Window comparator circuit

Design Goals

Input		Output		Supply		
V_{iMin}	V_{iMax}	V_{oMin}	V_{oMax}	V_{cc}	V_{ee}	V_{ref}
0V	5V	0V	36V	5V	0V	2.5V

V_L (Lower Threshold)	V_H (Upper Threshold)	Upper to Lower Threshold Ratio
1.66V	3.33V	2

Design Description

This circuit utilizes two comparators in parallel to determine if a signal is between two reference voltages. If the signal is within the window, the output is high. If the signal level is outside of the window, the output is low. For this design, the reference voltages are generated from a single supply with voltage dividers.
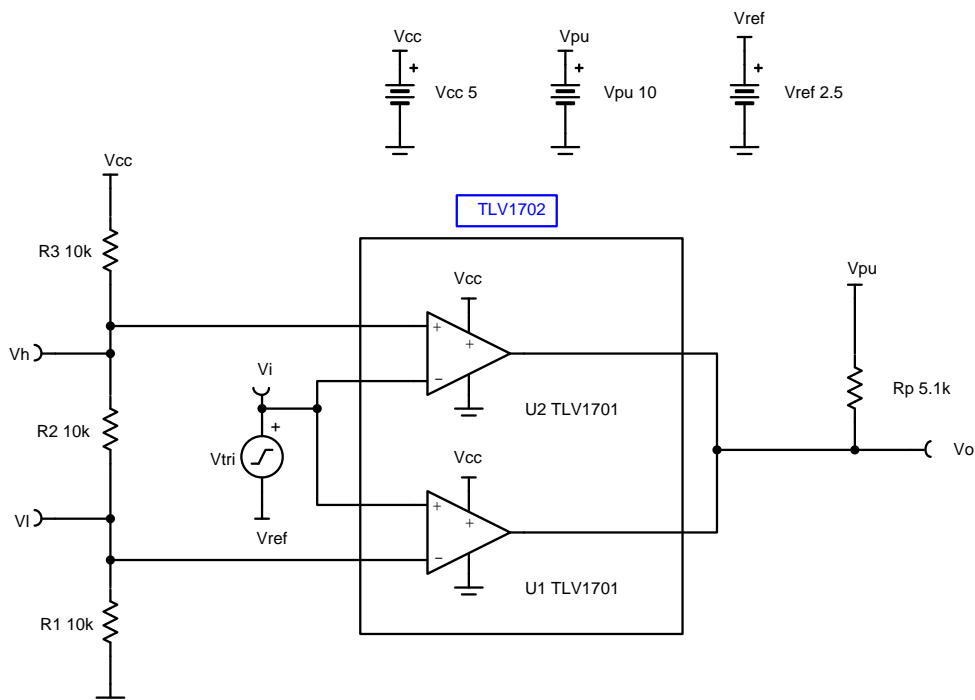
Design Notes

1. The input should not exceed the common mode limitations of the comparators.
2. If higher pullup voltages are used, R_p should be sized accordingly to prevent large current draw. The TLV1701 supports pullup voltages up to 36V.
3. Comparator must be open-drain or open-collector to allow for the ORed output.

Design Steps

1. Define the upper (V_H) and lower (V_L) window voltages.

$$V_H = V_{cc} \times \frac{R_1 + R_2}{R_1 + R_2 + R_3} = 3.33\,V$$

$$V_L = V_{cc} \times \frac{R_1}{R_1 + R_2 + R_3} = 1.66\,V$$

$$\frac{V_H}{V_L} = 1 + \frac{R_2}{R_1} = \frac{3.33V}{1.66V} = 2$$

2. Choose resistor values to achieve the desired window voltages.

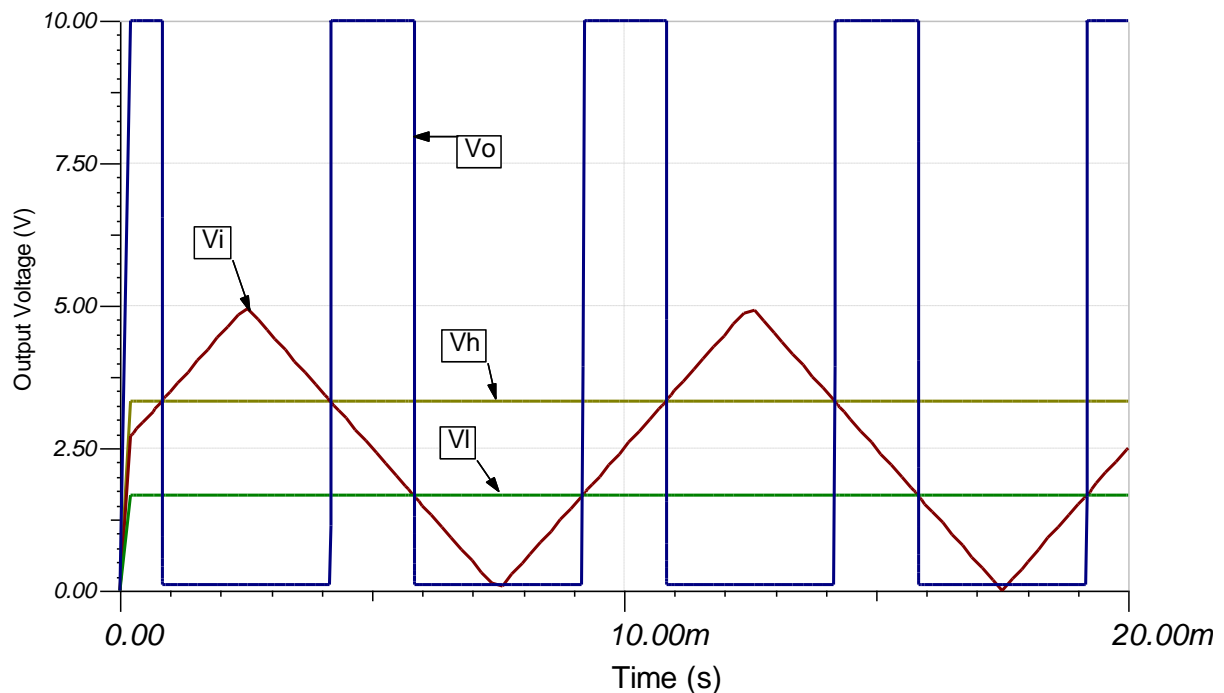$$\frac{V_H}{V_L} = 1 + \frac{R_2}{R_1} = 2,\ so\ R_2 = R_1$$

$$R_1 = R_2 = 10k\Omega\ (Selected\ standard\ values)$$

$$R_3 = \frac{R_1 \times V_{cc}}{V_L} - (R_1 + R_2)$$

$$R_3 = \frac{10k\Omega \times 5V}{1.66V} - 20k\Omega = 10.12\ k\Omega \approx 10k\Omega\ (Standard\ Value)$$

Design Simulations

Transient Simulation Results

Design References

See Analog Engineer's Circuit Cookbooks for TI's comprehensive circuit library.

See the circuit SPICE simulation file SBOC516.

See TIPD178, www.ti.com/tool/tipd178.

Design Featured Op Amp

TLV1702	
V_{cc}	2.2V to 36V
V_{inCM}	Rail-to-rail
V_{out}	Open Collector (36V Max)
V_{os}	2.5mV
I_q	75µA/Ch
I_b	15nA
Rise Time	365ns
Fall Time	240ns
#Channels	1, 2, 4
www.ti.com/product/tlv1702	

Revision History

Revision	Date	Change
A	February 2019	Downscale the title and changed title role to 'Amplifiers'. Added links to circuit cookbook landing page and SPICE simulation file.