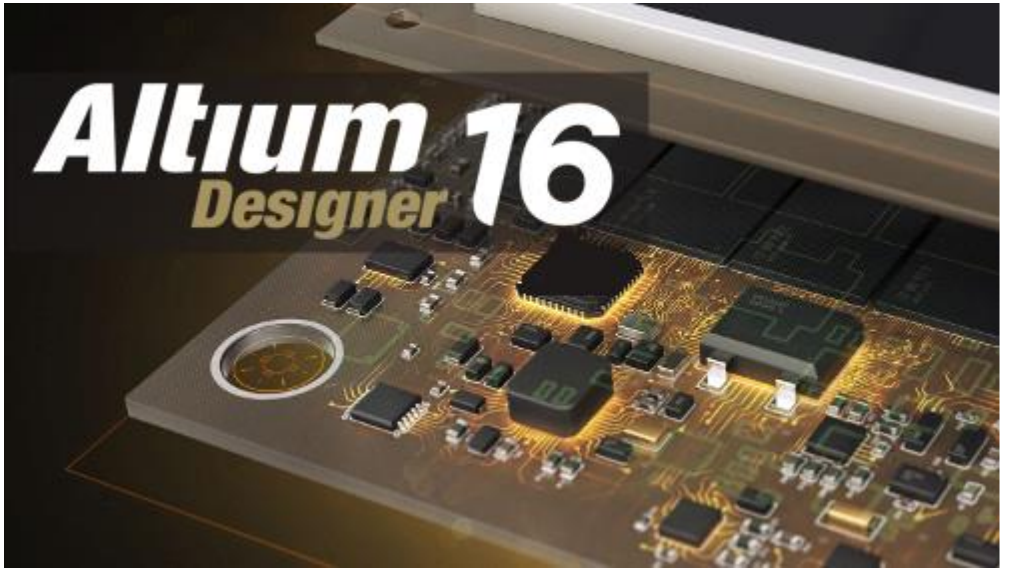



# Altium I

## (Design Capture & Simulation)

ELEC391

Spring 2017

# PCB Design support for ELEC391:

Altium 2016, 150 licenses

Lecture talks:

- Jan 30 Altium I (Design Capture + Simulation)
- Feb 6 Altium II (PCB Layout)
- Mar 13 Guest Lecture – PCB Production
- Support & submission instructions posted here

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

Mon: 13:00-15:00 / 16:00-18:00
Tue : 09:00-11:00 / 12:00-14:00 / 16:00-18:00
Wed: 13:00-15:00 / 16:00-18:00
Tue : 09:00-11:00 / 12:00-14:00 / 16:00-18:00

# Contents

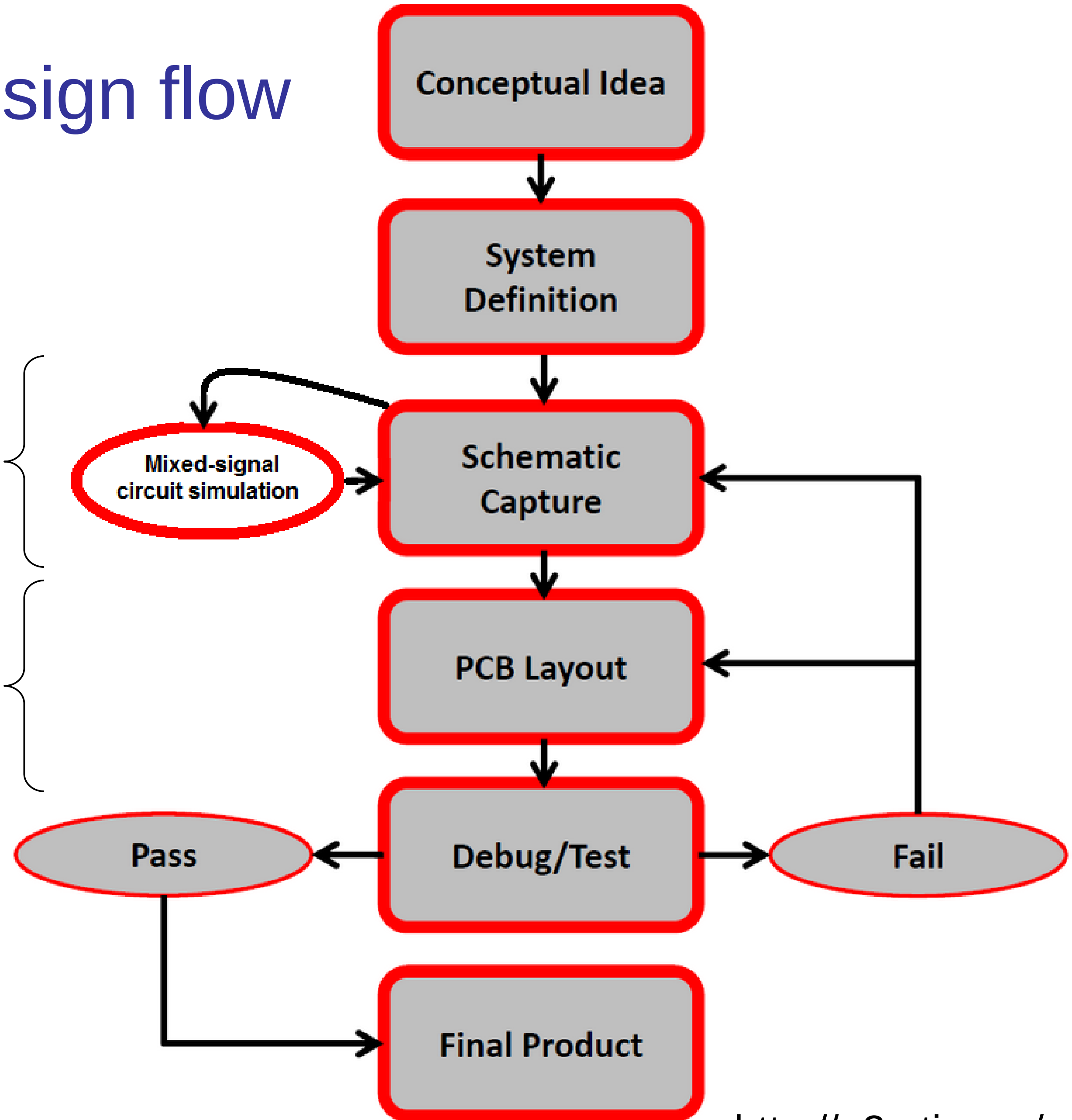
- How to install Altium Designer 2016
- Understanding Altium Designer
- Walk-through Tutorial
  - Schematic Capture
  - Mixed signal simulations
- SPICE basic concepts

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

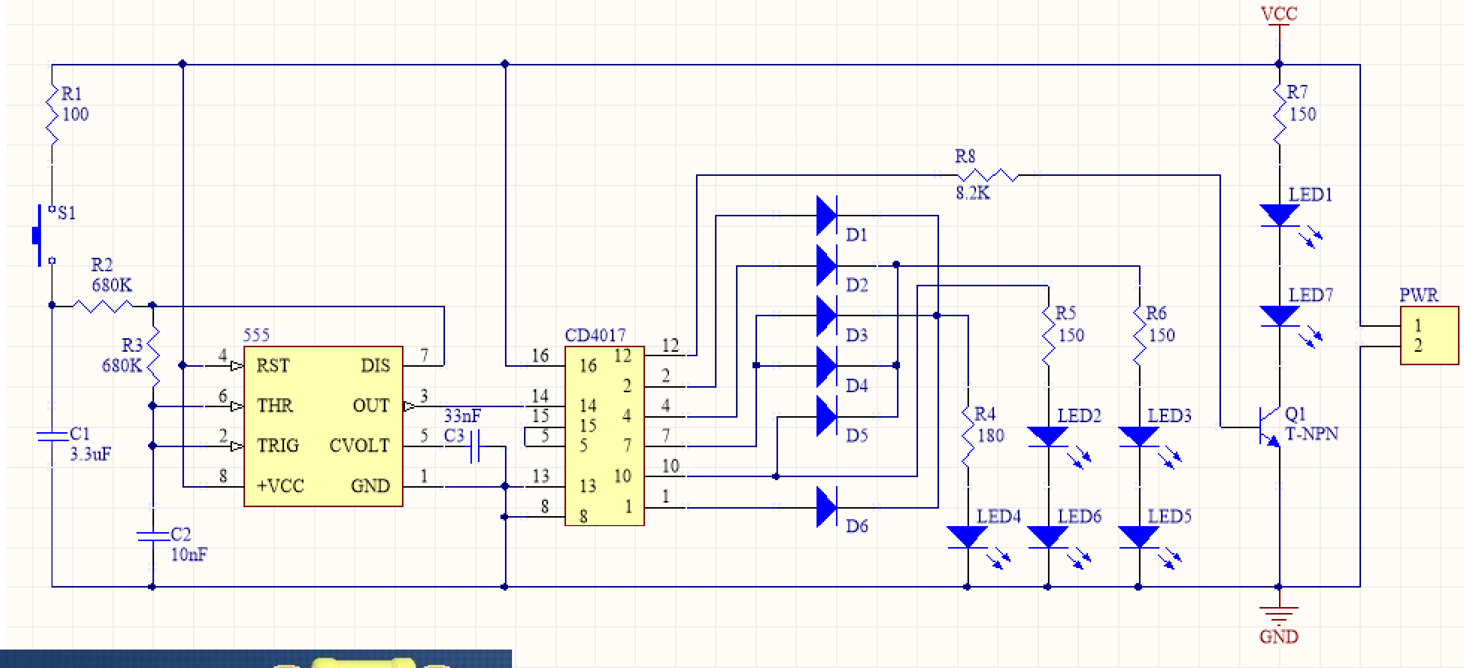
# Typical PCB Design flow

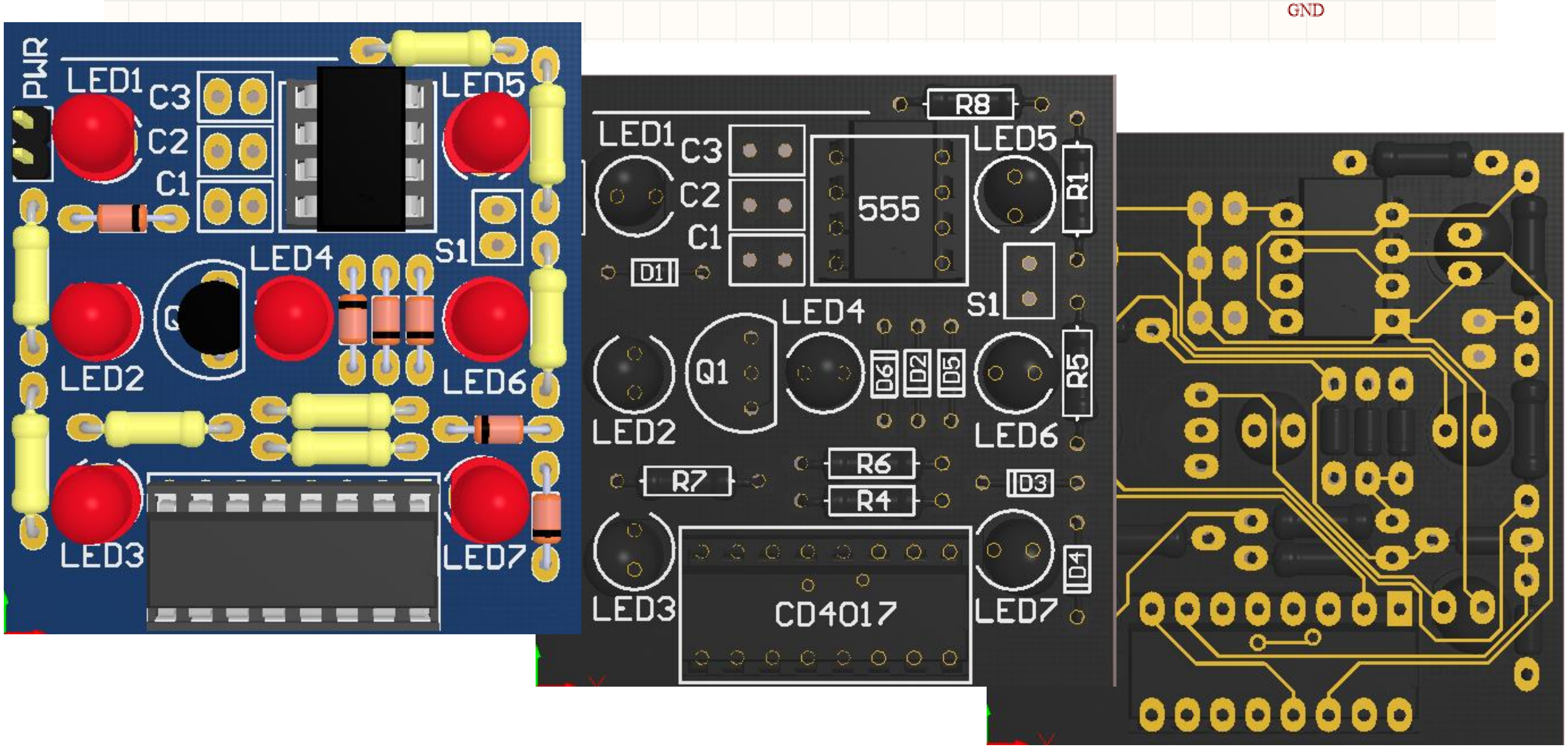
Front-end design and capture

Back-end design

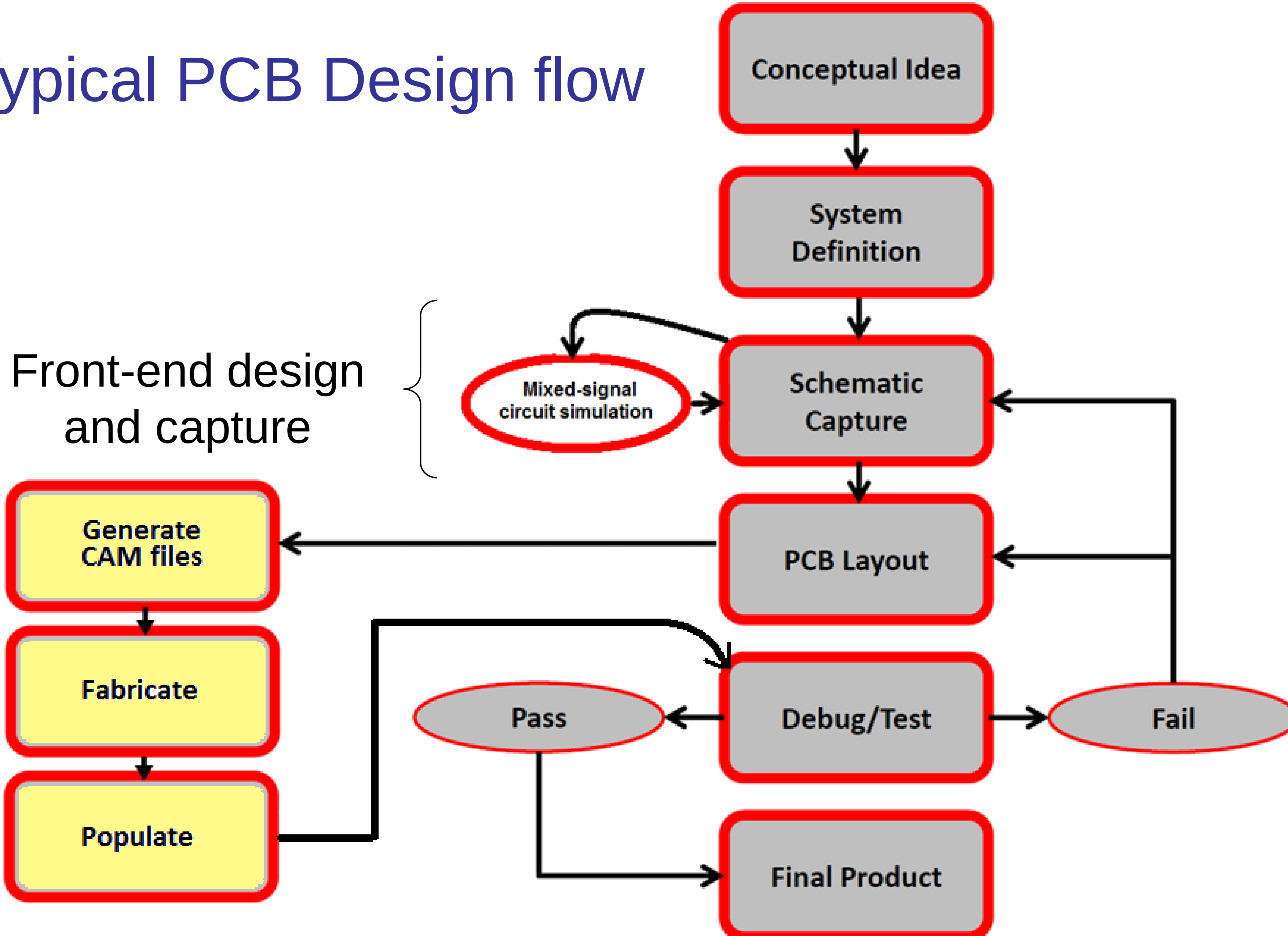



http://e2e.ti.com/


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


PWR
LED1
C3
C2
C1
LED5
LED4
S1
LED2
LED6
LED3
LED7
R8
555
R1
D1
Q1
D6
D2
D5
R5
R6
R7
R4
D3
D4
CD4017

# Typical PCB Design flow

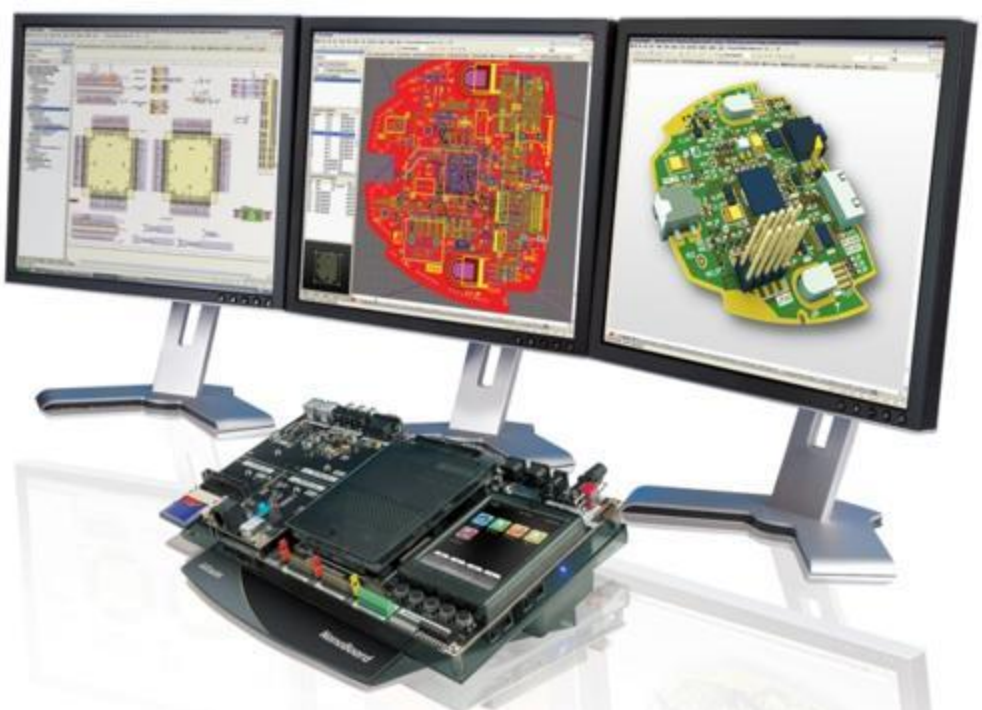



http://e2e.ti.com/

# Altium Designer 2016

## A complete product development system

System requirements (MS W7, W8, W10)



- Front-end design and capture
- Physical PCB design
- FPGA hardware design
- FPGA system implementation and debugging
- Embedded software development
- Mixed-signal circuit simulation
- Signal integrity analysis
- PCB manufacturing

# How to install Altium 2016

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

- Useful links:
  http://www.ece.ubc.ca/~leos/pages/tools/altium.html

- Create an account at Altium Live:
  http://live.altium.com/#signin (slow)
  email: engservices@ece.ubc.ca (fast)

# Install .zip file

THE UNIVERSITY OF BRITISH COLUMBIA

UBC Engineering — Electrical and Computer Engineering

## Electronic Software Distribution

**Search**

Enter search term | Go

**Admin**

- Groups
- Software
- Eligibility

**History**

- Previous Downloads
- Accepted Licenses

**Help**

- Login
- Eligibility
- ISO Files

### ALTIUM DESIGNER

Circuit Design Software

**External Links**

- Altium
- Altium Designer

| |
|---|
| Summer 09 Release |
| 10 |
| 2014 |
| 16 |

### ALTIUM DESIGNER 16

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

### 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:

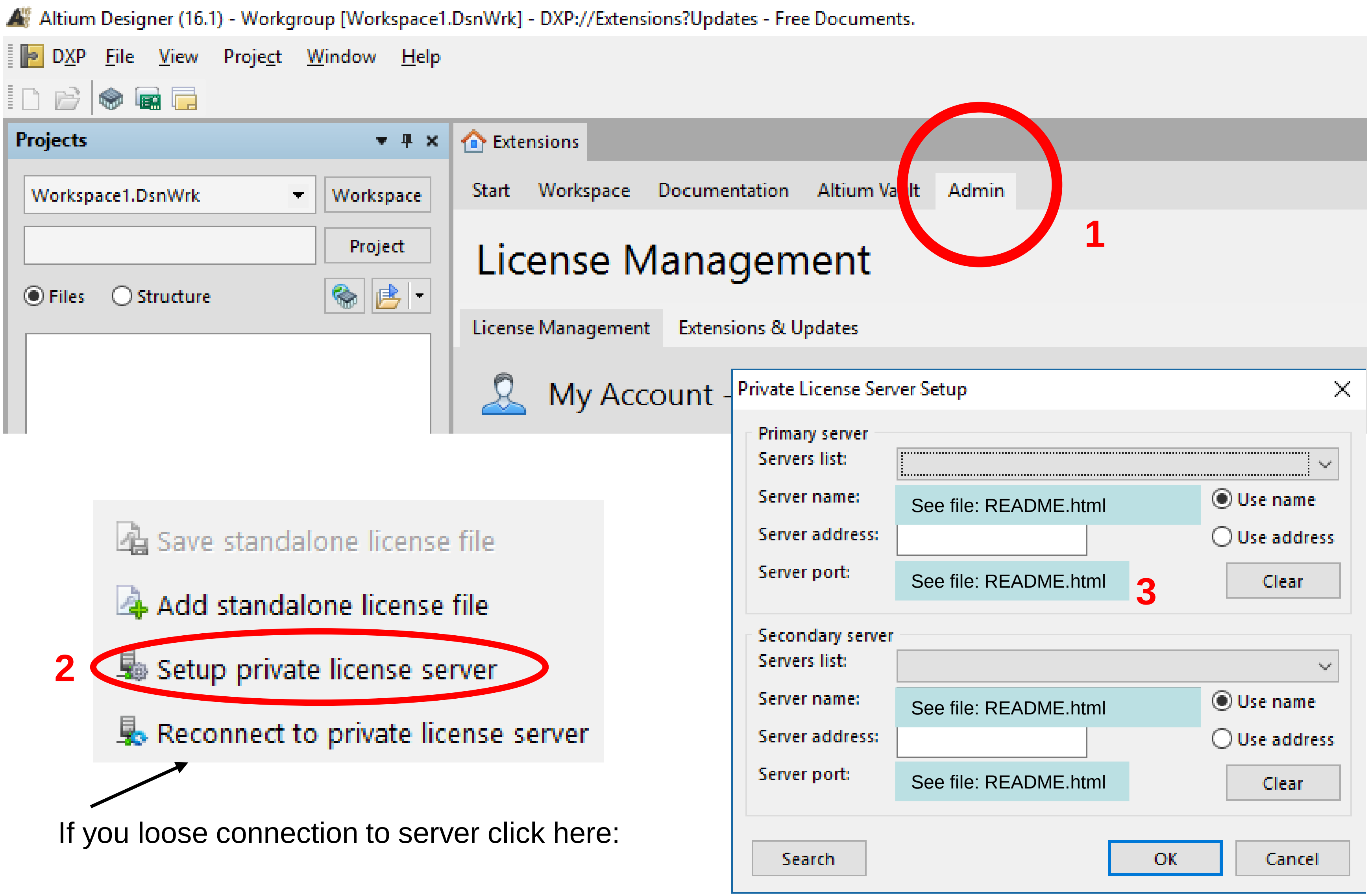
- A wired connection on the ECE network
- A wired connection on UBC ResNet
- A wireless connection at the UBC Vancouver campus on the `ubcprivate`, `ubcsecure`, or `ubc` network (`ubcvisitor` and `eduroam` are not sufficient)
- A myVPN connection to the UBC Vancouver network
- A myVPN connection to the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

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

| | |
|---|---|
| **Server name:** | See file: README.html |
| **Server port:** | |
| **Secondary server name:** | |
| **Server port:** | |

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

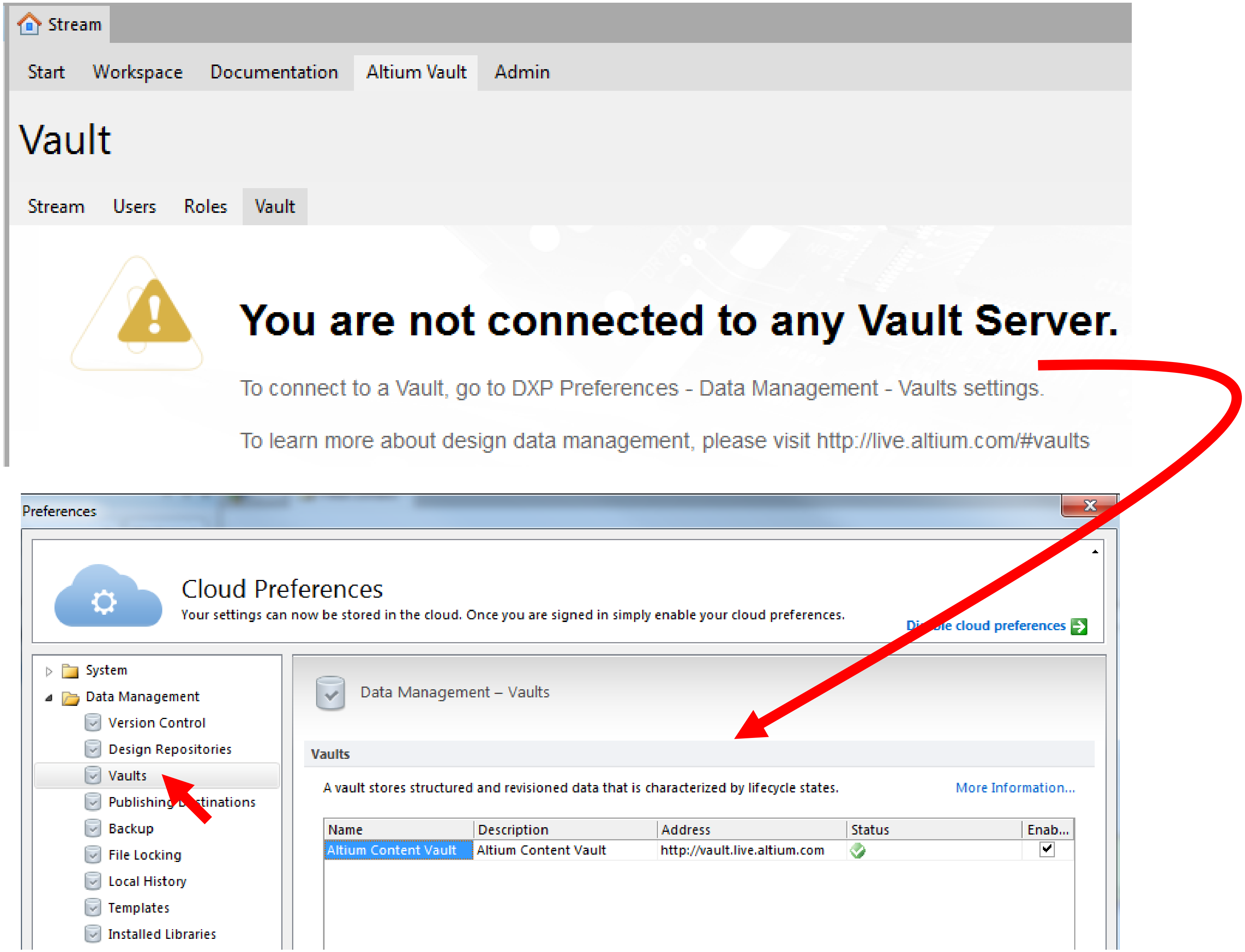
# To set license server




If you loose connection to server click here:

As per README.html file

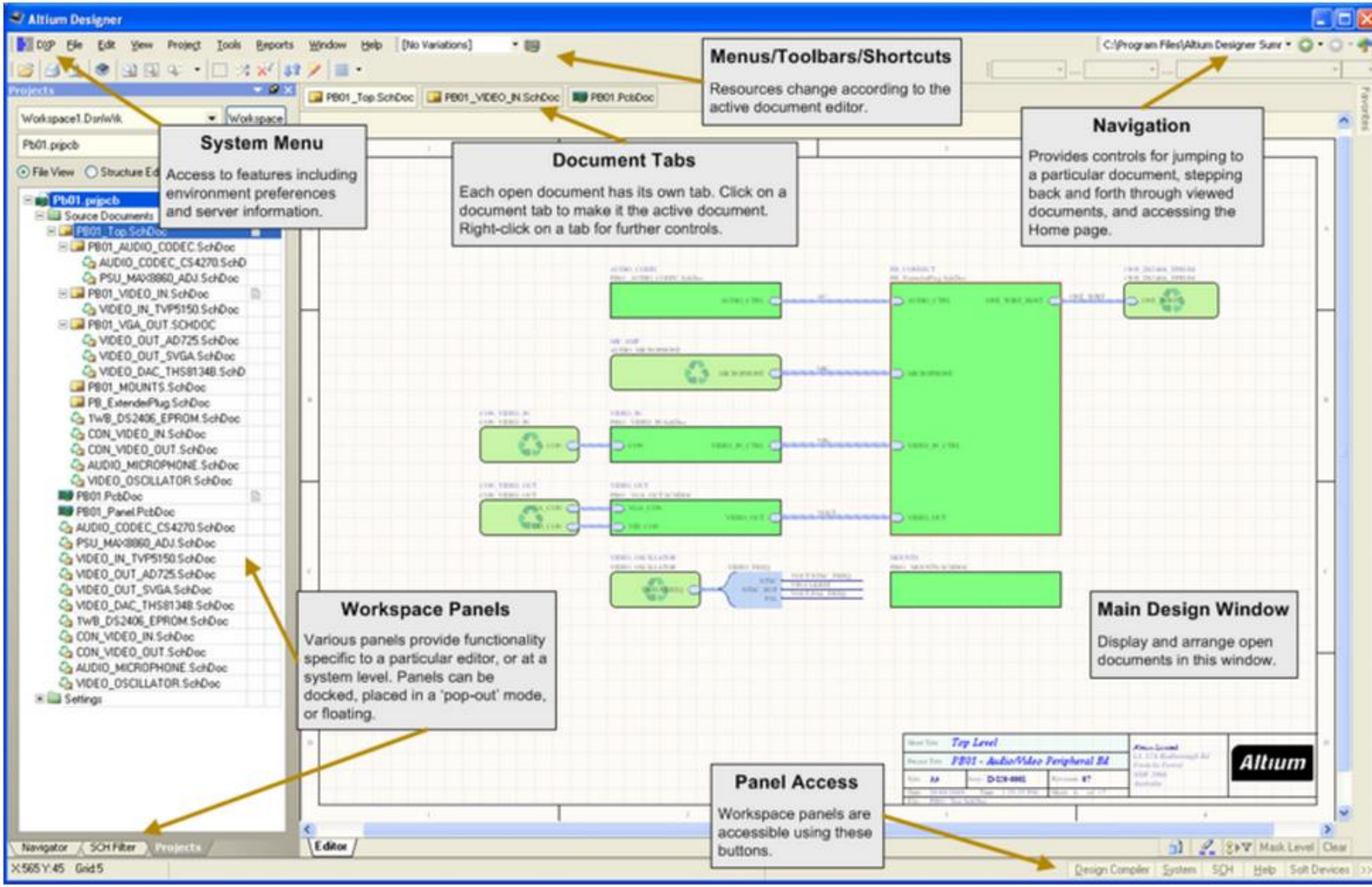
# Connecting to the Altium Vault



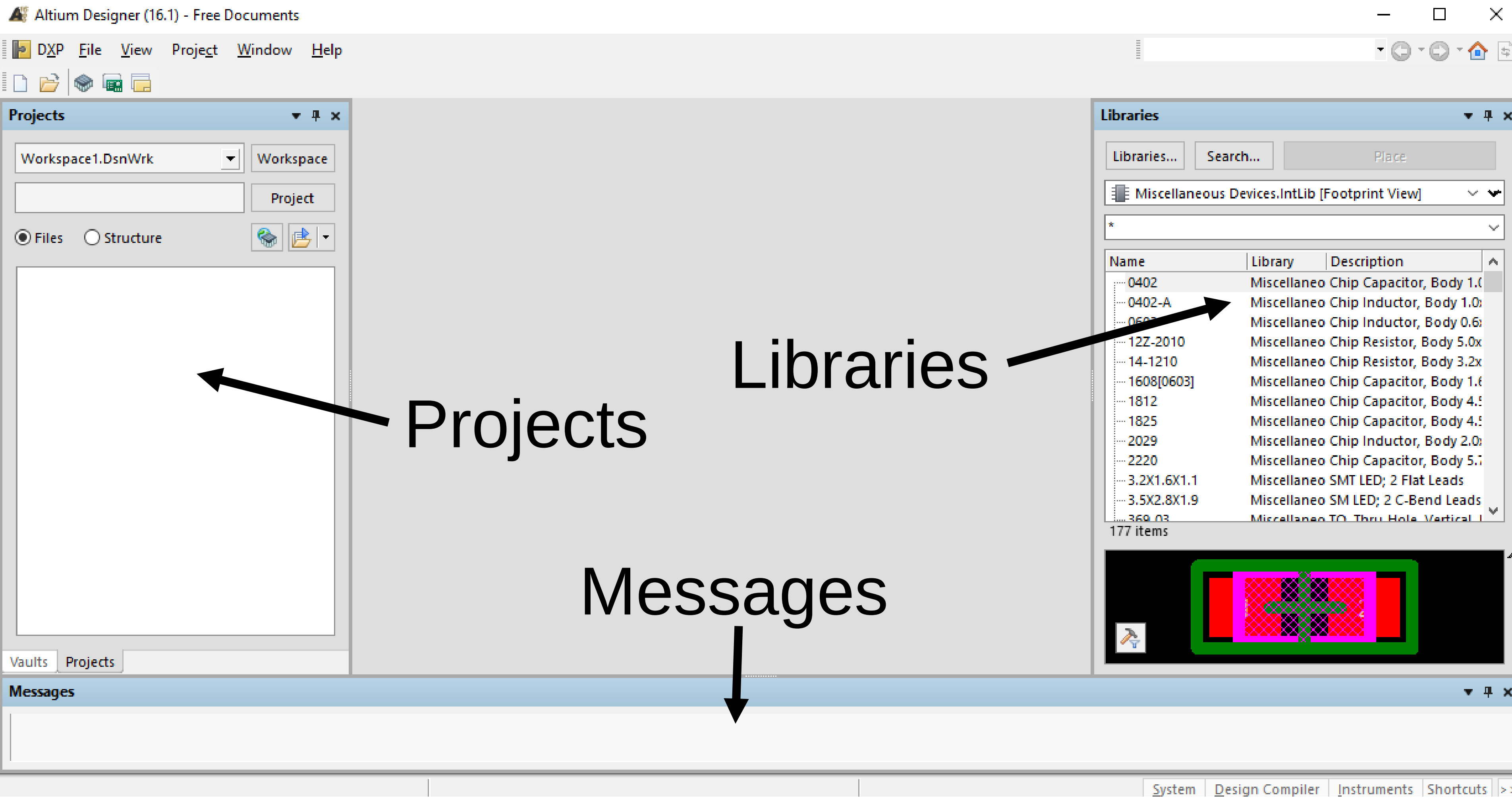
# Understanding Altium

- DXP (Design explorer): Unified platform
- Collaborative environment (corporate tool):
  - Multiple users, some with dedicated tasks
  - Design team incremental changes day-by-day
  - Built-in version control (SVN subversion or CVS concurrent versions system
  - Design repositories / Vaults (accessible by multiple users with different credentials
- Cloud oriented:
  - Save preferences
  - http://live.altium.com/ (forum, design content, blog)

# Altium Design Environment



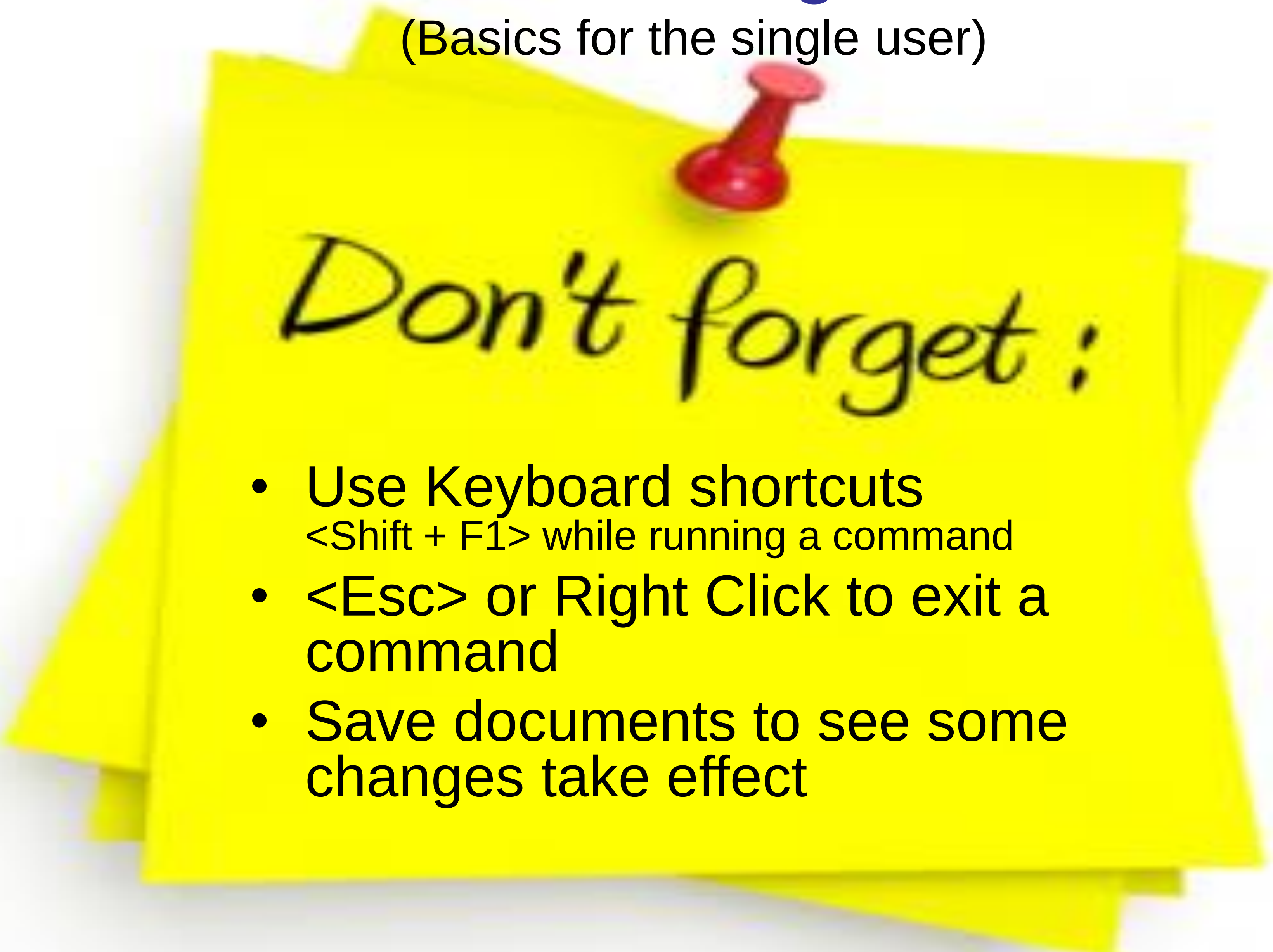
# Recommended basic panels




For more help working with panels read this

# Understanding Altium

(Basics for the single user)

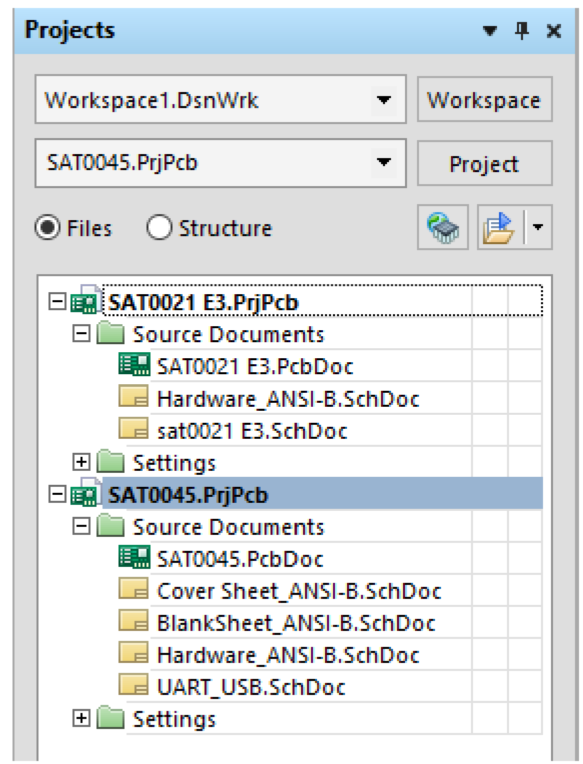



- Use Keyboard shortcuts
  <Shift + F1> while running a command
- <Esc> or Right Click to exit a command
- Save documents to see some changes take effect

# Understanding Altium

(Basics for the single user)

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



# Altium Projects

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

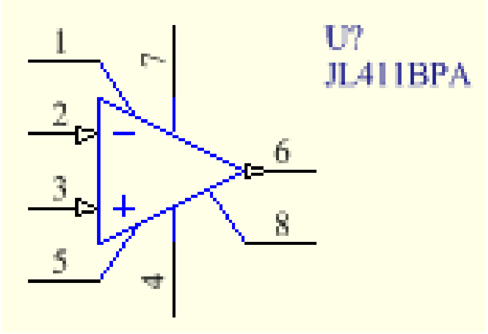
  (does not move the file creates a copy)
- The active project is highlighted
- Add/Remove documents to/from a project

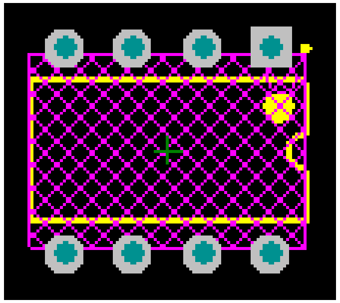
# Altium Projects: types

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

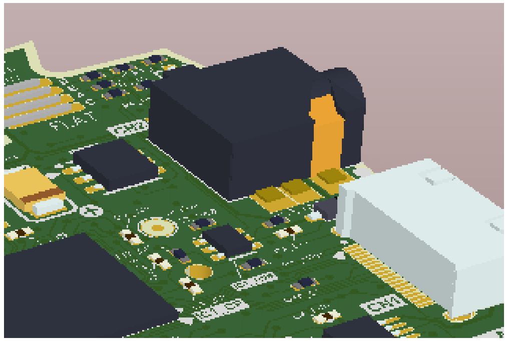
# Component, Model and Library Concepts

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



```
****************INPUT STAGE**************
*
IOS 2 1 25.0P
*^Input offset current
CI1 1 0 3P
CI2 2 0 3P
R1 1 3 1E12
R2 3 2 1E12
I1 99 4 1.0M
J1 5 2 4 JX
J2 6 7 4 JX
R3 5 50 650
R4 6 50 650
*Fp2=28 MHZ
```



# Component, Model and Library Concepts

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

Different component representations

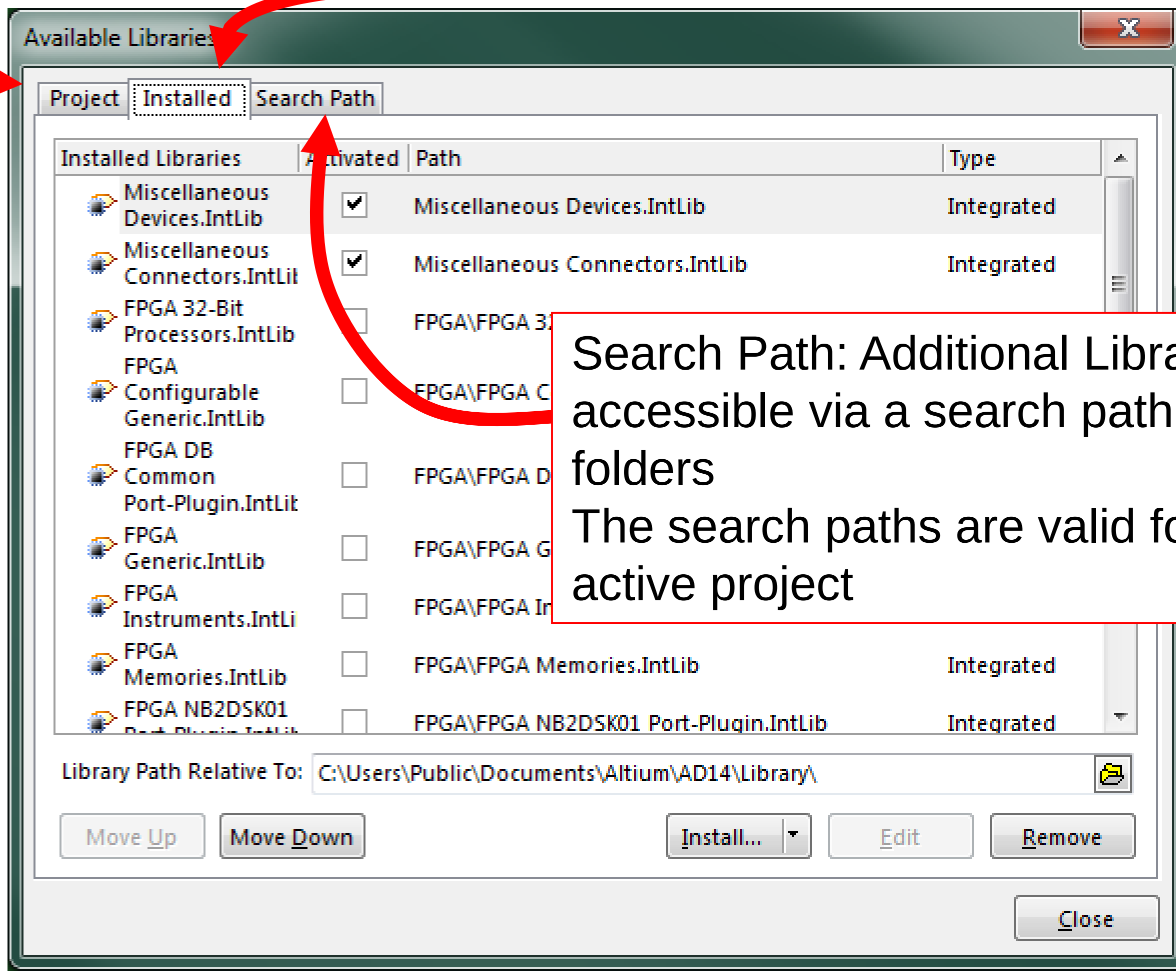
- A unified component is a container with links to all domain models + parametric information

# Libraries = collection of components

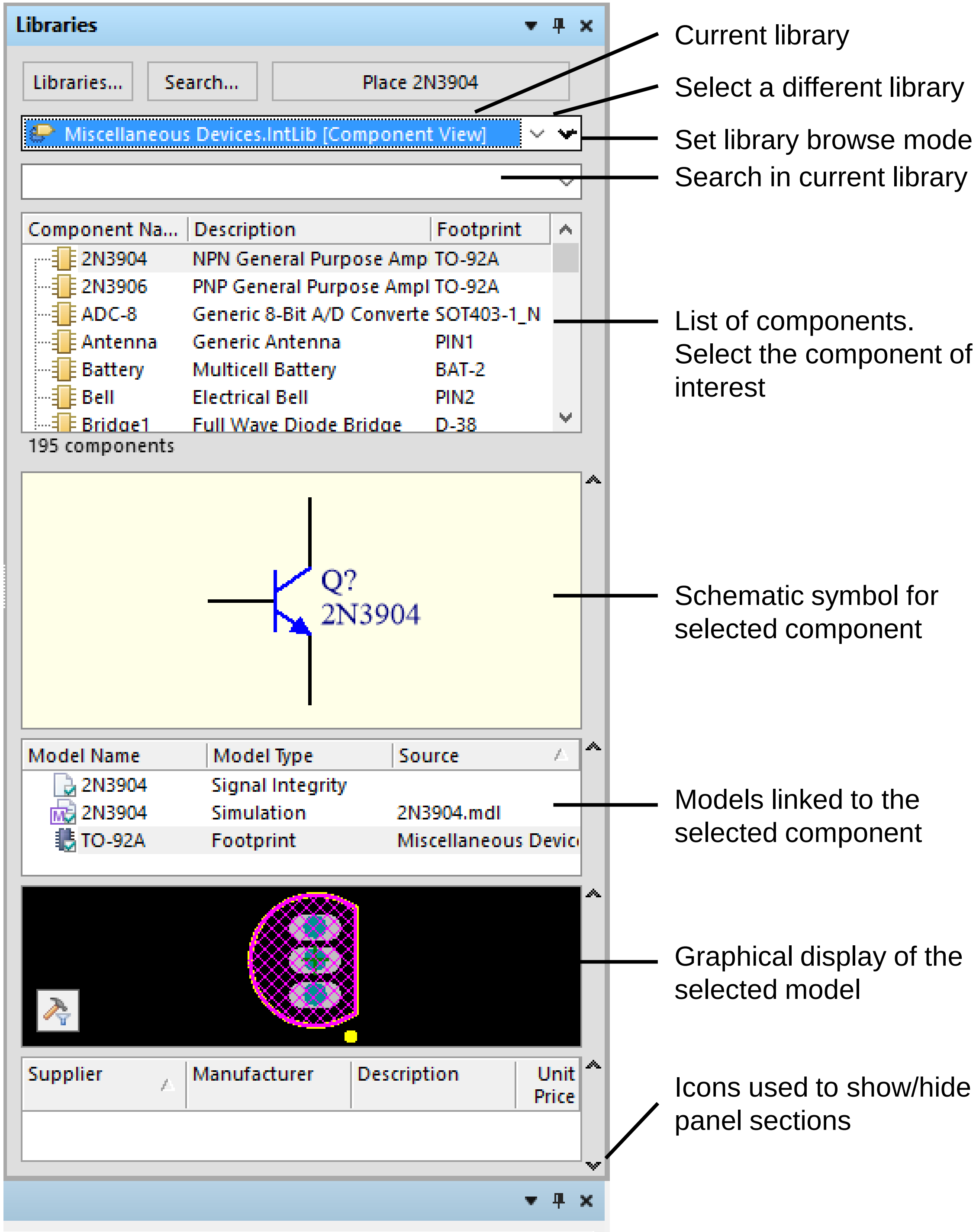
- Collection of components, models or both
- Model Libraries (*.MDL, *.CKT, *.PCBLib)
  - Simulation models are one file per model
- Schematic Libraries (*SchLib)
  - Symbol and a link to a model library
- Integrated Libraries (*.IntLib)
  - Symbol, footprint and other models are compiled into a single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

Project: part of and available only to the active project and its documents
You have to keep track of where these are if you move the project files

Installed: All installed libraries. Components are available to all open projects and list is persistent across design sessions




Search Path: Additional Libraries accessible via a search path and sub-folders
The search paths are valid for the active project




# Libraries Panel:

All libraries available to the active project

Project + Installed + Search Path

## When placing component:

<spacebar> to rotate

<x> or <y> to flip

<Tab> open properties dialog

<L> for PCB footprints
to flip component side

## To search across libraries:

Search …

# Obtaining integrated libraries

1. Frozen (legacy) libraries: from here
   you can install anywhere but it is a good idea to make a subfolder under:
   C:\Users\Public\Public Documents\Altium\AD16\Library
   or a cloud storage service if you work from more than one PC
2. AltiumLive website: Resources / Design Content

National Semiconductor

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.

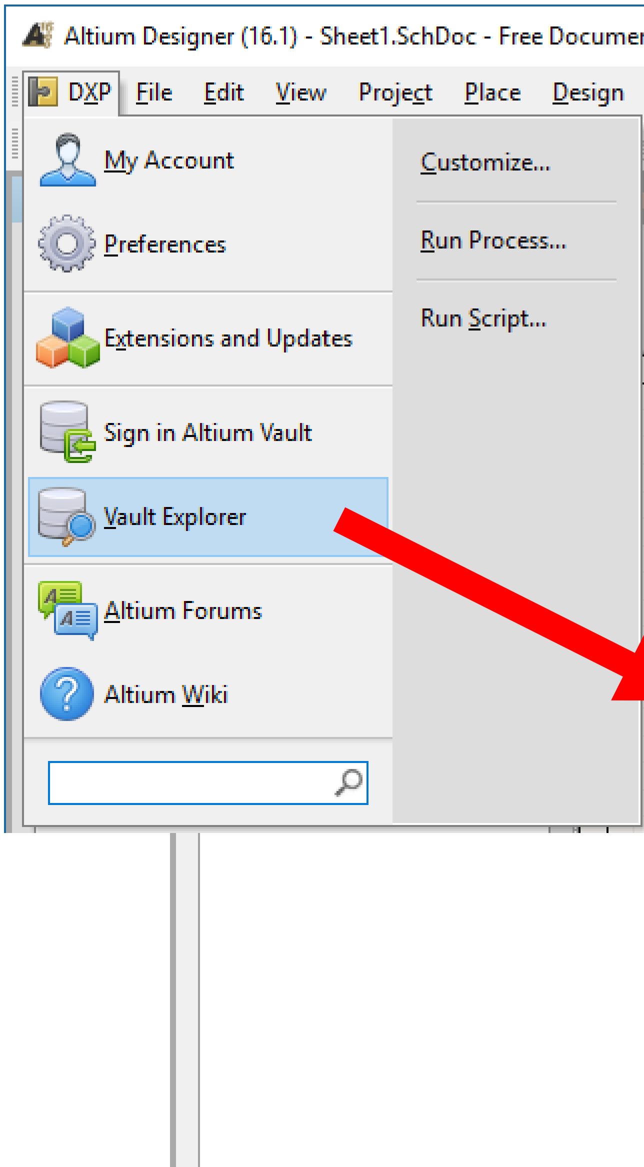
GO TO VAULT DOWNLOAD LIBRARY

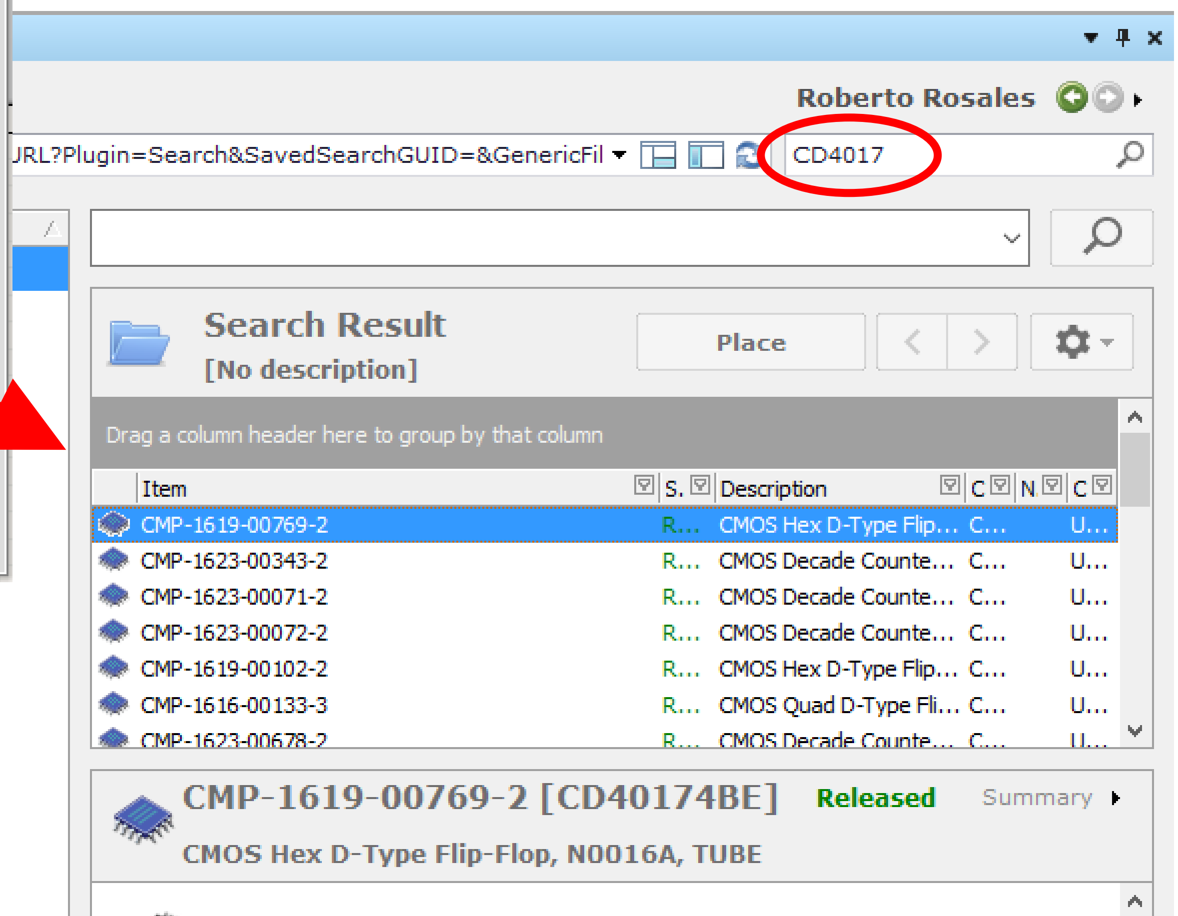
This is useful to preview component

This downloads a .zip file for the complete library

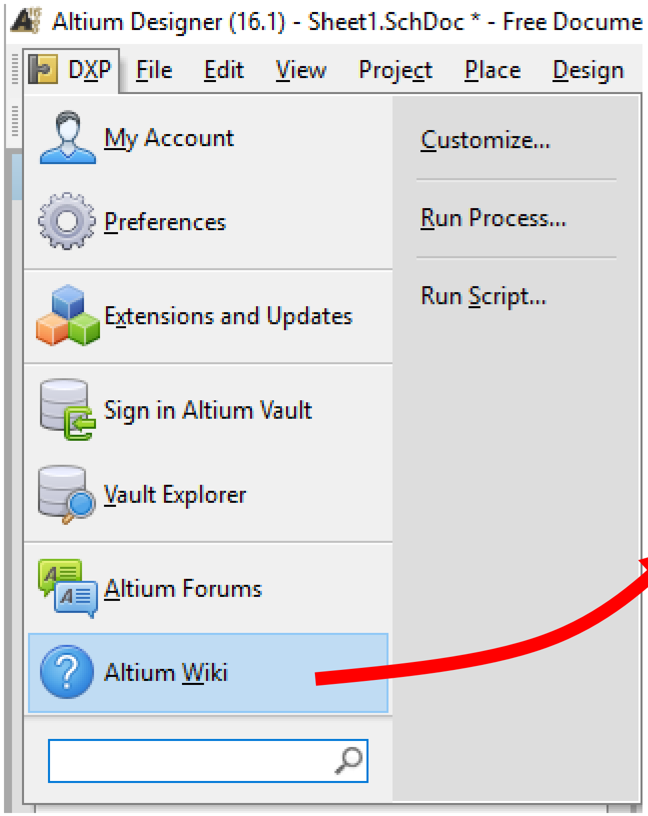
# Altium Vault

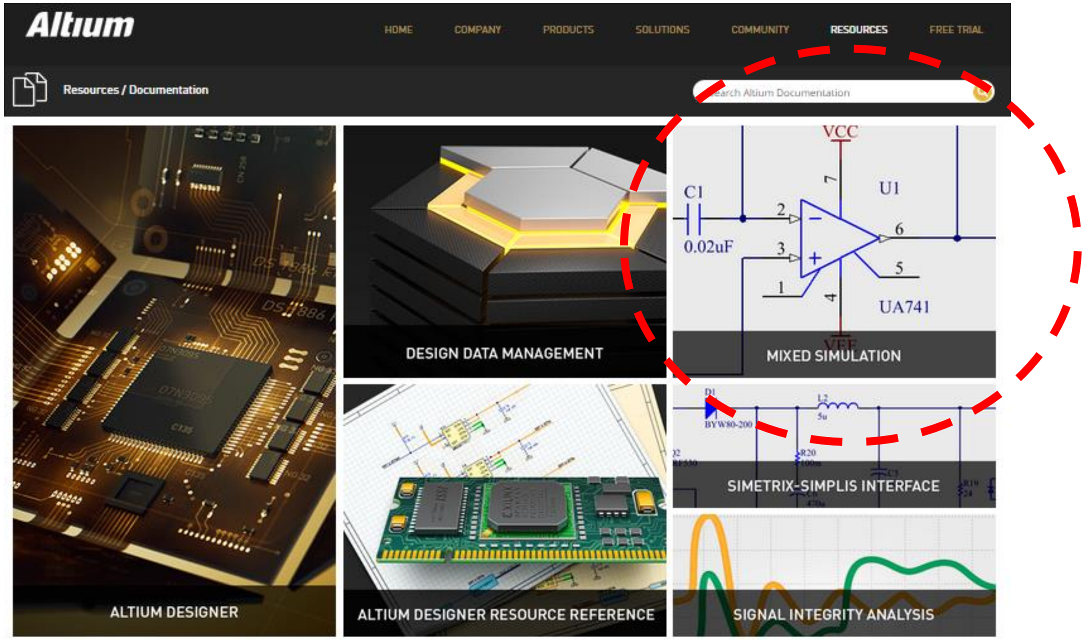
Altium's cloud library (repository of models)
Also includes real-time supply chain information



# Learning to use Altium



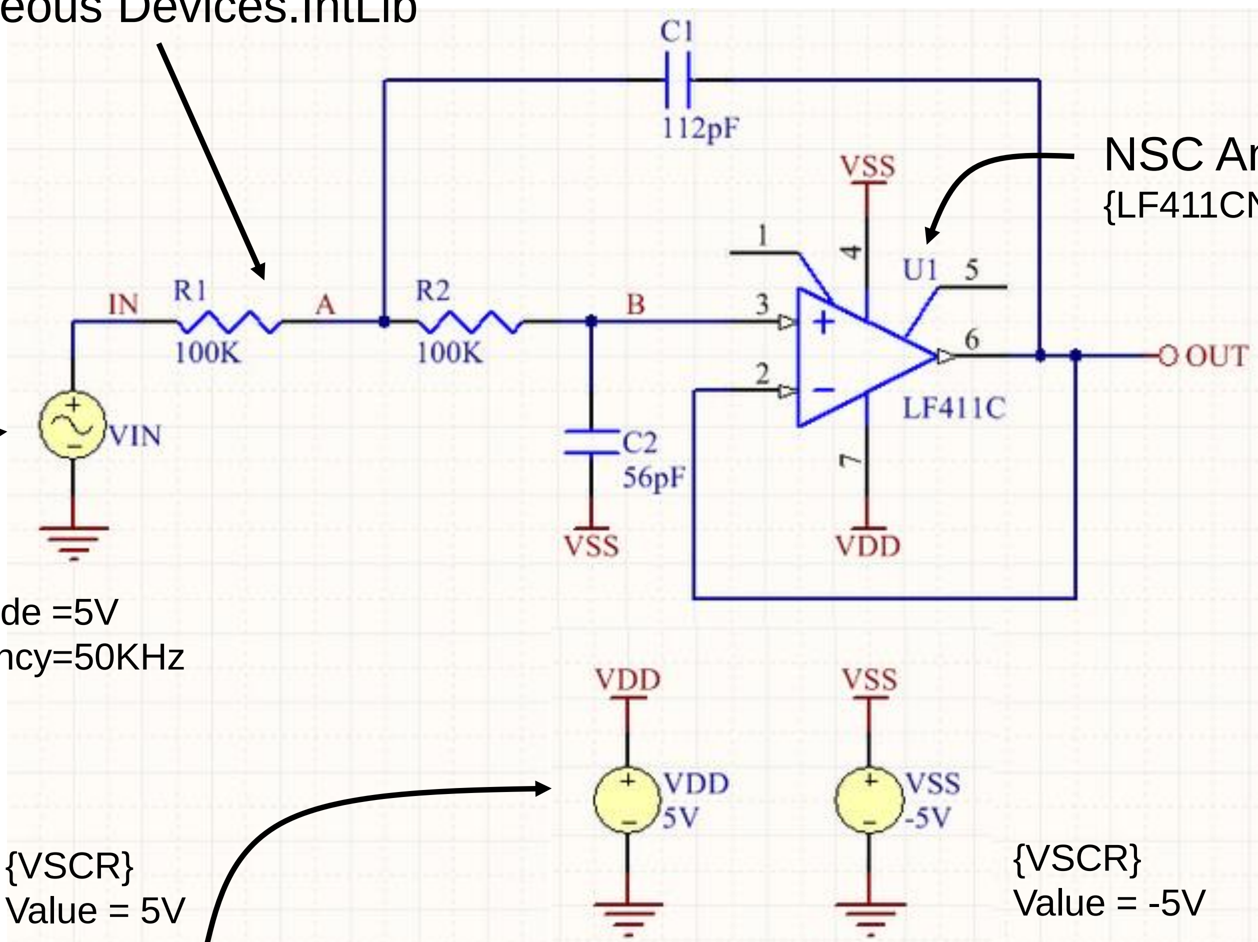

Best training material is on the Altium website
It is updated, but beware that menus and options slightly change between versions

# Demo: Schematic entry and Simulation

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

Miscellaneous Devices.IntLib
{Res1, Cap}

NSC Amplifier.IntLib
{LF411CN} flip in Y!



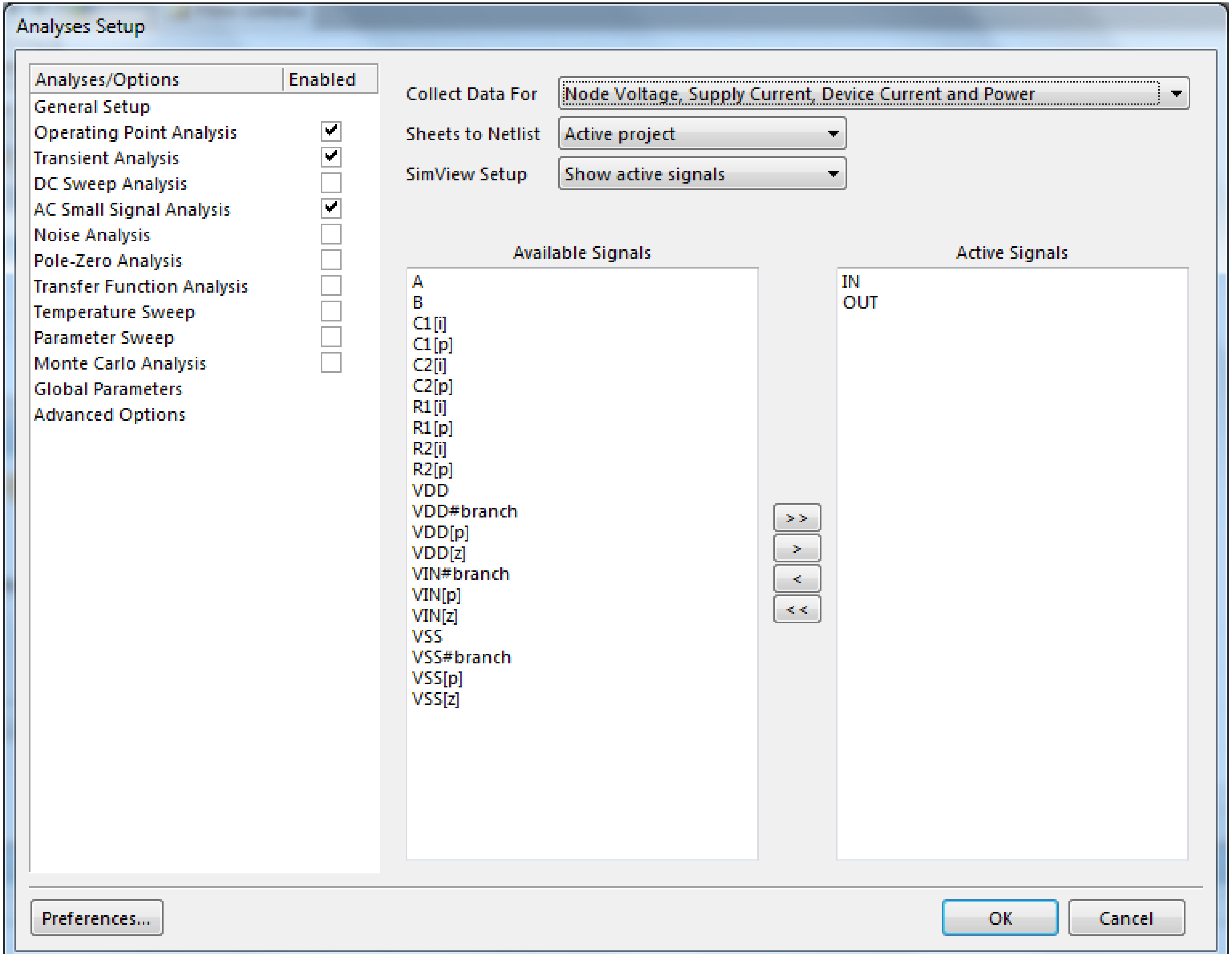

{VSIN}
Amplitude =5V
Frequency=50KHz

{VSCR}
Value = 5V

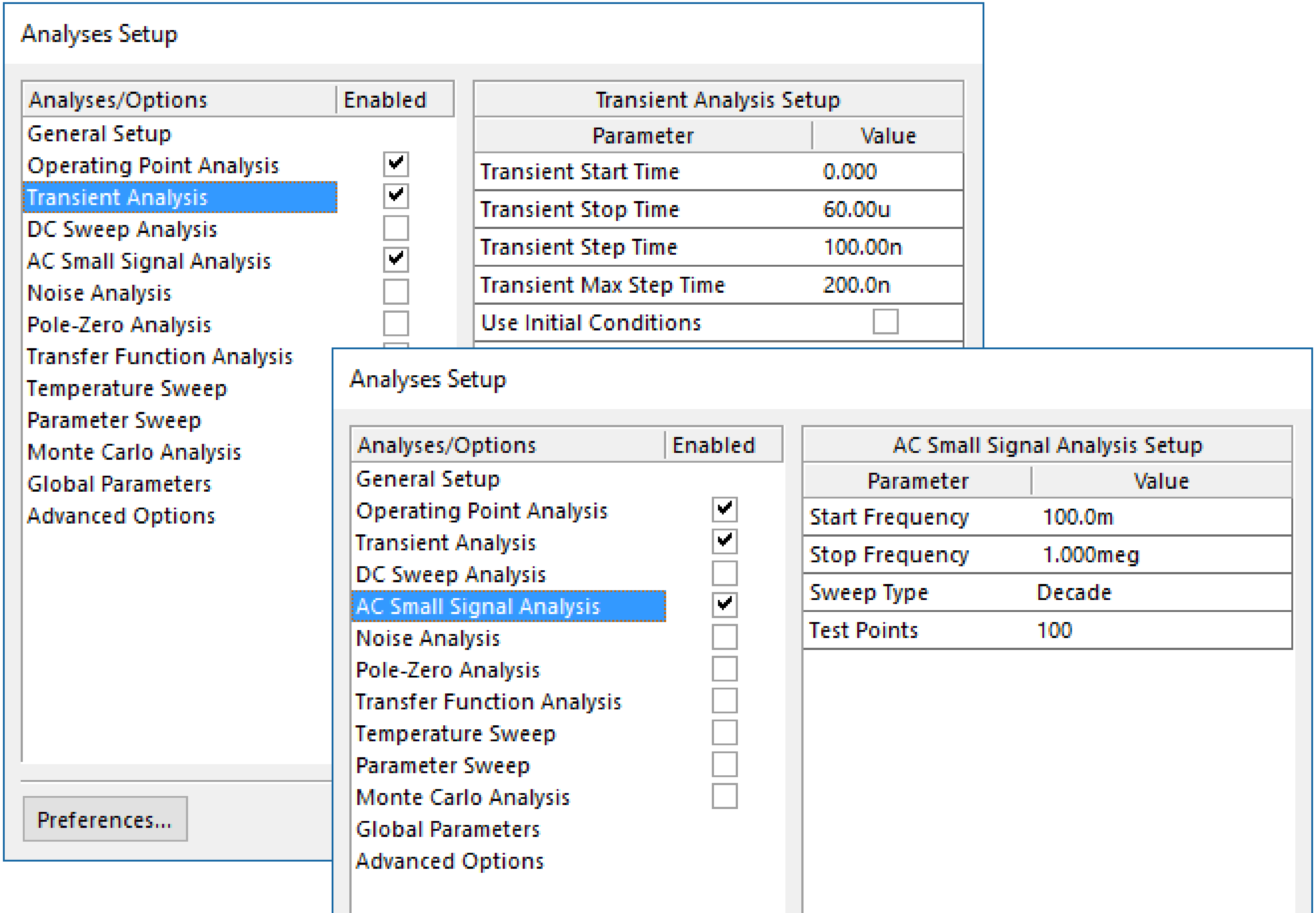
{VSCR}
Value = -5V

C:\Users\Public\Public Documents\Altium\AD16\Library\Simulation\Simulation Sources.IntLib

# Set simulation parameters



# 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.

# About SPICE

- Berkley (class project +Masters), CANCER
  Computer Analysis of Nonlinear Circuits Excluding
  Radiation
- Berkley (PhD) , Simulation Program with Integrated Circuit Emphasis
  → SPICE 1972 FORTRAN
  → SPICE 2 1975, SPICE 2G6 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

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 PJF(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 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><br>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><br>+ [<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><br>+ [L=<value>] [W=<value>]<br>+ [AD=<drain area value>] [AS=<source area value>]<br>+ [PD=<drain perimeter value>] [PS=<source perimeter value>]<br>+ [NRD=<value>] [NRS=< value>]<br>+ [IC=<initial D-S volt.>, <initial G-S volt.>, <initial B-S volt.>]<br>+ [TEMP=<temperature>] |
| **Q** | Bipolar Transistor | Q<refdes> <collector> <base> <emitter> <model> [<area>]<br>+ [IC=<initial B-E voltage>, <initial C-E voltage>]<br>+ [TEMP=<temperature>] |
| **R** | Resistor | R<refdes> <+node> <-node> [<model>] <value> |
| **S** | Voltage controlled switch | S<refdes> <+node> <-node> <+control><br>+ <-control> <model> [initial condition] |
| **T** | Transmission Line | T<refdes> <A+> <A-> <B+> <B-> Z0=<value><br>+ [TD=<value> \| F=<value>[NL=<value>]] |
| **V** | Voltage Source | V<refdes> <+node> <-node> [[DC] <value>]<br>+ [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 |