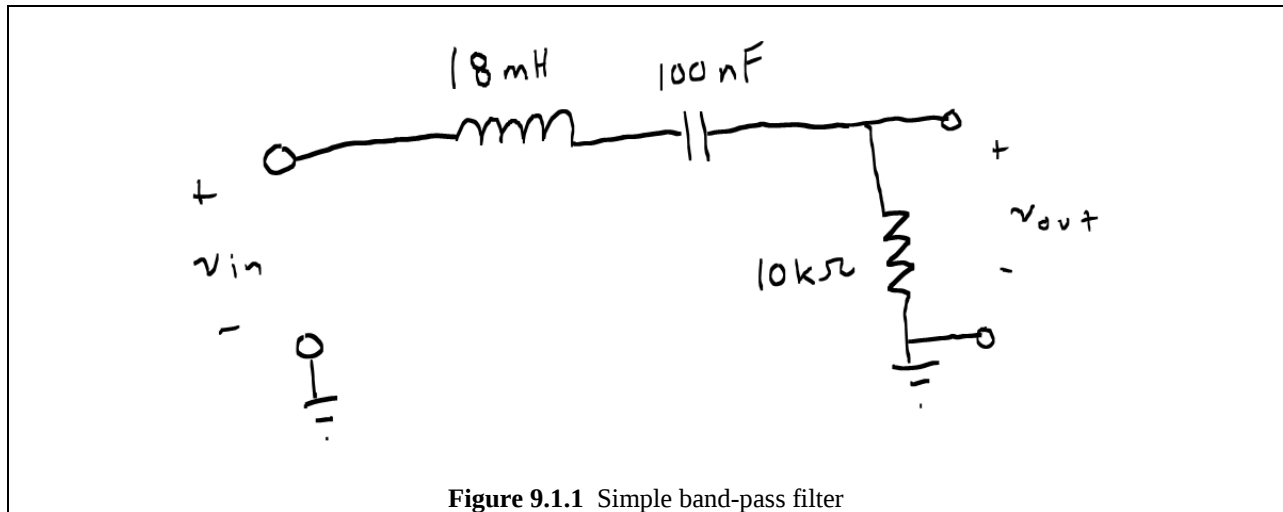


MULTISIM DEMO 9.1: INTRODUCTION TO FILTERS IN MULTISIM

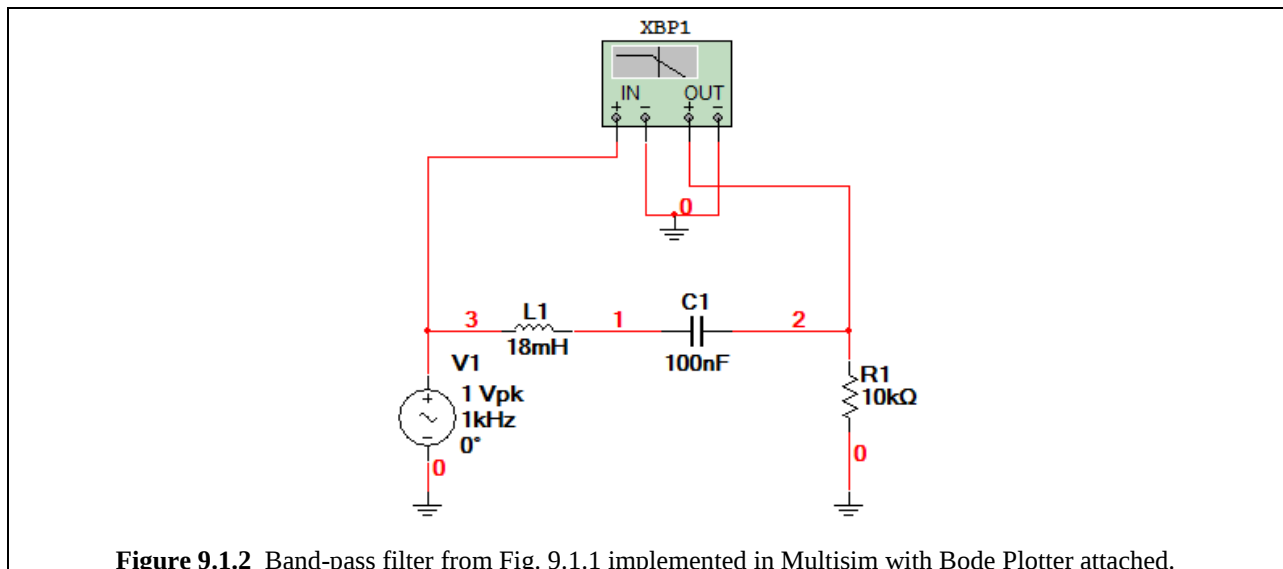
Filters are perhaps one of the most enjoyable things to learn in circuits. Additionally, you can readily see where filters can be applied in the “real” world. Multisim allows for the easy simulation of many different varieties of filters in many different ways.

Bode Plotter

To begin our study of filters here, let’s use some of the tools in Multisim to find the pass band attenuation and the corner frequencies of the band-pass filter shown in Figure 9.1.1.



Multisim actually has an instrument called the Bode Plotter which is used in the Interactive Simulation mode to generate, as its name implies, Bode Plots. This is a great way to easily get a Bode plot although the manipulation capabilities are somewhat limited. So let’s build the circuit shown in Fig. 9.1.2 (including the Bode Plotter Instrument).



Assuming you've got experience with Multisim at this point, the only problem you might have is with finding where the Bode Plotter Instrument is. You can, of course, go to Simulate>Instruments>Bode Plotter, or if you want the short way, click on the  symbol on the Instruments toolbar.

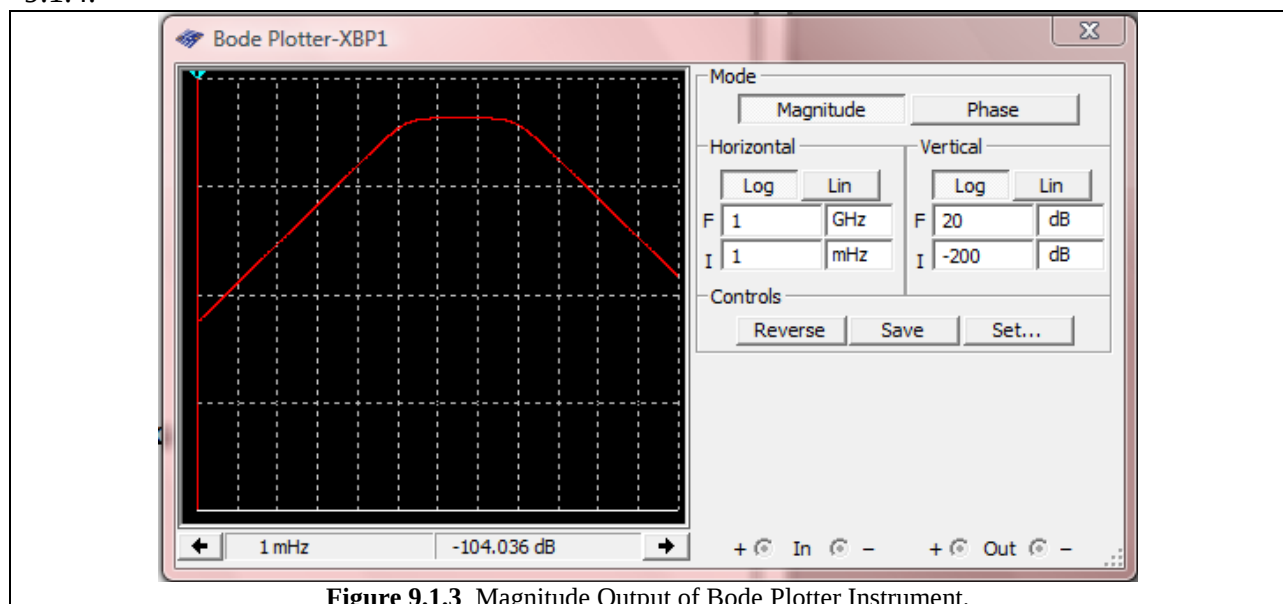
Generally speaking, the “-” terminals of both the IN and OUT should be tied together to give a common ground. Then, you simply need to attach the “+” terminal of IN to the input node of the circuit and the “+” terminal of OUT to the output node of the circuit.

For the Bode Plotter, in order to ensure that you measure everything you need, you must set the horizontal and vertical scales before starting the Interactive Simulation. So for this example:

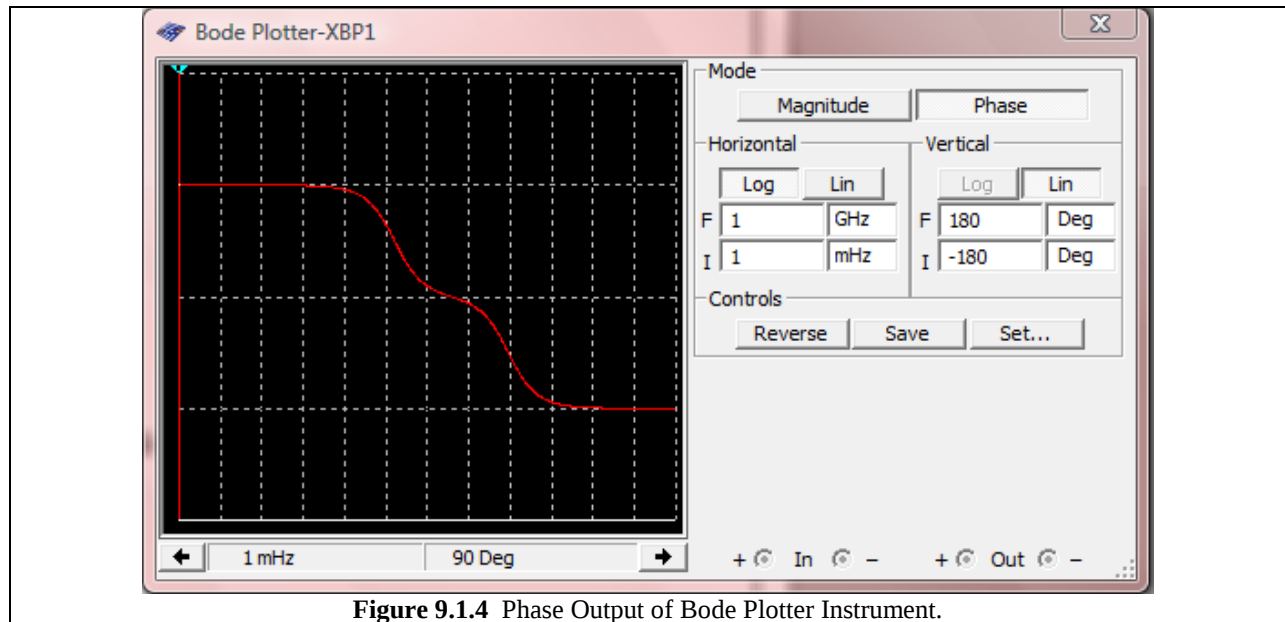
1. Open up the Bode Plotter window by double-clicking on the symbol in the schematic.
2. Set the Mode to Magnitude.
3. Under the Horizontal title, select Log.
4. Set I to 1 mHz (I stands for “Initial”)...yeah that's right we're using milliHertz.
5. Set F to 1 GHz (F stands for “Final”).
6. Under the Vertical title, select Log, and set I to -200 dB and F to 20 dB.
7. Set the Mode to Phase.
8. Under Vertical set I to -180 degrees and F to 180 degrees. (We can't set the vertical scale to log for phase).

Additionally, you can adjust the resolution of the Bode Plotter by clicking on the “Set...” button under the Controls title and adjusting the number of Resolution Points. (The number of Resolution Points is how many data points comprise the entire Bode plot...it is 100 by default but can be anything from 1 to 1000).

Run the Interactive Simulation. When the Mode is set to Magnitude, the screen should look like that in Fig. 9.1.3, and when the screen is set to Phase, the screen should look like that in Fig. 9.1.4.



With the simulation running, you can adjust the frequency range downward, meaning that you can increase I and decrease F, but you cannot expand outward the original bounds set before the Interactive Simulation began. If F was originally set to 10 MHz, you can't change it mid-simulation to 59 MHz, although you could change it to 8 MHz. You need to stop, expand the boundaries and then re-simulate.



Off to the side of the Bode Plotter screen, there is a cursor which you can move around just like in the cursor in the Grapher window. The frequency and magnitude or phase location of the cursor is presented at the bottom of the screen. For incremental movement of the cursor, you can use the left and right pointing arrows on either side of the value readout directly below the screen..

Using the cursors we can find the center frequency of the band-pass filter by right clicking on the cursor and selecting Go to next Y_MAX, keeping in mind which side of the maximum you are on to select which arrow suffix you select.

With the center frequency known, you can move 3 dB downward on both the left and right side of the center frequency to find the corner frequencies. Example values you may obtain are the following:

Center frequency:	3.921 KHz
Lower 3dB frequency:	157.689 Hz
Upper 3 dB frequency:	90.374 KHz

Bode Plots in AC Analysis

We can also do Bode plots using the AC Analysis function in Multisim. To do this, simply open up the AC Analysis window and set the start and stop frequencies to what you want. For the

Selected variables for output, insert an equation expressing the output voltage divided by the input voltage. In the circuit in Fig. 9.1.2, this corresponds to $V(2)/V(3)$.

Additionally, make sure that under the Frequency Parameters tab, you select the Vertical scale to be Decibel!

Simulate to get the output. If you don't like your scales on the Phase or Magnitude plots change them through the Properties menu:

1. Right click on the particular graph, and select Properties.
2. Click on the Left Axis tab.
3. Under the Range Menu, set Min. to -125 and Max. to 25 as shown in Fig. 9.1.5.
4. Click OK

Make sure that you adjust things so that the output of the plot is clear and the critical characteristics such as the pass-band and the cutoff frequencies are visible. We don't want smooshed graphs...those don't tell us anything valuable.

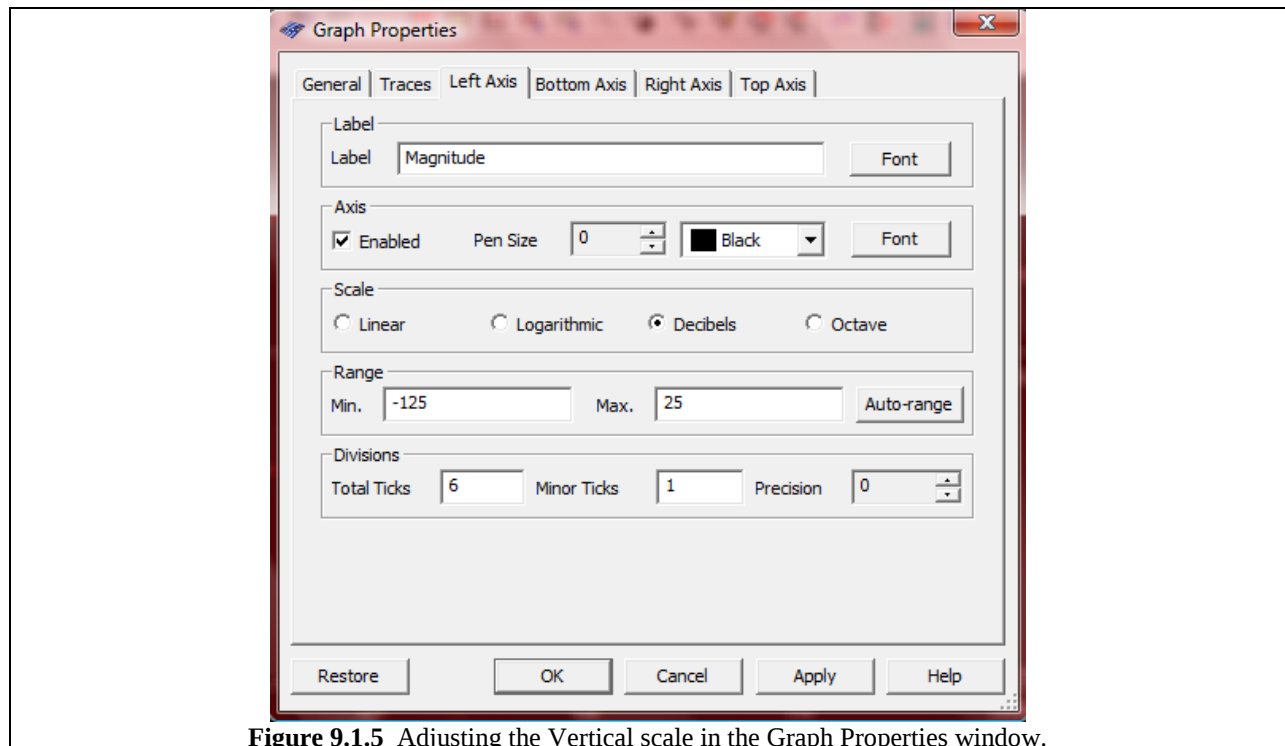


Figure 9.1.5 Adjusting the Vertical scale in the Graph Properties window.

So after we've adjusted things as we see fit, we should have a Bode plot similar to that in Fig. 9.1.6. You can use the cursors to find the center frequency as well as the corner frequencies. Doing so gives:

Center Frequency: 3.9811 kHz
 Lower 3dB frequency: 159.2283 Hz
 Upper 3dB frequency: 90.313 kHz

These values are nice and close to what we obtained with the Bode Plotter Instrument above.

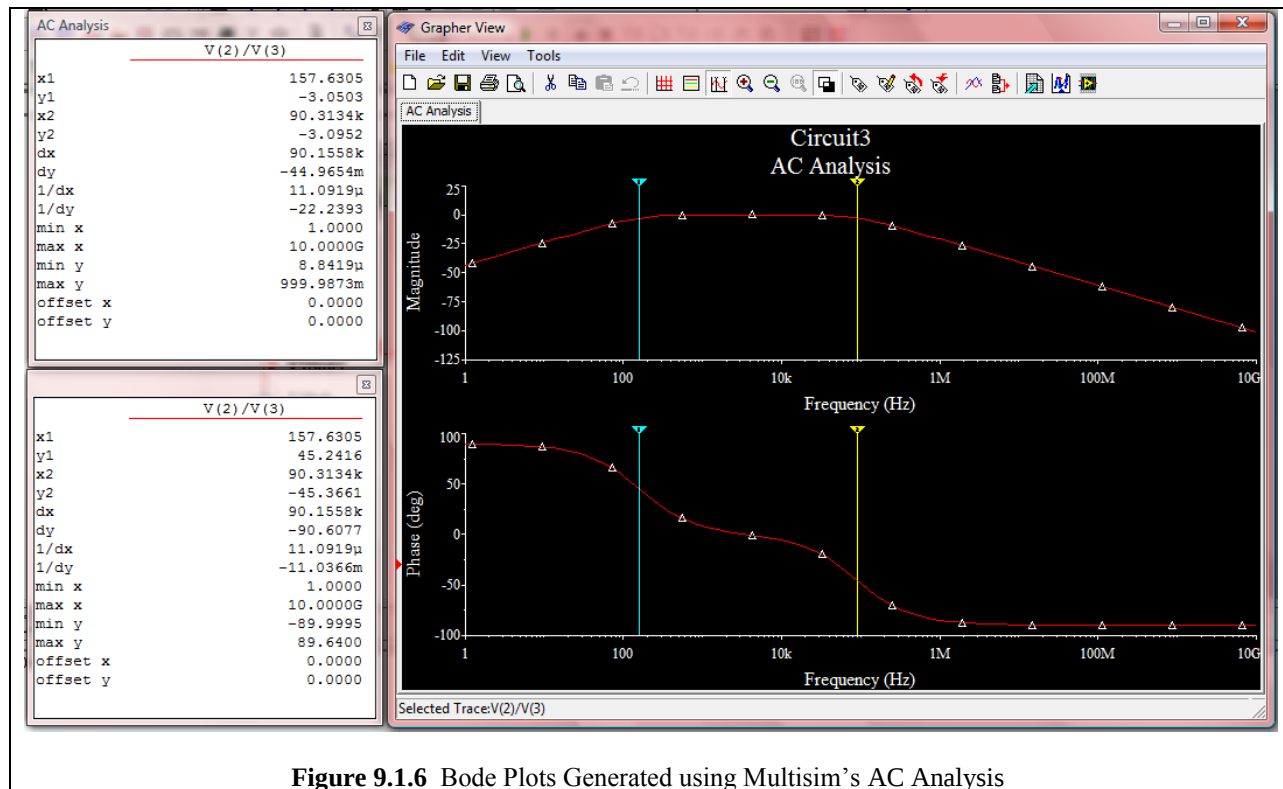


Figure 9.1.6 Bode Plots Generated using Multisim's AC Analysis