

Altium Designer Basics Course

Implementing a printed circuit board (PCB) in your FYP / research? Altium Designer is a powerful and complicated fully integrated PCB design suite which has a significant learning curve. The Monash IEEE student branch is organising a workshop consisting of two 3-hour sessions to teach the basics of using Altium to create a PCB. There are only 30 places available so register now at the link below.

Event Details:

Location: Room 213 and 215, Building 16 Alliance Lane (Building 35)
Dates: 8th September 2016, 6pm – 9pm

Registration is Compulsory! Limited to 30 places

There will be refreshment provided!

The workshop is free for IEEE members, \$10 for Non-IEEE members

Contacts:

Chang Wei – chang.tan@monash.edu

Ashan – ashan.senevirathne@monash.edu

Registration Page



Sign up now!

<https://goo.gl/omZRNF>

Altium
Designer®

