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

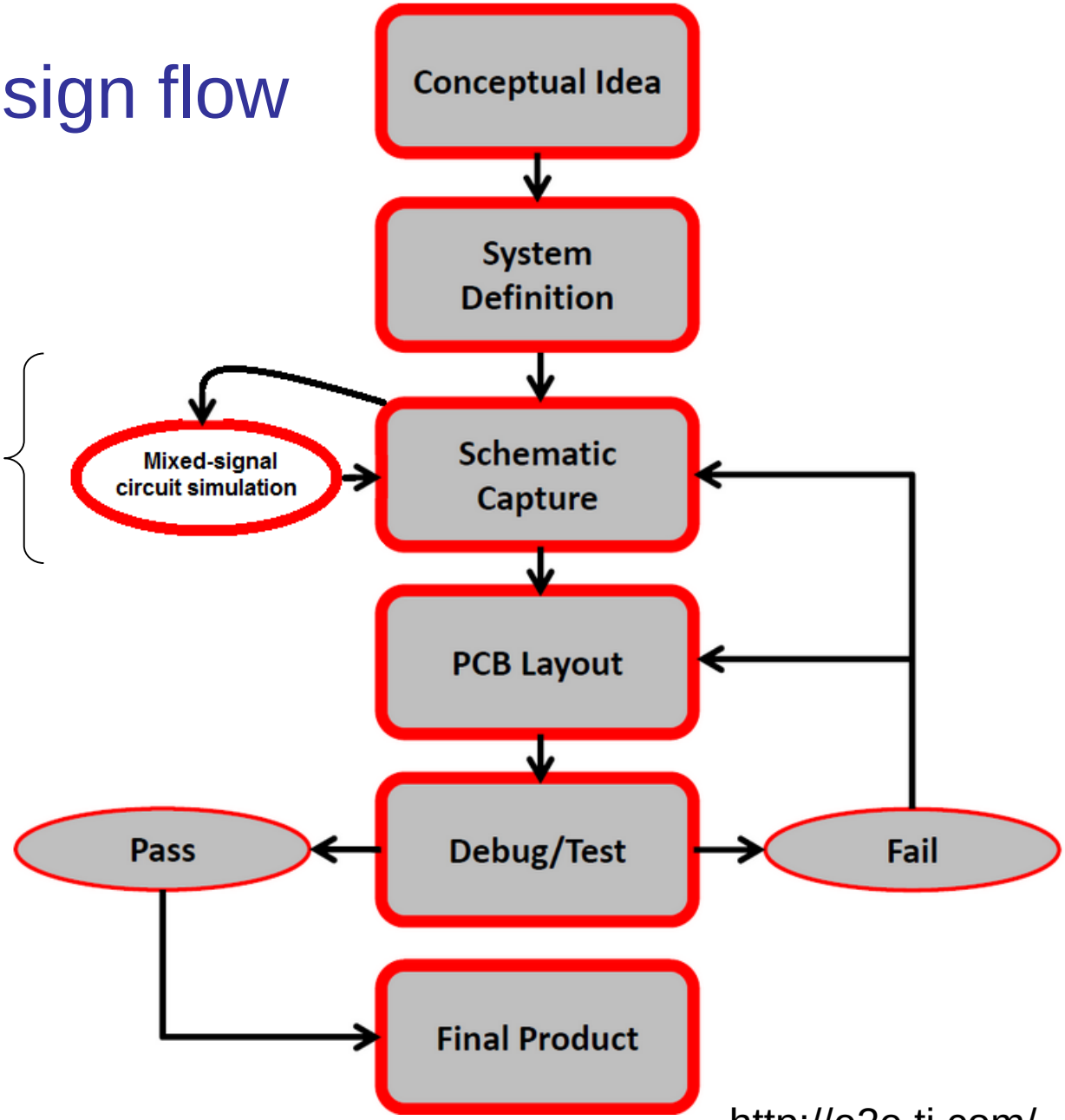
Contents

- 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.

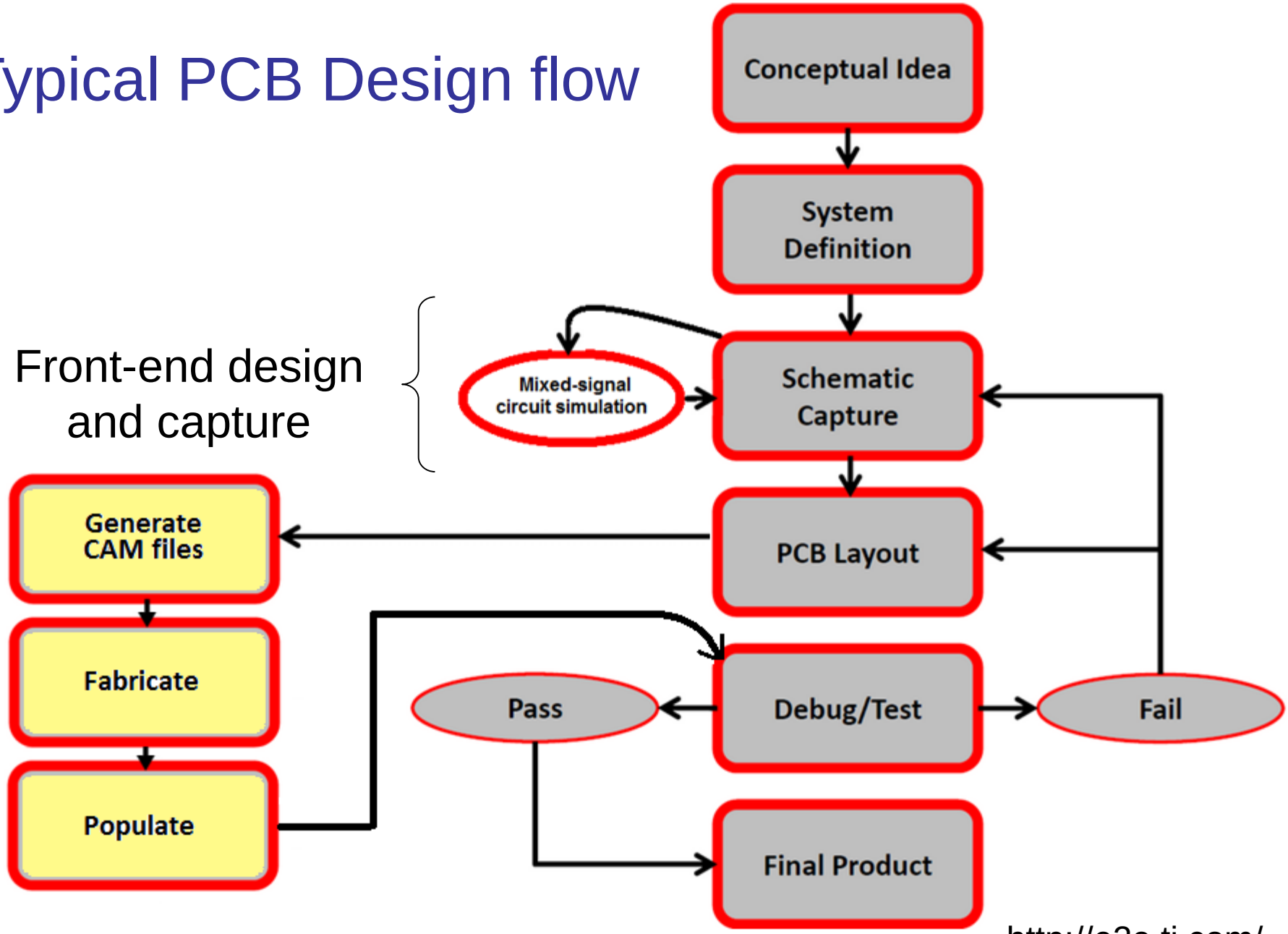
Typical PCB Design flow

Front-end design and capture



http://e2e.ti.com/

Typical PCB Design flow



http://e2e.ti.com/

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

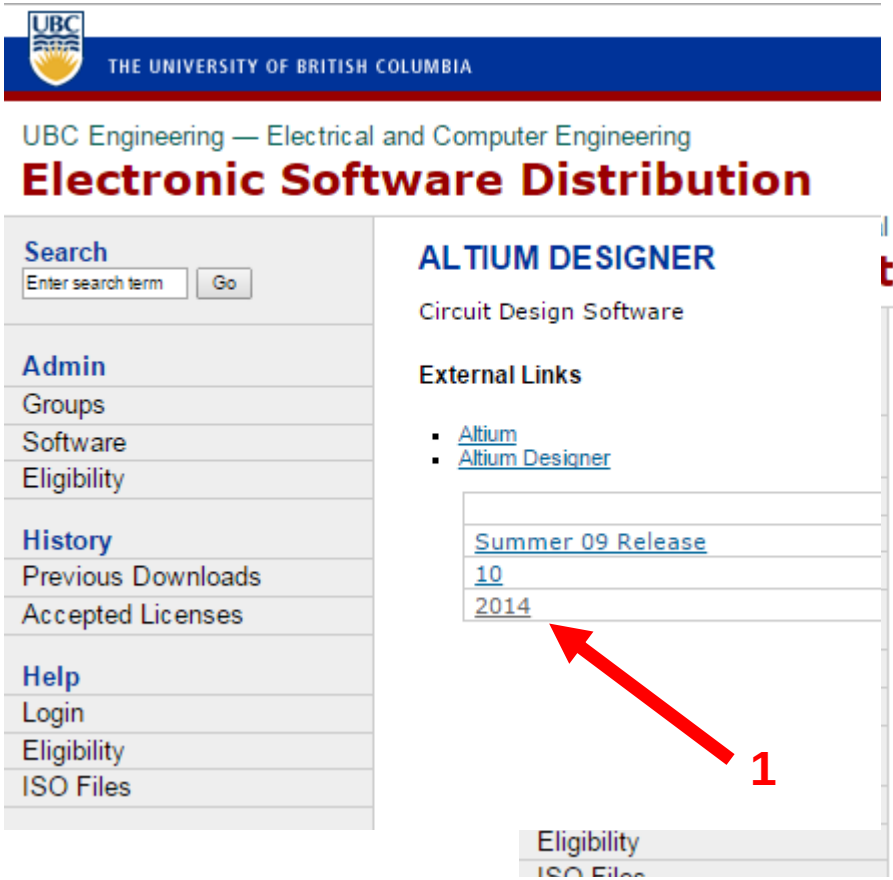
How to install Altium 2014

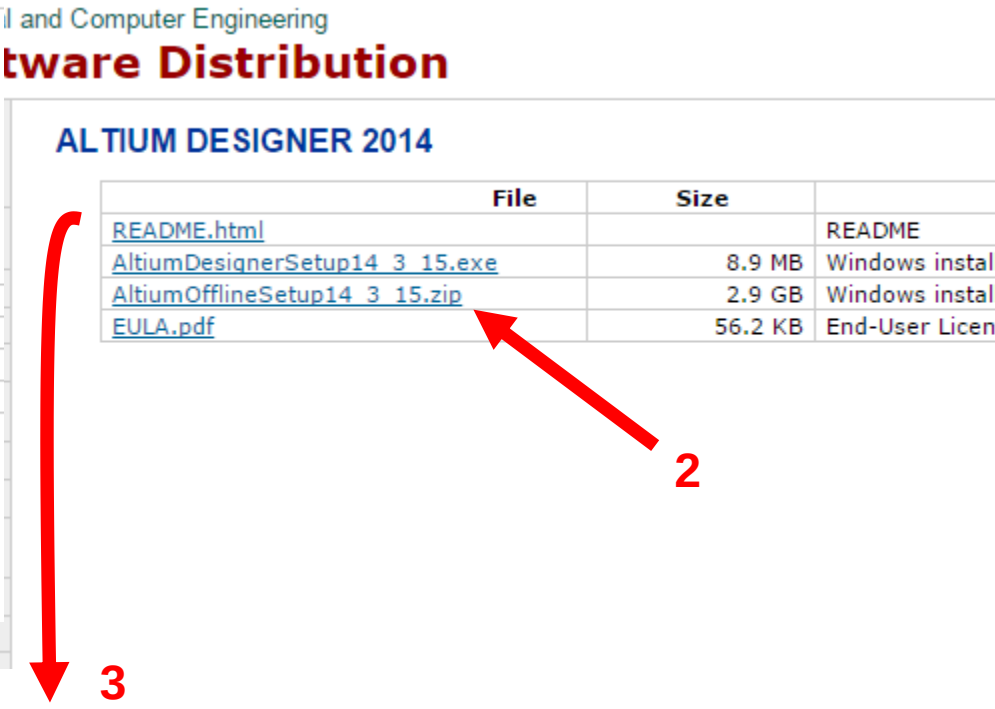
- Link to our download site:
  https://download.ece.ubc.ca/

- 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)

Install 2014v





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:

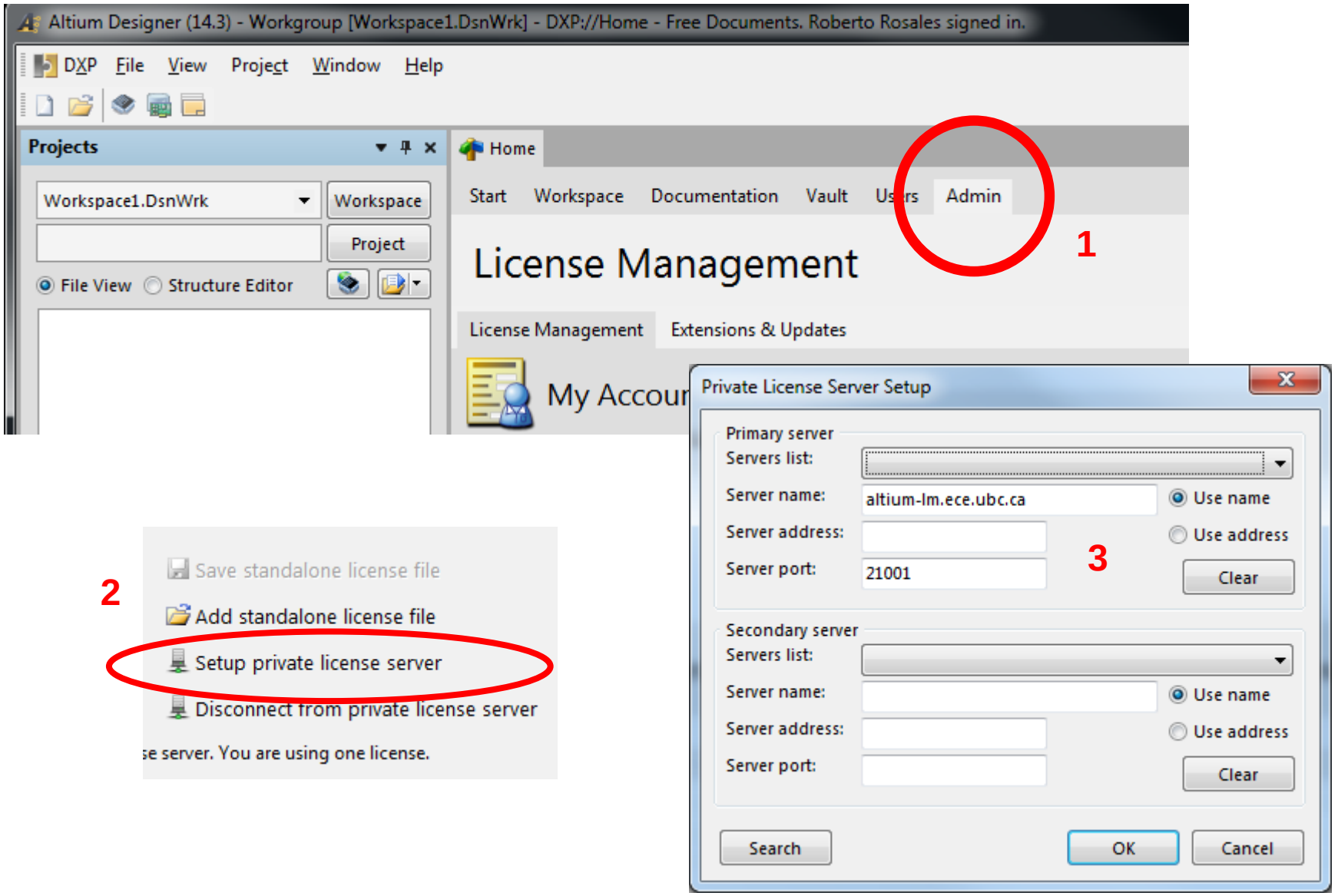
- A wired connection on the ECE network
- A wired connection on UBC ResNet
- A wireless connection at the UBC Vancouver campus on the `ubcprivate`, `ubcsecure`, or `ubc` network (`ubcvisitor` and `eduroam` are not sufficient)
- A myVPN connection to the UBC Vancouver network
- A myVPN connection to the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

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

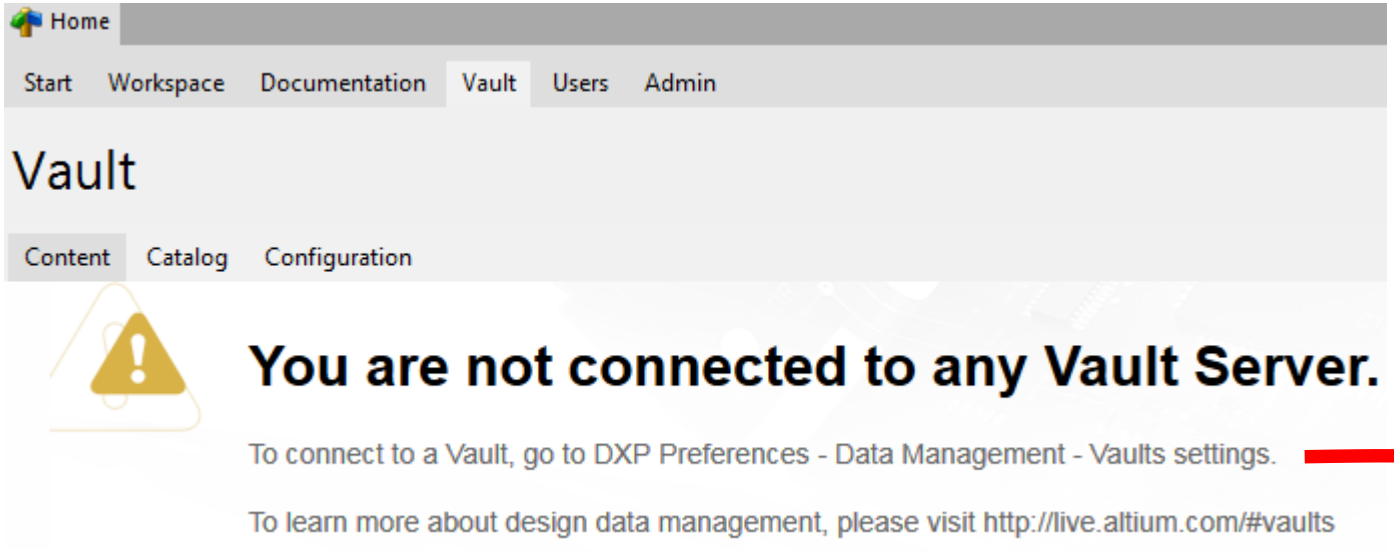
Server name:	`altium-lm.ece.ubc.ca`
Server port:	`21001`

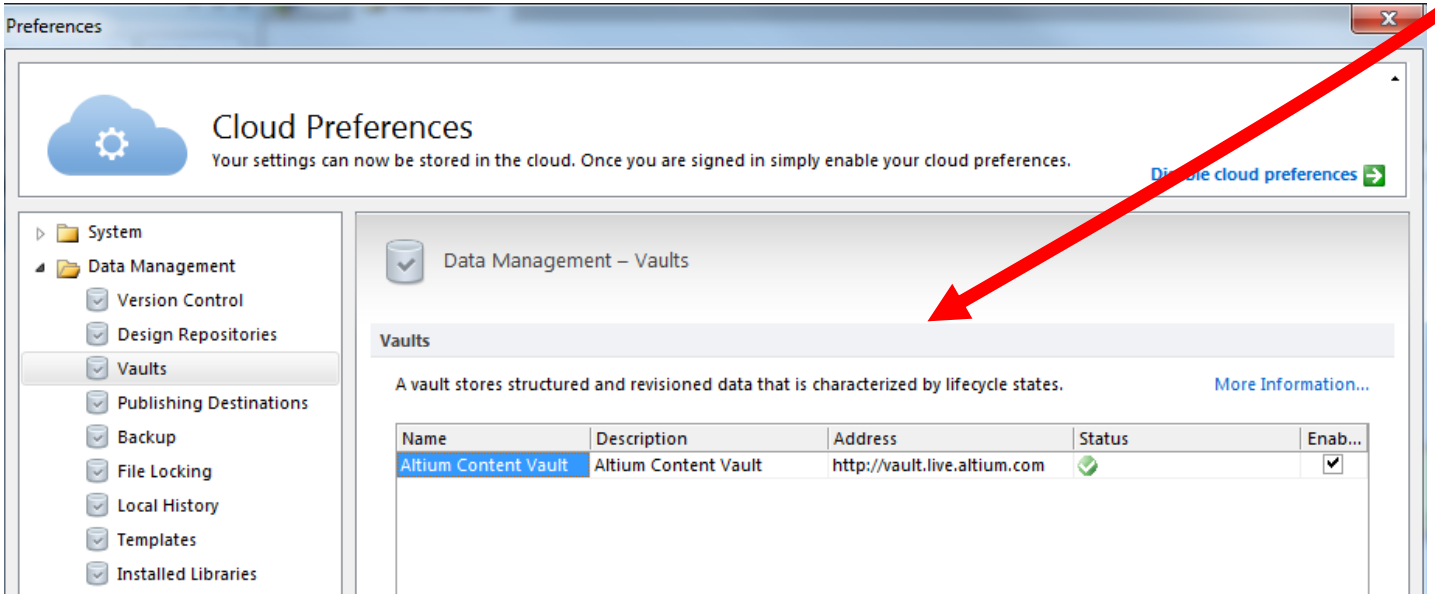
Select the new license that appears and click on "Use". You may as well also delete any old, expired licenses that are also showing.

To set license server



Connecting to the Altium Vault

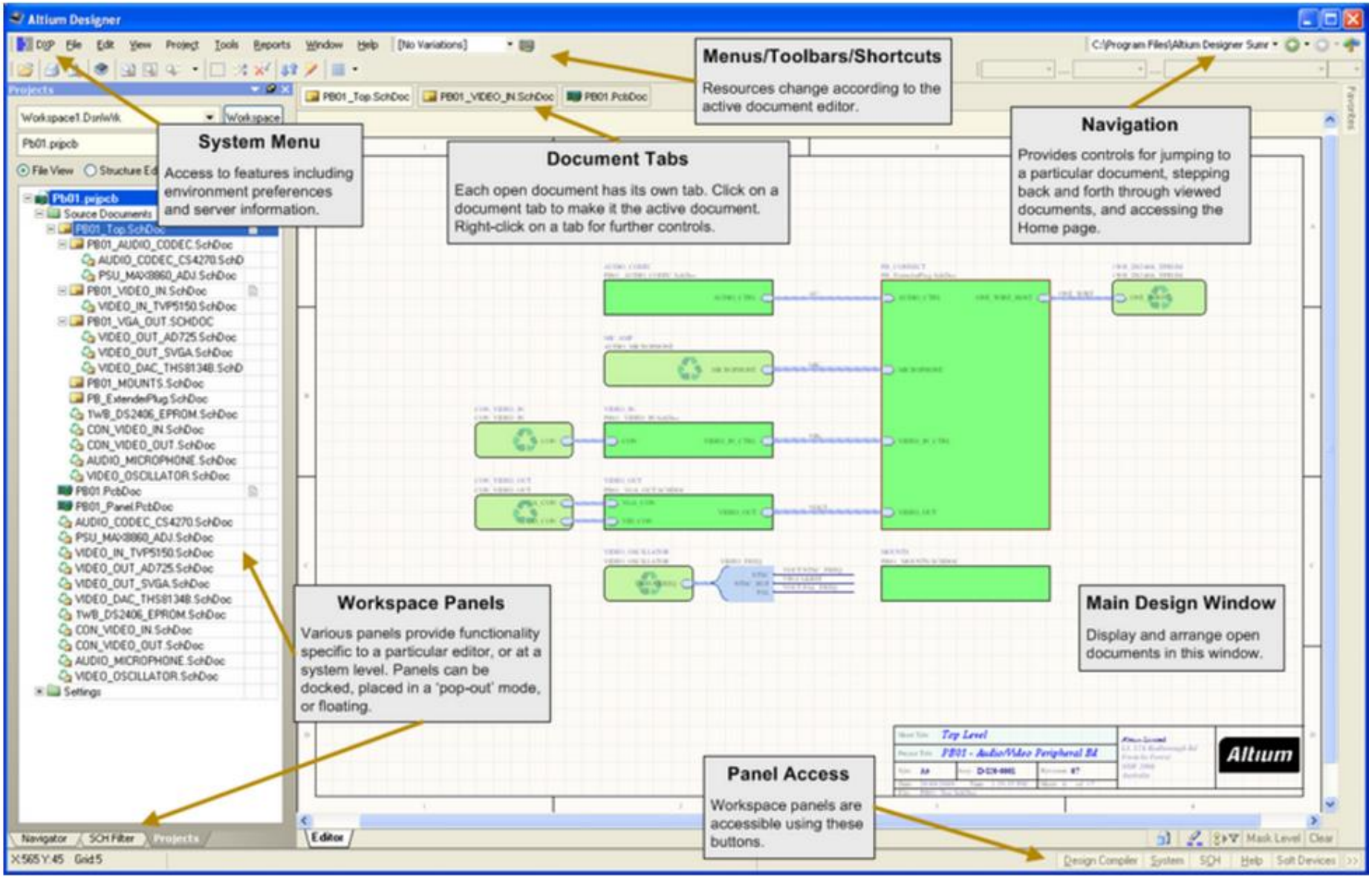




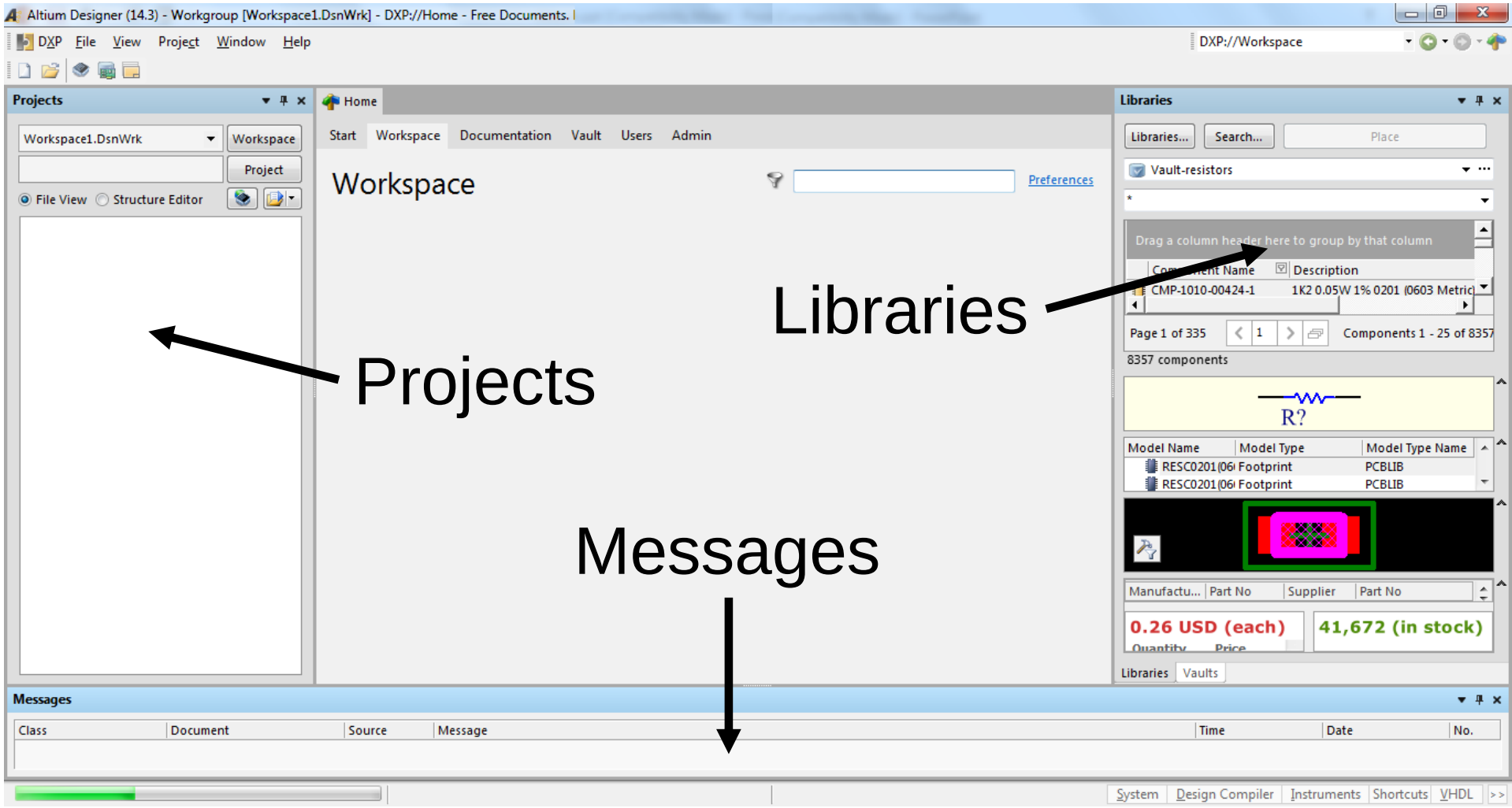
Understanding Altium

- 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

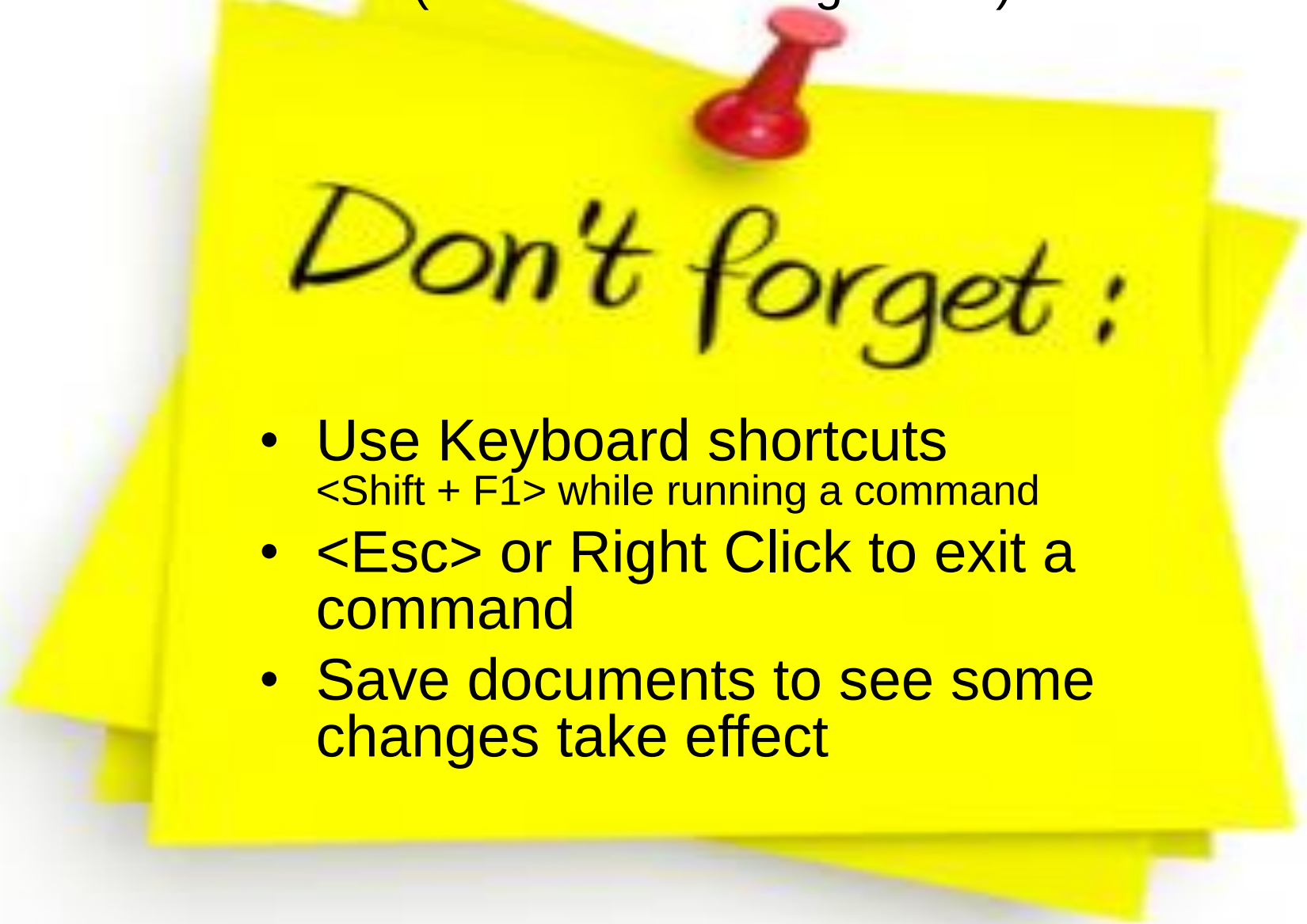


Recommended basic panels



Understanding Altium

(Basics for the single user)



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

Understanding Altium

(Basics for the single user)

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

Altium Projects

- Project: collection of design documents
  - 1 Project = 1 implementation
  - It stores links to all source documents
    - relative reference: same drive
    - absolute reference: different drive
  - It creates links to all output documents
  - Saves project options
- Create a PCB_Project, Save as: new name

(does not move the file creates a copy)

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

Altium Projects: types

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

Component, Model and Library Concepts

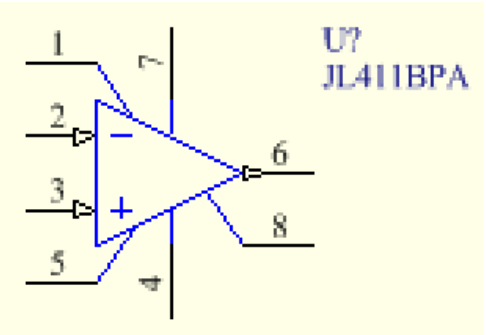
- Domains = Different phases of design
  - Schematic capture
  - PCB layout (2D / 3D)
  - SPICE simulation
  - Signal integrity analysis

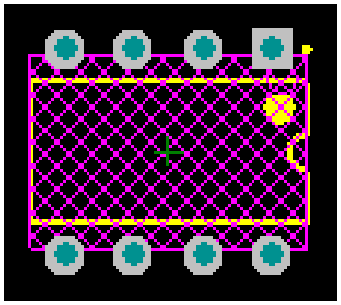
  } Different component representations

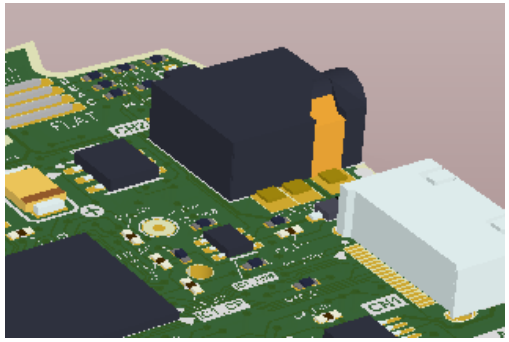
- A unified component is a container with links to all domain models + parametric information

Component, Model and Library Concepts

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

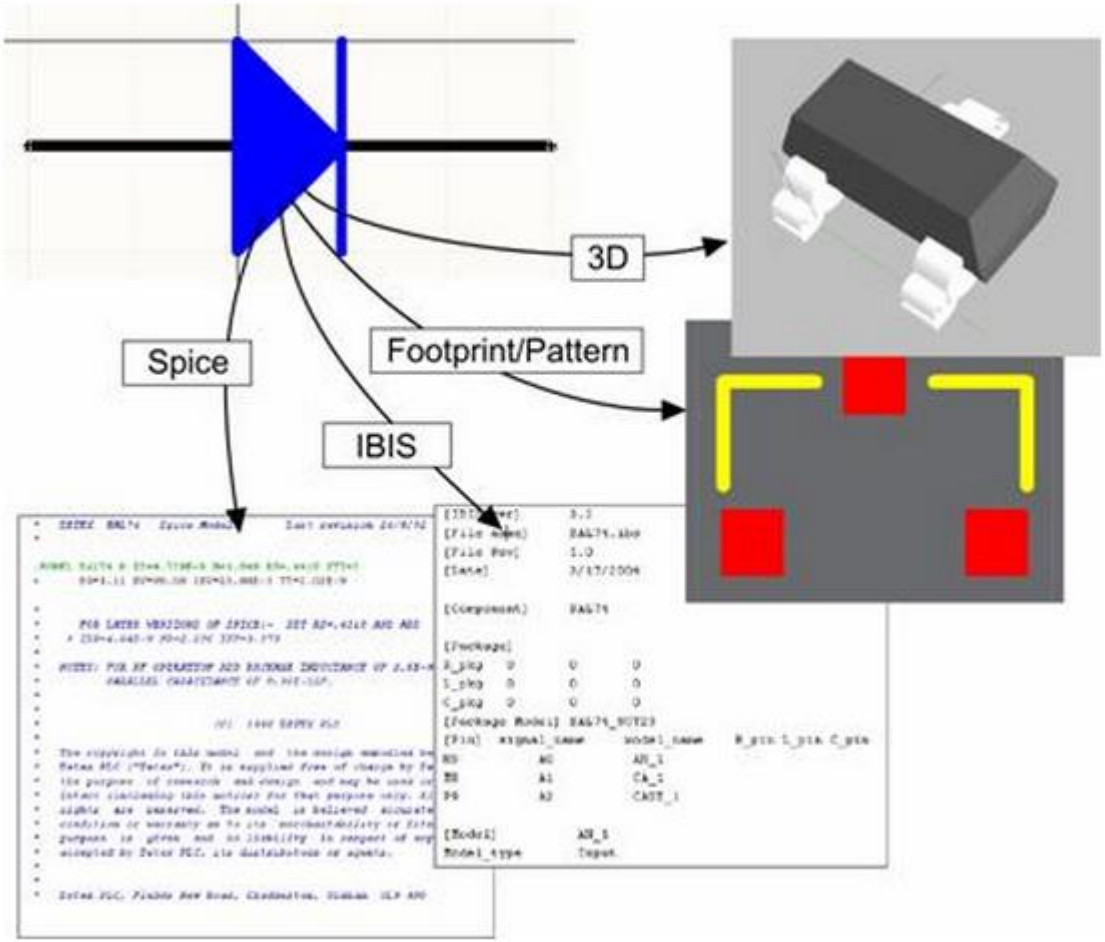






Component, Model and Library Concepts

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

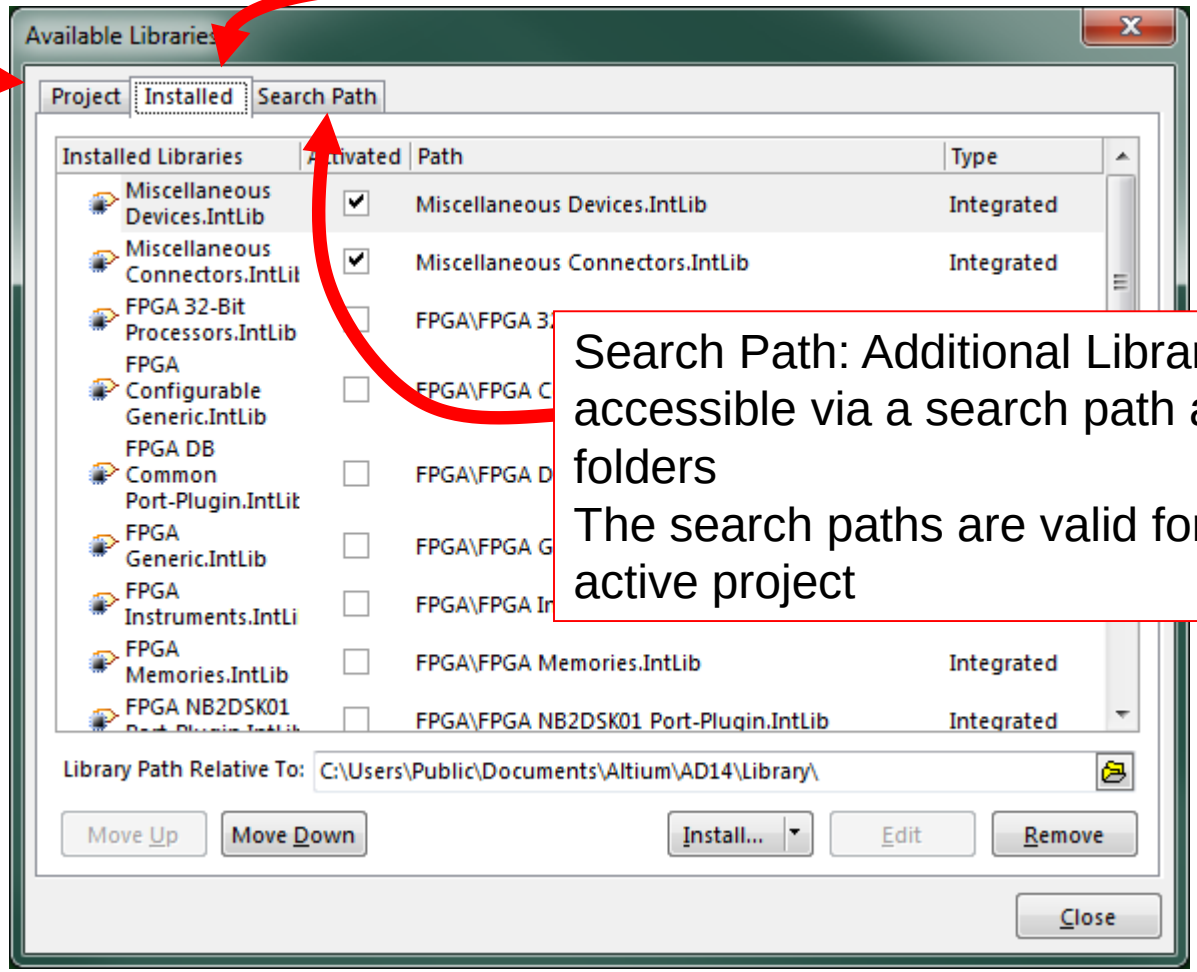


Libraries = collection of components

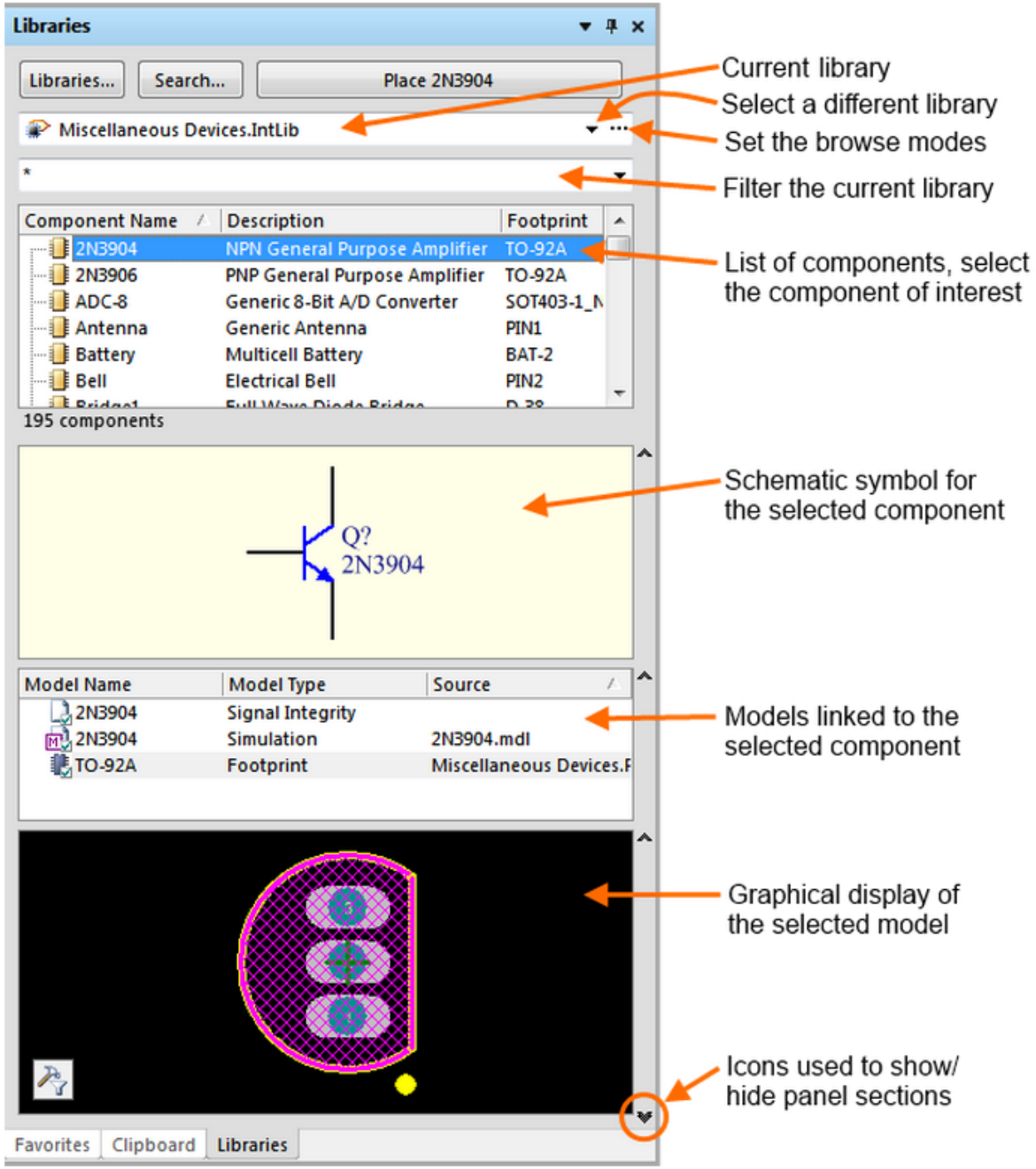
- Collection of components, models or both
- Model Libraries (*.MDL, *.CKT, *.PCBLib)
  - Simulation models are one file per model
- Schematic Libraries (*SchLib)
  - Symbol and a link to a model library
- Integrated Libraries (*.IntLib)
  - Symbol, footprint and other models are compiled into a single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



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

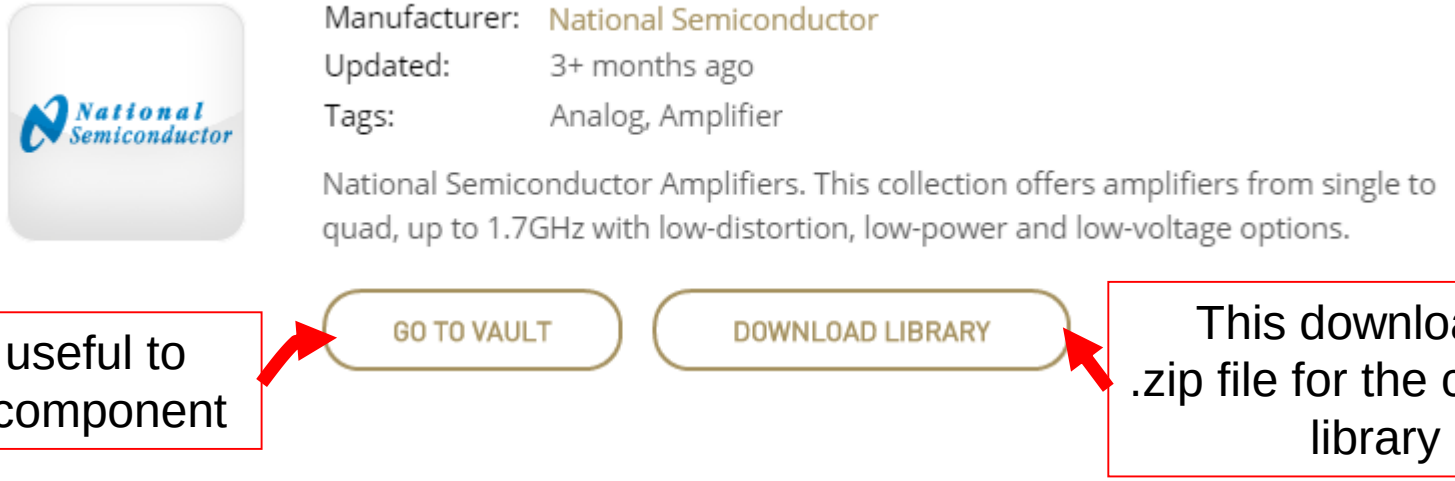
<L> for PCB footprints
to flip component side

To search across libraries:

Search …

Obtaining integrated libraries

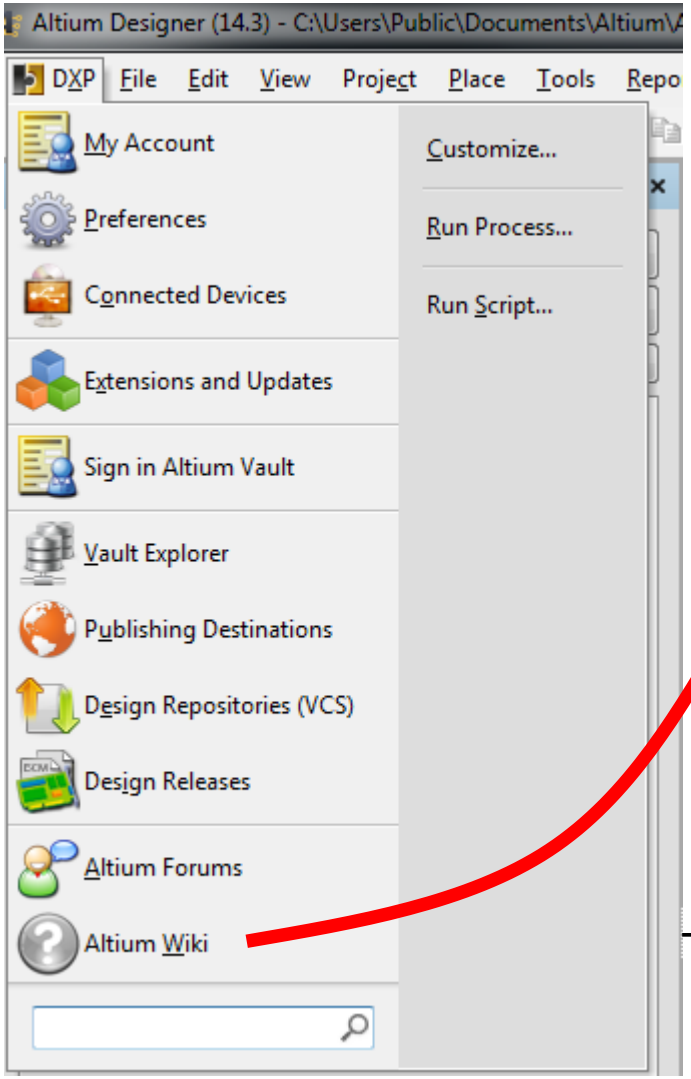
1. Frozen libraries: from here
   you can install anywhere but it is a good idea to make a subfolder under:
   C:\Users\Public\Documents\Altium\AD14\Library
   or a cloud storage service if you use more than one PC
2. AltiumLive website: Resources / Design Content

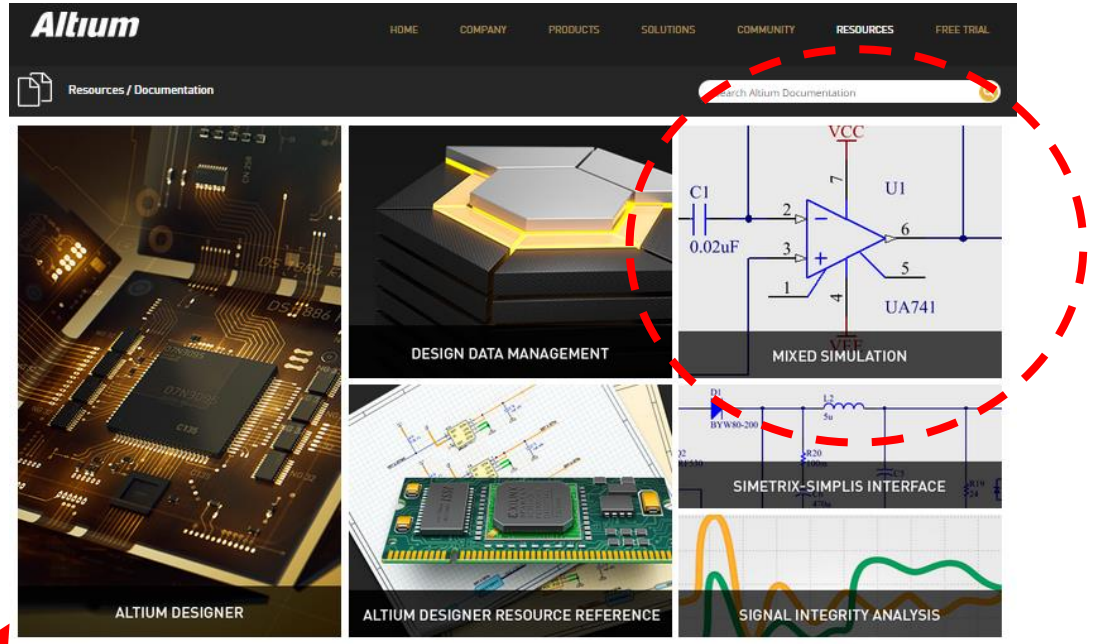


Altium Vault

- 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





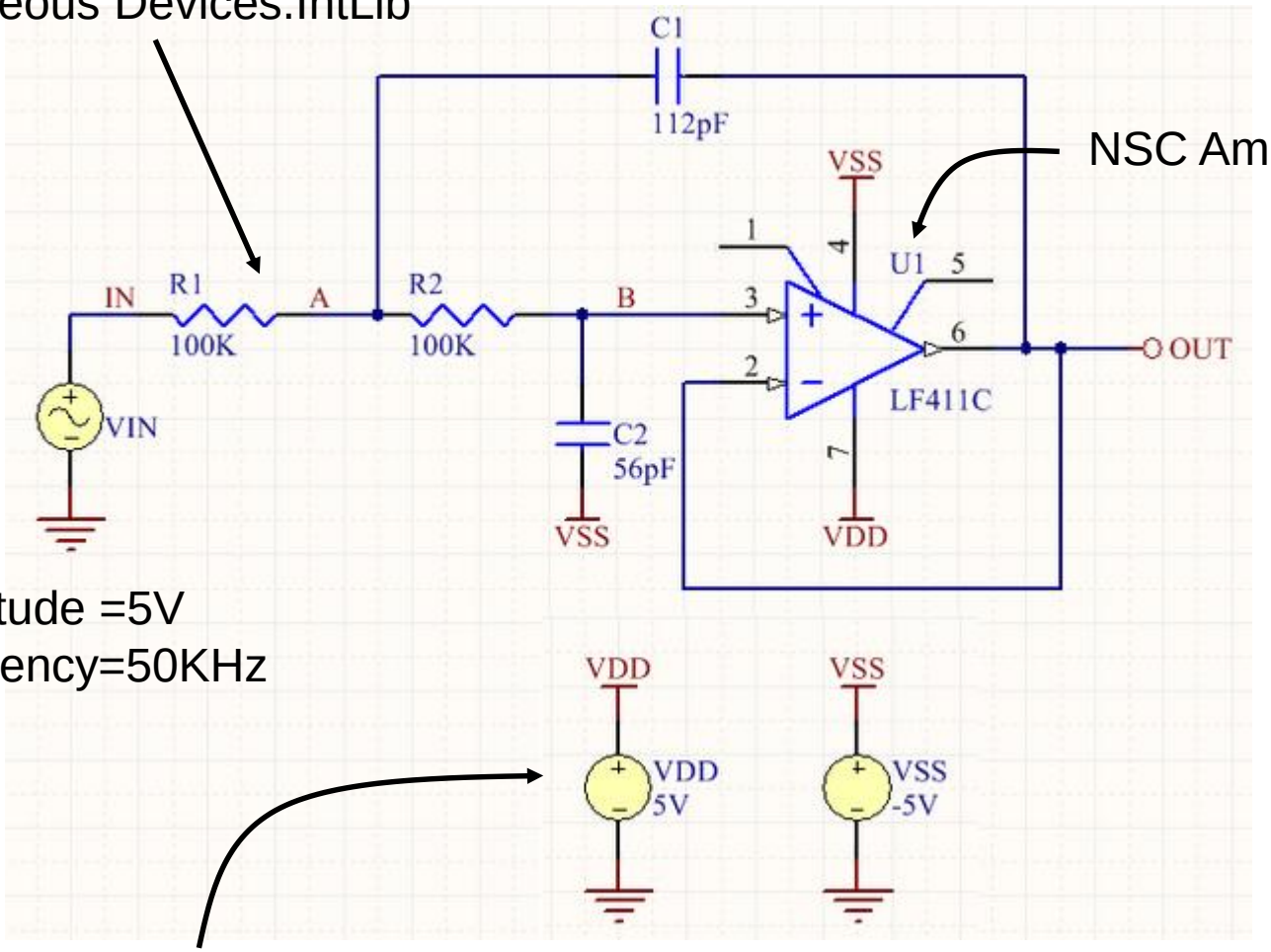
Until recently: best training guides were for Altium 2009
(pdf lesson files organized in chapters)
But DXP menus have changed since
The same information is now updated in the Altium_wiki

Demo: Schematic entry and Simulation

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

Miscellaneous Devices.IntLib

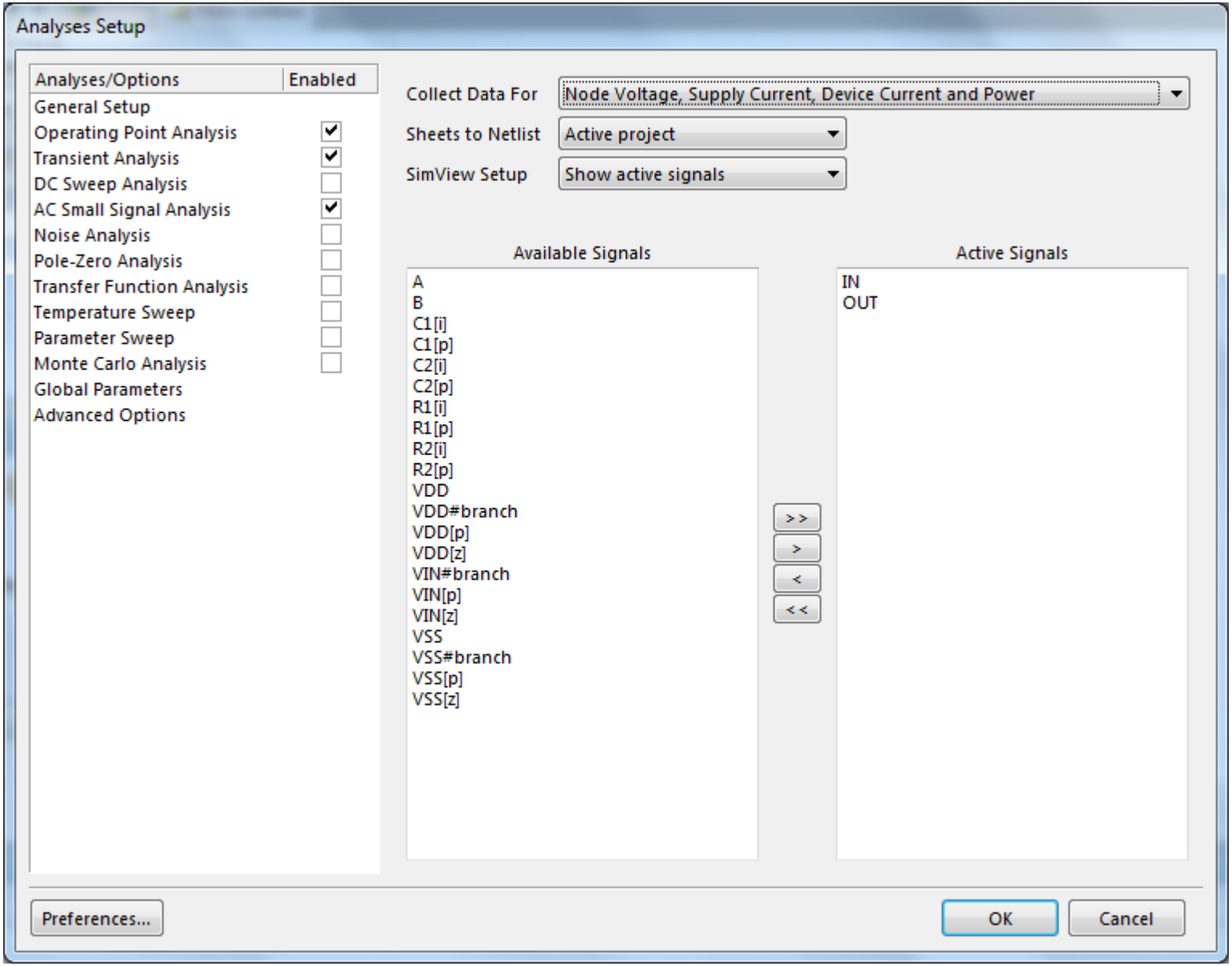
NSC Amplifier.IntLib



Amplitude =5V
Frequency=50KHz

C:\Users\Public\Documents\Altium\AD14\Library\Simulation\Simulation Sources.IntLib

Set simulation parameters



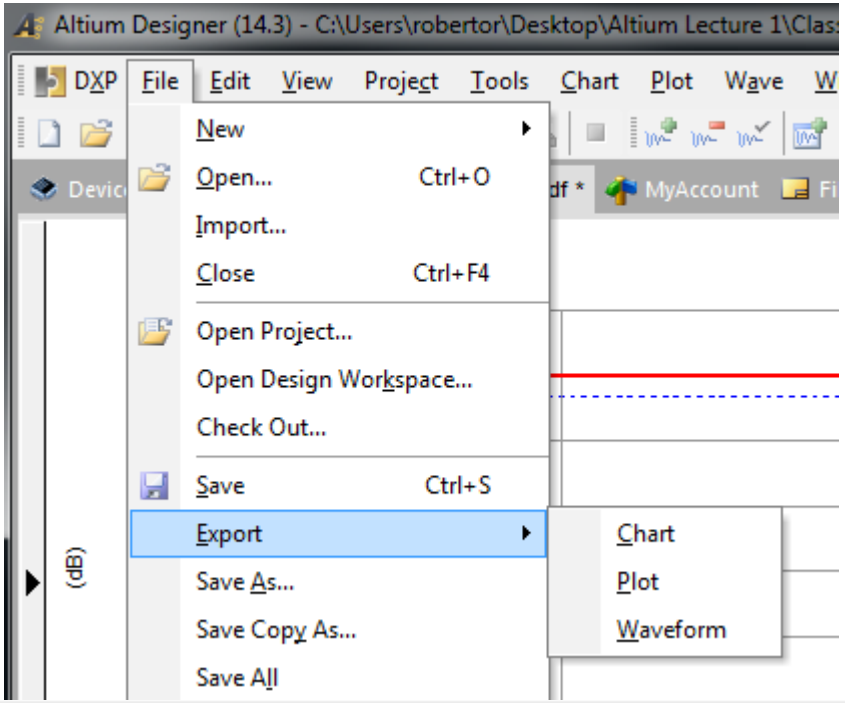
Wiring Tips

- 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

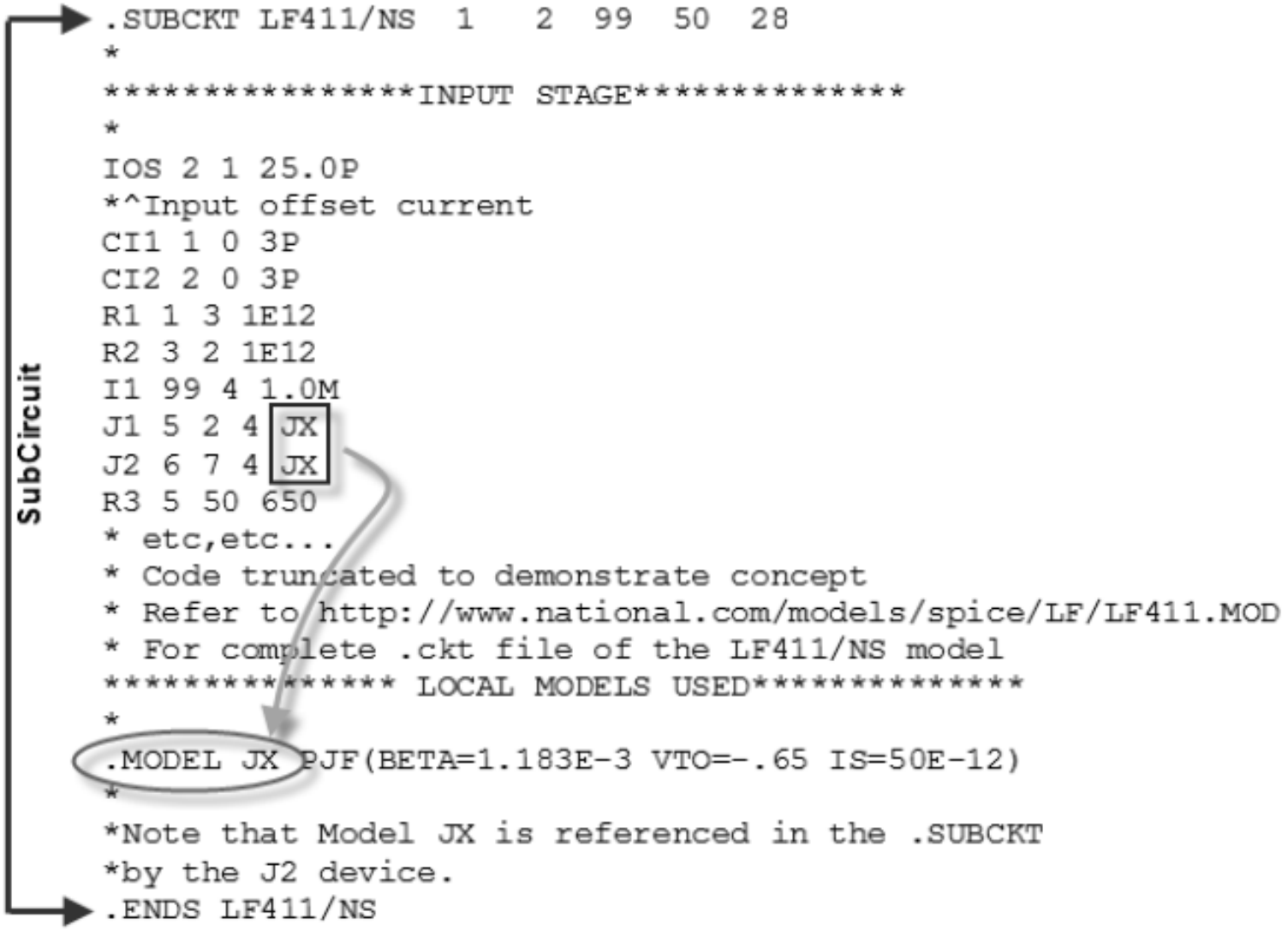
- Altium Designer is compatible with:
  - SPICE3f5 (Berkley SPICE)
  - XSPICE (Georgia Tech)
  - PSPICE (Micro/Sim/Orcad/Cadence)
- You may need to change the file extension to .mdl or .ckt

```
.MODEL Diode D
+(
+  AF=1.0 Bv=5.2 CJO=0.0 EG=1.11 FC=0.5 Ibvl=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



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 0vdcm 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