

Experiment 8

Free-Running (Astable) Multivibrator

Introduction

The purpose of this experiment is to investigate a free-running (astable) multivibrator circuit. This circuit is a relative of the one-shot (monostable) multivibrator and the flip-flop (bistable) multivibrator.

Equipment Needed

- Normal laboratory equipment
- PN2222 npn BJTs
-

Pre-Lab

Run a 100- μ s SPICE transient simulation of Fig. 1 using the 2N2222 transistor model included with the student version of PSpice. Try the simulation using zero initial conditions, and also setting the initial condition on any one capacitor to 1 V. With certain initial conditions, the multivibrator should break into sustained oscillation.

Procedure

1. Construct the circuit of Fig. 1. Use the RLC bridge to accurately measure the values of the *specific* components you use for R2, C1, R3 and C2. (If stock runs low on the values given in Fig. 1, you may substitute other standard values, but be sure to record the actual measured values of the parts used.)
2. The multivibrator should oscillate at nearly the same frequency predicted in your SPICE simulation. Use the dc-coupled two-channel scope to observe and record the collector

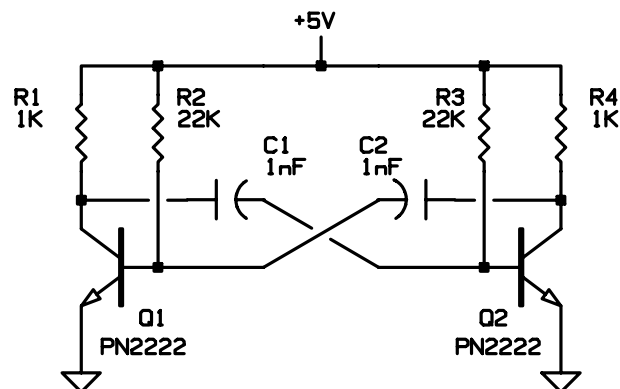


Fig. 1 Free-running (astable) multivibrator.

Experiment 8 Free-Running (Astable) Multivibrator

and base voltage waveforms on Q1.

3. Observe and record the collector and base voltage waveforms on Q2.
4. Compare these waveforms carefully with those predicted in your SPICE simulation. They should be very similar in their characteristics, and the voltage levels should agree to within about 15% at all points, assuming that the simulation was based on the same component values as those actually used.

Comments

The normal behavior of this circuit is to oscillate in a seesaw fashion, with one transistor cutoff and the other saturated, then *vice versa*. There is a theoretical possibility that the circuit could also rest at an equilibrium point at which both transistors would be saturated, and *no* oscillation would occur. (Did you observe this in the first SPICE simulation?) Upon application of the 5-V power to the actual lab circuit, there will be a startup transient, after which the circuit will either end up sitting at the equilibrium condition, or in repetitive oscillation. The first case is possible if the circuit is completely symmetric in all ways; the second occurs if there is any significant imbalance between the left- and right-hand sides of Fig. 1.

The lab instructor will assist you in deriving a timing equation, which is to be included in your report. In this derivation, you may assume that $V_{CE(SAT)} = 0.2 \text{ V}$, $V_{BE(SAT)} = 0.8 \text{ V}$, and $V_{BE(ON)} = 0.7 \text{ V}$. You may also assume that the switching time is negligible compared to an oscillation period, and that the capacitors achieve their dc equilibrium within a partial period.

Report

1. Compare the observed collector and base voltage waveforms with those predicted by the SPICE simulation. Make the comparison in terms of the observed frequency of oscillation, as well as the wave shapes and the voltages at key points in the waveforms.
2. Give the timing equation which will predict the two half-periods of the oscillation waveform. Calculate the oscillation period in terms of the measured component values and compare with your observations.
3. Discuss the factor(s) which determine whether or not the circuit will begin oscillation upon application of the power. (You might use additional SPICE simulations to explore the effects of unequal transistor betas, or unequal collector resistors, *etc.*) Is it likely that a laboratory circuit will fail to oscillate?