EasyEDA Handy Tips

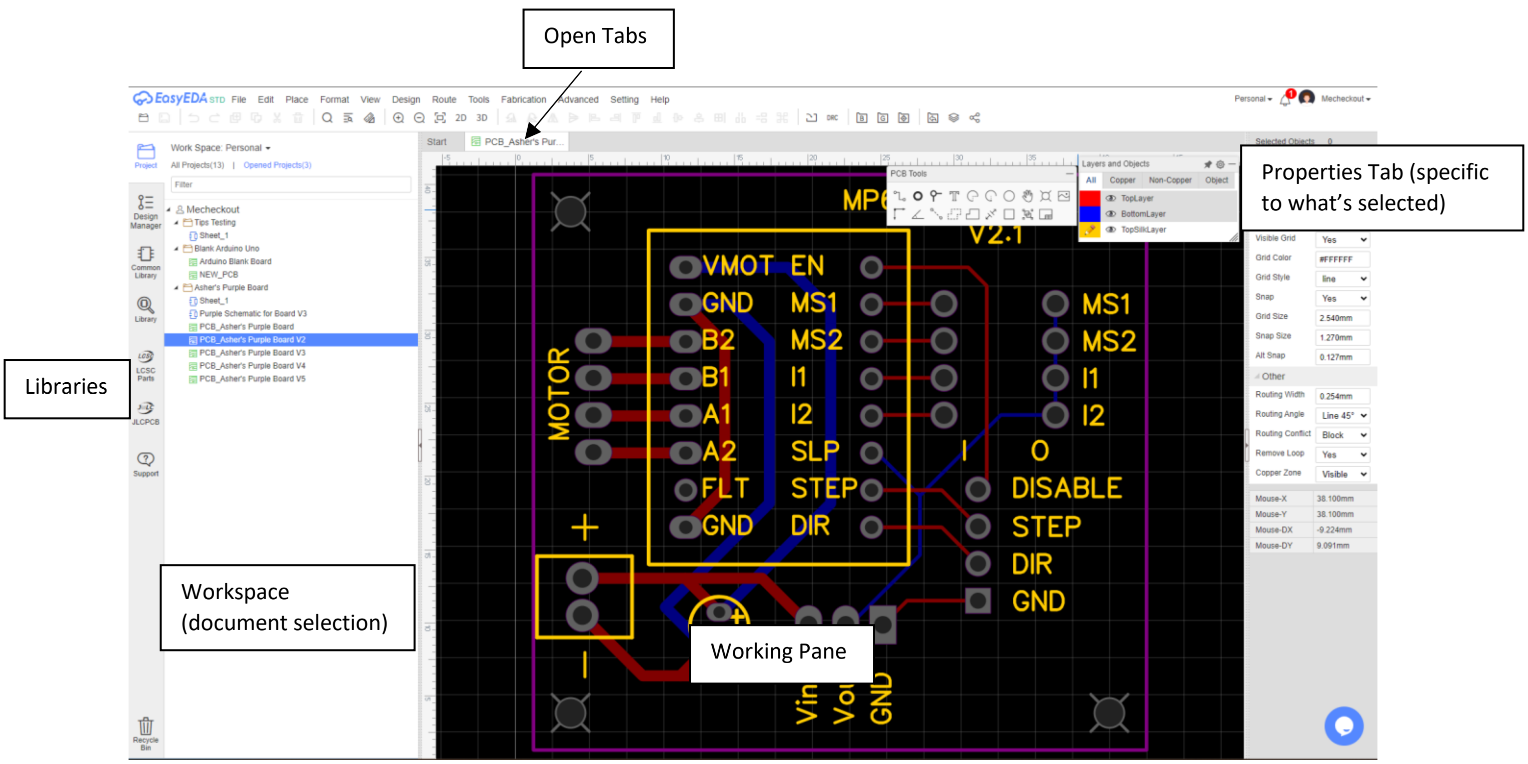
READ THIS BEFORE ANYTHING ELSE:

- You do not need to download ANYTHING to use EasyEDA. There is an online editor that works great!
- This guide is not exhaustive. There are many great videos and pages about how to do different things with EasyEDA, and you'll have the most success with this software learning by doing.

Getting Started

- When you first navigate to EasyEDA.com, you'll want to select "Design Online"
- You'll want to select Std Edition (Student Edition). An editor will open. At this point, you'll want to Log In (top right corner). If you're not logged in, you won't be able to save any projects.
- To open a new project, select "File" (top left corner) >> "New" >> "Project". A "Project" is just a folder that organizes/groups schematics with PCB Designs. When you open a new project, it will automatically create a new schematic. You're now ready to start designing!
- Before getting too far, read through the rest of this guide. It's full of tips and ideas that will hopefully save you lots of frustration and "Oh duh" moments:)

EasyEDA PCB Designer Layout



- This program doesn't autosave. It's a good idea to save your project frequently; I've had problems with it not letting me save sometimes.
- The software does have a good amount of training material and forum discussion, so if you have a question about what you're doing, Google is a good place to start.

How PCB Design Works

- PCBs are composed of different layers of different materials. This editor (and all PCB design tools) allows you to edit each layer individually.
- The most important layers will be the copper layers. Those are what connect the actual circuits and make the board useful.
- You'll also be able to edit the silkscreen, which is the print on the board, and some of the properties of the insulator, including the outline of the board.
- PCB Design tools allow you to edit the layout and properties of all of the components that you place on the board.
- Some PCBs, especially those that have lots of components, are easier to design with a circuit schematic before designing the PCB. These editors allow you to design a schematic, then load it into a PCB to edit the layout and connections.
- Pins that connect to each other are grouped into what are called "nets". Nets define the circuit relationships and help you keep your circuit in order. The software will automatically assign nets to your connections.

Schematic Design

- You don't necessarily need to place parts or even make a schematic to design a PCB, but it may be a helpful tool, especially for more complex designs
- When placing parts from the library, it's easiest to have a different browser tab open (to a site like LCSC.com) to find the part you want, then enter its product number into the EasyEDA's library search. The library search alone is pretty crummy.
- EasyEDA isn't hooked up to DigiKey's inventory, but it is connected to LCSC's library, which has most of the same parts. If you find the part on the LCSC website, you can search for it in the EasyEDA library using the LCSC Product number (C#####)
- When connecting wires, crossing over a wire won't create a node, but will make your schematic confusing. Currently, EasyEDA doesn't have any special symbol for wires crossing over each other, but nodes are labeled with a red dot.
- When putting together your schematic, the system will automatically assign nets (groups of connections). You can edit these nets later when you make the PCB layout; unfortunately, you cannot edit them in the schematic (as far as I know).
- The schematic editor has some nice tools for checking voltage, nets, etc.

PCB Design

- 1mil = 1/1000 inch = .0254mm
- When you start each new PCB Design, it will ask you to set the dimensions of your PCB and give you options for a rectangle or circle. You can edit the size later, but you won't be able to change the origin you set.

- If you don't like the rectangle it gives you or you want to do a more complex geometry, you can draw with the track or circle/arc tools, then change what you drew to the BoardOutline Layer. (You can preview your board with the "3D" button to tell if it got the shape right)
- When editing your PCB, you can change the snap size for dragging things around on the board. This is very helpful if you're looking for uniformity or you have a size convention you're trying to follow (standard pin spacing is 2.54mm or 100mil). To change this, right click the background grid with nothing selected and change the "snap size".
- For designs with a schematic, you can't connect pins with a track or trace unless they are part of the same net. You can change a pin's net in the properties tab.
- If you're dragging and dropping components into a new PCB design, you can connect pins all willy-nilly and nets will automatically be assigned.
- To export your design or generate Gerber files (used for PCB fabrication), select the "File" dropdown in the top left corner.
- EasyEDA is furnished by JLCPCB, so when you export your Gerber files, it gives you an option to order from them. You don't have to do this; you can just export the Gerber files.

Design Principles

- Higher current needs more copper to work safely. You can adjust the width of tracks in the properties tab while they're selected. It's also good to make the pads they connect to bigger as well. You could also choose a thicker layer height (like 2 oz.) but 1 oz. is standard and it will probably be cheaper to just make the tracks wider in the long run.
- Tracks running too close together (especially high current) can create impedance that interferes with signals. This generally isn't a huge issue, but you should check it when you're creating higher current lines.
- It's generally always a good idea to plumb ground pins together to make a common ground on the board.
- JLCPCB has specific PCB manufacturing capabilities that are listed on their website. It's a good idea to glance over these before starting your designs. Unfortunately, their system doesn't automatically check to see if your board is in compliance with these rules, even when you upload the Gerber for a quote. There are options to verify the design after the order is placed and your board will get flagged before production if one of the design rules is off, but that's a hassle. Fortunately, they have a Design for Manufacture (DFM) checking site that you can upload your Gerber files into (link below). If you're concerned about any of the rules, this is a good place to check before submitting an order.

Asher's Design Workflow

- Open a new Project, and create a new PCB Design. Don't bother with the schematic. (In the real world, you'll ALWAYS want to make a schematic, but for small projects, it will probably be more hassle than it's worth)
- When selecting the Board Dimensions, start with something large, make sure you have rounded corners (to minimize ouchies), and make sure your Start X and Y are both zero (EasyEDA won't

let you change the origin once it's place). You can redraw the boundaries with "Tools" >> "Set Board Outline…" or draw lines like traces and change their layer to "BoardOutline"

- Place mounting holes and any other large/non-circuit features.
- Place all of your parts (ICs, resistors, capacitors, diodes, etc.) to the PCB. Don't worry about placement, but DO worry about if you'll be able to solder them. (For reference, BYU has extremely limited surface mount capabilities; through holes will be your friends)
- Layout your parts with two things in mind: User Interface, and ease of circuitry.
  - You want to be able to connect all of the traces together reasonably, but you also want the placement of parts you interact with to make sense. I suggest leaning heavily to good user interface; you'll find that you can fit more traces in a small space than you think
  - Don't forget about the backside of the PCB. The cost of the PCB only depends on the size, using both layers is completely free (and a great idea)
- Assign nets by hand, starting from your power source and moving out. Make sure all of the circuits make sense.
- This is a good time to have a friend check your work, or to re-check it yourself. After assigning your nets, if you need to change your circuit, you'll have to delete a good chunk of your work.
- Draw traces and copper areas, starting with copper areas and lines that take higher current (remember that more current = bigger trace).
  - Unless you have a reason, don't connect the GND net until everything else is connected. You'll want to use the rest of the space on the layer for grounding
  - This step will require some creativity, especially if your parts are close together. Don't be afraid to delete and rework.
  - Avoid using vias on traces willy-nilly. They can be a point of failure in your final product, and make following/checking your work much harder. Vias do a much better job helping with heat dissipation and with small ICs
- With all of your nets (except for ground) connected, create a copper area for the GND net over the entire board, on both sides. This connects all the grounds on a big plane, the "ground plane". This will help with signal noise and heat dissipation.
- Run through your silkscreens, with two things in mind: Do I need this? and Will I be able to assemble/understand the board without this?
  - Most of the parts from LCSC will have silkscreens with them. What I've found is that they come with a lot of information that isn't very helpful (outlines, BOM assignment numbers), and are missing information that is very helpful (resistance, capacitance, which side is on, etc.)
  - Make sure to also earmark what header pins do (ex. A header pin that takes the step input for a stepper driver should be labeled something like "STEP")
- This is another great place to have a friend check your work. The more eyes the better.
- With your design all complete, export the gerbers and send it through JLC's DFM tool (link below).
  - The DFM checking tool can sometimes mis-flag errors (tht to smd is a frequent offender). If something is flagged, read through the Details section to see if its been flagged for an actual problem. Generally, silk screen errors actually need to be taken

    - care of, while some of the routing errors (especially relating to surface mounts) are just a misflag
- If you're satisfied with your design, and it passes JLC's DFM tool, you're good to submit an order (the tips below apply to JLCPCB ordering).
    - In the order form, you can change some of the parameters. Some of them probably won't apply (gold finders, castellated holes, via covering), but some are important.
    - Unless you like lead, you'll probably want to switch your surface finish to Leadfree HASL (or ENIG, but ENIG is $$$)
    - The Confirm Production File field allows you to have them send over the final design file before production. There's not really a need for this unless you have a pretty crazy design.
    - The Mark on PCB field changes how they mark your order on the actual PCB. I usually opt to remove the mark, because I'm not doing big batches or repeat ordering.

Helpful Websites (Especially if you still have questions)

- JLCPCB Capabilities
- Trace Width Current Guide
- JLCPCB DFM Verification Tool
- Altium's PCB Design Rules
- PCBCart Ten Golden Rules