Student research project

Circuit board design for interfacing imperix power modules

An electric drive contains the inverter, the electric machine and the control algorithm. After a new control method has been developed in simulation successfully, it must be also verified within a real-world experiment at the test bench. Therefore, a test bench needs to be built and in this case, imperix power modules, as shown in the figure below, can be used to build the power electronic inverter. Due to the modular system of imperix (each circuit board represents a half bridge) different converter topologies can be set up easily, e.g., a two-level three-phase inverter for electric drives. The control algorithm to be tested is executed on a dSPACE MicroLabBox II (MLB II). However, the I/O signal specifications and connector interfaces of these two components are not directly matching to each other. Thus, in this work an interface circuit board to connect these two systems must be designed, soldered and tested at the real test bench in the laboratory.

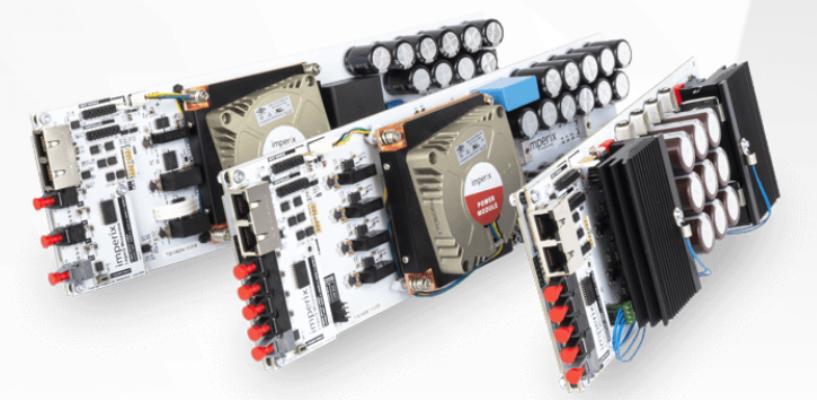

Figure 1: Exemplary imperix half-bridge modules.

Necessary requirements:

- Finished course work on power electronic fundamentals
- First experiences in design of circuit boards
- First experience with computer added design (CAD) programs like KiCad

WP 1: Design the schematic **[2 weeks]**
First, the schematic of the circuit board must be designed, which includes among others the handling of the PWM signal from dSPACE MLB II (digital) to the imperix board (fiber optics connector), a fault indicator (also fiber optic), thus a signal conversion must be done. In addition, the voltage and current sensors on the imperix board are interfaced via RJ45 connector, which must be also adapted for the MLB II. Finally, the schematic must be scaled to host multiple half-bridge power modules and bundle their signals towards the MLB II.

WP 2: Create the PCB layout **[2 weeks]**
In this WP the layout of the circuit board should be created. If the necessary footprint is already in the library, it can be used, otherwise the footprints of the components must be generated. Afterwards, according to the literature, the layout can be designed. At the end a bill of material (BOM) should be created automatically, so that the components can be ordered.

WP 3: Prototyping and hardware test **[2 weeks]**
The chair will purchase all components for the interface board, which then needs to be soldered on the designed PCB. Afterwards a first I/O test of the interface board should be conducted. Furthermore, the circuit board must be physically integrated in the test bench setup, so that a full test of the whole signal chain is possible (i.e., a suitable rack / mounting option must be chosen to host both the imperix modules and the self-designed interface card). Finally, the component interface should be tested in an end-to-end fashion by applying a simple switching pattern to the power modules (which will be connected to some dummy load) while also reading the sensor values.

WP 4: Documentation **[1 weeks]**
All work should be reported in a structured way directly within the schematic, layout and on the circuit board.

Gantt chart

The planned timetable is shown in the Gantt diagram below.

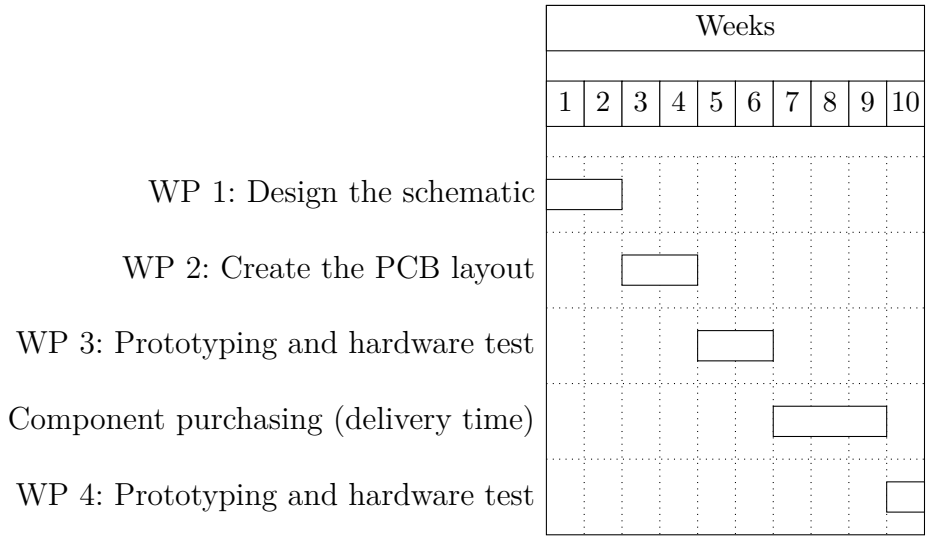

Figure 2: Gantt chart for the project (assuming student full time work on project).