

CERTIFICATION

This is to certify that

Benjamin Soltau

has successfully completed a course

Advanced Hardware Design

Date: 25 OCT 2021 - 02 JAN 2022

Course tutor

Robert Feranec



Software used: **Altium Designer**

Course content: 1) Starting a new project 2) Schematic design 3) Selecting components and creating libraries 4) Footprints, 3D models, starting a new PCB 5) Variants & BOMs 6) Checking libraries and schematic 7) PCB layout & Placement 8) PCB layout & Impedances, stackup 9) Stackup, finishing PCB, length matching 10) Generating output documentation

Certificate number: 1c654a07902c2a8f4780576f2