

# 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](http://www.altium.com))

# TABLE OF CONTENTS

## SECTION I – The Basics of Altium Designer 19

### 1 .... Introduction to Altium Designer 19

- 1.1 Review of schematic circuits and printed circuit boards
- 1.2 Introduction to the environment of Altium
- 1.3 A simple schematic circuit capture – **Lab01**

### 2 .... Schematic capture editor

- 2.1 Elements and tools of the schematic capture editor
- 2.2 10-step procedure of creating schematic captures
- 2.3 How to find desired schematic components (symbols)
- 2.4 How to install proper (schematic) libraries
- 2.5 Electrical rules setting -- for schematic captures
- 2.6 Netlist and bill of materials (BOM)
- 2.7 Project library versus general library
- 2.8 Troubleshoot schematic captures
- 2.9 A de-morgenized logic circuit capture – **Lab02**

### 3 .... PCB layout editor

- 3.1 Elements and tools of the PCB layout editor
- 3.2 10-step procedure of creating PCB layouts
- 3.3 PCB document wizard
- 3.4 How to find desired PCB components (footprints)
- 3.5 How to install proper (PCB) libraries
- 3.6 Import (netlist) changes versus Update PCB
- 3.7 Design rules setting -- for PCB layouts
- 3.8 Manual route versus auto route
- 3.9 Troubleshoot a PCB layout
- 3.10 Reshape a PCB board (re-size)
- 3.11 How to pour polygon
- 3.12 Fabrication outputs versus assembly outputs
- 3.13 A through-hole single-sided PCB design – **Lab03-04**

**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

<b>Instructor</b>	John YANG
<b>Office hours</b>	TBA
<b>Location</b>	TEC 268
<b>Phone</b>	250-370-4213 <b>Alternative:</b> _____
<b>E-mail</b>	yang@camosun.bc.ca
<b>Website</b>	www.camosun.ca