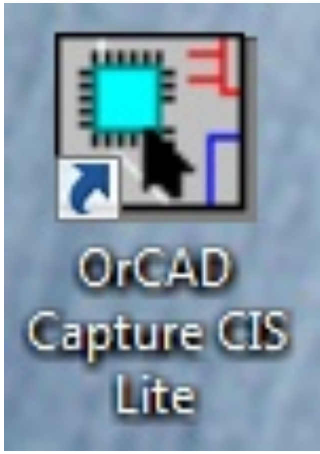
PSPICE Lecture #3

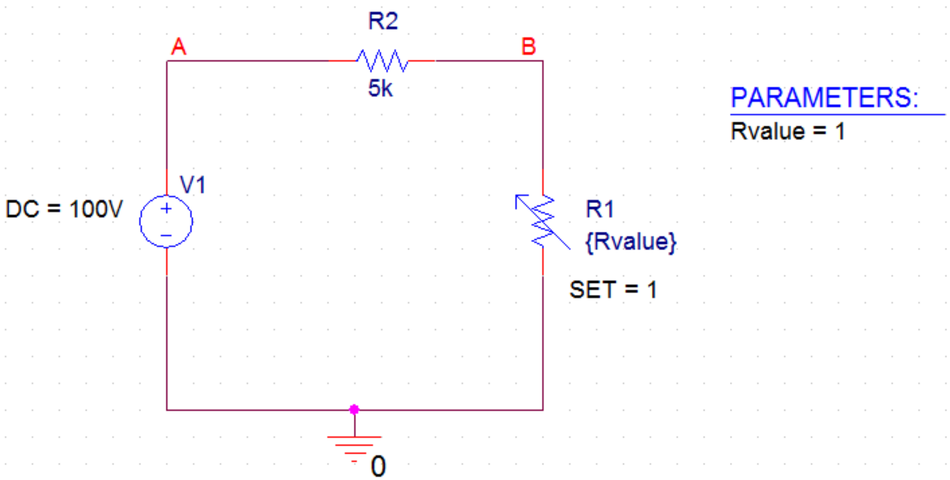


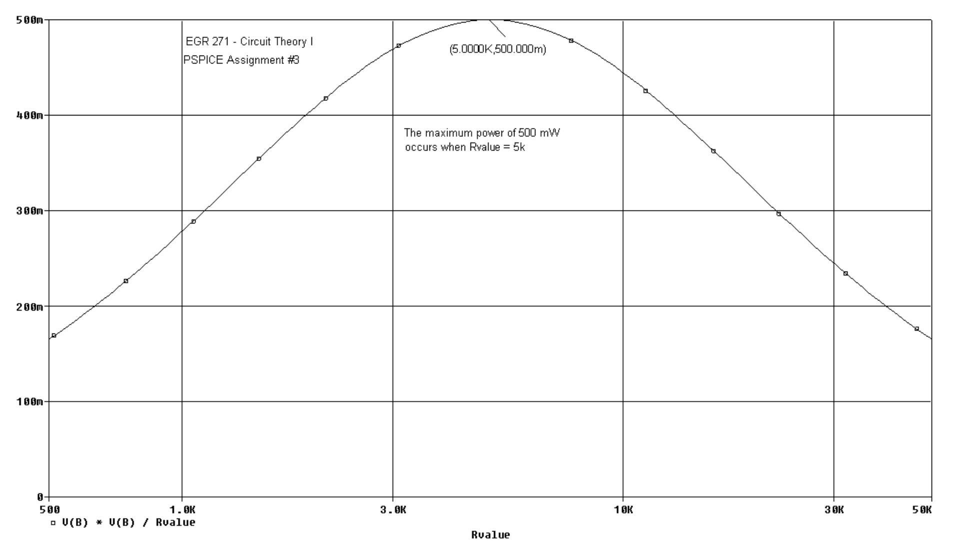
Reference: (see course web site)

- ***Sample PSPICE Report***
- PSPICE Example: ***Max Power Transfer - Varying a Resistor***
- PSPICE Assignment #3

Topics to be presented:

- Varying components in PSPICE
- Maximum Power Transfer Theorem





DC Sweep

A DC Sweep analysis in PSPICE can be used to vary a:

- Voltage source
- Current Source

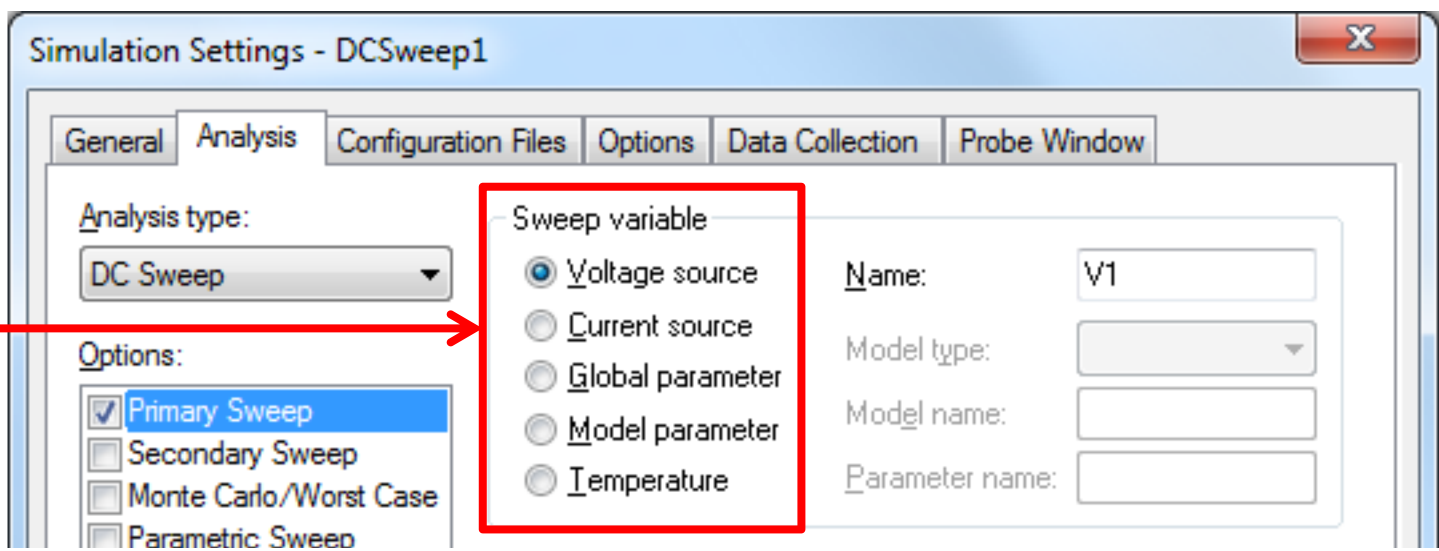
Covered in PSPICE Lecture #2

- Global parameter (such as a resistor, inductor, or capacitor)
- Model parameter (such as a constant in a PSPICE model)

Covered here in PSPICE Lecture #3

- Temperature

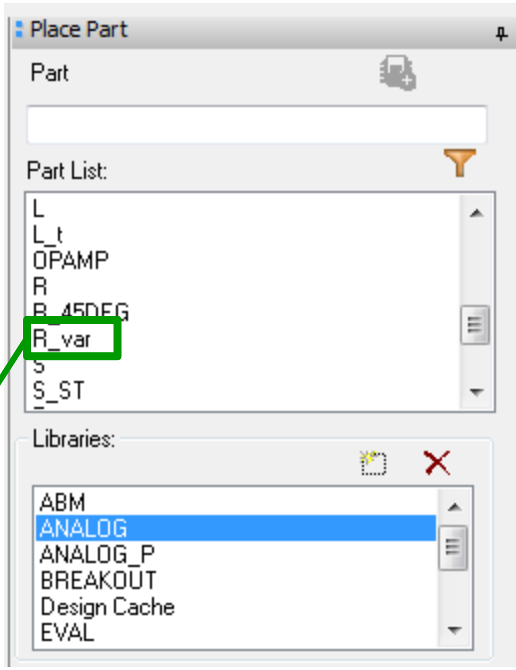
Quantities that can be varied using a DC Sweep

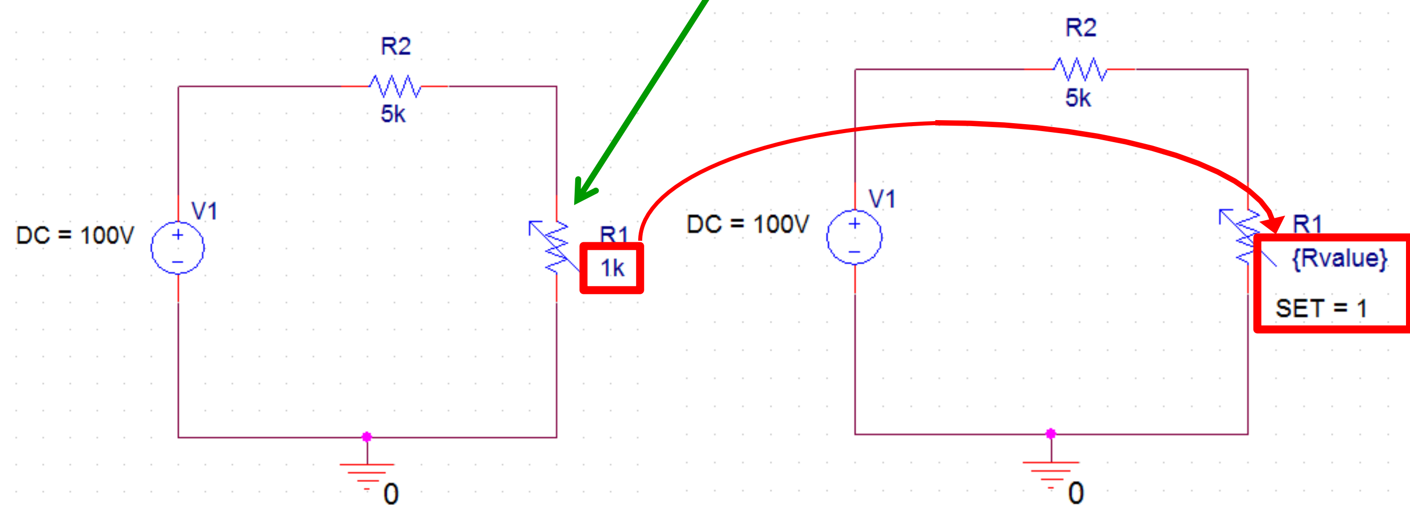


Varying a component (Global Parameter) in PSPICE

Follow these steps to vary a component in PSPICE:

1) Draw the schematic using a ***variable part*** for the component to be varied. For example, use part ***R_var***, not R. Similarly, use part C_var to vary a capacitor.
2) Change the value of the part to a name in braces. For example, change 1k to {Rvalue}.
3) Change SET. Double-click on the part and change the property named SET from 0.5 to 1. Also display this property. The value of R is actually multiplied by SET, so using SET = 0.5 is confusing.

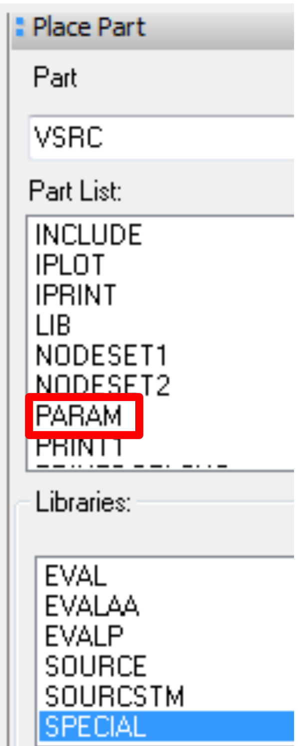


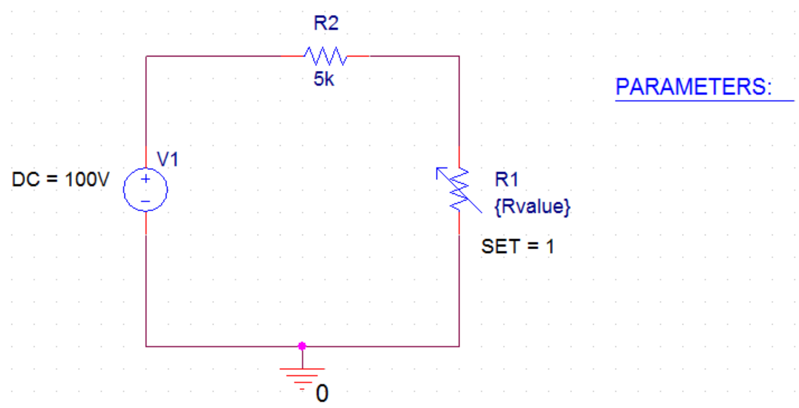


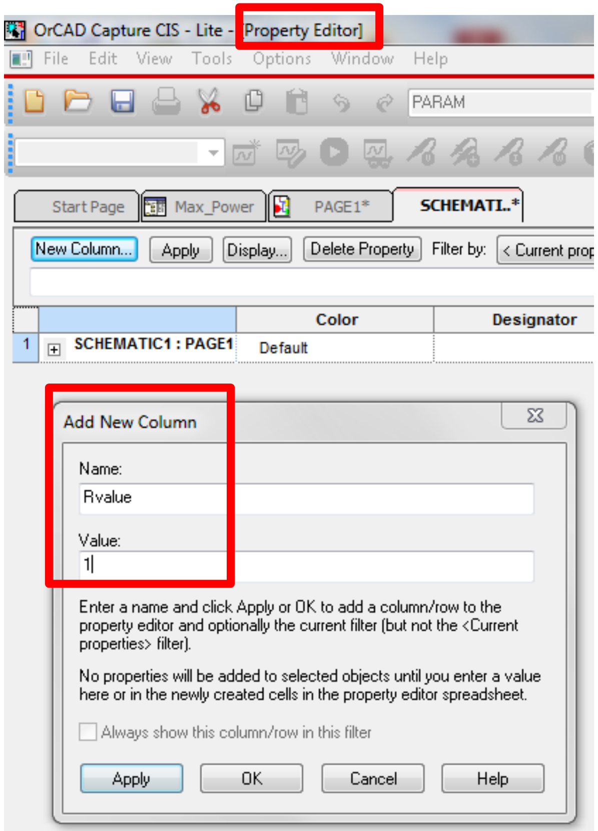
Varying a component (Global Parameter) in PSPICE (continued)

4) Add a part named PARAM. Place the part (it will appear as PARAMETERS) on the schematic next to the circuit. PARAM is located in the SPECIAL library.

5) Add a property (column) to PARAM.
- Double-click on PARAMETERS to open the Property Editor.
- Select ***New Column*** to add a new property to PARAM.
- Name the New Column ***Rvalue*** (the name usesd for the variable resistor.
- Give the New Column a Value of 1 (any value).



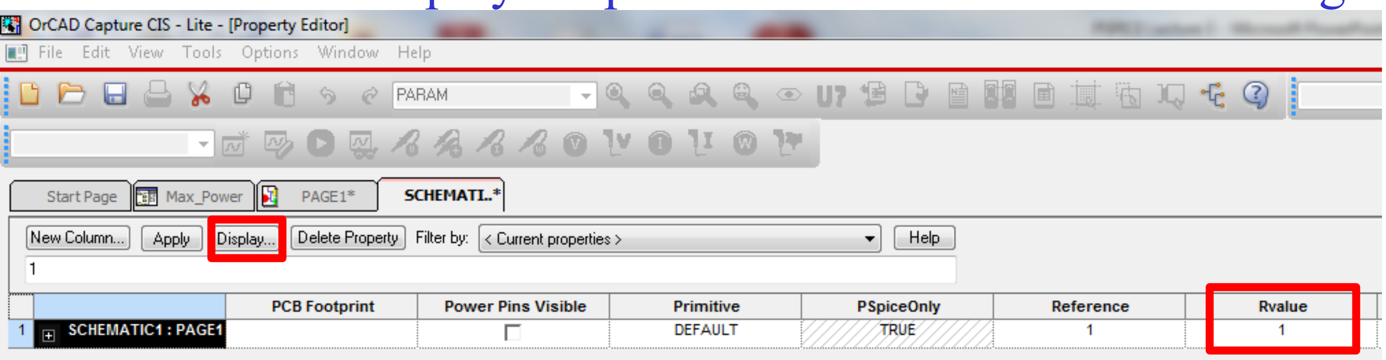


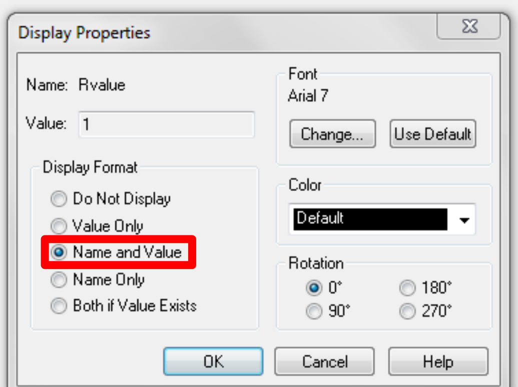


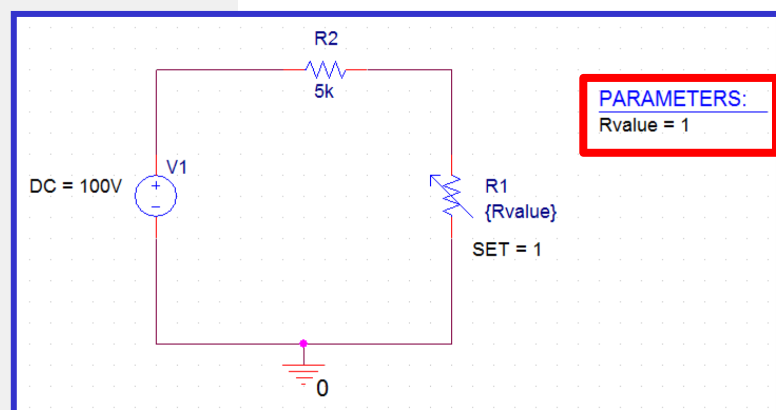
Varying a component (Global Parameter) in PSPICE (continued)

6) Display the new property. The new property just added will not be shown by default. In general, always display and new values added or any values that are altered in PSPICE. To display the property:

- Double-click on PARAMETERS to open the Property Editor.
- Scroll through the properties to find and select the new property added (Rvalue).
- Select ***Display*** to open the Display Properties window. Select ***Name and Value***.
- Close the Display Properties window and note the change to the schematic.



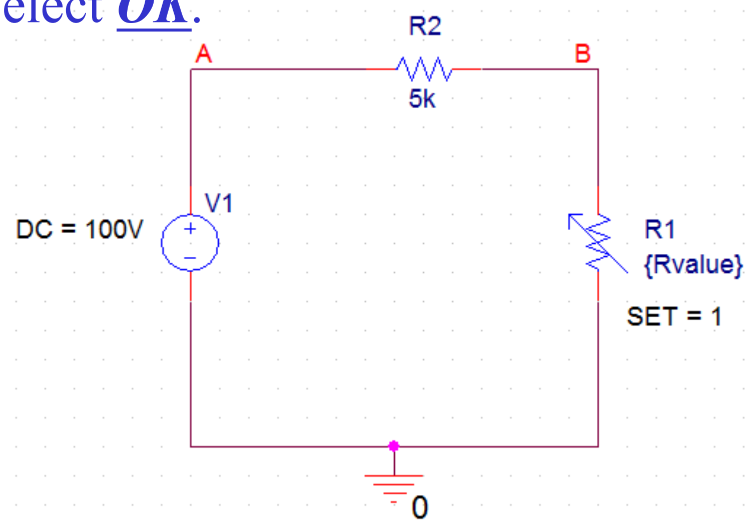


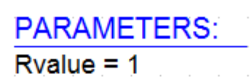


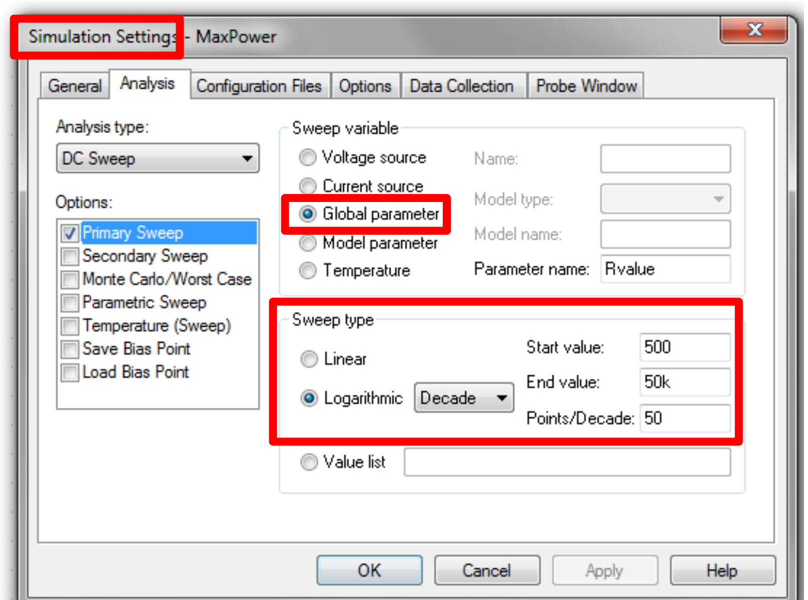
Varying a component (Global Parameter) in PSPICE (continued)

7) Create a Simulation Profile.

- Select ***PSPICE*** – ***New Simulation Profile***
- Change the ***Analysis type*** to ***DC Sweep***
- Select ***Global parameter***
- Select ***Logarithmic*** for this example and vary Rvalue from 500 to 50k.
- Using ***50 points/Decade*** will result in a total of 100 points in this example.
- Select ***OK***.







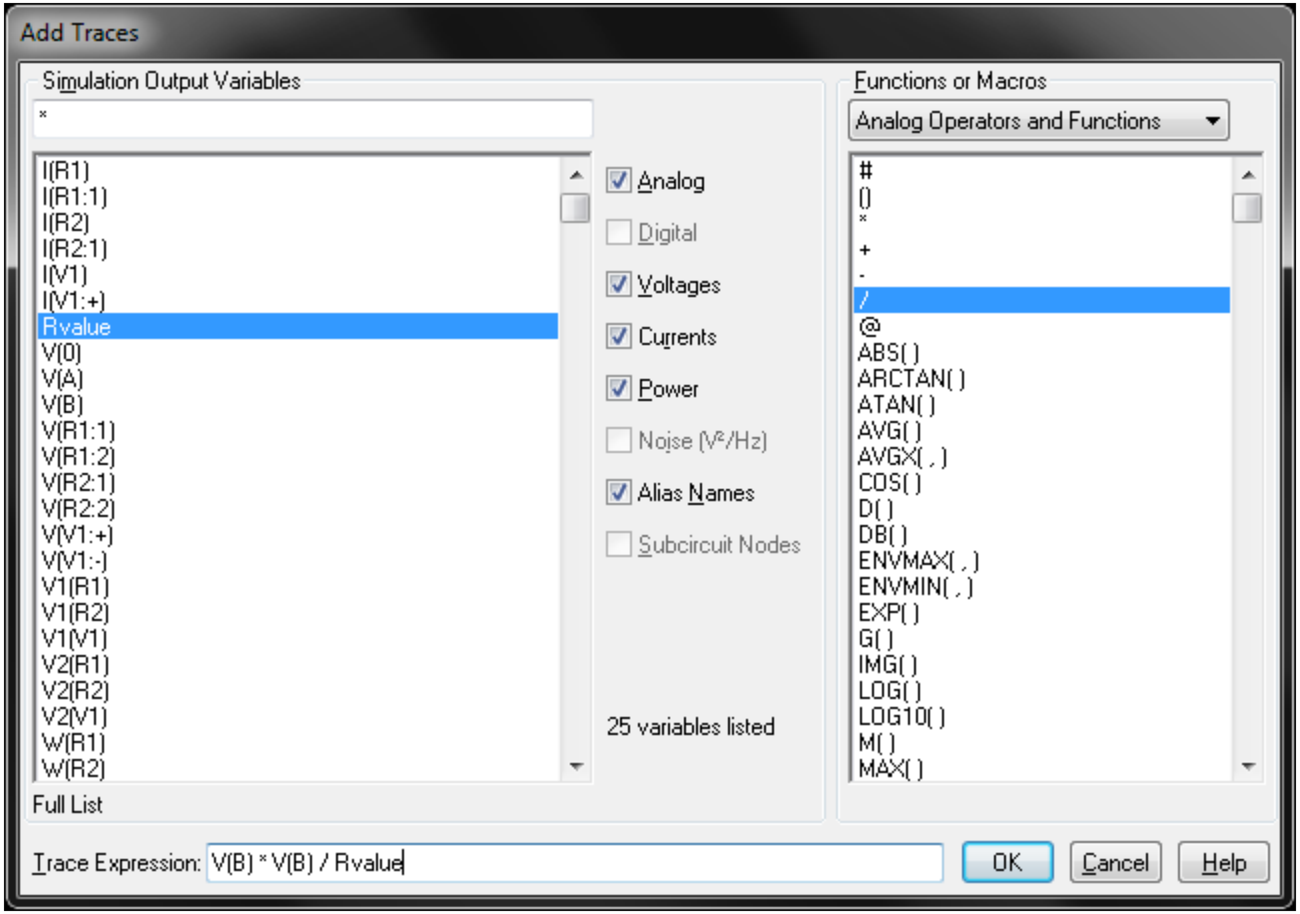
Note that nodes A and B were labeled on the schematic. This will make it easier to refer to the output voltage, V(B), later.

Varying a component (Global Parameter) in PSPICE (continued)

8) Analyze the circuit. Select ***PSPICE*** – ***Run***

9) Graph Power vs Resistance for R1

   a) Select Add Traces and enter the expression for the power to resistor R1: ***V(B)*V(B)/Rvalue***



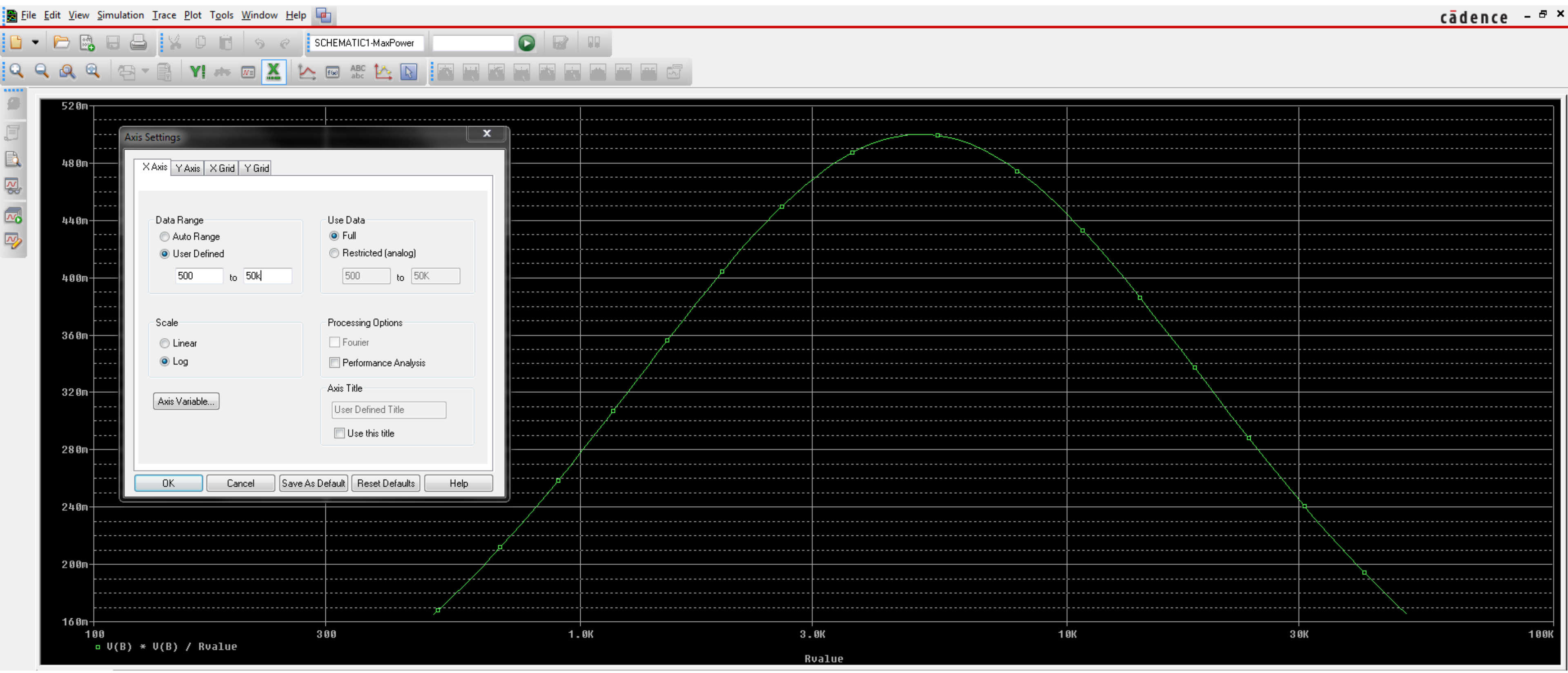
Varying a component (Global Parameter) in PSPICE (continued)

9. Graph Power vs Resistance for R1

   b) Select ***Add Traces*** (or use toolbar) and enter the expression for the power to resistor R1: ***V(B)*V(B)/Rvalue***

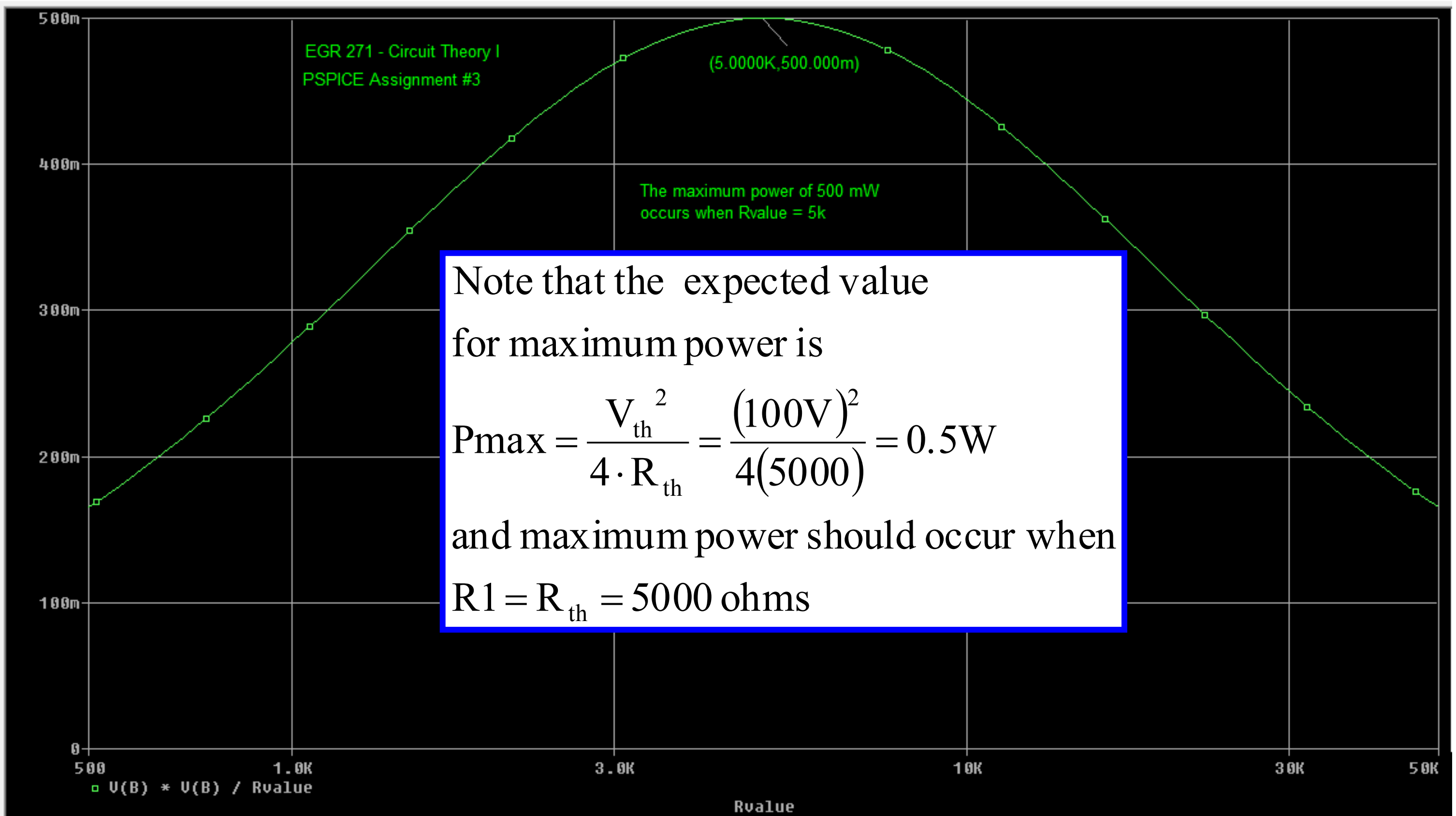
   c) Select ***Plot – X-axis*** and change the range to 500 to 50k

   d) Select ***Plot – Y axis*** and change the range to 0 to 500m.



Varying a component (Global Parameter) in PSPICE (continued)

9. Graph Power vs Resistance for R1
   e) Select ***Trace – Cursor – Display*** (or use toolbar) to turn on the cursor
   f) Select ***Trace – Cursor – Peak*** (or use toolbar) to move the cursor to the peak
   g) Select ***Plot – Label – Mark*** (or use toolbar) to mark the point
   h) Select ***Plot – Label – Text*** (or use toolbar) to add text to the graph



Note that the expected value for maximum power is

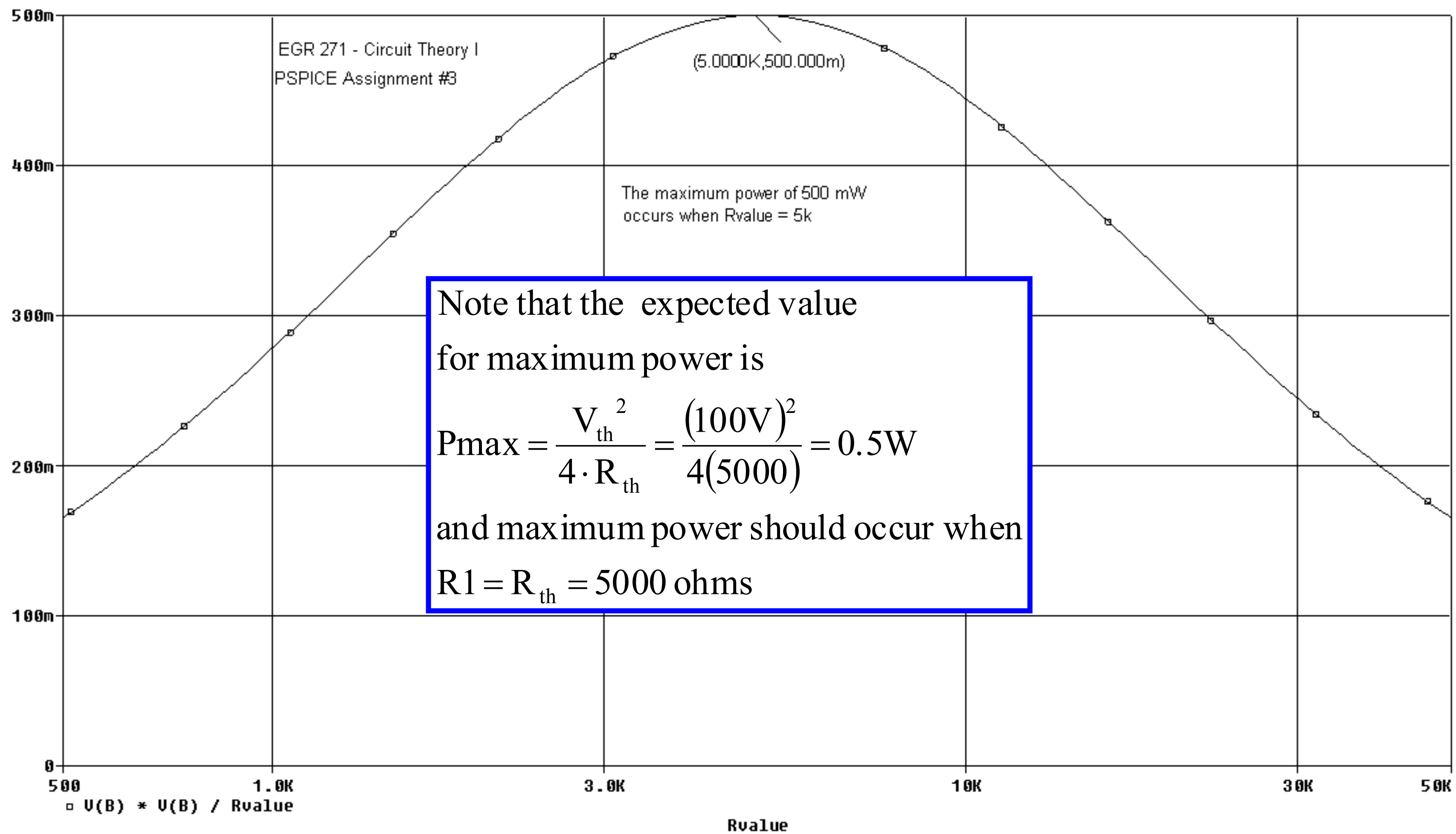
$$Pmax = \frac{V_{th}^{\ 2}}{4 \cdot R_{th}} = \frac{(100V)^2}{4(5000)} = 0.5W$$

and maximum power should occur when $R1 = R_{th} = 5000$ ohms

Varying a component (Global Parameter) in PSPICE (continued)

9. Graph Power vs Resistance for R1

i) Select ***Window – Copy to Clipboard*** to copy the graph (with a white background) to the clipboard where you can paste it into Word or elsewhere.



Note that the expected value for maximum power is

$$Pmax = \frac{V_{th}^{\,2}}{4 \cdot R_{th}} = \frac{(100V)^2}{4(5000)} = 0.5W$$

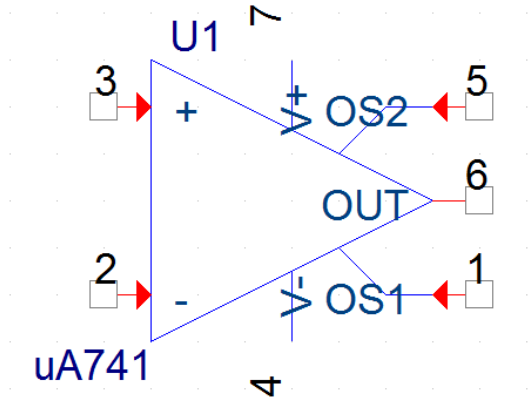
and maximum power should occur when

$R1 = R_{th} = 5000$ ohms

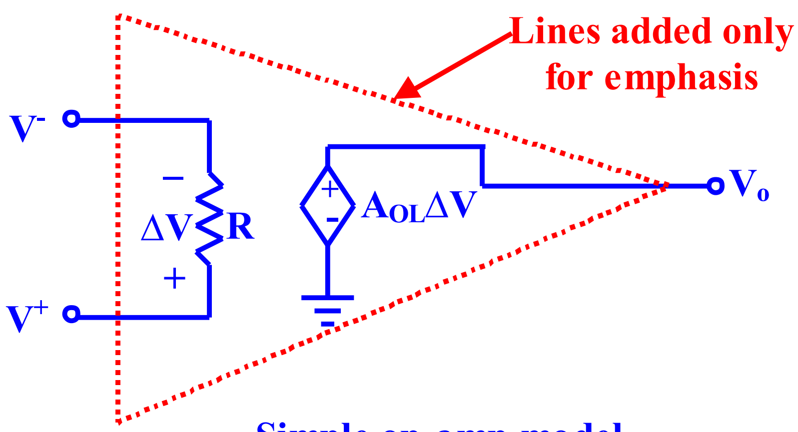
Operational Amplifiers

Operational amplifiers can be analyzed in PSPICE using different models, including:

1) Using specific part from the EVAL library, such as the uA741



2) Use a general op amp circuit model consisting of a dependent source and a resistor



Simple op-amp model

Typical values for the op amp model shown:

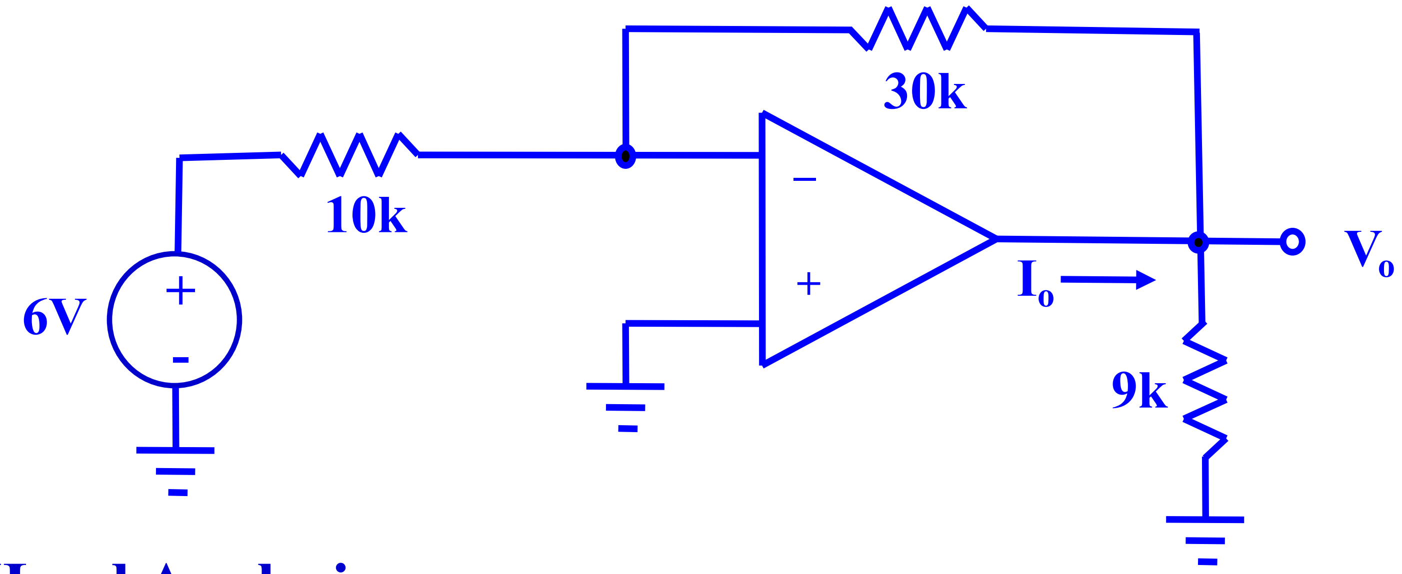
$A_{OL} = 100{,}000$

$R = 2M\Omega - 10M\Omega$

Example

Analyze the following op amp circuit (find V_o and I_o):

1) By hand
2) Using PSPICE with the uA741 op amp
3) Using PSPICE with a general op amp model (dependent source and resistor)



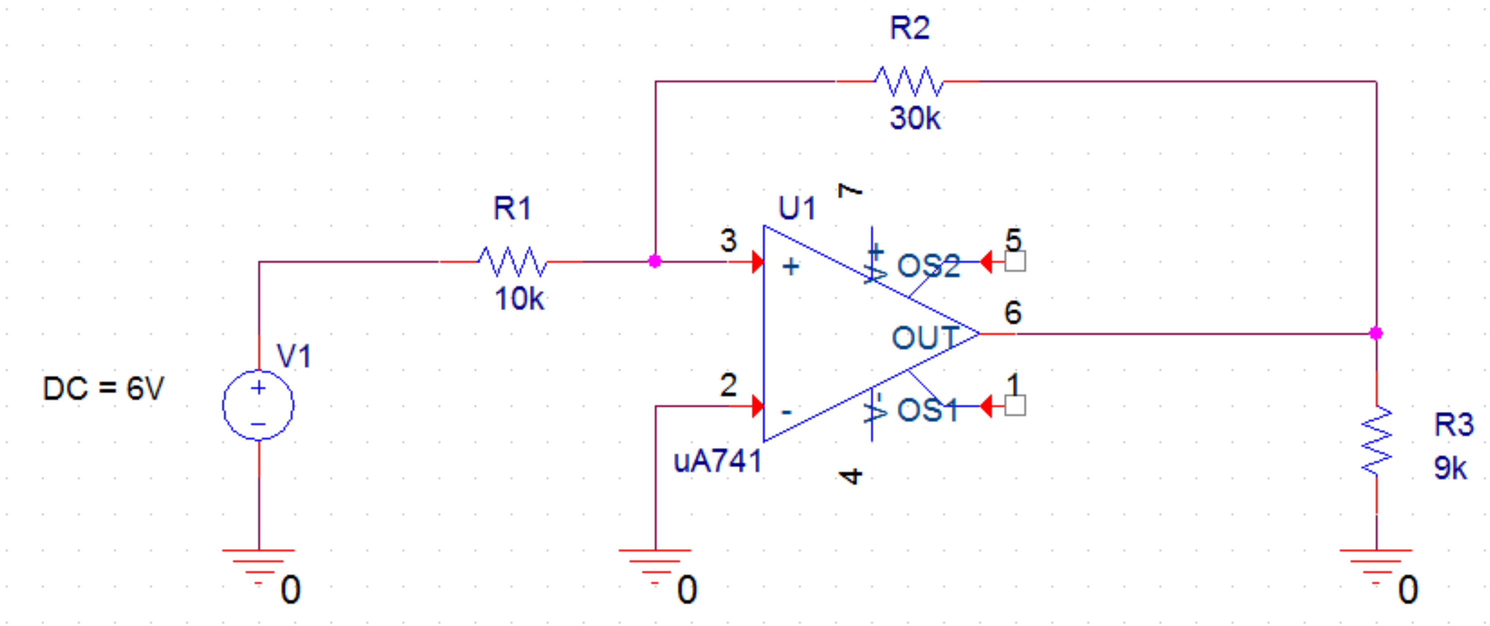
Hand Analysis:

$$\text{KCL, inverting input (out} = +): \frac{0-6}{10k} + \frac{0-V_0}{30k} = 0, \quad \text{so Vo} = -18\text{V}$$

$$\text{KCL, output (out} = +): \frac{-18-0}{30k} + \frac{-18}{9k} - I_o = 0, \quad \text{so } I_o = -2.6\,\text{mA}$$

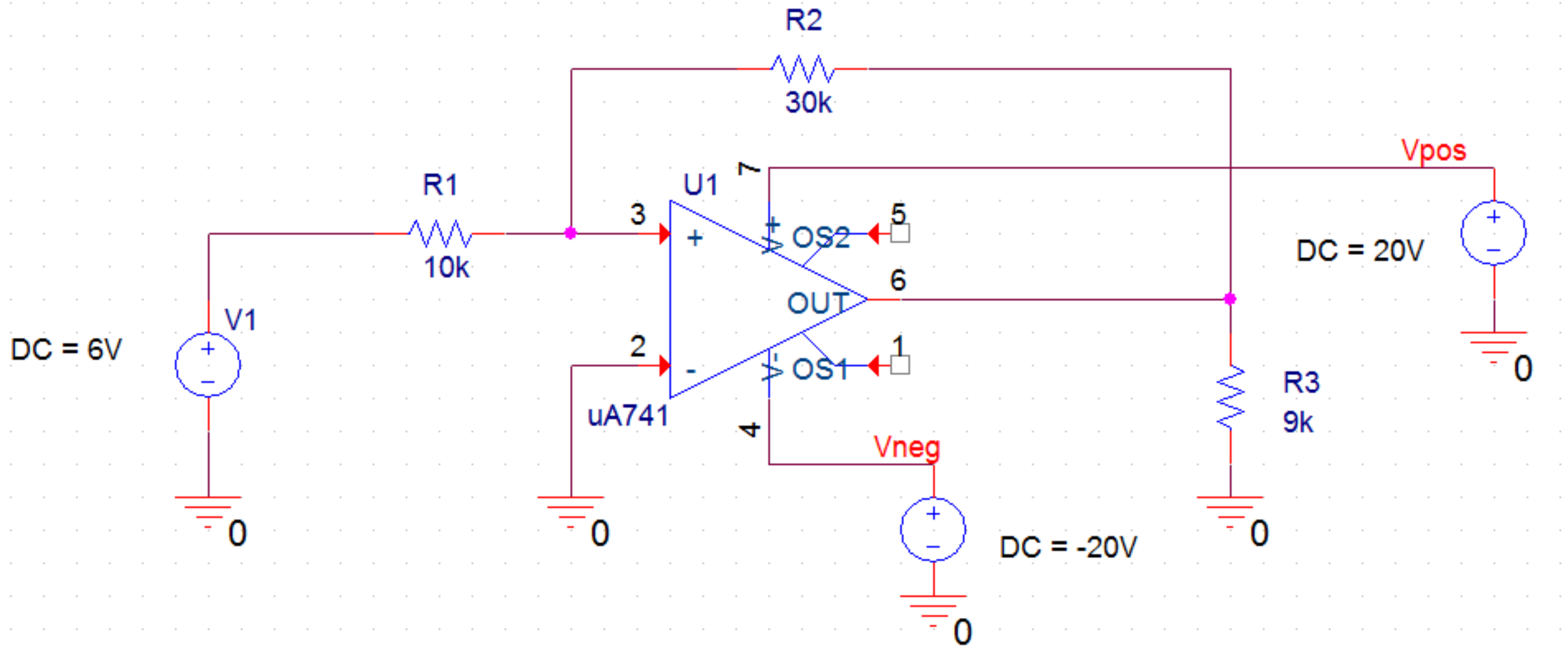
PSPICE analysis with the uA741 op amp:

1) Draw the schematic. Use the uA741 from the EVAL library.

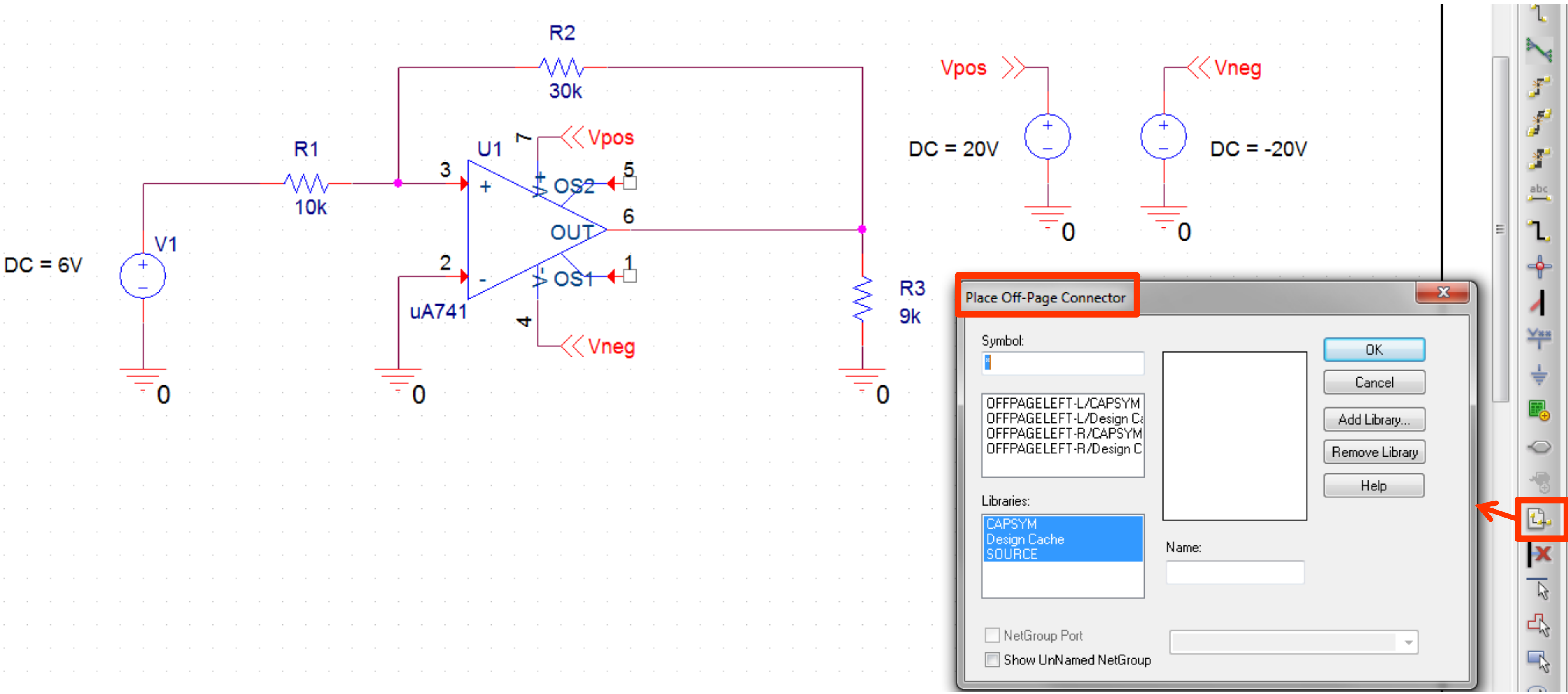


2) Ignore the connections labeled OS1 and OS2. In practical lab situations, an adjustable resistor (potentiometer) can be connected between these terminals to "zero" the op amp (to set the output to 0V when the input is 0V). This is somewhat like zeroing your bathroom scale.

3) Add voltage sources to power the op amp. The value of the voltage sources depends on Vo. In general, the source voltages should be greater than V_o. In practical situations, it is recommended that they be greater by at least 2V. Since V_o = -18V, supply voltages of +20V and -20V have been added below.

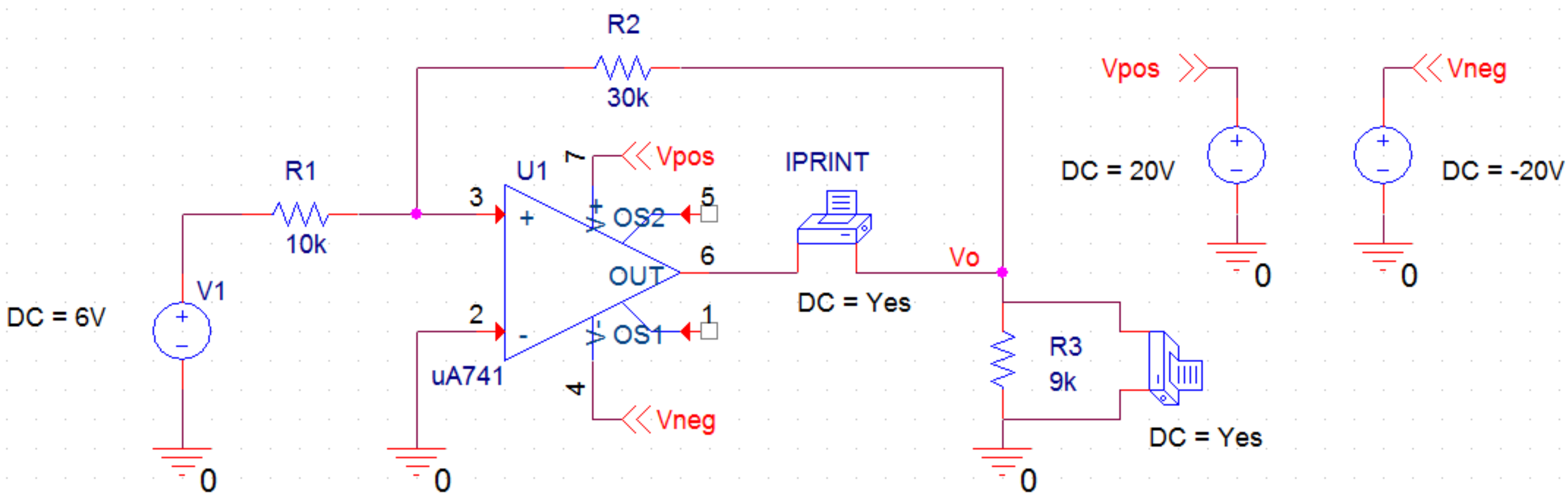


<u>Note</u>: To avoid crowding, the supply voltages can be placed to the side and connected to the circuit using OFFPAGE Connectors. Note the name of the OFFPAGE connector also serves as a node label.

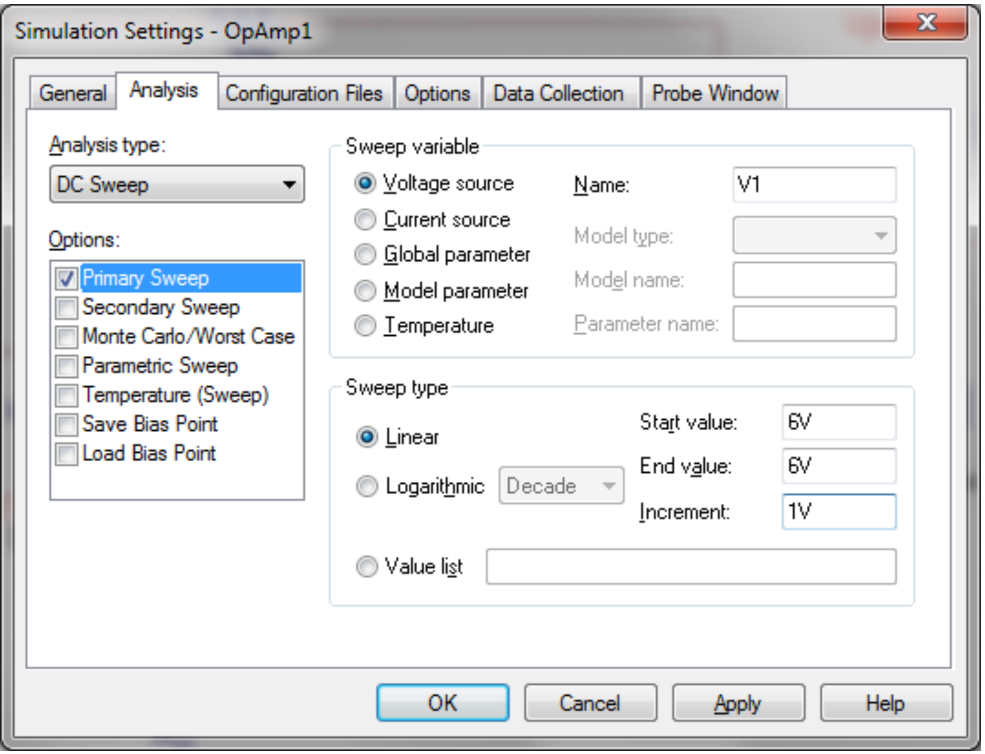


4) Add voltage and current printers to measure Io and Vo.

- Be sure to change the ***DC property*** on each printer to ***Yes*** and display the property.
- Be sure to place the ***current printer in series*** and place the ***voltage printer in parallel***.
- It is also a good idea to label the node for the output voltage as Vo.



5) Create a New Simulation Profile, Run PSPICE, and view the results in the OUTPUT file.



A portion of the .OUT file

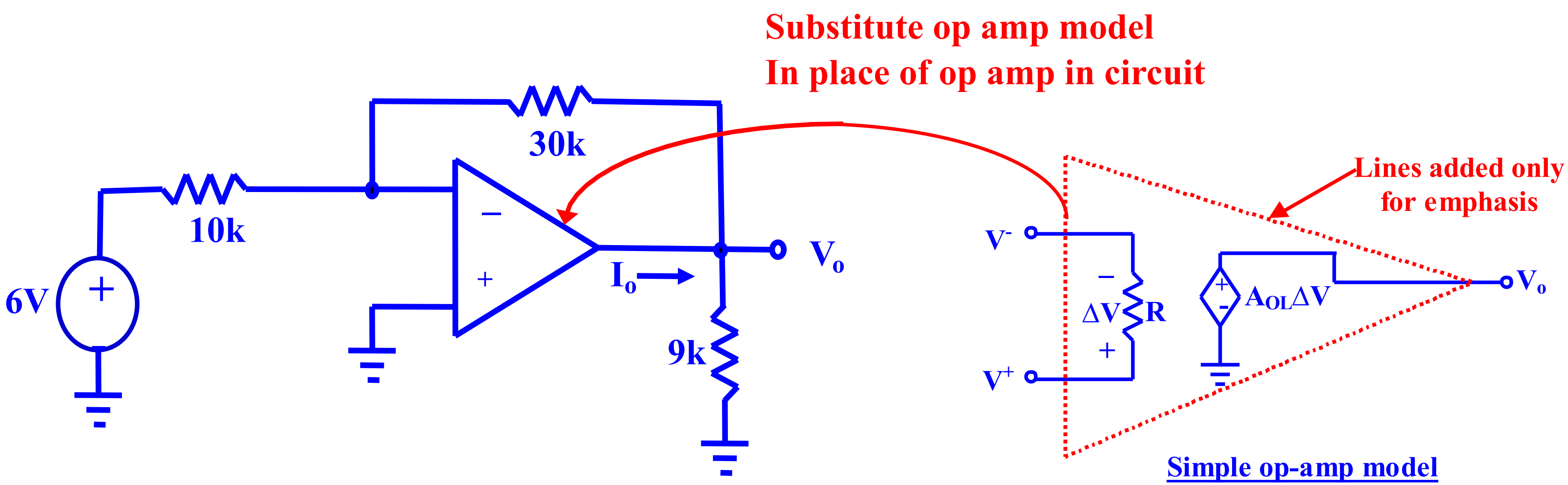
```
*Libraries:
* Profile Libraries :
* Local Libraries :
* From [PSPICE NETLIST] section of C:\OrCAD\
.lib "nomd.lib"

*Analysis directives:
.DC LIN V_V1 6V 6V 1V
.PROBE V(alias(*)) I(alias(*)) W(alias(*)) I
.INC "..\SCHEMATIC1.net"


 V_V1         I(V_PRINT1)

  6.000E+00   -2.600E-03


 V_V1         V(VO,0)

  6.000E+00   -1.800E+01
```

Note that the results match the hand analysis:

V_o = -18V and I_o = -2.6 mA

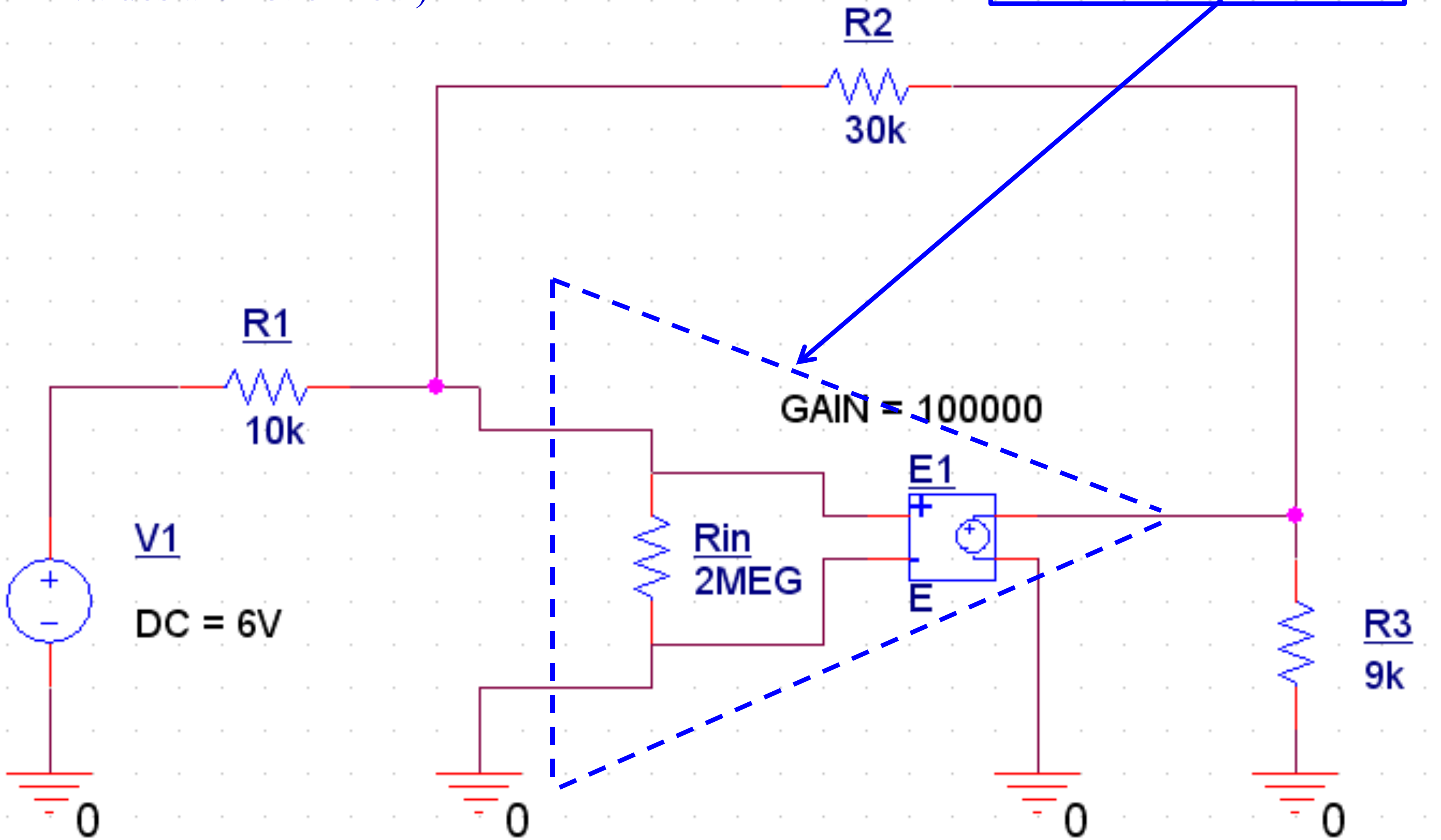
PSPICE analysis using a model consisting of a dependent source and a resistor:

1) Draw the schematic. Use R_{in} = 2MΩ and A_{OL} = 100,000



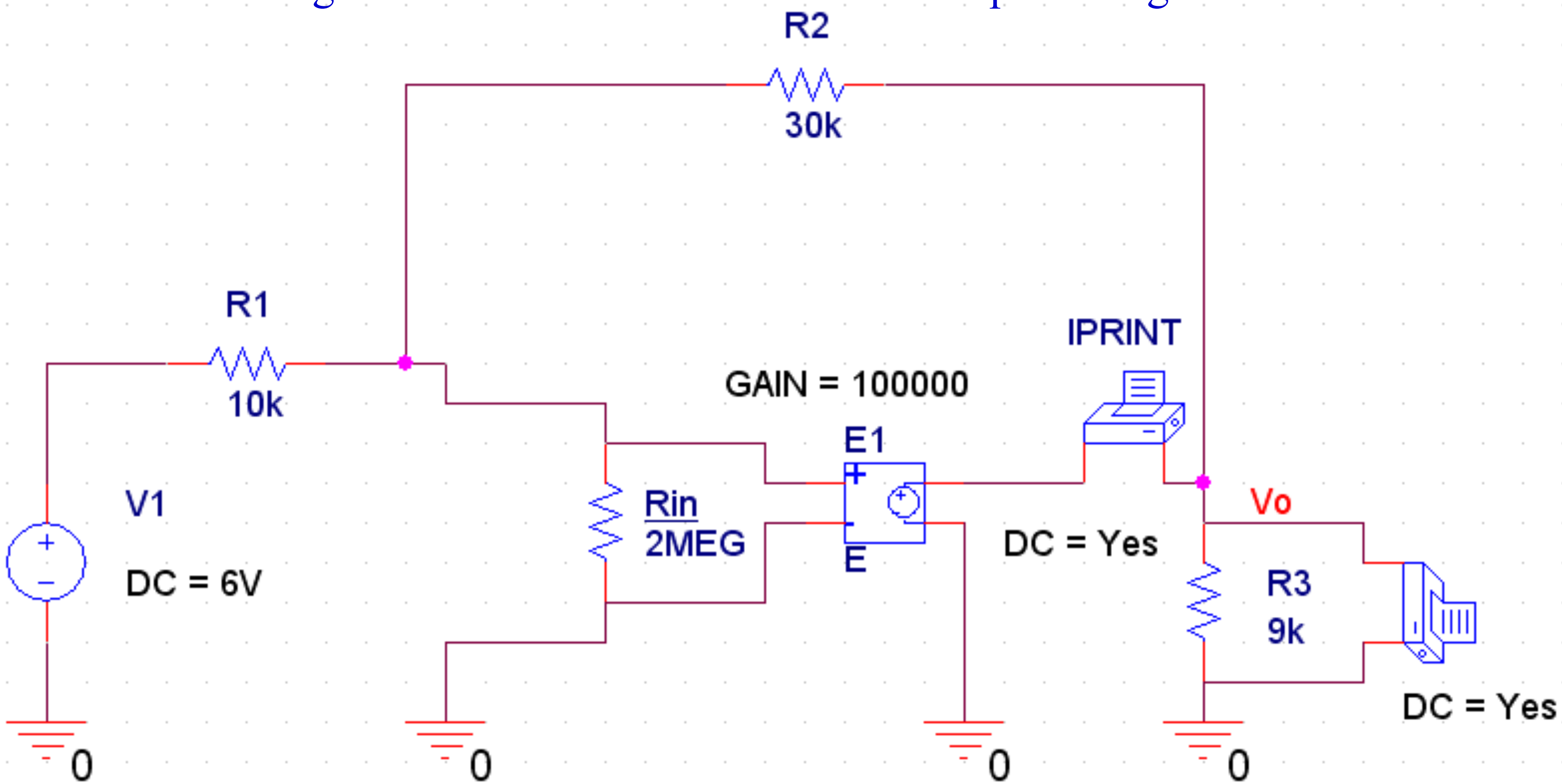
1) Draw the schematic (continued).
Use R_{in} = 2MΩ and A_{OL} = 100,000 (exact values are not critical)

Dotted lines not part of schematic. Added for emphasis.

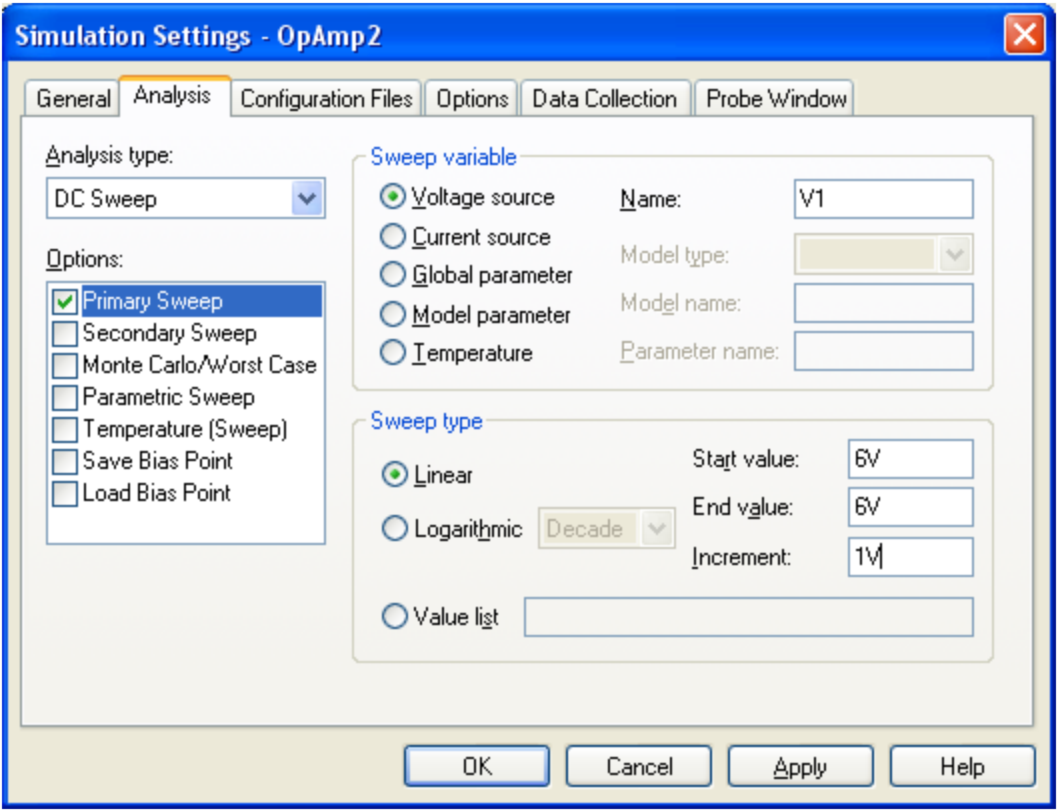


2) Add voltage and current printers to measure Io and Vo.

- Be sure to change the ***DC property*** on each printer to ***Yes*** and display the property.
- Be sure to place the ***current printer in series*** and place the ***voltage printer in parallel***.
- It is also a good idea to label the node for the output voltage as Vo.



3) Create a New Simulation Profile, Run PSPICE, and view the results in the OUTPUT file.




```
*Libraries:
* Profile Libraries :
* Local Libraries :
* From [PSPICE NETLIST] se
.lib "nomd.lib"

*Analysis directives:
.DC LIN V_V1 6V 6V 1V
.OPTIONS ADVCONV
.PROBE64 V(alias(*)) I(ali
.INC "..\SCHEMATIC1.net"

V_V1        I(V_PRINT1)

6.000E+00   -2.600E-03


V_V1        V(VO,0)

6.000E+00   -1.800E+01
```

Note that the results match the hand analysis:
V_o = -18V and I_o = -2.6 mA