David Barros Leonardo

Licenciatura em Ciências da Engenharia Electrotécnica e de Computadores

Design of High-Bandwidth and High-Linearity Input Buffers for ADCs

Dissertation submitted in partial fulfillment
of the requirements for the degree of

Master of Science in
Engenharia Electrotécnica e de Computadores

Adviser: João Carlos da Palma Goes,
Prof. Doutor, NOVA University of Lisbon

Examination Committee

Chairperson: André Mora
Raporteur: Luís Bica Oliveira

FACULDADE DE
CIÊNCIAS E TECNOLOGIA
UNIVERSIDADE NOVA DE LISBOA

November, 2020

Design of High-Bandwidth and High-Linearity Input Buffers for ADCs

This document was created using the (pdf)LaTeX processor, based in the "novathesis" template[1], developed at the Dep. Informática of FCT-NOVA [2].
[1] https://github.com/joaomlourenco/novathesis [2] http://www.di.fct.unl.pt

Acknowledgements

I would like to thank all the professors who were part of my academic path and who were kind enough to share some of their knowledge with me. I would like to give a special thanks to Professor Luís Oliveira for all the help given at the beginning of my master's degree and a huge thanks to Professor João Goes for having accepted to advise and to have advised me in this work.

I am thanking the people at Xilinx Ireland for the opportunity of an internship while working at my thesis. Special thanks to Vincent Callaghan, Bruno Vaz, Roswald Francis and Sohaib Afridi for their patience and will to teach me all of the important things for for my work there.

I will not forget my peers, friends and colleagues and their dedication, I would like to thank everyone who made me the person I am today. I would like to give special thanks to Francisco Matos, Diogo Pereira, Lucas Fioravanço and Clarisse Feio for all the support they gave me during my years in college and a special word for Miguel Calado for keeping my spirits high in the last stage of the thesis.

Finally, I would like to thank my parents, for all the work, effort and dedication that they have shown in all these years, and my little sister, for always being concerned with things that would never bother me.

Abstract

Nowadays on-chip Input Buffers (IBs) for direct conversion front-ends are realized with a higher voltage supply than that of the core voltage of the technology, mainly for linearity purposes. This, in turn, makes it mandatory to have more than one voltage source to supply a single chip in addition to having devices capable of handling higher voltages.

This work explores the possibility of having IBs supplied with the technology's core voltage to standardize all of the devices and reducing the different voltage supply sources and/or voltage regulators needed for operating the front-end drivers of the Analog to Digital Converters (ADCs).

A new input buffer architecture will be presented and compared to some prior input buffer implementations in the same conditions. This new architecture presents good linearity and bandwidth results and can be used for input buffers with the added benefit of not needing higher voltages nor special devices.

This new architecture is based off an existing one with another feedback loop to improved high-frequency peaking and linearity issues. This architecture achieves better results in bandwidth, a SNDR of 58 dB with and output voltage of 600 mV peak-to-peak differential. Furthermore, this buffer achieves a better efficiency linearity-wise when comparing to other buffers in the same conditions.

Keywords: ADC, Input Buffer, CMOS, High Linearity, High Bandwidth, Direct Conversion, Harmonic Distortion, Intermodulation Distortion.

Resumo

Nos dias que correm, Input Buffers (IBs) para interfaces de alta frequência dos circuitos de conversão directa em receptores de áudio são alimentados com tensões superiores à tensão de alimentação nominal da tecnologia, principalmente por questões de linearidade. Isto obriga ao uso de várias fontes de tensão para alimentar um único *chip* bem como a utilização de transístores capazes de suportar maiores níveis de tensão.

Neste trabalho é explorada a possibilidade de usar IBs com tensão de alimentação nominal para uniformizar todos os transístores utilizados dentro de um projecto e diminuir a necessidade de mais do que um nível de tensão para a operação dos *drivers* dos Analog to Digital Converters (ADCs).

Será apresentada uma nova arquitectura e esta será comparada, nas mesmas condições, com várias implementações de *input buffer* existentes na literatura. Esta nova arquitectura apresenta bons resultados em termos de linearidade e largura de banda, podendo ser utilizada para projectos de alta frequência sem a necessidade de diferentes tipos de dispositivos ou diferentes níveis de tensão.

Esta nova arquitectura é baseada noutra acrescentando uma malha de realimentação para melhorar a linearidade e peakings na largura de banda a altas frequências. Esta arquitectura tem melhores resultados em termos de largura de banda, um SNDR de 58 dB com um sinal de saída com 600 mV pico-a-pico diferencial. Para além disto, este buffer é mais eficiente em termos de linearidade quando comparado com outros buffers nas mesmas condições.

Palavras-chave: ADC, *Input Buffer*, CMOS, Linearidade, Largura de Banda, Converão Directa, Distorção Harmónica, Distorção de Intemodulação.

Contents

List of Figures

List of Tables

Acronyms

ADC	Analog to Digital Converter.
BB	Baseband.
BiCMOS	Bipolar Complementary Metal Oxide Semiconductor.
BJT	Bipolar Junction Transistor.
BW	Bandwidth.
CMOS	Complementary Metal Oxide Semiconductor.
CSF	Cascaded Source Follower.
Current Feedback IB	Current Feedback Input Buffer.
ENOB	Effective Number of Bits.
FFT	Fast Fourier Transform.
FOM	Figure of Merit.
FVF	Flipped Voltage Follower.
GBW	Product Gain Bandwidth.
HD2	2^{nd} Harmonic Distortion.
HD3	3^{rd} Harmonic Distortion.
HD5	5^{th} Harmonic Distortion.
I/Q	Inphase/Quadrature.
IB	Input Buffer.
IF	Intermediate Frequency.
IM3	3^{rd} order Inter-Modulation.
KCL	Kirchhoff's Current Law.
LNA	Low Noise Amplifier.

MOS	Metal Oxide Semiconductor.
MSAAC	Matlab tool for Symbolic Analysis of Analog Circuits.
NMOS	N-channel Metal Oxide Semiconductor.
OpAmp	Operational Amplifier.
OTA	Operational Transconductance Amplifier.
PMOS	P-channel Metal Oxide Semiconductor.
RF	Radio Frequency.
SC	Sampling-Capacitor.
SF	Source Follower.
SFDR	Spurious-Free Dynamic Range.
SNDR	Signal to Noise and Distortion Ratio.
SNR	Signal to Noise Ratio.
SSF	Super Source Follower.
TF	Transfer Function.
TH	Track-and-Hold.
THD	Total Harmonic Distortion.
VCMI	Input Common-Mode Voltage.

Chapter 1

Introduction

This Chapter provides the motivation and purpose for this project. It points out the main features of Input Buffers (IBs) and the challenges of its design. Finally, it presents the organization of this thesis and its main contributions.

1.1 Motivation and Background

Nowadays, most signal processing operations are done in the digital domain, implying the necessity of having Analog to Digital Converters (ADCs) serving as an A/D interface between the physical world and the digital world.

For this reason, ADCs have increased the sampling rate, improving the maximum allowed Bandwidth (BW) of the input signal, converting signals directly from Radio Frequency (RF). This is a practice to either reduce or eliminate the need for analog mixers and complex filters. We make these operations in the digital domain with the help of software.

Presently, the ADC is closer to the antenna on receiver systems. As a result, this kind of RF ADCs must be able to convert signals with frequencies in the GHz range. Besides the intrinsic speed necessary of the ADC, these conversions come with another level of complexity since the Low Noise Amplifier (LNA) might not be able to drive the ADC without rising significantly the power dissipation of the receiving system.

The driving of the RF ADCs is done by analog buffers, as shown in figure 1.1. This block drives the ADC without changing the input signal, serving as an interface between the LNA and the ADC, making those blocks independent of each other. A good buffer should have the following specifications:

- high linearity, so that it does not alter input signal properties;
- high bandwidth, because the input signal should be at RF range;

- low output resistance, so that it can drive the ADC;
- high input impedance, to limit the influence in the input signal;
- low power dissipation targeting handset applications;
- low footprint (area) for reducing cost.

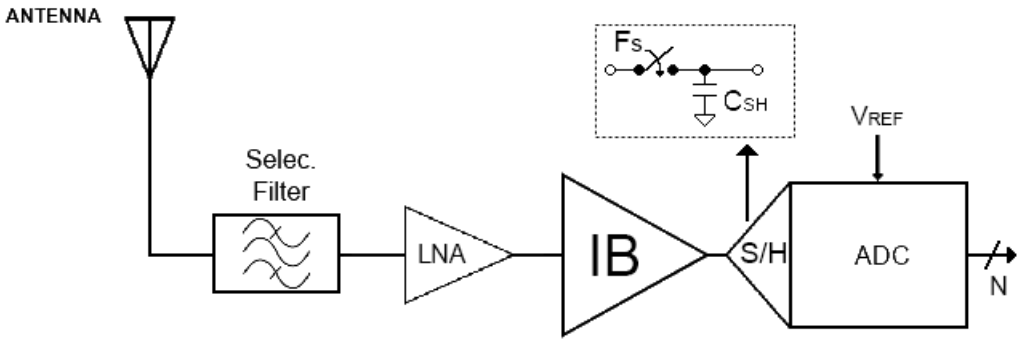
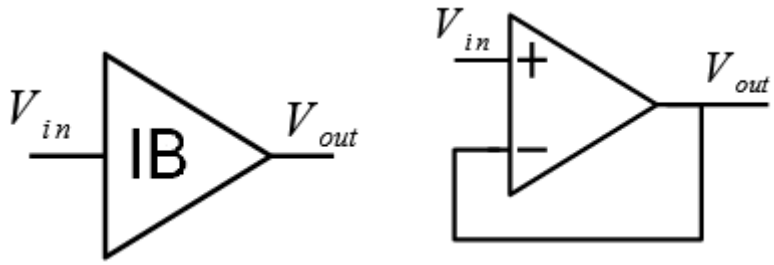


Figure 1.1: High level design of a direct-conversion Receiver System (Rx) comprising an IB and an ADC.

Figure 1.1 represents the direct-conversion front-end architecture. The direct-conversion is becoming more relevant for its advantages [11]. These advantages include flexibility, power dissipation, reduced system complexity and reduced weight [7, 10, 11]. The down-conversion process is done in the digital domain [10] and the Inphase/Quadrature (I/Q) demodulator can be replace by just one ADC while the demodulation is done in the digital domain [11].

The LNA on the figure is mandatory for impedance matching of the antenna. As stated earlier, the input buffer input impedance is desired to be with an high value. This block connects to the antenna (or the selection filter in the figure) and determines the performance of the receiver [2]. The LNA is supposed to amplify the input signal with the minimum noise figure possible [1] while achieving an input impedance matching in all of the signal bandwidth of 50 Ω [1, 2] to minimize any influence on the receiving signal [3]. Usually this amplifier is done with a common gate or common source configuration [23].

An ideal voltage buffer (figure 1.2 a) serves as an interface between two distinct circuits to eliminate any influence that they can have on one another. Meaning that the input impedance should be high, and the output impedance should be zero so that the buffer would be capable of driving whichever load [19], while achieving good linearity [13]. An Operational Amplifier (OpAmp) in unity gain configuration (figure 1.2 b) can implement an ideal voltage buffer.



(a) Ideal Input Buffer Representation.

(b) OpAmp voltage Input Buffer.

Figure 1.2: High level representation (a) and implementation (b) of a buffer.

Traditionally the most common voltage buffer is the well-known source-follower (Complementary Metal Oxide Semiconductor (CMOS) transistor in common-drain configuration) shown in figure 1.3, but with some studies, there have been new designs that implement different buffers with different strengths, that are efficient for some applications.

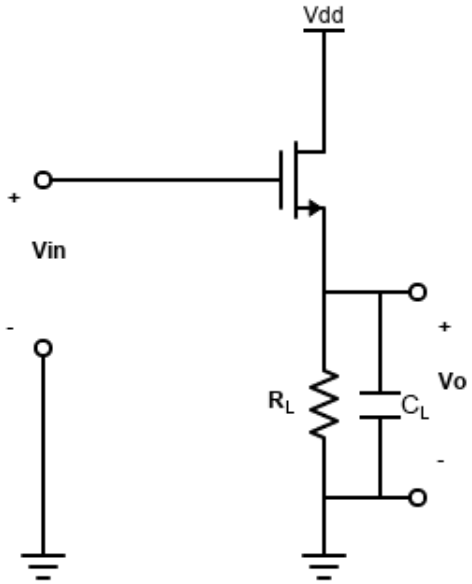


Figure 1.3: Basic common-drain topology [12].

Perfect analog unity buffers can be described as circuits characterized by a Transfer Function (TF) = 1, meaning that the output is a replica of the input. Typically the gain, defined by the ratio between the output signal and the input signal, is a ratio between 2 expressions. In voltage buffers, usually, there are some similarities between the numerator and the denominator. If both terms are equal then we have a perfect unity voltage buffer.

Let's imagine that a buffer's TF is determined by

$$\frac{g_m + s \cdot C_{gs}}{g_m + s \cdot \left(C_{gs} + C_{gd}\right)} \tag{1.1}$$

where we see similarities between the numerator and denominator. If the equation (1.1) is rearranged to

$$\frac{g_m + s \cdot C_{gs}}{\left[g_m + s \cdot C_{gs}\right] + s \cdot C_{gd}} \tag{1.2}$$

then the similarities are much easier to perceive. If in equation (1.2) we substitute the numerator by B and the remaining of the denominator by C we can simplify the expression as

$$\frac{B}{B + C}. \tag{1.3}$$

emphasizing the problem of analog buffers. Term C is the difference between a real buffer and an ideal perfect buffer. In a perfect buffer $C = 0$, which makes equation (1.3) $= 1$, for this reason $\frac{C}{B}$ is sometimes called follow-up error (FE) meaning the error that takes the buffer away from the ideal unitary buffer. However, the description of C is valuable in a real voltage buffer to check how it behaves and how it differs from a perfect buffer with different frequency ranges.

Since the perfect unity voltage buffer is not achievable, the perfect topology for this building-block does not exist. Usually, each case uses one architecture that satisfies the need for each implementation. The main differences rely on optimizing the buffer in some specifications and compare this new design with others in terms of simplicity, linearity, and power consumption. There are, however, some topologies that, instead of trying to achieve a unity gain, ultimately try to achieve higher gains to limit the need for extra gain blocks in the signal path.

1.2 Objectives and Original Contribution

The first objective of this thesis is to study, simulate, and compare different input voltage buffer designs already described in the literature. It will examine the ability of these buffers to work at high speed with a load of 1 pF with a voltage supply of 1.2 V, for 130 nm technology.

The second objective is to propose a new buffer design and do the same tests and simulations to compare its performance with the state-of-the-art input buffers.

This thesis presents a new functional buffer design described in chapter 5. This architecture will be explained, with all the considerations taken into account. The proposed new buffer achieves superior performance when compared with the state-of-the-art. Regarding this new architecture, a paper was submitted to ISCAS 2021 with the authorship of David Leonardo, João Goes and João Melo.

1.3 Thesis Organization

This work is organized as follows.

In chapter 2 is described as a general idea about voltage buffer designs. It will show some architectures already published in the literature, deriving their transfer function and some thoughts about each buffer.

In chapter 3 there will be presented the systematic buffer simulation analysis. It will show the simulation guidelines that every simulation followed, both in technology, capacitive load, and sizing of the devices. Furthermore, it explains all the simulations done and what can be expected to extract from them.

In chapter 4 the simulation results of some of the voltage input buffers, presented in chapter 2, will be shown. Here, it will be possible to see how each design in previous works compares to each other when carried out the same analysis.

In chapter 5 a new IB architecture will be presented and studied with the same type of simulations that every other buffer. The same chapter provides an in-depth analysis of the new architecture and an improvement to the design.

In chapter 6 will be drawn some conclusions about the work. This final chapter will reinforce the comparisons between all the buffers, concluding the best performances between all the designs.

CHAPTER 2

REVIEW OF STATE-OF-THE-ART

Analog voltage buffers were used for quite some time as output stages of gain blocks, like Operational Amplifiers (OpAmps) [8, 17, 27], since the gain block first stages' output impedance was high and was hard to drive whatever circuitry implemented after the block. High output impedance would degrade the performance of the circuit comprised of this block. By implementing a last-stage common-drain class A, B, or AB output buffer, the output impedance lowers and the block can drive circuits with more ease. Also, this output stage isolates the gain stages from the load effects.

Nowadays, analog buffers are used for different applications. With the need to down-convert signals from RF into Baseband (BB) the buffers are used as an isolator from the effects that come from analog designs like mixers (that could kickback large signals that could influence the LNA). Eventually, the analog mixer can be removed from the receiving chain and the ADC can move closer to the antenna. Now, the problem is that the front-end Track-and-Hold (TH) circuit that samples the input signal to be quantized by the ADC can be a source of interference and have quite low switched impedance. Therefore, an input buffer is used as an interface between the input signal and the ADC.

The input buffer needs to isolate the input from the Sampling-Capacitor (SC) sampling stage of the ADC while maintaining to feed the input signal to the ADC with low distortion. However, this can be difficult to implement as most of the buffers comprise non-linear devices. The non-linearities of the input buffer severely impact the performance of the ADC.

Bearing this in mind, the design of Input Buffer (IB) must follow the specification of the ADC to minimize the effect of the buffer on the overall ADC dynamic performance. An input buffer must not just have a high input bandwidth with unity-gain, but also have high enough Signal to Noise Ratio (SNR), Signal to Noise and Distortion Ratio (SNDR), Spurious-Free Dynamic Range (SFDR), as well as enough Effective Number of Bits (ENOB)

to allow driving a moderate to high effective resolution ADC.

This chapter describes several buffer projects in the literature as well as an analysis of the transfer function of most of these projects. Since some designs are difficult to analyze by hand, it was used the Matlab tool for Symbolic Analysis of Analog Circuits (MSAAC) toolbox. MSAAC mainly uses a netlist to derive the transfer function of the circuit - within a given error.

It is taken into consideration that some of the designs can be used for a multitude of tasks. As such, they will be shown in the first section as general-purpose designs. Then it is shown some of the buffers already designed to implement IB for properly driving ADC.

2.1 General Purpose Buffers

Generally, buffers can be used either to interface between circuits or to limit the influence of a part of the circuit in whatever comes next. For this reason, traditionally, voltage buffers have been used as the last stage of amplifiers, so that the output resistance of the amplifier circuit is low and to achieve higher speed. This section shows some possibility of voltage buffers that can be readily used as input buffers for driving ADCs but due to their simplicity, they are versatile for other applications.

2.1.1 Common Drain Input Buffer

The classic Source Follower (SF) topology, figure 2.1, can be used as one of the main building-blocks of current mirrors, differential Operational Transconductance Amplifiers (OTAs), and class AB output-stage [8], as well as, interfacing with a front-end for ADC.

Due to its simplicity, the common-drain configuration is the first design option for voltage buffers. The output voltage extracted on the source of a CMOS transistor is almost a perfect copy of its gate voltage, aside for the V_{gs} [12]. This fact makes this circuit a simple yet effective voltage buffer when nominal power-supply is not a big concern (e.g. $V_{DD} \geqslant 1.8$ V).

A CMOS implementation ensures a minimum input current [6] since the input resistance is defined by the input impedance of the gate of a transistor.

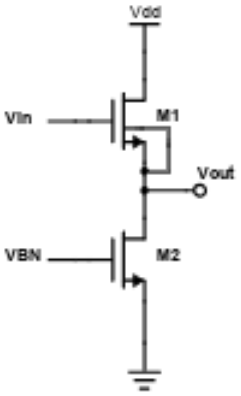


Figure 2.1: Source Follower Design.

Ignoring the body effect and the parasitic capacitances (low-frequency model) and assuming M2 as an ideal current source, figure 2.2 represents the small-signal equivalent of figure 2.1. With this, it is possible to extract an approximated TF of the SF.

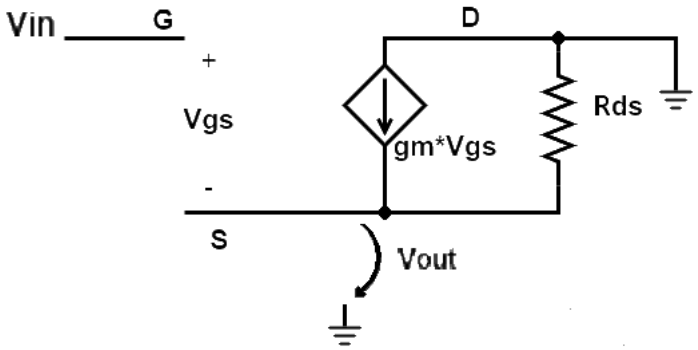


Figure 2.2: Small-Signal equivalent with Ideal Current Source (no parasitic effects, no body effect.)

By inspection on figure 2.2, it is possible to obtain an equation that relates v_{in} and v_{gs},

$$v_{gs1} = v_{in} - v_{out}, \tag{2.1}$$

as well as apply the Kirchhoff's Current Law (KCL) to the drain node to obtain another equation that relates v_{gs} and v_{out}

$$v_{out} \cdot R_{ds1} = g_{m1} * v_{gs1}. \tag{2.2}$$

By rearranging equation (2.2) and replacing R_{ds1} for $\frac{1}{g_{ds1}}$ it can be written

$$v_{out} = \frac{g_{m1}}{g_{ds1}} \cdot v_{gs1}, \tag{2.3}$$

if (2.1) is substituted in (2.3) then

$$v_{out} = \frac{g_{m1}}{g_{ds1}} \cdot (v_{in} - v_{out}), \tag{2.4}$$

where by extracting $\frac{v_{out}}{v_{in}}$ from (2.4) we get the familiar gain expression,

$$\frac{v_{out}}{v_{in}} = \frac{\frac{g_{m1}}{g_{ds1}}}{\frac{g_{m1}}{g_{ds1}} + 1}. \tag{2.5}$$

If the intrinsic gain $\frac{g_{m1}}{g_{ds1}}$ is far larger than 1, then (2.5) could be written as $\frac{v_{out}}{v_{in}} \approx 1$, which is the ideal buffer. Note, however, that even the SF doesn't have gain of 0 dB, but it can be approximately that.

The previous study was done without considering any parasitic effects to simplify the calculations. However, these should be taken into account since the purpose of this work is to design high bandwidth buffers. Therefore, it is important to assume the parasitic effects and find out their impact on the bandwidth. Taking into consideration some of these effects, the transfer function of the SF can be described as

$$\text{TF} = \frac{g_{m1} + s \cdot C_{gs1}}{g_{m1} + g_{ds1} + g_{ds2} + s \cdot \left(C_{gd2} + C_{gs1}\right)} \tag{2.6}$$

that even without much error (2.6) can still be approximated to a unity gain TF if $g_m >> g_{ds}$ and $C_{gs} >> C_{gd}$. However, as frequency increases, the effect of C_{gd2} becomes stronger and it limits the useful bandwidth of the buffer.

This design (Figure 2.1), however, has some limitations that were analyzed and optimized in different studies. Limitations in the output resistance have been studied in [9, 12, 18]. Linearity issues with this design were studied in [21, 25], as well as studies to try and overcome the offset at the output [9].

One of the major problems with this topology is the non-linearity that comes with a lack of isolation of just a single transistor [21] and the strong dependence of v_{gs} regarding signal variations.

On another note, even though this configuration theoretically has the lowest output resistance of all the three single transistor configurations, the output resistance (R_o) is roughly around $1/gm$, in [12, 22] the output resistance is calculated to be

$$R_{out} = \frac{1}{g_m + g_{mb}} \| R_L, \tag{2.7}$$

where R_{out} is the output resistance, gm is the intrinsic gain of the device, R_L is the load of the buffer and g_{mb} represents the body-effect transconductance. This value can be around some kΩ and thus may be higher than expected. This makes it necessary to increase the bias current and increase the aspect ratio $\frac{W}{L}$, increasing the power dissipation and the area of the buffer [16].

2.1.2 Cascaded Source Follower Input Buffer

One of the issues associated with the simple source follower is the DC level shifting. This can be more or less problematic depending on each specific problem and application.

In [26] is presented a design with offset cancellation on the output. This design uses two cascaded complementary source followers to reduce the offset at the output. As shown in figure 2.3 the Cascaded Source Follower (CSF) uses two level-shifters to cancel the offset effect that appears when using just one of them.

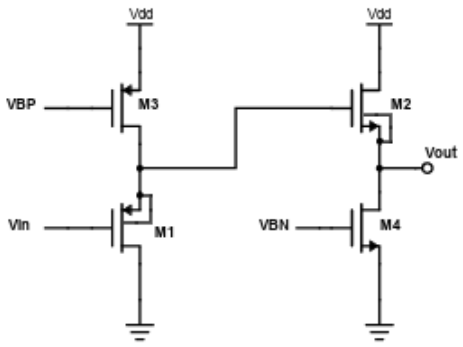


Figure 2.3: Cascaded Source Follower Design.

This can, ultimately, reduce the offset of $v_o - v_i$ to $v_{gs2} - v_{gs1}$ that can be theoretically zero but a simple change on the process or working temperature can alter the offset [26].

As far as the transfer function, assuming that M3 and M4 are ideal current sources, the TF can be approximated to

$$\mathrm{TF} = \frac{g_{m1} \cdot g_{m2} + s \cdot \left(C_{gs1} \cdot g_{m2} + C_{gs2} \cdot g_{m1}\right)}{g_{m1} \cdot g_{m2} + g_{m1} \cdot g_{ds2} + g_{m2} \cdot g_{ds1} + s \cdot \left(C_{gd2} \cdot g_{m2} + C_{gs1} \cdot g_{m2} + C_{gs2} \cdot g_{m1}\right)}. \tag{2.8}$$

Taking a look at equation (2.8) it is possible to describe it as a buffer by rearranging some of the terms

$$\mathrm{TF} = \frac{g_{m1} \cdot g_{m2} + s \cdot \left(C_{gs1} \cdot g_{m2} + C_{gs2} \cdot g_{m1}\right)}{\left[g_{m1} \cdot g_{m2} + s \cdot \left(C_{gs1} \cdot g_{m2} + C_{gs2} \cdot g_{m1}\right)\right] + g_{m1} \cdot g_{ds2} + g_{m2} \cdot g_{ds1} + s \cdot \left(C_{gd2} \cdot g_{m2}\right)}. \tag{2.9}$$

However, this TF was simplified by considering M3 and M4 as ideal current sources. If that was not the case, the corruption factor on (2.9) would be larger thanks to the parasitic capacitors on both of these transistors.

This architecture has some improvements in maintaining the Input Common-Mode Voltage (VCMI) at the output but it does still have the same problem with the output resistance of the basic Source Follower, meaning that power dissipation can be an issue.

2.1.3 Super Source Follower Input Buffer

Since some of the more modern designs operate with low positive power supply voltages the simple source follower might have a larger output resistance than what should be needed to drive some loads [12, 18]. So, in some cases, the Super Source Follower (SSF), as shown in Figure 2.4, is used as another variation of the traditional source follower.

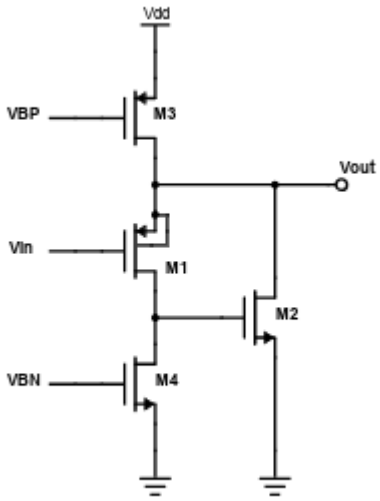


Figure 2.4: Super Source Follower.

This architecture can lower the output resistance of this buffer significantly thanks to the negative-feedback implemented through transistor M2 to approximately $\frac{g_{ds1}}{g_{m1} \cdot g_{m2}}$ [12, 18].

The transfer function of the SSF, considering M3 and M4 as ideal current sources, is given by

$$\mathrm{TF} = \frac{s^2 \cdot C_{gs1} \cdot \left(C_{gd1} + C_{gs2}\right) + s \cdot \left(g_{m1} \cdot \left(C_{gd1} + C_{gs2}\right) - C_{gd1} \cdot g_{m2}\right) + g_{m1} \cdot g_{m2}}{s^2 \cdot \left(C_{gs1} \cdot C_{gd1}\right) + s \cdot \left(g_{m1} \cdot \left(C_{gd1} + C_{gs2}\right) + g_{ds1} \cdot C_{gs1}\right) + g_{m1} \cdot g_{m2}} \tag{2.10}$$

this means that while the DC gain of the SSF is 1 V/V, dominated by the product of both g_{m1} and g_{m2}, the parasitic effect creates two poles and two zeros that can negatively impact the buffer, possible by creating a peaking at higher frequencies. Looking at equation (2.10) one might think that the second-order zeros might be stronger than the poles. However, the aforementioned equation considers ideal current sources that if were considered their impact would be felt more on the poles of the system.

Nevertheless, this buffer is a great improvement on the SF since it has a lower output resistance and the feedback loop on M2 corrects some of the non-linearity of the v_{gs} signal dependency.

2.1.4 Flipped Voltage Follower Input Buffer

Another variant of the simple source follower is the Flipped Voltage Follower (FVF). As shown in Figure 2.5, there is a second transistor (M2) that ensures, also relying on negative-feedback, that the current in M1 is held constant. This suppresses the variance on V_{gs} because of the output voltage, making the buffer even more linear [5, 20].

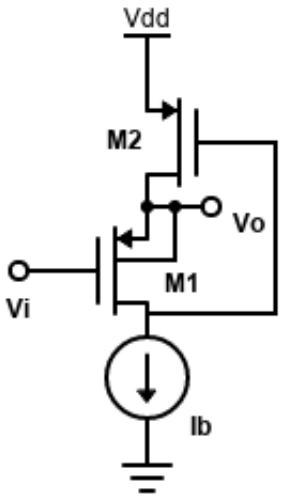


Figure 2.5: Flipped Voltage Follower [20].

This cell is used in some designs with low voltage sources (below 1V) [5]. This circuit TF can be approximated to

$$\text{TF} = \frac{g_{m1}}{g_{m1} + g_{ds1}} \tag{2.11}$$

given that the trans-conductance of transistor M2 is large enough compared to the parasitic capacitors and assuming, again, that the body-effect of device M1 is removed. It can be seen that equations (2.10) and (2.11) are approximately the same and if the trans-conductance is much larger than g_{ds} then a unity gain buffer can be achieved.

2.2 Input Buffers Design

In this section, some designs used particularly on input buffers for ADC will be shown. This doesn't mean that the architectures shown before can't be used as input buffers. However, in this section, there will be shown some of the particular designs are used for this kind of block.

2.2.1 Differential Source Follower Input Buffer

The Differential Source Follower was proposed as a way to limit mismatch error introduced by the use of two SF to create one differential input buffer at the input of the ADC.

This approach originally proposed in [16], instead of using two completely independent SF for each one of the differential inputs, introduces a differential loop between the two SF [16].

Using a fully-differential topology improves the performance of the design by ideally cancelling the even-order harmonics. Otherwise, merely duplicating most parts of the circuit may lead to area and power dissipation increase [4].

In figure 2.6 it is shown the design of the differential source follower input buffer.

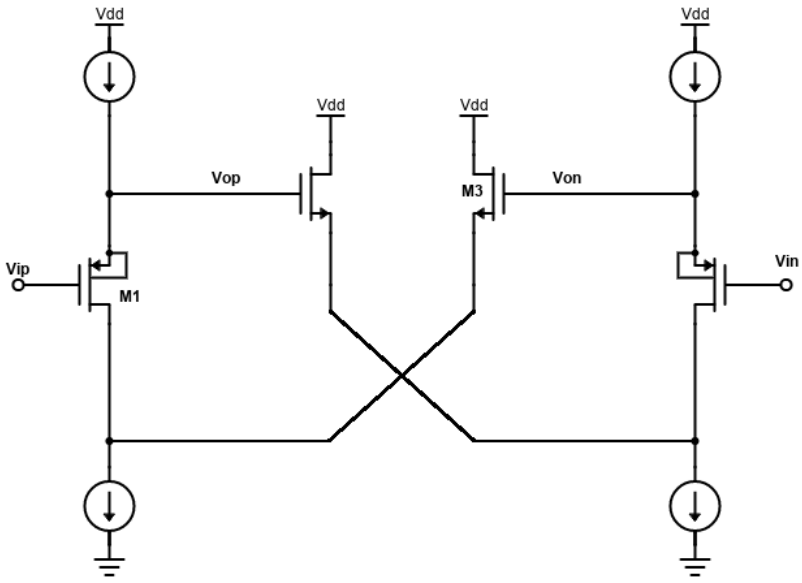


Figure 2.6: Differential Source Follower IB [16].

Using a cross-coupled pair at the output nodes with transistor M3 and its counterpart the even-harmonic distortion is further attenuated. However, this affects the performance of the circuit in terms of bandwidth and gain [16].

2.2.2 Differential Super Source Follower Input Buffer

The Super Source Follower was proposed to decrease output resistance and thus increase the bandwidth of the buffer, making it able to work at higher frequencies [16]. However, this approach was single-ended and, consequently, two independent circuits are used when buffering a fully-differential input. This can bring mismatch errors between the two circuits and, ultimately, it can degrade the linearity. In figure 2.7 it is represented by a differential super source follower design.

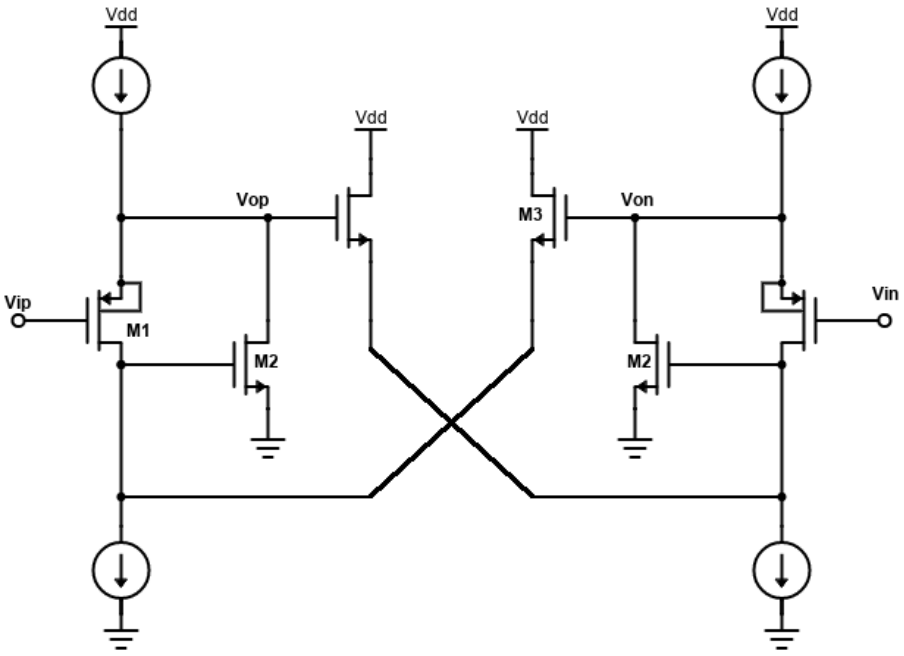


Figure 2.7: Differential Super Source Follower IB [16].

Similarly to the Differential Source Follower, the Differential Super Source Follower creates a cross-coupled loop between the two circuits, to make a fully differential circuit. In [16] it is reported that this technique improves buffer linearity and speed, while also having the added benefit of having active gain in the buffer.

2.2.3 Input Buffer with Current Feedback

The Current Feedback Input Buffer (Current Feedback IB) differs from the SSF since its feedback is done by current instead of voltage. This feedback maximizes the linearity of the simple Source follower architecture [24].

Figure 2.8 shows one design with current feedback. In it, the main device M1's drain voltage is used to regulate the V_{gs} of M2. This will regulate the current flowing on M2's branch that is mirrored by a wide-swing dynamic cascode current mirror composed by M3 and M4. This feedback is controlled by the current mirroring factor between devices M3A and M3B, and devices M4A and M4B.

Analyzing the feedback loop on figure 2.8 it is possible to conclude that there is some positive feedback. If the voltage V_{in} increases then increases the voltage V_{gs} on transistor M1 which makes the current flowing through the transistor larger. This in turn increases the voltage on the source V_{out} and lowers the one on the drain. By lowering the voltage at the drain of M1, the same voltage applied to the source of M2, this lowers the current flowing in M2 because the V_{gs} of this transistor decreases. By decreasing the current the voltage on the drain of M2 decreases which decreases the V_{gs} of M3A, making the current

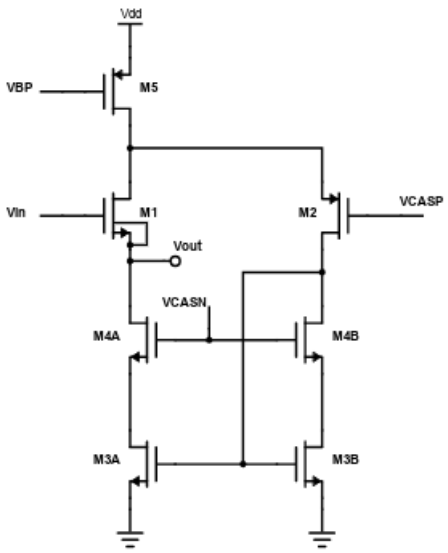


Figure 2.8: Current Feedback Input Buffer.

flowing on the cascode smaller and, consequently, increasing v_{out}.

Parasitic capacitance in the feedback loop needs to be minimized so that the AC current lost to charge these parasitic capacitors is kept at a minimum [24] and the feedback is fast enough to improve the signal bandwidth.

This input buffer transfer function is given by

$$\mathrm{TF} = \frac{2 \cdot g_{m2} \cdot (g_{m4} + 2 \cdot g_{ds4})}{2 \cdot g_{m2} \cdot (g_{m4} + 2 \cdot g_{ds4}) + s \cdot C_{gd5} \cdot g_{m4}} \tag{2.12}$$

this makes this circuit fairly dependent on frequency. Taking into consideration (2.12) this circuit acts as an ideal buffer until the effect of the parasitic capacitor C_{gd5} is felt.

2.2.4 Push-Pull Input Buffer

Another way of designing an input buffer is by using a push-pull configuration design [14].

As shown in figure 2.9 this buffer is an AC-coupled class-AB push-pull source follower. The decoupling of the input means that this block cannot be used as a low-frequency buffer. However, it does not need another low-pass filter for applications at higher frequencies, where a baseband might not be needed.

As stated before, this design has an AC coupling at the input to each gate of the devices. The bias point is achieved by generating each voltage from $VB1$ to $VB4$. The resistors need to be sized to not change the input value at each device. Since all the devices above V_{in} and V_{out} are N-channel Metal Oxide Semiconductor (NMOS) and all devices below are P-channel Metal Oxide Semiconductor (PMOS) then when the input rises the NMOS devices pull the output since the V_{gs} of said devices will increase, while the PMOS devices push it because their V_{gs} will decrease. If the input lowers then the

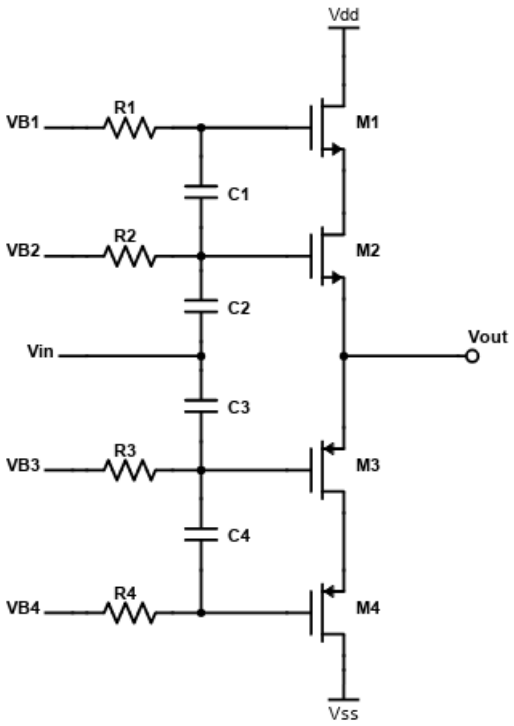


Figure 2.9: Push-Pull Input Buffer [14].

reverse happens. Because of this behavior, this configuration is called a "push-pull"input buffer.

Another drawback of this circuit is due to the requirement of a positive (high) supply voltage (V_{DD}) together with a negative V_{SS}. In [14], V_{DD} = 1.35 V and V_{SS} = -0.45 V have been used.

Push-pull source followers are commonly used as the last stages of amplifiers for their power dissipation optimization and providing the required low output resistance.

2.2.5 Vgs-controlled Cascaded Source Follower Input Buffer

The CSF tries to guarantee that the output common mode be equal to the input common mode. As stated before there are some mismatch problems with just using two cascaded source followers. There are a number of techniques that can improve the performance of th CSF [26]. One of these uses a negative feedback loop to control the V_{gs} of the cascaded common-drain device. Figure 2.10 shows the technique design.

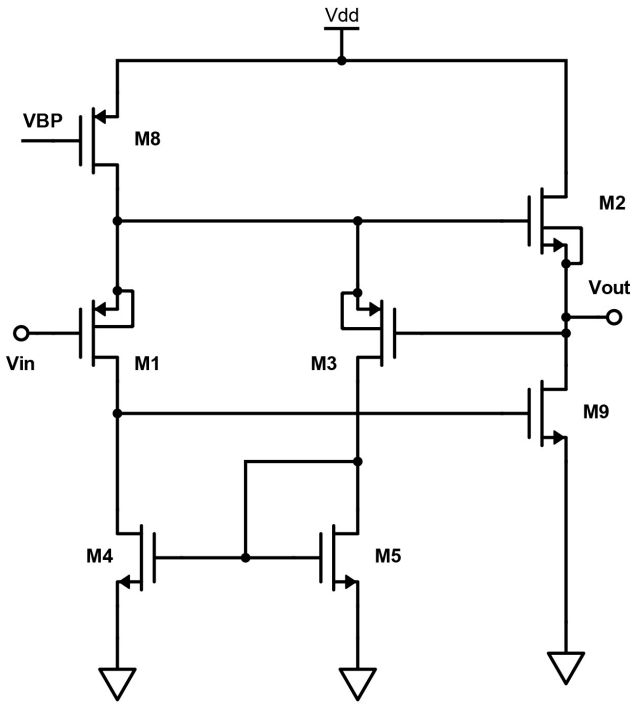


Figure 2.10: Cascaded Source Follower with Vgs control [26].

The negative feedback reduces output impedance and keeps $V_{gs2} \approx V_{gs1}$, reducing both constant and input-dependent component of the output signal. M3's gate and source are connected to M1's source and gate, respectively, to sense any variation on V_{gs2}. M1 and M3 form a differential-pair with a current-mirror load comprised of M4 and M5. If there is any variance in V_{gs} of the common-drain devices, the gate voltage of M9 is adjusted to change the current trough M2 until $V_{gs1} \approx V_{gs2}$.

2.3 Other implementations

Some other types of implementation were not covered in this chapter but can be considered.

2.3.1 BiCMOS solution

This solution (Figure 2.11), described in [6], uses BiCMOS technology to improve linearity while attempting to maintain a high speed. This solution was not considered since it uses BiCMOS technology while this work only focuses on CMOS technology.

This buffer comprises a two-stage OpAmp in a unity gain configuration, using a global feedback solution [6]. The first stage is a PMOS differential pair with some Bipolar Junction Transistor (BJT) as an active load. In the last stage, there are two more BJT in a Darlington structure. To keep the output with the same value as the input, the first is

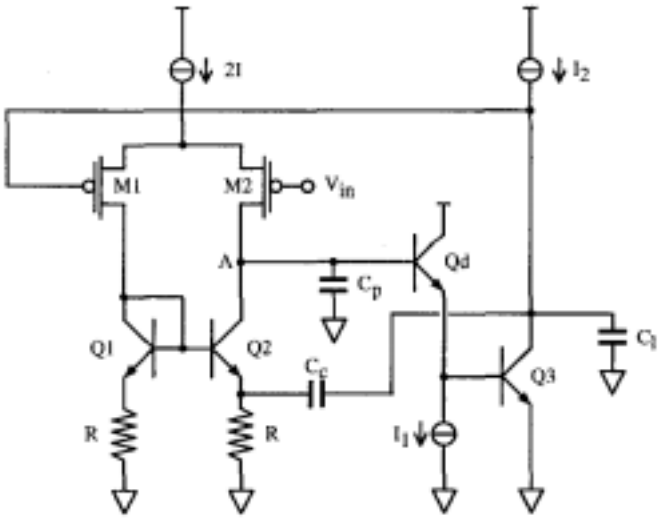


Figure 2.11: BiCMOS PMOS input buffer [6].

feedback to the M1 device that is the counterpart of the differential pair of the input. If the input changes, the current from 2I becomes unbalanced in the differential pair and the circuit will converge to the point where the output equals the input.

2.4 Final thoughts

Every implementation of input buffers shown and described in this chapter was trying to answer some particular problems of previous designs. In the following chapters of this work, some of these architectures will be simulated on the same basis to achieve some understanding of the best architectures for this work.

In the next chapter, it will be shown an overview of the simulation guidelines that are followed in this work. As well as want will be the priorities to achieve and what are the constraints that were imposed. It will be also shown, an overview of the simulation setup used in every simulated buffer and how every device on the design was sized.

CHAPTER 3

STUDY AND SIMULATION OF THE VOLTAGE BUFFERS

3.1 The Process Guidelines and Simulating Conditions

In this chapter, it will be discussed the sizing and the simulation of some of the buffers shown in Chapter 2 (review of state-of-the-art) as well as comparisons between them. With that in mind, this work was done with some specifications (Table 3.1), so that the comparisons can be done with fairness. This means that these specifications have been used to systematize the process for all the designs to be simulated in a comparable condition.

Table 3.1: Specifications of the simulation.

Technology	1.2 V, 130 nm CMOS
Capacitive Load	1 pF
Input impedance	> 10 MΩ
Open Loop Gain	≈ 1 V/V
Devices type	No IO devices; Only standard core devices are used
Voltage Supply	1.2 V ± 10%

Besides the contents on Table 3.1, there was the need to size all the transistors similarly. Otherwise, some misconceptions could be made regarding the operation of the circuits. For this reason, the transistors were divided into three different groups so that their sizing could follow a systematic design procedure.

Current Biasing - The transistors that implemented the ideal current source to bias the buffer;

Main Buffer Transistors - The main transistors that implemented the common-drain buffering function;

Feedback - The implemented current mirror to create either negative or positive feedback.

All the transistors in every different design but within the same group were sized in the same way, with NMOS transistors having a different sizing than the PMOS. The voltage setup was made by a similar circuit in every design, with some room to optimize said voltages, mainly so that all the transistors were in the saturation regime (i.e. the active region).

3.2 Sizing

This section will describe the sizing of each group talked about in Section 3.1. For this, the simplest design, the Source Follower (Section 2.1.1), will be used as a reference circuit and the process to size will be shown.

Figure 3.1 shows the design of the source follower. In this design, M1 is the buffer transistor NMOS while M2 acts as a current source.

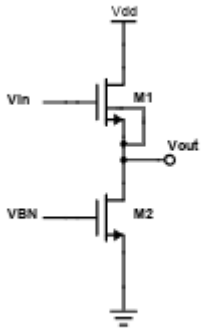


Figure 3.1: Source Follower.

This means that M1 will be sized as in the "Main Buffer"transistor group and M2 as in the "Current Biasing"group.

3.2.1 Sizing Main Buffer Transistors

Starting with M1, "Main Buffer"Transistors will be designed with speed as a major concern, this means that the channel length used will be set to the minimum possible. In the given technology it is 120 nm. The V_{dsat} will be chosen at around 100 mV ±25 mV so that the signal can have a dynamic range high enough for future designs. Lastly, the current used in buffer transistors is 1 mA as shown in Table 3.1.

With these values, knowing that the drain current, for the strong inversion region, of the transistor (I_D) can be approximated to

$$I_D = \frac{K}{2} \cdot \frac{W}{L} \cdot \left(V_{gs} - V_{th}\right)^2 \tag{3.1}$$

where V_{gs} is the voltage from the gate to the source of the transistor and V_{th} is the threshold voltage needed for the transistor to start conducting and assuming that the transistor is in strong inversion $V_{gs} - V_{th} \approx V_{dsat}$ then, if this is done to equation (3.1)

$$I_D = \frac{K}{2} \cdot \frac{W}{L} \cdot V_{dsat}^2 \tag{3.2}$$

where I_D is the biasing current of the transistor, K is a technological design parameter, W is the channel width and L is the channel length. Since the only value not know is the channel width it's possible to solve equation (3.2) regarding that parameter as

$$W = \frac{2 \cdot I_D \cdot L}{K \cdot V_{dsat}^2}. \tag{3.3}$$

The channel length (L) used for these transistors was the technology minimum channel length. Using the minimum value for L increases the speed of the transistor and, because of this, increases the bandwidth of the buffer. Since we want high speed input buffers it is recommended to use lower channel length. Solving equation (3.3) the designated values can be seen as

$$W = \frac{2 \cdot 1000 \cdot 120 \cdot 10^{-9}}{500 \cdot 0.1^2}(\text{m}) \tag{3.4}$$

which means that the channel width of NMOS buffer transistors should be $W = 48\ \mu$m. In Table 3.2 is shown the dimensions of NMOS and PMOS Buffer Transistors.

Table 3.2: Buffer Transistors Sizing.

Buffer Transistors			
NMOS		PMOS	
L	W	L	W
120 nm	48 μm	120 nm	160 μm

Some important actions were taken regarding the body-effect of these devices. Bearing in mind that the body-effect trans-conductance limits the gain of the block. This attenuation is particularly important in buffer designs since the buffer does not achieve high gains the body-effect will most assuredly create an attenuation. This attenuation can be less noticeable if the body effect is taken out of the main buffer devices. This means that every bulk of any transistor in a common-drain configuration has been shorted to the source of the same device, so that the voltage between source and bulk is zero and there is no body-effect, this is valid for every advanced "triple-well"process.

3.2.2 Sizing Current Biasing Circuitry

Current Biasing transistors define the current that flows in each branch of the circuit. The way it was implemented in this work is simply from the use of multiplier transistors. This means that the voltage bias circuit was implemented with certain sized transistors and the current biasing transistors are equal in size and replicated the number of times necessary

to provide a multiplied current. If, for example, we had a voltage biasing transistor M_{BIAS} that creates a voltage V_{BIAS} with a current I_{BIAS}, if in the branch that its implemented a current source with $10 \times I_{BIAS}$, then that means that the implementation of that source would be ten parallel transistors equal to M_{BIAS}.

In Table 3.3 there are discriminated all of the base transistors and their driving current for each of the BIAS voltages.

Table 3.3: Current Sources Sizing.

PMOS				NMOS			
VBP		VCASP		VCASN		VBN	
L	*W*	*L*	*W*	*L*	*W*	*L*	*W*
360 nm	48 μm	360 nm	5.3 μm	480 nm	2.7 μm	360 nm	14.4 μm

3.2.3 Sizing Feedback

The feedback sizing works very similarly to Current Biasing Sizing, with the difference that it is not made for biasing, but to achieve the desired performance. The way these transistors are sized in this work is, again, with the multiplying factor. The idea is to have a certain variation to be felt strongly or weakly in different parts of the circuit.

Let's take, for example, the feedback current mirror at Current Feedback IB (Figure 3.2). In this buffer, the feedback is achieved by mirroring the current from M3B to M3A in the cascoded current mirror.

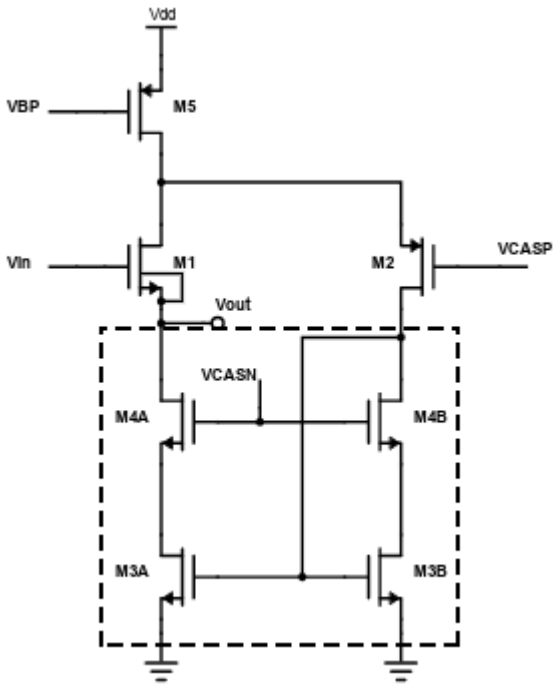


Figure 3.2: Current Feedback buffer (Current Mirror Feedback Highlighted).

If the feedback mirroring factor needs to be 1:4 [24], then this means that M3A needs to be made of 4 transistors the same size as M3B so that the size of the later is 4 times smaller than the first.

3.3 Simulation testbench

For all simulations, the same setup was used. This setup was a high-level design with all the simulated outside influence as shown in figure 3.3. It is shown that there is no input interference to the buffer, as well as no AC coupling. This was used so that the DC of the signal was the one injected at the input of the buffer and was easier to achieve the pretended value, given by the voltage source.

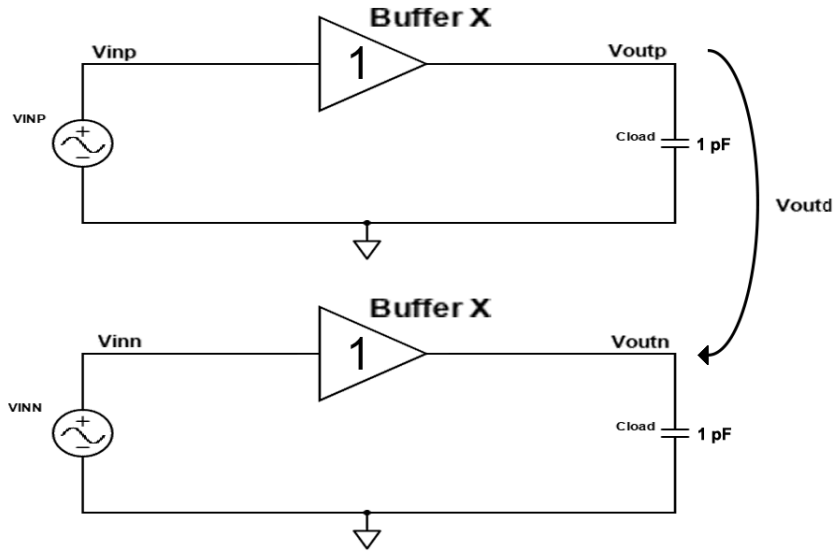


Figure 3.3: Simulation testbench for all the simulated buffers.

It is possible to check that the output is modeled as a single load capacitor with a normalized value of 1 pF. This model some of the effects of the circuit that follows the buffer. The value of the capacitor was the same between simulations. Because of this, bandwidth values were achieved in the same conditions between all simulations.

All of the buffers have been simulated in a differential simulation to increase the dynamic range of the output signal and some better noise and distortion values. The circuits are the same circuit replicated with differential input voltages. This simulation setup was used in all non-differential designs to achieve a pseudo-differential buffer.

Talking more about the DC operating point, the voltages were mainly achieved by a single circuit common to all simulated buffers. However, even though the schematic of the current mirrors was always the same, the sizing of the devices was not. This was necessary to achieve some variance between DC operating points between all of the designs. In figure 3.4 it is shown this same schematic and the only devices tuned in each

buffer simulation were M1 and M8, the ones achieving the cascode biasing voltages V_{CASP} and V_{CASN}, respectively.

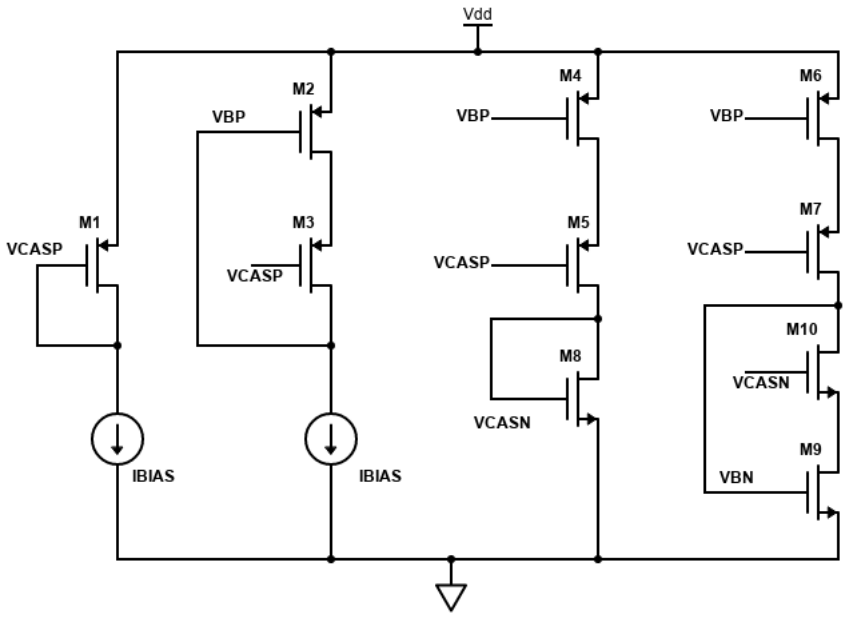


Figure 3.4: Current Mirror used to BIAS all the simulated circuits.

From the DC supply to the input supply, the input signal had three components that were simulated. The first one was the common-mode component of the signal. The common-mode was used to bias the circuit into the preferred operating point. This value was tuned with DC simulation until all the devices were in the desired region for their operation. The second input signal component simulated was the AC component. This one was used to simulate the buffer with varying frequency of the input signal to obtain the bode diagram and the bandwidth. Lastly, a sinusoidal wave was supplied to the input of the buffer to be used in a transient simulation. This wave's properties were adjusted to have an output signal wave with around 600 mV ($\approx V_{DD}/2$) amplitude and the frequency, although shared between all buffers, was chosen with the results of the AC simulations.

Even though all the simulation had the same simulation testbench, the input signal's components varied depending on the buffer. The only factor that was constant across every design was the frequency of the input signal. Both the common-mode and the input amplitude were changed depending on the desired performance of the buffer.

3.4 Simulations setup and objectives

In this section, some of the objectives behind every simulation will be presented. This section will not look at any result of any buffer, instead, it will indicate what were the ideas and thought process behind every simulation, as well as mentioning every few aspects worth noting in the next chapter.

3.4.1 DC Simulation

The main objective of the DC simulation was to verify and tune the operation point of every transistor. DC values of current and V_{gs} voltages were checked if they were within parameters or if there were any possible improvements.

The output common-mode was checked to see if the buffer would work with all the restrictions. This value would have to be enough to keep all of the transistors in saturation (i.e. active region) even with the variation of the signal.

Another important point was the output driving current, for comparison purposes. While the total DC current was necessary to compare the power dissipation between buffers, the output current should be around the same values regardless of the architecture so that the comparison between bandwidths of the designs is kept in the same fair conditions in every buffer.

3.4.2 AC Simulation

The AC simulation served two purposes. The first one being the bandwidth value of every simulated buffer and the second one being the choice of the input frequency for the transient analysis.

Regarding bandwidth, all of the buffers were simulated with a varying frequency from 1 Hz to 100 GHz and all bode's gain and phase diagrams were plotted. From these graphs, a few things were accounted for later comparisons. The DC gain, or lack thereof, to compare the need for higher amplitudes at the input. As a reminder, the transient-noise simulation was done to keep an output amplitude of 600 mV. Maintaining this value with a strong attenuation could be impractical at the input. Another important characteristic was the flatness of the band. Some designs have frequency peakings that can severely impact the expected performance of the buffers at higher frequencies.

Ultimately, the results of these simulations were used to get a value of frequency high enough to keep the circuits at some stress levels. The frequency chosen would need to be the same in all of the buffers. That meant that the stress level would not be equal in all designs. However, it was decided that keeping the same frequencies would suit better and fairer comparisons.

3.4.3 Transient-noise Simulation

The last analysis was transient-noise analysis. All the circuits were simulated with 100 cycles of the input with a constant frequency. These simulations were done following the results of the prior simulations.

The transient-noise simulation was used to simulate every circuit reaction to a "real"signal. This simulation was stressed to an input signal with a certain amplitude, to achieve the targeted output amplitude and frequency consistent between buffers.

This simulation has been used to calculate the Fast Fourier Transform (FFT) of the output signal. With the FFT calculated some values were extracted from it. Namely the SNDR and the Total Harmonic Distortion (THD) between others. Even though the distortion is usually compared regarding THD, the values of distortion of the first five harmonics were calculated independently, named 2^{nd} Harmonic Distortion (HD2) to 5^{th} Harmonic Distortion (HD5). As stated in the last section (AC Simulation), the amplitude of the output in these simulations was set to be about 600 mV ($\approx V_{DD}/2$). This was done so that the comparisons between buffers were, once again, done with similar conditions among buffers.

Another aspect of this analysis is that it was prepared to allow the double-tone simulation, by having two different input signals with similar frequency and the same amplitude. This simulation gives a metric of how the circuit reacts to a signal composed of two different frequencies close to each other. The maximum differential output swing was, again, set to 600 mV.

One important note to make when talking about FFT is to have some coherent window to minimize spectral leakage. In this work, the coherent window was achieved by making the input frequency a multiple of a frequency bin. This frequency bin is a function of the sampling frequency, in this work 2 GHz, and the number of points per period. These bins were also used for the inter-modulation analyzes, by having both frequencies one bin shift from the frequency chosen for the input signal on a regular transient simulation.

The next chapter describes the simulation study of some voltage-buffer architectures presented in chapter 2 - review of state-of-the-art - following the details of this chapter. In the end, the information is summarized in a table regarding the most important values of the simulations.

Chapter 4

Analysis and Results of the state-of-the-art

Continuing from the last chapter, this one presents the results of the electrical simulations. These reports discriminate each simulation in each design. Comparisons between the simulations' results are made. Finally, the designs will be categorized for their performance, advantages, and disadvantages.

Every simulated buffer was analyzed independently with the same procedure. Firstly, each architecture shows some results regarding the DC simulation. With this, we validate the DC bias operating point of each device and annotate currents values.

Secondly, we use the AC analysis to compare each bandwidth and acknowledge the input frequency for the transient-noise analysis, as stated in Chapter 3 - Study and simulation of the voltage buffers.

Finally, a more extensive transient-noise analysis is done. This simulation allows us to compare the different values of distortion and noise that affect each design.

4.1 Source Follower Input Buffer Analysis

This section will present the three analyses of the basic Source Follower (SF). Being the simplest circuit, there were no problems to size and simulate it.

4.1.1 DC Analysis (SF)

The sizing of the circuit was easy to achieve, as only two devices comprise this buffer, one of which is a current source to the buffer main-device. The main concern was to get the current to be as close as possible to the nominal 1 mA target. Figure 4.1 shows the current to be approximately the target current. The same figure shows room for the output to achieve 150 mV amplitude, pretended for the transient-noise analysis 600 mV peak-to-peak differential amplitude.

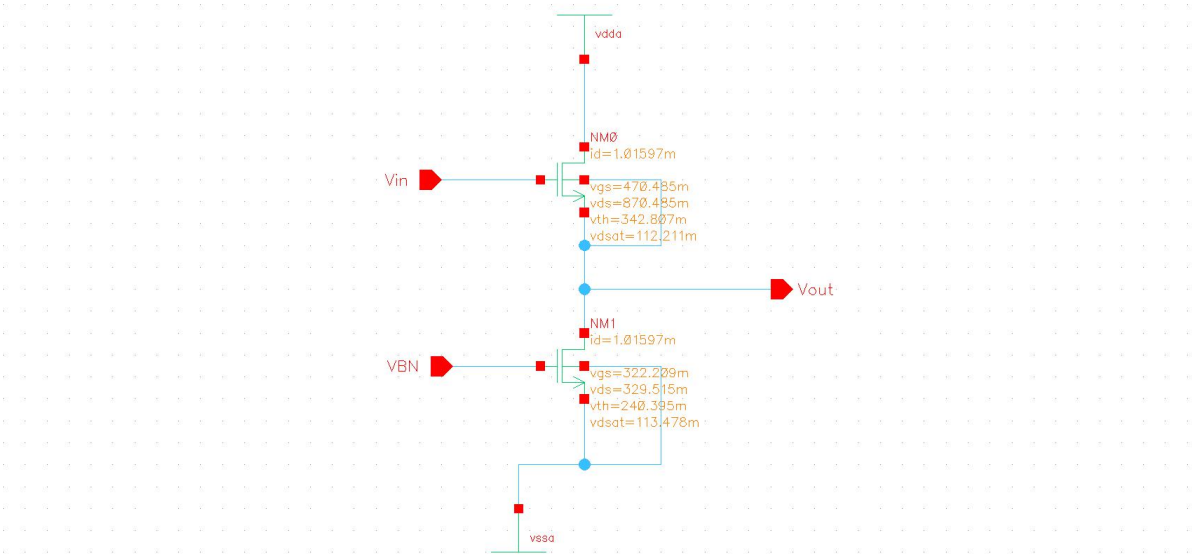


Figure 4.1: Source Follower DC Operating point.

Note that the bias voltage V_{BN} was achieved with a current mirror not shown in the schematic. The circuit used to achieve said voltage was shown in figure 3.4 on page 26. The sizing for this circuit was chosen to have the V_{BN} necessary for the desired operating point of the circuit.

4.1.2 AC Analysis (SF)

The AC analysis of the SF had not many problems. With only one pole at the output, the design is always stable. Figure 4.2 shows that the gain of the block is close to 1 V/V (0 dB). The system is dominated by one pole that influences the frequency response since the gain is constant until a frequency when it decays about 20 dB per decade.

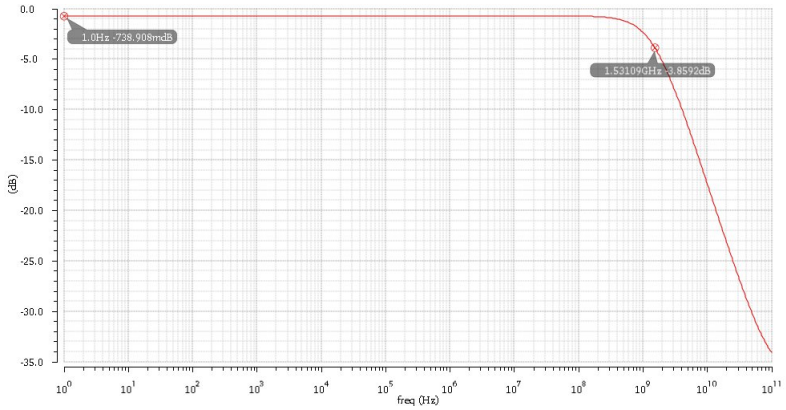


Figure 4.2: Source Follower Frequency Response.

Lastly, it is important to point out that the simulated bandwidth of the system is around 1.5 GHz. This is the point when the gain degrades 3 dB compared to the initial value.

4.1.3 Transient-noise Analysis (SF)

The transient-noise analysis for the SF was easy to tune for the target output amplitude voltage as shown in figure 4.3. This figure shows just the output signal and it looks like a perfect sinusoidal wave. However, to be able to measure the distortion value it is necessary to get the FFT of this signal.

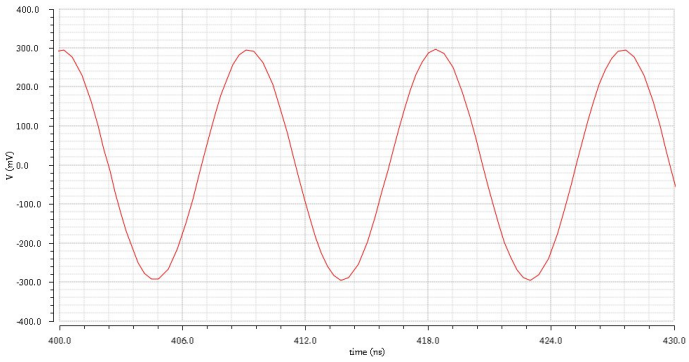


Figure 4.3: Source Follower Output (time).

The signal FFT is shown in figure 4.4. With this figure, we can see some trends. First, the DC power bin/spur (i.e. the offset bin) is minimal (around -100 dB) this happens because the output is differential, meaning that the common-mode at each branch cancels one another. Second, the strongest power spur comes from the frequency of the input signal meaning that the wave that we saw in figure 4.3 is close to the input frequency. There are another two points marked in figure 4.4, the second is with double the frequency of the input signal and the third with three times the frequency of the same signal. These points are harmonics of the signal, used to calculate HD2 and 3rd Harmonic Distortion (HD3) respectively, and the lower they are the lesser the harmonic distortion of the output signal.

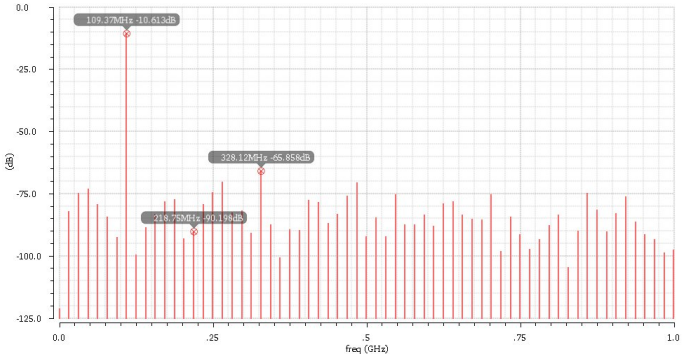


Figure 4.4: Source Follower Output (frequency).

Paying attention now to the inter-modulation analysis, figure 4.5 shows the mix of two different frequency signals.

The signal FFT is shown in figure 4.6. Here it is possible to see two different peaks; these are the fundamental harmonics of the two tones selected for this analysis. The two

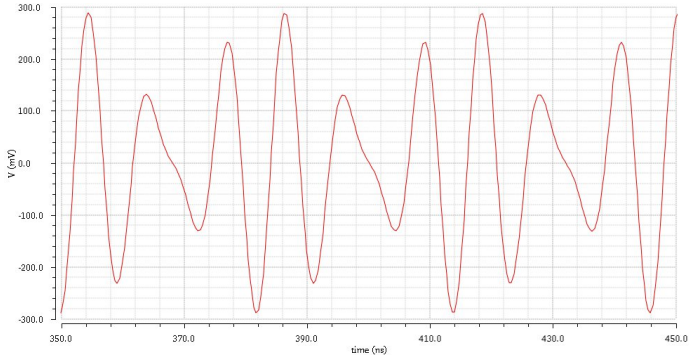


Figure 4.5: Source Follower 2 Tone Analysis Output (time).

other marked points are the points to calculate the 3rd order Inter-Modulation (IM3). This value is the difference between the lowest fundamental harmonic power (in this case 125 MHz) and the highest power of the inter-modulation (in this case 62.5 MHz).

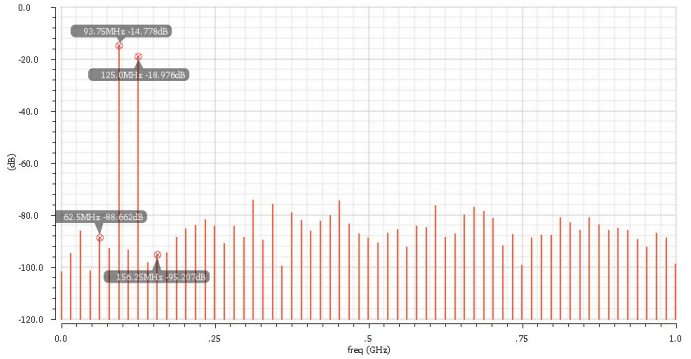


Figure 4.6: Source Follower 2 Tone Analysis Output (frequency).

All the results are compiled in table 4.1. These include results from all 4 different analyses.

Table 4.1: Summary from the Source Follower Simulation Analysis.

Current	2 mA
Bandwidth	1.5 GHz
DC Gain	-0.7 dB
Input Frequency	109 MHz
Offset Spur	-121 dB
HD2	-80 dB
HD3	-55 dB
IM3	70 dB
SFDR	55 dB
ENOB	8.0 bits

Current refers to the sum of all of the currents flowing through the differential circuit. In figure 4.1 the operation point shown is just one of the circuits meaning that the current

is doubled when a differential circuit is used. These values will be used as a reference for all future results.

4.2 Super Source Follower Input Buffer Analysis

This section will present the three analyses of the Super Source Follower. This circuit was meant to be an improvement over the last one. With the voltage feedback, the output resistance decreases and the bandwidth should increase.

4.2.1 DC Analysis (SSF)

The sizing of the circuit followed the following guidelines. The current through PM1, the main device, was set to be 1 mA. Meanwhile, the feedback current, through NM1, was set to be around $\frac{1}{4}$ of the value through the main device. The DC operation bias point of the circuit is visible in figure 4.7.

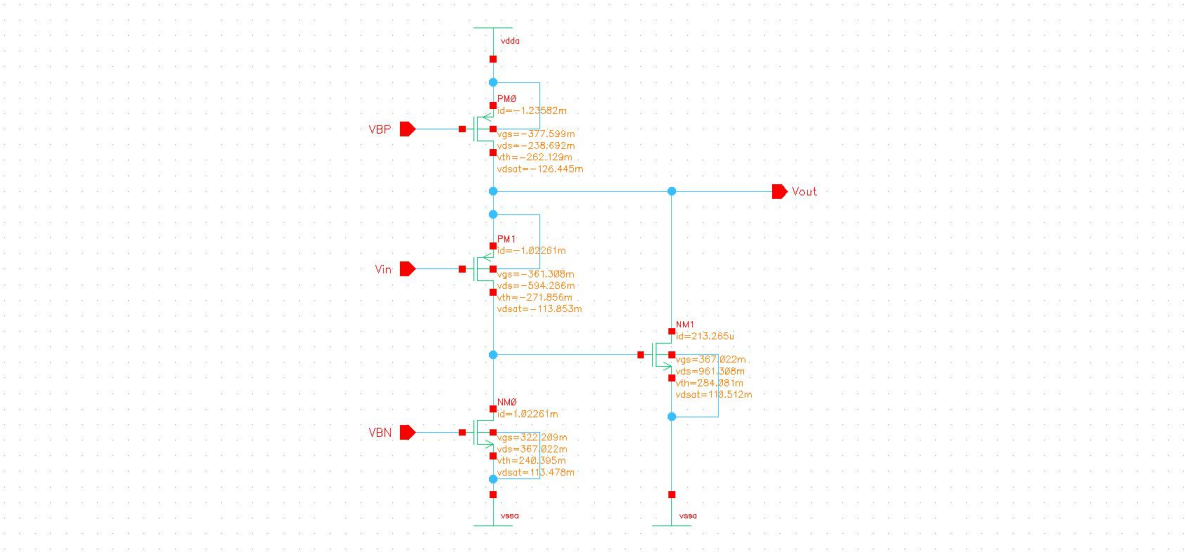


Figure 4.7: Super Source Follower DC Operating point.

4.2.2 AC Analysis (SSF)

The AC analysis of the SSF, figure 4.8, showed the first problems with complex buffer systems. With 2 poles and 1 zero, the frequency response of this buffer shows a peaking close to the cutoff frequency. This peaking can degrade some of the linearity performance of the buffer at a higher frequency.

It is also important to note that the registered bandwidth is 1.7 GHz. Even though increasing the current through the feedback device would improve the bandwidth, by doing it, the peaking shown would increase as well.

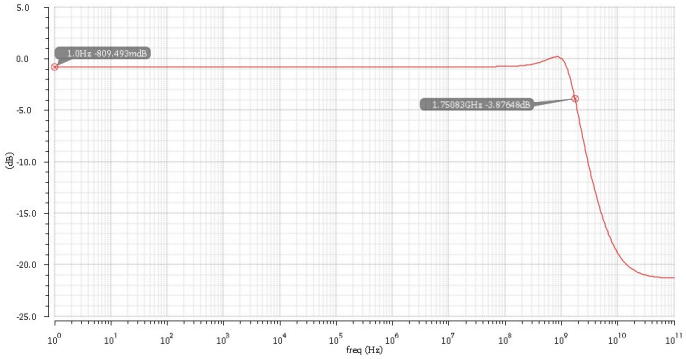


Figure 4.8: Super Source Follower Frequency Response.

4.2.3 Transient-noise Analysis (SSF)

In this simulation, the target output voltage of 600 mV was achieved without problems (Figure 4.9), and the results were as expected. Again, the sinusoidal wave is not able to give information about the linearity performance of the buffer, for that, it is necessary to transform the time signal to the frequency domain.

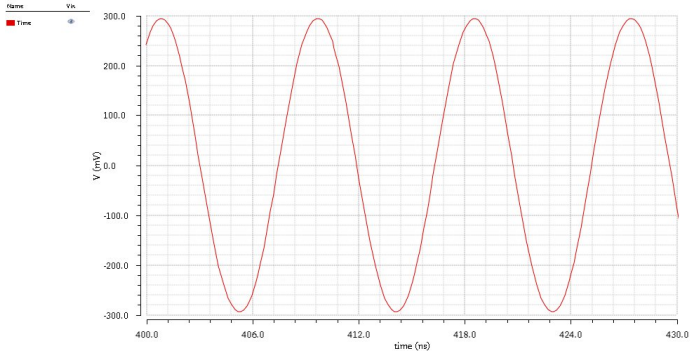


Figure 4.9: Super Source Follower Output (time).

The signal FFT is shown in figure 4.10. Like what was shown in the SF section, the DC value is negligible and the input frequency is the strongest signal shown. In this case, the HD2 is far more attenuated compared to other frequencies, while HD3 is still the strongest harmonic of the signal.

Regarding the two-tone analysis, the same pattern can be seen in time in figure 4.11. Again looking at the time wave gives little to no information.

Looking at the signal FFT, however, can show a lot more information. Again it is possible to see the two-tone frequency in addition to the power of the IM3.

Table 4.2 shows some of the numeric results of these simulations showing the performance of the simulated circuit.

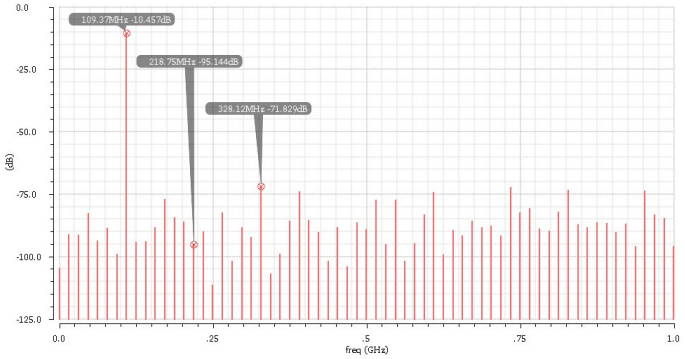


Figure 4.10: Super Source Follower Output (frequency).

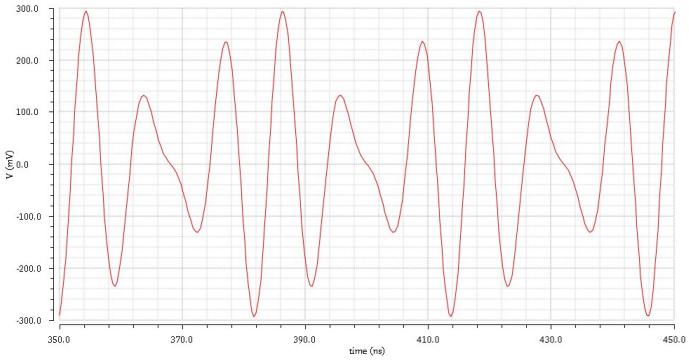


Figure 4.11: Super Source 2 Tone Analysis Follower Output (time).

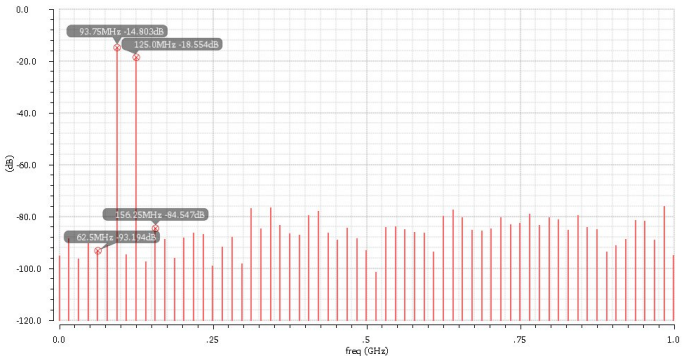


Figure 4.12: Super Source Follower 2 Tone Analysis Output (frequency).

Table 4.2: Summary from the Super Source Follower Simulation Analysis.

Current	2.6 mA
Bandwidth	1.7 GHz
DC Gain	-0.8 dB
Input Frequency	109 MHz
Offset Spur	-105 dB
HD2	-85 dB
HD3	-61 dB
IM3	66 dB
SFDR	61 dB
ENOB	8.5 bits

Comparing the results from table 4.2 and 4.1 on page 32, it is possible to understand that the SSF has some advantages compared to the SF. There is a 6 dB increase in harmonic distortion and the ENOB rises half a bit. However, the bandwidth decreases contrary to what was expected. As stated before the bandwidth of the SSF can increase but that will increase the frequency peaking, visible in figure 4.8. This decrease in bandwidth happened because the SSF was not sized to the best performance. Note that the current flowing through the main device need to be the same to derive some conclusions and that current can be, in some way, not ideal for this design, when compared to the SF.

4.3 Cascaded Source Follower Input Buffer Analysis

The Cascaded Source Follower is an improvement from a single-ended Source Follower. The main concept of this buffer is to eliminate the common-mode shift between output and input. Even though this problem is not as present in the differential signal domain, the same analysis was made to this buffer.

4.3.1 DC Analysis (CSF)

Since the CSF is in its essence two SF. The sizing of the circuit was made by having two 1 mA branches. However, this can be optimized since the first branch drives the second. Since the second branch load should be around the fF range, it is possible to reduce the first stage current to limit the overall circuit's power dissipation. However, the currents were sized to 1 mA in both stages, as shown in Figure 4.13.

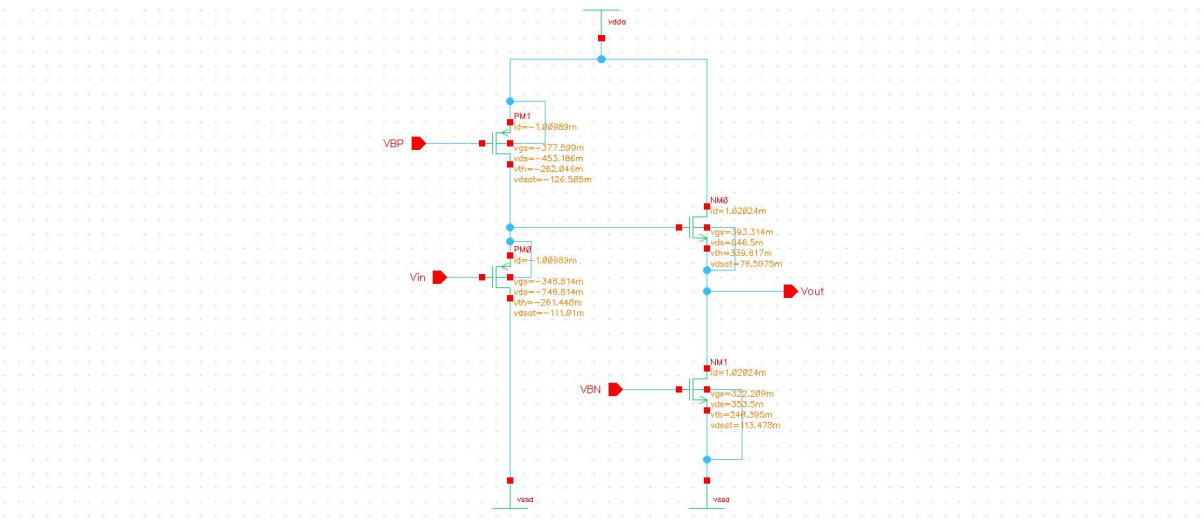


Figure 4.13: Cascaded Source Follower DC Operating point.

As stated before, this buffer meant to keep the common-mode from the input at the output. To achieve this both V_{gs} of the two main devices (PM0 and NM0) should be the

same. Since this circuit would be used in a differential topology this problem was not addressed on its sizing. However, with close inspection of the values of V_{gs} it is possible to see that, without the differential pair, the output common-mode would be shifted by 50mV compared to the input common-mode.

4.3.2 AC Analysis (CSF)

Since the output stage of this buffer is a regular Source Follower it was not expected to have any improvement in the bandwidth of this system and because of the absence of zeros from the system, it was not expected to have any peaking in the frequency. This assumption can be confirmed by figure 4.14 where the frequency response of the CSF is shown.

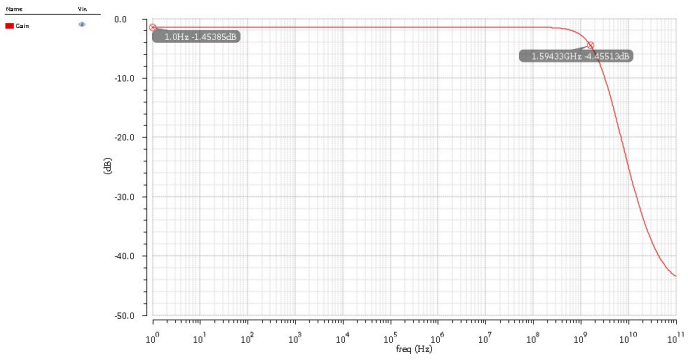


Figure 4.14: Cascaded Source Follower Frequency Response.

Lastly, it is important to point out that the bandwidth of the system is around 1.6 GHz.

4.3.3 Transient-noise Analysis (CSF)

The transient-noise analysis for the CSF was as simple to set up as the SF one. The cascaded nature does have a small impact on the attenuation of the signal however that was not a major problem for this circuit. Figure 4.15 shows the signal variance across time. The voltage achieved is around 600 mV peak to peak with a differential output.

The time-domain of the signal does not show the distortion caused by the circuit, for that it was used a FFT to transform the time domain to frequency domain shown in figure 4.16. Here it is possible to see the same peak at the input frequency and all of the output signal's harmonics. Note that, even though the CSF was supposed to decrease the common-mode of the system, the DC power in figure 4.16 is around 100 dB like every other circuit shown previously. This happens since the common-mode cancellation is done with a differential topology, meaning that the single-ended topology does not need any common-mode cancellation.

To check the inter-modulation distortion, a two-tone signal was used at the input. In figure 4.17 the output of the buffer is shown. In close inspection, it is possible to see

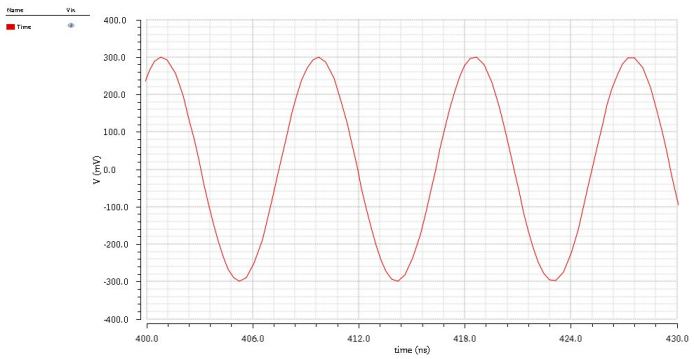


Figure 4.15: Cascaded Source Follower Output (time).

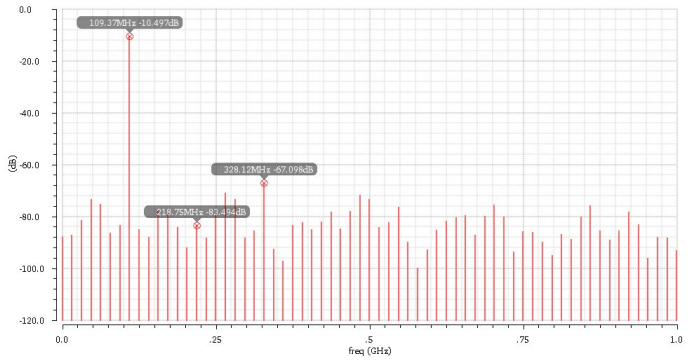


Figure 4.16: Cascaded Source Follower Output (frequency).

that the signal is somewhat different from the signal at figure 4.11 on page 35, yet this difference is expected and shown in the FFT of the signal.

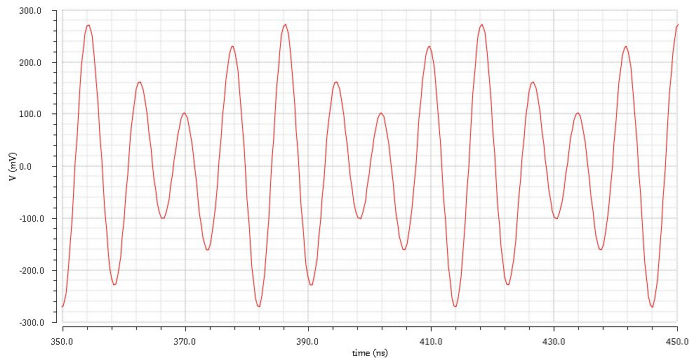


Figure 4.17: Cascaded Source Follower 2 Tone Analyse Output (time).

The FFT of the previous signal is a bit different from the previous ones because the tone with higher power at the output is the lower frequency one. However, this analysis does not require that the same frequency has the highest power across designs.

The measurements recorded in all of the three analyses are shown in the table 4.3.

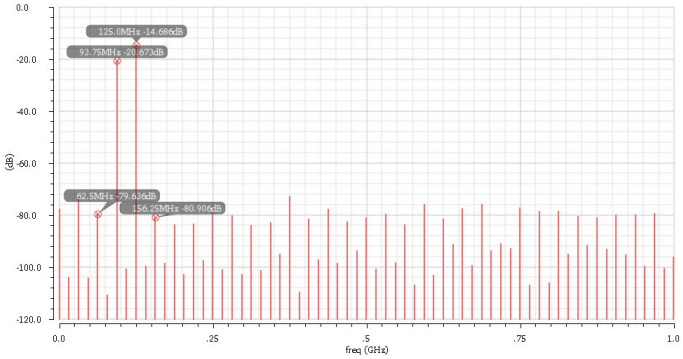


Figure 4.18: Cascaded Source Follower 2 Tone Analyse Output (frequency).

Table 4.3: Summary from the Cascaded Source Follower Simulation Analysis.

Current	4 mA
Bandwidth	1.6 GHz
DC Gain	-1.5 dB
Input Frequency	109 MHz
Offset Spur	-90 dB
HD2	-73 dB
HD3	-56 dB
IM3	59 dB
SFDR	57 dB
ENOB	8.1 bits

Comparing the results from the CSF and the SF (tables 4.3 and 4.1, respectively) it can be noted that they are pretty similar regarding linearity. Since the CSF is nothing more than two SF this was the expected result. By properly sizing the current on the first stage it's possible to reduce The power dissipation of the CSF.

4.4 Current Feedback Input Buffer Analysis

Current Feedback IB uses a current feedback loop instead of the voltage feedback of the SSF. This can increase the voltage dynamic range of the signal and have some improvements regarding noise and distortion.

4.4.1 DC Analysis (Current Feedback IB)

For the sizing of the circuit, [24] states that a $\frac{1}{4}$ ratio between the current of the main device and the feedback current provides the best noise-canceling results. However, the frequency response presents some irregularities at higher frequencies. This effect can be minimized by increasing the feedback current, that's why in figure 4.19 the feedback loop has a bit more current than expected. If the proposed ratio was kept, there would be a resonance at higher frequencies meaning that the linearity will suffer in the transient-noise analysis.

For these reasons, the feedback mirroring factor was increased to $\frac{1}{2}$, the least current that was able to negate the linearity issues with the transient-noise analysis.

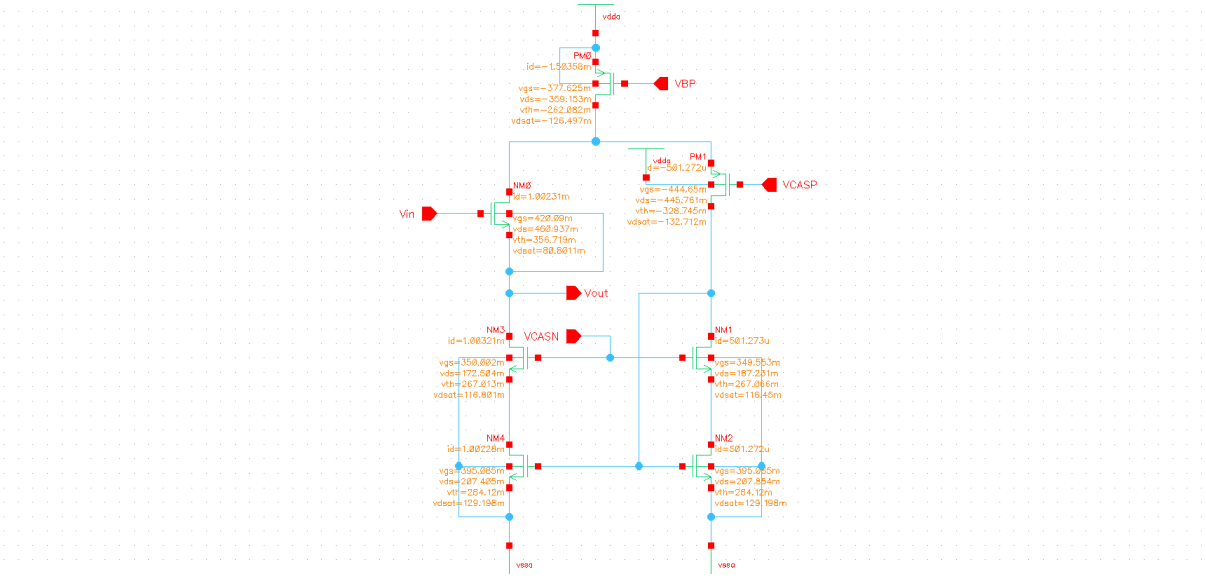


Figure 4.19: Current Feedback IB DC Operating point.

4.4.2 AC Analysis (Current Feedback IB)

The AC analysis of the Current Feedback IB shows the problem of this design. The pole-zero system of this circuit creates a peaking in frequency (Figure 4.20). This can be regulated by increasing or decreasing the currents in each branch. Notwithstanding, it is important to note that the change of the current ratio will have a major impact on the buffer's linearity.

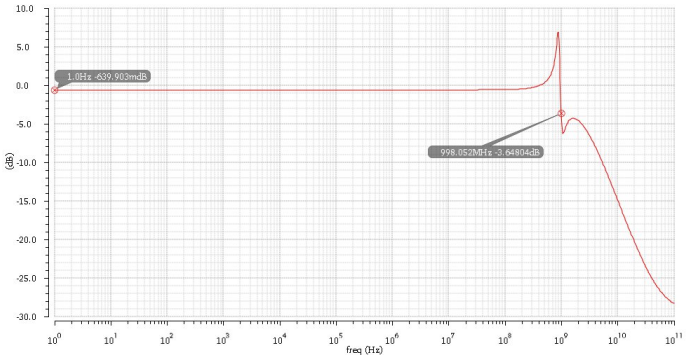


Figure 4.20: Current Feedback IB Frequency Response.

The bandwidth, calculated by the point with 3 dB decrease compared to the DC value, is 1.0 GHz but because of the abrupt peaking, the results at high frequency should be somewhat unpredictable if nothing more is used to regulate the frequency response.

4.4.3 Transient-noise Analysis (Current Feedback IB)

The transient-noise analysis did not present problems, the input frequency was fairly smaller compared to the frequency where the peaking exists. Figure 4.21 shows the differential output signal. Using an FFT the time domain signal was brought to the frequency domain, where it is possible to look at the power of each frequency.

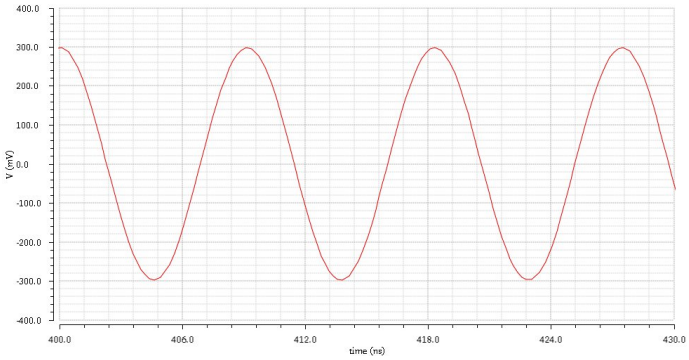


Figure 4.21: Current Feedback IB Output (time).

The signal FFT is shown in figure 4.22. The marked frequencies are the fundamental harmonic power, the second harmonic power, and the third harmonic power.

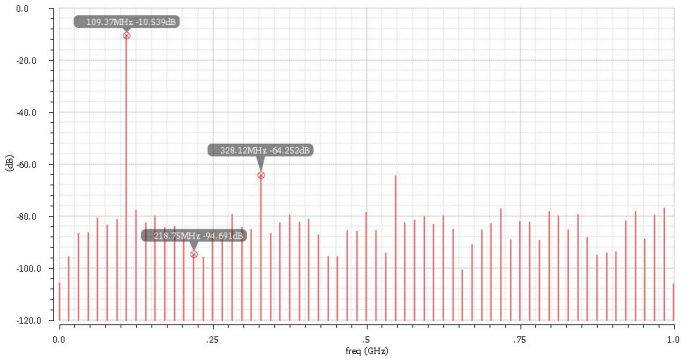


Figure 4.22: Current Feedback IB Output (frequency).

Setting up the 2 tone analysis and extracting the time wave presented no problem. The maximum differential output swing was set up to 600 mV and the result is presented in figure 4.23.

Checking the signal on frequency domain in figure 4.24, the same aspect as the other designs appears with two clear power spikes at the fundamental harmonics frequencies.

In the next table 4.4 the results of the analysis are shown.

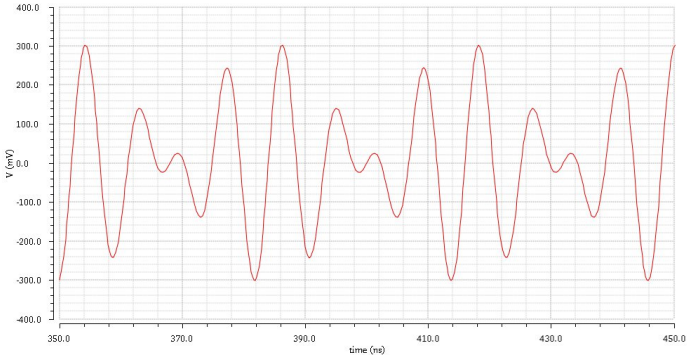


Figure 4.23: Current Feedback IB 2 Tone Analysis Output (time).

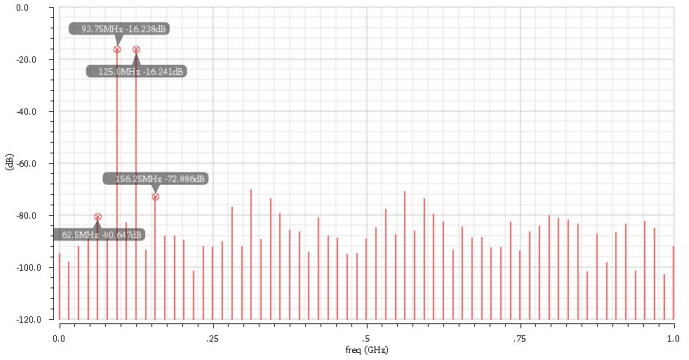


Figure 4.24: Current Feedback IB 2 Tone Analysis Output (frequency).

Table 4.4: Summary from the Current Feedback IB Simulation Analysis.

Current	3 mA
Bandwidth	1.0 GHz
DC Gain	-0.6 dB
Input Frequency	109 MHz
Offset Spur	-106 dB
HD2	-84 dB
HD3	-54 dB
IM3	57 dB
SFDR	54 dB
ENOB	7.9 bits

Comparing this buffer with the SSF (table 4.2 on page 35), it looks like this last buffer fails to improve the performance compared to the SSF. Yet, this happens because the input frequency is maybe too close to the frequency peaking of this last buffer. Keeping that in mind the results shown are not that bad and still are very close to every other buffer.

4.5 Vgs-controlled Cascaded Source Follower Input Buffer

The Vgs-controlled CSF IB uses a negative feedback loop to reduce the V_{gs} variances of each cascaded devices on the CSF design. This feedback loop is responsible to reduce the impact of input-dependant V_{gs} variations on the circuit, increasing linearity.

4.5.1 DC Analysis (Vgs-controlled CSF IB)

For the sizing of the circuit, the main objective was to ensure the output current through the NM1 device. All the currents and operating points of each device are shown in figure 4.25. For linearity and bandwidth results, the current used in both branches of the differential pair of PM5 and PM8 does not have the same current. Note that, even though PM8 is not is a traditional common-drain topology, [26] recommends to remove this device's body-effect.

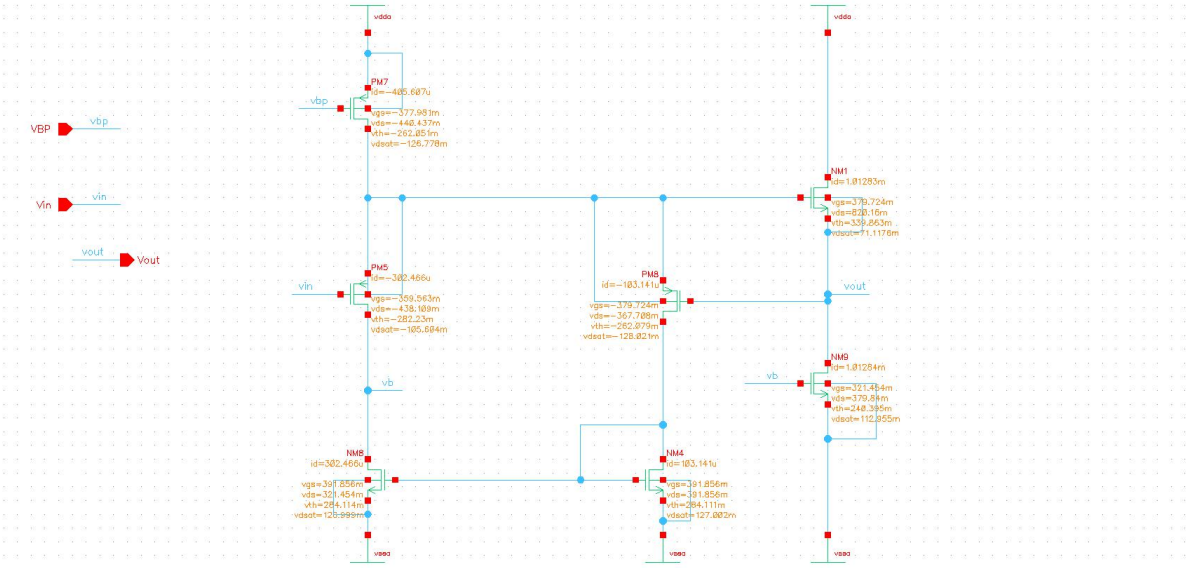


Figure 4.25: Vgs-controlled CSF IB DC Operating point.

It is reported in [26] the use of a compensation capacitor to increase the phase margin. However, since the simulated results were under the reported behavior, this capacitor was not used in these simulations.

4.5.2 AC Analysis (Vgs-controlled CSF IB)

The AC analysis of the Vgs-controlled CSF IB, shown in figure 4.26 shows that this design also has a peaking at higher frequencies. However, this peaking is not as noticeable as the Current Feedback IB's. Furthermore, the DC Gain of this design is comparable to the designs with one buffer stage, meaning that the feedback loop is also increasing the overall design's gain.

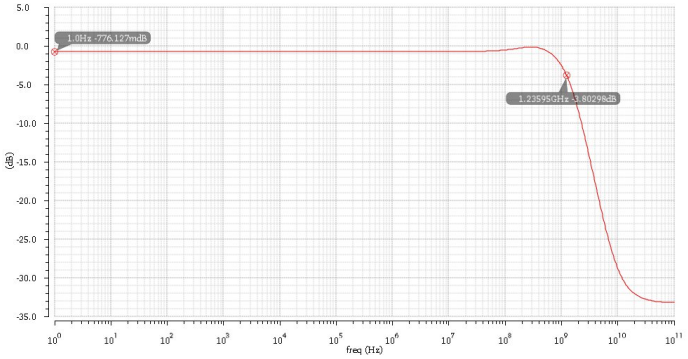


Figure 4.26: Vgs-controlled CSF IB Frequency Response.

The bandwidth value of this design, calculated by the point with a 3 dB decrease compared to the DC value, is 1.2 GHz. The peaking is felt around frequencies of the 100 MHz range.

4.5.3 Transient-noise Analysis (Vgs-controlled CSF IB)

The transient-noise analysis did not present problems, considering that the DC gain of this IB is comparable to every other one-staged buffer, the input voltage amplitude did not stress the operating point of any device. Figure 4.27 shows the differential output signal. Using an FFT the time domain signal was brought to the frequency domain, where it is possible to look at the power of each frequency.

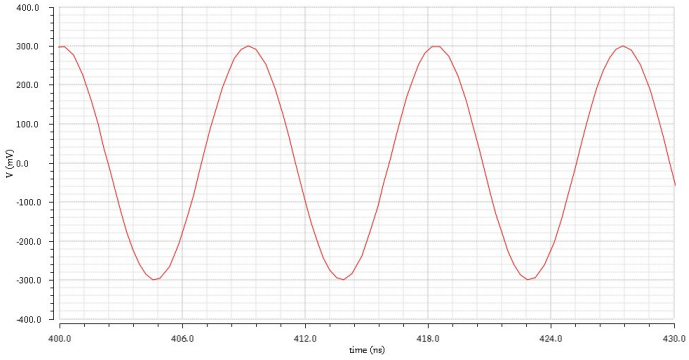


Figure 4.27: Vgs-controlled CSF IB Output (time).

The signal FFT is shown in figure 4.28. The marked frequencies are the fundamental harmonic power, the second harmonic power, and the third harmonic power.

Setting up the 2 tone analysis and extracting the time wave presented no problem. The maximum differential output swing was set up to 600 mV and the result is presented in figure 4.29.

Checking the signal on frequency domain in figure 4.30, the same aspect as the other designs appears with two clear power spikes at the fundamental harmonics frequencies.

In the next table 4.5 the results of the analysis are shown.

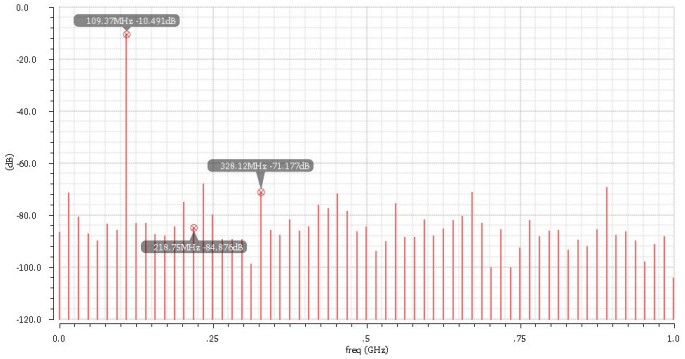


Figure 4.28: Vgs-controlled CSF IB Output (frequency).

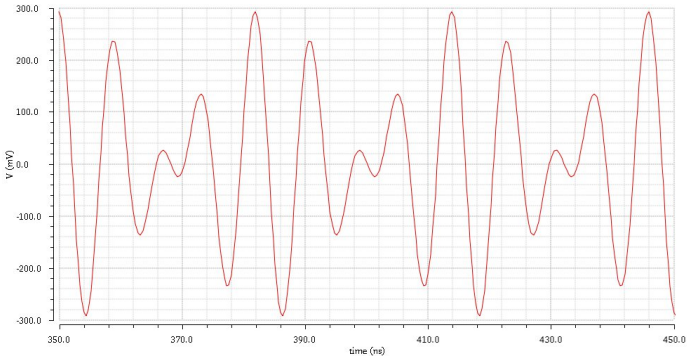


Figure 4.29: Vgs-controlled CSF IB 2 Tone Analysis Output (time).

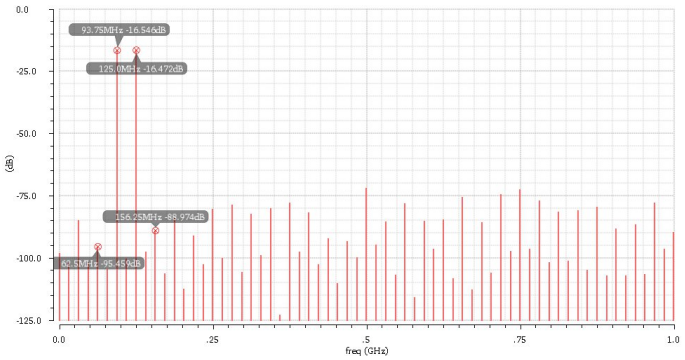


Figure 4.30: Vgs-controlled CSF IB 2 Tone Analysis Output (frequency).

Table 4.5: Summary from the Vgs-controlled CSF IB Simulation Analysis.

Current	2.8 mA
Bandwidth	1.2 GHz
DC Gain	-0.8 dB
Input Frequency	109 MHz
Offset Spur	-87 dB
HD2	-74 dB
HD3	-60 dB
IM3	72 dB
SFDR	57 dB
ENOB	8.1 bits

Comparing this buffer with the CSF (table 4.3 on page 39) there are many similarities between the two. One difference is regarding the bandwidth and the total current of the devices. Since this one has uses less current and has one more feedback loop it is expected that the bandwidth decreases as well. Another difference regards the DC Gain that, in this architecture, is doubled than that of the CSF.

4.6 Summary of all the analyses

In this chapter, some buffers were tested in the same conditions. Now, a fair comparison can be done between all of them. In table 4.6 such comparison is drawn.

Table 4.6: Comparison between tested designs analysis.

	SF	SSF	CSF	Current Feedback IB	Vgs-Controlled CSF IB
Current (mA)	2.0	2.6	4.0	3.0	2.8
Bandwidth (GHz)	1.5	1.7	1.6	1.0	1.2
DC Gain (dB)	-0.7	-0.8	-1.5	-0.6	-0.8
Input Frequency (MHz)	109	109	109	109	109
Offset Spur (dB)	-121	-105	-90	-106	-87
HD2 (dB)	-80	-85	-73	-84	-74
HD3 (dB)	-55	-61	-56	-54	-60
IM3 (dB)	70	66	59	57	72
SFDR (dB)	55	61	57	54	57
ENOB (bits)	8.0	8.5	8.1	7.9	8.1

Regarding current, most of all simulated designs use around the 2.5 mA. The CSF is the only one not on this mark, mostly because the first stage was not power-optimized and was drawing 1 mA like the second stage.

Bandwidth wise, there was a major performance boost on the SSF. There still is the problem regarding peaking on the SSF and Current Feedback IB designs. The was most assuredly from some pole-zero frequency response that probably should need some more study to prevent or minimize the effects without changing much of the current. Another way to reduce this effect is by using some outside regulation.

The DC Gain of all topologies was similar and around -0.7 dB. Yet, the CSF had -1.5 dB gain. This difference is explained by the use of two different buffer stages, while every other buffer had just one device in common mode configuration, the CSF had 2 devices. By assuming that each device attenuates the output by 0.7 dB having a gain of around -1.4 dB with 2 of these devices is expected. However, the Vgs-controlled CSF IB topology does not have this problem, this happens because of the feedback loops in the design.

Regarding HD2, or any even harmonic distortion, no design was much affected by this. The differential nature of the buffers reduces even harmonic distortion, as shown in this analysis. HD3, and odd harmonic distortion, is a major concern not easily solvable. Out

of all of the tested designs, the best performance was achieved by the SSF, this might have happened since it was the buffer with feedback that had the most output swing available.

The ENOB is mostly used for data converters. Yet, because the buffer will limit the effectiveness of the ADC it is used here to have a better understanding of the efficiency of the buffer. In this category, the best buffer is SSF while all the other four have pretty similar results.

Comparing SFDR the best result comes from the SSF, which is not surprising since this metric, in this case, would be mostly limited by the HD3, as it is possible to see in the figures representing the spectrum of each tested design.

The next chapter presents a new buffer architecture and the ideas behind the design. Furthermore, the proposed buffer will be subject to the same analysis done. Lastly, the proposed design will suffer some improvements to maximize its performance.

Chapter 5

Proposed Architecture

A new architecture is studied for a high-bandwidth and high-linearity IB. The main ideas and schematic are presented briefly, followed by the simulation of the new IB and some improvements. Lastly, at the end of this chapter, some comparisons are made between this architecture and the state-of-the-art.

5.1 Circuit Schematic and Proposed Idea

The schematic of the proposed architecture, shown in figure 5.1, is based on some previously shown buffer architectures with two different feedback branches. Firstly the transistors M1 and M2 form a cascaded source follower in which the two have a different branch of feedback loops. The first feedback loop happens in the loop of M7, M8, M9, and M4. If the voltage at the gate of M2 rises while $Vout$ is constant the V_{gs} of M2 rises and the current increases. With that increase the drain of M2 will decrease, decreasing as well the V_{gs} of M7 (VCASP is constant) and the current through M7. The decrease in current in M7 will decrease the drain voltage that is the gate voltage on M4, meaning that the V_{gs} decreases, its current decreases, and $Vout$ increases. This one serves as a positive feedback loop to guarantee some bandwidth by quickly raising $Vout$ when the input increases.

The second feedback loop starts from the voltage of the gate of M4 and M9 to M14. Completing the example above, if the voltage at the gate of M14 decreases then the current decreases, since V_{gs} decreases. This will increase the drain voltage of M14, which is the same as the M5 gate voltage. The increase of the gate voltage will reduce V_{gs}, decreasing the current and lowering the drain voltage of M5. The drain voltage of M5 is the gate of M2 contrary to the supposed rise of the last paragraph, meaning that this is negative feedback.

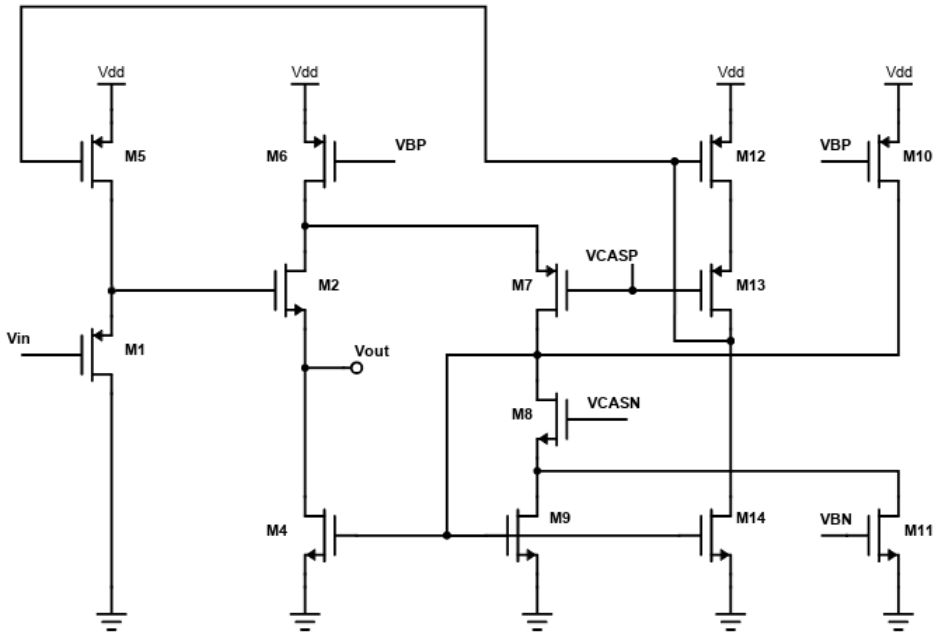


Figure 5.1: Proposed Input Buffer Architecture.

The last two devices that need introduction are M10 and M11. These devices are responsible to assure that M8 is always in the strong-inversion state.

5.2 DC Analysis

Since this buffer is the one with the most branches, it should be the one with the most power drained out of all simulated. The sizing was done to have a current of 1 mA flowing through M2, in accordance with the guidelines. There is no need to have a high current flowing through M1. In this case the current was sized to be 350 μA.

The current feedback ratio in the first feedback loop was sized to be around 1/3 of the current flowing through M2. This means that the current from M7 and M9 was set to be around 333 μA. The higher this ratio the higher the bandwidth and the higher the peaking at higher frequencies.

The second feedback loop was sized to be 1/1. This choice was made taking into account that the results were better with a higher ratio and, since the current in M1 will be lower compared to M2, it was decided that the same current could be used in this feedback loop.

Moreover, M10 and M11 were set to drive M8 with a significantly higher current so they were set to have 3mA. These values were simulated to have some significant improvements on the bandwidth with minimal increases in the power needed for the block. This current minimized the total peaking of the buffer.

Figure 5.2 shows the operating point that was used for the next simulations. Even though the sizing of the transistors was achieved and every device is in the strong inversion region, it is possible to see that there is no comfort range for most of the devices. One way to increase this range would be to decrease the use of cascode devices. However, since the desired operation point was achieved, it was decided to continue with said devices.

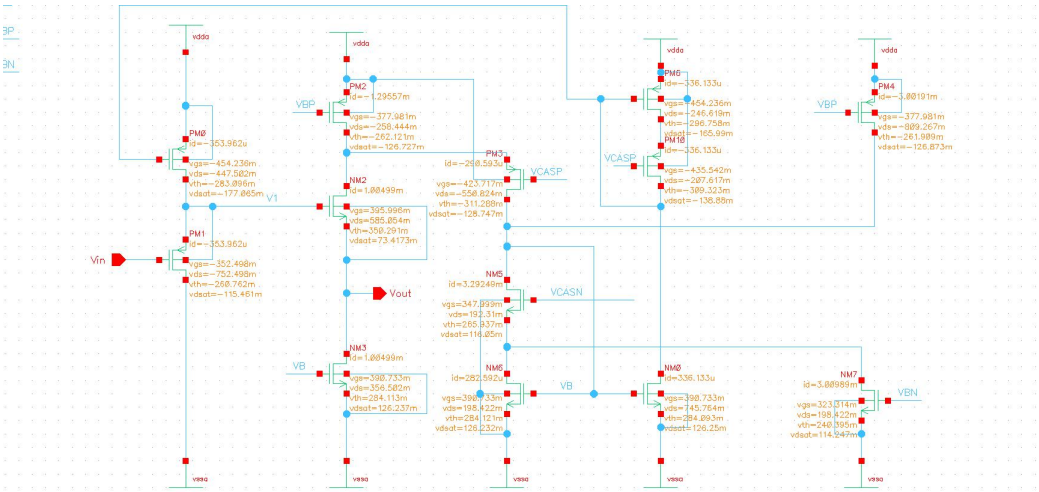


Figure 5.2: Operating point of proposed architecture.

Aside from that, there was caution in sizing M10 and M11, the devices used to bias M8 into the strong-inversion region. If there was a small variance of the current between the two of them then there would be the same variance between devices M7 and M9. The said variance could have some negative impact on the buffer's performance. As such, devices M10 and M11 were sized to have the same current flowing through them.

5.3 AC Analysis

In this AC analysis, the study was divided into 3 stages. The first one refers to the output/input of M1 controlled by the negative feedback, the second one refers to the output/input of M2 controlled by the positive feedback and the last one refers to the full buffer output/input.

Figure 5.3 represents the gain between the gate and the source of M2. This transfer function is similar to the gain of the Current Feedback IB in figure 4.20 on page 40. This trace represents the positive feedback and it is noticeable the peaking at a higher frequency. With the sizing presented in figure 5.2 the bandwidth between the gate and the source of M2 is about 2.6 GHz.

In figure 5.4 it is possible to see the outwards loop gain, between the gate and source of M1. This trace is the first to have a "negative"peaking mostly because it is a negative feedback loop. At the same higher frequency this loop will decrease the gain with a similar effect to the previous one. In this loop, the bandwidth achieves around 7 GHz. However, it is important to note that the node capacitance shouldn't be that high considering that

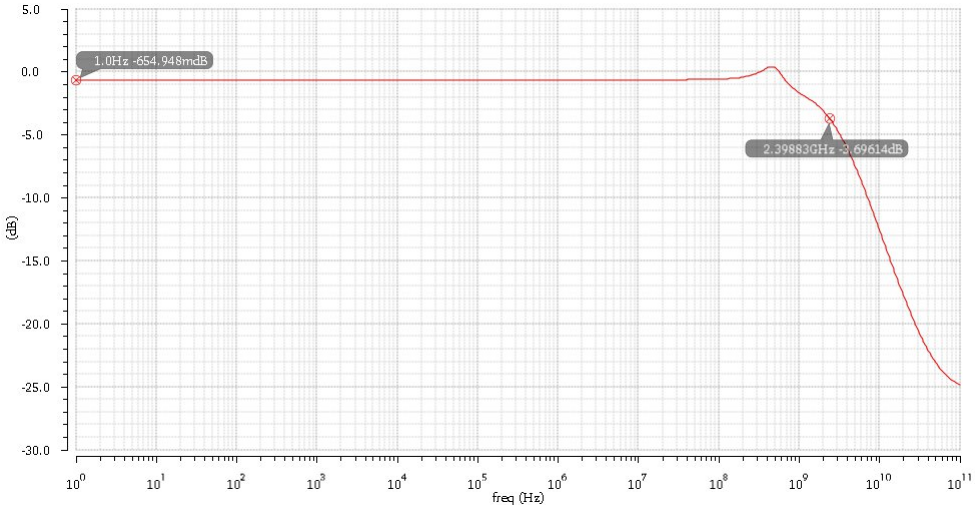


Figure 5.3: Gain on the positive feedback loop.

it is, essentially, the gate of M2. This assures the increase in bandwidth should increase, since the load is smaller, which validates the assumption that the current through M1 does not need to be that high, compared to M2.

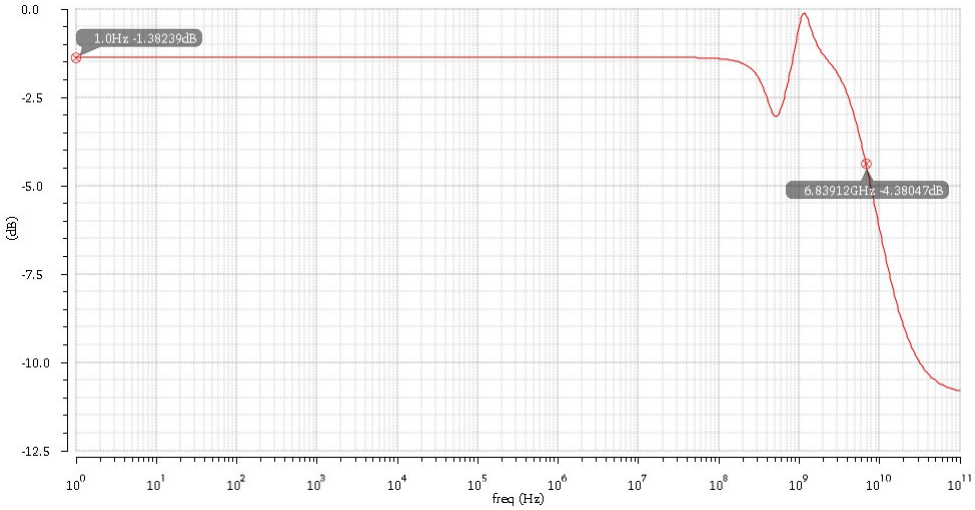


Figure 5.4: Gain on the negative feedback loop.

Figure 5.5 shows the two traces together and check that both peakings are achieved at similar frequency ranges. This is a good aspect of this architecture since, if the sizing is well achieved, the two peakings should cancel each other into a constant gain of the buffer until the cutoff frequency.

The result of the total buffer is shown in figure 5.6, here the gain is mostly flat until the cutoff frequency. The total bandwidth achieved is around 2.5 GHz. This means that the negative feedback is controlling the positive feedback peaking, as was expected. It is important to note that this value is smaller than any individual value of the stages, however, the gain is, overall, flatter than any other.

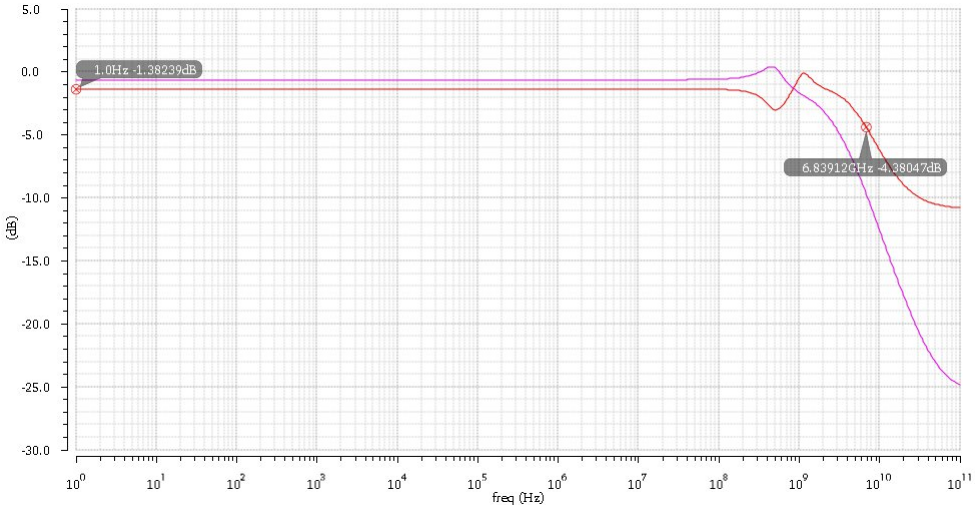


Figure 5.5: Comparison between positive and negative feedback loop gains.

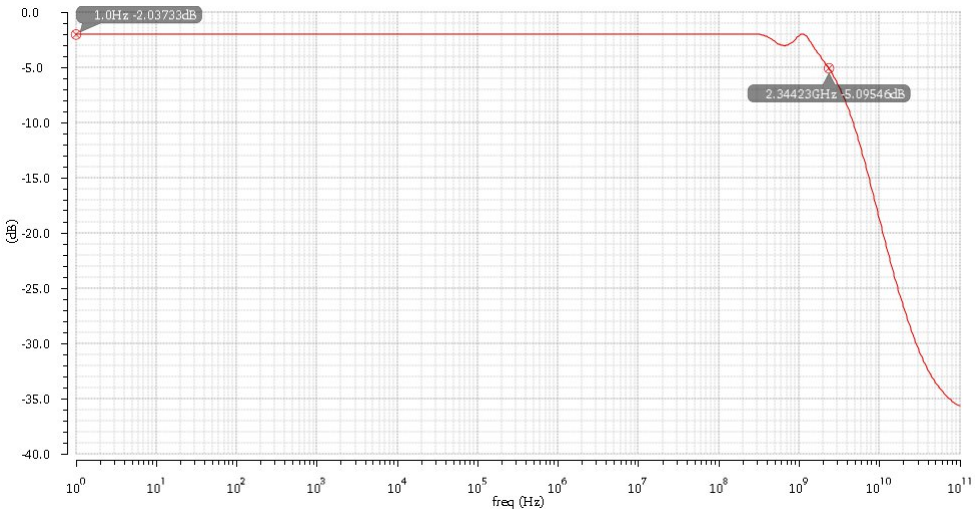


Figure 5.6: Proposed architecture frequency gain.

5.4 Transient-noise Analysis

The transient-noise analysis was used to calculate the linearity of the buffer. A single tone input wave was used to determine the harmonic distortion of the system. In figure 5.7 the output sine wave is visible. This wave was aimed to have 600 mV peak-to-peak differential voltage to be able to compare with all other buffers studied before.

The time wave was decomposed into all the frequency power FFT presented in figure 5.8. It's possible to see the fundamental harmonic peak, in this figure. Besides, are marked the 2^{nd} and 3^{rd} harmonics.

After the single tone analysis, the double-tone analysis was set and the process repeated. In figure 5.9 the time wave is shown as the output of the buffer when the input is composed of two different frequency signals. The target was to have an output wave with a maximum voltage of 600 mV peak-to-peak differential.

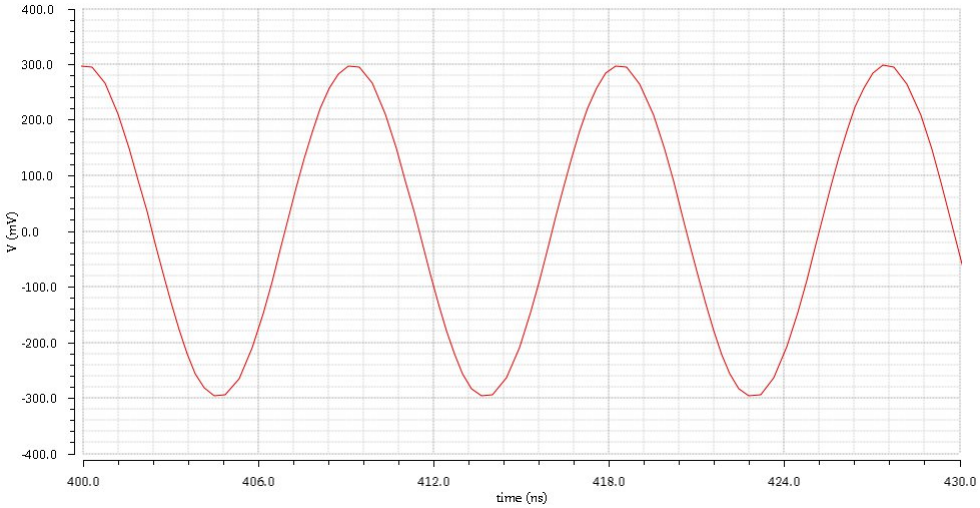


Figure 5.7: Proposed architecture Output (time).

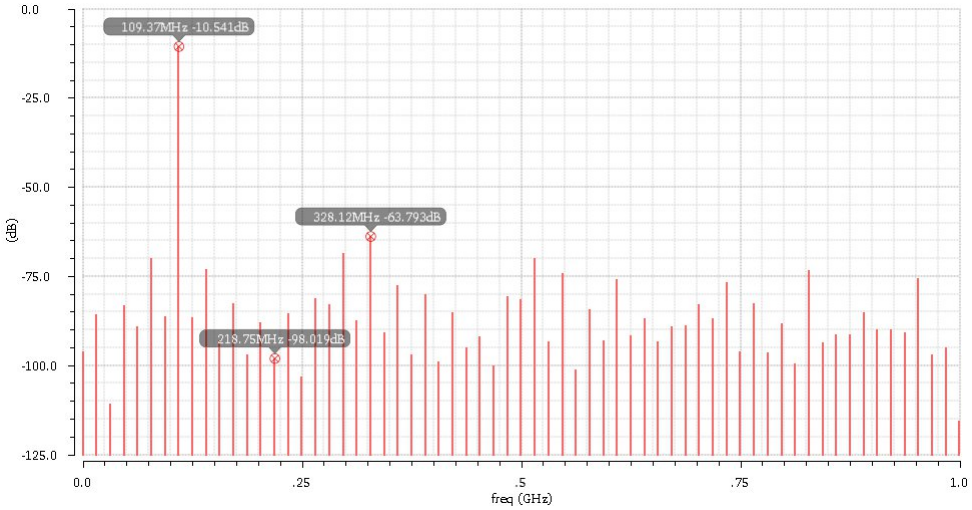


Figure 5.8: Proposed architecture Output (frequency).

Taking a look at the FFT of the previous signal, in figure 5.10 it is possible to see the fundamental harmonics of the different signals. Even though it looks like the inter-modulation is worst in this buffer compared to the previous ones, this is certainly not the case, since the inter-modulation frequencies are, in fact, better, yet the reason why it is fairly different from previous buffers is that most of the other frequencies got attenuated even more and, for the sake of showing up all the calculated points on the FFT, the scale is different on this one figure.

Lastly, in table 5.1 it's compiled some results of the previous analysis.

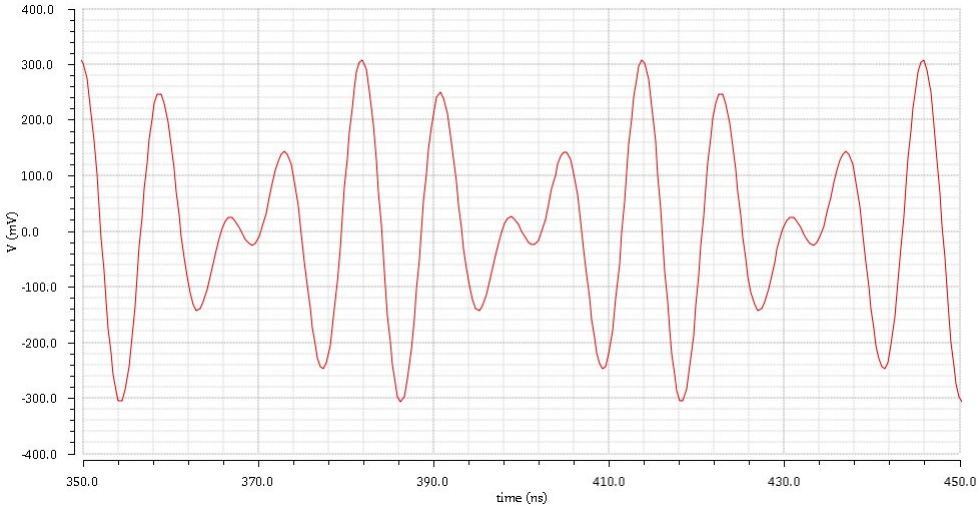


Figure 5.9: Proposed architecture 2 Tone Analysis Output (time).

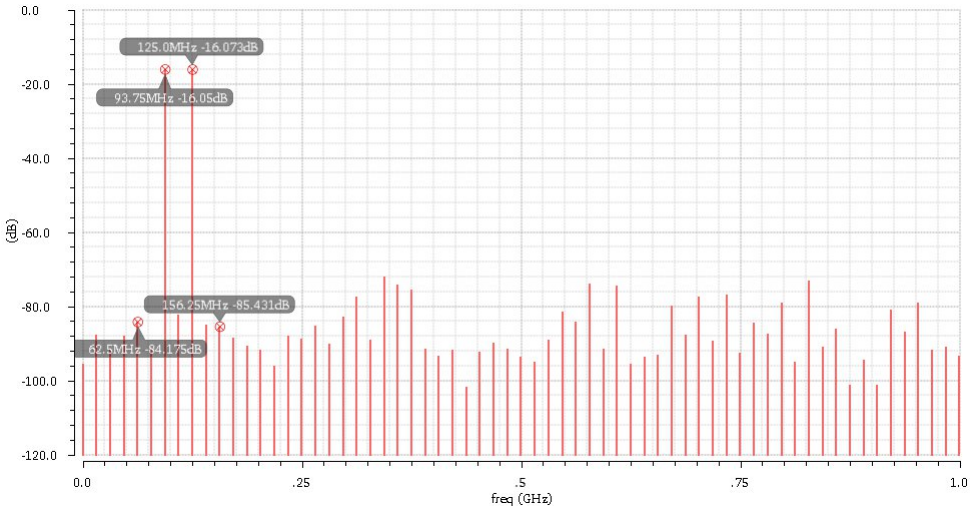


Figure 5.10: Proposed architecture 2 Tone Analysis Output (frequency).

Table 5.1: Summary from the Proposed Buffer Simulation Analysis.

Current	10 mA
Bandwidth	2.3 GHz
DC Gain	-2.0 dB
Input Frequency	109 MHz
Offset Spur	-96 dB
HD2	-87 dB
HD3	-53 dB
IM3	68 dB
SFDR	53 dB
ENOB	7.9 bits

Regarding current, it is important to note that most of it (6 mA) comes from the extra branch used to drive the cascoded NMOS. That branch can give more or less flatness to

the overall frequency gain by increasing or decreasing the current, respectively.

The last point can improve bandwidth by flattening the gain until higher frequencies but that would increase the power dissipation slightly.

In terms of linearity, the results were not as expected. That is why we made some variations to try and get some better performances.

5.5 Proposed Buffer with Improvements

As stated in section 5.2 - DC Analysis, some devices are at almost barely in strong inversion, and that some variation on the V_{ds} of said devices would take them out of the strong inversion, creating non-linearity issues. Trying to keep the same ideas behind the circuit, figure 5.11 shows the last circuit changes compared to figure 5.1. The only difference is that transistor M13 from 5.1 does not show up in figure 5.11.

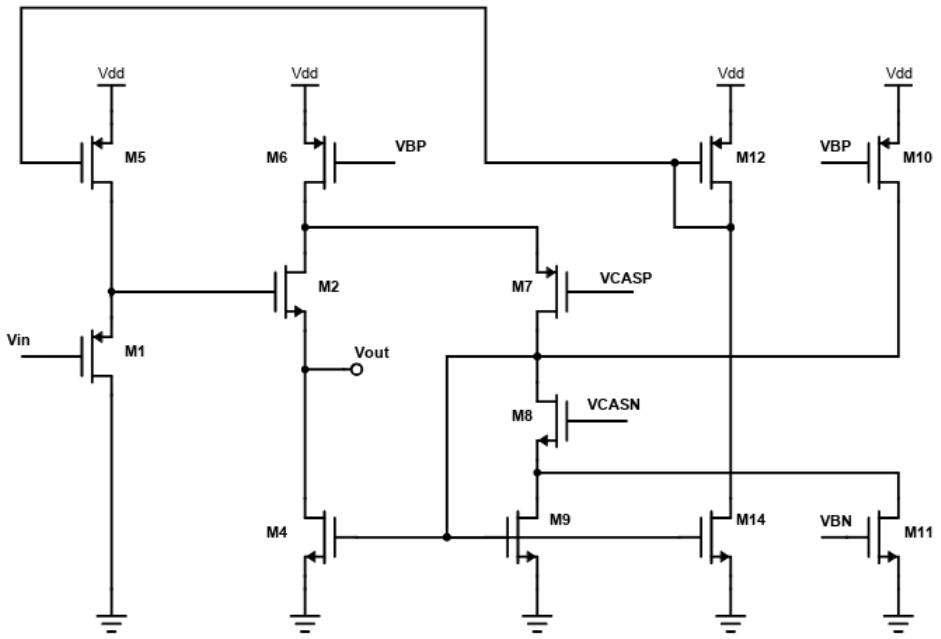


Figure 5.11: Proposed Improved Input Buffer Architecture.

Regarding the buffer's TF, it was used MSAAC to achieve some expression on the gain. However, even with a high tolerance for expression errors, it proved to be too complex to analyze. Yet, if we ignore the parasitic effect of every device, the DC Gain expression of the circuit is found to be

$$\text{DC Gain} = \frac{g_{m1} \cdot (g_{m4} + g_{m9})}{g_{m1} \cdot (g_{m4} + g_{m9}) + g_{m4} \cdot (g_{ds1} + g_{ds5}) + g_{m9} \cdot g_{ds1}}, \tag{5.1}$$

with 10% error. The DC Gain expression is affected by the first stage of the design, as it will be reinforced in the following analysis. It is also possible to check that most of the

corruption factor is dominated by g_{m4} meaning that, optimally, to achieve an ideal buffer these values should be the smallest possible.

5.5.1 DC Analysis

Naturally, since most of the problems were regarding linearity, taking out the M13 device proved to be a sound improvement. This device is a cascode for M12, responsible to protect the drain of the latter against significant voltage variations. However, it is significantly easier to remove than the M8 device. Since eliminating the latter one would mean to lose the extra freedom from the boosted current.

With this changes it was possible to reduce the total current of the circuit. For this DC operating point the current through M1 and M2 were unchanged, 350 μA and 1 mA respectively. The current through M9 and M7 was lower to about 1/4 of M2's current value (i.e. 250 μA). The second feedback loop's current was lower as well. It was chosen a current feedback ratio of 1/3 meaning that the current flowing in M14 and M12 is about 116 μA. Lastly the current in M10 and M11 was lowered to 1 mA.

All these changes were done to achieve the best results of bandwidth and linearity with the least possible total current. However, the same functions described earlier apply. Meaning that the current through M2 was chosen because of the guidelines. M1's current was chosen to achieve a certain bandwidth. M7's current was chosen for bandwidth and peaking concerns. Finally, M11's current was chosen to limit the total peaking of the buffer.

In figure 5.12 the operating point of the buffer is shown. Note that the currents on all branches have been further optimized to have the minimum necessary biasing current for this buffer. In the end, with this simplification, it was possible to achieve the same results with a little more than half of the previous power dissipation.

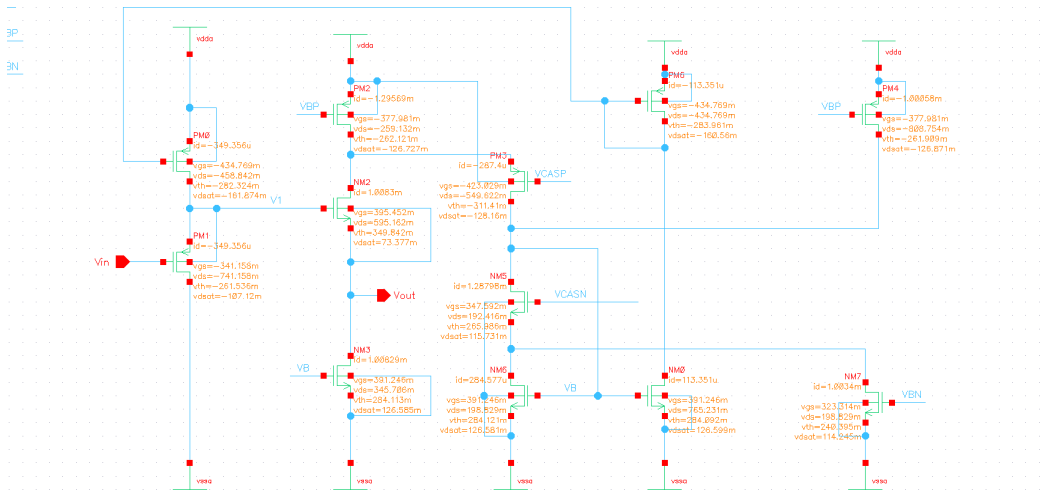


Figure 5.12: Operating point of proposed improved input buffer architecture.

This change does improve current consumption. However, this is working with almost the absolute minimum to have a stable and improved performance. The linearity performance is so noticeable that even with this low power budget it is relevant.

5.5.2 AC Analysis

Similarly, as with the first proposed architecture, the AC analysis was divided into three. The first one refers to the first stage buffer by M1 device, the second one to the second stage buffer by M2 device, and the last one with the total buffer.

Figure 5.13 represents the positive feedback loop gain. It has the peaking at high frequency and a bandwidth of around 2.2 GHz. This peaking is the main reason why an outside negative feedback loop is used.

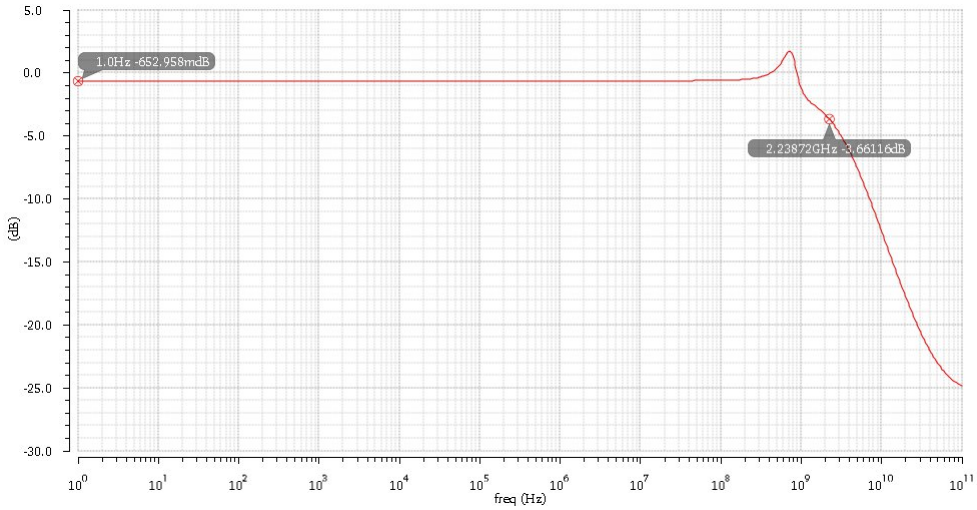


Figure 5.13: Gain on the positive feedback loop (Improved).

The negative feedback loop TF is shown in figure 5.14. Here the peaking is negative so that they would cancel each other when together. Since M1 only has to drive the gate of M2 the bandwidth of this buffer is better, even though there is less current through M1 compared to M2. The bandwidth of this branch is 6.1 GHz.

For easy comparison between the two figures 5.13 and 5.14 were compiled together in figure 5.15 where it is possible to confirm that both peakings are roughly at the same frequency.

Figure 5.16 shows the buffer total TF. Note that this operation point flattens the response much better than the one on figure 5.6 with the same current. This, again, might be a reason to decrease the circuit's biasing current. The overall bandwidth of the system is 2.2 GHz.

So far the overall bandwidth is almost the same between designs, however, the improved version does improve the flatness of the TF, making so that the buffer can have a better performance at higher frequencies without a need for outside calibrations. If the flatness is important then it is advised to consider increasing the current on devices PM4

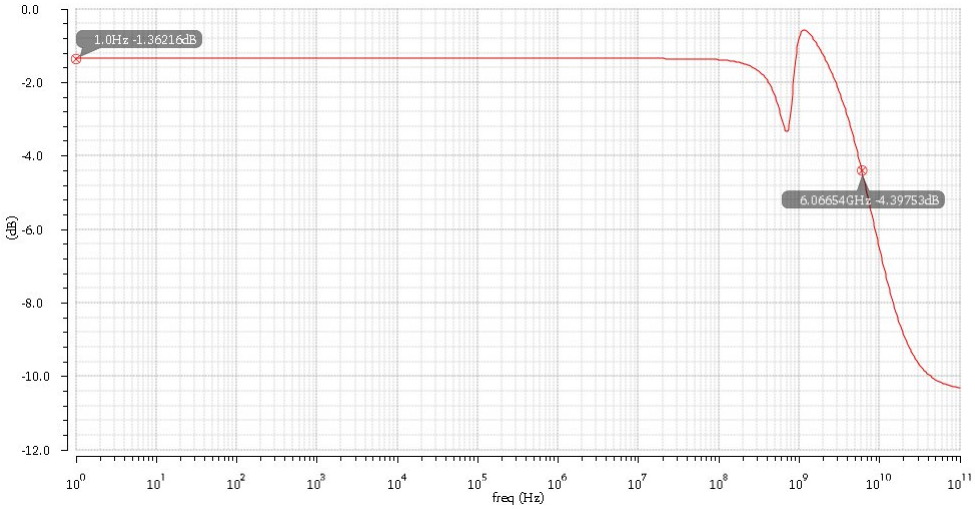


Figure 5.14: Gain on the negative feedback loop (Improved).

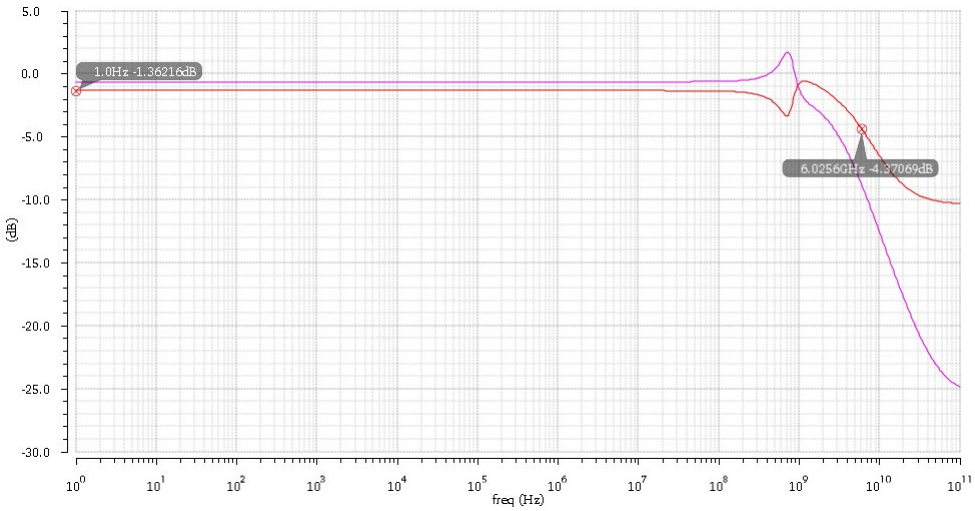


Figure 5.15: Comparison between positive and negative feedback loop gains.

and NM7 of figure 5.12 since higher currents on this branch tends to decrease the effect of the total peaking.

5.5.3 Transient-noise Analysis

The transient-noise analysis followed the same pattern as all the previous ones. Firstly a single tone input was used in a way to achieve a 600 mV peak-to-peak differential output voltage. The output signal is presented in figure 5.17.

This wave, aside from confirming the output maximum level, does not provide much information. It is necessary to use its FFT to extract some more information. Figure 5.18 shows the fundamental harmonic frequency as well as the second and third harmonic power. In this, it is possible to see that both of these harmonic distortions got slightly improved over figure 5.8.

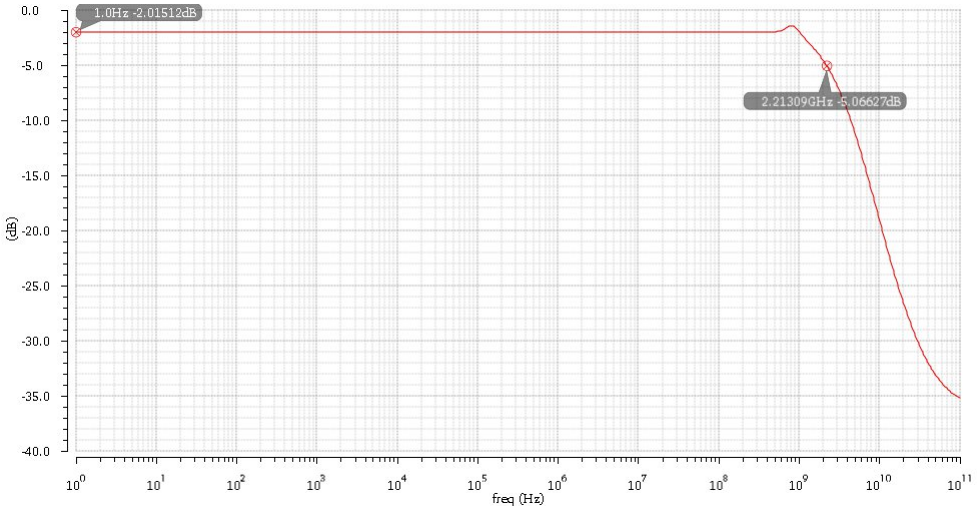


Figure 5.16: Improved Architecture frequency gain.

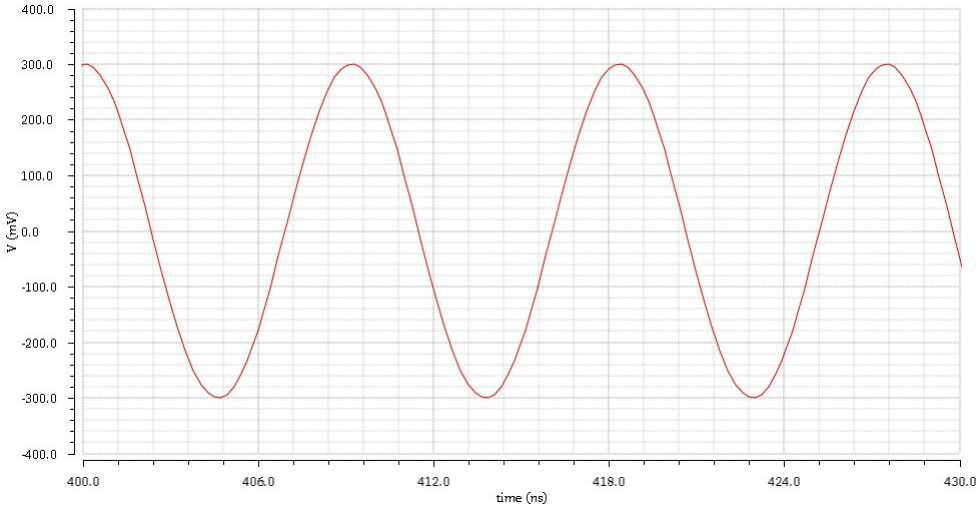


Figure 5.17: Improved Architecture Output (time).

After the single-tone analysis, the double-tone analysis could begin. Figure **??** shows the output signal with 2 similar frequency signals.

Using the FFT of the previous signal is it possible to check the IM3 of the buffer. In figure 5.20 it is possible to see the two fundamental harmonics of the signals present in the input. The other two marks show the power value of the IM3.

Lastly, in table 5.2, the results from all of these last analyses are shown.

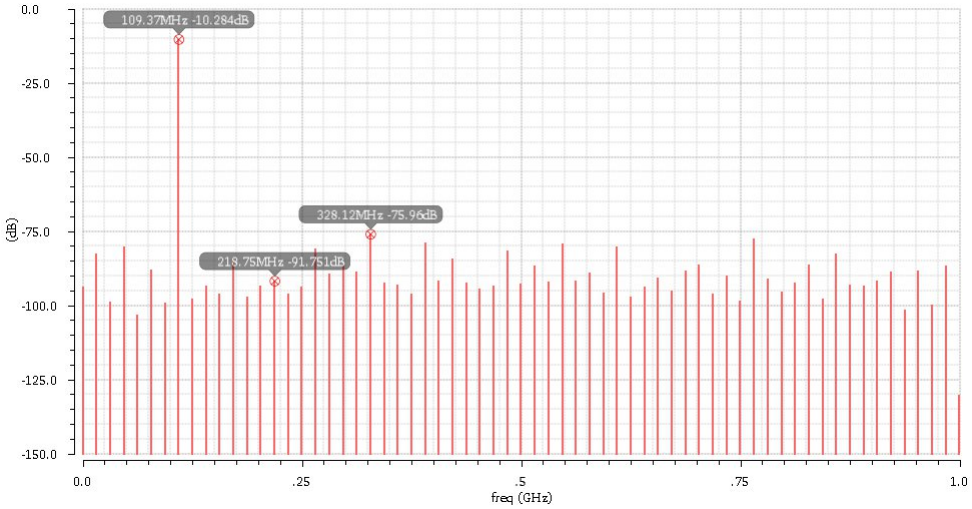


Figure 5.18: Improved Architecture Output (frequency).

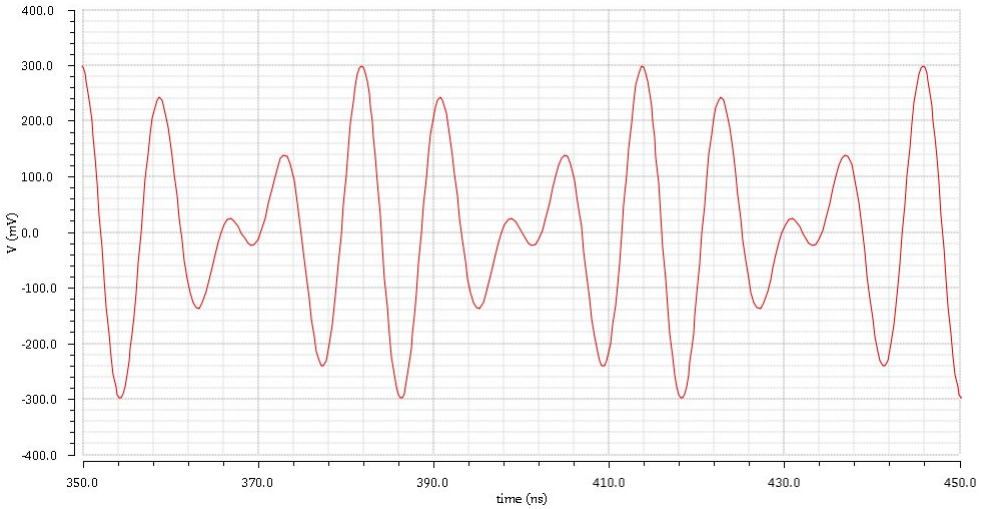


Figure 5.19: Improved Architecture 2 Tone Analysis Output (time).

Table 5.2: Summary from the Improved Buffer Simulation Analysis.

Current	5.5 mA
Bandwidth	2.2 GHz
DC Gain	-2.0 dB
Input Frequency	109 MHz
Offset Spur	-95 dB
HD2	-81 dB
HD3	-66 dB
IM3	65 dB
SFDR	66 dB
ENOB	9.3 bits

It is possible to see a noticeable improvement over the values on table 5.1. This improvement certifies that there was too much strain on some of the first design's devices.

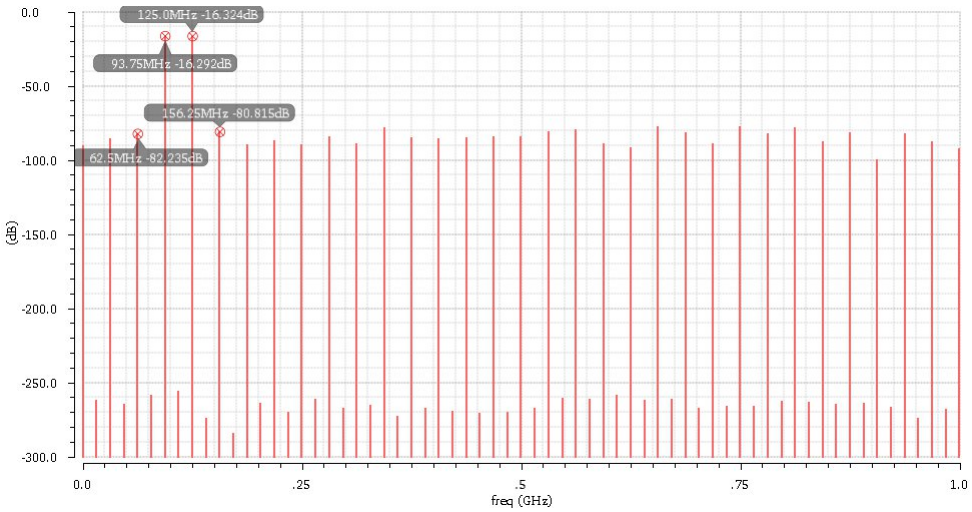


Figure 5.20: Improved Architecture 2 Tone Analysis Output (frequency).

Because the output needs to be at a predefined value besides this buffer having low DC gain, it was necessary to have an input signal with high amplitude. The high input voltage might have been responsible to cause all the non-linearity in the previous buffer.

Of course, since this one had such a better performance in both AC and transient-noise analysis, it would be recommended to try and decrease the current supply further to make it more competitive with the simulated buffers in chapter 4 - Analysis and Results of the state-of-the-art. However, this simple change was made to improve the proposed buffer.

Table 5.3 shows the improvement without the cascoded transistor.

Table 5.3: Comparison between simulated designs analysis.

	Proposed IB	Proposed Improved IB
Current	10 mA	5.5 mA
Bandwidth	2.3 GHz	2.2 GHz
DC Gain	-2.0 dB	-2.0 dB
Input Frequency	109 MHz	109 MHz
Offset Spur	-96 dB	-95 dB
HD2	-87 dB	-81 dB
HD3	-53 dB	-66 dB
IM3	68 dB	65 dB
SFDR	53 dB	66 dB
ENOB	7.9 bits	9.3 bits

It is possible to see that overall the improved proposed input buffer is 10 dB better compared to the proposed buffer in terms of SFDR. While the power dissipated by the improved design is about half of that of the first proposed design, both bandwidths are pretty much the same.

The DC gain is the same in both input buffers and is mostly limited by the first stage gain. Equation (5.1) shows that most of the DC gain depends just on the first stage

meaning that, to improve the overall gain it is most important to improve the first stage gain.

Linearity wise, the improved proposed buffer does have a better performance in all regards. This design is not as restrained by the change of the voltage levels of the input. However, in terms of IM3, the first proposed design does come out ahead but by a small margin.

Still about linearity, the value of HD3 is lower than that of the Current Feedback IB in table 4.4 in page 42. However, the value of HD2 is actually bigger than the value on that same table. This indicates that the second feedback loop may improve HD3 while degrading HD2 what, in this case, is preferable. The HD2 value will be attenuated since it is used a differential input buffer.

In the next chapter, the comparison between these two architectures will be done against all previous designs.

Chapter 6

Conclusion

Over the course of the last two chapters, there was a series of simulations done to several different buffer designs. In this chapter, the last comparisons will be drawn between all studied designs. This chapter presents an overall conclusion on the work done, problems faced, and future work that could be pursued to further improve this investigation.

6.1 Final Comparisons

In chapter 4 - Analysis and Results of the state-of-the-art - it was studied 4 different architectures. They were the Source Follower (Section 4.1), Super Source Follower (Section 4.2), Cascaded Source Follower (Section 4.3) and Current Feedback IB (Section 4.4).
In chapter 5 - Proposed Architecture - a new buffer design was presented, as well as an improvement more suited for low voltage supply.
At the end of both of these chapters was shown a table compiling the results of all of the analyses to make an easy comparison between designs of the same chapter, however, there was not a comparison between all the designs.
Table 6.1 shows all of the results into a single compacted table for easy comparison.

Table 6.1: Comparison between tested designs analysis.

	SF	SSF	CSF	Current Feedback IB	Vgs-Controlled CSF IB	Proposed IB	Proposed Improved IB
Current (mA)	2.0	2.6	4.0	3.0	2.8	10	5.5
Bandwidth (GHz)	1.5	1.7	1.6	1.0	1.2	2.3	2.2
DC Gain (dB)	-0.7	-0.8	-1.5	-0.6	-0.8	-2.0	-2.0
Input Frequency (MHz)	109	109	109	109	109	109	109
Offset Spur (dB)	-121	-105	-90	-106	-87	-96	-95
HD2 (dB)	-80	-85	-73	-84	-74	-87	-81
HD3 (dB)	-55	-61	-56	-54	-60	-53	-66
IM3 (dB)	70	66	59	57	72	68	65
SFDR (dB)	55	61	57	54	57	53	66
SNDR (dB)	50	53	50	49	51	49	58
ENOB (bits)	8.0	8.5	8.1	7.9	8.1	7.9	9.3
FOM $\left(\frac{\text{MHz}\cdot\text{pF}}{\text{mW}}\right)$	625	544	333	278	357	192	333
FOM-Walden (fJ)	3.10	2.51	5.81	7.82	4.83	11.3	2.31

As is it possible to see, the proposed design falls in the middle of all designs, overall. Even though it is the one with the higher bandwidth, it still falls short regarding some of the distortion parameters as well as current consumption.

Power-wise, the worst designs are the proposed and the proposed improved designs, however, the latter has a significant improvement over all the others. Aside from these two, the CSF is the worst in terms of power demand, but as stated in its section of this work, this design needed not that much power in the first branch to have similar results.

Regarding bandwidth, both the proposed and the proposed improved designs have the top performance. It is important to note that both the current and the load were the same across simulations. This means that the output resistance of the buffer was the difference-maker on these values.

The DC Gain is another major concern point for this new architecture. Even assuming that the DC gain could not be comparable to single-stage buffers, both the proposed and the proposed improved buffers have a significantly larger attenuation that the CSF.

In regards to linearity, the best design is, by far, the proposed improved design. Keeping SFDR at higher than 65 dB and SNDR improvement of 7 dB over the second best. However, the best HD2 comes out from the proposed design without improvement. Furthermore, even though the best IM3 value comes from the Vgs-Controlled CSF IB, all of the design had a similar performance concerning this point.

The ENOB metric is mostly used for data converters. However, in this work, it serves

as a metric for the performance of the design. Note that most of the buffers in these conditions were pretty constant with 8 bits with the SSF coming off around with 8.5 bits ENOB. The best one on this metric was once again the improved design with more than 9 bits ENOB.

The first Figure of Merit (FOM) was proposed in [15]. It describes a way of calculating a FOM by relating the Product Gain Bandwidth (GBW) with the capacitive load and the power dissipation of the OpAmp. In [15] it is used

$$\frac{\mathrm{GBW} \cdot \mathrm{C_L}}{Power} \tag{6.1}$$

to calculate the value of this FOM. In this work, since the DC gain is around 0 dB instead of using GBW we used BW. These values represent the efficiency of the block in term of BW. In this case, the higher the value means better performance. Naturally, because it had only one stage and had no feedback, the SF was the best design in this regard followed by the SSF. Note that this FOM does not consider linearity.

The FOM-Walden is used to categorizing ADCs. However, since it takes into account the linearity it was used here to categorize the buffer in terms of linearity. This FOM is calculated by

$$\frac{Power}{2^{\frac{SNDR-1.76}{6.02}} \cdot 2 \cdot BW} \tag{6.2}$$

to account for the power, bandwidth, and linearity of the ADCs. Notice that, in case of the Walden FOM, the lower the number the better the efficiency of the IB. Regarding its values, the proposed improved IB has the best FOM-Walden.

Finally, the same simulations were done with some variation at the supply voltage. These simulations serve the purpose of an overview of the variation of the buffer's working conditions. Table 6.2 represents the change of the supply voltage to less 10% of its original value.

Table 6.2: Comparison between tested designs analysis (@ 1.08 V supply voltage).

	SF	SSF	CSF	Current Feedback IB	Vgs-Controlled CSF IB	Proposed IB	Proposed Improved IB
Bandwidth (GHz)	1.5	1.7	1.6	1.0	1.1	2.2	2.1
IM3 (dB)	67	59	51	59	65	57	61
SFDR (dB)	55	58	49	53	57	50	59
ENOB (bits)	7.9	8.3	8.0	7.8	8.1	7.6	9.0

Overall all of the buffers suffered some degradation in their performance. This degradation was more significant in complex designs like the Current Feedback IB and both proposed architectures. Even though the improved buffer suffered the most degradation in its performance, it is still the best buffer overall.

Continuing checking the variation at the supply voltage, Table 6.3 presents the results with an increase of 10% of the supply voltage.

Table 6.3: Comparison between tested designs analysis (@ 1.32 V supply voltage).

	SF	SSF	CSF	Current Feedback IB	Vgs-Controlled CSF IB	Proposed IB	Proposed Improved IB
Bandwidth (GHz)	1.5	1.8	1.6	1.0	1.3	2.4	2.3
IM3 (dB)	67	67	66	56	72	68	65
SFDR (dB)	55	60	54	54	58	56	64
ENOB (bits)	8.0	8.4	8.0	7.9	8.2	8.0	9.3

With the increase of the supply voltage every buffer had some minor performance improvement. Note that the biggest change was Vgs-Controlled CSF IB IM3 improvement, however the SFDR on that buffer did not improve by the same rate, none the less this buffer was the one with an overall performance improvement with higher supply voltages.

6.2 Final conclusion of the work

In summary, the proposed improved design looks like a functional input buffer design for high linearity and high bandwidth purposes. It is important to note that the proposed improved buffer had such performance on this technology (130 nm) with the constraints labeled on table 3.1 on page 21. This means that in different technologies the proposed buffers can have some distinct performances. However, it is possible to predict that there should be some advantages of using more recent buffer designs in new technologies, mainly regarding bandwidth and die area.

One final note, most of the buffers used in the industry do not follow the guidelines of this work and usually are supplied with a voltage higher than the core voltage of the technology. The higher voltage improves the performance of the buffer, as seen comparing tables 6.1 and 6.3. However, this work was used to compare buffers with core devices. Using core devices improves the speed of the buffer and eliminates the need for higher voltages.

As a continuation of this work, further analysis of both proposed and proposed improved designs in deep nanoscale technologies to investigate if these designs scale better into newer FinFET and FD-SOI technologies.

Bibliography

[1] A. Bevilacqua and A. M. Niknejad. "An ultra-wideband CMOS LNA for 3.1 to 10.6 GHz wireless receivers." In: *2004 IEEE International Solid-State Circuits Conference (IEEE Cat. No.04CH37519)*. 2004, 382–533 Vol.1. DOI: 10.1109/ISSCC.2004.1332754.

[2] Bo-Yang Chang and C. F. Jou. "Design of a 3.1-10.6GHz low-voltage, low-power CMOS low-noise amplifier for ultra-wideband receivers." In: *2005 Asia-Pacific Microwave Conference Proceedings*. Vol. 2. 2005, 4 pp. DOI: 10.1109/APMC.2005.1606458.

[3] F. Bruccoleri, E. A. M. Klumperink, and B. Nauta. "Noise cancelling in wideband CMOS LNAs." In: *2002 IEEE International Solid-State Circuits Conference. Digest of Technical Papers (Cat. No.02CH37315)*. Vol. 1. 2002, 406–407 vol.1. DOI: 10.1109/ISSCC.2002.993104.

[4] J. Carrillo, M. Dominguez, F. Duque, and G. Torelli. "Low-voltage wide-swing fully differential CMOS voltage buffer." In: Aug. 2011, pp. 122–125. DOI: 10.1109/ECCTD.2011.6043292.

[5] R. G. Carvajal, J. Ramirez-Angulo, A. J. Lopez-Martin, A. Torralba, J. A. G. Galan, A. Carlosena, and F. M. Chavero. "The flipped voltage follower: a useful cell for low-voltage low-power circuit design." In: *IEEE Transactions on Circuits and Systems I: Regular Papers* 52.7 (2005), pp. 1276–1291. ISSN: 1558-0806. DOI: 10.1109/TCSI.2005.851387.

[6] P. Costa, C. Fiocchi, U. Gatti, and F. Maloberti. "High-performance BiCMOS output buffer design strategies." In: *1999 IEEE International Symposium on Circuits and Systems (ISCAS)*. Vol. 2. 1999, 168–171 vol.2. DOI: 10.1109/ISCAS.1999.780645.

[7] C. Erdmann, E. Cullen, D. Brouard, R. Pelliconi, B. Verbruggen, J. Mcgrath, D. Collins, M. De La Torre, P. Gay, P. Lynch, P. Lim, A. Collins, and B. Farley. "16.3 A 330mW 14b 6.8GS/s dual-mode RF DAC in 16nm FinFET achieving -70.8dBc ACPR in a 20MHz channel at 5.2GHz." In: *2017 IEEE International Solid-State Circuits Conference (ISSCC)*. 2017, pp. 280–281. DOI: 10.1109/ISSCC.2017.7870370.

[8] Fan You, S. H. K. Embabi, and E. Sanchez-Sinencio. "Low-voltage class AB buffers with quiescent current control." In: *IEEE Journal of Solid-State Circuits* 33.6 (1998), pp. 915–920. DOI: 10.1109/4.678659.

[9] F. A. Farag. "High performance CMOS buffer amplifier with offset cancellation." In: *2013 Saudi International Electronics, Communications and Photonics Conference*. 2013, pp. 1–4. DOI: 10.1109/SIECPC.2013.6551015.

[10] A. Glascott-Jones, N. Chantier, F. Bore, and M. Wingender. "Direct conversion techniques for radar systems." In: *2013 14th International Radar Symposium (IRS)*. Vol. 1. 2013, pp. 288–295.

[11] A. Glascott-Jones, N. Chantier, F. Bore, M. Wingender, O. Gentaz, and M. Torres. "High speed ADCs for direct conversion techniques in array radar systems." In: *2013 IEEE International Symposium on Phased Array Systems and Technology*. 2013, pp. 69–73. DOI: 10.1109/ARRAY.2013.6731801.

[12] P. R. Gray, R. G. Meyer, P. J. Hurst, and S. H. Lewis. *Analysis and Design of Analog Integrated Circuits*. 4th. USA: John Wiley & Sons, Inc., 2001. ISBN: 0471321680.

[13] Y. Ha, M. Li, and A. Q. Liu. "Low-voltage high driving capability CMOS buffer used in MEMS interface circuits." In: *ICECS'99. Proceedings of ICECS'99. 6th IEEE International Conference on Electronics, Circuits and Systems (Cat. No. 99EX357)*. Vol. 3. IEEE. 1999, pp. 1313–1316.

[14] B. Hershberg, D. Dermit, B. v. Liempd, E. Martens, N. Markulic, J. Lagos, and J. Craninckx. "3.1 A 3.2GS/s 10 ENOB 61mW Ringamp ADC in 16nm with Background Monitoring of Distortion." In: *2019 IEEE International Solid- State Circuits Conference - (ISSCC)*. 2019, pp. 58–60. DOI: 10.1109/ISSCC.2019.8662290.

[15] Hiok-Tiaq Ng, R. M. Ziazadeh, and D. J. Allstot. "A multistage amplifier technique with embedded frequency compensation." In: *IEEE Journal of Solid-State Circuits* 34.3 (1999), pp. 339–347. DOI: 10.1109/4.748185.

[16] I. Hwang, D. Kim, J. Roh, M. Lee, S. Nah, and M. Song. "An 8-b cascaded folding A/D converter with a new fully differential source follower." In: *2013 European Conference on Circuit Theory and Design (ECCTD)*. 2013, pp. 1–4.

[17] Joongsik Kih, Byungsoo Chang, and Deog-Kyoon Jeong. "Class-AB large-swing CMOS buffer amplifier with controlled bias current." In: *IEEE Journal of Solid-State Circuits* 28.12 (1993), pp. 1350–1353. DOI: 10.1109/4.262009.

[18] Y. Kong, S. Xu, and H. Yang. "An Ultra Low Output Resistance and Wide Swing Voltage Follower." In: *2007 International Conference on Communications, Circuits and Systems*. 2007, pp. 1007–1010. DOI: 10.1109/ICCCAS.2007.4348217.

[19] E. Ozalevli, M. Qureshi, and P. Hasler. "Low-voltage floating-gate CMOS buffer." In: June 2006, 4 pp. –1875. DOI: 10.1109/ISCAS.2006.1692974.

[20] J. Ramirez-Angulo, R. G. Carvajal, A. Torralba, J. Galan, A. P. Vega-Leal, and J. Tombs. "The flipped voltage follower: a useful cell for low-voltage low-power circuit design." In: *2002 IEEE International Symposium on Circuits and Systems. Proceedings (Cat. No.02CH37353)*. Vol. 3. 2002, pp. III–III. DOI: 10.1109/ISCAS.2002.1010299.

[21] H. Rapakko and J. Kostamovaara. "On the Performance and Use of an Improved Source–Follower Buffer." In: *IEEE Transactions on Circuits and Systems I: Regular Papers* 54.3 (2007), pp. 504–517. ISSN: 1558-0806. DOI: 10.1109/TCSI.2006.887609.

[22] B. Razavi. *Fundamentals Of Microelectronics*. 2nd. USA: John Wiley & Sons, Inc., 2014.

[23] A. Tanaka, H. Kodama, and A. Kasamatsu. "Low noise amplifier with center frequency hoping for an MB-OFDM UWB receiver." In: June 2004, pp. 420 –423. ISBN: 0-7803-8373-7. DOI: 10.1109/UWBST.2004.1321008.

[24] B. Vaz, A. Lynam, B. Verbruggen, A. Laraba, C. Mesadri, A. Boumaalif, J. Mcgrath, U. Kamath, R. De Le Torre, A. Manlapat, D. Breathnach, C. Erdmann, and B. Farley. "16.1 A 13b 4GS/s digitally assisted dynamic 3-stage asynchronous pipelined-SAR ADC." In: *2017 IEEE International Solid-State Circuits Conference (ISSCC)*. 2017, pp. 276–277. DOI: 10.1109/ISSCC.2017.7870368.

[25] Xianping Fan and P. K. Chan. "Analysis and design of low-distortion CMOS source followers." In: *IEEE Transactions on Circuits and Systems I: Regular Papers* 52.8 (2005), pp. 1489–1501. ISSN: 1558-0806. DOI: 10.1109/TCSI.2005.851711.

[26] G. Xing, S. H. Lewis, and T. R. Viswanathan. "A Unity-Gain Buffer with Reduced Offset and Gain Error." In: *IEEE Custom Integrated Circuits Conference 2006*. 2006, pp. 825–828. DOI: 10.1109/CICC.2006.320831.

[27] A. Zhuk, A. Bugakova, E. Ovsepyan, and N. Dmitriyenko. "The Complementary Buffer Amplifiers with Low Static Current Consumption for Low-Voltage Analog Microcircuits." In: June 2018, pp. 86–89. DOI: 10.1109/EDM.2018.8435041.