

CERTIFICATION

This is to certify that

Gerhard de Clercq

has successfully completed a course

Learn Altium Essentials

Date: 12 MAR 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: 1c654a67500c4a7f9730270f3