EEE334 Lab Guidelines

**NOTE:** *If you are repeating or retaking EEE334, YOU MUST RE-DO ALL THE LAB EXPERIMENTS regardless of what previous EEE334 lab grades you may have received.*

### The Lab Area

1. **EEE334 lab area is in GWC 273, ALL EXPERIMENTS ARE CONDUCTED AT THIS LOCATION**
2. If taking EEE334 off campus for undergraduate credit, you must come to GWC 273 at the main ASU campus to perform the experiments
3. There is **NO FOOD OR DRINK ALLOWED in GWC 273**
4. In order for the Department to estimate semester demand for TAs (Teaching Assistants), **ALL STUDENTS MUST SIGN IN AT THE FRONT DESK WHEN ENTERING GWC 273**
5. Lab work is under an **OPEN LAB SYSTEM**, that is, there is no specific required lab time in which you must work. You may perform your work anytime the lab is open, however, **it is strongly recommended to perform all lab experiments when an EEE 334 Lab TA is on duty since only she/he can verify the correctness of your lab data**
6. When conducting experiments, **at most 3 students are allowed per workstation, BUT** the collection of required results and observations onto a **DATASHEET during the experiment, must be done INDIVIDUALLY**

### Experiments

1. Experiments involve Calculations, LTSpice simulations and Laboratory exercises
2. There are 7 lab experiments and **a Lab Report is required per experiment**
   
   a. Labs 1-6 are mandatory for everyone and are graded by the EEE 334 Administrator TA
b. Lab 7 is only for Honor Students who choose to do it for Honors credit and is graded by the course Professor

3. Labs 4 through 6 require the use of an Electrostatic Sensitive Device (ESD), a Metal Oxide Semiconductor Field Effect Transistor Chip (MOSFET chip)

Prior to lab 4, arrangements will be made for students to sign out MOSFET ICs and a Ground Strap that is to be worn when handling these ESD devices

4. There is no lab final exam, thus lab scores are determined based on the average of the LAB REPORT GRADES

Conducting Lab Experiments

1. Prior to conducting an experiment, it is strongly recommended to:
   a. Read the experiment’s instructions
   b. Do required calculations on the DATASHEET
   c. Do required LTSpice simulations and record observations on the DATASHEET

Datasheet Rules:

a) The collection of at least ONE FULL PAGE of required results and observations IS MANDATORY PER STUDENT to receive FULL CREDIT

The Datasheet includes:

1. Calculations (Theoretical results)
2. LTSpice Observation, usually of the Transient and Frequency Analysis, (Simulated results)
3. Experimental Observations (Lab results)

The above allows for comparison between theoretical, simulated and lab results in order to confirm agreement of the three result types.
b) After conducting an experiment or part of it, ask the EEE 334 TA on duty to verify the correctness of your data:
   - If the data sheet is complete containing all required results and observations, the TA shall stamp, sign and date it.
   - If the data sheet is NOT complete, the TA may sign and date the partial work as a proof of work in progress; **ONLY COMPLETE DATA SHEETS RECEIVE STAMP AND CREDIT**
   - Data sheets brought to the TA without any evidence of doing lab work will **NOT** be stamped nor signed.

**NOTE 1:** Lab TAs from other courses may stamp and sign your lab data sheets but they **CANNOT GUARANTEE** the correctness of your data, and you will then bear full responsibility and penalty for collecting incorrect data.

**NOTE 2:** COPIED and PHOTOCOPIED Datasheets receive a grade of **0** and owner will be referred to the course professor.

2. In the lab, when collecting components to assemble circuits **USE THE METER TO ENSURE CORRECT COMPONENT VALUE**
3. All components and equipment used while conducting experiments **MUST ALWAYS BE RETURNED** to their proper location after use.

Lab Reports General Information

**NOTE 1:** Due to student volume per semester and TA working hours, instructions will be provided to write the lab reports individually or in groups of no more than three students.

**NOTE 2:** In order to obtain credit, the Lab Report Format Guidelines provided in the next section **MUST BE FOLLOWED CLOSELY** when writing lab reports for the experiments in this course. **Report formats from other courses DO NOT APPLY to EEE334 lab reports.**
1. ALWAYS include the stamped and signed data sheet or sheets with the lab report, MISSING DATA SHEETS will result in a 0 (zero) grade for the entire lab report
   - The Report for Lab 7 does not require a data sheet

2. Turn in ONLY ON-TIME Lab Reports into the proper slot of the wooden box inside the GWC 273. Placing reports in the wrong slot will result in a -10% penalty per day late

Late Report Policy

NOTE: Never place a LATE LAB REPORT into the wooden box, otherwise there will be a minimum of -50% in penalties.

a. Late lab reports must be given directly to an EEE334 Lab TA for she/he to record the turn in date in order to award the report the appropriate credit
b. Penalty for late labs is –10% per day late, not including Sundays or University holidays
c. Under special circumstances, if certain lab report has already been graded and returned, in order to turn in that report for full credit, it is necessary to obtain a written note from the course Professor instructing the Report Grader to accept the late report without penalty, otherwise the late report will not be accepted

Lab Report Grading Distribution

Discussed in the next section, are the multiple components of the lab report; their point distribution is tabulated below:

<table>
<thead>
<tr>
<th>Component</th>
<th>Points</th>
</tr>
</thead>
<tbody>
<tr>
<td>Title page</td>
<td>2 pts</td>
</tr>
<tr>
<td>Introduction</td>
<td>3 pts</td>
</tr>
<tr>
<td>Equipment and Components</td>
<td>3 pts</td>
</tr>
<tr>
<td>Course of Action</td>
<td>7 pts</td>
</tr>
</tbody>
</table>
### Results

<table>
<thead>
<tr>
<th></th>
<th>pts</th>
</tr>
</thead>
<tbody>
<tr>
<td>Results</td>
<td>30</td>
</tr>
<tr>
<td>Discussion</td>
<td>10</td>
</tr>
<tr>
<td>Closing Argument</td>
<td>5</td>
</tr>
<tr>
<td>Post lab questions</td>
<td>20</td>
</tr>
<tr>
<td>Stamped/Signed datasheet</td>
<td>20</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td><strong>100</strong></td>
</tr>
</tbody>
</table>

### EEE334 Lab Report Format Guidelines

This section describes the *mandatory format to follow when writing the EEE334 lab report*. As mentioned before, report formats learned for other courses do not apply to EEE334 lab reports and deviation from the correct format will result in severe credit penalty.

The lab report is a standalone document, in other words, there should be no reference made to the lab manual, the datasheet or the text book in order for its content to be interpreted. It must be computer generated and it consists of the following elements which are described in the appropriate lab report order:

1. **Title Page (2 Points)**
   
   Includes in the following order:
   
   a. Course: **EEE334**
   
   b. Experiment number
   
   c. Experiment Topic
   
   d. Experiment Due Date
   
   e. Student’s name(s)
   
   f. Course Professor(s)

### IMPORTANT NOTE APPLICABLE TO THE NEXT SECTIONS
2. Introduction (3 Points)
   In ONE PARAGRAPH, effectively describe the experiment’s objective. Be specific but concise. This may be achieved by considering the following questions in each experiment:
   a. What is this experiment about or what will be studied in this experiment?
   b. What components or special equipment will be used?
   c. What type of analysis or results will be researched?

3. Equipment and Components (3 Points)
   Provide a simple list or table of the Equipment and Component used per experiment section.
   a. Equipment used: Quantity, name and model
      Example of lab Equipment includes:
      Oscilloscopes, Multimeters, Signal Generators, DC Power Supplies, Curve Tracers and Transformers
   b. Components used: Quantity, name and value where applicable
      Examples of Components to be used are:
      Resistors, Capacitors, Zener Diodes, Diodes, Bridge Rectifiers, MOSFET devices and Potentiometers

4. Course of Action (7 Points)
   Each experiment is composed of multiple parts; in each part do the following:
a. List the number and name of the experiment section.

b. If a circuit is required, the first step must indicate the building of the circuit followed by its LTSpice schematic:
   i. Paste a copy of the LTSpice schematic ONLY, **NO DC voltage or current results on schematic ALLOWED at this point.**
   ii. Name the schematic appropriately according to the circuit’s function, i.e. Low Pass Filter, Common-Source Amplifier, etc.

c. After the well addressed circuit schematic, USE AT MOST 10 BULLETS TO DESCRIBE the logical steps of the COURSE OF ACTION taken to obtain **ALL** the required results:
   i. **Mention:** Where applicable, obtaining specific required calculations, LTSpice analysis and lab results.
   ii. **Do not mention:** Equipment set up or actual results in this section.

5. **Results (30 Points)**
   This is a collection of **ALL THE RESULTS** required throughout the experiment. All the results per experiment segment are collected and place in this section.

   **RELEVANT PARAMETERS Definition**
   These are important attributes used to describe a function such as:
   
   - **Waveforms:** i.e. a sine wave is described by its amplitude (Vpp, Vp, Vrms), Period, Frequency, Phase Shift, Offset, etc. In this lab only Vpp, Vrms and sometime Phase Shift will be required as relevant parameters.
   - **Plots:** i.e. a Frequency response plot is described by the Gain (in dB or dimensionless), Bandwidth, 3dB – Frequency (fH, fL where applicable), half-power Gain, Cut-off frequency, etc. In this lab only the dB Gain, Bandwidth and sometimes fH and fL will be required as relevant parameters.

Result sections MUST be presented as follows:

a. **Appropriate number and name of the result section**
   i. **Definitions,** when required
   ii. **Theoretical Calculations**
NOTE: *ALWAYS show calculations to prove work was NOT copied*

1. Use a computer program to generate equations.
2. **ALWAYS** specify **UNITS**.
3. Box or highlight Required Results.

iii. LTSpice

**PASTE** into this section where applicable:

1. Schematics with DC voltages and currents.
2. Simulation Plots

**NOTE:** *All plots MUST BE TRANSPARENT COPIES for clarity purposes, refer to the LTSpice Guide to learn about transparent copies (LTSpice results continued below).*

a. **Transient Analysis (Time Domain)**
   i. Name plot appropriately.
   ii. Plot MUST show at least ONE FULL OSCILLATION OR TEN where steady state is not instantaneous.
   iii. **ALWAYS** specify RELEVANT PARAMETERS on all plots.

b. **AC Sweeps (Frequency Analysis)**
   i. Name plot appropriately.
   ii. Plot MUST show a frequency sweep in logarithmic scale (Decades) from 1Hz to 1GHz.
   iii. **ALWAYS** specify relevant parameters on plots, i.e. Max. dB Gain, Bandwidth, etc.

iv. **Lab Results**

1. **Oscilloscope Screenshots**
   a. All waveforms displayed MUST include ALL RELEVANT PARAMETERS where applicable, i.e. Vp-p, Vrms, Phase Shift, etc.
b. Specify clearly which waveform belongs to the Input and which to the Output

2. Measurements and Calculations
   a. Use measured lab results to calculate required results where applicable and place them in this section
   b. Box or highlight required results

v. Comparison and % Error Table
   1. Tabulate Required Results (use Excel)
      a. The table must contain columns for:
         i. Obtained Theoretical, LTSpice and Lab results
         ii. % Error between Theoretical and LTSpice results
         iii. % Error between Lab and Theoretical results
         iv. % Error between Lab and LTSpice results

      NOTE: In all % Error cases show calculations or at least equation used.

6. Discussion (10 Points)
   The numerical results from the previous section are not enough for clarity purposes, thus a theoretical explanation of the results is also necessary. For each section of the results do the following (one paragraph per section):
   a. Number and name the Discussion of the result section appropriately.
   b. Discuss ALL THE REQUIRED RESULTS per section by pointing out:
      i. Significance of the results.
      ii. Unexpected outcomes and wisely speculate their reason (i.e. large error is due to measuring methods, components actual values, etc).
      iii. Agreement of all Theoretical, LTSpice and Lab results (i.e. % error).

      NOTE: When comparing results use ALWAYS % Error, i.e. “The LTSpice CS Amplifier Gain was 4.5% greater than the Theoretical Gain obtained, this could be due to inaccurate MOSFET parameters used in theoretical calculations, etc.”
iv. Section results conclusion (one line).

7. Closing Arguments (5 Points and possibility of extra credit)
   a. This is a paragraph commenting on the experience of carrying out the experiment.
   
   **NOTE:** This section IS NOT a conclusion of the results; therefore DO NOT COMMENT ON THE RESULTS. Commenting on results belongs to the Discussion section.

   b. The paragraph MUST include:
      i. A statement of whether the experiment goal was achieved or not.
      ii. What was learned from performing the experiment
      iii. Difficulties encountered with lab equipment or components, LTSpice, TAs, Manual, book, team members, etc.
      iv. Suggestions for improvement in future lab experiments.
      
      **NOTE:** Useful suggestions may be rewarded with extra credit

8. Post Lab Questions (20 Points)
   a. Provide brief but complete answers to all post lab questions.
   b. ALWAYS use LTSpice to support answers (where applicable).

9. Lab Data Sheets (20 Points)
   
   **NOTE:** Datasheets are covered in detail under EEE334 Lab Guidelines. As mentioned in that section, EACH STUDENT MUST OBTAIN AT LEAST ONE FULL PAGE of data during each experiment. COPIED and PHOTOCOPIED datasheets receive a grade of 0 and owners will be referred to the course Professor.

   Since lab reports may be written individually or as groups of no more than three students, a lab report may contain one or up to three Data Sheets.
      i. Each Data Sheet MUST include the owner’s name.
      ii. Attach ALL stamped and signed lab Data Sheets to the back of the lab report.
NOTE: Failure to include stamped and signed data sheets with the lab report will result in a 0 (zero) report grade since Datasheets are proof of lab work performed.
LTSpice Guide

You will need to master LTSpice Software for circuit simulation. This is a powerful and user friendly CAD tool available free here:

http://www.linear.com/designtools/software/

You can find several tutorials online. Some good sets can be found here:

http://denethor.wlu.ca/ltspice/


Below are the most used components when building schematic circuits in this course:

1. DC source: voltage
2. Resistor: res
3. capacitor: cap
4. inductor: ind
5. AC source/sine wave: signal
6. Nmos: nmos4
7. Pmos: pmos4

Some useful shortcuts:

1. To rotate an element, i.e., a resistor: Ctrl+R
2. Open components list: F2
3. Enable delete: delete

This concludes LTSpice Guidelines
Tektronix–DPO4032 Oscilloscope Guide

This Guide goes over brief descriptions of commonly used Measurements and Functions

Note: Tektronix DPO4032 Oscilloscope Instruction Manuals are available in the lab area.

Measurements

Time Measurements Description

1. **Period**: Time required to complete the first cycle in a waveform or gated region.
   
   \[ \text{Period} \rightarrow T = 1/f \quad \text{Period Units} \rightarrow \text{Seconds (s)} \]

2. **Frequency**: The first cycle in a waveform or gated region.
   
   \[ \text{Frequency} \rightarrow f = 1/T \quad \text{Frequency Units} \rightarrow \text{Hertz (Hz)} \]

3. **Delay**: The time between the mid reference (default 50%) amplitude point of two different waveforms.

4. **Rise Time**: The time required for the leading edge of the first pulse in the waveform or gated region to rise from the low reference value (default = 10%) to the high reference value (default = 90%) of the final value.

5. **Fall Time**: The time required for the falling edge of the first pulse in the waveform or gated region to fall from the high reference value (default = 90%) to the low reference value (default = 10%) of the final value.

6. **Positive Pulse Width**: The distance (time) between the mid reference (default 50%) amplitude points of a positive pulse. The measurement is made on the first pulse in the waveform or gated region.

7. **Negative Pulse Width**: The distance (time) between the mid reference (default 50%) amplitude points of a negative pulse. The measurement is made on the first pulse in the waveform or gated region.

8. **Phase**: The amount of time that one waveform leads or lags another waveform, expressed in degrees where 360° comprises one waveform cycle.
Amplitude Measurements Description

1. **Pk-Pk**: The absolute difference between the maximum and minimum amplitude in the entire waveform or gated region.
2. **Amplitude**: The high value less the low value measured over the entire waveform or gated region.
3. **RMS**: The true Root Mean Square voltage over the entire waveform or gated region.
4. **Mean**: The arithmetic mean over the entire waveform or gated region.
5. **High**: This value is used as 100% whenever high reference, mid reference, or low reference values are needed, such as in fall time or rise time measurements.
6. **Low**: This value is used as 0% whenever high reference, mid reference, or low reference values are needed, such as in fall time or rise time measurements.
7. **Max**: The most positive peak voltage. Max is measured over the entire waveform or gated region.
8. **Min**: The most negative peak voltage. Min is measured over the entire waveform or gated region.

Functions

**High Resolution**
This is used to clean up noisy waveforms, for noise may add up to 2V to a signal. Use this function before obtaining measurements for accuracy. To get to High Res, press:

   **Acquire > Mode > Hi Res**

**Measure**
This is used to obtain the relevant parameters of waveforms. To use this function press:

   **Measure >Select Measurement > (Page through 1 - 7 pages of measurements and select desired Measurement, on the right side of the screen)**

**Save** (to a USB memory device)
This is used to save a waveform’s image or the data that produces it. Saving the data allows for manipulation using Excel.

- Saving formats available to **Save Screen Image** are:
  - bmp, png, and tif. **It is recommended to save screen images in bmp format and with Ink Saver ON so you save printer ink and have nice legible plots.**

- To **Save Screen Image**:
  - Save/Recall: Menu > Save Screen Image > (File Format) .bmp (and Ink Saver) ON > OK Save Screen Image

- Saving formats available to **Save Waveform** are:
  - isf and csv (Spreadsheet File Format). **It is recommended to save waveforms in csv format, when later manipulation of this data is necessary.**

- To Save Waveform:
  - Save/Recall: Menu > Save Waveform > To File > (Use the **Multipurpose a** knob to select where to save to) > Spreadsheet File Format (.csv) > OK save

**Math**

Sometimes this function will be used to obtain currents by dividing a voltage waveform by a resistance value. To do this press:

Math > Advanced Math > Edit Expression > (Use the **Multipurpose a** knob to scroll to selections, then **Enter Selection** or **Backspace** as needed) > OK Accept

When using the Math function for any other purpose, follow the same criteria.

**XY Mode**

This is covered in detail in lab 3.

---

This concludes the Tektronix – DPO 4032 Oscilloscope Guide
**EEE334 Lab#1**

**LTSpice and Lab Orientation - Instruments and Measurements**

It is strongly recommended to perform lab experiments when an EEE334 Lab TA is on duty, since only she/he can verify the correctness of your lab data. *Other Lab TAs may stamp and sign your lab data sheets but CANNOT GUARANTEE the correctness of your data, and you will then bear full responsibility and penalty for collecting incorrect data.*

In this experiment, the student will become familiar with basic circuit implementation in LTSpice and with using the lab equipment to be utilized throughout this course. In LTSpice there will be DC and AC circuit simulation, Transient and AC analysis exercises. In the lab, the equipment to be used includes oscilloscope, multimeters, signal generators, and DC power supplies.

In this lab, the student is expected to learn the order lab data should be collected for report purposes, LTSpice and equipment setup and measurements, thus this experiment is structured and step by step instructions are provided to achieve those objectives, later labs will not include this format or level of detail except for special cases.

### 1.1 Simple DC circuit

<table>
<thead>
<tr>
<th>VS</th>
<th>6V DC</th>
</tr>
</thead>
<tbody>
<tr>
<td>VS</td>
<td>8V DC</td>
</tr>
<tr>
<td>R1</td>
<td>3.9kΩ</td>
</tr>
<tr>
<td>R2</td>
<td>3.9kΩ</td>
</tr>
<tr>
<td>R3</td>
<td>3.9kΩ</td>
</tr>
</tbody>
</table>

![Simple DC circuit diagram](Fig.1.1)
1. **Calculate** (hand/calculator calculation) $V_{out}$ using any appropriate method, e.g. Superposition, Thevenin, .etc.

2. **Calculate** the current through $R_2$.

3. **Simulate** (simulate the circuit in LTSpice) the circuit in **LTSpice** obtaining $V_{out}$ and the current through $R_2$ (See LTSpice Guide, **DC in Circuits**).

   NOTE: ALWAYS paste LTSpice outcomes onto the result section of the lab report (for this part, paste both the LTSpice circuit and the Operating Point results in the lab report)

   NOTE: i) place all parts including the ground; ii) label all nodes in the circuit; iii) run the simulation and choose DC operating point to check the DC voltages and currents

4. In the lab, construct the circuit (build the circuit on your breadboard) in Fig. 1.1, ensure resistor values by measuring them using the RLC meter.

Take a few minutes to get acquainted with the lab equipment. Refer to the equipment manuals if needed. The TA will come around to answer any questions. The equipment is user friendly. You will almost certainly be able to figure out the controls without reading much of the manuals.

Use the digital multimeter to:

5. Measure and record $V_{out}$

6. Measure and record the current through $R_2$

   **Hint:** You can measure the current (AC or DC) using one of the following ways:

   1. Measure the voltage across the resistor and divide by the value of the resistor

   2. Measure the current directly using an ammeter, but you need to break your circuit since the ammeter is connected in series.

7. Use % Error to compare the calculated, LTSpice and Lab results (pairwise).

   Example:

   $$\%Error = \frac{(MeasuredValue - CalculatedValue) \times 100}{MeasuredValue}$$

**1.2 Simple AC circuit**
1. Calculate the current through R\(_2\) and R\(_4\) in the circuit shown in Fig. 1.2.
2. Calculate the voltage across R\(_1\), R\(_2\), R\(_3\) and R\(_4\).
3. Define V\(_{\text{rms}}\) and calculate V\(_{\text{rms}}\) for R\(_2\).
4. Simulate the circuit in LTSpice.
   b. Set up for Transient Analysis (Click on Run and choose Transient, choose appropriate stop time to show at least two complete cycles) and obtain (Plot Setting -> Add trace) in one plot the Current waveforms through R\(_2\) and R\(_4\) (right click on the variable name on graph to enable cursor to measure relevant parameters, such as amplitude and peak-to-peak value).
   c. Obtain in one plot the Voltage waveforms through R\(_1\), R\(_2\), R\(_3\) and R\(_4\) (label points on the circuit to obtain the voltage drop).

NOTE: ALWAYS paste both the LTSpice circuit and the resulting Transient Analysis plot onto the result section of the lab report (use Tools to save plots, the color of graph can be edit under Color Preferences).

HINT: never put voltages and currents waveform in the same plot! Since the units are different.

5. In the lab, assemble the circuit in Fig. 1.2, ensure components values by using the RLC meter.
6. Set the Function Generator.

   **Note:** In this course, ALWAYS, set generator to HIGH Z

   a. Set the Function Generator (Agilent 33220A) to HIGH Z, as follows:
   
   b. Press “Utility” button.

   Navigate the menu to set the output termination.


   d. Press “Load” softkey to choose “High Z”

   e. Press “Done” softkey

   Notice the status message “High Z Load” is shown in the upper-right corner of the display

   f. Adjust the function generator, $V_s$, to provide a 2 Vpp sinewave at 100 Hz.

7. Use the digital multimeter to obtain Voltage across $R_2$ and $R_4$

8. Use the oscilloscope (Tektronix–DPO4032) to obtain the Voltage and Current waveforms across $R_2$ and $R_4$. See the Oscilloscope Guide to learn how to use the applicable functions:

   a. Use the **Measurement** function to obtain $V_{pp}$, and $V_{rms}$ of $R_2$ and $R_4$

   b. Use the **Save** function to save the screenshot with $V_{R2}$, $V_{R4}$ and their relevant values to a USB memory device **to paste onto the result section of the lab report**.

   c. Use the **Math** function to obtain the Current waveforms across $R_2$ and $R_4$.

   d. Use the **Save** function to save the screenshot with $I_{R2}$, $I_{R4}$ and their Pk-Pk, and RMS values to a USB memory device **to paste onto the result section of the lab report**.

9. Document the comparison between the digital multimeter Voltage results for $R_2$ and $R_4$ and the oscilloscope $V_{rms}$ values for the same resistors. This results should be equal since:

   **A digital multimeter always gives the RMS value of voltage and current**

10. Change the generator waveforms to a **Triangular wave** and then to a **Square wave**. In both cases repeat steps 8, 8a, 8b and 9 above.
11. Calculate the factor between the \textbf{V}_{\text{peak}} \text{ obtained from the oscilloscope and the }
\textbf{V}_{\text{rms}} \text{ value obtained from the multimeter for the Triangular and the Square
waves. For example for the Sine wave the factor is } 1/\sqrt{2}, \text{ since :}
\textbf{V}_{\text{rms}} = \left(\frac{1}{\sqrt{2}}\right) \times (\textbf{V}_{\text{peak}}) \quad \text{OR} \quad \textbf{V}_{\text{rms}} = (1/2 \times \sqrt{2}) \times (\textbf{V}_{\text{pk-pk}})

12. Use % Error to compare the calculated, LTSpice and Lab results.

### 1.3 Low Pass Filter Analysis

Fig. 1.3 shows a low pass filter network. The voltage Gain is the absolute value of the ratio of the output voltage to input voltage. The Gain of this filter is given by the \textit{magnitude} of the following transfer function:

\[ \frac{V_o}{V_i} = \frac{I}{j\omega R \ G \ I} \]  \hspace{1cm} (1.1)

<table>
<thead>
<tr>
<th>VS</th>
<th>2 Vpp</th>
<th>\text{f}=1kHz</th>
</tr>
</thead>
<tbody>
<tr>
<td>R</td>
<td>1kΩ</td>
<td></td>
</tr>
<tr>
<td>C</td>
<td>0.01μF</td>
<td></td>
</tr>
</tbody>
</table>

![Fig.1.3 Low Pass Filter Network](image)

1. Calculate the voltage Gain of the filter using eqn. (1.1) (voltage Gain units are
\text{V}/\text{V}).
2. Simulate the circuit in LTSpice
3. Do Transient Analysis of \text{V}_{\text{out}} and \text{V}_{\text{in}}. Use the results to:
   a. Calculate the filter’s Gain.
   b. Obtain the Phase Shift of \text{V}_{\text{out}} with respect to \text{V}_{\text{in}}.

\textbf{NOTE: ALWAY}S paste the resulting \textbf{Transient Analysis} plot onto the
\textbf{result section of the lab report}. 
4. Do **AC Analysis** of \((V_{out}/V_{in})\) (run -> AC analysis, Type of Sweep: **Decade**, and pick appropriate **start frequency** and **end frequency**). Use the results to:
   a. Obtain the filter’s Max Gain in dB (NOTE: in LTSpice, the y axis is in dB in AC sweep, i.e. \(20 \times \log_{10}(\text{Abs}(V_{out}/V_{in}))\)).
   b. Obtain the frequency point where:
      \[\text{Gain} = \text{Max Gain} - 3\text{dB}\]. This is referred as the \(f_{3\text{dB}}\) point. It is used to determine Bandwidth because it represents the frequency at which \(V_{out}\) has lost half of its original power.
   c. Obtain the \(\omega_{3\text{dB}}\) Bandwidth of the Low Pass filter by using:
      i. \[\omega_{3\text{dB}} = (2\pi f_{3\text{dB}}) \text{ rad/sec}\]
      **NOTE:** ALWAYS paste the resulting AC Analysis plot onto the result section of the lab report.
      **NOTE:** the solid line shows the magnitude.
   d. Calculate the \(\omega_{3\text{dB}}\) Bandwidth mathematically by using \(\omega_{3\text{dB}} = 1/RC\).
   e. Compare the values of \(\omega_{3\text{dB}}\) Bandwidth from simulation and calculation.

5. In the lab, construct the circuit shown in Fig.1.3, and measure the components to ensure their values using the RLC meter.

6. Set the Function Generator (AGILENT 33220A) to HIGH Z and adjust it to provide a 2 Vpp sine wave at 1 kHz.

7. Place channel 1 of the oscilloscope (Tektronix–DPO4032) at \(V_S\) and channel 2 at \(V_{out}\).

8. Display both waveforms on the oscilloscope screen
   a. Use the **Measurement** function to obtain the relevant measurements, such as \(V_{pp}, V_{rms}\) and in this case **Phase Shift of Vout only**.
   b. Save the screenshot with both waveforms and parameters to a USB memory device **to paste onto the result section of the lab report**.
   c. Use the measured results of Vout and the known value of Vin to calculate the filter’s Gain.

9. Use % Error to compare the calculated, LTSpice and Lab results.
1.4 High Pass Filter Analysis

Fig. 1.4 shows a high pass filter network. The voltage Gain is the absolute value of the ratio of the output voltage to input voltage. The Gain of this filter is given by the magnitude of the following transfer function:

$$\frac{V_o}{V_i} = \frac{1}{(1/j\omega R) + j}$$  \hspace{1cm} (1.2)

<table>
<thead>
<tr>
<th>VS</th>
<th>2Vpp</th>
<th>10kHz</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td>10nF</td>
<td></td>
</tr>
<tr>
<td>R</td>
<td>22 kΩ</td>
<td></td>
</tr>
</tbody>
</table>

Procedure:

1. Calculate the voltage Gain of the filter using eqn. (1.2) (voltage Gain units are V/V).
3. Do Transient Analysis of Vout and Vin. Use the results to:
   a. Calculate the filter’s Gain.
   b. Obtain the Phase Shift of Vout with respect to Vin.
   
   **NOTE: ALWAYS paste the resulting Transient Analysis plot onto the result section of the lab report.**
4. Do AC Analysis of (Vout/Vin). Use the results to:
   a. Obtain the filter’s Max Gain in dB.
   b. Obtain the frequency point where:
Gain = Max Gain – 3dB. This is referred as the $f_{3dB}$ point. It is used to determine Bandwidth because it represents the frequency at which $V_{out}$ has lost half of its original power.

c. Obtain the $\omega_{3dB}$ Bandwidth of the High Pass filter by using:
   
   ii. $\omega_{3dB} = (2\pi f_{3dB})$ rad/sec

   **NOTE:** ALWAYS paste the resulting AC Analysis plot onto the result section of the lab report.

d. Calculate the $\omega_{3dB}$ Bandwidth mathematically by using $\omega_{3dB} = 1/RC$.

e. Compare the values of $\omega_{3dB}$ Bandwidth from simulation and calculation.

5. In the lab, construct the circuit shown in Fig.1.4, and measure the components to ensure their values using the RLC meter.

6. Set the Function Generator (AGILENT 33220A) to HIGH Z and adjust it to provide a 2 Vpp sine wave at 10 kHz.

7. Place channel 1 of the oscilloscope (Tektronix–DPO4032) at $V_S$ and channel 2 at $V_{out}$.

8. Display both waveforms on the oscilloscope screen.

   d. Use the **Measurement** function to obtain the relevant measurements, such as $V_{pp}$, $V_{rms}$ and in this case Phase Shift of $V_{out}$ only.

   e. Save the screenshot with both waveforms and parameters to a USB memory device to paste onto the result section of the lab report.

   f. Use the measured results of $V_{out}$ and the known value of $V_{in}$ to calculate the filter’s Gain.

9. Use % Error to compare the calculated, LTSpice and Lab results.

**Post-lab Questions:**

1. With respect to AC parameters, what type of Voltage or Current measurement is taken with digital multimeters and what types with oscilloscopes?

2. Why is it dangerous to short a voltage source?

3. Use LTSpice to implement and discuss the outcomes of the followings:
a. Modifying the resistors, $R_1$ and $R_2$, to be $10\,\text{k}\Omega$ in the simple AC circuit shown in Fig. 1.2, instead of $1\,\text{k}\Omega$.

b. Applying a sinusoidal input signal of $20\,\text{Vpp}$ and $10\,\text{kHz}$ in the low pass filter circuit shown in Fig. 1.3, as a substitute of $2\,\text{Vpp}$ and $1\,\text{kHz}$.

**Note:** Always do Transient and AC analysis when there are frequency dependent components in the circuit.

c. Modifying the resistor $R$ to be $2\,\text{k}\Omega$ in the High Pass Filter circuit shown in Fig. 1.4, instead of $22\,\text{k}\Omega$.

**Note:** Always do Transient and AC analysis when there are frequency dependent components in the circuit.

4. Determine the nominal value of the resistor, along with its tolerance, equivalent to the following colors of the bands. Use Table 1.1 and eqn. (1.3).

   i. Band A, brown; band B, black; band C, red; band D, gold.

   ii. A, B, and C bands are yellow and band D is gold.

   iii. If the nominal value of the resistor is $10\,\Omega$, what would be the best choice of the color bands?

\[
R = (10A + B) \times C \pm D\% \quad \text{(1.3)}
\]

<table>
<thead>
<tr>
<th>Band A</th>
<th>Band B</th>
<th>Band C</th>
<th>Band D</th>
</tr>
</thead>
<tbody>
<tr>
<td>Color</td>
<td>First</td>
<td>Second</td>
<td>Color</td>
</tr>
<tr>
<td></td>
<td>Significant digit</td>
<td>Significant digit</td>
<td>Multiplication</td>
</tr>
</tbody>
</table>

**Table 1.1 Color codes for resistors**
<table>
<thead>
<tr>
<th>Color</th>
<th>Fig.</th>
<th>Fig.</th>
<th>Color</th>
<th>Fig.</th>
<th>Fig.</th>
<th>Color</th>
<th>Fig.</th>
<th>Fig.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Black</td>
<td>0</td>
<td>0</td>
<td>Black</td>
<td>1</td>
<td></td>
<td>No color (or black)</td>
<td></td>
<td>±20</td>
</tr>
<tr>
<td>Brown</td>
<td>1</td>
<td>1</td>
<td>Brown</td>
<td>10</td>
<td></td>
<td>Silver</td>
<td></td>
<td>±10</td>
</tr>
<tr>
<td>Red</td>
<td>2</td>
<td>2</td>
<td>Red</td>
<td>100</td>
<td></td>
<td>Gold</td>
<td></td>
<td>±5</td>
</tr>
<tr>
<td>Orange</td>
<td>3</td>
<td>3</td>
<td>Orange</td>
<td>1,000</td>
<td></td>
<td>White or green</td>
<td></td>
<td>±5</td>
</tr>
<tr>
<td>Yellow</td>
<td>4</td>
<td>4</td>
<td>Yellow</td>
<td>10,000</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Green</td>
<td>5</td>
<td>5</td>
<td>Green</td>
<td>100,000</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Blue</td>
<td>6</td>
<td>6</td>
<td>Blue</td>
<td>1,000,000</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Purple</td>
<td>7</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Gray</td>
<td>8</td>
<td>8</td>
<td>Silver</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>White</td>
<td>9</td>
<td>9</td>
<td>Gold</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
It is strongly recommended to perform lab experiments when an EEE334 Lab TA is on duty, since only she/he can verify the correctness of your lab data. Other Lab TAs may stamp and sign your lab data sheets but CANNOT GUARANTEE the correctness of your data, and you will then bear full responsibility and penalty for collecting incorrect data.

This lab involves a study of the properties of operational amplifiers connected in the inverting and non-inverting configurations, as well as the application of op-amps, such as integrating and differentiating amplifiers.

### 2.1 Inverting Amplifier

1. Calculate the closed loop voltage gain of the circuit, as shown in Fig. 2.1.

2. Simulate the circuit in LTSpice using Operational Amplifier UA741 (OP27 in LTSpice).
   a. Do a Transient Analysis, for setup see LTSpice Guide, Transient Analysis. Ensure to plot both $V_S$ and $V_{out}$ and calculate the Gain
   b. Do an AC Sweep and obtain the Frequency Response for the Transfer Function ($V_{out}/V_{in}$) in dB. For setup see LTSpice Guide, AC Sweep (same as lab 1).

3. From the frequency response, locate the $f_{3dB}$ point, or the amplifier’s Bandwidth.
4. From the frequency response, determine $f_t$. $f_t$ is the unity gain frequency at which the gain = 0dB.

5. In the Lab, implement the circuit in Fig. 2.1. The pin configuration of the UA741 operational amplifier is shown in Fig 2.2; connect the two power supplies as in Fig 2.3, and set $V_S$ to 1 Vp-p sine wave at 1 kHz.

6. Use $R_f=10k\Omega$ and $R_s=1k\Omega$. Measure their values using the RLC meter.
7. Circuit Power Up Sequence: First, turn on the negative supply (–15V), then the positive supply, followed by the Function Generator. Always turn on the DC power supply before applying the AC signal. Always turn off the AC signal before turning off the DC power supply.

8. Observe the input and output waveforms on the oscilloscope (Tektronix–DPO4032)

9. Verify that the input and output waveforms are out of phase.

10. Save a screenshot with the input and output waveforms and relevant parameters to be placed in the lab report.

11. Measure the input and output voltages using a digital multimeter. Determine the amplifier’s Gain.

---

![Inverting Amplifier Diagram](image-url)

**Fig. 2.1 Inverting Amplifier**
2.2 Non-Inverting Amplifier

1. Calculate the closed loop voltage gain of the circuit, shown in Fig. 2.4.
2. Simulate the circuit in LTSpice using Operational Amplifier UA741 (OP27 in LTSpice):
   a. Do a Transient Analysis. Ensure to plot both $V_S$ and $V_{out}$ and calculate the Gain
   b. Do an AC Sweep and obtain the Frequency Response for the Transfer Function ($V_{out}/V_{in}$) in dB.
3. From the frequency response, locate the $f_{3dB}$ point, or the amplifier’s Bandwidth.
4. From the frequency response, determine $f_t$. $f_t$ is the unity gain frequency at which the gain = 0dB.
5. In the Lab, construct the circuit and Adjust $V_S$ to 1 Vpp sinewave at 1 kHz.
6. Use $R_f=10k\Omega$ and $R_S=1k\Omega$. Measure their values using the RLC meter.
7. Observe the output waveform on the oscilloscope (Tektronix–DPO4032).
8. Verify that the input and output waveforms are in phase.
9. Save a screenshot with the input and output waveforms and relevant parameters to be placed in the lab report.
10. Measure the input and output voltages using a digital multimeter (remember, these will be RMS values) and determine the Amplifier Gain.

![Fig. 2.4 Non-inverting Amplifier](image)

2.3 Integrating Amplifier

1. Calculate the Gain of the Amplifier circuit shown in Fig. 2.5.
2. Simulate the circuit in LTSpice, but use a VSIN of 1 Vpp at 1 kHz
   a. Do a Transient Analysis of Vout and Vs.
   b. Determine the relationship between Vs and Vout.
      
      *Hint: Notice the name of the amplifier*
   
   c. Determine the Gain of the Amplifier, (RMS_Vout/RMS_Vs).
      
      NOTE: ctrl + click on the variable name to check the rms value.
3. In the lab, construct the circuit in Fig. 2.5 and adjust $V_s$ to 0.5 Vpp Square Wave at 1 kHz.
4. Use $C_F=1nF$ and $R_S=10k\Omega$. Measure their values using the RLC meter.
5. Observe both the input and output waveforms on the oscilloscope (Tektronix-DPO4032).

6. Save a screenshot with the input and output waveforms and relevant parameters to be placed in the lab report.

7. Measure the input and output voltages using a digital multimeter (remember, these will be RMS values) and determine the Amplifier Gain.

![Integrating Amplifier](image)

Fig. 2.5 Integrating Amplifier

8. What is the relationship between the input and output waveform?

9. For the measured and simulated results, is the ratio RMS(V_{out})/RMS(V_S) similar to what you expect? Explain.
   
   *Hint: Compare the calculated result to the simulation and measurement results.*

2.4 Differentiating Amplifier

1. Calculate the Gain of the Amplifier circuit shown in Fig. 2.6.

2. Simulate the circuit in LTSpice, but use a VSIN of 2 Vpp at 1 kHz
   
   a. Do a Transient Analysis of Vout and Vs.
   
   b. Determine the relationship between Vs and Vout.
c. Determine the Gain of the Amplifier, \( \frac{\text{RMS}_{\text{Vout}}}{\text{RMS}_{\text{Vs}}} \).

3. In the lab, construct the circuit in Fig. 2.6 and **Adjust \( V_S \) to 2 Vpp Triangular Wave at 1 kHz.**

4. Use \( C_S=1\text{nF} \) and \( R_F=10\text{k}\Omega \). Measure their values using the RLC meter.

![Differentiating Amplifier Circuit](image)

Fig. 2.6 Differentiating Amplifier

5. Observe both the input and output waveforms on the oscilloscope (Tektronix–DPO4032).

6. Save a screenshot with the input and output waveforms and relevant parameters to be placed in the lab report.

7. Measure the input and output voltages using a digital multimeter (remember, these will be RMS values) and determine the Amplifier Gain.

8. What is the relationship between the input and output waveform?

9. For the measured and simulated results, is the ratio \( \frac{\text{RMS}(V_{\text{out}})}{\text{RMS}(V_S)} \) similar to what you expect? Explain.

   *Hint: Compare the calculated result to the simulation and lab results*
Post-lab questions:

1. Determine the new values of $R_f$, without changing $R_s$ in the circuits, to achieve a theoretical Gain of 7.5 for the following two cases:
   a. Inverting amplifier
   b. Non-inverting amplifier

2. Use LTSpice to implement and discuss the outcomes of the followings:
   a. Applying a sinusoidal input signal of 4Vpp and 4kHz in the inverting op-amp shown in Fig. 2.1, as a substitute of 1Vpp and 1kHz.
   b. Modifying the capacitor $C_F$ to be 10nF for the integrating op-amp circuit with sinewave, $V_S$, instead of 1nF.

3. Discuss the purpose of connecting the op-amp as shown in the circuit of Fig. 2.7? Explain the function of the circuit.

Hint: Simulate the circuit using LTSpice and notice the relationship between the input and output waveform.
EEE334 Lab#3
PN Junction Diodes and Applications

It is strongly recommended to perform lab experiments when an EEE334 Lab TA is on duty, since only she/he can verify the correctness of your lab data. Other Lab TAs may stamp and sign your lab data sheets but CANNOT GUARANTEE the correctness of your data, and you will then bear full responsibility and penalty for collecting incorrect data.

The objective of this lab is to familiarize the student with basic properties of junction diodes as well as to provide an overview of some important diode applications.

Sections 3.1-3.3 of the lab involve the study of properties of the PN junction diodes and this requires knowledge of EXCEL. These experiments involve a study of the Current-Voltage characteristics of PN junction Zener diodes using a curve tracer (LEADER LTC–905) in conjunction with an oscilloscope (Tektronix DPO-4032) and EXCEL to determine Turn On and Break down Voltages. Also included are the study of the DC and small signal (AC) analysis of a diode.

Sections 3.4-3.10 of the lab involve an overview of some important diode applications. These experiments will enable a student to acquire additional experience in analyzing and evaluating diode circuits and also to observe typical examples of semiconductor diodes in use. The applications include wave shaping, such as rectifier, clamper, and limiter (clipper) circuits. A zener diode voltage regulator and voltage multiplier circuit are illustrated, as well. For these sections, keep in mind that you will need to use lab observations and information about these circuits in your book to describe their function in the Discussion section of the lab report.

This lab contains detailed instructions for equipment set up and LTSpice, and this will not be the case for the remaining labs.
3.1 Current-Voltage characteristics of PN junction diodes

A curve tracer is an instrument used in analyzing the current-voltage characteristics of a diode or transistor. This is achieved by varying the voltage or current applied to the device, while observing the other component. The LTC-905 curve tracer is used in this lab, and it is connected to the DPO-4032 oscilloscope in order to observe and study the I-V curve of different diodes. The connections, component placement, settings for the curve tracer and oscilloscope are described below.

Oscilloscope to Curve tracer Connections:

1. Connect the red (positive) input of Channel 1 of the oscilloscope (corresponding to the X or Horizontal axis) to the red HORIZONTAL output of the curve tracer.
2. Connect the black (negative) input of Channel 1 of the oscilloscope to the black HORIZONTAL output of the curve tracer.
3. Repeat the same connections for Channel 2 (corresponding to the Y or VERTICAL axis) of the oscilloscope and the VERTICAL outputs of the curve tracer as shown in Fig. 3.1.

![Connection diagram](image)

Fig. 3.1 Connection of curve tracer to Oscilloscope and the diode

The following NOTES pertain to the Curve Tracer:

Note 1: The upper SELECTOR switch can be toggled between Transistor, TRANS for a BJT, or FET according to the component that is being analyzed.
**Note 2:** The lower SELECTOR switch can be toggled between port A and port B according to the port into which the component has been inserted.

**Note 3:** Rotating the corresponding knob can vary the base current/gate voltage. The collector current/drain sweep voltage can be similarly varied. This is used to generate the characteristic curves of the component as the voltage and/or current is varied.

**Diode Forward Bias Analysis:**

Document in your data sheet and report the definition of **Turn on voltage**.

**Oscilloscope and Curve Tracer Settings for Forward Bias**

**Oscilloscope (Tektronix DPO 4032):**

1. Preparing the oscilloscope for this experiment:
   a. Turn Oscilloscope **ON**.
   b. If there are measurements appearing at the bottom of the screen, remove them all by doing the following:
      i. Press **Measurement**.
      ii. Press the soft key on the bottom of the screen corresponding to **Remove Measurement**.
      iii. Press the soft key on the right side of the screen corresponding to **Remove All Measurements**.

2. Setting the oscilloscope to display the XY mode:
   a. Press the **ACQUIRE** soft key located in the **Horizontal** knob and key controls.
   b. To set the XY mode **ON**:
      i. Press the soft key corresponding to the **XY Display** appearing at the bottom of the oscilloscope screen.
ii. Press the soft key corresponding to the **Triggered XY** appearing at the right hand side of the screen.

iii. Ensure an XY graph appears covering the half leftmost part of the oscilloscope screen.

iv. **LOOK FOR** a yellow small OR big fuzzy **SPOT** somewhere on the XY screen, most likely at the origin.

3. Set Channel 1 to 100 mV/Div, by turning the **Scale** knob for Channel 1.

4. Set Channel 2 to 100 mV/Div, by turning the **Scale** knob for Channel 2.

5. Invert Channel 2. To perform this:
   a. Press the soft key button for Channel 2.
   b. Press the soft key corresponding to **Invert** on the bottom of the screen and set it to **ON**.

6. Set the **Coupling** for Channels 1 and 2 to **Ground**:
   a. Press Channel 1 soft key.
   b. Press the corresponding soft key for **Coupling** at the bottom of the screen until the chassis **Ground** symbol is selected.
   c. Repeat steps 6a and 6b for Channel 2. **When the yellow SPOT becomes finer and smaller, it may be difficult to see.**

7. Adjust the finer **SPOT, which indicates the origin**, to the **lower left-hand corner** of the oscilloscope screen by using the **Position** knobs. This enables you to use the entire screen to see the Forward Bias I-V curve.

8. After setting the new origin, set Channels 1 and 2 to DC mode by doing the following:
   a. Channel 2 menu should already appear at the bottom of the screen, otherwise press the soft key for Channel 2.
   b. Press the corresponding soft key for **Coupling** until **DC** is selected.
c. Press Channel 1 soft key.

d. Press the corresponding soft key for **Coupling** until **DC** is selected.

   **NOTICE** the finer yellow **SPOT**, now located at the left-hand corner becomes, fuzzier again.

9. Set the Time/div to **4.0 ms** by adjusting the **Scale** knob of the **Horizontal** knob and key controls.

**Curve Tracer Settings:**

1. **COLLECTOR/DRAIN SWEEP VOLTAGE** is set to 10V
2. **POLARITY** is set to **DIODE FORWARD (NPN side)**
3. **SELECTORS:** Upper: **TRANS.** Lower: **A**
4. **BASE CURRENT / GATE VOLTAGE:** Any setting (NA)
5. **CURRENT LIMIT:** SIGNAL
6. **H LENGTH:** Rotate fully clockwise

**Experimental Procedure:**

1. Obtain the Zener diodes 1N5234, 1N5237, and 1N5239. The positive and negative terminals of each diode are as shown in Fig. 3.2 below.

2. Connect the 1N5234 Zener diode between the C (+) and E (−) terminals on the A side of the curve tracer. You may use wires for the Jack area OR if the diode leads are thin, the leads can be inserted into the small holes of the Transistor plug indicated as C (+) and E (−).
3. Set the Power Switch on the Curve Tracer to the **ON** position.

4. Ensure a curve resembling Fig. 3.3 above appears on the oscilloscope screen.
   
   a. **IMPORTANT.** This is an I–V curve, so you must know that the Voltage is on the X axis and the Current on the Y axis.
   
   b. The curve may be very fuzzy and faint.
5. To improve the definition and intensity of the curve do the following
   a. Press the **Intensity** button
   
   On the upper right hand corner, **a** Waveform Intensity and **b** Graticule Intensity percentages readout should appear.
   b. Turn Knob **a** until the Waveform Intensity readout shows **100%**
   c. Press **Acquire**
      i. On the bottom of the screen, press the soft key for **Mode**.
      ii. On the right hand side of the screen, press the soft key for **Hi Res**.
      iii. **Ensure that the curve trace is well defined and brighter.**

6. **Save the Waveform in .CSV format by doing the following:**
   a. Press **Menu** on the bottom of the screen.
   b. Press the **Save Waveform** soft key also on the bottom of the screen.
   c. Press the **Waveform Source** soft key on the right side of the screen.
      
      **Turn Knob **a** to select All Displayed Waveforms.**
   d. Press **To File** soft key again on the right side of the screen.
      i. **Turn Knob **a** to scroll down to your Memory media.**
      ii. **Press the Select button**
         You may also select a desired folder to save to by scrolling and selecting in the same manner just mentioned.
      iii. **Select the .CSV Spread Sheet File Format** by pressing the corresponding soft key on the right side of the screen.
         **IMPORTANT, if the data is not in .CSV format you will not be able to do the required analysis in Excel.**
   e. Press **OK Save All**, an hour glass appears indicating saving data.
7. Ensure the data has been saved.

8. **DO NOT CHANGE THE OSCILLOSCOPE’S SETTINGS AFTER THIS POINT UNTIL THE NEXT SECTION.**

9. Set the Curve Tracer Power Switch to the **OFF** position.

10. Use **Excel**, preferably on your laptop or, in the lab computer to open the data file obtained from the oscilloscope.
   
   a. It contains approximately 10015 rows of data points for TIME, CH1 and CH2!
   
   b. CH1 is the Voltage data and CH2 is the current data
     
     i. **IMPORTANT**, the Current data collected directly from the oscilloscope is in V/cm:
     
     ii. **Before plotting, use the Conversion Table 3.1 to convert the Current to its correct units/div, mA/cm or A/cm.**

11. Plot the data of CH1 (Voltage, X axis) and the converted data of CH2 (Current, Y axis).
   
   a. Use an XY scatter type graph.
   
   b. Label the graph **Diode Characterization-Forward Bias**.
   
   c. Label the axis appropriately including the units i.e. Current in A or mA, Voltage in V, etc.

12. Determine the value of the **Turn on Voltage**, which is the X axis intersection of the tangent line at the **knee** of the curve as shown in Fig. 3.3

   **IMPORTANT**, you need to add a linear TREND LINE at the knee of your plot to find the **Turn on voltage value accurately**

**Note:** The forward bias I–V curve, the understanding and value with proper units and sign of the **Turn on Voltage are THE OBJECTIVES OF THIS SECTION**
Diode Reverse Bias Analysis:

Document in the datasheet and in the report results the definition of Breakdown Voltage

Oscilloscope and Curve Tracer Settings for Reverse Bias

Oscilloscope:

From the Forward bias settings just do the following changes:

a. Use the Position knobs to set the finer yellow SPOT, representing the origin, to the upper Right hand corner of the screen. This will allow the use of the entire screen to see the reverse bias curve.

b. Set Channel 1 to 1 V/cm using the scale knob for Channel 1.

c. Set Channel 2 to 1 V/cm using the scale knob for Channel 2.

Curve Tracer Settings:

From the Forward bias settings just do the following changes:

1. COLLECTOR SWEEP is set to 20V.

2. POLARITY is set to DIODE BACKWARD (PNP side).

Experimental Procedure:

1. With the 1N5234 diode still between the C and E terminals on the A side of the curve tracer, set the Power Switch on the Curve Tracer to the ON position.
2. Ensure a curve resembling Fig. 3.4 appears on the oscilloscope screen.

3. Save the curve in .CSV Spread Sheet File Format following the instructions for saving given in the Forward Bias section.

4. Ensure the data has been saved.

5. Set the Curve Tracer Power Switch to the **OFF** position.

6. On your laptop or lab computer, use **Excel** to open the data file obtained from the oscilloscope.
   a. It contains approximately 10015 rows of data points for **TIME**, **CH1** and **CH2**!
   b. CH1 is the Voltage data and **CH2** is the current data
      i. **IMPORTANT**, the Current data collected directly from the oscilloscope is in mV/cm or V/cm:
      ii. **Before plotting, use the Conversion Table 3.1 to convert the Current to its correct units/div, A/cm or mA/cm.**

7. Plot the data of **CH1** (Voltage, X axis) and the **converted** data of **CH2** (Current, Y axis).
   a. Use an XY scatter type graph.
   b. Label the graph **Diode Characterization—Reverse Bias**.
   c. Label the axis appropriately including the units i.e. Current in A or mA, Voltage in V, etc.

8. Determine the value of the **Breakdown Voltage**, which is the point at the X axis as shown in Fig. 3.4 below, in our case there is no gradual knee but a **sharp bend instead.**

   **Note:** The reverse bias, the understanding and the value with proper units and sign of the **Breakdown Voltage** are **THE OBJECTIVES OF THIS SECTION**
9. **Repeat the above procedure for the remaining diodes, this means DO THE FWD BIAS AND REVERSE BIAS FOR ALL 3 DIODES, but see the next steps first.**

   a. To go back to Forward Bias settings just do the following:
      
      i. On the Oscilloscope:
         
         1. Adjust the finer yellow SPOT back to the lower **Left hand corner** of the screen.
         2. Set Channel 1 and Channel 2 back to 100 mV/cm.

      ii. On the Curve Tracer:
         
         1. Set the POLARITY back to **NPN**.
         2. COLLECTOR SWEEP back to 10.
3. Replace the diode.

4. Switch POWER to ON.

10. Go back up to the instruction mentioning to ensure to see the Forward bias curve on the screen. Repeat the procedure until all three Zener diodes are characterized.

Table 3.1 Conversion Table for Converting VOLTS to AMPS

<table>
<thead>
<tr>
<th>Vertical Y sensitivity on scope</th>
<th>Conversion using LTC-905 curve tracer</th>
</tr>
</thead>
<tbody>
<tr>
<td>50 mV/cm</td>
<td>0.5 mA/cm</td>
</tr>
<tr>
<td>0.1 V/cm</td>
<td>1 mA/cm</td>
</tr>
<tr>
<td>0.2 V/cm</td>
<td>2 mA/cm</td>
</tr>
<tr>
<td>0.5 V/cm</td>
<td>5 mA/cm</td>
</tr>
<tr>
<td>1 V/cm</td>
<td>10 mA/cm</td>
</tr>
<tr>
<td>2 V/cm</td>
<td>20 mA/cm</td>
</tr>
<tr>
<td>3 V/cm</td>
<td>30 mA/cm</td>
</tr>
</tbody>
</table>

3.2 DC analysis of a diode

1. Record the usually assumed $V_d$ for a diode, then calculate $V_{out}$, and $i_d$, the current passing through the diode, in the circuit shown in Fig. 3.5.

2. Construct the circuit:
   a) RLC meter and use $R=1k\Omega$.
   b) Use the 1N4005 (or 1N4003, perform the same in this lab) diode and adjust $V_T$ to 5V.

3. Measure $V_{out}$ across R using the digital multimeter.

4. Measure $i_d$, the current passing in the diode.

5. Measure $V_d$, the voltage drop across diode.
6. Simulate the circuit in LTSpice.
   a. Show the DC op pnt results. Show Vout and id.

7. Obtain the Current – Voltage characteristic of the diode by doing a DC Sweep of the LTSpice circuit:
   a. Go to Run >> DC Sweep, select DC Sweep and enter the following settings:
      i. Sweep Variable Type: Voltage Source
      ii. Name: V1 (DC source name)
      iii. Sweep Type: Linear Sweep
      iv. Start Value: 0
      v. End Value: 5
      vi. Increment: 0.1
      vii. Click O.K. after set up, and then Close.
   b. Plot the voltage across the diode and resistor, plot the current through the diode.

8. Once the trace is visible, measure the relevant parameters, such as the Voltage and Current at 5V in this case.
9. Save a copy of the trace with relevant parameters to be pasted directly onto your lab report LTSpice results.
10. Use % Error to compare the calculated with the measured and the LTSpice results.

3.3 Small signal (AC) analysis of a diode

1. **Calculate** $V_{out}$ assuming $V_d = 0.7V$, these are the DC voltage drops across the resistor and the diode respectively in Fig 3.6.
2. According to the $V_d$ **assumption** from above, calculate $I_d$, the DC current through the diode.
3. Construct the circuit.
   a. Use $R=1k\Omega$. Measure its value using the RLC meter.
   b. Use 1N4005 (or 1N4003, perform the same in this lab) diode.
   c. Adjust $V_T$ to 5V DC.
   d. Set The Function Generator, $V_s$, to High Z.
   e. Set $V_s$, to 1 Vpp sinewave at 1 kHz.
4. Use the oscilloscope to obtain:
   a. The waveform of $V_s$ and $V_{out}$, the AC voltages of the source and the drop across the resistor respectively.
      i. Pay close attention to waveform offsets, if there is any, record it. **What does an offset represent?**
      ii. Place the relevant parameters, such as $V_{pp}$ and $V_{rms}$, of both waveforms on the screenshot.
      iii. Save a bmp format copy of the screenshot including the relevant parameters.
      iv. Calculate the $V_{rms}$ for $V_s$ and $V_{out}$ from the $V_{pp}$ obtained from the oscilloscope.
      v. Are the calculated $V_{rms}$’ values equal to the oscilloscope values? Explain why or why not.
   b. Using the MATH function, obtain $i_d$, the current waveform passing through the diode
Hint: In order to get \( i_d \), obtain the \( V_{\text{out}} \) waveform first, then divide by the actual value of the resistor.

i. Place the relevant parameters, such as \( I_{\text{pp}} \) and \( i(\text{rms}) \), of the waveform on the screenshot.

ii. Save a bmp format copy of the screenshot including the relevant parameters.

c. To obtain the waveform of \( V_d \), the AC voltage drop across diode

i. Set the oscilloscope to 200mV/div and the Channel to DC Coupling

   **Note: At this resolution the waveform for \( V_d \) looks like noise, why?**

ii. Pay close attention to the waveform offset

   i. Measure the offset using the **Mean** in the oscilloscope.

   **ii. What does this offset represent?**

iii. Save a bmp screen shot of \( V_d \) and **relevant parameter** at this resolution.

iv. Set the oscilloscope to 5mV/div and the Channel to AC Coupling in order to observe \( V_d \)

v. Place the relevant parameters, such as \( V_{\text{pp}} \) and \( V_{\text{rms}} \), of the waveform on the screenshot.

vi. Save a bmp format screenshot including the relevant parameters of your results at this resolution.

5. Use the DMM to:

a. Measure and record the DC and AC values of \( V_{\text{out}} \).

b. Use the above measurements to determine the DC and rms values of \( i_d \).

c. Measure and record the DC and rms values of \( V_d \).
6. Simulate the circuit in LTSpice:
   a. Use VDC = 5V.
   b. Signal since wave settings:
      a. VOFF=0, VAMPL=0.5, FREQ=1k
      c. For LTSpice results, obtain a Transient Analysis with relevant parameters for
         Vd, Vout and id.

7. Use % Error to compare calculated, lab results and LTSpice results.

3.4 Half-Wave Rectifier

1. Construct the circuit as shown in Fig. 3.7. Adjust V_s to 15 Vpp sine wave at 1 kHz.
2. Use $R_S = R_L = 1\, \text{k}\Omega$. Measure their values using the RLC meter. Use the 1N4005 (or 1N4003, perform the same in this lab) diode.

3. Using the oscilloscope (Tektronix–DPO4032), obtain the input waveform of $V_S$ and the output waveform, $V_L$, and relevant parameters.

4. Simulate the circuit in LTSpice and compare with the oscilloscope results.

### 3.5 Full-Wave Rectifier

1. Connect the diode bridge rectifier (DB101) and the transformer as shown in Fig. 3.8. Use the transformer with the wooden base.
2. Adjust $V_S$ to 15 Vpp Sine Wave with a frequency of 50 Hz and observe the output waveform.

3. Obtain a screenshot of ONLY the output waveform with relevant parameters.
   
   Note: The input to the bridge rectifier must be connected as shown in Fig. 3.8.

3.6 Peak Detector (Rectifier)

1. Construct the circuit as shown in Fig. 3.9. Adjust $V_S$ to 15 Vpp sine wave at 1 kHz.

2. Use $C_L = 100\text{nF}$ and $R_L = 100\text{k}\Omega$. Measure their values using the RLC meter. Use the 1N4005 (or 1N4003, perform the same in this lab) diode.

3. Using the oscilloscope (Tektronix–DPO4032), obtain the input waveform of $V_S$ and the output waveform, $V_L$ and relevant parameters.

4. Simulate the circuit using LTSpice. Obtain the waveform of $V_S$ and $V_L$
   
   Note: Use signal sine wave for the AC source in the LTSpice schematic. $VOFF=0$, $VAMPL=7.5$, $FREQ=1k$.

5. Are your measured and simulated results matched? Compare the results.

![Fig. 3.9 Peak detector rectifier](image)

3.7 Diode Clamper

1. Construct the circuit as shown in Fig. 3.10. Adjust $V_S$ to 15 Vpp sine wave at 1kHz and $V_R$ to 5 $V_{dc}$.

2. Use $C = 100\text{nF}$. Measure its value using the RLC meter. Use the 1N4005 (or 1N4003, perform the same in this lab) diode.
3. Using the oscilloscope (Tektronix–DPO4032), obtain the input waveform of $V_S$ and the output waveform, $V_L$ and relevant parameters.

4. Simulate the circuit using LTSpice. Obtain the waveform of $V_S$ and $V_L$.

   *Note: Use signal sine for the AC source in the LTSpice schematic. $VOFF=0$, $VAMPL=7.5$, $FREQ=1k$.*

5. Are your measured and simulated results matched?

### 3.8 Diode Limiter (Clipper)

1. Construct the circuit as shown in Fig. 3.11. Adjust $V_S$ to 15 Vpp Sine Wave at 1kHz and $V_R$ to 5 Vdc.

2. Use $R=1k\Omega$. Measure its value using the RLC meter. Use the 1N4005 (or 1N4003, perform the same in this lab) diode.

3. Using oscilloscope (Tektronix–DPO4032), obtain the input waveform of $V_S$ and the output waveform, $V_L$.

4. Also, use the Tektronix DPO4032 oscilloscope to obtain the Voltage transfer characteristic $V_L$ versus $V_S$.

   Set the oscilloscope to XY mode as in lab 3 but do not invert channel 2.

   You may leave the Yellow SPOT at the center of the graph and set the Volts/div according to the voltage in this case.
5. Simulate the circuit using LTSpice and obtain the waveforms of $V_S$, $V_L$ (transient), and voltage transfer characteristics (Plot $V_S$ as your x-axis and $V_L$ as your y-axis, i.e. DC sweep or can change the axis setting).

*Note: Use signal sine for the AC source in the LTSpice schematic. $VOFF=0$, $VAMPL=7.5$, $FREQ=1k$.*

6. Are your measured and simulated results matched?

### 3.9 Zener Diode Voltage Regulator

1. Construct the circuit as shown in Fig. 3.12. Adjust $V_S$ to 5 Vpp sinewave at 1 kHz.
2. Use $R_S = 560\Omega$ and $R_L = 1k\Omega$. Measure their values using the RLC meter. Use the 1N5237 Zener diode.
3. Using the oscilloscope (Tektronix–DPO4032), obtain the input waveform of $V_S$ and the output waveform, $V_L$. 
4. Simulate the circuit using LTSPICE. Obtain the waveform of $V_S$ and $V_L$.
   
   *Note: Use signal sine for the AC source in the SPICE schematic. $VOFF=0$, $VAMPL=2.5$, $FREQ=1k$.*

7. Are your measured and simulated results matched?

### 3.10 Voltage Multiplier circuit

The circuit in Fig. 3.13 is a voltage multiplier. The circuit operation is similar to that of a full wave rectifier, when the capacitor voltages are superimposed.

**Procedure:**

1. Construct the circuit as shown in Fig. 3.13. Adjust $V_S$ to 20 Vpp Sine Wave at 1kHz.
2. Use $C_1=C_2=1000\mu$F, $R = 10k\Omega$. Measure its value using the RLC meter. Use the 1N4005 (or 1N4003, perform the same in this lab) diode.
   
   *Note: Pay attention to the capacitors polarities while connecting them.*

3. Using the oscilloscope (Tektronix–DPO4032), obtain ONLY the output waveform, $V_O$ with relevant parameters.
4. Simulate the circuit using LTSpice. Obtain the waveform of both $V_S$ and $V_O$.

*Note: Use signal sine for the AC source in the LTSpice schematic. VOFF=0, VAMPL=10, FREQ=1k.*

5. Determine the multiplication factor (MF) for measured and simulated results?

$$\text{The multiplication factor (MF)} = \frac{V_O\ \text{rms}}{V_S\ \text{rms}}$$

(3.1)

*Note: The input voltage is an AC voltage, while the output voltage is DC.*

6. Are your measured and simulated results matched?

**Post-lab questions:**

1. Explain the function of the Curve Tracer in this experiment

Use LTSpice to answer the following post lab questions (show LTSpice schematic and result graphs in your lab report):

2. In Fig. 3.5 (DC analysis of a diode), what would be $I_d$ if:
a. The resistor, R, was shunted (parallel) with a resistor, $R_{\text{shunt}}$, of equal value (1kΩ).

b. The resistor R was connected with another 1kΩ resistor in series.

c. The diode, D, were shunted (parallel) with a diode, $D_{\text{shunt}}$ (assumed to be matched).

3. In Fig. 3.6 (Small signal (AC) analysis of a diode), what would happen if the polarity of the DC voltage source were swapped?

4. What would happen if a capacitor $C=1\mu F$ were added (in parallel with R) to the diode circuit shown in Fig. 3.6?

5. Use LTSpice to implement and verify what would happen if the capacitor were increased ten times in the Peak Detector Rectifier? Explain.

6. Draw a circuit diagram to represent DB101 chip, as shown in Fig. 3.8. Make sure that you label the input and output voltage in your drawing.

   Note: You do not have to consider the transformer diagram in your drawing.

7. Discuss the operation of the voltage multiplier circuit.

8. What would happen if the capacitors ($C_1$ and $C_2$) were $1\mu F$ in the voltage multiplier circuit? Explain.
EEE334 Lab#4
MOS Characterizations

It is strongly recommended to perform lab experiments when an EEE334 Lab TA is on duty, since only she/he can verify the correctness of your lab data. Other Lab TAs may stamp and sign your lab data sheets but CANNOT GUARANTEE the correctness of your data, and you will then bear full responsibility and penalty for collecting incorrect data.

This experiment involves characterization of a Metal Oxide Semiconductor Field Effect Transistor (MOSFET). The student will be able to understand MOSFET operation, to measure the Current-Voltage characteristics and to determine the N-channel MOSFET (NMOS) parameters. Also, NMOS amplification will be illustrated.

Precaution when handling MOS devices
MOS transistors are easily damaged by Electro Static Discharge (ESD), thus you must wear a Ground Strap to rid yourself of ESD and be able to handle MOS devices without damaging them.

A CD4007 MOSFET array will be used in all remaining labs. It contains three NMOS and three PMOS (P-channel MOSFET) transistors as shown in Fig. 4.1. When employing this array, the substrate of the NMOS (bulk connection) is connected to Pin 7 and should be connected to the most negative voltage of the circuit, V_SS (or it can be shorted to the source node only if one NMOS transistor on the array is used). Pin 14 is the bulk of the PMOS transistor and should be connected to the most positive voltage in the circuit, V_DD (or a capacitor can be placed if one PMOS transistor on the array is used).
Fig. 4.1 CD4007 array

Fig. 4.2 Circuit symbol for NMOS and PMOS
Important note
The CD4007 MOSFET array should not be connected to circuits with power ON since high transient voltages may cause permanent damage. Therefore, to avoid seriously damaging the device, do not apply input signals until PIN 7 and 14 have been properly terminated.

4.1 Determination of \( V_t \) and \( k_n'W/L \) of NMOS
The drain current can be computed as a function of the gate voltage when the transistor is in saturation, using following equation:

\[
\sqrt{I_D} = \sqrt{\frac{1}{2} k_n' \frac{W}{L}} \left( V_{G} - V_t \right) \quad V_G > V_t \quad (4.1)
\]

Where, \( V_t \) is the threshold voltage and \( (k_n'W/L) \) is the Transconductance Parameter.

Eqn (4.1) is indeed linear between \( \sqrt{I_D} \) and \( V_G \), where the intersection with the horizontal x-axis gives the threshold voltage \( V_t \), and the value of the slope, whose value corresponds to,

\[
\text{s l o p e} = \frac{1}{\sqrt{2k_n' \frac{W}{L}}} \quad (4.2)
\]

Hint: Use an appropriate software program, such as MICROSOFT EXCEL to record the experimental data and then plot the curve. The slope will be determined easily.

Procedure:
1. Construct the circuit shown in Fig. 4.3. Use any of the three NMOS transistors of the CD4007 array. The circuit shown in Fig. 4.3 assumes that you use the NMOS transistor between the pins 3, 4 and 5.

Note: Use the “KEITHLEY” digital multimeter as the ammeter in your circuit.
2. Connect the substrate pin 7 to the ground (bulk is connected to ground). Place a 1μF capacitor between substrate pin 14 and the ground.

*Note: The NMOS transistor is always in the saturation since $V_{GS} = V_{DS}$.*

3. Sweep the gate voltage ($V_{GS} = V_{DS}$) from 0 to 8V in steps of 0.5V and record the corresponding drain current $i_D$.

4. Plot $\sqrt{i_D}$ against $V_{GS}$. Find the threshold voltage $V_t$ and the transconductance parameter ($k_nW/L$).

*Hint: When determining the slope, you should omit the values at which $\sqrt{i_D} = 0$.*

### 4.2 Determination of $r_o$ and $\lambda$ of NMOS

If the Channel Length Modulation, $\lambda$, is taken into account, for a MOSFET device in saturation the drain current $i_D$ can be written as:

$$i_D = \frac{I}{2} k_\alpha \frac{W}{L} (v_G - V_t)^2 (1 + \lambda v_D)$$  \hspace{1cm} (4.3)$$

where $\lambda$ is the Channel Length Modulation.

The output resistance $r_o$ in saturation ($V_{DS} > V_{GS} - V_T$ and $V_{GS} > V_T$) can be written as:

$$r_o = \frac{1}{I_D \lambda} = \frac{|V_A|}{I_D}$$  \hspace{1cm} (4.4)$$
The output resistance, $r_o$, is basically the inverse of the slope of the $i_D$ vs. $V_{DS}$, which be defined also as,

$$r_o = \left[ \frac{\partial i_D}{\partial V_D} \right]^{-1}$$

(4.5)

when $V_{GS}$ is constant.

The early voltage, $V_A$, is found from the intersection with the horizontal x-axis and the channel length modulation $\lambda$ can be defined from Equation (4.4) as:

$$\lambda = \left| \frac{1}{V_A} \right|$$

(4.6)

*Hint: Use an appropriate software program, such as MICROSOFT EXCEL to record the experimental data and then plot the curve. The slope will be determined easily.*

**Procedure:**

1. Construct the circuit shown in Fig. 4.4.
   
   *Note: Make sure that you use the same NMOS transistor as you used for the previous experiment*
   
   *Note: Use the “KEITHLEY” digital multimeter as the ammeter in your circuit.*

2. Connect the substrate pin 7 to the ground (bulk is connected to ground). Place a 1μF capacitor between substrate pin 14 and the ground.

3. Keep the gate voltage constant $V_{GS}$ at 3V and measure the drain current $i_D$ while varying the drain voltage. Sweep the drain voltage $V_{DS}$ from 0 to 7V in steps of 0.5V and record the corresponding drain current $i_D$.

4. Repeat step 3 when gate voltage constant $V_{GS}$ at 5V.

5. Plot $i_D$ against $V_{DS}$ (two curves in a graph). Determine the output resistance $r_o$ for both curves (at $V_{GS} = 3V$ and 5V). Find the corresponding value of $\lambda$ and $V_A$.
   
   *Note: Make sure that the slope is taken in the saturation region.*
   
   *Note: Since you have two values of the $\lambda$ and $V_A$ from the two curves, take the average value. $\lambda$ is a positive value for NMOS transistor, while $V_A$ is a negative value, which can be observed from $i_D$ vs $V_{DS}$ curves.*
4.3 Determination of $g_m$ of the NMOS transistor

$g_m$ is the Transconductance Factor of the MOS transistor and it is one of the parameters of the small-signal equivalent circuits. The goal is to measure the Transconductance of an NMOS transistor in saturation ($V_{DS} > V_{GS}-V_T$ and $V_{GS} > V_T$). The $g_m$ in saturation can be written as:

$$g_m = \left[ \frac{\partial i_D}{\partial V_G} \right]_{Q_S} = \sqrt{2I_Dk_n \frac{W}{L}} = \frac{2I_D}{V_G - V_T}$$  \hspace{1cm} (4.7)

The Average Transconductance $g_m$ is basically the slope of the $i_D$ vs. $V_{GS}$, when $V_{DS}$ is constant and MOS device is in saturation region. The slope is clearly different at different $I_D$.

*Hint: Use an appropriate software program, such as MICROSOFT EXCEL to record the experimental data and then plot the curve. The slope will be determined easily.*

**Procedure:**

1. Construct the circuit shown in Fig. 4.4. The same as in section 4.2.
Note: Make sure that you use the same NMOS transistor as you used for the previous experiment.

Note: Use the “KEITHLEY” digital multimeter as the ammeter in your circuit.

2. Connect the substrate pin 7 to the ground (bulk is connected to ground). Place a 1μF capacitor between substrate pin 14 and the ground.

3. Keep the drain voltage, \( V_{DS} \), constant at 10V and measure the drain current, \( i_D \), while varying the gate voltage. Sweep the gate voltage \( V_{GS} \) from 2 to 7V in steps of 0.5V.

   Note: Every time you sweep the \( V_{GS} \), a new value of Transconductance \( g_m \) is obtained by simply using eqn. (4.7).

4. Plot the curve, \( i_D \) vs. \( V_{GS} \), on a graph. Determine the average Transconductance \( g_m \).

### 4.4 Large signal operation: The transfer characteristics

Fig. 4.5 shows basic the structure of a common source amplifier. The objective here is to determine the transfer characteristic gain of the MOS amplifier circuits. The slope is basically the amplification gain of the amplifier, or the ratio of the output to the input voltage, in the saturation area.

Note: It could be useful if you take a look at Fig. 5.29 of the textbook (Sedra-Smith, 6th edition) before starting the experiment.

Hint: Use an appropriate software program, such as MICROSOFT EXCEL to record the experimental data and then plot the curve. The slope will be determined easily.

**Procedure:**

1. Construct the circuit shown in Fig. 4.5. Use any of the three NMOS transistors of the CD4007 array. The circuit shown in Fig. 4.5 assumes that you use the NMOS transistor between the pins 3, 4 and 5.

2. Connect the substrate pin 7 to the ground (bulk is connected to ground). Place a 1μF capacitor between substrate pin 14 and the ground.

3. Adjust \( V_{DD} \) to 15V. Use \( R_D=10k\Omega \). Measure its value using the RLC meter.
Fig. 4.5 Common source amplifier circuit

4. Vary the gate voltage $V_{GS}$ from 0 to 7V in steps of 0.5V and record the corresponding drain voltage $V_{DS}$.

   Note: You will notice that the output will change quickly at one point (specifically in the saturation region). When it happens, vary the input (gate) voltage in small steps, e.g. 0.2 or 0.25 V, in order to record the output (drain) voltage accurately and then the transfer characteristic curve.

5. Plot the output voltage (drain voltage $V_{DS}$) against the input voltage (gate voltage $V_{GS}$). Find the slope of the transfer characteristic around the point Q in the saturation area.

   Note: See Section 5.4.3 of the textbook (Sedra-Smith, 6th edition) where the point Q is defined. See also Fig. 5.29.

6. Simulate the circuit using LTSPICE.

   Note: Use NMOS4 for NMOS transistor which can be found from the parts list.
   Note: For LTSPICE (simulation), 1μF capacitor between substrate pin 14 and the ground is NOT needed. It is required ONLY for building a circuit and experimental measurements in the lab.
Important Note

In LTspice use Parts NMOS4 for the NMOS and PMOS4 for the PMOS. Before simulating NMOS or PMOS circuits, change the LTspice device parameters to reflect CD4007 NMOS or PMOS.

To run simulations of MOSFETs we need to at least set the values of parameters L (channel length), W (channel width), VT0 (zero-bias threshold voltage), KP (transconductance, μn/pCox), and LAMBDA (channel-length modulation coefficient, λ).

In LT Spice, these parameters can be specified by inserting the model into the schematic. Go to “Edit” on the menu bar and choose “Spice Directive”. In the pop-up window, type-in:

```plaintext
.MODEL TestN NMOS (KP=111u VT0=1.2 LAMBDA=0.01)
```

in the dialogue box to set KP, VT0, and LAMBDA of the NMOS transistor that we will use for this tutorial (as shown in Fig. 1).

**Note:** The above values are generic, so update the NMOS4 values by the Kp (this is Kn), Vt0 (this is Vt) and Lambda (this is λ) according to your measured NMOS parameters.

Click “OK” and place the model sentence onto the schematic. And similarly, for PMOS.

**Note:** For PMOS use:

```plaintext
.MODEL TestP PMOS (KP=55u VT0= -1.0 LAMBDA=0.04)
```

Notice that TestN and TestP are the model names and can be any name you give to the transistor model. The other names or variables in the .MODEL are standard and cannot be renamed.

The MOSFET we need to use in our simulation should be “nmos4” and “pmos4”. Then we need to set the values of each component. First, we set the NMOS transistor. Hold the “Ctrl” key and right-click on the NMOS symbol, then the “Component Attribute
Editor” window will pop-up. Double click on the value of “SpiceModel” and type in the name of our self-defined NMOS transistor, “TestN”. Click “OK” to confirm. Then, WITHOUT holding the “Ctrl” key, right click on the NMOS model. In the pop-up window, input the Length (L) and Width (W) values as \( W=30\mu m \), \( L=10\mu m \). Note that the bottom dialogue box shows “TestN” results, which means we have successfully set the NMOS model to our self-defined model, TestN and we have set the channel length and width to 10um and 30um, respectively.

NOTE: for PMOS use \( W=60\mu m \), \( L=10\mu m \)

Note: A more detailed setting procedure can be found:

Ask the lab TA for help if you have any question regrading the settings.

Reference: PSPICE setting (not for LTSPICE)
* .model N NMOS (Level=1 Gamma= 0 Xj=0 Tox=1200n Phi=.6 + Rs=0 Kp=111u Vto=1.2 Lambda=0.01 Rd=0 Cbd=2.0p Cbs=2.0p Pb=.8 Cgso=0.1p + Cgdo=0.1p Is=16.64p N=1 W=30u L=10u)
* .model P PMOS (Level=1 Gamma= 0 Xj=0 Tox=1200n Phi=.6 + Rs=0 Kp=55u Vto=-1.0 Lambda=0.04 Rd=0 Cbd=4.0p Cbs=4.0p Pb=.8 Cgso=0.2p + Cgdo=0.2p Is=16.64p N=1 W=60u L=10u)
* 7. Activate the DC Sweep by pressing the DC sweep button, and enter the following:

Sweep Variable Type: Voltage Source
Post-lab questions:

1. Discuss the impact of the temperature on the characteristics of NMOS transistors.

2. In the triode region of NMOS, the drain current can be written approximately as follows:

   \[ i_D \approx k_n \frac{W}{L} (V_G - V_{th}) V_D, \quad v_D < (V_G < V_{th}) \]  

   The resistance of the NMOS transistor \( r_{DS} \) can be defined as:

   \[ r_D \approx \frac{v_D}{i_D} \approx \frac{I}{k_n \frac{W}{L} (V_G - V_{th})} \]  

   a) Using your measured value of \( V_t \) and \( k_n \frac{W}{L} \), plot \( r_{DS} \) against \( V_{GS} \). Vary \( V_{GS} \) from 3 to 10V in steps of 1V. Comment on the relationship between \( r_{DS} \) and \( V_{GS} \).

   b) In the triode region, does the NMOS transistor act as a voltage-controlled resistor, provided that \( V_{GS} > V_t \)? Explain.

3. What would happen if the drain resistor \( R_D \) were 100kΩ in the common source amplifier circuit, shown in Fig. 4.5? Explain.

4. Propose a procedure to measure the threshold voltage, \( V_{th} \), and the transconductance parameter, \( k_p \) (W/L) for a PMOS transistor. Draw a circuit diagram illustrating your proposal experiment.

   *Hint: Note carefully the difference between NMOS and PMOS transistors, with respect to the relationship between \( V_{GS} \) and \( V_t \).*
Important note

You will DEFINITELY need to retain and keep your CMOS chip CD4007 along with all of its measurements and parameters obtained from lab experiment 5 in order to do lab experiments 4, 5 and 6. Protect your CD4007 chip from damage or loss.
EEE334 Lab#5
Single Stage MOS Amplifiers

It is strongly recommended to perform lab experiments when an EEE334 Lab TA is on duty, since only she/he can verify the correctness of your lab data. Other Lab TAs may stamp and sign your lab data sheets but CANNOT GUARANTEE the correctness of your data, and you will then bear full responsibility and penalty for collecting incorrect data.

This experiment involves the testing of three single stage amplifiers, such as the Common–Source (CS), Common–Gate (CG), and Common–Drain (CD) amplifiers using NMOS transistor. The main objectives can be summarized as follows:

1. To carry out an investigation of DC biasing of an NMOS transistor.
2. To illustrate testing of an NMOS Common–Source (CS) amplifier to measure the overall gain.
3. To show testing of an NMOS Common–Gate (CG) amplifier to measure the overall gain.
4. To demonstrate testing of an NMOS Common–Drain (CD) (Source–Follower) amplifier to measure the overall gain.

Precaution when handling MOS devices
MOS transistors are easily damaged by Electro Static Discharge (ESD), thus you must wear a Ground Strap to rid yourself of ESD and be able to handle MOS devices without damaging them.

As in lab 4, a CD4007 MOSFET array will be used. It contains three NMOS and three PMOS transistors as shown in Fig. 5.1. Remember when employing this array, the substrate of the NMOS (bulk connection) is connected to Pin 7 and should be connected to the most negative voltage of the circuit, $V_{SS}$ (or it can be shorted to the source node...
only if one NMOS transistor on the array is used). Pin 14 is the bulk of the PMOS transistor and should be connected to the most positive voltage in the circuit, $V_{DD}$ (or a capacitor can be placed if one PMOS transistor on the array is used).

Fig. 5.1 CD4007 array

Fig. 5.2 Circuit symbol for NMOS and PMOS

**Important note**
The CD4007 MOSFET array should not be connected to circuits with power ON since high transient voltages may cause permanent damage. Therefore, to avoid seriously damaging the device, do not apply input signals until PIN 7 and 14 have been properly terminated.

5.1 DC biasing of an NMOS transistor

Fig. 5.3 shows an NMOS Biasing circuit. The goal of this experiment is to correctly bias the NMOS transistor.

\[ \text{Fig. 5.3 DC biasing circuit} \]

Procedure:
1. Calculate the value of $R_{G2}$, so that the DC value of the $V_{DS}$ and $I_D$ are 10V, 0.6mA, respectively.

_Hint: Use following equation to determine $V_G$:_

$$
V_G = V_S + V_G = \frac{3}{2} V_S + V_t + \frac{2I_D}{k_n W} \sqrt{\frac{W}{L}}
$$

(5.1)

2. Construct the circuit shown in Fig. 5.3.
   a. TO AVOID DAMAGING the characterized NMOS in lab 4:
      i. Use a DUMMY CD4007 MOSFET to ensure the circuit will be biased properly first.
   b. Connect the substrate pin 7 to the source node.
   c. Place a 1$\mu$F capacitor between substrate pin 14 and the ground.
      i. Pay attention to the capacitor polarities, wrong polarities may cause MOSFET damage or circuit malfunction.
   d. For biasing resistor $R_{G2}$, use a potentiometer “resistance box”
      i. Set $R_{G2}$ to its calculated value in step 1.
   e. Use $R_{G1}=120k\Omega$, $R_S=5.1k\Omega$ and $R_D=3.3k\Omega$. Measure their values using the RLC meter.
   f. Place a multimeter between pins 4 and 5 to monitor and verify that $V_{ds} = 10V$. This is to ensure the circuit is properly biased at all times.
   g. Ensure the circuit is built correctly.
   h. Adjust $V_{DD}$ to 15 V and apply power to the circuit.
   i. Verify that $V_{ds}$ is approximately 10V, if so, the circuit is properly biased.
   j. Turn power off and replaced the DUMMY CD4007 MOSFET by the MOSFET used to characterize the NMOS in lab 4.
   k. Turn the power back on and verify that the circuit remains properly biased.
      i. If $V_{ds}$ is close to 10V, do minor adjustments to $R_{G2}$ until the required $V_{ds}$ is achieved.
Record the adjusted $R_{G2}$.

3. Measure and record the $V_{RD}$ and $V_{RS}$.

4. Calculate the corresponding currents $I_D$ and $I_S$.

5. Remove power from the circuit.

6. Simulate the circuit in LTSPICE to verify the operating point (.op).

7. Print schematic with relevant values or paste directly onto results part of lab report.

8. Use % error to compare calculated, lab and LTSPICE results.

### 5.2 Common-Source (CS) amplifier

1. Calculate the gain of the CS amplifier given in Fig. 5.4.

2. Determine the input ($R_{in}$) and output ($R_{out}$) resistances of the CS amplifier.

3. Modify the constructed circuit of Fig. 5.3 by adding the capacitors $C_{C1}$, $C_{C2}$, and $C_S$ as shown in Fig 5.4. The remaining parameters are the same.

   *Note: Pay attention to the capacitors polarities, wrong polarities can cause MOSFET damage or circuit malfunction.*

4. Use $C_{C1}=C_{C2}=C_S=1\mu F$. Measure their values using the RLC meter.

5. $R_L$ represents the internal resistance of the Oscilloscope and is about 1 MΩ.

6. $R_{sig}$ represents the internal resistance of the Function Generator and is about 50Ω, however, do set the Function Generator to High Z.

7. Turn on the DC supply ($V_{DD}$).

8. Set the Function Generator to 100mVpp and 10 kHz frequency, then connect it to the circuit.

---

**Important Notes**

Always check the circuit carefully before applying power.

Always turn ON the DC power supply before applying the AC signal.

Always turn off the AC signal before turning off the DC power supply.
9. Use the oscilloscope to:
   
   i. Obtain the waveform of $V_s$ and $V_o$.
   
   ii. Determine the phase shift of $V_o$ with respect to $V_s$.
   
   iii. Place the relevant parameters, such as $V_{pp}$ and $V_{rms}$, of both waveforms on the screenshot.
   
   iv. Save a bmp format copy of the screenshot including the relevant parameters.
   
   vii. Calculate the Gain of the CS amplifier with this result.
10. Remove all power from the circuit.

11. Simulate the circuit in LTSpice.

   **LTSpice Setup:**
   
i. For the AC and transient signal source (sine wave), set as follows:
      \[ \text{VOFF}=0, \text{AC}=0.05, \text{VAMPL}=0.05, \text{FREQ}=10k \]
   
   ii. Place 1Meg for \( R_L \) which represents the internal resistance of the Oscilloscope.
   
   iii. Set the NMOS model parameters as in lab 4, section 4.4.

   a. **Transient Analysis:**
      
i. Obtain waveforms for \( V_S \text{ a}=(\text{signal source}) \) and \( V_o \text{ (output voltage, VL)} \)
         including the relevant measurements.
      
      ii. From the obtained values calculate the Gain of the CS amplifier.
      
      iii. Print or paste plots directly onto the result section of the lab report.

   b. **AC Analysis:**
      
i. Obtain the frequency response for \( V_o/V_s \text{ in dB} \).
      
      ii. Determine the maximum Gain(dB), \( f_L, f_H \) and BW.
          
          Note:
          
          \( f_L \) and \( f_H \) are the frequencies at which the gain drops by 3dB below its mid-band.

          \[ \text{BW}=f_H-f_L \]
          
          and it defines the amplifier bandwidth.
      
      iii. Include all the above relevant measurements on the plot.
      
      iv. Print or paste the plot directly onto the result section of the lab report.

12. Use % error to compare calculated, lab and LTSPICE results

### 5.3 Common-Gate (CG) amplifier

1. Calculate the gain of the CG amplifier given in Fig. 5.5.

2. Determine the input (\( R_{in} \)) and output (\( R_{out} \)) resistances of the CG amplifier.

   *Note: Pay attention to the capacitors polarities, wrong polarities can cause MOSFET damage or circuit malfunction.*
Fig. 5.5 Common-Gate (CG) amplifier circuit

3. Modify the circuit of Fig. 5.3 by adding the capacitors $C_{C1}$, $C_{C2}$, and $C_S$ as shown in Fig 5.5. The remaining parameters are the same.
4. Use $C_{C1}=C_{C2}=C_S=1 \mu F$. Measure their values using the RLC meter.
5. $R_L$ represents the internal resistance of the Oscilloscope and is about $1 \text{ M}\Omega$.
6. $R_{\text{sig}}$ represents the internal resistance of the Function Generator and is about $50\Omega$, however, do set the Function Generator to High Z.

**Important Notes**

Always check the circuit carefully before applying power.
Always turn ON the DC power supply before applying the AC signal.
Always turn off the AC signal before turning off the DC power supply.

7. Turn on the DC supply ($V_{DD}$).
8. Set the Function Generator to 100mVpp and 10 kHz frequency, and then connect it to the circuit.
9. Use the oscilloscope to:
   i. Obtain the waveform of $V_s$ and $V_o$.
   ii. Determine the phase shift of $V_o$ with respect to $V_s$.
   iii. Place the relevant parameters, such as $V_{pp}$ and $V_{rms}$, of both waveforms on the screenshot.
   iv. Save a bmp format copy of the screenshot including the relevant parameters.
   v. Calculate the Gain of the CG amplifier with this result.
10. Remove all power from the circuit.
11. Simulate the circuit in LTSPICE.
    Setup:
    i. Use signal sine wave to simulate the AC source, set as follows:
       $VOFF=0$, $AC=0.05$, $VAMPL=0.05$, $FREQ=10k$
    ii. Place 1Meg for $R_L$ which represents the internal resistance of the Oscilloscope
    iii. Set the NMOS model parameters as in lab 4, section 4.4
    a. Transient Analysis:
       i. Obtain waveforms for $V_s$ and $V_o$ including the relevant measurements.
       ii. From the obtained values calculate the Gain of the CS amplifier.
       iii. Print or paste plots directly onto the result section of the lab report.
    b. AC Analysis:
       i. Obtain the frequency response for $V_o/V_s$ in dB.
       ii. Determine the maximum Gain(dB), $f_{1}$, $f_{4}$ and BW.
    Note:
\(f_L\) and \(f_H\) are the frequencies at which the gain drops by 3dB below its mid-band.

\(BW = f_H - f_L\) and it defines the amplifier bandwidth.

iii. Include all the above relevant measurements on the plot.

iv. Paste the plot directly onto the result section of the lab report.

12. Use \% error to compare calculated, lab and LTSPICE results.

5.4 Common-Drain (CD) amplifier (Source-Follower)

1. Calculate the gain of the CD amplifier given in Fig. 5.6.

2. Determine the input (R_{in}) and output (R_{out}) resistances of the CD amplifier.

   Note: Pay attention to the capacitors polarities, wrong polarities can cause MOSFET damage or circuit malfunction.

3. Modify the circuit of Fig. 5.3 by adding the capacitors \(C_{C1}\), \(C_{C2}\), and \(C_S\) as shown in Fig 5.6. The remaining parameters are the same.

4. Use \(C_{C1}=C_{C2}=C_S=1\mu F\). Measure their values using the RLC meter.

5. \(R_L\) represents the internal resistance of the Oscilloscope and is about 1 M\(\Omega\).

6. \(R_{sig}\) represents the internal resistance of the Function Generator and is about 50\(\Omega\); however, do set the Function Generator to High Z.

### Important Notes

- Always check the circuit carefully before applying power.
- Always turn ON the DC power supply before applying the AC signal.
- Always turn off the AC signal before turning off the DC power supply.

7. Turn on the DC supply (\(V_{DD}\)).

8. Set the Function Generator to 0.5Vpp and 10 kHz frequency, and then connect it to the circuit.

9. Use the oscilloscope to:
   a. Obtain the waveform of \(Vs\) and \(Vo\).
   b. Determine the phase shift of \(Vo\) with respect to \(Vs\).
c. Place the relevant parameters, such as Vpp and Vrms, of both waveforms on the screenshot.

d. Save a bmp format copy of the screenshot including the relevant parameters.

e. Calculate the Gain of the CG amplifier with these results.

10. Remove all power from the circuit.

11. Simulate the circuit in LTSPICE

   a. Setup:

       i. Use signal sine wave to simulate the AC source, set as follows:
           
           \[ \text{VOFF}=0, \text{AC}=0.25, \text{VAMPL}=0.25, \text{FREQ}=10\text{k} \]

       ii. Place 1Meg for \( R_L \) which represents the internal resistance of the Oscilloscope.

       iii. Set the NMOS model parameters as in lab 4, section 4.4.
b. Transient Analysis:
   i. Obtain waveforms for $V_S$ and $V_o$ including the relevant measurements
   ii. From the obtained values calculate the Gain of the CS amplifier.
   iii. Print or paste plots directly onto the result section of the lab report.

c. AC Analysis:
   i. Obtain the frequency response for $V_o/V_s$ in dB.
   ii. Determine the maximum Gain(dB), $f_L$, $f_H$ and BW.
       Note:
       $f_L$ and $f_H$ are the frequencies at which the gain drops by 3dB below its mid-band.
       $BW = f_H - f_L$ and it defines the amplifier bandwidth.
   iii. Include all the above relevant measurements on the plot.
   iv. Print or paste the plot directly onto the result section of the lab report.

12. Use % error to compare calculated, lab and LTSPICE results.

Post-lab questions:

1. Use LTSpice to observe and justify the outcomes of the followings (include all schematics and graphs in your lab reports):
   a. Doubling the size of the NMOS transistor in the DC biasing circuit, shown in Fig. 5.3.
      *Hint: Doubling the size of the NMOS transistor means the ratio of (W/L) is doubled, i.e., (W/L) is 6 instead of 3.*
   b. Having a load resistor ($R_L=3k\Omega$) in the CS, CG, and CD amplifiers shown in Figs. 5.4, 5.5 and 5.6.
      *Hint: $R_L$ was 1Meg\(\Omega\), which represents the oscilloscope probe or digital meter internal resistance.*
   c. Placing capacitors, $C_{C1}$, $C_{C2}$, and $C_S$, as 1000\(\mu\)F in the CS amplifier shown in Fig. 5.4, as a substitute of 1\(\mu\)F.
d. Removing the bypass capacitor $C_S$ in the CS amplifier shown in Fig. 5.4.

e. Adjusting $I_D$ (DC value), as 1mA in the CD amplifier shown in Fig. 5.6, as a replacement for 0.6mA.

2. Based on your results, discuss the main differences between the Common-Source (CS) amplifier, Common-Drain (CD) amplifier, and Common-Gate (CG) amplifier.

---

**Important note**

You will DEFINITELY need to retain your CD4007 MOSFET along with all of its measurements and parameters obtained from lab experiment 5 in order to continue to lab experiments 7 and 8, thus protect your CD4007 chip from damage or loss.
EEE334 Lab#6
Introduction to Digital Circuits

It is strongly recommended to perform lab experiments when an EEE334 Lab TA is on duty, since only she/he can verify the correctness of your lab data. Other Lab TAs may stamp and sign your lab data sheets but CANNOT GUARANTEE the correctness of your data, and you will then bear full responsibility and penalty for collecting incorrect data.

This experiment introduces the concept of some fundamental logic gates and their dynamic characteristics. You are required to understand the basic structure of the CMOS logic.

The main objectives can be summarized as follows:

1. To study the characteristics of the simple CMOS inverter.
2. To investigate the characteristics of the simple CMOS NAND and NOR gates.

Precaution when handling MOS devices
MOS transistors are easily damaged by Electro Static Discharge (ESD), thus you must wear a Ground Strap to rid yourself of ESD and be able to handle MOS devices without damaging them.

A CD4007 MOSFET array will be used. It contains three NMOS and three PMOS transistors as shown in Fig. 6.1. Remember when employing this array, the substrate of the NMOS (bulk connection) is connected to Pin 7 and should be connected to the most negative voltage of the circuit $V_{SS}$ (or it can be shorted to the source node only if one NMOS transistor on the array is used). Pin 14 is the bulk of the PMOS transistor and should be connected to the most positive voltage in the circuit, $V_{DD}$ (or a capacitor can be placed if one PMOS transistor on the array is used).
Fig. 6.1 CD4007 array

Fig. 6.2 Circuit symbol for NMOS and PMOS
**Important note**

The CD4007 MOSFET array should not be connected to circuits with power ON since high transient voltages may cause permanent damage. Therefore, to avoid seriously damaging the device, do not apply input signals until PIN 7 and 14 have been properly terminated.

### 6.1 Voltage transfer characteristics of the CMOS inverter “Static Operation”

Fig. 6.3 shows the basic structure of a CMOS inverter. $Q_1$ and $Q_2$ are NMOS and PMOS transistors, respectively acting as an inverter. The objective here is to determine the noise margin $N_{MH}$ and $N_{ML}$, as well as determining the voltage transfer characteristic ($V_{OUT}$ vs. $V_{IN}$) of the MOS amplifier circuit, and its gain.

![Fig. 6.3 The CMOS inverter](image-url)
Note that in the cutoff and triode regions of the voltage transfer characteristic, CMOS circuit approximates the operation of a switch. In between these two regions lies the saturation region, which makes an amplifier.

Note: It could be useful if you take a look at Fig. 13.5 and 13.20 of the textbook (Sedra-Smith, 6th edition) before starting the experiment. See Sections 13.1.3 and 13.2.2 to know about the definition and computation of noise margins.

Hint: Use an appropriate software program, such as MICROSOFT EXCEL to record the experimental data and then plot the curve. The slope will be determined easily.

Procedure:

1. Mathematically, predict the noise margin $NM_H$ and $NM_L$ for the circuit.
   
   Note: Note that $V_{tn} \neq V_{tp}$. For NMOS, use the measured value from lab 4. You may use generic values of PMOS transistor (in lab 4 in the table).

2. Construct the circuit shown in Fig. 6.3. The transistor pins are also shown.

3. Set $V_{DD}$ to 5V DC.
   
   Note: Make sure that you use the same NMOS transistor for $Q_1$ as you used for the lab#5; otherwise you will need to measure the parameters of the new NMOS transistor.

4. Connect the substrate pin 7 to the ground node and substrate pin 14 to $V_{DD}$.

5. Use C=100pF at the output terminal. Measure its value using the RLC meter.

6. Apply DC voltage source into $V_{IN}$. Vary the voltage $V_{IN}$ (DC) from 0 to 5V in steps of 0.5V and record the corresponding drain voltage $V_{OUT}$.
   
   Note: You will notice that the output will change quickly at one point (specifically in the saturation region). When it happens, vary the input voltage in small steps, e.g. 0.01 or 0.02 V, around 2.5V in order to record the output voltage accurately and then the transfer characteristic curve.

7. Plot the output voltage, $V_{OUT}$, against the input voltage, $V_{IN}$.

8. From the plot, find the slope of the transfer characteristic around the points $Q_N$ and $Q_P$ in the saturation area. The slope is the transfer characteristic gain.

9. Also, determine the noise margin $NM_H$ and $NM_L$ from the plot.
Note: See Fig. 13.20 and Section 13.1.3 in the textbook (Sedra-Smith, 6th edition).

10. Simulate the circuit using LTSpice:
   a. Use MbreakN3 for NMOS transistor and set the parameters as in lab 4, section 4.4.
   b. In LTSpice, a 1μF capacitor between substrate pin 14 and the ground is NOT needed. It is required ONLY for building a circuit and experimental measurements in the lab.
   c. Perform a DC sweep, and enter the following:
      i. Sweep Variable Type: Voltage Source
      ii. Name: V1 (VIN) Sweept Type: Linear Sweep
      iii. Start Value: 0 End Value: 5 Increment: 0.1
   d. Plot the output voltage, VOUT, against the input voltage, VIN.
   e. Find the slope of the transfer characteristic, NM_H and NM_L from the simulation results.
   f. Include these results in the report.

6.2 Dynamic operation of the CMOS inverter

Note for part 6.2: This semester, mathematical calculation (part 1-2) and LTSPICE only (parts 5-6), no need to perform hardware

1. Modify Fig. 6.3 by removing the DC supply Vin and applying a square wave input signal of 5Vpp and 1kHz frequency.
2. Mathematically, calculate the propagation delay (τp), power dissipation (PD), and the delay-power product (DP).
   Note: For calculations and SPICE use \( k_p' = 0.33mA/V^2, V_{tp} = -1.0V \) and \( \lambda_p = -0.04V^{-1} \)
3. Observe the output and input on the oscilloscopes. Save the waveforms.
Note: It could be useful if you take a look at Fig. 13.15 of the textbook (Sedra-Smith, 6th edition), to see how the output and input waveforms look like to determine $t_{PHL}$ and $t_{PLH}$.

4. Record $t_{PHL}$, $t_{PLH}$, $t_f$, and $t_r$, where $t_{PHL}$ and $t_{PLH}$ are high-to-low propagation delay and high-to-low propagation delay, respectively. $t_f$ and $t_r$ are the fall delay time and rise delay time.

Note: Probably you would see oscillations in the oscilloscope screen for the input and output waveforms. Use your judgment to estimate the values by ignoring the oscillations. If you cannot, ask TAs how help you out.

5. Simulate the circuit using LTSpice. Verify your measured readings ($t_{PHL}$, $t_{PLH}$, and $t_p$) with LTSpice output.

Note: Choose signal PULSE to model the input signal. The description of the voltage source $VPULSE$ is as follows $(0 \ 5 \ 0 \ 5n \ 5n \ 0.5m \ 1m)$.

6. From the transient, analysis of the output, determine $t_{PHL}$, $t_{PLH}$, and $t_p$.

6.3 CMOS characterization of NAND and NOR gates (SPICE ONLY)

In the previous section, we have studied the CMOS inverter. Now we will move on with more complex circuits using four transistors. Fig 6.4 shows two famous logic circuits, NAND and NOR gates. Both of them have four transistors, two NMOS and two PMOS transistors.

Procedure:

1. Determine the truth tables for each gate in Fig. 6.4. Type the Boolean expression of NOR and NAND gates.

2. For each gate separately, simulate the circuit using LTSpice ONLY. DO NOT BUILD THE CIRCUIT.

Note: Use generic values of the LTSpice model of the CD4007 chip for both NMOS and PMOS transistors.

3. Use $C=100pF$ at the output terminal.
4. Test your NAND gate with attaching a 1kHz VPULSE wave with 5 Vpp, to the first input (A) and a DC voltage of either zero or 5 V to second input (B).

*Note: Choose VPULSE to model the input signal. The description of the voltage source VPULSE is as follows (0 5 0 5n 5n 0.5m 1m). The DC and AC values for VPULSE are 0 (or ground). In SPICE, the pulse source is represented as follow: VPULSE (V1 V2 TD TR TF PW PER), where V1: initial value, V2: pulse voltage, TD: delay time, TR: rise time, TF: fall time, PW: pulse width, PER: period.*

5. In each case, attach the input and output waveforms (coming out from Y node) to the lab report. Voltages probes can be used to monitor the voltages at the input and output.

*Note: The first case is input (A) is pulse and input (B) is 5V. The second case is input (A) is pulse and input (B) is 0. By doing this, you have all cases to cover the truth table.*

6. If applicable, from the transient analysis of the output, determine the propagation delay for each case.

*Note: You will not be able to get $t_p$ in some cases.*
7. Obtain the output waveform in each case and explain how it corresponds to the NAND gate of the two inputs, using a truth table format.

8. Repeat the same steps (4 – 7) for NOR gate.

Post-lab questions:

1. Use LTSpice to implement and discuss the outcomes of the followings:
   a. Changing the capacitor to be 500pF, instead of 100pF, shown in Fig. 6.3 (with pulse input), CMOS inverter circuit.
   b. Adjusting the pulse frequency as 10kHz, instead of 1kHz, shown in Fig. 6.3 (with pulse input), CMOS inverter circuit.
   c. Matching the transistors Q_P and Q_N, shown in Fig. 6.3 (with pulse input), CMOS inverter circuit.

2. Draw an analog circuit, using NMOS and PMOS transistors, to represent the following Boolean expression:

   \[ Y = \overline{A + B} \]  \hspace{1cm} (6.1)

   Make sure that you label the inputs (A and B) and output (Y) voltages in your drawing.
EEE334 Lab#7
Design an Audio Amplifier using MOS transistor
For Honor Students only

It is strongly recommended that you perform all lab experiments when an EEE334 Lab TA is on duty since only the EEE334 Lab TA can verify the correctness of your lab data. This experiment (design) does not need a datasheet nor TA signature.

Name:
ASU ID:
Lab Partner:

This experiment involves a design of an audio amplifier using MOS transistor. This design will be an introduction to multistage amplifiers, since the overall gain is high.

This design requires only a LTSpice simulation and detail analysis only. Do not build the circuit. You do not need a datasheet for this experiment, so you may perform it in your appropriate time. However, you are welcome to do it in the lab area to make sure that your design is correct.

Design a MOS audio amplifier, under the following conditions:

1. \( A_v \geq 300 \text{V/V} \) (50dB), where \( A_v \) is the overall voltage amplification.
2. \( P_{\text{TOT}} \leq 200 \text{ mW} \), where \( P_{\text{TOT}} \) is total power assumption.
3. \( 1 \text{k}\Omega \leq R_L \leq 1 \text{M}\Omega \), where \( R_L \) is the load resistance.
4. \( f_L \leq 20 \text{ Hz} \) and \( f_H \geq 20 \text{ kHz} \), where \( f_L \) and \( f_H \) are the frequencies at which the gain drops by 3dB below its mid-band.
5. \( V_S \leq 5\text{mVpp} \), where \( V_S \) is the peak-peak source voltage.

Results of your design have to include the followings:
1. Schematic diagram of your design.

2. Following circuit parameters:
   a. Values of $V_{DD}$, $V_{SS}$, $R_{REF}$ (if active-load amplifier is used)
   b. Width (W) and Length (L) of each MOS transistor.
   c. Technology of MOS transistor uses in your design, including transistor characteristics, such as $V_t$, $\lambda$, and $k'$.  
   d. Capacitors and resistors values.

3. Transient analysis, $V_S$ and $V_{out}$, for each stage.

4. Frequency response analysis, $20 \times \log_{10} (\frac{|abs(V_{out}/V_s)|}{})$. Show the $f_L$ and $f_H$.

5. Details of your analysis that includes your calculations. State clearly any necessary assumptions.

Hints and suggestions:

1. Since the overall voltage amplification is relatively high, you might want to design/build multistage of MOS amplifier to minimize the total power consumption. Ask TAs for more details and recall the subject in the book.

2. Pick one of five technologies of MOS transistor for your design of MOS transistors, listed in Table 7.A.1, page 554, in your textbook (Sedra-Smith, 6th edition).

3. Any stage could be an amplifier with resistive load or with active load.

4. Make sure that the transistor(s) operate in the saturation region.

Type your analysis in your report in details along with the simulation using SPICE ONLY. DO NOT BUILD THE CIRCUIT.

Post-lab questions:

1. Discuss the advantages and disadvantages of using multistage amplifiers over single stage amplifiers.
2. In your design analysis, what would happen if the power supply (V_{DD} and V_{SS}) were 10V?

3. In your design analysis, what would happen if the overall voltage amplification (A_v) were increased to 500V/V, instead of 300V/V?

You have to submit this report directly to your professor, not to TA. DO NOT DROP IT INTO THE LAB DEPOSIT BOX. TAs have nothing to do with grading this report.
References:


