**Trim-potentiometers:**

With a trimpot, the resistance from the two outside pins is always the resistance indicated on the trimpot, 10k in this case. The resistance from the inside pin to one of the outside pins varies from 0 - 10k (let's call this x). The resistance of the other outside pin to the inside pin is then 10k(1 - x).

For lab 6, draw this out in a picture in terms of two resistors and look at what happens when you ground pin 1.

**To simulate a trim-potentiometer in Pspice:**

1) Build your circuit with the components you used before except for the trimpot.

2) Use the part, R_var, for the trimpot.

![Diagram of trimpot](image)

3) Change the numerical default value, 1k, to RVAR

4) Double click on the R_var resistor or right click and go to edit properties

5) Find the column that says SET, change value from 0.5 to 1

<table>
<thead>
<tr>
<th>Reference</th>
<th>SET</th>
<th>Source Library</th>
</tr>
</thead>
<tbody>
<tr>
<td>1)</td>
<td>R3</td>
<td>C:\PROGRAM FILES\ORCAD_DEMOCAPTURE\LIBRARY \</td>
</tr>
</tbody>
</table>

6) Add a new item called PARAM, it is in the SPECIAL library.

7) Double click on Parameters and click on New Column.
8) Enter the following information and press OK. (Note: the value does really matter, just make sure it is in the range of your trimpot.)

   ![Add New Column Window]

9) Click the X to close the window.

10) Set up your simulation profile for a DC sweep and use a global parameter.

   ![Simulation Settings Window]

10) Place a voltage probe on the node where you are looking for an output voltage as a function of the trimpot resistance.

   ![Probe Symbol]

   Now run the simulation and you should get a graph the same as you would have graphed with an equation.