University of Pennsylvania

Department of Electrical and Systems Engineering

ESE 216 LTSpice Transient Simulation and DC Sweep

This tutorial demonstrates the Transient simulation and DC sweep using LT Spice through an example of a half-wave rectifier.

Example

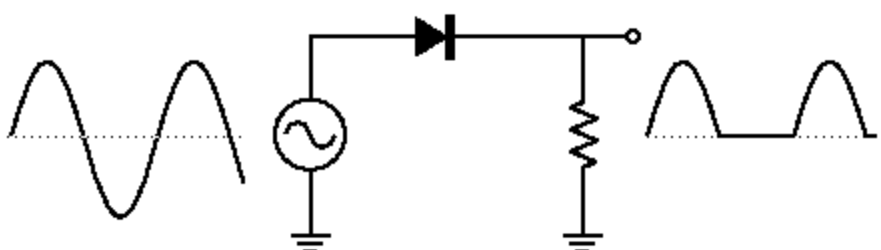

Fig. 1 A half-wave rectifier circuit

1: Design the schematic

To pick the diode from the design library, just click the "component" icon and choose the diode (Fig. 2). After placing the diode on the schematic, right click on the diode symbol and choose "Pick New Diode" to select the correct model for the diode (Fig. 3). For this example, we choose the 1N4148 diode model (Fig. 4), and click "OK" to confirm.

After inserting the diode, insert the load resistor (1kΩ), input voltage source, and make connections. The simulation schematic should look like Fig. 5 and the setup for the voltage source can be found in Fig. 6.

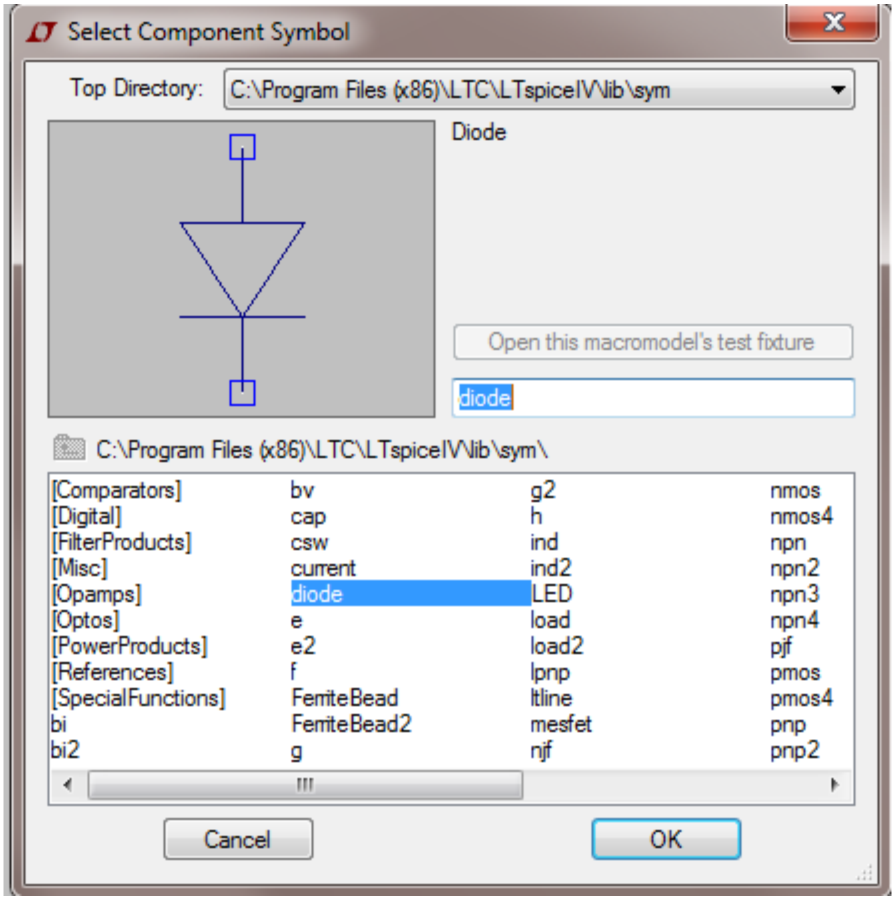

Fig. 2 Selecting the diode component symbol

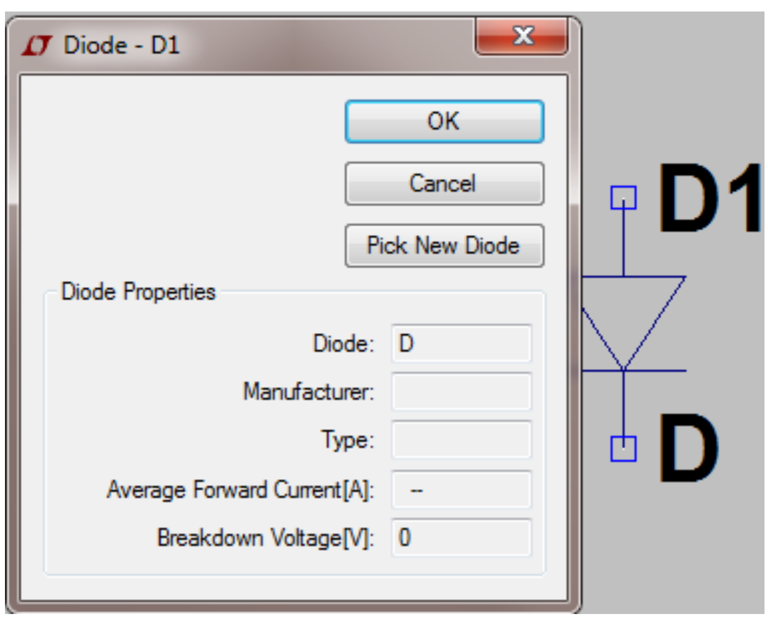

Fig. 3 Picking model for the diode

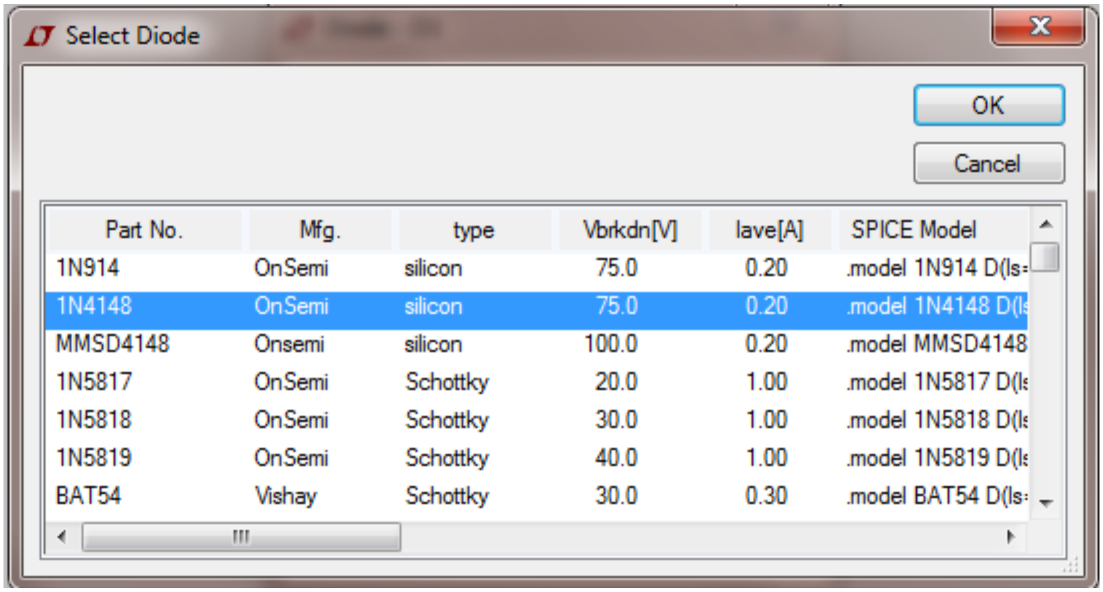

Fig. 4 Choosing 1N4148 diode model

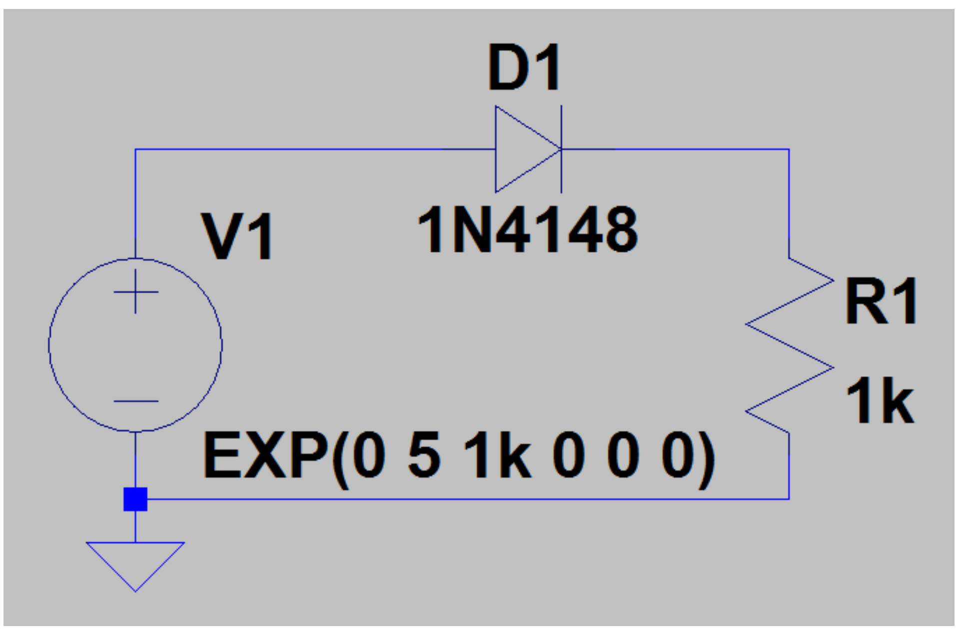

Fig. 5 Simulation schematic

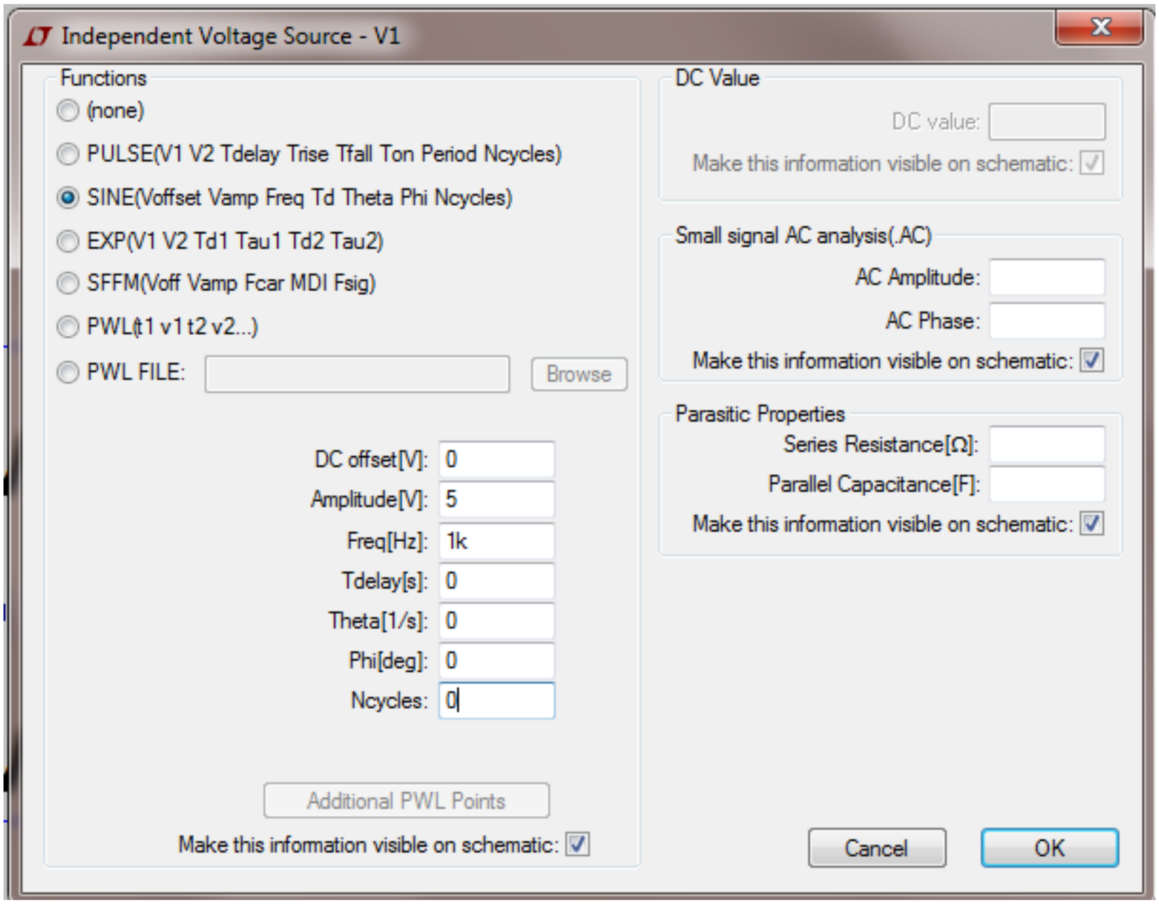

Fig. 6 Setup the voltage source

2. Simulate and Plot the Schematic

After drawing the schematic and setting the component values, we can proceed to simulations. We can setup the simulation by choosing “Simulation” on the menu bar and select “Edit Simulation Command”. This time we need to choose Transient simulation. The only thing we need to setup for the transient simulation is the stop time. Since we set the frequency of the voltage source to 1kHz, which corresponds to a period of 1ms, in order to see the full period signal in our simulation results, we should set the simulation “Stop Time” to at least two periods’ time. In this example, I set the stop time to 3ms, which means we can see the signals for 3 periods.

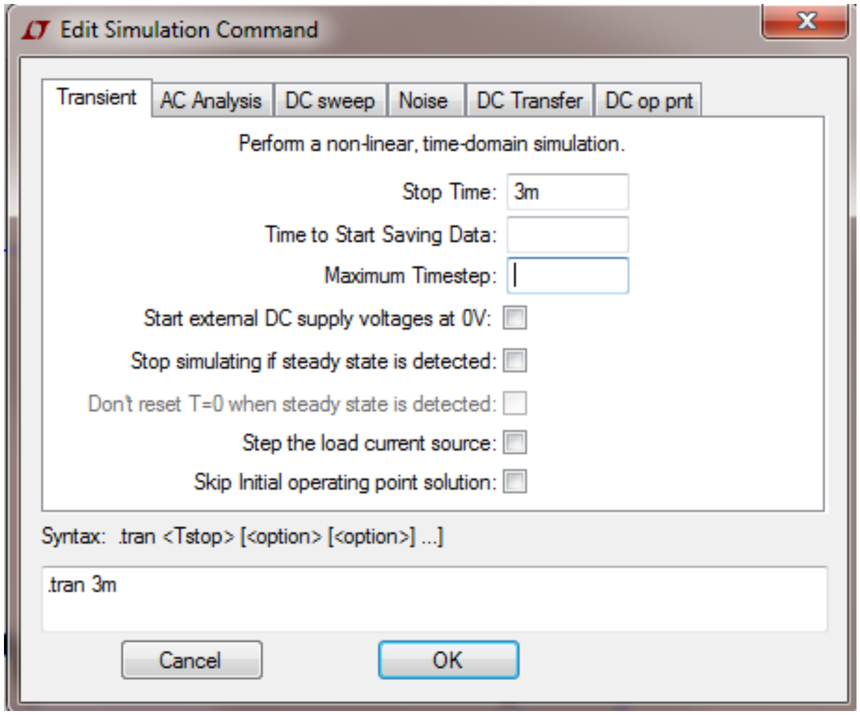

Fig. 7 Transient simulation setup

After setting the simulation, we can just click on the "run" button and plot the output signal. The output signal should be like Fig. 8. Note that the transient simulation result has time as the horizontal axis and voltage as the vertical axis.

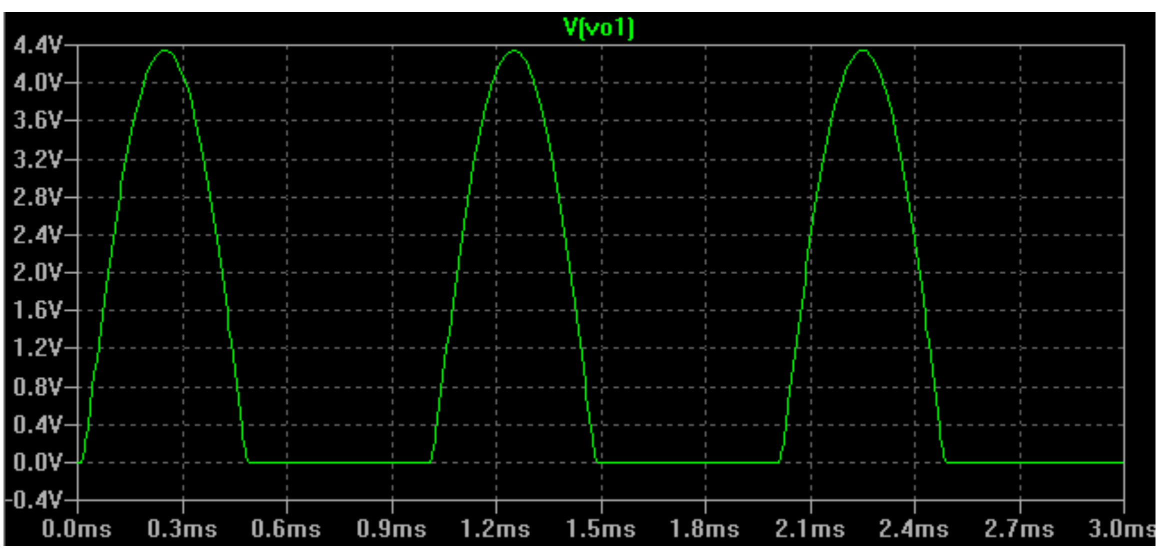

Fig. 8 Output signal from transient simulation

After we have done the transient simulation, we can move on to DC sweep simulations. The DC sweep is a function that sweep input voltage from a start value to a stop value for given steps. And it stores the DC voltage of every node in the circuit for each sweeping step. We can setup the DC sweep in the "Edit Simulation Command" window, as shown in Fig. 9. The "Name of Source to Sweep" is the name of the input source, V1, in our example. And the start value and stop value in the sweep is the voltage range of the sweep of the input voltage source. "Increment" is the step of the sweep. So the setting in Fig. 9 means we will have the input voltage source V1 to sweep from 0V to 5V at steps of 0.1V.

Keep in mind that the DC sweep disables the signal settings in the voltage source. This means even the signal source is set to output SINE wave at 1k Hz, the DC sweep simulation will automatically disable the SINE wave and make the signal source as a DC source sweeping from 0V to 5V at steps of 0.1V.

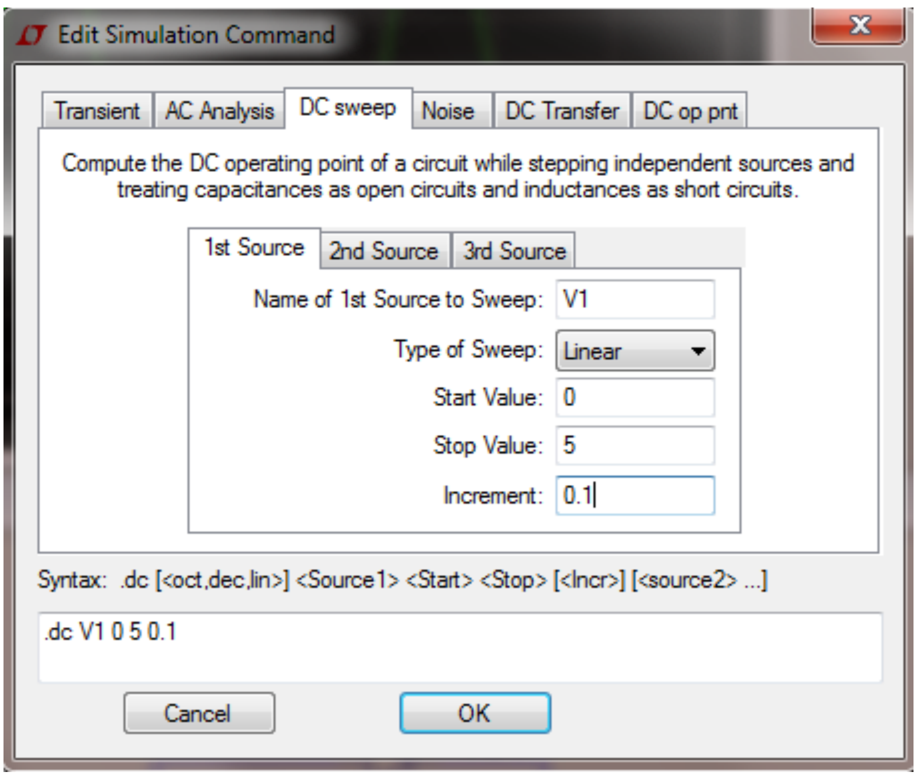

Fig. 9 DC sweep simulation setup

The result of the DC sweep simulation is shown in Fig. 10. Notice that the horizontal axis is the sweeping voltage from 0V to 5V, and the vertical axis is the output voltage. The plot is the output DC voltage corresponding to each step of the input sweeping DC voltage.

Fig. 10 DC sweep simulation plot