Final Project: PCB Design

A. OBJECTIVES

- Design and simulate a custom analog or mixed-signal circuit
- Capture the schematic using PCB layout software
- Design a printed circuit board (PCB) and submit files for fabrication
- Assemble the board and solder all component
- Test, troubleshoot, and demonstrate a fully functional prototype

B. EQUIPMENT REQUIRED

- Digital Multimeter
- Breadboard
- Dual Trace Oscilloscope
- Function Generator
- Multi-channel Power supply
- Miscellaneous Cables
- Computer Access
- Solder Station (Soldering Iron & Lead-Free Flux Core Solder preferred)
- Other Equipment as Requested and Available

C. PARTS REQUIRED

- Your BOM (Bill of materials) can vary widely on your Design choice
- $50-$75 budget is mandatory for outsourced manufacturing costs

D. PRIOR TO PROJECT

Project Proposal
Any lab assignment qualifies if it meets the minimum parts requirements, or you may propose a new circuit. Submit a short proposal (due date TBD) that includes:

- A preliminary schematic
- Simulation results & supporting calculations showing expected performance
- A tentative parts list

Circuit Design Requirements
Your custom circuit must contain at least:

- Ten passive components (resistors, capacitors, inductors, etc.)
- Three active components (operational amplifiers or discrete transistors)
- Minimum trace width: 10 mils
- Minimum trace space: 10 mils

Test Plan

You will develop your own test plan for this circuit. Your plan should describe (in sufficient detail to be understood by a test technician with your own level of lab experience) the tests which will be done to check the circuits performance against all the specifications. Your test plan should explicitly address the function of your chosen design.

E. PROJECT TUTORIAL

IMPORTANT NOTE: In this Tutoria1, we will be using EasyEDA. EasyEDA is a Free to use web-based/cloud-based EDA tool that is fully integrated with JLCPCB. This means you can go from schematic to ordering your PCB all in one place quickly and easily. EasyEDA, however, is not an industry standard EDA tool like Altium or Autodesk Eagle and is meant for small projects and/or quick prototyping. To learn more about hardware design it is highly recommended to research PCB design techniques and best practices from https://www.youtube.com/@AltiumAcademy . Also for a more detailed tutorial and introduction to EasyEDA it is recommended to watch this video: https://www.youtube.com/watch?v=MsdJgEinb34&t=22s

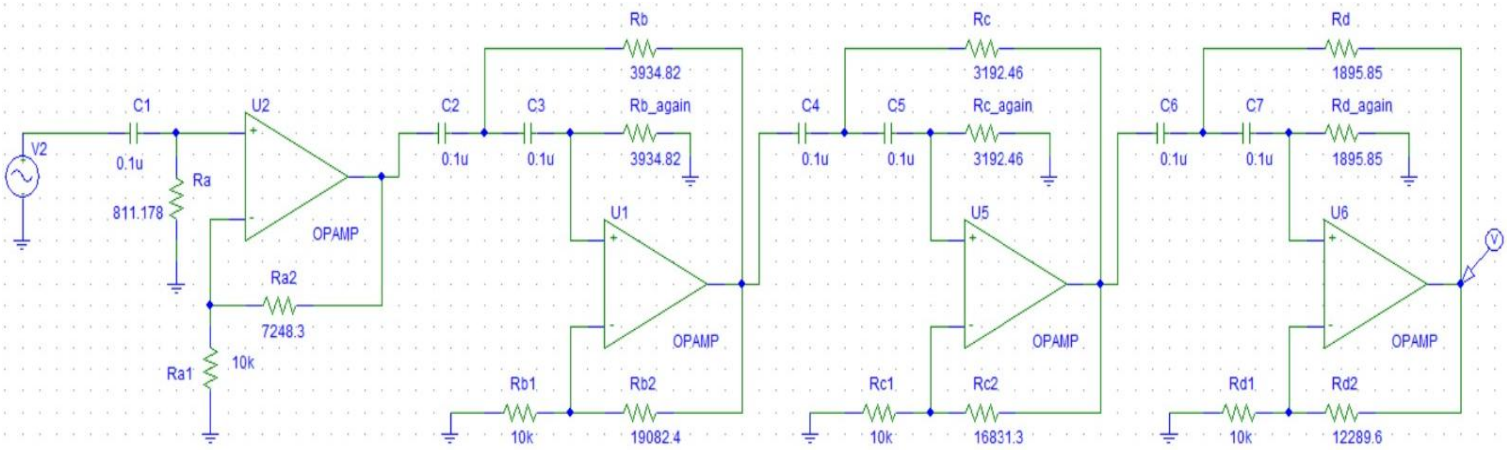
Project Proposal

(1) As stated in the "PRIOR TO PROJECT" section, your Proposal must include A preliminary schematic, simulations & supporting calculations showing expected performance, and a tentative parts list. **For example:**

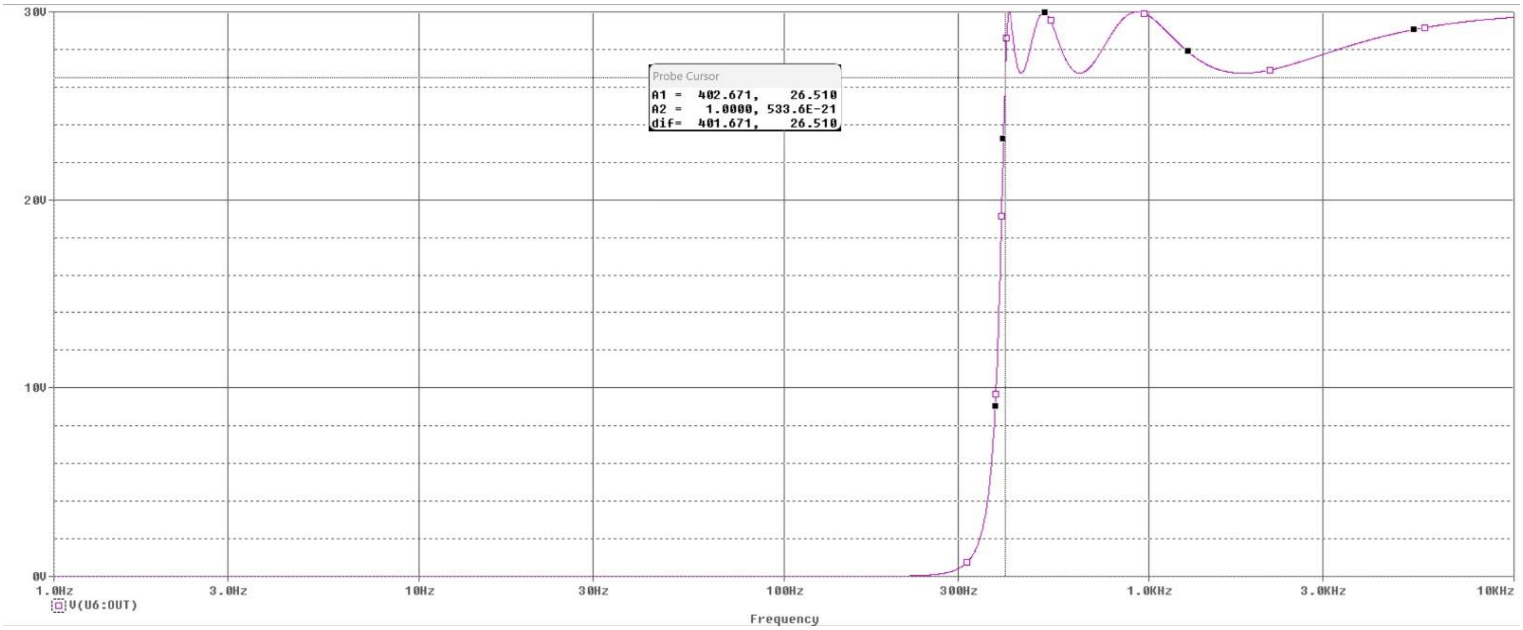
Final Project Proposal: Chebychev 1dB 7th order HPF

Filter	Specification
Chebyshev High-Pass	Cutoff Frequency: 403 Hz
	Passband Gain: 30 V/V
	Passband Ripple: ~1 dB

Table 1: Design target performance



Schematic 1: Preliminary Schematic



Simulation 1: Simulation showing expected performance

PART		
Opamp	**Resistor**	**Capacitor**
4xTL081 (or LM741)	15x 0.5W fixed resistor	7x 0.1uf Fixed capacitors

Table 2: Tentative parts list

Breadboard Prototype

(1) Before designing our PCB, it is crucial to build and test our preliminary schematic on a breadboard to ensure proper connections in your circuit and successful circuit operation. The **Example** Shown previously was fully assembled and tested for operation on a breadboard by running a frequency sweep shown below:

IMPORTANT NOTE: You will need to use your Engineering judgment to decide what methods of testing are best to use when testing your circuit.

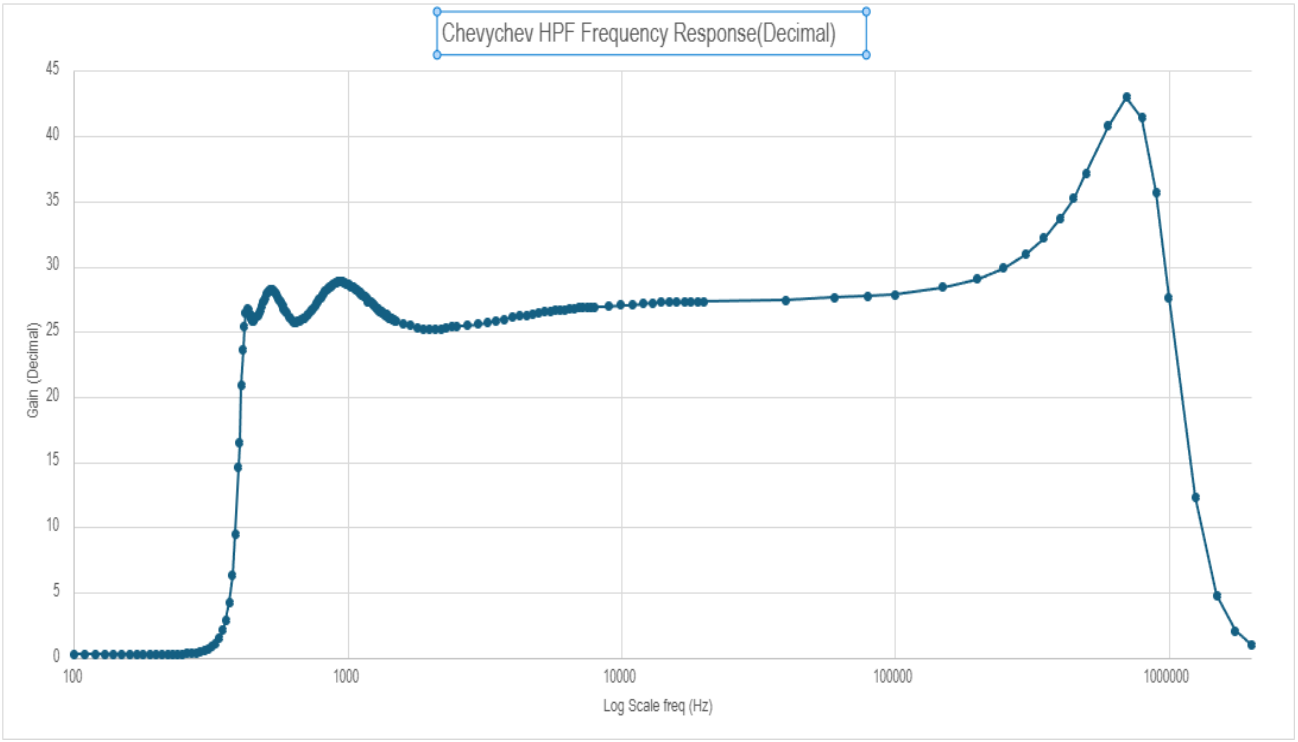


Chart 1: HPF Frequency response in Decimal

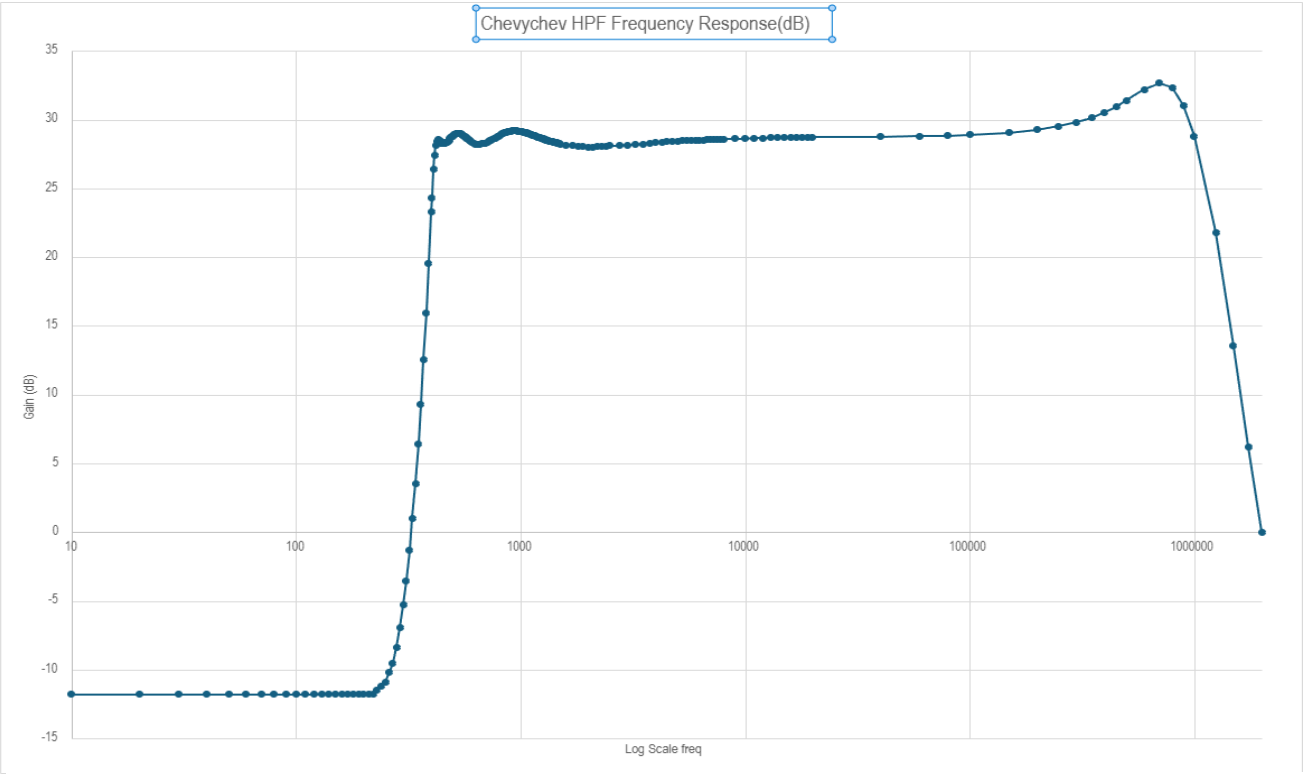


Chart 2: HPF Frequency response in dB

(2) Once you have confirmed the operational function of your schematic through both simulation testing and breadboard testing, we can now move onto the PCB design portion of your project.

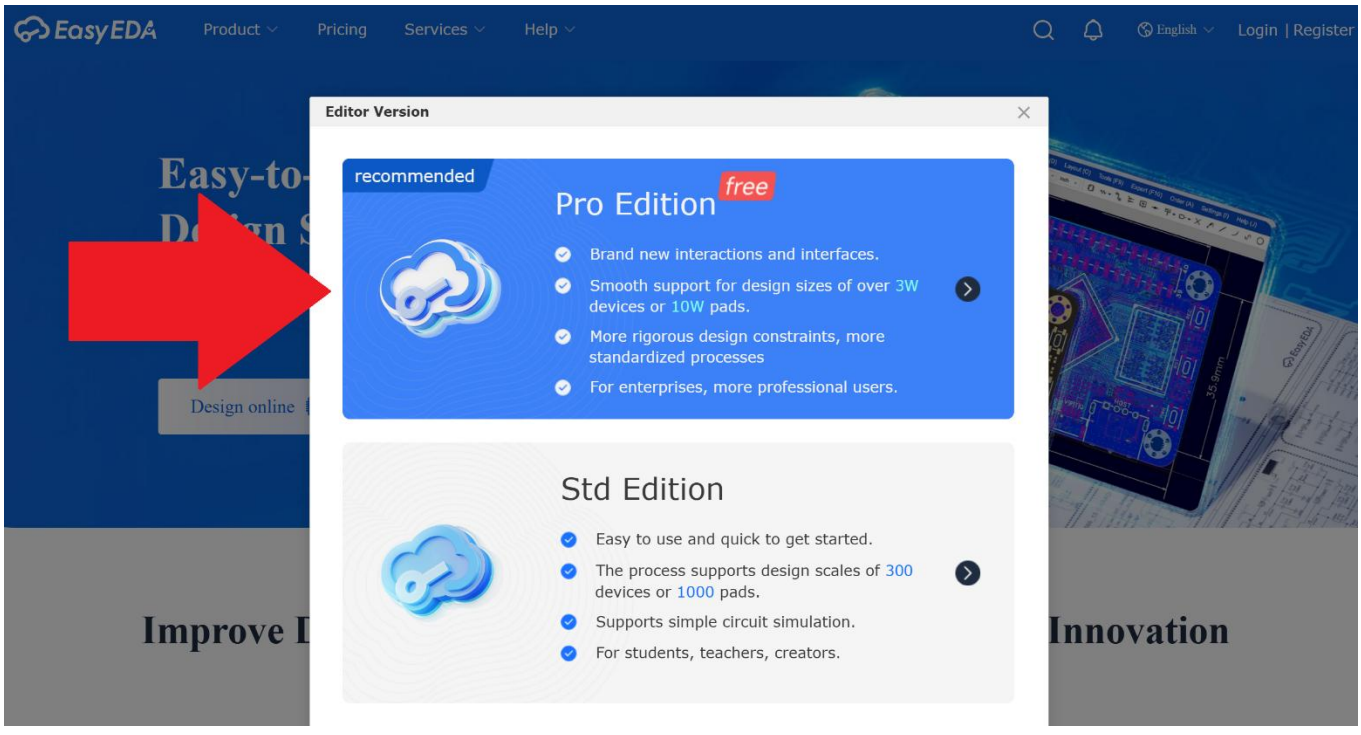
PCB Design

(1) Navigate to https://easyeda.com and select “Design online”.

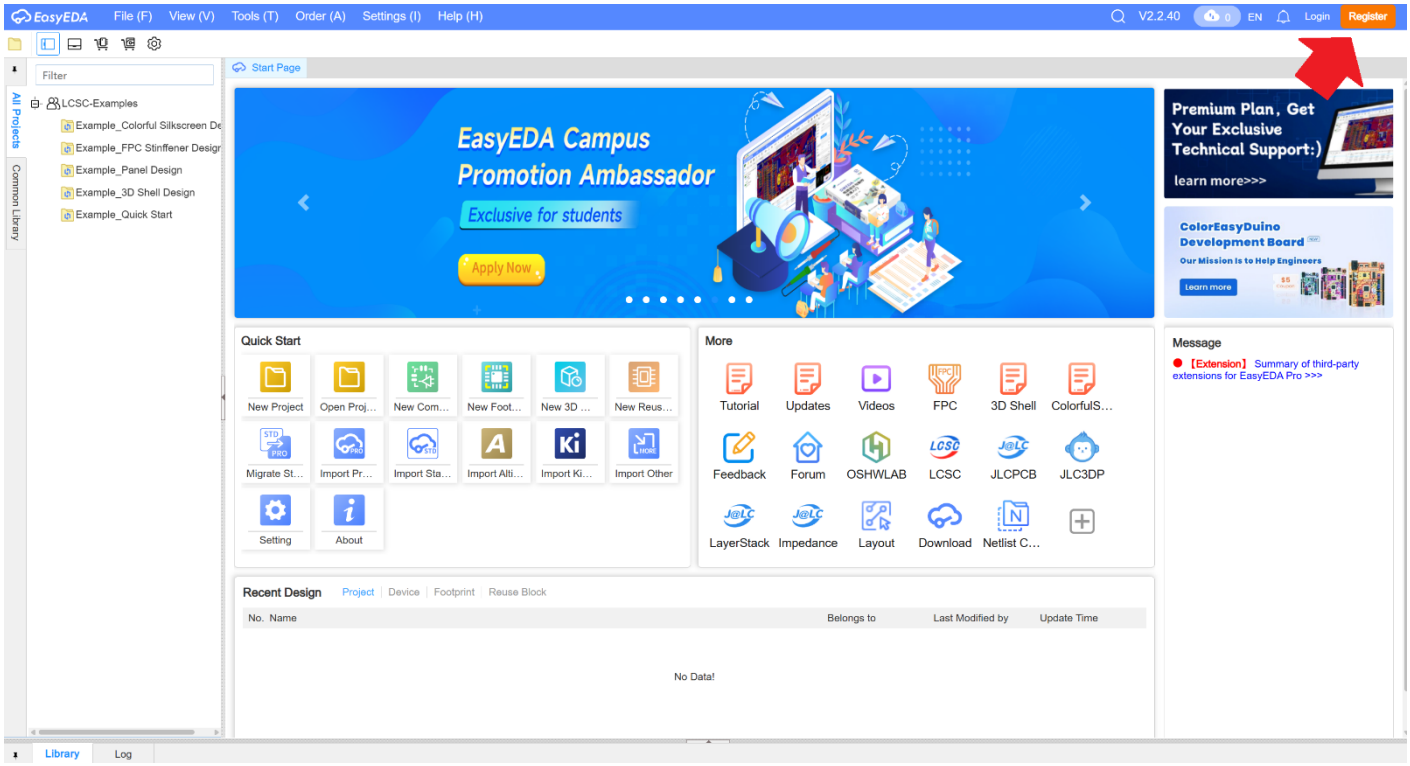
Improve Design Efficiency in Business and Accelerate Innovation

The world's first EDA software vendor with a full supply chain solution

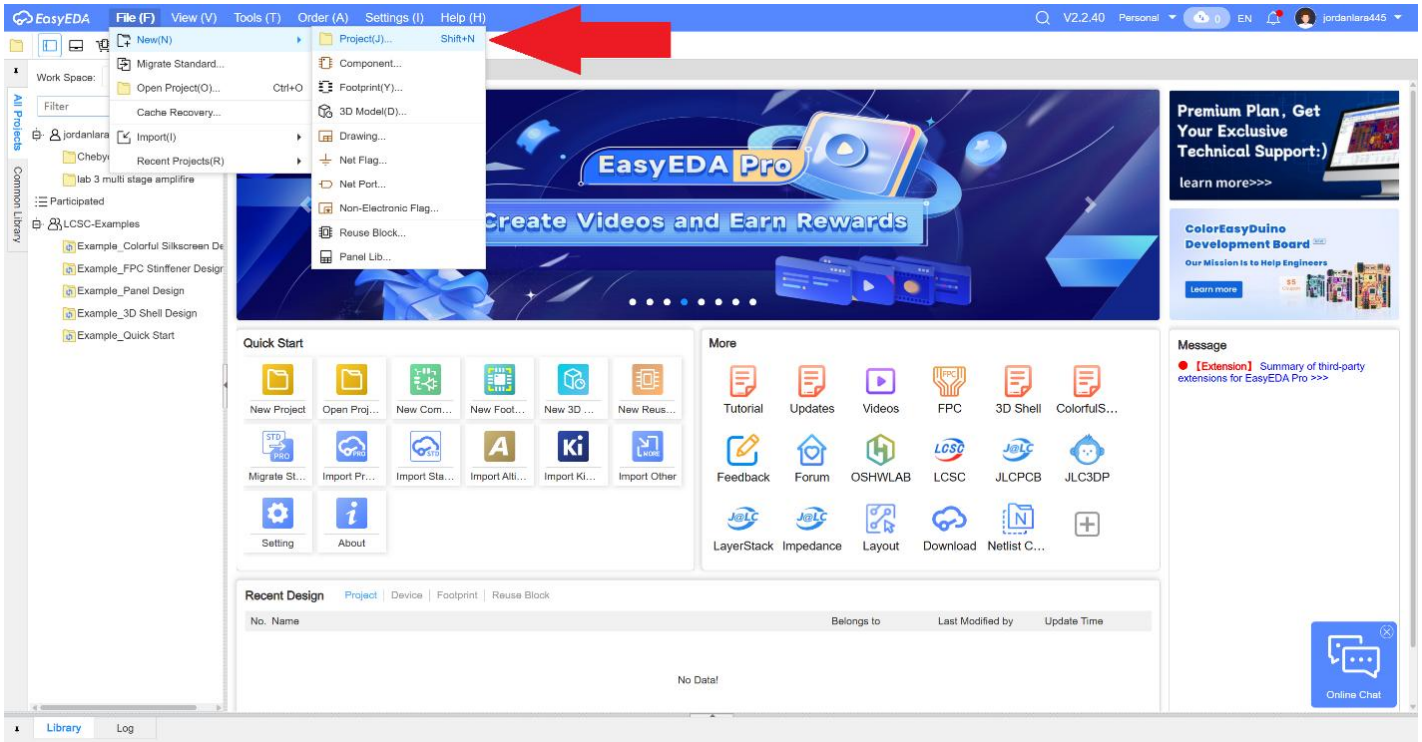
(2) You will be prompted to select an STD edition or Pro Edition. For this Tutorial we will use the Pro Edition, select the “Pro Edition”



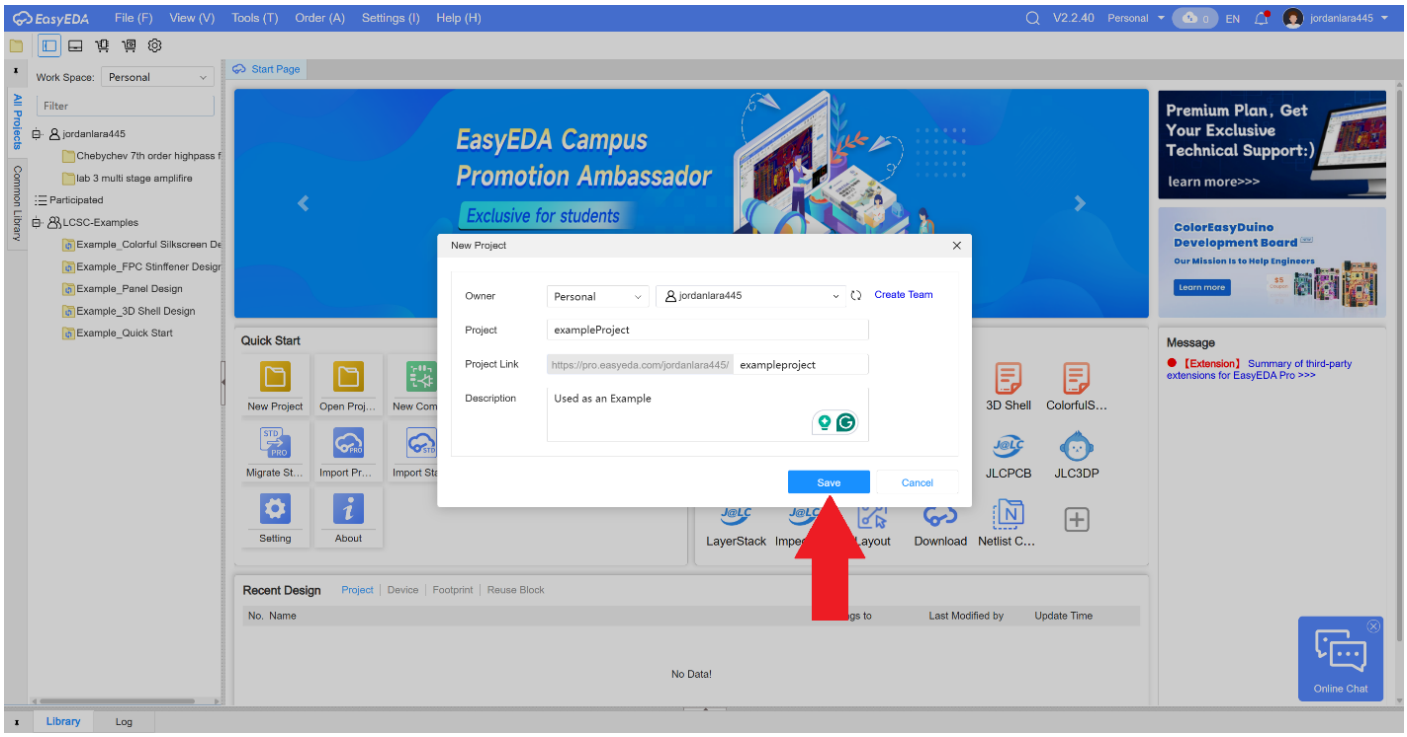
(3) It Is Very Important that you first Register a free account before creating any files, once you successfully register your account, you will be able to save your work to you account in the cloud and be able to work on and access your design on any device.



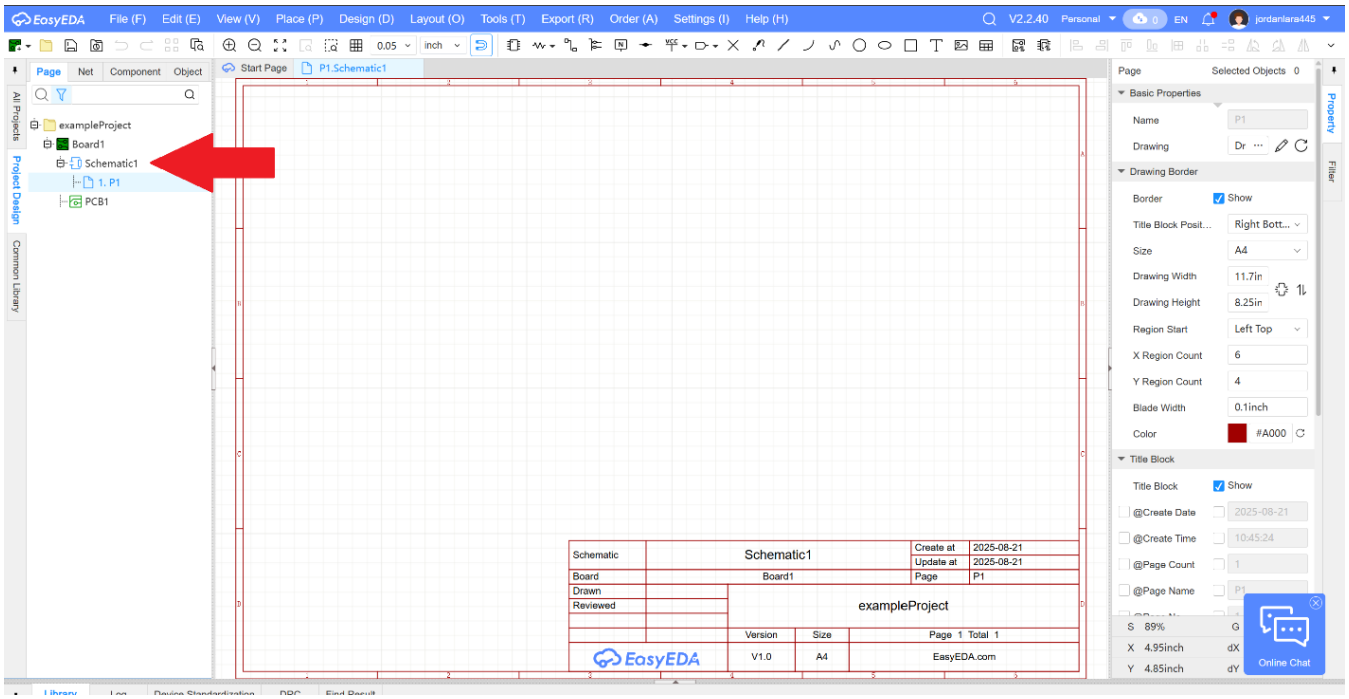
(4) Once you are signed in, go to File/New/Project and create a new Project



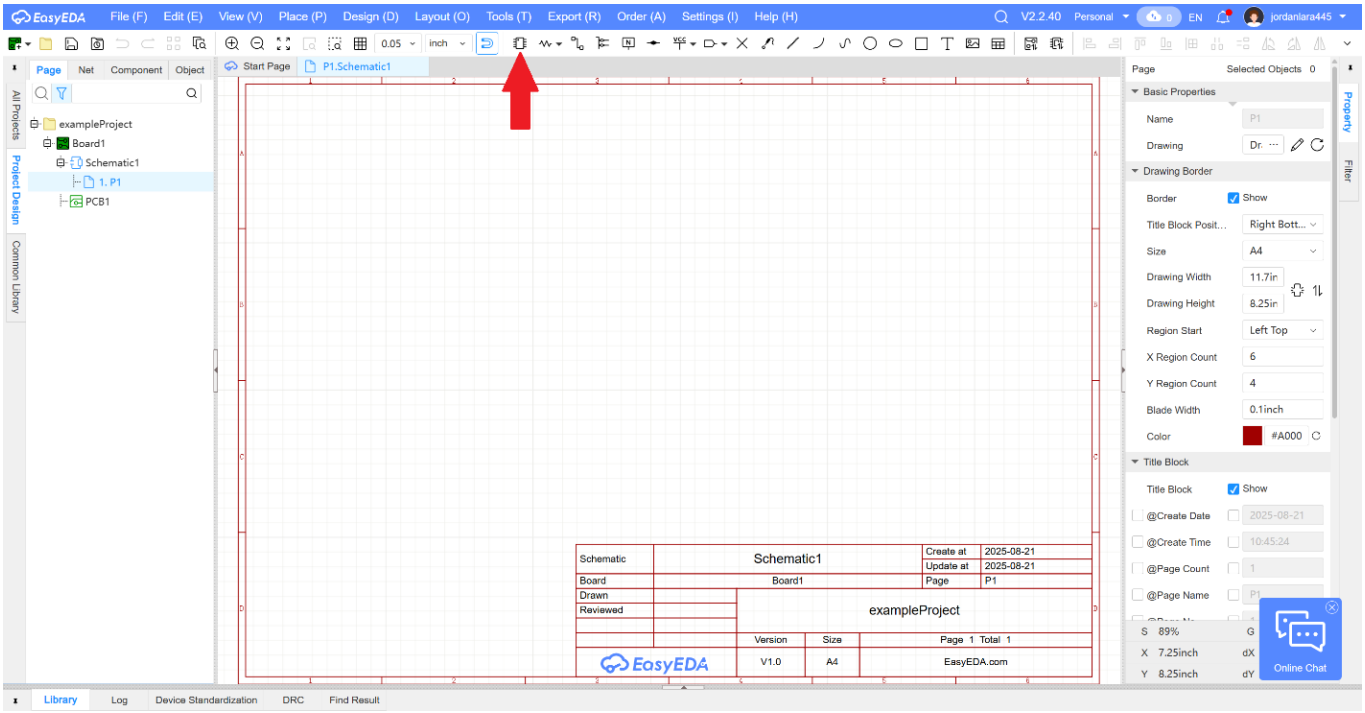
(5) Name your project and click save



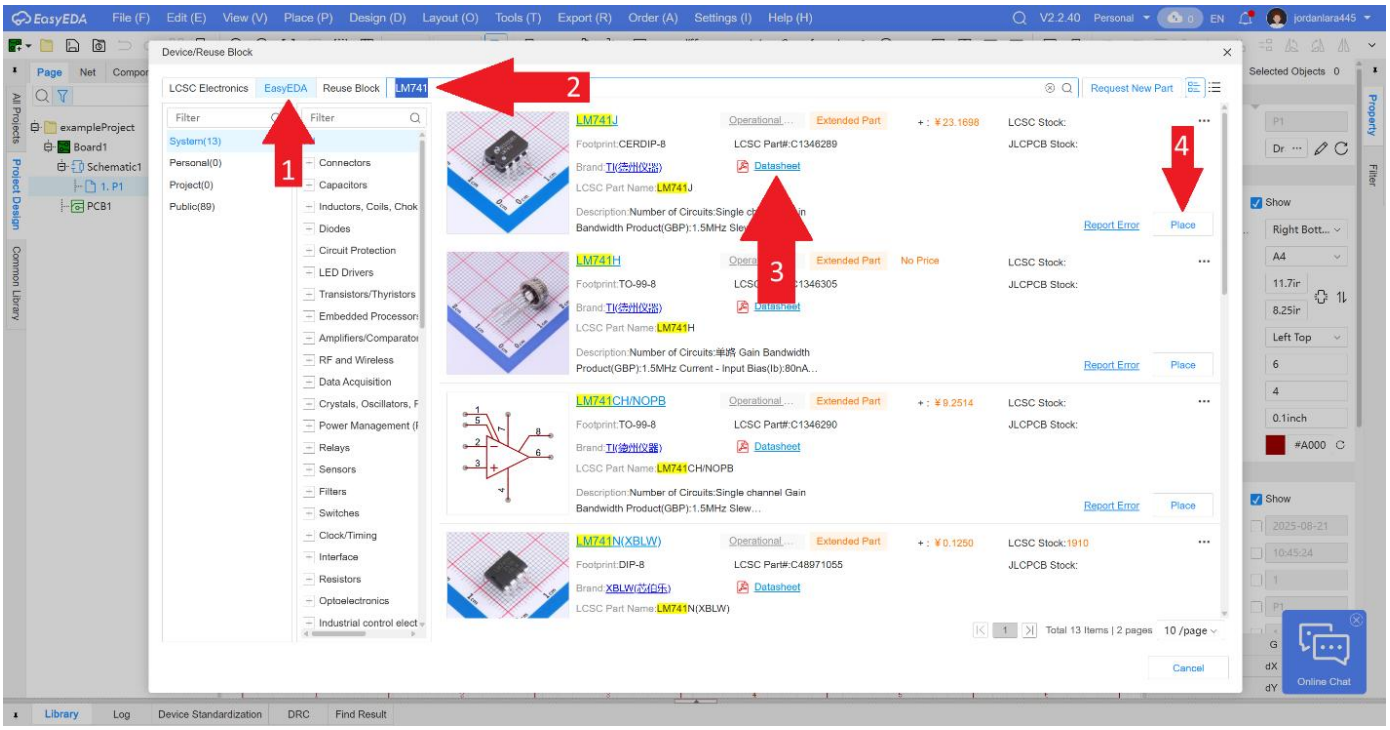
(6) You should now have your project open with default files already set up for you, double click the “Schematic1” file in the file tree to open the schematic editor. The schematic editor is where you will rebuild your schematic in EasyEDA using EasyEDA’s library of components.



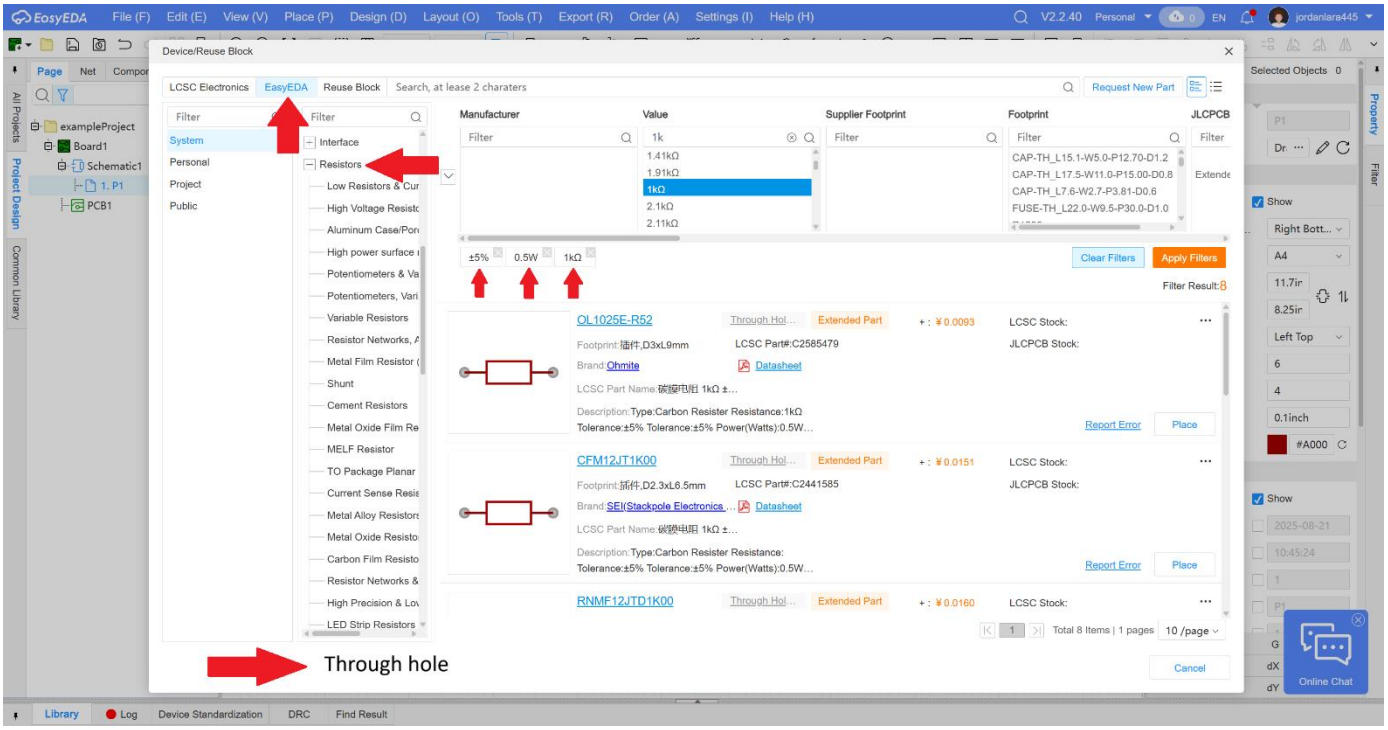
(7) For the purposes of this tutorial, I will walk through building a variable gain amplifier. Now Select the components icon.



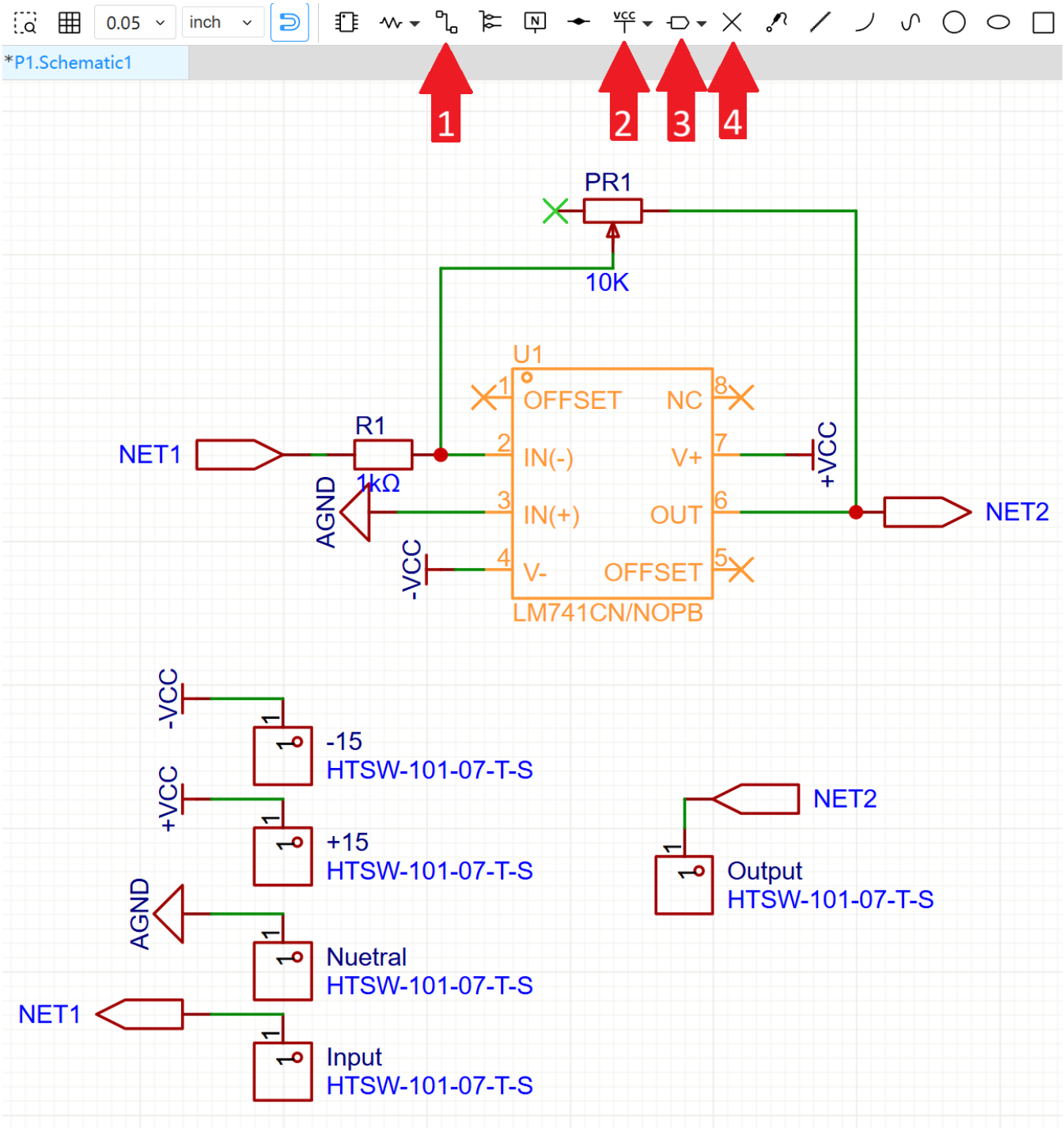
(8) First select EasyEDA in the top left for components only from the EasyEDA library. Second, if your component has a specific part name for example “LM741” or “TL081” you can use the search function to quickly narrow down the component library results. When looking at the many available components from the library, you may notice even after filtering there are many different versions of components that may be similar to what you need, this may include having a different footprint, different pin out, packaging, or other variations. For this reason, it is crucial to check the datasheet of each component you want to select to ensure it meets the specifications you need. Once you’ve ensured your part has the correct pin out and footprint, you can click place to add it to your schematic.



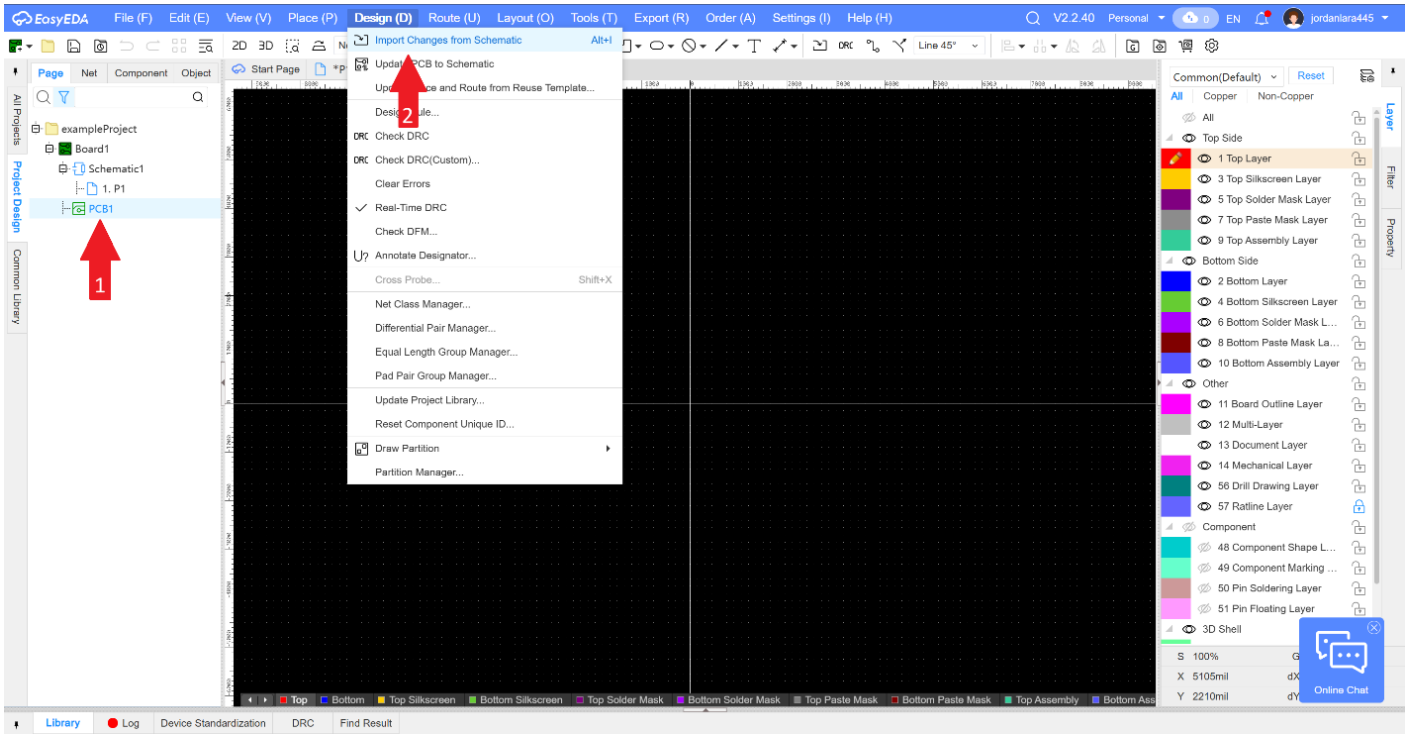
(9) To add passive components such as resistors and capacitors, it is a similar process but it is not as straight forward as using the search bar. Because there are so many variations of these passive components you must use EasyEDA's library filters to select the correct component.



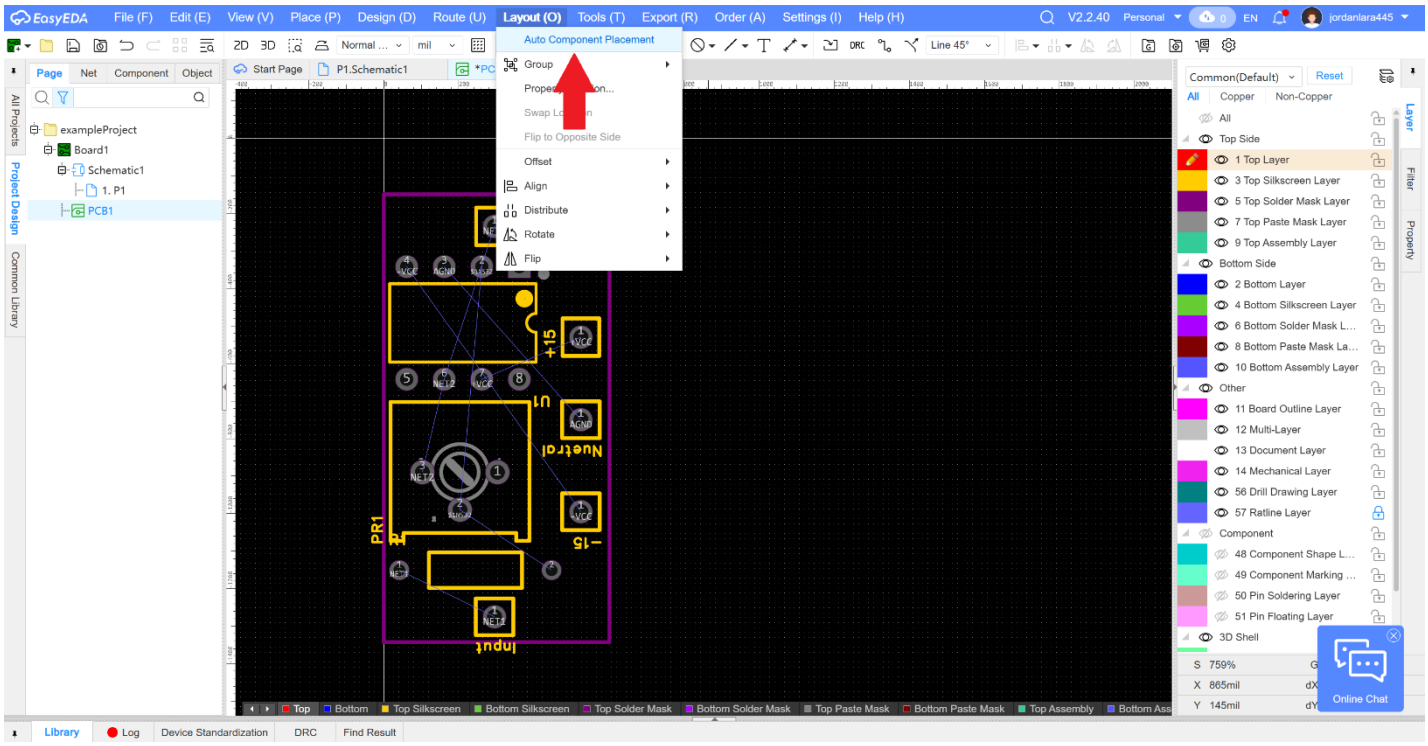
(10) Use different schematic tools to sketch the schematic. 1. Is the wire tool 2. Are the VCC and ground flags 3. Are in and out flags 4. Are no-connect flags. Each of the flags can be used to create NETs which allow you to connect components together to the same node without using a wire. If flags share a name, they are now combined in a NET or Node. No-connect flags must be used at every pin that is not used by the circuit.



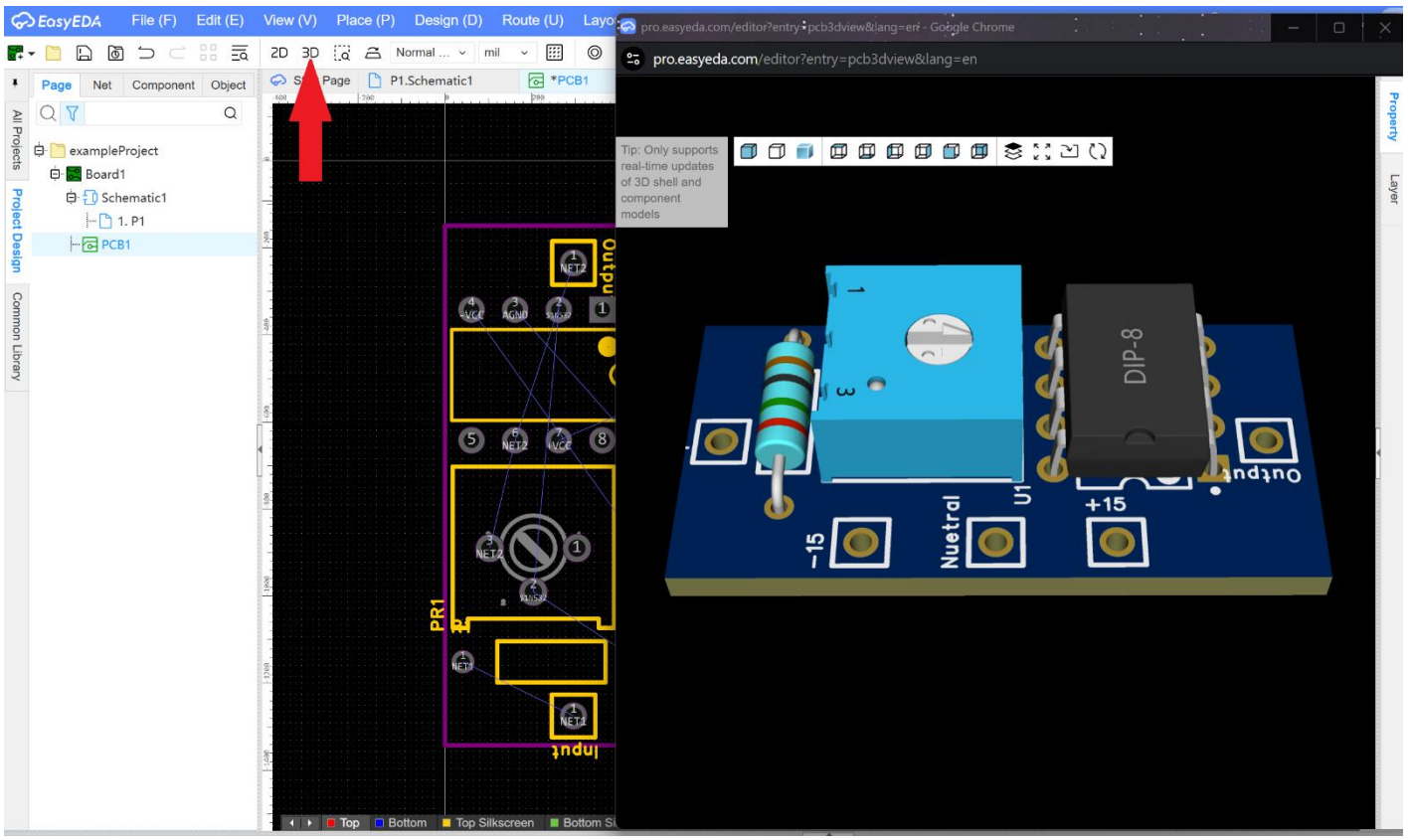
(11)Once you complete your schematic we can go to the PCB layout page and start designing the layout. 1. Double click the PCB file “PCB1” in the file tree to open the layout page 2. Go to Design then select import changes from schematic. You will be prompted to view the changes, just select “Apply Changes” in the bottom right.



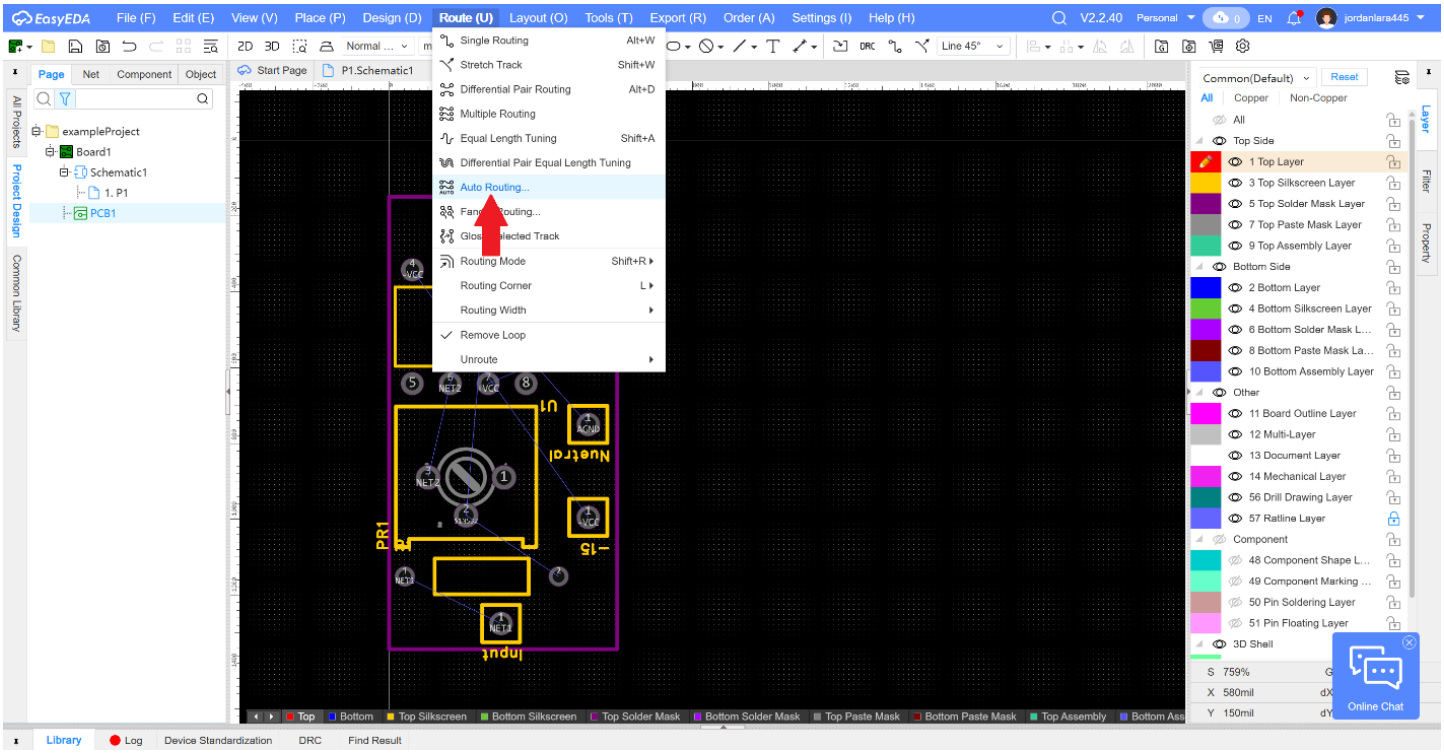
(12)Your components should now all be randomly scattered in your PCB layout file. We will now use EasyEDA’s Auto component placement tool to get a starting point layout which can be edited further based on your requirements.



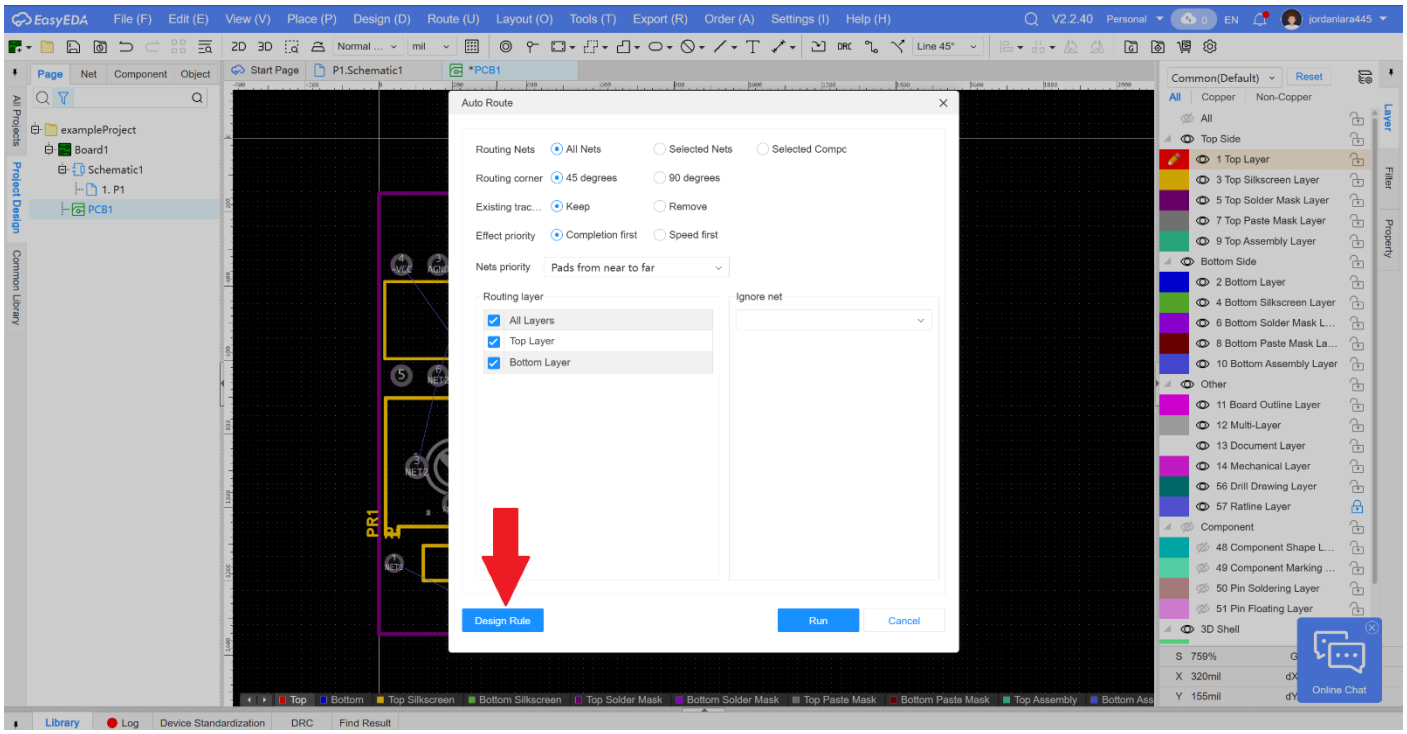
(13)You can view your PCB in 3D to make sure your components are where you need them and the component name text is clearly visible.

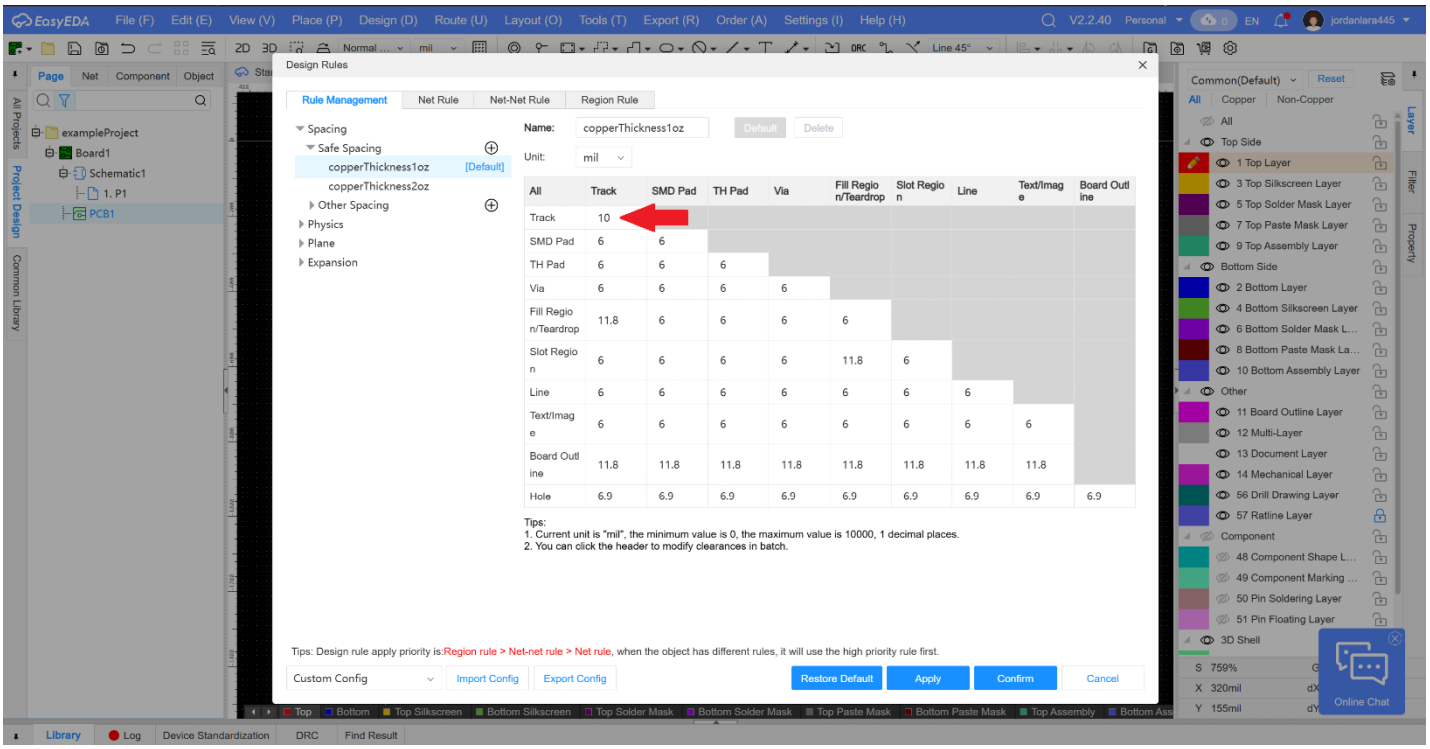


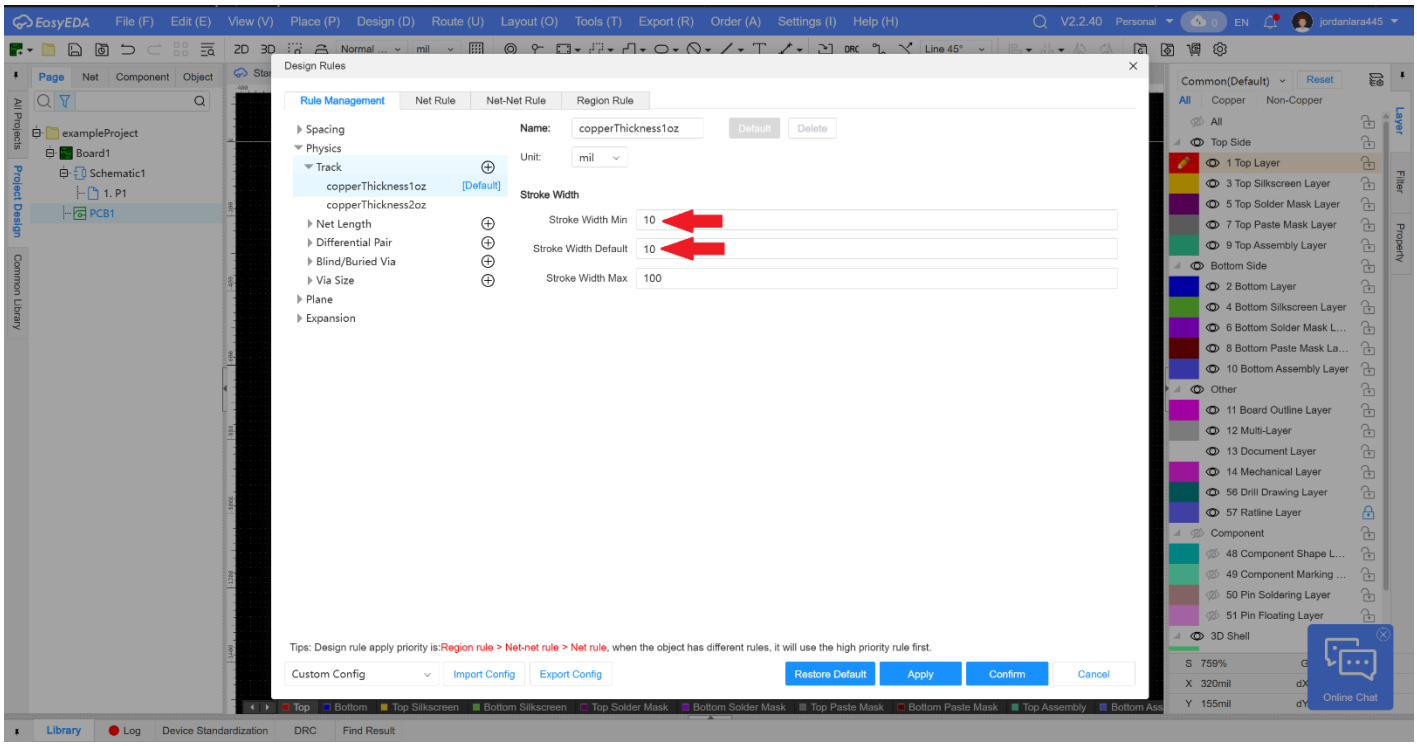
(14)Now we will be using EasyEDA's auto routing tool to rout our connections to each component. However, we have a requirement of 10 mil for transmission line thickness and spacing so we must make sure this is set in the auto routing rules. EasyEDA sets the thickness to 10 mil by default, but we will check it as well.



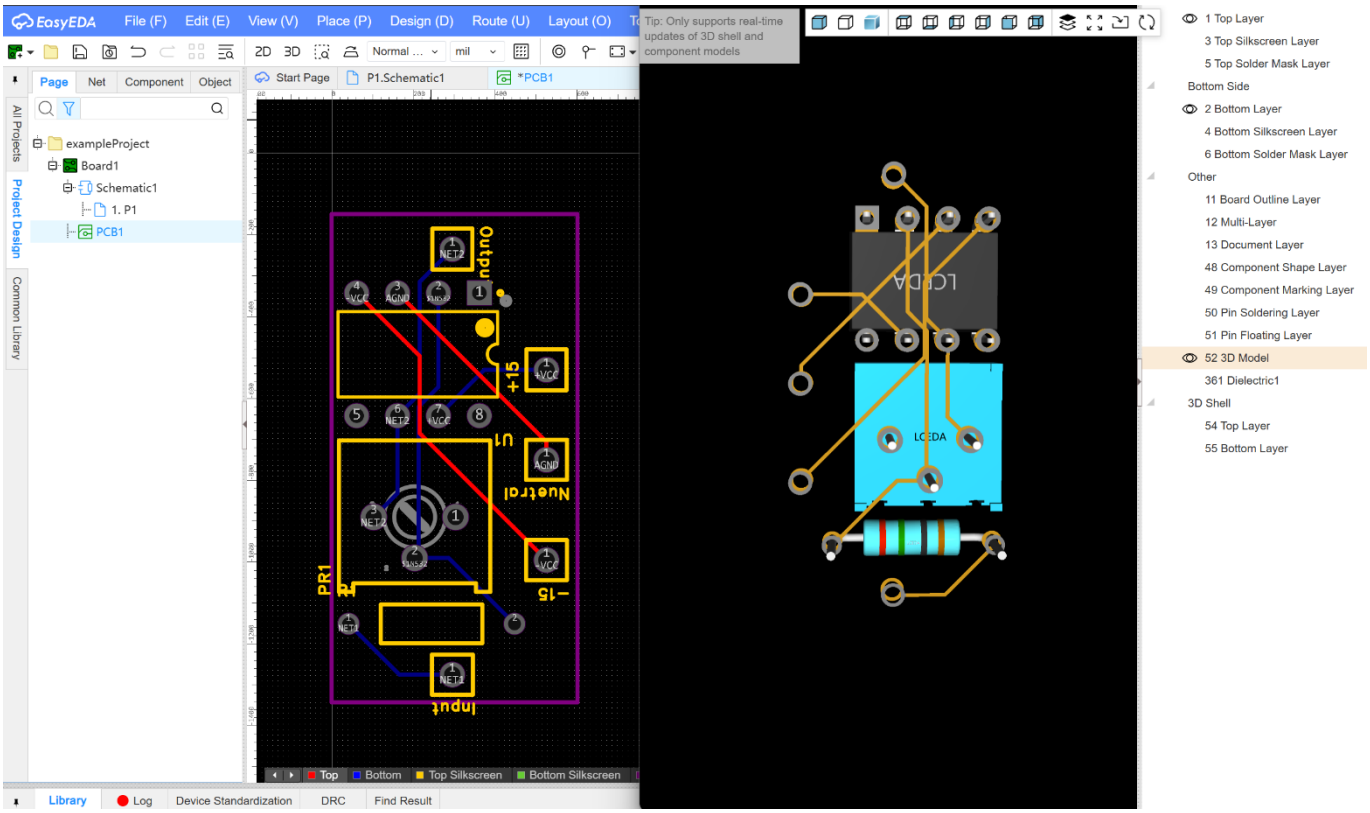
(15)Make sure the Track to Track Safe spacing is set to 10mil and the Track stroke width Min and default are 10 mil if they are not already then hit apply and confirm.


Auto Route
Routing Nets
All Nets
Selected Nets
Selected Compc
Routing corner
45 degrees
90 degrees
Existing trac...
Keep
Remove
Effect priority
Completion first
Speed first
Nets priority
Pads from near to far
Routing layer
All Layers
Top Layer
Bottom Layer
Ignore net
Design Rule
Run
Cancel

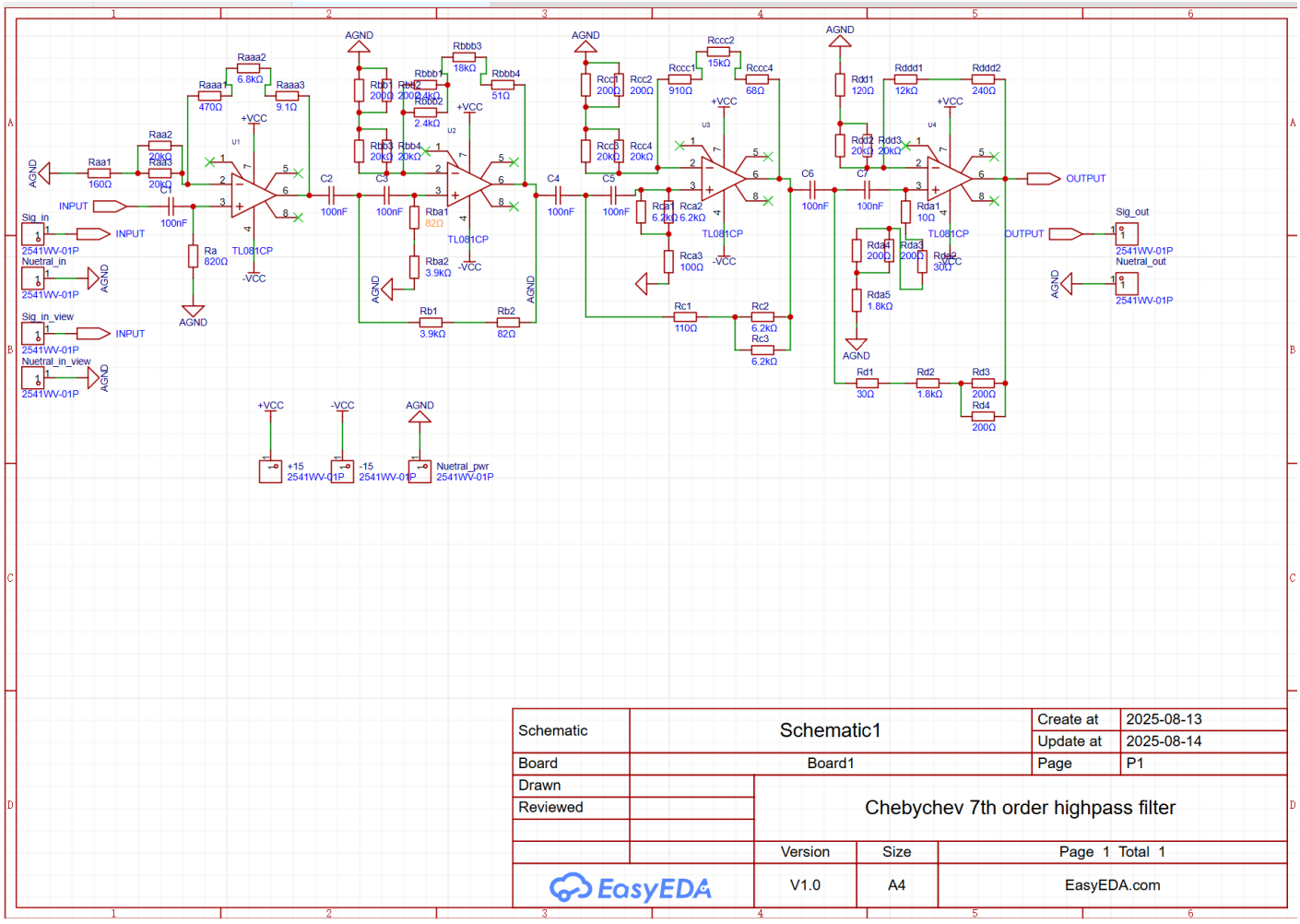

Design Rules
Rule Management
Net Rule
Net-Net Rule
Region Rule
Spacing
Safe Spacing
copperThickness1oz
[Default]
copperThickness2oz
Other Spacing
Physics
Plane
Expansion
Name:
copperThickness1oz
Default
Delete
Unit:
mil
All
Track
SMD Pad
TH Pad
Via
Fill Region/Teardrop
Slot Region
Line
Text/Image
Board Outline
Hole
Tips:
1. Current unit is "mil", the minimum value is 0, the maximum value is 10000, 1 decimal places.
2. You can click the header to modify clearances in batch.
Tips: Design rule apply priority is:Region rule > Net-net rule > Net rule, when the object has different rules, it will use the high priority rule first.
Custom Config
Import Config
Export Config
Restore Default
Apply
Confirm
Cancel



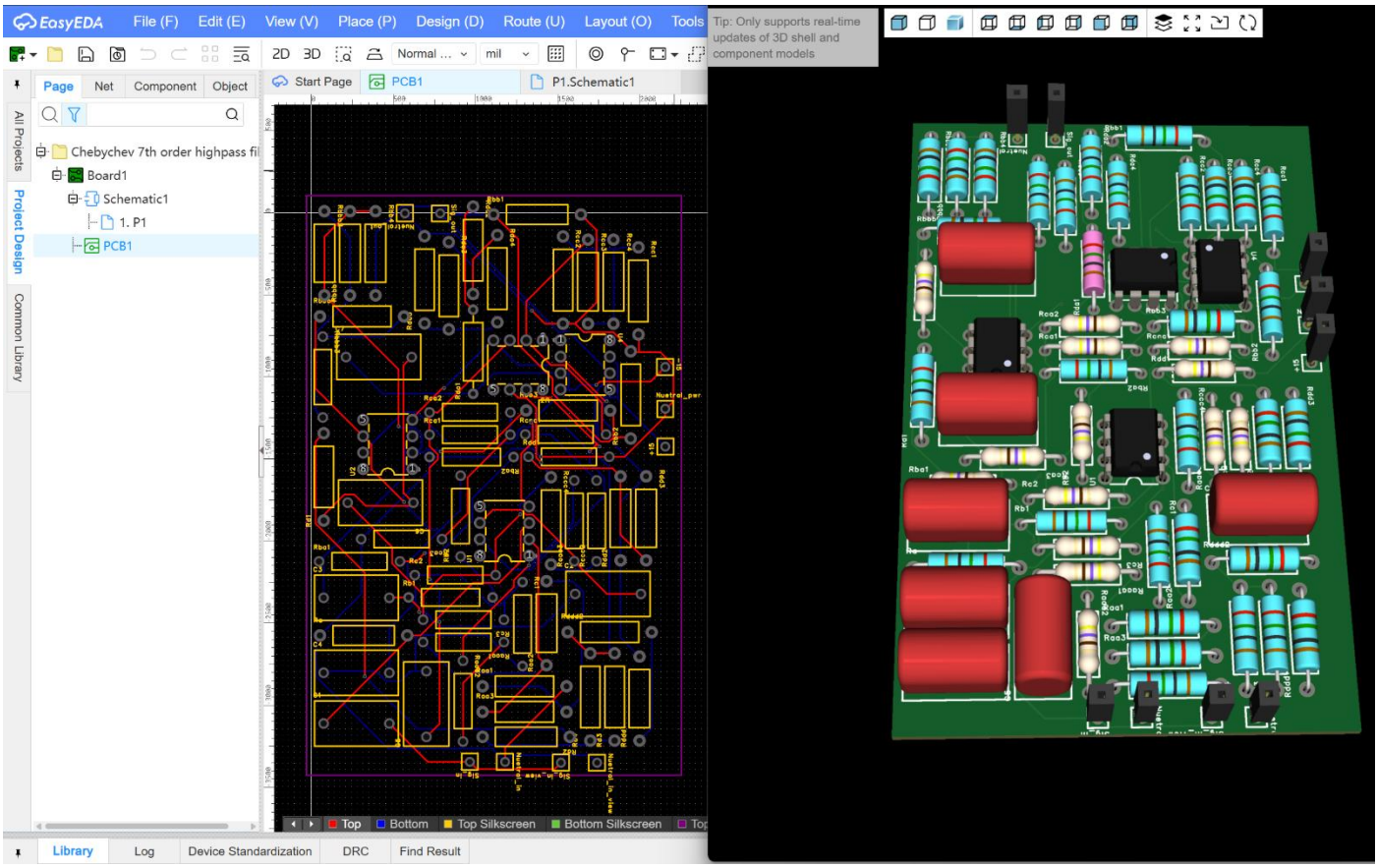
(16) We can now Click run on the Auto route. Depending on the complexity of your design this may take several minutes. You should now have all components on your PCB routed and connected.



(17)In The Beginning I used a Chebychev 7th order as an example for the proposal. I want to show the schematic for it in EasyEDA to show how a larger circuit may look:

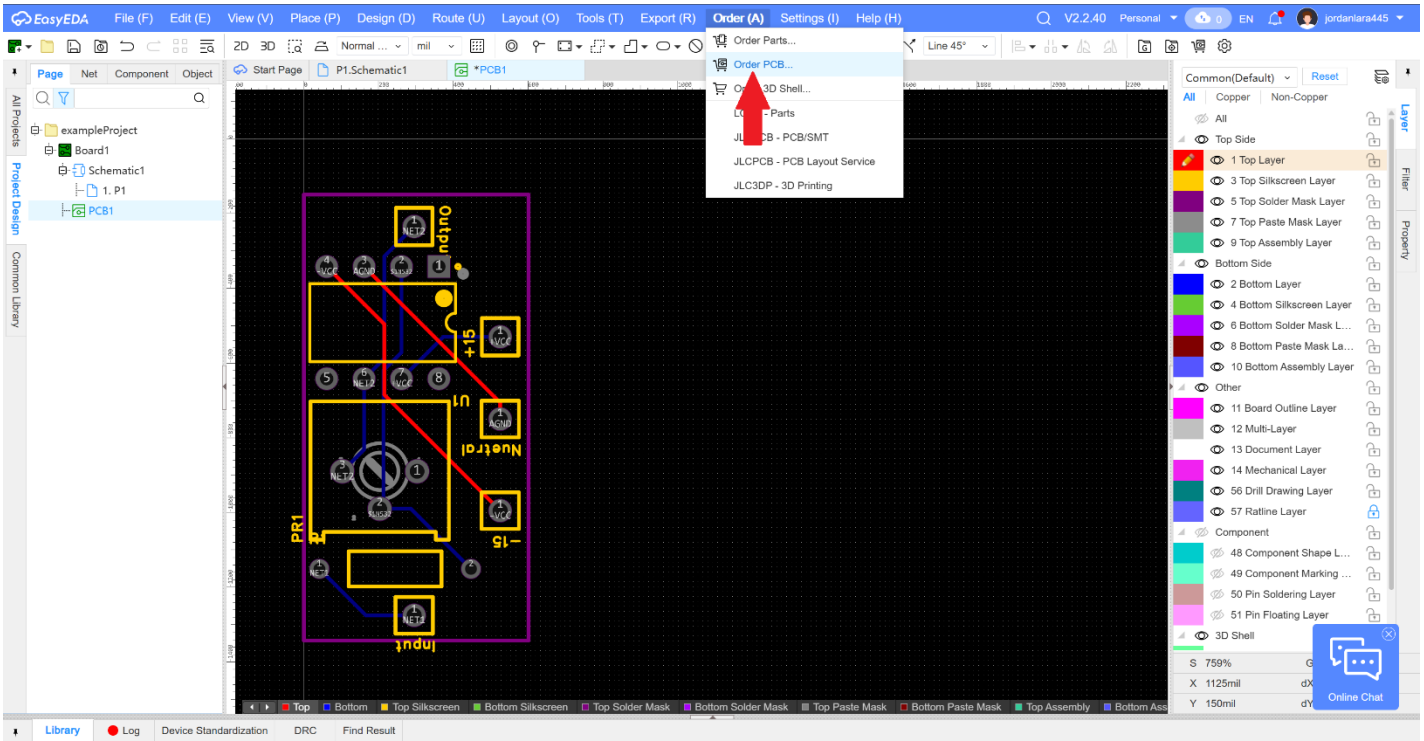


(18) I'll also show the accompanying PCB layout which used methods described previously to develop:

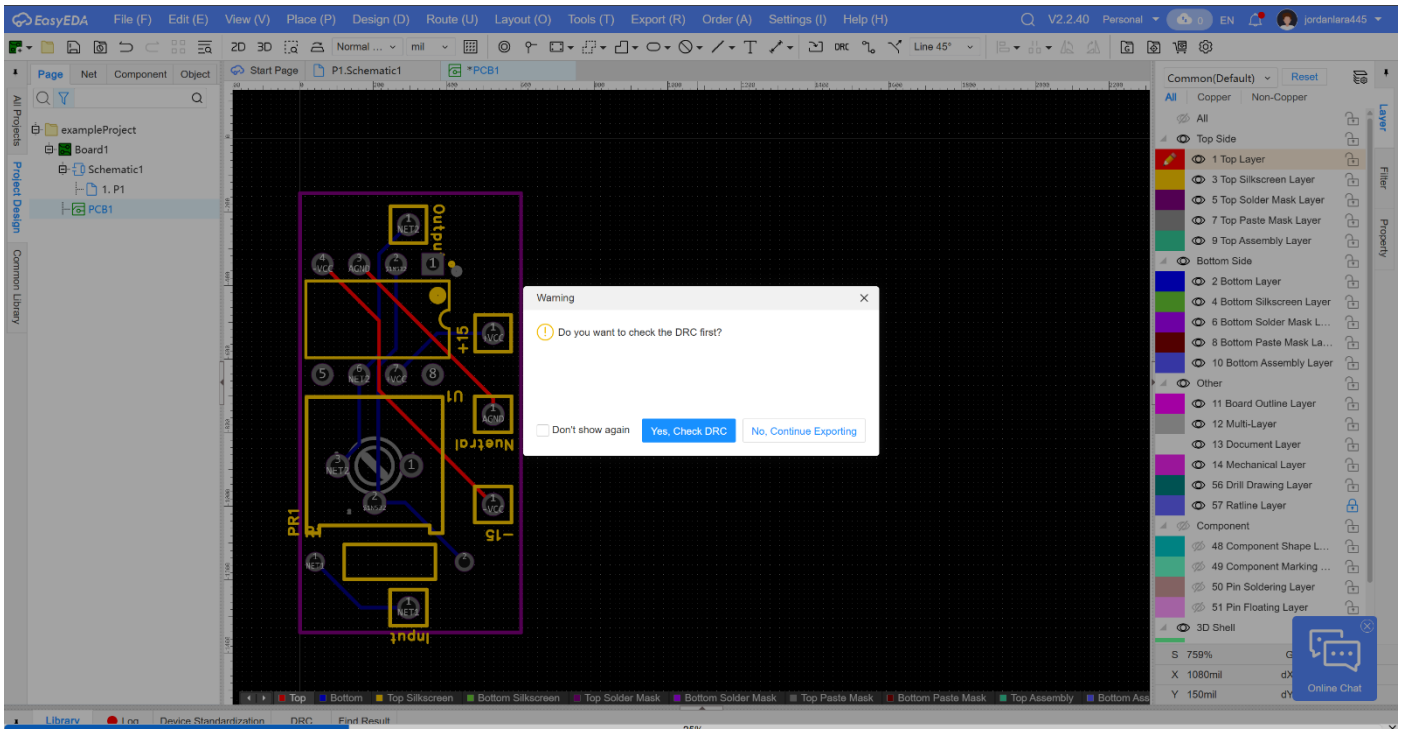


Outsourced PCB Manufacturing With JLCPCB Through EasyEDA

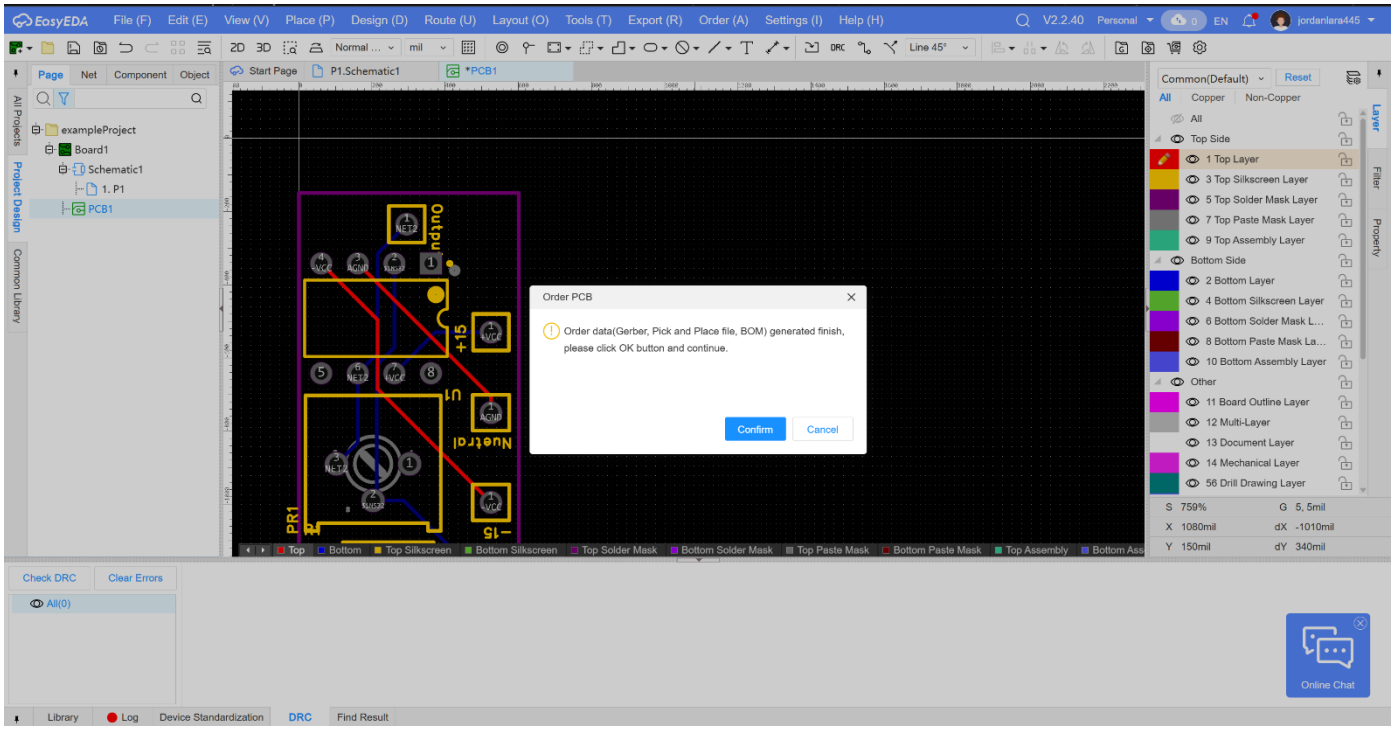
(1) Now that we have our PCB we can either download the Gerber files or Order it directly through EasyEDA From JLCPCB. Select Order/Order PCB.



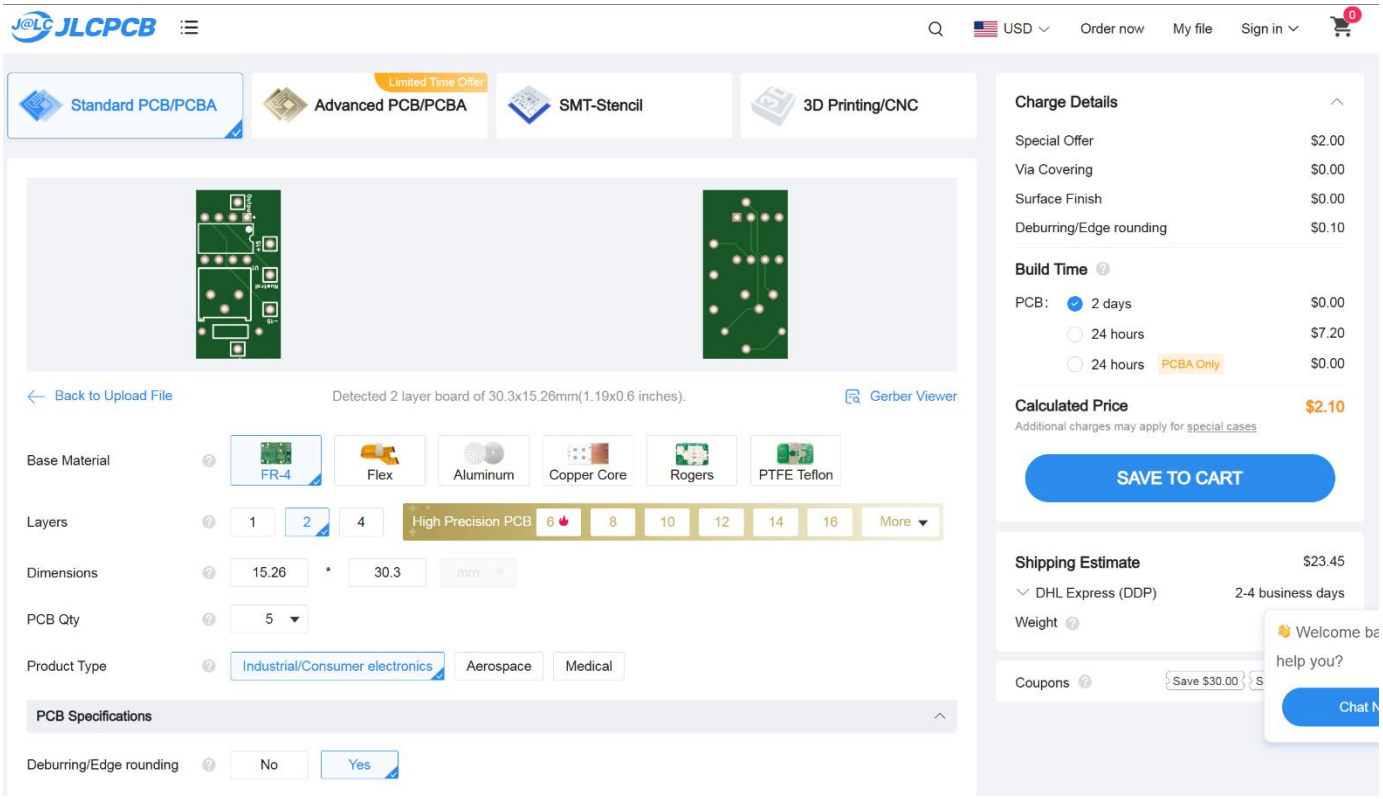
(2) Click “Yes, Check DRC”



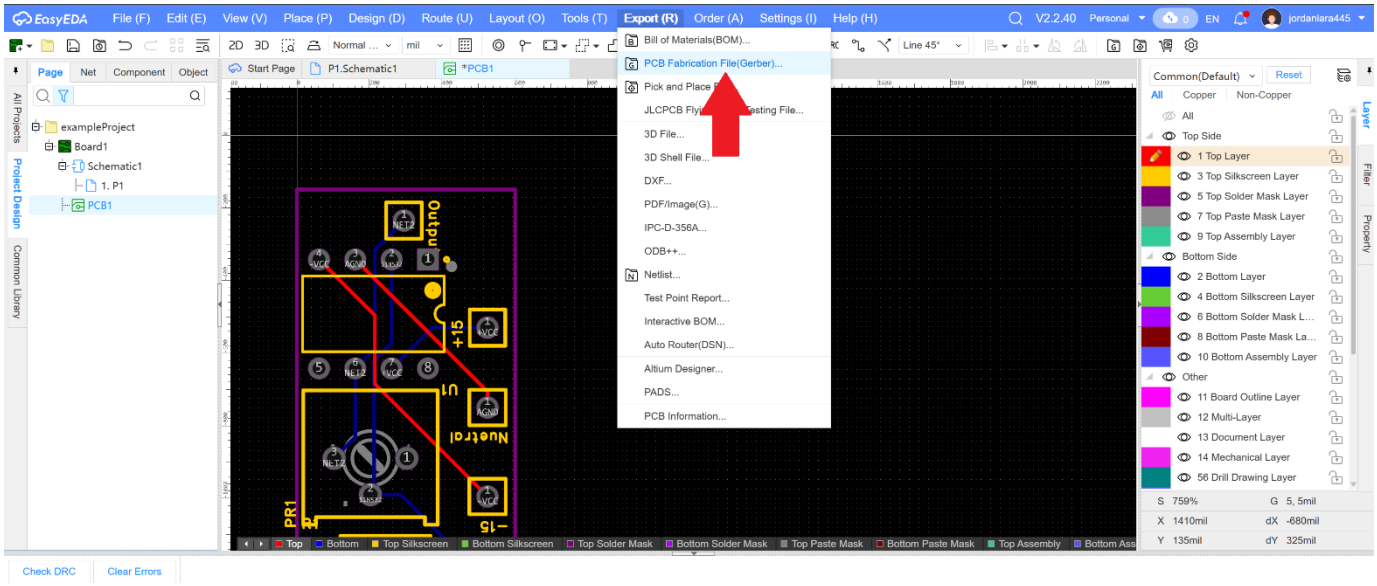
(3) Click “Confirm”



(4) You will be taken to the check out page to finalize your order



(5) To download the Gerber files, Select Export/Gerber.



Soldering & Assembly

(1) Once you've received your PCB from Manufacturing, you will need to go through a process of soldering your components to the board. It is crucial that you understand the

safe operation of soldering iron. A soldering iron can cause painful burns and should always be used with caution. Ensure you have access to a proper work area that is well ventilated. You may want to invest in “Helping Hands” soldering equipment to make the component placement and soldering process much smoother.

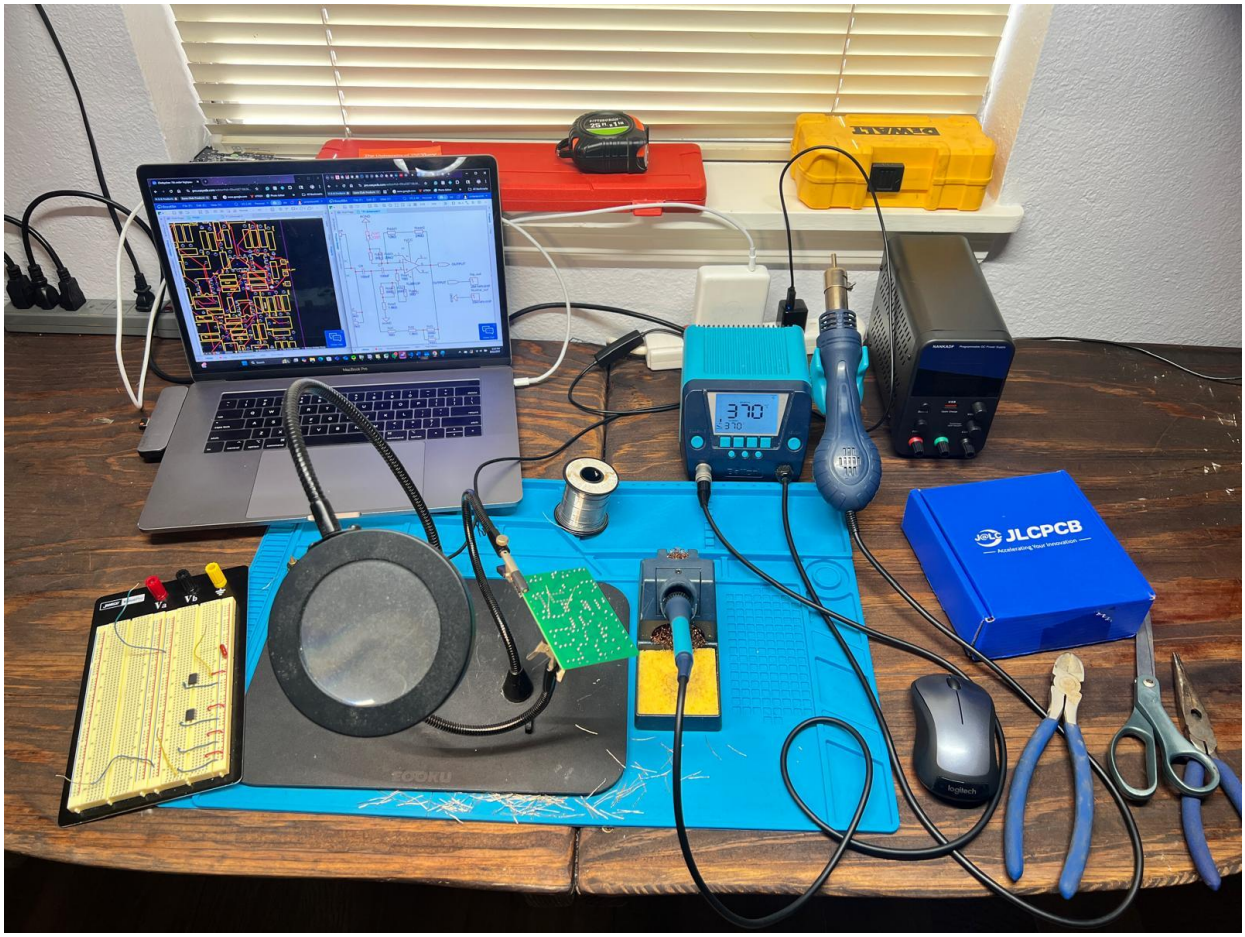
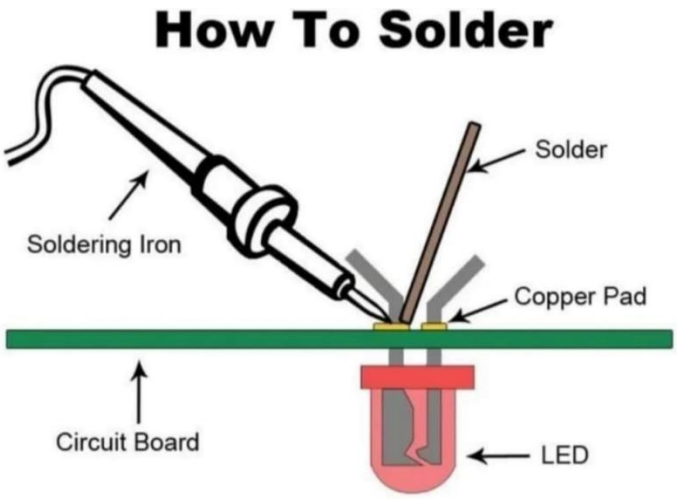
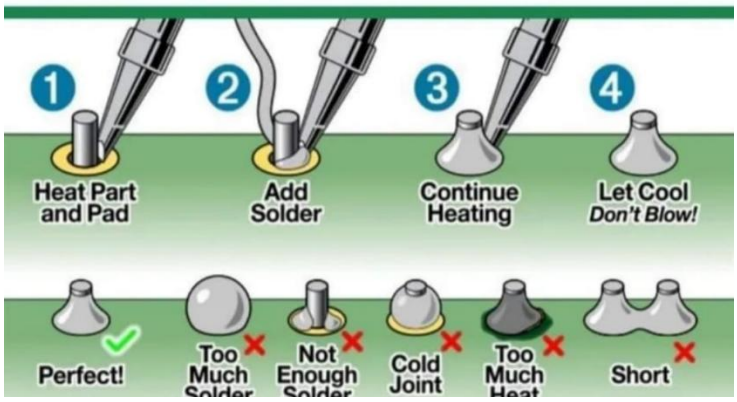


Figure 1: Soldering Area

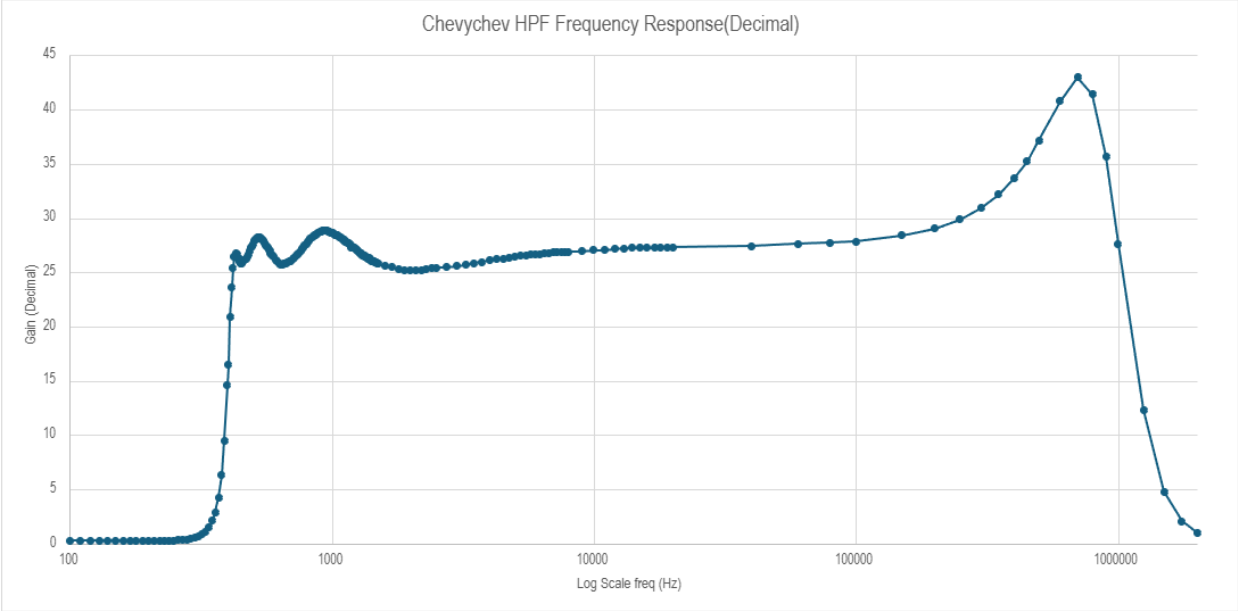
(2) The process of soldering is straight forward, you place your components in accordance to your designed PCB from the earlier steps, however it is a skill that is developed. It is very easy to ruin components or even the entire board if you do not set up your iron temperature correctly and do not wield it correctly. It is Suggested to Research your solder manufactures recommend temperature settings for your specific solder being used. Below are common rules of thumb for soldering.



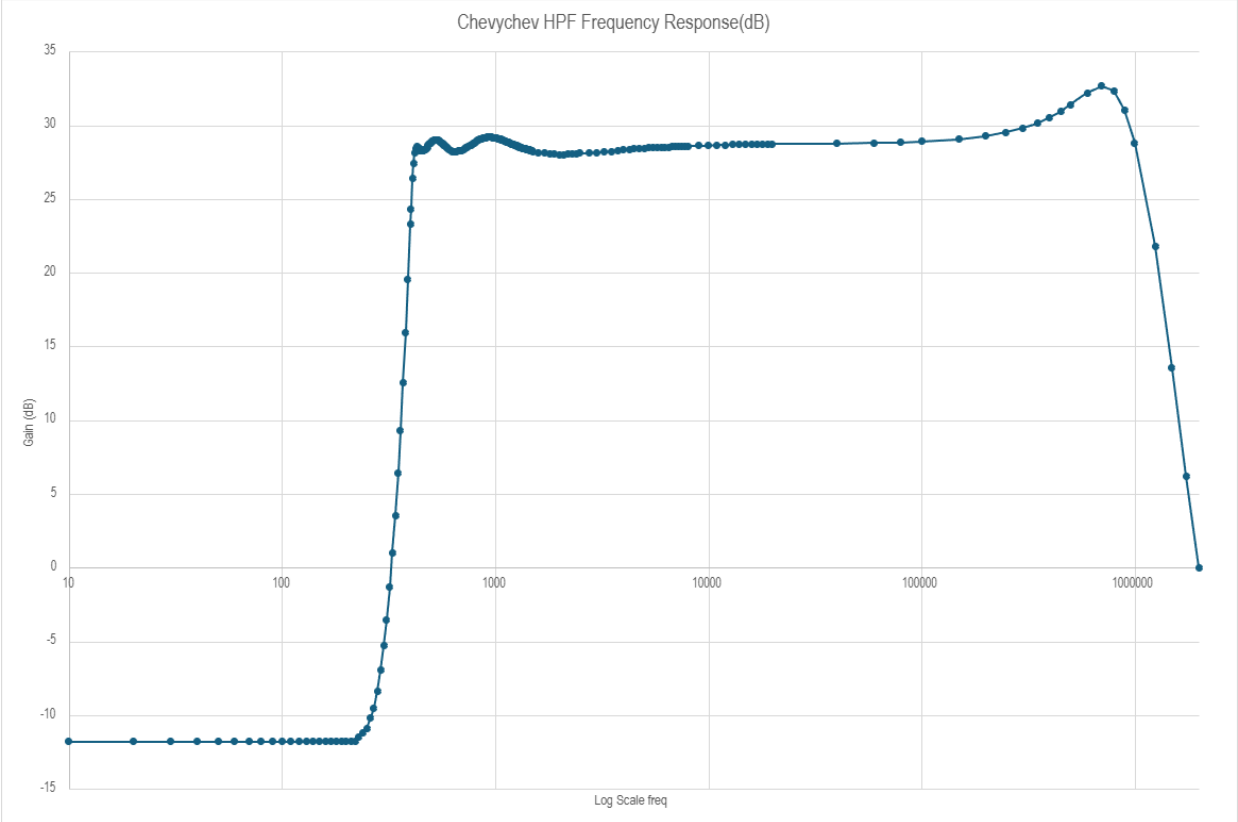


Project Testing

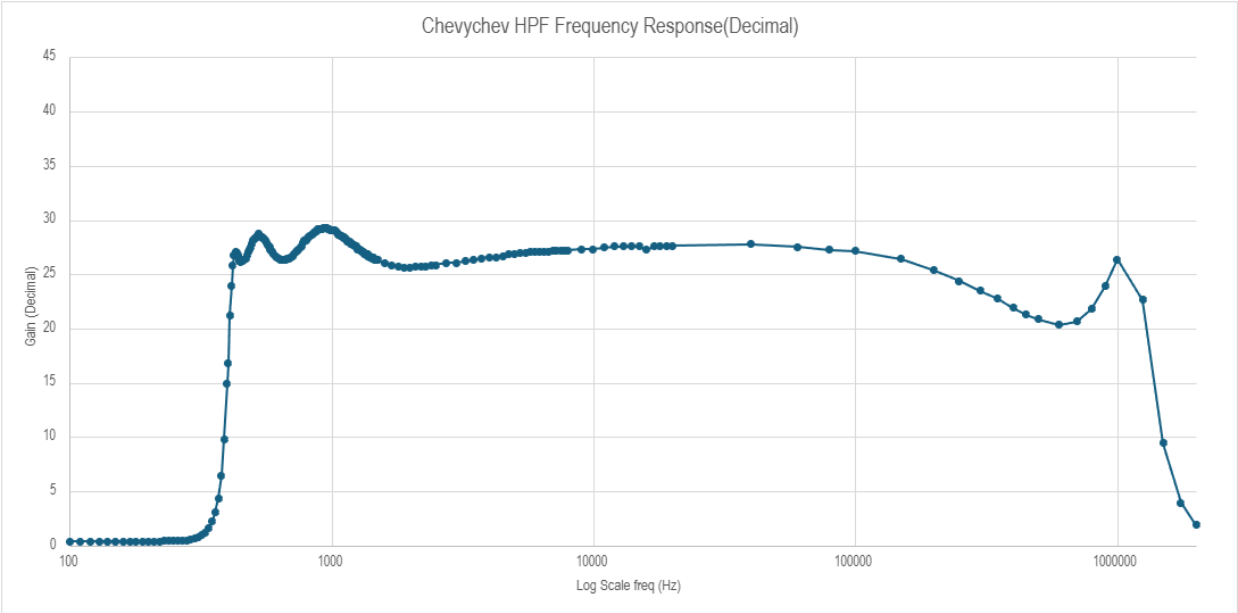
(1) As stated in the "PRIOR TO PROJECT" section, You will develop your own test plan for this circuit. Your plan should describe (in sufficient detail to be understood by a test technician with your own level of lab experience) the tests which will be done to check the circuits performance against all the specifications. Your test plan should explicitly address the function of your chosen design. **For example:** Using the same example of the 7th order Chebyshev HPF we've used throughout this tutorial, we will run a Frequency Sweep test on the Filter assembled as a PCB circuit to observe the difference in Functional operation between circuits designed and tested on a bread board and circuits designed and test with a PCB.



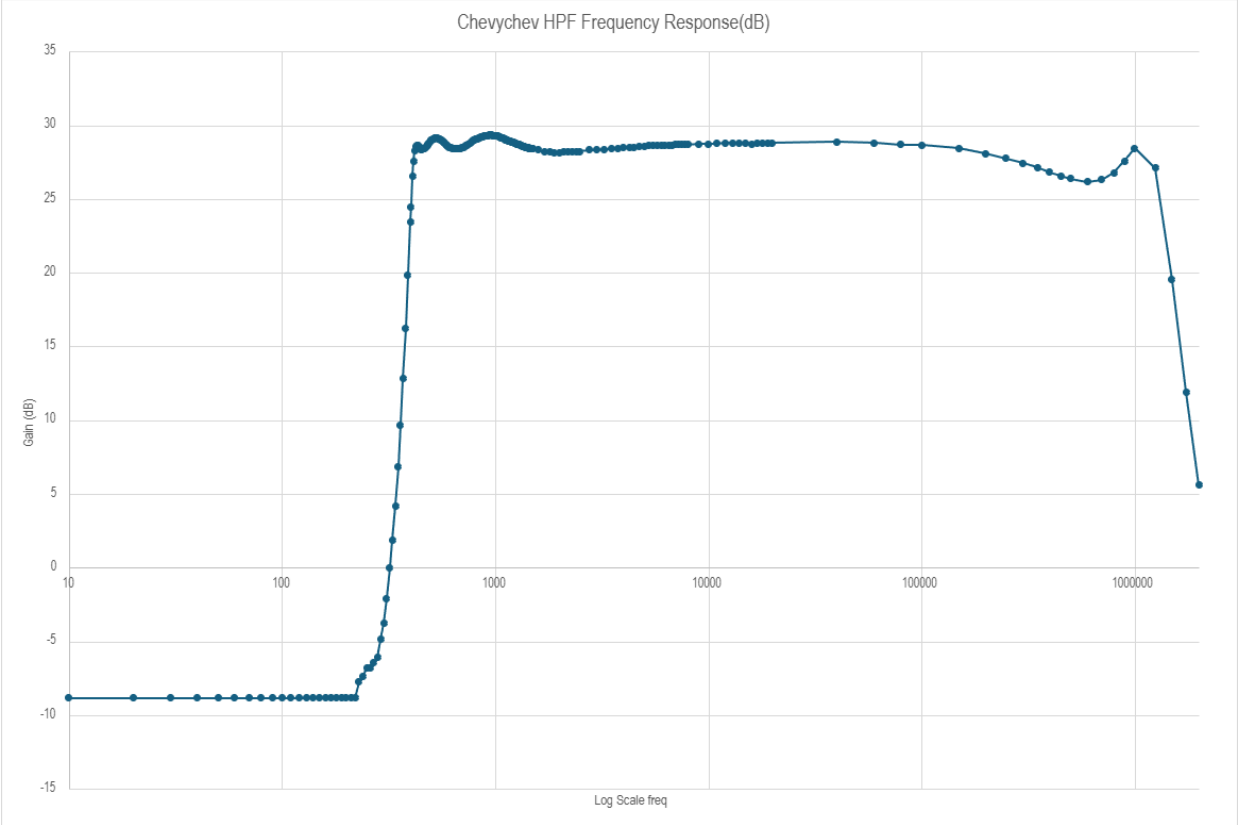
Graph 3: Chebychev 7th order highpass Frequency Response Breadboard Decimal



Graph 4: Chebychev 7th order highpass Frequency Response Breadboard Decibel



Graph 5: Chebychev 7th order highpass Frequency Response PCB Decibel



Graph 6: Chebychev 7th order highpass Frequency Response PCB Decibel

(2) We can observe that there are only minor changes in the performance of the filter at

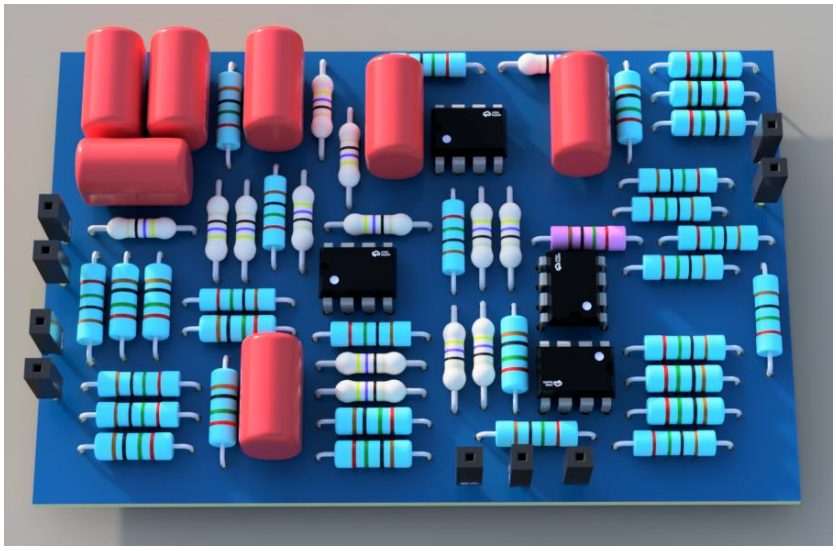
lower frequency ranges when comparing the Breadboard circuit and the PCB circuit. However in the higher frequency ranges approaching the Megahertz range, There is a very clear difference in performance between the Breadboard circuit and the PCB circuit. The PCB circuit has a much cleaner roll off. If it was not made clear previously, both of these circuits were tested using the exact same components ensuring the only change between them being the format of connection.

DEMONSTRATION: After you have completed testing, demonstrate to an instructor the functional operation of your PCB.

F. For Fun

Product design Task

(1) This Project is meant to serve as a introductory experience to Senior design. Senior design shares many similarities with product design. As an EE you may never be asked to use CAD software to make any fixtures. However, in a small team or company you may find it extremely beneficial to have multiple skills to be self sufficient and prevent delayed progress in your work. For this reason, it is important to understand how to make basic prototype level fixtures to go along with the development of your product. Your Task is to create a Protective case for your PCB that is user friendly and protects your circuit from damage.




ChebyChev 7th Order
HPF 403hz cut-off
30 V/V gain
VIN
AGND
VIN
AGND
AGND
VOUT
AGND
+15
-15