FEDEVEL ACADEMY

CERTIFICATION

This is to certify that

Federico Vittorio Lupo

has successfully completed a course

Learn Altium Essentials

Date: **16 AUG 2021 - 26 SEP 2021**

Course tutor Robert Feranec

Software used: **Altium Designer**
Course content: 1) Starting a new project, Creating a schematic symbol library 2) Drawing a schematic 3) Creating footprints and a PCB library 4) Doing PCB Layout 5) Improving PCB Layout 6) Finishing PCB, Generating manufacturing outputs

Certificate number: 1c653a17603c1a7f8750674f2