TIE
DESIGN OF ELECTRONIC MODULES & ASSEMBLIES
www.tie.ro
A WAY to turn your HOBBY into PROFESSION

Student professional contest:
The 26th Edition, Iasi, 25 - 28rd April 2017
Organizers:
"Gheorghe Asachi" Technical University of Iasi
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 purpose of this project is to design one **thermal monitoring system** which will be placed near the steam generator chamber in a nuclear power plant. The monitoring system is comprised from one infrared camera sensor, one data processing unit and one Wi-Fi module to stream the data outside the containment structure. The schematic of the proposed project is indicated in **Annex 1**.

The system consists of the following:

- One FLIR Lepton® sensor for thermal imaging acquisition. The sensor will *not be soldered directly on the PCB*, it will be placed in one **socket**.(See datasheet)
- The chosen data processing unit will be **Raspberry PI Zero**, which will acquire all the data from the video sensor and will manage all the wireless communication.
- One Wi-Fi module which will handle all the communication with the exterior.
  This module will be placed on the *opposite side* of the Raspberry PI.

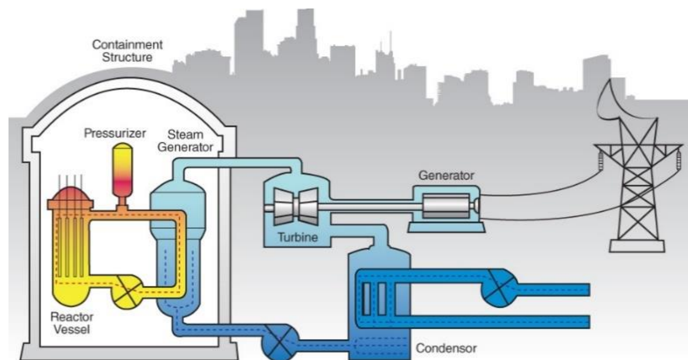


Figure 1. Power plant system

2. General requirements:

GEN-001	The design order is mandatory: libraries, schematic design, transfer procedure, layout design and postprocessing activities.
GEN-002	All dimensions shall be considered in mm.

Due to the complexity of this monitoring unit, and the lack of space available, the PCB must be built using ***rigid-flex technology***. It will be comprised of ***two rigid areas,*** one containing the Raspberry PI unit and the Wi-Fi module and the other containing the thermal sensor. Everything will be placed in one plastic housing:

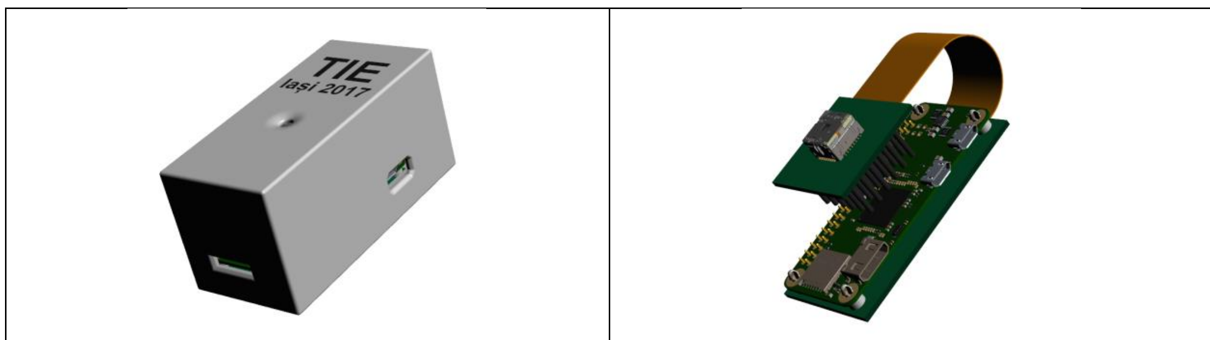


Figure 2. a). Mechanical housing of the module. b).How the PCB is slided into the housing.

! All the relevant mechanical and placement keepin/out's can be found in the mechanical annexes, in the contest documentation folders!

For simplicity we will split the PCB into three areas, as follows:

- ➔ **Area A :** contains the Raspberry Pi Zero, Wi-Fi module and other components;
- ➔ **Area B :** contains the flexible part**;**
- ➔ **Area C :** contains the thermal imaging sensor and adjacent components;

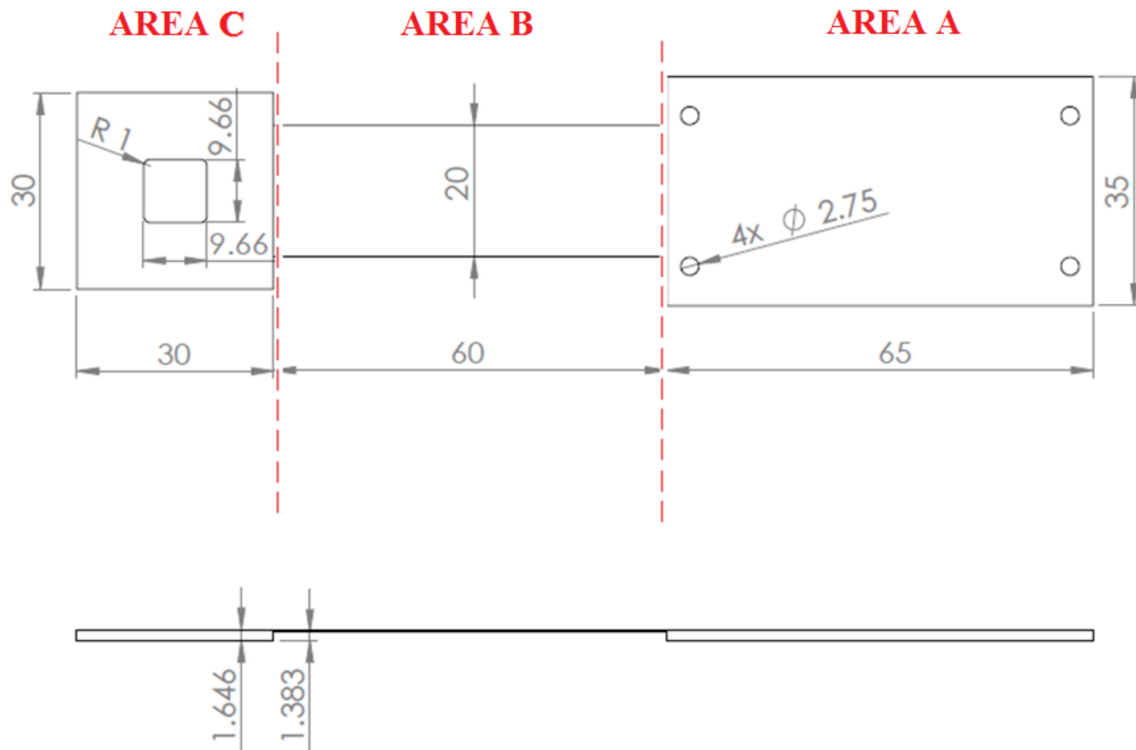
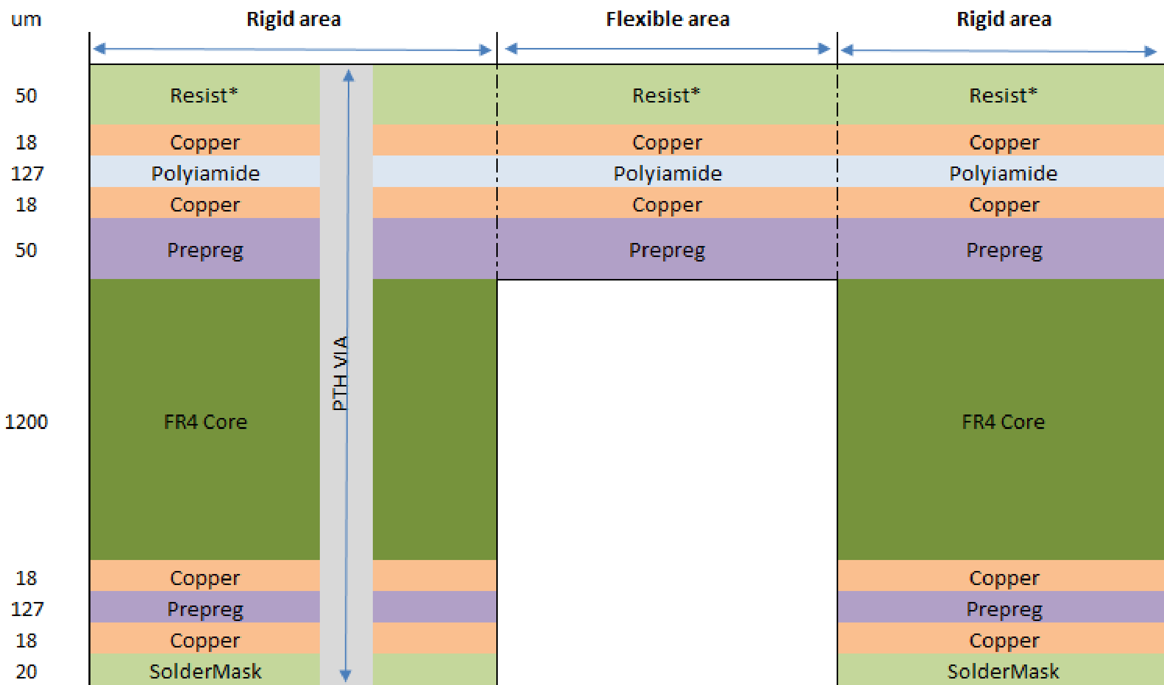


Figure 3. PCB outline mechanical informations and region splitting.

3. Schematic design specifications:

SCH-001	The schematic project will be created using any CAD system accepted in the contest (and respects all the minimum requirements published on the TIE official website).
SCH-002	The required components will be created in a new library named TIE2017.
SCH-003	The schematic must be drawn in a clear manner, e.g.: all references and values must have proper size and orientation, un-necessary crossings shall be avoided.
SCH-004	The schematic must be electrically correct, clean and readable. All reference designators must **strictly** follow **Annex 1**. The main purpose is to generate a correct netlist for PCB design but it must also provide a clear representation of functionality.
SCH-005	Test pads must be placed on the following nets (1 testpad per net): ***WAKE, CHIP_ENABLE, RESET, IQRN, PWR_DWN, WIFI_SDA, WIFI_SCL, SDA, SCL.***
SCH-006	Following completion of the schematic, a **Bill of Material (BOM)** must be generated, containing at least designators, values, quantity and footprint name.

4. Layout design specifications:



Resist* = Coverlay + Coverlay Adhesive

TOTAL RIGID PCB (um)	TOTAL FLEXIBLE FOIL (um)
1646	263

Figure 3. Proposed stack-up definition for the thermal monitoring system.

PCB-001	The PCB layout design will take into consideration the proposed stack-up from ***Figure 3***. Each participant should define in their individual CAD software ***only the rigid stack-up.*** Minimum copper width is 0.150mm and minimum clearance is 0.150mm. Accepted mechanical tolerance is +/-0.1 mm.
PCB-002	Component descriptions including standard footprints are indicated directly in Table1. Nonstandard footprints (for components with the manufacturer part number indicated in Table1, column3) must be determined from the provided datasheets. Accepted tolerance is +/-0.1 mm, except for pitch values where no tolerance is allowed.
PCB-003	All vias will have the following properties: 0.250mm hole diameter, 0.5mm soldermask opening and 0.650mm copper pads. The minimum distance, center-to-center between vias is 0.8mm.
PCB-004	In order to have the desired functionalities, all main components must be placed as indicated in the mechanical documentation files.
PCB-005	Minimum distance between two adjacent components is 0.5mm edge to edge.
PCB-006	The Raspberry PI Zero will be attached to the A area of the PCB using four FIX-MADA-3 plastic spacers(*see datasheet*). Use ***plated through holes*** to ease the plastic clamp's insertion. Minimum distance between any component and the spacers is 1mm.
PCB-007	The Wi-Fi module must be placed in such a way, that it's embedded antenna will be near a PCB's edge. Routing is ***strictly forbidden*** on any layer underneath the antenna. In case of copper plane usage on any layer under the antenna, one must provide a cut-out area with a minimum size of ***20*6mm*** symmetrically placed under the radiating element(***on all layers***).

PCB-008	The socket of FLIR sensor shall be placed in the middle of Area C.
PCB-009	Due to the heat sensitivity of area C, think carefully the placement of the two voltage regulators. Keep in mind also the fact that the camera sensor will need approximately ***330mA*** of constant current on the ***1V2 power line*** and ***30mA*** on constant current on ***2V8 power line***. In consequence, the participant shall determine the junction temperature (T_j) for both voltage regulators in order to determine whether or not a cooling area (***min. 1cm²***) is necessary. $T_{ambient}$=50°C and $R_{\theta JA}$=63°C/W.
PCB-010	If the voltage regulators are placed on area A, route the 1V2 and 3V3 in such a manner (trace width) that the DC voltage drop from one voltage regulator to the image sensor will be under 5% of the voltage's value on the respective track. (for e.g. on the 1V2 line, the maximum admissible voltage drop must not exceed 60mV). For copper, **ρ (Ω·m) at 20 °C is 1.68×10^{-8}.**
PCB-011	Due to flexfoil reliability concerns, all the traces must be ***straighten out at least 1mm before entering the flexible area***. If this rule is not respected, the flexfoil may break due to excessive bendings.
PCB-012	Given the fact that, areas A and C of the PCB will be slided in the mechanical housing please provide the PCB edges with minimum ***2.5mm*** of ground areas (not covered in soldermask) and place as many vias as possbile.
PCB-013	***MIPI_DATA_P/N*** and ***MIPI_CLK_P/N*** are differential pairs. Define them accordingly (as differential pairs) both in the schematic editor and layout editor. They will be routed in microstrip topology (Layer 1), at least in the flexible area, having the following properties: ***150um track width***, in-pair spacing ***160µm and 200µm*** differential pair to any other signal. The referece should be VDD or GND. The maximum admissible length differences between the MIPI_DATA_P/N pair and MIPI_CLK_P/N pair is 2.5mm.
PCB-014	Decoupling capacitors shall be placed as close as possible (max. 3mm) to the related pin and each shall be connected to the corresponding ground plane with two vias.
PCB-015	The C area of this PCB is the most thermal sensitive one. Due to this consideration, one heatsink must be placed under the FLIR sensor's socket, to prevent false data acquistions due to thermal fluctuations. ***Remove all the soldermask*** on the opposite side of the sensor (Area C) and assure that there are no components placed on that side.
PCB-016	A 5mm x 5mm copper area (separated from all nets) covered by solder mask, shall be placed on the PCB (for data matrix code).

5. Test specifications:

TST-001	Test pads must be ***1mm (1.1mm soldermask opening)*** in diameter and they must all be accessible for the needles of an In-Circuit Test system (ICT) (minimum distance between test pad centers must be ***1.27mm***)
TST-002	Global fiducial markers, having circular shape, must be introduced in a proper number, according to IPC recommendations.

6. Fabrication specifications:

FAB-001	In order to complete the manufacturing documentation, define one „documentation" layer, which will contain all the PCB dimensions and clear indications of which regions(A,B,C) are rigid or flexible. This is neccesary for the manufacturer to clearly understand our needs.
FAB-002	The necessary fabrication files (in extended Gerber format) must be provided.
FAB-003	Distinct drill file for holes must be provided.
FAB-004	Pick-and-place file for all SMT components must be generated.
FAB-005	A list of testpoint co-ordinates must be created, as a text file.
FAB-006	In order to manufacture this PCB, Annex2 file must have all the necessary information filled in.

Total: 300 points.

7. Bill of materials:

Table1:

C10, C12	10n	0603
C11, C13	1u	0805
C14, C16, C18	1n	0402
C15, C17, C19	100n	0805
C20	100n	0402
IC10	VREG 2V8	SOT 223
IC11	VREG 1V2	SOT 223
IC20	FLIR SENSOR	see datasheet
IC30	WI-FI MODULE	see datasheet
Q20	ASEMB-25.000MHZ-XY-T	see datasheet
R10, R11, R12, R13	100	0402
R14, R15, R32	10k	0603
R16, R17, R30, R31	1k	0603
R34	10k	0402
X10	PI ZERO	see datasheet