Class-A Power Amplifier Design Proposal Team Helios

Preston Maness
Alexander Kendrick
Mark McQuilken
Jonathan Wright

***Abstract*—The final project for Dr. Aslan's Fall 2012 Electronics I course was to design a power amplifier for use in audio applications. The goal was to apply knowledge of power supplies (diodes and transformers) and amplifiers (BJT transistors with hybrid pi model) to design a Class-A power amplifier from end to end. Particular emphasis was placed on collaborative design and CAD-assisted verification via Multisim software.**

Team Helios chose a common emitter to emitter follower design as a base. Choosing to employ a bipolar dual-rail power supply to the system –so as to increase maximum voltage swing and have zero power output in the absence an applied signal (no heat dissipation)– necessitated the introduction of a third transistor to act as a voltage shift. The result is an amplifier that has an approximate voltage gain of 142 and a power output of approximately five milliwatts across resistive loads varying from 2 to 16 ohms, as per the specification. Efficiency was measured as approximately five percent compared to the DC input. Total cost was determined to be X dollars. Lead times from manufacture to customer delivery are conservatively given at two months.

CONTENTS

I. INTRODUCTION AND PRODUCT SPECIFICATIONS

THE client, Good Speakers Inc., has commissioned the design of a Class-A Power Amplifier for use in audio applications. The following specifications are provided.

- blah
- blah
- blah
- blah

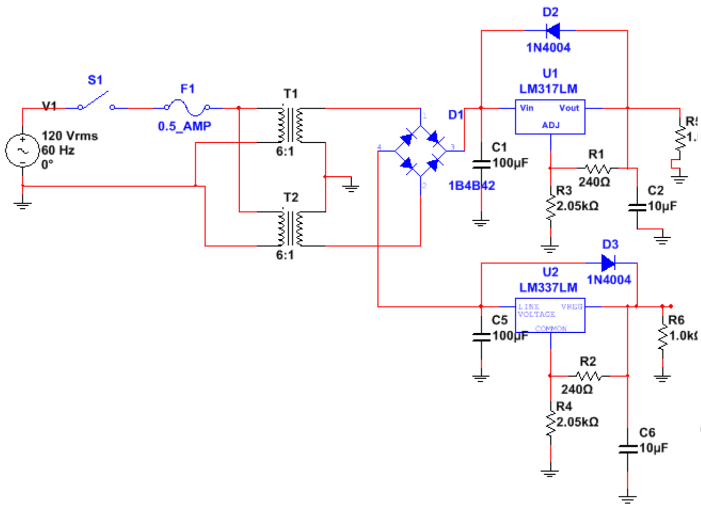
II. IMPLICATIONS OF SPECIFICATIONS

Given the audio application of the amplifier, one can reasonably assume that the frequency of the input signal will fall between 22 Hz and 22 kHz. While the client desires as wide a bandwidth as possible, the audio bandwidth is the most crucial to meet. The lower bound of 22 Hz would require significantly larger capacitors to place the lower cutoff value at said frequency. Team Helios assumes a lower bound of 100 Hz in calculations, knowing that the effect on these low frequencies past the 100 Hz cutoff is negligible. In addition, the chosen design that uses DC coupling to minimize the need for capacitors has an added effect of reducing attenuation at each cutoff frequency.

asdf

III. POWER SUPPLY DESIGN

The following design is an improvement on the basic models introduced at the start of semester. It takes into account various sources of noise and distortion common to mains input. It also attempts to mitigate common sources of design failure, such as inrush current and overvoltage. Finally, it further addresses safety concerns with the introduction of arc-resistant fuses and is a step in the right direction for approval by certifiers such as Underwriters Laboratories.



A. Block Diagram

asdf

B. Considerations

asdf

IV. Rejected Amplifier Designs

Various approaches and topologies were investigated. From single-transistor to CE-EF to op-amp implementations, each design was examined on a broad level with advantages and drawbacks determined.

A. Transformer Coupled

my transformer design

B. CE-EF Cap Coupled

similar design to chosen one that doesn't use bipolar supply (increases hum).

C. Op-Amp Implementation

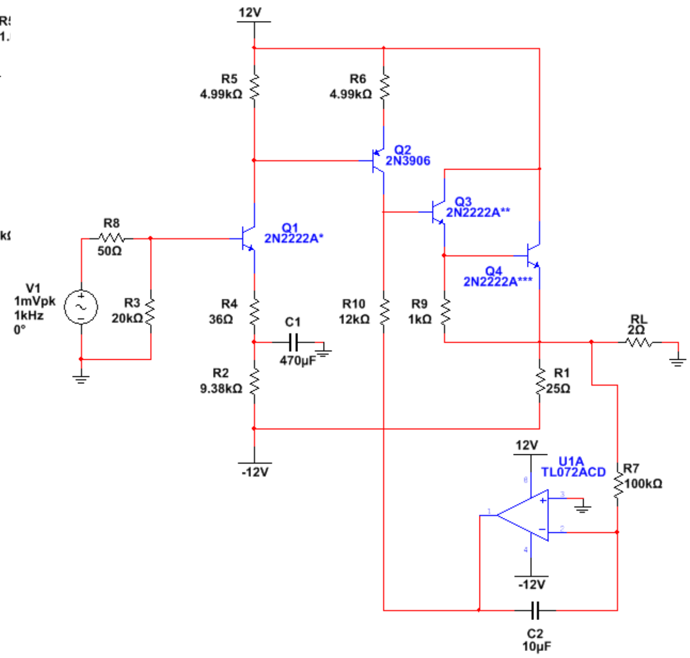
one of the designs that used lots of op-amps (usable, but lack of familiarity with op-amps resulted in using the selected design)

D. Comparison Summary

Advantages Drawbacks

V. Selected Amplifer Design

The full schematic is below



A. Block Diagram

adsf

B. Design Approach

The traditional approach for design, starting with KVL and $I_C = \beta I_B$, functions well for single-transistor implementations. It is thorough and rigourous and the hybrid pi model is very accurate for the frequency range under consideration. However, this design methodology's heavy dependence on assumptions for system behaviour that can change and are at times interdependent can often lead to line after line of dense calculation that must be revamped continuously for each iteration of the design. Performing such calculations manually is error prone, and writing software to automate the process is outside of the scope of this project.

Consequently, a work backward approach was taken, starting from arbitrarily selected target output values –for example, require 10 mW output power– and working backwards to determine requirements at each stage of the amplifier. For example: in the given design, a target output of 5 mW was selected. This implies an RMS voltage on the output. Knowing that this must come from the emitter-follower stage (Darlington pair in the chosen design), one can conclude that the RMS voltage entering the emitter-follower must be greater than $1.4V$, since V_{BE} is approximated as $0.7V$. Knowing that the voltage shift gain is approximately one, one can conclude that this voltage is also the input and output of the voltage shift stage and, consequently, that our common emitter (which performs the actual voltage gain) needs to output this. At this point, one may set up the DC conditions on the CE and tweak R_{E1} as necessary to influence the voltage gain.

This fast and loose approach permits a more rapid development process that can always be checked against the more formal mathematics once ballpark values are determined. In our case, design verification was performed using Multisim.

C. Calculations

Still haven't gotten any calculations. Granted, the above does a good job on the philosophizing.

D. Considerations

adsf

VI. Cost Analysis of Selected Design

asdf

VII. Conclusion

asdf