PCB Design Contest

Round 1

IEEE Robotics & Automation Society
BUET STUDENT BRANCH CHAPTER

Contest Rules and Submission Guidelines

Things to submit for each problem:

1. Gerber files zipped as a single file
2. Main project file
3. PDF files of each of your layers (1 layer in 1 PDF, name your file as Top copper, bottom copper, etc.). Make a subfolder and keep all your PDFs inside.
4. 3D view of your design. For this, go to the 3D visualizer of your PCB Designing software. View your PCB from different angles and record your screen. For recording, you may use Zoom or any other software as per your choice. Upload the video on YouTube. Submit the link of the video. (Note: While uploading to YouTube, ensure that the video is "Unlisted")
   [Screen recording tutorial], [Video Uploading Tutorial]

Place all your files related to a problem in a google drive folder. **Turn on access to your "Entire folder".** For this, right click on the folder, go to share -> general access -> select anyone with the link. Then copy the link and submit.

Please refer to the end of this document for design rules, judgement criteria, tips, and other information.

Problem 1: Arduino Uno and LCD Display [100 points]

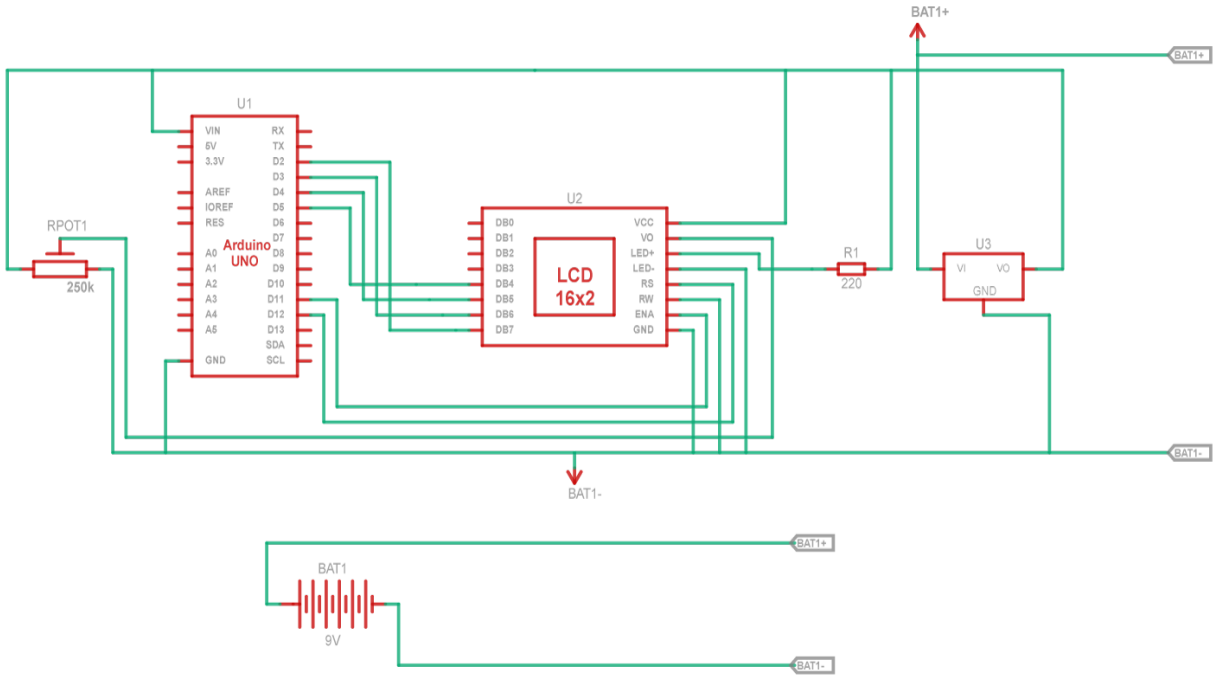


Figure 1: Schematics of a project with Arduino Uno and an LCD display

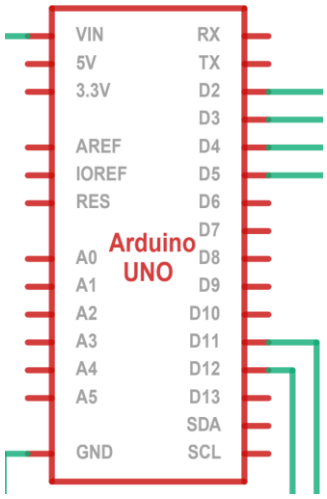


Figure 2: Arduino Uno pinout zoomed in

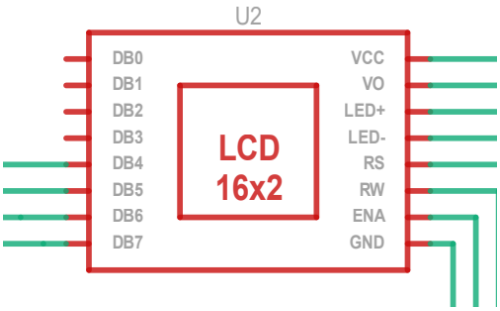


Figure 3: LCD display pinout zoomed in

You have completed this project with Arduino Uno which displays text on an LCD display. You want to use this whole setup as a part of another bigger project. But the wiring of the LCD display is too much crowded and not portable. Moreover, the connections made on the breadboard becomes loose so often and your setup stops working.

Suppose you decided to solve all these problems by bringing everything on a compact PCB. Design the PCB.

Note:

1. Your Designed PCB will be basically an Arduino Shield. To get an idea on shields, go to this link. On top of your designed shield, the other components like the LCD board, potentiometer, and the voltage regulator will be placed. You must make all the pins of the Arduino accessible through your shield.
2. You need not place the battery on the shield. Just ensure that there is a way to connect the battery +ve and -ve terminals to the shield. We recommend to use this connector on the PCB to connect battery wires to it. Remember to label the connector's -ve and +ve terminals correctly.

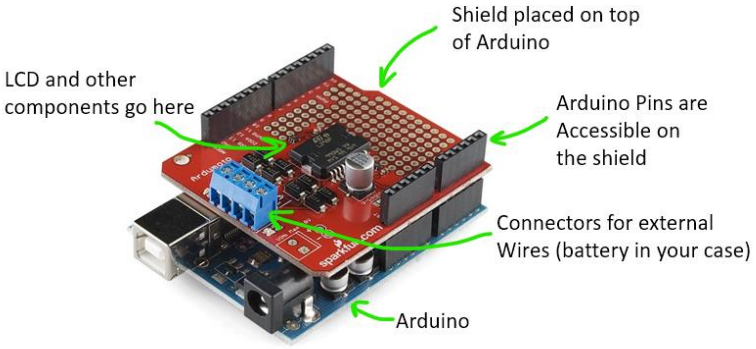


Figure 4: Reference image from Sparkfun

References:

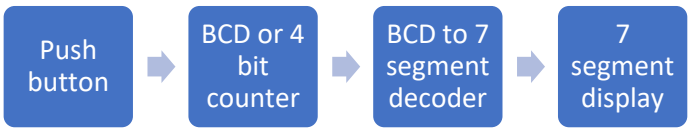
1. LCD Display 16X2 with Header Robotics Bangladesh (roboticsbd.com)
2. 10K Ohm Potentiometer Robotics Bangladesh (roboticsbd.com)
3. L7805 Voltage Regulator Robotics Bangladesh (roboticsbd.com)

4. 2 Pin Pitch 5.0mm Straight Pin Screw PCB Terminal Block Connector Robotics Bangladesh (roboticsbd.com)

Problem 2 Counter with Seven Segment Display [200 points]

Implement a seven segment display counter on a PCB board. Your system should have a minimum of one 7-segment display and a push button. When the button is pressed, the number showed in the 7 segment display increases.

There are different approaches. One of them is shown below:



Reference:

1. 7-segment Display Counter Tutorial (electronics-tutorials.ws)
2. IC 7493 4 Bit Binary Counter Robotics Bangladesh (roboticsbd.com)
3. HEF4511B BCD to 7-segment (IC 4511) Robotics Bangladesh (roboticsbd.com)

Problem 3: PCB layout Design of an USB optical mouse [500 points]

The objective of this open challenge is to design a PCB board, as shown in Figure 5, intended for a typical optical mouse. The complete circuit schematic is displayed in Figure 2, and the chip modules and associated components are listed in Table 1. Using your best judgements, standard design practices, and aesthetics, your task is to design an efficient PCB layout that can accommodate all the components and ensure the proper functioning of the mouse.

For this particular problem, “Design Rules as Specified by Taru PCB” can be relaxed, you can choose standard trace widths, and other specification to accommodate the SMD components. But use standard and consistent rules and define your design specification in the submitted report. That is, while submitting your solution, create an extra doc file and write your design specifications. Include the doc file along with other files in the Google drive folder. [Design specifications means the minimum dimensions and distances used in the design rule manager of your software].

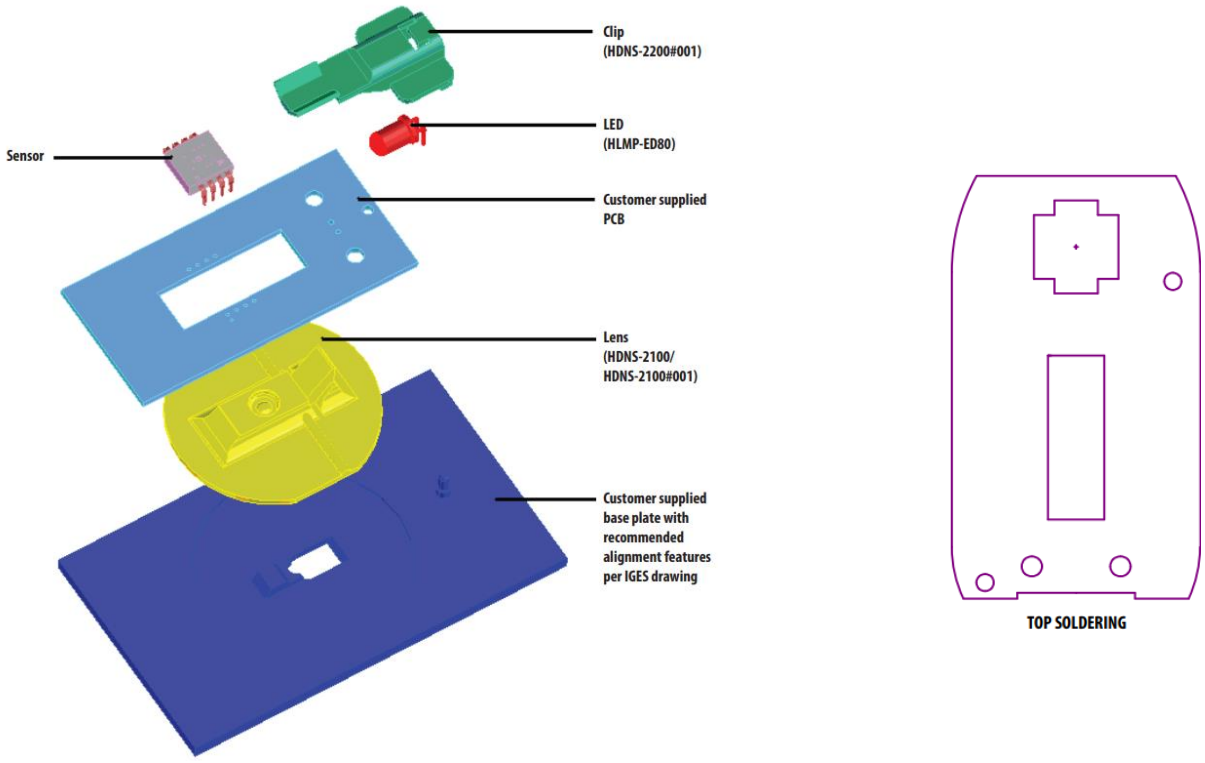


Figure 5: Conceptual view drawing of optical tracking engine with ADNS-5060 optical mouse sensor (left) and a conceptual PCB top soldering with Z-wheel and optical sensor placement slots (right)

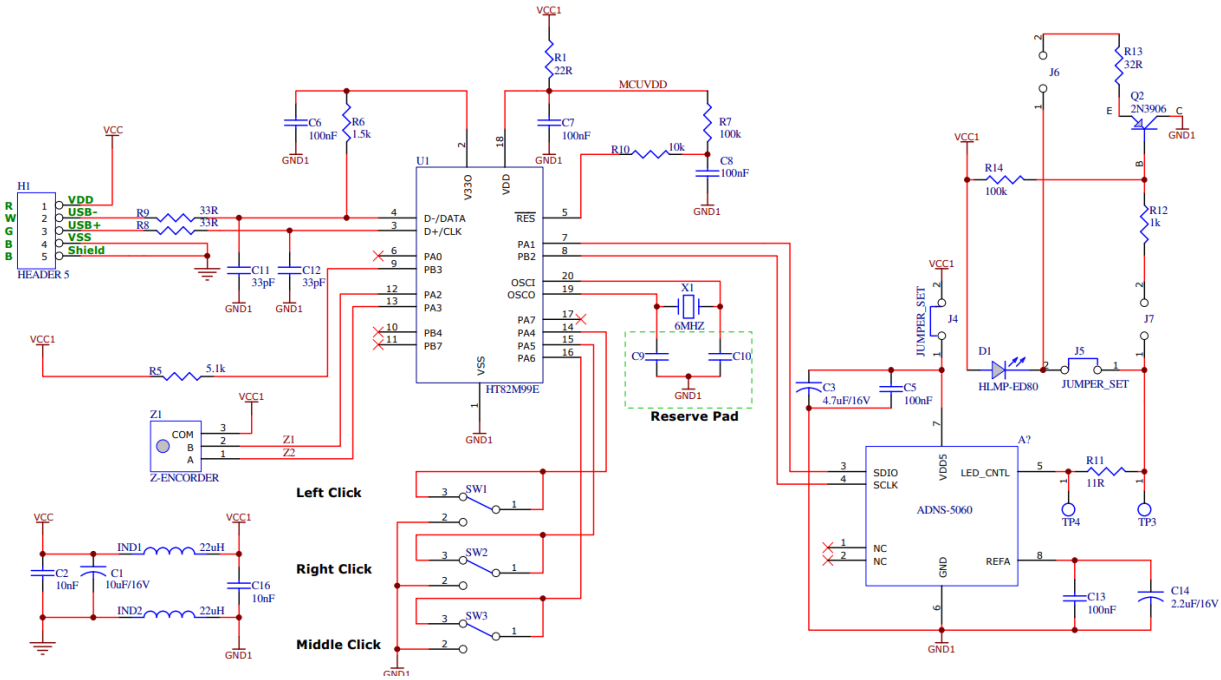


Figure 6: Schematic diagram of an optical mouse

Some of the important components in the schematic are

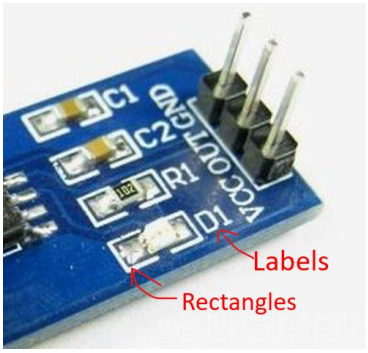
1. H1 Header 5: USB Interface port
2. HT82M99E: USB microcontroller
3. ADNS-5060: Optical sensor
4. Z-Encoder: Z-wheel movement is detected by the z-encoder
5. Left/Right/Middle Click: To detect the button press
6. Reserve Pad: To host the oscillator

No	Components	Footprint	Designator	Qty
1	Resistor 11R 1%	0603 [SMD]	R11	1
2	Resistor 22R 1%	0603 [SMD]	R1	1
3	Resistor 31R6 1%	0603 [SMD]	R13	1
4	Resistor 33R 1%	0805 [SMD]	R8.R9	2
5	Resistor 1k 1%	0603 [SMD]	R4.R12	2
6	Resistor 1k5 1%	0805 [SMD]	R6	1
7	Resistor 5k1 1%	0805 [SMD]	R5	1
8	Resistor 10k 1%	0603 [SMD]	R10	1
9	Resistor 100k 1%	0603 [SMD]	R7.R14	2
10	Inductor 22uH	0805 [SMD]	IND1.IND2	2
11	Capacitor Ceramic 33pF	0805 [SMD]	C11.C12	2
12	Capacitor Ceramic 10nF	0603 [SMD]	C2.C16	2
13	Capacitor Ceramic 100nF	0603 [SMD]	C5.C6.C7.C8.C15	5
14	Capacitor Ceramic 100nF	Radial	C13	1
15	Capacitor Electrolytic 2.2uF, 16V	Radial [6.8x4]	C14	1
16	Capacitor Tantalum 4.7uF, 10V	Case A	C3	1
17	Capacitor Tantalum 10uF, 10V	Case A	C1	1
18	IC Sensor A5060	-	U2	1
19	IC SMD C Crystal 6Mhz	SMD	X1	1
20	Transistor C PNP MMBT3906	SOT-23	Q1.Q2	2
21	LED HLMP-ED80	DIP	D1	1
22	LED Clip HDNS-2200	-		1
23	IC DIP C Holtek MCU HT82M99E	20 pin	U1	1
24	Lens HDNS-2100#001	-		1
25	Cable C USB (1 meter)	-		1
26	Switch C Micro OMRON (3 Way)	Straight	SW1.SW2.SW3	3
27	Mechanical Encoder	DIP	Z1	1

Figure 7: Bill of materials shown in the schematics

Things you must have in your designs:

1. Clearly mark the area of each of the components with rectangles of appropriate size.
2. Label each component area as well as the pins on the PCB. To label components, use the name of the component (e.g., LCD, Arduino, potentiometer, timer, etc.) To label pins, use the same name used in the main schematics or circuit diagram (e.g., VCC, GND, RS, RW, A0, A1, etc.)



Design Rules as Specified by Taru PCB:

If you order your PCB from a local manufacturer (like Taru PCB), they will ask you for the PDF files of each layer. They also have other restrictions and requirements. You must follow the following specifications while designing your PCB and exporting the outputs as PDF. For this online Round 1, we are asking you to submit your design files only. For the finals you will need to order your PCB from a manufacturer like Taru PCB. The requirements are as follows:

1. Minimum trace width 0.7 mm.
2. Separated layers in pdf format
3. Color format should be in Monochrome / Black on white
4. Pad to pad distances > 0.5mm
5. Line to plane distance > 0.5mm
6. Pad inner hole color should be white
7. Pad inner hole radius (white Circle) should be 40% smaller than actual size (e.g. You wanted 1mm hole so the inner circle should be 0.6mm)
8. For single layer PCB (Bottom Copper, Bottom Silk, Bottom Resist) should be Mirrored, Top Silk should be as it is.
9. For Double Layer PCB (Bottom Copper, Bottom Silk, Bottom Resist) Should be Mirrored and (Top Copper, Top Silk, Top Resist) should be as it is.

For any query related to minimum drill size, hole size, or other restrictions, contact Taru PCB [Asif Al Jamie - 01972120144] on WhatsApp. Please keep in mind that, they are not related to this contest or IEEE RAS BUET SBC. So, when you are contacting them, you are contacting them as an independent customer.

Judgement Criteria for this Contest

1. Board footprint/area (less is better)
2. Number of vias (less is better)
3. Cumulative length of wires/traces (less is better)
4. Component placement
5. Verification against PCB Design Rules
6. Robustness against interference, heating, power spikes, component separation
7. Aesthetics

Design tips:

1. Make your design as compact as possible. The bigger your PCB, the more it will cost you to manufacture it.
2. Use minimum number of vias. Because when you manufacture your PCB from a local manufacturer (like Taru PCB), they will not connect the top and bottom layers of the PCB with via. They will just drill a hole on the places of vias and you will need to manually solder small conductors within vias to connect the top and bottom copper layers. This is a very cumbersome task which you want to avoid as much as possible.
3. Try to design your PCB in such a way that the components are placed only on the top side and all the connections and soldering are made only on the bottom side. While designing your PCB, try to think from the soldering point-of-view. If you notice that you might need to solder on both sides of your PCB, try your best to modify your design so that you need to solder only on one side.
4. If you cannot find a 3d model of a component on the internet or in your design software's database, no need to worry. Just ensure that the footprint of the component is designed correctly. The most critical part of the footprint is the distance between pins. Make sure that the distance is correct.

Learning Materials:

1. PCB Design Workshop organized by IEEE RAS BUET SBC
2. Proteus Tutorial in Bangla
3. Custom part creation tutorial on Proteus
4. If you want to design in Altium, you are more than welcome. Using BUET's institutional mail address, you can get the official license of Altium (just like MATLAB). Google for Altium Student License and you will get all the steps to follow.
   Altium Tutorial for beginners: Fedevel Academy by Robert Feranec
   Altium also has official courses on their website (Like those of MATLAB in the MathWorks's website). You may follow them too.
5. How to export Gerber files in Proteus
6. For viewing exported Gerber files, you may use the free version of Viewmate