

Design an RC or LC oscillator to produce a sinewave with a 2 V amplitude and an oscillation frequency of 2 KHz. Simulate your design in Spice (PSpice, LTSpice, NGSpice or any other Spice version) using a Spice model for a real op amp or transistor. Show a schematic drawing of your circuit (hand drawn on engineering paper or computer drawn), your calculations for the oscillation frequency, the actual component values you used after tuning the simulation, and a plot of the voltage output (3 to 5 cycles in steady state) from your Spice simulation.