About the layout / design of the

1/4 H-Bridge xCh

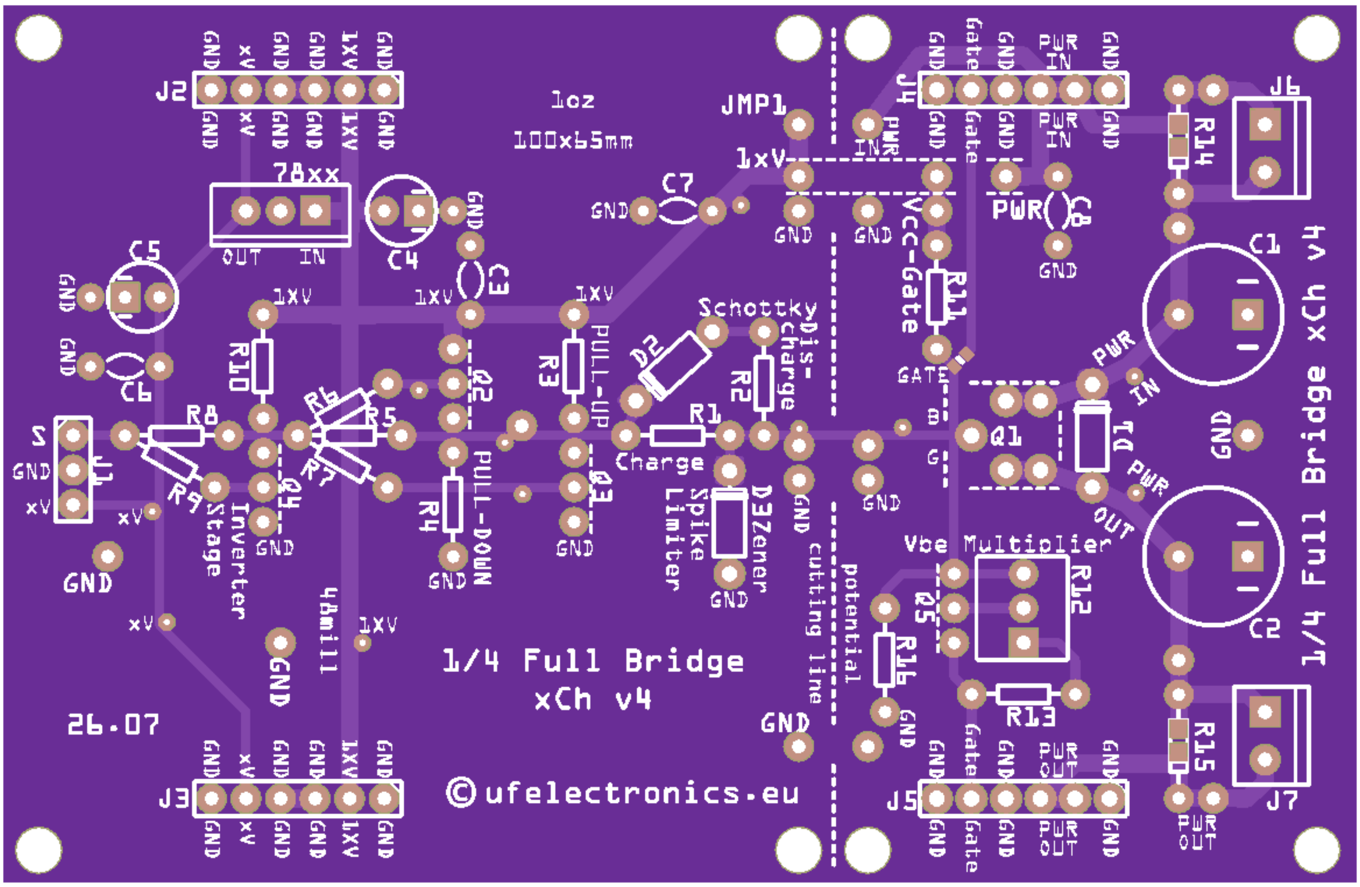
v4 - Full_Bridge_1-4_xCh_v4_2026.07.03-2200

Rev. 1 - 2026.08.05-1200

https://ufelectronics.eu/product/1-4-h-bridge-xch-v4-one-set-of-two-pcbs/

v4 has, compared to v1, improved in the following way:

- A clearer, and potential easy physical division of the left and right functionality.
- Better integration of two boards 'north-south' of each other.
- Screw terminals for accessing drain/collector and source/emitter.
- Option to add a resistor on the source/emitter to ground (R16).



Foreword

This board is intended for experimenting with MOSFETs and BJT power transistors, which throughout this document are collectively referred to as output transistors.

It is designed both for evaluating and testing output transistors, and for learning the fundamental concepts behind their operation, such as gate drive, the Miller plateau, switching losses, thermal compensation, and output stage biasing. Once you are familiar with these concepts, the board can also become part of your own circuit designs, for example as the output stage of a Class AB audio amplifier or other power electronics projects.

The PCB is designed so that two boards can be connected together using 90-degree male and female pin headers. Together they form the upper and lower halves of a push-pull output stage, using either MOSFETs or complementary BJT power transistors.

Power is distributed between the two boards through the interconnecting headers. Each board may have its own signal input while sharing the common supply rails (GND, xV and 1xV).

Using two boards connected in a north-south arrangement, you can build and experiment with a half H-bridge, also known as a totem-pole or push-pull stage.

Using four boards, you can build and experiment with a complete full H-bridge.

The board supports both N-channel and P-channel MOSFETs, as well as complementary NPN and PNP power transistors such as the TIP31/TIP32 family.

If you are unfamiliar with H-bridges, you can read more here:

https://en.wikipedia.org/wiki/H-bridge

This PCB is intended as an educational and experimental platform. It is not intended to be a production-ready motor driver or power amplifier PCB.

Overall Concept

This PCB is designed as an experimentation and learning platform for MOSFETs and BJT power transistors.

Through-hole components have been chosen throughout the design to make assembly, modification, and experimentation as easy as possible, especially for beginners.

Overall Layout

The components are arranged in a semi-schematic layout, making the transition from circuit schematic to physical PCB as intuitive as possible. This helps relate the schematic symbols to their physical implementation on the board.

With through-hole components, PCB traces would normally be routed on the bottom layer. For this board, however, the signal traces are deliberately routed on the top layer so they remain visible while working with the assembled board. This makes it much easier to follow the signal paths without having to turn the PCB over.

The entire bottom layer is used as a continuous ground plane to provide a low-impedance ground connection and improve noise performance.

There is intentionally no ground plane on the top (component) side. This makes the individual traces much easier to distinguish visually. The purple solder mask was also chosen because it provides good contrast, making the copper traces easier to follow during experiments.

Division of the PCB in Two Sections

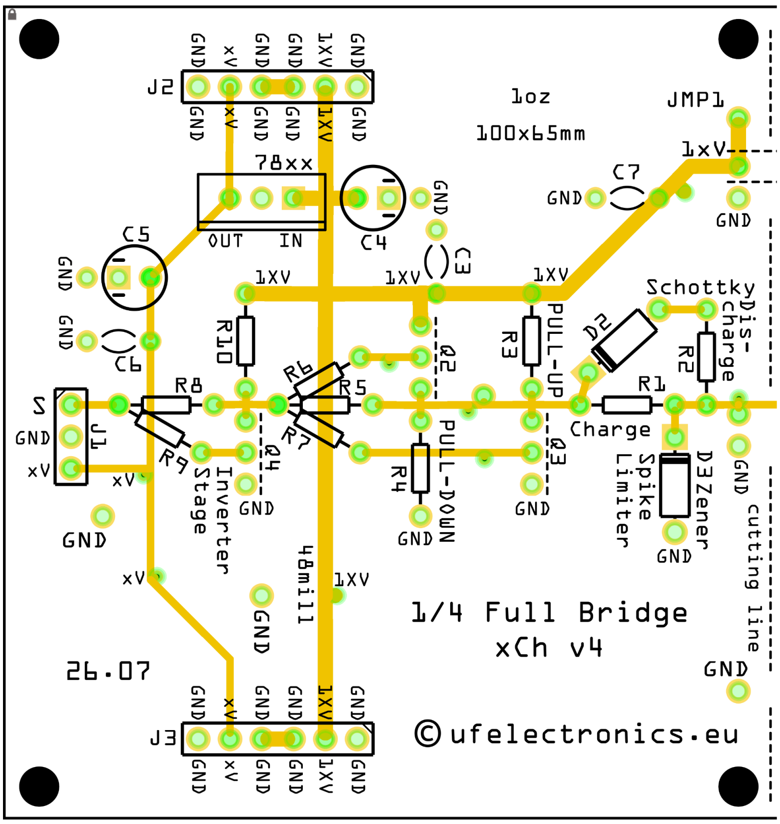
The PCB has been designed with an optional division into two sections, indicated by the vertical dotted line approximately 60% from the left edge.

For most applications, the board should be used as a single unit and the division can simply be ignored.

If your application only requires one of the two sections, however, the PCB can be cut along the marked line. This reduces the physical size of the board while retaining the functionality you need.

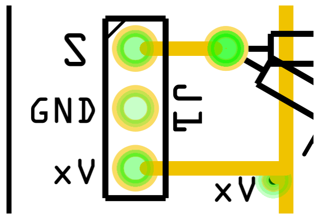
Both sections include footprints that allow power and signals to be reconnected if the split board is later used together again. This is described in the section "Bridging Vertical Split Boards."

The Left Section - The 'Cold' Side



Signal Source and Power to the Signal Source - J1 (GND, xV, S)

The digital control signal enters the board through the three-pin header J1 on the left side of the PCB.



J1 provides the following connections:

- GND - Ground reference.
- xV - Low-voltage supply (typically 3.3 V or 5 V).
- S - Digital control signal.

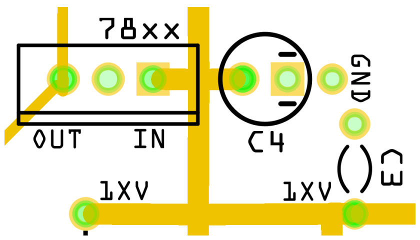
The xV pin can also be used to supply power to the signal source, for example a microcontroller or signal generator. This is useful when the control electronics require the same supply voltage that is available on the PCB.

The relationship between the xV and 1xV supply rails is explained in the next section, "Dual Power - 1xV, xV."

Dual Power - 1xV, xV

Some output transistors benefit from a higher gate-drive voltage than the 3.3 V or 5 V typically provided by a microcontroller. For this reason, the board provides two separate supply rails:

- 1xV - Higher supply voltage, typically 10 V to 19 V, for driving the output transistor.
- xV - Lower supply voltage, typically 3.3 V or 5 V, for the signal source and associated logic.



To generate the xV supply from the higher 1xV rail, the PCB includes a footprint for an optional 78xx linear voltage regulator. You can populate the regulator with the output voltage that best suits your application, for example 3.3 V or 5 V.

If your application does not require a higher gate-drive voltage, the regulator can simply be omitted. In that case, the xV and 1xV rails may be connected together, allowing both the control circuitry and the output transistor to operate from the same supply.

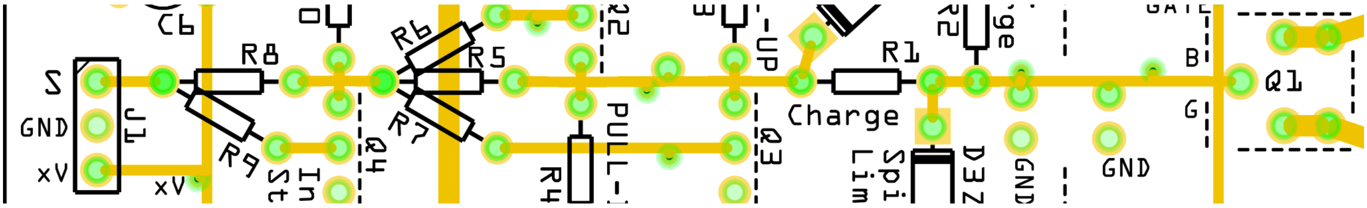
MCU 3.3V / 5V Direct Drive

Use case: Driving the output transistor directly from a 3.3 V or 5 V microcontroller I/O pin using only a gate/base resistor.

Component List	
MOSFET	FQP30N06L
R21	10 Ω

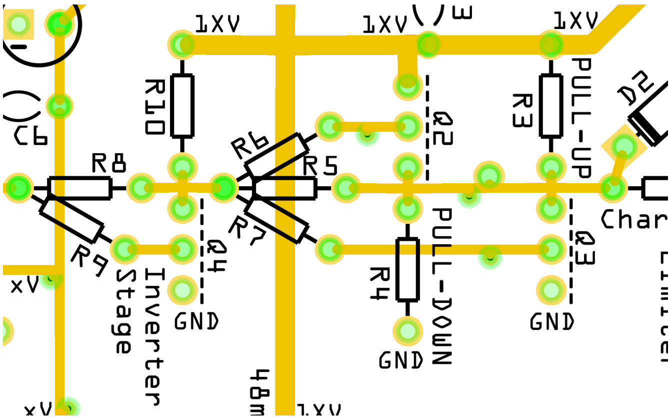
Many logic-level MOSFETs, such as the FQP30N06L, can be driven directly from a 3.3 V or 5 V microcontroller without requiring additional driver circuitry. In this configuration, only a single gate resistor is typically needed.

To use this configuration, route the signal from the S pin on J1 directly to the gate or base of the output transistor using the appropriate jumpers. No signal conditioning is required except of maybe a resistor.



Before selecting the jumper configuration, refer to the section "Gate Transition Control", which describes the available options for controlling the charging and discharging of the gate (or the base drive, when using BJTs).

Signal Conditioning



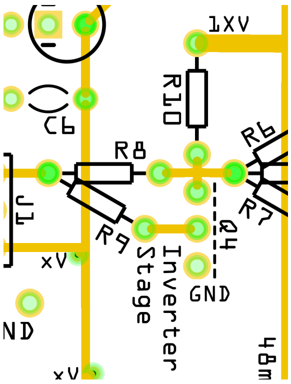
Before reaching the output transistor, the input signal can optionally pass through one or more signal-conditioning stages.

These stages allow the signal to be adapted to the requirements of your application, for example by:

- inverting the logic level,
- defining the default state after power-up,
- increasing the drive capability,
- or driving the output transistor from a higher supply voltage.

The following sections describe the available signal-conditioning blocks.

Inverting the Signal - R10, Q4



Some applications require the output transistor to respond with the opposite logic level. For example, a high input signal may need to turn the output transistor off instead of on.

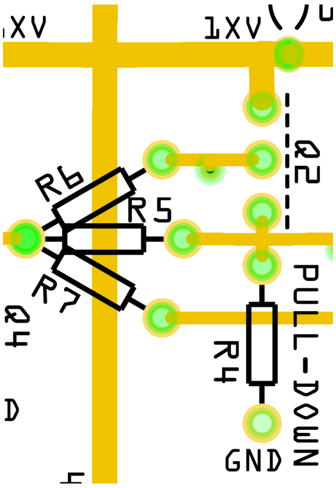
Populating R10 and Q4 inserts a simple inverter into the signal path, allowing the output logic to be reversed.

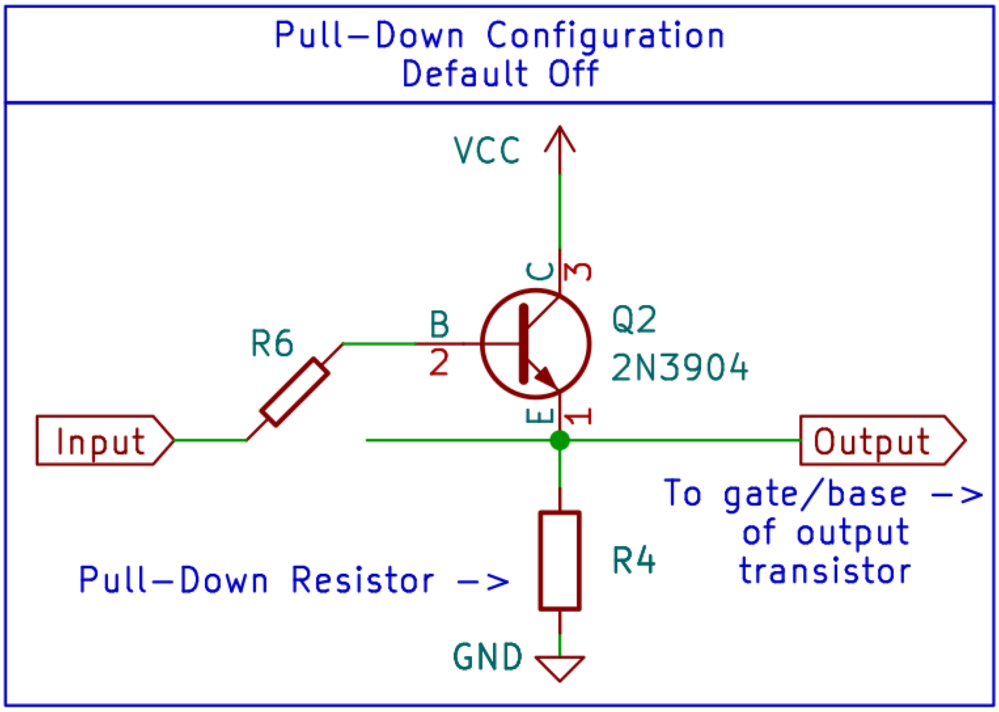
Not to be used in conjunction with pull-up or the pull-down configurations, except of the push-pull configurations.

Optional: A 39 kΩ resistor from the base of Q4 to ground may be added to ensure that Q4 remains OFF whenever no input signal is present.

Default Off - Pull-Down Resistor - Q2, R4, R6

Use case: The output transistor should remain OFF by default until instructed otherwise.





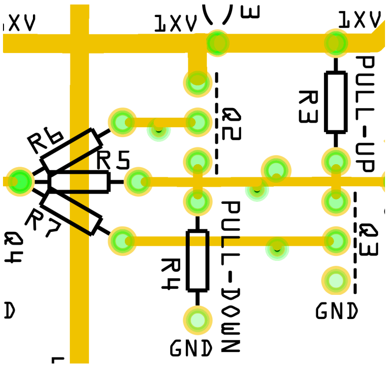
If the output transistor should remain OFF whenever the control signal is absent - for example during power-up or while the MCU is resetting - you can populate R4 and Q2.

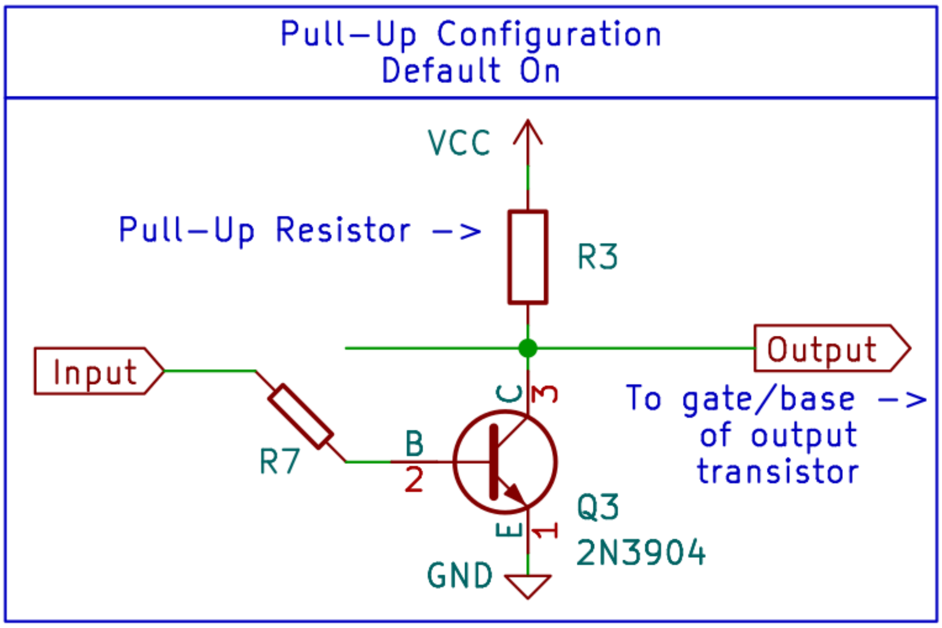
This configuration ensures that the output transistor defaults to the OFF state until an active control signal is applied.

This circuit can also be used when the output transistor must be driven from a higher voltage than the MCU can provide directly.

Default On - Pull-Up Resistor - R3, Q3, R7

Use case: The output transistor should remain ON by default until instructed otherwise.





Some applications require the output transistor to default to the ON state whenever the control signal is absent. This may be desirable during power-up or if the signal source becomes disconnected or loses power.

By populating R3, Q3 and R7, the output transistor is pulled into the ON state until an active control signal overrides it.

As shown in the schematic to the right, this circuit also allows the output transistor to be driven from a supply voltage higher than the MCU can provide directly.

Push-Pull (Totem-Pole) Driver - Q2, Q3

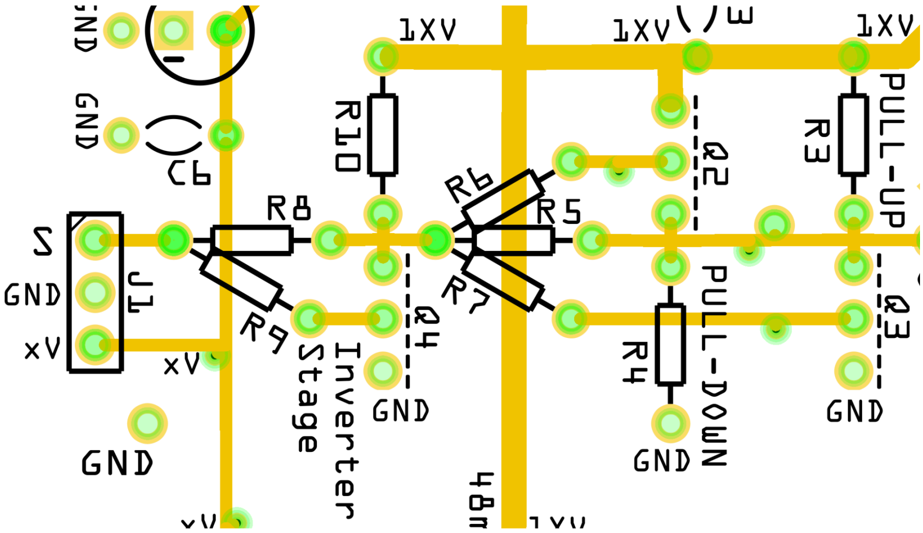
Use case: High switching frequencies, MOSFETs with large gate capacitance, or applications requiring a higher gate-drive voltage than the MCU can provide directly.

A push-pull (totem-pole) driver actively both charges and discharges the gate of a MOSFET. Compared to driving the gate through a resistor alone, this provides much higher peak current in both directions, resulting in faster switching and lower switching losses.

Faster gate charging and discharging reduce the time the MOSFET spends in its linear operating region. This reduces switching losses, improves efficiency, and enables reliable operation at higher switching frequencies.

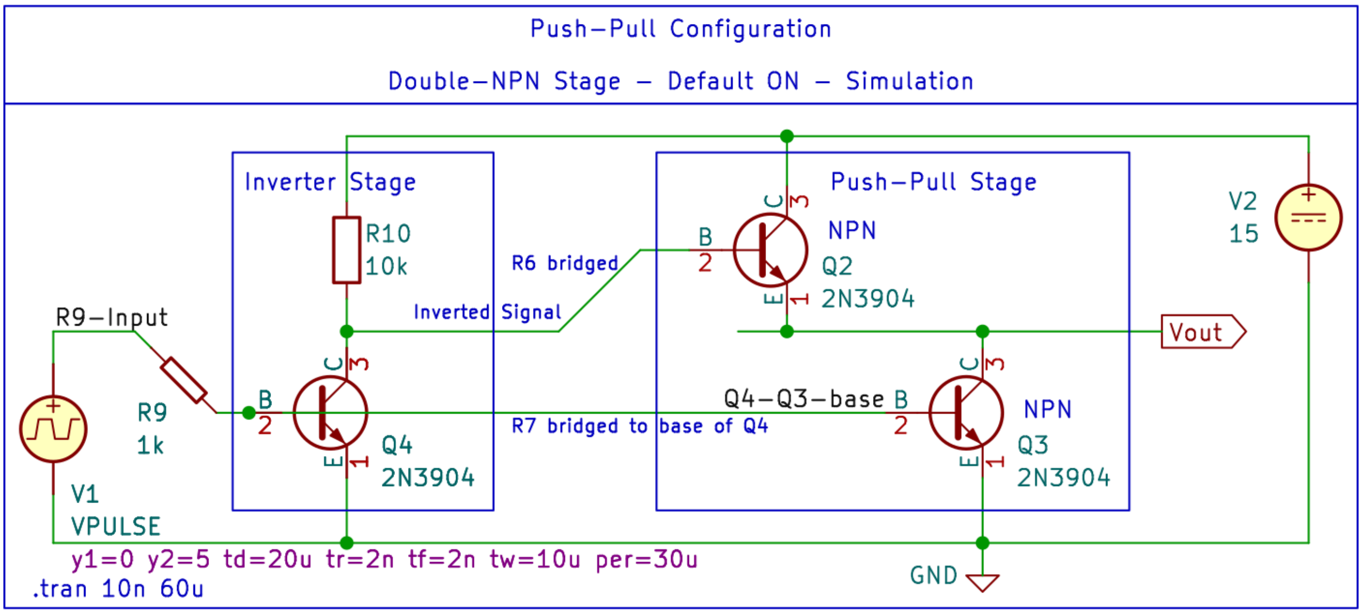
This configuration also allows the gate to be driven from the 1xV supply rather than directly from the MCU. For example, a logic signal of 3.3 V or 5 V can be used to switch a MOSFET with a 12 V or 15 V gate-drive voltage, provided the MOSFET's maximum gate-to-source voltage is not exceeded.

When using this configuration, R3 and R4 are normally omitted.

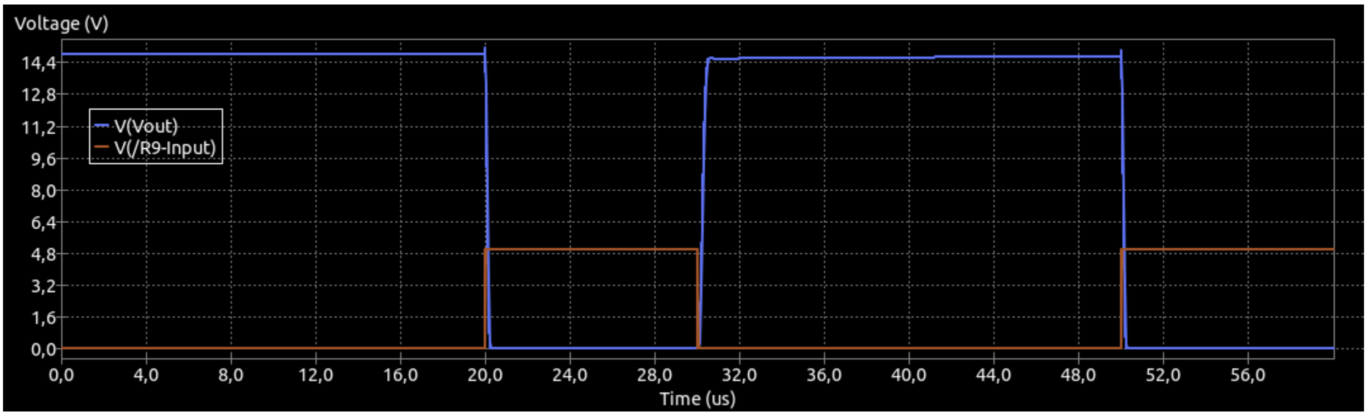


Schematic Example and Simulation

This is an example of the push-pull stage.



Example of push-pull (totem-pole) gate driver using the 1xV supply.

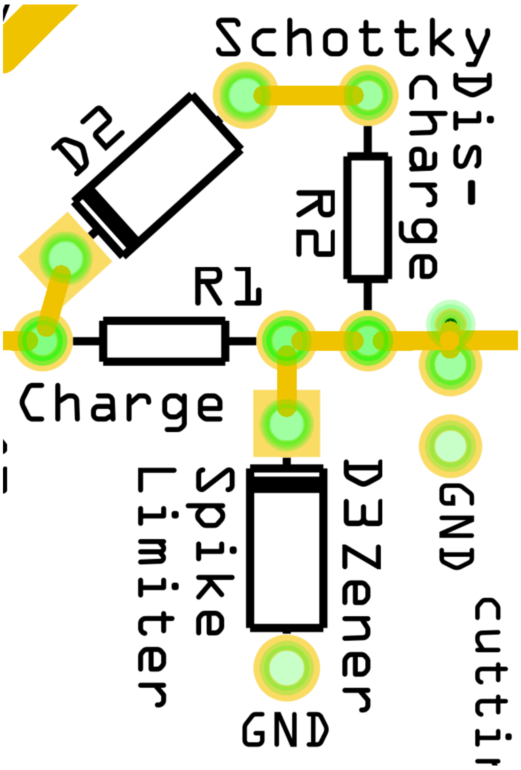


Simulation of the former circuit: Input -> Output

Driving Inductive Loads

Note: When driving inductive loads such as motors, relays or solenoids, always add a flyback diode across the load. Footprint D1 just right of Q1 is provided for this purpose.

Gate Transition Control - R1, R2, D2 (Schottky), D3 (Zener)



MOSFET gates behave like capacitors and must be charged and discharged whenever the transistor switches state. The speed of these transitions affects switching losses, electromagnetic interference (EMI), and the overall switching performance of the circuit.

This section allows the charging and discharging speeds to be adjusted independently while also providing optional gate-voltage protection.

Gate Charging - R1

R1 controls how quickly the MOSFET gate charges when switching the transistor ON.

Increasing the resistance slows the gate charging, reducing switching speed and potentially lowering EMI. Decreasing the resistance charges the gate more quickly, resulting in faster turn-on.

Gate Discharging - R2, D2 (Schottky)

By populating R2 and D2, the gate charging and discharging paths become independent.

This makes it possible to use one resistance for charging the gate and another for discharging it. In many applications, a faster discharge is desirable, as it turns the MOSFET OFF more quickly without unnecessarily increasing the turn-on speed.

A faster gate discharge reduces the time the MOSFET spends switching, helping reduce switching losses and allowing higher switching frequencies.

Gate Spike Limiter - D3 (Zener)

The D3 footprint allows an optional Zener diode to be connected between the gate and source.

The Zener diode protects the MOSFET gate from excessive gate-to-source voltage caused by voltage spikes or ringing during switching.

Choose a Zener voltage that is:

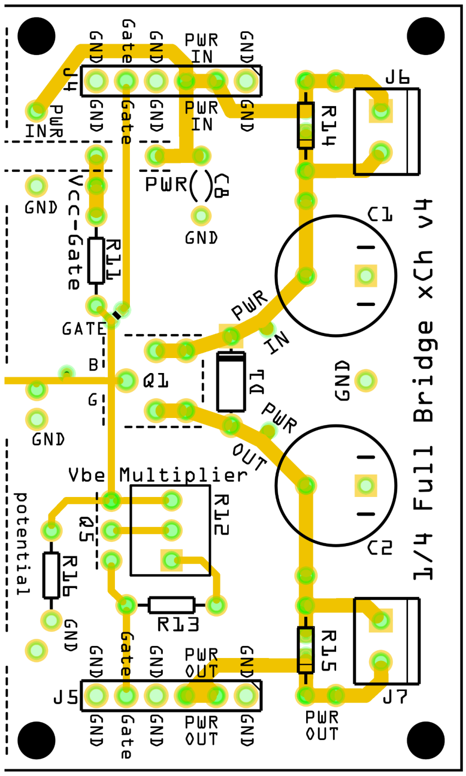
- higher than your normal gate-drive voltage,
- but lower than the MOSFET's maximum gate-to-source voltage (V_GS(max)).

For example, if the MOSFET is driven with 12 V and has a maximum V_GS rating of ±20 V, a 15 V Zener diode is a common choice.

The Right Section - The 'Hot' Side

This section contains the circuitry directly associated with the output transistor.

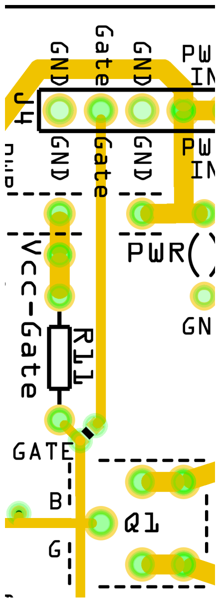
Here you will find the power connections, output terminals, protection components, and optional bias circuitry used to support the output stage.



'North' of the Output Transistor - Gate/Base Drive - R11

The area above the output transistor contains the connections used to drive the gate or base of the output transistor.

The drive can either be generated locally on the PCB or supplied externally through J4. The J4 connector carries the complete interface, including ground, supply voltage, and the gate/base drive signal, making it easy to connect another board or an external driver circuit.

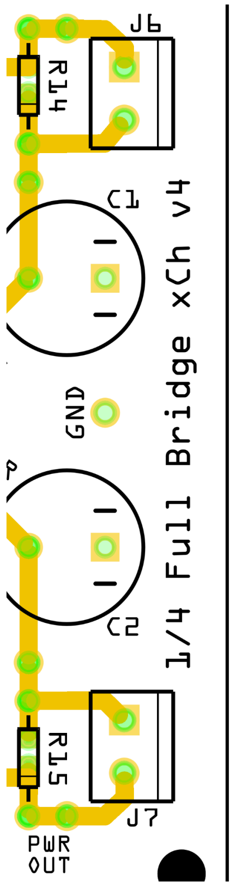


VCC Gate Biasing - R11

R11 provides an optional series resistor between the selected supply rail and the gate (or base) of the output transistor.

Depending on the application, this resistor can be used to limit the gate/base drive current, reduce ringing, or adjust the switching speed of the output transistor.

'East' of the Output Transistor - J6, J7, R14, R15, D1



This area provides convenient access to the output transistor terminals, optional series resistors, and protection against inductive loads.

Tap into Drain/Collector and Source/Emitter - J6, J7

Along the right edge of the PCB, connectors J6 and J7 provide direct access to the drain/collector and source/emitter of the output transistor.

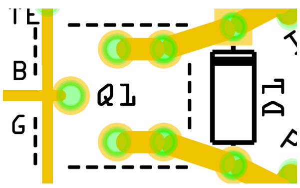
These connectors make it easy to connect the output stage to external circuits or measurement equipment. They also provide convenient locations for adding capacitors or other components as required by your application.

Optional Series Resistors - R14, R15

R14 and R15 provide footprints for optional series resistors in the drain/collector and source/emitter paths.

These resistors may be used for a variety of purposes, including current limiting, current sensing, damping, or simply as convenient insertion points for experimentation. Depending on the application, higher-power resistors may be required.

The Flyback Diode - D1

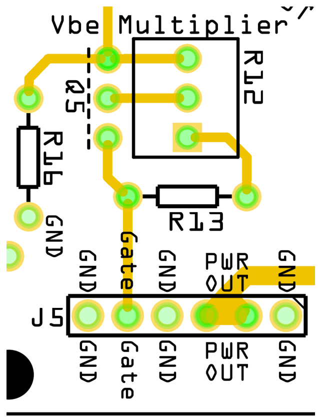


When switching inductive loads, such as motors, relays, or solenoids, the collapsing magnetic field generates a high reverse voltage that can damage the output transistor.

Footprint D1 allows an external flyback diode to be mounted across the load, protecting the output transistor from these voltage spikes.

'South' of the Output Transistor

Bias Voltage Generator - VBE Multiplier - R12, R13, Q5



A Class AB output stage requires a small voltage between the bases of the two output transistors to establish the correct idle (quiescent) current.

Traditionally, this voltage can be generated using two series-connected diodes. The VBE multiplier provides an adjustable alternative with the added benefit of thermal compensation.

The board includes an integrated VBE multiplier below Q1.

Typical starting values are:

- R13: 1 kΩ or 820Ω
- R12: 2 kΩ trimmer
- Q5: 2N3904, 2N2222, BC547, or a similar NPN transistor

The VBE multiplier generates the required base-to-base voltage and by that establish the correct idle (quiescent) current for the output transistors. By adjusting this voltage, the quiescent current of the Class AB output stage can be set, minimizing crossover distortion.

When the VBE multiplier transistor is mounted on the same heatsink as the output transistors, it also compensates for temperature changes, helping to keep the quiescent current stable.

Adjusting the Vbe Multiplier

1. Set R12 to its minimum resistance.
2. Apply power.
3. Measure the quiescent current by monitoring the collector current of Q3 or, alternatively, observe the total supply current.
4. Slowly increase R12 until the desired quiescent current is reached.

For the example circuit, 2-10 mA is a reasonable starting point. In general, the quiescent current should be adjusted to the lowest value that eliminates crossover distortion. Depending on the output transistor used, this might be higher than 2 - 10 mA.

Not Using the Vbe Multiplier - R16

The VBE multiplier is only required when building a Class AB BJT output stage.

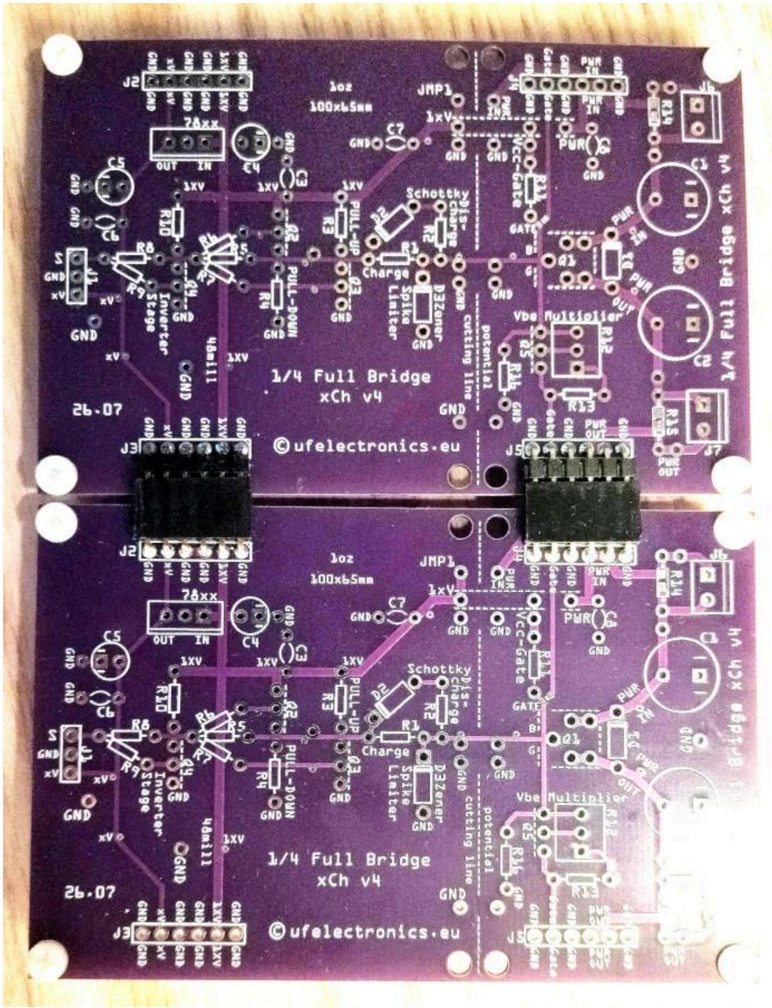
If the board is instead used as the lower output transistor in another circuit - for example with a MOSFET - the VBE multiplier can simply be omitted.

In that case, R16 may be populated with a resistor to ground, or replaced with a wire link if the source or emitter should be connected directly to ground.

Connecting Multiple Boards

A single board is well suited for experimenting with an individual output transistor. Typical experiments include:

- direct MCU drive,
- gate-drive circuits,
- switching speed,
- gate charge and discharge,
- switching losses,
- gate resistor selection,
- flyback diode protection,
- Class A and Class B transistor stages,
- and general MOSFET or BJT characterization.



The horizontal board interconnectors J2, J3, J4 and J5 make it easy to connect two boards together, extending the range of possible experiments.

Using two boards, you can build and experiment with:

- a half H-bridge,
- a push-pull (totem-pole) stage,
- or the output stage of a Class AB amplifier.

Using four boards, arranged as two half H-bridges, you can build and experiment with a complete H-bridge.

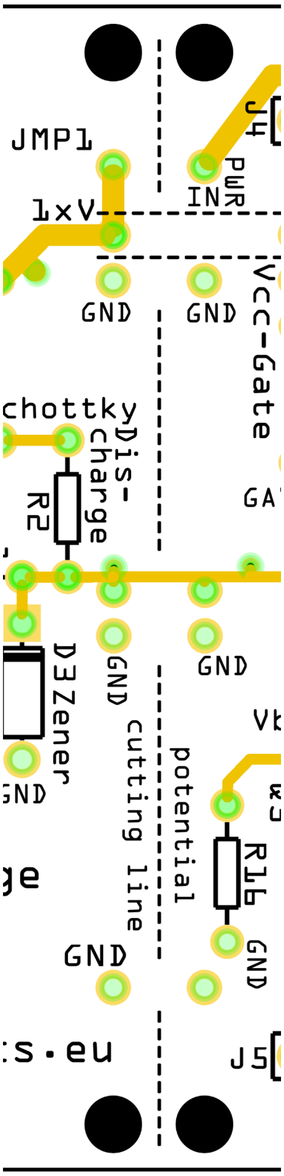
Power Supply

The board interconnectors are designed so that multiple boards can share both their power supplies and control signals.

- J2 and J3 distribute the common supply rails (GND, xV, and 1xV) between boards.
- J4 and J5 distribute the output-stage connections, including GND, the gate/base drive signal, and the power input.

When two boards are connected together, most of the required power and control signals are transferred automatically through these connectors, reducing the amount of external wiring

Bridging Vertical Split Boards



The PCB can be physically divided into two smaller sections to save space when only part of the circuitry is required.

If the two sections later need to be used together again, dedicated footprints allow power and signals to be bridged across the split.

Using 90-degree male and female pin headers, the two sections can simply be plugged together without any additional wiring, creating a compact and reusable connection.

This makes it possible to experiment with either section individually while retaining the option of quickly restoring the original functionality of the complete board.

Test Points

The PCB does not include dedicated, labeled test points. Instead, the design provides numerous accessible measurement points in the form of vias and single-pin header footprints distributed across the board.

This gives you the flexibility to probe almost any part of the circuit using an oscilloscope, multimeter, or logic analyzer, making the board well suited for experimentation and debugging.

Many of these locations can also be populated with single-pin headers, making it easier to connect oscilloscope probes or clip leads during experiments.

Decoupling Capacitors - C3, C4, C5, C6, C7, C8

These footprints are intended for decoupling capacitors. Each capacitor is placed as close as possible to the circuit it supports, helping to reduce supply ripple, provide local energy storage, and improve switching performance.

C3, C4, C7

These capacitors decouple the 1xV supply rail (typically 10 V to 19 V).

Depending on the selected configuration, the 1xV rail may also supply the gate/base drive through R11.

C5, C6

These capacitors decouple the xV supply rail (typically 3.3 V or 5 V), providing a stable supply for the signal-conditioning circuitry and other low-voltage electronics.

C8

C8 decouples the PWR-IN supply feeding the drain/collector of the output transistor.

Depending on the selected configuration, this supply may also be connected to the gate/base drive through R11.

Local Output Transistor Capacitors - C1, C2

Unlike the other decoupling capacitors, C1 and C2 are connected directly to the output transistor rather than to a dedicated power rail.

- C1 is connected to the drain/collector.
- C2 is connected to the source/emitter.

These footprints are intended for experiments requiring additional capacitive loading or filtering directly at the output transistor.

Placing decoupling capacitors close to the circuit they support minimizes the inductance of the current path, making them more effective at supplying the short bursts of current required during switching.

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

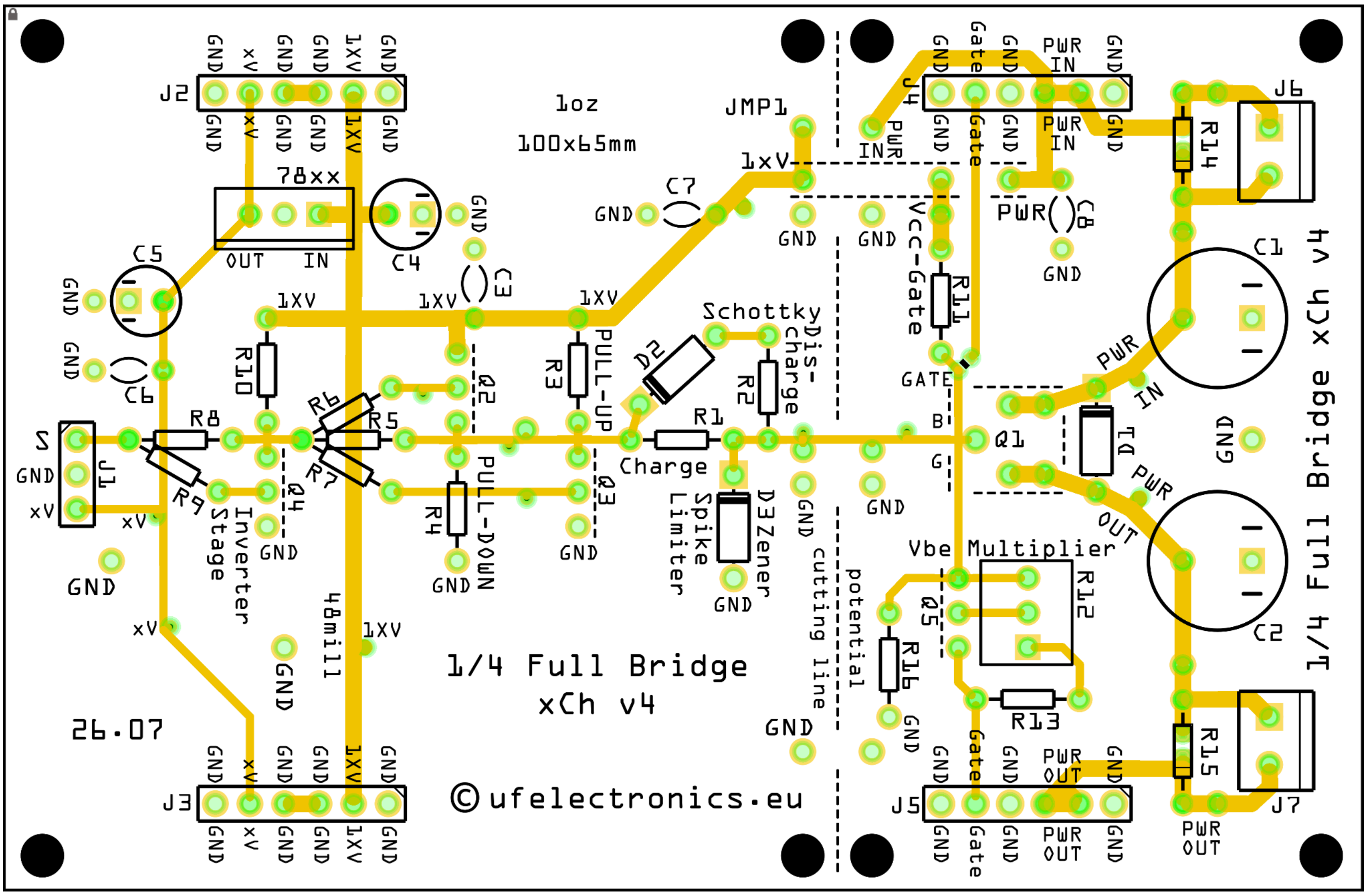
Discord: https://discord.com/channels/1373552306817339535/1373552307467325542

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

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

Semi-Schematic Layout

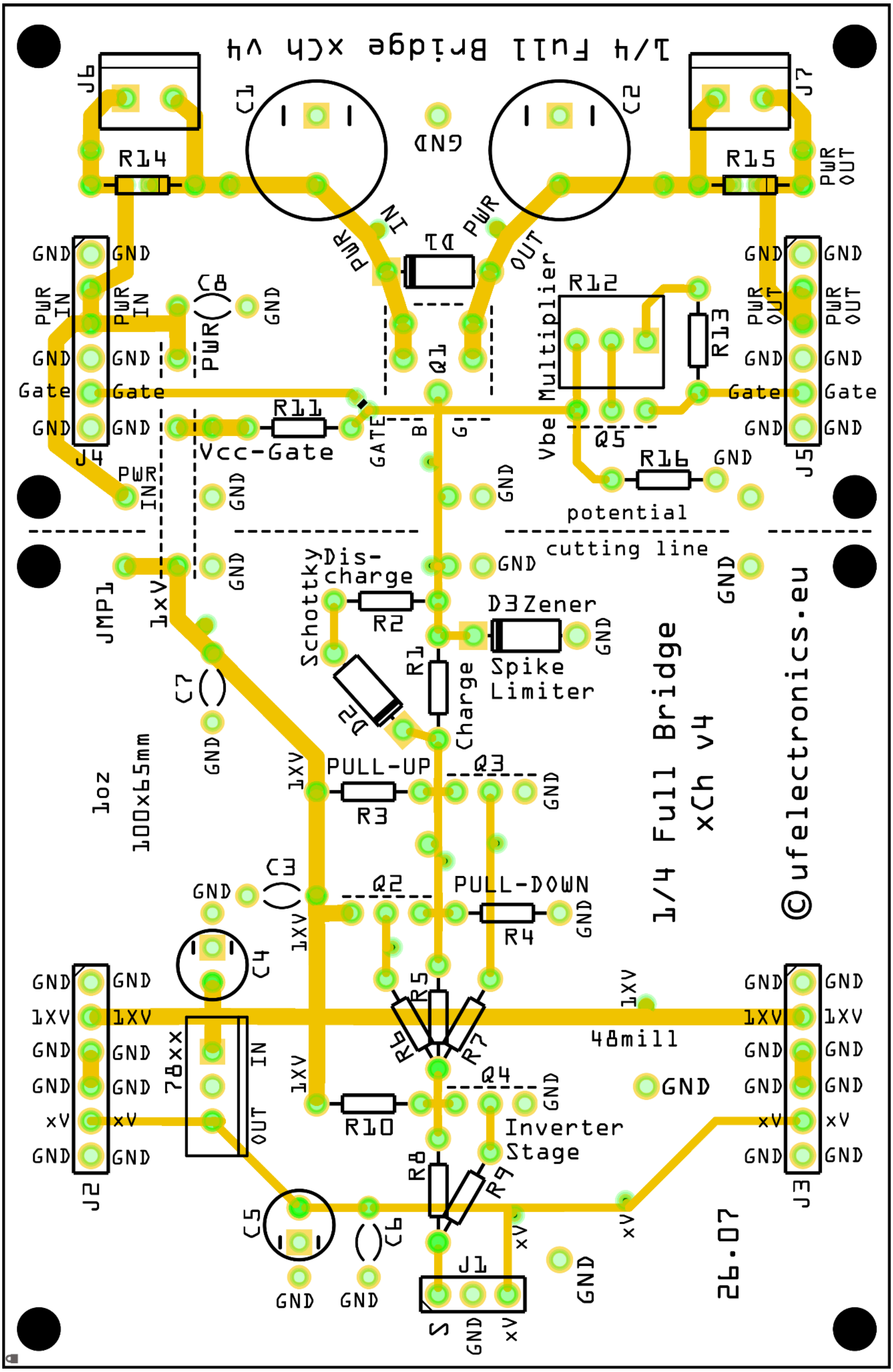
Horizontal



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.

Vertical



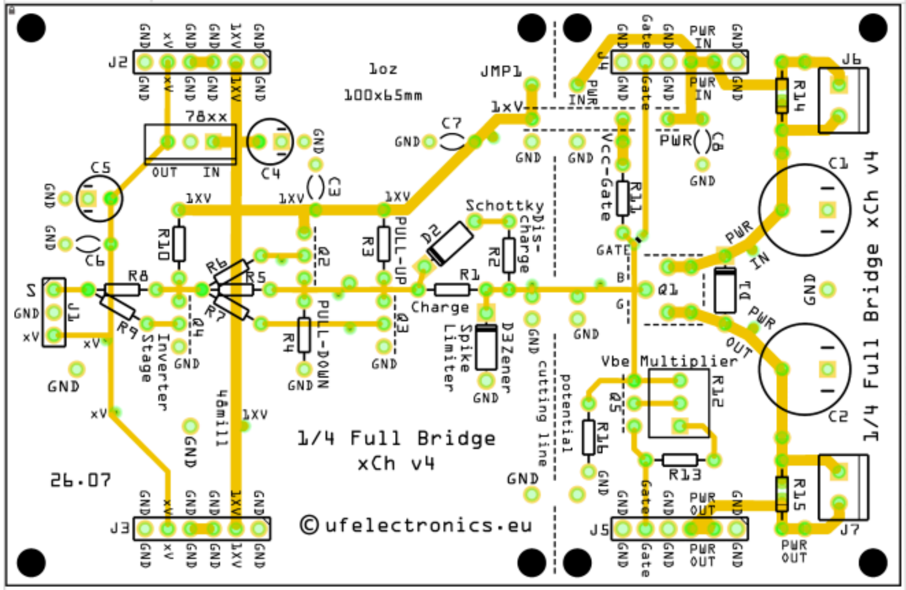
1/4 Full Bridge xCh v4
J6
J7
C1
C2
GND
R14
R15
PWR OUT
PWR IN
D1
PWR
OUT
Multiplier
R12
R13
C8
PWR
GND
Gate
J4
J5
Q1
R11
Vcc-Gate
GATE
B
G
Vbe
Q5
R16
potential
cutting line
JMP1
1xV
Schottky
Dis-charge
R2
R1
D3 Zener
Spike Limiter
Charge
D2
C7
1oz
100x65mm
1/4 Full Bridge xCh v4
© ufelectronics.eu
PULL-UP
R3
Q3
C3
Q2
PULL-DOWN
R4
C4
R5
R6
R7
1xV
48mill
78xx
IN
OUT
Q4
R10
Inverter Stage
R8
R9
J2
J3
xV
C5
C6
J1
S
26.07

Assembly Sheet

1/4 H-Bridge xCh - v4 - Assembly Sheet

Circuit .... :

Vers. / Date :



Notes Vcc:

Resistors	**Value**	**Capacitors**	**Value**
R1		C1	
R2		C2	
R3		C3	
R4		C4	
R5		C5	
R6		C6	
R7		C7	
R8		C8	
R9			
R10		Q1	
R11		Q2	
R12		Q3	
R13		Q4	
R14		Q5	
R15			
R16		78xx	
D1			
D2			
D3			