HOMEWORK SET 3

OBJECTIVE: The purpose of this lab is to give you experience simulating basic circuits using SPICE simulation tools.

Start out familiarizing yourself with the LTSPICE simulation environment. You can use any SPICE simulation environment you want, but LTSPICE gives you a lot of functionality and is offered free from Linear Technology. The tradeoff is that the GUI is a bit antiquated and you mostly have Linear Technology parts loaded in the library.

Lets start off by setting up a schematic environment that will allow you to build your circuit. This can be done by going to:

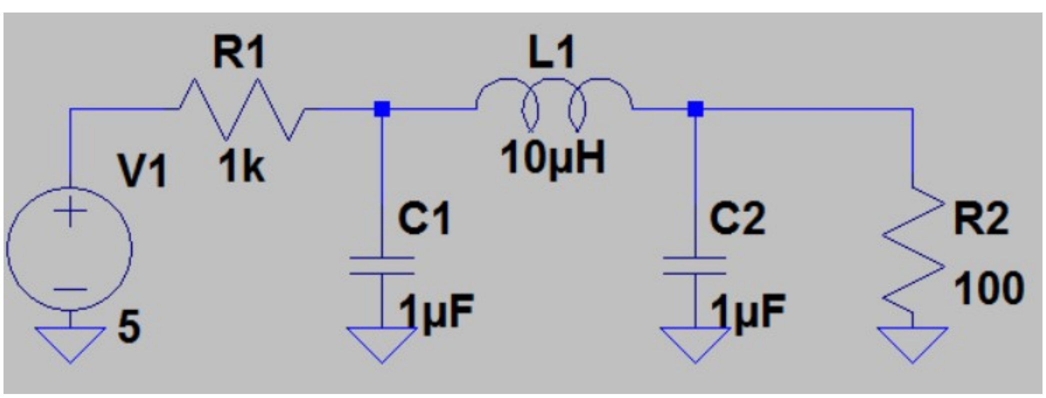
FILE → NEW SCHEMATIC

You will notice that a new menu bar has appeared giving you access to many of the core circuit elements. The most important options you will have are:

1. Place wire
2. Place GND
3. Resistor/Capacitor/Inductor/Diode (See the appropriate symbol)
4. Place component (See AND gate symbol)

There are plenty of hot-keys that will help make setting up the circuit quicker (ie: pressing “R” on the keyboard will create a resistor). To see these hot-keys, simply look at the “EDIT” menu.

Begin by creating a basic circuit like the one shown below. Special elements like the voltage source can be found in the “Place Component” button. Once you wire up all of the components, you will want to assign values to each of the components. This can be done by either right clicking on each element or double clicking on each element. For special values like the 1 micro-farad capacitor, you can use letters like “u” or “n” or “p” to denote micro, nano and pico.



Now that the your circuit is setup, you may notice that the source “V1” and source resistance “R1” can be simulated as a Thevenin equivalent circuit. The Pi-filter (Capacitor-Inductor-Capacitor) is a filter in-line with your load resistance “R2”. Analyzing circuits piecewise like this helps break down the various functionality of a larger circuit as shown to help look at individual elements within the circuit.

To run a simulation, you should be aware that LTSPICE allows you to perform several types of analysis. You have the ability to set which type you want by going to the following menus and selecting the appropriate one:

SIMULATE → Edit Simulation Command

1. **Transient**: Time dependent behavioral analysis
2. **AC Analysis**: Frequency dependent behavioral analysis
3. **DC Sweep:** DC operating point analysis with variable source
4. **Noise**: Stochastic noise analysis
5. **DC Transfer**: DC small signal transfer characteristics
6. **DC Operating Point**: DC behavioral analysis

SIMULATION 1: PERFORM A DC SWEEP ANALYSIS

First lets try to perform a “DC SWEEP ANALYSIS”. This type of analysis will evaluate your circuits steady-state DC behavior with sources that have varying properties. For this analysis, you have one source “V1”, so enter that into the simulation command window and step the voltage linearly from -15V to +15V in 0.1V steps. When you click “OK”, you will be brought back to your schematic environment where your mouse cursor will say something like “.dc V1 -15 15 0.1”. This means that the simulator will solve for the DC behavior across the entire circuit for all values between -15V to +15V in 0.1V steps.

This is a command line level entry to tell the tool that you are performing a DC analysis where source V1 is stepped between -15V through 15V in 0.1V steps. In the future, you could accomplish the same task clicking on the “SPICE DIRECTIVE” button and entering the same string.

The next step is to simulate the circuit behavior. Do this by going too:

SIMULATE → RUN

Once the simulation is complete, a blank graph window will pop-up. Here you will have the opportunity to tell LTSPICE what you would part of the circuit you would like to graph. This can be done two ways, first, you can go to your circuit and click on a:

NODE → For Voltage

ELEMENT → For Current

The second way is to:

RIGHT CLICK ON GRAPH → ADD TRACE

QUESTION 1:

i. Plot the voltage at your source "V1", the voltage at "C1" and "C2". Intuitively explain what you see and why it is expected.
ii. Plot the currents through each of the elements and intuitively explain what you see and why it is expected.

SIMULATION 2: PERFORM AN AC ANALYSIS

This analysis type is typically performed with AC signals presented by a source within your circuit. Its end results would show you the magnitude and phase relationship.

To perform this analysis:

1. Delete the last simulation directive (ie: .dc V1 -15 15 0.1)
2. Go into the “Edit Simulation Command” window and select “AC ANALYSIS”
3. Select a sweep type → Select “Decade”
    a. Octave: Evaluates frequency steps in x2 multiples
    b. Decade: Evaluates frequency steps in x10 multiples
    c. Linear: Evaluates frequency steps linearly
4. Enter number of points → “100”
    a. This is how many points to calculate between decade frequency steps
5. Enter start frequency → “1Hz”
6. Enter stop frequency → “100MHz”

Press OK and drop the simulation directive on the schematic but don’t run the simulation. Here you are effectively telling the simulator to evaluate the circuits behavior by stepping an AC source from 1Hz to 100MHz with 100 steps between decade frequency steps (ie: 1Hz to 10Hz to 100Hz to 1000Hz to etc.).

Since this is an AC analysis, you will need to setup an AC source within your circuit. This can be done by right clicking on your voltage source “V1” and clicking “advanced”. Here you will be able to tell the simulator that the source is no longer a basic “DC voltage source” and is to behave as something else. Your options setting up the voltage source are:

1. PULSE: A pulsed signal with properties you define in the appropriate text boxes
2. SINE: A sinusoid generator with properties you define in the appropriate text boxes
3. EXP: An exponential generator with properties you define in the appropriate text boxes

You will also note that you will need to enter values for “Small Signal AC Analysis”. Enter the following for this simulation:

1. SINE Wave
    i. DC Offset: 0V
    ii. Amplitude: 1V
    iii. Frequency: 10Hz
    iv. TDelay: 0
    v. Theta: 0
    vi. Phi: 0
    vii. NCycles: 0
    viii. AC Amplitude: 1
    ix. AC Phase: 0

Most of these settings are required to setup the simulation environment for their default values. When the simulator is run, the frequency values will be stepped over the range you defined previously.

QUESTION 2:

i. Simulate and sketch the voltage across “C1” and “R2”. Intuitively explain what you see.
ii. Simulate and sketch the current through “L1”. Intuitively explain what you see.

SIMULATION 3: PERFORM A TRANSIENT ANALYSIS

This analysis type allows you to simulate the transient "time varying" behavior of your circuit. In other words, as time goes on, how are the voltages and currents across your circuit behaving. This type of analysis is particularly useful if you are interested in how long it takes for your circuit to reach a steady state (stable) value.

To perform this analysis:

1. Delete the last simulation directive (ie: .ac dec 100 1 100MHz)
2. Go into the "Edit Simulation Command" window and select "TRANSIENT"
3. STOP TIME: This is the time when the simulator should stop evaluating the circuit behavior
    a. Enter "300mS" for 300 milli-seconds
4. TIME TO START SAVING DATA: This is the time when the simulator should start evaluating the circuit behavior
    a. Enter "0" for 0 seconds
5. MAXIMUM TIMESTEP: This is the time-step to evaluate the circuit behavior
    a. Enter "0.01" for 10mS

In this simulation case, you will need to have some type of time varying source in order to evaluate the time-dependent behavior of your circuit. Here you will want to go into the advanced properties of the voltage source and define it as some sort of an "AC source" otherwise the solver will treat it as a DC source and there will be no time varying behavior from your circuit.

For this simulation, define your voltage source as a "PULSE" generator with the following properties:

1. Initial Voltage: 0V
2. On Voltage: 15V
3. Tdelay: 1nS
4. Trise: 1nS
5. Tfall: 1nS
6. Ton: 1S
7. Tperiod: 2S
8. AC Amplitude: 15
9. AC Phase: 0

Run the simulation and plot the voltage across the source "V1". Overlay the voltage across the load resistance "R2" by clicking once at the node across "R2". Here you will see the stepped input signal presented to your circuit and the resulting output signal presented at your load.

For this circuit, the primary time dependent behavior occurs between 0 and 1mS. To get better resolution, click on the x-axis of the graph and change the scale too:

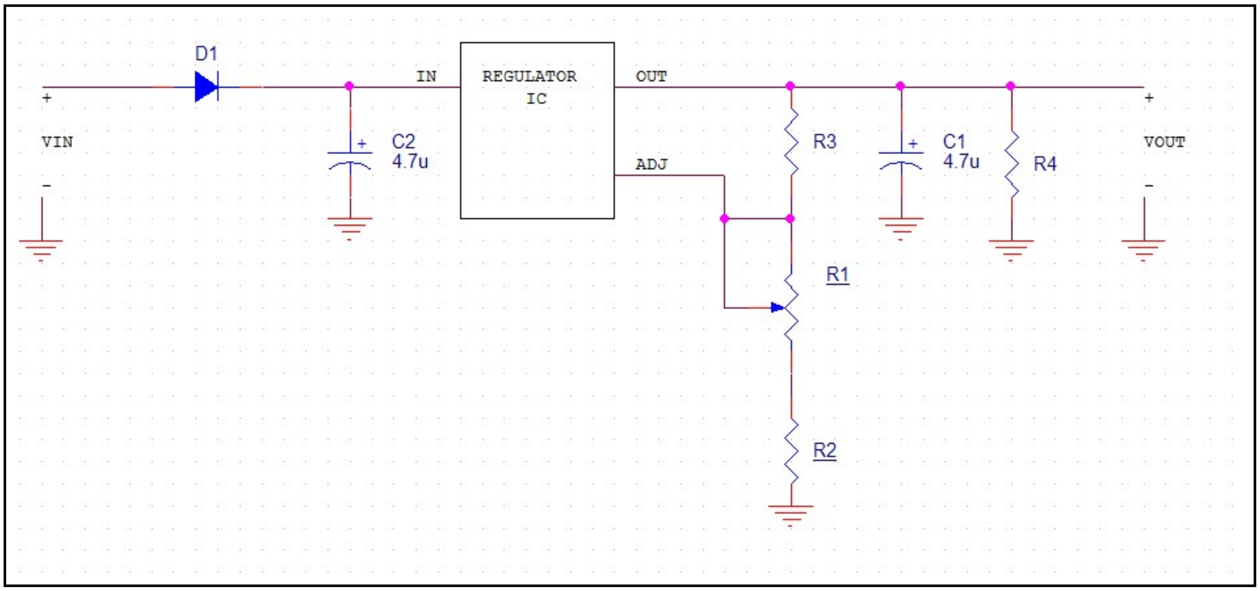
1. Left: 0s
2. Tick: 0.5mS
3. Right: 5mS

QUESTION 3:

i. Sketch the corresponding signal plot and explain the results you see.
ii. How long does it take for the output voltage to reach steady state from the turn-on step response of the input source. What is the steady state output voltage.
iii. Change the graph scale again and evaluate how long does it take for the output voltage to reach steady state from the turn-off step response of the input source. What is the steady state output voltage.

ASSIGNMENT 4: SIMULATE YOUR REGULATOR CIRCUIT

Now that you have some experience simulating electronic circuits, please draw out the regulator circuit you made from the last lab in LTSPICE. The regulator type you should have used is the "LT1085". For the resistor "R1" shown in the drawing below, you can use a fixed resistor value and step the resistance value in the simulation command window.



QUESTION 4:

i. Simulate the circuit using a DC-SWEEP analysis sweeping the input voltage from -25V to +25V
   a. Plot the output voltage and input voltage to the circuit on the same graph
   b. Plot the output voltage and the input voltage to the regulator on the same graph.
   c. Be sure and explain the results and differences between the two plotted graphs and determine if the simulated response is expected for this circuit.

ii. Simulate the circuit using a TRANSIENT analysis by configuring the input source as a stepped input transitioning from 0V to +25V
   a. Plot the output voltage and input voltage to the circuit on the same graph
   b. Plot the output voltage and the input voltage to the regulator on the same graph.
   c. Be sure and explain the results and differences between the two plotted graphs and determine if the simulated response is expected for this circuit.