ADC 64 channel Low Pass Filter PCB designing.

JINR Summer Student Program, 2014

Marika Kuczynska
University of Science and Technology, Poland

Supervisor
Andrey Terletskiy

19 September 2014

Abstract

JINR is the one of well developing research institute. In the coming years, it is planned to open few new detectors like EMC() or MPD(MultiPurpose Detector) on NICA(Nuclotron-based Ion Collider fAcility). To be sure, that the prototype detectors works correctly they need to be carefully tested. First step to examine the detector is to build electronic system, which will be processing the signal received from detector. The electronic system is consists of the so-called front-end and back-end electronic. In first step of examination these two electronics can be realized by very basic system like amplifier, low-pass filter and ADC (analogue-to-digital converter). The paper presents the results of 5 week practise in data acquisition system group from JINR. Main goal of work was to design the PCB (printed circuit board) with front-end electronic - low pass filter for 64 channel ADC. Scheme and layout of PCB was made in Altium Designer and will be further described later.

Contents

List of Figures

1 Introdution

1.1 Front-end electronics

The amplifier and the low pass filter are part of the front-end electronics. This means, that they are first elements through which the signal goes.

The amplifier is designed to prepare a small electrical signal for further processing, since the signal coming from the detector istoo weak. The amplifiern is often placed close to the sensor to reduce the effects of noise and interference.

The low pass filter passes only a signal of low frequency, thanks to this, its cutting off noise that are significantly higher than the signal frequency.There are several types of filters used in front-end electronics. Depending on the how the shaped bandwidth of the signal is wanted to receive. In this project, the Chebyshev filters are used. They are characterized by a very sharp edge (rise time and fall time), but there is also a small ripple in the bandwidth of the signal. The higher the filter rank is, the smaller the distortion bandwidth.

1.2 Altium Designer

Altium Designer is a powerful electronic design automation software package for printed circuit board, FPGA and embedded software design. It is developed and marketed by Altium Limited of Australia. It allows to acceleration and significantly simplification designing PCB proces.

One of the tasks of SSP was to familiarize with the operation and the various functions of the program. The end result was to design a PCB board, operating with the Altium Designer.

1.3 LT6600-20 - amplifier and low-pass filter

The LT6600-20 integrated circuit included a fully differential amplifier and a 4th order 20MHz lowpass filter approximating a Chebyshev frequency response . Most differential amplifiers require many precision external components to tailor gain and bandwidth. In contrast, with the LT6600-20, two external resistors program differential gain, and the filter's 20MHz cutoff frequency and passband ripple are internally set. The LT6600-20 also provides the necessary level shifting to set its output common mode voltage to accommodate the reference voltage requirements of ADC [1].

2 Scheme of ADC64LPF

ADC64LPF is the name of designing PCB board, which contains low pass filter and amplifier. All project has two parts: schematic of ADC64LPF and layout of its. At figure 1 there is presented scheme of LT6600-20 and elements like resistors and capacitors, which are proper connected.

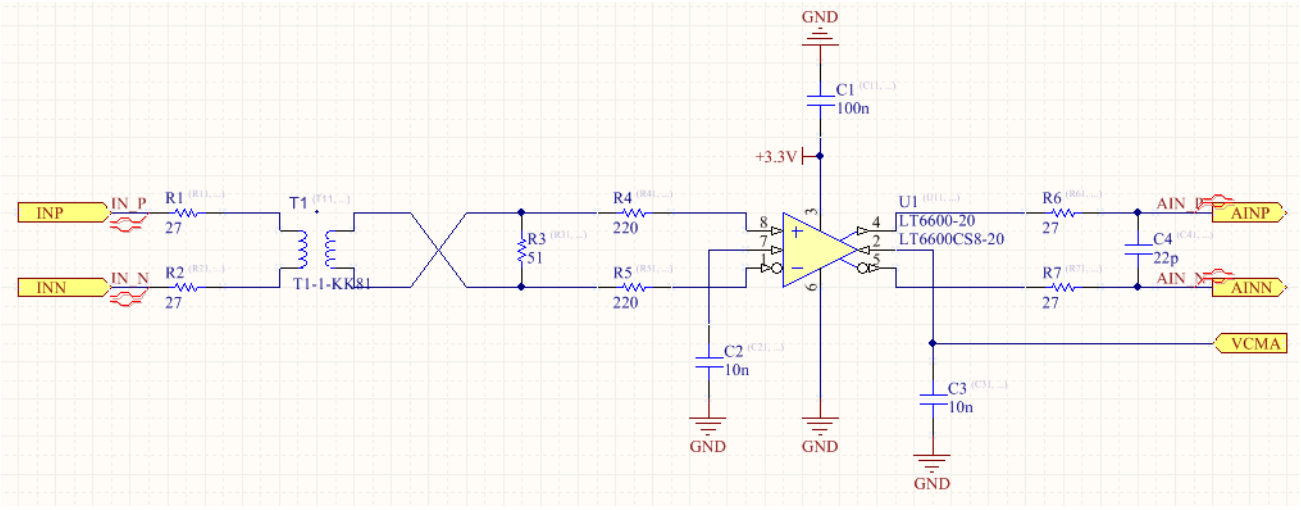


Figure 1: *Diagram of the setup containing LT6600-20.*

Pins 1, 8 are the digital input to the LT6600-20. Signals is applied to both input pins through identical external resistors, $R_{IN} = 27\Omega$.

Pin 2 (VOC) is the DC Common Mode Reference Voltage. It is driven by an external voltage reference with value of 0.8 V. It programs the common mode voltage of the differential output of the filter. Pin 2 is also bypassed with a 10 nF ceramic capacitor.

Pins 3, 6 are power supply pins. Pin 3 goes to ground and 6 is high voltage. In this case is equal to +3.3V. Pin 6 need to by bypass by 100 nF ceramic capacitor, while 3 goes straight to ground.

Pins 4 and 5 are the filter differential outputs.

Pin 7 is internally biased at midsupply. Is bypassed with a quality 10 nF ceramic capacitor to Pin 6. These pin sets the output common mode voltage of the 1st stage of the filter.

Differential output of the construction shown at figure 1 is connected to input of one channel of ADC. That is to say that this scheme need to be duplicated 64 times, because ADC has 64 channels. Realization of duplication is shown at figure 2.

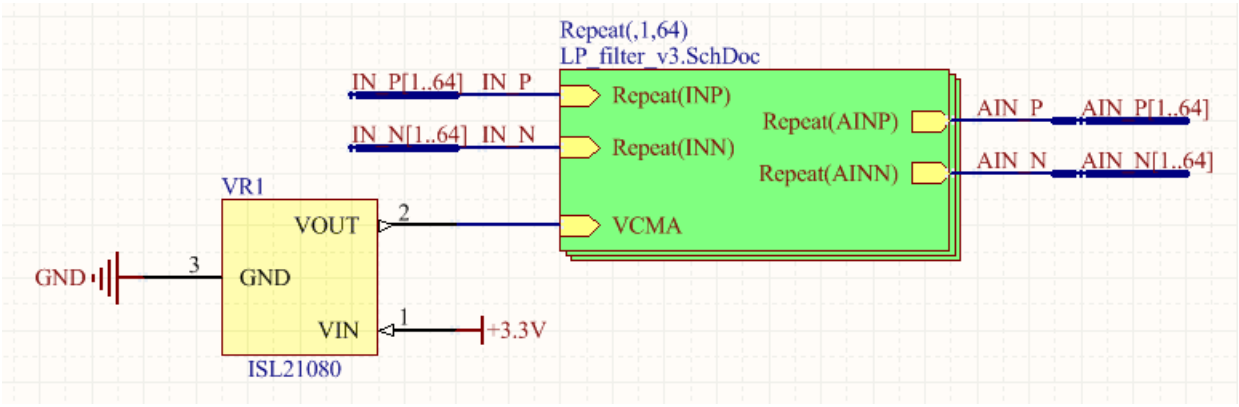


Figure 2: *Way of duplication of LT6600-20 diagram.*

Element ISL21080 is a analog voltage references. It defines reference voltage for common mode, and it has a value of 0.9 V. Since the element has not been used so far, it has been added to the library of Altium by drawing a diagram and footprint according to [2]. Both of them are shown at figure 3.

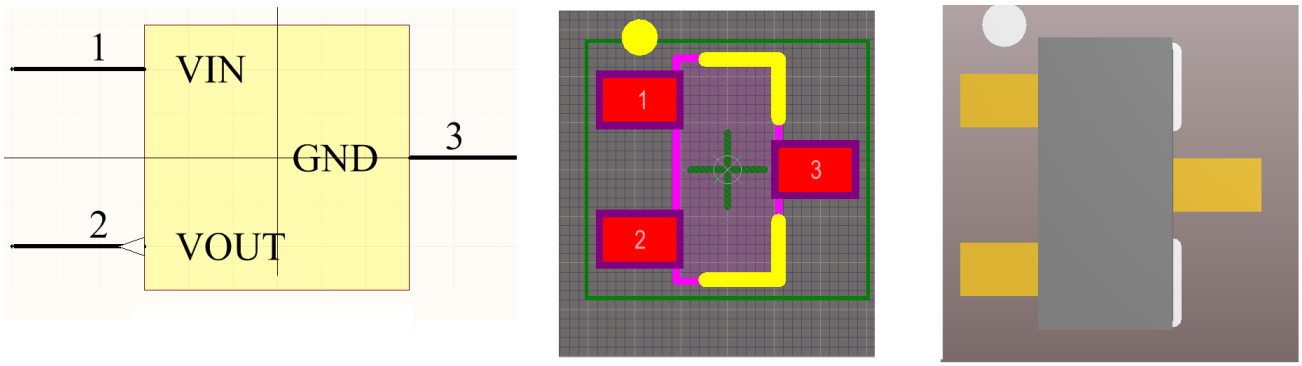


Figure 3: *Created symbol and footprint of ISL21080*

Schematic version of ADC64LPF is shown at figure 4. It consist of 4 input connectors and 4 output connectors (to obtain 64 for inputs and outputs entries), Voltage references - ISL21080 and low pass filter with amplifier - LT6600-20, multiplied 64 times.

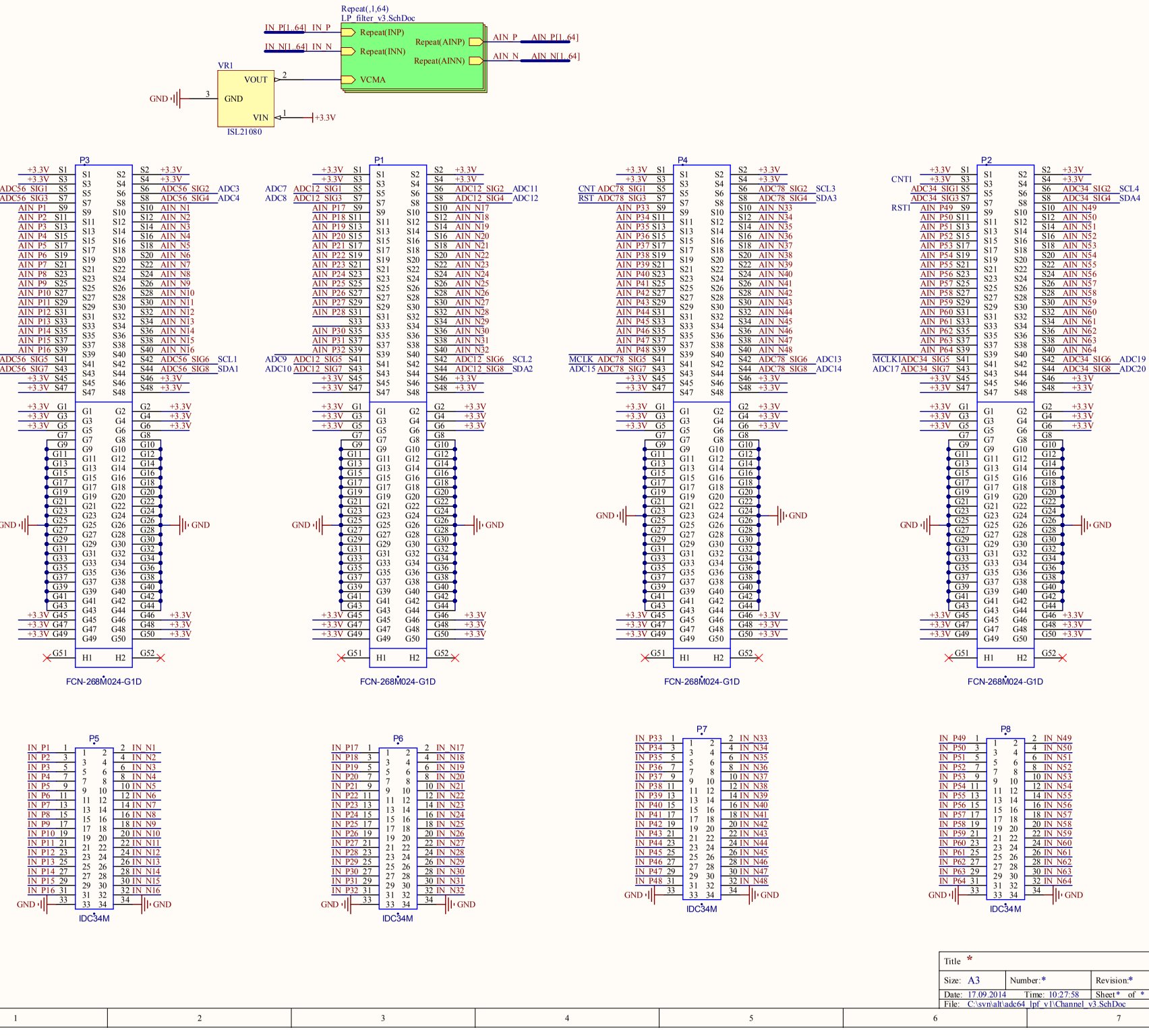


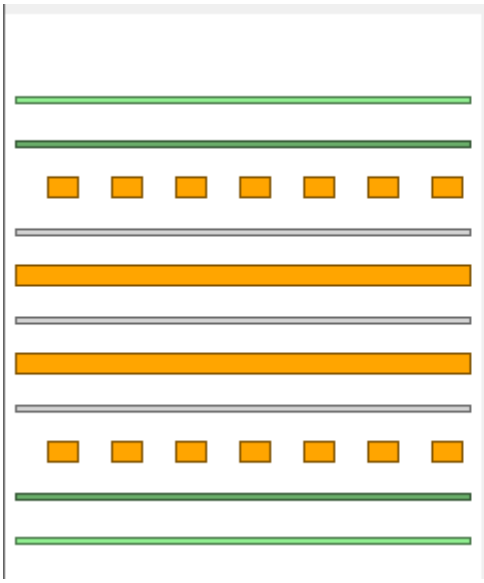
Figure 4: *Schematic of ADC64LPF project.*

3 Layout of ADC64lpf

Drawing layout is the most labor-intensive process. It requires a lot of attention, patience and accuracy, as each inexactness will result in deterioration in the quality of the signal.

3.1 Layers of PCB

To prepare PCB for designing, the layers must be defined at first. To design a low-pass filter's PCB, four signal layers are required (see figure 5).



Layer Name	Type	Material	Thickness (mm)
Top Overlay	Overlay		
Top Solder	Solder Mask/Co...	Surface Material	0.01
Top Layer	Signal	Copper	0.036
Dielectric 1	Dielectric	Prepreg	0.32
VCC	Internal Plane	Copper	0.036
Dielectric 3	Dielectric	Prepreg	0.32
GND	Internal Plane	Copper	0.036
Dielectric 2	Dielectric	Prepreg	0.32
Bottom Layer	Signal	Copper	0.036
Bottom Solder	Solder Mask/Co...	Surface Material	0.01
Bottom Overlay	Overlay		

Figure 5: *Table of ADC64LPF board's layers.*

Layers with thickness 0.036 *mm* are the signal layers: Top layer and Bottom Layer and Internal Planes: VCC and GND. They are mode of copper, while the splitting layers are made of prepreg and have a thickness 0.32 *mm*.

3.2 Layout

At first the placement of elements must be carefully arranged. It is very important that output connectors of ADC64lpf board fit the input connectors on the other PCB board with ADC. Outputs connectors need to by placed at first because them position is strictly defined. Next step was to find proper location for 64 rooms. One room contains the elements shown in the figure 6 They are respectively arranged and connected to form a so-called "room". Input connectors were placed in optimal configuration with other elements.

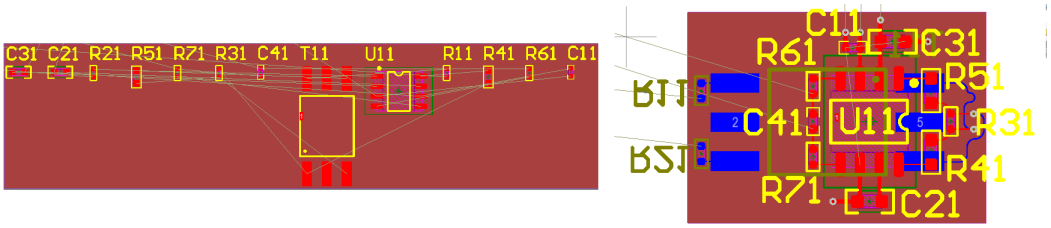


Figure 6: *Example of unrouted and routed "room"*

Next step was to route the connections between the elements. During this operation, at all times must be kept in mind that the path length of the differential pair, between a input and output connectors, need to be equal to an accuracy of 1 mm. Moreover a difference in length between the two paths one differential pair, need to be kept with accuracy 0.1 mm. To meet these expectations, each path must be carefully tuned. This has been done as snakes visible on some of the routs.

At figure 7 are visilble all layers. That is two signal layers, two internal planes, mask layers, mechanical layers, silkscreen layers and others. To make the view clearer, were also presented Top layer (see figure 8) and Bottom layer (see figure 9). At these two layer routes and them tuning are better visible. Average length of one differential pair routing (from input to output connector) is 140 mm with accuracy of 1 mm.

Only two of the many important rules that must be followed durring designing the PCB board had been just presented. There were also many others, like dostance beetewn various differential pair (0.84 mm), dostance beetewn routs of the same differential pair (0.15 mm) and others.

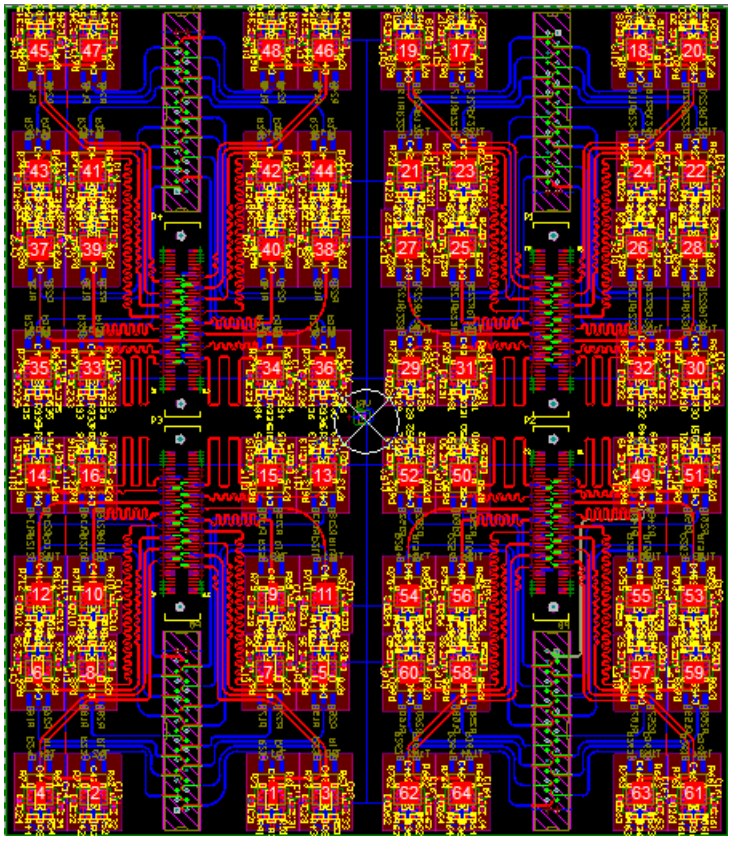


Figure 7: *All layer of ADC64LPF board*

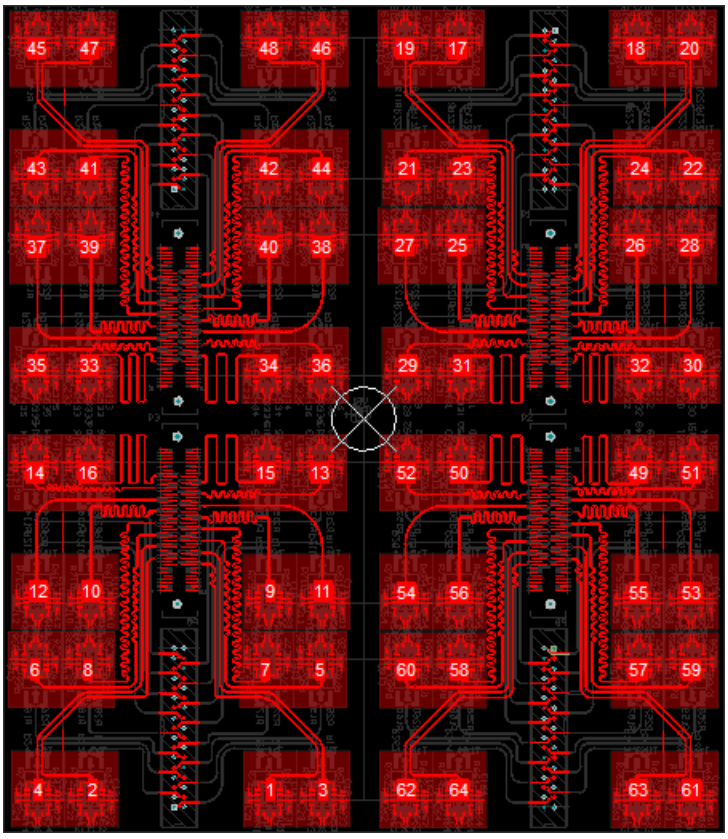


Figure 8: *Top layer of ADC64LPF board*

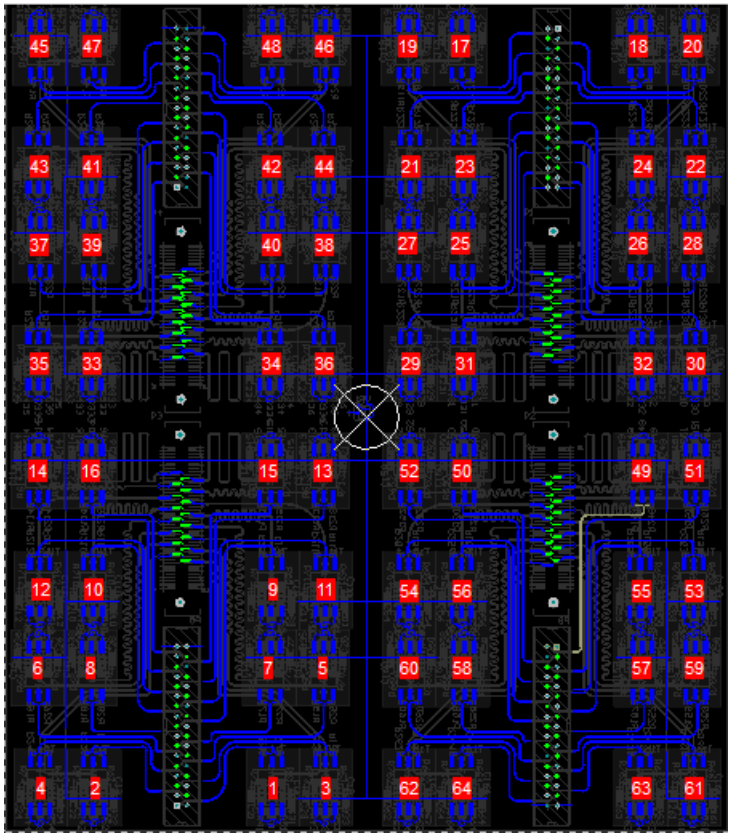


Figure 9: *Bottom layer of ADC64LPF board*

4 Summary

The main goal of the practices was to learn about PCB designing. During 5 weeks there was opportunity to work with one of the best programs to PCB designing - Altium Designer. The task was to design PCB board with low pass filter and amplifier, which will be used as a part of electronic to test prototypes detectors. Results of work have been presented in these paper.

References

[1] Documentation of LT6600-20: http://www.linear.com/product/LT6600-20

[2] Documentation of ISL21080 Voltage References:
http://www.intersil.com/content/dam/Intersil/documents/isl2/isl21080.pdf