# How to Vary a Component Value in an LTspice Netlist Simulation

It is sometimes desired to determine how varying the value of a particular component affects the response of a circuit. One might be tempted to do this by manually entering each value separately and resimulating the circuit to view the response for each value, one at a time. Fortunately, LTspice has a .step command that makes this much easier by allowing you to easily sweep a component through a range of values in a single simulation run. This command causes an analysis to be repeatedly performed while stepping through a model parameter, global parameter or independent source value.

Example 1:

Given the circuit shown below, determine how the amount of power absorbed by RL varies with the value of RL.
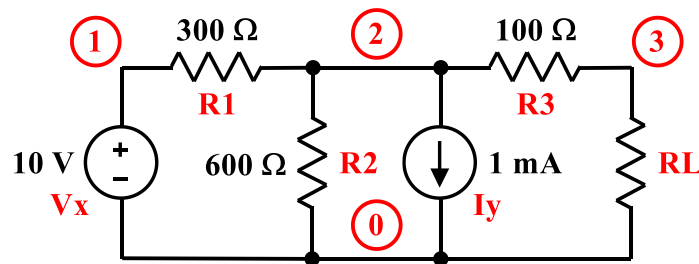
1. **Describe the Resistor**: Rather than a numerical value, assign the resistor's value to a parameter. In this case, the parameter is {R}. The name of the parameter must be enclosed in curly brackets.

    RL 2 0 {R}

2. **Specify the Sweep Range**: In this example, we sweep the value of parameter R from 1e-12 to 500, in steps of 10. (The lower limit cannot be zero.)

    .step param R 1e-12 500 10

3. **Specify the analysis:** This would determine the operating point for each value of RL.

    .op

4. **Include other lines to complete the description of your circuit:**

 | | | | | |
    |---|---|---|---|---|
    | Vx | 1 | 0 | DC | 10 |
    | R1 | 1 | 2 | 300 | |
    | R2 | 2 | 0 | 600 | |
    | Iy | 2 | 0 | DC | 1m |
    | R3 | 2 | 3 | 100 | |

5. **Run the simulation.**

When the simulation is done, the plot window will open. The value of parameter R is automatically assigned as the independent variable for the plot. Right click in the middle of the window, click on "Add Traces" in the menu, and then select variables from the list or enter the function you wish to plot. To plot the power absorbed by RL, enter the expression V(3)*I(RL) on the command line, and then click on "OK". Note that LTspice is "smart enough" to label the vertical axis with units of watts.
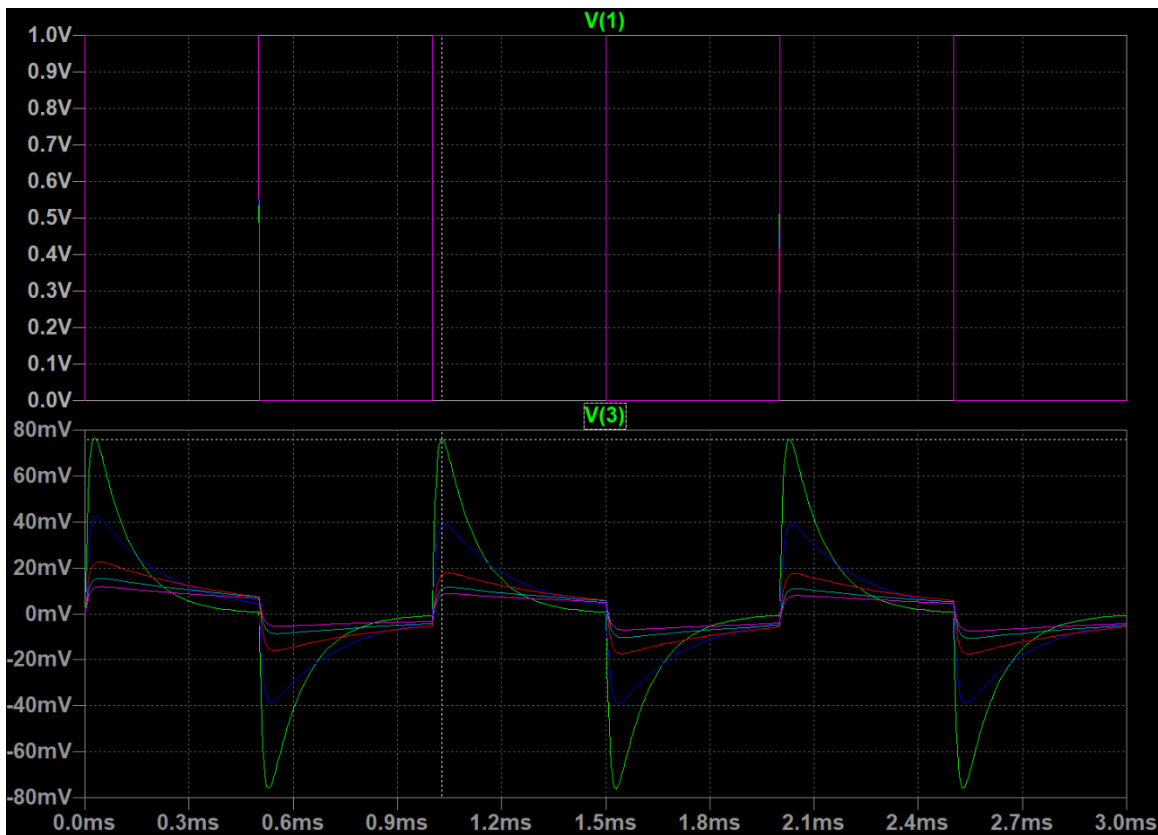

Example 2:

Consider the RLC circuit shown below. Here the objective is to see how the waveform for $v_C(t)$ changes as the capacitor value is varied.
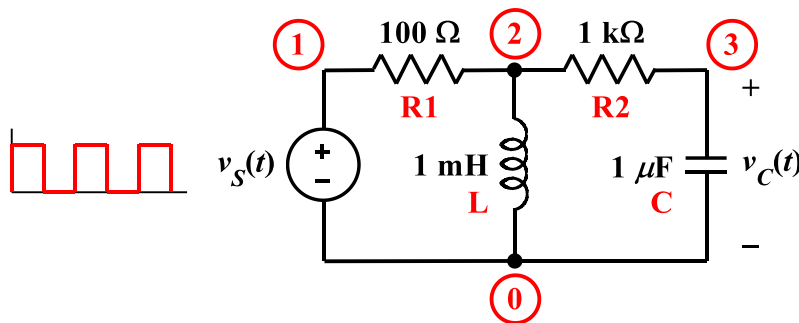
```
Vs      1       0       PULSE(0 1 0 1u 1u .5m 1m)
R1      1       2       100
L       2       0       1m
R2      2       3       1k
C       3       0       {X}
.step   param X         list    .1u     .2u     .4u     .6u     .8u
.TRAN           3m
```


Once you run the simulation and display the results in the Waveform Viewer you can determine the step information associated with a particular trace by attaching a cursor (click on the trace label at the top of the window), using the up and down arrow keys to move between traces, and then right-clicking on the cursor once it's on the trace you want.