About the layout / design of the

Common Emitter/Collector Amplifier Board

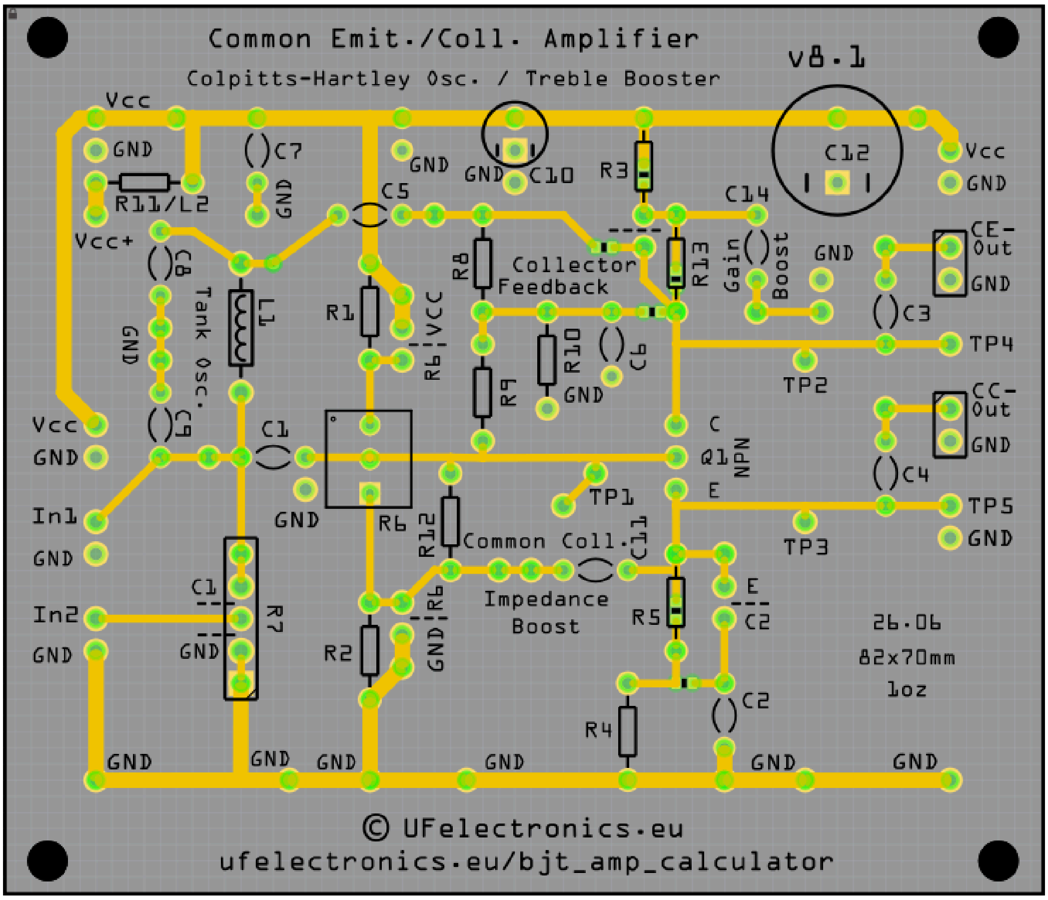
v8.1 - Commen_Emit_Coll_Amp_v8.1_2026.06.23-1210

Rev. 1 - 2026.06.28-1800

https://ufelectronics.eu/product/common-emitter-collector-amplifier-board-v8-1/

v8.1 has, compared to v8, additional footprints for:

- Vcc+ - R11/L2 - Facilitates a simple supply filter, isolating noise from/to the power rail.
- Gain Boost - R13, C14 - If you have an additional com. coll. stage.
- Impedance Boost - C11, R12 - If the stage is a com. coll. stage.
- A ground next to T5 - For a ground wire together with the feedback option to the gain boost.



To be updated: Section "Emitter Circuit Examples" - Add images of components mounted.

Foreword

A single-transistor amplifier may seem simple, and it is. But that simplicity is also its greatest strength as a learning tool.

When you build, tweak, and measure these circuits yourself, you quickly discover the gap between theoretical calculations and practical reality. Biasing is rarely perfect, gain is limited, and small changes can have large effects.

Once these limitations become clear through hands-on experience, the motivation to move toward more advanced topologies naturally arises. In that context, the differential amplifier stands out as a powerful next step, offering significantly improved stability, linearity, and control.

This board is therefore not just a tool for learning how transistor amplifiers work, but also for understanding why more advanced designs are needed.

In that sense, mastering the simple amplifier is not the end goal - it is the foundation for everything that follows.

It is through these limitations that real understanding begins.

Overall Concept

This is a PCB board for experimentation and testing of a single NPN BJT amplifier circuit.

Through-hole components are chosen to make it beginner friendly.

Overall Layout

The components are placed semi-schematic to make the mental shift between schematic and PCB as easy as possible; Helping users relate schematics to their physical implementation.

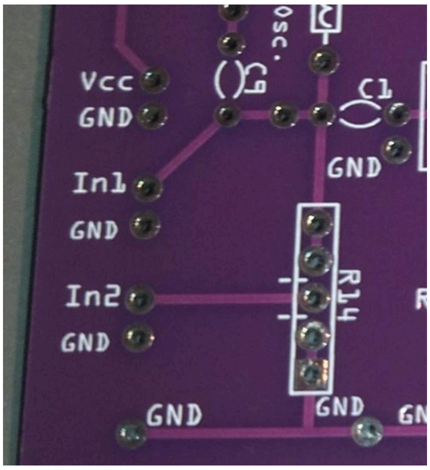
I would normally place the traces on the bottom side, when using through-hole components. But the traces are in this case by purpose placed on the top side, to make them visible without having to check the bottom for how the traces runs along.

The opposite site of the trace side, in this case the bottom side, is a full ground plane.

There is deliberately no ground plane on the top, the component side, to make it easier differentiate visually and to see the traces. For the same reason is the purple color of the solder mask chosen, so that the traces are easier to see.

The Various Parts of the Layout

Input/Output



Input Connectors - In1, In2, Vcc, Gnd

The left side of the PCB is intended to receive potential input signals. The right side to emit potential output signals. Both are spaced vertically so that one larger pin header, connecting to external circuits, can be used across the multiple input and outputs at the same time.

For the left input side we talk about the pairs Vcc-ground, In1-ground, and In2-ground.

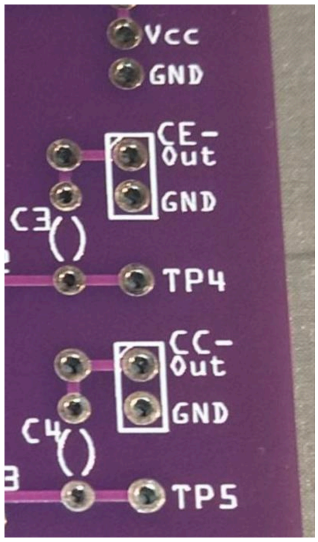
In1 goes directly to the coupling capacitor feeding into the base of the transistor. Where as In2 goes to a potential variable resistor/potentiometer, that can sway between feeding the same coupling capacitor as mentioned before, or ground.

Output Connectors - CE-out, CC-out, Vcc, TP4, TP5

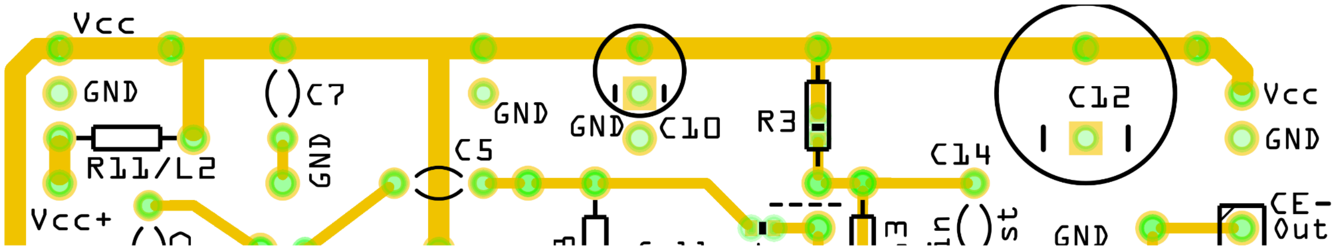
For the right output side we talk about the pairs Vcc-ground, CEout-ground, TP4, CCout-ground and TP5. - Where 'TP' stands for 'Test Point'.

On the right side, TP4 and TP5 are directly connected to collector and emitter of the transistor respectively. CEout and CCout are the decoupled signal of TP4 and TP5 respectively.

All power supply and signal connectors are followed by their own ground just next to them. This is to keep a potential magentic loop as small as possible and avoid noise. Partial exceptions are TP4.



Power supply and lab equipment connectors



Vcc

There are Vcc and ground connectors at the left-top and right-top. These are in pairs, and sitting close for connecting a single two-pin pin header, preferable male, as these make it easy to connect lab probes also.

There is an additional Vcc connector option a little separate from the the top-left Vcc-ground pair, so that the Vcc from a lab power supply can be attached here, without getting to close to a ground pin.

The Vcc trace has three footprints for decoupling capacitors. Also an unmarked option for a small fourth one.

Vcc+ - R11/L2

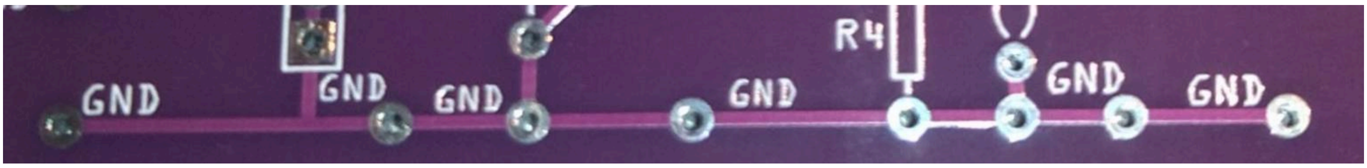
At the left-top side, you will now encounter 'Vcc+', added with the upgrade from version 8 to 8.1.

The Vcc+ input allows an optional resistor, ferrite bead, or inductor to be inserted in series with the supply voltage, R11/L2.

Together with the nearby decoupling capacitors, this creates a simple supply filter that helps isolate the circuit from noise on the power rail. It can also reduce the amount of noise that the circuit itself injects back into the power supply.

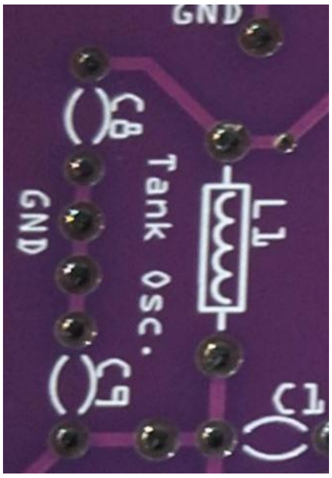
This makes it easier to experiment with supply filtering techniques and observe their effect on circuit performance.

Ground



At the lower edge of the PCB you will find several options for ground connected male pin headers. As with the Vcc, you also need the possibility to connect various lab equipment to ground. More even for ground than for Vcc.

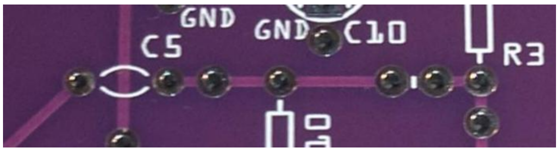
Colpitts osc. / Tank circuit - C8, C9, L1, C5



At the left side of the top-left quadrant, you will find the space for an tank circuit. The area labeled “Tank Osc.” represents a Colpitts oscillator configuration. This circuit consists of an inductor (L1) and two capacitors (C8 and C9).

The one capacitor (C9) and one side of the inductor (L1) is connected to the input side of the input coupling capacitor (C1). The same capacitor and inductor share a trace to In1 and one side of the potentiometer coming from In2.

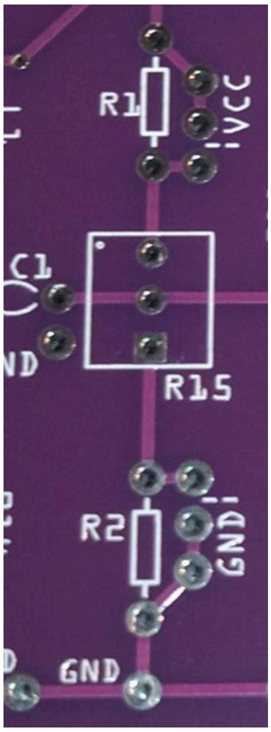
The top side of the Colpitts oscillator has a positive feedback from the collector via the capacitor footprint C5.



- **NOTE**: ***This feedback does not work, unless you solder the trace 'south' of C10 to the collector trace at R3***. - Give it a solder-blob.

A fixed trace is omitted here, to give the option of eliminating potential antenna effects of the feedback trace at higher oscillations.

Biasing the transistor - R1, R2, R6



R6

On the output side of the coupling capacitor (C1) going to the base of the transistor, you will see a footprint for a trimpot; R6.

R2

From R6 going to ground, you will find two options; Another footprint for another trimpot, next to a footprint for a standard resistor (R2). - Of course, only one of those are to be used.

R1

Again going from the trimpot R6 to Vcc, you will find an equal configuration for R1 as for R2.

Biasing

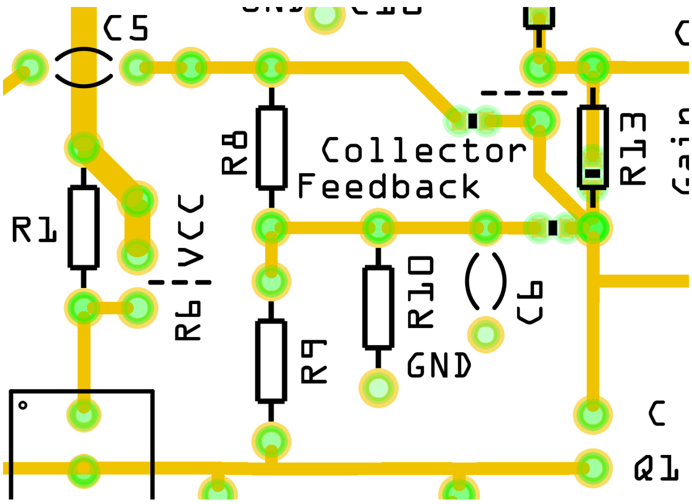
When you want to bias a transistor, you will often find that theory and practice are two different things. Even the best calculations can be hard to get correct, as not all factors can be included by ease without special tools.

That is the reason the various footprints for trimpots are included. It gives some flexibility after the components values are chosen.

A BJT Amplifier Calculator is available for getting rough estimates of the various resistors at:

https://ufelectronics.eu/bjt_amp_calculator - It's free to use and it's doesn't contain ads.

Feedback options



R8, R9, R10 and C6

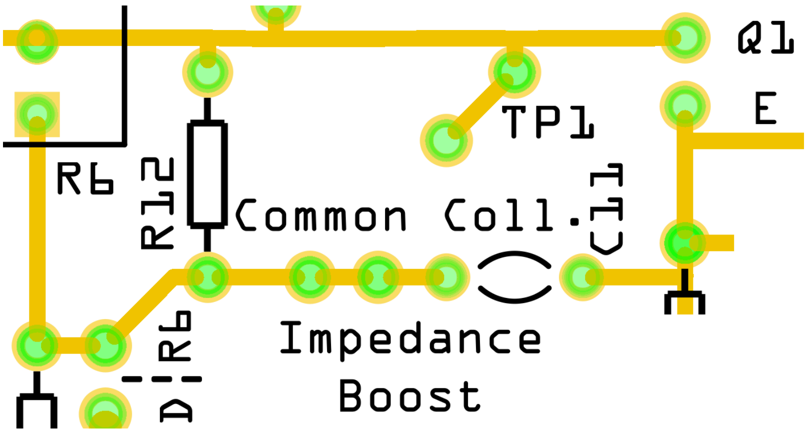
'North' of the trace going horizontally from R6 to the base of the transistor, can be used for various feedback circuits. This could be a simple treble booster or to create a crystal oscillator. Users can implement additional feedback networks based on their specific application.

C6

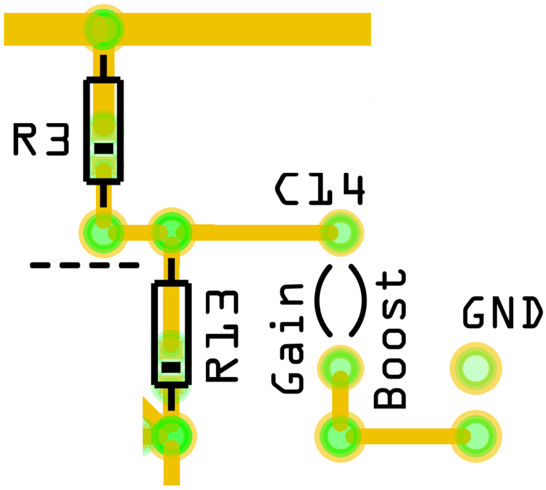
There is an option to make a solder bridge from the top pad of C6 to the collector of the transistor. Again, the use depends on your specific schematic.

C11, R12 - Impedance Boosting for Common collector, etc.

Just 'South' of the trace going horizontally from R6 to the base of the transistor, you will find another feedback option. Explore it yourself according to your wishes.



The collector resistor - R3, R13, C14



R3

This resistor R3 is either shorted when the circuit is configured as a common collector, or used as a fixed resistor for common emitter operation. - So here, isn't there implemented any footprint for a trimpot.

This resistor R3 might even get hot, depending on the values and transistor selected, which is even less ideal for trimpots.

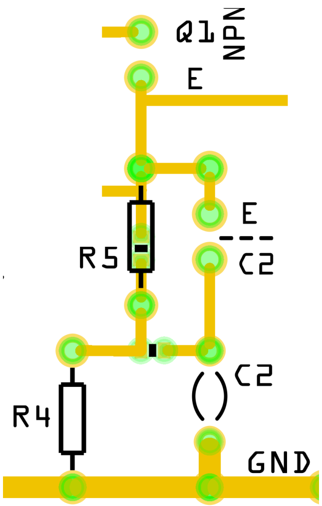
If you want a simple BJT amplifier, only using R3 as a single collector resistor, R13 is intended to be shorted by a solder-blob.

R13, C14 - and division of R3

If you of some reason want to divide a single R3 collector resistor into two resistors, you can so by using R3 and R13 in combination. - Now without shorting R13!

Using R3 and R13, together with C14, is it possible to implement gain boost at this place. The feedback to this place, 'south' of C14, would for instance have to come from the output of a common collector stage.- T5 and its GND-friend would be an optimal source.

- Pull a ground wire together with the signal wire. Even twist them. This reduces the magnetic loop of the feedback signal to a minimum.



The emitter side

SMD footprints for circuit variations - R5, C2

As there are various ways to shape the circuit on the emitter side, there is added SMD footprints to be able easily to connect it according to your circuit.

Below R5 you can see one of them. Give it a solder blob if you don't need R5 in your circuit.

To the left of the top of C2, you can see another SMD footprint. This serves as an optional place for adding a solder-blob, to connect the two traces.

R5

To the right of R5 is there an option for a trimpot. This trimpot adjusts the emitter degeneration and therefore controls the effective gain when C2 is used for bypassing. Inside the footprint of R5, as mentioned before, you find the SMD footprint for optional short circuiting it with a solder blob.

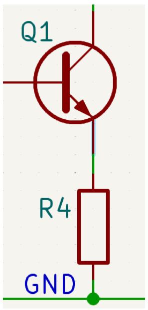
R4

This is the 'standard' emitter resistor, potentially bypassed by C2. No trimpot footprint included here. Just change the resistor if needed like you would do with R3.

C2

This is obviously the bypass capacitor for the emitter of the transistor circuit.

Emitter Circuit Examples



Ex. 1 - Degeneration Resistor

R4 is acting as a degeneration resistor. R4 improves the temperature stability of the gain, on the cost of overall less gain. Without R4, there is only the internal emitter resistance 're' in the transistor.

re is calculated as follows: re = 26 mV / Ie mA

The initial estimated DC gain (Av) is calculated as Av = R3 / (R4 + re).

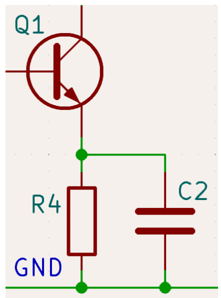
Image of the components on the PCB

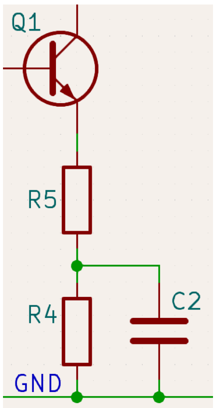
Image of the components on the PCB

Ex. 2 - Bypass Capacitor

This example is adding C2 as bypass capacitor, to improve gain to a "maximum".

C2 blocks the direct current (DC, from the biasing of R3 and R4), but lets alternating current (AC) through. This means that AC signals don't "see" R4, they go "directly" to ground via C2, and hence are amplified to a "maximum".





Ex. 3 - Splited Degeneration Resistor

Adding C2 can increase the gain so much, that the signal could distort. So adding R5 can reduce the gain of the bypass capacitor C2.

The DC voltage will se the sum of R4 and R5, while the AC signal only will see R5.

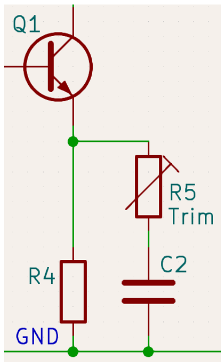
Image of the components on the PCB

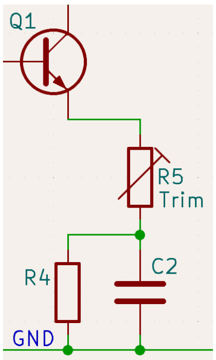
Image of the components on the PCB

Ex. 4 - Attenuated Bypass Cap. - Recommended Configuration

As a fixed value for R5 can be hard to determine from the start, an option to add a trimpot instead is available. This can eliminate R5 as being in series with R4, and makes the circuit easier to adjust, as the modulation of the amplification isn't tied to the overall resistance R4 and R5 in series.

In this example, DC sees only R4, and AC ses only R5.



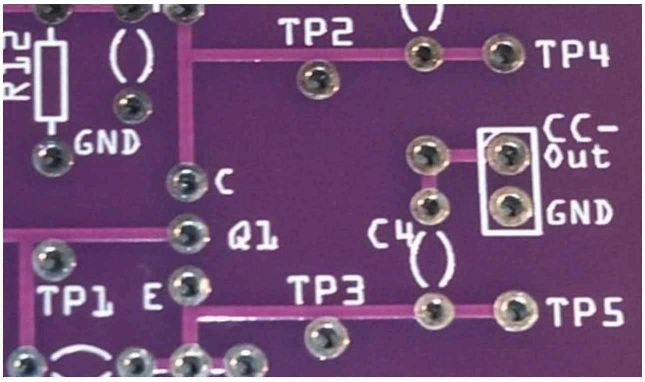


Ex. 5 - Variable Splited Deg. Resistor

If you liked example 3, this one gives you some flexibility by substituting the fixed value of R5 with an trimpot. Be aware that the gain will vary too, as you adjust the trimpot, due to the initial estimated DC gain is Av = R3 / (R4 + R5 + re).

Image of the components on the PCB

Test Points



Next to TP4 and TP5, there is TP1 on the base, TP2 on the collector and TP3 on the emitter.

TP4 is equal to TP2. Used as output for the common emitter configuration.

TP5 is equal to TP3. Used as output for the common collector configuration.

All the mentioned test points can be mounted with male pin headers for easy connection of test equipment. Without pin headers, they are easier to hold a probe on for a short quick measurement.

The test points are obviously not real PCB test points, as those are without a hole in the middle. The choice of the hole is due to it making it easier to fix a probe in, without the probe slipping around.

Suggestions?

Have suggestions for improvements, or need a personalized PCB design? - Feel free to get in touch:

Contact

https://ufelectronics.eu/about/contact/

Social

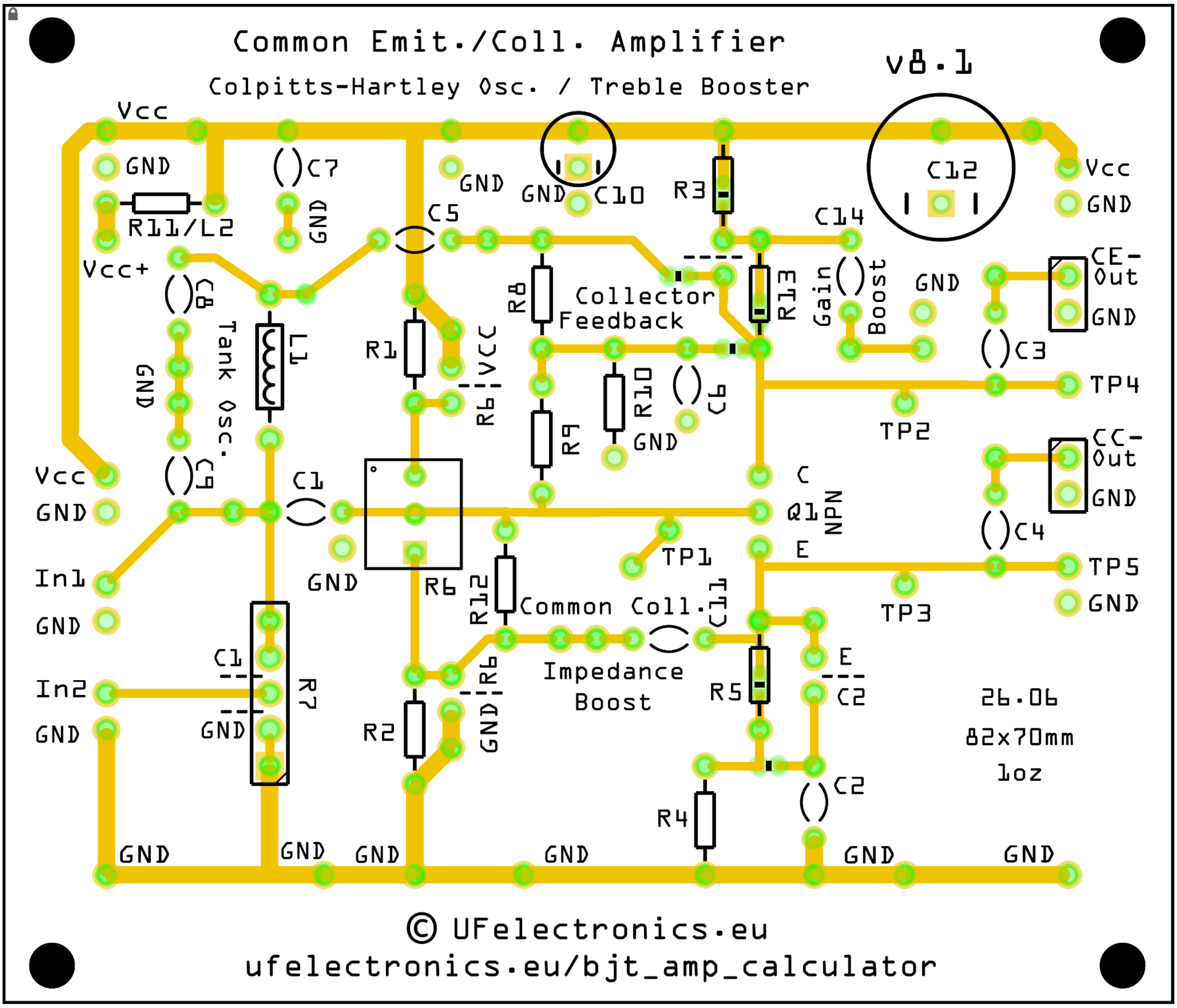
YouTube: https://www.youtube.com/@ufelectronics

Discord: https://discord.com/channels/1373552306817339535/1373552307467325542

Facebook: https://www.facebook.com/profile.php?id=61577320711532

LinkedIn: https://www.linkedin.com/company/ufelectronics/posts/?feedView=all

Semi-Schematic Layout



The figure above shows the routing on the top layer of the PCB.

Ground is routed on the bottom side of the PCB. The bottom layer with the ground plane is not visible in the figure above.