



Capstone Design Projects

PCB Design

Roberto Rosales

Jan 12th, 2016

Outline

- PCB Design at ECE UBC: Tools available, Design Flow
- How to install Altium 2014
- How to use Altium Designer
- Working units & SMD Footprints
- CAM files
- NC Drill Files
- Drill bit sizes
- Design hand off
- Recommended manufacturers

PCB Design at ECE UBC:

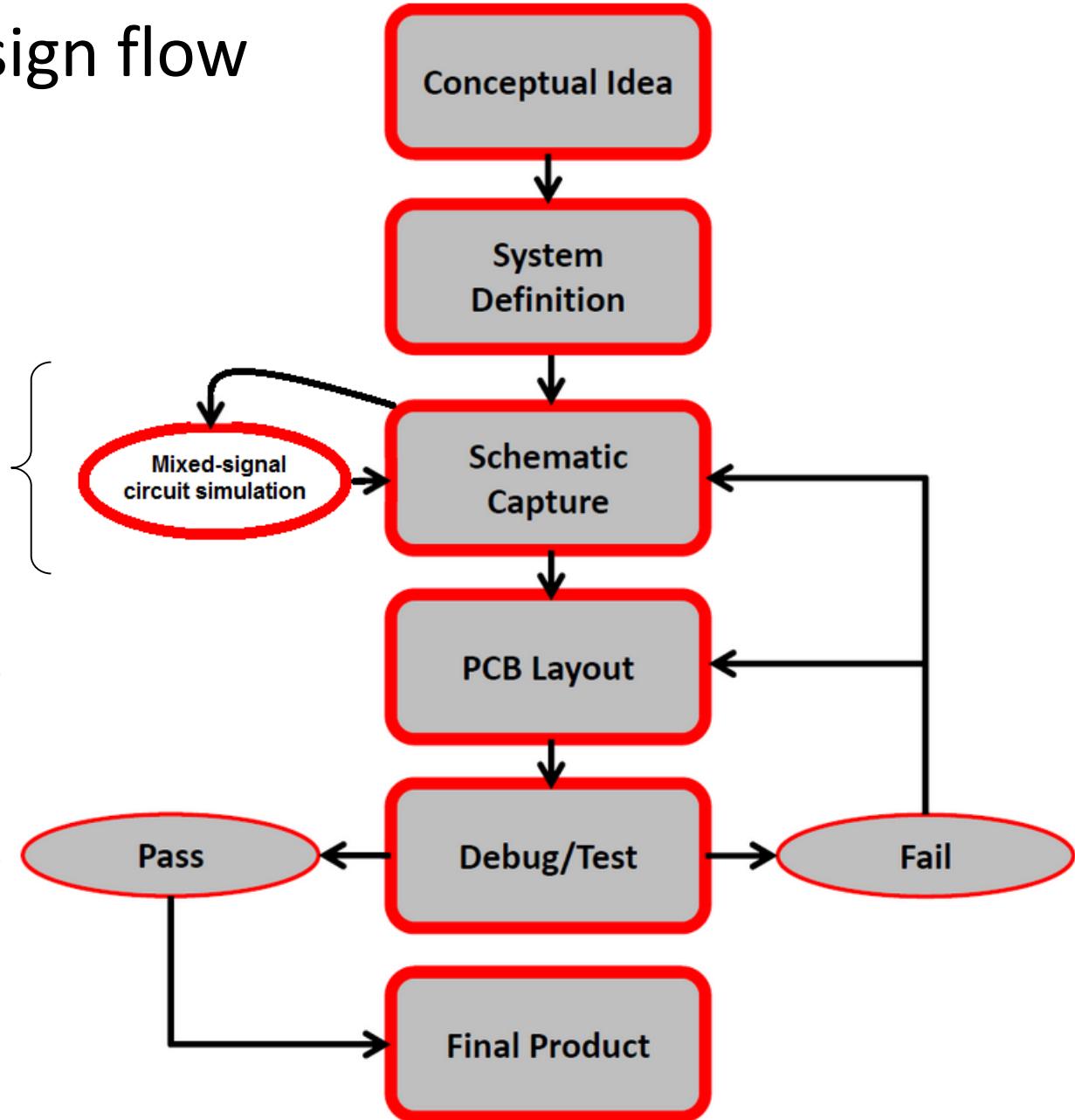
- Available to undergraduates:
 - 150 Licenses for [Altium Designer](#)
 - 150 Licenses for NI [Multisim](#) – [Ultiboard](#)
- Our preference is Altium designer v.2014
- Through CMC, Capstone students may gain access also to:
 - [Mentor Graphics PADS](#)
 - [Allegro](#)
 - [Upverter](#)
- Questions:
 - help@ece.ubc.ca , engservices@ece.ubc.ca

Licensing and support

- Our License agreements are for Academic use only (non-profit)
- When in doubt, ask Engineering Services to contact vendor
- Large license pools → restrict support requests through a local contact (ECE Engineering Services)
- You can use support forum, wiki pages etc.

Typical PCB Design flow

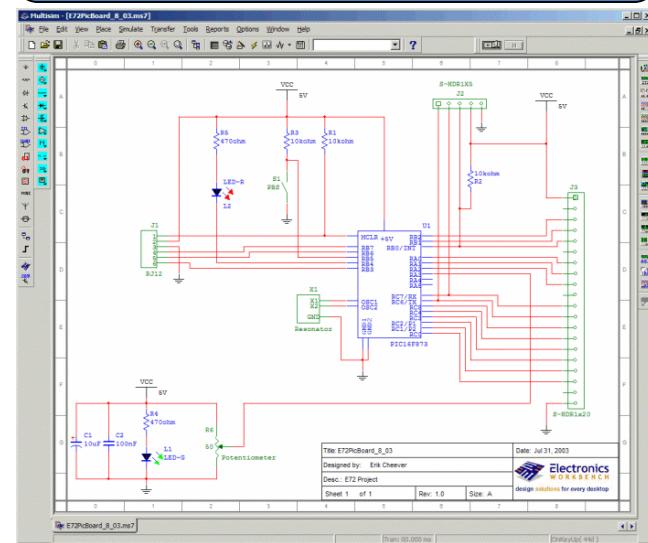
Front-end design
and capture



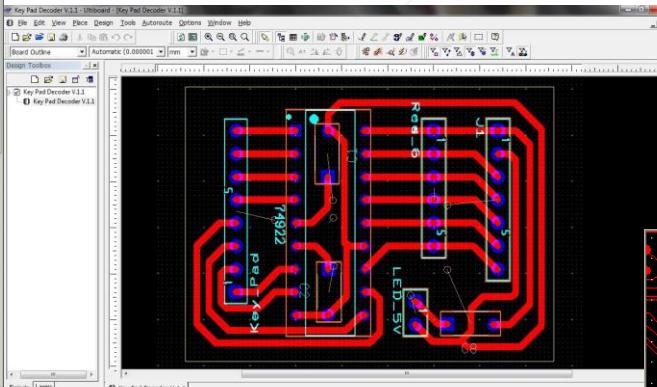
<http://e2e.ti.com/>

PCB Basic Design Flow

Schematic entry



PCB Layout

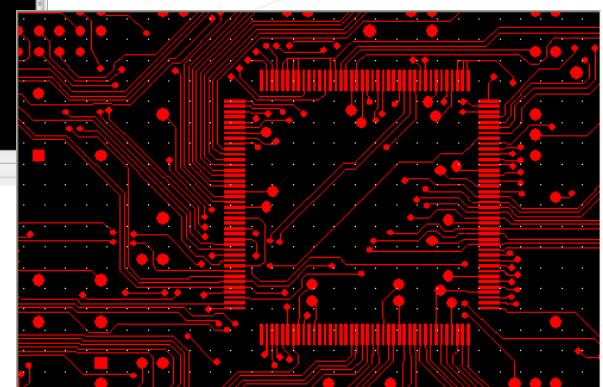


- Gerber: 274X

- Top, Bottom Cu
- Top, Bottom Solder
- Silkscreen
- Mechanical 1

- Drill files (tool list)

CAM files



- Symbol and Footprint creation
- Auto place
- Auto route ...

Altium Designer

A complete product development system

[System requirements](#) (MS WXP, W7, W8, problems with 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 2014

- Link to our download site:
<https://download.ece.ubc.ca/>
- Useful internal links:
<http://www.ece.ubc.ca/~leos/pages/tools/altium.html>
- <eng-services/fabrication/electrical-fabrication/pcb-design-and-assembly/>
- Create an account at Altium Live:
<http://live.altium.com/#signin> (may take several days)
Much faster: email engservices@ece.ubc.ca



Electronic Software Distribution

Search
<input type="text"/> Enter search term
<input type="button" value="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](#)[Eligibility](#)
[ISO Files](#)

1

Install 2014v

Electronic Software Distribution

ALTIUM DESIGNER 2014

File	Size	
README.html		README
AltiumDesignerSetup14_3_15.exe	8.9 MB	Windows instal
AltiumOfflineSetup14_3_15.zip	2.9 GB	Windows instal
EULA.pdf	56.2 KB	End-User Licen

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:	altium-lm.ece.ubc.ca
Server port:	21001

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

Altium Designer (14.3) - Workgroup [Workspace1.DsnWrk] - DXP://Home - Free Documents. Roberto Rosales signed in.

DXP File View Project Window Help

Projects Workspace1.DsnWrk Workspace

Start Workspace Documentation Vault Users Admin

License Management 1

File View Project

My Account - Roberto Rosales is now signed in

Private License Server Setup

Primary server

Servers list:

Server name: altium-lm.ece.ubc.ca 3

Server address:

Server port: 21001

Secondary server

Servers list:

Server name:

Server address:

Server port:

Search OK Cancel

2

Save standalone license file

Add standalone license file

Setup private license server

Disconnect from private license server

ece Electrical and Computer Engineering

Connecting to the Altium Vault

The screenshot shows the Altium Vault interface. At the top, there is a navigation bar with tabs: Home, Start, Workspace, Documentation, Vault (which is selected), Users, and Admin. Below the navigation bar, the word "Vault" is displayed in large letters. Underneath "Vault", there are three tabs: Content (selected), Catalog, and Configuration. A yellow warning icon with an exclamation mark is visible. The main message is "You are not connected to any Vault Server." Below this message, there is a link to "DXP Preferences - Data Management - Vaults settings". Another link below it provides more information about design data management, pointing to <http://live.altium.com/#vaults>.

The screenshot shows the "Preferences" dialog box. The title bar says "Preferences". Inside, there is a section titled "Cloud Preferences" with a "Cloud" icon and a note: "Your settings can now be stored in the cloud. Once you are signed in simply enable your cloud preferences." There is a "Disable cloud preferences" button. On the left, there is a tree view of preferences categories: System, Data Management (which is expanded), Version Control, Design Repositories, Vaults (which is selected and highlighted in blue), Publishing Destinations, Backup, File Locking, Local History, Templates, and Installed Libraries. On the right, under "Data Management – Vaults", there is a "Vaults" section with a note: "A vault stores structured and revised data that is characterized by lifecycle states." A red arrow points from the "Vaults" section in the main interface down to the "Vaults" section in the preferences dialog. Below this, there is a table:

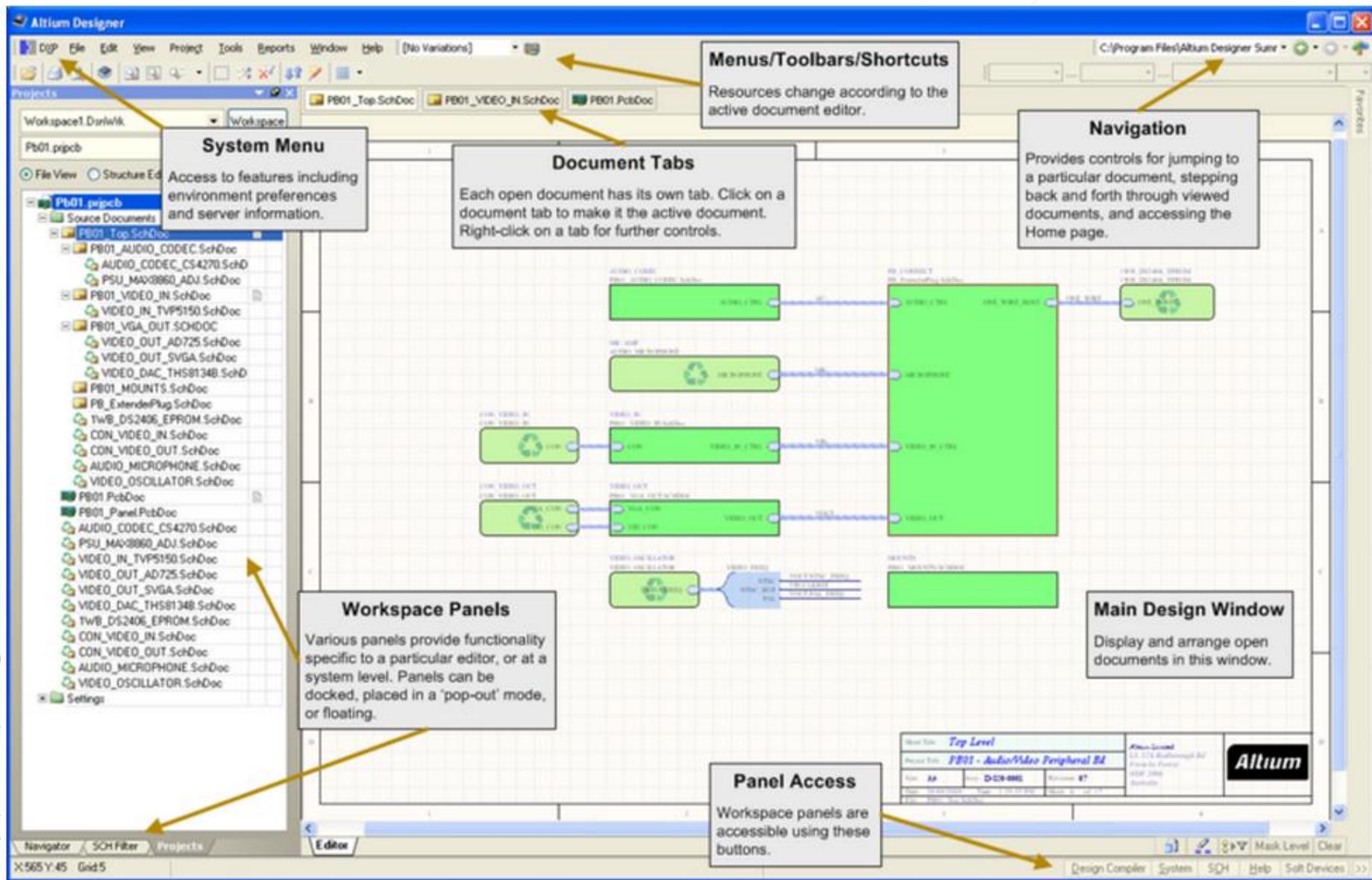
Name	Description	Address	Status	Enab...
Altium Content Vault	Altium Content Vault	http://vault.live.altium.com	✓	<input checked="" type="checkbox"/>

place of mind

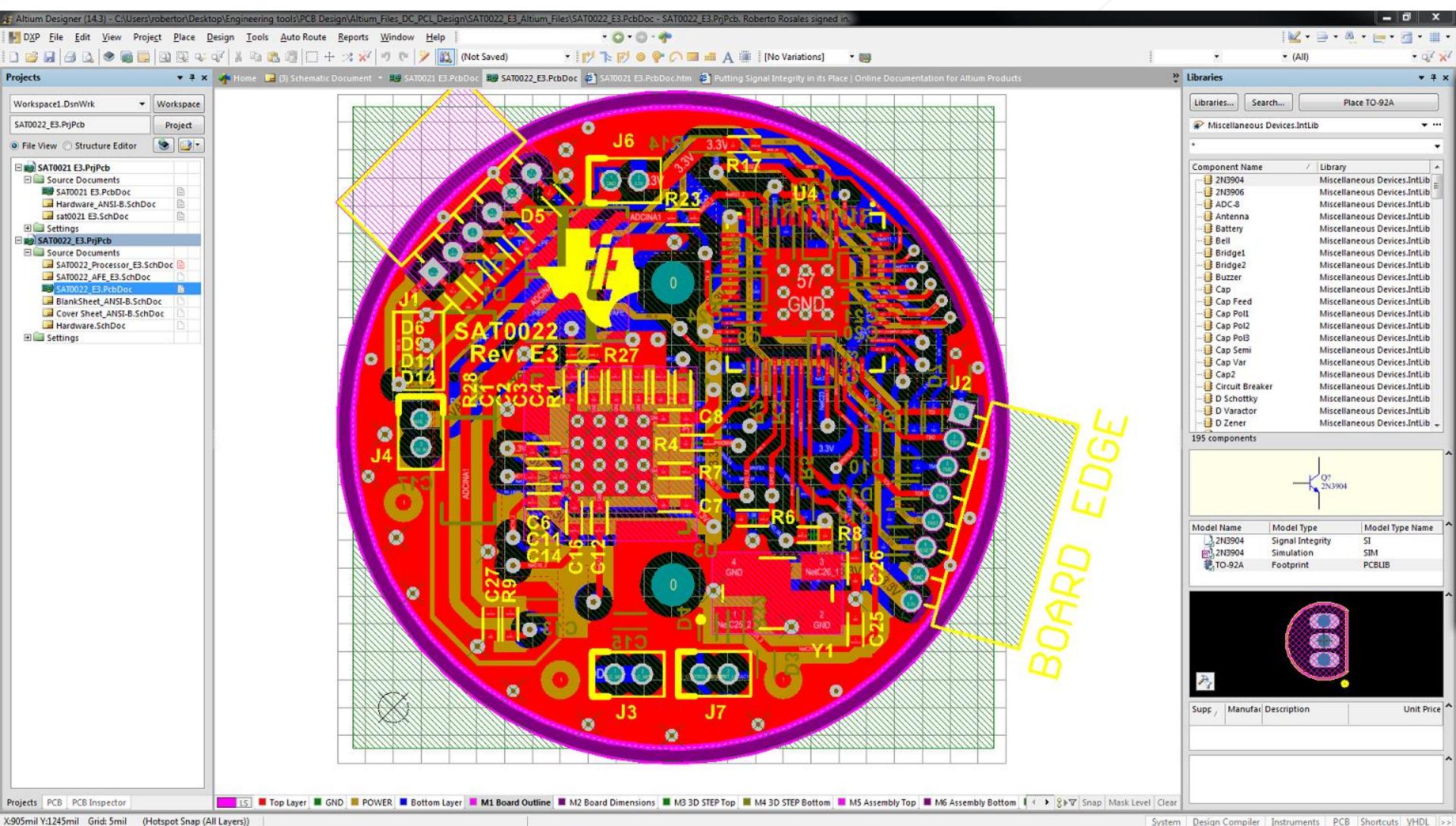
Understanding Altium

- 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:
 - Save preferences
 - <http://live.altium.com/> (forum, design content, blog)

Altium Design Environment



Example: TI Reference Design



Understanding Altium

(Basics for the single user)

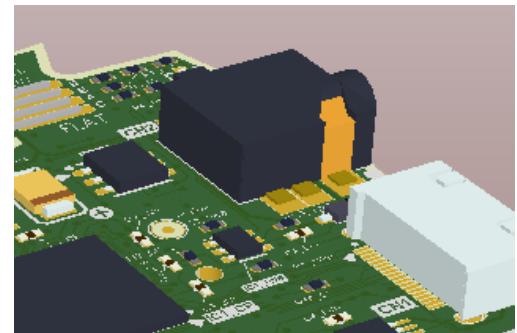
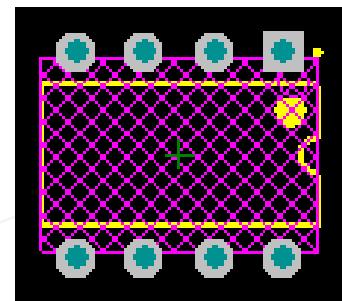
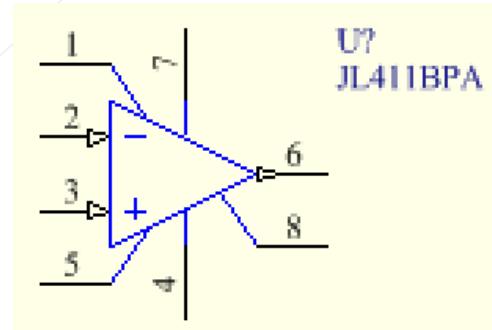
- Projects (project panel, active project)
- Workspace Panels (system-wide, editor-specific)
- Editors:
 - Schematic
 - Symbol editor
 - PCB layout
 - Footprint editor
 - CAM files (CAMtastic panel)
- Components and Libraries
- Keyboard shortcuts
 - <Shift + F1> while running a command
 - <Esc> or Right Click to exit a command
- Save documents to see some changes take effect

Altium Projects

- Project: collection of design documents (*.PrjPcb)
 - 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

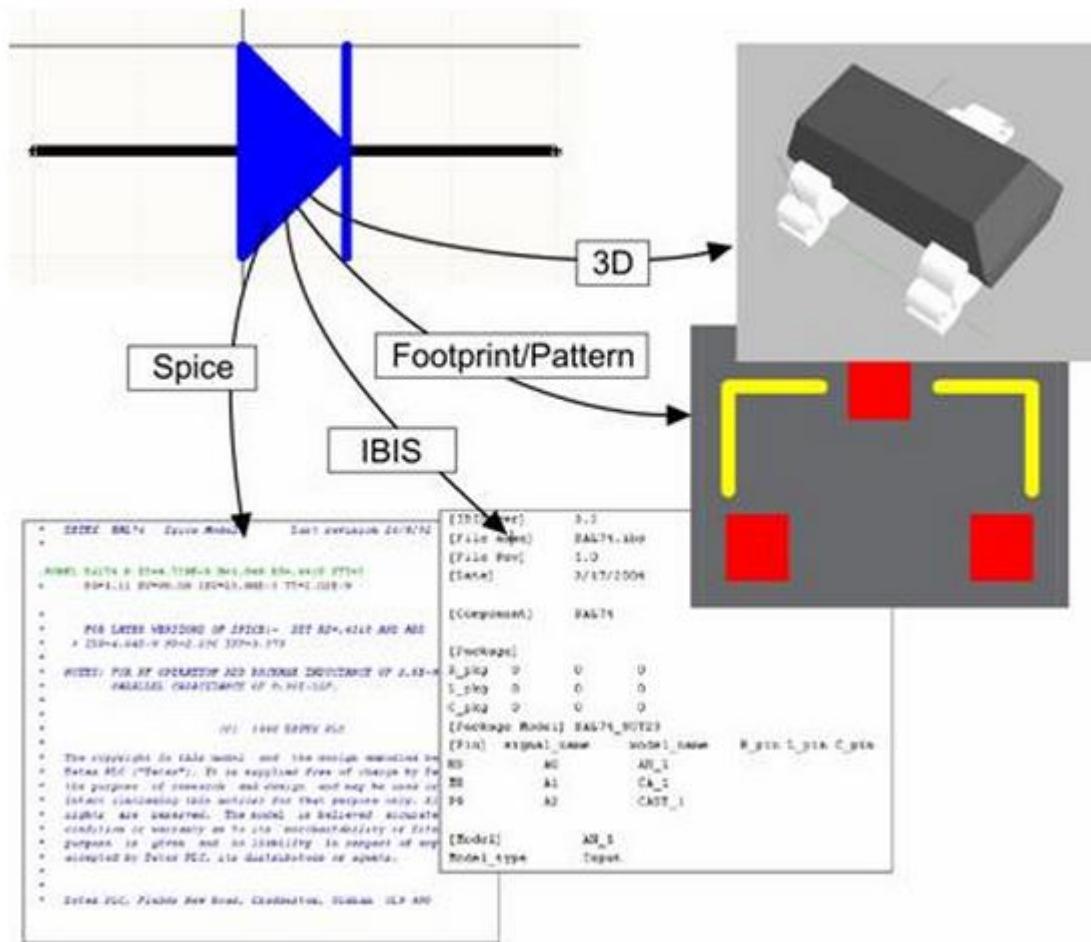
Component, Model and Library Concepts

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



Component, Model and Library Concepts

The built-in capability to create component visual representations, assign parameters, and create links between representations is very sophisticated

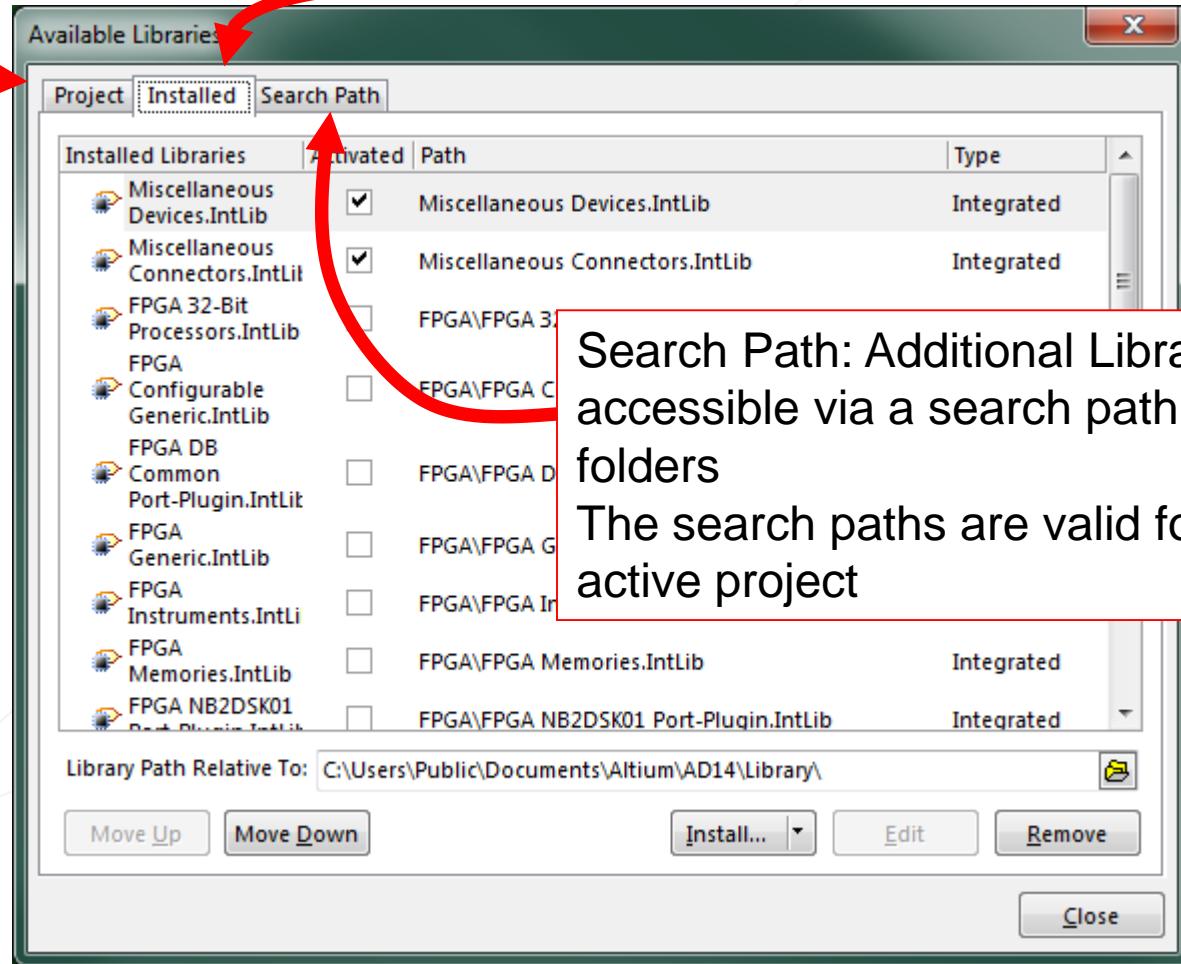


Libraries = collection of components

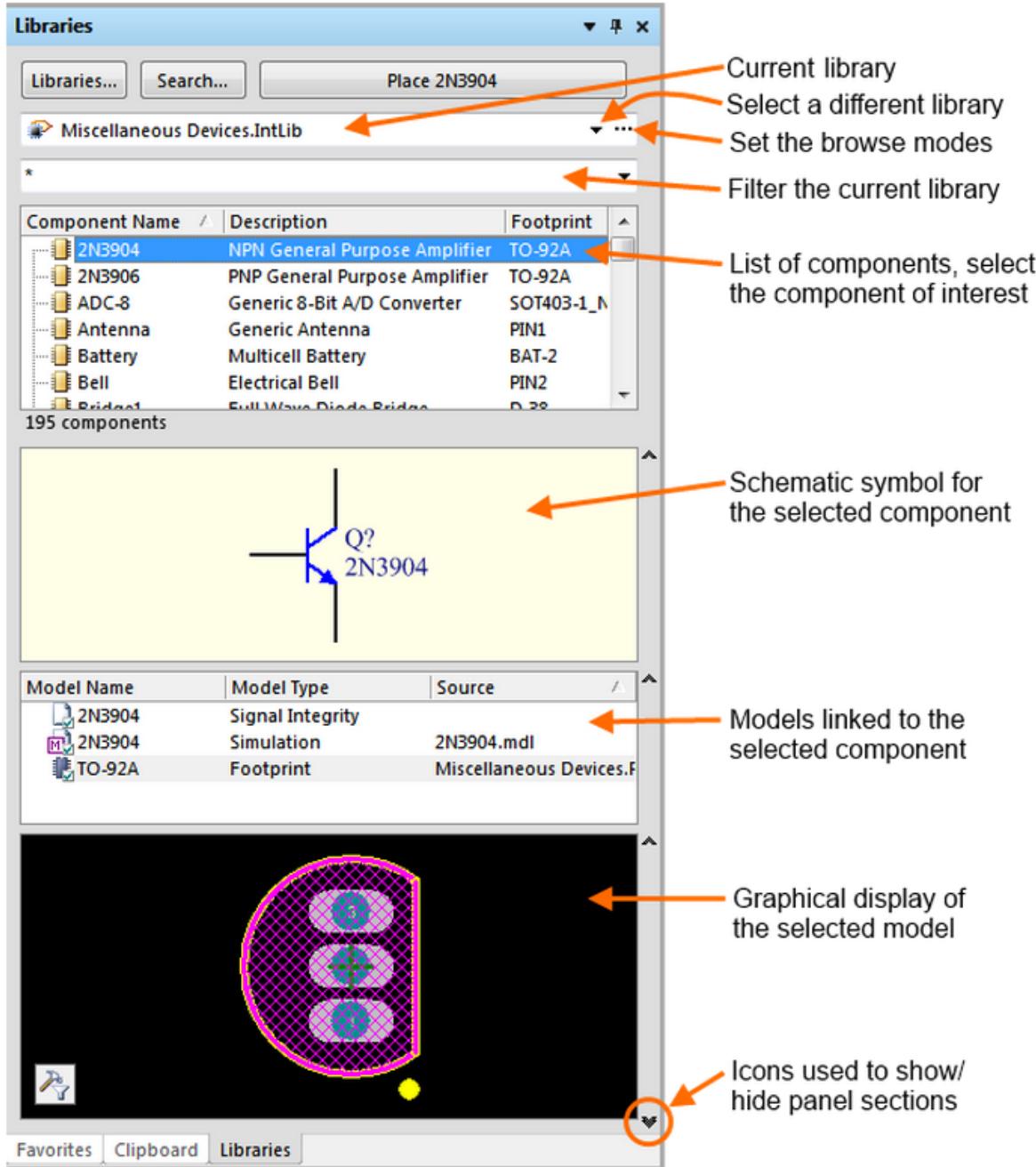
- Collection of components, models or both
- Model Libraries (*.MDL, *.CKT, *.PCBLib)
 - Simulation models are one file per model
- Schematic Libraries (*.SchLib)
 - Symbol and a link to a model library
- Integrated Libraries (*.IntLib)
 - Symbol, footprint and other models are compiled into a single 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



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 ...



a place of mind

Obtaining integrated libraries

1. Frozen libraries: [from here](#)

you can install anywhere but it is a good idea to make a subfolder under:

C:\Users\Public\Documents\Altium\AD14\Library

or a cloud storage service if you use more than one PC

2. AltiumLive website: [Resources / Design Content](#)



Manufacturer: National Semiconductor

Updated: 3+ months ago

Tags: Analog, Amplifier

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

This is useful to preview component

[GO TO VAULT](#)

[DOWNLOAD LIBRARY](#)

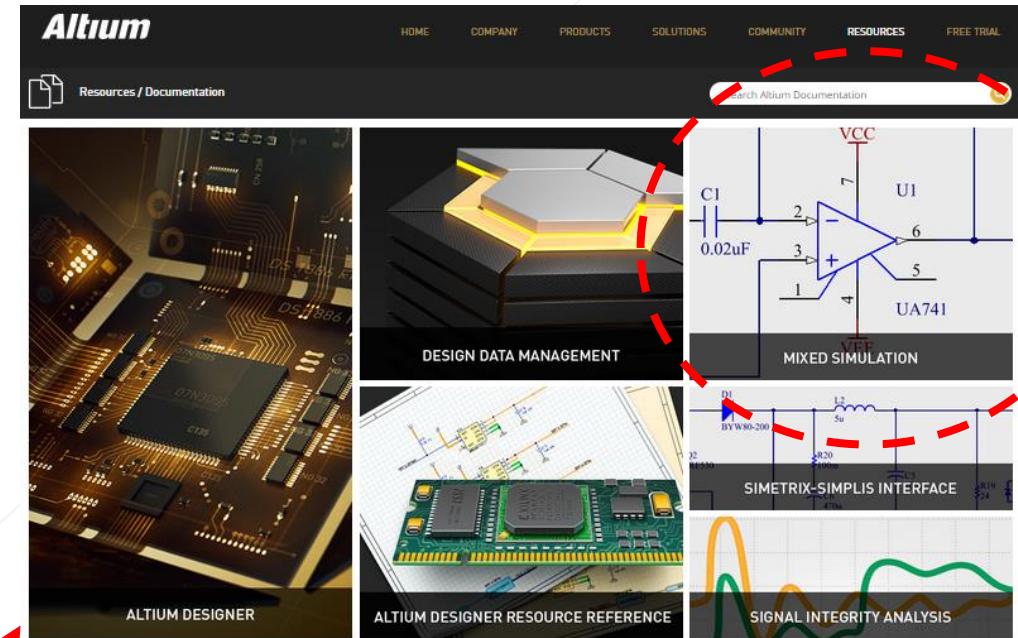
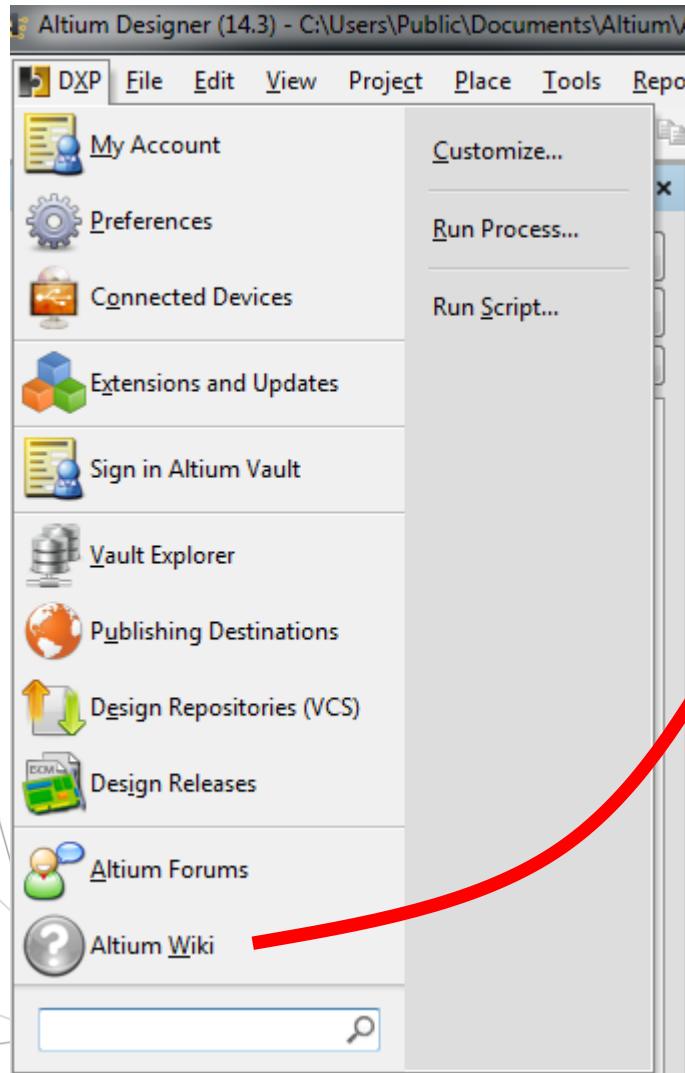
This downloads a .zip file for the complete library

3. [Altium Vault](#)



a place of mind

Learning how to use Altium



Until recently: best training guides were for Altium 2009
(pdf lesson files organized in chapters)
But DXP menus have changed since
The same information is now updated in the Altium_wiki

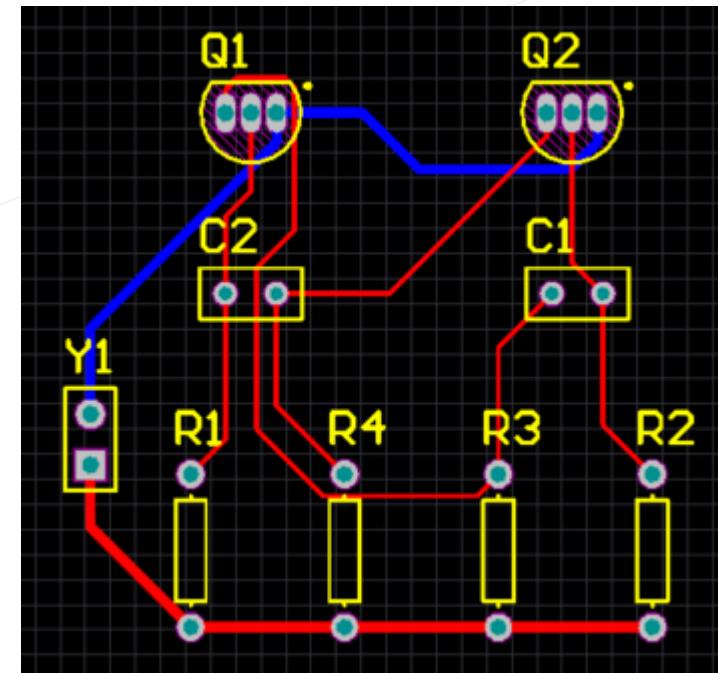
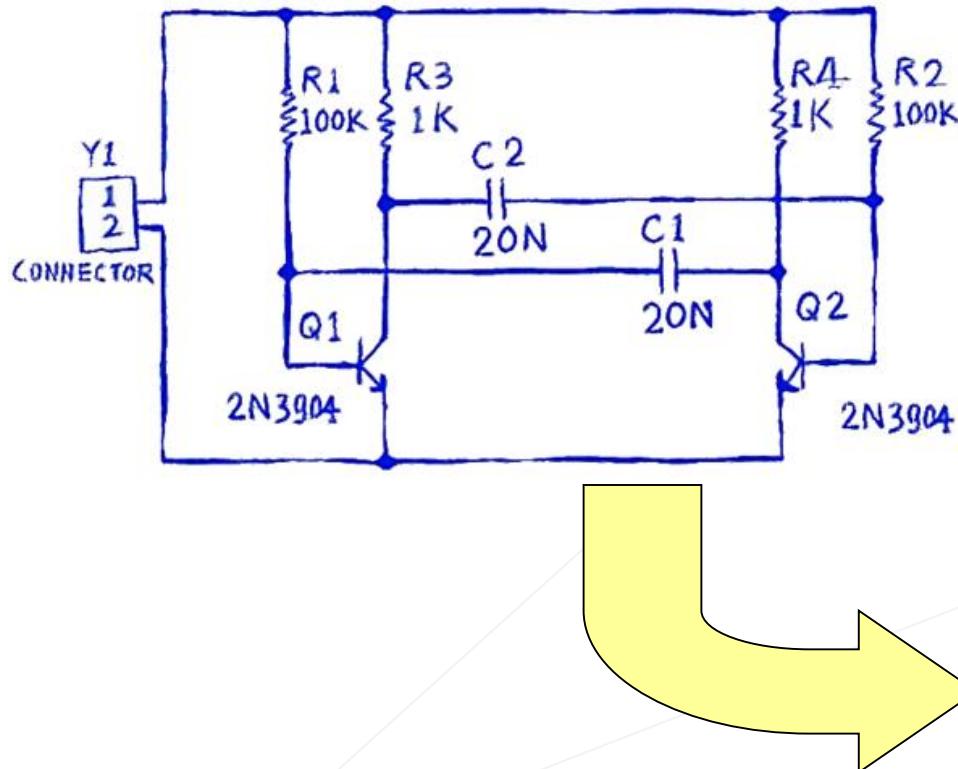


a place of mind

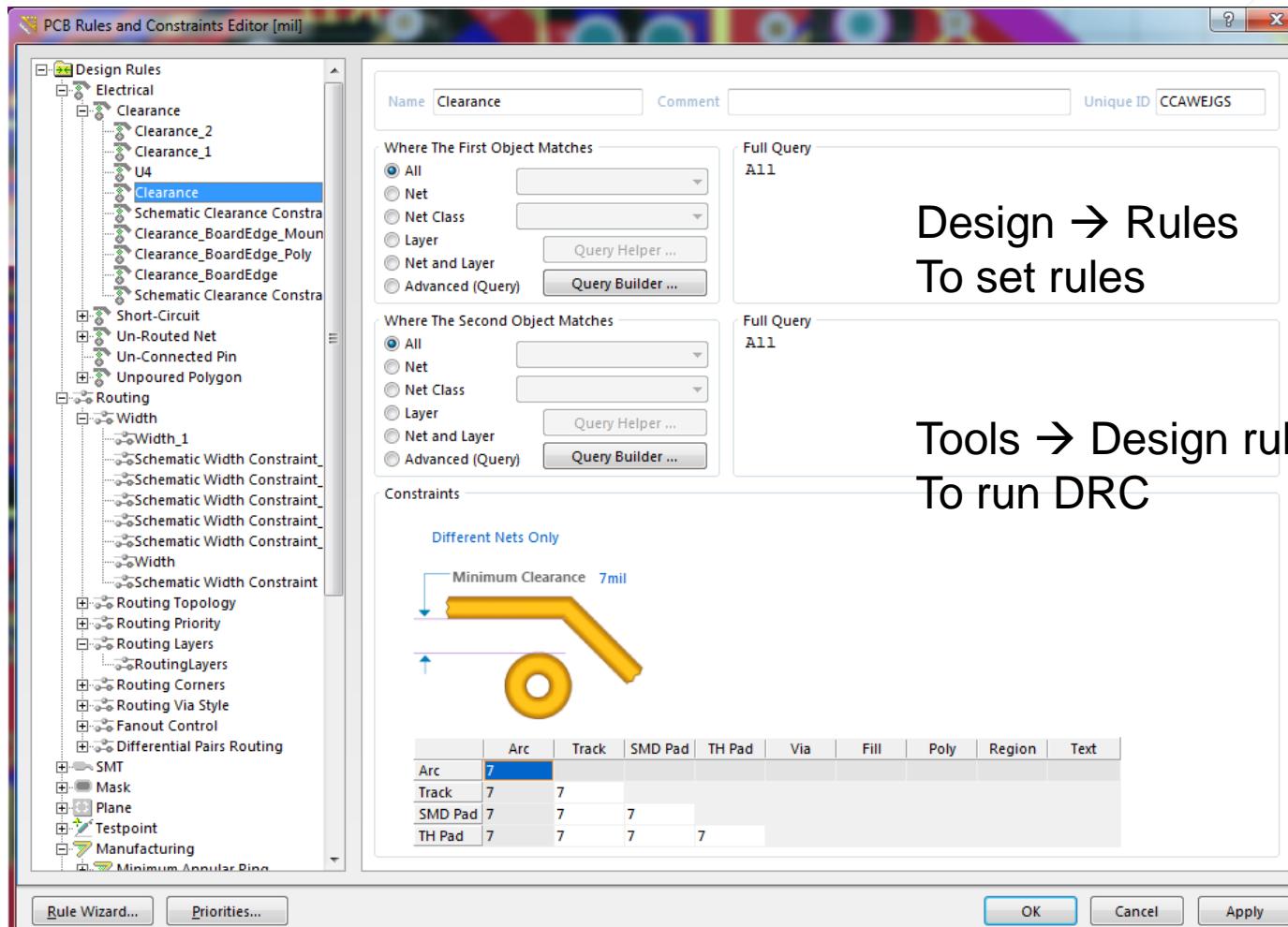
Board Implementation

Tutorial - Getting Started with PCB Design

<http://techdocs.altium.com/display/ADOH/Board+Implementation>



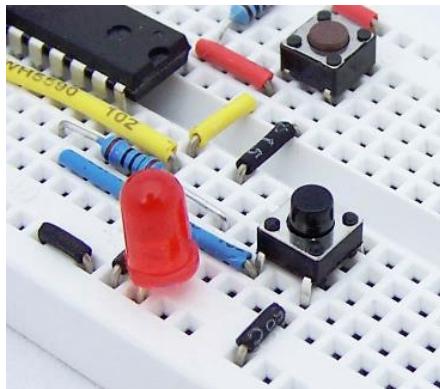
Design Rule Check



Working units

- Imperial (inches)

- $1/1000^{\text{th}}$ of an inch = 1 mil
= 1 thou
- 100mils (0.1") is a common dimension

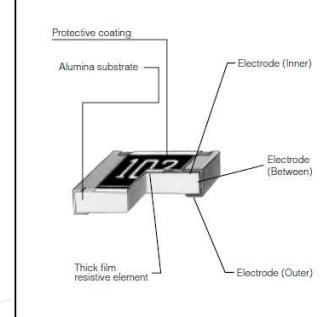


- Metric (mm)

- $1 \text{ mm} \neq 1\text{mil}$! ($\sim 40\text{mils}$)
- Common unit in SM parts

Panasonic

■ Construction



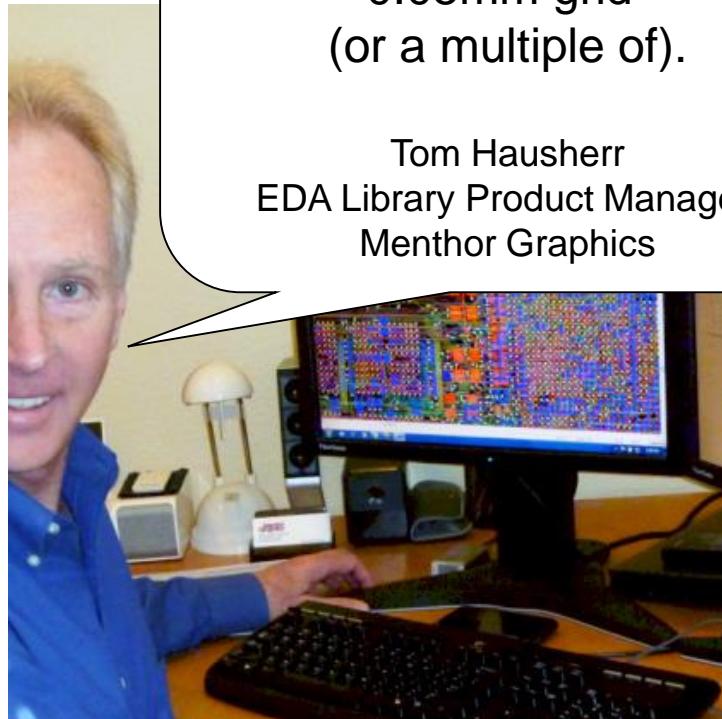
Thick Film Chip Resistors

■ Dimensions in mm (not to scale)

Type (inches)	Dimensions (mm)					Mass (Weight) (g/1000 pcs.)
	L	W	a	b	t	
ERJXG (01005)	0.40 ^{+0.05} _{-0.05}	0.20 ^{+0.05} _{-0.05}	0.10 ^{+0.03} _{-0.03}	0.10 ^{+0.03} _{-0.03}	0.13 ^{+0.05} _{-0.05}	0.04
ERJHG (0201)	0.60 ^{+0.05} _{-0.05}	0.30 ^{+0.05} _{-0.05}	0.10 ^{+0.06} _{-0.06}	0.15 ^{+0.06} _{-0.06}	0.23 ^{+0.03} _{-0.03}	0.15
ERJZG (0402)	1.00 ^{+0.05} _{-0.05}	0.50 ^{+0.05} _{-0.05}	0.20 ^{+0.10} _{-0.10}	0.25 ^{+0.05} _{-0.05}	0.35 ^{+0.06} _{-0.06}	0.8
ERJSG (0603)	1.60 ^{+0.10} _{-0.10}	0.80 ^{+0.10} _{-0.10}	0.30 ^{+0.20} _{-0.20}	0.30 ^{+0.16} _{-0.16}	0.45 ^{+0.10} _{-0.10}	2
ERJ6G (0805)	2.00 ^{+0.20} _{-0.20}	1.25 ^{+0.20} _{-0.20}	0.40 ^{+0.20} _{-0.20}	0.40 ^{+0.20} _{-0.20}	0.60 ^{+0.20} _{-0.20}	4
ERJ8G (1206)	3.20 ^{+0.25} _{-0.25}	1.60 ^{+0.25} _{-0.25}	0.50 ^{+0.20} _{-0.20}	0.50 ^{+0.20} _{-0.20}	0.60 ^{+0.20} _{-0.20}	10
ERJ14 (1210)	3.20 ^{+0.25} _{-0.25}	2.50 ^{+0.25} _{-0.25}	0.50 ^{+0.20} _{-0.20}	0.50 ^{+0.20} _{-0.20}	0.60 ^{+0.20} _{-0.20}	16
ERJ12 (1610)	4.50 ^{+0.25} _{-0.25}	3.20 ^{+0.25} _{-0.25}	0.50 ^{+0.20} _{-0.20}	0.50 ^{+0.20} _{-0.20}	0.60 ^{+0.20} _{-0.20}	27
ERJ12 (2010)	5.00 ^{+0.25} _{-0.25}	2.50 ^{+0.25} _{-0.25}	0.60 ^{+0.20} _{-0.20}	0.60 ^{+0.20} _{-0.20}	0.60 ^{+0.20} _{-0.20}	27
ERJ11 (2512)	6.40 ^{+0.25} _{-0.25}	3.20 ^{+0.25} _{-0.25}	0.65 ^{+0.20} _{-0.20}	0.60 ^{+0.20} _{-0.20}	0.60 ^{+0.20} _{-0.20}	45

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

Metric or Imperial?



"every element in a PCB design should reside on a 0.05mm grid"
(or a multiple of).

Tom Hausherr
EDA Library Product Manager,
Mentor Graphics

Old PCB wisdom:
“thou shall use thous”
David L. Jones
EEV blog



Comment driven by high density & modern surface mount technology

Comment driven by traditional 0.1" spacing between pins

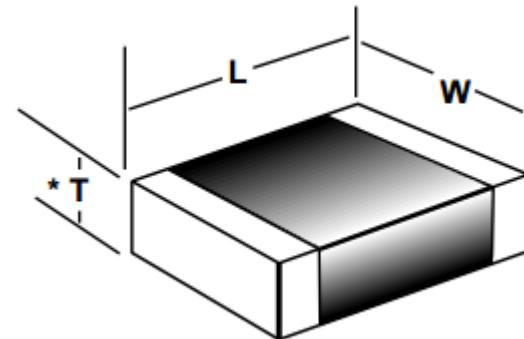
Surface Mount Packages

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

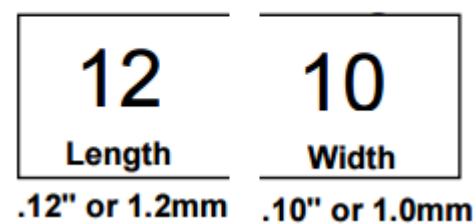
http://www.topline.tv/SMT_Nomenclature.pdf

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

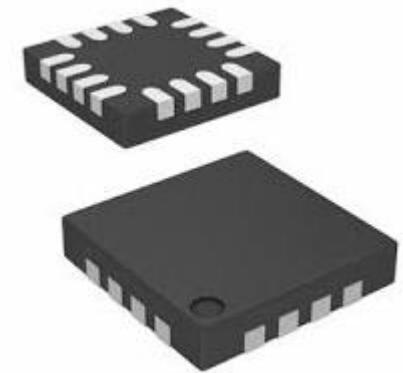
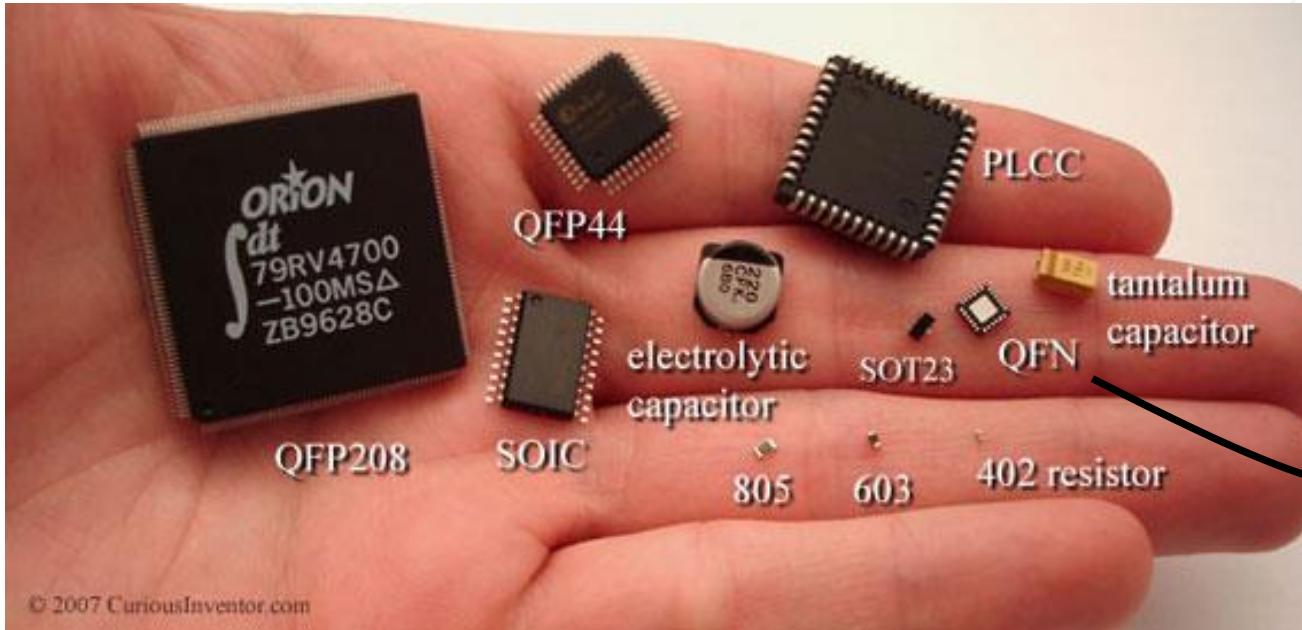
Caution: Overlapping size codes. Metric appears same as inches.



Example:



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
 - http://www.nxp.com/documents/other/Discretes_package_poster.pdf
 - http://www.nxp.com/documents/other/Discrete_Flat_No-leads_DFN_package_poster.pdf

Design Files

- PCB CAD/EDA Tools save design data in their own file format
- PCB Manufacturers employ specific file formats
 - One file per drawing layer and one file per drill layer, one aperture file, one text file for stack-up instructions, and one or more drawing files with instructions
 - Gerber and aperture: graphic file format used for each physical layer, drawing layer, and mask of the PCB.
 - NC (Numeric Control) Drill files: describes drilling coordinates and tool sizes
 - Bill of materials (BOM): detailed list of all needed components used for assembly

CAM {

CAM files

Examples of Altium CAM file extensions:

- *.GBl Gerber bottom layer
- *.GTL Gerber top layer
- *.GTO Gerber top overlay
- *.GBO Gerber bottom overlay
- *.GTS Gerber top solder
- *.GBS Gerber bottom solder
- *.GM1 Gerber Mechanical 1
- *.TXT NC Drill file
- *.DRL Drill tool list

CAM Files

- Several formats for Gerber (274D, 274X) and NC (Excellon I and II)
- Recent attempts to upgrade to IPC-2581, Gerber X2
- Units: imperial or metric, file formats can be 2.4 or 2.3, specify to remove leading or trailing zeros.
 - E.g. 10.7500 in 2.4 format = 107500, no trailing zeros: 1075
 - E.g. 00.03 in 2.3 format = 00030, no leading zeros: 30
- File extensions vary with PCB CAD tool

CAM Files

- Aperture codes:
 - Universal codes: D01, D02, D03 light on, light off, flash
 - Above D10 represent the aperture position on the wheel. These codes are user defined and must be provided to manufacturer.
- Gerber: X and Y coordinates and aperture commands
 - 274D does not define aperture, a separate aperture file must be provided
 - 274X aperture information is embedded in the Gerber files.

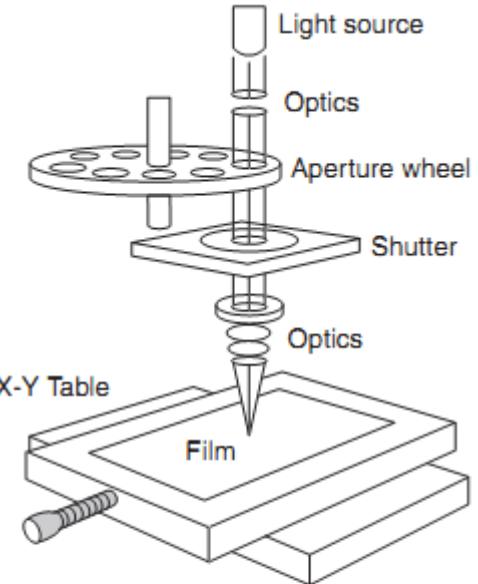


FIGURE 18.8 A photo plotter uses a stationary light source to deliver light through a rotating variable size aperture wheel (which determines the size of the light spot on the table) onto a table that can move in the x and y directions. A shutter shuts or allows light passage as required. Precision optics control the focus of the light spot.

CAM Files

inches

274X

b10	ROUNDED	10.000	10.000	0.000 FLASH
D11	ROUNDED	30.000	30.000	0.000 LINE
D12	ROUNDED	10.000	10.000	0.000 LINE
D13	ROUNDED	85.000	85.000	0.000 FLASH
D14	ROUNDED	95.000	95.000	0.000 FLASH
D15	ROUNDED	118.000	118.000	0.000 FLASH
D16	ROUNDED	110.000	110.000	0.000 FLASH
D17	ROUNDED	158.000	158.000	0.000 FLASH
D18	ROUNDED	86.000	86.000	0.000 FLASH
D19	OCTAGONAL	118.000	118.000	0.000 FLASH
D20	ROUNDED	68.000	68.000	0.000 FLASH
D21	OCTAGONAL	68.000	68.000	0.000 FLASH
D22	ROUNDED	70.866	70.866	0.000 FLASH
D23	ROUNDED	80.000	80.000	0.000 FLASH
D24	OCTAGONAL	80.000	80.000	0.000 FLASH
D25	ROUNDED	8.000	8.000	0.000 LINE
D26	ROUNDED	39.370	39.370	0.000 LINE
D27	ROUNDED	93.000	93.000	0.000 FLASH
D28	ROUNDED	103.000	103.000	0.000 FLASH
D29	ROUNDED	126.000	126.000	0.000 FLASH
D30	ROUNDED	166.000	166.000	0.000 FLASH
D31	ROUNDED	94.000	94.000	0.000 FLASH
D32	OCTAGONAL	126.000	126.000	0.000 FLASH
D33	ROUNDED	76.000	76.000	0.000 FLASH
D34	OCTAGONAL	76.000	76.000	0.000 FLASH

For Help, press F1

File Edit View Insert Format Help

%FSLAX23123*%
%MOIN*%
G70*
G01*
G75*
%ADD10C, 0.010*%
%ADD11C, 0.030*%
%ADD12C, 0.010*%
%ADD13C, 0.085*%
%ADD14C, 0.095*%
%ADD15C, 0.118*%
%ADD16C, 0.110*%
%ADD17C, 0.158*%
%ADD18C, 0.086*%
%ADD19P, 0.118XEX0*%
%ADD20C, 0.068*%
%ADD21P, 0.068XEX0*%
%ADD22C, 0.071*%
%ADD23C, 0.080*%
%ADD24P, 0.080XEX0*%
D10*
X9948Y8520D02*
D03*
D03*
X9886Y9155D02*
D03*

For Help, press F1

NC Drill files

- A separate Numeric Control (NC) format is used for hole drilling
- Most commonly used format is from Excellon Company → Excellon NC Drill.
- NC drill file is similar to a Gerber file: Tool information, X/Y coordinates, some special codes (e.g. M30 end of program)
- Separate files for allocation of plated and non-plated holes

NC Drill Files

Azure.DRR - WordPad

NCDrill File Report For: Azure.PCB 19-Oct-2011 23:13:38

Layer Pair : TopLayer to BottomLayer
ASCII File : NCDrillOutput.TXT
EIA File : NCDrillOutput.DRL

Tool	Hole Size	Hole Count Plated	Tool Travel
T1	44mil (1.1176mm)	112	19.22 Inch (488.10 mm)
T2	60mil (1.524mm)	121	40.26 Inch (1022.72 mm)
T3	67mil (1.7018mm)	36	20.41 Inch (518.52 mm)
T4	86mil (2.1844mm)	74	35.74 Inch (907.85 mm)
T5	86mil (2.1844mm)	2	NPTH 8.37 Inch (212.71 mm)
T6	110mil (2.794mm)	2	NPTH 2.26 Inch (57.31 mm)
T7	161mil (4.0894mm)	3	NPTH 15.03 Inch (381.64 mm)
Totals		350	141.29 Inch (3588.86 mm)

Total Processing Time : 00:00:00

For Help, press F1

Azure.TXT - WordPad

M72
M48
T1F00S00C0.044
T2F00S00C0.060
T3F00S00C0.067
T4F00S00C0.086
T5F00S00C0.086
T6F00S00C0.110
T7F00S00C0.161
%
T01
X3681Y318
X3681Y416
X3681Y514
Y613
X3827
X3827Y514
X3827Y416
X3827Y318
X3965
Y416
Y514
X4063
X4161
Y416
.....

Drill bit sizes

- Drill bit sizes:
 - Inches and mm. (try to not mix)
 - Some manufacturers charge extra for additional sizes
 - For Drill Bit Conversion Table refer to:
http://en.wikipedia.org/wiki/Drill_bit_sizes

Drill bit sizes

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

Design hand off

- Good example: TI DC PLC Reference Design
 - <http://www.ti.com/tool/24vdcplcevm>
- Be consistent with your naming convention
 - Choose a meaningful name for the design
- Provide
 - Top level design guide /summary document
 - schematics in pdf format (.zip)
 - Gerber files (.zip)
 - BOM (Bill of materials) (.zip)
(Schematic: Reports → Bill of materials)
 - Altium design files (.zip)
 - Libraries (PCB Editor: Design → Make PCB Library)

Recommended Manufacturers

Local high end

- Ominicircuits
<http://www.omnicircuitboards.com/>
- CCI Canadian Circuits
<http://www.canadiancircuits.com/>

Canada – simple and fast

- APC Alberta Circuit Boards (very useful price estimator)
<http://www.apcircuits.com/>
- DYCO
<http://www.dyco-circuits.com/>

If price is an issue

- MYRO PCB
<http://www.myropcb.com/>
- Speedy PCB
<http://www.speedypcb.com/>

There are many more ...