

CERTIFICATION

This is to certify that

Cheuk Yu Chau

has successfully completed a course

Learn Altium Essentials

Date: 25 OCT 2021 - 05 DEC 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: 1c653a87009c3a5f4730679f6