1. General description of the project:

The purpose of this project is to design one **wearable, biometric monitoring system** which will be used in blockchain-type wallet authentification process. The monitoring system is comprised from several sensors placed in pre-determined areas of the PCB, one processing unit(microcontroller), one bluetooth module and a wireless charging coil.

The schematic of the proposed project is indicated in **ANNEX_1**.

The system description is as following:

In order to function properly, our system needs to examine periodically, unique biometrical data in a fixed pre-determined time frame, in order to generate one individual and specific key string, to make the authentification process valid. To make this happen, the following were used:

- ➔ One **pulse oximeter sensor**, responsible for a noninvasive pulse and arterial oxygen saturation measurement. This sensor will be activated once every 1h or when the accelerometer will detect prolonged user physical activity. **(See datasheet for technical specifications of DCM03).**
- ➔ In order to properly detect and mediate quantitatively the physical activity of the user, one **accelerometer sensor** is needed.
  The placement of this sensor is limited only by the volume given by the casing and it's a matter of designer's personal choice. **(See datasheet for technical specifications TIE2018_DIG_MIC_GYRO).**
- ➔ One **digital microphone**, which sole purpose is to capture the unique characteristics of user's voice. This component should be placed as close as possible to the processing unit. **(See datasheet for technical specifications** TIE2018_DIG_MIC_GYRO**). Both accelerometer sensor and digital microphone are part of the same component.**
- ➔ Because this PCB will be fitted in a water-proof enclosure, the gadget will communicate with the exterior through a **bluetooth module (SoC - system on chip)**. The participant should carefully think and place this module because it has one embedded antenna, which will add more specific design constraints. **(See datasheet for technical specifications TIE2018_SoC).**
- ➔ All the sensor data acquistions and communication will be handled by one **microcontroller**, which will be placed in the central rigid PCB area. **(See datasheet for technical specifications TIE_2018_WPC_AFE).**
- ➔ The whole system will be powered by one **Lithium-Ion polymer based rechargeable battery**, which will be connected by wires on the PCB. This battery will have temperature monitoring to allow fast charging capabilites.

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 unit must be designed using one ***rigid-flex PCB***. It is comprised of ***two rigid sides***, one containing the bluetooth module and some of the sensors and and the other containing the processing unit and adjacent components.

The two buttons, the NTC, the pulse oximeter and the charging coil will be placed on flexible areas. Everything will be placed in an enclosed, rubber-made, water-tight housing:

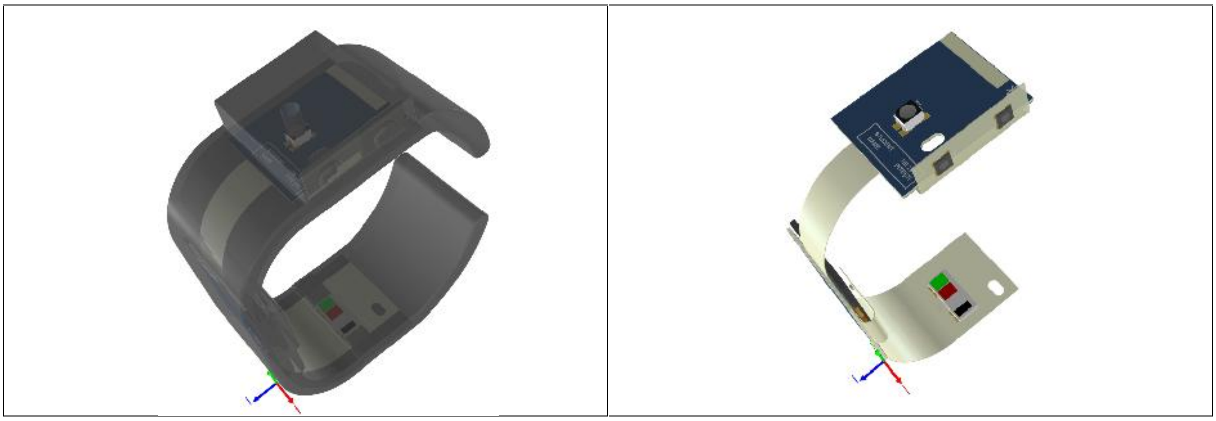

Figure 2 a). Mechanical housing of the module b).PCB without housing

!! All the relevant mechanical and placement keep-in/out's can be found in the mechanical annexes(ANNEX_2, ANNEX_2_3D), in the contest documentation folders!!

To create the PCB outline, the participants have two options:

1. **To import the attached .dxf/.idf file which contain the mechanical data - PCB outline. Pay attention to the import options and on what layer does the tool place the imported data. At the end, the outline must correspond to the proper layer or the information will be considered useless by the evaluators.**
2. **To create the outline manually as described below:**

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 (U100, U200, U300, U400, U600, PB300) must be created in a new library named TIE2018.
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, no overlap of texts, graphical elements and electrical objects is allowed.
SCH-004	The schematic must be electrically correct, clean and readable. All reference designators and values must ***strictly*** follow ***Annex_1,*** *footprint naming and other text elements are not mandatory*. 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 test pad per net): ***SWD_IO, RESET, SWD_CLK, 3V3 and GND*** for programing**.**
SCH-006	Following completion of the schematic, a ***Bill of Material (BOM)*** must be generated.
SCH-007	The same page size and count shall be used as the one received in ***Annex 1***.

4. Layout design specifications:

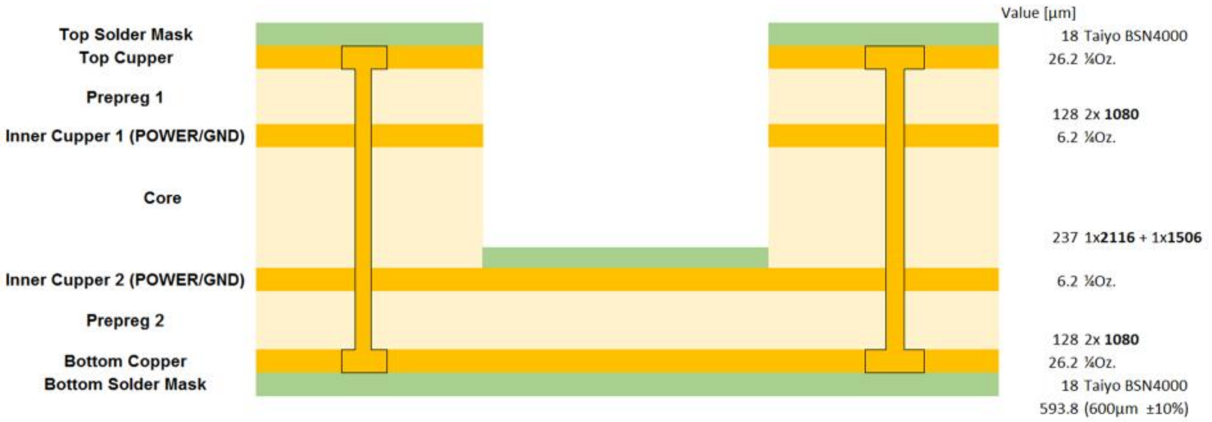

Figure 4. Proposed stack-up definition for the thermal monitoring system.

PCB-001	The PCB layout design will take into consideration the proposed stack-up from ***Figure 4***. Each participant should define in their individual CAD software ***only the rigid stack-up defintion.*** Minimum trace width is ***0.100mm*** and minimum clearance is ***0.100mm***.
PCB-002	Signal routing will be done on top and bottom layers. ***Inner layer_1*** and ***Inner layer_2*** will be used for ***power signals(GND and 3V3)***. Interconnects on flex foil mustbe routed on the layers indicated in the proposed ***stack-up definition table (Fig.4)***.
PCB-003	Component descriptions, including standard footprints are indicated directly in the ***ANNEX_3***. Non-standard footprints (components which have the manufacturer part number TIE2018_...) indicated in ***Table 1***, (***Column 3***), must be designed as specified in the provided datasheets and saved in the ***TIE2018 Library***. All footprints must have proper definition for copper pads, component outline and silk screen.
PCB-004	All VIA will have the following properties: Signal VIA:**200µm** hole diameter and **400µm** copper pad Power VIA:**300µm** hole diameter and **600µm** copper pad
PCB-005	In order to have the desired functionalities, all user interface components must be precisely placed, as indicated in the mechanical documentation (ANNEX_2). Component coordinates refer to the geometrical center.
PCB-006	The PCB origin shall be placed in the middle of the ***2mm NPTH (#A)***.
PCB-007	Minimum distance between two adjacent components is ***400µm*** (component outline to component outline), and pad to pad ***250µm***.

PCB-008	On flexible areas, due to manufacturing requirements the routing will be done with ***rounded traces***. Also, if layer transitions are needed (via usage is mandatory) it is absolutely a must to use ***teardrops*** in order to prevent trace-to-via detachments.
PCB-009	To mechanically enhance the flexibility of the PCB, please insert, in the bendable areas, where there are no traces present or copper areas connected to any net, any kind of ***homogenous copper hatch***. The pattern choosing may be either one available.
PCB-010	The Pulse Oximeter Sensor will be placed as indicated in the mechanical drawing **(ANNEX_2)** on bottom side.
PCB-011	For soldering the wireless power charging receiver coil, two copper pads of minimum ***1.5x1.5mm(1mm edge-to-edge spacing)*** will be placed at distance **≥*5mm*** below the Pulse Oximeter Sensor on the same side.
PCB-012	The Lithium-Ion Battery will be soldered on two copper pads of minimum ***1.5x1.5mm (1mm edge-to-edge spacing)*** placed on Rigid Region A on the same side as Pulse Oximeter Sensor.
PCB-013	TIE2018_SoC shall have a keep-out area under the antenna, free of copper of minimum ***2.5x13mm***.
PCB-014	Signals: ***AC1*** and ***AC2*** will be routed in a parallel manner, having ***400µm*** trace width and a spacing of maximum ***250µm***. Total length of any trace must not exceed ***70mm***!
PCB-015	Signals: {***RX_N, RX_P},{ TX_N, TX_P}*** are 100Ω differential pair signals. The routing shall be done with a trace width of ***110µm*** and ***120µm*** spacing. Differential signals shall have their corresponding shield and return path (***RX_REF*** and ***TX_REF***). The length of any differential pair must not exceed ***35mm***.
PCB-016	SPI interface (***SPI_SIMO, SPI_SOMI, SPI_CLK, SPI_EN)*** traces shall be routed ***as bus***, length shall not exceed ***60mm*** and must have a reference .
PCB-017	Signals ***ANTIN*** and ***ANTOUT***, are single ended 50Ω impedance, and must be routed with ***220µm*** trace width.
PCB-018	I2C interface signals must be routed as ***short as possible*** so the user shall be careful about critical component placement.
PCB-019	Decoupling capacitors and AC coupling capacitors must be properly placed.

5. Thermal considerations:

THERM-001	To prevent battery over-heating, a temperature monitoring thermistor will be used as indicated on the mechanical drawing. NTC signals will be routed with ***300µm*** traces, spaced at ***250µm***.
THERM-002	Calculate junction temperature for ***U100*** considering **RθJA = 37.7ºC/W** and dissipated power (Pd) is ***300mW***. See ***ANNEX_4*** for more details.
THERM-003	In order to properly cool U100, place a suitable number of thermal VIAs close to pins 40 and 44. Thermal resistance of one **VIA is 43.8ºC/W** and minimum required thermal resistance of the obtained VIA cluster is **8.1ºC/W**.

6. Test specifications:

TST-001	Test pads must be ***0.5mm (0.575mm solder mask 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 ***1.00mm***.
TST-002	Global fiducial markers, having circular shape, must be introduced in a proper number, according to IPC recommendations.
TST-003	A suitable number of test points for programing must be placed on bottom side.

7. Fabrication specifications:

FAB-001	In order to complete the manufacturing data, please 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 necessary 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 coordinates must be created, as a text file.

Total: 300 points

TIE Drive Content:

Name	Content
ANNEX_1.pdf	**Shematic definition**
ANNEX_2.pdf	**Mechanical drawing**
ANNEX_2_3D.pdf	**3D mechanical CAD data**
ANNEX_2.idf	**PCB Outline IDF format**
ANNEX_2.dxf	**PCB Outline DXF format**
ANEX_3.pdf	**BOM table**
ANNEX_4.pdf	**Semiconductor and IC Package Thermal Metrics**
ANNEX_5	**Footprints folder**

Component list

TIE 2018

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

#	Designator	Comment	Footprint Name	Quantity
1	C100, C101	10nF	0603	2
2	C102, C103, C104, C300, C400, C401	100nF	0402	6
3	C105	4.7uF	0603	1
4	C301	DNP	0402	1
5	C302	1pF	0402	1
6	L300	1.1nH	0402	1
7	L301	10nH	0402	1
8	L302	10uH	0603	1
9	PB300, PB301,	TIE2018_PB	See Datasheet	3
10	R100, R101, R301, R302, R400, R401	10k	0402	6
11	R102	3.83k	0402	1
12	R300, R303	4K7	0402	2
13	R600. R601. R602	100R	0402	3
14	U100	TIE_2018_WPC_AFE	See Datasheet	1
15	U200	DCM03	See Datasheet	1
16	U300	TIE2018_SoC	See Datasheet	1
17	U400	TIE2018_DIG_MIC_GYRO	QFN16(3x33mm)	1
18	U600	TIE2018_RGB	PLCC4	1
				34

Table 1 – Bill of materials: