

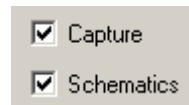
Introduction

This laboratory experiment introduces the PSpice (Student Edition 9.1) circuit simulation program with the schematic capture front end, Schematics. Basic features of the program will be explored using simple resistive circuit examples.

Procedure

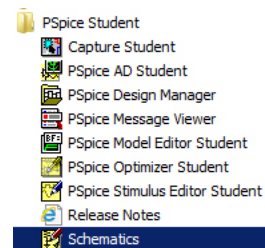
1. **Install PSpice 9.1** (Student Edition) on a computer or locate a computer onto which the software has already been installed.


If installing the software, select both the **Capture** and the **Schematics** modules.

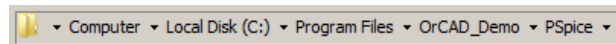


2. **Execute the Schematics** capture program.

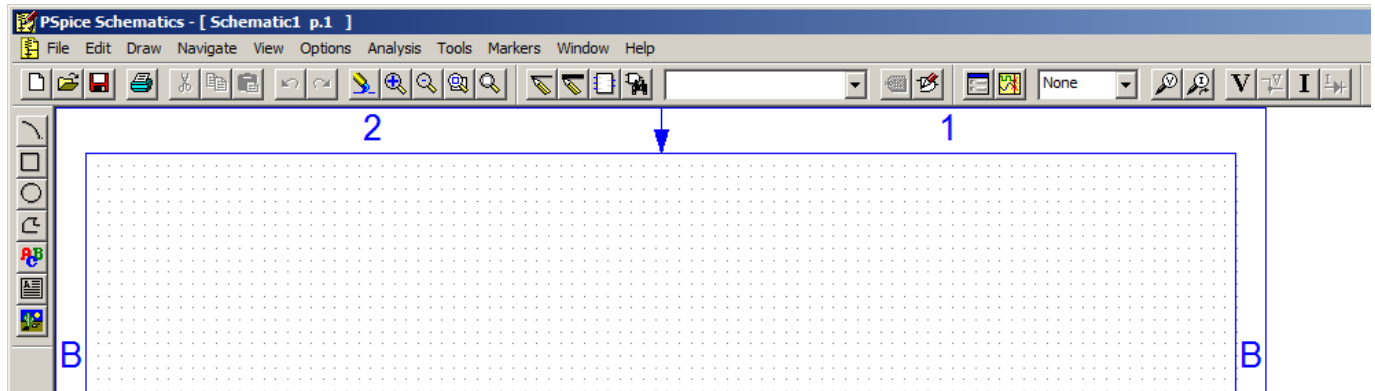
If a shortcut was created during the installation, Schematics can typically be executed by *left-clicking* **Start**→**Programs**→**PSpice Student**→**Schematics**.



If a shortcut is not available, locate and execute the program named **psched.exe**  **psched.exe**. It is typically located in a folder named PSpice:



A blank schematic worksheet should appear:



Additionally, several other windows may appear, such as:



You may **minimize**  these windows to minimize desktop clutter.

Schematics is a front-end program with a direct interface to other PSpice tools. Using Schematics it is possible to design, draw and simulate circuits. Some of the simulation results can be viewed directly within the Schematics window while other results can be analyzed using the PSpice waveform viewer (Probe).

The first circuit that will be entered into the Schematics program for simulation is shown to the right.

The circuit consists of a **40V** DC source that is connected in-series with a **1500Ω** resistor and a **500Ω** resistor.

The specific operations required to create this circuit are described in steps **3** through **10** of this procedure.

Note: for the remainder of this lab exercise, the term “*click*” will be used to specify *left-clicking* the mouse and the term “*right-click*” will be used to specify *right-clicking* the mouse.

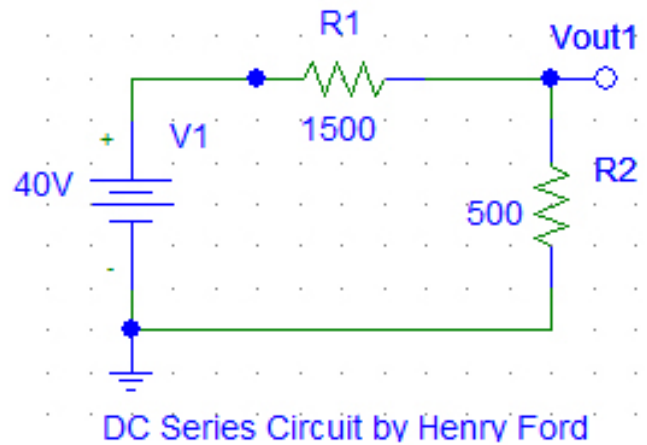
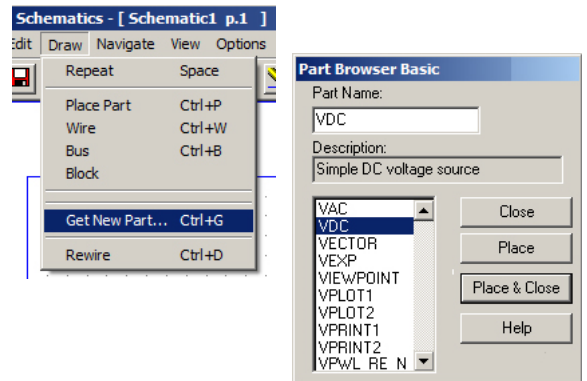


Figure 3.1 – DC Series Circuit

3. First, place a **DC voltage source, V1**, in the *Schematics* window by doing the following:

Click **Draw** from the *Schematics* menu bar, to open the *Draw* drop-down menu, and then click **Get New Part...** to open the *Part Browser Basic* dialog box.



In the *Part Browser Basic* dialog box, scroll the Full List menu until the **VDC** option appears. Click on the **VDC** option.


Note that the option **VDC** could also have been typed directly into the *Part Name:* text box instead of utilizing the scrolling menu.


Click the **Place&Close** command button. The part will now appear in black on the mouse pointer.

Move the mouse to position the DC voltage source in the left center of the schematic workspace and then *click* to place the source. It should appear on the schematic page as source **V1**.

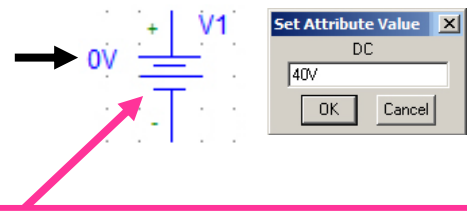
Note that the part still appears in black on the mouse pointer and that additional parts could be placed by repositioning the mouse pointer and *clicking* the mouse again if desired. PSpice will sequentially assign a name to each additional source.

Right-click the mouse to stop placing DC voltage sources. The mouse pointer will return to normal.

Note that the *Part Browser Basic* dialog box can also be opened by *clicking* , the “**Get New Part**” icon, which is located on toolbar above the schematic workspace.

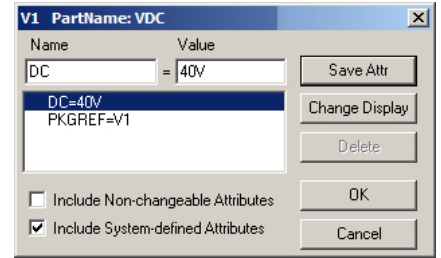
Also note that, once a part is placed, the part also appears in the **Get Recent Part** drop-down menu, , allowing additional parts of the same type to be quickly placed by selecting them from the available choices displayed in this drop-down menu.

- The magnitude of the DC voltage source, **V1**, will default to zero volts (0V). To set its voltage value to 40V, **double-click** on the 0V magnitude (displayed next to the part) and change the value from 0V to 40V in the **Set Attribute Value** dialog box that appears.



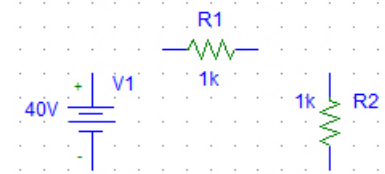
Note that **double-clicking** the part itself will cause a dialog box for the part to appear. All of the parts attributes, including the part's name (**V1**), can be changed within this box.

Be aware that changing many of these value may cause the part to no longer function correctly. For this reason, it is recommended that the **Include Non-changeable Attributes** box be unchecked so that only the user-definable attributes are displayed.



- Next, place **resistors R1** and **R2** by following a similar procedure to that used to place the source, but this time choose part “r” (for resistor) in the **Part Browser Basic** dialog box.

Appropriately position each resistor with mouse, using **<ctrl>-R** to rotate the part on the mouse pointer as needed, and **clicking** the mouse to place the resistor.



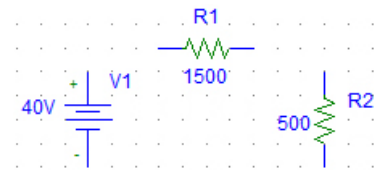
Do not place the parts such that they are all touching each other. Instead, leave a space between all of the parts. Wires will be drawn later in order to connect the parts into a complete circuit.

Right-click to stop placing resistors after both resistors have been appropriately placed.

Note that a placed part can be repositioned by first **clicking** on the part (it will turn red) and then **clicking-and-holding** the left button of the mouse, allowing the part to be dragged to a new position. Also note that a part can be deleted by first **clicking** on the part and then pressing the **<Delete>** key.

- The magnitude of both resistors will default to **1kΩ (1000Ω)**.

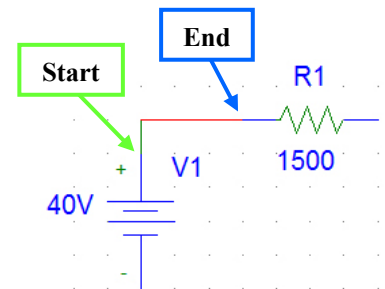
Change the value of **R1** to **1500Ω (or 1.5kΩ)** and change the value of **R2** to **500Ω**.



- Next, **draw wires** to connect the components **V1**, **R1**, and **R2** by performing the following steps:

Click Draw→**Wire** or **click** the **Draw Wire** icon, . A **pencil** will now replace the mouse pointer.

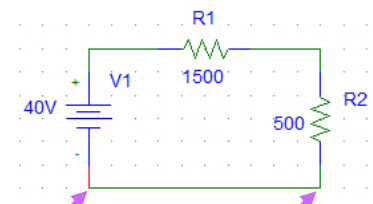
Position the pencil's point at the top of the source, **V1**, and **click** the mouse to begin drawing a wire. Move the pencil to point at the end of the left (blue) terminal of the resistor **R1** and then **click** the mouse again to end that wire. The last portion of the wire should be selected (red) and the pencil should move about freely without the wire moving with it.



If this is not the case, then the pencil was not pointed at the terminal-end of the part when the mouse was **clicked**... press **<ctrl>-Z** and try again.

Next, draw a wire from the right terminal of the resistor **R1** to the top terminal of the resistor **R2** in the same manner that the previous wire was drawn.

Then, draw the last wire from the bottom terminal of resistor **R2** to the bottom terminal of the source **VDC**.

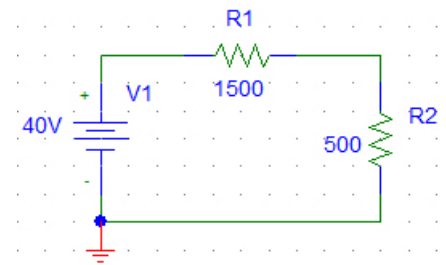


Note that, after starting to draw a wire, **clicking** the mouse at a blank location will draw the wire to that point, after which further movement of the mouse will cause the wire to continue from that location.

Right-click to stop drawing wires once all three of the wires have been drawn.

8. In order to PSpice to be able to analyze the circuit, a **ground node** must first be defined.

Ground the negative terminal of the source by choosing part “**GND_EARTH**” in the **Part Browser Basic** dialog box and appropriately positioning the part at the bottom-left node.

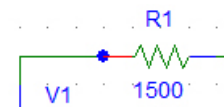


Right-click to stop placing ground symbols.

Note that, after the **GND_EARTH** part is placed, a blue dot will appear at the location where the ground symbol connects to the bottom wire. PSpice places a blue dot at any node to which **three or more** parts or wires are connected.

If a blue dot appears at any location where only two parts/wires seem to connect, an extra wire may have mistakenly been drawn on top of another. If the extra wire is not deleted, PSpice may report an error when analyzing the circuit.

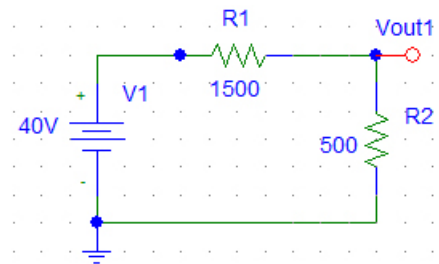
For example – in the case of the blue dot shown to the right, and extra wire stub (highlighted red) was mistakenly drawn on top of the left-side of resistor **R1**. If this wire stub is not deleted, PSpice may not be able to analyze the circuit,



9. Next, place an output connection bubble by selecting the new part “**BUBBLE**”, pressing **<ctrl>-R** three times to rotate the connection bubble to the correct orientation, and appropriately positioning the part using the mouse.

Right-click to stop placing bubbles.

Assign a name to the bubble by **double-clicking** on the symbol for the connection bubble and entering “**Vout1**” in the **Set Attribute Value** dialog box that appears.






10. Insert the text “**DC Series Circuit**” underneath the circuit by performing the following steps:

Click Draw to open the **Draw** drop-down menu and choose the **Text...** option. In the **Add Text** dialog box that appears, type “**DC Series Circuit by <yourname>**” and **click OK**.

An outline of a textbox should appear at the mouse pointer. Position the box at the desired location, **click** the mouse to place the box, and then **right-click** to stop placing text boxes. Your circuit should now look similar to the **DC Series Circuit** shown at the top of page two of this lab exercise.

11. Check to be sure that your circuit is the same as the circuit shown in Figure 3.1.

Note – to change the display size of the circuit shown in the *Schematics* worksheet, *click* on the **Zoom In** icon, , or the **Zoom Out** icon, .

Also, the **Zoom to Fit Page** icon, , will scale the entire drawing to fit the window.

12. To the right of the **DC Series Circuit** in the *Schematics* window, create the **DC Parallel Circuit** shown in Figure 3.2.

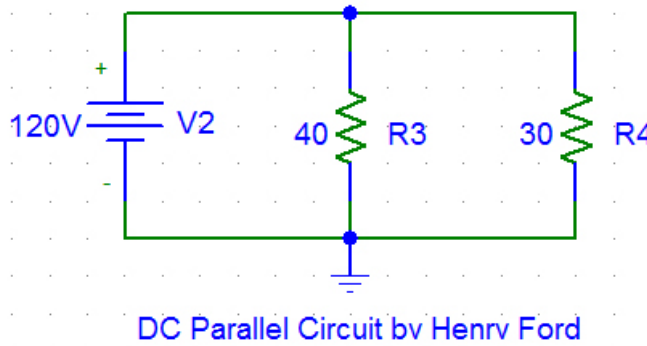


Figure 3.2 – DC Parallel Circuit


Note that the thickness of the lines representing the parts and wires in Figure 3.2 has been increased. To change the line thickness in your circuits, Click **Options** from the *Schematics* menu bar and then choose **Display Preferences...** to open the *Display Preferences* dialog box.

Click on the **Graphics** tab and adjust the line Width to the next larger setting for the Display Layers: Pins, Stimulus, Symbols, and Wires


Use the techniques that were performed in steps 3 through 10 to create the new **DC Parallel Circuit**. Set the value of the DC voltage source, **V2**, to **120V**. Set the value of resistor **R3** to **40Ω** and the value of resistor **R4** to **30Ω**. Be sure to create the appropriate text below the circuit.


Note that your parts may be named differently if you placed and then deleted extra parts when constructing the circuit shown in Figure 4.1. The name of any part placed within a circuit may be changed by first double-clicking on the part name (i.e. – **V2**) and changing the **Package Reference Designator** (name) that appears in the *Edit Reference Designator* dialog box.

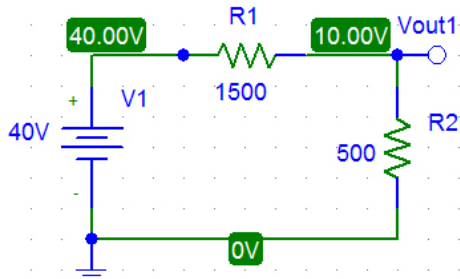
13. When both circuits are complete, save your schematic as “**3000_Lab04_LastName**” where “LastName” is your last name. You are now ready to have PSpice analyze the circuits.

14. *Click* **Analysis** from the *Schematics* menu bar and choose **Setup...** or *click* the **Setup** icon . The *Analysis Setup* dialog box that appears allow the user to select and configure the analysis types that PSpice will perform. **Bias Point Detail** should be the only analysis enabled. *Close* the window.

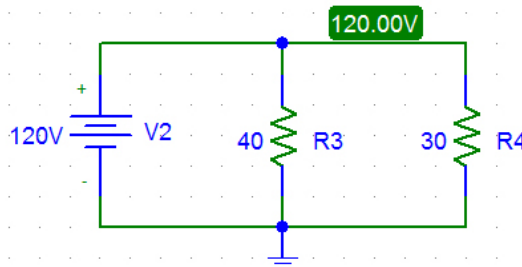
A **Bias Point Detail** instructs PSpice to determine the DC (average) voltage present as each node in the circuits along with the DC (average) current flowing into each of the circuit components.

15. **Click Analysis** from the *Schematics* menu bar and choose **Simulate** or click the **Simulate** icon .

16. Once the simulation is complete, **click** the **Enable Bias Voltage Display** icon  in the *Schematics* window to display the bias voltages on the circuits. (Note that the bias voltages may automatically display when the simulation is complete. When enabled, the bias voltages (node voltages) are shown for each node in the circuits.)



DC Series Circuit by Henry Ford




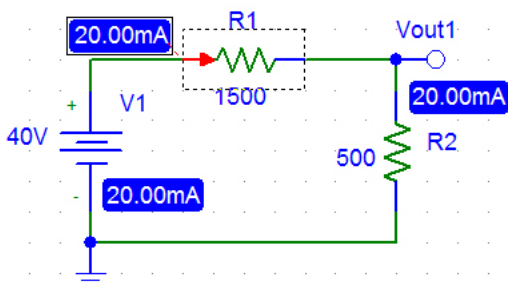
DC Parallel Circuit by Henry Ford

The locations of the displayed bias voltages can be changed by first **clicking** on the target voltage and then **dragging** the bias voltage to a new location. While dragging the displayed voltage, the associated node will be highlighted in pink and a dashed red line will connect the voltage to the node.

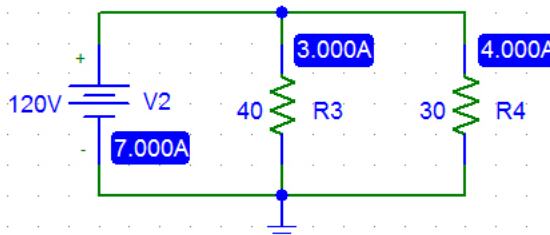
Print/capture a copy of the circuits displaying the bias voltage results for submission.

Click the **Enable Bias Voltage Display** icon  a second time to hide the bias voltages.

17. **Click** the **Enable Bias Current Display** icon  in the *Schematics* window to display the bias currents on the circuits. When enabled, the bias currents flowing into each of the components are shown on the circuits. Clicking on any one of the currents will cause an arrow to be displayed that shows the direction that the current is flowing.




DC Series Circuit by Henry Ford




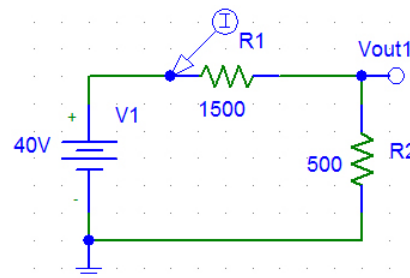
DC Parallel Circuit by Henry Ford

Print/capture a copy of the circuits displaying the bias current results for submission.


Click the **Enable Bias Current Display** icon  a second time to hide the bias voltages.

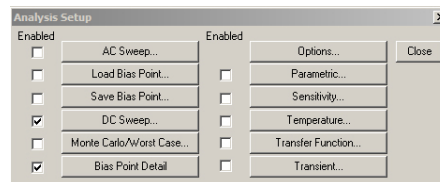
18. **Click** the **Current Marker** icon  and use the mouse to position the tip of a current marker on the left terminal of **R1**. By placing a marker (probe) at this position, PSpice is instructed to automatically plot the current at this location when a **sweep analysis** is performed.

Although you are not instructed to do so at this point in time, a voltage marker can be placed on any node in the circuit by **clicking** the **Voltage/Level Marker** icon .



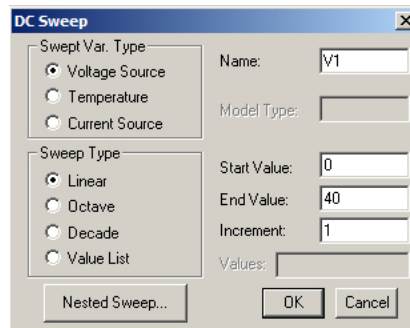
19. Complete a **DC Sweep** simulation on the series DC circuit by performing the following steps:


Click the **Setup** icon . Enable a **DC Sweep** and click on the **DC Sweep** button to configure the sweep parameters.



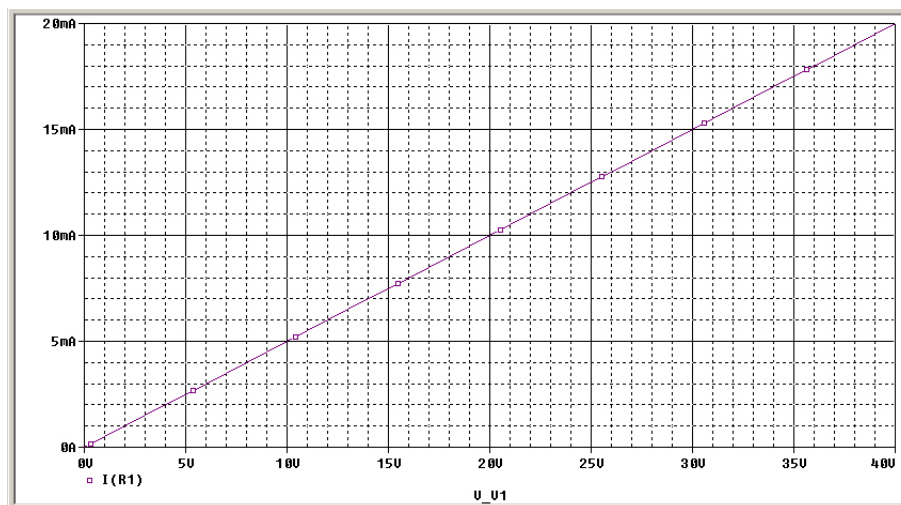
20. Configure the parameters in the **DC Sweep** dialog box.

Click **OK** when finished and then click the **Close** button in the **Analysis Setup** dialog box.



21. To start the simulation, click the **Simulate** icon .

When a successful simulation is finished, the PSpice waveform viewer (Probe) should display a plot of the current **I(R1)** [mA] as a function of the source voltage **V1** [V] similar to that shown below:



Print/capture a copy of the circuits displaying the bias current results for submission.

Close the PSpice waveform viewer by **clicking** the **Close** icon in the uppermost right-hand corner of the window. This should return you to the **Schematic** screen.

22. **Submit** a printed copy of:

- The circuits displaying the bias voltages,
- The circuits displaying the bias currents, and
- The final current vs. voltage plot

by the posted deadline for the lab experiment.