

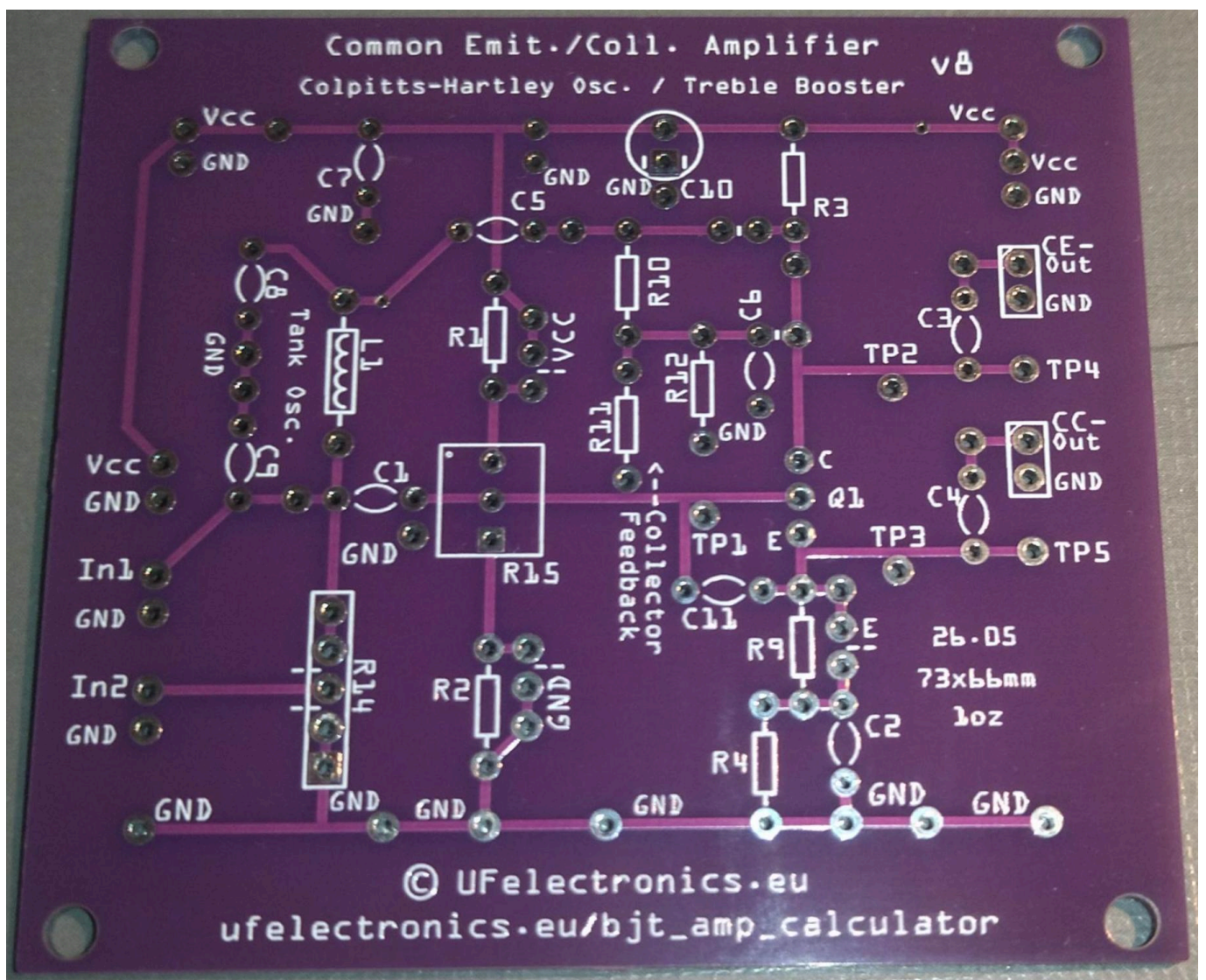
About the layout / design of the

Common Emitter/Collector Amplifier Board

v8 - Commen_Emit_Coll_Amp_v8_2026.05.25-1000

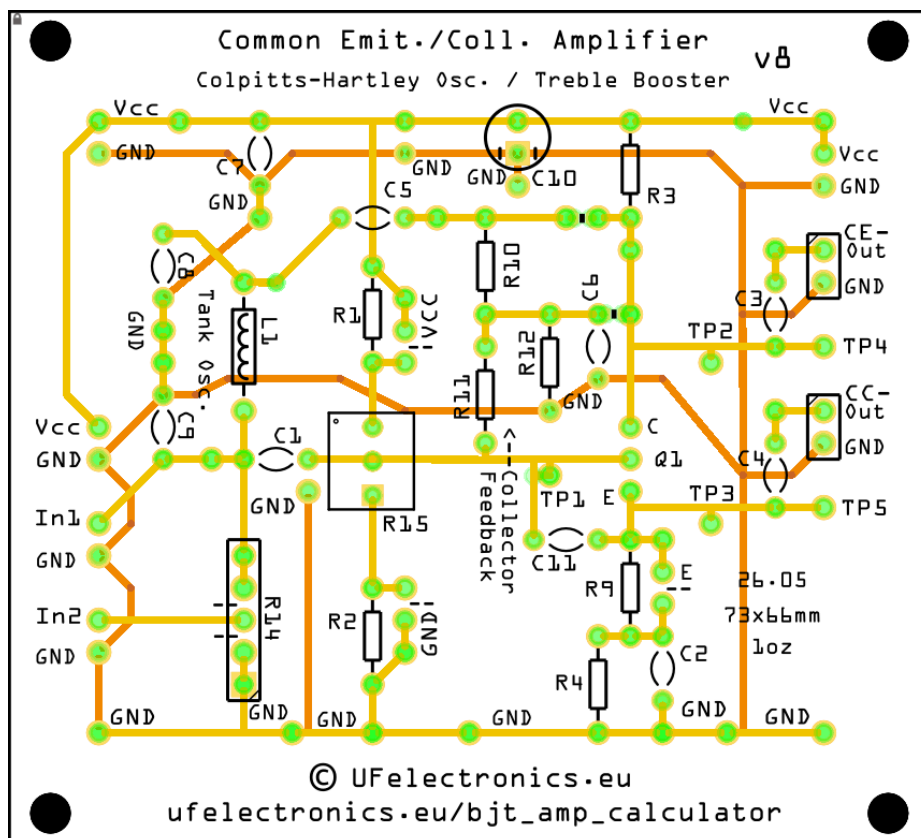
Rev. 2 - State: Under Revision

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



Semi-Schematic Layout	2
Foreword	3
Overall Concept	3
Overall Layout	3
The Various Parts of the Layout	4
Input/Output.....	4
Power supply and lab equipment connectors.....	4
Colpitts osc. / Tank circuit.....	5
Biasing the transistor.....	6
Feedback options.....	6
The collector resistor R3.....	6
The emitter side.....	7
Test Points.....	7
Suggestions?	8
Contact.....	8
Social.....	8
Semi-Schematic Layout in Big	9

Semi-Schematic Layout



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 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.

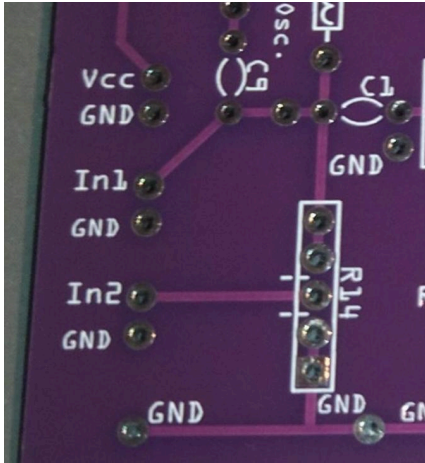
I would normally place the traces on the bottom side, when using through-hole components. But the traces are in this case by purpose placed on the top side, to make them visible without having to check the bottom for how the traces runs along.

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



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.

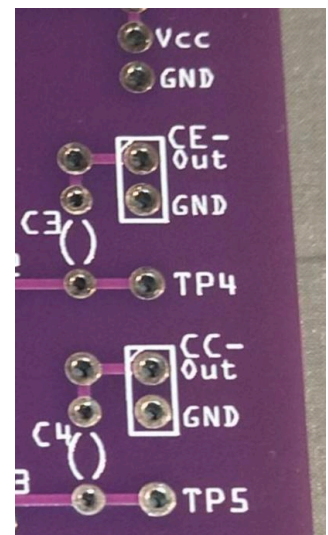
For the left input side we talk about the pairs Vcc-ground, In1-ground, and In2-ground.

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 magnetic loop as small as possible and avoid noise. Partial exceptions are TP4 and TP5.

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.



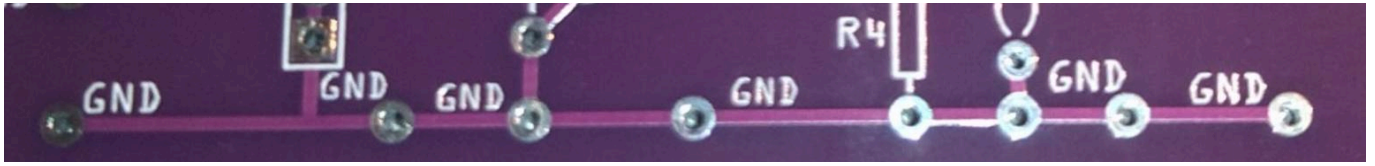
Power supply and lab equipment connectors



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.

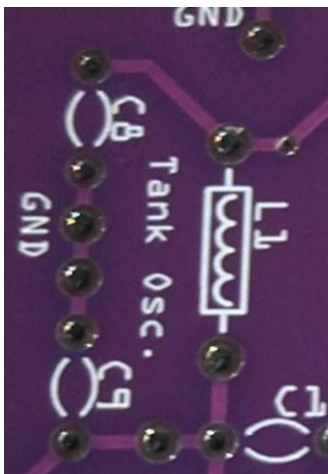
There is an additional Vcc connector option a little separate from the the top-left Vcc-ground pair, so that the Vcc from a lab power supply can be attached here, without getting too close to a ground pin.

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



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



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 C5.



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

A footprint for a double pin header for a jumper is available at that point. 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

R15

On the output side of the coupling capacitor (C1) going to the base of the transistor, you will see a footprint for a trimpot; R15.

R2

From R15 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 R15 to Vcc, you will find an equal configuration (R1).

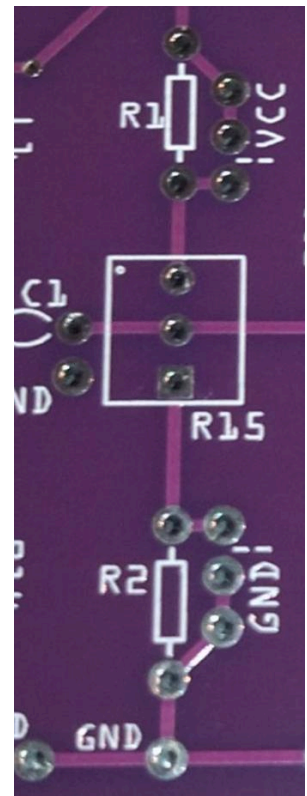
Biasing

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. Also without ads.



Feedback options

R10, R11, R12 and C6

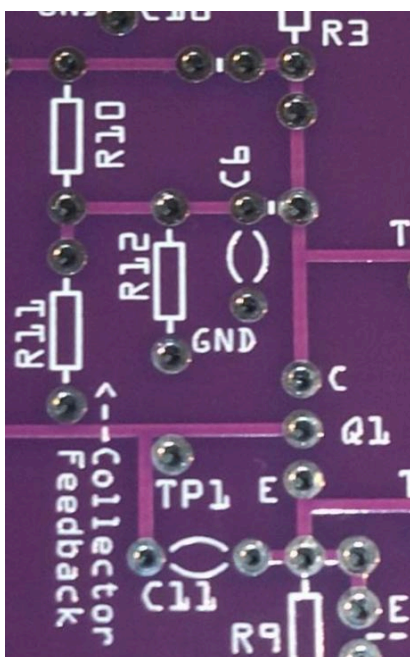
'North' of the trace going horizontally from R15 to the base of the transistor, can be used for various feedback circuits. This could be a simple treble booster or to create a crystal oscillator. Users can implement additional feedback networks based on their specific application.

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.

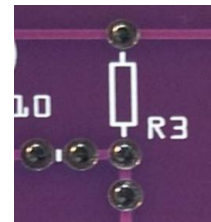
C11

Just 'South' of the same trace, you will find a feedback option too. Explore it yourself according to your wishes.

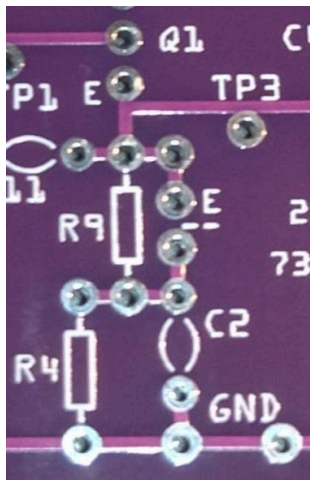


The collector resistor R3

This resistor is either shorted when the circuit is configured as a common collector, or used as a fixed resistor for common emitter operation. So here there is not implemented a footprint for a trimpot. This resistor might even get hot, depending on the values and transistor selected, which is even less ideal for trimpots.



The emitter side



R9

To the right of R9 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.

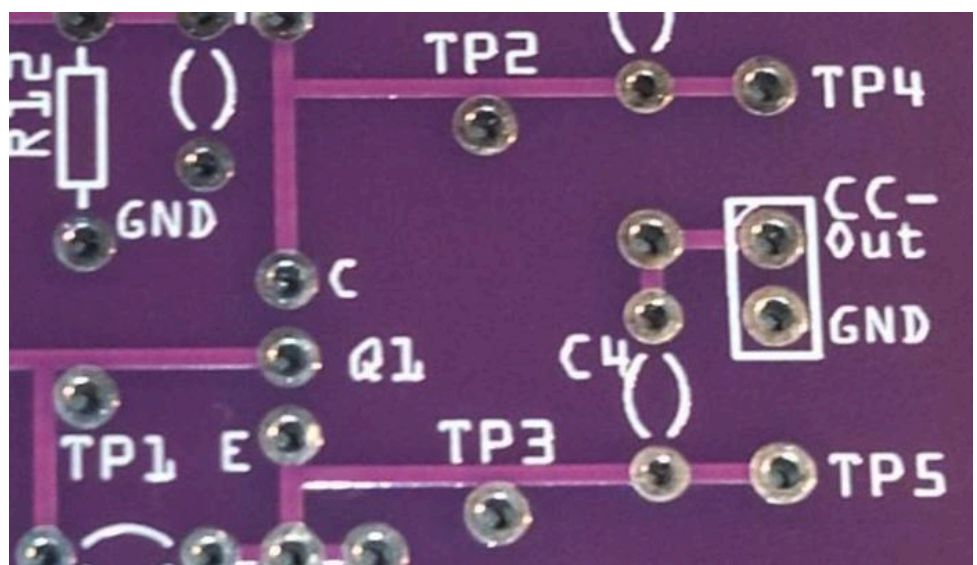
R4

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.

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, and TP5 is equal to TP3.



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 in Big

