
(footprints) around
on board

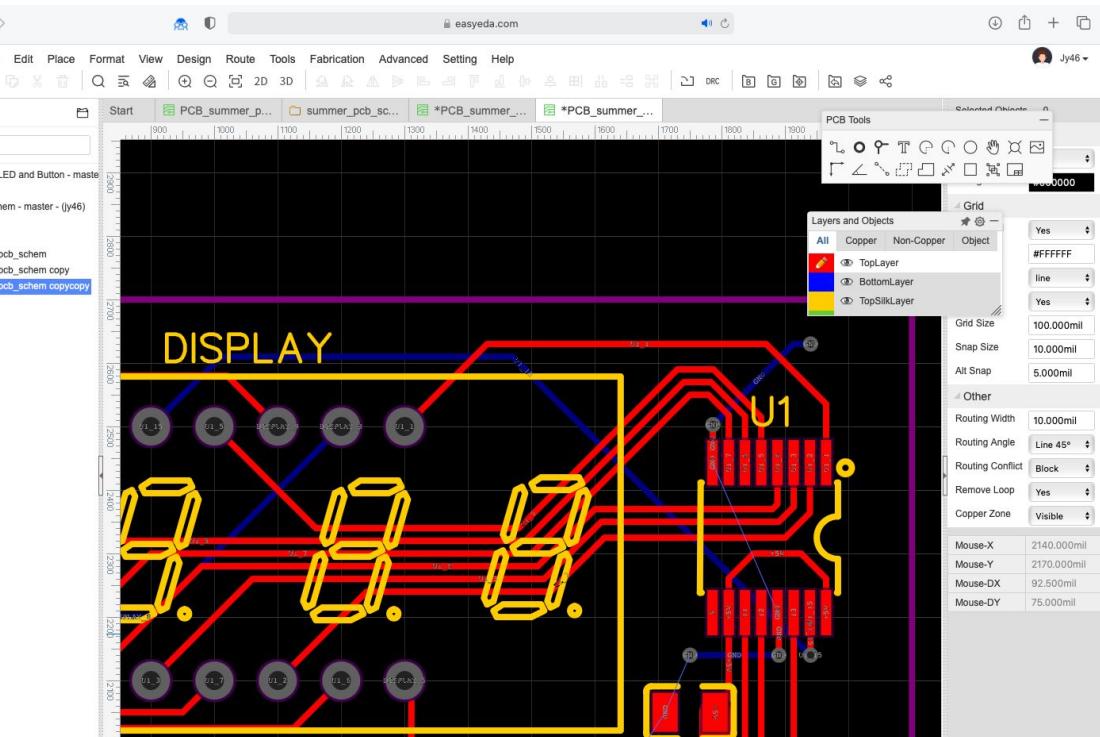
This view is called
“rats nest”



You've Finished a Schematic, What's Next?

Route wires (traces) on multiple layers between parts/pads

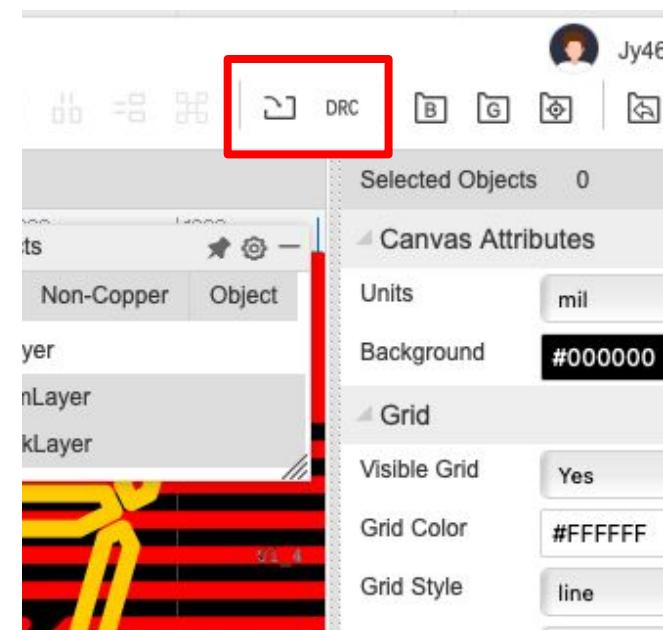
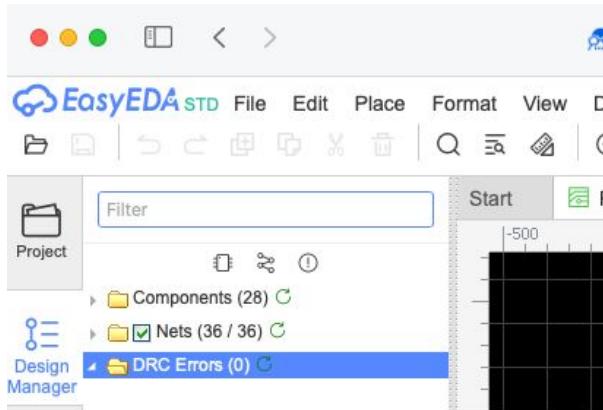
- Manually route critical traces
- Automatically route remaining traces via autorouter



You've Finished a Schematic, What's Next?

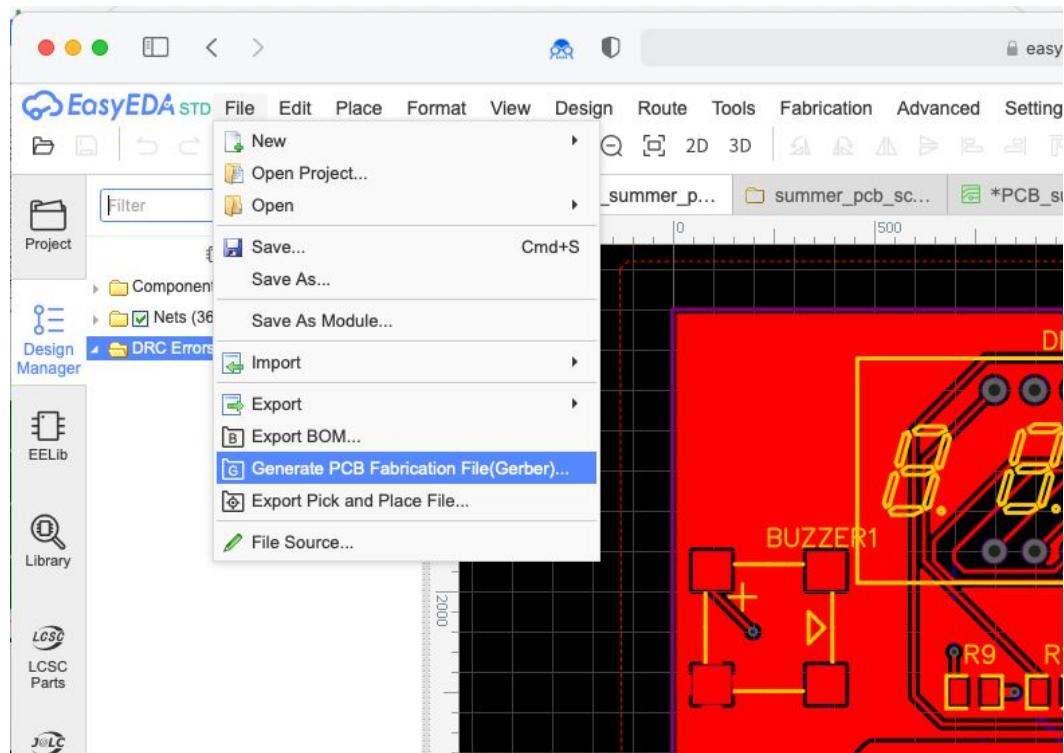
Perform design rules check (DRC)

- Can the fab make the PCB?
- Space between traces
- Keep out zones for components
- Space from edges

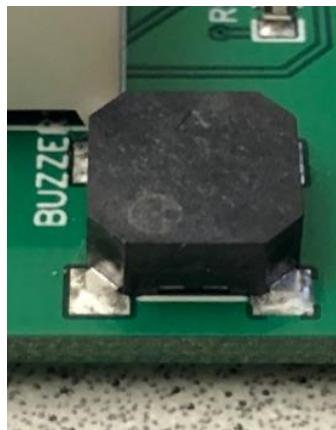


You've Finished Placement & Routing, What's Next?

- Produce files for manufacturing
 - “Gerber” files (1 per layer)
 - Metal, solder mask, silkscreen, pads, thru-hole plating
 - Drill files (hole locations, drill size)
- Produce bill of materials (BOM)
 - All components in the design

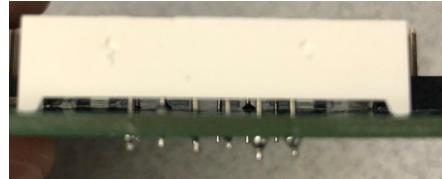


Component Sizes & Packages



Surface Mount
(SMT)

Side



Top

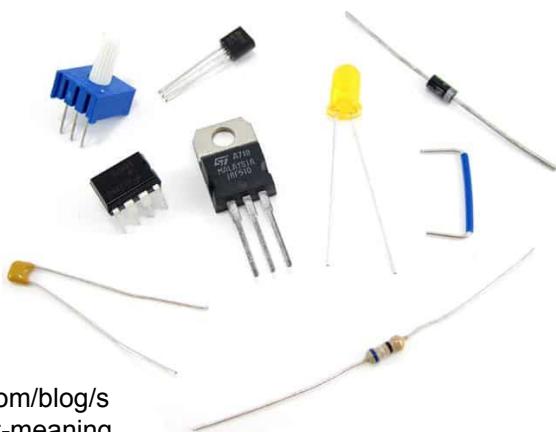


Bottom

Thru-hole

Thru-hole

- Older
- Requires more space
- Easy to hand-solder & de-solder
- Many parts no longer have thru-hole packages



IC Package - Through Hole

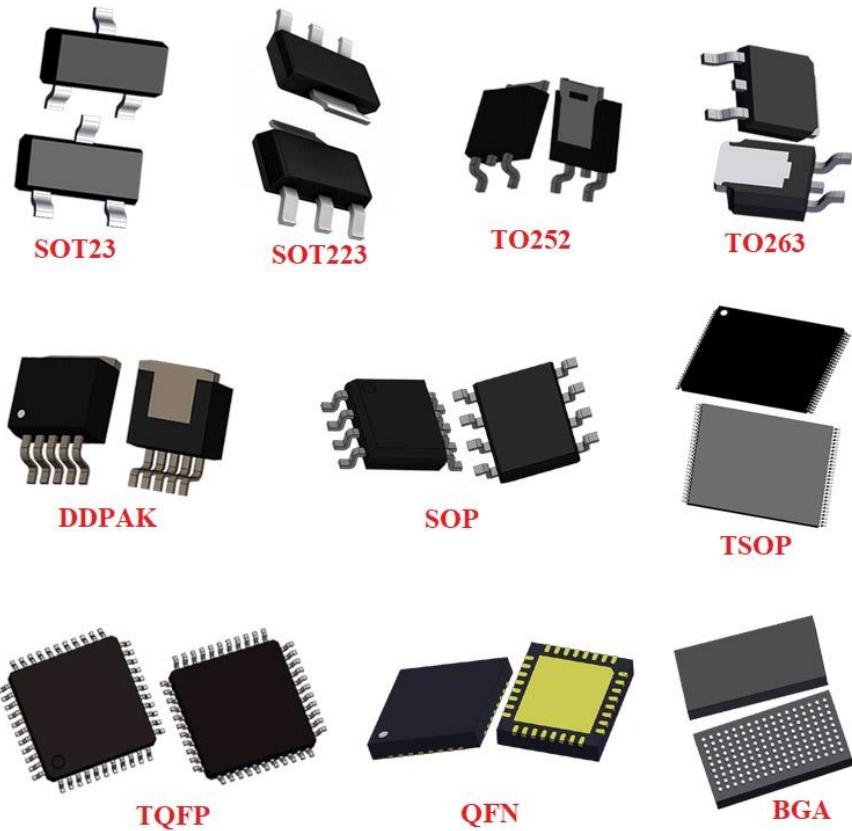


Surface Mount (SMT)

- Modern
- Requires less space
- Some can be hand-soldered

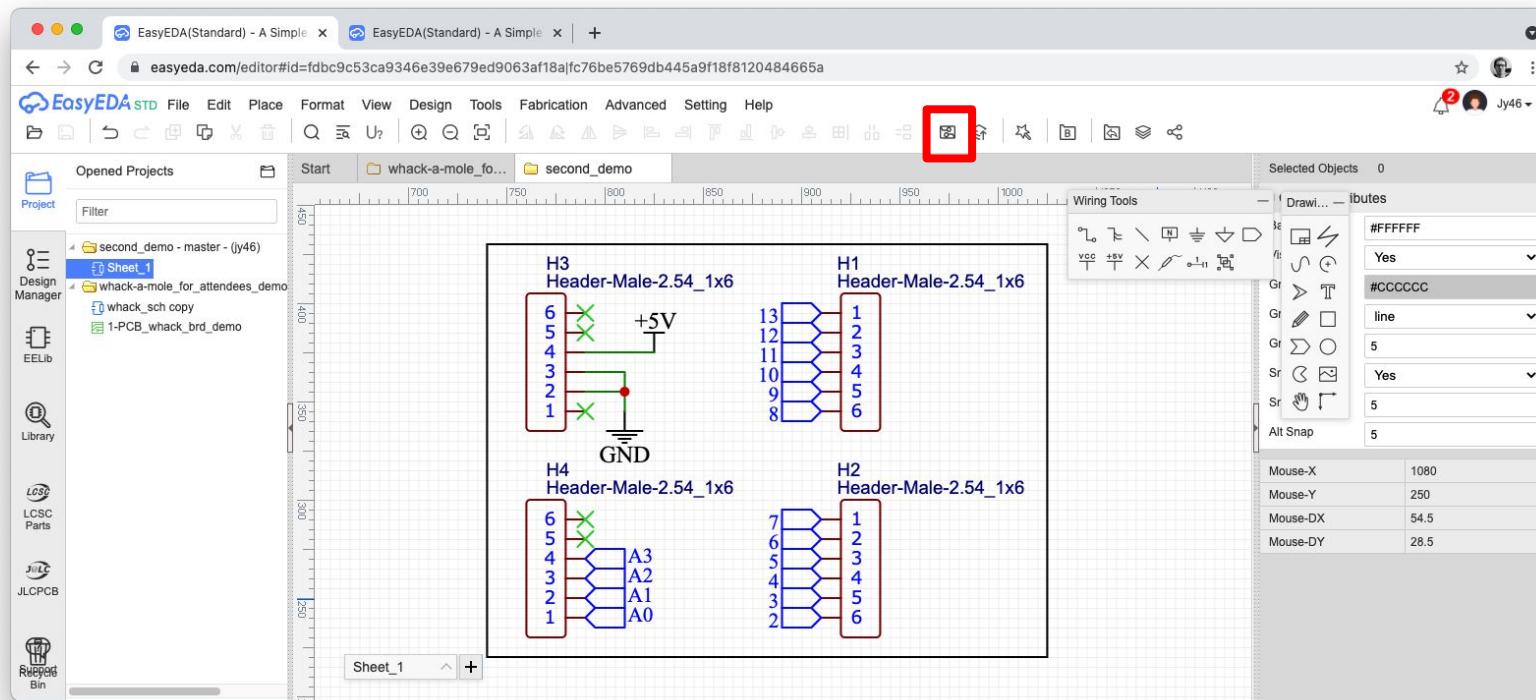


Horowitz & Hill. *The Art of Electronics*, 3rd ed. (2015)

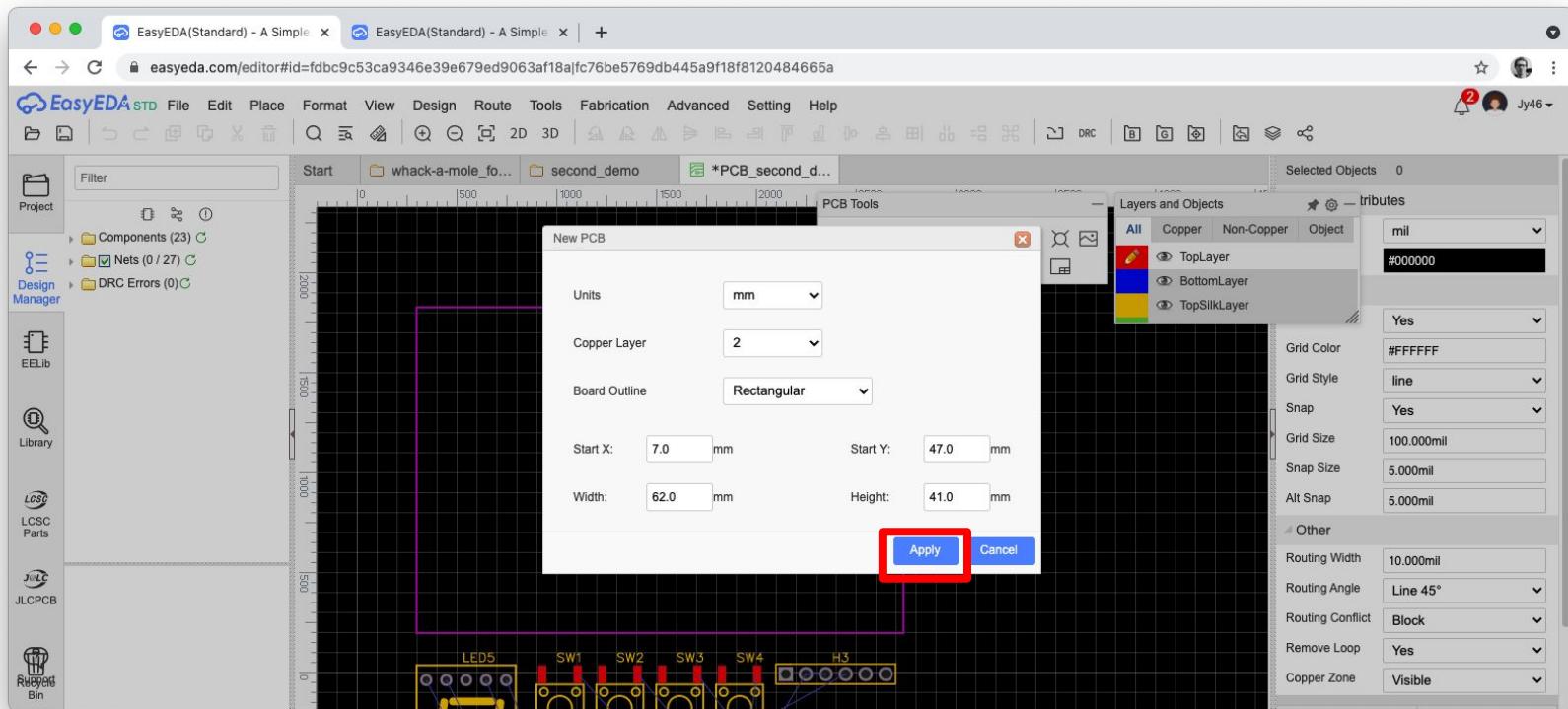


<https://components101.com/articles/different-ic-package-types-and-which-one-should-you-select>

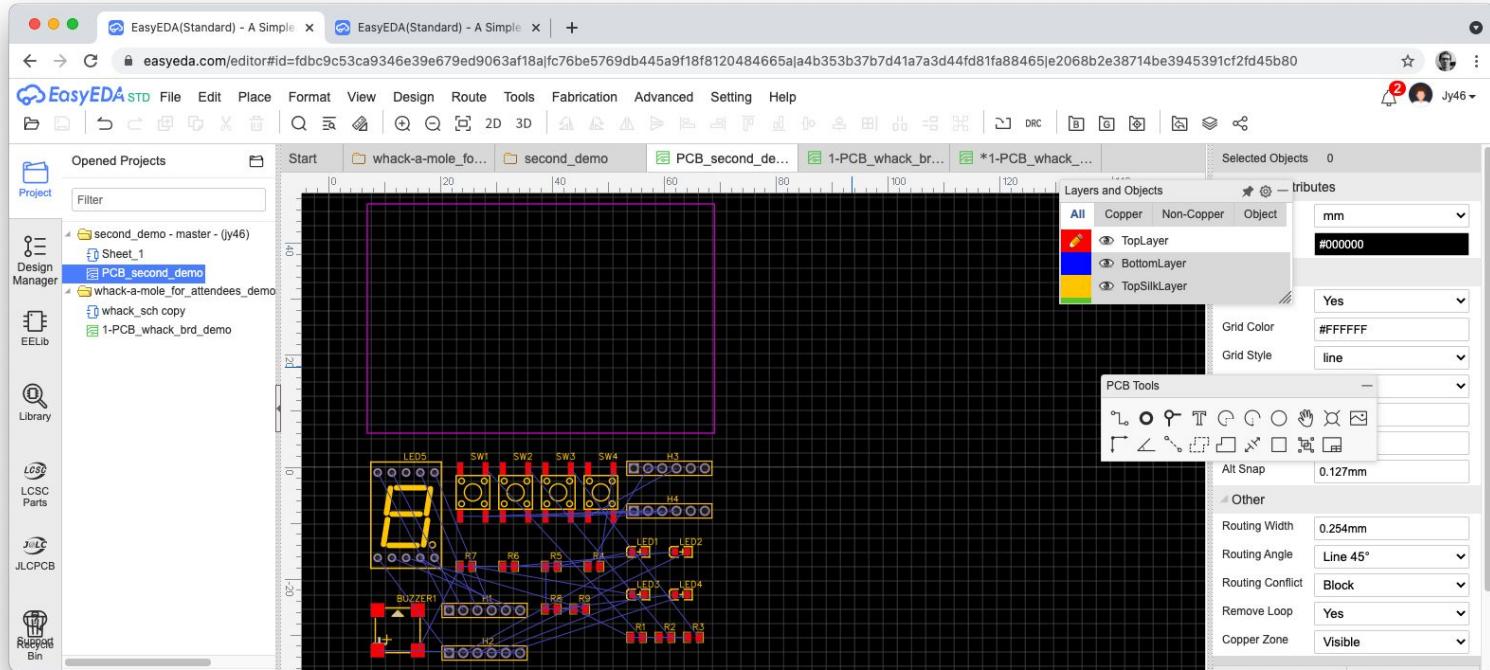
Now Convert Schematic to PCB



Click Apply for Board Sizing



Components are Ready for Placement However, Need Arduino Shield Layout

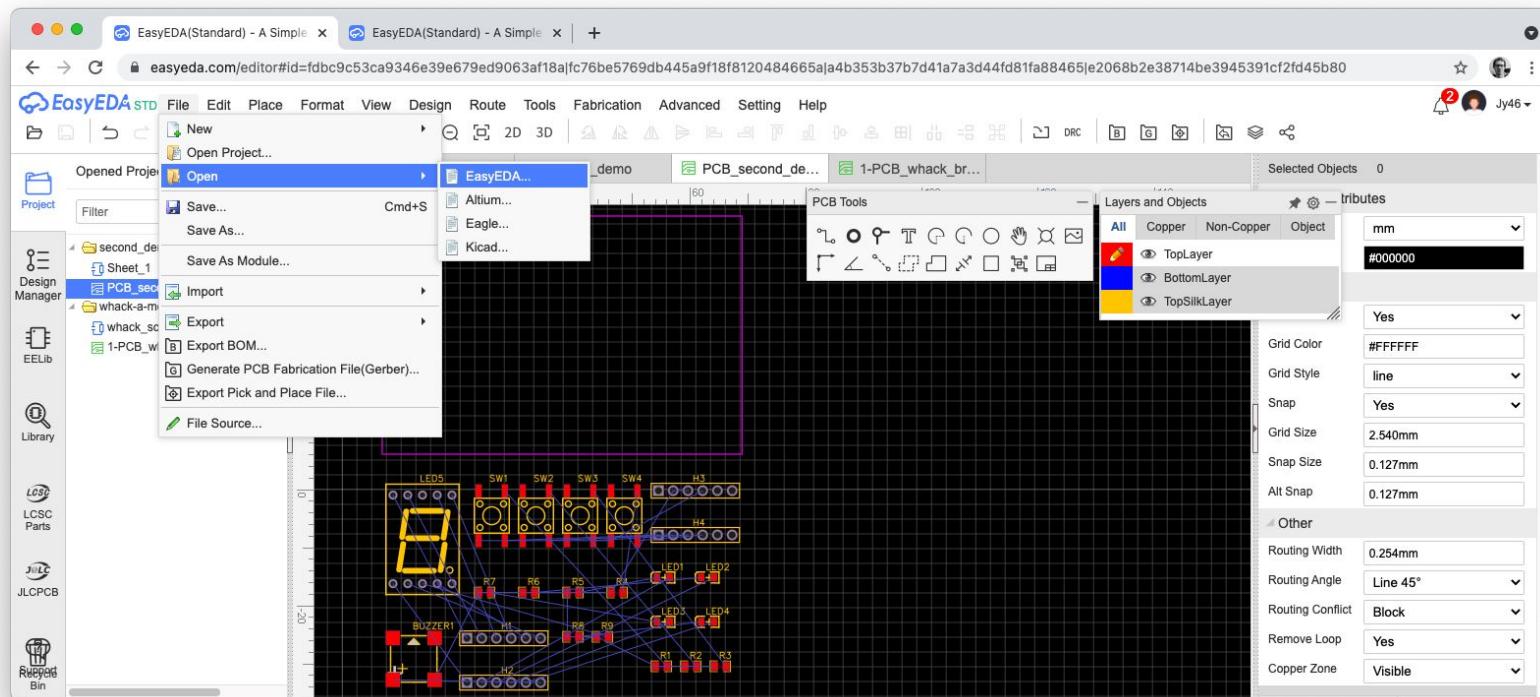


Download File with Arduino Shield Layout

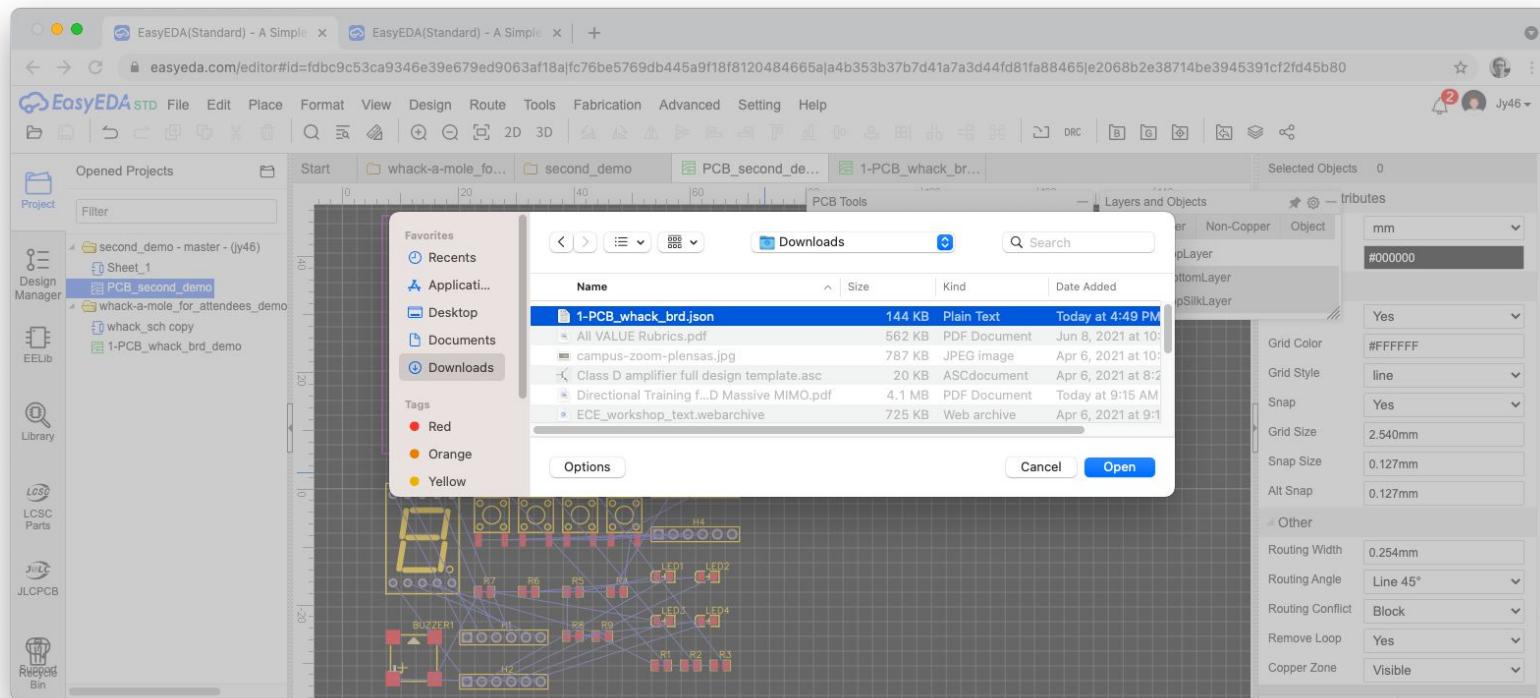
Download file from here:

https://drive.google.com/file/d/1bG_XOHcTlye5dCddasBaLiAk78t8u-0F/view?usp=sharing

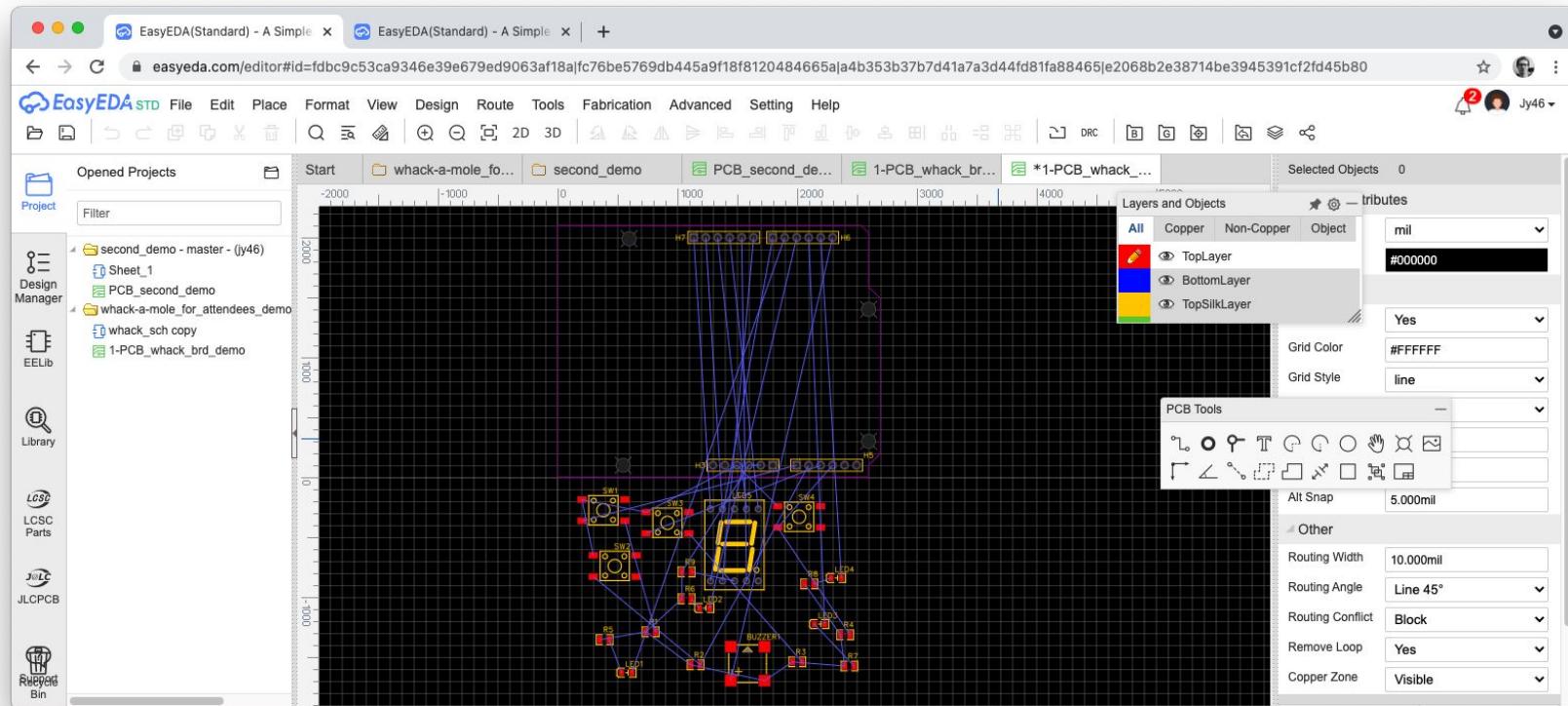
Import Downloaded File with Arduino Shield Layout



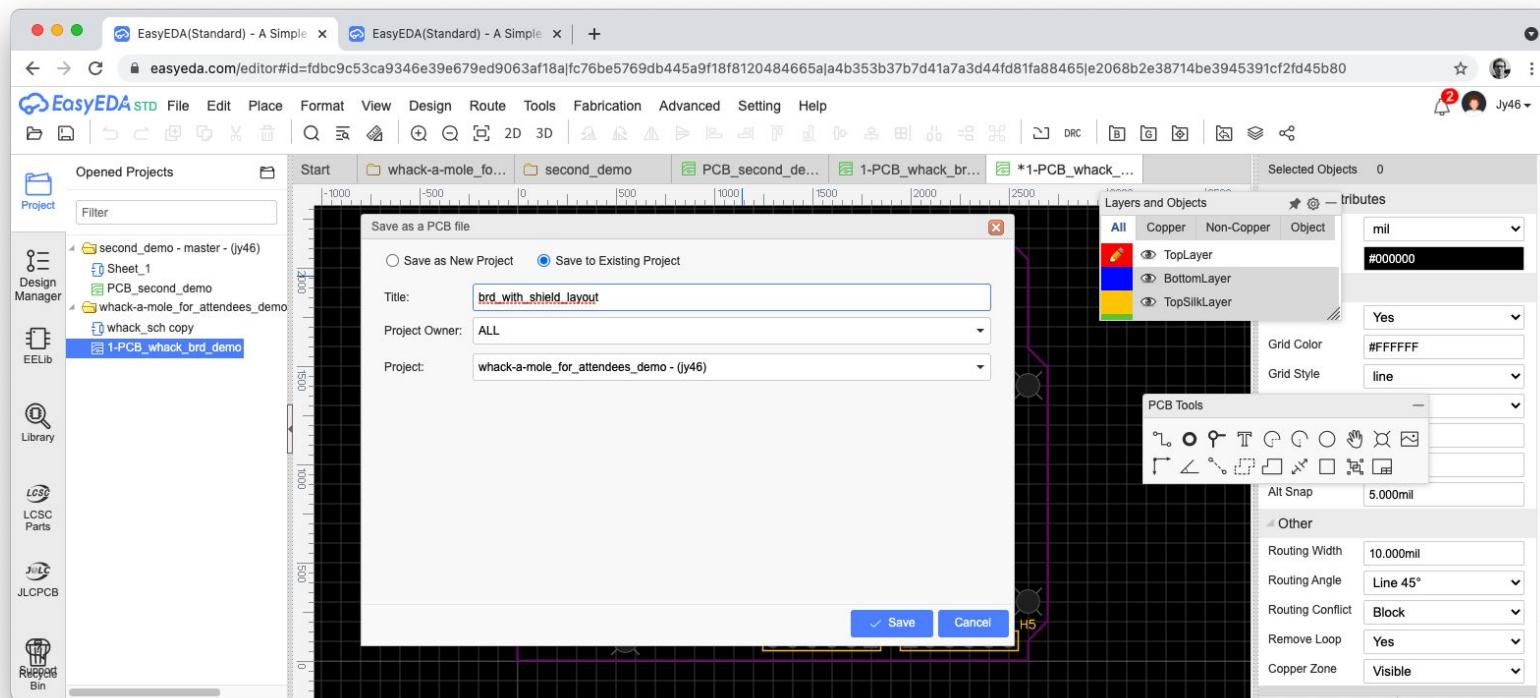
Import Downloaded File with Arduino Shield Layout



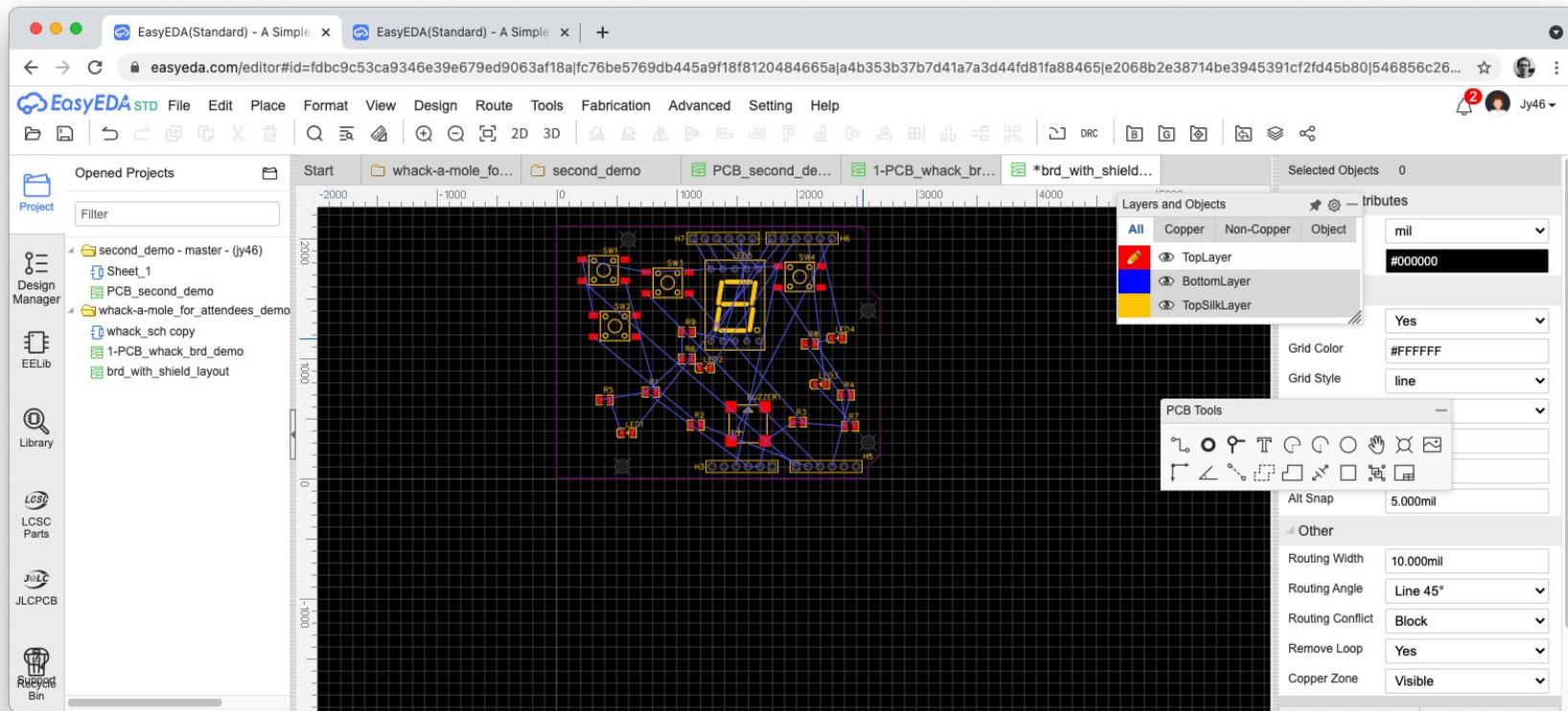
Import Downloaded File with Arduino Shield Layout



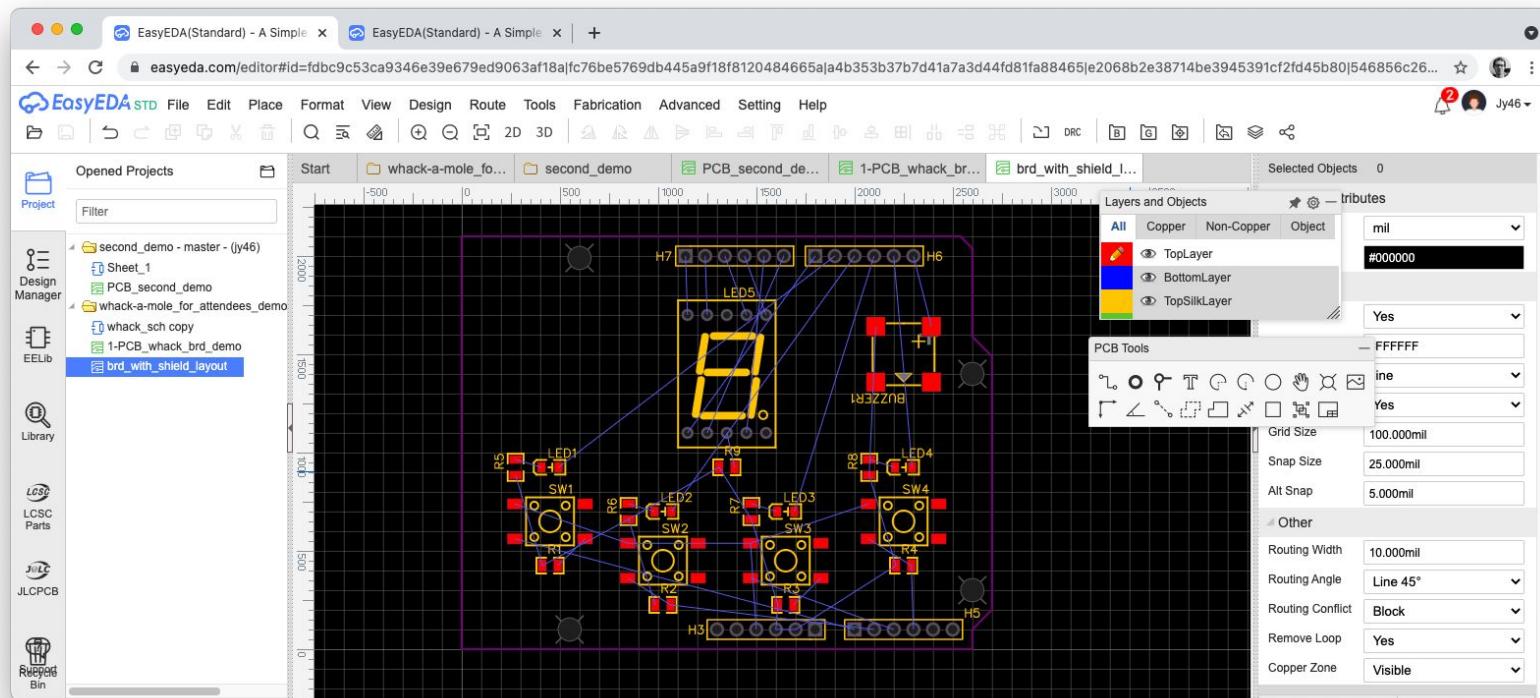
Save it in the Same Project



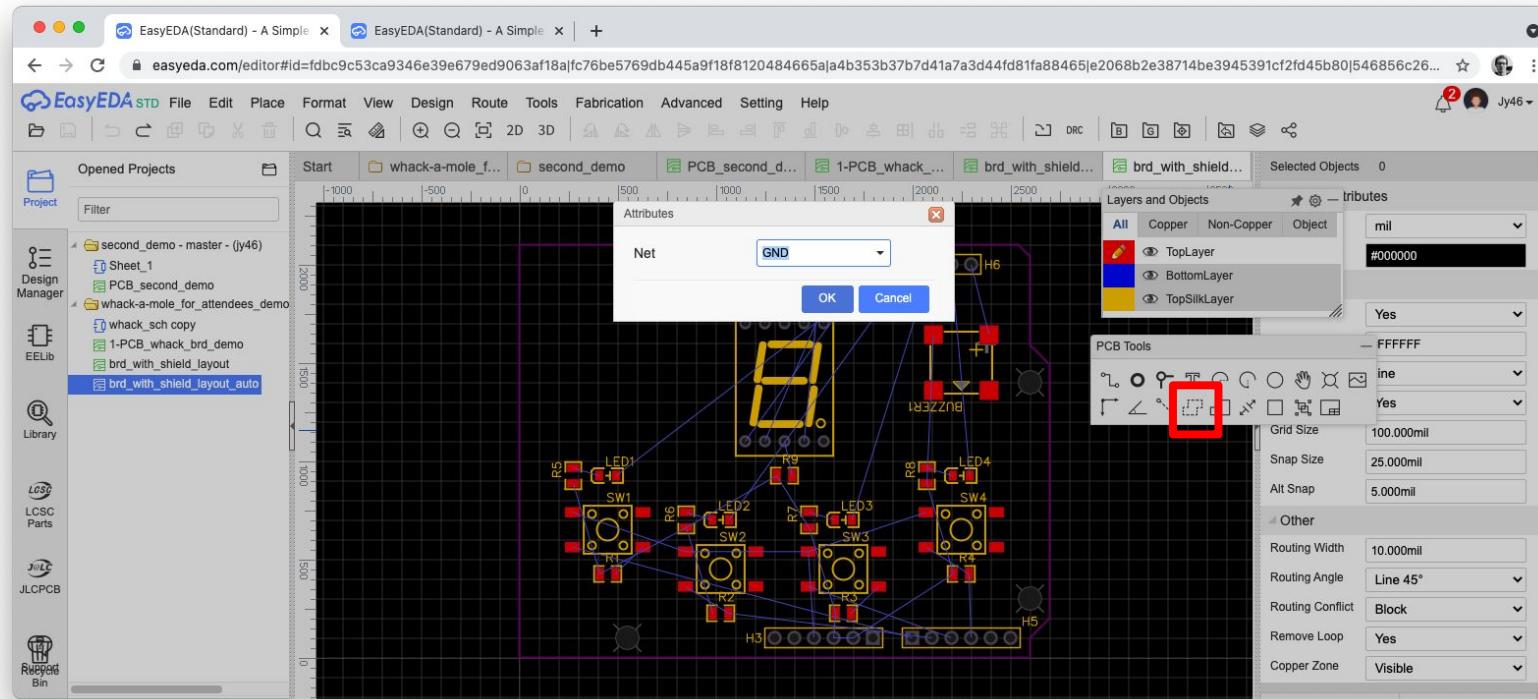
Now Move Components Onto the Board



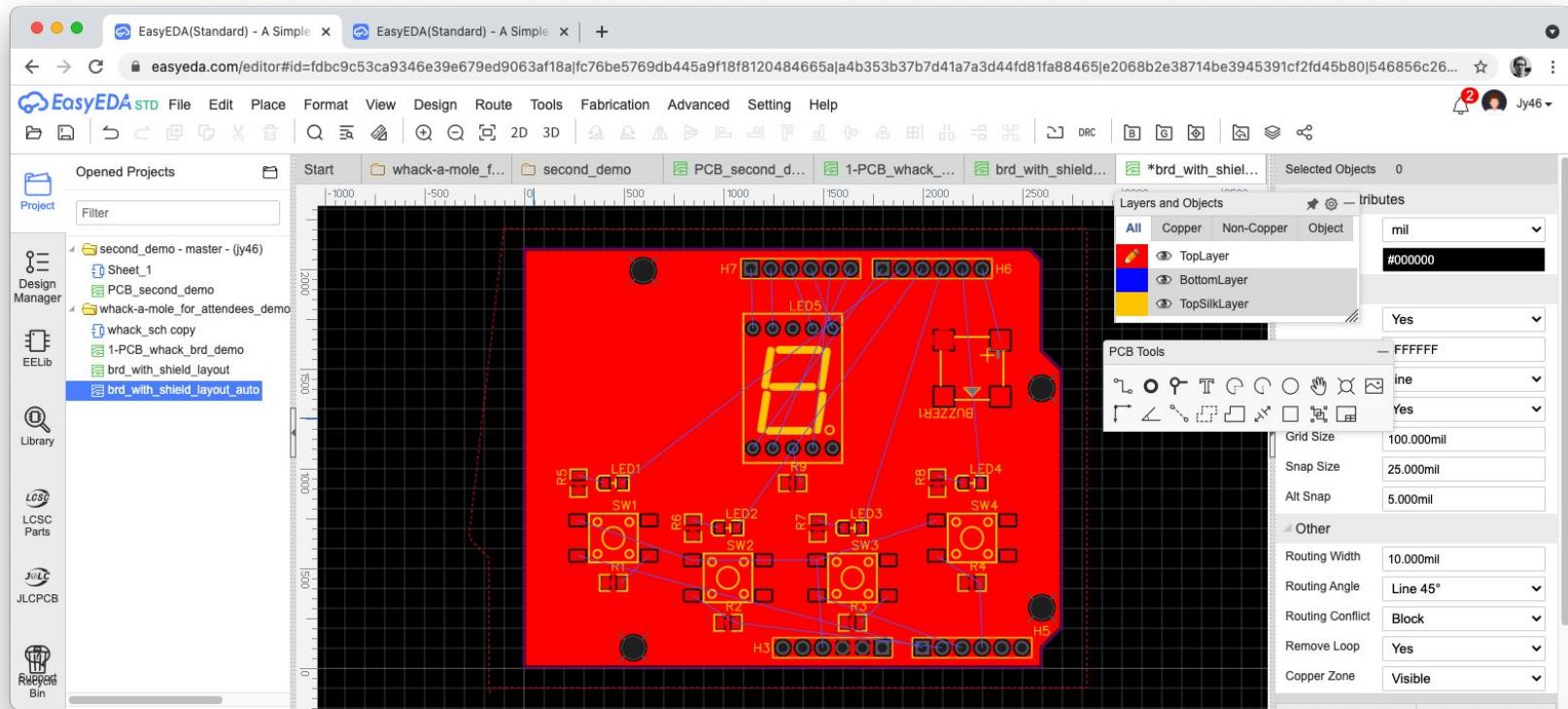
Move Components Around to Get a Design You Like



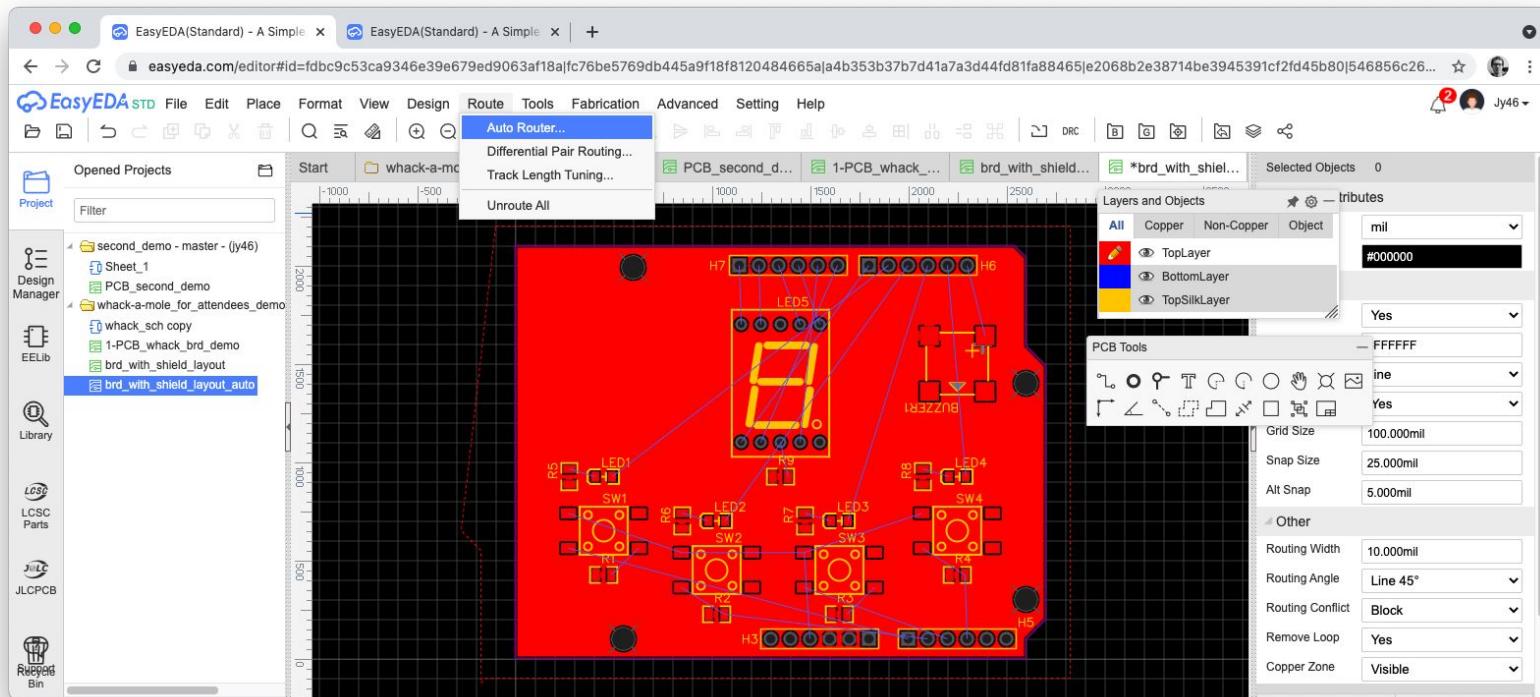
Copper Pour for GND



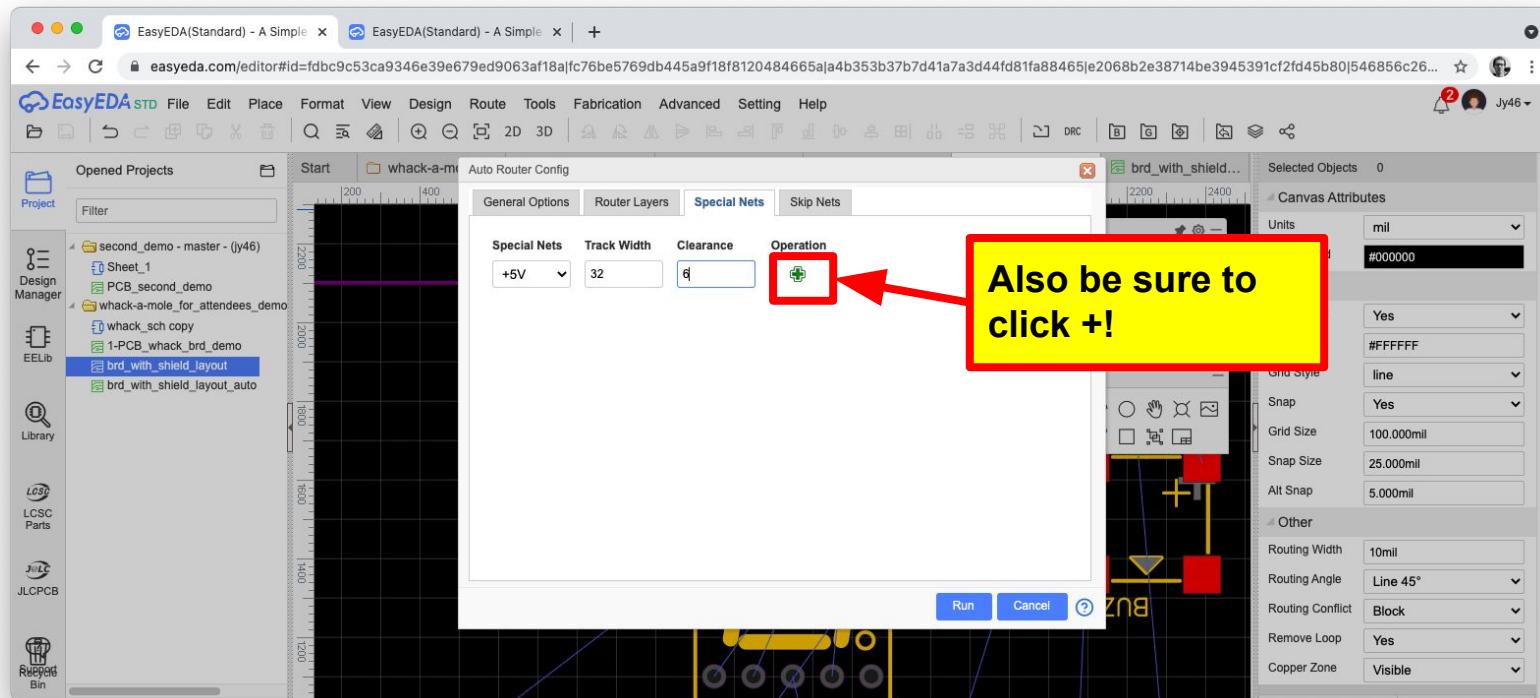
Copper Pour for GND



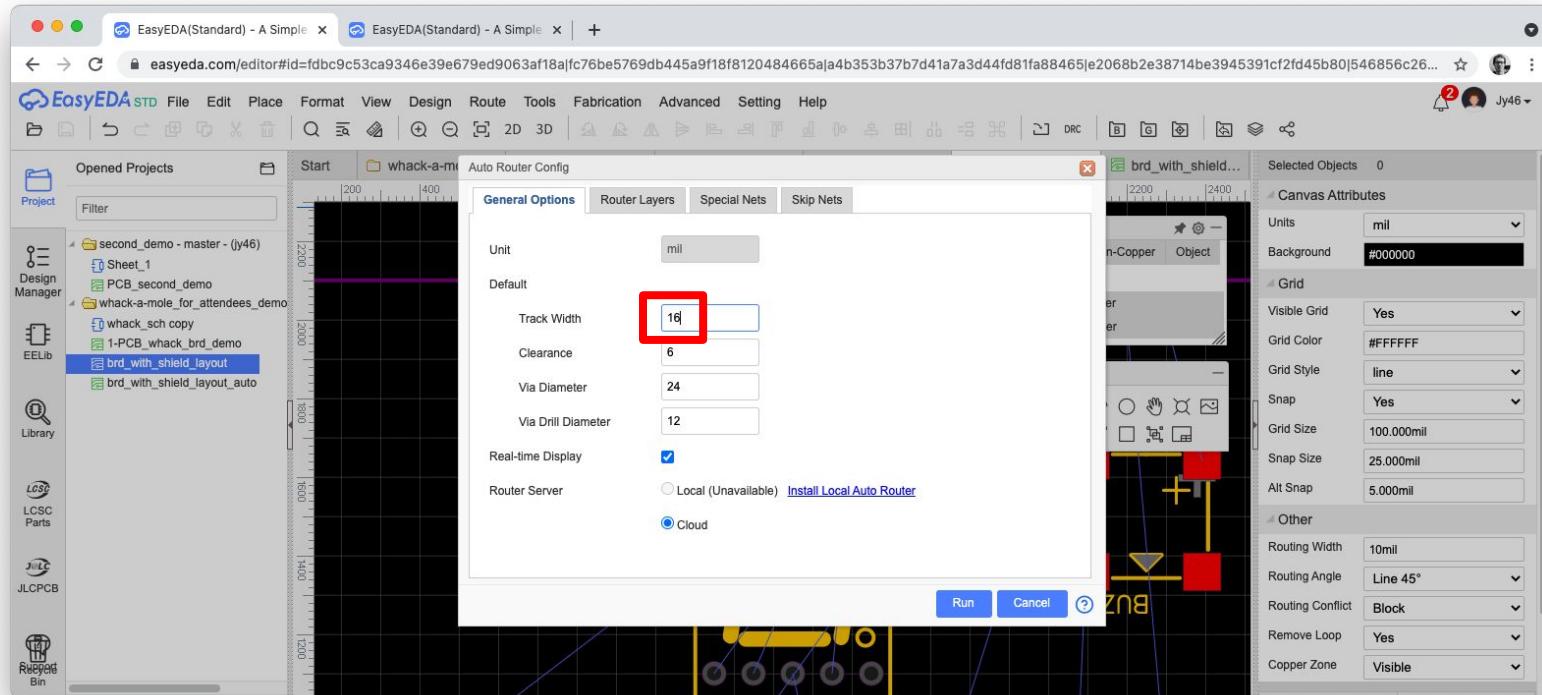
Then Could Just Auto Route



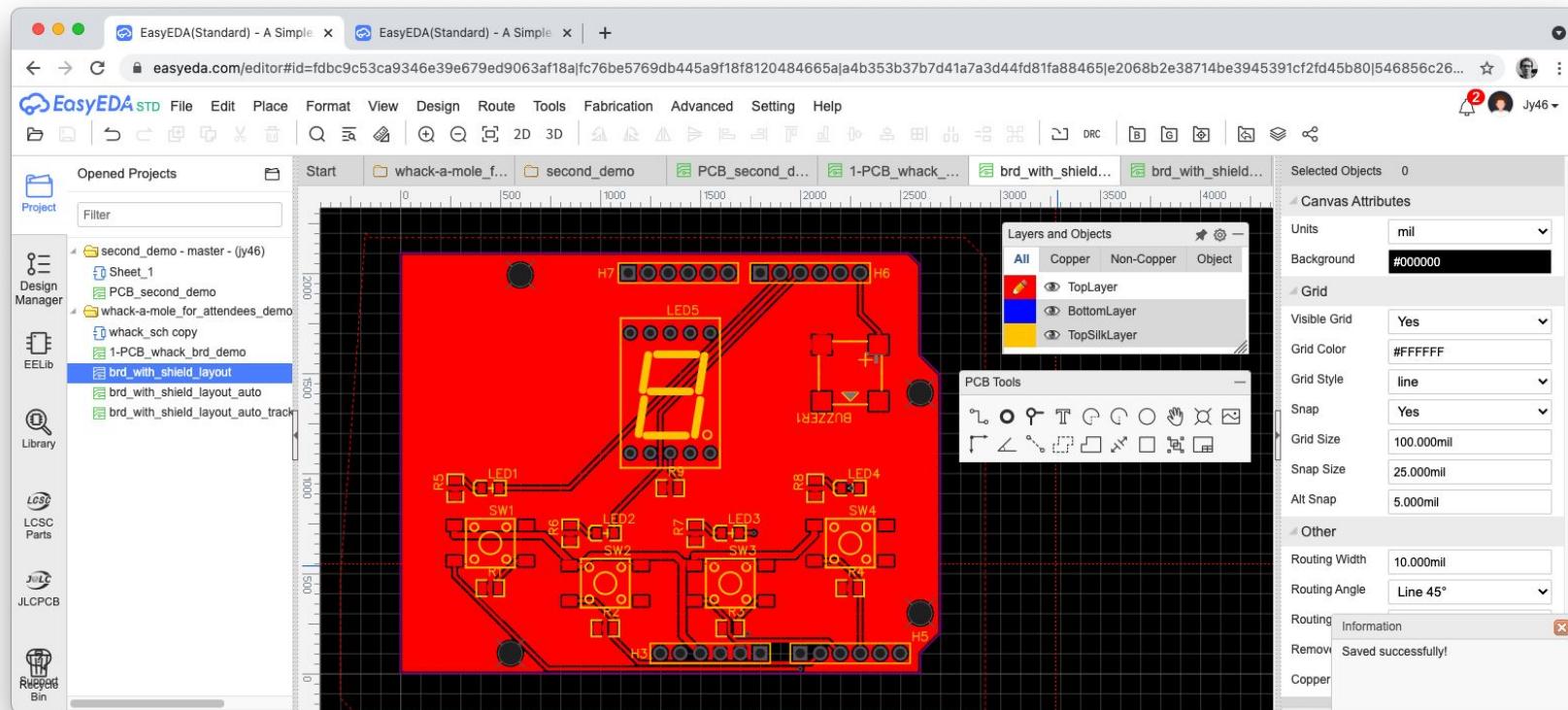
Make Sure +5 V Net is Wider: 32 mil



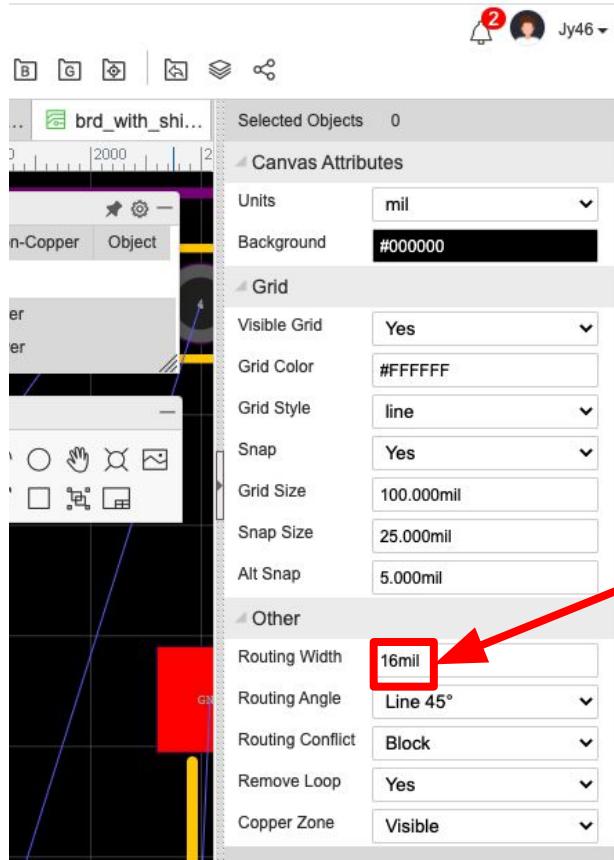
Set Other Track Widths to 16 mil



Auto Routing Result

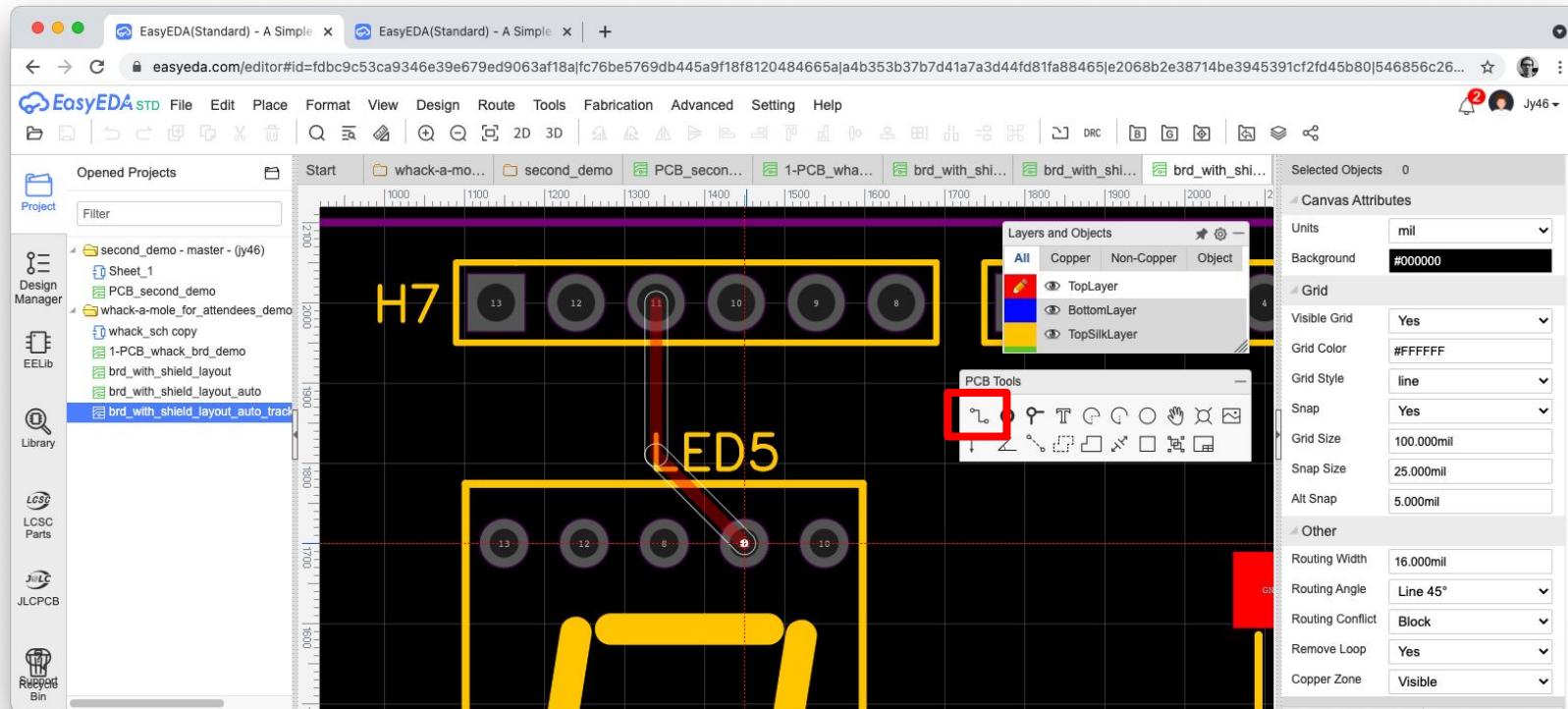


Final Comment: Can do manual routing instead

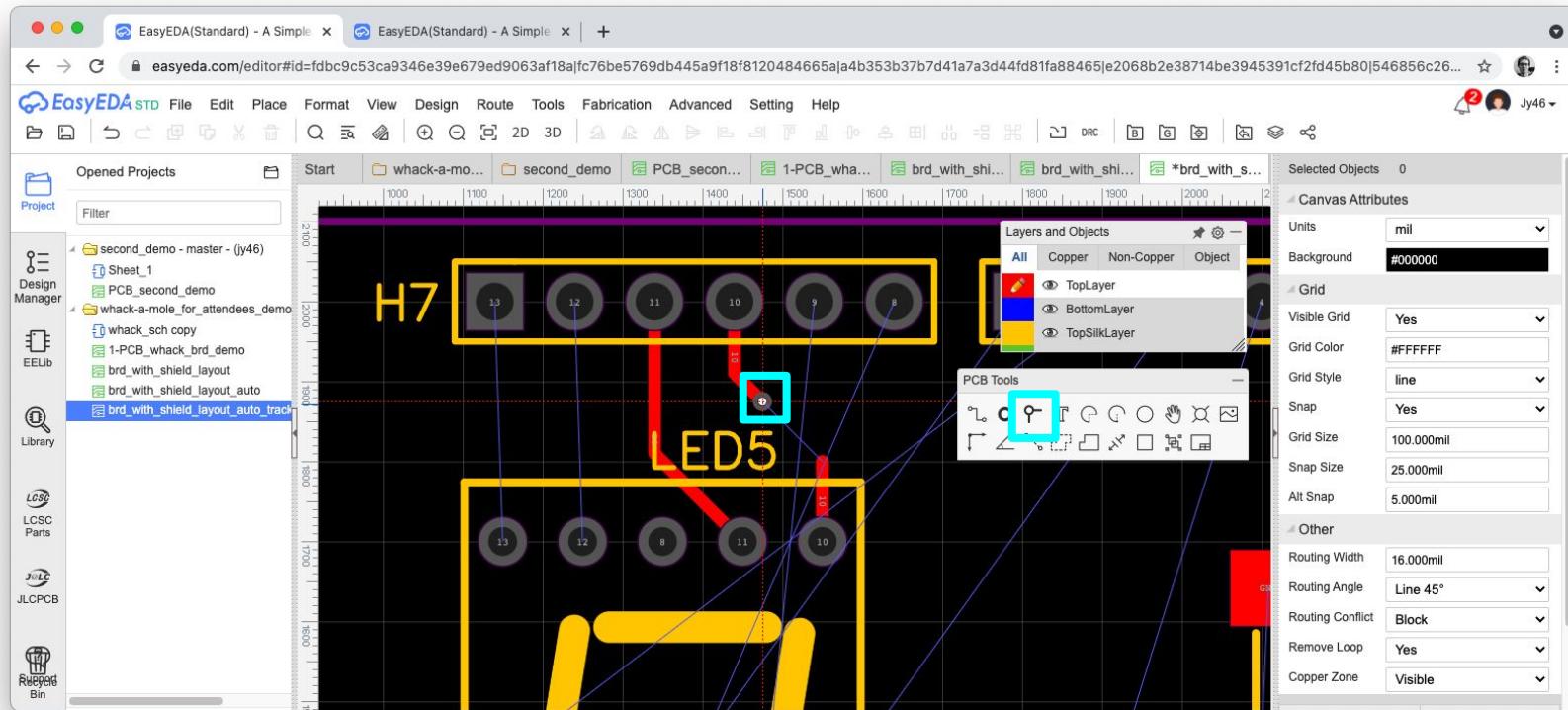


Be sure to set
routing width
appropriately

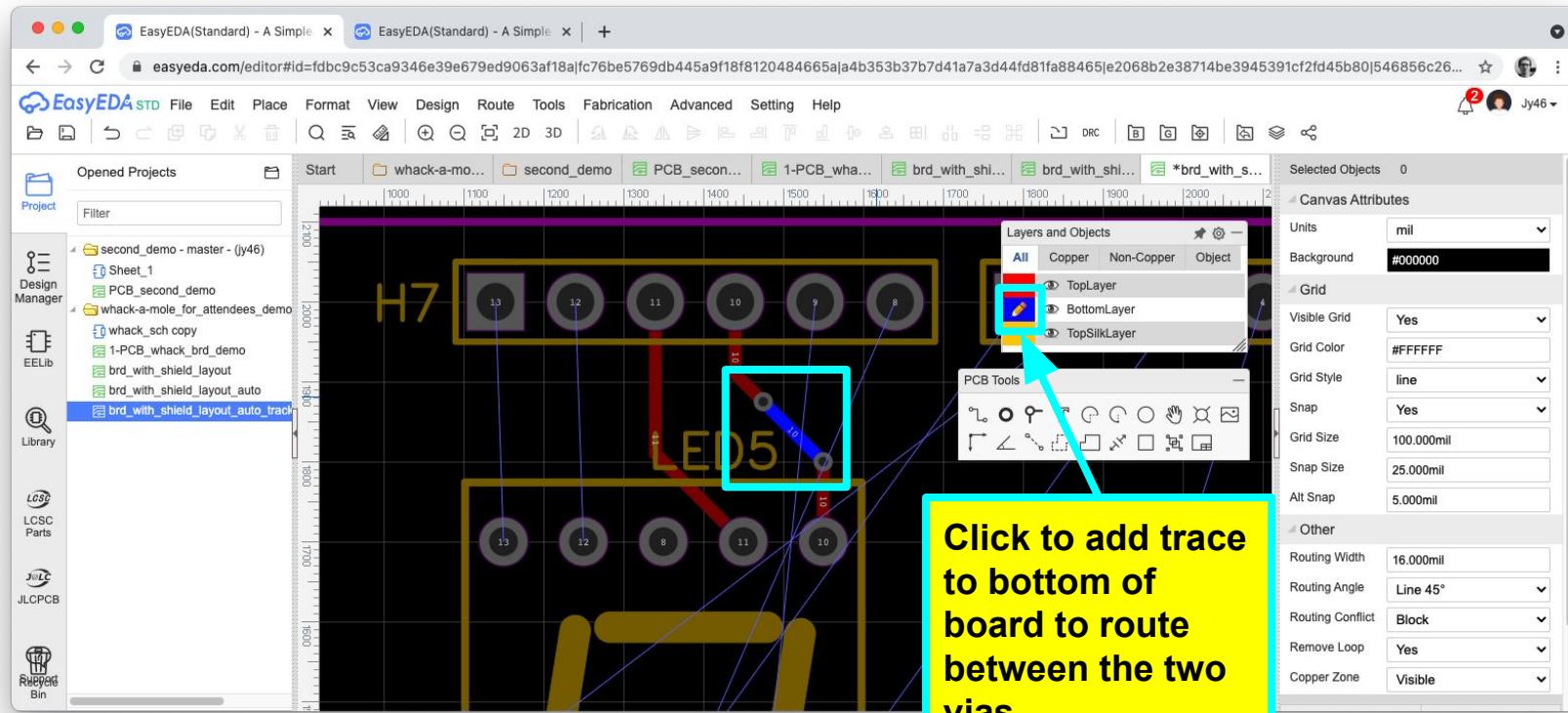
Final Comment 1: Can do manual routing instead



Final Comment 2: Can do manual via placement



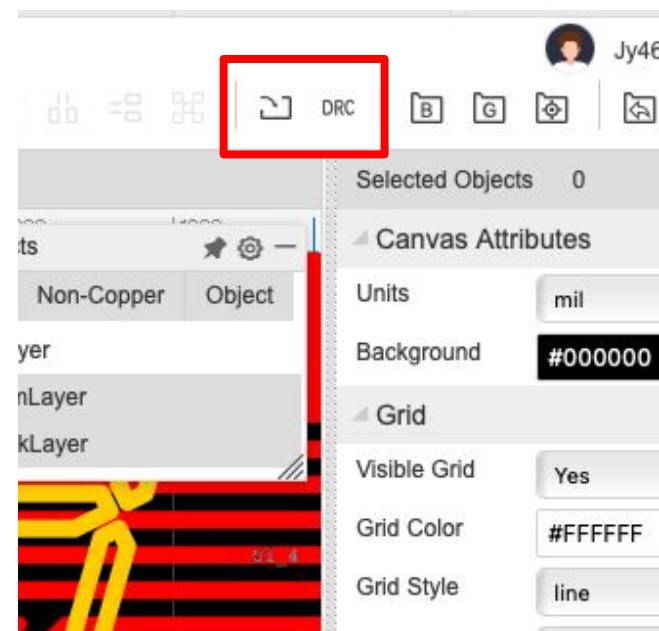
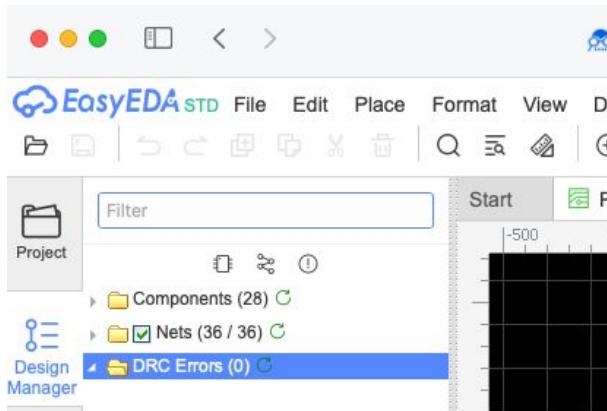
Final Comment 2: Can do manual via placement



Design Rules Check (DRC)

Perform design rules check (DRC)

- Can the fab make the PCB?
- Space between traces
- Keep out zones for components
- Space from edges



Feedback & Scheduling Follow-up

Please spend a few minutes providing feedback on your experience today using
[this form](#)

Email me now or later at jy46@rice.edu if you want to schedule a follow-up

Start with a Design from OSHWLab: <https://oshwlab.com>

The screenshot shows the OSHWLab website interface. At the top, there is a navigation bar with links for Home, Explore, Post, Forum, and a search bar containing the text "Arduino shield". Below the search bar is a dropdown menu titled "Recent searches" with the entry "Arduino shield". To the right of the search bar are buttons for "Share Project", "Login | Register", and a notification bell icon.

The main content area features a large banner with the text "OSHWLab" and "The open source hardware platform based on EasyEDA". Below the banner, there is a section titled "Recommended" with four project cards:

- Low cost logic analyzer probe...**: An image of a breadboard with various components. Statistics: 3.6k views, 5 comments, 11 likes. Posted by **faeton** on 2021-05-25.
- Augmented Reality -- Multim...**: An image of a breadboard with a microcontroller and sensors. Statistics: 96 views, 1 comment, 0 likes. Posted by **dirkx** on 2021-05-25.
- IV-25 Dispaly**: An image of a digital display showing the number "1234567890". Statistics: 183 views, 0 comments, 0 likes. Posted by **fan4tix** on 2021-05-24.
- ArduTest LED**: An image of a breadboard connected to a computer monitor displaying a control interface. Statistics: 31 views, 0 comments, 0 likes. Posted by **mstoffers** on 2021-05-24.

On the right side of the page, there is a sidebar with sections for "Active" and "Official" users, each listing five profiles:

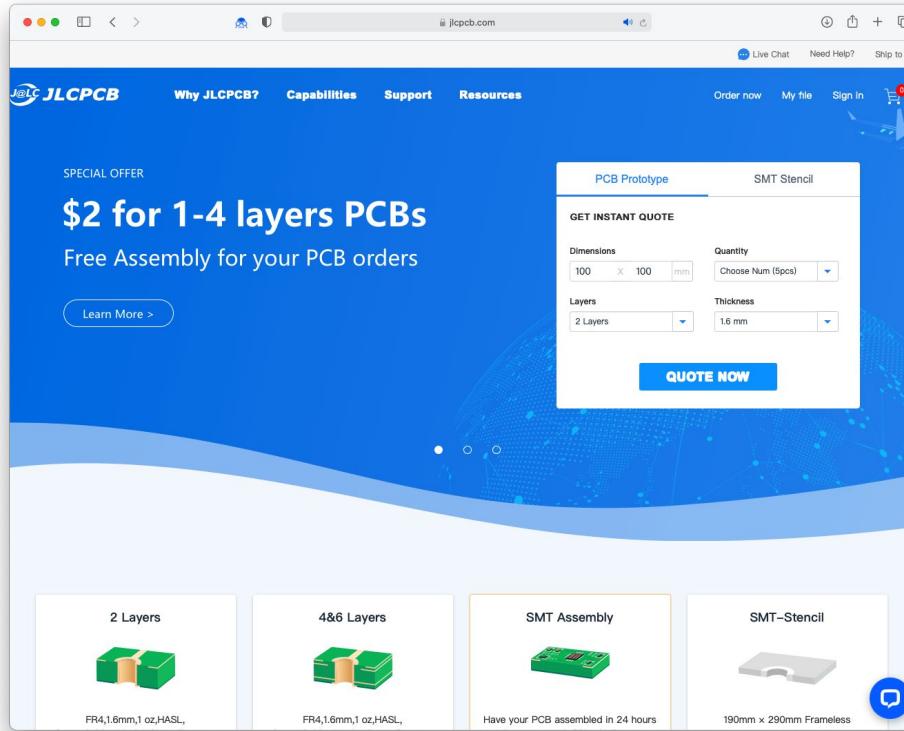
- Stefan Wagner**: Project: 99, Likes: 197
- adrirobot**: Project: 54, Likes: 15
- Daisy**: Project: 20, Likes: 0
- Little_Arc**: Project: 11, Likes: 20
- wegi1**: Project: 53, Likes: 71

At the bottom right of the page, the number "153" is displayed.

Arduino Shield: <https://oshwlab.com/MakotoNinja/arduino-shield>

The screenshot shows a project page on OSHWLab. At the top, there's a navigation bar with a menu icon, the OSHWLab logo, and a search bar. Below the header, the URL is displayed as "Home > Explore > Project detail". On the right side of the header, there's a blue button with the text "Open all in editor" which is highlighted with a red rectangle. The main content area features a user profile for "MakotoNinja" with a circular profile picture of a person with brown hair. Below the profile, the name "MakotoNinja" is displayed. Underneath the profile, there are four metrics: "0 Following", "4 Followers", "0 Likes", and "0 Scores". There are also two buttons: a red "+ Follow" button and a green "Send" button. Below this section, there's a large image of an Arduino shield, which is a breadboard-style PCB with many pins and components. To the right of the image, the project title "Arduino Shield" is shown, along with the creation date "3 years ago" and the open source license "Public Domain". At the bottom of the project card, there are engagement statistics: "3.1k" (views), "0" (comments), "0" (likes), and "11" (stars).

Manufacture: JLCPCB - <https://jlcpcb.com>

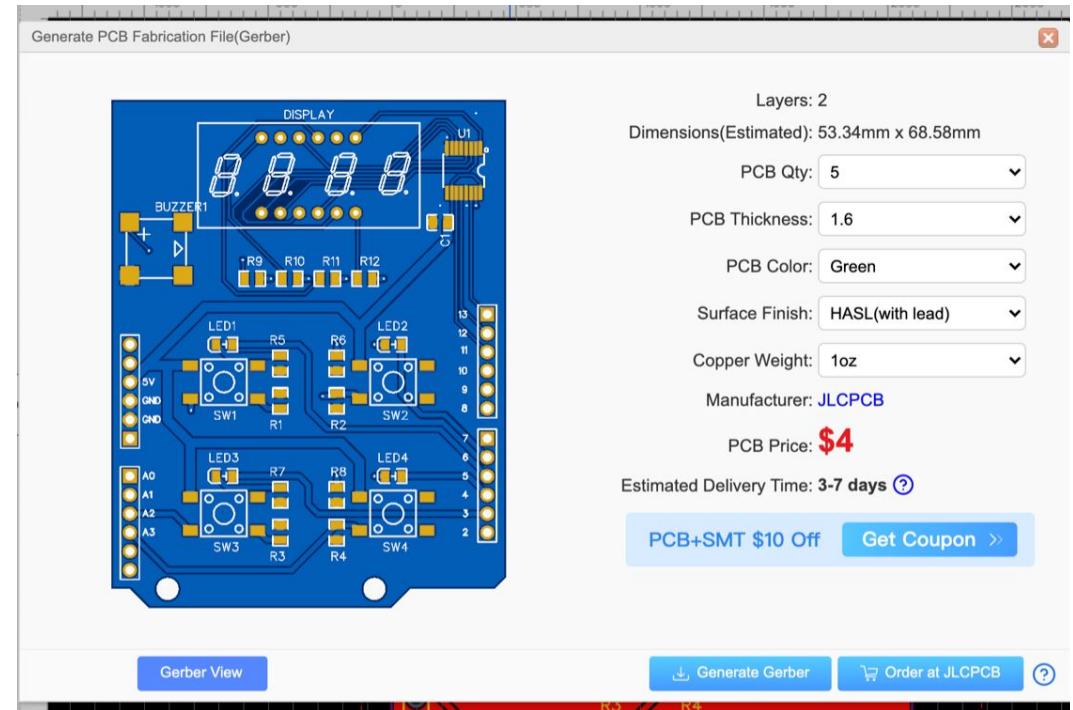
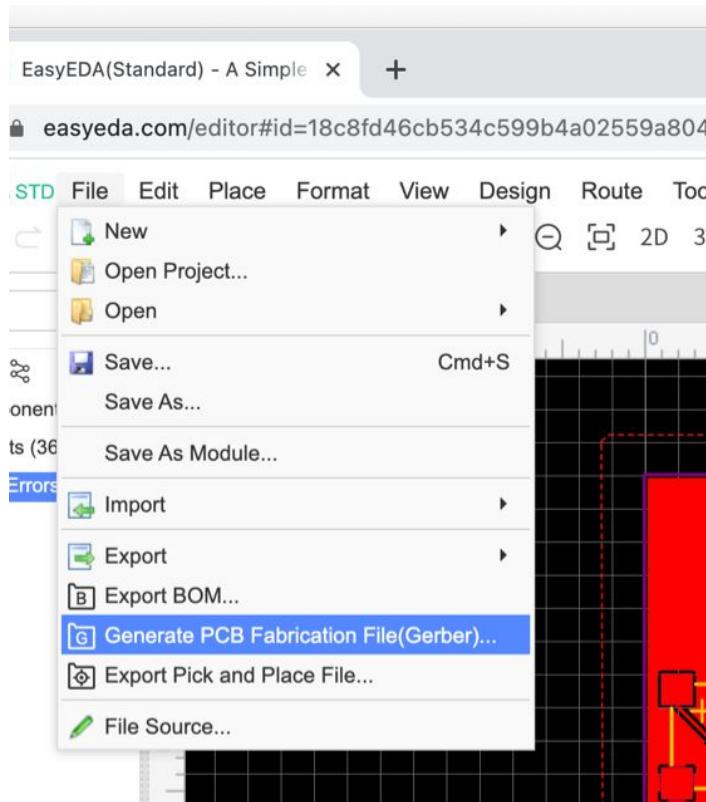


Low cost

Integrated with EasyEDA

Can pay for assembly

Ordering: Start with generating Gerber files



Ordering: JLCPCB site

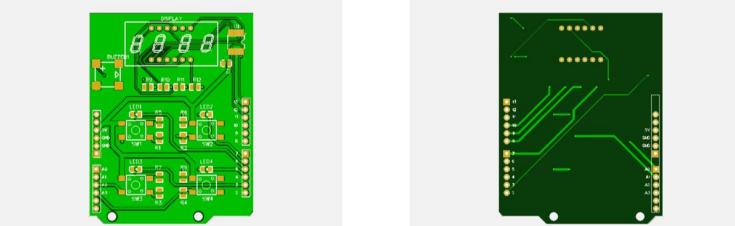
EasyEDA(Standard) - A Simple PCB Prototype & PCB Fabrication

cart.jlcpcb.com/?fileId=983c09a2e1764bb48bec553e45295275&uuid=ddca4ee4fb7a4db1bb2afea2b2b86a1b&achieveDate=72&eadLink=2&electropolishingOnlyNo=no

PCB

Detected 2 layer board of 69x53mm(2.7x2.1 inches).

Your upload has finished processing. Enter the project details below and we'll move on to checking all the individual layers to make sure that they're correct.



SMT-Stencil

Charge Details

Special Offer \$2.00

Build Time 2-3 days \$0.00

Calculated Price \$4.00-\$2.00 Additional charges may apply for [special cases](#)

Weight 0.06kg

SAVE TO CART

Shipping Estimate

Charge: [Choose destination country first](#)

Layers: 1, 2, 4, 6 (Layer 2 is selected)

Dimensions: 69 * 53 mm

PCB Qty: 5

Different Design: 1, 2, 3, 4

Gerber Viewer

Hello ! we are here to help, ask us anything.

JLCPCB

Ordering: Options

The screenshot shows the JLPCB.com PCB ordering interface. On the left, there is a large form for specifying PCB parameters. On the right, there are sections for 'Charge Details' and a 'Calculated Price' summary.

Charge Details

- Special Offer: \$2.00
- Build Time: 2-3 days
- PCB: \$0.00

Calculated Price

\$4.00 - \$2.00

Additional charges may apply for special cases

Weight: 0.06kg

SAVE TO CART

Shipping Estimate

Charge: Choose destination country first

Help Message

Hello ! we are here to help, ask us anything.

JLPCB

Form Fields (Left Side)

- Layers: 1, 2, 4, 6 (2 is selected)
- Dimensions: 69 * 53 mm
- PCB Qty: 5
- Different Design: 1, 2, 3, 4 (1 is selected)
- Delivery Format: Single PCB, Panel by Customer, Panel by JLPCB (Single PCB is selected)
- PCB Thickness: 0.4, 0.6, 0.8, 1.0, 1.2, 1.6, 2.0 (1.6 is selected)
- PCB Color: Green, Red, Yellow, Blue, White, Black (Green is selected)
- Surface Finish: HASL(with lead), LeadFree HASL-RoHS, ENIG-RoHS (HASL(with lead) is selected)
- Outer Copper Weight: 1 oz, 2 oz (1 oz is selected)
- Gold Fingers: No, Yes (No is selected)
- Confirm Production file: No, Yes (No is selected)
- Flying Probe Test: Fully Test, Not Test (Fully Test is selected)
- Castellated Holes: No, Yes (No is selected)
- Remove Order Number: No, Yes (No is selected), Specify a location

Ordering: SMT Assembly

Only accept single sided placement, we recommend you choose the side which has more SMD parts.

*The parts that are not in [JLCPCB SMT Parts Library](#) won't be placed on your board.

*Panels with V-cut can not be made with SMT assembly, please panelize PCBs with stamp holes.

Assemble top side Assemble bottom side

SMT QTY: 5 2

Tooling holes: Added by Customer Added by JLCPCB

I agree to the Terms and Conditions of JLCPCB SMT Service.

Charge Details

Special Offer	\$2.00
SMT Price	\$7.00
Setup Fee	\$7.00
Stencil	\$1.50
Panel	\$0.00
Large Size	\$0.00
Components	-
Extended components fee	-
SMT Assembly	-

Build Time

PCB:	<input checked="" type="radio"/> 2-3 days	\$0.00
SMT:	72 hours	

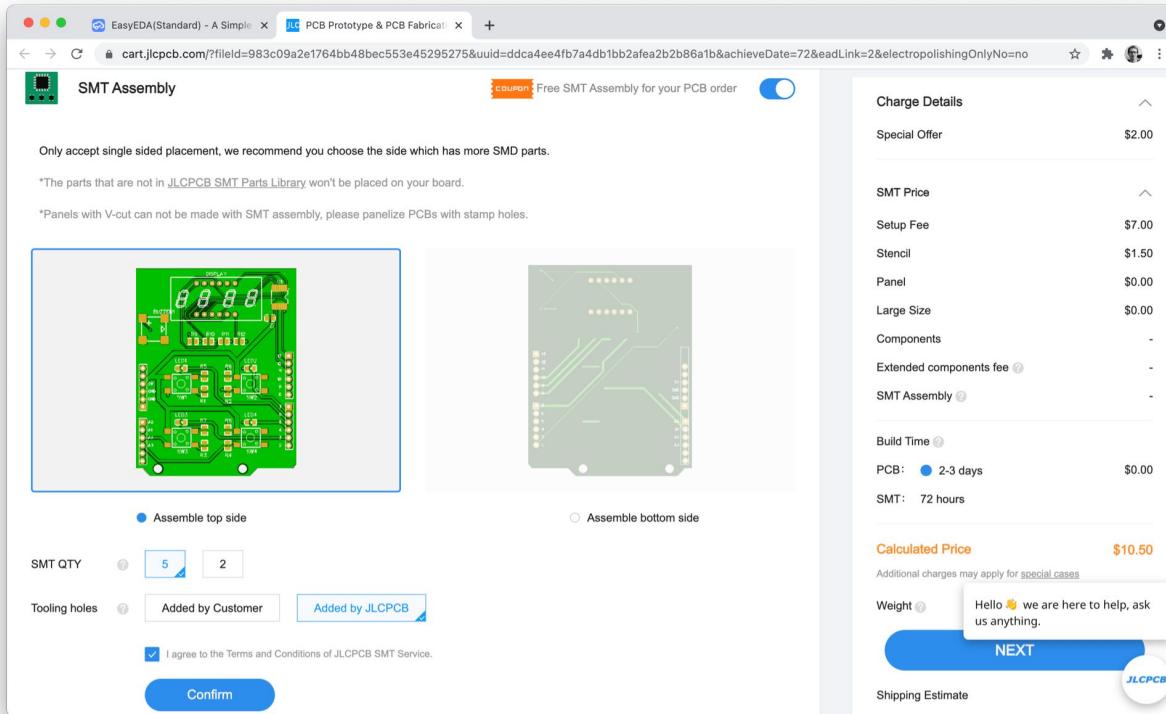
Calculated Price \$10.50

Additional charges may apply for [special cases](#).

Weight: Hello! We are here to help, ask us anything.

NEXT

Shipping Estimate



Ordering: Uploading Other Files

The image displays two screenshots illustrating the process of uploading files for PCB fabrication.

Left Screenshot (EasyEDA Software): Shows the software's main menu with the "File" option open, highlighting the "Export BOM..." option. The menu also includes "Import", "Generate PCB Fabrication File(Gerber)...", "Export Pick and Place File...", and "File Source...".

Right Screenshot (JLCPCB Ordering Page): Shows the "cart.jlcpcb.com/smtQuote" page. The top navigation bar includes links for "Why JLCPCB?", "Capabilities", "Support", "Resources", "Order now", "My file", and a user profile. The main area features four buttons: "PCB" (with a gear icon), "Upload BOM/CPL" (with a document icon), "Select Parts" (with a checkmark icon), and "Quote" (with a calculator icon). Below these buttons are two large blue "Add" buttons: "Add BOM File" (with an up arrow icon) and "Add CPL File" (with an up arrow icon). Each button has a note below it: "Only accept XLS,XLSX,CSV." for the BOM file and "Pick&Place File, Only accept XLS,XLSX,CSV." for the CPL file. There are also "View Sample BOM" and "View Sample CPL" links. A message at the bottom left says, "Not sure where to start? Check our [SMT FAQs page](#)." A tip at the bottom center states, "Tips: With EasyEDA, you can generate BOM/CPL files with a single click." A live chat window on the right side asks, "Hi, let us know if you have any questions." and provides a text input field with a "Write a message..." placeholder and a send button. A cookie consent banner at the bottom of the page reads: "We use cookies to offer you a better experience. Detailed information on the use of cookies on this website is provided in our [Privacy Policy](#). By using this site, you consent to the use of our cookies." with "Accept" and "Reject" buttons.

Ordering: JLC recognizes parts

The screenshot shows the JLCPCB SMT Quote tool interface. At the top, there are tabs for "EasyEDA(Standard) - A Simple" and "JLC PCB Prototype & PCB Fabrication". The main header includes the JLCPCB logo, navigation links for "Why JLCPCB?", "Capabilities", "Support", and "Resources", and user account links for "Order now", "My file", and "jy46". A shopping cart icon shows 0 items.

The workflow steps are displayed as icons: "PCB" (with a circuit board icon), "Upload BOM/CPL" (with an upload icon), "Select Parts" (with a checkmark icon), and "Quote" (with a calculator icon). Below this, a message says "Automatically saved, last updated by 13:56".

Under the "Top Side" section, it says "Select the parts you want to assemble on your boards. No restrictions on using extended parts for each order now." and shows statistics: "Total 9 parts detected", "9 Parts confirmed", and "0 parts not selected".

The main table is titled "Uploaded BOM Data" and "Review Matched Parts". It has columns for Designator, Name, Footprint, Matched Part Detail, Qty, Unit Price, and Operate (with a checkbox). The table lists three parts:

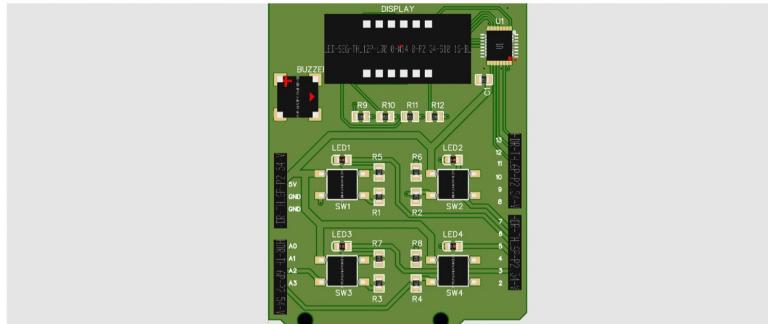
Uploaded BOM Data			Review Matched Parts				
Designator	Name	Footprint	Matched Part Detail	Qty	Unit Price	Operate	
SW1,SW2,SW3,SW4	K2-1102SP-C4SC-04	KEY-SMD_4P-L6.0-W6.0-P4.50-LS9.5	Korean Hroparts Elec K2-110... SMD-4_6.0x6.0x4.5P C127509	Extended Part 4	\$0.0297	<input checked="" type="checkbox"/>	
DISPLAY	FJ3461AH	LED-SEG-TH_12P-L30.0-W14.0-P2.54-S10.16-BL	Shenzhen Zhihao Elec FJ346... LED-SEG-TH_12P-L30.0-W14.0-P2.54-S10.16-BL [Hand-soldering] C10708	Extended Part 1	\$0.5985	<input checked="" type="checkbox"/>	
U1	CD74HC595SM96	SSOP-16_L5.0-W4.4-P0.65-L6.4-BL	Texas Instruments CD74HC5... SSOP-16_5.3x6.2x0.65P C553167	Extended Part 1	\$0.5607	<input checked="" type="checkbox"/>	

Ordering: Parts Placement

EasyEDA(Standard) - A Simple PCB Prototype & PCB Fabrication

cart.jlcpcb.com/smtQuote

Review Parts Placement



The parts placement is for reference purposes only. Our engineer will review and fix the components orientation in 4-6 hours after you place the order. You can check the result in your order history.

Selected Parts(9 items)

Part Detail	Selected By	Designator	Price
Shenzhen Zhihao Elec FJ346...			
Extended Part C10708			
4 0.36"(9.14mm) Cathode(Ne...			
[Hand-soldering]			
		DISPLAY	Qty: 5*1 Ext Price: \$3.5910

Download Selected Parts List

Charge Details

PCB Price	\$2.00
Special Offer	\$2.00
SMT Price	\$10.09
Setup fee	\$7.00
Stencil	\$1.50
Panel	\$0.00
Large Size	\$0.00
Components	\$10.09
Extended components fee	\$0.00
SMT Assembly	\$0.91
Hand-soldering labor fee	\$3.50
Manual Assembly	\$3.96

Build Time

PCB:	2-3 days	\$0.00
SMT:	120 hours	

Total Price

\$3.96

Weight: 110g

Ordering: Checkout

The screenshot shows a web browser displaying the JLCPCB shopping cart page at cart.jlcpcb.com/cart. The page is titled "SHOPPING CART" and "SUMMARY (2 items)".

SHOPPING CART

Item	Qty	Price
5a32ccedc3334f6fb58f0022e2aec267_16... PCB prototype: Y5-3454886A Green, 1.6 thickness, HASL(with lead) 2-3...	5	\$2.00 \$4.00
5a32ccedc3334f6fb58f0022e2aec267_16... SMT Assembly: SMT0210601206541-345... Assemble top side, 120	5	\$35.96

SUMMARY (2 items)

Subtotal	\$37.96
Weight	0.11kg

Shipping calculated at checkout

Secure Checkout

Add new item

Payment methods: VISA, MasterCard, AMEX, DISCOVER, PayPal, JPBank

SSL ENCRYPTED PAYMENT

COMPANY

- About JLCPCB

SUPPORT

- Help Center

NETWORK SITES

- EasyEDA

JLCPCB CONNECT WITH