Problem Set 5
Due Wednesday Oct. 13, 2010

Problem Set #6 will be due Monday 18, 2010

In the following problems, be sure to note the “ground rules” found at the beginning of the problem section on p. 334. Figure 2.30 is found on p. 106 and Fig. 2.35 is found on p. 112 in Gray and Meyer. Be sure to use the correct transistor model in doing the Spice simulations. In your answer, show both the circuit schematic with node numbers and the net list.

You should read sections 4.1, 4.2 and Appendix A of chapter 4.

1. Derive the expression for the output resistance of a common emitter transistor when the input is open circuited and the base is shorted to the collector.

2. Problem 4.1

3. Problem 4.5. When doing the SPICE analysis, include in the model for the MOSFETs $k'$, $t_{ox}$, $V_t$, $V_A$, $L$, and $W$. These values are obtained from your own calculations in the first part of the problem and numbers in Table 2.4 on page 135. A correlation between the device parameters used in the text and those actually used in SPICE are summarized in Appendix 4.2.1 on p. 163. In the SPICE analysis, attach a DC power supply to the output. It is through this that the output current is drawn.

4. Problem 4.9 SPICE part only. In this problem, assume $\beta = 200$, $I_s = 5 \cdot 10^{-15}$ A, and $V_A = 130$.

5. Problem 4.12 Be sure to use the values given in Figs. 2.30 and 2.35 as described at the beginning of the problems. In the SPICE part, evaluate the circuit for differential and common mode excitation and calculate CMRR by hand from the .TF results.

Exam 1 on chapters 1 – 3 on Monday Oct. 4.