APPENDIX C

Using PSpice with Analog Circuits

There are a large number of codes that can be used to simulate circuit performance. We will use the circuit simulation code PSpice.® Students can get a free version of PSpice, at least for some period of time, here: https://www.orcad.com/orcad-academic-program. You can also use PSpice via the University of Maryland's Virtual Computer Lab (VCL): https://eit.umd.edu/vcl. After you log in, click on: "Cadence 17_2 Design Entry CIS. You must have already downloaded the "Citrix Workspace App," otherwise, go here or similar for your operating system): https://www.citrix.com/downloads/workspace-app/windows/workspace-app-for-windows-latest.html.

In this tutorial, we will demonstrate how to simulate three different circuits. It is assumed, as you go through the tutorial, that you are seated in front of a PC that has PSpice, that runs some current version of Microsoft Windows™, and that you are basically familiar with that operating system.

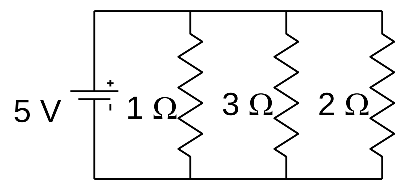


Fig. C.1. A DC circuit.

The first program to simulate is the DC circuit shown in Fig. C.1. From Windows™, open ORCAD Capture CIS (or CIS Lite if it is the free demo version). On the getting started page you can click on "New Project", or from the dropdown menu system click on File → New → Project. The drop-down menu that appears is shown in Fig. C.2. Type "test" for the project name and click on "Analog or Mixed A/D". Type in "C:\temp" for the location and click the "OK" button. Next, the

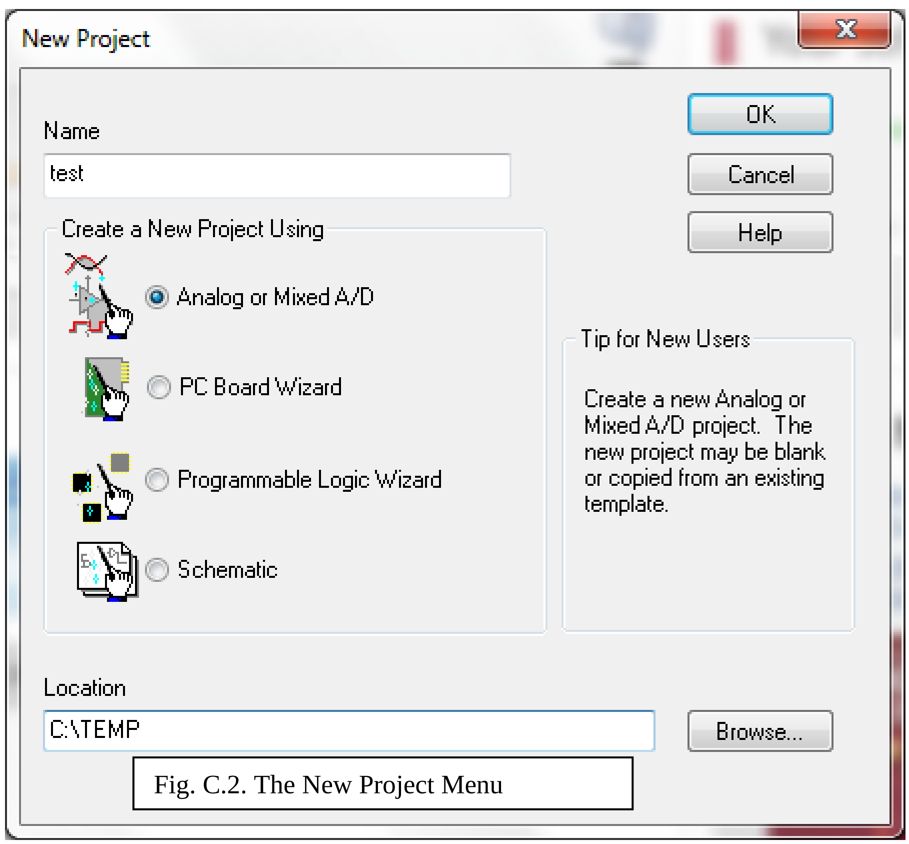


Fig. C.2. The New Project Menu

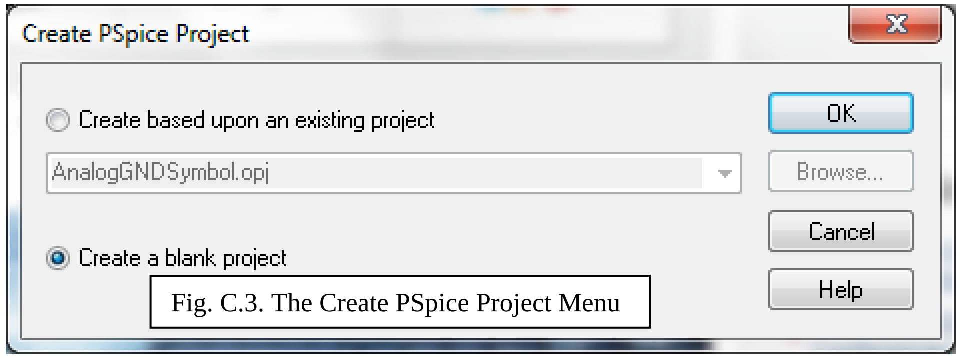


Fig. C.3. The Create PSpice Project Menu

"Create PSpice Project" menu will pop up (see Fig. C.3). click on create a blank project and click "OK".

At this point the OR-CAD Capture CIS drawing environment will open up on the computer screen and we are ready to begin drawing a circuit. Rather than give a comprehensive description of all of the menu items and options accessible on this page, we will take a more directed approach that will allow us to perform the simulation related to the circuit in Fig. C.1.

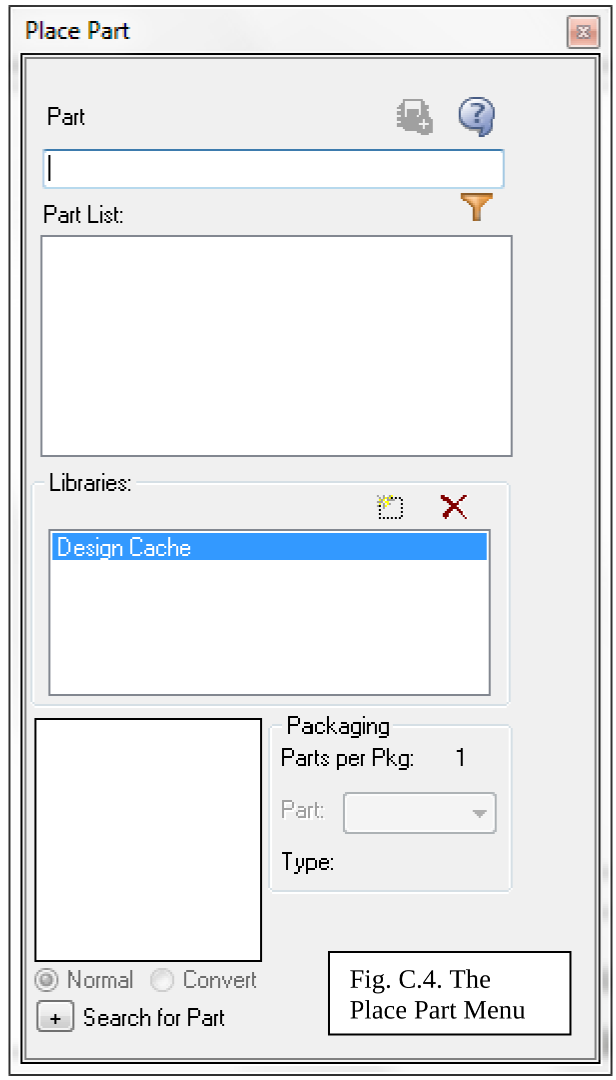


Fig. C.4. The Place Part Menu

We need to place four parts on the schematic drawing: One DC voltage source and three resistors. We need to connect the parts together with wires. PSpice requires that all circuits have grounds defined (zero volts), so we will need to place a ground as well. From the Place → Part drop down menu we will place the four components. After clicking on Place → Part sequence, the menu in Fig. C.4. will appear, normally on the right side of the screen. If the only Library shown is "Design Cache", you will have to add more libraries. Above the box of libraries are two icons. The right icon is a "delete library" icon. To the left is the "add library" icon. Click on the "add library" icon and add all of the available libraries. (Note: You can get other libraries on line from electronic component manufacturers to improve the accuracy of your circuit simulations for specific components.) When you have some libraries available, type "Vdc" in the part box and hit enter. Now the Place Part dialog box will appear as in Fig. C.5.

Move the mouse to the drawing area and right click to place a battery of zero volts, with the name "V1" in the schematic. Right-click the mouse and click on "End Mode" to stop placing batteries (see Fig. C.6.) Move the mouse over the "0V" and double-click the left mouse button. The "Display Properties Menu will appear (Fig. C.7). Change the voltage to 5Vdc and click "OK." You have just added the 5V battery.

Move the mouse over to the part box and replace Vdc with "R" for resistor. Hit enter. Move the mouse over to the drawing area and left click three times, moving the mouse to the right before each new click. Try to place the resistors exactly in the same line – that will make things a little easier later if you can do that. Right click and then select End Mode to stop placing resistors. The three resistors are incorrectly rotated by 90 degrees and have a value of 1kΩ each.

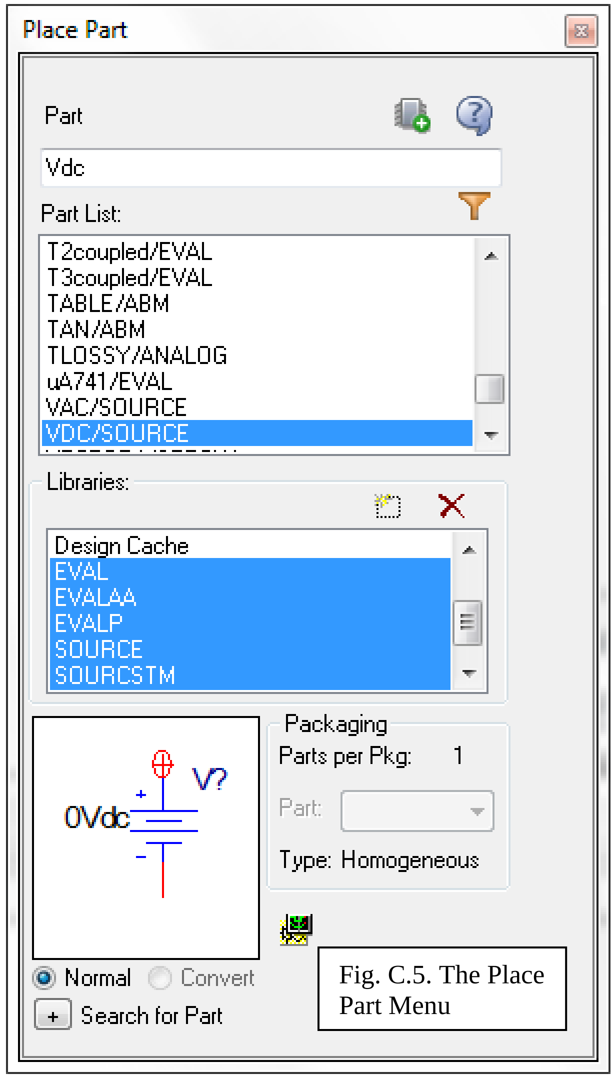


Fig. C.5. The Place Part Menu

The third resistor R3 should be highlighted (it is pink while the other two resistors are blue). Click on the drop down menu Edit → Rotate. The third resistor should now be parallel to the battery. Double-click on the resistance value and change it to "2" in the Display Properties box that appears. No units are necessary. If you had needed to include an SI prefix (p. n, u, m, k, MEG), there would be no space allowed between the component value and the prefix. Move the mouse over R1 and left-click. Repeat the procedures above to rotate the resistor and change the resistance to 1Ω. Repeat the procedure for the middle resistor, giving it a value of 3Ω. At this point you have placed all four components.

Next, the components must be connected by wires. Use the drop-down menu: Place → Wire. Cross-hairs will appear when you move the cursor over the schematic. There are many common errors that prevent PSpice from working correctly, and placing the wires incorrectly is a particularly frustrating error, so take care with this process.

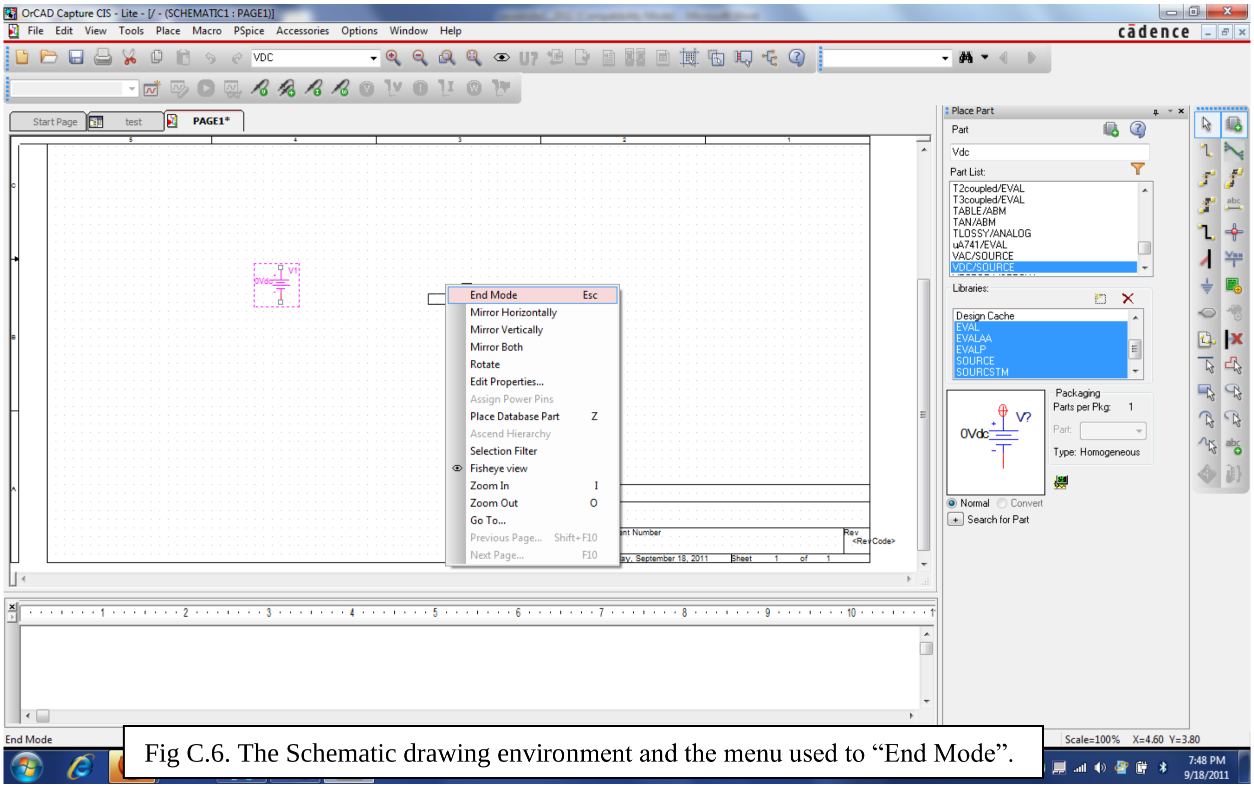


Fig C.6. The Schematic drawing environment and the menu used to "End Mode".

Place the cross-hair directly over the small square box at the top of the 5V battery (this square represents one of the two terminals of the battery (the positive terminal in this case) and left click the mouse. Move the cursor over to the right-most resistor. If you have lined up the resistors correctly, there should appear three red circles. These circles indicate that the wire will connect to the terminals of these components once you left-click the mouse. This is exactly what we want to do, so left-click again, and a wire should appear connecting one terminal of each of the four components together. If some reason the resistors are not aligned, you will have to repeat this process. Once you place one

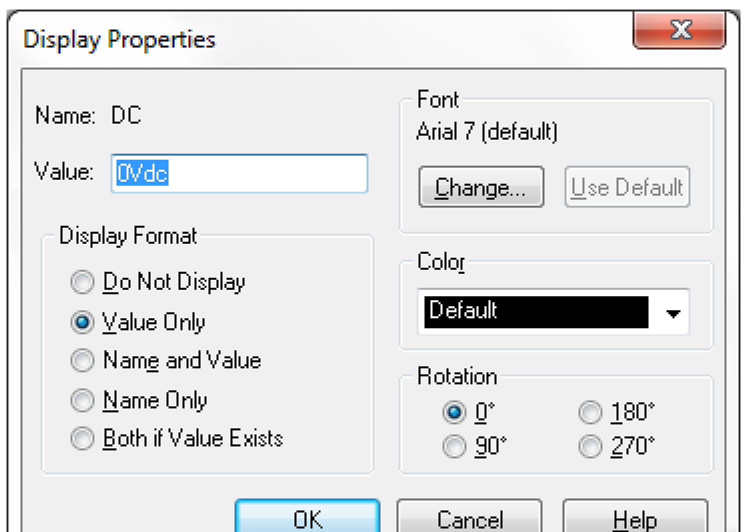


Fig. C.7. Display Properties Menu

wire, you remain in the place wire mode, so just repeat the two-click sequence to place the next wire. When you finish making the top node, make the bottom node by placing a wire between the lower terminals of all four components. Right click and select "End Mode" to stop placing wires.

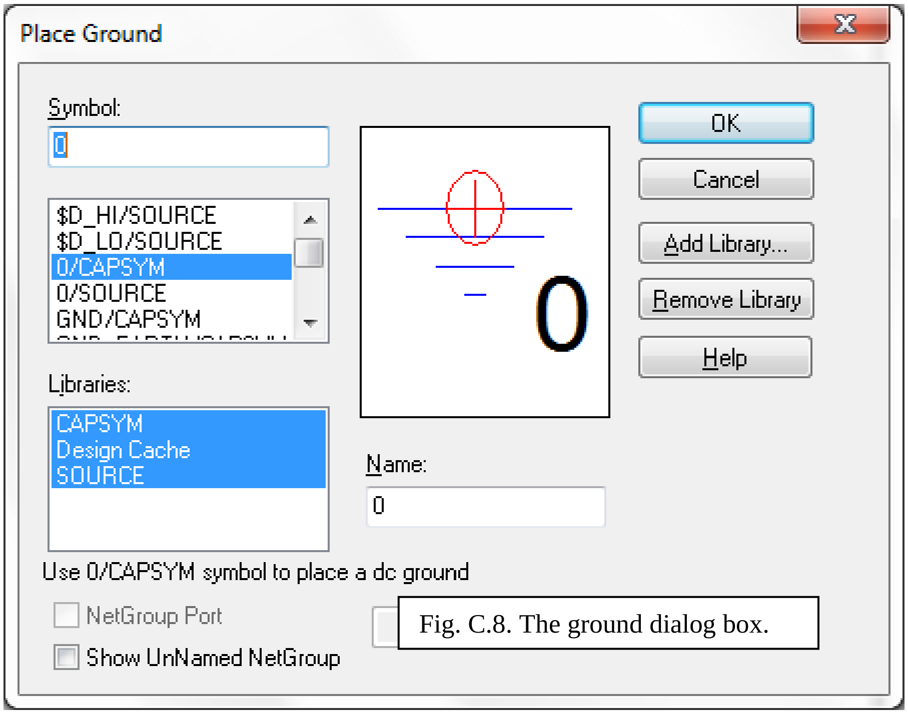


Fig. C.8. The ground dialog box.

The final component that we must add is the ground connection. Use the drop down sequence Place → Ground to produce the menu shown in Fig. C.8. Type in "0" to get the ground we need in press "OK". Place this ground anywhere along the bottom node with a left-click, and end mode with a right-click. Make cure the red circle appears to ensure that you are making a good connection to the node. We are now done drawing the circuit; you may want to save your work with File → Save or by pressing the Save Icon (the picture of the obsolete floppy disk).

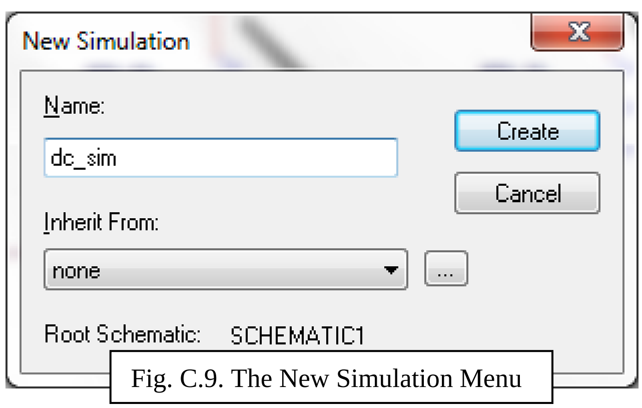


Fig. C.9. The New Simulation Menu

The next step is to tell PSpice what type of simulation we want to do, so we need to use the drop-down sequence PSpice → New Simulation Profile. In the "New Simulation" dialog box that appears. Type in any name that you wish and press the "Create" button (see Fig. C.9.).

To make the analysis part a little easier, we can indicate where we would like to measure circuit variables. On the lowest toolbar on the upper left part of the screen, there is a picture of different oscilloscope probes. From left-to-right, they are for measuring: voltages (with respect to ground), voltage differences, currents, and dissipated powers. Click on the current probe icon. Move the cursor so that the icon tip is touching the upper terminal of the voltage source and left-click. Place current probe at the upper terminals of all three resistors too. Then right-click to end mode. The circuit should look like the one in Fig. C.10. We have zoomed in on the circuit and moved the location of the resistor names and component values to make the circuit easier to see, so the drawing will not look just like yours unless you do the same.

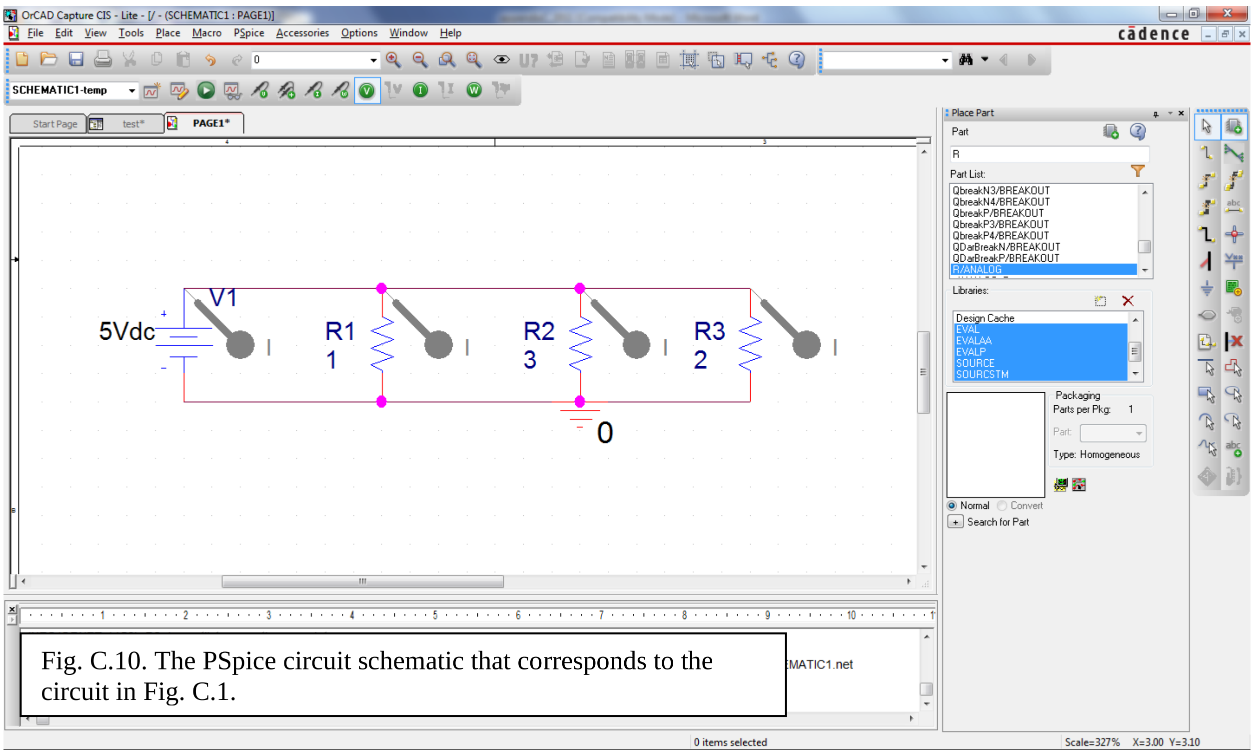


Fig. C.10. The PSpice circuit schematic that corresponds to the circuit in Fig. C.1.

Normally, one would have to edit the simulation profile, but since this is a simple DC analysis, we can just use the drop-down sequence PSpice → Run to initiate the simulation. A warning box will appear saying that you will not be able to "undo" anything after running the simulation. Push the "yes" button to proceed. The simulation will complete if there are no errors, and the probe program will launch. The current probes on the circuit schematic will change colors to indicate the color each measurement has in the probe window. The node voltages that correspond to the DC "bias point" will appear on the schematic – for this circuit the two nodes are at ground (0 volts) and 5 volts. Before we look at the simulation output, note that creating, editing, and viewing the simulation profiles, as well as initiating the PSpice simulation can also be done via the four icon buttons that are to the left of the four probe icons on the toolbar.

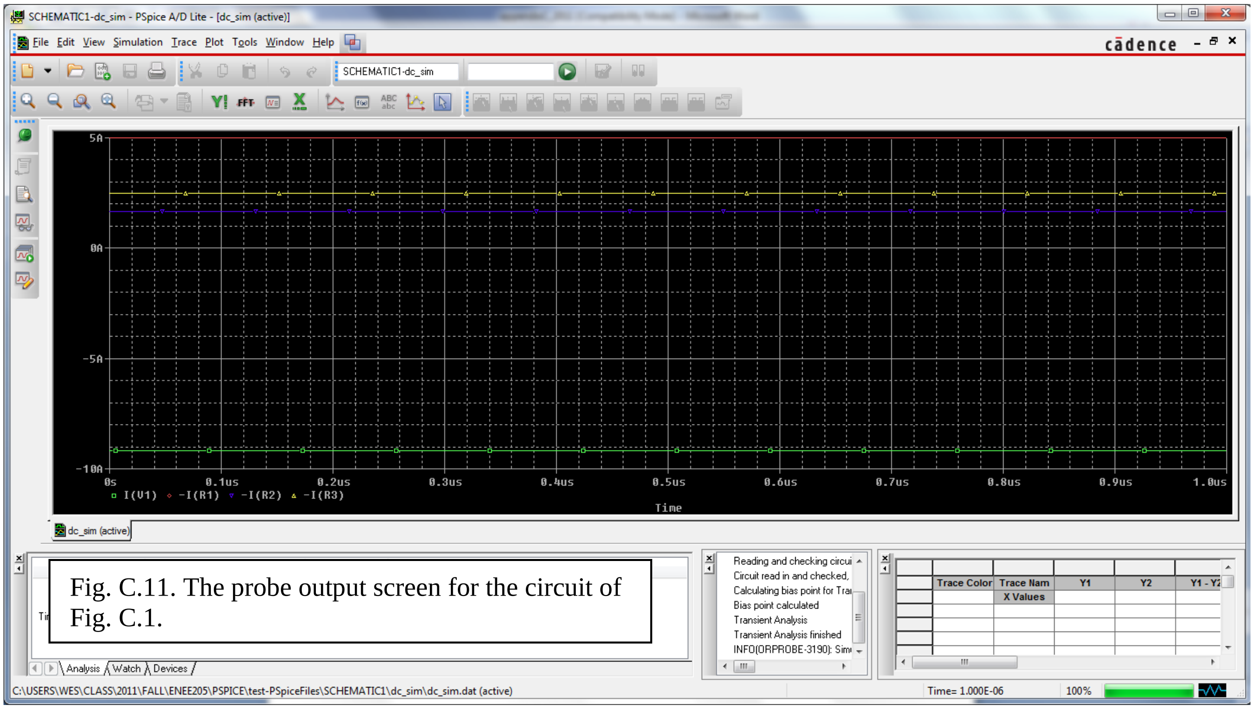


Fig. C.11. The probe output screen for the circuit of Fig. C.1.

The Probe output is shown in Fig. C.11. The results are plotted for 1 µs, but it is a DC circuit, so nothing varies with time. Note that the sum of all four currents is zero. This is a consequence of KVL, the way that we placed the components, and PSpice, which uses the passive sign convention for all components, including sources. Thus, the battery is supplying power to the circuit, since the current going into the positive terminal is negative.

Note that after the simulation is complete, in the schematics window, the three icons to the right of the probes are available and can be pressed to show (or hide) all node voltages, all component currents, and all component powers. Note the battery power displayed is negative, another consequence of using the passive sign convention for the battery.

The next circuit that we will model with PSpice is shown in Fig. C.12. While we could add this circuit to the current schematic, we will exit PSpice and enter again to start from scratch. Fewer steps will be presented in this part of the tutorial, as we will highlight only the parts that are new to this simulation.

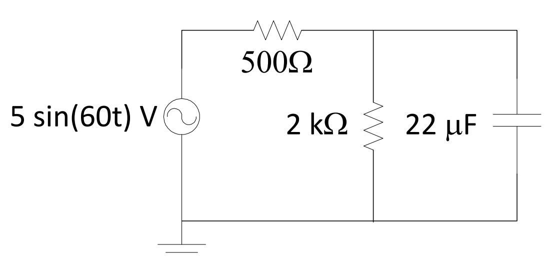


Fig. C. 12 The second circuit to simulate.

There are many types of voltage sources. We are going to perform a transient analysis with this circuit, so we will use one called VSIN, which is located in the SOURCE library. VSIN has four parameters that can be adjusted and are listed to the left of the source symbol. The formula for the voltage source is:

$$V(t) = VOFF + VAMPL*SIN(2\pi*FREQ*t);$$

so VOFF is an offset dc voltage, VAMPL is the peak sine wave voltage, and FREQ is the sine wave frequency. The fourth parameter, AC, is for a frequency analysis and we do not need to give a value for our analysis. Put VOFF=0V, VAMPL=5V, and FREQ=60/2π ≈ 9.55.

Place the resistors and the capacitor as indicated in the figure. The name for a capacitor is just "C" and it is in ANALOG library. Add the zero ground and connect all components with wires. The circuit should appear as in Fig. C. 13. The figure indicates that we will look for the voltages of the source and across the capacitor.

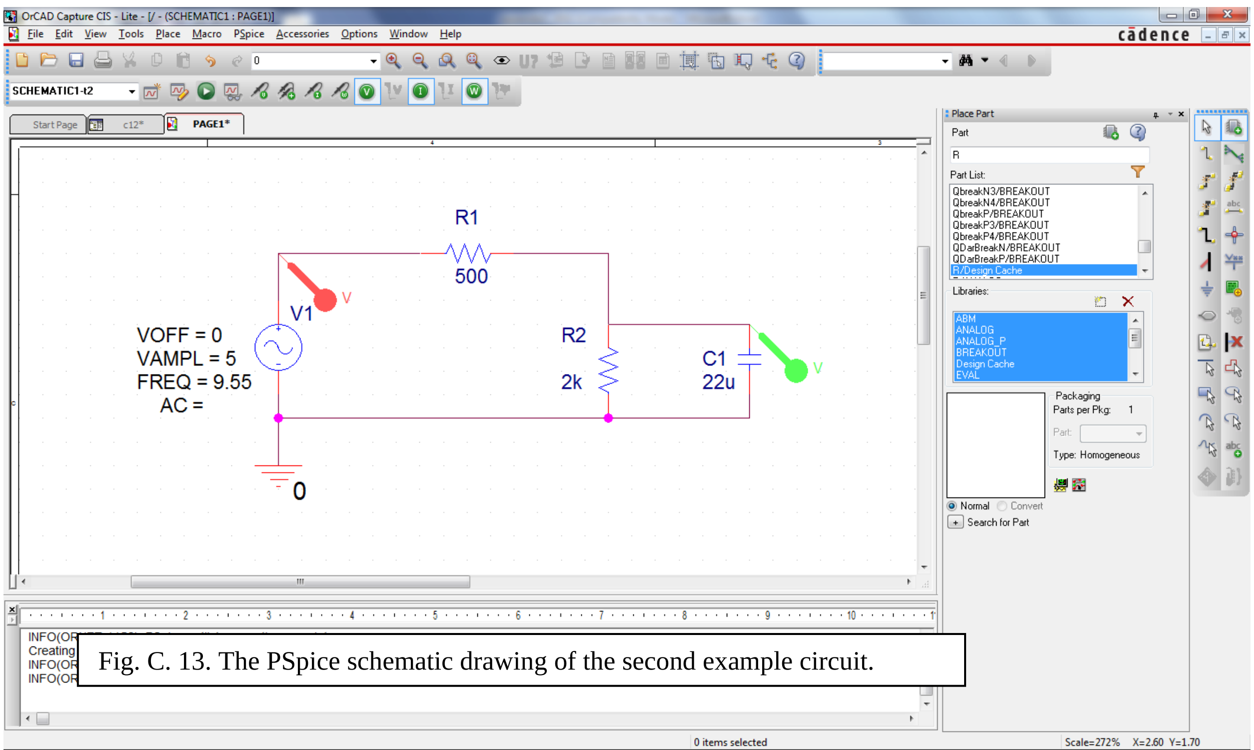


Fig. C. 13. The PSpice schematic drawing of the second example circuit.

Create a new simulation profile and edit it. Change the "Run to time" to 250 ms (as in Fig. C.14. and click on the "Skip the initial transient bias point calculation" button. This will force all voltages in the circuit to start from zero (except for the voltage source). Press o.k. Add the two voltage probes as indicated in Fig. C. 13 and run the simulation. The probe output should look as in Fig. C. 15.

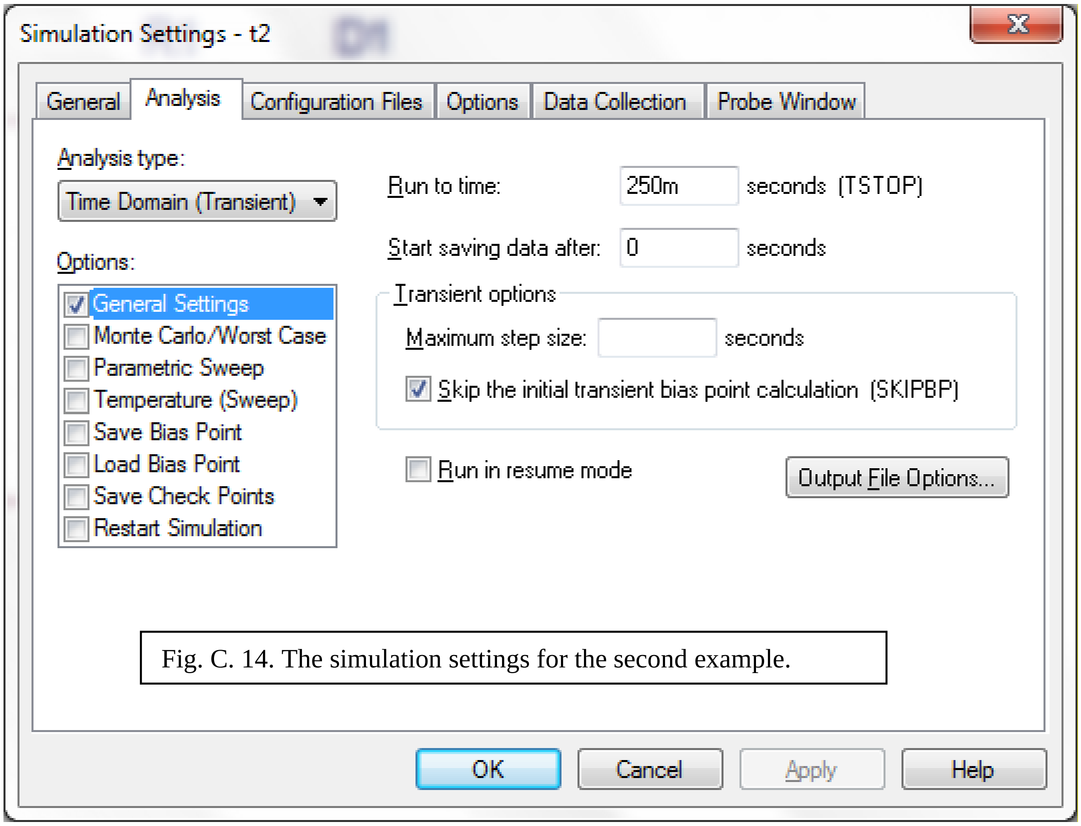


Fig. C. 14. The simulation settings for the second example.

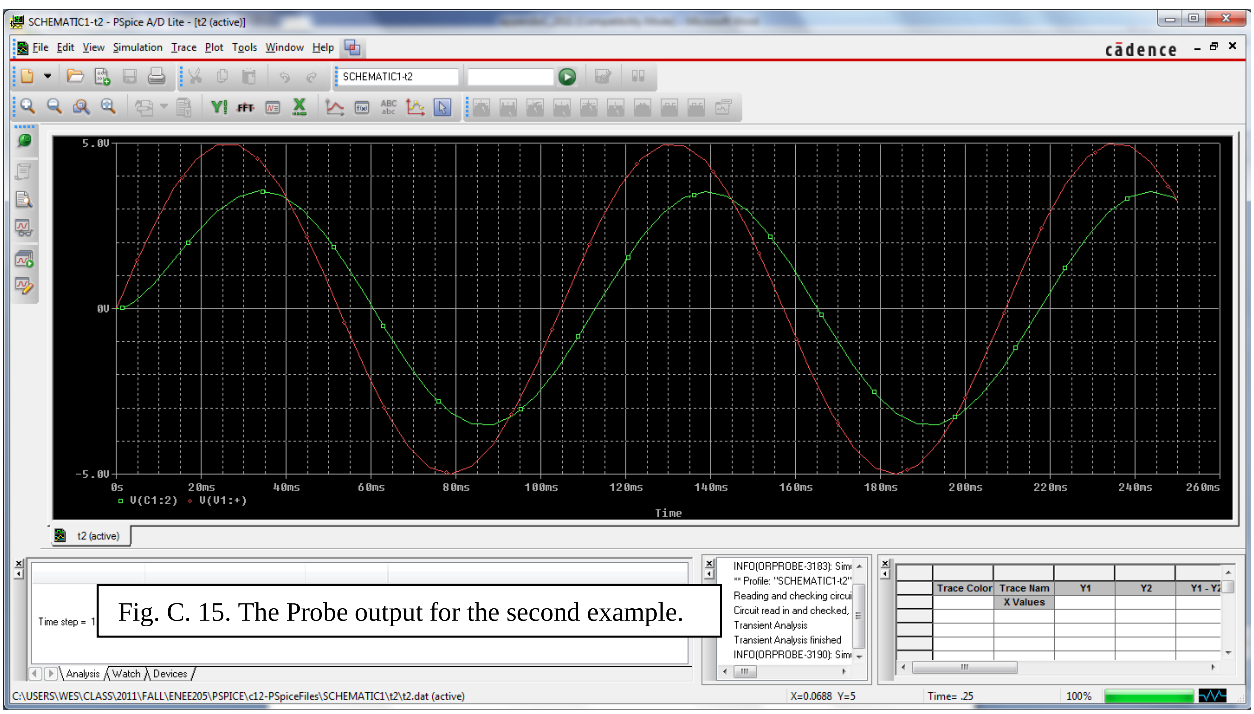
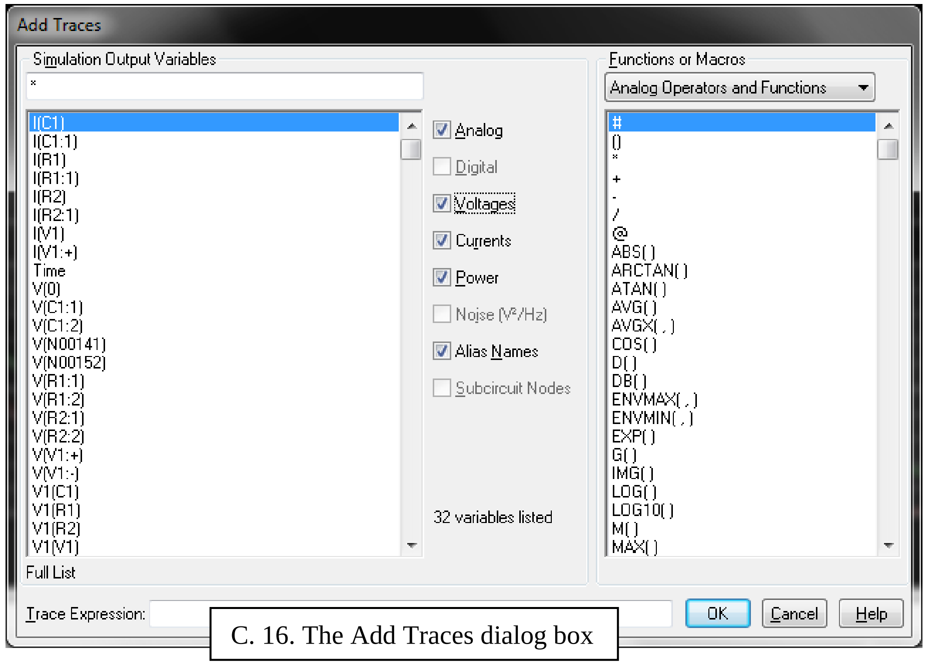


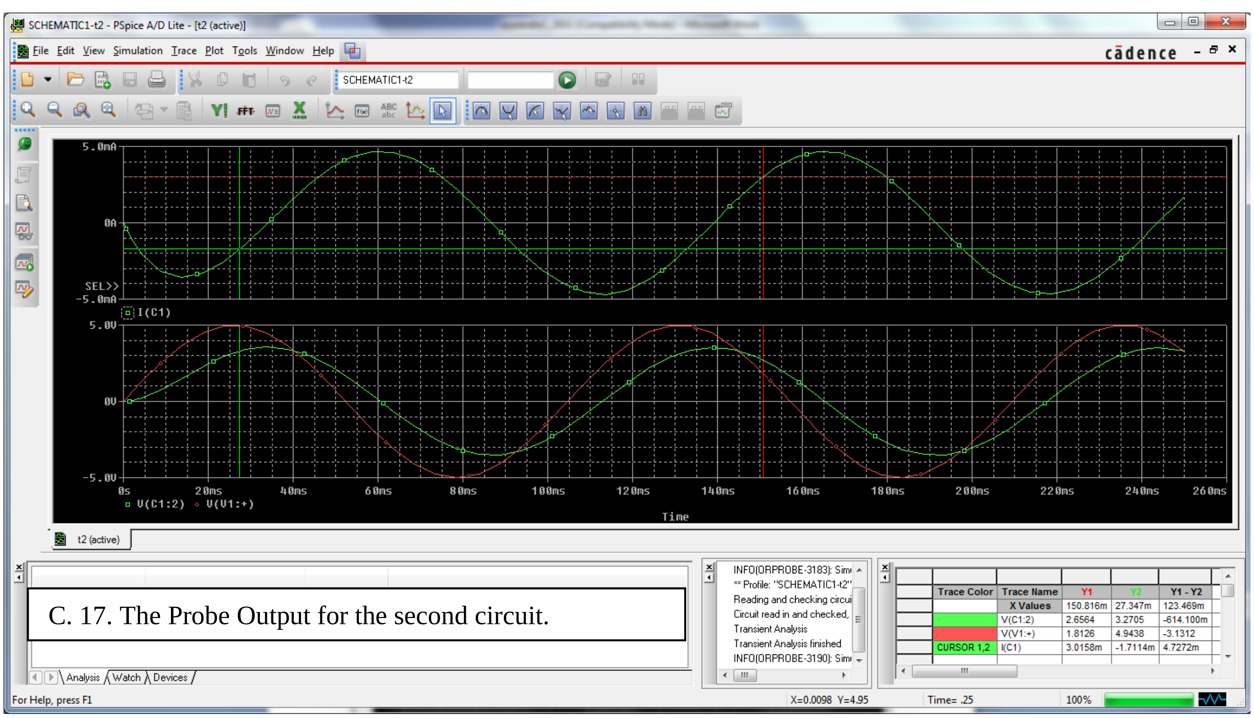
Fig. C. 15. The Probe output for the second example.

The probe program itself can be used to glean much information from the circuit, beyond what the initial circuit probes ask it to do. As an example, let's plot the current through the capacitor. In probe, use the drop-down menu Plot → Add plot to window. A new plot will appear above the original plot, and there will be a SEL>> to the left of the new plot. This indicates that we have selected this new plot for now, and are ready to make changes to it. Now use the drop-down menu Trace → Add trace. The add traces dialog box appears as in Fig. C. 16.



C. 16. The Add Traces dialog box

Click on "Voltages" and "Power" to remove the check marks and reduce the number of variable in the list. Click on I(C1) to plot the current through the capacitor. Finally, click on Trace → Cursor → Display. Left and right-click at various points on the plot and see how the cursor variables change in the box in the lower right corner of the probe window. The Probe output should now appear as in Fig. C. 17.



C. 17. The Probe Output for the second circuit.

The circuit for the final example is shown in Fig. C. 18. Start from scratch in PSpice to build the circuit. The name for an inductor is “L”. Use the VAC source, and leave the default values (1V maximum for an “AC” steady-state analysis.) The goal of this example will be to see how the voltage magnitude across each passive component changes with input source frequency.

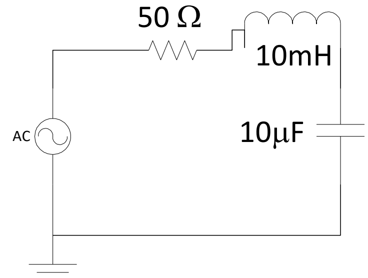


Fig. C. 18. The final example circuit.

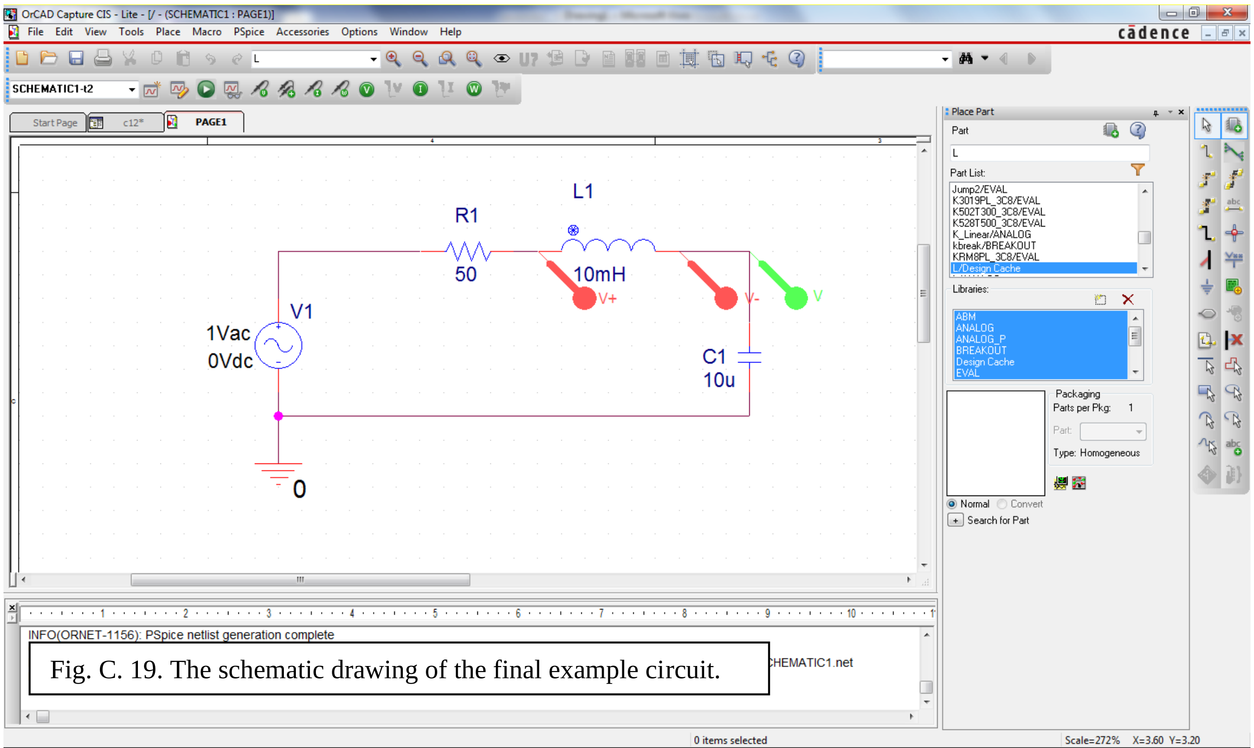


Fig. C. 19. The schematic drawing of the final example circuit.

The circuit as drawn in the schematic editor is shown in Fig. C. 19. Since one side of the capacitor is connected to ground, we can find the voltage from a single probe. To find the inductor voltage, we need to use differential voltage probes. When the circuit is drawn, create and edit a simulation profile. You need to select the analysis type; we will do an AC Sweep (see Fig. C. 20). We need to choose the frequency range – we will go from 10 Hz to 10,000 Hz. We also need to choose the number of frequencies to be simulated. Since this is such a large frequency range, we will use a logarithmic placement of frequencies, and plot 20 data points / decade.

The probe output is shown in Fig. C. 21. Note that the voltage across the resistor was added after Probe was initiated.

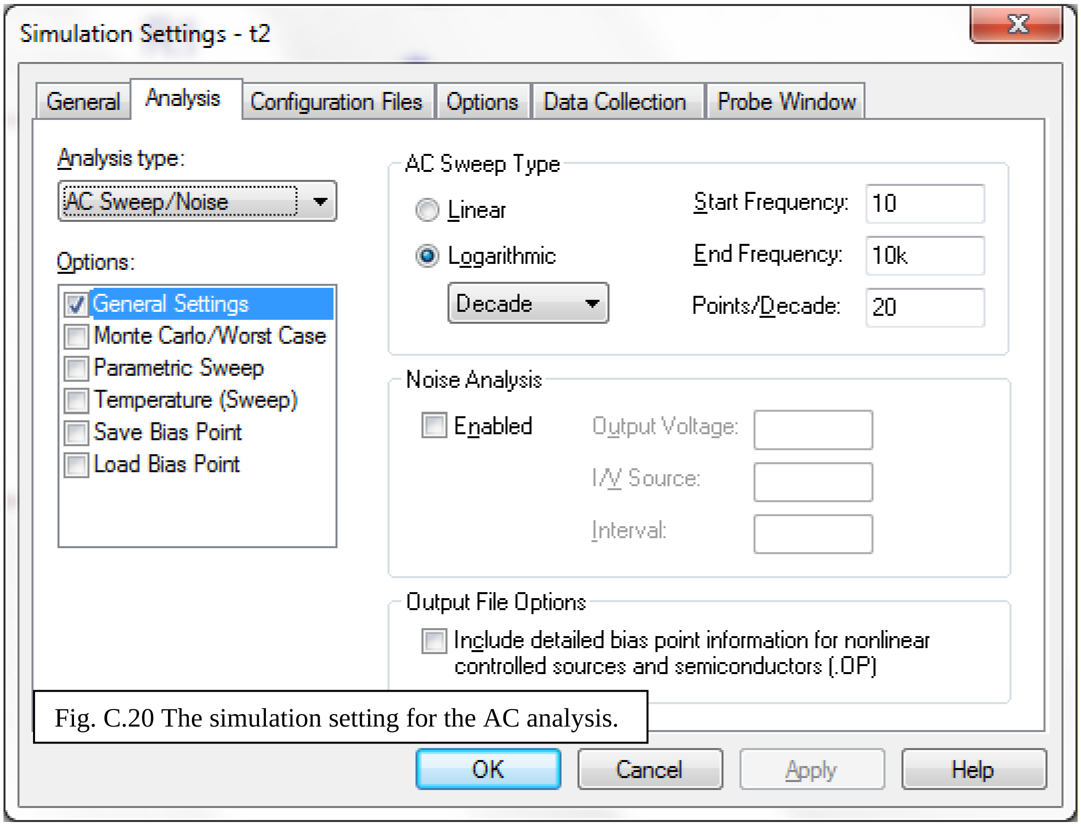


Fig. C.20 The simulation setting for the AC analysis.

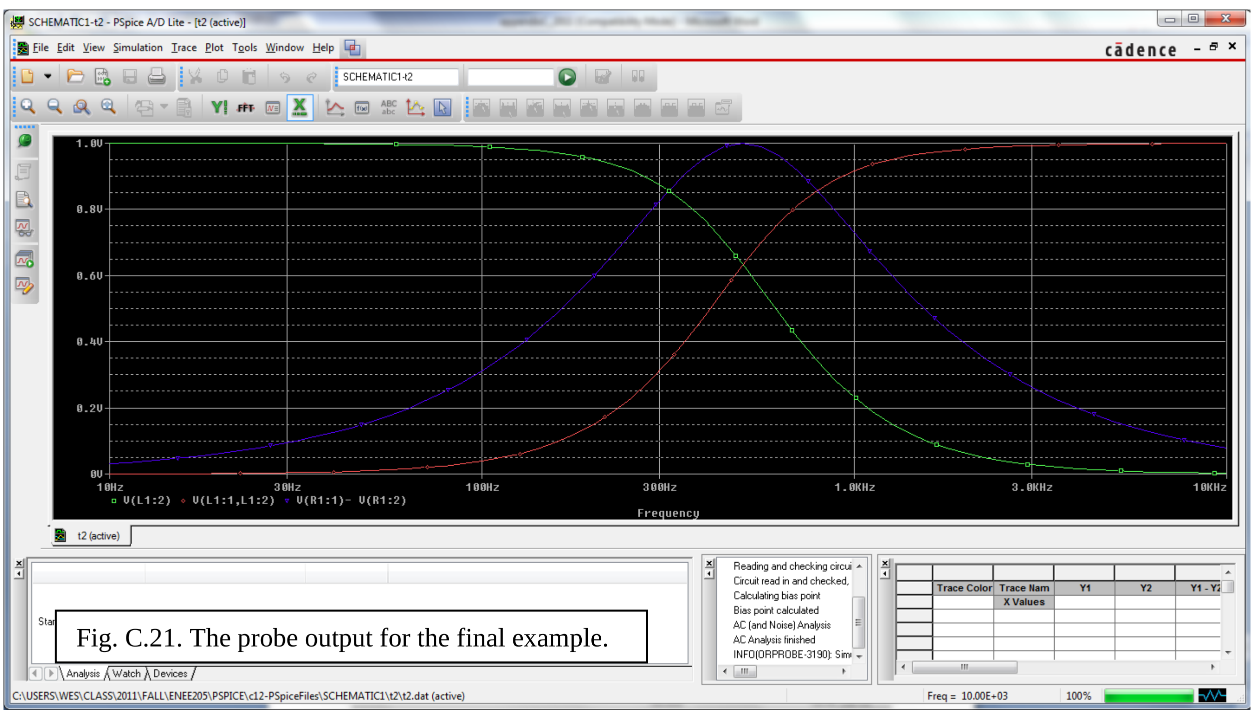


Fig. C.21. The probe output for the final example.