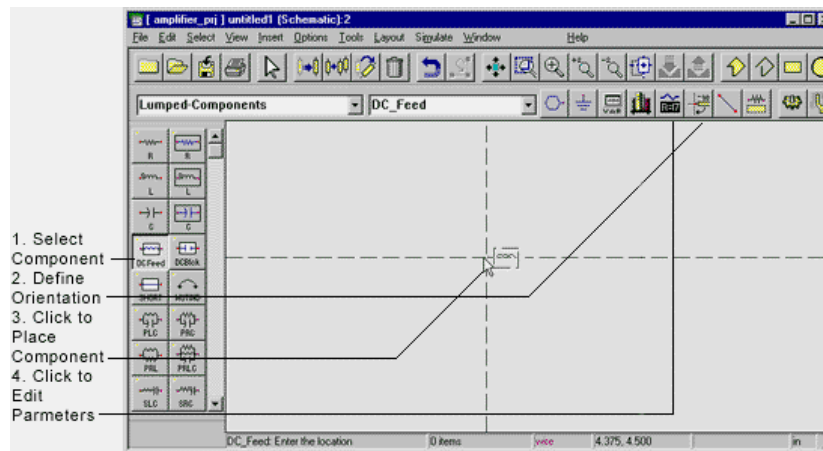


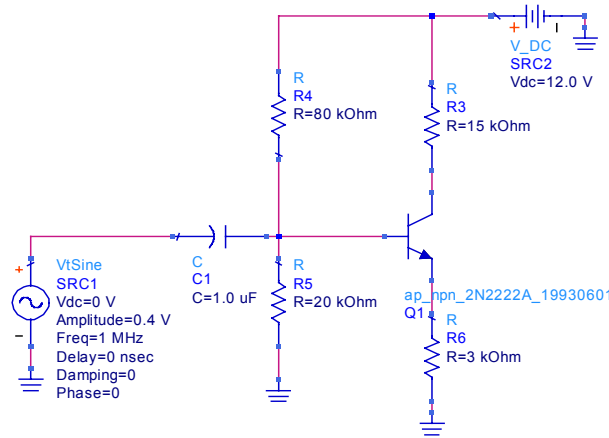
An Example Circuit Constructed and Simulated Using ADS v1.5

1. First read the Quick Tour for ADS at <http://eesof.tm.agilent.com/docs/adsd15/quicktour.html>.
2. Open ADS and select a name for your project. The project will contain your circuits (called “schematics”), data, plots and other information.
3. Add the components to your circuit. This is a three-step process and varies depending on the type of components you are adding.

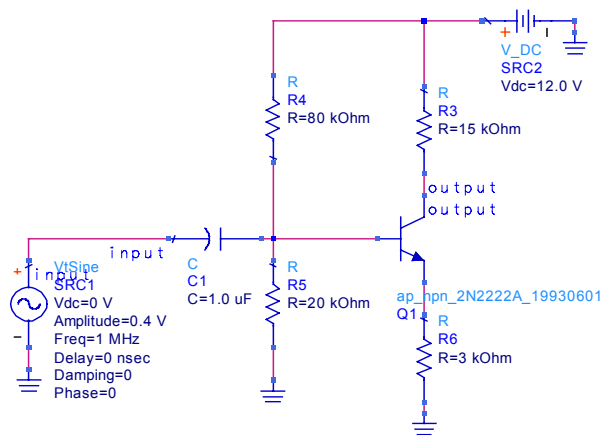


If you are adding generic components (resistors, capacitors, etc.), then first, select the family of components in the “Component Palette List” (the drop down list in the upper left corner of the schematic window). Then click on the desired component that appears below. Next, click in the schematic at the location you wish to place the component. Finally, hit Esc to deselect the process of adding this component.

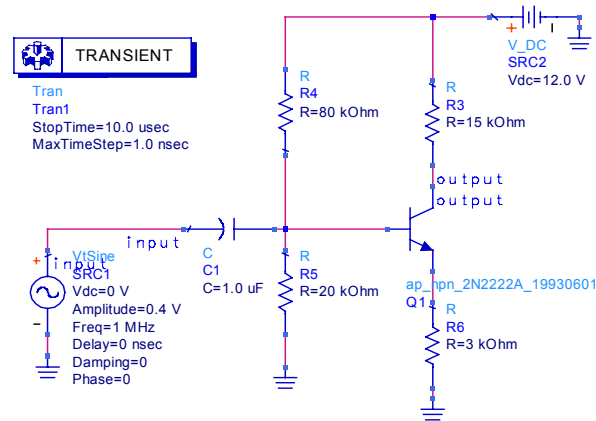
If you are adding a specific component such as a specific diode or transistor, then select Insert > Component > Component Library ... There are many families of libraries you can choose from, but the components you are interested in are probably located in the Analog Parts Library. Select a sub-family of parts in this library (for example, “AP Diodes (No Layout)”) and an extensive list of diodes will appear component window. Click on the component you want to use, then move back to the schematic window and click there to insert the component. Hit Esc to discontinue this insertion process.



4. After you have added all of your components and sources, you will need to name the nodes that you wish to measure the voltage. Click on Name icon near the top of the schematic window, enter a name for the node and then click on the node. Your name will now appear on the schematic.



5. Before you can begin the simulation process, you need to add a Simulation Controller to the schematic. In the above circuit, we will perform a transient analysis, so we'll add the Transient controller. To add the controller, click on Simulation-Transient in the Component Palette List. The Transient Simulation Controller is the icon labeled Trans that shows a wheel. Click on this icon, then click again in the schematic. You should edit the stop time to a few periods of your waveform.



- To run the simulation, select **Simulate** > **Simulate** from the menu. Once the simulation has finished, you can view a plot of your results by clicking on the rectangular plot icon on the left-hand side of the plotting window that pops up when the simulation is completed. Then click in the plotting area. A dialog box will appear. Add the nodes for which you want voltages plotted (“input” and “output” in this example) and clicking the >>Add>> button. Your plot will appear after you select Done.

