# EE2019 Analog Systems Lab

# Table of Contents

## List of Figures

# Chapter 1 Analog Systems Lab Overview

## Objective

Design of a composite analog system for synchronized light and sound.

## Learning Outcome

At the end of this lab, students should be understand following topics with their application in real world

- Feedback theory
- Open and closed loop system
- Stability of a closed loop system
- Compensating an unstable system
- Voltage and Current regulation
- Active-RC Analog Filters
- Op-amp based pre-amplifier
- Audio receiver
- Audio amplifier

## Brief Description

The system consists of following three main modules

1. DC-DC Converter based LED Driver
2. Electronic Stethoscope
3. Class-D Audio Amplifier

When these 3 modules are connected together, it can synchronized light with sound by changing the brightness of LED (Light Emitting Diode) with sound level. Sound can be heard over speaker driven by class-D amplifier. Typically, heart beat and lung sound is used as an input which is derived from stethoscope and processed in electronic stethoscope module. However, alternate audio signal such as fixed frequency tone from audio source or functional generator can also be used.

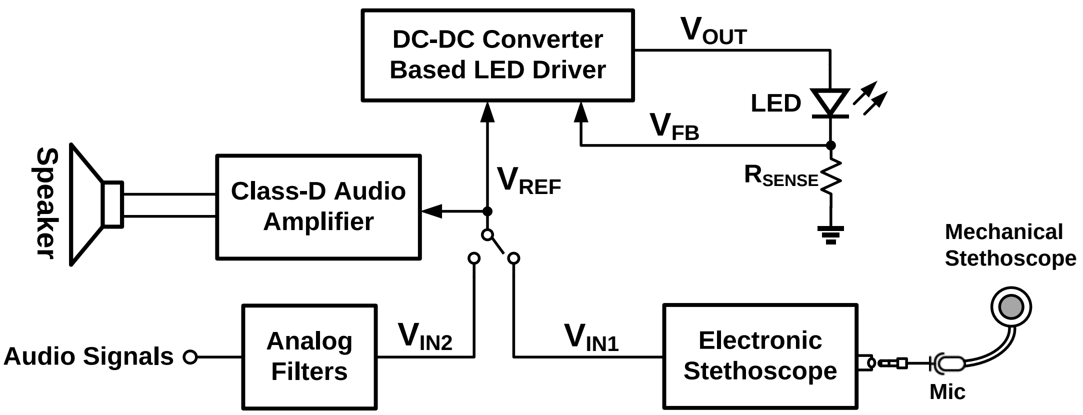



*Figure 1-1 Block diagram of the synchronized light and sound system*

## Evaluation

- Weekly pre-lab exercise and simulation results: 10%
- Weekly module demo: 30%
- Final system demo: 30%
- Final exam: 30%

## Important Instruction

- Pre-lab exercise and simulation results must be submitted before starting the lab experiment
- Students can use their choice of cad tool for simulation. Information about cad tools can be found at http://www.ee.iitm.ac.in/~nagendra/cadinfo.html
- All lab experiments are carried in group of two but reports are submitted individually
- Experiments start from week-2 as first week of the lab is used for orientation

# Chapter 2 DC-DC Converter Based LED Driver

## Introduction

LEDs are designed to operate with a constant current and brightness is usually proportional to the current. Since the V-I characteristic of LED as shown in Figure is exponential, a small change in voltage can cause a significant change in LED current. Since current higher than rated LED current may damage LED, it requires constant voltage over varying operation conditions. Accurate and constant voltage is achieved by voltage regulation (linear or switching). Switching regulator or dc-dc converter is often preferred over linear regulator due to higher efficiency.

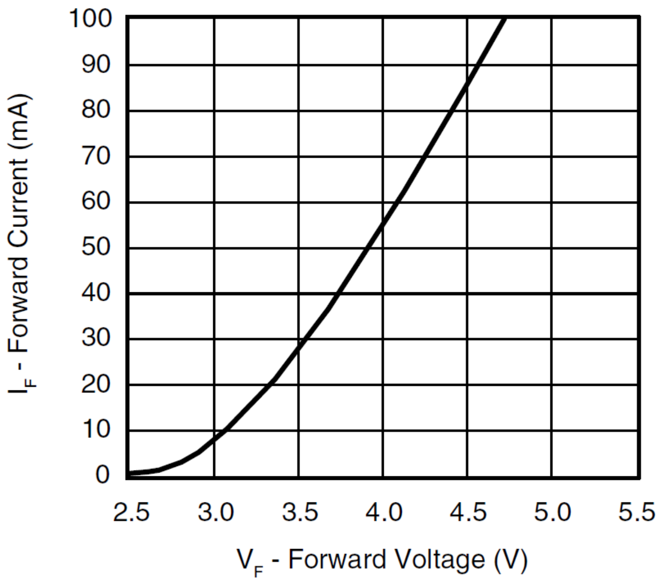



*Figure 2-1 Voltage Vs Current Characteristic of LED*

Figure 2-2 shows the block diagram of a switching dc-dc converter.

## Working Principle

Switching regulator works on the principle of Pulse Width Modulation (PWM) and output voltage, $V_{OUT}$ is expressed as:

Equation 2-1 $\quad V_{OUT} = D \cdot V_{IN}$

Where D is the duty cycle of PWM signal expressed as ratio of ON time over Time Period ($D=T_{ON}/T_{SW}$), $V_{IN}$ is the voltage level of PWM signal.

If $V_{IN}$ remains constant then desired VOUT can be achieved by simply generating a PWM signal with duty cycle $D=V_{OUT}/V_{IN}$ in an open loop system. However, in the real world, $V_{IN}$ varies depending upon the source. For instance if VIN is supplied from battery then voltage may be higher when battery is fully charge compared to when charge is low. Similarly if power source is solar panel voltage may vary based on the light. Therefore an open loop system may fail to work and closed loop system with negative feedback is required to regulate the output voltage with variable $V_{IN}$.

As shown in Figure 2-2, the feedback voltage, VFB which is scaled version of VOUT is compared with constant reference VREF to generate error signal VERR. Error signal is processed through compensator to generate the control signal VCTRL which is converted to PWM signal by PWM modulator. Since PWM modulator cannot supply high current, it requires a power stage to drive the large current. The switching PWM signal $V_{SW}$ is then passed through a low-pass filter which suppresses all the switching harmonics and converts the PWM signal into desired DC voltage (with small ripple content). $V_{OUT}$ is actually the average of the $V_{SW}$ (which is expressed by Equation 2-1) with small ripple content. The negative feedback automatically adjusts duty cycle D in case of varying $V_{IN}$ to ensure constant $V_{OUT}$.

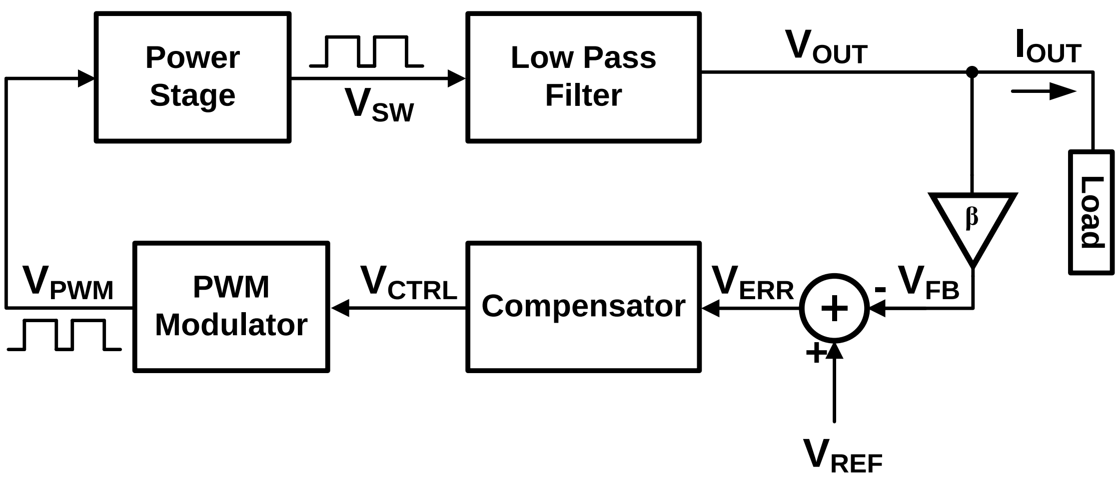



*Figure 2-2 Block diagram of a switching regulator*

The output voltage VOUT can be programmed either by changing feedback factor β or reference voltage $V_{REF}$ which can expressed as:

Equation 2-2 $\mathrm{V_{OUT}} = \frac{\mathrm{V_{REF}}}{\beta}$

## Building Blocks

As shown in Figure 2-2, a switching regulator consists of following blocks:

### 1. Low Pass Filter

Since filter has to supply the high load current, a very low loss filter is required. An ideal inductor has zero loss (zero impedance) at dc, hence LC low-pass filter makes an ideal choice for dc-dc converter.

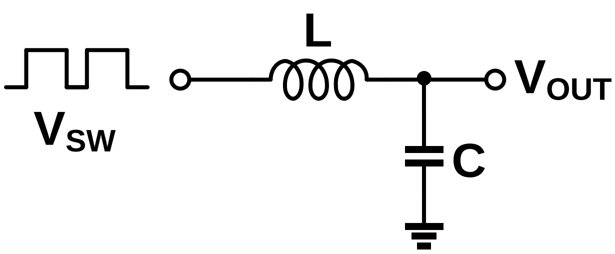



*Figure 2-3 An ideal LC Low-pass Filter*

In reality, inductor has a small series resistance call DCR and LC Low-pass filter in Figure 2-3 becomes a RLC filter as shown in Figure 2-4 which further modifies as Figure 2-5 with presence of resistive load $R_{OUT}$.

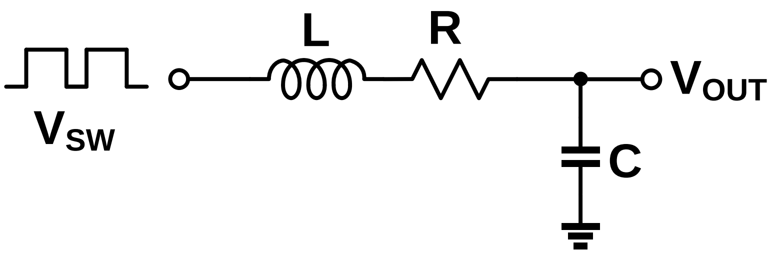



*Figure 2-4 A non-ideal LC Low-pass Filter*

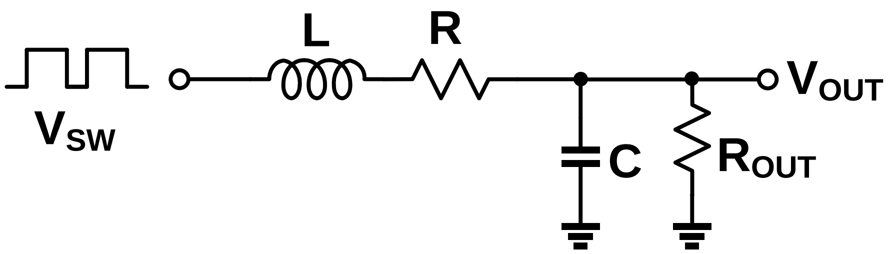



*Figure 2-5 A non-ideal LC Low-pass Filter with resistive load $R_{OUT}$*

***Exercise 2-1*** *Derive the AC transfer function of LC low-pass filters shown in Figure 2-3, Figure 2-4 and Figure 2-5. Find expressions for centre frequency $W_o$ and quality factor $Q_o$ for all the three filters. Study the effect of R and $R_{OUT}$ on $W_o$ and $Q_o$.*

## *Selecting L and C*

The values and inductor L and capacitor C is selected based on two factors (1) Switching frequency (2) Inductor ripple current. The cut-off frequency of LC filter is selected 50-100 times lower than switching frequency to minimize the output voltage ripple. Value of inductor is selected to minimize the inductor ripple current for reduced RMS losses and also prevent the inductor from getting saturated. Since larger inductor value comes at the cost of bigger area, there is always a trade-off between inductor size and efficiency. The minimum value of an inductor is quite often chosen such that peak-to-peak ripple current of inductor does not exceed 1.5-2 time of the maximum load current while maximum value depends upon the required light load efficiency.

The peak-to-peak inductor ripple current can be expressed as:

Equation 2-3 $$\Delta I_L = \frac{V_{IN} - V_{OUT}}{L} \cdot \frac{D}{F_{SW}}$$

Where D is the duty cycle and $F_{SW}$ is the switching frequency of the PWM signal $V_{SW}$. The output ripple voltage can be derived by integrating the inductor ripple current and expressed as:

Equation 2-4 $\Delta V_O = \frac{V_{IN}-V_{OUT}}{L} \cdot \frac{D}{8 \cdot C \cdot F_{SW}{}^2}$

The behaviour of inductor ripple current and output ripple voltage is shown in Figure 2-6.

There might be inductors with different dc and saturation current ratings for the same value and one should be careful in choosing the inductor to ensure that peak inductor current does not exceed the inductor saturation current under any operating conditions.

***Exercise 2-2*** *For a constant $V_{OUT}$, derive the duty cycle D for which $\Delta I_L$ is maximum. Plot the characteristic of $\Delta I_L$ Vs. D for D=0 to 1 for $V_{IN}$=5V, L=10uH and $F_{SW}$=500KHz.*

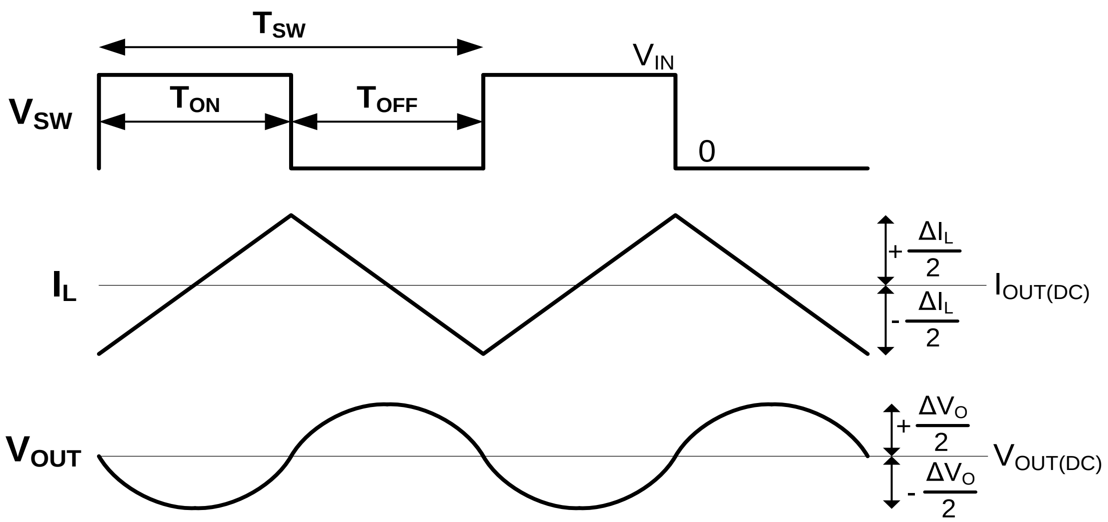



*Figure 2-6 Inductor ripple current and output ripple voltage of LC Low-pass Filter with PWM input*

## 2. Compensator

The RLC filter possesses double poles which are complex in nature hence causing 180 Degree phase shift. Negative feedback with 180 Degree phase shift makes the system unstable hence need to be compensated. As per the rule, in order to have a stable system, there could be only one dominant pole in a closed loop system with negative feedback. The compensator in a dc-dc converter can be used to either cancel one of the poles of LCR filter by using type-3 compensation or push both the poles outside unity gain bandwidth by using type-1 compensation.

### *Type-1 Compensation*

Type-1 compensation uses a single pole low pass filter or integrator such that the UGB of the loop is much less (5-10 times) of the double pole frequency of LC filter. Figure 2-7 shows a first order opamp-RC filter used as type-1 compensator.

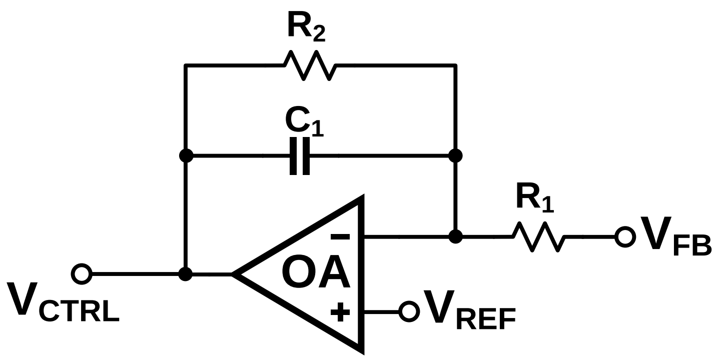



*Figure 2-7 Fist order opamp-RC filter as type-1 compensator*

Connecting positive terminal of opamp to VREF performs the function of subtraction ($V_{ERR}=V_{REF}-V_{FB}$) and low pass filter processes the error signal to get $V_{CTRL}$. Ideally, we desire zero dc error between VFB and VREF which can only be achieved by having infinite gain at dc. The feedback resistor R2 in the low pass filter limits the dc gain hence an opamp-RC integrator is preferred over lowpass filter as type-1 compensator.

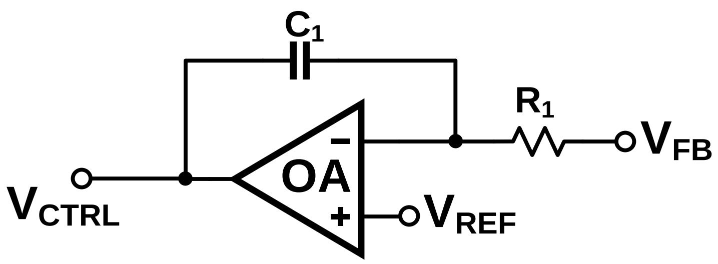



*Figure 2-8 Opamp-RC integrator as type-1 compensator*

Type-1 compensation can only be used with slower system where fast transient response or tracking speed is not needed as low bandwidth of the loop makes the system very slow.

***Exercise 2-3*** *Draw the bode plots of lowpass filter and integrator shown in Figure 2-7 and Figure 2-8, respectively. Find the expression for unity gain bandwidth (UGB) for the two circuits.*

### *Type-3 Compensation*

Unlike type-1 compensator which pushes the double LC poles out of UGB by reducing the loop bandwidth, type-3 compensator cancels one of LC poles and extends the loop bandwidth. Type-3 compensator offers fast transient response and tracking speed due to higher bandwidth. The compensator is also known as PID as it possesses Proportional (P), Integral (I) and Derivative (D) components. Circuit diagram of a type-3 compensator is shown in

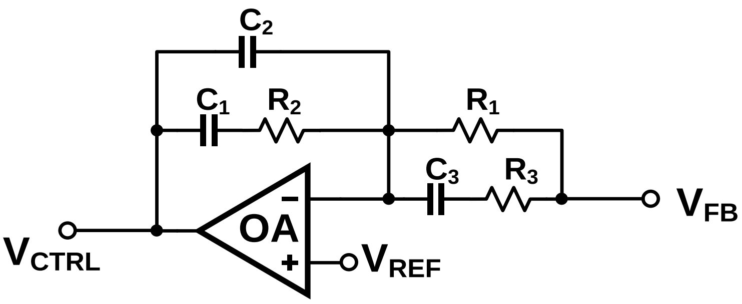



*Figure 2-9 Opamp-RC integrator as type-1 compensator*

## 3. PWM Modulator

PWM modulator is used to convert the control voltage, $V_{CTRL}$ to PWM signal by comparing $V_{CRTL}$ with a fixed frequency ramp signal as shown in Figure 2-10. Duty cycle of the PWM signal is proportional to $V_{CTRL}$ and can be expressed as:

*Equation 2-5* $$D = \frac{T_{ON}}{T_{SW}} = \frac{V_{CTRL}}{V_M}$$

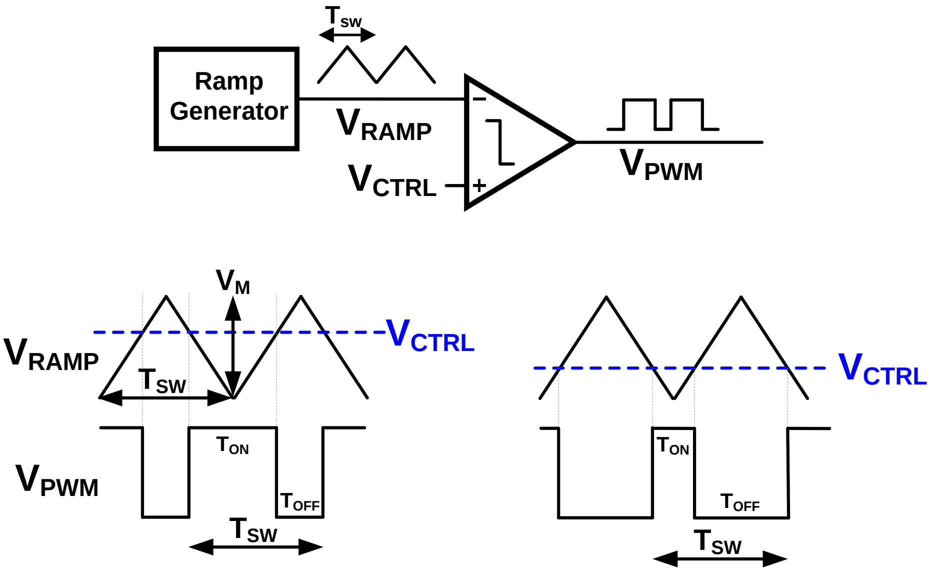



*Figure 2-10 PWM Modulator*

## 4. Power Stage

Since PWM comparator is not strong enough to drive high current, it requires high current complementary switches $M_P$ and $M_N$. These switches are usually power MOSFETs with high gate capacitance hence also require gate drivers to ensure small rise/fall times. Non-overlap clock generator is used to avoid any circuit current between VIN-GND via $M_P$-$M_N$ which may damage the circuitry. Non-overlap time can be adjusted by changing values if capacitors $C_P$ and $C_N$.

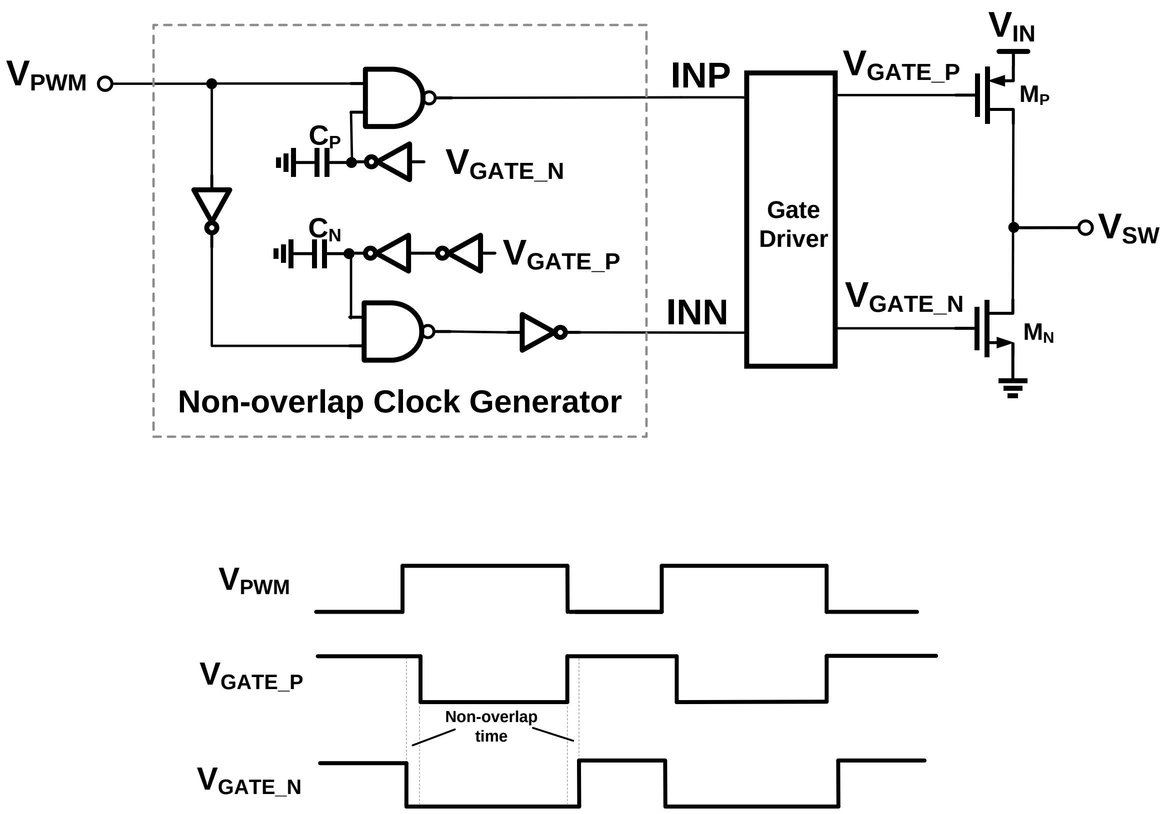



*Figure 2-11 Power Stage with Non-overlap Clock Generator and Gate Driver*

The complete LED driver using Type-I and Type-III compensator are shown in Figure 2-12 and Figure 2-13, respectively. Open loop or loop gain transfer function of the LED driver can be expressed as:

*Equation 2-6* $$H(s) = \beta \cdot H_{comp}(s) \cdot \frac{1}{V_M} \cdot V_{IN} \cdot H_{LS}(s)$$

Where,

$V_M$ is the peak-to-peak amplitude of ramp signal ($V_{RAMP}$), $V_{IN}$ is the input supply of power stage and β is the feedback factor and can be derived from Equation 2-2 as:

*Equation 2-7* $$\beta = \frac{V_{REF}}{V_{OUT}} = \frac{V_{REF}}{V_{REF}+V_{F\_LED}}$$

Where $V_{F\_LED}$ is the LED forward voltage, $V_M$ is the amplitude of the ramp signal and $V_{IN}$ is the power stage input supply voltage.

$H_{COMP}(s)$ is the transfer function of compensator and $H_{LS}(s)$ is the transfer function of LC low-pass filter.

Current into LED ($I_{OUT}$) can be expressed as:

*Equation 2-8* $$I_{OUT} = \frac{V_{REF}}{R_{SENSE}}$$

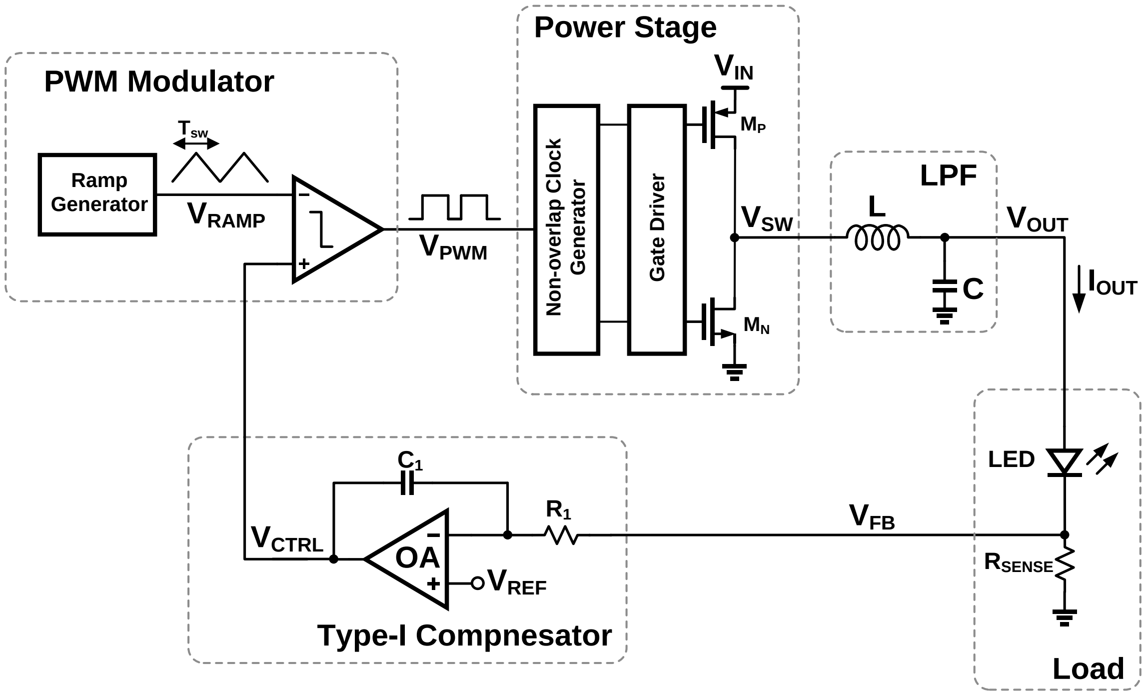



*Figure 2-12 Circuit diagram of a dc-dc converter based LED driver using Type-I compensator*

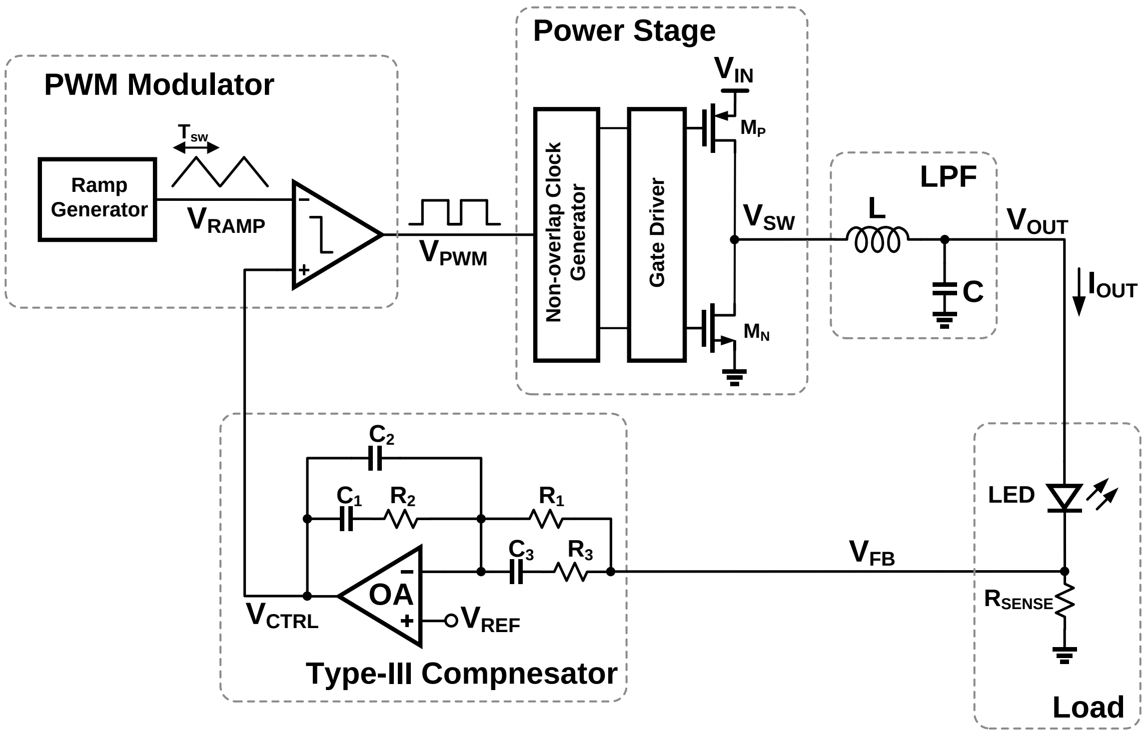



*Figure 2-13 Circuit diagram of a dc-dc converter based LED driver using Type-III Compensator*

## References:


1. http://www.electronics-tutorials.ws/opamp/opamp_6.html
2. https://www.allaboutcircuits.com/textbook/semiconductors/chpt-8/differentiator-integrator-circuits/
3. http://fab.cba.mit.edu/classes/961.04/topics/pwm.pdf
4. http://www.ti.com/lit/an/sloa020a/sloa020a.pdf
5. https://en.wikipedia.org/wiki/Phase_margin
6. http://www.mit.edu/afs.new/athena/course/2/2.010/www_f00/psets/hw3_dir/tutor3_dir/tut3_g.html
7. http://www.linear.com/solutions/4449
8. https://www.allaboutcircuits.com/technical-articles/negative-feedback-part-4-introduction-to-stability/
9. https://www.allaboutcircuits.com/technical-articles/negative-feedback-part-5-gain-margin-and-phase-margin/
10. https://www.allaboutcircuits.com/technical-articles/negative-feedback-part-6-new-and-improved-stability-analysis/
11. https://www.allaboutcircuits.com/technical-articles/negative-feedback-part-9-breaking-the-loop/

## Lab Experiments

LED driver module is divided into following three experiments:

- EXPERIMENT-1: RAMP GENERATOR (Week-1)
- EXPERIMENT-2: PWM MODULATOR AND POWER STAGE (Week-2)
- EXPERIMENT-3: COMPENSATOR and MODULE INTEGRATION (Week-3)

## EXPERIMENT-1: RAMP GENERATOR AND PWM MODULATOR

### Circuit Diagram

Ramp or triangle wave generator is actually an oscillator which is designed using opamp-RC integrator and Schmitt trigger.

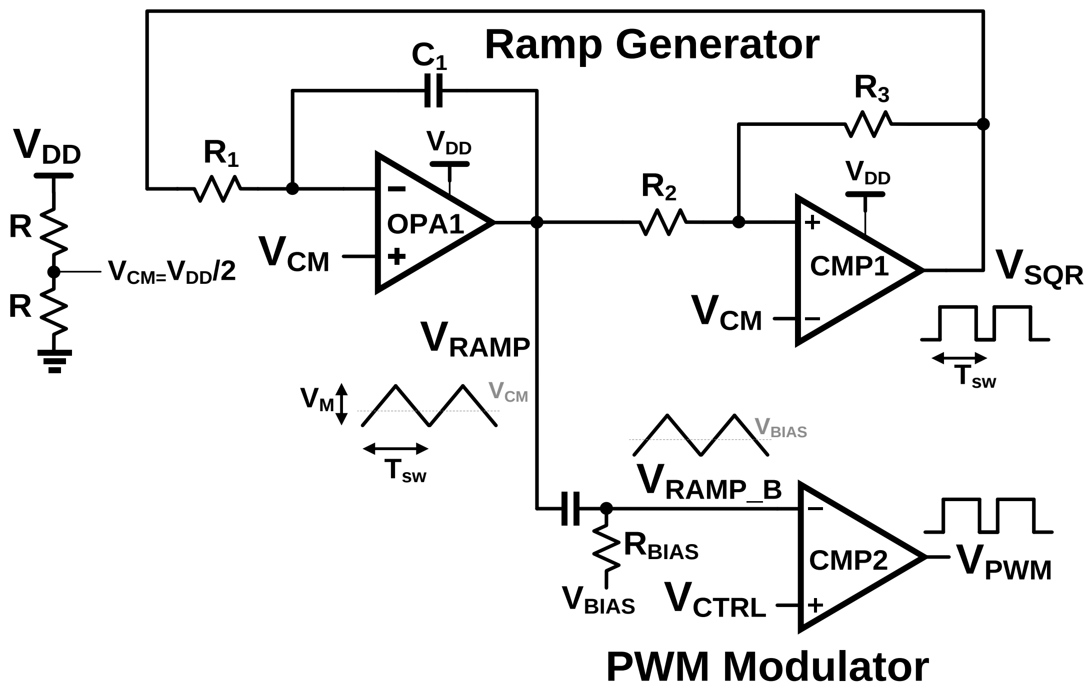



*Figure 2-14 Ramp Generator Circuit*

The peak-peak amplitude of the ramp is defined by the equation:

*Equation 2-9* $$V_M = 2 \cdot \frac{R_2}{R_3} \cdot V_{CM}$$

The oscillation frequency of the ramp is given by equation:

*Equation 2-10* $$F_{SW} \; or \; 1/T_{SW} = \frac{R_3}{4 \cdot R_2 \cdot R_1 \cdot C_1}$$

## Specifications

- Supply voltage ($V_{DD}$) = 3V-5V
- Frequency ($1/T_{SW}$) = 100KHz
- Peak-peak ramp amplitude ($V_M$) = 1V (min)-2V (max)

## List of Components

- OPA1: MCP6292
- CMP1 and CMP2: LM339 (open collector – requires a pullup resistor between $V_{DD}$ and $V_{OUT}$)

## List of Measurements

1. Set $V_{IN}$=4V, $V_{CM}$ = $V_{IN}/2$, $V_{CTRL}$ = $V_{BIAS}$
2. Capture integrator output ($V_{RAMP}$) and Schmitt trigger output (square wave)
3. Measure and record frequency of $V_{RAMP}$ and square wave
4. Measure amplitude of $V_{RAMP}$
5. Capture the ramp waveform $V_{RAMP\_B}$ and measure the amplitude and dc bias
6. Measure and record frequency of $V_{RAMP\_B}$
7. Capture $V_{PWM}$, measure frequency and duty cycle
8. Capture and measure $V_{BIAS}$
9. Sweep $V_{CTRL}$ between get duty cycles of 0%, 25%, 50%, 75% and 100%. Measure and record $V_{CTRL}$ and $V_{PWM}$ duty cycle.
10. Repeat 2-6 by for $V_{IN}$ = 3V and 5V.

## Pre-Lab Exercises

1. For the ramp generator circuit in Figure 2-14, derive the expression for ramp amplitude (Equation 2-9) and frequency (Equation 2-10)
2. Simulate the ramp generator circuit shown in Figure 2-14 and verify the expressions in Equation 2-9 and Equation 2-10. Sweep the values and observe the effect on ramp amplitude and frequency.
3. Perform measurements 1-10 using simulation.

# EXPERIMENT-2: POWER STAGE AND LPF

## Circuit Diagram

Power stage uses $V_{PWM}$ from PWM modulator as input and drives LC LPF through power MOSFETs $M_P$ and $M_N$. $V_{GATE\_P}$ and $V_{GATE\_N}$ must be non-overlapped (break before make) to avoid short circuit condition which may damage bread board and circuitry. Non-overlap time of the power stage can be adjusted by varying $C_P$ and $C_N$. It is recommended to disconnect power supply ($V_{IN}$) from $M_P$ for testing the non-overlap time. Once non-overlap time is verified, VIN can be connected back.

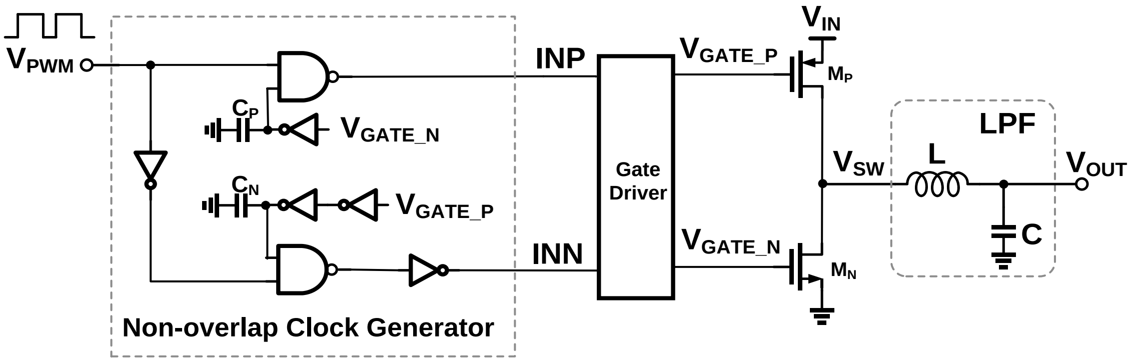



*Figure 2-15 Power stage and LPF*

## Specifications

- Supply voltage ($V_{IN}$=$V_{DD}$) = 3V-5V
- PWM Frequency ($1/T_{SW}$) = 100KHz

## List of Components

- NAND Gates: SN74AHC00N
- Inverters: CD4069UBE
- Gate Driver: TC427EPA
- Power MOSFETs: SFT1341 and STD17NF03L
- Inductor (L): RCH875NP-101K
- Capacitor (C): 47uF

## List of Measurements

1. Set $V_{DD}$=$V_{IN}$=4V, PWM duty cycle (D)=50%
2. disconnect $V_{IN}$ from MP and input $V_{PWM}$ from Experiment-1
3. Capture $V_{GATE\_P}$ and $V_{GATE\_N}$, measure non-overlap
4. Connect $V_{IN}$ back to $M_P$
5. Capture $V_{SW}$ and measure dead time, duty cycle and frequency. Observe the difference between $V_{PWM}$ and $V_{SW}$
6. Plot $V_{OUT}$, measure average value, ripple amplitude and frequency
7. Vary PWM duty cycle (D) from 0 to 100% with 25% step by adjusting $V_{CTRL}$ and repeat 6. Verify relationship, D = $V_{OUT}/V_{IN}$
8. Set D=50% and apply resistive load to draw 50mA from $V_{OUT}$

9. Observe difference in $V_{OUT}$ with and without load. What could be the possible reasons for differences?
10. Repeat 6-9 for $V_{IN}$ = 3V and 5V. For step 8, adjust duty cycle maintain 50mA load current

## Pre-Lab Exercises

1. For the power stage shown in Figure 2-15,
2. Simulate the ramp generator circuit shown in Figure 2-14 and verify the expressions in Equation 2-9 and Equation 2-10. Sweep the values and observe the effect on ramp amplitude and frequency.
3. Perform measurements 1-10 using simulation. Capture all the graphs

# EXPERIMENT-3: COMPENSATOR AND MDODULE INTEGRATION

## Circuit Diagram

For simplicity, type-I (integrator) compensator is used for loop compensation.

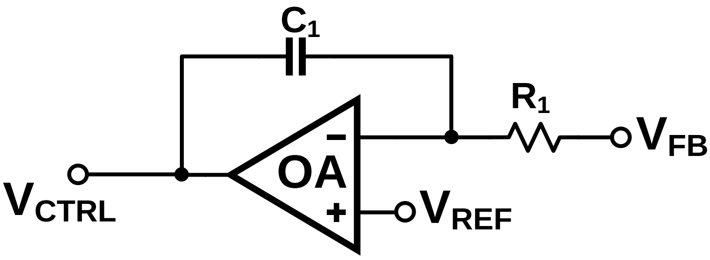



*Figure 2-16 Type-I (Integral) Compensator*

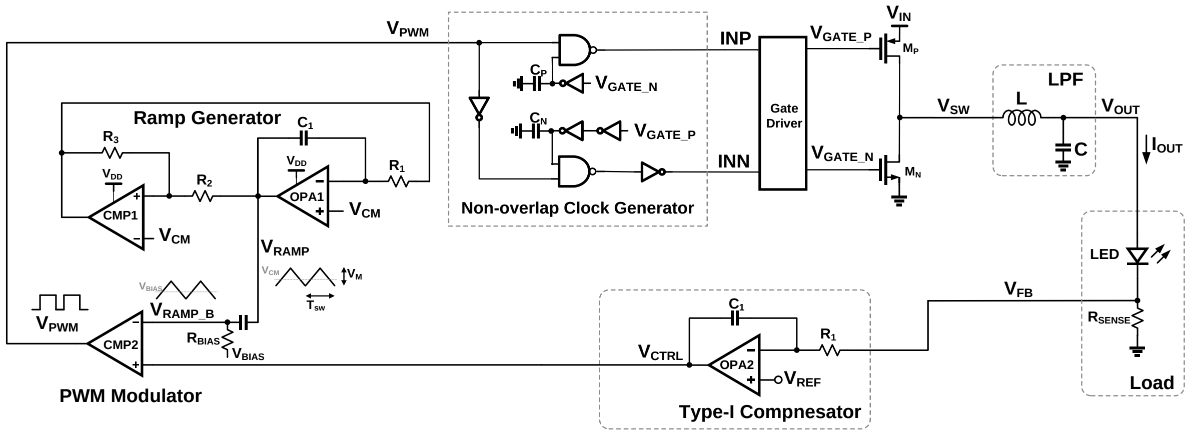



*Figure 2-17 Complete LED Driver*

## Generating $V_{REF}$

$V_{REF}$ applied at positive terminal of OPA2 determines the current into LED (see Equation 2-8). For standalone LED driver, VREF can be generated from external supply voltage. Instead of connecting power supply output directly to VREF, a first order RC filter can be employed to filter any noise in the supply. $V_{REF}$ should be turned ON only after the main supply $V_{IN}$.

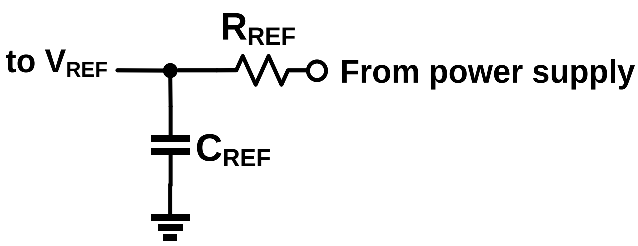



*Figure 2-18 $V_{REF}$ Generation*

The values of $R_{REF}$ and $C_{REF}$ should be chosen to keep the -3dB cutoff of the filter at 10Hz or below.

## Stability Analysis

Stability analysis of the complete LED driver ( is done by modelling the circuit in continuous domain to get the open loop transfer function of Equation 2-6 so that bode plot can be used to analyse the transfer function.

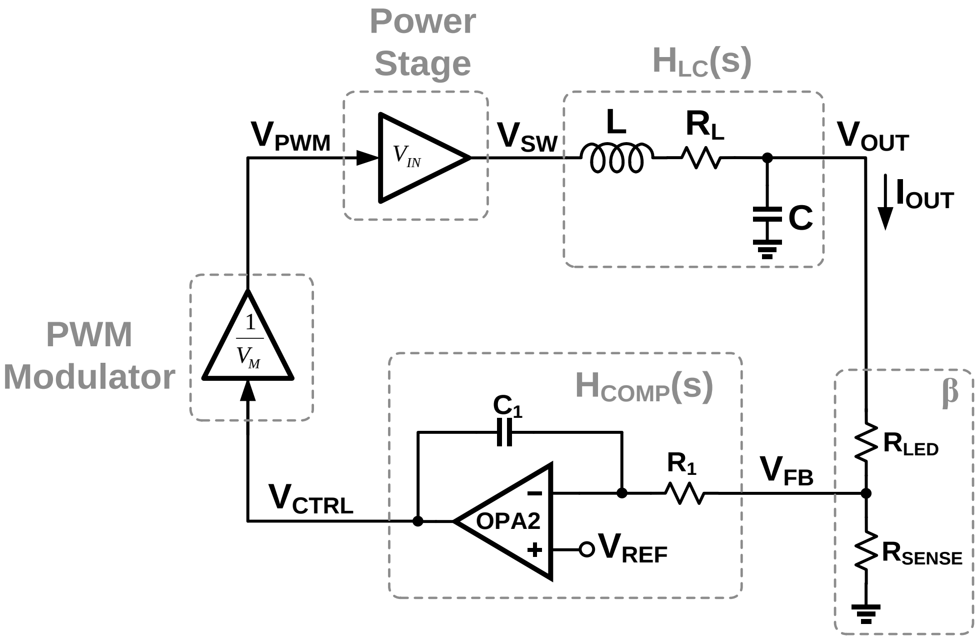



*Figure 2-19 Continuous time model of siwtching LED driver with Type-I compensator*

LED can be replaced by an equivalent resistor $R_{LED}$ and $V_{FB}$ can be expressed as:

*Equation 2-11* $$V_{FB} = \frac{R_{SENSE}}{R_{SENSE}+R_{LED}} V_{OUT}$$

Since $V_{FB}=V_{REF}$

*Equation 2-12* $$\beta = \frac{V_{REF}}{V_{OUT}} = \frac{R_{SENSE}}{R_{SENSE}+R_{LED}}$$

Using Equation 2-7 and Equation 2-12, $R_{LED}$ can be calculated as:

*Equation 2-13* $$R_{LED} = R_{SENSE} \cdot \frac{V_{F\_LED}}{V_{REF}}$$

Forward voltage of LED ($V_{F\_LED}$) can be found the datasheet and is usually in the range of 2V to 3.3V depending upon the current capacity and colour.

For PWM modulator and power stage gains can be implemented using voltage controlled voltage source (VCVS) or a simple ideal gain element if available in the simulator's ideal component library.

Once circuit is modelled, stability analysis can be performed by breaking the loop (to get the open loop transfer function). In, the loop is broken at output and should be at the output ac input of amplitude 1 is applied at $v_{in\_ac}$.

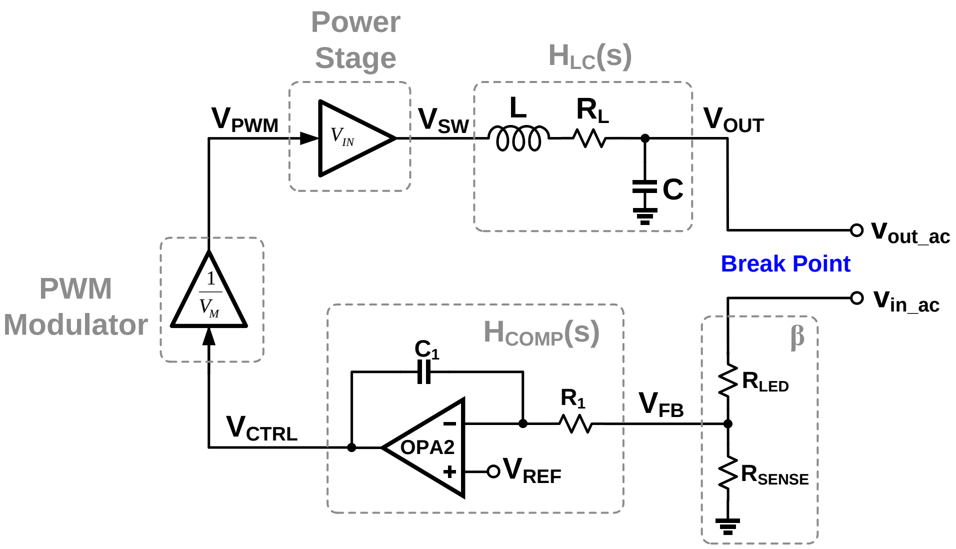



Figure 2-20 Breaking the loop for stability analysis

Since open loop transfer function is needed only for ac and dc operating points of the circuit should not be disturbed after opening the loop. In order to preserve the dc operating point of the circuit, loop is broken in such a way that it should behave like closed loop for dc but open loop for ac. This can be achieved by breaking the loop using inductor and capacitor as shown in Figure 2-21. Since inductor behaves like a short circuit at dc and capacitor as open circuit, loop will remain closed at dc. While for ac inductor behaves as open and capacitor as short, loop will open for ac.

Values $L_{break}$ and $C_{break}$ should be large (order of Mega Henry and Mega Farad) so that they don't interference with actual ac response of the circuit.

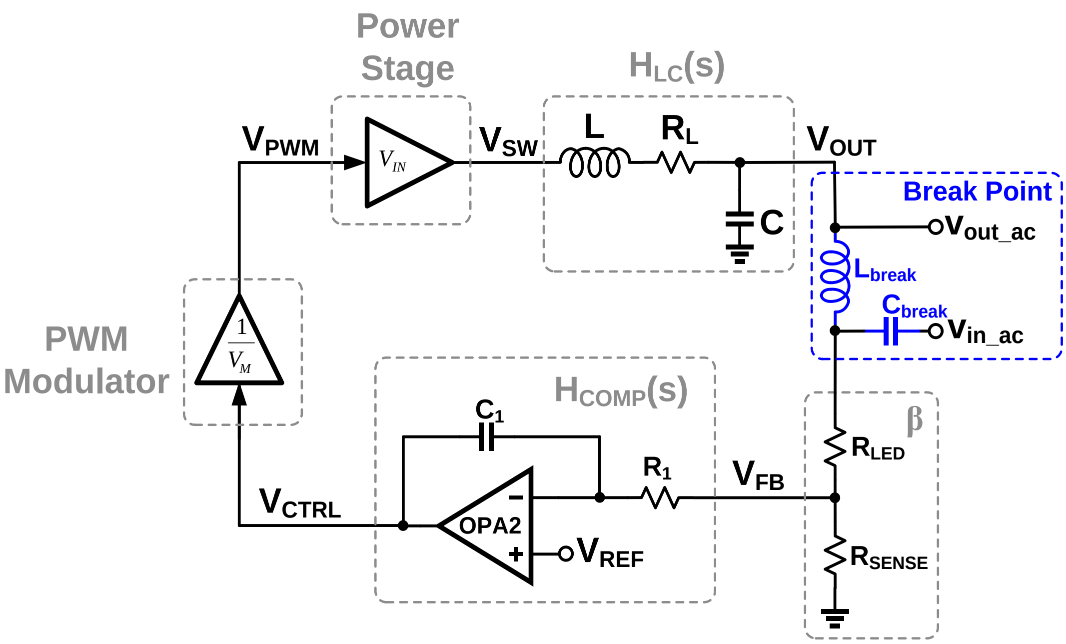



Figure 2-21 Breaking the loop using L and C

Stability of the circuit is checked by looking at the phase margin. Phase margin is defined as (phase difference of total loop phase shift from 0 or 360 degrees at unity gain (0dB). The frequency at unity gain is called unity gain bandwidth ($F_{UGB}$). Even though a system with > 0 degree phase margin is theoretically stable, in phase margin of a stable system should be greater than 45 degrees. However, it is recommended to have the phase margin ≥ 60 degrees for better transient response (without any ringing in the output).

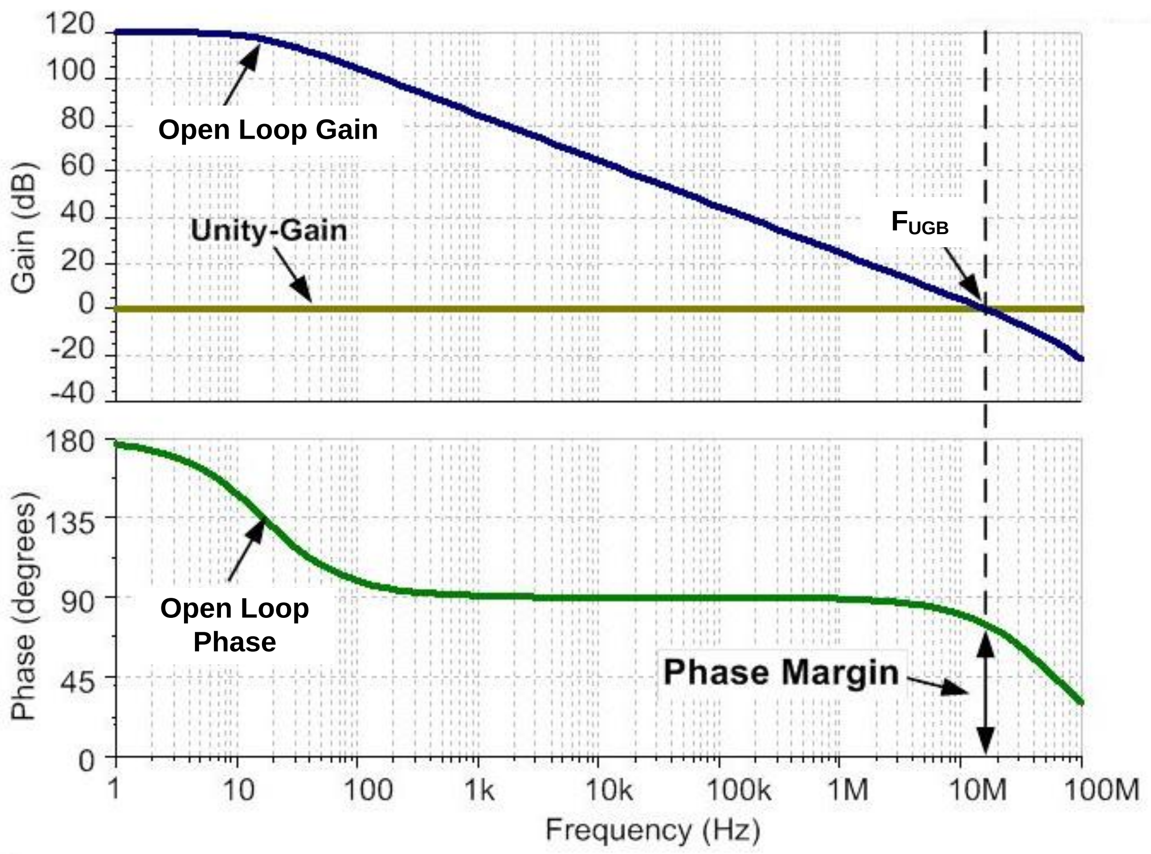



*Figure 2-22 Phase Margin of a feedback system*

## Specifications

- Supply voltage ($V_{IN}$=$V_{DD}$) = 3V-5V
- Phase Margin > 60 Degree
- $I_{OUT}$ ($I_{LED}$) = 50mA
- $R_{SENSE}$ = 5 Ohm

## List of Components

- Op-Amp (OPA2): MCP6292
- Inductor (L=100uH): RCH875NP-101K
- LED: 151053YS04500
- Sense Resistor ($R_{SENSE}$=5Ω): MOSX1CT52R5R1J

## List of Measurements

1. Set $V_{DD}$=$V_{IN}$=4V, $V_{REF}$=0V
2. Observe $V_{OUT}$=0V, LED is OFF ($I_{OUT}$=0) and there is no switching i.e. $V_{CTRL}$=0, $V_{PWM}$=$V_{SW}$=0
3. Slowly increase $V_{REF}$ to a value (few mV) where LED starts turning ON. Measure and plot $V_{OUT}$, $V_{FB}$, $V_{RAMP}$, $V_{CTRL}$, $V_{PWM}$ and $V_{SW}$. Verify that PWM duty cycle, D = $V_{CTRL}/V_M$ = $V_{OUT}/V_{IN}$
4. Repeat step 3 with 5 different values of $V_{REF}$ with uniform step. The maximum value of $V_{REF}$ should be such that LED current should not exceed specified current rating. Refer to the LED datasheet for maximum current rating.
5. Repeat 2-4 for $V_{IN}$ = 3V and 5V.
6. Turn OFF $V_{REF}$ first and then $V_{IN}$.
7. Change -3dB cutoff of $V_{REF}$ low-pass filter to 10KHz and use function generator to supply $V_{REF}$
8. Select square wave of amplitude 250mV (with low level=0V and high level=250mV), frequency = 1Hz, duty cycle = 25%
9. Set $V_{IN}$=4V and turn on $V_{IN}$ first and then $V_{REF}$ coming from function generator. Observe LED light. Increase duty cycle if LED does not blink. Measure and capture $V_{OUT}$, $V_{FB}$, $V_{RAMP}$, $V_{CTRL}$, $V_{PWM}$ and $V_{SW}$.
10. Now turn OFF $V_{REF}$ function generator output and set it to sinusoid with frequency 1Hz and amplitude 250mV (Vmin=0V, Vmax=250mV).
11. Turn ON $V_{REF}$ and observe LED light. It should follow the sinusoid pattern. Capture voltages $V_{OUT}$, $V_{FB}$, $V_{CTRL}$, $V_{PWM}$ and $V_{SW}$ for one cycle of sinusoid.
12. Sweep the sinusoid frequency from 1 Hz to 1KHz and observe LED light. Does LED stop blinking at higher frequency? What is that frequency?

## Pre-Lab Exercise

1. For the LED driver in Figure 2-17, Find the loop gain transfer function with and without type-I compensator. Calculate the values of $R_1$ and $C_1$ of the compensator for phase margin > 60 degrees. Use continuous time model (Equation 2-6, Equation 2-7, Equation 2-11, Equation 2-12, Equation 2-13 and Figure 2-19) and perform the AC or stability analysis using simulation tool (see Figure 2-20, Figure 2-21 and Figure 2-22). Capture AC magnitude and phase response with and without compensator.
2. Calculate $V_{REF}$ for LED current of 10mA, 50mA and 100mA. If $V_{REF}$ is fixed at 250mV, how will you program the LED current to 10mA, 50mA and 100mA?
3. Design switching LED driver shown in Figure 2-17 and perform measurements 1-11 using simulation tool. Observe LED current for LED light. Capture all the graphs.

# Chapter 3 Class-D Audio Amplifier

Class-D amplifier module is same as EE3703: Analog Circuits Lab with few minor changes. Details about class-d amplifier can be found at:

http://www.ee.iitm.ac.in/vlsi/courses/ec330_2011/finalproject/classdamp

## References:


1. Wikipedia article
2. Notes on Class D amplifier from Georgia Institute of Technology
3. Notes from Elliott Sound Products
4. Brett Forejt, Vijay Rentala, Jose Duilio Arteaga, and Gangadhar Burra, "A 700+-mW Class D Design With Direct Battery Hookup in a 90-nm Process," *IEEE Journal of Solid-State Circuits*, Volume 40, Issue 9, Sep. 2005, pp. 1880-1887.
5. Varona et al., "A Low-Voltage Fully-Monolithic ΔΣ-Based Class-D Audio Amplifier," *Proceedings of the 1999 European Solid State Circuits Conference*, pp. 545-548. (This has an example of switch sizing. This is not the type of class D amplifier you are required to design)
6. Putzeys B., "Digital audio's final frontier," *IEEE Spectrum* vol. 40, no. 3, Mar. 2008. pp. 34-41.
7. Berkhout M., "Audio at low and high power," *Proceedings of the 2008 European Solid State Circuits Conference* pp. 40-49.
8. Application notes from companies
    a. Texas Instruments: http://www.ti.com/audio/ (e.g. Class-D LC Filter Design, 07 Jan 2008; TPA3101D2 Mono Amplifier Configuration, 16 Apr 2007)
    b. Maxim Integrated Circuits: http://www.maxim-ic.com/appnotes.cfm/appnote_number/3977 (The bridged three level topology shown here may be a bit confusing. See the TI datasheet for a simpler topology- logically they are the same)
    c. Analog Devices: http://www.analog.com/library/analogDialogue/archives/40-06/class_d.html
    d. International Rectifier: http://www.irf.com/product-info/audio/classdtutorial.pdf
    e. http://www.infineon.com/dgdl/an-1071.pdf?fileId=5546d462533600a40153559538eb0ff1


## List of Difference between EE3703 and EE2019 Class-d Amplifier:

| **Parameters** | **EE2019 Class-d** | **EE3703 Class-d** |
|---|---|---|
| PWM Frequency | 100KHz | 300KHz |
| Ramp Generator | Op-Amp and Comparator based (used from experiment-1) | BJT based |

# EXPERIMENT-4: SINGLE ENDED-TO-DIFFERENTIAL INPUT CONVERTER AND PWM MODULATOR

## Circuit Diagram:

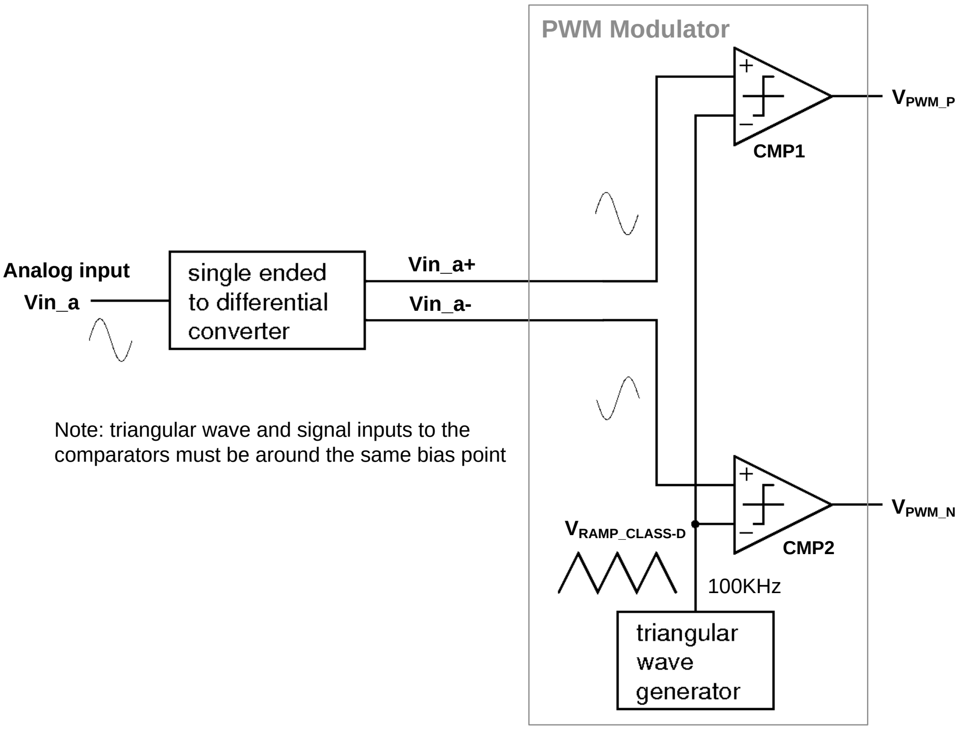



*Figure 3-1 Block diagram of single ended-to-differential converter and PWM modulator*

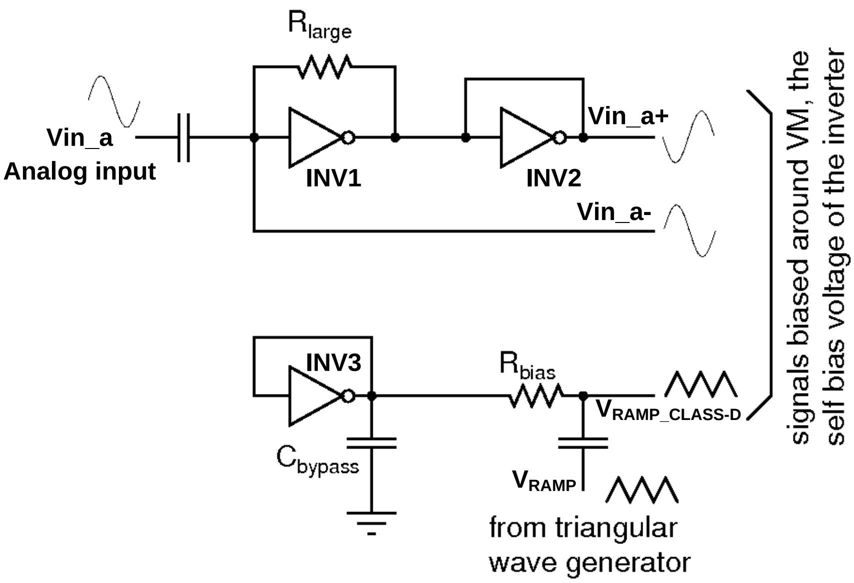



*Figure 3-2 Circuit diagram of single ended-to-differential converter*

## Specifications

- Supply voltage ($V_{IN}$=$V_{DD}$) = 3V-5V
- PWM Frequency = 100KHz

## List of Components

- CMP1 and CMP2: LM339 (open collector – requires a pullup resistor between $V_{DD}$ and $V_{OUT}$)
- INV1, INV2 and INV3: MC14069

## List of Measurements

1. Set $V_{DD}$=$V_{IN}$=4V
2. From function generator, set sinusoid wave of 1KHz and use as input to single ended-to-differential converter. Peak-to-peak amplitude of the sinusoid should be same as peak-to-peak amplitude of the triangular wave.
3. Measure amplitude and frequency of waveforms at input, Vin+ and Vin-. Capture oscilloscope waveform and verify that Vin+ and Vin- are 180 degrees out of phase and have same amplitude as input.
4. Measure and capture duty cycle at $V_{PWM\_P}$ and $V_{PWM\_N}$. Duty cycle should follow the same pattern as Vin_a+ and Vin_a-. Verify that $V_{PWM\_N}$ has inverter duty cycle (1-D) of $V_{PWM\_P}$ (D).
5. Add an RC filter at $V_{PWM\_P}$ and $V_{PWM\_N}$ with 3dB cut-off frequency of 10-20KHz and observe the output. Verify that output has the same shape as Vin_a+ and Vin_a-.
6. Repeat 2-5 for supply voltage, $V_{IN}$=3V and 5V. Observe the difference and comment.

## Pre-Lab Exercise

1. Drive the expression for Vin+ and Vin- in terms of input and prove that Vin+ and Vin- have sane amplitude but of opposite polarity.
2. Find the expression for differential PWM signal, $V_{PWM\_P}$-$V_{PWM-N}$ and prove that average output is amplified version of analog input to single ended to differential converter. Find the gain of amplifier.
3. Build the complete circuit shown in Figure 3-1 and Figure 3-2 in LTSpice. Verify the functionality by simulation with measurements 1-6.

# EXPERIMENT-5: H-BRIDGE DRIVER AND INTEGRATION

## Circuit Diagram:

Figure 3-6 shows the circuit diagram of half-bridge driver. The driver is the output stage of class-D amplifier and is the key to obtaining good efficiency. The switches (Qp and Qn) of the half-bridge driver are implemented using NPN and PNP transistors and driven with CMOS inverter buffers. Use a base resistance (bases of $Q_p$ and $Q_n$) of a few kilohms to limit the base current. If you find that the drive is insufficient (i.e. the transistors don't saturate with a heavy load), reduce the base resistances so that they saturate. If you find that the drive is still not sufficient, you can omit the base resistor, and connect two inverters in parallel to drive the base of the transistors. The non-overlap generator can be designed using the circuit in experiment-2 or the one shown in Figure 3-4.

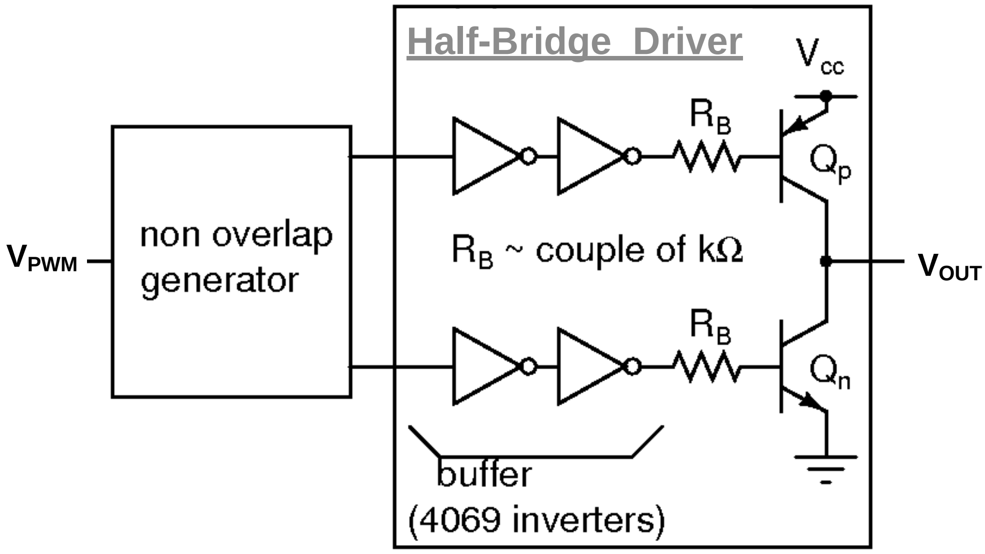



*Figure 3-3 Half-bridge speaker driver*

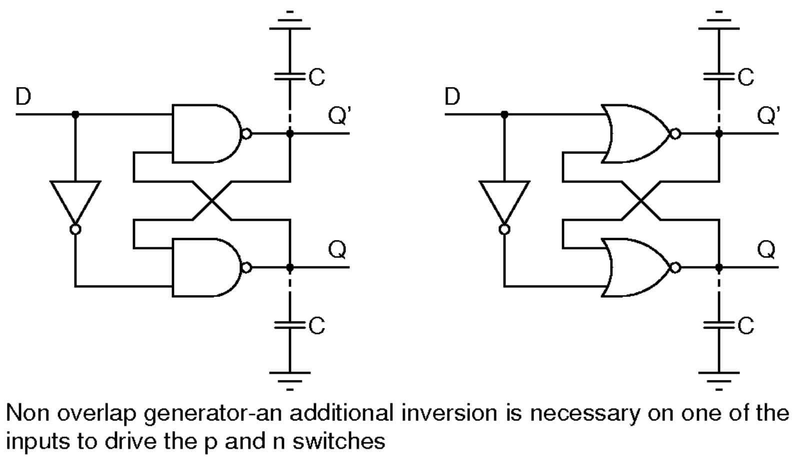



*Figure 3-4 Non-overlap clock generator*

In order to test the half-bridge circuit, $V_{PWM}$ from one of the PWM modulators ($V_{PWM\_P}$ or $V_{PWM\_N}$) of experiment-4 can be used as input. VOUT can be initially tested without load and then 32Ω resistive load is applied.

For simulation, actual electrical model of speaker can be used as shown in Figure 3-5. L is the coil inductance which is usually within the range of few 100s to a 1000 uH depending upon the size of coil. $R_L$ is coil resistance which depends upon power rating of the speaker.

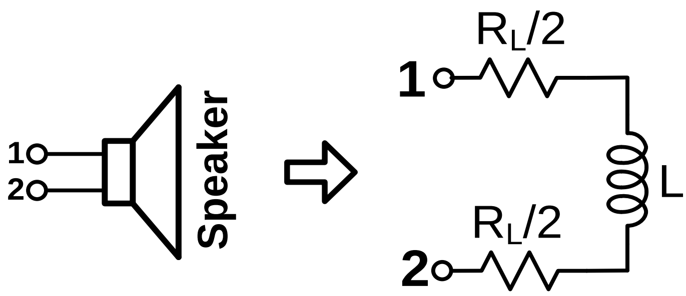
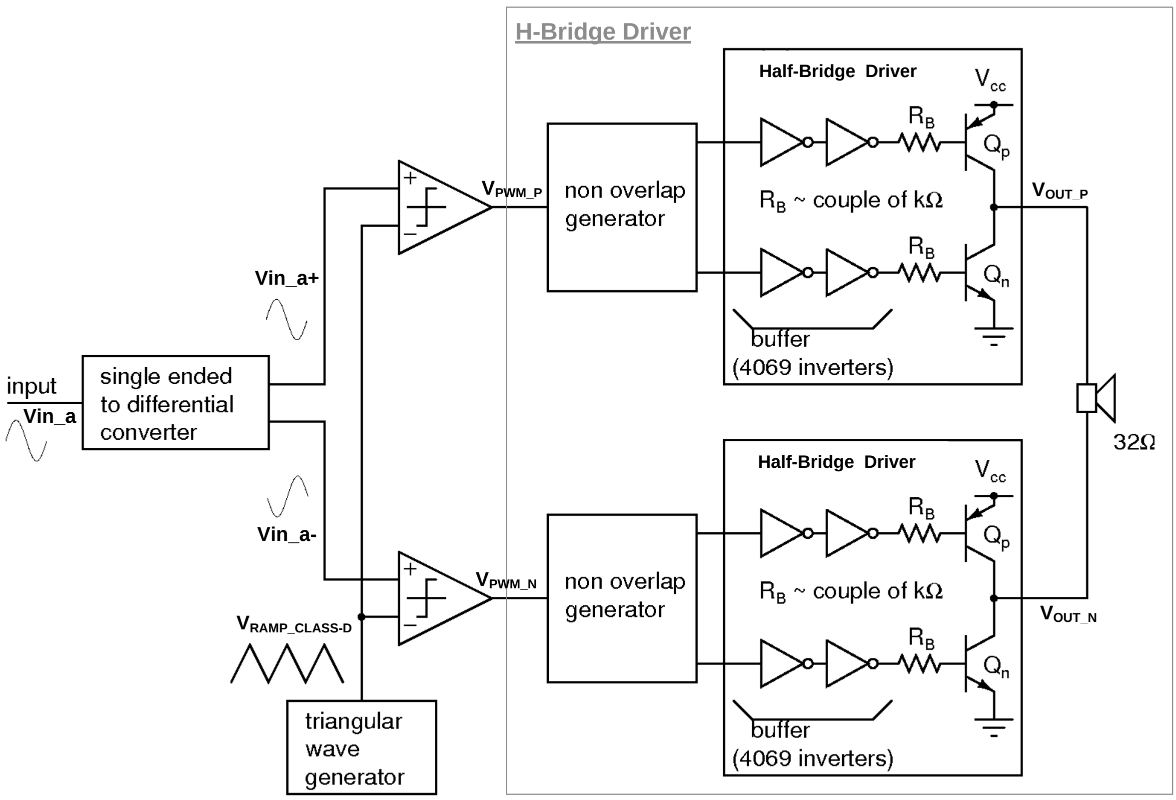



*Figure 3-5 Electrical model of a speaker*

Figure 3-6 shows the circuit diagram of complete class-d amplifier. The PWM output from single ended to differential converter and PWM modulator designed in experiment-4 is fed to H-Bridge driver which drives the speaker load. H-bridge driver consist of two identical half-bridge drivers. The complete class-D amplifier should be tested with resistive load first and then actual speaker.




Note: triangular wave and signal inputs to the comparator must be around the same bias point

*Figure 3-6 Circuit diagram of the complete class-d amplifier*

## Specifications

- Supply voltage ($V_{IN}$=$V_{DD}$) = 3V-5V
- PWM Frequency = 100KHz
- Load Resistance ($R_L$) = 32Ω

## List of Components

- CMP1 and CMP2: LM339 (open collector – requires a pullup resistor between $V_{DD}$ and $V_{OUT}$)
- Inverters: MC14069 or CD4069
- NAND Gates: SN74AHC00N
- BJTs: 2NXXXX series or alternate parts

## List of Measurements

1. Set $V_{DD}$=$V_{IN}$=4V, $R_L$=32 Ω
2. From function generator, set sinusoid wave of 1KHz and use as input to single ended-to-differential converter. Peak-to-peak amplitude of the sinusoid should be same as peak-to-peak amplitude of the triangular wave.
3. Measure and capture duty cycle at $V_{OUT+}$ and $V_{OUT-}$. Duty cycle should follow the same pattern as Vin_a+ and Vin_a-. Verify that $V_{OUT-}$ has inverter duty cycle (1-D) of $V_{OUT+}$ (D).
4. Add an RC filter at $V_{OUT+}$ and $V_{OUT-}$ with 3dB cut-off frequency of 10-20KHz and observe the output. Verify that output has the same shape as Vin_a+ and Vin_a-. RC filter is only to observe the average value of output hence should not be in the load path (i.e. load should be connected directly between $V_{OUT+}$ and $V_{OUT-}$).
5. Verify 2-4 with speaker and do hearing test. Reduce the amplitude of input sinusoid and observe the change in sound level. Repeat hearing test for 5 different frequency tones between 0.5KHz to 5KHz and observe the sound.
6. Verify operation for supply voltage, $V_{IN}$=3V and 5V. Observe the difference and comment.

**NOTE:** capture oscilloscope waveform only for one condition to show the functionality of circuit.

## Pre-Lab Exercise

1. Build the complete circuit shown in Figure 3-1 and Figure 3-2 in LTSpice. Verify the functionality by simulation with measurements 1-7. Use speaker model from Figure 3-5 as load and plot current through inductor. Inductor current should be average of differential output voltage ($V_{OUT\_P}$-$V_{OUT\_N}$) divided by $R_L$.

# Chapter 4 Analog Filter and Peak Detector

## Introduction and Circuit Diagrams

Analog filters are used to pass desired frequency signals and reject other frequencies. The objective of this module is to design a second order high-Q bandpass filter which will pass only a fixed frequency audio tone. Output of the filter is used as input signal to LED driver and class-D amplifier at later stage when we integrate all the modules and build complete system.

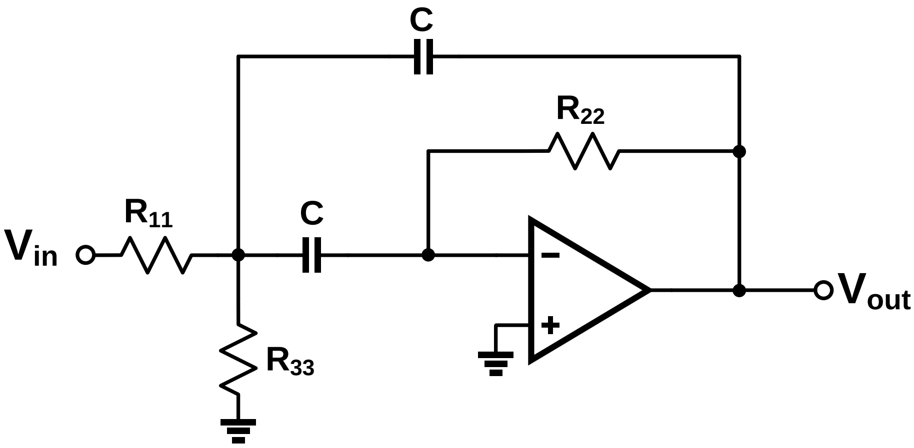



*Figure 4-1 A second order bandpass filter*

For basic theory and different types of filter, refer to the following documents:

- http://www.ti.com/lit/an/sbfa001c/sbfa001c.pdf
- https://focus.ti.com/lit/ml/sloa088/sloa088.pdf

Since, reference voltage to LED driver is dc, the ac output signal of the above bandpass filter must be converter to dc using a peak detector. Following document provides detailed description about the peak detector circuit.

- http://ww1.microchip.com/downloads/en/AppNotes/01353A.pdf

Figure 4-2 shows the circuit of a basic peak detector. It is based on a half-wave rectifier (AC-to-DC converter). Since $V_{IN}$ must be greater than forward voltage of diode (D1) for conduction, the circuit does not work for input voltages lower than diode forward voltage (~0.7V).

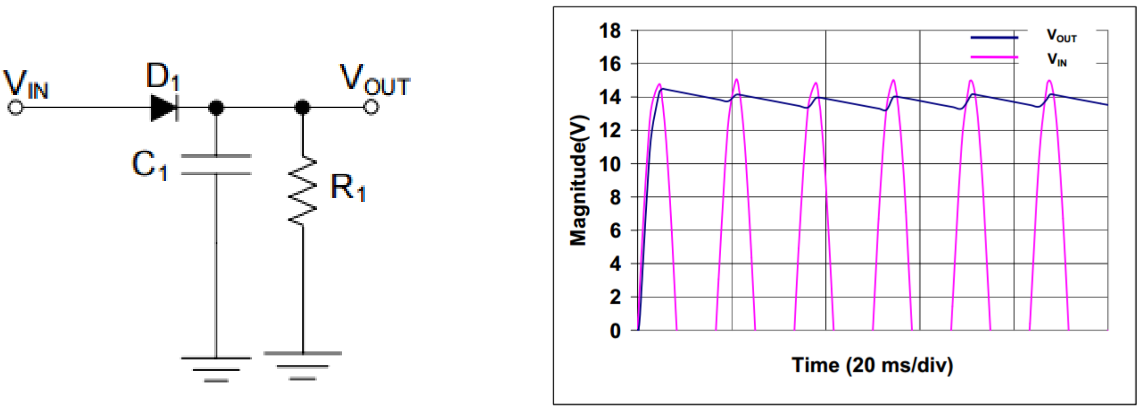



*Figure 4-2 A basic peak detector curcuit*

An op-amp based peak detector shown in Figure 4-4 is used in this module. First order high pass filter (R1-C1) is used to de-couple any dc bias of the input Vin. Feedback from Vpeak to op-amp inverting input ensures that D1 is always conducting for positive voltage of Vin_ac. Since diode remains reverse biased for negative voltage, capacitor (C2) holds the peak value of Vin_ac at Vpeak.

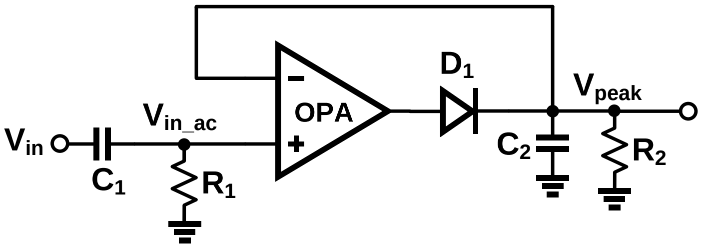



*Figure 4-3 Op-amp based peak detector curcuit*

Resistors R2 provides the discharge path to Vpeak so that output voltage can be reduced if amplitude of the input is reducing. The discharge rate of Vpeak depends upon the RC time constant defined as: $\tau = R2 \cdot C2$ and must be chosen high enough to ensure low ripple at Vpeak and low enough so that Vpeak can track any slow changes in the input signal amplitude. Generally, time constant ($\tau$) is kept around 10 times of the time period of input signal.

The dc voltage obtained at Vpeak may have higher voltage than the maximum specified value of $V_{REF}$ in the LED driver. Peak detector circuit of Figure 4-3 can be modified by splitting R2 into R2-R3 to form a voltage divider. The desired level of $V_{REF}$ can be achieved by adjusting the values of R2 and R3. The values of R2+R3 should be order of 10s of KOhms or higher as lower values may cause current drawn from op-amp output higher than its drive capability. Maximum output current of the op-amp can be checked from the datasheet before selecting the values of R2 and R3.

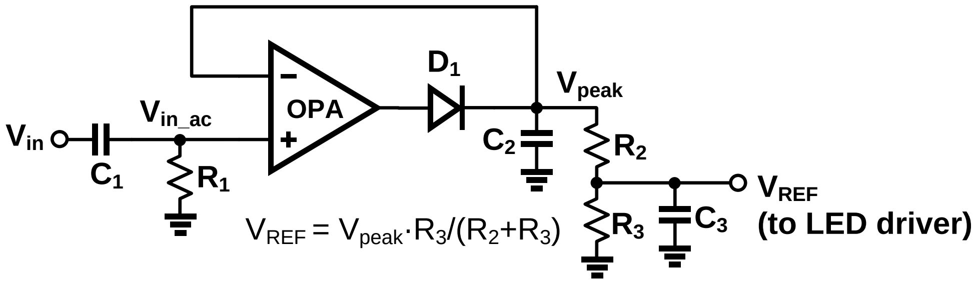



*Figure 4-4 Modified op-amp based peak detector curcuit*

Capacitor (C3) can also be added at $V_{REF}$ to filters out the ripple further and get a cleaner dc voltage.

# EXPERIMENT-6: BANDPASS FILTER AND PEAK DETECTOR

The objective of experiment-6 is to design a bandpass filter and peak detector shown in Figure 4-5. Audio input (Vin_audio), which is a fixed frequency sinusoid tone, is used as input to the bandpass filter. The bandpass filter is designed to respond to a desired frequency tone and reject other frequencies. Output of the bandpass filter is converted to dc voltage ($V_{REF}$) using the peak detector. The common mode signal ($V_{CM}$) is set to $V_{DD}/2$ which can be generated from a resistor divider as it was done in experiment-1 (Figure 2-14).

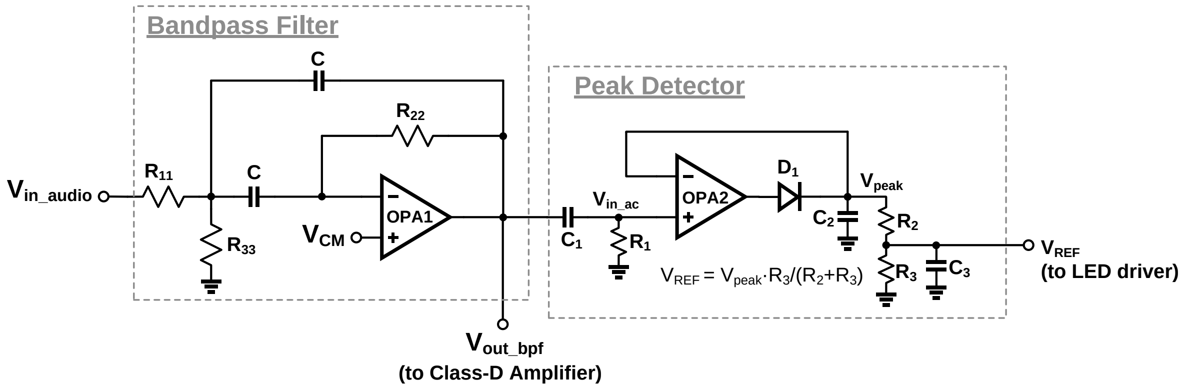



*Figure 4-5 Bandpass filter and peak detector curcuit*

## Specifications

- Supply voltage: $V_{DD}$=5V
- $V_{CM}=V_{DD}/2$=2.5V
- Bandpass filter Gain (Ao)=1 (0 dB)
- Bandpass filter Q-factor (Qo) = 20
- Bandpass filter center frequency (fo) = 1KHz
- Maximum peak-to-peak ripple at Vpeak = 100mV
- Maximum peak-to-peak ripple at $V_{REF}$ = 10mV
- Maximum peak to peak amplitude of Vin_audio = 0.9xVm (Vm is the peak-to-peak amplitude of the ramp signal obtained from experiment-1 at $V_{DD}$=5V)
- Maximum value of $V_{REF}$ (for maximum amplitude of Vin_audio) = 400mV

## List of Components

- OPA1 and OPA2: MCP6004 (Op Amps Quad 1.8V 1MHz)
- D1: 1N4148TR (Diodes - General Purpose)

## List of Measurements

1. Set $V_{DD}$=5V, $V_{CM}$=2.5V
2. Tune the bandpass filter for centre frequency (fo)=1KHz, gain (Am)=1 and Qo=10.
3. From function generator, set sinusoid wave of 1KHz and use as input to bandapss filter (Vin_audio). Peak-to-peak amplitude of the sinusoid should 0.9 times of peak-to-peak amplitude of the ramp signal of experiment-1 at $V_{DD}$=5V.

4. Measure and capture the output of bandpass filter (Vout_bpf) and verify the amplitude as per the gain (Ao=1).
5. Plot Vin_ac and verify that signal is biased around 0V.
6. Measure and plot Vpeak average and peak to peak ripple. Verify that average is approximately same as peak level of Vin_ac and prak-to-peak ripple is within the specification (100mV).
7. Measure and plot $V_{REF}$ and verify the average value is 400mV and ripple is within 10mV.
8. Reduce the amplitude of Vin_audio and verify the Vout_bpf and $V_{REF}$ follow the change in amplitude. Set the amplitude back to its maximum value (0.9xVm)
9. Now drift the frequency away from 1KHz (increase and decrease) and verify that Vout_bpf and $V_{REF}$ are reduced when Vin_audio frequency moves away from 1KHz
10. Change the Vin_audio frequency to 5KHz. Measure and plot Vout_bpf and $V_{REF}$. Verify the amplitude of Vout_bpf and $V_{REF}$ according to the frequency response of bandpass filter.

## Pre-Lab Exercise

1. Derive the transfer function of bandpass filter shown in Figure 4-1 and prove that it is a second order bandpass filter having transfer function equivalent to: $\mathrm{H}(s) = \frac{A_O \cdot \frac{w_O}{Q_O} \cdot s}{s^2 + \frac{w_O}{Q_O} s + w_O^2}$ . Find the values of resistors and capacitors for Ao, Qo and fo provided in the Specifications.
2. Calculate the values of R1, R2, R3, C1, C2 and C3 for the frequency and ripple provided in the Specifications.
3. Design and simulate the entire circuit shown in Figure 4-5 with above calculated values. Verify the operation with measurements 1 to 10.

**NOTE:**

- The gain of the bandpass filter at center frequency should be unity. If not then adjust the value of $R_{11}$. Alternatively, the amplitude of Vin_audio from function generator can be adjusted to get the peak-to-peak amplitude of Vout_bpf = 0.9xVm.
- The center frequency (fo) may be slightly off from simulation results when implemented on breadboard. This is mainly due to the tolerance in resistors and capacitors. In that case, you can tune the frequency of Vin_audio to match the center frequency of the bandpass filter. Exact center frequency can be found by sweeping the frequency of Vin_audio around 1KHz and look for the maximum amplitude of Vout_bpf.

# Chapter 5 TOP LEVEL INTEGRATION

Top level integration combines all the three modules (LED Driver, Class-D Amplifier and Filter+Peak Detector designed during experiments 1-6) to build the complete system. Figure 5-1 shows the block diagram of the complete system after integrating all the modules.

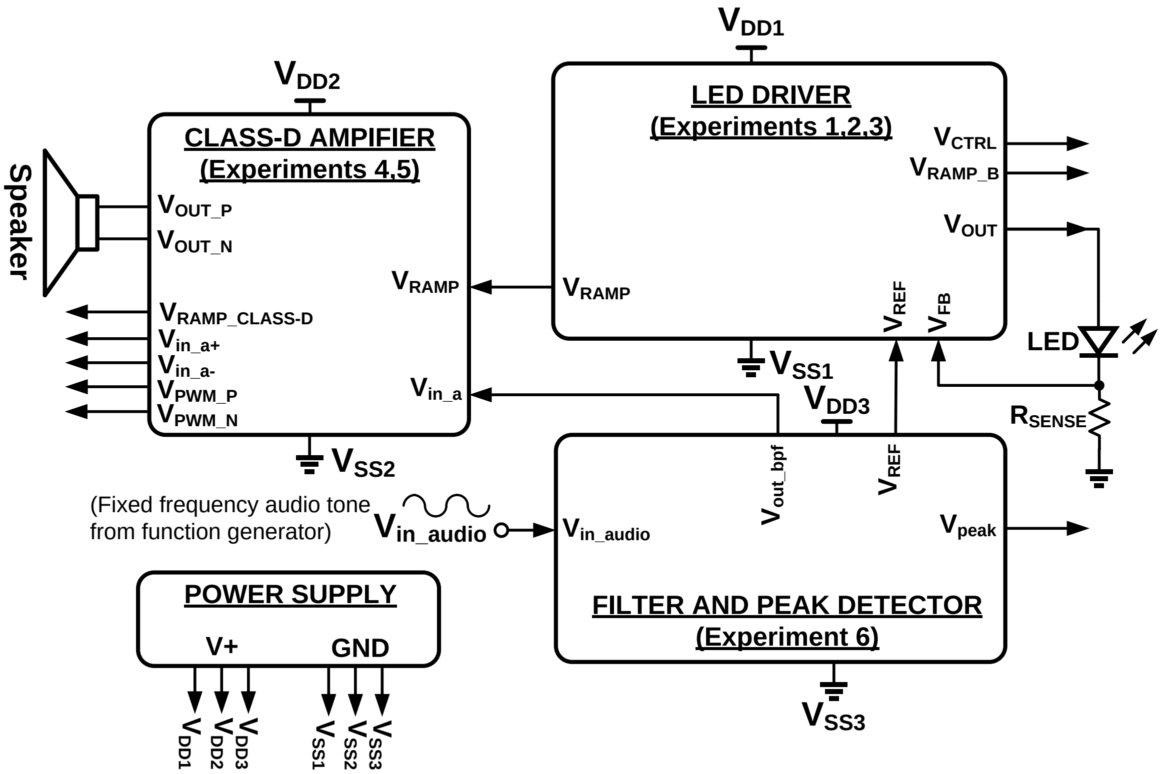



*Figure 5-1 Block diagram of the complete system after integration*

Vin_audio is a fixed frequency audio tone generated from function generated as it was used in experiment-6. All the interface signals going from one module to other should be connected properly. In order to prevent noise coupling from one module to other, $V_{DD}$ and GND ($V_{SS}$) of each module should be connected directly to power supply and not shorted locally on the breadboard. If required, decoupling capacitors of few µF can be connected locally between VDD and GND of each module. If analog modules (non-switching) within the modules are affected from switching noise then VDD and GND of each analog module can be separated as well and connected directly to the power supply.

## Integration Guidelines

1. Makes sure all the individual modules are working before integrating them together.
2. Before starting board level integration, integrate all the modules together on LTSpice and verify the functionality.
3. Label all the signals shown in the block diagram of Figure 5-1 using a small piece of paper and tape. Wires connecting to these labelled signals should be brought to for measurement. Rest of the signals can left inside the board.

4. Try to use different colour wires for VDD, GND and signals. For example, red can be used for VDD, black from GND and other colours for signals.
5. Putting tape around the circuits may help in keeping the connections intact. Signal wires which are brought out for measurement can also be fastened locally on board using tape to protect from popping out of the holes.
6. $V_{DD}$ and GND ($V_{SS}$) of each module should be connected directly to power supply and not shorted locally on the breadboard. If required, decoupling capacitors of few µF can be connected locally between VDD and GND of each module.
7. Check the short between VDD, GND and signals before turning the power supply ON.
8. Limit the power supply current to prevent the circuit from damaging in case of accidental short. Usually current limit is set slightly higher (1.5x or so) than the maximum total current drawn by the circuits.

## Final Demo and Viva

Final demo will be based on both LTSpice and board level design. Students will not be given extra time to work on circuits on the day of final demo hence all students should have their modules ready before start of the demo. Students will be asked to demonstrate following:

1. LTSpice simulation results. Must be implemented individually and not in group.
2. Hardware functionality demo (in group).
3. Probe signals listed in Figure 5-1.
4. Capability of operating instruments used in EE2019 lab (oscilloscope, power supplies, function generator etc.)
5. Answering questions related to circuits designed in EE2019 lab experiments.