



Organisers: "Lucian Blaga" University of Sibiu
"Politehnica" University of Bucharest - Faculty of Electronics, Telecommunications and Information Technology
Center for Technological Electronics and Interconnection Techniques

1 General description of the project

The project goal is to design and generate layout and fabrication files for the PCB panel of a temperature control unit, as shown in figure 1.



Figure 1. Temperature control unit (top view of the front panel).

The temperature control unit has three (3) parts:

1. Front panel: plastic housing with cutouts for LCD and 4 push-buttons;
2. **PCB panel** – electronic system of temperature controller – **this is the part that should be designed**;
3. Back panel: Aluminum housing.

The **block diagram** of the temperature controller for which the PCB panel should be designed is shown in figure 2. The complete schematic diagram is presented in page 4.

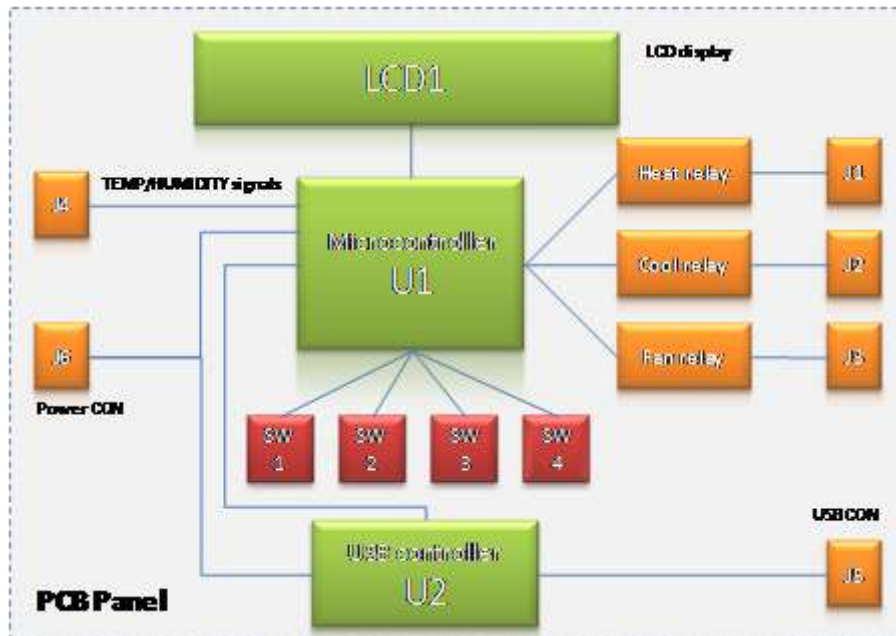


Figure 2. Block diagram of the temperature controller.

The **PCB panel** of the temperature controller consists in the following blocks:

- Microcontroller (U1) – controls 3 output relays based on temperature/humidity signals;
- USB controller (U2) – interfaces the temperature controller with a PC;
- Relays: three (3) relays for controlling heater, cooling unit and fan;
- Connectors: for wiring inputs and outputs;
- LCD: for displaying controller data;
- Push-buttons: four (4) push-buttons for user interface.

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.

3 Schematic design specifications (80 points)

SCH-001	The schematic project will be created using any CAD system accepted in the contest.
SCH-002	The components U1, U2 and LCD1 will be created in a new library named with the last name (surname/family name) of the contestant.
SCH-003	The schematic should be drawn in a clear manner, e.g.: all references and values should have proper size and orientation, un-necessary crossings should be avoided.

Note: the schematic should be electrically correct, clean and readable. The purpose of it is to generate a correct netlist for PCB design.

4 Mechanical design specifications (14 points)

MEC-001	Back panel (Aluminum housing) geometry is specified in figure 3. The PCB panel should fit the back panel.
MEC-002	The back panel has 4 tapped holes (3.2 mm diameter) for PCB fixing screws. The PCB panel should accommodate all 4 screw holes (non-plated) in order to be fixed firmly. Avoid placing components or copper on a 7mm radius around holes.
MEC-003	The LCD and push-buttons should be placed accordingly with mechanical drawings, in order to fit the cutouts on the front panel.

5 Layout design specifications (168 points)

PCB-001	Components footprints and layout guidelines are specified in the attached datasheets. Accepted tolerance is ± 0.1 mm, except for pitch values where no tolerance is allowed.
PCB-002	The layout design should take into consideration a double-sided PCB with 35 μ m copper thickness on each layer. Minimum copper width is 0.2 mm and minimum clearance is 0.2 mm.
PCB-003	Vias connecting electrical layers will have 0.3 mm drill and 0.7 mm pad diameter.
PCB-004	Placement should follow the functional blocks placement guidelines given in figure 2. Through-hole components (THDs) should be placed on top side and Surface Mounted components (SMDs) on the bottom side.
PCB-005	All components footprints are specified in table 1. Use provided datasheets for more information.
PCB-006	The trace width for each relay output should be sized for a maximum current of 1.5 A, with no vias. Allowable trace overheating is 10 °C above the ambient temperature.
PCB-007	USB signals (USB _{D+} and USB _{D-}) should be routed as a differential pair.
PCB-008	USB signals (USB1... USB8) should be routed with matching length, on the same layer.
PCB-009	All components are assembled using the wave soldering process. Solder thief pads should be placed for component U2, considering that the solder wave will travel from pin 1 to pin 10 (pin 20 to pin 11, respectively). All components should have proper orientation considering the solder wave direction.
PCB-010	Copper should have a clearance of minimum 1mm to board outline. All components should have a clearance of minimum 2mm to board outline.

6 Test specifications (18 points)

TST-001	Test pads (having proper size and shape, according to IPC recommendations) should be placed on a 2.54 mm spacing grid, on the bottom layer.
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 and U2.

7 Fabrication specifications (14 points)

FAB-001	The fabrication files (in extended Gerber format) should be provided (top, bottom, soldermask, silkscreen, drill drawing).
FAB-002	Distinct files for non-plated and plated holes should be provided.
FAB-003	Pick-and-place file for all SMDs must be generated.
FAB-004	A list of testpoint coordinates should be created, as a text file.

Item	Quantity	Reference	Part	Package
1	3	C1,C5,C11	10uF	D (7343)
2	2	C2,C6	100nF	C 0402
3	2	C3,C4	22pF	C 0402
4	4	C7,C8,C9,C10,C12	0.1uF	C 0402
5	6	D1,D2,D3,D4,D5,D6	CMDSH23	SOD-323
6	6	J1,J2,J3,J4,J5,J6	CON4	Header 0.1" 2x2
7	1	LCD1	GD5360P	GD5360P
8	3	L1, L2, L3	RELAY	DS (1 form C)
9	2	L4,L5	MMZ2012	L 0805
10	6	Q1,Q2,Q3,Q4,Q5,Q6	BSS123	SOT-23
11	1	R1	1M	R 0402
12	2	R2,R3	33R	R 0402
13	4	SW1,SW2,SW3,SW4	PUSHBUTTON	ESE 20C
14	1	U1	MC9S08LL16	LQFP-48
15	1	U2	MC9S08JS16	SOIC 20
16	1	Y1	32.76810E	TC-38
17	1	Y2	12 MHz	ACHL (8 DIP)

Table 1. BOM of the temperature controller unit.

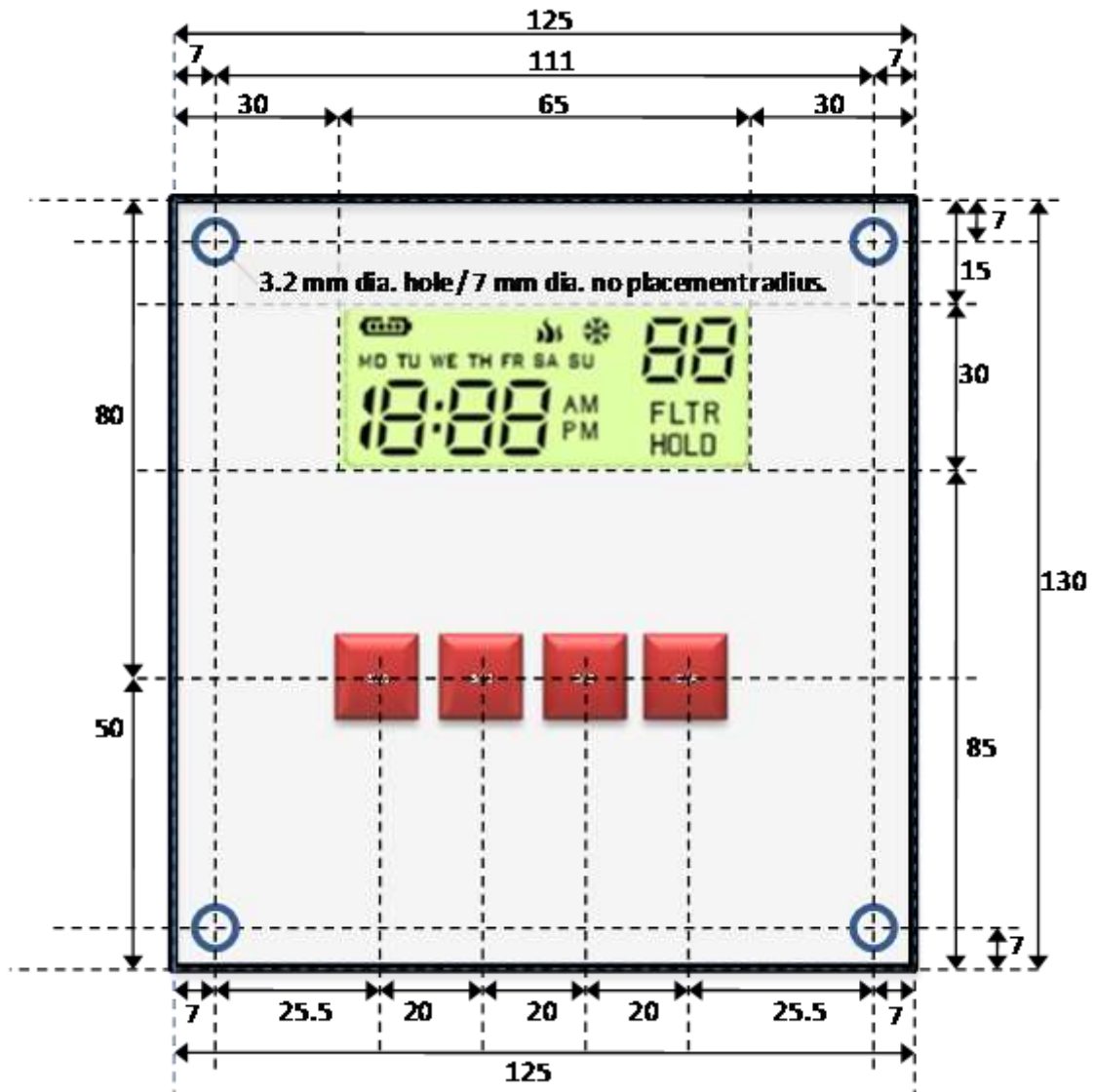


Figure 3. Mechanical drawing for back Aluminum housing (internal). All dimensions are in [mm].