File: h:/courses/Spring2014/307/In_Lab_final_307S14.doc RWN draft of 05/04/14e

ENEE 307 in Laboratory "Final" Spring 2014

This laboratory final involves the 555 timer.
The timers available are Fairchild LM555. PSpice model files for the 555 are available in the eval and anl_misc PSice libraries. The TI data sheet is available at www.fairchildsemi.com/ds/LM/LM555.pdf . As the chips are similar, also of interest is www.ti.com/lit/ds/symlink/ne555.pdf and www.ti.com/lit/ds/symlink/lm555.pdf .
Note that to swamp parasitics the external capacitanes should be greater than 500pFd and to limit internal currents Ra and Rb should be greater than 1kOhm; also Ra+Rb<3.3MegOhm is required.

a) Individually design, simulate in PSpice, and build on your own bread board a 555 timer circuit to give a square wave pulse sequence with time high, T_h, and time low, T_l, designed for an "output waveform duty" cycle,
   $Duty=T_h/(T_h+T_l)$,
   specified according to your name as per the TA provided sheet. Do this for a repetition rate of 10 milliSeconds and an output maximum of 6V measured with respect to ground on an output resistor of 2KOhm.
b) Obtain a "pulse position modulated" output, modulated by modulating your square wave of part a) by a sinusoid of period corresponding to a racing heart beat rate of 120 beats per minute. Be sure to determine an input offset to allow modulation over the full period.
c) Test a divide by three circuit using the numerical values in the above ne555 data sheet, Figure 17 or the lm555 data sheet Figure 18.
d) Submit print-outs of your Spice (circuit and runs)[30 points] and scope displays (include voltages at the output, threshold, and control nodes)[30 points] along with your design equations & calculations[15 points], and discussion on the principles of operation of the circuits[20points] and comparisons[5points].

Your signature certifies that the work is solely your own; only signed submissions will be graded.

Note: Spice path to eval models to be used in the library portion of the Pspice menu "Edit Simulation Profile, Configuration Files, Library" :
\cadence\SPB_16.6\tools\pspice\library\eval.lib