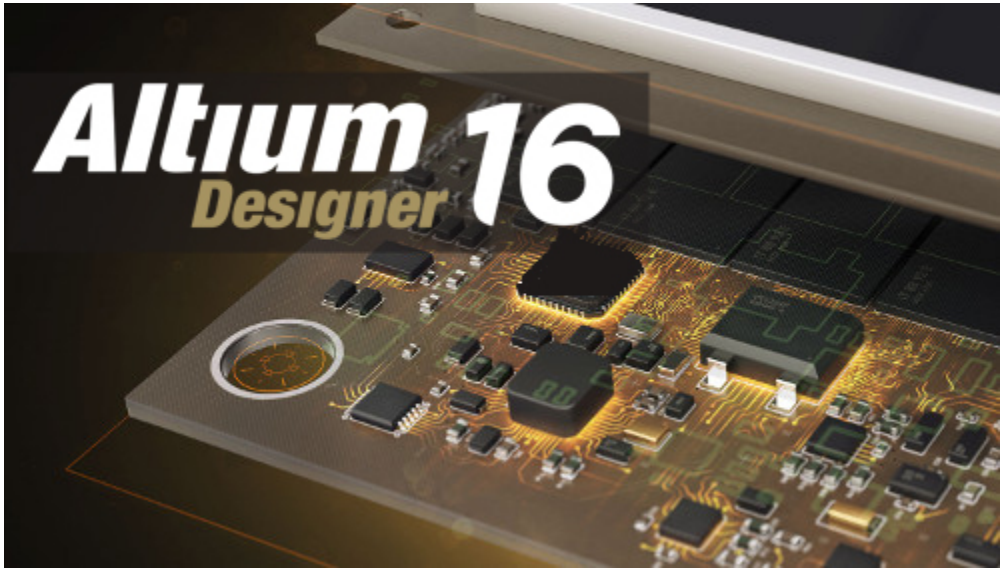


Altium I

(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

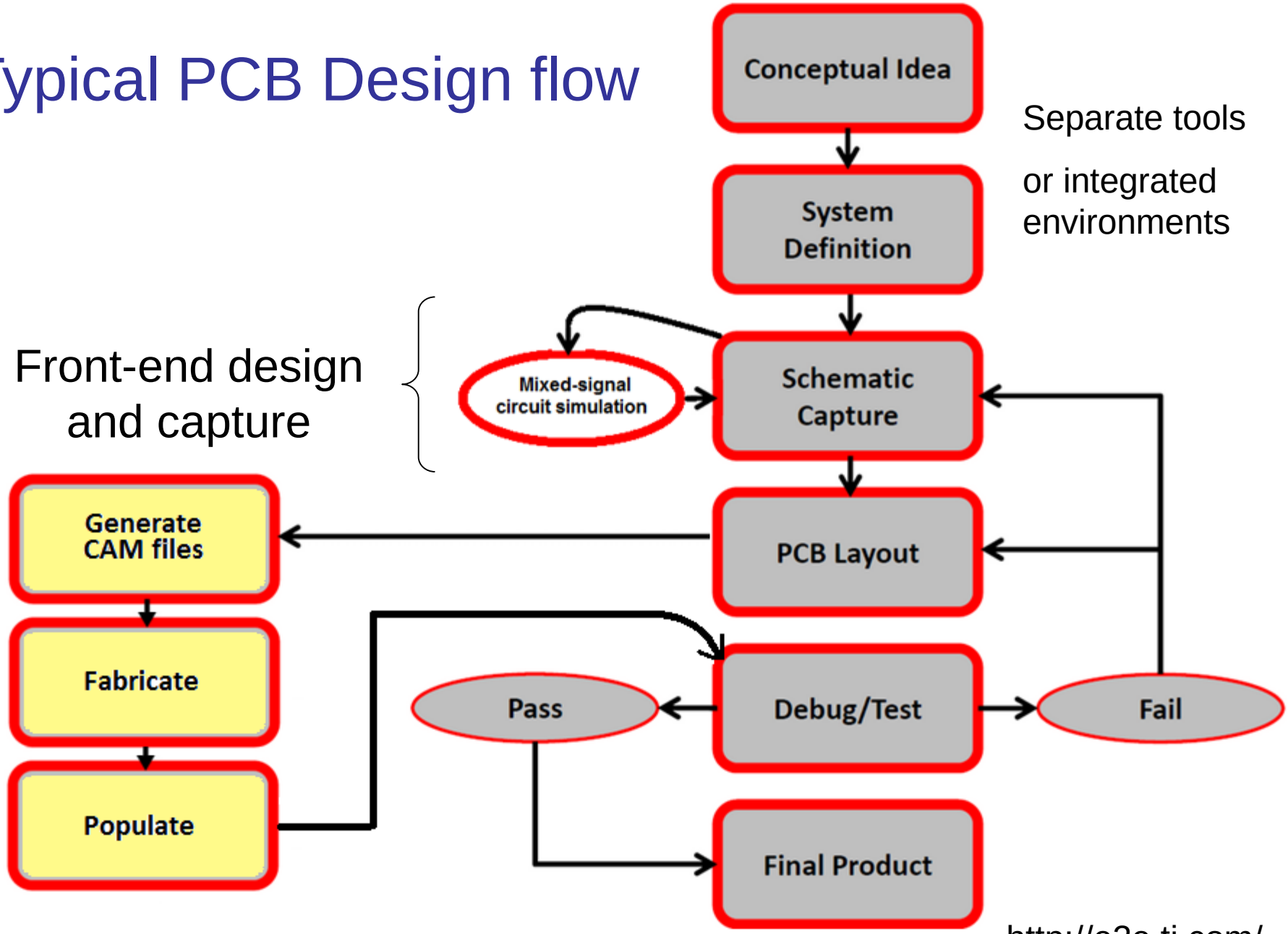
Contents

- 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

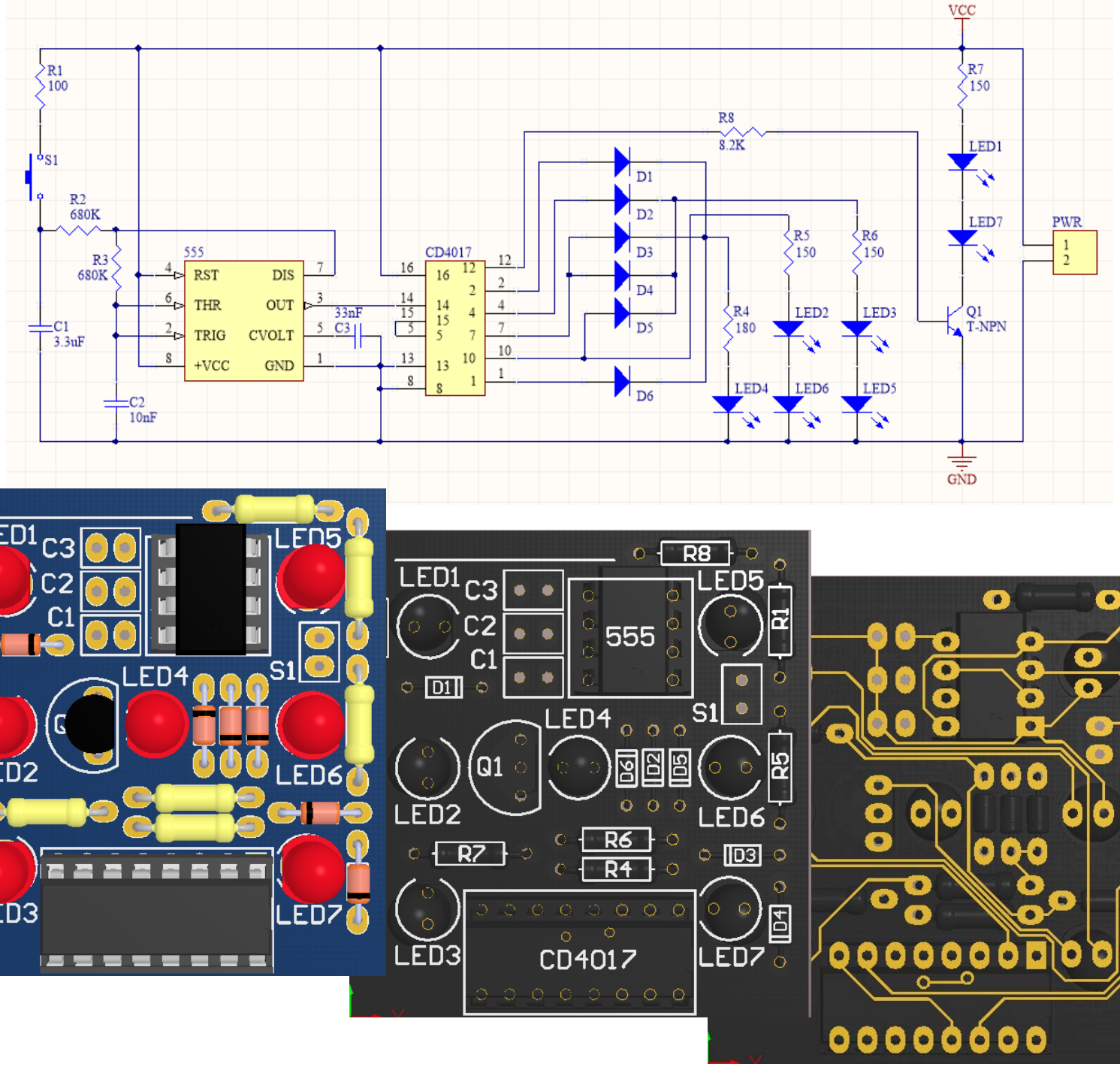


Separate tools

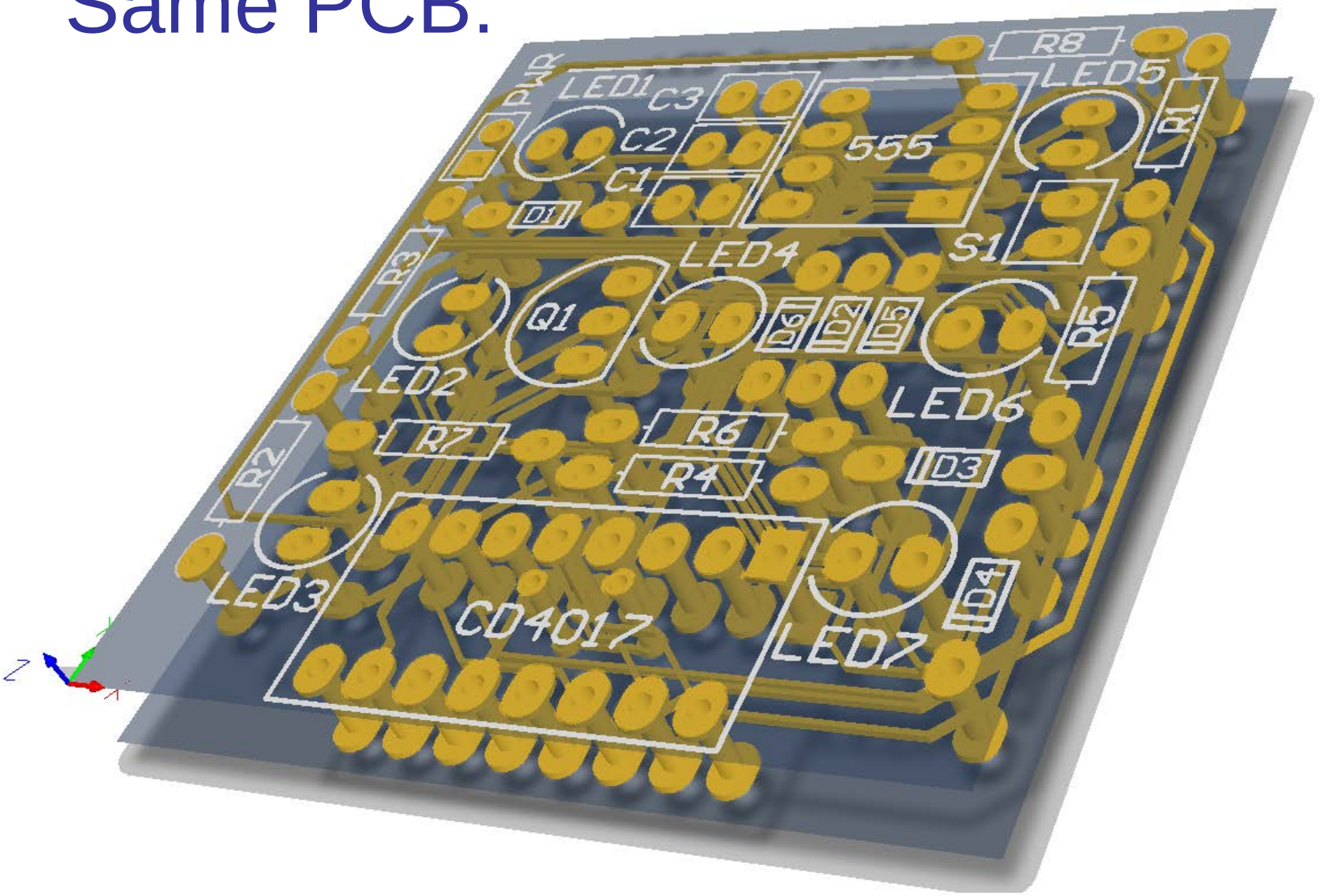
or integrated environments

Front-end design and capture

http://e2e.ti.com/


VCC
R1
100
S1
R2
680K
R3
680K
C1
3.3uF
C2
10nF
555
RST
THR
TRIG
+VCC
DIS
OUT
CVOLT
GND
33nF
C3
CD4017
R8
8.2K
D1
D2
D3
D4
D5
D6
R7
150
LED1
LED7
PWR
R5
150
R6
150
R4
180
LED2
LED3
LED4
LED6
LED5
Q1
T-NPN
GND
PWR
LED1
C3
C2
C1
LED5
S1
LED4
Q1
LED2
LED6
LED3
LED7
R8
555
R1
D1
R5
D6
D2
D5
R7
R6
R4
D3
D4
CD4017

Same PCB:


LED1
C3
C2
C1
555
R8
LED5
R1
D1
LED4
S1
R3
Q1
D1
D2
D5
R5
LED2
LED6
R7
R6
R4
D3
R2
LED3
CD4017
D4
LED7

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

THE UNIVERSITY OF BRITISH COLUMBIA

UBC Engineering — Electrical and Computer Engineering

Electronic Software Distribution

Search

Enter search term | Go

Admin
- Groups
- Software
- Eligibility

History
- Previous Downloads
- Accepted Licenses

Help
- Login
- Eligibility
- ISO Files

ALTIUM DESIGNER

Circuit Design Software

External Links

- Altium
- Altium Designer

Summer 09 Release
10
2014
16

ALTIUM DESIGNER 16

File	Size	
README.html		README
AltiumDesigner16Setup.exe	10.4 MB	Windows installer (requires
EULA.pdf	56.2 KB	End-User License Agreemer
OfflineSetupAD16_1_9.zip	3 GB	Windows installer

1

2

3

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

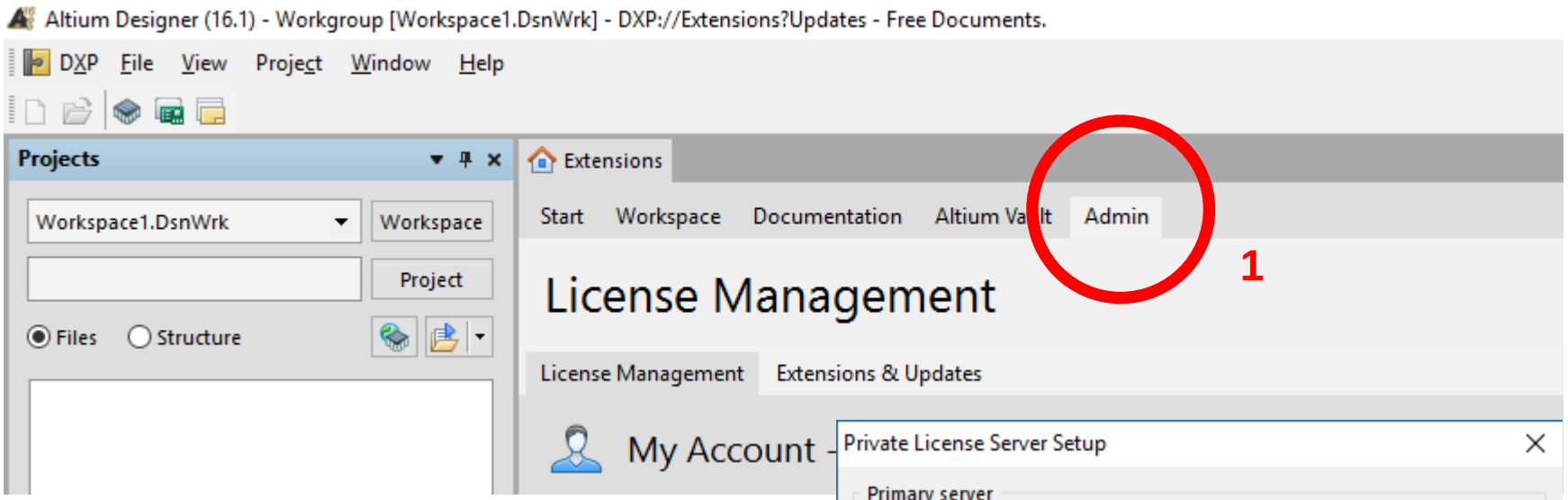
Start Altium, and from your "My Account" page, click on "Setup private license server". Enter:

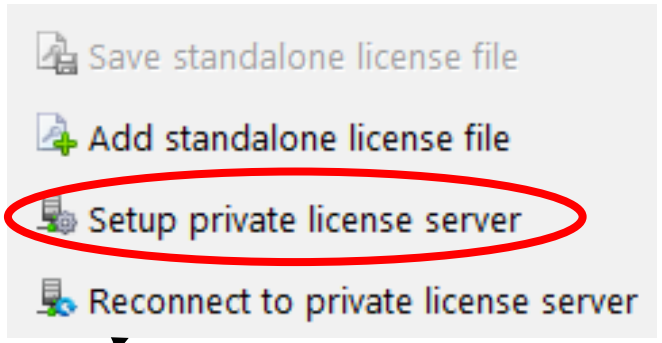
Server name:	
Server port:	
Secondary server name:	
Server port:	

See file: README.html

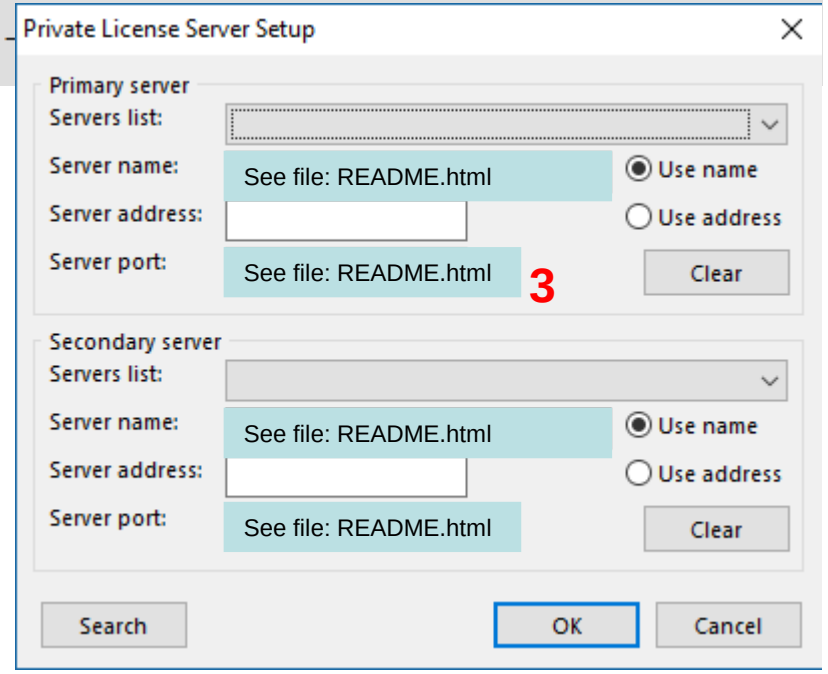
Select the new license that appears and click on "Use". You may as well also delete any old, expired licenses that are also showing.

To set license server



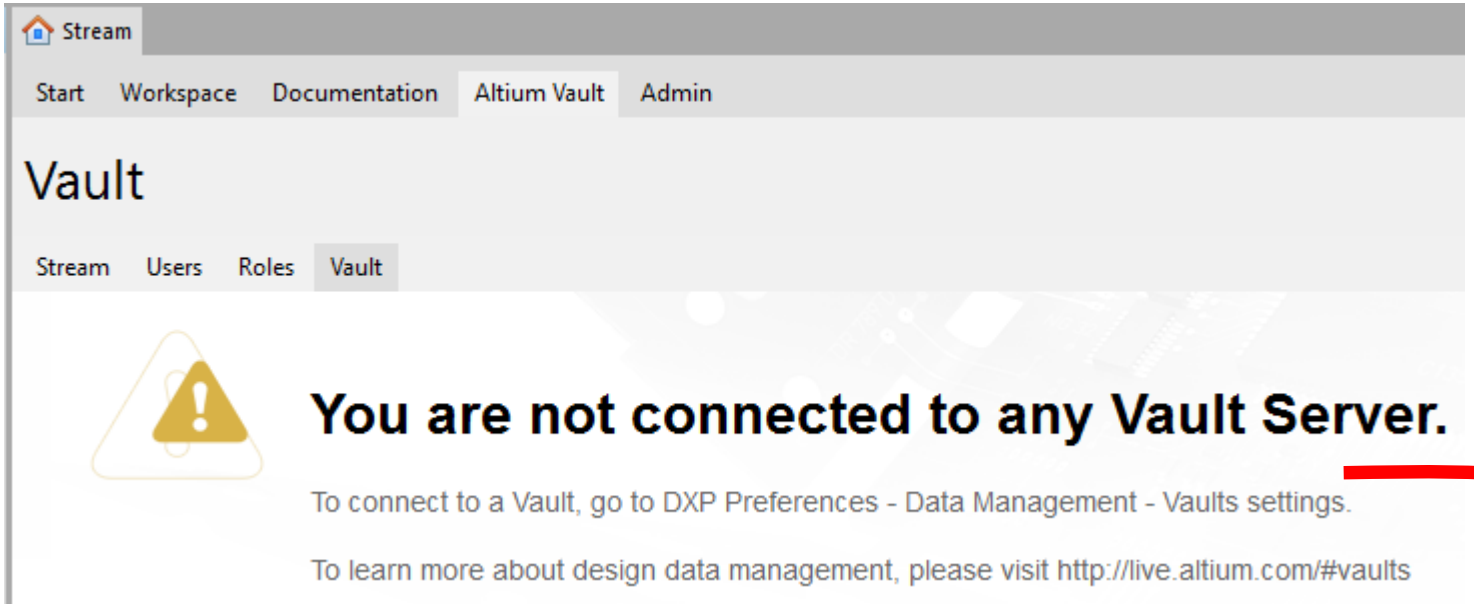


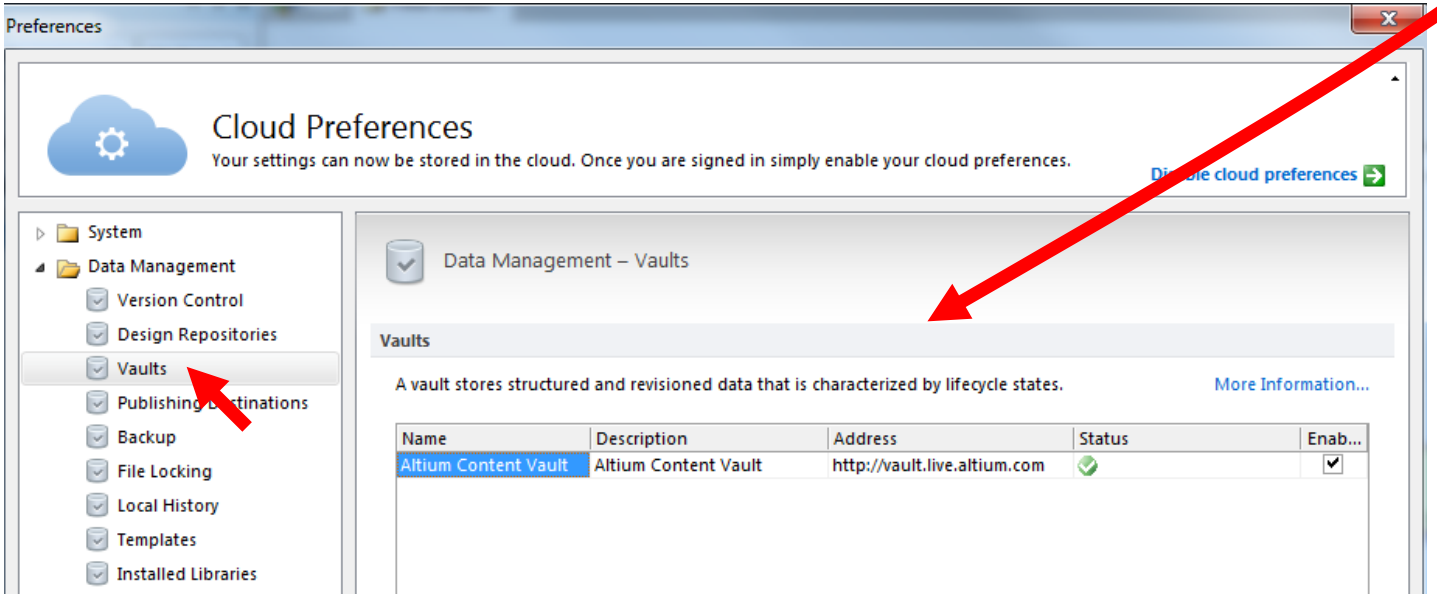
If you loose connection to server click here:



As per README.html file

Connecting to the Altium Vault

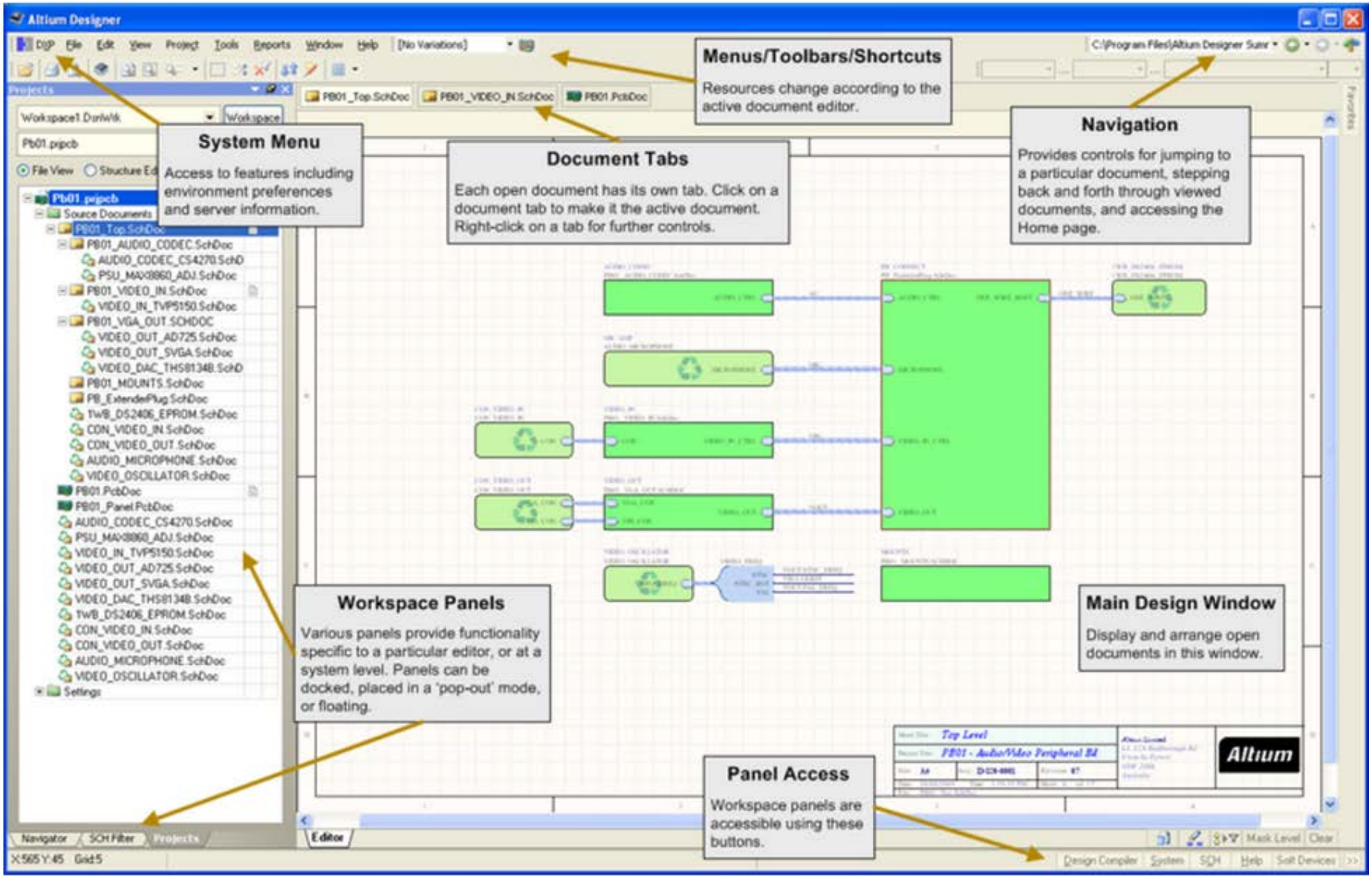




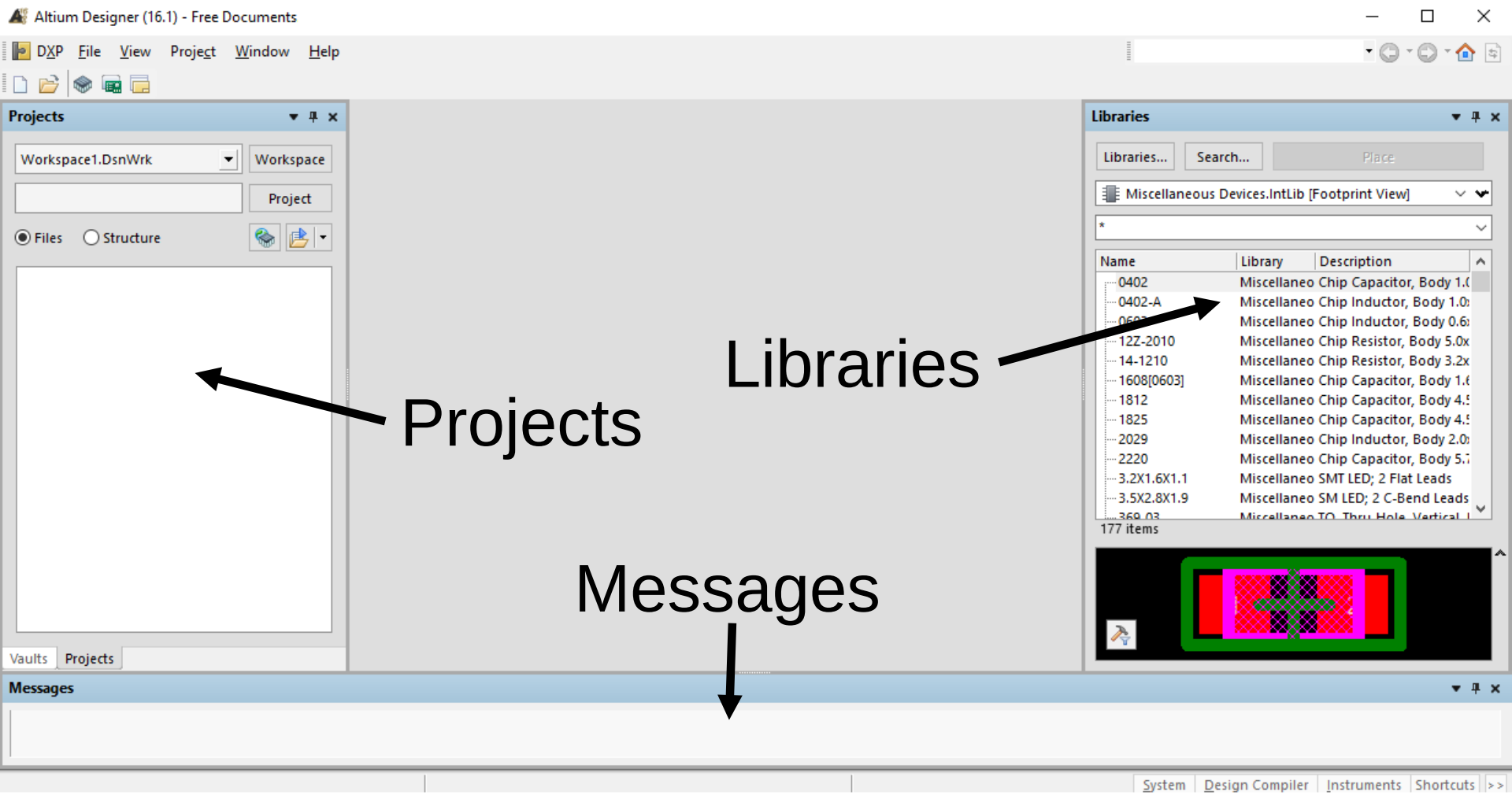
Understanding Altium

- 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



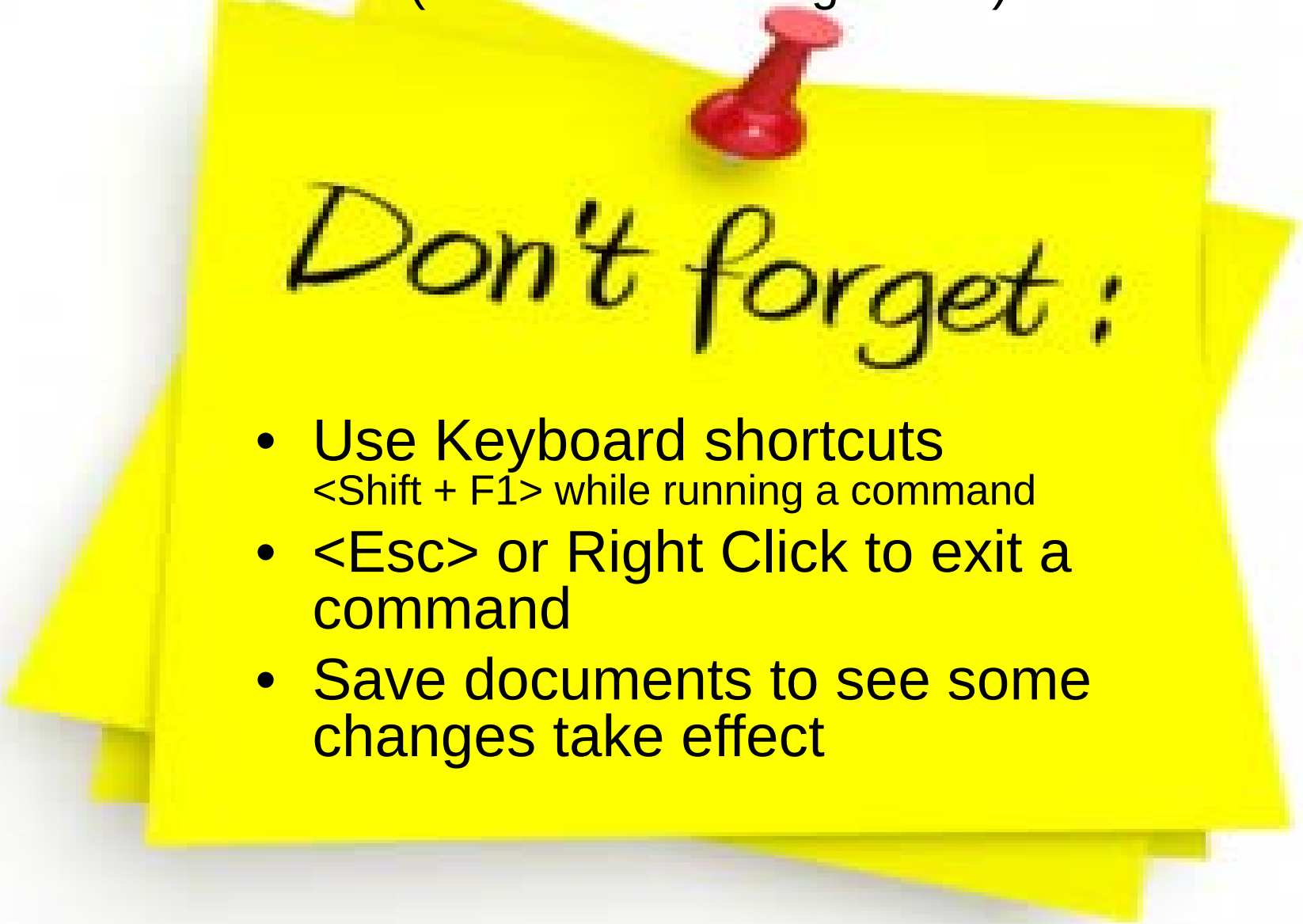
Recommended basic panels



For more help working with panels read this

Understanding Altium

(Basics for the single user)

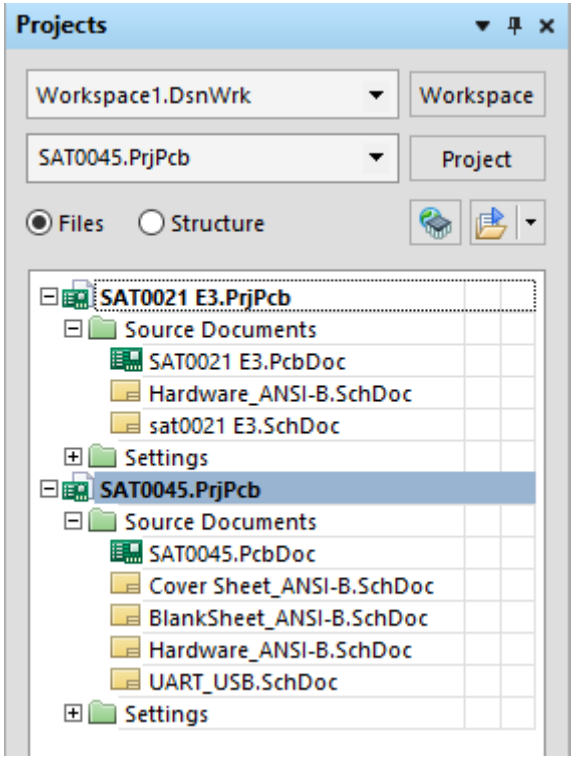


- Use Keyboard shortcuts
  <Shift + F1> while running a command
- <Esc> or Right Click to exit a command
- Save documents to see some changes take effect

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)

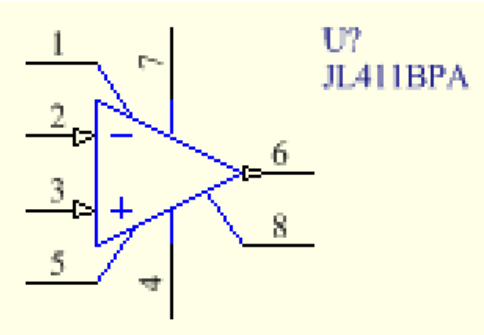
- The active project is highlighted
- Add/Remove documents to/from a project

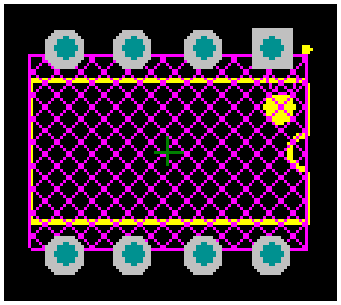
Altium Projects: types

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

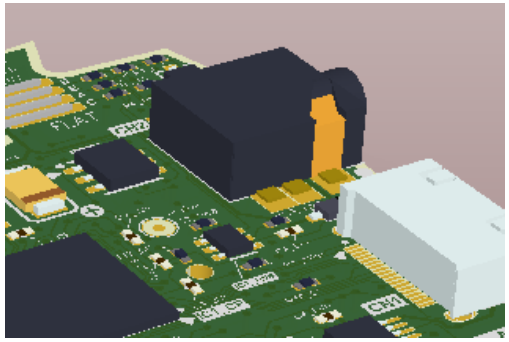
Component, Model and Library Concepts

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





```
***************INPUT STAGE**************
*
IOS 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 JX
J2 6 7 4 JX
R3 5 50 650
R4 6 50 650
*Fp2=28 MHZ
```



Component, Model and Library Concepts

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

  } Different component representations

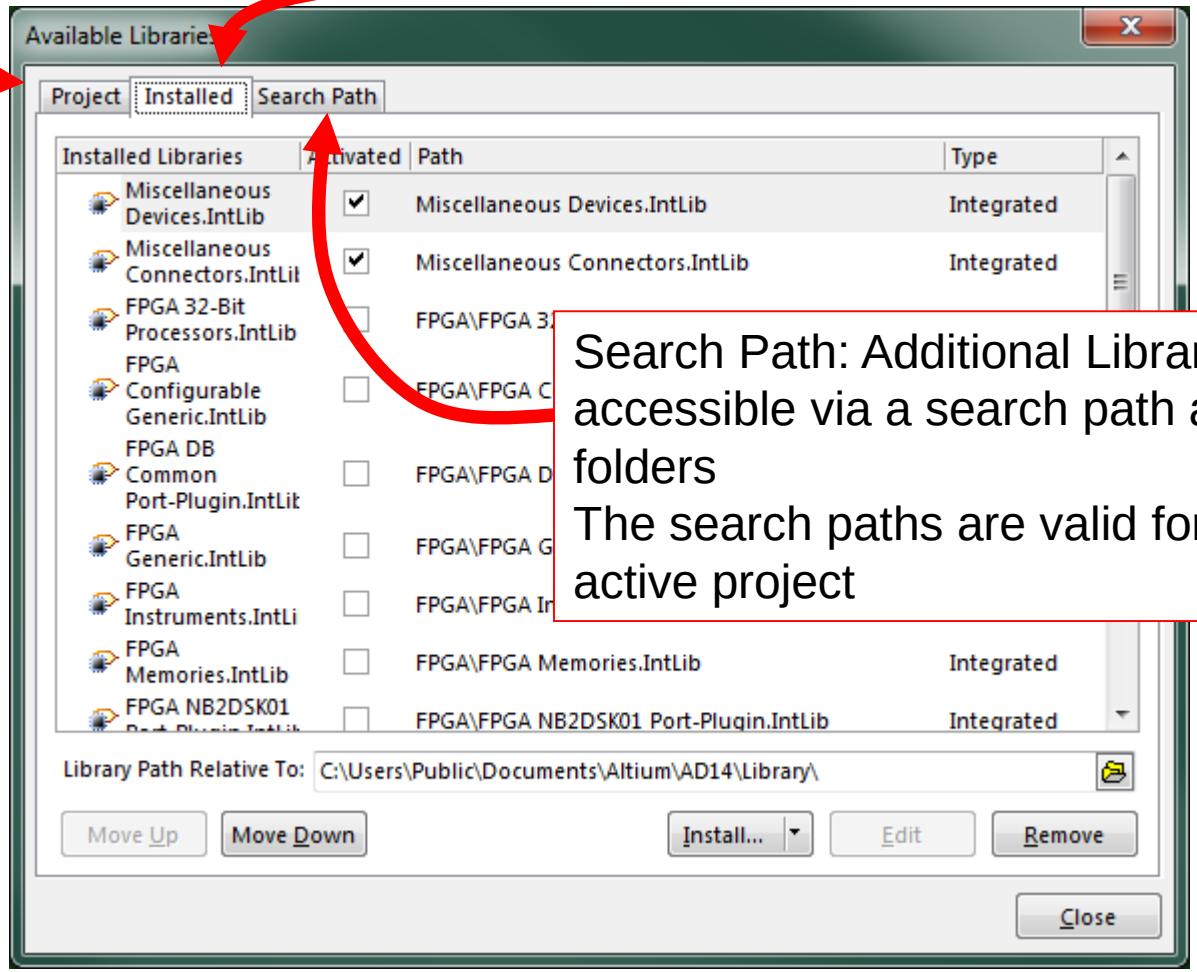
- A unified component is a container with links to all domain models + parametric information

Libraries = collection of components

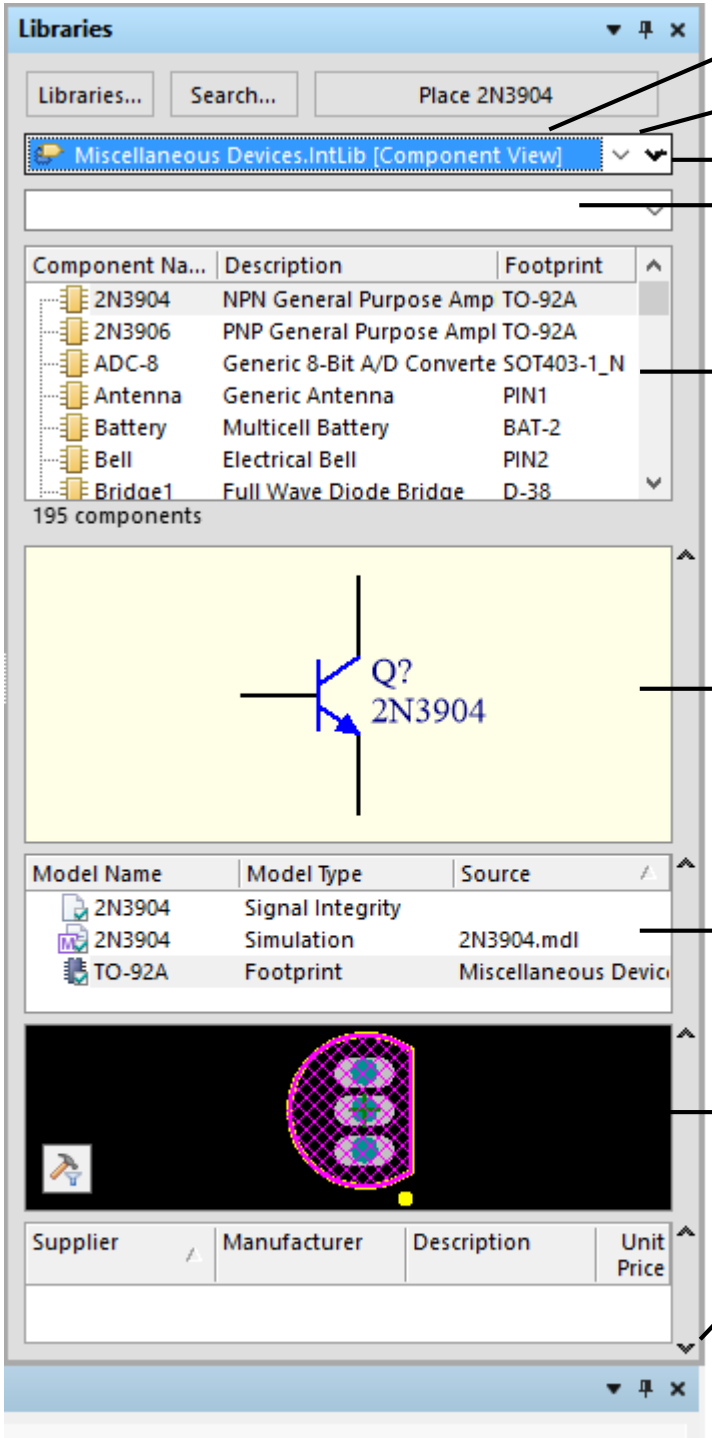
- Collection of components, models or both
- Model Libraries (*.MDL, *.CKT, *.PCBLib)
  - Simulation models are one file per model
- Schematic Libraries (*SchLib)
  - Symbol and a link to a model library
- Integrated Libraries (*.IntLib)
  - Symbol, footprint and other models are compiled into a single 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 sub-folders
The search paths are valid for the active project



Current library

Select a different library

Set library browse mode

Search in current library

List of components. Select the component of interest

Schematic symbol for selected component

Models linked to the selected component

Graphical display of the selected model

Icons used to show/hide panel sections

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

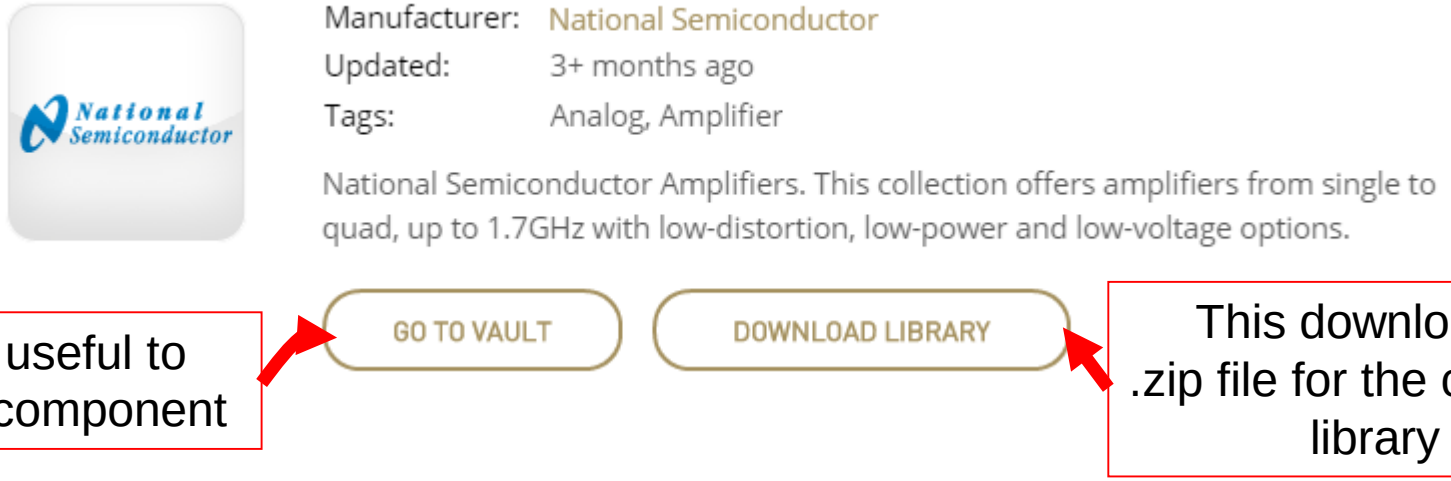
<L> for PCB footprints
to flip component side

To search across libraries:

Search …

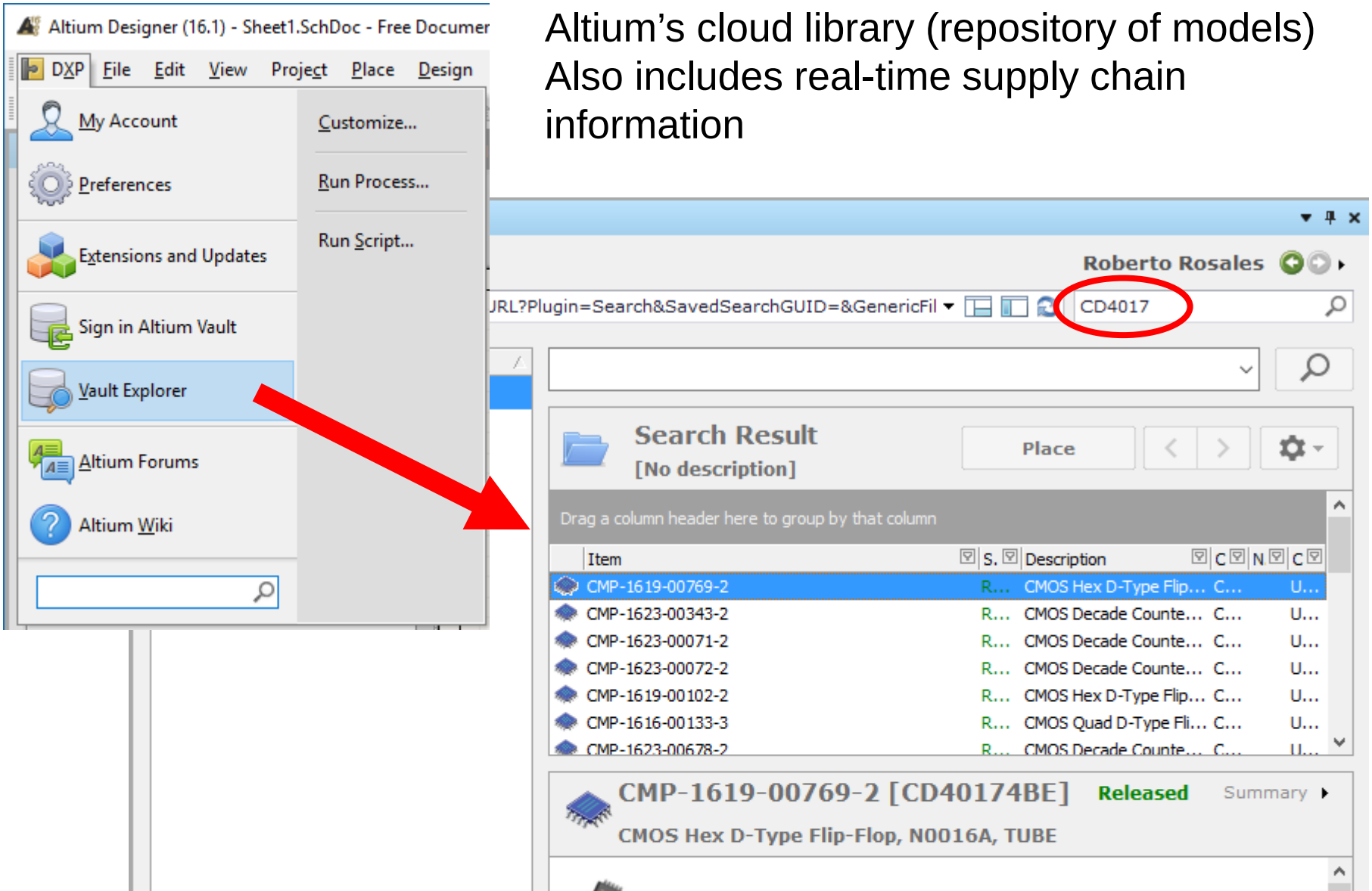
Obtaining integrated libraries

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

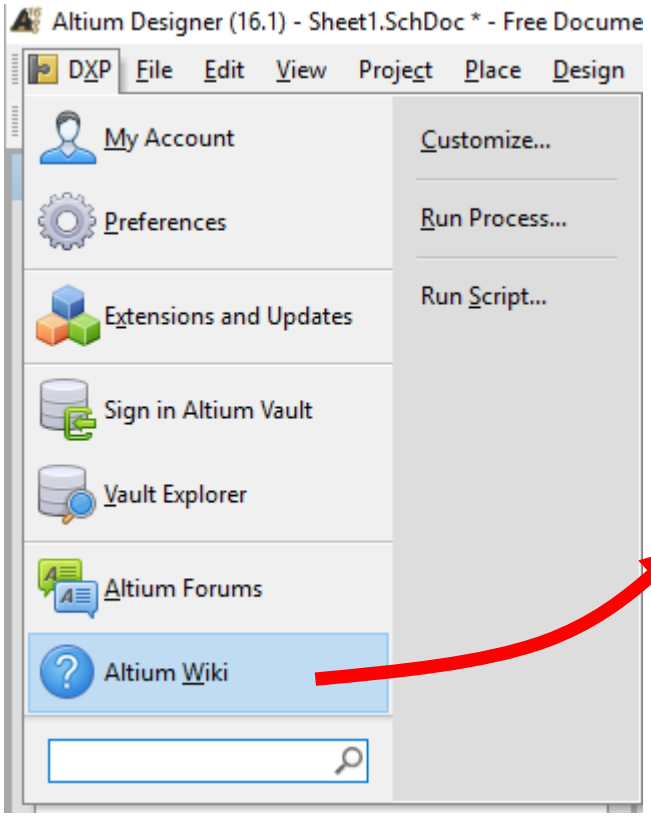


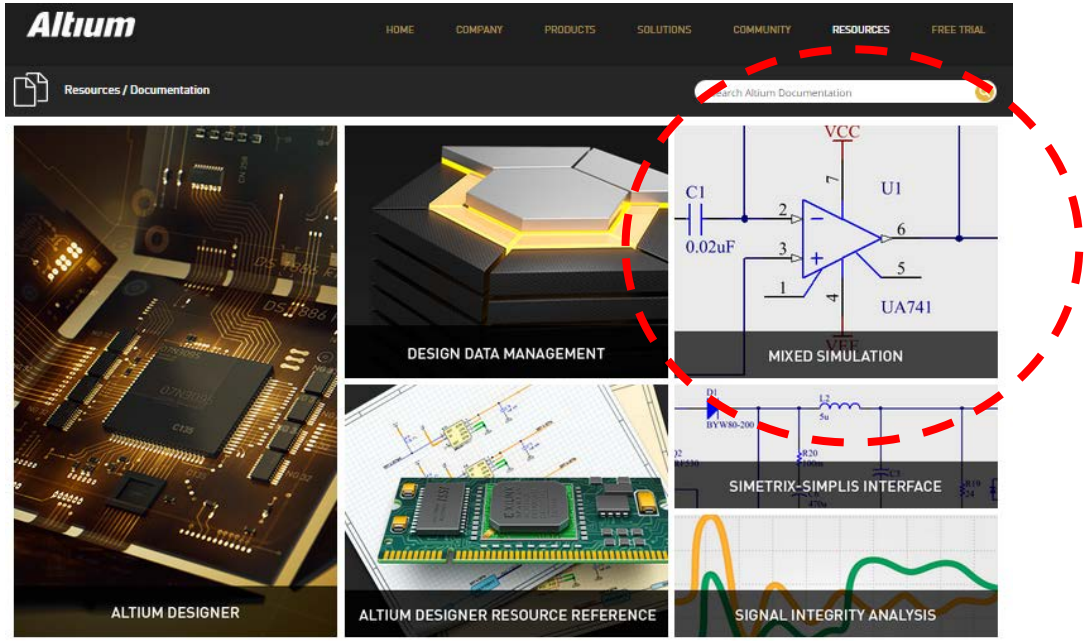
Altium Vault

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



Learning to use Altium

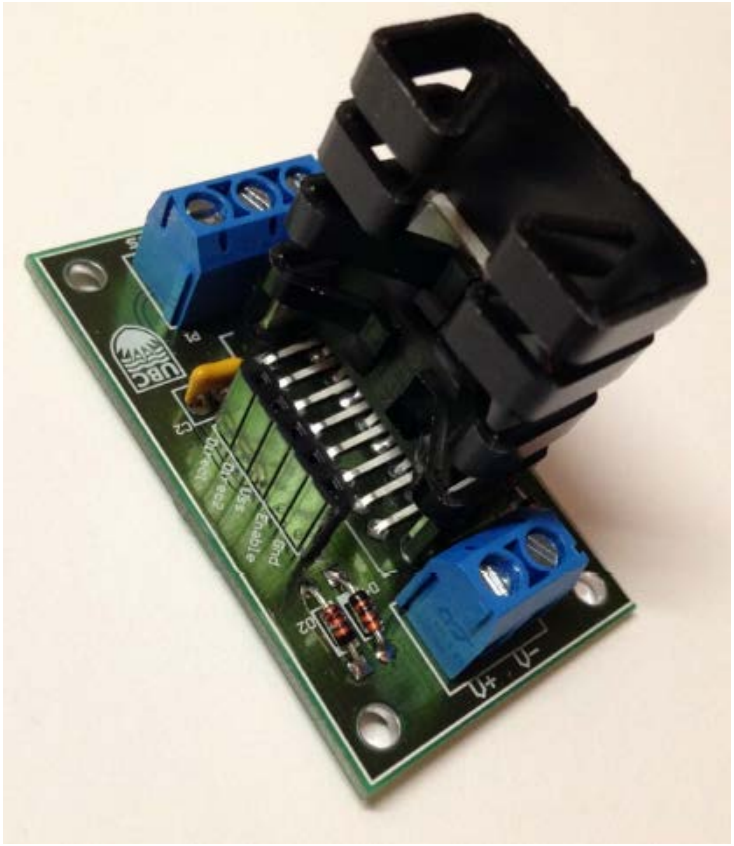




Best training material is on the Altium website
It is updated, but beware that menus and options slightly change between versions

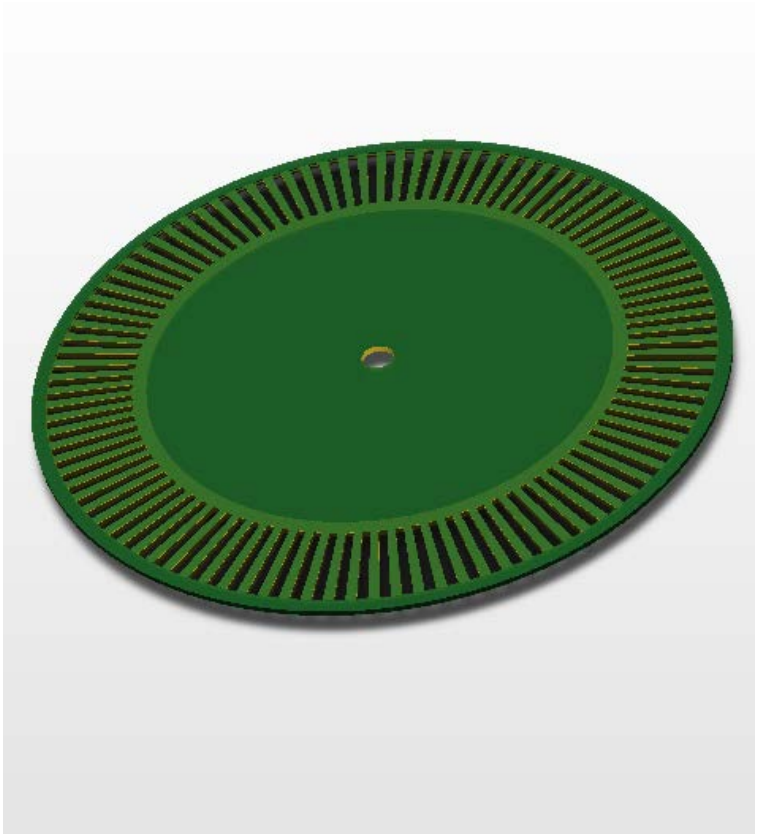
Walk-through examples

L298 Motor Driver Board
(by Matt Winship)



L298_Motor_Driver_Board_Datasheet.pdf

Encoder Wheel
(by Rob Hais)

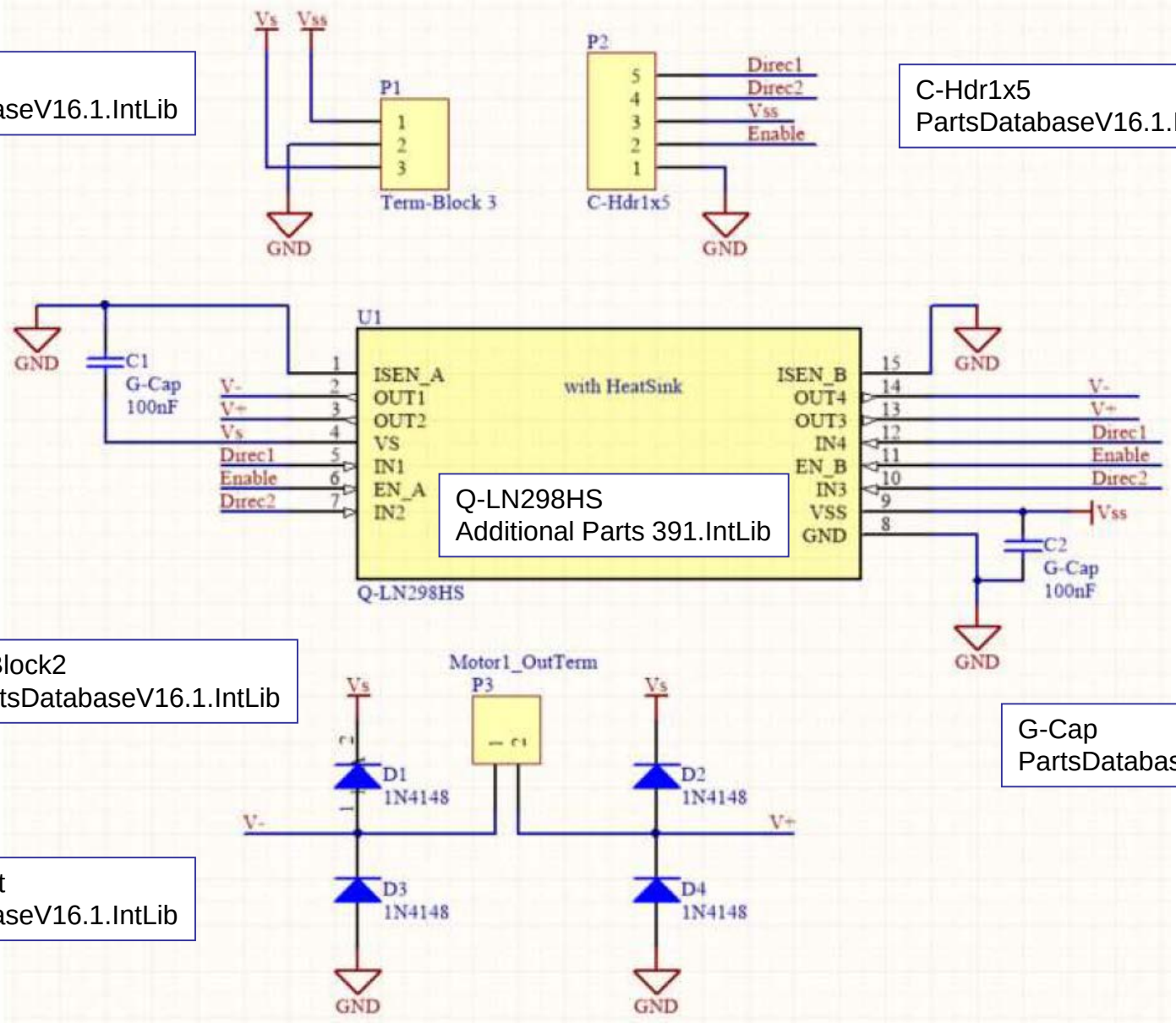


Encoder Layout Tutorial.pdf

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

LM298 Motor Driver Board Schematic

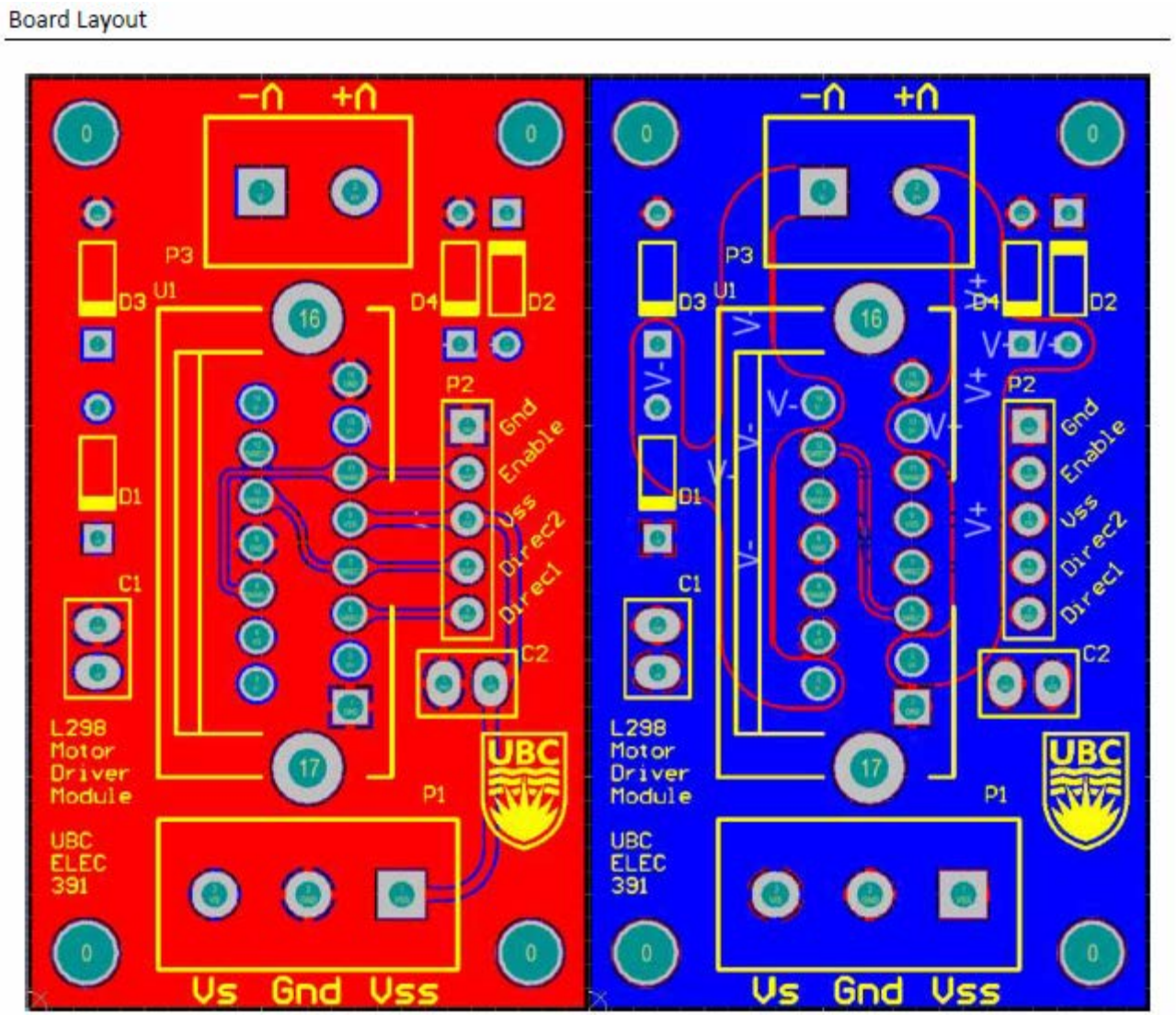
1. Load libraries
2. Draw the schematic
   Set electrical type for connector pins
3. Compile Project:
   Project → Project Options
4. Place no ERC labels if necessary
   Modify connection matrix with caution

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



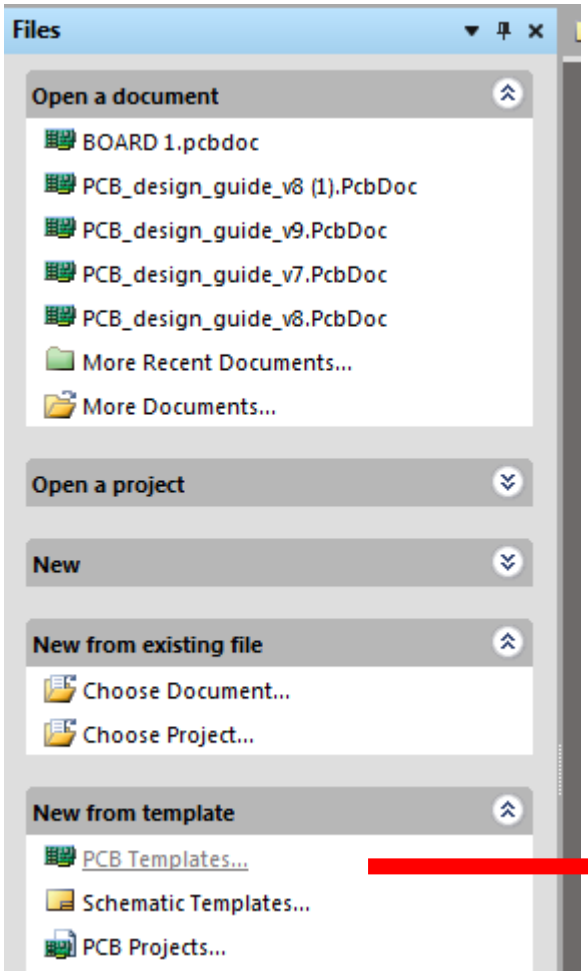
Top Metal

Bottom Metal

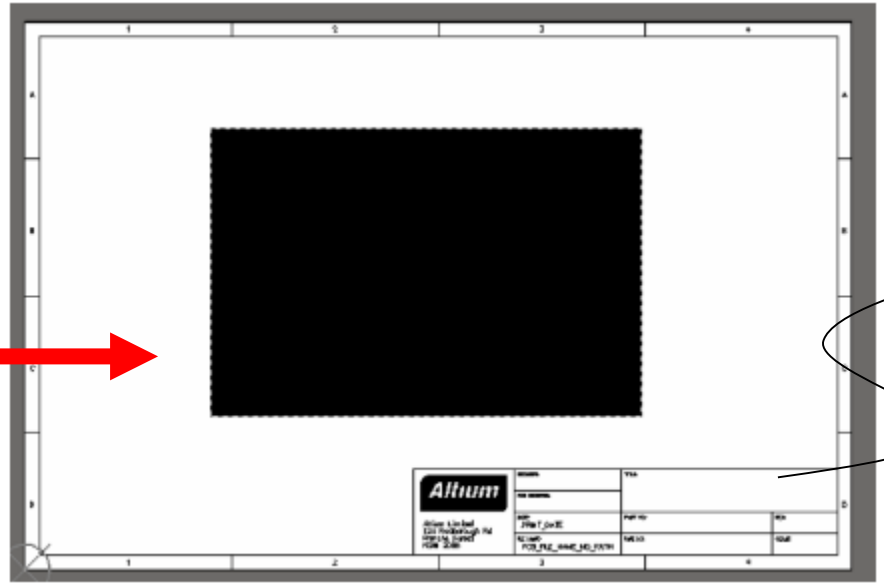
2 starting points for PCB design

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



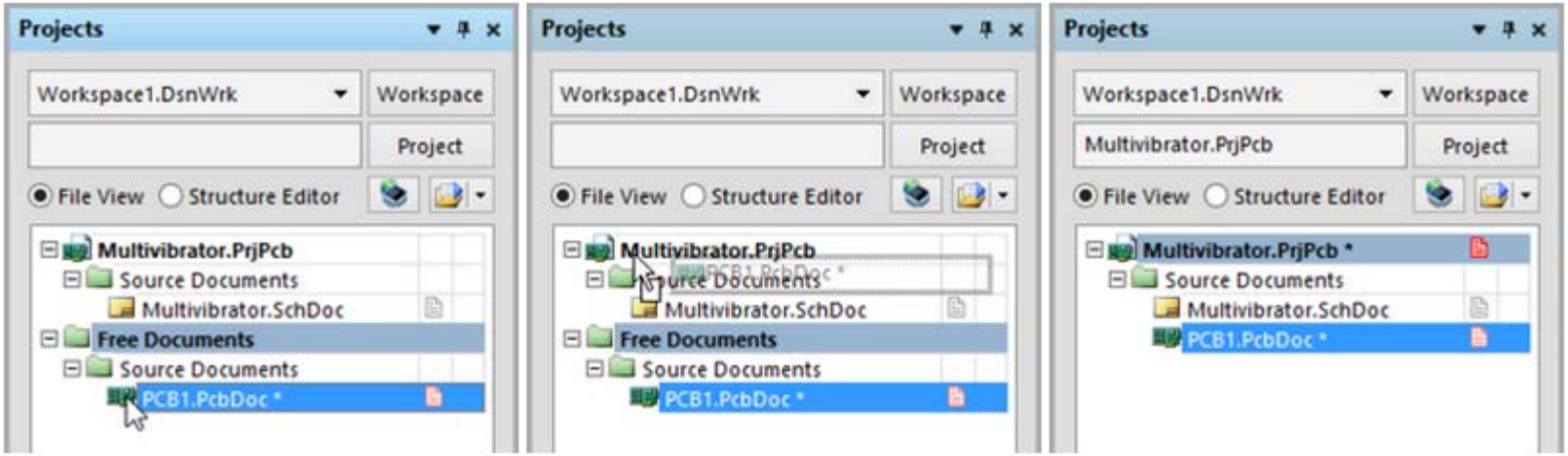
- Files Panel:
  - New from template
  - select the A4.PcbDoc template
  - Save as … same name and directory as SchDoc file



Mechanical 16

Adding PCB file to project

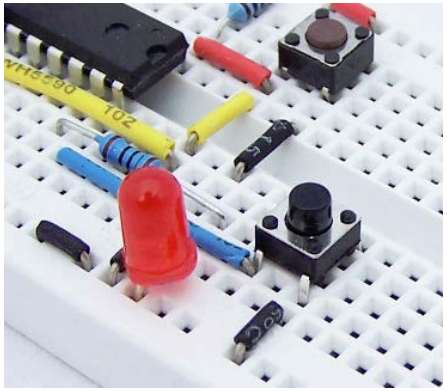
- Make the PCB board part of the project



- Rename the file
- Save the PcBDoc file and the project

First things first ... choosing working units

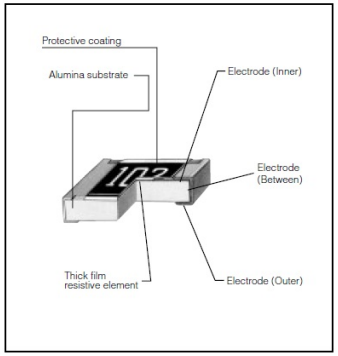
- Imperial (inches)
  - 1/1000th of an inch = 1 mil = 1thou
  - 100mils (0.1”) is a common dimension



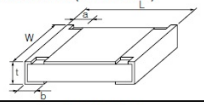
- Metric (mm)
  - 1 mm ≠ 1mil !
  - Common unit in SM parts

Panasonic Thick Film Chip Resistors

■ Construction



■ Dimensions in mm (not to scale)

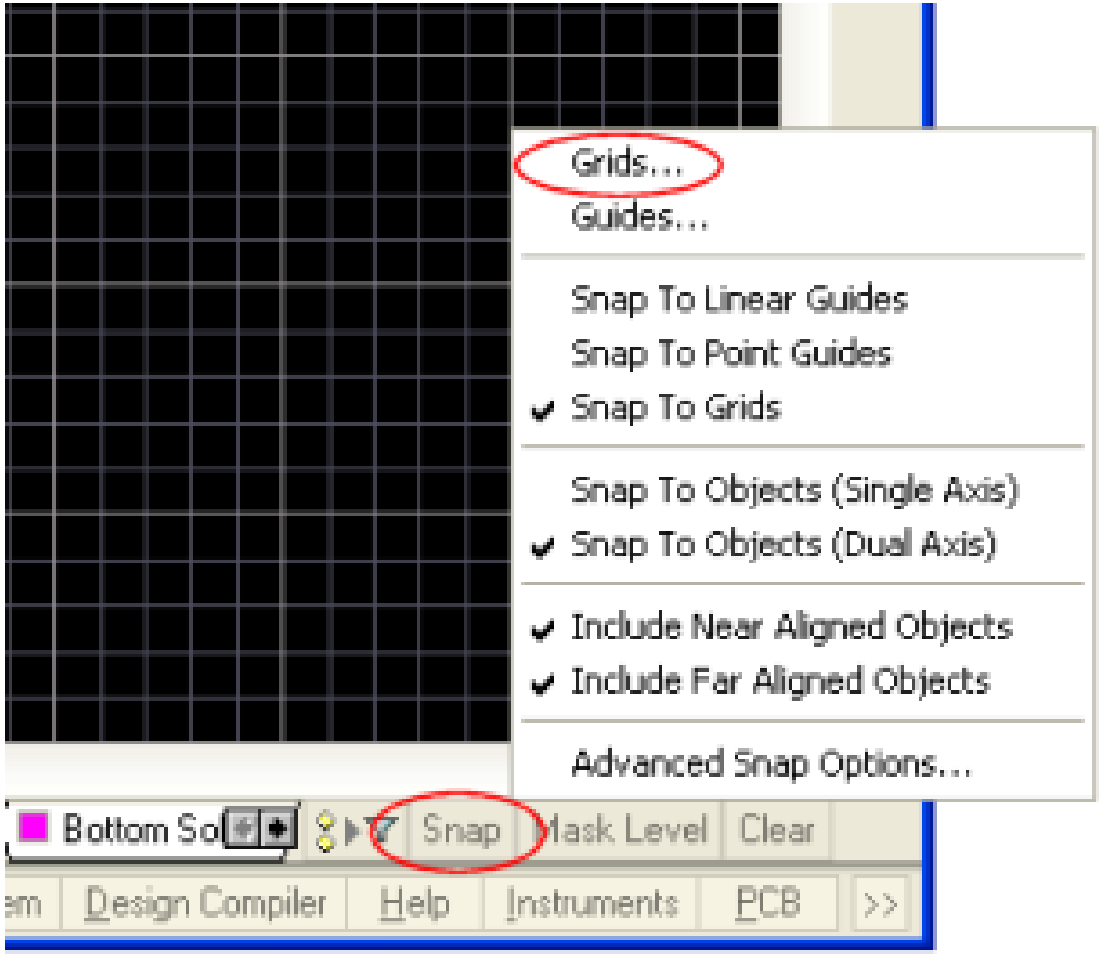


Type (inches)	Dimensions (mm) L	W	a	b	t	Mass (Weight) (g/1000 pcs.)
ERJXG (01005)	$0.40^{\pm0.02}$	$0.20^{\pm0.02}$	$0.10^{\pm0.03}$	$0.10^{\pm0.03}$	$0.13^{\pm0.03}$	0.04
ERJ1G (0201)	$0.60^{\pm0.03}$	$0.30^{\pm0.03}$	$0.10^{\pm0.05}$	$0.15^{\pm0.05}$	$0.23^{\pm0.03}$	0.15
ERJ2G (0402)	$1.00^{\pm0.05}$	$0.50^{\pm0.05}$	$0.20^{\pm0.10}$	$0.25^{\pm0.05}$	$0.35^{\pm0.05}$	0.8
ERJ3G (0603)	$1.60^{\pm0.15}$	$0.80^{+0.15}_{-0.05}$	$0.30^{\pm0.20}$	$0.30^{\pm0.15}$	$0.45^{\pm0.10}$	2
ERJ6G (0805)	$2.00^{\pm0.20}$	$1.25^{\pm0.10}$	$0.40^{\pm0.20}$	$0.40^{\pm0.20}$	$0.60^{\pm0.10}$	4
ERJ8G (1206)	$3.20^{+0.05}_{-0.20}$	$1.60^{+0.05}_{-0.15}$	$0.50^{\pm0.20}$	$0.50^{\pm0.20}$	$0.60^{\pm0.10}$	10
ERJ14 (1210)	$3.20^{\pm0.20}$	$2.50^{\pm0.20}$	$0.50^{\pm0.20}$	$0.50^{\pm0.20}$	$0.60^{\pm0.10}$	16
ERJ12 (1812)	$4.50^{\pm0.20}$	$3.20^{\pm0.20}$	$0.50^{\pm0.20}$	$0.50^{\pm0.20}$	$0.60^{\pm0.10}$	27
ERJ12Z (2010)	$5.00^{\pm0.20}$	$2.50^{\pm0.20}$	$0.60^{\pm0.20}$	$0.60^{\pm0.20}$	$0.60^{\pm0.10}$	27
ERJ1T (2512)	$6.40^{\pm0.20}$	$3.20^{\pm0.20}$	$0.65^{\pm0.20}$	$0.60^{\pm0.20}$	$0.60^{\pm0.10}$	45

- Remember: 100mils = 2.54mm
- To switch units in Altium Press <Q>

First things first … setting the snap grid

- PCBs are grid based objects

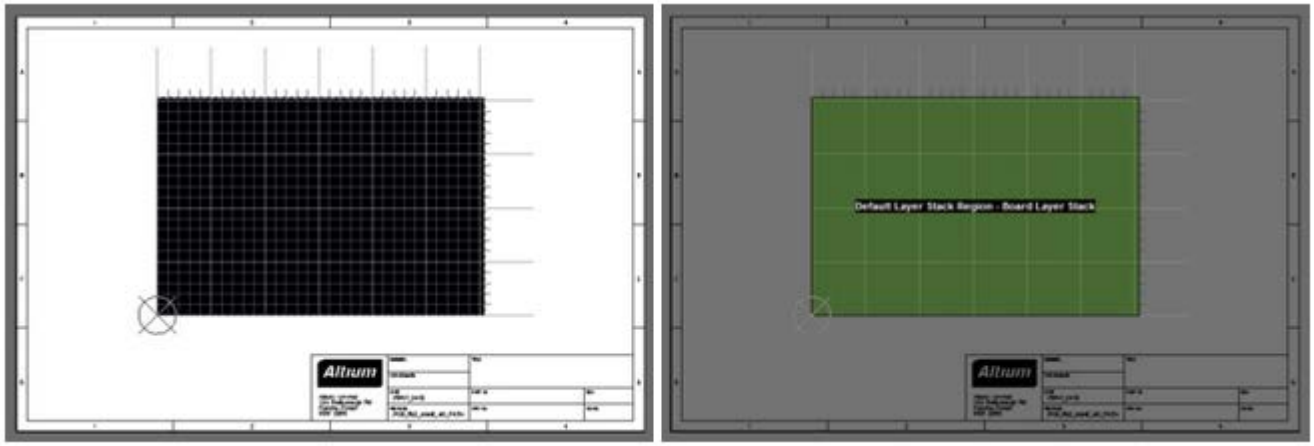


Unified Cursor-Snap System

- Selecting a suitable snap grid:
  - <Ctrl>+<G>
  - Start with a coarse grid to define board size

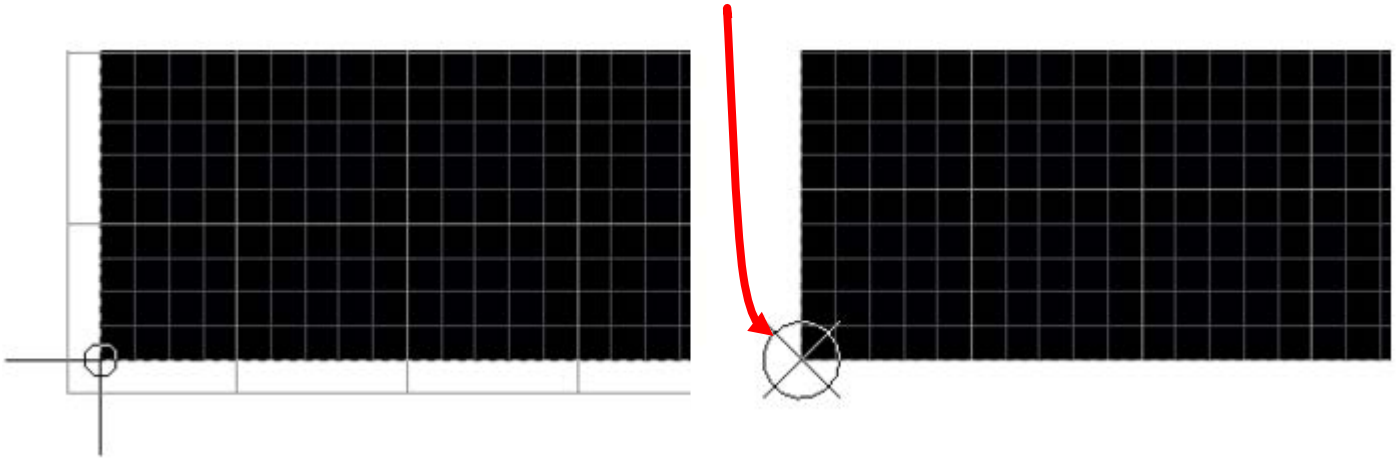
First things first … redefining the board shape

- 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**).



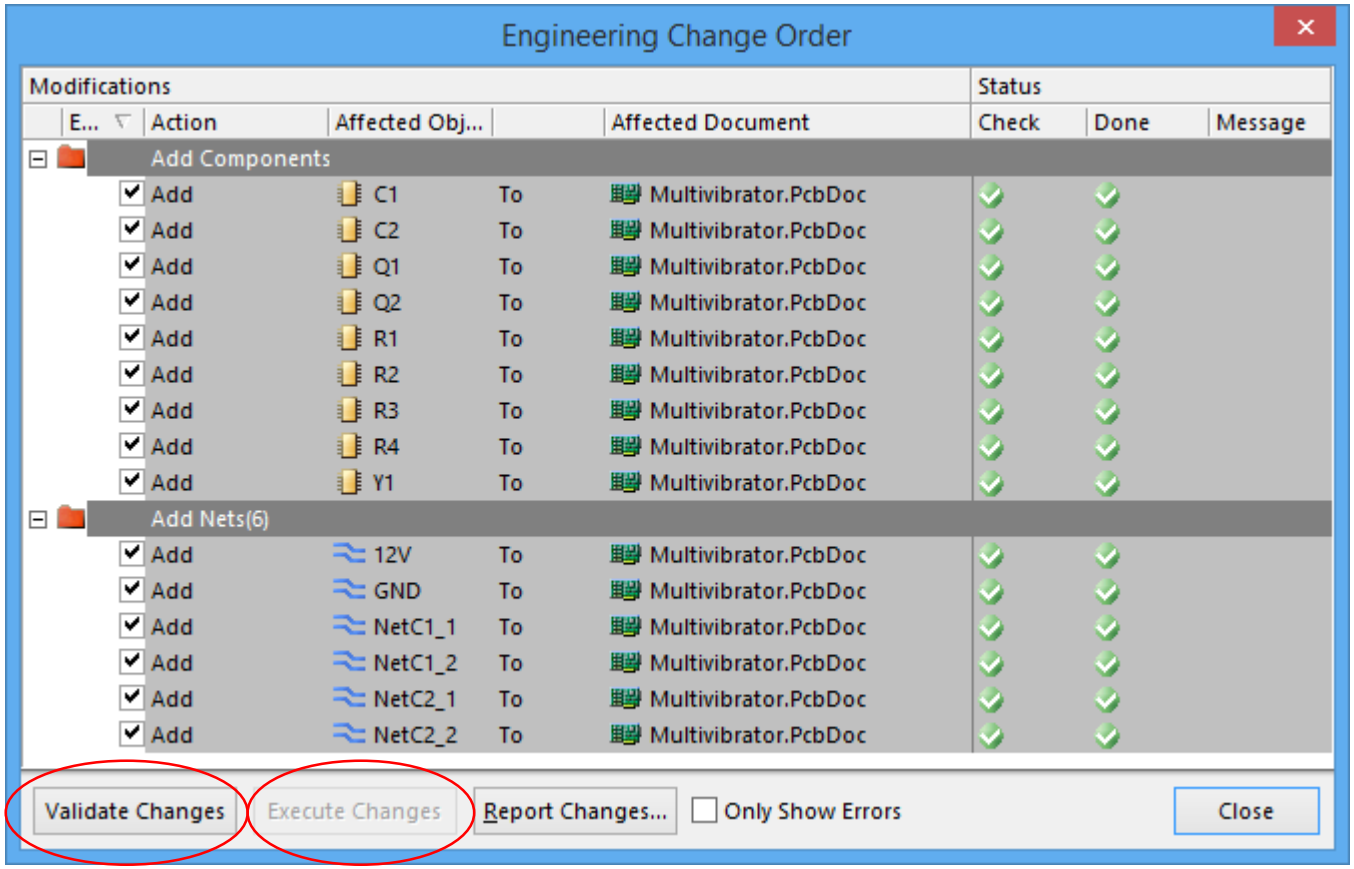
First things first … setting the board origin

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

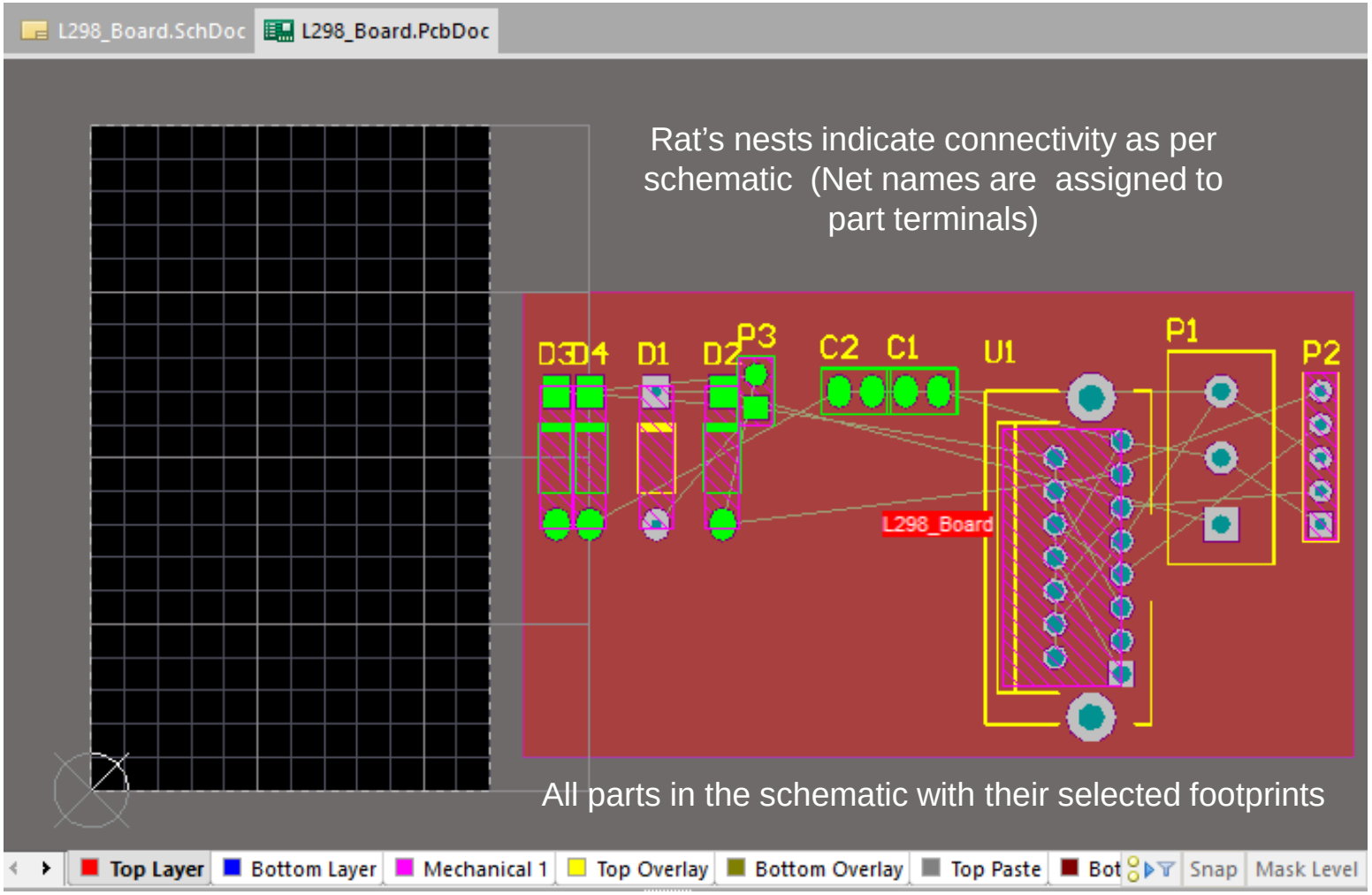


Design transfer

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

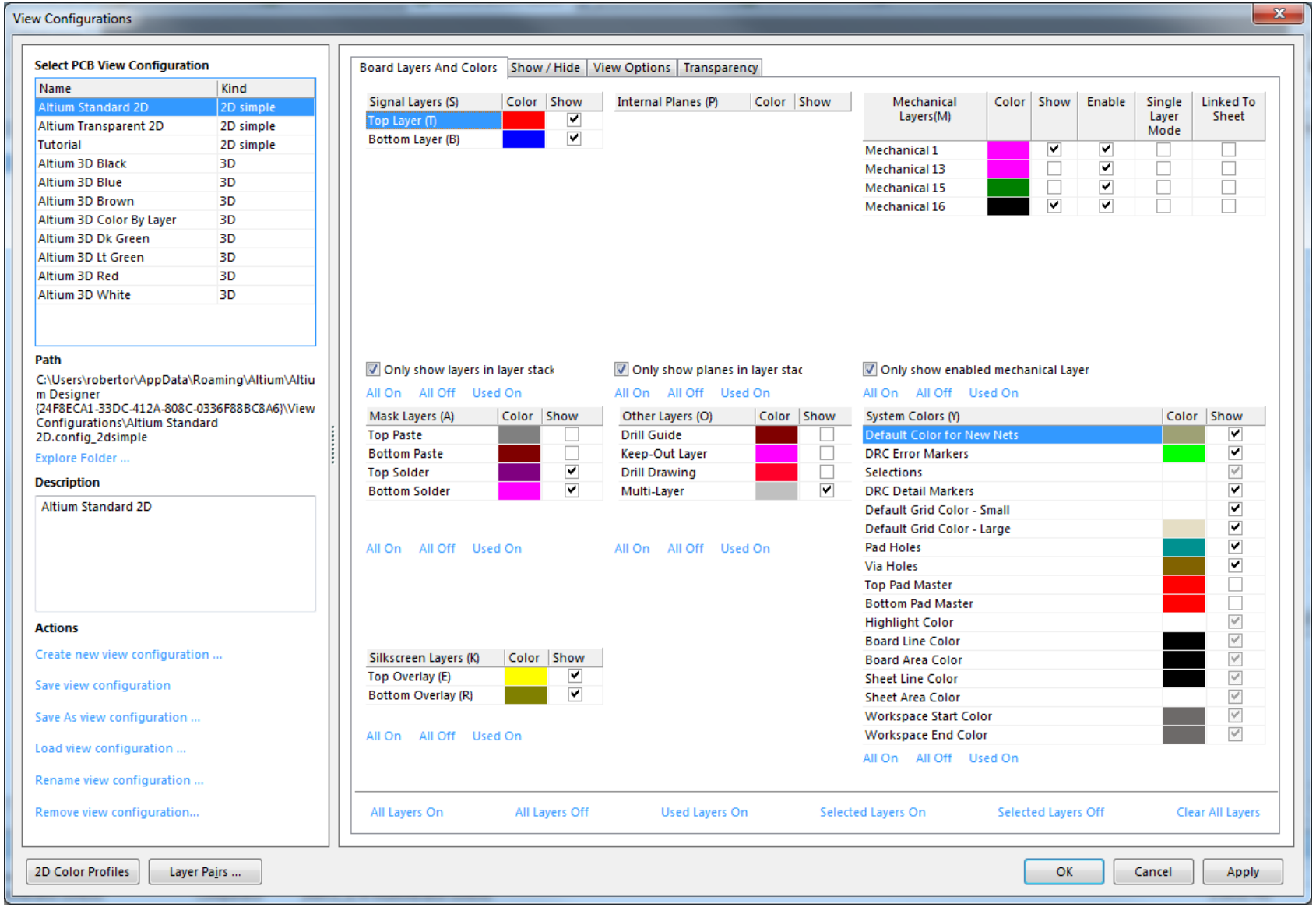


Design transfer



Configuring the Display Layers

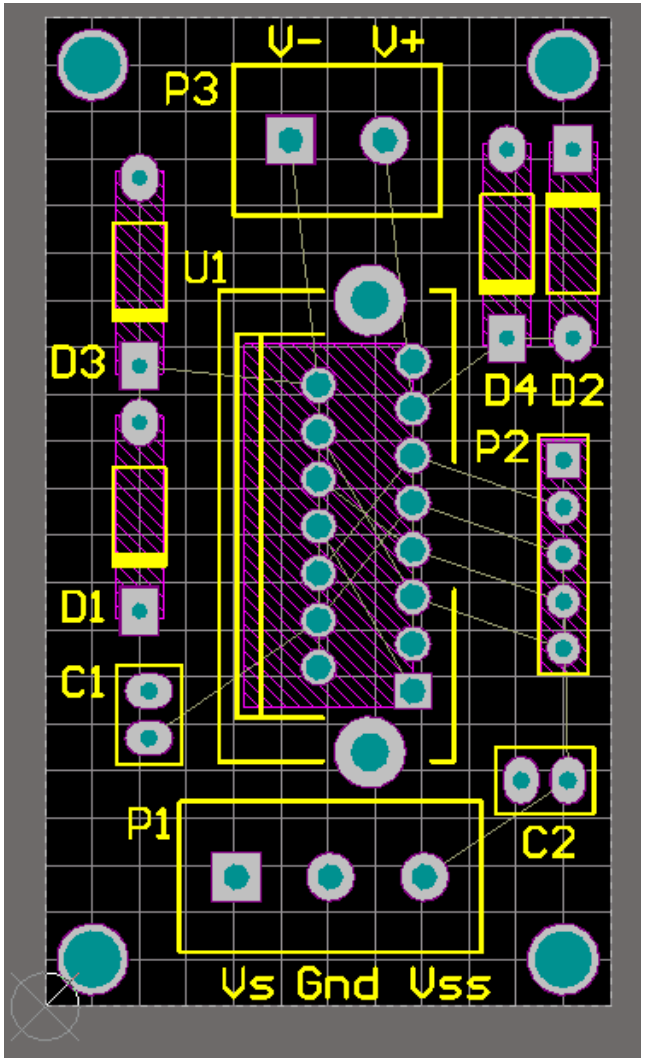
- Design » Board Layers and Colors



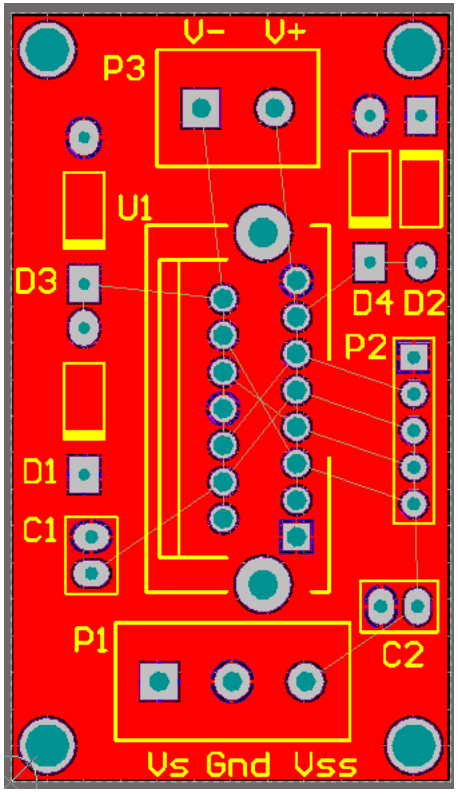
Configuring the Display Layers

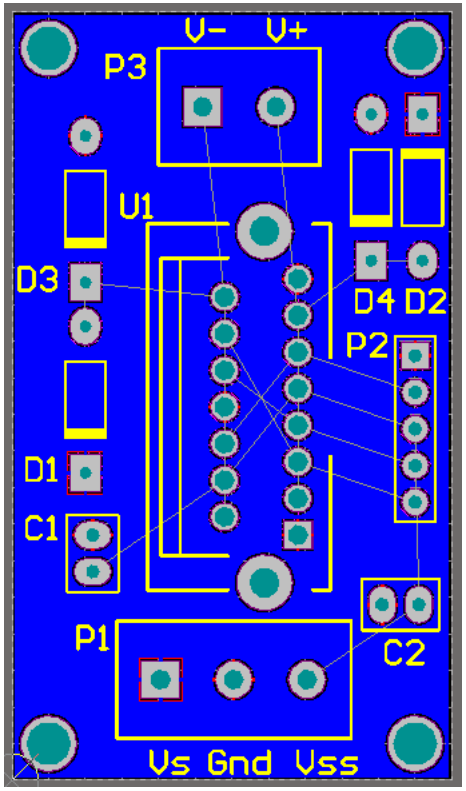
- Electrical layers
  32 signal layers and 16 internal power plane layers.
- Mechanical layers
  32 general purpose mechanical layers, used for design tasks such as dimensions, fabrication details, assembly instructions, or special purpose tasks such as glue dot layers. These layers can be selectively 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).

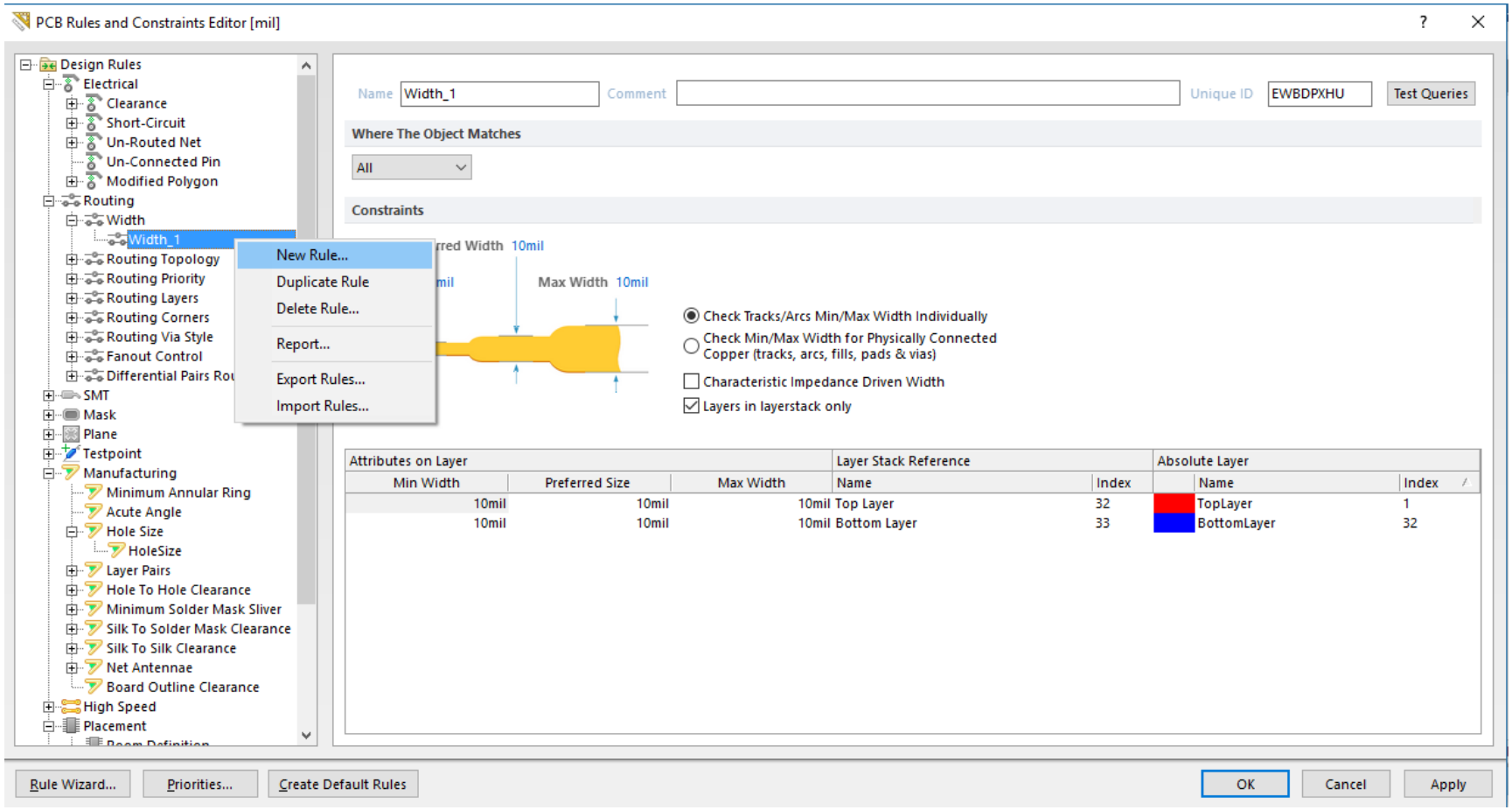
- **Shift**+**Spacebar** to cycle through the various track corner modes. The styles are: any angle, 45°, 45° with arc, 90° and 90° with arc. There is an option to limit this to 45° and 90° in the **PCB Editor - Interactive Routing** page of the *Preferences* dialog.

Design Rules

- Design >> Rules

Rule	Constrain	Query
Electrical, Clearance	Min clearance = 7mil	All
Routing, Width*	Min width = 7mils Max width = 500mils Preferred = 10mils	All
Routing, Width_IO	Min width = 7mils Max width =500mils Preferred =100mils	Advanced (Query) (InNet('V+') **OR** InNet('V-'))
Width_Vss	Min width = 7mils Max width =500mils Preferred =20mils	Net Vss

Custom Routing design rules



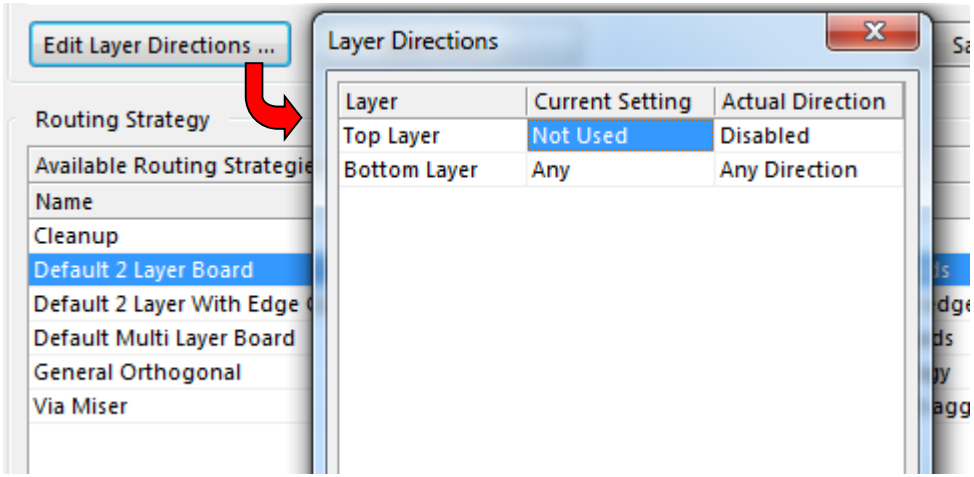
Rename to “Width_IO’

Use ‘Custom Query’ to set”
Belongs to net V+
OR
Belongs to V-

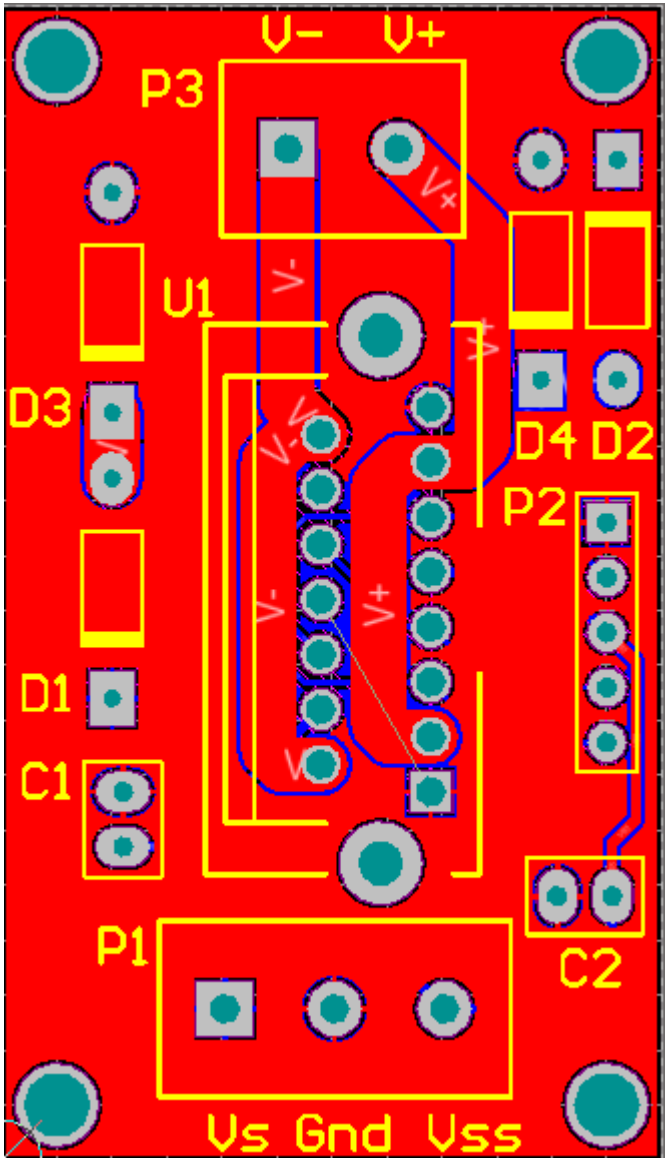
Set rule execution priority

Auto route

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



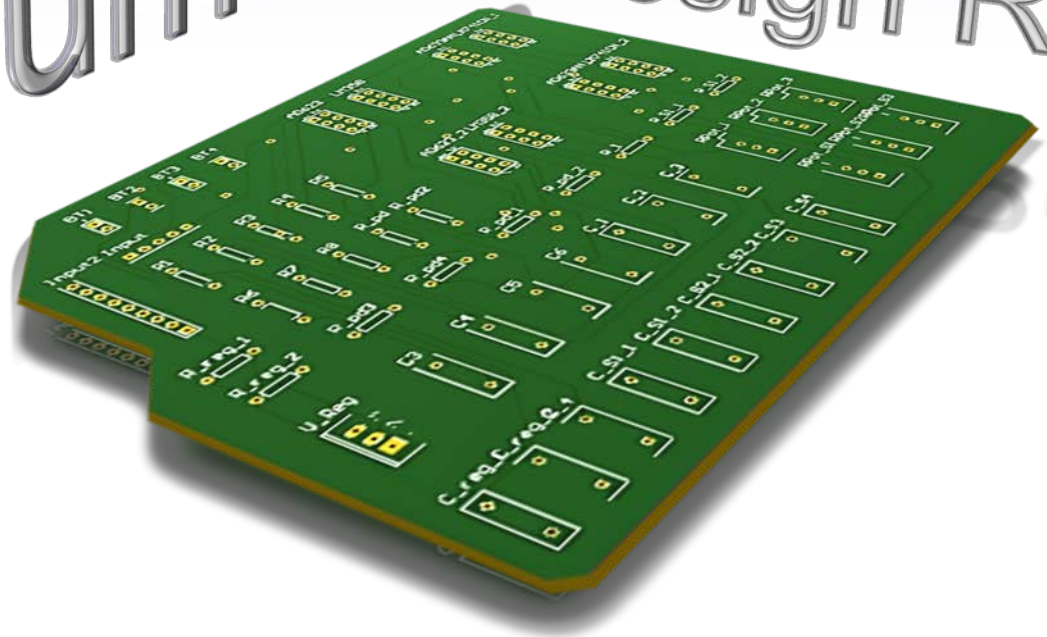
- You can also set single layer routing



ELEC 391

Altium PCB

Design Rules



Andy

Submission Instructions

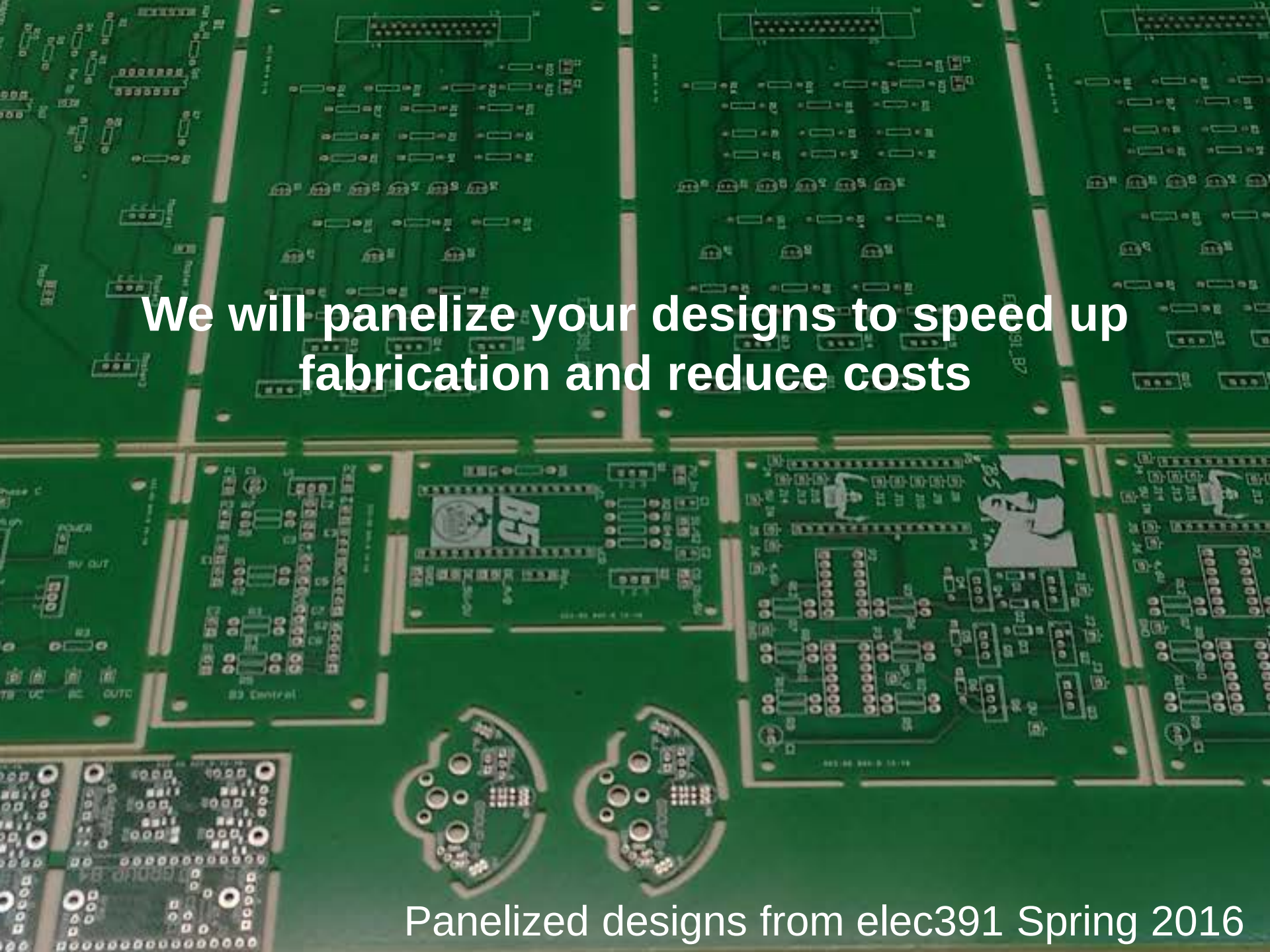
- No limits to PCB submissions (7 deadlines)
- Cost: $25 + $10/ sq-in, from project budget
- Submission dates:

  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

M	T	**W**	Th	F	Sa	S
M	T	**W**	**Th**	F	Sa	S

- 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
Panelized designs from elec391 Spring 2016

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 constrains

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

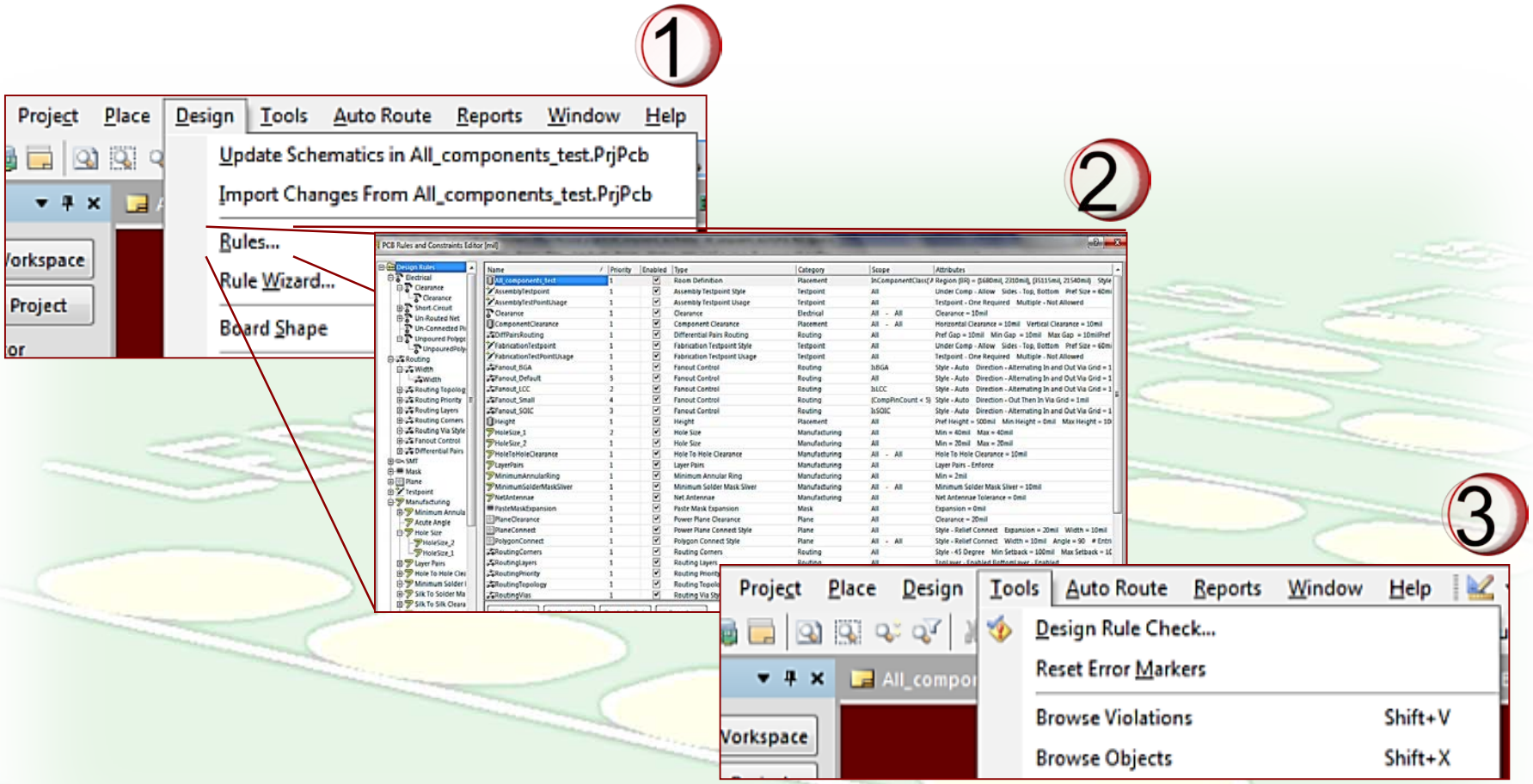
4) Use latest version of course component library available here
http://www.ece.ubc.ca/~eng-services/files/courses/elec391-spring2018/pcb-design/

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

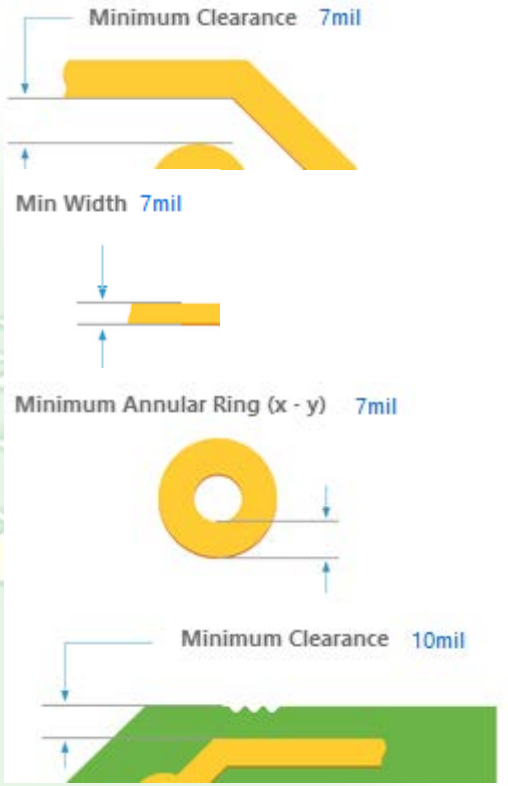


Rules – design rules

- 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

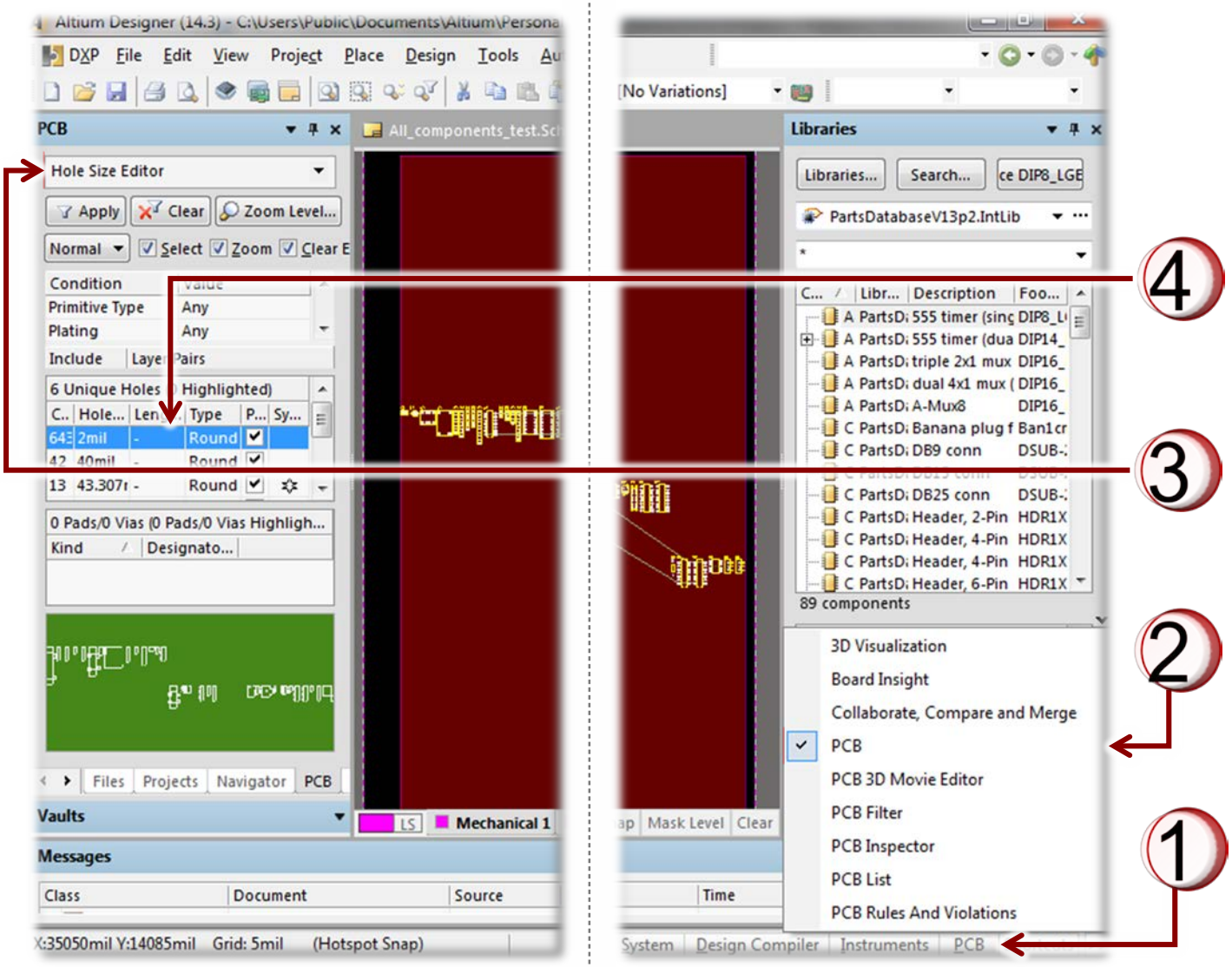


Rules – hole sizes

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

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

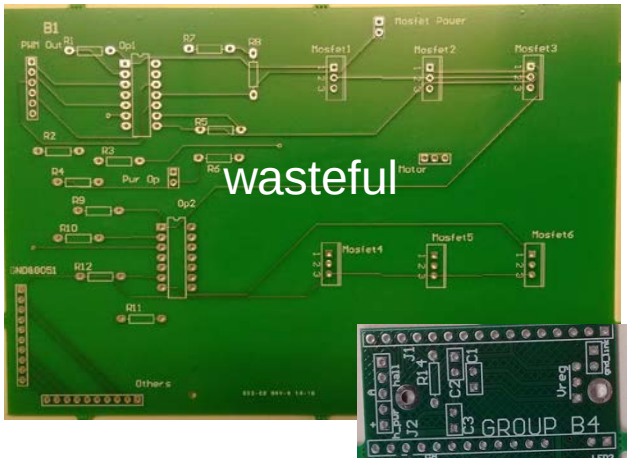
PCB Hole Size Editor




PCB Design Best Practices

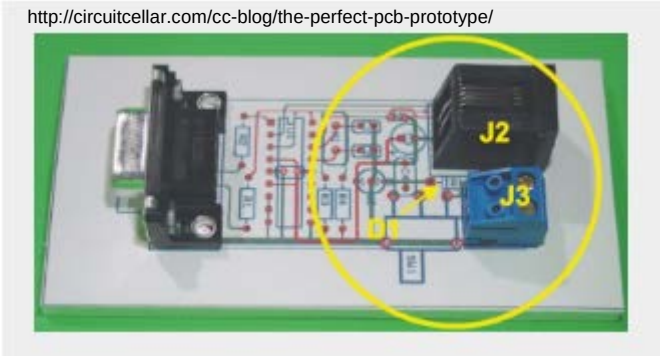
Best Practices: Estimating board size

- 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.



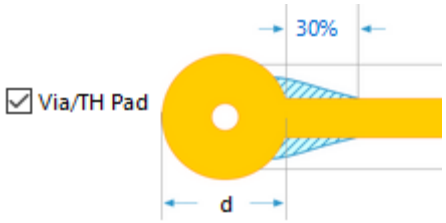
Best Practices: Floor planning

- 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-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

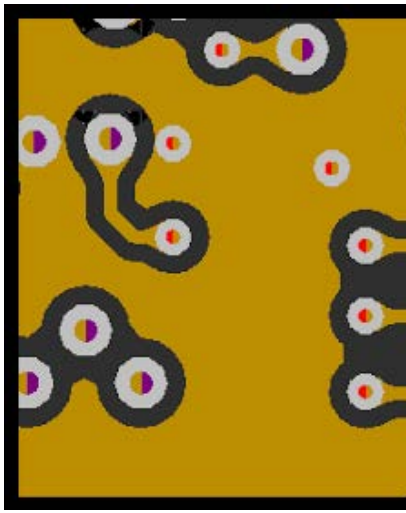
- On two sided boards keep traces perpendicular as much as possible
- Avoid 90 degree bends in tracks (?) (reduced chances of acid traps)
- Keep traces a short as possible
- Always connect a trace to the center of the pad
- Use teardrops (Tools >> tear drops), and use vias to avoid lockout
- Do not place vias under SMD pads



- Layout first all critical traces

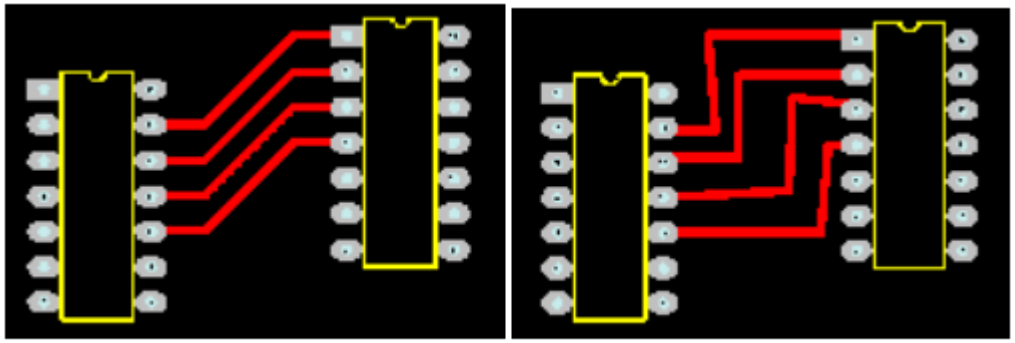
  e.g. CLK, diff pairs, controlled length

- Polygons as fills:
  Connect to GND (EMC), or do not leave ‘dead copper’



- Rout nicely

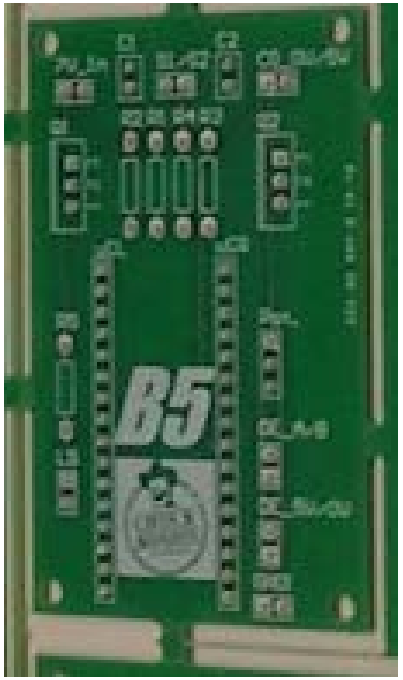
[Ref 3]

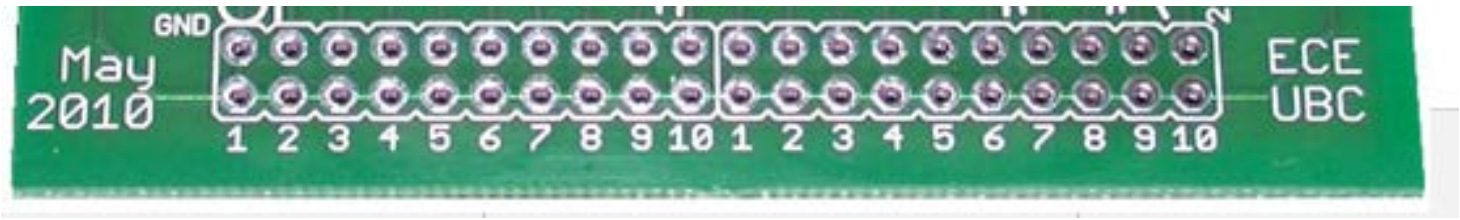


An example of GOOD routing (Left) and BAD routing (Right)

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

Ref [B1]

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

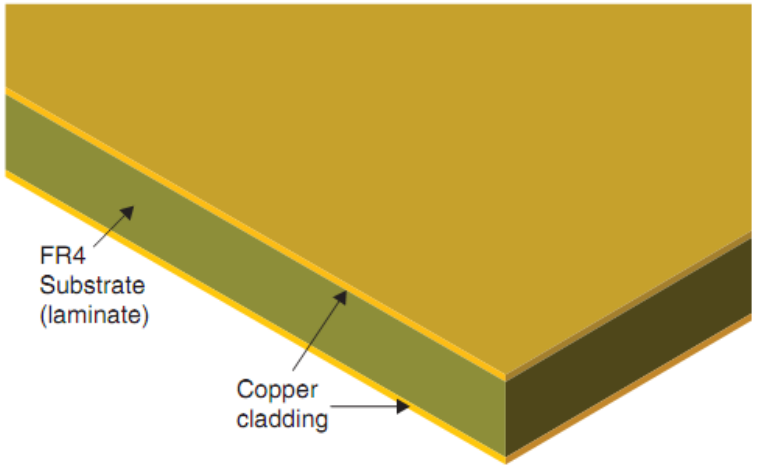


Figure 1-2 A double-sided copper clad FR4 substrate.

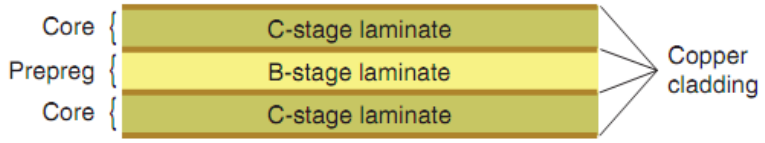


Figure 1-3 Cores and prepreg.

Cu thickness measured in weight oz/ft^2

½ oz → 0.7mils

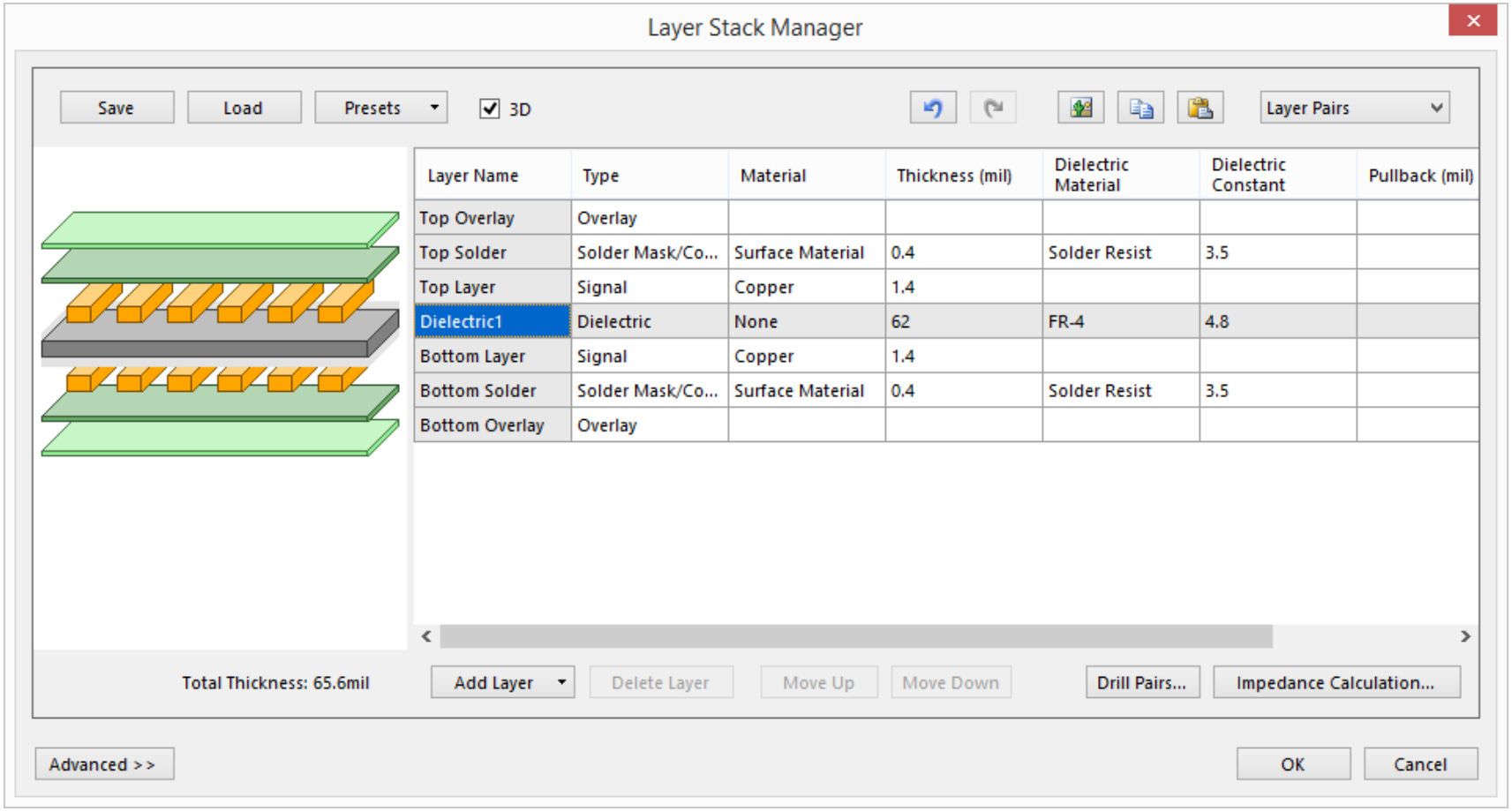
1 oz → 1.4mils

2 oz → 2.8mils

1mil = 25µm

PCB Anatomy: Layer Stackup

Design >> Layer Stack Manager …



Ref [B1]

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

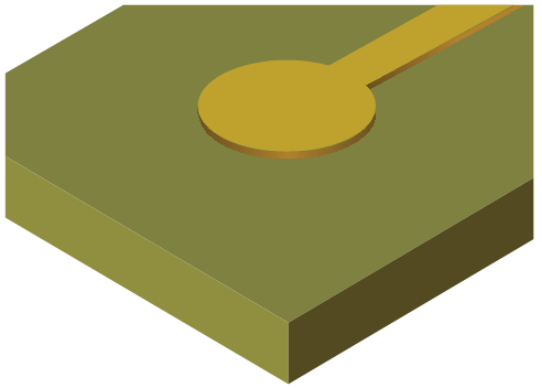


Figure 1-11 Copper pad and trace after etching and resist stripping.

Negative view:
Copper planes, Drill holes, Solder Masks

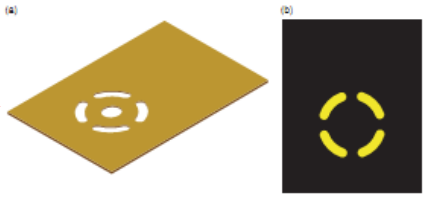


Figure 1-18 Copper in a plane layer (negative view without drill info). (a) Copper plane with thermal relief. (b) Negative view in Layout.

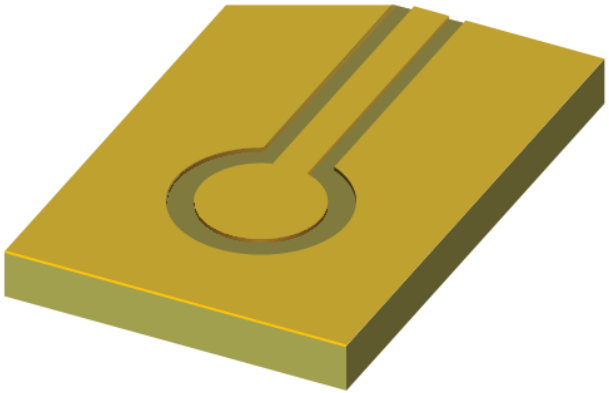
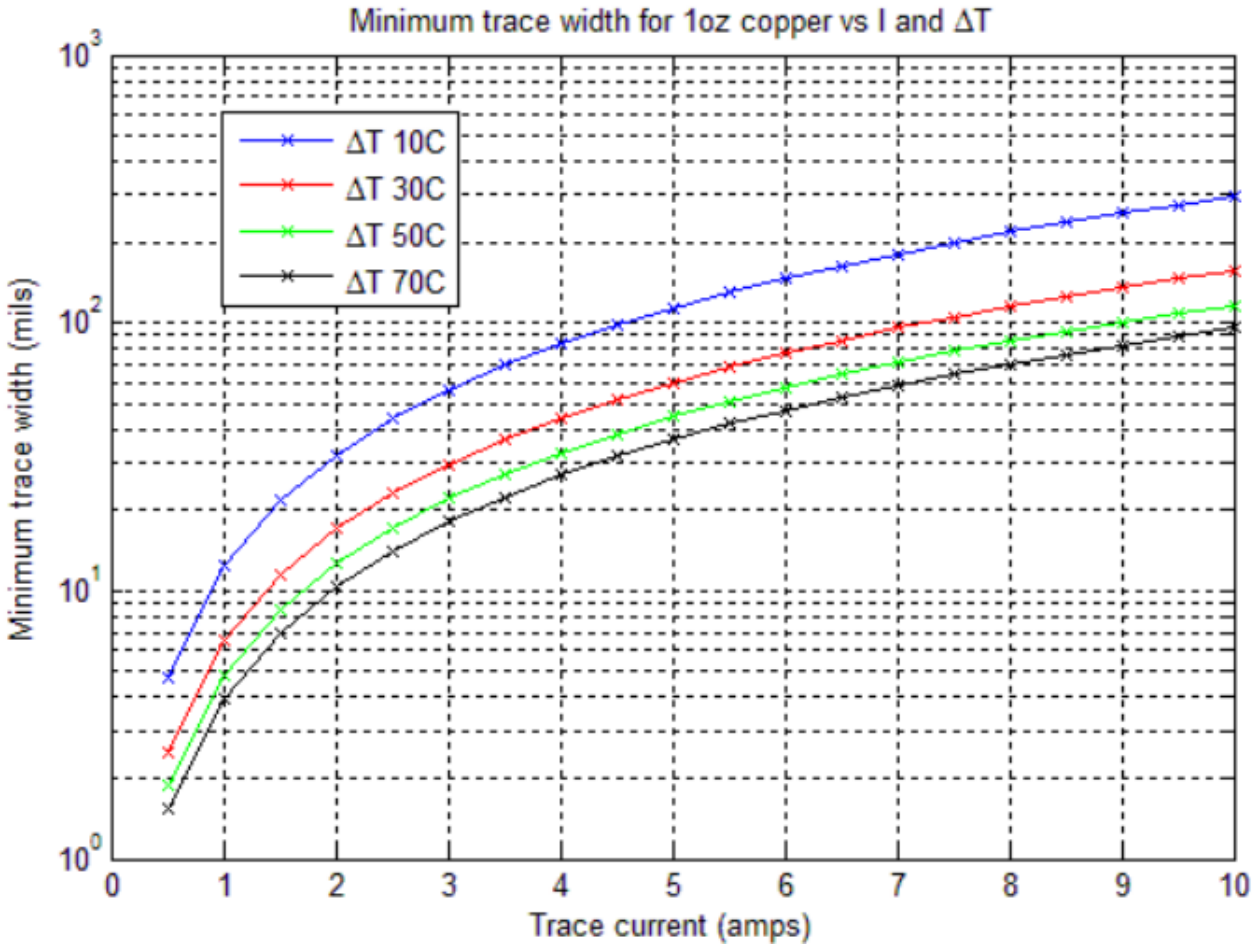


Figure 1-12 A mechanically milled trace.

PCB Anatomy: Trace width

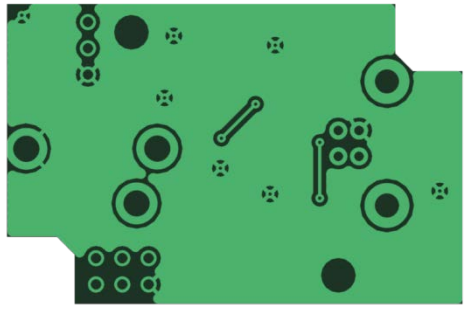


Use the following online trace width calculator:
http://circuitcalculator.com/wordpress/2006/01/31/pcb-trace-width-calculator

Ref [B1]

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



Source: wikipedia.org: Thermal pad

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

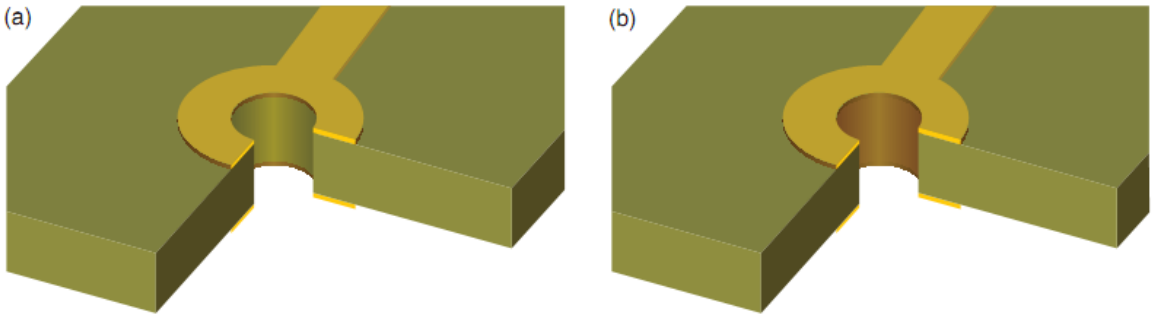


Figure 1-13 Holes are drilled into the board and then copper plated. (a) A nonplated through-hole. (b) A plated through-hole.

PWR and GND planes are commonly inner layers

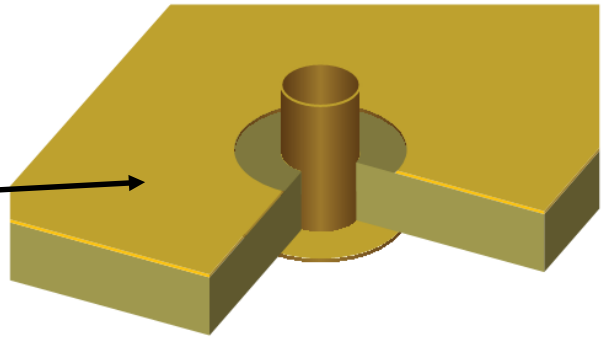


Figure 1-15 A clearance area provides isolation between a plated hole and a plane.

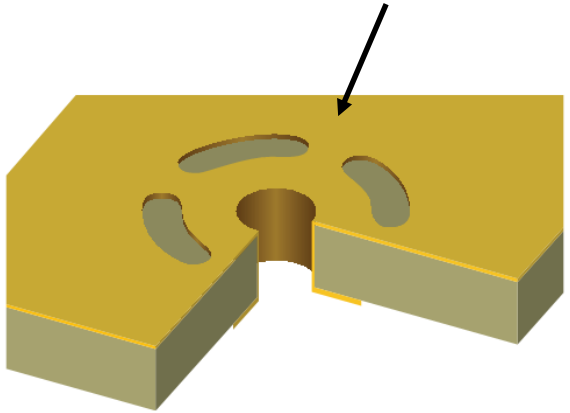
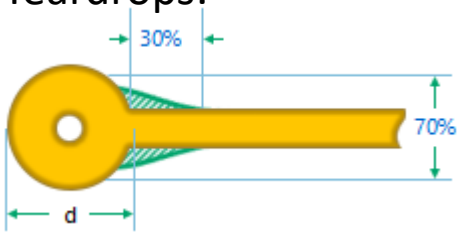


Figure 1-14 A connection to a plane layer through a thermal relief.

Teardrops:



Ref [B1]

PCB Anatomy: Vias

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

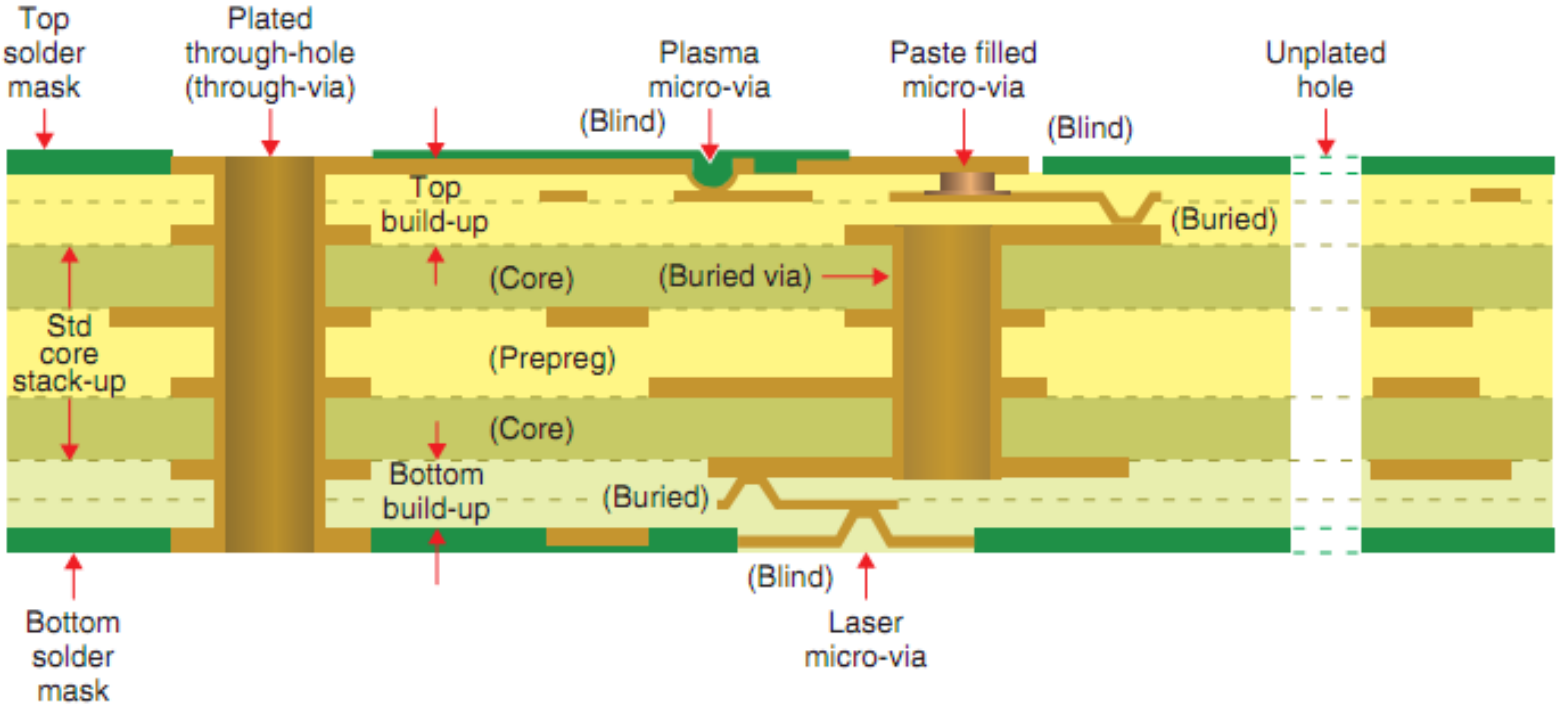


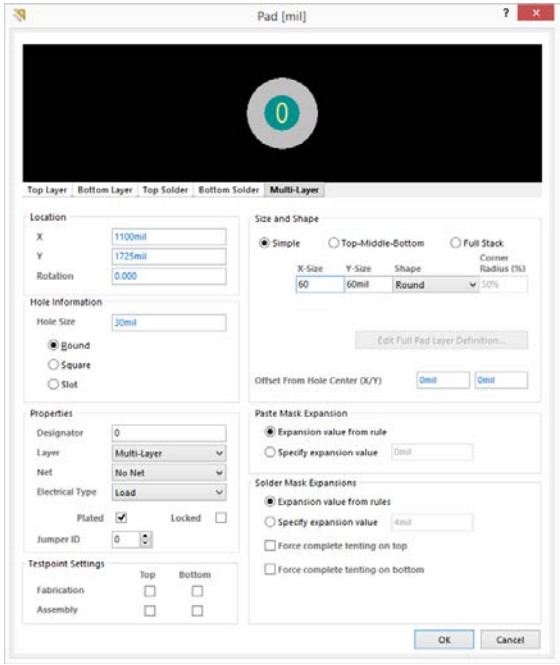
Figure 1-5 A built-up, multitechnology, PCB stack-up.

Ref [B1]

PCB Anatomy: Holes

Holes can be:

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



Altium pad properties dialog

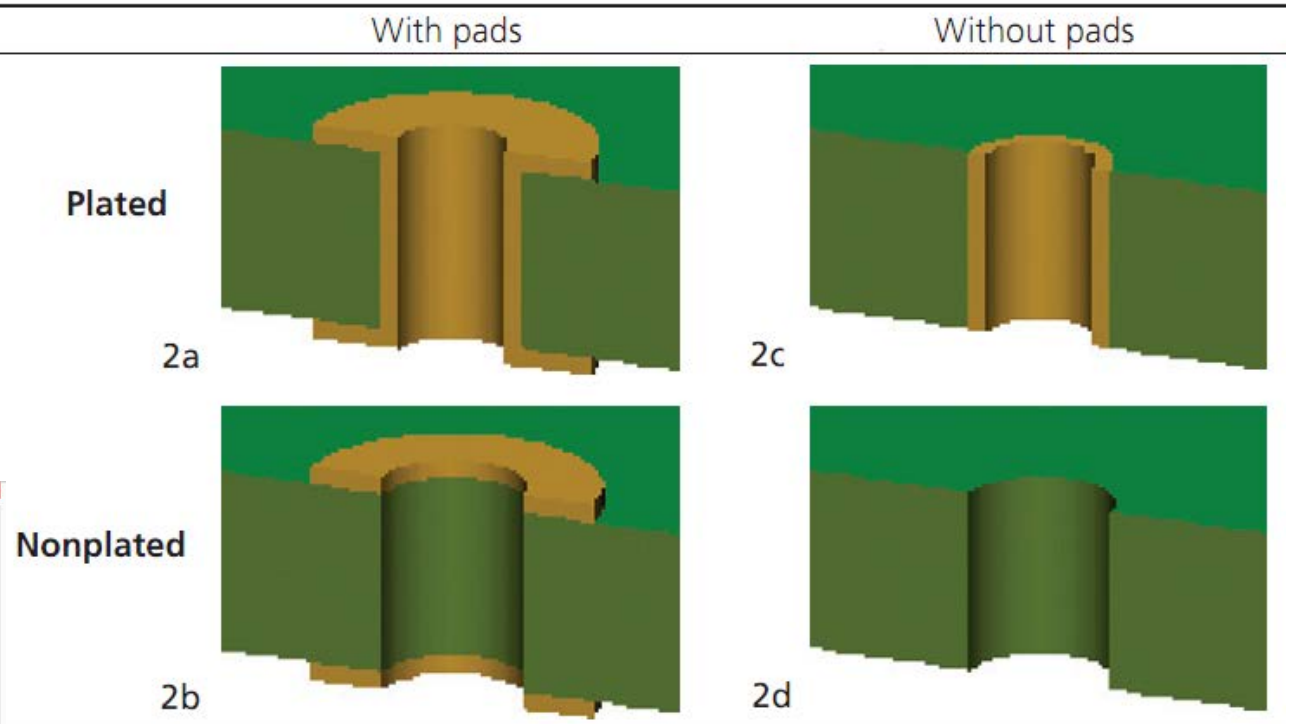


Table 8-2 Basic hole types

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

Plating reduces hole size by 0.003”

Ref [B1]

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

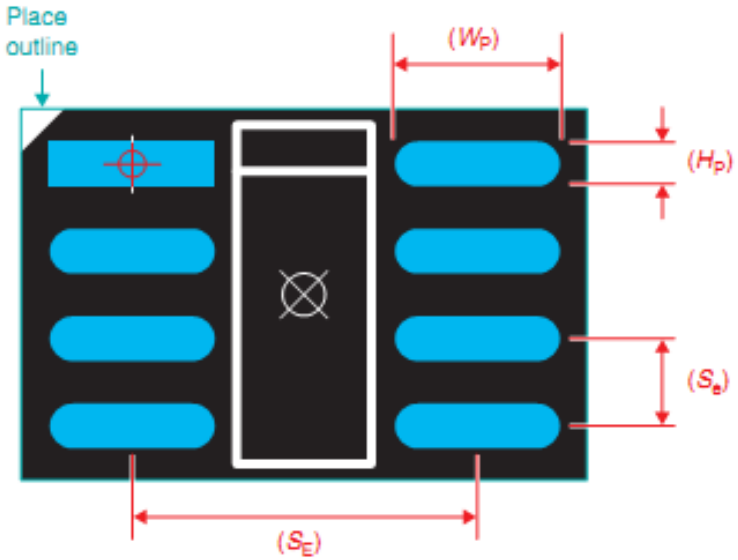


Figure 5-7 Footprint dimensions (typical convention).

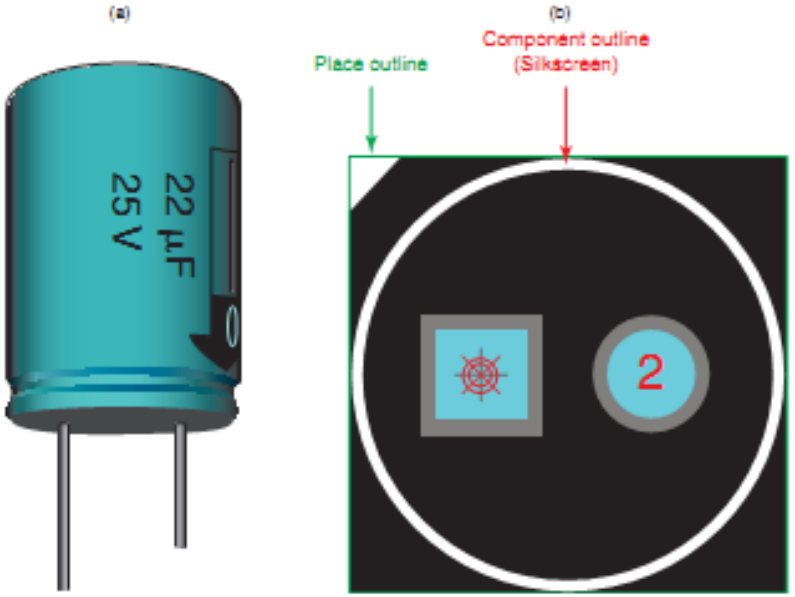


Figure 5-12 Radial-leaded through-hole device. (a) Axial-leaded capacitor. (b) Layout axial footprint

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

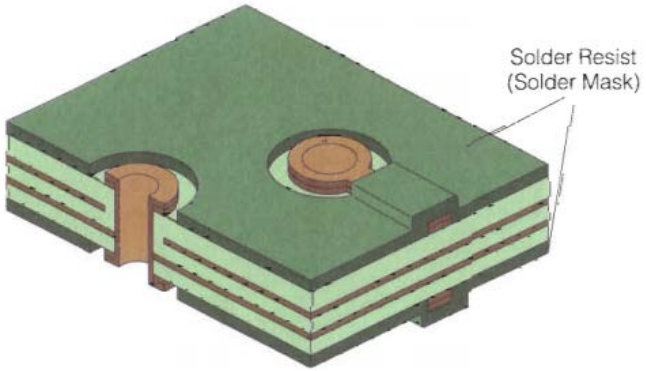


Illustration ML-14. *Apply solder resist.* The specified resist (either dry film, liquid photoimageable, or screen printed) is applied to the surfaces of the PCB or panel.

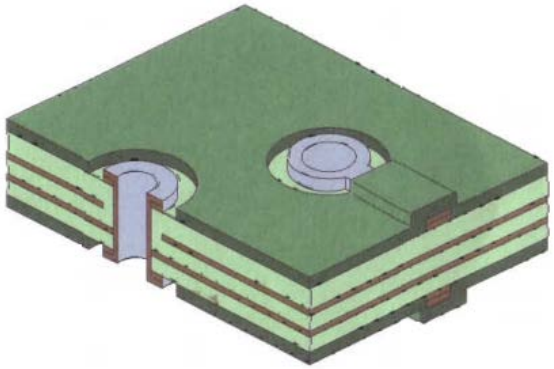
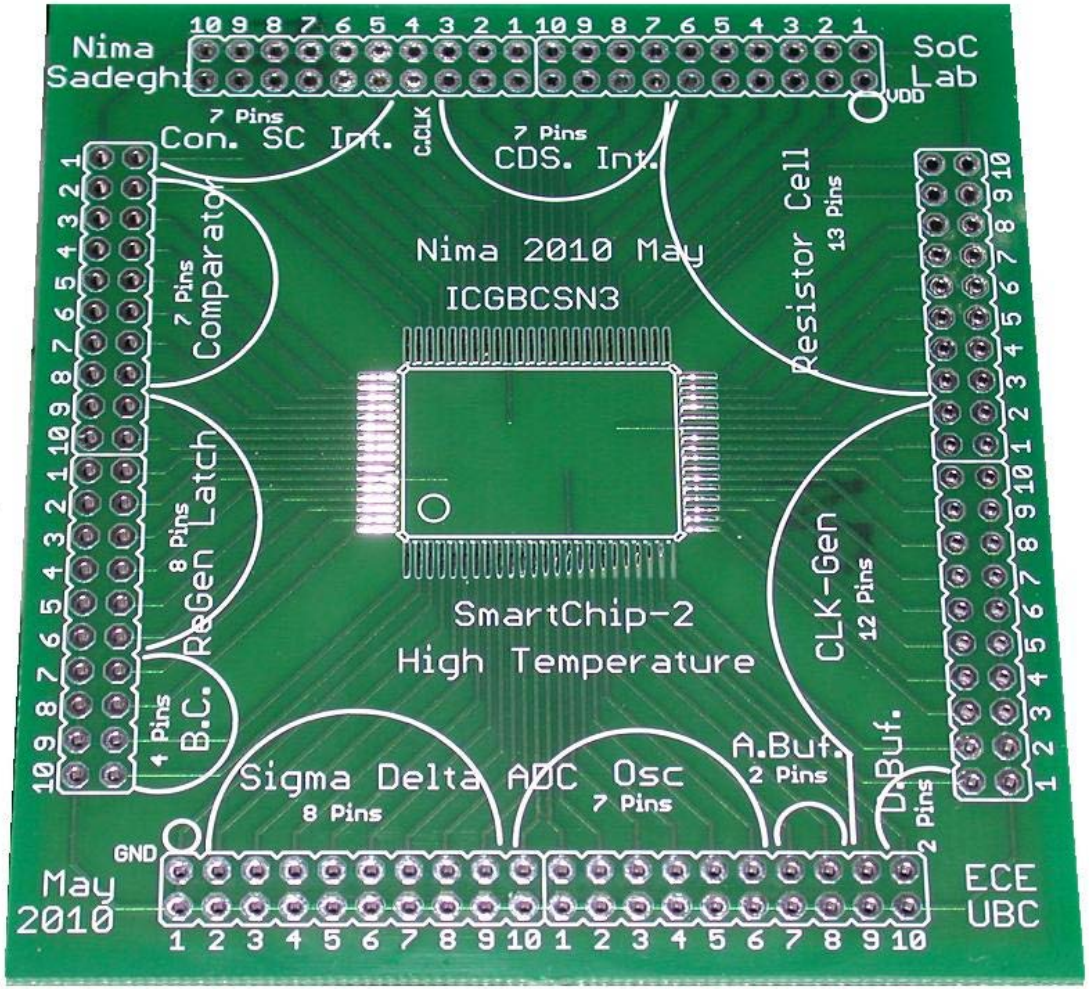


Illustration ML-15. *Solder coat.* Solder (tin/lead) is applied to the exposed copper areas, and the excess solder is removed.

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)

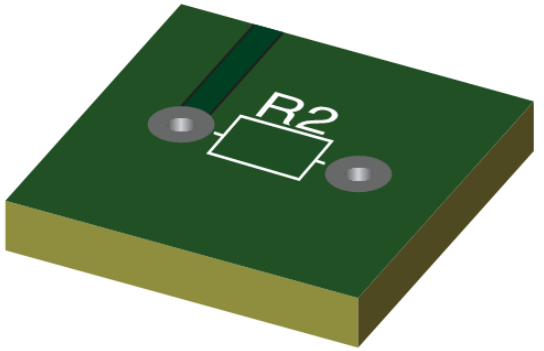
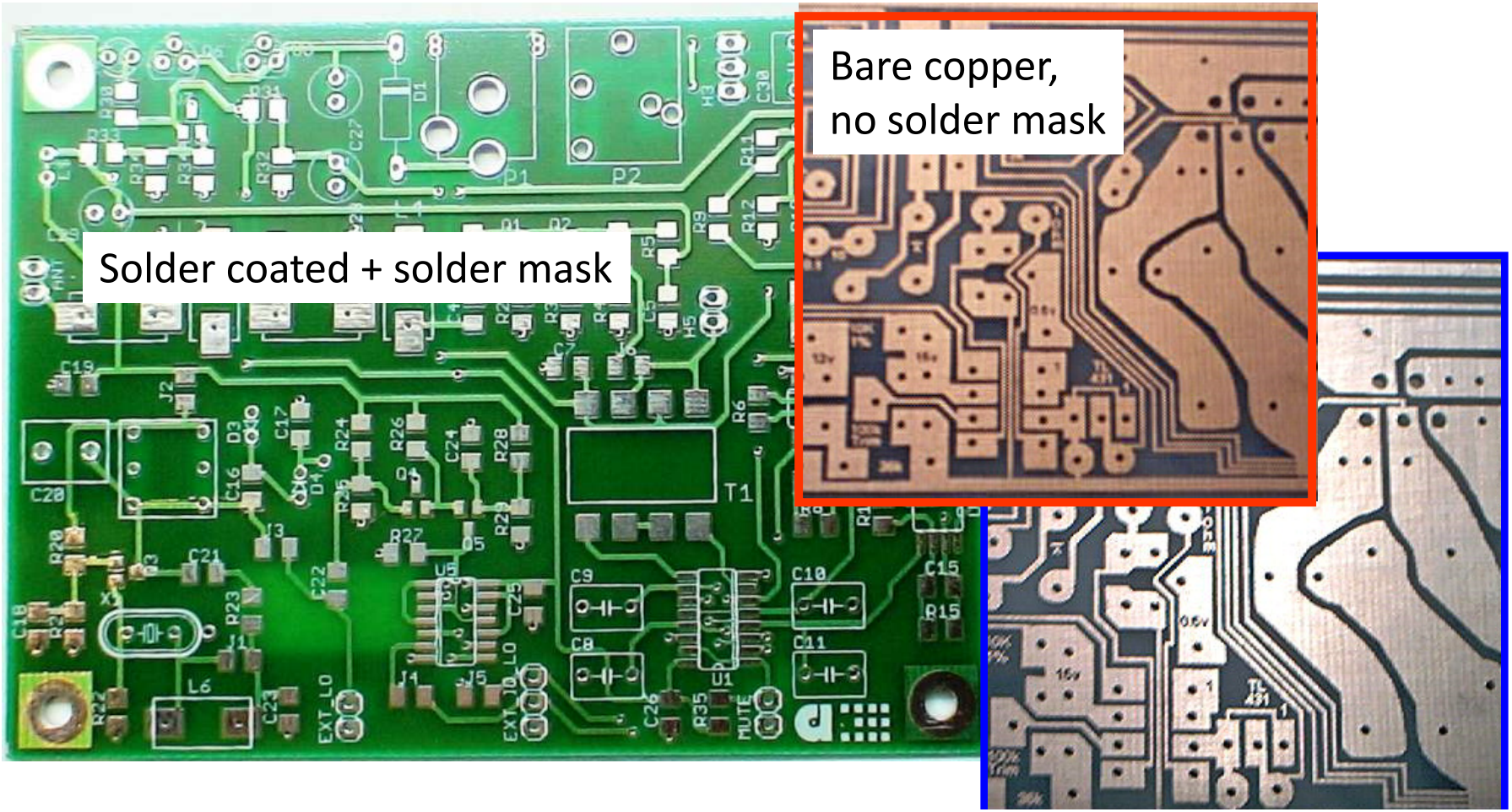


Figure 1-16 Final layers are the soldermask (green) and silk screen (white).

PCB Anatomy: Solder coat / thinning



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