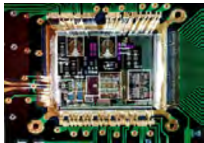
Lecture 6: Sigma-Delta Modulator and Evaluation of SPICE

CSCE 6933/5933 Advanced Topics in VLSI Systems

Instructor: Saraju P. Mohanty, Ph. D.

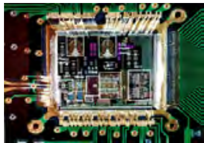
NOTE: The figures, text etc included in slides are borrowed from various books, websites, authors pages, and other sources for academic purpose only. The instructor does not claim any originality.

UNIVERSITY OF NORTH TEXAS
Discover the power of ideas



Outline

- Introduction
- Our Contribution
- High level Design of Sigma-Delta Modulator
- Design of Individual Components
- Comparative view of various Analog Circuit Simulators



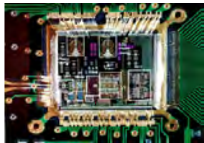
Two Points to Discuss

- The design of Sigma-Delta modulation is done using Verilog-A in Cadence and design of individual components (Diff-Amp, Op-Amp and Comparator) in CMOS transistor level in Spectre.
- And Comparison and Evaluation of existing analog circuit simulators. Circuit Simulators which are considered are Ngspice, Tclspice, Winspice and Spectre.

Analog Circuit Simulators

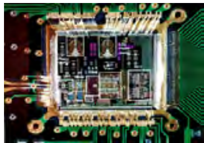
- Analog circuit simulator is a tool which is used to design and predict the circuit behavior before the fabrication.

- History
  - In **1970** , the First Analog circuit simulator named **CANCER** (Computer Analysis of Non-Linear Circuits Excluding Radiation), was developed by Ron Rohrer and his research team at University of California.
  - In **1972**, Nagel and Pederson released **SPICE1** (Simulation Program with IC Emphasis) into the public domain so that people can modify and upgrade the code.
  - In **1975**, Nagel, who upgraded the **SPICE1**, made some significant improvements and named it as **SPICE2.**
  - In **1985**, **SPICE3**, circuit simulator, was upgraded and written in C programming language.



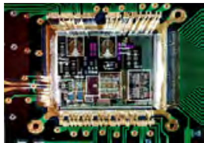
Analog Circuit simulator

- The circuit simulators which are compared and evaluated are as follows :
  - **Ngspice**
  - **Tclspice**
  - **Winspice**
  - **Spectre**



Ngspice

- Ngspice is a circuit simulator which is a continuation of SPICE3f5 and runs in UNIX environments.
- Ngspice was developed at the University of Rome by Paolo Nenzi.
- Ngspice is an open-source and the source-code is available online.
- The required package to Ngspice is Xgraph.

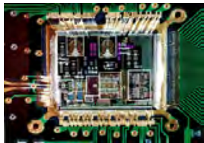


Tclspice

- Tclspice an improved version of SPICE3f5 is a circuit simulator to be used on Tcl/Tk scripting language and it runs in in UNIX environments.

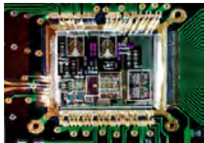
- Tclspice is also an open-source (BSD license) and the source-code is available online.

- The required packages to Tclspice are Tcl\Tk, BLT and Tclreadline.



Winspice

- Winspice an extension of Spice3f4 is an analog circuit simulator different from circuit simulators Ngspice and Tclspice.
- Winspice runs on win32.
- This is a tool developed to increase capability, adds new model and new applications.



Spectre

- Spectre is an analog circuit simulator which is very much efficient and stable.

- It is a commercial software developed by cadence and used by many VLSI companies.

- Spectre is developed to improve capability and also to add built-in models for semi-conductor.

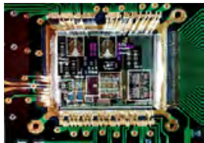
- It runs on UNIX environments and it has schematic editor to design the circuits.

Example Circuits

- Differential Amplifier: Differential amplifier is a circuit which amplifies the difference between two input signals.

- Operational Amplifier: Operational Amplifier (Op-Amp) is a popular device in linear circuits. It produces an output which is a product between inverting and non inverting inputs.

- Comparator: Comparator is a circuit which compares two analog input voltages and produces a digital output.



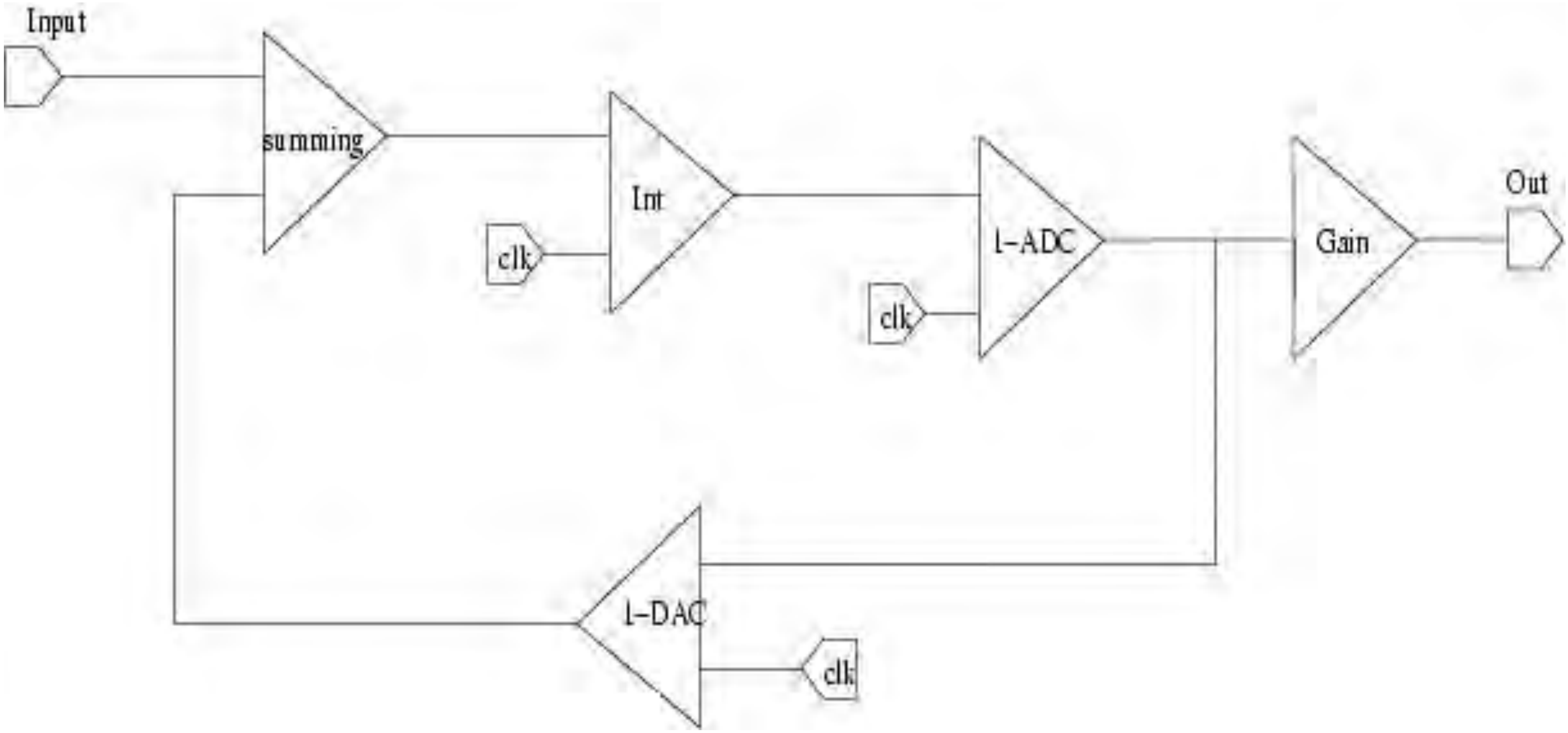
Sigma-Delta Modulator

- Sigma-Delta modulation based analog to digital (A/D) conversion technology is an effective alternative for high resolution converters.

- Such a technique is not only cost efficient but also can be integrated on DSP Ics.

- Increased use of digital technology in communication mechanisms propelled the introduction of cost effective high resolution A/D converters.

High Level Design of Sigma-Delta Modulator

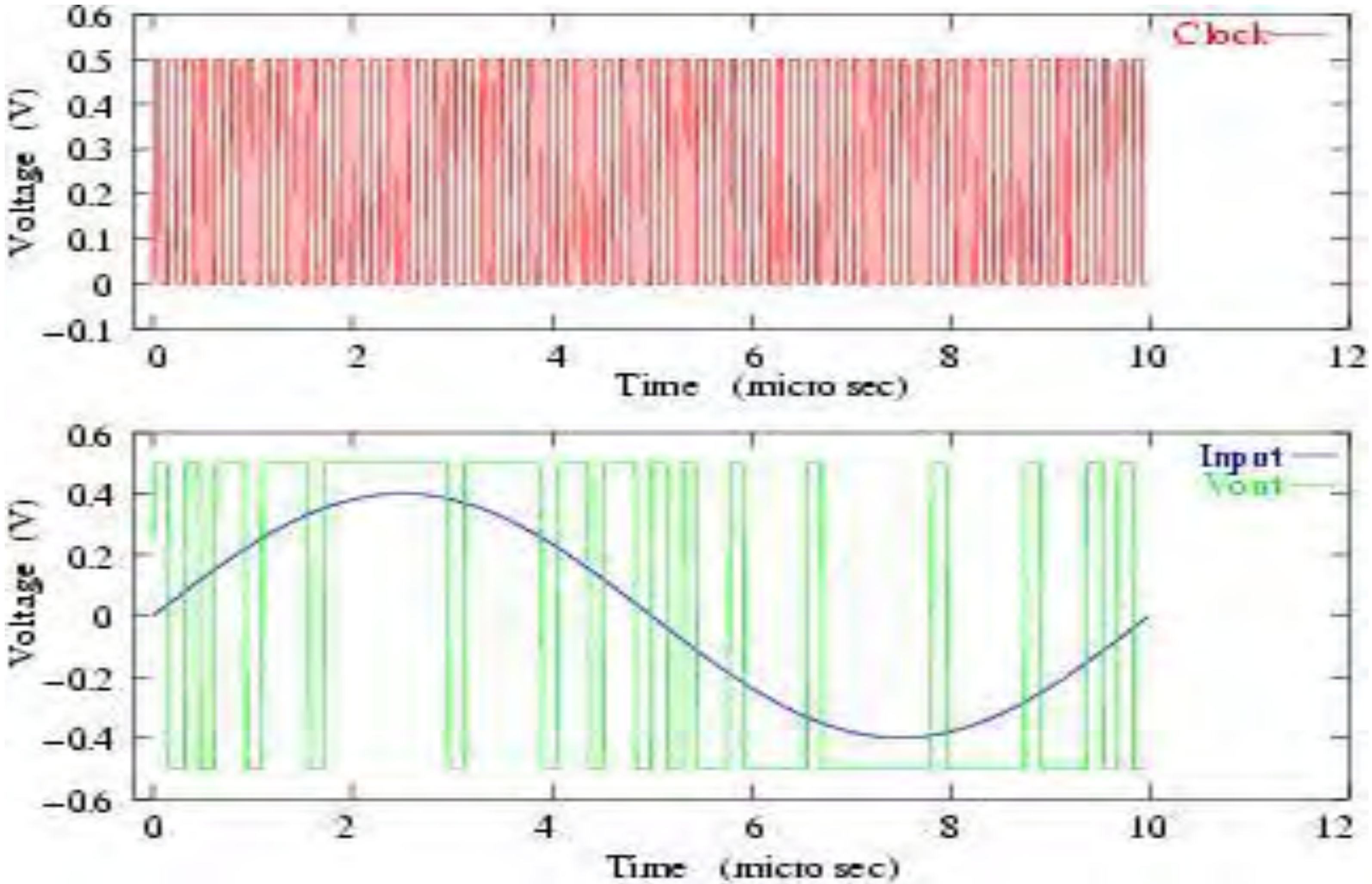


Block Diagram of Sigma-Delta Modulator

High Level Design of Sigma-Delta Modulator

- Design of an Ideal Sigma-Delta Modulator.
- Verilog-A code for the Ideal Sigma-Delta Modulator mainly consists of summing junction, Integrator, Quantizer and 1-bit DAC.
- Design all individual components of Sigma-Delta Modulator such as summing junction, Integrator, Quantizer and 1-bit DAC using Verilog-A.
- Connect all the individual components to design Sigma-Delta Modulator.

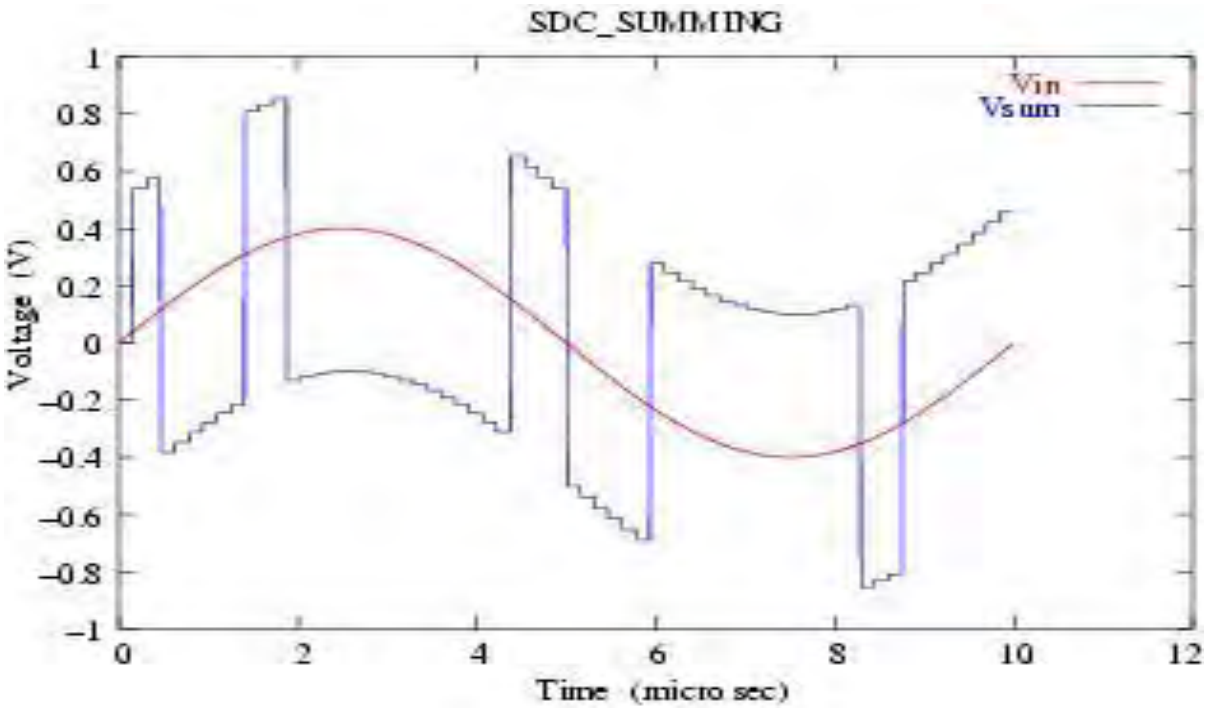
Output for Ideal Sigma-Delta Modulator



High Level Design of Summing Amplifier

- The working of summing amplifier is the sum of the input signal (Vin) of Sigma-Delta Modulator and Output of the 1-bit DAC (Vd).

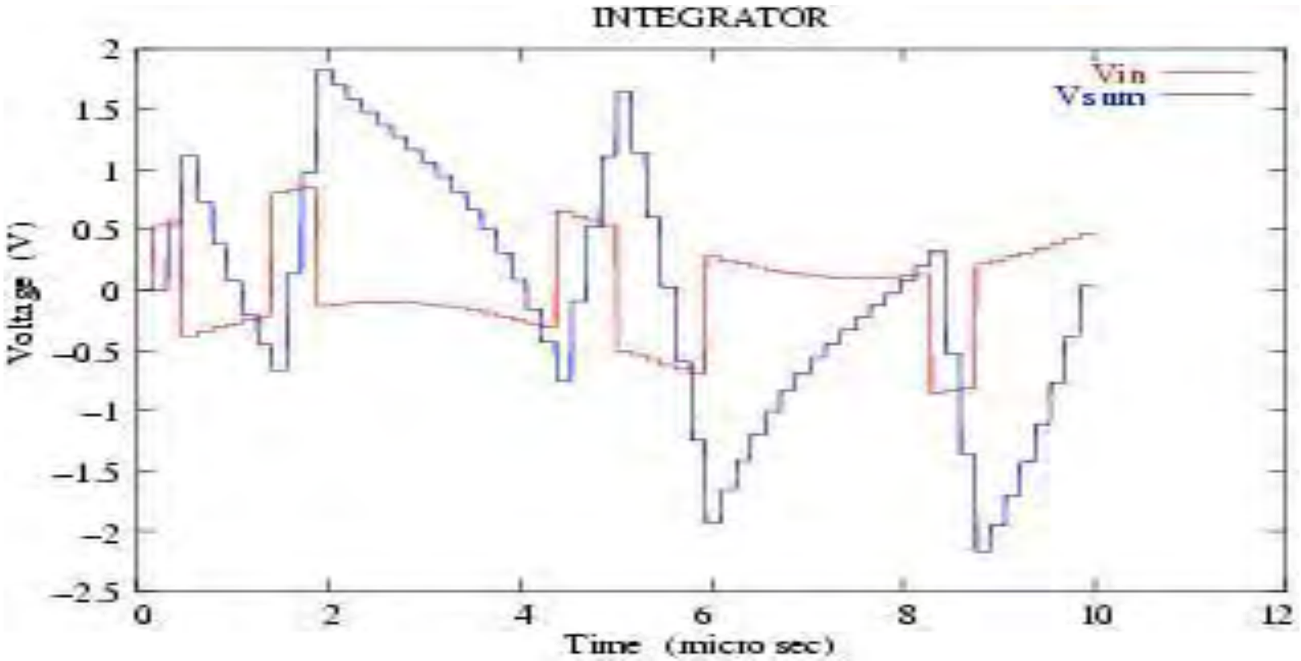
- Graph of the Summing Amplifier is shown below.



High level Design of Integrator

- The output voltage (vint) of the integrator will be the sum of the output voltage of the summing Amplifier (vsum) and output of the integrator (vint).

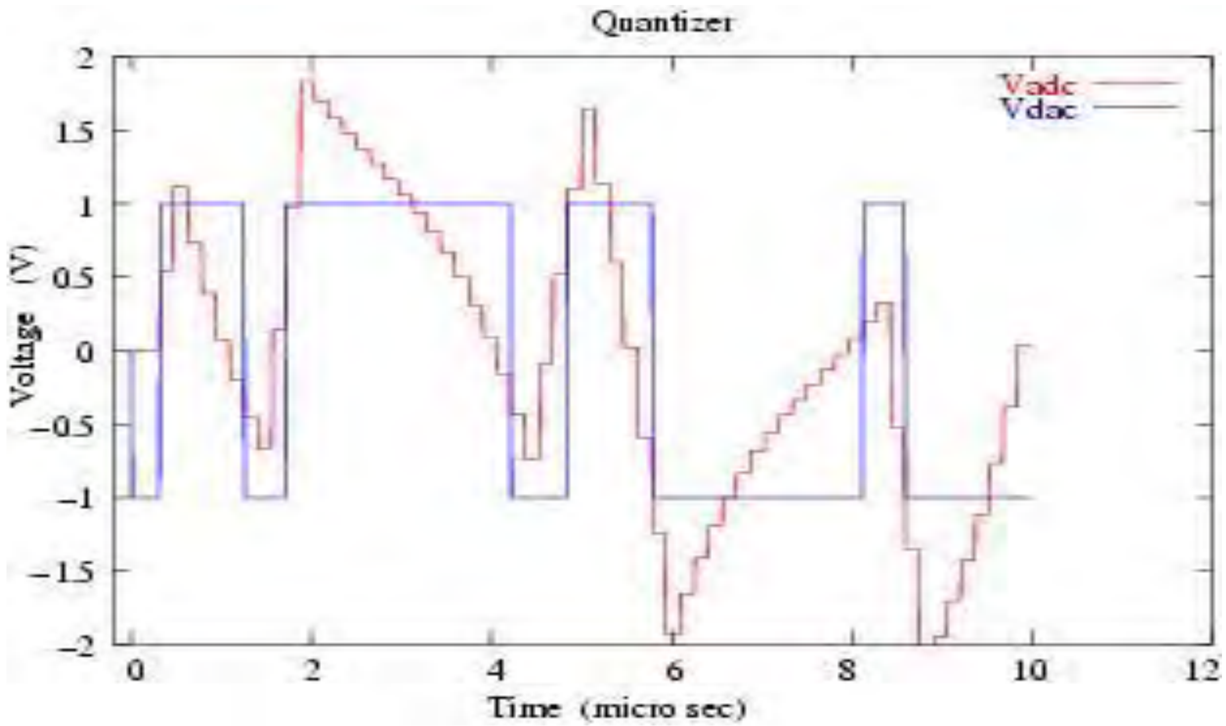
- Graph of the integrator is shown below.



High level Design of Quantizer

- The working of the Quantizer is if V(vin) > vth then the output for the quantizer will be vout_val = +1 otherwise Vout_val = -1. Where Vout_val is an instance parameter.

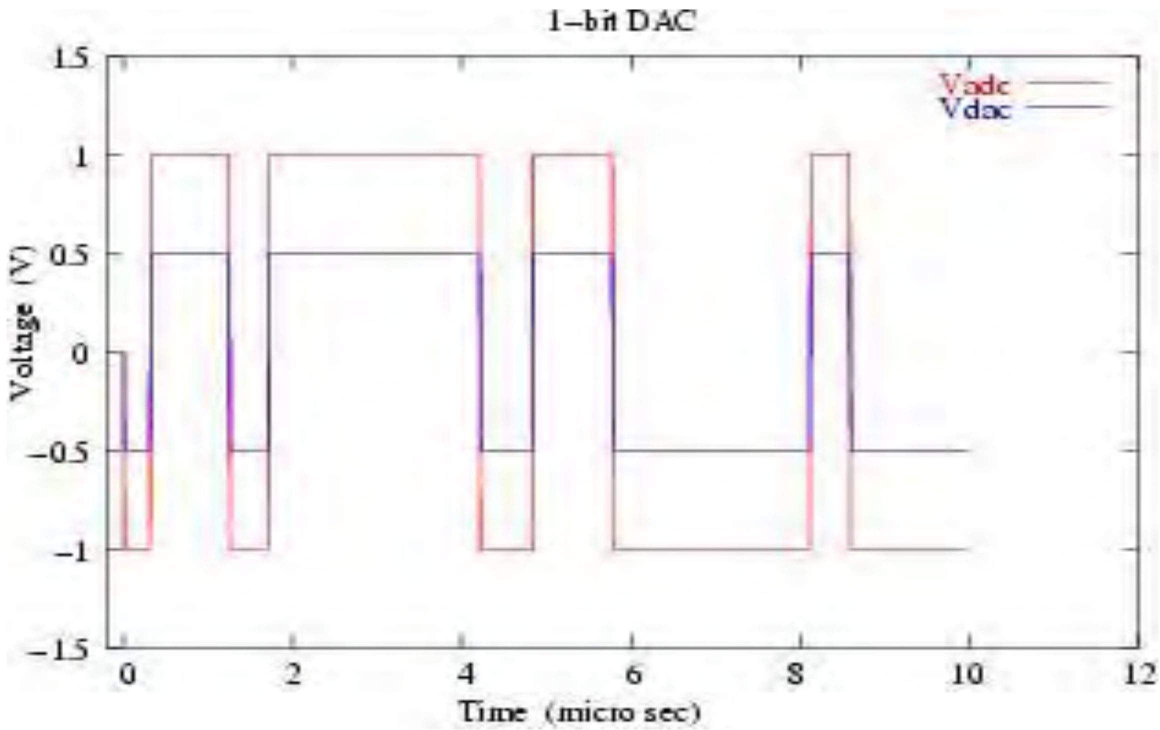
- Graph of quantizer is shown below.



High level Design of 1-bit DAC

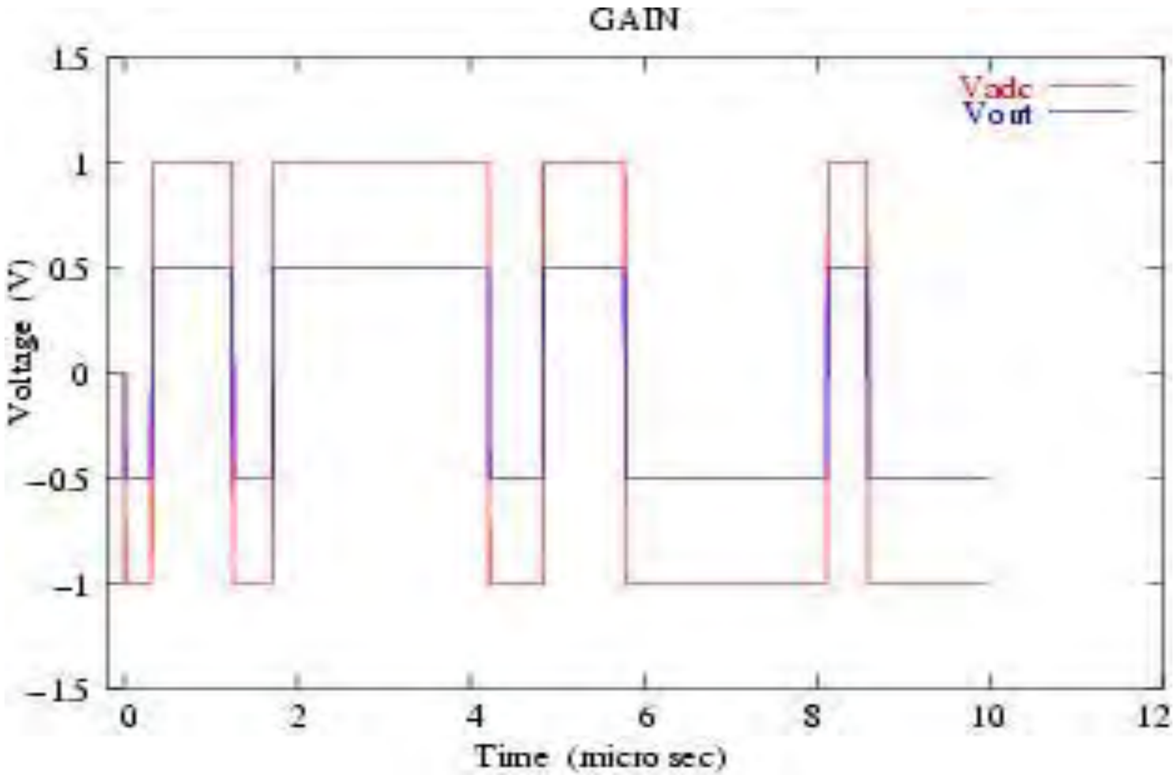
- The working of 1-Bit DAC is the product of input voltage V(vout_val) and vout_high. The V(vout) will be the output voltage vout_val for the 1-bit DAC. Where Vout_val and vout_high is an instance parameters.

- The graph of 1-bit DAC is shown below.



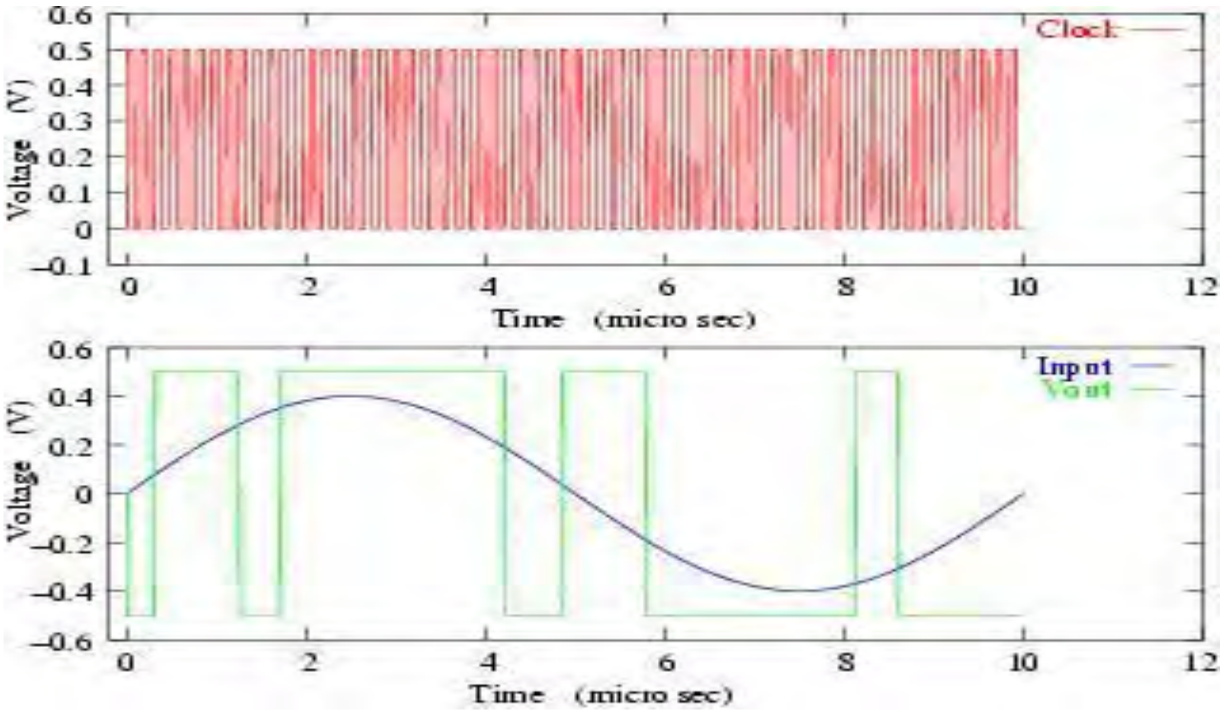
High level design of Gain Amplifier

- The working of the Gain amplifier is the product of input voltage V(vout_val) and vout_high. Where vout_high is an instance parameter.
- Graph of Gain amplifier is shown below.



High Level Design of Sigma-Delta Modulator using Individual Components

- Connecting all the individual components of Sigma-Delta Modulator such as Summing Amplifier, Integrator, Quantizer, 1-Bit DAC and Gain as shown in the block diagram.
- The output of the Sigma-Delta Modulator is shown.



Design of Individual Components

- The individual components which we designed are as follows:
    - Differential Amplifier
    - Operational Amplifier
    - Comparator

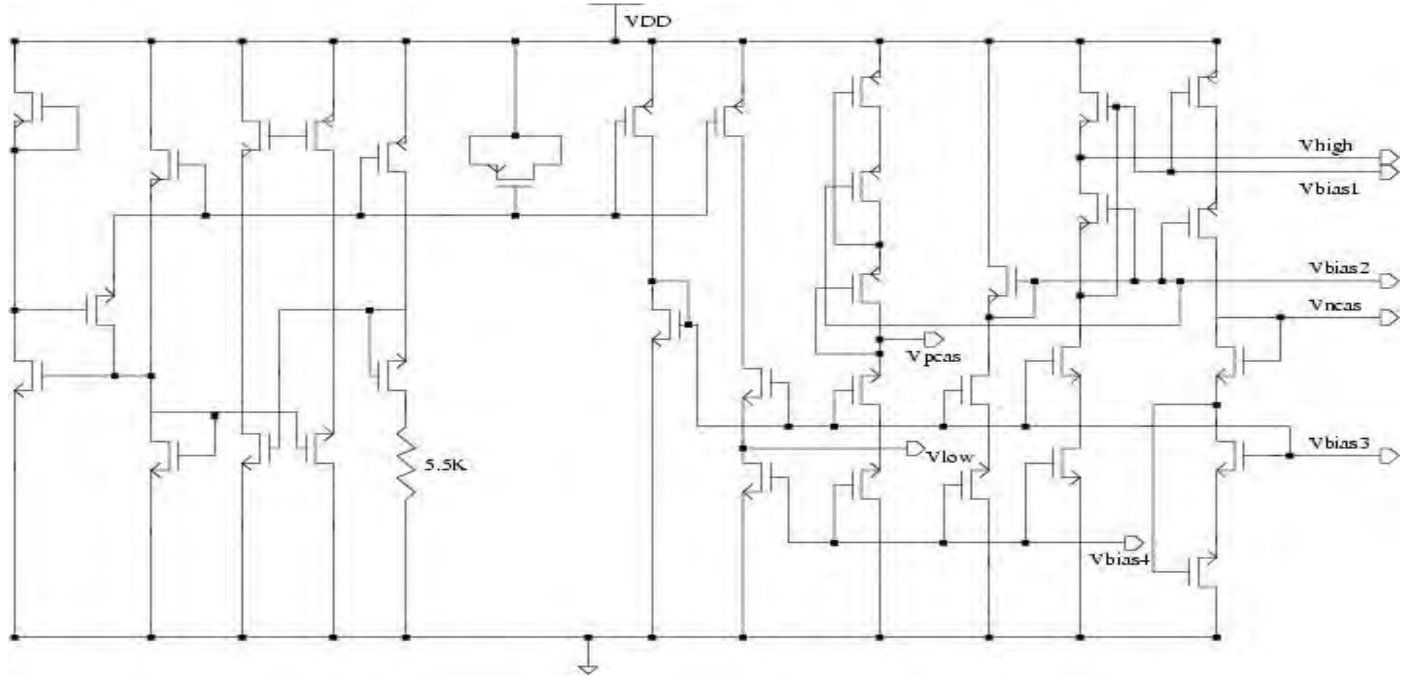
- All components are designed in CMOS transistor level design using BSIM4 Model.

- Differential Amplifier and Operational Amplifier are designed in 50 nm.

- Comparator is designed in 1um technology.

Design of Differential Amplifier (Diff-Amp)

- Differential amplifier is a circuit which compares two input signals and amplifies the difference between them.
- Diff-amp is designed in 50 nm technology using BSIM4 model file.



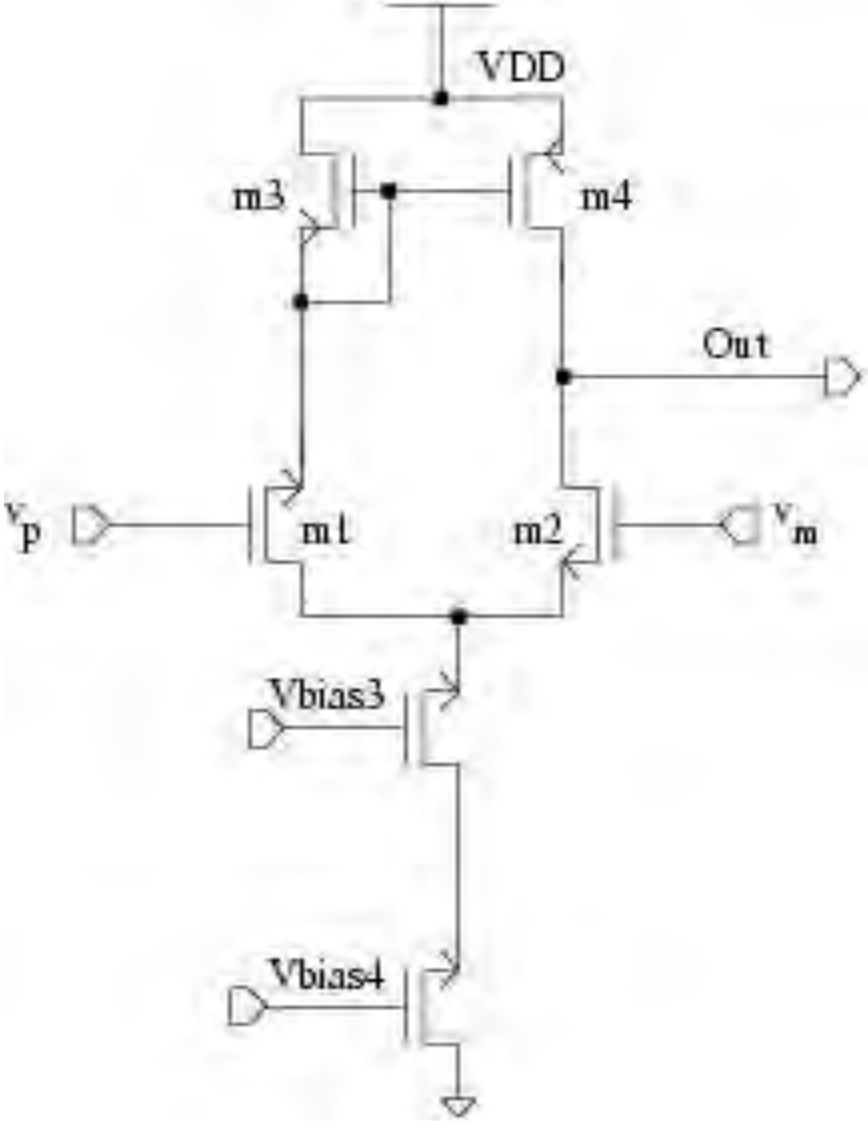
Circuit Diagram for Short-channel Biasing Circuit

Differential Amplifier: Circuit

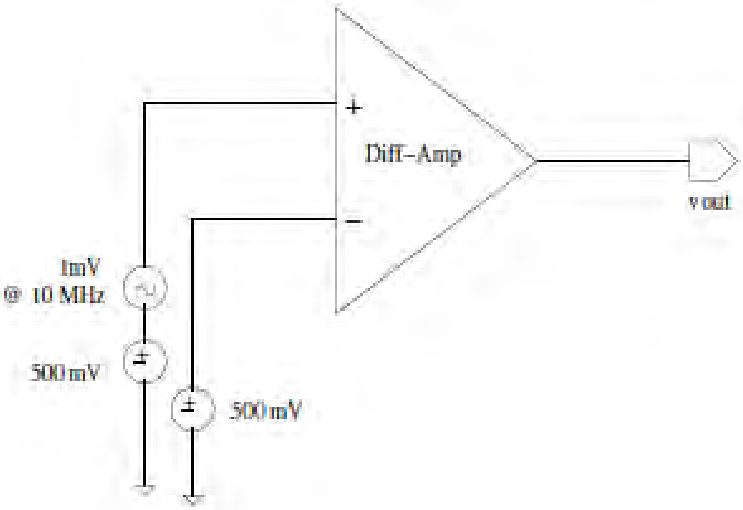
- Differential amplifier is a circuit which compares two input signals and amplifies the difference between them.

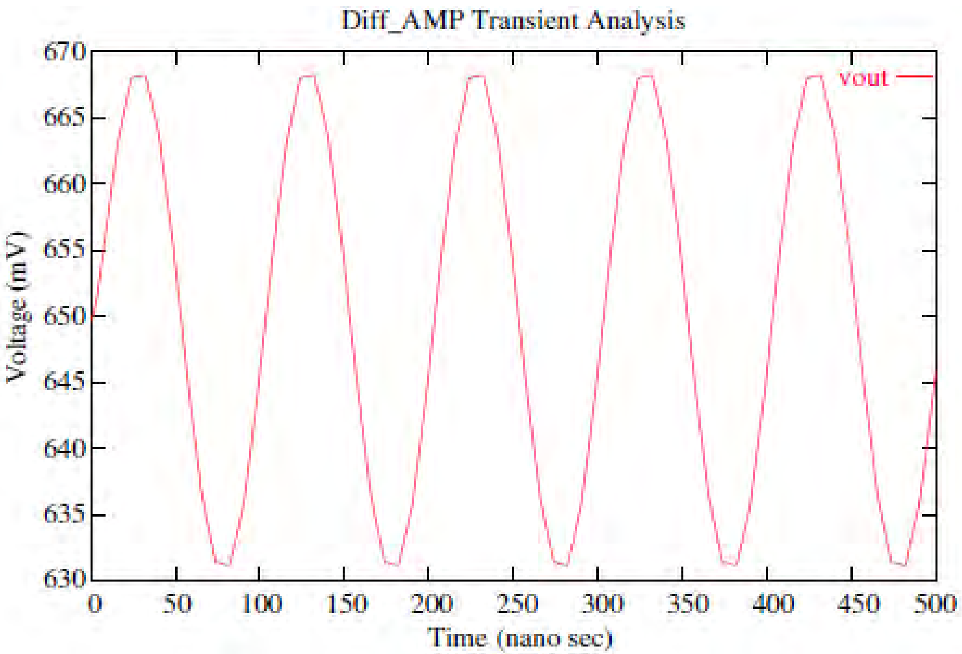
- The differential amplifiers have two gains one is common gain and the other is differential gain and the ratio of them gives Common Mode Rejection Ratio:

$$CMRR = 20log\left|\frac{A_d}{A_c}\right|$$

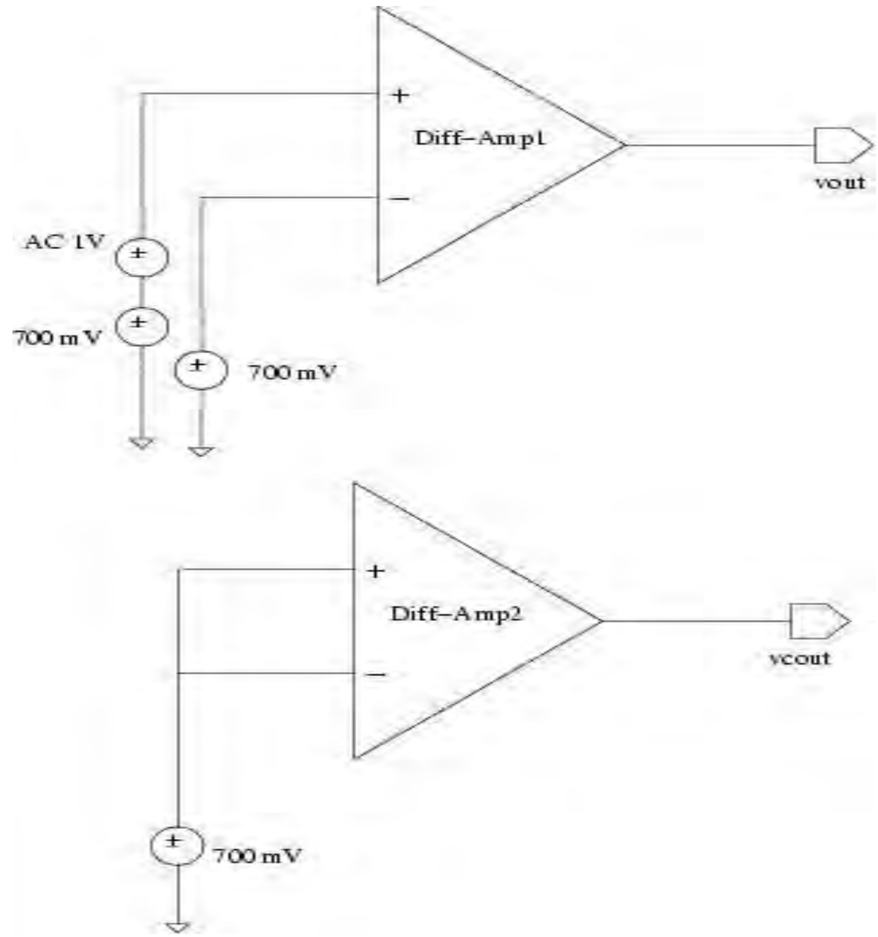


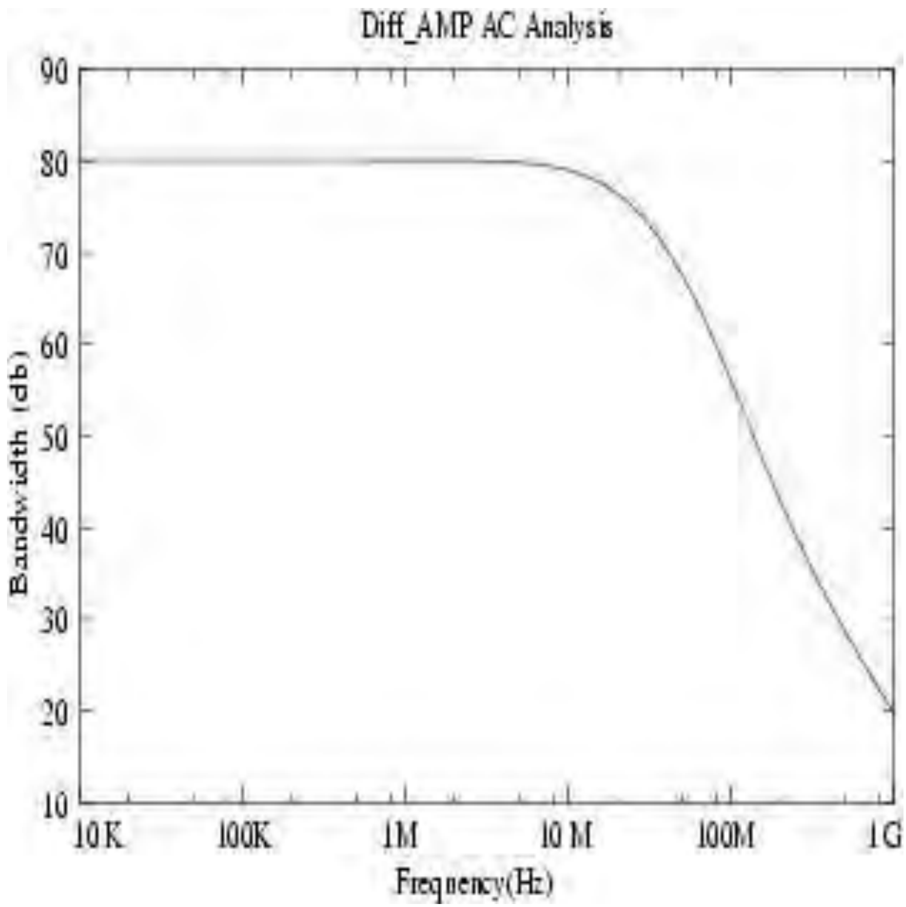
Differential Amplifier: Transient Analysis





Differential Amplifier : Common Mode Rejection Ratio



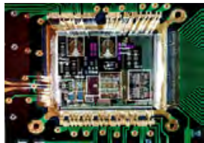


Block diagram of CMRR Diff-Amp

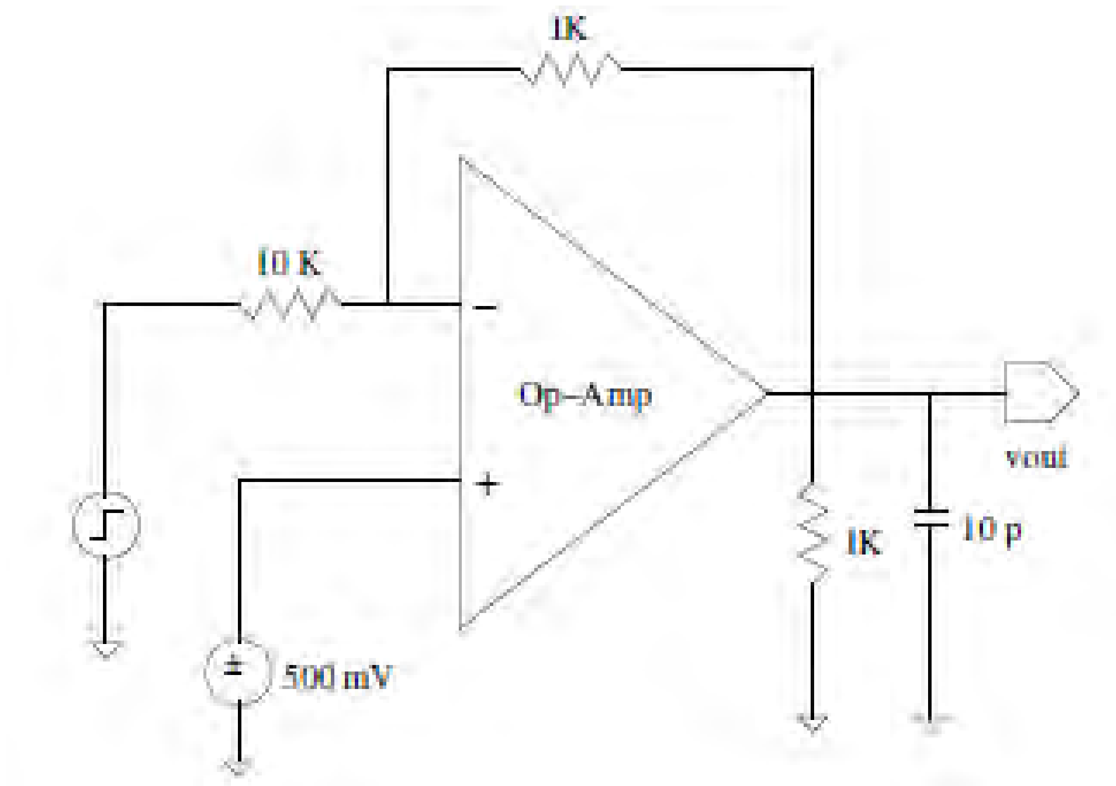
Output gain of CMRR Diff-Amp

Operational Amplifier

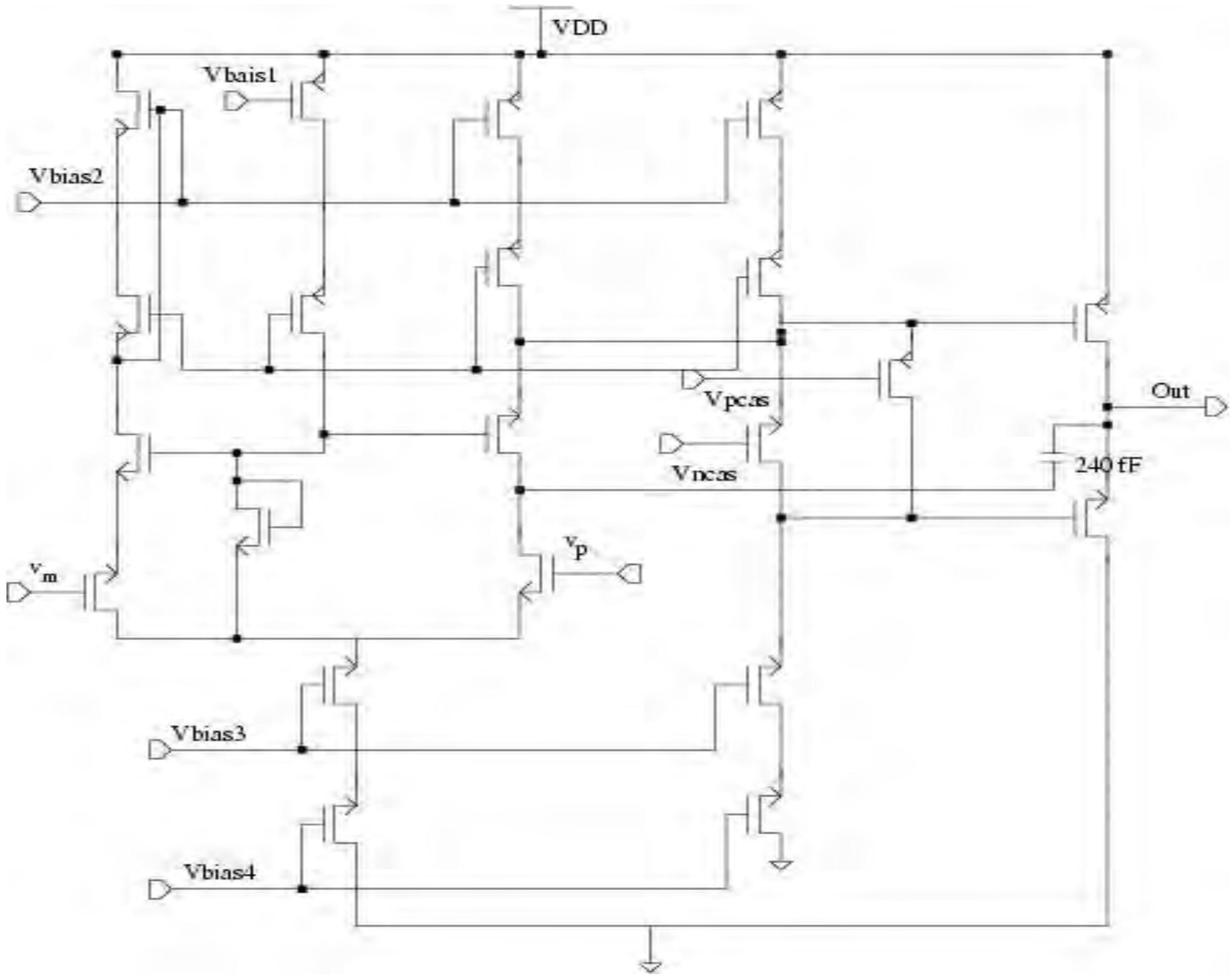
- To design a Operational Amplifier it requires a short-channel biasing circuit.
- The parameters given at scale for the NMOS the length L=2 and the width W=50; for PMOS length L=2 and the width W=100.
- Op-Amp is designed in 50 nm technology using BSIM4 model file.



Operational Amplifier: Circuit for AC Analysis

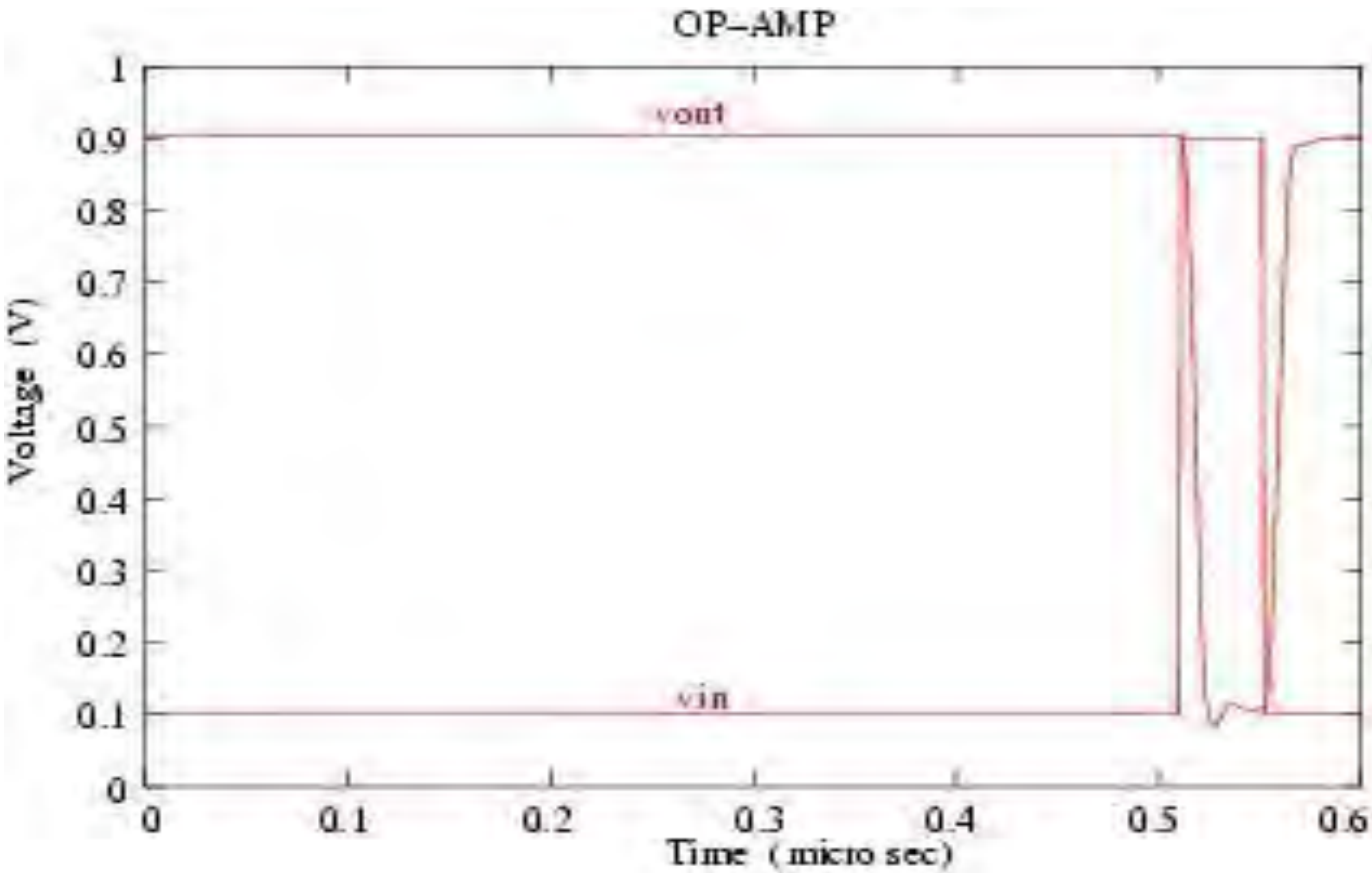


Operational Amplifier: Circuit Diagram



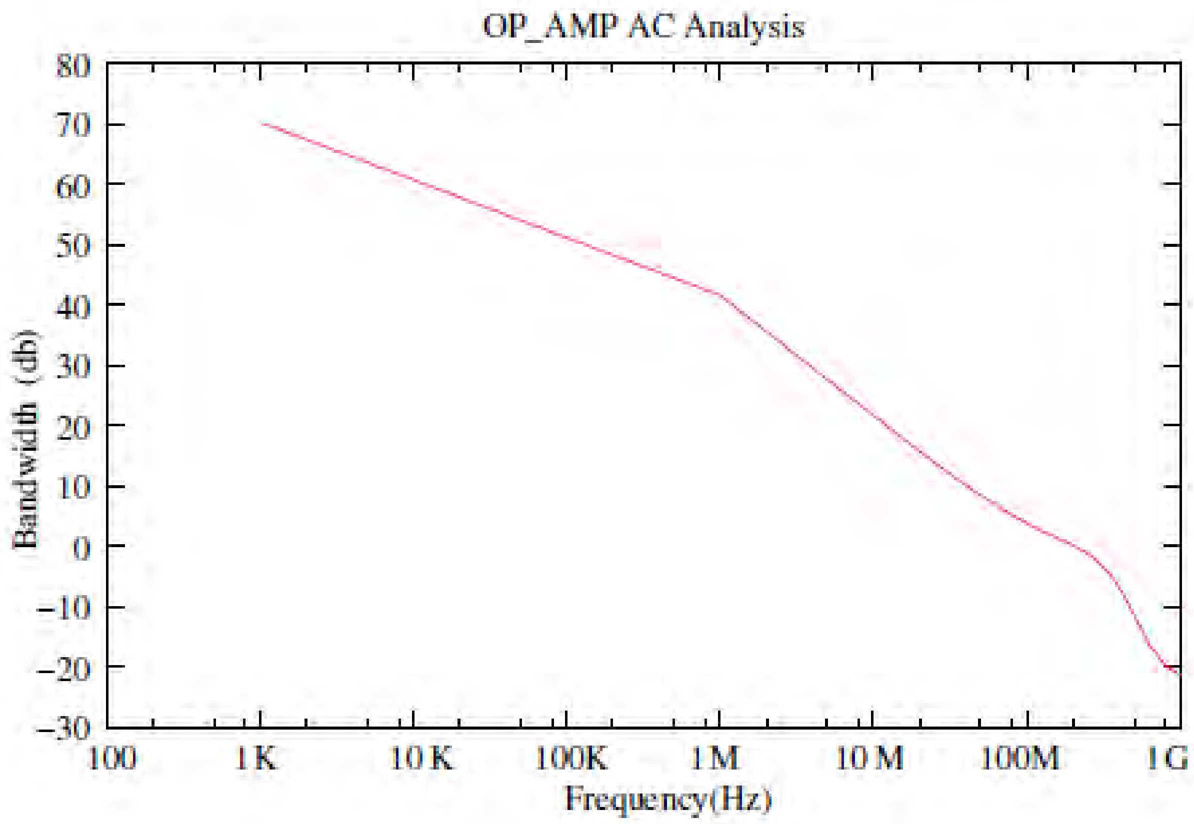
- The inputs Vbias1 ,vbias2 Vbias3 Vbias4, Vpcas and Vncas are the outputs of the short-channel biasing circuit.

Operational Amplifier: Transient Analysis

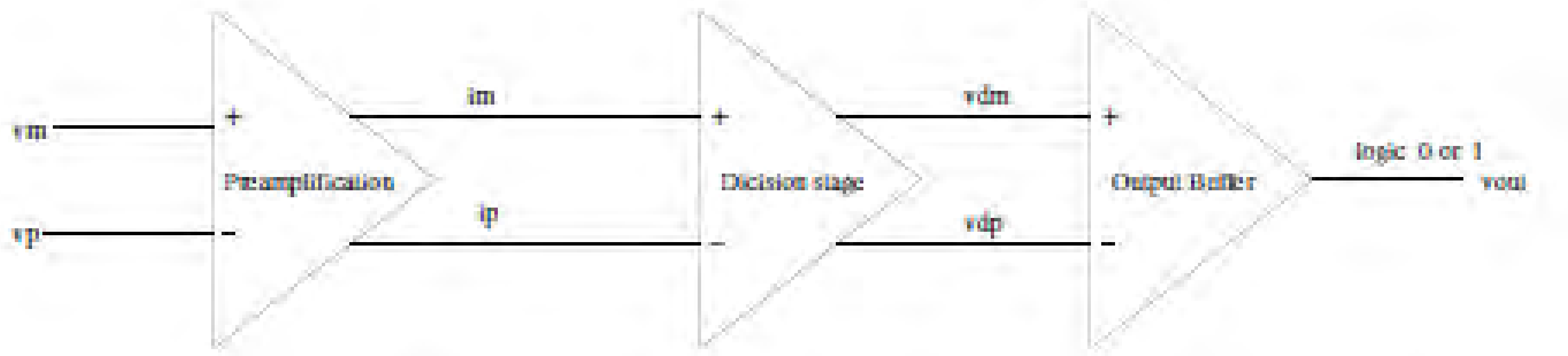


Here vin is pulse voltage of 0.1 V to 0.9 V at a delay of 510nsec and vm is 0.5 V.

Operational Amplifier: Frequency Vs Bandwidth

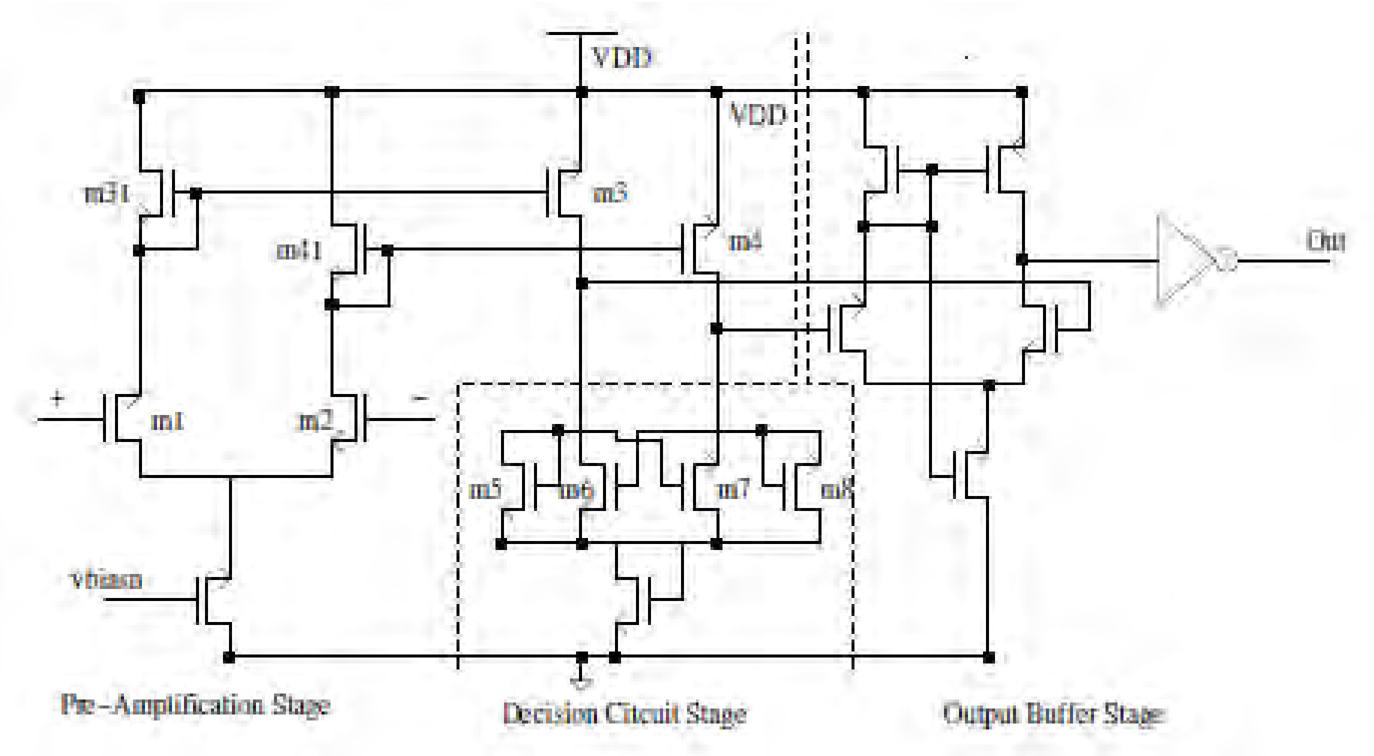


Comparator: Block Diagram



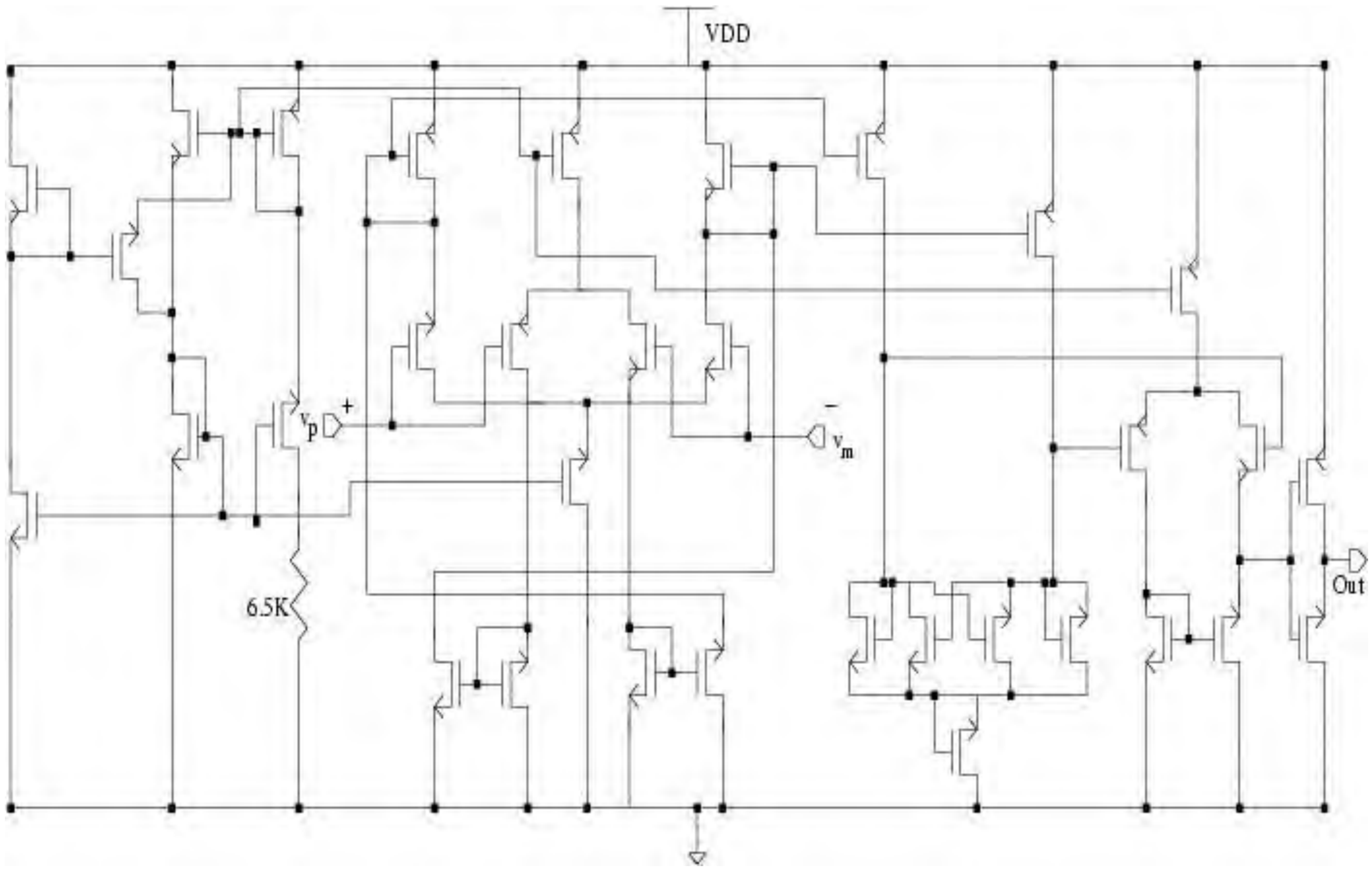
- The comparator mainly consists of 3 stages: preamplication, decision stage and the output buffer.
- The preamplication stage amplifies the input signal to improve the comparator sensitivity.
- The decision stage determines which of the input signal is larger.
- The output buffer amplifies the signal and gives the output.

Comparator: Circuit Diagram

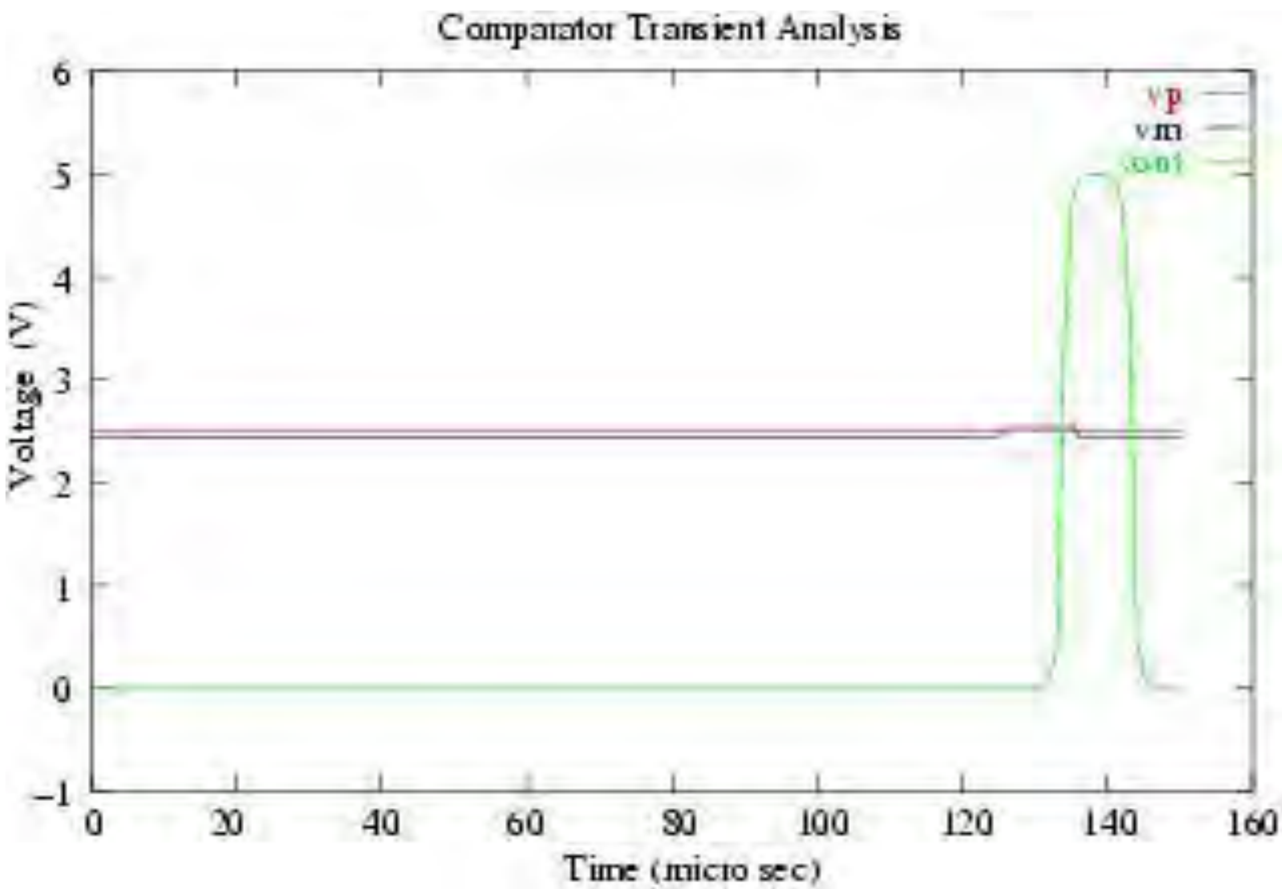


Comparator: Circuit Diagram …

- The working of comparator is if vp > vm then logic of the circuit is VDD otherwise 0.
- The circuit diagram of comparator is shown.



Comparator : Transient analysis



Here vp is 2.5 v and vm is pulse voltage of 2.45 v to 2.55 v at a delay of 120 micro sec using BSIM4 model file.

Comparative view of Various Analog Circuit Simulators

- Compare and evaluate the circuit simulators by considering the benchmark circuits.

- Simulate the benchmarks and plot the outputs.

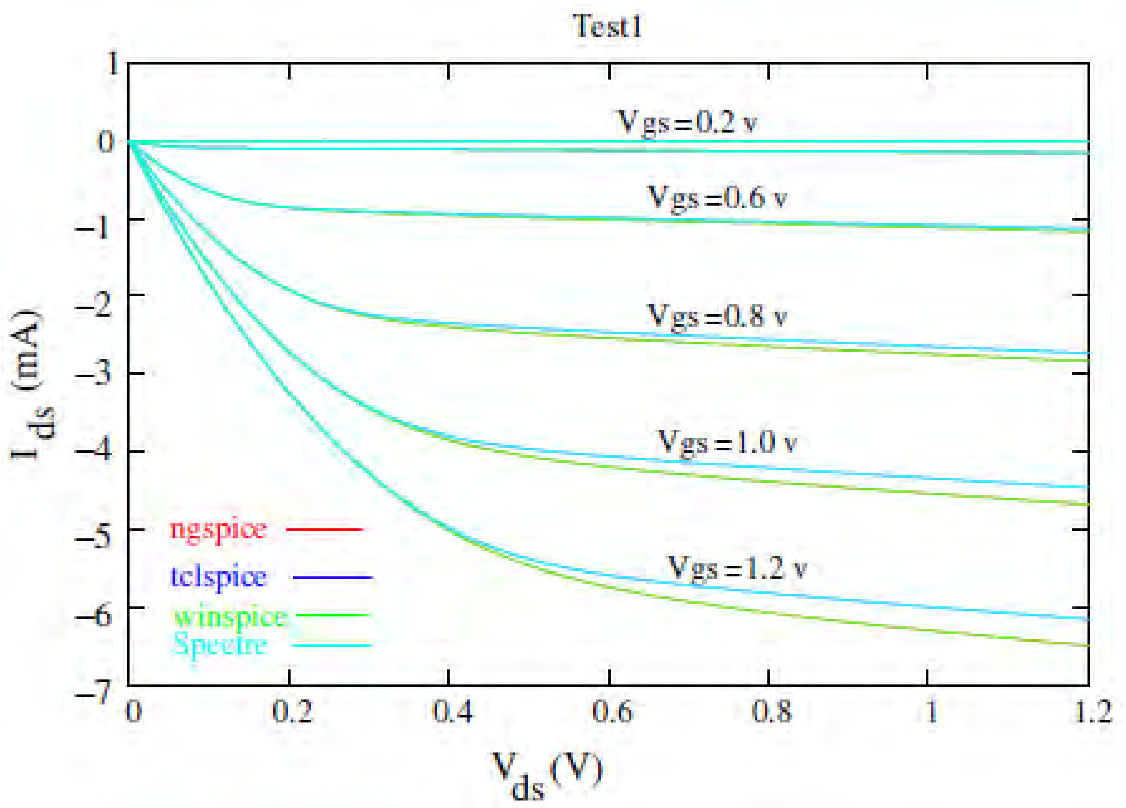
- Compare and evaluate the benchmarks by considering the metric quantities such as Execution Time, Accuracy and Convergence.

Execution Time : Comparison

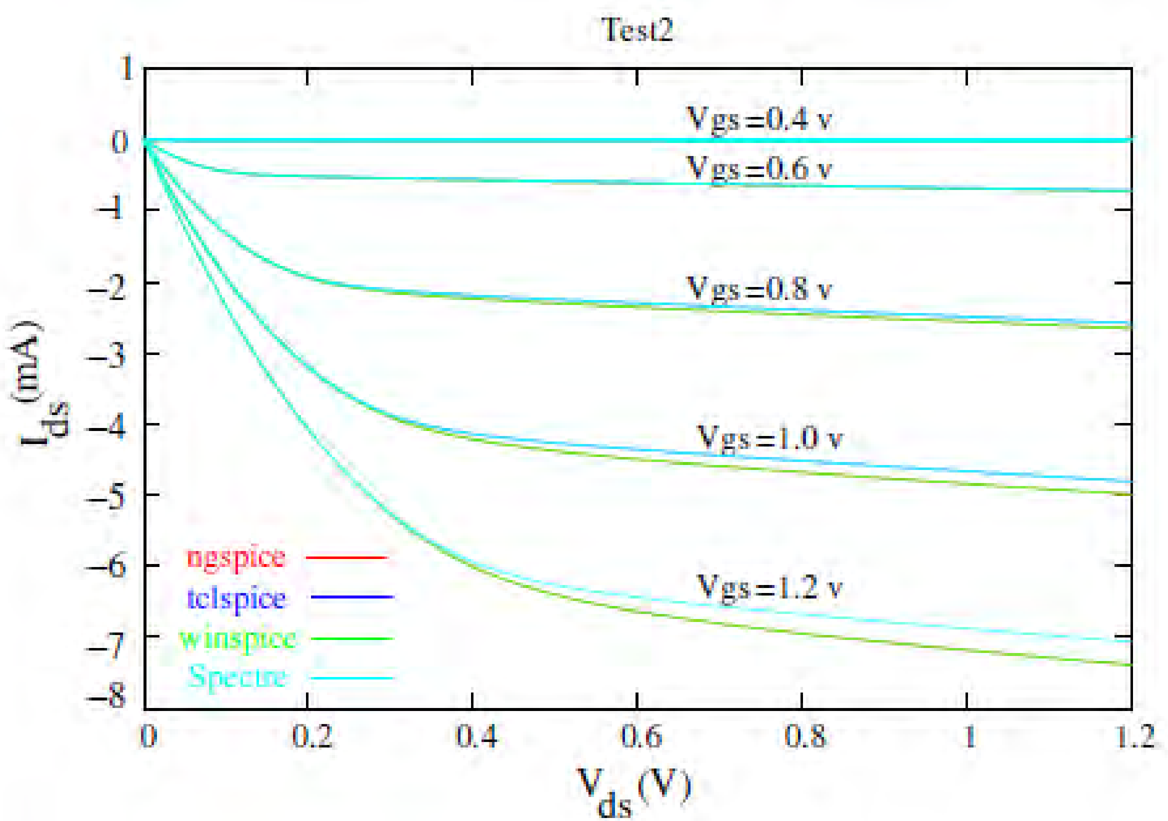
Benchmarks	Circuit Simulator (Time in seconds)			
	Ngspice	Tclspice	Winspice	Spectre
Test1	0.059	0.07	0.203	0.11
Test2	0.045	0.12	0.234	0.16
Test3	0.044	0.12	0.235	0.13
Test4	0.29	0.12	0.109	0.12
Test5	0.45	0.21	0.281	0.17
Test6	0.45	1.48	0.25	0.14
Test7	0.044	0.09	0.25	0.14
Test8	0.03	0.1	0.141	0.16
Test9	0.046	0.1	0.25	0.15
Test10	0.046	0.09	0.25	0.18
Test11	0.048	0.11	0.25	0.15
gstage	0.017	0.11	0.15	0.09
R0_17	2.634	0.14	3.532	1.86
Comparator			0.15	0.02
Op-Amp			24	0.48

Execution Time Table

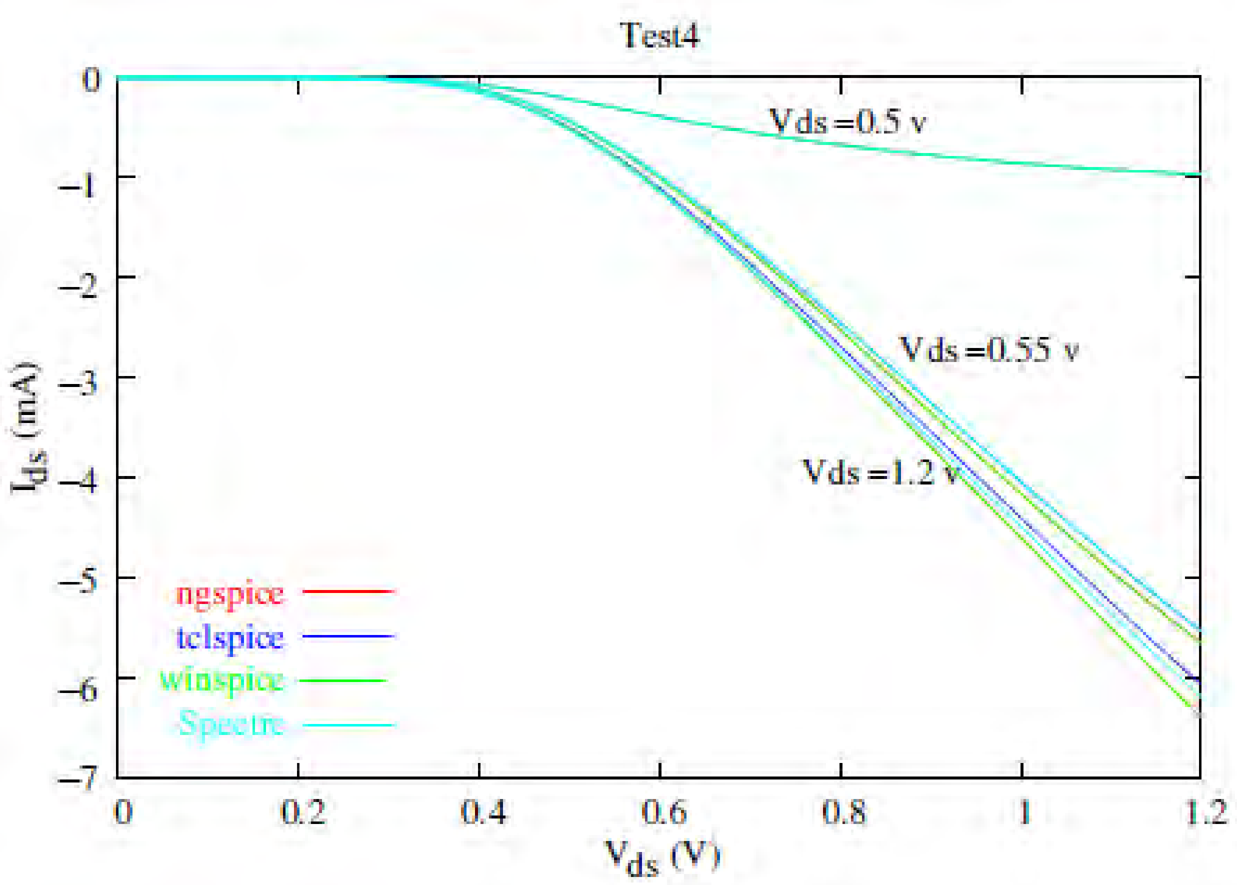
Output for Benchmark Test 1



Output for Benchmark Test 2



Output for Benchmark Test 4

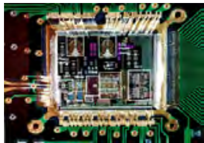


Accuracy Comparison

- The Accuracy of a circuit simulator is determined by calculating the outputs of the benchmarks and comparing the same with all the available circuit simulators.

- Compare the accuracy of various circuit simulators with that of the Spectre.

- Spectre is a efficient and stable circuit simulator.

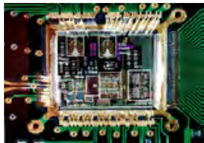


Convergence Comparison

- Convergence is calculated by changing the tolerance values for the benchmarks.

- By changing the tolerance value, it will change number of iterations then it change the convergence.

- For all the circuit simulators, the default tolerance values are as follows:
  - retol=1e-3
  - vabstol=1e-6
  - iabstol=1e-12



Conclusions

- Designed Sigma-Delta Modulator and individual components of Sigma-Delta Modulator.
- Compared and evaluated the existing analog circuits by considering some benchmarks.
- The proposed design of Sigma-Delta Modulator and its individual components of it can be used for the design of Sigma-Delta ADC.
- The comparison and evaluation of existing analog Circuit simulators is performed.
- In principle can be extends for a new Circuit Simulators.