EET 222  
Circuit Modeling Project

**Objective**

Often times we are given unknown circuits and asked to determine functionality or troubleshoot a problem. The task at hand may involve creating a schematic from the built circuit or populated board. It may require us to verify that the circuit that was built matches the designed schematic.

In all cases we need to probe around on the circuit board to determine the actual connections. We also need to have a way of converting this back into a schematic. This will allow us to compare the board's schematic to a desired schematic. It may also allow us to confirm that we have the correct schematic so that we may perform simulation to aid in the troubleshooting.

This type of task is very meticulous and requires extreme attention to detail. Today’s boards are not as simple as they used to be. Many of them contain multiple layers of routing or routing that cannot be traced by eyesight. This requires you to develop a system for “ohming out” the connections. This method requires the use of the DMM, usually in continuity mode. This method also requires you to document and keep track of all connections. Elements are often connected to more than one other element. A good method will allow the tester to discover all the connections between elements. Leaving out even one, can result in a non-functioning schematic, causing the complete task to be repeated.

**Task**

You will be given a phase tripler board. It is known that this board is powered by two power supplies, (+12V and -12V). The input of the board is a 12Vpp 60Hz sine wave. The board is supposed to produce three outputs that are all 120º out of phase of each other.

Your task is to probe this circuit board and create a functioning schematic so that we may simulate this design. You will need to use the DMM to verify connectivity between components. You should use the same board each time. You will have to return the boards to the lab office when you are not in lab. We do not have enough boards for everyone to take one home.

Once you have your schematic figured out. Simulate it in LTspice. Compare the results of this simulation to what the board actually produces when run. If your schematic is correct then the results should be the same.

You should explain your methodology for testing and finding the proper schematic. You will need graphical data from the oscilloscope and Ltspice to prove that the two board and your Ltspice schematic are identical. You will need to analyze this data and explain how the two graphs are the same. In this way you are proving that your LTspice schematic matches the actual built schematic.

**Additional Requirements**

To make your schematic easy to read:
- Use off-page connections in LTspice for your Vcc and VEE. Connections are made in LTspice automatically by labeling nodes with the same label. This happens with or without a visible “wire” on the schematic. These are especially useful for commonly used nodes, such as ground and your power rails.
- Part of your report grade will be based on the readability of the schematic.
EET 222  Circuit Modeling Project

- Put the 3 op-amps outputs on the right, so signals flow from left to right.
- You should not have any intersecting connections that are not connected.
- Verify your board works before starting the lab. See the instructions below under “Test and Calibrate”.
- Record the phase tripler board number, so that you can continue to work on the same board in future labs.
- Put your board number on the cover sheet of your report.
- Before starting make a plan for how you will trace all of the connections. It may help to create a BOM (Bill of Materials) or parts list before starting to actually trace the circuit.
- You must use the DMM to verify connectivity. The settings of the POTs may cause nodes to short together when they aren’t really shorted together.
- You should be able to trace the entire board and have most of the schematic done in week 1. Week 2 should be spent matching spice to your schematic.
- Make sure you record all possible data in your lab notebooks. It is always better to have more data than you need. If you are missing data you will have to get back into lab to retake the measurements.

Hints and Tips

- The op-amps on the board are all L272M. You will need to use the data sheet for this op-amp to find its pinout.
- In LTspice use UniversalOpamp2 to represent the L272M
- For testing and simulation purposes, adjust all pots to their maximum settings.
- Think about how you will guarantee the pots are at maximum setting when you actually test the board.
- Unscrewing the board from the posts will allow you easier access to the bottom of the board.

Test and Calibrate

1) Connect a function generator to the input and set it to produce a 12Vpp, 60Hz since wave.
2) Connect a 1k resistor between each output and ground.
3) Connect the +/- 12V DC power inputs to your power supply.
4) Connect the oscilloscope channel 1 to output P000. The resulting waveform should be exactly 12Vpp at 60Hz. Set the trigger to channel 1.
5) Connect channel 2 of the scope to output P240. Set the scope time base to 1ms per division. Center both traces. Adjust potentiometer P240 so that the output P240’s positive-going zero crossing is exactly 5.56ms after output P000’s positive-going zero crossing.
6) Adjust potentiometer A240 so that output P240’s amplitude is exactly 12Vpp.
7) Connect oscilloscope channel 2 to output P120. Set the scope time base to 2.5ms (or 2.0ms) per division. Center both traces. Adjust potentiometer P120 so that the output P120’s positive-going zero crossing is exactly 5.56ms after that of the P000’s output.
8) Adjust potentiometer A120 so that P120’s output amplitude is exactly 12Vpp. At this point, all 3 outputs should be 12Vpp and mutually 120 degrees apart.