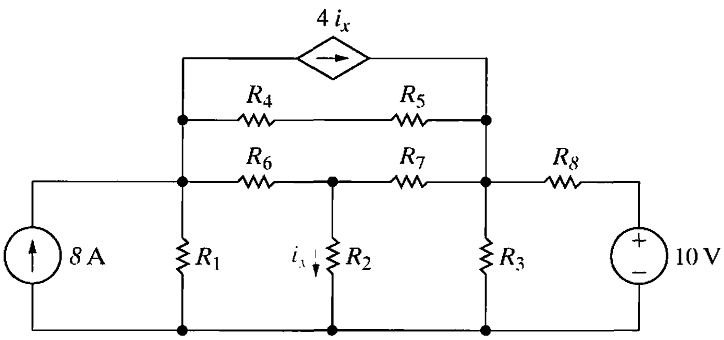
Problems

Section 4.1

4.1 For the circuit shown in Fig. P4.1, state the numerical value of the number of (a) branches, (b) branches where the current is unknown, (c) essential branches, (d) essential branches where the current is unknown, (e) nodes, (f) essential nodes, and (g) meshes.

Figure P4.1



4.2 a) If only the essential nodes and branches are identified in the circuit in Fig. P4.1, how many simultaneous equations are needed to describe the circuit?

b) How many of these equations can be derived using Kirchhoff's current law?

c) How many must be derived using Kirchhoff's voltage law?

d) What two meshes should be avoided in applying the voltage law?

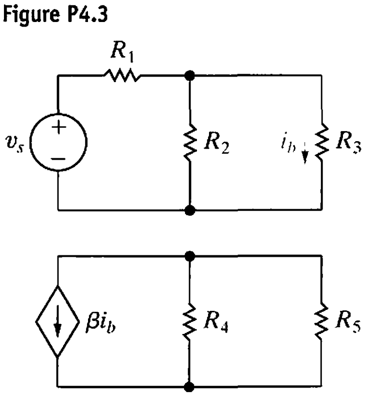
4.3 a) How many separate parts does the circuit in Fig. P4.3 have?

b) How many nodes?

c) How many branches are there?

d) Assume that the lower node in each part of the circuit is joined by a single conductor. Repeat the calculations in (a)–(c).

Figure P4.3



4.4 Assume the current i_g in the circuit in Fig. P4.4 is known. The resistors $R_1 - R_5$ are also known.

a) How many unknown currents are there?

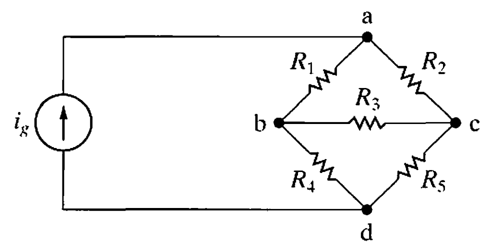
b) How many independent equations can be written using Kirchhoff's current law (KCL)?

c) Write an independent set of KCL equations.

d) How many independent equations can be derived from Kirchhoff's voltage law (KVL)?

e) Write a set of independent KVL equations.

Figure P4.4



4.5 A current leaving a node is defined as positive.

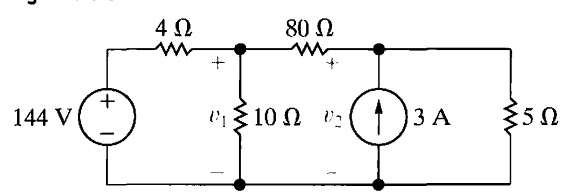
a) Sum the currents at each node in the circuit shown in Fig. P4.4.

b) Show that any one of the equations in (a) can be derived from the remaining three equations.

Section 4.2

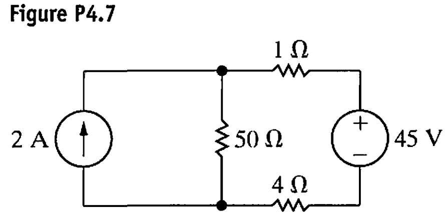
4.6 Use the node-voltage method to find v_1 and v_2 in the circuit in Fig. P4.6.
PSPICE
MULTISIM

Figure P4.6



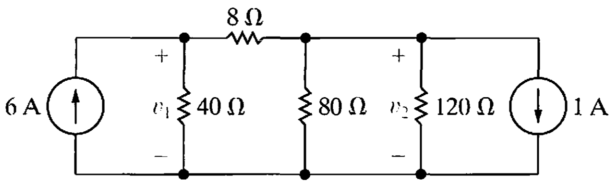
4.7 Use the node-voltage method to find how much power the 2 A source extracts from the circuit in Fig. P4.7.
PSPICE
MULTISIM

Figure P4.7



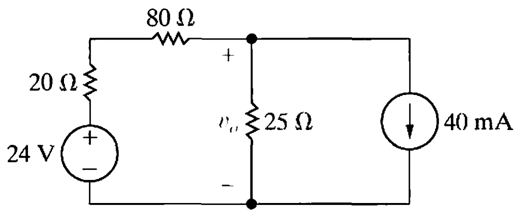
4.8 Use the node-voltage method to find v_1 and v_2 in the circuit shown in Fig. P4.8.
PSPICE MULTISIM

Figure P4.8



4.9 Use the node-voltage method to find v_o in the circuit in Fig. P4.9.
PSPICE MULTISIM

Figure P4.9

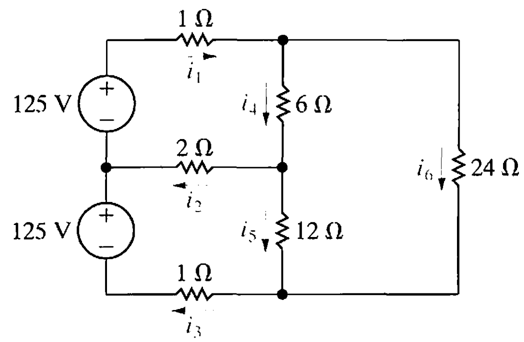


4.10 a) Find the power developed by the 40 mA current source in the circuit in Fig. P4.9.
PSPICE MULTISIM
b) Find the power developed by the 24 V voltage source in the circuit in Fig. P4.9.
c) Verify that the total power developed equals the total power dissipated.

4.11 A 50 Ω resistor is connected in series with the 40 mA current source in the circuit in Fig. P4.9.
PSPICE MULTISIM
a) Find v_o.
b) Find the power developed by the 40 mA current source.
c) Find the power developed by the 24 V voltage source.
d) Verify that the total power developed equals the total power dissipated.
e) What effect will any finite resistance connected in series with the 40 mA current source have on the value of v_o?

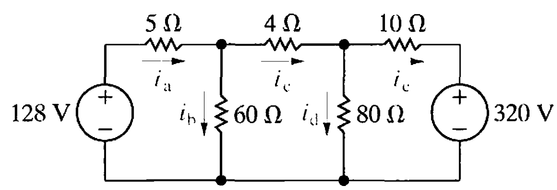
4.12 The circuit shown in Fig. P4.12 is a dc model of a residential power distribution circuit.
PSPICE MULTISIM
a) Use the node-voltage method to find the branch currents $i_1 - i_6$.
b) Test your solution for the branch currents by showing that the total power dissipated equals the total power developed.

Figure P4.12



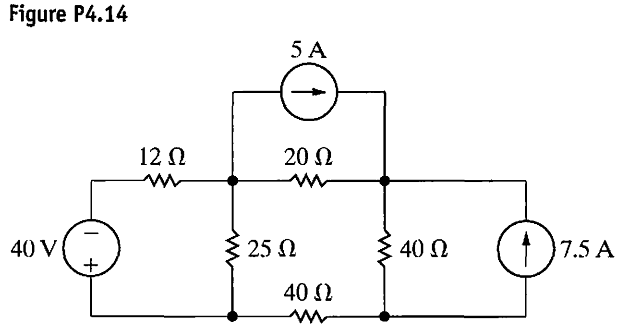
4.13 a) Use the node-voltage method to find the branch currents $i_a - i_e$ in the circuit shown in Fig. P4.13.
PSPICE MULTISIM
b) Find the total power developed in the circuit.

Figure P4.13



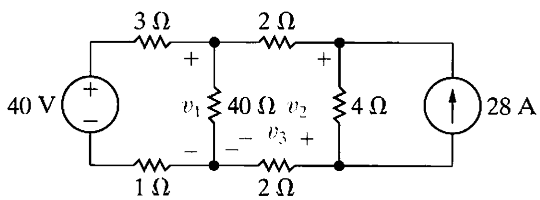
4.14 Use the node-voltage method to find the total power dissipated in the circuit in Fig. P4.14.
PSPICE MULTISIM

Figure P4.14



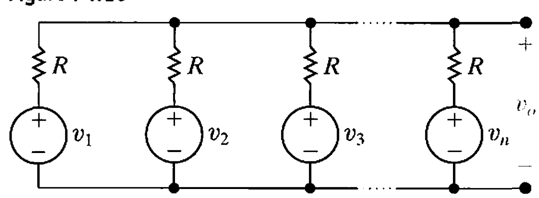
4.15 a) Use the node-voltage method to find v_1, v_2, and v_3 in the circuit in Fig. P4.15.
PSPICE MULTISIM
b) How much power does the 40 V voltage source deliver to the circuit?

Figure P4.15



4.16 a) Use the node-voltage method to show that the output voltage v_o in the circuit in Fig. P4.16 is equal to the average value of the source voltages.
PSPICE MULTISIM
b) Find v_o if $v_1 = 100$ V, $v_2 = 80$ V, and $v_3 = -60$ V.

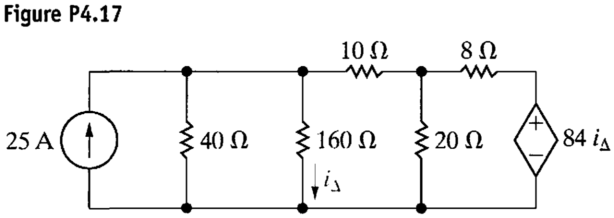
Figure P4.16



Section 4.3

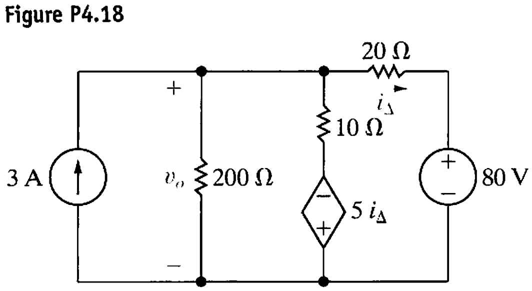
4.17 a) Use the node-voltage method to find the total power developed in the circuit in Fig. P4.17.
PSPICE MULTISIM
b) Check your answer by finding the total power absorbed in the circuit.

Figure P4.17



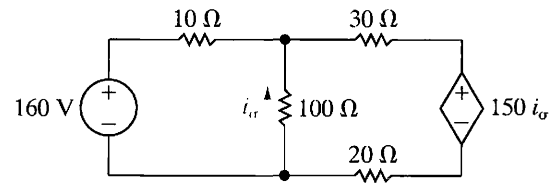
4.18 a) Use the node-voltage method to find v_o in the circuit in Fig. P4.18.
PSPICE MULTISIM
b) Find the power absorbed by the dependent source.
c) Find the total power developed by the independent sources.

Figure P4.18



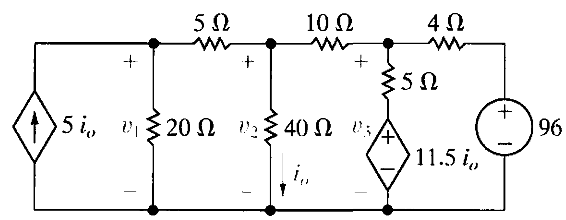
4.19 Use the node-voltage method to calculate the power delivered by the dependent voltage source in the circuit in Fig. P4.19.
PSPICE MULTISIM

Figure P4.19



4.20 a) Find the node voltages v_1, v_2, and v_3 in the circuit in Fig. P4.20.
PSPICE MULTISIM
b) Find the total power dissipated in the circuit.

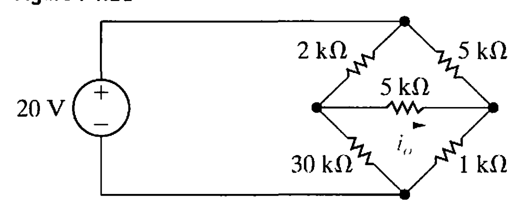
Figure P4.20



Section 4.4

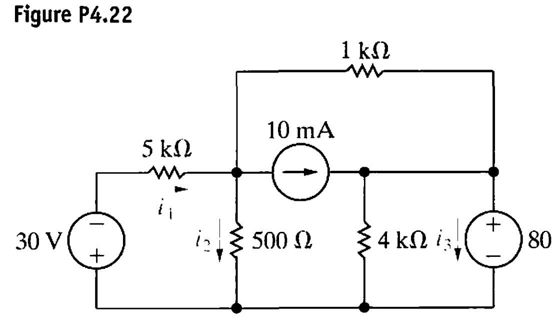
4.21 Use the node-voltage method to find i_o in the circuit in Fig. P4.21.
PSPICE MULTISIM

Figure P4.21



4.22 a) Use the node-voltage method to find the branch currents i_1, i_2, and i_3 in the circuit in Fig. P4.22.
PSPICE MULTISIM
b) Check your solution for i_1, i_2, and i_3 by showing that the power dissipated in the circuit equals the power developed.

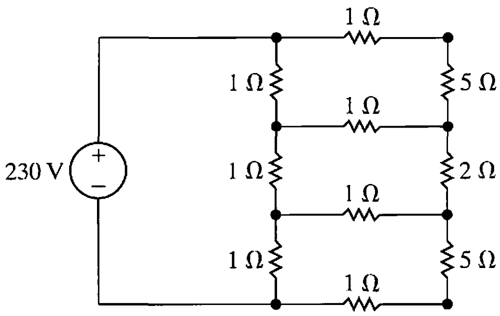
Figure P4.22



4.23 a) Use the node-voltage method to find the power dissipated in the 2 Ω resistor in the circuit in Fig. P4.23.

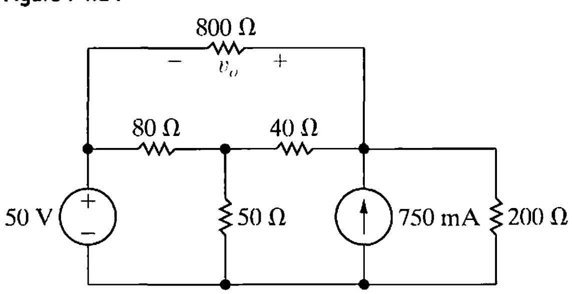
b) Find the power supplied by the 230 V source.

Figure P4.23



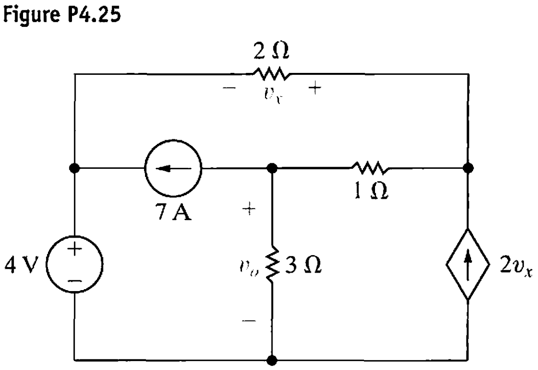
4.24 Use the node-voltage method to find the value of v_o in the circuit in Fig. P4.24.
PSPICE MULTISIM

Figure P4.24



4.25 Use the node-voltage method to find the value of v_o in the circuit in Fig. P4.25.
PSPICE MULTISIM

Figure P4.25

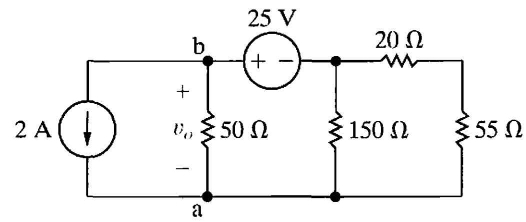


4.26 a) Use the node-voltage method to find v_o and the power delivered by the 2 A current source in the circuit in Fig. P4.26. Use node a as the reference node.
PSPICE MULTISIM

b) Repeat part (a), but use node b as the reference node.

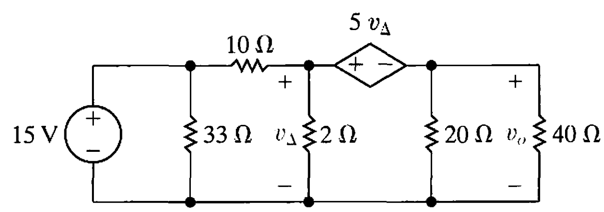
c) Compare the choice of reference node in (a) and (b). Which is better, and why?

Figure P4.26



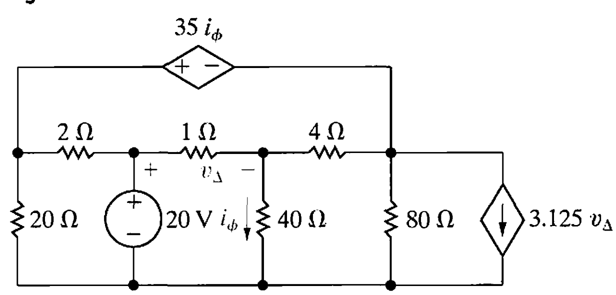
4.27 Use the node-voltage method to find v_o in the circuit in Fig. P4.27.
PSPICE MULTISIM

Figure P4.27



4.28 Use the node-voltage method to find the power developed by the 20 V source in the circuit in Fig. P4.28.
PSPICE MULTISIM

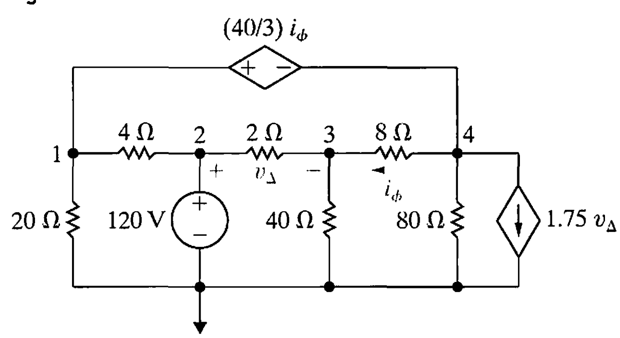
Figure P4.28



4.29 Assume you are a project engineer and one of your staff is assigned to analyze the circuit shown in Fig. P4.29. The reference node and node numbers given on the figure were assigned by the analyst. Her solution gives the values of v_3 and v_4 as 108 V and 81.6 V, respectively.
PSPICE MULTISIM

Test these values by checking the total power developed in the circuit against the total power dissipated. Do you agree with the solution submitted by the analyst?

Figure P4.29



4.30 Show that when Eqs. 4.16, 4.17, and 4.19 are solved for i_B, the result is identical to Eq. 2.25.

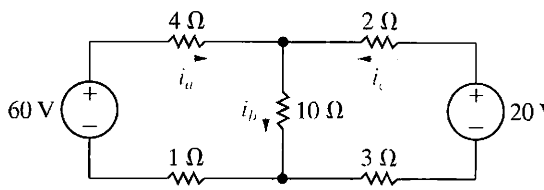
Section 4.5

4.31 Solve Problem 4.12 using the mesh-current method.

4.32 Solve Problem 4.13 using the mesh-current method.

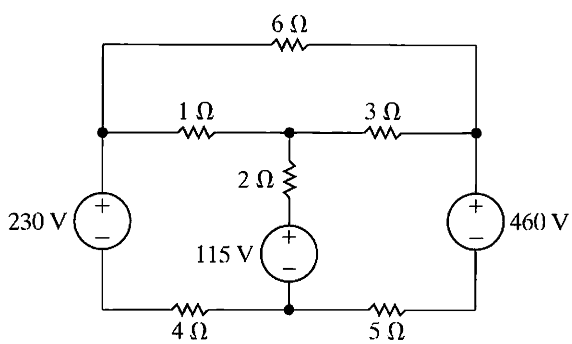
4.33 PSPICE MULTISIM
a) Use the mesh-current method to find the branch currents i_a, i_b, and i_c in the circuit in Fig. P4.33.
b) Repeat (a) if the polarity of the 60 V source is reversed.

Figure P4.33



4.34 PSPICE MULTISIM
a) Use the mesh-current method to find the total power developed in the circuit in Fig. P4.34.
b) Check your answer by showing that the total power developed equals the total power dissipated.

Figure P4.34



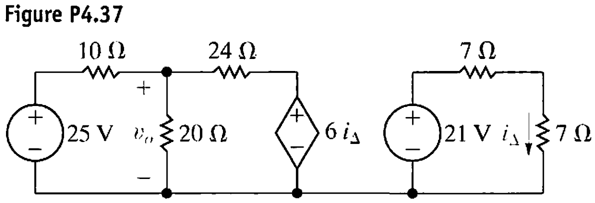
4.35 Solve Problem 4.21 using the mesh-current method.

4.36 Solve Problem 4.23 using the mesh-current method.

Section 4.6

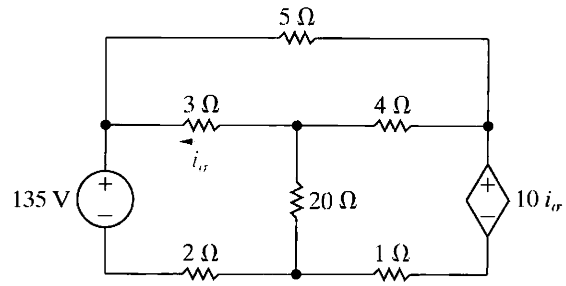
4.37 PSPICE MULTISIM
a) Use the mesh-current method to find v_o in the circuit in Fig. P4.37.
b) Find the power delivered by the dependent source.

Figure P4.37



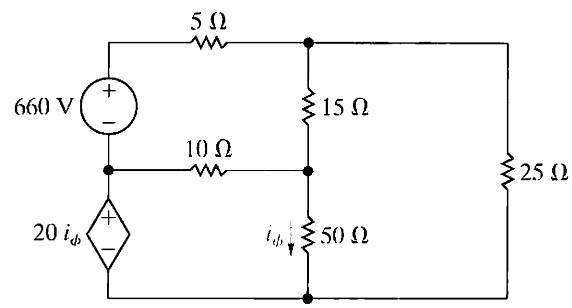
4.38 PSPICE MULTISIM Use the mesh-current method to find the power dissipated in the 20 Ω resistor in the circuit in Fig. P4.38.

Figure P4.38



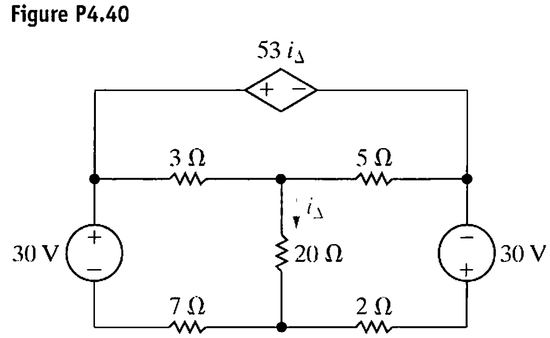
4.39 PSPICE MULTISIM Use the mesh-current method to find the power delivered by the dependent voltage source in the circuit seen in Fig. P4.39.

Figure P4.39



4.40 PSPICE MULTISIM Use the mesh-current method to find the power developed in the dependent voltage source in the circuit in Fig. P4.40.

Figure P4.40

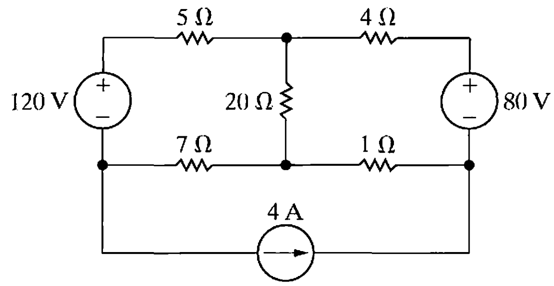


Section 4.7

4.41 Solve Problem 4.8 using the mesh-current method.

4.42
a) Use the mesh-current method to find how much power the 4 A current source delivers to the circuit in Fig. P4.42.
b) Find the total power delivered to the circuit.
c) Check your calculations by showing that the total power developed in the circuit equals the total power dissipated

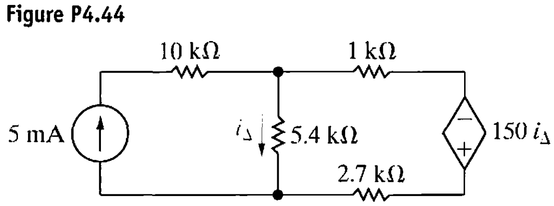
Figure P4.42



4.43 Solve Problem 4.20 using the mesh-current method.

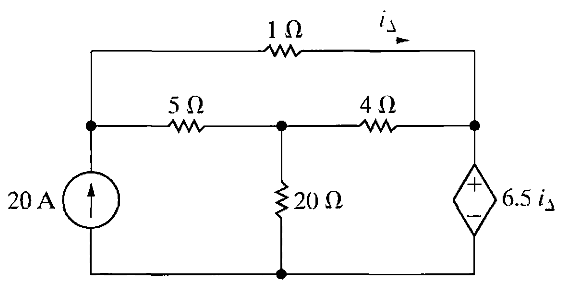
4.44 PSPICE MULTISIM
a) Use the mesh-current method to solve for i_Δ in the circuit in Fig. P4.44.
b) Find the power delivered by the independent current source.
c) Find the power delivered by the dependent voltage source.

Figure P4.44



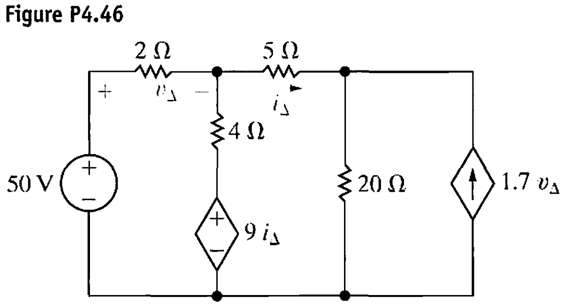
4.45 PSPICE MULTISIM
Use the mesh-current method to find the total power developed in the circuit in Fig. P4.45.

Figure P4.45



4.46 PSPICE MULTISIM
a) Use the mesh-current method to determine which sources in the circuit in Fig. P4.46 are generating power.
b) Find the total power dissipated in the circuit.

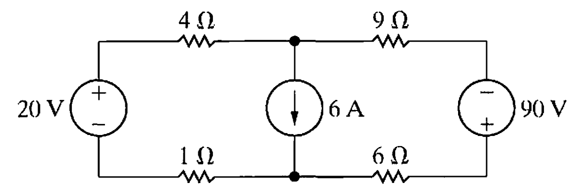
Figure P4.46



4.47 Solve Problem 4.22 using the mesh-current method.

4.48 PSPICE MULTISIM
Use the mesh-current method to find the total power dissipated in the circuit in Fig. P4.48.

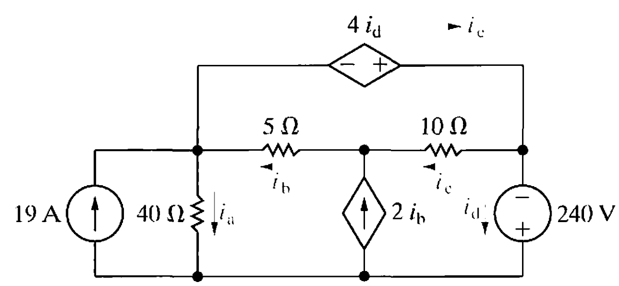
Figure P4.48



4.49
a) Assume the 20 V source in the circuit in Fig. P4.48 is changed to 60 V. Find the total power dissipated in the circuit.
b) Repeat (a) if the 6 A current source is replaced by a short circuit.
c) Explain why the answers to (a) and (b) are the same.

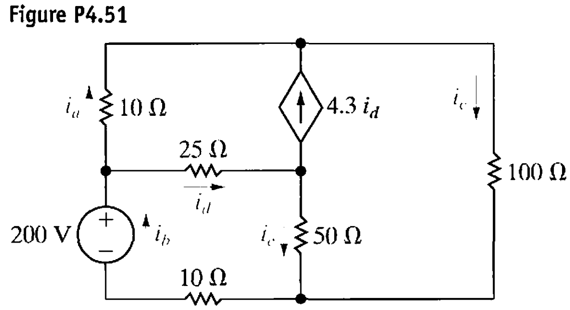
4.50 PSPICE MULTISIM
a) Find the branch currents $i_a - i_e$ for the circuit shown in Fig. P4.50.
b) Check your answers by showing that the total power generated equals the total power dissipated.

Figure P4.50



4.51 PSPICE MULTISIM
a) Use the mesh-current method to find the branch currents in $i_a - i_e$ in the circuit in Fig. P4.51.
b) Check your solution by showing that the total power developed in the circuit equals the total power dissipated.

Figure P4.51

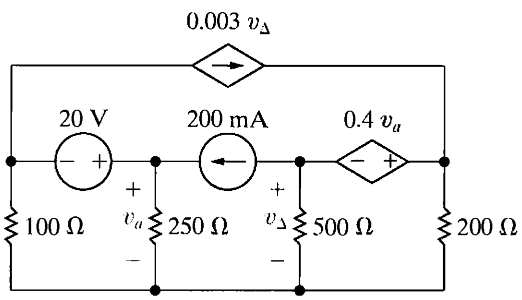


Section 4.8

4.52 PSPICE MULTISIM a) Would you use the node-voltage or mesh-current method to find the power absorbed by the 20 V source in the circuit in Fig. P4.52? Explain your choice.

b) Use the method you selected in (a) to find the power.

Figure P4.52



4.53 PSPICE MULTISIM Assume you have been asked to find the power dissipated in the 1 kΩ resistor in the circuit in Fig. P4.53.

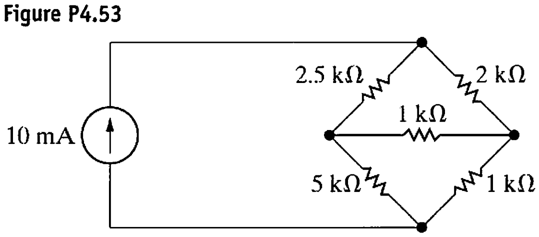
a) Which method of circuit analysis would you recommend? Explain why.

b) Use your recommended method of analysis to find the power dissipated in the 1 kΩ resistor.

c) Would you change your recommendation if the problem had been to find the power developed by the 10 mA current source? Explain.

d) Find the power delivered by the 10 mA current source.

Figure P4.53



4.54 PSPICE MULTISIM A 4 kΩ resistor is placed in parallel with the 10 mA current source in the circuit in Fig. P4.53. Assume you have been asked to calculate the power developed by the current source.

a) Which method of circuit analysis would you recommend? Explain why.

b) Find the power developed by the current source.

4.55 PSPICE MULTISIM The circuit in Fig. P4.55 is a direct-current version of a typical three-wire distribution system. The resistors R_a, R_b, and R_c represent the resistances of the three conductors that connect the three loads R_1, R_2, and R_3 to the 125/250 V voltage supply. The resistors R_1 and R_2 represent loads connected to the 125 V circuits, and R_3 represents a load connected to the 250 V circuit.

a) What circuit analysis method will you use and why?

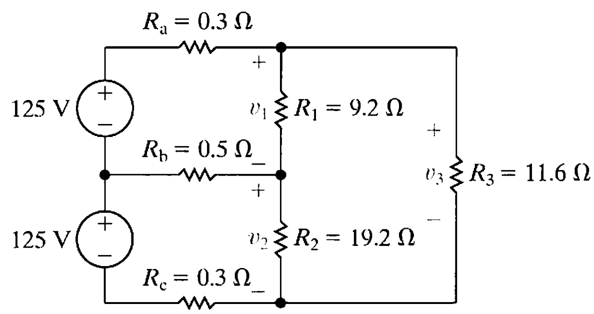
b) Calculate v_1, v_2, and v_3.

c) Calculate the power delivered to R_1, R_2, and R_3.

d) What percentage of the total power developed by the sources is delivered to the loads?

e) The R_b branch represents the neutral conductor in the distribution circuit. What adverse effect occurs if the neutral conductor is opened? (*Hint:* Calculate v_1 and v_2 and note that appliances or loads designed for use in this circuit would have a nominal voltage rating of 125 V.)

Figure P4.55



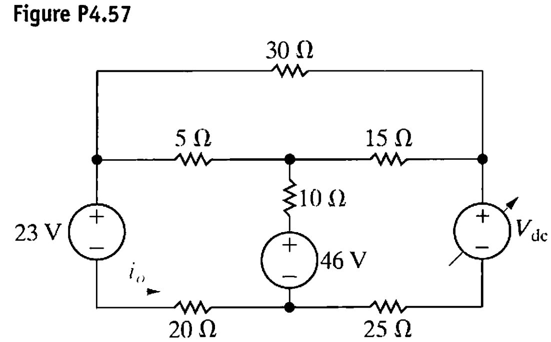
4.56 PSPICE MULTISIM Show that whenever $R_1 = R_2$ in the circuit in Fig. P4.55, the current in the neutral conductor is zero. (*Hint:* Solve for the neutral conductor current as a function of R_1 and R_2.)

4.57 PSPICE MULTISIM The variable dc voltage source in the circuit in Fig. P4.57 is adjusted so that i_o is zero.

a) Find the value of V_{dc}.

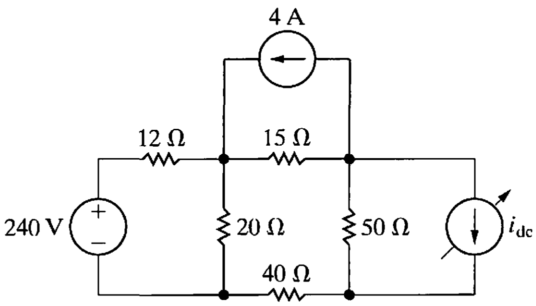
b) Check your solution by showing the power developed equals the power dissipated.

Figure P4.57



4.58 PSPICE MULTISIM The variable dc current source in the circuit in Fig. P4.58 is adjusted so that the power developed by the 4 A current source is zero. Find the value of i_{dc}.

Figure P4.58

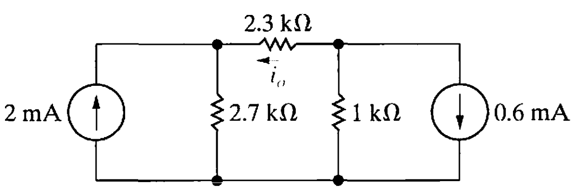


Section 4.9

4.59 PSPICE MULTISIM a) Use a series of source transformations to find the current i_o in the circuit in Fig. P4.59.

b) Verify your solution by using the node-voltage method to find i_o.

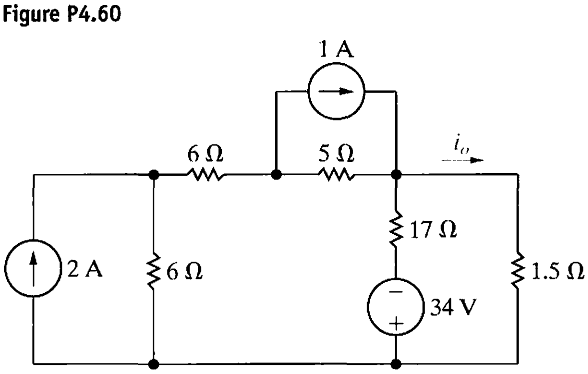
Figure P4.59



4.60 PSPICE MULTISIM a) Use a series of source transformations to find i_o in the circuit in Fig. P4.60.

b) Verify your solution by using the mesh-current method to find i_o.

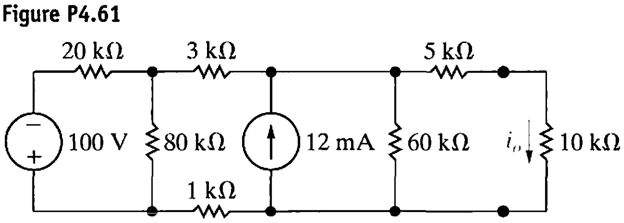
Figure P4.60



4.61 PSPICE MULTISIM a) Find the current in the 10 kΩ resistor in the circuit in Fig. P4.61 by making a succession of appropriate source transformations.

b) Using the result obtained in (a), work back through the circuit to find the power developed by the 100 V source.

Figure P4.61



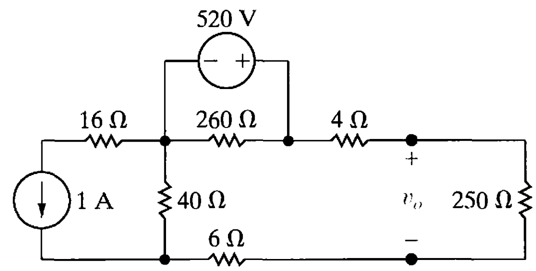
4.62 PSPICE MULTISIM a) Use source transformations to find v_o in the circuit in Fig. P4.62.

b) Find the power developed by the 520 V source.

c) Find the power developed by the 1 A current source.

d) Verify that the total power developed equals the total power dissipated.

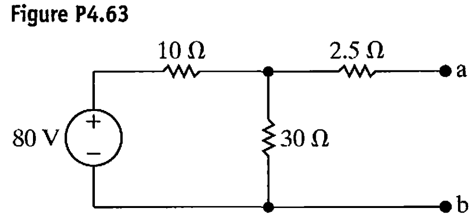
Figure P4.62



Section 4.10

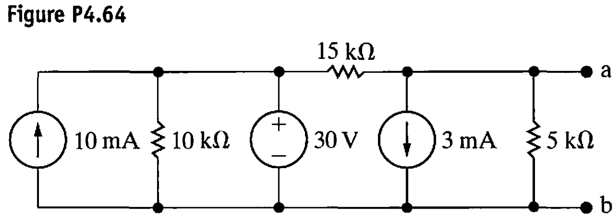
4.63 PSPICE MULTISIM Find the Thévenin equivalent with respect to the terminals a,b for the circuit in Fig. P4.63.

Figure P4.63



4.64 PSPICE MULTISIM Find the Norton equivalent with respect to the terminals a,b in the circuit in Fig. P4.64.

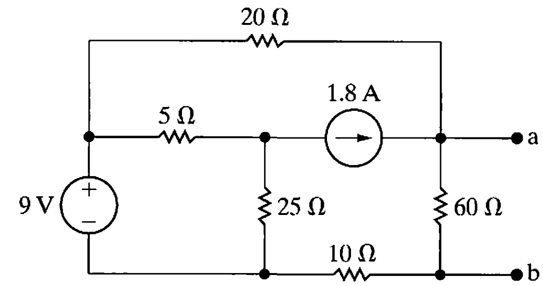
Figure P4.64



4.65 PSPICE MULTISIM a) Find the Thévenin equivalent with respect to the terminals a,b for the circuit in Fig. P4.65 by finding the open-circuit voltage and the short-circuit current.

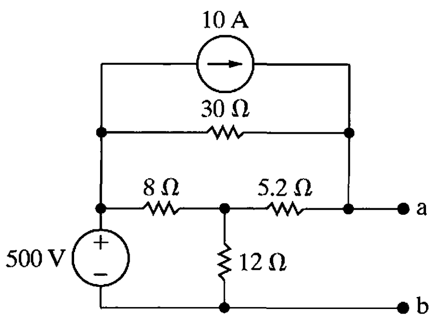
b) Solve for the Thévenin resistance by removing the independent sources. Compare your result to the Thévenin resistance found in (a).

Figure P4.65



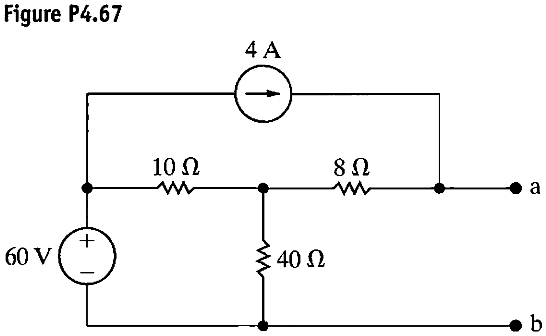
4.66 Find the Thévenin equivalent with respect to the terminals a,b for the circuit in Fig. P4.66.
PSPICE MULTISIM

Figure P4.66



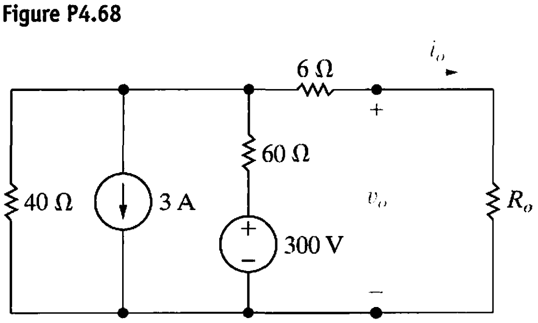
4.67 Find the Norton equivalent with respect to the terminals a,b for the circuit in Fig. P4.67.
PSPICE MULTISIM

Figure P4.67



4.68 Determine i_o and v_o in the circuit shown in Fig. P4.68 when R_o is a resistor from Appendix H whose value is less than 100 Ω.
PSPICE MULTISIM

Figure P4.68



4.69 An automobile battery, when connected to a car radio, provides 12.5 V to the radio. When connected to a set of headlights, it provides 11.7 V to the headlights. Assume the radio can be modeled as a 6.25 Ω resistor and the headlights can be modeled as a 0.65 Ω resistor. What are the Thévenin and Norton equivalents for the battery?

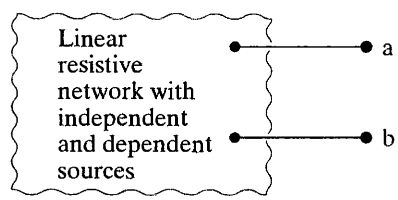
4.70 A Thévenin equivalent can also be determined from measurements made at the pair of terminals of interest. Assume the following measurements were made at the terminals a,b in the circuit in Fig. P4.70.

When a 20 Ω resistor is connected to the terminals a,b, the voltage v_{ab} is measured and found to be 100 V.

When a 50 Ω resistor is connected to the terminals a,b, the voltage is measured and found to be 200 V.

Find the Thévenin equivalent of the network with respect to the terminals a,b.

Figure P4.70

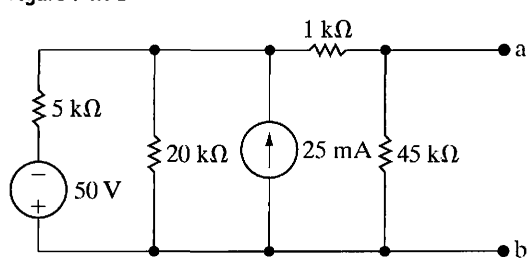


4.71 A voltmeter with a resistance of 85.5 kΩ is used to measure the voltage v_{ab} in the circuit in Fig. P4.71.
PSPICE MULTISIM

a) What is the voltmeter reading?

b) What is the percentage of error in the voltmeter reading if the percentage of error is defined as [(measured − actual)/actual] × 100?

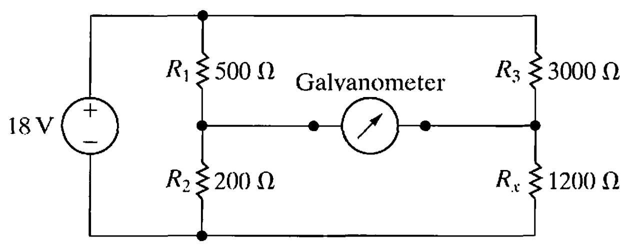
Figure P4.71



4.72 The Wheatstone bridge in the circuit shown in Fig. P4.72 is balanced when R_3 equals 3000 Ω. If the galvanometer has a resistance of 50 Ω, how much current will the galvanometer detect, when the bridge is unbalanced by setting R_3 to 3003 Ω? (*Hint:* Find the Thévenin equivalent with respect to the galvanometer terminals when R_3 = 3003 Ω. Note that once we have found this Thévenin equivalent, it is easy to find the amount of unbalanced
PSPICE MULTISIM

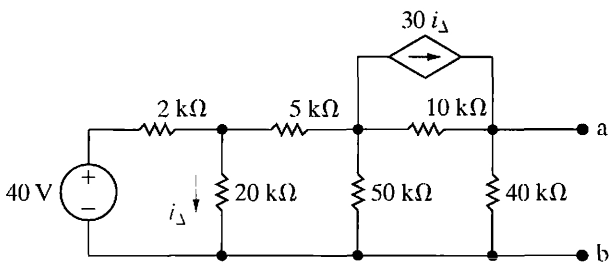
current in the galvanometer branch for different galvanometer movements.)

Figure P4.72



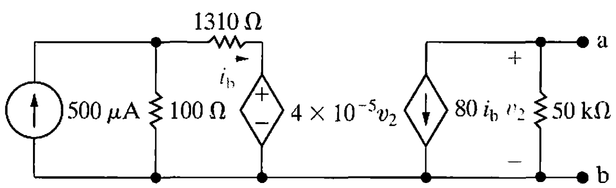
4.73 Find the Norton equivalent with respect to the terminals a,b for the circuit seen in Fig. P4.73.
PSPICE MULTISIM

Figure P4.73



4.74 Determine the Thévenin equivalent with respect to the terminals a,b for the circuit shown in Fig. P4.74.
PSPICE MULTISIM

Figure P4.74

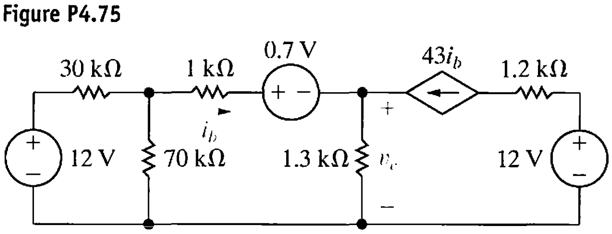


4.75 When a voltmeter is used to measure the voltage v_e in Fig. P4.75, it reads 5.5 V.
PSPICE MULTISIM

a) What is the resistance of the voltmeter?

b) What is the percentage of error in the voltage measurement?

Figure P4.75

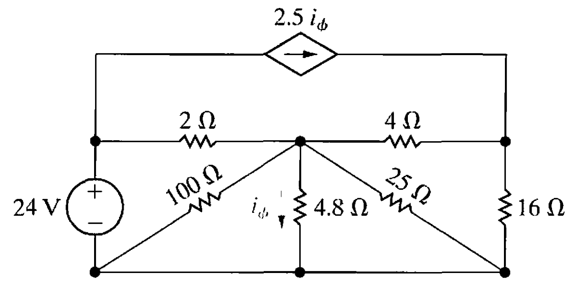


4.76 When an ammeter is used to measure the current i_ϕ in the circuit shown in Fig. P4.76, it reads 6 A.
PSPICE MULTISIM

a) What is the resistance of the ammeter?

b) What is the percentage of error in the current measurement?

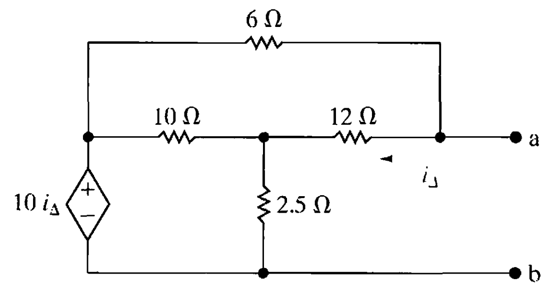
Figure P4.76



Section 4.11

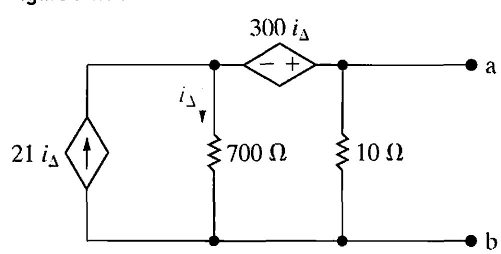
4.77 Find the Thévenin equivalent with respect to the terminals a,b in the circuit in Fig. P4.77.

Figure P4.77



4.78 Find the Norton equivalent with respect to the terminals a,b for the circuit seen in Fig. P4.78.

Figure P4.78



Section 4.12

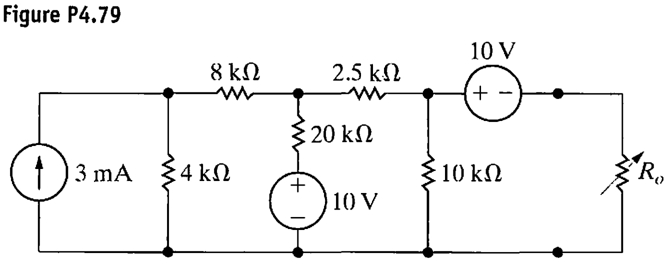
4.79 The variable resistor in the circuit in Fig. P4.79 is adjusted for maximum power transfer to R_o.
PSPICE MULTISIM

a) Find the value of R_o.

b) Find the maximum power that can be delivered to R_o.

c) Find a resistor in Appendix H closest to the value in part (a). How much power is delivered to this resistor?

Figure P4.79

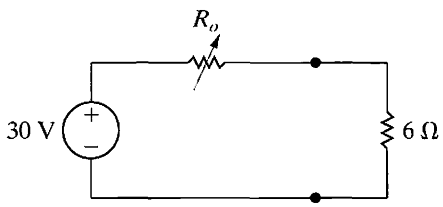


4.80 PSPICE MULTISIM What percentage of the total power developed in the circuit in Fig. P4.79 is delivered to R_o when R_o is set for maximum power transfer?

4.81 a) Find the value of the variable resistor R_o in the circuit in Fig. P4.81 that will result in maximum power dissipation in the 6 Ω resistor. (Hint: Hasty conclusions could be hazardous to your career.)

b) What is the maximum power that can be delivered to the 6 Ω resistor?

Figure P4.81



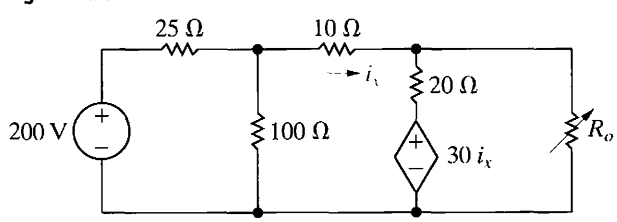
4.82 a) Calculate the power delivered for each value of R_o used in Problem 4.68.

b) Plot the power delivered to R_o versus the resistance R_o.

c) At what value of R_o is the power delivered to R_o a maximum?

4.83 PSPICE MULTISIM The variable resistor (R_o) in the circuit in Fig. P4.83 is adjusted until the power dissipated in the resistor is 250 W. Find the values of R_o that satisfy this condition.

Figure P4.83



4.84 PSPICE MULTISIM A variable resistor R_o is connected across the terminals a,b in the circuit in Fig. P4.73. The variable resistor is adjusted until maximum power is transferred to R_o.

a) Find the value of R_o.

b) Find the maximum power delivered to R_o.

c) Find the percentage of the total power developed in the circuit that is delivered to R_o.

d) Find the resistor from Appendix H closest in value to the R_o from part (a).

e) Find the percentage of the total power developed in the circuit that is delivered to the resistor in part (d).

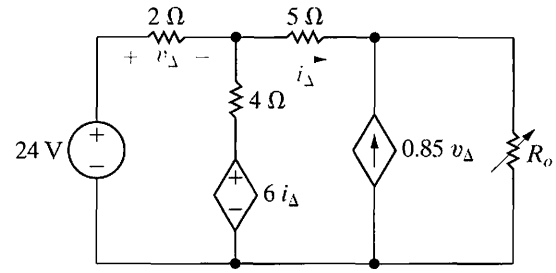
4.85 PSPICE MULTISIM The variable resistor (R_o) in the circuit in Fig. P4.85 is adjusted until it absorbs maximum power from the circuit.

a) Find the value of R_o.

b) Find the maximum power.

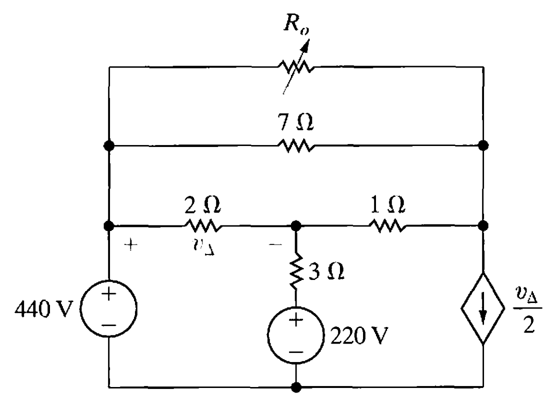
c) Find the percentage of the total power developed in the circuit that is delivered to R_o.

Figure P4.85



4.86 PSPICE MULTISIM The variable resistor (R_o) in the circuit in Fig. P4.86 is adjusted for maximum power transfer to R_o. What percentage of the total power developed in the circuit is delivered to R_o?

Figure P4.86

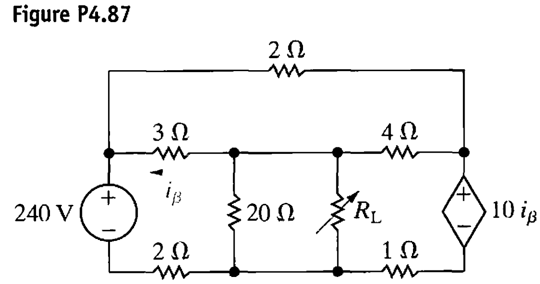


4.87 PSPICE MULTISIM The variable resistor (R_L) in the circuit in Fig. P4.87 is adjusted for maximum power transfer to R_L.

a) Find the numerical value of R_L.

b) Find the maximum power transferred to R_L.

Figure P4.87



4.88 The variable resistor in the circuit in Fig. P4.88 is adjusted for maximum power transfer to R_o.

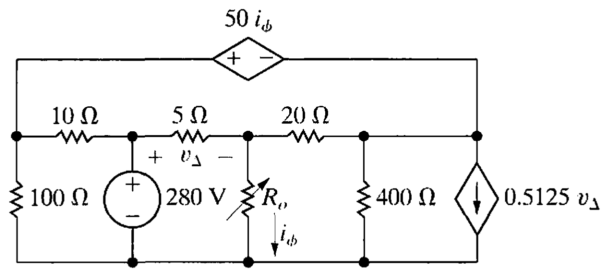
PSPICE MULTISIM

a) Find the numerical value of R_o.

b) Find the maximum power delivered to R_o.

c) How much power does the 280 V source deliver to the circuit when R_o is adjusted to the value found in (a)?

Figure P4.88



4.89 The variable resistor (R_o) in the circuit in Fig. P4.89 is adjusted for maximum power transfer to R_o.

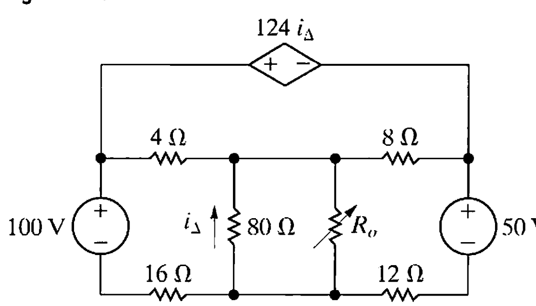
PSPICE MULTISIM

a) Find the value of R_o.

b) Find the maximum power that can be delivered to R_o.

c) If R_o is selected from Appendix H, which resistor value will result in the greatest amount of power delivered to R_o?

Figure P4.89



4.90 What percentage of the total power developed in the circuit in Fig. P4.89 is delivered to R_o found in Problem 4.89(a)?

PSPICE MULTISIM

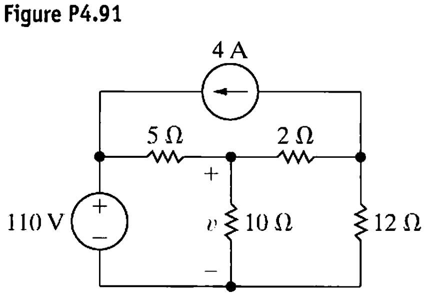
Section 4.13

4.91 a) Use the principle of superposition to find the voltage v in the circuit of Fig. P4.91.

PSPICE MULTISIM

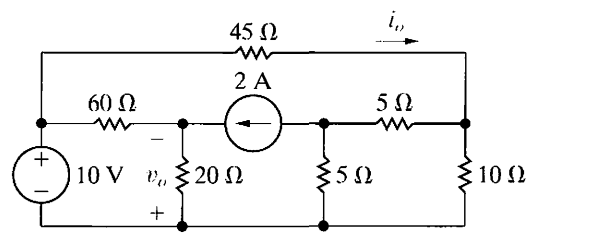
b) Find the power dissipated in the 10 Ω resistor.

Figure P4.91



4.92 Use superposition to solve for i_o and v_o in the circuit in Fig. P4.92.

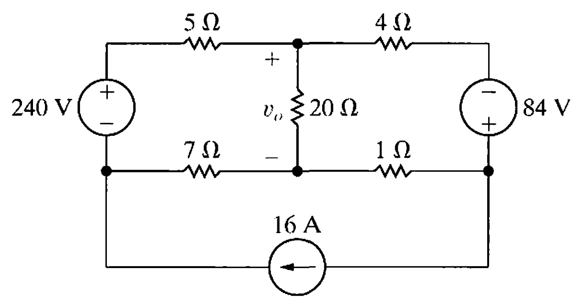
Figure P4.92



4.93 Use the principle of superposition to find the voltage v_o in the circuit in Fig. P4.93.

PSPICE MULTISIM

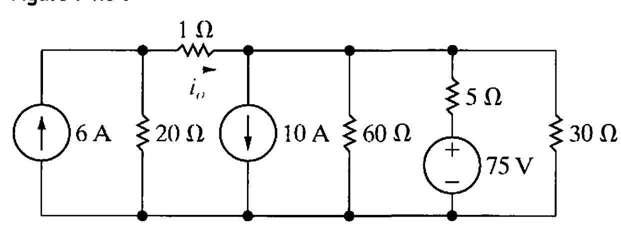
Figure P4.93



4.94 Use the principle of superposition to find the current i_o in the circuit shown in Fig. P4.94.

PSPICE MULTISIM

Figure P4.94

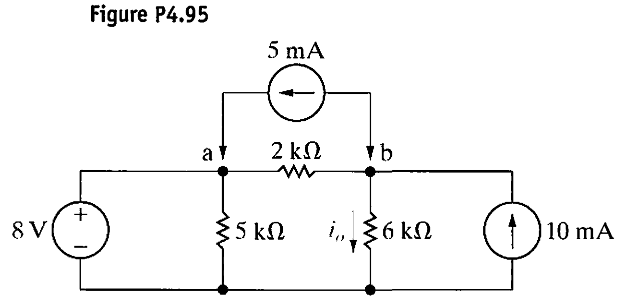


4.95 a) In the circuit in Fig. P4.95, before the 5 mA current source is attached to the terminals a,b, the current i_o is calculated and found to be 3.5 mA. Use superposition to find the value of i_o after the current source is attached.

PSPICE MULTISIM

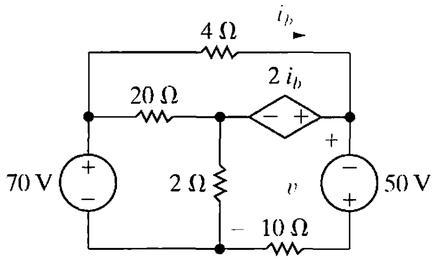
b) Verify your solution by finding i_o when all three sources are acting simultaneously.

Figure P4.95



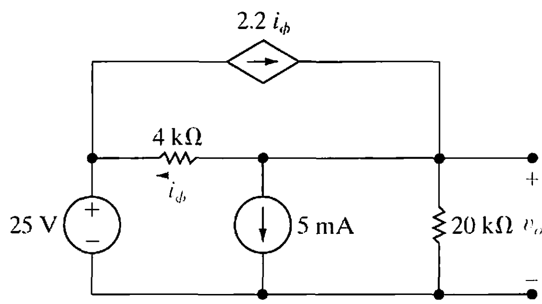
4.96 Use the principle of superposition to find the voltage v in the circuit of Fig. P4.96.
PSPICE MULTISIM

Figure P4.96



4.97 Use the principle of superposition to find v_o in the circuit in Fig. P4.97.
PSPICE MULTISIM

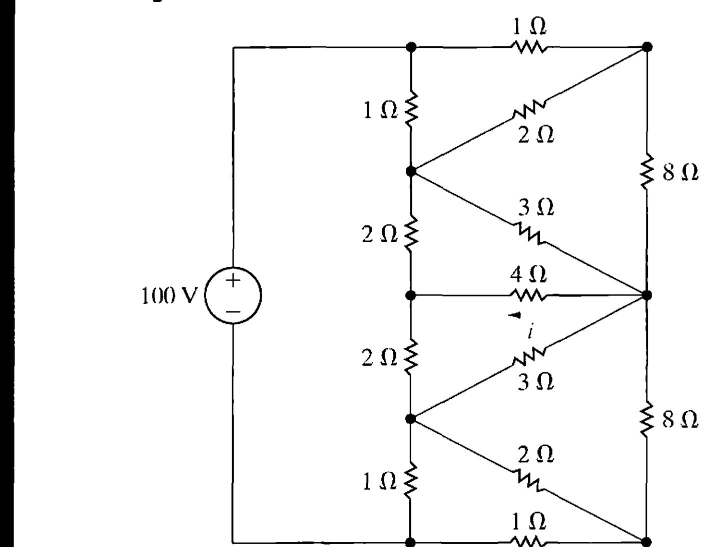
Figure P4.97



Sections 4.1–4.13

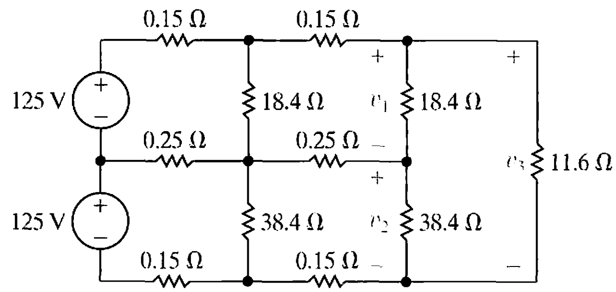
4.98 Find i in the circuit in Fig. P4.98.
PSPICE MULTISIM

Figure P4.98



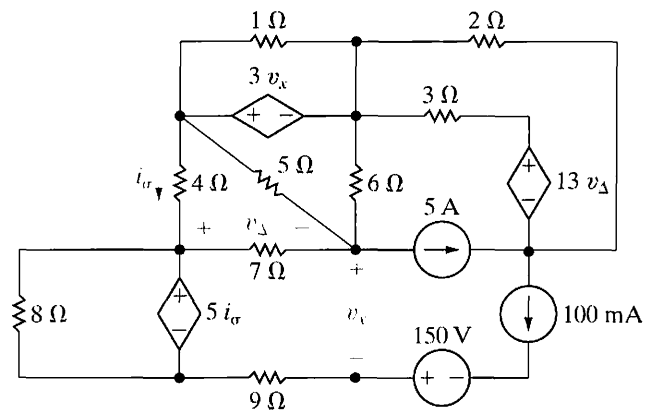
4.99 Find v_1, v_2, and v_3 in the circuit in Fig. P4.99.
PSPICE MULTISIM

Figure P4.99



4.100 Find the power absorbed by the 5 A current source in the circuit in Fig. P4.100.
PSPICE MULTISIM

Figure P4.100

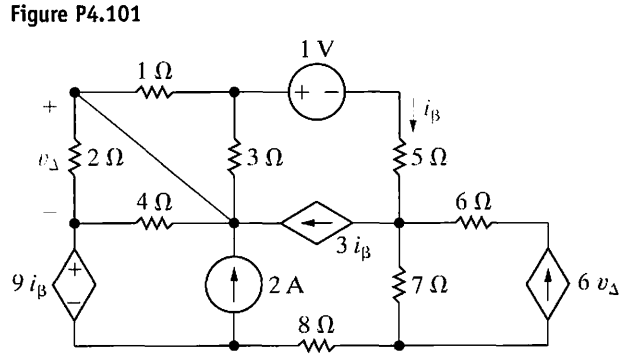


4.101 Assume your supervisor has asked you to determine the power developed by the 1 V source in the circuit in Fig. P4.101. Before calculating the power developed by the 1 V source, the supervisor asks you to submit a proposal describing how you plan to attack the problem. Furthermore, he asks you to explain why you have chosen your proposed method of solution.

a) Describe your plan of attack, explaining your reasoning.

b) Use the method you have outlined in (a) to find the power developed by the 1 V source.

Figure P4.101

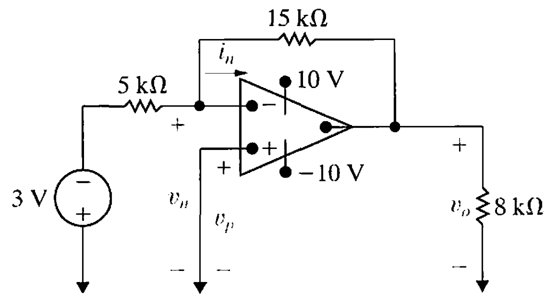


Problems

Sections 5.1–5.2

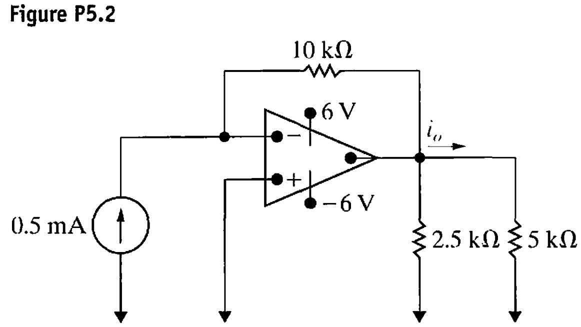
5.1 The op amp in the circuit in Fig. P5.1 is ideal.
PSPICE MULTISIM
a) Label the five op amp terminals with their names.
b) What ideal op amp constraint determines the value of i_n? What is this value?
c) What ideal op amp constraint determines the value of $(v_p - v_n)$? What is this value?
d) Calculate v_o.

Figure P5.1



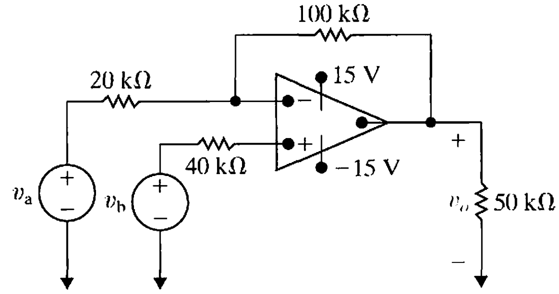
5.2 Find i_o in the circuit in Fig. P5.2 if the op amp is ideal.
PSPICE MULTISIM

Figure P5.2



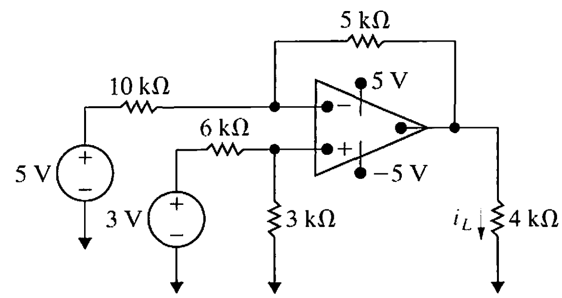
5.3 The op amp in the circuit in Fig. P5.3 is ideal.
PSPICE MULTISIM
a) Calculate v_o if $v_a = 4$ V and $v_b = 0$ V.
b) Calculate v_o if $v_a = 2$ V and $v_b = 0$ V.
c) Calculate v_o if $v_a = 2$ V and $v_b = 1$ V.
d) Calculate v_o if $v_a = 1$ V and $v_b = 2$ V.
e) Calculate v_o if $v_a = 1.5$ V and $v_b = 4$ V.
f) If $v_b = 1.6$ V, specify the range of v_a such that the amplifier does not saturate.

Figure P5.3



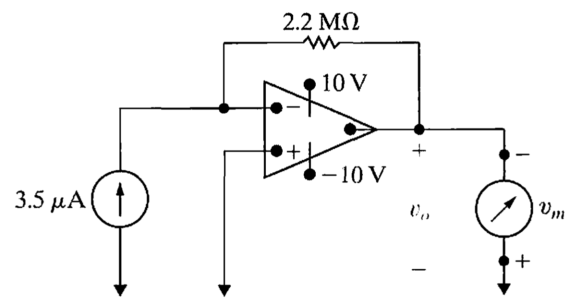
5.4 Find i_L (in microamperes) in the circuit in Fig. P5.4.
PSPICE MULTISIM

Figure P5.4



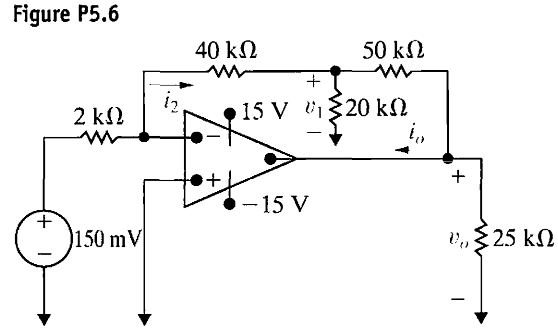
5.5 A voltmeter with a full-scale reading of 10 V is used to measure the output voltage in the circuit in Fig. P5.5. What is the reading of the voltmeter? Assume the op amp is ideal.
PSPICE MULTISIM

Figure P5.5



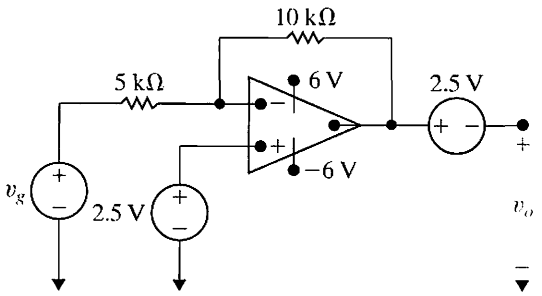
5.6 The op amp in the circuit in Fig. P5.6 is ideal. Calculate the following:
PSPICE MULTISIM
a) v_1
b) v_o
c) i_2
d) i_o

Figure P5.6



5.7 A circuit designer claims the circuit in Fig. P5.7 will produce an output voltage that will vary between ±5 as v_g varies between 0 and 5 V. Assume the op amp is ideal.
PRACTICAL PERSPECTIVE PSPICE MULTISIM
a) Draw a graph of the output voltage v_o as a function of the input voltage v_g for $0 \leq v_g \leq 5$ V.
b) Do you agree with the designer's claim?

Figure P5.7



Section 5.3

5.8 DESIGN PROBLEM a) Design an inverting amplifier using an ideal op amp that has a gain of 3. Use a set of identical resistors from Appendix H.

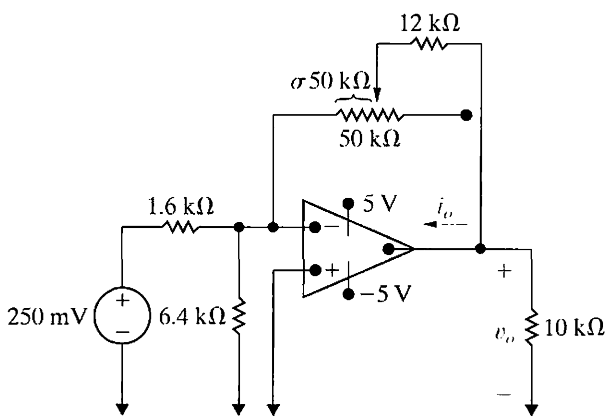
b) If you wish to amplify a 5 V input signal using the circuit you designed in part (a), what are the smallest power supply signals you can use?

5.9 PSPICE MULTISIM The op amp in the circuit in Fig. P5.9 is ideal.

a) Find the range of values for σ in which the op amp does not saturate.

b) Find i_o (in microamperes) when $\sigma = 0.272$.

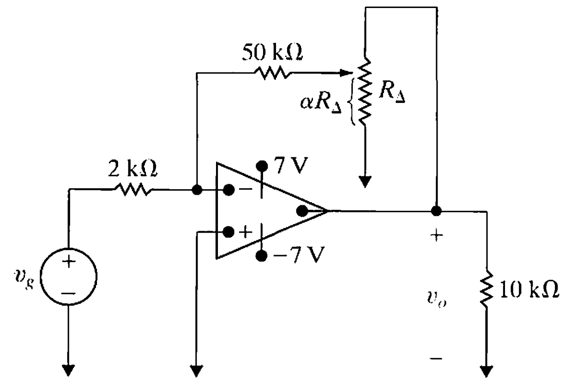
Figure P5.9



5.10 PSPICE MULTISIM a) The op amp in the circuit shown in Fig. P5.10 is ideal. The adjustable resistor R_Δ has a maximum value of 100 kΩ, and α is restricted to the range of $0.2 \le \alpha \le 1$. Calculate the range of v_o if $v_g = 40$ mV.

b) If α is not restricted, at what value of α will the op amp saturate?

Figure P5.10



Section 5.4

5.11 PSPICE MULTISIM Refer to the circuit in Fig. 5.12, where the op amp is assumed to be ideal. Given that $R_a = 4$ kΩ, $R_b = 5$ kΩ, $R_c = 20$ kΩ, $v_a = 200$ mV, $v_b = 150$ mV, $v_c = 400$ mV, and $V_{CC} = \pm 6$ V, specify the range of R_f for which the op amp operates within its linear region.

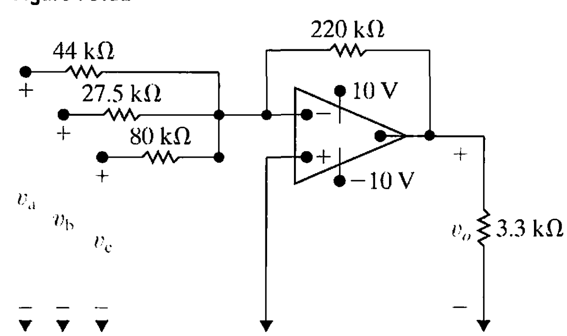
5.12 PSPICE MULTISIM The op amp in Fig. P5.12 is ideal.

a) What circuit configuration is shown in this figure?

b) Find v_o if $v_a = 1$ V, $v_b = 1.5$ V, and $v_c = -4$ V.

c) The voltages v_a and v_c remain at 1 V and −4 V, respectively. What are the limits on v_b if the op amp operates within its linear region?

Figure P5.12



5.13 DESIGN PROBLEM PSPICE MULTISIM Design an inverting-summing amplifier so that

$$v_o = -(3v_a + 5v_b + 4v_c + 2v_d).$$

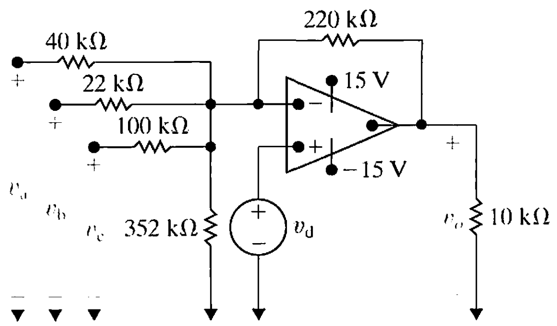
Start by choosing a feedback resistor (R_f) from Appendix H. Then choose single resistors from Appendix H or construct resistor neworks from resistors in Appendix H to satisfy the design values for R_a, R_b, R_c, and R_d. Draw your final circuit diagram.

5.14 a) The op amp in Fig. P5.14 is ideal. Find v_o if $v_a = 4$ V, $v_b = 9$ V, $v_c = 13$ V, and $v_d = 8$ V.

PSPICE MULTISIM

b) Assume v_a v_c, and v_d retain their values as given in (a). Specify the range of v_a such that the op amp operates within its linear region.

Figure P5.14



5.15 The 220 kΩ feedback resistor in the circuit in Fig. P5.14 is replaced by a variable resistor R_f. The voltages $v_a - v_d$ have the same values as given in Problem 5.14(a).

PSPICE MULTISIM

a) What value of R_f will cause the op amp to saturate? Note that $0 \leq R_f \leq \infty$.

b) When R_f has the value found in (a), what is the current (in microamperes) into the output terminal of the op amp?

Section 5.5

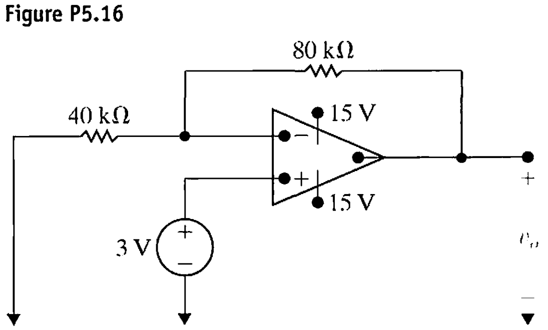
5.16 The op amp in the circuit of Fig. P5.16 is ideal.

PSPICE MULTISIM

a) What op amp circuit configuration is this?

b) Calculate v_o.

Figure P5.16



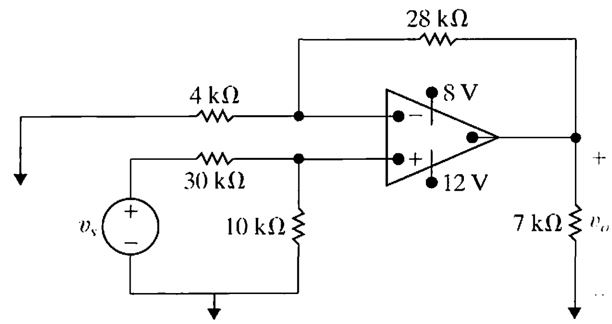
5.17 The op amp in the circuit of Fig. P5.17 is ideal.

a) What op amp circuit configuration is this?

b) Find v_o in terms of v_s.

c) Find the range of values for v_s such that v_o does not saturate and the op amp remains in its linear region of operation.

Figure P5.17



5.18 The op amp in the circuit shown in Fig. P5.18 is ideal.

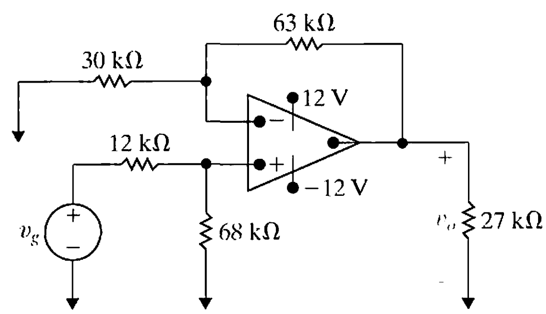
PSPICE MULTISIM

a) Calculate v_o when v_g equals 4 V.

b) Specify the range of values of v_g so that the op amp operates in a linear mode.

c) Assume that v_g equals 2 V and that the 63 kΩ resistor is replaced with a variable resistor. What value of the variable resistor will cause the op amp to saturate?

Figure P5.18



5.19 a) Design a non-inverting amplifier with a gain of 4. Use resistors from Appendix H. You might need to combine resistors in series and in parallel to get the desired resistance. Draw your final circuit.

b) If you use ±12 V power supplies for the op amp, what range of input values will allow the op amp to stay in its linear operating region?

5.20 The op amp in the circuit of Fig. P5.20 is ideal.

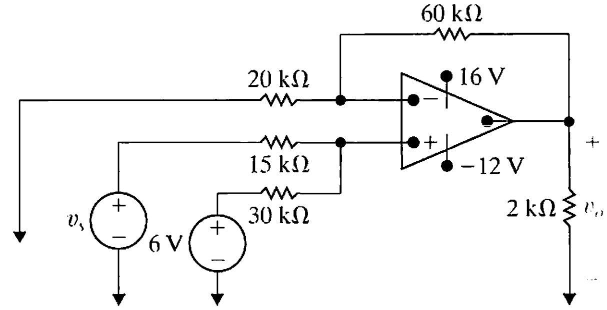
PSPICE MULTISIM

a) What op amp circuit configuration is this?

b) Find v_o in terms of v_s.

c) Find the range of values for v_s such that v_o does not saturate and the op amp remains in its linear region of operation.

Figure P5.20



5.21 PSPICE MULTISIM The op amp in the circuit shown in Fig. P5.21 is ideal. The signal voltages v_a and v_b are 800 mV and 400 mV, respectively.

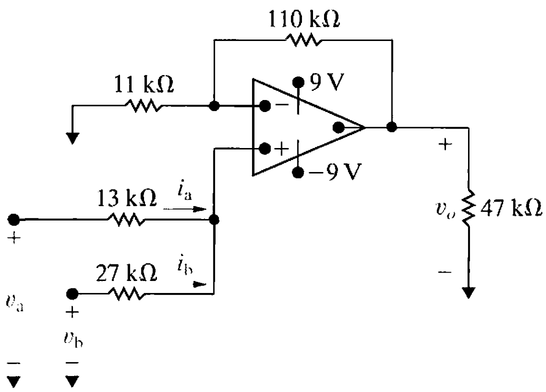
a) What circuit configuration is shown in the figure?

b) Calculate v_o in volts.

c) Find i_a and i_b in microamperes.

d) What are the weighting factors associated with v_a and v_b?

Figure P5.21



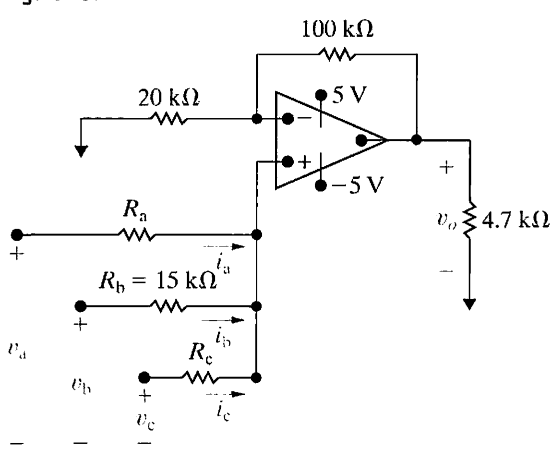
5.22 DESIGN PROBLEM PSPICE MULTISIM The circuit in Fig. P5.22 is a noninverting summing amplifier. Assume the op amp is ideal. Design the circuit so that

$$v_o = v_a + 2v_b + 3v_c.$$

a) Specify the numerical values of R_a and R_c.

b) Calculate i_a, i_b, and i_c (in microamperes) when $v_a = 0.7$ V, $v_b = 0.4$ V, and $v_c = 1.1$ V.

Figure P5.22



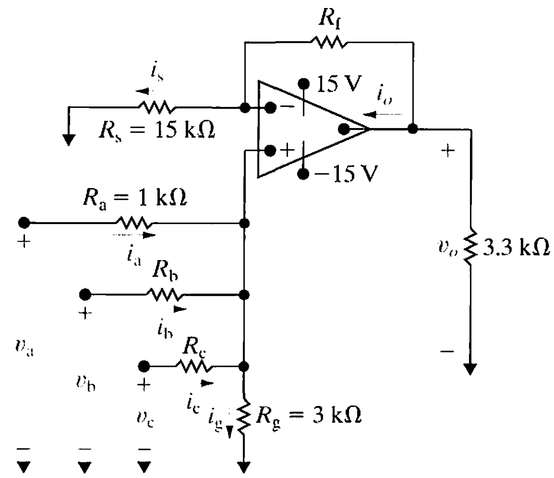
5.23 PSPICE MULTISIM The op amp in the noninverting summing amplifier of Fig. P5.23 is ideal.

a) Specify the values of R_f, R_b, and R_c so that

$$v_o = 6v_a + 3v_b + 4v_c.$$

b) Using the values found in part (a) for R_f, R_b, and R_c, find (in microamperes) i_a, i_b, i_c, i_g, and i_s when $v_a = 0.5$ V, $v_b = 2.5$ V, and $v_c = 1$ V.

Figure P5.23



Section 5.6

5.24 a) Use the principle of superposition to derive Eq. 5.22.

b) Derive Eqs. 5.23 and 5.24.

5.25 PSPICE MULTISIM The resistors in the difference amplifier shown in Fig. 5.15 are $R_a = 24$ kΩ, $R_b = 75$ kΩ, $R_c = 130$ kΩ and $R_d = 120$ kΩ. The signal voltages v_a and v_b are 8 and 5 V, respectively, and $V_{CC} = \pm 20$ V.

a) Find v_o.

b) What is the resistance seen by the signal source v_a?

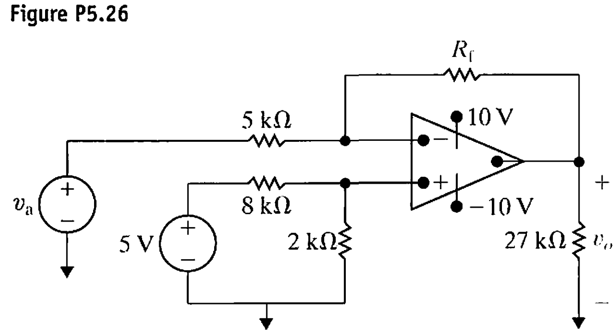
c) What is the resistance seen by the signal source v_b?

5.26 The op amp in the circuit of Fig. P5.26 is ideal. What value of R_f will give the equation

$$v_o = 5 - 4v_a,$$

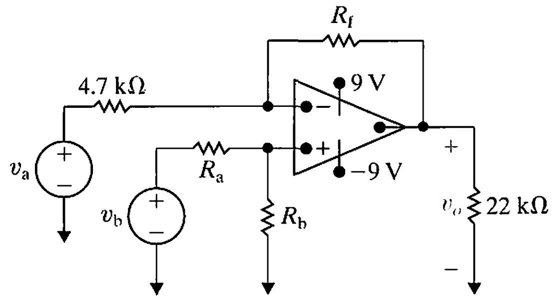
for this circuit?

Figure P5.26



5.27 DESIGN PROBLEM PSPICE MULTISIM Design the difference-amplifier circuit in Fig. P5.27 so that $v_o = 10(v_b - v_a)$, and the voltage source v_b sees an input resistance of 220 kΩ. Specify the values of R_a, R_b, and R_f using single resistors or combinations of resistors from Appendix H. Use the ideal model for the op amp.

Figure P5.27

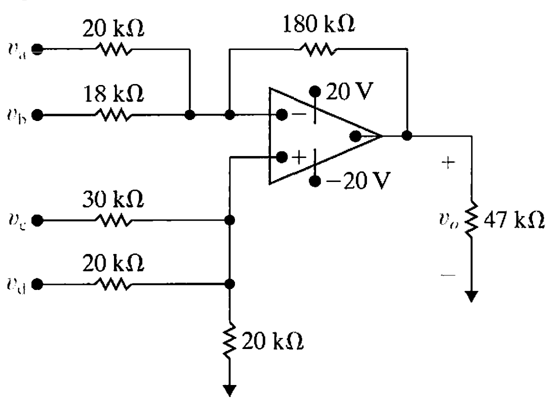


5.28 PSPICE MULTISIM The op amp in the adder-subtracter circuit shown in Fig. P5.28 is ideal.

a) Find v_o when $v_a = 1$ V, $v_b = 2$ V, $v_c = 3$ V, and $v_d = 4$ V.

b) If v_a, v_b, and v_d are held constant, what values of v_c will not saturate the op amp?

Figure P5.28

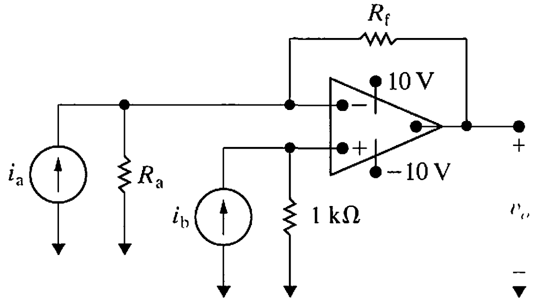


5.29 DESIGN PROBLEM PSPICE MULTISIM Select the values of R_a and R_f in the circuit in Fig. P5.29 so that

$$v_o = 5000(i_b - i_a).$$

Use single resistors or combinations of resistors from Appendix H. The op amp is ideal.

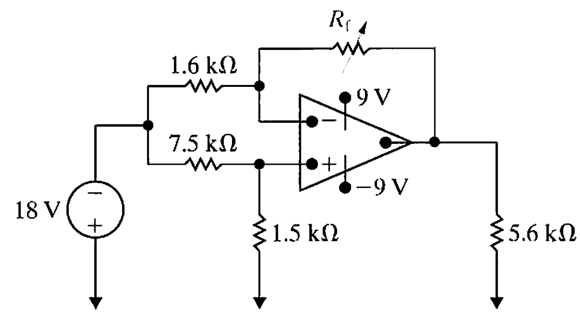
Figure P5.29



5.30 DESIGN PROBLEM PSPICE MULTISIM Design a difference amplifier (Fig. 5.15) to meet the following criteria: $v_o = 3v_b - 4v_a$. The resistance seen by the signal source v_b is 470 kΩ, and the resistance seen by the signal source v_a is 22 kΩ when the output voltage v_o is zero. Specify the values of R_a, R_b, R_c, and R_d using single resistors or combinations of resistors from Appendix H.

5.31 The resistor R_f in the circuit in Fig. P5.31 is adjusted until the ideal op amp saturates. Specify R_f in kilohms.

Figure P5.31



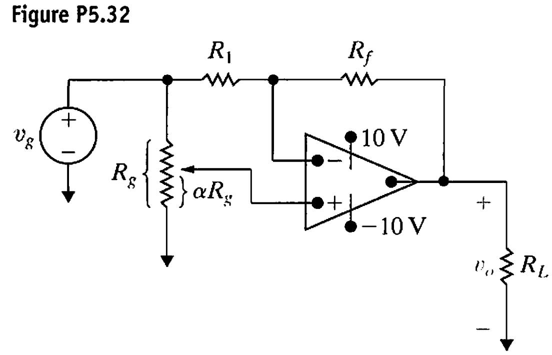
5.32 The op amp in the circuit of Fig. P5.32 is ideal.

a) Plot v_o versus α when $R_f = 4R_1$ and $v_g = 2$ V. Use increments of 0.1 and note by hypothesis that $0 \le \alpha \le 1.0$.

b) Write an equation for the straight line you plotted in (a). How are the slope and intercept of the line related to v_g and the ratio R_f/R_1?

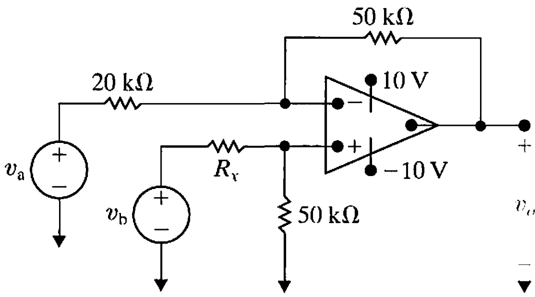
c) Using the results from (b), choose values for v_g and the ratio R_f/R_1 such that $v_o = -6\alpha + 4$.

Figure P5.32



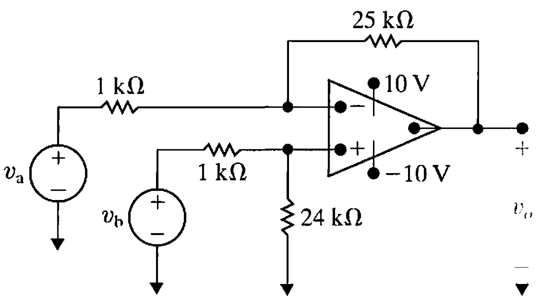
5.33 In the difference amplifier shown in Fig. P5.33, what range of values of R_x yields a CMRR $\geq$ 1000?

Figure P5.33



5.34 In the difference amplifier shown in Fig. P5.34, compute (a) the differential mode gain, (b) the common mode gain, and (c) the CMRR.

Figure P5.34



Sections 5.1–5.6

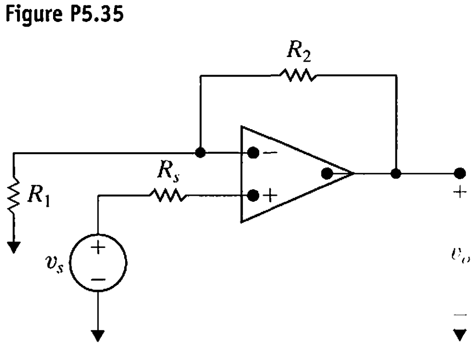
5.35 Assume that the ideal op amp in the circuit seen in Fig. P5.35 is operating in its linear region.

a) Show that $v_o = [(R_1 + R_2)/R_1]v_s$.

b) What happens if $R_1 \to \infty$ and $R_2 \to 0$?

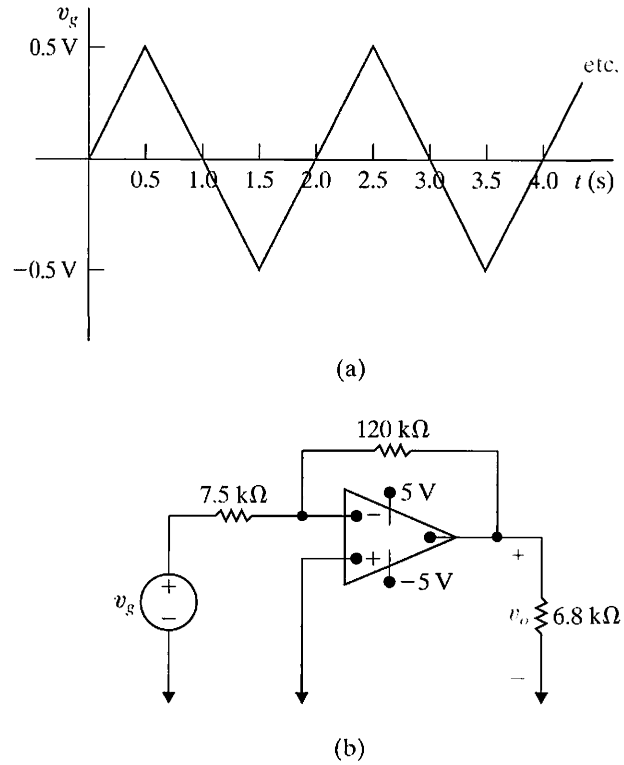
c) Explain why this circuit is referred to as a voltage follower when $R_1 = \infty$ and $R_2 = 0$.

Figure P5.35



5.36 PSPICE MULTISIM The voltage v_g shown in Fig. P5.36(a) is applied to the inverting amplifier shown in Fig. P5.36(b). Sketch v_o versus t, assuming the op amp is ideal.

Figure P5.36



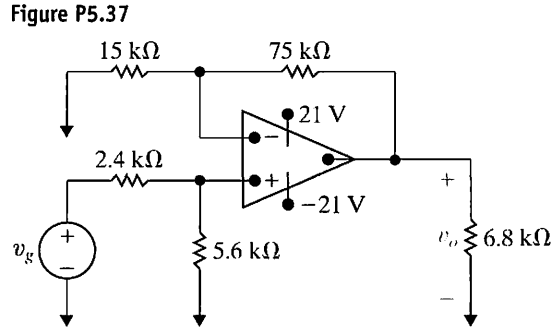
5.37 PSPICE MULTISIM The signal voltage v_g in the circuit shown in Fig. P5.37 is described by the following equations:

$$v_g = 0, \qquad t \leq 0,$$

$$v_g = 10 \sin(\pi/3)t \text{ V}, \qquad 0 \leq t \leq \infty.$$

Sketch v_o versus t, assuming the op amp is ideal.

Figure P5.37



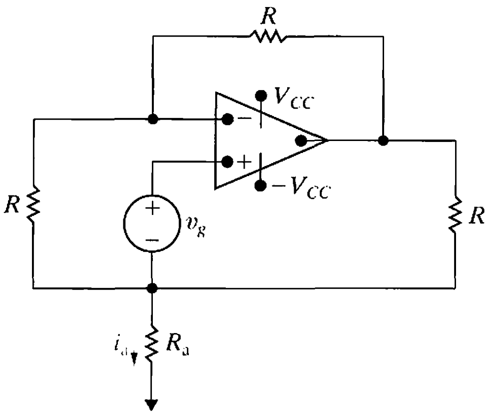
5.38 a) Show that when the ideal op amp in Fig. P5.38 is operating in its linear region,

$$i_a = \frac{3v_g}{R}.$$

b) Show that the ideal op amp will saturate when

$$R_a = \frac{R(\pm V_{CC} - 2v_g)}{3v_g}.$$

Figure P5.38



5.39 Assume that the ideal op amp in the circuit in Fig. P5.39 is operating in its linear region.
PSPICE MULTISIM

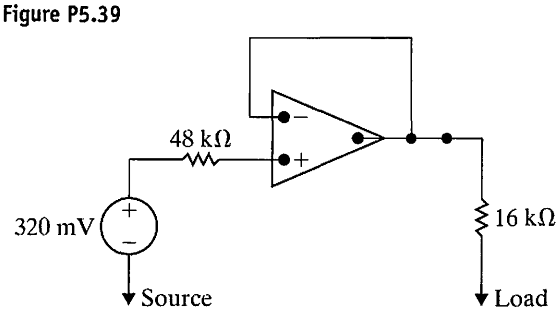
a) Calculate the power delivered to the 16 kΩ resistor.

b) Repeat (a) with the op amp removed from the circuit, that is, with the 16 kΩ resistor connected in the series with the voltage source and the 48 kΩ resistor.

c) Find the ratio of the power found in (a) to that found in (b).

d) Does the insertion of the op amp between the source and the load serve a useful purpose? Explain.

Figure P5.39



5.40 The circuit inside the shaded area in Fig. P5.40 is a constant current source for a limited range of values of R_L.
PSPICE MULTISIM

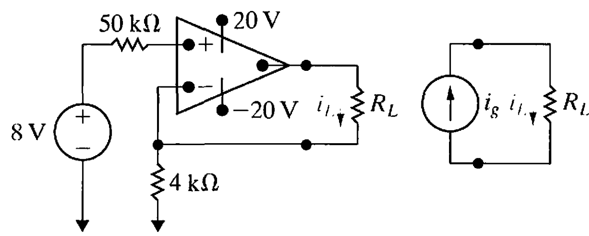
a) Find the value of i_L for $R_L = 4$ kΩ.

b) Find the maximum value for R_L for which i_L will have the value in (a).

c) Assume that $R_L = 16$ kΩ. Explain the operation of the circuit. You can assume that $i_n = i_p \approx 0$ under all operating conditions.

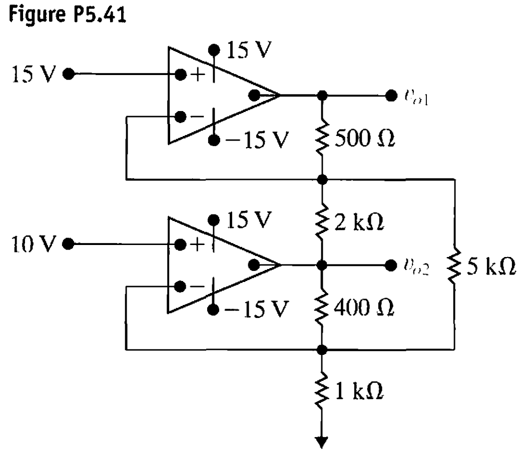
d) Sketch i_L versus R_L for $0 \leq R_L \leq 16$ kΩ.

Figure P5.40



5.41 The two op amps in the circuit in Fig. P5.41 are ideal. Calculate v_{o1} and v_{o2}.
PSPICE MULTISIM

Figure P5.41

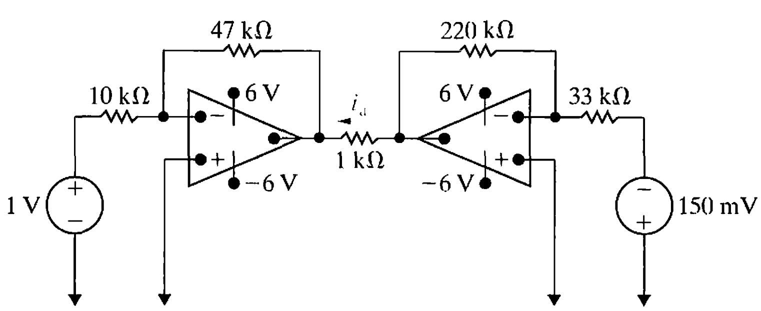


5.42 The op amps in the circuit in Fig. P5.42 are ideal.
PSPICE MULTISIM

a) Find i_a.

b) Find the value of the left source voltage for which $i_a = 0$.

Figure P5.42



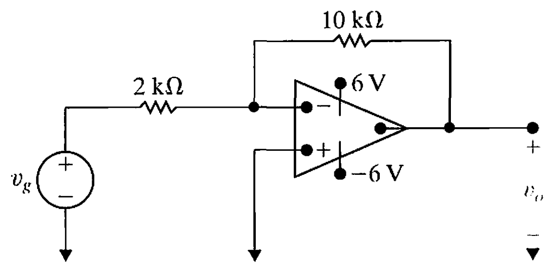
Section 5.7

5.43 PSPICE MULTISIM Repeat Assessment Problem 5.6, given that the inverting amplifier is loaded with a 500 Ω resistor.

5.44 PSPICE MULTISIM Assume the input resistance of the op amp in Fig. P5.44 is infinite and its output resistance is zero.

a) Find v_o as a function of v_g and the open-loop gain A.
b) What is the value of v_o if $v_g = 1$ V and $A = 150$?
c) What is the value of v_o if $v_g = 1$ V and $A = \infty$?
d) How large does A have to be so that v_o is 99% of its value in (c)?

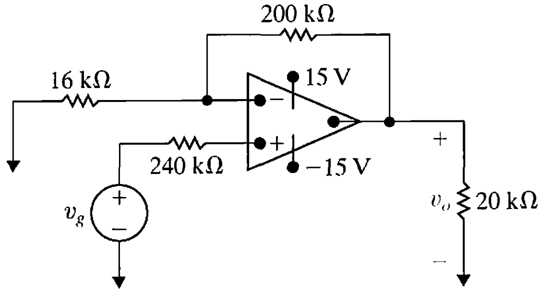
Figure P5.44



5.45 PSPICE MULTISIM The op amp in the noninverting amplifier circuit of Fig. P5.45 has an input resistance of 560 kΩ, an output resistance of 8 kΩ, and an open-loop gain of 50,000. Assume that the op amp is operating in its linear region.

a) Calculate the voltage gain (v_o/v_g).
b) Find the inverting and noninverting input voltages v_n and v_p (in millivolts) if $v_g = 1$ V.
c) Calculate the difference ($v_p - v_n$) in microvolts when $v_g = 1$ V.
d) Find the current drain in picoamperes on the signal source v_g when $v_g = 1$ V.
e) Repeat (a)–(d) assuming an ideal op amp.

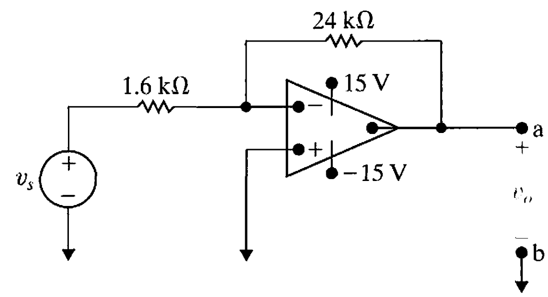
Figure P5.45



5.46 PSPICE MULTISIM a) Find the Thévenin equivalent circuit with respect to the output terminals a,b for the inverting amplifier of Fig. P5.46. The dc signal source has a value of 880 mV. The op amp has an input resistance of 500 kΩ, an output resistance of 2 kΩ and an open-loop gain of 100,000.

b) What is the output resistance of the inverting amplifier?
c) What is the resistance (in ohms) seen by the signal source v_s when the load at the terminals a,b is 330 Ω?

Figure P5.46



5.47 PSPICE MULTISIM Repeat Problem 5.46 assuming an ideal op amp.

5.48 Derive Eq. 5.60.

Sections 5.1–5.7

5.49 PRACTICAL PERSPECTIVE Suppose the strain gages in the bridge in Fig. 5.21 have the value 120 Ω ± 1%. The power supplies to the op amp are ±15 V, and the reference voltage, v_{ref}, is taken from the positive power supply.

a) Calculate the value of R_f so that when the strain gage that is lengthening reaches its maximum length, the output voltage is 5 V.
b) Suppose that we can accurately measure 50 mV changes in the output voltage. What change in strain gage resistance can be detected in milliohms?