Altium I
(Circuit Design + Layout)

ELEC391
Spring 2018
PCB Design support for ELEC391:

Altium 2016, 150 licenses

Lecture talks:
• Jan 22 Altium I (Circuit Design + Layout)
• Mar 12 Guest Lecture – PCB Production
• Support & submission instructions posted [here](#)

Mechanical and PCB design support available 2hrs per lab session, rooms MCLD315,306

Tue & Thu : 09:00-11:00 / 12:00-14:00 / 16:00-18:00
Contents

• PCB design flow
• How to install Altium Designer 2016
• Understanding Altium Designer
• Walk-through examples
• Instructions for elec391 fabrication submissions

• PCB design best practices
• Anatomy of a PCB

Credits: Unless explicitly stated all source material is from the Altium website and Altium training documents.
Typical PCB Design flow

Front-end design and capture

- Generate CAM files
- Fabricate
- Populate

Separate tools or integrated environments

- Conceptual Idea
- System Definition
- Schematic Capture
- PCB Layout
- Debug/Test
- Final Product
- Pass
- Fail

Mixed-signal circuit simulation
Same PCB:
Altium Designer 2016

A complete product development system

System requirements (MS W7, W8, W10)

- Front-end design and capture
- Physical PCB design
- FPGA hardware design
- FPGA system implementation and debugging
- Embedded software development
- Mixed-signal circuit simulation
- Signal integrity analysis
- PCB manufacturing
How to install Altium 2016

• Link to our download site:
  https://download.ece.ubc.ca

• Useful links:
  http://www.ece.ubc.ca/~leos/pages/tools/altium.html

• Create an account at Altium Live:
  http://live.altium.com/#signin (slow)
  email: engservices@ece.ubc.ca (fast)
Install .zip file

### Install .zip file

**Altium Designer**

Circuit Design Software

**External Links**

- Altium
- Altium Designer

<table>
<thead>
<tr>
<th>File</th>
<th>Size</th>
<th>Desc.</th>
</tr>
</thead>
<tbody>
<tr>
<td>README.html</td>
<td></td>
<td>Windows installer (requires Altium License)</td>
</tr>
<tr>
<td>AltiumDesignerSetup16.exe</td>
<td>10.4 MB</td>
<td>End-User License Agreement</td>
</tr>
<tr>
<td>EULA.pdf</td>
<td>56.2 KB</td>
<td></td>
</tr>
<tr>
<td>OfflineSetupAD1619.zip</td>
<td>3 GB</td>
<td>Windows installer</td>
</tr>
</tbody>
</table>

**Summer 09 Release**

- 10
- 2014
- 16

---

**Using the ECE License Server**

The ECE license server for Altium is accessible only from the UBC network. Before starting Altium, you should be connected by one of the following means:

- A wired connection on the ECE network
- A wired connection on UBC ResNet
- A wireless connection at the UBC Vancouver campus on the ubcprivate, ubcsecure, or ubc network (ubcvisitor and eduroam are not sufficient)
- A myVPN connection to the UBC Vancouver network
- A myVPN connection to the ece.prof pool

Start Altium, and from your "My Account" page, click on "Setup private license server". Enter:

<table>
<thead>
<tr>
<th>Server name:</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Server port:</td>
<td></td>
</tr>
<tr>
<td>Secondary server name:</td>
<td></td>
</tr>
<tr>
<td>Server port:</td>
<td></td>
</tr>
</tbody>
</table>

Select the new license that appears and click on "Use". You may as well also delete any old, expired licenses that are also showing.

See file: README.html
To set license server

As per README.html file

If you loose connection to server click here:

As per README.html file
Connecting to the Altium Vault

You are not connected to any Vault Server.

To connect to a Vault, go to DXP Preferences - Data Management - Vaults settings.

To learn more about design data management, please visit http://live.altium.com/#vaults
Understanding Altium

• DXP (Design explorer): Unified platform
• Collaborative environment (corporate tool):
  – Multiple users, some with dedicated tasks
  – Design team incremental changes day-by-day
  – Built-in version control (SVN subversion or CVS concurrent versions system)
  – Design repositories / Vaults (accessible by multiple users with different credentials)
• Cloud oriented support:
  – Save preferences online
  – http://live.altium.com/ (forum, design content, blog)
Altium Design Environment

**System Menu**
Access to features including environment preferences and server information.

**Menus/Toolbars/Shortcuts**
Resources change according to the active document editor.

**Document Tabs**
Each open document has its own tab. Click on a document tab to make it the active document. Right-click on a tab for further controls.

**Workspace Panels**
Various panels provide functionality specific to a particular editor, or at a system level. Panels can be docked, placed in a ‘pop-out’ mode, or floating.

**Navigation**
Provides controls for jumping to a particular document, stepping back and forth through viewed documents, and accessing the Home page.

**Main Design Window**
Display and arrange open documents in this window.

**Panel Access**
Workspace panels are accessible using these buttons.
Recommended basic panels

For more help working with panels read [this](#)
Understanding Altium
(Basics for the single user)

Don't forget:

- Use Keyboard shortcuts
  <Shift + F1> while running a command
- <Esc> or Right Click to exit a command
- Save documents to see some changes take effect
Understanding Altium
(Basics for the single user)

• Projects (project panel, active project)

• Workspace Panels (system-wide, editor-specific)

• Editors:
  – Schematic
    • Symbol editor
  – PCB layout
    • Footprint editor
    • CAM files (CAMtastic panel)

• Components and Libraries
Altium Projects

• Project: collection of design documents
  – 1 Project = 1 implementation
  – It stores links to all source documents
    • relative reference: same drive
    • absolute reference: different drive
  – It creates links to all output documents
  – Saves project options

• Create a PCB_Project, Save as: new name
  (does not move the file creates a copy)
• The active project is highlighted
• Add/Remove documents to/from a project
Altium Projects: types

- PCB Project (*.PrjPcb)
  - Schematic, libraries, PCB layout
- FPGA Project (*.PrjFpg)
- Embedded Project (*.PrjEmb)
- Core Project (*.PrjCor)
- Integrated Library (*.LibPkg) & (*.IntLib)
- Script Project (*.PrjScr)
Component, Model and Library Concepts

- Component representations:
  - Schematic **symbol**
  - PCB **footprint**
  - SPICE model definitions
  - Signal integrity description
  - 3D graphical description

```plaintext
***********INPUT STAGE***********
* I0 2 1 25.0P
^Input offset current
CI1 1 0 3P
CI2 2 0 3P
R1 1 3 1E12
R2 3 2 1E12
I1 99 4 1.0M
J1 5 2 4 3X
J2 6 7 4 3X
R3 5 50 650
R4 6 50 650
*Fp2=28 MHz
S1 5 6 12739
```
Component, Model and Library Concepts

• Domains = Different phases of design
  – Schematic capture
  – PCB layout (2D / 3D)
  – SPICE simulation
  – Signal integrity analysis

• A unified component is a container with links to all domain models + parametric information
Libraries = collection of components

- Collection of components, models or both
- Model Libraries (*.MDL, *.CKT, *.PCBLib)
  - Simulation models are one file per model
- Schematic Libraries (*.SchLib)
  - Symbol and a link to a model library
- Integrated Libraries (*.IntLib)
  - Symbol, footprint and other models are compiled into a single portable file
Project: part of and available only to the active project and its documents. You have to keep track of where these are if you move the project files.

Installed: All installed libraries. Components are available to all open projects and list is persistent across design sessions.

Search Path: Additional Libraries accessible via a search path and subfolders. The search paths are valid for the active project.
Libraries Panel:

All libraries available to the active project

Project + Installed + Search Path

When placing component:

- <spacebar> to rotate
- <x> or <y> to flip
- <Tab> open properties dialog
- <L> for PCB footprints to flip component side

To search across libraries:

Search …
Obtaining integrated libraries

1. **Frozen (legacy) libraries:** From here you can install anywhere but it is a good idea to make a subfolder under:
   C:\Users\Public\Public Documents\Altium\AD16\Library or a cloud storage service if you work from more than one PC

2. **AltiumLive website:** Resources / Design Content

   ![National Semiconductor](image)

   Manufacturer: National Semiconductor
   Updated: 3+ months ago
   Tags: Analog, Amplifier

   National Semiconductor Amplifiers. This collection offers amplifiers from single to quad, up to 1.7GHz with low-distortion, low-power and low-voltage options.

   - This downloads a .zip file for the complete library
   - This is useful to preview component
Altium Vault

Altium’s cloud library (repository of models)
Also includes real-time supply chain information
Learning to use Altium

Best training material is on the Altium website. It is updated, but beware that menus and options slightly change between versions.
Walk-through examples

L298 Motor Driver Board
(by Matt Winship)

Encoder Wheel
(by Rob Hais)

L298_Motor_Driver_Board_Datasheet.pdf

Encoder/Layout Tutorial.pdf
LM298 Motor Driver Board Schematic

C-Block3
PartsDatabaseV16.1.IntLib

C-Hdr1x5
PartsDatabaseV16.1.IntLib

Q-LN298HS
Additional Parts 391.IntLib

C-Block2
PartsDatabaseV16.1.IntLib

G-Cap
PartsDatabaseV16.1.IntLib

T-DiodeRect
PartsDatabaseV16.1.IntLib
1. Load libraries
2. Draw the schematic
   Set electrical type for connector pins
3. Compile Project:
   Project → Project Options
4. Place no ERC labels if necessary
   Modify connection matrix with caution
Wiring Tips

- Left-click or <Enter> to anchor the wire at the cursor position.
- <Backspace> (←) to remove the last anchor point.
- <Spacebar> to toggle the direction of the corner.
- <Shift+Spacebar> to cycle through all possible corner modes.
- Right-click or <Esc> to exit wire placement mode.
- To graphically edit the shape of a wire, Click once to select it first, then Click and hold on a segment or vertex to move it.
- Whenever a wire crosses the connection point of a component, or is terminated on another wire, a junction will automatically be created.
- A wire that crosses the end of a pin will connect to that pin, even if you delete the junction.
- To move a placed component and drag connected wires with it, hold down the Ctrl key while moving the component, or select Move » Drag.
LM298 Motor Driver Board Layout

- Size 1.2” x 2”
- 2 layers
- Mounting holes
- Thick traces for V- and V+
- Power planes for Vs and GND
2 starting points for PCB design

1. From a companion schematic package
   – Prepare project schematics
   – Import schematic design
   – Component footprints are added automatically
   – Connectivity is indicated with rats nests
   – Net names are imported from the schematic

2. Directly from the PCB editor
   – You need to select and place manually each component footprint from a library
   – No rats nest – connectivity
   – You must assign nets manually (at least GND)
Creating a New Board from a template

- **Files Panel:**
  - New from template
  - select the A4.PcbDoc template
  - Save as ... same name and directory as SchDoc file
Adding PCB file to project

• Make the PCB board part of the project
• Rename the file
• Save the PcbDoc file and the project
First things first … choosing working units

- **Imperial (inches)**
  - $1/1000^{th}$ of an inch = 1 mil = 1 thou
  - 100mils (0.1”) is a common dimension

- **Metric (mm)**
  - 1 mm ≠ 1 mil!
  - Common unit in SM parts

- Remember: 100mils = 2.54mm
- To switch units in Altium Press <Q>
First things first … setting the snap grid

- PCBs are grid based objects

Unified Cursor-Snap System

- Selecting a suitable snap grid:
  - <Ctrl>+<G>
  - Start with a coarse grid to define board size
First things first ... redefining the board shape

- Viewing modes:
  - Board Planning Mode (1)
    - Design » Edit Board Shape (resize to 1.5” x 1.5”)
    - Design >> Move Board Shape (Relocate the origin)
  - 2D Layout Mode (2)
  - 3D Layout Mode (3).
First things first … setting the board origin

• Absolute origin (lower left corner)
• User-defined relative origin
  • Edit >> Origin >> Set
Design transfer

- Design transfer
  - On Schematic file
    - Design >> Update PCB Document …
Design transfer

Rat’s nests indicate connectivity as per schematic (Net names are assigned to part terminals)

All parts in the schematic with their selected footprints
Configuring the Display Layers

- Design » Board Layers and Colors
Configuring the Display Layers

- **Electrical layers**
  32 signal layers and 16 internal power plane layers.

- **Mechanical layers**
  32 general purpose mechanical layers, used for design tasks such as dimensions, fabrication details, assembly instructions, or special purpose tasks such as glue dot layers. These layers can be selectively included in print and Gerber output generation. They can also be paired, meaning that objects placed on one of the paired layers in the library editor, will flip to the other layer in the pair when the component is flipped to the bottom side of the board.

- **Special layers**
  these include the top and bottom silkscreen layers, the solder and paste mask layers, drill layers, the Keep-Out layer (used to define the electrical boundaries), the multilayer (used for multilayer pads and vias), the connection layer, DRC error layer, grid layers, hole layers, and other display-type layers.
Positioning components & routing

Place a plane on top for GND
Place a plane bottom for Vs
Handy shortcuts for routing

• Press * on the numeric keypad while routing to cycle through the available signal layers. A via will automatically be added, in accordance with the applicable Routing Via Style design rule. Alternatively, use **Ctrl+Shift+Roll** shortcuts to move back and forth through the available signal layers.

• **Shift+R** to cycle through the enabled conflict resolution modes, including Push, Walkaround, Hug and Push, and Ignore. Enable the required modes in the **PCB Editor - Interactive Routing** page of the **Preferences** dialog.

• **Shift+S** to cycle single layer mode on and off, ideal when there are many objects on multiple layers.

• **Spacebar** to toggle the corner direction (for all but any angle mode).

• **Shift+Spacebar** to cycle through the various track corner modes. The styles are: any angle, 45°, 45° with arc, 90° and 90° with arc. There is an option to limit this to 45° and 90° in the **PCB Editor - Interactive Routing** page of the **Preferences** dialog.
# Design Rules

- **Design >> Rules**

<table>
<thead>
<tr>
<th>Rule</th>
<th>Constrain</th>
<th>Query</th>
</tr>
</thead>
<tbody>
<tr>
<td>Electrical, Clearance</td>
<td>Min clearance = 7mil</td>
<td>All</td>
</tr>
<tr>
<td>Routing, Width*</td>
<td>Min width = 7mils Max width = 500mils</td>
<td>All</td>
</tr>
<tr>
<td></td>
<td>Preferred = 10mils</td>
<td></td>
</tr>
<tr>
<td>Routing, Width_IO</td>
<td>Min width = 7mils Max width = 500mils</td>
<td>Advanced (Query) (InNet('V+')) OR InNet('V-'))</td>
</tr>
<tr>
<td>Width_Vss</td>
<td>Min width = 7mils Max width = 500mils</td>
<td>Net Vss</td>
</tr>
<tr>
<td></td>
<td>Preferred = 20mils</td>
<td></td>
</tr>
</tbody>
</table>
Custom Routing design rules

Rename to “Width_IO’”
Use ‘Custom Query’ to set”
Belongs to net V+
OR
Belongs to V-
Set rule execution priority
Auto route

- Tools » Un-Route » All
- Auto Route » All

- You can also set single layer routing
Submission Instructions

• No limits to PCB submissions (7 deadlines)
• Cost: $25 + $10/ sq-in, from project budget
• Submission dates: Midnight, Mondays Jan 29, Feb 5-12-26, Mar 5-12-19 we will check submissions and accept fixes until 5PM the following Tuesday
• Turn around: 5-6 business days
• Work within the given guidelines
• Verify PCB layout and design - prior to design submission
• Submissions will be rejected if guidelines are not followed
We will panelize your designs to speed up fabrication and reduce costs.
Submission Instructions

• You can send several different boards per submission.
• You can request several copies of each but that increases your area.
• Email pcb@ece.ubc.ca
  Subject: [PCB] ELEC391, Group #, submission#
• Attach: Zipped file with your PCB Project file (*.PrjPcb) and all associated files, also include the latest DRC report. (make sure all files are under the same directory)

Body:
Total number of boards to fabricate:
Name of boards to fabricate and number of copies for each
Design constraints

1) Layers:
   1) Maximum number of electrical layers = 2
   2) Bottom overlay (PCB underside text) will not be manufactured - please use "bottom layer" for bottom text

2) Try to minimize the size of your PCB
   Components can be placed side by side (recommend 50-100 mil IC's separation for most cases)

3) Do not forget to:
   Add your group number on the top overlay – make it visible
   Draw a board outline on Mechanical 1
   if several boards in a single file, draw a board outline for each
   (min spacing from edge of board for any feature is 10mils)
Design constrains

4) Use latest version of course component library available here
   http://www.ece.ubc.ca/~eng-services/files/courses/elec391-spring2018/pcb-design/

5) Using other libraries
   If you do, make sure parts will pass Design-Rule-Checking
   These two Altium libraries contain useful parts:
   -Miscellaneous Devices.IntLib
   -Miscellaneous Connectors.IntLib

6) Install provided Design-Rules file - please do not modify base rules, but you can add custom routing rules.

Submissions that do not pass DRC will be rejected
Rules and Checks

1. Update Schematics in All_components_test.PrjPcb
   Import Changes From All_components_test.PrjPcb

2. Rules...
   Rule Wizard...
   Board Shape

3. Design Rule Check...
   Reset Error Markers
   Browse Violations
   Browse Objects
Rules – design rules

• DRC file available [here](http://www.ece.ubc.ca/~eng-services/files/courses/elec391-spring2018/pcb-design/)

• Download and save as “.RUL” file

• On your PCB design select: Design >> Rules

• On the 'PCB Rules and Constrains Editor', Right click anywhere on the left column
  – Select: Import Rules
  – Select all rules in window (using shift and mouse) → OK
  – Browse to select .RUL file
  – Clear existing rules prior to import? → NO
Rules – design rules

- Component clearance and (electrical) clearance:
  - Minimum distance = 7 mil

- (Routing) width:
  - Minimum trace width = 7 mil

- Annular ring size:
  - Minimum annular ring size = 7 mil
  - Minimum annular ring size for vias = 5 mil

- Board outline clearance: 10mils

- Limited set of allowed hole sizes
### Rules – hole sizes

- Pre-selected hole and drill sizes: non-plated vs. plated sizes

<table>
<thead>
<tr>
<th>Drill Number Set</th>
<th>Drill Size</th>
<th>Finished Size</th>
<th>Approximate Use</th>
</tr>
</thead>
<tbody>
<tr>
<td>#76</td>
<td>.020&quot;</td>
<td>.017&quot;</td>
<td>via holes</td>
</tr>
<tr>
<td>#70</td>
<td>.028&quot;</td>
<td>.025&quot;</td>
<td>via holes, fine lead devices such as trim pots etc.</td>
</tr>
<tr>
<td>#65</td>
<td>.035&quot;</td>
<td>.032&quot;</td>
<td>IC's, 1/4 watt resistors, small diodes, ripple caps etc.</td>
</tr>
<tr>
<td>#62</td>
<td>.038&quot;</td>
<td>.035&quot;</td>
<td>Square posted pins that measure .025&quot; on the flat.</td>
</tr>
<tr>
<td>#58</td>
<td>.042&quot;</td>
<td>.039&quot;</td>
<td>TO-220 packages, IDC type square posted headers, 1/2 watt resistors, 1N9000 series diodes, IC chip carriers, etc.</td>
</tr>
<tr>
<td>#55</td>
<td>.052&quot;</td>
<td>.049&quot;</td>
<td>larger connectors, transformer leads, etc.</td>
</tr>
<tr>
<td>#53</td>
<td>.060&quot;</td>
<td>.057&quot;</td>
<td>similar to .052&quot; above</td>
</tr>
<tr>
<td>#44</td>
<td>.086&quot;</td>
<td>.083&quot;</td>
<td>TO-220 mounting holes, screw holes, general mounting</td>
</tr>
<tr>
<td>1/8 in.</td>
<td>.125&quot;</td>
<td>.122&quot;</td>
<td>mounting holes</td>
</tr>
<tr>
<td>#24</td>
<td>.152&quot;</td>
<td>.149&quot;</td>
<td>mounting holes</td>
</tr>
</tbody>
</table>
PCB Hole Size Editor

1. Select the PCB Editor.
2. Navigate to the Hole Size Editor.
3. Choose the desired condition and primitive type.
4. Set the hole size and select the rounding option.
PCB Design Best Practices
Best Practices: Estimating board size

• Before starting layout it is good to have an idea of the target size of the PCB board and all other relevant dimensions.

• It is very helpful to have the components at hand to plan the floor-plan.

• An old good trick of the trade is to print the PCB layout at a 1:1 scale, place the printout on a foam and stick on the through hole components.
Best Practices: Floor planning

- Choose your units and set the grid
- Carefully plan the placement of components
  - Place analog and digital sections apart
  - Group components into ‘functional blocks’
  - Place ICs in the same direction
  - Align ICs, resistors, labels, capacitors etc.
  - Place de-cap close by their ICs
  - Place Op-amp resistors near the Op-amp
  - Plan for mounting holes and heat sinks
- Aim for symmetry when possible
- Do use Design Rule check
Best Practices: Routing strategy

- On two sided boards keep traces perpendicular as much as possible
- Avoid 90 degree bends in tracks (?) (reduced chances of acid traps)
- Keep traces a short as possible
- Always connect a trace to the center of the pad
- Use teardrops (Tools >> tear drops), and use vias to avoid lockout
- Do not place vias under SMD pads
- Layout first all critical traces
  - e.g. CLK, diff pairs, controlled length
- Polygons as fills:
  - Connect to GND (EMC), or do not leave ‘dead copper’
- Rout nicely

[Ref 3]
Best Practices: Labelling

- Always sign your design: add date, version, and name of board
- Label all relevant inputs and outputs
- Default sizes for comments and designators are 60mils x 10mils
- If you have silkscreen on both sides add a 'TOP' label to the top overlay.
Best Practices: Finishing touches

• Add mounting holes
• Run: Reports >> Board Information
  – Board specification → to confirm board size
  – Non-plated hole size
  – Plated hole size
• Using the hole size editor:
  – Minimize the total number of holes sizes
  – Verify that all vias are the same size (if possible)
• Verify that there are no unwanted leftovers on any Mechanical layer
Online resources

1. Ten best practices of PCB design – EDN Magazine, Edwin Robledo & Mark Toth

2. Circuit Board Layout Techniques – Texas Instruments, Chapter 17 of Op-amps for everyone

3. PCB Design Tutorial – David L. Jones
PCB Anatomy: Substrate

• **Substrate (laminate)**
  - Rigid board of insulating material
  - Provides structural support to the circuit components
  - Most commonly used material type is FR4, 62-63mils thick
  - Laminates are available in different thicknesses

![Diagram of PCB Substrate](image)

*Copper cladding*

---

Cu thickness measured in weight oz/ft²
- ½ oz → 0.7mils
- 1 oz → 1.4mils
- 2 oz → 2.8mils

1mil = 25μm
PCB Anatomy: Layer Stackup

Design >> Layer Stack Manager ...

Layer Stack Manager

<table>
<thead>
<tr>
<th>Layer Name</th>
<th>Type</th>
<th>Material</th>
<th>Thickness (mil)</th>
<th>Dielectric Material</th>
<th>Dielectric Constant</th>
<th>Pullback (mil)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top Overlay</td>
<td>Overlay</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Top Solder</td>
<td>Solder Mask/Co..</td>
<td>Surface Material</td>
<td>0.4</td>
<td>Solder Resist</td>
<td>3.5</td>
<td></td>
</tr>
<tr>
<td>Top Layer</td>
<td>Signal</td>
<td>Copper</td>
<td>1.4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Dielectric1</td>
<td>Dielectric</td>
<td>None</td>
<td>62</td>
<td>FR-4</td>
<td>4.8</td>
<td></td>
</tr>
<tr>
<td>Bottom Layer</td>
<td>Signal</td>
<td>Copper</td>
<td>1.4</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Bottom Solder</td>
<td>Solder Mask/Co..</td>
<td>Surface Material</td>
<td>0.4</td>
<td>Solder Resist</td>
<td>3.5</td>
<td></td>
</tr>
<tr>
<td>Bottom Overlay</td>
<td>Overlay</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Total Thickness: 85.6mil

Advanced >>
PCB Anatomy: Traces / Tracks

- Copper traces are patterned either by:
  - Photolithography: requires photomasks
  - Laser: used to draw patterns on photoresist
  - Mechanical milling: Cu is removed to isolate the traces.

- Trace width and thickness determines:
  - Ampacity (current carrying capacity)
  - Characteristic impedance for RF designs

- Practical limitations:
  - Minimum trace width and gap

Negative view:
Copper planes, Drill holes, Solder Masks

Figure 1-11  Copper pad and trace after etching and resist stripping.

Figure 1-12  A mechanically milled trace.

Ref [B1]
PCB Anatomy: Trace width

Use the following online trace width calculator:
http://circuitcalculator.com/wordpress/2006/01/31/pcb-trace-width-calculator
PCB Anatomy: Vias

- Connection between layers is accomplished with via holes.
- After the holes are drilled, their inner walls are plated.
- Top and bottom traces are patterned after plating.

Thermal relief is needed when connecting a via to a copper plane.

PWR and GND planes are commonly inner layers.

Teardrops:

```
Figure 1-13: Holes are drilled into the board and then copper plated. (a) A nonplated through-hole. (b) A plated through-hole.
```

```
Figure 1-14: A connection to a plane layer through a thermal relief.
```

```
Figure 1-15: A clearance area provides isolation between a plated hole and a plane.
```
PCB Anatomy: Vias

- Types of via holes:
  - Plated and un-plated through-hole, blind, buried

Figure 1-5 A built-up, multitechnology, PCB stack-up.

Ref [B1]
PCB Anatomy: Holes

Holes can be:

- Vias, multi-layer pads, mounting holes, or cuts
- Plated or non plated

You must specify whether a hole is plated or non plated during the design process.

Plating reduces hole size by 0.003”
PCB Anatomy: Pads

- Pads: contact areas for soldering components, test points, and solder traps
- Pads can have any shape
- Single layer pads: Top/bottom layer, common for SMT, end launch connectors
- Multi-layer pads: for through hole components
- Footprints are a collection of pads

Figure 5-7 Footprint dimensions (typical convention).
Figure 5-12 Radial-led through-hole device. (a) Axial-led capacitor. (b) Layout axial footprint
PCB Anatomy: Solder mask

- Solder mask or solder resist:
  - Thin polymer layer deposited on top and bottom layers
  - Protects outer layers from oxidation and prevents solder bridges
  - Allows for wave or reflow soldering of components
  - Holes are opened with photolithography wherever components will be soldered
  - Default color is green, but any other color is possible

Source: Printed Circuit Board Basics: An Introduction to the PCB Industry, by: Michael Flatt
PCB Anatomy: Legend / Silkscreen / Overlay

- Legend or silkscreen:
  - Applied on top of the solder resist
  - Can be applied to one or both outer layers
  - Default color is white but any other color is possible

Tip: add (Top) and (Bottom)

*Figure 1-16: Final layers are the soldermask (green) and silk screen (white).*
PCB Anatomy: Solder coat / thinning

Solder coated + solder mask

Bare copper, no solder mask

Solder coated + no solder mask
PCB Anatomy: Mechanical Layers

• Multi-purpose layers
• E.g. Altium supports 32 Mechanical layers: M1 … M32
• Typically
  – M1  Board outline
  – M2  PCB manufacturing info
  – M11-M12 Top and bottom layer dimensions
  – M13 Top layer 3D models and mechanical outlines
  – M14 Bottom layer 3D models and mechanical outlines
  – M15 Top layer assembly information
  – M16 Bottom layer assembly information