

Freitag, 23. Januar 2009 DIY | studio | mail

Enhanced Pultec Equalizer Project

Enhanced Pultec EQ V3 untested



Pultec GainStage for NE5532, OPA2604...



Description

Here you can get the files for a very easy diy project (hopefully) ;-)

Its the Filter Circuit of the well know Pultec Program Equalizer EQP1A. The circuit is not original. No tubes and other dangerous or expensive parts are involved here. The idea was to have a circuit close to the original (the beloved inductor based filter circuit from pulse technologies) but with easy to find components.

In the beginning the filter was passive. The gainstage was added later. I'd seen pultecs without tubes but with a discrete opamps. So if the pultec would be designed these days... maybe they wouldn't use tubes, too? 0:-) But thats a question for Eugene Shenk from Pulse Technologies himselfs. ;-)

If you want the tube version of this EQ you could either google for "pultec eqp" and you'll find informations about how to build it with tubes and close to the original. Another Idea is to use the nice SRPP Gainstage from Jakob Erland (Gyraf Audio). So you don't need the opamps pcb. But everything will be muuch more expensive... Lundahl Transformers, Tubes, Mains transformers...

The PCB Layout

The pcb layout of the filter pcb fits on a 160x100mm pcb (eurocard). The gainstage/input output stage fits on a half eurocard format.

There are pinouts for two inductors. You have to use one at a time of course! Really, the sound will not be better with two inductors! ;-)

Here you can download the pcb files and component placements:

Layout for the Filter PCB Layout for the Gainstage PCB

The inductor

The pcb layout is made for RM8 cores from "epcos". Max (ioaudio) can make inductors, too! They'll fit into the pcb. You can also wind the inductors yourselfs. Its not that hard. This can be done by hand. Steffen has some informations about how to wind your own inductors on his website:

http://diygallery.de/DIYsites/inductor.html

If you want to diy the inductor:

You have to calculate the turns for each tap/pin. It depends on which core material you're using. There are different core materials available for the RM8 core from epcos. If you choose a higher value core material you'll need less turns and vice versa. If the core is filled with wire but you need to add some turns... either your wire thickness is too big or your core material is too low. (or both) ;-) 1600nH and 0,14mm wire worked for me. This may work for you. But measure the values in the end if they're right:

26mH=127 turns 39mH=156 turns 78mH=221 turns 155mH=311 turns 312mH=442 turns

The capacitors

You can use wima capacitors, orange drops or styroflex capacitors if you want. The layout is made for wima capacitors. Some capacitors are wired in parallel because the aren't available for with the exact value. You could match some capacitors if you want to have the 100% correct value. Capacitors can have around 20% tolerance.

Potentiometers and switches

You can use what you like. Lorlins will do. Its nice to have a shortened switch on the low frequency switch. That because you'll here some plops when switch through the frequencies. (only when the potentiometer is clockwise). Grayhill or elma's will do a great job, too. Manley are using Grayhill for example.

The Gain Stage

The PSU is on the gainstage pcb. You can use 7818/7918 (18V Voltage Regulators) to use NE5532 opamps or 7824/7824 (24V Voltage Regulators) if you want to use burr brown OPA2604 opamps. You'll need a power transformer with two secondary windings. 18V AC/AC should work for 18V. There is a gnd lift circuit on the board. You can add a switch on the backside of the case. Connect "chassis gnd" to the chassis. Don't forget to remove the powder coating or laquere where you want to connect the gnd at the case. If you want to have a power switch, just switch one phase of the mains connections. The earth of the mains connector must be connected to the case all the time! Maybe you need to adjust the gain when the filter is turned on. Replace the 18K resistor next to the "OUT, GND, IN1,IN2" connections with a trimpot and adjust the level while the filter is NOT in bypass mode. Input and output level must be the same then. Either you just leave the trimmer in the circuit or measure the resistor value and replace the trimpot with a resistor. If your see smoke on the pcb something is wrong. Probably the 10R resistors next to the voltage regulators are on fire ;-) They act like a fuse. There is a short somewhere. Connect your multimeter to GND and then measure the 10R resistor at the side where the ground lift switch is... if there is a short (e.g. low resistance on the multimeter) find the short and remove it. :-)

Modifications

Instead of having one switch for the low frequency you can use two lorlin switches. You can change the cut frequency and the boost frequency seperately. (if this helps) ;-)



