Workshop 2 - Altium Designer

In this workshop we will give you an introduction into Altium Designer, largely giving you experience around schematic capture and PCB layout.

Technical learning objectives:

· Schematic capture with Altium Designer
· PCB layout with Altium Designer
· To understand the value of circuit simulation as a part of the design cycle
· Symbol and component library management
· Bill of Material (BoM) creation and export
· Supply chain management
· Gerber file export
· Panelisation tools and techniques

Tutorials and Recommended Resources
- [Altium Documentation](#)
- [Keyboard Shortcuts](#)
- [Altium Tricks and Standards](#) **(Must Read)**
- [Creating and Optimising a Bill of Materials](#) **(Must Read)**
- [Altium Bugs and Things to Watch Out For](#)
- [Bunnie's Blog: The Factory Floor Part 1](#)
- [Bunnie's Blog: The Factory Floor Part 2](#)
- [Bunnie's Blog: The Factory Floor Part 3](#)
- [Bunnie's Blog: The Factory Floor Part 4](#)
- [PCB Design Rules Reference](#)
- [From Idea to Manufacture (Altium)](#)
- [More About Components and Libraries](#)
- [Working with Integrated Libraries](#)
- [Linking to Supplier Data](#)
- [Gerber Setup](#)

Advanced Track Resources:
- [Fundamentals of Injection Molding](#)
- [Autodesk Inventor Guided Tutorials](#)
- [3D Printing Tutorial](#)
- [PCB Panelisation Video](#)
- [Embedded Board Array Enhancements](#)

Electronic System Design Process

Electronic systems design should always be an iterative process. Just like writing code, you would never hit the debug button after you have written all 10,000 lines of code for a piece of software! In this subject, you will gain experience running through a single iteration of the electronics design process.

Note: This process might take you a whole semester during this subject, however it is possible to complete this in as little as a week if you were working on it through a full-time position. Cheap and fast access to PCB manufacturing services online means you can have a PCB design ready within an hour, export the Gerber files, pay ~$50 to a service such as PCB Wing or ITEAD Studio, and have multiple prototype PCBs ready for assembly in a week or so.

Advanced Track: If you would like to go one step further, it is always good practice to understand the environment in which your PCB will operate within. In the Recommended Resources section of this document we have included some links to learn about plastic injection moulding design considerations.

If you have a lighter subject load this semester, we suggest you download a student copy of Autodesk Inventor and design a 3D printed case for your DSO. Think about the design of the final product, where you will need to place any physical interfaces such as terminals or buttons. Perhaps you would prefer to use a non-standard PCB board shape, or perhaps you would like to split your design to contain multiple stacked PCBs using a header component.

If you are interested, you can download a student copy of Autodesk Inventor for free using your student details. The Research Bazar also run a very good introductory course on campus from time to time.

From Idea to Manufacture

Part 1: Building a Simple Astable Multivibrator PCB

Altium Designer is a powerful tool for an electronics engineer. Many students simplify the design process to the two main features of PCB design: schematic capture and PCB layout. This oversimplification misses many of the important steps of the full design process. Here is an example of a simplified design process:
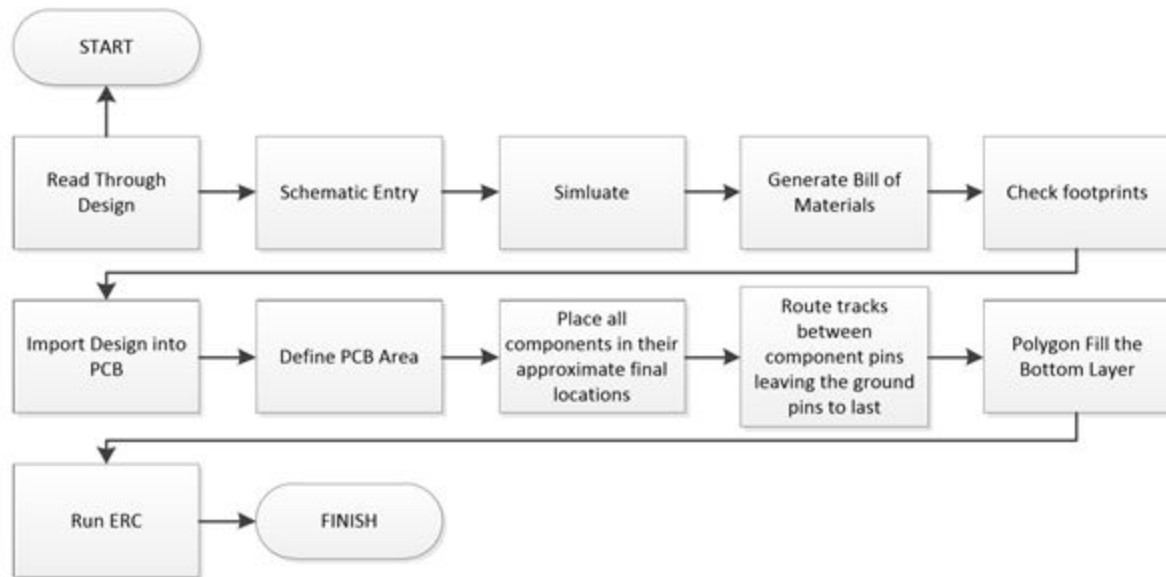
This tutorial will take you through the above steps, following the documentation and examples from Altium's website: From Idea to Manufacture (Altium)

You should attempt to complete the above tutorial within the 3 hour session, but you may find it worthwhile to take more time outside of your allocated workshop time to thoroughly gain experience with the full tutorial. This will place you well for next week's workshop where we will introduce you to the DSO project, and all the weeks that follow.

Each section of the Altium tutorial has a main article for additional background. Make sure you go through these articles thoroughly rather than skimming through the main tutorial alone.

Part 2: Library Management

Once you have completed the tutorial, you should also become familiar with creating your own schematic and component libraries, creating your own schematic symbols and component footprints from mechanical detail of a datasheet and using the IPC Compliant Footprint Wizard.

Part 3: Supplier Detail and Bill of Materials

Practice adding supplier details to the components of your schematic library, and can successfully export a Bill of Materials of every component of your Astable Multivibrator circuit design.

This Bill of Material file can be imported directly by suppliers such as Digi Key, to make ordering components much more simple.

Part 4: Bonus Information

Be sure to read through the other recommended resources as well. In particular, Bunnie's Blog is a gold mine of good information.