

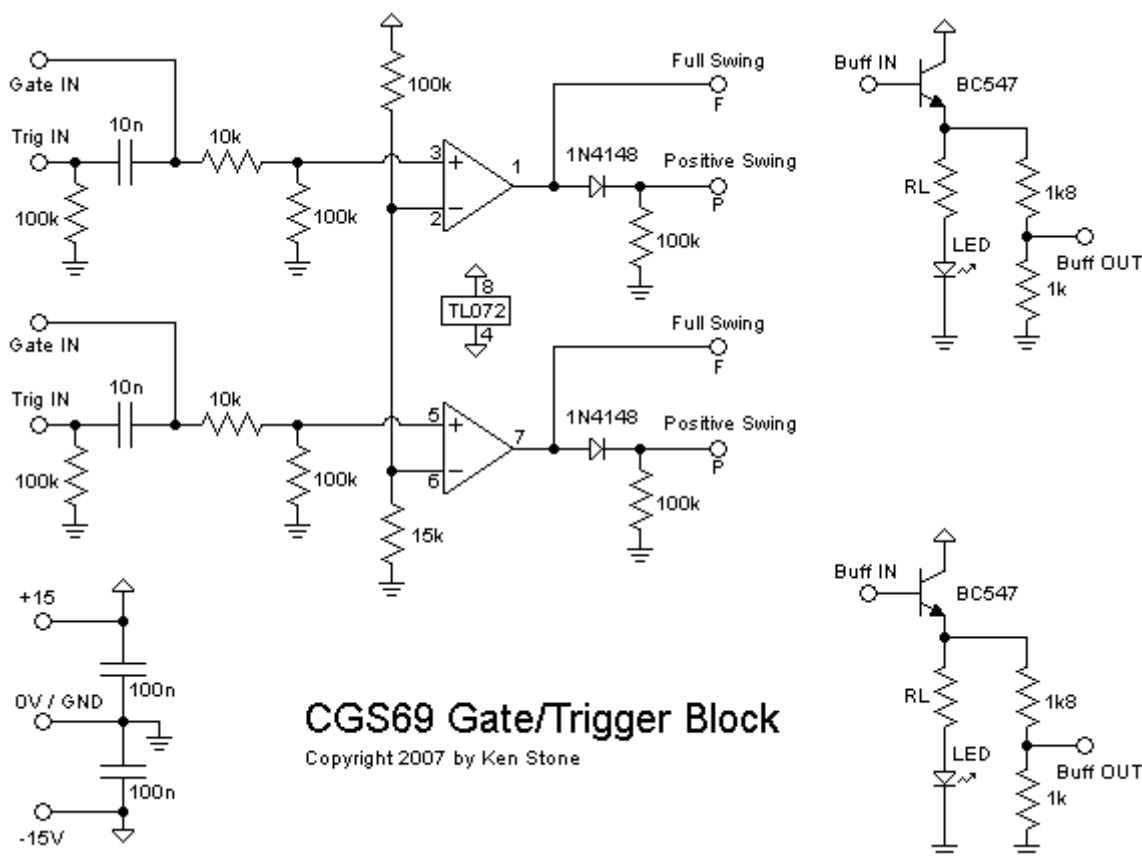
Dev-mod Gate/Trigger block

for music synthesizers.



While this "development module" does replicate the [Gate to Trigger Converter](#) to some extent, the intended use is a little different. When prototyping new designs, one frequently finds the need to build certain component blocks over and over again. The options are simple - waste time doing it again, waste a good board to use the subsections that are required, or use something like this, that can be easily salvaged after each use. Of course there is nothing to prevent it becoming a permanent part of a one-off design either.

A little on how it works:



The schematic for the Gate/Trigger Block.

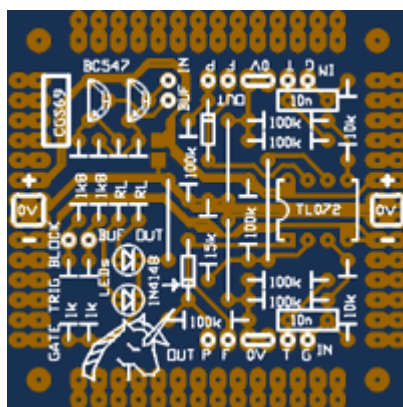
Each of the blocks is functionally identical and can be configured in several ways. First there is a comparator and its associated network designed to convert a positive going signal into a positive going gate signal (detection point is around 1.5V)). The range of this signal can be -VE to +VE (less a couple of volts) or 0V to +VE.

A couple of extra components (resistor and capacitor) allow the blocks to be used as gate to trigger converters that to correspond to positive going transitions. The exact detection point depends on rate and level of voltage change - for example a rapid change from -10V to 0V would trigger it just as readily as one that changed from 0V to 10V.

Note that it is NOT possible to use a block as both a gate detector and a trigger converter at the same time. One input OR the other must be chosen. As there are two blocks on the PCB, if you require both functions at once, use one as a gate detector and the other as a trigger converter. If you are not planning to use the trigger function, omit the 10n and the corresponding 100k pull-down resistor.

There are also two uncommitted output buffers to drive either directly from the "P" positive outputs, for stand alone use, or from the outputs of whatever logic you add between.

Construction



The component overlay for the Gate/Trigger Block .

Before you start assembly, check the board for etching faults. Look for any shorts between tracks, or open circuits due to over etching. Take this opportunity to sand the edges of the board if needed, removing any splinters or rough edges. (With the boards supplied by me, the edges are already milled, and etching faults are very rare.)

When you are happy with the printed circuit board, construction can proceed as normal, starting with the resistors first, followed by the IC socket if used, then moving onto the taller components.

Take particular care with the orientation of the polarized components such as electrolytics, diodes, transistors and ICs.

Decoupling capacitors are mounted directly on the rear of the PCB under the chip. You can use surface mount devices, or with a little lead bending, monoblock ceramics.

The LED resistors (RL) can be selected as needed. If you are using common LEDs, 1k to 2k2 would be appropriate. If you are using any of the super-bright modern LEDs for their unique colors, the resistors can be much higher, 10k to 33k for example, so that the LEDs don't dazzle you.

If you need to increase the output swing of the buffers, change the values of the 1k8/1k resistors used in the divider.

Around the edge of the board are rows of pads suitable for adding headers to allow dev-blocks to be linked together using the CGS72 joiner or plain old pin header jumpers. Apart from the power rails, these are not assigned, so as to allow maximum versatility. You will need to link these pads to the inputs and outputs of the circuit blocks as required.

Board markings

T (IN)	Signal input for using the block as a gate-to-trigger converter
G (IN)	Signal input for using the block as a gate detector
P (OUT)	Raw block output with voltage swing between 0 and approx +13 volts
F (OUT)	Raw block output with voltage swing between approx -13 and +13 volts
BUF IN	Input to the buffer. Drive with a positive signal (0V = off, +13V to +15V = on)
BUF OUT	Output from buffer. 0V = off. 4.5V to 5V output = on.

Notes:

- **PCB info:** 2" x 2" with four 3mm mounting holes 0.15" in from the edges.
- Please [email me](#) if you find any errors.

Parts list

This is a guide only. Parts needed will vary with individual constructor's needs.

If anyone is interested in buying these boards, please check the [PCBs for Sale](#) page to see if I have any in stock.

Can't find the parts? See the [parts FAQ](#) to see if I've already answered the question. Also see the [CGS Synth discussion group](#).

Part	Quantity
Capacitors	
10n	2
100n 1206smt	2
Resistors	
RL Choose to suit LED	2
1k	2
1k8	2
10k	2
15k	1
100k	7
Semi's	
3mm or 5mm LED	2
1N4148	2
BC547 or similar	2
TL072 or similar	1
Misc.	
0.1 headers	as required
CGS69 PCB	1

Article, art & design copyright 2007 by [Ken Stone](#)

[Modular Synth Home](#)

[Disclaimer](#)

