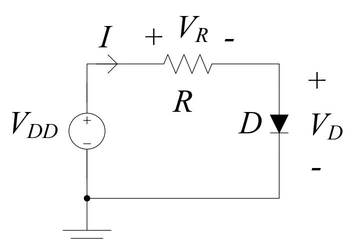
Lecture 5: Introduction to *B2 Spice* from Beige Bag Software.

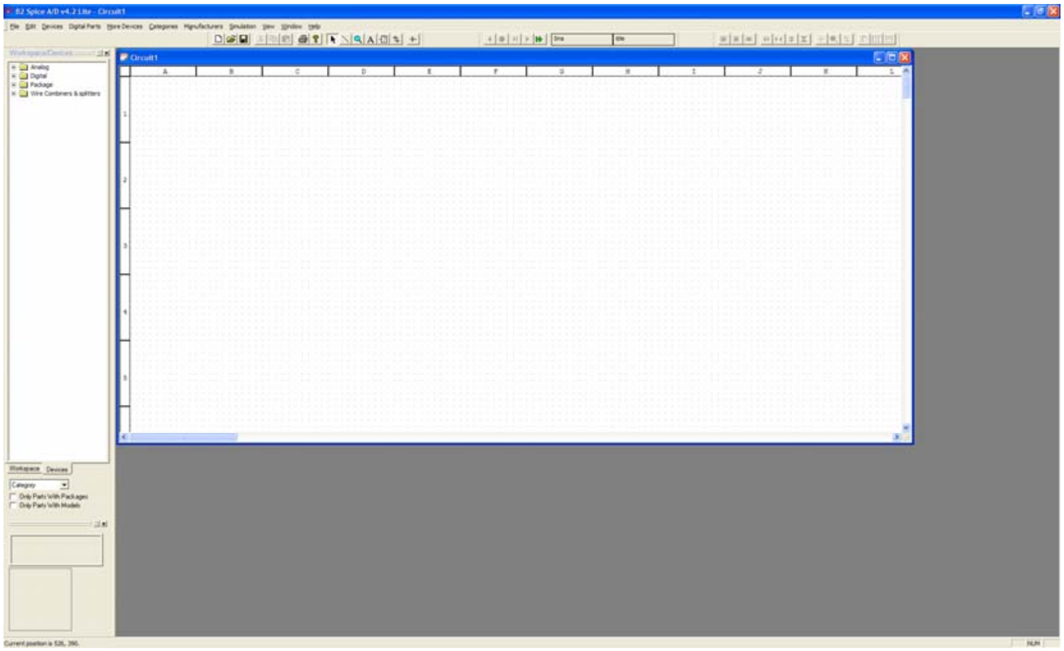
There are many, many circuit simulation packages available today. SPICE and PSPICE are two common examples. In this course, we will be using *B2 Spice* from Beige Bag Software. A free version, B2 Spice A/D V4 Lite, is available for you to install on your computer.

B2 Spice Simulation of a Simple DC Diode Circuit

To illustrate the use of *B2 Spice*, we will analyze the simple series resistor and diode circuit considered earlier in Lecture 3:

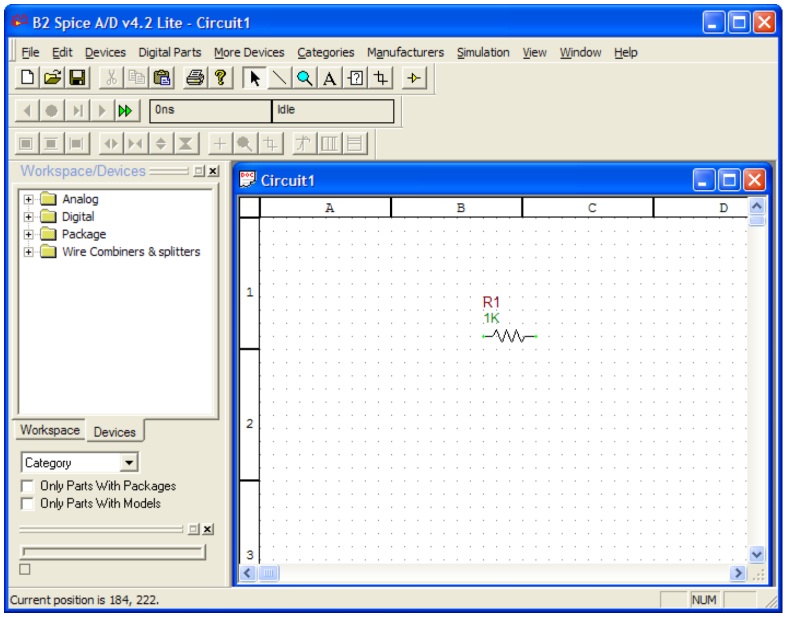


B2 Spice startup window:



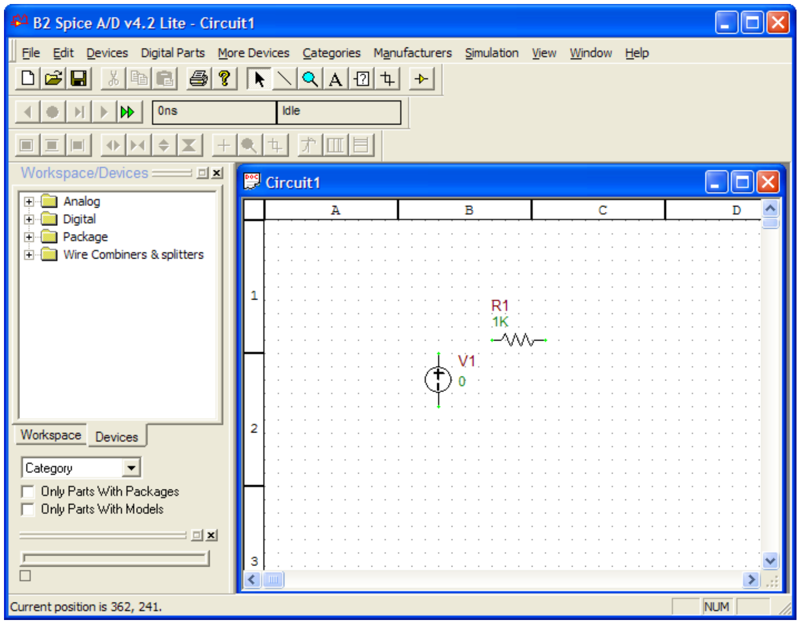
The circuit is assembled in the schematic editor shown above. You graphically add electrical parts to your circuit by choosing them from the menus, then draw wires to interconnect these devices. You can adjust the properties of these devices by double clicking the component once it's placed in the schematic.

We'll first place a resistor. Select **Devices | Resistor (simple) (R)** then click on the schematic to place the resistor:

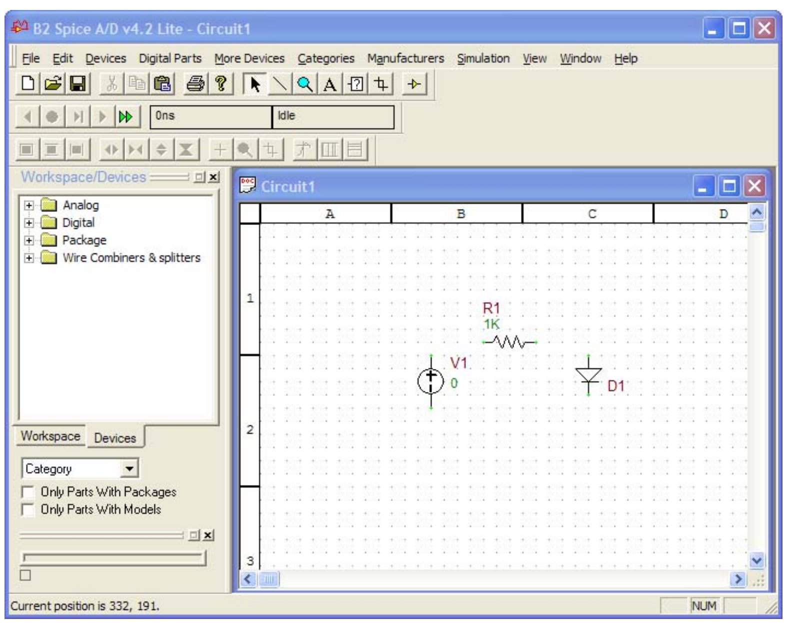


You can double click the component to change its values, such as the resistance of the resistor in this case. However, the default value of 1 kΩ is the value we need here.

To add the DC voltage source, we select **Devices | Voltage Source (V)** then click once to place the voltage source on the schematic. Selecting Ctrl + r once rotates the source vertically giving:

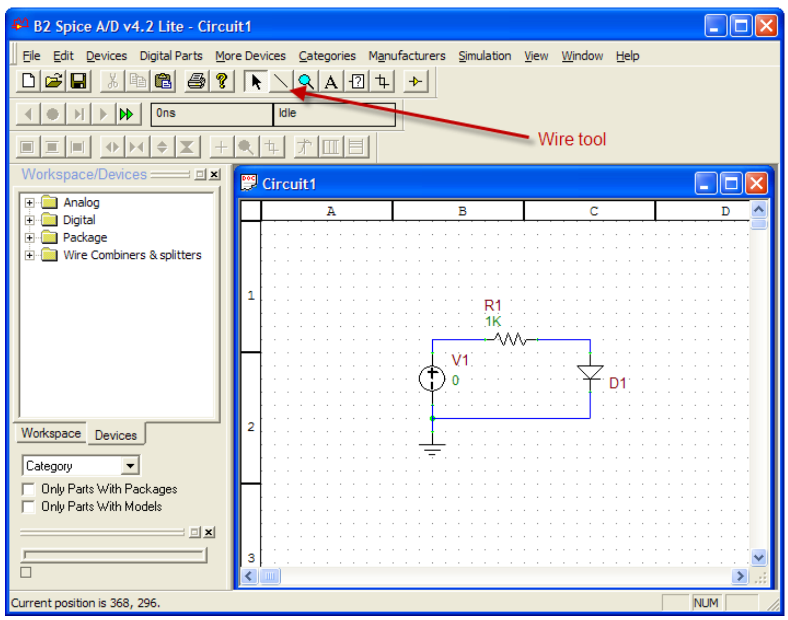


Next, we’ll add the diode. In this case, we’ll add the generic diode available in *B2 Spice* in **Devices | Diode (D)**:

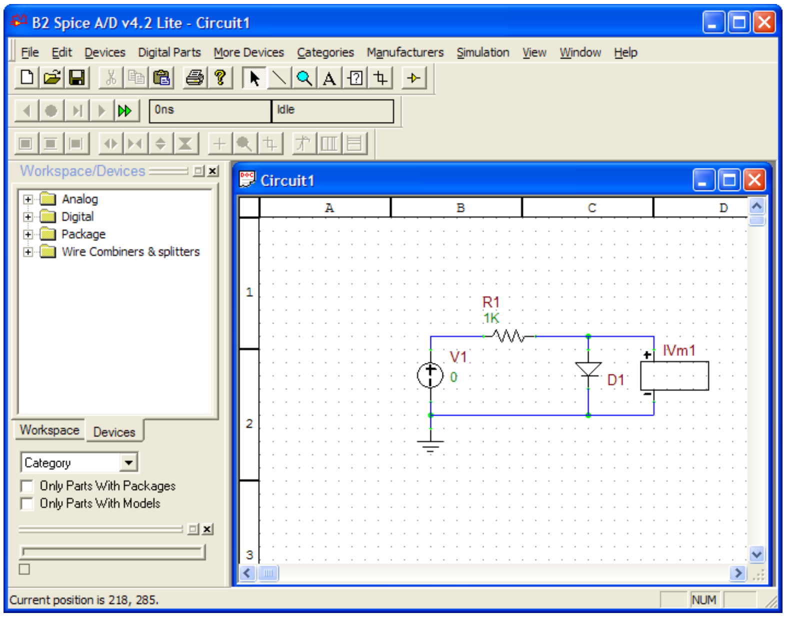


In the full version of *B2 Spice* there is a huge library of component libraries available, but in the Lite version there is only two: the generic diode and the 1N4007 diode, which can be found in **Categories | Diode…** .

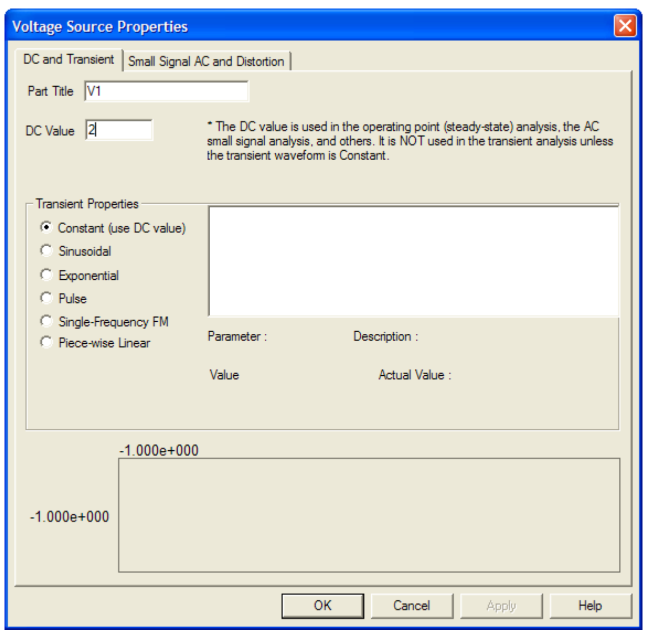
The next steps are to connect the circuit components together using the wire tool and then add a circuit ground from **Devices | Ground (0)**. The wire tool is accessed from the toolbar as illustrated below:



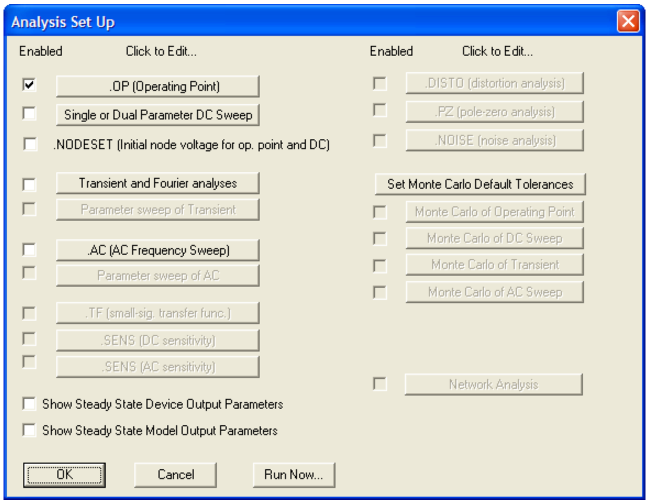
The final step in setting up this simulation in *B2 Spice* is to add a voltmeter to measure the diode voltage. Select **Devices | Voltmeter Vertical (2)**, place the component, then wire it into the circuit as shown using the wiring tool:



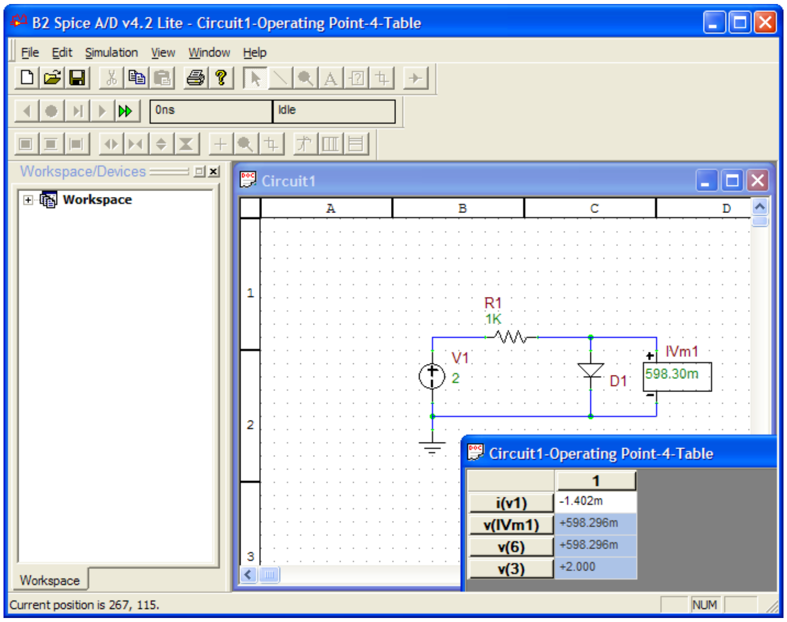
Before running the simulation we will change the voltage source to 2 V:



The final step is to tell *B2 Spice* what type of simulation you would like to perform. Select **Simulation | Set Up Simulations** and select .OP (Operating Point) for a DC simulation:



At this point the circuit is ready to be analyzed by *B2 Spice*. To run the simulation, simply select **Simulation | Run Simulations…** (or press F5):

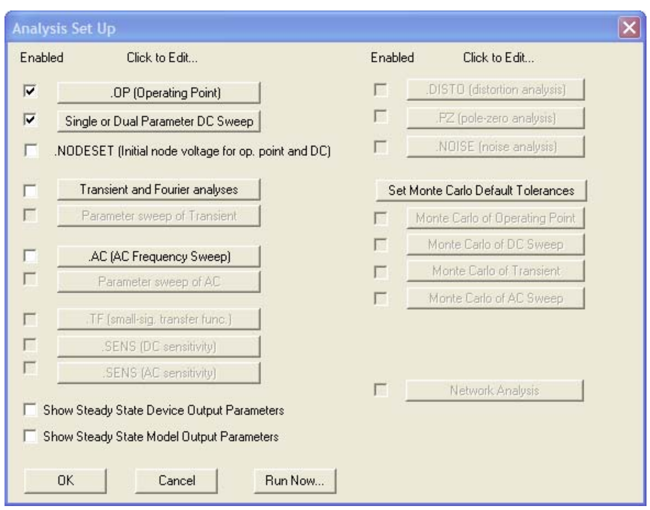


Notice that a voltage of 598.3 mV now appears in the voltmeter device, indicating the DC voltage across the diode. A table of data also pops up when the simulation has finished. In this particular case it indicates the current **into** the voltage source V1 is -1.402 mA, or equivalently, the current **out** of the source is +1.402 mA.

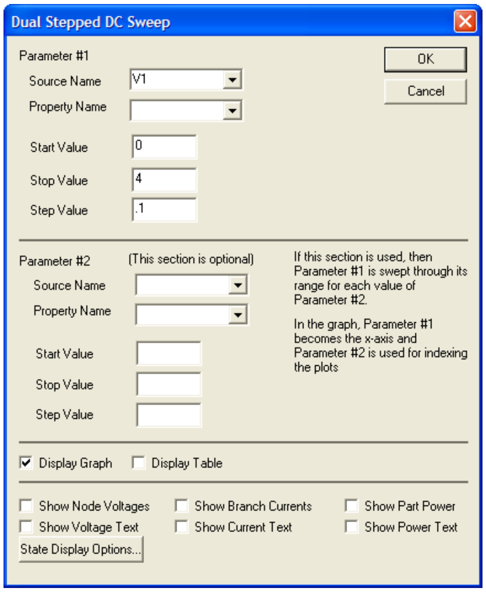
Plot V_D Versus Variable Source Voltage

For the next simulation, we will plot the diode voltage V_{D1} as the source voltage changes from 0 V to 4 V. We'll then make a plot of this diode voltage.

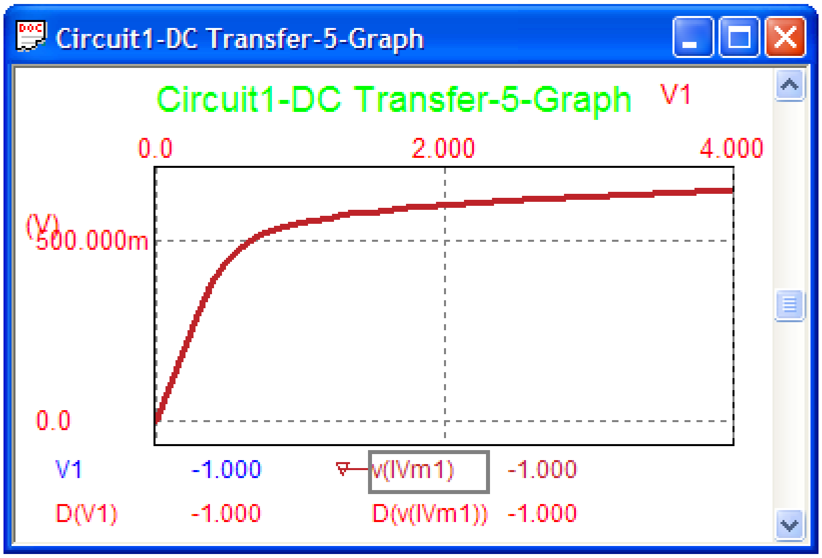
To begin, we select **Simulation | Set Up Simulations…** and Enable the Single or Dual Parameter DC Sweep as indicated:



Then we select the Single or Dual Parameter DC Sweep button, which brings up the Dual Stepped DC Sweep window. The source we wish to vary is the voltage V1 with a Start Value of 0 V and a Stop Value of 4 V. We'll choose a step voltage of 0.1 V for this sweep:



After simulating by selecting **Simulation | Set Up Simulations…**, the following plot will appear showing the variation of the diode voltage as the source voltage varies from 0 to 4 V, as desired.


Circuit1-DC Transfer-5-Graph
Circuit1-DC Transfer-5-Graph
V1
0.0
2.000
4.000
(V)
500.000m
0.0
V1
-1.000
v(IVm1)
-1.000
D(V1)
-1.000
D(v(IVm1))
-1.000