# 1 General description of the project

The project goal is to design and generate layout and fabrication files for the PCB of a **5W wireless charger**; the schematic diagram is presented in Appendix 1.

The **wireless charger** consists of:

1. a single chip wireless power transmitter U2;
2. a full bridge inverter: integrated half bridge U2 and external half bridge Q3;
3. an EEPROM U3;
4. an energy transfer coil L3;
5. connector X1 and the ESD protection circuit U1;

# 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 metric system. Accepted tolerance is ±0.1 mm, unless stated otherwise. |

# 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, Q1 and Q3 will be created in a new library named with the last name (surname/family name) of the contestant. |
| SCH-003 | The schematic must be drawn in a clear manner, e.g.: all references and values shall be easily readable; un-necessary crossings shall be avoided; all reference designators shall be identical to those in appendix 1. It is not mandatory to indicate footprint information in the schematic. |
| SCH-004 | The schematic must be electrically correct; the main purpose is to generate a correct netlist for PCB design but it must also provide a clear representation of functionality. |

# 4 Mechanical design specifications (15 points)

| | |
|---|---|
| MEC-001 | PCB geometry is specified in figure 2. |
| MEC-002 | The X1 connector must be placed on the bottom side of the PCB, aligned to the right edge, as illustrated in figure 1. The center of pin 3 of the connector shall be exactly 7mm away from the upper edge of the PCB and its orientation must allow attachment of an external cable. |
| MEC-003 | The LED shall be placed on the horizontal axis of the PCB (as shown in figure 2), with its center 6 mm from the edge. |

**The PCB** of the wireless charger shall fit in the plastic enclosure presented in figure 1.

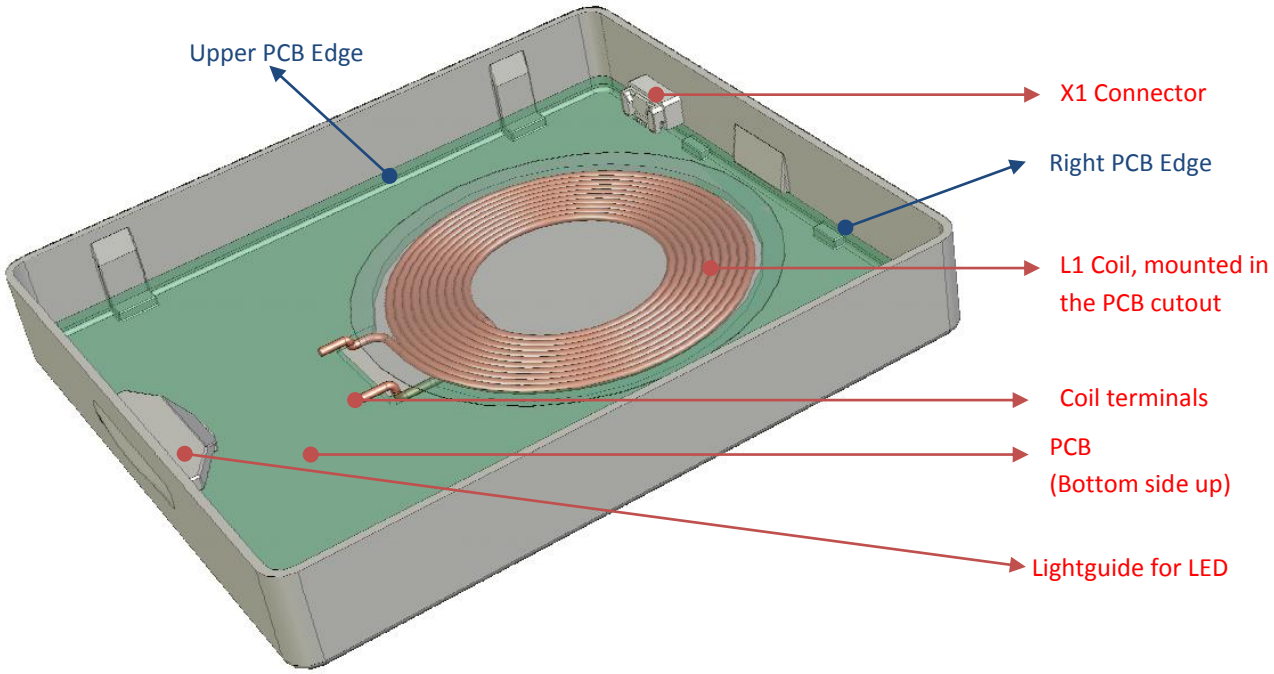


*Figure 1.The 3D view of the wireless charger's enclosure.*

The **PCB geometry** is detailed in figure 2.

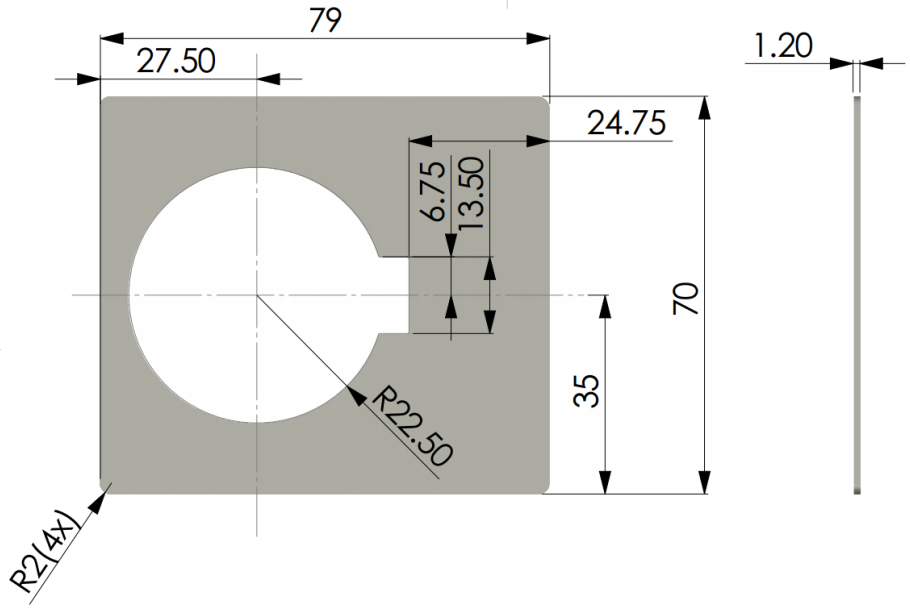


*Figure 2.Mechanical drawing of the wireless charger PCB (top view).*

# 5 Layout design specifications (180 points)

| | |
|---|---|
| PCB-001 | The layout design will take into consideration a FR4 double-sided PCB with 35µm copper thickness on each layer. Minimum copper width is 0.150 mm and minimum clearance is 0.150 mm. |
| PCB-002 | Component footprints are indicated on the schematic.<br>Parts marked with "NP" are not populated, but footprints must exist on the PCB.<br>Use provided datasheets for more information. No tolerance is allowed for pitch values. |
| PCB-003 | Vias connecting electrical layers will have 0.4 mm drill and 0.7 mm pad diameter. |
| PCB-004 | The L3 coil will be placed in the cutout of the PCB with the coil terminals soldered on the bottom side.<br>All other components will be placed on the bottom side. |

| | |
|---|---|
| PCB-005 | Minimum distance between 2 adjacent components is 0.5 mm edge to edge. |
| PCB-006 | Minimum distance between components (including test pads) and edges of the PCB is 3 mm edge to edge, except for L3, X1 and LED1. |
| PCB-007 | A 10mm x 10mm copper area, covered by solder mask, shall be placed on the PCB for data matrix barcode. |
| PCB-008 | The D+ and D- signals shall be routed as differential pair (maximum acceptable track length difference 1mm) and with these additional constraints: track width 0.2mm, clearance between tracks 0.25mm (zero tolerance !) |
| PCB-009 | Components U2 and Q3 must be provided with proper thermal pads/areas/clearance for cooling. The PCB area under the exposed pad (EP) of U2 must be covered with copper and must be connected with a 5x5 via matrix to a copper area on the opposite side of the PCB of at least 4.1mm x 4.1mm. Thermal area for Q3 must follow the indications of Figure 1 on page 10 in the datasheet of the component. |
| PCB-010 | Maximum current flowing through the L3 coil is 7A. All tracks on the PCB carrying this current must be dimensioned to provide this capability at a maximum local temperature rise of 40°C. Clearance between these tracks and other small signal tracks must be kept larger than 2mm, except in an area of 2mm around the pins of the following components (when, due to pitch or other footprint constraints it is allowed to decrease this clearance to 0.25mm): U2, Q1, Q3, D1, X1 |
| PCB-011 | Each track connecting any component to L3 inductor must be kept shorter than 5mm. |
| PCB-012 | ESD protection devices must be placed in such a way to fit their purpose: tracks to and from these devices must be routed in such a way that any potential discharge is forced to first reach the ESD protection device before it would have the chance to reach the protected device. |
| PCB-013 | Decoupling non-polarized capacitors shall be placed as close as possible (max. 3mm) to the related pin and shall be connected to the ground plane through one via. |
| PCB-014 | Connection between signal ground and power ground shall be done as close as possible (within 2mm) to the corresponding pin of L2 ferrite bead. |
| PCB-015 | Track stubs caused by not mounting the components marked in schematic as “NP” (not placed), must be below 5mm in order to minimize the effect of creating unintended antennas |

# 6 Fabrication and test specifications (25 points)

| | |
|---|---|
| FT-001 | Test pads (having 1mm copper diameter) must be placed on the bottom side of the PCB, for all the test points specified in schematic and they must all be accessible for the needles of an In-Circuit Test system (ICT). Minimum distance between test pad centers must be minimum 2 mm. |
| FT-002 | Global fiducial markers, having circular shape, must be introduced in a proper number, according to IPC recommendations. |
| FT-003 | Local fiducial markers will be placed for component U2. |
| FT-004 | The necessary fabrication files (in extended Gerber format) must be provided. |
| FT-005 | Distinct drill file for holes must be provided. |
| FT-006 | Pick-and-place file for all SMT components must be generated. |
| FT-007 | A list of testpoint co-ordinates must be created, as a text file. |

Total: 300 points.