Neoy2k Boards: Assembly Guide, Schematics...

Sabre DAC Main board, Top view
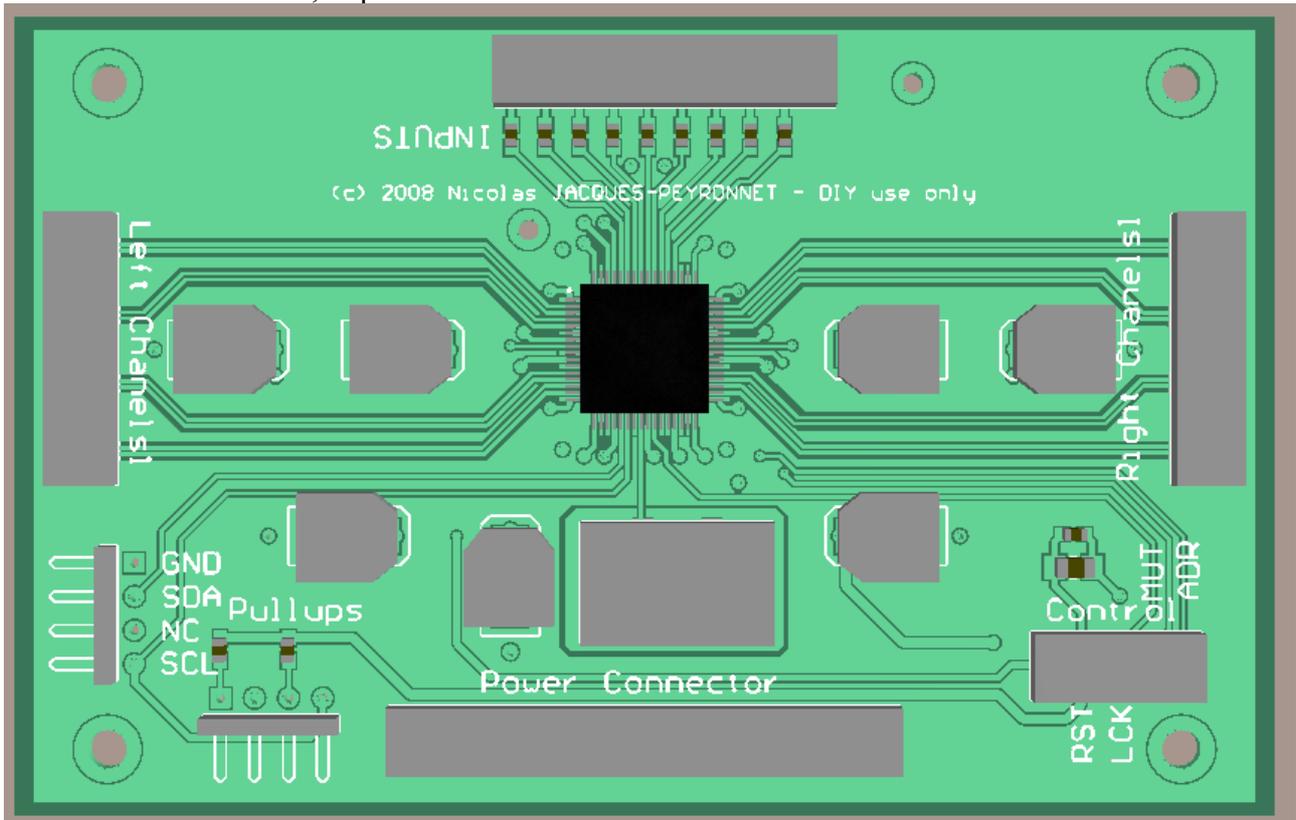
Bottom View
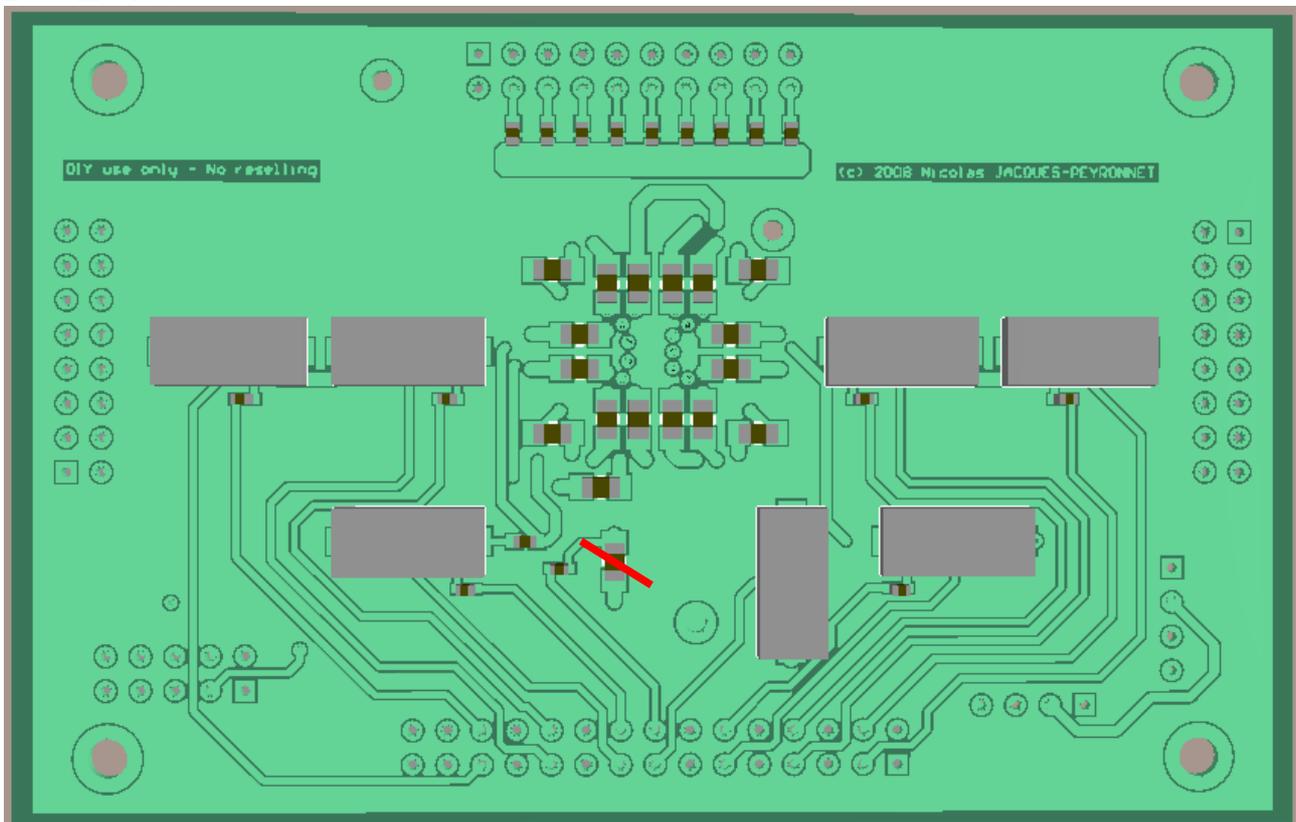
Schematics: See Mainboard.pdf. PCB: See screenshots Mainboard_layer.

BOM:

Sabre	ES9018	1	ESS	9008	Mouser ref
Crystek cchd-950	80Mhz	1	Crystek	CCHD-950-25-80.000	549-CCHD-950-25-80
Header Control	5*2	1	3M	30310-6002HB	517-30310-6002
Header Inputs	10*2	1	3M	30320-6002HB	517-30320-6002
Header Outputs	8*2	2	3M	N3408-6202RB	517-N3408-6202RB
Header Power	15*2	1	3M	D2530-6002-AR	517-D2530-6002-AR
Header i2c	4H	2	Molex	22-05-2041;	538-22-05-2041
Capa C1206	100nF	19	Murata	GRM31C5C1E104JA01L	81-GRM31C5C1E104JA01
Solid Cap	100µF	7	Cornell Dubilier	ESRD101M06B	598-ESRD101M06B
FairRite	600Ohms	7	Fair Rite	2744555577	623-2744555577
Shunts 0402	0 Ohms	7	Vishay/Dale	CRCW04020000Z0ED	71-CRCW0402-0-E3
In Res 0603	22 Ohms	9	Vishay/Dale	CRCW060322R0FKEA	71-CRCW0603-22-E3
In res g 0603	47kOhms	9	Vishay/Dale	CRCW060347K0FKEA	71-CRCW0603-47K-E3
Res gen 0603	3,3kOhms	4	Vishay/Dale	CRCW06033K30FKEA	71-CRCW0603-3.3K-E3

/!\ The Outputs header is a wrong one. It is a latching one, very usefull, but too large: it blocks mounting holes and forces to solder the i2c header on the bottom side. Use a standard 16 pins 2 rows header to solve the problem.

Building Guide:

A] TOP
1) Solder the ESS9008 or ESS9018. Beware of orientation! There is a pin 1 mark, follow it.
2) Solder the clock adapter: have the board in the orientation shown. The adapter board will be in this orientation. You must match the pads of the bottom of this board to the pads for the clock on the mainboard. Use the two left vias as guides. You may heat them to transfer heat to solder paste underneath but I only tested oven reflowing. Solder one C0G 100nF cap at the left, match clock output pin to the upper left pad (vertical orientation). /!\ In the datasheet, the pin drawing is from BOTTOM VIEW (that's why I made the mistake and we now need an adapter....).
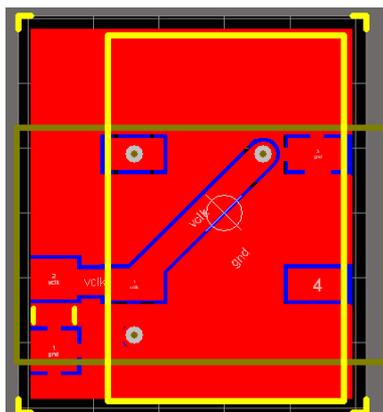
3) Solder 9* 0603 22R Resistors near the input connector (Current limiting resistors)
4) Solder 3* 3k3 0603 Resistors at Pullups and up Control header. (Reset)
5) Solder 1* 100nF C0G 1206 cap near the Control Header (Reset)
6) Solder the 7 100µF Caps.

B] BOTTOM
1) Solder the 16* 100nF C0G 1206 caps (Decoupling). Do not solder the lower one (with a red cross on the 3d screenshot of first page). You may solder an additional one if you want though (clock decoupling).

2) Solder the 9* 47k 0603 resistors at the input connector (pull-down resistors)
3) Just on the right of the lower left FairRite Bead must be a 3k3 resistor (Address pull up).
4) Solder the 7 Fair Rite beads
5) Solder 0402 shunts or put blobs of solder on the lines you want remote sensing.

C] Solder Headers.

On right of input connector are 2 holes, shunt them with a wire to connect EMI RFI screens to ground (or connect each directly at GND of your power supply).

On top left of the ESS is a hole, connect a pin: This is the GND pin of the whole board. It was placed very close to the chip for best performance, may not be the most handy but...

D] You're done! This is not an easy to solder board so check and double check with the tester....

Ask me if you want the original Altium Designer files. I couldn't export PCB in PDF...

How to use:

Power Connector:
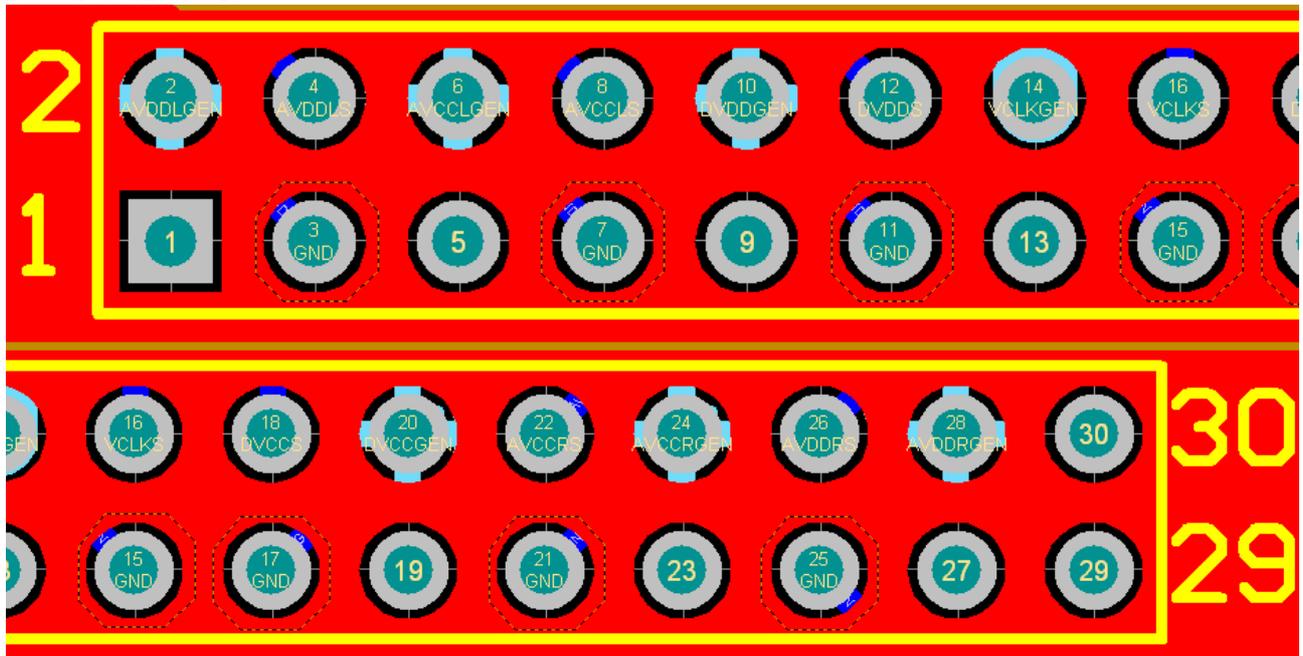
Unassigned pins are not connected on the board. On your cable, you sould pull them to ground to provide isolation (ground connected only on outputs, not on inputs, to avoid current noise crossings).

GND labelled pins are ground sense pins to refer regulators locally near the chip ground pins.This is optionnal.

XxxS labelled pins are the voltage sense lines, that are connected to the load supply only if you put a blob of solder on the 0402 footprints. Otherwise, do not solder (it would put an antenna on the supply line), and connect them to ground on your cable (no floating trace). Optionnal, of course.

XxxGEN labbeled pins are the power force inputs. Labelled to what they actually connect to (refer to schematics and datasheet).

The ground pin is on top left of the ESS chip, not on the connector. It may not be very handy, but at least ground is close to the chip....


Control Header
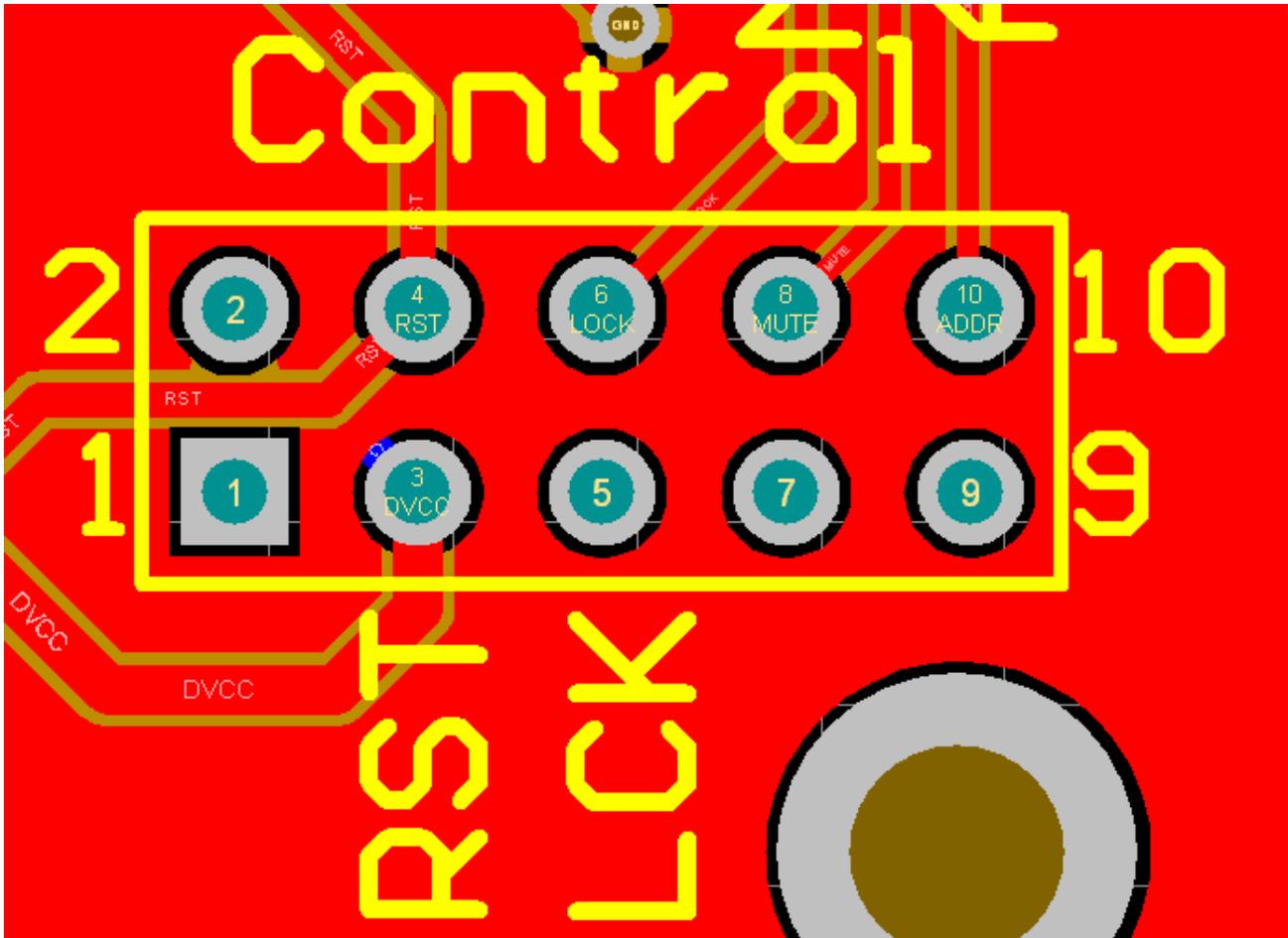
RST (4): Tie to DVCC(3) for a short time to reset the board (I never needed it....).
Lock: Will output 3v3 when locked on input signal
Mute: Will output 3v3 when AUTOMUTE enabled
Addr: Tie to ground to change the i2c address.
Pin 5, 7, 9 should have been ground, this is an error. I don't remember if it was corrected prior to sending to the board, so check continuity to ground if you want to use them, they may be connected...

I2C
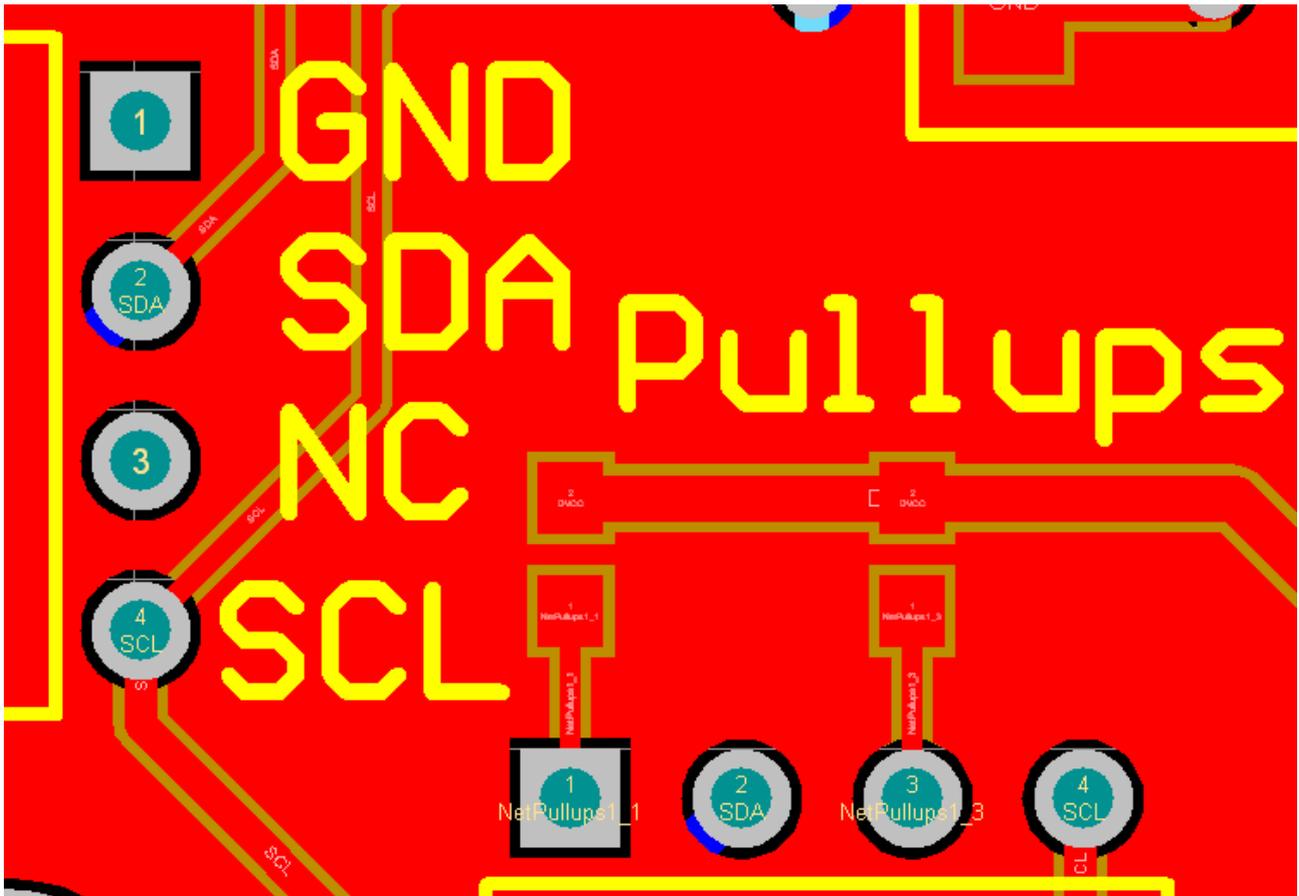
Well, you get the idea...

Use shunt on the connector to add local pull up resistors to SDA and SDL if needed (if you don't already have on your i2c master or on ANY other device).

Connecting them if they already are to another ref may destroy these pullups resistors, and maybe more...

Output connectors:

No need for a drawing... Follow schematics. 1, 3, 5.... pins are connected to ground (this is an output) and should be connected as reference for the IV stage.

Input connector:
Follow ESS datasheet and schematics. No ground on this connector as it is an input.
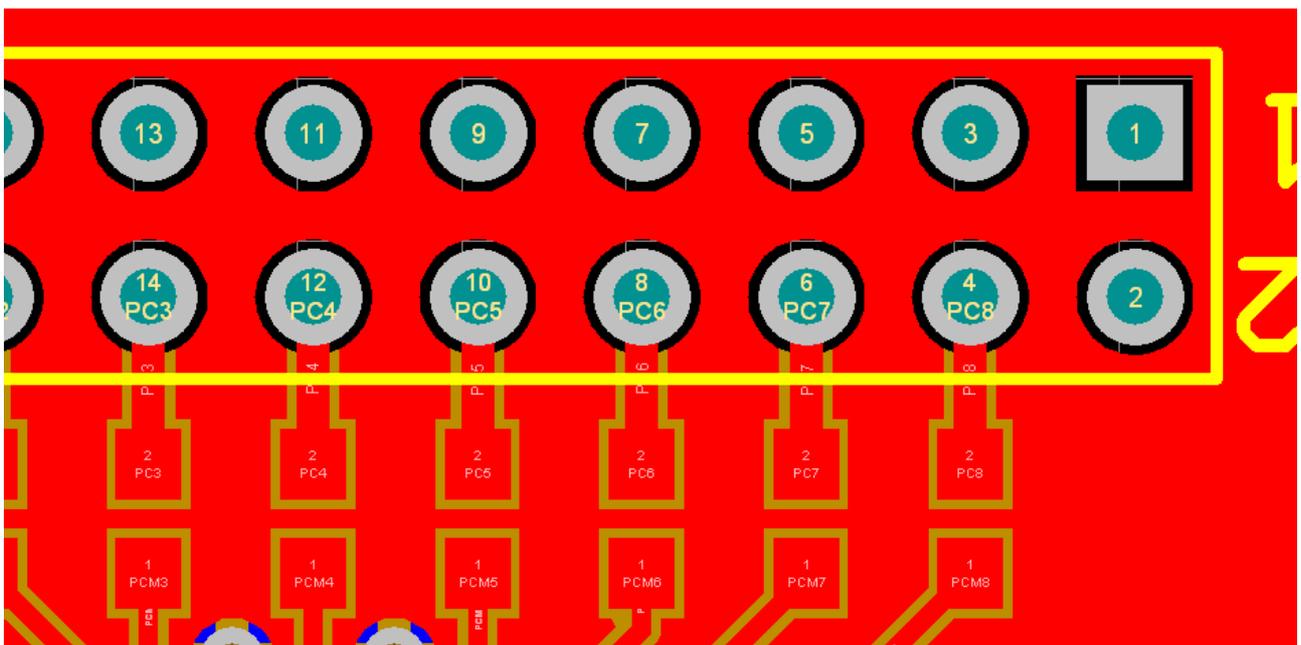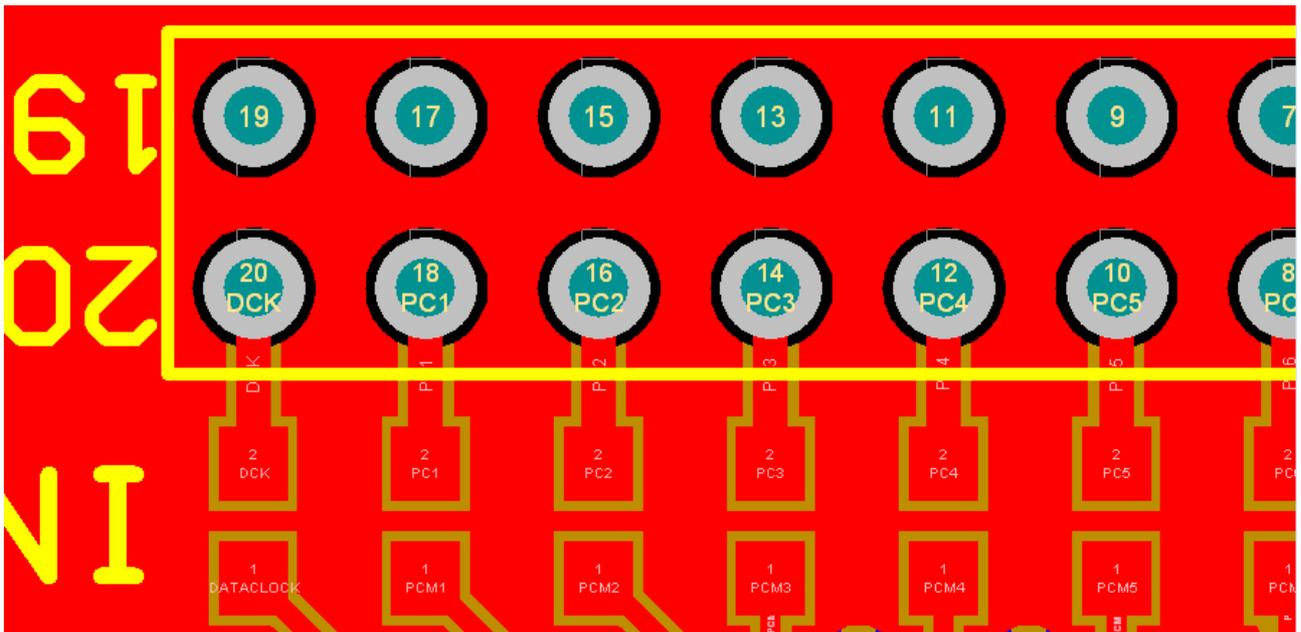
It will automaticcaly detect SPDIF/I2S. It will need software programming via i2c to switch to DSD.

Enjoy the sound! Beware, this DAC has a greater dynamic range than your hearing...