



CERTIFICATION

This is to certify that

Pablo Arguinzones

has successfully completed the course

Learn Altium Essentials

Date **22 AUG 2025**

Course tutor

Robert Feranec

Robert Feranec



Course duration: 9h 45min **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