Summer 2019

Laboratory Manual for Engineering Electronics

Sandip Das
Kennesaw State University, sdas2@kennesaw.edu

Walter Thain
Kennesaw State University, wthain@kennesaw.edu

Sheila Hill
Kennesaw State University, sdoneho1@kennesaw.edu

Open Textbook
Kennesaw State University

UNIVERSITY SYSTEM OF GEORGIA

Sandip Das, Walter Thain, and Sheila Hill

Laboratory Manual for Engineering Electronics

Affordable Learning Georgia

Georgia's Virtual Library GALILEO
An Initiative of the University System of Georgia

Table of Contents

Using LTspice

EE 3401 Laboratory Exercise
Electrical Engineering Department
Kennesaw State University

Objectives:

Students will learn how to use the LTspice circuit simulator, including schematic entry, selecting and running different simulation types, and how to produce simulation output for reports. Example circuits will be simulated to demonstrate the capabilities of LTspice.

Introduction:

LTspice is a fully-functional, freely-available circuit simulator. Linear Technology, Inc. originally designed it so engineers could simulate their switching power supply controller integrated circuits. It is an excellent SPICE simulator, rivaling costly commercial products like Electronic Workbench and PSpice. Some important advantages to LTspice are that it is free, circuit sizes are unlimited, it is very easy to add new models, and the user can easily modify the simulator's behavior. However, PSpice and Electronic Workbench have other advantages and are better at mixed analog/digital circuits than LTspice.

In this lab exercise you will:

- Examine the documentation
- Adjust the schematic window's user interface with control panel options
- Enter a simple schematic
- Explore the LTspice component library
- Run simulations for
  - dc operating point
  - ac small-signal frequency response
  - time-domain
- Format and create schematic and plot outputs for reports

Equipment:

Computer with LTspice installed.

Procedure:

Starting LTspice

1. Start LTspice by left clicking on the icon  . When it starts up, LTspice may ask if you would like to download an update. Do not do that now.

   The screen should look like Fig. 1 if you have a MS Windows PC.

   Note: Mac OS users do not have the toolbar under the menu bar.

   Note: For both Windows and MAC OS users by right-clicking on the main window and using that context-based menu.

   These instructions will use the context-based menu rather than the toolbar.

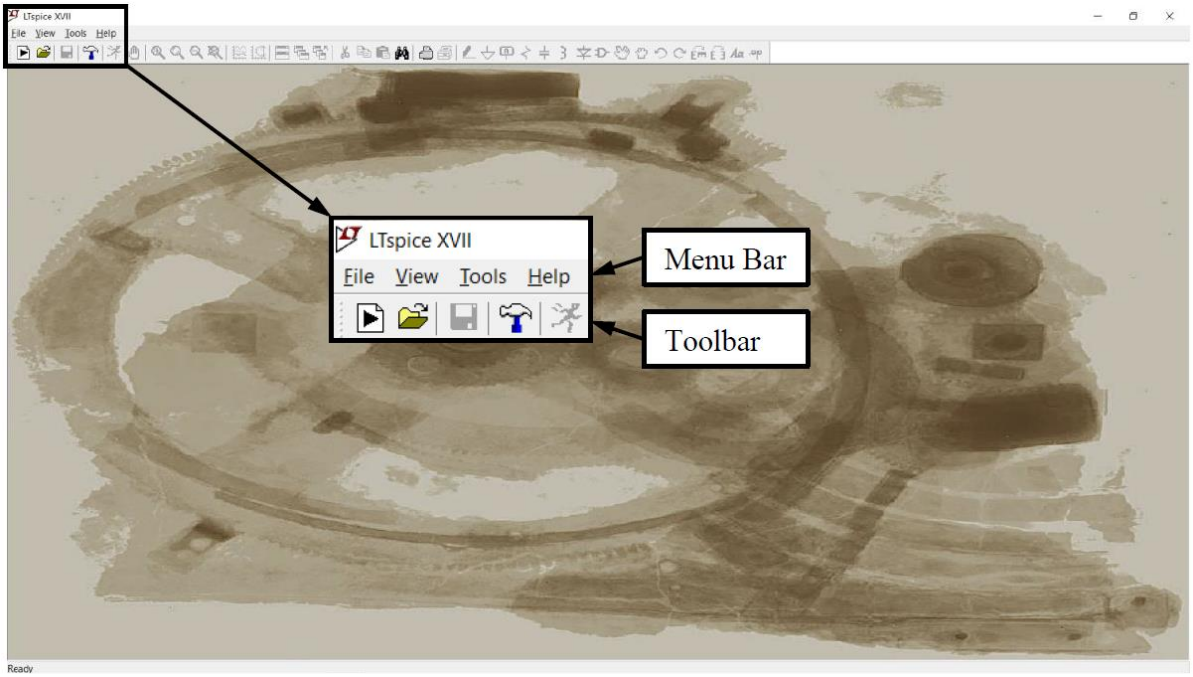


Fig. 1. MS Windows version startup window with expanded menu and toolbar insert.

Examine the Documentation

A quick overview of the help documentation follows. It serves as the manual but there are some other online resources and a user's group that can help with more advanced topics.

2. On the menu bar, left click Help > Help Topics and expand the LTspice XVII heading.

   Mac Users: Help > LTspice Help can be found on the File/Edit/Window/Help menu at the top of the screen.

   a. Expand the Schematic Capture topic.

      - Left click the first four subtopics so you can see what is covered.

   b. Expand the Waveform Viewer topic. This is the waveform plotting utility. Examine Data Trace Selection, Waveform Arithmetic, Axis Control, and Attached Cursors.

   c. Expand the LTspice topic (**Mac Users:** LTspice Simulator topic). This is where you find the detailed settings for simulations and the definitions of the circuit elements.

      - Expand Dot Commands (**Mac Users:** Simulator Directives). These are used to set simulation parameters and select the simulation types.
        - In most cases you choose basic simulation settings graphically using the menu or toolbar.
        - However, special commands are implemented by placing command text on the schematic itself. An example of using a dot command is adding a special model you want to use for one of your circuit elements.
      - Expand Circuit Elements. These are the basic circuit element building blocks used in circuits and subcircuits. The element options are explained in these subtopics.

d. Expand the Control Panel topic. These functions control simulator program operation, aspects of the user interface, color schemes for schematics and plots, and plotting parameters.

e. Close the Help window.

Adjust the Schematic Window User Interface

After opening a new schematic, you will make some useful adjustments to the default schematic graphical user interface settings. There are many other adjustments that you can make to suit your preference.

3. Left click on the menu bar File > New Schematic. Fig. 2 shows the default schematic window for MS Windows users. The menu and toolbar change to the one shown in the insert. There are more menu items.

   Mac Users: The Mac version does not have a toolbar. All actions are available by right-clicking on the window and selecting the desired action. These same actions are also available to Windows users.

   a. Left click Tools > Control Panel

   b. Left click the Drafting Options tab.

   - Left click the Reset to Default Values button so you can see what these settings are in case they have been changed.
   - The following changes will make schematics look better, particularly in reports
     - Under Pen thickness[*] drop down menu, select 4.

       Mac Users: Drafting > Line Thickness should be 2.

       Waveforms > Line Thickness should be 4

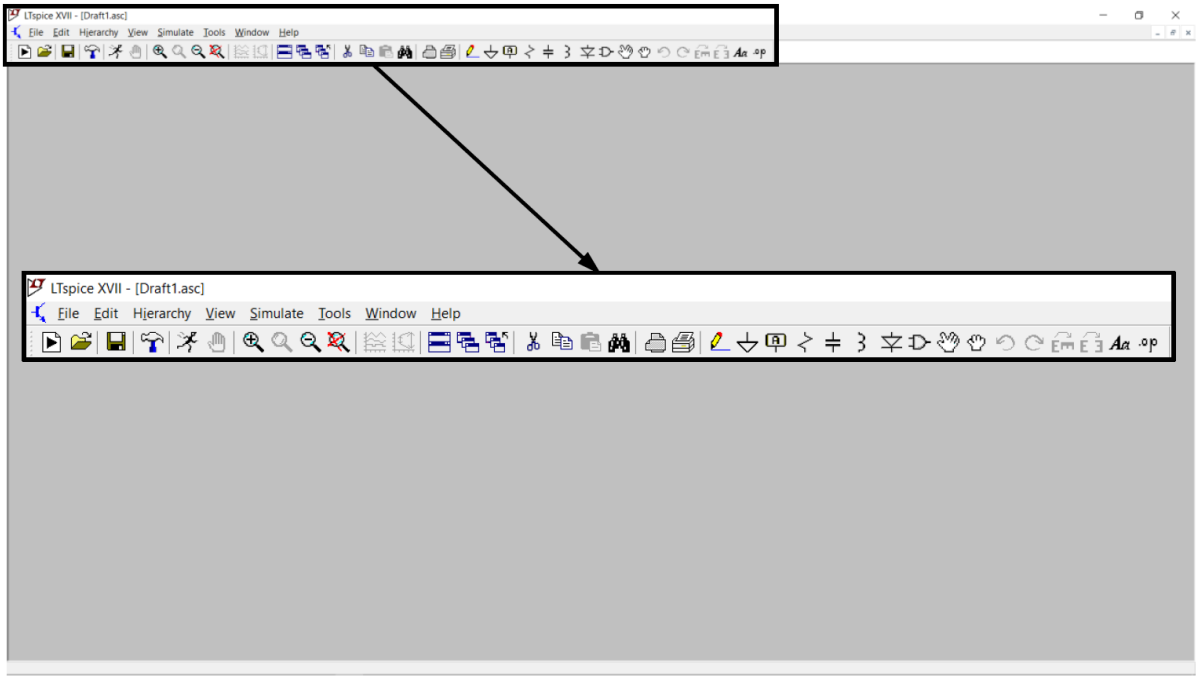


Fig. 2. Default schematic window with expanded menu and toolbar insert.

- The grey schematic background does not look good in reports if you copy the schematic to the clipboard, so change it to white as follows.
  - Left click Color Schemes[*] (Mac Users: Configure Colors).
  - In the Selected Item box, use the drop-down arrow to select Background.
  - Move the three RGB sliders all the way to the right so they are at 255.
  - Left click OK.

- Other settings you can change in the Drafting Options are:
  - You may want to change the font properties (type, size, or bold).

    Mac Users: There is no way to change font properties such as type, bold, italics, etc. Size can be changed by right-clicking an individual component name or value.
  - You may also want to select Show Schematic Grid Points while you are drawing a schematic, but these will show up in the schematic if you copy it to the clipboard.

    Mac Users: Schematic Grid Points are toggled on or off using the View menu after right-clicking the schematic to reveal the menu

c. Close the Control Panel.

Enter a Schematic

You will construct the circuit in Fig. 3 or one chosen by your instructor. The circuit is a simple model of an op amp inverting amplifier using standard SPICE elements.

4. Add components to the schematic as follows.

   Note: For MS Windows users resistors, capacitors, inductors, diodes, and ground elements have their own toolbar buttons. All other elements and models are accessed through the component library button (looks like an AND gate).

   a. Place the resistors.

   - Right click on the schematic and select Draft > Component. The Select Component Symbol window will appear as shown in Fig. 4. This directory level has all the standard LTspice elements. Select res followed by OK. A resistor will attach to the cursor.
   - Drag the resistor on the schematic and left click to place it. It doesn't matter where it is placed or how it is oriented since it will be moved later.
   - Move the mouse and left click to place the second resistor.
   - Place a third resistor. You will delete it shortly.
   - Right click to end the resistor placement (or press the ESC key).

   Note: The F2 key will activate the select component function without having to enter a menu.

   b. Deleting a component.

   - Right click on the schematic (not the resistor itself) and select Edit > Delete.
   - The cursor will change to a scissor icon.

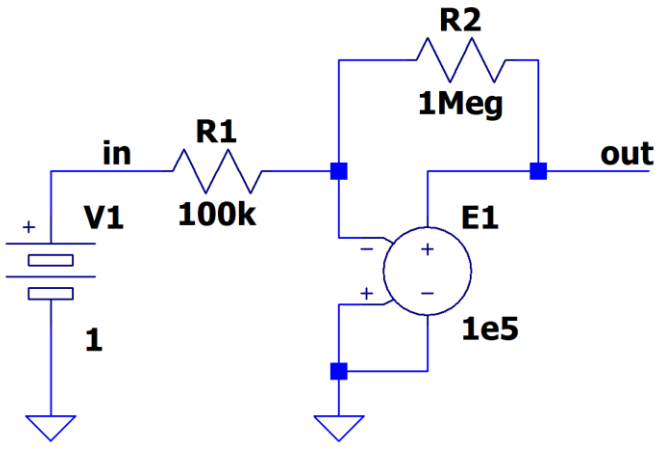


Fig. 3. Op amp inverting amplifier schematic

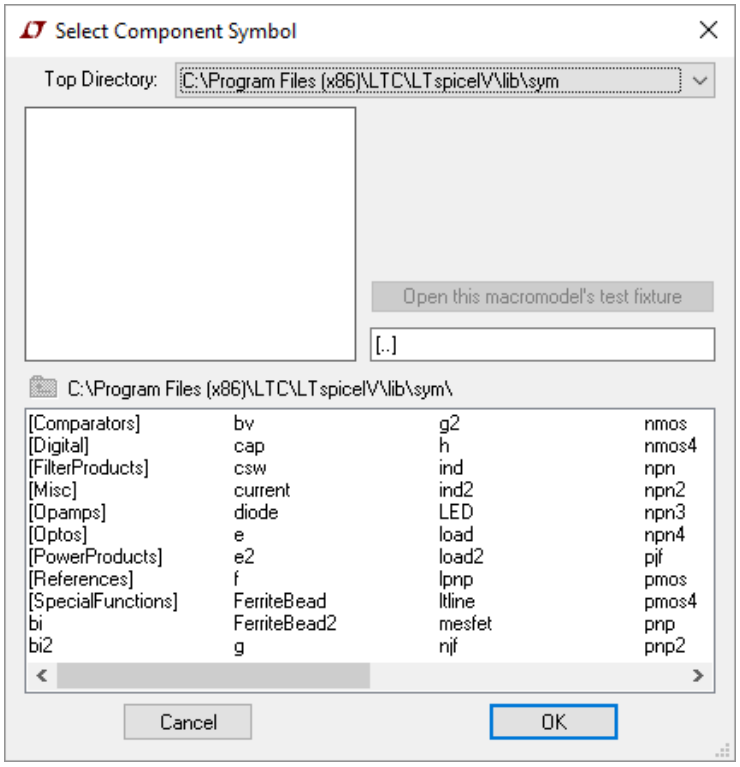


Fig. 4. Select Component Symbol window, top level directory.

- Move it over the third resistor, left click to delete it, then right click to end the delete mode.

Note: The F5 key will activate the delete function without having to enter a menu. Also, the MS Windows version has a scissors icon on the toolbar.

c. Place the voltage-controlled voltage source (VCVS).

- Right click on the schematic and select Draft > Component.

- VCVS elements are identified by the letter “e” in all versions of SPICE. LTspice has two symbols, “e” and “e2”.
    - First click on the e and the symbol will appear in the top left box of the Select Component Symbol window.
    - Then click on the e2 and that symbol will appear. Note the difference is just the orientation of the + and – signs on the VCVS input leads.
    - The circuit in Fig. 3 uses e2, so left click on that one and then click OK.
    - Place it on the schematic as you did with the resistor. It will be moved later.
    - Right click to end the VCVS placement (or press the ESC key).

d. Place the independent voltage source.

- Right click on the schematic and select Draft > Component.
- Navigate to the “voltage” element in the top-level directory and left click on it.
    - Its symbol is a standard independent voltage source and not one that looks like a battery as in the schematic. However, it can be used for a dc source too.
    - You will use a battery symbol and not this one, but the standard voltage source symbol is more popular.
- Locate and double left click on the [Misc] directory link.
    - Select the “battery” symbol and then left click OK.
    - Place it on the schematic and then right click when you are done.

e. Place the ground (0-volt node) reference symbol.

- Right click on the schematic and select Draft > Label Net.
- Select GND (global node 0), then OK. Place the GND symbol on the schematic.
- You can place more than one ground symbol on a schematic, which can make the wiring neater.
- Right click when you are done placing grounds.

Note: The MS Windows version has a ground icon on the toolbar.

5. Arrange components and connect them with wires as follows.

a. To move and rotate components, use the Move tool (looks like a hand with open fingers).

- Right click on the schematic and select Edit > Move. Or you can use F7.
- Click on one of the two resistors.
- While the resistor is “active”, press Ctrl-R to rotate it to horizontal and move it to its final position. Then left click to place it.
- Repeat the process to place the second resistor.
- Move the voltage sources to their final positions.

b. Wire the schematic using the wire tool (looks like a pencil).

- Right click on the schematic and select Draft > Draw Wire. Or you can use F3.
- Move the mouse position to one of the element leads using the crosshairs to find it.
- Left click to start the wire. You do not have to hold the mouse button down.
- Drag the wire to the next element. You can make bends by left clicking and changing the wire direction.
- When you reach the next element, left click on its lead and the wire will terminate.

Note: If you want to finish a wire open-ended and not on an element, left click, then right click where you want to end it.

c. More on moving components.

- The Move tool will move components and individual wire segments when you left click on them.
- You can move a group of components by first drawing a select box with the Move tool hand.
  - Position the Move hand where you want to start the select box. Click and hold the left button and drag it to select the components you want. Then release the left button.
  - Then drag the selected components where you want them to go.

d. Dragging components.

- The Drag tool is the hand symbol that looks like a fist.
- Dragging leaves wires connected and can be more convenient at times.

6. Elements are automatically given reference designator labels as they are placed on the schematic. LTspice uses them as unique identifiers. Elements usually do not have default values, but if they do, they usually need to be changed. Fig. 5 shows circuit element labels and values.

a. Changing resistor values and designator labels is done as follows.

- You can change the designator label by right clicking on it and entering text. You do not need to change it for this exercise, but resistor designator labels must start with the letter R.

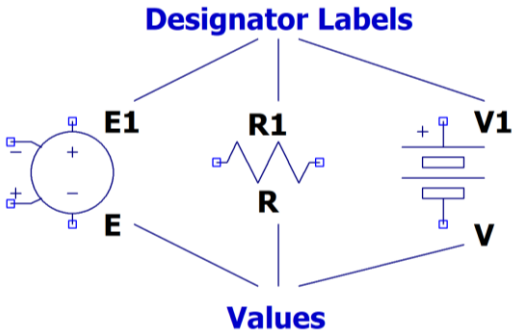


Fig. 5. Element designator labels and values.

- The initial value for all resistors is the placeholder letter R. It does not represent a numerical value.
    - Right click on the R and change the value to what you want.
    - Numbers like 1e5 or 1E5 are equivalent to 100k, 100000, and 0.1Meg.

    Note: Exponential notation follows the syntax above. Characters like "×10" and "^" are meaningless and can cause errors. See Table 1.

Table 1: SPICE number notation.

Suffix	Multiplier
T	1e12
G	1e9
Meg	1e6
K	1e3
mil	25.4e-6
m	1e-3
u(or μ)	1e-6
n	1e-9
p	1e-12
f	1e-15

b. Changing the independent voltage source value and designator label is done as follows.

- The designator is changed the same way as with resistors. You do not need to change it for this exercise, but independent voltage source designator labels must start with the letter V.
- To change the value, right click on the placeholder letter V. Enter the dc voltage you want.
- You can also change the value by right-clicking on the voltage source element, which opens the Voltage Source dialog box.
    - Enter the dc value you want in the box.
    - Note that if you left click the Advanced button, you can give the source values for other simulation types.
        - "Functions" waveform types are for Transient (time-domain) simulations.
        - "DC Value" is for operating point and transient simulations.
        - "Small signal AC" is for ac analysis (frequency sweep) simulations.

c. Changing the VCVS scale factor and designator label is done as follows.

- The designator is changed the same way as with resistors. You do not need to change it for this exercise, but VCVS designator labels must start with the letter E.
- To change the scale factor, right click on the placeholder letter E. Enter the scale factor you want.
    - The scale factor does not have to be a constant value

- It can be an equation involving other circuit parameters. See the Help section on Voltage Dependent Voltage Source.

- You can also change the scale factor by right-clicking on the element which opens the Component Attributes Editor box.
  - Change the scale factor in the Value row, Value column.

d. Labeling circuit nodes makes it easier to find them in the simulation results.

- Label the input node between V1 and R1 as follows.
  - Right click on the wire, then left click Label Net.
  - The Net Name dialog box will appear. Type "in" (without quotes) in the box, then left click OK.
  - A box will attach to the cursor that has a small square anchor point. Position the anchor point on the wire and left click to anchor it to the wire. Then right click to stop the mode.
  - You can remove net names with the Delete tool.
- Label the output node "out".

e. Save your circuit schematic by first left clicking File > Save As. Navigate to a desired folder or create a new one and enter a filename. Then click Save.

Note: You should save files frequently and use different filenames when appropriate.

Running Simulations and Documenting Results

You will run three simulations, a dc operating point analysis, a time-domain analysis, and a small-signal frequency response analysis. You will also learn how to plot waveforms, use cursors, and format plots for reports.

7. *DC operating point simulation.* The schematic you just entered uses an independent dc voltage source, so you will perform a dc analysis first.

a. Set up the dc operating point analysis and save results as follows.

- On the menu at the top of the window, left click Simulate > Edit Simulation Command.

  Mac Users: Right-click, Draft > Spice Directive

- Left click the DC op pnt tab (**Mac Users:** Type .op into the empty text box). There are no parameters to specify, so left click OK.
  - A box will attach itself to the cursor.
  - Move the cursor to an empty part of the schematic and left click.
  - The dot command .op will appear. You can use the Move function to move the .op command text somewhere else.

b. Run the .op simulation by clicking Simulate > Run (looks like a running person) on the main menu.

- For MS Windows users, a window with the results will appear.

Mac Users: Operating point simulation results are found in the SPICE Error Log. Right-click to reveal the menu, then View > SPICE Error Log.

- Reduce the size of the window to include just the output data.
  - Put the cursor over the lower right corner, until a double arrow appears.
  - Left click, hold and drag the window to an appropriate size.

c. Open a new document in MS Word or a similar word processor, then copy and paste the results in it as follows.

- Click on the window with the .op simulation results to make it active.
- MS Windows can copy the .op results window to the clipboard by pressing keys Alt-PrtScr at the same time when that window is active.

  Mac Users: Select the relevant text, right-click and copy
- In the document, paste the clipboard bitmap of the .op results window.
- Resize it in the document so the results are readable.

d. Copy the schematic to the document as follows.

- Return to LTspice and copy the schematic to the document as follows.
- Close the .op results window.
- Right click on the schematic window and select View > Zoom to Fit  . This will expand the schematic to maximum size.
- On the menu, left click Tools > Copy bitmap to Clipboard.

  Mac Users: Use Screenshot, save the photo, then add the photo to the document.
- Then paste the clipboard bitmap into the document.
- Crop and resize it as needed to make it an appropriate size. This is how the schematic figure images were generated for these instructions.

8. *Time-domain simulation.* The next simulation is a time-domain analysis with the independent voltage source changed to a sine wave. You could use the battery source for this since it is a fully-functional independent voltage source. However, you will change it to the general voltage source you saw earlier.

a. Change the voltage source.

- Delete the battery source.
- Open the Select Component Symbol window as done earlier.

  Mac Users: Draft > Component > type vo into the search box. The independent voltage source will be selected.
- Find and select the Voltage independent source and place it on the schematic where the battery used to be.

b. Adjust the source to be a 1 Vp-p, 1 kHz sine wave as follows.

- Right click on the source and the Voltage Source dialog box will appear.
- Left click Advanced and the Independent Voltage Source box will appear.
- Select SINE and then fill in the information as follows.
  - DC offset: 0
  - Amplitude: 1
  - Freq (Hz): 1000
  - Leave all others blank. You should have values appearing as in Fig. 6.
- Left click OK. The schematic should now appear as in Fig. 7.

c. You will now set up the transient simulation to run for four sine wave cycles. A 100 µs maximum timestep is specified to ensure a smooth curve. You do not always have to specify a maximum timestep, but it is useful at times.

- Left click Simulate > Edit Simulation Command.

  Mac Users: Draft > Spice Directive. Type .tran 100u 4e-3 into the text box. Note that the timestep precedes the stop time.
- Left click the Transient tab and enter the simulation parameters below.
  - Stop Time: 4e-3
  - Maximum Timestep: 100u
- Left click OK.

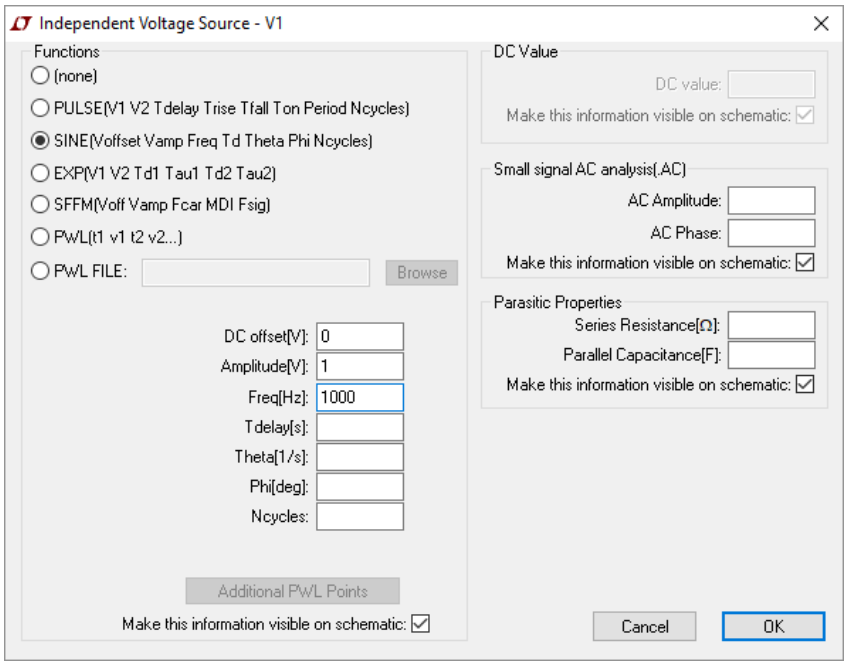


Fig. 6. Sine wave voltage source parameters.

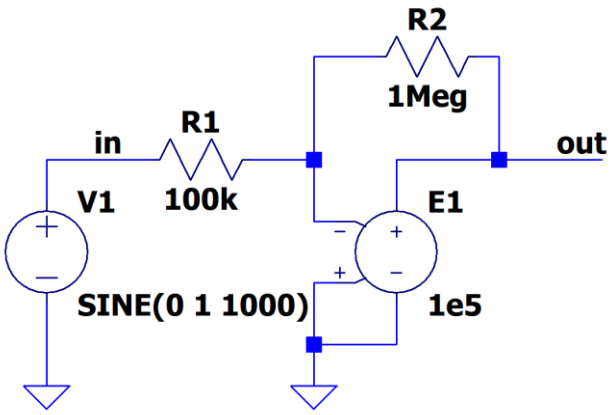


Fig. 7. Schematic with sine wave source.

- Place the .tran statement on the schematic. If the .op statement is still there, it will now have a semicolon in front, which comments it out and makes it inoperative.

  Mac Users: The Mac version does not add a semicolon to previous directives. Use Edit > Delete to remove the .op directive.

Note: Some versions of LTspice incorrectly handle this automatic commenting function and actually merge the specifications of two simulation types. This is noticeable when reselecting .op when a .tran simulation is active. In this case, you may have to delete existing simulation commands on the schematic and re-enter.

You can manually add and remove semicolons and periods to select the one simulation you want by editing each simulation command independently.

d. Run the simulation and examine the waveforms.

- Left click the menu Simulate > Run and a blank plot window will appear.
- Right click in the middle of the plot window and select Add Trace.
- Left click on the output node voltage and the input node voltage, then left click OK. A plot with both waveforms will appear.
- Add a grid to the plot as follows
  - Right click on an empty part of the plot window then select View and check the box next to Grid.
  - You can also select Plot Settings on the menu and check the Grid box.
- From the schematic, determine the ideal amplifier gain magnitude and whether it is non-inverting (0° phase shift) or inverting (180° phase shift).

  Gain: _____ Inverting or non-inverting: _____

- Before copying the plot to the clipboard for insertion in the document, it is essential to remove the black background. *Never use plots with black backgrounds* in documents.

- On the menu bar, left click Tools > Color Preferences.
  - Left click the Waveform tab.
  - In the Select Item box, choose Background.
  - Move the three sliders all the way right so the numbers in the boxes are 255.
  - Left click OK and you will return to the plot.
- The axes are grey and a better choice for a white background is black.
  - Return to the Color Preferences box.
  - Select Axis and move the sliders all the way to the left so the numbers in the boxes are 0.
  - Click Apply.
  - Now select Grid and make it black.
- The trace colors that look good with a black background may not with a white background, particularly light green. You will change this trace to the same color as V[11] but you need to see what its color numbers are first.
  - Select Trace V[11] and note the color numbers.
  - Select Trace V[1] and change the color numbers to the ones you just found.
  - Left click OK.

- The trace font size should be made larger than the default value.
  - On the menu bar, left click Tools > Control Panel.
  - Click on Reset to Default Values to see what those settings are.
  - Under Pen thickness[*] drop down menu, select 4.
  - Change the Font to Arial with 20-point size and select Bold font.

    Mac Users: This step cannot be done on the Mac version.
  - Left click OK.
  - The plot should look like the one in Fig. 8.

e. Adjust the plot axis scaling and copy it to the document.

- Adjust the time axis as follows.
  - Move the mouse over the horizontal axis until it displays a small ruler.
  - Right click and the Horizontal Axis dialog box appears.
  - Replace 4 ms with 2 ms.
  - Click OK.
- Copy the plot to the clipboard.
  - On the menu, left click Tools > Copy bitmap to Clipboard.

Mac Users: Use the Screenshot function.

- Paste the plot in the document.
- Resize the plot so it is 5.5-inch wide.

f. Copy the schematic to the clipboard and then to the document as before.

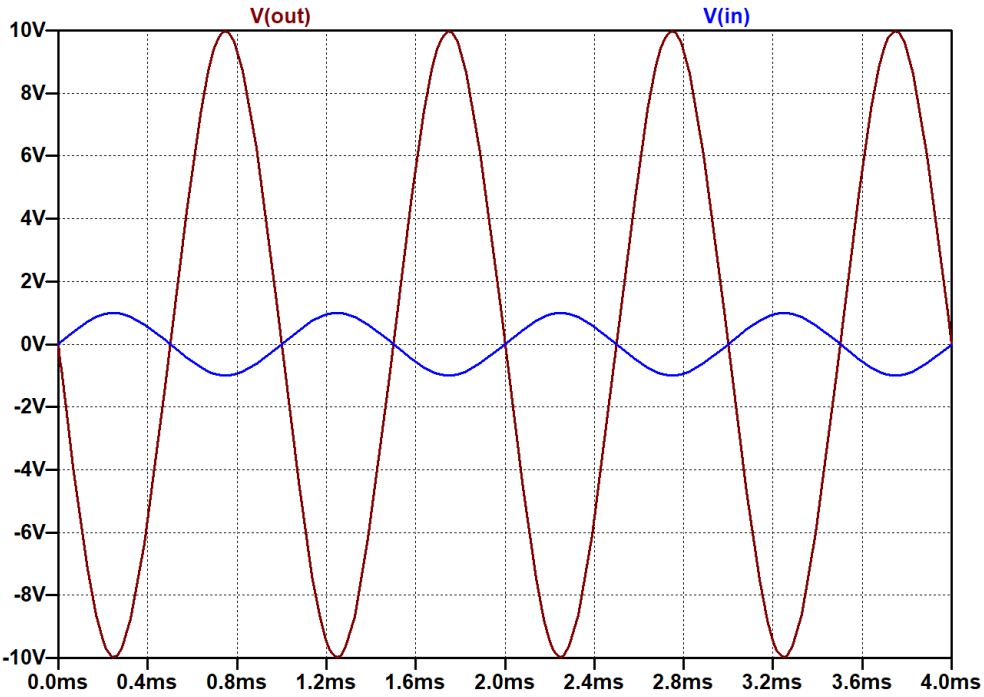


Fig. 8. Input and output waveforms from transient simulation.

9. *AC analysis simulation.* The last simulation is an ac analysis frequency sweep with the independent voltage source changed to an ac version with 1 V amplitude. The circuit is changed to be a single-pole lowpass filter with a 10 kHz cutoff frequency. You will make a Bode plot to display the amplifier gain vs. signal frequency, and using cursors, find the 3 dB frequency where the gain magnitude is 3 dB lower than the maximum at 0 Hz.

   a. Convert the amplifier to a single-pole lowpass filter as follows.

   - Left click on the Capacitor button on the toolbar and place a capacitor just above the 1 MΩ feedback resistor. Rotate it to be horizontal.

     Mac Users: Right-click Draft > Component and type c in the search box to select the capacitor. Click OK and hit Ctrl-R to rotate before placing on the schematic.

   - Connect the capacitor in parallel with the feedback resistor.
   - Calculate the value needed to set the cutoff frequency to f_c = 10 kHz where the response will be down 3 dB. Eq. 1 shows the relationship between the cutoff frequency, R_2, and C values.

$$C = \frac{1}{2\pi R_2 f_c} \tag{1}$$

- Change the capacitor value to the one you calculated.

b. Adjust the voltage source to be a 1 V ac source as follows.

- Right click on the source and the Voltage Source dialog box will appear.
- Left click Advanced and the Independent Voltage Source box will appear.

  Mac Users: Right-click on the source and the Edit Voltage Source box will appear.
- In the Small signal AC analysis area, set the amplitude to 1.
- Leave the sine wave time domain configuration as is. It will not affect the ac analysis simulation.
- The schematic should appear as in Fig. 9 with the placeholder letter C replaced with the value you calculated.

c. Now set up the ac analysis simulation to sweep from 100 Hz to 1 MHz:

- On the menu, left click Simulate > Edit Simulation Command.

  Mac Users: Right-click Draft > Spice Directive and type .ac dec 101 100 1e6 into the text box.
- Left click the AC Analysis tab and enter the simulation parameters below. A decade frequency sweep uses logarithmic point spacing over each 10:1 frequency decade. By choosing 101 points per decade, the plot should be smooth ad there is a point on each decade boundary and 100 frequencies in between.
  - Type of Sweep: Decade
  - Number of points per decade: 101
  - Start Frequency: 100
  - Stop Frequency 1e6
- Left click OK and place the .ac command on the schematic (**Mac Users:** Use Edit > Delete to delete the previous .tran directive)

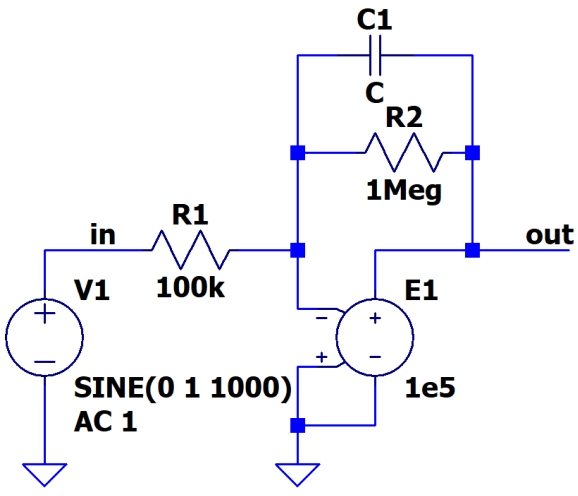


Fig. 9. Schematic with ac source and feedback capacitor.

d. Run the simulation and examine the waveforms.

- Left click the Simulate button and a blank plot window will appear.
- Traces for the input and output may already be present. If so, delete the input trace using the Delete tool.
- If no traces are present,
  - Right click in the middle of the plot window and select Add Trace.
  - Left click on the output node voltage then left click OK. Plots of the output magnitude and phase will appear as shown in Fig. 10.
  - Because your input signal had 1 V amplitude, *a plot of the output signal is the same as the gain, which is why 1 V was used.*
- Place two cursors on the plot and measure the 3 dB cutoff frequency as follows.
  - Right click on the trace name at the top of the plot and the Expression Editor will appear.

    Mac Users: Left click on the trace name to enable cursor 1. Left click again to enable cursor 2. No numbers appear on either cursor; use the menu window that appears to distinguish between cursors. No need to attach cursors – they will appear and be easy to see.
  - In the Attached Cursors box, select 1st & 2nd.
  - Left click OK.
  - The cursors will be dashed crosshairs that are sometimes difficult to find.

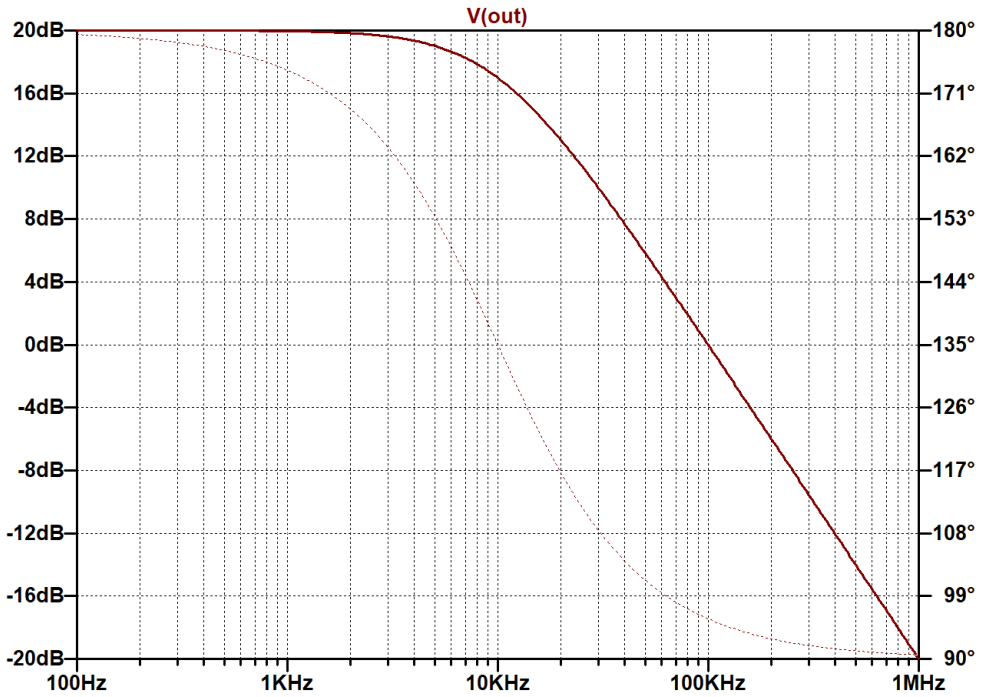


Fig. 10. Lowpass filter frequency response.

- Move the mouse around the plot until a yellow "1" appears.
- Left click and hold and drag the cursor all the way down to 100 Hz.
- Note the cursor 1 magnitude.
  - This is the gain magnitude at 100 Hz and is very close to the maximum gain that occurs at 0 Hz.

    Gain at 100 Hz in dB: _____
  - Convert the gain in dB to a non-dB gain magnitude.

    Gain at 100 Hz (non-dB): _____
  - Is this expected value?
- Move the mouse around the plot until a yellow "2" appears.
- Left click and hold and drag the cursor until the Ratio (Cursor2/Cursor1) magnitude is -3.00 ± 0.02 dB.
- Read the cursor 2 frequency. Determine the percent error between 10 kHz and the cursor 2 frequency.

  Cutoff frequency: _____
- Place a marker at the 3 dB point as follows.
  - On the plot menu, select Plot Settings > Notes and Annotations > Label Curs. Pos.

  Mac Users: Use Draw > Text to create a text label (this can't be edited so type carefully. Use Draw > Arrow to make an arrow from the label to the point on the trace.
  - You can right click on the label text and change or delete it.
  - You can also delete the label and arrow using the Delete tool.

e. Copy the plot and paste it in the document.

- Left click Tools > Copy bitmap to Clipboard (**Mac Users:** Use Screenshot)
- Paste the plot in the document.
- Resize the plot so it is 5.5-inch wide.

Results Documentation Summary

10. You should have the following documentation from the procedure steps.

a. You should have three schematics, one from each simulation.

b. You should have an image of the dc simulation results and one plot from the transient simulation and one from the ac simulation.

c. Be sure that you recorded data and calculated results in steps you were asked to.

d. Check with your instructor for additional documentation requirements.

Instrumentation

EE 3401 Laboratory Exercise
Electrical Engineering Department
Kennesaw State University

Objectives:

Students will learn how to make measurements using a function generator and oscilloscope. The basic features of both instruments are examined. Also, methods for improving displayed waveform quality are explored.

Introduction:

The typical equipment you find in an electronics laboratory are a DC power supply, a multimeter, a function generator, and an oscilloscope. In this exercise, you will use some of the most popular features of the function generator and oscilloscope to make measurements. You will build a voltage divider with a low voltage output and use features of the oscilloscope to improve the displayed waveform.

Equipment:

Agilent (Keysight) 33220A Function/Arbitrary Waveform Generator
Agilent (Keysight) DSO 5014A Oscilloscope
Cables and oscilloscope probes as needed
Solderless breadboard and required components – see instructions
Components:
Resistors (Qty): 1 kΩ (1), 1 MΩ (1)

Procedure:

Function Generator

You will start with the function generator. The Agilent 33220A Function/Arbitrary Waveform Generator (abbreviated FG or AWG) produces the usual function generator waveforms (sine, square, triangle, etc.) plus arbitrary waveform shapes downloaded into its memory. The most important AWG specifications are given in the appendix, Table 1.

1. Set up the AWG for a 1 kHz, 1 $V_{peak\text{-}peak}$ sine wave as follows.
   a. First you need to change the generator's configuration so that the output amplitude value you see on the display will be more accurate for the typical load impedances you connect it to.

      By default, the function generator expects to be connected to a 50 Ω load. Since it also has a 50 Ω output impedance, this forms a voltage divider at the output connector with a ratio of ½. The equivalent circuit is in Fig. 1. The generator takes this assumption into account, so if you set the output to 1 $V_{p\text{-}p}$, the AWG sets V_{gen} to 2 $V_{p\text{-}p}$ so that V_{out} = 1 $V_{p–p}$.

      When you connect a high load impedance like 10 kΩ, the divider ratio is actually 10,000/10,050 = 0.995 and V_{out} = 1.99 Vp-p. But, the display will still read 1 Vp-p because of the 50 Ω load assumption.

      The solution is to change the generator's assumed load impedance to open circuit by putting it in High-Z mode.

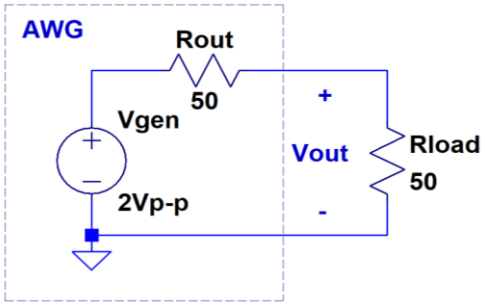


Fig. 1. AWG output equivalent circuit

On the AWG's front panel push Utility > Output Setup. You will see the display in Fig. 2 with the default 50 Ω assumed load specified. Then push the grey button under the highlighted Load indicator and it will switch to High-Z. Then press the blue button under Done.

b. Push the Sine button and you will see a display like in Fig. 3. The Freq setting is active so you can just type in numbers on the keypad or use the large dial to change the frequency. The buttons under the dial will move the cursor position on the display. The default output frequency is 1

Fig. 2. AWG output setup display.

Fig. 3. AWG sine wave display.

Fig. 4. Entering the amplitude level.

kHz, which is what you want, so don't change it.

Note: It's best to use Freq, Ampl, and Offset settings and not Period, HiLevel and LoLevel settings.

c. Push the blue button under Ampl and type in 1 with the keypad; then push the button under $V_{p\text{-}p}$. See Fig. 4. Now press the Output button and this will enable the signal at the output connector.

Note: At power-up the generator output is disabled and it is easy to forget to enable it. But it is a convenient feature because you can disable the output when you need to without disconnecting the cable or turning off the generator.

Oscilloscope:

The oscilloscope displays voltage waveforms as a function of time. In this course the waveforms you will display are either periodic or DC levels. To display waveforms, the oscilloscope needs to *trigger* on one of them. With *edge* triggering, you tell it what voltage level on the waveform to use for the trigger threshold. When the signal reaches that level, the oscilloscope draws a trace. When done, it waits a short time and looks for another triggering event and draws the next trace. The traces overlap each other exactly. Traces are updated many times per second.

You need to adjust the oscilloscope settings to optimize the display. The three main tasks are to (1) select the waveform's peak-peak amplitude, (2) select the time-per division so it displays the number of periods you want, and (3) select the trigger parameters needed to stabilize the waveform.

Once you optimize the display, you can have the oscilloscope make waveform measurements for you. Waveform data and the display image can be saved to USB drives or uploaded to a computer application. Table 2 in the Appendix has some of the key specifications of the Agilent DSO5014A oscilloscope used in the lab.

The oscilloscope front panel is shown in Fig 5 with the main adjustment controls indicated. The most important settings are shown along the top of the display as in Fig. 6.

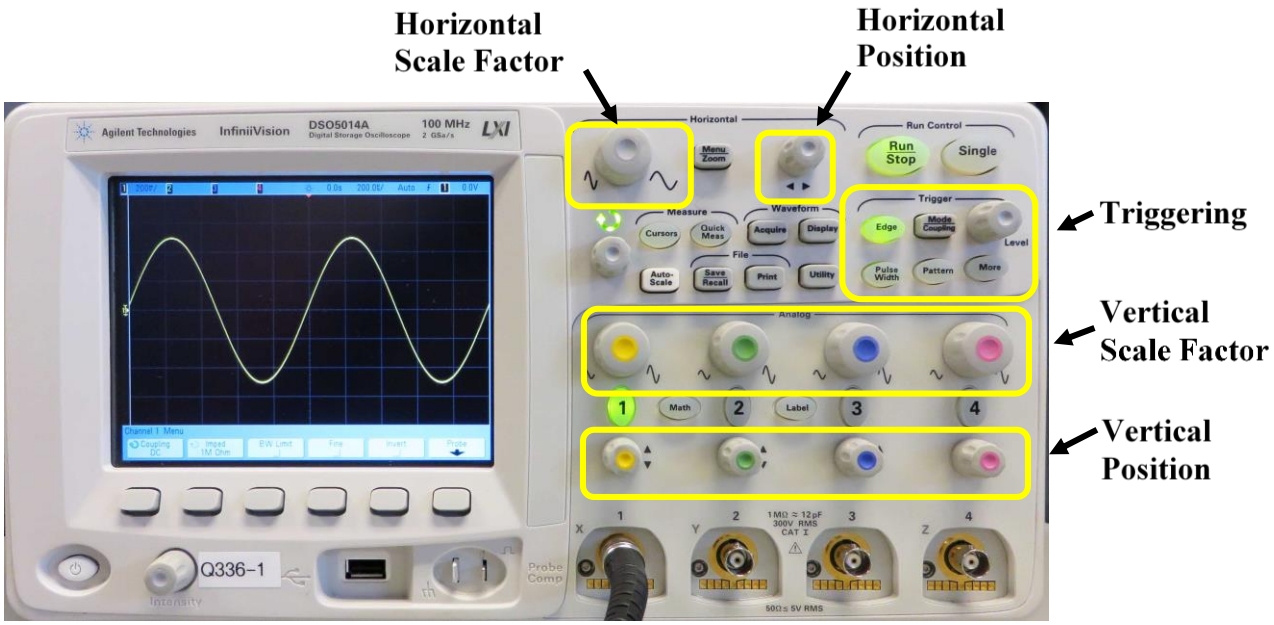


Fig. 5. Oscilloscope front panel.

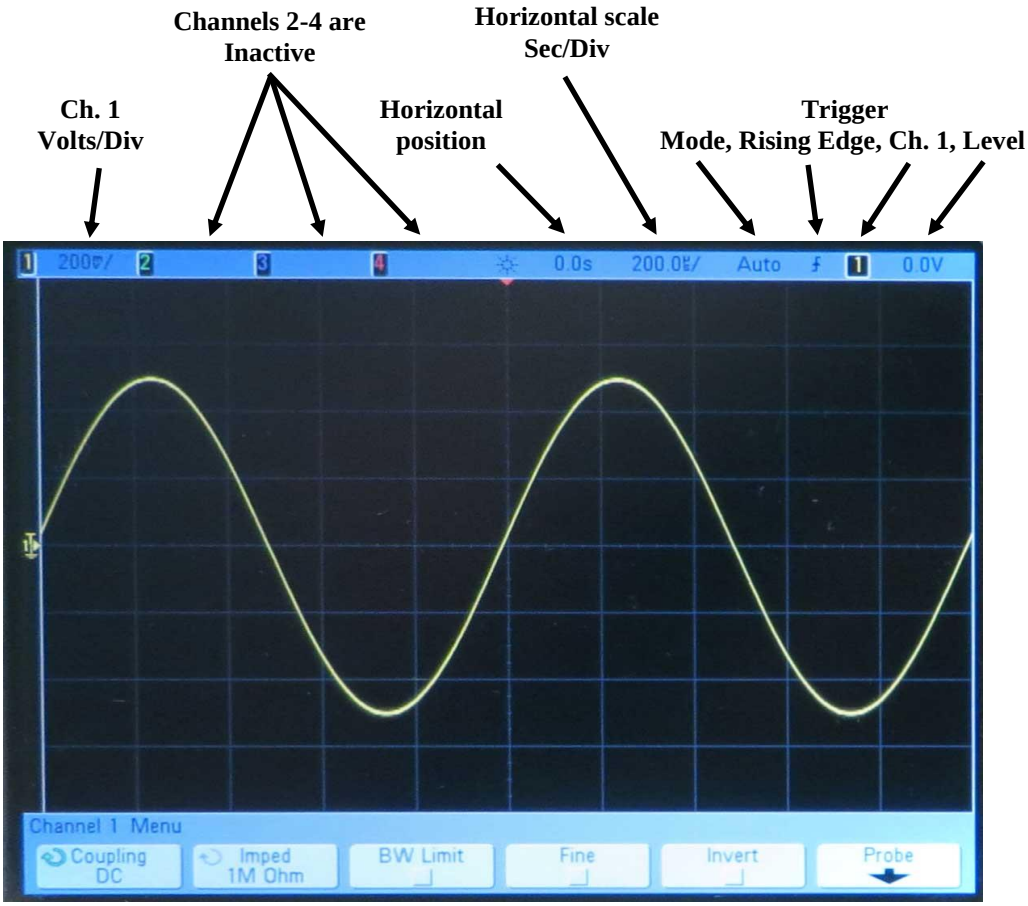


Fig. 6. Display with settings indicated.

2. You will set up the oscilloscope to display the AWG waveform on Channel 1.

   When you choose the oscilloscope vertical, horizontal, and trigger settings, you need to have some idea of what the waveform should look like. Three guidelines are: (don't do these on the oscilloscope yet).

   - Choose a vertical scale so that the waveform occupies 3 to 6 divisions.
   - Choose a horizontal scale so that 2 to 4 periods are displayed.
   - Choose edge triggering. Select rising or falling edge. Select auto trigger mode. Select a trigger voltage in the middle of the waveform's voltage swing (approximately the waveform's average voltage).

   Once you have a stable display you can readjust it as desired.

   Note: The normal, or Norm trigger mode will not display a trace at all if a trigger event is not detected, so it can be hard to set up the display in this mode. The automatic or Auto mode will always display a trace but if a trigger event is not detected the trace will be unstable.

   a. Connect the AWG output (not the sync) to the oscilloscope Ch. 1 input with a cable. The waveform may not look like the one in Fig. 6 at this time.

   b. The AWG output should be a 1 Vp-p, 1 kHz sine wave, so calculate appropriate settings:

      Calculate the vertical scale factor so the peak-peak waveform amplitude will span 5 divisions.

      Calculated scale factor (volts/div): _____

      Now calculate the waveform period.

      Waveform period: _____

      From the calculated period, determine the horizontal scale factor that results in 2 periods across the 10 horizontal divisions.

      Horizontal scale factor (sec/div): _____

      Calculate the voltage that is midway between the waveform positive and negative peaks:

      Midpoint voltage: _____

   c. Now adjust the settings as in the procedure below. The displayed waveform may not be stable until you complete the settings adjustments.

      - Press the "1" button in the Ch. 1 vertical control area so it is lit. This activates Ch. 1 (Fig. 7.)
      - Adjust the Ch. 1 vertical position so the trace ground reference is at the display center. Fig. 8 shows the ground reference level indicator (has a "1" next to it) and the trigger level indicator (has a "T" next to it) on the left side of the display. The ground level in Fig. 8 is above the center so you can see it.

Fig. 7. Ch. 1 Analog vertical controls.

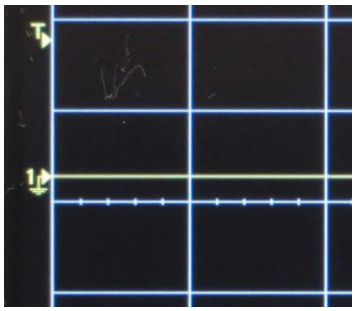


Fig. 8. Ch. 1 ground reference and trigger level. No signal present.

- Adjust the Ch.1 vertical scale until it is the value you calculated or as close as possible to that value.

  The adjustment increments follow a 1 – 2 – 5 most-significant digit progression, so your calculated value may not match the setting values, but that is fine. Just use the closest scale factor. The waveform should not clip at the top or bottom of the display.

- Adjust the horizontal scale factor knob (Fig. 9) to your calculated value.

  Again, the increments follow a similar 1 – 2 – 5 pattern, so you may not find the exact value you calculated. Just make sure you see at least two periods.

- Press the Mode/Coupling button in the trigger control area (Fig. 10) and the menu at the bottom of the display will appear as in Fig. 11. If the mode is not Auto, press the button under the Mode menu item and select it. Press multiple times until the check mark appears next to Auto, then stop. The menu will disappear.

- Press the Edge button in the trigger section. The Edge Trigger menu will appear across the bottom of the display as in Fig. 12.

- Since the only waveform is on Ch. 1, you must trigger on it. If the Source is not Ch. 1, press

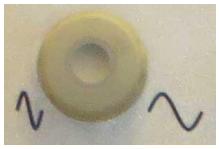


Fig. 9. Horizontal time base scale control.

Fig. 10. Trigger controls.

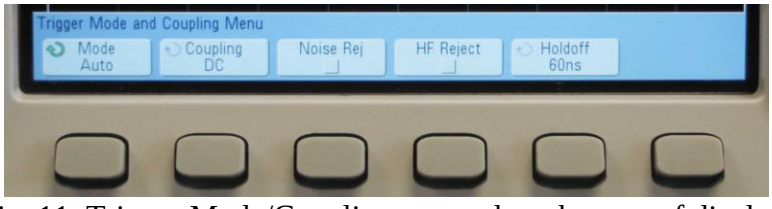


Fig. 11. Trigger Mode/Coupling menu along bottom of display.

the button under the Source indicator until the selection menu pops up. Press multiple times until the check mark appears next to Ch. 1, then stop. The menu will disappear.

- Choose the positive-going slope (arrow below Slope points up).
- Now adjust the trigger voltage level to the midpoint voltage value you calculated in part 2(b) by turning the small Level knob in the trigger section. Once this is done, the waveform should be stable.

Note: If you are on a low V/div scale like 10 mV/div and the trigger level is way off, it can take many Level knob turns to set the level to what you want. You can speed up the process by changing the vertical scale to a higher value and then adjusting the trigger level to bring it close. Then reset the vertical scale to the one you want and fine-tune the trigger.

- Have your instructor verify that your display is stable.

d. You can have the oscilloscope automatically scale a waveform and trigger on it by pressing the Autoscale button (Fig. 13.).

While this is convenient, you should not have to resort to this method just to display a waveform.

Note: *Autoscale can produce unexpected results*. For example, the oscilloscope may trigger on a high-frequency noise signal. You can tell if this happened when your waveform does not look right. When looking at the amplitude and horizontal scale settings you will likely see a very small vertical scale and a very small time/div (usually nanoseconds/div).

- Press the Autoscale button and see what happens.

3. Making automatic measurements.

The oscilloscope can display up to four simultaneous automatic. It's easier than using time and voltage cursors to make the measurements manually. The measurements can be made on any waveform and between waveforms. The most popular ones are peak-peak amplitude, average level (same as DC level), frequency, period, and phase difference.

a. Press the Quick Meas button shown in Fig. 13 and the Measurement Menu will appear. From the menu, you choose the source channel, the measurement you want, and then display it. When you press the Select button the menu in Fig. 14 appears. You can keep pressing Select until the arrow points to the desire measurement, or you can use the Select Knob next to the Autoscale button to do the same.

b. Have the oscilloscope measure the AWG waveform peak-peak amplitude and frequency. You will have to select two measurements, one at a time. You may have to clear existing measurements with the Clear Meas menu.

Frequency: _____ Peak-peak voltage: _____

Fig. 12. Edge Trigger menu along bottom of display.

Fig. 13. Autoscale and Measure controls.

Do these values match the settings on the AWG? If they do not, correct the problem.

4. Measuring a voltage divider's gain.

   You will measure the input and output waveforms of the voltage divider circuit in Fig. 15 and explore some other features of the oscilloscope.

   Note: Always monitor the input waveform along with the output waveform.

   a. Build the circuit on a breadboard and connect cables to the equipment.

   - Put the two resistors on the breadboard, connecting one end of each together on one row. The other ends should be on different rows.

     Note: The 5 holes on a single row *are* connected. (For example, holes with coordinates 1-A, 1-B, 1-C, 1-D, 1-E). Rows on a panel *are not* connected across the gap (For example, hole 1-E does not connect to hole 1-F). Adjacent rows *are not* connected. The holes in columns of the long, narrow strips *are* connected.

   - Connect a cable from the AWG output connector to the circuit. You will need a BNC to mini-grabber or BNC to alligator clip test cable. The black lead is the ground, so it goes to R2 and the red lead goes to R1.

   - Connect test lead cables or true oscilloscope probes from the voltage divider to the oscilloscope so that you can monitor the AWG input signal on Ch. 1 and the voltage across R2 on Ch. 2.

     Note: Probes are better than test lead cables for connecting signals to the oscilloscope. The most common variety is a ×10 probe which has a 10 MΩ input resistance, but it also acts as an attenuator, reducing the signal amplitude at the oscilloscope input by a factor of 10. Agilent probes are automatically detected by Agilent oscilloscopes and the attenuation is corrected in the vertical scale factor. The non-Agilent probes have selectable ×1 (no attenuation) or ×10 attenuation but the ×10 ratio is not accounted for by the oscilloscope. You must do that in the channel menu Probe setting.

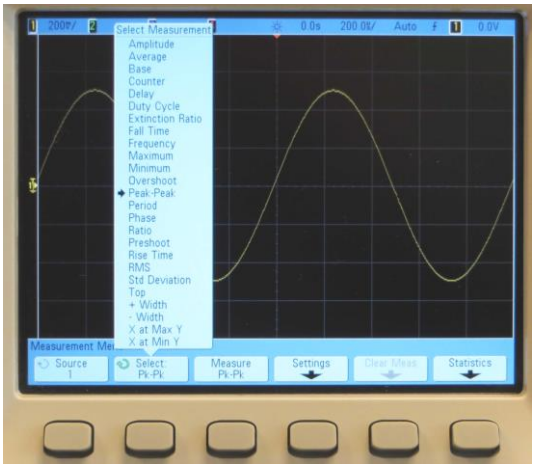


Fig. 14. Select measurement menu.

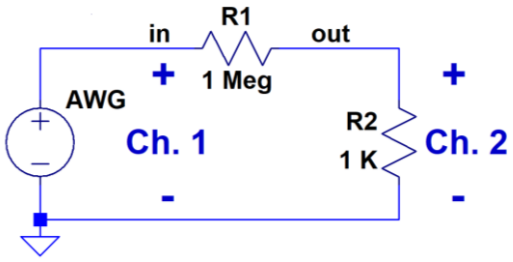


Fig. 15. Voltage divider.

- The completed setup should look like the one in Fig. 16. All black leads *must* be connected to the same node, in this case the bottom of R2. Fig. 17 is a close-up of the connections on the breadboard. Fig. 18 shows how you can make connections to components less cluttered with a short jumper wire.

b. Set the AWG amplitude to 10 Vp-p and enable its output.

c. Set the oscilloscope to trigger on the AWG signal.

d. Measure the peak-peak amplitude of both waveforms.

Input signal amplitude: _____ Output signal amplitude: _____

e. Calculate the theoretical and measured voltage divider gain (or loss) $Vout_{p\text{-}p}/Vin_{p\text{-}p}$. Compare the

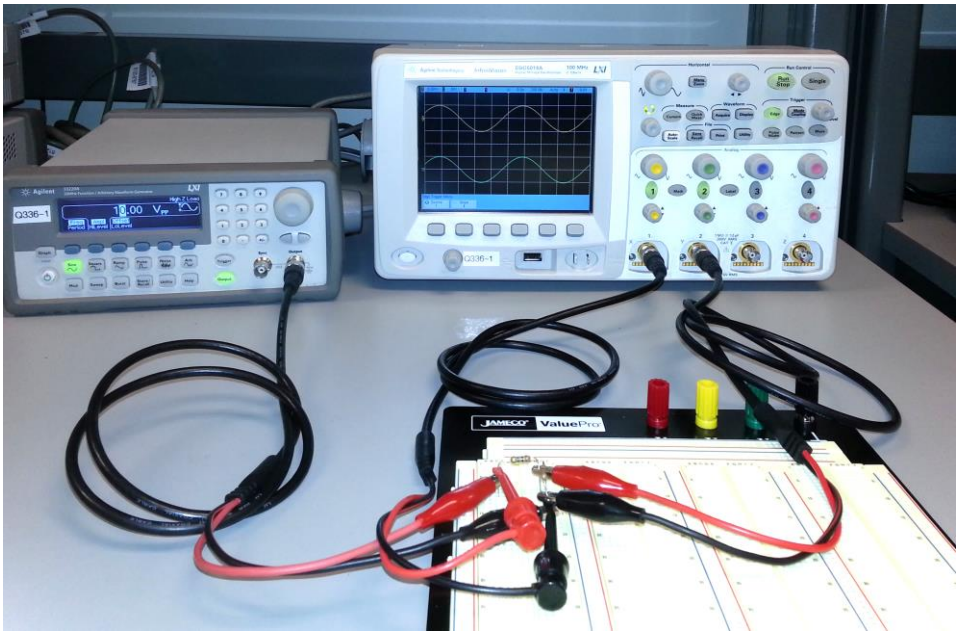


Fig. 16. Voltage divider setup.

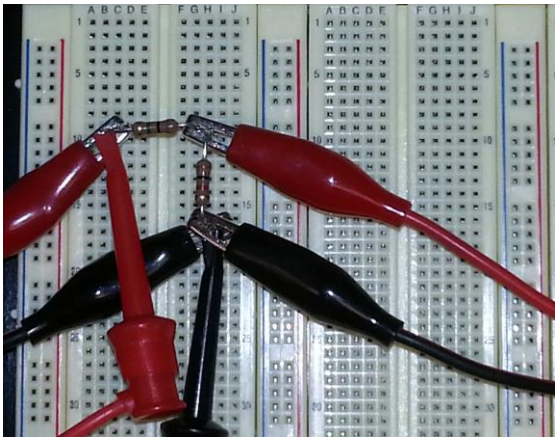


Fig. 17. Close up of test lead connections.

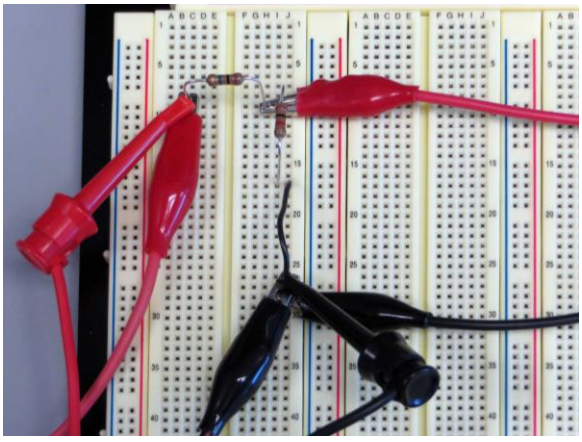


Fig. 18. Using a jumper to make connections easier.

two using a percent error calculation as in Eq. 1.

$$Percent\ error = \frac{Measured - Theoretical}{Theoretical} \times 100\% \quad (1)$$

Theoretical divider gain: _____ Measured divider gain: _____

Percent error: _____

f. Low-level signals like the one on Ch. 2 can have noise on it, which makes the automatic measured peak-peak voltage value too large.

- Go into the Ch. 2 menu and turn on the bandwidth limit (BW Limit). When activated the small square under BW Limit will be dark.
- Remeasure the output signal amplitude and recalculate the measured divider gain.

  BW Limit output signal amplitude:_____

  Remeasured divider gain: _____

  Percent error from calculated divider gain: _____

g. Averaging is the best way to reduce noise on a trace. But, the triggering must be stable for it to work. High resolution mode (Hi Res) is about as good and does not require stable triggering. In these two modes, all waveforms are affected, not just one channel.

- Turn off BW Limit on Ch. 2.
- In the Waveform section, press the Acquire button. Activate the Acq Mode menu and select Hi Res or Average. Then try the other to see which removes the most noise.
- Remeasure the output signal amplitude and recalculate the measured divider gain.

  Averaged or Hi Res output signal amplitude: _____

  Remeasured divider gain: _____

  Percent error from calculated divider gain: _____

  Account for the difference between the calculated and measured divider gain.

h. Now turn off waveform averaging or high resolution mode.

5. More on triggering.

The best signal to trigger on is the Sync output of the function generator. After that the next best signal is the one with the largest amplitude. However, you may find that you must trigger on low-level, noisy signals. The trigger circuit has low pass filters you can select to help with triggering on small signals.

a. Change your trigger source to the divider output on Ch. 2 and adjust the trigger level to the center of the waveform.

b. The waveform may not be stable. If it is, lower the AWG amplitude by 1 Vp-p increments until the waveform *is not* stable.

c. Go into the trigger Mode/Coupling menu and try activating high-frequency reject (HF Reject) or Noise Reject.

   Did HF Reject or Noise Reject stabilize the triggering?

6. Saving waveforms.

   Saving oscilloscope display images is the best way to document your measurements. If the oscilloscope is connected to a computer running Keysight (Agilent) BenchVue, it is the most convenient way to save waveforms.

   If not, waveforms can be saved to a USB drive inserted in the front panel connector.

   Note: when saving a display image, invert the graticule colors so that the display's background is white. It saves toner/ink when printing and looks better on white paper.

   a. Insert a USB flash drive in the front panel connector.

      Note: The largest capacity USB drive the oscilloscope can read is 4 GB.

   b. Press the Save/Recall button in the File section (Fig. 19) and the Save/Recall menu will appear on the display.

Fig. 19. Save/Recall button.

   c. Press the button under Save and its menu will appear on the display.

   d. In the Save menu, press the button under Format and select either a bitmap (BMP) or portable network graphics (PNG) image file format. You will automatically return to the Save menu in a few seconds.

   e. In the Save menu, make sure Save To says usb0. You will automatically return to the Save menu in a few seconds.

   f. Changing the file name is optional. If you want to, press the button under File Name and make the changes. Return to the Save menu by pressing the button under the up arrow ↑.

      Note: The oscilloscope will use a default name starting with "scope_0" and will increment the number automatically each time you save a display image.

   g. In the Save menu, press the button under Settings and select Invert Grat (the small square will turn dark). Return to the Save menu by pressing the button under the up arrow ↑.

   h. In the Save menu, press the button under Press to Save to save the file.

      Note: Always check your saved files to be sure they look right.

Appendix:

The important function generator and oscilloscope specifications are in tables A-1 and A-2.

Table 1. Important AWG Specifications.

Specification	Value
Standard waveforms	Sine, square, triangle, ramp, pulse, DC, noise
Square wave:	Frequency range: 1 µHz to 20 MHz Duty cycle: 20% to 80% Rise and fall times: < 13 ns
Triangle and ramp	Frequency range: 1 µHz to 20 MHz Variable symmetry: 0% to 100% (50% is triangle)
Pulse	Frequency range: 500 µHz to 5 MHz Pulse width: 20 ns minimum, 10 ns resolution
Amplitude range	10 mV peak-peak to 10 V p-p into 50 Ω load 20 mV peak-peak to 20 V p-p into open circuit
DC offset range (DC + peak AC)	±5 V into 50 Ω load ±10 V into open circuit
Waveform output impedance	50 Ω
Sync output	50 % duty cycle, TTL-level square wave at selected waveform frequency

Table 2. Important Oscilloscope Specifications.

Specification	Value
Number of channels	4
Channel bandwidth	DC to 100 MHz, 3.5 Hz to 100 MHz when AC-coupled
Channel digitizer sample rate	2 Gs/s
Waveform memory	4 million points per channel
Input impedance	1 MΩ or 50 Ω, selectable
Vertical resolution	8 bits (256 levels), 12 bits in Hi Res or Average acquisition modes (16,384 levels)
Vertical scale factor range	2 mV/div to 5 V/div
Coupling	DC or AC (AC coupling blocks the DC part of a waveform)
Bandwidth limit	20 MHz
Time scale factor range	5 nsec/div to 50 sec/div

Operational Amplifiers

EE 3401 Laboratory Exercise
Electrical Engineering Department
Kennesaw State University

Objectives:

Students analyze, build, test, and simulate several operational amplifier (op-amp) circuits to develop a fundamental understanding of circuits using them. Students also design an amplifier circuit to specifications.

Introduction:

Integrated circuit (IC) Op-amps are used in many applications such as fixed- or variable-gain amplifiers, filters, integrators, differentiators, and voltage regulators. This exercise focuses on classic op-amp applications such as basic inverting and non-inverting amplifiers, including a cascade topology. A potentiometer is used in two situations: to create an adjustable amplifier gain and to create an adjustable reference voltage.

In the prelab assignment, students analyze the amplifier circuits they build and test in the lab. The prelab assignment includes a variable-gain amplifier circuit design problem that is simulated later. The circuits built I the lab include decoupling capacitors to help reduce interference signals that could appear on the op-amp power supply pins.

The op-amp used is the LM741. It was very popular in the past when higher dc supply voltages were more common. Multiple manufacturers produce the 741 op-amp and these other models have different prefixes such as uA741. Today, many op-amps have better performance and can operate from dc supplies of only a few volts. But the LM741 is robust and this is an important characteristic in instructional labs when circuit connection errors occur.

Equipment:

Function/Arbitrary Waveform Generator
Oscilloscope
Digital Multimeter
DC power supply with ±15 V and +5 V outputs
Cables and oscilloscope probes as needed
Solderless breadboard
Components:

- Resistors (Qty): 10 kΩ (3), 56 kΩ (1), 100 kΩ (1)
- Potentiometer (Qty): 100 kΩ (1)
- Op-amp (Qty): LM741 (2)
- Capacitors (Qty): 0.1 μF (3)

Prelab:

Perform the following analyses and calculations before coming to lab. For all circuits in this assignment, assume that the op-amps are ideal and that they use ± 15 V power supplies for biasing. Fig. 1 shows how the op-amp's schematic symbol maps to its physical IC pinout.

1. For the five amplifier circuits in Figs. 2, 3, 4, 5, and 6:
   a. Write an expression for the output voltage, v_o, in terms of the resistor symbols and the input voltage, v_{in}.

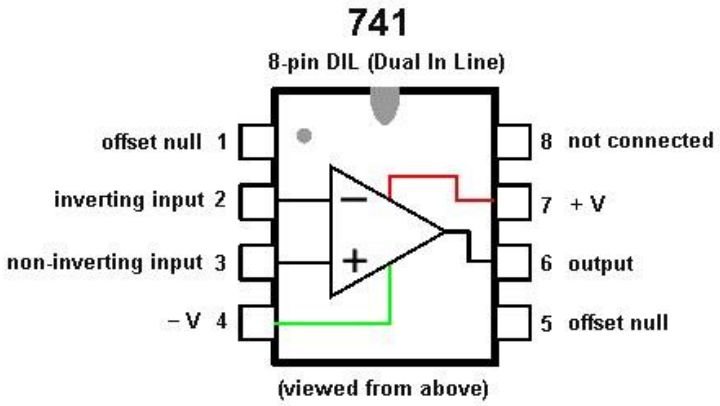


Fig. 1. LM741 op-amp pinout looking at it from above.

- For the first and second circuits (Figs. 2 and 3), the feedback resistor is the series combination of R2 and R3. Use both of these resistor symbols in your expression.
- For circuit 5 (Fig. 6), assume that v_{in} = 5V and ignore potentiometer R1 and C3.

b. Using resistor values and input voltage amplitudes, calculate the peak-peak output voltage, v_o (not just the peak voltage) and numerical voltage gain v_o/v_{in}.

- Calculate three numerical gains for the circuit 2 (Fig. 3) for R3 = 30 kΩ, 100 kΩ, and 0 Ω
- Record these results in Table 1 for circuits 1 through 4 (Figs. 2 through 5) and in Table 2 for the circuit 5 (Fig. 6).

2. Design an op-amp circuit that meets the following specifications. Your design can be done with one or two op-amps. Your instructor may give you a different design.

Note: Show all work. Draw a schematic of your design and label component values and op-amp pin numbers.

a. The op-amp circuit must have a non-inverting overall voltage gain that is adjustable from +3 to +12. The circuit has a 47 kΩ load resistor connected from the last op-amp's output terminal to ground.

b. The circuit input resistance must be ≥ 50 kΩ.

c. Your design is limited to a maximum of two op-amps and seven resistors, one of which is a variable 100 kΩ resistor (potentiometer). The potentiometer resistance is 0 Ω when adjusted to one extreme of its range and 100 kΩ at the other extreme.

d. When the potentiometer is adjusted to one extreme position the overall circuit voltage gain should be +3. When the variable resistor is adjusted to its other extreme position the overall circuit voltage gain should be +12.

Procedure:

General Information

1. This is important information about connecting the LM 741 op-amp in a circuit and making waveform measurements.

a. Refer to the LM741's pinout drawing in Fig. 1.

Note: The Offset Null (pins 1 and 5) as well as pin 8 are left unconnected in this lab exercise.

- When constructing the circuits, you may have to combine resistors to realize the required values as closely as you can.
- Use $\pm V_{CC} = \pm 15$ V for all circuits; that is +V = 15V and -V = -15V. For all circuits except circuit 3 in Fig. 4, make sure that the input voltage, $v_{in,}$ has no dc-offset voltage.

b. Measure sine wave input and output waveforms in peak-peak values. The oscilloscope can make the measurements automatically. Record your data in Table 1.

Note: Use Hi Res or Average acquisition mode for best oscilloscope measurements.

Note: Always turn off both power supplies before making circuit changes.

Note: Be careful not to damage the op-amp pins when it is inserted or removed from the proto-board. When removing, you should carefully pry it out, not pull it with your fingers. An IC puller tool is best.

Op Amp Circuit Measurements

2. The circuit in first op-amp circuit in Fig. 2a uses a potentiometer in the feedback path for adjustable gain. R1 and R2 set the minimum gain and R3 provides the adjustable gain range.

A potentiometer is a variable resistor with three terminals. The wiper is the center terminal and it connects to the potentiometer's internal resistance at an adjustable location set by the knob or screw.

- Let the resistance between the wiper and the left end terminal of R3 be R_x. Then the resistance between the wiper and the right end terminal is $R_y = R_{max} - R_x$, where R_{max} is the resistance between the two end terminals, in this case 100 kΩ.
- Only one of the two resistances is needed in this circuit, so you will connect the wiper terminal with a wire to left end terminal. This shorts out one of the two resistances, leaving the other as R3 = R_y in Fig. 2a.

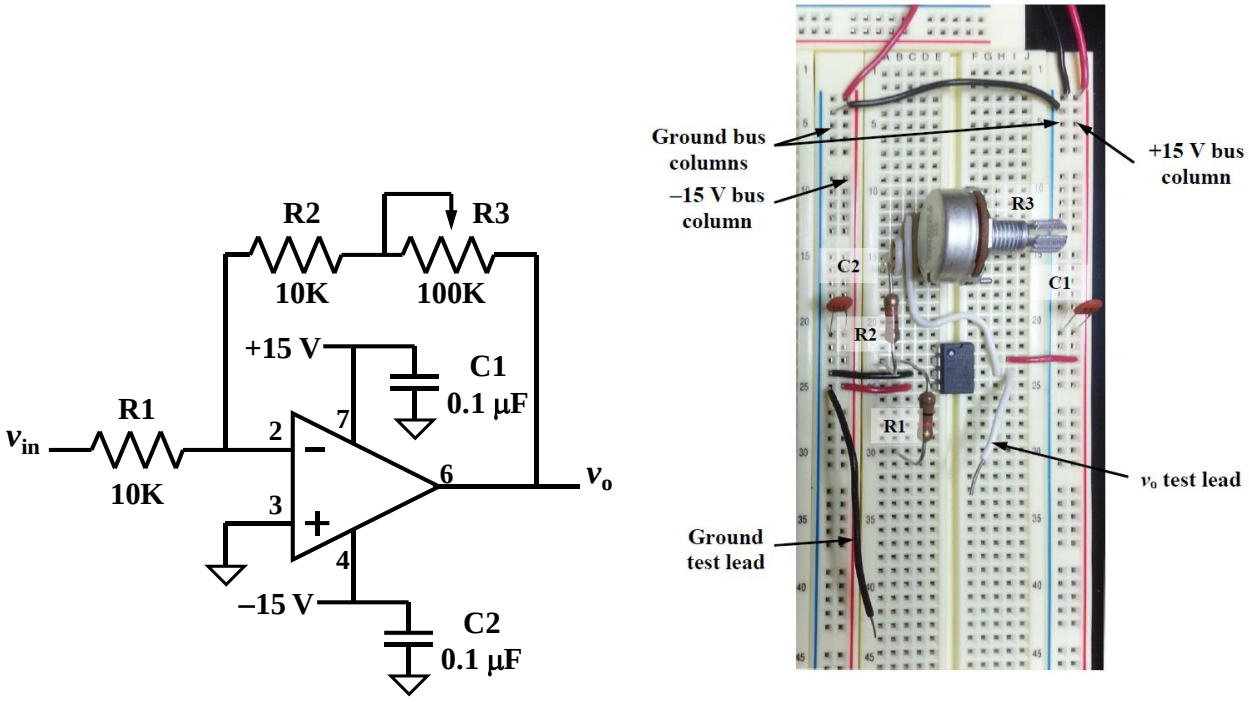
Note: Only use a potentiometer when adjustable gain is necessary. Otherwise use a fixed resistor.

a. Adjust the potentiometer so that the resistance from the wiper to the right end terminal is 30 kΩ. The view is looking at the adjusting wiper's shaft.

b. Place the components on your breadboard. A possible layout is shown in Fig. 2b. The breadboard busses (long connected columns) are used for the power supply connections. Connect C1 between the +15 V bus and the ground (GND) bus and one ground bus and C2 between the -15 V bus and the other ground bus. C1 and C2 should be within 1 inch of the LM741 IC.

Connect the other components. Minimize the number of wires. *Always connect components directly to each other if you can.* For example, one end of R1 and R2 should connect directly to the 741's pin 2.

c. Connect the power supplies to the op-amp and the function generator to the v_{in} node.



(a) (b)

Fig. 2. First op-amp circuit (a) schematic and (b) example breadboard layout.

d. Set the function generator so that v_{in}=1sin(1000πt) V. *Verify that the input waveform is correct using the oscilloscope*.

e. Turn on the power supply.

f. Adjust the oscilloscope to display two periods of v_{in} and v_o. Measure their peak-peak voltages and record them in Table 1. Save a copy of the oscilloscope display. Each waveform should span at least 2 vertical divisions.

3. Consider the second circuit in Fig. 3.

a. Be sure the power supply is turned off before changing the circuit.

b. Build the circuit. Leave the potentiometer setting the same as in the first circuit.

c. Connect the power supplies to the op-amp and the function generator to the v_{in} node.

d. Set the function generator so that v_{in}=1sin(1000πt) V. *Verify that the input waveform is correct using the oscilloscope*.

e. Turn on the power supply.

f. Adjust the oscilloscope to display two periods of v_{in} and v_o. Measure their peak-peak voltages and record them in Table 1. Save a copy of the oscilloscope display. Each waveform should span at least 2 vertical divisions.

g. Adjust R3 for maximum gain (100 kΩ). Measure the peak-peak voltages of v_{in} and v_o and record them in Table 1. Save a copy of the oscilloscope display. Each waveform should span at least 2 vertical divisions.

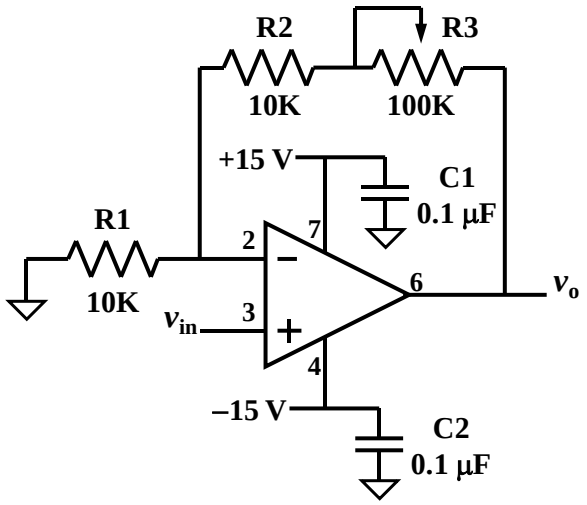


Fig. 3. Second op-amp circuit

h. Adjust R3 for minimum gain (0 Ω). Measure the peak-peak voltages of v_{in} and v_o and record them in Table 1. Save a copy of the oscilloscope display. Each waveform should span at least 2 vertical divisions.

4. Consider the third circuit in Fig. 4.

   a. Be sure the power supply is turned off before changing the circuit.

   b. Build the circuit. *Use a fixed resistor for R2, not the potentiometer*. Connect the power supplies to the op-amp and the function generator to the v_{in} node.

   c. Set the function generator so that v_{in}=1sin(1000πt) V. *Verify that the input waveform is correct using the oscilloscope*.

   d. Turn on the power supply.

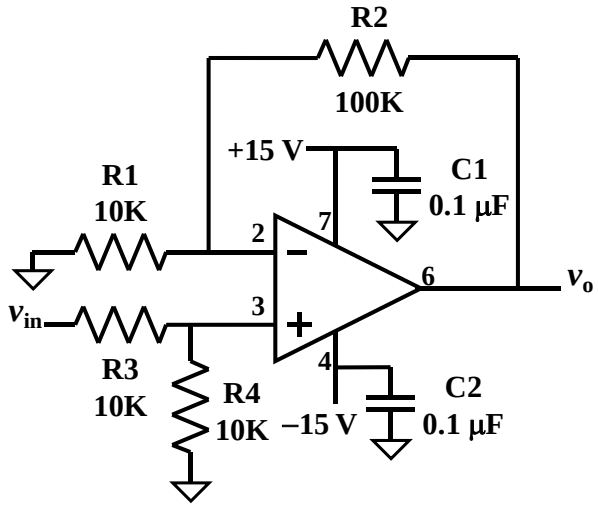


Fig. 4. Third op-amp circuit.

e. Adjust the oscilloscope to display two periods of v_{in} and v_o. Measure their peak-peak voltages and record them in Table 1. Save a copy of the oscilloscope display. Each waveform should span at least 2 vertical divisions.

5. Consider the fourth circuit in Fig. 5.

   a. Be sure the power supply is turned off before changing the circuit.

   b. Build the circuit. Connect the power supplies to the op-amp and the function generator to the v_{in} node. Leave C1 and C2 connected where they are.

   c. This circuit has a large gain. Set the function generator so that $v_{in}=0.1\sin(1000\pi t)$ V so the output waveform does not saturate. *Verify that the input waveform is correct using the oscilloscope.*

   d. Turn on the power supply.

   e. Adjust the oscilloscope to display two periods of v_{in} and v_o. Measure their peak-peak voltages and record them in Table 1. Save a copy of the oscilloscope display. Each waveform should span at least 2 vertical divisions.

6. Consider the fifth circuit in Fig. 7.

   a. Be sure the power supply is turned off before changing the circuit.

   b. Build the circuit. This time v_{in} is set to 5 V dc. The 5 V is derived from the +15 V supply as follows.

      Connect one end of the potentiometer to ground and the other end to +15 V. Connect the wiper to the op-amp's non-inverting terminal and C3.

      C3 and the lower part of the potentiometer resistance create a low-pass filter that attenuates interfering signals above a cutoff frequency. See question 3 in the Analysis section.

   c. Turn on the power supply.

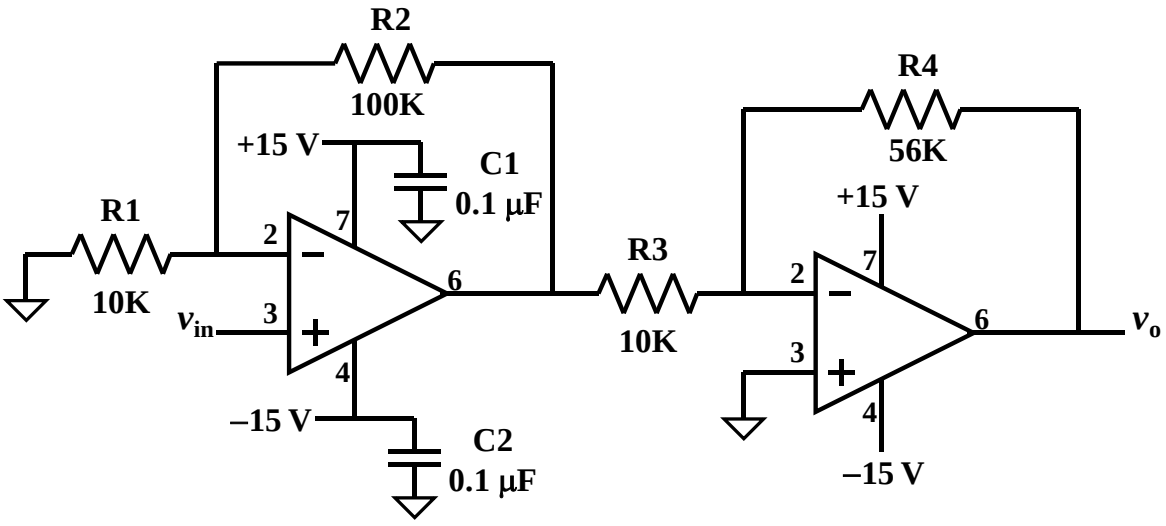


Fig. 5. Fourth op-amp circuit.

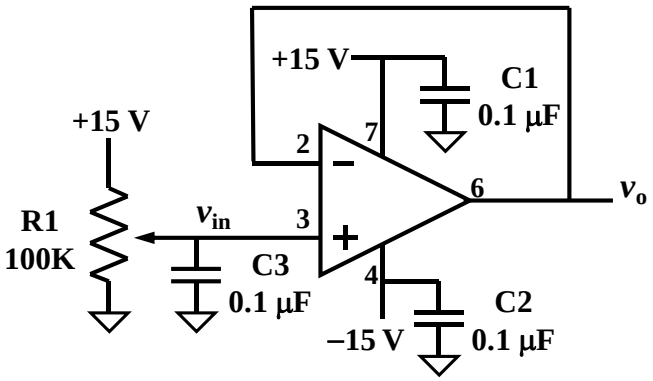


Fig. 7. Fifth op-amp circuit.

d. Measure v_{in} with the DMM, *not the oscilloscope*. Adjust it to 5 ± 0.1 V. Then measure v_o with the DMM. Record the values in Table 2.

Designed Circuit Simulation

7. Consider the circuit you designed in the prelab assignment.

   a. Run time domain simulations for two circuit configurations. For the first configuration replace the potentiometer by its 0 Ω minimum resistance. For the second, replace the potentiometer by its 100 kΩ maximum resistance.

   b. Use a detailed 741 op-amp model that has power supply pin connections. It may be called LM741 or uA741 or similar. You must use independent dc voltage sources set to +15 V and –15 V as power supplies for the op-amp model. Leave the offset adjustment pins 1 and 5 of the 741 model unconnected.

      Replace your fixed resistors with the closest commercially-available 5% tolerance resistor value. The resistor value does not have to be one in your parts kit. Keep the potentiometer's maximum resistance at 100 kΩ.

      Note: if you use LTspice, there are no 741 op-amps in the library. Choose an LT1001 op-amp instead.

   c. Use a 1 V peak, 1 kHz sine wave input. Make one plot for each configuration showing the input and output waveforms.

      - Mark the peak input and output waveform voltages on the plot.
      - Calculate the voltage gain from each simulation.
      - Save copies of the schematics and plots.

Results Documentation Summary

8. You should have the following documentation from the procedure steps.

   a. You should have six oscilloscope screen shots, one each from circuits 1, 3, and 4 as well as three from circuit 2.

   b. You should have a schematic and annotated waveform plot from the simulated circuit.

c. Be sure that you recorded data and calculated results in steps you were asked to.

d. Check with your instructor for additional documentation requirements.

Analysis:

1. Calculate the voltage gain using your measured data from procedure steps 1 through 6 and record the results in Table 1. Calculate the percent error between measured and calculated gains using Eq. 1 and record it in Table 1.

$$\%\, error = \frac{measured\ voltage - calculated\ voltage}{calculated\ voltage} \times 100\% \qquad (1)$$

2. Determine the percentage error between the ideal and simulated gain from procedure step 7.
3. Circuit 5 uses a potentiometer as a voltage divider to derive 5 V dc from 15 V dc. R_x is on one side of the wiper and R_y is on the other, as shown in Fig. 8. C3 creates a low-pass filter that attenuates some of the interfering signals that can appear on the +15 V power supply.
   a. Calculate values for R_x and R_y.
   b. Calculate the low-pass filter cutoff frequency in Hz.

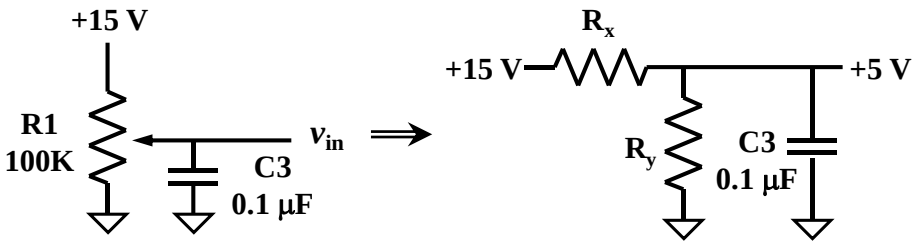


Fig. 8. Voltage divider and low-pass filter in circuit 5.

Appendix:

Table 1. Calculated and measured data for circuits 1 through 4.

Circuit No.	Calculated v_o (peak-peak)	Measured v_{in} (peak-peak)	Measured v_o (peak-peak)	Calculated Gain (Prelab)	Measured Gain	Gain Percent Error
1						
2 R3 = 30 kΩ						
2 R3 = 100 kΩ						
2 R3 = 0 Ω						
3						
4						

Table 2. Calculated and measured data for circuit 5.

Circuit No.	Calculated v_o (dc)	Measured v_{in} (dc)	Measured v_o (dc)	Calculated Gain (Prelab)	Measured Gain	Gain Percent Error
5						

Differentiator, Integrator, and Pulse-Width Modulator Circuits Using Op Amps

EE 3401 Laboratory Exercise
Electrical Engineering Department
Kennesaw State University

Objectives:

Students analyze, build, and test integrator, differentiator, and pulse-width modulator circuits using op amps to develop a fundamental understanding of their operation.

Introduction:

Op amps are used in circuits to perform various mathematical operations, such as amplitude scaling (gain), addition, subtraction, integration, and differentiation. This exercise focuses on the inverting integrator and inverting differentiator circuits, which are used in circuits such as waveform generators, active filters, proportional-integral-differential (PID) control systems, and pulse-width modulators (PWM). Within these more complex systems, the individual op amp circuits are "building blocks" connected to perform the desired system-level functions.

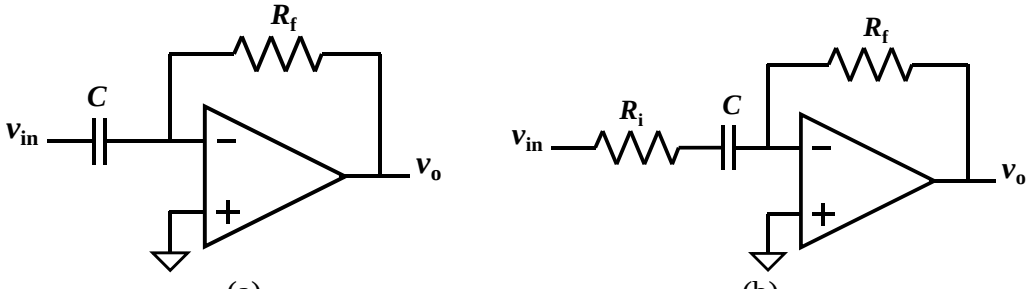
The PWM circuit in this exercise uses a triangle waveform produced from an integrator circuit and an op amp configured as a voltage comparator to demonstrate the building-block design approach. Example PWM applications include dc motor control, switched-mode power supplies, D/A and A/D converters, and light dimmer circuits.

Inverting Differentiator

The ideal op amp inverting differentiator circuit is based on the inverting amplifier with the input resistor replaced by a capacitor as in Fig. 1(a). Inverting differentiators produce an output voltage proportional to the *rate-of-change*, or *derivative* of the input signal voltage, but with inverted polarity. The circuit produces an output voltage when the input voltage is changing and will produce zero voltage when the input voltage is constant.

In the time domain, the ideal inverting differentiator's output is:

$$v_o(t) = -R_f C \frac{dv_{in}(t)}{dt} \tag{1}$$



(a) (b)

Fig. 1. Ideal inverting differentiator (a) and practical differentiator (b).

The ideal differentiator's gain as a function of radian frequency ω is

$$H(j\omega) = \frac{V_o(j\omega)}{V_{in}(j\omega)} = -\frac{R_f}{\frac{1}{j\omega C}} = -j\omega R_f C = \omega R_f C \angle -90°. \quad (2)$$

Letting $\omega = 2\pi f$ we have:

$$H(f) = \frac{V_o(f)}{V_{in}(f)} = \frac{-R_f}{1/(j2\pi f C)} = -j2\pi f R_f C = 2\pi f R_f C \angle -90°. \quad (3)$$

The gain decreases as f decreases, so $H(f) \to 0$ as $f \to 0$. But, the gain increases as f increases, so $H(f) \to \infty$, as $f \to \infty$. This is a problem for many applications, so the gain at high frequencies is usually limited by adding a resistor in series with the input capacitor as shown in Fig. 1(b). R_i converts the differentiator to a single-pole high-pass filter, which behaves as an *approximate*, but *practical* inverting differentiator over a limited frequency range.

The transfer function for a practical differentiator is:

$$H(f) = \frac{V_o(f)}{V_{in}(f)} = \frac{-R_f}{R_i + 1/(j2\pi f C)} = \frac{-j2\pi f R_f C}{1 + 2\pi f R_i C} \quad (4)$$

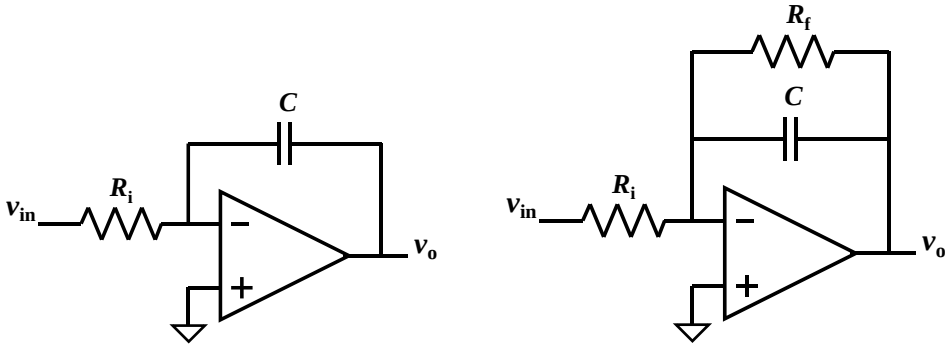
Now as $f \to \infty$, the gain is limited to $H(f) \to -R_f / R_i$, but when $f \to 0$, $H(f) \to 0$ as before. The high-pass filter's –3 dB frequency is

$$f_3 = \frac{1}{2\pi R_i C} \quad (5)$$

The single-pole high-pass filter can be used as a differentiator for frequencies below about $0.1 f_3$.

Inverting Integrator

The ideal inverting integrator in Fig. 2(a) reverses the positions of the differentiator's resistor and capacitor. Inverting integrators produce an output voltage that is proportional to the *integral* of the input signal voltage, but with inverted polarity.



(a) (b)

Fig. 2. Ideal inverting integrator (a) and practical integrator (b).

In the time domain, the ideal inverting integrator's output is:

$$v_o(t) = -\frac{1}{R_i C}\int_0^t v_{in}(t)\,dt + v_c(0) \tag{6}$$

Where $v_c(0)$ is the initial voltage on the capacitor at $t = 0$. The ideal integrator's gain as a function of ω is

$$H(j\omega) = \frac{V_o(j\omega)}{V_{in}(j\omega)} = -\frac{1}{j\omega R_i C} = \frac{1}{\omega R_i C}\angle 90°. \tag{7}$$

Letting $\omega = 2\pi f$ gives:

$$H(f) = \frac{V_o(f)}{V_{in}(f)} = -\frac{1}{j2\pi f R_i C} = \frac{1}{2\pi f R_i C}\angle 90°. \tag{8}$$

The ideal inverting integrator also has problems in practical applications because its gain $H(f) \to \infty$ as $f \to 0$. Real op amps have small dc bias currents flowing in or out of their input terminals. These bias currents are in the pA or nA range and neglected in most applications. Bias currents in an ideal integrator charge the feedback capacitor, causing the output voltage to drift and usually saturate.

Therefore, a resistor is connected in parallel with the feedback capacitor as shown in Fig. 2(b), providing an alternative path for the bias current to flow. R_f converts the integrator to a single-pole low-pass filter, which behaves as an *approximate*, but *practical* inverting integrator over a limited frequency range.

The transfer function for a practical integrator is:

$$H(f) = \frac{V_o(f)}{V_{in}(f)} = \frac{-R_f / R_i}{1 + j2\pi f R_f C} \tag{9}$$

Now as $f \to 0$, the gain is limited to $H(f) \to -R_f / R_i$, but when $f \to \infty$, $H(f) \to 0$ as before. The low-pass filter's –3 dB frequency is

$$f_3 = \frac{1}{2\pi R_f C} \tag{10}$$

The single-pole low-pass filter can be used as an integrator for frequencies above about $10 f_3$.

Pulse-Width Modulator

Fig. 3(a) shows a PWM circuit using an open-loop op amp as a voltage comparator. Negative feedback is not used in comparators, but many times *positive* feedback is used to improve noise immunity.

A triangular waveform is applied to the non-inverting input and a variable reference voltage waveform V_{ref} is applied to the inverting input. Since the op amp is a differential amplifier, it *compares* the triangular wave voltage with that of V_{ref}. The comparator output is high when the voltage at the non-inverting input is greater than V_{ref} and the output is low when the voltage at the non-inverting input is less than V_{ref}.

When V_{ref} changes, the pulse width of the output changes. Fig. 3(b) shows the superimposed triangular and dc V_{ref} input waveforms and the resulting output PWM waveform from the comparator.

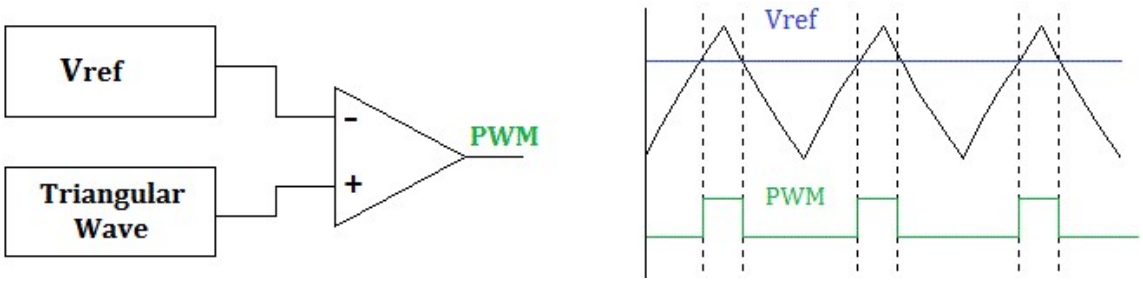


Fig. 3. Pulse-width modulator (a) and associated waveforms (b).

Equipment:

Function/Arbitrary Waveform Generator
Oscilloscope
Digital Multimeter
DC power supply with ±15 V and +5 V outputs
Cables and oscilloscope probes as needed
Solderless breadboard
Components:
Resistors (Qty): 470 Ω (1), 1.5 kΩ (1), 10 kΩ (1), 82 kΩ (1), 180 kΩ (1), 1 MΩ (1)
Op amp (Qty): LM741 (2)
Capacitors (Qty): 0.1 μF (2), 10 nF (1)

Prelab:

1. Use Eq. 1 to derive an expression for the expected output waveform from an ideal differentiator circuit having input waveform $v_{in}=1\sin[(2\pi)1000t]$ V. Let R_f=1.5 kΩ and C=10 nF.
2. Use Eq. 3 to find the peak-peak output amplitude of the ideal differentiator of question 1 for a 2 $V_{p\text{-}p}$ sine wave input at 1 kHz, 2 kHz, and 3 kHz. Put the results in the Calculated Output column of Table 1 in the Appendix.
3. Use Eq. 6 to derive an expression for the expected output waveform from an ideal integrator circuit having input waveform $v_{in}=1\sin[(2\pi)1000t]$ V. Let R_i=10 kΩ and C=10 nF. Assume $v_c(0)$ = 0.
4. Use Eq. 8 to find the peak-peak output amplitude of the ideal integrator of question 3 for a 2 $V_{p\text{-}p}$ sine wave input at 1 kHz, 2 kHz, and 3 kHz. Put the results in the Calculated Output column of Table 3.

Procedure:

General Information

1. a. Use ± V_{CC} = ± 15 V for all circuits; that is +V = 15V and -V = -15V.

   Note: If the overload light for any supply is lit, *turn off the supply immediately* and find the short circuit that caused it.

   Note: Always turn off power supplies before making circuit changes.

b. Use 0.1 μF decoupling capacitors C2 and C3 between each power supply and ground.

c. Measure input and output waveforms in peak-peak values. The oscilloscope can make the measurements automatically.

d. Leave LM741 pins 1, 5, and 8 unconnected.

Note: Be careful not to damage the op amp pins when it is inserted or removed from the breadboard. When removing, you should carefully pry it out, not pull it with your fingers. An IC puller tool is best.

e. Use Hi Res or Average acquisition mode for best oscilloscope measurements.

Inverting Differentiator Measurements

2. In the first part of this procedure you will observe the inverting differentiator's response to different types of input waveforms. In the second part you will see how it responds as a sine wave input's amplitude and frequency are changed. Construct the practical inverting differentiator circuit in Fig. 4 as follows.

   a. Place and connect components.

   - Use the breadboard busses for the power supply connections.
     - Connect C2 between the +15 V bus and a ground (GND) bus.
     - Connect C3 between the -15 V bus and a ground bus.

     Note: C2 and C3 should be within 1 inch of the LM741.
   - Connect the other components. Minimize the number of wires. *Always connect components directly to each other if you can.* For example, one end of R2 and C1 should connect directly to the 741's pin 2.

   b. Connect the power supplies to the op amp and the function generator to the v_{in} node.

   c. Connect oscilloscope channel 1 to the input and channel 2 to the output.

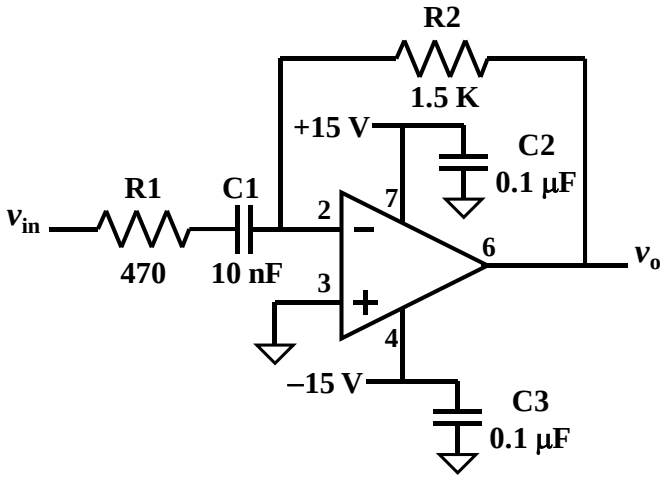


Fig. 4. Practical differentiator circuit for procedure step 2.

d. Turn on the power supply.

e. Set the function generator so that $v_{in}=1\sin[(2\pi)1000t]$ V. *Verify that the input waveform is correct using the oscilloscope.* Adjust the oscilloscope to display two periods of v_{in} and v_o.

f. Describe the output waveform as it relates to the input and record your description in Table 1. You do not have to record the amplitudes.

- For example, if the input waveform was a cosine, you would describe the output of a *non-inverting* differentiator as "inverted sine" or "–sine".
  - Remember that this is an *inverting* differentiator.
- *For the sine wave input only,* record the phase difference between the output and input waveforms in degrees and indicate whether the output is leading or lagging.
- **Save an image of the oscilloscope display.**

g. Repeat step (f) for the other waveforms in Table 1. **Be sure to save oscilloscope display images.**

i. Set the function generator so that $v_{in}=1\sin[(2\pi)1000t]$ V. *Verify that the input waveform is correct using the oscilloscope.* Adjust the oscilloscope to display two periods of v_{in} and v_o.

j. Record the output voltage in Table 2. You do not need to record the phase difference between input and output waveforms. **You do not need to save oscilloscope display images.**

k. Repeat the measurement for the other input amplitude and frequency settings in Table 2.

l. Turn off the power supply. Leave the power supply connected to the op amp.

Inverting Integrator Measurements

3. In the first part of this procedure you will observe the inverting integrator's response to different types of input waveforms. In the second part you will see how it responds as a sine wave input's amplitude and frequency are changed. Construct the practical inverting integrator circuit in Fig. 5 as follows.

   a. Place and connect components. Minimize the number of wires. *Always connect components directly to each other if you can.* For example, one end of R1, R2 and C1 should connect directly to the 741's pin 2.

   b. Connect the function generator to the v_{in} node.

   c. Connect oscilloscope channel 1 to the input and channel 2 to the output.

   d. Turn on the power supply.

   e. Set the function generator so that $v_{in}=1\sin[(2\pi)1000t]$ V. *Verify that the input waveform is correct using the oscilloscope.* Adjust the oscilloscope to display two periods of v_{in} and v_o.

   f. Describe the output waveform as it relates to the input and record your description in Table 3. You do not have to record the amplitudes.

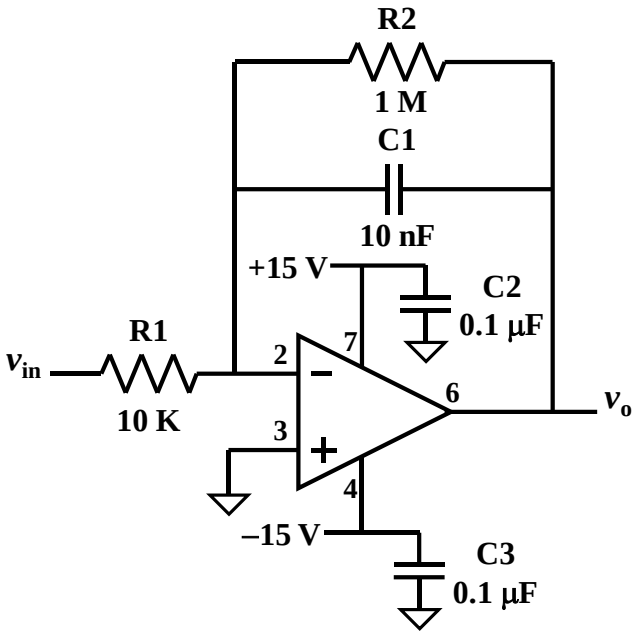


Fig. 5. Practical integrator circuit for procedure step 3.

- For example, if the input waveform was a cosine, you would describe the output of a *non-inverting* integrator as "sine".
  - Remember that this is an *inverting* integrator.
- *For the sine wave input only*, record the phase difference between the output and input waveforms in degrees and indicate whether the output is leading or lagging.
- **Save an image of the oscilloscope display**.

g. Repeat step (f) for the other waveforms in the Table 3. **Be sure to save oscilloscope display images.**

h. Set the function generator so that $v_{in}=1\sin[(2\pi)1000t]$ V. *Verify that the input waveform is correct using the oscilloscope.* Adjust the oscilloscope to display two periods of v_{in} and v_o.

i. Record the output voltage in Table 4. You do not need to record the phase difference between input and output waveforms. **You do not need to save oscilloscope display images.**

j. Repeat the measurement for the other input amplitude and frequency settings in Table 4.

k. Turn off the power supply. Leave the power supply connected to the op amp. *Do not dismantle the integrator circuit; you will use it next in the PWM circuit*.

Pulse-Width Modulator Measurements

4. Refer to the block diagram of the PWM circuit in Fig. 6. The PWM circuit uses a second LM 741 op amp as the comparator. The triangle wave output from the integrator circuit is used as one of the comparator inputs, but with a dc offset added to it. The triangle wave is connected to a resistor divider offset circuit by blocking capacitor C_b. A third adjustable power supply voltage is used as V_{ref}.

a. Temporarily disable the function generator's output or disconnect it from the integrator's input.

b. Place a second LM 741 op amp on the breadboard. *You must connect its power supply pins to* ±15 V.

c. Connect one lead of 0.1 μF capacitor C_b to the comparator's pin 3 and the other lead to the integrator circuit's output.

d. Create a voltage divider between +15 V and ground. Use a 1 MΩ resistor for R1. R2 is a series combination of a 180 kΩ and a 82 kΩ resistor. Do not connect it to C_b or the comparator yet.

- Calculate the expected voltage across R2. Measure it with the DMM and verify that it is within ±10 % of the calculated value. Record the calculated and measured values below.

  Calculated R2 voltage: _____ Measured R2 voltage: _____

- Now connect the voltage divider output to the junction of C_b and comparator pin 3.

e. Set the function generator for a 500 Hz, 1.0 Vp-p square wave (50% symmetry pulse).

- Connect the function generator's output to the integrator. Measure the peak-peak voltage at the integrator's output using oscilloscope channel 1 and record it below.

  Integrator output voltage (peak-peak): _____

f. Measure the waveform on the right side of C_b with channel 2.

- Use cursors to find the maximum and minimum voltages of the offset triangle wave on channel 2.
- **Save an oscilloscope screen image**.

Note: The lowest voltage of the triangle wave should be greater than 0 V. Check the circuit for errors if it is not.

g. Set up the oscilloscope so the comparator input and output waveforms can be observed using three channels.

- Channel 1:
  - Connect it to the node with the offset triangle wave on the right of C_b.
  - Set the vertical scale to 1 V/div.

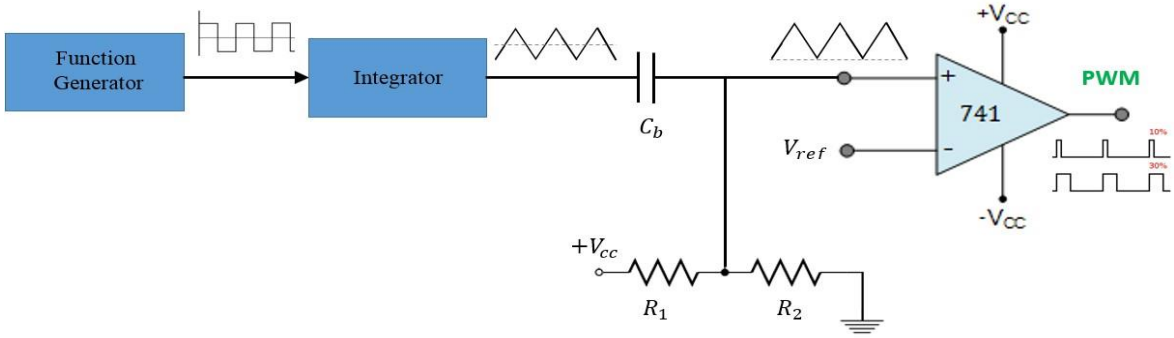


Fig. 6. PWM circuit for procedure step 4.

- Move the channel 1 reference to the first vertical division above the bottom.
- dc-couple the channel.

- Trigger the oscilloscope on channel 1 or the function generator's sync pulse.
- Channel 2:
  - Use a probe. If the probe's scale factor is selectable, make sure it is on ×10. If the oscilloscope does not automatically read the probe's scale factor, change it in the channel menu.
  - Connect it to the comparator's output PWM waveform.
  - Set the vertical scale to 10 V/div.
  - Move the channel 2 reference to the second vertical division from the top.
  - dc-couple the channel.
- Channel 3:
  - Connect it to V_{ref}.
  - Set the vertical scale to 1 V/div.
  - Move the channel 3 reference to the first vertical division above the bottom.
  - dc-couple the channel.

g. Adjust Vref to create the comparator output's duty cycle to the values in Table 5.
- Record the Vref value in the table.
- **Save an oscilloscope screen image for 25 % and 75 % duty cycles**.

5. Results documentation summary.
   a. You should have ten oscilloscope screen shots, four for the differentiator, three for the integrator, and three for the pulse-width modulator.
   b. Be sure that you recorded data and calculated results in steps you were asked to.
   c. Check with your instructor for additional documentation requirements.

Analysis:

1. Use Eq. 11 to calculate the percent error between measured and calculated differentiator output amplitudes for 2 Vp-p sine wave input voltages at 1 kHz, 2 kHz, and 3 kHz and put the results in the last column of Table 2.

$$\%\, error = \frac{measured\ voltage - calculated\ voltage}{calculated\ voltage} \times 100\% \qquad (11)$$

2. Calculate the percent error between measured and calculated integrator output amplitudes for 2 Vp-p sine wave input voltages at 1 kHz, 2 kHz, and 3 kHz and put the results in the last column of Table 4.
3. Explain how the expression calculated in prelab question 1 relates to the differentiator's output phase difference recorded in Table 1, row 1.

4. Explain how the expression calculated in prelab question 1 relates to the integrator's output phase difference recorded in Table 3, row 1.
5. From the measured data in Table 2, what is the dependence of the output amplitude on the signal frequency?
6. From the measured data in Table 2, what is the dependence of the output amplitude on the input signal's amplitude?
7. From the measured data in Table 4, what is the dependence of the output amplitude on the signal frequency?
8. From the measured data in Table 4, what is the dependence of the output amplitude on the input signal's amplitude?
9. Use Eq. 5 to calculate the high-pass –3dB frequency for the practical differentiator. Then calculate the ratio of the worst-case (highest) signal frequency you used to the high-pass f_3 frequency. Does it meet the criterion for using a single-pole high-pass filter as a differentiator?
10. Use Eq. 10 to calculate the low-pass –3dB frequency for the practical integrator. Then calculate the ratio of the worst-case (lowest) signal frequency you used to the low-pass f_3 frequency. Does it meet the criterion for using a single-pole low-pass filter as an integrator?

Appendix:

Table 1. Differentiator input and output waveform descriptions

Input Waveform Description	Output Waveform Description	Phase Difference
Sine wave		
Square wave		
Triangle (ramp with 50% symmetry)		
Sawtooth (ramp with 25% symmetry)		

Table 2. Differentiator sinusoidal input and output waveform measurements

Frequency	Input Voltage (peak-peak)	Calculated Output Voltage (peak-peak)	Measured Output Voltage (peak-peak)	Percent Error
1 kHz	2			
	4			
	5			
2 kHz	2			
	4			
	5			
3 kHz	2			
	4			
	5			

Table 3. Integrator input and output waveform descriptions

Input Waveform Description	Output Waveform Description	Phase Difference
Sine wave		
Square wave		
Ramp (0 % symmetry)		

Table 4. Integrator sinusoidal input and output waveform measurements

Frequency	Input Voltage (peak-peak)	Calculated Output Voltage (peak-peak)	Measured Output Voltage (peak-peak)	Percent Error
1 kHz	2			
	4			
	5			
2 kHz	2			
	4			
	5			
3 kHz	2			
	4			
	5			

Table 5. PWM duty cycle vs. reference voltage.

Duty Cycle	Reference Voltage, V_{ref}
5%	
25%	
50%	
75%	
95%	

Diode Characteristics

EE 3401 Laboratory Exercise
Electrical Engineering Department
Kennesaw State University

Objectives:

Students analyze, build, and test simple circuits using typical p-n junction diodes to better understand diode behavior. MATLAB is used to match an exponential function to measured diode current vs. voltage data, allowing comparison to calculated values.

Introduction:

Diodes are the fundamental semiconductor device. Understanding their behavior provides a foundation for investigating more complex devices and non-linear circuits. There are various types of diodes, such as p-n junction diode, Schottky diode, Zener diode, etc. In this lab exercise, you will investigate the behavior of p-n junction diodes and perform circuit analysis using the diode equation and the simplified *constant voltage drop* (CVD) model.

MATLAB curve-fitting capability is used to obtain a best-fit exponential function to the measured diode current vs. voltage data. The resulting exponential function's parameters are used to determine the diode's reverse saturation current I_s and emission coefficient n.

Equipment:

Agilent (Keysight) 34410A digital multimeter (DMM)
Agilent (Keysight) DSO 5014A oscilloscope or second DMM
DC power supply with ±20 V and +6 V outputs
Cables and oscilloscope probes as needed
Solderless breadboard
Components:
Resistors (Qty): 330 Ω (1), 1.0 kΩ (1)
Diodes (Qty): 1N4007 (3)

Prelab:

1. For the circuit in Fig. 2:

   Use Shockley's equation (Eq. 1) to solve for the diode current as a function of the diode voltage and fill in the "Diode Equation" column in Table 1.

$$i_D = I_S\left(e^{v_D/nV_T} - 1\right) \tag{1}$$

   Where v_D = voltage across the diode.
   I_s = reverse saturation current. Let $I_s = 15 \times 10^{-9}$ A.
   n = emission coefficient. Let $n = 2$.
   V_T = thermal voltage. Let $V_T = 26$ mV.

   Note: These parameters are realistic for a typical 1N4007 diode over the v_D measurement range in this experiment.

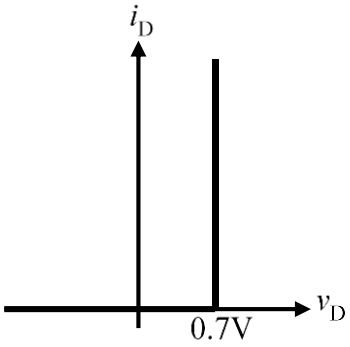


Fig. 1. Diode constant-voltage drop (CVD) model transfer curve.

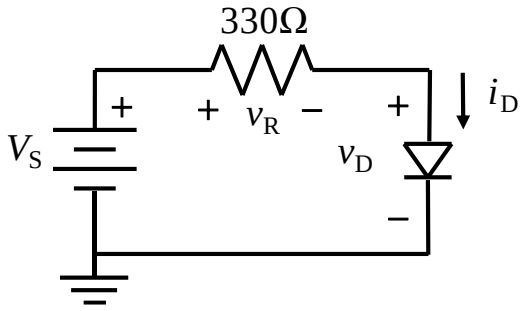


Fig. 2. Circuit for measuring the forward diode characteristic.

2. Using the constant voltage drop model:

   Complete the Prelab Calculations columns of Table 2 assuming the diode is characterized by the CVD model transfer curve given in Fig. 1.

3. For the circuit in Fig. 5:

   Complete the Prelab Calculations columns of Table 3 for the dc input voltages indicated assuming the diodes behave as in Fig. 1. In the columns labeled D_1, D_2, and D_3 list the bias condition of the specified diode ("F" for forward biased and "R" for reverse biased).

   You may find it useful to redraw the circuit and add dc voltage sources or grounds for v_1, v_2, and v_3.

Procedure:

General Information

1. When measuring the current-voltage characteristics of a diode in the next section, a DMM is used for dc current measurement and an oscilloscope for dc voltage measurements. If available, a second DMM would be a better choice to measure dc voltage.

   Instructions refer to settings for an Agilent 34410A DMM.

   Instructions refer to settings for an Agilent DSO 5014A oscilloscope. This oscilloscope can measure average voltage automatically.

   Alternatively, the dc current can be obtained from the voltage across the 330 Ω series resistor, so a single DMM can be used to measure diode current and voltage. Measure the 330 Ω resistor value so the calculated current value is more accurate.

2. When using the oscilloscope for dc voltage measurements, use Hi Res or Average acquisition mode and use the smallest V/div scale factor that keeps the trace on the screen.

Current-Voltage Characteristics of a Typical Diode

3. You will construct the circuit in Fig. 3 on your protoboard to measure the forward diode characteristic, but not just yet. Read the information below and set up the test instruments for the measurement first.

a. The lab dc power supply has three adjustable outputs: 0 to +6 V, 0 to +20 V and 0 to –20 V. V_S in Fig. 3 uses the +6 V output. The black COM terminal serves as the reference for all three and the circuit's 0V ground.

   Set up the power supply as follows.

   - Make sure nothing is connected to its terminals.
   - Push the +6 V button in the Meter section so that the meter will monitor that supply voltage and current.

     Note: The three buttons in the meter section select the supply the meter monitors; they do not turn that supply on or off. The power switch turns all supplies on or off.
   - Set the +6 V and ±20 V to 0 V by turning their control knobs fully counterclockwise. Turn the Tracking Ratio knob fully clockwise.
   - Turn off the power supply.

b. Configure the oscilloscope to measure v_D on Ch. 1. Use the following settings.

   - Ch. 1 menu: choose DC coupling and BW Limit.
   - Vertical scale: select 100 mV/div.
   - Horizontal scale: select some convenient value like 1 ms/div.
   - In the Trigger > Mode/Coupling menu: select Auto.

     Note: You cannot use Normal triggering because the oscilloscope will not trigger on the dc waveform (except for noise artifacts); and it will not display a trace if it cannot trigger.
   - In the Waveform>Acquire menu: choose Hi Res or Average, whichever results in the least noise on the trace.
   - In the Quick Meas menu select Source > Ch. 1 and Measure > Average.

     Note: The average value is the dc value.

c. Configure the as an ammeter to measure i_D. This requires connecting to the meter red I and black LO terminals as in Fig. 4.

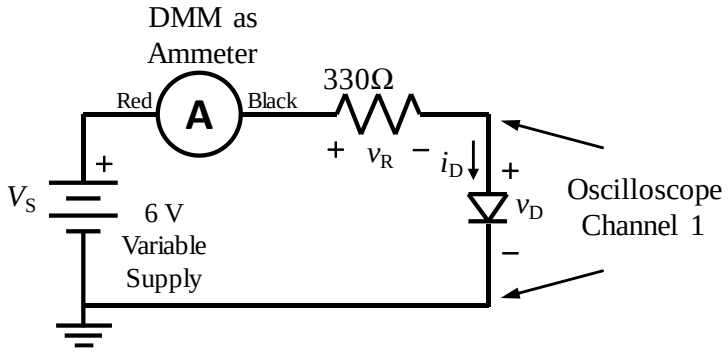


Fig. 3. Circuit for measuring the diode forward characteristic.

Fig. 4. Multimeter configured as an ammeter to measure DC current.

Note: The ammeter must be connected in series with the current path. Never connect an ammeter in parallel with a component.

- Choose dc current mode by pressing and releasing Shift, then pressing DC I, which appears above the DC V button.

  Have your instructor verify the ammeter location and operating mode are correct.

d. Build the Fig. 3 circuit using a 1N4007 diode. Include the multimeter/ammeter. Minimize the number of wires to avoid mistakes. Connect oscilloscope channel 1 across the diode with the ground lead connected to common.

4. Measure the diode forward transfer characteristics as follows.

   a. Turn on the power supply.

   b. Adjust V_S to set v_D as measured on the oscilloscope to the values in Table 1 as accurately as possible. *Start with v_D = 0.7 V first so you can check that your measurements are OK*. Record i_D from the meter and v_D from the oscilloscope.

      Note: Do not set your power supply voltage to the column 1 v_D values. Your power supply voltage settings will be higher than v_D due to the voltage drop through the resistor. The exception is row 1.

The Constant Voltage Drop Model

5. The circuit of Fig. 3 is also used in this part. V_S uses the +6 V output of the DC power supply.

   a. Turn off the power supply.

   b. Turn the +6 V control knob fully counterclockwise, setting it to 0 V.

   c. For the first four rows, adjust the supply to the values in the first column. Each time, record v_D, i_D. and v_R in Table 2. Where $v_R = V_S - v_D$.

   d. For the last two rows, you must reverse the polarity of the +6 V supply to create negative voltages. Just switch the two cables on the +6 V and COM connectors. *Measure the diode voltage with the oscilloscope and then remove the oscilloscope connection and measure the current.* Do not measure current; assume it 0.

      Note: The oscilloscope input impedance is much less than that of the reverse-biased diode. It appears in parallel with the diode, so leaving the oscilloscope connection in place when making the current measurement is a mistake. In fact, the typical ammeter is not capable of

measuring the actual reverse biased diode current, so these values are 0 as far as this experiment is concerned.

Large-Signal Diode Characteristics

6. Construct the circuit in Fig. 5 and measure its performance as follows. The power supply COM terminal is the measurement ground reference.

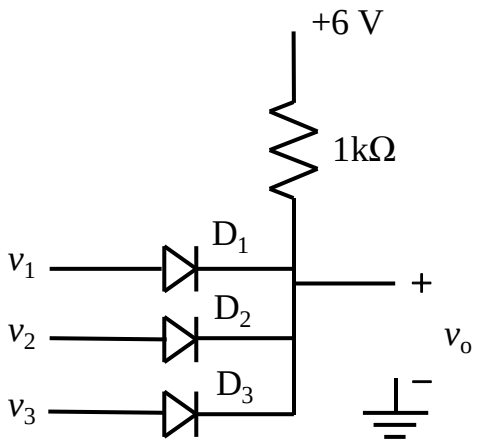


Fig. 5. Large-signal diode circuit

   a. Turn off the power supply.

   b. Assemble the circuit on the breadboard.

   c. Connect the +6 V, +20 V, –20 V and COM terminals to the breadboard binding posts. Then connect wires from the binding posts to the circuit so that you can connect the appropriate voltages to the diodes.

   d. Monitor the output voltage with the multimeter connected as a DC voltmeter. Select the DCV mode by pressing that button.

   e. Measure the output voltage v_O for the input voltages v_1, v_2, and v_3 of Table 3.

      Note: You need to change the tracking ratio to get a negative voltage from the –20V supply with magnitude less than that of the +20V supply. Set the positive voltage to the desired value, then while monitoring the negative voltage, change the tracking ratio until you reach the desired value. The magnitude of the negative voltage is always less than or equal to the magnitude of the positive voltage.

7. Results documentation summary.

   a. Be sure that you recorded data and calculated results in steps you were asked to.

   b. Check with your instructor for additional documentation requirements.

Analysis:

1. For this question, you will plot three diode characteristics. One is the calculated data from Eq. 1, the second is the measured data, and the third is an exponential curve that is fit to the measured data.

   Refer to the partial MATLAB file in the appendix. The file has zero-filled arrays *vdmeas* and *idmeas* for measured v_D, and i_D data. Replace the data in those arrays with your measured data from Table 1.

   The program uses the "exp1" exponential fitting function to create an exponential function to match your data as closely as possible. The resulting fitted function is given by Eq. 2, where the *a* and *b* terms are found by the algorithm.

$$f = ae^{bx} \tag{2}$$

   Once the fitting function is calculated, the program creates an array *vd1* of finely-spaced voltage points from 0 V to 0.7 V. Then the program uses Eq. 2 to find the fitted current array *id1*.

   The program plots the measured data (*idmeas* vs. *vdmeas*) as 'o' points and the fitted current array *id1*.

   Add statements to the program to calculate Eq. 1 using the voltage array *vd1* as input. Call your array of currents *idcalc*. Modify the plot statement so all three curves are plotted. Fig. B-1 is an example plot.

   Generate the plot and indicate which curve is from Eq. 1.

   Compare the curves and account for differences.

2. In the MATLAB command window, after running the curve matching and plotting code, type "f" at the prompt followed by Enter. The *a* and *b* values of Eq. 2 are displayed. Use these values to find estimates of the reverse saturation current I_S and the emission coefficient *n*. Let V_T = 26 mV.

3. Compare the calculated CVD model values and the measured values for the diode current i_D of Table 2 by calculating the % error with the theoretical values as reference (Eq. 3). Put the percent errors with 2 digits of precision in the last column of Table 2. Explain the errors.

$$\%Error = \frac{Measured - Theoretical}{Theoretical} \times 100\% \tag{3}$$

4. Consider the Table 3 data. Explain why the three diodes are biased the way they are for the case where the input voltages are 12V, 0V, and 6V and the case where the inputs are all 12V.

Appendix 1: Data Tables

Table 1. 1N4007 forward-bias characteristics.

Nominal v_D (V)	Diode Equation i_D (mA)	Measured v_D (V)	Measured i_D (mA)
0			
0.2			
0.4			
0.45			
0.5			
0.55			
0.6			
0.65			
0.7			

Table 2. Series diode circuit measurements and the CVD model data.

	Prelab Calculations			Experimental			
V_S (V)	CVD Model v_D (V)	CVD Model v_R (V)	CVD Model i_D (mA)	Measured v_D (V)	Calculated v_R (V)	Measured i_D (mA)	i_D Percent Error (Eq. 3)
0.5							
1							
5							
-1.0					Omit	0	Omit
-5.0					Omit	0	Omit

Table 3. Large signal diode circuit data.

Input Voltages			Prelab Calculations				Experimental
v_1 (V)	v_2 (V)	v_3 (V)	D_1 Bias (expected)	D_2 Bias (expected)	D_3 Bias (expected)	v_o Calculated	v_o Measured
0	0	0					
-12	6	12					
12	0	6					
12	12	12					
-8	12	0					

Appendix 2: MATLAB Code for Curve Fitting and Plotting

```
% Diodecurv.m
% Diode exponential curve matching and plotting
%
% Plots measured data and exponential curve fit to measured data
%
%
clear
idmeas=[0 0 0 0 0 0 0 0 0];  %Measured id values in amps for testing fit
vdmeas=[0 0 0 0 0 0 0 0 0];  %Measured vd values for testing fit

% Create the exponential fit to measured data
f=fit(vdmeas',idmeas','exp1','TolFun',1e-10);  %Tighter tolerance than default

vd1=[0:0.005:0.7];  %Create voltage point array for matched function values
id1=f(vd1);  %Calculate matched function values

% The current values are plotted in mA by scaling x1000
plot(vdmeas,idmeas*1000,'o',vd1,id1*1000)
title('Diode Forward Characteristic')
xlabel('V_{D}, V')
ylabel('I_{D}, mA')
grid
```

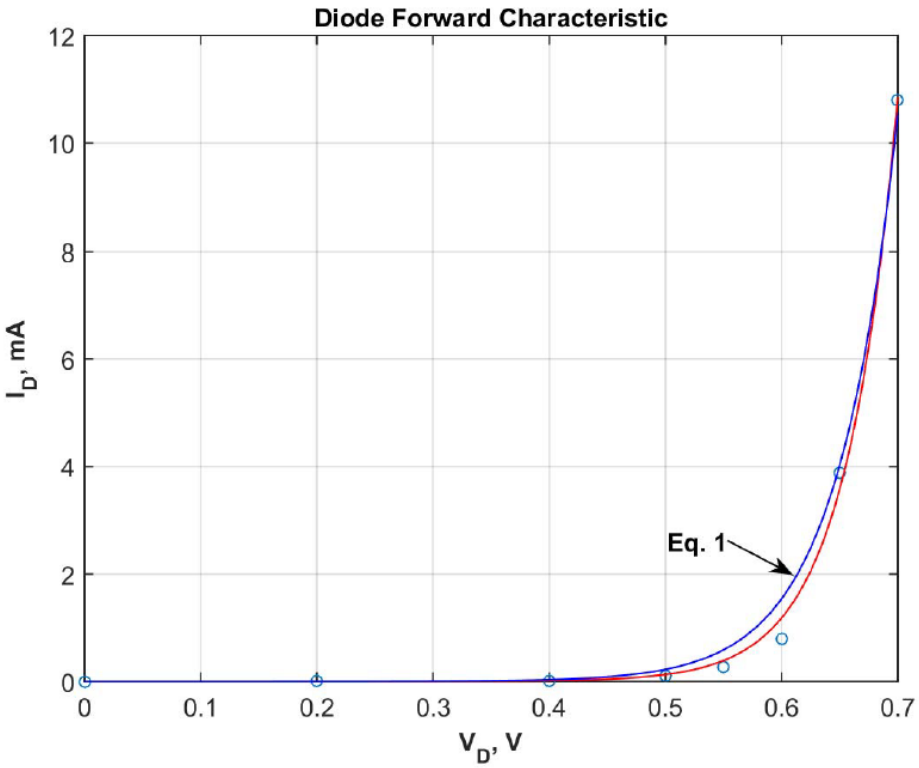


Fig. B-1. Example plot.

Diode Rectifiers and Regulators

EE 3401 Laboratory Exercise
Electrical Engineering Department
Kennesaw State University

Objectives:

Half-wave and full-wave rectifier circuits and a Zener diode regulator circuit as well as the performance improvement of the rectifier circuits caused by adding a filter capacitor are investigated. The Zener diode regulator's performance with different input voltages is examined. Theoretical and measured results are compared.

Introduction:

Half- and full-wave rectifiers are commonly used in power supply circuits. Rectifying an ac waveform creates a dc component. Filtering the rectified ac waveform with a capacitor increases the dc level and reduces harmonic content, improving the quality of the rectifier's output.

Let the half-wave rectifier circuit in Fig. 1a have input waveform $v_I = V_m \sin(2\pi f t)$ with period $T = 1/f$. The resulting output waveform is shown in Fig. 1b with the horizontal axis scaled both in time and angle. The average, or dc value of the periodic output waveform is

$$V_{avg} = V_{dc} = \frac{1}{T}\int_{t_1}^{t_2}\left[V_m \sin\left(2\pi f t\right) - V_d\right]dt \tag{1}$$

Where V_m is the sine wave amplitude and V_d is the diode's forward voltage drop. The peak output voltage is $v_{o,peak} = V_m - V_d$. Integration limits t_1 and t_2 are the times the diode starts and ends conduction. Normalizing the period to angle 2π radians simplifies the integration because the period and the limits integration limits θ_1 and θ_2 are independent of frequency and time period T. Therefore,

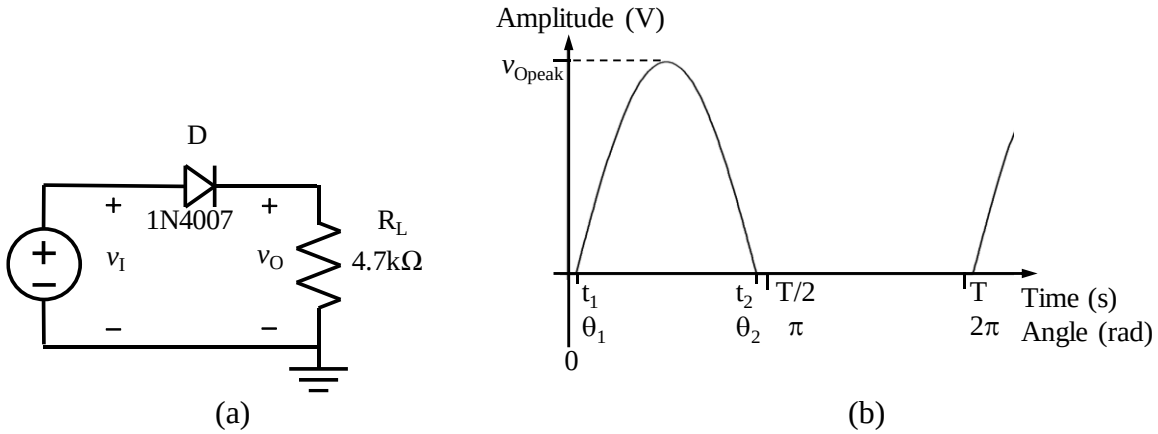


Fig. 1. Half-wave rectifier (a) circuit and (b) output waveform.

$$V_{avg} = \frac{1}{2\pi}\int_{\theta_1}^{\theta_2}\left[V_m \sin(\theta) - V_d\right]d\theta \tag{2}$$

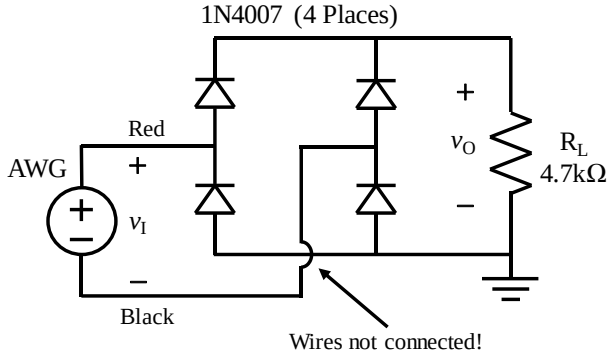
The diode begins conducting at angle θ_1 and it turns off at and θ_2:

$$\theta_1 = \sin^{-1}\left(V_d / V_m\right) \quad \text{and} \quad \theta_2 = \pi - \theta_1 \tag{3}$$

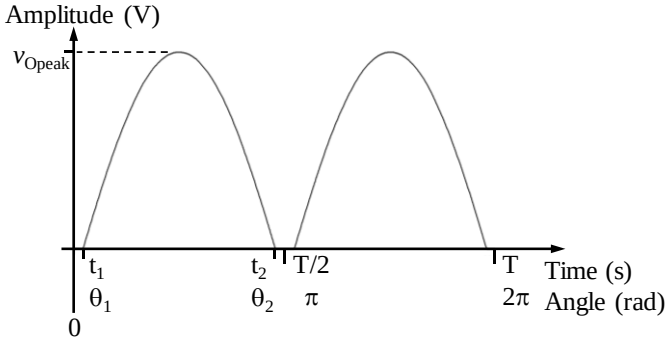
The difference $\varphi = \theta_1 - \theta_2$ is the *conduction angle* of the half-wave rectifier.

A full-wave bridge rectifier circuit is shown in Fig. 2a with a "floating" signal generator and grounded load resistor, which is the most common configuration of this rectifier. Its output waveform appears in Fig. 2b. Note that the output frequency is twice the input frequency, but the time period *T* and angular period 2π on the horizontal axis refer to the input waveform. Following a similar development to the half-wave rectifier, the dc output voltage from the full-wave bridge rectifier is

$$V_{avg} = \frac{1}{\pi}\int_{\theta_1}^{\theta_2}\left[V_m \sin(\theta) - 2V_d\right]d\theta \tag{4}$$



(a)



(b)

Fig. 2. Full-wave bridge rectifier (a) circuit and (b) output waveform. Time and angular periods refer to the input waveform.

where the integration is over half the original waveform's period and the $2V_d$ term is due to the load current flowing through two diodes. The peak output voltage is $v_{o,peak} = V_m - 2V_d$. The integration limits are found using Eq. 5 with the angles referred to the original input waveform.

$$\theta_1 = \sin^{-1}\left(2V_d / V_m\right) \quad \text{and} \quad \theta_2 = \pi - \theta_1 \tag{5}$$

However, the full-wave rectifier's conduction angle must be referred to the output waveform's period, which is half that of the input so $\varphi = 2(\theta_1 - \theta_2)$.

If a large filter capacitor is added in parallel with the load resistor for either the half- or full-wave rectifier, the resulting ripple waveform is approximately a sawtooth superimposed on the dc level. Fig. 3 shows a filtered full-wave bridge rectifier circuit and the peak-peak ripple waveform. Each "tooth" in the ripple waveform can be approximated as a right triangle with maximum value equal to $v_{o,peak}$. It can be shown that the approximate *peak-peak* ripple amplitude for the filtered half-wave rectifier is

$$V_{r,p-p} \approx \frac{v_{o,peak}}{fRC} \quad \text{(half-wave)} \tag{6}$$

And the approximate peak-peak ripple amplitude for the filtered full-wave rectifier is

$$V_{r,p-p} \approx \frac{v_{o,peak}}{2fRC} \quad \text{(full-wave)} \tag{7}$$

Where f is the input waveform's frequency in Eqs. (6) and (7). In either case, the filtered rectifier's dc level is approximately

$$V_{avg} \approx v_{o,peak} - \frac{V_{r,p-p}}{2} \tag{8}$$

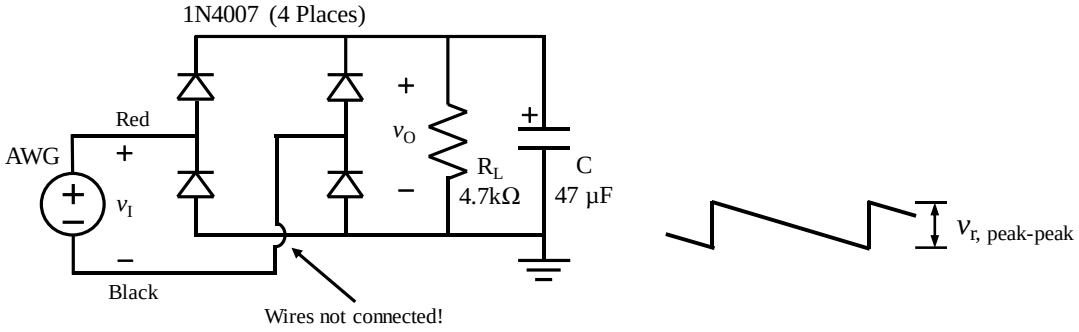


Fig. 3. Full-wave bridge rectifier with filter capacitor (a) circuit and (b) output ripple waveform.

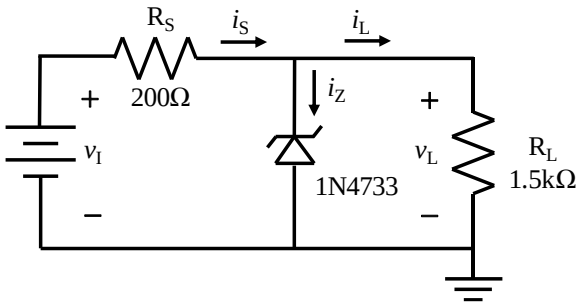


Fig. 4. Zener diode voltage regulator.

A Zener diode is sometimes used as a simple dc voltage reference or voltage regulator using a circuit like the one in Fig. 4. It is an inexpensive alternative to monolithic reference and regulator integrated circuits, but the performance is not as good. The Zener diode's incremental ac resistance makes this type of regulator's output voltage vary with load current and applied input voltage. A good voltage regulator acts like a voltage source so its output resistance must be very low, ideally 0. The Zener diode used here has an incremental resistance of a few ohms, which while low, is still too high to consider it as behaving like a voltage source.

The Zener diode's approximate incremental resistance r_z can be calculated from measurements of the diode's voltage and current at two different operating points (v_{z1}, i_{z1}) and (v_{z2}, i_{z2}).

$$r_z \approx \frac{\Delta v_z}{\Delta i_z} = \left| \frac{v_{z1} - v_{z2}}{i_{z1} - i_{z2}} \right| \tag{9}$$

Equipment:

Function/Arbitrary Waveform Generator (AWG)
Oscilloscope
Digital Multimeter (DMM)
DC power supply with +12 V output
Cables and oscilloscope probes as needed
Solderless breadboard
Components:
- Resistors (Qty): 200 Ω (1), 1.5 kΩ (1), 4.7 kΩ (1)
- Capacitor (Qty): 47 μF (1)
- Diodes (Qty): 1N4007 (4), 1N4733A (1)

Prelab:

Assume the following for calculations in prelab steps 1-3.

- The function generator (AWG) produces a waveform $v_I(t) = 10 \sin(2\pi(60)t)$ when it is loaded as indicated.
- Use the constant voltage drop (CVD) model for the diodes with $v_{fwd} = V_d = 0.7$ V.

1. Perform the following calculations for the half-wave rectifier in Fig. 1. Put your results in Table 1.
   a. Determine $v_{o,peak}$, θ_1, and θ_2, for the rectifier output waveform of Fig. 1b.
   b. Calculate the average (dc) value of $v_o(t)$ using Eq. 2.
   c. Calculate the conduction angle φ.
2. Perform the following calculations for the full-wave rectifier in Fig. 2. Put your results in Table 1.
   a. Determine $v_{o,peak}$, θ_1, and θ_2 for the rectifier output waveform of Fig. 2b.
   b. Calculate the average (dc) value of $v_o(t)$ using Eq. 4.
   c. Calculate the conduction angle φ.
3. Perform the following calculations for the full-wave rectifier with filter capacitor in Fig. 3. Put your results in Table 1.
   a. Determine the maximum value of $v_o(t)$, $v_{o,peak}$.
   b. Determine the peak-to-peak ripple voltage, $v_{r,\ peak\text{-}peak}$.
   c. Determine the approximate average value of $v_O(t)$. You do not have to use integration. Assume the ripple waveform has an ideal sawtooth shape with each "tooth" being a right triangle having amplitude $v_{r\ peak\text{-}peak}$ as in Fig. 3b.
   d. Sketch a *realistic* output waveform including ripple and superimpose it on the unfiltered full-wave rectifier output waveform. Label the maximum and minimum output voltages, not just the peak-peak value.
4. Perform the following calculations for the Zener-diode voltage regulator in Fig. 4. Use the CVD model for the diode in reverse bias assuming it has a constant voltage drop of 5.1 V. Put your results in Table 2.
   a. If v_I = 8 V dc, determine i_S, i_L, i_Z, and v_{RL}, the power dissipated in R_S and the power dissipated in the Zener diode.
   b. Repeat part (a) for v_I = 12 V dc.

Procedure:

General Information

1. a. Some oscilloscope measurement functions refer to command sequences for the Agilent (Keysight) DSO 5014A.
   b. When using the oscilloscope for dc voltage measurements, use Hi Res or Average acquisition mode and use the smallest V/div scale factor that keeps the trace on the screen.
   c. The function generator referenced in the instructions is the Agilent (Keysight) 33220A. This generator's output waveform connector is floating (not connected to power system ground). This enables it to be used as a floating sine wave source for a full-wave bridge rectifier.

Half-Wave Rectifier

2. Perform measurements of the half-wave rectifier circuit shown in Fig. 1 as follows.

a. Construct the circuit shown in Fig. 1.

b. Set up the function generator.

- Select a 60 Hz, 20 Vp-p sine wave.
- Make sure the function generator is in High-Z mode.
- Enable the function generator output.

c. Set up the oscilloscope.

- Monitor the function generator output waveform on the oscilloscope Ch. 1 and the resistor voltage on Ch. 2.
- Use DC coupling on both channels.
- Trigger on Ch. 1 using Auto mode.
- Display at least 2 periods and line up the channel ground references together so the waveforms are superimposed.

d. Have the oscilloscope measure the peak (not peak-peak) values of the Ch. 1 and Ch. 2 waveforms and the average value and of the Ch. 2 waveform.

- Verify that the input waveform's amplitude is 20 Vp-p.
- Have your instructor verify that the display is correct.
- Save a copy of the oscilloscope display.
- Determine the difference between the Ch. 1 and Ch. 2 peak amplitudes.

  Amplitude difference between input and output: _____

Full-Wave Rectifier

3. Perform measurements of the full-wave rectifier circuit shown in Fig. 5 as follows.

   a. Construct the circuit shown in Fig. 5.

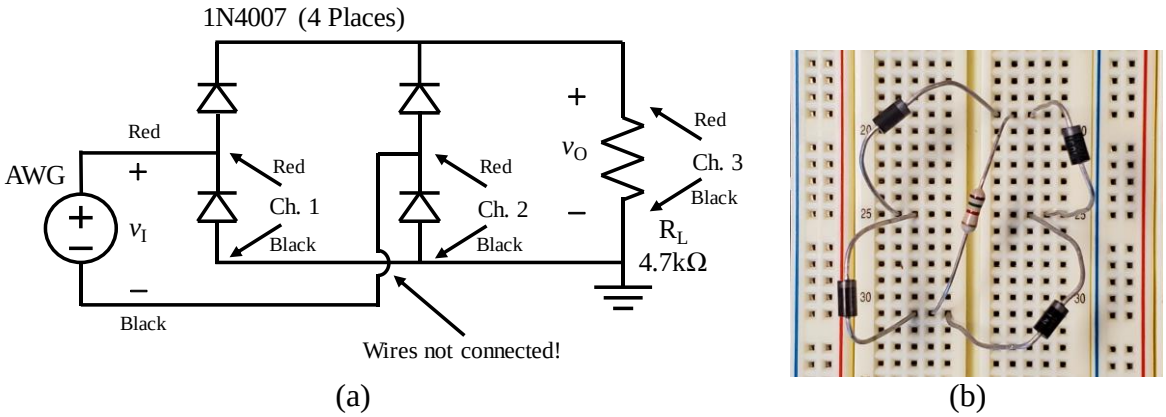
   Note: You cannot connect the function generator to ground in this circuit. It must "float". If you do the rectifier will not work.

   Since oscilloscope connectors are connected to power line ground internally, that establishes the circuit 0 V ground reference.

   For function generators that are connected to ground internally, the full-wave rectifier circuit can be configured for a floating load and the load voltage measured differentially.

   b. Set up the function generator.

   - Select a 60 Hz, 20 Vp-p sine wave.
   - Make sure the function generator is in High-Z mode.
   - Enable the function generator output.



(a) (b)

Fig. 5. Full-wave rectifier circuit (a) schematic showing oscilloscope connections and (b) suggested breadboard layout.

c. Set up the oscilloscope.

- You will use a differential measurement technique to monitor the function generator output because you cannot connect oscilloscope ground leads to either function generator lead. You can use test leads or oscilloscope probes for the connections.
    - Connect Ch. 1 from the function generator "Red" lead to circuit ground at the resistor. Use DC coupling.
    - Connect Ch. 2 from the function generator "Black" lead to the circuit ground at the resistor. Use DC coupling.
    - Connect Ch. 3 across the resistor. Use DC coupling.
    - Set the Ch. 1, 2, and 3 vertical scale factors to be the same, like 5 V/div.
    - Trigger on Ch. 1 using Auto mode.
    - Use the Math function and subtract Ch. 2 from Ch. 1 (Ch. 1 – Ch. 2). A new waveform of a different color will appear. Turn off Ch. 1 and Ch. 2 by pressing the "1" and "2" buttons each twice. The Math waveform will still be displayed.
    - In the Math menu you can change the displayed waveform scale factor and offset.
- Display at least 2 periods and line up the channel ground references together so the waveforms are superimposed.

d. Have the oscilloscope measure the peak (not peak-peak) values of the Math and Ch. 3 waveforms and the average value and of the Ch. 3 waveform. Use Quick Meas > Select Meas > Max for the peak measurement.

- Have your instructor verify that the display is correct.
- Save a copy of the oscilloscope display.
- Determine the difference between the Math and Ch. 3 peak amplitudes.

  Amplitude difference between input and output: _____

Filtered Full-Wave Rectifier

4. Perform measurements of the full-wave rectifier circuit with filter capacitor as follows.
   a. The full-wave rectifier in Fig. 5 is modified by connecting a 47 μF capacitor in parallel with the load resistor.
      - The capacitor is an electrolytic type and must be connected so the DC voltage has the right polarity. Look at the capacitor case and you will see some "–" signs near one of the leads. **This lead must be connected to the low voltage side, in this case the circuit ground**.

        Note: Connecting an electrolytic capacitor backwards could damage it. If the current is high enough it can fail catastrophically.

   b. Disable or disconnect the function generator output (don't turn the function generator off) and connect the capacitor across the load resistor with the correct polarity.
   c. Enable the function generator output and observe the Ch. 3 waveform. Because the ripple amplitude is small, the output voltage is mostly dc. The average value of the Ch. 3 waveform should be higher than it was in step 2.

      Note: When the capacitor charges near the top of the rectified sine wave peaks, it acts like a low impedance. The 50 Ω internal impedance of the function generator will drop a significant amount of the voltage. The result is distorted peaks in the Math waveform.

   d. Save a copy of the display with the Ch. 3 and Math waveforms superimposed.
   e. To measure the ripple waveform more carefully, change Ch. 3 to AC coupling. It will then center the waveform at the Ch. 3 ground reference. Change the Ch. 3 scale factor so the ripple is obvious. Have the oscilloscope measure the Ch. 3 peak-peak amplitude this time.
   f. Save a copy of the display with just the Ch. 3 waveform displayed.
5. Perform measurements of the Zener diode voltage regulator circuit in Fig. 4 as follows.
   a. Construct the circuit.
   b. Use the +20 V output of the dc power supply. Set it to +8 V first.
   c. Use the multimeter as a voltmeter and measure v_L and v_{RS} (the voltage across R_S). Use these values to calculate currents i_S, i_L, i_Z. Enter the appropriate values in Table 2.

      v_L: _____ v_{RS}: _____
   d. Calculate the power dissipated in R_S and in the Zener diode and put the results in Table 2.
   e. Set the power supply voltage to +12 V and repeat steps (c) and (d).

      v_L: _____ v_{RS}: _____
6. Results documentation summary.
   a. You should have three oscilloscope screen shots, one each from steps 2, 3, and 4.
   b. Be sure that you recorded data and calculated results in steps you were asked to.
   e. Check with your instructor for additional documentation requirements.

Analysis:

1. Consider the half-wave rectifier. Compare theoretical and measured results for the average and the peak output voltages using percent error. Use the theoretical value as reference by subtracting it from the measured value and dividing that result by the theoretical value; then convert to percent.
2. Consider the full-wave rectifier without filter capacitor. Compare theoretical and measured results for the average and the peak output voltages using percent error.
3. Consider the full-wave rectifier with filter capacitor. Compare theoretical and measured results for the average output voltage and the peak-peak ripple voltage using percent error.
4. Determine the approximate diode turn-on voltage from the half-wave rectifier waveforms.
5. How might the function generator's internal impedance affect the filtered full-wave rectifier's average output voltage?
6. Consider the Zener diode voltage regulator. Use percent error to compare theoretical and measured values of v_L, i_Z, and P_Z at $v_I = 12$ V.
7. Consider the Zener diode voltage regulator. Use data measured at $v_I = 8$ V and 12 V to calculate the diode on resistance r_Z. Does the Zener diode voltage regulator act as a good voltage source (source resistance less than 1 Ω)?

Reference:

[1] A. Sedra and K. Smith, *Microelectronic Circuits*, 7ed., Oxford University Press, New York, 2015.

Appendix:

Table 1. Calculated and measured rectifier data.

Rectifier	Prelab Calculations				Experimental		
	$v_{o,peak}$ (V)	Conduction Angle, φ (degrees)	V_{avg} (V)	$V_{r,p-p}$ (V)	$v_{o,peak}$ (V)	V_{avg} (V)	$V_{r,p-p}$ (V)
Half-wave							
Full-wave							
Filtered full-wave							

Table 2. Zener diode voltage regulator data.

	Prelab Calculations						Experimental					
v_I (V)	i_S (mA)	i_Z (mA)	i_L (mA)	v_L (V)	P_{RS} (mW)	P_Z (mW)	i_S (mA)	i_Z (mA)	i_L (mA)	v_L (V)	P_{RS} (mW)	P_Z (mW)
8												
12												

Table 3. Errors between measured and calculated rectifier voltages.

Rectifier	Peak Voltage Error (%)	Avg. Voltage Error (%)	Peak-Peak Ripple Voltage Error (%)
Half-wave			
Full-wave			
Filtered full-wave			

Table 4. Errors between measured and calculated Zener diode circuit parameters at v_I = 12 V.

Parameter	Error (%)
Load voltage, v_L	
Zener diode current, i_Z	
Zener diode power, P_Z	

MOSFET Behavior

EE 3401 Laboratory Exercise
Electrical Engineering Department
Kennesaw State University

Objectives:

You will study MOSFET behavior by measuring and analyzing NMOS transistor behavior in the saturation region. MATLAB is used to fit curves to the measured data. In addition, simple logic circuits using NMOS transistors are constructed and the transistor behavior in the ohmic region is observed.

Introduction:

The BS170 is designed for low-voltage, low-current switching applications such as power MOSFET gate drivers. You will measure its dc transconductance curve I_D vs. V_{GS} and two common-source curves in the saturation region. The I_D vs. V_{GS} curve for large MOSFETs like the BS170 is parabolic and Eq. 1 should be a close approximation. Using MATLAB, a parabolic equation is matched to the transconductance curve and V_{tn} determined. MATLAB is also used to fit lines to the measured common-source curves and values of channel-length modulation factor λ and drain-source resistance r_o are found. Transconductance g_m at a fixed V_{DS} is calculated.

$$I_D = \frac{k_n'}{2}\left(\frac{W}{L}\right)\left(V_{GS} - V_{tn}\right)^2\left(1 + \lambda V_{DS}\right) \qquad V_{GS} > V_{tn} \tag{1}$$

Two simple logic circuits are built and their voltage input-output tables are measured. The $r_{DS(on)}$ resistance of the transistors is also calculated.

Equipment:

Digital Multimeter (DMM)
DC power supply with +6 V and +20 V outputs
Cables as needed
Solderless breadboard
Components:
- Resistors (Qty): 1 kΩ (2)
- MOSFET (Qty): BS170 (2)

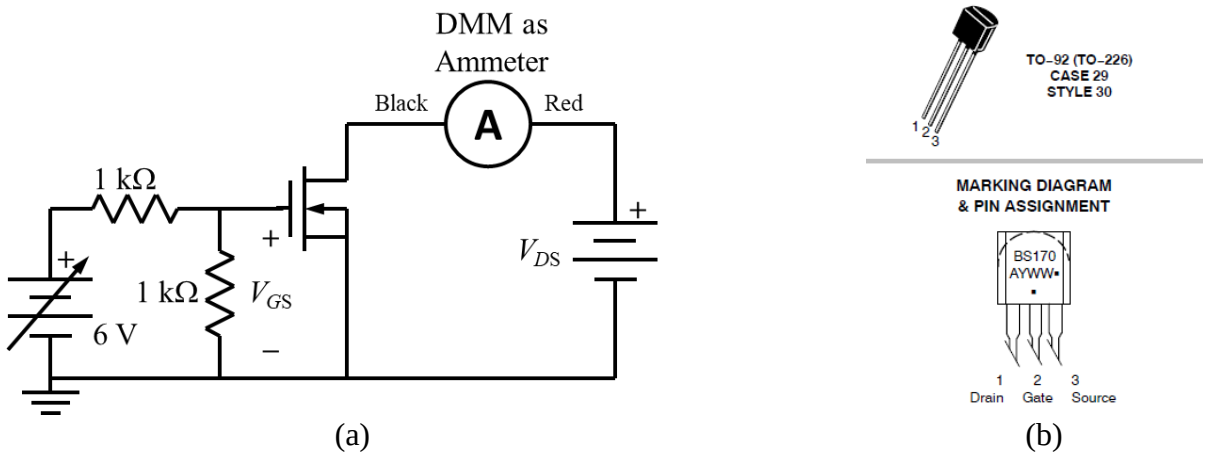
Procedure:

General Information

1. Measurements like the ones in this lab exercise can result in too much power dissipation in the transistor if you are not careful. **The continuous power rating of the BS170 is 500 mW**.

I_D vs. V_{GS} Measurements

2. Measure the BS170 I_D vs. V_{GS} transconductance characteristics for constant V_{DS} using the circuit in Fig. 1 as follows.
   a. Construct the circuit in Fig. 1 using the +6V supply for V_{GS} and the +20 V supply for V_{DS}. Make sure the supplies are set to 0 V before connecting them to the circuit.



(a) (b)

Fig. 1. NMOS transistor (a) test circuit and (b) package drawing.

b. Connect the multimeter *in series* with V_{DS} to measure the drain current I_D.

c. Use the 6 V power supply's built-in meter to calculate V_{GS}. Since the two 1 kΩ resistors form a voltage divider with ration 0.5, V_{GS} is half the supply voltage.

d. Set V_{DS} to 4.0 ± 0.1 V. Then adjust V_{GS} to produce I_D current values close to the "nominal" values listed in Appendix 1, Table 1. You do not have to set the current precisely, but try to be within 15%. Record V_{GS} and I_D using 3 digits of precision.

Note: Do not let I_D exceed 45 mA.

Note: The current may drift due to temperature effects, so record your data quickly after you set the current values.

e. Turn the power supply voltages to 0 V.

I_D vs. V_{DS} Measurements

2. Measure BS170 common-source I_D vs. V_{DS} transfer curves for two constant V_{GS} values using the circuit in Fig. 1 as follows.

a. Use the same test instrument setup as for step 1.

b. Set V_{DS} to 4.0±0.1 V and adjust V_{GS} until I_D = 5 ± 1 mA. Record the value of V_{GS} in the title of Table 2 with 3 digits of precision. Record the I_D and V_{DS} values in the table to 3 digits of precision.

c. Set V_{DS} to the Table 2 column values and record I_D and V_{DS}. Be sure to keep V_{GS} at the same setting as step (b) for each measurement.

d. Set V_{DS} to 4.0±0.1 V and adjust V_{GS} until I_D = 10 ± 1 mA. Record the value of V_{GS} in the title of Table 3 with 3 digits of precision. Record the I_D and V_{DS} values in the table to 3 digits of precision.

e. Set V_{DS} to the Table 3 column values and record I_D and V_{DS}. Be sure to keep V_{GS} at the same setting as step (d) for each measurement.

Logic Circuit and Ohmic-Region Measurements

3. Characteristics of two MOSFET logic gates are measured as follows. It is desired that MOSFETs in logic circuits operate either in the triode (ohmic) or cutoff regions.
   a. Construct the circuit of Fig. 2. Use the 6V supply for the 5V power supply and the two input voltages. Use the multimeter to measure V_{out} with 2 digits of precision.
   b. Apply the input voltage levels for V_1 and V_2 and record the output voltage V_{out} as shown in Table 4.
   c. Construct the circuit of Fig. 3 and repeat the measurements, recording data in Table 5.
4. Results documentation summary.
   a. Be sure that you recorded data and calculated results in steps you were asked to.

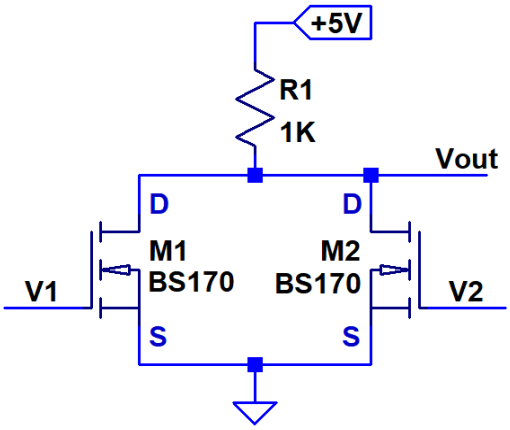


Fig. 2. Logic circuit 1.

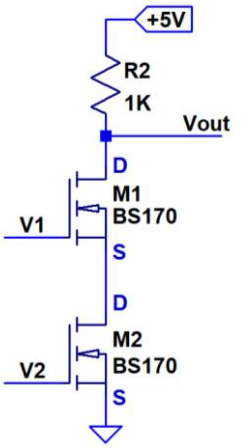


Fig. 3. Logic circuit 2.

   e. Check with your instructor for additional documentation requirements

Analysis:

1. You will plot all your measured dc transconductance curve data points from Table 1 and fit a parabola to a subset of the data using MATLAB or similar software. If done well, the threshold voltage, V_{tn}, occurs at the parabola's minimum value, where ideally $I_D = 0$, as predicted by Eq. 1.

   An example MATLAB file is included in Appendix 2 for curve fitting and plotting. Replace the zeros in the *idmeas*, *vgsmeas*, *idfit*, and *vgsfit* arrays (known as vectors in MATLAB) with your data. You may have more data points than the number of zeros in the arrays.

   Fig. 4 is an example of the resulting plot but with a different V_{GS} increment and range. The "polyfit" function calculates the parabola's coefficients and assigns them to the array variable *p*. Eq. 2 is the resulting parabola's equation using program variables, where the *p*(*n*) are constants in the *p* array with array index *n*. See MATLAB's online documentation for specifics.

$$I_D = p(1)(V_{GS})^2 + p(2)(V_{GS}) + p(3) \quad \text{mA} \tag{2}$$

Here I_D = *id1* and V_{GS} = *vgs1*.

Note: The parabola will be a good fit if you only use one or two points for which I_D is less than 1 mA. A good fitting parabola will have its minimum very close to the V_{GS} axis where $I_D = 0$.

If it dips too far below, check your data or program for errors. Then remove the data point with the lowest value of current *idfit* and associated *vgsfit* you are using for the parabola fit and rerun the program.

If the parabola does not intersect the V_{GS} axis, check your data or program for errors. Then add one more low-current data point (or zero current point) to the *idfit* and *vgsfit* arrays and rerun the program. If the parabola still doesn't reach the axis, its minimum should still be very close to the axis.

Once you have the best-fit parabola for your data, extract the coefficients of p and calculate the derivative of Eq. 2 by hand, $d(I_D)/d(V_{GS})$. Setting the derivative equal to 0 allows you to find the voltage $V_{GS} \approx V_{tn}$ resulting in the minimum current $I_{Dmin} \approx 0$. Remember that I_D is in mA.

The Fairchild Semiconductor BS170 datasheet specifies V_{tn} to be "typically" 2.1 V with a range of 0.8 V to 3.0 V when I_D = 1 mA and $V_{DS} = V_{GS}$. Your calculated V_{tn} is for $I_{Dmin} \approx 0$ and $V_{DS} \neq V_{GS}$. Nevertheless, your calculated value should be close to the typical value.

Determine the percent error between your value and the typical specification using the specification as reference.

2. Plot the measured data from Tables 2 and 3 on the one graph of I_D vs. V_{DS}. Use linear regression analysis on each plot separately to find best-fit lines matching the data. Determine the values of $V_A = 1/\lambda$ where the lines intersect the $-V_{DS}$ axis. Perform the analyses using points on the flattest parts of the curves. Most likely this is for V_{DS} between 4 V to 9 V, inclusive, but may be different.

   The slope from each line is the saturation-region small-signal drain-source conductance g_o. Find the corresponding r_o values.

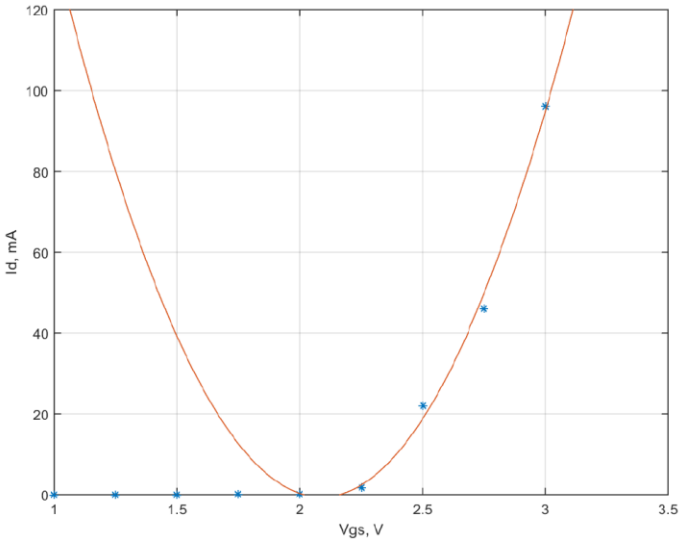


Fig. 4. Measured data and parabola fit to data.

You can use a similar program as the one used for question 1 with polyfit matching a first order polynomial (line) to the data subset. If you use that program, remember that I_D is in mA.

3. The derivative you calculated in question 1 is an expression of the transistor's transconductance g_m as a function of V_{GS}. Calculate g_m at V_{GS} = 2.6 V and V_{GS} = 2.8 V in mA/V (or mS).
4. Determine the logic functions of the circuits in Figs. 2 and 3 from their input/output tables. Assume output voltages above 2.5 V are logic 1 and those below 2.5 V are logic 0.
5. Calculate the on resistance r_{DS} of each BS170 from data in Table 4, rows 2 and 3 using the V_{DS} values you measured.
6. Why is V_{out} in Table 4, row 4 less than V_{out} of either row 2 or 3?

Appendix 1: Data Tables

Table 1. I_D vs. V_{GS}, V_{DS}=4V

Important: Do Not Let I_D Exceed 45 mA

Parameter	Nominal I_D (mA)										
	0.1	0.2	0.4	0.7	1	2	4	7	10	20	40
Measured I_D (mA)											
Measured V_{GS} (volts)											

Table 2. I_D vs. V_{DS}, V_{GS}= _____

Parameter	Nominal V_{DS} (V)								
	1.0	2.0	3.0	4.0	5.0	6.0	7.0	8.0	9.0
Measured V_{DS} (V)									
Measured I_D (mA)									

Table 3. I_D vs. V_{DS}, V_{GS}= _____

Parameter	Nominal V_{DS} (V)								
	1.0	2.0	3.0	4.0	5.0	6.0	7.0	8.0	9.0
Measured V_{DS} (V)									
Measured I_D (mA)									

Table 4. Logic circuit 1 Measurements.

V_1 (V)	V_2 (V)	V_{out} (V)
0	0	
0	5	
5	0	
5	5	

Table 5. Logic circuit 2 Measurements.

V_1 (V)	V_2 (V)	V_{out} (V)
0	0	
0	5	
5	0	
5	5	

Appendix 2: MATLAB Parabola Matching Code Example

```
% MOSFET Parabola Matching
%
% The parabola match will be best if you include only one or two points
% with current less than 1 mA.
% Current values are in mA.
% Replace the zeros with your data. You may have a different number of
% points than the number of zeros.
%
idmeas=[0 0 0 0 0]; %All measured id values in mA
vgsmeas=[0 0 0 0 0]; %All measured vgs values
idfit=[0 0 0 0];   %Measured id values used for curve fit
vgsfit=[0 0 0 0];  %Measured vgs values used for curve fit
p=polyfit(vgsfit,idfit,2);  %Find 2nd order polynomial coefficients
vgs1=[0:0.02:3.5];  %vgs range for parabola calculation, increment=0.02V
id1=polyval(p,vgs1);
plot(vgsmeas,idmeas,'*',vgs1,id1)  %Points only for meas, line for calc
axis([1 3.5 0 120])  %Typical axis settings, you can change
xlabel('Vgs, V')
ylabel('Id, mA')
grid
```

CMOS Inverter and Amplifier

EE 3401 Laboratory Exercise
Electrical Engineering Department
Kennesaw State University

Objectives:

Students simulate SPICE Level 1 NMOS and PMOS transistor models using LTspice and determine the effects of various parameters on their performance. A complementary MOS (CMOS) inverter is simulated and the effect of transistor transconductance is observed. A feedback amplifier based on the inverter is simulated and its closed loop gain is examined and compared to theoretical expectations.

Introduction:

The Level 1 MOS transistor models in SPICE are based on the Shichman-Hodges model equations discussed in class. You will change some of the parameters of the default PMOS and NMOS models and run simulations using the models. Then a simple CMOS inverter circuit is simulated. CMOS integrated circuits make use of complementary architecture. That is, NMOS and PMOS transistors are used in symmetric push-pull configurations. The inverter is a basic element of most CMOS digital integrated circuits. The voltage transfer characteristic (VTC) of a simple inverter made up of generic PMOS and NMOS transistors will be simulated. Finally, you will test your digital inverter in a linear amplifier configuration.

Equipment:

Computer running LTspice software

Procedure:

MOSFET Model Parameters and Common Source Characteristics

Measured MOSFET common-source (CS) characteristics are obtained from a circuit like the one in Fig. 1. Simulating the circuit will show the CS characteristics for a MOSFET model. To plot each curve, the gate-to-source voltage V_{GS} is set to a given value and the drain-to-source voltage V_{DS} is swept across the desired range.

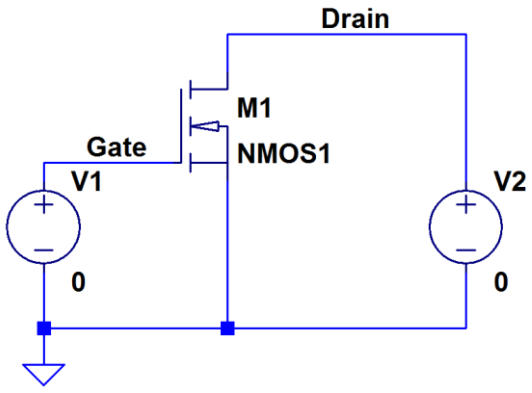


Fig. 1. NMOS transistor CS characteristics test circuit.

1. Prepare a NMOS transistor model.

   a. Calculate the parameters of a NMOS transistor model that is based on the default model but with following non-default parameters:

      - Specify a k' (KP) value calculated from the 3µm channel length process parameters given in Table 1 in Appendix 1 using Eq. 1 below.

$$k' = k'_n = \mu_n C_{ox} = \frac{\mu_n \varepsilon_{ox}}{t_{ox}} \quad \text{A/V}^2 \tag{1}$$

        Where $\varepsilon_{ox} = 3.45\times10^{-11}$ F/m, $\mu_n = 7\times10^{-2}$ m^2/(V·s) and $t_{ox} = 4\times10^{-8}$ m.

      - Specify a channel width of 180 µm, a channel length of 6 µm, a zero-bias threshold voltage of V_{t0} = 1Vand a channel length modulation factor of 0.01V^{-1}.

      - Give the model the name NMOS1.

   b. The model statement is shown below with question marks for the values you need to provide. Enter the model statement using the ".op" SPICE Directive toolbar button or Right Click>Draft>SPICE Directive.

      .model NMOS1 NMOS(KP=? VTO=? W=? L=? LAMBDA=?)

      Note: SPICE uses W and L dimensions in meters and the "O" in VTO is not "zero". Also, SPICE does not use case-sensitive model names and parameters.

2. Use the LTspice DC Sweep analysis type to plot the common-source characteristics of the NMOS1 transistor model.

   a. Create an LTspice schematic of the circuit in Fig. 1.

      - Select the "nmos" component for the transistor. When you place the transistor, the model will be specified as the default "NMOS". Change it to a custom model NMOS1 by right clicking on "NMOS" and changing it.

      - Be sure to label the drain and gate nodes so their data will be easy to find after the simulation is finished.

      - Set the DC voltage values "V" of each source to 0. It doesn't matter what values they are since the DC Sweep analysis will use the voltage ranges you specify for the simulation.

   b. Set up a nested DC sweep analysis that will sweep the drain-source voltage linearly from 0 to 16 V in 20 mV steps and sweeps the gate-source voltage linearly from 0 to 6 V in 500 mV steps.

      - In the DC Sweep analysis setup, click on the 1st Source tab. Enter the following parameters:

        Name of 1st Source to sweep: V2

        Type of sweep: Linear

        Start value: 0

        Stop value: 16

        Increment: 20e-3

- Click on the 2nd Source tab. Enter the following parameters:

  Name of 2nd Source to sweep: V1
  Type of sweep: Linear
  Start value: 0
  Stop value: 6
  Increment: 500e-3

c. Run the simulation.

d. Use the Waveform Viewer to display the drain current.

- The Waveform Viewer will display all the CS characteristic curves on one plot.
- Place a cursor at the quiescent point (Q-point) specified by V_{DS} = 5 V and V_{GS} = 3.5 V, and use the I_D value there to estimate the dc transconductance G_M where

$$G_M = \frac{I_D}{V_{GS}} \tag{2}$$

  Use the up/down arrows to step the cursor to a different curve. The cursor box will tell you the V_{DS} and I_D value but not V_{GS}. To be sure you are on the right V_{GS} curve, right click on the cursor itself and a box will open that tells you.

- Record the Q-point values for V_{DS}, V_{GS}, and I_D, as well as G_M in the NMOS1 row of Table 2 in Appendix 2.
- Save a copy of the NMOS1 transistor's CS characteristics.
- Save a copy of the schematic.

e. Use the two cursors to estimate the transistor drain-to-source output resistance, $r_o \approx \Delta V_{DS}/\Delta I_D$ at the quiescent point in part (d).

- Put both cursors on the V_{GS} = 3.5 V curve, one on either side of the Q-point. Use V_{DS} values of 4 V and 6 V for the cursor positions. The cursor box will tell you the slope $g_o = \Delta I_D/\Delta V_{DS}$.
- Record g_o and r_o in Table 2.

3. Repeat step 2 for a PMOS transistor model. You will have to redraw the circuit so V1's polarity is $+V_{SG}$ and V2's polarity is $+V_{SD}$. Make sure the transistor's source terminal is connected to ground. See Fig. 3.

a. Create the PMOS1 model.

- The PMOS transistor has $\mu_p = 3.5\times10^{-2}$ m^2/(V·s), half the value of μ_n. Recalculate k' for the PMOS transistor. All other model parameters are the same, except V_{t0} should be negative, that is V_{t0} = -1.
- Create a new model called PMOS1 with these parameters.

b. Run the simulation and plot $-I_D$ instead of I_D.

- Calculate the parameters G_M and r_o of the PMOS1 model at V_{DS} = –5 V and V_{GS} = –3.5 V, and NMOS1 models.

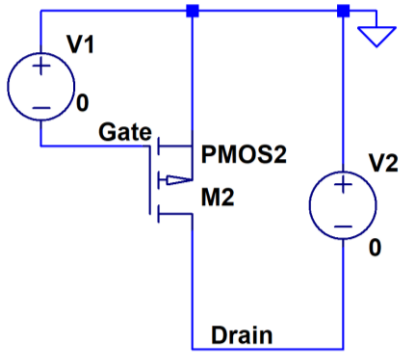


Fig. 3. PMOS transistor CS characteristics test circuit.

- Record your data in the PMOS1 row of Table 2.
- Save a copy of the PMOS1 transistor's CS characteristics.
- Save a copy of the schematic.

4. Modify the PMOS1 transistor model so that its transconductance matches that of the NMOS1 model.

   a. Adjust the PMOS transistor channel width so that its G_M value is the same as that of the NMOS1 transistor at the specified quiescent point. Call this model PMOS2.

   b. Then repeat step 3 to verify the model's performance.

   - Record your data in the PMOS2 row of Table 2.
   - Save a copy of the PMOS2 transistor's CS characteristics.

CMOS Inverter Model

5. Standard CMOS logic is designed for a logic level threshold at half the supply voltage. So, a CMOS inverter with a 5 V supply should have a voltage transfer curve that transitions from one logic state to the other when the input voltage is 2.5 V.

   a. Draw the schematic in Fig. 4 but use the PMOS1 model instead of the PMOS2 model shown. The load capacitor represents the output capacitance of the inverter.

   b. Perform a DC sweep (not a nested sweep) on the circuit. Sweep V1 from 0 to 5 V using 100 µV steps. Plot the output voltage V(out), which will trace the inverter's voltage transfer curve.

   Note: The simulation will take a while since the voltage step is so small.

   c. Measure the point where V(out) = V(in) on your transfer characteristic by plotting V(in) on the same graph as the transfer curve. This is the input voltage that causes the output logic state to change.

   - Record your data in the NMOS1-PMOS1 row of Table 3 in Appendix 2.
   - Save a copy of the voltage transfer characteristic plot including the plot of V(in). Mark the point where the two curves intersect and label the input/output voltages there.

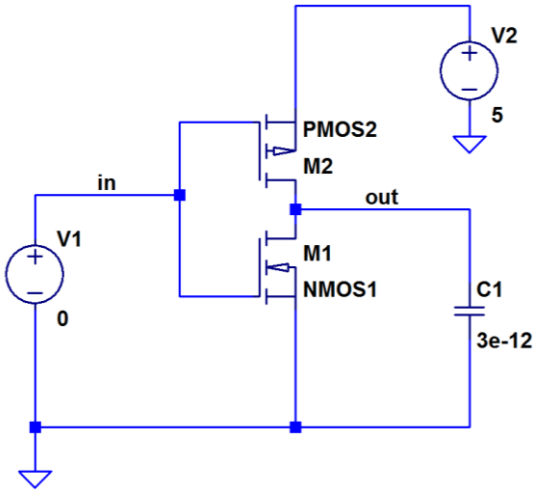


Fig. 4. CMOS inverter.

- Save a copy of the schematic.

6. Repeat step 5 with the PMOS2 transistor model. Put your data in the NMOS1-PMOS2 row of Table 3. You do not have to save a copy of the schematic.

CMOS Inverting Amplifier Model

7. The inverter can be converted into an inverting linear amplifier by adding a feedback resistor and an input resistor, in this case 10MΩ and 1MΩ respectively.

   a. Draw the schematic in Fig. 5 and use the PMOS2 model. Do not draw the triangular amplifier symbol; it is not a circuit element.

   b. Generate the voltage transfer characteristic for the amplifier in Fig. 5 by sweeping V1 from –5V to 5V using 1 mV steps.

   c. On a second plot, plot the derivative of the transfer curve. The latter plot shows the closed-loop gain of the amplifier as a function of input voltage.

   - Determine the maximum inverting gain from the derivative plot, which should occur when V(in) = 0V.
   - Determine the range of input voltages over which the gain is the same as maximum, within 1%.
   - Record your data in Table 4.
   - Save a copy of the transfer characteristic and gain plots. Mark the maximum gain on the derivative plot.
   - Save a copy of the schematic.

   c. Remove the 10MΩ resistor leaving an open circuit and repeat step (b). This simulation will show the open-loop gain of the inverter. Record your data in Table 4 and save copies of the plots.

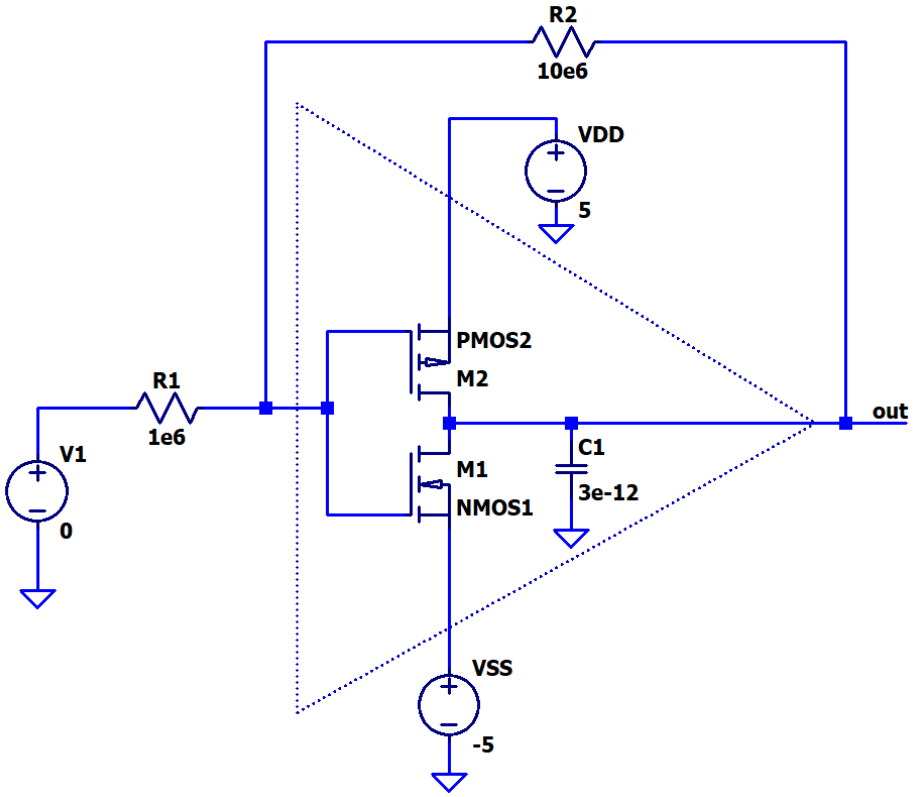


Fig. 5. Linear inverting amplifier using a digital inverter. Note that the triangle is symbolic and not part of the circuit.

Results Documentation Summary

8. You should have the following documentation from the procedure steps.
   a. You should have plots and schematics from steps 2(d), 3(b), 5(c) and 7(c). You should have only plots from steps 4(b) and 6, and 7(c).
   b. Be sure that you recorded data and calculated results in steps you were asked to.
   c. Check with your instructor for additional documentation requirements.

Analysis Questions:

1. Compare the transfer curves from the two inverter simulations. Which one has the better value of threshold voltage? Justify your answer.
2. When simulating the closed-loop inverting amplifier, was the input range for maximum gain larger or smaller than that of the open-loop amplifier? Why?
3. Consider the closed-loop inverting amplifier. Calculate the closed-loop gain if the inverter had infinite open-loop gain.
4. Using the open-loop amplifier's maximum gain determined in step 7, calculate the closed loop gain using the input and feedback resistors in Fig. 5. How does this value compare to the simulated gain of the inverting amplifier from step 7?

Appendix 1: Typical MOSFET Process Parameters

Table 1. Typical process parameters for 3μm minimum allowed channel length [1].

Parameter	Symbol	Value *n*-Channel Transistor	Value *p*-Channel Transistor	Units
Substrate doping	N_A, N_D	1×10^{15}	1×10^{16}	Atoms/cm^3
Gate oxide thickness	t_{ox}	400	400	Å
Metal-silicon work function	ϕ_{ms}	−0.6	−0.1	V
Channel mobility	μ_n, μ_p	700	350	cm^2/V-s
Minimum drawn channel length	L_{drwn}	3	3	μm
Source, drain junction depth	X_j	0.6	0.6	μm
Source, drain side diffusion	L_d	0.3	0.3	μm
Overlap capacitance per unit gate width	C_{ol}	0.35	0.35	fF/μm
Threshold adjust implant (box dist)				
impurity type		P	P	
effective depth	X_i	0.3	0.3	μm
effective surface concentration	N_{si}	2×10^{16}	0.9×10^{16}	Atoms/cm^3
Nominal threshold voltage	V_t	0.7	−0.7	V
Polysilicon gate doping concentration	N_{dpoly}	10^{20}	10^{20}	Atoms/cm^3
Poly gate sheet resistance	R_s	20	20	Ω/□
Source, drain-bulk junction capacitances (zero bias)	C_{j0}	0.08	0.20	fF/μm^2
Source, drain-bulk junction capacitance grading coefficient	n	0.5	0.5	
Source, drain periphery capacitance (zero bias)	C_{jsw0}	0.5	1.5	fF/μm
Source, drain periphery capacitance grading coefficient	n	0.5	0.5	
Source, drain junction built-in potential	ψ_0	0.65	0.65	V
Surface-state density	$\frac{Q_{SS}}{q}$	10^{11}	10^{11}	Atoms/cm^2
Channel-length modulation parameter	$\left\lvert\frac{dX_d}{dV_{DS}}\right\rvert$	0.2	0.1	μm/V

Reference:

[1] P. Gray, P. Hurst, S. Lewis, R. Meyer, *Analysis and Design of Analog Integrated Circuits*, 5ed, New York, John Wiley and Sons, 2009.

Appendix 2: Data Tables

Table 2. MOSFET Model Q-Points, G_M and r_o values.

Model Name	Q-Point V_{DS} (V)	Q-Point V_{GS} (V)	Q-Point I_D (mA)	DC G_M (mS)	Output Conductance	Output Resistance r_o
NMOS1	5.0	3.5				
PMOS1	–5.0	–3.5				
PMOS2	–5.0	–3.5				

Table 3. CMOS Inverter Input Logic Level Threshold Voltage

Inverter Circuit	Logic Level Threshold Voltage
NMOS1-PMOS1	
NMOS1-PMOS2	

Table 4. Inverting Amplifier Maximum Gains and Input Range

Configuration	Maximum Gain	Input Range for Max Gain
Closed-loop		
Open-loop		

BJT DC Characteristics

EE 3401 Laboratory Exercise
Electrical Engineering Department
Kennesaw State University

Objectives:

Students measure the common-emitter output characteristics of a NPN transistor and derive transistor parameters from the data. A diode-connected configuration is also examined. MATLAB is used to fit curves to measured data.

Introduction:

The 2N2222 is a popular NPN small-signal transistor. It is a general-purpose transistor with good bandwidth, current gain and switching characteristics. It comes in two popular packages, the metal TO-18 and the plastic TO-92. Figs. 1 and 2 show the packages and lead configurations. The two versions have slightly different specifications.

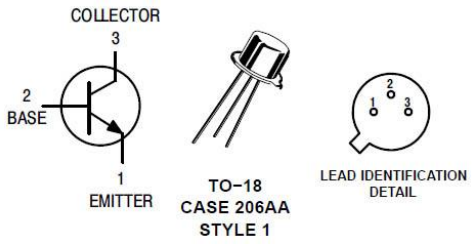


Fig. 1. TO-18 case version. **Note that the case is connected to the collector lead**. The tab is closest to the emitter.

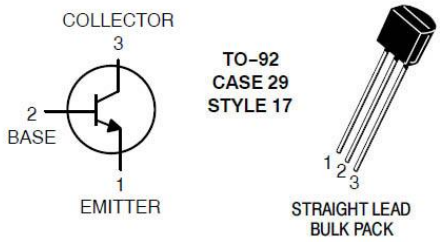


Fig. 2. TO-92 case version.

The first part of the procedure involves measuring the transistor's output characteristics in a common-emitter configuration for two different base currents. Linear regression analyses are performed on data for each base current using MATLAB. Transistor parameters β, α, g_m, r_o and V_A are obtained.

In the second part of the procedure, characteristics of a diode-connected configuration are measured. This configuration is used in current mirrors because the transistor is always in the active region and it has very low V_{CE} variation over a broad range of collector current. MATLAB is used to plot the output resistance of the configuration.

Equipment:

One or two digital multimeters (DMM)
Oscilloscope
Triple output power supply
Cables and oscilloscope probes as needed
Solderless breadboard

Components:

Resistors (Qty): 100 Ω (1), 10 kΩ (1), 100 kΩ (1)
NPN transistor (Qty): 2N2222 (1)

Procedure:

General Information

1. Consider the following when building and testing the circuits.
   a. The base voltage source and series 100 kΩ resistor used in the first part of the procedure approximates a current source. Monitor the current carefully and keep it close to constant.
   b. When using an oscilloscope to measure V_{CE}, select Hi Res or Average acquisition mode for best measurements.

      Note: it is best to use a x 10 probe with the oscilloscope.

Common-Emitter Output Characteristics of the 2N2222

2. Construct the circuit in Fig. 3 for measuring the common-emitter output characteristics as follows.
   a. If you are using the TO-18 version, the collector is connected to the case. Wrap some tape around the case to avoid short circuits if resistor leads accidentally touch it.
   b. Use the +6 V supply for V_{BB} and the +20 V supply for V_{CC}. Make sure both supplies are set to 0 V dc.
   c. Connect instruments to measure base current, collector current and collector-emitter voltage.
      - Components with "A" and "V" represent a DMM uses as an ammeter and a DMM used as a voltmeter.
      - If two ammeters are available, use them to measure I_B and I_C. Note that current can be calculated from multimeter voltage measurements made across resistors R_B and R_C. Measure R_B and R_C with the multimeter if you are using them to find current.

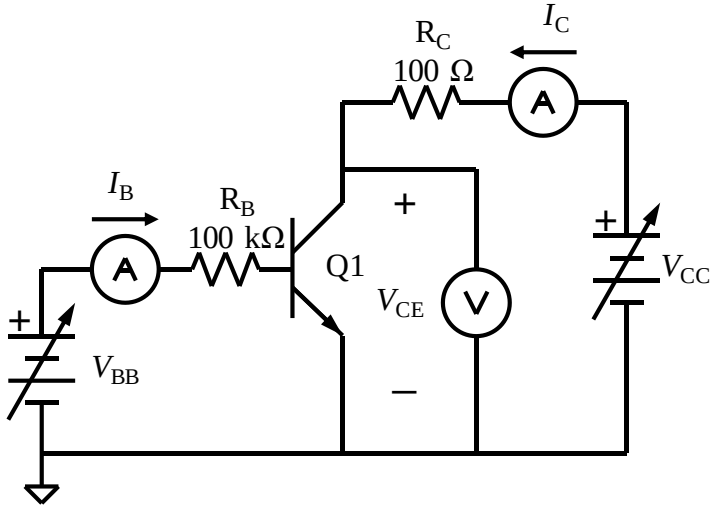


Fig. 3. Circuit for common-emitter output characteristic measurements.

R_B value: _____ R_C value: _____

- Use the oscilloscope to measure V_{CE}. Select Hi-Res or Average acquisition mode, Auto triggering, 10 msec/div, and have the oscilloscope measure the Average value.

3. Make measurements as follows, using Tables 1 and 2 to record your data. Your measurements will drift with temperature changes in the transistor, so record the data as soon as your voltage and current adjustments are complete.

   a. Adjust V_{CC} to set $V_{CE} = 2 \pm 0.1$ V.

   b. Adjust V_{BB} to set $I_B = 20\ \mu A \pm 1\ \mu A$. Record the actual value in the title of Appendix 1, Table 1. Check that $V_{CE} = 2$ V and readjust V_{CE} if necessary. Then verify that I_B is still within tolerance. Measure and record the value of I_C and V_{CE} in the table. Use 3 digits of precision.

      Note: If measuring base current using the voltage across R_B, calculate the expected voltage $V_{RB} = I_B R_B$ using the measured R_B value. Then adjust V_{BB} to set that voltage.

      Note: $V_{CE} \neq V_{CC}$ due to the voltage drop across R_C.

   c. Adjust V_{CC} to set $V_{CE} = 4 \pm 0.1$ V. Measure and record the value of V_{CE} and I_C.

   d. Repeat procedure step 3(c) for $V_{CE} = 6$ V, 8 V, 10 V, 12 V and 14 V.

   e. Adjust V_{CC} to return V_{CE} to 2 ± 0.1 Vdc. Then adjust V_{BB} to set $I_B = 40\ \mu A \pm 1\ \mu A$. Record the actual value of I_B in the title of Table 2. Use 3 digits of precision.

   f. Repeat procedure steps 3(c) and 3(d) for the new value of I_B.

Diode-Connected Transistor Circuit

4. Modify the circuit as shown in Fig. 4 to measure the characteristics of a diode-connected transistor as described below. Your measurements will drift with temperature changes in the transistor, so record the data as soon as your voltage and current adjustments are complete.

   a. Use the +20 V supply for V_{CC} as before. Make sure the supply is set to 0 V dc.

   b. Replace R_C with a 10 kΩ resistor.

   c. Connect instruments to measure I_E and V_{CE}.

      - You *must* use the multimeter to measure V_{CE} accurately.
      - I_E can be calculated from a multimeter voltage measurement made across R_C. If you are calculating I_E, measure R_C with the multimeter to find its actual resistance and use it in the calculation.

        R_C value: _____

        Note: The current through R_C is $I_E = I_C + I_B$ for this transistor configuration.

5. Make measurements as follows, using Table 3 to record your data.

   a. Adjust V_{CC} to set $I_E = 250\ \mu A \pm 2\%$. Record the actual value or I_E. Measure V_{CE} with the multimeter and record the value. Use 3 digits of precision.

   b. Adjust V_{CC} to set $I_E = 500\ \mu A \pm 2\%$. Record I_E and V_{CE}.

c. Repeat procedure step 5(b) for the remaining I_E values.

Results Documentation Summary

6. You should have the following documentation from the procedure steps.
   a. Be sure that you recorded data and calculated results in steps you were asked to.
   b. Check with your instructor for additional documentation requirements.

Analysis:

1. Use linear regression analysis on the measured data from Tables 1 and 2 to find best-fit lines matching the data. Select adjoining points on the flattest parts of the curves for the analysis.

   Plot the data from Tables 1 and 2 and the best-fit lines on *one* graph of I_C vs. V_{CE}. Plot only points for the measured data with no interpolated curve between the points. Save a plot that has horizontal V_{CE} axis range from 0 to 15 V.

   Find the two Early voltage V_A values by calculating the points where the lines intersect the $-V_{CE}$ axis from the equations determined by the linear regression algorithm. Remember that V_A is positive. Do not simply locate the intersection points graphically.

   The slope from each line is the active-region small-signal collector-emitter conductance g_o. Find the corresponding r_o values.

   You can use a similar MATLAB program as the one included in the MOSFET Behavior lab instructions with the polyfit function matching a first order polynomial (line) to the data subset. If you use that program, remember that I_C is in mA.

2. Plot the measured data from Table 3 on a graph of V_{CE} vs. I_E. Create a 4th-order polynomial curve fit to the data and plot that curve on the same graph. Restrict the I_E axis to range used for measurements. On a separate graph, plot $R_o = dV_{CE}/dI_E$. Figs. 5 and 6 show example plots. Appendix 2 has the MATLAB code used to obtain the example with measured data removed from the arrays.

3. Using measurements in Tables 1 and 2, calculate β, α and g_m at V_{CE} =6 V for the two base currents and put the results in Table 4.

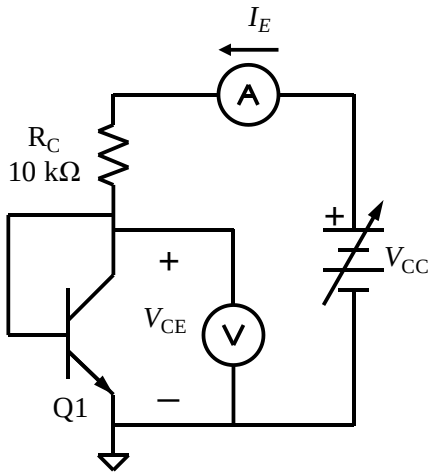


Fig. 4. Diode-connected transistor circuit.

4. Compare the load current values between the current mirrors with and without ballast resistors. How did the load current change with temperature for each circuit?

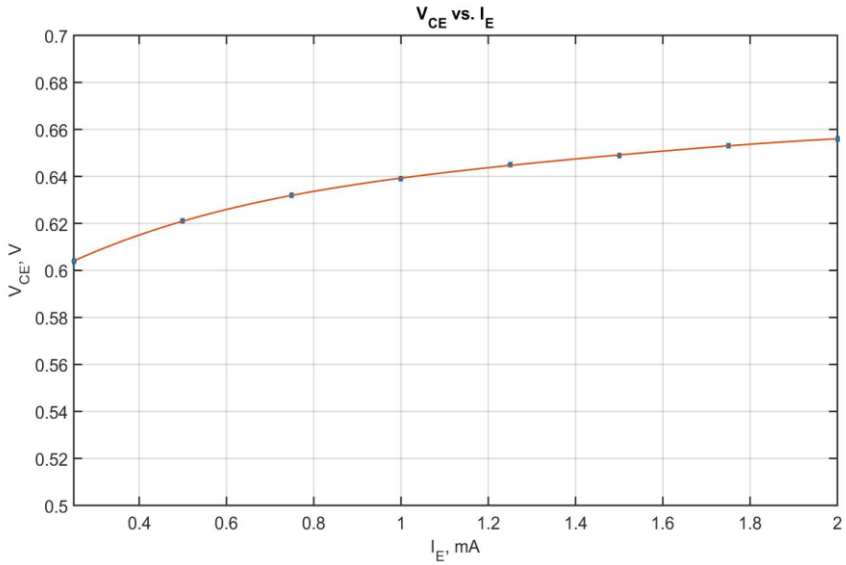


Fig. 5. Diode-connected transistor V_{CE} vs. I_E, measured data with 4th – order polynomial curve fit.

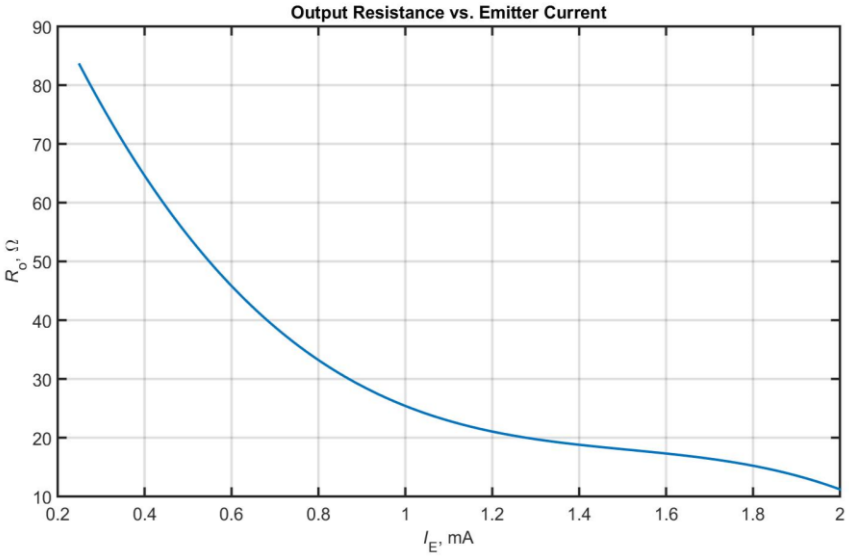


Fig. 6. Diode-connected transistor R_o vs. I_E.

Appendix 1: Data Tables

Table 1. I_C vs. V_{CE}, I_B = _____

Parameter	Nominal V_{CE} (V)						
	2.0	4.0	6.0	8.0	10.0	12.0	14.0
Measured V_{CE} (V)							
Measured I_C (mA)							

Table 2. I_C vs. V_{CE}, I_B = _____

Parameter	Nominal V_{CE} (V)						
	2.0	4.0	6.0	8.0	10.0	12.0	14.0
Measured V_{CE} (V)							
Measured I_C (mA)							

Table 3. Diode-Connected Transistor Output Characteristics V_{CE} vs. I_E.

Parameter	Nominal I_E (mA)							
	0.25	0.50	0.75	1.0	1.25	1.50	1.75	2.0
Measured I_E (mA)								
Measured V_{CE} (V)								

Table 4. Calculated Transistor Parameters at V_{CE} = 6 V.

Nominal I_B (μA)	β	α	g_m
20			
40			

Appendix 2: MATLAB Code for Diode-Connected Transistor Plots

```
% Diodeconn.m
% BJT diode connected transistor Vce vs. Ie data and curve fit.
%
% Current values are in mA.
%
% The figure(2) graph shows how to specify italic and subscript text in labels.
%
IeMeas=[0 0 0 0 0 0 0 0]; %Measured Ie
VceMeas=[ 0 0 0 0 0 0 0 0];  %Measured Vce
Ie=polyfit(IeMeas,VceMeas,4);  % 4th-order polynomial fit

IeRange=[0.25:0.002:2];    % Ie range in mA

VceFit=polyval(Ie,IeRange);  % Make smooth curve over measured Ie range

% dIe is derivative of Ie polynomial found manually from Ie coefficients
dIe=[ 0 0 0 0 ];  % Derivative of Ie polynomial, 4 terms

% Ro is output resistance curve from 4th order polynomial fit
Ro=polyval(dIe,IeRange)*1000;

figure(1)
plot(IeMeas,VceMeas,'*',IeRange,VceFit)
axis([0.25 2 0.5 0.7])
title('V_{CE} vs. I_{E}')
xlabel('I_{E}, mA')
ylabel('V_{CE}, V')
grid

figure(2)   % Use separate figure
plot(IeRange,Ro)
title('Output Resistance vs. Emitter Current')
xlabel('\itI\rm_{E}, mA')
ylabel('\itR\rm_{o}, \Omega')
grid
```

BJT Amplifiers

EE 3401 Laboratory Exercise
Electrical Engineering Department
Kennesaw State University

Objectives:

You will analyze, build and test NPN transistor amplifiers in order see how they perform. Three common-emitter (CE) amplifiers and one common collector (CC) amplifier are examined. The effect of an emitter resistor on the quality of the bias circuit and the amplifier gain are observed.

Introduction:

The CE amplifier in Fig. 1 has no emitter degeneration resistor. The resulting dc bias circuit quiescent point (Q-point) that is sensitive to β variation. However, it has high ac voltage gain. The function generator's signal is connected to the base through a dc blocking capacitor. The second CE amplifier in Fig. 2 has an emitter resistor, which makes the Q-point much less sensitive to β variation. But, the gain is much lower than that of the first circuit. The third circuit uses a capacitor to "bypass" the emitter resistor for ac signals, increasing the gain. The capacitor does not affect the dc bias circuit, so the Q-point is the same as for the second circuit.

The common-collector amplifier in Fig. 4 is a modification of the Fig. 2 circuit with the collector resistor removed.

Equipment:

Function/Arbitrary Waveform Generator
Oscilloscope
Digital Multimeter (DMM)
DC power supply with +20 V output
Cables and oscilloscope probes as needed
Solderless breadboard
Components:
- Resistors (Qty): 100 Ω (3), 1 kΩ (1), 56 kΩ (1), 180 kΩ (1)
- Capacitors (Qty): 4.7 μF (2)
- NPN transistor (Qty): 2N2222 (1)

Prelab:

1. For the three amplifier circuits in Figs. 1, 2, 3, and 4:
   a. Calculate the Q-point values of collector current (I_C) and collector-emitter voltage (V_{CE}). Assume β_{DC}=150, V_{BE} = 0.65 V, $V_{CE(SAT)}$ =0.2 V, and $V_A = \infty$. Put your results in the prelab columns of Table 1.
   b. Calculate the small-signal voltage gain V_{out}/V_{in}. Assume β_{AC}=150 and the capacitors are short circuits. Put your results in the prelab columns of Table 2. Record the gain magnitude in column 2 and the phase shift in column 3. A non-inverting gain has a phase shift of 0° and an inverting gain has a phase shift 180°.

Procedure:

General Information

1. Consider the following when building and testing the circuit.
   a. Electrolytic capacitors are used in the circuits. These are polarized and must be installed in the correct direction as indicated by the "+" sign on the schematics.
   b. Measure sine wave input and output waveforms in peak-peak values. The oscilloscope can make the measurements automatically.

      Note: Use Hi Res or Average acquisition mode for best oscilloscope measurements.

      Note: Always turn off the power supply before making circuit changes.

Common-Emitter Amplifier Circuits

2. Consider the circuit in Fig. 1.
   a. Measure the value of R_C.

      R_C value: _____
   b. Construct the circuit. Minimize the number of wires.
   c. Use the DMM to measure V_{CE} and I_C. For I_C, measure the voltage across R_C and divide by its value. Put the results in Table 1.
   d. Connect the oscilloscope to measure V_{in} on channel 1 and V_{out} on channel 2. Use a ×10 probe to measure V_{out}. Set channel 2 for ac coupling so the output signal's dc component does not appear on the oscilloscope.

      Note: Some probes can be switched between ×1 and ×10. Be sure it is set to ×10. If the probe scale factor is not automatically sensed by the oscilloscope, use the channel 2 menu to set the probe scale factor to ×10.

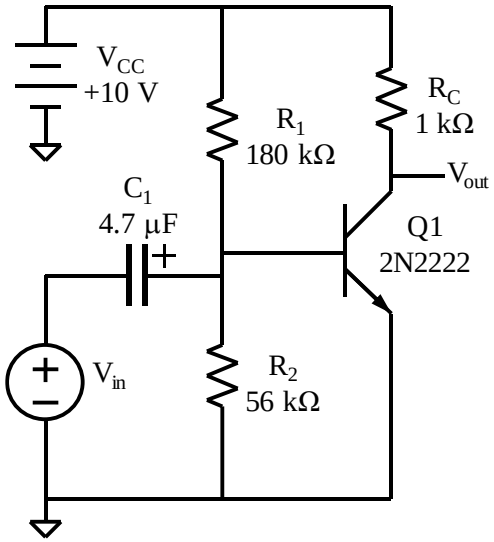


Fig. 1. Common-emitter amplifier without emitter resistor.

e. Set the function generator to create the waveform $V_{in} = 0.01\sin[2\pi(5000)t]$ V. Connect it to the circuit and verify the peak-peak amplitude on the oscilloscope.

Note: The oscilloscope may have trouble triggering on the channel 1 waveform. Connect a BNC-BNC cable form the generator's sync output to channel 3 and trigger on that square wave instead. Once you set up the triggering, turn off channel 3 so the square wave does not appear on the display.

f. Verify that the V_{out} waveform is a sine wave with very little distortion.

Note: If the waveform flattens at the top or bottom, you need to reduce the amplitude of V_{in}. Do this by adding two 100 Ω resistors in parallel from the function generator's connection point at the left side of C_1 to ground. This creates a voltage divider with scale factor 0.5 between the generator's internal 50 Ω resistance and the 50 Ω resistance you created.

g. Measure the peak-peak amplitudes of V_{in} and V_{out} and the phase shift of V_{out} relative to V_{in}. Record these in Table 2. Calculate the measured gain magnitude $|V_{out}/V_{in}|$ and place the result in the table. Save an image of the oscilloscope screen showing the measured amplitudes and phase shift.

Note: Be sure to use Hi Res or Average acquisition mode so that the waveforms are clean with minimum noise.

h. If you added 100 Ω resistors to reduce the input signal amplitude, remove them. Change the input signal to $V_{in} = 0.02\sin[2\pi(5000)t]$ V. The waveform should now flatten at the top or bottom or both. Save an image of the oscilloscope screen.

3. Consider the circuit in Fig. 2.

a. Construct the circuit.

b. Disable the function generator output or disconnect it from the circuit. Use the DMM to measure V_{CE} and I_C. Put the results in Table 1.

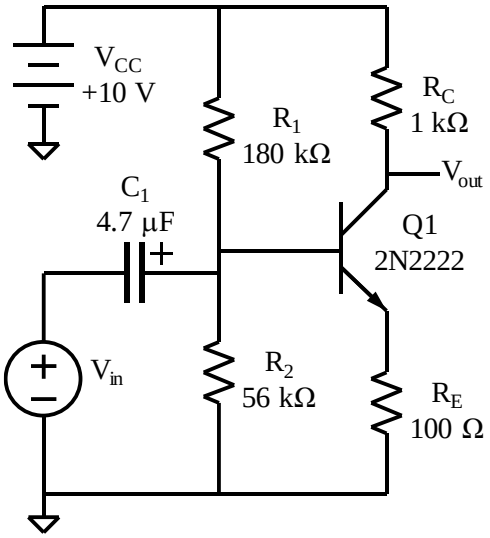


Fig. 2. Common-emitter amplifier with emitter resistor.

c. Set the function generator to create the waveform $V_{in} = 0.02\sin[2\pi(5000)t]$ V. Connect it to the circuit and verify the peak-peak amplitude on the oscilloscope.

d. Measure the peak-peak amplitudes of V_{in} and V_{out} and the phase shift of V_{out} relative to V_{in}. Record these in Table 2. Calculate the measured gain magnitude $|V_{out}/V_{in}|$ and place the result in the table. Save an image of the oscilloscope screen showing the measured amplitudes and phase shift.

4. Consider the circuit in Fig. 3.

a. Construct the circuit.

b. Disable the function generator output or disconnect it from the circuit. Use the DMM to measure V_{CE} and I_C. Put the results in Table 1.

c. Set the function generator to create the waveform $V_{in} = 0.02\sin[2\pi(5000)t]$ V. Connect it to the circuit and verify the peak-peak amplitude on the oscilloscope.

d. Measure the peak-peak amplitudes of V_{in} and V_{out} and the phase shift of V_{out} relative to V_{in}. Record these in Table 2. Calculate the measured gain magnitude $|V_{out}/V_{in}|$ and place the result in the table. Save an image of the oscilloscope screen showing the measured amplitudes and phase shift.

Common-Collector Amplifier Circuit

5. Consider the circuit in Fig. 4.

a. Remove R_E from the circuit and measure its value. Do not measure the resistor while it is in the circuit.

R_E value: _____

b. Construct the circuit.

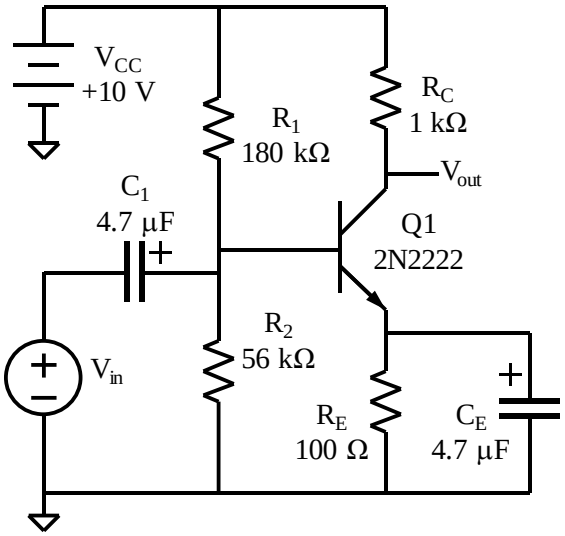


Fig. 3. Common-emitter amplifier with bypassed emitter resistor.

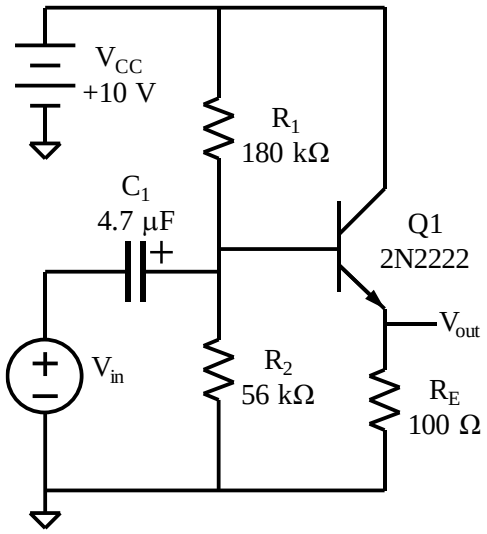


Fig. 4. Common-collector amplifier.

c. Disable the function generator output or disconnect it from the circuit. Use the DMM to measure V_{CE} and I_E. For I_E, measure the voltage across R_E and divide by its value. Assume $I_C \approx I_E$ since β is large. Put the V_{CE} and I_C results in Table 1.

d. Set the function generator to create the waveform $V_{in} = 2\sin[2\pi(5000)t]$ V. Connect it to the circuit and verify the peak-peak amplitude on the oscilloscope.

e. Measure the peak-peak amplitudes of V_{in} and V_{out} and the phase shift of V_{out} relative to V_{in}. Record these in Table 2. Calculate the measured gain magnitude $|V_{out}/V_{in}|$ and place the result in the table. Save an image of the oscilloscope screen showing the measured amplitudes and phase shift.

Results Documentation Summary

6. You should have the following documentation from the procedure steps.

   a. You should have five oscilloscope screen shots, two from the circuit 1 and one each from circuits 2, 3, and 4.

   b. Be sure that you recorded data and calculated results in steps you were asked to.

   c. Check with your instructor for additional documentation requirements.

Analysis:

1. Examine your oscilloscope image from step 2(h) and refer to the BJT common-emitter characteristics in Fig. 5. If the waveform flattens at the top, is the transistor in saturation or cutoff? If the waveform flattens at the bottom, is the transistor in saturation or cutoff?

2. Calculate the percent error between measured and calculated I_C and V_{CE} values in Table 1. Put the results in the appropriate columns in the table. Explain the cause(s) of the differences.

3. Calculate the percent error between measured and calculated gain magnitude values in Table 2. Put the results in the appropriate column in the table. Explain the cause(s) of the differences.

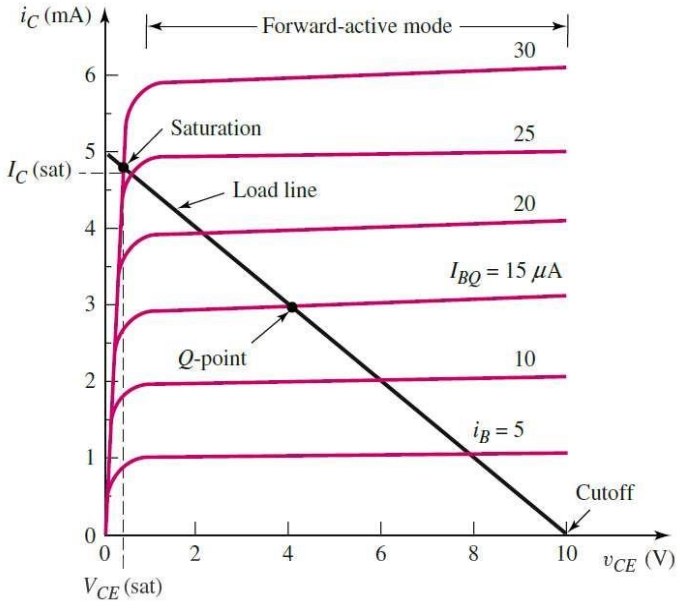


Fig. 5. Typical BJT common-emitter characteristics with a CE amplifier load line.

4. The measured phase shift between V_{out} and V_{in} for the CE amplifier with bypassed R_E (Fig. 3) is not the expected 180°. This is because the capacitor's impedance at 5 kHz *does not* approximate a short circuit well enough. By accounting for the capacitor's actual impedance in the theoretical gain calculation, the result will show a gain magnitude and phase shift closer to the measured values.

   - Recalculate the theoretical gain $H(f)=V_{out}(f)/V_{in}(f)$ at f = 5 kHz after replacing R_E with Z_E. where

$$Z_E = R_E \parallel Z_{C_E} = R_E \parallel \left(\frac{1}{j\omega C_E} \right) \tag{1}$$

     Express your answer in polar (magnitude and phase) form with

$$H(f) = |H(f)| \angle H(f) \tag{2}$$

     Use the gain magnitude and gain phase equations derived in Appendix 2. Enter these calculated results in Table 3.

   - Enter the measured gain magnitude and phase shift from Table 2 in Table 3. Find the percent error between the measured and newly calculated values of gain magnitude. Do the same for the phase shift. Enter the results in Table 3.

Appendix 1: Data Tables

Table 1. Calculated and measured amplifier Q-point data.

Amplifier	Calculated I_C (mA) (Prelab)	Calculated V_{CE} (V) (Prelab)	Measured I_C (mA)	Measured V_{CE} (V)	I_C Percent Error	V_{CE} Percent Error
CE, no R_E, Fig. 1						
CE, with R_E, Fig. 2						
CE, bypassed R_E, Fig. 3						
CC, Fig. 4						

Table 2. Calculated and measured amplifier gain data.

Amplifier	Calculated Gain Magnitude $\lvert V_{out}/V_{in}\rvert$ (Prelab)	Calculated Gain Phase Shift (Prelab)	Measured V_{in} (V)	Measured V_{out} (V)	Measured Gain Magnitude $\lvert V_{out}/V_{in}\rvert$	Measured Gain Phase Shift	Gain Magnitude Percent Error
CE, no R_E, Fig. 1							
CE, with R_E, Fig. 2							
CE, bypassed R_E, Fig. 3							
CC, Fig. 4							

Table 3. Calculated and measured gain data for CE amplifier with bypassed emitter resistor.

Amplifier	Calculated Gain Magnitude $\lvert V_{out}/V_{in} \rvert$ (Analysis Question 4)	Calculated Gain Phase Shift (Analysis Question 4)	Measured Gain Magnitude $\lvert V_{out}/V_{in} \rvert$	Measured Gain Phase Shift	Gain Magnitude Percent Error	Gain Phase Percent Error
CE, bypassed R_E, Fig. 3						

Appendix 2: Effect of Emitter Bypass Capacitor's Impedance on CE Amplifier Gain

The CE amplifier in Fig. 2 has a voltage gain transfer function

$$H(f)=\frac{V_{out}(f)}{V_{in}(f)}=\frac{-g_m R_C}{1+g_m R_E} \tag{A-1}$$

Where it is assumed that C_1 acts like a short circuit, which is a good assumption for this circuit.

In the Fig. 3 circuit, a capacitor is added in parallel with R_E. If the capacitor is assumed to be a short circuit, then the resistor R_E is shorted out and Eq. 1 becomes

$$H(f)=\frac{V_{out}(f)}{V_{in}(f)}=\frac{-g_m R_C}{1+g_m(0)}=-g_m R_C \tag{A-2}$$

Where again it is assumed that C_1 acts like a short circuit, which is a good assumption for this circuit.

However, the 4.7 μF capacitor C_E *is not* a good approximation to a short circuit at 5 kHz. To account for this, Eq. 1 is modified by replacing R_E with Z_E.

$$H(f)=\frac{V_{out}(f)}{V_{in}(f)}=\frac{-g_m R_C}{1+g_m Z_E} \tag{A-3}$$

Where

$$Z_E=R_E \parallel Z_{C_E}=R_E \parallel \left(\frac{1}{j\omega C_E}\right)=\frac{R_E\left(1/j\omega C_E\right)}{R_E+\left(1/j\omega C_E\right)} \tag{A-4}$$

Multiplying numerator and denominator of Eq. A-4 by $j\omega C_E$ gives

$$Z_E=\frac{R_E\left(1/j\omega C_E\right)}{R_E+\left(1/j\omega C_E\right)}\left(\frac{j\omega C_E}{j\omega C_E}\right)=\frac{R_E}{1+j\omega R_E C_E} \tag{A-5}$$

Substituting Eq. A-5 into Eq. A-3 yields

$$H(\omega)=\frac{V_{out}(\omega)}{V_{in}(\omega)}=\frac{-g_m R_C}{1+g_m\left(\frac{R_E}{1+j\omega R_E C_E}\right)} \tag{A-6}$$

Now multiply the numerator and denominator by $1+j\omega R_E C_E$.

$$\begin{aligned} H(\omega)=\frac{V_{out}(\omega)}{V_{in}(\omega)}&=\frac{-g_m R_C}{1+g_m\left(\frac{R_E}{1+j\omega R_E C_E}\right)}\left(\frac{1+j\omega R_E C_E}{1+j\omega R_E C_E}\right)=\frac{-g_m R_C\left(1+j\omega R_E C_E\right)}{\left(1+j\omega R_E C_E\right)+g_m R_E} \\ &=\frac{-g_m R_C\left(1+j\omega R_E C_E\right)}{\left(1+g_m R_E\right)+j\omega R_E C_E}=-g_m R_C\left[\frac{\left(1+j\omega R_E C_E\right)}{\left(1+g_m R_E\right)+j\omega R_E C_E}\right] \end{aligned} \tag{A-7}$$

Where the numerator and denominator are separated into real and imaginary terms. The $g_m R_C$ term in the numerator is purely real and is factored out.

The magnitude of $H(f)$ is found From Eq. A-7 by dividing the magnitude of the numerator by the magnitude of the denominator and letting $\omega = 2\pi f$.

$$\left|H(\omega)\right| = g_m R_C \frac{\sqrt{1^2 + \left(\omega R_E C_E\right)^2}}{\sqrt{\left(1 + g_m R_E\right)^2 + \left(\omega R_E C_E\right)^2}} \tag{A-8}$$

$$\left|H(f)\right| = g_m R_C \frac{\sqrt{1 + \left(2\pi f R_E C_E\right)^2}}{\sqrt{\left(1 + g_m R_E\right)^2 + \left(2\pi f R_E C_E\right)^2}} \tag{A-9}$$

The phase of $H(f)$ in is found from Eq. A-7 by subtracting the denominator phase from the numerator phase. The "–" sign due to the inverting gain is a phase shift of ±π, or ±180°. Choose +180° or –180° for the inverting gain term's phase so that the overall phase $\angle H(f)$ is within the range –180° to +180°. Note that the numerator in Eq. A-7 is the product of two terms, so the phases of the two terms add.

$$\begin{aligned}\angle H(\omega) &= \left[\text{numerator phase}\right] - \left[\text{denominator phase}\right] \\ &= \left[\pm 180° + \tan^{-1}\left(\frac{\omega R_E C_E}{1}\right)\right] - \left[\tan^{-1}\left(\frac{\omega R_E C_E}{1 + g_m R_E}\right)\right] \\ &= \left[\pm 180° + \tan^{-1}\left(\omega R_E C_E\right)\right] - \left[\tan^{-1}\left(\frac{\omega R_E C_E}{1 + g_m R_E}\right)\right]\end{aligned} \tag{A-10}$$

$$\angle H(f) = \left[\pm 180° + \tan^{-1}\left(2\pi f R_E C_E\right)\right] - \left[\tan^{-1}\left(\frac{2\pi f R_E C_E}{1 + g_m R_E}\right)\right] \tag{A-11}$$

If the sign of $\angle H(f)$ is positive, the output waveform leads the input and if it is negative, the output waveform lags the input.