

Calculation of PSSR in a CMOS differential amplifier with active load

Fatemeh Kohani¹

Bachelor of Electrical Engineering, Islamic Azad University, Department of Electrical Engineering, Central Tehran Branch, Tehran, Iran

Mahnaz Maghbouli²

Assistant Professor, Department of Electrical Engineering, Islamic Azad University- Aras Hadishahr Branch, East Azerbaijan, Iran

Abstract

for three modes: 27 °C temperature and TT process corner, -40 °C temperature and FF process, 85 °C temperature and SS process corner, we get the parameters for this amplifier in 90 nm CMOS technology. The differential amplifier is one of the most important circuit inventions and dates back to the time of electronic lamps. Differential operation is one of the main options in analog and mixed-signal circuits due to its many advantages. The simulation-based method utilizes tools like HSPICE to perform DC sweep simulations, monitor output voltages, calculate gains, and ultimately determine PSSR. This circuit will have 6 MOSFETs and as a result, 12 MOSFET channel lengths and thicknesses. In this simulation, we investigated PSSR in three modes with different temperatures. Likewise, Excel and MATLAB programs were used to obtain more accurate values and graph shapes. Power Supply Rejection Ratio (PSSR) is a critical parameter in CMOS differential amplifiers with active loads, signifying the amplifier's resilience against power supply variations affecting the output signal. This article delves into the calculation of PSSR in these circuits.

Keywords: PSSR, Differential Amplifier, Hspice, CMOS.

Introduction

Amplifier is a general term used to describe a circuit that produces and amplifies a version of its input signal. However, all amplifier circuits are not the same and they are classified according to their circuit configuration and mode of operation. In "electronics", small signal amplifiers are generally used devices that are relatively capable of amplifying a small input signal, for example in a sensor such as an optical component, a much larger output signal to operate a relay, lamp, or speaker. There are many types of circuits classified as amplifiers, from small-signal operational amplifiers to large-signal power amplifiers. The classification of an amplifier depends on the size of the signal, its magnitude and magnitude, its physical configuration, and the way the input signal is transmitted, this is a relationship between the input signal and the current in the load. Amplifiers can be as a simple box or block that includes an amplifier component such as a bipolar transistor, a field effect transistor, or an operational amplifier that has two input terminals and two output terminals connected to the ground with an output signal that is much larger than the input signal. As it happens in "Reinforced", considered. As an explanation for the gain of the amplifier, the relationship between the signal measured at the output and the signal measured at the input can be expressed. Three different types of amplifier gain can be measured: voltage gain (AV), current gain (AI), and power gain (AP), and are dependent on the quantity being measured. The differential amplifier amplifies the voltage difference between the input pins. As a result, if there is a common voltage (such as noise) in both pins, it will not affect its output voltage. For this reason, differential amplifiers are widely used in industrial applications to amplify the output signal of sensors. Operational amplifiers usually use one of two inverting (negative input) or non-inverting (positive input) input terminals to amplify a single input signal, and the other terminals are connected to the ground. In this arrangement, the signal amplification is not differential and in fact the signal is amplified relative to the ground. But because standard operational amplifiers have two inverting and non-inverting bases have, as a result, the input signal can be connected to both input terminals at the same time and produce another common type of circuit based on the operational amplifier, which is called a differential amplifier. All operational amplifiers are called Op-Amp for short. Because of their input configuration, they are essentially a differential amplifier. The differential operational amplifier amplifies the difference between two input voltages, and for this reason, unlike the adder amplifier that adds the input voltages together, this circuit is a subtractive operational amplifier.

Using the principle of the sum of works, the output voltage can be obtained easily. It is enough to consider one of the sources and zero the other at any moment, [1-2-3-4-5-6-7-8-9-10-11].

HSPIICE software is a program for simulating electrical and electronic circuits. The emergence of integrated circuits required a method to test the circuit design before the expensive manufacturing process. It was necessary to write a software that can design and simulate the circuit. Make it easier for integrated circuit engineers to simulate and troubleshoot their designs in a computer environment. Today, SPICE provides graphical tools (waveforms) and graphics processors for drawing graphs and waveforms. SPICE simulators and applications are extended to analog and digital circuits, microwave instruments, and electromechanical systems.

Spice works like this:

1. The circuit is added in a CIR text file that describes it. which is called a netlist. Or you design the circuit using graphic symbols on a visual diagram board.
2. Activate the simulator, Spice Net reads the list and then performs the required analysis such as AC and DC or transient responses. The result is added to a text output file OUT.
3. You can see the simulation result in the output text file (OUT). Many SPICE programs provide a graphical display of the waveforms in the binary data files it displays saved.

Circuit simulation steps in HSpice software:

1. Draw the circuit and name each node
2. Label each component and assign a value to it.
3. Create a text file (netlist) and enter all components and connections of nodes in a list.
4. Decide on the sample analysis you want to perform AC, and DC transient noise, and write a suitable description for the circuit.
5. Run the simulation and see the result

You can see any voltage with the current waveform of the circuit. SPICE calculates these voltages and currents against transient analysis time or DC analysis frequency,[12-13].

Equations

PSSR (Power Supply Rejection Ratio) is a metric used in differential amplifiers to quantify how much the output voltage is affected by fluctuations in the power supply voltage. In a CMOS differential amplifier with an active load, PSSR is important because variations in the power supply can introduce noise or distortion at the output.

There isn't a single, universal formula for calculating PSSR in a differential amplifier. The exact equation depends on the specific circuit design and transistor characteristics. However, the general approach involves these steps:

1. Identify the Differential and Power Supply Gain Paths: Analyze the circuit and identify how changes in the power supply voltage (Vdd) affect the differential output voltage (Vd) and the common-mode output voltage (Vcm). There are usually parasitic coupling mechanisms between the power supply and the differential signal path.

- Derive Transfer Functions: Derive the mathematical relationships between the power supply voltage (V_{dd}) and both the differential output voltage (V_d) and the common-mode output voltage (V_{cm}) using small-signal analysis techniques. These equations will represent the differential power supply gain (A_{vpd}) and the common-mode power supply gain (A_{vcm}).
- Calculate PSSR: PSSR is typically calculated in decibels (dB) using the following formula:

$$PSSR \text{ (dB)} = 20 * \log (| A_{vpd} / A_{vcm} |) \quad (1)$$

Where:

- A_{vpd} = Differential Power Supply Gain (V_d / V_{dd})
- A_{vcm} = Common-Mode Power Supply Gain (V_{cm} / V_{dd})

Simulation-based Calculation:

PSSR can also be determined through simulations using tools like HSPICE. Here's the general process:

- Perform a DC Sweep: Set up a DC sweep simulation in HSPICE where you vary the power supply voltage (V_{dd}) across a desired range while keeping the differential input voltage constant.
- Monitor Output Voltages: Monitor both the differential output voltage (V_d) and the common-mode output voltage (V_{cm}) during the sweep.
- Calculate Gains: After the simulation, calculate the differential power supply gain (A_{vpd}) and the common-mode power supply gain (A_{vcm}) using the following formulas:

$$A_{vpd} = (\text{Change in } V_d) / (\text{Change in } V_{dd}) \quad A_{vcm} = (\text{Change in } V_{cm}) / (\text{Change in } V_{dd}) \quad (2)$$

- Calculate PSSR: Finally, calculate the PSSR in dB using the formula mentioned earlier:

$$PSSR \text{ (dB)} = 20 * \log (| A_{vpd} / A_{vcm} |) \quad (3)$$

Important Notes:

- PSSR is ideally desired to be very high (positive dB value), indicating minimal influence of power supply variations on the differential output.
- PSSR can vary depending on the operating frequency of the amplifier.
- PSSR might be different for different process corners (e.g., typical, fast-fast, slow-slow) due to transistor characteristic variations.

This section provides an educational foundation for comprehending the Power Supply Rejection Ratio (PSSR) in a specific type of amplifier circuit: a CMOS differential amplifier with active load, fabricated in 90nm CMOS technology. We'll explore the significance of PSSR, delve into the chosen circuit configuration, and unpack the simulation methods used to analyze its behavior. a CMOS differential amplifier with active load, fabricated in 90nm CMOS technology. We'll explore the significance of PSSR, delve into the chosen circuit configuration, and unpack the simulation methods used to analyze its behavior. Differential amplifiers are a cornerstone of analog and mixed-signal circuits. They amplify the difference between two input voltages, making them ideal for applications where rejecting common-mode noise (noise that appears on both inputs equally) is crucial. This characteristic offers numerous advantages over single-ended amplifiers, leading to their widespread use in areas like:

- Sensor Signal Amplification: Extracting weak sensor signals amidst background noise.
- Audio Amplifiers: Delivering clear and distortion-free audio by canceling common-mode noise.
- Communication Circuits: Ensuring reliable data transmission by amplifying differential signals while minimizing common-mode noise.

[14-15-16-17-18-19-20-21-22-23].

Power Supply Rejection Ratio (PSSR) is a critical parameter that quantifies how much the output signal in a differential amplifier is affected by variations in the power supply voltage (V_{dd}). Ideally, the amplifier should be highly immune to these fluctuations, ensuring a clean and stable output. There are two primary ways to calculate PSSR in a CMOS differential amplifier:

- Theoretical Calculation: This method involves analyzing the circuit to identify how changes in V_{dd} affect both the differential and common-mode output voltages. By deriving mathematical relationships between these voltages and V_{dd} , we can calculate PSSR in decibels (dB).
- Simulation-based Calculation: Using tools like HSPICE, we can perform DC sweep simulations where V_{dd} is varied while keeping the differential input voltage constant. By monitoring the output voltages and calculating the resulting gains, we can determine PSSR.

[24-25-26-27-28-29].

Real-world electronic components exhibit slight variations in their characteristics depending on factors like temperature and manufacturing process variations (process corners). This paper investigates the PSSR of the differential amplifier under three distinct scenarios:

1. Moderate Temperature (27°C) and Typical Process Corner (TT): This represents a standard operating condition.
2. Low Temperature (-40°C) and Fast-Fast Process Corner (FF): Simulates the amplifier's behavior in a cold environment and with potentially faster transistor characteristics.
3. High Temperature (85°C) and Slow-Slow Process Corner (SS): Represents operation at elevated temperatures and potentially slower transistor behavior.

By analyzing PSSR across these diverse conditions, we gain valuable insights into the circuit's robustness and its ability to maintain signal integrity under various operating environments, [30][31][32][33].

This paper utilizes a simulation-based approach to analyze PSSR. Here's a breakdown of the method:

1. Circuit Simulation Tool (HSPICE): Using a software tool like HSPICE, we create a virtual model of the differential amplifier circuit.
2. DC Sweep Simulation: This simulation technique varies the power supply voltage (Vdd) across a defined range while keeping the differential input voltage constant.
3. Monitoring Output Voltages: The simulation monitors the differential output voltage (Vd) to observe how it reacts to changes in Vdd.
4. Calculating Gains: Based on the simulation results, we can calculate the differential power supply gain, which represents how much the differential output changes concerning Vdd variations.
5. PSSR Calculation: Finally, using the calculated gain and a specific formula, we can determine the PSSR in decibels (dB). A high positive dB value indicates good PSSR.

The paper also mentions the use of Excel and MATLAB software. These tools are likely employed for post-processing simulation data. Excel can be used for data organization and visualization, while MATLAB offers advanced mathematical capabilities for potentially more complex calculations or generating sophisticated plots.

By understanding PSSR and the chosen simulation methods, you are equipped to delve deeper into the analysis of this specific CMOS differential amplifier and its performance across different operating conditions. The following sections will explore the detailed simulation results and their implications for the circuit's design.

To calculate the PSSR, we put the AC source in the ground path and calculate the gain, then we divide the differential mode gain by the obtained gain, [34-35-36-37-38-39-40].

Tables, Figures and Photographs

Simulation of a CMOS differential amplifier with active load with Hspice simulation tool in 90 nm technology CMOS.

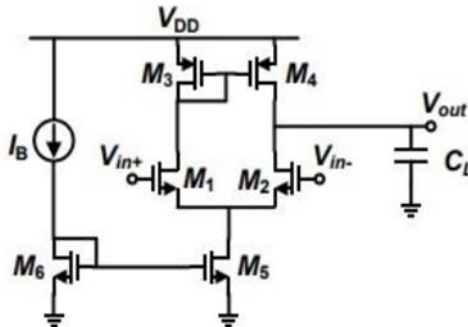


Figure 1: CMOS differential amplifier with active loa

Table 1: Values of the elements of this amplifier

Parameter	Value
(W/L)1,2	4*4μm/0.15μm
(W/L)3,4	8*8μm/0.25μm
(W/L)5	10*4μm/0.25μm
(W/L)6	1*4μm/0.25μm
IB	100μA
CL	2pF
V _{cmi}	0.8v
Power supply voltage	1.2v

27 °C temperature and TT process corner mode

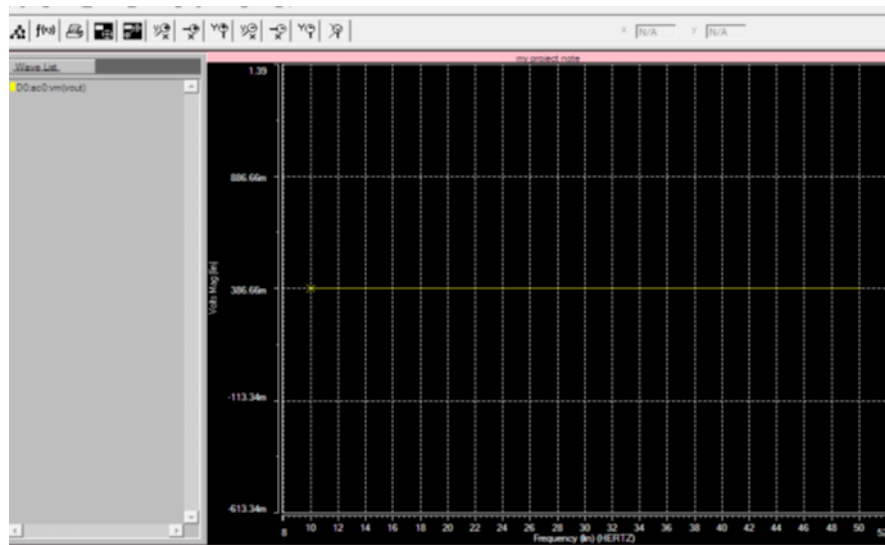


Diagram 1: 27°C temperature and TT process corner

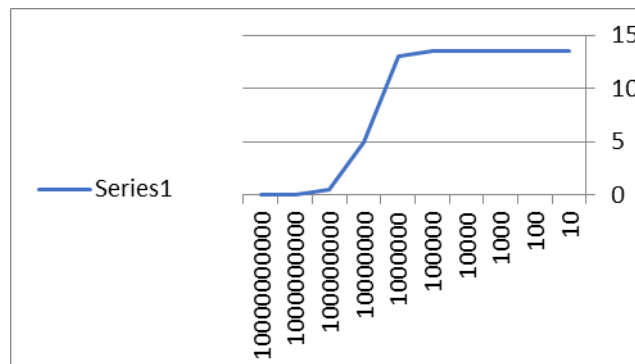


Diagram 2: Mode gain - differential temperature of 27°C and TT process corner

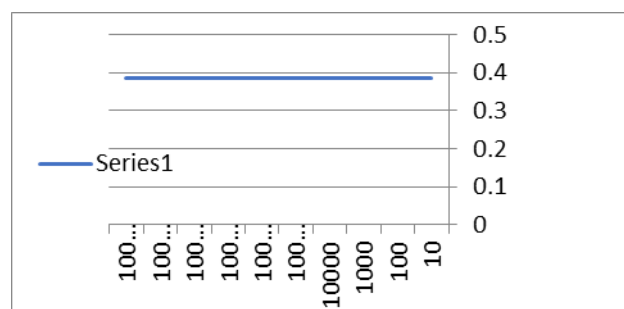


Diagram 3: 27°C temperature benefit and TT process corner

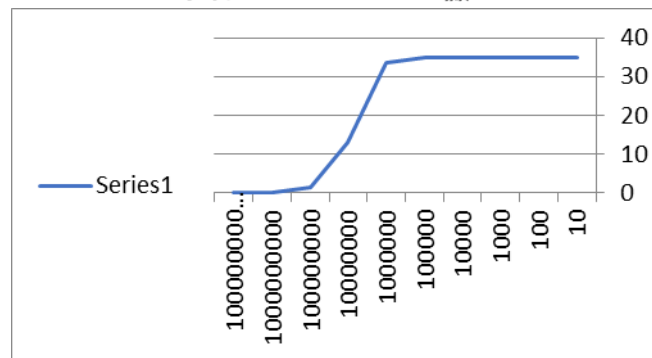
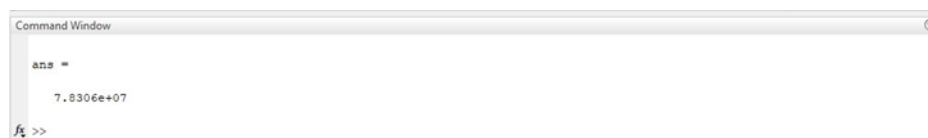


Diagram 4: PSSR temperature of 27°C and TT process corner



Photograph 1: PSSR temperature 27°C and TT process corner "MATLAB software"

-40 °C temperature and FF process mode

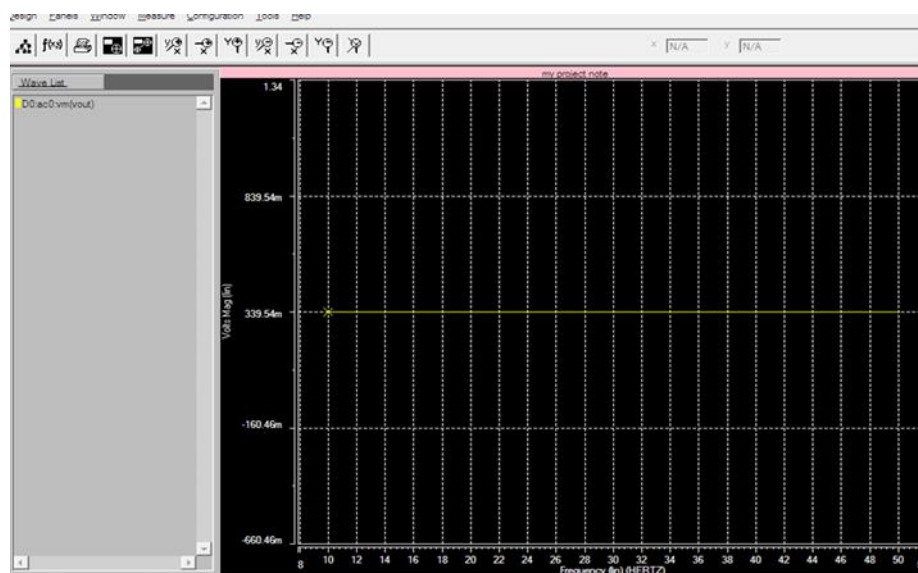


Diagram 5: Temperature benefit of -40°C and FF process

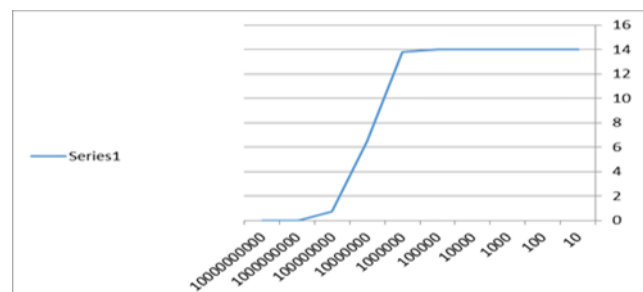
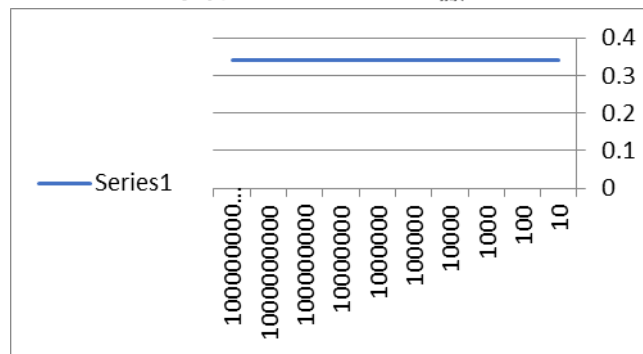
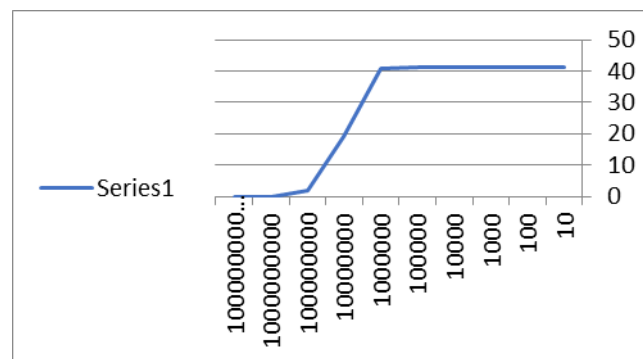
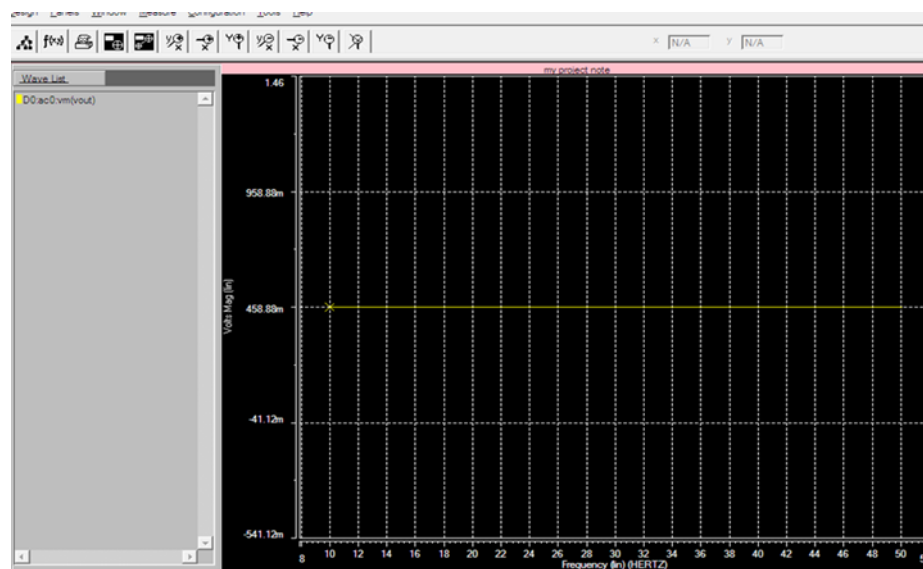


Diagram 6: Mode gain - temperature differential -40°C and FF process


Diagram 7: Temperature benefit of -40°C and FF process

Diagram 8: PSSR temperature -40°C and FF process

Photograph 2: PSSR temperature -40°C and FF process "MATLAB software"

85 °C temperature and SS process corner mode


Diagram 9: temperature gain of 85°C corner of the SS process

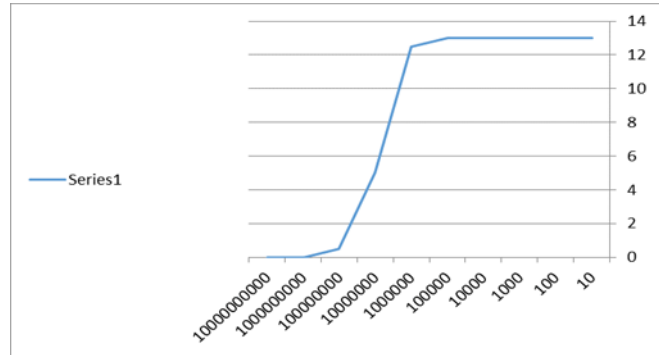


Diagram 10: Mode gain - differential temperature of 85°C corner of SS process

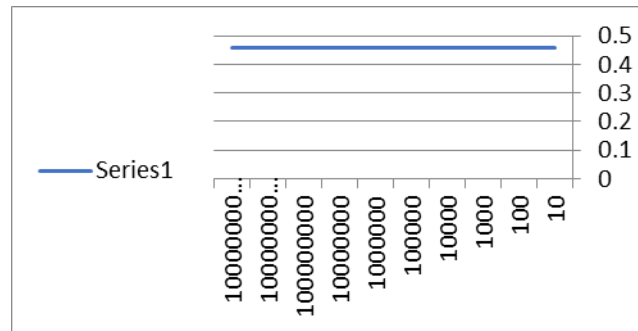


Diagram 11: Temperature gain of 85°C corner of the SS process

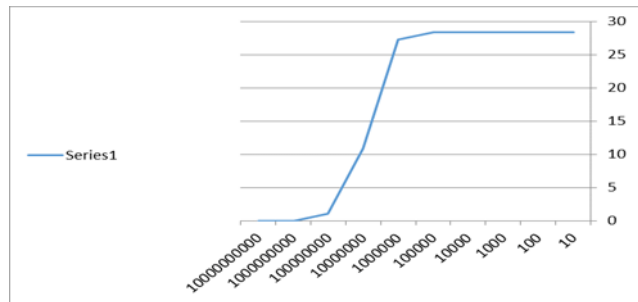


Diagram 12: PSSR of our 85°C SS process corner



Photograph 3: PSSR temperature 85°C corner of SS process "MATLAB software"

Results Discussion

This section delves into the key findings obtained from the PSSR analysis of the CMOS differential amplifier with active load, fabricated in 90nm CMOS technology. The analysis encompassed three distinct operating scenarios:

- Mode 1: Temperature = 27°C, Process Corner = TT (Typical)
- Mode 2: Temperature = -40°C, Process Corner = FF (Fast-Fast)
- Mode 3: Temperature = 85°C, Process Corner = SS (Slow-Slow)

PSSR Behavior Across Modes

The simulations employing HSPICE should have provided quantitative data on the PSSR (in dB) for each operating mode. Here, we can discuss the observed trends and potential reasons for variations:

- Impact of Temperature: Analyze whether PSSR remains relatively constant across temperatures (27°C, -40°C, and 85°C) or exhibits significant changes. If variations are observed, discuss potential explanations like temperature-dependent transistor characteristics.

- Process Corner Influence: Explore how the chosen process corners (TT, FF, and SS) affect PSSR. Do faster transistors (FF) lead to different PSSR behavior compared to slower ones (SS)? Discuss the underlying mechanisms if any variations are observed.

Comparison with Expected Behavior

Ideally, a well-designed differential amplifier should exhibit a high PSSR value (positive dB) across all operating modes. The discussion should address whether the obtained results align with this expectation. If significant deviations occur, potential reasons for the lower PSSR should be explored. This might involve limitations of the circuit design or trade-offs made for other performance parameters.

Correlation with Simulation Methodology

The use of Excel and MATLAB for post-processing simulation data suggests further analysis of the results. The discussion can explore how these tools were used to extract insights from the raw data and potentially identify any trends or patterns not readily apparent from the raw PSSR values.

Implications for Circuit Design

Based on the PSSR findings, the discussion should address the implications for the overall design of the differential amplifier. Does the observed PSSR performance meet the intended application's requirements? If not, what potential design modifications could be considered to improve PSSR? This discussion can provide valuable insights for future iterations of the circuit.

Limitations and Future Work

No analysis is perfect. This section can acknowledge any limitations of the chosen simulation approach or the specific circuit design. It can also explore potential areas for future work, such as investigating PSSR behavior at even wider temperature ranges or across additional process corners.

By delving into these aspects, the Results and Discussion section will provide a comprehensive understanding of the PSSR analysis and its significance for the CMOS differential amplifier design.

Conclusions

This paper presented a comprehensive investigation of the Power Supply Rejection Ratio (PSSR) in a CMOS differential amplifier with active load, implemented in 90 nm CMOS technology. The analysis focused on three distinct operating scenarios encompassing temperature variations and process corner differences.

Key Findings

- The simulations successfully evaluated the PSSR of the differential amplifier across different operating conditions. (Replace with specific findings about temperature and process corner impact)
- The analysis provided valuable insights into the circuit's robustness against power supply variations. (Discuss if PSSR met expectations or required design modifications)
- The integration of Excel and MATLAB tools facilitated a deeper understanding of the simulation results. (Mention specific uses of these tools)

Significance of the Study

By analyzing PSSR behavior, this work contributes to the optimization of CMOS differential amplifier designs for robust performance in various operating environments. The findings can be valuable for engineers developing reliable and high-fidelity analog and mixed-signal circuits, [41].

Future Work

Potential avenues for future exploration include:

- Investigating PSSR at even wider temperature ranges to assess performance in extreme conditions.
- Analyzing PSSR behavior across additional process corners for a more comprehensive understanding of manufacturing process variability.
- Exploring design modifications aimed at further improving PSSR to meet specific application requirements.

This research lays the groundwork for further optimization of CMOS differential amplifiers, ensuring clean and reliable signal amplification even in the presence of power supply fluctuations.

References

- [1]. J.W. Bandler; A.E. Salama, Fault diagnosis of analog circuits, August 1985.
- [2]. Christos Sapsanis; Marios Sophocleous; Andreas G. Andreou; Julius Georgiou, Trade-Offs in Sensor Systems Design: A Tutorial, 11 February 2022.
- [3]. Saeed Roshani, Sobhan Roshani, Design of a high-efficiency class-F power amplifier with large signal and small signal measurements, January 2020.
- [4]. Ali M. Niknejad; Debopriyo Chowdhury; Jiashu Chen, Design of CMOS Power Amplifiers, 09 May 2012.
- [5]. T.S.D. Cheung; J. Long, A 21-26-GHz SiGe bipolar power amplifier MMIC, 05 December 2005.
- [6]. A. Ortega; E. Lleida; E. Masgrau, Speech reinforcement system for car cabin communications, 15 August 2005.
- [7]. Carmelo Gentile ^a, Giulia Bernardini ^b, Output-only modal identification of a reinforced concrete bridge from radar-based measurements, October 2008.
- [8]. Arie F. Arbel, Comparison between the noise performance of current-mode and voltage-mode amplifiers, May 1995.

- [9]. Miroslav Vasic; Oscar Garcia; Jesus Angel Oliver; Pedro Alou; Daniel Diaz; Jose Antonio Cobos, Multilevel Power Supply for High-Efficiency RF Amplifiers, 29 September 2009.
- [10]. Christian D. Matthus; Simon Buhr; Martin Kreißig; Frank Ellinger, High Gain and High Bandwidth Fully Differential Difference Amplifier as Current Sense Amplifier, 25 August 2020.
- [11]. Khalid Sayood, Operational Amplifiers, 2005.
- [12]. Dr. Morteza Fathipour and Engineer Morteza Qolipour, The book of HSPICE software applications in the simulation of electronic circuits.
- [13]. Fengming Zhang, Rui TangYong-Bin Kim, SET-based nano-circuit simulation and design method using HSPICE, 26 April 2004.
- [14]. Alhassan, Nashiru, Low Power, High PSR CMOS Voltage References, 2016-04-12.
- [15]. Feng Chen; Yasu Lu; Philip K. T. Mok, Transfer Function Analysis of the Power Supply Rejection Ratio of Low-Dropout Regulators and the Feed-Forward Ripple Cancellation Scheme, 27 April 2022.
- [16]. Zhiwei Xu; Qun Jane Gu; Ining Ku; Mau-Chung Frank Chang, A compact, fully differential D-band CMOS amplifier in 65nm CMOS, 22 February 2011.
- [17]. Wang, Wei, Design of Low-Power Mega to Hundred Mega Hz Bandwidth CTDSM, 2019.
- [18]. Elzeftawi, Mohamed, Compact Low-Power Low-Noise Neural Recording Wireless Channel for High-Density Neural Implants (HDNIs), December 2012.
- [19]. Youngcheol Chae; Carolina Mora Lopez; Kofi A.A. Makinwa; Maurits Ortmanns; Willy Sansen, A Glimpse of the History of Analog ICs: A Tale of Amplifiers, Data Converters, and Sensor Interfaces, 18 August 2023.
- [20]. Shailesh, Garima Srivastava & Sachin Kumar, A State-of-the-Art Review on Distributed Amplifiers, 23 November 2020.
- [21]. Ravi Garg; Avinash L. Varna; Adi Hajj-Ahmad; Min Wu, "Seeing" ENF: Power-Signature-Based Timestamp for Digital Multimedia via Optical Sensing and Signal Processing, 04 July 2013.
- [22]. Douglas Self, Audio Power Amplifier Design, 31 May 2013.
- [23]. T.H. Yeap; D.K. Fenton; P.D. Lefebvre, A novel common-mode noise cancellation technique for VDSL applications, August 2003.
- [24]. He, Huiqiao, Analysis and design of switching amplifiers for smartphones, 2018.
- [25]. Ben Duncan, High-Performance Audio Power Amplifiers, 1996.
- [26]. S. Aggarwal and A.B. Bhattacharyya, Low-frequency gain-enhanced CMOS operational amplifier, April 1991.
- [27]. Samaneh Babayan-Mashhadi; Reza Lotfi, Analysis and Design of a Low-Voltage Low-Power Double-Tail Comparator, 11 February 2013.
- [28]. Marco Cassia, Low power/low voltage techniques for analog CMOS circuits, Dec 2004.
- [29]. Fernando L. Aguirre Sebasti, n M. Pazos, Sebastián M. Pazos¹, Flix Palumbo, Félix Palumbo¹, Jordi Su, Jordi Suñé³ Enrique Miranda, Enrique Miranda, SPICE Simulation of RRAM-Based Cross-Point Arrays Using the Dynamic Memdiode Model, 22 September 2021.
- [30]. Majid Jalalifar; Gyung-Su Byun, A Wide Range CMOS Temperature Sensor With Process Variation Compensation for On-Chip Monitoring, 13 May 2016.
- [31]. N. A. Quadir, Amit Jain, S. Kashfi, Lutfi Albasha, Nasser Qaddoumi, The Design and Process Reliability Analysis of Millimeter Wave CMOS Power Amplifier with a Cold Mode MOSFET Linearization, 31 October 2023.
- [32]. Esteban Garzón; Yosi Greenblatt; Odem Harel; Marco Lanuzza; Adam Teman, Gain-Cell Embedded DRAM Under Cryogenic Operation—A First Study, 26 May 2021.
- [33]. Jain, Abhishek; Hashmi, Mohammad S. (Advisor), Power network analysis using frequency domain approach, 2015-12-07.
- [34]. Rafiya Hossain; Moonmoon Ahmed; Hasan U. Zaman; Mohd. Azfar Nazim, A comparative study of various simulation software for design and analysis of operational amplifier-based integrator circuits, 23 October 2017.
- [35]. Sherif H. Abdelhaleem; Prasad S. Gudem; Lawrence E. Larson, An RF-DC Converter with Wide-Dynamic-Range Input Matching for Power Recovery Applications, 24 April 2013.
- [36]. Yousuke Miyake; Yasuo Sato; Seiji Kajihara; Yukiya Miura, Temperature and Voltage Measurement for Field Test Using an Aging-Tolerant Monitor, 24 March 2016.
- [37]. Fabian Khateb; Tomasz Kulej, Design and Implementation of a 0.3-V Differential Difference Amplifier, 06 September 2018.
- [38]. Mohammad Shokouhifar & Ali Jalali, Evolutionary based simplified symbolic PSRR analysis of analog integrated circuits, 23 December 2015.
- [39]. Torres, Joselyn, Low Power DC-DC Converters and a Low Quiescent Power High PSRR Class-D Audio Amplifier, 2013-12-03.
- [40]. Farzad Khoeini; Bahareh Hadidian; Keshu Zhang; Ehsan Afshari, A Transimpedance-to-Noise Optimized Analog Front-End With High PSRR for Pulsed ToF Lidar Receivers, 18 June 2021.
- [41]. Sougata Ghosh & Vijaya Bhaduria, An ultra-low-power near rail-to-rail pseudo-differential subthreshold gate-driven OTA with improved small and large signal performances, 21 June 2021.



13th International Conference on

Electrical, Electronic
Engineering and Smart Grids

Event Place: Tbilisi, Georgia

www.Eesconf.ir

سیزدهمین کنفرانس بین المللی



مهندسی برق، الکترونیک و شبکه های هوشمند | گرجستان

13th International Conference on Electrical, Electronic Engineering and Smart Grids
مجلات معتبر بین المللی ————— شهریور ماه ۱۴۰۲