# Class D Amplifier Instructions

Joe Hao

January 8, 2026

# Contents

# Introduction

The purpose of this educational circuit board exercise is designed to educate participants about electronics, soldering, and some basic signal processing methods. The project is tailored towards individuals of all backgrounds seeking to develop a stronger understanding of circuit construction as well as gain valuable hands-on experience working with electronics.

The class D amplifier was chosen for this exercise as it contains a good blend of fundamental components such as transistors and inductors, as well as more complex integrated circuits, while not being overly difficult to understand from a theoretical point of view. Furthermore, the audio output of the amplifier provides clear physical feedback to the participant as they tune the various parts of the amplifier.

Thus, the PCB is designed with modularity as a major focus. Participants are able to measure and view the audio signal as it passes through each module using an oscilloscope to explore the analog and digital signal processing aspect of the project. In addition, a custom development area has been included for personal and creative additions to the circuit. The custom development area follows the wiring of a standard prototyping "breadboard", and allows for easy addition of an additional source of audio.

# Background

## Soldering Technique

Soldering is the process of joining two metals together by melting an alloy (solder) between them. This method will be used to mount electronic components to the printed circuit board (PCB) throughout this exercise. To melt the solder, a "soldering iron" will be used.

The tip of a soldering iron can have many different shapes, such as a blunted point, a wedge, or a sharp point. However, the general procedure to using any soldering iron remains the same.

First, securely mount the component that is being soldered onto the PCB in its correct orientation. To hold the component in place, either bend the legs of the component to prevent it from falling out or use a piece of tape across the front of the component to hold it against the PCB.

When soldering, it is highly recommended that flux is applied to the solder junction before it is heated and soldered. Solder flux is a chemical, usually a paste, that removes oxides from metallic surfaces and helps the solder alloy form stronger bonds. However, after the solder joint is complete, excess flux should be cleaned from the board using isopropyl alcohol (IPA) as it leaves behind a sticky resin which may be corrosive. To use flux, apply a small, pea-sized ball of flux to the joint being soldered.

Then, use the soldering iron to heat up the component being soldered, which should take no more than 5 seconds. It is important to heat up the component itself because the solder alloy will not bond well to cold components. If flux is used, it will liquefy when heated and flow across the joint as well as the board. Once the component is hot, touch the tip of the solder wire to the now-hot junction between the component and the PCB. The solder should melt on contact with the component. If it is not melting, remove the solder and continue heating the component with the soldering iron. Apply solder until a uniform "tent" has formed on the PCB and the component.

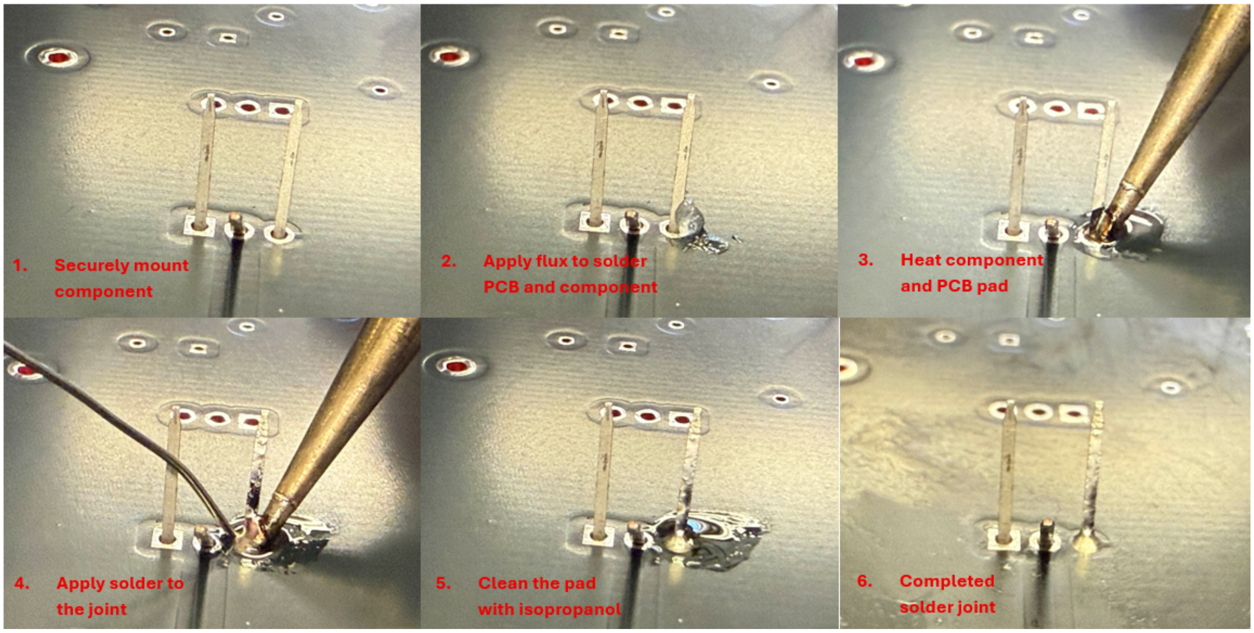


Figure 1: Soldering technique

There are two common mistakes often seen in soldering: cold joints and excess solder. A cold joint occurs when the component or pad is insufficiently heated, and solder does not bond well to form a good electrical contact. To avoid this, apply solder directly to the component and avoid touching the tip of the soldering iron to make sure the component is heated.

There is enough solder to form a solid connection when it forms a small "tent". Excess solder can overflow from the component, cause short circuits, impede de-soldering, and is wasteful. Additionally, excess solder can conceal a possible cold joint which can cause connectivity issues.

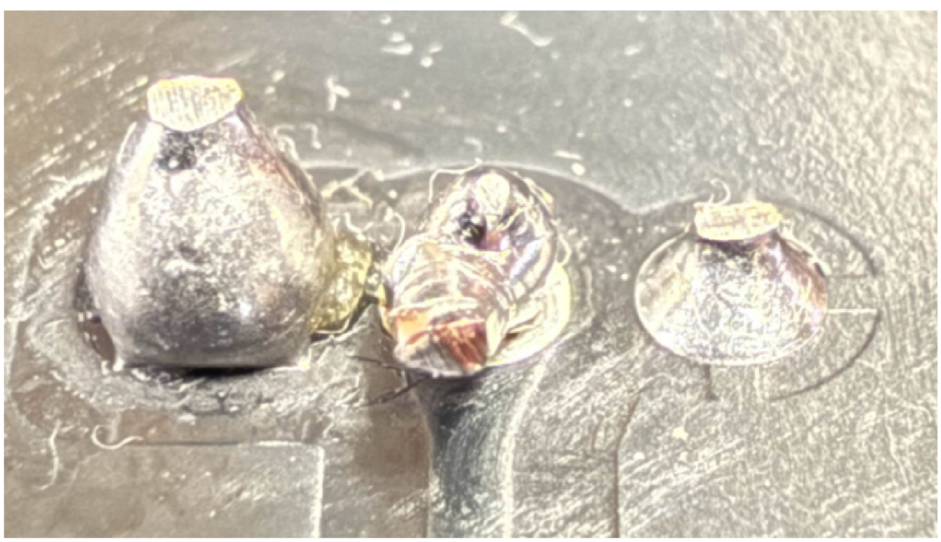

Figure 2: From left to right: excess solder, cold joint, and proper solder joint

Oftentimes these two problems accompany each other. When a cold joint does form, it is a common response to add additional solder to attempt to fortify the joint, but that will only cause further problems. Instead, in the event of a cold joint, continue heating the part and the pad with the tip of the soldering iron. The present solder should flow onto the joint when it has been sufficiently heated.

At the end of the proverbial day, though, situations will arise where you may have to improvise. If it works, it works!

The techniques for de-soldering, or removing solder from a component, will not be covered under the scope of this exercise.

# Safety

This exercise involves the use of soldering irons, solder, and electricity. There are various precautions one must take to be able to work safely with these tools.

## Soldering Iron

The soldering iron is extremely hot during operation, and contact with skin will cause severe burns. Always place the soldering iron in the holder when it is not being used, and make sure that the soldering iron tip, as well as the exposed metal barrel, touches no surface except for the PCB when it is turned on. **The entire front metal portion of the soldering iron is heated, and any part of it will burn as well as cause heat damage.** When using the soldering iron, only grip it by its plastic handle.

Keeping the work area free of clutter and stray cables will greatly reduce the chance of anything being damaged and anyone being injured. Always assume a soldering iron is hot, even when it is turned off. Tie back any loose hair as any loose strands will burn when close to a hot iron.

## Solder

Most electronics solder is an alloy containing 63% tin and 37% lead. Lead is a heavy metal which is toxic to the human body, and exposure should be as limited. When soldering, work in a ventilated space and use some form of fume extraction at all times. Do not touch the face, eat, or drink before a thorough hand wash. When handling a completed PCB, handle the board by its edges to limit lead exposure as much as possible.

## Electricity

A "short circuit" may occur when two points on the circuit are connected incorrectly, especially in the case of excess solder. Short circuits can cause damage to components, and in the worst of cases, generate large amounts of heat,and stress the circuit's power supply. Make sure to carefully inspect the completed PCB to see that each solder joint only connects a single pin to a single pad on the PCB. Additionally, a bench top power supply should be used to check the function of any circuit in the "Custom Development Area" before it is connected to the rest of the PCB. The maximum output of the USB-C Power Supply is 3A, 5V and any excess current draw may cause damage to the power supply.

# List of Materials

- **Transistors**

  - PNP transistor (2N2907A) — 2 (Q4, Q6)
  - NPN transistor (2N4401) — 4 (Q1, Q2, Q3, Q5)
  - N-channel MOSFET (IRLZ14) — 2 (Q7, Q8)

- **Resistors**
  - 10kΩ resistor — 1 (R1)
  - 10Ω resistor — 7 (R4, R7, R8, R11, R12, R13, R14)
  - 1kΩ resistor — 4 (R5, R6, R9, R10)
  - 5.1kΩ resistor — 2 (R2, R3)

- **Trimpots**
  - 20kΩ trimpot — 1 (RV2)
  - 5kΩ trimpot — 1 (RV1)
  - 2kΩ trimpot — 1 (RV3)

- **Capacitors**
  - 1000$\mu$F electrolytic capacitor — 1 (C3)
  - 100nF ceramic capacitor — 4 (C1, C4, C5, C6)
  - 10nF capacitor — 1 (C2)
  - 2.2$\mu$F film capacitor — 1 (C7)
  - 3.3$\mu$F film capacitor — 1 (C8)
  - 3300$\mu$F electrolytic capacitor — 1 (C9)

- **ICs**
  - TLC555 — 1 (U2)
  - LM393 — 1 (U1)

- **Other Parts**
  - 22$\mu$H Inductor — 1 (L1)
  - 3.5mm Jack — 1 (J1)
  - Schottky diode — 1 (D1)
  - 3W 4Ω speaker — 1 (N/A)
  - Terminal block — 1 (LS1)
  - USB-C terminal — 1 (J2)
  - test point — 9 (TP1 - 9)
  - 3 pin connector - 1 (SW1)
  - bridge jumper - 1 (N/A)
  - class-D amplifier PCB - 1
  - 8-Pin IC Socket - 2 (U1, U2)

# Procedure

This section will largely contain a collection of things to note for each individual module. It is meant to be viewed alongside the instructional videos on the project GitHub, which can be reached through the QR code on amplifier PCB.

Most component footprints on the PCB include their respective schematic symbols. Familiarity with these conventions will aid you in determining the orientation of polarized components! The component footprints also line up with the physical size of the components, so if there is a big mismatch between the component and the footprint something has probably gone awry.

If more clarification is needed, please reach out to the email posted on the GitHub. It is the same one you emailed to obtain a kit. Good luck!

Testing instructions will be posted separatly.

## USB-C Power Supply

- USB-C power adapter - 1 (J2)
- 5.1kΩ resistor - 2 (R2, R3)
- 1000$\mu$F electrolytic capacitor - 1 (C3)

For the USB-C power adapter, refer to the guide video which can be found on the project GitHub as it requires a different technique to solder than the rest of the components. Make sure that the electrolytic capacitor has the correct polarity before soldering it in place!

If any pins on the USB-C power adapter are connected accidentally, you can retouch them using a clean soldering iron. Excess solder should flow towards areas of lesser concentration which will be your soldering iron if it is properly cleaned. For this reason, it is recommended that you solder this component first to avoid having to work around other components.

## Audio Input Processing

- 3.5mm Jack - 1 (J1)
- 10kΩ resistor - 1 (R1)
- 10Ω resistor - 1 (R4)
- 5kΩ trimpot - 1 (RV1)
- 20kΩ trimpot - 1 (RV2)
- 0.1$\mu$F ceramic capacitor - 2 (C1, C4)
- 3 pin connector - 1 (SW1)
- bridge jumper - 1 (N/A)

For how to read the values of the ceramic capacitors and the trimpots please refer to the description of the associated video. It is explained in-depth there. Tape is highly recommended for soldering the trimpots and capacitors.

## Triangle Wave Generator

- TLC555 - 1 (U2)
- 10nF ceramic capacitor - 1 (C2)
- 2kΩ trimpot - 1 (RV3)
- DIP-8 IC socket - 1 (U2)

Do not solder the TLC555 directly onto the PCB! Instead, solder the IC socket into U2 instead and plug the TLC555 into the socket afterwards. This allows for easy removal of the IC in the event that it becomes damaged. When soldering the IC socket, use tape to hold it against the PCB.

Refer to the video to double check correct orientation to place the TLC555. Pin 1 of the IC, denoted by a dot its surface, is associated with the square pad on the PCB. This is true for both the ICs used in this project.

## PWM Modulator

- LM393 - 1 (U1)
- 1kΩ resistor - 2 (R5, R6)
- 10Ω resistor - 2 (R7, R8)

Same tips as the Triangle Wave Generator module. Besides the IC, this section just has resistors. You have soldered a bunch of them already, just treat the ones here as additional practice.

## PWM-Switching MOSFETs

- PNP transistor (2N2907A) - 2 (Q4, Q6)
- NPN transistor (2N4001) - 4 (Q1, Q2, Q3, Q5)
- N-channel MOSFET (IRLZ14) - 2 (Q7, Q8)
- 10Ω resistor - 4 (R11, R12, R13, R14)
- 1kΩ resistor - 2 (R9, R10)
- 100nF ceramic capacitor - 2 (C5, C6)
- 2.2$\mu$F film capacitor - 1 (C7)

Looking at the PNP and NPN transistors, you can see that there is a flat and round side. The correct orientation of the component is marked by its footprint on the PCB, which also has a flat and round side. Just line it up and solder in place! Make sure the right transistors are being used as the PNP and NPN transistors look extremely similar to each other.

The diode D1 has a silver stripe near one of the terminals, and is black everywhere else. Observing the schematic symbol of D1 on the PCB, you can see that it looks like a triangle pointing into a line. The line corresponds to the silver stripe on the diode and shows the correct orientation to solder the diode. Additionally, the leg that corresponds to the silver stripe should slot into a square pad.

The protruding black package on the front of the MOSFETS (Q7 and Q8) must face each other on the completed module. Use some tape to hold these parts down while soldering.

The value of the film capacitor C7 can be determined using the same scheme as the previously used ceramic capacitors. Also, only one of the two film capacitors (the shorter one) will fit into this footprint!

The rest of this module is populated by ceramic capacitors and resistors, more practice, yay!

## Audio Output

- 3300$\mu$F electrolytic capacitor - 1 (C9)
- 3.3$\mu$F film capacitor - 1 (C8)
- 22$\mu$H inductor - 1 (L1)
- terminal block - 1 (LS1)

The inductor (L1) looks very similar to an electrolytic capacitor, but instead of a line of negative symbols indicating its negative terminal there is instead a line of dots.

Again, double check the polarity of the electrolytic capacitor (C9) before soldering it in place.

If you want to use the speaker box design included in the GitHub, solder the terminal block on the BACKSIDE of the PCB with the wire terminals facing TOWARDS the PCB. If not, Solder the terminal block on the front side of the PCB with the terminals facing away from the PCB.