FAMU/FSU College of Engineering

Department of Electrical and Computer Engineering

Operation Manual

Team 303 Software Defined Radio

Names: Kira Bronstein

Simon Charry

Christian Pollock

Jaryd Walton

Evan Woodard

Date: March 29, 2021

Table of Contents

1.0 Project Overview

This Operation Manual focuses on the process of Software Radio Transmitter through designing, building and testing of four PCBs. The first PCB, “Regulator Board”, focuses on converting the 12 voltages into many different desired voltages for each component such as FPGA, DAC, and etc. The second PCB, “Digital Board”, focuses on transmitting the data from the JTAG interface to FPGA and converting the digital into analog signal. The third PCB, “Op-amp Board”, focuses on amplifying the signal as well as eliminating any noise. The fourth and last PCB, “Modulator Board”, focuses on the RF modulator process of the signal.

2.0 Component/Module Description

This section provides details of each key component from each PCB.

2.1 Regulator Board

This subsection focuses on the description of switcher, split rail, and linear regulator.

2.1.1 Switcher Regulator: TPS561201DDCR

The switchers received an input of 12 voltage and converted through a step-down process into 3.3V and 1.2V for FPGA, Clock oscillator and DAC.

2.1.2 Split Rail Switcher: LTC3388EMSE

The switcher received an input of 12 voltage and spilt the voltage as well as stepping down the voltages to 5 and -5 volts

2.1.3 Linear Regulator: ADP-121AUJZ30R7

The switcher will receive an input voltage of 5 and -5 voltage and use a quiescent current dropout linear voltage to a 3 and -3 volts

2.2 Digital Board

This subsection focuses on the description of Clock Oscillator, Buffer, FPGA, and DAC.

2.2.1 Clock Oscillator: SIT8103AI-23-33E-100.0000X

The SIT8103 is a MEMS oscillator with frequency stability of ± 20 PPM. The pin 1,OE/ST, is connected to the 3.3 V regulator as well the FPGA. The output of the oscillator produces a 100 MHz to the buffer.

2.2.2 Buffer: SI5330F-B00214-GMR

The buffer sends the clock signal to the DAC and the FPGA while being powered by 3.3 V regulator. The SI5330 supports single-ended or differential input clock signals as well as generating four differential or eight single-end outputs.

2.2.3 FPGA: XC6SLX9-TQG144

The FPGA has the most connections as it is connected to DAC, buffer, JTAG interface connector, 3.3 V and 1.2 V voltage regulator. The 16 pins of FPGA are used for two 8-bit signals to the DAC. The FPGA has 9,152 Logic Cells, 144 pins, and 102 User I/Os.

2.2.4 DAC: MAX5852-ETL+

The DAC takes the input of 16 pins from the FPGA, three different voltage supplies, and a clock signal. The DAC is a 40 pins, Dual, 8-bit. 165Msps, Current-Output, and supports single-ended and differential modes of operation.

2.3 Op-amp Board

This subsection focuses on the description of the op-amp.

2.3.1 Op-amp: LT1818CS5-TRMPBF

The op-amp takes an input of I and Q signal and amplifies it as well eliminating any unnecessary noise. The LT1818 has 400 MHz Gain Bandwidth Product and 2500 V/us slew rate.

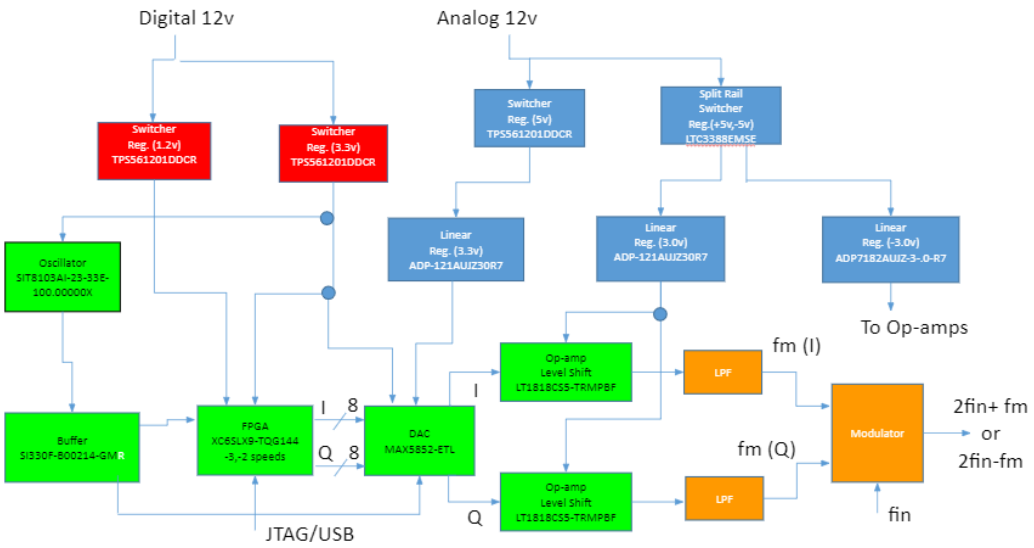
2.4 Modulator Board

The modulator takes in the amplified and filtered signal from the DAC and combines this signal (which is defined by the FPGA programming) with a carrier signal. This carrier signal effectively boosts the range of transmission and decreases power consumption by modulation practices. The modulator operates at 2.6 GHz frequency and combines with two RF sources operating at 40 MHz frequency to get the desired output for the modulated signal.

3.0 Integration

This section focuses on the details of connection with the four PCBs

3.1 Connection of each PCB



The figure above is the design overview on how all of the PCB boards are going to be connected with each other. We are going to use ST XH2.54-2P 2.54mm Pitch 2 Pin Female Connectors Cable Plug Header Expansion Wire 100mm that is going to be connected to the PCC02SAAN which is a header male 2 pin connector that is going to be soldered on the FPGAs, the regulators and the op-amp. For the RF signal we are going to use the EMPCB.SMAFSTJ.B.HT which is a coaxial connector that will be soldered on the op-amp, FPGA, DAX, and the modulator. The JTAG interface 6-pin connector, 3-644695-6, will be soldered on the Digital Board. The JTAG connector is for the FPGA communication to convert the signal into two 8-bits.

4.0 Operation

This section outlines the necessary steps to generate the Gerber and Drill files to submit to the manufacturer. Alongside detailing how to add a schematic symbol. After finding an affordable board house for fabrication the following details should be followed:

Layers:	4 layer
Material detail:	FR4-Standard Tg 140C
Board Type:	Single unit
Thickness:	0.8mm
Surface Finish:	HASL
Solder Mask Color:	Green
Copper weight finish:	35um
Surface Finish:	ENIG
Min trace/spacing :	0.2 mm
Solder Mask:	both sides
Silk Screen Legend:	1-Side
Silk Screen Color:	White
Smallest Hole:	0.4 mm

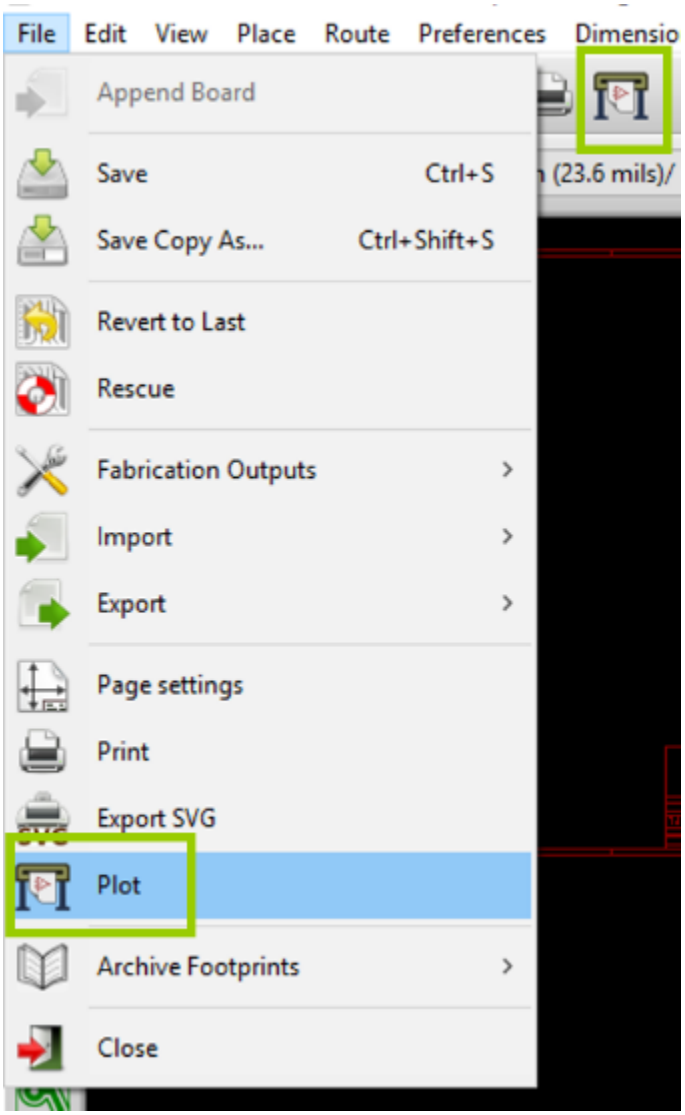
Fiducials: No Fiducials
Stencil Framework: None

Once the board is in hand the components should be soldered to the board. A few Test points are included on the board to ensure that once the board can be easily tested and to make sure that it is together all the necessary components are receiving the correct amount of current.

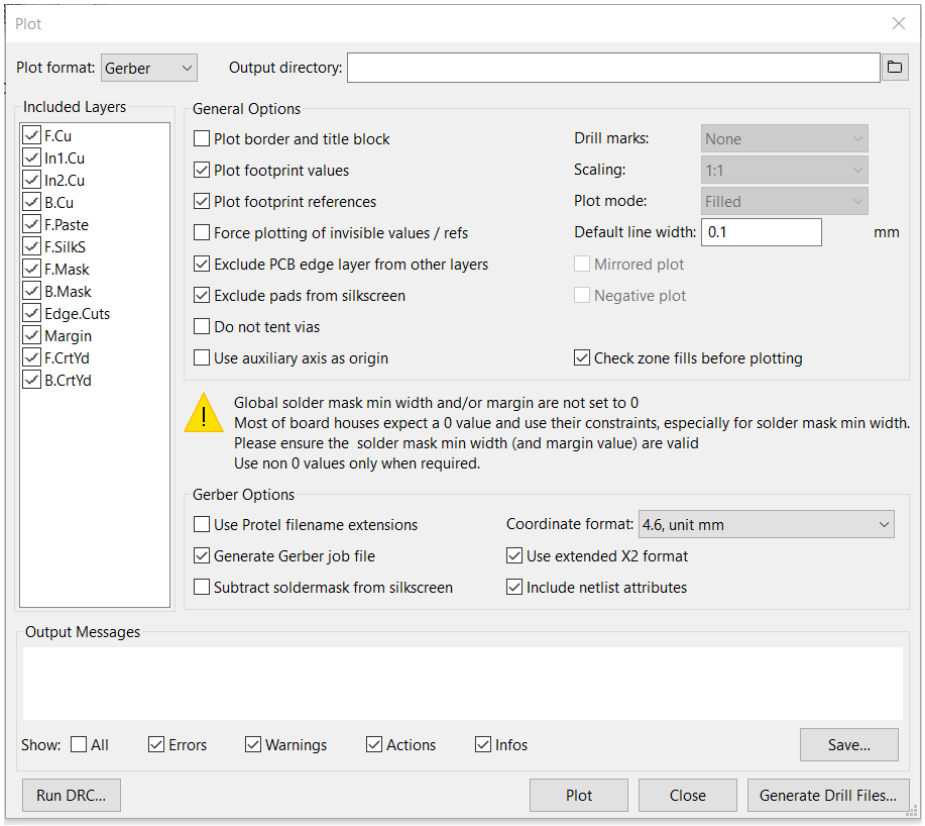
4.1 Generating Gerber and Drill files from KiCAD

Provided below is the reference used for the following steps:
https://support.jlcpcb.com/article/44-how-to-export-kicad-pcb-to-gerber-files

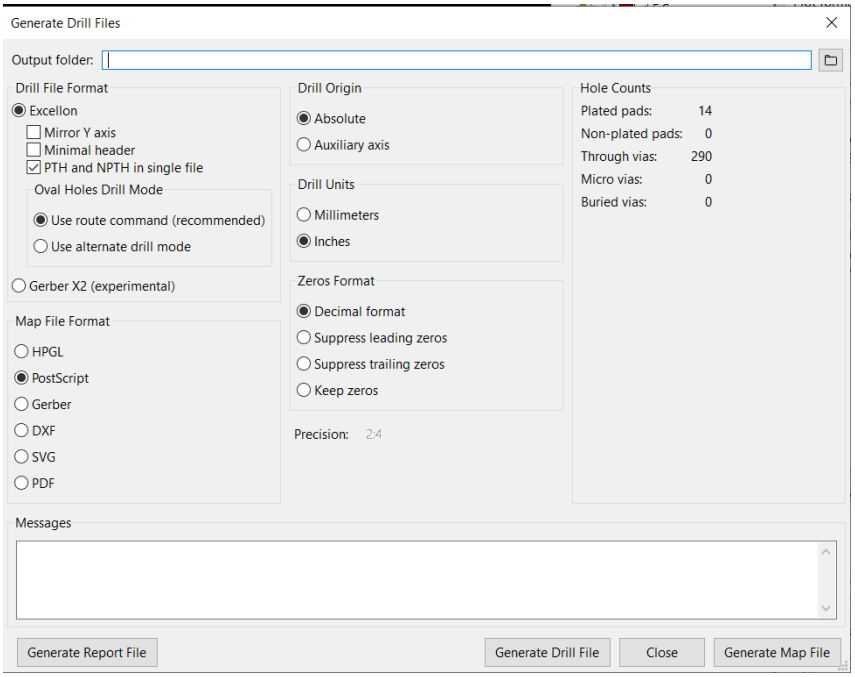
1. Open your PCB design file in KiCad and go to File -> Plot or the icon



2. In the following window, select a directory for gerber files, select your desired layers and leave the general option default.



3. Click on Generate Drill Files, in the following window, select “Merge PTH and NPTH holes into one file” box. Then click to Generate Drill Files.



4. Click the “Plot” button to generate the gerber files.

4.2 Adding the Schematic symbol library into KiCAD

Please note the following steps work for KiCAD version 5.1.5.

1. Open Eeschema
2. Click Preferences → Manage Symbol Libraries
3. In the following window ensure that the Global Libraries tab is selected. At the bottom right of the window select Add existing library to table.
4. Browse to where .lib file is located and select it.
5. Select Ok.

5.0 Troubleshooting

When importing the .lib files into KiCAD ensure that they are also located in the directory that the project file is located. The team has dealt with symbols being undefined because the .lib file was not located in the same directory as the project file. KiCAD does not work well on MAC this is why you want to have a windows computer, to save time and prevent unnecessary headaches it is strongly recommended that KiCAD. If you are unfamiliar with KiCAD, the link provided below is a playlist by Digi-key on learning KiCAD using a simple project. Tutorials range from creating custom schematic symbols to soldering components on the PCB.

https://www.youtube.com/playlist?list=PLEBQazB0HUyR24ckSZ5u05TZHV9khgA1O