

LTSPICE - A Beginner's Guide

Kevin Alessandro Bautista

Version 1.2

Last updated: November 1, 2023

Contents

| | | |
|----------|-----------------------------------------------------------|-----------|
| 1 | Author's Note | 4 |
| 2 | Introduction | 4 |
| 2.1 | What is LTSPICE? | 4 |
| 2.2 | Content Expectations and Supplemental Resources | 5 |
| 2.3 | Suggested Materials | 5 |
| 2.4 | Getting Started | 5 |
| 3 | Circuit Building in LTSPICE | 7 |
| 3.1 | Accessing the Components | 7 |
| 3.2 | Accessing Passive Components | 8 |
| 3.3 | Moving and Connecting the Components | 9 |
| 3.4 | Component Values | 10 |
| 4 | Time Varying Signals in LTSPICE | 12 |
| 4.1 | Time Varying Voltage Signals | 12 |
| 4.2 | Time Varying Current Signals | 12 |
| 5 | Running an AC Simulation in LTSPICE | 14 |
| 5.1 | Setting Up Your Components | 14 |
| 5.2 | Spice Directives | 15 |
| 5.3 | Running the Simulation | 16 |
| 6 | Frequency Analysis | 18 |
| 6.1 | Setting up Your Sources | 18 |
| 6.2 | Spice Directives | 19 |
| 6.3 | Running your Simulation | 20 |
| 7 | Running a .OP (DC) Analysis | 22 |
| 7.1 | Setting up your Sources | 22 |
| 7.2 | Spice Directives | 23 |
| 7.3 | Running your Simulation | 23 |

List of Figures

| | | |
|---|------------------------------------|---|
| 1 | LTSPICE Intro Screen | 5 |
| 2 | LTSPICE Schematic Screen | 6 |
| 3 | Component Symbol | 7 |
| 4 | Component Table | 7 |
| 5 | Filter Components | 9 |
| 6 | Rearranged Components | 9 |

| | | |
|----|------------------------------------------|----|
| 7 | Connected Components | 10 |
| 8 | DC Voltage Menu | 12 |
| 9 | AC Voltage Menu | 12 |
| 10 | Current Voltage Menu | 13 |
| 11 | AC Current Menu | 13 |
| 12 | Sine Menu | 14 |
| 13 | Spice Directive Menu | 15 |
| 14 | Circuit with Spice Directive | 16 |
| 15 | Transient Simulation Example | 17 |
| 16 | RLC filter | 18 |
| 17 | RLC filter | 18 |
| 18 | Filter with Spice Directive | 20 |
| 19 | Filter with AC Analysis | 21 |
| 20 | DC Circuit | 22 |
| 21 | DC Circuit with Values | 23 |
| 22 | Circuit with .OP Directive | 23 |
| 23 | Result of Running .OP Analysis | 24 |
| 24 | Circuit with .OP Analysis | 24 |

1 Author's Note

This document is meant to serve as a beginner's guide to using LTSPICE. It's not meant to be an extensive guide on everything LTSPICE can do, rather it's a helping tool that can get a person started on fundamental simulations and tools that LTSPICE has. I initially started this document about a few months ago, but had this idea for a solid year now. I still consider myself a beginner user of LTSPICE, but found that a lot of the tutorials needed to actually run simulations or do analyses were scattered across multiple resources that made them a bit hard to find. As such, I created this guide as a way to help other students / LTSPICE users grab a foothold in this software as it does have its uses, but also as a way to not discourage people from exploring circuits as a whole.

At its core, there will likely be a few revisions to this document as time goes on, with more sections being added or edited depending on content. I do not think that this document is perfect by any means, but I do hope that it provides some use and relevance to you on your journey into simulations. I've learned that it's better to start imperfectly than to be paralyzed by the idea of perfection, otherwise this document likely wouldn't have gotten finished. If you have any questions, comments, or concerns, I'm happy to hear them at: kbautis@purdue.edu. With all that said, I hope you benefit from this document as I had writing it.

Thank you for your time.
Kevin Alessandro Bautista
kbautis@purdue.edu

2 Introduction

2.1 What is LTSPICE?

At its core, LTSPICE is a circuit simulator that allows us to represent a lot of circuit models, and with the right parameters, we can assume both ideal and non-ideal cases. It is used for both simulation and schematic purposes. While other free software such as AutoDesk EAGLE exists, LTSPICE is a friendly introduction to circuit theory. Depending on when you're reading this, we will be using **LTSPICE XVII v17.0.27**. The download link can be found here: <https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>. If there is a newer version, feel free to use that. The versions shouldn't be too different anyways.

It is also important to note that the images here may sometimes look different from what you would see depending on what operating system (OS) you're running. The images or examples will have been taken using a Windows 10 OS, which could look very different from a Mac OS. However, underlying methods should be the same, and key processes such as how to access different features and additional distinctions, will be clarified if they differ

based on OS.

2.2 Content Expectations and Supplemental Resources

As mentioned previously, this document is essentially a survival-guide on how to use LTSPICE. It is not in any way, shape, or form, a circuit theory document. I am not skilled nor eloquent enough to explain in-depth circuit theory in a concise manner, and so I'll stick to explaining **“how”** to use LTSPICE, and leave the **“why”** of how a circuit functions the way it does to your coursework and labs.

While you may benefit from a more active form of learning, I will also attach a link to videos that I feel are useful. Watching these is not required to understand the document, but could benefit someone who might prefer to learn this way rather than reading about it. I will not be referencing the videos in the later sections, just note that they will cover roughly the same amount of content.

2.3 Suggested Materials

Really, the only material you need is a laptop, but a mouse might help you work a bit quicker. Additionally, if you're working on a desktop, then the mouse is probably a necessity. Also, downloading LTSPICE is a good place to start! Once you have it downloaded, you're ready to go.

2.4 Getting Started

When first opening up LTSPICE, you'll encounter this screen:

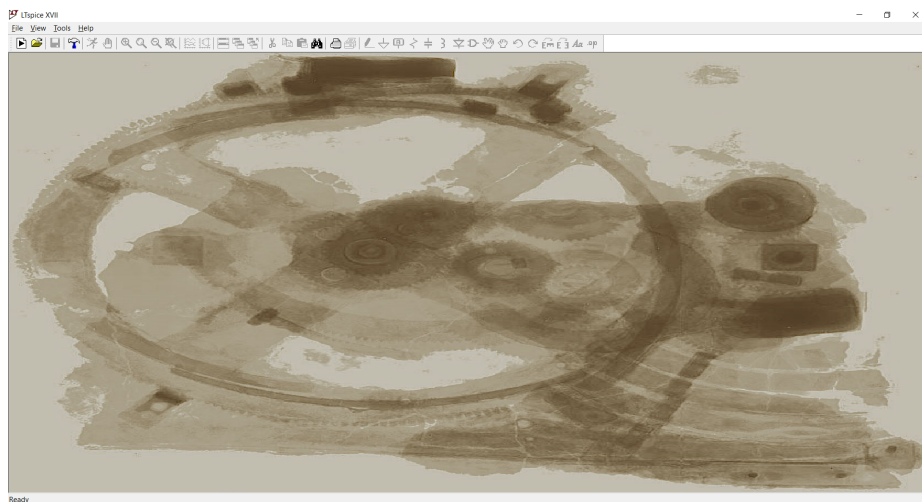


Figure 1: LTSPICE Intro Screen

For Mac: right click the screen and click “New Schematic” to get started.

For Windows: you can do the same, or go to File in the upper left corner, and click “New Schematic” to get started.

Once you do so, you should see the following screen. Note that if you’re on a Mac, the toolbar will be missing at the top. This screen ensures that you’re on a new schematic and are ready to now begin building and simulating your circuits.

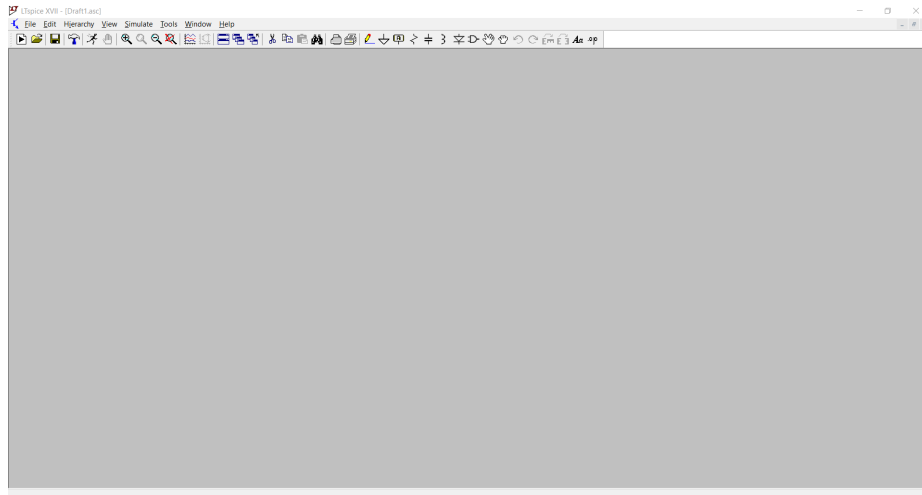


Figure 2: LTSPICE Schematic Screen

This is the screen we’ll be working on for the rest of the guide. It’s where you can build your circuit and run all the simulations you need.

3 Circuit Building in LTSPICE

3.1 Accessing the Components

For Mac: right click the screen and click “Draft” and then click “Component”.

For Windows: you can do the same, or go to the toolbar at the top of your screen and click on the symbol shown below.



Figure 3: Component Symbol

Doing so gives you a LTSPICE’s table of all the components it has to offer. While there are shortcuts which we’ll discuss later, this table also has a nice search bar as shown below so you can search all the components manually.

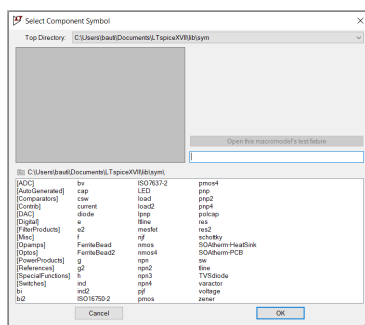


Figure 4: Component Table

Simply type in a component you want (i.e. Res or Cap for a resistor or a capacitor) and press enter or click on it. For the most part, you’ll only be dealing with RLC circuits and op-amps at most, but LTSPICE also offers MOSFETS, transistors, BJTs, and so on and so forth. LTSPICE has variations of all of these, which will likely vary based on your application. For the practical purposes and so this report doesn’t take up 30 pages, we’ll be focusing on RLC circuits and basic op-amp components, but feel free to play around with it as we go along.

For the purposes of this document, we’ll be going over **how to build a standard first-order RLC filter** as it provides a good indication of all the tools you would need in order to succeed with LTSPICE.

3.2 Accessing Passive Components

Now that you're able to access the Component option and pick out your components, we can go ahead and get started with laying out our components. Commonly, you will be using these components when building your circuit:

- Voltage source
- Ground
- Resistors
- Capacitors
- Inductors

The following table demonstrates what to search for in the Components tab as well as shortcuts on your keyboard that you can press while on the schematic screen:

| Component | Component Tab Name | Shortcut |
|-----------|--------------------|----------|
| Voltage | Voltage | v |
| Ground | N/A | g |
| Resistor | res | r |
| Capacitor | cap | c |
| Inductor | ind | l |

Table 1: Useful table of keyboard shortcuts and component names of common components in circuit design

From 1, you can type in the specific component tab name in the component search bar and hit Enter or click OK. Or, you can simply press the keyboard shortcut on your schematic. Once you do either of these things, you'll notice that the component is attached to your mouse. Click anywhere to place it.

Go ahead and place a voltage source, GND, a resistor, a capacitor, and an inductor side by side as shown in the figure below.

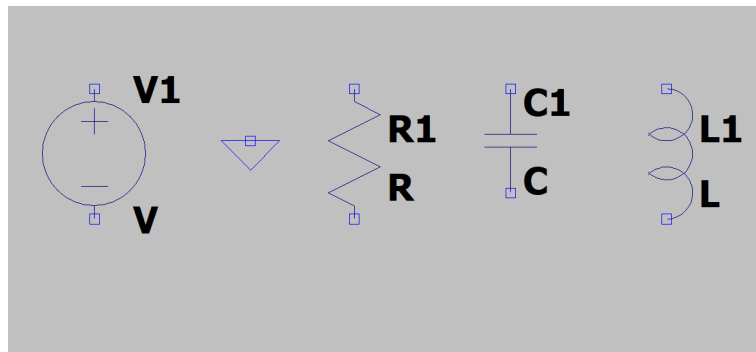


Figure 5: Filter Components

3.3 Moving and Connecting the Components

To move the components:

For Mac: right click the screen and click “**Edit**” and then click “**Move**”.

For Windows: you can do the same, or go to the toolbar at the top of your screen and click on the **Open Hand** symbol on your toolbar.

From there, you can simply click on any component you’d like to move and it should move with your mouse. **If you’d like to rotate, you can click CTRL+R or CMD+R to rotate your component by 90 degrees each time. You can also do this when placing your components for the first time.**

Move and rotate each component such that it looks like the figure below.

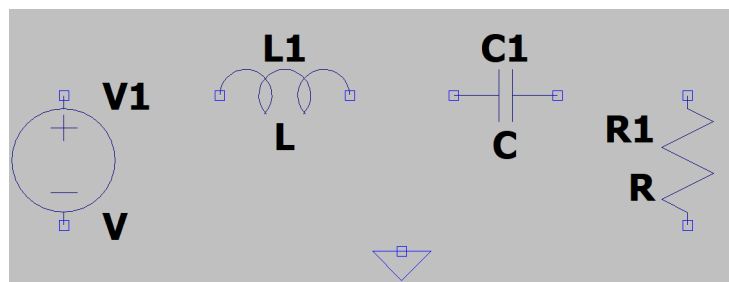


Figure 6: Rearranged Components

Notice how each component has a box on each of its ends. These are its connecting points to signify where you can connect things to that component. LTSPICE connects things using Wires. To connect:

For Mac: right click the screen and click “**Draft**” and then click “**Draw Wire**”.

For Windows: you can do the same, or go to the toolbar at the top of your screen and click on the **Pencil** symbol.

Doing this brings up a crosshair-like cursor. **To connect components, click on one box of one component then connect it to another box of the desired component.** Note that you can only draw straight wires. You can click outside of boxes to create 90 degree turning points for your wires, but it’s often easier to just place your components the right way so you do less work trying to figure out wiring.

Connect your component such that it matches the figure below. Note that for the voltage source, I had to make a 90 degree turn once I was in line with the GND symbol.

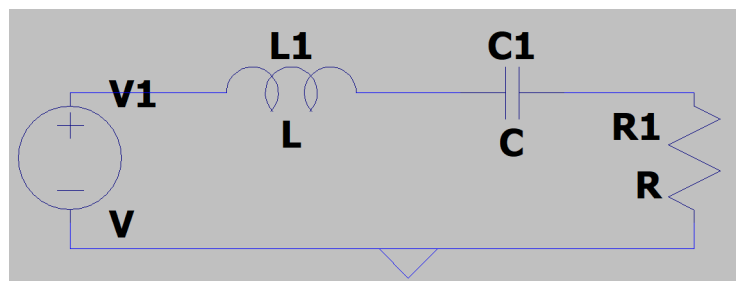


Figure 7: Connected Components

3.4 Component Values

To change the values of the components, simply right click the middle of the component you’d like to change. Note: Right clicking the text (i.e. R1, L1, V1, etc.) changes the designation / name of the component NOT the value. In the table below, there is a list of prefixes you can attach at the end of the numerical value as a multiplier.

| Prefix | Value | Shortcut | Example |
|--------|-------|----------|---------|
| Mega | 1e6 | Meg | 10Meg |
| Kilo | 1e3 | k | 3k |
| milli | 1e-3 | m | 1m |
| micro | 1e-6 | u | 12u |
| nano | 1e-9 | n | 20n |
| pico | 1e-12 | p | 100p |

Table 2: Useful table of prefixes for your component values

Right clicking the component pulls up a separate menu where you can adjust the value of each component. **For current sources or voltage sources, you're initially greeted with the DC values, but clicking "Advanced" pulls up a more AC based menu.** We'll explore these parameters once we start running simulations.

4 Time Varying Signals in LTSPICE

4.1 Time Varying Voltage Signals

To create a time varying voltage signal, right click on the middle of any voltage source to pull up the menu as shown below.

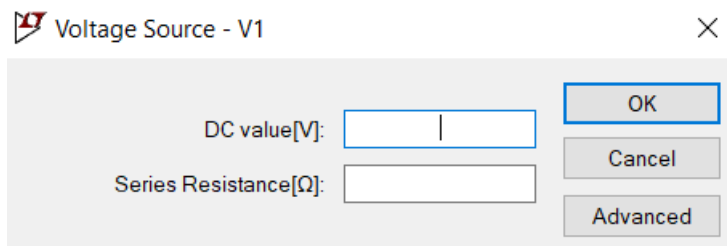


Figure 8: DC Voltage Menu

This is the DC menu for voltage. You can adjust factors here when dealing with a **non-time varying signal**. Click "Advanced" to show the AC menu shown below.

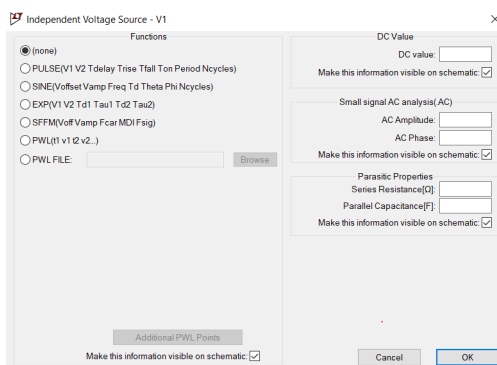


Figure 9: AC Voltage Menu

Here, you can see the AC menu for accessing various types of time-varying signals that you can play around with. We'll get into how to use some of them later. For the most part, you'll likely be sticking to the sine-wave function or pulse if you're starting out.

4.2 Time Varying Current Signals

To create a time varying current signal, right click on the middle of any current source to pull up the menu as shown below.

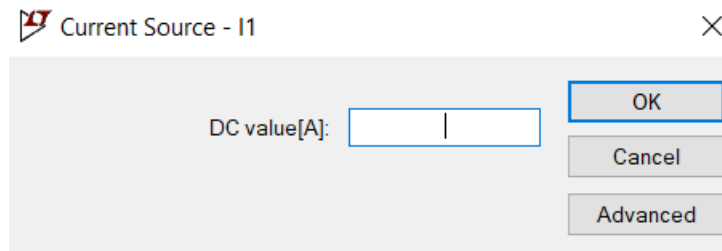


Figure 10: Current Voltage Menu

This is the DC menu for current. You can adjust factors here when dealing with a **non-time varying signal**. Click "Advanced" to show the AC menu shown below.

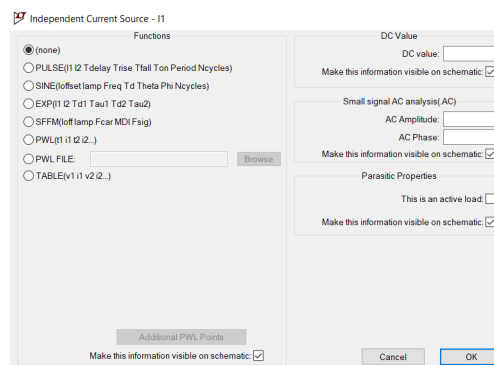


Figure 11: AC Current Menu

Similar to the voltage source, you can also access much of the same settings when picking AC simulations. Because of their similarity and for convenience, examples using time-varying signals will be done with AC voltage sources.

5 Running an AC Simulation in LTSPICE

It's important to be able to run time-varying simulations because you want to see how your circuit responds over time, giving you a bunch of key information about your circuit's properties.

5.1 Setting Up Your Components

For this example, we'll be using the same figure from 3.3 (Figure 7). First, we'll set up our voltage source.

Right click the voltage source, and click "Advanced" and click the circle right beside "Sine". Note that you can do this for any of the time signals in the options, but for this example we'll be using Sine for ease.

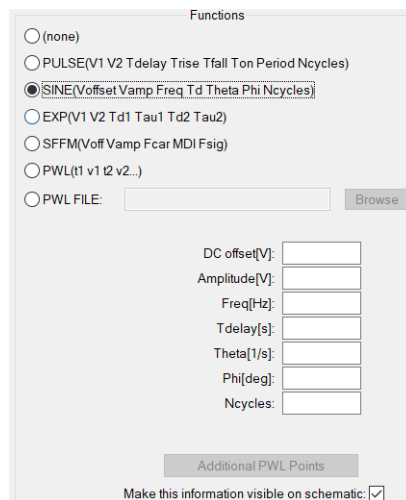


Figure 12: Sine Menu

Here, do the following:

- Set DC offset = 0
- Set Amplitude = 1
- Set Freq = 1000
- (Optional but I recommend): Check the box that says "Make this information visible on schematic"
- Click OK

You can leave the rest blank. I found that these variables aren't necessary for running ideal and basic simulations.

Now that you have your source set up, it's time to set up our passive components.

- Set $R = 10$
- Set $C = 3e-6$ (type 3u)
- Set $L = 0.01$ (type 1m)

Since we're dealing with ideal cases, do not add any non-ideal factors or parasitic components.

Note: When doing your own schematics, these can be any combination of values that fit whatever simulation you're trying to run. We'll cover how to setup other types of signals in a later section.

5.2 Spice Directives

Spice directives are a big part of how LTSPICE can run simulations. To access Spice Directives:

For Mac: right click the screen and click **Draft** and then click **Spice Directive**. Or, you can press **"s"** on your keyboard.

For Windows: you can do the same, or go to the toolbar at the top of your screen and click on the symbol that says **".op"**.

Doing that correctly brings up the following menu where you can type in what you would like LTSPICE to simulate:

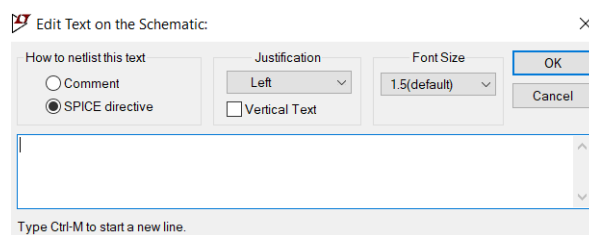


Figure 13: Spice Directive Menu

To analyze AC signals as they vary through time, we need to run a *transient* simulation. A transient simulation uses the following directive:

.tran X

The ".tran" is to let LTSPICE know you want to run a transient simulation and the X denotes the desired time you'd like to run the simulation for. You can put any prefix with X. For our example. type the following in the directive: **.tran 5m** and click OK. The directive will hover with your mouse and you can click to place it where convenient. Your circuit should now look like the one shown below.

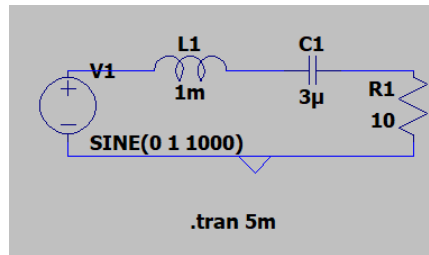


Figure 14: Circuit with Spice Directive

5.3 Running the Simulation

For Mac: right click the screen and click **"Run"**.

For Windows: you can do the same, or go to the toolbar at the top of your screen and click on the symbol that looks like a running figure.

The first time you run a simulation you will likely just see a black screen. To see figures, you would have to select which components you'd like to specifically measure. **Note: If you want to measure VOLTAGE - hover your mouse on the nodes (connections between components) of your circuit until you see a RED probe. If you hover over the middle of your component, you'll see a BLACK circle indicating you're measuring CURRENT.**

For this example. measure the current and the voltage across R1. Your output should look like the snippet below if you followed along - where the simulation ran for 5 ms and you can see the characteristic phase shift between current and voltage. This indicates that your filter is working and you're measuring the right thing!

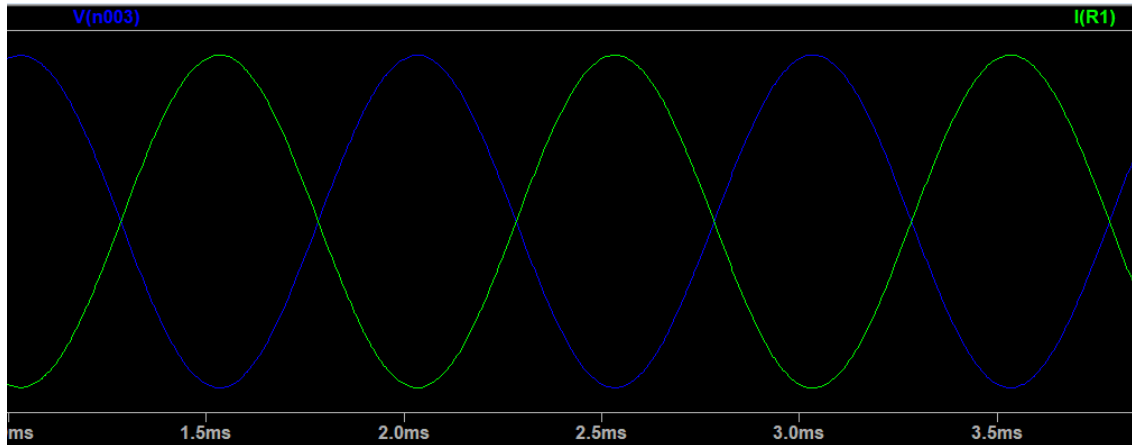


Figure 15: Transient Simulation Example

6 Frequency Analysis

A key part of a beginner's toolkit is being able to run a frequency analysis, which is particularly beneficial when dealing with your filters. This section will be using the RLC filter we've been working on as an example but can be applied to any circuit whose frequency component you're analyzing.

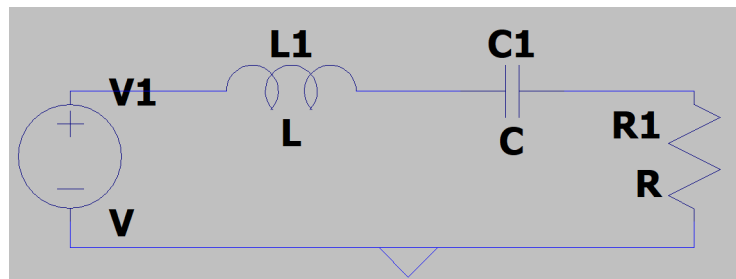


Figure 16: RLC filter

6.1 Setting up Your Sources

First, right click your voltage source to bring up the menu. Click "Advanced" and go to the section labeled "Small Signal Analysis (AC)".

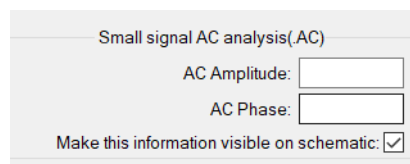


Figure 17: RLC filter

Set the AC Amplitude equal to 1, and the AC Phase equal to 0. These are just the conditions of the AC signal of our filter. AC Amplitude denotes the amplitude of the AC wave, and the AC Phase simply denotes the phase that this wave has.

You can then change your component values to your desired parameters. For this example, change the RLC values to the following:

- Set $R = 10$
- Set $C = 3\text{e-}6$ (type 3u)
- Set $L = 0.01$ (type 1m)

6.2 Spice Directives

For Mac: right click the screen and click **Draft**” and then click **“Spice Directive”**. **Or, you can press ”s” on your keyboard.**

For Windows: you can do the same, or go to the toolbar at the top of your screen and click on the symbol that says **”.op”**.

In LTSPICE, a frequency analysis is also known as a .ac analysis, and follows the following format:

.ac X-label pts start-freq end-freq

- .ac denotes that we want to run a frequency analysis
- X-label denotes what time of X-axis we want (Octave, Decade, Linear, List) - we'll choose decade as it's using base 10
- pts dictates how many points in between each tick mark of the X axis we want (more points means smoother and more data but longer run time)
- start-freq is your starting frequency (**it's a good habit to let this be 100x smaller than your low frequency cutoff**)
- end-freq is your ending frequency (**it's a good habit to let this be 100x larger than your high frequency cutoff**)

As stated above, it's a good practice to sample 100x smaller and larger your cutoff frequency to ensure that you're getting the entire changes in your signal. You can add prefixes to make it easier. Going too small would mean that you're not capturing the changes well, and going too high, your circuit could start to demonstrate some non-ideal characteristics you're not interested in.

For this tutorial, if you're still following along, go ahead and type in the following into your Spice Directive and click OK:

.ac decade 100 0.1 100k

Place this directive where it is convenient. **Note: If you have a Spice Directive prior to this one, you will need to delete it as it won't run if you have two active directives.**

For Mac: right click the screen and click **Edit**” and then click **“Delete”**.

For Windows: you can do the same, or go to the toolbar at the top of your screen and click on the symbol that looks like a **pair of scissors**.

From there, you can click on the Spice Directive you want to delete and it'll be gone. If done right, your circuit should now look like the one below.

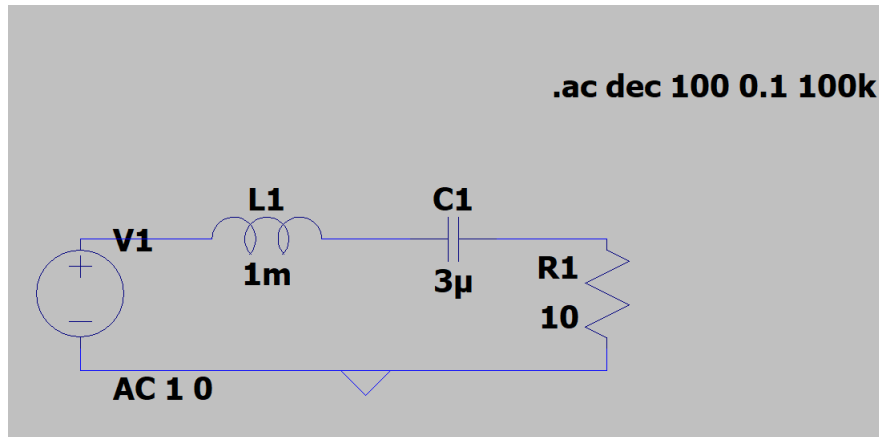


Figure 18: Filter with Spice Directive

6.3 Running your Simulation

For Mac: right click the screen and click **"Run"**.

For Windows: you can do the same, or go to the toolbar at the top of your screen and click on the symbol that looks like a running figure.

The first time you run a simulation you will likely just see a black screen. To see figures, you would have to select which components you'd like to specifically measure. **Note: If you want to measure VOLTAGE** - hover your mouse on the nodes (connections between components) of your circuit until you see a **RED** probe. If you hover over the middle of your component, you'll see a **BLACK** circle indicating you're measuring **CURRENT**.

For this example. measure the current and the voltage across **R1**. Your output should look like the snippet below if you followed along - where the simulation ranges from 0.1 Hz to 100k Hz and you can see the characteristic resonant frequencies as a result of the RLC circuit. This indicates that your filter is working and you're measuring the right thing!

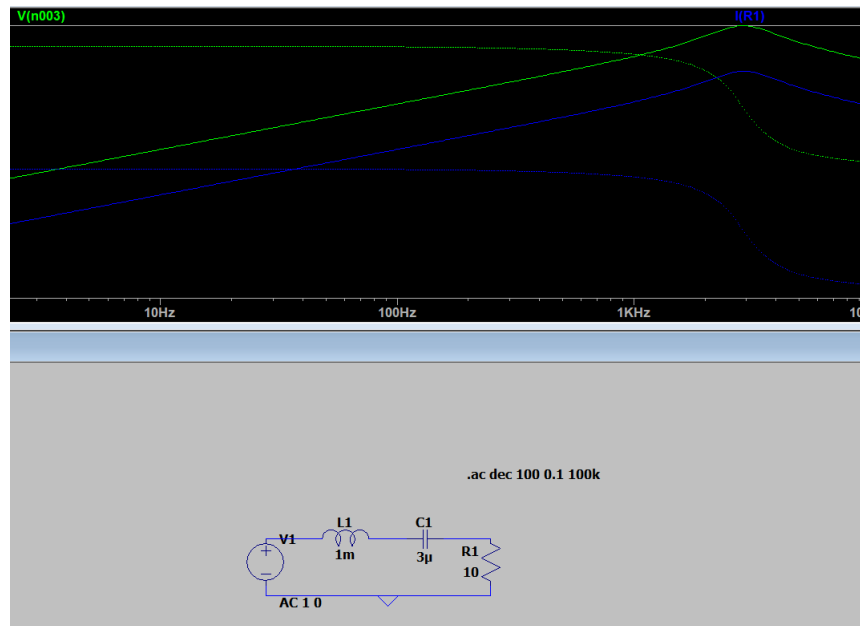


Figure 19: Filter with AC Analysis

7 Running a .OP (DC) Analysis

The past few examples and sections have been dedicated to analyzing AC or time-varying signals. This section will involve analyzing DC circuits as LTSPICE provides a spice directive (**.op**) that gives you all the key information you need to be able to analyze a DC circuit. All the steps here can be applied to any DC circuit, but if you wanted to follow along, the circuit we'll be using is shown below.

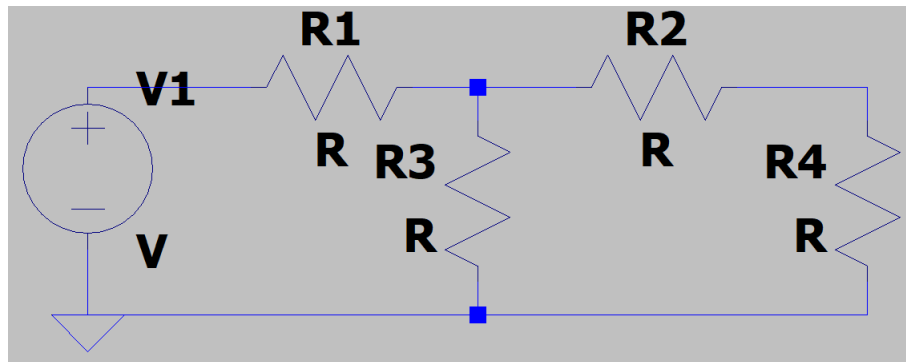


Figure 20: DC Circuit

7.1 Setting up your Sources

First, **right click your voltage source to bring up the DC menu. Input any voltage you want; for this example, you can use 5 V (just type in 5).** Note that we're only using DC voltages here - your .op simulation won't work if you're running an AC signal. That's why we have other sections dedicated to AC analysis!

From there, you can put any value you want for your components. For this example, make them the following:

- $R1 = 10$
- $R2 = 20$
- $R3 = 5$
- $R4 = 35$

If you're following the example, your circuit should now look like the one below:

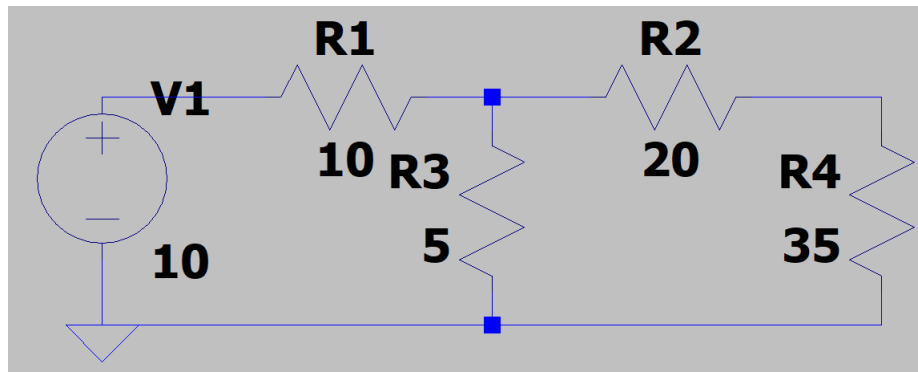


Figure 21: DC Circuit with Values

7.2 Spice Directives

For Mac: right click the screen and click **Draft** and then click **Spice Directive**. Or, you can press **"s"** on your keyboard.

For Windows: you can do the same, or go to the toolbar at the top of your screen and click on the symbol that says **".op"**.

Running a .op simulation is one of the easier simulations you can run on LTSPICE. Once you have the Spice Directive menu pulled up, simply type in **".op"** and click OK. It'll be attached to your mouse and you can place it anywhere you want. All that's left to do is run the simulation!

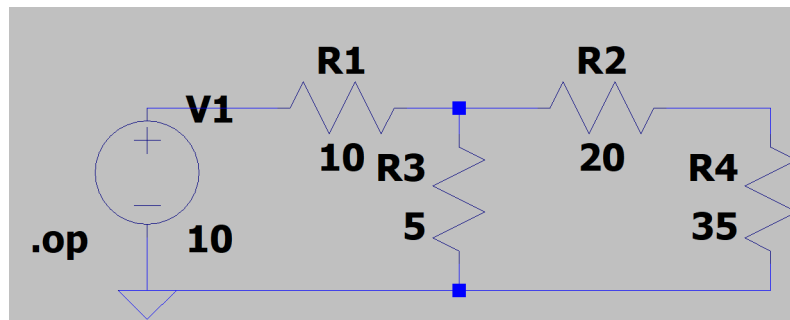


Figure 22: Circuit with .OP Directive

7.3 Running your Simulation

For Mac: right click the screen and click **"Run"**.

For Windows: you can do the same, or go to the toolbar at the top of your screen and click on the symbol that looks like a running figure.

If you did everything correctly, you should see a table similar to the one below. This table includes voltage and current values for each node (connection point) of your circuit.

```

--- Operating Point ---
V(n001) :      10      voltage
V(n002) :    3.14286   voltage
V(n003) :      2      voltage
I(R4) :    -0.0571429  device_current
I(R3) :    -0.628571   device_current
I(R2) :    -0.0571429  device_current
I(R1) :    -0.685714   device_current
I(V1) :    -0.685714   device_current

```

Figure 23: Result of Running .OP Analysis

Notice that you may encounter values that have **negative current**. There's nothing wrong here. **I believe it's just because LTSPICE assumes a passive sign notation for its resistors and you can just take the absolute value of it if you're assuming active sign convention (current from your voltage to the resistor).**

One thing you can also do is see these voltages on your circuit. If you close out of that table (if you want to see it you can always run it again!), you can click twice on the node you're interested in and it should show the voltage at that node! An example is shown below. It's a useful tool for analyzing voltages of your components.

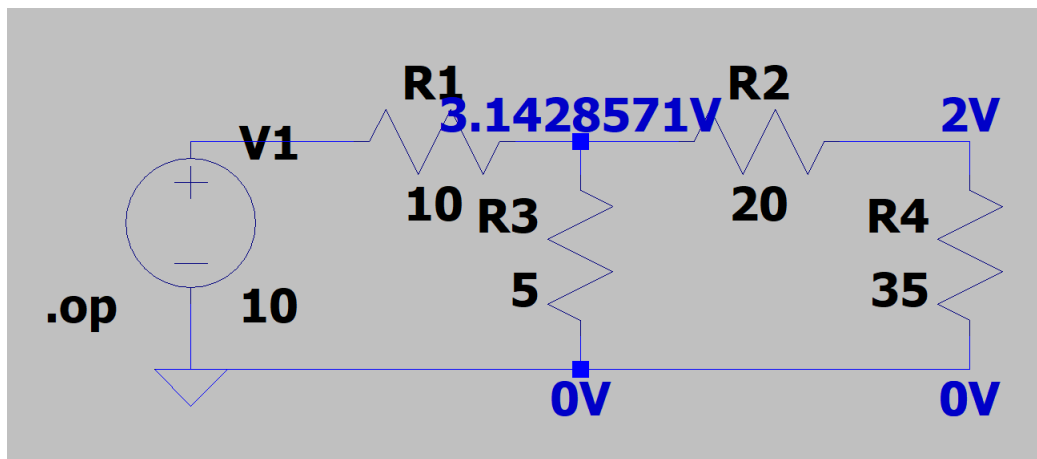


Figure 24: Circuit with .OP Analysis