

Q-meter

Background

In polarized target experiments, it is important to be able to measure the polarization of nuclear targets. The Q-meter is a method of doing this. The nuclear target is placed in the inductive part of an LC circuit, and the circuit is tuned to have the same resonant frequency as the nuclear target. The frequency of the RF voltage is modulated, from which one can detect changes in the circuit's Quality Factor (the "Q" in "Q-meter"). From this quantity, one can obtain a measurement of the polarization of the target. For more detail on the method, see the Liverpool Q-Meter Guide.

The particular design for this Q-meter comes from the University of Liverpool. There are five printed circuit boards (referred to as PCB 1, PCB 2, ..., etc.) which constitute the Q-meter circuit. The University of Virginia's PT Group has several of these Q-meters which were built some time ago. Some of them are fully functional, while others are not - for instance, some do not produce a curve, while some produce an inverted curve. The PT Group also has access to the Q-Meter Guide (previously mentioned), as well as the Q-meter's design documents, which detail the designs of the metal frames and the PCB's.

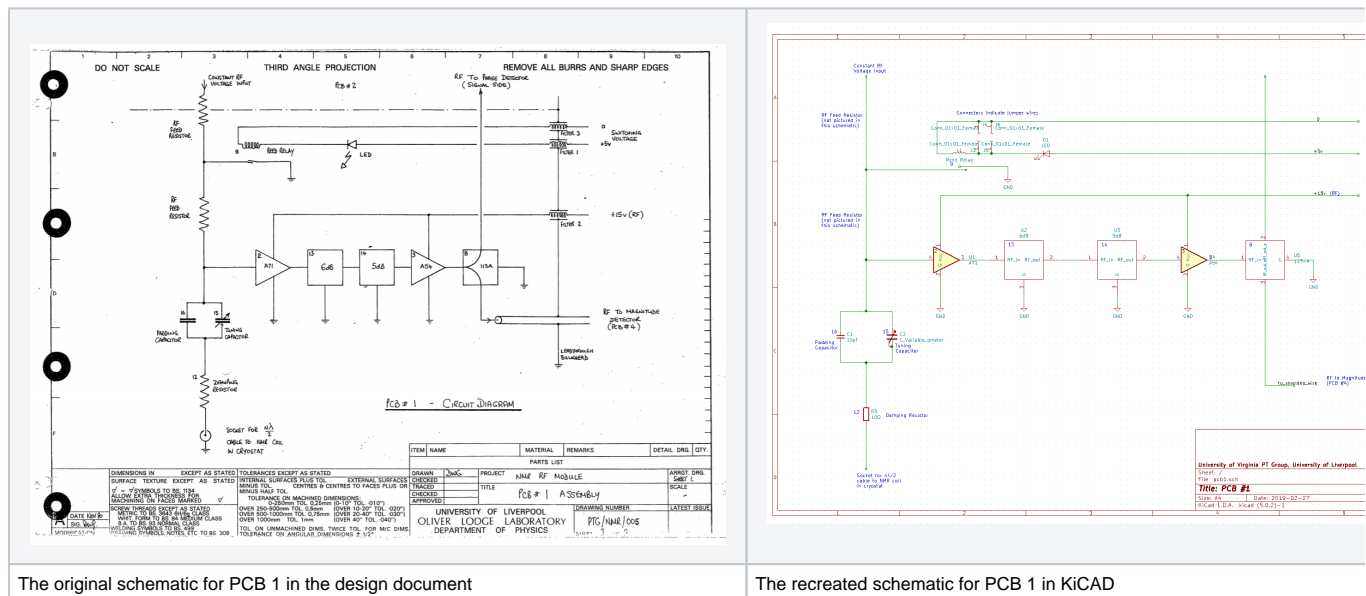
Because of the convenience of this device, and the access to the design documents, it could be a productive effort to try to fix the non-functional Q-meters, and to construct new Q-meters. The way to go about this is to make new PCBs, preferably with off-the-shelf components or modern replacements for obsolete components.

Method

The first step was to design the PCBs in a modern CAD software. Following the Q-meter's design documents, each of the five PCB's was reconstructed in KiCAD, a free and open-source CAD software for designing printed circuit boards.

Designing a circuit board in KiCAD follows several steps which will be briefly explained:

1. Schematics



The original schematic for PCB 1 in the design document

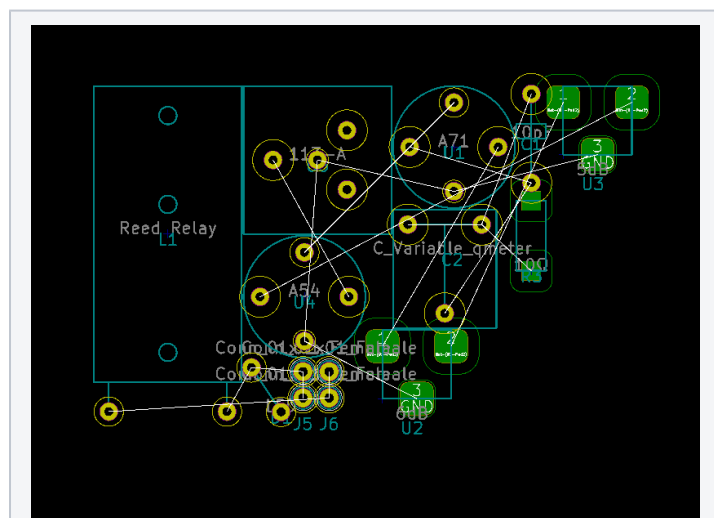
The recreated schematic for PCB 1 in KiCAD

First, one must construct the schematic for the circuit. Fortunately the design documents contained schematics for each PCB. Often, the user will need to construct a custom Symbol for one of the circuit's components, because the KiCAD library mostly contains common Symbols (resistors, capacitors, etc.) and likely will not have one tailored to the exact component. For instance, the 113A Symbol in the above schematic (rightmost square) had to be drawn from scratch, taking into account the number of pins and labels for each. KiCAD has a built in tool for drawing new components.

2. Laying out the PCB

The next step is to actually lay out how the circuit will look. The difference between this and designing the schematic is that, while the schematic tells KiCAD what components should be connected to which other components, the PCB editor actually determines how the board will be physically laid out.

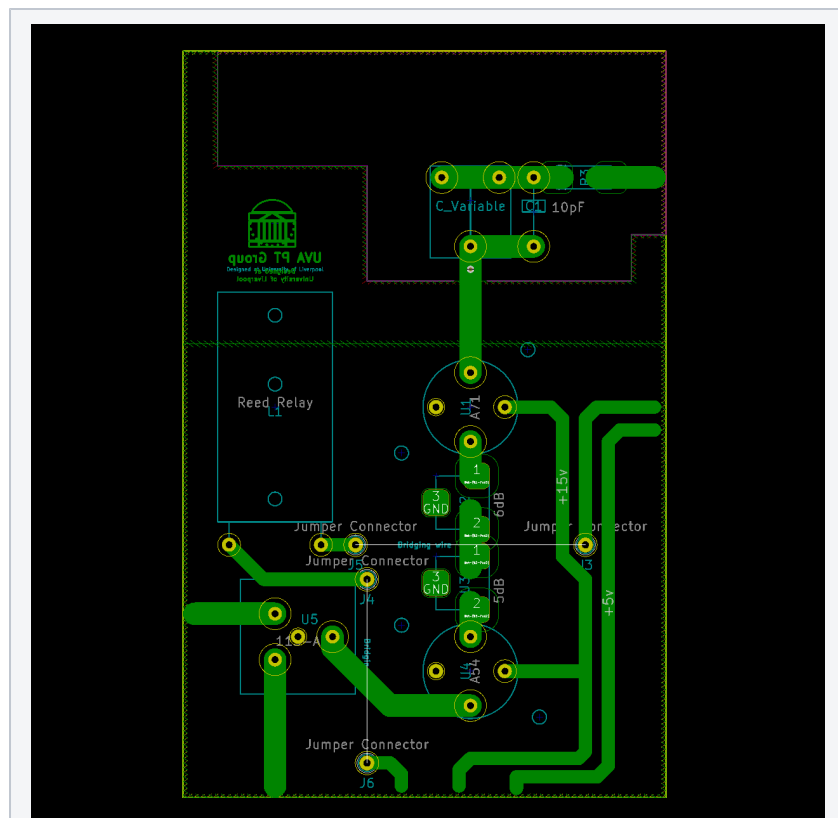
To illustrate, the following image is what the PCB looks like when one first loads their schematic into the PCB editor:



PCB 1. The footprints are initially disorganized in the PCB editor.

There are white lines which denote connections between components - this information is received from the schematic (specifically the Netlist file, which is generated from the Schematic). The user must move these components around, and draw traces to complete a connection between components.

The completed PCB design is shown below, with the ground planes made transparent so that the traces (the thick green lines) are more clearly visible.

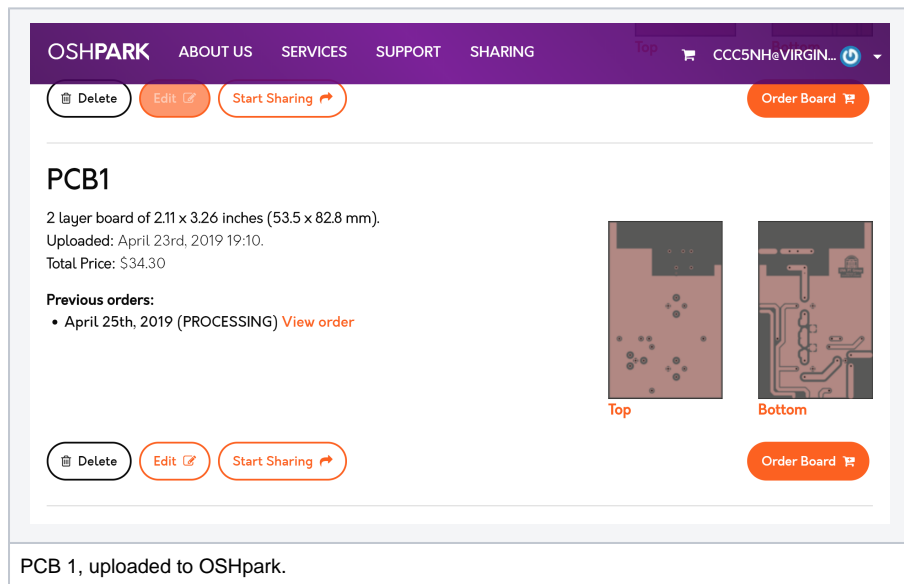


Finished PCB 1.

As with the schematic editor, there are representations of the circuit components. They are called "Footprints" here, and they are different from Symbols in that they must represent how the component will physically exist on the board. The Footprint will tell the KiCAD software (and, therefore, the manufacturer of the PCB) where the holes for the pins must be drilled and where to place copper for its connections. KiCAD has an editor to create custom Footprints.

The layout of each PCB was designed to match the original PCBs as closely as possible. Some things to keep in mind were the boards' dimensions, the positions of traces which connect to other PCB's (such as the two traces in the middle of the right edge, of the PCB pictured above), the thicknesses of the traces, the clearance of the traces (space between the trace and the ground plane), and the area of the ground plane.

3. Manufacturing

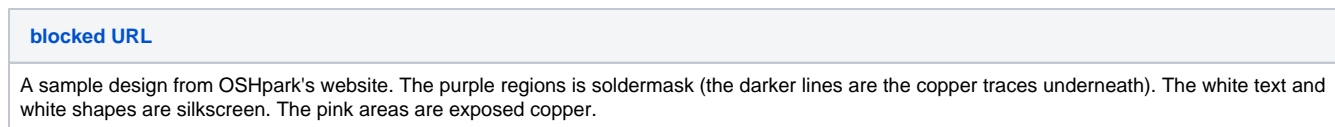


Once the PCB layout is complete, one must create "Gerber files" which are the industry standard format for designs of PCBs to be manufactured. KiCAD has the ability to create these from the PCB Layout editor, as well as to create "Drill files" which tell the manufacturer where holes should be drilled on the PCB.

The service used was called OSHpark. They have the advantage of decent prices, the ability to order a small number of PCBs at a time, and a simple and straightforward web interface. One simply must upload the Gerber files, and then place the order.

There are a few important things to note:

- PCB designs consist of "layers." For example, on a two-sided PCB, there will be a "Front Copper" layer and "Back Copper" layer, which have the traces meant for the front and back of the PCB, respectively. There is also the "Edge Cuts" layer which tells the manufacturer the perimeter of the board to cut around. There are "Silkscreen" layers which have useful symbols/text to be printed onto the board. There are also "soldermask" layers, explained below.
- A PCB typically has something called a "Soldermask" which is a layer of polymer that shields the copper traces underneath from oxidation. The Liverpool Q-meter design does not call for any soldermask, opting instead for gold plating. To match the original design, we ordered our PCBs with no soldermask layer. In CAD software, soldermask layers are typically "negative," meaning that rather than placing soldermask wherever the design is filled in, it actually places soldermask wherever the design is not filled in and keeps it away from filled zones. To order a PCB from OSHpark with no soldermask, one must go into KiCAD and place a filled zone over the entire board on both front and back soldermask layers.
- OSHpark offers gold-plated nickel over copper as a further way to protect the copper from oxidation. Even without soldermask, it is our expectation that this layer will protect the copper on our PCBs from oxidation.



List of Parts

Item	Description	Quantity per board	Modern replacement (if applicable)
RF Amplifier	A71	1	
RF Amplifier	A54	1	
RF Amplifier	GPD-402	1	
RF Amplifier	GPD-403	3	
RF Splitter	113A	1	
Double Balanced Mixer	SRA-1	1	
Detector Diode	BD3, back diode	2	

Chip Resistor	CR15, type B, 5%, 27	2	
Chip Resistor	CR15, type B, 5%, 50	1	
Chip Resistor	CR15, type B, 5%, 10	1	
Chip attenuator	AC0103, 3dB, 50	3	
Chip attenuator	AC0105, 5dB, 50	2	
Chip attenuator	AC0106, 6dB, 50	2	
Tuning capacitor	AP14E1, with insulated shaft	1	
Capacitor	10 pF S. Mica	1	
Capacitor	220 pF ceramic	4	
Capacitor	1000 pF ceramic	8	
Capacitor	0.1 μ F ceramic	2	
Resistor	150 thick film	1	
Resistor	220 thick film	1	
Resistor	330 thick film	2	
Resistor	2.2k thick film	12	
Resistor	4.7k thick film	6	
Trimpot	10k, 3262W-1-103	2	
Dual Op Amp	OP-10 EY / LT1002	2	
Op Amp	LF356N	2	
RF Inductor	10 μ H	2	
Lead Through Filter	1251-001, EMI suppression	14	
SMA Jack	R.125-403, 4 hole flange socket	3	
SMA Plug	R.125-052, straight solder plug	4	
Multiway connector	DB-25P	1	
Multiway connector	DB-25S	1	
SMA Jack	2004-7985, straight b'head jack	1	
Cable	BA50085, semi-rigid	150 cms	
LED	10 mA @ 2V, to fit 4mm hole	1	
Capacitor Drive	Type H	1	
Capacitor Drive knob	412, multi-turn, lockable	1	
Capacitor Drive sleeve	Rubber coupling	2 cms	
Solder tag	H3	1	
Screw	M2.5 * 6mm, cheese head	65	
Screw	M2.5 * 16mm, c/sunk head	7	
Screw	M3 * 6mm, cheese head	3	
Interconnecting wire	7/0.2mm, insulated copper		
Interconnecting wire	1/0.6mm, bare copper	15 cms	
Sleeving	As required, for above		
Solder			
Reed switch	Mini-25, 7-12 AT		
SMA Phase adjuster	901-508, In-line adaptor	1	
Module frame assembly	PTG/NMR/002		
Capacitor drive assembly	PTG/NMR/003		
Reed relay assembly	PTG/NMR/004		
Lead through b'head assy.	PTG/NMR/011		
Star washer	9.5mm 1/D, 17mm 0/D (max)		
Washer	M2.5, plain, brass		
Screw	M4 * 6mm, c/sunk head		

Files

Liverpool Q-meter Guide: [LiverpoolQMeterGuide.pdf](#)

Q-meter design document: [qmeter_blueprint.pdf](#)

KiCAD files (Most recent update: 4/23/2019): [qmeter Boards.zip](#) (Note: pcb3_fabrication and pcb4_fabrication contain the PCB layouts which were ordered. pcb3 and pcb4 contain schematics which more closely resemble the design document, but which are less convenient to construct in KiCAD because of how the dual op-amps are separated).

Just the Gerber files (Most recent update: 4/23/2019): [Gerber_Files_for_PCBs.zip](#)