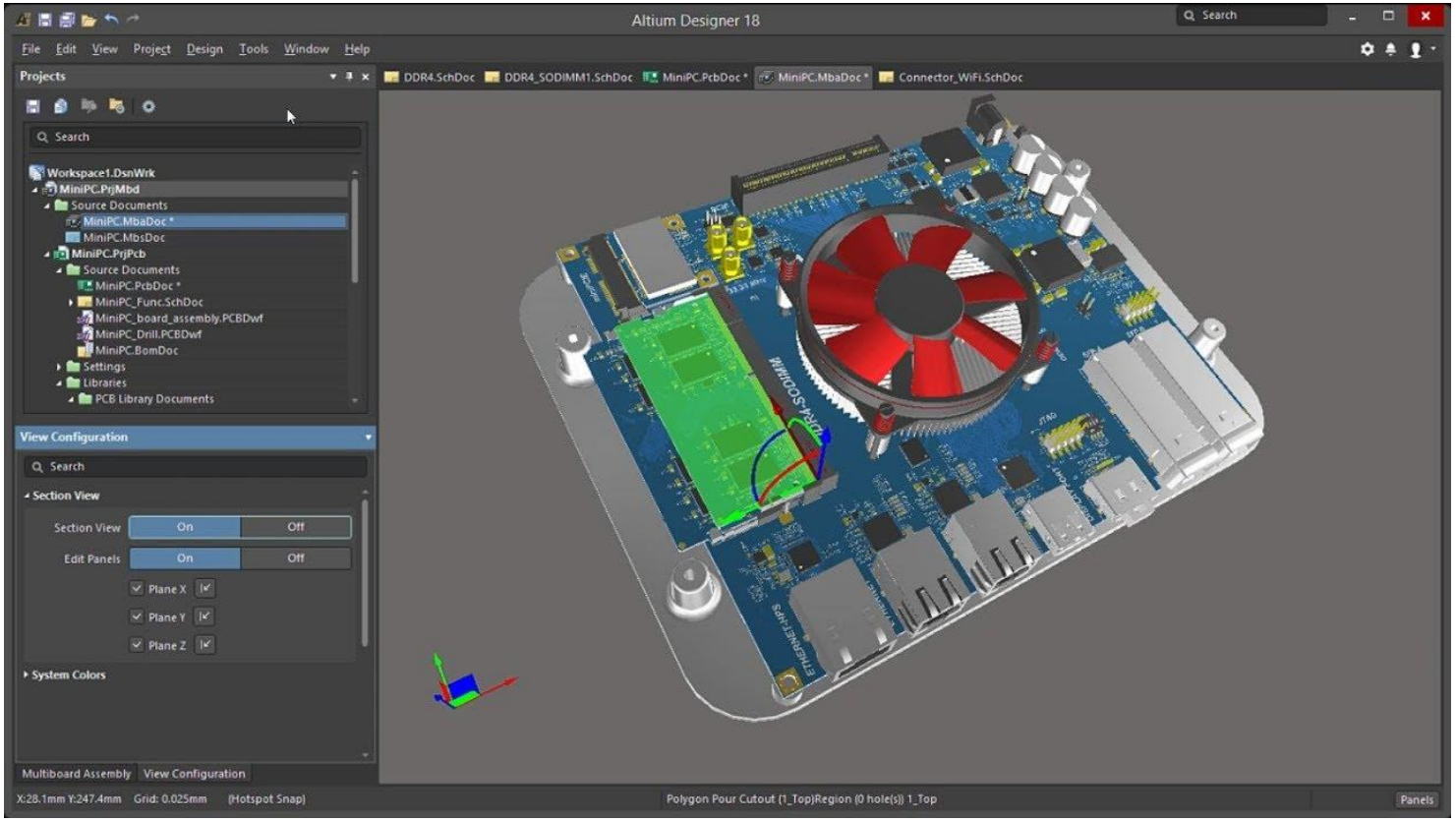


Introduction to PCB layout

ELEC391 - Spring 2020

PCB Design support for ELEC391:

Altium 2018, 150 licenses

Talks:

- Jan 21 Introduction to PCB layout
- Mar 16 (TBC) Invited talk: PCB Fabrication

Fabrication deadlines, Mondays:

Feb 3, Feb 10, Feb 17, Feb 24, Mar 2, Mar 9, Mar 16

Support & submission instructions posted here

Contents

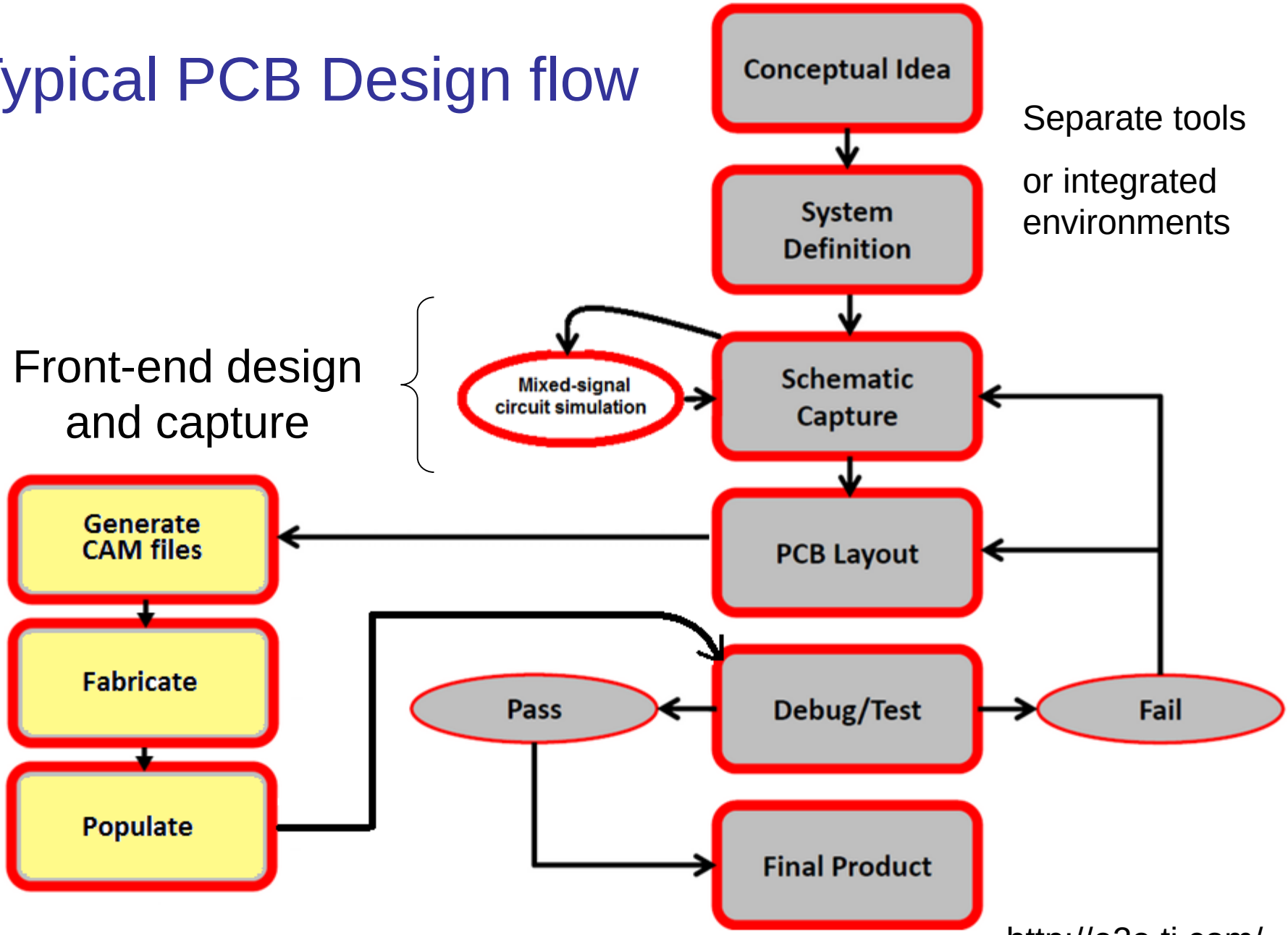
- PCB design flow
- How to install Altium Designer 2018
- Overview of Altium Designer
- Design example as backdrop to introduce PCB concepts
- Instructions for ELEC391 fabrication submissions
- Reference section: PCB design best practices

Credits:

- Unless explicitly stated all source material is from the Altium website and Altium training documents.
- [B1] Complete PCB Design Using OrCad Capture and Layout \ Kraig Mitzner, 2007

PCB Design Flow

Typical PCB Design flow



Separate tools

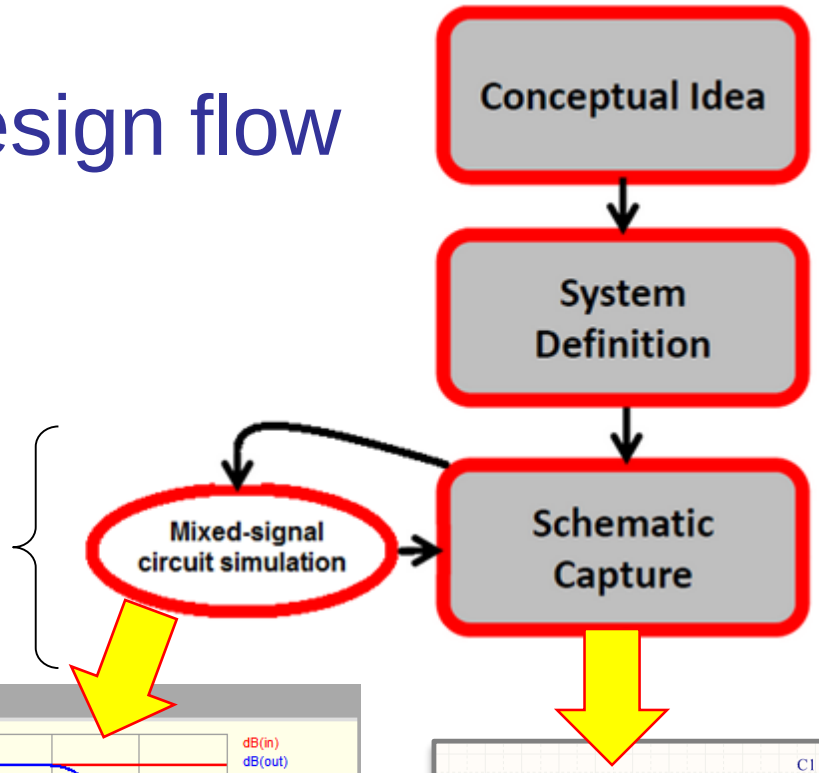
or integrated environments

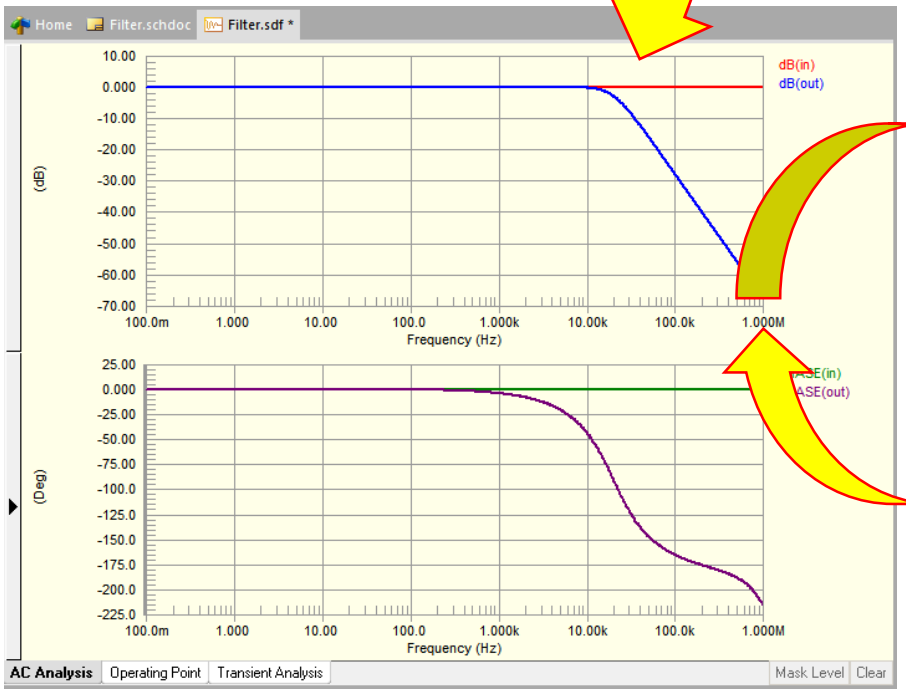
Front-end design and capture

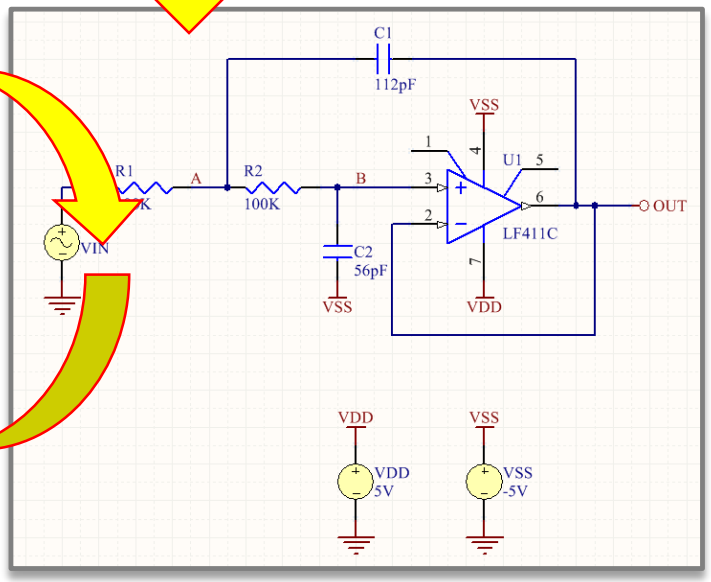
http://e2e.ti.com/

Typical PCB Design flow

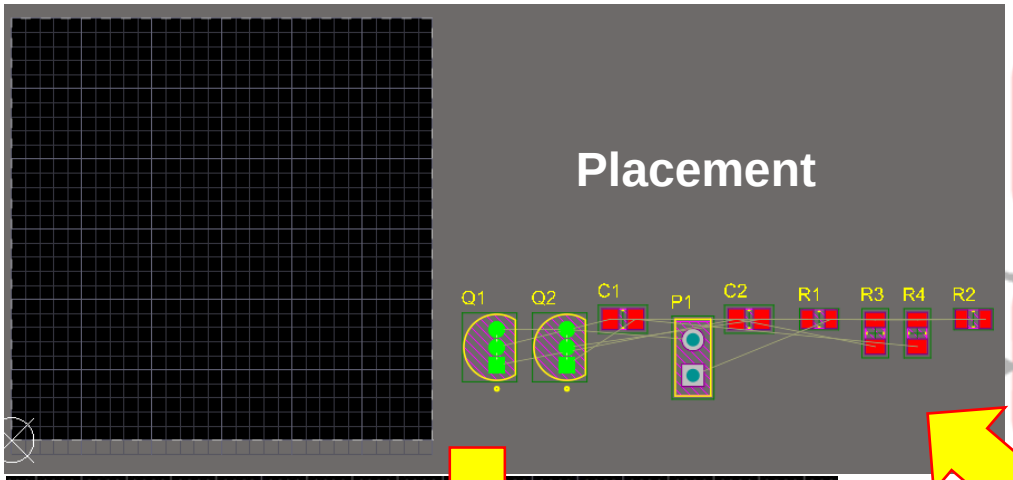
Front-end design and capture

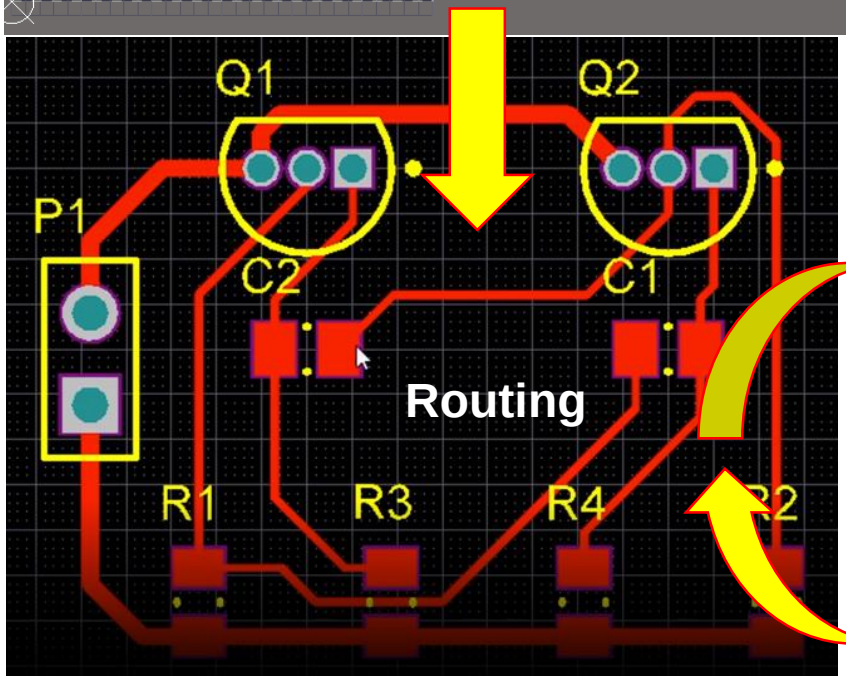


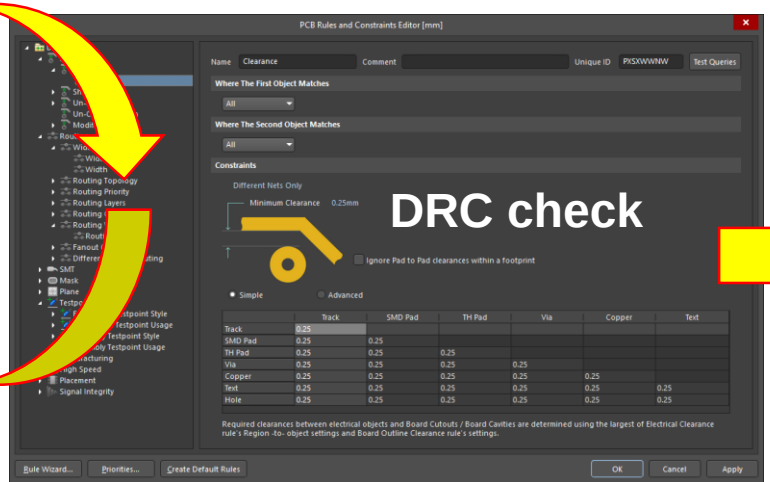




Typical PCB Design flow







Conceptual Idea

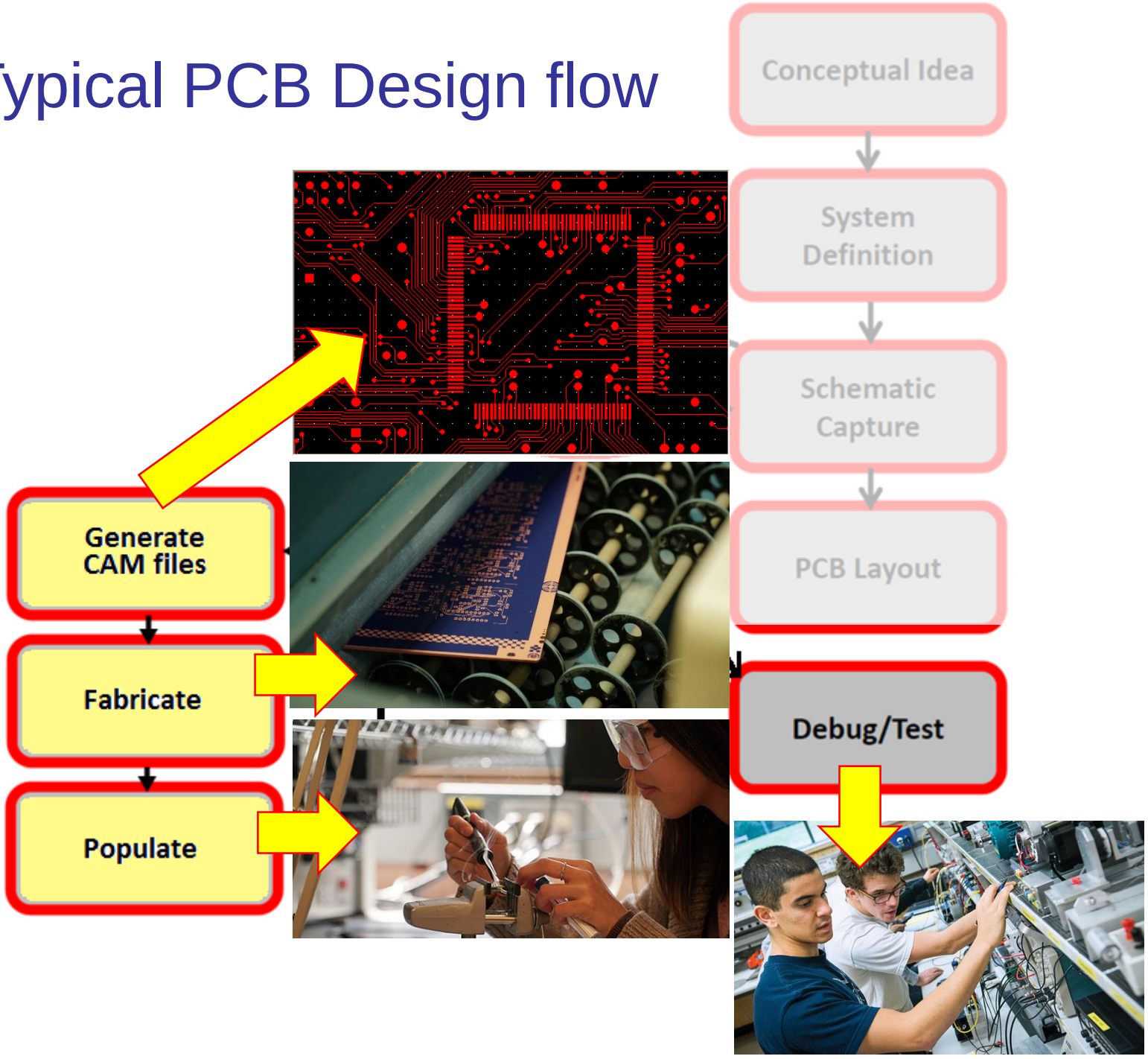
System Definition

Schematic Capture

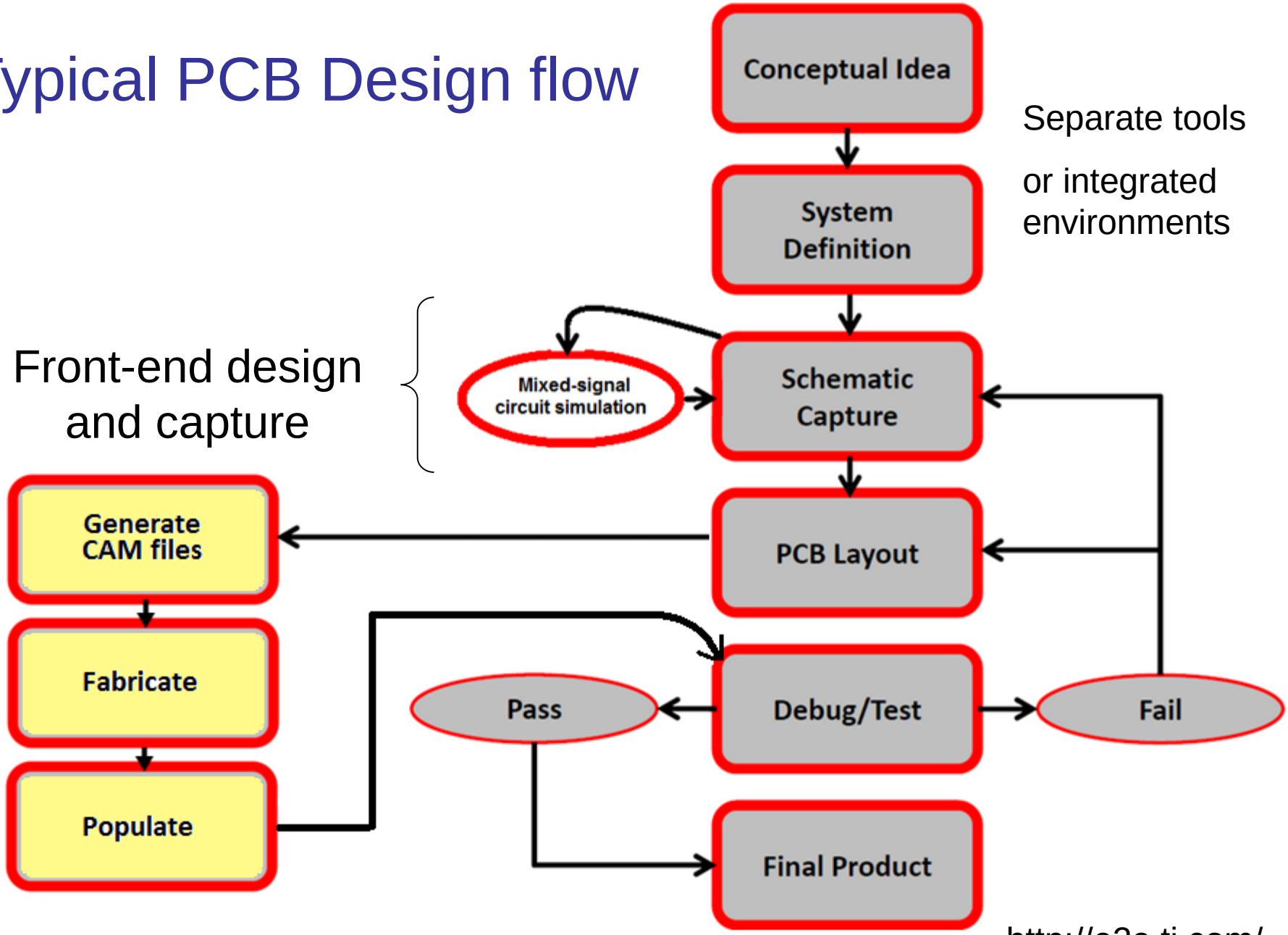
PCB Layout

Physical PCB Design

Typical PCB Design flow



Typical PCB Design flow



Separate tools

or integrated environments

Front-end design and capture

http://e2e.ti.com/

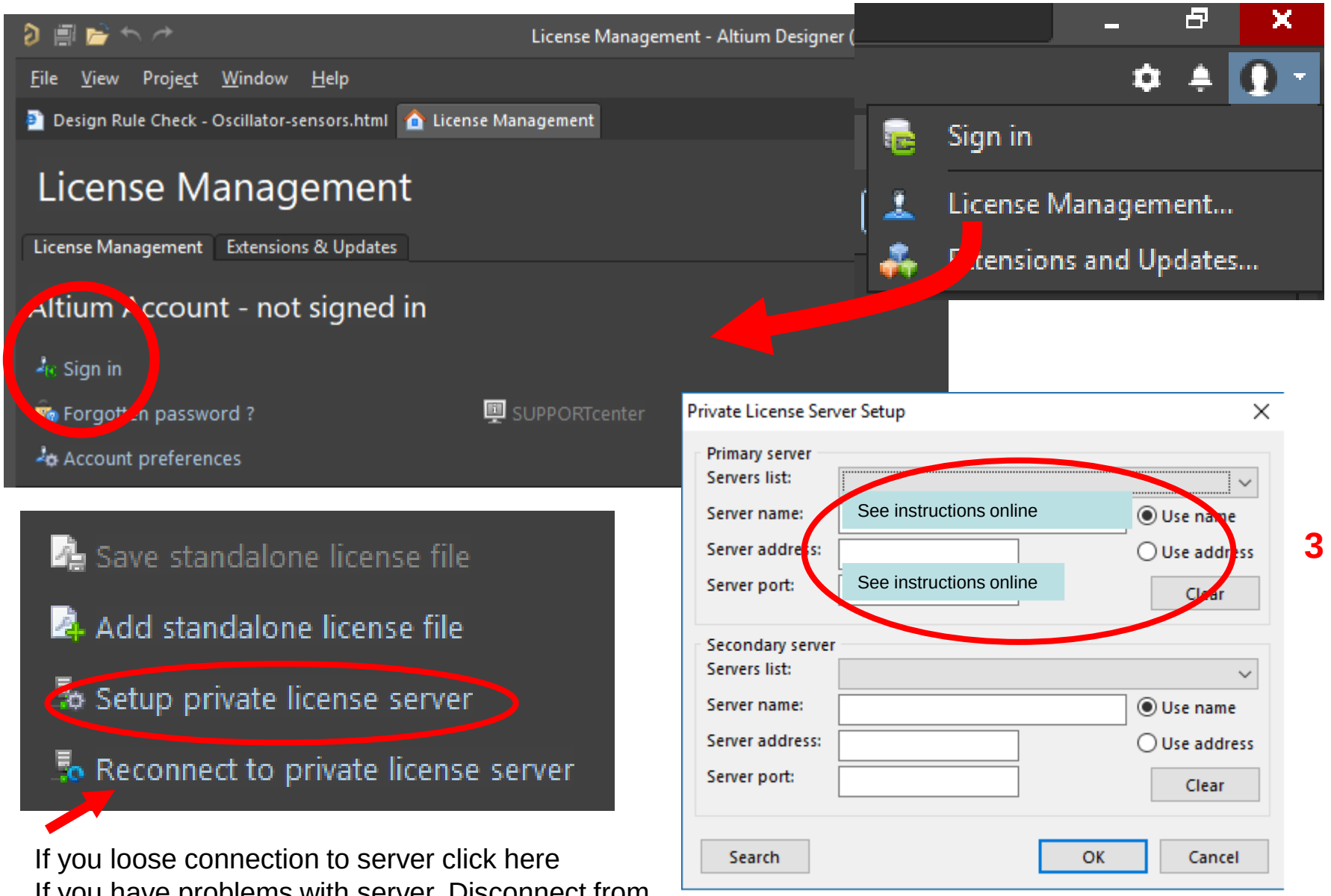
How to install Altium Designer 2018

How to install Altium 2018

- Link to instructions:
  https://www.ece.ubc.ca/~eng-services/Altium/
  - Create an account at Altium Live:
    email: engservices@ece.ubc.ca (fast)
    http://live.altium.com/#signin (slow & not linked to our license)
  - Download Altium v2018
  - Connect to license server

- Access to license server:
  - Lab PC or a wired connection to ECE network
  - Wireless connection at UBC (ubcprivate, ubcsecure)
  - My VPN connection

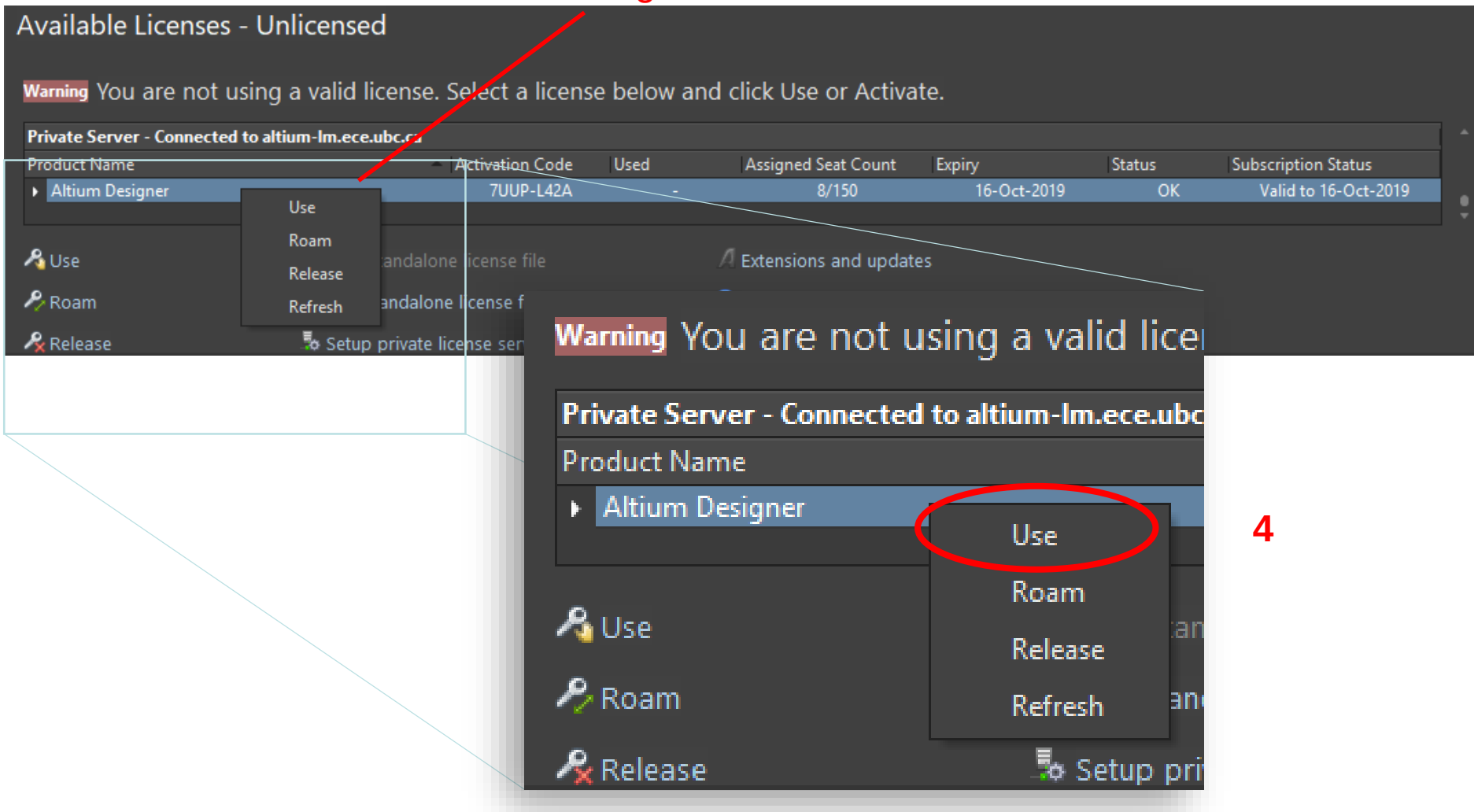
To set the license server



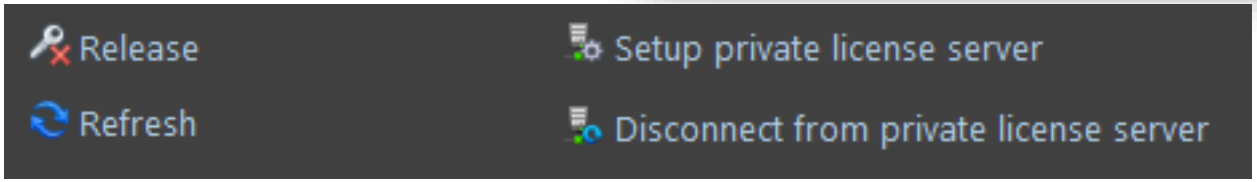
If you loose connection to server click here
If you have problems with server, Disconnect from & Reconnect to server

To use a license

Right click



4



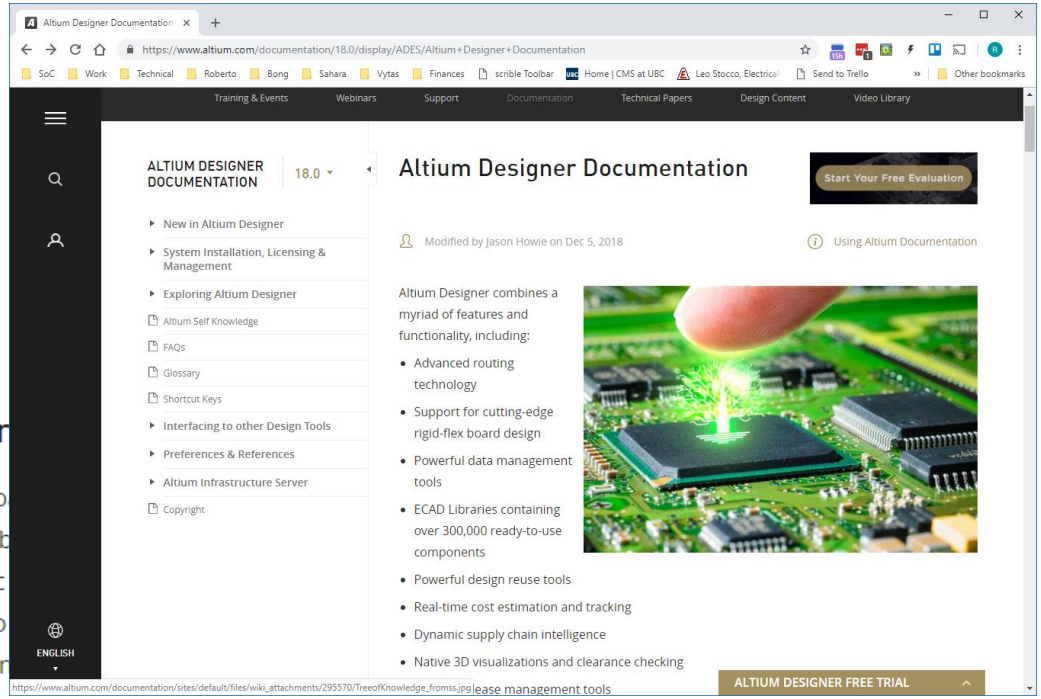
If you are having problems with the license server: Disconnect & Reconnect

Getting started with Altium Designer 2018

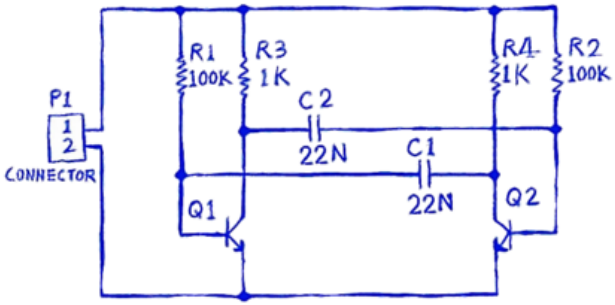
Exploring Altium Designer

Altium Designer documentation:

Do complete this tutorial



Jumping into PCB Design (and Altium Designer



New to bo
where to b
schematic
needed to
componen

The design you will be capturing and then designing a printed circuit board (PCB) for, is a simple astable multivibrator. The circuit - shown to the left - uses two general purpose NPN transistors, configured as a self-running astable multivibrator.

The tutorial touches on many of the areas of the software that you will need to become conversant with, in your day-to-day design work - what better way to familiarize yourself with, and get to know, your powerful design 'partner'.

Best training material is on the Altium website. Info is updated, but beware that menus and options change between versions. ver 2018 <> ver 2016

Overview of Altium Designer

Altium Designer 2018

A unified design environment

System requirements (MS W7, W8, W10)

- Front-end design and capture
- Physical PCB design
- Mixed-signal circuit simulation
- Signal integrity analysis
- Multi-Board Assembly
- Power Integrity Analyzer
- PCB manufacturing

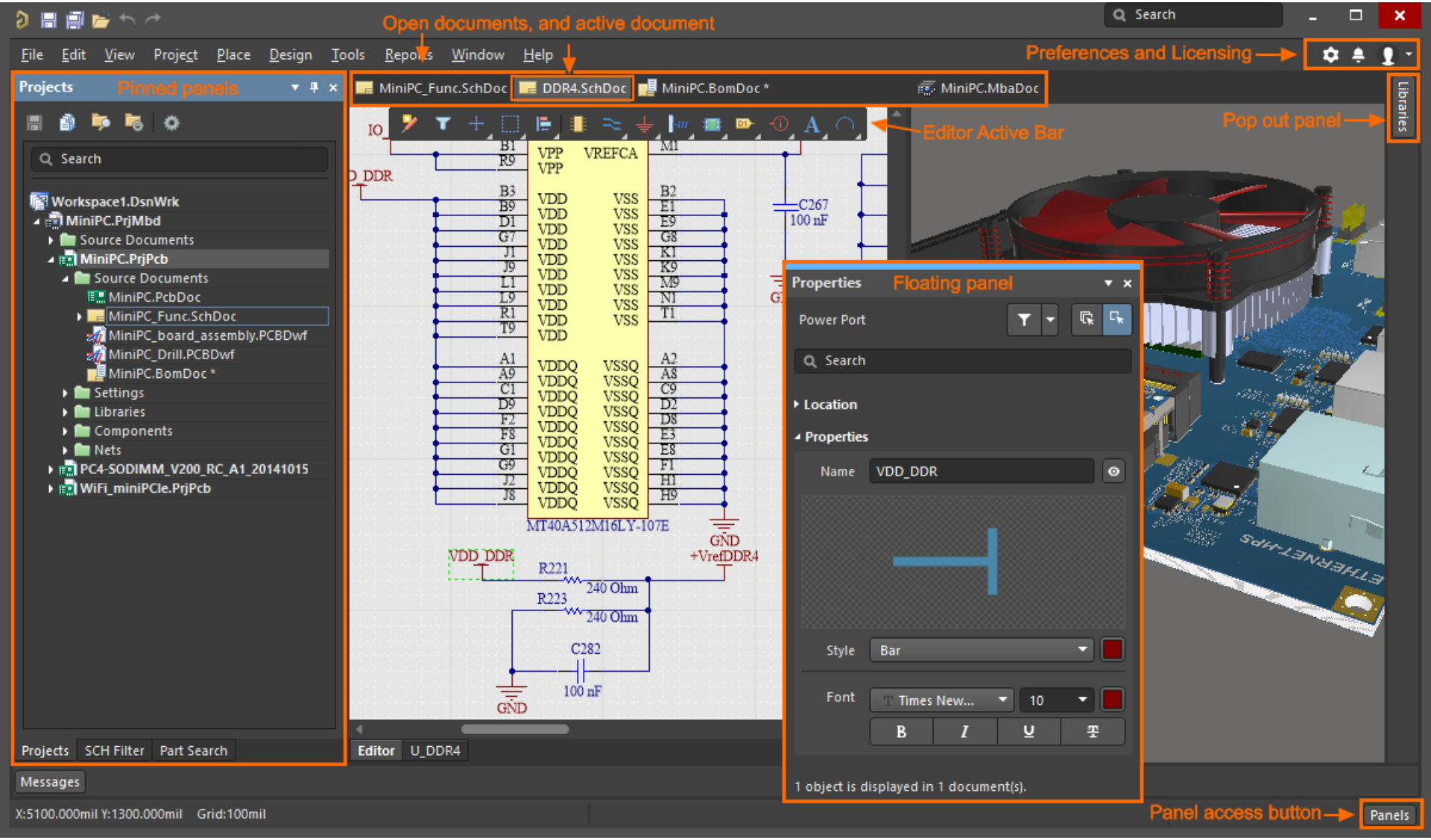
Professional Design Environment

- Unified platform for individual or corporate use
- 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 Vault (dynamic library of components)

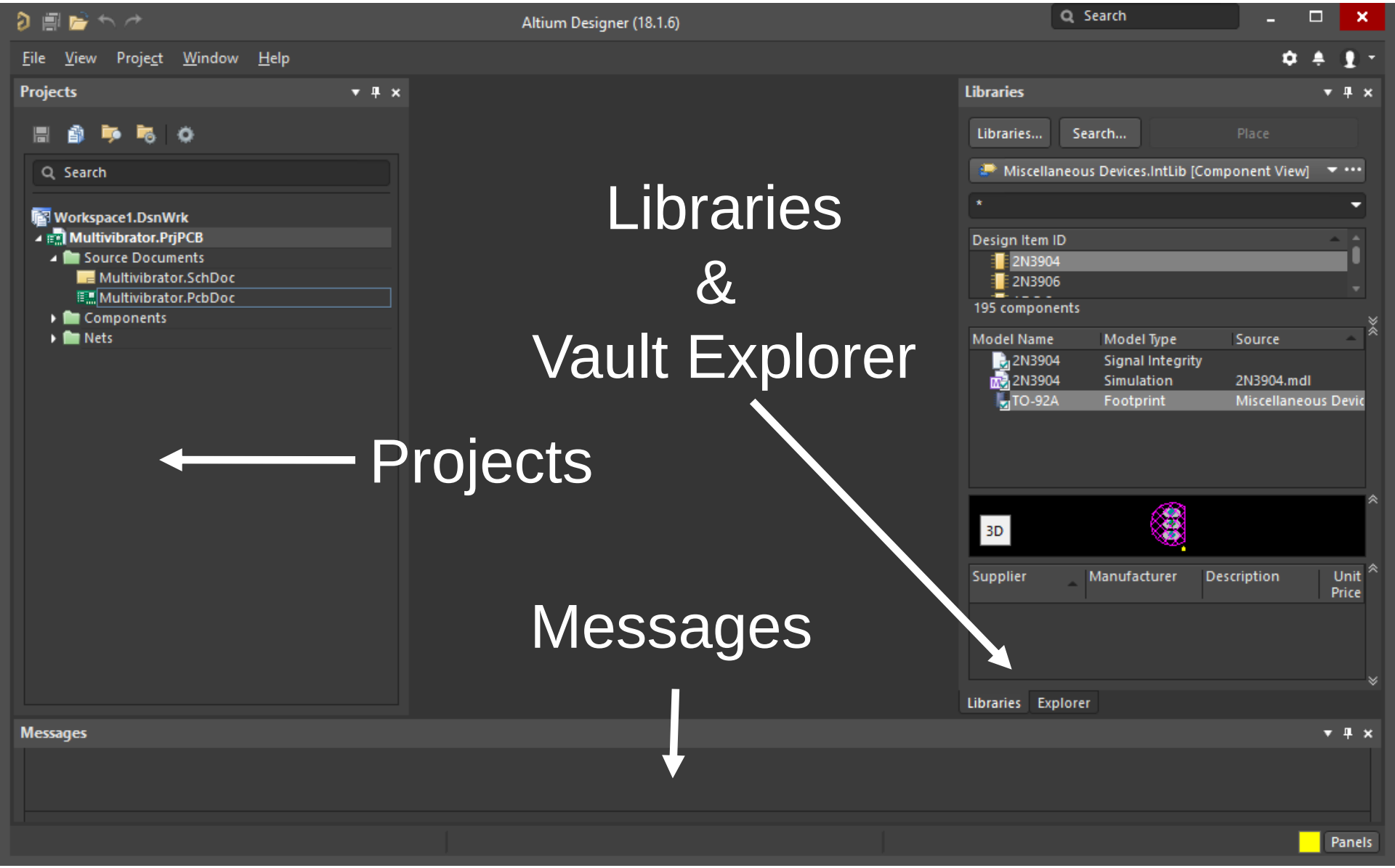
Advice for ELEC391

- Keep it simple
- Focus on schematic entry & layout
- One page schematics
- Locate the right library models for your parts
- Stay away from very small SMD components
- Remember that ECE will take care of CAD file generation and fabrication

Altium Design X2 Environment

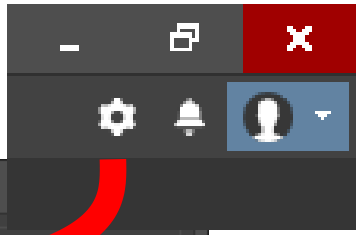


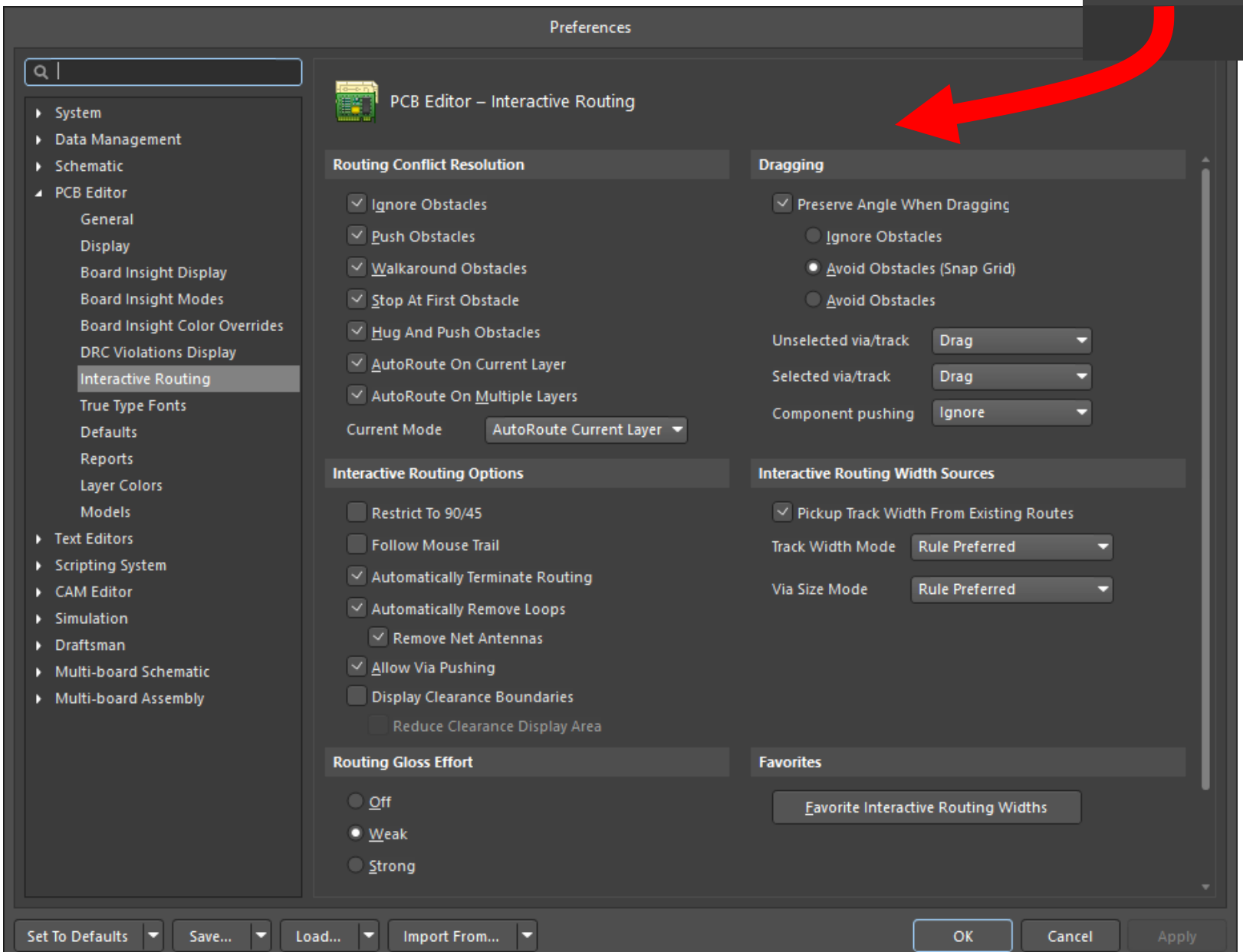
Recommended basic panels



For more help working with panels read this

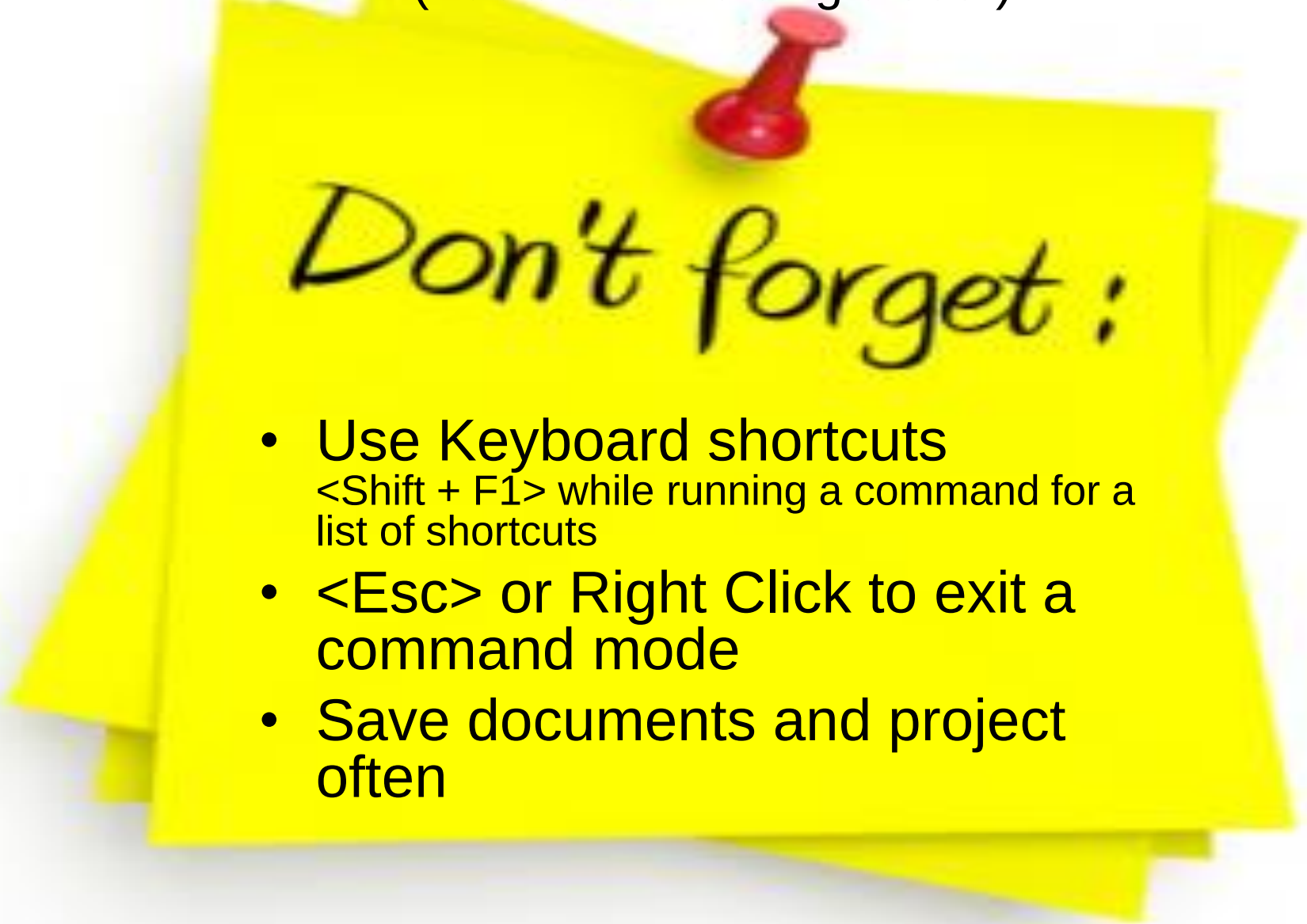
Preferences Dialog





Tips

(Basics for the single user)

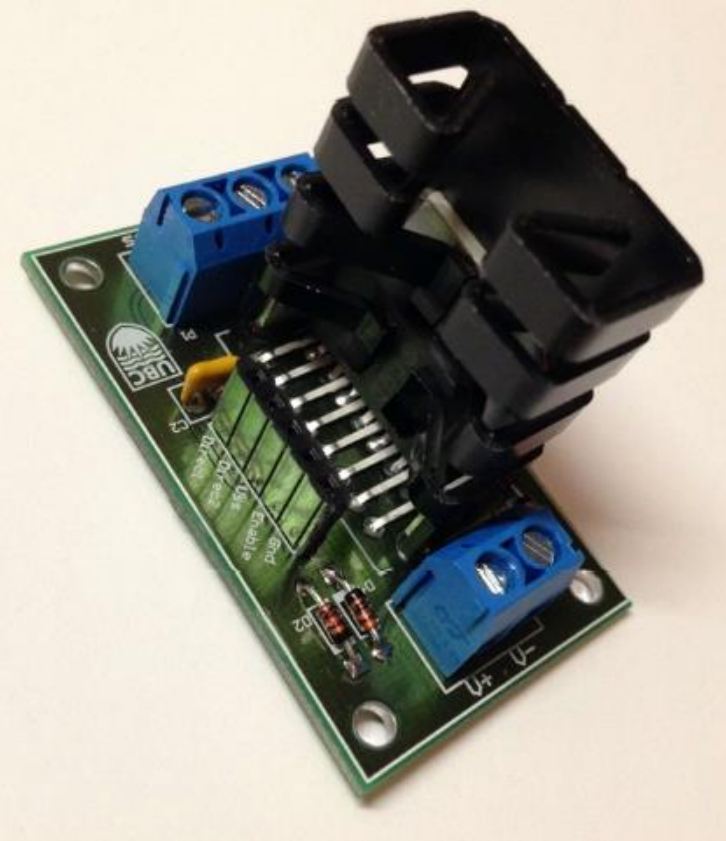


- Use Keyboard shortcuts
  <Shift + F1> while running a command for a list of shortcuts
- <Esc> or Right Click to exit a command mode
- Save documents and project often

Design Example

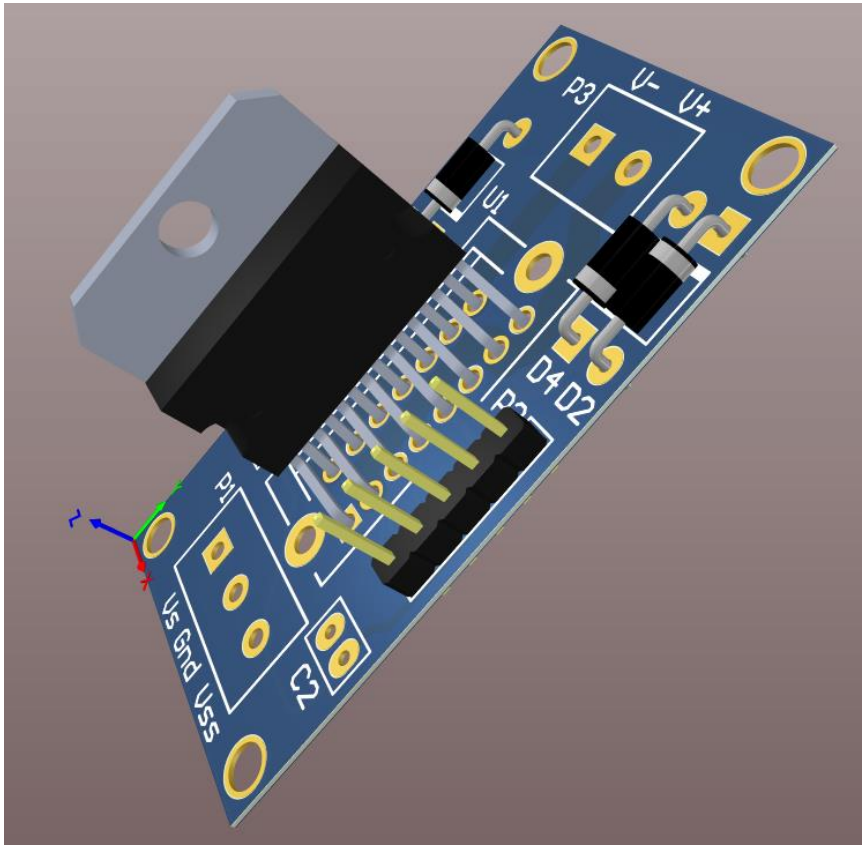
Design example

L298 Motor Driver Board
(by Matt Winship)

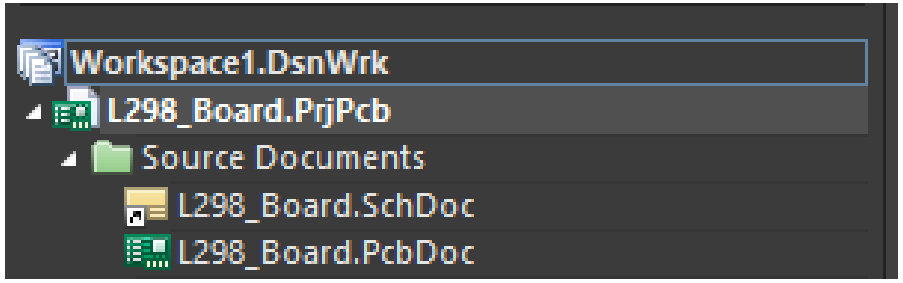


L298_Motor_Driver_Board_Datasheet.pdf

3D View on Layout Editor

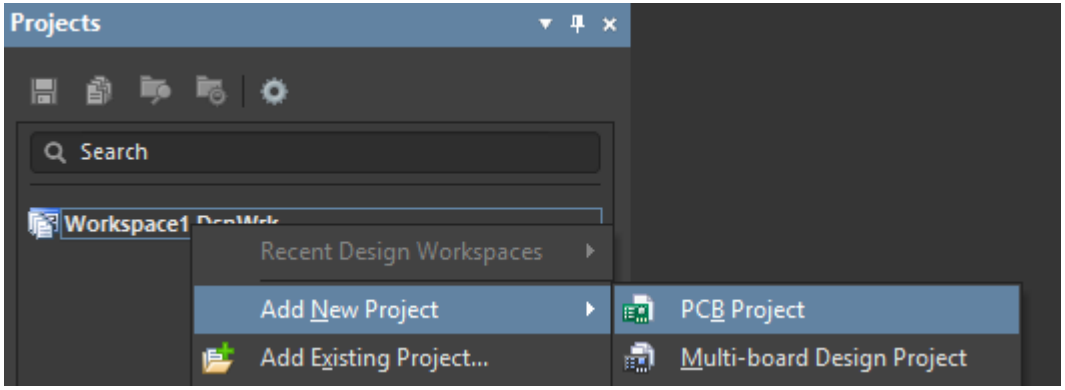


Create project files

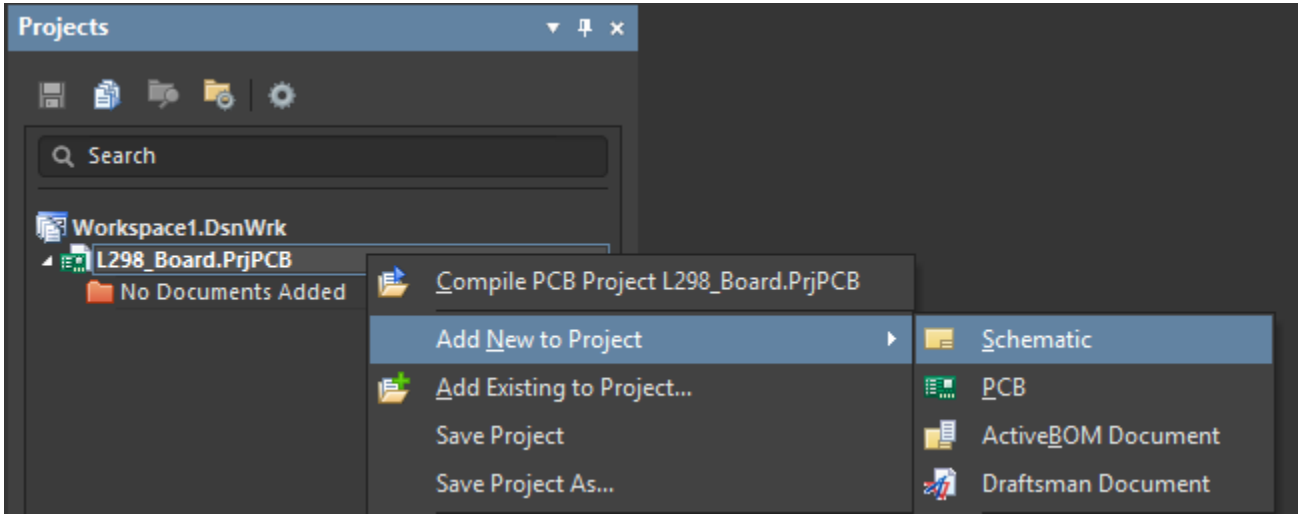


1. Project file

2. Schematic file
3. PCB file



When creating the project file remember to use “Save As” to rename. Otherwise your project will be called “PCB_Project1”

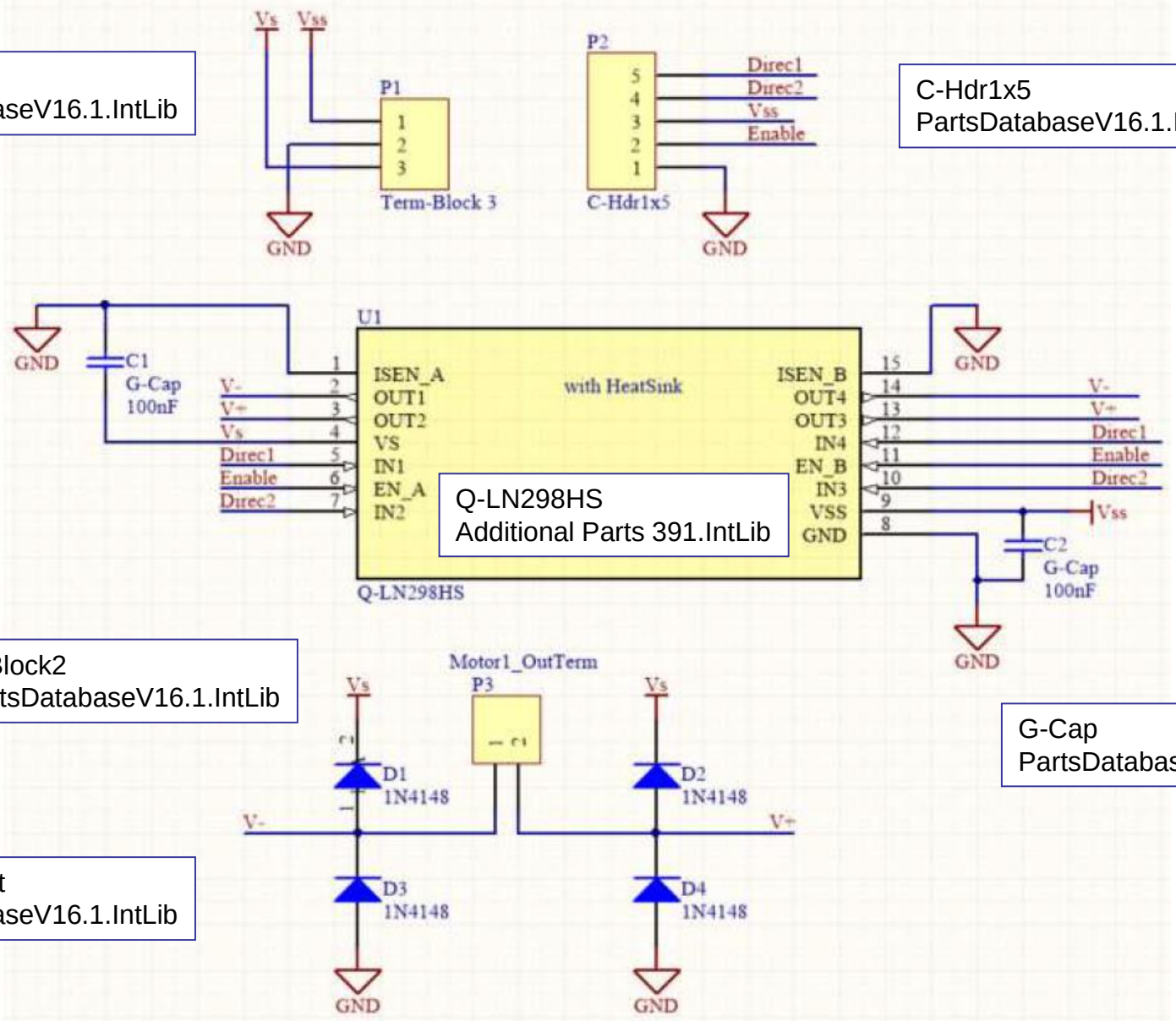


Create Schematic & PCB files.
Remember to rename accordingly.
It is good practice to save all files in the same directory

L298 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

Steps to create the Schematic

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

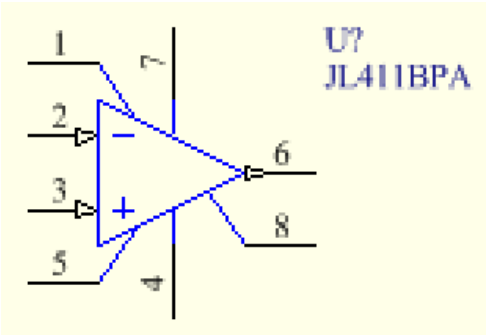
Wiring Tips

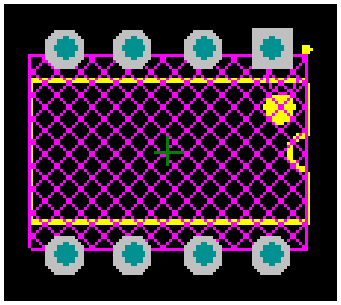
- Left-click or <Enter> to anchor the wire at the cursor position.
- <Backspace> (←) to remove the last anchor point.
- <Spacebar> to toggle the direction of the corner.
- <Shift+Spacebar> to cycle through all possible corner modes.
- Right-click or <Esc> to exit wire placement mode.
- To graphically 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.

About Libraries

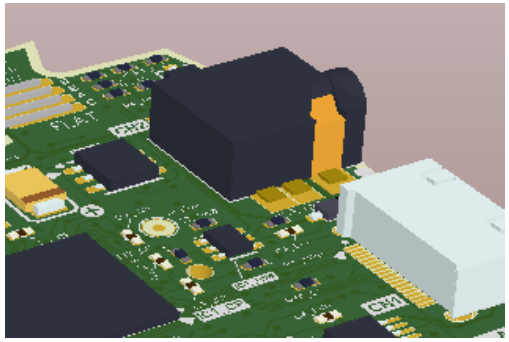
More on: Understanding Models, Components and Libraries

- Component representations for different phases of design:
  - 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
```



Libraries = collection of components

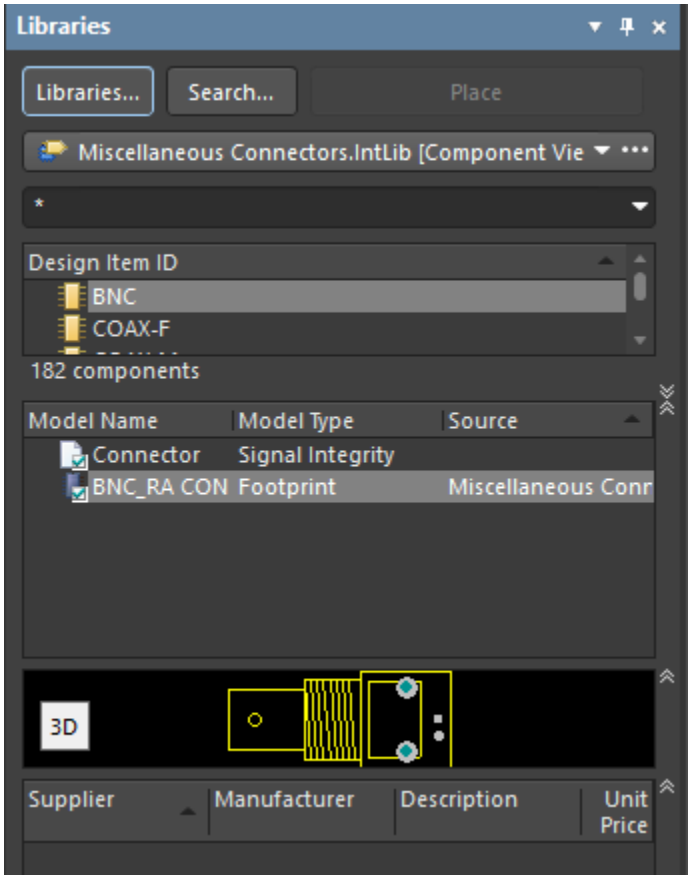
- Collection of components, models or both
- Model Libraries (*.MDL, *.CKT, *.PCBLib)
  - Simulation models are one file per model
  - PCBLib libraries are typically a collection of footprints
- Schematic Libraries (*SchLib)
  - Symbol and a link to a model library
- Integrated Libraries (*.IntLib)
  - Symbol, footprint and other models are compiled into a single portable file to create …
  - Unified components with links to all domain models + parametric information

Obtaining integrated libraries

1. Altium default libraries
   Miscellaneous Connectors
   Miscellaneous Devices
2. Altium Vault
   Cloud dynamic collection of unified components
   includes real-time supply chain information
3. 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\AD18\Library
   or a cloud storage service if you work from more than one PC
4. AltiumLive website: Resources / Design Content
5. IC Manufacturer sites (specially simulation models)
6. ELEC391 libraries from last year: here
7. Make your own

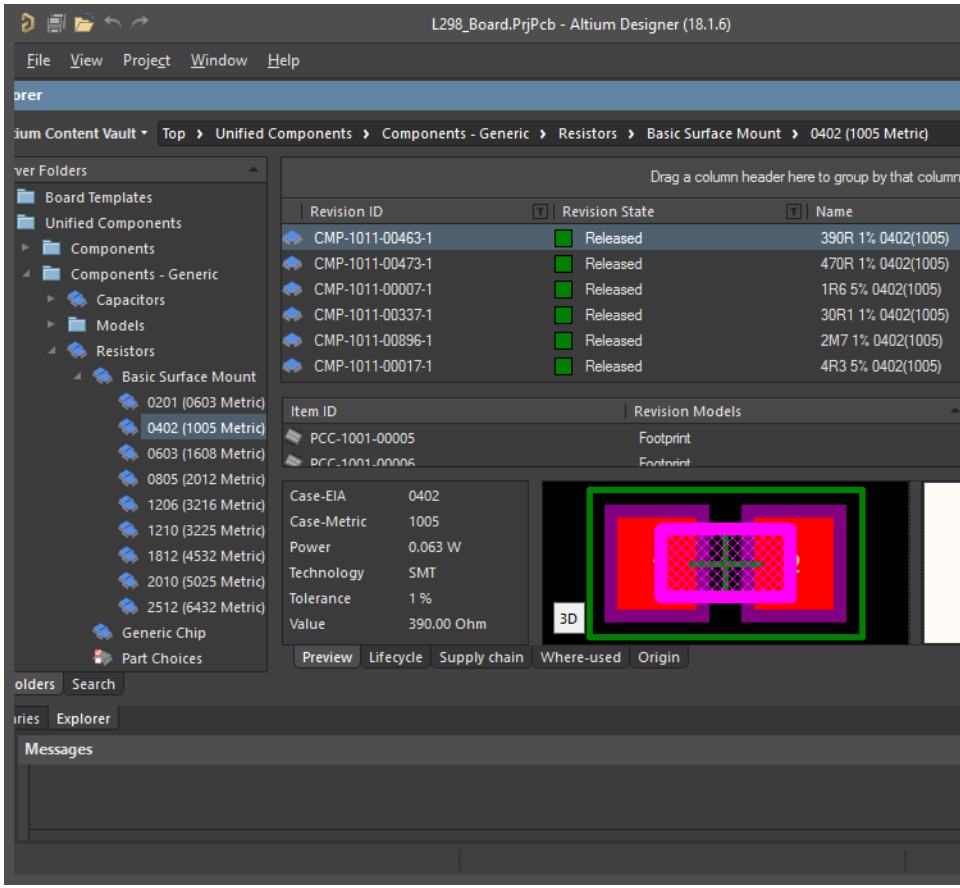
Accessing libraries

1. Library Panel



2. Explorer Panel

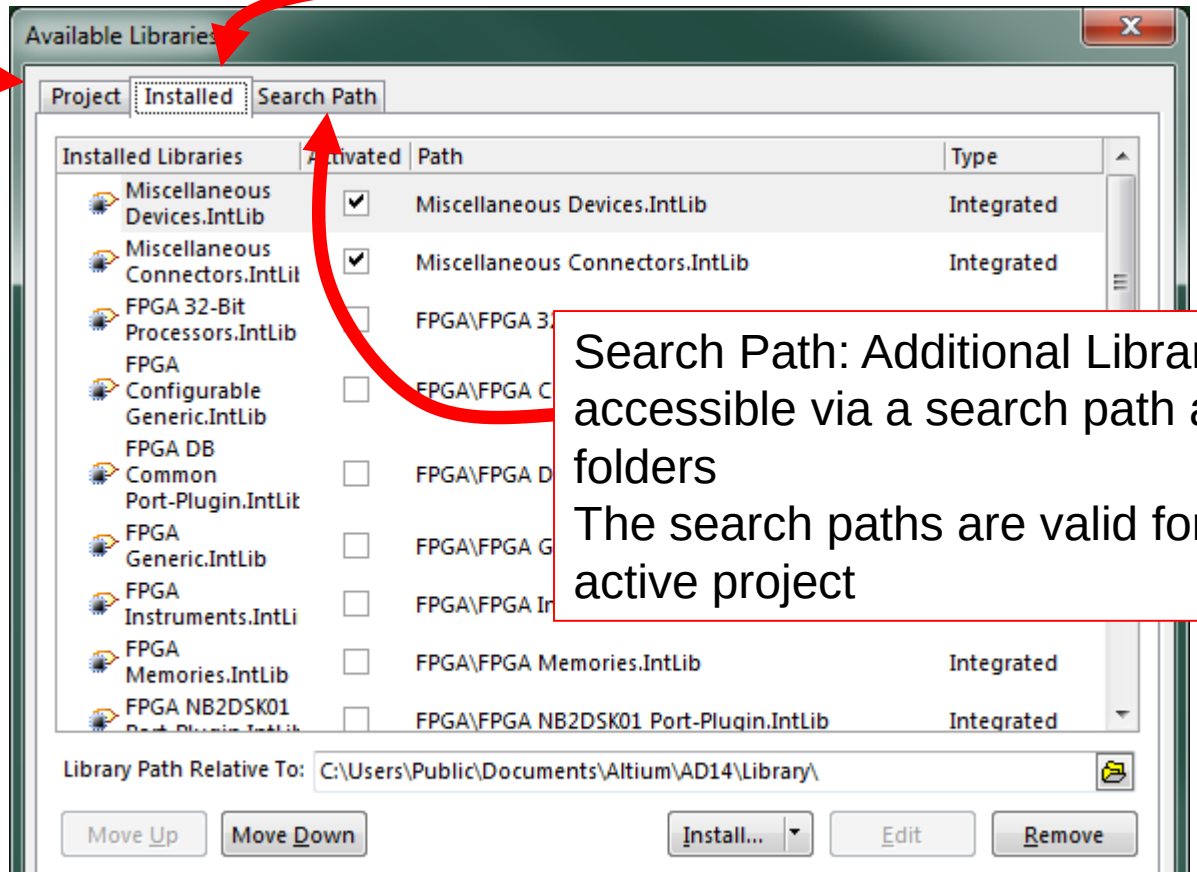
(with an Altium Live account)



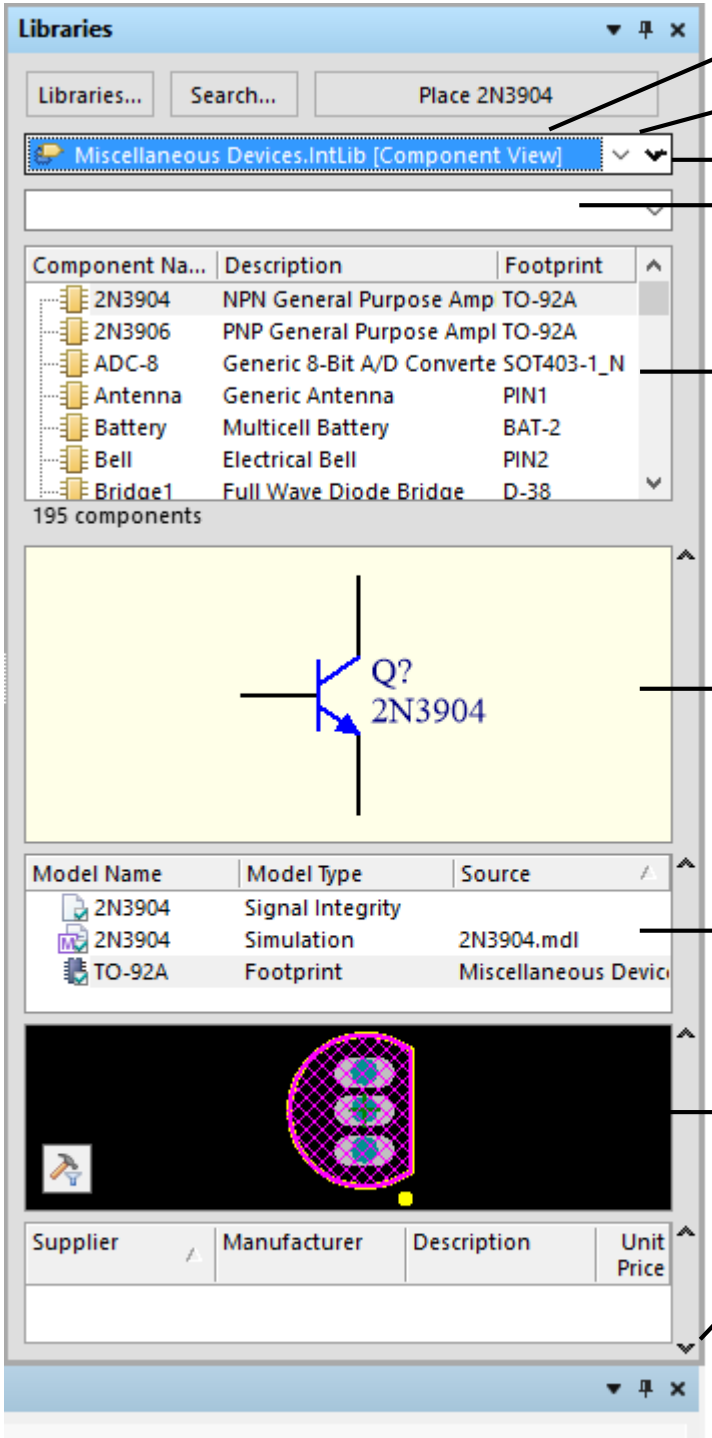
Configuring libraries

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 …

How to select items from the Vault?

Often you will get several choices for the same part, e.g.

Revision ID	Revision State	Name	Description
CMP-1048-01464-1	Released	BC547CG	Amplifier Transistor, NPN Silicon, 3-Pin TO-92, Pb-Free, Bulk Box
CMP-1048-01437-1	Released	BC547C	Amplifier Transistor, NPN Silicon, 3-Pin TO-92, Bulk Box
CMP-1048-01606-1	Released	BC547CZL1G	Amplifier Transistor, NPN Silicon, 3-Pin TO-92, Pb-Free, Ammo Box

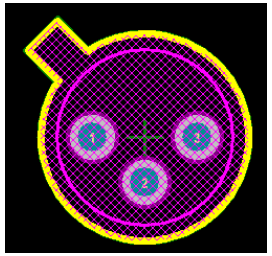
These are typically different component or shipping packaging options. Read the description and check the footprints carefully

Bulk (TO-92) = loose parts

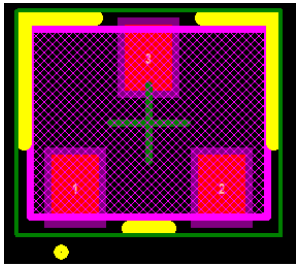
Ammo Box (TO-92)

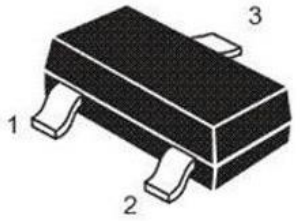
RoHS = Compliance

2N2222 (TO-18)



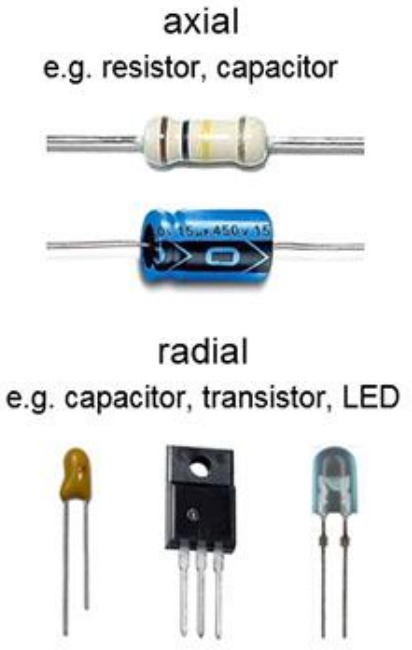
2N2222 (3-Pin SMD)

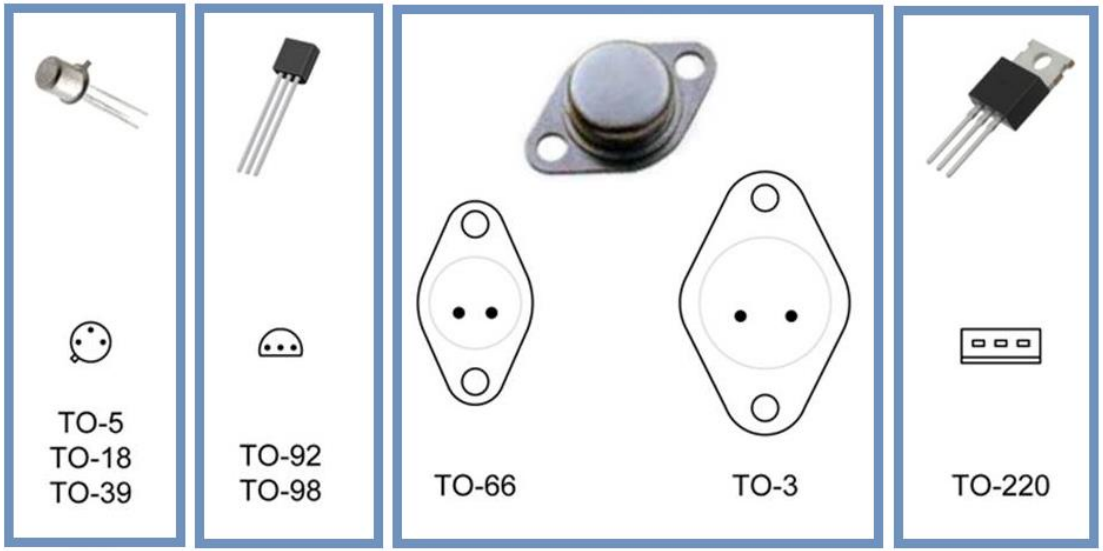




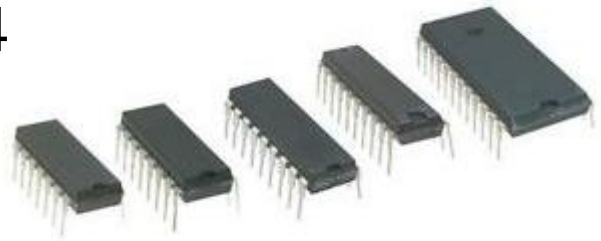
Through-hole Packages & Components

- Use holes drilled through the PCB for mounting the components which are (typ.) soldered on the bottom layer





- Dual in-line package, DIPn → DIP14
  Pin spacing is 0.1”
  TTL, CMOS, Linear (Analog) ICs



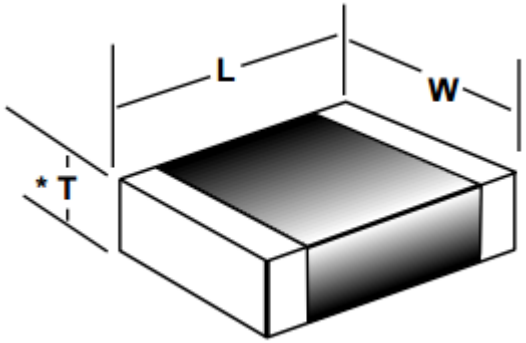
Surface Mount Packages

- Use SMD for high density, small parasitics, or a specific IC
- Passives Flatchip (ceramic Rs, Cs) 4-digit size code

  http://www.topline.tv/SMT_Nomenclature.pdf

Size Code		Approximate Size (LxW)	
Inch	**Metric**	**Inch**	**Metric**
0402	1005*	.04" x .02"	1.0 x 0.5mm
0504	1210*	.05" x .04"	1.2 x 1.0mm
0603	1508	.06" x .03"	1.5 x 0.8mm
0805	2012	.08" x .05"	2.0 x 1.2mm
1005*	2512	.10" x .05"	2.5 x 1.2mm
1206	3216	.12" x .06"	3.2 x 1.6mm
1210*	3225	.12" x .10"	3.2 x 2.5mm
1812	4532	.18" x .12"	4.5 x 3.2mm
2225	5664	.22" x .25"	5.6 x 6.4mm

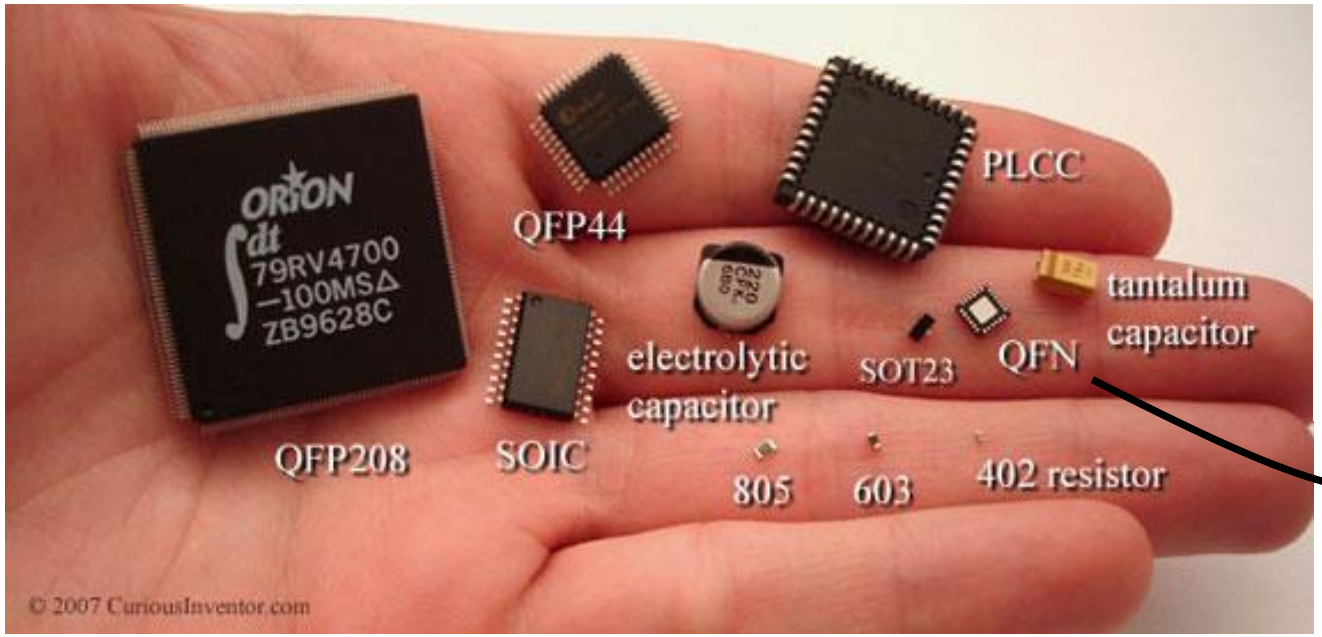
Caution: Overlapping size codes. Metric appears same as inches.



Example:

12	10
Length	**Width**
.12" or 1.2mm	**.10" or 1.0mm**

Surface Mount Packages



Difficult to solder by hand

- There are many types of packages (with leads, lead-less)
- List of integrated circuit packaging types (wikipedia)
  List of integrated circuit package dimensions
- Good visual reference is the NXP posters
  Discretes_package_poster.pdf
  Discrete_Flat_No-leads_DFN_package_poster.pdf

Ref [B1]

Footprints are made out of 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

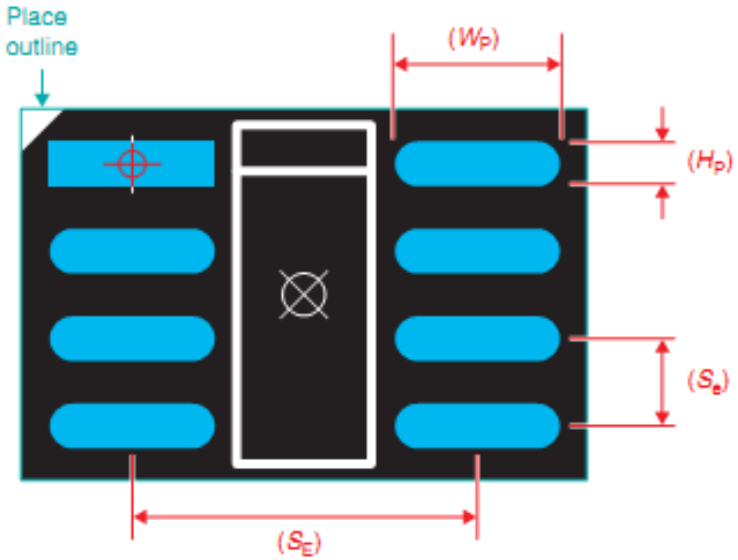


Figure 5-7 Footprint dimensions (typical convention).

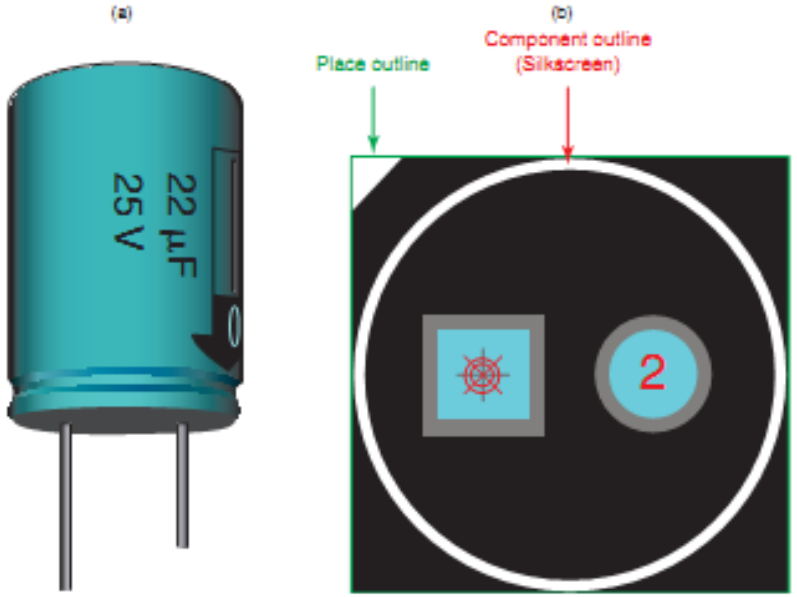
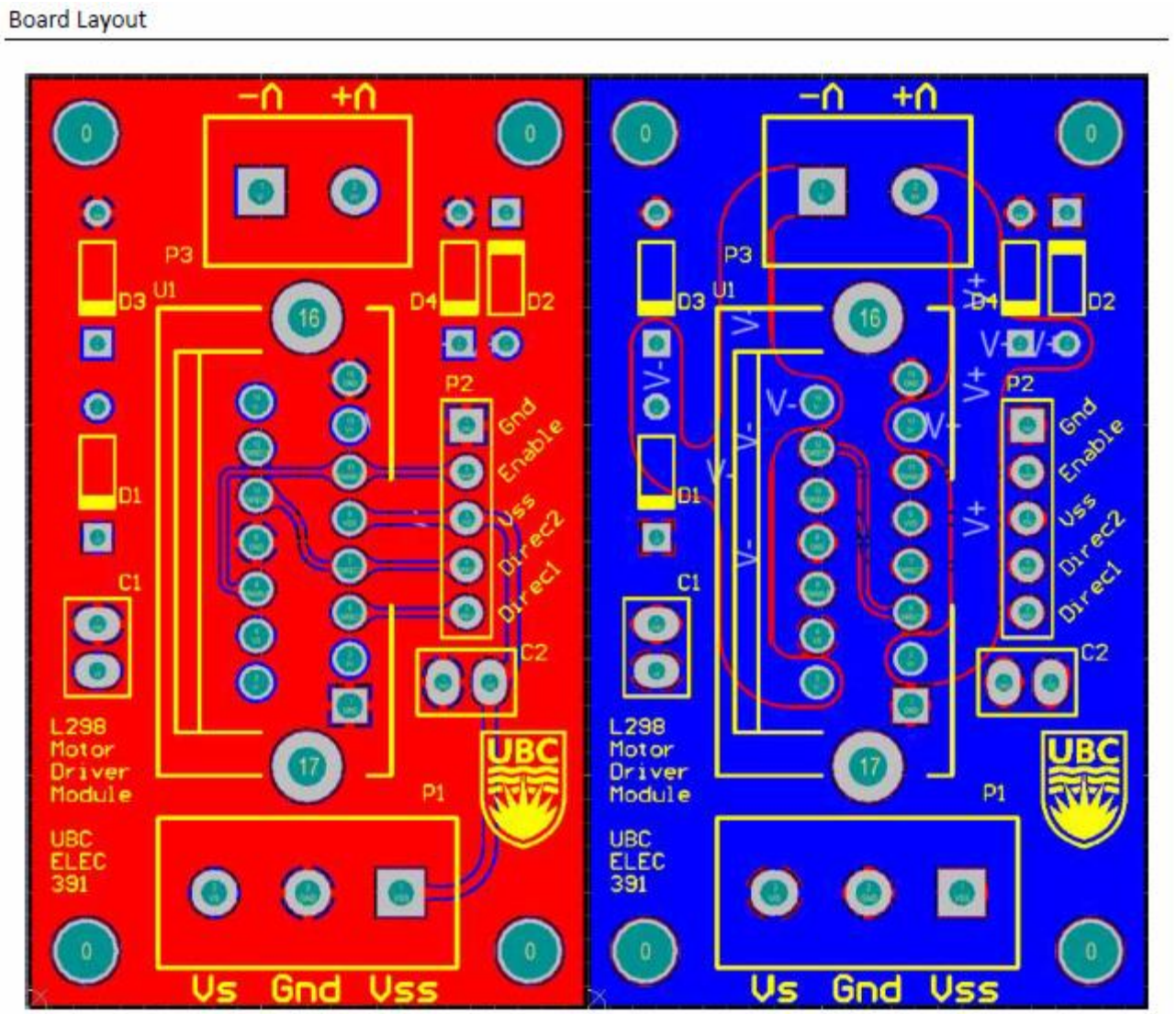


Figure 5-12 Radial-leaded through-hole device. (a) Axial-leaded capacitor. (b) Layout axial footprint

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)

Steps to complete the Layout

Setup

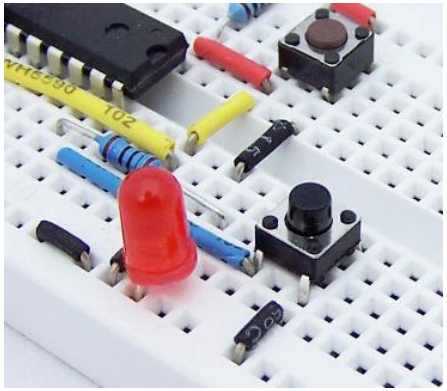
1. Choose work units (metric or imperial)
   you can switch at any time
2. Set the grid
3. Define the board shape
   plan for mounting holes and account for connectors
4. Set the origin
   although you can move it for convenience as you work
5. Transfer the design from the schematic

Layout

6. Place components and mounting holes
7. Route
8. Draw a board outline, add labels (group #)
9. DRC Check

Working units

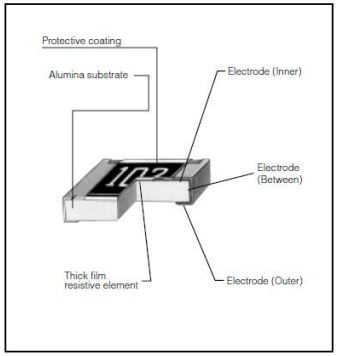
- Imperial (inches)
  - 1/1000th of an inch = 1 mil = 1thou
  - 100mils (0.1”) is a common dimension



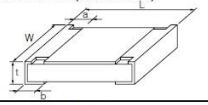
- Metric (mm)
  - 1 mm ≠ 1mil !
  - Common unit in SM parts

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

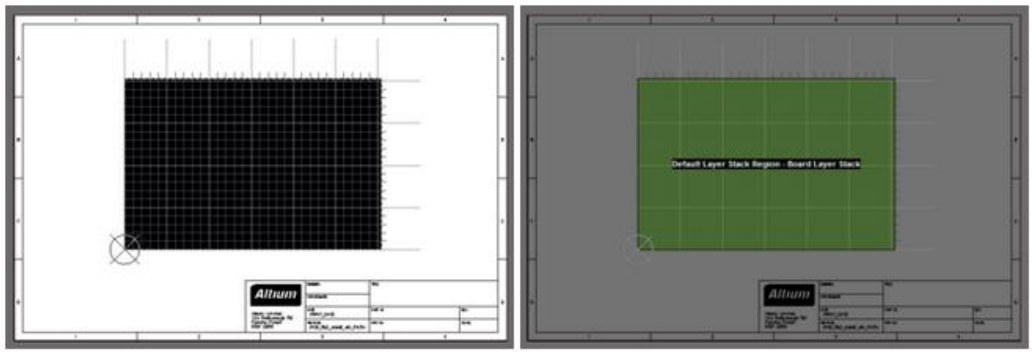
- Remember: 100mils = 2.54mm
- To switch units in Altium Press <Q>

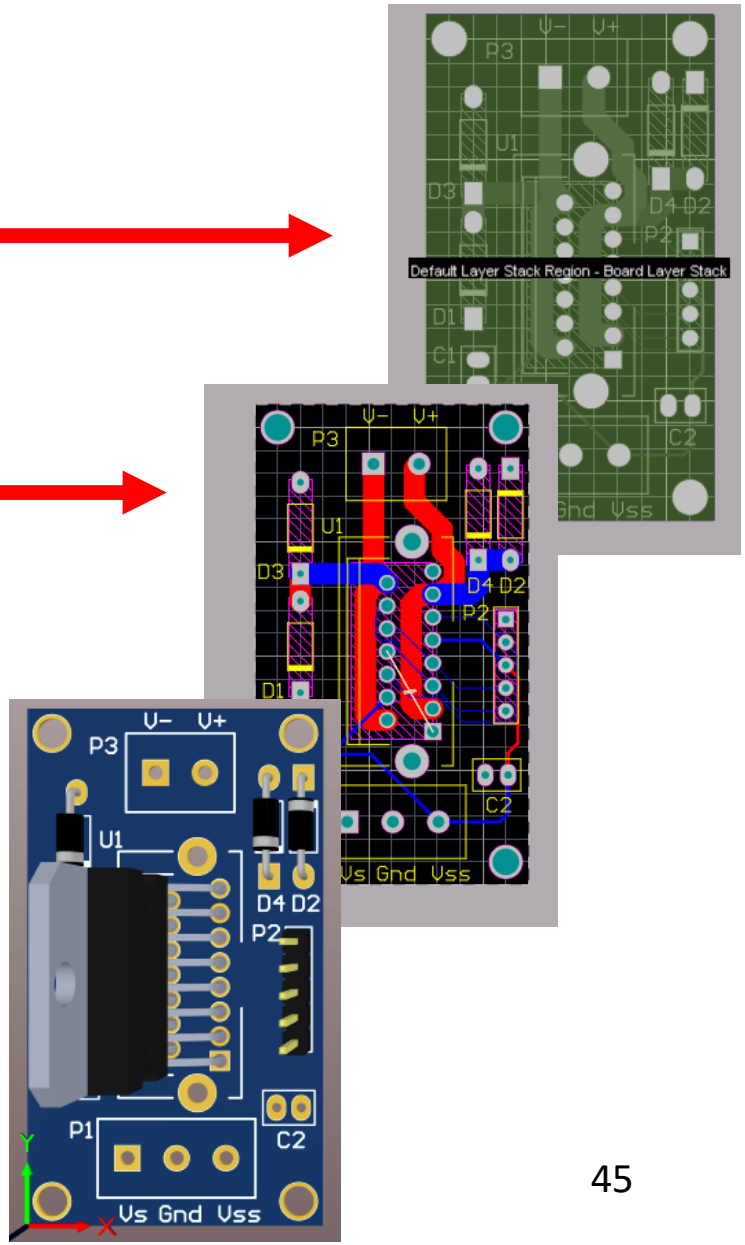
About the grid …

- The PCB Editor is a grid based environment
- Objects are placed on a ‘placement or snap grid’
- Placement is assisted such as the user doesn’t need to aim with absolute accuracy
- Unified Cursor-Snap System >> Properties Panel
  - User defined grids (Cartesian & Polar), that can start at any location and cover different size areas
    Default snap grid is called: Global Board Snap Grid
  - Press <G> to set Global grid step, define grids etc.
- You can work with just the Global Grid, and adjust the step to coarse when placing parts, and fine values when routing
  - More about the PCB Grid system here

PCB Editor view modes

- Viewing modes:
  - Board Planning Mode <**1**>
    - **Design » Edit Board Shape**
    - **Design >> Move Board Shape**
  - 2D Layout Mode <**2**>
  - 3D Layout Mode <**3**>

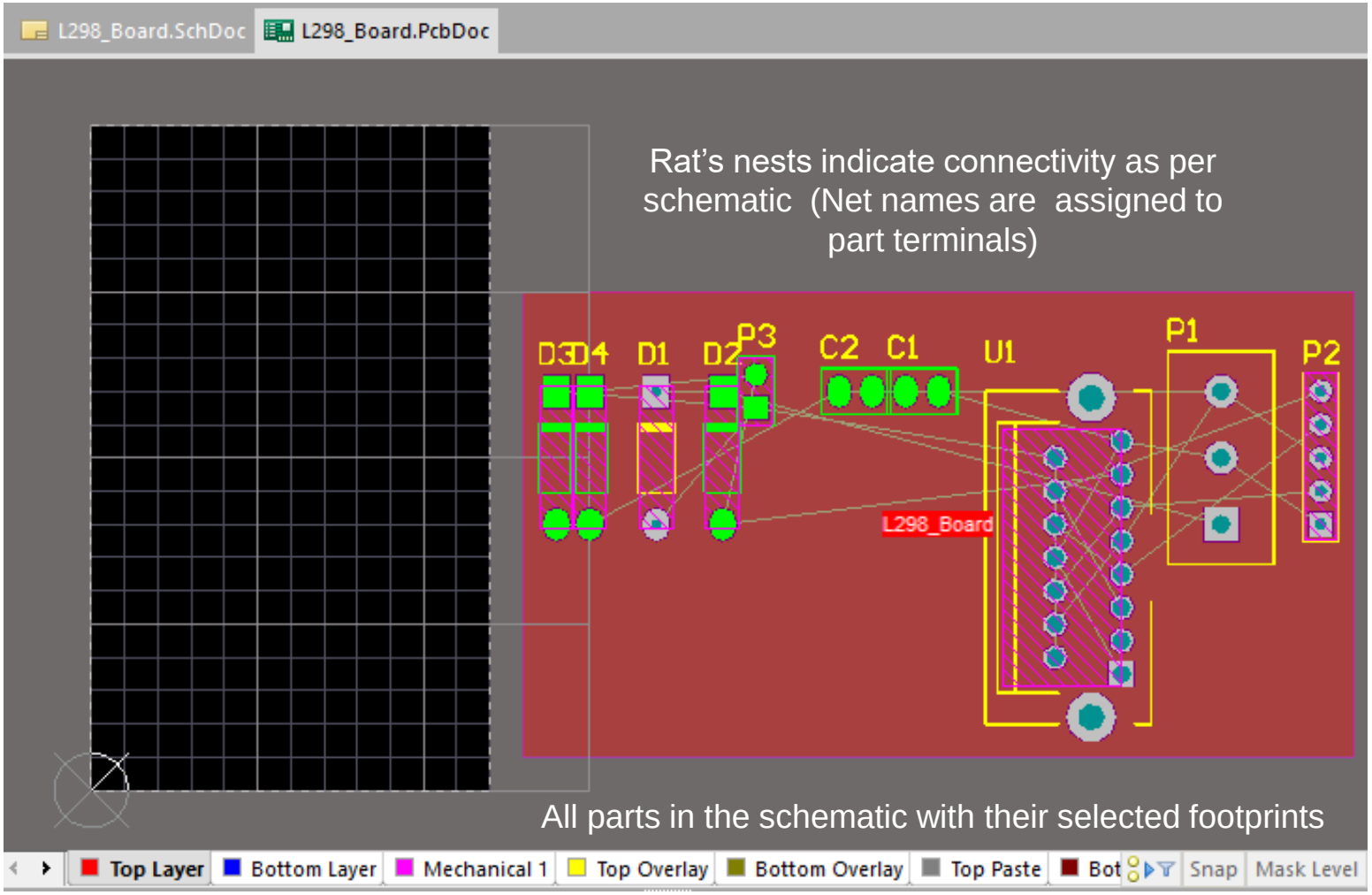




Design transfer

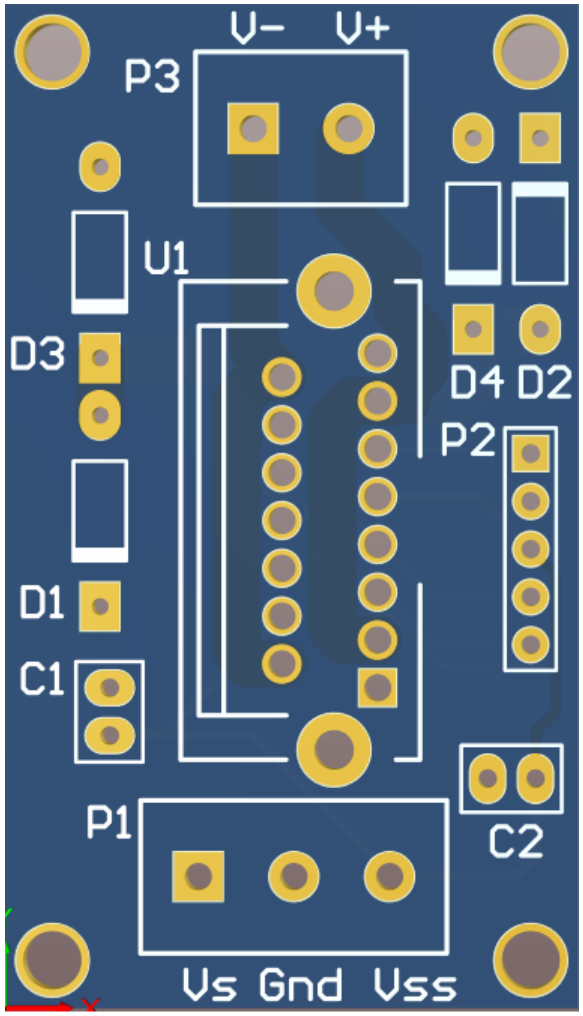
Design >> Update PCB Document …

Setup steps

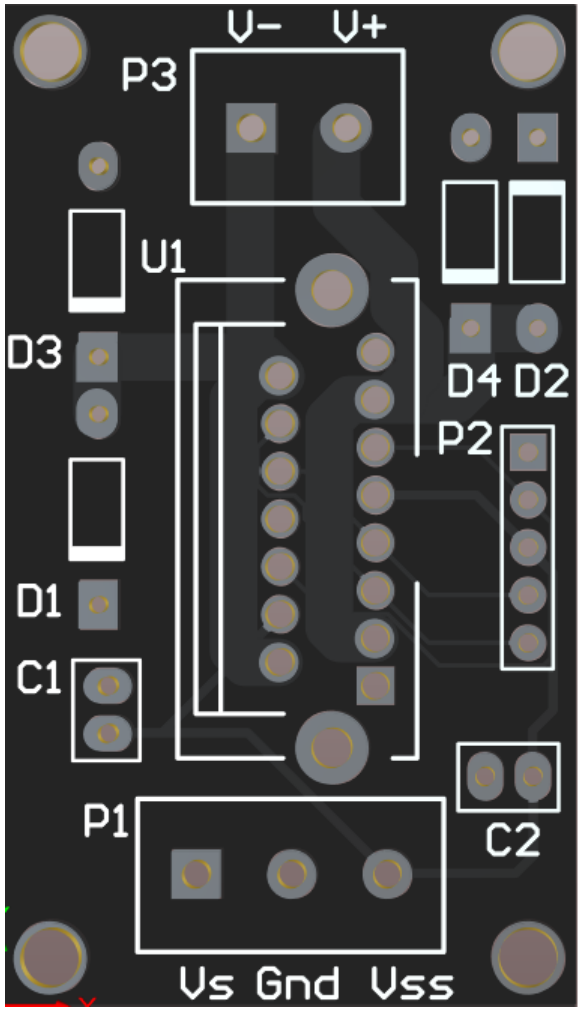


PCBs are multi-layer entities

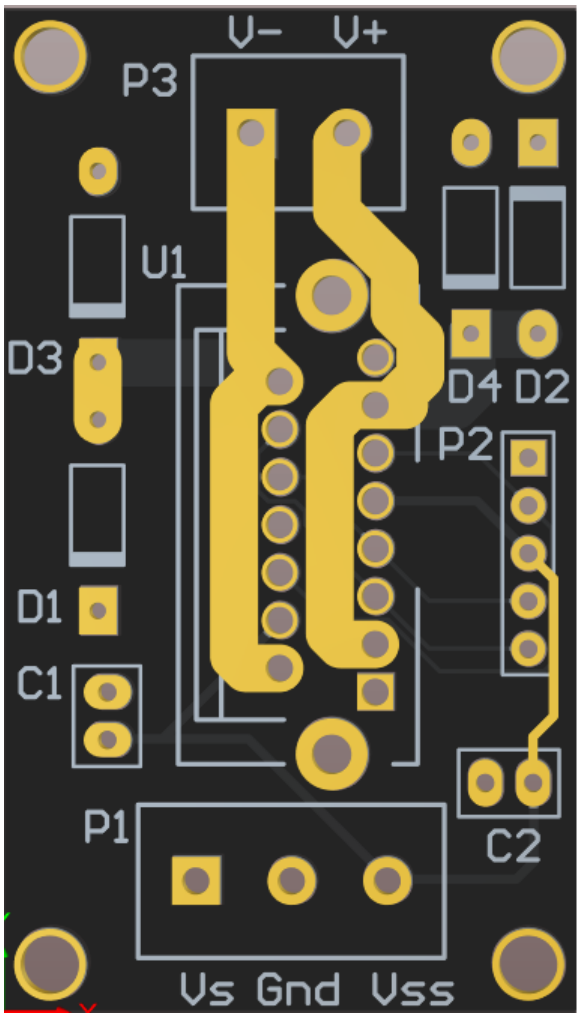
Top view all layers



Top overlay layer



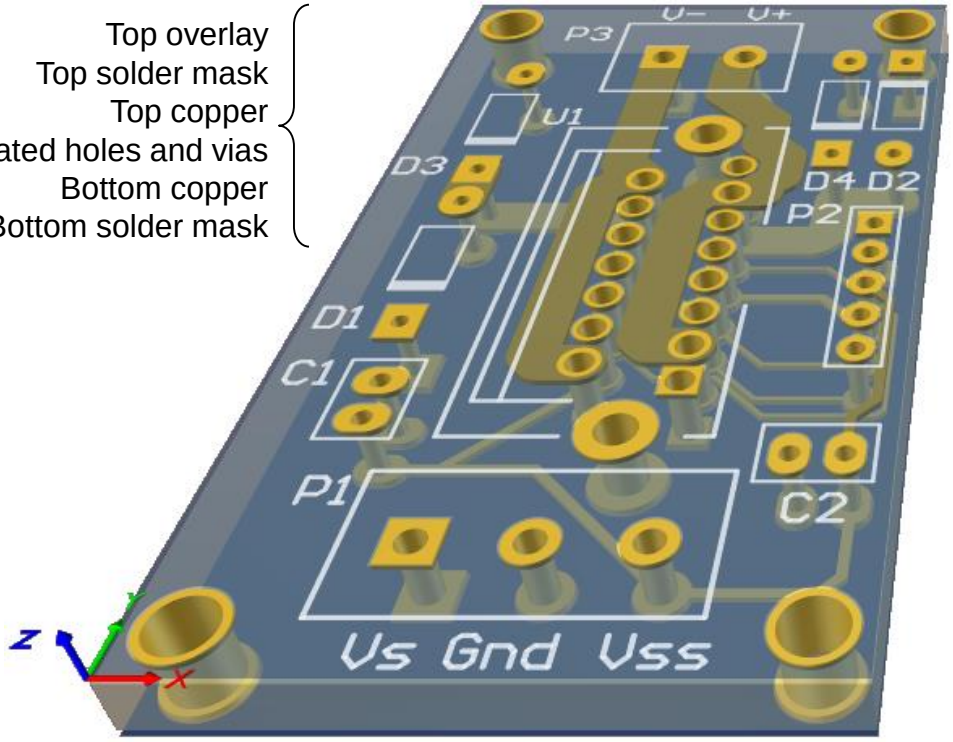
Top copper layer



This is a 2 layer PCB

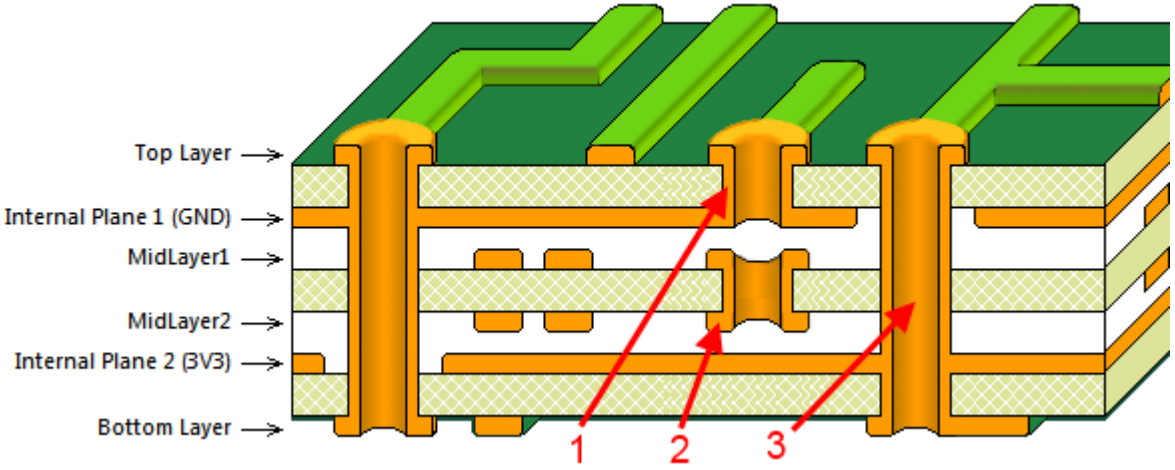
1 dielectric substrate with
top and bottom copper

Top overlay
Top solder mask
Top copper
Plated holes and vias
Bottom copper
Bottom solder mask



You as a designer specify the location of objects and features of every layer

Holes, vias, and footprints are grouped multi-layer objects



More complex PCBs have a stack-up of multiple boards and hence more layers

Ref [B1]

PCB Laminates

- Substrate (laminate)
  - Rigid board of insulating material, provides structural support.
  - Available in different materials and thicknesses, e.g:

    FR4, Rogers/Duroid, Teflon (PTFE) 8mils (0.2mm) → 240mils(6.1mm)
  - Most commonly used material type is FR4, 62mils thick
  - Available with different copper thicknesses

    Cu thickness measured in weight oz/ft^2

    ½ oz → 0.7mils

    1 oz → 1.4mils

    2 oz → 2.8mils

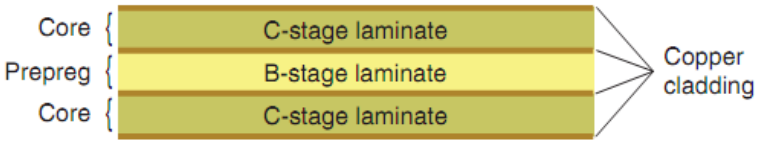


Figure 1-3 Cores and prepreg.

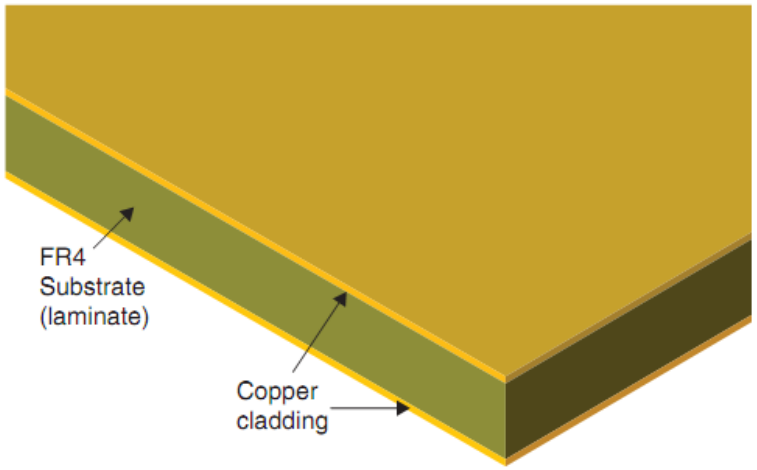


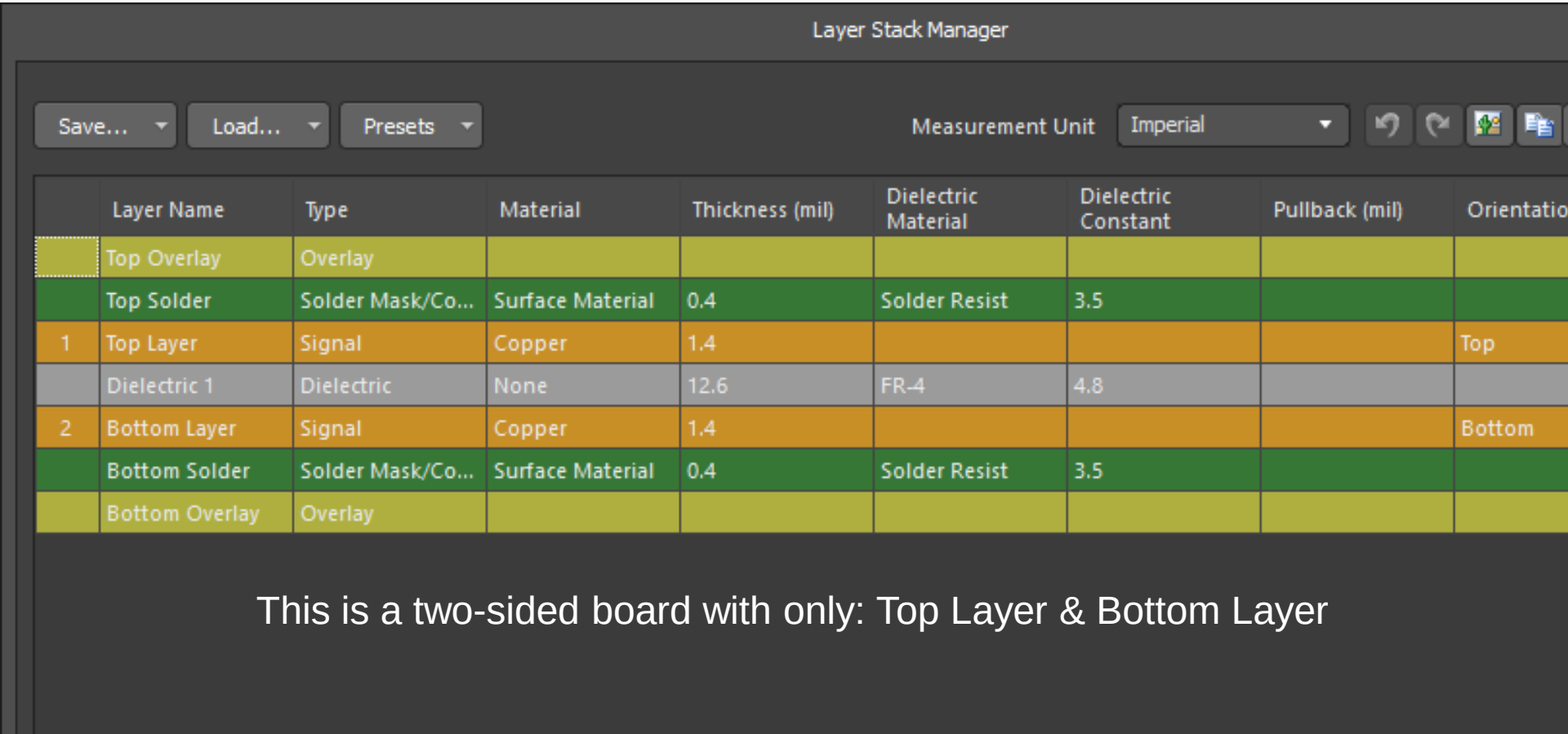
Figure 1-2 A double-sided copper clad FR4 substrate.

1mil = 25µm

PCB Anatomy: Layer Stackup

Design >> Layer Stack Manager …
This will enable additional layers as required

Information on substrate thickness, type and dielectric is not included in Gerber CAM files, these need to be explained separately to the manufacturer.



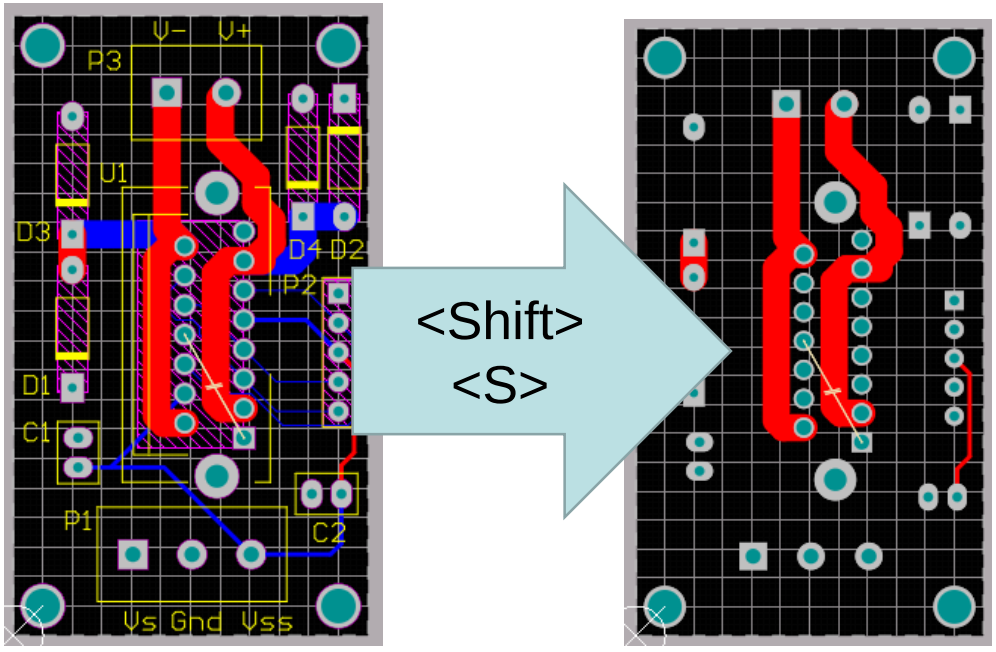
	Layer Name	Type	Material	Thickness (mil)	Dielectric Material	Dielectric Constant	Pullback (mil)	Orientatio
	Top Overlay	Overlay						
	Top Solder	Solder Mask/Co...	Surface Material	0.4	Solder Resist	3.5		
1	Top Layer	Signal	Copper	1.4				Top
	Dielectric 1	Dielectric	None	12.6	FR-4	4.8		
2	Bottom Layer	Signal	Copper	1.4				Bottom
	Bottom Solder	Solder Mask/Co...	Surface Material	0.4	Solder Resist	3.5		
	Bottom Overlay	Overlay						

This is a two-sided board with only: Top Layer & Bottom Layer

Working with Layers

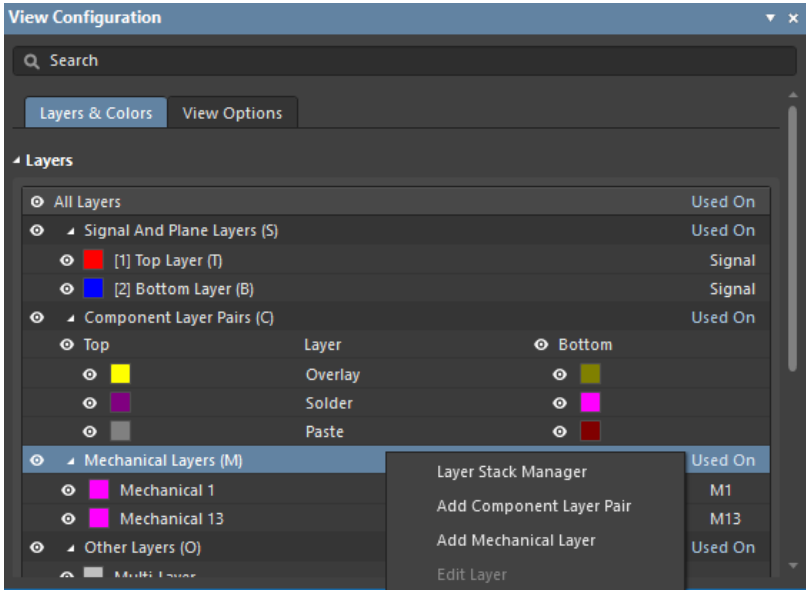
- Use the tabs at the bottom of the editor to switch between layers

To switch to single layer mode



To set relevant layers

View » Panels » View Configuration
or press <L>



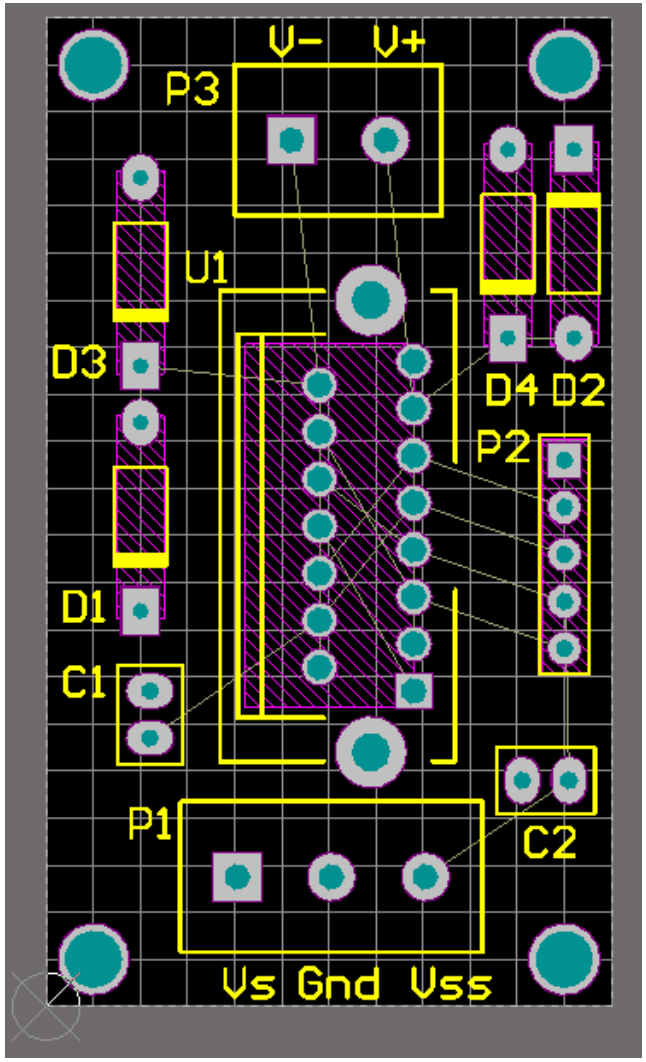
Configuring the Display Layers

- 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.

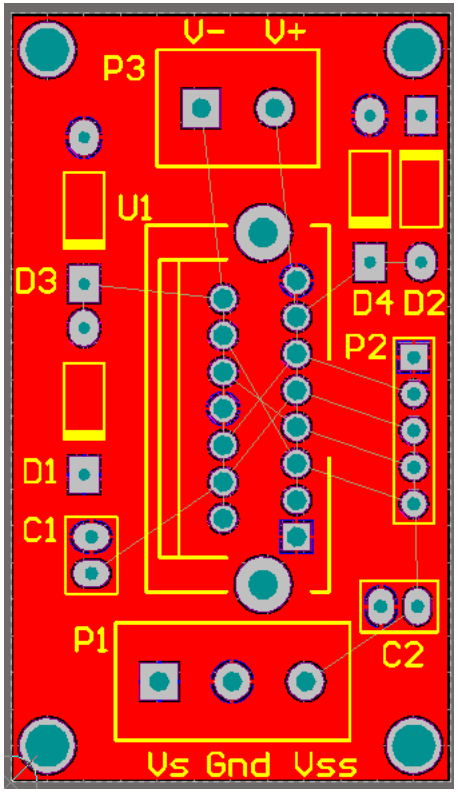
Mechanical Layers

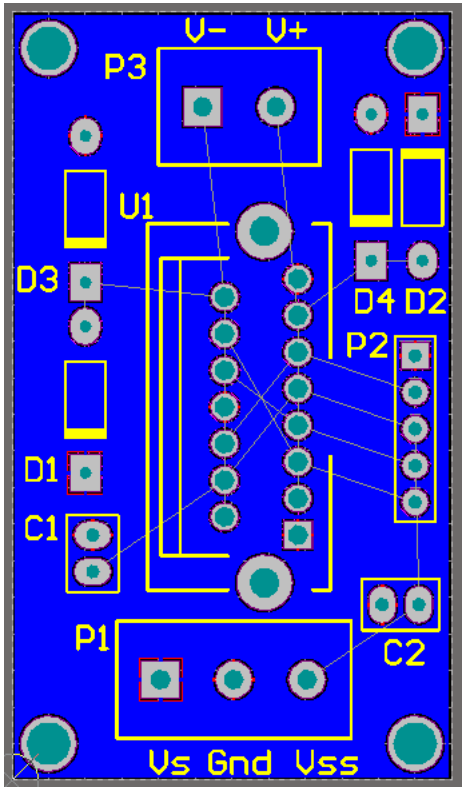
- Multi-purpose layers
- E.g. Altium supports 32 Mechanical layers: M1 … M32
- Typically
  - M1 Board outline
  - M2 PCB manufacturing info
  - M11-M12 Top and bottom layer dimensions
  - M13 Top layer 3D models and mechanical outlines
  - M14 Bottom layer 3D models and mechanical outlines
  - M15 Top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

Positioning components & routing



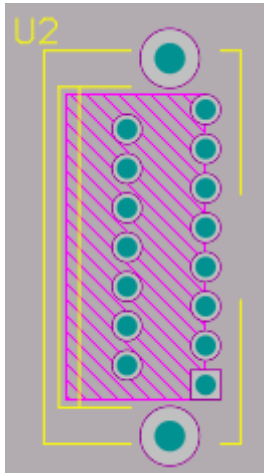
Place a plane on top for GND
Place a plane bottom for Vs



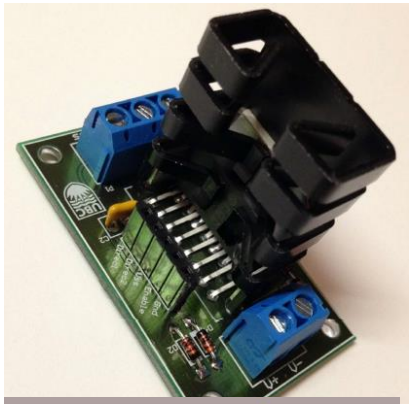


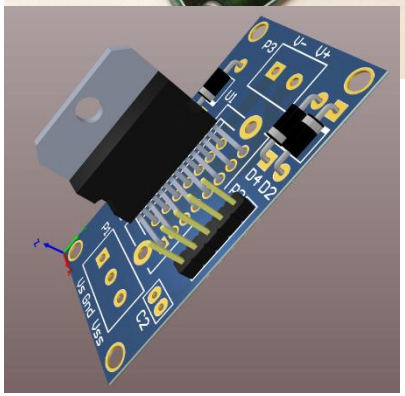
Positioning components & routing

- When placing components:
  - Try to align parts (same orientation)
  - Ensure that labels are all set in the same direction
  - Beware of bulky connectors (use 3D view with caution)



Yellow (top overlay) lines
in footprints typically mark
the physical boundaries of a part
However you must be cautious and
think about it.
This footprint includes the space taken
by a heatsink, but does not show it in
the 3D view





Ref [B1]

About 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 for e.g. 7mils/7mils is typical fab. spec.

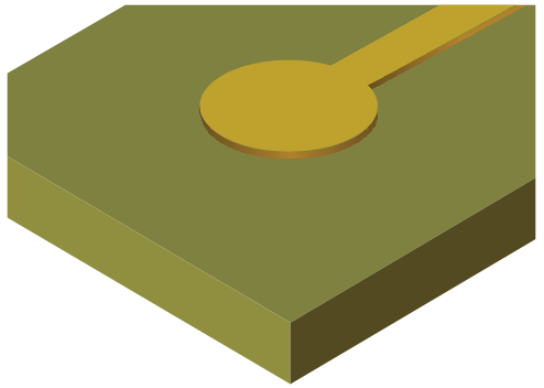


Figure 1-11 Copper pad and trace after etching and resist stripping.

Negative view:
Copper planes, Drill holes, Solder Masks

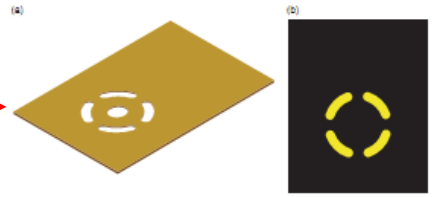


Figure 1-18 Copper in a plane layer (negative view without drill info). (a) Copper plane with thermal relief. (b) Negative view in Layout.

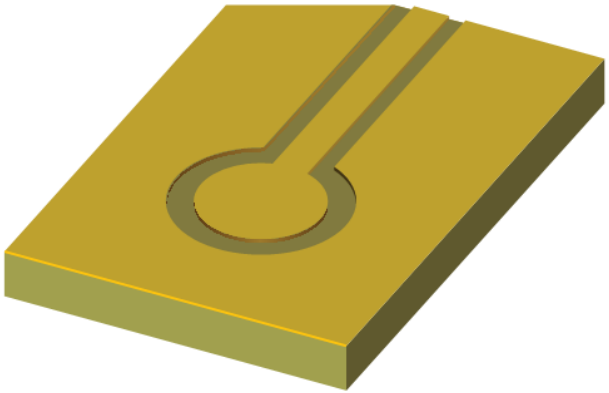
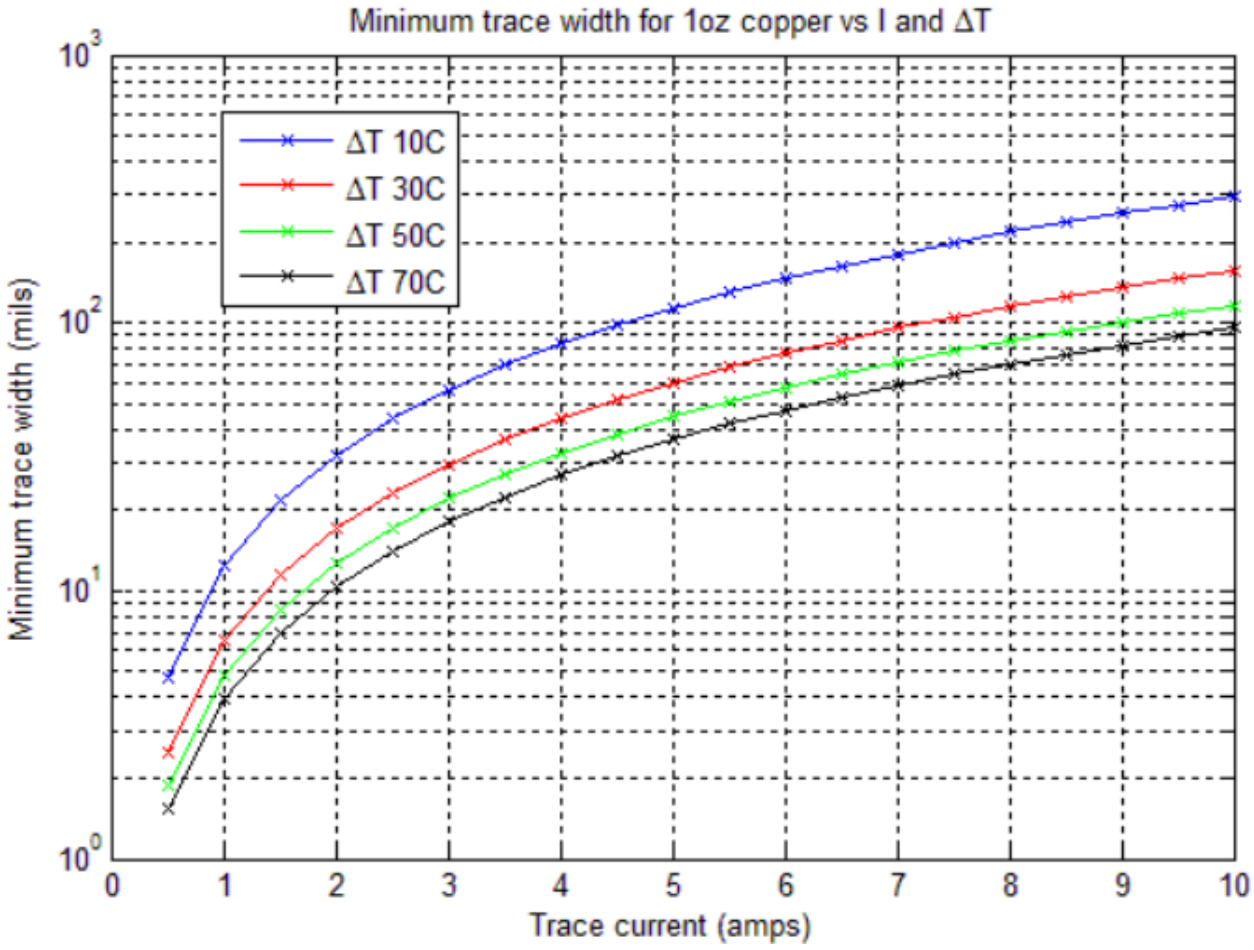


Figure 1-12 A mechanically milled trace.

Determining trace width



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

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).

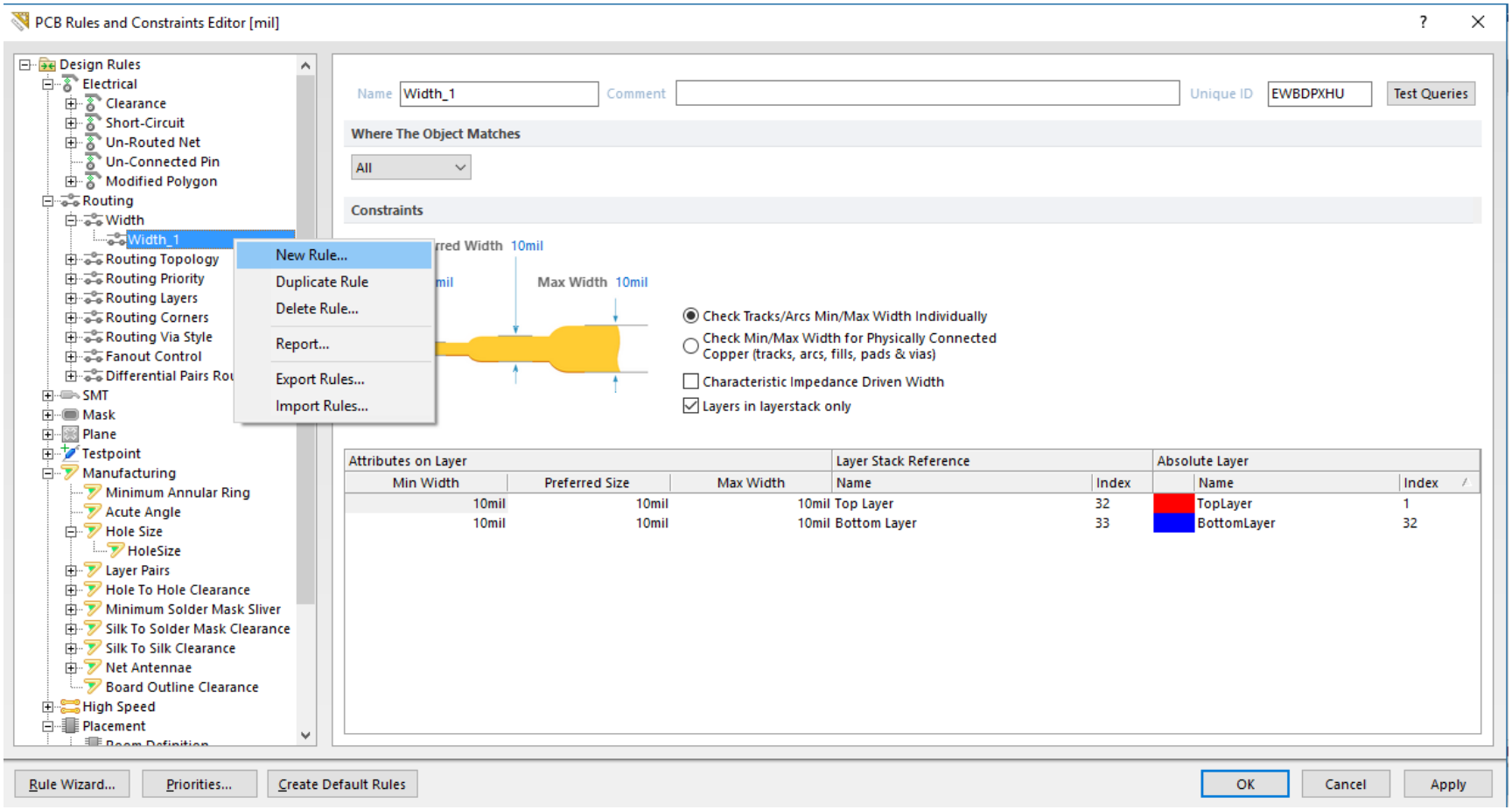
- **Shift**+**Spacebar** to cycle through the various track corner modes. The styles are: any angle, 45°, 45° with arc, 90° and 90° with arc. There is an option to limit this to 45° and 90° in the **PCB Editor - Interactive Routing** page of the *Preferences* dialog.

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



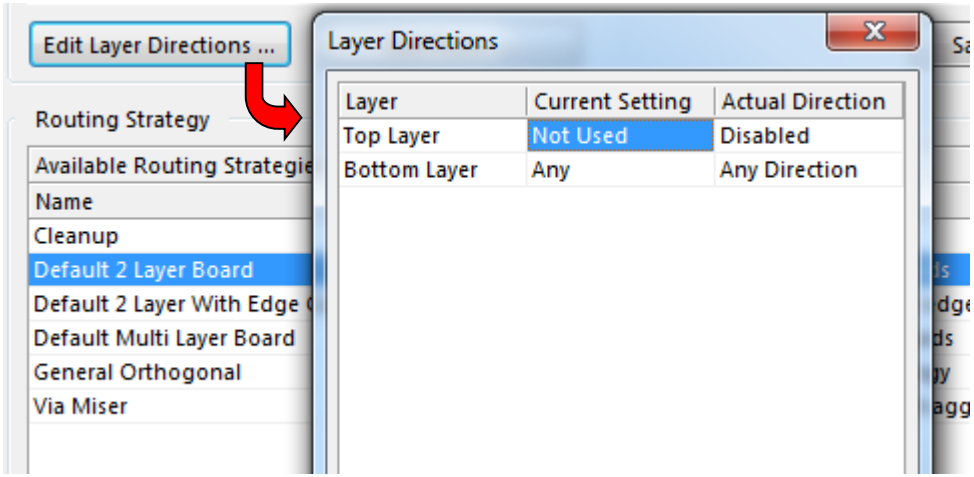
Rename to “Width_IO’

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

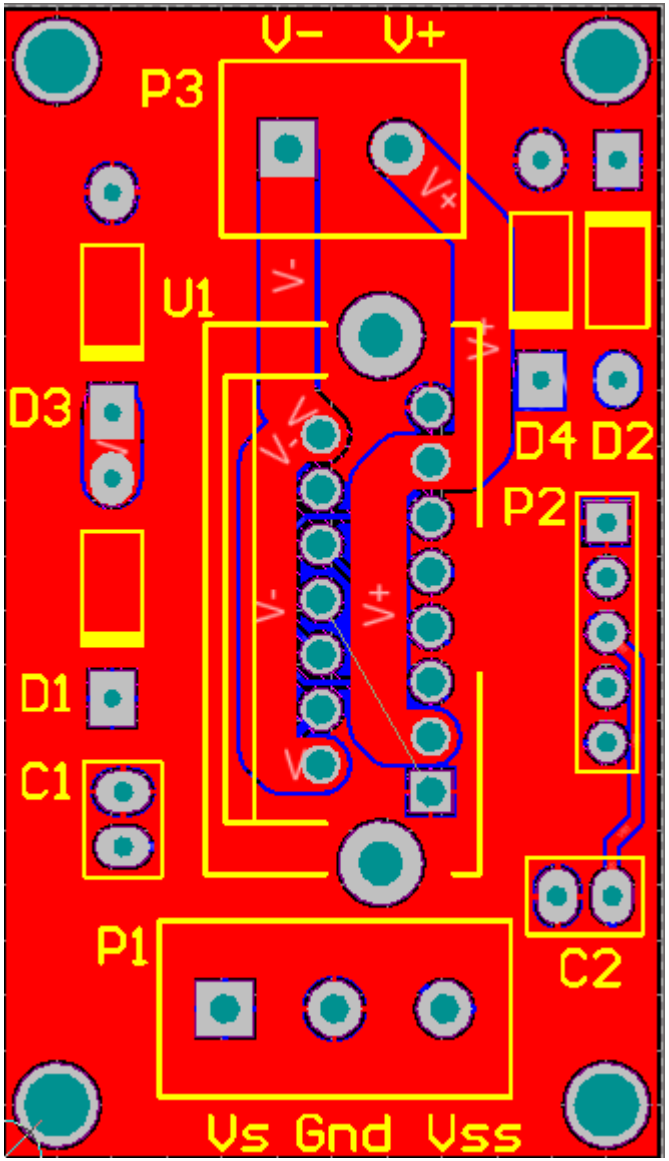
Set rule execution priority

Auto route

- Rout » Un-Route » All
- Auto Route » All



- You can also set single layer routing



Ref [B1]

Holes

Holes can be:

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

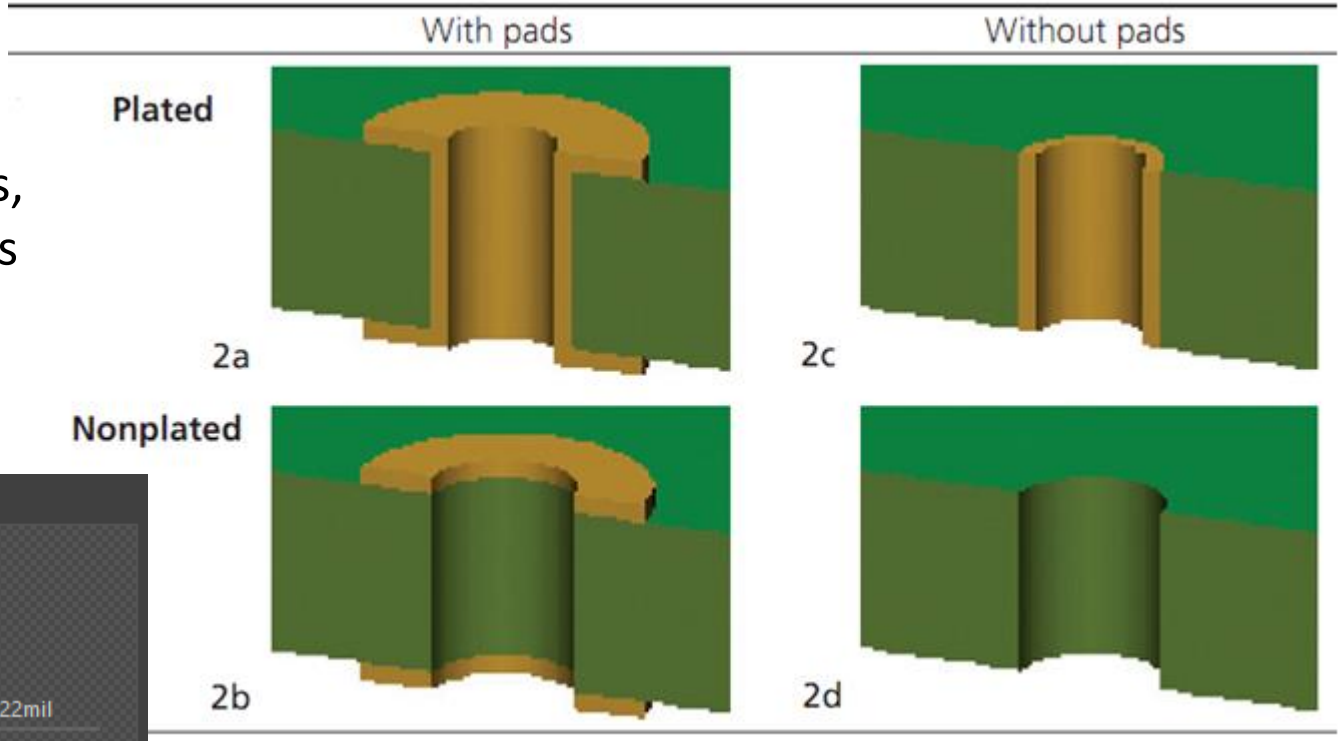
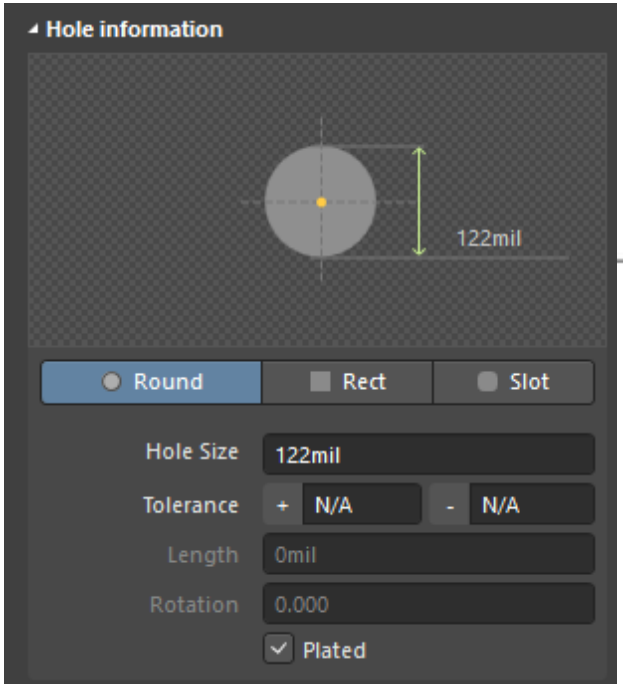


Table 8-2 Basic hole types



Altium pad properties dialog

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

Plating reduces hole size by 0.003”

Hole sizes

- Comon hole 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

Ref [B1]

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

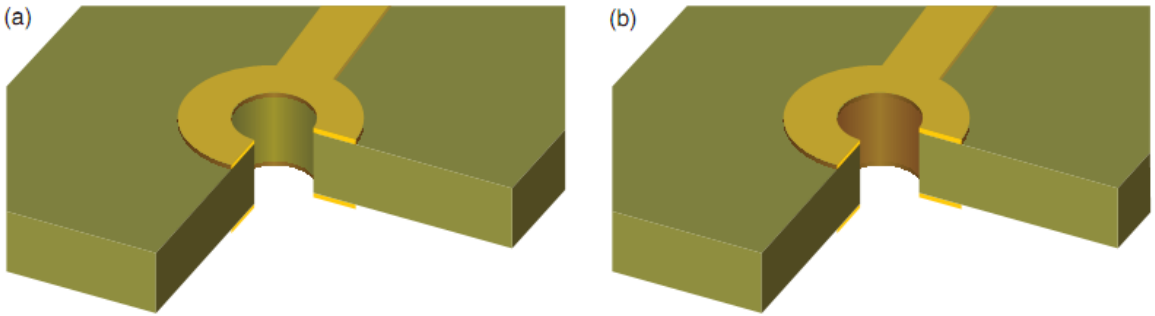
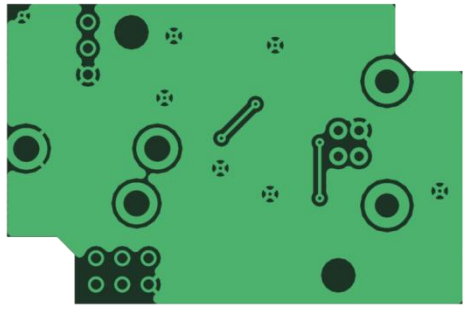


Figure 1-13 Holes are drilled into the board and then copper plated. (a) A nonplated through-hole. (b) A plated through-hole.



Source: wikipedia.org: Thermal pad

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

PWR and GND planes are commonly inner layers

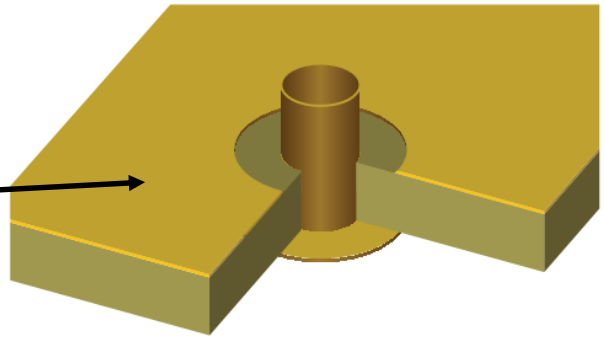


Figure 1-15 A clearance area provides isolation between a plated hole and a plane.

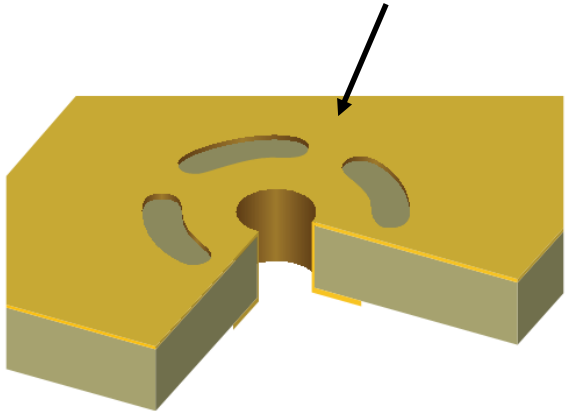
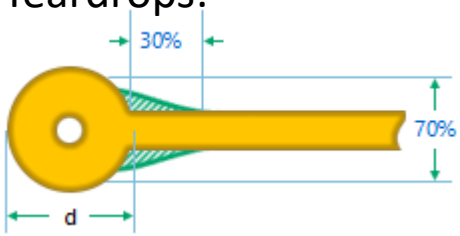


Figure 1-14 A connection to a plane layer through a thermal relief.

Teardrops:



Ref [B1]

Types of Vias

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

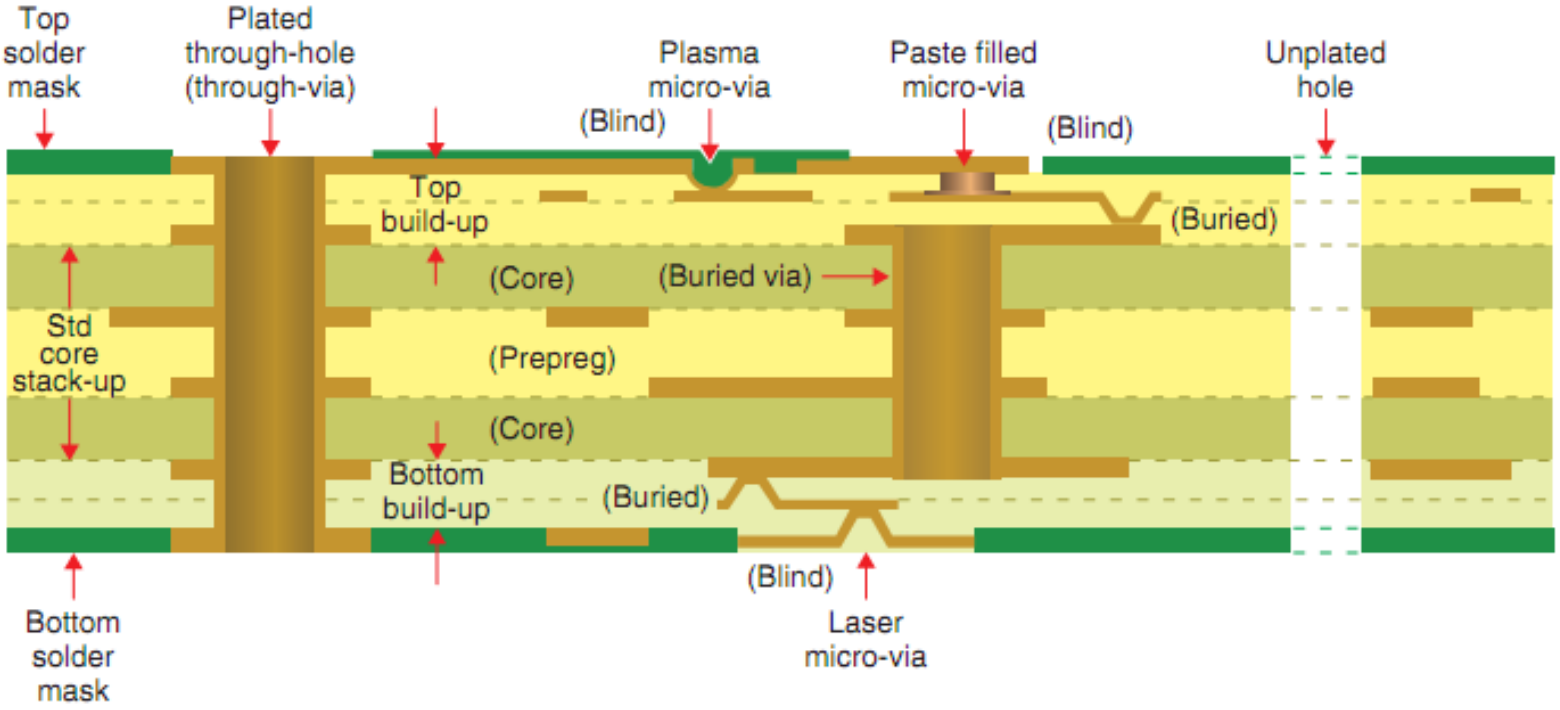


Figure 1-5 A built-up, multitechnology, PCB stack-up.

Ref [B1]

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

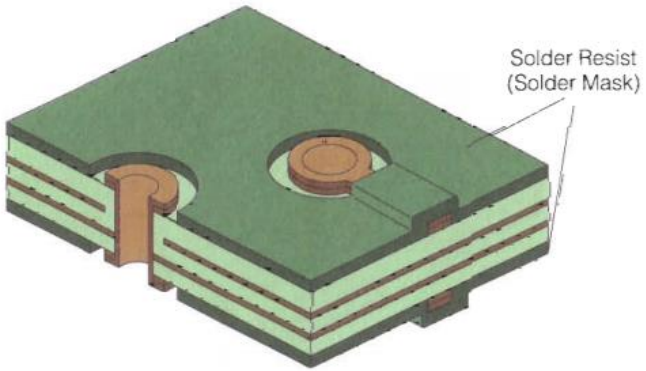


Illustration ML-14. *Apply solder resist.* The specified resist (either dry film, liquid photoimageable, or screen printed) is applied to the surfaces of the PCB or panel.

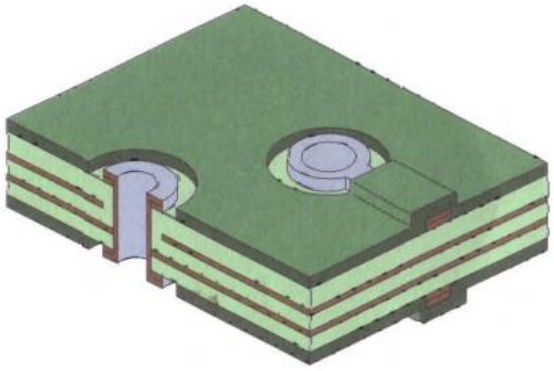
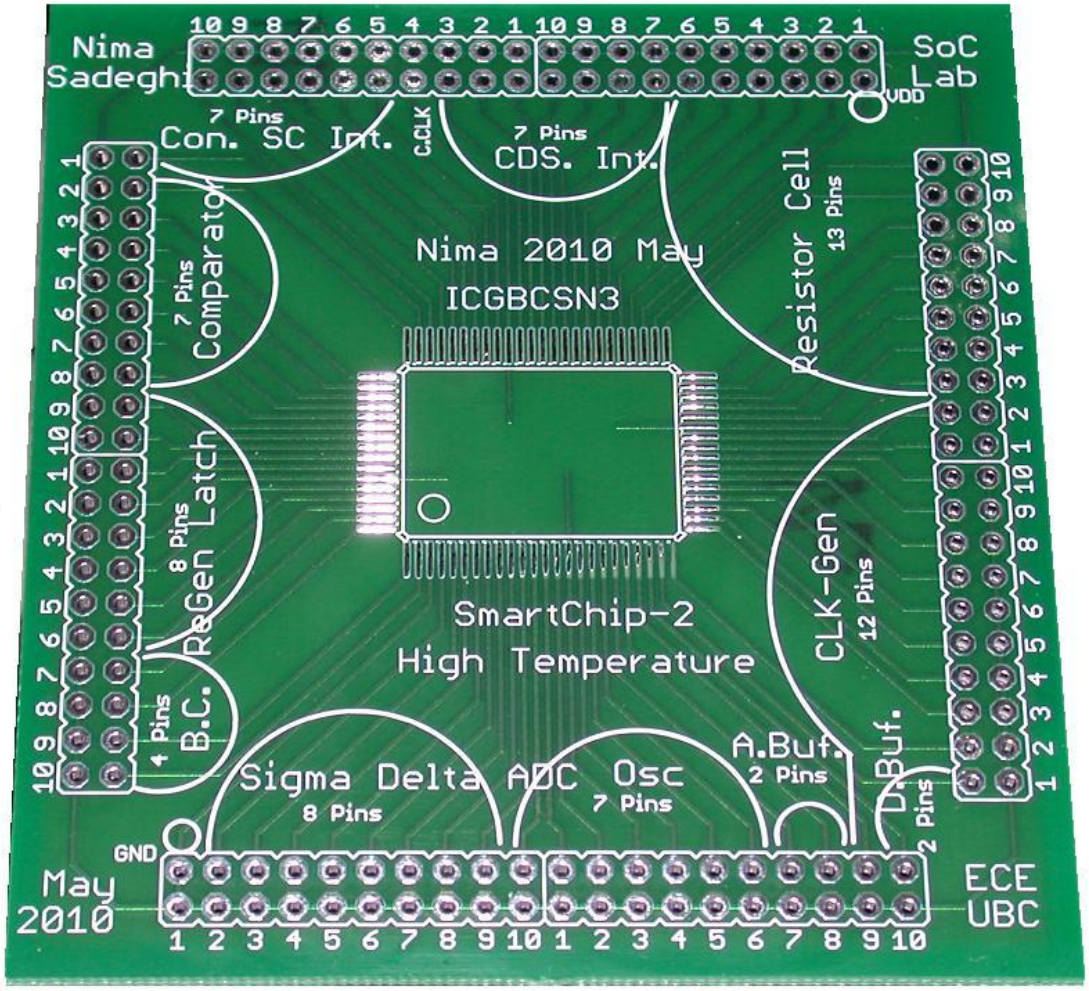


Illustration ML-15. *Solder coat.* Solder (tin/lead) is applied to the exposed copper areas, and the excess solder is removed.

Source: Printed Circuit Board Basics: An Introduction to the PCB Industry, by: Michael Flatt

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)

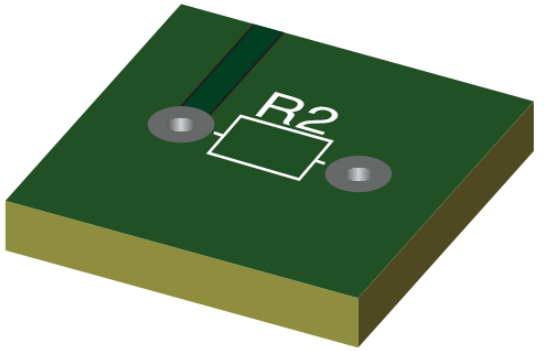


Figure 1-16 Final layers are the soldermask (green) and silk screen (white).

Instructions for ELEC391 fabrication submissions

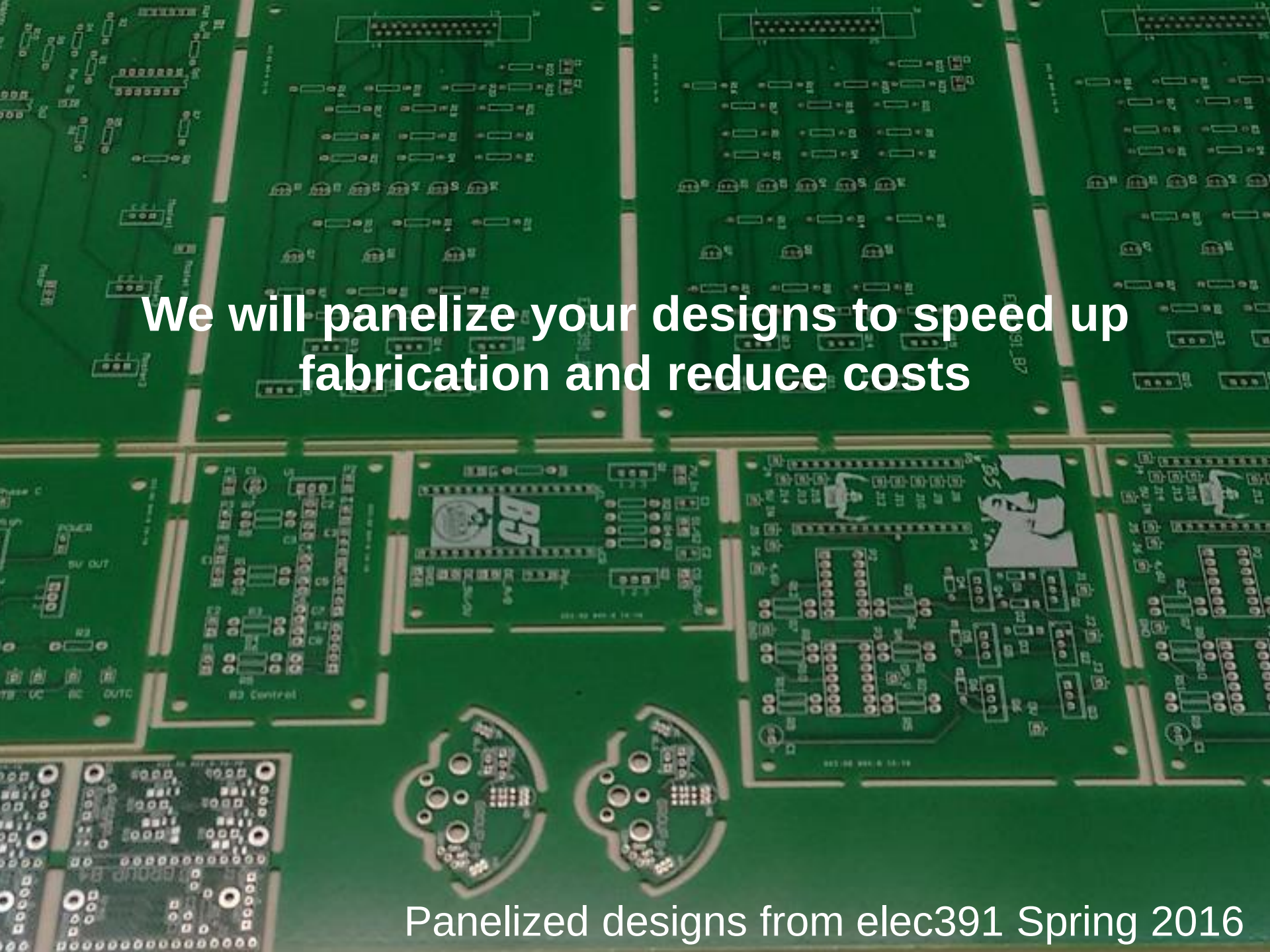
Submission Instructions

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

  Midnight, Mondays
  Feb 3, Feb 10, Feb 17, Feb 24,
  Mar 2, Mar 9, Mar 16
  we will check submissions and accept fixes
  until 5PM the following Tuesday

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

- 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
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 & cost
- 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 1/2

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

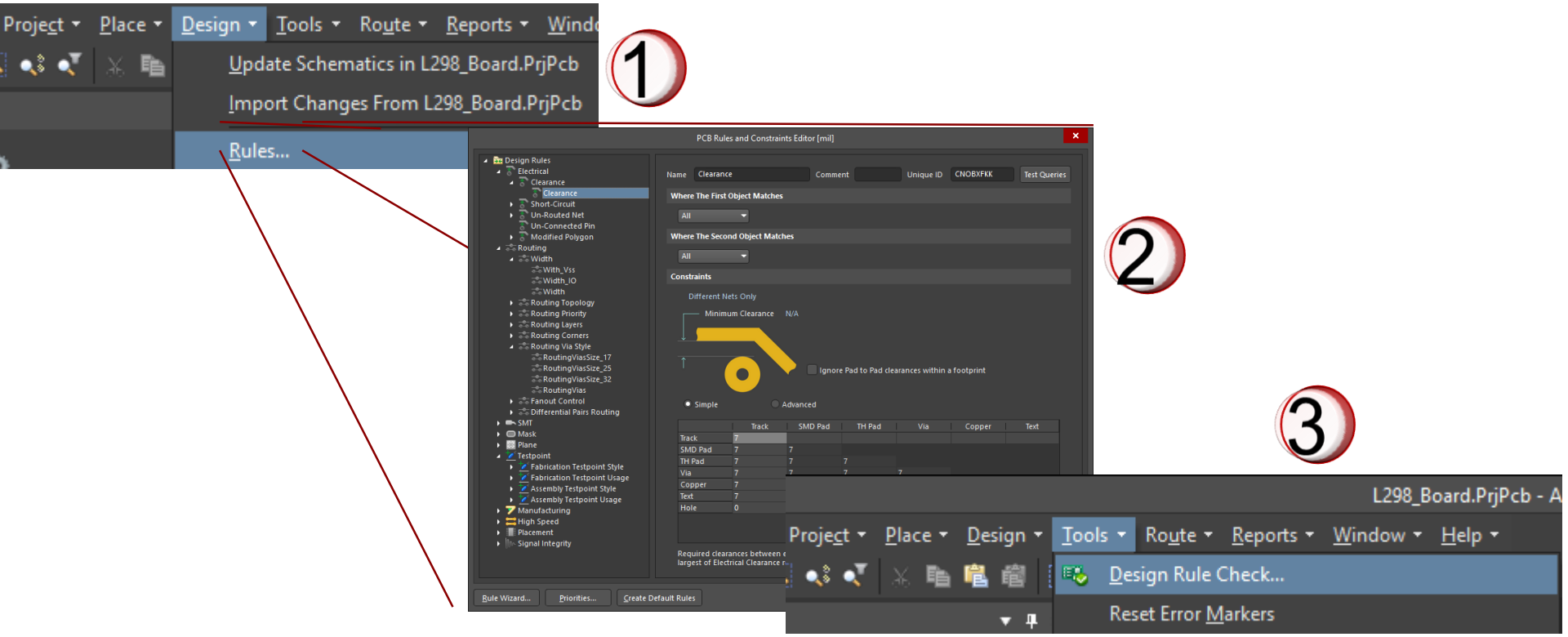
Draw a board outline on layer "Mechanical 1"

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 2/2

4) Install provided Design-Rules file (available here)
please do not modify base rules, but you can add custom routing rules.
Submissions that do not pass DRC will be rejected



How to load design rule file

- DRC file and these same instructions from here
- 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 or <Ctrl> +<A>) → OK
  - Browse to select .RUL file
  - Clear existing rules prior to import? → NO

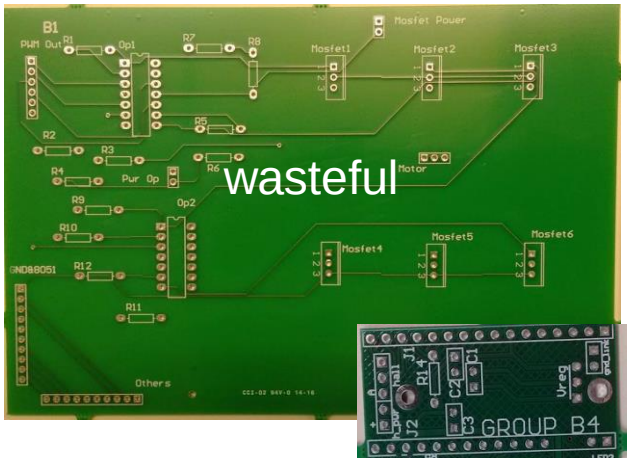
Rules – design rules

- 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
- No limit to allowed hole sizes


PCB Design Best Practices

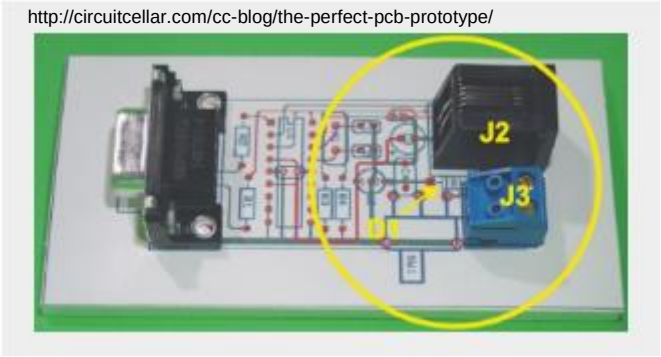
Best Practices: Estimating board size

- 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.



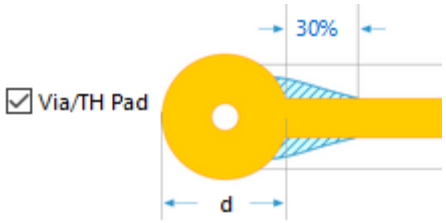
Best Practices: Floor planning

- 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

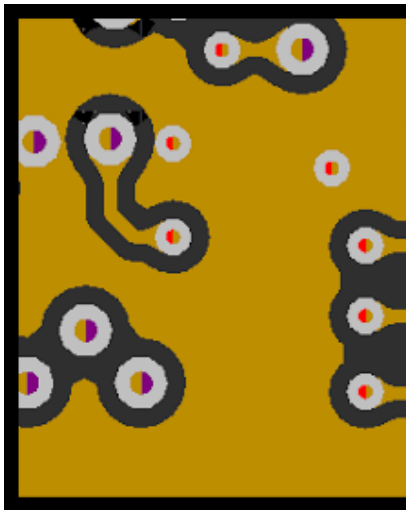
- On two sided boards keep traces perpendicular as much as possible
- Avoid 90 degree bends in tracks (reduced chances of acid traps)
- Keep traces a short as possible
- Always connect a trace to the center of the pad
- Use teardrops (Tools >> tear drops), and use vias to avoid lockout
- Do not place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



- Layout first all critical traces

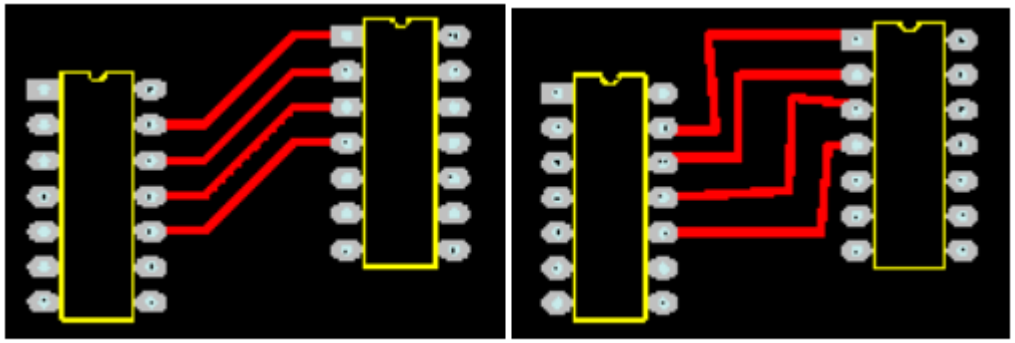
  e.g. CLK, diff pairs, controlled length

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



- Rout nicely

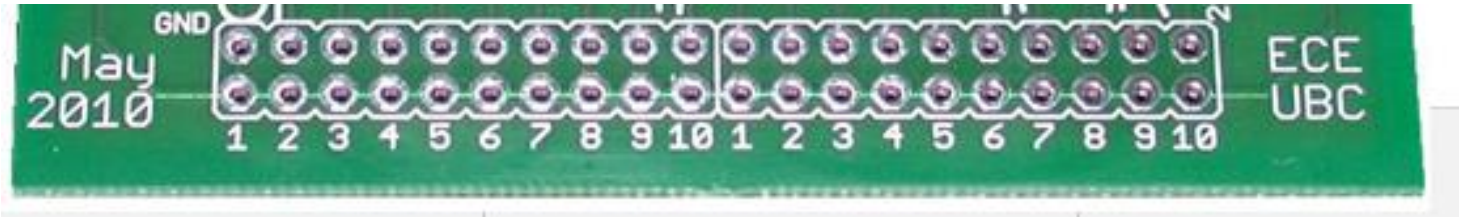
[Ref 3]



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
- Confirm Board Information
  - *Properties panel*: *Board information* section (when no objects are selected)
  - Click on *Reports* and select:
    - Board Specifications
    - 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