Altium I
(Circuit Design + Layout)

ELEC391
Summer T1 2018
Contents

• PCB Design support for ELEC391
• PCB design flow
• How to install Altium Designer 2016
• Understanding Altium Designer

• Walk-through example

• PCB design best practices
• Anatomy of a PCB

Credits: Unless explicitly stated all source material is from the Altium website and Altium training documents.
PCB Design support for ELEC391:

Altium 2016, 150 licenses

- Jun 4 Altium I (Circuit Design + Layout)
- PCB Submissions Jun 10
We will panelize your designs to speed up fabrication and reduce costs.
Submission Instructions

• FR4 62mils, 7mils/7mils, 2-layers, top overlay only.
• Designs must pass DRC
• 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: *PcbDoc files only
  Body:
  Total number of designs (not copies) to fabricate
  Name of designs to fabricate and number of copies for each
Rules and Checks
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
Typical PCB Design flow

Separate tools or integrated environments

Front-end design and capture

- Generate CAM files
- Fabricate
- Populate

Mixed-signal circuit simulation

- Schematic Capture
- PCB Layout
- Debug/Test
- Final Product
- Pass
- Fail

http://e2e.ti.com/
PCBs are multi-layer objects
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

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

**ALTUM DESIGNER**
Circuit Design Software

**External Links**
- Altium
- Altium Designer

<table>
<thead>
<tr>
<th>File</th>
<th>Size</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>README.html</td>
<td></td>
<td>Windows installer (requires</td>
</tr>
<tr>
<td>AltiumDesigner16Setup.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>Windows installer</td>
</tr>
<tr>
<td>OfflineSetupAD16 1 9.zip</td>
<td>3 GB</td>
<td></td>
</tr>
</tbody>
</table>

**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>
</tbody>
</table>

Secondary server name:                  Server port:

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

1. Admin

License Management

2. Setup private license server

If you lose connection to server click here:

3. See file: README.html

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

Cloud Preferences
Your settings can now be stored in the cloud. Once you are signed in simply enable your cloud preferences.

Data Management – Vaults

<table>
<thead>
<tr>
<th>Name</th>
<th>Description</th>
<th>Address</th>
<th>Status</th>
<th>Enabled</th>
</tr>
</thead>
<tbody>
<tr>
<td>Altium Content Vault</td>
<td>Altium Content Vault</td>
<td><a href="http://vault.live.altium.com">http://vault.live.altium.com</a></td>
<td>✔️</td>
<td>✔️</td>
</tr>
</tbody>
</table>
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/](http://live.altium.com/) (forum, design content, blog)
Altium Design Environment

- **System Menu**: Access to features including environment preferences and server information.
- **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.
- **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.
- **Menus/Toolbars/Shortcuts**: Resources change according to the active document editor.
- **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
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
Project types

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

- **Component representations:**
  - Schematic symbol
  - PCB footprint
  - SPICE model definitions
  - Signal integrity description
  - 3D graphical description
Libraries = collections 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)
  – Unified components: Symbol, footprint and other
domain models + parametric information are compiled
into a single portable file
To setup libraries in Altium

Libraries panel

Available Libraries Dialog
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 sub-folders. 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

   ![Image of National Semiconductor](image.jpg)

   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.

   - **GO TO VAULT**
   - **DOWNLOAD LIBRARY**

   This is useful to preview component

   This downloads a .zip file for the complete library
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 example

L298 Motor Driver Board
(by Matt Winship)

L298_Motor_Driver_Board_Datasheet.pdf

Altium introductory tutorial
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
LM298 Motor Driver Board Schematic

C-Block3
PartsDatabaseV16.1.IntLib

C-Hdr1x5
PartsDatabaseV16.1.IntLib

C-Block2
PartsDatabaseV16.1.IntLib

Q-LN298HS
Additional Parts 391.IntLib

G-Cap
PartsDatabaseV16.1.IntLib

T-DiodeRect
PartsDatabaseV16.1.IntLib
Drawing the schematic

You can select among different GND symbols

You need to input the capacitance value
Set the electrical type of pins

On the Q-LN298HS Symbol, pins Direct1, Direct2, and Enable are inputs. You need to set the pins of connector P2 to Output, or I/O, to provide a compatible electrical type net.

Unlock pins

Select pin, change electrical type

Graphical

Location X: 5250mil
Orientation: 180 Degrees
Mode: Normal

Logical

Display Name: 4
Designator: 4
Electrical Type: I/O

Parameters

Part Number: 1
Compile Project

- Project >> Compile Project

- This error is caused by having 2 pins of the LM298 connected to V- and 2 pins connected to V+

- In this case this was done intentionally

- To ignore this error place a No ERC label on the net
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 2.1”
- 2 layers
- Mounting holes
- Thick traces for V- and V+
- Power planes for Vs and GND

Top Metal

Bottom Metal
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 ≠ 1mil!
  - 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 1.2” x 2.1”)
    – 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

View Configurations

Select PCB View Configuration

- Altium Standard 2D
- Altium Transparent 2D
- Tutorial
- Altium 3D Black
- Altium 3D Blue
- Altium 3D Brown
- Altium 3D Color By Layer
- Altium 3D Dk Green
- Altium 3D Lt Green
- Altium 3D Red
- Altium 3D White

Path:
C:\Users\robfor\AppData\Roaming\Altium\Altium Designer\24F6E6A8-3DCC-412A-808C-0366588BC9A6\View Configurations\Altium Standard 2D\config_2dsimple

Description:
Altium Standard 2D

Actions:
Create new view configuration...
Save view configuration...
Save As view configuration...
Load view configuration...
Rename view configuration...
Remove view configuration...

Board Layers And Colors

- Signal Layers (5)
  - Top Layer (T)
  - Bottom Layer (B)
- Internal Planes (P)
  - All On
  - All Off
  - Used On

Show / Hide

- All On
- All Off
- Used On

View Options

- All On
- All Off
- Used On

Transparency

- All On
- All Off
- Used On

Mechanical Layers (M)

- Mechanical 1
- Mechanical 2
- Mechanical 3
- Mechanical 4
- Mechanical 5
- Mechanical 6

Show

- All On
- All Off
- Enable
- Single Layer Mode
- Linked To Sheet

Path:
C:\Users\robfor\AppData\Roaming\Altium\Altium Designer\24F6E6A8-3DCC-412A-808C-0366588BC9A6\View Configurations\Altium Standard 2D\config_2dsimple
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

122mils mounting holes
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, Preferred = 10mils</td>
<td>All</td>
</tr>
<tr>
<td>Routing, Width_IO</td>
<td>Min width = 7mils, Max width = 500mils, Preferred = 100mils</td>
<td>Advanced (Query) (InNet('V+') OR InNet('V-'))</td>
</tr>
<tr>
<td>Width_Vss</td>
<td>Min width = 7mils, Max width = 500mils, Preferred = 20mils</td>
<td>Net Vss</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
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.

http://circuitcellar.com/cc-blog/the-perfect-pcb-prototype/
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-caps 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

Background: Apple Macintosh PCB from http://www.digibarn.com/collections
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 as 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
Anatomy of a PCB

PCB Anatomy: Laminate

- Laminate (substrate)
  - Rigid board of insulating material
  - Available in different thicknesses & materials
  - Covered with copper foil or cladding
  - Provides structural support and insulation to circuit components
  - Most commonly used material type is FR4, 62-63mils (1.6mm) thick

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

1mil = 25μm
PCB Anatomy: Laminate

Copper foil

Prepeg

Copper clad

Common glass weaves

1080

2116

7628
PCB Anatomy: Layer Stack-up

Ref [B1]  Figure 1-3  Cores and prepreg.

Design >> Layer Stack Manager …
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

Manufacturing limitations:
- Minimum trace width and gap (e.g. 7/7)

Negative view:
Copper planes, Drill holes, Solder Masks

Teardrops:
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

**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.

PWR and GND planes are commonly inner layers

Ref [B1]
Source: wikipedia.org: Thermal pad
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.*
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).](image1)

![Figure 5-12 Radial-leded through-hole device. (a) Axial-leded capacitor. (b) Layout axial footprint](image2)

Ref [B1]
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

Illustration ML-14. Apply solder resist. The specified resist (either dry film, liquid photoinageable, or screen printed) is applied to the surfaces of the PCB or panel.

Illustration ML-15. Solder coat. Solder (tin/lead) is applied to the exposed copper areas, and the excess solder is removed.

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: Surface finishing / thinning

There are different types of finishes, eg:

HASL (tin), ENIG (Nickel and Gold), Silver immersion
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
PCB Basic Terminology

- **Laminate / Substrate**
  - Material (FR4, Rogers)
  - Thickness (62-63mil, 16mm)
  - Prepeg, Core, Foil, Clad
  - Stack-up

- **Traces / tracks**
  - Width & gap
  - Ampacity
  - Characteristic impedance

- **Vias & Hole**
  - Types of vias
  - Plated vs non-plated holes
  - Thermal relief
  - Annular ring

- **Pads**
  - Multi-layer, single layer
  - Tear drop

- **Components / Parts**
  - Footprints
  - Symbols
  - Libraries

- **Layers**
  - Mechanical, board outline
  - Top metal, Bottom metal
  - Inner layers / Planes
  - Top / Bottom Solder Mask
  - Legend / Silkscreen / Top overlay
  - Top / Bottom pads
  - Multi-layer

- **Surface Finish**
  - HASL, ENIG

- **Fabrication CAM files**
  - Gerber, 274X
  - Excellon NC drill files