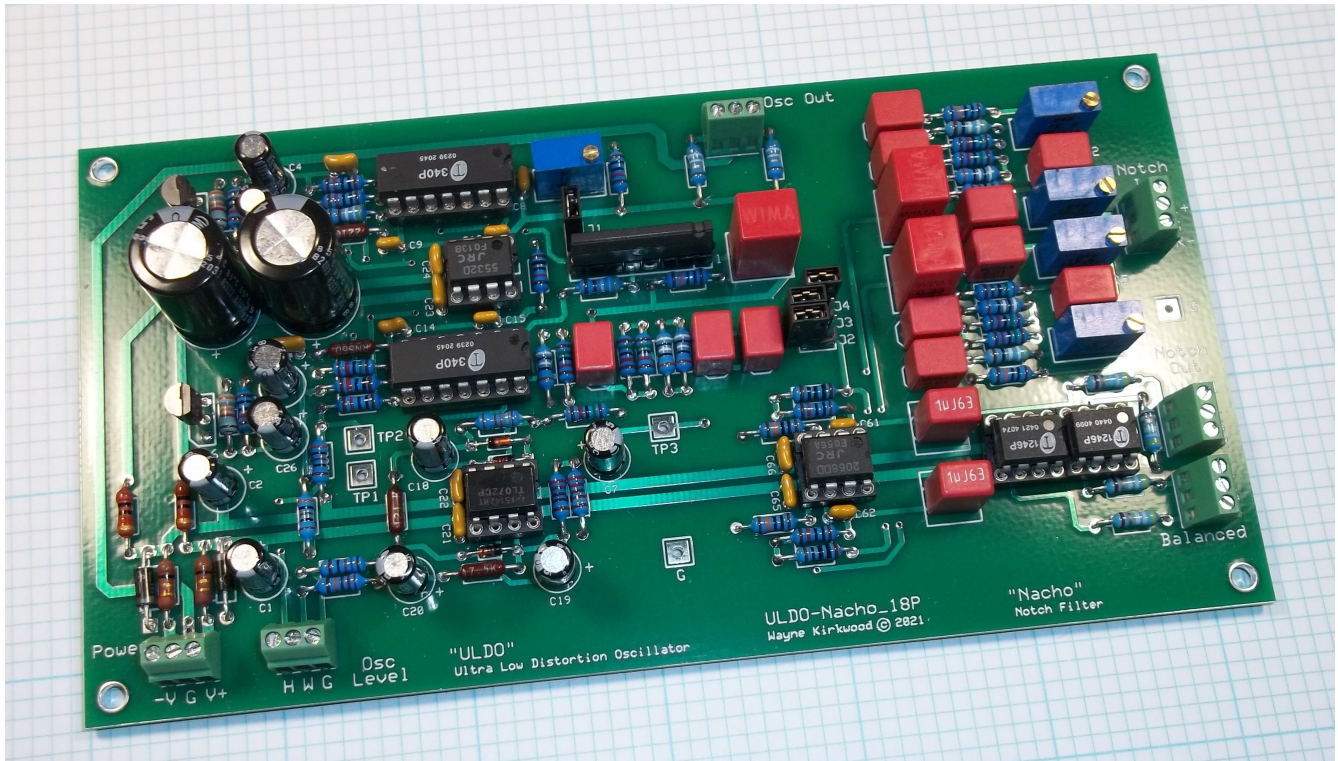


Assembly Instructions for the KA Electronics Ultra Low Distortion Oscillator – Notch Filter

9-2-2023 ULDO-Nacho 20/21 PCB



ULDO-Nacho PC Board

Install IC sockets

Place the PC Board on the bench silkscreen side face up.

Place each of IC sockets into their respective locations but do not solder them.

On the dual inline sockets observe orientation of the notch which should point to the right with the PC board oriented as shown above. Verify that socket IC3, IC5 and IC8 are not inserted into the adjacent bypass capacitor holes.

Install **five** 8 pin IC sockets at IC3, IC5, IC8, IC9 and IC10.

Install **two** 14 pin IC sockets and IC2 and IC4.

A piece of cardboard about the same size as the PC board will be used in the next step to solder the IC sockets at one time. The cardboard is used to hold the sockets in place so the board can be turned over without the sockets dropping out.

Lift the board up and place the cardboard on top of the board to form a "sandwich" of PC board, sockets and cardboard.

With the sandwich held in place flip the board over to the solder side.

Tack solder only two of the corner IC pins.

Once all the IC sockets are tack soldered in place on two pins, flip the board over to the component side. Make certain that each socket is correctly oriented, fully seated on the board and square.

If you're satisfied with the positioning of the sockets, solder all of the remaining pins.

IC1 requires a SIP socket to be installed. The SIP sockets are sold in 20 pin lengths. Count 8 pins and snap off the other 12 pins.

Install the 8 pin SIP socket at IC1. Tack solder IC1 at one pin, flip the board over and make sure the SIP socket is square and flush. Solder the remaining pins.

Visually check each pin's connection particularly those to the ground plane. Reheat any pins if needed.

Install resistors and diodes

Install **two** 1Ω 1W "fusible" resistors at R1 and R2. (These fusible resistors are 1W resistors in a 1/4W size.)

Install **five** 100KΩ 1% resistors at R3, R51, R57, R63 and R68.

Install **one** 499Ω 1% resistor at R4.

Install **one** 2.21KΩ 1% resistor at R5.

Install **two** 4.99KΩ 1% resistors at R6 and R12.

The following step requires ratio-matching two 10KΩ 1% resistors to within 10-20Ω. The BOM has 100 resistors in this value to allow matching. The resistors are ratio-matched to provide adequate adjustment range of VR1.

To ratio-match the resistors pick a resistor, measure it, then find a second resistor that is within 10-20Ω of the first. Because they are ratio-matched the values don't have to be exactly 10KΩ, they only have to be close in value to each other. If you can't find a close match to the first resistor you chose, pick another at random and match to it. If the values aren't exactly matched note which one is the higher value of the two.

Install **two** 10KΩ 1% ratio-matched resistors at R7, and R8. Put the lower value in R7 and the higher value in R8.

Install **eight** 10KΩ 1% resistors at R10, R11, R20, R21, R34, R36, R65 and R67.

Install **two** 221Ω 1% resistors at R9 and R19.

Install **five** 909Ω 1% resistors at R13, R14, R18, R55 and R61.

The following step requires value-matching three 15KΩ 1% resistors to within 10-20Ω of 15KΩ. The BOM has 100 resistors in this value to allow matching. The resistors are value-matched to increase frequency accuracy. If you can't find three that are exactly 15KΩ find three with near-identical values close to 15KΩ.

Install **three** 15K value-matched resistors at R15, R16 and R17.

Install **six** 15K Ω 1% resistors at R22, R25, R53, R54, R59 and R60.

Install **six** 49.9 Ω 1% resistors at R23, R24, R69, R70, R71 and R72.

Install **one** 24.3K Ω 1% resistor at R26.

Install **one** 121K Ω 1% resistor at R27.

Install **one** 221K Ω 1% resistor at R28.

Install **one** 47.5K Ω 1% resistor at R29.

Install **one** 332 Ω 1% resistor at R30.

Install **two** 10 Ω 1W fusible resistors at R31 and R32. (These fusible resistors are metal oxide 1W resistors in a 1/4W size.)

Install **two** 38.3K Ω 1% resistors at R33 and R35.

Install **two** 301 Ω 1% resistors at R37 and R38.

Install **one** 150K Ω resistor at R39.

Install **two** 499 Ω 1% resistors at R52 and R58.

Install **two** 7.5K Ω 1% resistors at R56 and R62.

Install **two** 100 Ω 1% resistors at R64 and R66.

Install **four** 1N4148 diodes at D1-D3 and D6. Observe polarity.

Install **two** 1N4004 diodes at D4 and D5. Observe polarity.

Install ceramic capacitors

Install **nine** 100 nF (0.1 μ F) at C6 (near J1), C16, C17, C21, C22, C23, C24, C65 and C66.

Install **one** 33 pF at C8. (Near VR1.)

Install **four** 220 pF at C9, C10, C14 and C15.

Install **one** 10 pF at C29.

Install **two** 100 pF at C61 and C62.

Install jumper headers

Before soldering the headers, install jumper shunts onto the pins. The shunts serve as insulators which allows you to position them while soldering without burning your fingers.

When installing the jumpers, tack solder only one pin and reheat it to adjust the position of the header so that its square and flush with the board. Once you're satisfied with the orientation of the headers solder the remaining pins.

You will need **four** shunts to install on the headers.

Install **one** 2 pin header at J1. Link J1 with a shunt while soldering and keep it installed.

A 6 pin header is used for jumpers J2-J4. Install three shunts horizontally on J2-J4.

Install the 6 pin header.

Install Phoenix three pin connectors

When installing the Phoenix connectors make sure that the wire openings point to the outside edge of the PC board.

Install **seven** 3 pin Phoenix connectors on the PC board.

Install small electrolytic capacitors

Note: The + (positive) terminals for the electrolytic capacitors have a square pad. The polarity of these capacitors are critical.

Install **six** 47 μF 35V at C1, C2, C3, C4, C26 and C27. **Note that the orientation of C4 is reversed from the other capacitors in the group.**

Install **two** 10 μF 35V at C7 and C20.

For ULDO-Nacho 20 versions of the PC board C18 and C19 are small electrolytic capacitors. ULDO-Nacho 21 PC boards allow either electrolytic or 5 mm film capacitors to be used. The Project Manager BOM for ULDO 21 specifies film.

Install **one** 0.47 μF 100V at C18.

Install **one** 1 μF 100V at C19.

C25 and C28 will be installed in a later step.

Install film capacitors

Install **eleven** 10 nF (0.01 μF) 1% film capacitors at C11, C12, C13, C51, C52, C53, C54, C55, C56, C57 and C58.

Install **two** 1 μF film capacitors at C63 and C64.

Install **three** 0.33 μF film capacitors at C5, C59 and C60.

Install the Trim Pots

Final test is simplified if VR1 the 200 Ω gain pot is preset to its mid-point.

With an Ohmmeter measure the resistance between the outer pins. The end-to-end resistance will not necessarily be the printed value due to tolerance.

Move one meter lead to the wiper which is center pin.

Adjust the trim pot to half the measured end-to-end value.

Install the 200 Ω Ohm trim pot at VR1. The adjustment screw should be on the right and is marked with a circle on the silkscreen.

Install **four** 1K Ω trim pots at VR2, VR3, VR4 and VR5.

Install TL431 Shunt Regulators

When installing the shunt regulators fan the two outside leads slightly to increase their lead pitch to 0.1 inches. The bottom of the device case, when installed, should be about 0.2" off the surface of the PC board.

Install **two** TL431 shunt regulators at IC6 and IC7. Be careful to observe device orientation.

Install large electrolytic capacitors

Note: The + (positive) terminals for the electrolytic capacitors have a square pad. The polarity of these capacitors are critical.

Install **two** 470 μ F 50V at C25 and C28.

Note: Do not install the ICs at this time.

Check all solder connections and reheat or re-flow them if necessary.

It is recommended that the PC board be stripped and cleaned of flux. When removing flux make sure that areas near the IC pins and it's nearby components are very clean and that no flux bridges exist. Acetone used sparingly on a Q-tip works well with rosin-based flux but make sure that it does not contact any plastic on the top side of the PC board or contaminate the IC sockets, jumpers and trim pots.

Initial Tests

In this section DC and signal tests check basic functionality.

DC Tests

Connect a source of bipolar DC power.

Ground is in the middle of the three pin power connector. Observe polarity.

If a variable power supply is used, slowly raise the voltage to about +/-15V.

There should be a small current draw, about 10 mA from the TL431 regulators. If high current is drawn stop immediately and check the board for solder bridges and for correct polarity of D4, D5, C1-C4, C25 and C28. If all is well, then proceed.

Check the voltage at TP1. It should read about +12V.

Check the voltage at TP2. It should read about -12V.

The following will check the voltages at the IC sockets before they are installed.

Check the voltages at pin 8 of IC3, IC5 and IC8's sockets. They should be +15V. The voltages at pin 4 of the sockets should be -15V.

Check the voltages at pin 7 of IC9 and IC10's sockets. They should be +15V. The voltages at pin 4 should be -15V.

Check the voltage at pin 7 of IC1's socket. It should be +12V. The voltage at pin 5 should be -12V.

Check the voltages at pin 5 of IC2 and IC4's socket. It should be +12V. The voltages at pin 3 should be -12V.

If the voltages are correct on some pins but absent on others check to make sure the IC socket pins were actually soldered.

Temporarily remove power.

Install the ICS

Install a THAT2180B at IC1. Be careful inserting the pins the first time in the SIP socket as they may be quite tight. Pin 1 goes to the right.

Install **two** THAT340 at IC2 and IC4.

Install an NJM5532 at IC3 and IC8.

Install a TL072 at IC5.

Install **two** THAT1246 (INA137) at IC9 and IC10.

Install the jumper links

Configure the jumpers for testing:

J1 open. Store shunt on one pin.

J2 linked.

J3 linked.

J4 linked.

Operating Current, Voltage and DC Offset Tests

Reconnect power.

If a variable power supply is used, slowly raise the voltage to about +/-15V.

Measure the DC voltage drop across R1 and R2.

The voltage drop across R1 and R2 should be about 35 mV indicating a current draw of 35 mA.

Check the voltage at TP1. It should read about +12V.

Check the voltage at TP2. It should read about -12V.

Measure the DC voltages relative to ground at pins 5 of IC2 and IC4. The voltages should read approximately +0.6 to +0.7V.

Measure the DC voltages relative to ground at pins 3 of IC2 and IC4. The voltages should read approximately -0.6 to -0.7V.

Measure the DC voltages relative to ground at pins 1 and 7 of IC3. The voltages should read approximately 0V \pm 10 mV.

Measure the DC voltages relative to ground at pins 1 and 7 of IC8. The voltages should typically read -2.0V or less. A high amount offset is normal

in this stage. If the offset is significantly more than -2V try another NJM5532.

Measure the DC voltages relative to ground at pin 6 of IC9 and IC10. The voltages should read 0V \pm 10 mV.

Remove power.

If the voltages above are out of range check the board to make sure that the proper resistor values are installed and that there are no solder bridges or bad connections.

Signal Tests

Set the Oscillator Gain

Connect a 10K Ω potentiometer to the Output Level Phoenix connector. The clockwise end connects to "H", the wiper to "W" and the counter-clockwise end to "G".

Apply power.

Measure the wiper voltage relative to ground and set the Level control to -4.2V DC.

Remove the shunt on J1. Store the shunt on one pin.

Connect an oscilloscope to the Oscillator Out "+" terminal.

If oscillation is not observed slowly rotate VR1 clockwise. At some point oscillation should start and slowly increase in level until it saturates. Decrease the level until it stops and then increase it again until it starts once more. Adjust VR1 about 1/8th a turn more past the start point. The point is to adjust VR1 to provide slightly more gain than it needs to oscillate.

Remove power in order to discharge the level detector capacitors C18-C20.

Install the shunt at J1 and apply power.

For ULDO-20 PC boards:

If a steady-state oscillation is observed adjust VR1 to have +300 mV at TP3.

For ULDO-21 PC boards:

If a steady-state oscillation is observed adjust VR1 to have +1.5 V at TP3.

If it is not oscillating increase VR1 slightly. (Clockwise rotation.)

If the oscillator level is not steady-state but ramps up and then falls, decrease VR1 slightly. (Counter-clockwise rotation.)

Once steady-state oscillation occurs set VR1 at TP3 to the correct voltage. (After warm-up occurs it may be necessary to re-trim VR1.)

The output level at the "+" terminal should be approximately 10V P-P.

Check the level at the output "-" terminal. It should also be about 10V P-P.

Connect a 604 Ω load resistor from the "+" and "-" outputs and monitor the "+" output with the oscilloscope.

Vary the Output Level control throughout its range to make sure that the oscillator stabilizes at each setting. At the very low end of adjustment stabilization takes longer. If the output is allowed to saturate at maximum level for a long period recovery will also take longer.

After a warm-up period re-adjust VR1 so that TP3 reads +300 mV. In normal operation the voltage at TP3 will vary slightly with temperature, output level and load.

Adjust the notch filters

The notch filters can be used balanced or unbalanced depending on jumpers J2-J4. The following tests adjust the notch filter depth for the unbalanced configuration.

Make certain that J2, J3 and J4 are linked.

Connect the balanced Oscillator Output to the Notch Filter Input connector's "+" and "-" terminals with a pair of short jumper wires.

Connect an AC voltmeter and adjust the output level to +20 dBu measured from the "+" output to ground.

To reduce signal contamination disconnect the voltmeter.

Connect an oscilloscope to the Left Notch Output and adjust VR2 and VR3 interactively to minimize the 1 kHz fundamental. When you reach a dip or larger null on one pot move to the other. Repeat. Adjustment for maximum notch depth (minimum output) may take 5-6 iterations.

Connect an oscilloscope to the Right Notch Output and adjust VR4 and VR5 to minimize the fundamental.

Readjust the notch filter once the oscillator has warmed up.

To switch the Notch Filter to balanced input, remove the shunts at J2 and J3.

The Balanced Notch Output is used to provide common mode rejection and read differential distortion. If the A/D converter used for analysis has a single-ended unbalanced input use only the "+" and "G" connections.

The notch depth is optimized for single-ended use. If VR2-VR5 are adjusted while being used in the balanced configuration they must be re-calibrated for single-ended use when J2-J4 are linked.

J2 and J3 can also be connected to a front panel switch.

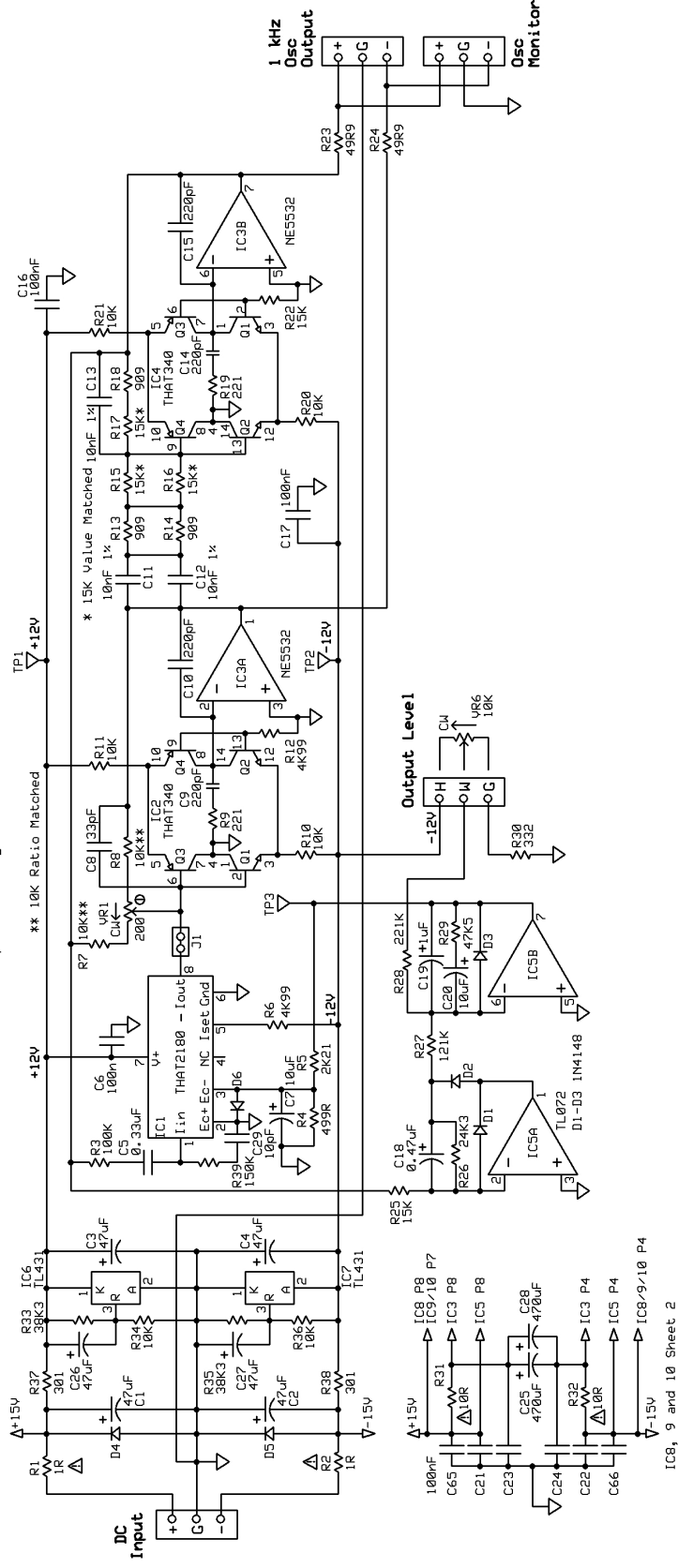
Note that J4 is always linked for switchable balanced/unbalanced use and is only removed when the notch filter components are populated with components for balanced-only operation.

This completes checkout of the Oscillator and Notch Filter.

"ULD0" Ultra Low Distortion Oscillator

(c) 2023 Wayne Kirkwood
www.proaudiodesignforum.com

Δ Vishay PR01 Fusible Resistor



IC8, 9 and 10 Sheet 2

"ULD0" Ultra Low Distortion Oscillator

Pro Audio Design Forum

Wayne Kirkwood
(c) 2023

Rev 1.8 ULD0-21

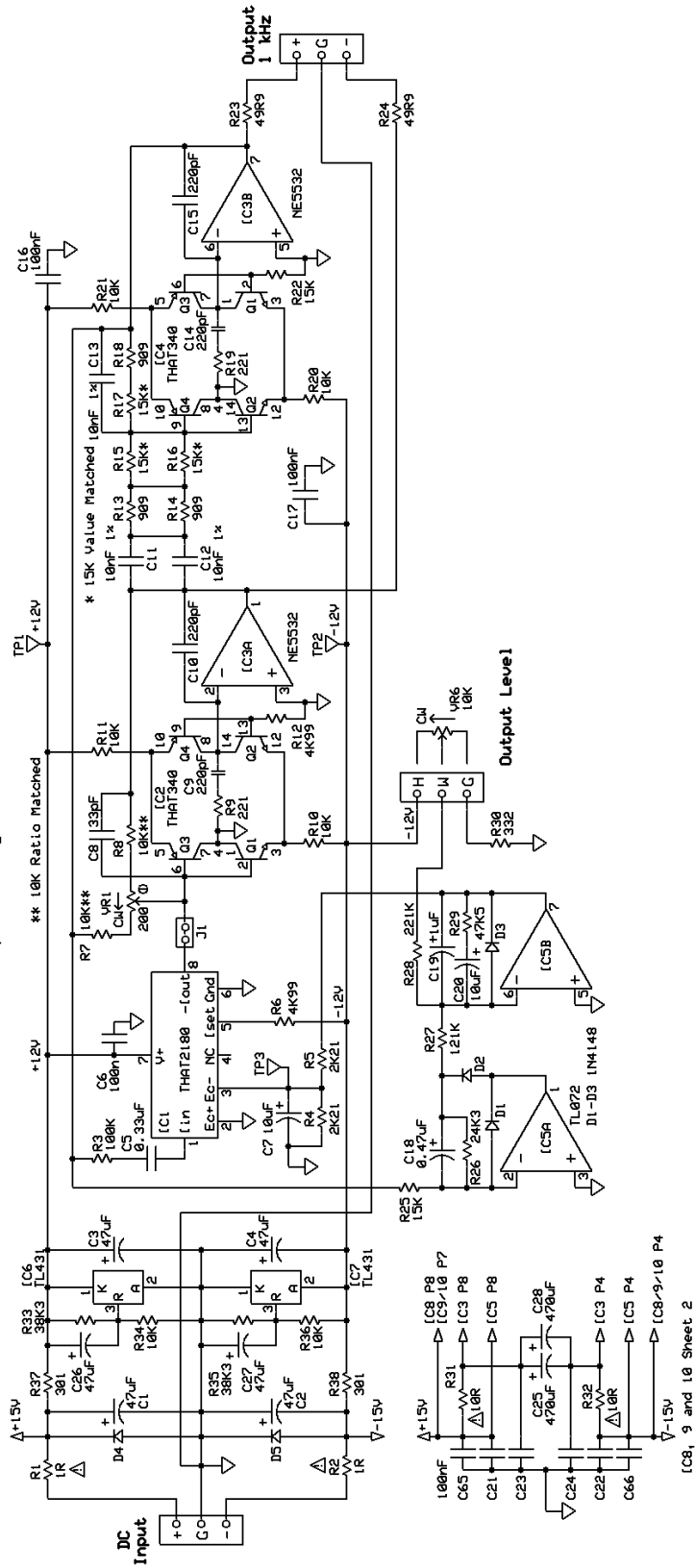
9/2/2023

Sheet 1 of 3

"ULD0"
Ultra Low Distortion Oscillator

(c) 2021 Wayne Kirkwood
www.proaudiodesignforum.com

△ Vishay PR01 Fusible Resistor



"ULD0" Ultra Low Distortion Oscillator

Pro Audio Design Forum

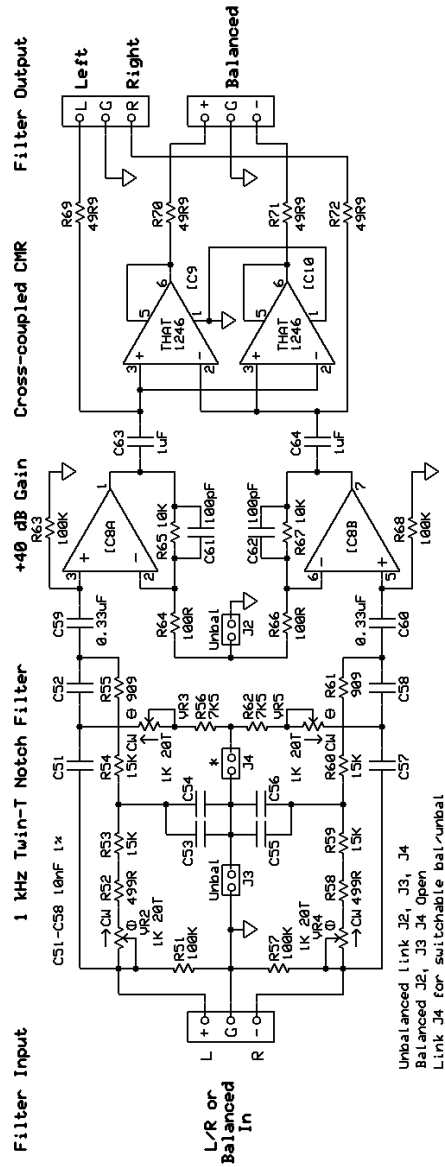
Wayne Kirkwood

Rev 1.5

Sheet 1 of 2

7/8/2021

"Nacho" 1 kHz Notch Filter
(c) 2021 Wayne Kirkwood



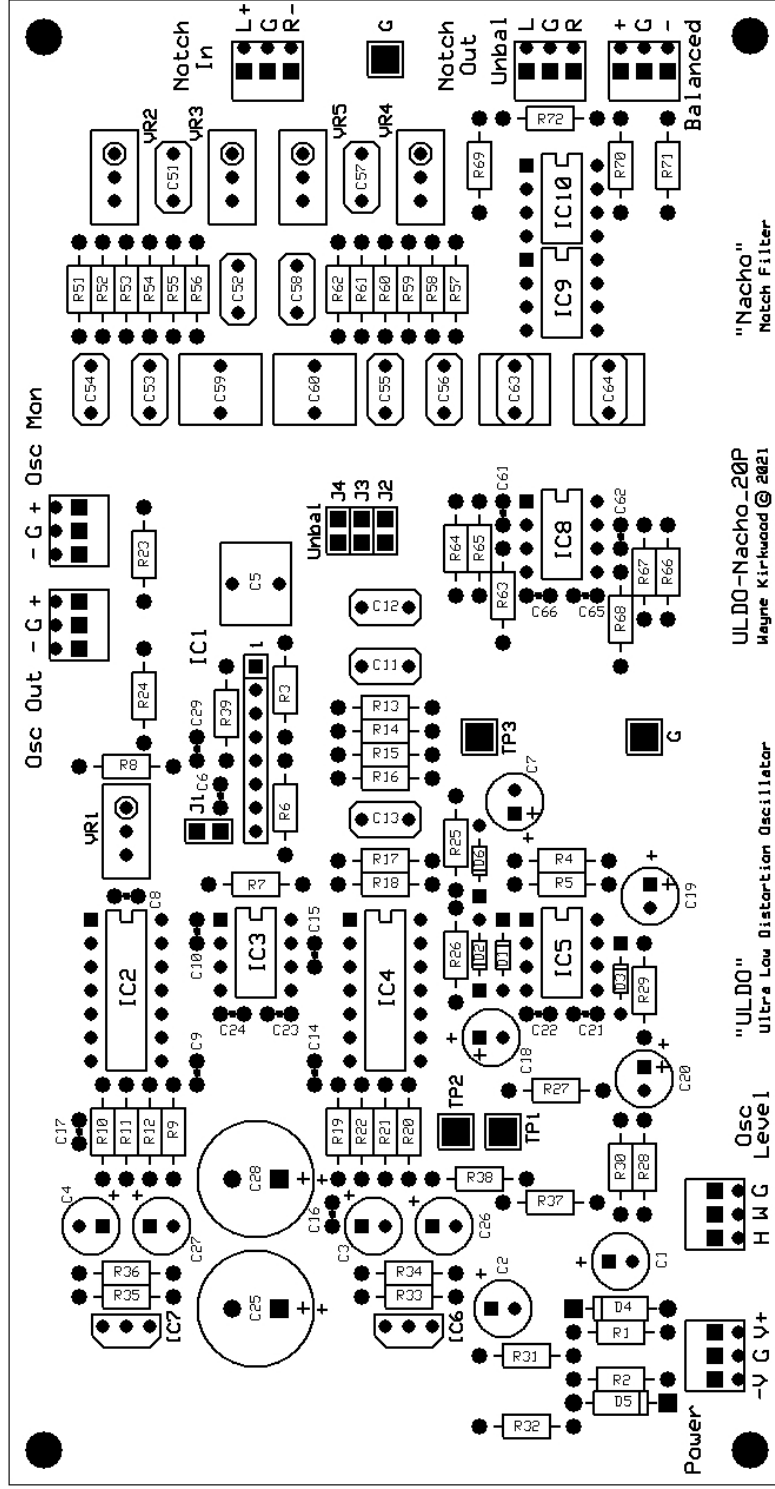
"Nacho" 1 kHz Notch Filter

Pro Audio Design Forum

Wayne Kirkwood
(c) 2021

Rev 1.3
5/3/2021

Sheet 2 of 2



Detailed Parts List

A complete bill of materials is available from Mouser Electronics:

ULDO-Nacho 20 PC Board with THAT ICs:

<https://www.mouser.com/ProjectManager/ProjectDetail.aspx?AccessID=c98f74bf17>

ULDO-Nacho 21 PC Board with THAT ICs:

<https://www.mouser.com/ProjectManager/ProjectDetail.aspx?AccessID=2eba50a733>

Atto Attenuator PC Board:

<https://www.mouser.com/ProjectManager/ProjectDetail.aspx?AccessID=315cb79440>

Other Resources

Pro Audio Design Forum Build Thread:

<https://www.proaudiodesignforum.com/forum/php/viewtopic.php?f=7&t=1265>

Pro Audio Design Forum Design Thread:

<https://proaudiodesignforum.com/forum/php/viewtopic.php?f=6&t=887>

THD calculator to interpret Nacho's FFT readings:

<https://proaudiodesignforum.com/forum/php/viewtopic.php?f=6&t=887&start=130#p16443>

For more information or technical support contact: sales@ka-electronics.com