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

Score: ___/100

Electroencephalograph (EEG)

Final Project Part 2: PCB Layout

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:

I.   Project Deadlines

A reminder that the deadlines for this project are as follows:

- Schematic Design and Simulation: **Due Nov. 7-12** (on day of your normal lab)
- Printed Circuit Board Layout: **Due Nov. 14-19** (on day of your normal lab)
- Final Soldered Board, Demonstration, and Lab Write Up: **Due December 3rd**

You must meet all deadlines for this project ON TIME. If you miss the deadline for any of these items you may not finish the project on time, and if you miss the deadline for the Printed Circuit Board Layout you may fail the project portion of the class.

II.   Multisim Component Selection

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:

_____

[1] If you are using the SMT voltage regulator, then make sure you choose the SMT footprint for the simulation
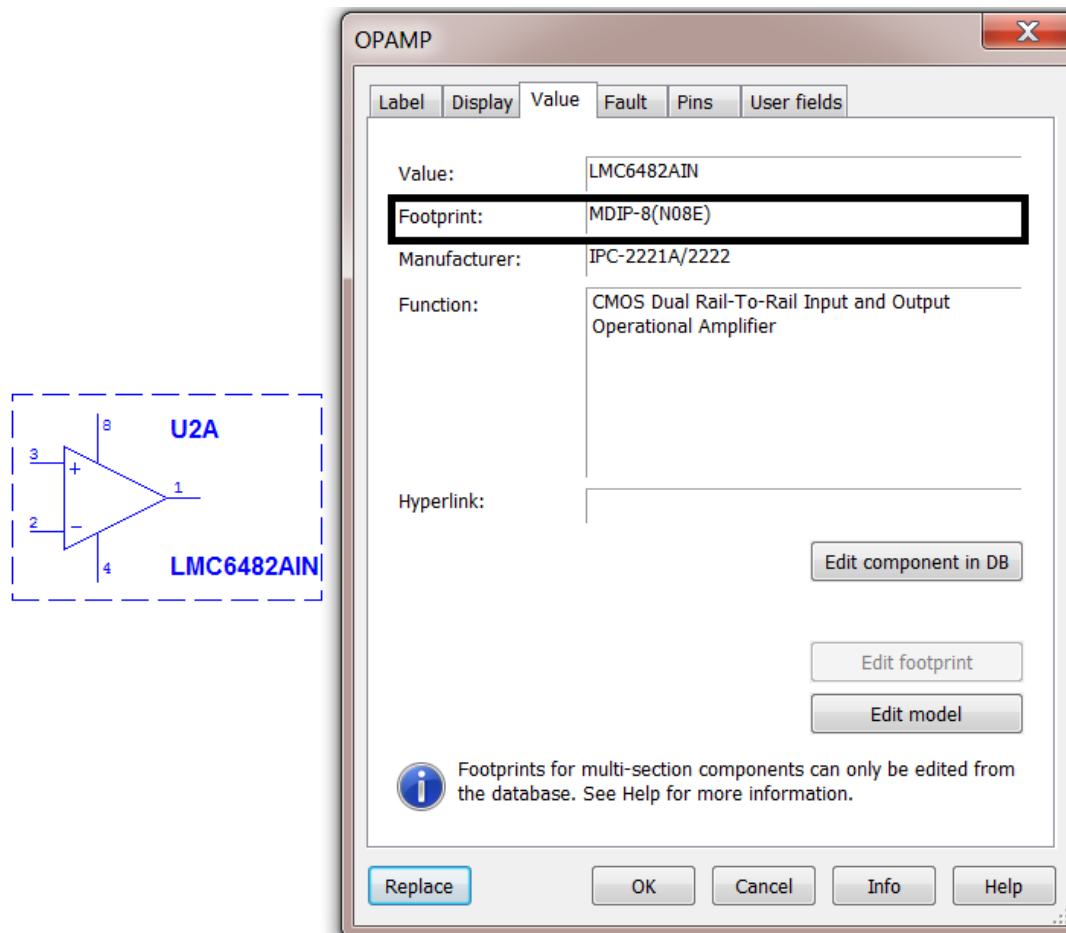
3

Figure 1 - Multisim Component for Project

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 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.
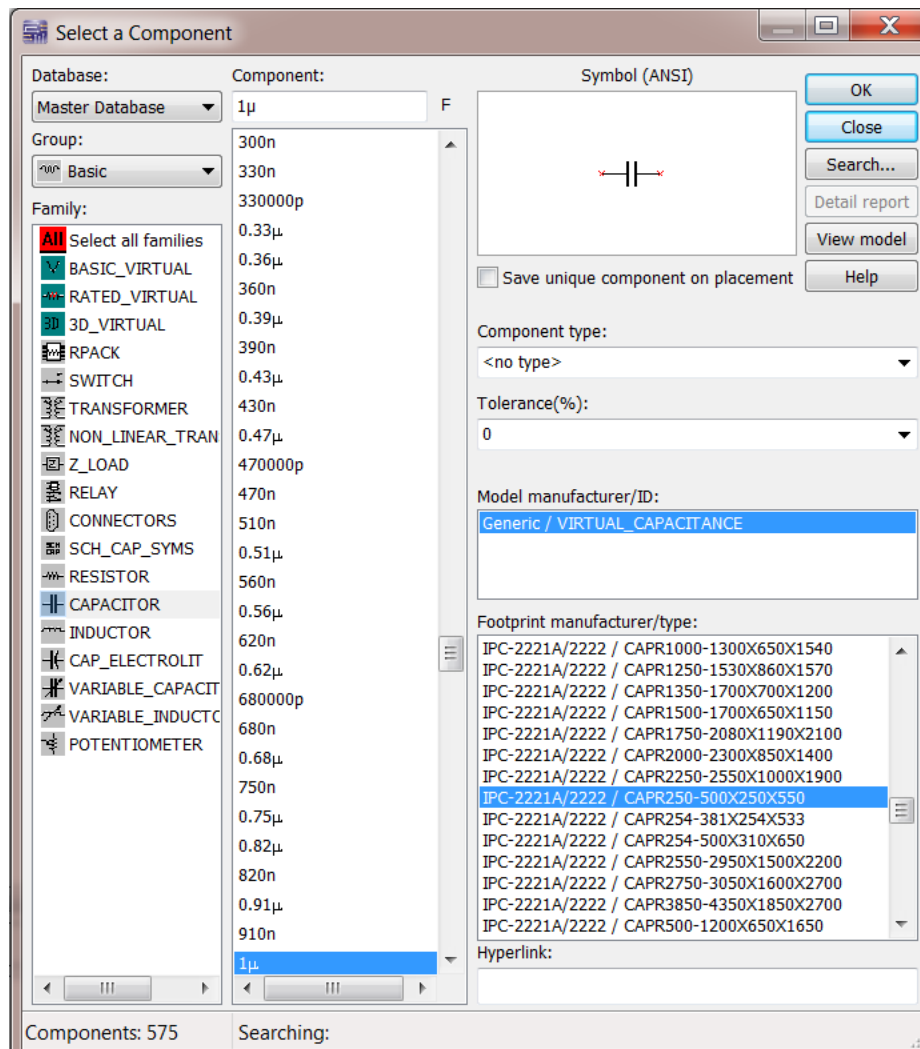

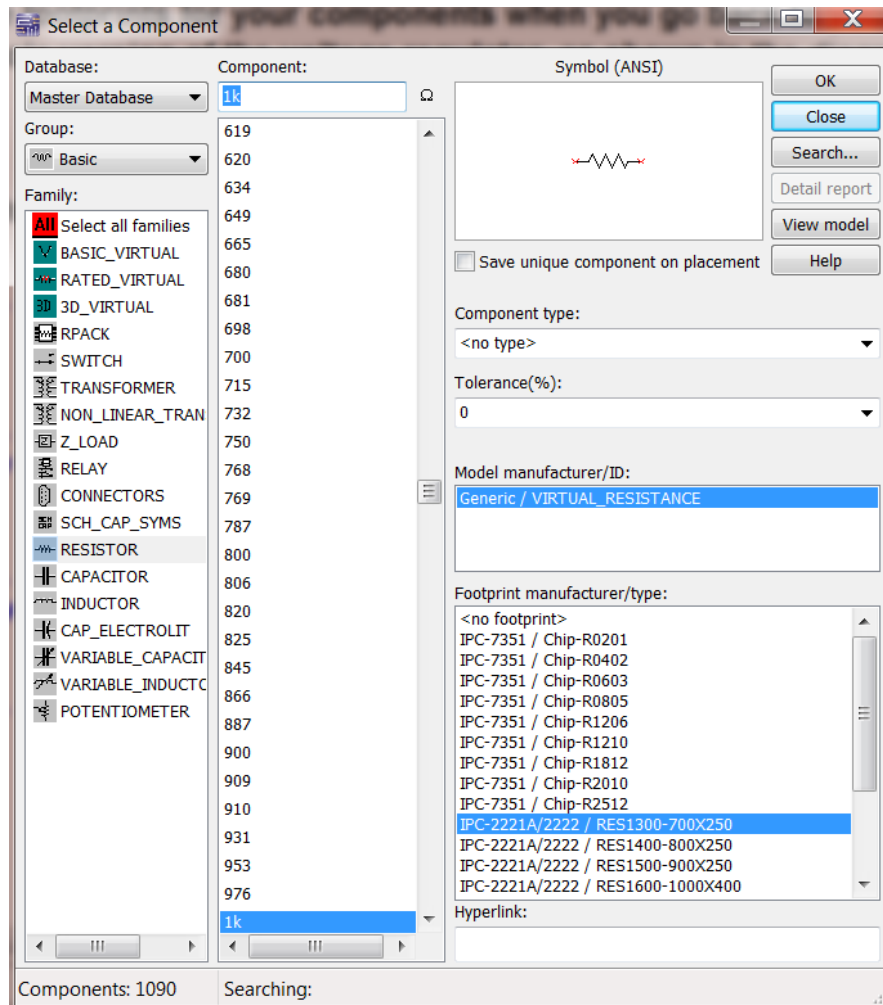**Figure 2 - Capacitor for use in project**

Figure 3 - Resistor for use in project

3. Now we move onto the potentiometers and the voltage regulator. The footprints for these components, unfortunately, are not standard, 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.
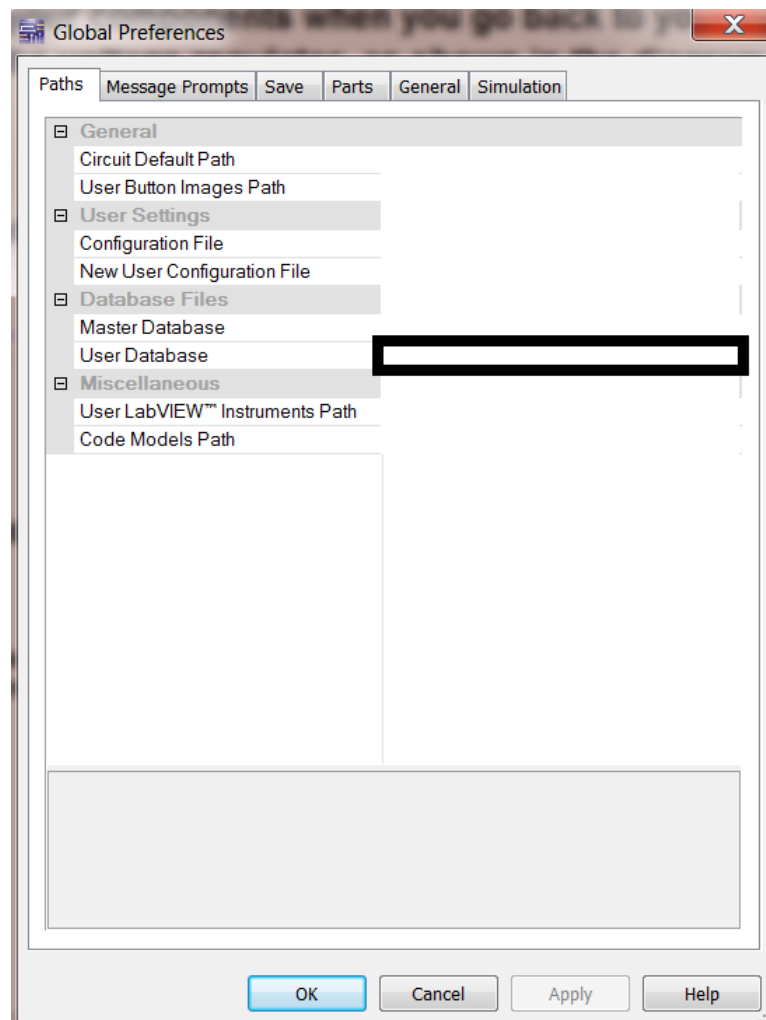

Figure 4 - Modifiying User Database in Multisim

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".

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.
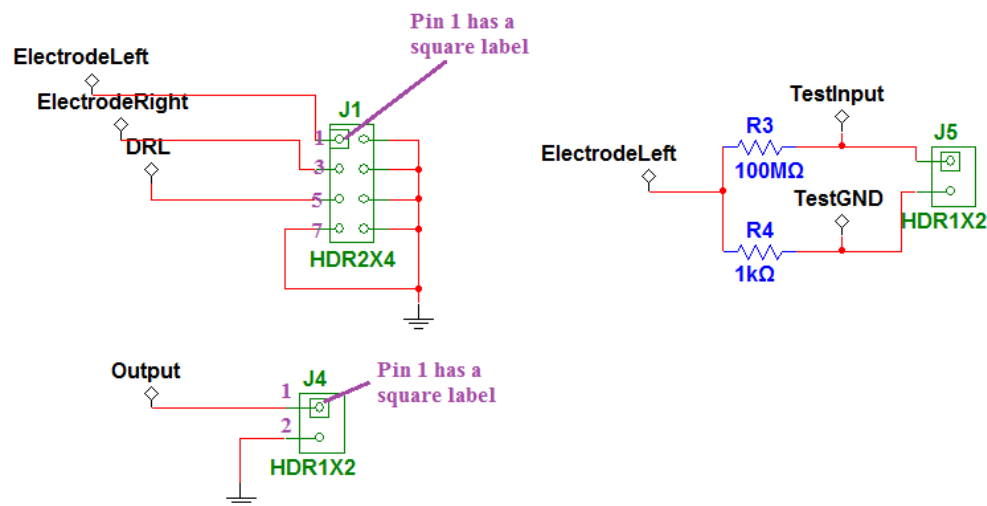

Figure 5 - Test ouputs for final project

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.

III.   Ultiboard Layout

A.   Transferring components 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.
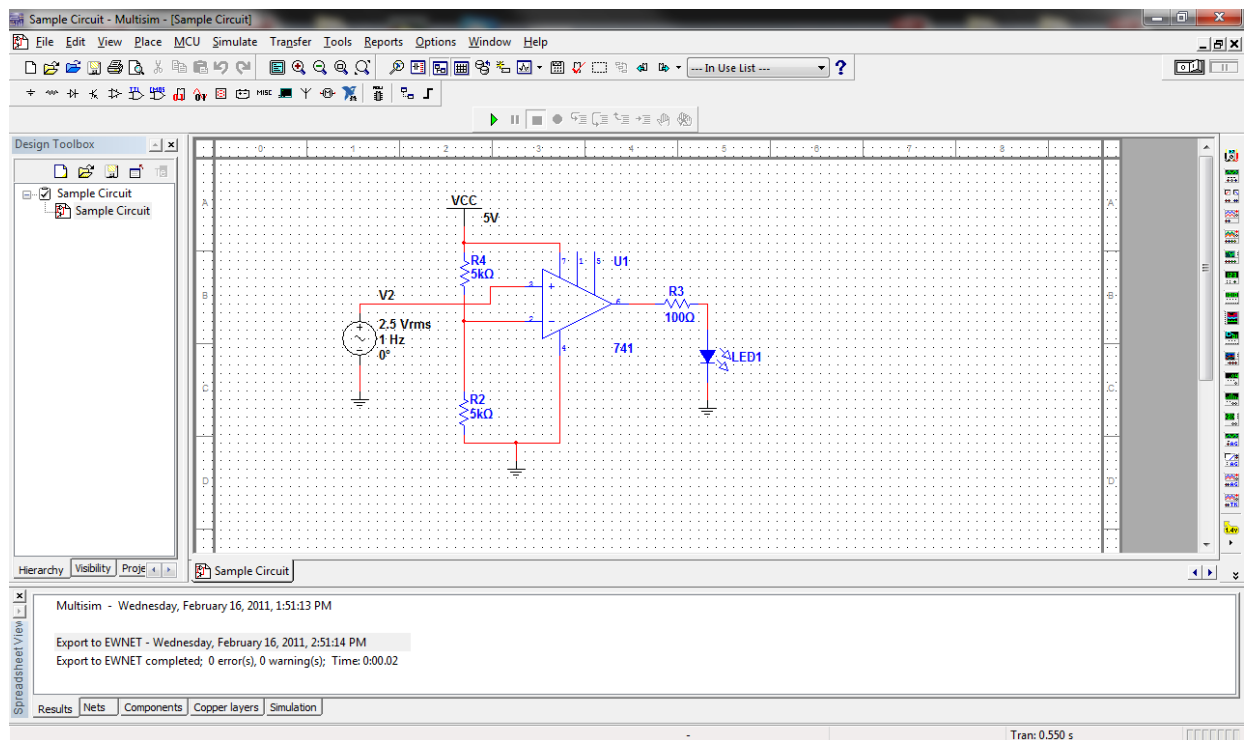

Figure 6 - Sample circuit in Multisim

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.

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.

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.
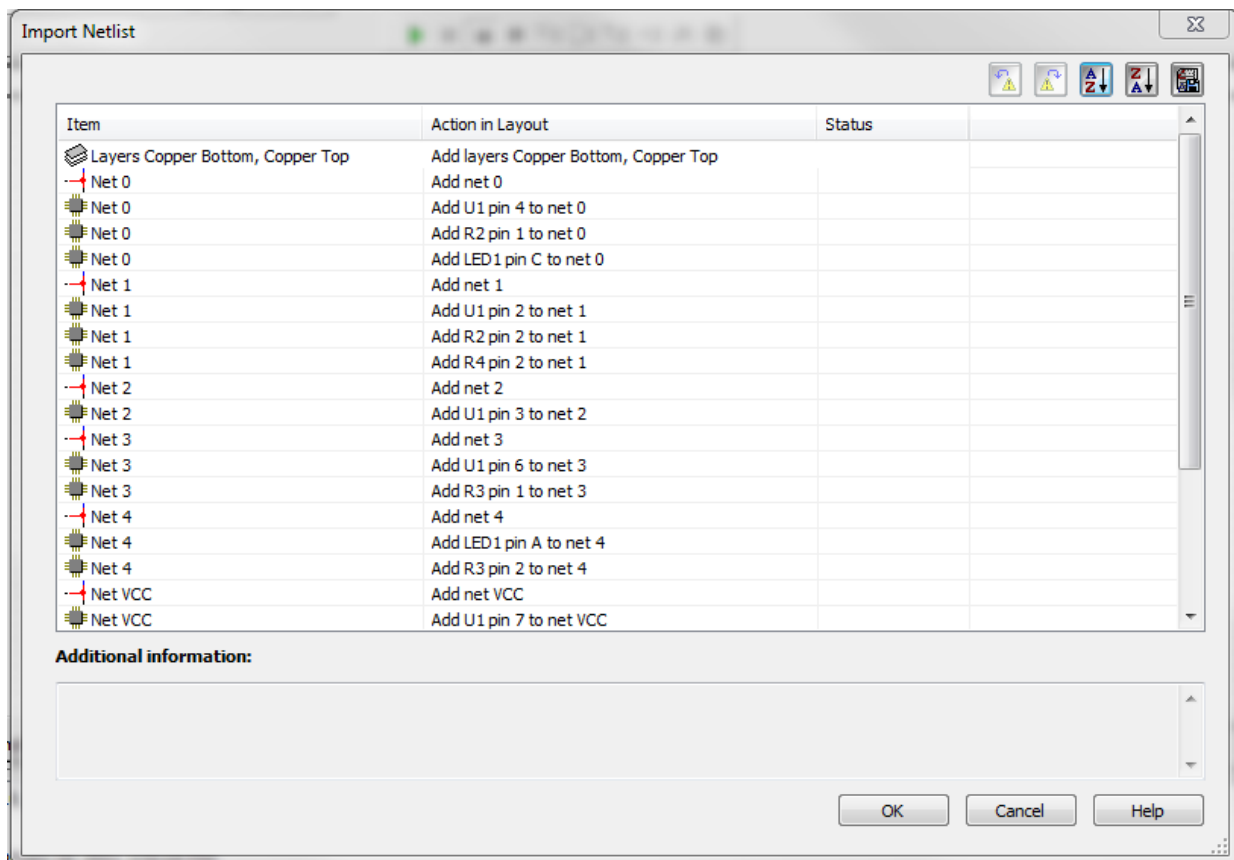

Figure 7 - Import Netlist Window

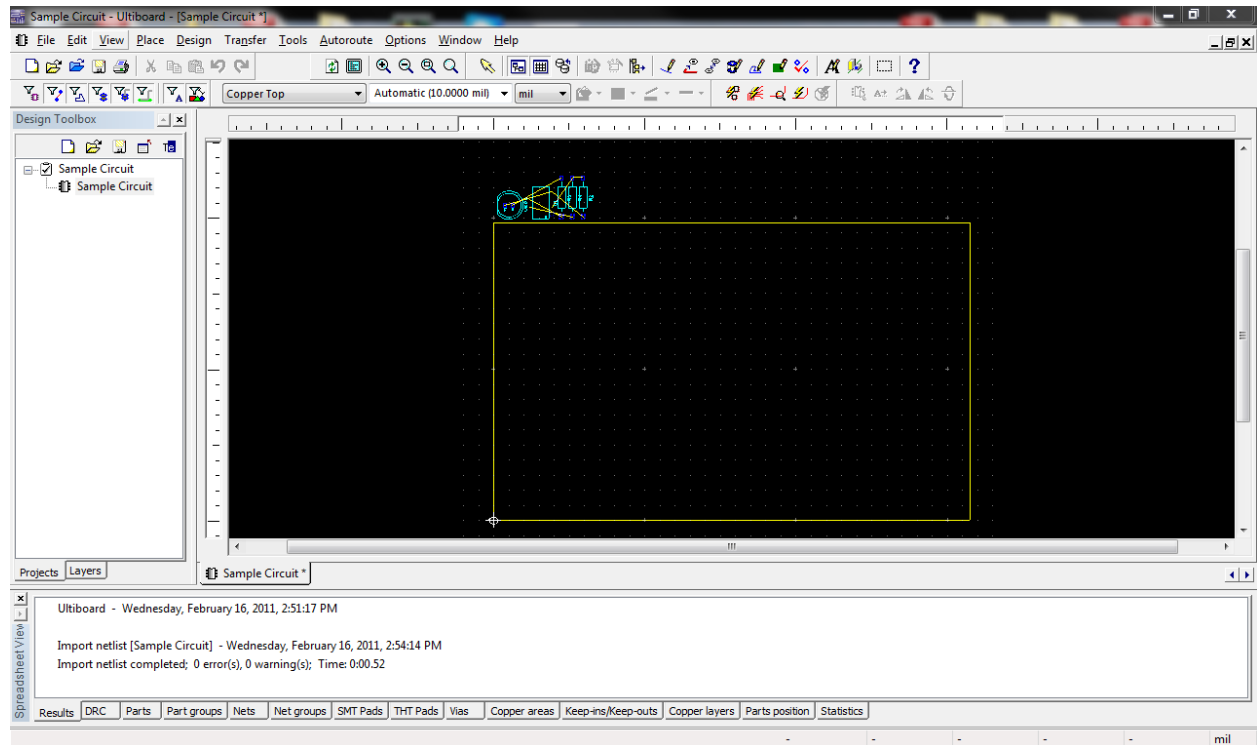Click OK and you should get a window that looks similar to the one shown below.

10

Figure 8 - Ultiboard after importing components from Multisim

B.   Initializing Ultiboard

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 (note that it can be somewhat difficult to exactly click on the yellow rectangle).

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.
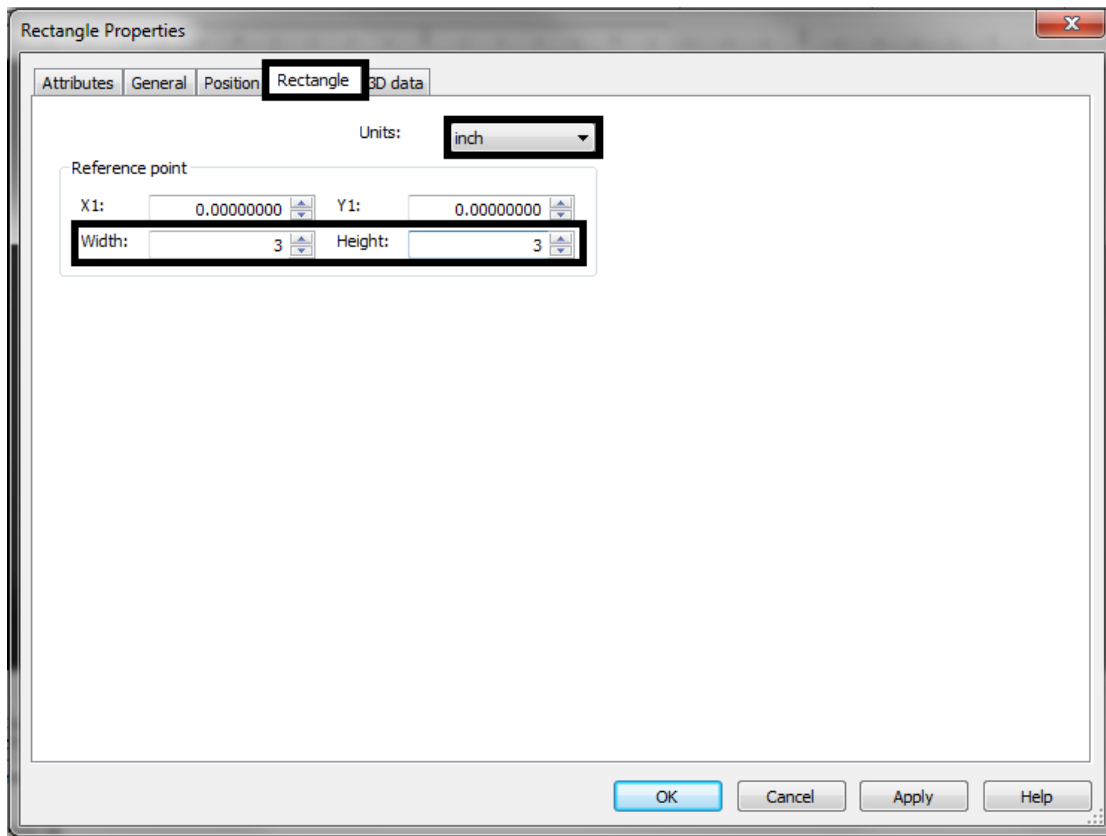
Figure 9 - Rectangle Properties Window

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.
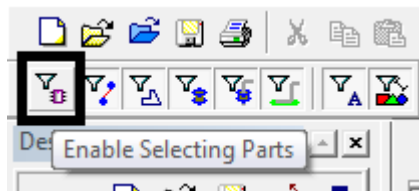

Figure 10 - Enable Selecting Parts

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:
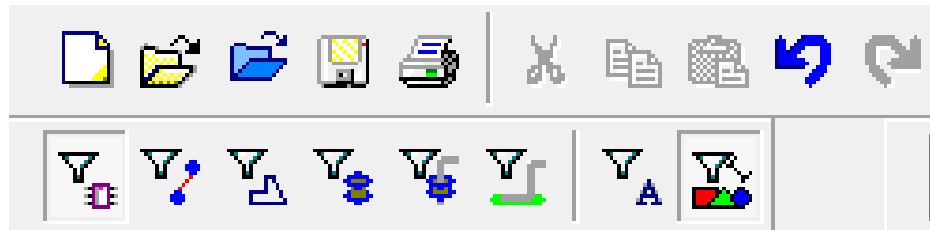

Figure 11 - Toolbar settings for project

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

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.

C.   Component Layout on PCB

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.

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.
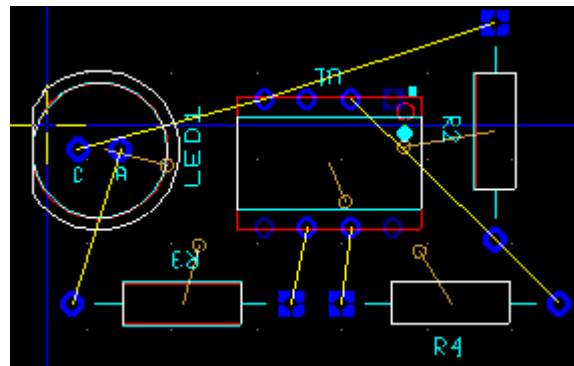

Figure 12 - Components on Sample Circuit

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.

---

[2] Note that the examples in the next couple of figures do obey this practice
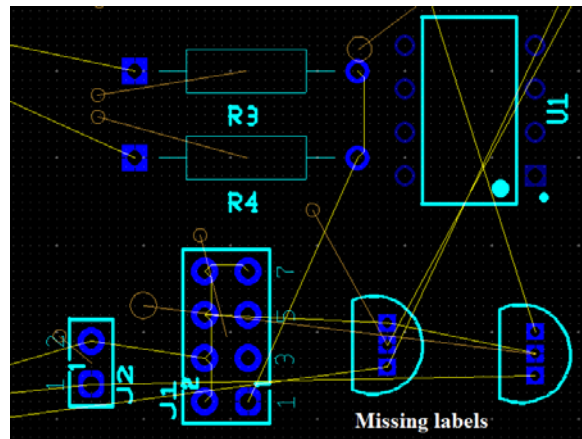
Figure 13 - Missing labels on voltage regulators

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".
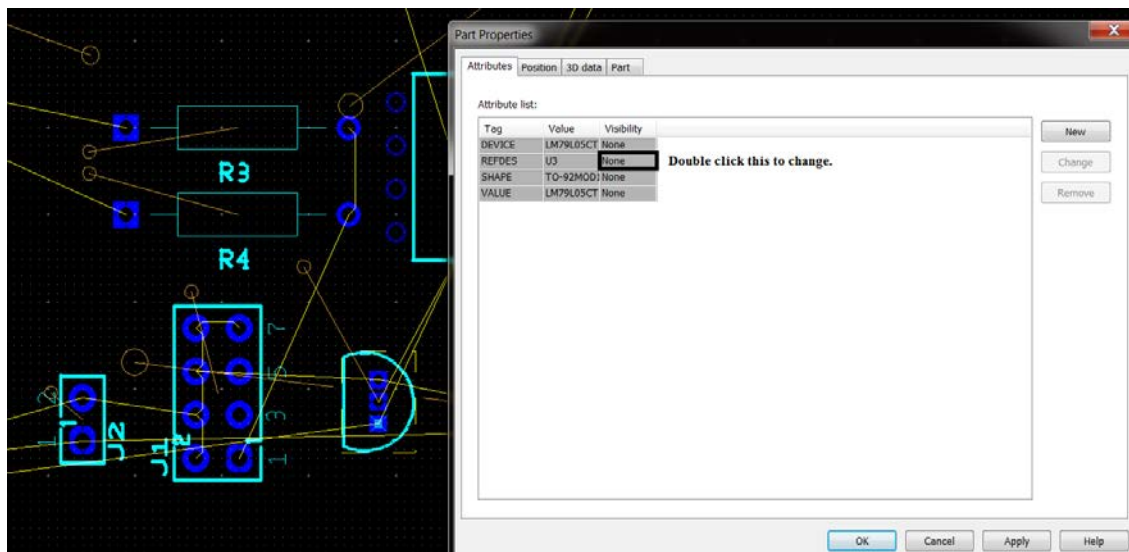

Figure 14 - Adding labels to components

D.   Routing Traces on the Board

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

- BEFORE you actually route your wires, please set up the design rules. In Ultiboard, click "Options", and select "PCB properties". Under the tab "Design rules", you should be able to set your design rules, such as the trace width, clearance etc. Please follow the figure below for your setting. Make sure the units is "mil".
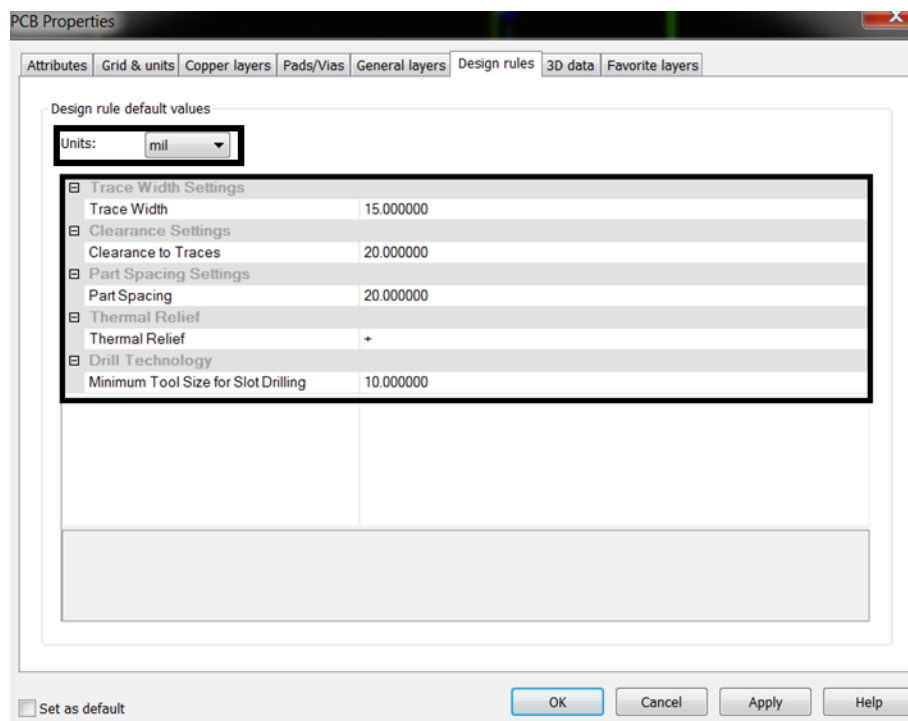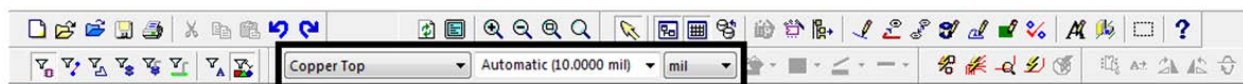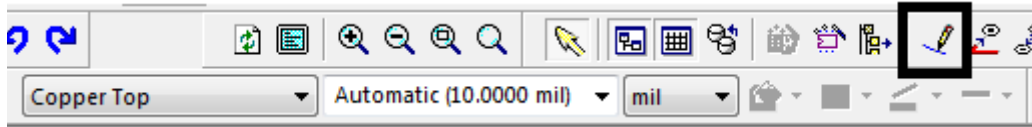

Figure 15 - Design rules menu

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.
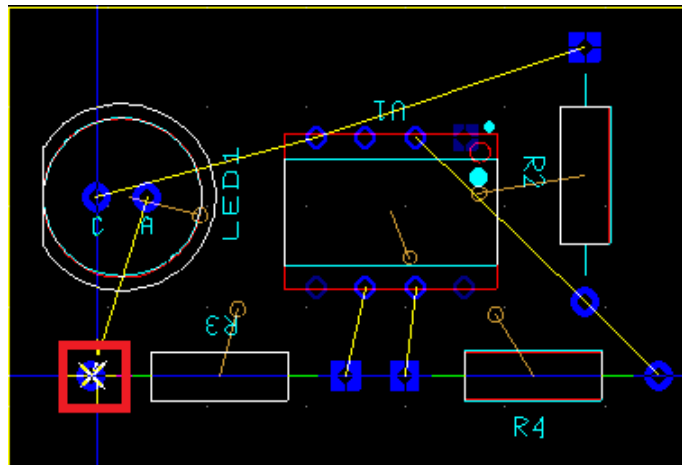

Figure 16 - Place Wire Tool Locks Onto Pad

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.
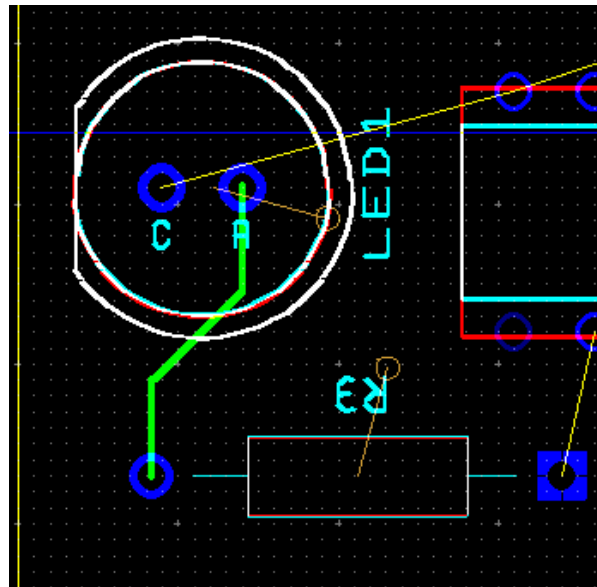
Figure 17 - Routed Trace

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.
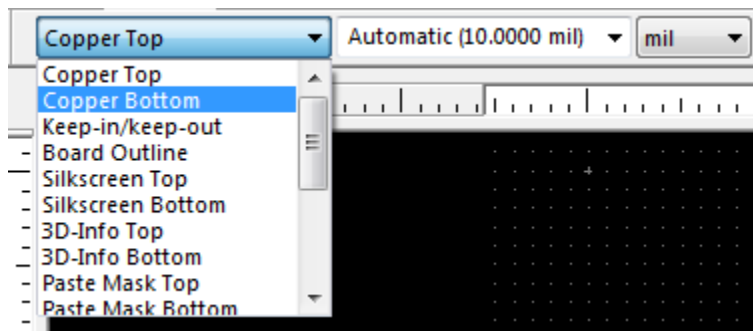

Figure 18 - Select Copper Bottom Layer

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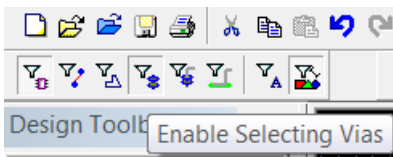
There are several more important considerations that we will want to implement in our design. In all PCB layouts, the maximum angle that a trace can bend is 45 degrees. This rules out all right angle bends, therefore, you will need to ensure that all traces on your PCB obey this rule.

Good Trace

BAD Trace
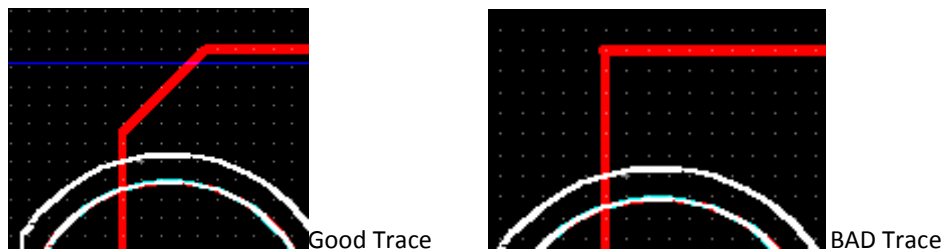
Another important consideration is the practice of biasing the board. Biasing the board means that we try and route all connections going up or down on one side of the board, and route the rest of the connections going left or right on the other side. This practice allows us to introduce a degree of 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]

- When you finish all the routing, please do a DRC (design rule check) and Connectivity Check. There should not be any error. If error shows up, you either violate the design rule, or left some wires unrouted. Please fix these errors before you submit your zip file to your GSI.
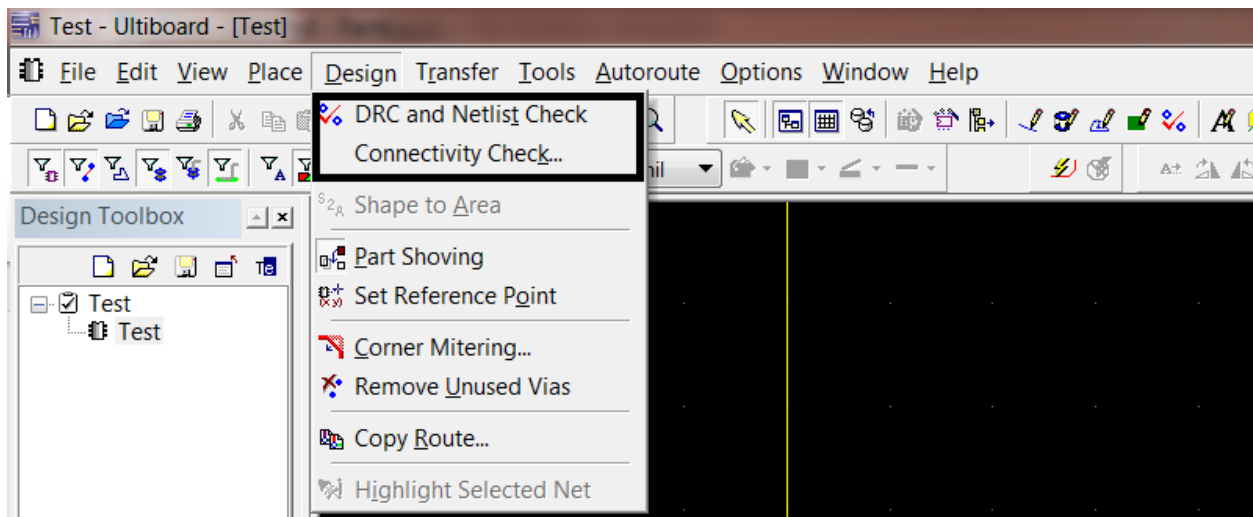
Figure 19 - DRC Selection

E.   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

---

[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.

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.

Figure 20 - 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.

F.   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.)
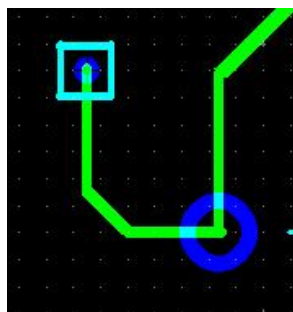

Figure 21 - Placing Test Point

G.   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>

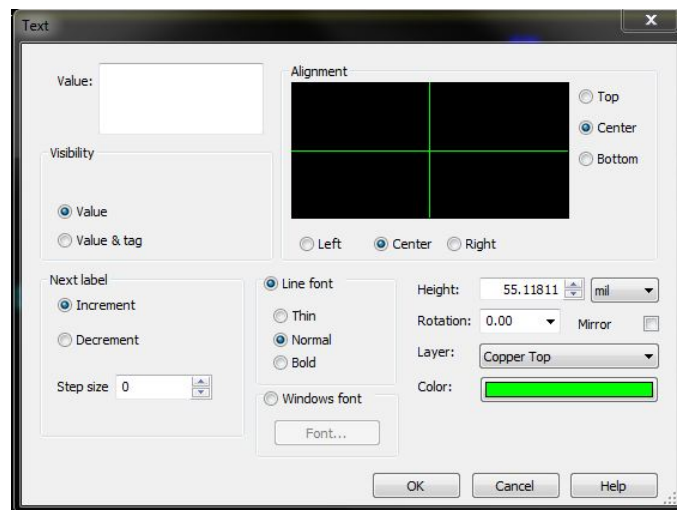Line 2: "Section: " + <Your lab section day> + <Your lab section hours> + ",TA: " + <Your TA's full name>

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.

H.   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.)
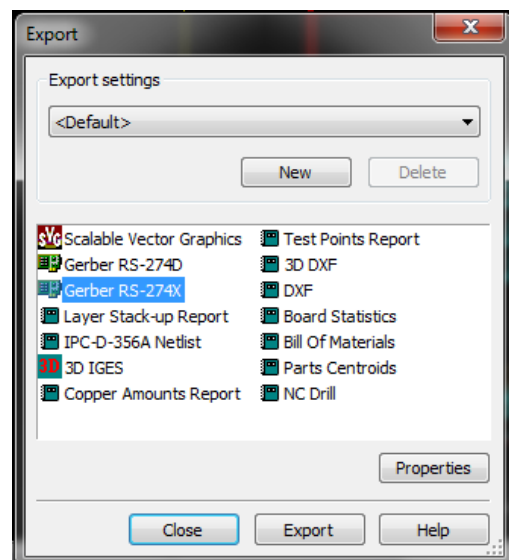

Figure 22- Export Window

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.
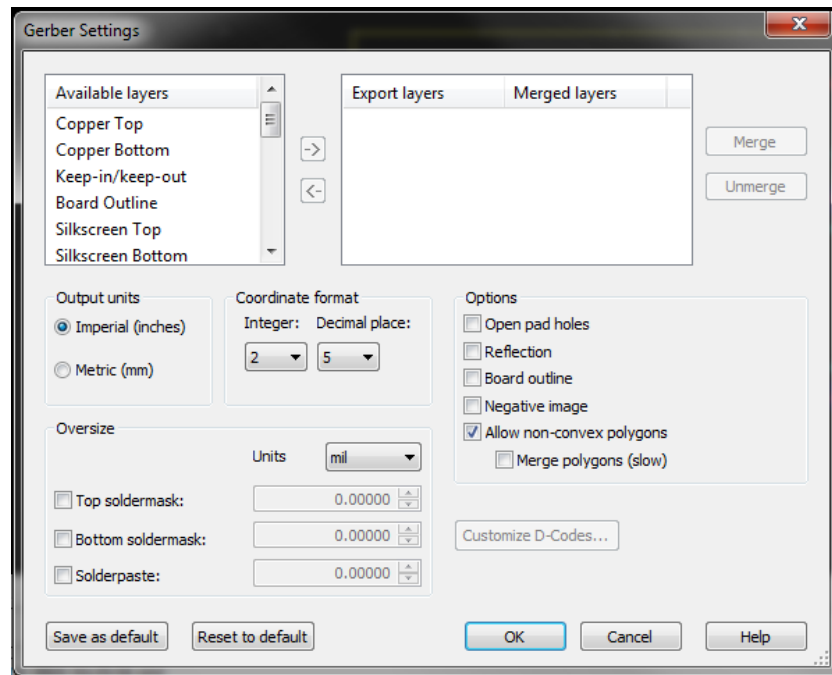
Figure 23 - Layer Select

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"

IV.   PCB Specifications

Every PCB manufacturing process has certain design specifications that need to be met for the final manufactured PCB to function correctly. These specifications are listed below for the project. IF YOU DO NOT ADHERE TO THESE SPECIFICAITONS, YOU WILL LOSE POINTS AND YOUR PCB MAY NOT FUNCTION CORRECTLY. Note: 1 mil = 1/1000 inch.

-   Your layout be no larger than 5 inches width by 3 inches height
-   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 (at the output of each circuit module)
-   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
-   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 at least **15 mils** wide.
-   Your traces and pads must satisfy a minimum of **20 mils** clearance with other components, and at least **50 mils** from the edge of the board
-   Using the Autorouter is prohibited. It also does not correctly autoroute consistently so if you do, you probably will lose points

V.   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

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.

TA Signs here when completed Gerber files are submitted: