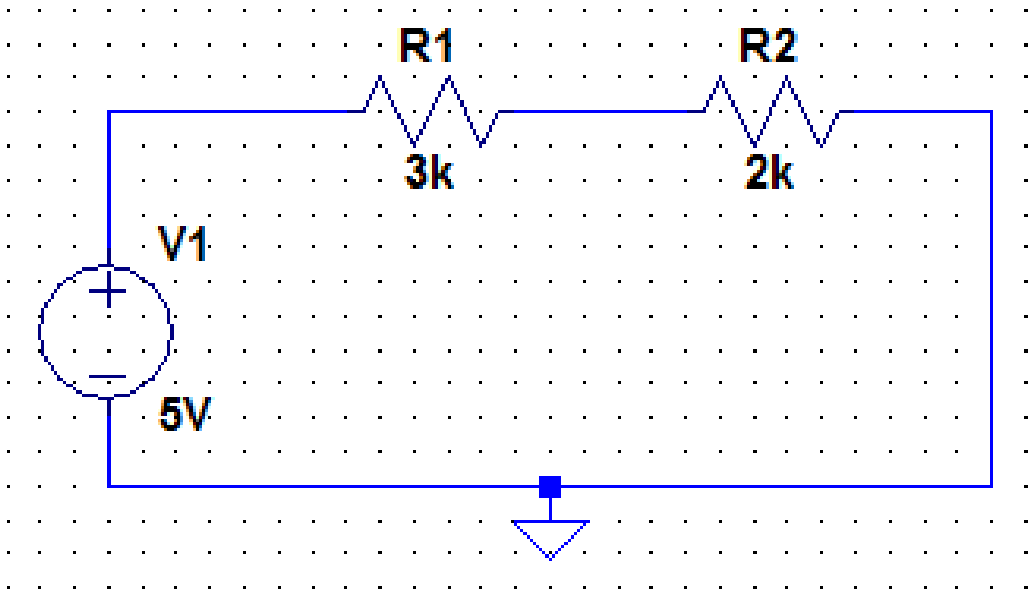
More LTspice

Lab 2

LTspice

- Build the following circuit in LTspice and simulate it:

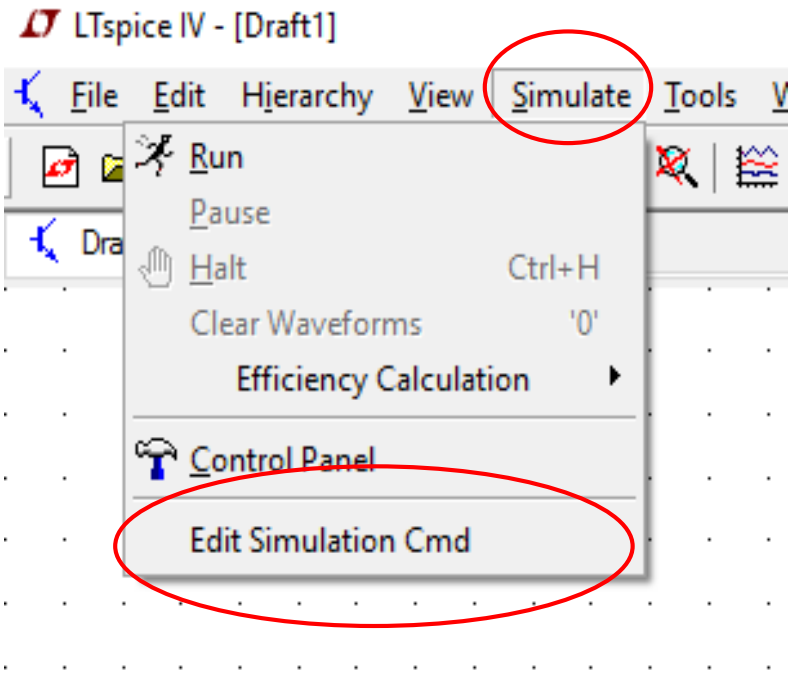


DC Sweep

- DC Sweep is used when you want to vary the value of one of the sources or resistances in the circuit
  - Use DC sweep to figure out the response of the circuit over a range of values

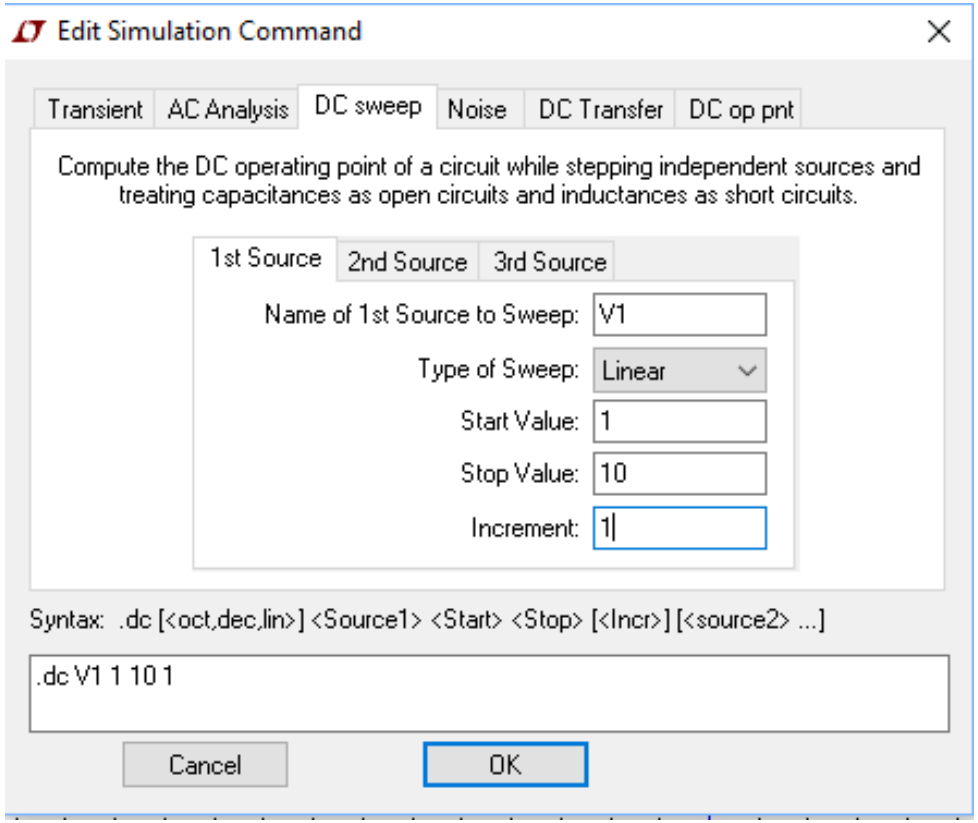
Running a DC sweep

- Run the DC sweep, by selecting the Simulate menu and clicking *Edit Simulation Cmd*



Running a DC sweep

- In the dialog box click the DC Sweep tab and set the following parameters and click OK



DC Sweep parameters

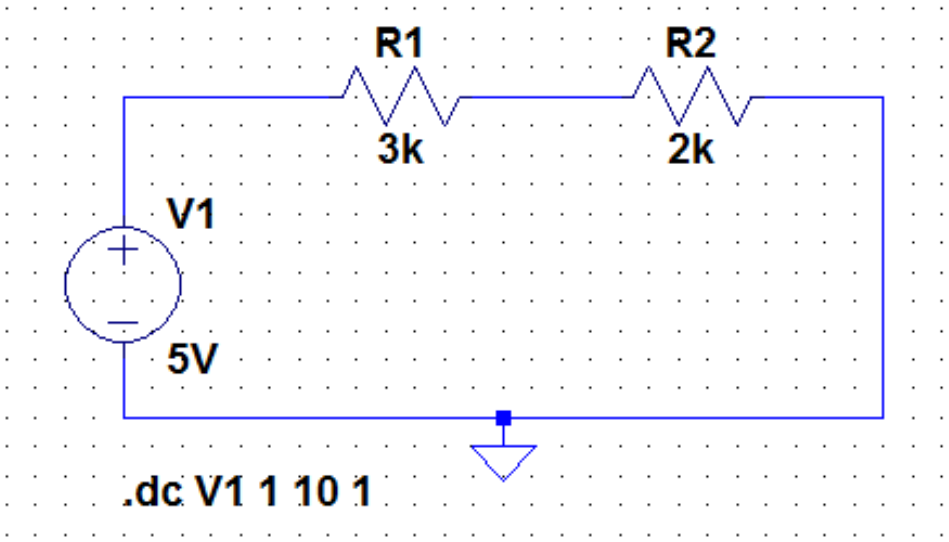
- Sweep type is linear. We will try other options later.

- For the Name text box, enter the name of your voltage source in the schematic. **NOT** the type of source it is (e.g. voltage) but what you call it: “V1,” or “VSource1”

- Enter the desired start, end and increment values. We choose, 1, 10 and 1 respectively. Note that you do NOT have to enter the units

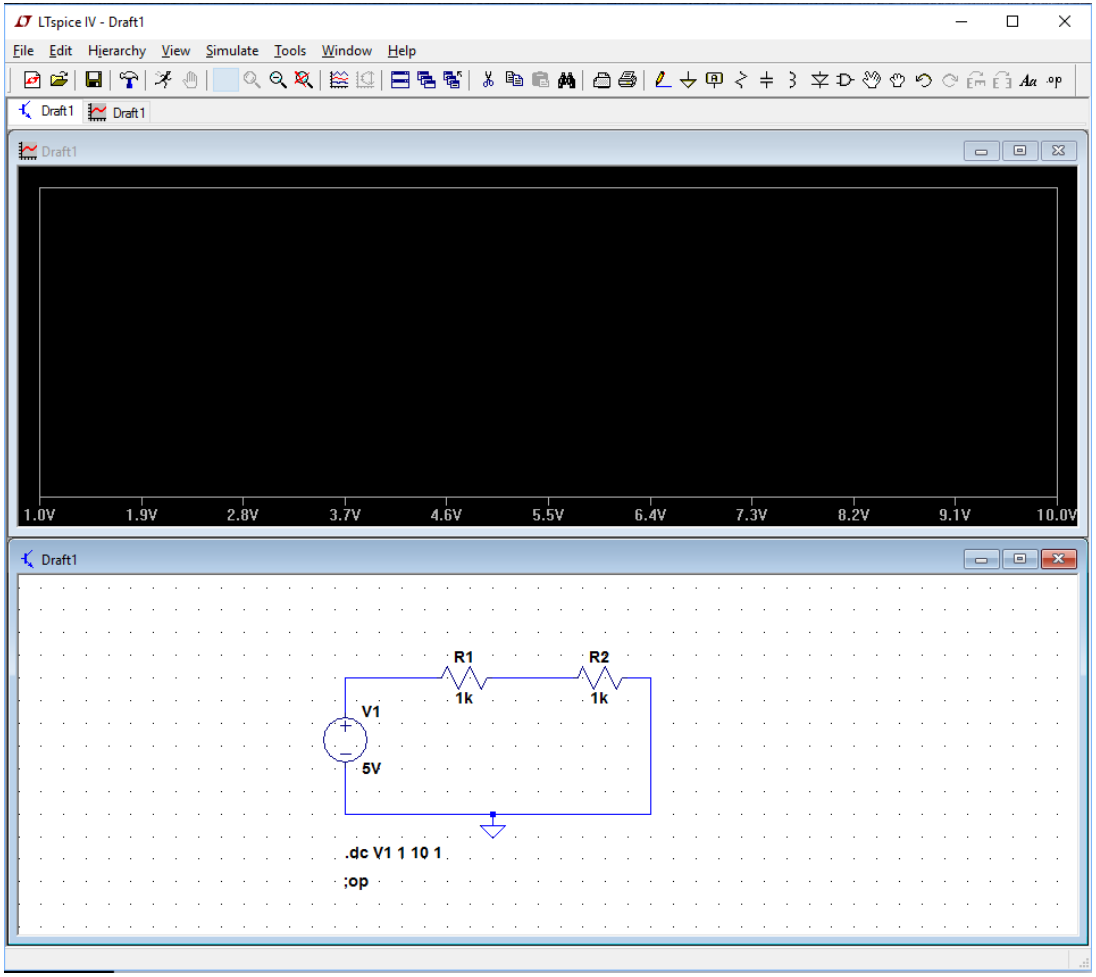
DC Sweep

- Running a DC Sweep implies that the static value supplied on the schematic for the voltage supply does not matter.



Simulate

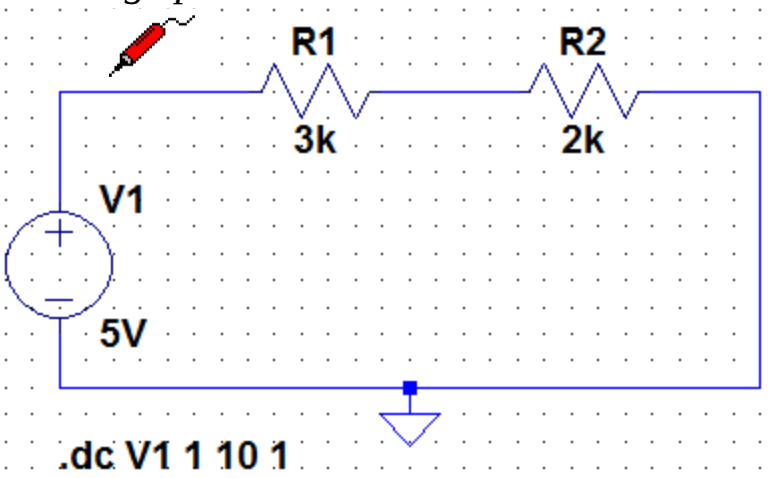
- Once you have set the DC Sweep, simulate the circuit. The Probe window will pop up with a blank black graph.



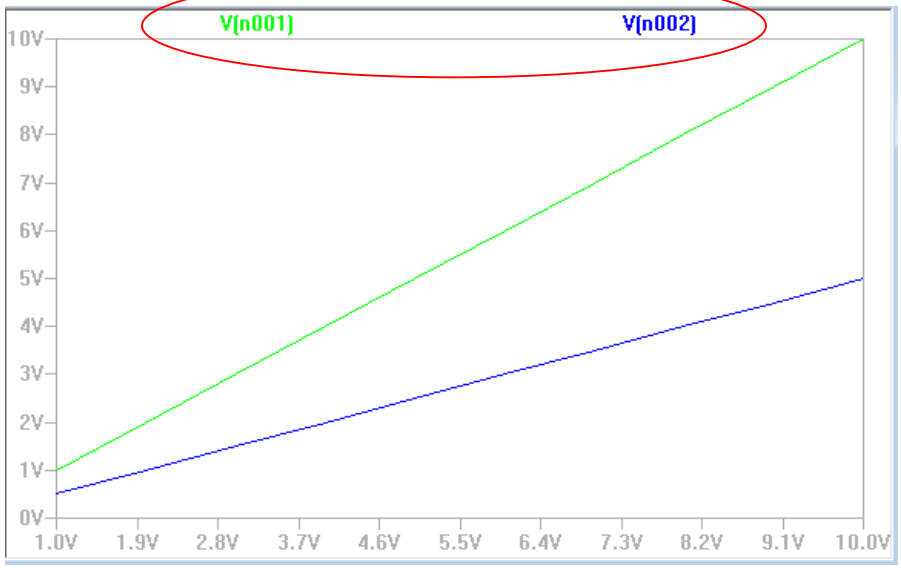
Adding a Trace

- Left-click on a wire to plot the voltage for that node. **This will measure the voltage <u>with respect to ground</u>.**

Voltage probe cursor



Node voltages



Adding a Trace

- We will plot the voltage drops across all three components using node voltages. Here you can see the two voltages are V(n001) and V(n002).

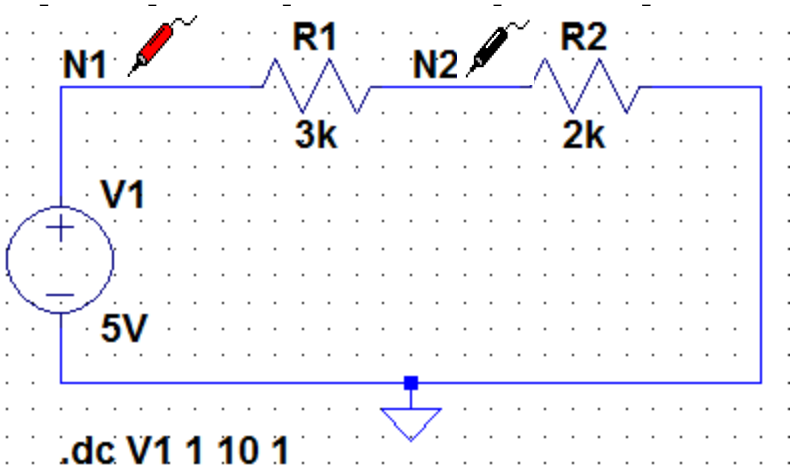
- For more complicated circuits, it can be a problem to recognize the nodes without labels.

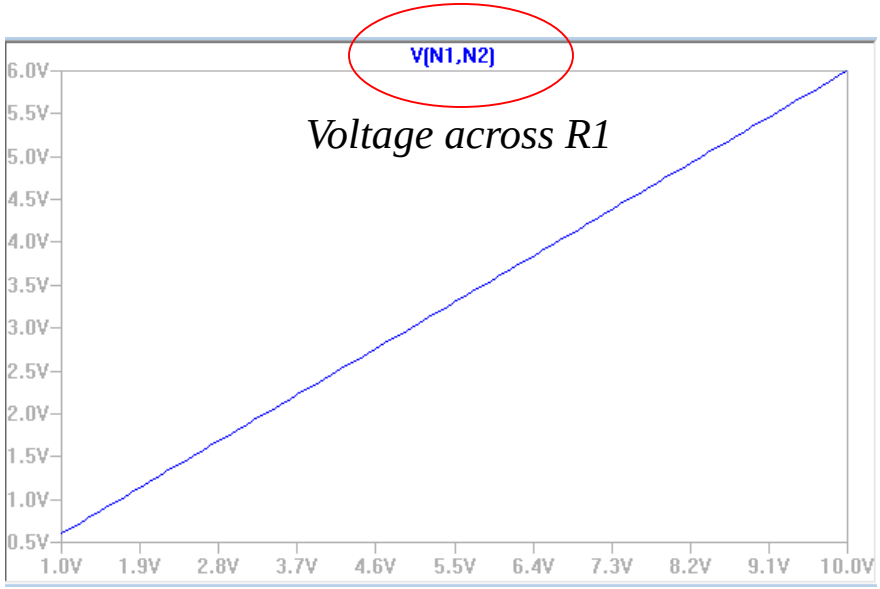
- **It is important to label nodes when adding traces for the homework problems.**

Adding a Trace

- Go back to the schematic, rename the wires N1 and N2 and simulate your schematic again.
- Plot the three voltage drops across the components. For example, the voltage difference across nodes N1 and N2 will provide the voltage across R1.

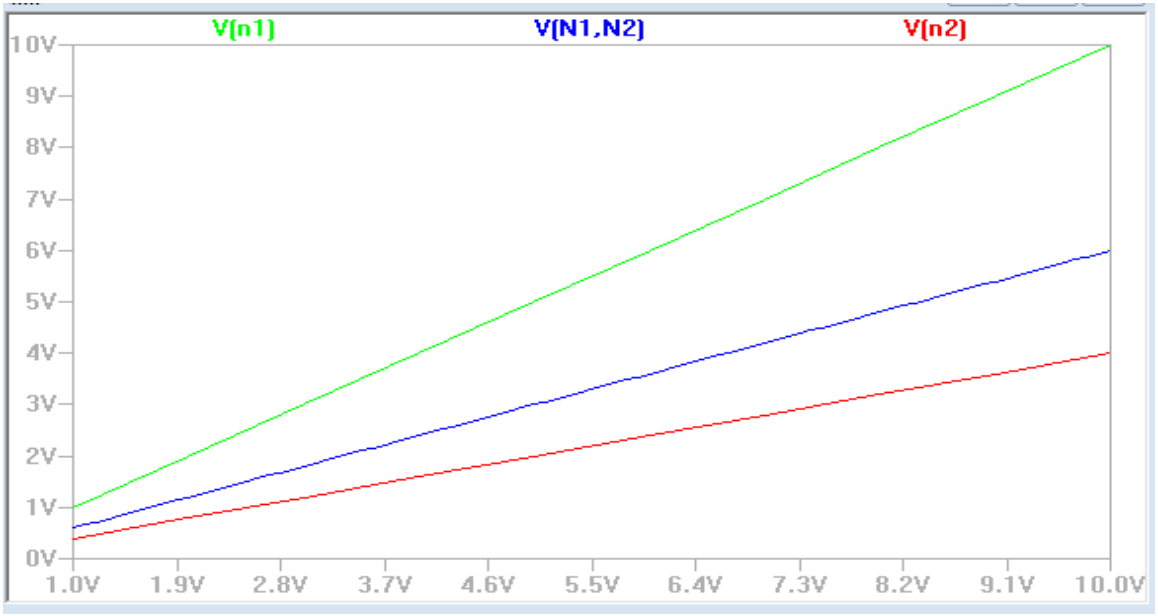
Voltage difference across nodes: Left-click and hold on the node and





Adding a Trace

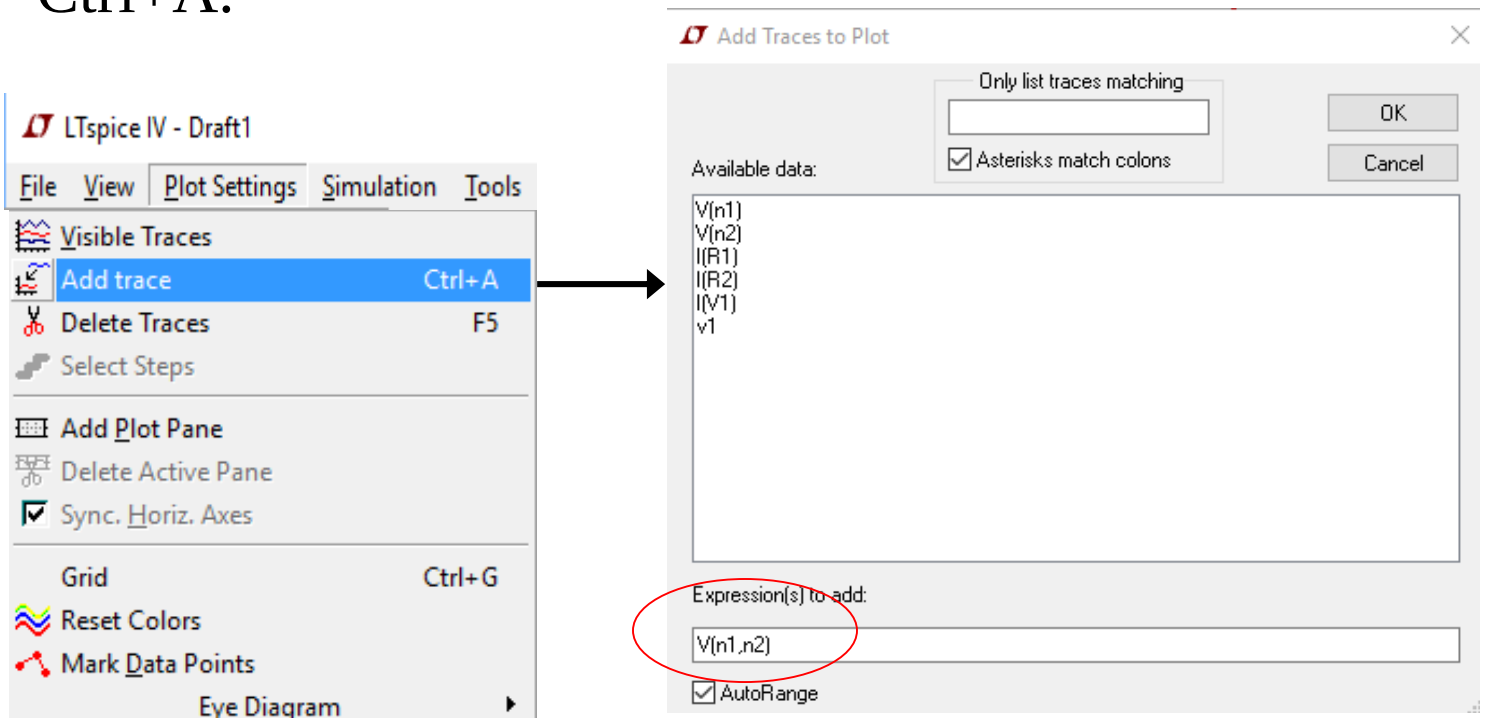
- You should see the following plot



  - Note that the variable for the x-axis has automatically been chosen to be the voltage source. This is because it is the swept variable.
  - In the *Tools* drop down menu, choose *Copy bitmap to Clipboard* to paste the graph in word processing applications.

Adding a Trace (with expressions)

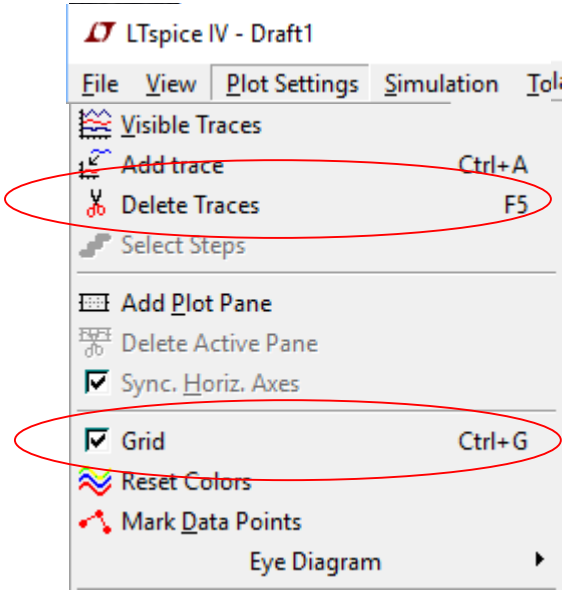
- To add a plot, after selecting the plot window, click on the *Plot Settings* menu and select *Add trace* or you can press Ctrl+A.

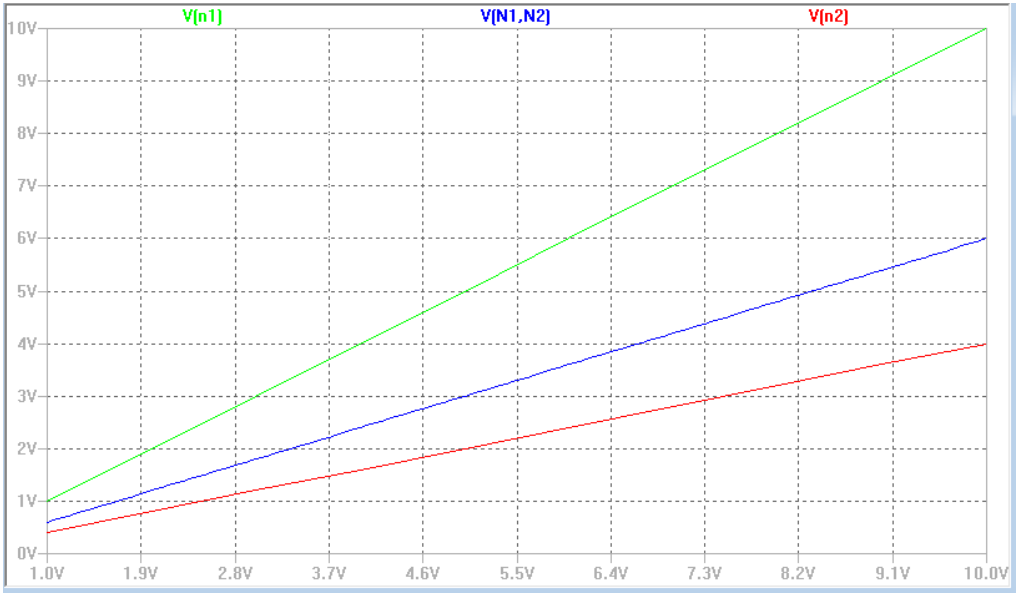


This expression gives the voltage difference between nodes N1 and N2.
=> V(n1)-V(n2)

Modifying/Deleting a Trace

- To modify a specific trace, right-click the legend, and change it.
- To turn the grid on, under *Plot Settings* click on *Grid* (Ctrl+G).
- To delete a trace, under *Plot Settings* click *Delete Traces* (F5) or click and delete the legend.

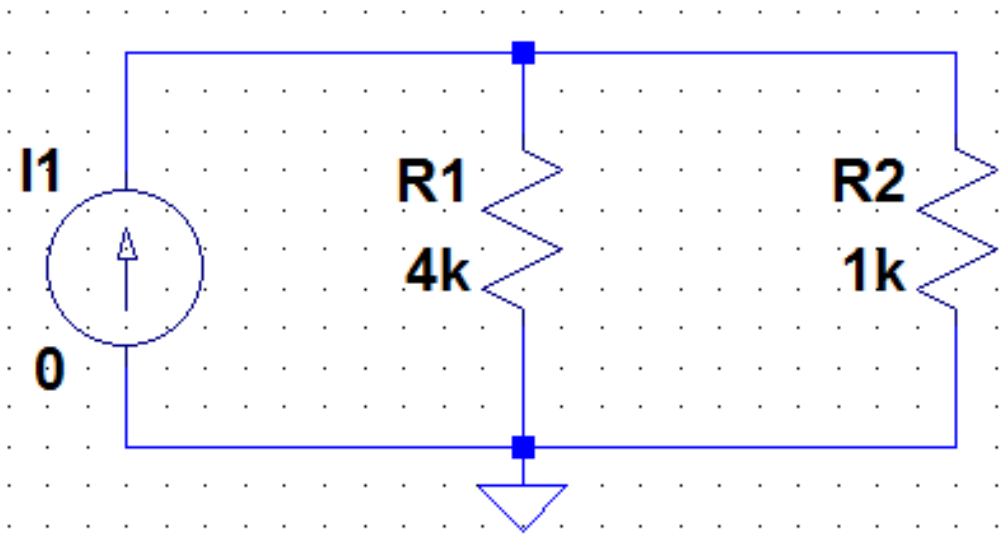




Understand the Traces

- Note that the voltage dropped across the resistance R2 is exactly half of the voltage supplied by the source at any point on the trace
- Record on your data sheet the reason for this.

Build a Current Divider



- In the DC sweep dialog box, sweep the current source linearly from -5mA and 5mA in steps of 0.1mA

Add Traces

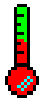
- Simulate the circuit and plot the following currents: I(I1), I(R1) and I(R2).
- To plot current, left-click on the body of the component. You will see the current probe.

- Note that you can also use the currents supplied by LTspice in the *Visible Traces* dialog box.

- You do not need to name the wires to obtain currents alone
  - This is because voltage is dropped ACROSS a component and the current flows THROUGH the component.

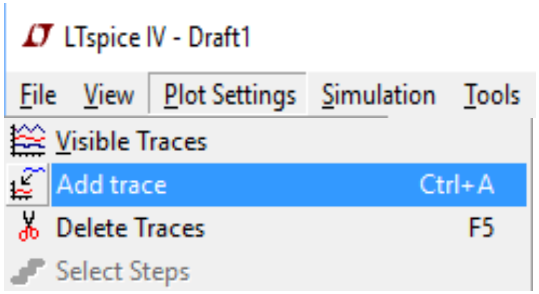
Plotting Power

- Delete all three traces.
- For instantaneous power dissipation, hold down the *Alt* key and left-click on the component.
  This will calculate the power by using V*I



Power probe

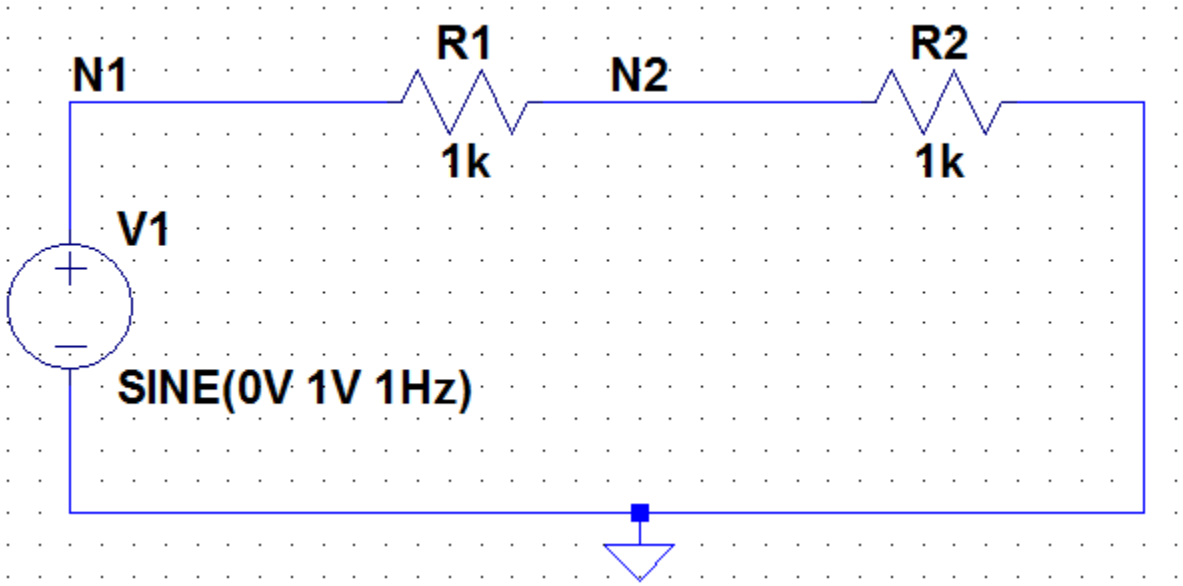
- There are three formulas for power
  - VI,
  - V^2/R, and
  - I^2R



- We will also use the I^2R formula to plot power.
  - Type in I(R1)*I(R1)*4000 to plot the power dissipated by R1. (4000 is the value of R1)

Transient Analysis

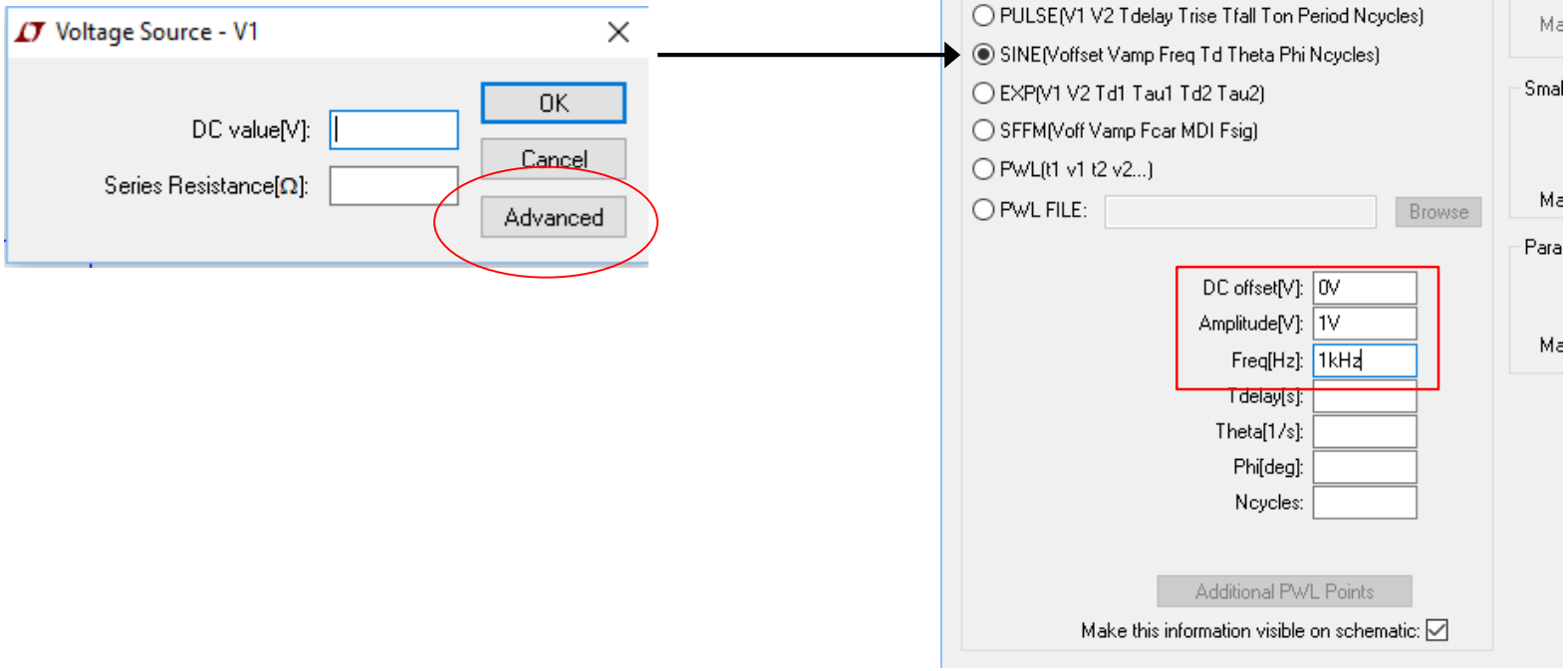
- Transient analysis is used when the signal in the circuit - current or voltage - changes with time
- Draw this circuit in LTspice. Use SINE as the source function.



Transient Analysis

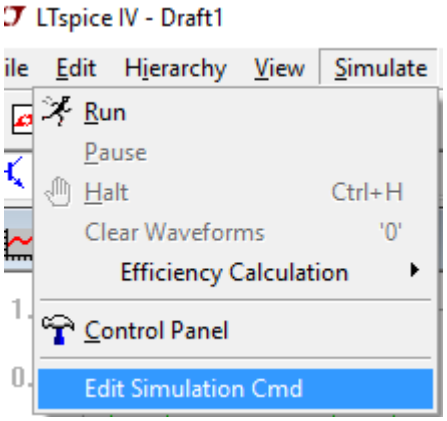
- Set the SINE source to have a frequency of 1kHz, with a peak to peak voltage of 2V (amplitude 1V) and 0 offset.

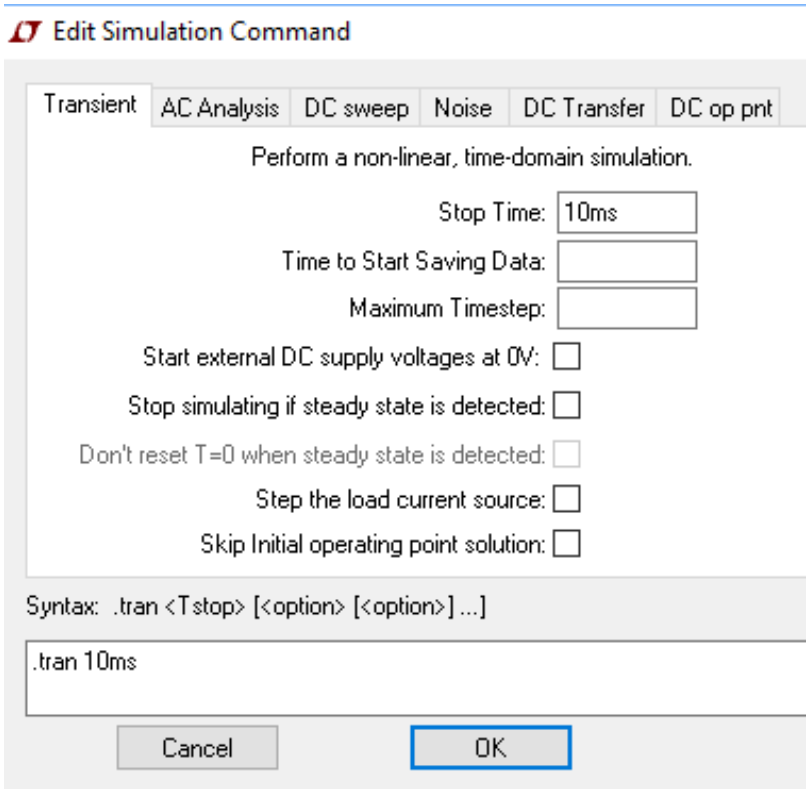
Right-click on the voltage source and click *Advanced*



Transient Analysis

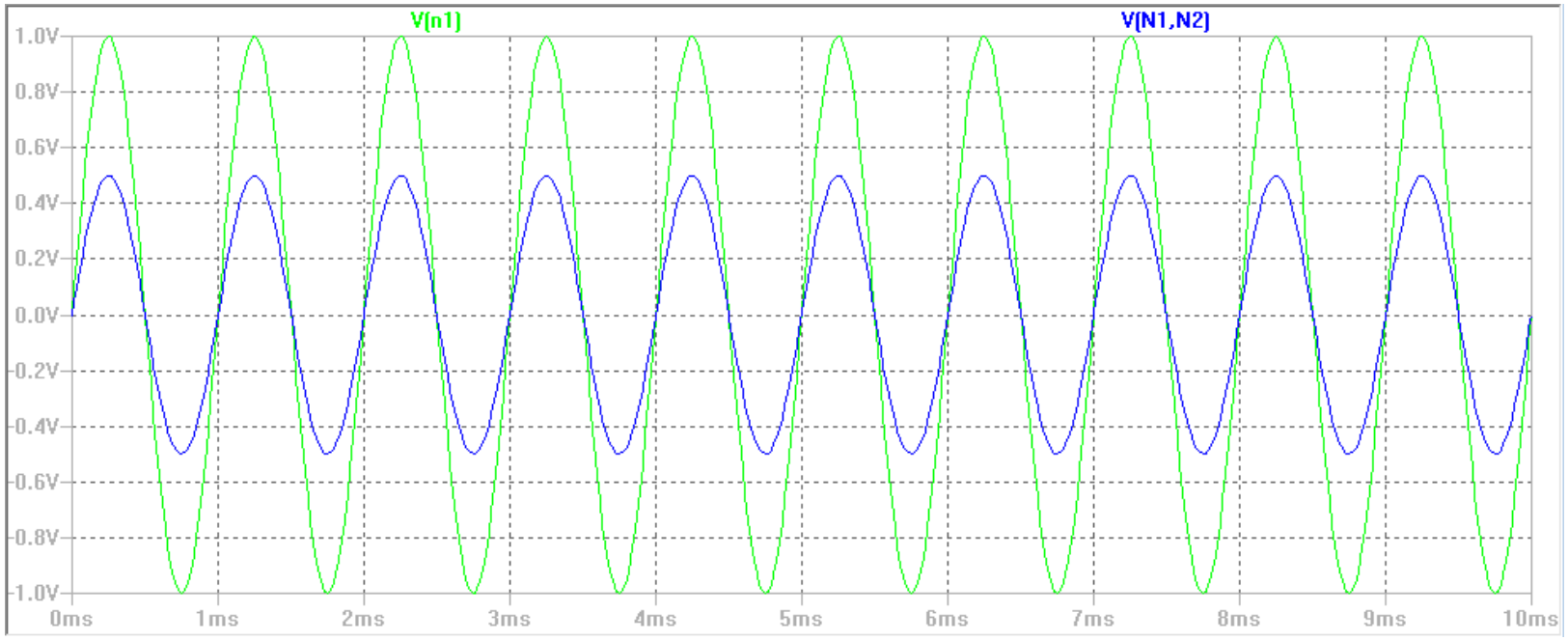
- Under the *Simulate* menu, click *Edit Simulation Command*. Click on the Transient tab. Enter10ms for Final time. Click OK.





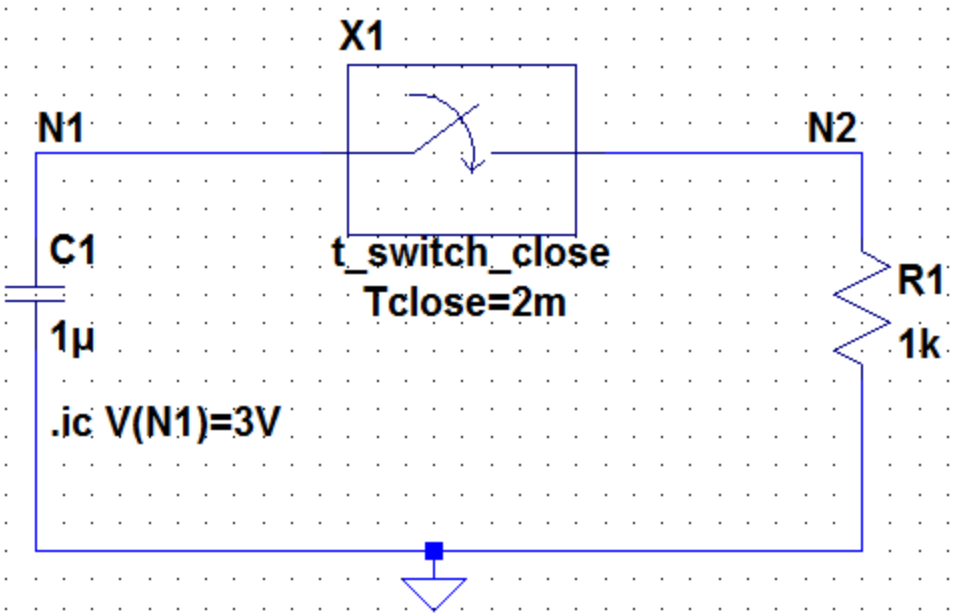
Transient Analysis

- Simulate your circuit and in the Probe window, plot the source voltage and the voltage dropped across R1.



Switching Transients

- Build the circuit shown below:
  - The switch is called t_switch_close. This is a third party component. Follow the instructions in class to access the component.

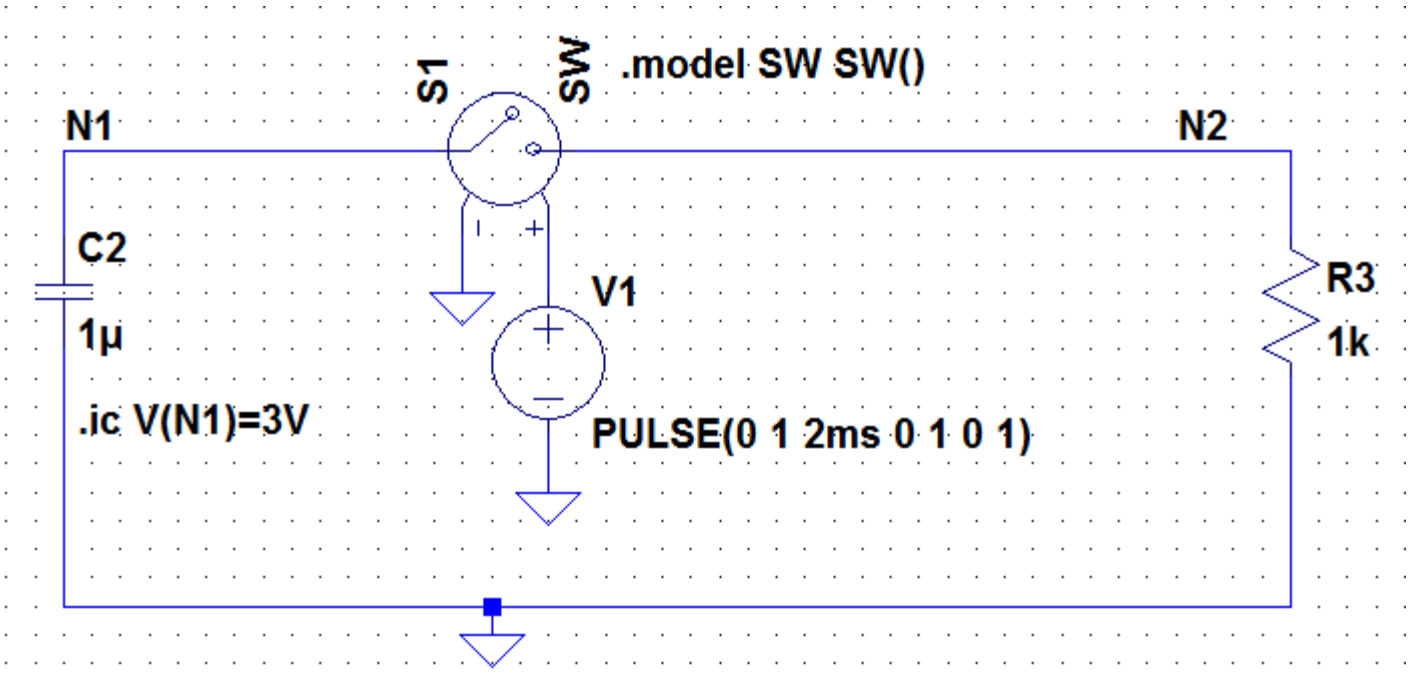


Settings

- Edit the attributes of the switch. Set Tclose to 2 milliseconds
  - This means that the switch is open initially but will close at time *t*=2 ms
- Set the initial condition or initial voltage of the capacitor to 3V
  - Press or ‘S’ to add a SPICE directive.
  - Enter ***.ic V(N1)=3V*** . This will be the initial condition for the capacitor.
- Set up a *Transient analysis* with *Final time*=10ms.
- Simulate the circuit and plot V_C and V_R.

Switching Transients (Alternative)

- Build the circuit shown below:



Settings (Alternative)

- Use a voltage controlled switch (SW).
  - Press .op or ‘S’ to add a SPICE directive.
  - Enter ***.model SW SW()*** .

- Use a voltage source and set it to PULSE with the given settings.

- Set the initial condition or initial voltage of the capacitor to 3V
  - Press .op or ‘S’ to add a SPICE directive.
  - Enter ***.ic V(N1)=3V*** . This will be the initial condition for the capacitor.

- Set up a *Transient analysis* with *Final time*=10ms.

- Simulate the circuit and plot VC and VR.