



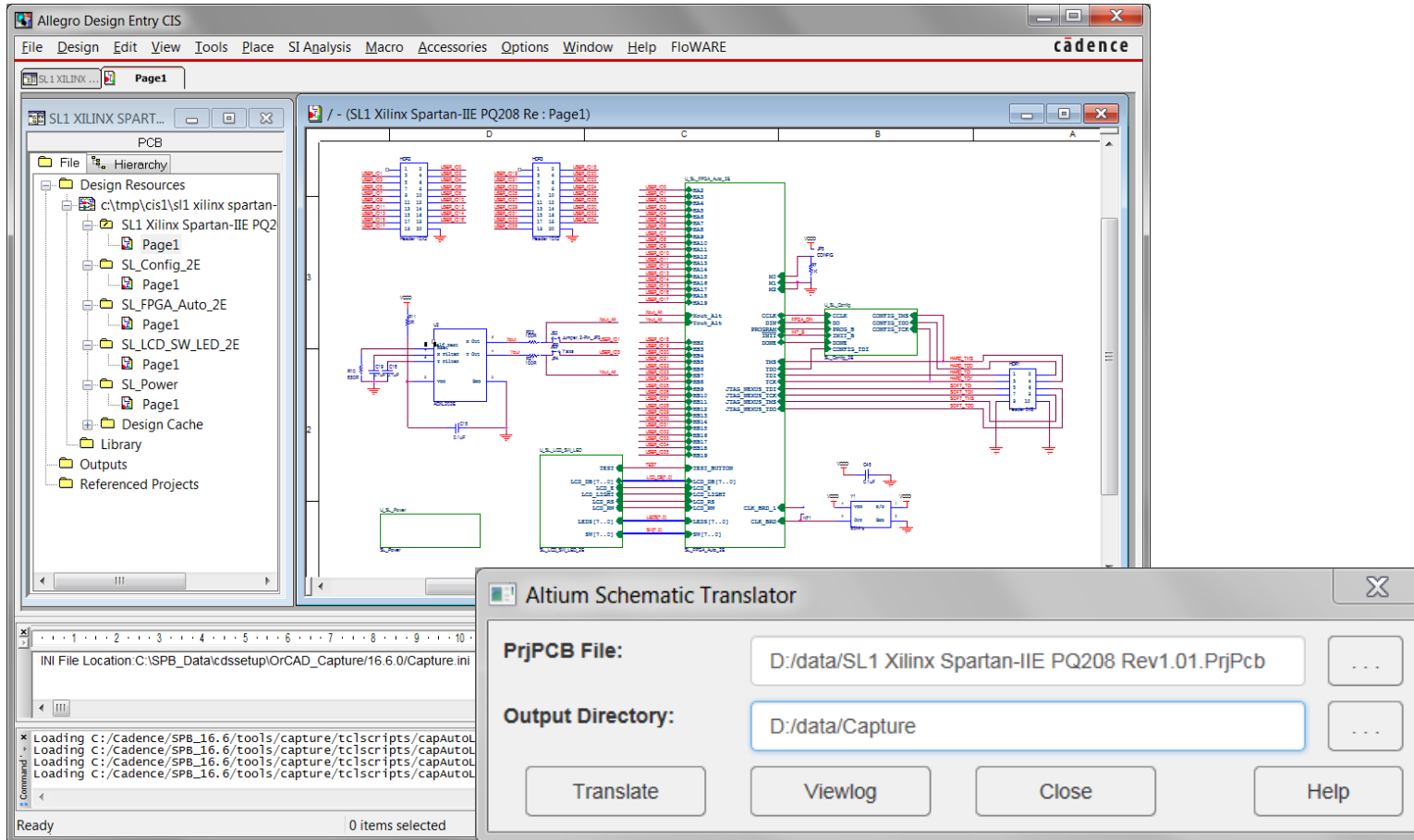
Altium Schematic Translator - Capture

January 2016

cādence®

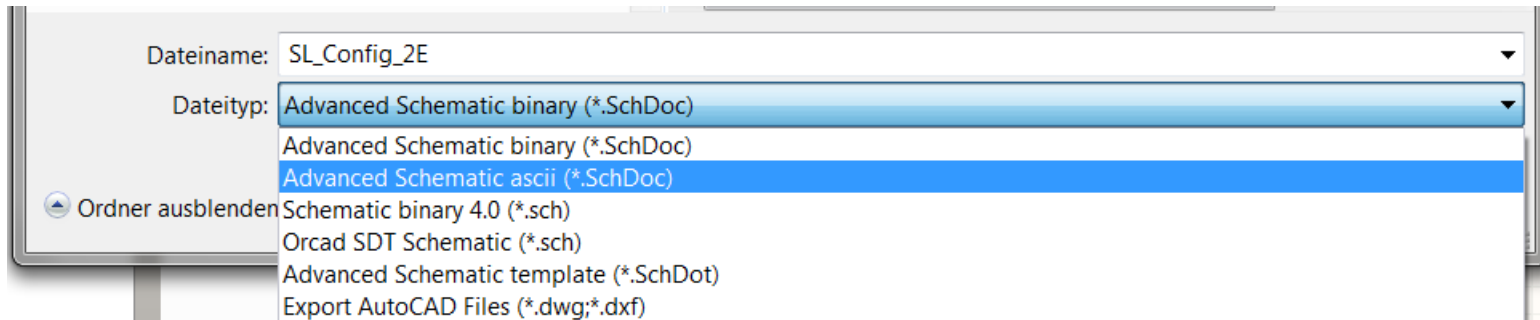
Altium Schematic Translator – Capture

- Translates Altium schematics to OrCAD Capture



Prerequisites: ASCII Schematic








- Schematic has to be saved to ASCII format within Altium. This will replace the original binary file by its ASCII equivalent. Location and file extension is the same.



Prerequisites: Project structure

- Schematics can be translated only if they are embedded in a PCB project (***.PrjPCB**) which manages the design documents needed to manufacture a PCB design.
- Furthermore a valid structure file (***.PrjPCBStructure**) is needed. It will be generated after compiling the PCB project and references the individual schematic pages (***.SchDoc**).
- If the project file does not exist:
 - Select *File » New » Project* from the menus, the *New Project* dialog will open. From the list of available project types, choose *PCB Project*
 - Add the schematic documents ***.SchDoc** to the project
 - Compile the project (e.g. *RMB » Compile PCB Project*)
 - For more documentation about setting up PCB projects refer to Altium documentation.

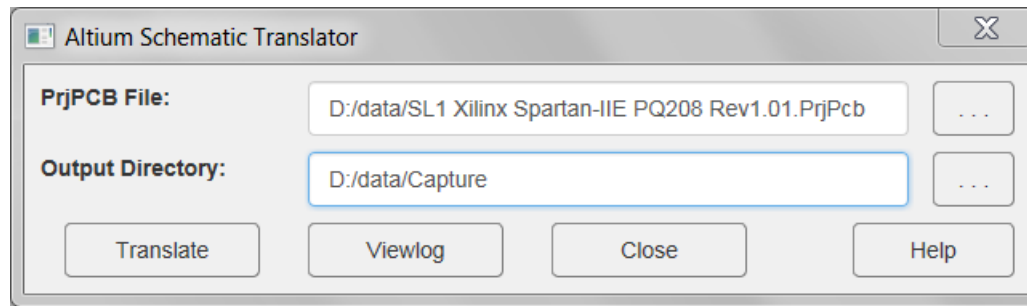
Project structure example

Name	Größe	Änderungsdatum
 SL_Config_2E.SchDoc	55 KB	14.11.2013 16:21
 SL_FPGA_Auto_2E.SchDoc	297 KB	13.11.2013 15:47
 SL_LCD_SW_LED_2E.SchDoc	177 KB	19.11.2013 08:29
 SL_Power.SchDoc	279 KB	19.11.2013 08:35
 SL1 Xilinx Spartan-III PQ208 Rev1.01.PcbDoc	5.647 KB	21.01.2014 10:53
 SL1 Xilinx Spartan-III PQ208 Rev1.01.PrjPcb	43 KB	15.02.2011 19:20
 SL1 Xilinx Spartan-III PQ208 Rev1.01.PrjPcbStructure	2 KB	12.11.2013 14:32

- Each schematic page * **.SchDoc** is saved in ASCII format.
- Project file * **.PrjPCB** acts as project master.
- File * **.PrjPCBStructure** has references to schematic pages (relative path).
- File * **.PcbDoc** is only needed when PCB has to be translated.

Running the translator

- Launch **Altium-Capture** translator
 - Select Altium project file **.prjPCB**
 - Specify output directory for OrCAD project
 - Translate



- Once finished open the project and check schematic for translation errors

Synchronize with PCB

- If you plan to translate a complete project including PCB
 - Run *Tools – Create Netlist* and specify an empty database
 - Open this database with PCB Editor
 - Within PCB Editor run ***Altium-PCB*** translator and navigate again to **.prjPCB** file. The translator will retain netlist and device logic and translate only the remaining data in order to complete the board.

cā dence[®]