Fachhochschule
Dortmund
University of Applied Sciences and Arts

Hardware Project

Audio Mixer
(A case study on development of a PCB using Circuit Maker)

Guided by Prof. Dr. Peter Schulz

Author:

Mariam Jamal:7206873

TABLE OF CONTENTS:

Introduction

Audio Mixing is one of the major kinds of Signal processing and an interesting study of all time. We have moved and improved from a technology where Audio Mixers weren’t compact, but enormous and ponderous, to the technology where even the AR/VR audio professionals could use it in their devices. The audio engineer, the maestro behind such developments, can be thought of being analogous to an orchestra conductor, ensuring that all of the individual audio sources mesh together.

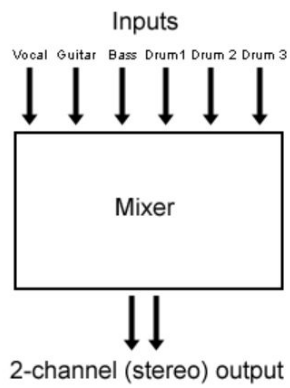


Fig. 1 Audio MIxer

Audio Mixer

An audio mixer is a device with the primary function to accept, combine, process and monitor audio. Apart from combining signals, mixers also allow to adjust levels, enhance sound with equalization and effects, create monitor feeds, record various mixes, etc.

Applications:

Some of the most common uses for sound mixers include:

-Music studios and live performances: Combining different instruments into a stereo master mix and additional monitoring mixes.
-Television studios: Combining sound from microphones, tape machines and other sources.
-Field shoots: Combining multiple microphones into 2 or 4 channels for easier recording.

Fachhochschule Dortmund
University of Applied Sciences and Arts

Problem Statement

The given task is to design a PCB schematic diagram for an FPGA based audio mixer which has 4 input channels each of 16 bits. Each of these input channels are multiplied by different gain factors each of 10 bits for the target channel. Additionally, each channel is also multiplied by master volume control. These target channels are the sum of weighted input channels and are then sent as a TDM stream to generate a 24 bit output.

FPGA design and development:

- Input:
  4 channels: 16 bit
  Serial TDM, 48kHZ Frame Rate
  Format: 1.15 fixed point, signed

- Output
  2 channels: 24 bit
  Serial TDM, 48kHZ Frame Rate
  Format: 1.23 fixed point, signed

- Gain Controller:
  -30dB to +30dB
  Format: 5.5 fixed point, unsigned

- Master Volume Controller:
  -30dB to +30dB

- Total number of multiplier instances = 2

System Architecture Block Diagram

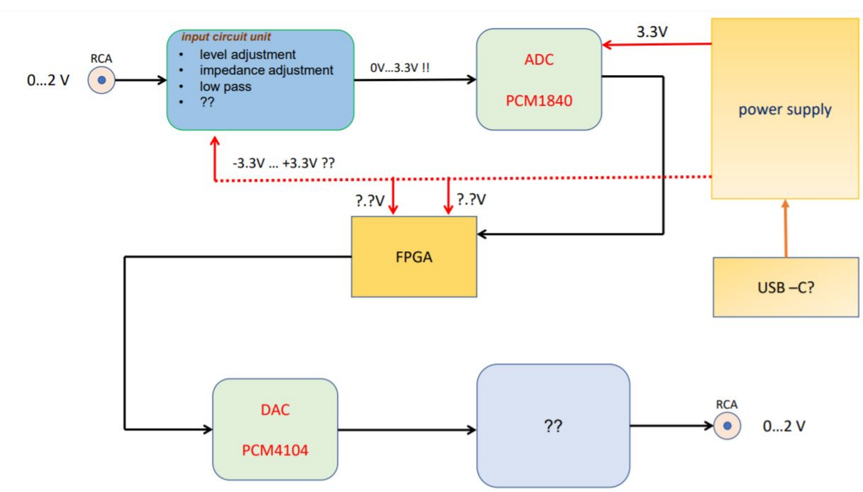


Fig. 2 Block Diagram

Project Task

1. Defining the project and initial design idea development through a block diagram
2. Research and selection of component/s
3. Cost Management of Project
4. Reading and understanding the datasheet
5. Setup of PCB design environment (Circuit Maker)
6. Schematic Design and Footprint of a component
7. Overall Layout and Schematic Diagram

Design Flow

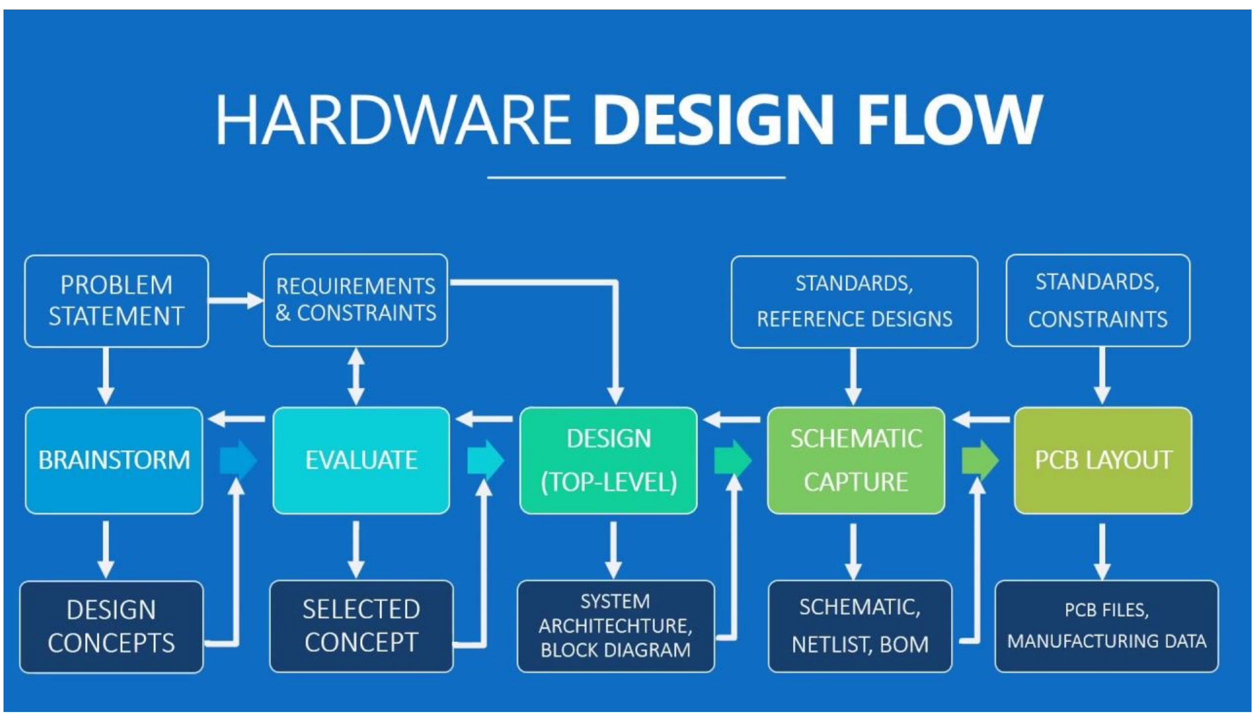


Fig. 3 Design Flow of PCB Design [2]

PCB Design

What is PCB?

> "A **printed circuit board (PCB)** is a rigid structure that mechanically supports and electrically connects electrical or electronic components using conductive tracks, pads and other features etched from one or more sheet layers of copper laminated onto and/or between sheet layers of a non-conductive substrate. Components are generally soldered onto the PCB to both electrically connect and mechanically fasten them to it." [1]

A single-layer PCB is very restrictive; the circuit realization will not make efficient use of available area, and the designer may have difficulty creating the necessary interconnections. Incorporating additional conductive layers makes the PCB more compact and easier to design.

Structure/Composition

PCB generally consists of four layers, which are heat laminated together into a single layer as mentioned below:

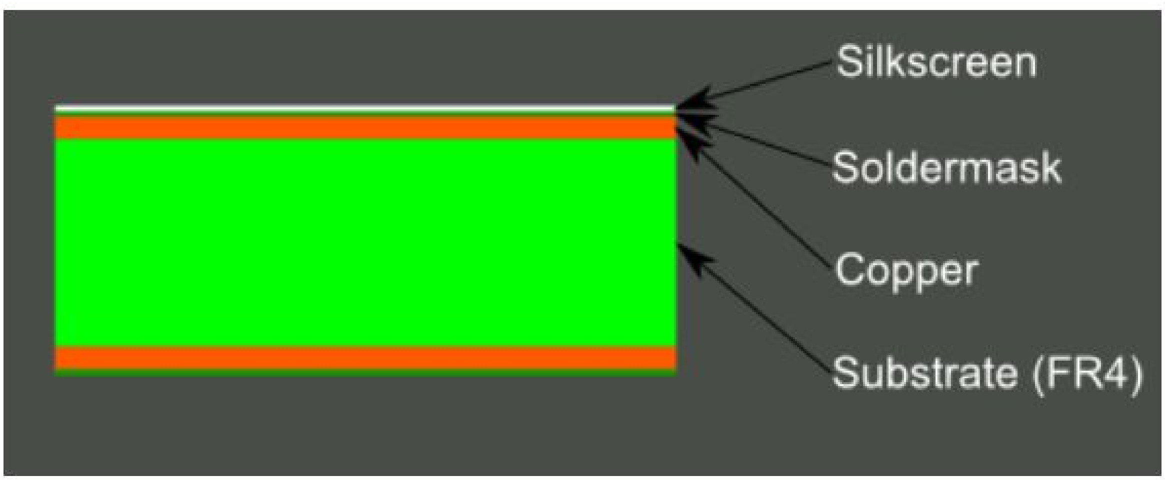


Fig. 4 Composition of PCB

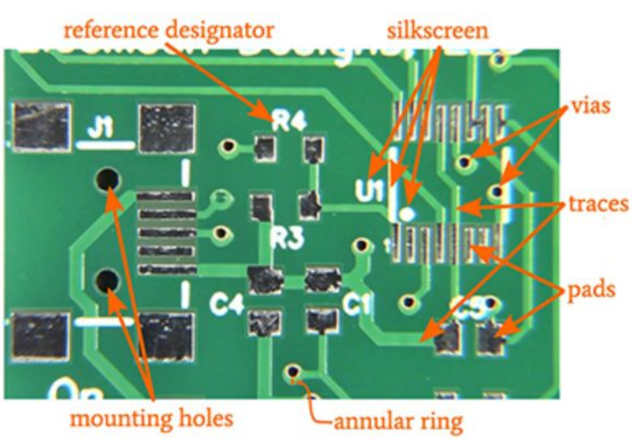
1. **Substrate**
   It provides a solid foundation for the circuit board. A dielectric core material and poor electrical conductivity make the transmission as pure as possible. The typical dielectric material for circuit boards is a flame resistant composite of woven fiberglass cloth and epoxy resin known as FR-4..

2. **Copper foil**
   The next step after finishing the substrate is a piece of copper foil that is coated with an adhesive and then heat-fixed in the foundation. A piece of renewed resin or prepreg can stand between the copper foil / copper foil and the main part. Prepreg is a resin and That was coated with epoxy resin. Then sandwiched between the base and the copper foil layer or between the copper foil layers.

3. **Soldermask**
   The solder mask gives the circuit board its color. Typically the PCB solder mask layer is green while other manufacturers use other colors, e.g. Red. Insulated pads are used by manufacturers to secure the copper path and pads that are insulated from other pieces of metal made from solder or other conductive material. It is also used to control the user by showing which parts of the board can be repaired(soldered) and which parts cannot be repaired.

4. **Silkscreen**
   It is usually in white ink. The Silkscreen is used to add information to the board that makes it easier to put together and makes it easier for people to understand. Usually these descriptions include numbers, letters, and other symbols, such as: function notations for the pins or LEDs.

PCB Terminology



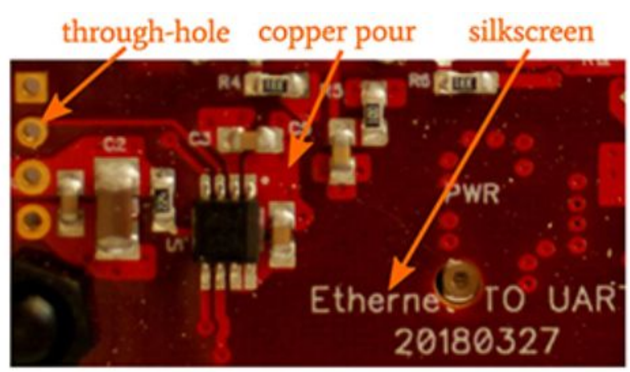


Fig. 5 PCB features

1. **Trace, pads and holes:**
   The connection is called a trace, and the connections for the parts are called pads (for pins placed on the board) and through-holes (for pins in holes drilled in the board). Outline a PCB design that consists of arranging pads and holes so parts can be installed properly, then connect the pads and through holes with traces

2. **Vias**
   Not all holes are designed for inner-hole sections. It is usually necessary to transfer a signal or voltage supply from one part of the circuit board to another, and this is accomplished by the use of small holes, the vias.

3. **Mounting holes**
   Many circular boards also have holes that have a mechanical but not an electrical use and therefore do not need to be plated. The word “plating” in this context refers to material stored in the hole.

4. **Copper pours**
   The copper pours is a very large part of a circuit board filled with conductive material. Copper castings can be used to form a very low resistance or low-inductance connection between the components and improve the operating temperature.

5. **Through-hole**
   A through-hole or via begins like a copper circle and then becomes a hole when a drill bit passes in the circle (ideally in the middle of the circle).

6. **Annular ring**
   This term refers to the amount of copper left after the hole.

7. **Reference Designator and Silkscreen**
   PCB’s contain a lot of "added" information that is useless to the electrical performance of the device. For example, reference designators uniquely identify components, points indicate the appropriateness of component orientation, and project names or serial numbers help us record the number of PCB’s accumulated in them in the laboratory. We see this as silkscreen.

Design Process

1. **Requirement analysis and Component selection**
   The first step in PCB design is to assess needs and select appropriate components such as the processor and power supply. Create a design that fulfills all the requirements.

2. **Circuit Schematic Capture**
   The first step in the board design process is drawing the schematics of the board. You can do this by using a capture tool that allows you to create images by inserting your ICs, vias and connections into the tool, and then plugging in any wires you need. You can also look for a package that you can use to run simulations. This way you can test the active circuit boards before the production process.

   With the acquisition program, the entire electrical design of the department is contained in the file and can be converted into a so-called "netlist". The netlist is the information network and is actually the group of pins and the circle of the node or network that each pin is connected to.

3. **PCB Component Placement**
   The next step in circuit planning is to plan the location of components (ICs, passives, connectors,etc.) on the board. You will be roughly arranging them on your board to see if there's enough space and can fit the items you need the way you want. Once you have this difficult setup done, you can get details of the setup and exact location of each component depending on the number of nets, temperature analysis, and other requirements your product may have (e.g. antenna location. or power supply).

4. **Routing**
   The next step is to plan the path and methods for your communications take what comes from one pin to another or from one component to another. The process usually takes a long time because the data management software you provide and the website map the route on the department head to the various sections you describe. Networks should be shorter (for example: clock signals) or the same length (for example: pair difference) - you should start your sequence action with a signal with a high priority level.

5. **Design Rule Check**
   The PCB layout software can help you verify if your PCB can be made by the PCB manufacturer. All manufacturers have different basic rules for circuit board drawings. By reviewing the regulations, you can quickly identify conflicts and correct errors that may be causing a problem with the circuit board manufacturing process. Such errors can be:
   Two ICs are too close to each other so that the assembly machine cannot reach and place them on the circuit board.

6. **PCB files: Gerber Files etc**
   At the end of this process, you will likely find Gerber Files, a standard for PCB-linked files that contains visual info and plot for your circuit. These files contain not only visual images, but also information about the drilling and the assembly instructions.

Software and Tools

Circuit Maker

CircuitMaker is electronic design automation software for printed circuit board designs targeted at the hobby, hacker, and maker community.

Lattice Diamond

The XP2 chip that we have created our design for, is made to go through HDL design flow of Simulation, Synthesis, logic analysis, Mapping, Place and Route. The tool has been used for verifying if the code can successfully undergo synthesis, could be mapped, placing and route design as per the given processor requirements. It gives the Synthesis Library support before the Synthesizer tool, Synplify Pro is used for it.

For the timing verification, Diamond tool gets the timing data from routing and follows a reverse engineering process to get back the intended result. Thus, this tool has served as the environment for design entry of the audio mixer.

Project Details

1. FPGA based solution

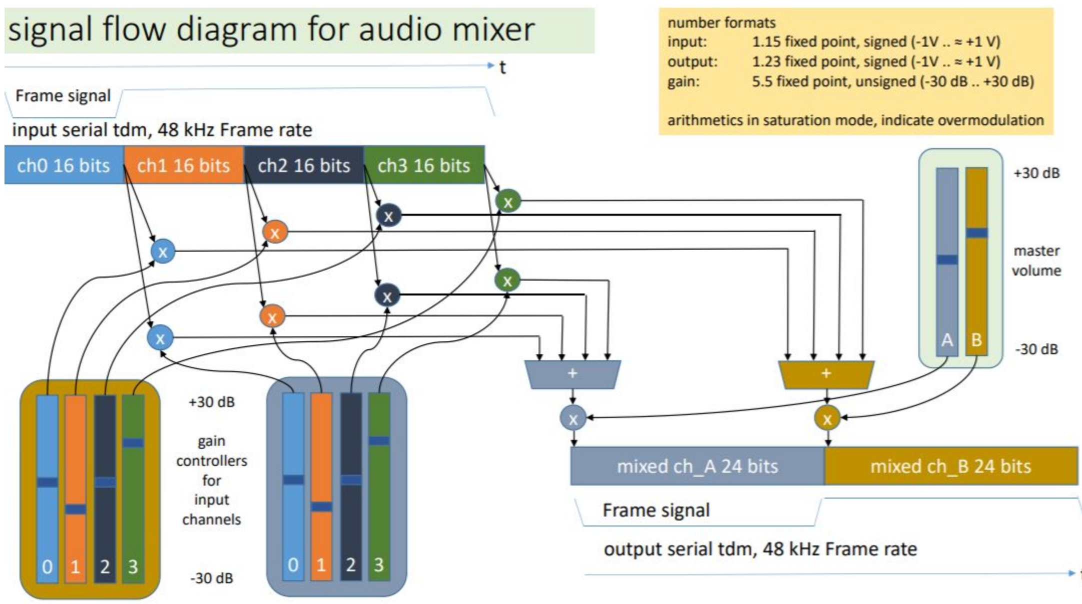


Fig. 6 Signal Flow Diagram for FPGA

RTL Schematic Diagram:

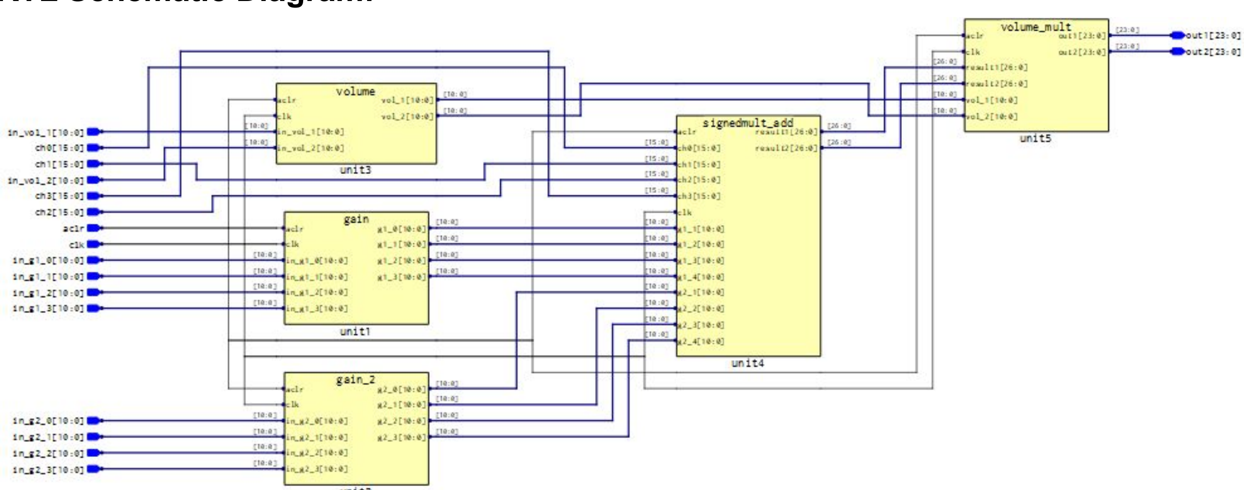


Fig. 7 RTL Schematic Diagram

2. PCB Design Environment (Circuit Maker)

<u>Setup:</u>

1. Download and Install Altium Circuit Maker
2. Create a new account
3. Login to the account and create a new project

3. Component Selection Process

a. Research:

- Internet based research through a search engine such as Google
- Through Circuit Maker

  a. Go to Components tab, and search for the desired components

OR

  b. View > Libraries > Octopart (from drop-down menu) and search for the desired components

- Directly through Octoparts website

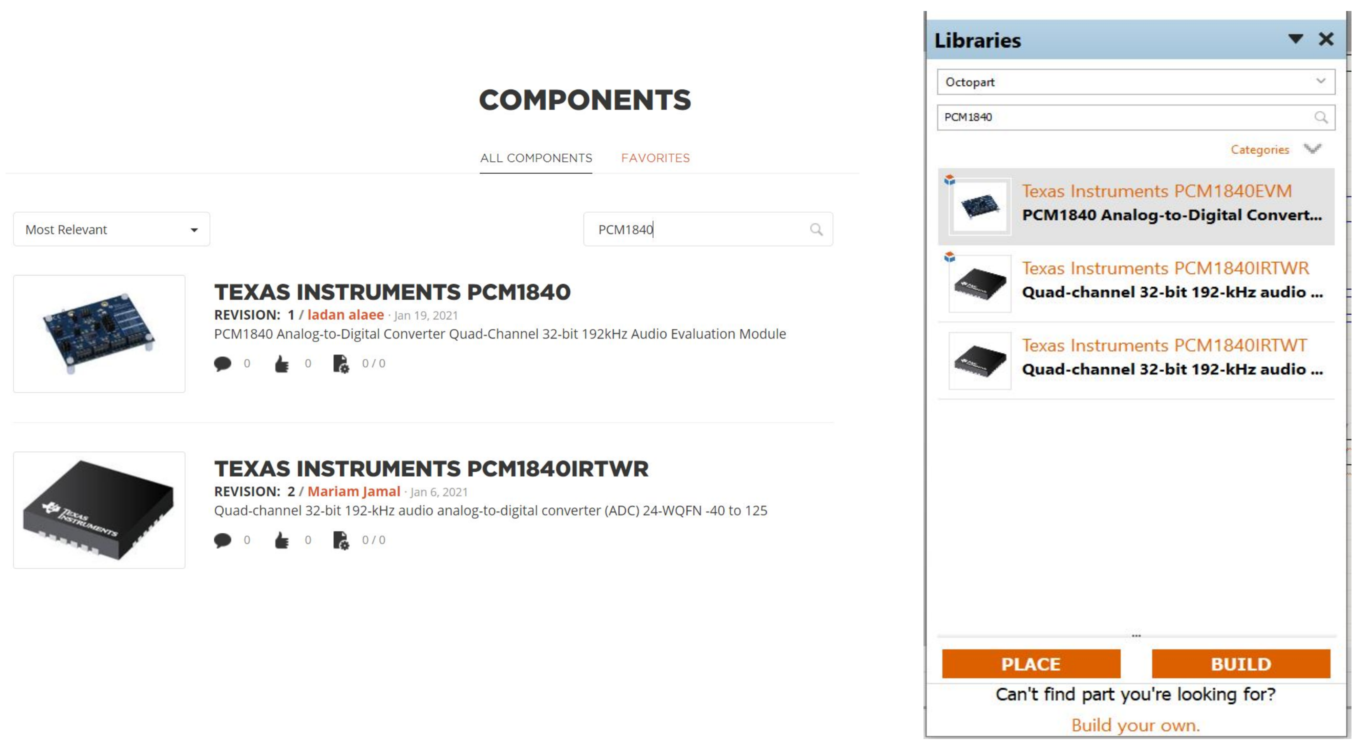
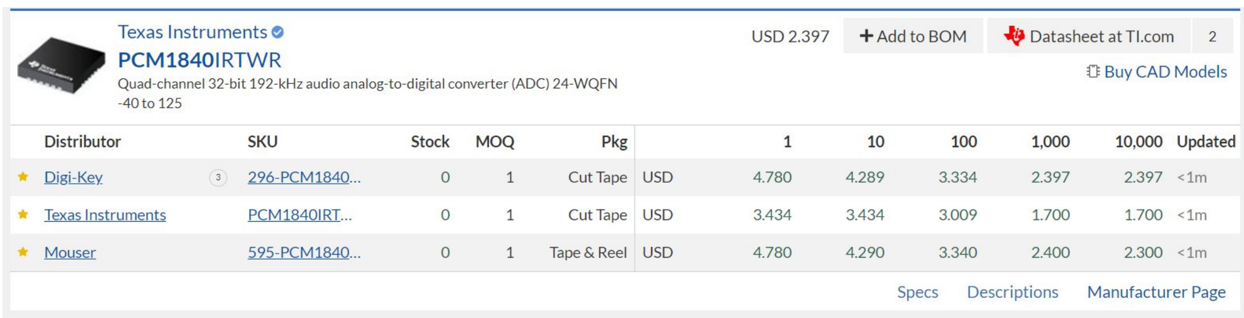


Fig. 8 Component Selection in Circuit Maker

b. Compare

- Compare and select from different options available based on requirements, cost, discussions or other criterias
- The cost per piece and per multiple of 10 pieces can be easily seen on Octopart's website.
- The datasheet can also be accessed easily



Distributor	SKU	Stock	MOQ	Pkg		1	10	100	1,000	10,000	Updated
Digi-Key (3)	296-PCM1840...	0	1	Cut Tape	USD	4.780	4.289	3.334	2.397	2.397	<1m
Texas Instruments	PCM1840IRT...	0	1	Cut Tape	USD	3.434	3.434	3.009	1.700	1.700	<1m
Mouser	595-PCM1840...	0	1	Tape & Reel	USD	4.780	4.290	3.340	2.400	2.300	<1m

Fig. 9 Comparison of different component and prices

c. Place

Simply drag it on to an open Schematic design document, click the place button, or right-click the entry and select Place from the context menu.

4. Component Generation Process

If you cannot find the component on circuit maker, then we can build our own schematic symbol and footprint for the component which will be saved in the vault of circuit maker and can be used by other users.

A Schematic Symbol is a component's graphical object that represents the component when it is placed in a schematic document. The Symbol includes electrical connection information (indicated by component Pins) that allows a circuit to be logically wired together, and to be matched to its equivalent Footprint connections (Pads, etc) in the PCB domain.

Steps to generate the Schematic Symbol:

1. Click on build your own and then add the name and description
2. Click on add new symbol
3. Symbols are created by placing shapes and applying the drawing tools, and importantly, by including connection Pins that define the component's electrical wiring points in a schematic document.

4. To edit the schematic symbol's properties, open the Schematic Symbol Properties dialog by selecting Home | Library » Component Properties
5. Save the symbol using File » Save

A PCB Footprint is a component's graphical object that represents its physical and connectivity form when it is placed in a PCB document. The Footprint includes both electrical and mechanical connection information (primarily indicated by PCB Pads) and allows the components to be interconnected by tracks in a board layout design.

Steps to generate the Footprint:

1. Click on add new footprint
2. Footprints are created by placing pads, tracks, lines, arcs, 3D elements, etc on suitable PCB layers to accurately represent the physical and electrical attributes of the component in the PCB domain.
3. To edit the footprint's properties, open the PCB Library Component properties dialog by selecting Home | Library » Component Properties
4. Save the symbol using File » Save

5. Project Components

a. RCA:

It is the interface to the input of the system.

b. Signal Conditioning Circuit for ADC:

First, the signals made by the sensor should be as free of noise as possible. In addition, the normal frequency of the signal, such as its bandwidth, must be limited to a reasonable extent due to some constraints. This is often required to use what is known as an anti-aliasing filter.

Second, signals generated by signal processing devices, whether voltages, currents or electrical current, often have weak currents. In order for the signal to process correctly and to further strengthen the system, the signal must be amplified.

In addition to filtering and amplification, the need to convert the signal to digital numbers using an analog-to-digital converter (ADC) increases some additional conditional requirements. In addition to amplifying the signal, it may also be necessary to convert the signal to match the different ADC voltage. Also, most ADCs ,only have unipolar inputs working. That is, the correct voltage cannot vary between positive and negative conditions with respect to the ground. In such cases, a level shifter is required.

For our project, the following Signal conditioning circuit is used which has LM358B Op-Amp with a gain factor of 1.65 and LT5400-4 Quad Matched Resistor Series Network

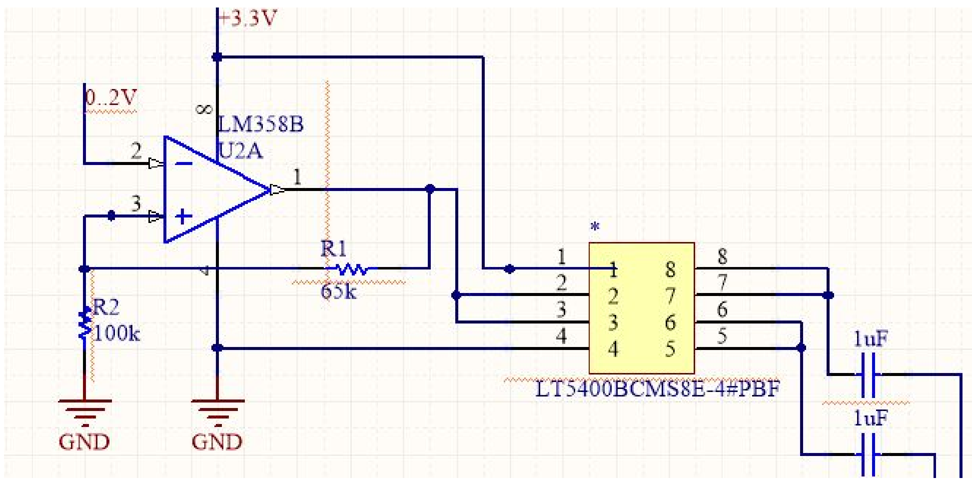


Fig. 10 ADC Signal Conditioning Circuit

c. ADC (Analog to Digital Converter)

"Analog-to-Digital converters (ADC) translate analog signals, real world signals like temperature, pressure, voltage, current, distance, or light intensity, into a digital representation of that signal. This digital representation can then be processed, manipulated, computed, transmitted or stored." [13]

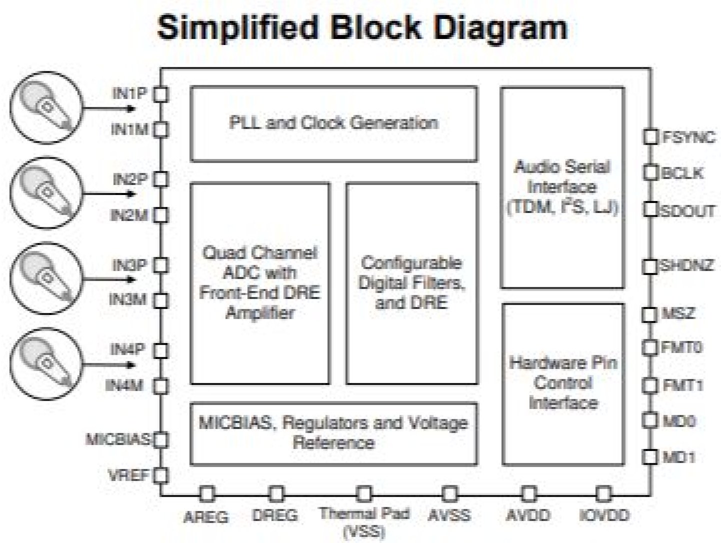


Fig. 11 PCM1840 ADC Block Diagram

ADC selected for this project is PCM1840 which is a Quad Channel, 32-Bit, 192-kHz, Burr-BrownTM Audio ADC.

-It is connected to Bank 6 of the FPGA device
As seen in the block diagram,
-It has 4 analog differential inputs, 5 control pins (MSZ,FMT0,FMT1,MD0,MD1), 1 serial output pin SD0 and other pins.
-SHDNZ is the device hardware shutdown and reset (active low) pin
-It works on 3.3V
-This device can be operated in two modes, Master and Slave.
In Slave mode, FSYNC and BCLK are inputs and in Master mode, outputs.
FSYNC: 48Khz
BCLK: 6.144 MHz

-This can be configured using MSZ pin
We use the Slave Mode.

Table 1. Master and Slave Mode Selection

MSZ	MASTER AND SLAVE SELECTION
LOW	Slave mode of operation
HIGH	Master ode of operation

-The bus protocol TDM, I2S, or left-justified (LJ) format can be selected by using the FMT0 and FMT1 pins.
We use 4-Channel TDM mode.

Table 2. Audio Serial Interface Format

FMT1	FMT0	AUDIO SERIAL INTERFACE FORMAT
LOW	LOW	4-channel output with time division multiplexing (TDM) mode
LOW	HIGH	2-channel output with time division multiplexing (TDM) mode
HIGH	LOW	2-channel output with left-justified (LJ) mode
HIGH	HIGH	2-channel output with inter IC sound (I^2S) mode

-The device record channel includes a high dynamic range, built-in digital decimation filter to process the oversampled data from the multibit delta-sigma (ΔΣ) modulator to generate digital data at the same Nyquist sampling rate as the FSYNC rate. The decimation filter can be chosen from two different types only in slave mode, depending on the required frequency response, group delay, and phase linearity requirements for the target application. The selection of the decimation filter option can be done by the MD0 pin.

Table 7. Decimation Filter Mode Selection for the Record Channel

MD0	DECIMATION FILTER MODE SELECTION (Supported Only in Slave Mode)
LOW	Linear phase filters are used for the decimation in slave mode. For master mode, the device always use linear phase filters for the decimation.
HIGH	Low latency filters are used for the decimation in slave mode. For master mode, the device always use linear phase filters for the decimation.

-The device integrates an ultra-low noise front-end DRE gain amplifier with 123-dB dynamic range performance with a low-noise, low-distortion, multibit delta-sigma (ΔΣ) ADC with a 108-dB dynamic range. The dynamic range enhancer (DRE) is a digitally assisted algorithm to boost the overall channel performance. The DRE monitors the incoming signal amplitude and accordingly adjusts the internal DRE amplifier gain automatically. The DRE achieves a complete-channel dynamic range as high as 123 dB. At a system level, the DRE scheme enables farfield, high-fidelity recording of audio signals in very quiet environments and low-distortion recording in loud environments.

The DRE can be enable only in slave mode by driving high to the MD1 pin.

Table 20. DRE Selection for the Record Channel

MD1	DRE SELECTION (Supported Only in Slave Mode)
LOW	DRE is disabled in slave mode. For master mode, DRE is always disabled.
HIGH	DRE is enabled with DRE_LVL = –36 dB and DRE_MAXGAIN = 24 dB in slave mode. For master mode, DRE is always disabled.

d. DAC (Digital to Analog Converter

A Digital to Analog Converter (DAC) converts a digital input signal into an analog output signal. The digital signal is represented with a binary code, which is a combination of bits 0 and 1.

DAC selected for this project is PCM4104 which is a High Performance, 24-bit, 4 Channel Audio Digital to Analog Converter

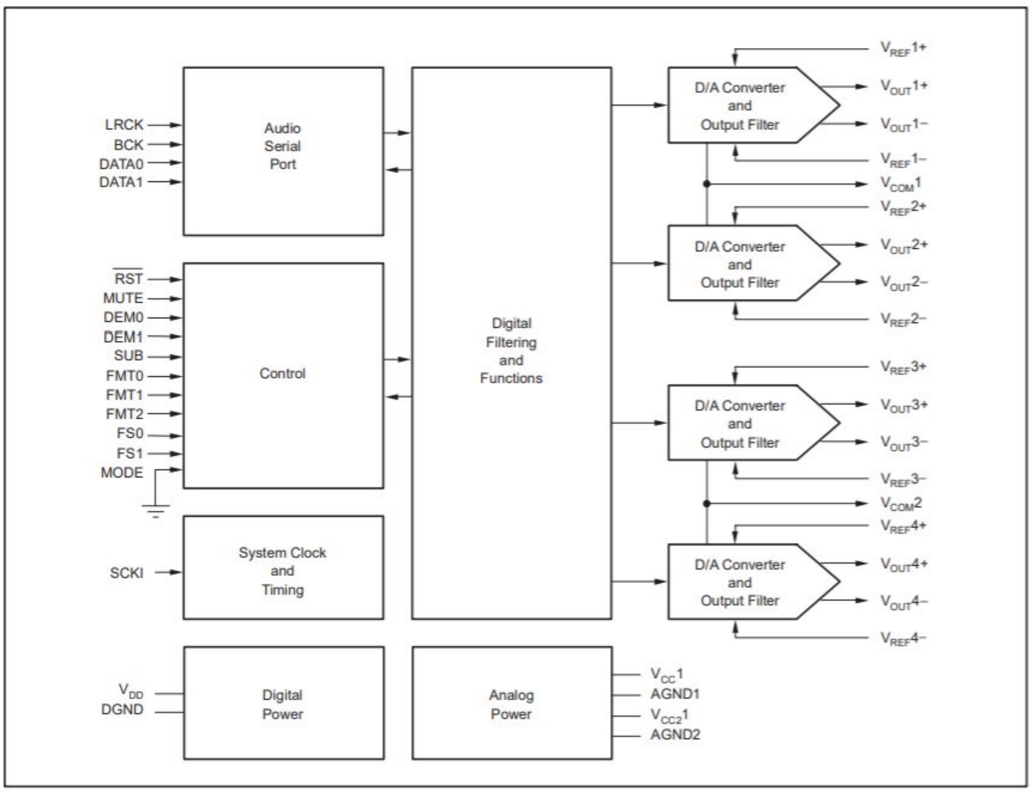


Fig.12 PCM4104 DAC Block Diagram

-It is connected to Bank 5 of FPGA.
As seen in the block diagram,
-Modes of operation; Standalone (MODE=0) and Software(MODE=1). We use the Standalone mode.
-Inputs: 2 inputs DATA0 and DATA1, when using TDM data formats, DATA0
carries the audio data for all four channels, while the DATA1 input is ignored
-Outputs: 4 differential voltage outputs
-Sampling mode: We use Single Rate as it is for <54kHz

Table 2. Sampling Mode Configuration

FS1	FS0	SAMPLING MODE
0	0	Single Rate
0	1	Dual Rate
1	0	Quad Rate
1	1	– Not Used –

-Audio Data Format:

Table 3. Audio Data Format Configuration

FMT2	FMT1	FMT0	AUDIO DATA FORMAT
0	0	0	24-bit left justified
0	0	1	24-bit I^2S
0	1	0	TDM with zero BCK delay
0	1	1	TDM with one BCK delay
1	0	0	24-bit right justified
1	0	1	20-bit right justified
1	1	0	18-bit right justified
1	1	1	16-bit right justified

-Soft Mute: To simultaneously mute the four output channels

Table 4. Mute Function Configuration

MUTE	ANALOG OUTPUTS
0	On (mute disabled)
1	Muted

-Digital De-emphasis: provides de-emphasis of the higher frequency content within the 20kHz audio band. We use the 48kHz mode

Table 5. Digital De-Emphasis Configuration

DEM1	DEM0	DIGITAL DE-EMPHASIS MODE
0	0	Off (de-emphasis disabled)
0	1	48kHz
1	0	44.1kHz
1	1	32kHz

e. Signal Conditioning Circuit for DAC:

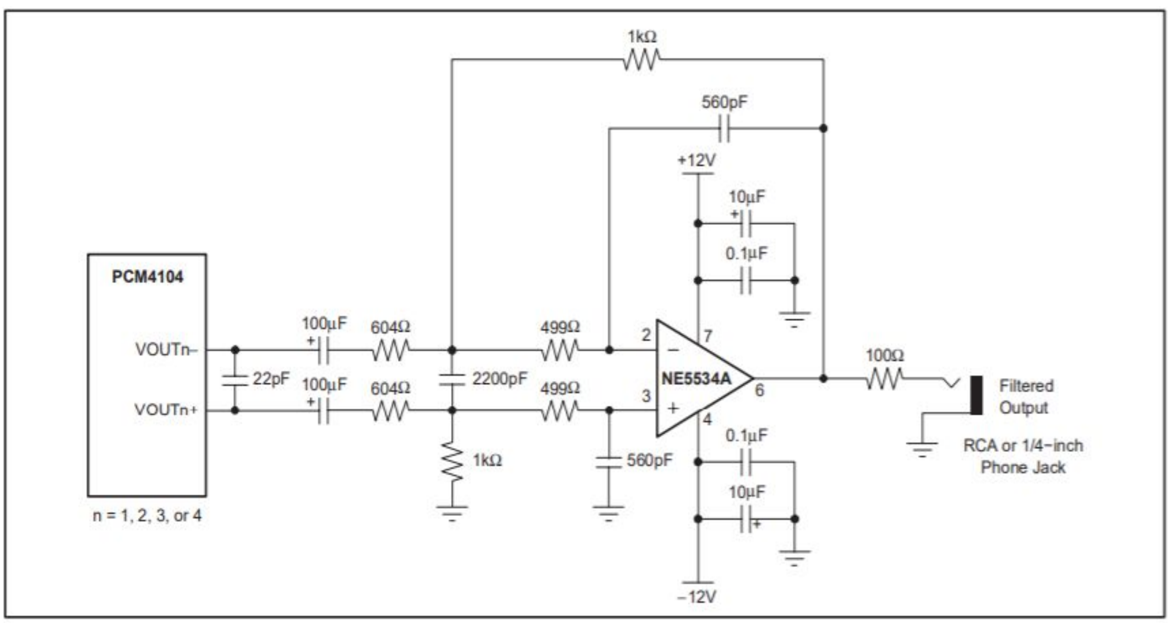


Fig.12 Signal Conditioning Circuit for DAC

6. Schematic Diagram

The entire schematic diagram can be accessed from the attached project files.
It consists of 8 sheets namely:

1. ADC (Bank 5)
2. Connectors for input
3. Connectors for output
4. DAC (Bank 6)
5. Power Supply
6. Filter
7. Control Potentiometer (Bank 4)
8. FPGA

Conclusion

The schematic diagram for the case study on Audio Mixer was successfully designed on Circuit Maker. The design and development of PCB was learned comprehensively.

References

[1] https://en.wikipedia.org/wiki/Printed_circuit_board#Characteristics

[2] https://www.youtube.com/watch?v=7y2JDnO8e28

[3] https://resources.pcb.cadence.com/blog/2019-printed-circuit-board-an-introduction-and-the-basics-of-printed-circuit-boards

[4] https://www.allaboutcircuits.com/technical-articles/what-is-a-printed-circuit-board-pcb/

[5] https://learn.sparkfun.com/tutorials/pcb-basics/all

[6] https://emsginc.com/resources/basics-pcb-design-composition/

[7] https://hardwarebee.com/brief-guide-pcb-design-flow/

[8] https://www.electronics-notes.com/articles/analogue_circuits/pcb-design/how-to-design-pcb-board-basics.php

[9] https://www.tronicszone.com/blog/steps-pcb-design-manufacturing/

[10] https://octopart.com/

[11] https://workspace.circuitmaker.com/Components

[12] https://radiolocman.com/shem/schematics.html?di=278655

[13] https://wiki.analog.com/university/courses/electronics/text/chapter-20

[14] Datasheets mentioned in Appendix

Appendix

1. Datasheets
    a. PCM1840 (ADC)
    b. PCM4104 (DAC)
    c. Quartz 75391
    d. EN5322
    e. Lattice XP2 (FPGA)

2. Schematic Files
3. Bill of Materials
4. Lattice XP2-17 Pinout
5. Circuit Maker Sheets

Affidavit

I, Mariam Jamal, herewith declare that I have composed the present report and work myself and without use of any other than the cited sources and aids. Sentences or parts of sentences quoted literally are marked as such ;other references with regard to the statement and scope are indicated by full details of the publications concerned.