SSR01 Sjöström Super Regulator

Contributed by Per-Anders Sjöström
Wednesday, 08 April 2009
Last Updated Tuesday, 27 July 2010

SSR-01 The Sjöström Super Regulator

Click on the picture to get a larger view.

Some background can be found in my JSR-01 project.

This design is based on my JSR-01 project and the goal was to make the pcb with hole mounted parts as small and versatile as possible.Interesting features
- 4-layer pcb with 70 um (2 oz.) copper.
- Power traces 140 um!

- Gold pads.
- Extremely low noise.
- Extremely low output impedance in the audioband.
- Small size.
- Easy to change voltage, also to negative voltage but this particular design (pcb) requires two different pcb's, one for positive voltages and one for negative.
- Well-known and well-tested in serious and demanding applications.
- Option for TO92 or SOT23 transistors for the small signal transistors.
- Option for DIL08 or SO08 opamp.
- LM431 (and similar) reference or devices like LM329. The pcb has a universal footprint.
- Trimming options for both the reference and the feedback. Room for extra resistors and trimpots.

- One output with sense inputs.
- All resistors 10 mm so it is possible to use a cut and bending machine for those parts.
- Option for using higher voltage than the max supply voltage for the opamp.

{mospagebreak title=Background} The background

This design has been known at www.diyaudio.com and elsewhere and it has been bescribed thouroughly in couple of articles in The Audio Amateur and also in other magazines, like EDN. The design presented below is a developed/altered version (not necessarily better) and not identical to the one in the articles nor to the pcb from Audio Express. Some of the changes has Andrew L Weekes done and some have I done.Changes Walt Jung -> Andrew Weekes

Andrew added the preregulator and redesigned the pcb and added a starground in a more clear way. He also made a higher quality pcb with thicker copper.Changes Andrew Weekes ->Per-Anders Sjöström
- Added all parts around the preregulator
- Added component print
- Added a footprint for SO08 opamps
- Added compensation for the opamp

- Added a footprint for LM431 plus LM329
- Added footprints for SOT23 transistors
- Added trimming facilities, bith with fixed resistors and with trimpots.
- 4 layer pcb with 70 um copper resulting in 140 um for all high current traces
- Added parts for tying the sense inputs to the outputs
- Added solder pads for the sense inputs if you wanted to "park" them and left them unused.
- Added option for using higher voltage than the max supply voltage for the opamp.
- Added option for feeding the opamp from before the pass transistor enabling a very low output voltage.
- Gold on all exposed copper surfaces
- All connections for thick wires

The regulator from 1961

The basic idea has been around for at least 48 years.

Bob Pease has dig up some info about the Philbrick tube super regulator . The circuit solution can be traced back to 1961 at least. Change all triodes to transistors and you'll see that this is really a super regulator.

The regulator from 1977

The Super Regulator from 1977, design by Mr. Kaneda.

An another example can be found in book written by Mr. Kaneda from 1977. What has changed since then? The zener references are nowadays much, much better and the opamps are even more much better. In other words there are great opportunities to increase the performance. One more change, rather important also is that the 680 ohms resistor at the top is changed to a constant current generator. This is very important for the output current capability also to the bandwidth, the speed of the regulator.   Is "Super" too much?

Does the design deserve the epithet "super"? Yes, indeed! Can anyone come up with any regulator with less noise and lower output impedance? I don't think so. I would be happy if someone can prove me wrong. This design is extreme in the true sense when it comes to these two parameters. I believe that a couple of more parameters also are state of the art.

I think this design is very interesting from an engineering point of view, a challenge to make it as good as it can be. Therefore I have decided to have it for spin.

Andrew L Weekes ALW has made a very good hole mounted pcb together with a very good documentation. He has also had a group purchase run at www.diyaudio.com and he came up with 700-800 pcb's, but he has left the DIY scene years ago for other activities but you can get his boards though.

aos (Andrija Ifkovic), has also made an pcb to a very good price but has stopped selling pcb's.

BrianGT (Brian Bell) has integrated the regulator in his DAC design. The thread is here. An another nice picture is here. The design did never leave the computer.

Coffin's friend has also made two pcb's, can't say how good they are but they looks nice at least.

Warren Young has designed a Young-Jung regulator along with a nice pcb. Can't say how good it is but the pcb design looks correct.

There are also a couple of commercial sources, really expensive ones.

{mospagebreak title=The design}   The design

The design is based nearly to 100% of the original super regulator. My contribution is transferring it to a more advanced pcb and making the pcb more universal. I have also made options for three different references. There will be no trouble at all to build the different versions.   The schematics

Click on the picture to get a larger view.

I recommend that you download the schematics and print it so you can follow the describing text more carefully.

{mospagebreak title=Circuit description}Circuit description   The pre-regulator

The first part is the pre-regulator, IC1 (or IC2) which has a rather odd connection. The sensing network, R1 and R2 is not connected to ground as usual. It's connected to the output so the LM317 regulates the voltage across the pass transistor T2. The purpose of this is to create a constant voltage drop over this transistor. With this connection you will get a more flexible minimum input voltage which is 2.5 volts (according to the values of R1 and R2) plus the minimum voltage across the LM317 which is 2-3 volts. In total you will need approx. 5 volts more than the regulated output voltage.

The main purpose of this regulator is to increase the power supply rejection ratio, PSRR. If you of some reason don't want this pre-regulator just omit, C1-C3, R1, R2, IC1 (or IC2). Solder in a wire between Vin and Vout. See the schematics. The rest of the regulator consists of a reference of some sort, opamp, output stage and feedback network.
The reference

As reference I have chosen to have maximum flexibility. Virtual any reference will fit as long as the voltage is above the common mode limit for the used opamp. 2.5-7 V will work and in some cases also 1.2 V. The original LM329 can only be delivered in TO92 package and not in any kind of SMD. Therefore I have made options for TO92 (hole mounted), SO08 and SOT23. I have used LM431 in SO08 and this very cheap reference is rather good, although not as nearly as good as

LM329 but much, much better than a regular zener diode (which also can used). As a little help if you use LM431 hole mounted, I have written down a small note at the pcb, just under C7. It says 431/RAC which is the pinning, Ref, Anode, Cathode. You will have to twist the LM431 in order to fit. No reference can be used for negative voltage just by turning them around or something so therefore all three types are doubled for negative voltages. Please notice that the resistors which are setting the reference voltage must be interchanged, R4 and R5 swap it's positions for negative voltage. R3 is carefully selected and may be changed if especially lower voltages are to be chosen. See instructions for this.

If you want to trim the reference voltage just change R5. If you want exactly 15 volts out, you should have slightly under 5.0 kohms. You could either have 5.1 kohms of solder in one 56 kohms on top of the R5. I chose 5.6 kohms because it's a more common value.

The reference voltage is filtered by R6 and C7, C8 in order to remove the noise and as a side effect you will get a smooth startup. R6 should optimally be R7 and R8 in parallel but I consider this not extremely important. You can either choose 499 ohms or simply solder two 1 kohms on top of each other. If you have matched input impedance's you will minimize the input offset and also the drift with temperature (not very important in audio applications). It may also be good when it comes to common mode rejection. Since the whole design is placed on a groundplane I gather that high frequency problems are not so big so you must take this into account. R9 and C9 are parts for "just in case", to tailoring the frequency properties. Normally R9 is only a tin jumper and C9 is omitted.   The opamp

The opamp can be almost any type as long it's not too fast, faster than the output stage. The maximum speed of the opamp at unity gain is around 30-40 MHz. The AD825 has the bandwidth of 34-37 MHz at unity gain and the AD797 has unity gain bandwidth of approx. 30 MHz. If extremely low noise is important AD797 might be suitable but if "low" noise is sufficient I think any good audio opamp will do, like AD8610, OPA134 etc. Even good old NE5534 may work but I haven't tested this opamp. The only thing you must think of is the common mode limits for the inputs of the opamp. At start-up it's important that the opamp has predictable behaviour, especially sensitive is this for a negative regulator.

At the moment only AD797 and AD825 are tested for 5-14 volts, both positive and negative output voltages.   Power for the opamp

Since the pcb is made both for positive and negative voltages the opamp must have different connections for the power supply voltage, pin 4 and 7 at the opamp. You can either take the power from the output or before the pass transistor T2. What you choose is dependent of used opamp and how low the minimum supply voltage is (see the datasheet for this parameter). Minimum voltage for AD797 is 10 volts so if you want to be really sure you must feed it from the unregulated side if you want less than 10 volts out. Parts which are involved in this are R10-R13 and C10, C11.   The feedback network

The feedback network is very simple, only R7 and R8. C6 reduces the output impedance half of it's the value. This capacitor is hardly necessary in real life but if you want lower output impedance you should use it. The values of the feedback resistors should be a low as possible considering the max power dissipation. It serves to purposes. The first is to create a voltage divider with good high frequency properties (avoiding unstability) and also to draw current out of the pass transistor, T2. If current flows all the time also the output impedance can be kept low. It's also important for stability reasons.

The output voltage is determined by the reference voltage * (R7/R8 + 1)   The output stage

The output stage is rather unusual. Normally you have only one emitter follower, maybe together with a driver transistor forming a Darlington transistor. This type of output stage is rather slow, too here. To speed things up the driver transistor is changed to a class An emitter follower as the driver for the pass transistor. The load of the emitter follower (T3) is a current source formed by a reference voltage, the LED H1 (makes a nice glow) and R14 together with T1. The current through the LED should be 1-3 mA. If you want very low voltages you must change the value of R14.

The driver is also a bit backwards. This creates a very special and also very important feature, the possibility to a safe power up of the opamp. The output stage deliver max output voltage when the opamp is inactive. This makes the opamp come alive. The zener DZ1 (or DZ2) increases the startup voltage and creates also a necessary DC shift so the output of the opamp works at ideally at half the output voltage. This zener voltage must be decreased when lower output voltages are wanted. For 5 volts it's sufficient with a diode in the forward direction, creating only 0.7 volts. T3 is added compared to Walt Jung's original (but Mr. Kaneda had it) and unloads the opamp from the drive currents. C12 takes down the impedance of the zener.   No short circuit limiting

I have increased the current of the driver T1 and T2, for higher speed but this makes also the regulator more vulnerable against short circuit so I suggest you are careful when you are connecting.   Capacitive load and C13

The faster opamps you have, the more sensitive against capacitive load the regulator will get, meaning low loss capacitive load. The regulator has a minimum load of 0.3-0.5 ohms and if you have a 100 nF connected very close to the output transistor you can get an impedance lower than 0.5 ohms. This frequency will be at 3 MHz. Therefore the C13 must not be too "good". In fact a "normal" cap is sufficient and if you choose a low impedance cap you may experience unstability. If you use a fast opamp, it simulates a cap rather high up in frequency so serious decoupling this close to the output transistor isn't necessary but if you only have 30 mm wire the regulator can take 100 nF or more. The short wire creates a small inductance which is sufficient in order to limit the lowest impedance.   The "sense" connections

The sense connections are a bit overkill but if anyone wants these I have added this feature. The sense function is mainly to subtract or eliminate the losses in the power wires, which won't be especially large. It's just milliamps. If you want this feature make sure that the sense wires are close to the power wires so instability can be avoided. If you don't want to use this feature, just put two tin jumpers, close to the connector (you can't miss it) on the solderside of the pcb. If you are sure that you never will use the sense function you can also omit R16, R17, C14 and C15.

!!!!!! more to come......

{mospagebreak title=The pcb layout}The pcb layout

Please download the pdf of the pcb layout for more info.

{mospagebreak title=Building directions}   Building directions

This design is very easy to build. . You can solder the parts in the order you'll like but it may be practical to start with all low parts such as resistors and then take higher parts.

Start with all low parts except for the output power amp and the opamp.
 - Opamp if it's a SMD type

 - Resistors, jumpers

 - Zener diodes
 - Plastic capacitors
 - Transistors, voltage reference

 - LED

 - Trim pot (if you use it)

 - Capacitors, elelctrolytic
 - Regulator, power transistor

The pcb has four layers and therefore it requires much more heat than regular two-layer pcb. Ecpecially the ground connections are heat demanding. Use a 50 watts (at least) temperature controlled soldering iron and warm the pad so it really melts and tin flows to the component side. As you can see in my pictures I haven't succeeded to make 100% perfect solder joints and especially hard are the ground connections. If you check the R3, here I should have burned with the soldering iron a lot more. You can warm up the pcb with a hot air gun if you have a weak soldering iron. This will make a lot better result.

Besides from this instructions the regulator is pretty easy to build but pay attention how you should mount all polarized parts. See pictures below and also the pdf file. {mospagebreak title=5V, LM431} 5 V with LM431

Click on the picture to get a larger view.

The yellow stripe is cathode or the short leg of the LED. Notice also the yellow dot at the opamp.

{mospagebreak title=9V, LM431} 9 V with LM431

Click on the picture to get a larger view.

The yellow stripe is cathode or the short leg of the LED. The zener DZ1, cathode up. Notice also the yellow dot at the opamp.

{mospagebreak title=12V, LM431}12 V with LM431

Click on the picture to get a larger view.

The yellow stripe is cathode or the short leg of the LED. The zener DZ1, cathode up. Notice also the yellow dot at the opamp.

{mospagebreak title=15V, LM431}15 V with LM431

Click on the picture to get a larger view.

The yellow stripe is cathode or the short leg of the LED. The zener DZ1, cathode up. Notice also the yellow dot at the opamp.

{mospagebreak title=18V, LM431}18 V with LM431

Click on the picture to get a larger view.

The yellow stripe is cathode or the short leg of the LED. The zener DZ1, cathode up. Notice also the yellow dot at the opamp.

{mospagebreak title=24V, LM431}24 V with LM431

Click on the picture to get a larger view.

The yellow stripe is cathode or the short leg of the LED. The zener DZ1, cathode up. Notice also the yellow dot at the opamp.

{mospagebreak title=9V, LM329} 9 V with LM329

Click on the picture to get a larger view.

The yellow stripe is cathode or the short leg of the LED. The zener DZ1, cathode up. Notice also the yellow dot at the opamp. The LM329, VR1 is placed in the alternative position.

{mospagebreak title=12V, LM329}12 V with LM329

Click on the picture to get a larger view.

The yellow stripe is cathode or the short leg of the LED. The zener DZ1, cathode up. Notice also the yellow dot at the opamp. The LM329, VR1 is placed in the alternative position.

{mospagebreak title=15V, LM329}15 V with LM329

Click on the picture to get a larger view.

The yellow stripe is cathode or the short leg of the LED. The zener DZ1, cathode up. Notice also the yellow dot at the opamp. The LM329, VR1 is placed in the alternative position.

{mospagebreak title=18V, LM329}18 V with LM329

Click on the picture to get a larger view.

The yellow stripe is cathode or the short leg of the LED. The zener DZ1, cathode up. Notice also the yellow dot at the opamp. The LM329, VR1 is placed in the alternative position.

{mospagebreak title=24V, LM329}24 V with LM329

Click on the picture to get a larger view.

The yellow stripe is cathode or the short leg of the LED. The zener DZ1, cathode up. Notice also the yellow dot at the opamp. The LM329, VR1 is placed in the alternative position.

{mospagebreak title=Test} Test

If you don't plan to use the "sense" function, put a tin blob on the solder side at J1 and J2 (see the schematics also).
- Use a regulated power supply and set the current limit to 100 mA or something low.
- Connected a voltmeter at the output.
- Turn up the voltage slowly and observe the output. When the LED H1 starts to shine you should have a stable output voltage.

If you have the preregulator LM317/337 installed the output voltage should be stable with 4-5 volts more in than set output voltage.

{mospagebreak title=Technical data} Technical dataOperating voltage:Max 35 V, less current at high voltage in. Min 4 volts more than output voltage.Output voltage:5-30 V, down to 3 volts is possible.Hum and noise at full output current:Down to 0.9 µV or below depending of component choicesMax current:Approx 1 A peakMax continuous current:Approx 1 A depending of ambient temperature and mounting.Dimensions:58,4 (2.3") x 38,1 (1.5") mm

Noise measurements
Used parts
LM329, AD825
LM329, AD797
LM329, LT1028
No filter:
15.14 µV 5.67 µV 6.95 µVA-Weighted: 2.38 µV 1.03 µV1.00 µVCCIR-486: 6.40 µV 2.04 µV2.02 µVCCIR-2K(Avg): 3.03 µV 0.94
µV22-22K: 3.52 uV
2.37 uV
1.70 uV