Instructions to connect Trenz TE0715 SoM to Vivado 2018.2

Context

The Trenz TE0715 is a System on Module (SoM) that, together with other features, can be configurated to contain a FPGA and a microcontroller inside. This flexibility makes it usefull to implement complex designs that need both of them.

Important caveats about powering board

To use the baseboard you must power it using a power jack with +5V and GND. However, you must be careful to not limit the current too much, as when programming the Trenz, there is a peak in current consumption, and if that current is not delivered to the board, the firmware load will fail. I suggest to put the current limit up to 5Amps, as the peak current consumed may vary depending on the firmware; this limit ensures to not have firmware load errors because of limited current.

Connection with PC

The Trenz TE0715 has no USB connector to be directly wired to a PC; it uses JTAG as communication protocol. In order to connect it to a PC –to programm it using Vivado–, a “middleman board” mus be used. This middleman board will translate the USB communication to the JTAG protocol that is used by Trenz TE0715.

In the TGC-Charge Monitoring Board project there are 2 options to connect the TrenzTE0715 with the PC:

1. Use the Carrier Board Trenz TE0703
    1. An image of it is:
        1.

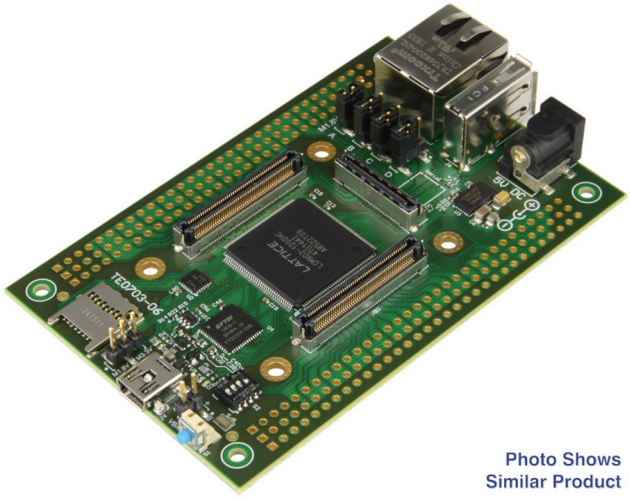

Figure 1

2. Use the test board for 2 mezzanines, together with a Trenz TE0790
    1. An image of the test board with boards TE0790 and TrenzTE0715:
        1.

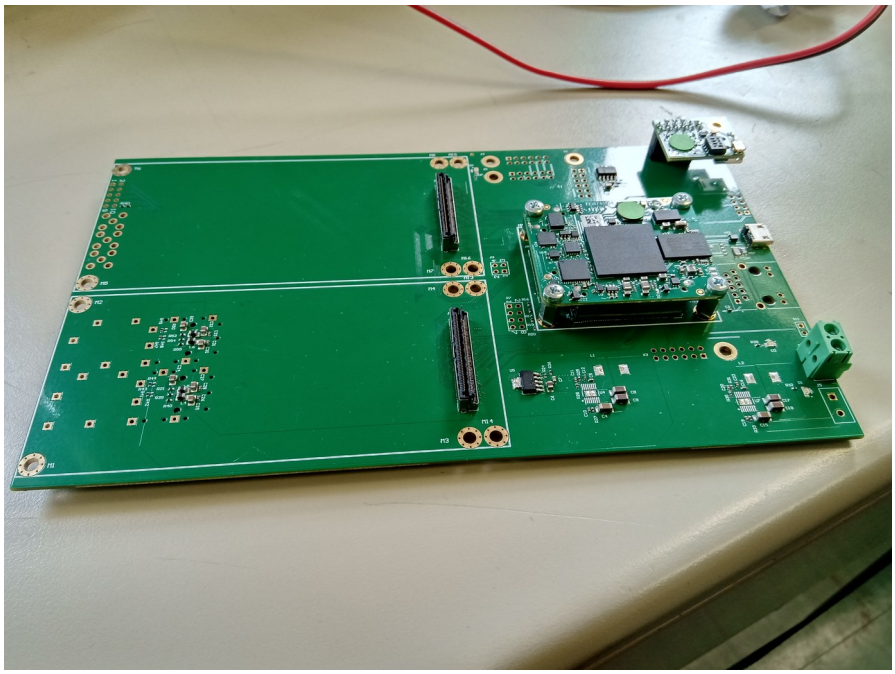

Figure 2

Connection using Carrier Board Trenz TE0703

For this connection, the first step is to plug the SoM into the central connector of the TE0703. After that you must set the jumpers and the DIP switch, so the connection would be safely set for the SoM.

The DIP switches in the TE0703 must be in the following position:

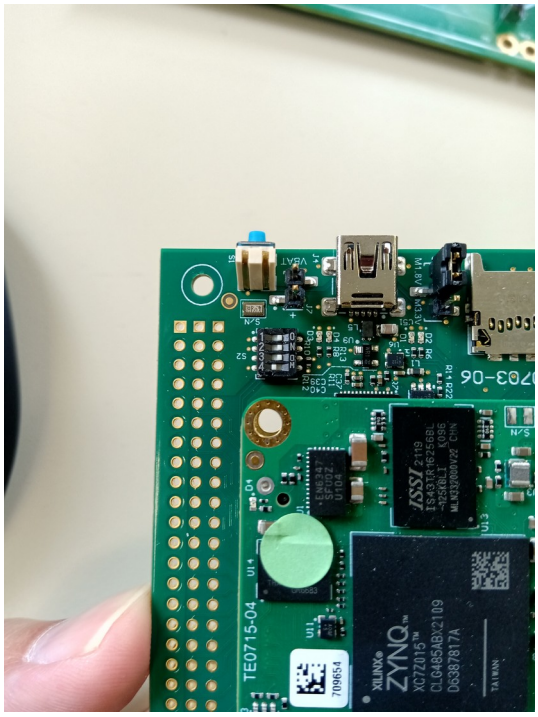

Figure 3

And the jumpers must be all in 1.8Volts. Probably tehre are room for increasing the voltage of part or even all the jumpers up to 3.3v, but for the scope of this project, it is not necessary and it has not been tested.

After completing the previous steps, you can power the Carrier board with 5 volts and GND through the power jack besides the USB connection. After that you may connect the complete system to your PC

Connection using test board for 2 mezzanines, together with a Trenz TE0790

For this connection you should connect the board Trenz TE0790 and the SoM Trenz TE0715 to the testing board; follow the connection seeing in Figure 2.

The board Trenz TE0790 has a DIP switch that will set the type of communication between the PC and the SoM. In order to correctly communicate the SoM with Vivado, the DIP switches must be in the following configuration:

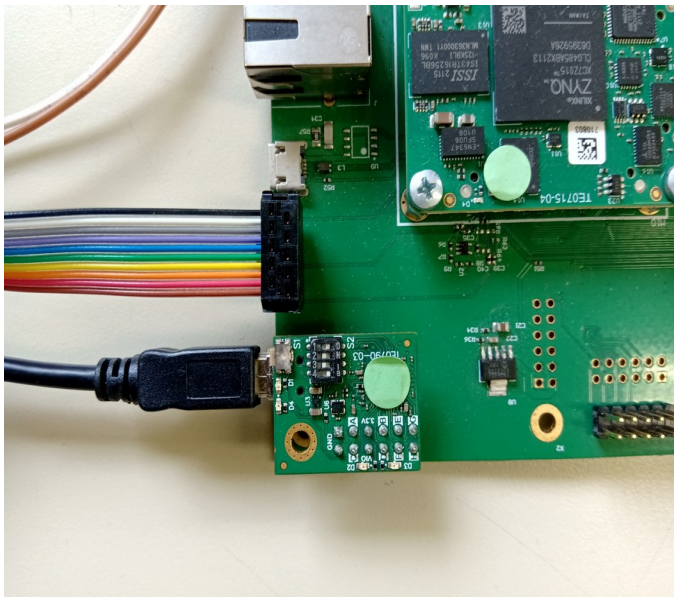

Figure 4

After connecting the boards on the test board, you should power the system using 5 volts and GND. For that, use the Terminal blocks (T-blocks) as the obe shown in the following image.

Figure 5

You must check what is the positive and negative terminal in the T-block (which is 5V and which is GND), but as an initial guess, the side labeled as J5 in the PCB is commonly the 5V terminal, and the side labeled as J6 is the GND terminal.

After powering the test board with 5V and GND, you can connect it to the PC through the USB connection in the board Trenz TE0790.

Vivado Configuration

For Vivado to recognize the SoM, you should add the boards to the available boards. For that you should enter into this Gitlab and download the files inside directory “Boards”: charge-monitoring-system/vivado_proj/board_files .

After that, follow the instructions in the README.md file.

Example “Hello World” program as a first test with Trenz board

1. To connect the board to your computer follow these instructions:
    1. https://www.youtube.com/watch?v=3cwHU7S1v7Q&list=PL_Nji0JOuXg3QpmlzqZmDFDBcIIflpdwA&index=6
2. To program the microcontroller instantiated in the Trenz:
    1. https://www.youtube.com/watch?v=CNnrFVWveek&list=PL_Nji0JOuXg3QpmlzqZmDFDBcIIflpdwA&index=6&t=333s

1. Missing steps in this instruction: In order to be able to open SDK in project's directory, before launching SDK, you must export the hardware to the project's directory. To make that, follow these steps:
    1. In Vivado, from the main Vivado File menu, select `File > Export > Export Hardware`.
    2. Ensure that the Include Bitstream check box is checked and that the Export to field is set to the default option of
    3. Click OK.
    4. To launch SDK, select `File > Launch SDK`.
    5. The Launch SDK dialog box opens.
    6. Accept the default selections for Exported location and Workspace.
    7. Click OK.
        - Reference: https://support.xilinx.com/s/question/0D52E00006iHoZaSAK/unable-to-create-a-new-hardware-platform-in-sdk-from-vivado?language=en_US