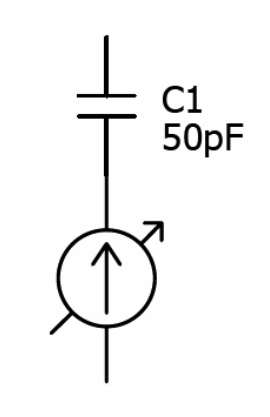
Lab 12: 5Spice and Eagle

This week's lab focuses on the use of the 5Spice v1.22 analog circuit simulator the Eagle v4.16r2 circuit layout editor.

1. Use the 5Spice circuit simulator to simulate the ideal theoretical behavior of the circuit in Lab 10 part 4a – choose a feedback capacitor which optimizes the theoretical behavior of the circuit. You will replace the photodiode with the following sub-circuit.



1a. Determine the quiescent steady-state behavior of the circuit for photocurrent of 1 μA.

1b. Determine the response of the circuit to a photocurrent with the following functional form: $I(t)=10 + cos(\omega t)$ μA.

1c. Determine the response of the circuit to a square-wave photocurrent with a minimum current of 0 μA, a maximum current of 10 μA, and a frequency of 100 kHz.

2. Repeat part 1 (a, b, c) using the circuit of Lab 10 part 4b (i.e. use an OP27).

3. Design an RC low-pass filter with a 3 dB frequency of 100 kHz. Use the Eagle 4.16r2 layout editor to produce a schematic, board, and Gerber file. The input and output of the circuit should each use a female BNC connector.