FEDEVEL EDUCATION

CERTIFICATION

This is to certify that

Erdi Can YILDIRIM

has successfully completed
FEDEVEL ACADEMY course

Schematic & PCB Design

Date: **21 FEB 2024**

Course tutor: Robert Feranec

Course duration: 10h 50min
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: 1c750a87000c8a5f3720572f3