

# ***Boot Camp – Solidworks PCB Course Agenda***

---

## **Course Overview**

Boot Camp for SOLIDWORKS PCB is a comprehensive course that brings users up to speed on all essential aspects of the SOLIDWORKS PCB software. The target audience for this course is wide ranging:

- Electrical Engineers with little PCB tool experience
- Electrical Engineers and Designers who's organizations prefer to stay under the Solidworks banner
- Altium Designer users who are considering SOLIDWORKS PCB as an alternative
- Mechanical Engineers or Solidworks Premium users who occasionally need to do PCB design
- Electrical or Mechanical Engineers who wish to become Mechatronics Engineers

Students will gain experience by working through a real-life design example. Design files are supplied along with the training manual, and consist of a PCB outline and a simple set of schematics. This course provides the knowledge necessary to see the design through from inception to complete fabrication and assembly files and drawings. The class is taught by well seasoned Altium Designer instructors who are familiar with both tool sets.

The course is 3 full days. By conclusion of the third day, students will have experience with all the editors and many of the powerful features within SOLIDWORKS PCB. Most importantly, students will become well acquainted with the software and aware of its many capabilities and the possibilities it brings to their organization.

## **Day 1 - Environment**

- 1.1 - Preface - Introduction to Boot Camp for Solidworks PCB
- 1.2 - Installation of Solidworks PCB and Solidworks PCB Services
- 1.3 - SVN (Version Control) Primer
- 1.4 - Project Creation
- 1.5 - Project Workflow and SWPCB Environment
- 1.6 - Libraries, Availability and Integrated Libraries
- 1.7 - PCB Basics
- 1.8 - PCB Layers, Board Shape and Board Setup
- 1.9 - PCB Options, Preferences and Grids



# ***Boot Camp – Solidworks PCB Course Agenda***

---

## **Day 2 - Schematic**

- 2.1 - Schematic Basics
- 2.2 - Project Options
- 2.3 - Schematic Options and Preferences
- 2.4 - Schematic Libraries
- 2.5 - Schematic Construction
- 2.6 - Schematic Placing and Wiring
- 2.7 - Page Numbers, Annotation and Compiling
- 2.8 - Schematic Outputs
- 2.9 - Updating to the PCB

## **Day 3 - Printed Circuit Board**

- 3.1 - Design Rules and DRC Detail Markers
- 3.2 - Component Placement
- 3.3 - Routing and Other Copper Features
- 3.4 - Design Rule Check
- 3.5 - PCB Outputs
- 3.6 - Solidworks PCB Connector

You will be provided with a training manual (326-pages).

