

## Lecture 8

# Introduction to Altium Designer IDE

### Objectives:

- Introduce Altium Designer for schematic & PCB development.
- Create a new project and generate a new schematic to add components, wire connections, net labels, supplier links, and net class blankets.
- Create / modify custom schematic symbols & component footprints.
- Create a PCB file, set its dimensions, populate it with components from a schematic design, and modify the PCB rules to aid with design.
- Discuss fabrication file (Gerber) output generation, as well as Bill of Materials (BOM) generation and 3D CAD model exporting.

### Keywords:

Schematic symbol

Component footprint

Net label

Supplier link

Net Classes (blankets)

Bill of Materials (BOM)

Printed Circuit Board (PCB)

PCB Rules

Layer Stack Manager

Gerber files (fabrication outputs)

STEP file (3D CAD model)

## Installing / Working with Altium Designer

Altium Designer is one of the industry's standard software packages for professional circuit board layout and design, developed by Altium Limited in Australia.



It consists of several design environments all merged together into a common program. Users can develop Printed Circuit Boards (PCBs), design software and systems for Field Programmable Gate Arrays (FPGAs), and also includes a compiler and development environment for embedded software development. Altium Designer can also perform circuit simulations – all from within the same program.

The core of Altium Designer has been in development since the 1980's, where it was previously named *Protel*. Altium Designer was the first PCB layout tool to incorporate 3D visualization and clearance checking of PCB assemblies.

## Starting a New PCB Project

## Adding a Schematic and Placing Parts