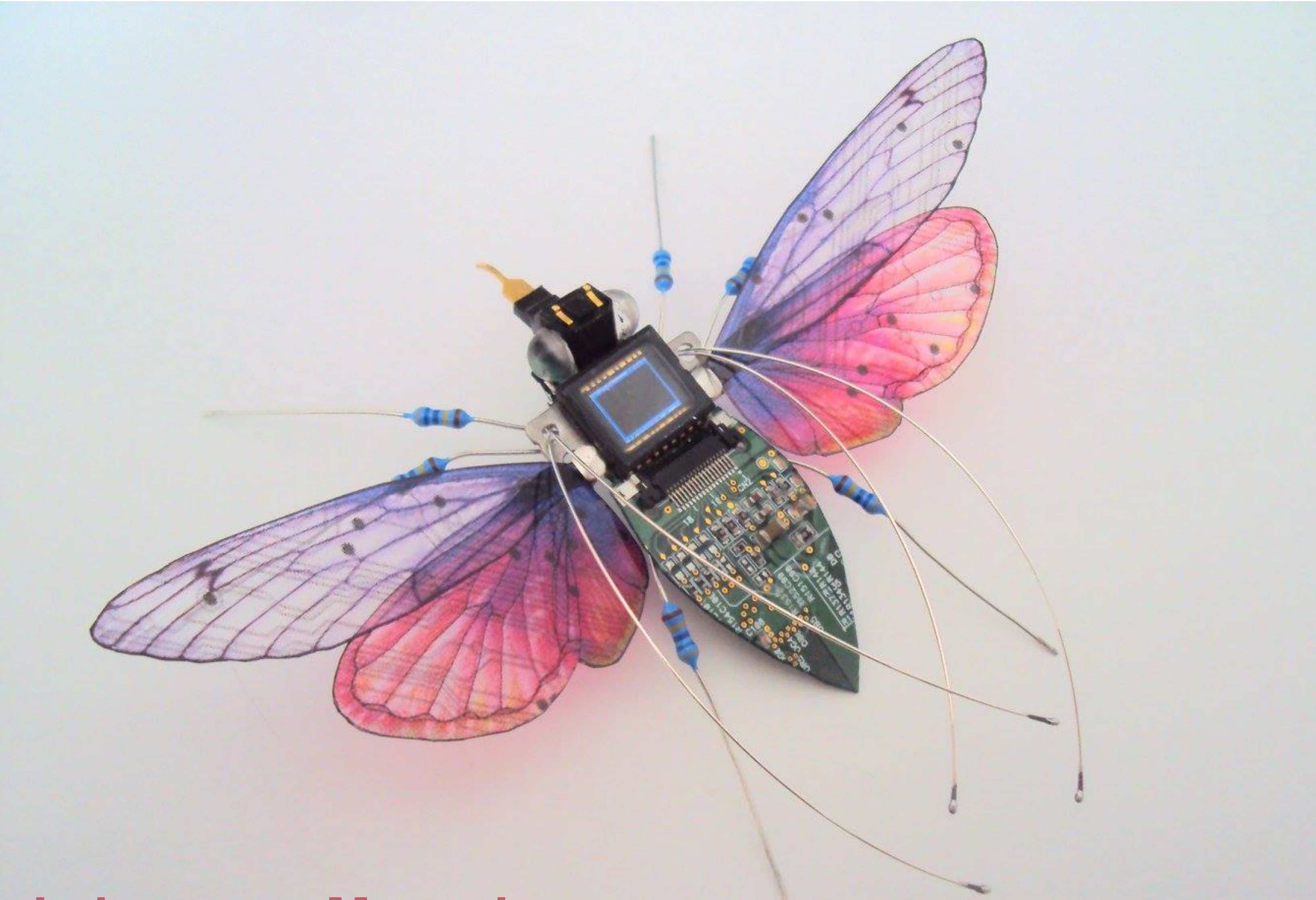


Laboratory Manual
ECE 3410: Microelectronics I

Spring 2016

Chris Winstead
Associate Professor
Electrical and Computer Engineering
chris.winstead@usu.edu

UtahStateUniversity

“Any sufficiently advanced technology is indistinguishable from magic.”
– Arthur C. Clarke

“It’s still magic even if you know how it’s done.”
– Terry Pratchett

Cover art by Julie Alice Chappell
DewLeaf Shop on Etsy: `https://www.etsy.com/shop/DewLeaf`
Artist Facebook Page: `https://www.facebook.com/Julie-Alice-Chappell-Artist-146001165481339/`

Contents

Introduction

Lab Documentation

Each student must have his/her own lab book. Lab experiments should be conducted individually. In this course, we will use rigorous lab documentation procedures that are followed in the most competitive industries. These procedures are a relaxed version of procedures used in a bio-electronics manufacturing facility.

Rigorous lab procedures exist for several reasons:

1. To enable exact replication of laboratory procedures at a later time.

2. To help you understand your own work after some time delay, when you may have forgotten significant details.

3. To communicate your work to another person, such as a supervisor, assistant, team member, or your replacement (assume you will be favorably transferred or promoted, and you must hand off your work to someone else).

4. To assist in tracing the root causes of major problems that may appear in subsequent experiments, or in products.

5. To provide a legally admissible and precise record of R&D activity that can be used in patent litigation.

6. To satisfy demands of regulatory agencies such as the FCC, FDA or ISO.

Following these procedures can be time consuming, but it will give you important practice in the disciplines expected by many large industrial firms. In your career you may find yourself working in an environment with more relaxed expectations. In that case you are free to downgrade your laboratory discipline, but it is always better to be overprepared than unprepared for your professional career.

Laboratory Notebooks

- Must be *bound* (not spiral).
- Should have three blank pages in the front for a table of contents.
- All pages should be numbered.
- Use only the front of each page, so the ink doesn't bleed through.
- The back of each page may be used for pencil scratch work.
- Sign and date each page after completing your work.
- Draw a line through any blank sections. For instance, if you do not completely fill a page, draw a line through the empty portion.
- If a page is left blank (e.g. as a spacer between sections), write "This page intentionally blank" and sign and date the page.
- Avoid writing any jokes, doodles, unprofessional remarks or other extraneous materials.

For each lab experiment:

1. Begin with an abstract describing the experimental objectives and procedures. All writing should be in third-person, professional style.

2. All equipment used for the experiment should be recorded in the lab book. Record the manufacturer and model number.

3. Record a list of components and other parts for each experiment.

4. Draw/reproduce complete schematic diagrams for all circuits used in the experiment.

5. Record careful tables of numerical results. Always record units and precision for experimental measurements.

6. Graphs should be carefully drawn in the lab book. They may be printed (e.g., from Matlab) and then taped into the lab book, but the student must initial across the edge of any content which is taped into the lab book.

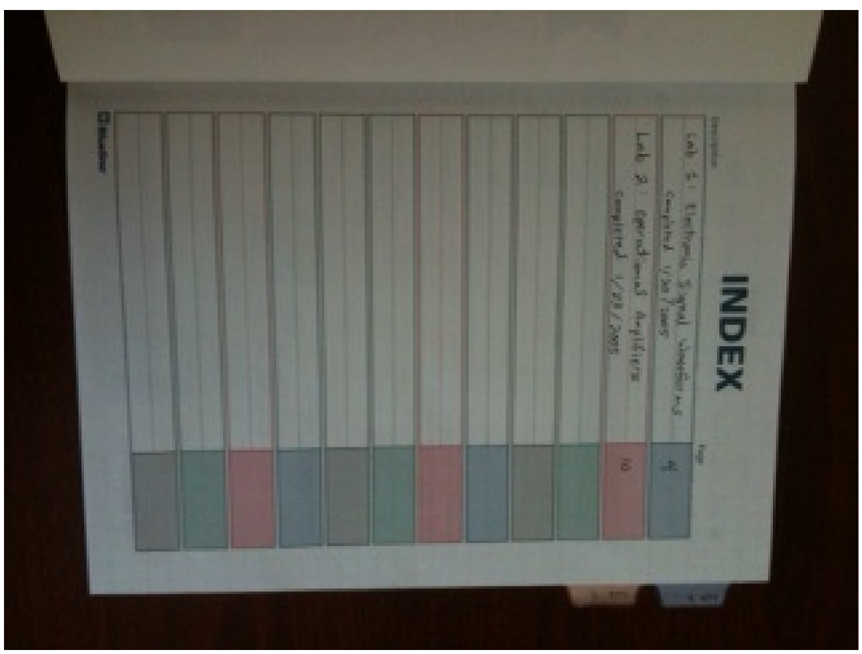
7. Conclusions: Explain your results with reference to the stated objectives of the lab. Were there any noteworthy or unusual observations that arose from the experiments? Did your results match theoretical expectations? If not, give reasonable hypotheses as to what might have gone wrong. Support your hypotheses with plausible arguments.

8. Have the TA/instructor sign and date the last page of each lab to certify your results.

Suggested Lab Book Product

The USU Bookstore carries a collection of Lab Notebooks manufactured by *Blueline*. These notebooks feature colored label stickers, section tabs, a form for contact information, and a table of contents. The Blueline notebook is recommended but not required.

Here are some images of the Blueline notebook:



Computing Resources

The Electronics Wiki

General information about USU's facilities, resources and courses for electronics can be found at the USU Electronics Wiki page:

http://electronics.wiki.usu.edu

The Electronics Wiki is a community document, and students are permitted to edit pages. If you figure something out or have some knowledge to add, please do so. If you find any erroneous information on the site, please correct it. Each Wiki page has a login option and help links to assist you in creating and editing pages.

Linux and the DAL

Today, most high-end electronic design is produced using Linux/Unix design software. Linux is therefore emphasized in the electronics and VLSI courses at USU. You may use the Linux computers in the Design Automation Lab, where you may login to a machine using your A-number and USU password.

If you do not already have access permissions for the DAL, please send a message to your instructor providing your A-number and indicating that you need access permission.

You are encouraged to attend a Linux tutorial session, and to study the Linux tutorial located on the Electronics Wiki:

http://electronics.wiki.usu.edu/Linux_Tutorial

SPICE Simulators

You will be assigned to perform a number of simulations using SPICE (*S*imulation *P*rogram with *I*ntegrated *C*ircuit *E*mphasis). There are several implementations of SPICE (called "flavors"), and you may use any version. It is recommended that you use NGSpice, which is available in the ECE Design Automation Lab (EL 105). NGSpice is free software, and can be installed on your personal computer if you wish. To get started with NGSpice, refer to the tutorial located on the ECE Electronics Wiki:

http://electronics.wiki.usu.edu/NGSpice

Preparing Lab Reports

Organization of the Report

Introduction

A lab report should begin with a short introductory section (one or two paragraphs) that describes the general activities and briefly summarizes findings. In this summary, you should identify the most important or interesting aspects of the lab. Try to use objective professional language. For example, do not say "I hooked up X and was supposed to get R but when I did Y it didn't work so I tried Z and then it worked." Instead, say something like "X was studied using a Z configuration, with result R."

Pre-Lab Analysis

Provide a summary of your pre-lab calculations, focusing specifically on the calculations that affect your experimental design. Note that some calculations are purely instructive; only include them in the report if they directly affect your experiment. If you performed experiment X and plan to write about it in the report, then you should fully describe any pre-lab analysis related to X.

Simulation Results

If the lab includes a session with the SPICE simulator, place the results here. Again, you do not need to be exhaustive. Focus on what is important. If you are reporting on specific experiment X, then provide the simulation results related to X and explain how they match with the pre-lab analysis related to X. All of your sections should tie-in with each other.

Experimental Results

Here's the part where you place your actual measurements. If you choose to discuss experiment X, then provide quantitative data resulting from the experiment, and compare those results to the predictions made in pre-lab analysis and simulation.

Conclusions

In this section, provide a summary and interpretation of your results. If your experimental results did not match your predictions, then offer plausible hypotheses to explain the discrepancy. If your results point toward a grand conclusion or generalization, state it here.

Appendix: Data Tables

If you need to include data tables in the report, place them in an appendix at the end. The main body of your report should not be over-burdened with minutia. The report should communicate knowledge and insights. The minutia can go at the end for the reader's reference.

Use Scientific Writing Style

In the introduction, stick to the lab's scientific themes rather than the educational goals. The educational outcomes are *my* objectives. The scientific activities are *your* objectives. For instance, it would be good to say, "Three circuits were studied with respect to their DC and AC characteristics: a voltage divider, a low-pass RC circuit and a high-pass RC circuit. Transfer functions were obtained through theoretical analysis and SPICE simulation, and were experimentally tested." Similarly, in your Conclusions you should stick to the professional facts.

Be careful making subjective judgement calls like "the measurements were close enough." Instead, propose an explanation for observed discrepancies. Your measured cutoff frequency was lower than expected. Could this be explained by the measured resistor value? Or is it because the measurement procedure is not precise enough to obtain the exact value?

Preparing the Report Using LyX/LaTeX

You are strongly encouraged to use the LyX/LaTeX document processor for preparing your reports. This is a soft requirement; you can prepare reports in MS Word but the results are usually of lower quality. Here are some tips for good writing in LyX:

When using LyX, I recommend that you start by directly editing my template document. In my template, I've already configured document settings to give you nice margins, to automatically scale included figures, and to correctly process my provided figures. If you start from scratch with a new LyX file, it will not benefit from all my settings. Please carefully study the template, and then take note of the additional tips below.

In equations, you should enter standard functions like sin by selecting them from the function sub-menu at the bottom of the LyX window in math mode. This will typeset them properly in Roman font. By typing *sin* directly, it is interpreted as a product of three variables named 's', 'i' and 'n', so the formatting looks a little off. Simiarly, you should usually set subscripts like "IN" in v_{IN} as Roman font. This can be done by highlighting the letters and selecting from the font submenu at the bottom of the LyX window when in Math mode. You will also want to place standard units like V and hertz in Roman font.

When inserting a cross reference, LyX will only print the number. So if you make a reference to Figure 2, you should type "Figure X" and then insert the cross-reference in place of 'X' (and be sure to put a space before the cross-reference). When referencing equations, it is customary to enclose the cross-reference in parentheses, as in (X). You may also use the long form "Equation X" but again be sure to put a space before the cross-reference.

Parts Inventory

Electronic Components and Parts

To complete the exercises in this manual, you will need (at minimum) the following components:

Value	Description	Quantity
100 Ω	resistor	1
300 Ω		1
1 kΩ		2
2 kΩ		1
10 kΩ		5
100 kΩ		1
1 nF	capacitor	1
10 nF		1
1 μF		4
20 μF		1
uA741	op amp	2
1N914	diodes	4
CD4007	MOS array	1
10k	potentiometer	1

Additional parts:

- Solderless breadboard or superstrip
- Breadboard hookup wire kit
- Potentiometer adjust tool

Laboratory Equipment

The laboratory exercises require access to the following standard equipment:

- Digital Multi-Meter (DMM)
- Dual-Channel Oscilloscope with two probes (must have math function with FFT)
- Function Generator
- Dual-Output Power Supply
- Equipment cables: banana-to-banana, banana-to-alligator, BNC-to-BNC, BNC-to-alligator

Linux and Unix in Electronic Design

In this course, we make extensive use of the free NGSpice simulator, based on Berkeley SPICE, which was originally developed in the Unix environment. Although you can obtain a version of NGSpice for Windows, and there are numerous SPICE-based simulators available on the Windows platform, the most advanced industry-grade electronic design software runs only on Unix or Linux operating systems. For that reason, lab simulations in this course will take place in our Linux-based Design Automation Laboratory.

At this level, we will focus our attention on the NGSpice simulator. In more advanced courses you may have opportunities to use the more advanced Electronic Design Automation (EDA) tools that are favored in the semiconductor industry. Here is a short list of EDA vendors to be aware of:

- Cadence Design Systems makes a massive array of tools used for all corners of electronic design industry. They are particularly used for custom design of analog and radio-frequency products, automated design of digital circuits, and assembly of large system-on-chip designs. Their leading products include the Virtuoso tools for analog and mixed-signal design, the Spectre analog simulator, the Incisiv mixed-signal (analog+digital) circuit simulation platform, and the Encounter digital place-and-route system.

- Mentor Graphics also produces a suite of products for IC design. Their leading products include the ModelSim simulator and the Calibre design-rule verifier.

- Synopsys is another leading player that is best known for their DesignCompiler tool for digital synthesis, and the HSPICE circuit simulator.

This is only a partial list of major players in the EDA industry and the products that are applicable for advanced system design, especially integrated circuits. For smaller printed circuit board projects or for personal use, some of the most respected tools are as follows:

- OrCAD is a suite of PCB design tools from Cadence. It includes the popular PSPICE simulator with schematic entry, along with a variety of tools for PCB layout and verification. **OrCAD offers a free (limited) student version**.

- EAGLE from CADSoft is very popular for designs with moderate complexity. It is **free for making small boards**, and is therefore a popular choice for students, hobbyists, and small-scale research projects as well as professional design.

- gEDA is a **fully free, open-source suite of EDA tools that uses NGSpice as its core simulator.** It supports graphical schematic entry and has PCB layout tools. It is available only on Linux and Mac platforms (some people have got it to work on Windows). It is not remotely as powerful as the commercial tools listed above, but is often valued by students who are interested in Linux and open-source software.

What is Unix?

Unix and "Unix-like" operating systems have played a key role in computer engineering for several decades. Today, every engineer should be comfortable working in a Unix/Linux environment (sometimes collectively called *nix). Unix system also come with a universe of powerful terminal commands and standard tools that must be learned. There are many ways to get started with Unix. In the 90's a PC-based "flavor" of Unix appeared called "Linux" (named for its creator, Linus Torvalds). Linux is a free, open-source software product that you can install on almost any computer (as an alternative to Windows). Today, Linux plays an important role in many engineering applications, including:

- Embedded computers such as media players (iPod, Apple TV), network equipment, automation controllers and GPS devices.
- Smart phones like the Android OS (based on Linux), and the iPhone (based on BSD Unix).
- Servers for a variety of network and telecommunication needs.
- Scientific computing and engineering simulation.
- Engineering design, especially integrated circuits and complex products.

Another flavor of Unix called "BSD" (short for Berkeley Software Distribution) is also the basis of many products, including Apple's Mac OS X operating system, and the iPhone and iPad products.

Getting Started

You can start using Linux immediately by setting up an account in our Design Automation Lab at USU. To get an account, contact Scott Kimber at *scott.kimber@usu.edu*. Once your account is enabled, you can login with your campus A-number and password. If you want to explore Linux on your own (strongly recommended!), there are many ways to start working with Linux on your personal computer. To start, you need to select a *distribution*, download and install it. A distribution is a packaging of the Linux operating system along with a variety of standard tools and configurations. Popular free distributions include Fedora, CentOS, Debian and Ubuntu. To install one of these distributions, there are several approaches:

- Download the install disk image and burn it onto a bootable disk or thumbdrive. Then boot up the target machine and install (this will usually replace the OS on your system). Here is a friendly download page for Fedora.
- If you have enough disk space, you can install a *dual-boot* system so that Linux and Windows live together on the same machine. Here are some example instructions for Ubuntu.
- If you have a Mac, OS X is already a Unix-based product. Most Linux software programs can be installed on a Mac using the Fink Project or the MacPorts Project.
- If you use Windows, you can install CygWin, a Unix-like environment that runs on top of Windows. Most of the important Linux software is available for Windows thanks to CygWin. A similar system called MinGW is also available.

Linux Packages

When you install Linux, you have the option to select packages. As an engineer, you probably want to become a power user, so the default packages will not be enough. Here are some things to select:

- Development packages. Especially the GNU compilers (gcc, g++) and development libraries (GNU Scientific Library, for instance).
- Editors. Emacs, XEmacs, vi, nano and pico are all good tools to have.
- Graphical tools. You will want XFig for drawing technical illustrations. You may also want to check out the Gimp (kind of like Photoshop).
- Scientific packages. Scan through the available packages. If it looks good, get it.

Using the Terminal

Forget pointing and clicking. Learn to use the terminal (aka console). All the power of Unix lies in the terminal command interface. Once you are comfortable using the terminal, you will be able to do 100 times as much, 10 times as fast. On any Unix computer, you can make yourself useful immediately just by finding the terminal. For example, some power-users like to hack their iPhones by accessing the terminal interface inside the device. This gives the user a lot of power to change the device's settings (in this case it also voids the warranty).

Here is a short list of essential terminal commands that you need to learn:

Command	Description
ls	List files in the current directory
cd	Change to a different directory
pwd	Print the present working directory
man	Lookup manual page for a command
cat	Print a text file's contents
more	Print a text file with pauses
less	Browse a text file (better than more)
ps	List IDs of processes that you started
kill	Kill a running process by ID
killall	Kill running processes by name
cp	Copy a file
mv	Move a file
rm	Remove a file
mkdir	Make a new directory
rmdir	Remove an *empty* directory
rm -R	Remove a directory tree (use caution!)
ssh	Secure connection to a remote server
scp	Securely copy files to/from a remote server

Command	Description
tar	Create or expand a backup archive
zip, unzip	Create or expand a ZIP file
gzip, gunzip	Create or expand a GZIP file
grep	Search for text patterns in files
find	Search for specific files
sed	Bulk search-and-replace
gedit	Text editor similar to Notepad (GUI)
emacs	Powerful guru-class text editor (GUI)
vim	Powerful guru-class text editor
evince	PDF document viewer (GUI)
pstopdf	Convert PostScript documents to PDF
epstopdf	Convert PostScript image to PDF

Exercises

Before doing these exercises, please study the Linux Tutorial available on the USU Electronics Wiki web site, at `http://electronics.wiki.usu.edu/Linux_Tutorial`. To pass this lab assignment, you must carry out the following activities using only Terminal commands. After completing each section, show your work to the instructor or TA by running the `history` command in the terminal. The `history` command will display all of the commands you have typed into the terminal, so that the instructor can verify your work.

Basic File Manipulation

1. Directory navigation: Using the `mkdir` and `cd` commands, create a directory tree that looks like this:

   ```
   ~/linux_tutorial
   ~/linux_tutorial/dir1
   ~/linux_tutorial/dir1/subdir1
   ~/linux_tutorial/dir1/subdir2
   ~/linux_tutorial/dir2
   ```

   Then verify your directory tree by running the command

   ```
   find -type d
   ```

   Once you have completed this part, show your result to the instructor or TA.

2. Terminal text editing: Change to directory `subdir1`, and run the command `pwd` to verify that you are in the right directory. Then create a file called "`Readme.txt`" using the `cat` command. The files contents should be as follows:

   ```
   ECE students are Linux gurus.
   ```

   Remember that when you are done typing your text in `cat`, you end the file by pressing `Enter`, then `Ctrl-D`.

3. Copying, renaming and deleting: Using the `cp` command, make a copy of `Readme.txt` called `Copy.txt`. Then use the `mv` command to rename `Copy.txt` to `Useless.txt`. Finally, delete the copy by using the command `rm Useless.txt`.

4. Environment variables. Many Linux programs use *environment variables* to control their configuration. Make an environment variable via the following commands::

   ```
   x=15
   echo $x
   ```

   Do not use any spaces around the = sign in the `x=15` statement. The `echo` command prints the environmental variable's value onto the terminal. The dollar-sign is used to indicate that x is a variable. Try running these commands to see the difference between `x` and `$x`:

   ```
   echo x
   echo $x
   ```

Environment Variables and Basic Programming

1. Exporting variables: When you make an environment variable, it only exists in your current shell. Other programs outside your shell (such as programs you launch from the Applications menu) will not be able to see this variable. To set a variable so that it can be seen by all your programs, use the `export` command:

   ```
   export x=15
   ```

   When a variable is set using the export command, it will exist for all programs that you launch until you logout. The variable will not exist any more after you log out.

2. The `$PATH` and `$HOME` variables: Some variables are set by the system every time you login. These include the `$HOME` variable, which says where your user's directory is located, and the `$PATH` variable, which tells the system where to look for executable files. Try these commands:

   ```
   echo $HOME
   echo $PATH
   ```

   The `PATH` variable is fomatted in a very simple way: `PATH=<dir1>:<dir2>:<dir3>:...` and so on. It is often helpful to include the current working directory in the PATH listing, so that the system knows to look in your current directory for executable programs. To add this to your path, use this command:

   ```
   export PATH=.:$PATH
   ```

   This command simply adds the characters "`.:`" to the front of PATH, so that the system will always look in the "`.`" directory first when searching for commands.

3. Launching a GUI: Go to directory `subdir2`, and create a file called `hello.cpp` using the command `gedit hello.cpp &`. The "`&`" tells the program to run in the *background*, so that you can continue using the terminal while the editor runs. Enter the following text exactly as shown:

   ```
   // hello.cpp
   // A C++ program written for the ECE Linux Tutorial

   #include <iostream>
   using namespace std;

   int main() {
     cout << "Hello world from the ECE Design Automation Lab!\n";
     return 0;
   }
   ```

4. Using the GNU compiler: Now, while still in `subdir2`, compile the `hello.cpp` program using the GNU g++ compiler. To do this, use this command

   ```
   g++ hello.cpp -o hello
   ```

   This will create an executable file called `hello` in `subdir2`. Now run your program by typing

   ```
   ./hello
   ```

If you correctly added the "." directory to your path (exercise 2), then you should be able to run the program by simply typing

```
hello
```

Please try this and verify that it works.

5. Creating permanent environment variables: You will sometimes need to create environment variables and settings that are permanent – i.e. they need to be set every time you use the terminal. Permanent changes can be made by editing a file called `.bashrc` in your home directory. To permanently add "." to your $PATH variable, edit your `.bashrc` file using this command:

```
cp ~/.bashrc ~/.bashrc_backup
nano ~/.bashrc
```

Note that you should always create a backup before changing `.bashrc`, in case you make some mistake. Add this line to the end of the file:

```
export PATH=.:$PATH
```

To save the change, type `Ctrl-O`. To exit the editor, type `Ctrl-X`. The `$PATH` variable will now include "`.`" every time you open a terminal. To test your new settings, type `bash` and press return. This relaunches the bash shell within your terminal. If you did everything right, you should be able to run your program by typing `hello` from within `subdir2`. If something goes horribly wrong, and your bash shell fails to relaunch, try typing `Ctrl-C` or `Ctrl-D` to cancel the new shell. This should take you back to your safe working shell, where you can restore your `.bashrc` file by typing `cp ~/.bashrc_backup ~/.bashrc`.

Manipulating Inputs and Outputs, and Searching

1. Output redirection: Now change directories to `dir1`. Perform a recursive directory listing using `ls`, and direct the output to a file called `listing.txt`. Then view the `listing.txt` file using the `more` command. Your command sequence should look like this:

```
ls -R > listing.txt
more listing.txt
```

2. Pipes: Now do the listing again, only use the *pipe* operator to send the listing directly to the `more` command, without needing to create a file:

```
ls -R | more
```

3. Using grep: You can use the grep command to search for text within a file. Change to subdir2 and run the following commands:

```
grep -n "Hello" hello.cpp
grep -l "Hello" hello.cpp
grep -i "heLlO" hello.cpp
grep -in "hello" hello.cpp
```

Note the affect of the options -i, -l and -n. Also note that they can be used in combination.

4. Using find: You can search your entire directory tree using the find command. Change to the linux_tutorial directory, and run these commands:

```
find -type d
find -type f
find -name hello.cpp
```

The “-type d” option searches for directory names. The “-type f” option searches for regular files. The “-name hello.cpp” option searches for a specific filename.

5. Using grep and find together: The Bash shell allows various ways to use output from one command as input to another. For example, the grep syntax is

```
grep <pattern> <filename>
```

For the `<filename>` option, you can insert a list of files returned by the `find` command. This is done by using backward-ticks (`) to surround the find command, like this:

```
grep -li "hello" `find -type f`
```

Run this command from the `linux_tutorial` directory, and it should print out a list of files that contain the text “hello”. The `grep` options are “i”, which specifies a case-insensitive search, and “l”, which requests that command print out names of files that contain the word “hello”.

6. Man: Many commands, like `grep` and `find`, have a large number of options and configurations. To find out more about a command, you can type `man <command>` in the terminal (“`man`” is short for “manual”). Try studying the grep options using this command:

```
man grep
```

Scripts and Archives

1. Scripts: You can automate a sequence of commands using a *script* file. Change to the directory `dir2`. In this directory, use gedit to create a file called `hello.sh`, with the following contents:

```
#!/bin/bash

echo "Hello World from the ECE Design Automation Lab!"

# Change to your tutorial's root directory:
cd ..

# List all the files you've created:
for x in `find -type d`
do
  echo $x "is a directory"
done
```

```
for x in `find -type f`
do
  echo $x "is a regular file"
done
```

Then, to run your script, first change the file's permissions to allow the user (you) to execute the script:

```
chmod u+x hello.sh
```

With the permissions properly changed, you may run the script with the command

```
./hello.sh
```

You should see that it prints out the hello message, and gives you a customized listing of your directory tree.

2. Archives: You can create an archive of your directories and files by using the `tar` command. Try running the following commands:

```
cd ~
tar -c -v -f linux_tutorial.tar linux_tutorial
ls
```

The `tar -c` command creates an archive called a "tarball". The name of the tarball is specified using the `-f` option. The -v option tells tar to print verbose messages while it works (you may want to omit this when archiving large directory trees). You can *extract* the tarball using the `tar -x` command:

```
mv linux_tutorial old_tree
tar -x -v -f linux_tutorial.tar
```

This creates an exact replica of the files stored in `linux_tutorial.tar`. The extracted files are now placed in a directory called `linux_tutorial`, hence restoring the entire file tree. You can use tar archives to create backups of your work, to exchange your files with other people, and to transfer files between different computers.

3. Compressed archives: Because tar archives can be quite large, they are usually *compressed* using the `gzip` program, which reduces the file's size. The best way to make a compressed archive is to add the `-z` option after the `tar` command:

```
tar -c -v -z -f linux_tutorial.tgz linux_tutorial/
```

Notice that the file is now called "`linux_tutorial.tgz`" instead of "`linux_tutorial.tar`". The `tgz` file extension indicates that the archive is compressed. You may also see files with names like "`file.tar.gz`". This is the same as "`file.tgz`". To expand a compressed archive, simply add the `-z` option to the `tar -x` command:

```
mv linux_tutorial old_tar_tree
tar -x -v -z -f linux_tutorial.tgz
```

Things to Check Off

As you complete these exercises, please check with your instructor or TA and check off the following items:

1. Directory listing.
2. `history` from Section 1.
3. Program `hello.cpp` compiles and runs.
4. Changes to `.bashrc`.
5. `history` from Section 2.
6. `history` from Section 3.
7. Script file `hello.sh` runs properly.
8. `history` from Section 4.

1 Lab Procedures, policies and Equipment

Objectives

- **To prepare materials needed for the semesters' lab assignments.**
- **To review techniques and procedures required for professional laboratory work.**
- **To gain experience using the function generator, power supply, oscilloscope and multimeter located in the ECE Circuits Lab (EL 104).**

Parts and Equipment Required

Components and Materials Needed:

- 10kΩ resistor (1)
- 1kΩ resistor (1)
- 1nF capacitor (1)
- solderless breadboard/superstrip (1)

Equipment to be Used:

- Digital Multimeter (1)
- Oscilloscope (2 channels)
- Function generator (1)
- Banana-to-alligator cables (1 pair, red and black)
- BNC-to-BNC cable (2)
- BNC-to-alligator cable (1)
- Oscilloscope probe (2)

1. Pre-Lab Exercises

Exercise 1.

Consider the following signal:

$$v_{\text{IN}}(t) = 1\text{V} + (2\text{V})\sin(2\pi f t),$$

for a frequency of $f = 10\text{kHz}$. Describe the magnitude spectrum, $|V_{\text{IN}}(f)|$, associated with this signal.

Exercise 2.

Suppose the signal from Exercise 1 is provided as input to the circuit shown in Figure 1.1. Give expressions for the output signal, $v_{\text{OUT}}(t)$.

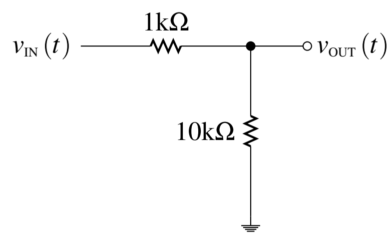


Figure 1.1. Circuit for Exercise 2.

Exercise 3.

Suppose the signal from Exercise 1 is provided as input to the circuit shown in Figure 1.2. Evaluate the gain $|H(f)|$ (in dB), the phase shift $\phi = \angle H(jf)$ (in degrees) and the output signal $v_{\text{OUT}}(t)$ (in Volts) at three frequencies, $f_1 = 1\text{kHz}$, $f_2 = 10\text{kHz}$, $f_3 = 50\text{kHz}$. What kind of filter circuit is this, and what is its 3dB cutoff frequency, in kHz? Finally, evaluate the phase ϕ at the 3dB cutoff frequency. Hint: The 3dB cutoff frequency is where the output amplitude falls to 0.707 of its maximum value. Also recall that the gain, in decibels, at specific frequency f is given by

$$\text{Gain(dB)} = 20\log_{10}\left(\frac{v_{\text{OUT}}(f)}{v_{\text{IN}}(f)}\right),$$

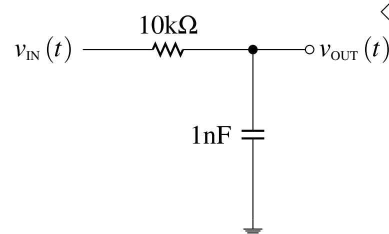


Figure 1.2. Circuit for Exercise 3.

Exercise 4.

Suppose the signal from Exercise 1 is provided as input to the circuit shown in Figure 1.3. Evaluate the gain $|H(f)|$ (in dB), the phase shift $\phi = \angle H(jf)$ (in degrees) and the output signal $v_{\text{OUT}}(t)$ (in Volts) at three frequencies, $f_1 = 1\text{kHz}$, $f_2 = 10\text{kHz}$, $f_3 = 50\text{kHz}$. What kind of filter circuit is this, and what is its 3dB cutoff frequency, in kHz? Finally, evaluate the phase ϕ at the 3dB cutoff frequency.

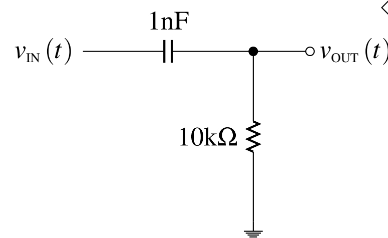


Figure 1.3. Circuit for Exercise 4.

2. Physical Experiments

Procedure 1. Connect the banana cables to the Digital Multimeter on your lab bench. Set the Multimeter to measure Ohms, and obtain precise measurements for the two resistors used in this lab. Record the values in your lab book.

Procedure 2. Using a BNC cable, connect the `Main` signal output from the Function Generator to the `CH 1` input on the oscilloscope. Also connect the `SYNC` output from the Function Generator to the `EXT TRIG` input on the oscilloscope. Perform the following steps:

Step A. Press the `TRIG MENU` button. Using the Menu buttons along the right side of the oscilloscope display, press the `Source` button until it displays `Ext`.

Step B. Press the `Channel 1` button. You should see the `Channel 1` Menu options appear along the right side of the scope display. Make sure that it reads `DC Coupling`, `BW Limit OFF`, `Probe 1X` and `Invert OFF`.

Step C. Adjust the `Volts/Div` and `Seconds/Div` knobs to get 1 V per division and 100 µs per division, respectively.

Step D. Adjust the `Amplitude`, `Offset` and `Frequency` knobs on the Function Generator until the settings match the waveform specified in Prelab Exercise 1. Set the Function Generator to a sine wave signal source. Use the oscilloscope display to verify your settings. <u>Note:</u> To adjust the offset voltage on the Function Generator, you will have to pull out on the knob to unlock its setting.

Step E. Press the `Measure` button on the oscilloscope. Set the `Source` to be `Channel 1` using the Menu buttons on the side of the scope. Using the measurement options, obtain precise measurements for the signal's *frequency*, its *peak-to-peak amplitude*, and it *average value* (i.e. the offset voltage). Record these values in your lab book.

Procedure 3. Press the `Math` button on the oscilloscope. Perform the following steps:

Step A. Using the Menu buttons along the right side of the oscilloscope, set the Math operation to `FFT`. Adjust the `Seconds/Div` and `Volts/Div` knobs to read 50 kS/s and 10 dB per division, respectively.

Step B. Record the spectrum displayed on the oscilloscope. Does it match your expectations? Take note of any anomalies, and offer hypotheses to explain them.

Step C. On the Function Generator, change the signal type to a square wave. How does this change the magnitude spectrum?

Step D. Now change the signal type to a triangle wave. How does this change the magnitude spectrum?

Procedure 4. Using your breadboard, connect a circuit to match the schematic in Figure 1.1. Perform the following steps:

Step A. Set the Function Generator to produce a sine wave. Press `CH 1 Menu` to turn off the FFT display on the oscilloscope. Verify that the Function Generator settings still match the waveform specified in Exercise 1.

Step B. Using a BNC-to-alligator cable, connect the Function Generator to your circuit. The alligator clips should connect to v_{IN} (red) and ground (black).

Step C. Connect the two probes to the oscilloscope. Connect both of the probes' black alligator clips to ground on your circuit board. Then connect the Channel 1 probe to v_{IN}, and the Channel 2 probe to v_{OUT}.

Step D. Adjust the oscilloscope so that both signals are visible. Adjust both Channels to display 1 V per division. Using the `Vertical Position` knobs, position the signals so that their reference points match (look for the little arrow on the left side of the scope display).

Step E. Using the `Measure` key, obtain precise measurements for the peak-to-peak amplitude and average value of each signal. Do the measurements match your expectation? Explain any difference.

Procedure 5. Change the circuit on your breadboard to match the one shown in Figure 1.2, and perform the following steps:

Step A. Connect the Channel 1 probe to v_{IN} and the Channel 2 probe to v_{OUT}. Set the input frequency to a low value of $f_1 = 1\text{kHz}$.

Step B. Press the `Math` button and activate the `FFT` function. Set the `Source` to be `CH 2`. Adjust the `VOLTS/DIV` and `SEC/DIV` knobs so that the display reads 10 dB per division and 50 kS/s, respectively.

Step C. You should see a peak at 1 kHz. Adjust the `Horizontal Position` knob so that the peak is centered on the oscilloscope display, then zoom in by pressing the `FFT Zoom` button. Zoom in on the magnitude display by adjusting the `Vertical Position` knob above `CH 2`. Change the `VOLTS/DIV` setting to 2dB per division and locate the peak on the display.

Step D. Press the `CURSOR` button and set the `Type` to `Magnitude`. Position `Cursor 1` at the top of the peak, and position `Cursor 2` 3dB below the magnitude of `Cursor 1` (watch the `Delta` measurement on the right side of the display to see when you reach 3dB).

Step E. Now change the `FFT Zoom` to `X1` and slowly increase the frequency on the Function Generator until the peak precisely touches `Cursor 2`.

Step F. Re-center the peak on the display by adjusting the `Horizontal Position` knob, and change the `FFT Zoom` to `X10`. Change the cursor `Type` to `Frequency`. Adjust one of the cursors to align with the new peak on the FFT display. This is the 3dB cutoff frequency for the circuit. Record this frequency in your lab book. Does it match your expectation from Prelab Exercise 3? Explain any differences.

Procedure 6. Change the circuit on your breadboard to match the one shown in Figure 1.3, and perform the following steps:

Step A. Repeat the steps from Procedure 5. This time, you will sweep from a high frequency to a lower frequency.

Step B. Adjust the `SEC/DIV` so that the display reads 100 kS/s.

Step C. Set the Function Generator to 40kHz. As before, locate the peak magnitude with `Cursor 1`, and position `Cursor 2` to a point 3dB below `Cursor 1`.

Step D. Slowly *decrease* the frequency on the Function Generator until the peak precisely touches `Cursor 2`.

Step E. Switch the cursors to `Type Frequency`, and align one of the cursors with the peak. This is the 3dB cutoff frequency of your high-pass circuit. Record this value in your lab book. Does it meet your expectations? Offer a hypothesis for any deviation.

Procedure 7. A digital oscilloscope takes discrete samples.When the sample rate is lower than double the frequency you are looking at, the apparent frequency will be lower than actual frequency. To observe this phenomenon,leave your circuit in the configuration of Procedure 6 Increase the frequency beyond the right edge of the FFT display. Continue increasing the frequency, up to 200kHz, and observe the FFT display. What happens as you increase the frequency? Explain what you observe, and comment on how this phenomenon may affect future lab experiments.

3. Post-Lab

In your lab book, write a summary of your findings. In particular, comment on the measured behavior of the voltage divider, low-pass and high-pass circuits. Can your measurements be precisely predicted by using formulas and your measured resistor values? In your opinion, how precise is your measurement of the cutoff frequencies for the low-pass and high-pass circuits? What steps could be taken to improve the precision? Also, comment on the effect observed in Procedure 7. What problems could this cause for future experiments, and how can you avoid those problems?

Finally, write a brief report describing your objectives, methods and major findings. Submit this report to the TA or Instructor along with your Lab Book.

2 Operational Amplifiers: Part A

Objectives

- **To demonstrate the principle of superposition by examining the characteristics of a weighted summer op amp configuration.**
- **To observe the effects of non-ideal amplifier characteristics, including finite input resistance, finite open-loop gain, and systematic offset voltages.**
- **To demonstrate methods of offset cancellation via capacitive coupling.**

Parts and Equipment Required

Components and Materials Needed:

- 741 Operational amplifiers (2).
- 10nF capacitor (1).
- 10kΩ resistors (5).
- 10kΩ potentiometer (1).
- Breadboard and hookup wire.

Equipment to be Used:

- Banana cable sets (4).
- Oscilloscope probes (2).
- Potentiometer adjustment tool (1).
- BNC-to-BNC cable (1)
- BNC-to-alligator cable (1)

1. Pre-Lab Exercises

Exercise 5.

Consider the inverting weighted summer circuit shown in Figure 2.1. Using only 10kΩ resistors, design the amplifier stage to implement the function $v_{\text{out}} = -v_1 - 2v_2$. Note that the input v_1 is supplied by a 10kΩ potentiometer. Predict the input resistances seen at terminals v_1 and v_2.

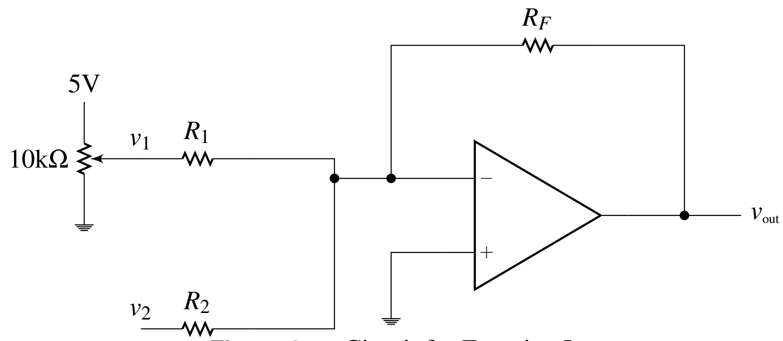


Figure 2.1. Circuit for Exercise 5.

Exercise 6.

Consider the revised weighted summer circuit shown in Figure 2.2. In this circuit, an AC coupling capacitor, $C = 10\text{nF}$, is inserted in the signal path of v_2. This creates a high-pass filter which rejects the DC offset of v_2, allowing only the AC part to be summed with v_1. What is the cutoff frequency (i.e. the *lowest* frequency that will be passed) for this high-pass configuration?

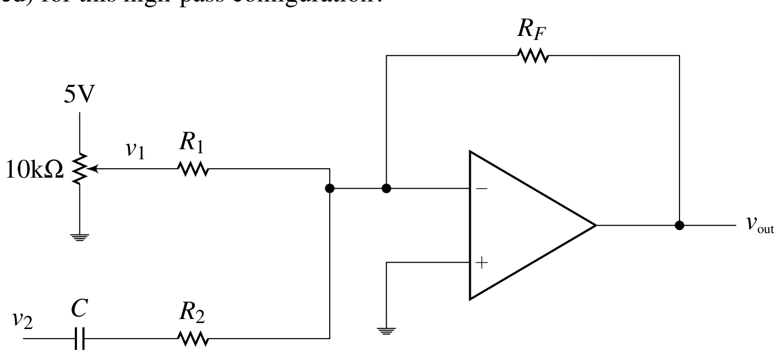


Figure 2.2. Circuit for Exercise 6.

Exercise 7.

Consider another circuit, shown in Figure 2.3, which is similar to the circuit from Exercise 6. In this circuit, a unity-gain follower is used to isolate the high-pass offset-reject filter from the input resistance in the second stage. Note that a 10MΩ resistor is inserted with the coupling capacitor in order to pass the op amp's non-ideal bias current. Predict the high-pass cutoff frequency for this two-stage configuration.

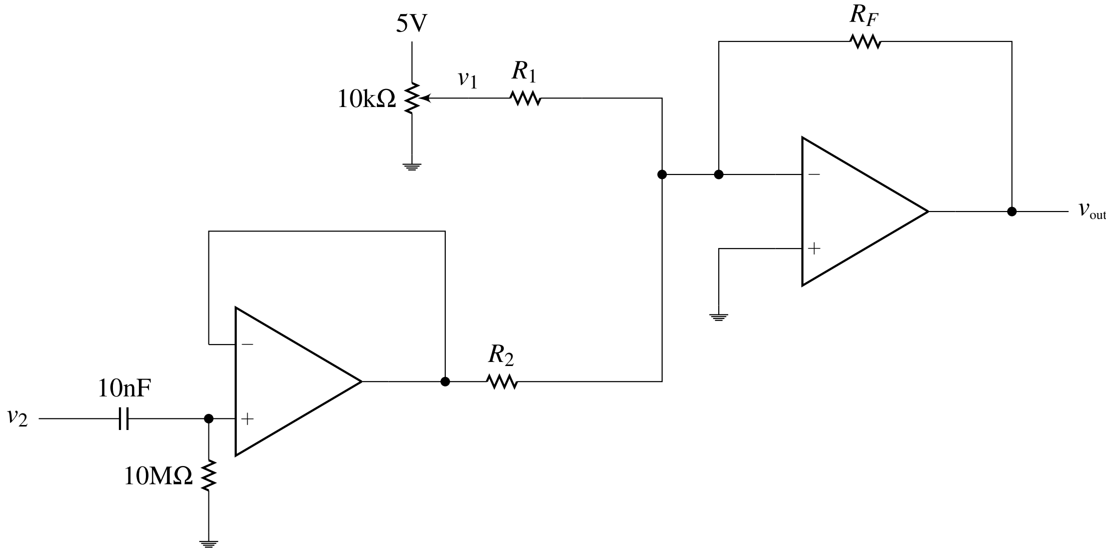


Figure 2.3. Circuit for Exercise 7.

2. Physical Experiments

Procedure 1. Using a breadboard, construct the circuit described in Figure 2.1. Use $+15$V and -15V for the 741's power supplies. Perform the following experiments:

Step A. Adjust the potentiometer so that $v_1 = 1$V. Connect v_2 to the 5V supply. Predict the amplifier's output for these input values. Measure the actual input and output values using the digital multimeter. How closely do the values match? Offer explanations for any discrepancy.

Step B. Connect v_2 to ground (zero potential). By adjusting the potentiometer, vary v_1 from 0V to 5V in steps of 1V. Record a table that includes precise measurements of both v_1 and v_{out}. In your table, also record the expected value of v_{out} for each measured v_1, and record the error (i.e. the difference between the expected and measured v_{out}).

Step C. Record two plots in your lab note book. In both plots, let the horizontal (x) axis be the input voltage v_1. In the first plot, draw graphs showing the measured and expected values of v_{out} for each measured v_1. In the second plot, draw a graph of the error. Answer the following questions, and justify your answers using features from your graphs:

i Does the circuit's gain differ from the designed value?

ii Does the op amp exhibit a systematic offset voltage?

Procedure 2. Construct the circuit described in Figure 2.2. Using a BNC-to-alligator cable, connect the function generator's output to the circuit's input at v_2. Perform the following experiments.

Step A. Offset Cancellation:

i Set the function generator to provide a sinusoidal waveform with 1V peak-to-peak amplitude and 50kHz frequency.

ii Using an oscilloscope probe, record a precise measurement of the peak-to-peak amplitude and offset voltage at v_2.

iii What is the gain at this frequency? Is it the same as the DC gain measured in Proc. 1?

iv Vary the function generator's DC offset and describe how v_2 and v_{out} respond. Note: you may need to *pull out* the offset adjustment knob in order to change the setting.

v While keeping the function generator's offset voltage fixed, vary v_1 by adjusting the potentiometer. For three separate values of v_1, record measured values for v_1, and predict the effect that the value will have on v_{out}. Using the oscilloscope, measure and record the offset voltage, the peak-to-peak amplitude and the frequency of v_{out} at each value. Do the measured values agree with your predictions? Offer explanations for any discrepancies.

Step B. Frequency Response:

i Sweep the function generator's input frequency *logarithmically* from 1kHz to 4MHz, doubling the frequency at each point in the sweep (i.e. measure at 1kHz, 2kHz, 4kHz, 8kHz, ...). At each point, measure the peak-to-peak amplitude of v_1 and v_{out}. Record a table including these values and the gain magnitude at each point (the gain magnitude is ratio of the measured amplitudes). Record the gain magnitude in both V/V and dB.

ii Using your measured data, create a Bode plot showing the magnitude (in dB) as a function of the frequency. Plot the frequency on a *log scale*.

iii Based on your Bode plot, what is the lower cutoff frequency of this circuit? Does the plot provide enough data to estimate the upper cutoff frequency? If so, record its value and state the total bandwidth (difference between upper and lower cutoff frequencies) for this circuit.

Procedure 3. Construct the circuit shown in Figure 2.3. Repeat the frequency-sweep measurement described in Proc. 2 B. This time, begin your sweep at a low frequency of 500Hz. After completing the measurement, explain any observed differences between the measured frequency responses, the cutoff frequencies, and the bandwidth of these two circuits.

3. Post-Lab

In your lab book, write a brief summary of your findings. Prepare a formal report describing the objectives, methods and major findings of this lab experience. Your report should compare results from pre-lab analyses, SPICE simulations and physical experiments. Submit this report online in Canvas, and have the TA examine and grade your lab book.

3 Operational Amplifiers: Part B

Objectives

- **Measurement of non-ideal op amp characteristics such as DC open-loop gain, slew rate, full-power bandwidth, and input offset voltage.**

Parts and Equipment Required

Components and Materials Needed:

- 741 Operational amplifier (1).
- 100Ω resistor (1).
- 10kΩ resistor (2).
- 100kΩ resistor (1).
- Data sheet for the uA741 Operational amplifier.

Equipment to be Used:

- Banana cable sets (5).
- Oscilloscope probes (2).
- Potentiometer adjustment tool (1).
- BNC-to-BNC cable (1)
- BNC-to-alligator cable (1)

1. Pre-Lab Exercises

In this lab, you will measure several non-ideal characteristics of the 741 op amp. In practice, these characteristics are very challenging to measure. The exercises below introduce some practical measurement techniques that can be used in our laboratory.

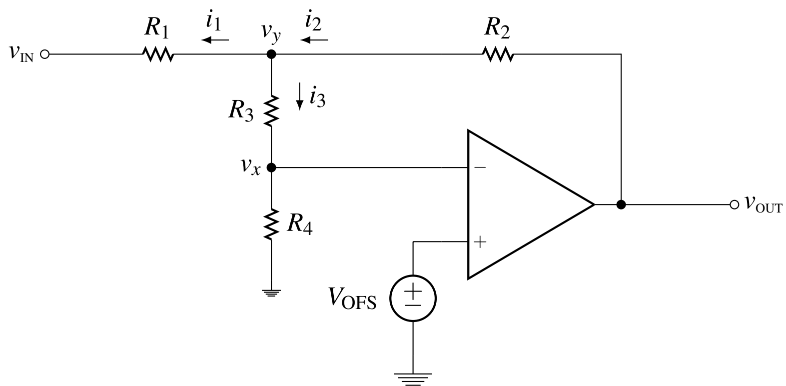


Figure 3.1. Circuit used for measuring the op amp's open-loop gain with resistor values $R_1 = 10\text{k}\Omega$, $R_2 = 10\text{k}\Omega$, $R_3 = 100\text{k}\Omega$, and $R_4 = 100\Omega$.

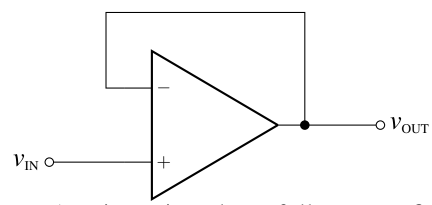


Figure 3.2. A unity-gain voltage follower configuration.

Exercise 8.

The circuit shown in Figure 3.1 is configured to allow measuring the op amp's offset voltage and open-loop gain. In this exercise, you will analyze the circuit's response to the offset voltage when $v_{\text{IN}} = 0$. Assume that the op amp has infinite open-loop gain and zero input bias current. Then there should be a perfect virtual short so that $V_X \approx V_{\text{OFS}}$ (note that we are using the all-capital notation to indicate a DC offset). Examine the circuit and verify that the following equations are correct:

$$I_2 = I_1 + I_3$$
$$I_1 = \frac{V_Y}{R_1}$$
$$I_2 = \frac{V_{\text{OUT}} - V_Y}{R_2}$$
$$I_3 = \frac{V_{\text{OFS}}}{r_4}$$

From these equations, solve for V_{OUT} and show that it is equal to

$$V_{\text{OUT}} = V_{\text{OFS}} \left(\left(1 + \frac{R_3}{R_4} \right) \left(1 + \frac{R_2}{R_1} \right) + \frac{R_2}{R_4} \right). \tag{3.1}$$

Using the resistor values stated in Figure 3.1, evaluate the expression in parenthesis and state the numerical value for the offset gain $G_o = V_{\text{OUT}} / V_{\text{OFS}}$.

Exercise 9.

The circuit shown in Figure 3.1 can also be used to estimate the op amp's open-loop gain A. To do this, we apply a sinusoidal input at v_{in} and measure the gain as the *ratio of amplitudes* $v_{\text{out}}/v_{\text{in}}$ (here we are using the all-lowercase notation to indicate AC signal amplitudes). To analyze the AC behavior, we assume finite open-loop gain but we still assume zero input bias current (this assumption is acceptable for the uA741 since I_{bias} is extremely small, around 10 nA).
By analyzing the currents in this circuit, it can be shown that

$$v_y \approx v_{\text{in}} \left(\frac{R_2 (R_3 + R_4)}{(R_1 + R_2)(R_3 + R_4) + R_1 R_2 + A R_1 R_4} \right) \tag{3.2}$$

$$\approx v_{\text{in}} \left(\frac{R_2 (R_3 + R_4)}{A R_1 R_4} \right), \tag{3.3}$$

where the approximation is made because the op amp's open-loop gain A is expected to be very large. (You are encouraged, but not required, to try and derive this result as an exercise). By using this approximation, the op amp's open-loop gain can be estimated:

$$A \approx \left(\frac{v_{\text{in}}}{v_y} \right) \left(\frac{R_2 (R_3 + R_4)}{R_1 R_4} \right) \tag{3.4}$$

To abbreviate this result, we define a constant

$$G_a = \left(\frac{R_2 (R_3 + R_4)}{R_1 R_4} \right),$$

so that the open-loop gain can be quickly computed as

$$A \approx \frac{v_{\text{in}}}{v_y} G_a.$$

Suppose that $v_{\text{in}} = 1\,\text{V}$ and v_y is measured to be $0.01\,\text{V}$. Based on Equation 3.4, calculate the op amp's open-loop gain A. Once you have obtained an estimate for A, substitute it into both (3.2) and (3.3), and compare the differences in these results. Based on your comparison, how accurate is the approximation used to obtain Equation 3.3?

Exercise 10.

Suppose the op amp has a *slew rate* $\text{SR} = 0.5\,\text{V}/\mu\text{s}$ and the rail voltages are $\pm 15\,\text{V}$. Calculate the *Full Power Bandwidth* (FPBW):

$$\text{FPBW} = \frac{\text{SR}}{2\pi |V_R|}. \tag{3.5}$$

Now suppose the op amp is configured as a voltage follower as shown in Figure 3.2, and the input is a sinusoid with zero offset:

$$v_{\text{IN}} = (2\,\text{V}) \sin(2\pi f t). \tag{3.6}$$

What is the maximum operating frequency f_{rmmax} for which slewing is avoided?

2. Physical Experiments

In your hardware session, you will attempt to measure the op amp's input offset voltage, open-loop gain and slew rate. When you complete each of your measurements, **write your result in a table on the white board** so that you can compare your measurements with those obtained by other students. This will give us a picture of how much variation occurs in these parameters.

Procedure 1. Connect your op amp in the configuration shown in Figure 3.1. Use the DMM to precisely measure all resistor values in your circuit. Using the measured values, evaluate the expressions in Equation 3.1 and 3.4.

Procedure 2. Connect the input v_{IN} to ground. Measure V_{OUT} and infer the value of V_{OFS} based on Equation 3.1.

Procedure 3. Now measure the DC open-loop gain of the op amp. To do this, use the function generator to supply an input signal at v_{in} with a frequency below 1 Hz. To observe such low frequencies on the oscilloscope, you will need to adjust the time setting to about `SEC/DIV` and use the `RUN/STOP` button to capture a snapshot of the waveforms. Adjust the amplitude and offset of v_{in} to ensure that v_{out} is not saturated or distorted, and is large enough to obtain a precise amplitude measurement. Use the oscilloscope to measure the peak-to-peak amplitudes at v_{in} and v_y, then estimate A using Equation 3.4.

Procedure 4. Wire your op amp in a unity-gain voltage follower configuration. Using the function generator, provide an input signal v_{in} with a peak-to-peak amplitude of about 4 V and a frequency of 50 kHz. On the oscilloscope, observe the waveform at v_{out} and estimate the slew rate by using the cursors to obtain Δv_{out} and Δt in the linear segments. AC coupling is recommended for this measurement.

Procedure 5. Reduce the signal frequency to about 40 kHz and observe the waveform at v_{out}. Can you perceive any distortion in the output waveform? Switch the oscilloscope to the FFT display. Can you see evidence of distortion in the FFT display? Repeat these observations at 30 kHz.

3. Post-Lab

In your lab book, write a summary of your findings, and write a brief report describing your objectives, methods and major findings. Submit this report to the TA or Instructor along with your Lab Book.

4 Diodes

Objectives

- **Gain practical experience with the behavior of silicon diodes under forward and reverse bias.**
- **Explore the use of diode devices in rectifying, limiting, and DC restoration circuits.**

Parts and Equipment Required

Components and Materials Needed:

- 10kΩ resistor (1).
- 10nF capacitor (1).
- 1N914 diodes (4).

Equipment to be Used:

- Banana cable sets (5).
- Oscilloscope probes (2).
- BNC-to-BNC cable (1).
- BNC-to-alligator cable (1).

1. Pre-Lab Exercises

In this lab we will perform experiments on six circuits, which are shown below. In each circuit, assume that R is 10k ohms, and C is 10nF. The 1N914 diode has n=2 and a forward voltage drop of 655mV at a forward current of 1mA.

Exercise 11.

For both of the circuits shown in Figure 4.1, identify the logical function (if V_1 and V_2 represent binary signals). Determine the value of v_{out} when $V_1 = 3\text{V}$ and $V_2 = 2\text{V}$. Determine the value of v_{out} when $V_2 = 2.8\text{V}$, 3.2V and 4V.

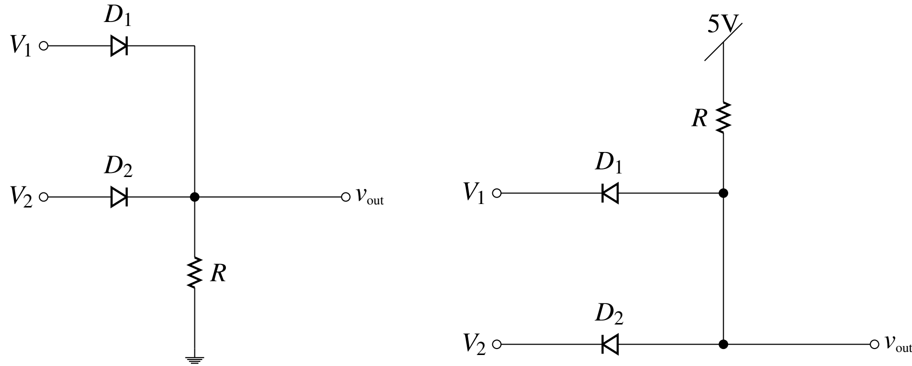


Figure 4.1. Diode logic circuits.

Exercise 12.

Figure 4.2 shows the schematic diagram for a half-wave rectifier circuit. If v_{in} is a 1kHz sinusoid with zero offset and a zero-to-peak amplitude of 2V, predict the waveform of v_{out} as accurately as possible without using iterative analysis or SPICE simulations. Sketch the predicted waveform in your lab book.

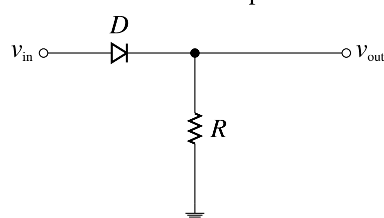


Figure 4.2. Half-wave rectifier circuit.

Exercise 13.

Figure 4.3 shows the schematic diagram for a half-wave peak rectifier circuit. Again assume that v_{in} is a sinusoid with zero offset and zero-to-peak amplitude of 2V. Predict the output waveforms when v_{in} has a frequency of 1kHz, 10 kHz, and 100kHz. Sketch the predicted waveforms in your lab book.

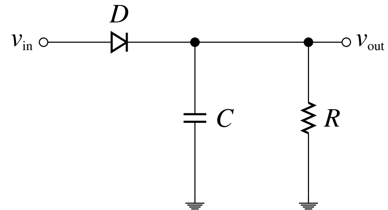


Figure 4.3. Peak rectifier circuit.

Exercise 14.

Figure 4.4 shows the schematic diagram for a voltage-limiting circuit. Assume that v_{in} is a 1kHz sinusoid with zero offset and zero-to-peak amplitude of 3V. Using the constant voltage- drop model, predict the output waveform as accurately as possible, and sketch the prediction in your lab book.

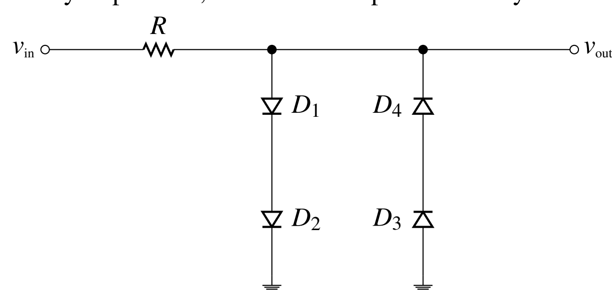


Figure 4.4. Voltage limiter circuit.

Exercise 15.

Figure 4.5 shows the schematic diagram for a DC restoration circuit (a.k.a. a clamped capacitor circuit). Assume that v_{in} is a 1kHz sinusoid with a 1V offset and a zero-to-peak amplitude of 3V. Using the constant voltage drop model, predict the output waveform as accurately as possible. Also predict the offset of the output waveform. Sketch the predicted waveform in your lab book.

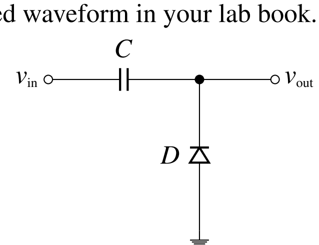


Figure 4.5. DC restoration circuit.

2. Physical Experiments

Procedure 1. Repeat the following procedures for both of the diode gates described in Figure 4.1.

Step A. Holding V_1 fixed at 3V, adjust V_2 from 2V to 4V in steps of 0.5V.

Step B. Plot the results in your lab book. How do the measured results compare with your predictions?

Procedure 2. Set up the half-wave rectifier circuit. Use the function generator to supply v_{in} as specified in the prelab.

Step A. Measure and record the output waveform, and sketch the result in your lab book. Be sure to accurately measure the duration of forward bias, the peak output voltage, and the output voltage under reverse bias.

Step B. How do the measured results differ from your predictions?

Procedure 3. Set up the half-wave peak rectifier circuit. Use the function generator to supply v_{in} as specified in the prelab.

Step A. Carefully measure and record the output waveforms for v_{in} at 1kHz, 10kHz and 100kHz.

Step B. Measure and record the peak-to-peak amplitude of the ripple in the output waveform at 1kHz, 10kHz and 100kHz.

Procedure 4. Set up the limiter circuit as specified in the prelab.

Step A. Measure and record the output waveform when v_{in} has a peak-to-peak voltage of 1 V.

Step B. Repeat the above measurement when v_{in} has a peak-to-peak voltage of 4 V.

Step C. How do the results compare to your expectations? Did you accurately predict the clipping voltage? If not, suggest reasons for the discrepancy.

Procedure 5. Set up the DC restoration circuit as specified in the prelab.

Step A. Measure and record the output waveform. Make sure you have `DC Coupling` in `Channel 1` and `Channel 2` settings. Ideally, the output offset should equal half its amplitude (i.e. the low peak of the sinusoid should be at ground).

Step B. Measure the actual minimum value of the output waveform. This value is expected to be slightly lower than the minimum value of the input waveform. Explain the measured result.

3. Post-Lab

In your lab book, write a summary of your findings, and write a brief report describing your objectives, methods and major findings. Submit this report to the TA or Instructor along with your Lab Book.

5 A MOSFET Active Balun Circuit

Objectives

- **To use knowledge of common-source, source-follower and common-gate amplifier configurations to design a practical active balun circuit.**
- **To confirm design predictions using hand analysis and simulation tools.**
- **To build and test the designed balun circuit; to practice systematic laboratory troubleshooting skills.**

Introduction

A *balun* is a circuit that converts a single-ended signal to a differential one. The single-ended signal could be generated by a sensor or antenna. Differential signals are often better for transmitting across long wires because they cancel out environmental interference and noise. Differential signals are also required for signal processing with differential circuits like modulators.

Most baluns are made using passive devices – transformers, resistors, inductors, etc. An *active balun* uses an amplifying device like a MOSFET or BJT. A standard MOSFET active balun circuit is shown in Figure 5.1. The MOSFET device in this circuit is biased in **saturation**, and **works both as a common-source (CS) amplifier and as a source-follower (SF)**. When considering the signal transferred from v_g to v_{o1}, we have a CS circuit. When considering the signal transferred from v_g to v_{o2}, we have a SF circuit. **To make a good balun, we want the CS and SF circuits to have equal gain, so that the differential output is *balanced*.**

1. Data Tables

Table 5.1. Given Data for the Active Balun Circuit.

Component	Description	Value
C_1	Input bypass capacitor.	20μF
R_1	Input high-pass filter resistor.	100kΩ
C_2	Output bypass capacitor.	1μF
R_L	Load resistance.	2kΩ
V_{DD}	Main supply voltage	10V

Table 5.2. Device data for CD4007 NMOS component.

Parameter	Description	Value
$\mu_N C_{ox}\left(\frac{W}{L}\right)$	Scale constant.	$333\mu\text{A}/\text{V}^2$
V_{T0}	Threshold voltage.	2.0V

Parts and Equipments Required

Parts

- (1) CD4007 chip.
- (4) 1μF capacitors.
- (1) 100kΩ resistor.
- (2) 1kΩ resistors.
- (1) 300Ω resistor.
- (4) calculated resistor values.

Equipment

- Dual power supply – use to supply V_{DD} and V_G.
- Digital multimeter – use to verify bias voltage and current.
- Function generator – use to generate v_{in}.
- Dual-channel oscilloscope – use to measure the input and output signals.

2. Pre-Lab

Exercise 16 (Active Balun Design Problem)**.**

You will design the active balun circuit of Figure 5.1 by choosing appropriate resistor values and bias voltages. Several of the resistances and bypass capacitor values are given in Table 5.1. The MOSFET $M1$ is an N-type device from the CD4007 package. The characteristics of $M1$ are given in Table 5.2.

Derive small-signal expressions for the mid-band Common-Source and Source-Follower gains of the balun circuit:

$$A_{CS} \triangleq \frac{v_{o1}}{v_g}$$

$$A_{SF} \triangleq \frac{v_{o2}}{v_g}.$$

Use the small-signal equivalent circuit model shown in Figure 5.2. Note that “mid-band” means you can ignore all capacitances in this analysis. **Record the derivations in your lab book, and put the expressions in your final report.**

We want the two gains to be $A_{SF} = 0.5\text{V/V}$ and $A_{CS} = -0.5\text{V/V}$. Then the overall differential gain will be $A_d = A_{SF} - A_{CS} = 1.0\text{V/V}$. Suppose the MOSFET has $g_m = 1\text{mA/V}$. **Solve for $\mathbf{R_S}$ and $\mathbf{R_D}$** under the assumption that r_o is very large (i.e. $r_o \to \infty$), and **verify that the gains are $\pm$0.5V/V**. Also **determine the MOSFET's large-signal bias current, $\mathbf{I_D}$** (use the formula $g_m = \sqrt{2\mu_n C_{ox}(W/L) I_D}$), and **determine the gate voltage, $\mathbf{V_G}$**, needed to generate this current. **Put all your analysis and calculations in your lab book**. List your final results in a table:

Parameter	Value	Parameter	Value	$M1$ Operating Region	(check one)
R_D		V_{S1}		Linear	
R_S		V_{G1}		Triode	
I_D		V_{D1}		Saturation	
$V_{GS1} - V_{T0N}$		V_{DS1}		Cutoff	

Include this table in your lab book and in your final report. You should find that $M1$ is operating in saturation. Recall that saturation is where $V_{DS} \geq V_{GS} - V_{Th}$. If your result is not in saturation, you should re-do all your design calculations.

Exercise 17 (Matched Impedance Design Problem)**.**

Suppose the balun's input signal is delivered from a 300Ω transmission line, and the balun is required to act as a matched line termination, i.e. the balun's input resistance should equal 300Ω. The NMOS active balun circuit has a very high input impedance due to the gate of $M1$, which causes reflections in the transmission line. To provide the proper input resistance, a common-gate amplifier circuit is inserted in series with the balun's input, as shown in Figure 5.3. The common-gate circuit is designed to have a total input impedance of 300Ω and a low gain.

Derive expressions for the gain and input resistance of the Common-Gate stage. Using these expressions, calculate values of R_2, R_3 and $M2$'s bias current, I_{D2}, to satisfy the following constraints:

- $g_{m2} = 1.5mA/V$.
- $A_{CG} = 1$V/V, where $A_{CG} \triangleq v_{g1}/v_{s2}$.
- $R_{in} = 300\Omega$.

Use the mid-band equivalent small-signal circuit model shown in Figure 5.4. Note that in the mid-band model, the C_1 acts like a short circuit, so $v_{d2} = v_{g1}$. The input resistance, R_{in}, is the effective resistance seen looking into the common-gate circuit. Derive an expression for R_{in} using by applying a test voltage v_x to the input, as illustrated in Figure 5.5. Then solve for the total current, i_x, that goes into the amplifier. The input resistance is $R_{in} = v_x/i_x$. Using the small-signal model, show that the input resistance is $R_2 \parallel 1/g_m$, and the gain (from source to drain) is approximately $g_m R_3$ (note that R_1 is ignored in this analysis because it is very large, so that $R_3 \parallel R_1 \approx R_3$). Calculate the values for R_2 and R_3 that are required to yield the specified input resistance and gain.

Once your resistor values are chosen, solve the DC values of V_{D2}, V_{S2} and V_{G2} and verify that $M2$ is in saturation. Record your final expressions and calculated values in a table like this one:

Parameter	Expression	Calculated Value	Parameter	Value
R_{in}			$V_{GS2} - V_{T0}$	
A_{CG}			V_{S2}	
R_2			V_{G2}	
R_3			V_{D2}	
I_{D2}			V_{DS2}	

$M2$ Operating Region	(check one)
linear	
triode	
saturation	
cutoff	

Record all derivations and calculations in your lab book. In your final report, give the derived expressions and calculated values in a table.

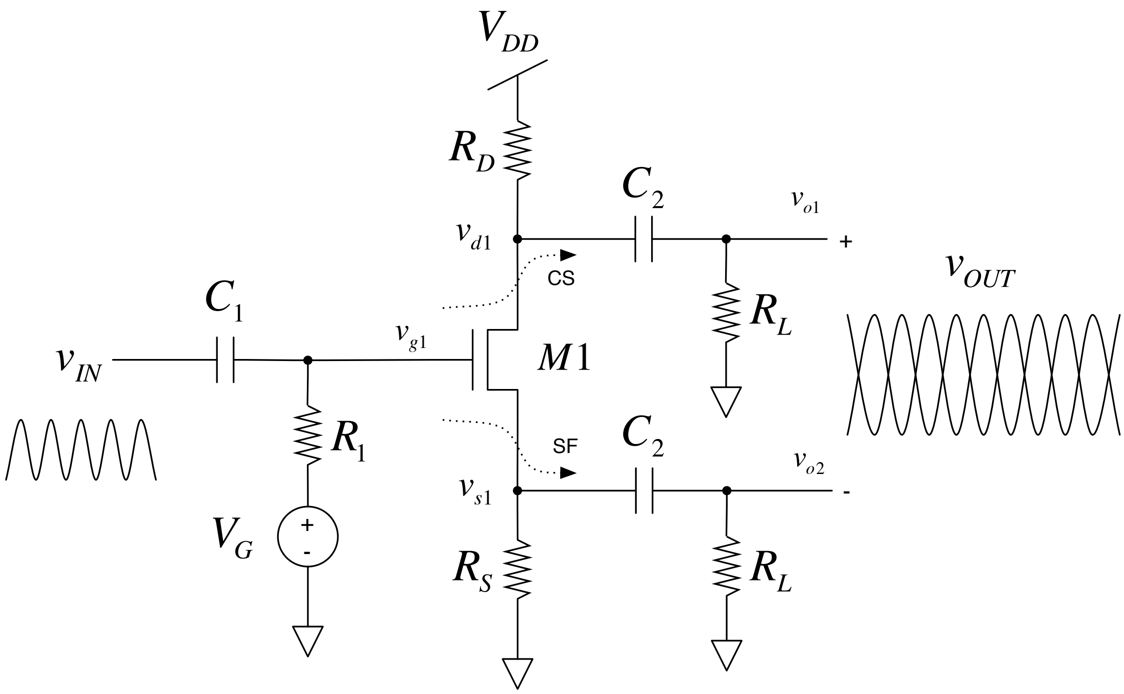


Figure 5.1. A MOSFET active balun circuit.

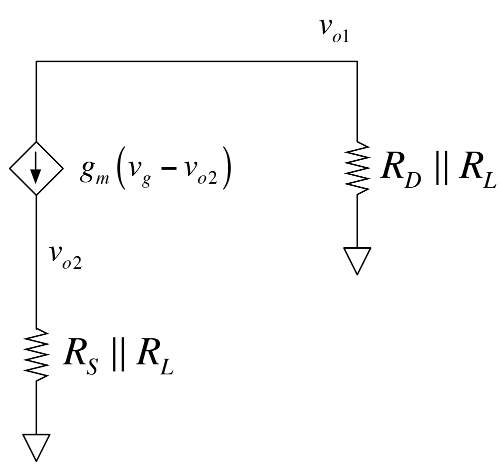


Figure 5.2. Small-signal mid-band equivalent circuit for the active balun.

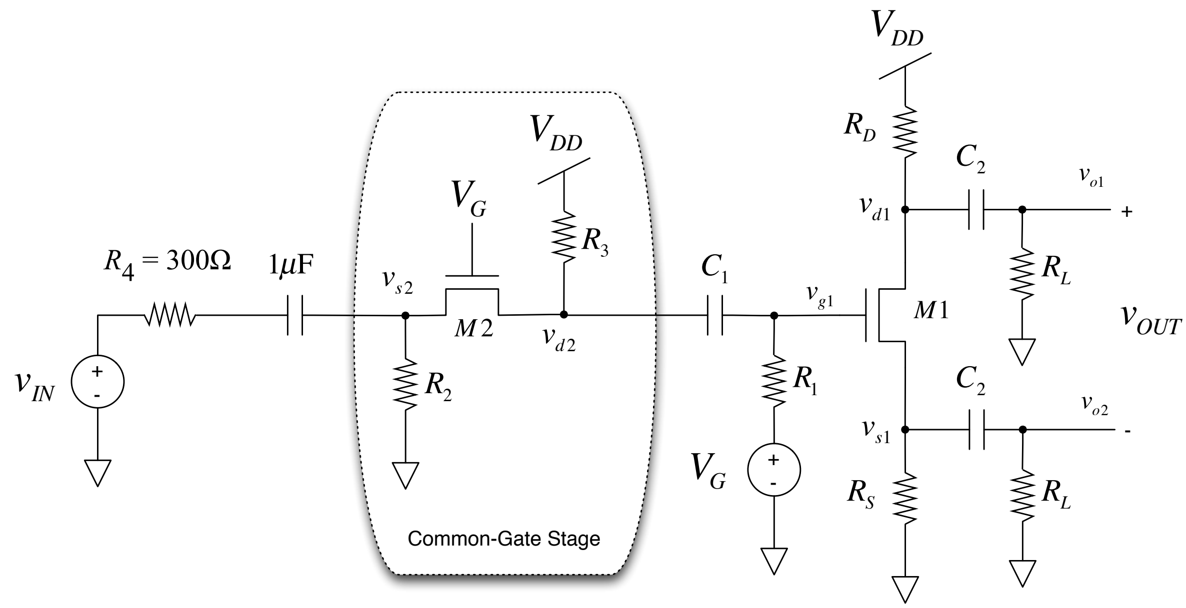


Figure 5.3. A common-gate circuit is inserted to match the balun's input impedance.

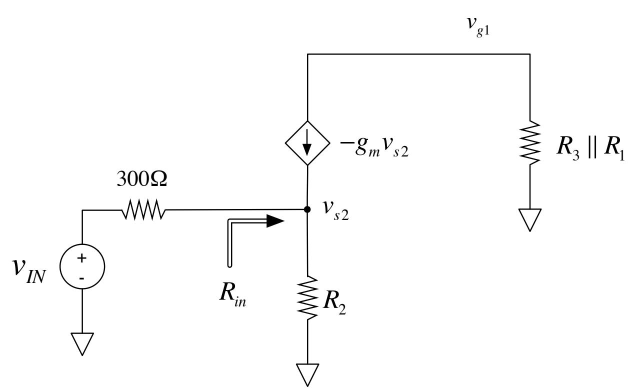


Figure 5.4. Small-signal mid-band equivalent circuit for the common-gate stage.

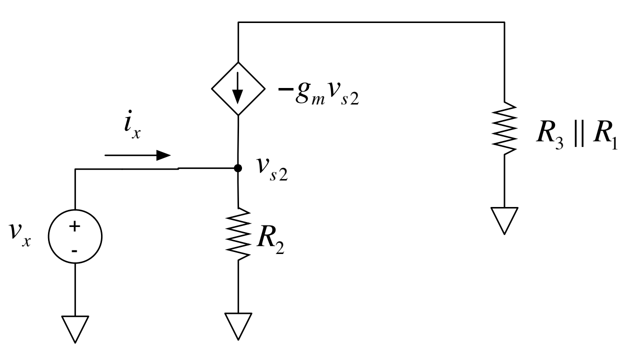


Figure 5.5. Model for calculating the effective input resistance, R_{in}, of the Common-Gate amplifier.

Exercise 18 (SPICE simulation)**.**

You should now have a complete active balun design with values for all components and bias voltages. Create a SPICE description of the active balun circuit with Common-Gate input stage (Figure 5.3). For the input signal, use a 50kHz sinusoidal voltage source with a peak-to-peak amplitude of 1V. Perform a transient simulation covering three periods of the input signal. **Plot the single-ended input and the differential output signals, and include these plots in your lab book and final report.**

Verify that the output signals are *amplitude-balanced* and *phase-balanced*. To do this, measure the amplitude of v_{o1} and v_{o2}. If their amplitudes are equal, then the signal is amplitude balanced. If they are not equal, **record the amount of amplitude imbalance in your lab book, and explain what design change could be made to improve the amplitude balance**.

Measure the amplitude imbalance as the difference in amplitudes, in dBm:

$$\Delta A = 20\log\left(\frac{|A_{CS}+A_{SF}|}{0.001}\right),$$

where A_{CS} is assumed to be a negative quantity, and A_{SF} is assumed to be positive.

To measure the phase balance, zoom in to where the transient output curves cross zero. They should ideally cross zero at the same time. If they don't, measure the time difference, Δt, between the zero-crossings. Then the phase imbalance is given by

$$\Delta\phi = 2\pi f \Delta t.$$

Record the phase imbalance in your lab book, and include it in your final report. Also include your SPICE input file as an appendix to the report.

3. Physical Experiments

Procedure 1. First construct the active balun circuit of Figure 5.1[1]. Use the bench power supply to provide $V_{DD} = 10V$, and to provide the desired gate bias voltage V_G.Use the digital multimeter to precisely measure the value of all resistances. Record these measurements in a table in your lab book.

Procedure 2. Measure the DC operating point.

Step A. Step Use the digital multimeter to measure the DC voltage at every node in your circuit.

Step B. Use the multimeter to measure the bias current, I_D, by connecting the meter probes *in series* between R_D and V_{DD}.

Step C. Record the measurements in a table and compare them with your design predictions.

[1] **Caution**: CMOS devices are very sensitive to electro-static discharge (ESD). To avoid damaging your components, always touch a grounded surface before handling any MOS chips. Ideally, you should wear a grounded wrist strap when handling sensitive components.

Step D. Re-calculate your design predictions using the measured resistor values, and add these results to your table.

Step E. Take note of any remaining discrepancies, and try to explain them in your report.

Procedure 3. Verify the small-signal behavior.

Step A. Connect the function generator to create a 50kHz signal with a peak-to-peak amplitude of 1V. Verify the waveform using the oscilloscope before connecting it to your circuit.

Step B. After the waveform is verified, connect it to your circuit. Use the oscilloscope probe to measure the output waveforms v_{o1} and v_{o2}.

Step C. If you are using a digital-capture oscilloscope, record the waveform and include it in your lab book and final report. Alternatively, use a digital camera to take a snapshot of the scope display. As a last resort, you may sketch the waveform, but be sure to include as much detail as possible.

Procedure 4. Measure the amplitude balance.

Step A. Using the oscilloscope, obtain precise measurements of v_{in}, v_{o1} and v_{o2}.

Step B. Record a table of gain values in your lab book:

$$
\begin{aligned}
A_{CS} &= \frac{v_{o1}}{v_{in}} \\
A_{SF} &= \frac{v_{o2}}{v_{in}} \\
A_d &= \frac{v_{o1}+v_{o2}}{v_{in}}.
\end{aligned}
$$

Step C. Do the measured results agree with the predictions? How much amplitude imbalance is observed?

- Record the measurements in a table and compare them with your design predictions.
- Re-calculate your design predictions using the measured resistor values, and add these results to your table.
- Take note of any remaining discrepancies, and try to explain them in your report.

Step D. Include the amount of amplitude imbalance in your final report.

Procedure 5. Measure the phase balance.

Step A. Use the zero-crossing procedure to measure the phase imbalance of your circuit.

Step B. Record this value in your lab book and include it in your final report.

Procedure 6. Build and verify the Common-Gate stage.

Step A. Disconnect the function generator from your circuit, and construct the Common-Gate circuit of Figure 5.3.Use the digital multimeter to measure the precise value of all resistors.Record these measurements in a table in your lab book.

Step B. Measure the DC voltages and bias current of the Common-Gate stage.

- Record the measurements in a table and compare them with your design predictions.
- Re-calculate your design predictions using the measured resistor values, and add these results to your table.
- Take note of any remaining discrepancies, and try to explain them in your report.

Step C. Connect the signal generator to your circuit as indicated in Figure 5.3.

Step D. Verify the input impedance as follows. First, use the oscilloscope to precisely measure the amplitudes of v_{in} , v_{g2} and v_{g1}.

- The ratio of amplitudes is $v_{g2}/v_{in} = R_{in}/(R_{in} + R_4)$.
- Using this formula and your measurement of R_4, determine the value of R_{in}.
- Also calculate the gain $A_{CG} = v_{g1}/v_{g2}$.
- Record this measurement in your lab book, and in your final report.

4. Report

Your final report should be a carefully written, formal summary of your lab results. It should include the following specific information:

- A short introduction (one or two paragraphs) to the balun circuit, identifying its applications and explaining how it works.
- Your derived expressions for the gains and input resistance: A_{CS}, A_{SF}, A_d, A_{CG} and R_{in}. It is not necessary to include the details of your derivation if they are in your lab book and have been checked by the instructor or TA.
- A table reporting the DC operating point, the small-signal gains, and the input resistance of your circuit. Include three columns:
  - Your original design predictions.
  - Your revised predictions based on measured resistor values.
  - Your actual measurements.
- Your measured amplitude and phase imbalance, as measured from SPICE simulations and the actual circuit.
  - If you observed significant imbalance, offer a reasonable hypothesis to explain why. Suggest at least one solution to remove the imbalance.
- A plot showing your SPICE transient simulation results.
- A figure showing the oscilloscope waveform from the lab.
- Conclusions should summarize the success of your design choices and succinctly report any pitfalls that may affect users of this circuit in the future.
- Your SPICE circuit description should be attached as an appendix.