Exercises on Static Circuits

1. *Modelling a compact disc player.* The (left or right channel) output of a typical CD player can be modeled as a voltage source that is able to produce voltage between -5V and $+5$V and currents between -10mA and $+10$mA without distorting. The CD input on a receiver can be modeled as a resistance of $R\Omega$.

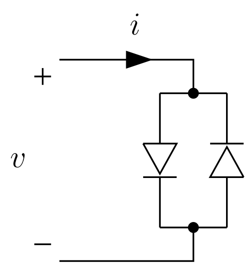
   (a) For what values of R is the maximum output voltage of the CD player limited by the ± 5V voltage limit, and for what values of R is it limited by the ± 10mA current limit? (Typically, R exceeds 10kΩ.)

   (b) For $R = 10$kΩ, find the maximum power in Watts that can be transferred from the CD player (say, from the Left channel output) into the receiver (via the Left channel CD input).

   These values are typical of all consumer line-level audio electronics.

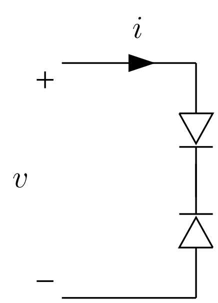
2. *Connecting two diodes.* Consider two diodes characterized by the exponential model $i = i_0(e^{v/v_{\mathrm{T}}} - 1)$, with $i_0 = 10^{-14}$A and $v_{\mathrm{T}} = 26$mV.

   (a) Suppose we form a two-terminal circuit element by connecting the diodes in parallel but oppositely oriented, as shown below:

   Give the $v - i$ relation for this element and sketch it, or plot it using Matlab. What is the slope of this curve at $v = i = 0$? Can you think of any practical use for this element?

   (b) Suppose we form a two-terminal circuit element by connecting the diodes in series but oppositely oriented, as shown below:

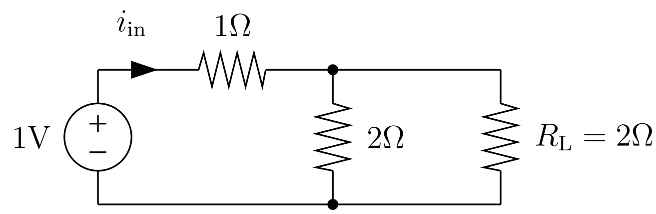
   Give the $v - i$ relation for this element and sketch it, or plot it using Matlab.

3. A typical 10MΩ 1/8W resistor is 7mm long. As the voltage across it is increased, which is likely to happen first:

   (a) the 1/8W power rating is exceeded, or

   (b) an arc forms from lead to lead?

4. Consider the circuit below:



   (a) Find i_{in}.

   (b) What fraction of the power delivered by the voltage source is dissipated in resistor R_{L}?

5. *Deriving the $v - i$ relation for a light bulb.* Many circuit element models are derived from physics, *e.g.*, semiconductor physics, E&M, etc. In this problem you use some simple physics to derive an electrical model of an ordinary light bulb.

   An incandescent lamp works by heating up its filament to a temperature at which the net heat lost from the filament equals the electrical power input $p_{\text{in}} = vi$. The net heat lost is a function of the filament temperature T, say, $f(T)$ (in units of Watts, with T in degrees K). The function f increases with T and satisfies $F(T_{\text{amb}}) = 0$ where T_{amb} is the ambient temperature (usually 300°K), *i.e.*, at ambient temperature, no net heat is lost from the filament. The filament temperature of a typical incandescent lamp operating at its standard voltage is between 2800°K and 3400°K.

   A simple thermal model that includes only radiated heat is $f(T) = \alpha(T^4 - T_{\text{amb}}^4)$ where $\alpha > 0$ is a constant that depends on the filament (surface area, emissivity). More complicated thermal models would include terms for conduction and convection, the effects of emissivity varying with temperature, and so on.

   Electrically, the filament is a resistor, *i.e.*, $v = Ri$. Because of the extreme variation in filament temperature, however, we must take into account the variation in R with T. it turns out that the filament resistance is accurately modeled by $R = R_0 + c(T - T_{\text{amb}})$ where R_0 is the filament resistance at ambient temperature and $c > 0$ is a constant called the *resistance temperature coefficient.*

   When you first turn on an incandescent lamp, its resistance is R_0 since its filament temperature starts at $T = T_{\text{amb}}$. This resistance is lower than its steady-state or equilibrium value (which it reaches in a fraction of a second, as the filament reaches its operating temperature). Hence when you first turn on an incandescent lamp, a larger current flows (for a short period) than you'd expect from the power rating of the lamp. This is called the *cold inrush current,* and it is an important practical effect. The cold inrush current is often a factor of eight or more times the steady-state current.

From the relations given above we can derive the (steady-state) $v - i$ relation for an incandescent lamp. First verify that the relation is symmetric, *i.e.*, if (v, i) lies on the curve, then so does $(-v, -i)$. So we will assume that v and i are positive. Now show that

$$i = \left(\frac{f(T)}{R_0 + c(T - T_{\text{amb}})} \right)^{1/2}, \quad v = (f(T)(R_0 + c(T - T_{\text{amb}})))^{1/2}.$$

These equations parametrize the v–i curve by the parameter T; by varying T from T_{amb} to the maximum filament temperature we can trace out the v–i curve (by hand, or using Matlab, etc.).

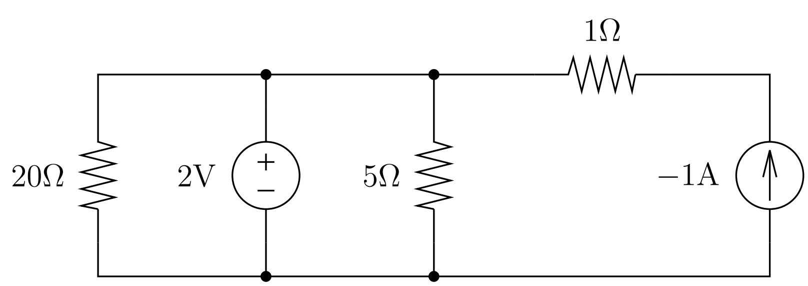
Now consider a typical 125V/100W lamp that operates at 3000°K (when $v = 125$V). The cold inrush current is eight times the operating current. You can use the simplified f described above.

(a) Use Matlab to plot the v–i curve of this lamp. Verify that it has the general shape shown in the notes.

(b) What voltage results in a power that is one-half the rated power, *i.e.*, 50W? Compare this with the voltage that yields half-power for a (linear) resistor that dissipates 100W at 125V.

(c) As we discussed in class, every model has limits of applicability. Briefly decribe some of the limits of applicability for the model of a 125V/100W lamp found above. For example, do you think that the model predicts the current accurately for $v = 400$V? Does it accurately predict the current when the voltage is rapidly varying? Roughly how accurately would you expect the model to predict i given v, over the range $|v| \leq 125$V? (*i.e.*, 0.001%, 1%, or 10%?) You can give educated guesses as your answers.

6. Consider the circuit below:



(a) Find the power p_{vs} delivered by the voltage source.

(b) Find the power p_{cs} delivered by the current source.

7. *Does current takes the path of least resistance?* The idea of the phrase "current takes the path of least resistance" is essentially the approximation $R_1 \| R_2 \approx \min\{R_1, R_2\}$, *i.e.*, the parallel connection of two resistors yields a resistance about equal to the minimum of the two resistances. Of course this approximation is never exact. But how far off can it be?

   (a) Show that whenever R_1, R_2 are positive we have

   $$\frac{1}{2}\min\{R_1, R_2\} \leq R_1 \| R_2 \leq \min\{R_1, R_2\}.$$

   Thus, the approximation is never off by more than 100%.

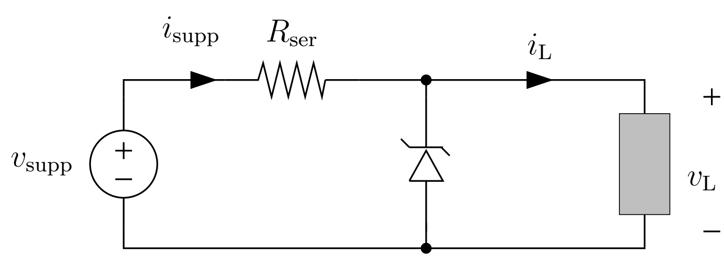
   (b) Find an example where the approximation yields a 100% error.

   (c) Find the conditions on R_1 and R_2 such that the approximation $R_1 \| R_2 \approx \min\{R_1, R_2\}$ is accurate to 10%.

   (d) Use Matlab to plot the relative error of the approximation as a function of the resistance ratio R_2/R_1 (assuming both are positive). The relative error is given by

   $$\frac{|\min\{R_1, R_2\} - R_1 \| R_2|}{|R_1 \| R_2|}.$$

8. *Zener diode voltage regulator.* The circuit below shows a simple *voltage regulator* based on a zener diode with zener voltage v_z. In this circuit, the voltage across the load, v_L, is equal to v_z over a range of input supply voltage v_{supp} and a range of load current i_L. Thus, the voltage v_L has been *regulated* against variations in supply voltage and load current. The resistor R_{ser} is called the *series resistor.*



   In this problem you will explore the design of such a regulator. You are given a maximum load current and a range of supply voltages, *i.e.*, you know i_{max}, v_{min}, and v_{max} such that:

   $$0 \leq i_L \leq i_{max}, \qquad v_{min} \leq v_{supp} \leq v_{max}.$$

   You may assume that the zener diode is described by the ideal zener diode characteristic shown on page **??** of the notes.

(a) First assume the circuit is regulating, *i.e.*, $v_L = v_z$. Find expressions for the power dissipated in the series resistance, the zener diode, and the load, in terms of R_{ser}, i_L, v_{supp}, and v_z. Given the voltage source and load current ranges above, find the maximum power that can be dissipated in the zener diode. Repeat for the series resistor.

Note: The answers to these questions are used to properly size the diode and the resistor. A component that can safely dissipate a large power (*e.g.*, 1W) will cost more and be larger than one that handles a smaller power (say, 1/8W), so it's desirable to find the smallest safe power-rating for a component.

(b) Now we consider the conditions under which $v_L = v_z$ (which you assumed in part (a)). Show that this happens provided

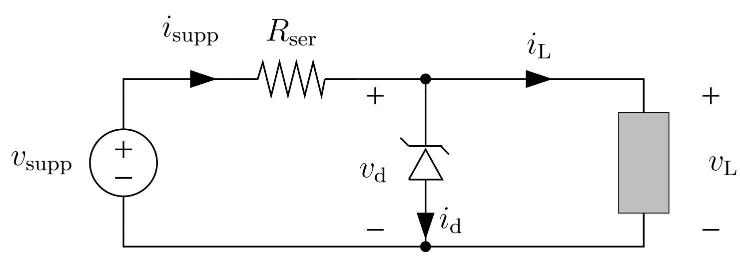
$$v_{supp} - i_L R_{ser} \geq v_z.$$

What conditions on R_{ser} guarantee that we will have $v_L = v_z$ over the full range of v_{supp} and i_L given above?

(c) Design a zener diode voltage regulator that operates for load currents between 0 and 100mA at $v_L = 15$V, for supply voltages ranging between 18V and 25V. (By design, we mean: find R_{ser} and say what power ratings the resistor and diode must have.) Your design should make the power rating of the series resistor and diode as small as possible.

Note: these values are realistic.

In this problem, we investigate the design of a proper voltage regulator using a zener diode. The problem walks us though the design of a circuit that keeps the voltage on the load constant even while the source voltage and load current change. Also, bounds on the source voltage and load current provide a means to design the circuit with the least expensive series resistor and diode by *sizing* them appropriately.



(a) First, we assume the zener diode is operating at the zener voltage, $v_d = v_z$, and develop expressions for the power dissipation in the specified circuit elements.

- Series Resistor: The voltage across the resistor is the source voltage minus the zener/load voltage. Using $p = \frac{v^2}{R}$,

$$p_{ser} = \frac{(v_{supp} - v_z)^2}{R_{ser}}.$$

- Zener Diode: Let the voltage across the diode be $v_{\mathrm{z}} = v_{\mathrm{L}}$ and let the current through the diode be $i_{\mathrm{d}} = i_{\mathrm{supp}} - i_{\mathrm{L}}$. i_{supp} can be found by applying Ohm's Law:
$$i_{\mathrm{supp}} = \frac{v_{\mathrm{supp}} - v_{\mathrm{z}}}{R_{\mathrm{ser}}},$$
so that,
$$p_{\mathrm{d}} = v_{\mathrm{z}} \left[i_{\mathrm{supp}} - i_{\mathrm{L}} \right] = v_{\mathrm{z}} \left[\frac{v_{\mathrm{supp}} - v_{\mathrm{z}}}{R_{\mathrm{ser}}} - i_{\mathrm{L}} \right].$$

- Load: The power transferred to the load is simply:
$$p_{\mathrm{L}} = v_{\mathrm{z}} i_{\mathrm{L}}.$$

We can now find the maximum possible power dissipated in the diode and the series resistance by using the limits on load current and supply voltage given in the problem description. The maximum power dissipated in both devices occurs when the supply voltage is at a maximum ($v_{\mathrm{supp}} = v_{\max}$) and when the load current is at a minimum ($i_{\mathrm{L}} = 0$).

$$\begin{aligned} \max_{v_{\mathrm{supp}}, i_{\mathrm{L}}} p_{\mathrm{ser}} &= \frac{(v_{\max} - v_{\mathrm{z}})^2}{R_{\mathrm{ser}}}, \\ \max_{v_{\mathrm{supp}}, i_{\mathrm{L}}} p_{\mathrm{d}} &= v_{\mathrm{z}} \left[\frac{v_{\max} - v_{\mathrm{z}}}{R_{\mathrm{ser}}} \right]. \end{aligned}$$

(b) The v–i curve of the ideal zener diode appears on page 2-6 of the notes. The reference polarity for the curve in the notes is not the same as the polarity definition for the diode in this solution (i.e. v_{d} and i_{d}). With these reference polarities, the zener diode operates at its zener voltage when the voltage drop across the diode equals v_{z} and the current through the diode is positive:

$$\begin{aligned} v_{\mathrm{d}} &= v_{\mathrm{z}} = v_{\mathrm{supp}} - i_{\mathrm{supp}} R_{\mathrm{ser}}, \\ i_{\mathrm{d}} &\geq 0. \end{aligned}$$

Since the current flowing down through the diode is positive, we can conclude that $i_{\mathrm{supp}} \geq i_{\mathrm{L}}$, so that

$$v_{\mathrm{supp}} - i_{\mathrm{L}} R_{\mathrm{ser}} \geq v_{\mathrm{supp}} - i_{\mathrm{supp}} R_{\mathrm{ser}} = v_{\mathrm{z}}.$$

Hence, the load voltage is regulated at v_{z} when,

$$v_{\mathrm{supp}} - i_{\mathrm{L}} R_{\mathrm{ser}} \geq v_{\mathrm{z}}.$$

Solving for the series resistance, we determine an upper bound on the possible values for R_{ser} such that the above equation holds:

$$R_{\mathrm{ser}} \leq \frac{v_{\mathrm{supp}} - v_{\mathrm{z}}}{i_{\mathrm{L}}}.$$

Given the variations in supplied voltage and load current, we need to find the most conservative bound on R_{ser}. This translates to finding the smallest upper bound:

$$R_{\text{ser}} \leq \min\left\{\frac{v_{\text{supp}} - v_{\text{z}}}{i_{\text{L}}}\right\} = \frac{v_{\text{min}} - v_{\text{z}}}{i_{\text{max}}}.$$

In other words, for any variations in v_{supp} and i_{L}, a series resistance that holds under the above condition will keep the voltage regulator working.

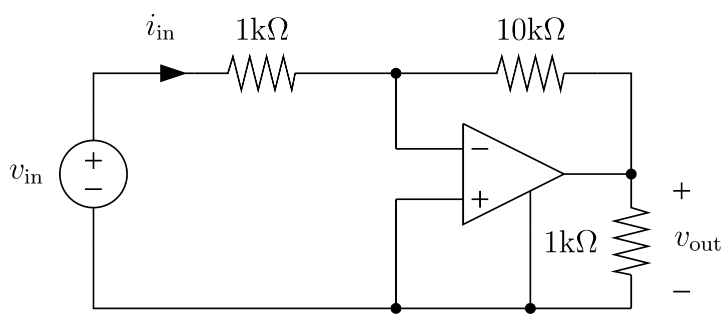
(c) Given values for i_{max},v_{min}, and v_{max}, we can find the range of possible series resistances using the result in (b):

$$R_{\text{ser}} \leq \frac{18 - 15}{.1} = 30\Omega$$

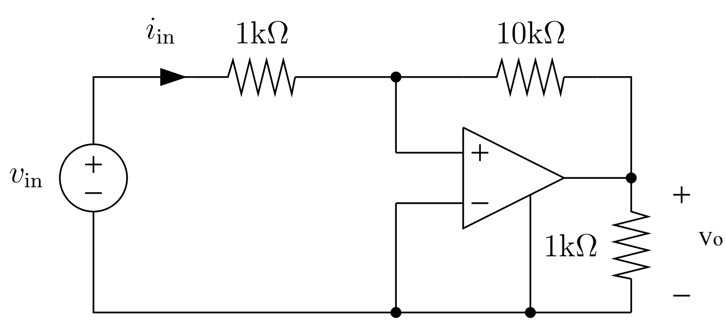
Therefore, any resistance less that 30Ω will work. However, by examining the results in part (a), one can see that the maximum power dissipated in the diode and resistance is less when R_{ser} is large. Hence, we choose $R_{\text{ser}} = 30\Omega$ to minimize the necessary power rating:

$$\begin{aligned} \max p_{\text{z}} &= 5\text{W} \\ \max p_{\text{ser}} &= 3.33\text{W} \end{aligned}$$

9. *An inverting amplifier using an op-amp.* This problem concerns the two op-amp circuits shown below:



Circuit A



Circuit B

The only difference is the orientation of the input terminals on the op-amp.

(a) Using the ideal op-amp model (page **??** of lectures notes), find i_{in} and v_{out} in circuit (A).

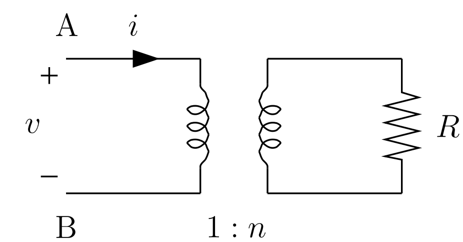
(b) Repeat for circuit (B).

(c) A more realistic model of an op-amp is a VCVS with a gain of 10^5, *i.e.*, $v_{\text{out}} = 10^5\tilde{v}$ where $\tilde{v}$ is the voltage difference across the + and − input terminals of the op-amp. Using this op-amp model, find i_{in} and v_{out} in the circuit (A).

(d) Repeat for circuit (B).

(e) With a real op-amp, the circuit (A) will work (meaning, i_{in} and v_{out} predicted by the ideal op-amp model (a) will be very close to the actual current and voltage in the real circuit) while the circuit (B) won't. Do either of the op-amp models considered in this problem predict this?

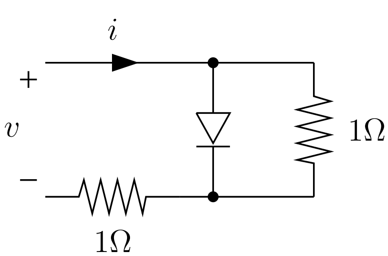
10. *Reflected resistance seen through a transformer.* Consider the circuit shown below:



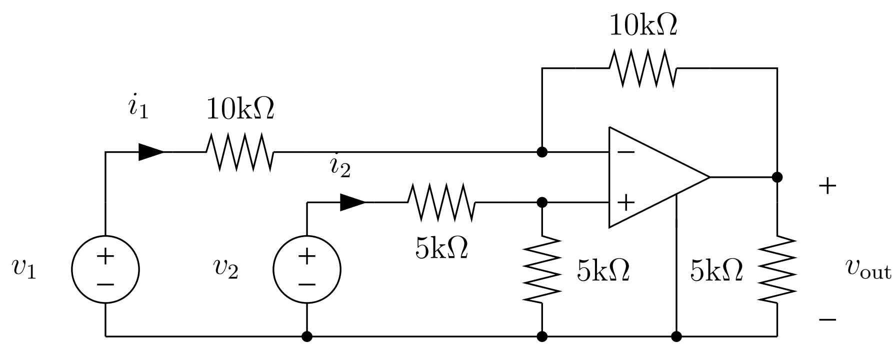
Show that $v = iR_{\text{eff}}$ for some appropriate R_{eff} (the subscript stands for "effective").

This means that this circuit, from the point of view of the terminals A and B, is electrically equivalent to a resistor of value R_{eff}. R_{eff} is often called the "reflected resistance of R seen through the transformer."

11. Sketch the $v - i$ characteristic of the two-terminal element below. The diode is characterized by the ideal diode model.

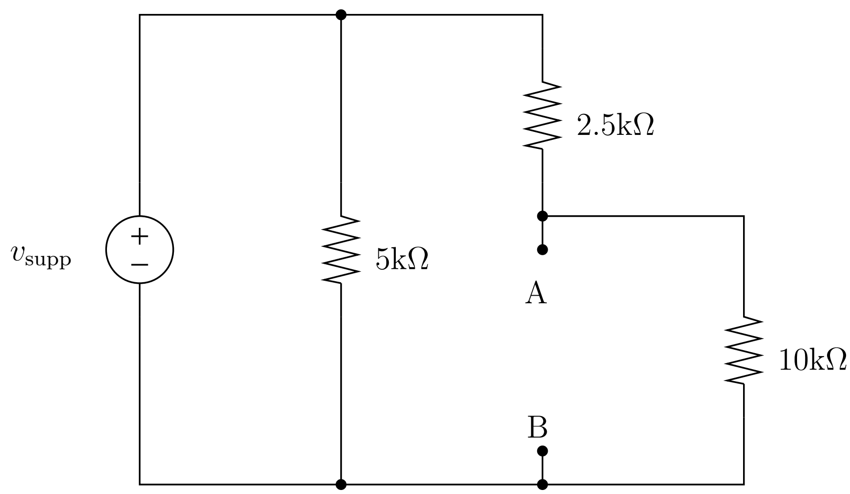


12. The op-amp in the circuit below is characterized by the ideal op-amp model. Find i_1, i_2, and v_{out}. What does this circuit do? Can you think of a practical application of this circuit?



13. This problem concerns the two related circuits shown below.

    (a) Find the Thevenin voltage v_{th} and Thevenin resistance R_{th} of the circuit shown below, with respect to the terminals A and B. (Your answers should be as explicit as possible, but may contain the parameter v_{supp}.)

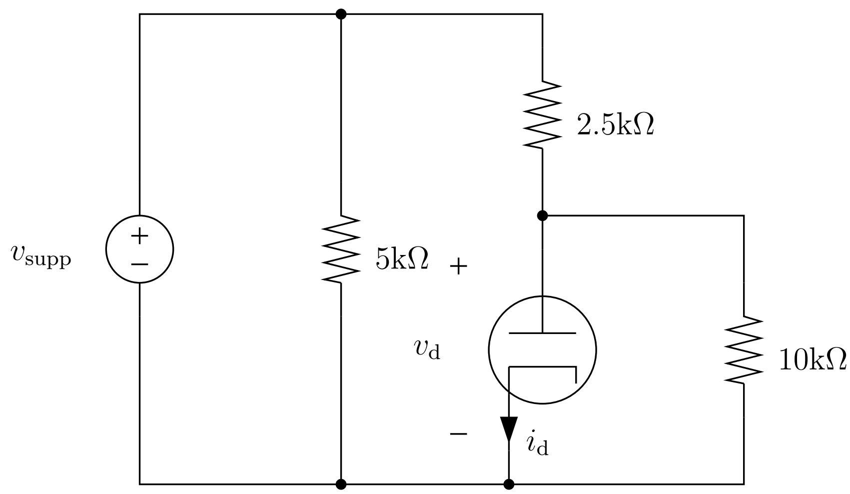


    (b) Now consider the circuit below. The vacuum tube rectifier is characterized by

$$i_{\text{d}} = \begin{cases} kv_{\text{d}}^{3/2} & v_{\text{d}} \geq 0 \\ 0 & v_{\text{d}} < 0 \end{cases}$$

where $k = 0.3$ mA/V$^{3/2}$. (Please note the units: i_{d} is in milliamps and v_{d} is in volts.)

Find the value of v_{supp} that results in $i_{\text{d}} = 30$mA.



(c) Let v_{supp} have the value you found in part (b), so that $i_{\text{d}} = 30\text{mA}$. Find the power p_{res} dissipated in the 2.5kΩ resistor. and the power p_{supp} delivered by the voltage source.

(d) Suppose that v_{supp} is 5 volts *less* than the value you found in part (b). Give an *estimate* of v_{d}. Explain what you are doing.

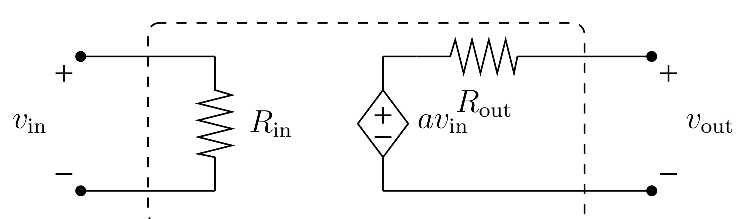
14. *Bias and small signal analysis of a MOS amplifier.* This problem concerns the MOS amplifier circuit shown at the top of page **??** of the notes, except we do not assume that $v_{\text{in}} \approx 4\text{V}$. The MOS transistor characteristic is given on page **??** of the notes, with parameters $v_{\text{th}} = 2\text{V}$, $\beta = 2\text{mA}/v^2$.

    (a) *Bias calculation.* Determine what v_{in} must be so that $v_{\text{out}} = 2\text{V}$.

    (b) *Forming a linearized model.* Determine the linearized model of the MOS transistor accurate near the bias condition found in part (a). Give equations describing the approximation and also a circuit model of the approximation.

    (c) *Small signal analysis.* Find an expression for v_{out} in terms of v_{in} that is accurate for v_{in} near the bias value found in part (a).

15. *Amplifiers.* The schematic diagram below shows a general model of an (ideal, linear) amplifier. The two terminals on the left are called the *input port* and the two terminals on the right are called the *output port.* An amplifer is characterized by its *input resistance* R_{in}, its *voltage gain* a, and its *output resistance* R_{out}.

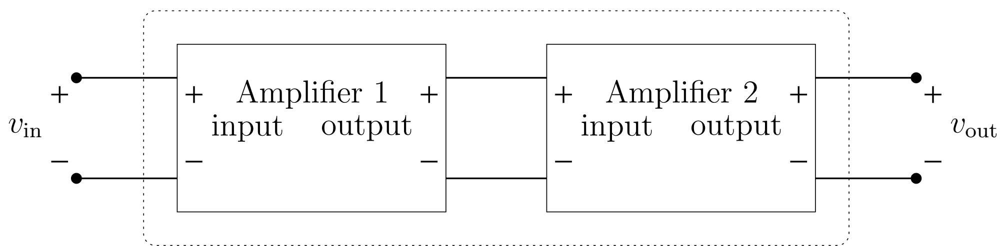


Note that when $R_{\text{out}} = 0$ and $R_{\text{in}} = \infty$, the amplifier reduces to a VCVS. Also note that the voltage and current at the input port is completely unaffected by the voltage and current at the output port.

Here are some typical values for amplifiers:

- For a line-level audio amplifier, we might have $R_{\text{in}} \approx 10\text{k}\Omega$, $R_{\text{out}} \approx 100\Omega$ or less, and a gain in the range 1 to 10.
- For an audio power amplifier (*i.e.*, one that accepts a line-level input and drives a speaker), we might have $R_{\text{in}} \approx 10\text{k}\Omega$, $R_{\text{out}} \approx 0.01\Omega$, and a gain in the range 10 to 100. **Note:** the output resistance of an audio power amplifer is *not* 4Ω or 8Ω.
- For high-frequency (*e.g.*, video) circuits, we might have $R_{\text{in}} = R_{\text{out}} = 50\Omega$ or 75Ω and a gain in the range of 1 to 10.
- A typical real op-amp is pretty well modeled by an amplifier with $R_{\text{in}} \approx 100\text{k}\Omega$, $R_{\text{out}} \approx 100\Omega$, and a gain of about 10^5. (This model gives better predictions than the ideal op-amp model, but is considerably harder to work with for hand analysis.)

(a) Suppose we hook up two amplifiers in cascade, as shown below.



Amplifier 1 has parameters $R_{\text{in}}^{(1)}$, $R_{\text{out}}^{(1)}$, and $a^{(1)}$; Amplifier 2 has parameters $R_{\text{in}}^{(2)}$, $R_{\text{out}}^{(2)}$, and $a^{(2)}$.

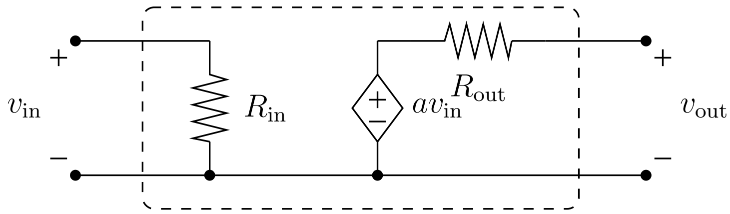
Show that the resulting four-termimal element (enclosed by the dotted box) is equivalent to a single amplifer. Find the parameters of the equivalent amplifer (*i.e.*, R_{in}, R_{out} and gain).

(b) Consider a video amplifier with $R_{\text{in}} = R_{\text{out}} = 75\Omega$ and a voltage gain of 10. Suppose the amplifier is driven by a voltage source (*i.e.*, a voltage source is connected across the input terminals), and the output terminals are connected to a load resistor which is 75Ω. Find the *power gain* of the amplifier, which is defined as the ratio of the power flowing out of the output port to the power flowing into the input port.

(c) The power gain of an amplifier depends on the load resistance. Find an expression for the power gain of an amplifier in terms of the parameters R_{in}, R_{out}, a, and the load resistance R_{L}. What load resistance maximizes the power gain? The resulting power gain is called the *maximum power gain* of the amplifier. Give an expression for maximum power gain in terms of R_{in}, R_{out}, and a.

16. *Grounded (single-ended) amplifiers.* In the amplifier model described in problem 15, no current can flow between the input terminals and the output terminals, *i.e.*, the input terminals and output terminals form two separate "ports". Sometimes this is called a "floating" or "isolating" amplifier to emphasize this fact.

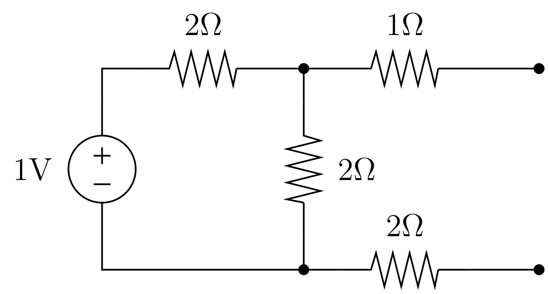
    Many real amplifiers are better modeled by the circuit shown below:

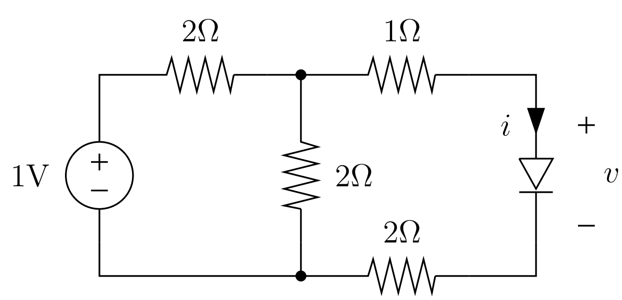
    Note that you can make a floating amplifier into a single-ended or grounded amplifier by connecting its input and output − terminals together.

    Are floating and grounded amplifiers electrically equivalent? If so, explain why. If not, find a simple circuit that contains an amplifier, and behaves differently when the amplifier is floating or grounded.

17. (a) Find the Thevenin equivalent for the circuit below:

    (b) Find v and i in the circuit below. The diode is characterized by the ideal diode model.

18. *Maximum power from a source with nonlinear $v - i$ characteristic.* Suppose that the $v - i$ characteristic of a battery is given by $v = 10 - 5i - 5i^2$ for $0 \le i \le 1$, where the current reference direction for i is *out of* the positive battery terminal. (For batteries,

power supplies, and the output of power amplifiers, it is common to use these reference polarities, which are *opposite* the normally used ones. In this case $p = vi$ is the power *delivered by* or *drawn from* the battery or power supply.)

(a) Sketch the $v - i$ characteristic of this battery, or use Matlab to plot it.

(b) Give an approximation of the $v - i$ characteristic valid for small currents. Show a Thevenin circuit that has the same $v - i$ characteristic as your approximation.

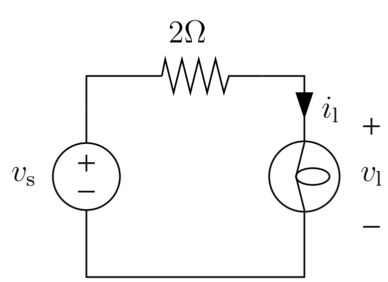
(c) If the battery is terminated in a resistance whose value is equal to the Thevenin resistance found in part (b), what is the power drawn from the battery?

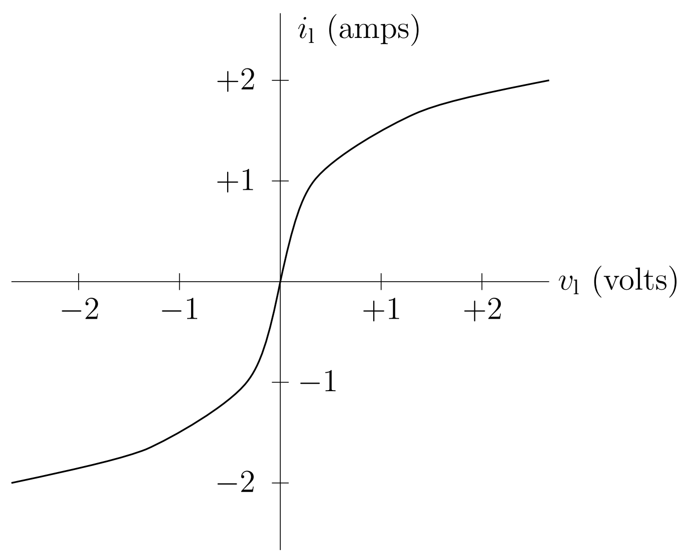
(d) Find the resistance value that maximizes the power drawn from the battery. Find the corresponding voltage, current, and power. Verify your answer by plotting some constant power curves (with dotted line type) on top of the $v - i$ characteristic.

(e) Find an approximate Thevenin equivalent of the battery that is valid near the voltage and current found in part (d).

(f) Conjecture a generalization of the maximum power transfer theorem to nonlinear $v - i$ characteristics.

19. Find a value of v_s such that the lamp in the circuit below dissipates 1.5W. The lamp characteristic is plotted at right. You can make reasonable estimates from the plot.



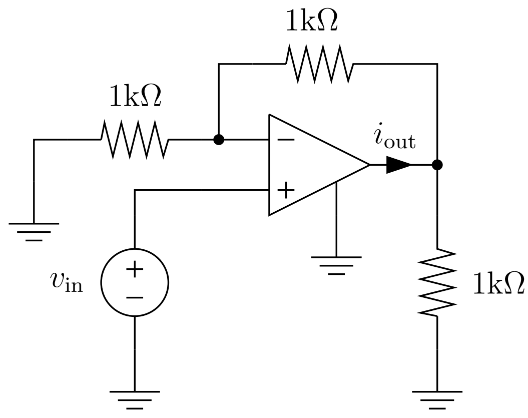


20. In the circuit below, v_{in} can vary over the range ± 5V.

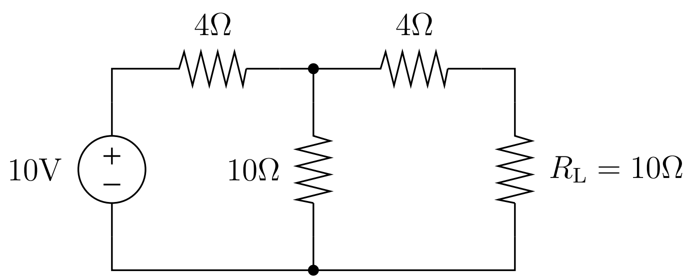
What is the largest (magnitude) output current the op-amp must supply? I.e., what is the maximum value of $|i_{out}|$ for $-5\text{V} \le v_{in} \le 5\text{V}$?

(This is an important practical question, since real op-amps have output current limits. A typical value is ± 10mA.)

You can use the ideal op-amp model.



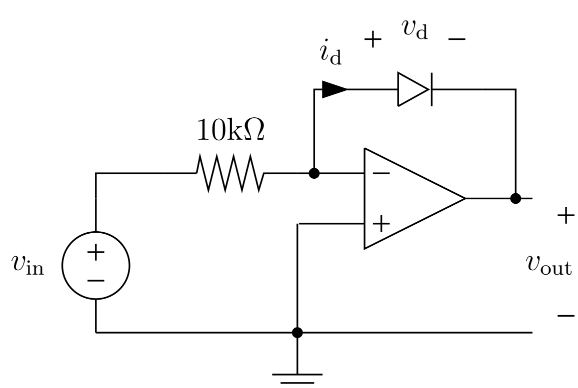
21. In the circuit shown below, the voltage source delivers power p_{src} and the load resistor R_{L} dissipates power p_{L}.



Find the power transfer efficiency, *i.e.*, $p_{\text{L}}/p_{\text{src}}$.

22. The diode in the circuit below is characterized by the exponential model $i_{\text{d}} = i_0(e^{v_{\text{d}}/v_t} - 1)$, with $i_0 = 10^{-14}$A and $v_t = 26$mV.

You can use the ideal model for the op-amp. The input voltage v_{in} varies from 10mV to 10V (always positive).

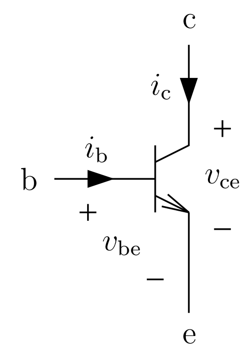


Express v_{out} as an explicit function of v_{in}.

23. An NPN transistor is described by the model

$$i_{\rm b} = i_0 e^{v_{\rm be}/v_t}, \quad i_{\rm c} = \beta(1 + v_{\rm ce}/v_a) i_{\rm b}$$

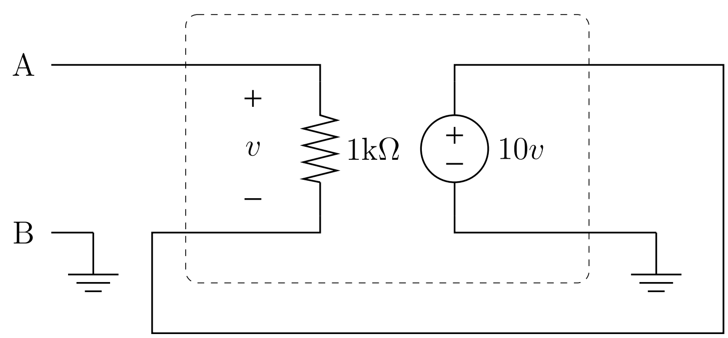
where $i_0 = 10^{-14}$A, $v_t = 26$mV, $\beta = 100$, and $v_a = 50$V, and the terminal currents and voltages are defined below. (This model captures the 'Early effect', in which $i_{\rm c}$ depends more strongly on $v_{\rm ce}$ than the Ebers-Moll model predicts. The parameter v_a is called the 'Early voltage'.)



Find the linearized circuit model of this transistor at the bias condition $v_{\rm be}^{\rm bias} = 0.7$V, $v_{\rm ce}^{\rm bias} = 10$V.

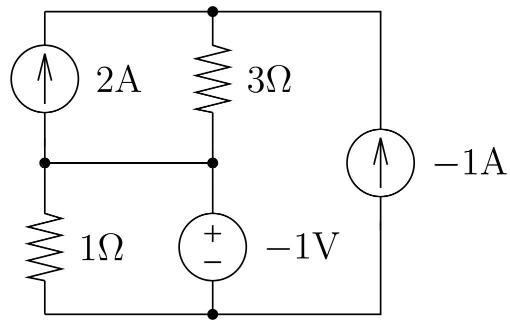
Express the linearized model as a circuit. Your circuit may contain voltage and current sources, resistors, and dependent sources. Be sure to clearly label the terminals c, b, and e, and to indicate the values of elements (*e.g.*, resistance, transconductance) in your circuit.

24. The circuit below shows an amplifier (in the dashed-line box) with input resistance 1kΩ, output resistance 0Ω, and gain 10, connected in a 'negative feedback' configuration (since the output is connected to the − input terminal).



Find the Thevenin resistance $R_{\rm th}$ looking into the terminals A and B.

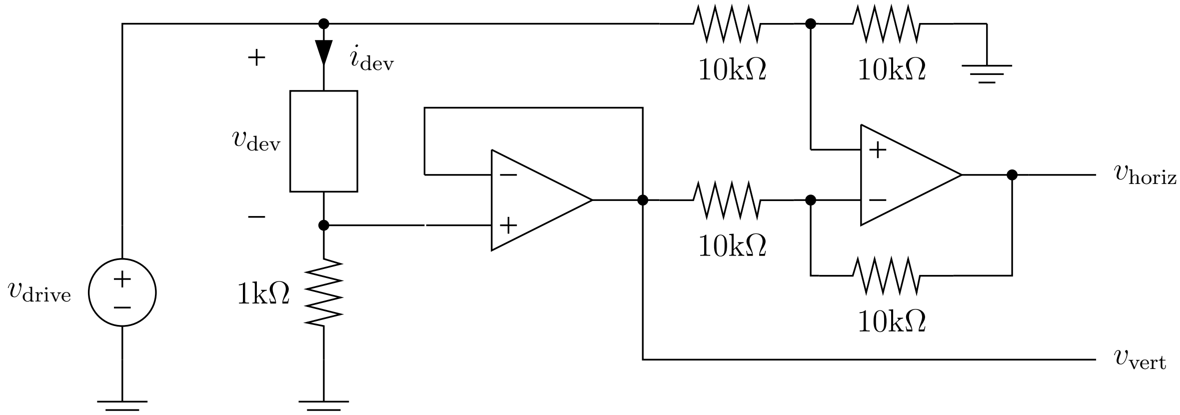
25. *Conservation of power.* Consider the circuit shown below.



Determine the power dissipated in each element. Which elements are supplying (positive) power? Which elements are absorbing (positive) power? Verify that the total power being supplied (by the elements supplying positive power) is equal to the total power absorbed (by the elements absorbing positive power).

You must explain the steps in your circuit analysis.

26. *$v - i$ curve tracer.* The following is a circuit used for curve tracing:



The box is the device to be tested, sometimes called the 'device under test' (D.U.T.). The output voltage v_{vert} is connected to the vertical deflection input of the oscilliscope, and the output voltage v_{horiz} is connected to the horizontal deflection input of the oscilliscope. The oscilliscope is set to 1V per division, horizontal and vertical. The voltage source v_{drive} is used to sweep out the $v - i$ curve. During the demo it had the form $v_{\text{drive}}(t) = 10 \sin 300t$ (approximately).

To analyze this circuit you can assume the op-amps are ideal.

Express the output voltages v_{vert} and v_{horiz} in terms of the device voltage v_{dev}, device current i_{dev}, drive voltage v_{drive}, and any other relevant parameter. Describe the steps in your analysis of this circuit.

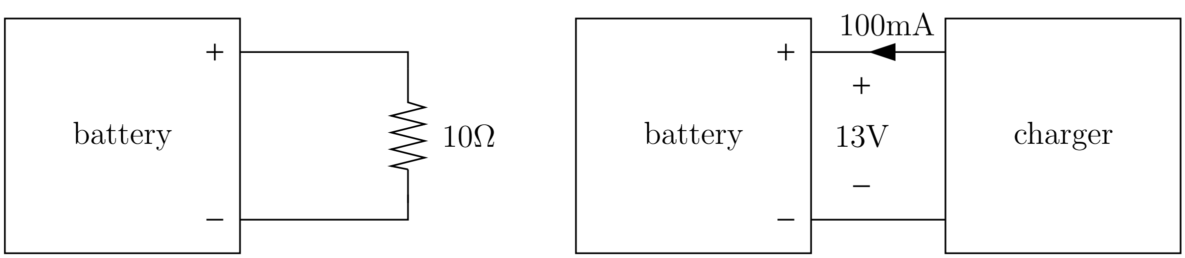
Explain how this circuit, connected to the oscilliscope as described, can be used to plot the v — i curve of the device. What are the resulting axis sensitivities (in V and A per oscilliscope division) and total range?

27. *Extracting maximum power from a battery.*

Consider a battery that can be modeled using a Thevenin equivalent.

When a 10Ω load resistance is connected to the battery, as shown at left below, 10W is dissipated in the load resistor.

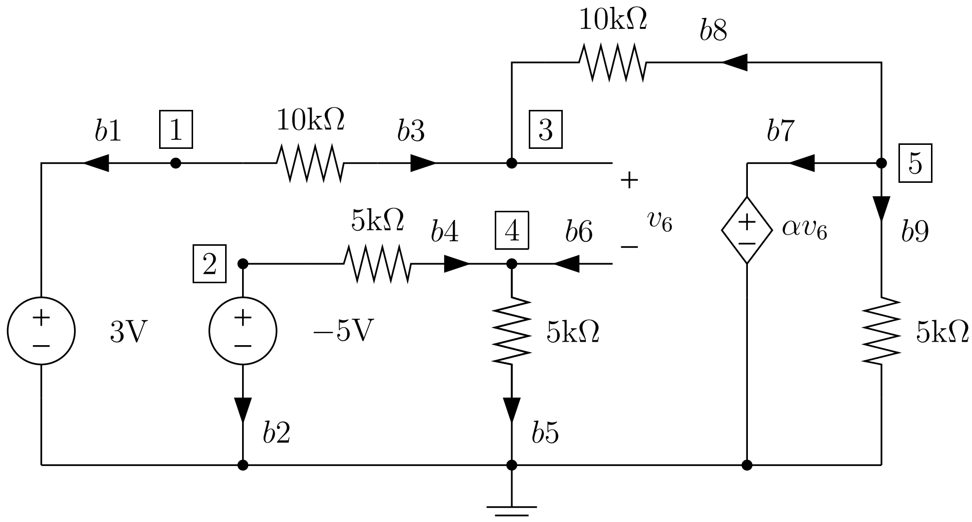
When the battery is connected to a 100mA charger, as shown at right below, the voltage across the battery terminals is 13V.



A load resistance R is connected to the battery. Find the value of R that results in maximum power dissipation in the load resistor.

Exercises on Static Circuits: Part II

1. This problem concerns the circuit shown below, with the branches oriented and labeled b1,. . . , b9, the nodes labeled $\boxed{1}$, . . . , $\boxed{5}$, and the voltage sources given specific numeric values. In addition, the op-amp has been replaced by a VCVS with a gain of $\alpha = 100$, with a sense branch b6 and an output branch b7.



   (a) Find the reduced incidence matrix for this circuit.

   (b) Write out KCL, KVL, and the branch relations for this circuit. Express these circuit equations in the form of a giant matrix equation $Fx = g$ where F is a large square matrix and x is a big vector consisting of $i_1, \ldots, i_9, v_1, \ldots, v_9, e_1, \ldots, e_5$. In writing down this big matrix, you don't need to explicitly write zeros. What percentage of the entries in the big matrix F are zero?

   (c) Solve the equations you found in part 1b, *i.e.*, find the solution vectors i, v, and e. You can do this by hand (it's not so bad!) or using Matlab.

   (d) Verify that the matrix equations describing KCL and KVL hold: $Ai = 0$ and $v = A^T e$.

   (e) Which branches are dissipating (absorbing) power and which branches are supplying power?

   (f) Calculate the quantity $p = v^T i$. Can you explain the answer that you get?

2. The reduced incidence matrix of a circuit is:

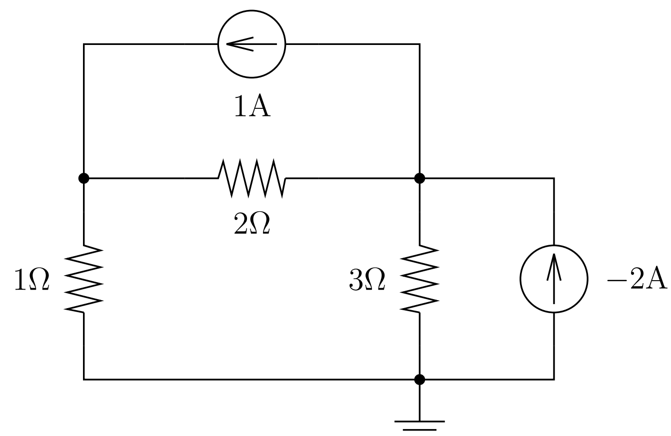
$$A = \begin{bmatrix} 1 & 1 & 0 & 0 & 0 \\ 0 & -1 & 1 & 1 & 0 \\ 0 & 0 & 0 & -1 & 1 \end{bmatrix}.$$

The branch relations are

$$\begin{bmatrix} -1 & 0 & 0 & 0 & 0 \\ 0 & -2 & 0 & 0 & 0 \\ 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -3 \end{bmatrix} i + \begin{bmatrix} 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 2 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 1 \end{bmatrix} v = \begin{bmatrix} 1 \\ 0 \\ 0 \\ 4 \\ 0 \end{bmatrix}.$$

Draw a conventional schematic diagram of this circuit. Label the branches and nodes, and show the orientation of the branches. Give the numerical values of the elements, *e.g.*, give resistor values in ohms, the value in volts of any voltage source, etc.

3. Consider the circuit below:



(a) Label the nodes and branches and assign orientations. Find the reduced node incidence matrix A.

(b) Write out the equations that describe this circuit. Give these equations in explicit form, *i.e.*, as a set of 12 equations in 12 unknowns. Also give the equations as one large matrix equation, with a 12×12 matrix. You can leave entries in the matrix that are zero as blank.

(c) Formulate the node voltage equations in the form $Ye = \tilde{i}$. Solve them.

(d) You know that if both of the current sources double in value, then all node voltages, branch voltages and branch currents will also double. Suppose that the three resistors in this circuit double in value. What happens?

4. *A generalization of power conservation.* Consider a circuit with b branches and n nodes, with associated reference directions assigned. As in the notes, v denotes the vector of branch voltages, i denotes the vector of branch currents, and e denotes the vector of node potentials (excluding the reference node). We saw in the notes that no matter what the branch elements are, we always have $v^T i = 0$, *i.e.*, the total net power dissipated in the circuit is always zero.

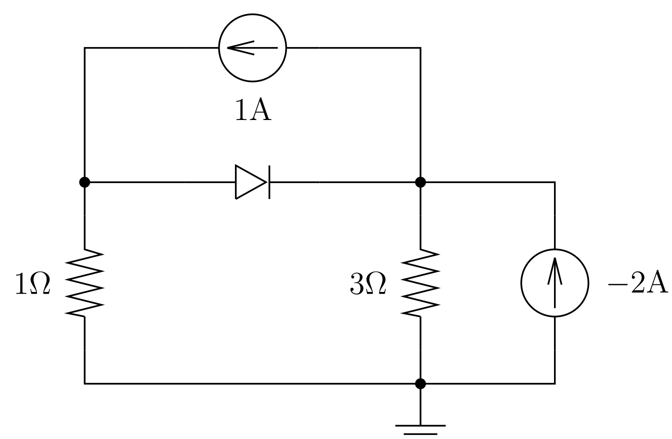
Now suppose we change one or more of the elements in this circuit, but we do not change the topology (*i.e.*, which branches are connected to which nodes, in what orientation). For example, we might substitute a diode for a voltage source, change the value of some resistors, and so on. We'll suppose that this modified circuit has a solution; let $\tilde{v}$ denote the branch voltages, $\tilde{i}$ denote the branch currents, and $\tilde{e}$ denote the node potentials in this modified circuit. We know that $\tilde{v}^T\tilde{i} = 0$, *i.e.*, the modified circuit also satisfies power conservation.

This modified circuit can be quite different from the original circuit, so there is no reason to think there is any relation at all between the circuit variables in the first circuit (v, i, and e) and the circuit variables in the second circuit ($\tilde{v}$, $\tilde{i}$, and $\tilde{e}$).

Remarkably, there is a relation, called *Tellegen's theorem.* It is: $\tilde{v}^T i = 0$. Note that $\tilde{v}^T i$ looks very much like $v^T i$ and also $\tilde{v}^T\tilde{i}$, both of which we know to be zero by power conservation. But Tellegen's theorem relates the voltages in one circuit with the currents in another!

(a) Prove Tellegen's theorem, *i.e.*, explain why it is true.

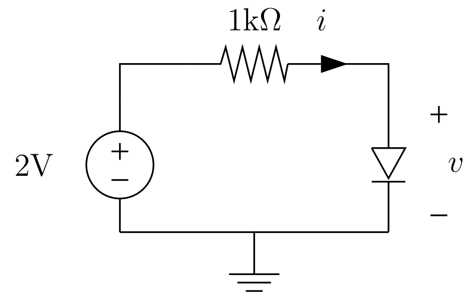
(b) First, find the vector i of branch currents in the circuit of problem 3. Now consider the circuit below, which is the same as the one problem 3 except that an ideal diode is substituted for the 2Ω resistor.



Find $\tilde{v}$, the vector of branch voltages for this modified circuit. Use the node and branch labels and reference directions that you used in problem 3.

Verify that Tellegen's theorem holds.

5. *Newton-Raphson procedure for a resistor-diode circuit.* Consider the nonlinear circuit shown below:



The diode is given by the exponential model with $i_0 = 10^{-14}$A and $v_T = 26$mV. We will try to find v and i using the Newton-Raphson method.

(a) We start with the initial guess $v^{(0)} = 0.7$V. Form the linearized model of the diode accurate near $v \approx 0.7$V, and show the circuit that results when you substitute this linearized model into the circuit above.

(b) Solve the resulting circuit, *i.e.*, calculate the voltage that appears across the linearized model of the diode. Call this voltage $v^{(1)}$ (the superscript stands for "v after one iteration").

(c) Do one more iteration of the Newton-Raphson method to find $v^{(2)}$.

(d) Do one more iteration of the Newton-Raphson method to find $v^{(3)}$.

(e) Now let's see what happens if we start with a very bad initial guess. Repeat parts (a-c) starting with the initial guess $v^{(0)} = -1.0$V. Note the values of current that you encounter. Can you guess what will happen if you keep going?

6. *A linear circuit solver.* Describe, in rough outline form, the code needed to make a version of SPICE that analyzes circuits that contain linear elements and sources. You may assume that you have a function (subroutine) that computes the solution x of the linear equation $Fx = g$. (Since this is a vague problem, answers ranging from one page of verbal description to complete C source will be accepted.)

7. Design a circuit with one op-amp and five resistors that produces output voltage $v_{\text{out}} = v_1 + 2v_2 - 3v_3$ at the output of the op-amp (with respect to the ground or reference node, to which the bottom output terminal of the op-amp is connected). v_1, v_2, and v_3 are the voltages of three voltage sources that have negative terminals grounded (*i.e.*, connected to the reference node). You may use the ideal op-amp model.

8. *Sensitivity of load power to variations in load resistance.* Suppose we have a load resistor R_{L} which we would like to dissipate p_{des} watts. (The load resistor might be a heating element in some experiment.) One simple way to do this is to connect it to a current source of $\sqrt{p_{\text{des}}/R_{\text{L}}}$ amps.

   We will consider an important practical consideration: variation in R_{L}. Suppose that R_{L} varies $\pm 10\%$, *i.e.*, $0.9R_{\text{nom}} \le R_{\text{L}} \le 1.1R_{\text{nom}}$, where R_{nom} is the *nominal value* of the load resistance. Such variation can arise in various ways. For example, when many load resistors are manufactured, the individual load resistors might have values that vary $\pm 10\%$ from the nominal value R_{nom}. As another example, the resistance of the load resistor might depend on some environmental variables such as ambient temperature that we cannot control.

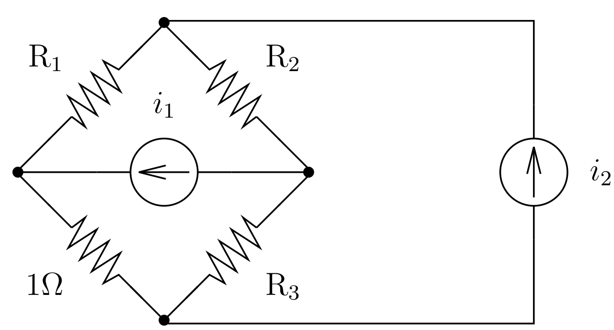
Now suppose we have a circuit consisting of linear elements, sources, and the load resistor R_{L}. Suppose that when $R_{\mathrm{L}} = R_{\mathrm{nom}}$, the desired power p_{des} is dissipated in R_{L}. Of course, the power dissipated in the load resistor will depend on the value of R_{L}; hence as R_{L} varies $\pm 10\%$, the power dissipated in it will also vary by some amount. As an example, consider the simple circuit described above, *i.e.*, a current source of value $\sqrt{p_{\mathrm{des}}/R_{\mathrm{nom}}}$ hooked up to the load resistor. In this circuit, as the load resistor varies $\pm 10\%$, the power dissipated in it also varies $\pm 10\%$ (which is not too surprising!).

We'd like to design a circuit such that the variation in power dissipated, as the load resistance varies $\pm 10\%$, is as small as possible. Thus, our circuit will have the nice property of minimizing the effect of $\pm 10\%$ variations in load resistance on the power dissipated in it.

The problem is: design such a linear circuit. Fully explain the reasoning behind your design and all steps of your calculations. You do not have to find the absolute best circuit; a circuit that comes very close will do. Many circuits work well, so try to find a simple one. For the circuit you design, give the percentage variation in the power dissipated in the load resistor as the load resistor varies $\pm 10\%$. Since I've already described a circuit in which this variation is $\pm 10\%$, your circuit should have a *smaller* varation. The values of elements in your circuit cannot depend on R_{L}, but can depend on R_{nom}.

Note: this is not an easy problem; you won't find the answer (directly) in the notes. For experts, we can pose a further problem: can you design a circuit containing *non-linear* elements that outperforms the circuit you designed above?

9. *Superposition for powers?* Consider the circuit shown below:



We want to find the power p that will be dissipated in the 1Ω resistor with $i_1 = 3$A and $i_2 = 1$A. Unfortunately we can only find two 1A current sources in the laboratory. The resistor values R_1, R_2, and R_3 are not known.

Three experiments are performed. In the first experiment, the source i_1 is turned on ($i_1 = 1\text{A}$), i_2 is turned off ($i_2 = 0\text{A}$), and the power dissipated in the 1Ω resistor is measured to be 0.25W. In the second experiment, i_1 is turned off, i_2 turned on, and the power dissipated in the 1Ω resistor is 0.25W. In the last experiment, i_1 is turned on, i_2 is also turned on but in reverse direction ($i_2 = -1\text{A}$) and the power dissipated in R_L is measured to be 0W. The following table summarizes these experiments:

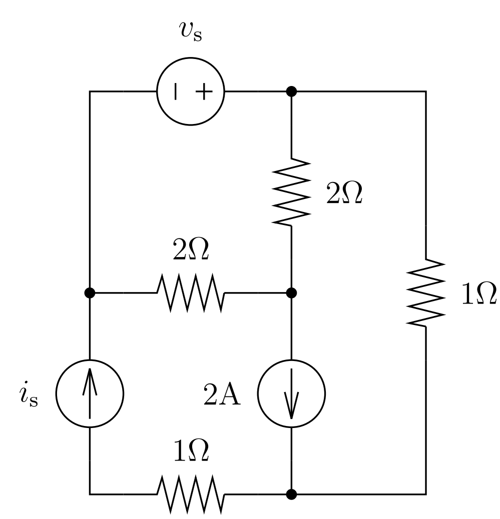
Experiment	i_1	i_2	Pwr. diss. in 1Ω res.
1	1	0	0.25
2	0	1	0.25
3	1	-1	0.00
—	3	1	p

(The bottom line of the table shows the experiment we'd perform if we could find a 3A current source.)

Can the power p be determined from these experiments? If your answer is yes, find the power p. If your answer is no, then answer this further question: if we performed a fourth experiment with $i_1 = i_2 = 1\text{A}$, could the power p be determined?

Note: Of course this problem is unlikely to arise in practice, but it does involve some important concepts so understanding it is worthwhile.

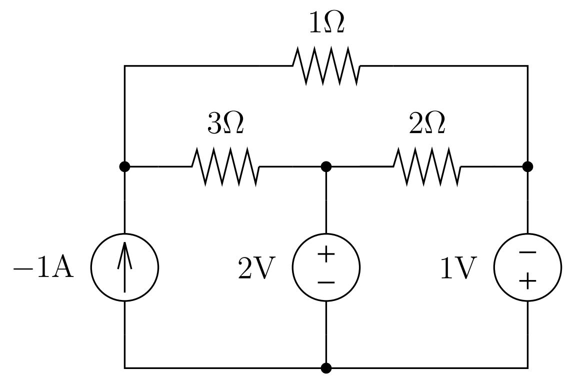
10. Consider the circuit below:



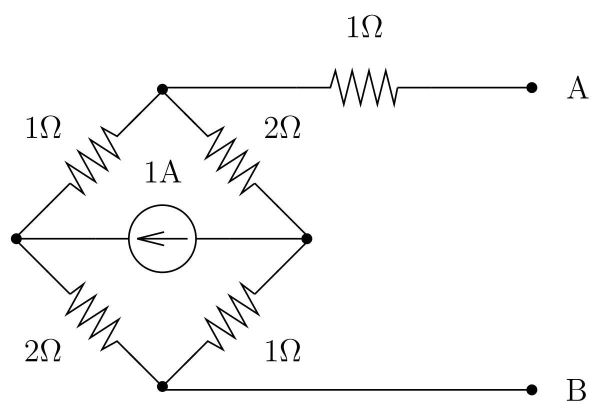
(a) Find an explicit expression for the power p *supplied by* the 2A current source. (i_s and v_s may appear in your answer.)

(b) Now suppose that you can adjust i_s over the range 0A to 1A, and v_s over the range 0V to 5V. Find the settings of i_s and v_s that *maximize* the power *supplied by* the 2A current source.

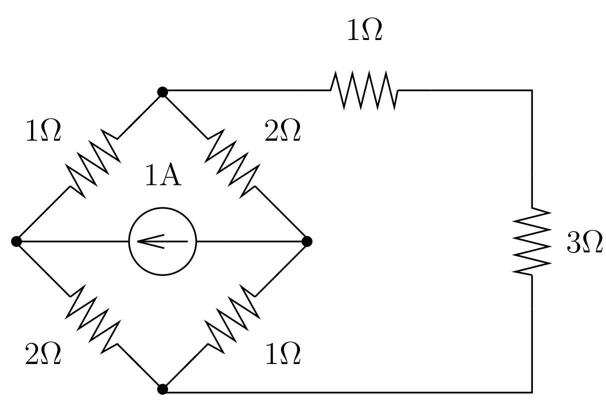
11. Find the power dissipated in the 3Ω resistor in the circuit below:



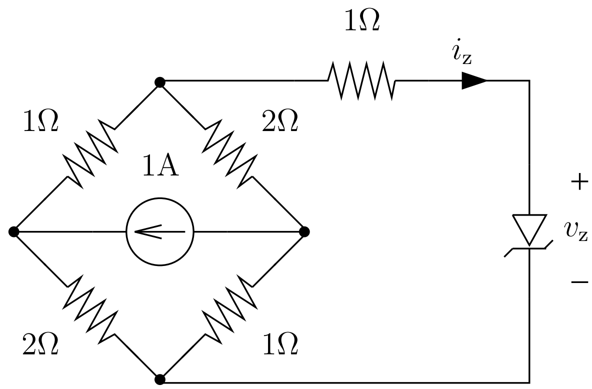
12. (a) Find the Thevenin equivalent of the circuit shown below.



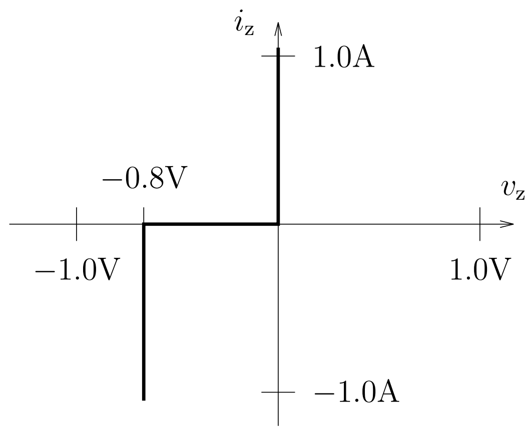
(b) Find the power p dissipated in the 3Ω resistor at the right in the circuit below:



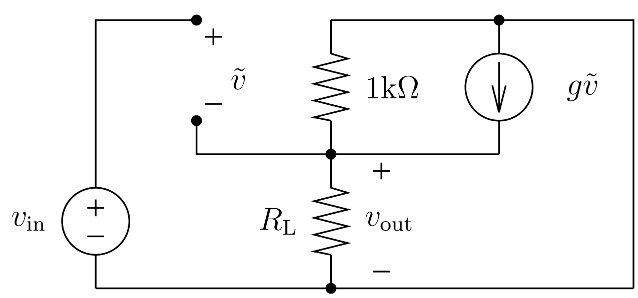
(c) Find the voltage v_z that appears across the zener diode in the circuit below.



The zener diode characteristic is shown below:

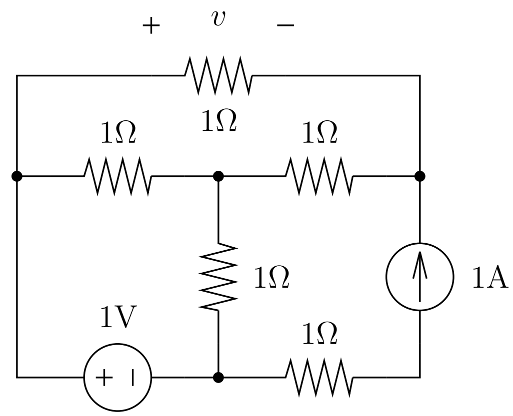


13. Consider the circuit below, which is called a *following amplifier*. The transconductance is given by $g = 10\text{mA/V}$.

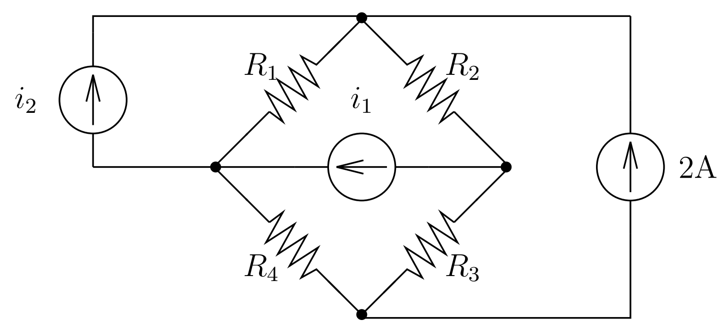


   (a) Find v_{out} when $R_L = 1\text{k}\Omega$.
   (b) Find the value of R_L that maximizes the power dissipated in it when $v_{in} = 3\text{V}$.

14. Find the voltage v shown in the circuit below:



15. The circuit shown below is designed to operate with $i_1 = 1\text{A}$ and $i_2 = 1\text{A}$. We need to determine the power p supplied by the 2A current source in this operating condition, but unfortunately we can find only one 1A current source in our laboratory. The resistor values R_1, R_2, R_3, and R_4 are not known.



Three experiments are performed. In the first experiment, the source i_1 is turned off ($i_1 = 0\text{A}$), i_2 is turned on ($i_2 = 1\text{A}$), and the power supplied by the 2A source, p_1, is measured. In the second experiment, i_1 is turned on, i_2 turned off, and the power supplied by the 2A source, p_2, is measured. In the last experiment, both i_1 and i_2 are turned off and the power supplied by the 2A source, p_3, is measured. The following table summarizes these experiments:

Experiment	i_1	i_2	Power supplied by 2A source
1	0	1	p_1
2	1	0	p_2
3	0	0	p_3
-	1	1	p

(The bottom line of the table shows the experiment we'd perform if we could find the other 1A current source.)

Which one of the following statements is true? Explain your choice.

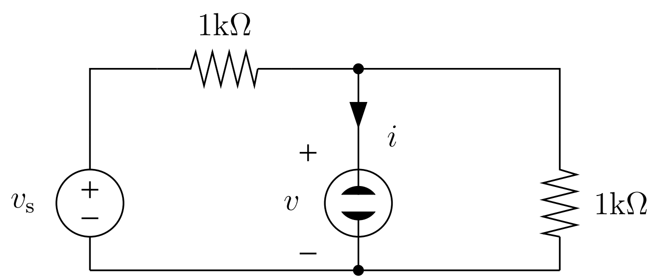
(a) $p = p_1 + p_2$

(b) $p = p_1 + p_2 + p_3$

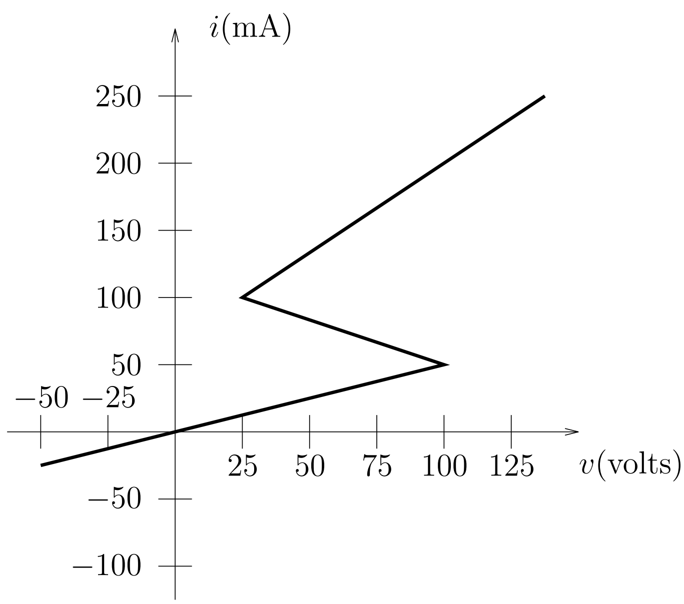
(c) $p = p_1 + p_2 - p_3$

(d) It is not possible to determine p from these experiments.

16. Consider the circuit below.



The v–i characteristic of the neon lamp is shown below.

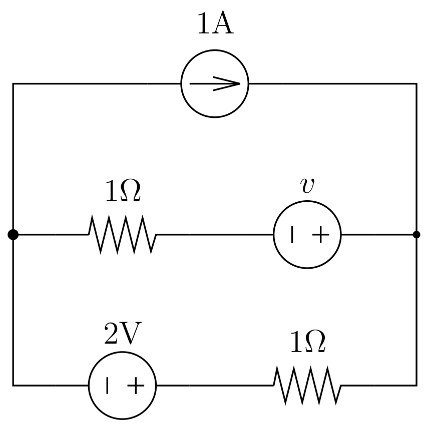


(a) For what values of v_s does this circuit have multiple solutions?

(b) Suppose that $v_s = 0$. In this case there is only one solution: every branch voltage and current is zero. In particular, $v = i = 0$. Suppose we use the Newton-Raphson (N-R) method to try to find this solution, using the initial guess $v^{(0)} = 100$, $i^{(0)} = 200\text{mA}$. Which one of the following statements is correct:

i. N-R will not converge because the initial guess and the solution are on different sides of the kinks in the v-i curve.

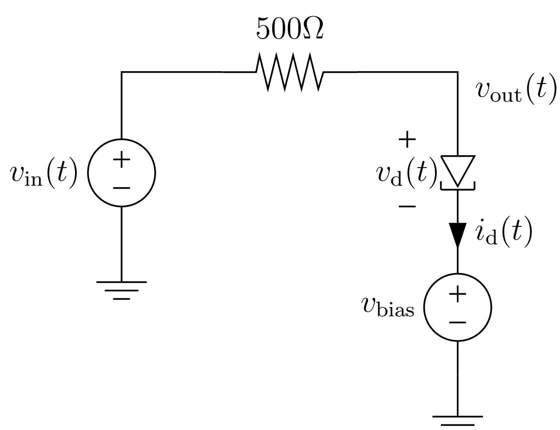
ii. N-R will converge to the solution in one iteration.
iii. N-R will converge to a wrong solution.
iv. N-R will converge to the solution very slowly (after many iterations).
v. None of the above.

17. 

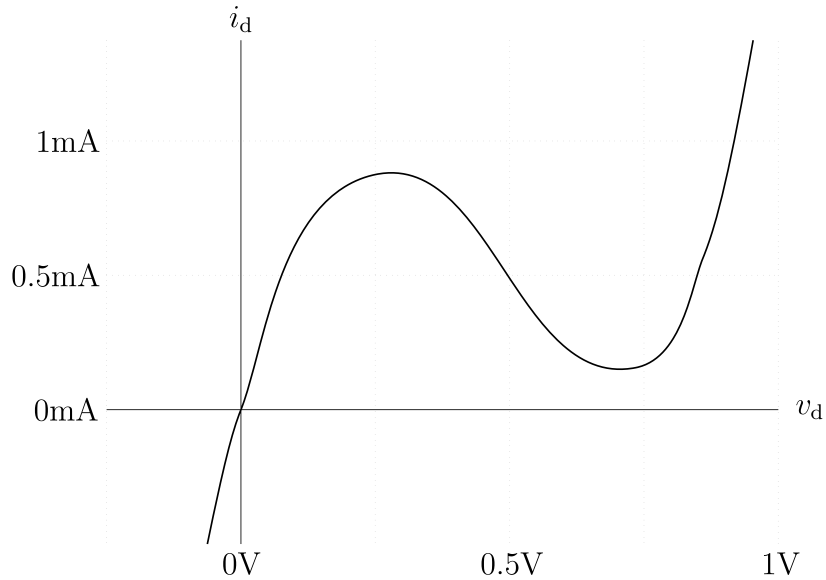
Find the value of v that maximizes the power dissipated in (*i.e.*, absorbed by) the voltage source v.

18. *Tunnel diode negative resistance amplifier.*

Consider the circuit shown below.



The voltage v_{out} is with respect to ground. The $v-i$ characteristic of the tunnel diode is shown below. In solving this problem you'll need to make estimates from this plot.



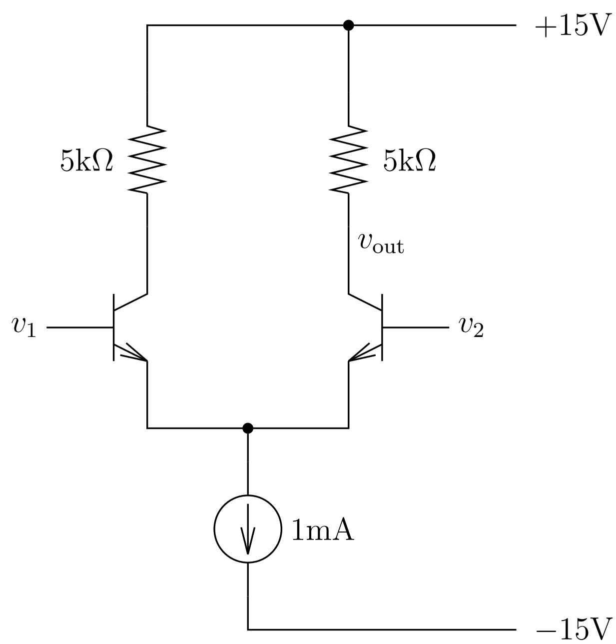
(a) *Bias calculation.* Suppose that $v_{\mathrm{in}}(t) = 0$. Find the value of v_{bias} so that the diode voltage v_{d} is 0.5V. We'll use this value of v_{bias} in the remainder of this problem.

(b) Find the linearized model of the tunnel diode at the bias point $v_d = 0.5\text{V}$. Express the linearized model as a circuit. Clearly label the two terminals of the circuit.

(c) Suppose that $v_{\mathrm{in}}(t) = 0.03\sin(400t)$. Give a (good) approximation of $v_{\mathrm{out}}(t)$.

19. *Differential input pair.*

The circuit shown below is widely used, for example, as the first stage of an op-amp.



All voltages shown are with respect to ground. The wire at top, which is connected to a +15V source called the *positive supply rail*; the wire at bottom is connected to a −15V source called the *negative supply rail.* The wires labeled v_1 and v_2 are the *input voltages.* They are connected to voltage sources with values v_1 and v_2. Note that current can flow into any of these wires; they are connected to things that are not shown in this schematic.

v_{out} (*i.e.*, the voltage at the point marked with respect to ground) is the *output voltage.*

You can assume that v_1 and v_2 are such that the following simple transistor model is accurate:

$$i_b = i_0 e^{v_{be}/v_t}, \quad i_c = \beta i_b$$

where $i_0 = 10^{-14}$A, $v_t = 26$mV, and $\beta = 100$. The currents and voltages are defined in the standard way: v_{be} is the voltage from base to emitter, i_b is current flowing into the base, and i_c is the current flowing into the collector.

Derive an expression for v_{out} as a function of v_1 and v_2. Simplify your expression as much as possible. Do not leave parameters like β in your answer: use the numerical values given.

20. *Duplex transmission with mismathed resistances.* Consider the duplex transmission circuit described in lecture 8. Our analysis was based on the assumption that the two 600Ω resistors are exactly matched. What if one of them, say the left one, is actually 500Ω instead of 600Ω? Analyze the circuit, expressing the two output voltages v_{out} and $\tilde{v}_{\text{out}}$ in terms of the two input voltages v_{in} and $\tilde{v}_{\text{in}}$. Do you think it would still work in practice, in a telephone circuit?

Exercises on Dynamic Circuits

1. *Energy analysis of the charge/fire circuit.* Consider the circuit on pages 11-17 and 11-18 of the notes. Assume that at $t = 0$ the capacitor is uncharged ($v_c(0) = 0$), and at $t = 1000$ the switch is thrown from "charge" to "fire." For the questions below, we want numerical answers.

   (a) Find the total energy supplied by the 100V source.

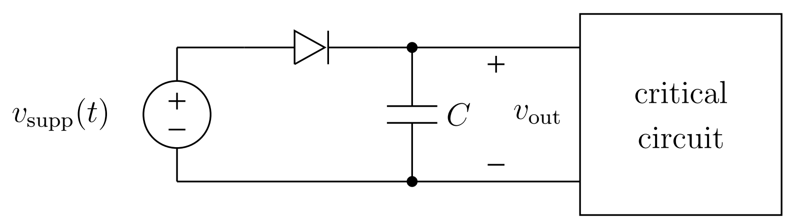
   (b) Find the total energy dissipated in the 100kΩ resistor.

   (c) Find the total energy stored in the capacitor at $t = 1000$.

   (d) Find the total energy dissipated in the 10Ω load resistor.

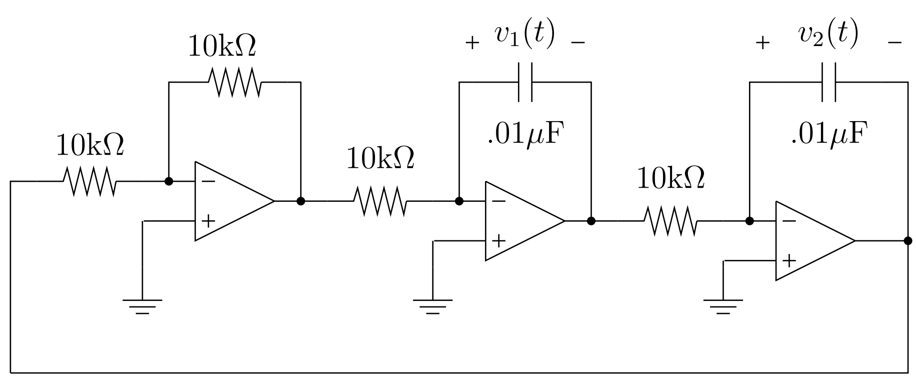
   (e) What is the earliest time the switch could be thrown to "fire" and still have a peak power of 300W dissipated in the load resistor?

2. *Coasting through power interruptions.* The circuit below is used to maintain some power supply voltage for brief periods during which the normal supply fails. In normal operation, $v_{\rm supp}(t) = v_{\rm out}(t) = 15$V. For periods up to 3 seconds long, with no warning, $v_{\rm supp}(t) = 0$ (but thereafter returns to 15V). The current drawn by the critical circuit varies between 0.3A and 1.5A. The critical circuit will work as long as $v_{\rm out} \geq 13V$. Find the smallest value of C that will allow the critical circuit to continue working during a 3 second long power supply failure. You can assume the diode is ideal, and that the time between power supply failures is very long.



3. What does the following circuit do? Assume $v_1(0) = 0$ and $v_2(0) = 1$.

   Hint: Show that $d^2v_1(t)/dt^2 + \omega^2 v_1(t) = 0$, where $\omega = 1/(10\text{k}\Omega \cdot 0.01\mu\text{F})$.



4. *A real problem involving inductors.* This problem concerns the magnet for a commercial magnetic resonance imaging (MRI) machine. The magnet creates a strong magnetic field (about 0.4 Tesla) over a fairly large volume—very roughly, about 1m by 1m by 0.5m. The magnet consists of about 100 turns of an aluminum conductor which is about 1/8in thick and 11in wide. The heat dissipated in the conductor is carried away by forced water cooling. The magnet and its steel housing weighs about 13 tons.

   A good electrical model of the magnet is an inductance of 15mH in series with the resistance of the aluminum conductor and wires leading to the power supply, 0.012Ω. The power supply that drives the magnet has a maximum output current of 2000A, and a maximum output voltage of 26V. The operating condition of the magnet is 1850A.

   Yes, some of these numbers are outside the range of typical electrical values that I told you we encounter. The wires hooking up the power supply to the magnet are gauge 0000 (twice as thick as your thumb), with about 10 in parallel to carry the enormous current. The AC input power to the power supply is about 69kW.

   Note (for cultural enrichment only): Many other MRI magnets are superconducting, so the resistance is essentially zero. After the initial current is established, the magnet terminals are shorted together, forming a superconducting loop. The power supply is then disconnected. Provided the loop remains superconducting, the magnet current will flow indefinitely.

   (a) What is the total energy in Joules stored in the magnetic field at operating condition?

   (b) How long does it take for this amount of energy to be dissipated as heat? The ratio of stored energy to power dissipation gives a rough idea of a turn-over time, relating the energy stored to the rate of energy turn-over.

   (c) What is the voltage of the power supply at operating condition? What is the power being supplied to the magnet?

   (d) Suppose that the power supply puts out its maximum voltage, and the inductor current is initially zero. How long does it take before the inductor current reaches the operating condition of 1850A? How much longer does it take before the power supply maximum current (2000A) is reached?

   (e) Do you think it's safe or wise to suddenly reduce the power supply voltage from its maximum, 26V, to the operating voltage found in part (c), when the magnet current first hits 1850A?

   (f) What would happen if the bolts clamping the large wires to the aluminum magnet conductor became loose, causing a bad connection between the power supply and the magnet?

5. *Truncation stability of simulation methods.* Consider a simple LC circuit with $L = 1\text{H}$ and $C = 1\text{F}$, with zero initial current and 1V across the capacitor at $t = 0$. $v(t)$ will

denote the voltage across the capacitor and $i(t)$ the current flowing out of the capacitor (*i.e.*, v and i are *not* associated references).

Of course you know the exact solution of this circuit. We'll study the simulation method described in the notes.

(a) The voltage and current don't change too much in about 1msec, so it seems reasonable to use a time step of 1msec. Show that the simulation method reduces to the following recursion:

$$\begin{aligned}\hat{v}(0.001(k+1)) &= \hat{v}((0.001k) - 0.001\hat{\imath}(0.001k),\\ \hat{\imath}(0.001(k+1)) &= \hat{\imath}((0.001k) + 0.001\hat{v}(0.001k),\end{aligned}$$

with initialization

$$\hat{\imath}(0)) = 0, \qquad \hat{v}(0) = 1.$$

We use the hat on the symbols $\hat{v}$ and $\hat{\imath}$ to emphasize the fact that these are *approximations* to the true voltage v and current i, respectively.

(b) *We* know that the total energy in this circuit is 0.5J and does not change with time (p12-4), but our *simulation algorithm* doesn't know this. So one check on the accuracy of the simulation is to keep track of the total energy in the circuit according to the simulation; hopefully this number should remain close to 0.5J.

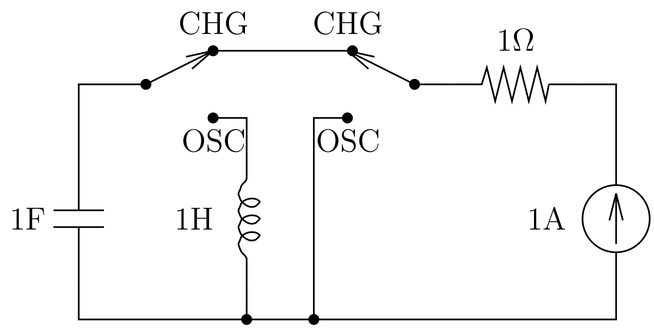
Find a recursion for $\hat{E}(kh)$, the energy in the circuit based on the approximate voltage and current from the simulation, *i.e.*,

$$\hat{E}(kh) = \frac{\hat{v}(kh)^2 + \hat{\imath}(kh)^2}{2}.$$

Then give an explicit formula for $\hat{E}(kh)$. Is it constant? Does it remain near 0.5J?

(c) One student pointed out a potential flaw in the philosophy behind the simulation method: each step is an approximation, so future steps are based on approximations of approximations of approximations ... and there is no reason to believe that the error doesn't build up. The technical term for the build-up of error is *truncation instability.* Does this problem provide an example of truncation instability?

6. In the circuit below, the capacitor voltage and the inductor current are zero at $t = 0$. The two switches are in the "charge" (CHG) position from $t = 0$ until $t = T_{\text{sw}}$ (as shown in the schematic) and are in the "oscillate" (OSC) position for $t > T_{\text{sw}}$. Thus, T_{sw} denotes the time at which the switches are thrown from CHG to OSC. For $t > T_{\text{sw}}$ the LC circuit simply oscillates, with constant total energy E.

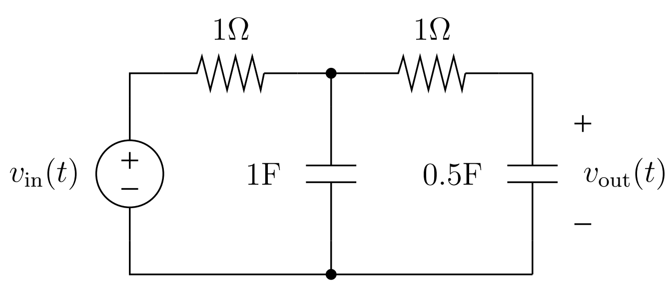


(a) Find T_{sw} such that $E = 8\text{J}$.

(b) Assume that the switch is thrown at the time T_{sw} found in part (a). What is the earliest (*i.e.*, smallest) time T_{ind} at which 8J is stored in the inductor?

7. *An RCRC circuit.*

Consider the circuit shown below. For $t < 0$, $v_{\text{in}}(t) = 1\text{V}$ and the circuit was in static conditions. For $t \geq 0$, $v_{\text{in}}(t) = 0\text{V}$.

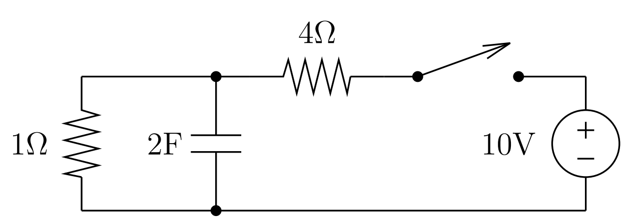


(a) Find v_{out} at $t = 0$, immediately after v_{in} has switched to 0V.

(b) Find $\dfrac{dv_{\text{out}}}{dt}$ at $t = 0$, immediately after v_{in} has switched to 0V.

(c) Find v_{out} at $t = 1$.

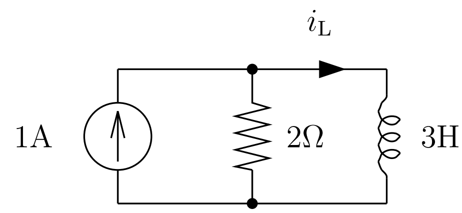
8. In the circuit below, the switch is closed for $t < 1\text{sec}$ and open for $t \geq 1\text{sec}$. The voltage across the capacitor is zero at $t = 0$.



(a) Find the maximum energy stored in the capacitor over the time interval $t \geq 0$.

(b) Find the maximum power dissipated in the 4Ω resistor over the time interval $t \geq 0$.

9. Consider the circuit shown below, in which $i_{\rm L}(0) = -2$A.



(a) Write out the equations that describe this circuit.

(b) Find $i_{\rm L}(t)$.

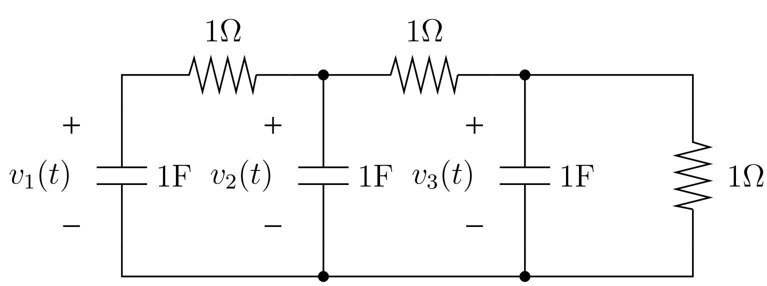
(c) Suppose the 1A current source were replaced with a 0A current source. Find $i_{\rm L}(t)$ in this case.

(d) Suppose that the initial inductor current were 0A instead of -2A (the current source is still 1A, however). Find $i_{\rm L}(t)$ in this case.

(e) Verify that the solutions found in parts (c) and (d) add up to the solution found in part (b).

(f) Use the simulation method described in the notes, with a time step-size of $h = 0.01$sec, to find (approximations of) $i_{\rm L}(t)$ for $t = 0, 0.01, 0.02, \ldots$ Compare the exact and (approximate) simulation values of the inductor current at $t = 1$sec.

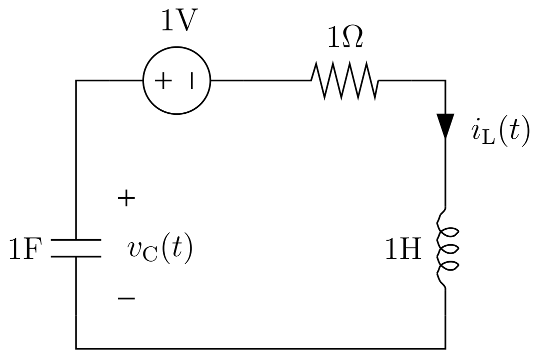
10. In the circuit below, $v_1(0) = 1$, $v_2(0) = -1$, and $v_3(0) = 1$.



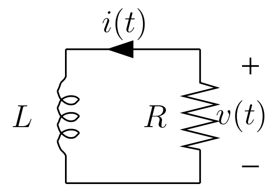
(a) Estimate $v_1(0.01)$, $v_2(0.01)$, and $v_3(0.01)$. **Note:** your estimate should be more accurate than $v_1(0.01) \approx 1$, $v_2(0.01) \approx -1$, $v_3(0.01) \approx 1$.

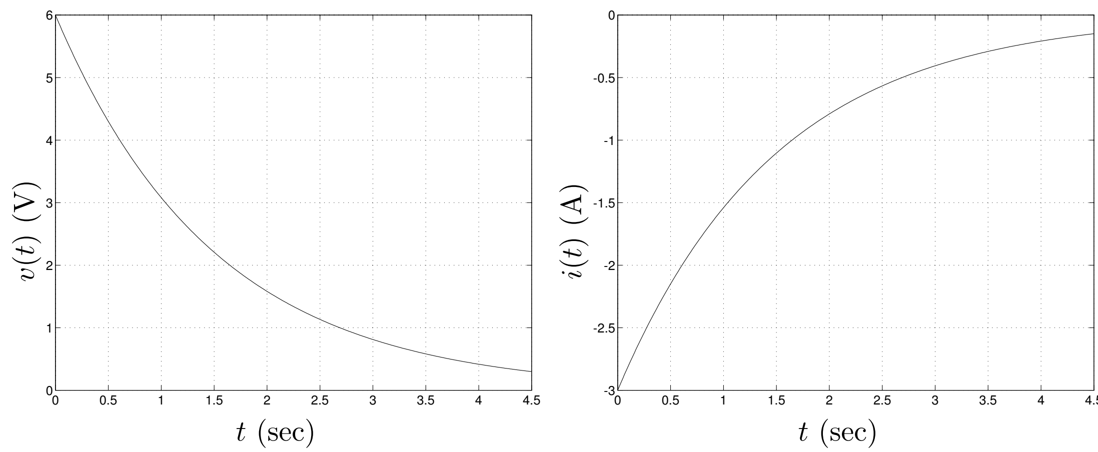
(b) Estimate the decrease in total stored energy over the time interval $[0, 0.01]$, *i.e.*, quantity $E(0) - E(0.01)$, where $E(t)$ denotes the total energy stored in the circuit at time t. **Note:** your estimate should be more accurate than $E(0) - E(0.01) \approx 0$.

11. In the circuit below, $v_{\rm C}(0) = 1$V and $i_{\rm L}(0) = 1$A. Estimate $v_{\rm C}(0.01\text{sec})$ and $i_{\rm L}(0.01\text{sec})$.

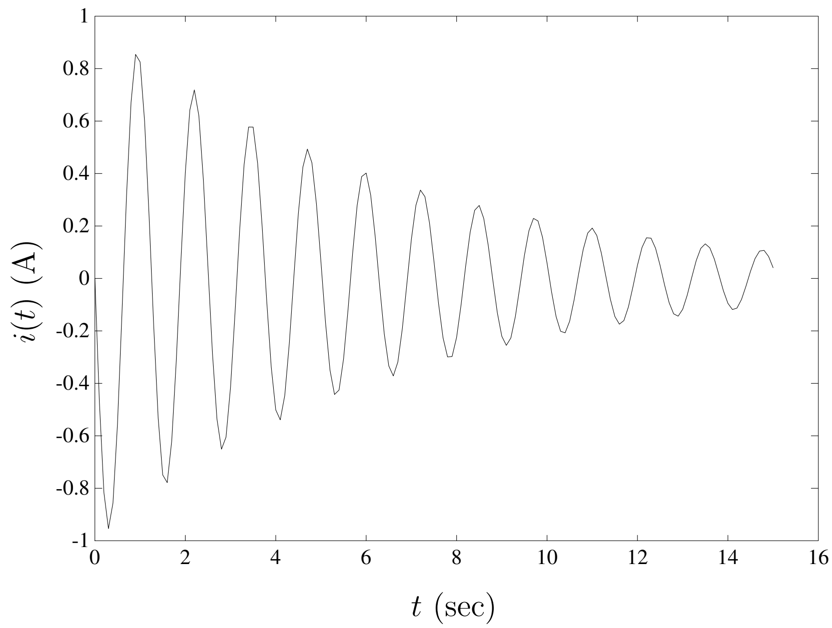


12. The voltage and current in the circuit at right are shown in the plot below. Estimate both the inductance L and the resistance R. Make sure to give the units for your answers (e.g., ohms or kilohms).





13. The waveform shown below is the current in a series RLC circuit. The value of the resistor is 100Ω.



(a) Estimate L and C.

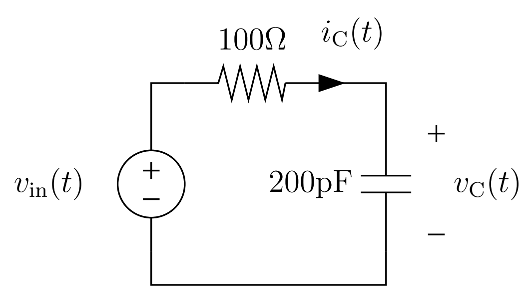
(b) About how long will it be before 99% of the initial stored energy in the circuit has dissipated?

14. In the circuit shown below, v_{in} switches from 5 to 0 volts at $t = 0$:

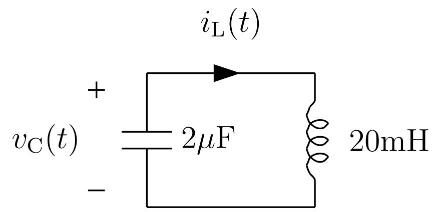
$$v_{\text{in}}(t) = \begin{cases} 5 & t < 0 \\ 0 & t \geq 0 \end{cases}$$

You may assume that prior to $t = 0$, the circuit had reached its steady-state condition, that is, $i_{\text{C}}(t)$ is zero for $t < 0$.

Find the smallest time T_{d} for which $v_{\text{C}}(T_{\text{d}}) = 3.5$. (The subscript "d" stands for "delay".)



15. This problem concerns the circuit shown below. Let $E_{\text{C}}(t)$ denote the energy (in Joules) stored in the capacitor at time t and let $E_{\text{L}}(t)$ denote the energy (in Joules) stored in the inductor at time t. The initial conditions are $v_{\text{C}}(0) = 4\text{V}$ and $i_{\text{L}}(0) = 30\text{mA}$.



(a) Find $E_L(0)$.

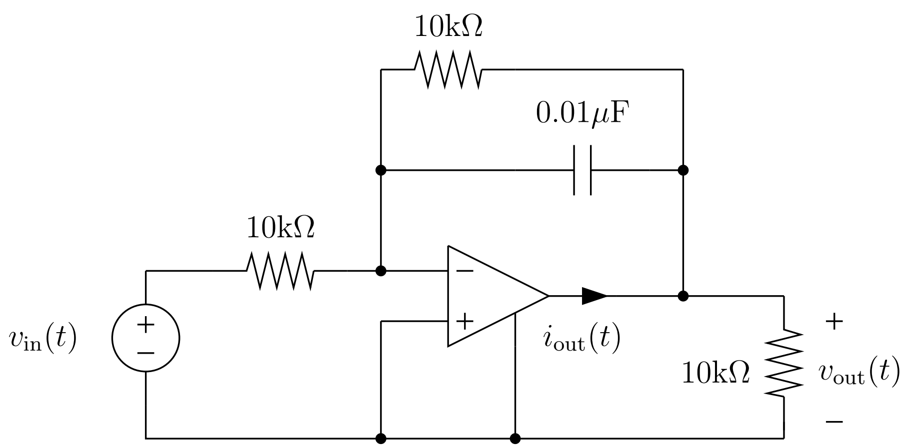
(b) Is $i_L(t)$ increasing or decreasing at $t = 0$?

(c) Find $\frac{dE_C}{dt}(0)$.

(d) Find the maximum magnitude of the inductor current, that is, find $\max_t |i_L(t)|$.

16. This problem concerns the circuit shown below. The capacitor is initially uncharged (that is, has no voltage across it at $t = 0$). The op-amp is described by the ideal op-amp model. The voltage source is given by:

$$v_{\rm in}(t) = \begin{cases} 0 & t < 0 \\ 5 & t \geq 0 \end{cases}$$

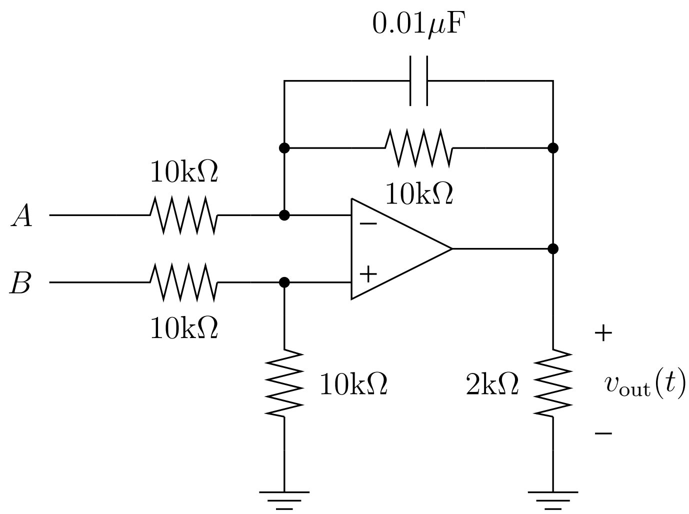


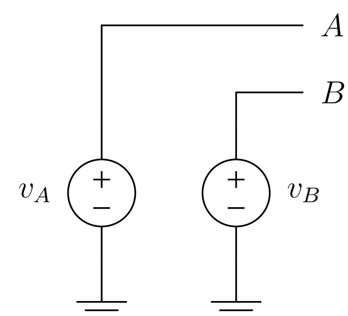
(a) Find an explicit expression for $v_{\rm out}(t)$ valid for $t > 0$.

(b) Find the maximum magnitude of the op-amp output current, that is, $\max_t |i_{\rm out}(t)|$. (As you know, real op-amps have limits on this quantity. Knowing this quantity can help us determine the type of op-amp necessary for the application.)

17. *Differential amplifier.*

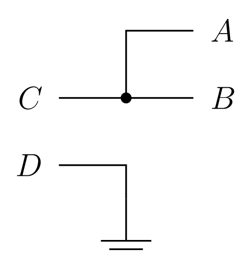
The circuit below shows a differential amplifier. You may use the ideal static op-amp model in your analysis. In the following questions, the input terminals A and B are connected in various ways to the circuit.





(a) Suppose that A and B are connected to grounded voltage sources v_A and v_B, respectively, as shown at right. Find an expression for v_{out}, assuming static conditions.

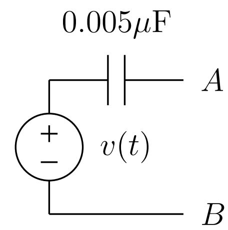
(b) We still assume static conditions. The resistance seen looking into the terminals A and B in the differential amplifier circuit is denoted R_{diff} and called the *differential input resistance.* (Perhaps more precisely, R_{diff} is the Thevenin resistance of the differential amplifer circuit for the terminals A and B.) Find R_{diff}.



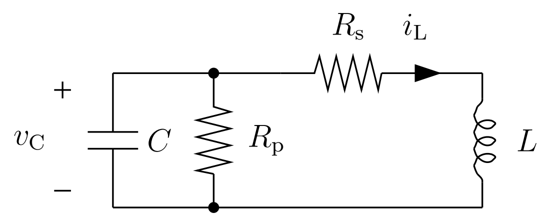
(c) We still assume static conditions. The resistance seen between ground and terminals A and B, tied together, is denoted R_{cm} and called the *common-mode input resistance.* (More precisely, R_{cm} is the Thevenin resistance for the terminals C and D shown at right, with A and B connected to the differential amplifier.) Find R_{cm}.

(d) We no longer assume static conditions. Suppose that A and B are connected to ground and $v_{\text{out}}(0) = 2\text{V}$. Find $v_{\text{out}}(t)$.

(e) Now assume that the circuit is operating in sinusoidal steady-state and the terminals A and B are connected to the circuit shown at right, where $v(t) = 2\cos(10^4 t)$. Find the voltage $v_{\text{out}}(t)$. Express your answer in the form $v_{\text{out}}(t) = v_m \cos(10^4 t + \phi)$.



18. *The RLRC circuit.* In the series RLC circuit, current flow causes power to be dissipated in the resistor as heat. In the parallel RLC circuit, voltage causes power to be dissipated. In the RLRC circuit shown below, power is dissipated by both mechanisms.



(a) Find a differential equation of the form

$$\frac{d^2 v_c}{dt^2} + b\frac{dv_c}{dt} + cv_c = 0$$

that describes this circuit.

(b) Find an expression for the rate of change of the total stored energy, *i.e.*,

$$\frac{d}{dt}\left(\frac{Li_L(t)^2}{2} + \frac{Cv_C(t)^2}{2}\right),$$

in terms of $i_L(t)$ and $v_C(t)$. Give one sentence interpreting your result.

19. *Transition from overdamped to critically damped to underdamped.*

Consider three series RLC circuits with the same inductance and capacitance, $L = 1$H, $C = 1$F, and the same initial conditions: 1V across the capacitor and zero current in the circuit. The resistors in the three circuits differ slightly: in the first circuit we have $R = 1.99\Omega$; in the second circuit we have $R = 2\Omega$, and in the third circuit we have $R = 2.01\Omega$.

You know from lecture 12 that the *formulas* for the solution $v(t)$ (the voltage across the capacitor) of these three circuits are quite different: In the first case, $v(t)$ is an exponentially decaying sinusoid; in the second, it is sum of an exponential and a strange term involving t times an exponential; in the last case, $v(t)$ is a sum of two decaying exponentials.

One student says:

> The voltage response is quite different in these three cases: in the first case the voltage crosses the value zero infinitely often; in the second and third cases, just once or maybe twice. So the solutions of these three circuits are indeed very different, even though the three resistor values are so close. The reason is that the value $R = 2\Omega$ is a "critical value" for this circuit, as seen in the formulas for lecture 12. It's not surprising that the solution of a circuit changes drastically as the resistance varies near a "critical value".

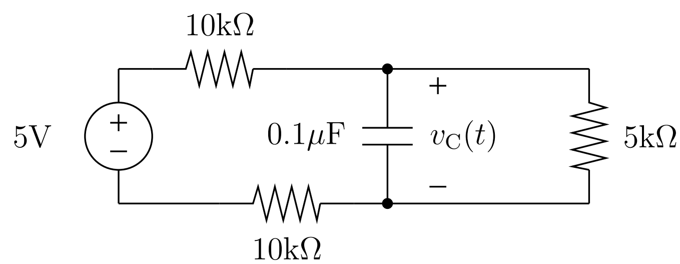
A second student then responds:

> Something is fishy here. I don't see how such a miniscule change in the resistor value can have such a great effect on the voltage across the capacitor. It just doesn't make physical sense to me.

Who is right? Discuss.

(At the very least, you should spend some time thinking about this problem, and maybe discuss what you'd do to resolve it. Feel free to give plots, examples, or mathematical proofs that support your discussion.)

20. In the circuit below, $v_C(0) = 0$.



Find an explicit expression for $v_C(t)$.

21. *Dynamic model of op-amp.* The simplest model of a real op-amp is the ideal op-amp model. The ideal op-amp model makes hand calculations and circuit analysis easy, and often gives good predictions of what will happen in a circuit with a real op-amp. But you already know that in some cases, the ideal op-amp model makes wrong predictions about real circuits containing op-amps. For example, the ideal op-amp model makes no distinction between the + and − input terminals: according to the ideal op-amp model you can swap them without affecting the behavior of the circuit! In real circuits containing op-amps, you can *never* swap the + and − input terminals (*i.e.*, swapping the + and − terminals will always have a drastic effect on the operation of the circuit). As a specific example we considered two inverting amplifier circuits labeled circuit A and circuit B, which differ only in the + and − input terminals being swapped. We told you then that circuit A works (with a real op-amp), *i.e.*, results in $v_{\text{out}}(t) \approx -10V_{\text{in}}(t)$, but circuit B doesn't work with a real op-amp. In that problem you discovered that a more complicated static model of a real op-amp (as a VCVS) still doesn't predict

that circuit B won't work. In this problem we give a simple *dynamic* model of a real op-amp, and once again analyze the two circuits.

A reasonable dynamic model of a real op-amp is that the current flowing into the + and − terminals are both zero, and

$$v_{\text{out}} + T\frac{dv_{\text{out}}}{dt} = g\tilde{v}$$

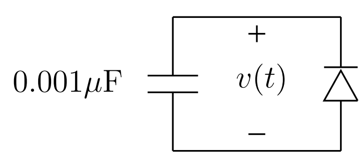
where g is the gain of the op-amp and T is its time constant. Typical values are $g = 10^5$ and $T = 1\text{sec}$. Note that under static conditions (*i.e.*, currents and voltages constant) this model reduced to a VCVS with gain g.

Assume that $v_{\text{in}} = 1$ for all t, and that $v_{\text{out}}(0) = 0$ in circuits A and B of problem 9. (Note that since our op-amp model is given by a differential equation, we have to specify an initial condition for it!) Find $v_{\text{out}}(t)$ for each circuit, as predicted by this dynamic model. Use the typical values for g and T mentioned above. Does this dynamic model of an op-amp predict that something is fishy with circuit B?

Note: we say that circuit A is *stable* whereas circuit B is *unstable.* We'll see alot more about this in EE102.

22. *A simple nonlinear dynamic circuit.* In class we studied several simple circuits containing capacitors, inductors, and resistors. For these simple circuits we found analytic ("closed-form") solutions for the voltages and currents as a function of time. In this problem you study a simple dynamic circuit with a *nonlinear* element—an exponential diode.

    The diode in the circuit below is characterized by the exponential diode model with $I_s = 10^{-12}\text{A}$ and $v_{\text{t}} = 26\text{mV}$. We have $v(0) = 1\text{V}$.

    (a) Give an intuitive, qualitative analysis of what happens. Try to avoid using any equations, and especially differential equations, in your discussion. I will begin the discussion for you: "Initially, the diode is reverse-biased, so a current of about 10^{-12}A flows out of the capacitor. This leaks charge away from the capacitor, so the voltage across it initially decreases at a rate of $10^{-12}\text{A}/0.001\mu\text{F} = 10\text{V/sec}$ ... You finish.

        Compare what happens in this circuit to what happens in an RC circuit.

    (b) Find a differential equation that v obeys.

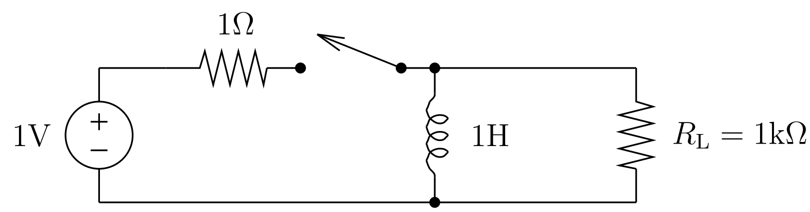
    (c) Solve it. Compare the solution to your qualitative analysis in part (a). *Hint for solving it:* first separate variables to write the differential equation in the form

$g(v)dv = dt$ where g is some appropriate function. Integrate to get $f(v(t)) - f(v(0)) = t$ where f is the integral of g, *i.e.*, $f' = g$. Now solve the resulting equation for $v(t)$. You may find it useful to recall that for $x > 0$ the derivative of $\log(1 - e^{-x})$ is $1/(e^x - 1)$.

Remark: Congratulations, you've just solved one of about three nonlinear dynamic circuits (in the whole world) that has an analytic solution. And it was not a pretty sight. Recall that most nonlinear static circuits do not have analytic solutions; well, even fewer nonlinear dynamic circuits have analytic solutions! In practice, nonlinear dynamic circuits (such as this one) are "solved" by computer, and never analytically. We'll see later how this is done.

23. *The spark coil circuit.* In the lectures we studied a charge/fire circuit which is based on the capacitor's ability to store energy over a long period and then release the stored energy in a short time interval. We noted that the output current could be very large compared to the charging current. For example, using a small battery that can put out, say, 100mA, we can charge a capacitor and then, for a brief period, put out a current of many amperes!

In this problem we explore a similar circuit, shown below, that uses an inductor instead of a capacitor to store and then release energy.



The inductor current is zero at $t = 0$. The switch (which is shown open) is closed from $t = 0$ until $t = 10$, at which point it opens (and stays open).

(a) Find $v_L(t)$, the voltage across R_L. Of course you will have two different expressions, one for $0 \le t \le 10$, and one for $t > 10$. Are you surprised by your answer?

(b) Find the peak (*i.e.*, maximum) power supplied by the battery. When does this occur?

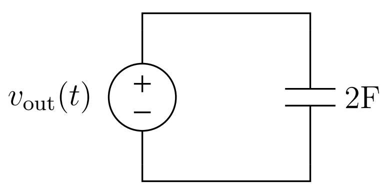
(c) Find the peak power dissipated in R_L. When does this occur?

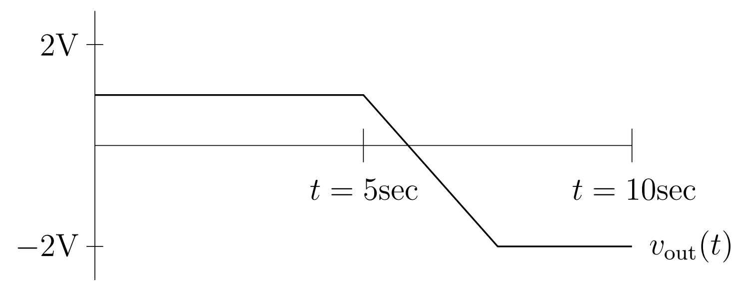
(d) Find the peak energy stored in the inductor. When does this occur?

You may use reasonable approximations in this problem, but state what they are as you use them.

Remark: this is (roughly) how a 12V car battery can generate a 20kV pulse required for the spark plugs. There is a practical lesson in this problem: generally, you would feel safe poking your fingers around in a circuit powered by a 1.5V battery; but if there are inductors in the circuit you could get a big surprise.

24. An amplifier (modeled as the voltage source shown at left) drives a 2F capacitive load. The plot at right shows the amplifier output voltage as a function of time t, for $0 \le t \le 10$. Let $p(t)$ denote the power *supplied by* the amplifier.





(a) Find the **maximum power** supplied by the amplifier over the time interval shown, *i.e.*,

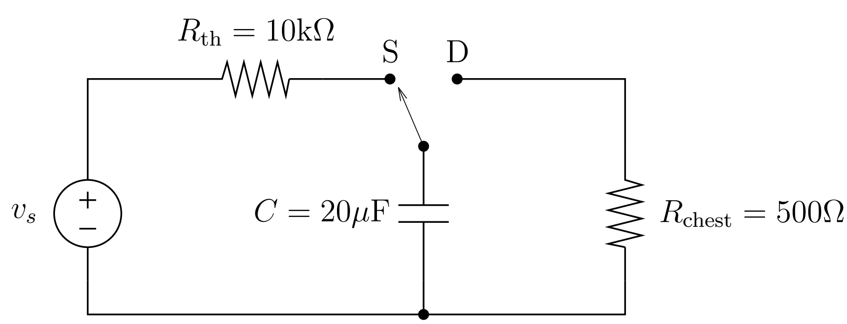
$$p_{\max} = \max_{0 \le t \le 10} p(t).$$

(b) Find the **average power** supplied by the amplifier over the time interval shown, *i.e.*,

$$p_{\mathrm{avg}} = \frac{1}{10} \int_0^{10} p(t)\, dt.$$

25. *Defibrillators.*

A defibrillator is used to deliver a strong shock across the chest of a person in cardiac arrest or fibrillation. The shock contracts all the heart muscle, whereupon the normal beating can (hopefully) start again. The first defibrillators used the simple circuit shown below.



With the switch in the standby mode, indicated as 'S', the 20μF capacitor is charged up by a power supply represented by a Thevenin voltage v_s and Thevenin resistance $R_{\mathrm{th}} = 10\mathrm{k}\Omega$. When the switch is thrown to 'D' (for 'defibrillate'), the capacitor discharges across the patient's chest, which we represent (pretty roughly) as a resistance of 500Ω. (The connections are made by two 'paddles' pushed against the sides of the chest.)

On most defibrillators you can select the 'dose,' *i.e.*, total energy of the shock, which is usually between 100J and 400J.

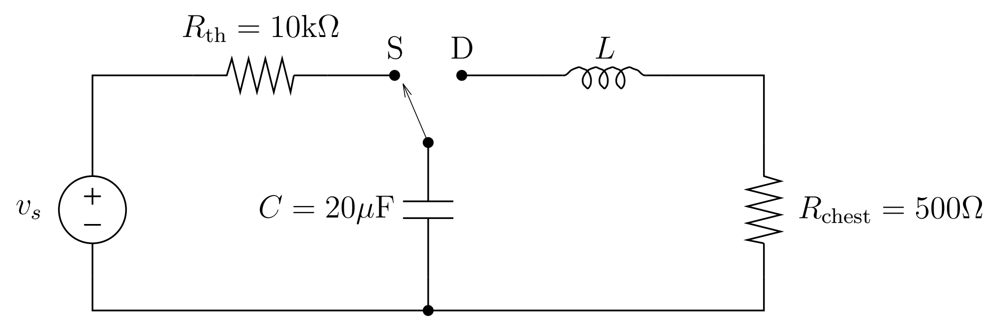
(a) Find v_s so that the dose is 100J. You can assume the capacitor is fully charged when the switch is thrown to 'D'. We'll use this value of v_s in parts 1b, 1c, and 1d.

(b) How long after the switch is thrown to 'D' does it take for the defibrillator to deliver 90% of its total dose, *i.e.*, 90J?

(c) What is the maximum power p_{max} dissipated in the patient's chest during defibrillation?

(d) Our model of the chest as a resistance of 500Ω is pretty crude. In fact the resistance varies considerably, depending on, *e.g.*, skin thickness. Suppose that the chest resistance is 1000Ω instead of 500Ω. What is the total energy E dissipated in the patient during defibrillation?

26. *An improved defibrillator.* One problem with the defibrillator described in problem 1 is that the maximum power p_{max} (which you found in part 1c) is large enough to sometimes cause tissue damage. An electrical engineer suggested the modified defibrillator circuit shown below. The inductor is meant to 'smooth out' the current through the chest during defibrillation, and yield a lower value of p_{max} for a given dose.



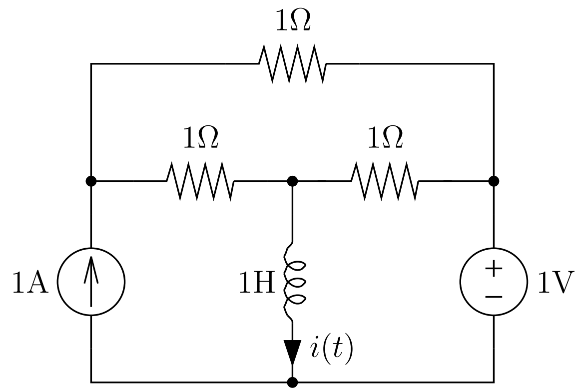
(a) Find the value of L that yields critical damping. We'll use this value of L in parts 2b and 2c.

(b) Find v_s so that the dose is 100J. You can assume the capacitor is fully charged when the switch is thrown to 'D'.

(c) Suppose v_s is equal to the value found in part 2b. What is the maximum power p_{max} dissipated in the patient's chest during defibrillation?

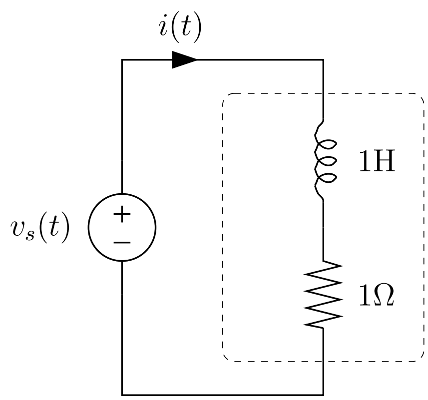
27. The current i through the inductor is zero at $t = 0$. Find $i(t)$ for $t \geq 0$.

Remember to show us what you are doing.



28. *Shutting down an electromagnet.*

An electromagnet is modeled as an inductance of 1H in series with a resistance of 1Ω. The electromagnet is driven by a programmable power supply (voltage source) which is limited to ±5V. The circuit is shown below, with the dashed box showing the electromagnet.



For $t < 0$, the electromagnet is in static steady-state with $i(t) = 2$A. The goal is to turn the electromagnet off, *i.e.*, reduce $i(t)$ to 0, as rapidly as possible after $t = 0$. This is done using a voltage supply waveform of the form

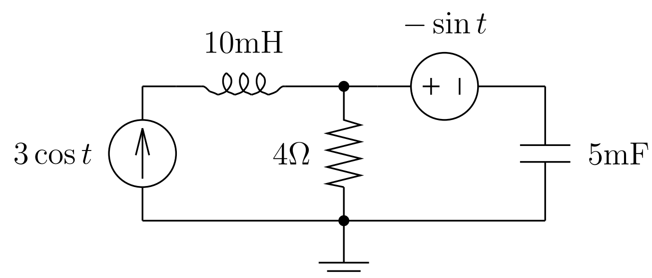
$$v_s(t) = \begin{cases} \alpha & 0 \le t < T \\ 0 & t \ge T \end{cases}$$

where $-5 \le \alpha \le 5$ and $T > 0$ are constants.

Find the values of α and T that result in $i(t)$ being reduced to zero as rapidly as possible after $t = 0$.

Exercises on Sinusoidal Steady States

1. *Circuit equations for a SSS circuit.* Write down a set of equations that completely describes the circuit below, which is operating in sinusoidal steady-state. You must label the nodes and branches, find the reduced node incidence matrix A, and find complex matrices $\mathbf{M}$, $\mathbf{N}$, and a complex vector $\mathbf{s}$ such that the branch equations are given by $M\mathbf{I} + N\mathbf{V} = \mathbf{s}$.



Use Matlab to solve this set of (complex, linear) equations.

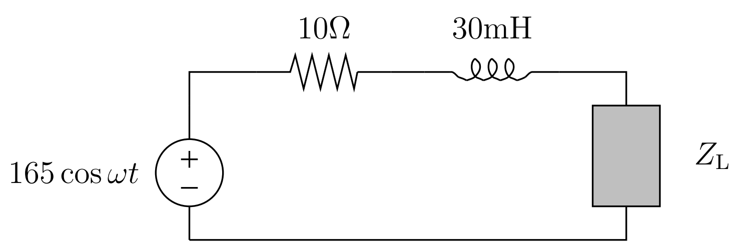
2. *Sinusoidal steady-state version of problem 16.* Consider the circuit given in problem 16 of *Problems on dynamic circuits*, with $v_{\text{in}}(t) = a\cos(\omega t)$. You may assume the circuit is operating in sinusoidal steady-state.

   (a) Find an explicit expression for $v_{\text{out}}(t)$ in terms of a and ω.

   (b) Find the value of ω for which the amplitude of the output voltage is half the amplitude of the input voltage.

   (c) If the circuit is driven at $\omega = 1$, what is the maximum op-amp output current, *i.e.*, $\max_t |i_{\text{out}}(t)|$? (Still assuming sinusoidal steady-state.)

3. *Maximum power transfer at two frequencies.* This problem concerns the circuit below:



   (a) Suppose that the voltage source has a frequency of 50Hz. What value of Z_{L} maximizes the average power dissipated in it?

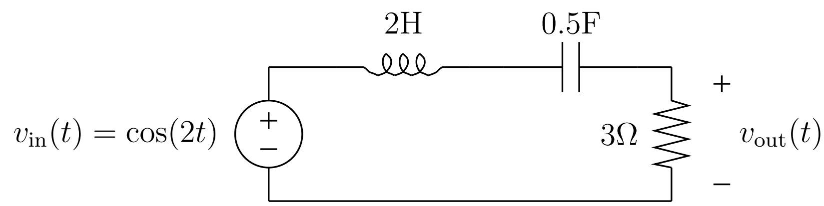
   (b) Suppose that the voltage source has a frequency of 60Hz. What value of Z_{L} maximizes the average power dissipated in it?

(c) Can you design a simple circuit with inductors, capacitors, and resistors, that has an impedance equal to the value found in part (a) at 50Hz and the value found in part (b) at 60Hz? (Note that such a circuit has the nice property of dissipating maximum average power whether the frequency is 50Hz or 60Hz.)

4. The output voltage of a CD player is sinusoidal, with amplitude 10V and frequency 20kHz. The CD player drives a power amplifier through a shielded cable that has a capacitance of 50pF/ft. The input resistance of the power amplifier is 10kΩ. The CD player can produce currents of ±10mA without distorting. (These values are realistic.)

   What is the maximum cable length the CD player can drive without distorting?

5. The circuit below is operating in sinusoidal steady-state. Find $v_{\text{out}}(t)$. Express your answer in the form $v_{\text{out}}(t) = a\cos(\omega t + \phi)$.

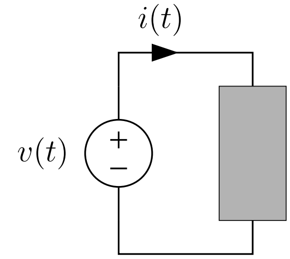


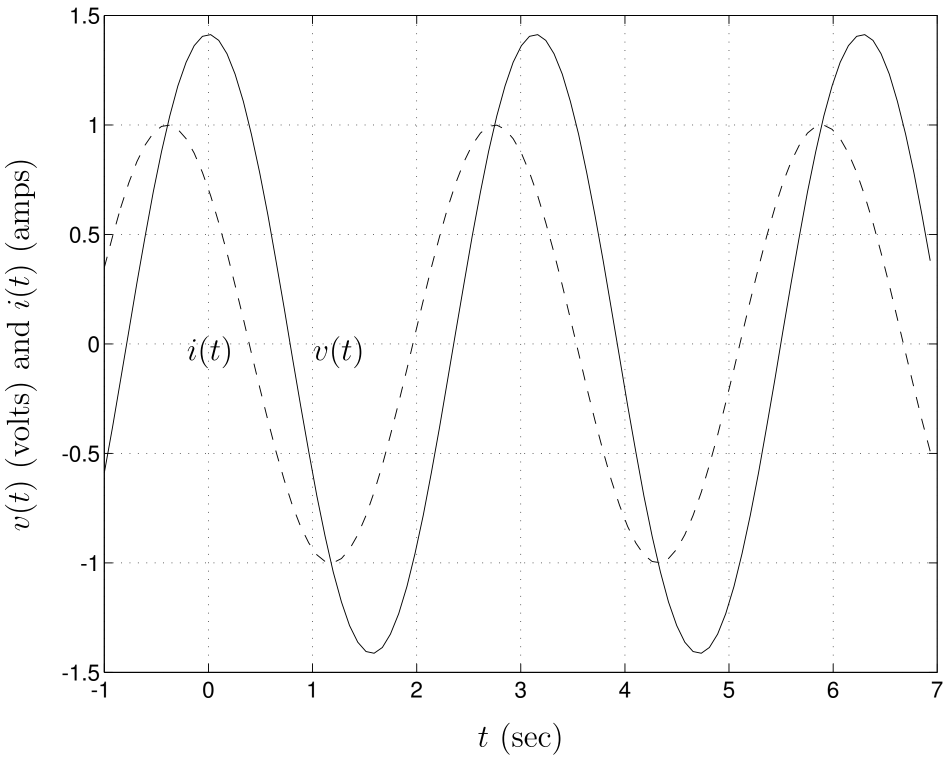
6. *Impedance of an exponential diode.* When the voltage across a linear element such as an inductor, capacitor, or resistor is sinusoidal, we can describe the element by $\mathbf{V} = Z\mathbf{I}$, where $\mathbf{V}$ and $\mathbf{I}$ are the phasors corresponding to the voltage and current, respectively, and Z is a complex number (the impedance) which depends on the type of element and the frequency of the sinusoidal voltage. This description can be considered as a sort of extension of Ohm's law to cover dynamic elements (in sinusoidal steady-state). The nice part about this description is that many of the formulas you know for resistors and resistances remain true for impedances. For example, the formulas for series connections, parallel connections, and $\Delta - Y$ transformations are the same as for resistances (except that the numbers can be complex).

   Now suppose a sinusoidal voltage with phasor $\mathbf{V}$ appears across an exponential diode with characteristic $i(t) = I_s(e^{v(t)/V_t} - 1)$. It is reasonable to guess that the current flowing through the diode can be characterized by the relation $\mathbf{I} = I_s(e^{\mathbf{V}/V_t} - 1)$. In words, we just plug in the appropriate phasors where the voltage and current appeared in the static case. Of course, $\mathbf{V}$ can be complex, but we know what the exponential of a complex number is, so the formula above does make sense.

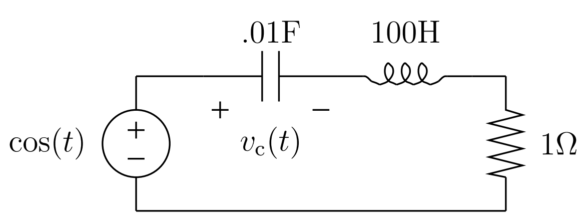
   Is this true? Discuss briefly.

7. The voltage and current in the circuit at right are shown in the plot below.
   Estimate the voltage phasor **V** corresponding to $v(t)$, the current phasor **I** corresponding to $i(t)$, the impedance **Z** of the shaded device, and the freqency ω (in radians per second).
   Your answers need to be accurate to only ±20%, so a calculator is unnecessary.





8. The circuit below is operating in sinusoidal steady-state.



   (a) What is the average power supplied by the voltage source?
   (b) What is the average energy stored in the inductor?

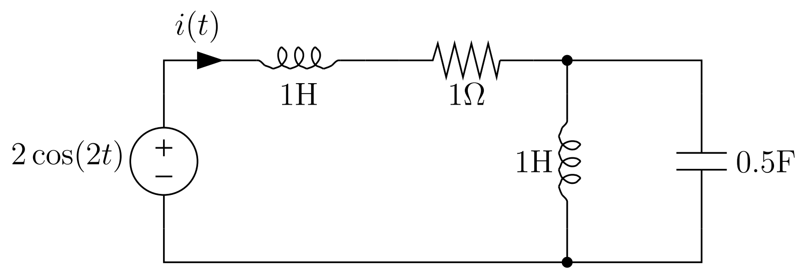
(c) What is the maximum value of $|v_c(t)|$? (Real capacitors have a *maximum voltage rating* which should not be exceeded. Usually, a capacitor with a higher voltage rating is larger and costs more than a capacitor with the same capacitance but lower voltage rating.)

9. Consider the circuit below, which is in sinusoidal steady-state. Find

   (a) the maximum value of the current, $\max_t |i(t)|$, and

   (b) the average power dissipated in the resistor, P_R.

   (c) the average energy stored in the capacitor.



10. An electrical element that is described by an impedance at 100rad/sec is subjected to four experiments. In the first experiment, the element is connected to a voltage source and the steady-state current through the element is determined. In the second experiment, the element is connected to a current source and the average power flowing into the element is determined. In the third experiment, the element is connected to a current source and the steady-state voltage across the element, and average power dissipated in the element are determined. In the last experiment, the element is connected to the voltage source and the steady-state current is determined. The experimental data is shown below:

Experiment	$v(t)$	$i(t)$	P_{avg}
1	$10\sin(100t)$	$1.414\cos(100t-45°)$	—
2	—	$2\sin(100t)$	10
3	$\cos(100t-45°)$	$\cos(100t)$	2.5
4	$\cos(100t)$	$-0.1\sin(100t)+0.1\cos(100t)$	—

Here, $v(t)$ and $i(t)$ denote the steady-state voltage across the element and current flowing through the element, respectively, with standard associated reference directions. P_{avg} denotes the average power dissipated in the element.

One (and only one) of these experiments was not conducted properly; the data from that experiment is not correct.

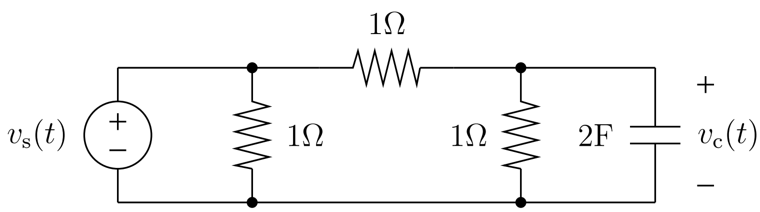
   (a) Find the impedance Z of the element at 100rad/sec.

(b) Identify the bad experiment.

11. In the circuit below,

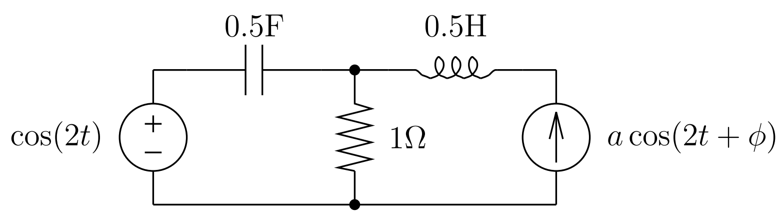
$$v_{\mathrm{s}}(t) = \begin{cases} \cos t & t < 0 \\ 0 & t \geq 0 \end{cases}$$

(You may assume that the circuit was in sinusoidal steady-state for $t \leq 0$.)

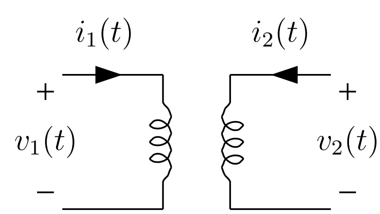


Find an explicit expression for $v_{\mathrm{c}}(t)$ for $t > 0$.

12. The circuit below is operating in sinusoidal steady-state. Find the amplitude a and phase ϕ of the current source that minimizes the average power dissipated in the resistor.



13. *Mutual inductance and transformers.* We encountered the ideal (static) transformer in lecture 4, where we noted that real transformers *never* operate under static conditions, *i.e.*, constant voltages and currents. In this problem we'll see a much better model of a real transformer as a pair of coupled inductors. The circuit variables for a transformer are labeled in the schematic diagram below. We will refer to the left-hand pair of terminals (*i.e.*, port) as winding 1 or the primary winding and the right-hand port as winding 2 or the secondary winding. (The term *winding* comes from the way real transformers are made, *i.e.*, by winding wire around a core, which is often made of iron. Of course, which winding you call primary and which secondary is just a matter of labeling.)



The model is

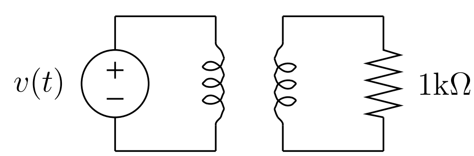
$$v_1(t) = L_1 \frac{di_1}{dt} + M \frac{di_2}{dt}, \qquad v_2(t) = L_2 \frac{di_2}{dt} + M \frac{di_1}{dt}.$$

L_1 is called the inductance of winding 1, and similarly for L_2. M is called the *mutual inductance* between the two windings. Note that when $M = 0$ this model reduces to a pair of independent inductors. The *turns ratio* of the transformer is defined as $n = \sqrt{L_2/L_1}$, and the *coupling coefficient* is defined as $k = M/\sqrt{L_1 L_2}$. Simple physics arguments can be used to establish that L_1 and L_2 are positive, and $|k| \leq 1$.

(a) Find the stored energy in the transformer at time t as a function of $i_1(t)$ and $i_2(t)$. The stored energy should satisfy the following equation: the integral of the total power entering the transformer (through both ports) over an arbitrary time interval is equal to the increase in stored energy over the time interval. Does your formula agree with the stored energy in two inductors when $M = 0$?

(b) Suppose that the voltages and currents are sinusoidal, with $\mathbf{V}_1$ denoting the phasor corresponding to $v_1(t)$, and so on. Derive expressions for the secondary voltage and current phasors (*i.e.*, $\mathbf{V}_2$ and $\mathbf{I}_2$) in terms of the primary voltage and current phasors (*i.e.*, $\mathbf{V}_1$ and $\mathbf{I}_1$). Compare these expressions to the equations for the ideal static transformer model, *i.e.*, $\mathbf{V}_2 = n\mathbf{V}_1$, $\mathbf{I}_2 = -\frac{1}{n}\mathbf{I}_1$. What happens as $k \to 1$ and ω becomes large?

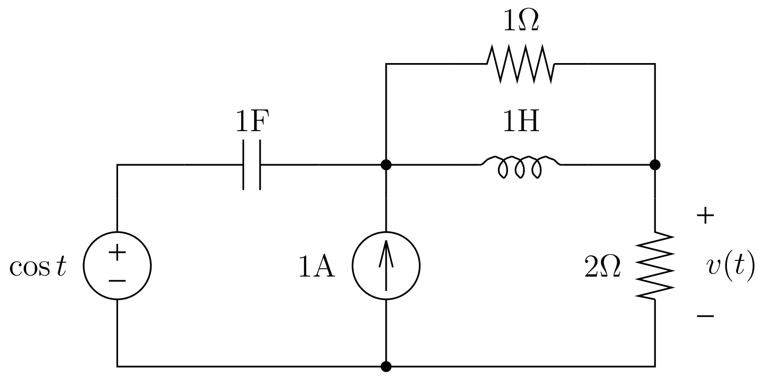
(c) The circuit below is operating in sinusoidal steady-state, with $v(t) = 165 \cos \omega(120\pi t)$. Find the voltage across the resistor.



The transformer has a primary inductance of 5H, a turns ratio $n = 2$, and a coupling coefficient of 0.95.

(d) Repeat the analysis of part (c), but assume the transformer is described by the ideal static model with a turns ratio of 2. Compare your answer to what you obtained in part (c).

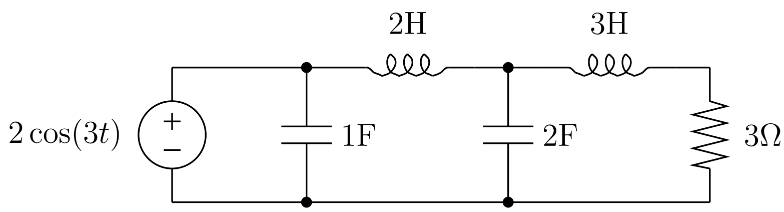
14. *Circuit with DC and sinusoidal sources.* You may assume the circuit below is in steady-state.



(a) Find the voltage $v(t)$.

(b) Find the average power dissipated in the 2Ω resistor.

15. The circuit below is operating in sinusoidal steady-state.



In this problem:

- $P_{\mathrm{s}}(t)$ is the power *supplied* by the voltage source at time t.
- $P_{\mathrm{r}}(t)$ is the power *dissipated* by the resistor at time t.
- $E(t)$ is the total energy stored in the inductors and capacitors at time t.

For any periodic function $f(t)$ we will use $\mathrm{AVG}(f)$ and $\max(f)$ to denote the time average value of $f(t)$ and the maximum value of $f(t)$, respectively. For example, $\max(P_{\mathrm{r}})$ is the maximum power dissipated in the resistor and $\mathrm{AVG}(P_{\mathrm{r}})$ is the average power dissipated in the resistor.

Consider the following statements:

(a) $\mathrm{AVG}(P_{\mathrm{s}}) = \mathrm{AVG}(P_{\mathrm{r}})$.

(b) $\max(P_{\mathrm{s}}) = \max(P_{\mathrm{r}})$.

(c) $\max(P_{\mathrm{s}}) \geq \max(dE/dt)$.

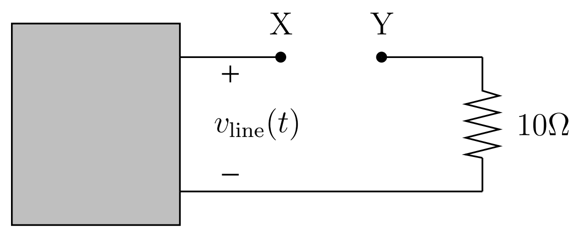
(d) $\max(dE/dt) \leq 2\omega\mathrm{AVG}(E)$.

(e) $\max(P_{\mathrm{s}}) = 2\mathrm{AVG}(P_{\mathrm{s}})$.

(f) $\max(P_{\mathrm{r}}) = 2\mathrm{AVG}(P_{\mathrm{r}})$.

Decide whether each statement is TRUE or FALSE.

16. Consider the circuit below:



The box at left is an electrical model of a power generation and distribution system. You can assume that it consists of inductors, capacitors, resistors, transformers, and several sinusoidal voltage sources with a frequency of 60Hz. You do not know the model.

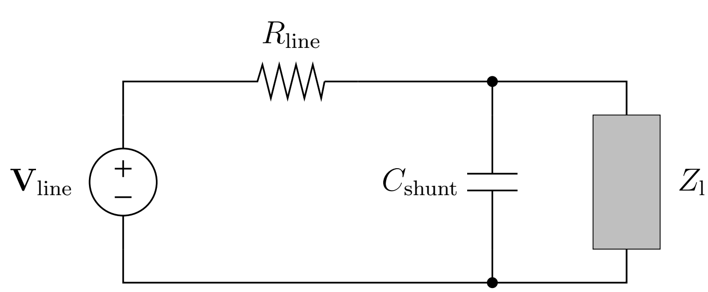
When the nodes X and Y are not connected we find that $v_{\text{line}}(t) = 165\cos(120\pi t)$. When the nodes X and Y are connected by a wire (of zero resistance) we find that $v_{\text{line}}(t) = 155\cos(120\pi t + 10°)$.

You are allowed to put either a capacitor or an inductor between the nodes X and Y (with positive capacitance or inductance, respectively). Your goal is to maximize the average power dissipated in the 10Ω load.

Specify the type of element (inductor or capacitor) and its value (in Henrys or Farads, respectively). Note that you *cannot* change the 10Ω load resistor.

17. *Designing an optimal shunt capacitance.*

The voltage source phasor $\mathbf{V}_{\text{line}} = 180\text{V}$ and resistance $R_{\text{line}} = 0.8\Omega$ in the circuit below are the Thevenin equivalent of a power generation and distribution system that is operating at 60Hz. The load impedance is $Z_{\text{l}} = (10 + 3j)\Omega$. It is common practice to compensate for the load reactance by adding a capacitor across the load, called a *shunt capacitor,* as shown in the circuit below. If the shunt capacitor is properly designed, the average power delivered to the load will be larger than if there were no shunt capacitor.



(a) Find the average power P delivered to the load impedance with no shunt capacitor, *i.e.*, $C_{\text{shunt}} = 0$.

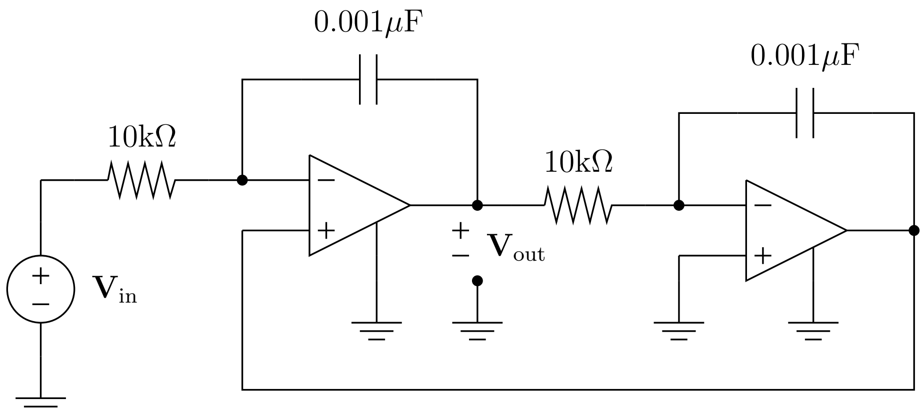
(b) The load impedance Z_l can be represented as a *parallel* connection of a resistor R and an inductor L (at 60Hz). Find R and L.

Note: we really do mean parallel, not series!

(c) Find the value of the shunt capacitor that maximizes the average power delivered to the load impedance.

Hint: it may be useful to transform to a Norton equivalent and use the result of part b.

18. *Resonance in an active circuit.* The circuit below is operating in sinusoidal steady-state at a frequency ω rad/sec. You can assume the op-amps are given by the ideal static op-amp model.

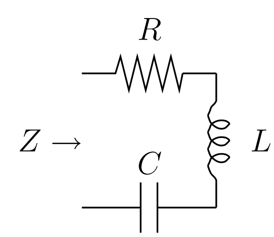


(a) Find the ratio of the output voltage phasor $\mathbf{V}_{\text{out}}$ to the input voltage phasor $\mathbf{V}_{\text{in}}$ as a function of frequency. This ratio is called the *transfer function* from the input voltage to the output voltage.

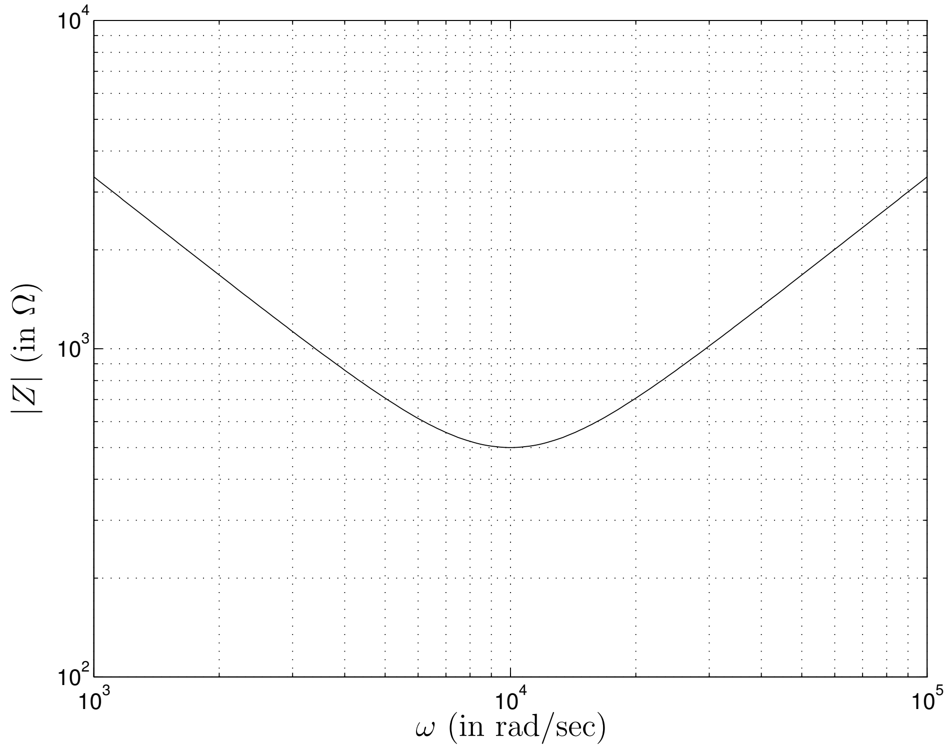
(b) Find the frequency ω_0 for which the magnitude of the transfer function is largest.

(c) Find the two frequencies $0 < \omega_1 < \omega_2$ for which the magnitude of the transfer function is a factor of $\sqrt{2}/2$ times the magnitude at the frequency ω_0.

Hint: if you're clever you can use some formulas from the notes.



19. The plot below shows the magnitude of the impedance Z of a series connection of a resistance R, an inductance L, and a capacitance C, as shown at right. Estimate R, L, and C. Note that frequency and impedance are given on a logarithmic scale.



20. *Six-phase power.* Some systems operate on six power lines which are separated by 60° phase shifts:

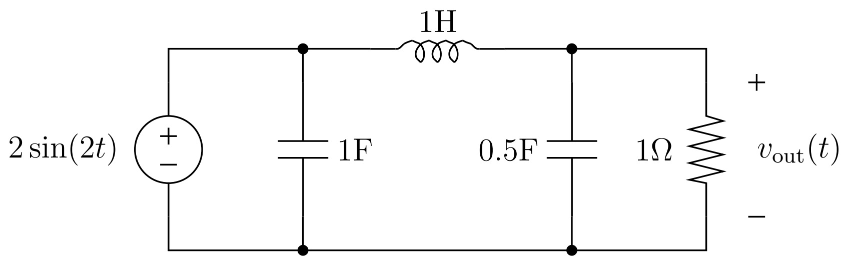
$$
\begin{aligned}
v_0(t) &= 170\cos(120\pi t) \\
v_1(t) &= 170\cos(120\pi t + 60^\circ) \\
v_2(t) &= 170\cos(120\pi t + 120^\circ) \\
v_3(t) &= 170\cos(120\pi t + 180^\circ) \\
v_4(t) &= 170\cos(120\pi t + 240^\circ) \\
v_5(t) &= 170\cos(120\pi t + 300^\circ)
\end{aligned}
$$

Find the amplitude and phase of the voltage from each leg to leg 0, *i.e.*, $v_1 - v_0, \ldots, v_5 - v_0$.

Enter your answers in the table below. Express the phases in degrees.

voltage	amplitude	phase
$v_1 - v_0$		
$v_2 - v_0$		
$v_3 - v_0$		
$v_4 - v_0$		
$v_5 - v_0$		

21. The circuit below is operating in sinusoidal steady-state.



Find $v_{\text{out}}(t)$, expressed in the form $v_{\text{out}}(t) = a\cos(\omega t + \phi)$, with ϕ in radians.

22. *Series and parallel resistance/reactance representations.*

Consider an impedance $Z = R + jX$. We can represent it as a resistance R in series with a reactance jX. We can also represent it as a resistance $\tilde{R}$ in *parallel* with a reactance $j\tilde{X}$.

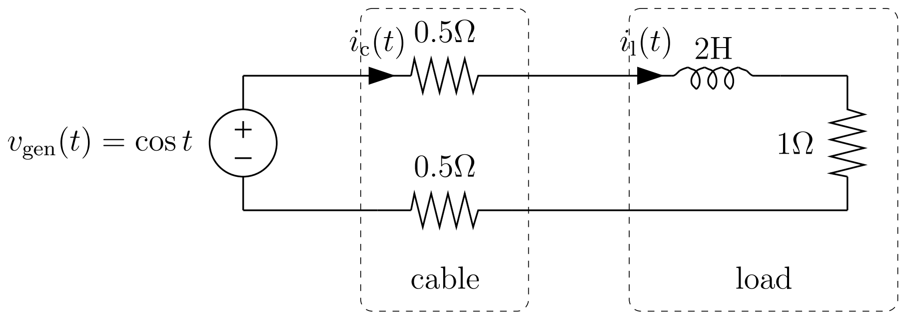
Express $\tilde{R}$ and $\tilde{X}$ in terms of R and X. Try to simplify your expressions as much as possible.

23. The function $z(t)$ is sinusoidal with frequency 2 rad/sec, and satisfies the differential equation

$$\frac{d^2z}{dt^2} + \frac{dz}{dt} + 2z + \sin(2t) = 0.$$

Find the amplitude of z, *i.e.*, $\max_t |z(t)|$.

24. In this problem we consider a generator that drives a load through a two-wire cable. The generator is modeled as a sinusoidal voltage source, $v_{\text{gen}}(t) = \cos t$. The cable is modeled as a resistance of 0.5Ω in each wire, and the load is modeled as a resistance of 1Ω is series with an inductance of 2H. The circuit, which is in sinusoidal steady-state, is shown below.



(a) Find the maximum current in the cable, *i.e.*, $\max_t |i_c(t)|$.

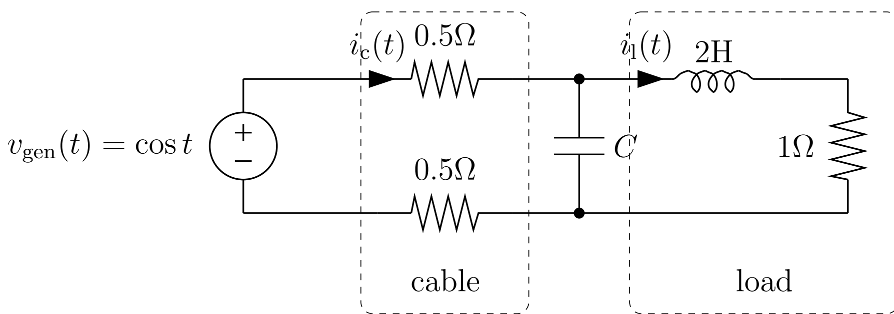
(b) Find the maximum current in the load, *i.e.*, $\max_t |i_l(t)|$.

(c) Find the average power p_{gen} *supplied by* the generator.

(d) Find the average power p_l *dissipated in* the load (*i.e.*, the load inductor and the load resistor).

(e) What fraction of the time does the generator absorb energy (*i.e.*, dissipate positive power)? Express your answer as a percentage.

25. This problem continues from the previous one. It is very common in practice to add a shunt capacitance across the load, as shown below.



(a) Find the value of C that results in the cable current i_c being in phase with the generator voltage $v_{gen}(t)$. We will use this value of C in the rest of this problem.

(b) Find the maximum current in the cable, *i.e.*, $\max_t |i_c(t)|$.

(c) Find the maximum current in the load, *i.e.*, $\max_t |i_l(t)|$.

(d) Find the average power p_{gen} *supplied by* the generator.

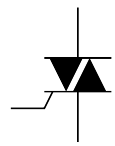
(e) Find the average power p_l *dissipated in* the load (*i.e.*, the load inductor and the load resistor).

Exercises on Fourier Series

1. What is the fundamental period and fundamental frequency of the function $f(t) = |\cos 2\pi t|$? Find its Fourier series. Express the answer in both sine/cosine and complex exponential form. This function is sometimes called a *full-wave rectified* sinusoid.

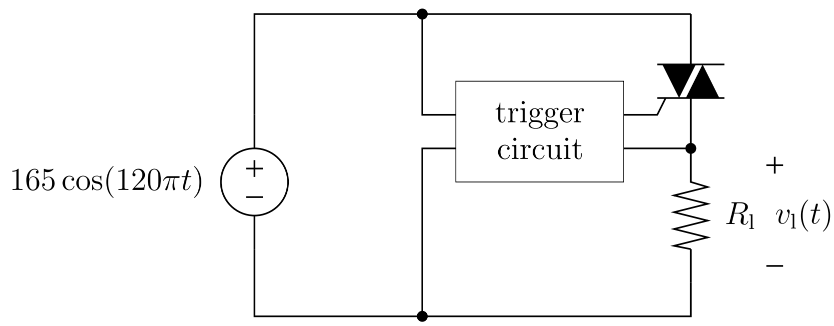
2. *SCR and TRIAC dimmers.* Dimmers for incandescant lights, power controllers for heaters, and speed controllers for some motors are all based on circuits that use devices called SCRs or TRIACs. These circuits work by varying the RMS value of the load voltage, which in turn varies the average power supplied.

   TRIACs (which are made from SCRs—silicon controlled rectifiers) are more common in residential light dimmers and speed controllers, so we'll describe them. The schematic symbol for a TRIAC is shown below.

   A simple model of a TRIAC is a switch that can be controlled electronically by a third terminal called the trigger input. When the TRIAC is "on" or "conducting" it behaves like a wire (closed switch) connected between the terminals A and B; when it is "off" or "open" it behaves like an open circuit between terminals A and B. If an appropriate voltage is applied between the trigger terminal and terminal B, the TRIAC will turn on, and remain on until the magnitude of the current flowing from A to B becomes zero (even if the trigger voltage is removed). Thus a TRIAC can be turned on or "triggered" by applying a pulse to the trigger terminal; it turns off ("extinguishes") whenever the current flowing from A to B is zero. It's essentially a switch that can be turned on at any time, but only goes off by itself, when the current through it becomes zero.

   The circuit below shows a basic TRIAC dimmer circuit. The trigger circuit provides a pulse of voltage sufficient to trigger the TRIAC on each half-cycle, whenever $\cos(120\pi t - \theta)$ passes through zero. θ is called the *firing angle,* and can be adjusted between zero and 180°. Note that the TRIAC turns off whenever $165\cos(120\pi t)$ passes through zero, so $\theta/120\pi$ gives the delay between the TRIAC turning off (because the load current becomes zero) and the TRIAC turning on again (because of the trigger pulse). You can check that when the firing angle is very small and positive, the TRIAC is on most of the time; when the firing angle is a little less than 180°, the TRIAC is off most of the time, and when the firing angle is 90°, the TRIAC is on half the time (for the second half of each half-cycle).



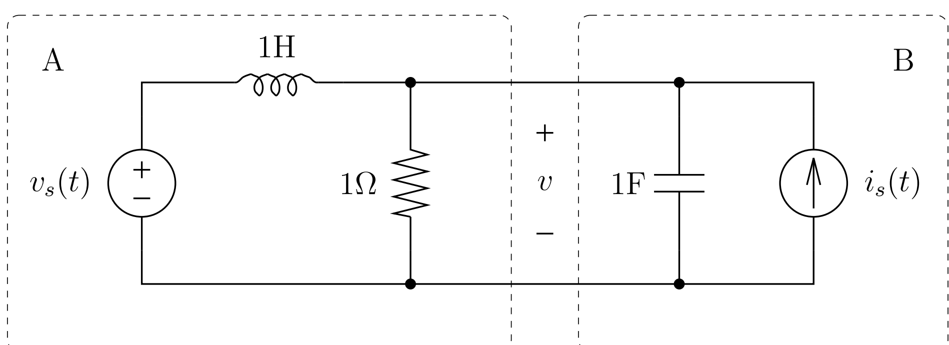
(a) Sketch the waveform of the load voltage v_l for several different firing angles. (You can use Matlab if you like.)

(b) Find the RMS value of v_l as a function of the firing angle θ. Give a sketch of RMS(v_l) versus θ. (You can use Matlab if you like.)

(c) Find the average power dissipated in the TRIAC as a function of the firing angle and load resistance (you can neglect the power of the trigger pulses). Comment briefly on the practical implications. (This partially explains why a dimmer that can handle many hundreds of watts of lighting load can fit inside a light switch box without causing a fire.)

(d) Find the Fourier series of v_l for the firing angle $\theta = 90°$.

3. The circuit below is operating in periodic steady-state. The voltage source is $v_s(t) = \sin t + 2\cos 3t$ and the current source is $i_s(t) = \cos t - \sin 2t$.



(a) Find the Fourier series (coefficients) of the voltage v.

(b) Find the average power flow from subcircuit A to subcircuit B at DC, the fundamental ($w_0 = 1$) and all higher harmonics.

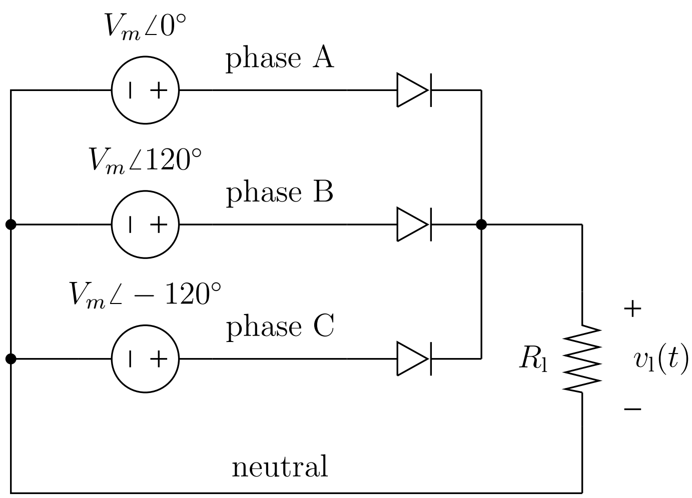
(c) Find the total average power flow from subcircuit A to subcircuit B.

(d) Find the average energy stored in the inductor.

(e) Could you find the maximum power flow from subcircuit A to subcircuit B? You don't need to find the specific number, but explain how you would do it.

4. The voltage across an exponential diode is $v(t) = 0.65 + 0.001\cos(2000\pi t)$. *Estimate* the DC, fundamental, and second harmonic of the current i. (Please explain your estimate; zero is not an acceptable answer.) You may use the parameters $V_T = 26\text{mV}$, $I_0 = 10^{-14}\text{A}$ for the diode.

5. *Three-phase rectified power.* The circuit below is often used to connect three-phase AC power to a load that can only handle voltages of one polarity. It is used in X-ray machines and many electrochemical industrial processes.



The line-to-line voltage is 208V RMS, so $V_m = 208\sqrt{2/3} = 170\text{V}$. The line frequency is 60Hz. You can assume the diodes are ideal. (By the way, when diodes are used in high power circuits such as this one, they are more commonly called *rectifiers.*)

(a) Sketch the waveform of v_l.

(b) What is the fundamental frequency of v_l?

(c) Find the average value and the RMS value of v_l.

(d) The *ripple* of the voltage v_l is defined as v_l minus its average value. What is the RMS value of the ripple of v_l? Find the *percent ripple,* which is defined as the ratio of the RMS ripple to the RMS value of v_l.

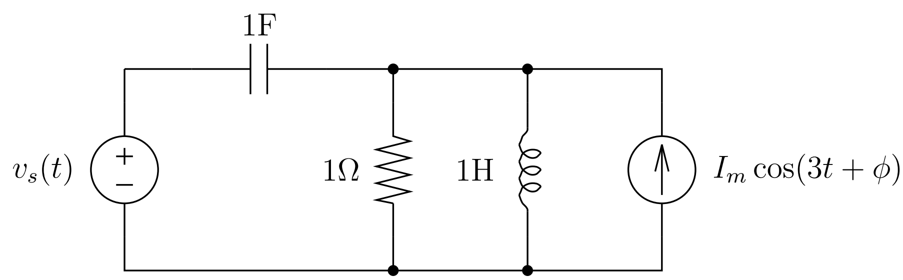
(e) What fraction of the total average load power is contained in the DC and fundamental components?

6. Suppose that f is periodic with period $T > 0$ and has Fourier coefficients $a_0,\ a_1, \ldots,$ $b_1,\ b_2, \ldots,$ and complex Fourier coefficients $c_0,\ c_1, \ldots$

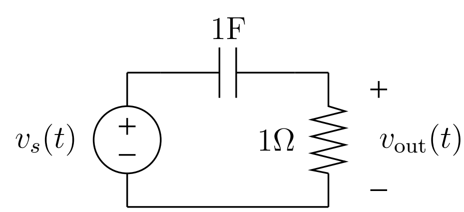
(a) What is the Fourier series of the derivative of f? (You may assume that the derivative exists and has a Fourier series.) Give your answer in terms of both of the sine and cosine Fourier coefficients and also the complex Fourier coefficients.

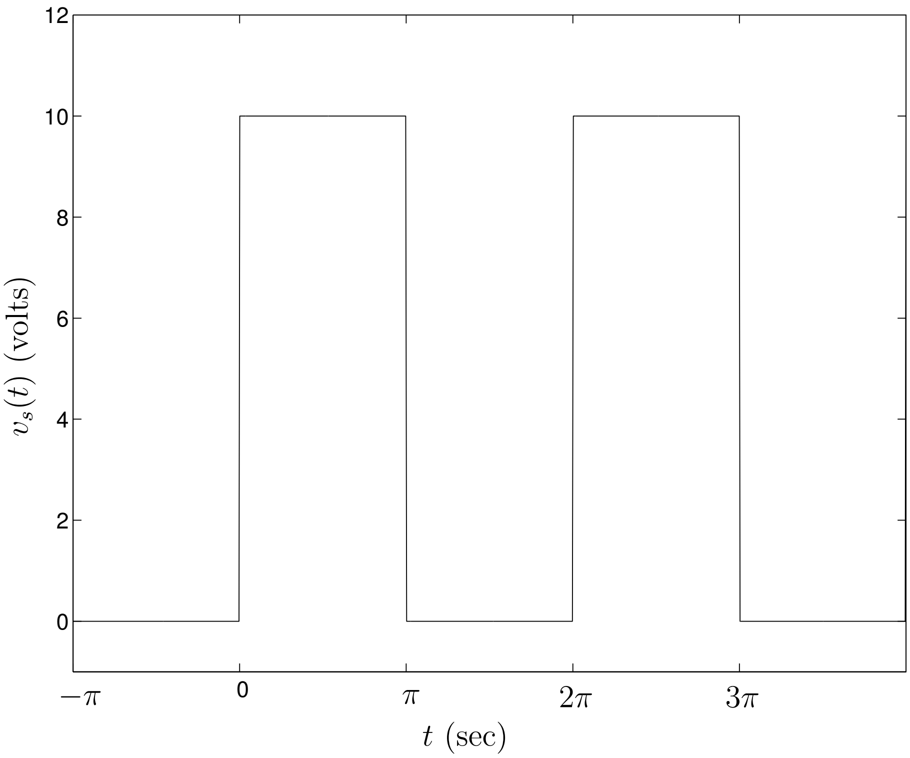
(b) Let g be the integral of f, *i.e.*, $g(t) = \int_0^t f(\tau)d\tau$. Under what conditions on f is the function g periodic? (*i.e.*, always? never?) When g is periodic, give its Fourier series.

7. Consider the circuit shown below. The voltage source is a sawtooth waveform with fundamental frequency $\omega_o = 1\text{rad/sec}$ and maximum amplitude 1V. Find the amplitude and phase of the current source that minimizes the average power dissipated in the resistor. How much smaller is this minimum average power than the average power dissipated in the resistor when the current source is turned off? Explain what you are doing.

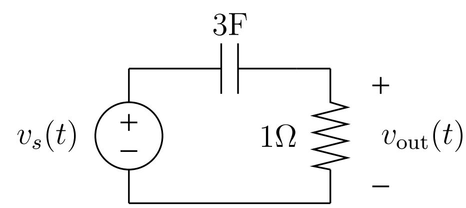


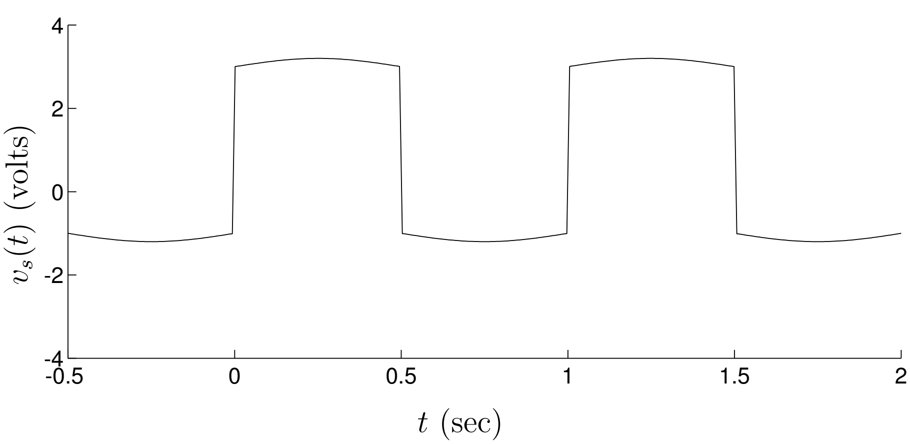
8. The circuit at right is in periodic steady-state. The voltage $v_s(t)$ is plotted below. What is the average value of v_{out}? Find the fundamental component of v_{out}, expressed in the form $V_m\cos(\omega_0 t + \phi)$.



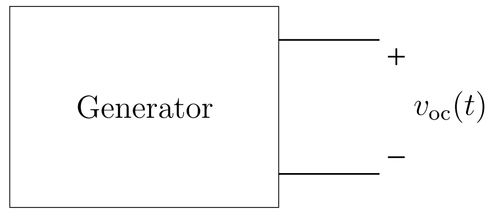


9. The circuit at right is in periodic steady-state.
   Estimate the RMS values of v_s and v_{out}.
   An accuracy of $\pm 20\%$ is sufficient.



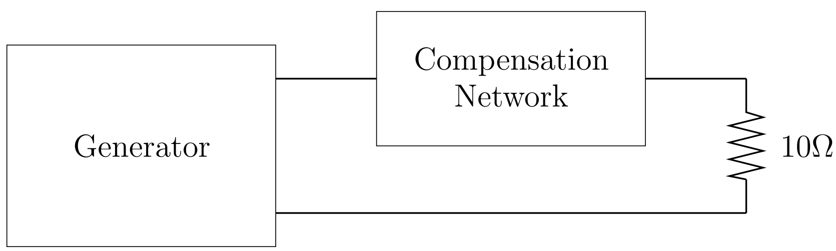


10. *Extracting maximum power from a periodic generator.* The power generator shown below can be modeled as a circuit that contains a periodic voltage source, inductors, capacitors, and resistors. You do not know the circuit or any of the component values.



You measure the (periodic) voltage waveform $v_{\text{oc}}(t)$, the open-circuit voltage that appears at the generator output with no load connected. You also measure the (periodic) voltage waveform $v_{\text{ld}}(t)$, the "loaded" voltage that appears at the generator output with a 10Ω load connected. Roughly speaking, the loaded voltage waveform is a bit smaller than the open-circuit voltage and has a bit of a different shape. You can assume that both waveforms are pretty well described by a partial Fourier series with five harmonics.

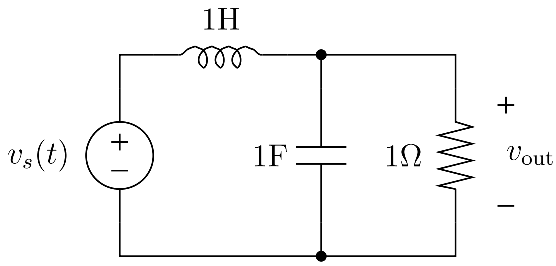
You are asked to design a series compensating network that results in maximizing the total average power dissipated in the 10Ω load resistor in the circuit shown below. Your compensating network can contain inductors, capacitors, and resistors.



Carefully explain *how* you would solve this problem. Describe the major steps and reasoning involved. (Obviously you can't actually solve the problem, *i.e.*, find a specific compensation network, since we haven't given you the specific open-circuit and loaded voltage waveforms.) Please address the following topics in your discussion: Do you have enough information to design such a network? If not, what additional information would you like to have? One simple compensation network is just a wire; hopefully your network results in more average power being dissipated in the 10Ω resistor. How much more?

Note: This problem is a bit vague but very important. It requires combining several things you know from 101 and 102 along with some careful thinking.

11. The circuit below is operating in periodic steady-state. The voltage source is $v_s(t) = 1 + \cos t$.



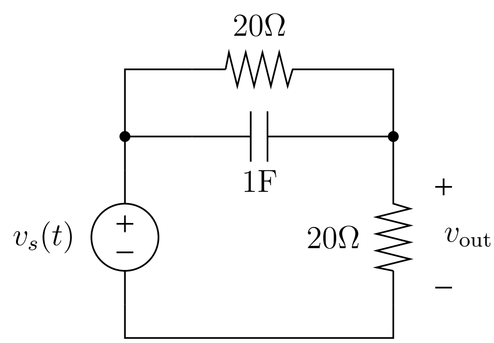
(a) Find $v_{\mathrm{out}}(t)$. Express any sinusoidal terms in your answer in the form $V_m \cos(\omega t + \phi)$.

(b) Find E, the average energy stored in the inductor.

(c) Find the percent RMS ripple of v_{out}, defined as

$$\%\ \text{ripple} = 100\frac{\mathrm{RMS}(v_{\mathrm{out}} - \mathrm{AVG}(v_{\mathrm{out}}))}{\mathrm{RMS}(v_{\mathrm{out}})}.$$

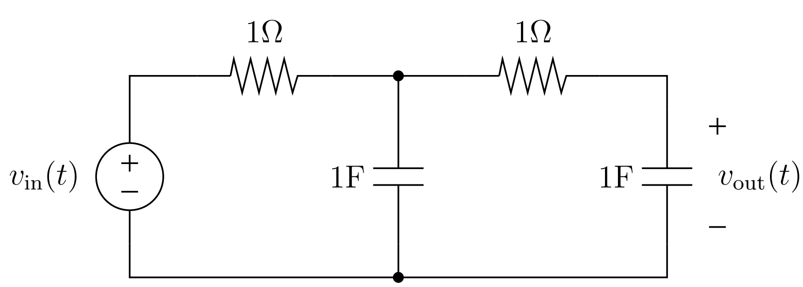
12. The circuit below is operating in periodic steady-state. The voltage source is a sawtooth with period 2π and amplitude 1, *i.e.*, $v_s(t) = t/(2\pi)$ for $0 \le t < 2\pi$. Sketch $v_{\mathrm{out}}(t)$. Your sketch does not have to be perfect, but the important features of v_{out} should be clear (*e.g.*, approximate maximum and minimum values, approximate shape).



You may need the Fourier series of v_s: $v_s(t) = \dfrac{1}{2} - \displaystyle\sum_{k=1}^{\infty} \frac{1}{\pi k} \sin kt$.

13. The circuit below is operating in periodic steady-state. The voltage source is a square-wave of frequency 1Hz between values 0 and 1, *i.e.*,

$$v_{\mathrm{in}}(t) = \begin{cases} 1 & k \le t \le k + 1/2, \quad k = 0, \pm 1, \pm 2 \ldots \\ 0 & k + 1/2 < t < k + 1, \quad k = 0, \pm 1, \pm 2 \ldots \end{cases}$$



Find the Fourier series of v_{in} and v_{out}.

Give a simple, approximate description of v_{out}.

14. *Numerical experiments with Fourier Series.*

You'll need to use Matlab for the following problem. Matlab is installed on the workstations in Sweet Hall, as well as the Macintoshes in most public clusters on campus. A student version of Matlab is available at the Stanford Bookstore for about the cost of a textbook. For information on how to use Matlab, you can consult the Matlab primer available from `http://www-leland.stanford.edu/class/ee101/primer.ps`.

For this particular problem you'll need three files: `ekg.m`, `fourier_coeff.m`, and `fourier_sum.m`. They are available at `/afs/ir/usr/class/ee102/hw2/` from your leland account (or any machine with `afs`); you can copy them from there to your working directory.

The first file is a Matlab m-file, and the latter two are Matlab functions (that we wrote for you). A Matlab m-file is a sequence of Matlab commands in an ascii file (*e.g.*, created using a text editor) with a name that ends in ".m" You can execute an m-file from within matlab by typing its name (without the ".m" extension) at the command prompt. [1] Matlab then executes the commands within the m-file exactly as if they were typed individually in the command window. For instance, you can type `ekg` to run `ekg.m`.

In this problem you will also need the two functions `fourier_coeff.m` and `fourier_sum.m`, which we wrote for you, and encourage you to examine. You can invoke them exactly like you would use built-in functions like `sin` or `abs`. You can even type `help fourier_coeff` to see the first few lines of fourier_coeff.m, which explains the details of how the function operates.

Matlab manipulates matrices and vectors: it cannot directly handle periodic signals. In Matlab we can approximately represent a periodic signal by giving a large vector which is a "sampled version" of the periodic signal. To represent a periodic signal $f_c(t)$, which has period T, we use a Matlab vector `f` whose coefficients are the values of f_c at equally spaced time intervals:

$$\mathtt{f}(k) = f_c((k-1)T/N), \quad k = 1, \ldots, N,$$

where N is the length of `f`, *i.e.*, the number of time samples. For the problems we'll encounter in this class, we can take N to be between 100 and 1000.

The time t in the periodic signal $f_c(t)$ that corresponds to the index k in the Matlab vector `f(k)` is given by $t = (k-1)T/N$. As k ranges from 1 to N, t ranges from 0 to $(N-1)T/N$. We can make up a Matlab vector that gives the sample times by

[1] Actually, when a name is typed into Matlab, Matlab first checks to see if it is a defined variable in memory, then checks if it is a built-in function, then looks in the working directory (the directory where you started Matlab) for an m-file of the same name, and finally looks in the Matlab path for an m-file of the same name.

```
t=0:T/N:T*(N-1)/N; % vector of times of length N
```

Then to plot one cycle of the periodic function f_c, you could use the following command:

```
plot(t,f);
```

(If you type just `plot(f)`, you'll get the wrong X-axis scale — it will use the index k.)

When working with periodic signals in Matlab, we often have to evaluate integrals (in RMS calculations, Fourier coefficients calculations, etc.). This is handled by simply approximating the integral as a sum: we use the approximation

$$\int_0^T f_c(t)\, dt \approx \frac{T}{N} \sum_{k=1}^{N} f_c((k-1)T/N)$$

which is accurate provided N is big enough that f_c doesn't change too much between time samples. (If you examine our function `fourier_coeff`, you'll see that we used such an approximation.)

Finally, we get to the problems:

(a) Generate a sawtooth waveform in a Matlab vector called `sawtooth` with period 2sec and amplitude 4. Your sawtooth should be stored in a row vector containing one period of the waveform, as described above. Use a sufficiently large number of samples, perhaps $N = 200$. Plot one or two cycles of your waveform.

(b) Use `fourier_coeff` to (approximately) compute the first 30 Fourier coefficients of your sawtooth signal. Check the computed coefficients against the coefficients found analytically in the notes, and briefly explain any discrepancy.

(c) Use `fourier_sum` to reconstruct a waveform from the first 30 Fourier coefficients. Plot two cycles. Does your plot resemble your original sawtooth waveform?

(d) Repeat steps 2-3, this time using only 10 coefficients.

(e) For fun, change the term $\frac{1}{\pi 3}\sin(3\omega t)$ into $\frac{2}{\pi 3}\cos(3\omega t)$, reconstruct, and plot. Does the plot change alot or little?

(f) Now let's consider a real periodic signal: Jon Carter's EKG. The m-file `ekg.m` defines a vector `ekg` and also `t` which is a sampled version of Jon's EKG and the corresponding times, over one heartbeat period, which is one second. Plot two cycles.

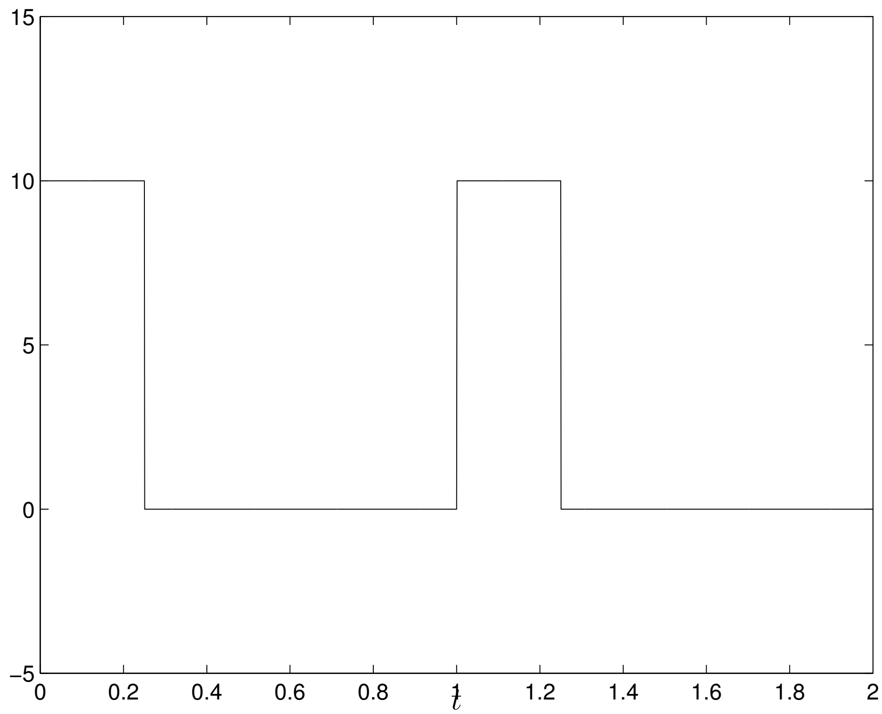
(g) Find the RMS value of Jon's EKG. You will have to approximate the integral as a sum.

(h) Find the first 30 fourier coefficients of Jon's EKG.

(i) Plot the spectrum, up to the 30th coefficient. What fraction of the total energy in Jon's EKG is contained in the first 10 terms? The first 30?

(j) Reconstruct the partial Fourier series of Jon's EKG using 10 terms and also 30 terms. Plot these and compare to the original EKG. For each case, find the RMS value of the error function (as defined in the class notes), and express it as a fraction of the total RMS value. Is it consistent with your calculations from the spectrum?

15. Consider the periodic function f with period $T = 1$, two cycles of which are shown below. This function is sometimes called a "25% duty cycle squarewave" since it is "on" (*i.e.*, assumes its high value) 25% of the time.



Find its average value a_0, its fundamental frequency coefficients a_1 and b_1, and its root-mean-square value RMS(f).

16. Suppose f is periodic and satisfies

$$f'' + 2f' + f = q(t),$$

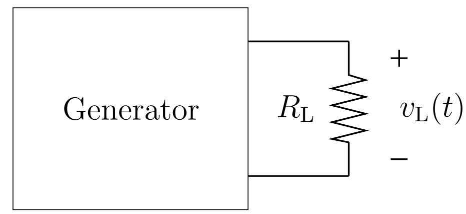
where q is a (periodic) 'reverse triangle wave' with frequency 5Hz, *i.e.*,

$$q(t) = 1 - 5t \quad \text{for} \quad 0 \le t < 0.2.$$

What is the average value of f?

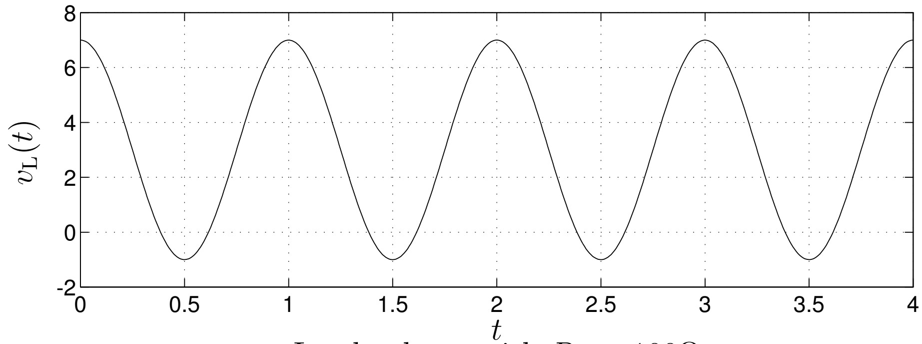
17. *Output voltage waveform as load resistance varies.*

The generator shown at right can be modeled as a circuit containing a DC (constant) voltage source, a sinusoidal current source, and several resistors, capacitors, and inductors. You do not know the circuit. Throughout this problem we assume periodic steady-state.

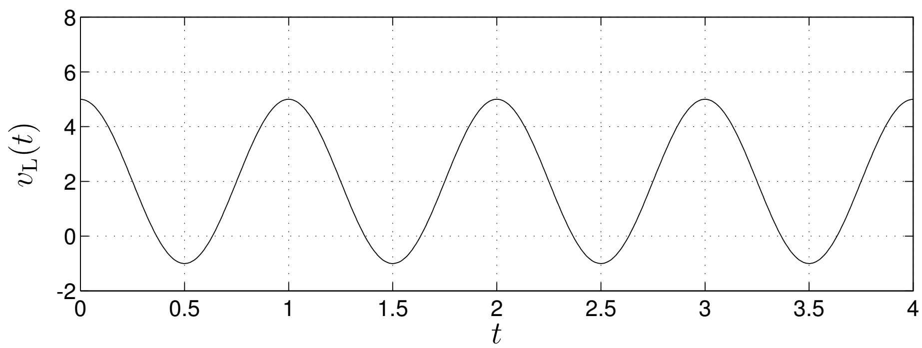


The two plots below show the load voltage waveform $v_L(t)$ for two different values of load resistance, $R_L = 200\Omega$ and $R_L = 100\Omega$.

Load voltage with $R_L = 200\Omega$



Load voltage with $R_L = 100\Omega$



(a) From the data given, can you determine the voltage waveform for a load resistance $R_L = 50\Omega$? If you can, do so. If not give "can't determine" as your answer.

(b) Suppose the frequency of the sinusoidal source in the generator exactly doubles, and the load resistance is $R_L = 100\Omega$. From the data given above, can you determine the load voltage waveform? If you can, do so. If not give "can't determine" as you answer.

18. *Some problems involving RMS value of periodic signals.*

(a) Suppose f is a periodic function with period 2π. We know that $\mathrm{RMS}(f) = 2$, and $\mathrm{RMS}(f+1) = 1$. ($f+1$ denotes the periodic function g given by $g(t) = f(t)+1$.)

What is AVG(f)? Either find (the specific number) AVG(f) or state 'cannot be determined' if it cannot be determined from the information given.

(b) Consider two functions p and q, with period 1, and $\mathrm{RMS}(p) = \mathrm{RMS}(q) = 2$. These two functions are never 'on' at the same time, *i.e.*, whenever $p(t) \neq 0$, we have $q(t) = 0$.

Can you determine $\mathrm{RMS}(p - 2q)$ from the information given? Either give (the specific number) $\mathrm{RMS}(p - 2q)$, or state 'cannot be determined' if it cannot be determined from the information given. ($p - 2q$ is the periodic function r defined by $r(t) = p(t) - 2q(t)$.)

Exercises on Bode Plots

1. Draw a rough sketch of the Bode magnitude and phase plot of the transfer function

$$H(s) = \frac{s^2 - 0.1s + 4}{s^2 + 0.1s + 1}.$$

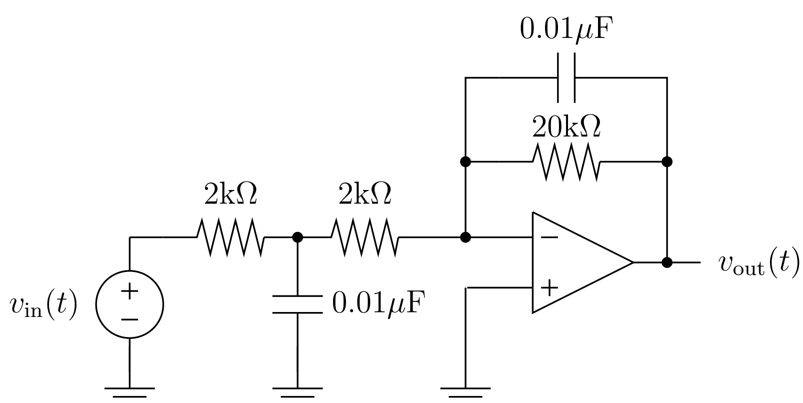
   Use a magnitude range of -40dB to $+40$dB, and a phase range of $-360°$ to $+360°$. Don't worry about corrections on the order of a few dB or a few tens of degrees.

   Label the key features of your plot. Check your plot at a few obvious frequencies.

2. An amplifier with a transfer function H has a DC gain $H(0) = 10^3$, poles at $s = -100$rad/sec and $s = -10^6$rad/sec, and a zero at $s = +10^4$rad/sec. (Note the signs of the poles and zeros!)

   Sketch the Bode magnitude and phase plots of H. Draw both a straight-line approximation and a "smooth" curve.

3. This problem concerns the circuit shown below. You can assume that the op-amps is ideal.



   (a) Find the transfer function H from v_{in} to v_{out}.
   (b) Plot the poles and zeros in the complex plane. Verify that this circuit is stable.
   (c) Sketch the Bode plot of H. Label important features of the plots. Would you describe this system as low-pass, band-pass, high-pass, or none of these?
   (d) Assume the capacitors are initially uncharged. Suppose that for $t \geq 0$, v_{in} is a sinusoid with amplitude 1V, frequency 3kHz, and phase 0°. You know that v_{out} will approach the sinusoidal steady-state response as $t \to \infty$. But how long will it take? Find a time T such that for $t \geq T$ the actual and steady-state responses are within about 1% of the amplitude of the steady-state response. Your number T does not have to be the smallest possible such T, just within a factor of two or three.

(e) Can you find appropriate initial capacitor voltages such that the system is in sinusoidal steady-state *immediately*, *i.e.*, from $t = 0$ on?

4. *A delay system.* Consider a system with input u and output y described by $y(t) = 0$ for $0 \le t < 1$ and $y(t) = u(t-1)$ for $t \ge 1$. Thus the output is the same as the input but delayed one second. Find the transfer function H of this system. What is its DC gain? Sketch the Bode plot of H. Can you sketch its poles and zeros in the complex plane?

5. *A system with undershoot.*

   In this problem we consider a system described by the transfer function
$$H(s) = \frac{1-s}{(1+s)(1+2s)},$$
   with input u and output y.

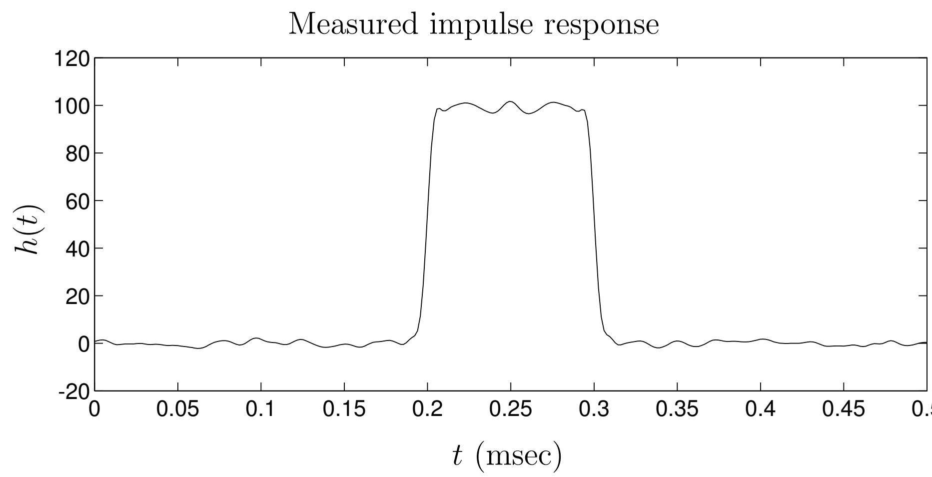
   (a) Sketch the Bode plot of H. Be careful with the phase plot. Does the magnitude plot look like the magnitude plot of a simpler transfer function? Can you explain this?

   (b) Sketch the step response. Make sure the final value and the slope at $t = 0+$ are correct. The interesting effect you see for small t is called *undershoot.*

   (c) Suppose that at $t = 200$ the input switched from the value 3 to -1, *i.e.*, $u(t) = 3$ until $t = 200$; after that $u(t) = -1$. Sketch $y(t)$ for t near 200, say, several seconds before to several seconds after. Systems with undershoot are sometimes descibed this way: "when you change the input rapidly from one constant value to another, the output first moves in the wrong direction". Does this make sense?

   (d) Can you find u such that $y(t) = 1 - e^{-t/2}$? Any comments about the u you found? Can you trace the interesting feature of u to some particular property of H, *e.g.*, its DC gain, pole locations, etc.?

6. The impulse response of a system described by a transfer function H is measured experimentally, and plotted below:



(a) Estimate $H(0)$, *i.e.*, the DC gain of this system.

(b) Estimate $H(j\omega)$ for $\omega = 2\pi \cdot 500\text{Hz}$, $\omega = 2\pi \cdot 5\text{kHz}$, $\omega = 2\pi \cdot 10\text{kHz}$, and $\omega = 2\pi \cdot 1\text{MHz}$. Explain your approximations. An answer of the form "small" is OK provided you give some rough maximum as in "$H(j\omega)$ is small, probably less than 10^{-4} or so".

(c) Sketch the step response of this system.

(d) The (10%-90%)*rise-time* of a system is defined as the time elapsed between the first time the step response reaches 10% of its final value and the last time the step response equals 90% of its final value. Estimate the (10%-90%) rise-time of this system.

7. *Using Matlab for impulse, step, and Bode plots.*

In Matlab, it's very easy to plot the impulse response, step response, or frequency response of a transfer function $H(s) = b(s)/a(s)$ when b and a are polynomials. We will the two Matlab code examples below for illustration:

```
num = [ 0.3, 1 ]
den = [ 1, 0.3, 1 ]
impulse( num, den )
print
step( num, den )
print
```

```
num2 = 3 * poly( [ -3, +1 ] )
den2 = poly( [ 1+3*j,1-3*j,-2 ] )
bode( num2, den2 )
print saveplot
```

First, you have to know how to enter polynomials into Matlab. Polynomials are represented by a vector of their coefficients, stored in descending order of the exponent. In other words, the polynomial

$$b(s) = b_m s^m + b_{m-1} s^{m-1} + \cdots + b_1 s + b_0$$

is represented by the row vector

$$[\ b_m\ b_{m-1}\ \cdots\ b_1\ b_0\ ]$$

in Matlab. So in the above examples, `num = [0.3,1]` represents $0.3s + 1$, and `den = [1,0.3,1]` represents $s^2 + 0.3s + 1$.

You can also create a polynomial from its roots using the `poly` command. To do so, you simply supply `poly` with a vector of the roots, stored in any order. For example, the line `den = poly([1+3*j,1-3*j,-2])` above is equivalent to `den = [1,0,6,20]` (try it and see!) Likewise, `num = 3 * poly([-3,+1])` is equivalent to `num = [3,6,-9]`.

It's quite easy to find the roots of a polynomial in Matlab; just type `roots(`*<poly>*`)` to get a vector that contains the roots of the polynomial *<poly>*. In the first example above, typing `roots(den)` would produce

```
ans =    -0.1500 + 0.9887i   -0.1500 - 0.9887i
```

Of course, for many polynomials you can find the roots yourself. But Matlab can compute the roots of a 20th order polynomial very quickly, and you can't.

Once you have constructed the numerator and denominator polynomials in the above fashion, it is quite simple to make some useful plots with them. Above, we have used:

- `impulse(num,den)` to generate an impulse response plot,
- `step(num,den)` to generate a step response plot, and
- `bode(num2,den2)` to (you guessed it!) generate a Bode plot (both magnitude and phase).

There is also an analogous command `nyquist` for making Nyquist plots.

To print the current plot, you can usually just type `print`. To choose a specific printer, however, type `print -P`<*printername*> instead. And to save a plot to the PostScript file <*filename*>`.ps` to print later, type `print` <*filename*>.

If you want to explore Matlab a bit more, type `demo` at the prompt; for help on any particular command, type `help` <*command*>.

Once you start using Matlab for these chores, you might resent the fact that we ask you to know how to (approximately) sketch these plots by hand. The important part is that you *understand* the plots, what they mean, and how they relate. (As far as we know, Matlab does not understand what the plots mean ...)

Go ahead and run Matlab (on a UNIX workstation, typing `matlab` will do the trick). Try out this examples above; but save a tree—don't `print` just yet! Once you get the feel of it, move on to the problem:

(a) Use Matlab to plot the Bode plot of the transfer function of problems 1 and 2, to verify your sketches.

(b) Consider the transfer function $H(s) = (s+1)/(s^2+s+1)$. Find the poles and zeros, and plot the impulse response, step response, and Bode plot using Matlab.

(c) Now consider the transfer function

$$G(s) = H(s)\frac{s+3}{s+3.1}.$$

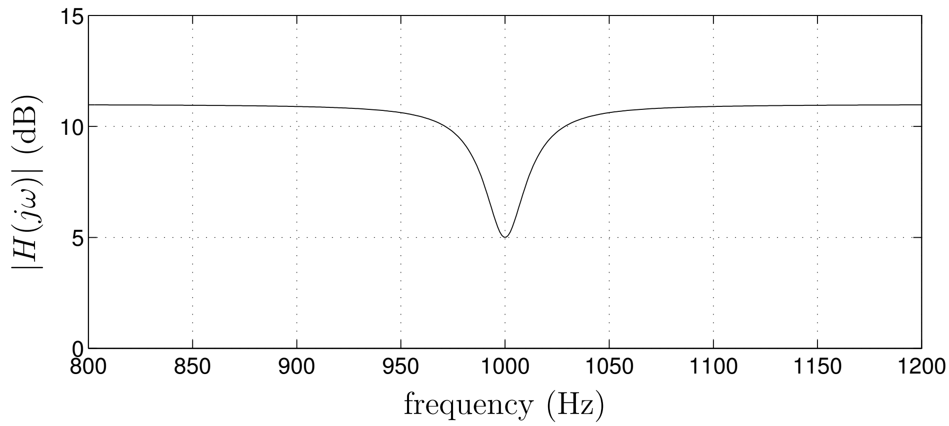
Intuition suggests that G is not much different from H since we have added a pole and a zero that *almost* cancel each other out. *Before* doing the next part, *guess* how the Bode plots of G and H will differ. Give a geometric explanation. Give the partial fraction expansion of H, and compare it to the partial fraction expansion of G.

(d) Now use Matlab to plot the impulse response, step response, and Bode plot of G using Matlab. Compare with your prediction.

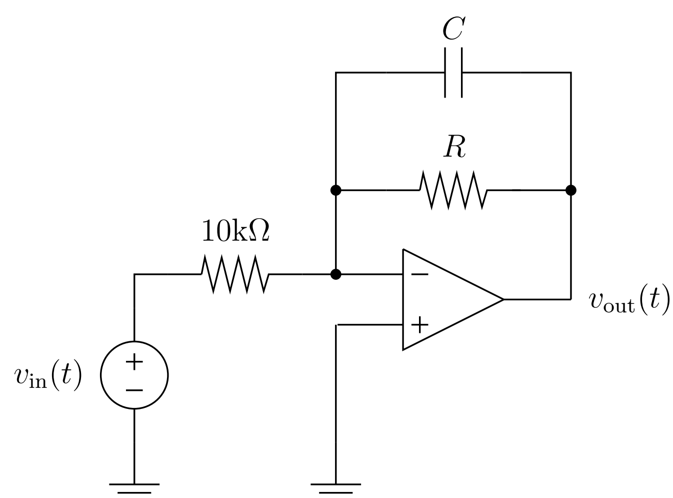
8. *Notch filter design.* Find a transfer function H that has the Bode magnitude plot shown below. Note that the vertical axis is given in dB and the horizontal axis, which is linear, is given in Hz.

   Express your answer as the ratio of two unfactored polynomials. *Justify* your choice of poles and/or zeros. An accuracy of $\pm 10\%$ for the coefficients is acceptable.

   Once the design is complete, use Matlab to create a full (magnitude and phase) Bode plot for the filter.



9. The circuit below is a simple one-pole lowpass filter.

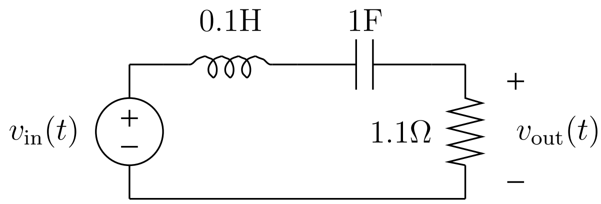


   Find (positive) R and C such that:

   - The (magnitude of the) DC gain is +12dB.
   - The magnitude of the transfer function at the frequency 1kHz is 3dB less than the magnitude of the DC gain.

You can assume the op-amp is ideal. Give numerical values for R and C. Since you can't use a calculator, an accuracy of 10% will suffice.

10. For parts (a)–(e) you can assume the circuit is initially relaxed, *i.e.*, the capacitor voltage and the inductor current are both zero at $t = 0$.



   (a) Find the transfer function H from v_{in} to v_{out}. Please check your answer carefully since other parts of this problem may depend on it. Try to express H in simple form.

   (b) Find the poles, zeros, and DC gain of H.

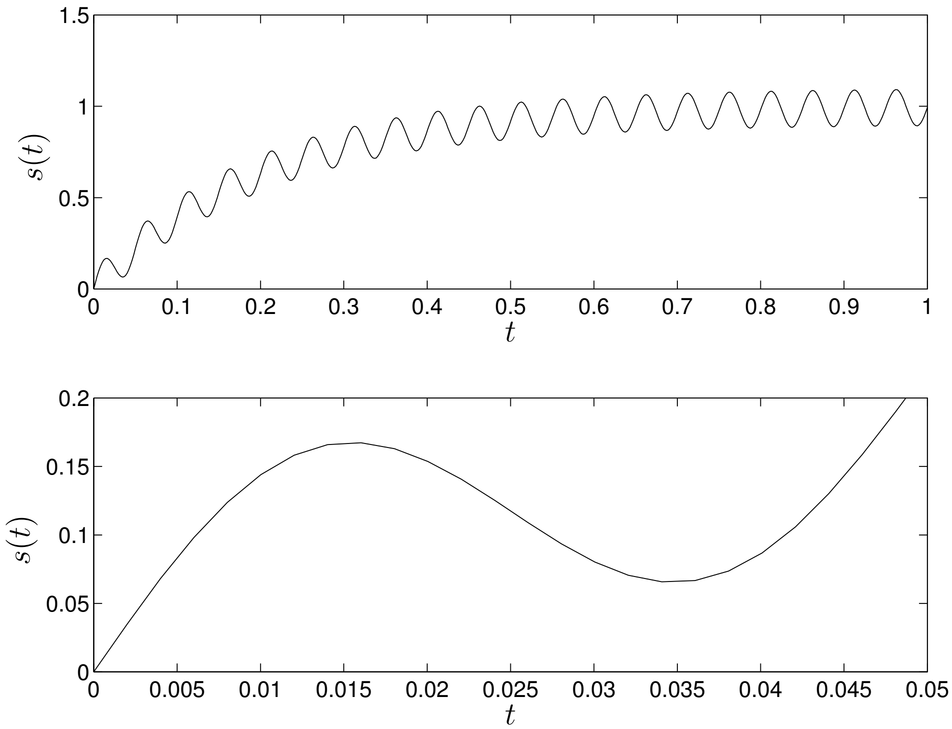
   (c) Find the unit step response $s(t)$ from v_{in} to v_{out}.

   (d) Sketch the Bode magnitude and phase plot of H at the bottom of this page. Be sure to clearly label the axes and key features of your plot.

   (e) Suppose that the input voltage is constrained to have a peak value less than one, *i.e.*, $|v_{\text{in}}(t)| \leq 1$. How large can $y(3)$ be? Briefly give your reasoning.

   (f) Suppose that $v_{\text{in}}(t)$ is a periodic and the circuit is in periodic steady-state. (We no longer assume that the capacitor voltage and inductor current are zero at $t = 0$.) Suppose that the RMS value of v_{in} is less than one. How large can the RMS value of v_{out} be? Briefly give your reasoning.

11. The unit step response $s(t)$ of a system described by a transfer function H, which has three poles, is shown in the two plots below. The two plots have different ranges; the second plot allows you to see details for small t.

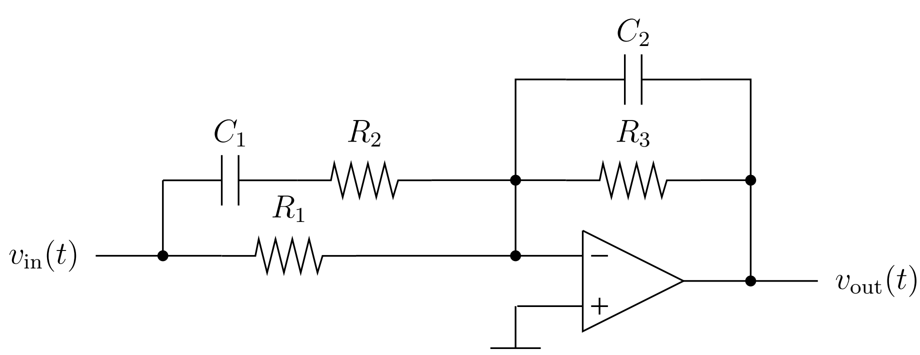


(a) Estimate the poles. An accuracy of $\pm 20\%$ is acceptable.

(b) At high frequencies $|H(j\omega)|$ becomes small. From the data given, can you determine the rate at which it decreases for large frequency (*e.g.*, 12 db/octave)? Either give the rate (in dB/octave) or state "cannot determine" if the data given is not sufficient to determine the high-frequency rolloff rate.

12. *An op-amp filter circuit.*

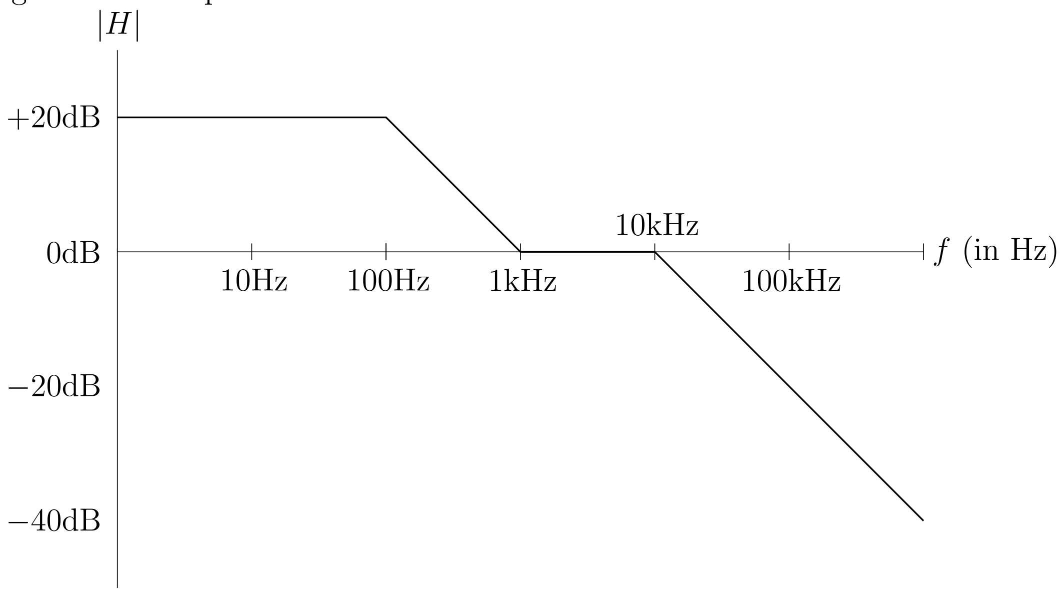
This problem concerns the filter circuit shown below. The voltages v_{in} and v_{out} are with respect to ground, and the op-amp is ideal. The transfer function from v_{in} to v_{out} will be denoted H.



(a) Find the DC gain, poles, and zeros of H. (Express them in terms of the component values R_1, R_2, R_3, C_1, and C_2.) If there are no zeros (or poles), give your answer as ‘none’. Express your answers in a simple form, and check them carefully, since you may want to use them in parts b and c.

(b) Suppose that $R_1 = R_2 = R_3 = 1\Omega$ and $C_1 = C_2 = 1\text{F}$. Find the unit step response $s(t)$ of the filter. (Assume zero initial voltage across C_1 and C_2.)

(c) *A filter synthesis problem.* For an audio application a filter is required with the magnitude Bode plot shown below:



For this application, the phase of H does not matter.

The resistor R_3 is fixed to be $10\text{k}\Omega$. Find (numerical, explicit values for) R_1, R_2, C_1, and C_2 so that the magnitude Bode plot of H matches (at least approximately) the required form shown above. (Needless to say, you cannot use negative values for R_1, R_2, C_1, and C_2.)

Exercises on Laplace Transform

1. Find the Laplace transform of the following functions.

   (a) $f(t) = (1 + t - t^2)e^{-3t}$.

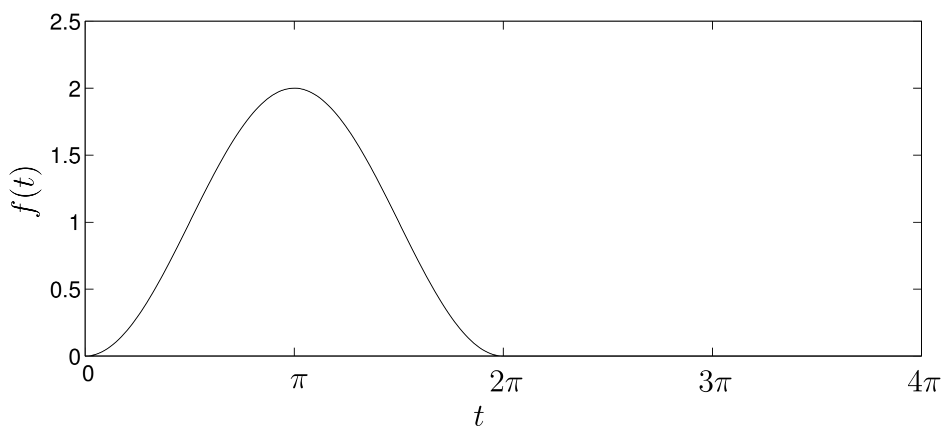
   (b) $f(t) = \begin{cases} 0 & 0 \le t < 1 \\ 1 & 1 \le t < 2 \\ -1 & 2 \le t \end{cases}$

   (c) $f(t) = 1 - e^{-t/T}$ where $T > 0$.

2. The "raised cosine pulse" is a signal used in applications such as radar and communications. It is defined by

$$f(t) = \begin{cases} 1 - \cos t & 0 \le t \le 2\pi \\ 0 & t > 2\pi \end{cases}$$

   and plotted below.

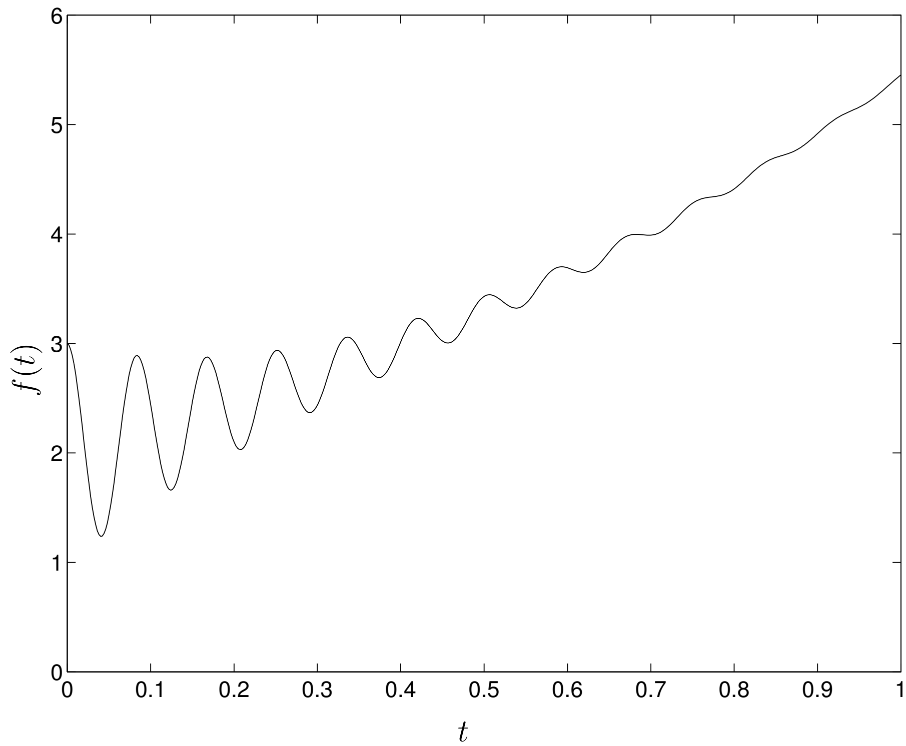


   Find F, the Laplace transform of f.

3. Can you find the Laplace transform of a function that is periodic for $t \ge 0$, given its Fourier series? (The expression you write down should be explicit, if not easy to work with or use ...)

4. The plot below shows $f(t)$. Its Laplace transform, F, has three poles. *Estimate* the specific numerical values of the poles. An accuracy of $\pm 30\%$ is sufficient.

   Note: We are looking for three specific complex numbers, *not* just qualitative descriptions of the pole locations! You do not have to estimate the residues.



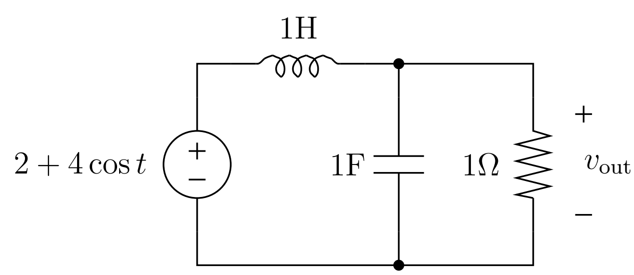
5. Solve the following differential equations using Laplace transforms. Verify that the solution you find satisfies the initial conditions and the differential equation.

   (a) $dv/dt = -2v + 3$, $v(0) = -1$.

   (b) $d^2i/dt^2 + 9i = 0$, $i(0) = 1$, $di/dt(0) = 0$.

6. *Time scale property of Laplace transform.* Let $\alpha > 0$ and define g by $g(t) = f(\alpha t)$. Pick some particular waveform f, and plot it and the corresponding waveform g for $\alpha = 1/2$. Repeat for $\alpha = 2$. Returning to the general case, find the Laplace transform of g in terms of the Laplace transform of f. Check your result by finding the Laplace transform of $\cos \omega_0 t$, using the Laplace transform of $\cos t$ derived in class.

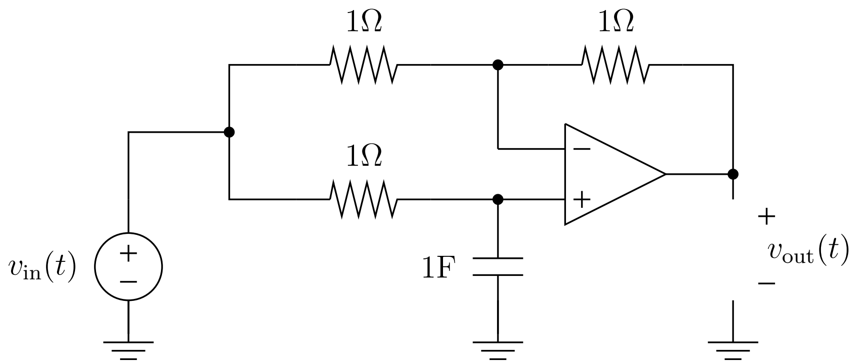
7. Consider the circuit shown below.



   (a) Suppose that the circuit is in periodic steady-state. Find $v_{\text{out}}(t)$.

(b) Suppose that the inductor current and capacitor voltage at $t = 0$ are both zero. Find $v_{\text{out}}(t)$. Does $v_{\text{out}}(t)$ converge to the periodic steady-state solution you found in part (a), as $t \to \infty$? What are the poles of the Laplace transform of v_{out}?

8. In the circuit shown below, the op-amp is ideal and the initial capacitor voltage (*i.e.*, at $t = 0$) is zero.



(a) Find the transfer function $H(s)$ from v_{in} to v_{out}.

(b) Now assume that $v_{\text{in}}(t) = e^{-2t}$ for $t \geq 0$. Find $v_{\text{out}}(t)$.

9. Suppose that f satisfies $d^3f/dt^3 = f$, $f(0) = 1$, $df/dt(0) = d^2f/dt^2(0) = 0$. Find $f(t)$.

10. *Positive real zeros and sign changes in f.* Suppose that $F(z) = 0$ for some real, positive z. You may assume that z is such that the defining integral for the Laplace transform converges. Show that f must change sign, *i.e.*, assume both negative and positive values at various times. Another way to say this is, f cannot be nonnegative for all $t \geq 0$ or nonpositive for all $t \geq 0$.

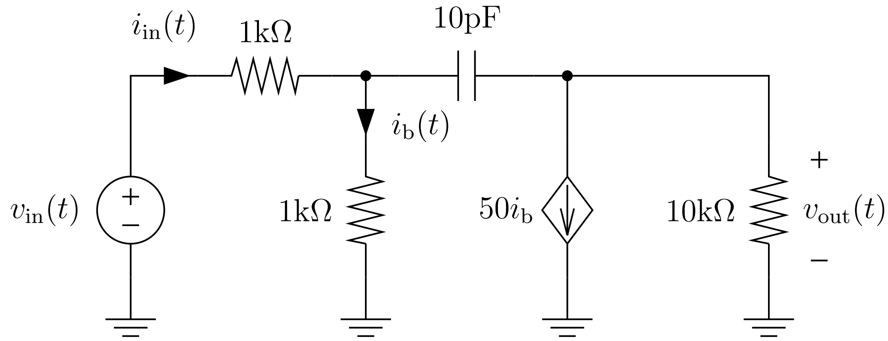
11. For each of the following rational functions, find the poles and zeros (giving multiplicities of each), the real factored form, the partial fraction expansion, and inverse Laplace transform. (In some cases, the expression may already be in one of these forms.)

(a) $\dfrac{1}{s+1} + \dfrac{1}{s+2} + \dfrac{1}{s+3}$

(b) $\dfrac{s^2+1}{s^3-s}$

(c) $\dfrac{(s-2)(s-3)(s-4)}{s^4-1}$

12. The circuit below is a small signal model of a typical transistor amplifier. You can assume zero initial voltage across the capacitor.

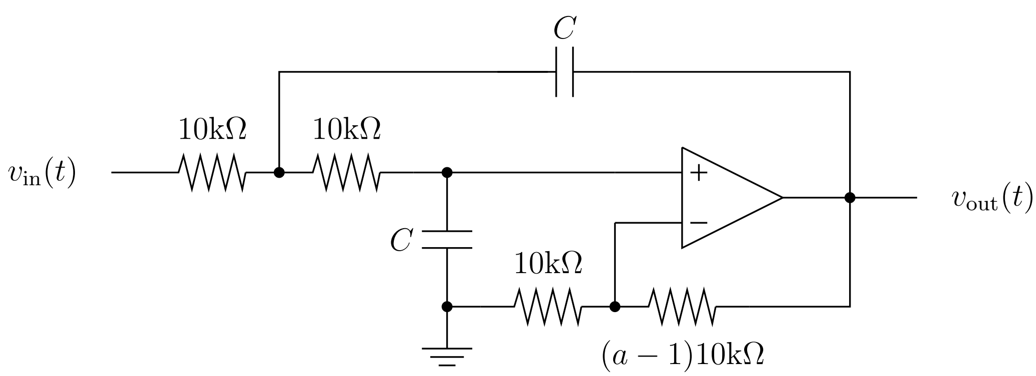


(a) Find the transfer function from v_{in} to v_{out}. (This is usually simply refered to as the transfer function of the amplifier.) Find its poles and zeros.

(b) Find $Z_{\text{in}}(s)$, the input impedance of the amplifier, defined as the ratio of $V_{\text{in}}(s)$ to $I_{\text{in}}(s)$. (Z_{in} is the transfer function from i_{in} to v_{in}.)

(c) Suppose $v_{\text{in}}(t) = 10\text{mV}$ for $t \geq 0$ (*i.e.*, the input is a 10mV step at $t = 0$). Find $v_{\text{out}}(t)$. How long does it take before v_{out} settles to within 90% of its limiting value?

13. The circuit below, called a Sallen-Key filter section, is widely used. You can assume the op-amp is ideal, and both capacitors have zero initial voltage. Note that there are two free design parameters: the capacitance C (which of course must be positive) and the (gain) a, which is required to satisfy $a \geq 1$.



(a) Find the transfer function from v_{in} to v_{out}.

(b) Pick C and a to yield poles at $(-10^4 \pm j10^4)$rad/sec.

(c) Suppose we hook up two of the filters you designed in part (b) in cascade, *i.e.*, connect v_{out} of one to v_{in} of the other. Find the transfer function from the remaining input to the remaining output.

(d) Continuing part (c), suppose that $v_{\text{in}}(t) = 1\text{V}$, *i.e.*, a step of one volt is applied at $t = 0$ to the (free) input. Find the output voltage.

14. *Stability analysis of a general second order system.* Consider the transfer function

$$H(s) = \frac{b(s)}{a(s)}, \qquad b(s) = b_1 s + b_0, \qquad a(s) = a_2 s^2 + a_1 s + a_0.$$

You can assume that b and a have no common roots and $a_2 \neq 0$, so the degree of a is two. What are the conditions on a_0, a_1, a_2, b_0, and b_1 under which this system is stable? Try to express your answer in the simplest form. **Hint 1:** in your analysis you'll have to consider the cases $a_1^2 \geq 4a_0a_2$ and $a_1^2 < 4a_0a_2$ separately. **Hint 2:** the answer is very simple; it can expressed in one short sentence.

15. *Stability analysis of the Sallen-Key filter.* Consider the Sallen-Key filter studied in problem 13.

    (a) Find the values of C and a that render the filter stable. If all values of a and C result in stability, say so. If not, find the conditions that ensure stability. You can assume that $C > 0$ and $a \geq 1$.

    (b) The actual resistance of a real (physical) resistor varies a little bit from its stated or *nominal* value. The maximum deviation is called the *tolerance.* Some typical tolerances are $\pm 20\%$, $\pm 10\%$, $\pm 5\%$, and $\pm 1\%$. Components with small tolerances are more expensive. For example, a (physical) 1kΩ, $\pm 20\%$ resistor can have a real resistance anywhere between 800Ω and 1200Ω, but is less expensive than a 1kΩ, $\pm 1\%$ resistor, which has a resistance between 990Ω and 1010Ω.

    Now suppose you design a Sallen-Key filter with poles at $-100 \pm 10^4 j$. How accurate (*i.e.*, what tolerance) does the feedback resistor (*i.e.*, the one with value $(a-1)10\text{k}\Omega$) have to be ensure that the real filter remains stable? Express your answer as a percentage of its value. Any comments?

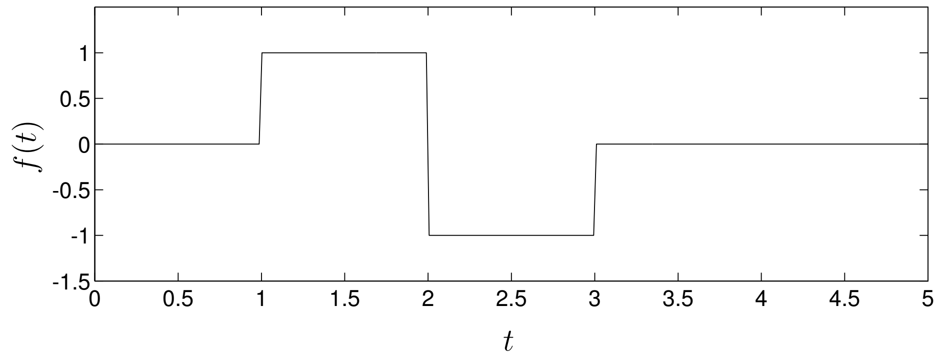
    To simplify your analysis, you can assume that all the other resistors (and the capacitors) are perfect, *i.e.*, have zero tolerance. It would be more difficult, but more correct, to take into account the variations in *all* component values; this is what is done in practice.

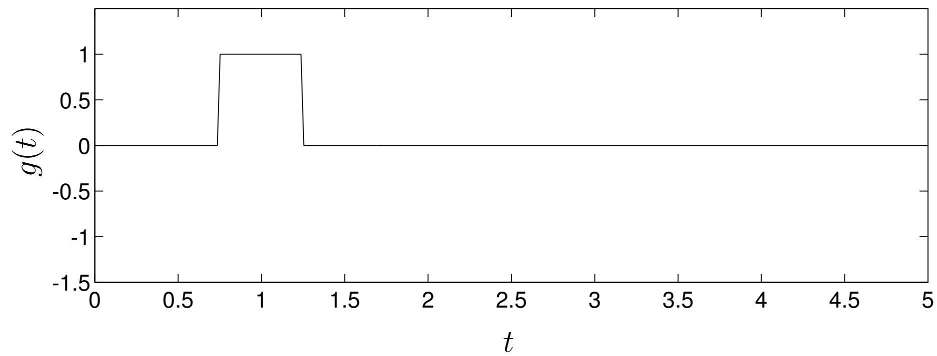
    How do you think this filter would behave as the temperature varies between $-10°$C and $+50°$C? (The value 23°C is often used as the nominal temperature, *i.e.*, the temperature at which a component is supposed to have its nominal value.) The temperature coefficient for a typical composition resistor is on the order of $-0.15\%/°$C. There are special (and more expensive) resistors that have a much smaller temperature coefficient.

16. What is $e^{-t} * e^{-2t}$? (These signals are not defined for $t < 0$.) Do this two ways: via Laplace transforms and also via direct integration. Sketch the two signals and their convolution.

    Repeat for $e^{-t} * (3\delta(t-1) - 2\delta(t-3))$.

17. The signals f and g are plotted below. Plot $f * g$.





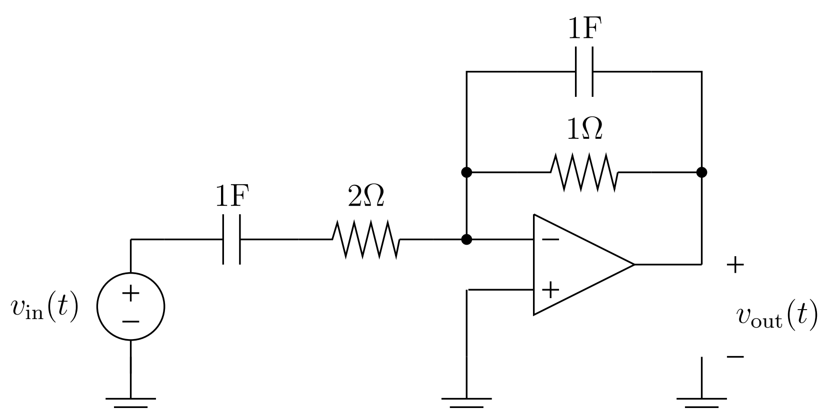
18. So, you thought problem 12 was safely behind you ...

    (a) Find the impulse response of the amplifier in problem 12 (from v_{in} to v_{out}).

    (b) Find the DC gain of the amplifier in problem 12 (from v_{in} to v_{out}). Verify that it is consistent with the transfer function and the final value of the response to the 10mV step you found. Verify also that your DC gain is consistent with a static analysis of the amplifier circuit.

    (c) Repeat parts (a) and (b) for the Sallen-Key filter. (Just the single filter, not the cascaded version.)

19. In the circuit shown below you may assume the op-amp is ideal, and the voltage across each of the capacitors is zero at $t = 0$.

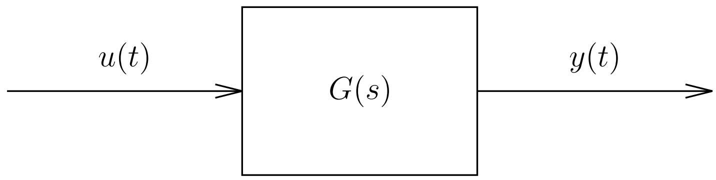


(a) Find the transfer function H from v_{in} to v_{out}. Try to express H in simple form.

(b) Find the poles, zeros and DC gain of H.

(c) Suppose that $v_{\text{in}}(t) = 1$ for $t \geq 0$. Find $v_{\text{out}}(t)$.

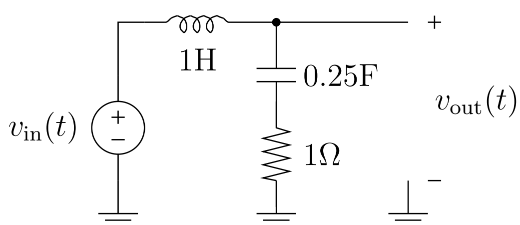
20. The system shown below is described by a transfer function G. The poles of G are at $s = -1$ and $s = -4$; G has only one zero, at $s = -2$. The DC gain of G is 1.



(a) Find the impulse response $g(t)$ of this system.

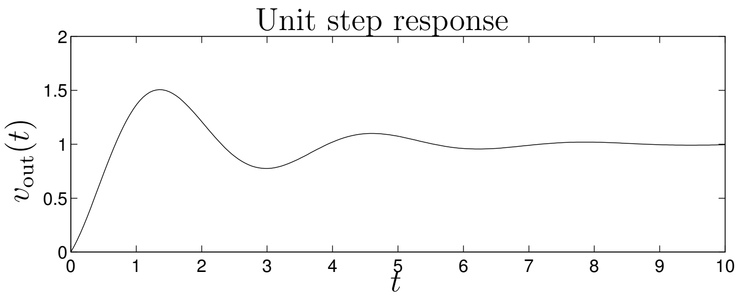
(b) Suppose that $u(t) = e^{-2t}$ for $t \geq 0$. Find $y(t)$.

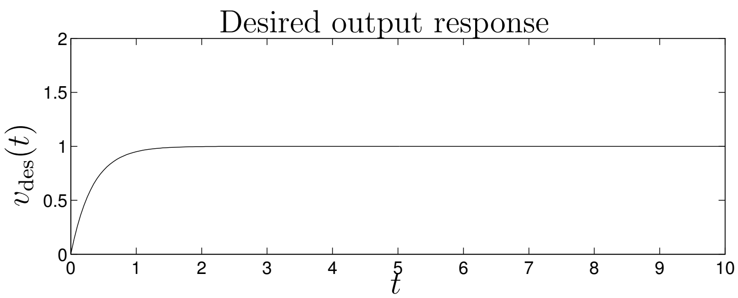
21. Consider the circuit shown below. You can assume the capacitor voltage and the inductor current are zero at $t = 0$.



Two plots are shown below. The top plot shows the unit step response from v_{in} to v_{out}, *i.e.*, $v_{\text{out}}(t)$ with $v_{\text{in}}(t) = 1$. Note that it exhibits some ringing, *i.e.*, oscillation, and settles (converges) in about 5sec or so.

The bottom plot shows the *desired* output voltage, which is $v_{\text{des}}(t) = 1 - e^{-3t}$. Note that it exhibits no oscillation and settles quite a bit faster than the step response, *i.e.*, in about 1sec.





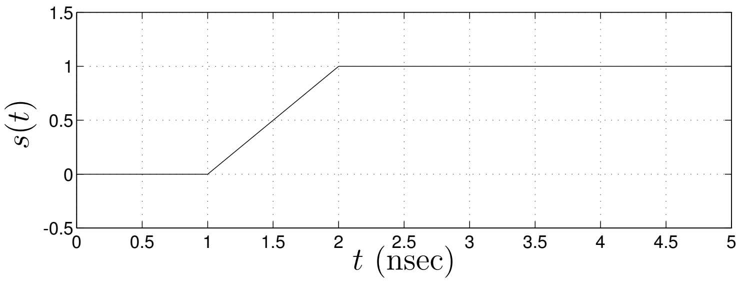
Finally, the problem: can you find an appropriate $v_{\text{in}}(t)$ such that we have $v_{\text{out}} = v_{\text{des}}$? If there is no such v_{in}, give your answer as "impossible". Otherwise, give v_{in} that yields $v_{\text{out}} = v_{\text{des}}$.

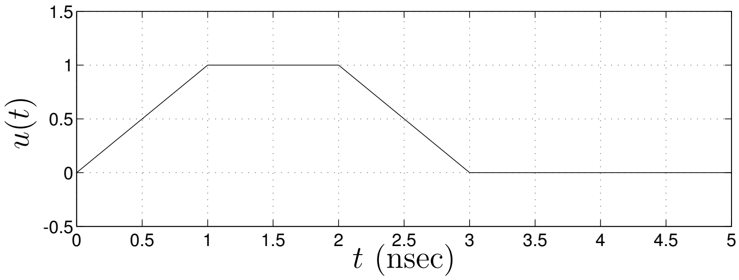
22. An engineer is looking for a function v that satisfies

$$\frac{d^4v}{dt^4} - v = 0, \qquad v(0) = 2, \qquad \lim_{t\to\infty} v(t) = 0.$$

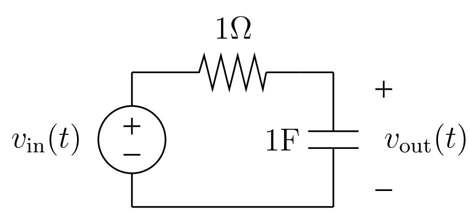
What can you say about such a v? If you believe no such v exists, give your answer as "impossible". If you can give v expicitly, do so. If you can give a qualitative description of what such a v would look like, do so. Give the most specific answer you can.

23. The top plot below shows the step response of a system described by a transfer function. Below that is a plot of an input $u(t)$ that we apply to this system. Sketch the response (output) $y(t)$.

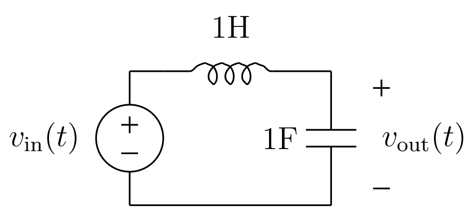




24. In the circuit at right, $v_{\text{out}}(0) = 0$ and $v_{\text{in}}(t) = 1 - e^{-2t}$ for $t \geq 0$.
    Find the Laplace transform $V_{\text{in}}(s)$ of $v_{\text{in}}(t)$.
    Find the output voltage, $v_{\text{out}}(t)$, for $t \geq 0$.
    (**Not** just its Laplace transform.)



25. Consider the L-C filter circuit shown below.



Part 1. Find the transfer function $H(s)$ from v_{in} to v_{out}.

Part 2. Find the unit step response $s(t)$ from v_{in} to v_{out}.

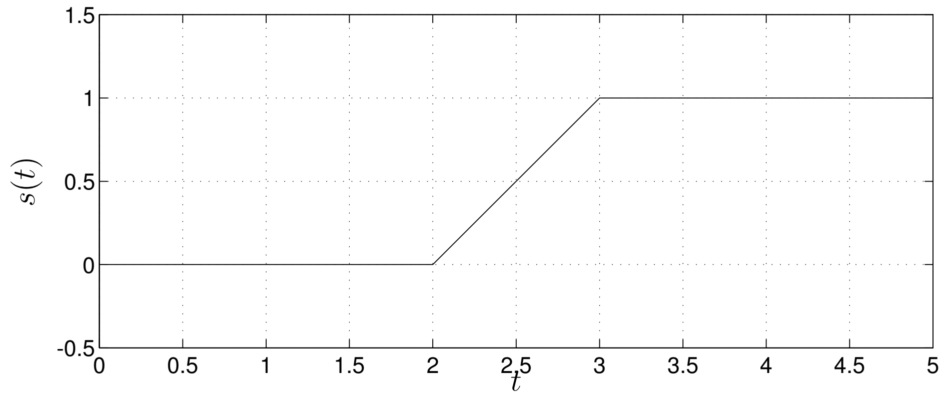
26. Suppose that the raised cosine pulse in problem 2 is applied as the input to the system of question 6. Discuss what the output will look like, as a function of the parameter T.

    For example, for T large enough, y will also have a shape close to a raised cosine pulse. Give more details, *e.g.*, how large does T have to be? What is the approximate maximum amplitude of the output? At what time does this maximum occur?

What happens if T is very small? What does the output look like then? Roughly how small does T have to be for your analysis to hold?

When you can, give your discussion in both the time and frequency domain.

27. The plot below shows the unit step response $s(t)$ of a system described by a transfer function.



Suppose the input $u(t) = \sin(2\pi t)$ for $t \geq 0$ is applied to this system. Sketch the resulting output at the bottom of this page. Make sure you clearly label the axes and key features of your plot.

28. *Transfer function from rainfall to river height.*

The height of a certain river depends on the past rainfall in the region. Specifically, let $u(t)$ denote the rainfall rate, in inches-per-hour, in a region at time t, and let $y(t)$ denote the river height, in feet, above a reference (dry period) level, at time t. The time t is measured in hours; we'll only consider $t \geq 0$.

Analysis of past data shows that the relation between rainfall and river height can be accurately described by a transfer function:

$$Y(s) = H(s)U(s), \quad H(s) = \frac{10}{(3s+1)(30s+1)}$$

(You don't need to know any hydrology to do this problem, but you might be interested in the physical basis of this two-pole transfer function. The fast pole is due to runoff from surface water and small tributaries, which contribute a relatively small amount of water relatively quickly. The slow pole is due to flow from larger tributaries and deeper ground water, which contribute more water into the river, over a much longer time scale.)

A brief but intense downpour. (Parts a and b.) Suppose that after a long dry spell (*i.e.*, no rain) it rains intensely at 12 inches-per-hour, for 5 minutes. This causes the river height to rise for a while, and then later recede.

(a) How long does it take, after the beginning of the brief downpour, for the river to reach its maximum height? We'll denote this delay as $t_{\max}$ (in hours).

(b) What is the maximum height of the river? We'll denote this maximum height as $y_{\max}$ (in feet).

Note: you can make a reasonable approximation provided you say what you are doing.

A continual rain. (Parts c and d.) Suppose that after a long dry spell it starts raining continuously at a rate of 1 inch-per-hour (and doesn't stop). This causes the river height to rise.

(c) What is the ultimate height of the river, *i.e.*, $y_{\text{ult}} = \lim_{t\to\infty} y(t)$?

(d) A flood occurs when the river height $y(t)$ reaches 8 feet. How long will it take, after the onset of the steady rain, to reach flood condition? We'll denote this time as t_{flood}. If the river never reaches 8 feet, give your answer as 'never'.

Note: you can make a reasonable approximation provided you say what you are doing.

29. *Reducing the rise-time of a signal.*

In a certain digital system a voltage signal should ideally switch from 0V to 5V infinitely fast, *i.e.*, with zero rise-time. But due to the finite bandwidth of the electronics that generates the signal, it has the form

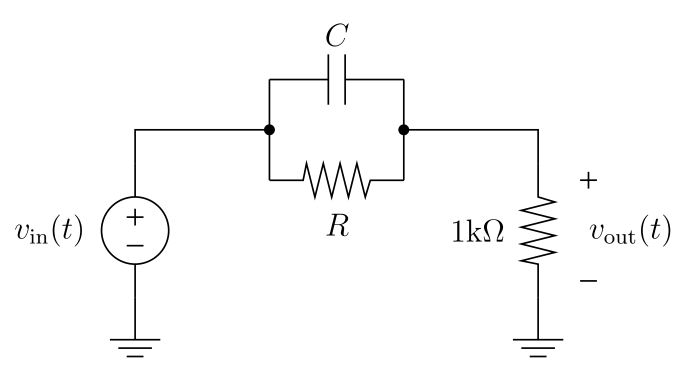
$$v_{\text{in}}(t) = 5\left(1 - e^{-t/T}\right) \quad \text{for } t \geq 0$$

where $T = 1\mu$sec. Thus, the signal has a rise-time around a few μsec.

An engineer claims that the circuit shown below can be used to reduce the rise-time of the signal, provided the component values R and C are chosen correctly. Specifically, the engineer claims that by choosing R and C correctly, we can have

$$v_{\text{out}}(t) = a\left(1 - e^{-10t/T}\right) \quad \text{for } t \geq 0$$

where a is some nonzero constant. Thus, the rise-time of v_{out} is a factor of 10 smaller than the rise-time of v_{in}, *i.e.*, a few hundred nsec.



Here is the problem: determine whether the engineer's claim is true or false. If the claim is true, find specific, numerical values of R and C that validate the claim. If the claim is false, briefly explain why the engineer's idea will not work.

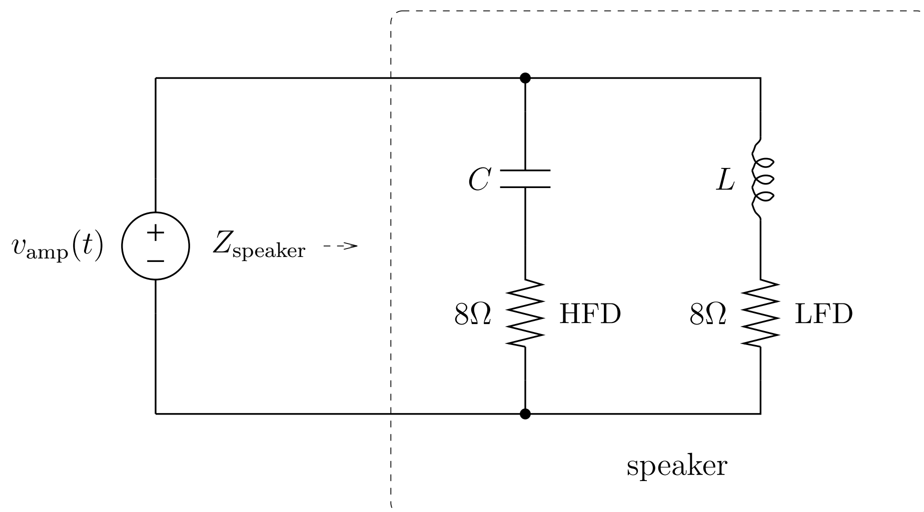
(You can assume the circuit starts in the relaxed state, *i.e.*, no charge on the capacitor. And no, you cannot use negative R or C.)

30. *A simple two-way crossover circuit.*

A typical high-fidelity speaker has separate drivers for low and high frequencies. (The driver is the physical device that vibrates to create the sound you hear. The old terms for the low and high frequency drivers are *woofer* and *tweeter*, respectively.)

The circuit shown below, called a *speaker crossover network*, is used to divide the audio signal coming from the amplifier into a low frequency part for the low frequency driver (LFD) and a high frequency part for the high frequency driver (HFD). Since the audio spectrum is divided into two parts, this is called a two-way system (three-way are also common).

The amplifier is modeled as a voltage source (which is a very good model), and the low and high frequency drivers are modeled as 8Ω resistances (which is not a good model of real drivers, but we will use it for this problem).



The crossover network is designed so that the transfer function from the amplifier to each driver has magnitude -3dB at a frequency ω_c called the *crossover frequency* of the speaker.

(a) Choose C and L so that the crossover frequency is 2kHz. Do this carefully as you will need your answers in part b.

(b) Using the values found in a, find $Z_{\text{speaker}}(s)$, the impedance of the two-way speaker seen by the amplifier (as indicated in the schematic).

Exercises on Fourier Transform

1. Find the Fourier transform of the following signals. In each case sketch a plot of the signal and its spectrum, *i.e.*, the magnitude-squared of its Fourier transform. For each signal give a rough idea of its "time-width" and "bandwidth".

   (a) $f(t) = te^{-|t|}$. *Hint:* try to use Laplace transforms ...

   (b) The signal

   $$f(t) = \begin{cases} \sin 10\pi t & 0 \le t \le 1 \\ 0 & t > 1 \text{ or } t < 0 \end{cases}$$

   which is called a *tone-burst.*

   (c) $f(t) = e^{-0.2|t|} \cos 2\pi t$.

   (d) A raised cosine pulse of duration T, *i.e.*,

   $$f(t) = \begin{cases} 1 + \cos(2\pi t/T) & |t| \le T/2 \\ 0 & |t| > T/2 \end{cases}$$

2. Find the inverse Fourier transform of $e^{-|\omega|-j\omega}$.

3. Two signals u and y are related by

   $$u(t) = -y(t) + 2\int_0^\infty e^{-\tau} u(t-\tau)\, d\tau.$$

   (a) Express this relation in the frequency domain.

   (b) What can you say about the two quantities

   $$\alpha = \int_{-\infty}^{\infty} u(t)^2\, dt, \qquad \beta = \int_{-\infty}^{\infty} y(t)^2\, dt \quad ?$$

   For example, is one always less than the other, no matter what u is?

4. Consider the function

   $$f(t) = \begin{cases} e^t & t \le 0 \\ 0 & t > 0 \end{cases}$$

   (a) Find its Fourier transform, $\hat{f}$.

   (b) Let g denote the function

   $$g(t) = \frac{1}{2\pi} \int_{-2}^{2} \hat{f}(\omega) e^{j\omega t}\, d\omega,$$

   which can be considered an approximation of the Fourier inversion formula in which we integrate only over the frequency band $-2 \le \omega \le 2$ instead of all frequencies.

Let $e(t) = f(t) - g(t)$, *i.e.*, e is the error between f and the approximation g. Find the total energy of the error, *i.e.*,

$$E_{\text{err}} = \int_{-\infty}^{+\infty} e(t)^2 dt.$$

Is the function g a fairly good approximation of f? Why or why not?

5. *Fourier coefficients of convolution of periodic signals.*

   Suppose f and g are both periodic signals with period T. We define the convolution of f and g as

   $$h(t) = \int_0^T f(\tau)g(t-\tau)\, d\tau.$$

   The function h is also periodic with period T.

   Let c_k denote the kth complex Fourier coefficient of f (where $k = 0, \pm1, \pm2, \ldots$). Similarly, let d_k denote the kth complex Fourier coefficient of g, and let e_k denote the kth complex Fourier coefficient of h.

   How do you think e_k is related to c_k and d_k? First guess the relation, and then verify your answer.

6. *Fourier coefficients of product of periodic signals.*

   Here is yet another time/frequency domain relation we haven't encountered, but won't surprise you. Suppose f and g are both periodic signals with period T. Let h denote the product of f and g, *i.e.*,

   $$h(t) = f(t)g(t),$$

   which is also periodic with period T.

   Find an expression for the Fourier coefficients of h in terms of the Fourier coefficients of f and g (which we'll call c_k and d_k, respectively).

   Hint: If x_k and y_k are both sequences of real or complex numbers, then we define the convolution of the two sequences, denoted $z = x * y$, as

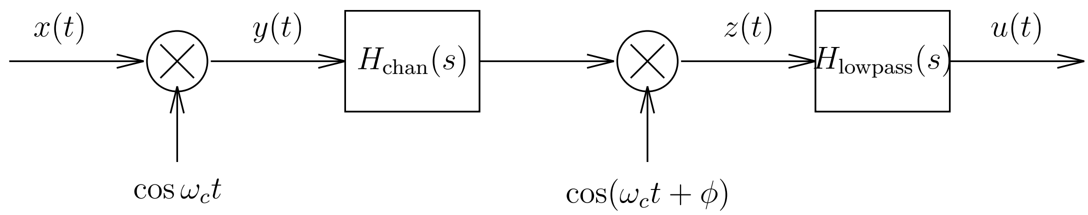
   $$z_k = \sum_{i=-\infty}^{\infty} x_i y_{k-i}$$

   (which shouldn't surprise you; it looks just like our definition of convolution of continuous signals, with summation substituted for integration).

7. *Analysis of synchronous demodulation including channel transfer function.*

   In the lecture notes we analyzed synchronous demodulation assuming the local oscillator signal was exactly the same as the carrier signal, $\cos \omega_c t$. We also assumed that the channel was just a wire, *i.e.*, the modulated signal was available directly to the demodulator. In this problem we (that is to say, you) analyze what happens when these assumptions don't hold.

As in the notes, $x(t)$ will denote the signal, which is bandlimited to W. The modulated signal will be $y(t) = x(t)\cos\omega_c t$, where $\omega_c > W$. The modulated signal then passes through the channel, which has transfer function H_{chan}. This signal is demodulated by multiplication by $\cos(\omega_c t + \phi)$, then lowpass filtering through H_{lowpass}. This is shown in the block diagram below.



For this problem we will make two simplifying assumptions. You can assume that H_{lowpass} is a perfect lowpass filter:

$$H_{\text{lowpass}}(j\omega) = \begin{cases} 1 & |w| \leq W \\ 0 & |w| > W \end{cases}$$

(although a more realistic analysis includes a non-ideal lowpass filter ...). You may also assume that over the band of frequencies between $\omega_c - W$ and $\omega_c + W$, the channel transfer is approximately constant, and equal to $H_{\text{channel}}(j\omega_c)$ (which is a complex number!). This second assumption is often realistic, if $W \ll \omega_c$, and the channel transfer function doesn't change too much with frequency.

(a) Find an expression for $Z(\omega)$ and $U(\omega)$. Your answer will involve the local oscillator phase angle ϕ and also the complex constant $H_{\text{channel}}(j\omega_c)$.

(b) Give an interpretation of what you found in (a). For example, how is $u(t)$ related to the input signal $x(t)$, and what effect does the phase angle ϕ have?

(c) Find the 'best' phase angle ϕ for the demodulator. You will have to first think about, decide, and explain what 'best' means here, and then solve the problem.

(d) Consider the case where the channel is a wire, *i.e.*, $H_{\text{channel}}(s) = 1$. What happens if the local oscillator is 90° out of phase with the carrier, *i.e.*, $\phi = \pm 90°$?

8. *Time and frequency widths of a signal.*

One specific definition of the time-width of a signal f is the smallest number T such that 90% of the total energy in the signal is contained in the time interval $[-T, T]$, *i.e.*,

$$\int_{-T}^{T} f(t)^2 \, dt = 0.90 \int_{-\infty}^{\infty} f(t)^2 \, dt.$$

In a similar way we can define the frequency width as the smallest number W such that 90% of the total energy in the signal is contained in the frequency interval $[-W, W]$, *i.e.*,

$$\frac{1}{2\pi}\int_{-W}^{W} |\hat{f}(\omega)|^2\, d\omega = 0.90\frac{1}{2\pi}\int_{-\infty}^{\infty} |\hat{f}(\omega)|^2\, d\omega,$$

where $\hat{f}$ denotes the Fourier transform of f.

Using these definitions, find the time width T and frequency width W for the signal

$$f(t) = \begin{cases} e^{-at} & t \geq 0 \\ 0 & t < 0 \end{cases}$$

where $a > 0$ is a (constant) parameter. (Obviously, your answer will depend on a.)

Hint: you may need the following indefinite integral:

$$\int \frac{1}{1+x^2}\, dx = \arctan x$$