Hardware Design and Testing of a 50 kW T-Type Active Rectifier for Fast Charging Applications

Candidate: Alessandro LOCATELLI **Supervisor:** Prof. Radu BOJOI **Advisor:** Fabio MANDRILE

***Abstract*— This thesis deals with the schematic and PCB design process of the active front-end (AFE) for a 50 kW ultrafast battery charger for electric vehicles. Ultrafast chargers are and will increasingly be a key technology, being at the heart of the transportation electrification. The thesis work has been carried out in collaboration with the interdepartmental Power Electronics Innovation Center (PEIC) at the Politecnico di Torino and it consists of the detailed hardware design of the AFE, from its schematics to the final printed circuit boards (PCBs) and the testing of the converter. The hardware development of this prototype is a work carried out in sequential stages and has required transversal engineering skills, both electrical, electronic and mechanical. In fact, the process started from the theoretical study phase of the active rectifiers, going through the design of the schematic and PCBs, until the assembly and testing of the boards.**

I. Introduction

The second decade of this century has been marked by the technological revolution of vehicles' electrification, both with hybrid (PHEV) and fully electric (BEV) solutions. The reasons are multiple, such as higher efficiencies of the traction line elements and the emissions of pollutants and greenhouse gases are absent in the EVs or very low in the case of hybrids. Nowadays the technology is the traction propelled by batteries, which require high durability and short recharging times. The main problem is the time needed to recharge batteries with high capacity, necessary to guarantee the autonomy. Therefore, off-board chargers have been developed with the possibility of recharging with high power direct current, which reduces the charging time. A ultrafast battery charger is a power electronic converter formed by power units, as can be seen in Fig.1. The converter, object of this thesis, is a 50 kW Active Front End 3 Levels T-Type Inverter, which represents the AC/DC stage of the ultrafast battery charger. The choice of this structure is due to the following reasons:

- Higher efficiency;
- Possibility of vehicle to grid (V2G) operation (bidirectional topology).

The goal of this work is the design, assembly and testing of the active rectifier prototype. Hence, this thesis deals with the detailed hardware design of the AFE, from its schematics to the final printed circuit boards (PCBs) and the testing of the converter. In particular, **my personal contributions** to this project are:

- Analysis of the literature about the battery chargers and the active rectifiers;
- Realization of the converter schematics;
- Choice of the components and sizing of part of the electronic cicuits and measures;
- Routing of the PCBs;
- Bill of Materials' definition;
- Assembly of the modules;
- Preliminar experimental testing of the boards.

The schematics and PCB routing have been realized with the software Altium Designer 20.

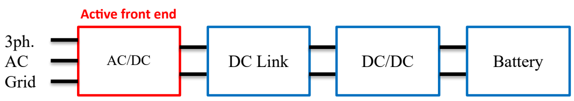

Figure 1: Charger block scheme with AFE.

II. Hardware Design

The Active Front End project has been organized following a PCB (Printed Circuit Board) oriented structure. This means that the overall project is divided into sections representing its sub-modules, which are the PCBs composing the converter. Each of these sub-modules has a specific role inside the functionality of AFE, as can be noticed in Fig.2. The hardware design of the boards shown have been carried out. The hardware design of a converter is a process that needs the iteration of multiple steps, in order to obtain the final prototype of each board, as described in Fig.3.

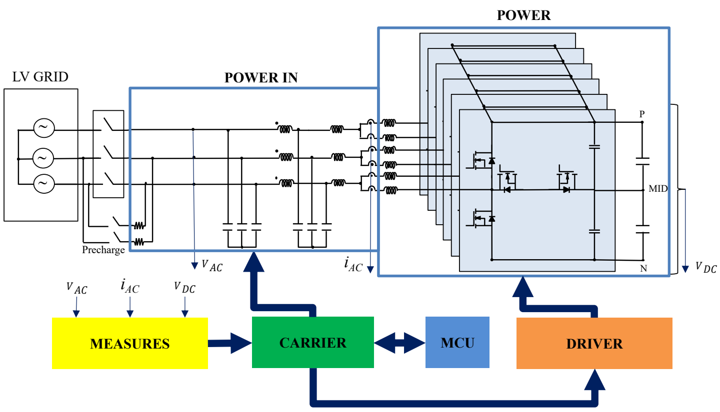

Figure 2: Active Front End boards and schematic.

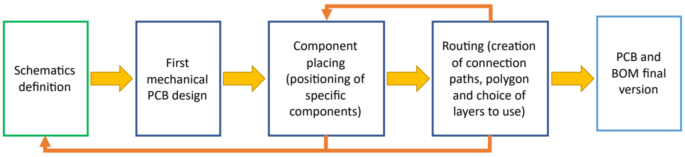

Figure 3: Hardware design process.

III. Schematic design and components' choice

The first step of the hardware design of a converter is the definitions of the electrical schematics. A schematic is the process of creating a logical representation of an electronic circuit. This means connecting a group of symbols (i.e. the components) together, creating an electronic prototype. In this stage the sizing of the circuits and the choice of the components is fundamental because it characterises the operation of each circuit. Some examples of components designed are the following: the power supply of the ICs, the filtering systems of both signals and supplies; the connectors definitions; the circuits for the measures (shunt resistors or voltage dividers, analog conditioning, isolated ADCs); the desaturation circuit design and the gate driver circuit verification; the MCU connections; the precharge circuit.

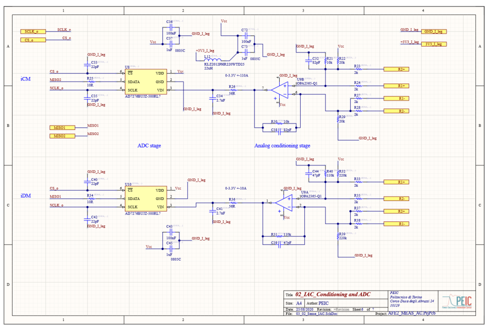

Figure 4: Example of schematic with conditioning and ADC for CM and DM currents measures of one phase.

IV. Printed Circuit Board Design

The design of a PCB is a task where many aspects related to different engineering fields must be considered such as the electrical and the mechanical aspects of the devices used in the PCB design. The technologies adopted in the project are in depth analyzed in the thesis and also the problems related to electromagnetic disturbances are examinated because they are common in every electric and electronic circuit and must be controlled. The PCB design is an iterative process characterized by two main tasks, which are the placing of the components and the tracks routing. After the definition of the board size and the setting of electrical and mechanical rules and grids, the design stage starts with the components' placing, that can be performed directly moving, rotating and arranging each component on the PCB area, according to engineering considerations. The routing process is characterized by a set of rules that impose the main features of the tracks and the isolation of each part, avoiding unexpected failures and critical behaviours. Altium Designer provides a specific editor for the PCB design, where the available 2D view is optimized for the design, while the 3D view is needed to have a complete vision of the board (as reported in Fig.5).

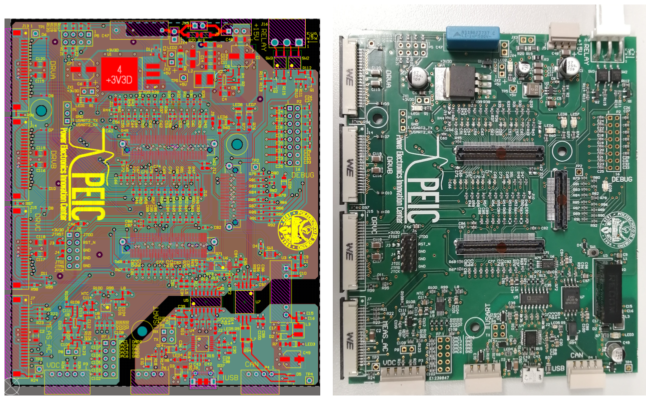

Figure 5: Example of 2D view and the real PCB of the Carrier board.

V. Assembly of the PCBs and experimental testing

After the definition of the BOM, the purchasing of the components and the ordering of the boards, the mechanical assembly of the available components to be soldered and parts of the PCBs with the cable wiring has been carried out in laboratory. Consequenty the preliminar testing of the communications and the power supply system for each board has been performed, in order to validate the main functionalities of the converter (power supply of the control boards, PWM signals and interleaving functionality), as can be seen in Fig.6.

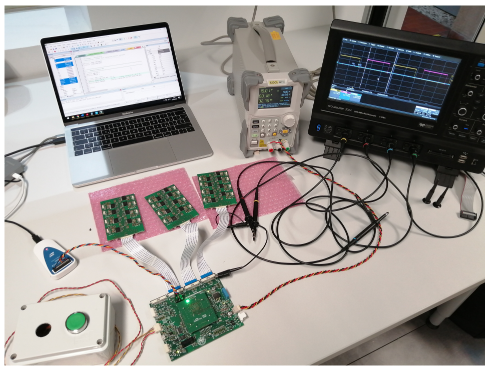

Figure 6: PWM testing (control boards and driver boards)

VI. Conclusions

This master thesis analyzed the schematic and PCB design process of the active rectifier for an ultrafast battery charger for electric vehicles. The hardware development of this prototype is a work carried out in sequential stages and requires transversal engineering skills, both electrical, electronic and mechanical and the team cooperation is needed. Therefore, the master thesis has been an opportunity for both academic, professional and personal development. In addition the tests carried out have been passed and the boards work within the specifications.