National Yang Ming Chiao Tung University Institute of Electronic Engineering

CAD HW3 -

Modeling Mixed-Signal System and Simulating with AMS

Outline

- Introduction to AMS
- AMS Simulation Setup
  - Analog Simulation with Verilog-A Model
  - Mixed-Signal Behavioral Model Simulation
    - Lab1: Mixed-Signal Model Simulation with AMS
    - Lab2: 4-bit ADC with Verilog-AMS

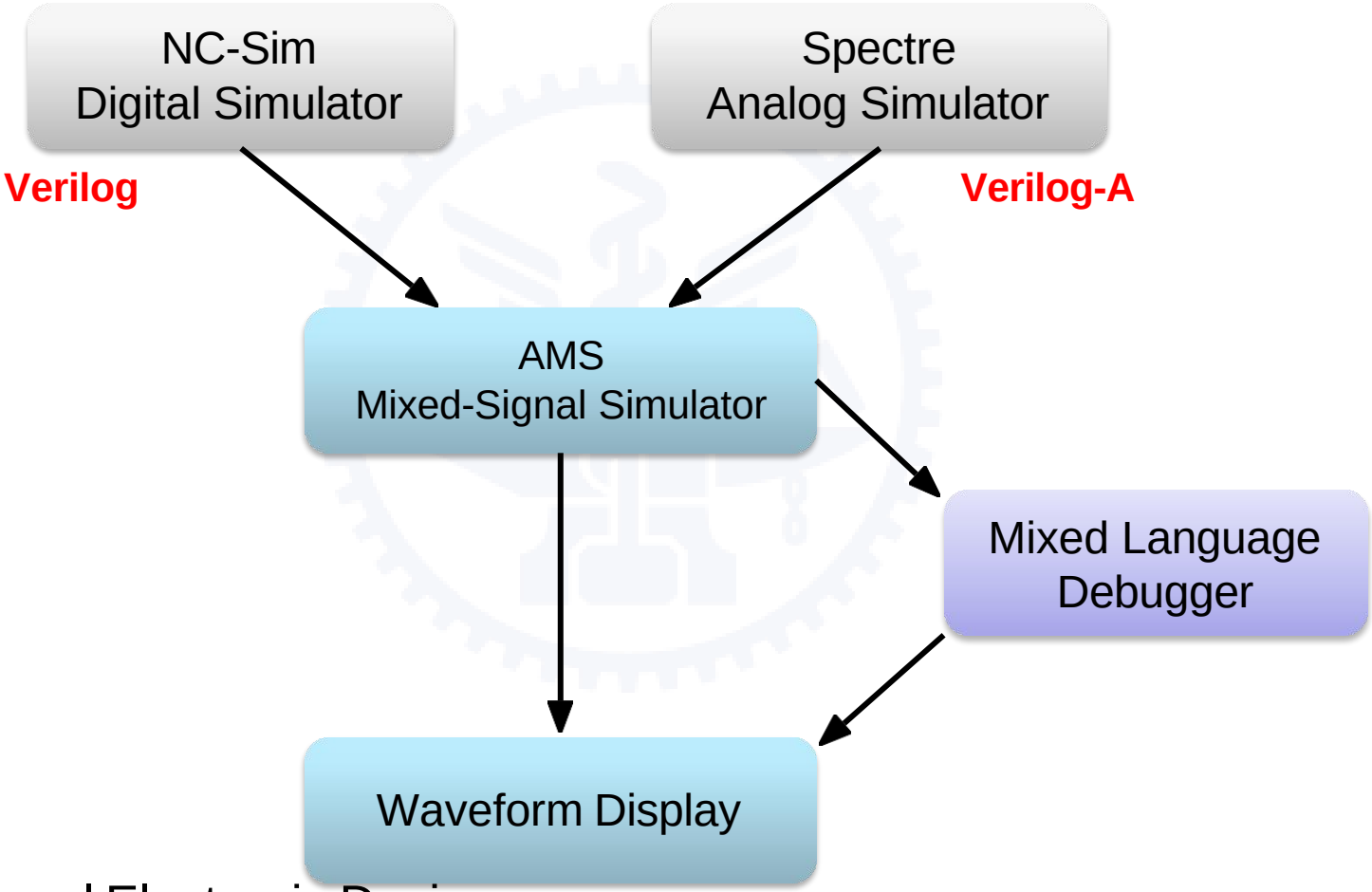
Outline

- Introduction to AMS
- AMS Simulation Setup
  - Analog Simulation with Verilog-A Model
  - Mixed-Signal Behavioral Model Simulation
    - Lab1: Mixed-Signal Model Simulation with AMS
    - Lab2: 4-bit ADC with Verilog-AMS

What is AMS Designer

- Top-down system-on-chip simulation for complex mixed-signal designs
- A single executable simulator incorporating the fastest in digital and most flexible analog simulation capability
  - Digital: NC-Sim
  - Analog: Spectre
- Simulation of complex designs incorporating any and all of the following:
  - Verilog, VHDL
  - Verilog-A, Verilog-AMS, VHDL-AMS
  - Spectre
  - SPICE
  - Composer schematics

What is AMS Designer(cont.)



Mixed-Signal Simulation with Model

- The mixed-signal behavioral model simulation can verify:
  - System behavior is correct or not?
  - System requirement is met or not?
  - System performance is satisfied or not?

- Weaknesses:
  - Only time domain information can be obtained directly
    - All behavioral model should be converted into time domain
    - Other characteristics might be calculated from time domain data

Outline

- Introduction to AMS
- AMS Simulation Setup
  - Analog Simulation with Verilog-A Model
  - Mixed-Signal Behavioral Model Simulation
    - Lab1: Mixed-Signal Model Simulation with AMS
    - Lab2: 4-bit ADC with Verilog-AMS

Environment Setting

- NC-Verilog / Verilog-XL
  - e.g > source /usr/cad/cadence/CIC/incisiv.cshrc
- Spectre
  - e.g > source /usr/cad/cadence/CIC/mmsim.cshrc
- Composer / Virtuoso
  - e.g > source /usr/cad/cadence/CIC/ic_06.17.709.cshrc

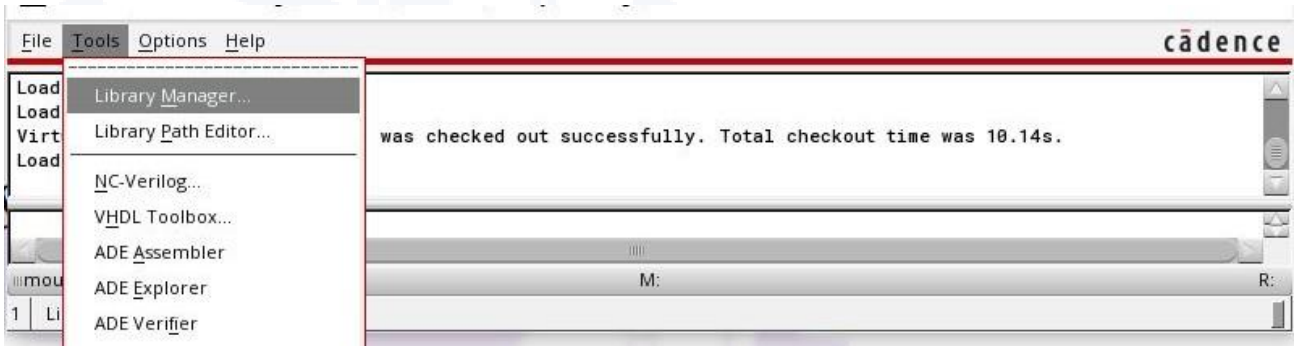
Simulation Flow

Step 1. > virtuoso &

Step 2. Open library manager

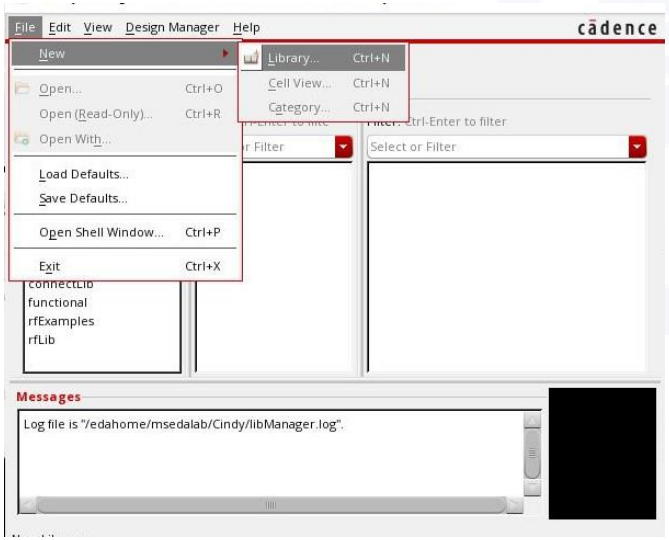
- Create library
- Create cell view & symbol
  - Verilog-A models
  - Verilog models
  - Schematic
  - Config
- Set ADE

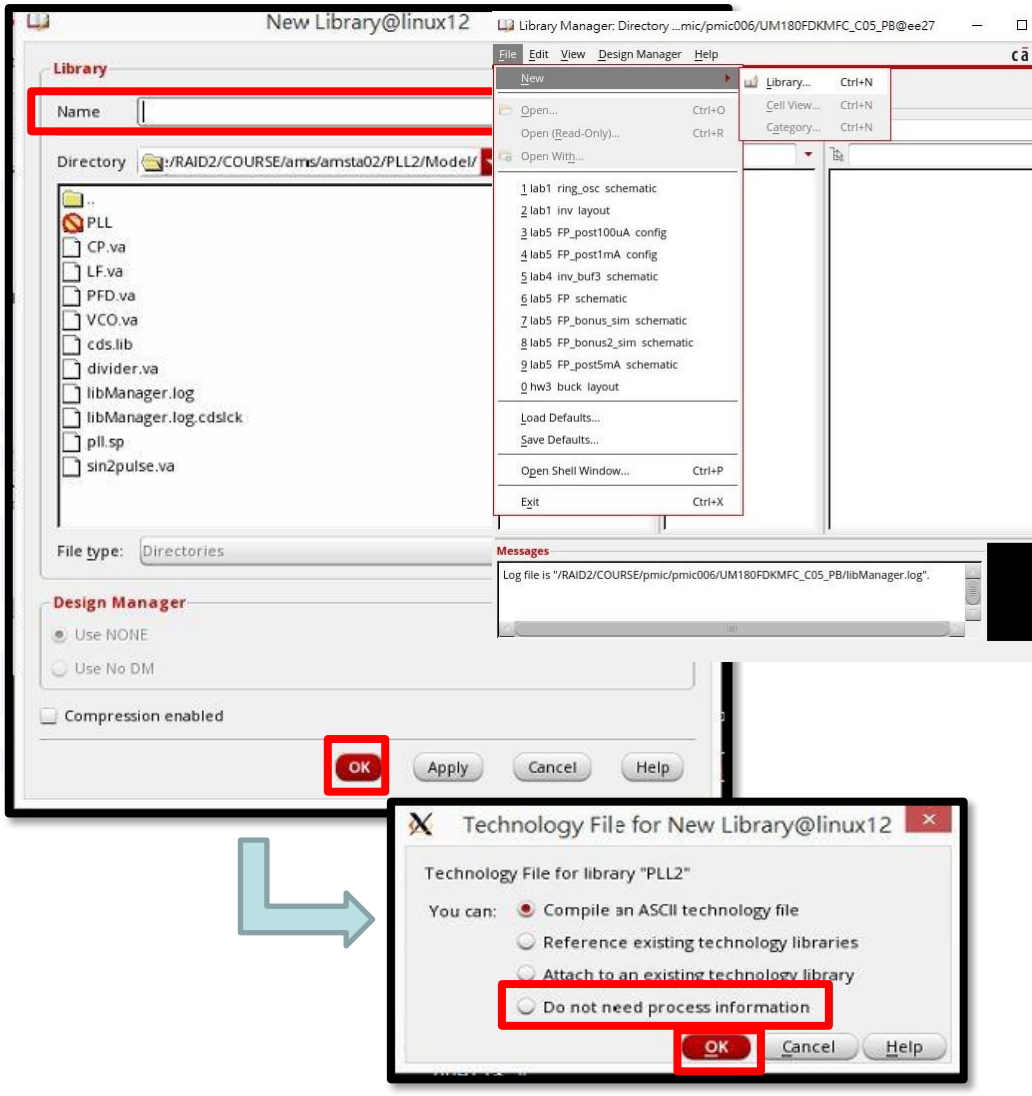
Open library manager



Simulation Flow

- Create library
- Create cell view & symbol
  - Verilog-A models
  - Verilog models
  - Schematic
  - Config
- Set ADE

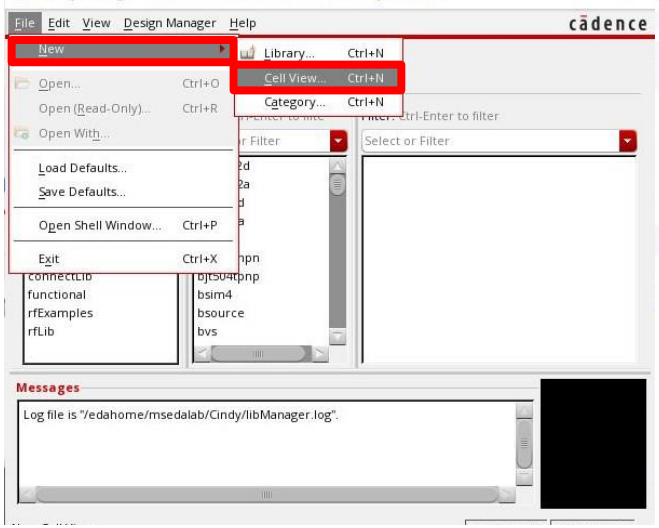


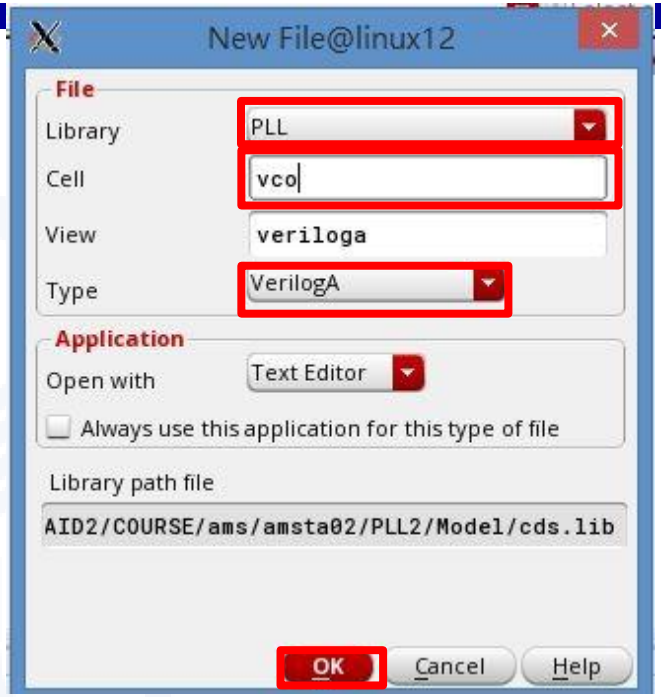


Create Verilog-A Cells

- Create library
- Create cell view & symbol
  - Verilog-A models
  - Verilog models
  - Schematic
  - Config
- Set ADE

Create a new cell view



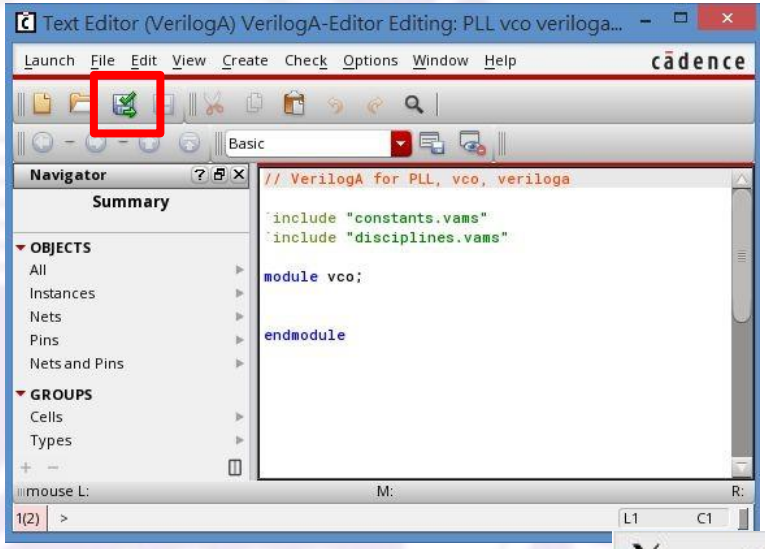


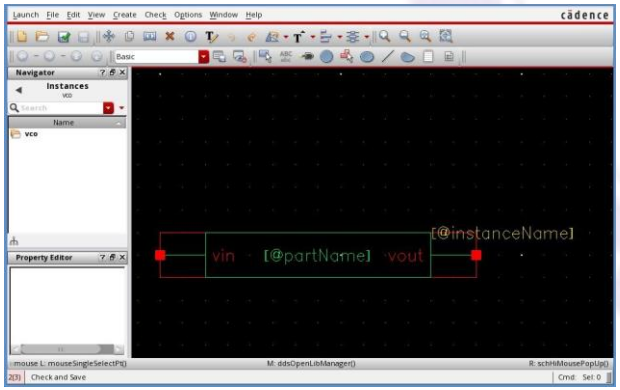
1. Choose your library
2. Input the name on the Cell Name column
3. Choose the **VerilogA** type for Analog model
4. OK

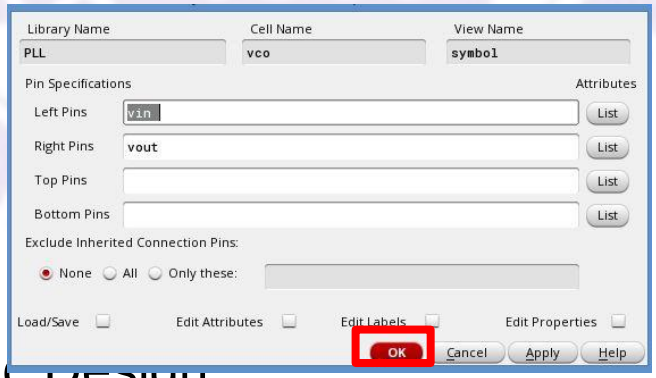
Designing with Verilog-A

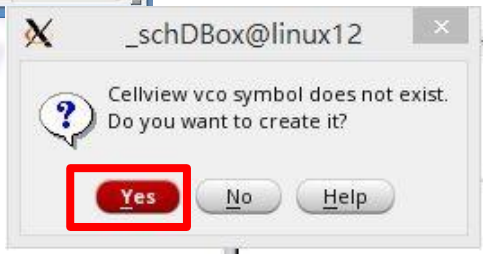
- Create library
- Create cell view & symbol
  - Verilog-A models
  - Verilog models
  - Schematic
  - Config
- Set ADE

Enter behavioral description





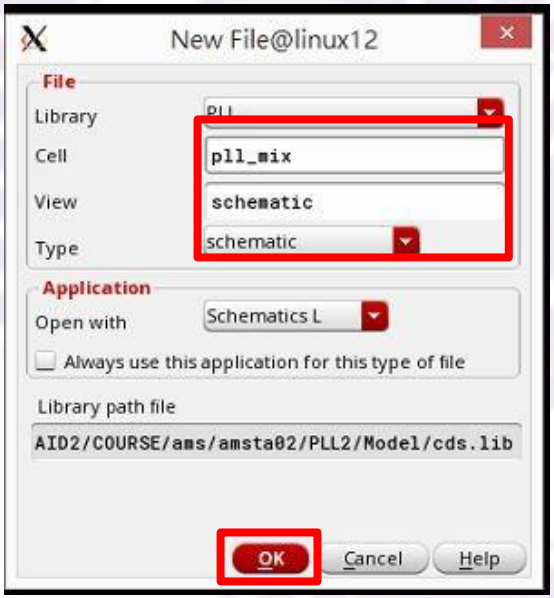


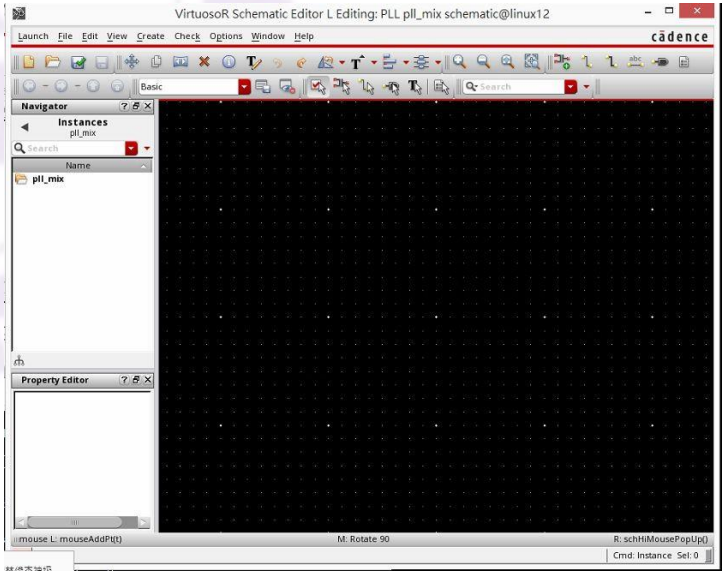


Create Top Cell - Schematic

- Create library
- Create cell view & symbol
  - Verilog-A models
  - Verilog models
  - Schematic
  - Config
- Set ADE

Input the name on the Cell Name column and choose the Schematic

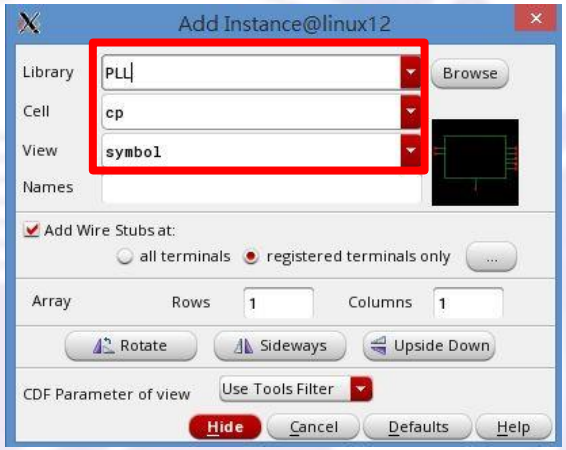




Create Top Cell - Schematic

- Create library
- Create cell view & symbol
  - Verilog-A models
  - Verilog models
  - Schematic
  - Config
- Set ADE

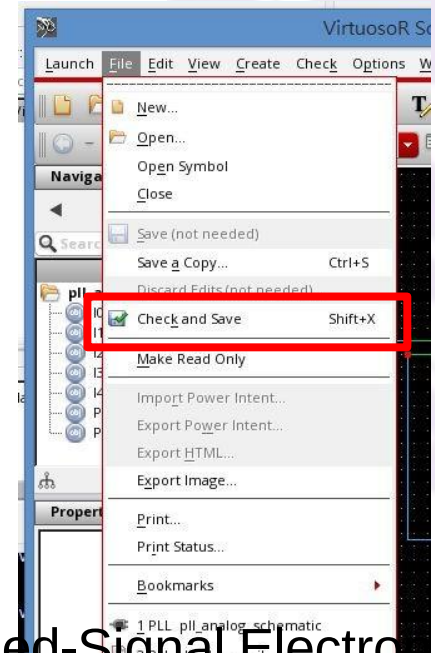
Add Instance: Create→instance (i)

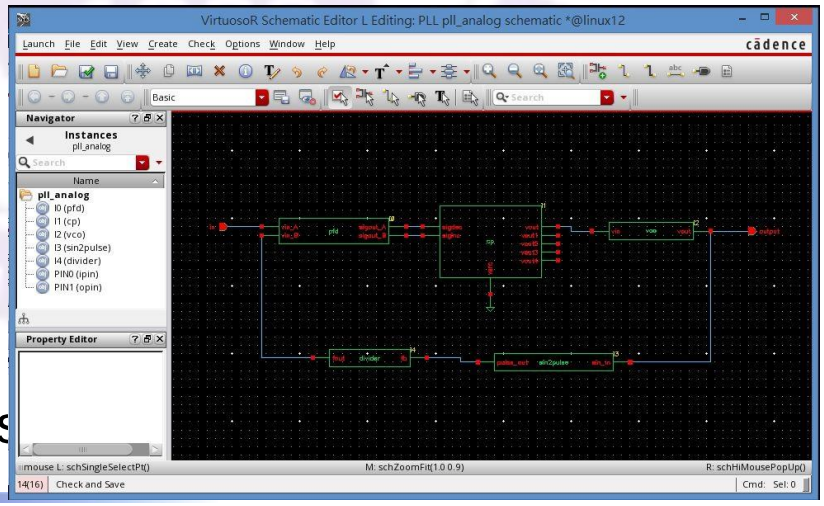


Add connection: Create → wire (w)

Add input/output: Create → pin (p)

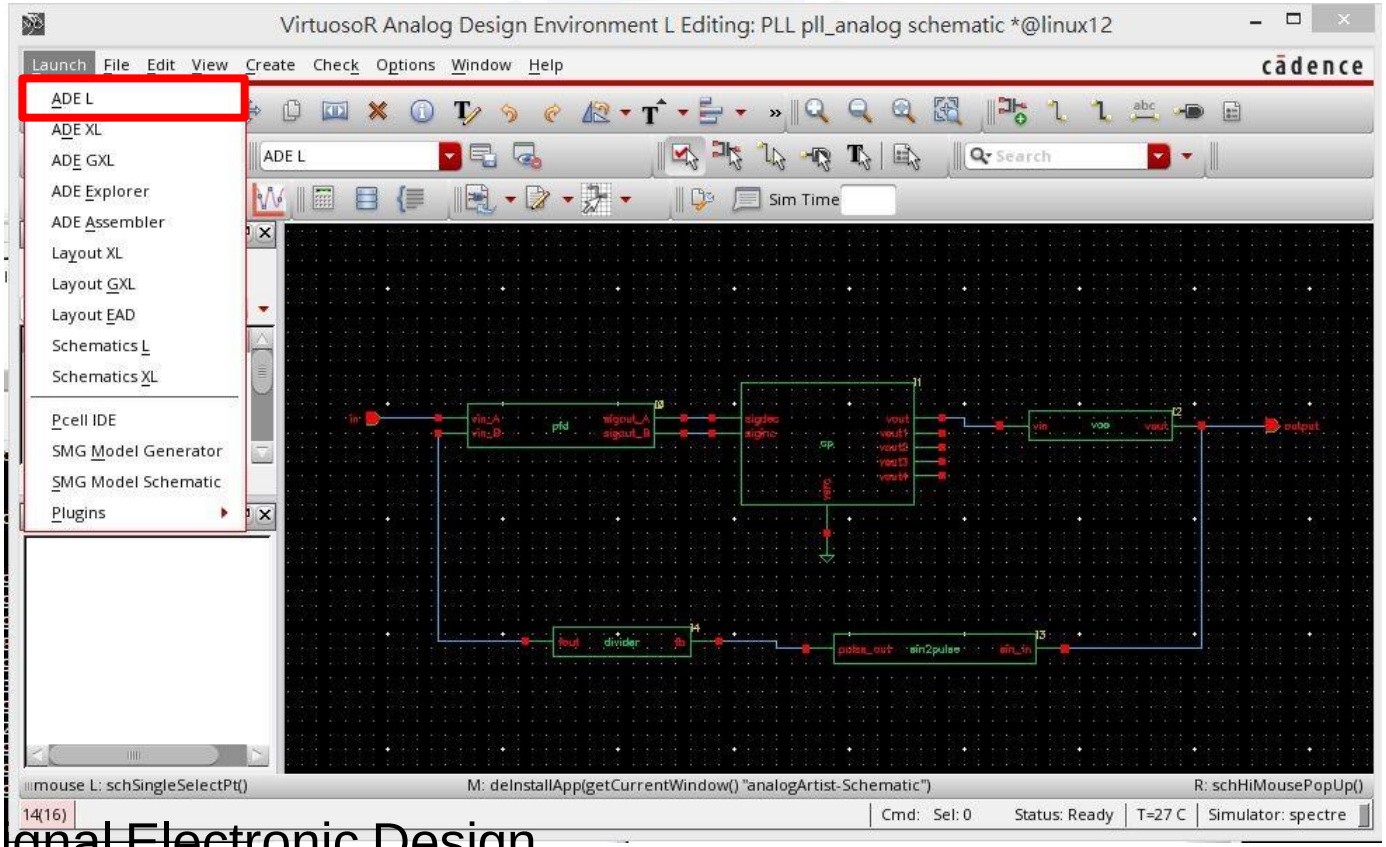
gnd in the“basic” library





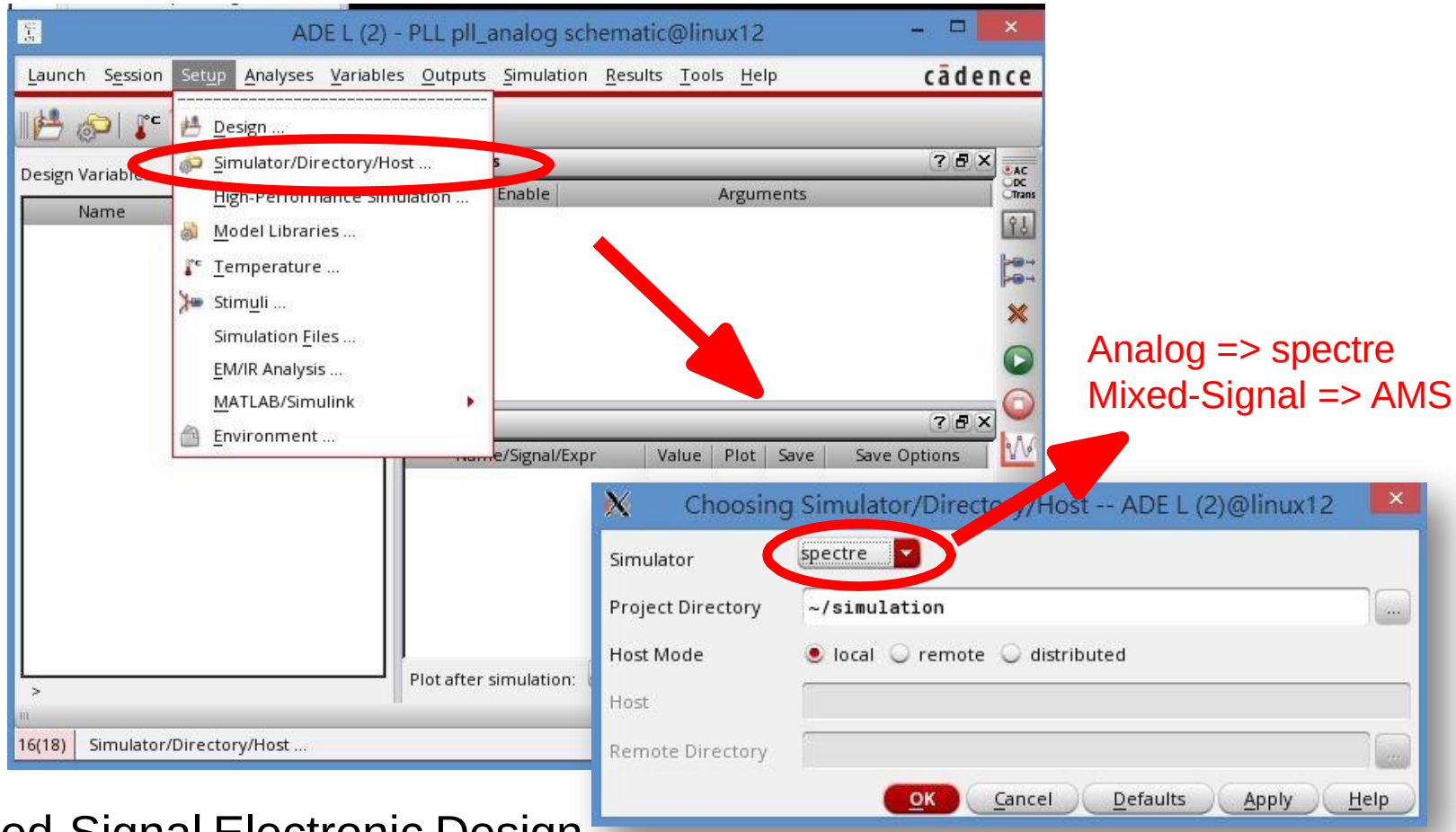
Simulation Environment

- Open Analog Design Environment (ADE) in schematic editing window

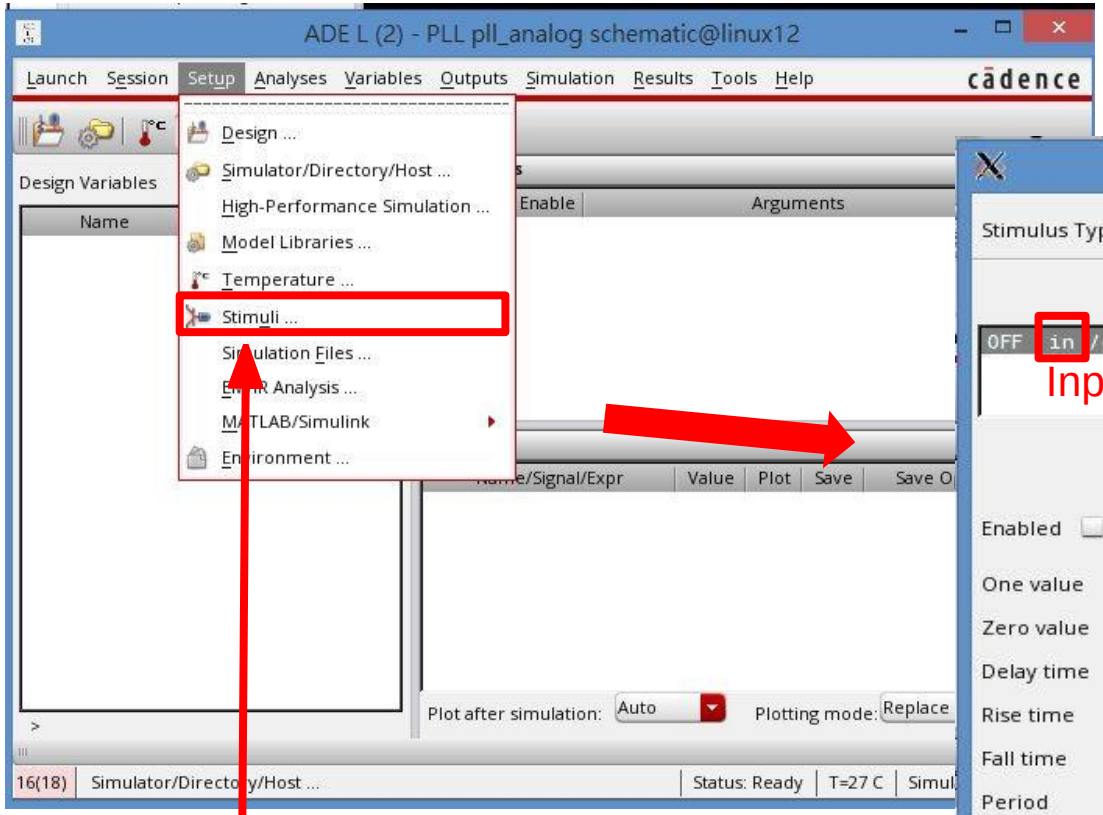


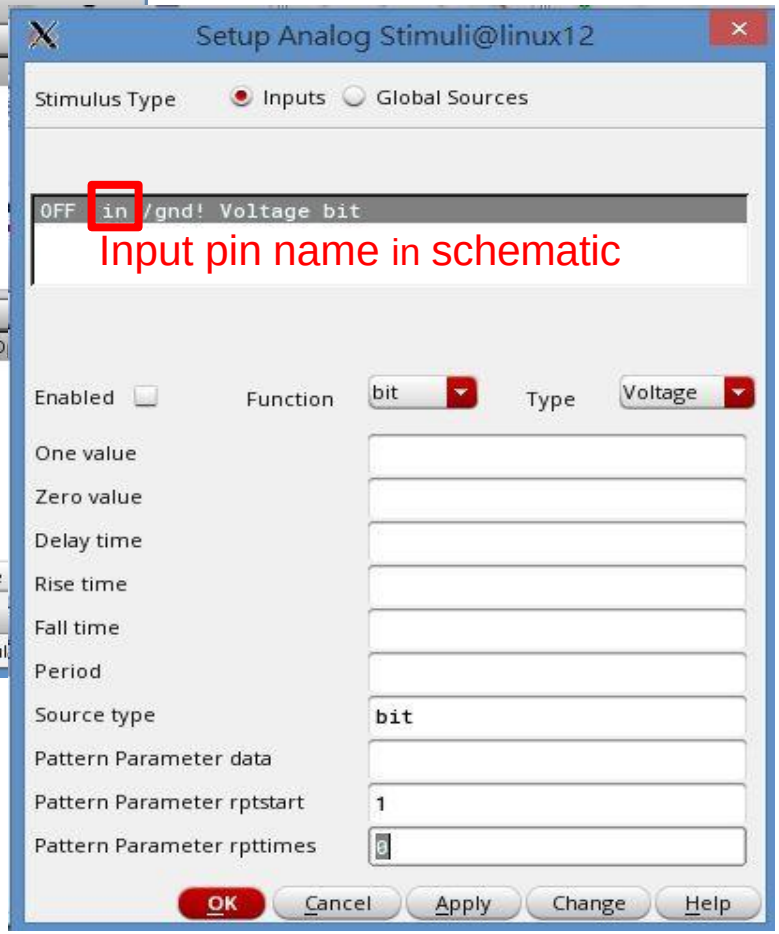
Simulation Environment

- Through Setup -> Simulator/Directory/Host



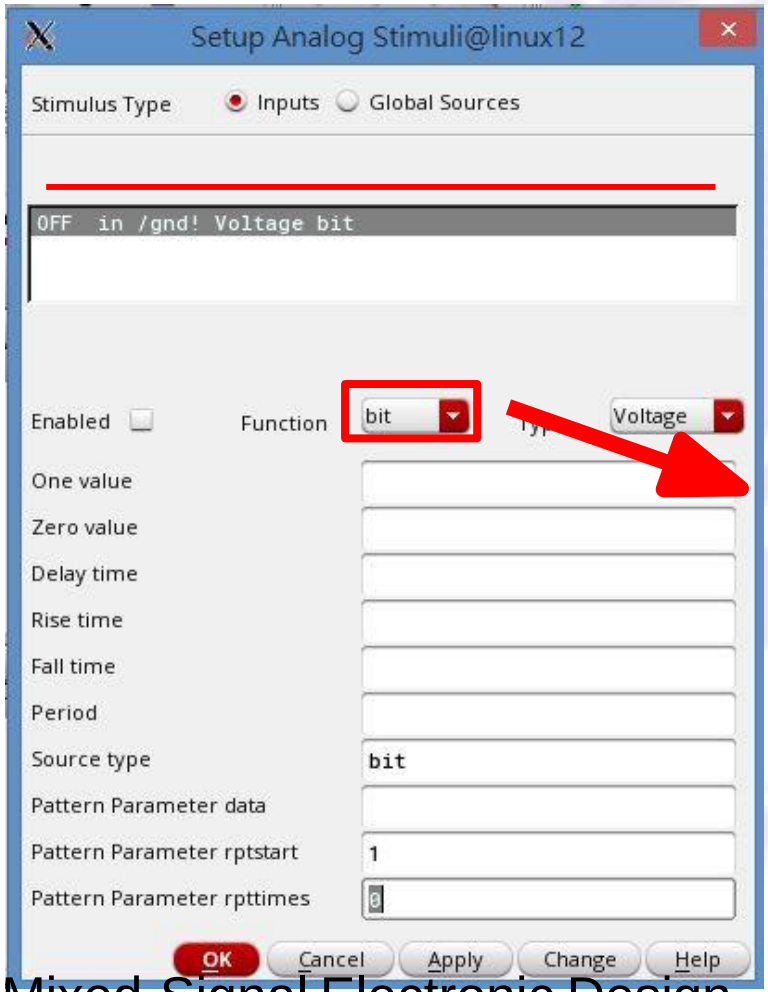
Give input information(1/2)



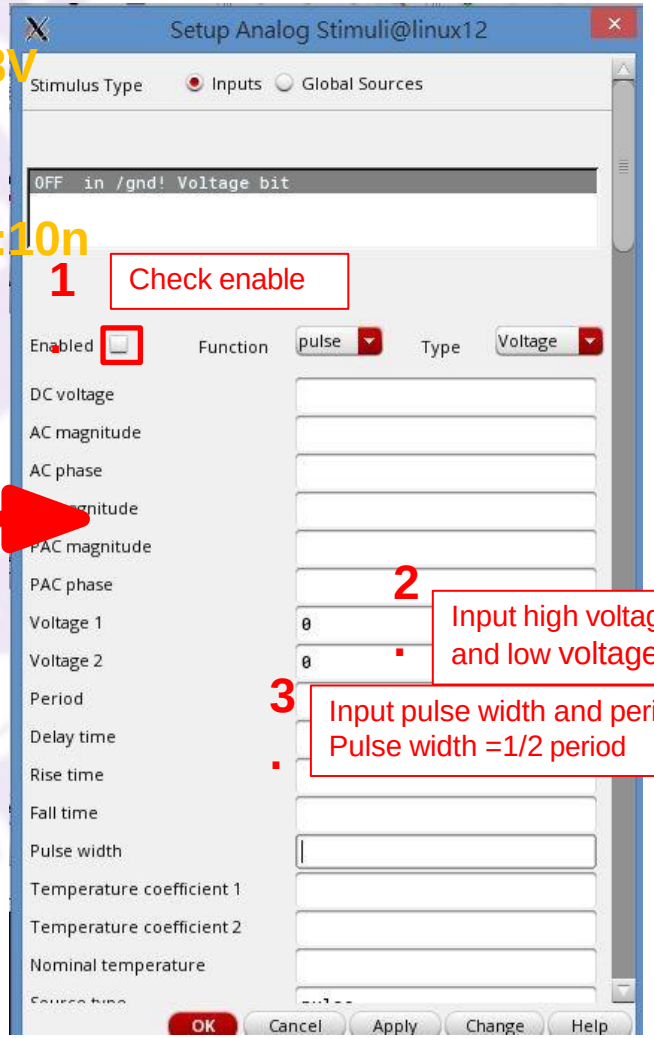


If choose spectre as simulator, there only has stimuli

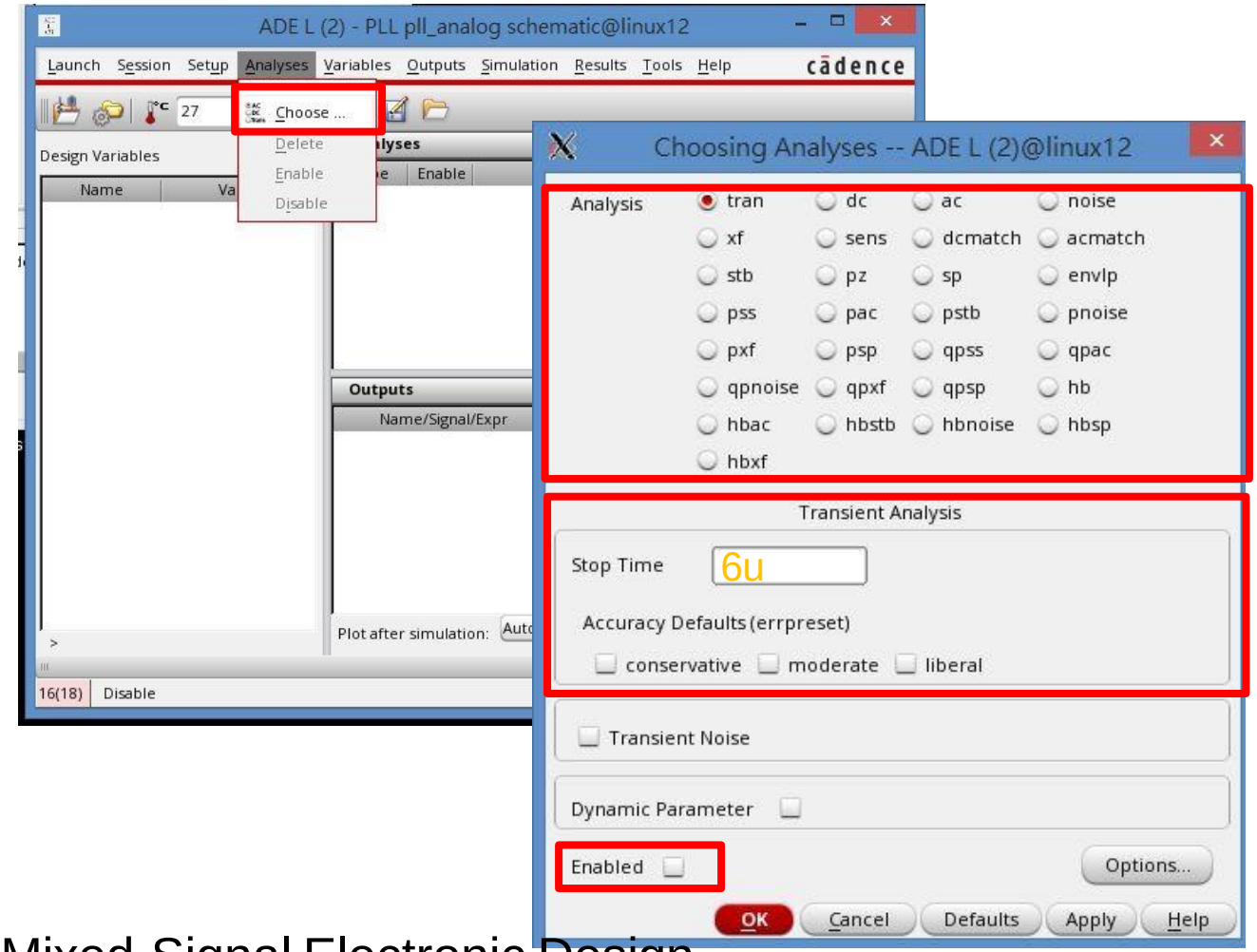
Give input information(2/2)



Voltage1:1.8V
Voltage2:0V
Period:20n
Pulse width:10n



Choose Analysis Type

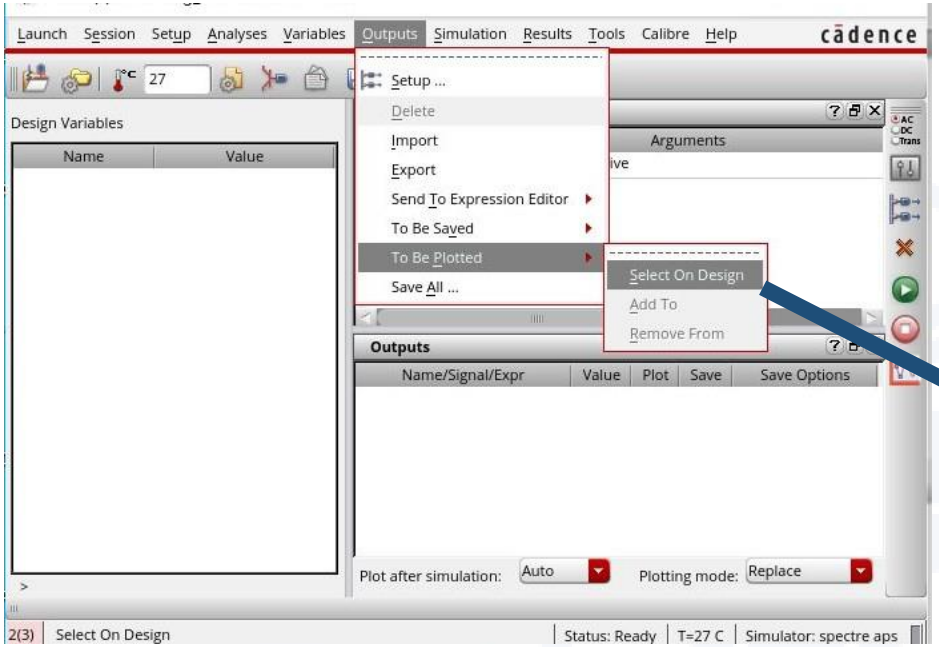


Choose tran analysis

Set the simulation time
and the accuracy flag

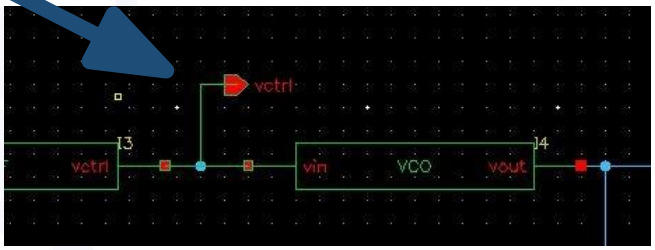
Check enable to simulation

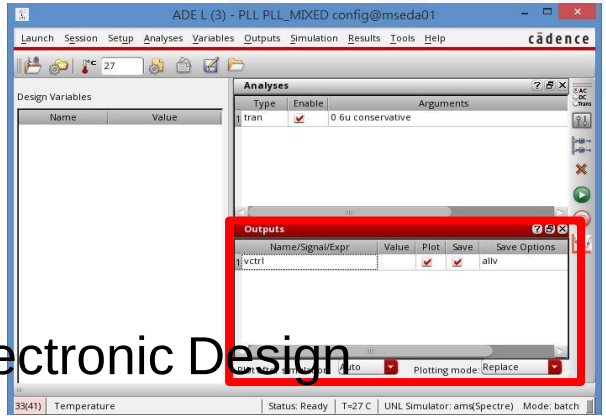
Save Output Nodes



To Be Plotted:

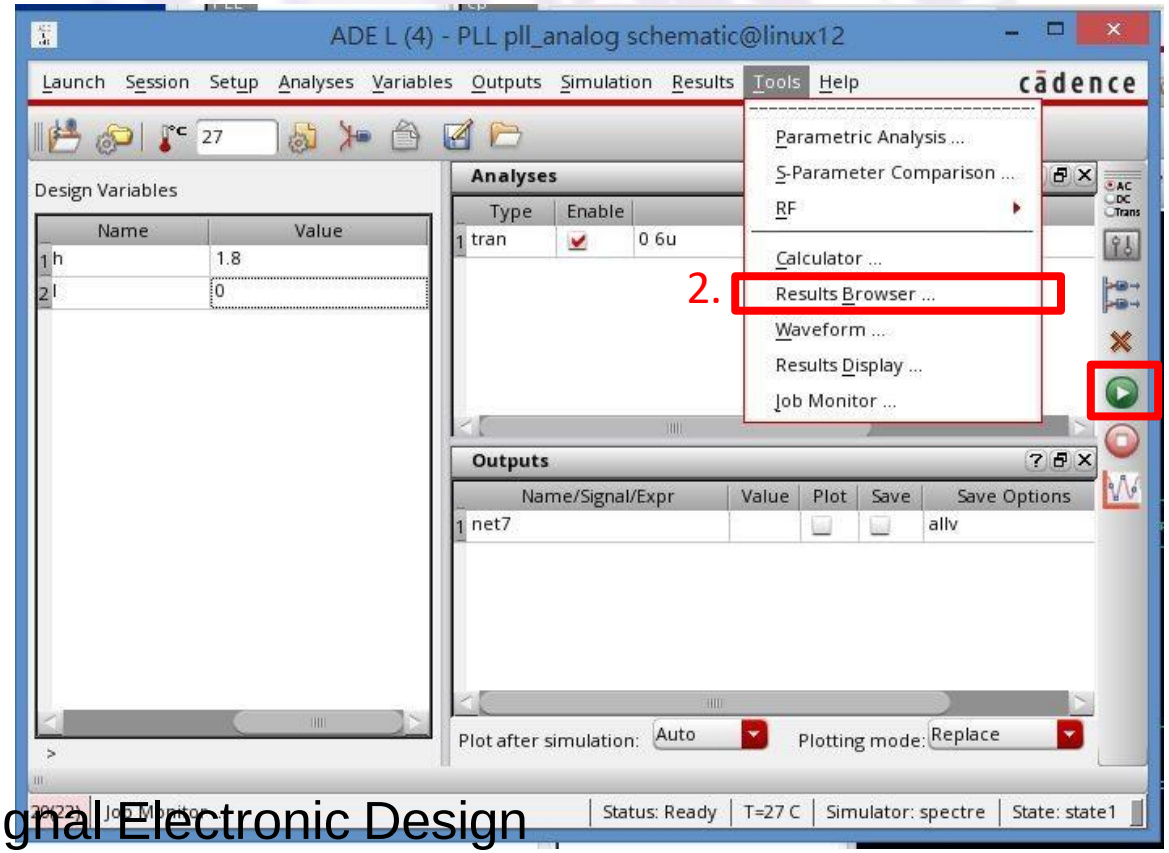
- Select on **line or node**
- Press **Esc** to cancel





Submit the Simulation

- Execute the simulation job with Run
- Tools → Results Brower

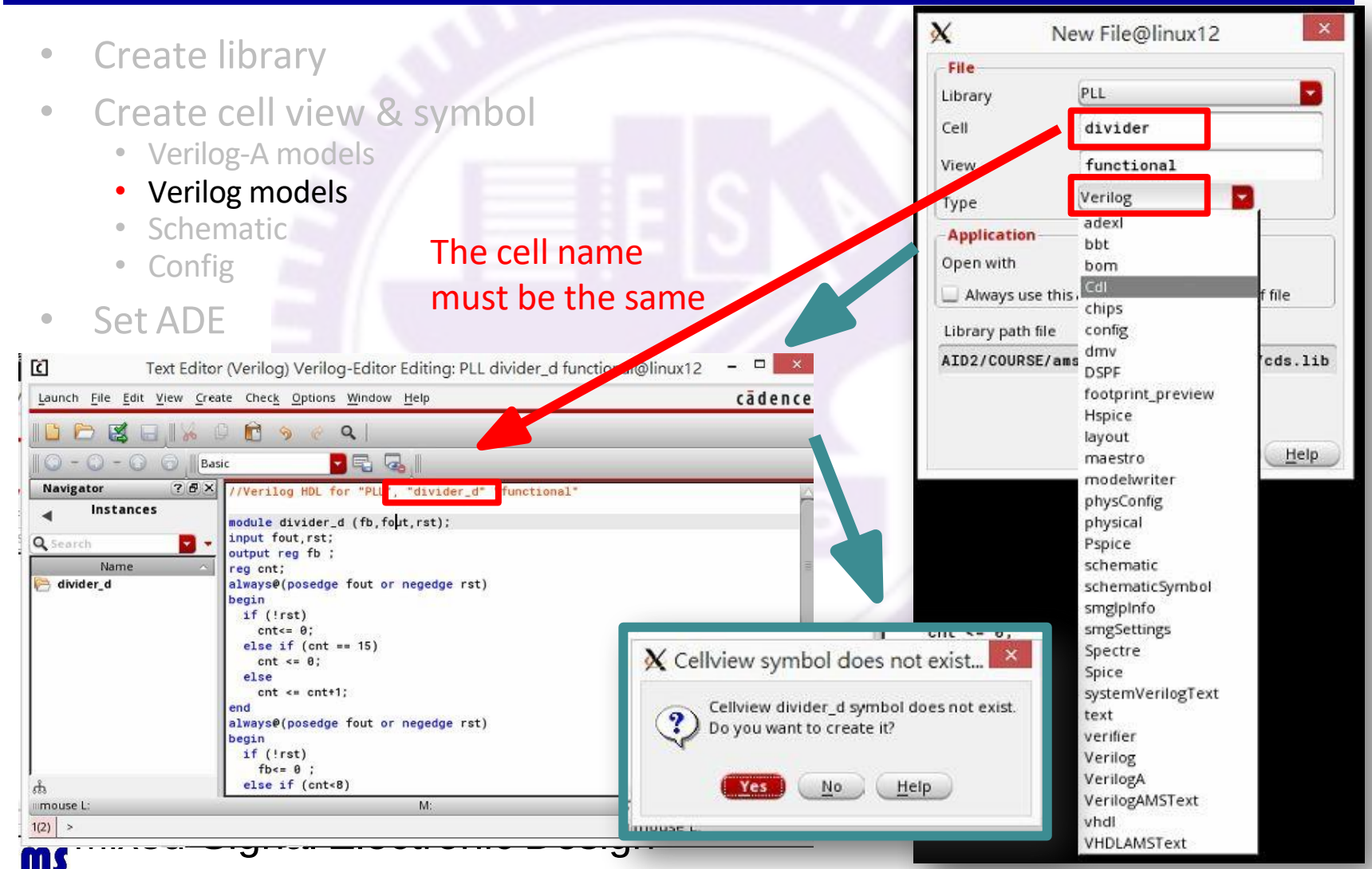


Outline

- Introduction to AMS
- AMS Simulation Setup
  - Analog Simulation with Verilog-A Model
  - Mixed-Signal Behavioral Model Simulation
    - Lab1: Mixed-Signal Model Simulation with AMS
    - Lab2: 4-bit ADC with Verilog-AMS

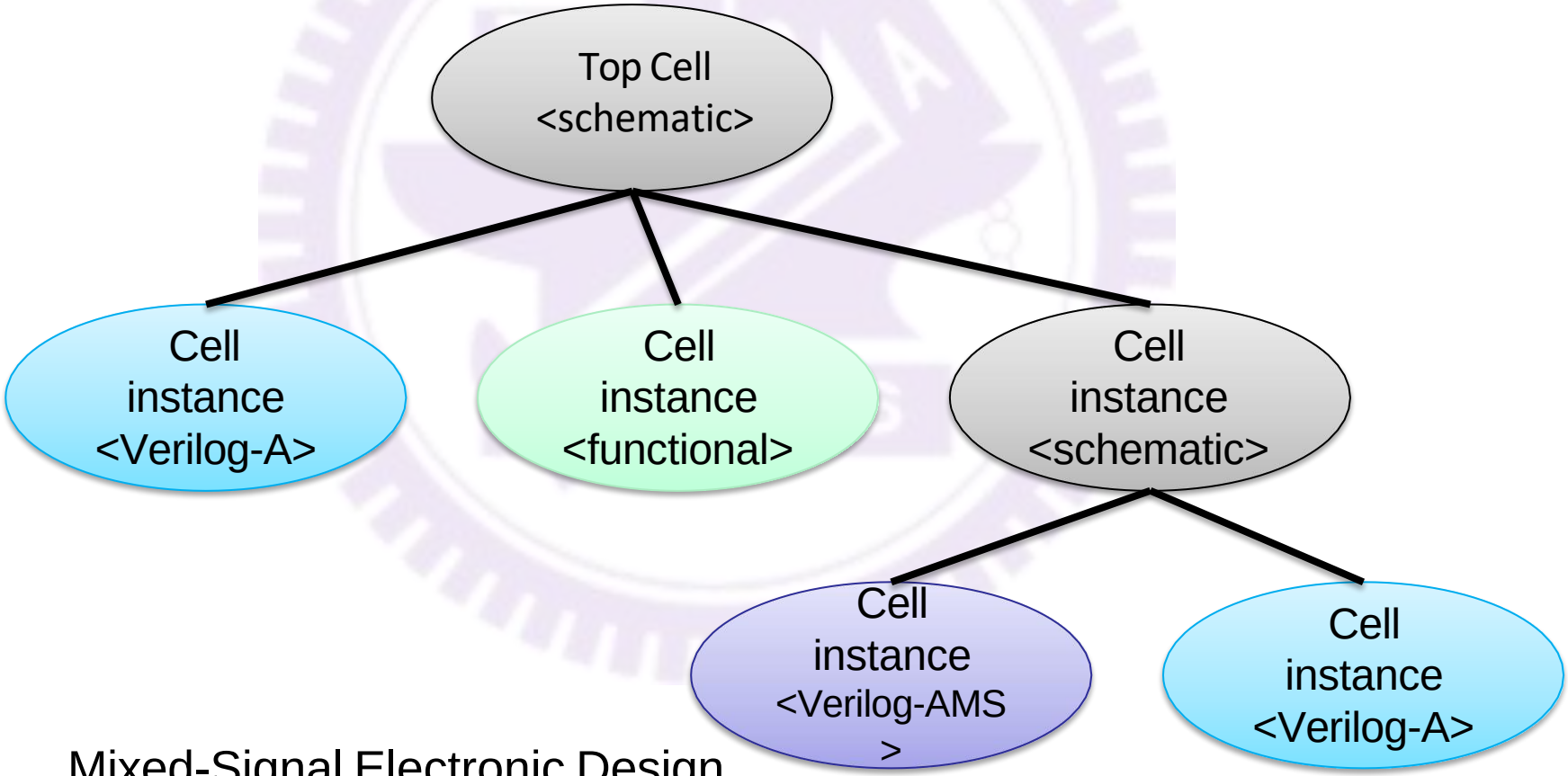
Create Verilog Cells

- Create library
- Create cell view & symbol
  - Verilog-A models
  - Verilog models
  - Schematic
  - Config
- Set ADE



Design Hierarchy – AMS

- Before creating the top schematic cell (add instance and connection), creating a config view for AMS simulation



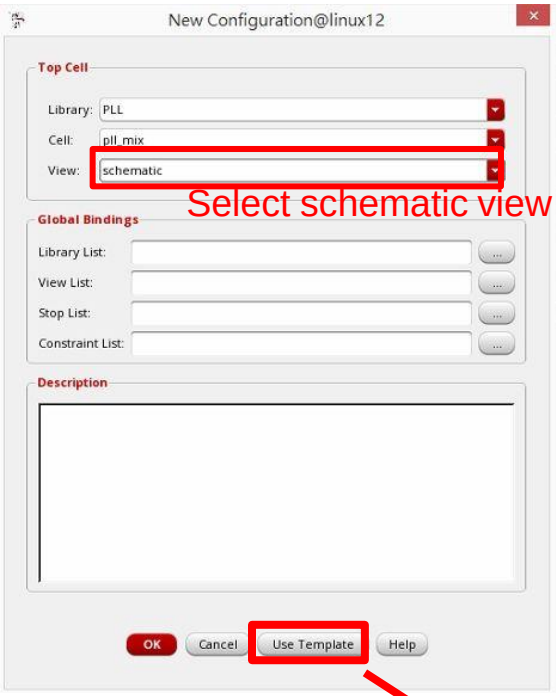
Create Config View for Simulation

- The mixed-signal simulation hierarchy is controlled by **Hierarchy-Editor**
  - It must have to be defined in the **config** cell view.

Cell name is top schematic cell for simulation

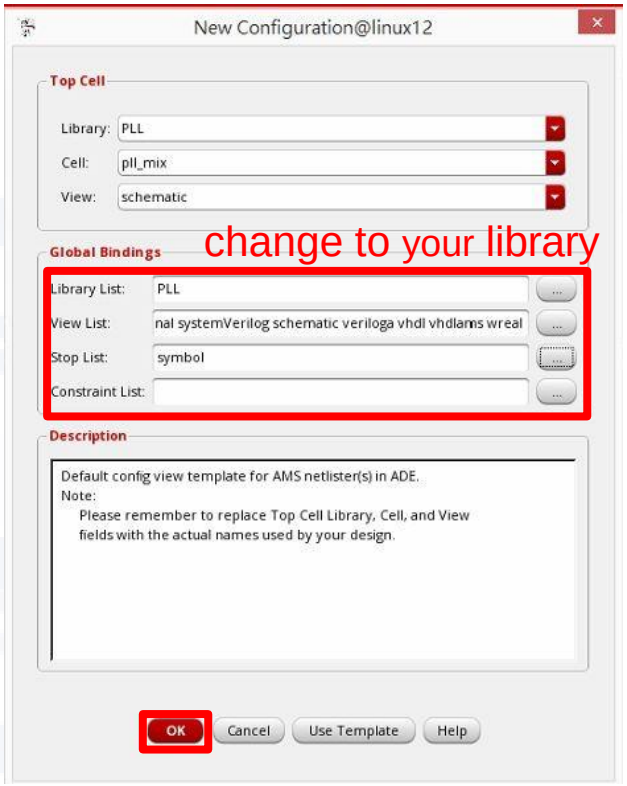
Choose **config** type

Set New Configuration



Select schematic view of top cell

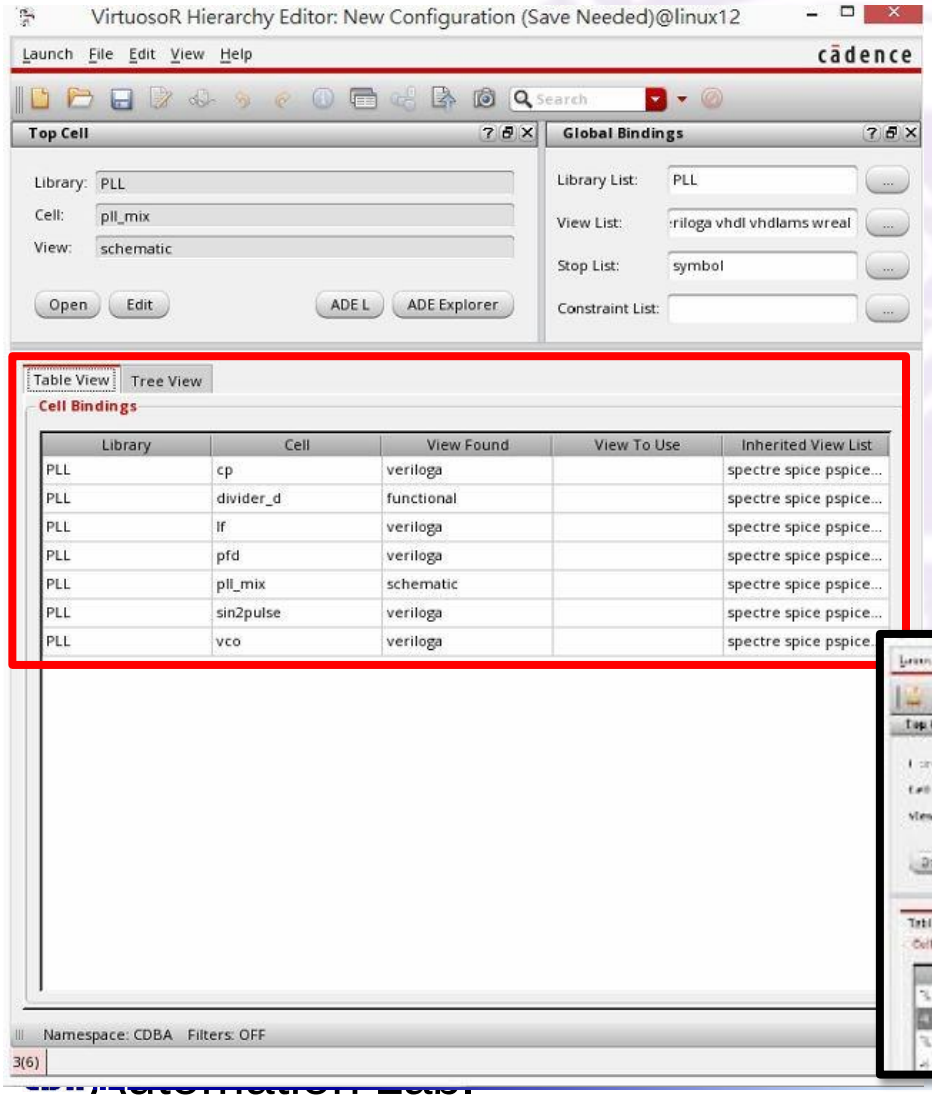
Bottom view



change to your library

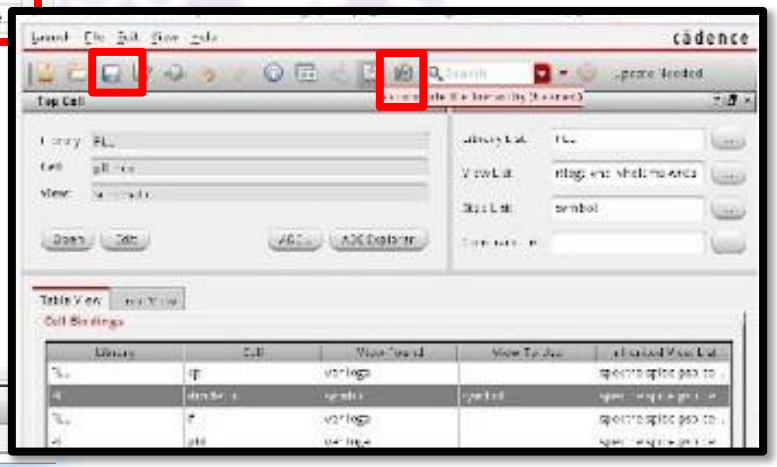
Select AMS simulator in template

Configuration Setting



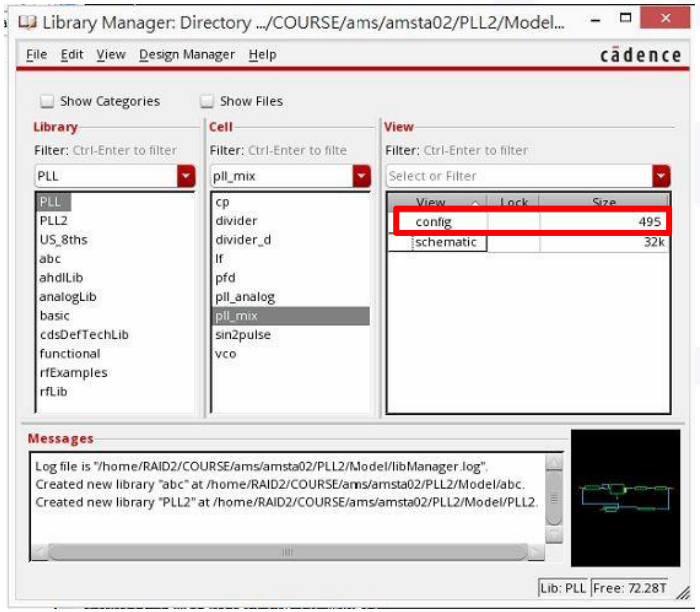
The hierarchy-editor will list all cells and views in the cell bindings

Must to save and recompute!

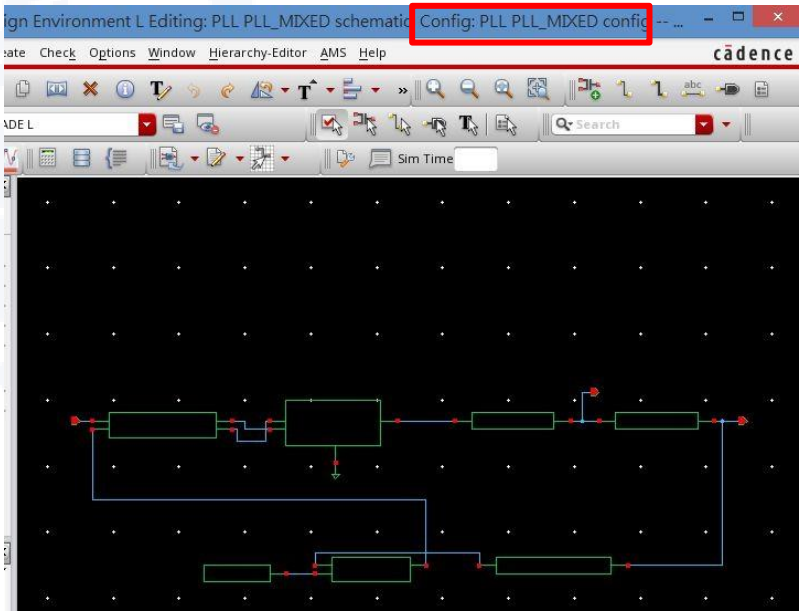


Open Simulation Tool

- Finish create config
  - Click config at library manager to open simulation tool
  - The simulation steps are the same as analog
    - Except give input information



Confirm the title has “config”



Digital Stimulus

- Create a behavioral or functional view for the stimulus block
  - The stimulus (Verilog) could be created to symbol view as the same procedure with digital cell

```
//Verilog HDL for "PLL", "stimulus_D" "functional"

`timescale 1ns/10ps
module stimulus_D (rst);
output rst;
reg rst;
initial begin
    rst=1'b0;
    #1 rst=1'b1;
end
endmodule
```

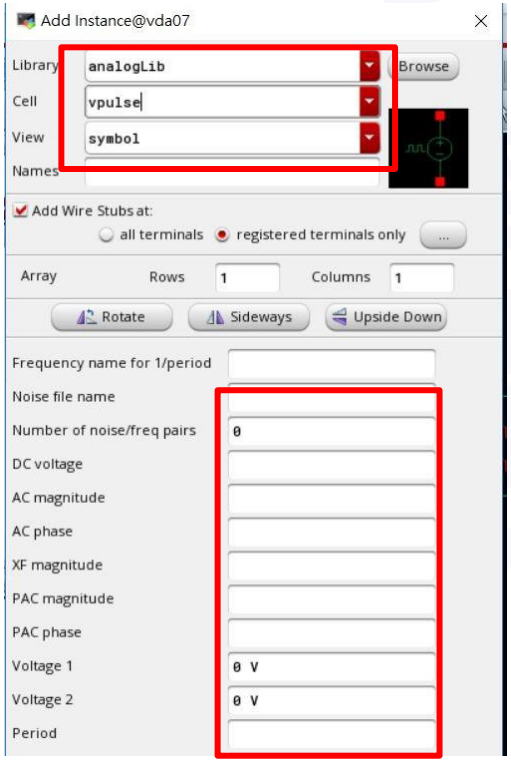
Analog Stimulus

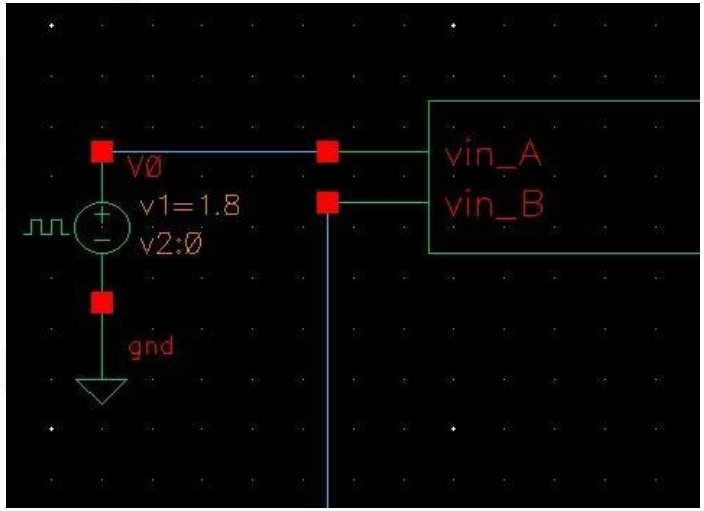
- The analog stimulus can be added as circuit instance

Add Instance

Choose analogLib & vpulse cell

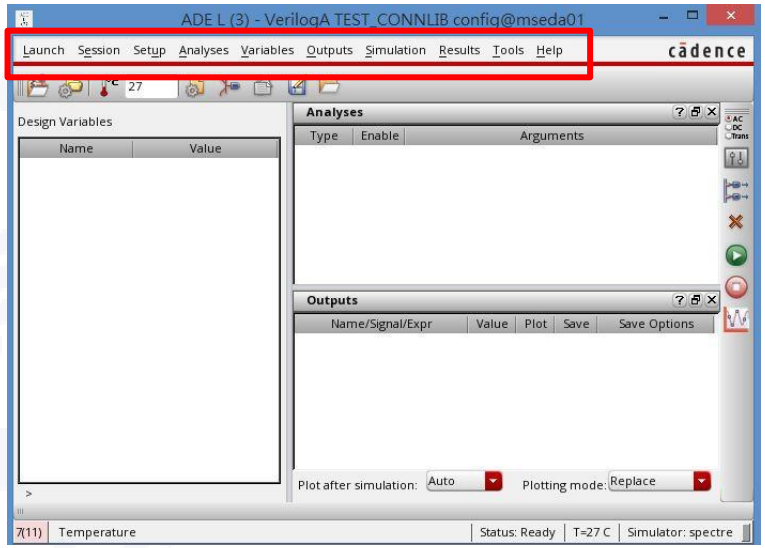
Set voltage & frequency





Analog Stimulus

1. Setup
   - ✓ Simulator choose AMS
2. Analysis
   - ✓ Tran analysis
   - ✓ Set simulation time and enable
3. Outputs
   - ✓ Save all or select on design
4. Run
5. Waveform viewer

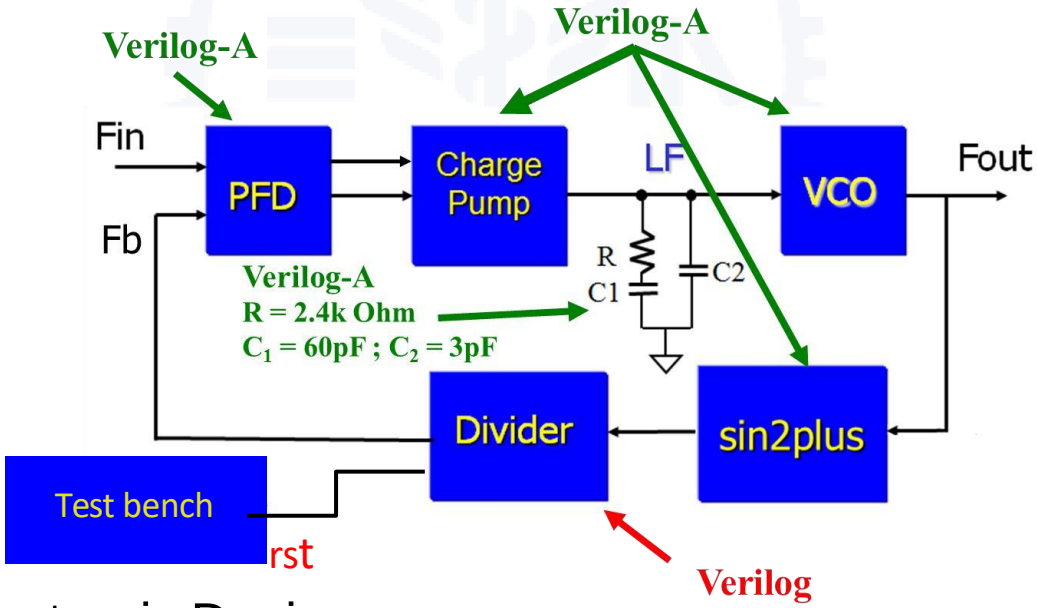


Outline

- Introduction to AMS
- AMS Simulation Setup
  - Analog Simulation with Verilog-A Model
  - Mixed-Signal Behavioral Model Simulation
    - Lab1: Mixed-Signal Model Simulation with AMS
    - Lab2: 4-bit ADC with Verilog-AMS

PLL Mixed-Signal Model

- All models are given
  - Analog model (Verilog-A): PFD, CP, LF, VCO, sin2plus
  - Digital model (Verilog): divider
- Import all models and create a testbench
  - Testbench : generate rst signal for divider



Additional Description

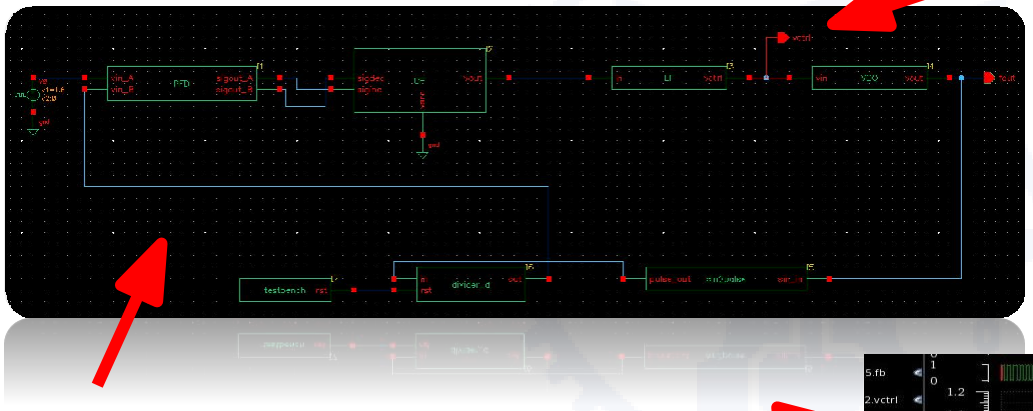
- Connection
  - **Connect *up* (*sigout_A*) signal of *PFD* to *siginc* signal of *CP***
  - **Connect *dn* (*sigout_B*) signal of *PFD* to *sigdec* signal of *CP***
  - **Connect *in* signal of *divider* to *sin2pulse***
  - **Charge pump(CP)**
    - Connect **vsrc** to **GND**
- VDD = 1.8V GND = 0V
- Simulator: **AMS**
- Input
  - **Fin=50M Hz, Adding voltage source instance**
- Simulation time ≧6μs
  - Related to the lock time of PLL

Results

- Show the waveform of vctrl, output of the divider and schematic view

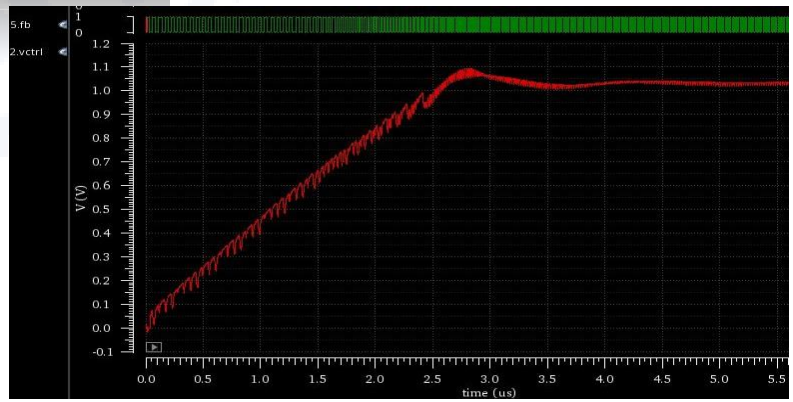
Top schematic cellview

Observe the waveform of Vctrl



Observe the output of divider.v

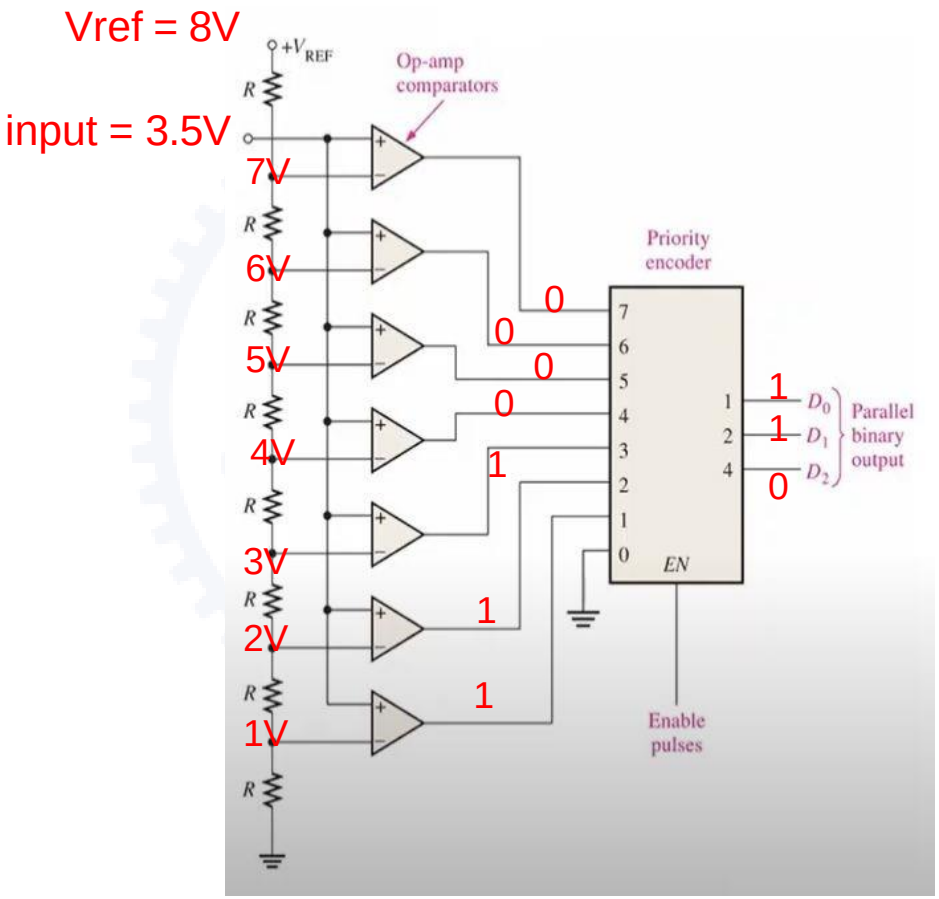
Open Results Brower



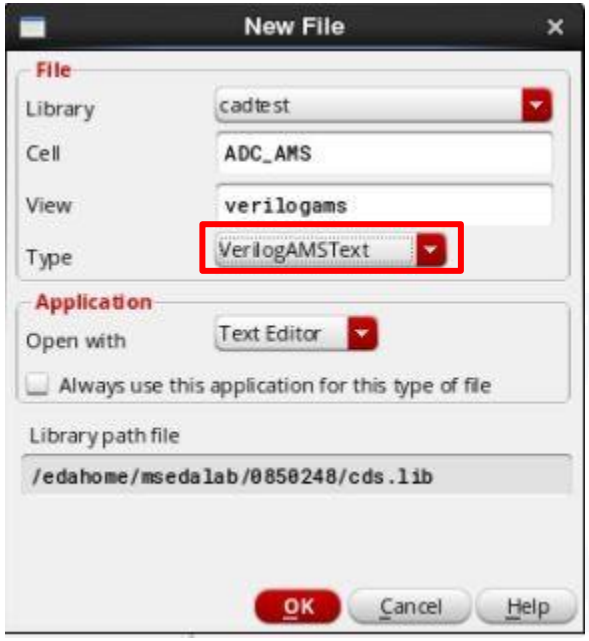
Outline

- Introduction to AMS
- AMS Simulation Setup
  - Analog Simulation with Verilog-A Model
  - Mixed-Signal Behavioral Model Simulation
    - Lab1: Mixed-Signal Model Simulation with AMS
    - Lab2: 4-bit ADC with Verilog-AMS

Description



Type : AMS




```
connectmodule a2d(i,o);
   parameter vdd = 1.0;
   ddiscrete   o;
   input       i;
   output      o;
   reg         o;
   electrical  i;
   always begin   @(cross(V(i) - vdd/2,+1))o = 1; end
   always begin   @(cross(V(i) - vdd/2,-1))o = 0; end
endmodule
```

Support output reg

Parameters of ADC Model

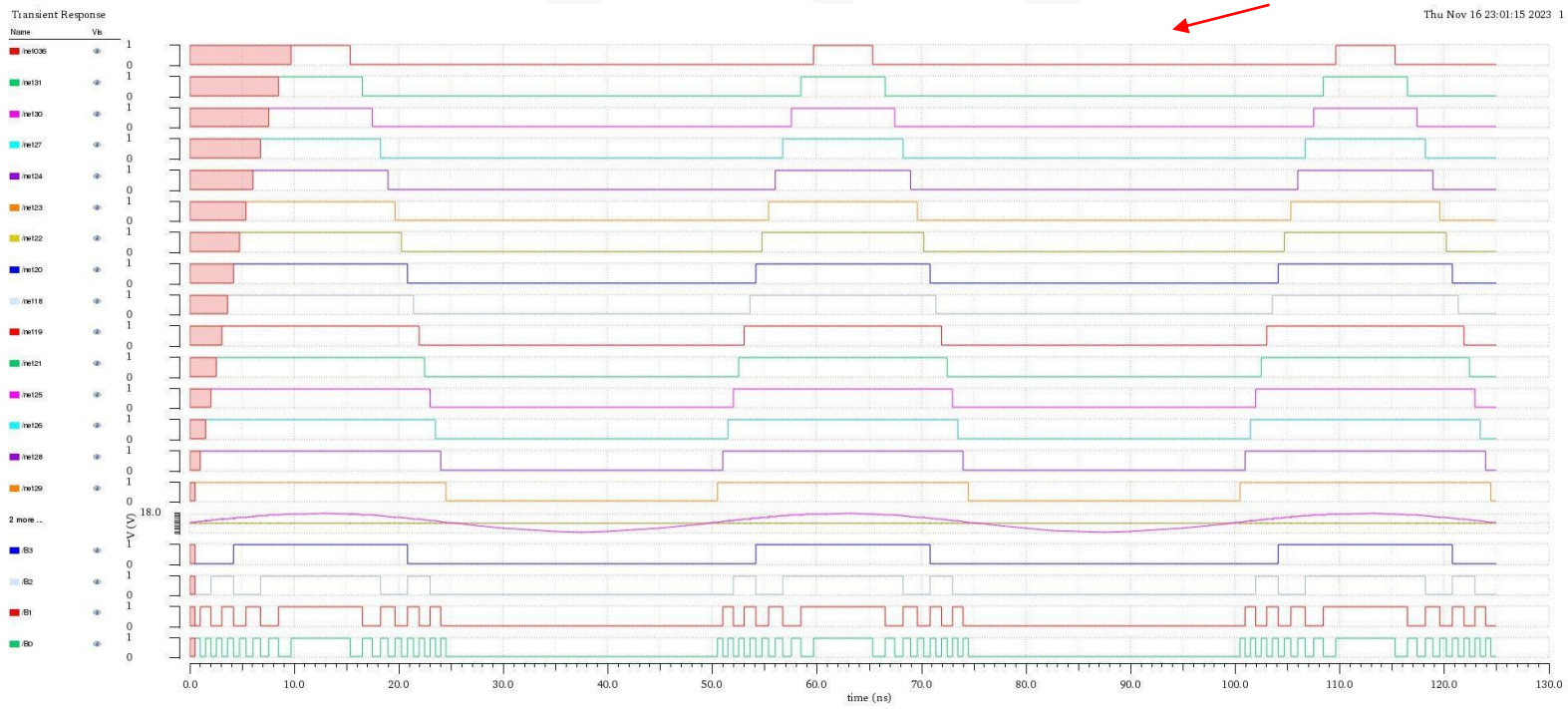
- Input : sin wave
- Amplitude : 16V, Frequency : 20M
- Output : 4 bit output
- R=0.5k, Vref=16V
- Analysis Type : tran, 125n
- Simulator: **AMS**

B3,B2,B1,B0 (bit)	Input (V)
0000	0
0001	1V
0010	2V
0011	3V
0100	4V
0101	5V
0110	6V
0111	7V
1000	8V
1001	9V
1010	10V
1011	11V
1100	12V
1101	13V
1110	14V
1111	15V

Results

- Hand in:
  - The waveform of outputs of comparators, encoder and input
  - The waveview need to clear shows the signal name

This just an example, please make sure the signal name is clear to see



Hand in

- Please upload a compressed file (studentID.zip) includes:
  - **Programming files** (Verilog and Verilog-A files)
    - Lab1 (35%): Stimuli (testbench.v)
    - Lab2 (45%): Encoder file(.vams or .v), comparator file(.vams or .va)
  - **Video files** (.mp4)**:** show the simulated process of lab 1&2 (5%)
    - From running ADE L to waveform creating
  - **Mini report** (studentID.pdf)
    - Simulation waveforms for two labs (6%)
      - Lab1: Mixed-signal simulation (Vctrl & output of the divider)
      - Lab2: ADC (outputs of comparators, encoder and input)
    - Two schematic cellviews (6%)
    - What you have learned from this course (2%)
    - Suggestion for this course (1%)
- **Deadline: 2024/8/31 (Sat.) 11:59 a.m.**