



Altium I

(Circuit Design & Simulation)

ELEC391

PCB Design support for ELEC391:

Altium 2014, 150 licenses

Lecture talks:

- Jan 22 Altium I (Circuit Design + Simulation)
- Feb 1 Altium II (PCB Layout)
- TBA Guest Lecture – PCB Production
- Support & submission instructions posted [here](#)

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

Mon: 16:00-18:00

Tue : 09:00-11:00 / 14:00-16:00 / 16:00-18:00

Wed: 09:00-11:00 / 16:00-18:00

Thu : 09:00-11:00 / 14:00-16:00 / 16:00-18:00

Fri : 09:00-11:00

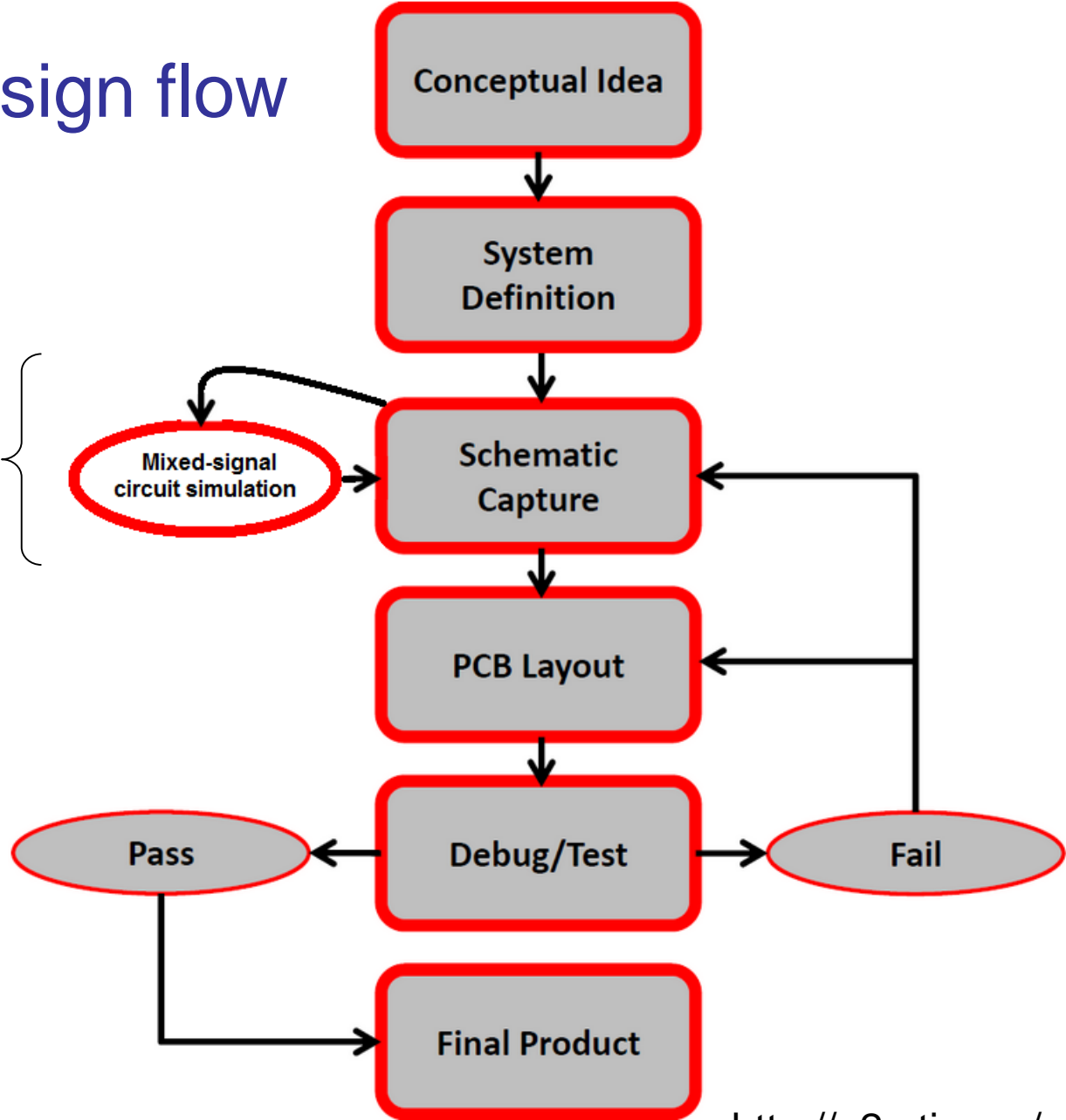
Contents

- How to install Altium Designer 2014
- Understanding Altium Designer
- Walk-through Tutorial
 - Schematic Capture
 - Mixed signal simulations
- SPICE basic concepts

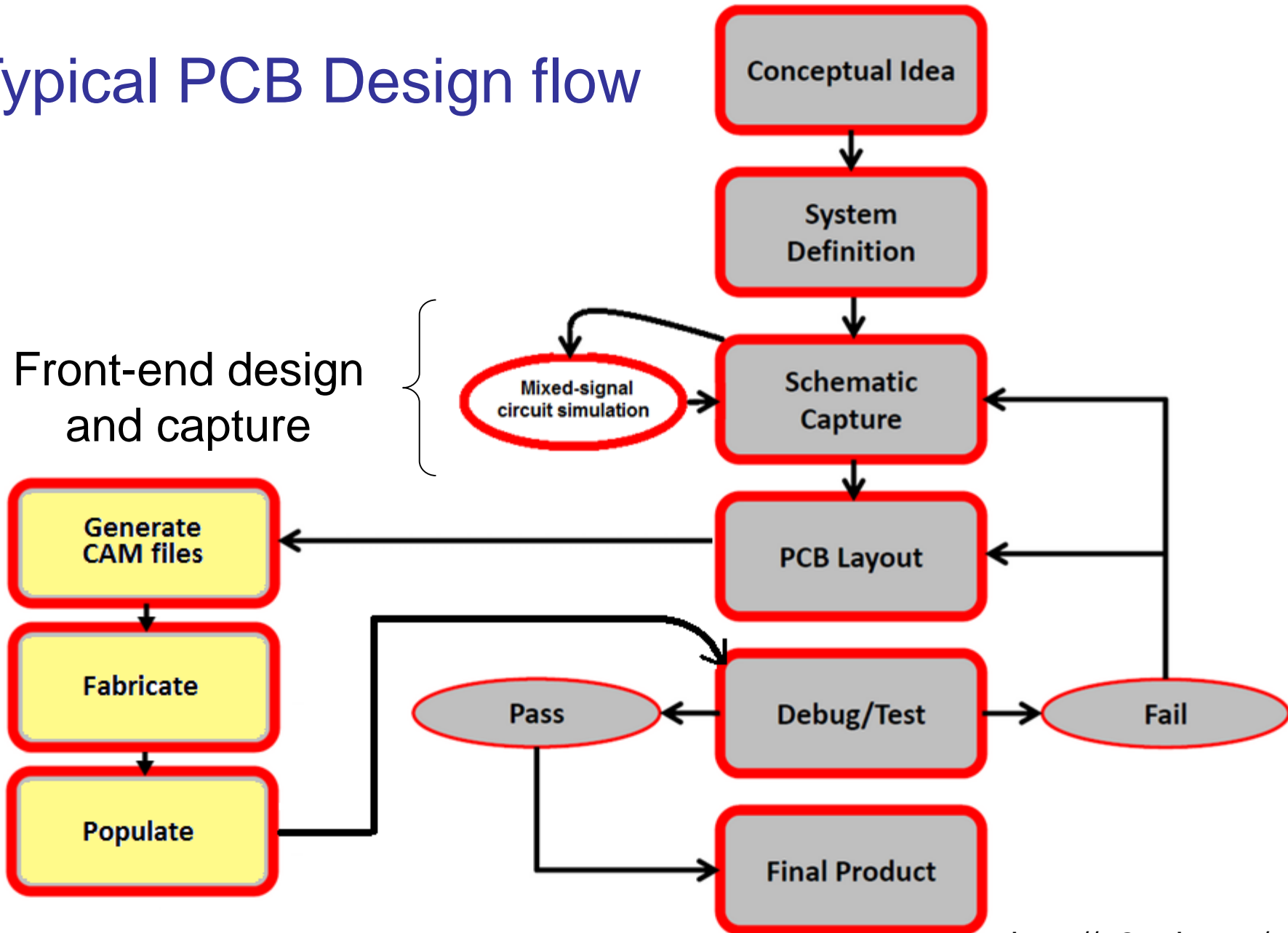
Credits: Unless explicitly stated all source material is from the Altium website and Altium training documents.

Typical PCB Design flow

Front-end design
and capture



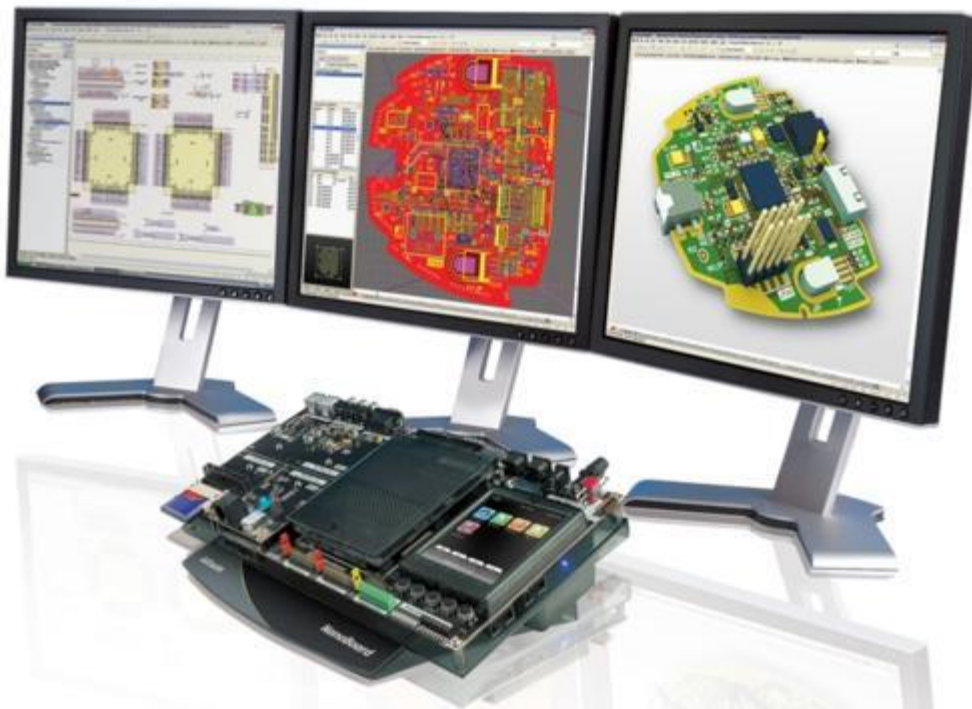
Typical PCB Design flow



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 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)



Electronic Software Distribution

Install 2014v

Search

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

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

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:

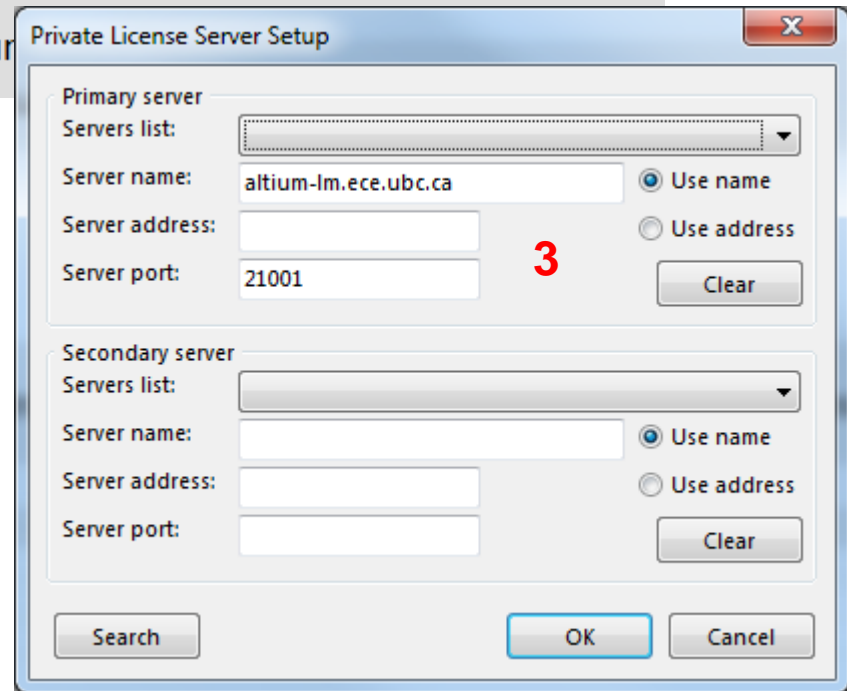
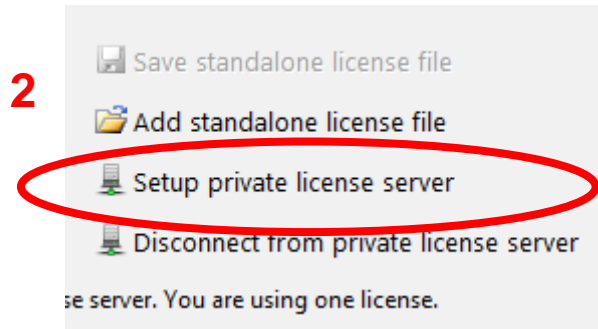
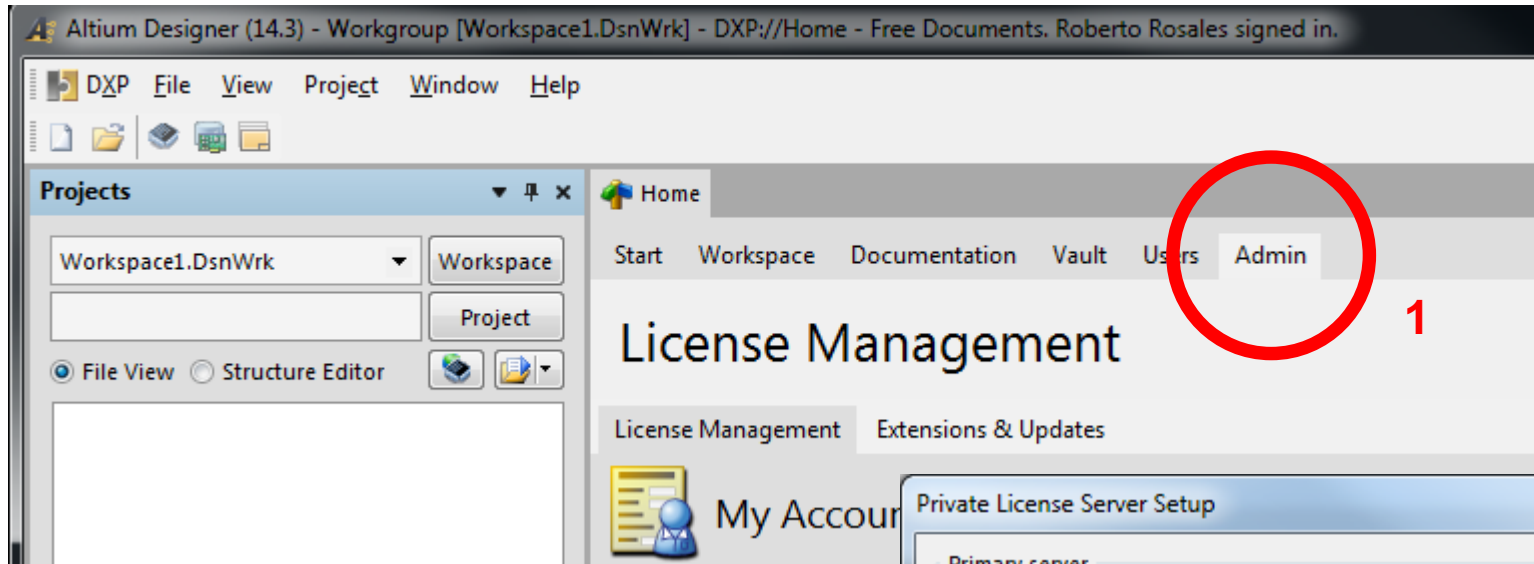
- A wired connection on the ECE network
- A wired connection on UBC ResNet
- A wireless connection at the UBC Vancouver campus on the ubcprivate, ubcsecure, or ubc network (ubcvisitor and eduroam are not sufficient)
- A [myVPN](#) connection to the UBC Vancouver network
- A myVPN connection to the ece.prof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

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

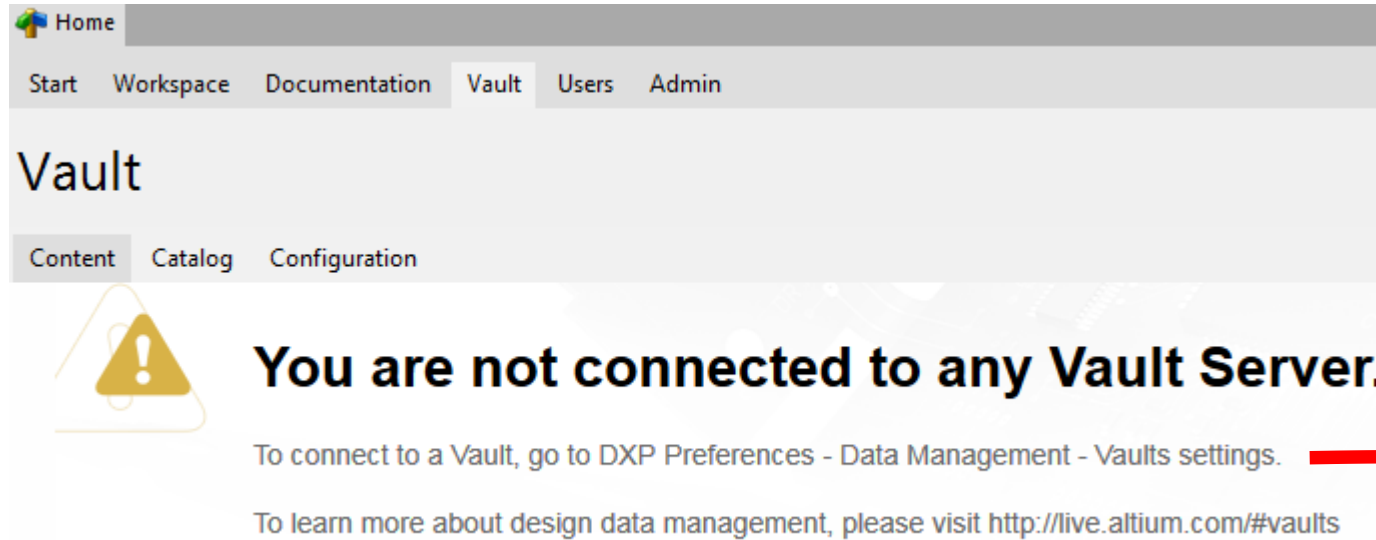
Server name:	altium-lm.ece.ubc.ca
Server port:	21001

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




Connecting to the Altium Vault



Home Start Workspace Documentation Vault Users Admin

Vault

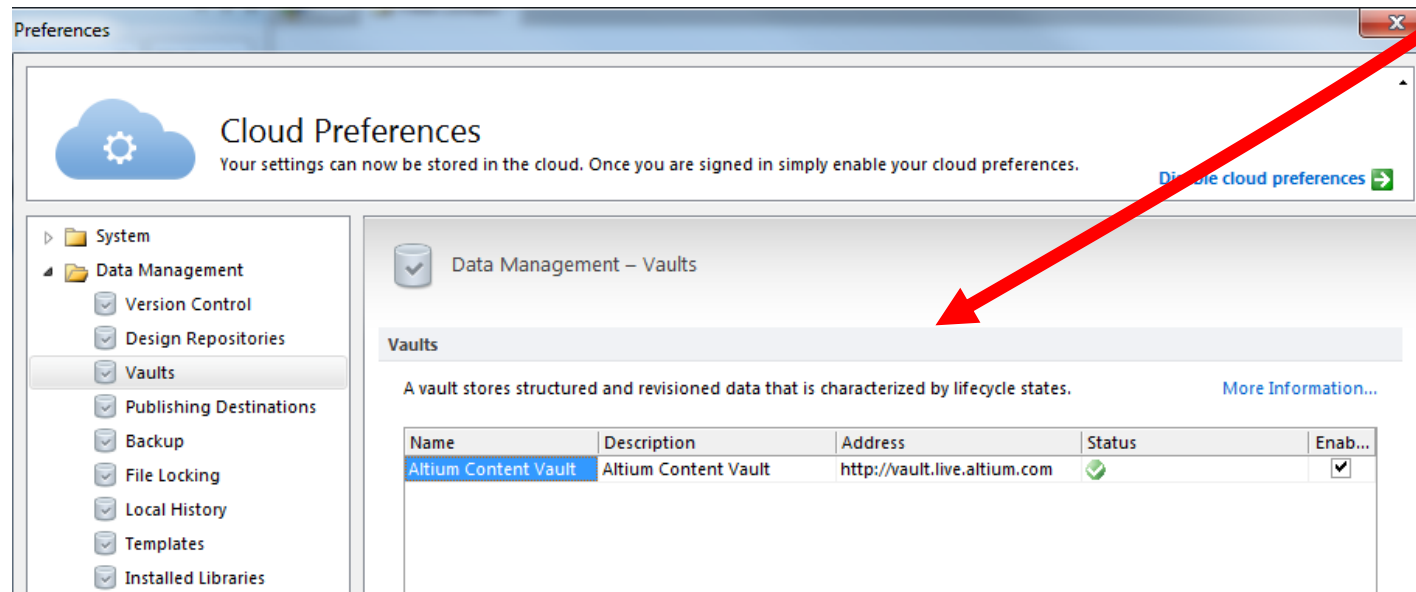
Content Catalog Configuration



You are not connected to any Vault Server.

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

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



Preferences

Cloud Preferences

Your settings can now be stored in the cloud. Once you are signed in simply enable your cloud preferences. [Disable cloud preferences](#)

- System
 - Data Management
 - Version Control
 - Design Repositories
 - Vaults**
 - Publishing Destinations
 - Backup
 - File Locking
 - Local History
 - Templates
 - Installed Libraries

Data Management – Vaults

Vaults

A vault stores structured and revisioned data that is characterized by lifecycle states. [More Information...](#)

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

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

Altium Designer

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

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

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

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

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

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

Panel Access
Workspace panels are accessible using these buttons.

System Information Panel:

View Title	Top Level	Altium Support	
Project Title	PB01 - Audio/Video Peripheral B1	61, 574, 600, 600, 600, 600	
File	AP	Device	BT
Project	D-01-001	Version	01
Project Path	C:\Program Files\Altium Designer\Projects\PB01 - Audio/Video Peripheral B1	Project Path	C:\Program Files\Altium Designer\Projects\PB01 - Audio/Video Peripheral B1

Altium

Design Compiler | System | SD1 | Help | Soft Devices >>>

Recommended basic panels

The screenshot shows the Altium Designer (14.3) interface. The main workspace is labeled "Workspace". Three panels are highlighted with arrows and labels:

- Projects:** Located on the left side, showing a tree view of the project structure.
- Libraries:** Located on the right side, displaying a list of components. The "Vault-resistors" library is selected, showing a table of components with columns for Component Name and Description. A resistor symbol is visible above the table.
- Messages:** Located at the bottom of the interface, showing a table with columns for Class, Document, Source, Message, Time, Date, and No.

The Libraries panel also shows a table with the following data:

Model Name	Model Type	Model Type Name
RESC0201_06i Footprint		PCBLIB
RESC0201_06i Footprint		PCBLIB

Below the table, the price and stock information is displayed:

0.26 USD (each)	41,672 (in stock)
Quantity	Price

Understanding Altium

(Basics for the single user)



Don't forget:

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

Understanding Altium

(Basics for the single user)

- Projects (project panel, active project)
- Workspace Panels (system-wide, editor-specific)
- Editors:
 - Schematic
 - Symbol editor
 - PCB layout
 - Footprint editor
 - CAM files (CAMtastic panel)
- Components and Libraries

Altium Projects

- Project: collection of design documents
 - 1 Project = 1 implementation
 - It stores links to all source documents
 - relative reference: same drive
 - absolute reference: different drive
 - It creates links to all output documents
 - Saves project options
- Create a PCB_Project, Save as: new name
(does not move the file creates a copy)
- The active project is highlighted
- Add/Remove documents to/from a project

Altium Projects: types

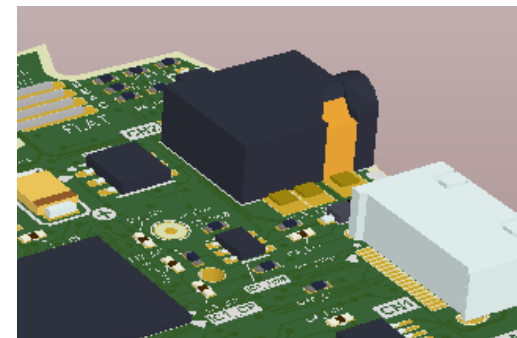
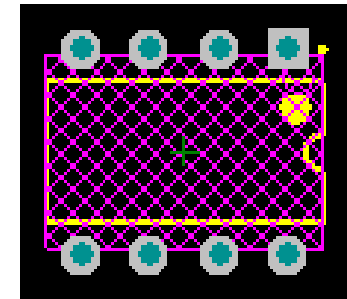
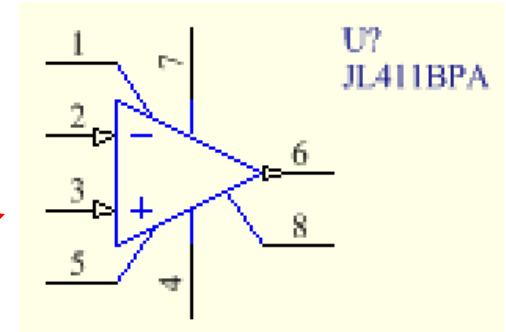
- PCB Project (*.PrjPcb)
 - Schematic, libraries, PCB layout
- FPGA Project (*.PrjFpg)
- Embedded Project (*.PrjEmb)
- Core Project (*.PrjCor)
- Integrated Library (*.LibPkg) & (*.IntLib)
- Script Project (*.PrjScr)

Component, Model and Library Concepts

- 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

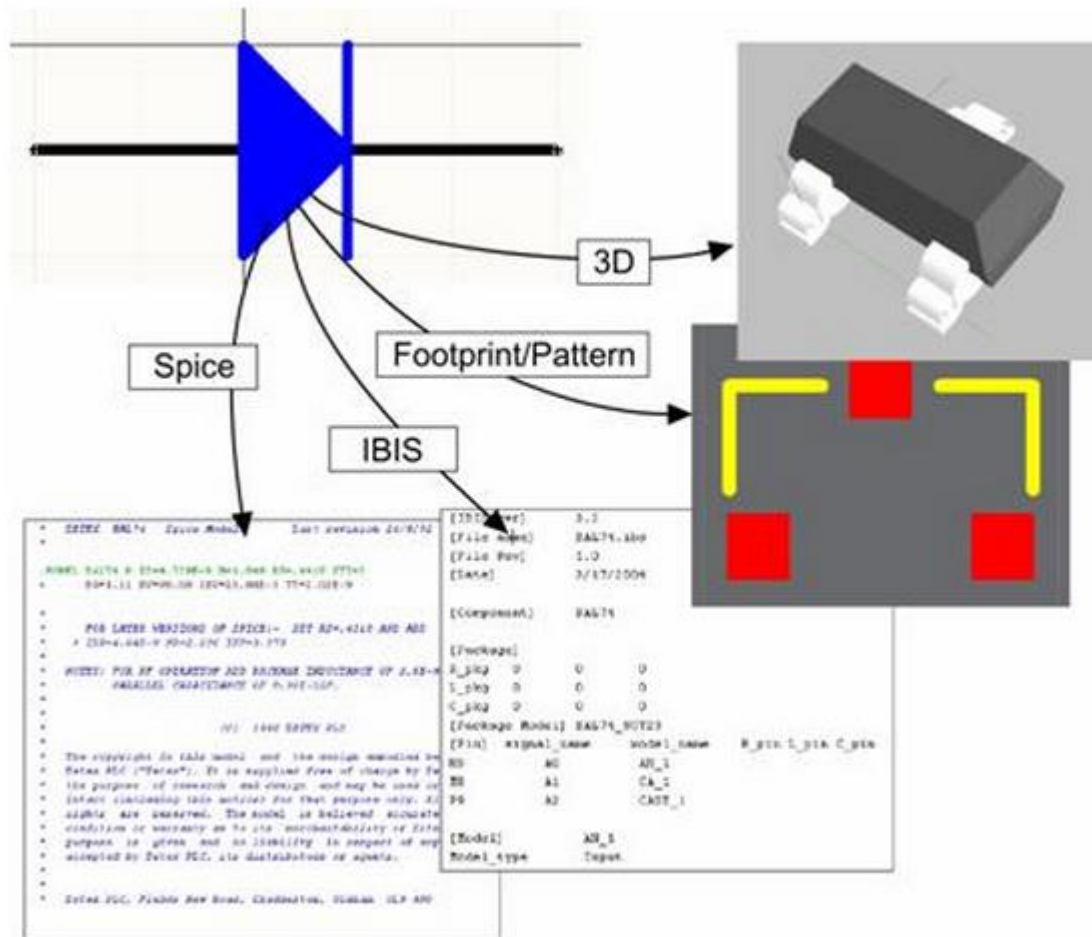
Component, Model and Library Concepts

- 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

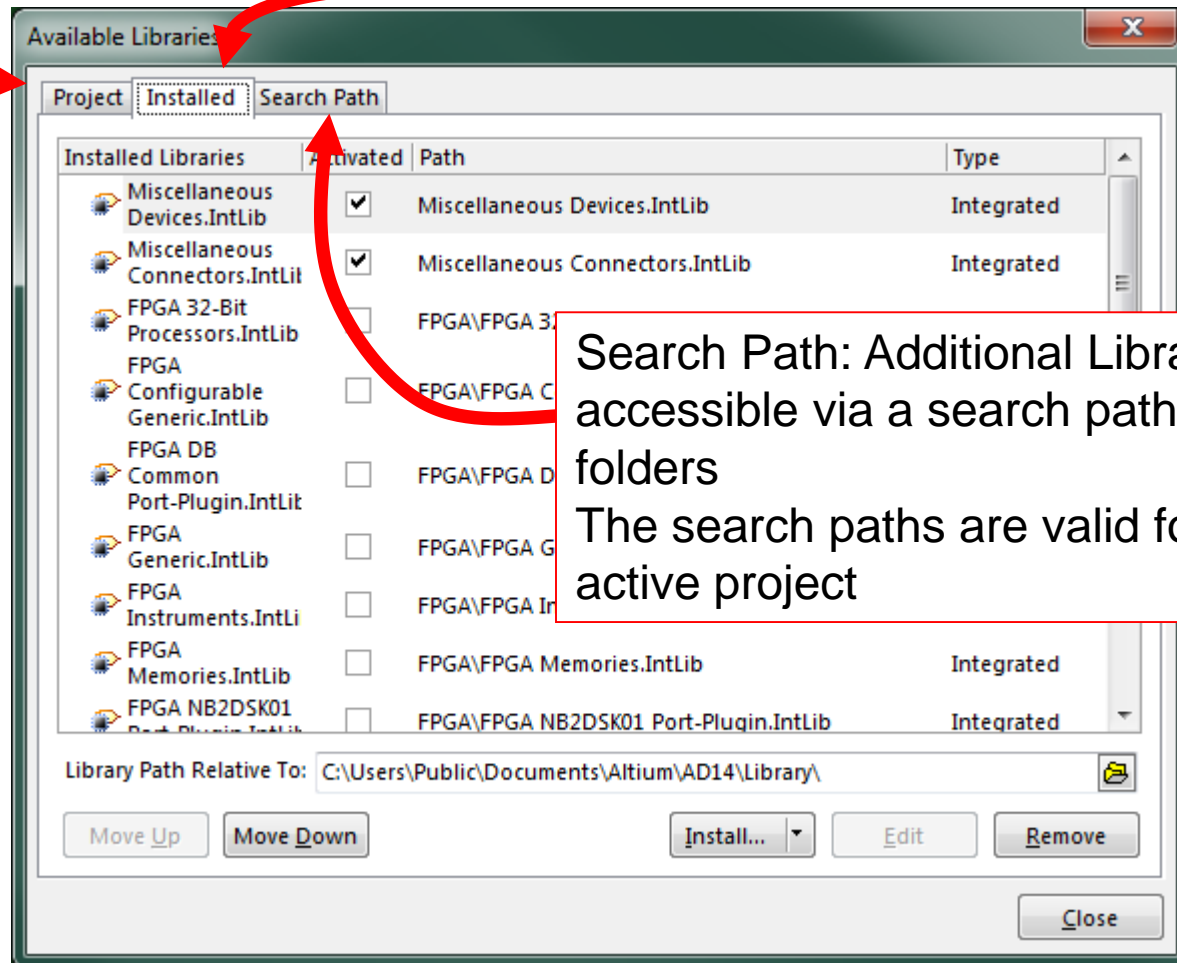


Libraries = collection of components

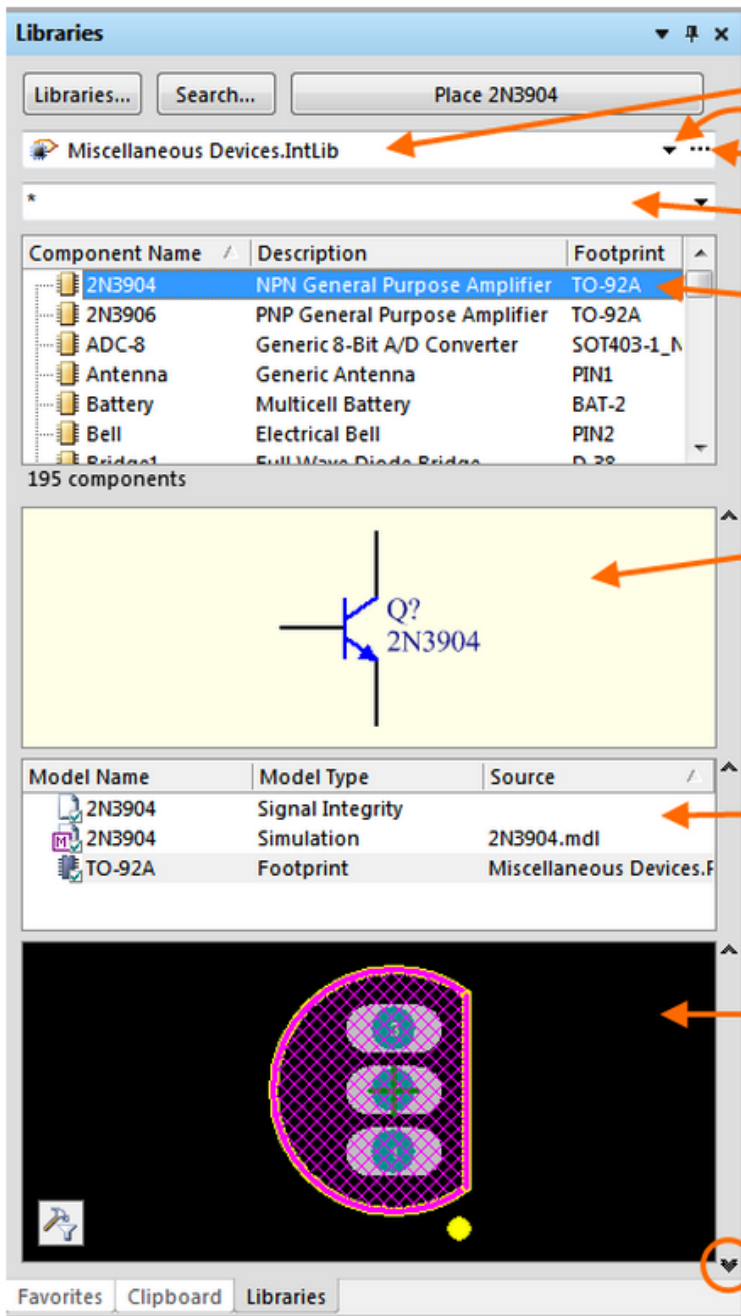
- Collection of components, models or both
- Model Libraries (*.MDL, *.CKT, *.PCBLib)
 - Simulation models are one file per model
- Schematic Libraries (*.SchLib)
 - Symbol and a link to a model library
- Integrated Libraries (*.IntLib)
 - Symbol, footprint and other models are compiled into a single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



- Current library
- Select a different library
- Set the browse modes
- Filter the current library

List of components, select the component of interest

Schematic symbol for the 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 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.

GO TO VAULT

DOWNLOAD LIBRARY

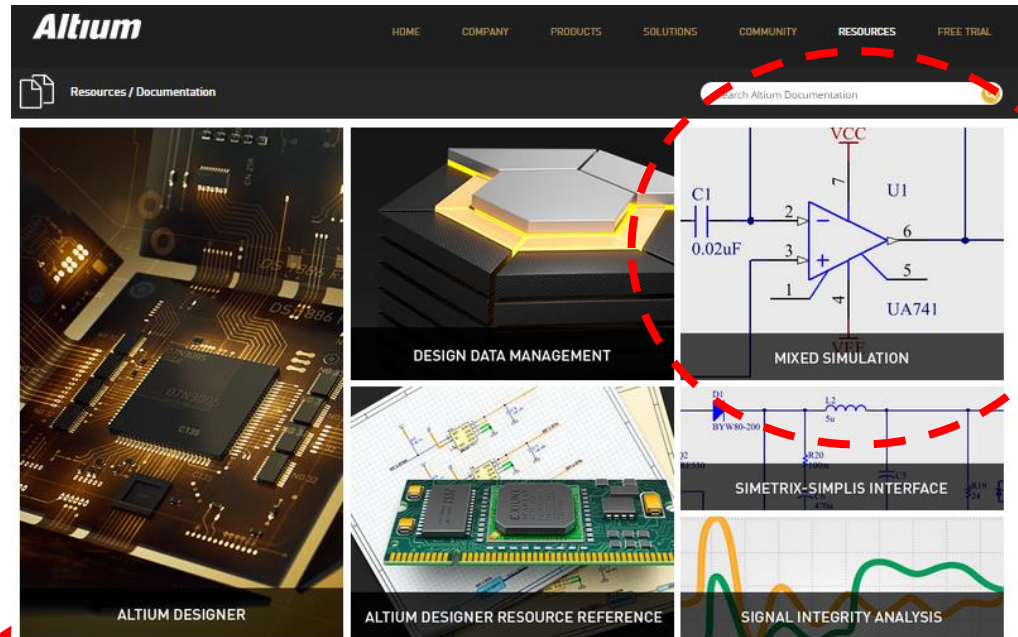
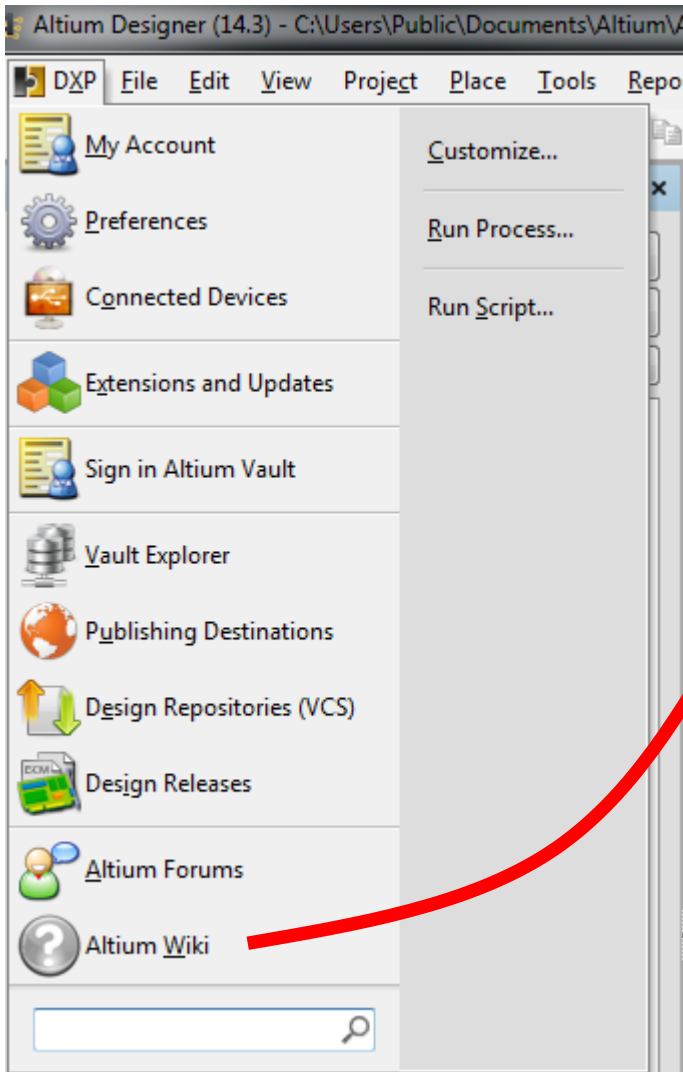
This is useful to preview component

This downloads a .zip file for the complete library

Altium Vault

- Altium is a unified development environment →
Philosophy: Design for reuse
- Vault is a cloud repository of models, components, schematics, design modules etc.
- “Vault-driven” electronic design: release to and source from Vault
- Vault-based components not only include all models, but also include real-time supply chain information.

Learning how to use Altium

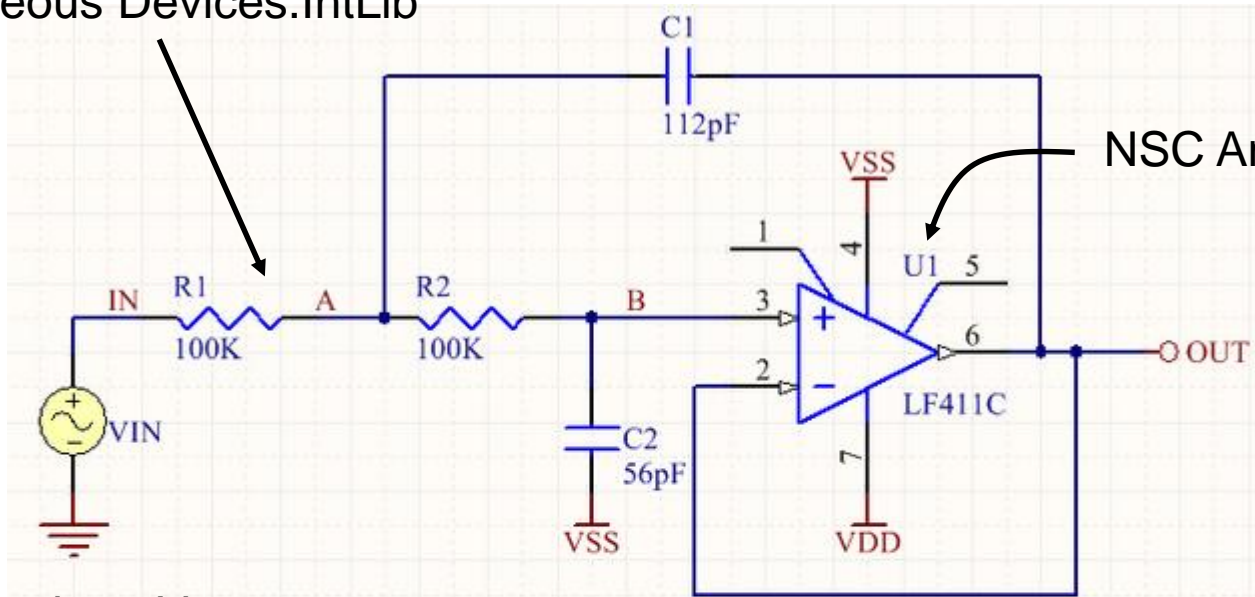


Until recently: best training guides were for Altium 2009
(pdf lesson files organized in chapters)
But DXP menus have changed since
The same information is now updated in the Altium_wiki

Demo: Schematic entry and Simulation

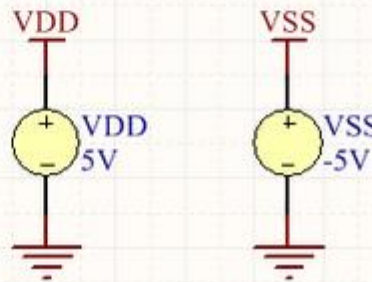
<http://techdocs.altium.com/display/AMSE/Defining+&+Running+Circuit+Simulation+Analyses>

Miscellaneous Devices.IntLib

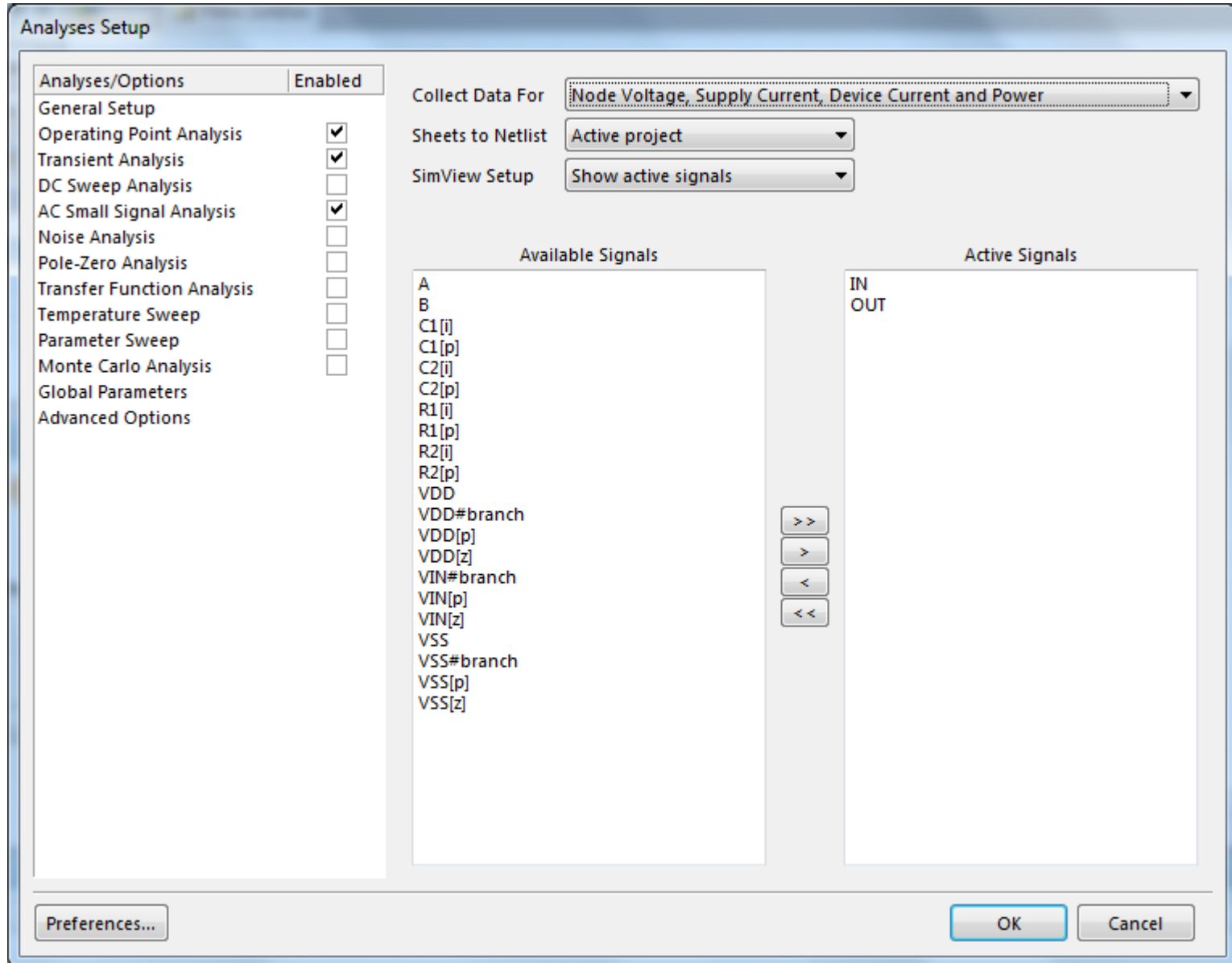


NSC Amplifier.IntLib

Amplitude =5V
Frequency=50KHz



Set simulation parameters



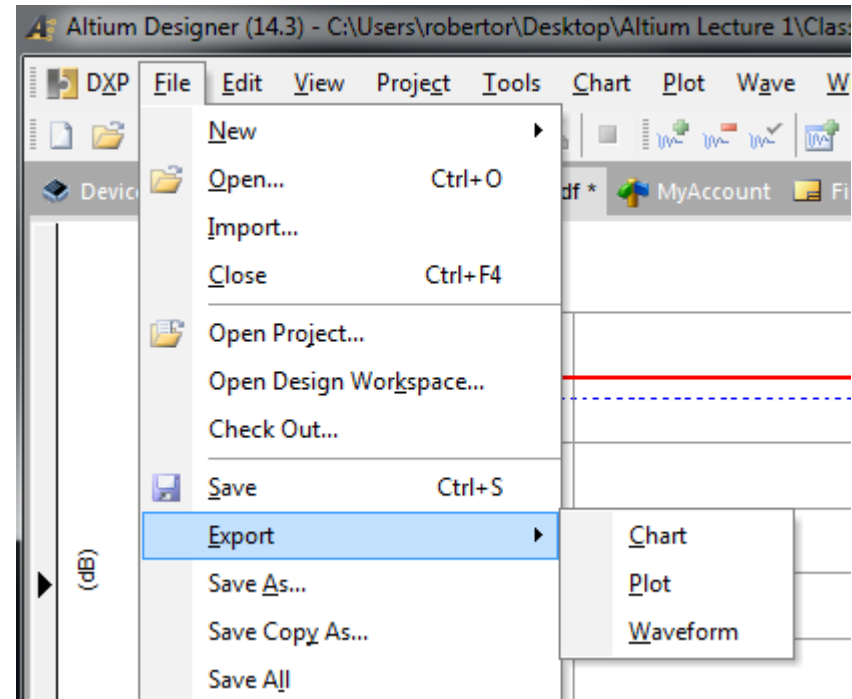
Wiring Tips

- **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**.

How to save results

- How to export a file
- Plugin for printing (Altium Live account)

SIMVIEW



Category: **Output Generators**
Version: 10.1531.27391
Updated: Jan 31, 2013

This plugin adds support for printing simulation data obtained by running a mixed-signal simulation on a circuit, or a pre-/post-layout signal integrity analysis, and stored in a simulation data file. This includes **Page Setup**, **Print** and **Print Preview** via the SimData Editor's main **File** menu, and the ability to add a SimView Print output in the

About SPICE

- **U.S DOD, CANCER**
Computer Analysis of Nonlinear Circuits Excluding Radiation
- **Berkley**, Simulation Program with Integrated Circuit Emphasis
 - SPICE 1972 FORTRAN
 - SPICE 2 1975, SPICE 2G.6 1983
 - SPICE 3 1989 C, **SPICE 3F5** 1993
 - SPICE 4 2004 (RF)
- **Proprietary versions of SPICE**
SPICE-like simulators or “Alphabet SPICE”
HSpice, **XSPICE** (Georgia Tech), **PSPICE**, etc

Altium and SPICE

- Altium Designer is compatible with:
 - SPICE3f5 (Berkley SPICE)
 - XSPICE (Georgia Tech)
 - PSPICE (Micro/Sim/Orcad/Cadence)
- You may need to change the file extension to .mdl or .ckt

```
.MODEL Diode D
+(
+ AF=1.0 Bv=5.2 CJO=0.0 EG=1.11 FC=0.5 Ibv1=0.2 Ibv=5 Ikf=10 IS=1E-14
+ Isr=1.8n KF=0.0 M=0.5 N=1.0 Nbv=3.1779 NBVL=1.0 Nr=1.5 Rs=.5875
+ TBV1=0.0 TBV2=0.0 TIKF=0.0 TRS1=0.0 TRS2=0.0 Vj=.75 XTI=3.0
+)
```

SUBCKT / .ENDS

- Other models need to be manually converted!

SPICE Models and Subcircuits

SubCircuit

```
.SUBCKT LF411/NS 1 2 99 50 28
*
*****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
* etc,etc...
* Code truncated to demonstrate concept
* Refer to http://www.national.com/models/spice/LF/LF411.MOD
* For complete .ckt file of the LF411/NS model
***** LOCAL MODELS USED*****
*
.MODEL JX PNF(BETA=1.183E-3 VTO=-.65 IS=50E-12)
*
*Note that Model JX is referenced in the .SUBCKT
*by the J2 device.
.ENDS LF411/NS
```

SPICE Netlist

- Subcircuits, models + analysis command + graphical output settings

```
*SPICE Netlist generated by Advanced Sim server
```

```
Cload 0 LLTRA_OUT 10pF
TLLTR1 LLTRA_IN 0 LLTRA_OUT 0 Z0=75 TD=19.6ns
Rload 0 LLTRA_OUT 75
Rs LLTRA_IN VS 5
Vinput VS 0 DC 0vdc PWL(0U 0V 10ns 2V 300ns 2V) AC 1vacm 0

.SAVE 0 LLTRA_IN LLTRA_OUT VS Vinput#branch @Vinput[z] @Cload[i] @Rload[i] @Rs[i]
.SAVE @Cload[p] @Rload[p] @Rs[p] @TLLTR1[p] @Vinput[p]

*PLOT TRAN -1 1 A=LLTRA_IN
*PLOT OP -1 1 A=LLTRA_IN

*Selected Circuit Analyses:
.TRAN 1.2E-9 3E-7 0 1.2E-9
.OP

.END
```

Asterisks (*) = Comments, Plus (+) = Line continuation, Period (.) = Command
Letters (A to Z) are used to represent elements, D= Diode, R = Resistor etc.




SPICE Syntax Reference (1/2)

Letter	Device	Syntax
A	Xspice / SimCode	Digital SimCode models
B	Non-Linear Dependent Voltage Source	B<refdes> <+node> <-node> V=<EQUATION> EQUATION denotes the expression defining the source waveform
C	Capacitor	C<refdes> <+node> <-node> [<model>] <value> [IC=<initial voltage>]
D	Diode	D<refdes> <+node> <-node> <model> [AREA] [IC=<initial voltage>] [TEMP=<temperature>]
I	Current Source	I<refdes> <+node> <-node> [[DC] <value>] [AC <magnitude> + [<phase>]]
J	Junction FET	J<refdes> <drain> <gate> <source> <model> [area] [initial on/off starting condition] [IC=initial D-S voltage, initial G-S voltage]
K	Inductor Coupling	K<refdes> L<name1> < L<name2> > <coupling>
L	Inductor	L<refdes> <+node> <-node> [model] <value> [IC=<initial current>]

SPICE Syntax Reference (2/2)

Letter	Device	Syntax
M	Mosfet	M<refdes> <drain> <gate> <source> <substrate> <model> + [L=<value>] [W=<value>] + [AD=<drain area value>] [AS=<source area value>] + [PD=<drain perimeter value>] [PS=<source perimeter value>] + [NRD=<value>] [NRS=<value>] + [IC=<initial D-S volt.>, <initial G-S volt.>, <initial B-S volt.>] + [TEMP=<temperature>]
Q	Bipolar Transistor	Q<refdes> <collector> <base> <emitter> <model> [<area>] + [IC=<initial B-E voltage>, <initial C-E voltage>] + [TEMP=<temperature>]
R	Resistor	R<refdes> <+node> <-node> [<model>] <value>
S	Voltage controlled switch	S<refdes> <+node> <-node> <+control> + <-control> <model> [initial condition]
T	Transmission Line	T<refdes> <A+> <A-> <B+> <B-> Z0=<value> + [TD=<value> F=<value>[NL=<value>]]
V	Voltage Source	V<refdes> <+node> <-node> [[DC] <value>] + [AC <magnitude> [<phase>]]
X	Sub-circuit call	X<refdes> [<node>]* <sub-circuit name>

SPICE Unit multipliers

Unit Multiplier	Value	Nomenclature	Measurement System
T	10^{12}	Tera	Metric
G	10^9	Giga	Metric
 Meg	10^6	Mega	Metric
K	10^3	Kilo	Metric
 mil	25.4^{-6}	Mils	English
 m	10^{-3}	Milli	Metric
u	10^{-6}	Micro	Metric
n	10^{-9}	Nano	Metric
p	10^{-12}	Pico	Metric
f	10^{-15}	Femto	Metric