

# A Student Guide to Introduce Multisim, LTSpice, and Mathcad for ECE 3043 at the Georgia Institute of Technology



Archit R Jain  
School of Electrical and Computer  
Engineering Student

# Table of Contents

1 Introduction.....	3
2 Backgrounds.....	4
2.1 Multisim.....	4
2.2 LTSpice.....	4
2.3 Mathcad.....	5
3 Multisim.....	6
3.1 Hints.....	12
4 LTSpice.....	13
5 Mathcad.....	17
6 References.....	19

# 1 Introduction

---

As a ECE student there are many challenging classes filled with new material that we have to take. These classes introduce anything from new type of circuits to new software and new programming languages. This guide is to help students learn new lab softwares. So in ECE 3043 students have to learn how to use Multisim, LTSpice, and Mathcad, in order to complete all of the future homeworks assignments. As a current student in ECE 3043 I know how much of a hard time I had doing the first homework, because I did not know how to use these softwares.

The purpose of this user manual is to teach current ECE 3043 students how to use Multisim, LTSpice, and Mathcad. Additionally there will be hints given as to their future homeworks. The first thing discussed in this manual will be background for three software, than go through step-by-step instruction on how to use this three softwares.

## 2 Backgrounds

---

### 2.1 What is Multisim

NI Multisim is an electronic schematic and simulation program that is part of circuit design class, along with NI Ultiboard. Multisim is one of the few circuit design programs to employ the original Berkeley SPICE based software simulation [1]. Multisim is widely used in academia and industry for circuit's education, electronic schematic design and SPICE simulation [1]. The latest version of Multisim enriches powerful simulation technology with the ability to analyze analog, digital, and power electronic across education.



**Figure 1.** Multisim logo [1]

### 2.2 What is LTSpice

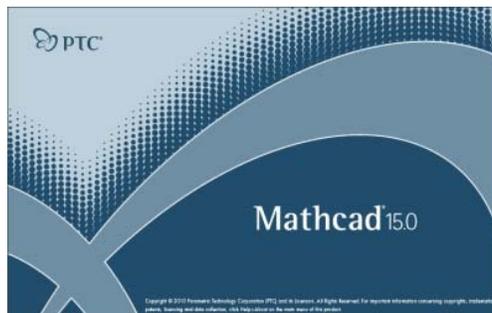
LTSpice is high performance SPICE simulator, schematic capture and waveform viewer with enhancements and models for easing the simulation of switching regulators [2]. LTSpice is node-unlimited and 3<sup>rd</sup> party models can be imported. Circuit simulations based on transient, AC, noise and DC analysis can be plotted as well as Fourier analysis [2]. LTSpice IV is used in many fields including radio frequency electronics, power electronics, digital electronics, and other disciplines.



**Figure 2.** LTSpice logo [2]

## 2.3 What is Mathcad

PTC Mathcad is the industry standard software for solving, analyzing, and sharing your most vital engineering calculations [3]. It has mathematical notation, unit's intelligence, and powerful calculation capabilities, presented within an easy-to-use interface, allows engineers and design teams to capture [3]. Mathcad also integrates these human-readable, live calculations with plots, graphs, text, and images into a single presented document.

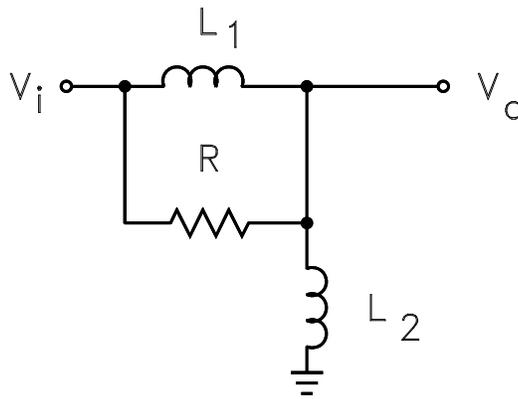


**Figure 3.** Mathcad logo [3]

### 3 Multisim

---

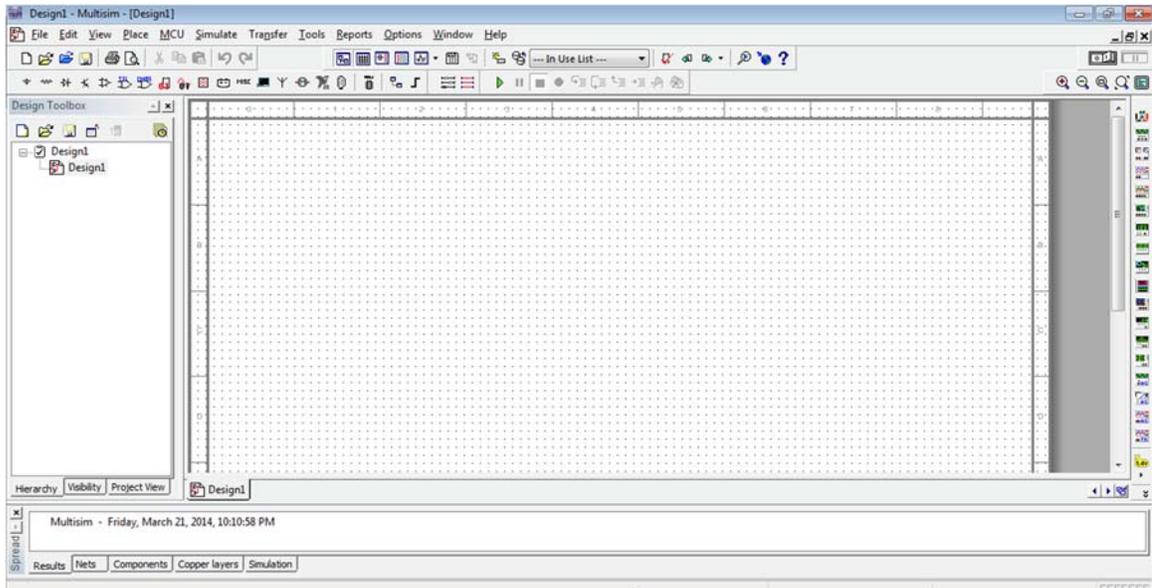
As stated earlier Multisim is an electronic schematic and simulation program. The best way to learn this program is to go through a step-by-step instruction on how to draw the following circuit (Figure 4) on Multisim.



**Figure 4**

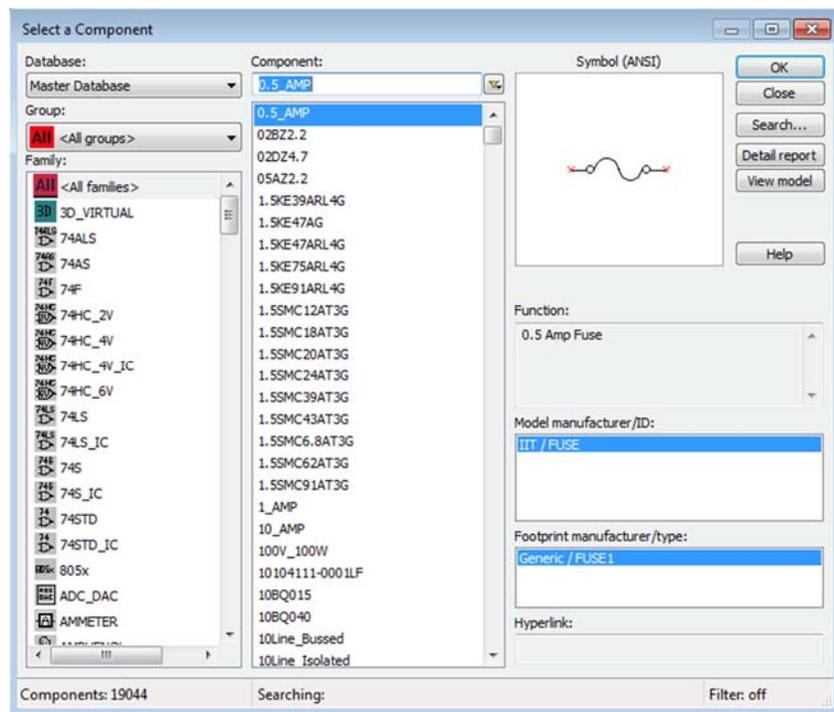
In Multisim, all the components are required to have value for the circuit before the schematic. If the values are not given on the homework sheet then the professor will provide equations in class for it to be solved for the specific values. But for this specific circuit the values that will be using are  $R = 4.3\text{k}\Omega$ ,  $L_1 = 7.3\text{ H}$ , and  $L_2 = 3\text{ mH}$ .

1. Begin by opening Multisim software from start menu. For the logo refer back to Figure 1. Once the software is opened it should look like Figure 5.



**Figure 5:** Starting screen of Multisim

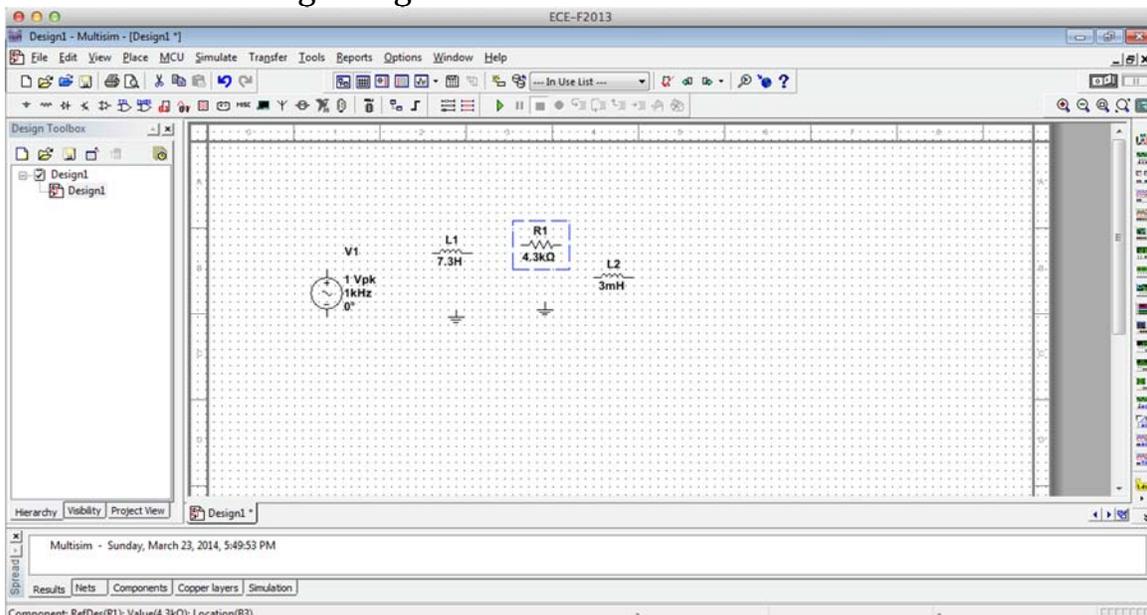
2. After the program is opened gather all components from the folder. To do so click on **PLACE** on the menu bar then click on **COMPONENT** and the screen should look like Figure 6.



**Figure 6:** Component Folder

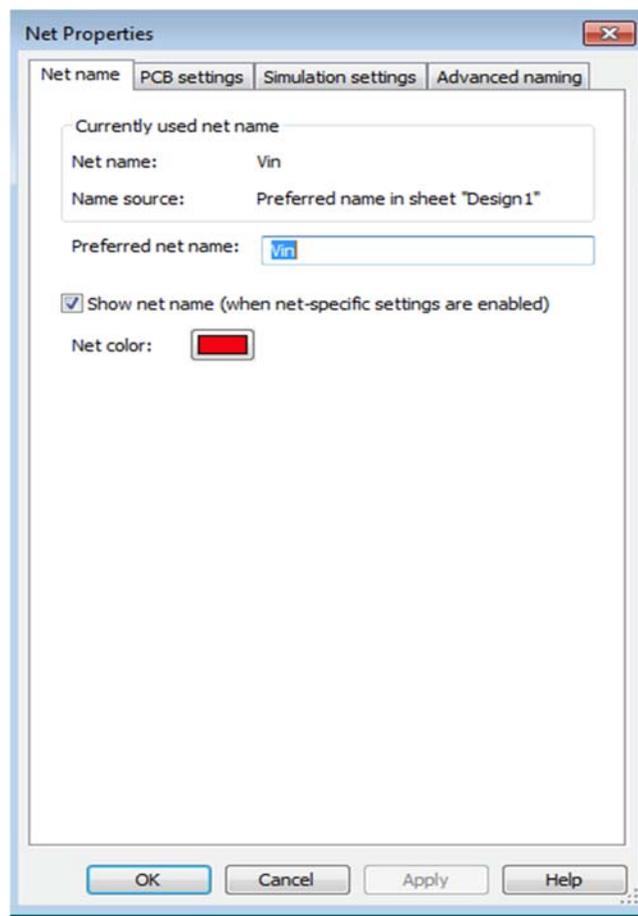
Now that the screen is up, click on the drop down menu under **GROUP**, click on **SOURCES**, click on **POWER SOURCES** then **GROUND** and click **OK**.

- Once all the grounds symbols are placed on the schematic, navigate to the same screen (Figure 6) but this time under **GROUP** and **SOURCES**, click on **SIGNAL VOLTAGE SOURCE**. In this class the only source that will be used is AC source, so therefore this class uses **AC VOLTAGE** under **COMPONENT**. {Later in the semester Piecewise Linear source is going to be required in order to solve the homework problem and also add in initial voltage.}
- Navigate to the same screen (Figure 6) this time under **GROUP** click on **BASIC** and under the **GROUP** there something called **FAMILY**. Under **FAMILY** there is a different type of components such as resistors, capacitors, and inductors, etc. But for this specific circuit its required to use inductors and resistor, and should look like in Figure 4. So in **FAMILY** click on **RESISTOR (R)** and under **COMPONENT** write the value of 4.3k and click **OK**. Do the same thing for the two other inductors (But in the homework there will have different values).
- After the completion of step 5, your window in Multisim should look like the following in Figure 7.

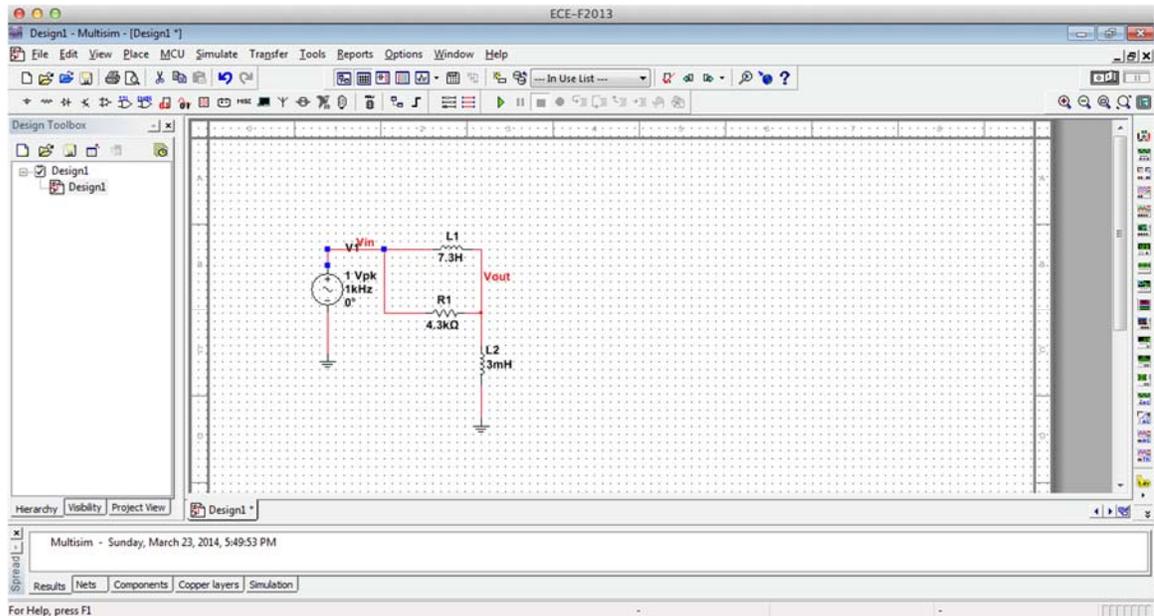


**Figure 7:** With all the components

6. Now lets rearrange the components to match Figure 4. Also inductor (L2) needs to be rotated. To rotate the component **HIGHLIGHT** the element and **RIGHT CLICK** and select **ROTATE 90° CLOCKWISE. CLICK** on it. This should rotate the component. After that, just connect components like it was required in **QUARTUS** using lines.
7. Once all the components have been rearranged to match Figure 4, the schematic needs labeling of Vin and Vout. To do so, **RIGHT CLICK** on the line between the **SOURCE** and the **NODE** and click on **PROPERTIES**, then under the tab **NET NAME, PREFERRED NET NAME**. Select write Vin, and **CHECK** the box for **SHOW NET NAME** so it can appear on your schematic.
8. Repeat the step for Vout between the inductor and resistor line, and your properties box should look like Figure 8 and your drawing should look like Figure 9.

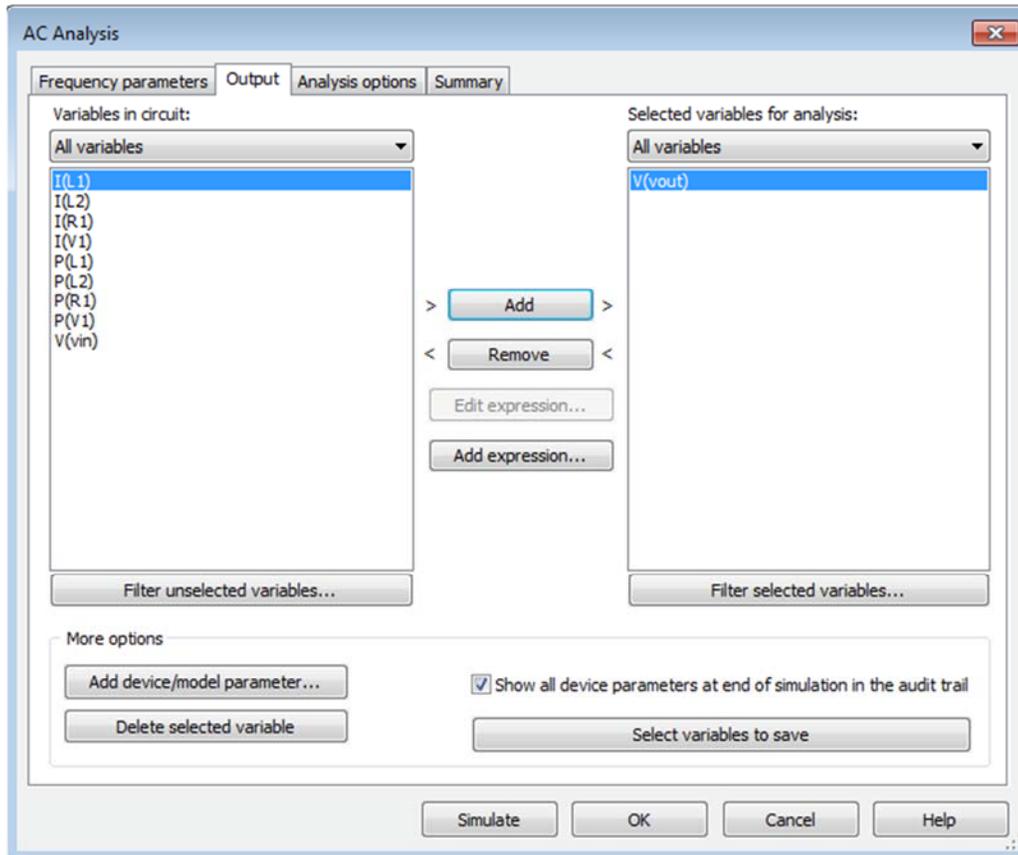


**Figure 8:** Properties box



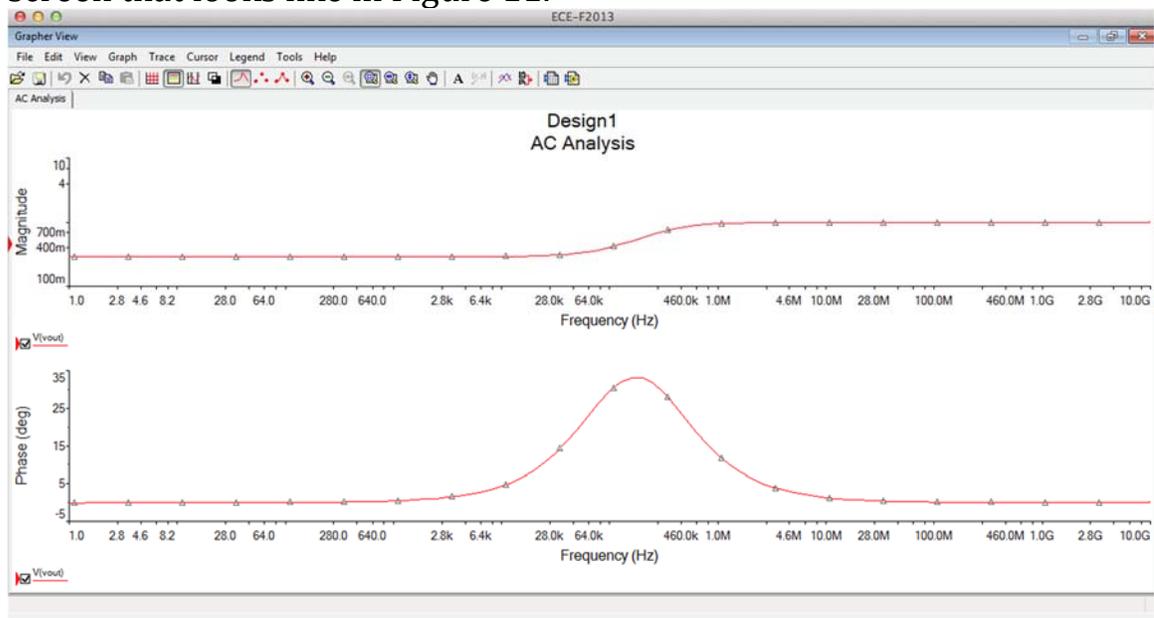
**Figure 9:** Component rearranged and connected with line

9. After the schematic has been created it needs to be saved to the folder. Once it's saved, graph the bode plot for this circuit (Figure 5). To do so, on the menu bar **CLICK** on **SMIULATE** and then **CLICK** on **ANALYSIS**. This will display another drop menu and click on **AC ANALYSIS**. Once that is clicked, get a box where the first tab - **FREQUENCY PARAMETERS**. Change the parameters based on the professor's preference, but for this circuit keep the parameters unchanged (as given in the box). Right next to the **FREQUENCY PARAMETERS** locate something called **OUTPUT** and **CLICK** on it. Once the tab is opened, locate **ALL VARIABLE** and select **Vout** and **CLICK** on **ADD** for analysis. The screen should look as following in Figure 10.



**Figure 10: AC analysis output**

- After Vout is on the analysis side, **CLICK** on **SIMULATE** and get the screen that looks like in Figure 11.



**Figure 11. Graph after simulating**

### 3.1 Hints:

1. When doing the first order circuit homework in Multisim, for the voltage component use piecewise linear voltage source.
2. Also, in the first order circuit homework, make sure primary and secondary y-axes are enabled.
3. After the schematic is finalized, click on **FILE**, then **SNIPPET**, and **SAVE SNIPPET AS SELECTION**.
4. After the simulation is done, you can save it as a **PDF** when you click on print.
5. In the homework when it says plot **bode plot**, it is the same as plotting **AC VOLTAGE ANALYSIS** under the **SMILULATION** tab.

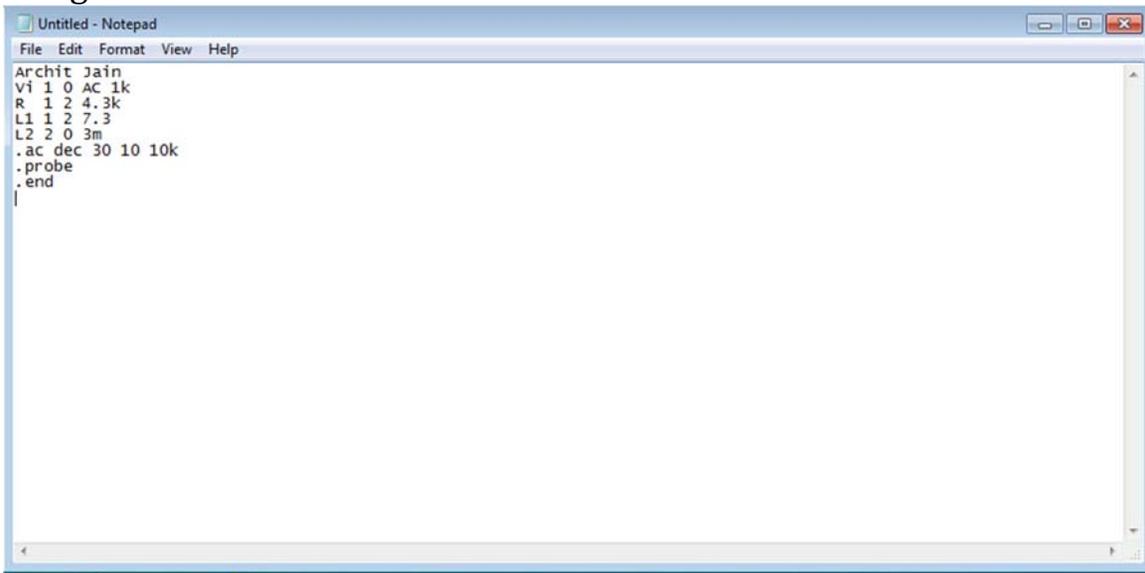
## 4 LTSpice

---

As mentioned earlier, LTSpice is a schematic capture and waveform viewer with enhancements and modes for easing the simulation. LTSpice can be used in two specific ways - first is by drawing schematic, and the second one is by writing code using notepad than open it in LTSpice. In this section of the introductory manual the coding part of LTSpice will be more appropriate to study. The same schematic from Figure 4 can be applied to look at the coding in LTSpice.

1. The example circuit Figure 4 used in Multisim will also be used in LTSpice. The code will be written in notepad and then **SAVED** as a **.CIR** file.
2. The first step in coding in LTSpice is to look at the node in Multisim. Once the node has been gathered from Multisim for this specific circuit, writing the code is easy in LTSpice. {The node in the homework will be different according to the circuit.}
3. First line of the code is always a comment line with the students name and/or the name of the graph or the homework.
4. The second line is stating the input voltage at node 1 and 0 and the AC stands for **AC VOLTAGE SOURCE** used in Multisim and 1k stands for input **FREQUENCY**.
5. In the next three line of codes, the left hand side of the code is **LETTER** of the components, the center part is the node of the components represented in the circuit, and the right hand side of the code is the value of the component given in homework or the calculated value.
6. The line that comes after all the component values is **.ac dec 30 10 10k**, in LTSpice **.ac** means that the graph is going to be **AC VOLTAGE ANALYSIS**, **dec 30 10 10k** just means that LTSpice is using decimal values and with 30 number of pints and starting value is 10 Hz frequency and end value is 10 kHz frequency.

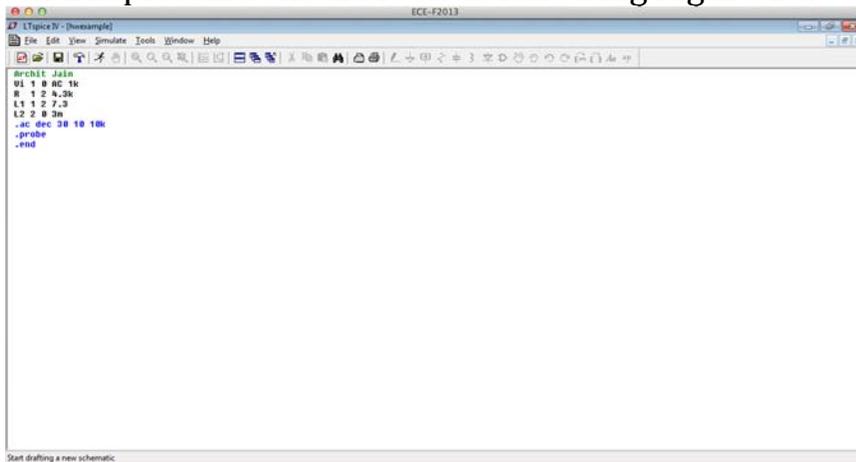
7. Finally the last two lines of code is **.probe** and **.end**; first the **.probe** tells LTSpice to write the result of **DC, AC, or TRANSIENT SIMULATION** in probe graphic form, second the **.end** is telling LTSpice to end the code here. {For majority of the homework professor Brewer will provide the outline of the code.}
8. Now save the code in notepad and it should look as the following Figure 12.



```
Archit Jain
vi 1 0 AC 1k
R 1 2 4.3k
L1 1 2 7.3
L2 2 0 3m
.ac dec 30 10 10k
.probe
.end
```

**Figure 12.** Code Written in Notepad

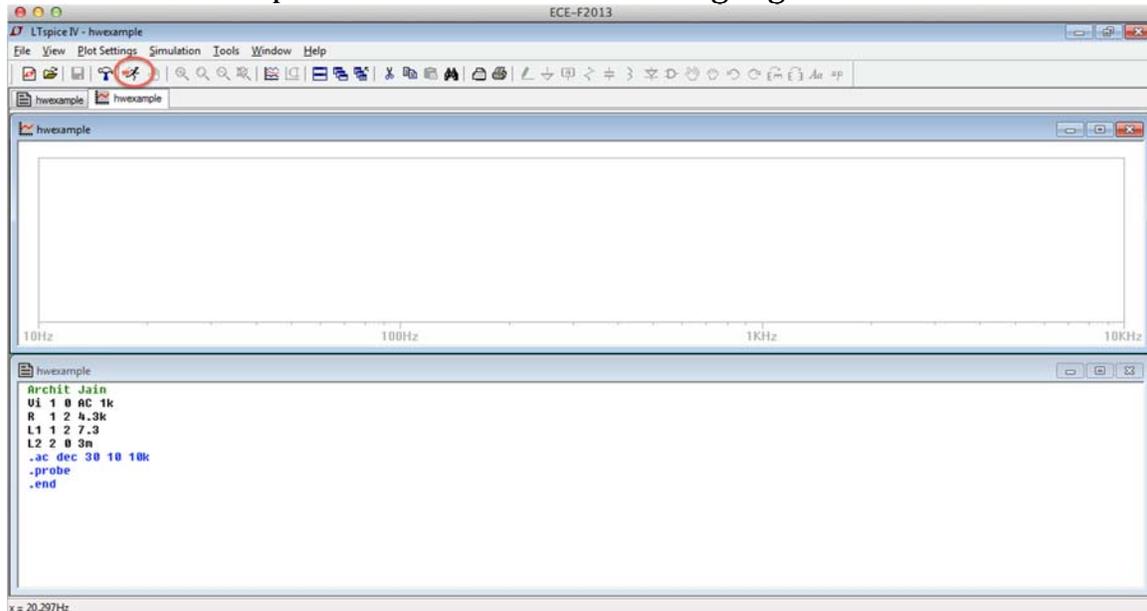
9. After all the of code is written once again save it as a **.CIR** file, then open it in LTSpice and should look as following Figure 13.



```
Archit Jain
vi 1 0 AC 1k
R 1 2 4.3k
L1 1 2 7.3
L2 2 0 3m
.ac dec 30 10 10k
.probe
.end
```

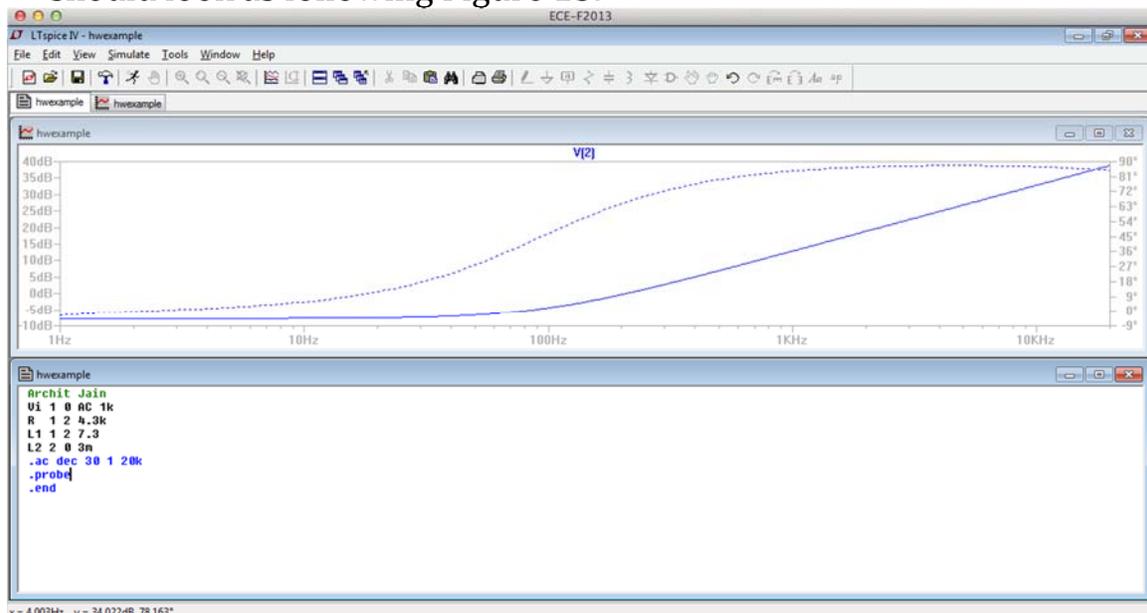
**Figure 13.** How LTSpice should look

10. Now **CLICK** on the run button that looks like a guy running on the menu bar indicated with the red circle, in the Figure 14, and the screen on LTSpice should look as following Figure 14.



**Figure 14.** After you **CLICK** run without graph

11. Now **RIGHT CLICK** on the graph and **CLICK** on **VISIBLE TRACE** and pick out the output **VOLTAGE**. But for this specific circuit the output voltage is going to be V[2]. After selecting the output the screen should look as following Figure 15.



**Figure 15.** Screen capture with graph

12. After the simulation is done save the coding and graph as a **.PDF** file so later it will be easier to print later on.

## 5 Mathcad

As mentioned earlier, Mathcad is a standard software for solving and analyzing the most vital engineering calculations. Mathcad has mathematical notation, unit's intelligence, and has an easy-to-use interface. The same circuit (Figure 4) that was applied in Multisim and LTSpice will be used here, to see how the software works.

1. In Mathcad, when the **RED PLUS** sign appears on the screen you start typing in the equations given in the class or the calculated equation in the homework.
2. For the circuit represented in Figure 4, lets write down all the component values and the equations that were given to us.
3. For example, the way to write in a component in Mathcad is to say  $R := 4.3k\Omega$ , by writing  $:=$  its stating that R is equal to that value and to get ohm symbol or any other symbol in Mathcad simply just **CLICK** on **MEASURING CUP** on the **COMMAND BAR**.
4. Once all the equations and the component values are written in Mathcad, the screen should look as following Figure 16.

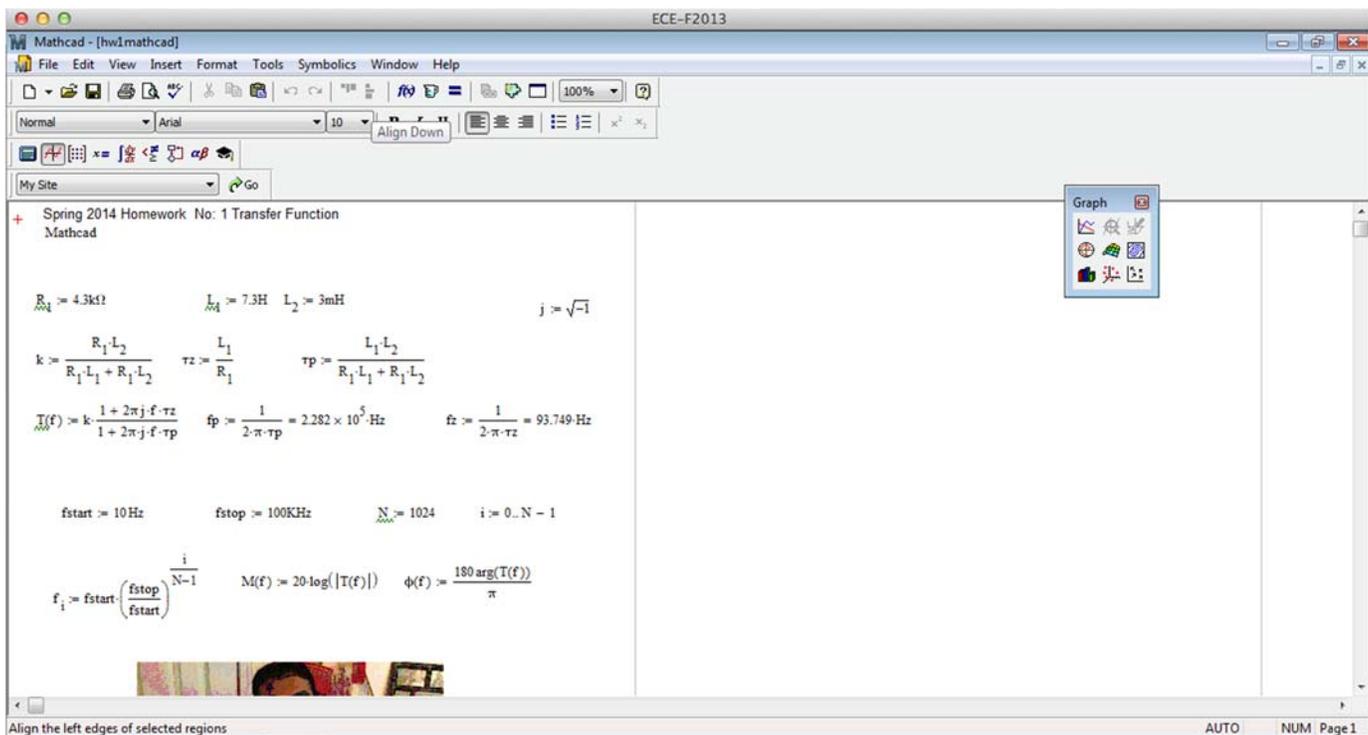


Figure 16. After all the equations

5. Now **CLICK** on the **GRAPH SYMBOL** on the **COMMAND BAR**, and a blank graph screen should appear. In order for the line to appear, **FREQUENCY** has to be mentioned in the x-axes, which is represented by  $f_i$ , and on the y-axes we have the magnitude and phase equation that are represented by **M** and **PHI**. After they are inputted, the screen should look as following Figure 17.

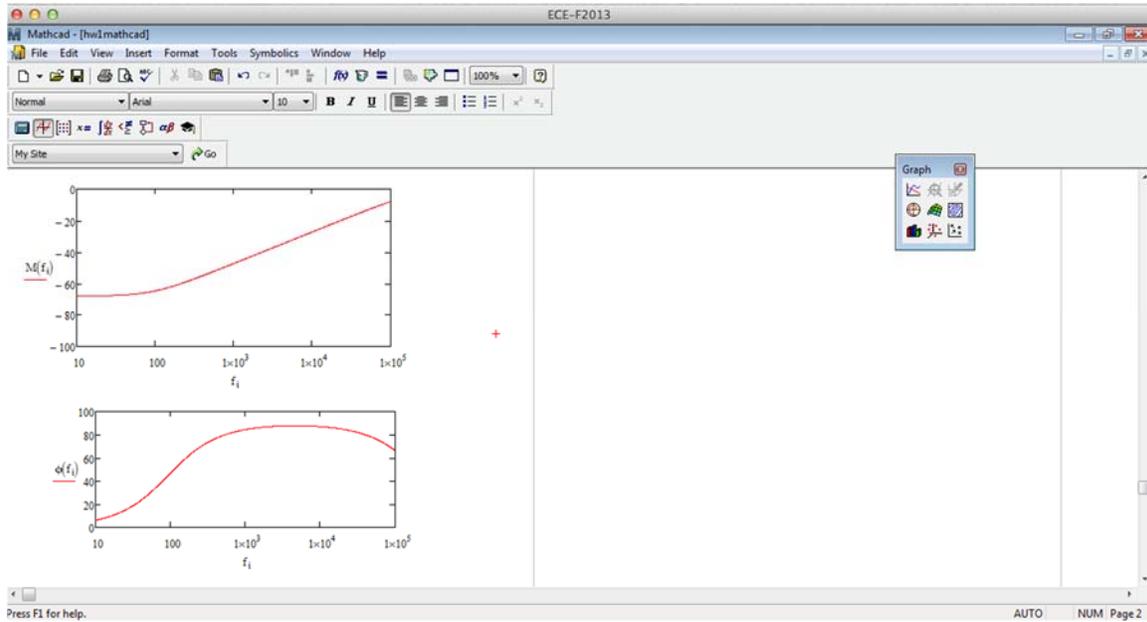


Figure 17. Mathcad with graphs

## 6 References:

---

[1] (2014, March 2). What is Multisim? [National Instrument]. Available: <http://www.ni.com/multisim/what-is/>

[2] (2014, March 3). What is LTSpice? [Linear Technology]. Available: <http://www.linear.com/designtools/software/>

[3] (2014, March 3). What is Mathcad? [Product & Service Advantages]. Available: <http://www.ptc.com/product/mathcad/about/>