National Chiao Tung University
Institute of Electronic Engineering

NCTU Mixed-Signal Electronic Design Automation Lab. 2018

# CAD HW4 - Modeling Mixed-Signal System and Simulating with AMS

Teaching Assistant: Tung Lin

Date: 12/13, 2022

Lab: ED-413

# Outline

- Introduction to AMS
- AMS Simulation Setup
  - Analog Simulation with Verilog-A Model
  - Mixed-Signal Behavioral Model Simulation
    - Lab1: Analog Model Simulation with AMS
    - Lab2: Mixed-Signal Model Simulation with AMS
    - Lab3: 2-bit ADC with Verilog-AMS

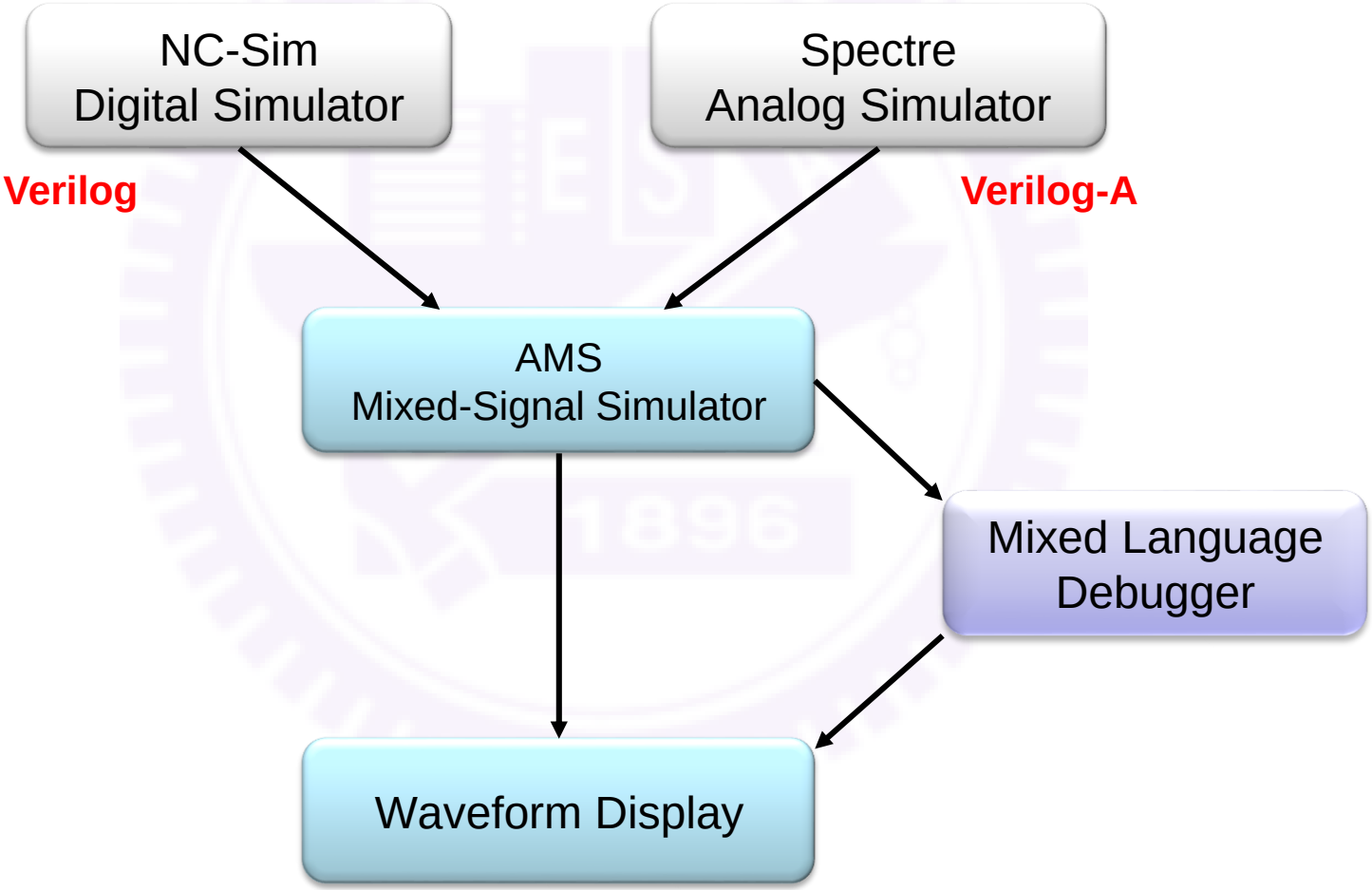
# Outline

- Introduction to AMS
- AMS Simulation Setup
  - Analog Simulation with Verilog-A Model
  - Mixed-Signal Behavioral Model Simulation
    - Lab1: Analog Model Simulation with AMS
    - Lab2: Mixed-Signal Model Simulation with AMS
    - Lab3: 2-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: Analog Model Simulation with AMS
    - Lab2: Mixed-Signal Model Simulation with AMS
    - Lab3: 2-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

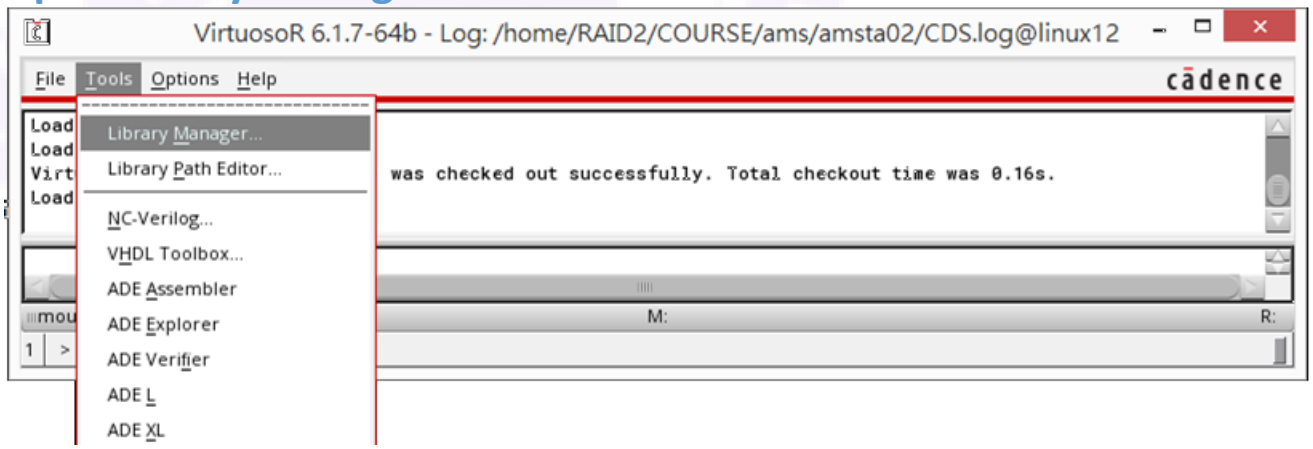
# Simulation Flow

Step 1. > virtuoso &

Step 2. Open library manager

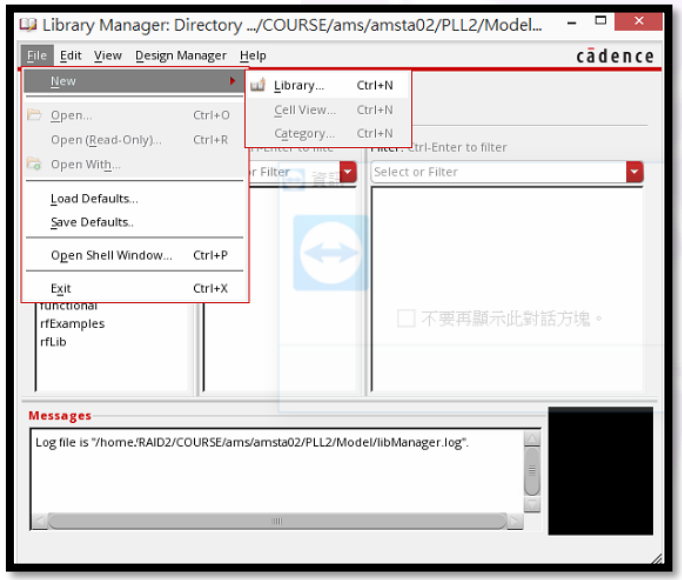
- Create library
- Create cell view & symbol
  - Verilog-A models
  - Verilog models
  - Schematic
  - Config
- Set ADE

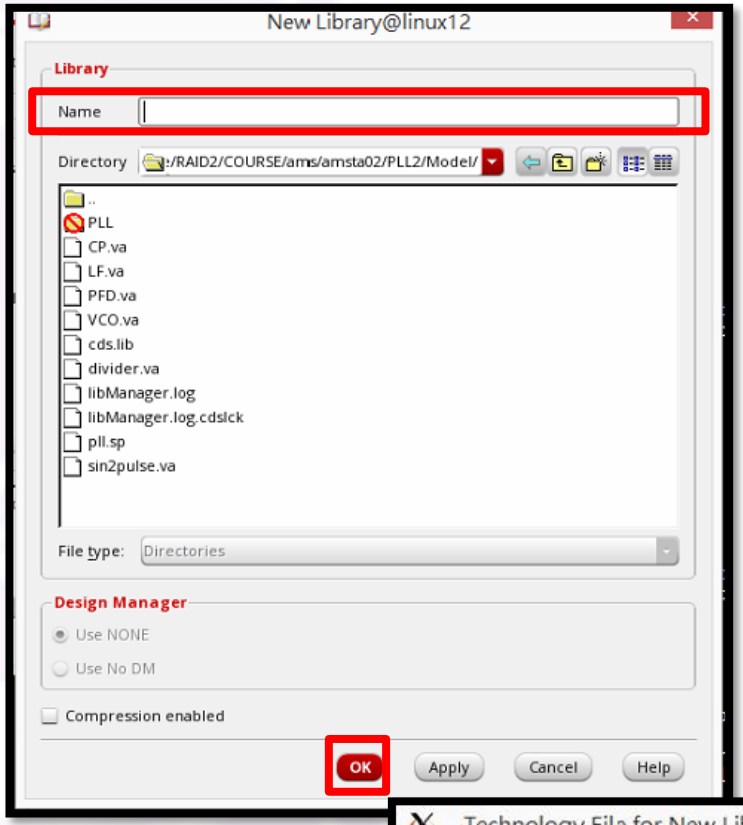
Open library manager

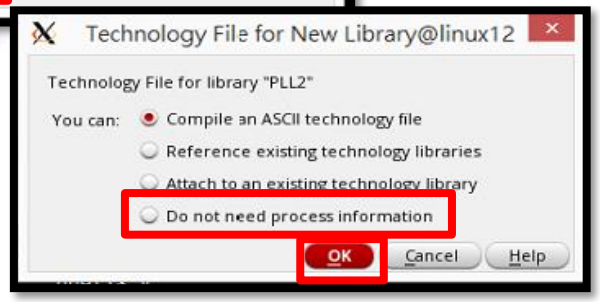


# Simulation Flow

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



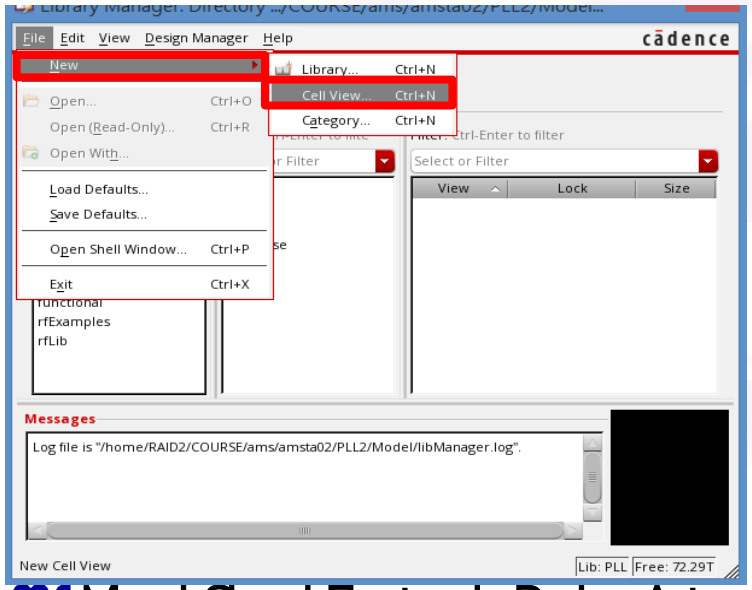



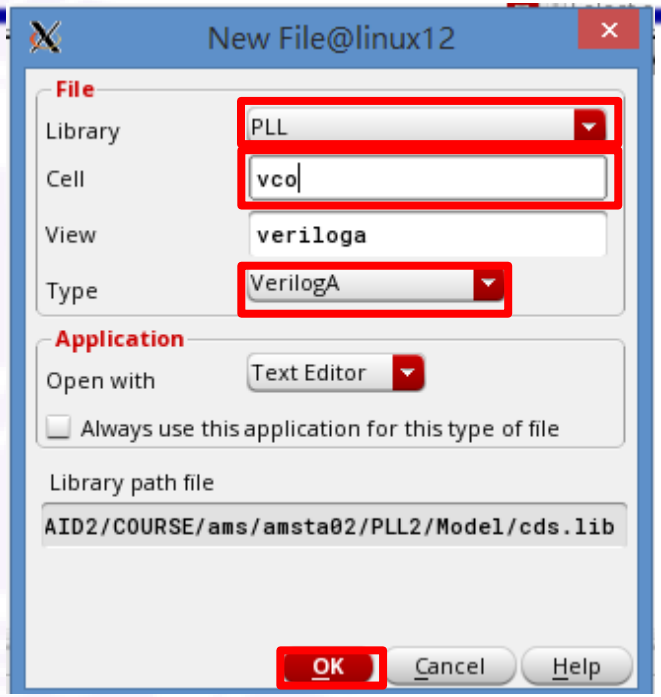



# Create Verilog-A Cells

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

Create a new cell view



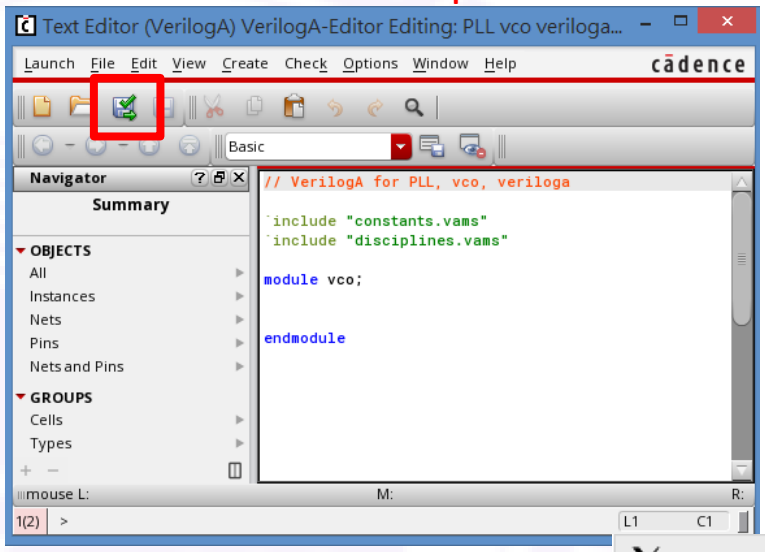



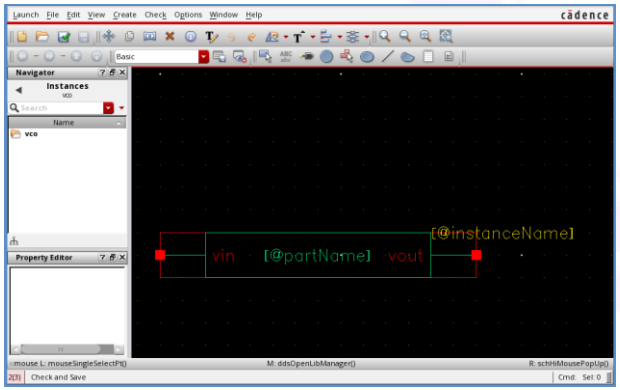

1. Choose your library
2. Input the name on the Cell Name column
3. Choose the **VerilogA** type for Analog model
4. OK

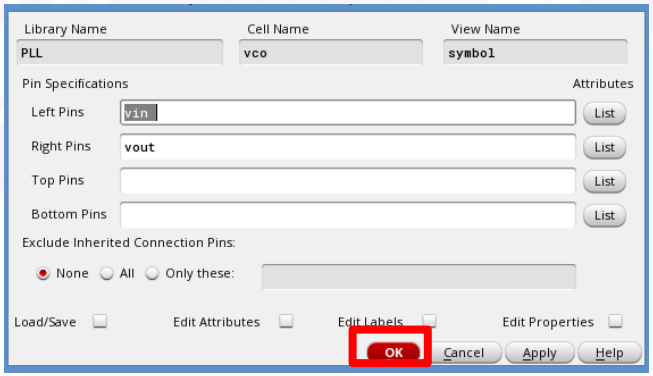
# Designing with Verilog-A

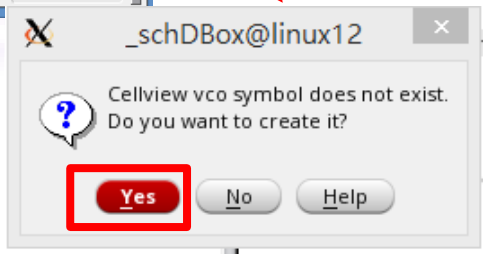
- Create library
- Create cell view & symbol
  - Verilog-A models
  - Verilog models
  - Schematic
  - Config
- Set ADE

Enter behavioral description



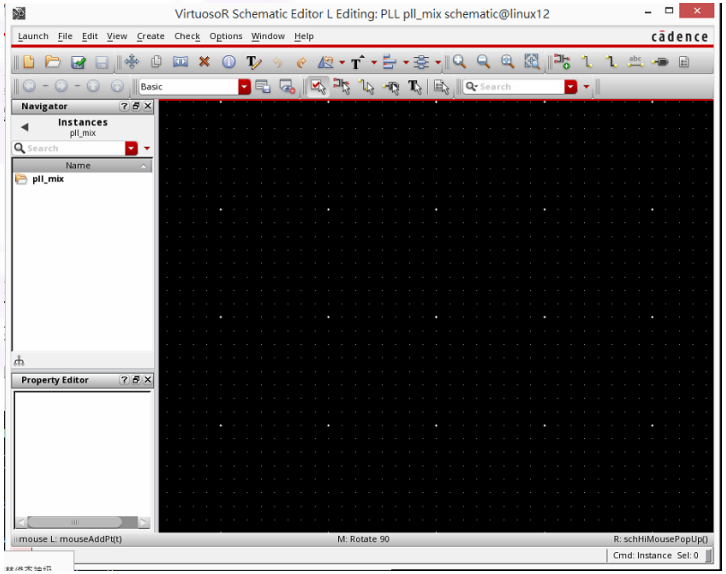



# Create Top Cell - Schematic

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

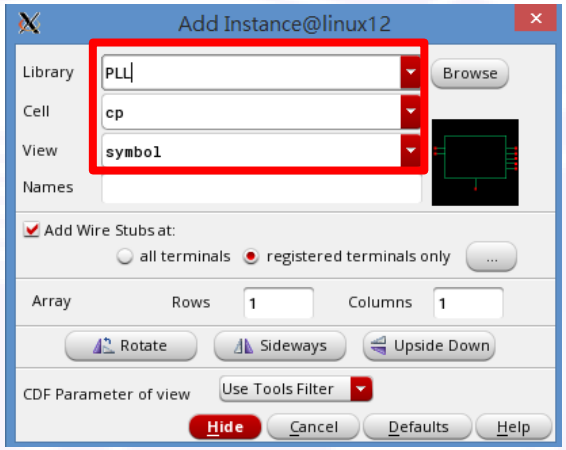
Input the name on the Cell Name column and choose the Schematic

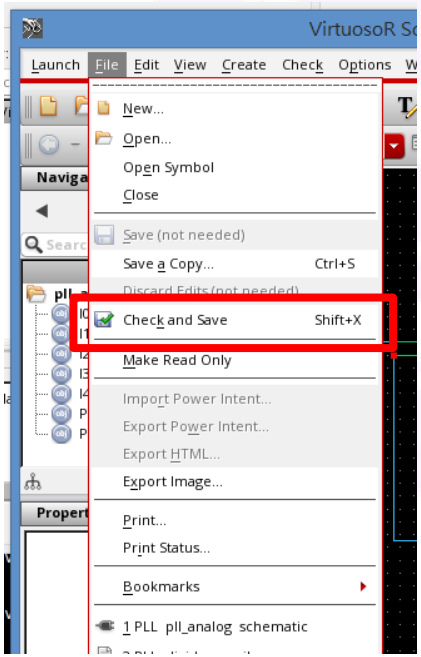



# Create Top Cell - Schematic

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

Add Instance: Create→instance (i)

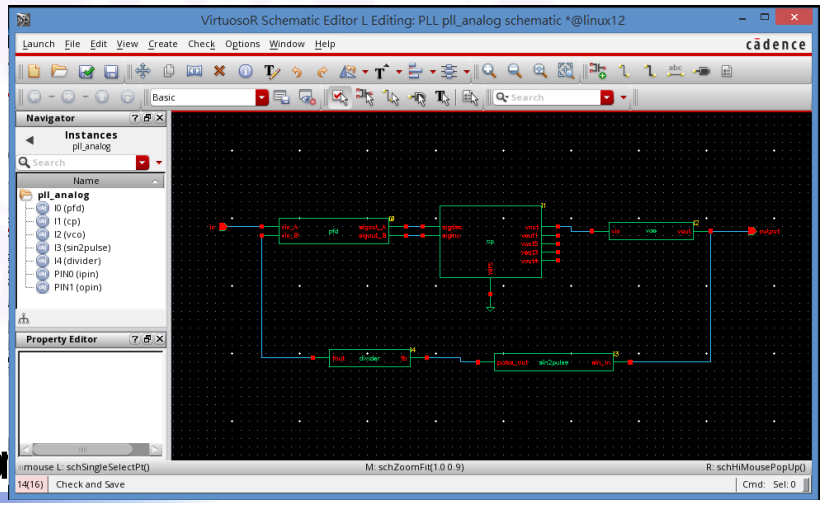



Add connection: Create → wire (w)

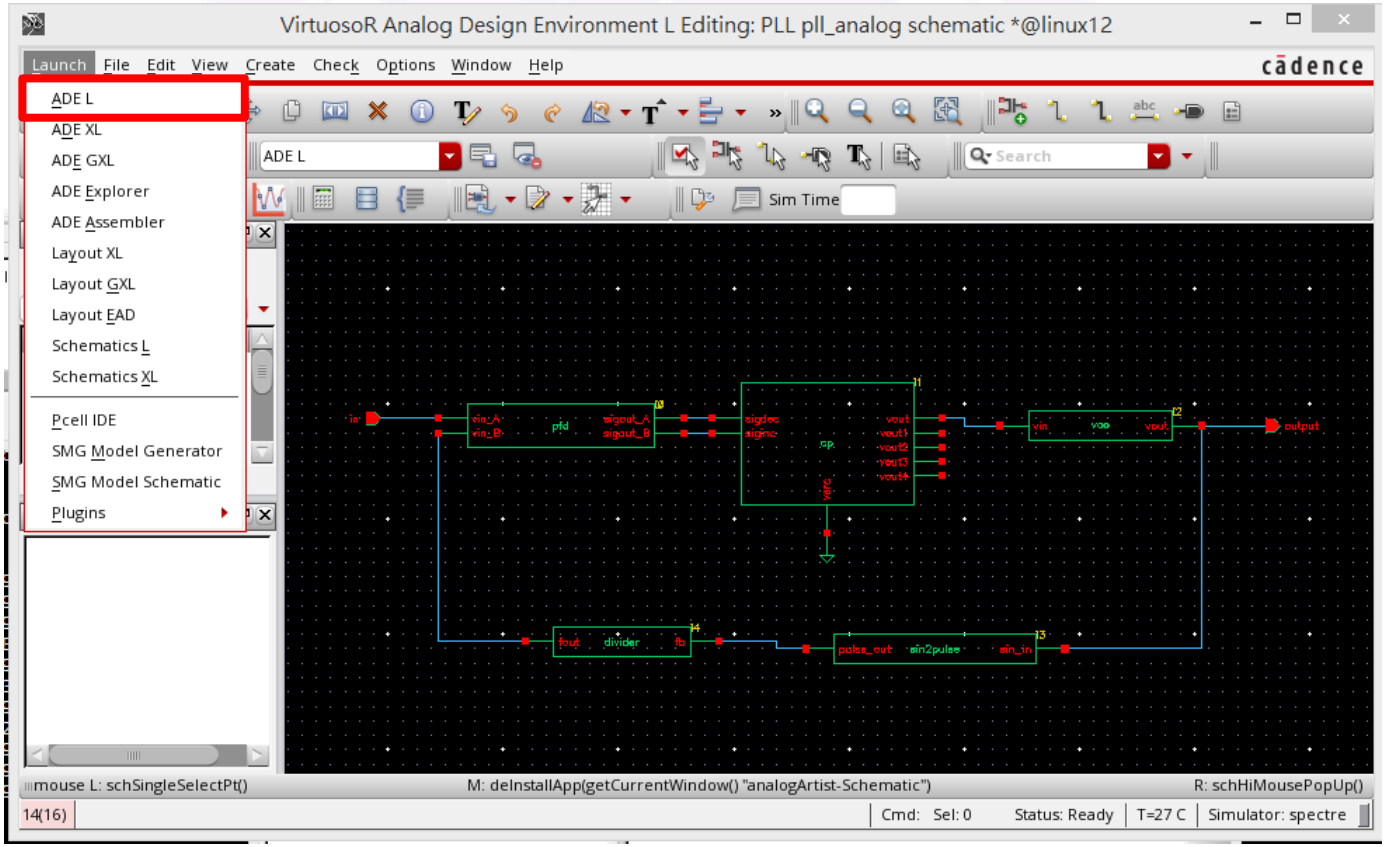
Add input/output: Create → pin (p)

gnd in the“basic” library

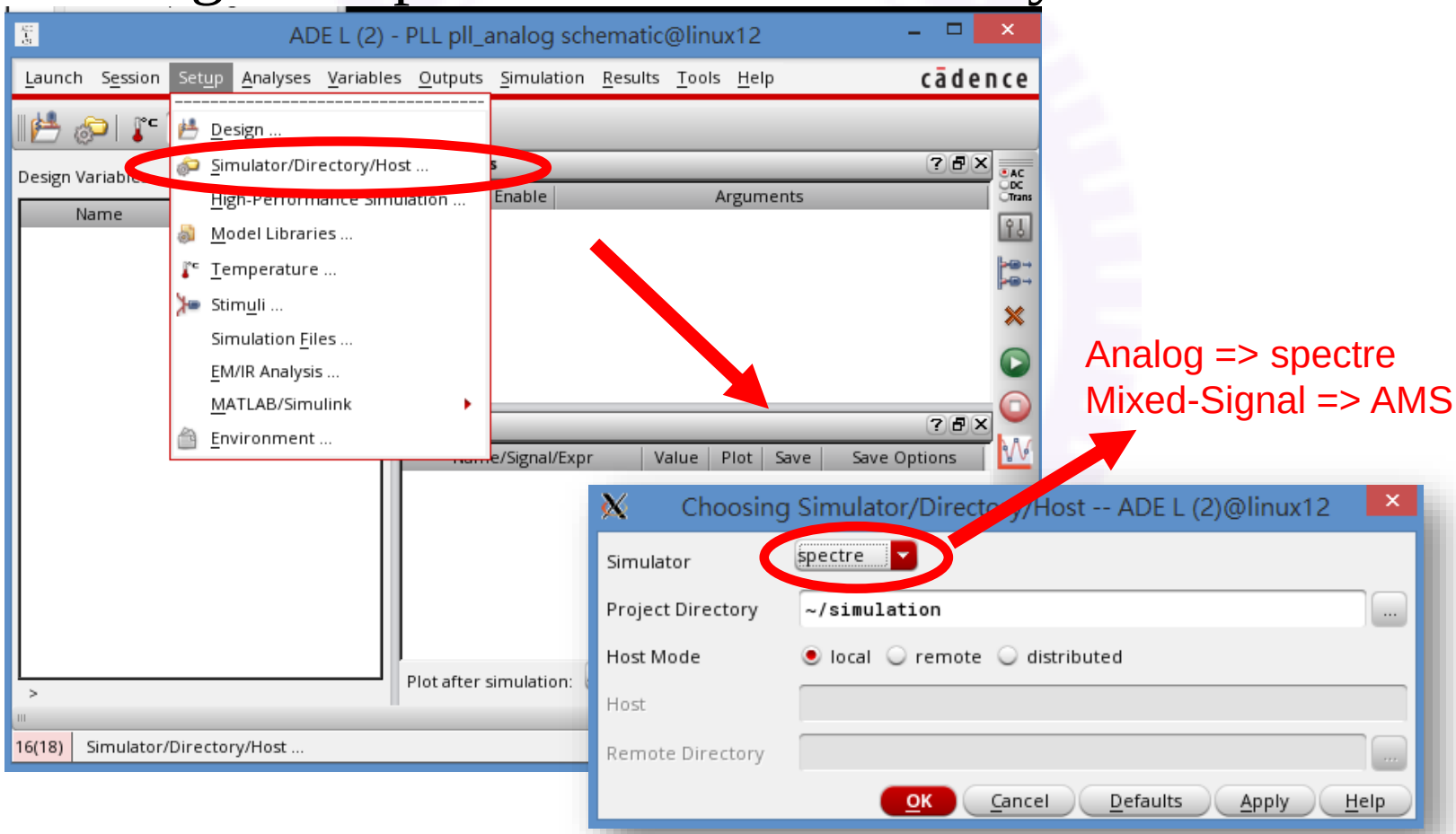


# Simulation Environment

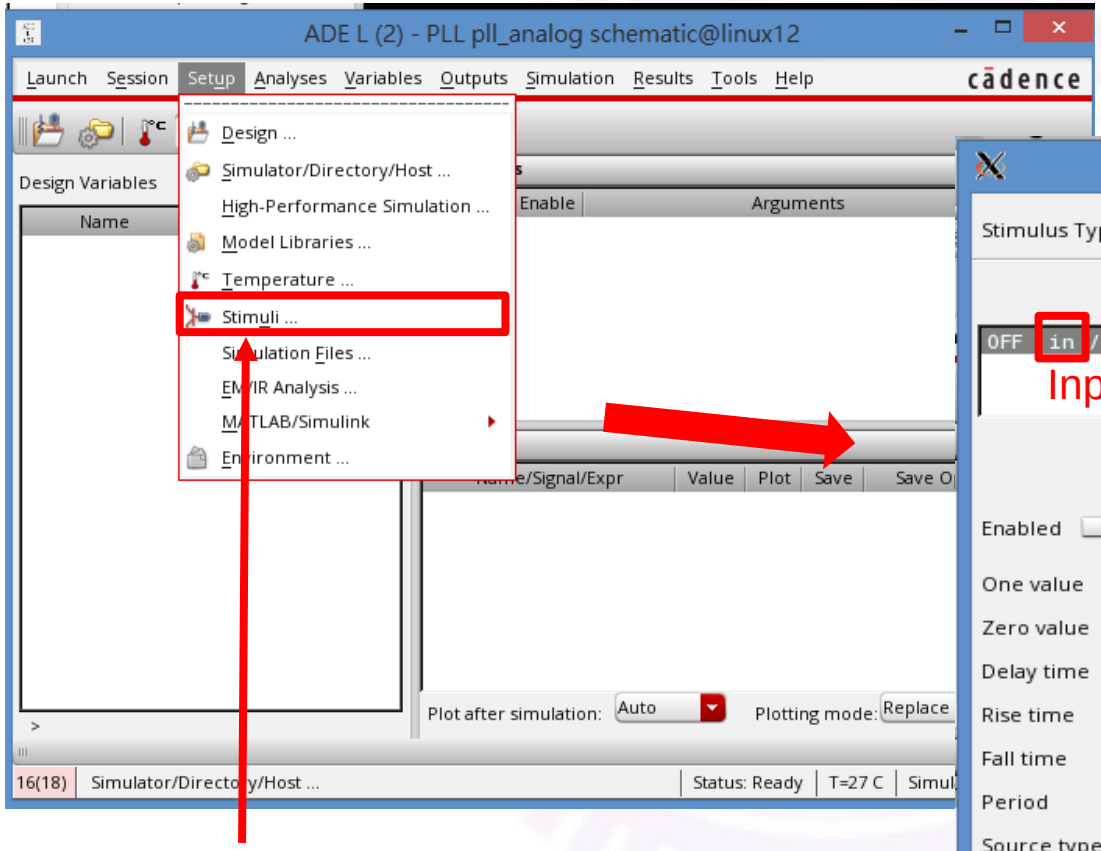
- Open Analog Design Environment (ADE) in schematic editing window

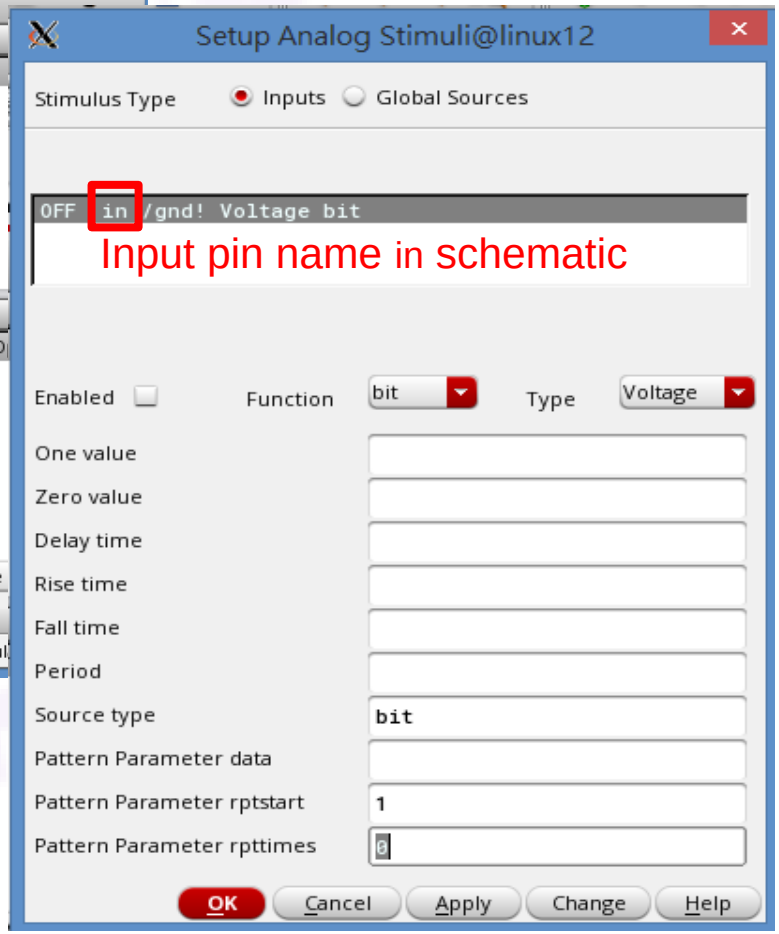


- Through Setup -> Simulator/Directory/Host



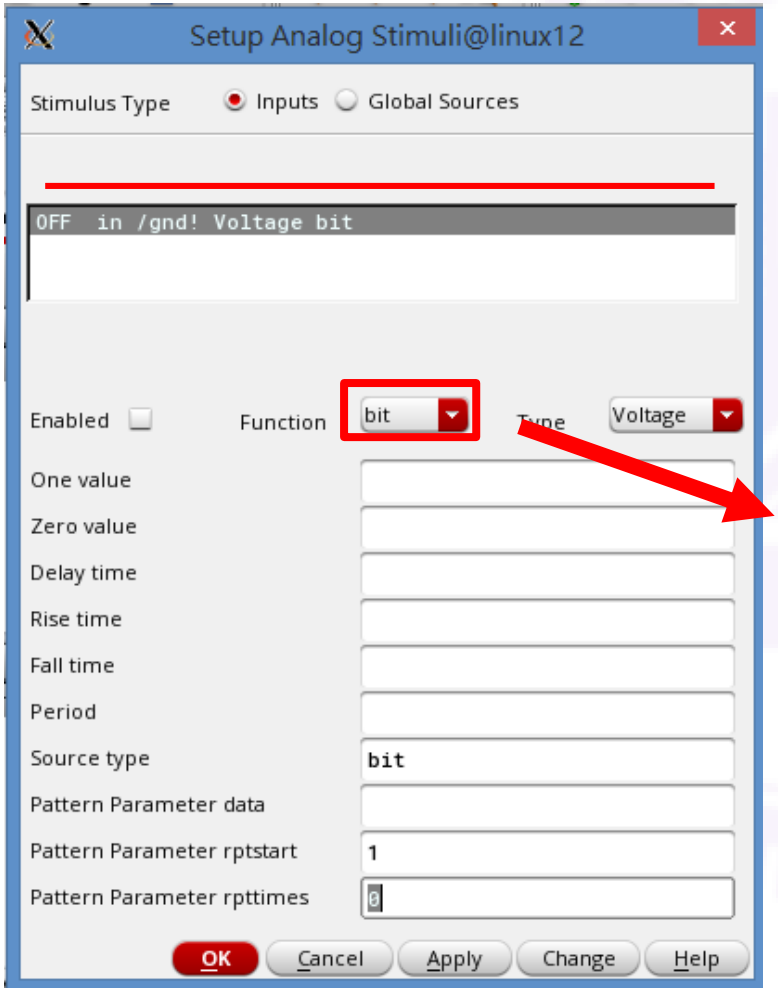
# Give input information(1/2)



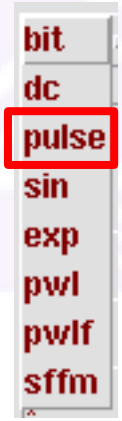



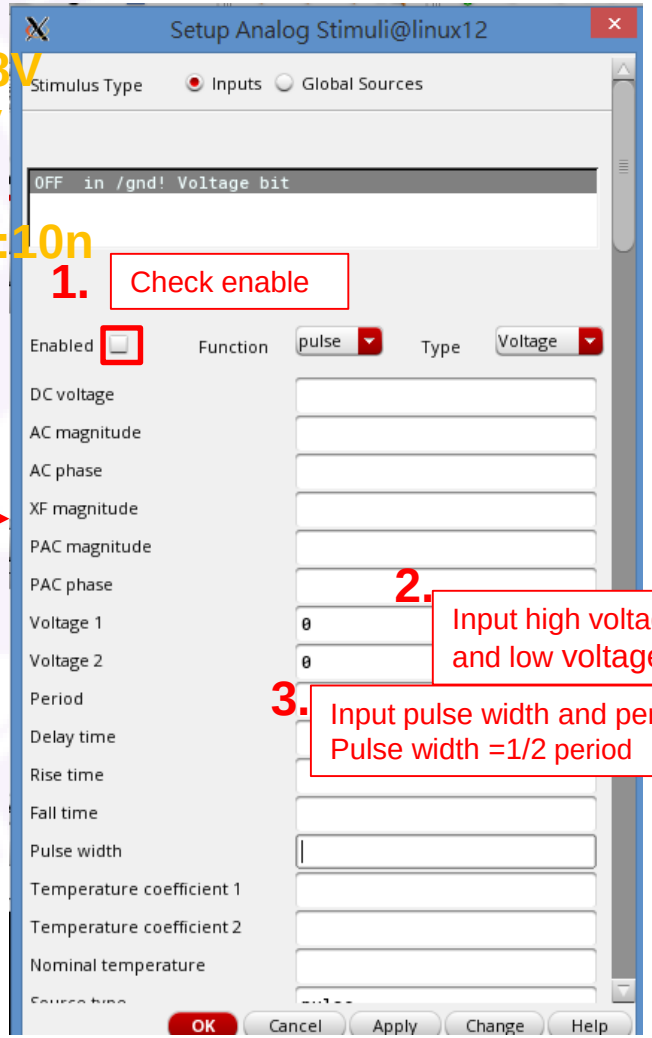

If choose spectre as simulator, there only has stimuli

# Give input information(2/2)

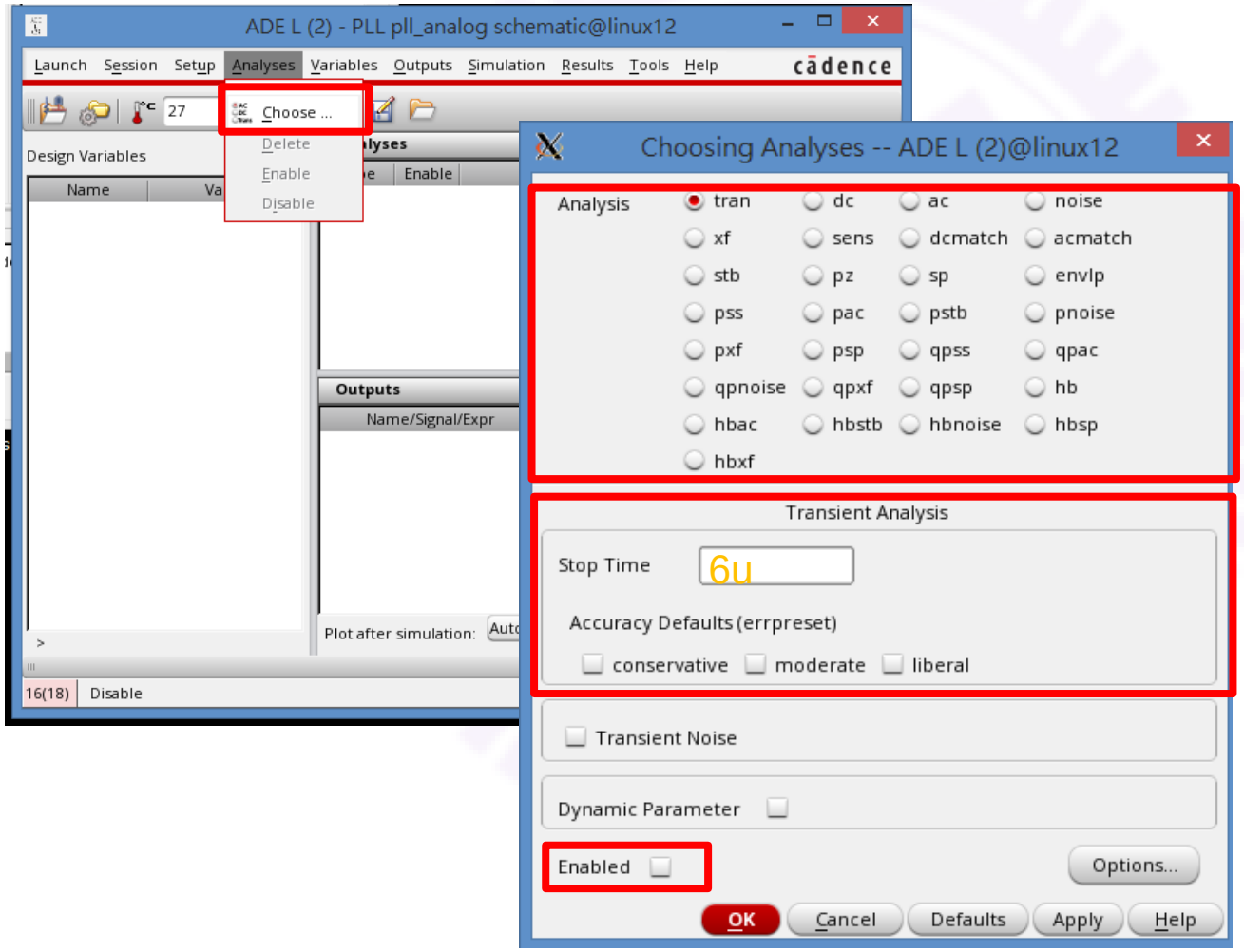



Voltage1:1.8V
Voltage2:0V
Period:20n
Pulse width:10n



# Choose Analysis Type

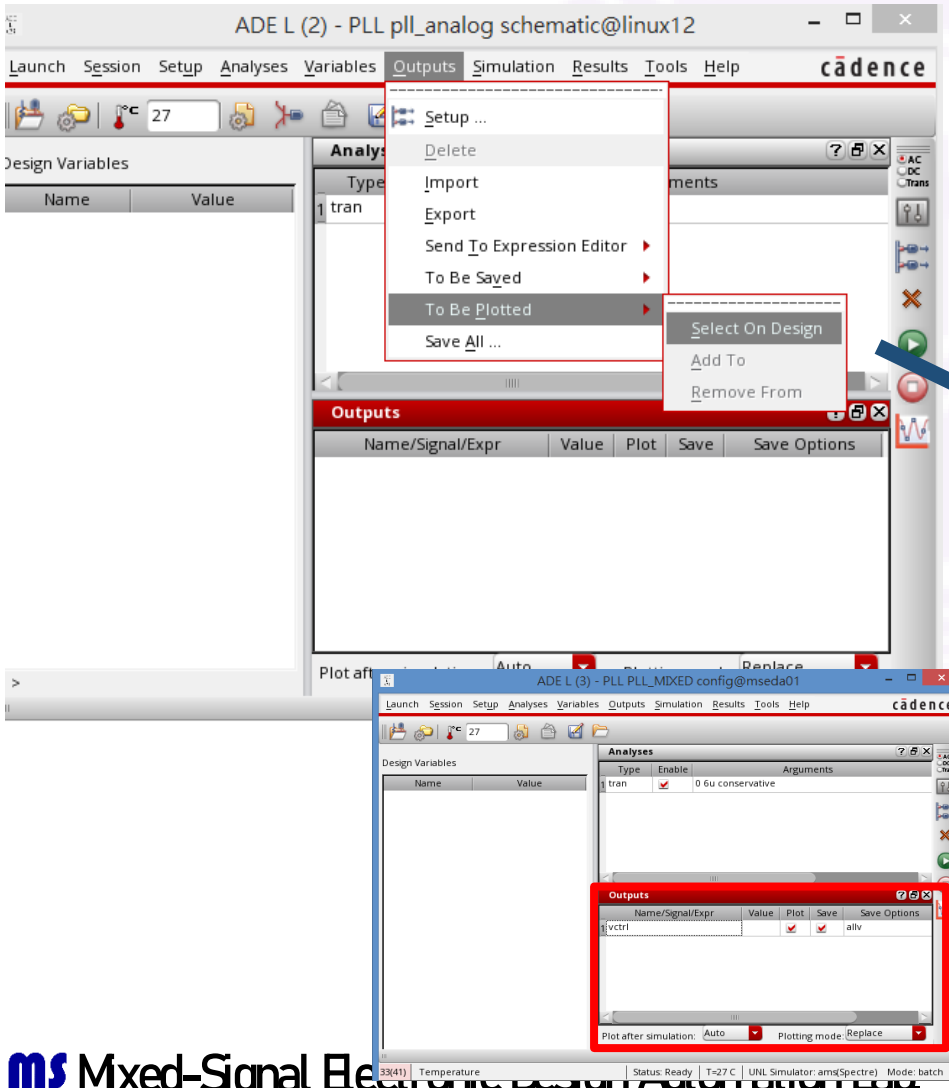



Choose tran analysis

Set the simulation time and the accuracy flag

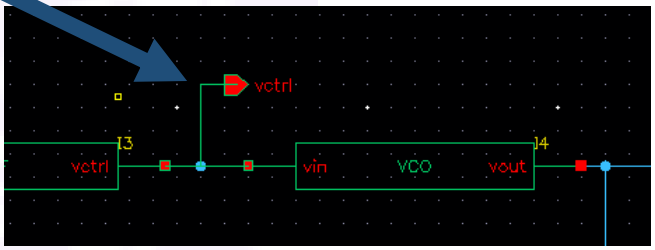
Check enable to simulati

# Save Output Nodes



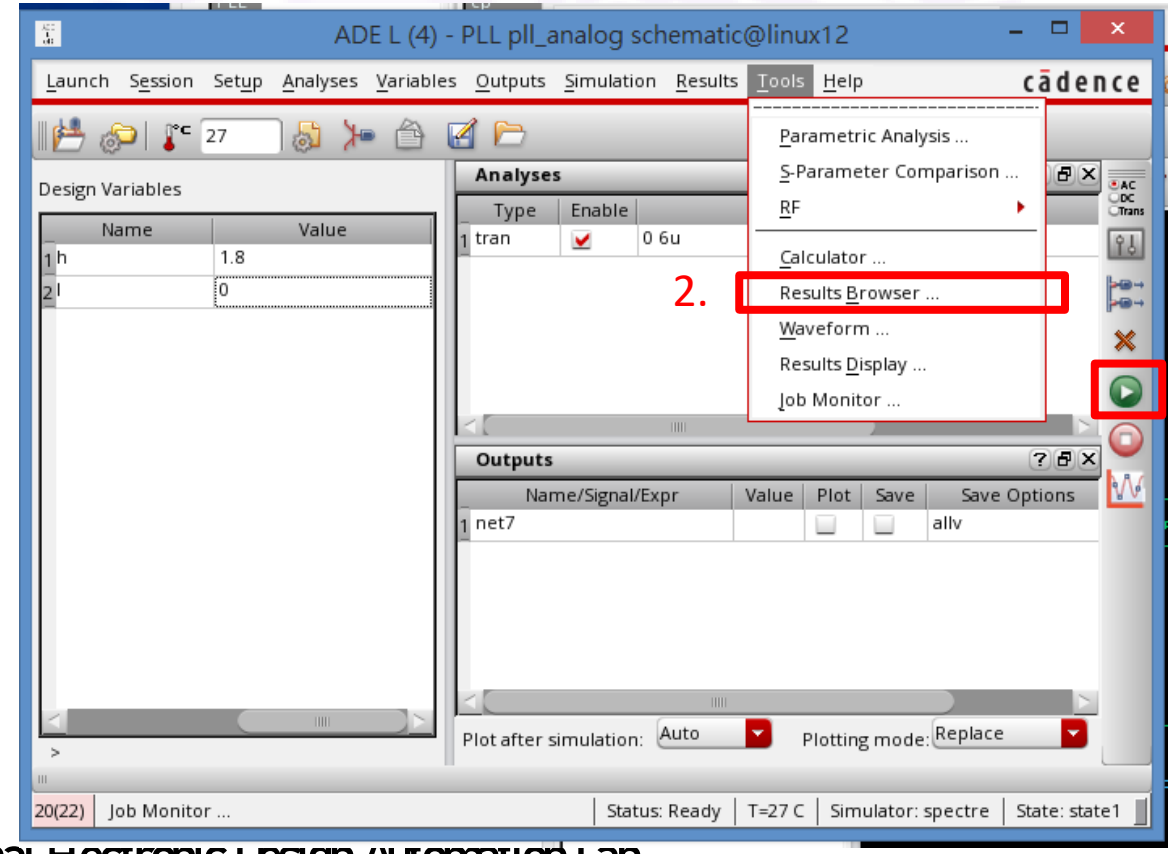

To Be Plotted:

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



# Submit the Simulation

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

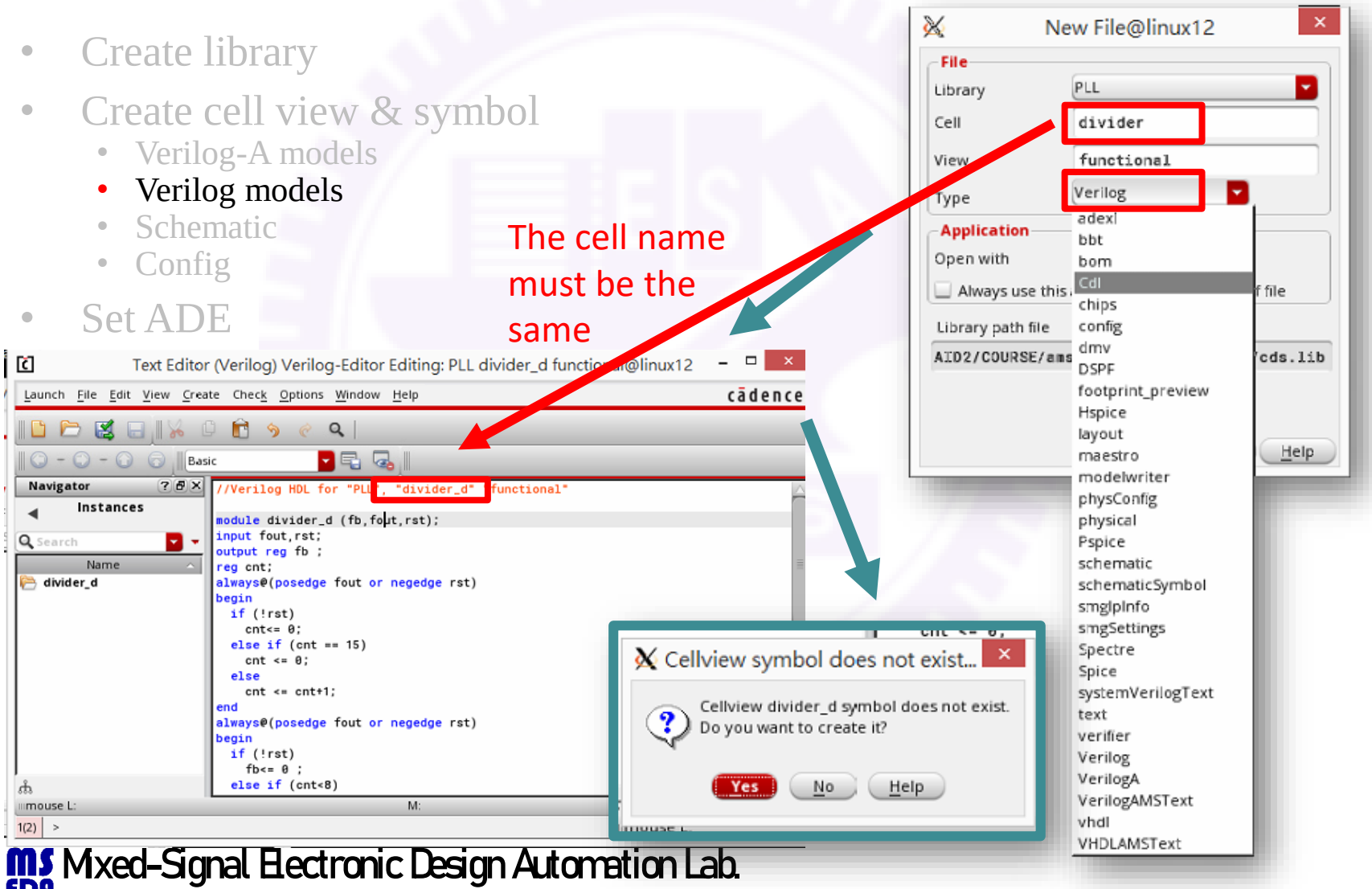


# Outline

- Introduction to AMS
- AMS Simulation Setup
  - Analog Simulation with Verilog-A Model
  - Mixed-Signal Behavioral Model Simulation
    - Lab1: Analog Model Simulation with AMS
    - Lab2: Mixed-Signal Model Simulation with AMS
    - Lab3: 2-bit ADC with Verilog-AMS

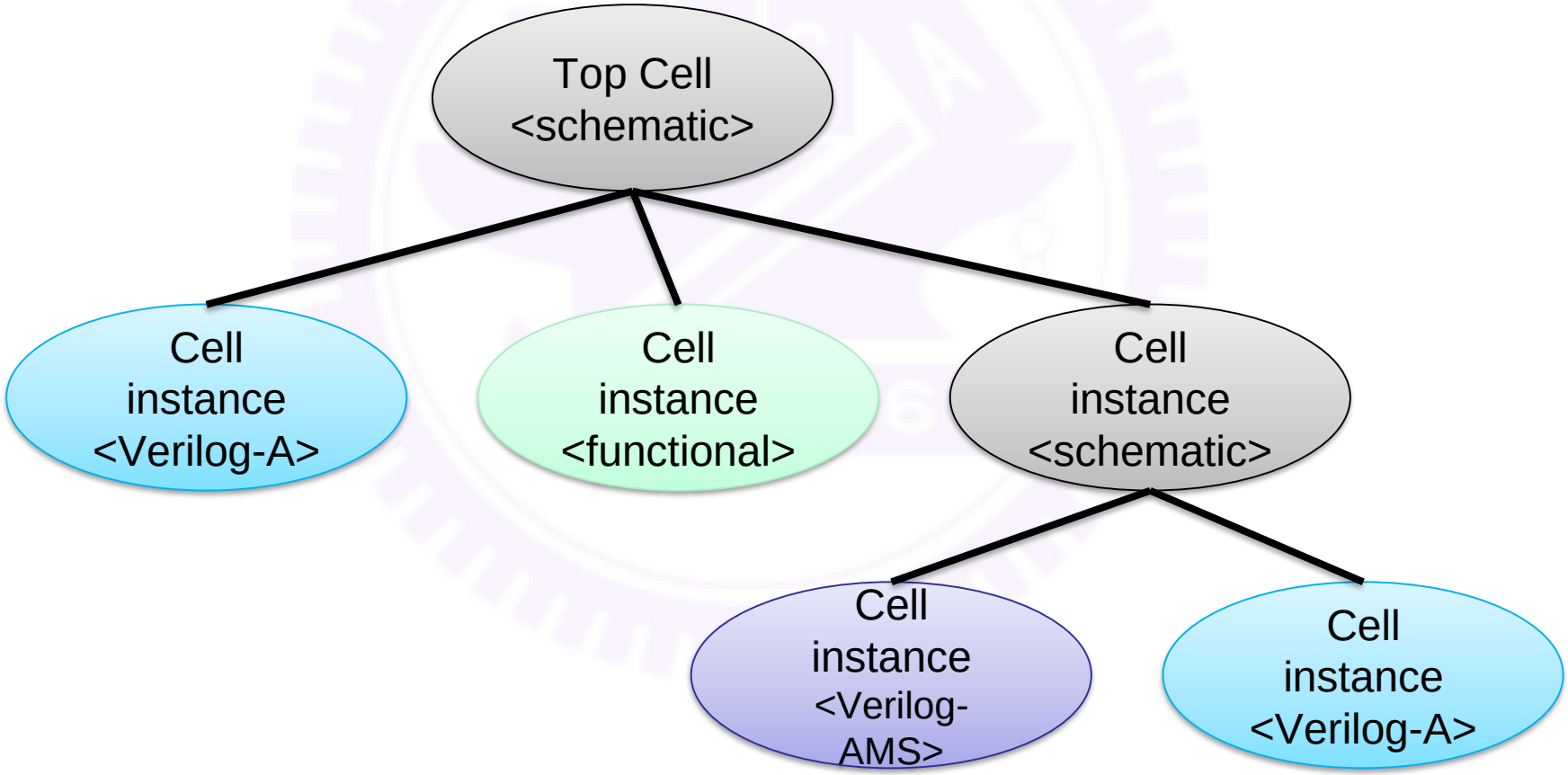
# Create Verilog Cells

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



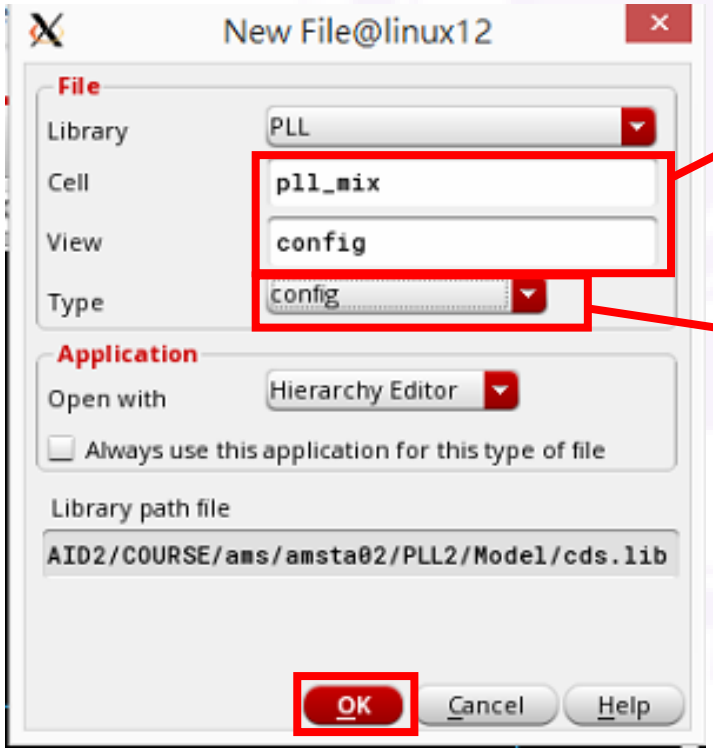
# Design Hierarchy – AMS

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



# Create Config View for Simulation

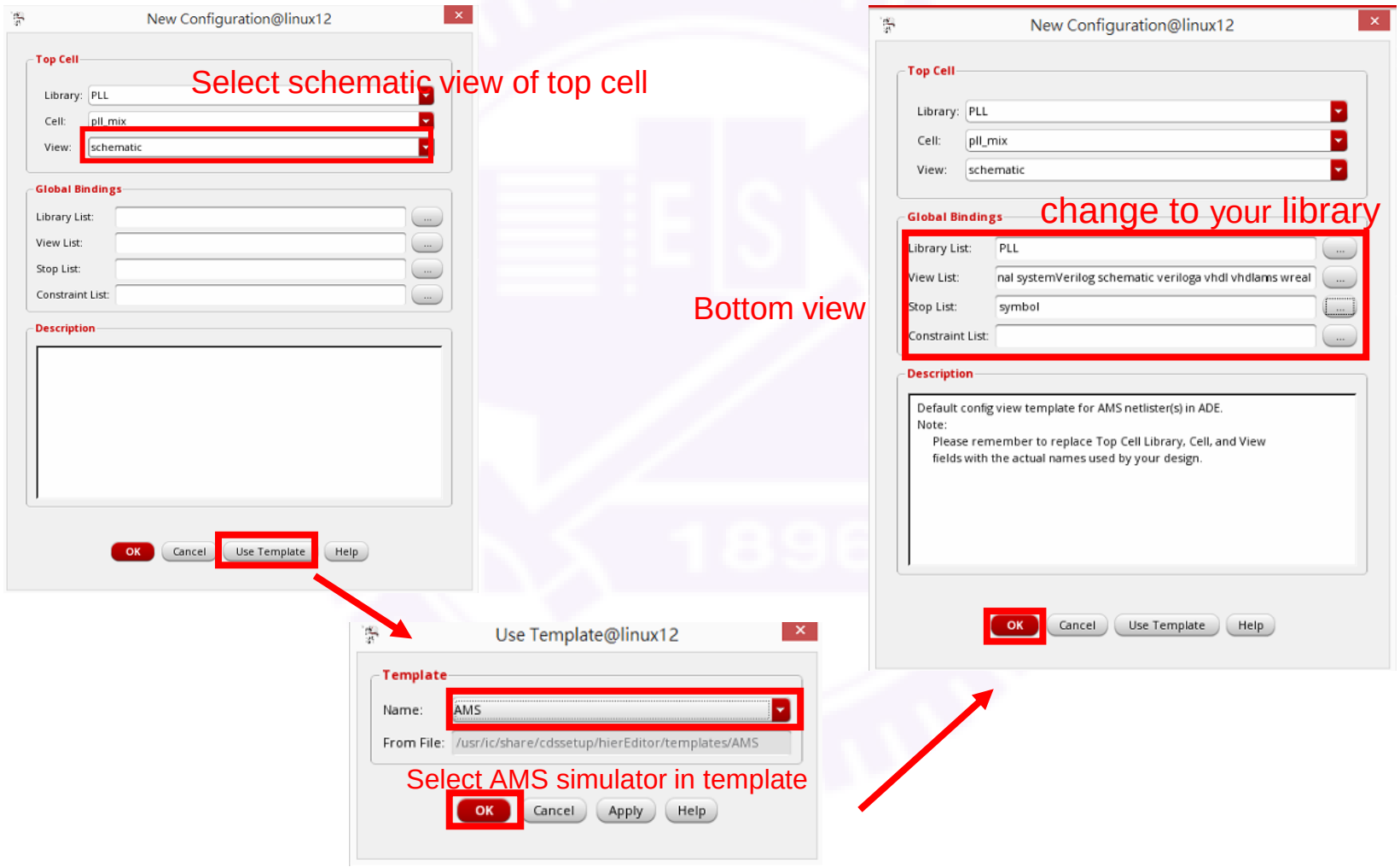
- The mixed-signal simulation hierarchy is controlled by **Hierarchy-Editor**
  - It must have to be defined in the **config** cell view.



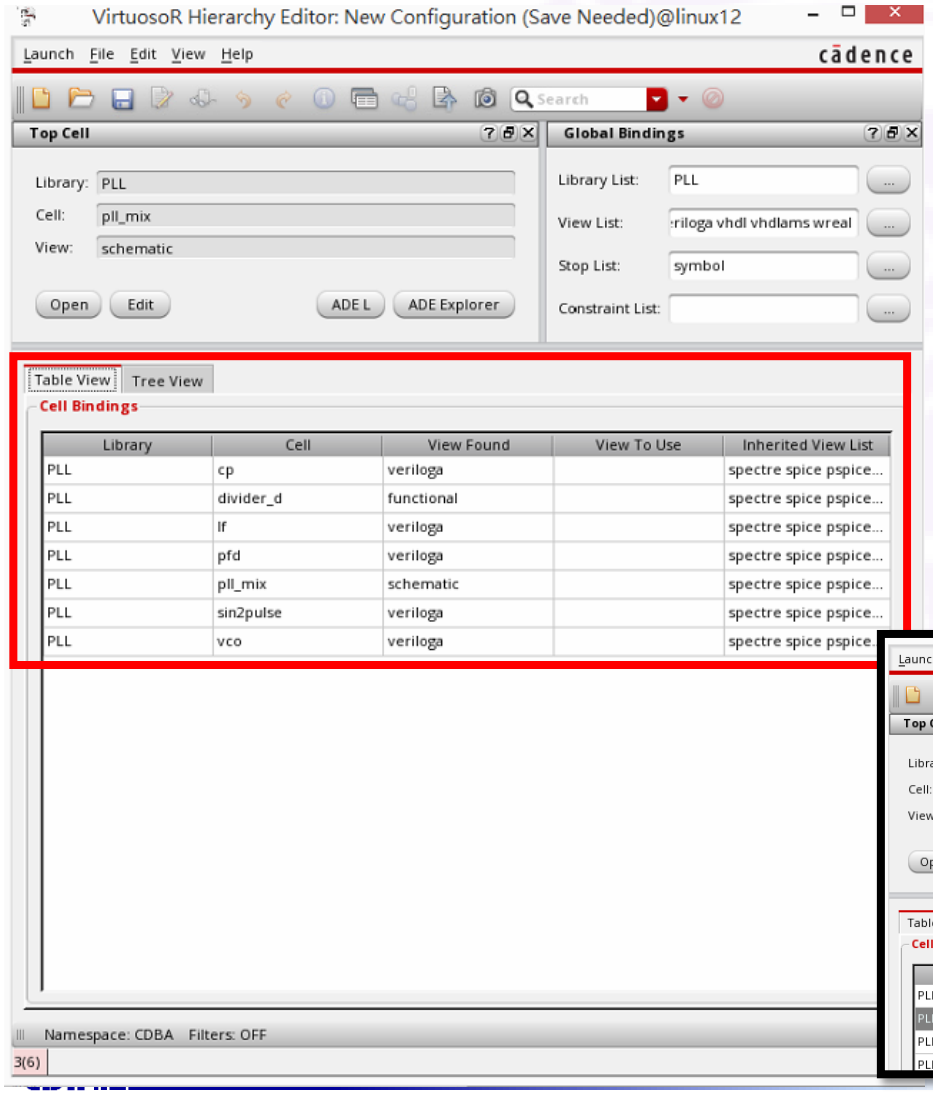

Cell name is top schematic cell for simulation

Choose **config** type

# Set New Configuration

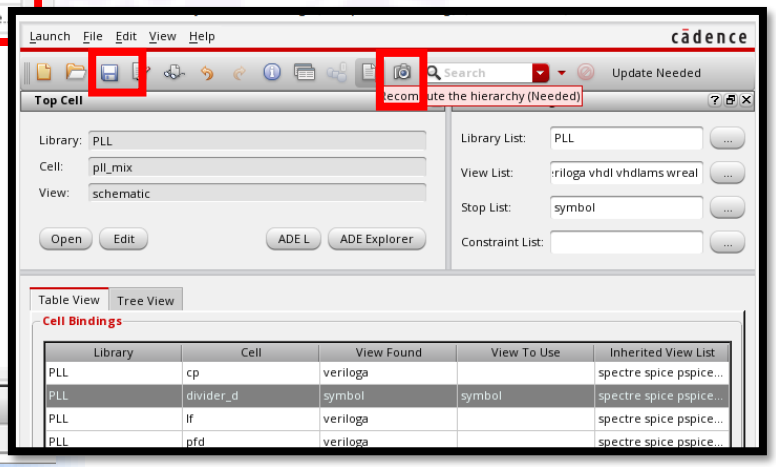


# Configuration Setting



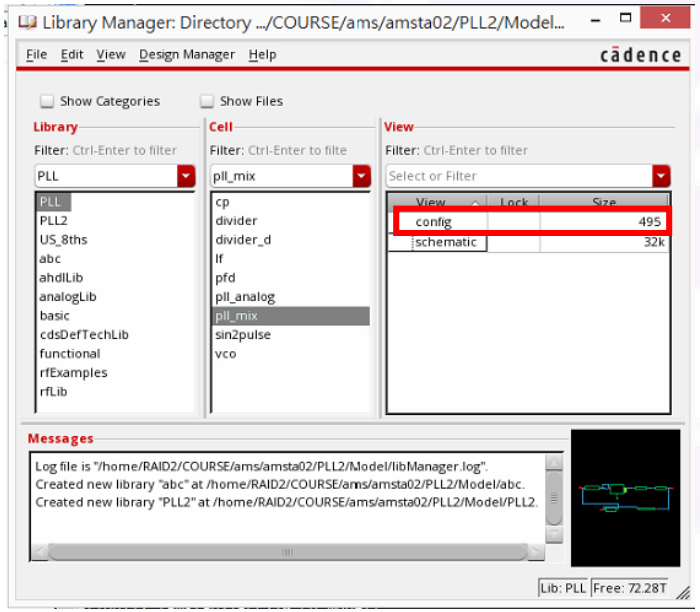

The hierarchy-editor will list all cells and views in the cell bindings

Must to save and recompute!

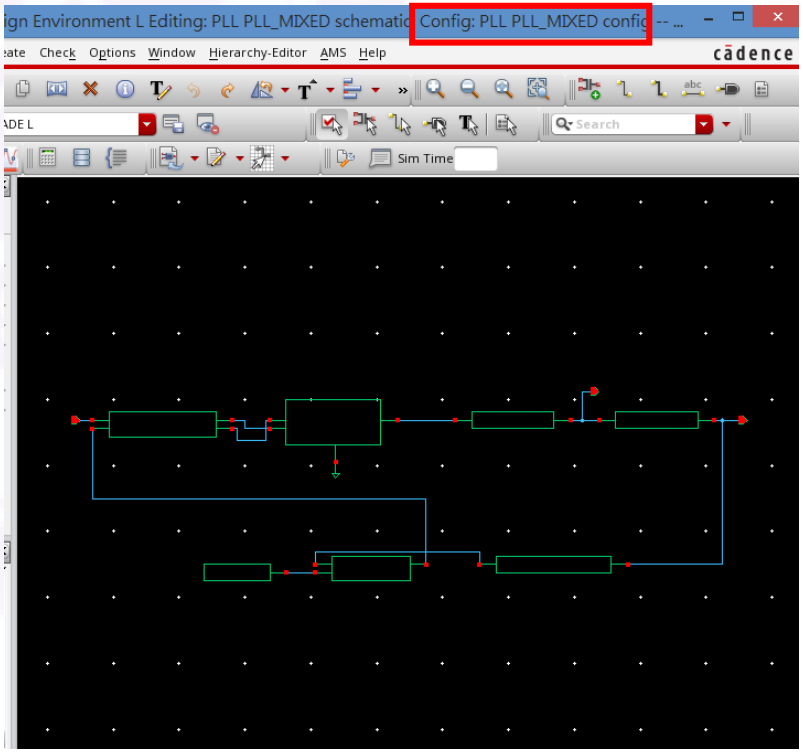


# Open Simulation Tool

- 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
```

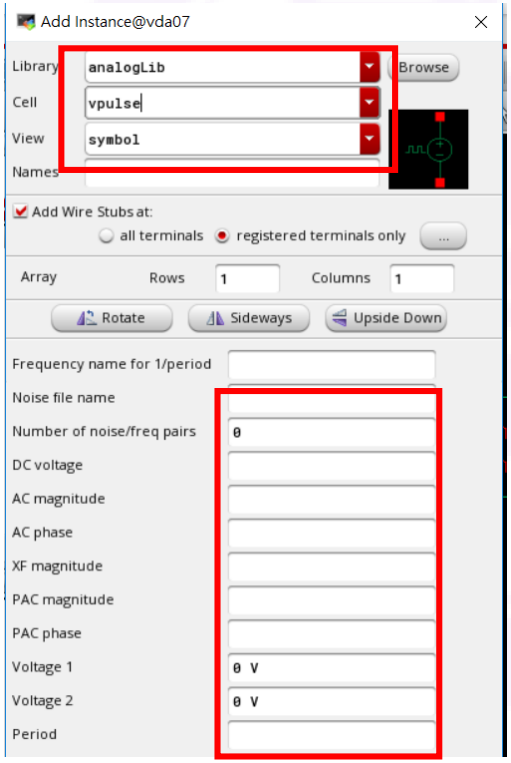
# Analog Stimulus

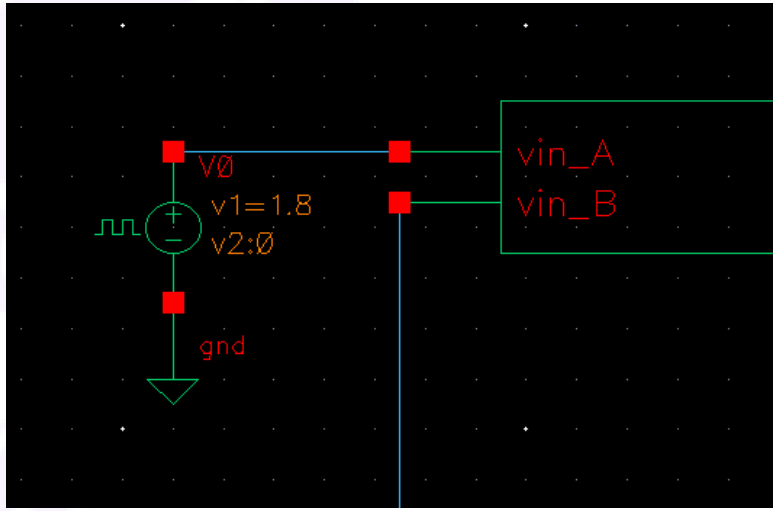
- The analog stimulus can be added as circuit instance

Add Instance

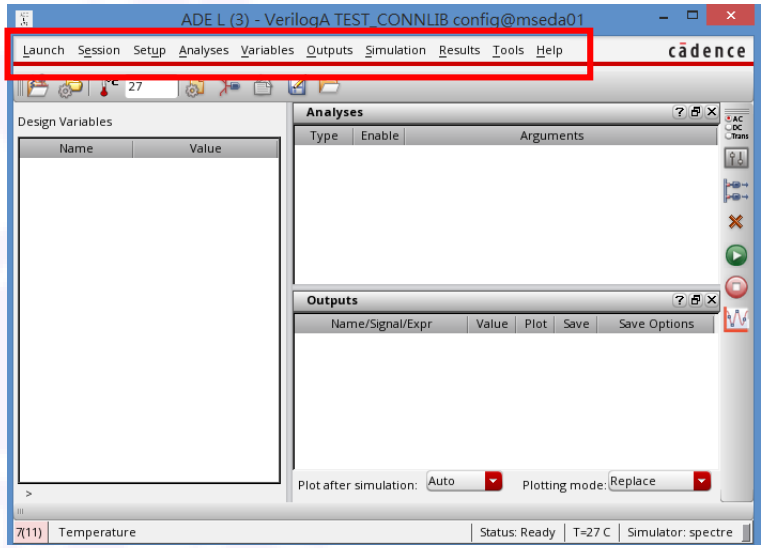
Choose analogLib & vpulse cell

Set voltage & frequency



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

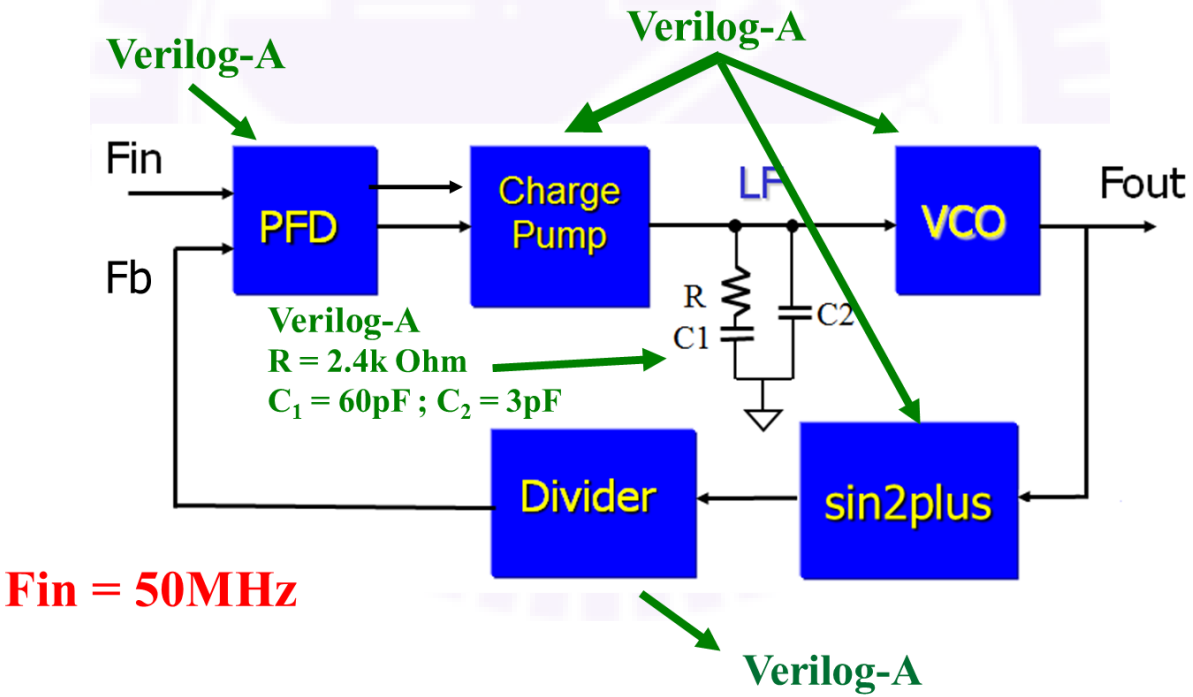


# Outline

- Introduction to AMS
- AMS Simulation Setup
  - Analog Simulation with Verilog-A Model
  - Mixed-Signal Behavioral Model Simulation
    - Lab1: Analog Model Simulation with AMS
    - Lab2: Mixed-Signal Model Simulation with AMS
    - Lab3: 2-bit ADC with Verilog-AMS

# PLL Analog Model

- All Verilog-A models are given
  - PFD, CP, LF, VCO, sin2plus, divider
- Import the Verilog-A models and setup ADE



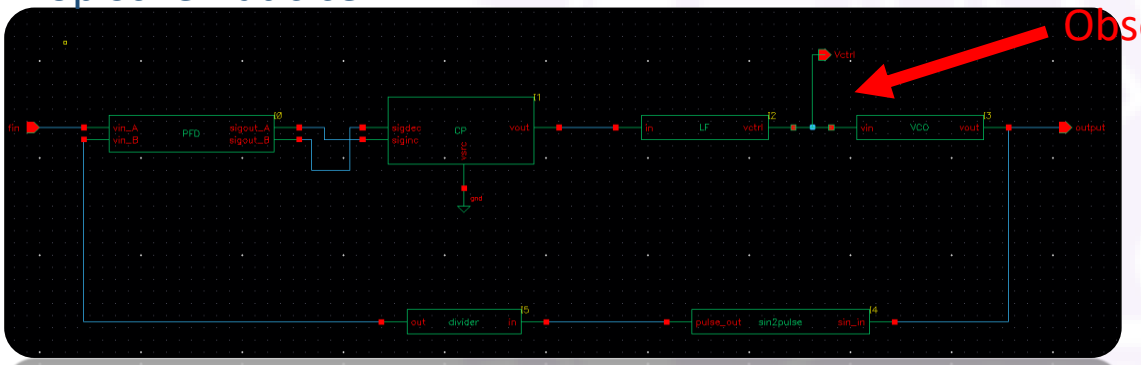
# Additional Description

- 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: **spectre**
- Input
  - **Fin=50M Hz : Set the ADE Stimuli**
- Simulation time $\geqq$ 6μs
  - Related to the lock time of PLL

# Results

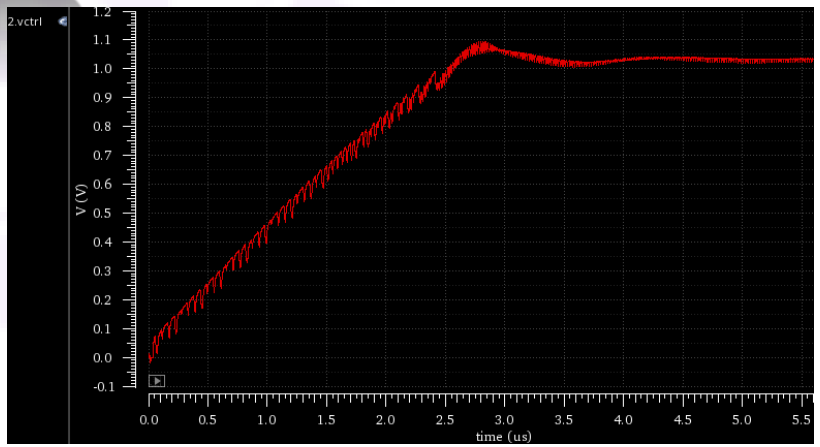
- Show the waveform of vctrl signal and schematic view

Top schematic cell




Observe the waveform of Vctrl

Open Results Brower

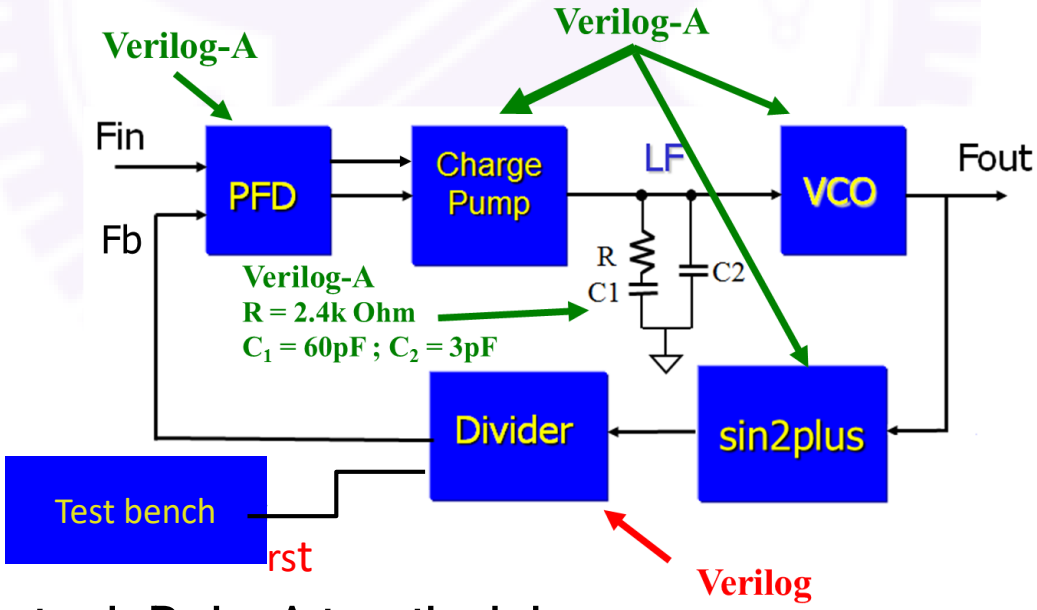


# Outline

- Introduction to AMS
- AMS Simulation Setup
  - Analog Simulation with Verilog-A Model
  - Mixed-Signal Behavioral Model Simulation
    - Lab1: Analog Model Simulation with AMS
    - Lab2: Mixed-Signal Model Simulation with AMS
    - Lab3: 2-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



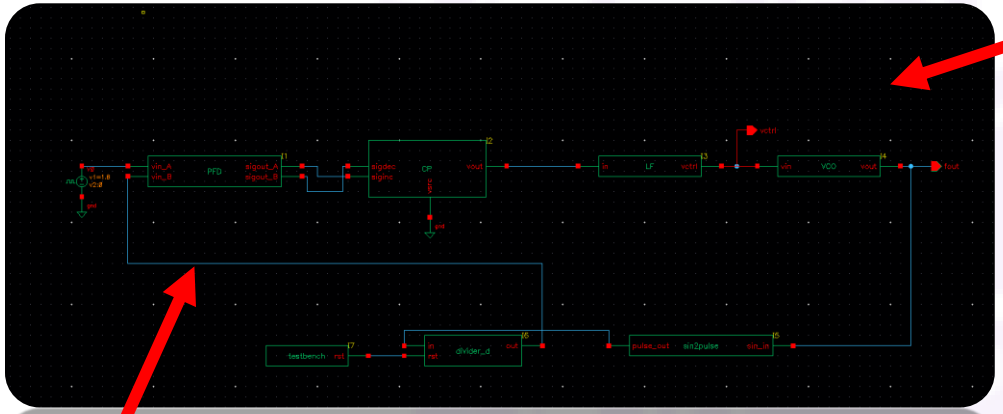
# Additional Description

- 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 $\geqq$ 6μs
  - Related to the lock time of PLL

# Results

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

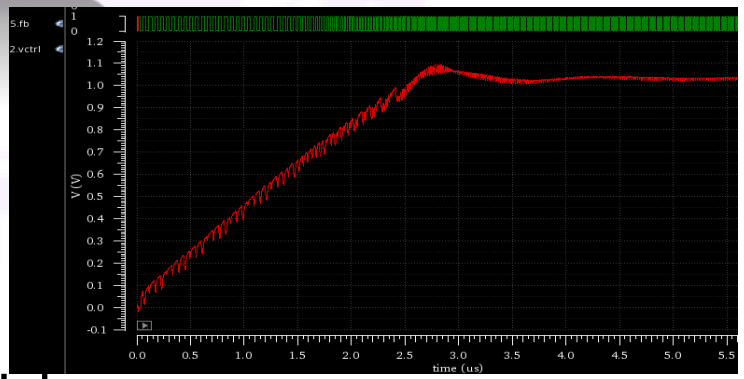
Top schematic cellview




Observe the waveform of Vctrl

Observe the output of divider.v

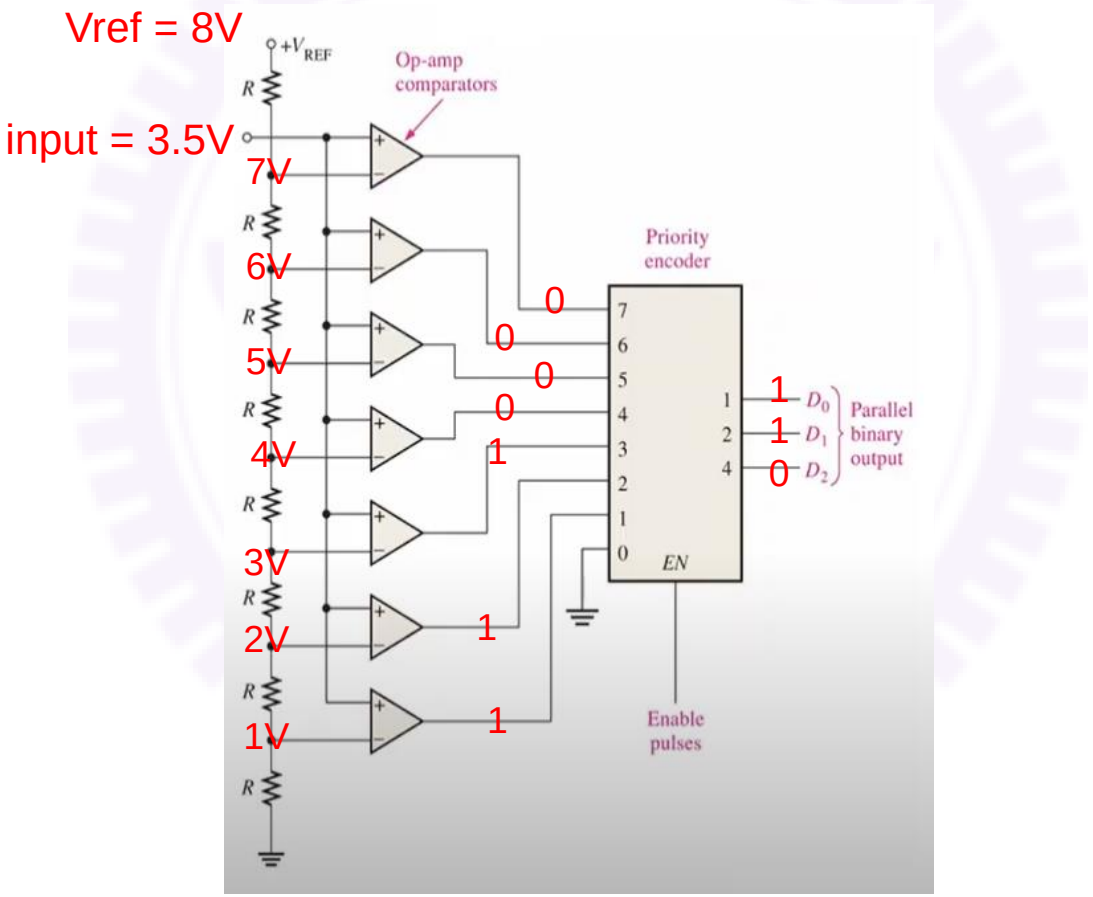
Open Results Brower



# Outline

- Introduction to AMS
- AMS Simulation Setup
  - Analog Simulation with Verilog-A Model
  - Mixed-Signal Behavioral Model Simulation
    - Lab1: Analog Model Simulation with AMS
    - Lab2: Mixed-Signal Model Simulation with AMS
    - Lab3: 2-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

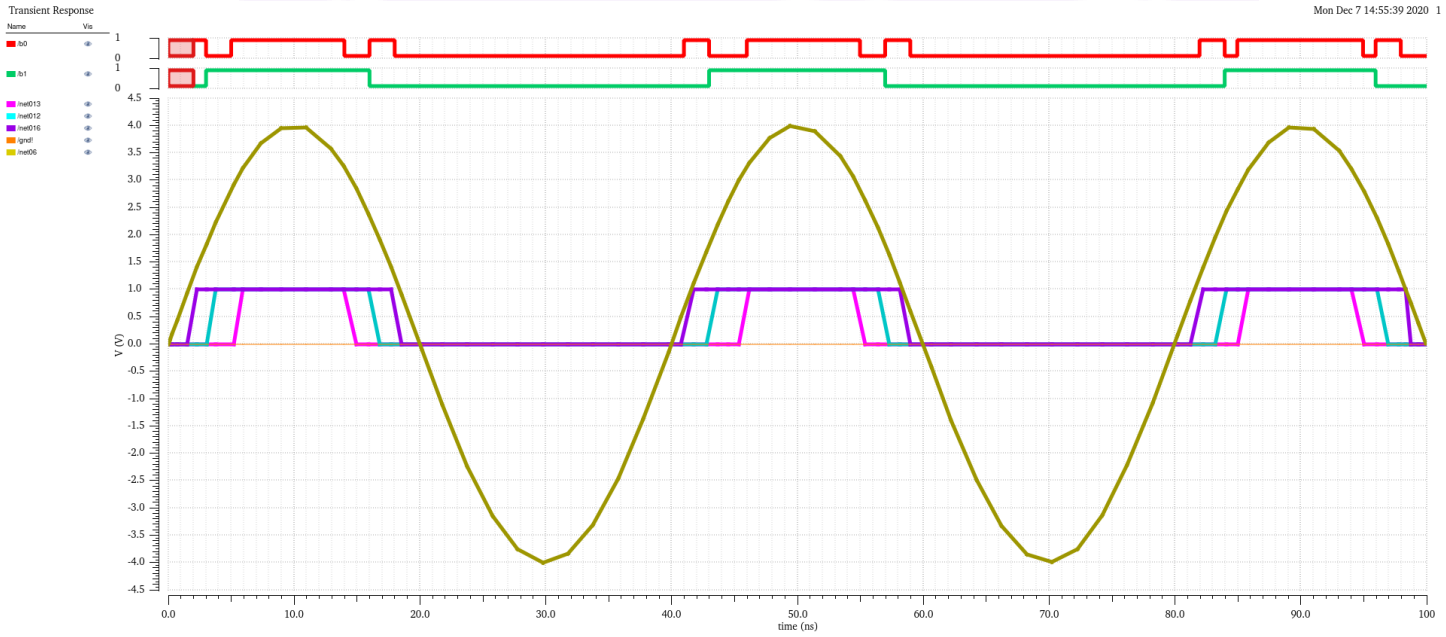
# Parameters of ADC Model

- Input : sin wave
- Amplitude : 4V, Frequency : 25M
- Output : 2 bit output
- R=1k, Vref=4V
- Analysis Type : tran, 100n
- Simulator: **AMS**

| B1,B0 (bit) | Input (V) |
|---|---|
| 00 | 0 |
| 01 | 1V |
| 10 | 2V |
| 11 | 3V |

# Results

- Hand in:
  - The waveform of outputs of comparators, Encoder and Input
  - Schematic cellview



# Hand in

- Please upload a compressed file includes:
  - **Programming files** (Verilog and Verilog-A files)
    - Lab2: Stimuli (testbench.v)
    - Lab3: Encoder file(.vams or .v), comparator file(.vams or .va)
  - **Mini report**
    - Three simulation waveforms
      - Lab1: Analog simulation (Vctrl)
      - Lab2: Mixed-signal simulation (Vctrl & output of the divider)
      - Lab3: ADC(outputs of comparators, Encoder and Input )
    - Three schematic cellviews
    - What you have learned from this homework
    - Questions and solutions
- **Deadline: 23:55, January 8, 2023**