

CERTIFICATION

This is to certify that

Ronny Lanes Silveira

has successfully completed a course

Learn Altium Essentials

Date: 26 FEB 2022

Course tutor

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

Certificate number: 1c654a57406c0a9f1700172f6