MIC&MOD

DIY Vintage Microphones

Mic & Mod U67 microphone.

The U67 microphone is no doubt one of the most iconic microphones ever produced.
Even although the U67 was designed in 1960, as a multipattern succesor of the U47, it is still seen in many studios around the world.
The U67 has a couple of unique features:

- **HF de-emphasis through feedback instead of passive filtering**
- **Low pass filtering through feedback**
- **In cardioid mode, the rear diaphragm of the capsule is completely decoupled, with as a result a higher output and a better signal to noise ratio.**
- **Use of a penthode tube used as a triode for less noise**
- **A very big output transformer, with low distortion**

This is the list of materials you should have to build a U67 microphone:

Parts list (The last band shows the resistor tolerance, in most cases brown = 1%)

Resistors		
1	680 ohm	Blue/Gray/Black/Brown
1	1.5 K.ohm	Brown/Green/Black/Red
2	10 K.ohm	Brown/Black/Black/Red
1	100 K.ohm	Brown/Black/Black/Orange
2	1 M.ohm	Brown/Black/Black/Yellow
2	100 M.ohm	Brown/Black/Purple
2	470 M.ohm	Yellow/Purple/Purple
2	1 G.ohm	Brown/Black/Gray

Capacitors		
1	270 pF Styroflex	Silver, axial
1	1000 pF Styroflex	Silver, axial
1	100 pF / Wima	Red, rectangular*
1	120 pF / Wima	Green, rectangularial*
1	10 nF /Wima	Red, 0.01µF*

1	**100 nF/Wima RM7.5**	**Red, 0.1 µF***
1	**10 nF /Vishay, axial**	**Yellow***
2	**1 µF / 450 V, radial**	**Green, 1 µF***
1	**47µF/ 16V or higher**	**Black or blue***

Miscellaneous		
1	**EF86 tube, selected**	**Electron tube in box, tested for moise**
1	**Noval tube socket, PCB**	**White tube socket with 9 pins**
1	**Tube PCB**	**Small printed circuit board for tube socket**
1	**Switch PCB**	**Small printed circuit board for the switches**
1	**Main PCB**	**PCB for the amplifier circuit**
2	**Solder pin, 1 mm.**	**Test pins**
3	**Screw, M2x10 or M2x12**	**To mount the switch PCB**
3	**Spacer 6 mm.**	**To mount the switch PCB**
1	**Screw M2 x 20**	**To mount the transformer**
1	**Nut M2**	**To mount the transformer**
2	**Screw M1.6x6**	**To mount the capsule in the holder**
4	**Screw M2x4**	**To mount the Main PCB to the frame**
1	**Microphone capsule**	**The part that will be capturing the sound!**
1	**Capsule mount**	**To mount the capsule in the headbasket**

Depending on the manufacturer, **colors may be different**.

First a word of warning:

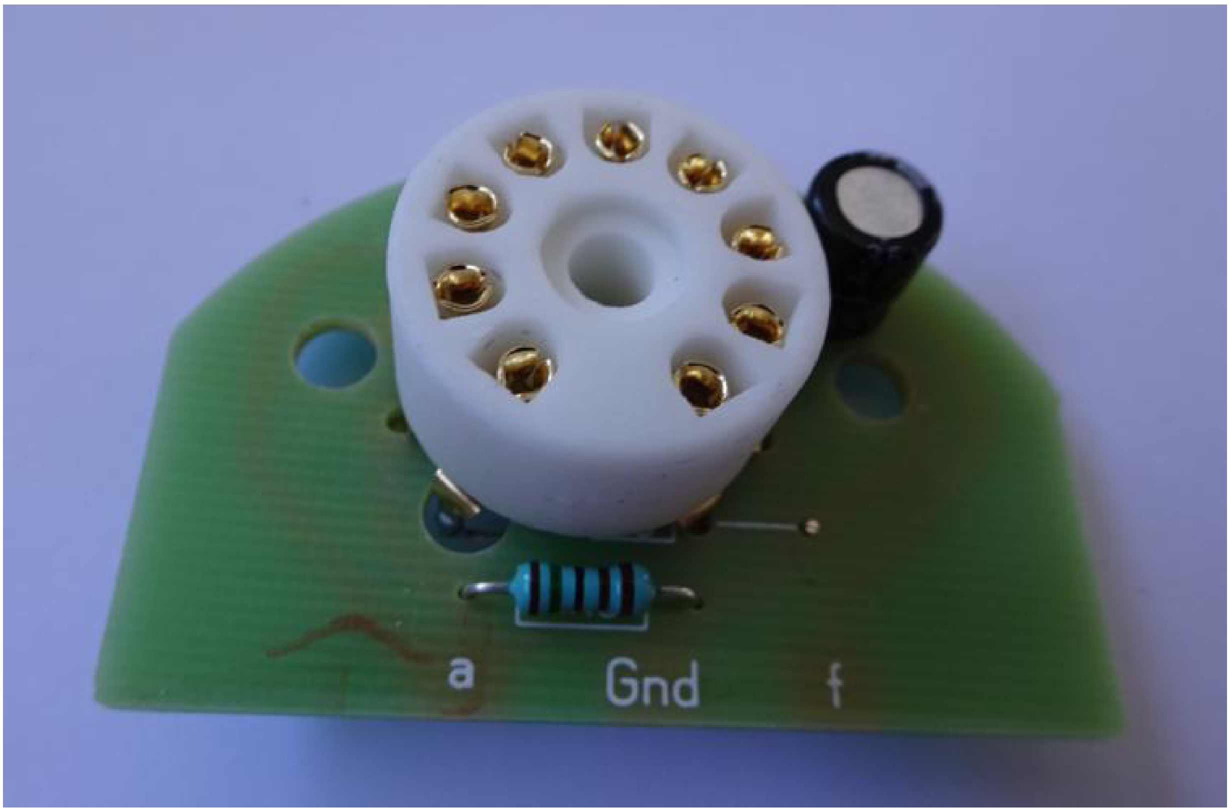
Do not touch anything inside the microphone when the power supply is connected. The microphone works with high voltages, that in some situations can be lethal. When you work on the microphone, ALWAYS disconnect the power supply. Even when the power supply is switched off, the capacitors inside the power supply will keep a high voltage for a long time.

In order to recreate a U67 microphone, we had to adapt some minor things, that do by no means affect the sound quality!
The original U67 used a very complicated transformer, with a 'tertiary' winding for the feedback loop. Such a transformer is ***very*** hard to find and even if you find them, they are very expensive. Because of this reason, we decided to use a more common, but very high quality transformer, with two windings.
Since a transformer is nothing else than a 'translator' for impedances, we made a clever design that takes the feedback from the primary winding. The behaviour of the transfomer is also reflected in this winding.
The U67 had a steep fixed low-cut filter, that could be bypassed by removing a jumper wire inside the microphone. Because there is nowadays no specific reason to have a fixed low-cut filter inside the microphone (every mixer or audio interface has a low-cut filter!), we 'pre opened' the S2 jumper inside the microphone, so to speak. After all, we want to hear and record the **full** frequency spectrum!
Of course we kept the switchable 'high-pass' filter on the microphone, to reduce too prominent low frequency content if a soundsource is very close to the microphone. (Proximity effect.)
The rest of the circuit is completely identical to the original U67 microphone.
We use a K67 microphone capsule and the same EF86 tube as in the original U67, a penthode tube used as a triode, because triodes have the advantage of producing less noise.
In fact, we even added an optional feature, that will be described later on.
At the time the U67 was designed, it was very problematic to find reliable high value resistors.
Nowadays high value resistors (1 Giga –ohm or so) are much more common.

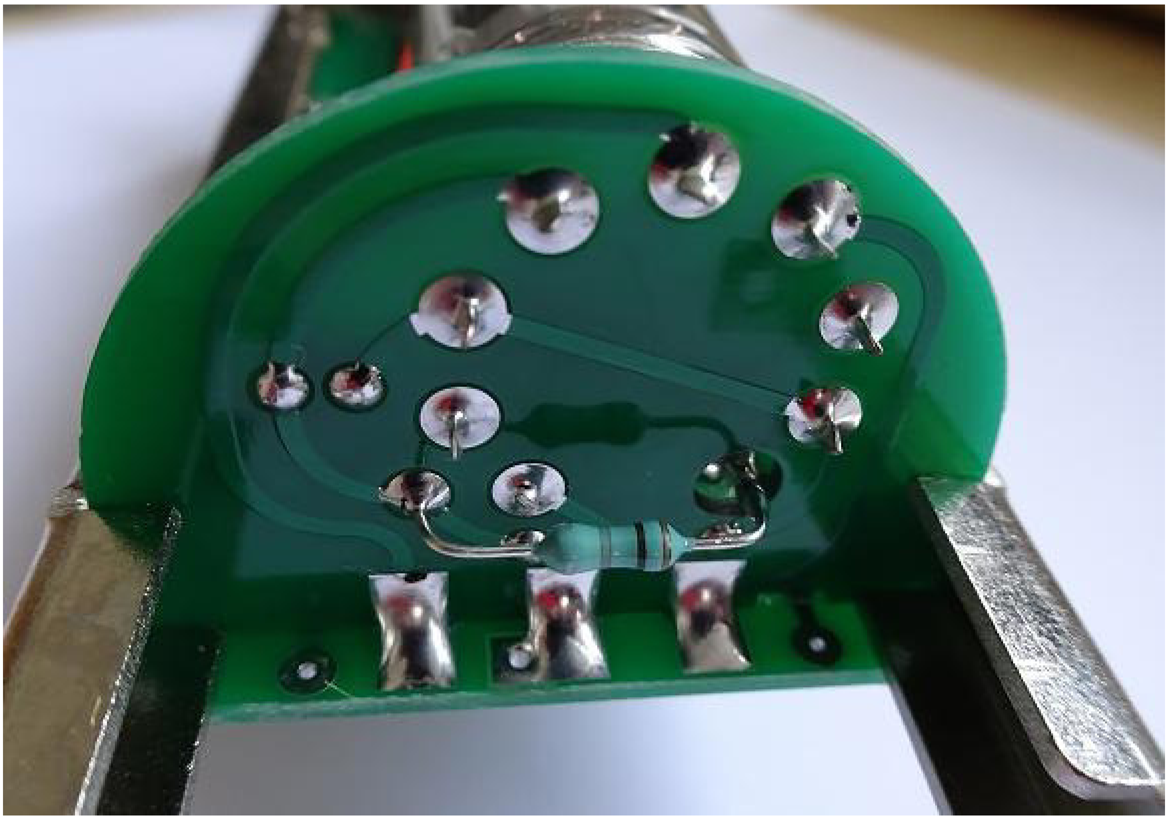
Because the use of a high(er) value grid leak resistor for the tube has certain advantages (lower self noise of the microphone, improved low-frequency response) we took the freedom to raise the value of these resistors, because it improved the quality of the finished microphone.

First a word of warning: the U67 is not the easiest microphone to build!
Not that it is very complicated, but you will need certain skills to succesfully assemble the microphone.
Also, **the microphone uses high voltages that can be lethal**.
For this reason, always **be very careful** when working on the microphone when the power supply is connected. Even if the power supply is switched off, the capacitors inside the power supply will keep their high voltage for a long time.
This is especially important if you want to take measurements when the microphone is active.
It is always a good idea to remove the cable connected to the power supply, when working on the microphone.

Now, let's start to build the "U67"!
The microphone consists of three small printed circuit boards that we will assemble one by one.
We start with the **tube PCB**. This is the easiest PCB, to 'get in the mood'.
First we mount the 1.5 K.ohm resistor (/green/black/brown/brown) and the 47 uF capacitor.
Take care of the right **polarity** of this capacitor! The positive (+) side should be close to the tube socket, the negative (-) side should be near the edge of the PCB.
On most electrolytic capacitors, the negative ('-')side is indicated with a white or black **stiripe** on one side. This means the the wire on the opposite site is the positive ('+') connection.



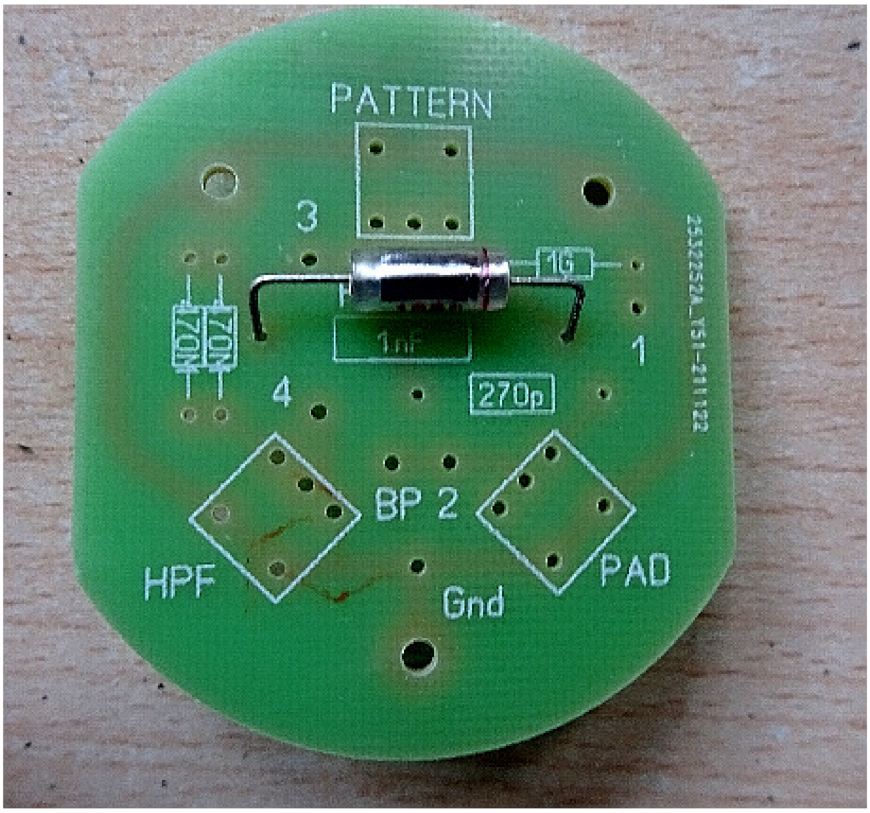
When these two components are in place, we place and solder the 9-pin tube socket.
Take care that the socket is flat on the printed circuit board and that one pin protrudes through the big hole, without touching the printed circuit board material.
When the tube socket is soldered in place we take a one GigaOhm (1 G..ohm, brown/black/gray/silver) resistor ald solder it between the pin that peeks through the big hole, and the PCB. Cut the end of the resitor flush on the component side of the PCB.
It should look like this. (In a later stage the tube PCB is soldered on the main PCB):



After we have mounted the high value resistor, this PCB is finished for the moment.
We can put it aside for later use.

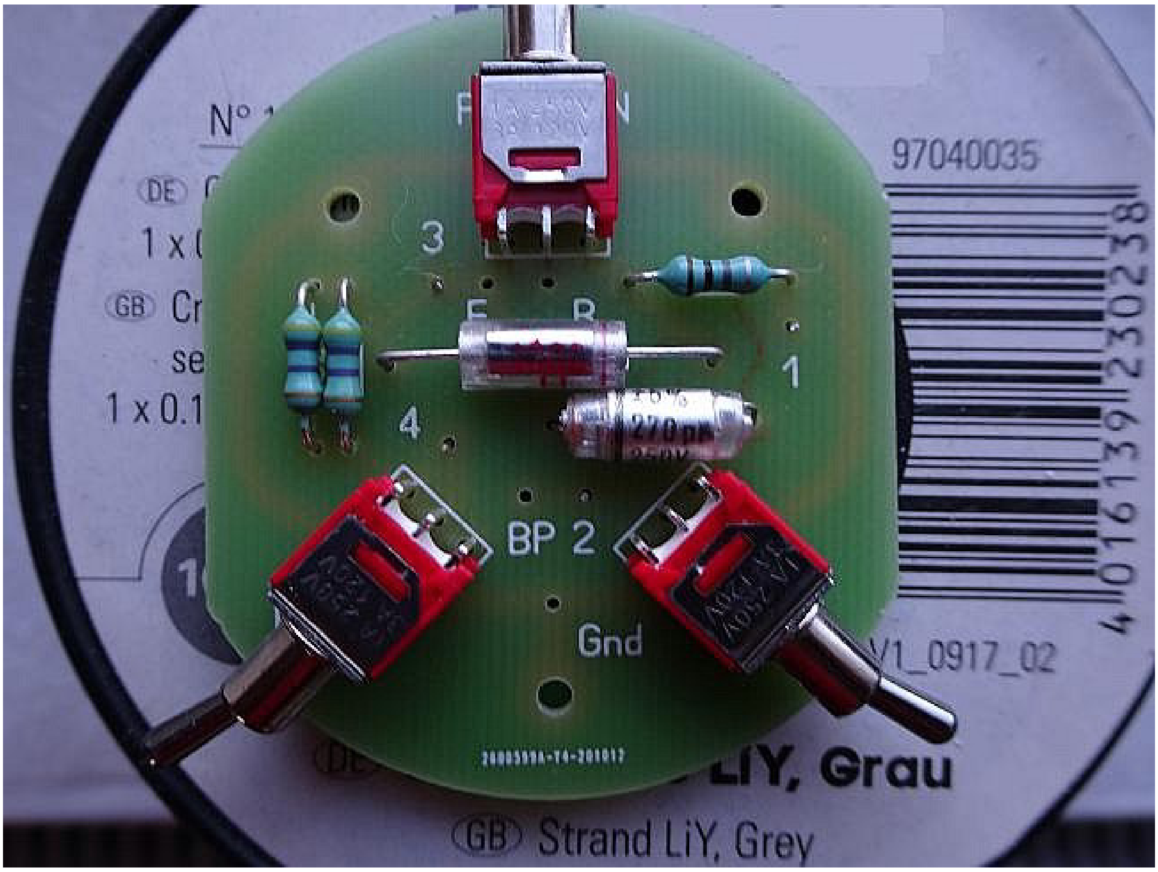
Now we assemble the **switch PCB**. Not very much parts here.
We start with the two 470 M.ohm resistors (yellow/purple/purple, gold or silver) and the 1 G.ohm resistor (brown/black/gray/silver).
It is recommended to mount these resistors a little 'floating' above the PCB.
A little distance (1 millimeter) is more than enough, it is important that these resitors don't touch the PCB.
After soldering the resistors in place, we place the two styroflex capacitors.
One capacitor is 270 pF, the other capacitor is 1000 pF. The value is printed on the capacitors.
Styroflex capacitors are sensitive to heat, so don't overheat them and try to solder them as quickly as possible.

If your kit contains a capsule mount with only **<u>one screw</u>** on the bottom, there will be a chance that the head of the screw will touch the 1000 pF capacitor. In this case it is a good solution to move the 1000 pF capacitor a bit to the side, like this:



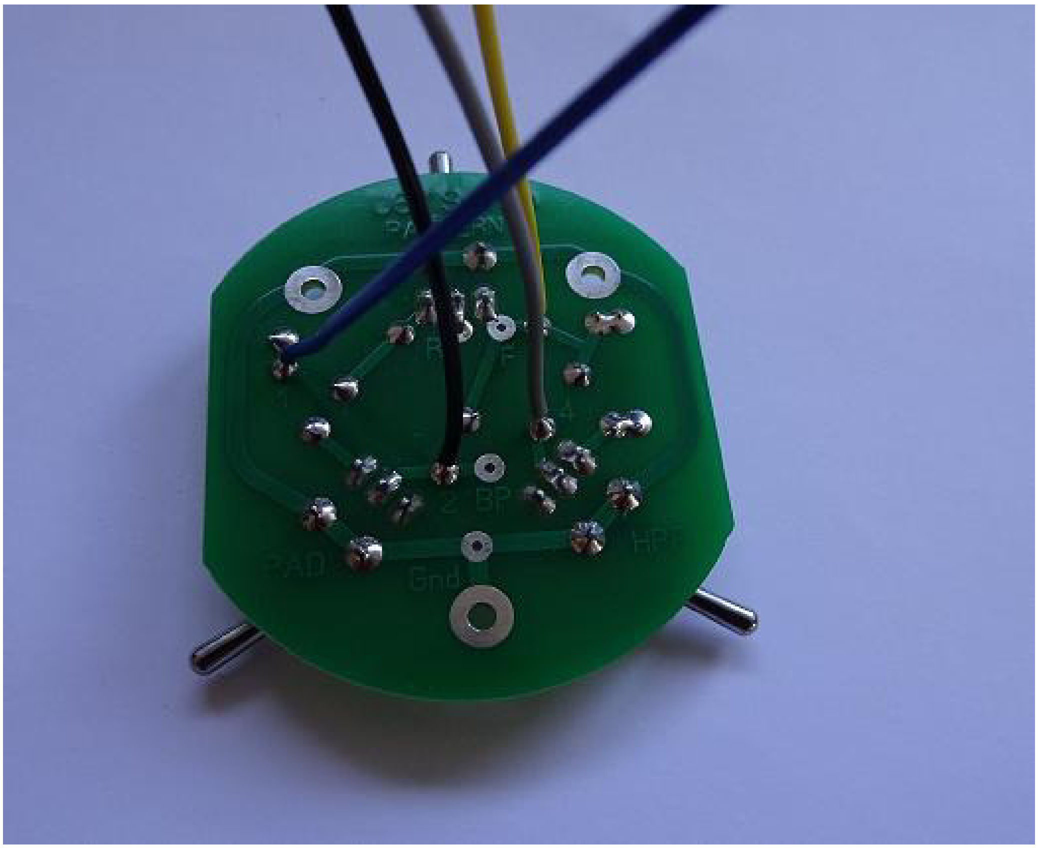
This will prevent the head of the screw touching and eventually damaging the 1000 pF capacitor

Last step is to place the three switches. Take care that you place the **3-position** switch (the other two switches only have **two positions**) on the correct spot, that is the position marked: ‘pattern’.
The other 2-position switches are placed on the spots marked ‘HPF’ and ‘PAD’.
Switches and components are placed on the side without conductive traces. (The side where the white screenprint is.)
Solder the switches as flat on the PCB as possible. If you dont do this, you will later have problems, because if the switches are too far from the PCB, the handles of the switches will not be in the correct position in the microphone body.
When the complete PCB is finished, it should look like this:

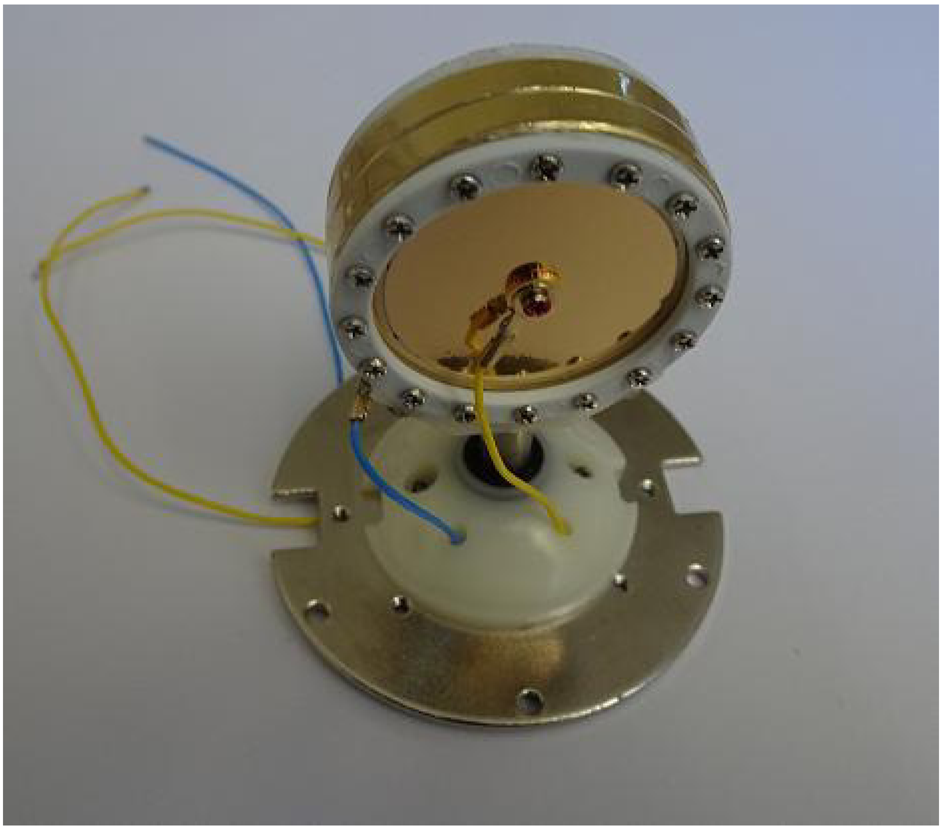


It is a good idea to solder some thin wires to the points 1, 2 3 and 4.
The wires are soldered on the **back** of the PCB.
These wires are used later to connect the switch PCB to the main PCB.
In this case, different colors are used to make identification easier.

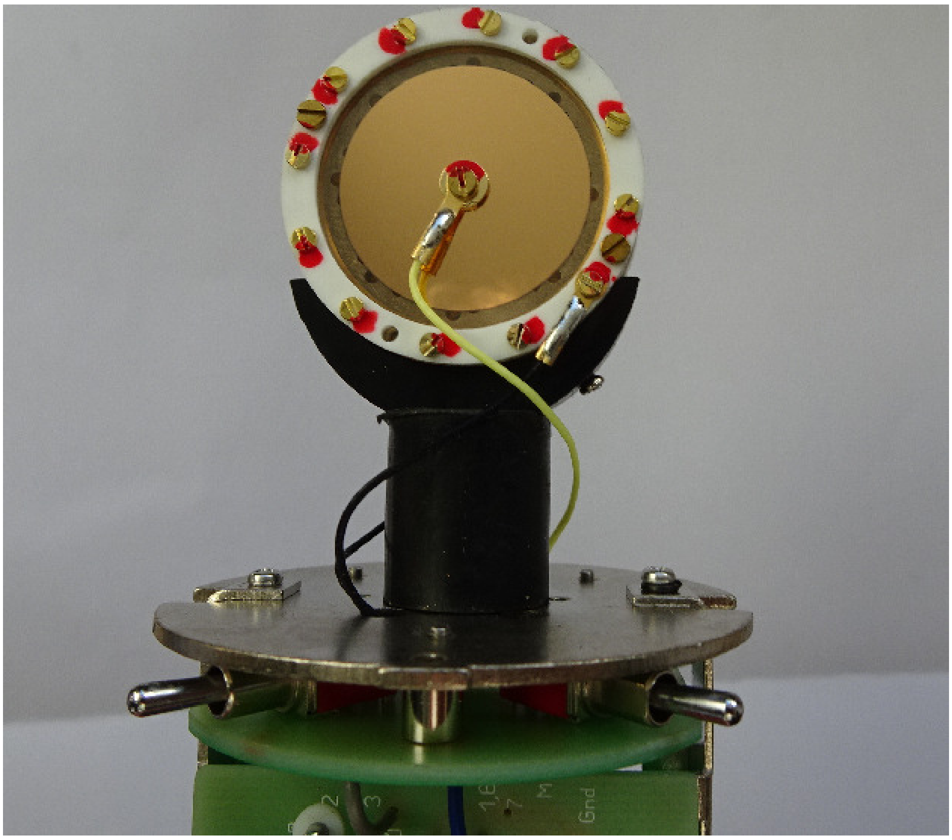
(Colors in the picture are random.)



At this stage, we can mount the microphone capsule inside the headbasket of the microphone body.

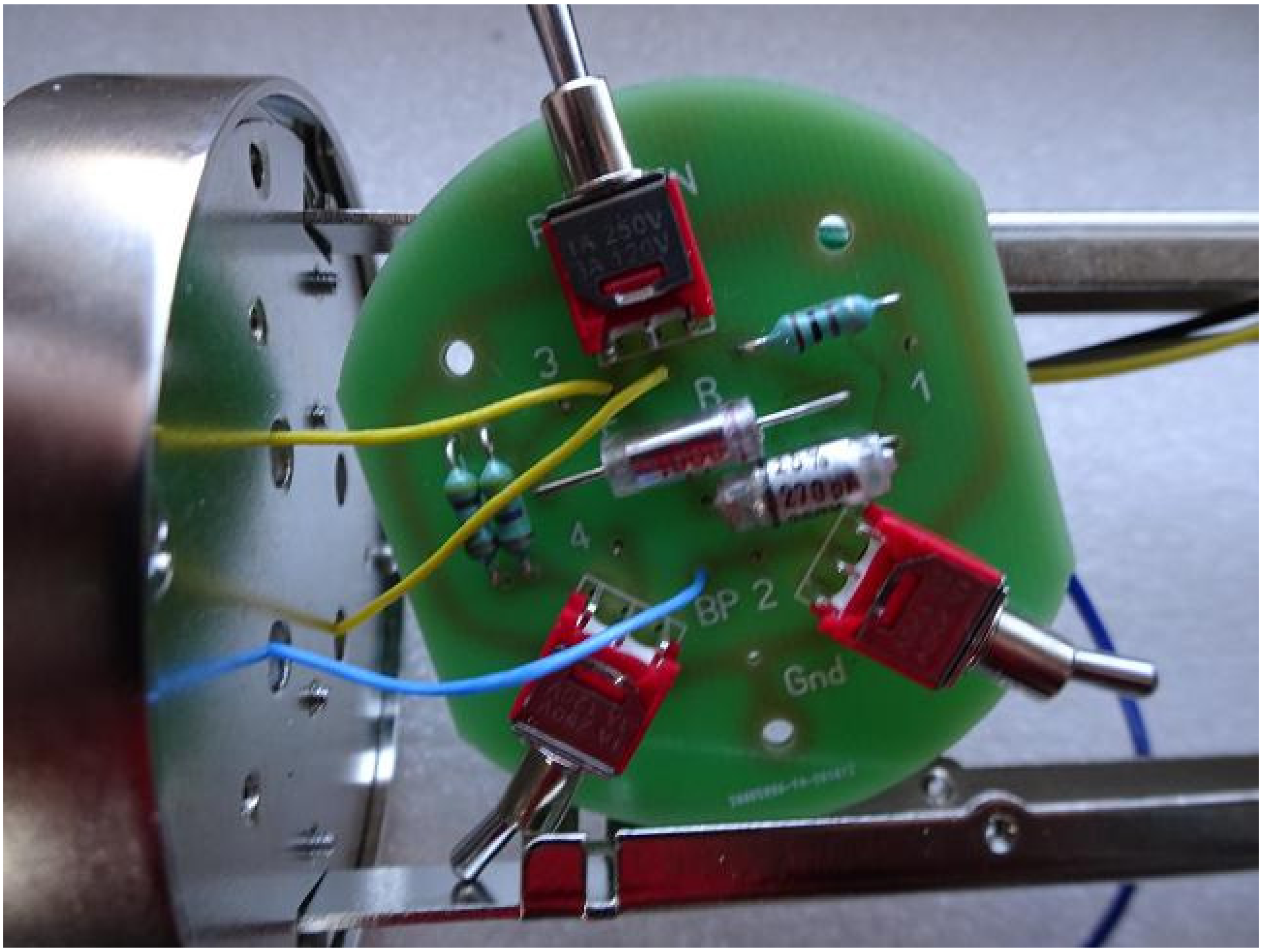


The newer capsule mounts are fastened with only one screw in the center of the metal plate.



Note that you use one screw in the ‘thin’ part of the saddle, and one screw in the ‘thick’ part.

There is no ‘front’or ‘rear’ side of the microphone capsule, technically both sides are identical.
Feed the capsule wires through the holes in the bottom plate.
Do this in such a way, that you can later identify which wire is coming from the front and which one is coming from the rear diaphragm. The color of the backplate is blue or black, so this one is easy to identify.
Mount the headbasket in place to prevent damage to the capsule!
Now we can carefully shorten and tin the capsule wires and solder them to our finished switch PCB. Front diapfragm connects to ‘F’, rear diaphragm connects to ‘R’ and the backplate wire connects to ‘BP’.

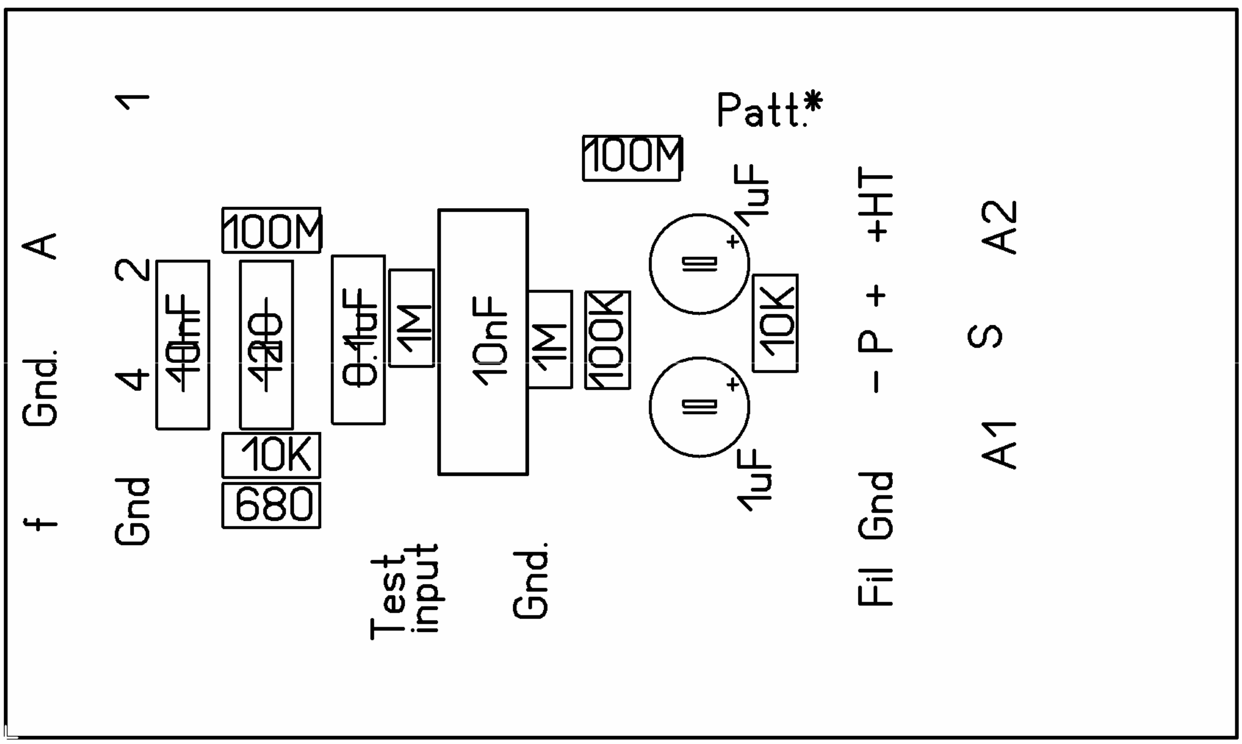


The switch PCB will later be mounted to the metal plate where the microphone capsule is mounted on. The PCB is mounted with three (10 or 12 mm) long screws and the 6 mm metal spacers.
The switches will be **between** the metal plate and the PCB. (Switches will be facing up.)
It is better to wait until the tube PCB and the main PCB are soldered together in a later stage.

Next step is the **main PCB.**
In principle, all component positions are clearly marked on the PCB.
Start with the resistors, followed by the capacitors.
This is more like ‘painting by numbers’.

Resistor values:

680 ohm = blue/gray/black/black/brown
10K = brown/black/black/red/brown
100K = brown/black/black/orange/brown
1M = brown/black/black/yellow/brown

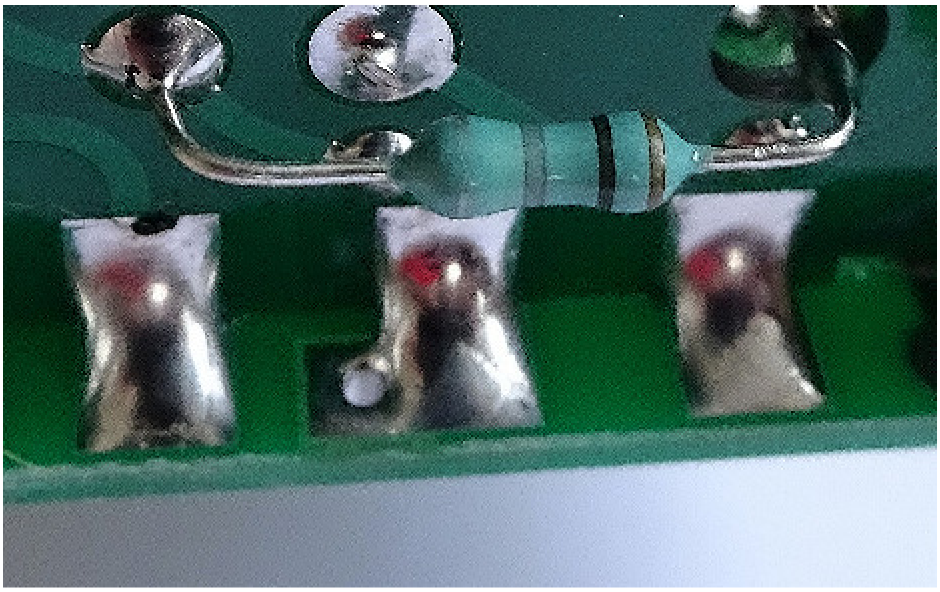


Place the two 1 mm solder pins for the connections to the ‘test input’.

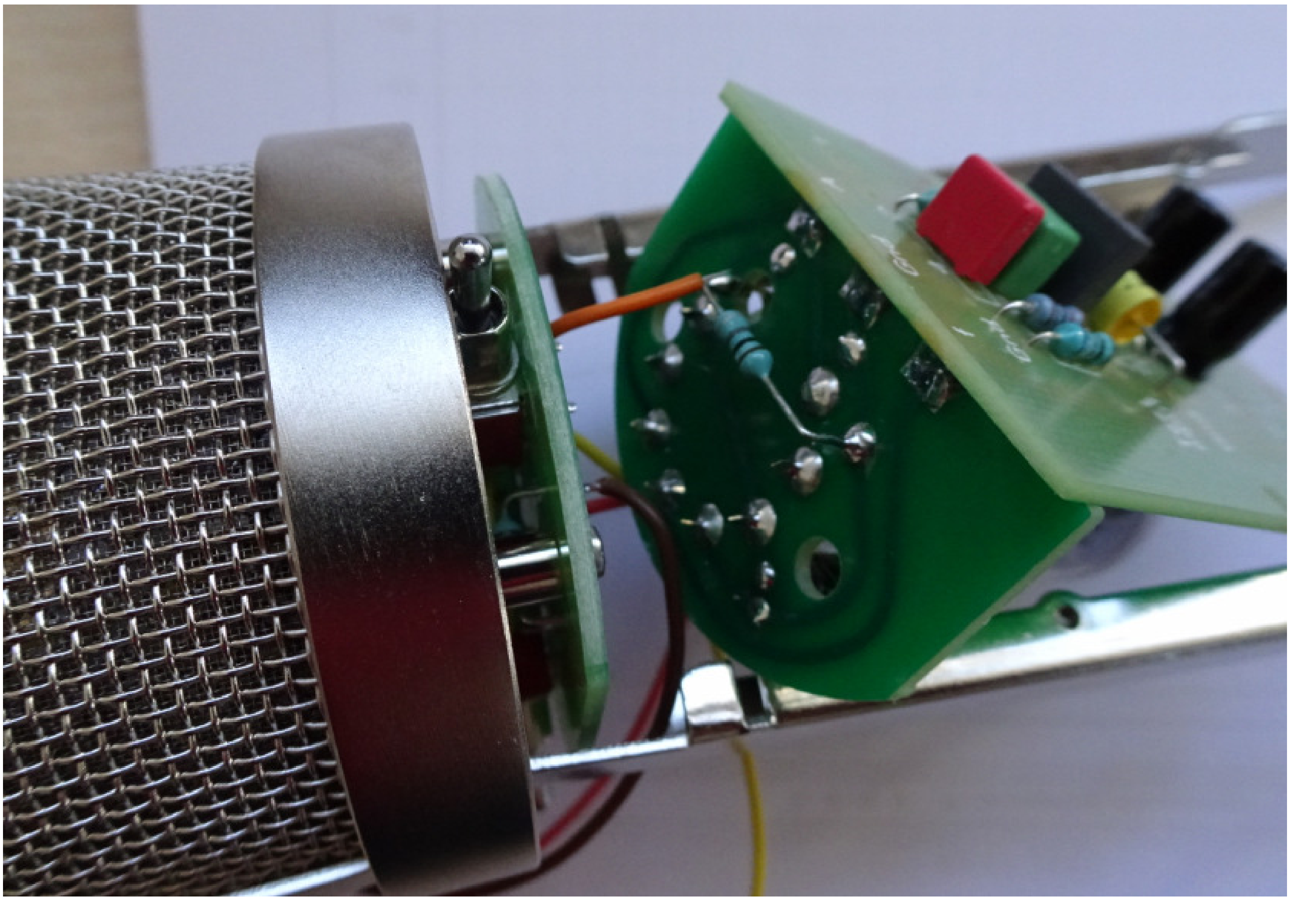
Take care of the polarity of the electrolytic capacitors! The ‘+’ side is marked on the PCB.

Note: with the 120 pF de-emphasis capacitor you will get the same frequency response as the original U67. If you prefer a slightly brighter microphone, place a 100 pF capacitor instead of the 120 pF one.

When the main PCB is finished, it should be soldered together with the tube PCB.
This is easier when you temporarily remove the headbasket and the metal base where the capsule mount is screwed on.
Slide the tube PCB in the slots of the ‘rails’.
Mount the main PCB with the four M2x4 screws on the rails.
Position the tube PCB in such a way that the solder point are as close together as possible.
Solder the three points together with some nice ‘blobs’ of solder.



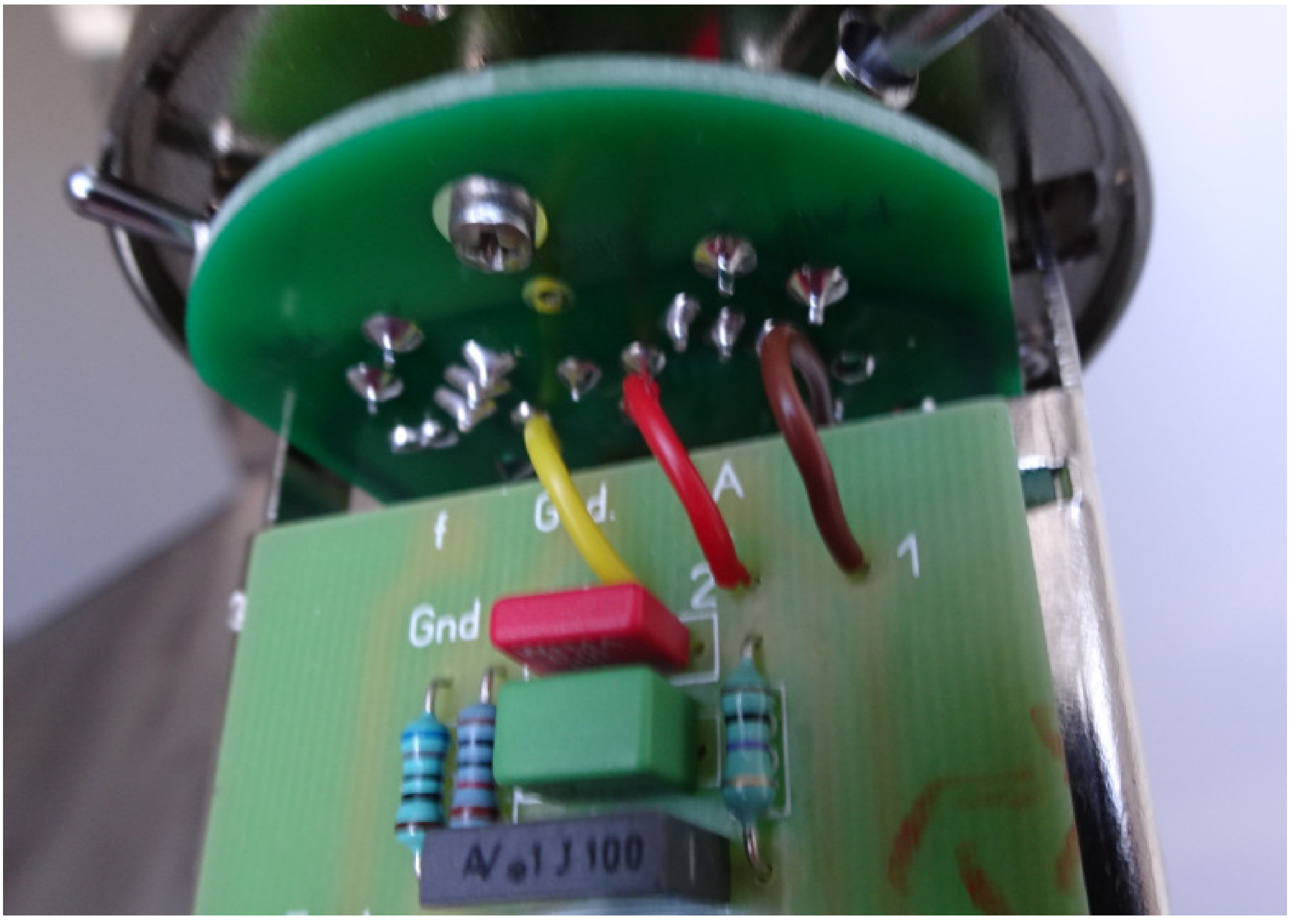
When this is done succesfully, mount the metal plate with the capsule again and put the headbasket back in place.
Now is the time to fasten the switch PCB with the long (10 or 12 mm) screws and the 6 mm. spacers.
It is easier if you take out the two PCBs that are soldered together now for the moment.
Now you can connect the wires from position 1, 2 and 4 of the switch PCB to the corresponding points 1, 2 and 4 on the main PCB.
Wire 3 from the switch PCB is connected **to the pin of the tube socked that is soldered to the 1 G.ohm resistor**. (The pin in the ‘big hole’.)



Also solder the wires to the corresponding points 1, 2 and 4

1 on the switch PCB should go to 1 om the main PCB, etc.

The colors in the picture can be chosen as you want, but different colors will make it easier to find which point of the switch PCB should be connected to which point on the main PCB.
It is a good idea to note the colors that are connected to the different numbers.

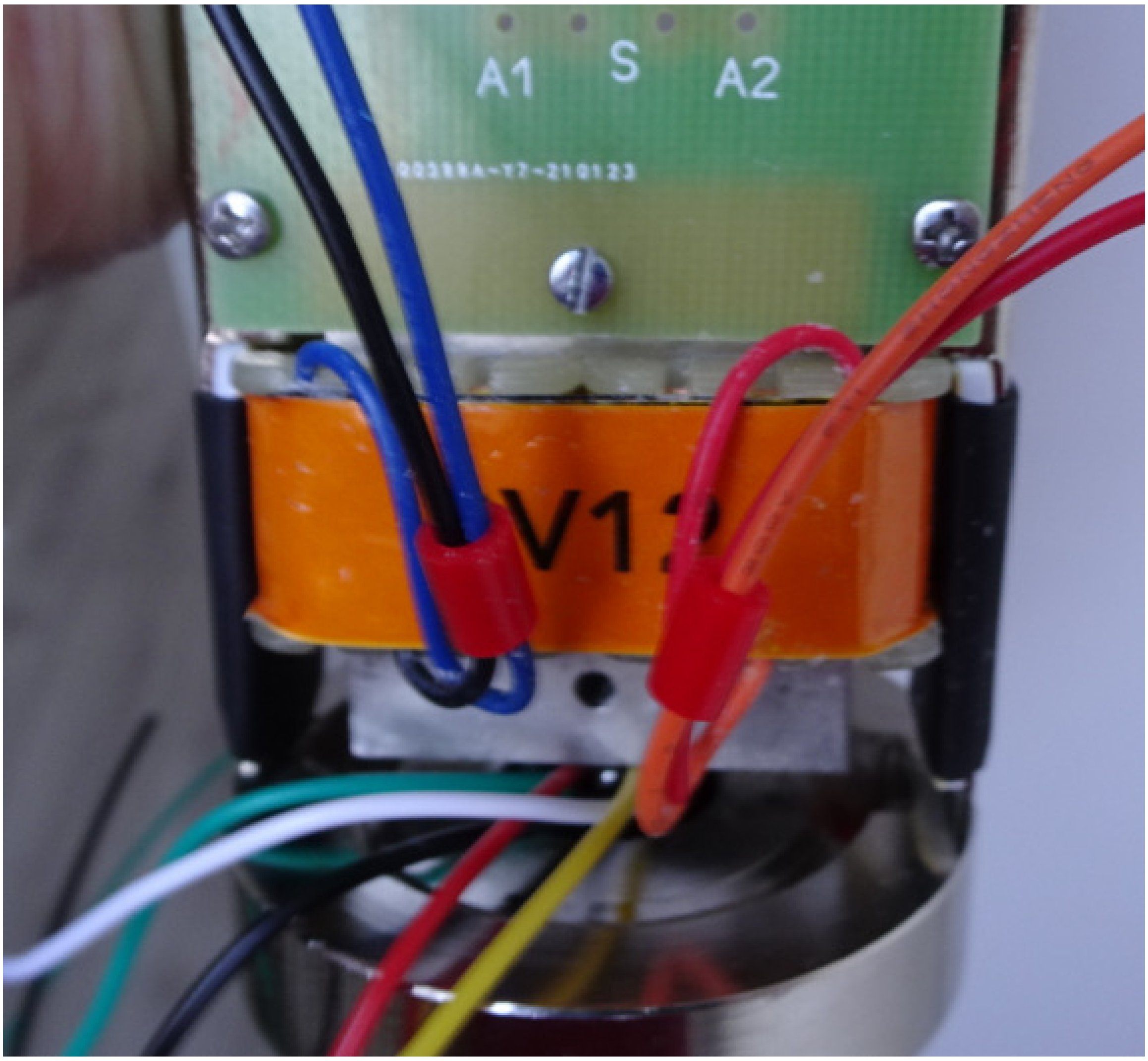


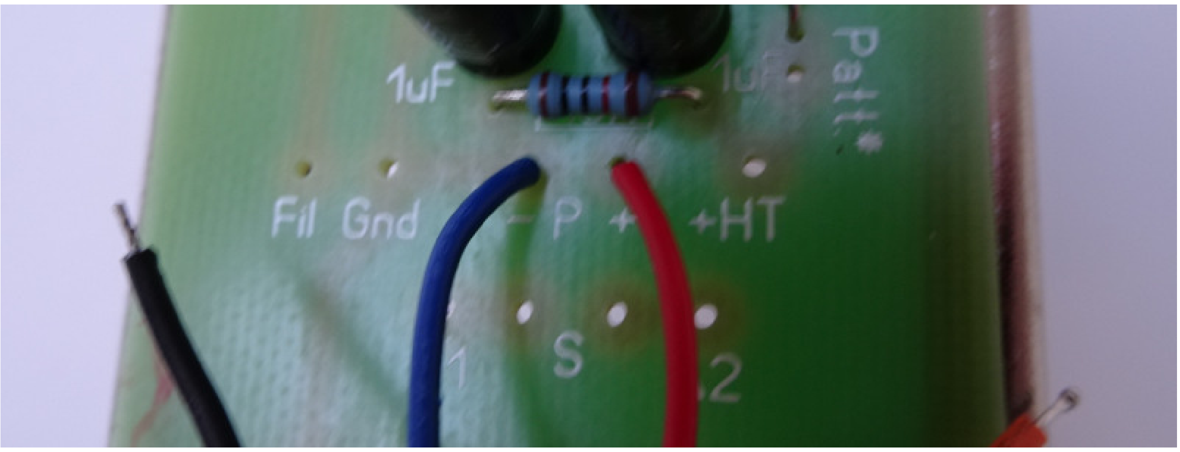
In this picture we followed the resistor color code: brown = 1, red = 2, yellow = 4.

If you have done this succesfully, we can mount the big transformer to the bottom of the main PCB. Use the (20 mm) long screw with nut to do this. Take care that the metal core of the transformer doesn't touch metal parts of the frame of the microphone body.
Maybe it is even a good idea to place a little piece of adhesive tape on the bottom side of the transformer, to avoid possible contact between the transformer core and the connections of the XLR connector. You may even add a drop of glue between the transformer core and the PCB to further secure the transformer in place. It is a tight fit, but it is possible!
Before glueing, remove the wax from the laminations of the transformer core where you are intending to put glue. It is also possible that the hole for the screw in the transformer core contains some wax. Usually this is easy to remove by simply pushing the screw in the hole of the core.

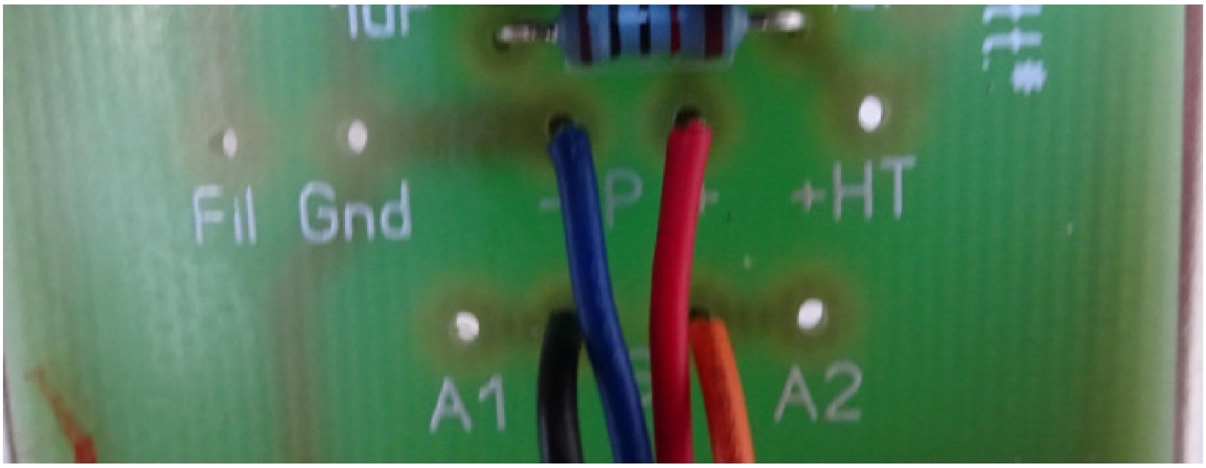
Do not overtighten the screw, because this may damage the laquer layer on the laminations of the transformer. The laquer layer isolates the laminations from each other.

Now solder the transformer wires in place. You may want to shorten the transformer wires (carefully!) a little. **Never pull** on the transformer wires, since this may damage the transformer! The red wire of the transformer is connected to the point 'P +' , the blue wire to 'P –'.



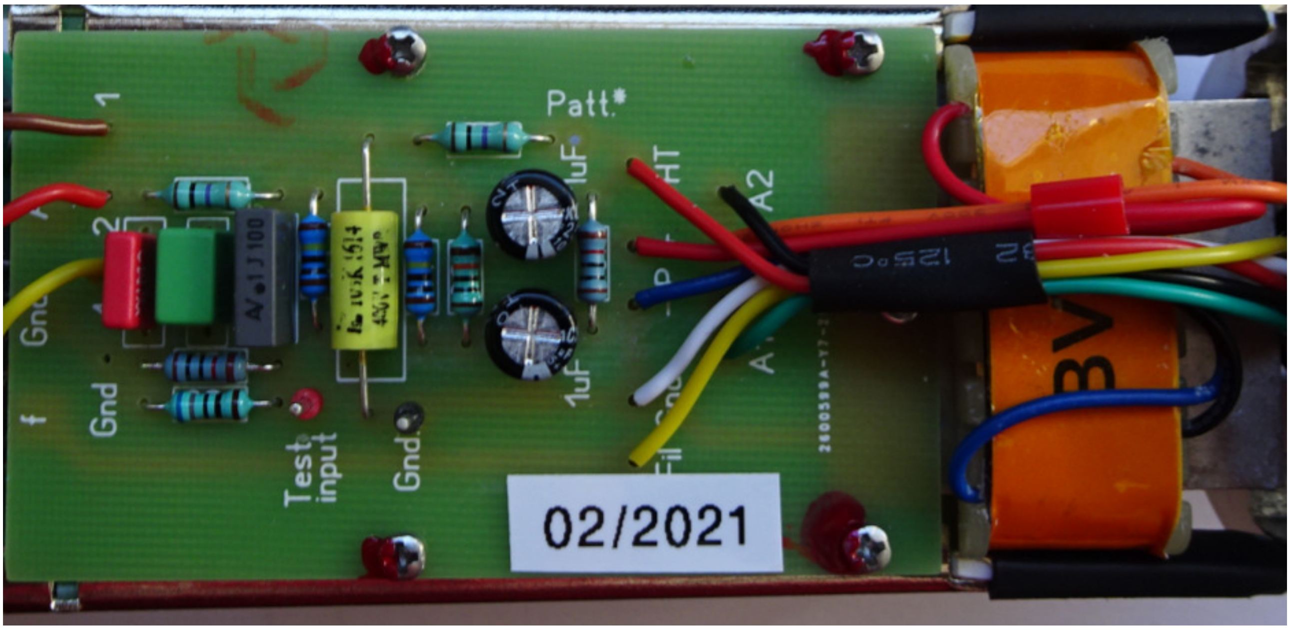


The black and orange wire from the transformer connect to ‘S +’ and ‘S –‘ (in no specific order).



Now the last thing to do is to connect the wires coming from the 7-pin XLR connector. Eventually, you may take of the bottom part of the microphone body, by removing the four countersunk screws on the bottom at the sides of the ‘rails’.

The way the wires should be connecteed is as follows:



Red (+120 V) to +HT on the main PCB
Yellow (+6,3V) to ‘Fil’ on the main PCB
White (Ground) to Gnd on the main PCB
Green (Audio) to one of the ‘S’ connections where the Black wire from the transformer is connected
Black (Audio) to one of the ‘S’ connections where the Orange wire from the transformer is connected.
You will notice that there is a **blue** wire left. You dont need this wire, unless you want to use the ‘remote pattern option’, described later on.

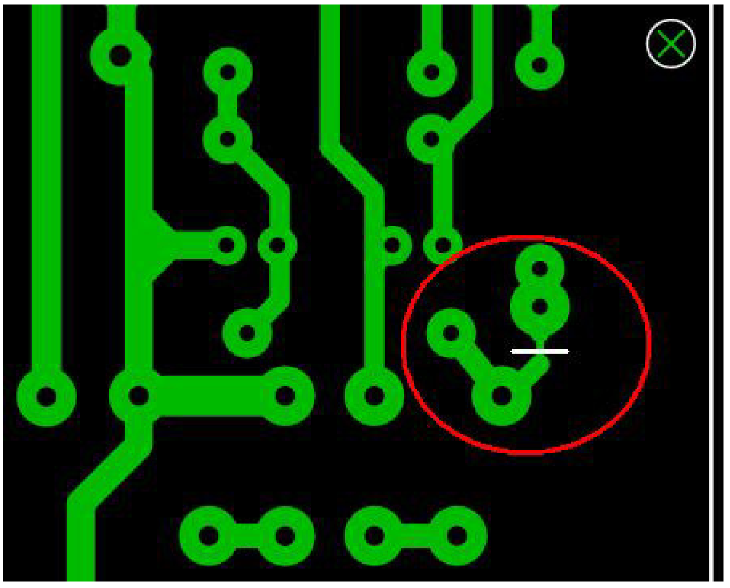
In principle the microphone is now finished and sholuld work.

You can now put the EF86 tube in the tube socket. Because the tube socket was never used before, the contacts are a bit stiff. It helps to fold a paperclip open and push the end of the paperclip is every pin socket. This will make it easier to put the tube in later.

Remote pattern control option.

We have added an option, to make it possible to change the directional pattern from the 9-position switch on the power supply. (If available.)
If you want to use this option, you have to do two things:

1- Interrupt the very small trace on the main PCB.



The white line shows where to cut.

2- Connect the blue wire to the spot marked: 'Patt*'.

The microphone now works as follows:

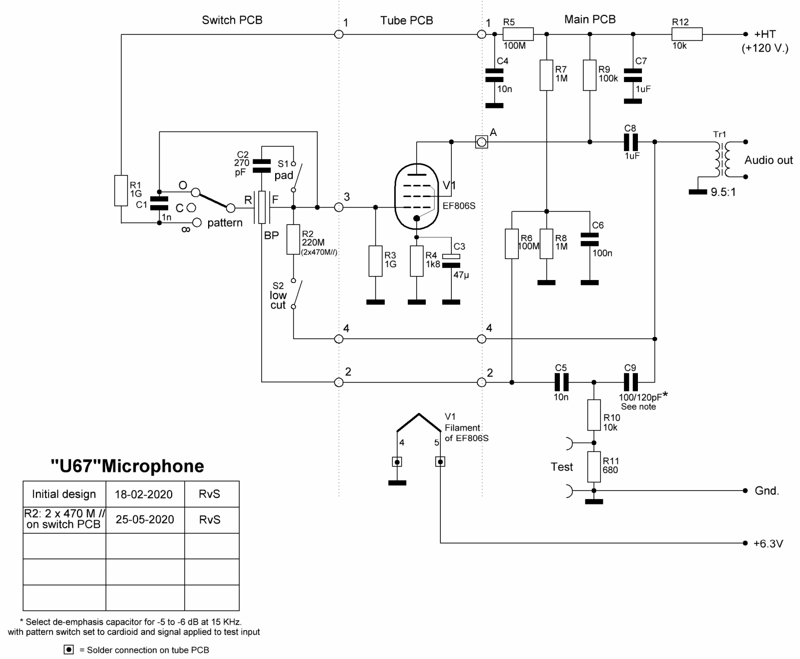
If you put the switch **on the microphone** on 'Figure-8', you can select from **omni** via **cardioid** to **figure-8** and positions in between(!), for example 'wide cardiod', with the knob on the power supply.

If you want to select the directional patters **on the microphone**, the rotary switch on the power supply should be in the **'Figure-8'** position.

Note that if you select the patters on the power supply, the cardioid position is no longer 'pure cardioid' and the output is in cardioid position lower than when you would have selected 'cardioid' on the microphone itself.

Version 1.4 12/2021-RvS

Addendum: schematic.



Finished microphone. (Prototype)