Bias Point Calculations:

Schematics Version 9.1
After Launching Schematics

Select Draw/Get New Part, which will cause a pop-up window to open with a list of all of the parts available in the student version of Schematics.

- Note that there is a key stroke shortcut.
Pick a Part

Type the name of the part in the box located by Part Name or Scroll through the list of parts and click on the name of the one that you want put into the circuit.

Then, click Place if you have more parts to select or Place and Close if this is the last part.
If You Know the Name of Your Part

You can type it in directly into the box below the Toolbar Help and then hit Enter.

– Either way, a symbol for a dc battery will appear on the schematic when you move your cursor on to it.
  • Left Click to place the part in a specific location on the schematic.
  • Right Click to end the process of placing the part Vdc.
# Names of Components

<table>
<thead>
<tr>
<th>Component</th>
<th>PSpice Name</th>
<th>Component</th>
<th>PSpice Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>DC voltage supply</td>
<td>Vdc</td>
<td>Resistor</td>
<td>R</td>
</tr>
<tr>
<td>DC current supply</td>
<td>Idc</td>
<td>Capacitor</td>
<td>C</td>
</tr>
<tr>
<td>Ground</td>
<td>gnd_earth</td>
<td>Inductor</td>
<td>L</td>
</tr>
</tbody>
</table>
Click on the pencil.

Then click the pencil tip to one end of a part and then to the end of another part to place a wire between them.
Save Your Schematic

• PSpice will not run a simulation unless your circuit is saved.
  – You can select any directory in which to save the .sch file.
  • Note: You can send the .sch file to the PSpice GTA or to me if you need assistance finding errors that prevent the simulation from running.
Select Type of Simulation

• Click on Analysis/Setup or the button.
  – Bias Point Detail should be clicked. If not, do so.
Run the Simulation

- Click Analysis/Simulate or click on the button.

A pop-up window should open, text should scroll by in the bottom left box. After “Simulation complete” is printed, the run is finished.
To Display DC Values

• Click on Analysis/Display Results on Schematic and enable the voltage and/or current display.
Comment on Analysis

• Note that the calculation performed by PSpice Schematics is a nodal analysis. The voltage dropped across a component is the difference between node voltages.