

IEEE Hamilton PCB Layout Workshop

Overview

The PCB Layout workshop provides an introduction to laying out printed wiring boards, and preparing the files necessary for production. It is suitable for design engineers and managers who need an overview of the process connecting designs to manufactured parts. It is suitable for hobbyists and experimenters who want to know how to create their own boards. At the end of this course, circuit designers will be prepared to implement their prototype designs on double sided PCB.

Cost

\$100.00
\$ 75.00 (IEEE Members)
\$ 40.00 (IEEE Student Members)

Duration

One Day

Date

Saturday 2007-April-21

Location

Computer Lab (t.b.d.)
Information Technology or Engineering Building
McMaster University
1280 Main St. W.
Hamilton, ON

Prerequisites

Experience with Microsoft Windows operating system.
Experience with or knowledge of electronics.

Products Used During the Course

Cadsoft Eagle Schematic (trial version)
Cadsoft Eagle Layout (trial version)
PentaLogix Viewmate

Instructor

John Harris, P.Eng., Ph.D., SMIEEE, has 12 years experience designing low-cost high-volume single-sided PCB; and 6 years experience designing 2-layer and 4-layer PCB with surface mount technology. He has also designed small IC's, parts using steel and flexible boards, and circuits using printed thick film components.

Registration

Please contact John Harris j.harris@ieee.org

Course Outline

1. Setting up a Design
2. History of PCB Design
3. PCB Materials and Layers
4. Pre-design Preparation
5. Database and Design Rules
6. PCB Board Outline
7. Importing Design
8. Parts Placement
9. Trace Routing
10. Tricks to Enable Layout
11. Design Cleanup
12. Back Annotation
13. Preparing Files to Order PCB
14. Manufacturing Documentation