ezDAC Construction Guide

Introduction

Most of the necessary information for building the ezDAC is available at the project website (http://www.ezdiyaudio.com). If you can read schematics and understand PCB drawings, you should have no problem understanding the design of the ezDAC (it is "ez", afterall). This document can serve as a handy reference or maybe just some reassurance that you are doing things correctly.

Powering the ezDAC

The ezDAC has several active components (labeled with "U" followed by a number), which must be powered using specific (DC) voltage levels. There are several ways to provide power to the ezDAC. The easiest way is probably to populate the onboard regulator sections, V1-V4, with adjustable 3-pin voltage regulators, LM317 for the positive rails (V1-V3) on the top of the board, and LM337 for the negative rail V4 on the bottom of the board. The negative rail is only used to power the op amps, which each have a positive and negative voltage pin. If using the LM317/LM337 voltage regulators, you must solder the resistors and capacitors in those sections. Each regulator has two resistors, which set the output voltage, and three capacitors which are used for bypassing and smoothing the voltage. For example, for U8, these components are R2, R3 and C1-C3. Now, the nature of the 3-pin adjustable regulators used here require that the input voltage to each regulator is approximately 2V higher than the output voltage. These are not low dropout regulators. For example, using LM317, if the desired output voltage were 10V, you would want to supply at least 12V, or else, risk damaging the regulator. On the other hand, more heat is generated or dissipated by these regulators as the voltage difference is increased. Especially for the low voltage sections of the DAC (U8=3.3V, U9=5.0V), this means proper heatsinking should be used. In the original ezDAC (v.1.0), the positive rail was designed to be powered by a single external positive DC power supply. In this version (v.1.5), the three positive power rails (V1-V3) can still be powered by a single DC supply, but the thick traces above the regulators must be jumpered together by wire or zero-ohm resistors. If using wire, which is recommended, use the thickest and shortest piece of wire you can get your hands on that will insert into the pads. Ideally, you want the resistance of the jumpers to be as close to zero ohms as possible. So, if you jumper these sections (and you could jumper all three or just V1-V2, or V2-V3, you get the idea...), your power supply must be able to supply enough current and a high enough voltage. Next, I will describe some specific cases:

Jumpering V1-V2-V3

This would be the bog standard way of doing things. Your power supply must be a couple of volts higher than the output voltage for the op amp regulator. Let's assume you want the output of the V3 regulator (U10) to be +12V. Your DC supply should be rated >= 14V and several hundred milliamps. In this case, the regulator that will generate the most heat will be U8, but even a relatively small bolt-on heatsink will do a good job of dissipation. In other words, you will be able to touch the heatsink without be scorched!

Jumpering V2-V3

U9 and U10 power the analog sections of the DAC, so it may be of some benefit to have separate supplies for these sections. For example, you could power U8 (the 3.3V regulator) with a relatively low voltage external supply (~5V, >200 mA rated), and U9, U10 with a higher-rated supply, for example, 15V, and ~100 mA or so. There would be several benefits of this approach, among them: 1) Less heat generated by U8; 2) better separation of analog/digital sections; 3) U9 and U10 are not required to source as much current as U8, so they can benefit from a better lower-current supply than may be possible with all three sections combined.

No Jumpers

Of course, you can provide each onboard regulator with its own power supply, thus tailoring the power source basically to each individual chip. Just remember, U8 outputs 3.3V, U9 outputs 5V. U10 can output anything above about +3 V (it must be greater than the output line-level signal from the op amps). My recommendation would be to power U10 with something greater than +9V (as many op amps actually perform better with greater voltage). Just make sure the voltage you supply is within the rated range for the particular op amp that you are using.

Bypassing Onboard Regulators Completely

You do not even need to use the LM317/37 regulators. If you have a better way to provide a good, low-impedance source of regulated power directly, then by all means, you do not need to use the onboard regulators. Simply connect your power supplies directly to the pads on the output sections where each regulator would go. For U8-U10, that would be the middle pad (you can see the power traces running to the chips), and for the negative rail (U11), that would be the pad on the left (closes to the "U11" label). If you go this route, make sure to connect the ground and/or sense leads appropriately, so that you avoid ground loops and such.

The Negative Rail

As I said above, the negative rail is only used to power the negative supply pins of the op amps in the output section. The only guideline here is to supply the same voltage (in magnitude, but opposite polarity) as for U10, whether you are using the onboard regulator or bypassing the regulator and connecting the supply directly.

Powering the XO (Oscillator Clock)

There is also quite a bit of modification possible here. The standard build would be to use the Abracon 4-pin (half-size) XO can, connected to the 3.3V regulator. Instead of the 4-pin can, however, it is possible to use a surface mount oscillator (Crystek makes one, and it is specified in the BOM). In this case, it is simply a part-for-part exchange. No other mods are necessary. Some of you may want to use an external clock, of which there are many to choose from (Tent, Kwak, Flea, on and on). To use these external clocks, you can simply connect the lead from the clock to the clock output pad on the ezDAC board (the one closest to the R36). In this case, you would still need to solder R35-36 (otherwise, the signal wouldn't get to the chips!), although you may need to change the nominal values for those resistors. You will be on your own with that choice. You will not need to connect F2, since no power is being drawn from the regulator.

Choosing the Op Amps

Some of you may be keen on trying different op amps from those recommended in the BOM. No problem. Just make sure that YFOA (your favorite op amp) has the same pin outs (pin compatible) with the AD8610. AFAIK, virtually all single channel op amps are pin-compatible, but not all operate on a dual (+/-) supply. Also, the voltage range may vary between op amps. There have already been many different op amps that have been shown to work correctly in the ezDAC, so you will likely have no problem trying a different one from the BOM.

Bypassing the I/V and Output Stage of the ezDAC

Some of you may want to mess around with external I/V, gain, and buffering of the PCM1794/8 output.

No problem. There are two ways to do this. You can tap the current output directly using the pads for the onboard I/V resistors (but do not solder those resistors), or you can solder the resistors, and tap the voltage, using the pads for the filter caps (C33 for the left channel, C36 for the right channel). In either case, you will have differential output, so you will need to figure out how to use them properly. Do not try to drive headphones or other low impedance loads directly from these outputs.

Connecting I2S Inputs

Some of you may want to bypass the S/PDIF receiver (CS8416), and connect I2S lines directly to the ASRC. This is possible. You can try to solder the leads directly to the 0805 pads (R19-R21) that go to the ASRC. If you are doing this, make sure to read the schematics/datasheets carefully. This is a mod that I would only recommend for those who are experienced and know what they are doing. Oh, and if you do not know what I2S means, please do not attempt this in any case.

Construction Guidelines

The ezDAC contains many surface mount parts, the smallest being 0805. With practice it is certainly possible to solder all the components by hand. I have done it, and many others have done it. However, I cannot honestly say it is an easy thing to do, although it does get easier with practice. Before attempting to build the ezDAC, you should a) have experience building other boards with similar smt packages or b) figure out a way to practice this type of soldering.

In general, when constructing a board with surface mount components, you want to start with the components at the center of the board and work your way out. Of course, don't forget the components on the underside of the board. Otherwise, you risk "trapping" yourself. You'll know what I mean by this, if it has happened to you before. Normally, I would suggest soldering the main IC's first (U1, U2, U3). However, in this case it may be easier to solder R11-R15, C26-27, and C31 first, and then proceed in the regular fashion. Leaded (through-hole components) should be done after all the smt components, and regulators last. Please note these are my recommendations. If you have a different way of doing things that works for you, by all means, proceed in your normal routine. After you are done, check all soldering very carefully, and proceed with powering up the board. (I always stand back several feet, just in case things blow up!) Hopefully, you will have passed the "smoke test", and then you can connect the board to your preamp or amp. Make sure the volume in your system is turned all the way down. If nothing unusual has happened so far, turn up the volume slowly, and listen for some music. At this point, the DAC is hopefully working. From there, I have nothing left to offer, except...Happy Listening!