



# CERTIFICATION

This is to certify that

**Marcelo Llomplat**

has successfully completed the course

**Learn Altium Essentials**

Date **20 MAR 2026**

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