

CERTIFICATION

This is to certify that

Ivan Liu Zhu

has successfully completed a course

Learn Altium Essentials

Date: **03 MAY 2021 - 13 JUN 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: 1c652a37108c6a6f0710270f9