PHYS404 Circuit Simulation

Physics department, University of Illinois Urbana-Champaign

April, 2025, updated September 2026

Contents

1 KiCAD **2**
1.1 Exercise: PCB for the Lockin Amp 3
1.2 Schematic Capture 4
1.3 PCB Layout 6
1.3.1 Edge Cuts 7
1.3.2 Route 7
1.3.3 Ground planes 8
1.3.4 Add labels 9
1.4 PCB fabrication 9
1.5 SPICE simulation 9

2 EasyEDA **11**

1 KiCAD

KiCAD is a circuit software for PCB designs and manufacturing. New versions of KiCAD come with integrated SPICE simulation. With KiCAD, I will demonstrate the PCB functions, including creating circuit schematics, design and layout printed circuit boards, 3D visualization, and show you its nice integrated SPICE capability. First, you need to download KiCAD from https://www.kicad.org/download/; pick the right installation file for your computer to download and install. The current version is KiCAD 10.0.6.

The KiCAD interface, the first time you open it, looks like:

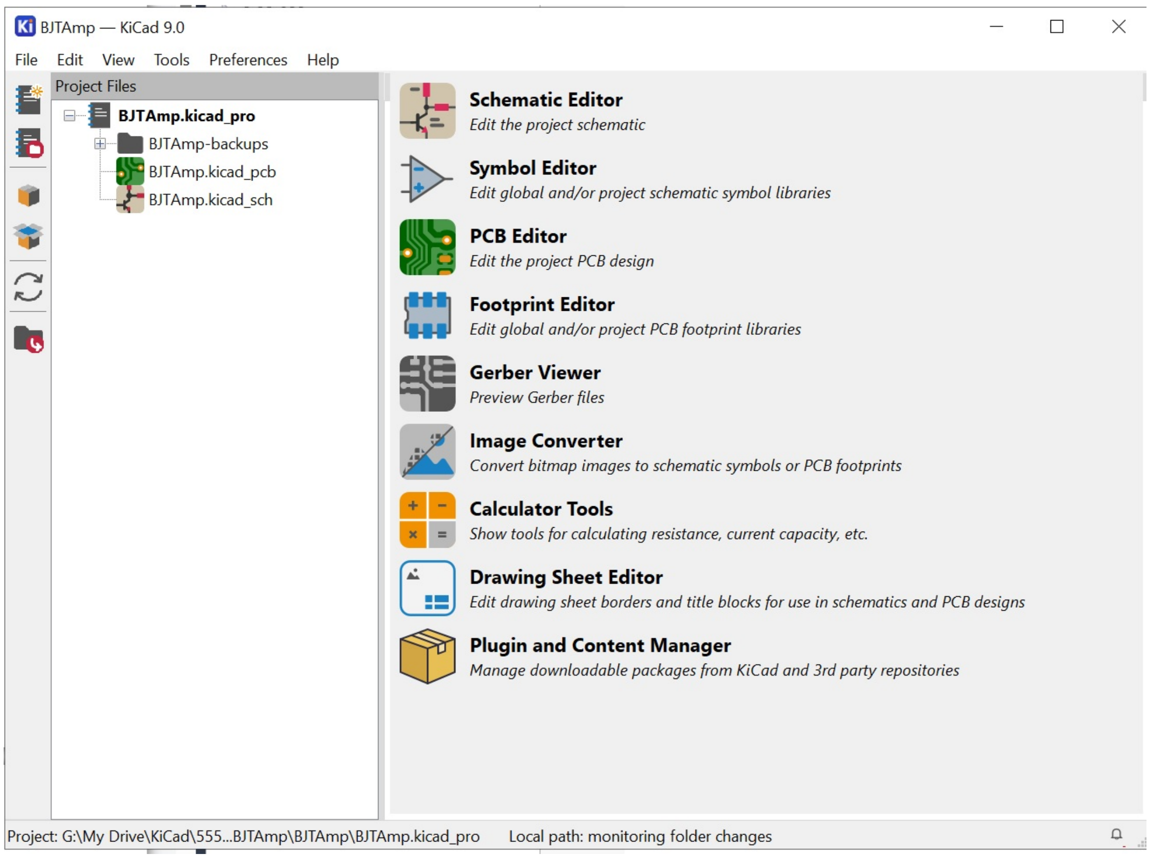


Figure 1: The KiCAD interface window

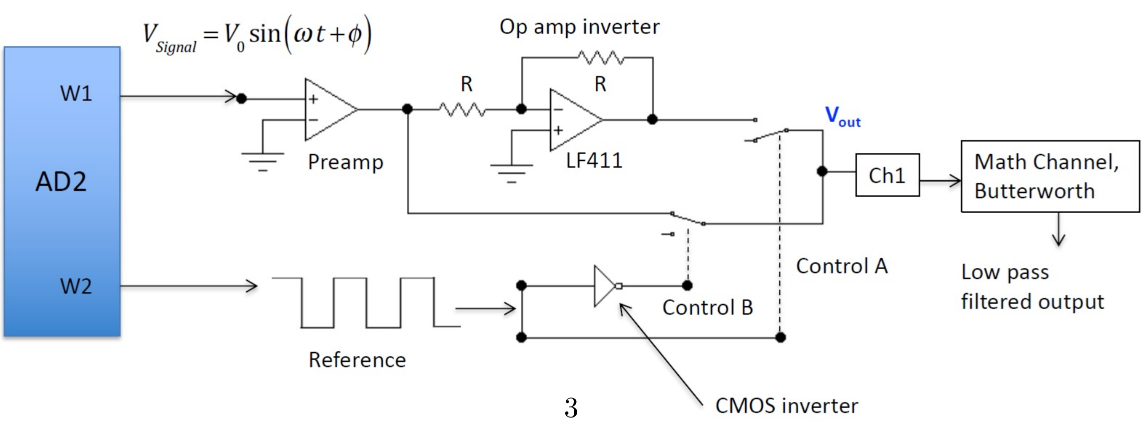
KiCAD is in fact a collection of different software for electronic design automation (EDA). The software suite facilitates the design and simulation of electronic hardware for PCB manufacturing. It features an integrated environment for schematic capture, PCB layout, manufacturing file viewing, ngspice-provided SPICE simulation, and engineering calculation. Tools exist within the package to create bills of materials, artwork, Gerber files, and 3D models of the PCB and its components. To create a PCB design, the two most important functions are the Schematic editor and PCB editor. We will start by creating a circuit schematic, using the Schematic Editor. On the editor, the key shortcuts on windows are listed here. For Mac, see https://hotkeysworld.com/kicad-for-macos.

- a: add symbols (components); after you add one component, you can do Ctrl +right-click to open the component window again to select the next symbol to add to the schematics.
- p: power
- w: wire
- m: move
- r: rotate
- x: mirror
- e: edit properties
- Mouse wheel up/down: zoom in/out
- Mouse click+drag to select components, then the selected components are highlighted.
- Ctrl+d: duplicate the selected components
- Ctrl+z: undo
- Ctrl+s: save
- Esc, or right-click+cancel to get out of the current mode.

Many of these key shortcuts are similar to those in LTSpice.

1.1 Exercise: PCB for the Lock-in Amp

In this exercise, we will create a PCB design for the lock-in amplifier we build in the PHYS404 lab. Here is the circuit diagram:



In this lab, we use several integrated circuit (IC) chips:

- **INA126** (Instrumentation Amplifier) for the preamp,
- **LF411** (JFET-input opamp) for the opamp inverter,
- **CD4066** (Quad bilateral switch) for the switches: only two switches are used
- **CD4049** (6-ch, 3-V to 18-V inverters) for the CMOS inverter: only 1 inverter is needed.

To recreate a full circuit schematic, we need the symbol and footprint for each of the IC chip. Note that the ICs used in the lab are the dual in-line package (DIP); they are larger than the surface mount IC packages (SOIC, SO, TSSOP).

Open the Schematic Editor, type $\boxed{\text{a}}$ to bring out a pop-up window that allows you to choose symbol among the built-in KiCAD library. You will find **CD4066** and **CD4049**. However, **INA126** and **LF411** are missing from the library. We need to download the missing symbol and footprint files. A good website to search for them is digikey (https: //www.digikey.com/), an online distributor of electronic components.

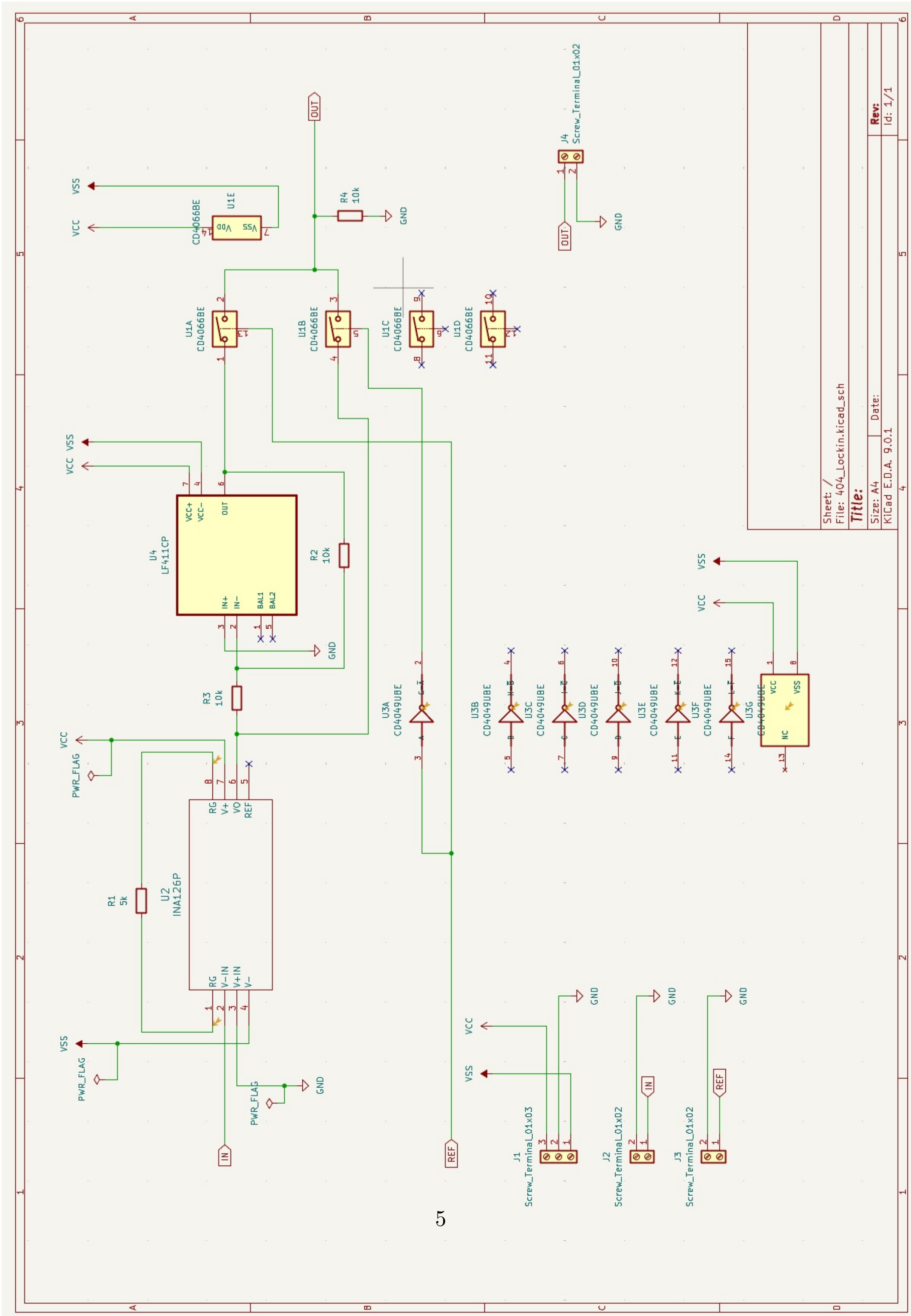
Once you have identified the part on the digikey database, go to $\boxed{\text{EDA/CAD Models}}$, you can find files for the Symbol, footprint, and 3D model of the particular part. At this point, you can find files created by Ultra Librarian and SnapMagic. Save the downloaded files into your local KiCAD project directory and unzip them.

Here is another useful link https://www.rs-online.com/designspark/pcb-part-libraries that provide three libraries: SnapMagic, Samacsys, and ultra librarian to download files for symbol, footprint, 3D model, and SPICE model.

To make these symbol and footprint files available in the KiCAD editors, you need add them to the KiCAD libraries. Bring back the main KiCAD interface window (shown in Fig. 1), and go to $\boxed{\text{Preferences}} \to \boxed{\text{Manage Symbol Libraries}}$. Click the file folder icon to find the directory of your downloaded symbol file (XXX.kicad_sys) and click $\boxed{\text{Open}}$ to have it included in the symbol library. Similarly, go to $\boxed{\text{Preferences}} \to \boxed{\text{Manage Footprint ibraries}}$ to add the new footprint files to the footprint library.

1.2 Schematic Capture

Now go back to the Schematic Editor, and try to build a circuit of the Lockin Amp by wiring up the proper pins between the symbols. For your reference, I am attaching the circuit diagram I have created. I have added one 3-pin terminal block to bring in the 5V (VDD) and $-5V$ (VSS) to the circuit from AD2, and three 2-pin terminal blocks for the signal in, reference in, and signal out. To make the schematic neat, I will suggest you to use the Power Symbol for VCC and VSS, and Global Labels to define the inputs (signal in, reference in) and the output.


U1A
CD4066BE
U1B
CD4066BE
U1C
CD4066BE
U1D
CD4066BE
U1E
CD4066BE
U2
INA126P
R1
5k
U4
LF411CP
R2
10k
R3
10k
R4
10k
U3A
CD4049UBE
U3B
CD4049UBE
U3C
CD4049UBE
U3D
CD4049UBE
U3E
CD4049UBE
U3F
CD4049UBE
U3G
CD4049UBE
J1
Screw_Terminal_01x03
J2
Screw_Terminal_01x02
J3
Screw_Terminal_01x02
J4
Screw_Terminal_01x02
VCC
VSS
GND
PWR_FLAG
IN
REF
OUT
Sheet: /
File: 404_Lockin.kicad_sch
Title:
Size: A4
Date:
Rev:
KiCad E.D.A. 9.0.1
Id: 1/1

Once you have finished the whole schematic, run the 'Electrical Rules Checker' by clicking the icon (with a check mark) on the tool panel. You will get errors, mostly due to two issues. The first type of error arises from the unused and unconnected pins on the IC chips. Place 'No Connect Flag' to terminate each of the unused pin. The remaining errors come from the power pins for VCC, VSS and GND. The error message is 'Input Power pin not driven by any Output Power pins'. Add a 'PWR_FLAG' symbol to the power pin; this indicates that the power for these pins comes off the board.

Once your schematic is error-free, save the schematic file with Ctrl + s, and click the PCB icon (the last icon on the tool panel) to start the PCB layout.

1.3 PCB Layout

An empty PCB Editor will show up on your screen. Click the icon 'Update PCB from Schematic...', which is on the left of the 'Design Rules Checker', the icon with a check mark. Click Update PCB towards the bottom of popup window. If there are no errors, then your parts will show up in the middle of the editor.

Errors likely occur, as one or more parts might be missing footprints. You will need to go back to the Schematic Editor, click the symbol until it gets highlighted, and then type e to bring out the 'Symbol Properties' window. On the Footprint entry, choose the right footprint to associate with the symbol.

Once you have all the footprints, then you will see all the parts and the wires connecting the pins. You can do a 3D rendering of the board by pressing Alt + 3. You can use your mouse to click and drag the window to change the viewing angle of the board. On the board, you will see the footprint, and the associated part showing up. If you don't see the physical part, then it is probably missing the file for the 3D model. You can import 3D model by following these steps:

1. Go back to the main KiCAD window, open the Footprint Editor.
2. On the left panel, search for the part you want to modify. Load the footprint to the editor.
3. Once the footprint shows up, click the window, and type e to open 'Footprint Properties' or go to File → Footprint Properties → 3D Models. Find the appropriate folder on the computer to link to the CAD file (.step) of the IC chip part. Once the part is loaded, you can use Rotation and Offset to place the part in the right place to line up with the footprint.
4. Once you are happy with the 3D rending, save and close the footprint directory. Update PCB again, and you should see the part added to the board with Alt + 3.

1.3.1 Edge Cuts

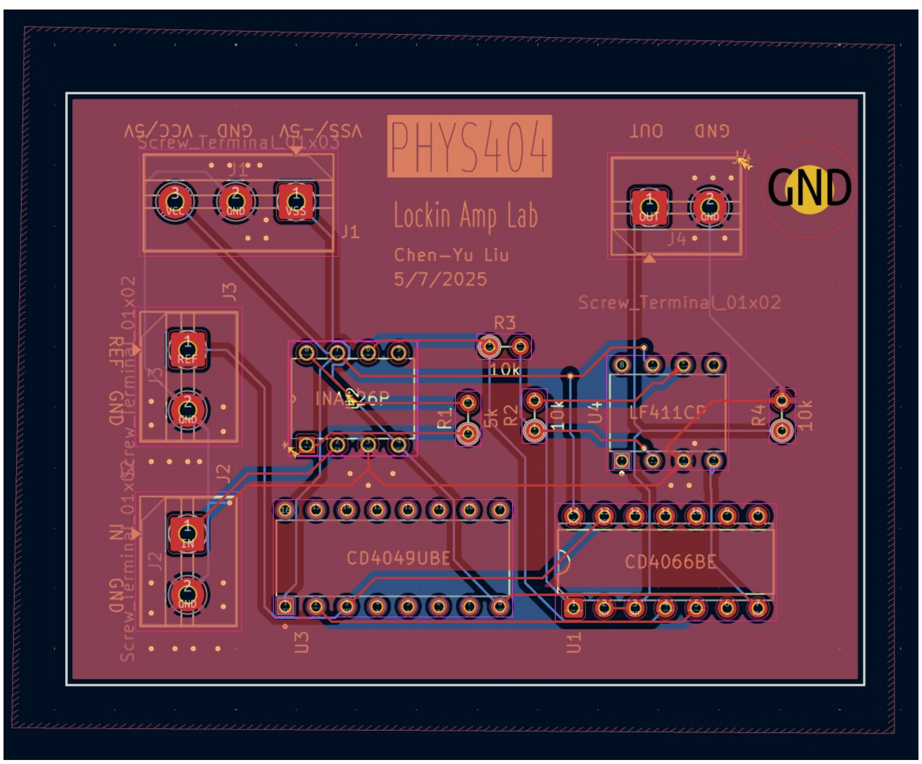
In the PCB Editor, you can investigate different layer of the PCB by clicking the various available layers listed on the Appearance panel on the right of the window. You can also hide specific layer(s) by clicking on the eye icon for each layer. We need to add a physical board. Click the 'Edge.Cuts' layer, and add a rectangle by selecting the 'Draw Rectangles' tool on the right tool panel. Draw a rectangle to enclose all the parts you have; arrange them inside the rectangle by moving the parts around and rotating to change their orientations.

The rectangle defines the boundary of the board, and it is important that it belongs to the *Edge.Cuts* layer. Otherwise, you will get an error message.

1.3.2 Route

Once you have organized the parts to their final positions, you can do routing of the connecting wires. Instead of routing individual wire, it is more efficient to perform autorouting to let the computer algorithm figure out optimized wire paths and inter-layer connections.

To enable auto routing, you need to install the Freerouting plugin. Go back to the main KiCAD window, click 'Plugin and Content Manager'. In the Repository, find Freerouting, and install it. Once the plugin is install, you will see another icon added to your tool panel in the PCB Editor. When you first run freerouting, your computer might also need to install Java JRE. After autorouting, the wire traces are getting more organized as shown:

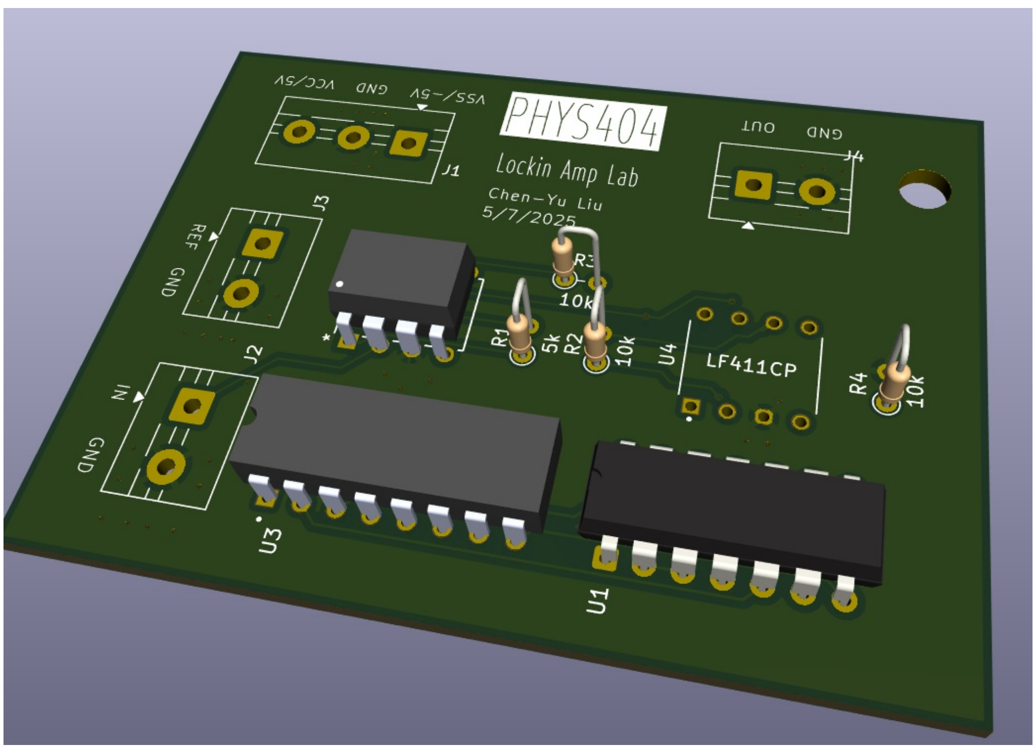


1.3.3 Ground planes

To ensure a better noise performance, it is recommended that you cover the bottom surface of the PCB with a pour layer of solid copper. On the PCB editor, switch to the bottom layer, and select the 'Draw Filled Zones' tool on the right tool panel. It will provide a tool for you to draw a polygon on the editor window. Draw a closed polygon to enclose all visible area of your board (as shown by the dashed line in the photo above), then press the $\boxed{\text{b}}$ key. It will automatically fill the bottom layer with copper, while avoiding the power wires and signal wires.

A ground plane in PCB design is a low-impedance, conductive layer that provides a return path for electrical currents, reducing loop area and thus minimizing noise and interference. It's essentially a large, interconnected ground wire that acts as a common reference point for all signals.

For optimal noise rejection, you might also want to add a pour layer on the top surface, too. Add a generous amount of Vias (the small golden circle icon on the right tool panel) around all the GND pads; these vias connect copper layers on the front and bottom surfaces. As a result, the two ground planes are stitched together and shorten any possible electrical path to ground.



At this point, you can run the 'Design Rules Checker' by clicking the icon (with a check mark) in the tool panel on top of the editor window. You might have errors, if the wire traces overlap each other or with the pads. You can move the wire traces by clicking and dragging or deleting to solve any overlapping issues. If you reposition the wires, then make sure to refill the copper zones on the front and bottom layers.

1.3.4 Add labels

You want to label the individual parts and add information on the PCB, including the model number of each IC chip and the value of the resistors, capacitors, and inductors; the information will greatly facilitate the work to assemble the components on the board. To print the values, you will need to click each component and type $\boxed{\text{e}}$ to bring up the 'Footprint Properties' window. On the 'Value' field, change the layer from 'F.Fab' to 'F.Silkscreen'.

I also used the 'Draw Text' tool to indicate the power connection, signal in, reference in, and signal out to the specific pins on the terminal blocks. You might want to add additional text for your project name, designer, dates, revision version, etc.. The 3D image of the final board is shown in the picture above.

1.4 PCB fabrication

There are many PCB manufacturers in the US and overseas. To order the PCB, we need to submit the design files in the format of Gerber job files.

On the PCB Editor, go to $\boxed{\text{File}} \rightarrow \boxed{\text{Fabrication Outputs}} \rightarrow \boxed{\text{Gerbers}}$. A 'Plot' window is opened. You can check the different layers to include. You can also $\boxed{\text{Run DRC}}$ again to check for errors. Plot (to generate many .gbr files, one for each active layer) and 'Generate Drill Files' (to generate .drl file). You can also generate a Bill of Materials (BOM) spreadsheet to help you order components needed to populate the PCB.

You can view the gerber files using the Gerber Viewer utility on the main KiCAD window. The gerber files and the drill file should be compressed into a single .zip file, which can be used to submit to PCB companies for online quote and order.

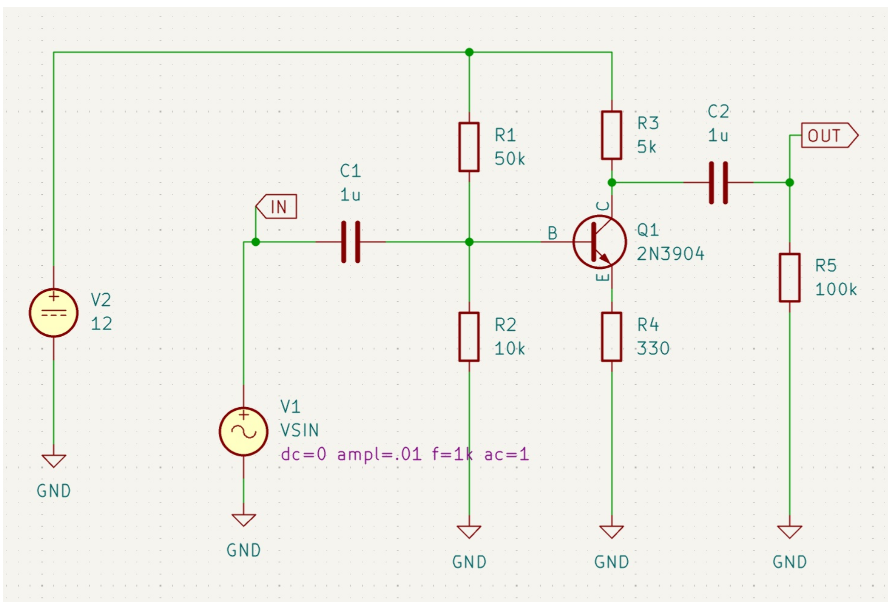
Order printed circuit board: https://jlcpcb.com/

1.5 SPICE simulation

KiCAD has ngspice incorporated. It suggests a list of places to find Spice model parameters for parts: https://ngspice.sourceforge.io/modelparams.html. To run the SPICE simulation, one has to make sure that each symbol has a SPICE model loaded. If the SPICE model is not available, then one needs to be downloaded. The best source to download the SPICE model is the official IC chip manufacturer. SnapMagic (https://www.snapmagic.com/) has a good database of parts with SPICE models. To import and associate the SPICE model, do the following:

- Go to the Schematic Editor, click the part you want to edit.
- Type e to open 'Symbol Properties' window; on the bottom of the window, click Simulation Model... .
- Import the SPICE model by specifying the path to the model file (.lib, .sub, or .ibs)
- Don't forget to click the 'Pin Assignments' tab to check that the symbol has the correct pin assignments corresponding to the model.

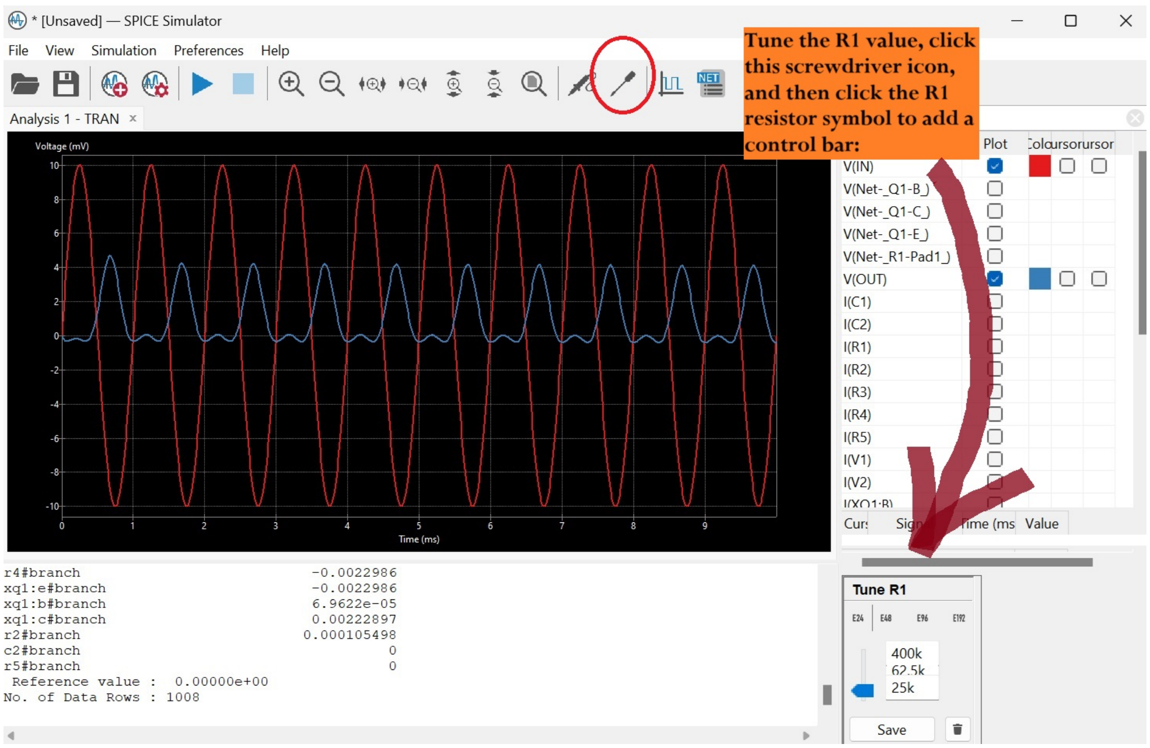
Once all the SPICE models are loaded, then running the SPICE simulation by clicking the 'simulator' icon on the top tool panel. It will open another popup window to define the simulation and display the results. I quickly drew a BJT amplifier circuit, and tried to download the model of transistor 2N3904 (from SnapMagic: https://www.snapeda.com/search/?q=2n3904&search-type=parts) that is used in the P404 transistor lab.



When first starting the simulator, I received an error that indicated that Pi is not defined, so I replaced Pi in the model by 3.14. With this change, the simulation runs, giving similar results as what is available in LTSpice using a generic NPN transistor, but with a different saturation current. There are a few nice features available in the KiCAD simulation environment:

1. We can perform scope measurements on the signal, by left-clicking the selected signal and then right-click to bring up measurements: MAX, MIM, Average, RMS, Integral, etc.., and fourier analysis.

2. We can also tune the value of the circuit components using a tuner function. To change the value of R1, first click the tuner icon (the screwdriver), and then go back to the Schematic Editor to click R1. As shown in the figure below, a tuner bar is added to the SPICE Simulator window. After I slide the bar to change the R1 value, a new simulation is performed and the waveform traces are updated.

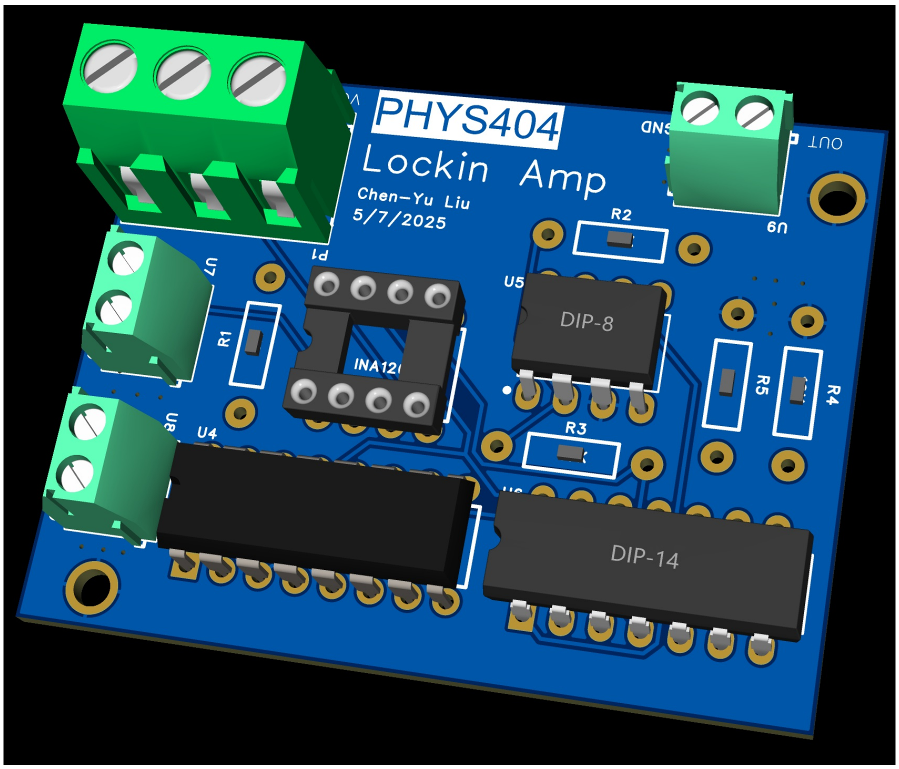


To learn more about in-depth functions of kiCAD, digikey provides some useful resources and tutorial videos: https://www.digikey.com/en/resources/design-tools/kicad

2 EasyEDA

EasyEDA (https://easyeda.com/) is an online PCB design & circuit simulator tool. It runs on remote servers and your design files are stored on the cloud. EasyEDA follows a similar workflow as KiCAD. The Pro Edition improves integration between the various editors and utilities. The best feature of easyEDA is its integrated online searchable databases of components and their files for symbol, footprint, 3D model, and SPICE model. This greatly reduces the amount of time and effort needed to collect and import model files.

It has all the desired functions, like autorouting and many other routing features. It has managers for layers, 3D models, SPICE library, etc. making it easier to find things and assign features across parts. The interface has direct links to order from LCSC (for parts) and JLCPCB (for PCB layout and 3D printing), two the the largest Chinese suppliers. Below is a screen capture of the PCB model I have created using easyEDA on https://pro.easyeda.com/ for the same PHYS404 Lockin Amp circuit, and I am quite impressed by the functionalities and the ease of use of this online tool.



EasyEDA can also simulate circuits (based on LTSpice), however, the simulation mode (only available in the Std Edition) seems to be separated from the PCB mode with no easy integration between the two.