Introduction:
In this project, we were asked to build an audio amplifier with resistors, capacitors, and transistors. The amplifier should have a gain above 10, big input impedance, small output impedance, and a large open loop gain so that the closed-loop gain is controlled by the feedback. Upon careful observation of our initial design, we have come up with a few adjustments to enhance the amplifier with better overall performance. The reason for building this amplifier was to understand the fundamentals of electric circuit theory by applying the concept-based knowledge to the real-life application.

This project gave us the opportunity to apply what we learned in our theoretical class to a real life project. By doing this, we gained a more realistic point of view of how some engineers work. Through this process we created an initial design of our circuit and did a simulation using OrCAD PSpice. Then we analyzed our simulation to make sure it was what we wanted and proceeded to test it on a bread board. Once that was done, we fixed whatever issues that came across and went on to fabricate it on a PCB board. Then we tested it again, and fixed any problems that we had. These are the steps that real engineers take and we were able to experience that.

Other than our design, there were better ways to get a power amplifier or even build a better one using more sophisticated parts available to us. But by having design constraints added to our project, we learned that this is how real engineering projects are sometimes given, and that we must follow certain specifications whether it be for safety reasons or other issues.
The circuit:
Our preliminary design

DC Voltages
Input Signal vs. Output Signal:

The plot below shows the input signal vs. output signal with respect to time.

As we can see, the input signal of 20mV is increased to approximately 460mV, providing our desired gain of approximately 23.
Gain vs. Frequency:

The first plot shows both Vin and Vout vs. Frequency and the second plot shows the gain (Vout/Vin) vs. Frequency.

We get a steady gain of slightly above 20 from 10Hz to about 100Khz.
Input Impedance vs. Frequency:

Input impedance is very high at about 45K ohms as we desired.

Output Impedance vs. Frequency:

Output impedance is about 630 milliohms. We achieved our desired output impedance value of below 1 ohm.
We made a few adjustments in our final design. We added a voltage divider to bring down the voltage coming in from the CD player: voltage from the CD player was about 50mV so we added a voltage divider with 200ohm and 300ohm resistors, so that we could bring the voltage down to about 20mV before entering the differential amplifier stage. Also, we decreased the DC supply voltages from 12V to 8.5V and this adjustment took care of the noise that was evident in the output signal (we could observe this from the oscilloscope and also listening to the sound coming out of the speaker).

Our PC board layout had some problems at first. The lines were too thin and they were placed too close to each other. This might prevent the currents from flowing smoothly or direct the currents in wrong direction. After we made a few adjustments with the PC board layout, we could come up with a very neat, organized, and workable PC board.
**Experiment:**

The first step in building our PC board was to wash the PC board and avoid touching the board with our fingertips. Then, we ironed the blue paper (which contains our PC board layout) onto the PC board itself. This task was more difficult than we thought because depending on how long we iron it, the outcome was different. We had to do it so that it won’t stick to the board too much, but at the same time, we had to ensure that it was printed on the board thoroughly and does not get taken off when removing the blue paper. We found that ironing for about 2 minutes gives us the best result.

After ironing, we put the board in the etching solution for about 30 minutes, until the black lines (PC board layout) turn into grayish silver color. While the board was in the etching solution, we could work on some other parts of the construction of the circuit, such as preparing all the components (resistors, capacitors, transistors) and running a simulation on the breadboard, etc. After we take the board out of the solution, we scrubbed off the gray lines of the board to reveal the copper.

The next step was to drill the holes for the placement of the components. Using our drilling machine, we could easily accomplish this task without any problem.

The last step of the PC board construction was the soldering which is the most important and tricky part. We had to make sure the metal completely touched the copper lines so they would conduct the currents well. Sometimes, the holes were drilled too far and this resulted in improper placement of the components. We had to get another board and redo the previous steps to finally fix the distance between the holes, ensuring the proper placement of the components. After placing each component, we checked resistance values using multimeter in order to check they are soldered appropriately.
Our final (enhanced) design and DC voltages (PSpice Simulation) Once we made the small changes to our PCB board and also the change of DC input voltages, we ran PSpice again using these changes to see how close the our actual measurement was to a simulation.
Our final (enhanced) design and DC voltages (from actual experiment, using multi-meter). As you can see some of the actual measurements are close to the that of the PSpice measurements.
Input impedance of our final design and shown through PSpice. We did not get a chance to do this during our lab session due to missing bread board.
This is the output impedance of our enhanced circuit. This a change from our previous output impedance only because we had to change the DV voltage.

Frequency dependence of voltage gain and phase. There is a incredible offset in our graph which we can not account for because when we tested it using the oscilloscope, we had no DC offset.
Discussion:

Something that we noticed was that when we did our initial testing on a bread board just to make sure that our circuit would work, we had perfect sound and measurements using the values given on our first PSpice design. But when we put it on our PCB board, the outcome was different. We initially had a 0.5 ohm resistor near the output just so that our output impedance would go down. In the simulation everything worked fine and our gain was at the good level. On the breadboard everything was fine as well and perfect sound came out of the speaker. On the other hand, the PCB board did not act in the proper manner as we wanted it to. When we tested it on the oscilloscope, we noticed that we had a gain of just over one and when we tried it on the speaker, we had a very low sound come out. This indeed made certain that we had a very small gain coming out. We determined (after speaking to our professor) that the 0.5 ohm was what caused our small gain. We removed it from our PCB board and our circuit had a larger gain. We decided that it could have been possible, that on the bread board we could have misplaced the 0.5ohm resistor and that made better sound quality.

The other interesting thing was that when we tested our PCB board (after correcting the first problem), we had a lot of noise coming speaker and this was shown in the oscilloscope. We fixed this by decreasing our input voltage from 12V to 8.5V. It was strange that this had to be done because in all our tests we used the 12V input voltage and everything worked fine. By decreasing the input voltage though we were able to remove the noise from our signal.

For future improvements of our circuit, we would use JFETs in our differential amp. The reason we did not use this is because PSpice was not giving the results that we desired and we were getting the right results using regular BJTs. Another improvement would probably have to be to use capacitors. This most likely would have fixed our noise problems in the circuit and probably would have given us a better sound as well. The reason we did not do this is because of time constraints and we took a faster approach of rectifying our problem.

Overall the lab was very fun and informative. The only other advice I would give for the future is to use a different fabrication method. We experienced a lot of problems when ironing on the blue paper. This was also due to the thin lines that we drew for our layout, but I feel that a better program that can do layouts would definitely help make the whole processes go quicker because we spent a lot of time fixing our layout, ironing the blue paper, and soldering all the components on. Otherwise, this lab was a very good experience for all of us.