Organisers: **"Transilvania" University of Brasov** and
"Politehnica" University of Bucharest – Faculty of Electronics, Telecommunications and Information Technology
Center for Technological Electronics and Interconnection Techniques

With support of: Association for Promoting Electronics Technology (APTE)

1 General description of the project

The project goal is to design and generate layout and fabrication files for the PCB of an **electric bike control module**. The electric bike components are shown in figure 1.

Figure 1. Electric BIKE components.

The electric bike is propelled by a brushless DC (BLDC) motor with integrated Hall sensors. Based on the input signals provided by throttle, brake and torque sensors, an electronic control unit (**electric BIKE control electronic module**) will generate a proper PWM sequence for the BLDC motor in order to propel the bike from the battery pack or to recover braking energy back into the battery.

The **electric BIKE control module** contains a microcontroller (uC) used to acquire signals from sensors (torque, throttle and brake) via a connector (J2).

The microcontroller will calculate the necessary PWM signals required for BLDC motor rotation based on sensors signals and on Hall position signals received via J1 connector. The PWM signals will control an H bridge (via driver circuitry) and will power up the BLDC motor windings (connected to J1).

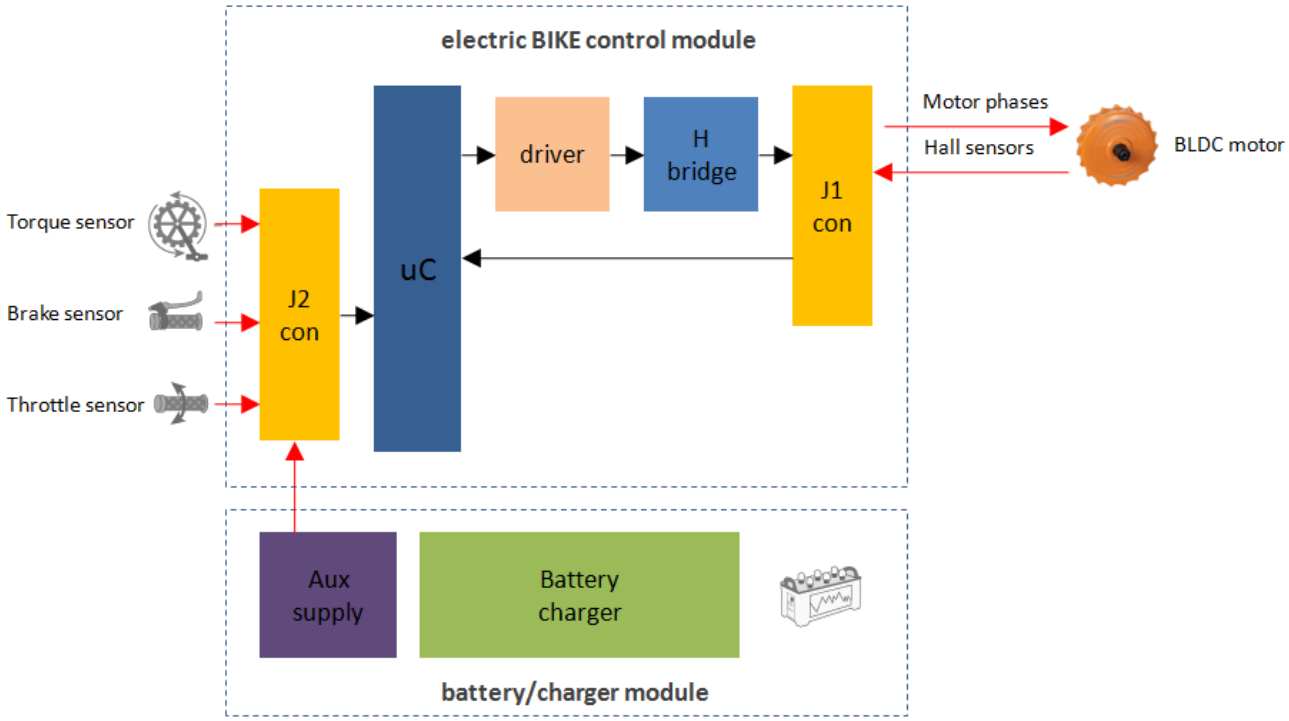

Figure 2. Block diagram of the electric bike modules.

The schematic of the electric BIKE control module is provided on page 4.

The PCB of the electric BIKE control module should have the dimensions detailed in figure 3 and should accommodate 6 non-plated holes (5mm diameter). The J1 and J2 connectors should be placed as in figure 3, equally spaced between top and bottom edges of the PCB.

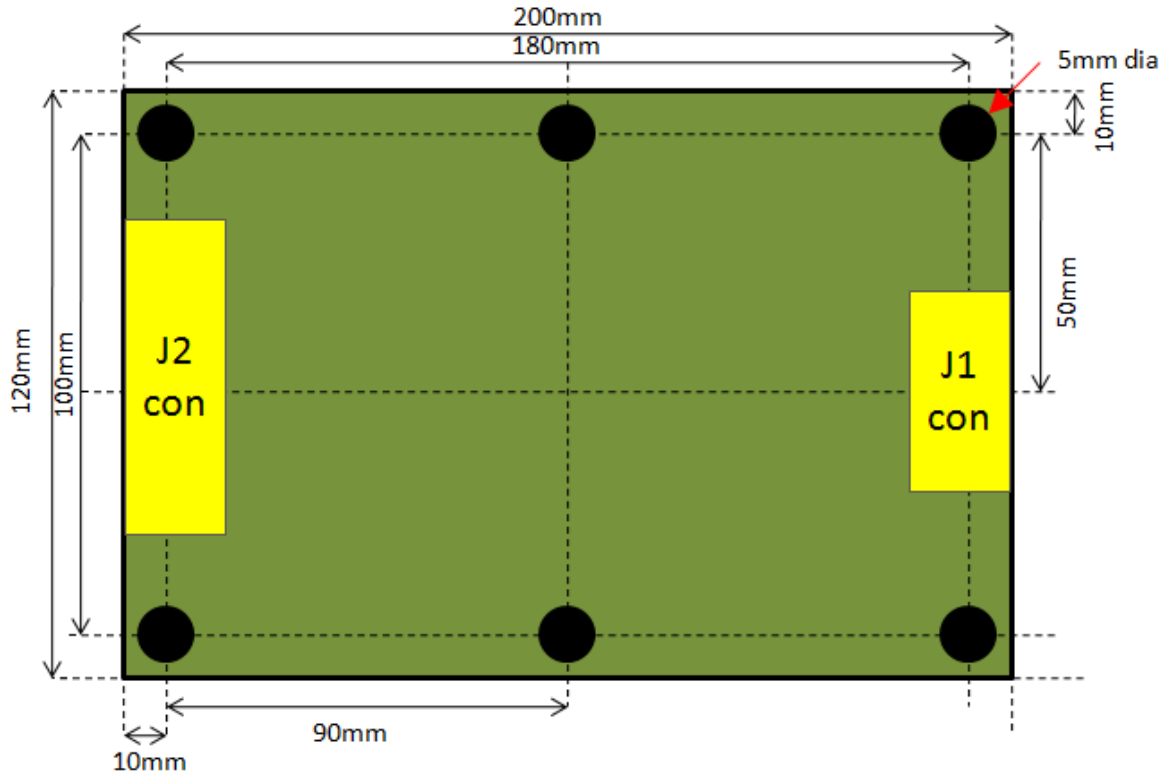

Figure 3. Mechanical drawing of the electric BIKE control module PCB.

2 General requirements

GEN-001	The design order is mandatory: libraries, schematic design, transfer procedure, layout design and post-processing activities.
GEN-002	All dimensions should be considered in metric system.
GEN-003	The electronic project (SCH and PCB) will be created using any CAD system accepted in the contest.

3 Schematic design specifications (75 points)

SCH-001	The components Z8FMC16100 and FAN7382M will be created in a new library named with the last name (surname/family name) of the contestant.
SCH-002	The schematic should be drawn in a clear manner, e.g., all references and values should have proper size and orientation, electrical connections should not be drawn over components and other un-necessary crossings should be avoided.

Notes: the schematic should be electrically correct, clean and readable. The main purpose is to generate a correct netlist for PCB design but it should also provide a clear representation of functionality.

4 Mechanical design specifications (14 points)

MEC-001	The PCB geometry is specified in figure 3.
MEC-002	The PCB should have 6 non-plate holes (5 mm diameter) for PCB fixing screws and should accommodate all 6 screws in order to be fixed firmly. Avoid placing components on a 10mm radius around center of the holes.
MEC-003	The J1, J2 connectors should be placed according to the mechanical drawings (equally spaced between top and bottom edges of the PCB) in order to fit the matching connectors.

5 Layout design specifications (150 points)

PCB-001	Components footprints and layout guidelines are specified in the attached datasheets.
PCB-002	The layout design should take into consideration a double-sided PCB with 35µm copper thickness on each

	layer. Minimum copper width is 0.2 mm and minimum copper clearance is 0.2 mm.
PCB-003	Vias connecting electrical layers will have 0.4 mm drill and 0.8 mm pad diameter (thermal vias are excluded from this rule).
PCB-004	All through-hole components will be placed on top layer and all surface-mounted-devices will be placed on bottom layer.
PCB-005	All components footprints are specified in table 1. Use provided datasheets for more information. If needed, use TO-220 package with bended terminals (Q1, Q2, Q3, Q4, Q5 and Q6) for high currents and increased electrical isolation.
PCB-006	The copper width for VBAT+, ground and phase signals of the BLDC motor should be sized for a maximum current of 10 A for each phase (max. 20^{o} C temp. rise).
PCB-007	Each MOSFET transistor (Q1, Q2, Q3, Q4, Q5 and Q6) should be mounted with a screw to a PCB heat sink area of at least $400mm^2$ on each PCB side, with at least 6 thermal vias. The RT1 thermistor should be placed as close as possible to these areas (maximum 15mm from any heat sink area).
PCB-008	The HS1, HS2 and HS3 nets should be routed with traces of the same length ($\pm$1mm tolerance).
PCB-009	The (CS-, CS+) signals should be routed as differential pair.
PCB-010	Connection between AGND and GND should be done as close as possible to pin 5 (U1) and connection between AVCC and 3V3 should be done as close as possible to pin 4 (U1) – maximum 5mm from each pin.
PCB-011	A ground area should be placed under the crystal oscillator. This ground land should be connected directly to the microcontroller ground pin.
PCB-012	The TEMP connection should be guarded by a ground trace on the opposite PCB side, keeping a minimum clearance of 0,5mm to other traces.
PCB-013	A ground plane should be placed under U1 (microcontroller) and decoupling capacitors.
PCB-014	Non-polarized decoupling capacitors should be placed as close as possible (maximum 3mm trace length) to the related pin and should be connected to the ground plane through at least one via.

6 Test specifications (14 points)

TST-001	Test pads (having proper size and shape according to IPC recommendations) should be placed on bottom layer, on a 2.54 mm spacing grid, for the following signals: AH, AL, BH, BL, CH, CL, GATE_AH, GATE_AL, GATE_BH, GATE_BL, GATE_CH, GATE_CL, CS-, CS+, VBAT+ and TEMP.
TST-002	Global fiducial markers, having circular shape, must be introduced in a proper number, according to IPC recommendations.
TST-003	Local fiducial markers will be placed for components U1, U2, U3 and U4, according to IPC recommendations.

7 Fabrication specifications (9 points)

FAB-001	The necessary PCB fabrication files (in extended Gerber format) should be provided.
FAB-002	Distinct files for non-plated and plated holes should be provided.
FAB-003	A pick-and-place file for all SMT components should be generated.
FAB-004	A list of testpoint co-ordinates should be created, as a text file.

Total: 260 points

Item	Quantity	Reference	Part	Footprint
1	11	C1,C4,C6,C9,C10,C12,C13,C16,C17,C19,C21	100nF	SMC0402
2	9	C2,C3,C7,C8,C11,C14,C15,C18,C20	10uF	B (EIA 3528)
3	1	C5	1000uF/63V	CYL 16x25 (DxL)
4	5	C22,C23,C24,C25,C26	100pF	SMC0402
5	3	D1,D2,D3	BAV19	DO-35
6	1	J1	CON8	J1_con
7	1	J2	CON15	Header 0.1" 1x15
8	1	L1	600E	SML1206
9	6	Q1,Q2,Q3,Q4,Q5,Q6	FQP85N06	TO-220
10	1	RT1	NTC10K	SMR0402
11	6	R1,R2,R6,R8,R13,R14	22R	SMR0402
12	6	R3,R4,R5,R9,R10,R12	10K	SMR0402
13	1	R7	0R005	SMR1825
14	1	R11	5K1	SMR0402
15	3	R15,R16,R17	4K7	SMR0402
16	1	R20	0R	SMR0402
17	1	R19	100K	SMR0402
18	1	U1	Z8FMC16100	QFN32
19	3	U2,U3,U4	FAN7382M	SOP8
20	1	Y1	20MHz	SMD Y1-XTAL

Table 1. BOM for electric BIKE control module.