# CAD FOR ELECTRONICS

(ECET 231)

## COURSE OUTLINE

### CALENDAR DESCRIPTION

**This course introduces Altium Designer 19 (or later edition), a computer-based electronics design software for engineers/technologists to design printed circuit boards (PCB), FPGA, and embedded software, and to manage project data. Students learn how to create the electronic schematic of a circuit, how to use schematic libraries to select and design electronic components, and how to plan a printed circuit board layout. Students use these skills to create a printed circuit board from a schematic diagram.**

| | |
|---|---|
| CREDIT | 2.0 |
| IN-CLASS WORKLOAD: | 3 hours / week |
| OUT-OF-CLASS WORKLOAD: | 3 hours / week |
| PREREQUISITES | ECET 242, 260 |

---

### OBJECTIVES

- Be able to draw schematic diagrams (simple, parallel, hierarchical)
- Be able to find, select, and create schematic components (schematic library editor)
- Be able to lay out multi-layer printed circuit boards (PCB)
- Be able to find, select, and create PCB footprints (PCB library editor)
- Be able to create 3D view of a PCB
- Be able to manage the PCB project data

### EVALUATION* *[* Delay levy (labs/assignments): **-10% per day**]*

Attendance to all classes and satisfactory completion of all assignments and labs are **mandatory**. The lab grade, the theory grade, and the final exam must be over 50% to pass the course. The final grade will be determined by the following components:

- Lab exercises (13) 50%
- Midterm Exam 20%
- Final Exam 30%
- **Grading: in accordance with Camosun College Policy**

### TEXT BOOKS AND REFERENCES:

- ECET 231 Course Notes (check it with your instructor)
- Altium website *(www.altium.com)*

# TABLE OF CONTENTS

## SECTION II – Library Editors

### 4 .... Schematic Library Editor

4.1 What is the schematic library editor?
4.2 Elements and tools of the schematic library editor
4.3 Three ways to create a schematic component (electrical symbol) – ***Lab05***

### 5 .... PCB Library Editor

5.1 What is PCB library editor?
5.2 Elements and tools of the PCB library editor
5.3 Three ways to create a PCB footprint (PCB symbol) – ***Lab06***

## SECTION III – Advanced Designs, Considerations, and Data Management

### 6 .... PCB with multiple signal layers & multiple internal planes

6.1 Parallel format of a multi-channel schematic capture
6.2 Hierarchical format of a multi-channel schematic capture – ***Lab07-08***
6.3 PCB layout with two signal layers (double sided)
6.4 PCB layout with two signal layers and two internal planes (power/ground)
6.5 Split internal planes – ***Lab09-10***
6.6 PCB layout with more than two signal layers and/or two internal planes
6.7 Layer stack management

### 7 .... PCB 3D Modeling

7.1 3D modeling has changed electronics design forever
7.2 Creating 3D component bodies in a footprint library
7.3 Embedding a 3D STEP model in a footprint – ***Lab11***
7.4 3D Measuring in Altium Designer

### 8 .... PCB Fabrication Documents

8.1 Bill of materials (BOM)
8.2 Gerber/ODB++
8.3 NC drill setup
8.4 Getting the layers stack right
8.5 File export centroid
8.6 PCB Panelization
8.7 How to do “Embedded Board Array” (Panelize) -- procedure
8.8 An example from Epec Engineered Technologies

**9 .... The advanced knowledge of PCB design**

9.1 Schematic template management
9.2 Board layers and Colors management (PCB)
9.3 Track width management
9.4 Holes and vias management
9.5 Solder paste/mask management
9.6 Room & Classes management
9.7 Mechanical layers management

9.8 Grounding and de-coupling
9.9 EMI/EMC considerations
9.10 High speed signal considerations
9.11 Differential pair & length matching
9.12 PCB Power distribution network (PDN) analyzer – ***Lab12****
9.13 BGA components layout – ***Lab13****

9.14 PCB project management

**Appendix: Protel 2004/DXP shortcut keys**

# Labs

Lab #1 **Introduction to Altium Designer**

Lab #2 **Simple logic Circuit Schematic Capture**

Lab #3-4 **Single-sided PCB Layout**

Lab #5 **Schematic Library Editor** – *create schematic symbols*

Lab #6 **PCB Library Editor** – *create PCB footprints*

Lab #7-8 **Multi-sheet Schematic Design**

Lab #9-10 **Multi-layer PCB Design**

Lab #11 **3D PCB Design**

Lab #12* **PCB Power Distribution Network** (PDN) **Analyzer**

Lab #13* **BGA PCB Design** *(* when time is allowed)*

==========

# Instructor Information


| | | | |
|---|---|---|---|
| **Instructor** | John YANG | | |
| **Office hours** | TBA | | |
| **Location** | TEC 268 | | |
| **Phone** | 250-370-4213 | **Alternative:** | |
| **E-mail** | yang@camosun.bc.ca | | |
| **Website** | www.camosun.ca | | |