**Steps to use variable resistor and how to simulate circuit using variable resistor**

1. "get new part" choose "R_var" and place it on the schematic.
2. Double click on "R_var" symbol and Change the **value** of the part (not the name!) to {RL} (use curly braces, name is arbitrary) and then press "Enter". Click "OK".
3. Go to "get new part" select "PARAM" and place it on the schematic.
4. Double click on "PARAMETERS"
5. Set the name to RL (same name as in "a" but with no curly braces) to NAME1
6. Set VALUE1=1k and then "OK" to exit the window.

Simulation Settings:

1) Click on "Set-up Analysis"
2) Analysis type: DC Sweep
3) Sweep variable: Global parameter
4) Parameter name: RL (or name of the parameter you used without curly braces)
5) Sweep Type: Linear
6) Fill in the Start, End, and Increment values. Note for resistance, the start value cannot be 0! Use 0.1 instead.

Press OK and simulate