EE313 Project

A. Electronic compass showing North and South

Design and build an analog compass using a Hall-effect sensor, UGN3503. We would like to measure the Earth's magnetic field using this sensor. Earth's magnetic field is about 0.47 Gauss in Ankara. The Hall effect sensor has a typical sensitivity of 1.3 mV/Gauss. A red LED should turn on when the Hall effect sensor looks at the North. A green LED should turn on when the sensor looks at the South. The Hall effect sensor has a nominal output voltage of about half the supply voltage, between 2.75 V and 3.25 V, with a supply voltage of 6 V. Note that 6V is the maximum allowed supply voltage. Refer to the datasheet of UGN3503 for more information. The load resistance connected to UGN3503 output should be larger than 10 KΩ.

The output voltages of DC amplifiers tend to *drift*. This very low-frequency voltage variation is due to temperature changes affecting circuit parameters or aging of components. To compensate for the drift, your circuit has to have an auto-zero switch to set the voltages when the Hall-effect sensor is looking to the west or east. When the auto-zero switch is released, it should work as specified for at least three minutes. Power supply: 15 ± 1 V (it should work with any voltage in this range; fine-tuning of this voltage should not be necessary).

Use a +6V voltage regulator to feed UGN3503. But that does not save you from small variations in the output voltage.

Specifications:

1. Current consumption from +15V supply < 30 mA
2. It should operate for at least three minutes after auto-zero.
3. LEDs should turn on within ± 45° of North or South.
4. LEDs should not flicker while turning on or off (should have some hysteresis).
5. PCB size not greater than 75mm × 60mm (one-layer board with a minimum number of jumpers).

B. Low-Frequency oscillator driving a 50Ω load

Design a 1 KHz (±20%) oscillator to drive a 50 Ω load. The output of the oscillator needs to have low distortion. The single supply voltage is 18V. The output voltage should be 15V peak-to-peak. The largest harmonic content of the output voltage should be no more than −20 dB.

You may use an OPAMP-based phase shift oscillator (refer to the OPAMP oscillator document) to generate the waveform. You may use a Class-B power amplifier like the one in Lab 3 to drive the 50Ω load.

The supply current should be less than 75mA.

Specifications:

1. Current consumption from +18V supply < 80mA (while driving 50Ω load)
2. Output frequency 1 KHz (± 20%)
3. Peak-to-peak output voltage > 15V onto 50Ω load.
4. Largest harmonic component < −20dB below the fundamental
5. PCB size not greater than 75mm × 60mm (one-layer board with a minimum number of jumpers).

Available components: UGN3503, OPAMPs: LM358 (two OPAMPs), TL062 (two OPAMPs, FET input), TL084 (four OPAMPs, FET input), LM324 (four OPAMPs); 6V voltage regulator: 7806; zener diodes; BJTs: BC238, BC308, BD135, BD136, JFET: BF245 (with njf symbol in LTSpice, n-Channel, VT=−2.3V), MOSFET: 2N7000 (with nmos symbol, n-channel, VT=1.6V), red LED, green LED, standard resistors, standard capacitors, 10K multiturn trimpot, push-button switch.

Datasheets are in Moodle. LTSpice model files of most components are also in Moodle.

Design Phase (Due April 18, 2024)

Simulate your design using LTSpice to show the performance under the abovementioned conditions. (Use the spice files available in Moodle for simulation.)

Upload your LTSpice source file *.asy into Moodle. Use Diptrace to generate the schematic of your design, showing a component list.

A. Electronic compass: You can simulate a switch in LTSpice using “sw” (a voltage-controlled switch). Include a Spice directive to define the properties of the switch: [.model SW SW(Ron=0.1 Roff=1000Meg Vt=5 Vh=0)] This statement defines a threshold voltage with V_t. You need to apply a control voltage source that is larger than V_t to turn on the switch. The switch has an on resistance of Ron and an off resistance of Roff.

   You can simulate UGN3503 with a voltage source as a sinewave generator with an offset (Vcc/2). Use a 10 Hz sine wave of 0.6 mV amplitude to simulate the rotation of your circuit with respect to the earth’s magnetic field. Let the rotation start after the auto-zero switch is turned on and off. After auto-zero switch activation, the LEDs should turn on and off at the correct angles. When the temperature of the circuit varies, the circuit should still work after activating the auto-zero switch. The provided model files do not model temperature variations.

B. Oscillator: You need to simulate for a while to find the steady-state value of the oscillator’s output voltage. For an oscillation to start, the total phase shift around the loop at the desired frequency should be 0° or −360°, and the loop gain should be larger than unity. You may first perform a small-signal AC analysis under the open-loop condition to find whether these conditions are satisfied. Find the frequency where the phase shift is 0° or −360°. The oscillation will start if the gain is greater than 0dB at this frequency. If the loop gain is slightly larger than 0dB, reaching steady-state will take a long time, but the harmonic content will be low. If the loop gain is 10dB or higher, the transient time will be low, but the harmonic content will be high.

   Then, close the loop and perform a transient analysis. The oscillation will grow slowly until it reaches a steady state.

   Note that the Class-B amplifier needs to operate from a single supply. For this purpose, an AC-grounded voltage divider is needed to generate half of the supply voltage.

PCB Design Phase (Due April 28, 2024)

Before designing the PCB, test your design on a breadboard. Then, using the Diptrace, generate a PCB layout. Use a one-layer board of size 75 mm × 60 mm. Spend time on placement. The projects are so simple that a PCB with no jumpers is possible. A good placement is very important for a good layout. Upload the gerber file and report showing your PCB design to Moodle before the deadline.

Experiment and demonstration (Due May 12, 2024)

Build your design on the PCB. Be very careful while soldering the components. Since the fabricated PCB has no (green-colored) solder mask, you may easily short-circuit the nodes to each other. Using a magnifying glass, inspect the solder joints to ensure you do not have any cold solder joints or short circuits.

Test the four specifications given above. Demonstrate your work.

Youtube video (Due May 16, 2024)

Prepare a 3-minute video, demonstrating your circuit. Upload the video to Youtube. Provide the youtube link in Moodle. You can use this link as an entry in your CV.

Grading criteria:

Design Phase (10 pts)

Nice looking schematic with component list: 2pts

Satisfaction of first four criteria in LTSpice: 8pts, 2 pts each

PCB Design Phase (10 pts)

A nice looking PCB without errors and a minimum number of jumpers. All components are neatly placed on one side of the PCB.

You lose 1pt for each error. You lose 0.5pt for each jumper.

You lose 4 pts each time you order a new PCB.

Experimental and Demonstration: (10 pts)

Nice looking PCB with good solder joints: 2pts

Experimental satisfaction of the first four criteria: 8 pts, 2pts each