YOUR NAME:	YOUR SID:
YOUR PARTNER'S NAME:	YOUR PARTNER'S SID:

Score: ___/200

Electroencephalograph (EEG)

Final Project: Electroencephalograph

ELECTRICAL ENGINEERING 40

INTRODUCTION TO MICROELECTRONIC CIRCUITS

University Of California, Berkeley

Department of Electrical Engineering and Computer Sciences

Professor Michel Maharbiz, Professor Vivek Subramanian, Professor Bharathwaj Muthuswamy, Vincent Lee, Weijian Yang, Dr. Winthrop Williams

Lab Contents:

Printed Circuit Board Layout Phase

You should now have a functional circuit design of your EEG that is 100% bug free (at least we hope). The next phase of the project will be to design a printed circuit board layout. We will be using Ultiboard as our board layout editor because of its compatibility with Multisim.

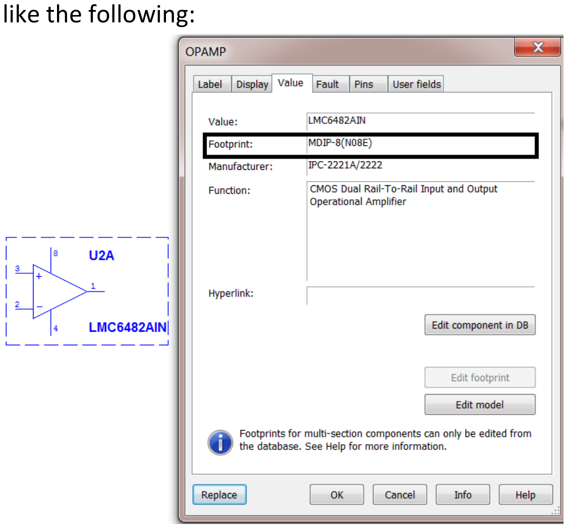
We will provide you a library of components for you to use in your simulation so that they will transfer to Ultiboard correctly. Please download "EE40_FinalProjectComponents.ms11", and "UsrComp_S_Alice.usr" from bspace.

Before we transfer to Ultiboard, we want you to do a couple of final checks to ensure that the transfer goes smoothly.

First, we want you to make sure that you are using all of the connections on each of your operational amplifier packages. Remember that the LMC6482 or TLC277 are dual packages and have two operational amplifiers per package in the simulator and should show up as UXA and UXB where X is a number. In your simulation, make sure you use both operational amplifiers in the package, otherwise you will have too many half used ICs and violate the specifications.

Secondly, make sure that you choose non-virtual and through-hole components for your part footprints.[1] These footprints will appear on your final PCB layouts, so it is important that the footprint for each component is correct. To ensure this point, please follow the guidance below step by step.

1. Download the MultiSim file "EE40_FinalProjectComponents.ms11" from bspace. Open this file, and you will see all the different components that we will use for the final project. Double click the operation amplifier, and you can see its footprint, like the following:



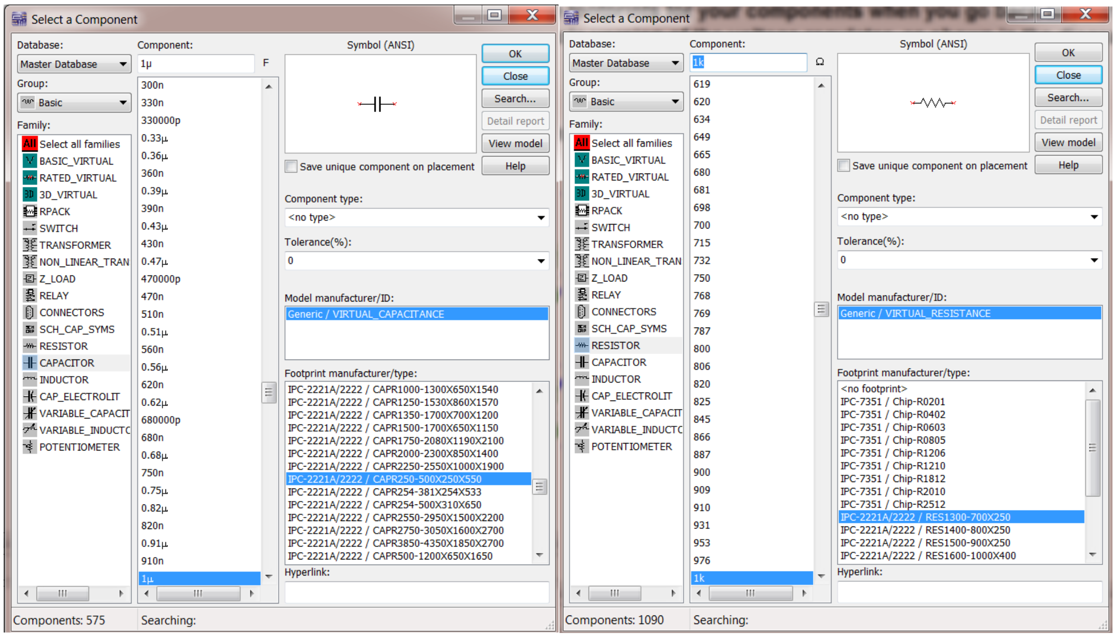
   Please check that the operation amplifier in your own MultiSim file has the same footprint. If not, you probably chose a wrong amplifier, and please replace it by TLC277CP.

2. Do the same thing for all the capacitors and resistors (forget about the potentiometer at this moment). First check the footprint on "EE40_FinalProjectComponents.ms11", and then go back to your own file to check. Most likely you are not using the same footprint or you are using a virtual component. To correct this, simply delete your original capacitors / resistors, and place a new one with the correct footprint (see the following

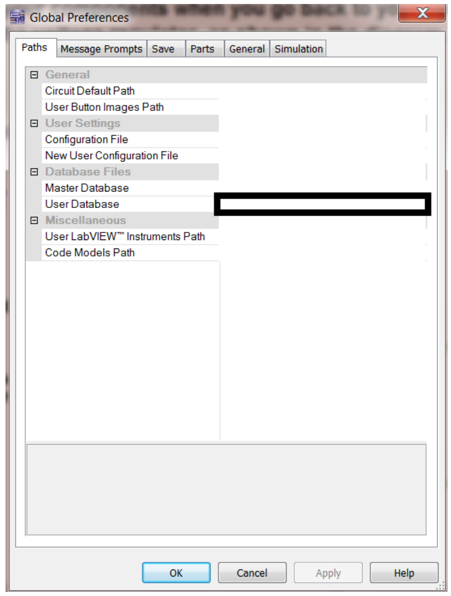
[1] If you are using the SMT voltage regulator, then make sure you choose the SMT footprint for the simulation

figure for two examples). Please make sure you do not mess up with the connection and the value of the capacitance / resistance. Please also note that different capacitors (0.01 μF, 0.22 μF, 1 μF etc.) may have different footprints. Refer to "EE40_FinalProjectComponents.ms11". After this step, all your capacitors and resistors should be in blue color in MultiSim.

If you use parallel or series resistors to make up new resistance values in your circuits, please make sure you include the actual parallel or series resistors layout in your MultiSim.



3. Now we move onto the potentiometers and the voltage regulator. The footprints for these components, unfortunately, are not standard for our final project. So we have to customize the footprint by measuring the physical dimension of the real components. This is done by the EE40 teaching team and saved in a database. You will need to import this to your own computer. To do this, please first download the database file "UsrComp_S_Alice.usr" from bspace. Then in your MultiSim interface, go to Option→Global Preferences, under that tap "Paths". You will see where your own "User Database" is stored. Open your Windows file browser, and go to where your own user database is stored. You will see your own user database file "UsrComp_S_YourWindowsLoginName.usr". Now close the MultiSim software, and place this file into another folder as a backup, and put "UsrComp_S_Alice.usr" into the original folder. Change its file name from "UsrComp_S_Alice.usr" into "UsrComp_S_YourWindowsLoginName.usr". We are now done with the database import. Please note that this import method is really brute force. There are smarter ways to do this, but those functions might not be enabled in our software package.

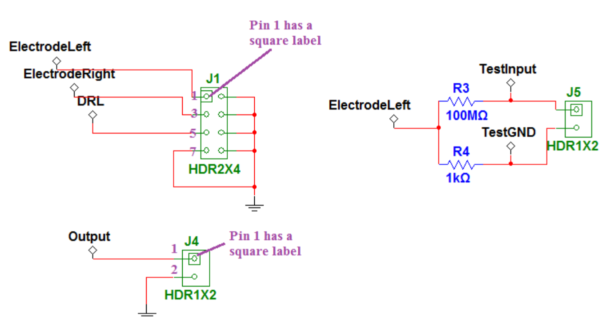


4. Go back to “EE40_FinalProjectComponents.ms11”. Copy the potentiometer component into your own MultiSim file, and replace your original ones with it. When you do this, please make sure that you do not mess up with the circuit connection, and have the correct resistance value for the potentiometer. Please note that the potentiometer will still be in black color, and it is fine. You will have one potentiometer for the instrumentation amplifier stage and one for the active low-pass filter stage.

5. Do the same for the voltage regulator. Replace the ones in your file with those in “EE40_FinalProjectComponents.ms11”. Please note that we provide both through-hole footprint as well as surface mount (SMT). “LM78L05CT” and “LM79L05CT” are through-hole footprint, while “LM7805CT” and “LM7905CT” are SMT. Please only choose either through-hole or SMT.

6. It will be a good idea to use a header strip to organize your input signal and output signal port. These header strips are available in “EE40_FinalProjectComponents.ms11” for you to copy to your own file. When you copy over, please make sure the “On-page Connectors” are correct in your own circuit. The pin numbers of the header strips are shown in purple in the following diagram (they will NOT show up in your MultiSim file). These numbers will show up in your PCB layout.

7. To facilitate circuit testing, besides input signal and output signal ports, please place header strips as test pads for the output of instrumentation amplifier, as well as active low pass filter. Please also place a voltage divider as a testing port for your circuit. The following shows an example.

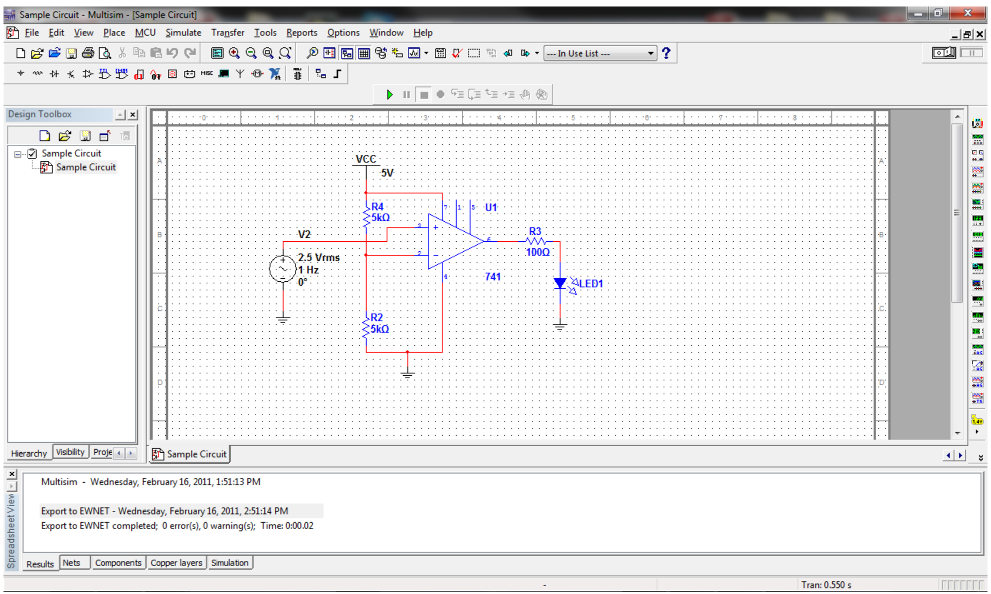


8. If you have power supplies in Mulitsim, it will show in black color. There would not be any footprint for it, and it will not show up in the PCB layout.

Transferring to Ultiboard

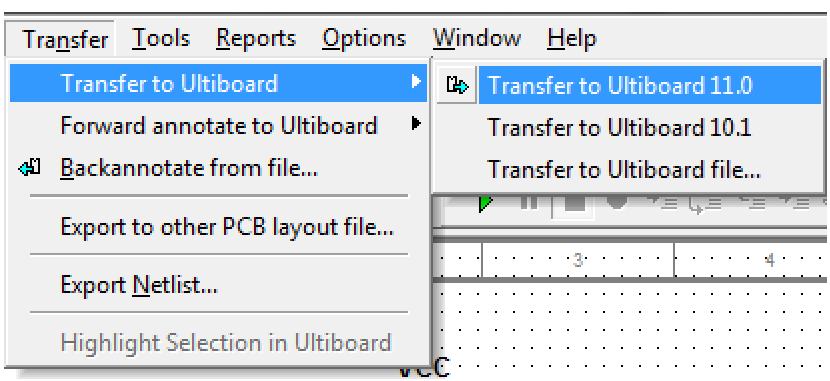
What makes Ultiboard nice is that we can directly transfer our schematic from Multisim. To transfer your schematic to Ultiboard…

Start by opening up your schematic. The schematic we used is a Sample Circuit so your schematic and board layouts will look different from the ones shown in this procedure.



Make sure that all the parts in your design are non-virtual component and are outlined in blue (except for potentiometers and header strips). Ground and V_{cc}, V_{ss} connections are excepted from this condition.

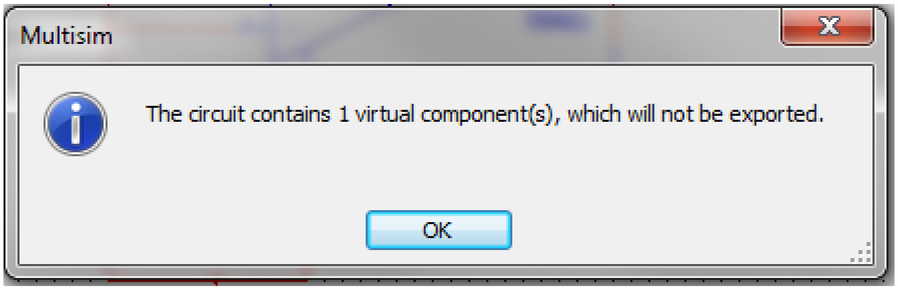
To transfer the components, go to Transfer->Transfer to Ultiboard->Transfer to Ultiboard 11.0.



Transferring to Ultiboard

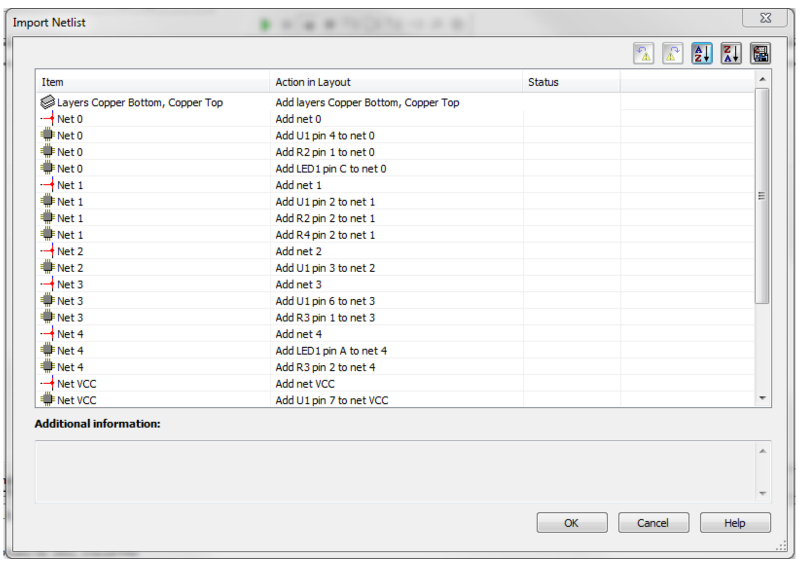
A prompt will appear requesting you to name the Ultiboard file you want to associate with your schematic. Once you name the file, Multisim will export your design to an Ultiboard file and a new Ultiboard window will show up.

Unfortunately for our project, if you have power supply or voltage source in your MultiSim file, you will get an error window that looks like the following for each virtual power supply we have in Multisim. Your pop up window should read "The circuit contains X virtual component(s), which will not be exported" if you have X virtual power supplies. If you have more than X virtual components that are not transferred, you did something wrong and need to go "un-virtualize" some of your components. Once you fix the errors, try this step again.



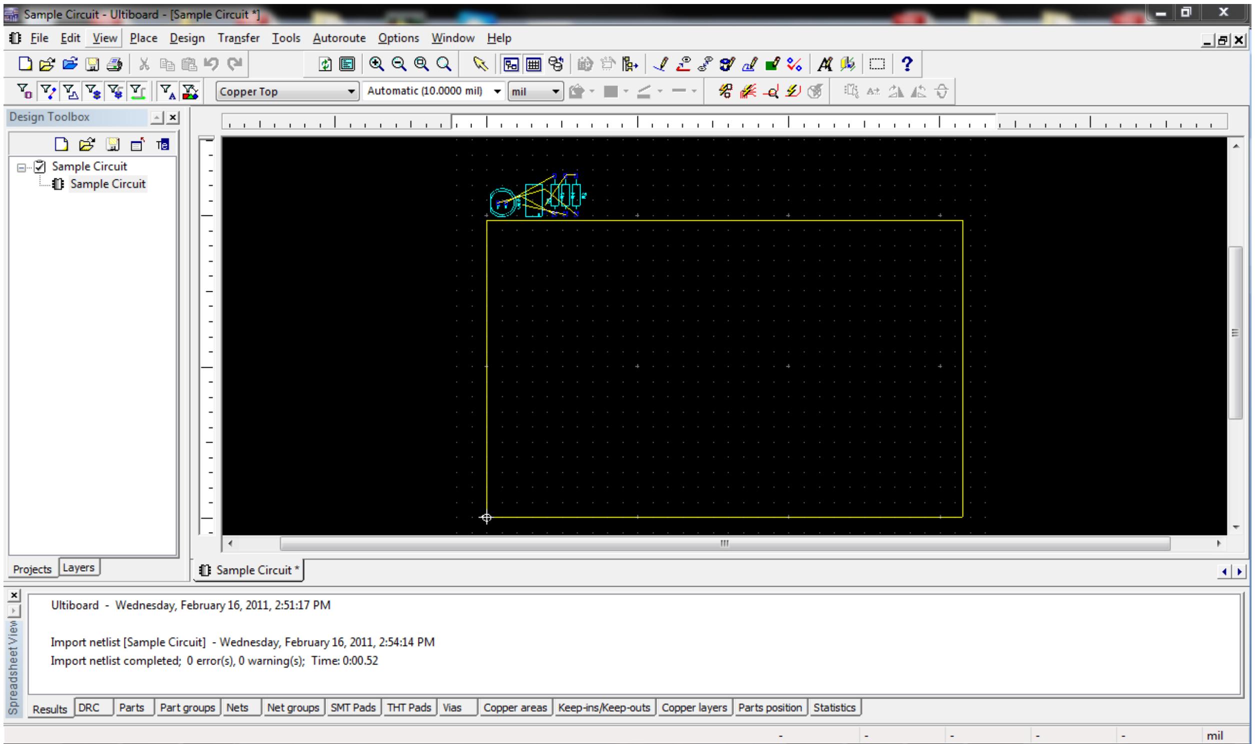
Virtual Component Warning

When Ultiboard starts up, you will see an Import Netlist window that summarizes the connections that are in the schematic such as the one shown below.



Import Netlist Window

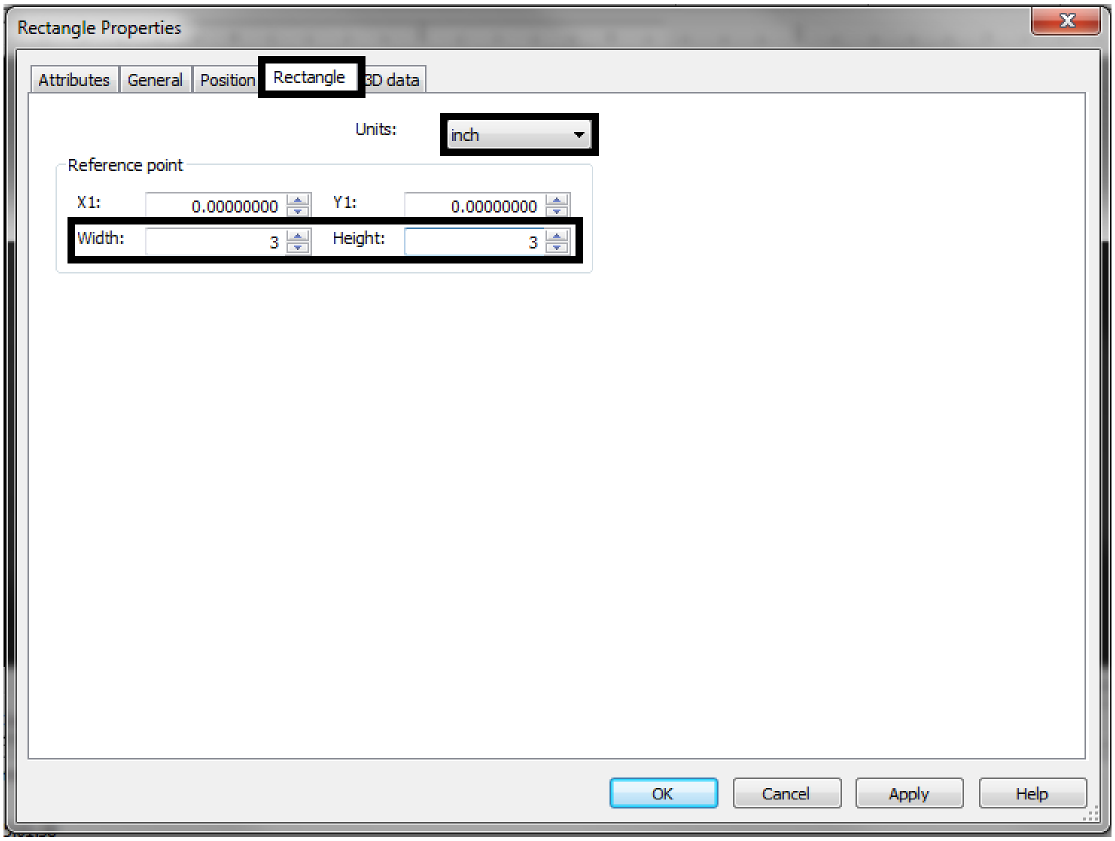
Click OK and you should get a window that looks similar to the one shown below.



You will notice that the circuit components that you wired in your schematic have been converted to show how they would appear on a printed circuit board. At this point you will want to check to make sure that none of your components are surface mounts with the exception of the voltage regulator for those who opted to use the surface mount version. If you have SMT components where they shouldn't be, you will have to go back to Multisim and either change the footprint or choose an equivalent component with a through-hole footprint.

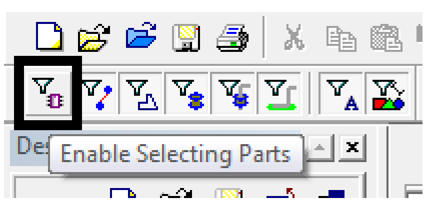
Before we start moving parts onto the board, we need to adjust the dimension of the region outlined by the yellow box, which represents the board. To adjust the board dimensions, right click anywhere on the yellow rectangle and select Properties.

A window named Rectangle Properties will appear. Navigate to the Rectangle tab and adjust the width and height parameters to the specifications (width = 5 inch; height = 3 inch). Make sure to use the correct units.



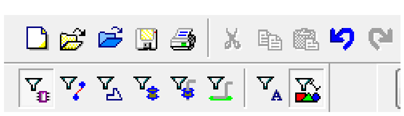
Rectangle Properties Window

We next want to start moving parts onto the board. In the top left area of the Ultiboard window, there is a toolbar that turns on and off the accessibility of certain aspects of the component layer. In order to select and move components around, we want to Enable Selecting Parts by clicking the button shown below.



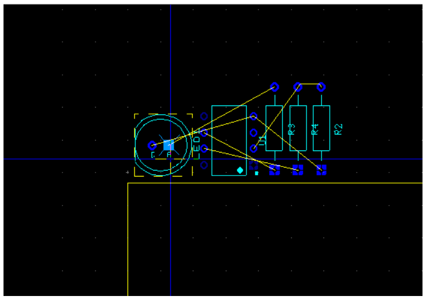
Enable Selecting Parts

You will now be able to select components and move them. Notice that the lines that connect each pair of nodes moves with the component accordingly. To avoid mistakenly deleting a wire, we turn off most other accessibility. You should have your tool bar look like the following before continuing:

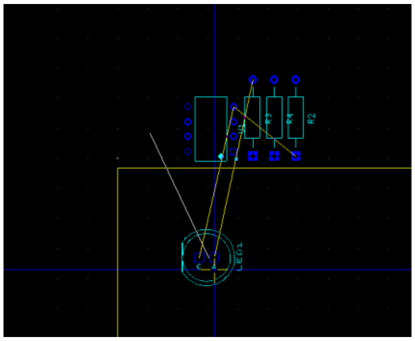


Only turn them on when necessary (e.g. when adding vias, when you try to move the labels, etc).

Placing Components on the Board



Select Component



Move Component onto Board

Move all of your components onto the board. To make our lives easier later for the routing process, you will want to minimize the number of intersections between each of the wires. Like in Multisim, you can rotate a component with Ctrl+R.

There are a few considerations that we need to consider when we place these components on the board. These are for organizational purposes that will make routing the wires on the board easier.

The first concerns component placement and alignment. During the placement phase, it is very important to make sure that we place our components in an organized manner because it will make the routing phase easier. One of the most annoying aspects of the routing phase is to make connections that span all the way across the board. Although sometimes this may be necessary, the more mileage we accumulate during our routing process, the harder it becomes to fit all of our wires and components on the board.

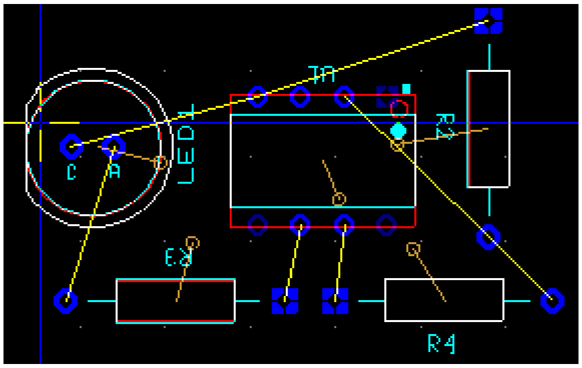
With that being said, one of the ways we can optimize our wiring is by placing components with a large number of connections closer to each other. We can also align ICs and circuit components with respect to each other (we will see how this helps later).

Also we expect you to orient all of your text and IC in the same direction. For instance, all components with text going in the horizontal direction should be placed so that if you read the board from the bottom side, all the text upside down or right side up. The same goes for the vertical direction.[2] Note that if you use the built in autoplace option in the toolbar, your components will not obey this constraint and we will be able to tell if you got lazy and used autoplace.

[2] Note that the examples in the next couple of figures do obey this practice

We also want you to align your components on the board relative to each other so that they line up and look nice. (This is not a joke). It makes routing a lot easier if your resistors are in columns so that you don't constantly have to route around randomly placed components.

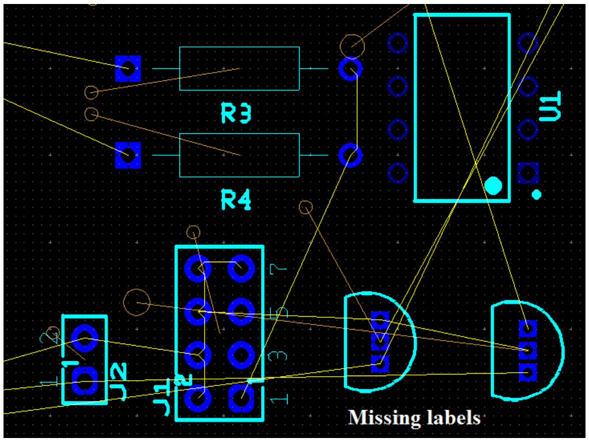
Finally, we want to minimize board space and use every square nanometer at efficiently as possible. If we need more space, we could always move parts around.



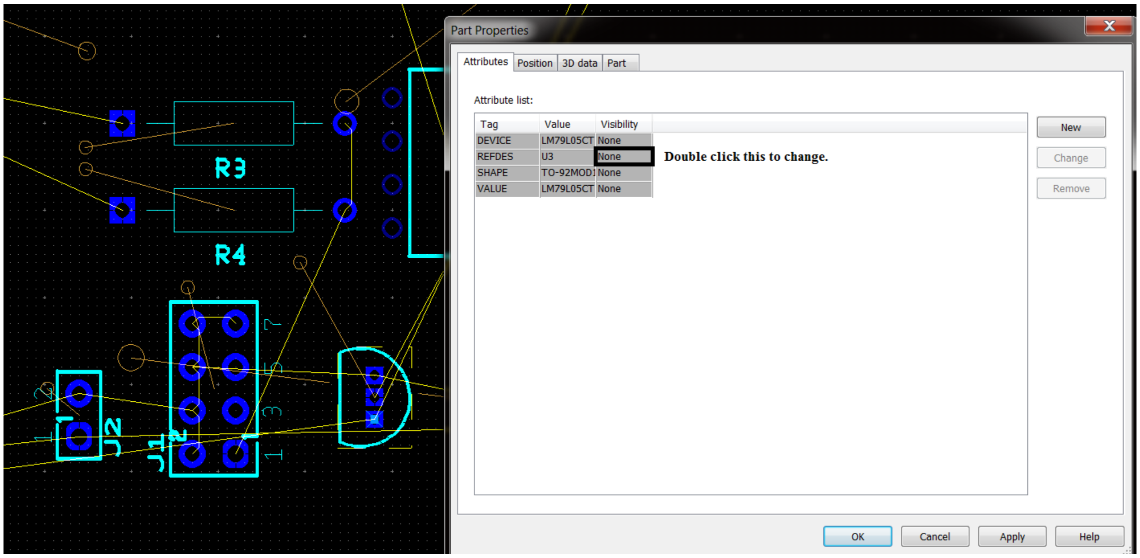
Components on Sample Circuit

Checking Labels for Each Component

Each component on the PCB should have its own labels, such as U1, J1, J2, R3, R4 shown in the following. These labels are important because they will guide you when you later solder each component on the PCB. Some of the components, however, might be missing labels, as shown below for the two voltage regulators.



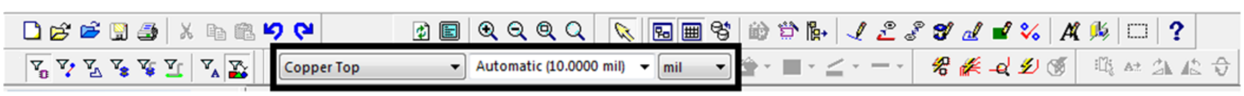
In order to fix these problems, double click the components that are missing labels. This will lead to the "Part Properties" window. In "Attributes" tap, you should be able to change the visibility of different labels. We need to turn on the label for "REFDEF".



Routing Traces on the Board

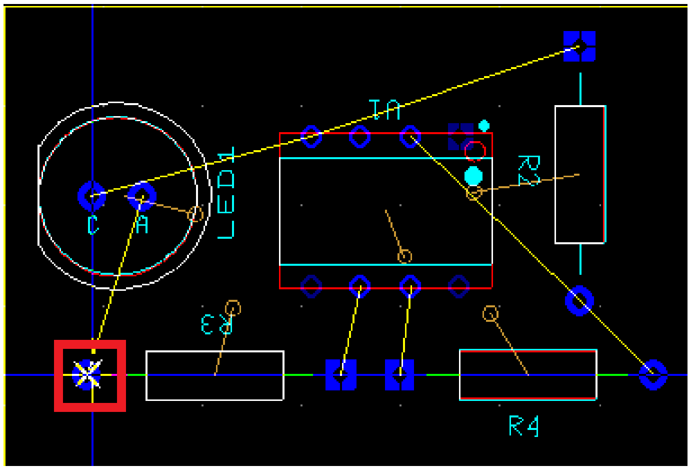
Once we have placed all of our components on the board, we need to route all of the wires.

To being the routing process, make sure that you have the Copper Top layer selected for your traces.



Next, select the Place Wire tool. This will enable you to select pads and begin placing the traces that connect each pad together.

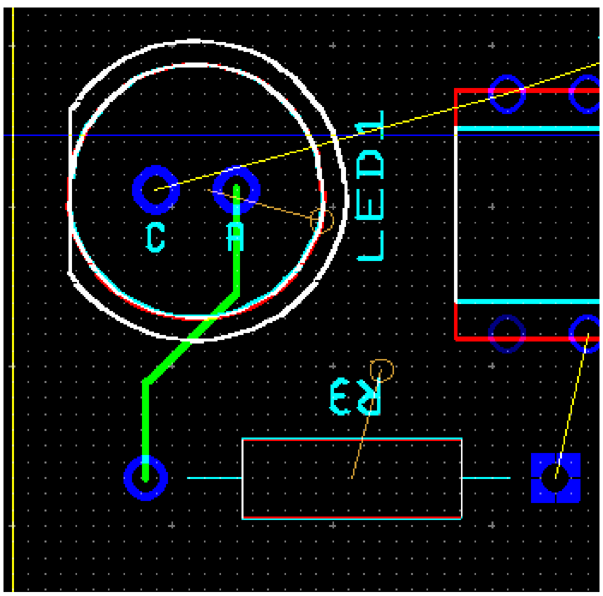
Your cursor should now lock on and turn into an ‘X’ when you hover over a pad as shown below.



Place Wire Tool Locks Onto Pad

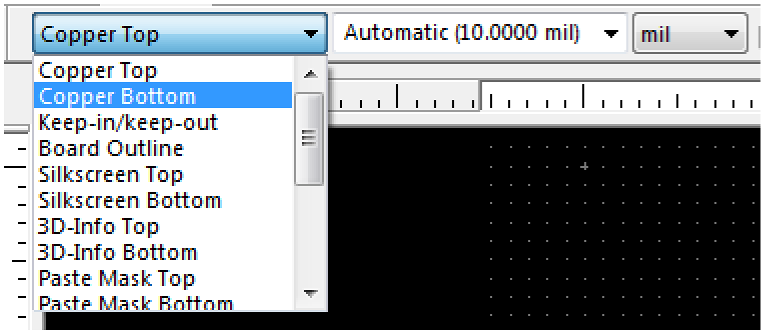
Click on the pad and connect the wire to the other pad. You will notice that the yellow line will disappear and a green line (representing a top layer trace) will takes its place.

Once you have connected the two pads together, press ESC or right click and select Cancel. This will terminate the routing of that trace.



Routed Trace

The lines we have made so far are green, indicating that they run on the Copper Top layer. But what if we have to cross one of the traces on the top layer? In order to solve this, we will need to route traces on the bottom layer. To do this, simply select Copper Bottom from the toolbar menu as shown below.

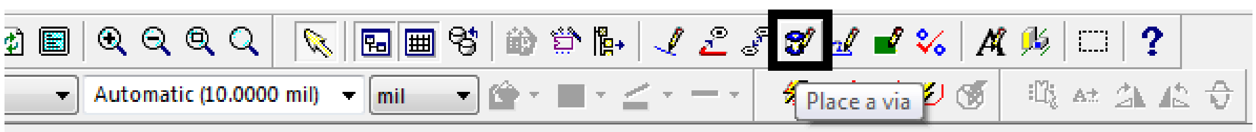


Select Copper Bottom Layer

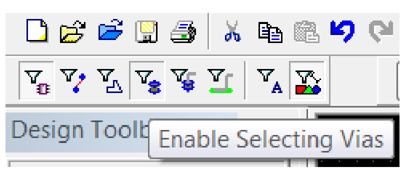
Now when you route the traces you will notice the resulting traces are red.

Another issue that we may encounter when routing our board traces is the need to switch layers in the middle of a trace. In other words, we have a wire running on one layer and need to cross over to the other layer in order to complete the trace. To solve this problem, we use what is called a **via**. Vias are simply holes in the board that appear as pads on both sides of the board.

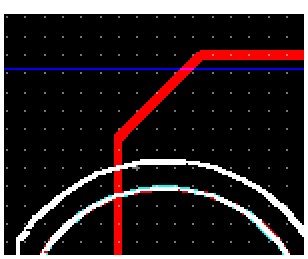
To place a Via, select the Place Via tool in the toolbar as shown below.

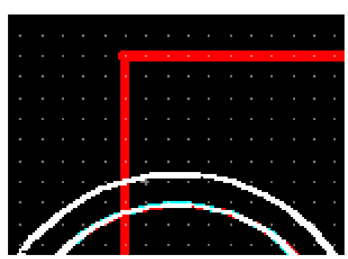


Then click the position on the board that you want to place the via. A window will appear prompting you to specify which layers the via will correspond to. For our purposes we simply want to go from Copper Top to Copper Bottom or vice versa; but in more complicated layouts that use multiple layer PCB, this becomes important. You may also want to "Enable Selecting Vias".



There are several more important considerations that we will want to implement in our design. In all PCB layouts, the maximum angle that a trace can bend is 45 degrees. This rules out all right angle bends, therefore, you will need to ensure that all traces on your PCB obey this rule.


Good Trace


BAD Trace

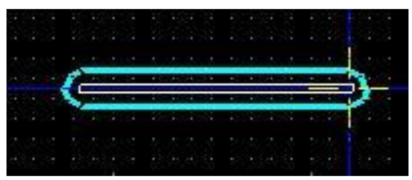
Another important consideration is the practice of biasing the board. Biasing the board means that we try and route all connections going up or down on one side of the board, and route the rest of the connections going left or right on the other side. This practice allows us to introduce a degree of organization into our layout. It also tends to reduce the number of difficult connections. Note that we don't have to follow these rules stringently; for instance, we might be forced to route traces certain directions simply because there is no other way to route them.

Finally, do NOT use the autorouter. The autorouter introduces a lot of unnecessary bends and does not adequately express the 45 degree bends that you are required to implement. In other words, the autorouter fails and you will be docked if you use it.[3]

Adding Jumpers

Sometimes after routing most of the board, you find that there is one connection that is absolutely impossible to route without crossing another wire. Or you might find that you have to use an unreasonable number of vias to accomplish this last routing. In either case, in such dire circumstances, we introduce a solution called a jumper which is essentially an air wiring that shorts two points on the board.

To place a jumper on your schematic, first make sure that a copper layer is selected in Ultiboard. Go to Place -> Jumper and your cursor will turn into a Jumper tool (a light blue circle). Click where you want the jumper to start and click where you want the jumper to end as shown below.



Placing a Jumper

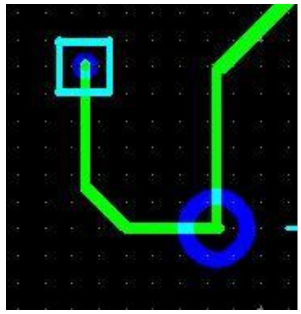
Again, we only use jumpers if absolutely necessary. We've given you enough space for the board, so if you need to use jumpers you're doing it wrong.

Adding Test Pads

For debugging purposes, we want you to include test pads on your design. A test pad allows us to check voltage outputs at critical nodes in your circuit and facilitates the debugging process. You should include test pads at critical points in you circuit so that you can check to make sure it is behaving properly after the soldering phase.

To add a Test pad, go to Place -> Test Point. Put your Test Point near the part of the circuit you want to test later and connect it using the Line tool. For example, the test point shown below (square box) is connected to a pin that we want to test (large blue circle). (Alternatively, add header strips in MultiSim for these test pads before you do the PCB layout. They will then get transferred to Ultiboard during "**Transferring to Ultiboard"** section.)

[3] The point of this part of the project is so that you can go through the process of routing the board. Using the autorouter defeats the point so do not do it. We're also not stupid and can distinguish autorouted boards from hand routed boards.



Placing Test Point

Adding Text

In addition to the default labels for the parts that you placed on the board, you will need to place addition text on your board such as pin out labels for the electrode connectors or your information. Note that the specifications require you to at least put your name, your partner's name, and lab section must be clearly marked on your board in the following format:

Line 1: <Your name> + " & " + <Your partner's name>

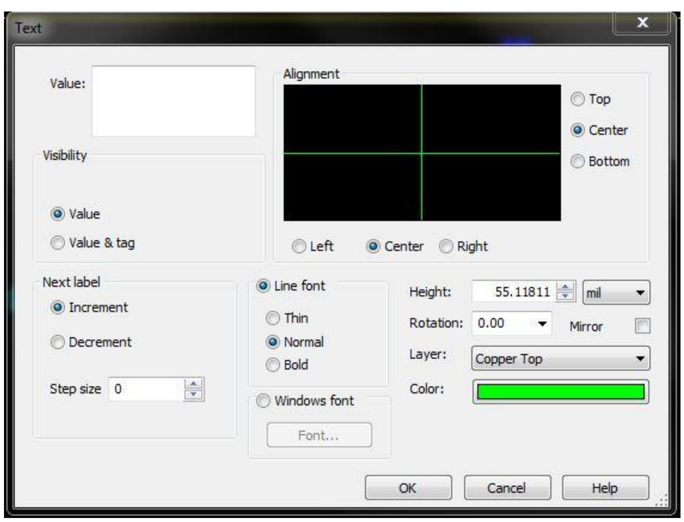
Line 2: "Section: " + <Your lab section day> + <Your lab section hours> + ",TA: " + <Your TA's full name>

Where + is the standard concatenation operator.

For example, Alice and Bob in Monday 8-11am Section with John as a TA should have the following label:

Alice & Bob
Section: Monday 8-11am, TA: John

To place text on your board, go to Place -> Graphic -> Text. A window like in the figure below should appear…



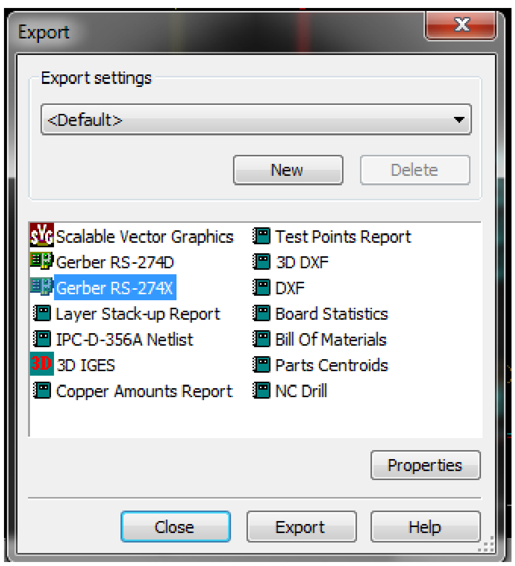
Input the appropriate values and place the text on your circuit layout. Make sure that the text faces in the same direction as the text for your component designators. In addition, make sure that your text is on the top layer of the board and not the bottom.

You can add text if appropriate to other parts of the board, however only add labels where appropriate. Do not flood your board with text.

Exporting to Gerber RS-274X File Format

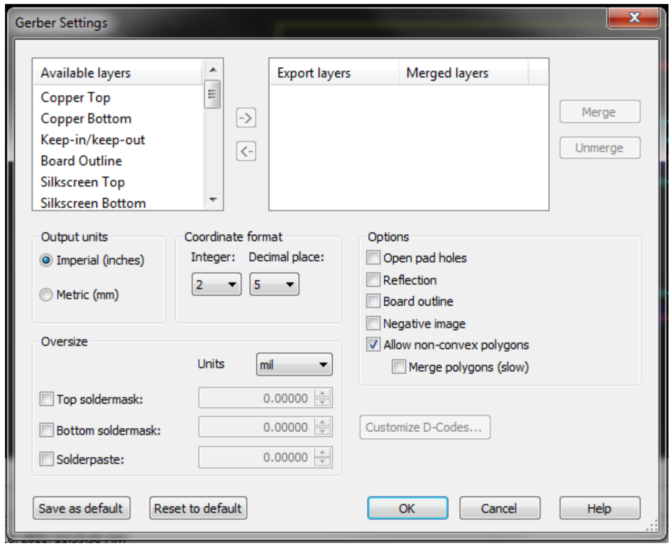
Once you have successfully completed the layout process and routed all of the connections on your board, we need to export the board to a Gerber file. Gerber files are one of the standard types of file formats that fabrication companies take so we will have to conform to their standards.

To export your file to a Gerber file in Ultiboard, select File->Export. A window such as the one shown below should appear. Select Gerber RS-274X and select Export. (Please note that some of the software packages may not have the Gerber RS-274X export option. In that case, please go to EE40 lab computer to do this final export task.)



Export Window

Once you hit Export, you will be prompted to select the layers that you want to export to the Gerber file format as shown below.



Layer Select

You will want to export the following layers into the Gerber file formats…

- Copper Top
- Copper Bottom
- Silkscreen Top
- Silkscreen Bottom
- Board Outline
- Drill
- Drill Symbols
- Solder Mask Top
- Solder Mask Bottom
- Paste Mask top
- Paste Mask Bottom

You will then be prompted to save each layer as a separate Gerber file. Make sure to keep track of these since you will need to submit them for the manufacturing process. You will be required to submit them, as well as your MultiSim file (*.ms11), Ultiboard files (*.ewprj and *.ewnet) in a .zip file. The .zip file name should be in the following format:

“EEG-“ + <your last name> + ”-“ + <your partner’s last name> + “.zip”

Your Assignment

Your assignment is to now implement your design that you have in Multisim and prepare it in Ultiboard. Following the instructions given above, transfer the components from Multisim and route the board in Ultiboard. Do NOT use autoroute or autoplace, they do not adhere to the specifications and it is very easy to tell if you did.

You will be required to submit both the Multisim schematic file and the Ultiboard PCB files.

If at any point in the PCB layout process you are unclear, attend office hours and ask your TAs. The board layout is the final step in the process and once we send your board out for fabrication it either works or it doesn't… Therefore, be very careful when doing your board layout.

A few things you should check before you submit your circuit…

- **Make sure that power traces are larger than regular traces. This means that the traces connecting the V_{cc} and GND nodes must be larger. See the specifications for details.**
- Add test pads between modules in your circuit. That way if you assemble your PCB and it does not work, we can debug and possibly fix your layout. Remember, the easier it is for us to trace your circuit, the better your chances are at receiving partial credit.
- You may want to add unconnected pads in your schematic. In the event that you forget a bypass capacitor, these extra pads will allow you to make any last minute changes once you receive the PCB from fabrication. You can even make a small section of "breadboard" this way if you have extra space.
- Make sure your name and your partner's names are on the PCB or else it'll be impossible to return the correct boards to you after the manufacturing process.
- Check that your PCB satisfies all clearance specifications and DRC checks outlined in the specification
- Read the specifications and make sure your board adheres to it
- Read the specifications again and make sure your board adheres to it

Printed Circuit Board Layout Specifications is shown below again:

- Your layout be no larger than 3 inches wide by 5 inches long
- Your layout may only use circular vias
- Your layout must use IC packages that have the same type of through-hole pads
- Your layout must incorporate test pads at key points in your design
- All components on your printed circuit board must be appropriately labeled with component number (such as "R5" for resistors and "C2" for capacitors).
- Your layout may NOT use surface mounts components except for the voltage regulator if you opt in.
- Your name and your partner's name, and lab section should be clearly marked on your PCB in a legible manner
- **Power traces for supplies and ground must be at least 50 mils wide, other traces must be 15 mils wide**
- Your traces and pads must satisfy a minimum of 20 mil clearance with other components
- Using the Autorouter is prohibited. It also does not corretly autoroute consistently so if you do, you probably will lose points
- When you solder the board, you must solder the sockets before putting in the IC chips to avoid heat damage to the chips

Once you are done, you will be required to Email your zip file to your GSI so that we can prepare them for manufacturing. During your lab section, your GSI will inform you the email address. The deadline for the submission is INSERT DEADLINE HERE

Important: THIS IS YOUR LAST CHANCE TO IMPLEMENT ANY FIXES TO YOUR CIRCUIT. AFTER YOU SUBMIT YOUR CIRCUIT IT WILL BE IMPOSSIBLE TO MODIFY.

Soldering Phase

You should now have a manufactured version of the PCB that you designed and submitted. Make sure that you have received the correct board that corresponds with the Schematic that you designed.

Now, as in Lab 1, we will solder the components to the board. You should already have these components in your kit and it should simply be a matter of stuffing and soldering the components into the right holes.

You may want to brush up on the basic of soldering and refer back to Lab 1. We only have one copy of your board; so if you somehow manage to break or destroy the board, we don't have extras.

As always with soldering, remember to…

- Not burn yourself or your partner… it hurts
- Not touch the leads the you are soldering because metal conducts heat
- Make sure that polarized components are placed in the correct orientation
- Allow time for the joints to cool so that you get good electrical connections
- Not breathe in the solder fumes (solder fumes are not particularly good for you…)
- Avoid splashing molten solder

Once you have soldered all of the components together on your PCB, it is now time to test your PCB for functionality.

We will NOT be providing you electrodes for you to test the EEG on yourself. Do NOT do this. You should first test your EEG using the test signal source that we built.

If you can confirm that your EEG responds correctly to your test signal source, than demonstrate it to your GSI for approval. Please do the measurement and fill in the table below for the whole EEG circuit. Peak-to-peak value of input refers to the real input to the instrumentation amplifier stage, and should be in the range of µV~10s µV depending on the gain of the circuit.

Frequency	Peak-to-peak value of input	Peak-to-peak value of output	Gain/Attenuation Factor	Gain/Attenuation in dB
DC				
1 Hz				
3 Hz				
10 Hz				
30 Hz				
50 Hz				
60 Hz				
70 Hz				
100 Hz				
300 Hz				
1000 Hz				

Your TA Signs Here If Circuit Works

We will be holding a special section which will be announced on bspace for you to test the EEG on your unfortunate professor (we're not joking).

If your EEG is not working as intended, then you need to try and debug, and hopefully fix the problems.

Here are a few suggestions on how to debug your PCB…

- Check electrical connections and make sure that your active circuit elements are receiving power. This can be accomplished by making sure signal values at the test pads are correct.
- Check for missing bypass capacitors or filter capacitors. These may be necessary if you are seeing a muffled or distorted output.
- Check solder joints for cold joints. Remember cold joints can potentially be open circuits that will most likely kill the functionality of your circuit.
- If you need to add a component, you can air wire the components to a spare pad if you created them otherwise you will have to jumper the appropriate nodes to an external breadboard.
- Make sure the coin cell battery holder is polarized and inserted in the correct direction. This can be tested by simply making testing the voltage across the terminals.

If you need to add another IC chip to your PCB, please note that this is almost impossible to do cleanly. If you need to add an IC chip or other components that is not easy to add, you may need to consider wiring your PCB to an external breadboard.

If you absolutely cannot determine what is wrong with your PCB, ask your one of the GSIs and we will do our best to help you. Please note that if you did not follow our recommendations and made your board impossible to trace, we cannot guarantee that we can fix your board.

If you are unable to fix or get a semi-functional board, write an explanation identifying what the problem is in your design and how this issue could be addressed. Test the circuit as much as you can from your test pads, and write down your measurement results. In addition, revise your schematic and PCB layout, annotating any changes you would have made and what these changes would fix your circuit.

Final Project Submissions

You will be required to demonstrate your working, semi-functional, or not working circuit to your GSI for evaluation. After you show your GSI your circuit, you may take it home with you and keep it.

In addition to demonstrating your working circuit to your GSI, you will be required to submit a final lab report. In this submission, you will be required to attach an original print out of your initial design, a revised schematic for your design, and the PCB layout of your design.

Your final lab write up is due during your lab session on the week of April 30thth to your lab GSI.

Congratulations, you have finished the laboratory component of EE40!

We hope you enjoyed/survived this final project.

GL HF on the final ☺!

--Your Teaching Staff

Extra Credit

To make things a little more interesting, this semester we're going to offer a little bit of extra credit for students who want to go beyond the scope of the project.