Building the GTDT audio amp from the kit.

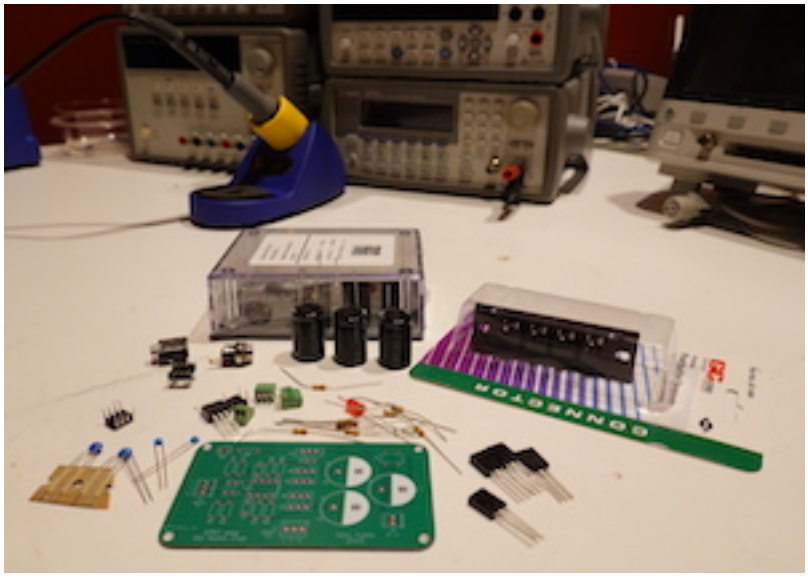
Following are step-by-step instructions for soldering the components of the GTDT amp onto the PCB provided in the kit. A guide like this is probably overkill — most people can probably figure out how everything goes together using the schematic. But it can be nice to see what the board looks like at each step, particularly if the builder is inexperienced. There are a couple of slightly tricky spots — like orienting the transistors. Hopefully, this guide lessens the chances of making mistakes.

Most of the soldering is quite easy. As always, we start with the small components (ceramic capacitors and resistors) and work towards the biggest components (electrolytic capacitors). Note that when soldering any connection to ground, we will soldering to the ground plane on the PCB. These connections require more heat & time to get the leads hot enough to melt the solder. Be patient and hold the soldering iron longer than normal. You may even want to increase the temperature[1].

It is always a good idea to check things as the build progresses. Use the continuity-test function of the multimeter to check solder connections. At appropriate points in the process, we can even connect power and check voltages.

0. Gather the parts

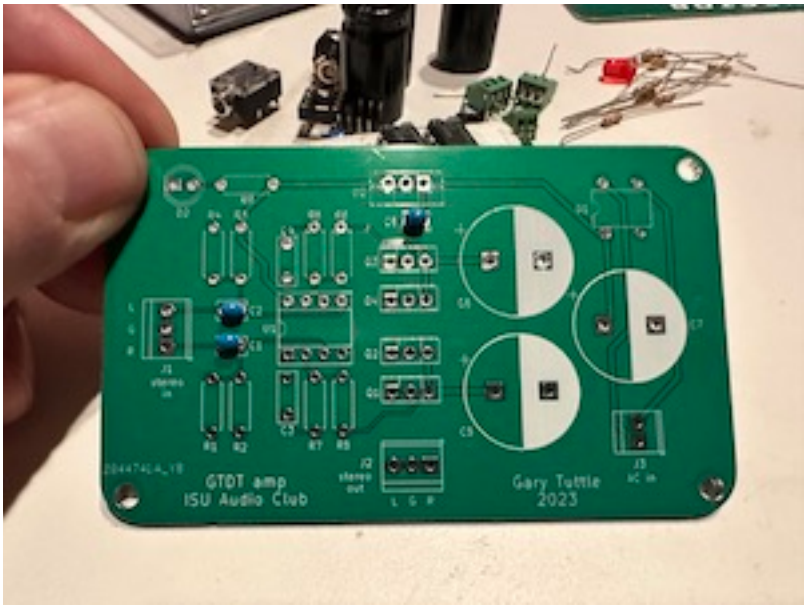
Consult the BoM to see the complete list.



[1] Generally, I use a soldering tip temperature of around 650°C. If the ground connections are "being stubborn", I may turn the temperature up to 700°C or 725°C to heat things up a bit more.

1. Small capacitors

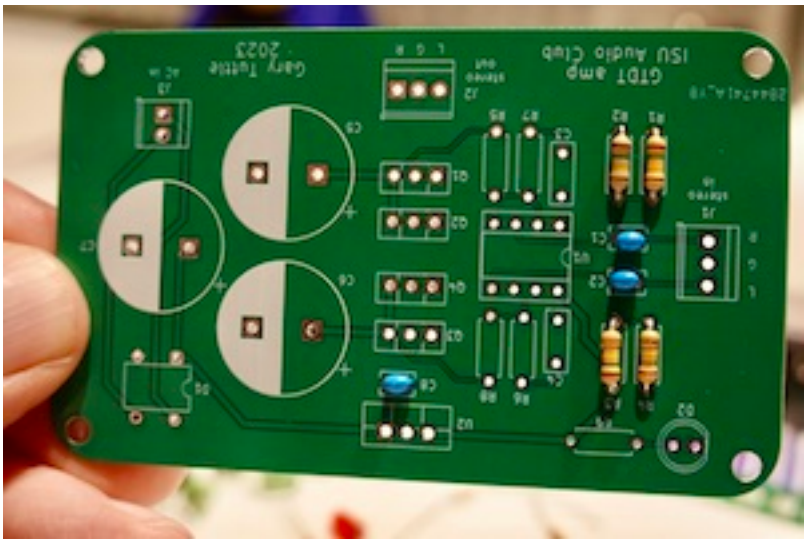
There are three 0.33-μF capacitors — two for the inputs, labeled C1 and C2 and one for voltage-regulator stability, labeled C8.



2. Biasing resistors

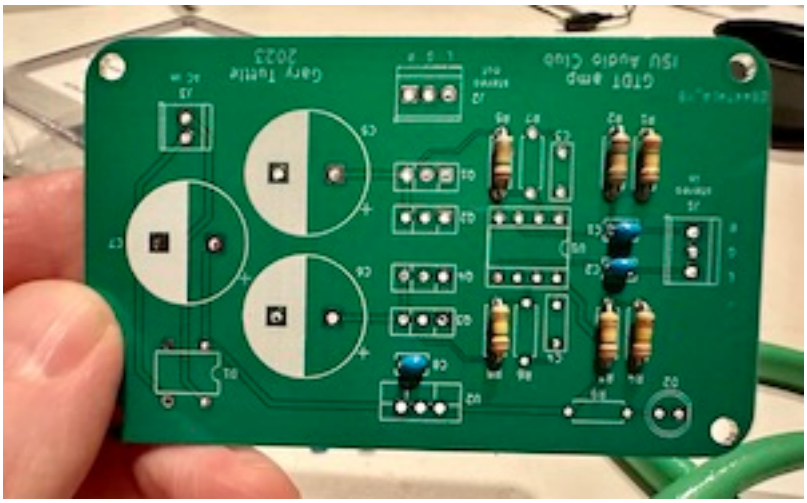
These are four 150-kΩ resistors, labeled R1, R2, R3, and R4.

(For some reason, I started holding the PCB “upside down”. 🤷‍♂️)



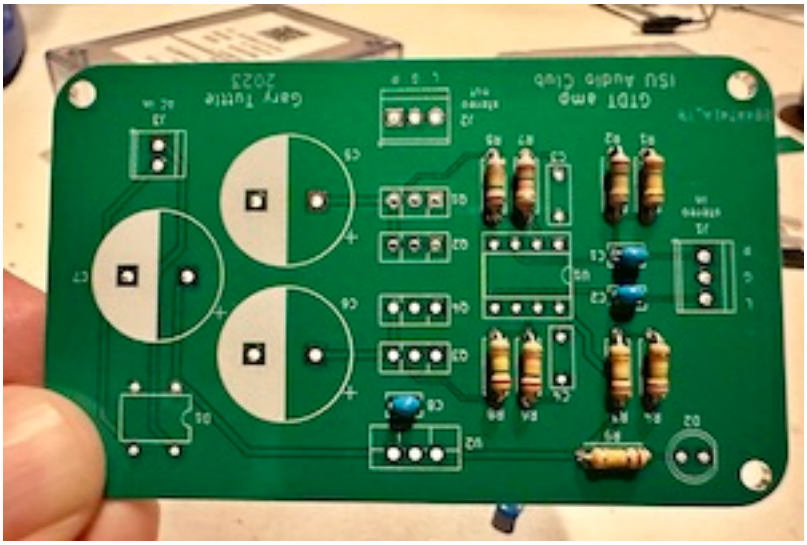
3. First feedback resistors

These are also 150-kΩ resistors and are labeled R5 and R8.



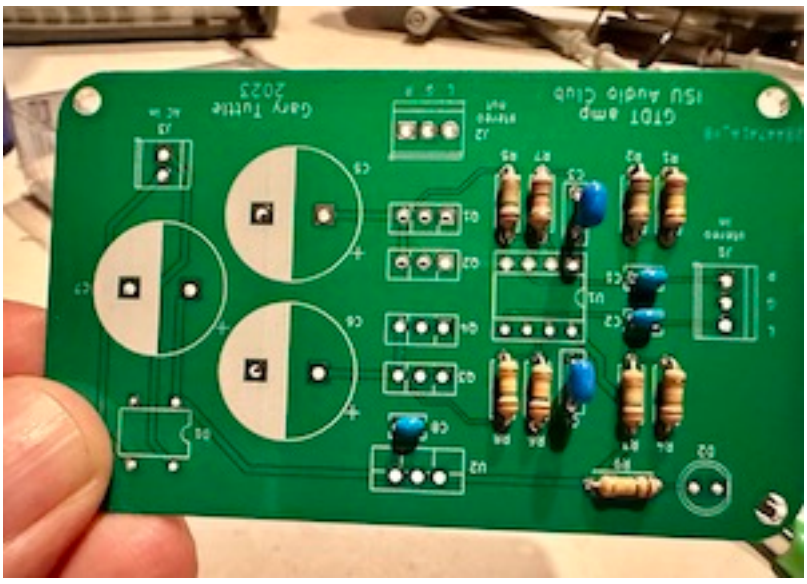
4. More resistors

These three are all 10-kΩ resistors. Two are for the feedback networks for the amps, labeled R6 and R7 and one is the current-limiting resistor for the LED, labeled R9.



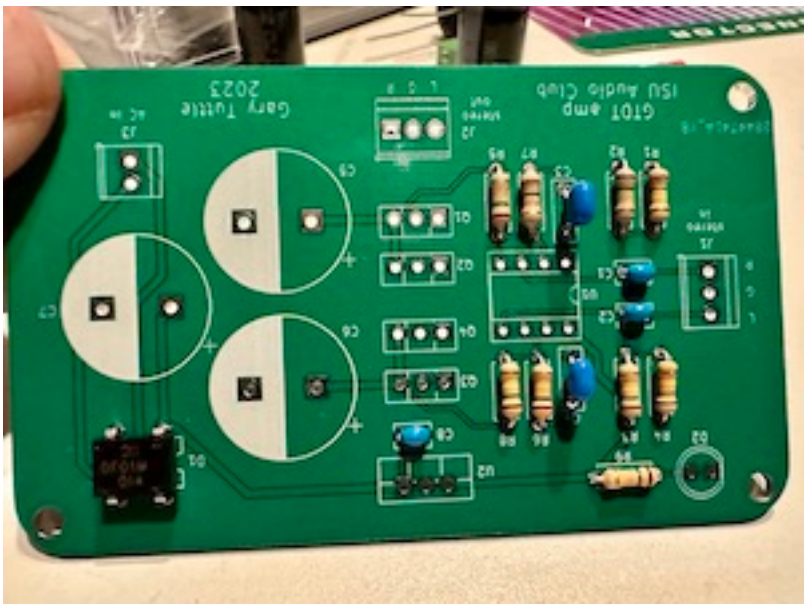
5. Feedback capacitors

Next are the two 3.3-μF capacitors used in the feedback networks, labeled C3 and C4.



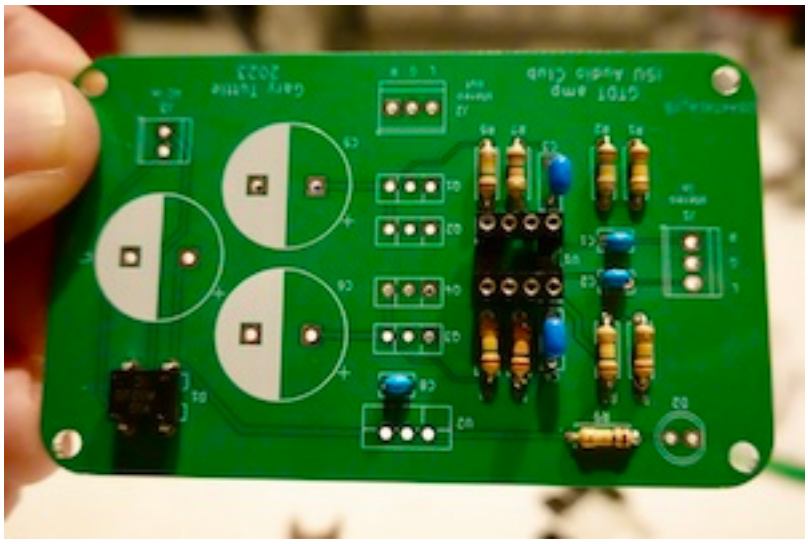
6. Diode bridge

Now solder the diode bride for the input power. Bend the pins underneath to keep the chip from following out while soldering It is important to get the orientation correct. The inputs (labeled with little sine waves) are nearer the PCB edge. The outputs (labeled with + and –) should be to the inside. Follow the PCB traces to see the connections.



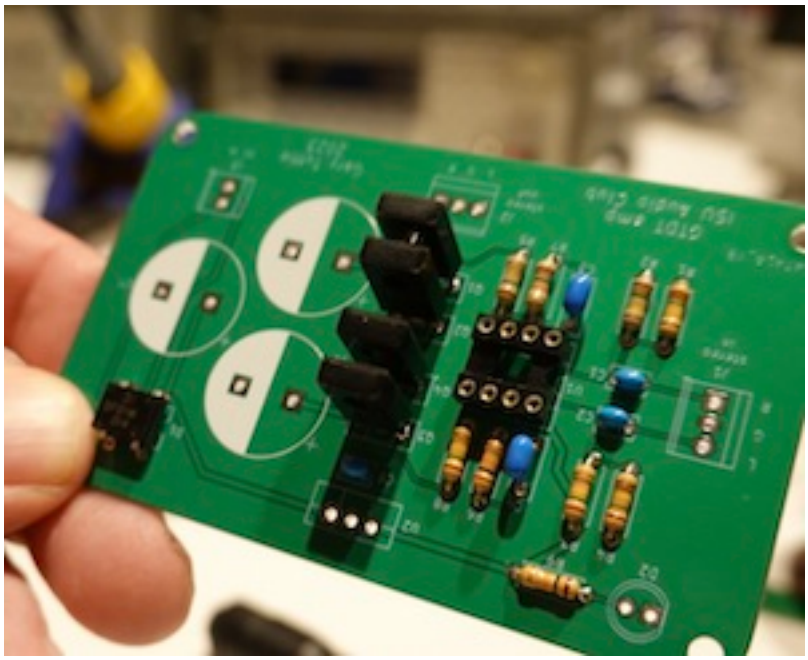
7. Op-amp socket

Solder in the socket for the op amp. The socket has a notch indicating the orientation. (Since the socket is symmetric, so the orientation is not critical. Still, it is good to keep everything “aligned”.) It is helpful to use a bit of tape to hold the socket to the board while it is being soldered. Or maybe a third hand



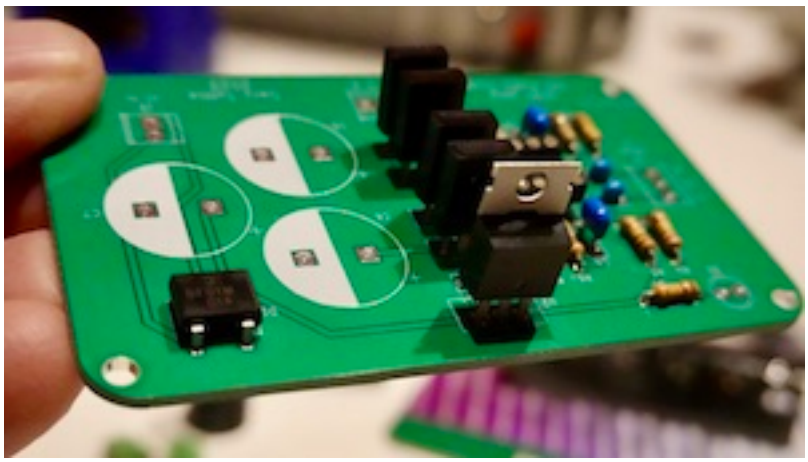
8. Transistors

This is the trickiest part. Be sure to consult the schematic as well as the data sheets for the MJE172 (pnp) and MJE182 (npn) transistors. If these are connected incorrectly, bad things can happen. Q1 and Q3 are the npn transistors. Q2 and Q4 are pnp. The ordering of the leads for both types is emitter–collector–base, left-to-right as you “face” the transistor. (Look for 3 dots arranged in a triangle as you face it.) For all four transistors, the base connections are closest to the op-amp socket, and the emitter connections are nearest the three big capacitors. Proceed carefully, double-check, and maybe have someone else triple-check before soldering.



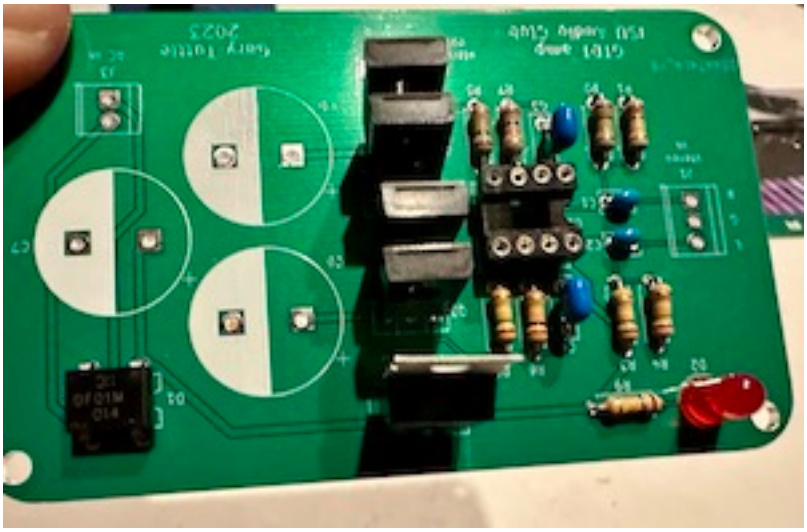
9. Voltage regulator

Solder in the 7815 voltage regulator. This would also be problematic if installed backwards. Check the data sheet. Pin 1 is the input and should should nearest the diode bridge. Pin 3 is the output and should be on the side were all the resistors are located. The heat sink should be on the side nearest the transistors.



10. LED

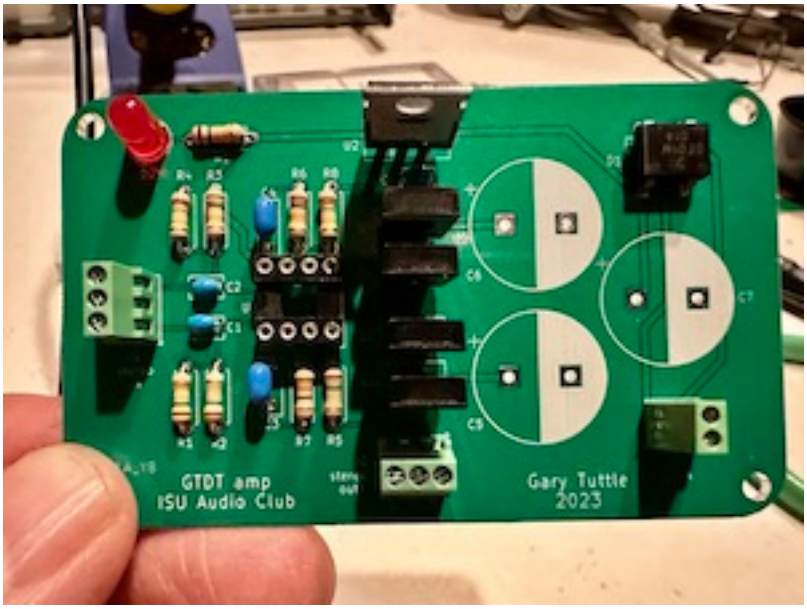
Add the power indiction LED. Again, check the orientation before soldering. The cathode is next to the flat edge on the package. Look at the silk screen image on the PCB.



11. Input and output connectors

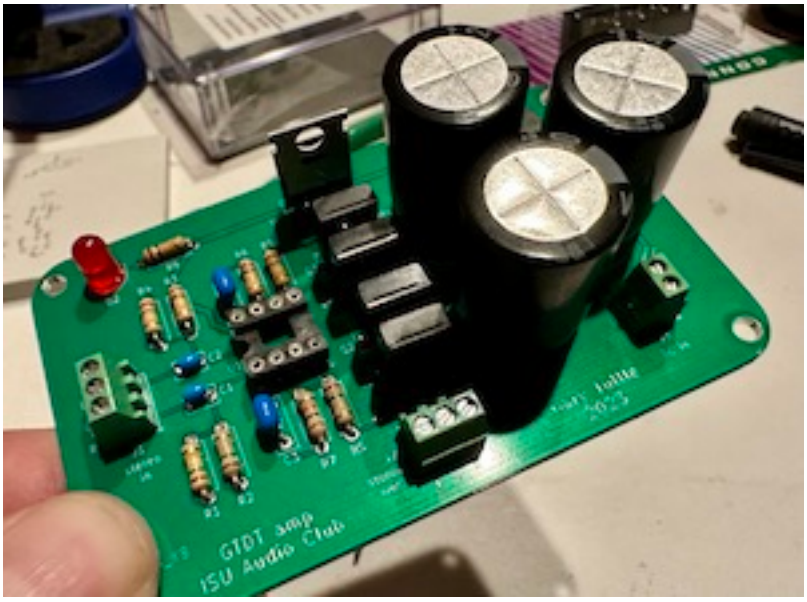
Solder in the three terminal blocks — two are three-pin versions for the audio input and output and the other is a two-pin version for power input. Make sure the holes are facing “out” to avoid awkward wiring. Bend the pins or use some tape to hold the blocks in place while soldering.

(Hey! The PCB is right-side up again.)



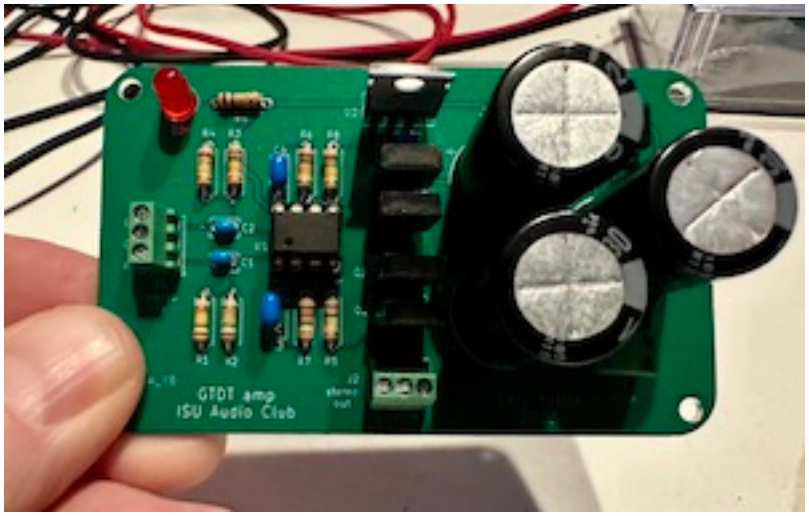
12. Large capacitors

This is the last soldering. Add the three big 3300-μF capacitors. Be sure to get the correct polarity — unless you enjoy seeing capacitors explode.



13. Op amp

Last, but not least, insert the op amp into the socket. As always, we probably need to carefully adjust the pins to line up with the holes. Insert carefully — it is easy to bend the pins. Also, make sure to get the correct orientation. (Look for the small circle next pin 1, or the half-moon cutout on the edge between pins 1 and 8.) Unlike the socket, the op-amp orientation is crucial. Getting it backwards may kill the op amp.



Before moving on to the enclosure and connectors, we should do a bit of testing. Here power is provided by an 18-V DC bench supply. The input signal is a 1-kHz sine wave, applied to the right channel. A 4-Ω, 10-W load resistor (simulating a speaker) is attached to the right-channel output. The input level is increased until the output just begins to clip, and then backed off slightly. The maximum input to prevent clipping is 165-mV RMS, and the corresponding output voltage is 2.53 V RMS.

The gain is $G = (2.53\text{ V})/(0.165\text{ V}) = 15.3$, close to the expected value of 16. The delivered power is $P = (2.53)^2/(4\ \Omega) = 1.6$ W. It won't blow off anyone's toupee, but it should be enough to drive set of small speakers. The left channel performs similarly.