1 General description of the project

The project goal is to design and generate layout and fabrication files for the PCB module of a **solar battery charger**. The schematic diagram is indicated in Annex 1.

The **solar battery charger** consists of:

1. a single chip boost converter with dynamic Maximum Power Point Tracking (SPV1020);
2. an output voltage and current controller (SEA05LTR);
3. a comparator circuit to detect a low-battery condition (around TL331KDBVR);
4. a smart high side driver with built in overcurrent protection function (AUIR3313S);

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 shall be considered in mm.

The PCB of the charger shall fit in the following metallic enclosure:

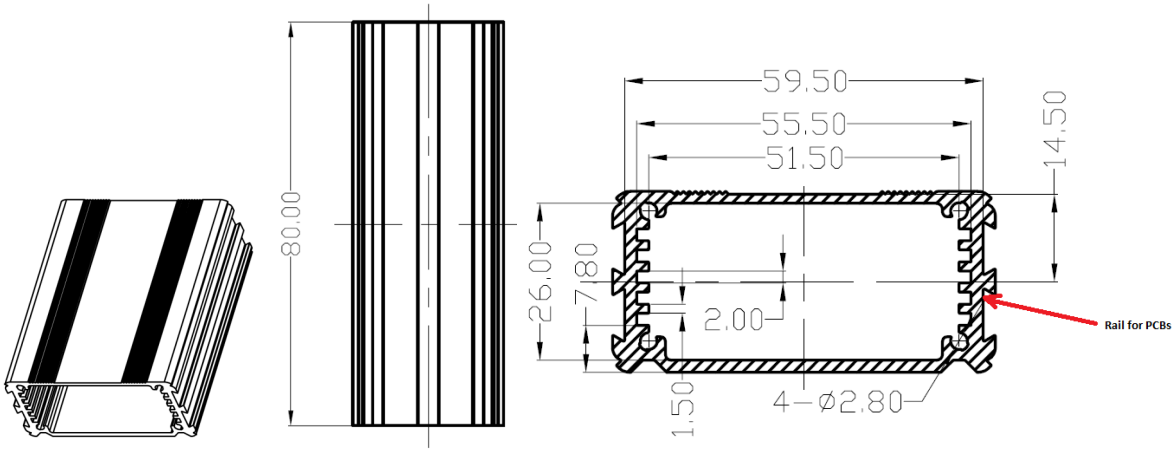


Figure 1. Dimensions of the enclosure

The PCB positional constraints for some of the components are detailed in Figure 2:

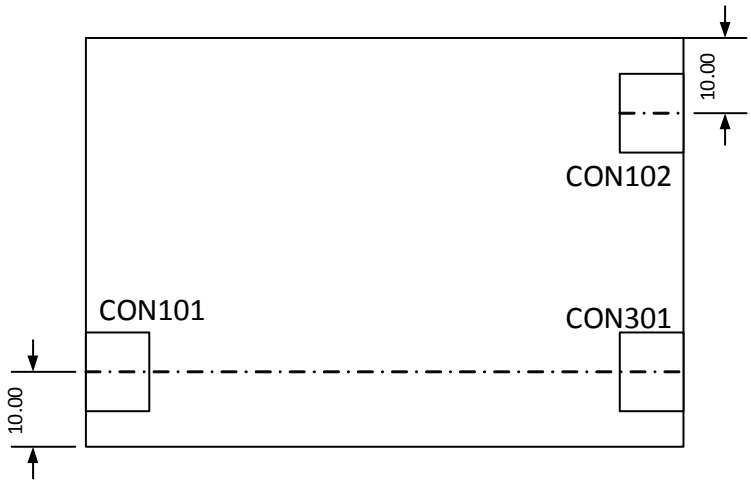


Figure 2. Position constraints for some of the components

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 U101, U201, U302 components will be created in a new library named TIE2016.
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. It is not mandatory to indicate footprint information in the schematic. Tolerance must only be indicated for resistors for which it is different than +/-1%.
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 pad per net): VIN, VCC, VIN_SNS, VREG, PZ_OUT, LOAD+.
SCH-006	Following completion of the schematic, a Bill of Material (BOM) must be generated.

4 Mechanical design specifications (25 points)

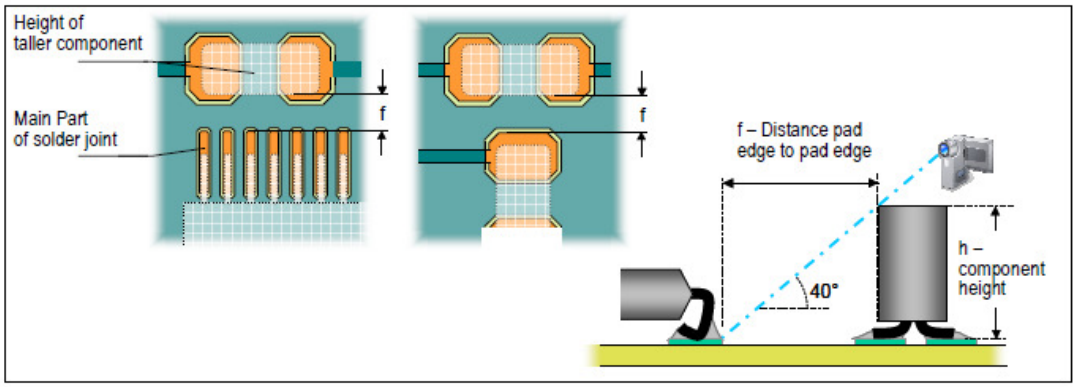
MEC-001	The housing dimensions indicated in Figure 1 are fixed (0% tolerance) and the fabrication capabilities in terms of PCB dimensions are +/-0.1mm tolerance. PCB must fit in the second rail from the bottom of the enclosure indicated in Figure 1. Choose the dimension the PCB so that: • The portion of the GND ring defined in requirement (PCB-007) along the long edges of the PCB sits completely on the rail, even under worst case conditions (PCB shifted from its nominal position). • At least 0.1mm space remains between each of the long sides of the PCB and the corresponding enclosure wall when the middle axis of the PCB is aligned with the middle axis of the housing (PCB not shifted from its nominal position). • The length of the PCB is 0.2mm shorter than the long edge of the enclosure, in order to ensure sufficient margin.
MEC-002	Connectors must be placed according to the constraints indicated in Figure 2.
MEC-003	For electrical testing of the PCBs, tooling holes are mandatory. The tooling holes shall be placed asymmetrically, as far away from one another, on the edges of the board to avoid reverse insertion of the assembly. Regardless of the testing method two tooling holes must be present. The following points must be taken into account when placing the holes : • The holes shall be freely accessible and not be covered by labels, components, etc. • Diameter of the tooling holes shall be 3.0mm • Distance of tooling holes edge to component edge or testpoint edge must be at least 2.0mm. • Distance of tooling holes edge to board edge must be at least 2.0mm
MEC-004	All components must be placed so that even under worst case conditions (PCB shifted from its nominal position) there is at least 1mm between each component and the nearest housing feature.

5 Layout design specifications (170 points)

PCB-001	The layout design will take into consideration a FR4 double-sided PCB with 70µm copper thickness on each layer. Minimum copper width is 0.150 mm and minimum clearance is 0.140 mm.
PCB-002	Component descriptions including standard footprints are indicated directly in Table 1. 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	Vias connecting electrical layers will have 0.3 mm drill and 0.6 mm pad diameter.
PCB-004	All components must be placed on the top side of the PCB.
PCB-005	Minimum distance between 2 adjacent components is 0.5 mm edge to edge.
PCB-006	A 0.6mm wide (GNDPWR) ground ring (not covered with solder mask) must be routed on all layers along the PCB outline at 0.5mm distance from the PCB edge. The ground rings from different layers must be connected to each other with a number of at least 10 (in total) vias spaced at least 15mm from each other.
PCB-007	A 10mm x 10mm copper area (separated from all nets) covered by solder mask, shall be placed on the PCB (for data matrix code).
PCB-008	The following tracks are defined as being high current paths: • The VBAT path between CONN102 and CONN301 (excluding the remaining portions of the VBAT net)

	• The GNDPWR path between CONN102 and CONN301 (excluding the remaining portions of the GNDPWR net) The minimum width for these paths must be 1mm (larger is allowed, planes are allowed)
PCB-009	The tracks connecting L101 and L103 to U101 must be routed symmetrically, having identical length and width (and hence impedance). Differences of +/-1mm for length and +/-0.1mm for width are acceptable. The tracks connecting L102 and L104 to U101 must be routed symmetrically, having identical length and width (and hence impedance). Differences of +/-1mm for length and +/-0.1mm for width are acceptable.
PCB-010	U101 must be provided with proper thermal pads/areas/clearance for cooling. The PCB area under its exposed pad must be covered with copper and must be connected with a 4x3 via matrix to a copper area on the opposite side of the PCB of at least 11.5mm x 7.5mm.
PCB-011	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-012	The circuit made up of R201, R202 and R203 must be placed as close as possible to the corresponding pins of U201. The tracks connecting R201 to R203 and R202 to R203 must be routed symmetrically, having identical length and width (and hence impedance). The tracks connecting R201 to U201 and R202 to U201 must be routed symmetrically, having identical length and width (and hence impedance). Differences of +/-0.3mm for length and +/-0.1mm for width are acceptable.
PCB-013	There are four voltage dividers used for correct functionality of the schematic. Identify these circuits and place them as close as possible (less than 10mm) to the corresponding IC.

6 Test specifications (15 points)

TST-001	Test pads must be 1mm 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 2.54mm)
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 component U101.
TST-004	In order to allow appropriate line of sight for a camera from the AOI system to the IC pins, there must be a minimum distance between the C104, C105, C106 and C107 components and pins of U101. This distance is at least1.1xHeight of the blocking components (the capacitors). 

7 Fabrication specifications (10 points)

FAB-001	The necessary fabrication files (in extended Gerber format) must be provided.
FAB-002	Distinct drill file for holes must be provided.
FAB-003	Pick-and-place file for all SMT components must be generated.
FAB-004	A list of testpoint co-ordinates must be created, as a text file.
FAB-005	In order to manufacture this PCB, Annex2 file must have all the necessary information filled in. The purpose of this file is to specify all relevant properties of the PCB to the manufacturer.

Total: 300 points

Table 1 - BOM

Component Reference	Description	Manufacturer_PN
R202	RESISTOR, THICK FILM, 4K7, ±1%, 0.1W, 0603	
R206	RESISTOR, THICK FILM, 22K, ±1%, 0.1W, 0603	
R207	RESISTOR, THICK FILM, 220K, ±1%, 0.1W, 0603	
R203	RESISTOR, METAL FILM, 0.07R, ±1%, 0.125W, 0805	
R201	RESISTOR, THICK FILM, 15R, ±1%, 0.2W, 0603	
R105	RESISTOR, THICK FILM, 1K, ±1%, 0.1W, 0603	
R310	RESISTOR, THICK FILM, 510R, ±1%, 0.1W, 0603	
R302	RESISTOR, THIN FILM, 78K7, ±0.1%, 0.166W, 0603	
R305	RESISTOR, THICK FILM, 910K, ±1%, 0.1W, 0603	
R304	RESISTOR, THICK FILM, 2K2, ±1%, 0.2W, 0603	
R101, R301	RESISTOR, THIN FILM, 240K, ±0.1%, 0.063W, 0603	
R103	RESISTOR, THICK FILM, 330K, ±1%, 0.1W, 0603	
R102, R104, R205, R303	RESISTOR, THIN FILM, 11K, ±0.1%, 0.166W, 0603	
R204	RESISTOR, THIN FILM, 118K, ±0.1%, 0.166W, 0603	
C104, C105, C106, C107, C201, C301, C302	CAPACITOR, CERAMIC, 100nF, ±10%, 50V, X7R, 0603	
C303, C304, C305	CAPACITOR, CERAMIC, 1nF, ±10%, 50V, X7R, 0603	
C202	CAPACITOR, CERAMIC, 4.7nF, ±10%, 50V, X7R, 0603	
C103, C112	CAPACITOR, CERAMIC, 2.2nF, ±10%, 50V, X7R, 0603	
C113	CAPACITOR, CERAMIC, 470nF, ±10%, 25V, X7R, 0603	
C114, C203	CAPACITOR, CERAMIC, 22nF, ±10%, 25V, X7R, 0603	
C102	CAPACITOR, CERAMIC, 1uF, ±10%, 50V, X7R, 0805	
C101, C108, C109, C110, C111	CAPACITOR, CERAMIC, 4.7uF, ±10%, 50V, X7R, 1210	
L101, L102, L103, L104	INDUCTOR, 68uH, ±20%, 2.6A	MSS1278T-683MLD
D102	DIODE, SCHOTTKY, POWER RECTIFIER, 1A, 60V, SMB	
D101	DIODE, TVS, 40V, 600W, SMB	
U101	VOLTAGE REGULATOR, DC-DC, BOOST, 40V, 320W	SPV1020
U303	SMART HSD, CURRENT SENSE, PROGRAMMABLE OVERCURRENT PROTECTION, 32V, 23A, 7mOhm	AUIR3313S
U302	COMPARATOR, SINGLE, SOT23	
CONN302	CONNECTOR, THD, HEADER, MALE, 2 PIN SIL	961102-6404-AR
CONN101, CONN102, CONN301	CONNECTOR, THD, TERMINAL BLOCK, 2 POSITION	PM5.08_2_90
CONN103, CONN104, CONN201, CONN202, CONN203, CONN303, CONN304	CONNETOR, THD, HEADER, MALE, 1 PIN SIL	90120-0761
D103, D104	IC, COOL BYPASS SWITCH, 12.5A, 40V	SPV1001N40
U201	IC, CC-CV CONTROLLER, 36V	SEA05LTR
U301	IC, SHUNT VOLTAGE REFERENCE, 3V, SOT23	