Robust to PVT, High DC Gain Amplifier for CT $\Sigma\Delta$ Modulators on SOI CMOS Technology

by

Héctor Iván Gómez Ortiz

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

MASTER ON SCIENCE WITH MAJOR ON ELECTRONICS

from

National Institute for Astrophysics, Optics and Electronics
October 2011
Tonantzintla, Puebla

Supervised by:

Ph.D. Guillermo Espinosa Flores-Verdad, INAOE

“ To my parents Arleth and Luis Héctor
To my sister Mabel and my brother Diego
To Jennifer ”

Héctor.

Acknowledgment

First of all to God for all his blessings.

I want to thank my parents Arleth and Luis Héctor and my sister Mabel and my brother Diego for all their understanding and support. Without them, nothing that so far I've managed, it had been possible.

I would like to thank my girlfriend Jennifer because despite the distance always she knew giving me her love and support.

I would like to thank my advisor Ph.D. Guillermo Espinosa Flores-Verdad for his useful advices and support.

I would like to thank Juan Carlos, Ricardo, Luz Karine, Jhoan and Fabian for their friendship and their collaboration in some way with development of this project.

I want to thank CONACyT for my scholarship.

In general, I want to thank INAOE.

ABSTRACT

TITLE: Robust to PVT High DC gain Amplifier for CT $\Sigma\Delta$ Modulators on SOI CMOS Technology [1].

AUTHORS: HÉCTOR IVÁN GÓMEZ ORTIZ

KEY WORDS: SOI, PVT, OTA, buffer, $\Sigma\Delta$ modulator, continuous time.

DESCRIPTION: SOI technology is particularly good for low voltage, low power and high speed digital systems because it has reduced drain to substrate capacitance due to the inclusion of an insulator layer between substrate and active region. Nonetheless, SOI is a pure digital technology making the analog design a challenging task. This work present the design of a relatively high DC gain amplifier on SOI technology, which must be robust to PVT variations. In order to evaluate the performance of amplifier, it is used in a continuous time sigma delta modulator.

Two OTAs was proposed in this work. As a first consideration, self-cascode transistor was used to improve the output impedance and reduce the influence of voltage supply in the amplifier gain. Also, a current shunt technique was applied to first proposed amplifier to enhance DC gain. However, the PVT variations considerably affects the performance of amplifier, so a second proposal was accomplished. Sums of transconductance was used together self-cascode transistor to reduce sensitivity to PVT variations. The last proposed amplifier achieves a DC gain of 53 dB with GBW=575 MHz and a power consumption of 1.25 mW. Simulation results shows that the amplifier has a DC gain over 50 dB despite PVT variations.

To drive resistive load of the first integrator of the modulator, different buffer was proposed in order to have wide input and output voltage excursion and low output impedance. Final Buffer implementation is a feedback amplifier which has complementary input differential pairs and a class AB output stage. Open loop gain of the amplifier is 57 dB while has close loop output impedance of 55 Ω. Buffer's harmonic distortion is around 0.1 % for all PVT variations with a voltage excursion of 600 mV.

A comparison between two-stage Miller amplifier and proposed amplifier was done to see how PVT variations affect the performance of a commonly used amplifier. Finally, both amplifiers are included in the modulator to validate the robustness of the proposed amplifier. Simulations of the whole modulator show that the SNDR of modulator is improved over 9 dB using the proposed amplifier.

The design was done on SOI 45 nm IBM technology and simulations were carried out in Hspice.

[1] Master thesis.

RESUMEN

TÍTULO: Amplificador robusto a *PVT* con alta ganancia en *DC* para moduladores $\Sigma\Delta$ en tiempo continuo en tecnología *SOI* [2]

AUTOR: HÉCTOR IVÁN GÓMEZ ORTIZ

PALABRAS CLAVES:*SOI*, *PVT*, *OTA*, *Buffer*, modulador $\Sigma\Delta$, tiempo continuo.

DESCRIPCIÓN: La tecnología *SOI* es particularmente buena para bajo voltaje, baja potencia y sistemas digitales de alta velocidad, por que tiene una capacitancia de drenaje a substrato reducida debido a la inclusión de una capa de aislante entre el substrato y la región activa. Sin embargo, *SOI* es una tecnología puramente digital lo que hace de el diseño analógico un tarea desafiante. En este trabajo se presenta el diseño de un amplificador con relativa alta ganancia en *DC* en tecnología *SOI*, él cual debe ser robusto a variaciones de *PVT*. Con el fin de evaluar el desempeño del amplificador, este se usa en un modulador sigma delta en tiempo continuo.

En este trabajo son propuestos dos *OTAs*. Como primera consideración, se usan transistores compuestos para aumentar la impedancia de salida del amplificador y reducir la influencia del voltaje de alimentación en la ganancia del amplificador. Además, se usa una técnica de derivación de corriente en el el primer amplificador propuesto para aumentar la ganancia de *DC*. Sin embargo, las variaciones de *PVT* afectan considerablemente el desempeño del amplificador por lo cual se propone un segundo amplificador. Se usa una suma de transconductancias junto con transistores compuestos para reducir las efectos de las variaciones *PVT*. El último amplificador propuesto logra una ganancia en *DC* de 53 dB con *GBW*=575 MHz y un consumo de potencia de 1.25mW. Los resultados de simulación muestran que el amplificador mantiene una ganancia en *DC* de mas de 50 dB a pesar de las variaciones de *PVT*.

Para manejar la carga resistiva del primer integrador del modulador, se propusieron diferentes *buffers* con el fin de obtener amplios rangos de excursión de voltaje a la entrada y a la salida y una baja impedancia de salida. La implementación final del *buffer* es una amplificador realimentado el cual tiene pares diferenciales complementarios a la entrada y una etapa de salida clase AB. La ganancia en lazo abierto del amplificador es 57 dB mientras tiene una impedancia de salida en lazo cerrado de 55 Ω. La distorsión harmónica del *buffer* es alrededor de 0.1 % para todas las variaciones de *PVT* con una excursión de voltaje de 600 mV.

Se hace una comparación entre un amplificador Miller de dos etapas y el amplificador propuesto para ver como las variaciones de *PVT* afectan el desempeño de un amplificador comúnmente usado. Finalmente, ambos amplificadores se utilizan en un modulador para validar la robustez del amplifi-

[2] Tesis de maestría.

cador propuesto. Las simulaciones del modulador muestran que si mejora la *SNDR* del modulador en mas de 9 dB usando el amplificador propuesto.

El diseño fue hecho en una tecnología *SOI* de 45 nm de IBM y las simulaciones se realizaron en Hspice.

Contents

List of Figures

List of Tables

List of Abbreviations

ΣΔ Sigma Delta

ADC Analog to Digital Converter

CMFB Common Mode Feedback

CT Continuous Time

DAC Digital to Analog Converter

DR Dynamic Range

DT Discrete Time

ELD Excess Loop Delay

ENOB Effective Number of Bits

GBW Gain Bandwidth

IBN In Band Noise

IM3 Third Order Intermodulation Distortion

OTA Operational Transconductance Amplifier

PVT Process Temperature and Voltage

SCE Short Channel Effect

SFDR Spurious Free Dynamic Range

SNDR Signal to Noise and Distortion Ratio

SNR Signal to Noise Ratio

SOI Silicon On Insulator

SQNR Signal to Quantization Noise Ratio

THD Total Harmonic Distortion

Chapter 1

Introduction

The technology scaling has originated a contrast on integrated circuit design. While the digital circuits have a performance improvement related to their higher speed operation and power consumption saving, the analog circuits show some negative effects such as reduced dynamic range, higher noise and loss in the intrinsic gain of transistors. As a consequence, there has been a trend of moving more functionality into the digital domain.

Although much of the processing is done in the digital domain, there is still the need of an element that provides a link between digital and analog domain: the analog-to-digital converter (ADC). Its design has been investigated in the last years motivated by the rapid growth of the market for portable systems and the integration on-chip of the analog-to-digital interface. However, the required specifications in different applications make the ADC design a challenging task because it is necessary to achieve high linearity, dynamic range and bandwidth capabilities, which conflicts with the low-power requirements for long stand-by time [1].

There exist a lot of possibilities to implement analog to digital (A/D) conversion depending on the resolution or the speed of data conversion. One favorable option, especially in very large scale integration (VLSI) systems, is the sigma–delta ($\Sigma\Delta$) ADC which shows a very low sensitivity to the non-idealities of its building blocks, besides simplifying analog functions like pre-filtering [2]. It is common to find two kind of implementations of a $\Sigma\Delta$ (modulator) ADC: discrete-time (DT) and continuous-time (CT). The switched capacitor (SC) technique made very popular the DT $\Sigma\Delta$ topology because of the similarity between its mathematical modelling and its physical implementation, but its restriction in the input frequency motivated the use of continuous-time implementations [3].

The advantages of the CT $\Sigma\Delta$ ADC are all based on the displacement of the sampler inside the modulator loop allowing the filters to reduce their speed requirements by means of a continuous-time implementation. Additionally, this condition provides an implicit anti-aliasing filter (AAF) and avoids the need of an accurate input sample and hold circuit [3]. Consequently, the CT architectures show a better power efficiency and the ability to make very high sampling rate $\Sigma\Delta$ modulators.

Despite having the advantages above mentioned, there are two main reasons why the CT modulators are not very commonly implemented. First, the advances in DT modulator implementation have made possible converters with high-resolution and high-bandwidth using low oversampling ratios. Second, and more important, the continuous-time modulators are affected by circuit non-idealities such as variations of the gain and bandwidth integrator (which are process dependent) and timing errors [3], making their practical implementation not straightforward.

The integrators can be implemented by using transconductors or operational amplifiers (OpAmps). The former implementation is made in open loop with a load capacitance, but this fact makes the linearity a critical design consideration. The later is made in close loop which improves the linearity, however, the mismatch in RC constants considerably affects its performance. They both are not only impacted by physical factors, which during manufacture result in permanent variations of the interconnections and devices, but also environmental factors, among them the variations in power supply, switching activity, and temperature of the chip or across the chip [4]. These changes in the fabrication process and typical conditions of operation are called PVT (process, voltage and temperature) variations which largely determine the specifications that can be achieved by the modulator.

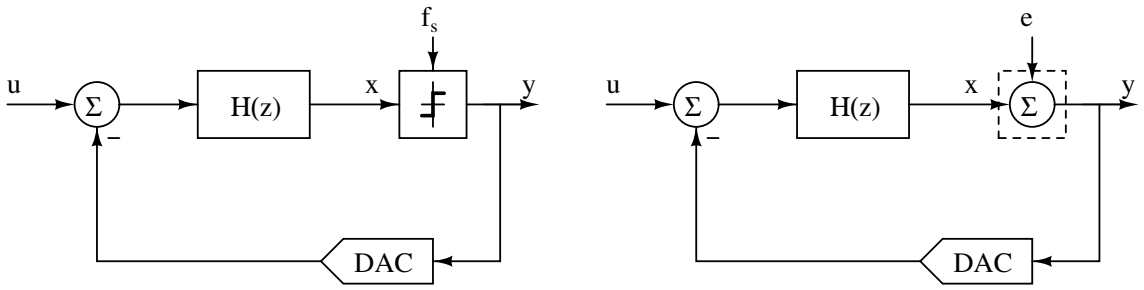
This scenario motivates us to propose an operational amplifier that minimizes the effects of PVT variations and overcomes the problem of the loss in the intrinsic gain of transistors, in order to make robust CT $\Sigma\Delta$ implementations. The design and validation are made on SOI CMOS 45 nm technology.

1.1 Basic theory of $\Sigma\Delta$ modulation

In a conventional A/D conversion, the analog input is uniformly sampled in time and its amplitude is quantized in discrete levels. Due to the finite number of quantization levels, the quantization process causes errors, which set the maximum achievable resolution [1]. Therefore, the resolution can be improved by increasing the number of quantization levels. To achieve higher accuracy, in a $\Sigma\Delta$ modulator additional techniques are used: oversampling and noise-shaping. The former is related to the *Nyquist theorem*, which states that the minimum sampling frequency, for a perfect reconstruction of sampled signal, is twice the signal bandwidth; thus, oversampling means that the sampling of the analog input signal is done with frequency higher than the *Nyquist Frequency* [3]. Accordingly, the oversampling ratio (OSR) is defined as [1]

$$m = \frac{f_s}{2 * f_b} \tag{1.1}$$

where m is the oversampling ratio, f_s is the sampling rate and f_b is the signal bandwidth. Noise-shaping implies filtering the quantization errors, in order to shape their frequency response. As a



(a) Real model. (b) Linearized model.

Figure 1.1: ΣΔ DT model.

result, the analog input signal is modulated into a digital word sequence whose spectrum approximates that of the analog input well in a narrow (desired) frequency range, but which is otherwise noisy. As a consequence, high resolution can be obtained in a relatively small bandwidth [1].

Despite the focus of this work is the CT ΣΔ modulation, the basic theory can be easily explained in DT as follows. The general model of a single-loop ΣΔ modulator is shown in figure 1.1(a) [5]. Basically, a ΣΔ modulator consists of a loop filter, performing the noise-shaping; a low resolution clocked quantizer, which is oversampled; and a feedback digital-to-analog converter (DAC)[1]. The first-order low pass filter (which also can be a bandpass filter) is an accumulator in the discrete-time domain or an integrator in the continuous-time domain. The quantizer, which is characterized by its strong nonlinearity, makes the ΣΔ analysis to be complicated; however, the quantizer can be replaced by a noise additive model (Figure 1.1(b)) where the quantization error e is independent of the circuit input u. The output y may now be written as [5]

$$Y(z) = \frac{H(z)}{1+H(z)}U(z) + \frac{1}{1+H(z)}E(z) \tag{1.2}$$

$$= STF(z)U(z) + NTF(z)E(z) \tag{1.3}$$

where $STF(z)$ and $NTF(z)$ are the so-called signal transfer function and noise transfer function. From (1.2) we see that the poles of $H(z)$ become the zeros of $NTF(z)$, and that for any frequency where $H(z) >> 1$ [5],

$$Y(z) \approx U(z). \tag{1.4}$$

[1] Here, the DAC is modeled by a unity gain

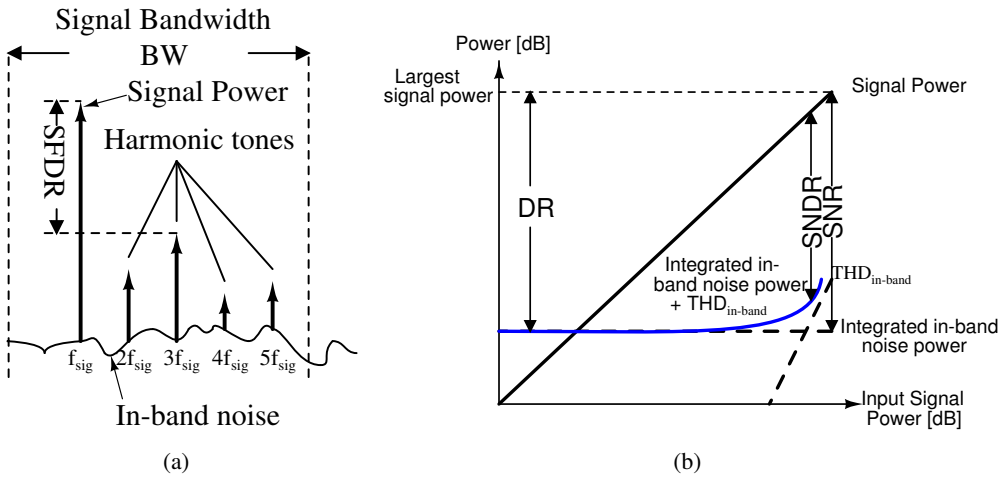
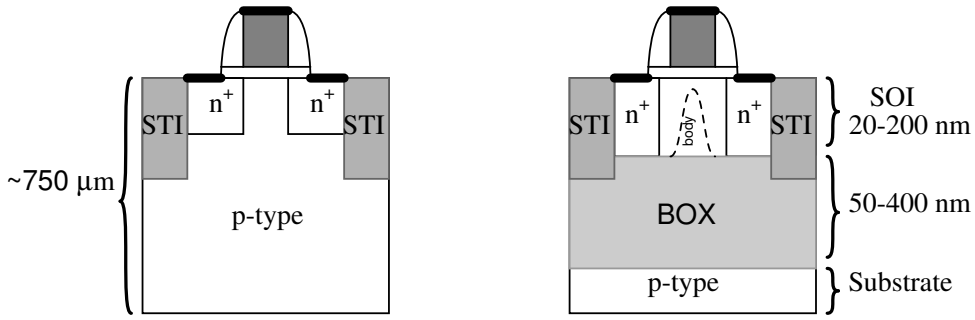


Figure 1.2: Critical performance of an ADC (a) output frequency spectrum (b) output power component v.s. input signal power. [6].

In other words, the input and output spectra are in greatest agreement at frequencies where the gain of $H(z)$ is large.

1.1.1 Performance metrics

Figure 1.2(a) illustrates the general output power spectrum of an ADC. The output of the ADC is composed by the signal, noise, and harmonic distortions which defines its performance. Different kind of noise contribute to the overall noise performance such as thermal noise from transistors or resistors, quantization noise from quantizer and jitter noise from sampling clock. The relation between power of those components and input signal power is depicted in Figure 1.2(b). While the integrated in-band noise power is fixed, the total in-band harmonic distortion ($\mathrm{T_HD}$ in-band) increases when input signal power grows. Signal-to-noise-ratio (SNR) is defined as the ratio between signal power and integrated in-band noise power while Signal-to-quantization-noise-ratio (SQNR) is defined if only quantization noise is considered. The harmonic tones should be excluded when computing SNR, thus, signal-to-noise-and-distortion-ratio (SNDR) includes this spurious tones by summing $\mathrm{T_HD}$ in-band and in-band noise. SFDR is the spurious-free dynamic range which is the power difference between signal and the largest harmonic tone or intermodulation tone as shown in 1.2(a); dynamic range (DR) means the power range of the input signal where the upper limit and lower limit are the largest allowable power that would not saturate the system and the power level equal to the integrated in-band noise power, respectively, as shown in Figure 1.2(b). IM3 is the power difference between signal and third intermodulation tones when employing two-tone test. Overall, SNDR and DR are the most critical indicators showing all the performance related to non-idealties of an ADC and the working range of the signal power [6].



(a) Bulk CMOS. (b) PD-SOI.

Figure 1.3: Cross-section of bulk and SOI MOS devices.

1.2 Silicon-On-Insulator technology

The trend of reducing power consumption in LSI systems has specially grown in last years, but, it has been slower than expected because the shrinkage of devices in bulk CMOS technology has had fundamental limitations due to decrease in carrier mobility and increase of tunneling gate and p-n junction leakage. As a consequence, the threshold voltage can not be enough reduced and to achieve a desired speed, the operating voltage must be set higher than what a scaled-down device was expected. These physical constraints make conventional scaling less and less feasible and has encouraged the industry to look for others alternatives. That is how silicon-on-insulator (SOI) technology has become an interesting alternative because it is particularly good for low-voltage, low power and high speed digital systems [7]. This technology has also proved to be effective in various niche and growing markets. Accordingly, SOI process has another advantages such as the reduced susceptibility to radiation, high temperature and high voltage operation, and it also enables the fabrications of micro-electro-mechanical systems (MEMS) for control systems.

In the Figure 1.3, bulk CMOS (a) and SOI CMOS (b) transistors can be seen. The main difference between them is the buried oxide (BOX) layer, that is why SOI devices have lower capacitance to substrate than comparable devices on bulk silicon. The BOX is a layer of silicon dioxide just below the surface or top Si layer, which is known as the SOI layer. Finally, the Si substrate beneath the BOX is called just Si substrate [8]. Depending on the thicknesses of SOI and BOX layers there are different applications and different kinds of devices. Thus, there exist MEMS sensors, power/high voltage devices, BiCMOS, partially depleted (PD) SOI CMOS and fully depleted (FD) SOI CMOS. The two last have the most thin layers. The centre of interest will be the PD-SOI CMOS because it has advantages such as manufacturability, reduced short channel effects (SCE) and multiple threshold voltages. Additionally, PD-SOI may be thought of as evolutionary rather than revolutionary device structure because it has targeting parameters which are similar to bulk CMOS.

The PD-SOI has particular characteristics that influence in the devices operation. The SOI body, which refers to the part of SOI layer that constitutes the body of MOSFET, is called floating body (FB) because when the depletion region is formed, it does not reach the bottom part of top Si layer. FB originates different phenomenons that should be studied in order to have a good understanding of technology. Below, these effects are reviewed along with others related to process.

1.2.1 SOI MOSFET's junction diode

PD-SOI MOSFET has two junction diodes identical in concept to those in the bulk device: the body-source diode is often weakly reverse biased and the drain-body diode is usually reverse biased. The body-source diode is a determinant mechanism in moving charge out of isolated body due to its biasing conditions and it is known as diode action. Nonetheless, there are other effects from the SOI diodes which are junction capacitances and junction leakages.

The junction capacitances are dependent on biasing conditions and they can be an important influence on SOI MOSFET behavior because in some cases, they have the ability to couple changes in gate, source, or drain potential into the body's voltage instantaneously. Junction leakage arises form three major mechanisms which are the electron/hole recombination in the space charge region, defects/impurities in the space charge region which disrupt the diode doping gradient from n-type to p-type and high energy carriers which exceed the diode's electronic barrier height. In bulk technologies, the connection of substrate or N-well to supply rail and the source or drain node driven by preceding circuitry on a second terminal, uphold the junction leakage. However, in the PD-SOI devices, junction leakage directly affects body potential due to its floating nature and thus it affects the performance, over an extended period. The junction current are widely influenced by the operation conditions due to its dependence on voltage and temperature, as represented simplistically in the classic diode equation.

1.2.2 Impact Ionization

SOI technology presents the problem of impact ionization and it continues to cause long-term wear out, but also has more immediate effect on the device's performance. Majority carriers have the ability to induce damage in the host FET structure when they are excited by high electric fields. The damage is accumulative and it modifies the threshold voltage of NFET and PFET devices. As consequence, the drain current is reduced in NFETs and increased in PFETs, which affects the performance and can eventually cause failures.

The are three mechanisms which induce degradation by hot carrier: conducting hot-carrier, non-conducting hot-carrier and substrate hot-carrier. In SOI, the first two are predominant. Conducting hot-carrier degradation occurs as the device is turned on. At pinch-off voltage, the electric field across the excessively short uninverted remainder of channel is then extremely high and essentially 'heats'

the electrons to an effective high temperature. The kinetic energy originated by the above mentioned makes some electrons in the case of NFETs not to be collected by the drain and the carriers energy will be injected into the insulator. In addition to the damage at silicon interface, impact ionization of silicon lattice atoms leaves behind positive charge, which accumulates in the isolated body of the device.

The nonconducting hot carrier degradation occurs in a similar way but when the gate voltage is less than V_T. Here, a portion of the inverted carriers present in subthreshold or punchtroungh current has sufficient energy to produce damage in the FET gate insulator oxide. The substrate hot electron effects are absent for the most part in SOI. The first two modes mentioned originated a charge accumulation in the body region which is significant in SOI due to FB.

1.2.3 Floating body effects

History effect and threshold voltage variability

The electrical property that largely determines the behaviour of a PD-SOI device is history effect. This makes the I-V characteristic not to be constant because it depends on charge of the body. This charge and its distribution caused by gate, source and drain potentials widely affect the device performance. Charge in the body is directly related to the potential of body. Due to the body bias the junctions are reverse-biased which must be overcome by gate drives. It can be seen as a modification in the threshold voltage. The changes in charge contained in the body are associated to a number of factors among them the previous state of transistor, the schematic position of transistor, slewing of the input and the load capacitance, channel length and PVT variations, junction temperature and operating frequency, and specific switch factor.

There are different paths to convey charge into and out of the body. Three paths move charges into the body which are usually slow and they are related to impact ionization and the junction leakage across the drain-body or source-body diode. However, there are only two paths that move charges out of the body that can be fast, and they are related to forward-biases of junction diodes. These transport mechanisms of charge affect the dynamic behaviour of the device and they can originate an effect named kink which consist in the increment of I_{DS} by decreasing of threshold voltage. It occurs when suddenly, a considerable amount of positive charge is injected into the body.

Short Channel effects

In Figure 1.4 the dependence of threshold voltage (V_T) roll-off with the channel length on bulk-CMOS is shown, which is known as short channel effect (SCE) for very short channel devices [9]. It has been observed a decrease in threshold voltage roll-off with elevated body voltage which is achieved with FB. Therefore, the SCEs are reduced or disappear in PD-SOI. Figure 1.4 superimposes the threshold

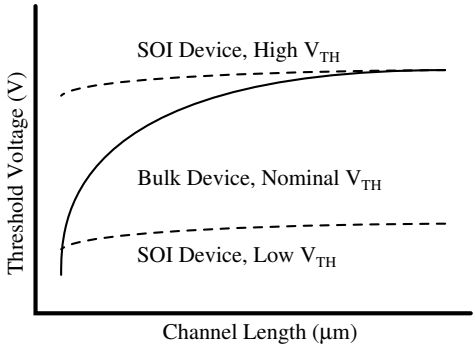


Figure 1.4: Roll-off of threshold voltage with L_{eff} for bulk and SOI devices [9].

voltage dependence on channel length for the SOI MOSFET for high and low threshold options. This feature increases scalability and reduces variability of performance across the process window of L_{eff}.

For short-channel devices, Drain Induced Barrier Lowering (DIBL) is a prominent effect in MOSFET which changes with the introduction of FB. While in bulk-CMOS, due to drain voltage, the effective channel length is reduced by the expansion of space-charge depletion region in drain-to -body junction, in PD-SOI device with FB, the channel length remains long because the body bias makes shrink the space region surrounding the drain. However, DIBL becomes artificially high due to strong influence of body in the threshold voltage which is reduced by the injection of positive charge into body through junction diodes. Depending on process, an SOI device with its body contact connected to a stable potential will revert to the bulk-CMOS DIBL values.

Bipolar Device Action

In bulk CMOS effects like latch-up arise from the presence of parasitic bipolar junction transistor (BJT). Devices in bulk-CMOS or PD-SOI form a bipolar structure where the drain acts as the BJT collector, the body as the BJT base and the source as the BJT emitter. However, an SOI device with FB merely makes the presence of the bipolar parasitics substantially more prominent. It occurs because the body can rise sufficiently high in voltage with respect to the source or drain to forward bias the respective junction diodes, permitting bipolar gain. If the body has been highly charged and the source of device is suddenly drawn low during a switch, the body-source junction diode can become instantaneously forward biased, and will conduct a current I_B momentarily from body (base) to source (emitter) until the charge in the body is enough discharged. At the same time, bipolar action cause a current of $\beta \cdot I_B$ to briefly pass from drain (collector) to source (emitter). Generally, the value of β is low, but depending on the value of V_{DD}, this bipolar gain can be higher than 1. It is important to maintain low the bipolar current because it can discharge a given node which originates logical errors in dynamic and pass-gate circuits.

1.3 State of the art for CT ΣΔ Modulators.

Despite the fact that CT ΣΔ modulators are strongly affected by circuit's non-idealities, the literature shows that many of these issues are not being taken into account in the design, especially in regard to PVT variations. However, the effects of non-idealities, such as excess loop delay (ELD) and jitter noise, are widely discussed in many papers. ELD is compensated by including additional feedback paths [10–13] while the influence of jitter noise in performance is dealt with by using non-return-to-zero (NRZ) feedback DAC [10, 13] waveform or implementing multi-bit DACs [11, 14]. On the other hand, calibration methods [15–18] are used in order to compensate or mitigate problems among them mismatch in time constants of the active-RC integrators, changes in gain-bandwidth (GBW) and DC gain of amplifiers, but these methods increase area and power consumption. PVT variations are considered in some papers [19–21], but only [19, 21] show how the performance of modulators is affected. In [19] the influence of power supply variations is shown while in [21] the temperature is also considered and a bulk-driven amplifier is proposed in order to operate with low power supply. Nevertheless, process variations are only taking into account in [20][2].

$$FOM = \frac{Power}{2 \cdot BW \cdot 2^{ENOB}} \tag{1.5}$$

$$ENOB = \frac{SNDR - 1.76}{6.02} \tag{1.6}$$

Ref. year.	f_s [MHz]	Power [mW]	SNDR [dB]	V_{DD} [V]	BW [MHz]	Technology CMOS [nm]	FOM [pJ/conv.]
[14] 2004	300	70	63.7	1.5	15	130	1.86
[10] 2005	1000	10	63.4	1.1	8	90	0.517
[11] 2006	640	20	74	1.2	20	130	0.122
[19] 2008	950	40	72-67	1.2	10-20	130	0.546[a]
[17] 2008	256	50	70	1.3	8	65	1.21
[15] 2008	12	1.2	76.5	1.1	0.02	45	5.49
[16] 2008	640	100	82	1.8	10	180	0.485
[18] 2009	100	13.7	60	1.8	1	180	8.4
[12] 2010	300	5.32	62.5	1.1	10	110	0.24
[13] 2010	320	36	74	1.8	10	180	0.235
[21] 2010	6.4	0.35	98	0.5	0.025	130	0.107
[20] 2010	400	44	67.7	1.8	25	180	0.444

Table 1.1: Literature survey of the reported ΣΔ ADCs.

[a] Calculated with 20 MHz of bandwidth.

[2] The performance of amplifiers is evaluated with regard to process and temperature variations.

In the Table 1.1 are listed the LP $\Sigma\Delta$ ADCs reported in last years. The main trend observed is higher frequency operation with low OSR, which implies the use of multibit ADCs and DACs to improve the power efficiency. However, there are works as [21] that has a low bandwidth with a good figure of merit (FOM)(1.5) [3]. The better FOM are achieved in pre or post-layouts simulations [12, 13, 21] and [11] shows the best FOM reported with measurement results.

1.4 Dissertation Organization

This document is oriented to design a high DC amplifier on SOI CMOS technology that must be robust to PVT variations in order to verify its performance in a CT $\Sigma\Delta$ modulator. This development is done on this document by identifying how the modulator behaviour is affected by circuit's non-idealities and evaluating how to overcome this problems. To fully understand the procedure, below is shown the organization of this document.

In chapter 2, a review of the modulator's theory is done. Also, the effects of circuit's non-idealities is presented and how they can be included in the model of the modulator. In chapter 3, the design of proposed circuits is performed, including the explanation of how to increase the gain of amplifiers and finally, a comparison with a conventional amplifier. Chapter 4 shows simulations results of the CT $\Sigma\Delta$ modulator using the proposed circuits. A brief comparison is done to see how the modulator is affected by PVT variations in the used amplifiers. Finally, some conclusions and the future work are drawn in chapter 5.

[3] ENOB refers to Effective Number of Bits

Chapter 2

Theory and modeling of CT $\Sigma\Delta$ modulators

2.1 Issues and Operation of the Modulator

In previous chapter, the basic theory of $\Sigma\Delta$ modulation in discrete time was explained; however, the implementation of a modulator can be made in continuous time too. Figure 2.1 [3] shows an illustrative schematic of a CT $\Sigma\Delta$ converter where the input signal can be subject to anti-aliasing filtering or not. The loop filter $H(s)$ now consists of CT filters, which can be active RC-filters using OpAmps, operational transconductance amplifiers (OTA), gmC-filters or even LC-resonator structures. The internal quantizer is a sampled or latched circuit, which is clocked at the sampling frequency f_s of the modulator. Below, the operation of these building blocks will be explained showing their differences with the blocks that are used by its counterpart in DT.

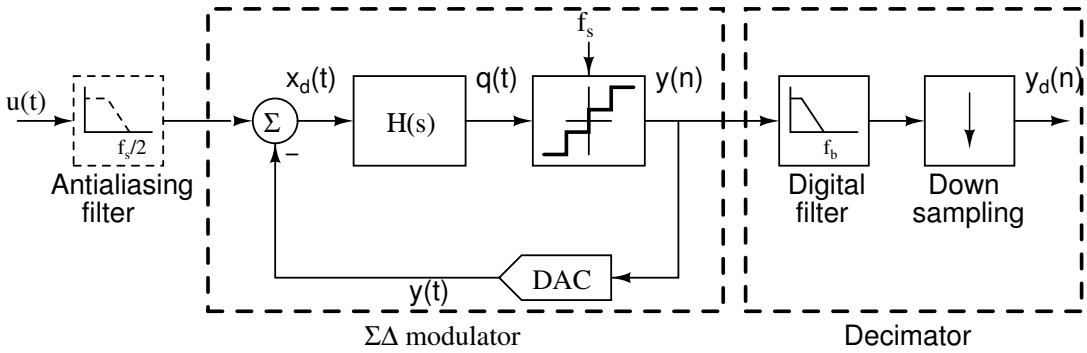


Figure 2.1: Block diagram of a CT $\Sigma\Delta$ A/D converter [3].

2.1.1 Sampling Operation

As the S/H circuit is placed at the input of the converter in DT modulators, every error of this block is added to the input signal. In contrast, the sampling operation in CT modulators takes place inside the ΣΔ loop, therefore, all non-idealities of the sampling process are subject to noise-shaping. It is an interesting feature because S/H operation at high f_s requires demanding specifications difficult to achieve.

The implementation of the sampling operation inside modulator loop, after the loop filter, results to some extent in implicit anti-aliasing filtering. This attribute can heavily reduce the required specifications of a front-end AAF and even makes it sometimes unnecessary. Mainly in high-speed circuits or architectures with very low oversampling ratios, this can be an strong argument to choose a CT ΣΔ implementation.

Timing errors of the sampling clock, such as jitter, have a significant impact on the DT modulator sampling operation. The S/H circuit can sample the input at a wrong time instant, which produces an equivalent amplitude error that degrades the SNR of the converter. Thus, the front-end S/H sets an upper limit on the performance of the entire DT modulator. On the other hand, the noise-shaped S/H errors in CT modulators make the sampling operation to be less sensitive to jittered clock. Nonetheless, the CT modulator are very sensitive to timing variations or delays within the feedback path. This is not an issue in DT modulators because the filters in SC circuits are usually designed as to safely settle within half sampling period.

Implicit Anti-aliasing Filter

This feature is easily understood if the first-order modulator of figure 2.1 is considered ($H(s) = I(s)$) without additional AAF. The input to the quantizer at a sampling instant depends on the feedback signal and additionally on an integrated version of the input signal $\int_{nT_s}^{(n+1)T_s} u(t)dt$ [3]. This integration can be expressed by a convolution of the input signal with a rectangular window which is a multiplication in frequency domain between the input spectrum and a $sinc(\pi f/f_s)$-function. This $sinc$-function attenuates the signal spectrum exactly at multiples of the sampling frequency and therefore shows the function of an AAF. Thus, it can be expected that higher order modulation and larger loop-gain results in higher order anti-aliasing property.

The figure 2.2 shows the AAF of the input signal in a CT ΣΔ modulator. From this equivalent model, the anti-aliasing filter can be approximately given by

$$F_{AAF}(w) = \frac{FF(s = jw)}{FF(z = e^{jwT_s})} \tag{2.1}$$

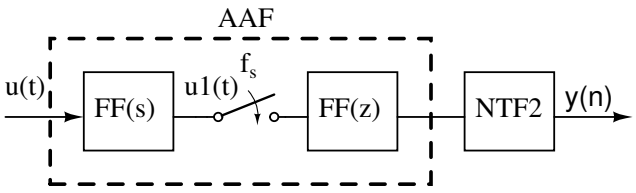


Figure 2.2: Equivalent representation of a CT $\Sigma\Delta$ modulator considering its implicit AAF [3].

where $FF(s)$ is the feed-forward filter of CT modulator and $FF(z)$ its DT equivalent. From (2.1), it can be seen that the AAF-behavior for different loop filter characteristics such as low-pass, band-pass, all poles at DC or optimally spread inside the in-band is preserved.

2.1.2 Filter Realization

The loop filters in DT and CT modulators are integrators or resonators which are commonly implemented with SC circuits and continuous-time integrators respectively. The signals in DT circuits are quickly changing, therefore, the modulators employing the SC technique have a maximum clock rate, which is limited by the OTA bandwidth and the need of several time constants to settle the charge transfer with given accuracy. In contrast, in CT modulators all signals are continuous-time waveforms; thus the OpAmp speed restrictions are drastically relaxed, and CT modulators can be clocked at higher frequencies that actually tend to lie between three and five times their DT counterparts.

Linearity behaviour concerning the virtual ground node is a matter of both DT and CT architectures. Despite the fact that DT modulators have large glitches in this node due to the fast SC pulses, its effects in linearity can be neglected as the signal of interest in DT is the finally settled value. In the case of CT modulators, the virtual ground node can be kept quiet; characteristic that must be permanently kept because the integration of a continuously changing waveform requires permanent linearity . Another issue is the voltage dependency (nonlinearity) of the passive and active components used. This imperfection leads to harmonic distortion at the modulator output, therefore, limiting the performance of CT $\Sigma\Delta$ modulators due to nonlinearity of resistor and transconductances stages. In particular, the first transconductor must have the overall accuracy of the system.

The transfer function of the loop filter is heavily affected by integrators implementation, *i. e.*, the utilization of discrete or continuous-time circuits. The former implementation defines its gain by capacitors ratio which is very precise since absolute mismatch of both capacitors does not affect the integrator gain. In contrast, CT circuits define the integrator gain by a RC or g_mC product; they both are subject to large process dependent variations, originating strongly loss of performance or even unstable system in worts cases.

2.1.3 Quantizer Implementation

A common characteristic in both DT and CT ΣΔ modulators is that all non-idealities of the quantization process are suppressed by noise shaping. However, the decision time of the internal quantizer and its signal dependent variation (metastability) have a significantly different influence: while the decision time in DT systems is subject to half of sampling period, a CT ΣΔ ideally needs infinitely fast quantization because the result is needed instantaneously to generate the continuous-time feedback signal. Thus, severe performance limitations can arise.

Generally, low resolution flash ADCs are used in CT ΣΔ modulators due to its high speed. Nonetheless, the exponentially increasing number of comparators, if a higher resolution is desired , essentially increases power and area consumption. Additionally, the capacitive load seen by the last integrator in the loop filter increases; therefore, the drive capability is pretty demanded by this load, increasing even more the power consumption.

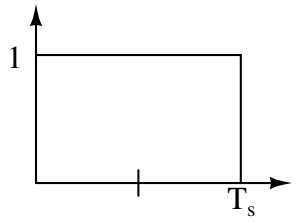
In order to relax the requirements imposed by flash ADCs, other conversion methods have been adopted , such as successive approximation and tracking as internal quantizers. Successive approximation quantizers offer significant reduction in the number of comparators to only one; however, this conversion method requires higher sampling frequencies for wide bandwidths which increases the power consumption. On the other hand, the tracking quantizer approach takes advantage of the fact that in a multi-bit ΣΔ modulator the feedback signal is usually changing only by one LSB at every sampling instant. Thus, full quantization range of the internal quantizer is not needed, but only the decision at a certain interval within the full-scale range. But it is possible that two LSB steps occur which makes necessary the use of more than one comparator. Despite improving the stability of the modulator, it still has to be ensured that the input signal to the tracking ADC within the modulator loop is not slewing too fast. Otherwise, the ADC loses the input signal and again the modulator becomes unstable. Consequently, both techniques impose their own advantages and drawbacks.

2.1.4 Feedback Realization

In DT modulators, the feedback signal is applied by charging a capacitor to a reference voltage and discharging it onto the integrating capacitance. In contrast, the analog continuous-time feedback waveform is integrated over time and thus the ΣΔ modulator is sensitive to every deviation from the ideal waveform of the feedback signal, originating some severe non-idealities.

The feedback DAC waveforms more commonly used are shown in Figure 2.3. Their classification is defined by how long they are presented during a sampling period. In general, these waveforms, can be expressed by

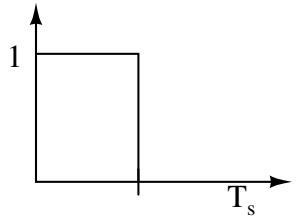
$$r(\alpha,\beta) = \begin{cases} 1 & \alpha T_s \leq t < \beta T_s, \quad 0 \leq \alpha T_s < \beta T_s \leq T_s \\ 0 & otherwise. \end{cases} \tag{2.2}$$



$$r_{NRZ}(t) = \begin{cases} 1, & 0 \le t < T_s \\ 0, & otherwise \end{cases}$$

$$R_{NRZ}(s) = \frac{1 - e^{-sT_s}}{s}$$

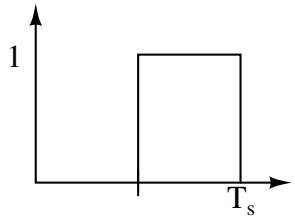
(a) NRZ pulse.



$$r_{NRZ}(t) = \begin{cases} 1, & 0 \le t < T_s/2 \\ 0, & otherwise \end{cases}$$

$$R_{NRZ}(s) = \frac{1 - e^{-s/2T_s}}{s}$$

(b) RZ pulse.



$$r_{NRZ}(t) = \begin{cases} 1, & T_s/2 \le t < T_s \\ 0, & otherwise \end{cases}$$

$$R_{NRZ}(s) = e^{-s/2T_s} \frac{1 - e^{-s/2T_s}}{s}$$

(c) HRZ pulse.

Figure 2.3: Feedback DAC waveforms.

where for a Non-Return-to-Zero (NRZ), a Return-to-Zero (RZ) and a Half-delayed Return-to-Zero (HRZ) pulse, the function becomes $r(0,1)$, $r(0,0.5)$ and $r(0.5,1)$, respectively.

A rectangular DAC pulse is particularly advantageous because it can be generated by simply switching current or voltage sources using the system's clock. However, the rectangular feedback realization is strongly affected by the purity of the sampling clock because unlike the ADC, the resulting DAC errors are directly introduced into the modulator input. Clock jitter directly modulates the ADC decision point as well as the rising and falling edges of the DAC. These issues modify the quantity of the feedback signal, resulting in a statistical integration error and consequently in increased noise. This issue has motivated to propose others feedback waveforms realizations but none of them has become popular.

There exist other performance limitations, among them, the delay between DAC and quantizer clock originated by the finite time of switching, non-linearity due to slew rate (SR) limitations and mismatch in multi-bit DACs. In spite of this, there are compensation techniques for these issues such as dynamic element matching (DEM) and data weighted averaging (DWA) [6]. However, These techniques have limitations in themselves that can impact negatively the performance.

2.2 Modeling and Simulation at System Level

In order to have a better approximation in modeling of modulators, the most of the non-idealities must be taken into account. Generally, the model of the circuit's non-idealities can be done by using MATLAB and Simulink, which is widely reported. As a consequence, with a correct behavioural model is possible to find the specifications of the building blocks to achieve the required specifications. Next, it will be described how it is done.

2.2.1 Considering the Effects of Gain and Finite Bandwidth

There are two main repercussions that are introduced by the gain and GBW. Due to the finite gain, the integrator pole is shifted to a value that depends on the sampling frequency and DC gain of amplifier. Consequently, the CT integrator transfer function can expressed as [22]:

$$ITF_{\varepsilon_{A_0}} = \frac{f_s A_0}{s(1+A_0)+f_s} = \frac{\alpha f_s}{s+\gamma} \tag{2.3}$$

$$\alpha = \frac{A_0}{1+A_0}, \quad \gamma = \frac{f_s}{1+A_0} \tag{2.4}$$

where A_0 is the OpAmp's DC gain, and α and γ represent the integrator gain error and the pole displacement, respectively. The effect of finite gain can be evaluated by calculating the total in-band quantization noise power (IBN) which, for an arbitrary order (M) single loop modulator, is given by [22]:

$$IBN(A_0, M) = \frac{\Delta^2}{12 k_1^2 k_q^2} \cdot \left[\frac{1}{A_0^{2L} OSR} + \sum_{m=1}^{M} \frac{\pi^{2m} M(M-1)...(M-m+1)}{(2m+1) OSR^{2m+1} A_0^{2(L-m)} m!} \right] \tag{2.5}$$

with k_q the effective quantizer gain and k_1 the feedback coefficient. For a third order modulator the IBN can be approximated by

$$IBN(A_0) \approx \frac{\Delta^2}{12 k_1^2 k_q^2} \cdot \left[\frac{\pi^6}{7 OSR^7} + \frac{3\pi^4}{5 OSR^5 A_0^2} \right] \tag{2.6}$$

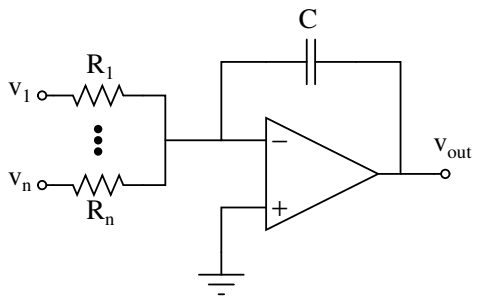


Figure 2.4: OpAmp RC integrator with multiple inputs.

where it is possible to calculate a critical DC gain to minimize the additional noise caused by the integrator leakage. However, if integrated distortion is considered, the required DC gain has to be much higher [22]. For single loop modulators, the minimum gain required is less than in multi-loop architectures.

In the case of GBW, the influence over modulator is studied by using the single pole approximation of amplifier (2.7). Accordingly, and neglecting the effect of DC gain, the entire ITF under the influence of finite GBW considering an integrator with multiple inputs (Figure 2.4), can be expressed as (2.8), where GBW is given in radians per seconds and k_j are the scaling coefficients of all input paths, which are mapped into different integrator resistors [23].

$$A_0 = \frac{A_0}{1 + \frac{s}{p_{dom}}} \tag{2.7}$$

$$ITF_{\varepsilon_{GBW}} \approx \frac{kif_s}{s} \frac{\frac{GBW}{GBW+\sum_{j=0}^{n}|k_j f_s|}}{\frac{s}{GBW+\sum_{j=0}^{n}|k_j f_s|}+1}. \tag{2.8}$$

From (2.8), the induced errors of finite GBW can be seen as a gain error (GE_j) and a parasitic pole (ω_j) defined by:

$$GE_i = \frac{GBW}{GBW + \sum_{j=0}^{n}|k_j f_s|} \;\; ; \;\; \omega_j = \frac{s}{GBW + \sum_{j=0}^{n}|k_j f_s|}. \tag{2.9}$$

An estimation of IBN as a function of GBW for single loop modulators is shown in (2.10). GBW_i denotes the gain-bandwidth product of the respective integrator and K_Q is $\frac{\Delta^2}{12k_1^2k_q^2}$. It can be shown that the required GBW is smaller than in the switched capacitor counterparts [22].The modeling of gain and finite GBW is made by using transfer functions as shown in Figure 2.5.

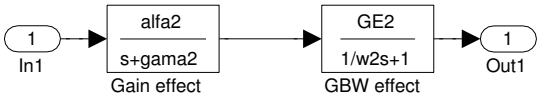


Figure 2.5: Real integrator model [23].

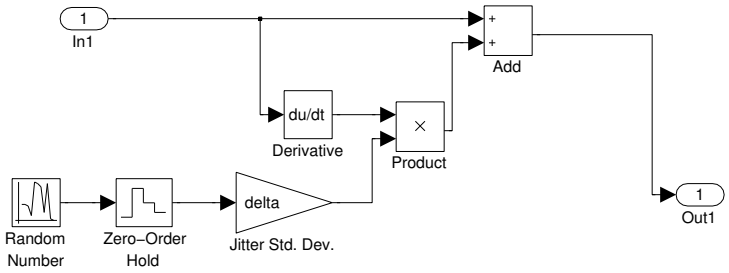


Figure 2.6: Modeling a random sampling jitter [24].

$$IBN(GBW, M) = K_Q \cdot \left[\frac{\pi^{2M-1}}{(2M+1)OSR^{2M+1}} \left(1 + \frac{k1 f_s}{GBW_1}\right)^2 \cdot \prod_{i=2}^{M} \left(1 + \frac{f_s}{GBW_i}\right)^2 \right] \quad (2.10)$$

2.2.2 Clock Jitter

Despite the fact that the effect of clock jitter in CT and DT ΣΔ ADCs is very different, it can be modeled in a similar way taking into account the effect at the input of the DT modulator and at the feedback path of the CT modulator. Mainly, sampling clock jitter results in increasing the total error power at the input of the modulator due to feedback DAC errors. The magnitude of this error is a function of both the statistical properties of jitter and the input signal of the modulator. Considering a sinusoidal signal with amplitude A and frequency f_{in}, the error introduced when this signal is sampled at a time instant with an error of δ is given by

$$x(t+\delta) - x(t) \approx 2\pi f_{in} \delta A \cos(2\pi f_{in} t) = \delta \frac{dx(t)}{dt}. \quad (2.11)$$

Here it is assumed that the sampling uncertainty δ is a Gaussian random process with standard deviation $\Delta\tau$ (parameter delta in Figure 2.6). While in DT modulators the jitter noise can be reduced by increasing de OSR, in CT counterpart the errors of feedback DAC are worse when f_s is increased, therefore its influence in IBN can not be easily reduced.

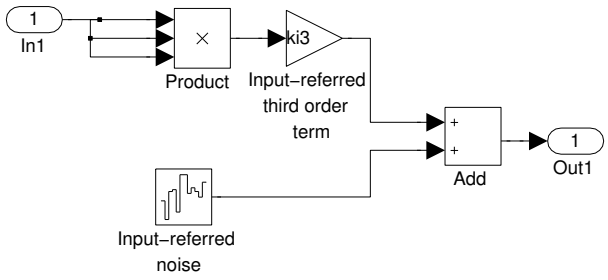


Figure 2.7: Input-referred noise and distortion model [6].

2.2.3 Other Non-Indealities

Features such as noise and non-linearity of the amplifier are critical in the performance of a modulator. The additional noise can not be enough suppressed, therefore, the reduction of the SNR is compromised. On the other hand, distortion directly affects the maximum achievable resolution by degrading the SNDR. These issues can be modeled as in Figure 2.7 where white noise is equivalent to thermal noise and the cubic power term followed by a gain stage is the harmonic distortion if a fully differential implementation is accomplished. Other non-idealities such as saturation of amplifiers due to power supply and excess loop delay (ELD) [3] can be included in the model by using saturation blocks and continuous time processing delay, respectively.

2.2.4 Example of Modeling a Modulator

To have an intuitive understanding of modeling, for this example the system proposed in [13], shown in Figure 2.8 is used. The method to obtain the modulator coefficients is as follow: firstly, the optimized noise transfer function in discrete time is obtained by using a delta-sigma toolbox of Simulink [13]. In this step, it is necessary to know the specifications that are required which define the order of the filter loop: OSR and, in this case, the local feedbacks for obtaining a pair of optimal complex zeros in NTF. The resulting NTF is given by (2.14):

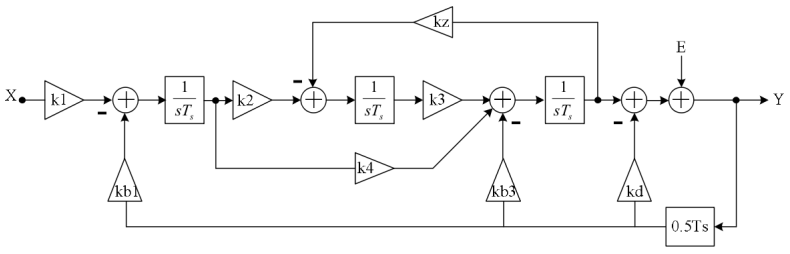


Figure 2.8: Third order modulator including ELD compensation [13].

k_1	k_2	k_3	k_4	k_z	$k_{b1,3}$	k_d
9/4	3/8	3/4	3/4	9/80	9/4	9/8

Table 2.1: Coefficients of modulator.

$$NTF = NTF(order, OSR, zero_{opt}, H_{inf}) \tag{2.12}$$

$$= NTF(3, 16, 1, 4) \tag{2.13}$$

$$NTF(z) = \frac{z^3 - 2.977z^2 + 2.977z - 1}{z^3 - 0.6323z^2 + 0.3992z - 0.05593} \tag{2.14}$$

For including the ELD compensation, it is necessary to re-sample at $T_s/2$ (here, the sampling frequency is normalized to 1) the loop filter obtained from (2.14). Next, the impulse invariant transformation is applied to (2.16), accomplishing an equivalent transfer function in CT (2.17).

$$H(z) = \frac{1 - NTF(z)}{NTF(z)} \tag{2.15}$$

$$H(z^{1/2}) = d2d(H(z)) \tag{2.16}$$

$$H(s) = d2c(H(z^{1/2})) \tag{2.17}$$

Finally, the NTF in CT is derived from (2.17). The relationship between coefficients of modulator and (2.17) can be easily found after applying the Mason's rule to 2.8 where (2.20) is obtained. The coefficients are presented in Table 2.1.

$$NTF(s) = \frac{1}{1 + H(s)} \tag{2.18}$$

$$NTF(s) = \frac{s^3 + 0.02401s}{s^3 + 2.321s^2 + 1.705s + 0.6147} \tag{2.19}$$

$$NTF(s) = \frac{s^3 + k_z k_3 s}{s^3 + k_{b3}s^2 + (k_z k_3 + k_{b1} k_4)s + k_{b1} k_2 k_3} \tag{2.20}$$

Simulation Results

There are some considerations that must be taken into account to have accurate simulation results. First, the algorithm used for calculating the Power Spectral Density (PSD) is the Fast Fourier Trans-

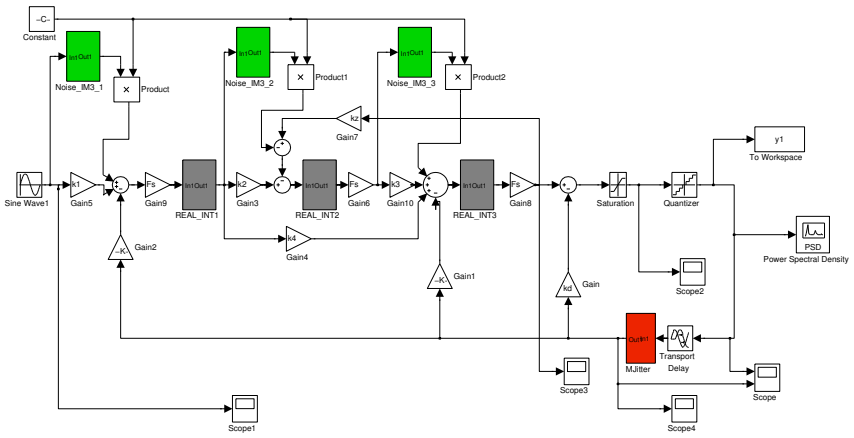
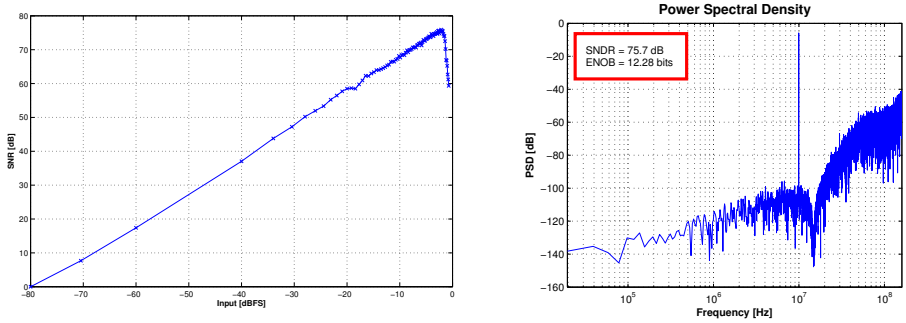


Figure 2.9: Simulink model of the third order modulator.

form, so a number of points $N = 2^n$ should be used [25]; second, for avoiding spurious tones, the number of cycles of the input signal within N points should be a prime number. The number of points chosen in next simulations is 2^{14} and the number of cycles are chosen depending on the desired input frequency.

The whole model is shown in Figure 2.9, where non-idealities are considered like gain and finite GBW, noise and non-linearity and jitter in the feedback loop. The bandwidth of the modulator is chosen to be 10 MHz, therefore, the frequency denormalization is made by using f_s =320 MHz or T_s=3.125 ns.

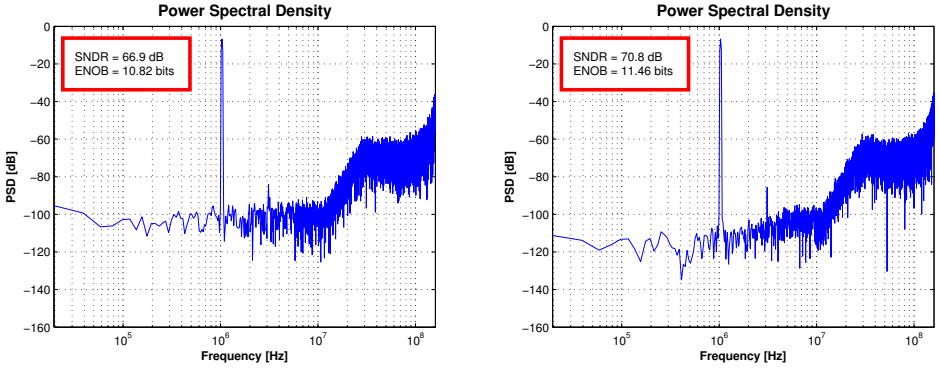
The simulation results show in Figure 2.10 are for an ideal modulator, *i.e.* without considering non-idealities. The dynamic range with input at 10 MHz (a) and PSD with level input of -2.2 dBFS (b) are presented. Figure 2.10(a) shows that the achieved DR is close to 77 dB while the peak SNR is above 75.7 dB. Next simulations will consider the effects of non-idealities in order to see how the performance is degraded.



(a) Dynamic range of the system used.

(b) Peak SNR at 10 MHz.

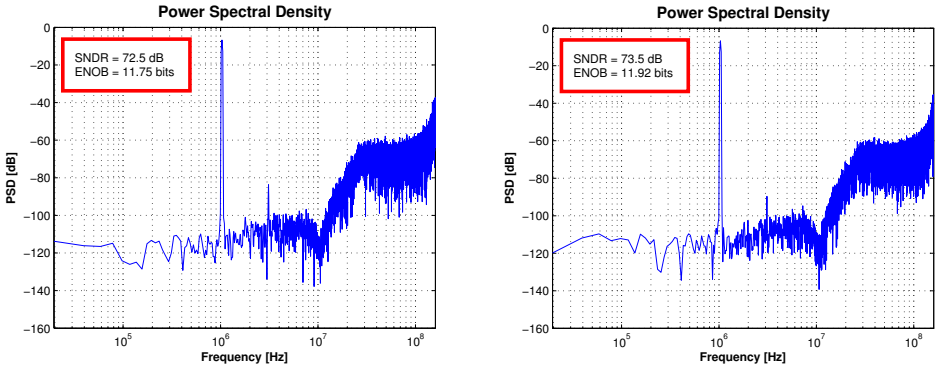
Figure 2.10: Ideal dynamic range and peak SNR for third order modulator.



(a) First integrator noise. HD3=-56 dB.

(b) Second and third integrator noise. HD3=-56 dB.

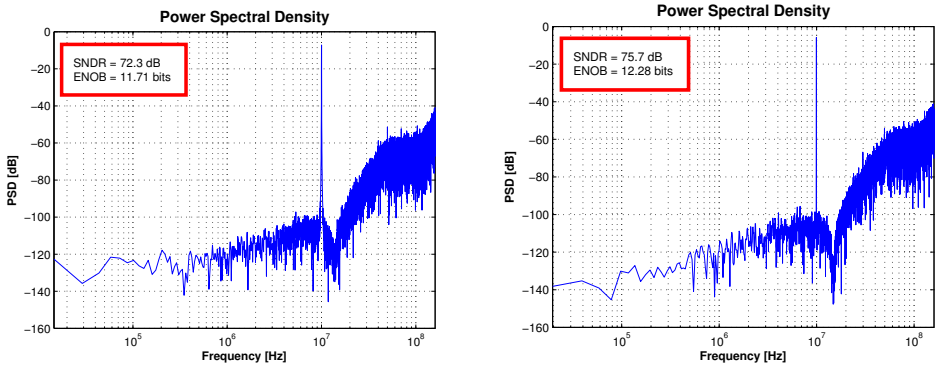
Figure 2.11: Effect of white noise added to integrators for third order modulator.



(a) First integrator non-linearity. HD3=-50 dB without including noise.

(b) Second and third integrator non-linearity. HD3=-50 dB without including noise.

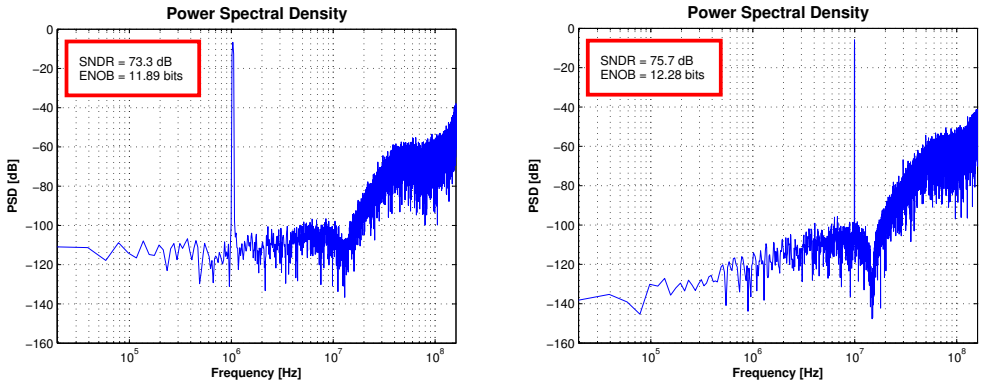
Figure 2.12: Influence of harmonic distortion of amplifiers for third order modulator.



(a) Effect of parasitic poles. Amplifiers gain Gain 50 dB and GBW 800MHz.

(b) Ideal spectrum.

Figure 2.13: Considering the effect of GBW and finite gain for third order modulator.



(a) Including jitter noise in feedback loop. σ=2 $\mathrm{ps_{rms}}$.

(b) Ideal spectrum.

Figure 2.14: Effect of jitter noise for third order modulator.

SNR vs Non-Idealities

The mainly issue of a CT $\Sigma\Delta$ modulator is the sensitivity to circuit's non-idealities. However, the effects of this non-ideal performance are more important in some particular building blocks. For example, Figure 2.11 is the SNR under the effects of thermal noise in the integrators, where is assumed equal power density for every one. 2.11(a) corresponds to noise considered on the first integrator, while 2.11(b) considers noise in second and third integrators. It is clear that the influence of first integrator widely defines the maximum SNR achievable. If non-linearity is considered, again the first integrator has the most prominent effect over performance as it can be see in 2.12. Although, Figure 2.12(a) and 2.12(b) have not so different SDNR, the performance degradation due to non-linearity of first integrator is higher than due to other integrators. The effects of finite GBW can be seen in Figure 2.13(a). Here, it is assumed that amplifiers have a gain of 50 dB and 800 MHz of GBW [13]. In 2.2.1, it was explained how this non-ideal feature affects the IBN, thus, it was expected that SNR was degraded.

DACs are the other critical blocks in the modulator which are affected by clock jitter. Figure 2.14 shows the PSD of modulator subject to jitter noise with a standard deviation of 2 $\mathrm{ps_{rms}}$ (a) and with ideal performance (b). As expected when jitter noise is considered the floor noise is increased degrading again the modulator performance.

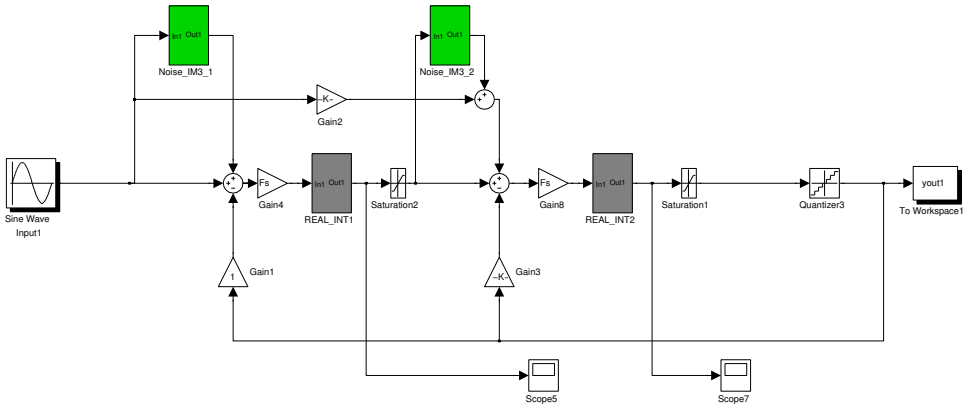


Figure 2.15: CT ΣΔ modulator at system level.

2.3 Simulations at system level of the proposed modulator

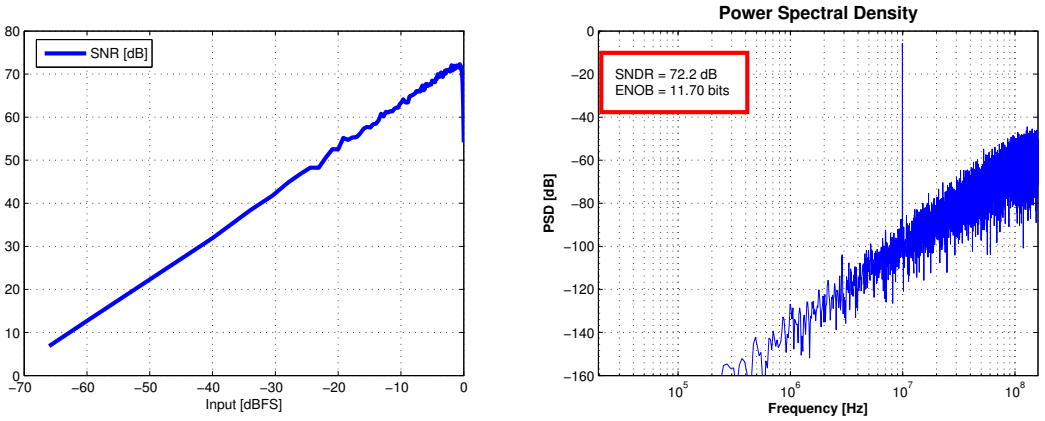
In last section a third order modulator was analyzed where many circuit's non-idealities were taken into account. However, the implementation of this modulator is not practical because signal excursion of integrators can not be maintained within real values. Also, as the interest of this work is the influence of amplifiers non-idealities in the modulator, a simpler system will be proposed.

The definition of a system architecture must be made according with the specifications required. In [13] it is proposed that, for achieving a given resolution, it is reasonable to set an over design specification of at least 2 bits. Equation (2.21) defines the maximum DR as a function of OSR, filter loop order L and resolution of internal quantizer. As the focus of this work is the performance of amplifiers and how they can affect the performance of the entire modulator, many non-idealities are considered in the model of amplifiers. Next, the target of this work and the specifications needed to accomplish the target will be defined.

$$DR = \frac{3}{2}\left(\frac{2L+1}{\pi^{2L}}\right)(2^N-1)^2 \times OSR^{(2L+1)} \tag{2.21}$$

In this work, a reasonable target is considered , based on Table 1.1, a modulator with 10 MHz of bandwidth and 10 bits resolution with OSR of 16 and internal quantizer of 4 bits. The minimum filter order is 2 in order to establish ideally an ENOB of 12 bits, according to [13]. Thus, the maximum achievable resolution is 72 dB or 11.66 bits.

The Simulink model of the modulator is presented in Figure 2.15. Here, the coefficients were calculated according to [26] and their values are 1 and 1.5 for feedback paths and the coefficient of feedforward path is 0.5. A very simple system was chosen in order to evaluate the effects of integrators in the whole system. The model includes non-idealities of the amplifiers such as saturation due to power supply (saturation block), non-linearity and thermal noise (green box), gain and finite bandwidth (gray box) which give us the specifications of amplifiers.



(a) Dynamic range of the proposed modulator.

(b) SNDR at 1.5 MHz.

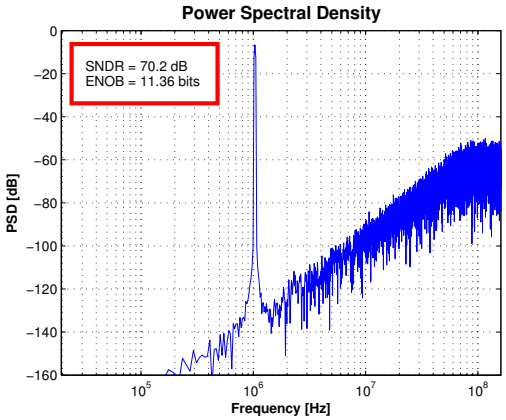
Figure 2.16: Ideal dynamic range and peak SNDR of proposed modulator.

In the Figure 2.16 the ideal performance of proposed modulator can be seen. Simulations were made by considering a frequency input signal of 1.5 MHz and the obtained maximum stable amplitude was -2 [dBFS]. DR range of the modulator is around 68 dB while the obtained peak SNR is 72.2 dB, which is equivalent to have a ENOB of 11.7 bits. This results are very close to the estimated values with numerical calculations.

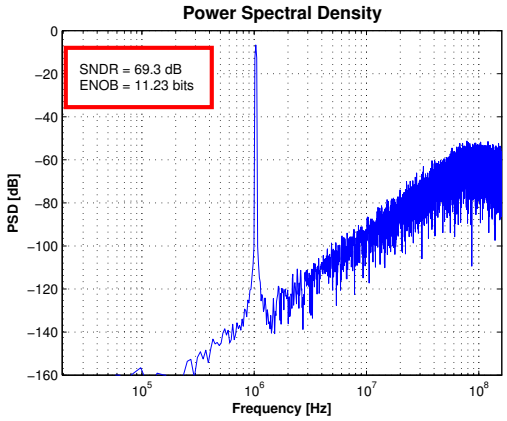
To observe the influence of amplifier gain in the modulator, some simulations was made including the DC gain and GBW model. In the Figure 2.17, the effect of gain variations are shown. Simulations were made for different gain values from 60 dB to 44 dB where can be seen how the maximum achievable SNR is degraded by reducing the amplifier gain. When DC gain is 60 dB, the peak SNR is 70 dB, but as the gain is reduced the maximum SNR falls close to 60 dB which gives a ENOB of 9.83. As a result, the specification of 10 bits is not achieved which implies that is necessary to maintain a gain in a safe margin to avoid an important degradation in modulator performance.

Figure 2.18 shows the DR and peak SNDR of the proposed system taking into account the non-idealities mentioned. With input level of -2 dBFS, the maximum SNDR and DR achievable are 66 dB and 67 dB respectively. The specification for amplifiers gain is based on (2.3) where, to have a pole displacement 1 decade away from modulator bandwidth, it is required at least 50 dB. The GBW is set at $2 \cdot f_s$. As the idea is to achieved 10 bits resolution, thermal noise and IM3 are set at values to maintain a peak SNDR above 65 dB. The maximum thermal noise and IM3 to achieve the specifications above mentioned are 31 nV/$\sqrt{\text{Hz}}$ and -62 dB, respectively.

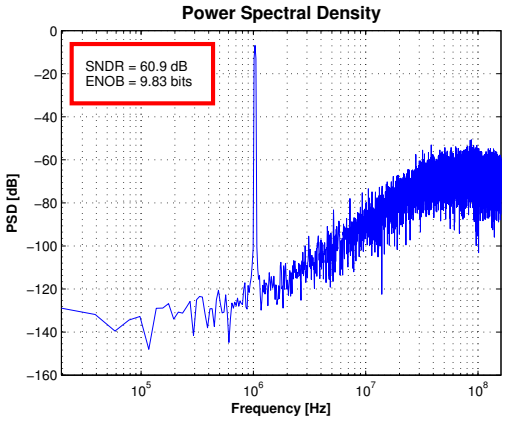
The performance obtained with the model is considered enough to use this in the design at circuit level. In the next chapter the design of amplifiers will be made based on specifications accomplished with simulations at system level.



(a) Peak SNR considering amplifier gain of 60 dB.

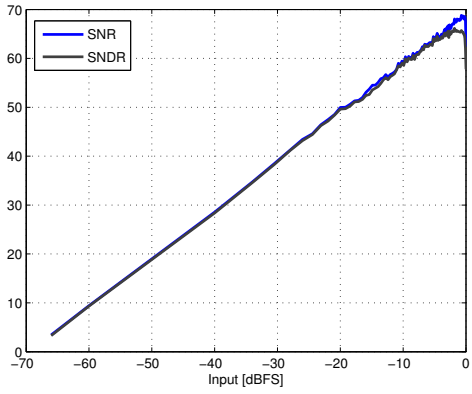


(b) Peak SNR considering amplifier gain of 50 dB.

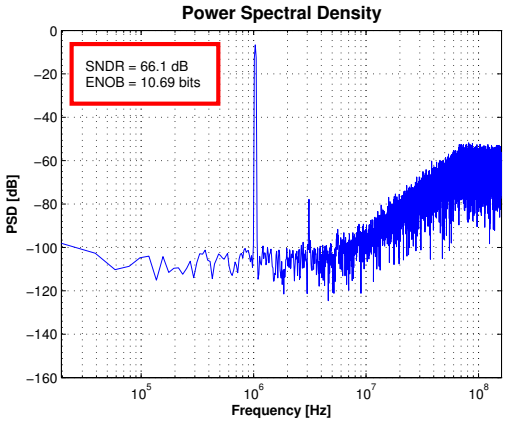


(c) Peak SNR considering amplifier gain of 44 dB.

Figure 2.17: Amplifier gain effect in the performance of the proposed modulator.



(a) Dynamic range of proposed modulator.



(b) SNDR at 1.5 MHz.

Figure 2.18: Dynamic range and peak SNDR including amplifier's non-idealities in the proposed modulator.

Chapter 3

Design and Simulations of the Proposed Circuits

In the last chapter, the main building blocks of a CT $\Sigma\Delta$ modulator and their principle of operation and more relevant issues were presented. An important conclusion is that the modulator is very sensitive to circuit's non-idealities unlike its counterpart in DT. The non-ideal performance of filter loop, *i.e.* the integrators, can significantly impact the specifications achievable with a modulator by increasing the IBN or even causing unstable operation. These drawbacks are mainly due to mismatch in filter coefficients (RC products) and the variations of DC gain and GBW of OpAmp. While calibration methods are widely used to overcome mismatch in RC products, compensating or mitigating PVT variations in OpAmps is a more difficult task. This chapter will focus in proposing an OpAmp that reduces PTV variations while providing high DC gain. The design of the circuits will be made on SOI CMOS 45 nm technology.

3.1 Gain-Enhancement

In section 2.3 the required specifications for the amplifiers were presented. It was found that the minimun gain and GBW must be 50 dB and 640 MHz, respectively. As first consideration, implementation of fully differential amplifiers will be made in order to reduce distortion and its characteristic of noise rejection [27]. Secondly, the technology used has not problem with GBW specification because of the high cutoff frequency (f_t) of its transistors. On the other hand, to achieve gain specification is a challenging task due to SCE and reduction of power supply. Below, a possible solution to the problem was briefly explained.

In the literature, there are many techniques used to improve DC gain of amplifiers. A commonly used technique is cascode transistors but it is not longer available in scaled technologies due to the restrictions in voltage supply. Instead cascode, cascade is an available alternative which has the draw-

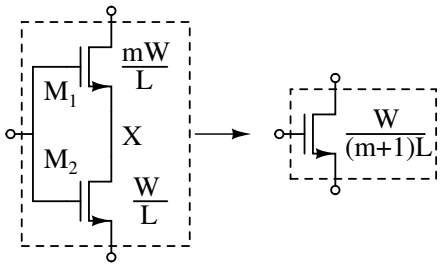


Figure 3.1: Self-cascode transistor.

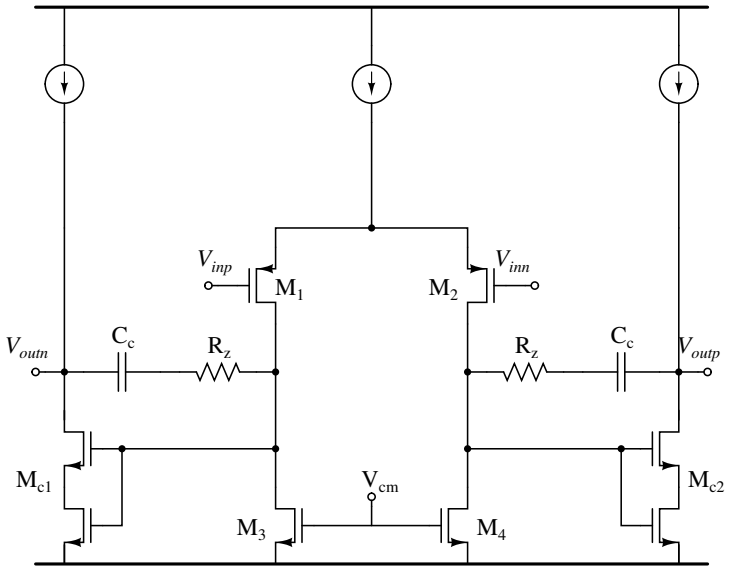


Figure 3.2: Two stage Miller amplifier.

back of making more difficult the frequency compensation. So, to improve DC gain without using many cascade stage, a possible solution is used self-cascode transistors (Figure 3.1) which can be seen as a single transistor with a higher output impedance. This can be explained considering that M_1 operates in saturation region while M_2 operates in linear region. Thus, transconductance of M_1 is given by (3.1) where V_{GX} can be approximated to V_{GS2} due to V_{DS2} is low. Thus, g_{ds2} can be calculated as in (3.2). The equivalent g_{ds} (3.3) and g_m (3.4) correspond to a transistor with channel length of $(m+1)$ times the M_1 channel length, therefore achieving a high output impedance.

$$g_{m1} = \mu C_{ox} \frac{W_1}{L_1} (V_{GX} - V_{TH}) \tag{3.1}$$

$$g_{ds2} \approx \frac{g_{m1}}{m} \tag{3.2}$$

$$g_{ds,eq} \approx \frac{g_{ds1}}{(m+1)} \tag{3.3}$$

$$g_{m,eq} \approx \frac{g_{m1}}{(m+1)} \tag{3.4}$$

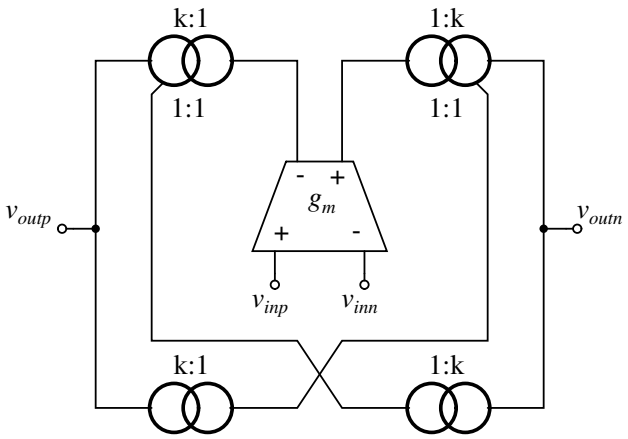


Figure 3.3: Amplifier with bidirectional output drive.

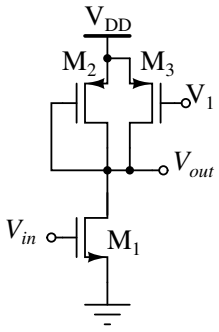


Figure 3.4: Current shunt.

Self-cascode transistor can be used in a two stage miller amplifier (Figure 3.2), which is a very common circuit. Despite improving output impedance, the amplifier gain is still limited beside being a sensitive circuit to PVT variations. So, the main idea will be reduced this sensitivity while maintain a relatively high DC gain.

As an alternative can be used a current mirror amplifier with bidirectional output drive (Figure 3.3 [27]) which is expected that has a better behaviour than miller amplifier with regard to PTV variation because its output is driven by the input transconductance stage (NMOS transistor) and current mirror 1:1 PMOS which are added to the output. However, this amplifier has only a gain stage making necessary to improve its gain beside using self-cascode transistors.

[28] proposes an enhanced current mirror amplifier by current shunt. This can be understood analyzing Figure 3.4. Generally, the gain of this stage is given by (3.5) which can be increased by making g_{m2} smaller than g_{m1} (3.6). This can be achieved as the most M_1 current flow through M_3.

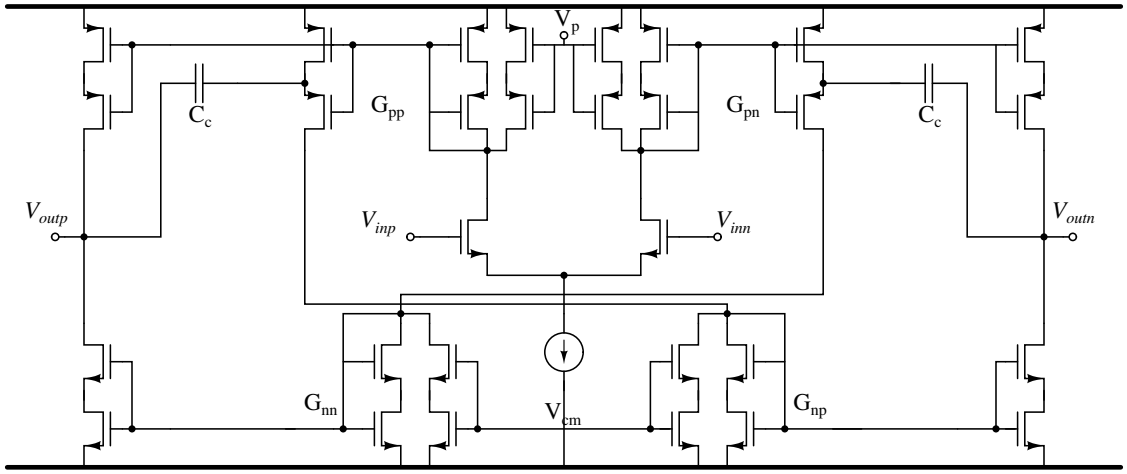
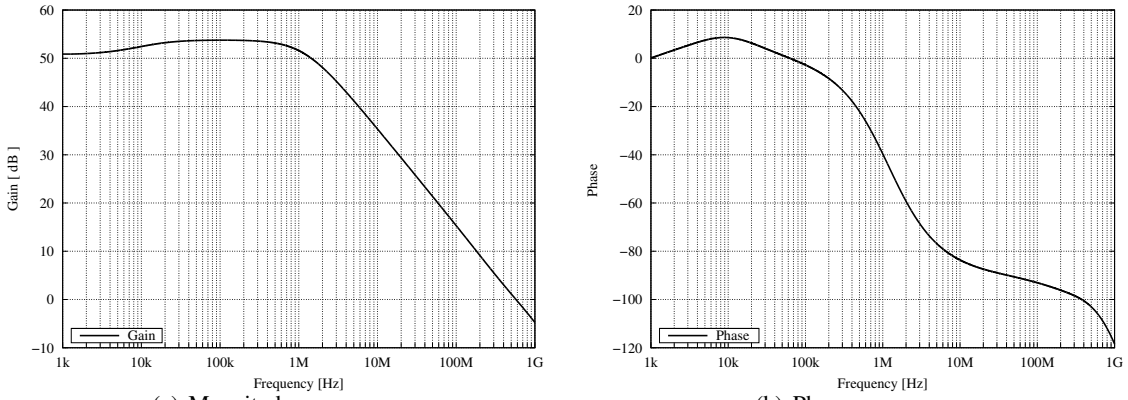


Figure 3.5: Amplifier with gain improvement.



(a) Magnitude response. (b) Phase response.

Figure 3.6: Frequency response of the Amplifier with gain improvement.

$$A_{DC} = \frac{g_{m1}}{g_{m2}} \tag{3.5}$$

$$g_{m2} = \frac{g_{m1}}{k} \tag{3.6}$$

Finally implementation of the amplifier with gain improvement is shown in Figure 3.5. The amplifier achieves a high DC gain by amplifying signal in two pair of nodes ($\mathrm{G_{pp}}$, $\mathrm{G_{np}}$ and $\mathrm{G_{nn}}$, $\mathrm{G_{pn}}$) before they are added to outputs. Vcm voltage is generated by a Common Mode Feedback (CMFB) and frequency compensation is made by using low impedance nodes of composite transistors and capacitances Cc. Frequency response of amplifier is shown in Figure 3.6.

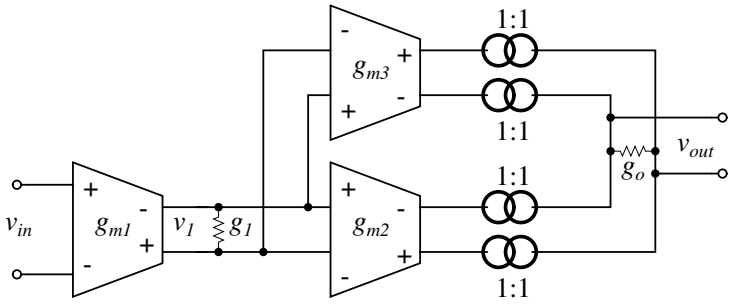


Figure 3.7: Block diagram of OTA.

From Figure 3.6, it can be seen that gain is over 50 dB and its GBW is close to 650 MHz with a PM over 80°. However, it can be observed an unusual characteristic that is not presented in CMOS technologies. The amplifier has a midband gain because transistors have zeros and poles at low frequency. The number of poles and zeros is equal but the poles have major cutoff frequency than zeros which results in gain attenuation as well as a phase increment at low frequency. These issues can affect directly of the performance of modulator causing distortion because the DC gain of amplifiers defines the noise shaping characteristic. It will be observed by simulating the whole system.

Despite having achieved the needed specifications, the amplifier has many problems in controlling its common mode which causes its gain to have wide changes with PVT variations. In order to solve this problem, in next section another amplifier finally will be proposed that is used in the whole system implementation.

3.2 Final implementation of the amplifier

In Figure 3.7 the amplification scheme is shown conceptually. Supposing that the output transconductance of the first stage is g_1, v_1 voltage can be calculated by $\frac{V_{in}g_{m1}}{g_1}$. Assuming g_o is the output transconductance, v_{out} is given by $\frac{(g_{m2}+g_{m3})v_1}{g_o}$. Thus, the overall gain is as follows

$$A_{DC} = \frac{v_{out}}{v_{in}} = \frac{g_{m1}(g_{m2} + g_{m3})}{g_1 g_o}. \tag{3.7}$$

If $g_{m2} = g_{m3}$ the gain is improved by a factor of two or 6 dB in comparison to amplifiers of two stages. As current mirrors add only non-dominant poles, the amplifier can be seen as a two stage amplifier with gain enhancement. This characteristic makes available miller frequency compensation, which is a very common technique. So, the design of the amplifier is very simple. Below, it will be explained how this scheme can be implemented at circuit level.

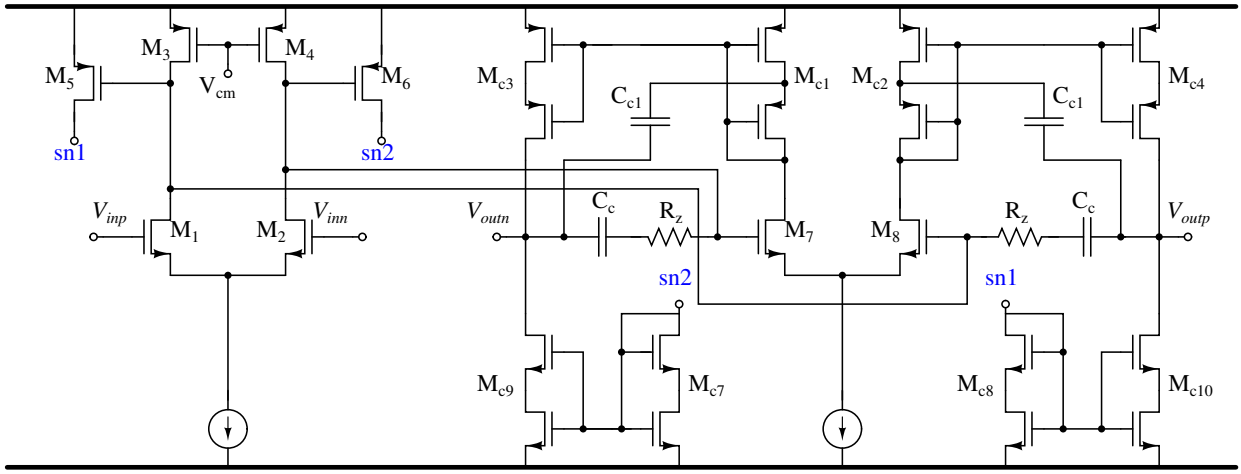


Figure 3.8: Enhanced amplifier with gain improvement.

3.2.1 Circuit implementation of proposed topology

In Figure 3.8 the OTA at circuit level is presented whose gain is given by (3.8). Some considerations were take into account in order to obtain an amplifier with high gain and reduced PVT variations. Firstly, the use of differential stage whenever it is possible since tail current avoids wide changes in transconductance of transistors. In spite of the fact that $\mathrm{M_{5,6}}$ are grounded transconductors, the CMFB circuit acts over them by adjusting the output node of the first transconductance stage through $\mathrm{V_{cm}}$, voltage which reduces wide variations of their transconductance. Finally, the signal flow of overall transconductance to the output depends on PMOS and NMOS transconductance, so it is expected that variations of transconductance at the output, with process variations, can be compensated or reduced mainly in crossed process corners.

$$A_{DC} = \frac{g_{m1,2}(g_{m5,6} + g_{m7,8})}{g_{out1}g_{out}} \tag{3.8}$$

Composite transistors are used in current mirrors in order to improve the output impedance. Miller compensation with nulling resistor is applied to maintain a safe PM. However, the use of composite transistors make additional compensation to be necessary to ensure an appropriate phase margin. Thus, additional capacitance $\mathrm{C_{c1}}$ is added between the output adn the low impedance node of transistors $\mathrm{M_{c1,c2}}$. Below, a comparison will be made between the amplifier proposed in this section and the two stage Miller amplifier.

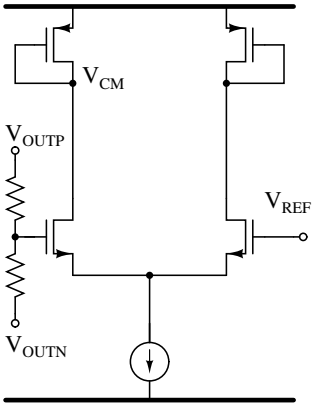


Figure 3.9: CMFB circuit.

Variable	Value [µm]	M
$W_{1,2}$	3.2	-
$W_{3,4}$	9.6	-
$W_{5,6}$	3.6	-
$W_{7,8}$	4	-
$W_{c1,2}$	7.2	3
$W_{c3,4}$	7.2	3
$W_{c7,8}$	4.8	3
$W_{c9,10}$	4.8	3

Table 3.1: Dimensions of enhanced amplifier.

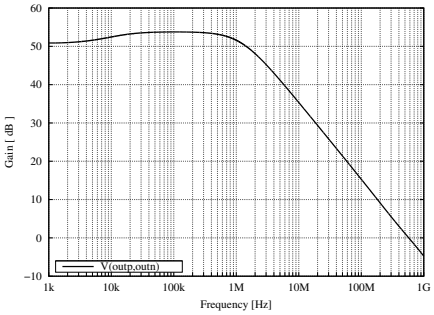
Variable	Value [µm]	M
$W_{1,2}$	2.8	-
$W_{3,4}$	4	-
$W_{c1,2}$	14.4	3
W_{tail}	43	3
W_{output}	43	3

Table 3.2: Dimensions of Miller amplifier.

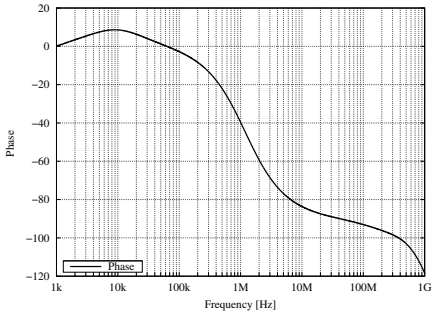
3.2.2 Design and simulations

To do the comparison, the CMFB circuit used for Miller amplifier and enhanced amplifier is the same and is shown in Figure 3.9. CMFB circuit has two resistors to obtain the output common mode voltage by averaging the differential outputs. Then, it compares the common mode voltage with a reference voltage which originates a current unbalance that finally is feedback through $\mathrm{V_{CM}}$ voltage.

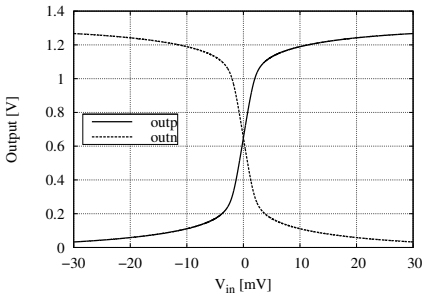
To design both circuits a channel length of 200 nm will be used because it has an appropriate trade off between $\mathrm{f_t}$ of transistors and DC gain. The first amplifier of the system used a buffer which can include excess phase. Thus, an over specification in PM is considered in order to maintain a safe margin. For enhanced amplifier, a transconductance of 1 mA/V is required to achieve GBW specification considering a load capacitance $\mathrm{C_L}$=200 fF and a compensation capacitance $\mathrm{C_c}$=200 fF. Fixing transconductances of $\mathrm{M_{5,6}}$ and $\mathrm{M_{7,8}}$ at 500 µA/V, to ensure a PM over 70°, the zero nulling resistor must be approximately 3 kΩ. To reduce distortion, transistors $\mathrm{M_{1,2}}$, $\mathrm{M_{5,6}}$ and $\mathrm{M_{7,8}}$ are biased with an overdrive voltage of 150 mV. The resulting biasing current for the first transconductance stage is 240 µA while for second transconductance stage ($\mathrm{M_{7,8}}$) is 120 µA. Biasing current of $\mathrm{M_{5,6}}$ is the same as $\mathrm{M_{7,8}}$ as they have the same transconductance. As the current mirrors are implemented with composite transistors, the m factor [29] is chosen 3 which gives an output impedance high enough for the required gain. By using, voltage supply of 1.3 V the resulting power consumption is 1.25 mW.



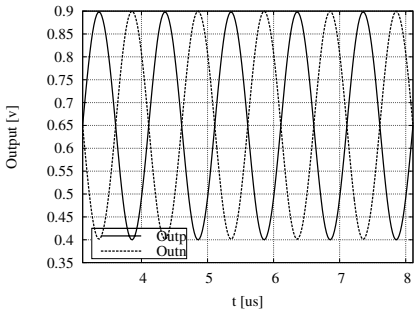
(a) Typical case magnitude response.



(b) Typical case phase response



(c) Input DC sweep.



(d) Typical case transient response

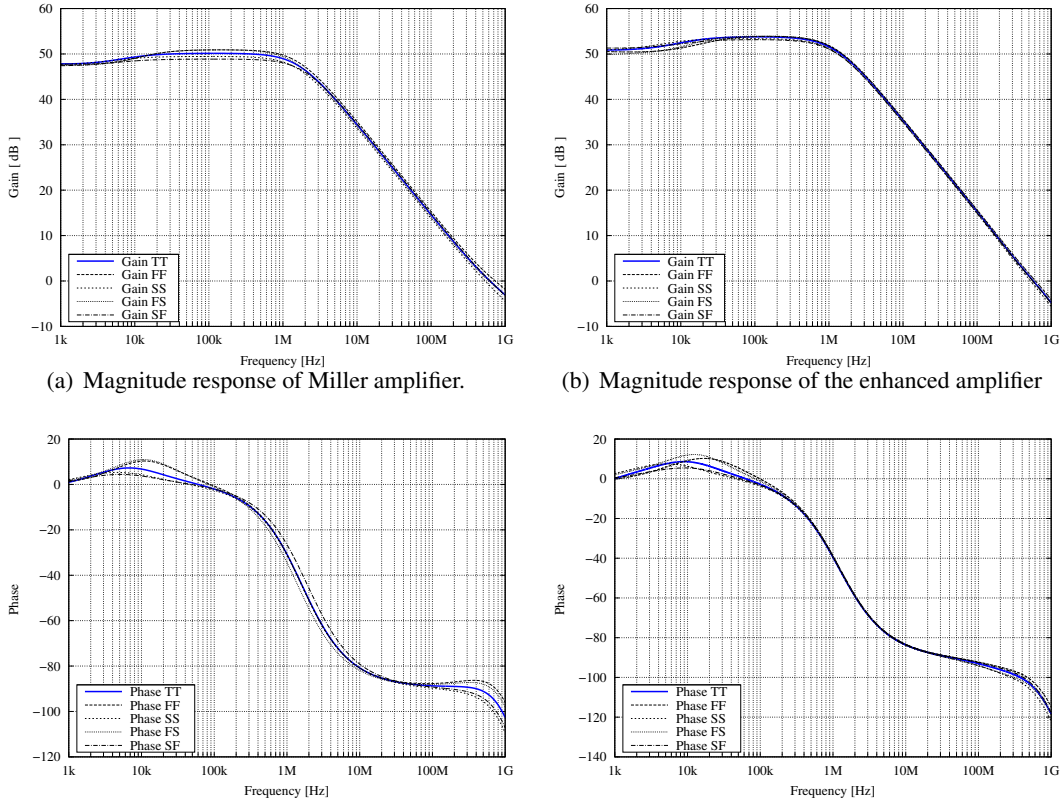
Figure 3.10: Performance of the enhanced amplifier : (a) and (b) show frequency response; (c) output voltage excursion; (d) transient response with output excursion of 500 mV.

Additional compensation capacitance C_{c1} with value 200 fF is used to improve the PM. Table 3.1 shows a summary of dimensions.

In order to compare both amplifiers, the power consumption is restricted to be similar. For Miller amplifier, the required transconductance of the first and second stage are 1 mA/V in order to achieve a safe PM and the GBW specification. By using overdrive voltage of 150 mV the resulting biasing current for the first stage is 240 µA while for each output branch it is 120 µA. The load capacitance used is the same as the above design. The compensation capacitance needed is 300 fF and zero nulling resistor is 1.2 kΩ. Transistor $M_{c1,2}$ are used with M=3 to improve output impedance the same as current sources. By using the same voltage supply, the associated power consumption is 1.35 mW. Table 3.2 summarizes the used dimensions.

Simulation results of the enhanced amplifier

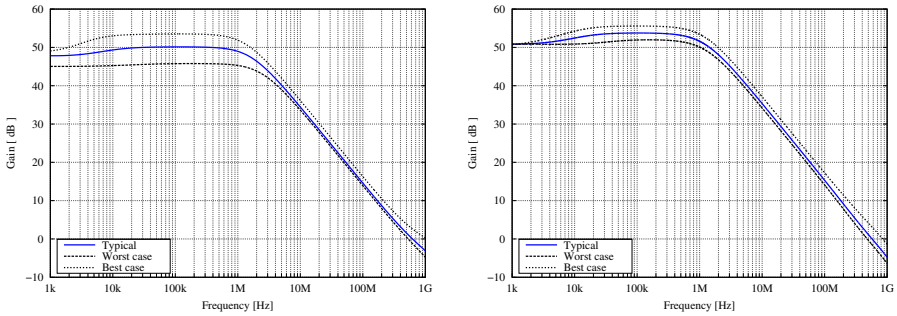
The simulations were made by setting the device temperature to 60 °C with power supply of 1.3 V maintaining typical process. The performance of the amplifier can be summarized by figures showing in 3.10. The maximum gain achieved is 53.7 dB while it has GBW of 575 MHz (Figures 3.10(a) and 3.10(b)) with PM of 75°. Figure 3.10(c) shows the output dynamic range while 3.10(d) shows the



(a) Magnitude response of Miller amplifier.
(b) Magnitude response of the enhanced amplifier
(c) Phase response of Miller amplifier.
(d) Phase response of the enhanced amplifier

Figure 3.11: Process variations with temperature and nominal voltage:(a) and (c) frequency response of Miller amplfier, (b) and (d) frequency response of enhanced amplifier.

transient response where the output range is fixed approximately to 500 mV. Harmonic distortion for transient analysis in 3.10(d) is around 0.7 %.



(a) Magnitude response of the Miller amplfier.
(b) Magnitude response of the enhanced amplifier.

Figure 3.12: PVT variations of studied topologies.

Specification	Typ.	Min.	Max.
Gain	50.1 dB	45.7 dB	53.5 dB
GBW	624 MHz	531 MHz	1.05 GHz
PM	83°	88.9°	78.5°

Table 3.3: Circuit specification of the Miller amplifier

Specification	Typ.	Min.	Max.
Gain	53.7 dB	51.9 dB	55.5 dB
GBW	574 MHz	480 MHz	845 MHz
PM	74.9°	73.5°	76.4°

Table 3.4: Circuit specification of the enhanced amplifier.

3.3 Comparison of topologies

In this section, the studied topologies will be compared in order to evaluate how they are affected by PVT variations. The specifications achieved with two stage Miller amplifier were A_{DC}=50 dB with GBW=624 MHz and PM=83°. Harmonic distortion is less than 0.7 % for a output voltage excursion of 500 mV. The comparison considers only the effects on midband gain and GBW of amplifiers.

3.3.1 Process variations

In Figure 3.11 the frequency response of OTA is shown considering only process variations. Simulations were carried out by using power supply of 1.3 V and temperature of 60°. Figures 3.11(a) and 3.11(c) show frequency response of the two stage Miller amplifier. The typical gain is 50 dB and its maximal variations are from 48.9 dB to 50.9 dB which is equivalent to -14% and 9%, respectively. The typical GBW is 624 MHz and it changes from 536 MHz to 733 MHz which is 15% and 17% . In contrast, for the enhanced amplifier, the minimum and maximum gain are 53.1 dB and 53.9 dB which in percentage is -7% and 2%. Its typical GBW is 574 MHz and it varies from 526.82 MHz (-8%) to 614 MHz (7%). The PM in proposed circuit varies 3° while in two stage Miller amplifier it changes in 10°.

3.3.2 Process, Voltage and Temperature Variations

Here, variations on voltage and temperature are also taken into account.A temperature range from -20°C to 100°C and a voltage range ± 10% from nominal power supply is used. Figure 3.12 shows typical, worst and best case of gain variations from topologies studied. In both amplifiers the worst case is observed when PMOS and NMOS transistor have slow corner, -10% in power supply and temperatures of 100°C. The best case is when PMOS and NMOS transistor have fast corner, +10% in power supply and temperatures of -20°C. Figure 3.12 shows that enhanced amplifier has significantly reduction in gain variations. For the worse case variations of GBW are not so different but for the best case Miller amplifier has wide change. Maximal variations en GBW and DC gain of Miller amplifier are 68% and 60%, while for proposed circuit are 47% and 23%. Harmonic distortion of the enhanced amplifier in the worst case is 1.2 % while for Miller amplifier is 1.3 %. Tables 3.3 and 3.4 summarize the comparison results with regard to DC gain, GBW and PM.

R_1	R_2	C	C_1	R_{DAC1}	R_{DAC2}
3.125 KΩ	3.125KΩ	1 pF	0.5 pF	3.125KΩ	2 KΩ

Table 3.5: Resistor and capacitor values.

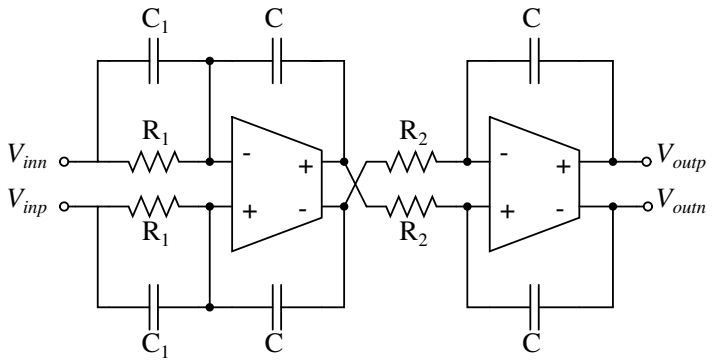


Figure 3.13: Loop filter.

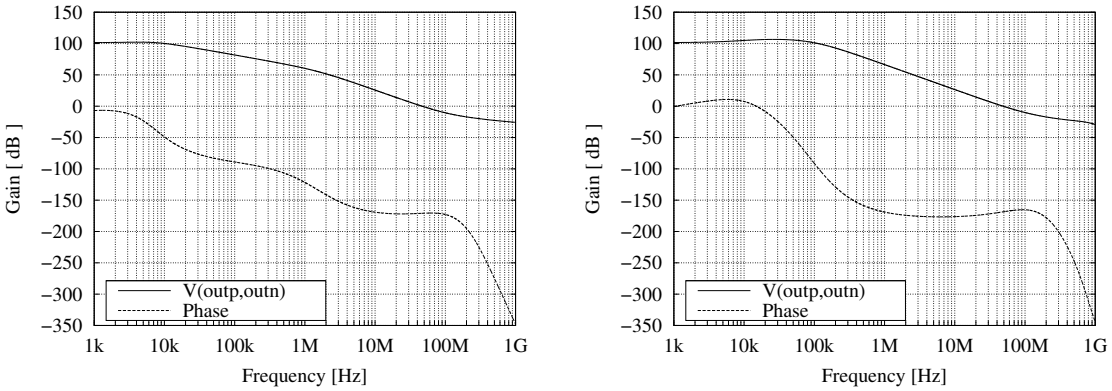
3.4 Loop Filter

Figure 3.13 shows the implementation of the loop filter. Time constants of the integrators can be calculated from (3.9) [3] where k_i corresponds to filter coefficient and the product R_iC_i defines time constant of the respective integrator. Resistance of DACs are computed in the same way. The feed-forward coefficient is given by (3.10) [30]. By fixing integrator capacitors to 1 pF, the values of other elements can be calculated. Table 3.5 show obtained values.

$$f_s = \frac{1}{C_i R_i k_i} \tag{3.9}$$

$$k_{feedforward} = \frac{C_1}{C} \tag{3.10}$$

A few number of papers mention the use of output buffers for amplifiers, *i.e.*, they use OTA as OpAmps in filter loops. However, the implementation of the loop filter with the enhanced OTA shows that exits significant difference. Figure 3.14 shows the frequency response of the loop filter using buffer (b) and without using buffer (a). Clearly, the time constants are affected by the output impedance of amplifiers. The desired frequency response, with magnitude roll-off of 40 dB and phase of 180°, is not achieved without a buffer. The first amplifier has resistive load unlike the second amplifier, therefore, it is necessary to use buffer a only in the first amplifier. In literature, an OTA in open loop is sometimes used for the next integrators to avoid that first amplifier has resistive load. Other implementations propose the use of a class AB output stage in order to drive the resistive load of the first integrator [11]. In this work the use of a buffer which will be described below is proposed.



(a) Loop filter implementation without using a buffer for the first integrator.

(b) Loop filter implementation using a ideal buffer for the first integrator.

Figure 3.14: Effect of OTA output impedance in filter frequency response.

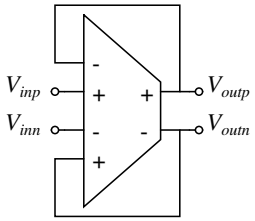


Figure 3.15: Block diagram of the fully differential buffer.

3.4.1 Proposing a Buffer

The buffers used in the literature have limitations in voltage excursion and in some case they have DC displacement. The desired buffer must have a wide input and output voltage excursion and no displacement in DC. Therefore, the first idea was to propose a fully differential buffer. Based on the error amplifier used in [31] can be modified to use as a fully differential (3.15) buffer which is shown in 3.16. It achieves a wide voltage excursion by using both input differential pairs to generate the output. This can be explained as follows: when V_{inn} is falling, M_1 tends to be turned off, however, due to fully differential operation, transistor $\mathrm{M}_{\mathrm{b}2}$ has a high V_{gs} which prevents the output current being zero. A similar operation is achieved when V_{inp} is falling. Besides having wide DR, the proposed buffer has a drawback: the output impedance is not so small because the open loop gain is small. The output impedance value is close to 2 kΩ.

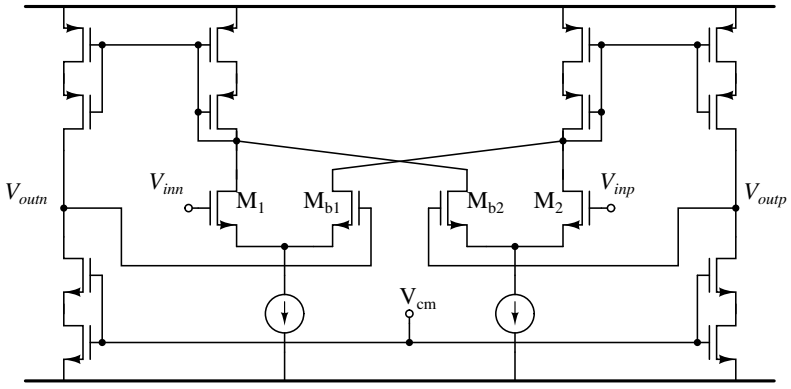


Figure 3.16: Fully differential buffer.

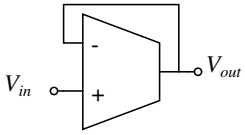
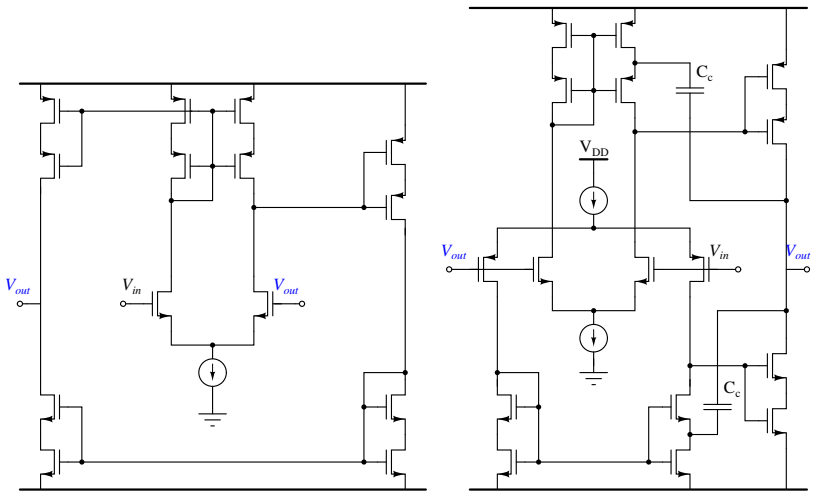


Figure 3.17: Block diagram of a single ended buffer.

In order to increase the open loop gain and reduce the output impedance, the circuit in Figure 3.18(a) is proposed. Basically, It is a feedback amplifier and achieves a high open loop gain (around 47 dB) by using a similar concept that in Figure 3.8. This circuit also is able to operate without DC displacement and has and output impedance of 200 Ω which is an important reduction in comparison to circuit in Figure 3.16 . However, the input voltage excursion is limited.

The final proposal is shown in Figure 3.18(b). This buffer uses two complementary differential pairs in order to improve input voltage excursion and a class AB output stage that improves SR. The open loop gain is close to 57 dB and frequency compensation is made by using indirect compensation [29]. The value of C_c is 500 fF while the load capacitance is 1pF.

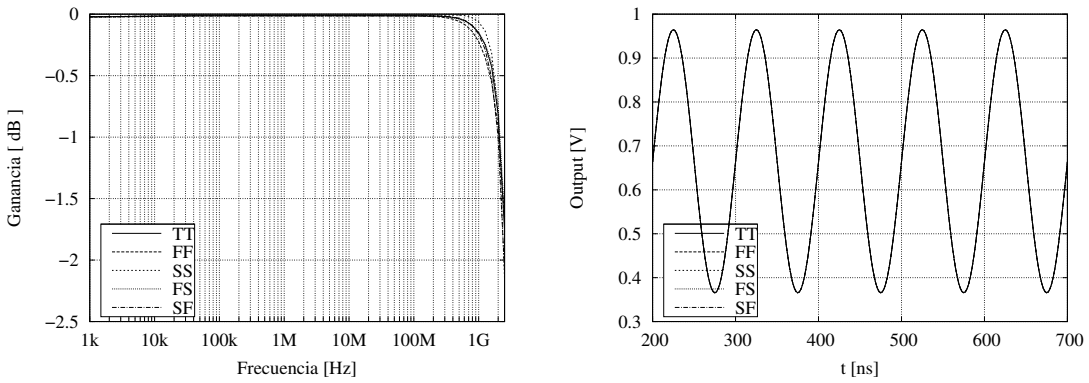
The tail current of the differential pair are 600 µA and the voltage supply is 1.3 V. Considering process variations and maintaining temperature and nominal voltage the simulations results are shown in Figure 3.19. Transient analysis was made with input of 300 mV and harmonic distortion for this conditions is less than 0.1 % for all process corners while the output impedance is not larger than 55 Ω and variations on unit gain is around 0.008 dB. Simulations with temperature and voltage corners show that distortion is kept lower than 0.1% for the considered input range. This is enough in order to avoid degrade linearity of amplifier.



(a) Buffer with open loop gain improved (b) Buffer with input voltage excursion improved

Figure 3.18: Single ended buffers.

In the next chapter, the simulations results of the whole system using the circuits designed in this chapter will be presented . VerilogA will be used to model ADC and DACs. The influence due to PVT variations in amplifiers over the modulator performance will be evaluated. It is expected to reduce the performance degradation associated to the enhanced amplifier.



(a) Magnitude response. (b) Transient response with output voltage excursion of 600 mV.

Figure 3.19: Process variations for proposed the buffer.

Chapter 4

Results

In the last chapter the analysis of PVT variations for the circuits designed was done . Simulation results show that the enhanced OTA accomplishes the gain specification while the variations of GBW are considered acceptable. In this chapter, analysis of PVT variations in the whole system will be done. VerilogA is used to model DACs and ADCs in order to implement the whole system. Finally, a brief comparison between Miller amplifier and the enhanced OTA is made.

4.1 Whole System Implementation

Figure 4.1 shows the whole modulator implementation. The resistors and capacitors values were computed in chapter 3. To verify the correct operation of the modulator, the whole system is firstly implemented in VerilogA.

4.1.1 VerilogA Implementation

VerilogA is a very useful tool that allows easy analog system and circuit descriptions. Here, description of amplifiers was done by modeling characteristics such as gain and finite GBW, output resistance and SR. The simulations are done with DC gain of 50 dB and GBW of 640 MHz according to chapter 2. Non-idealities of DACs and ADCs are not taken into account to implement their model in VerilogA. ADC is modeled with differential input and single ended output. Therefore, logic inverters were modeled to have complementary outputs. Simulations are done by using 2^{14} points for FFT and sinusoidal input of 1.5 MHz with input level of -2 dBFS. If critical circuit phenomenons are correctly described, circuit level simulations should present the required behaviour.

In the resulting spectrum (Figure 4.2), the correct operation of the modulator can be evaluated by observing the rising slope out of the band noise. For the second order modulator this slope must be 40 dB/dec which is achieved with the proposed system. Also, it can be seen that the value of SNR is according with simulations done in Simulink . Thus, the characteristics accomplished achieve the

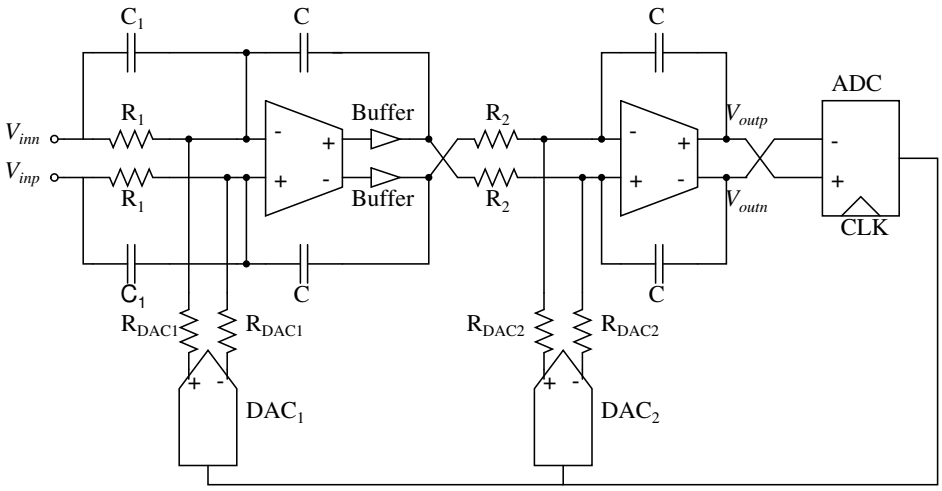


Figure 4.1: Whole system implementation.

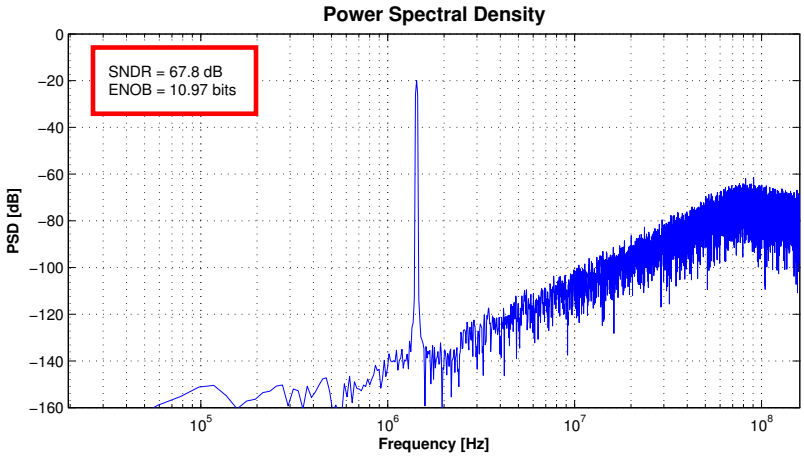


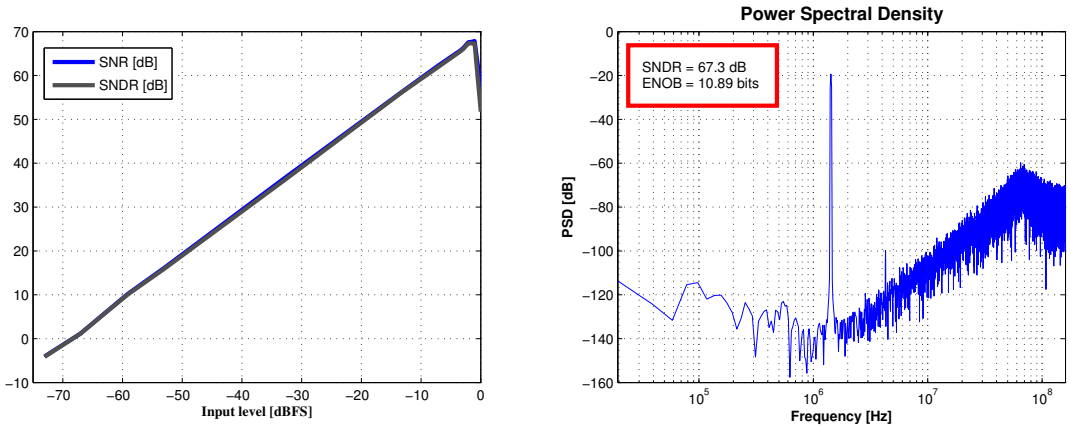
Figure 4.2: Power spectrum of system described in VerilogA.

behaviour required for circuit simulations. Below, the models of DAC, ADC and inverter will be used for whole system simulations.

4.1.2 Simulations Results Using the Proposed Circuits

As shown in Figure 4.1, only the first integrator includes buffers. The second integrator is only implemented with OTA because it does not have resistive load and its linearity requirements are relaxed. 450 mV of full scale range of the modulator is defined based on output dynamic range of the OTA to avoid introducing significant distortion. The simulations are performed with the same number of points in section 4.1.1.

Figure 4.3 show peak SNDR and DR of modulator. DR is close to 66 dB while peak SNR and SNDR are 67.7 dB and 67.3 dB respectively where the maximum input level is -1.6 dBFS. It can be



(a) Dynamic range of proposed modulator.

(b) SNDR at 1.5 MHz.

Figure 4.3: Dynamic range and peak SNDR of proposed modulator.

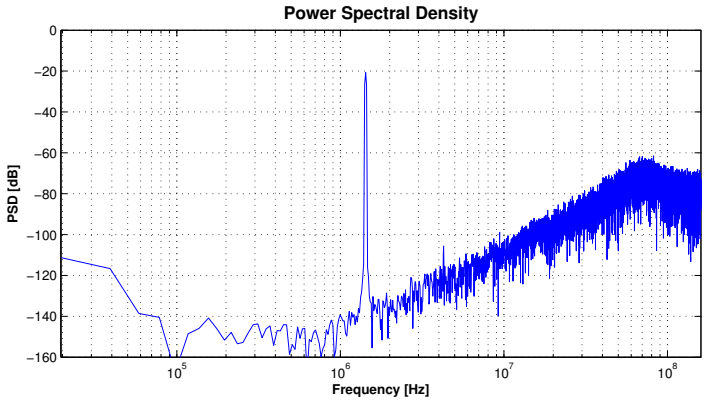


Figure 4.4: Power spectrum with increment of IBN at low frequency.

seen that the proposed full scale range widely reduces distortions of the amplifier. For implementation of ADC it can be concluded that full scale range can be increased in order to relax the quantization step that must be resolved by ADC. From Figure 4.3(b), an stable operation of the modulator can be seen by observing the slope of the noise shaping characteristic which for simulations results is 40 dB/dec.

In chapter 3 it was seen that DC gain of amplifiers has an attenuation at low frequency. The effect of this issue is shown in Figure 4.4. The simulation was done by ensuring the setting of modulator's output. From figure 4.4, it can be see that IBN noise at low frequency is increased due to the shape of the amplifier's gain. It is possible that this characteristic influences the similitude of SNR and SNDR peak because the power of distortion becomes similar to the IBN at low frequency.

The specifications achieved with the modulator using the enhanced amplifier shows agreement with the results obtained at system level simulations. Resolution required can be accomplished by the

Temperature	TT		FF		SS		FS		SF	
[°C]	SNR	SNDR	SNR	SNDR	SNR	SNDR	SNR	SNDR	SNR	SNDR
100	65.5	65.3	66.3	66.1	65.8	65.5	66.6	66.3	67.5	66.9
60	66.1	65.8	65.7	65.5	65.9	65.7	66.9	66.7	66.2	65.9
-20	67.6	67.3	67.2	67.1	67.3	67.1	67.2	67.1	67.1	66.8

Table 4.1: Performance summary. SNR and SNDR are expressed in dB.

system proposed while the amplifier has a total power consumption of 6 mW. Below, it will be shown the influence of PVT variations on the system.

4.1.3 PVT analysis

Robustness of amplifiers was analyzed in chapter 3 where for all corners of process, temperature and voltage variations the DC gain was maintained around 50 dB. Now, to evaluate the performance of amplifiers in the modulator, PVT variations are studied in the whole system. Simulation setup is the same that above simulations but the input level is set to -3 dBFS.

Table 4.1 show the SNR and SNDR achieved for all process and temperature corners. It can be see that the specification of 62 dB or 10 bits can be accomplished with the system. Minimum resulting SNR and SNDR is 65.5 dB and 65.3 dB respectively. From simulations performed, it was seen that for FF process corner the settling time of modulator was increased in comparison with other process corners. Therefore, simulation results obtained for FF process corner can not be accurate. It is necessary to review the settling time of the modulator and to do more simulations in order to improve the accuracy. However, it is expected that maximum achievable resolution does not have significant changes.

The effects of voltage supply variations on the modulator performance can be seen in Figure 4.5. These simulations was performed by fixing FS process corner and a temperature to 100°. The temperature was chosen by observing that 100° increases the distortion more significantly that others. From Figure 4.5, it can be seen that modifications in SNR for the variation range is not so wide. However, its effect in SNDR is clearly shown. As the voltage supply is reduced distortion becomes more prominent. This result was expected due to the non-linearity dependence of circuits with voltage supply. As a conclusion, the voltage supply can be reduced to around 1.1 V without strong loss in the resolution.

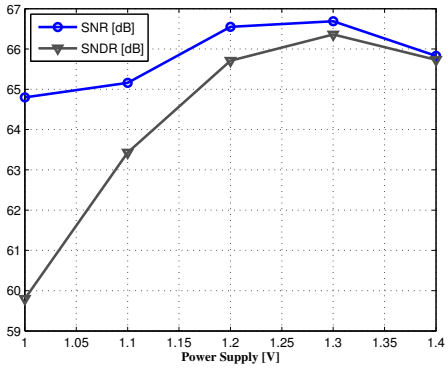
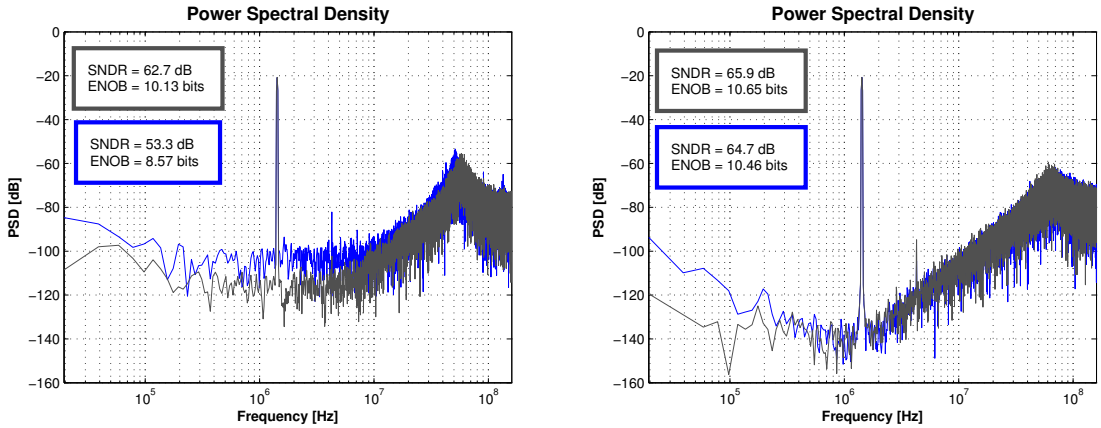


Figure 4.5: Effect of variations on voltage supply.



(a) Typical case (gray) and worst case (blue).

(b) Typical case (gray) and worst case (blue).

Figure 4.6: Output Power spectrum of the modulator: with miller amplifier (a) and with enhanced OTA (b).

4.2 Comparison of Topologies Used

Figures 4.6(b) and 4.6(a) show typical and worst case power spectrum. It can be observed that the variation in OpAmp's DC gain widely modifies the IBN and therefore degrades the SNR. Worst case refers to minimum DC gain and GBW of amplifiers which were presented in Figure 3.12. Simulations performed show that the modulator using the enhanced OTA has a minimum SNDR of 64.7 dB while for Miller amplifier it is 53.3 dB which is significant lower than the required specifications. Clearly, the enhanced OTA has better performance in comparison with Miller amplifier because it improves over 10 dB the modulator's SNDR. Also, from Figure 4.6(b) it can be seen that noise shaping has not significant variations despite having variations in DC gain. On the other hand, in Figure 4.6(a) a wide variation in noise shaping is observed .

Chapter 5

Conclusions

In this work two OTA are proposed. First amplifier achieves a DC gain of 50 dB but it is still sensitive to PVT variations and has problems with its common mode voltage. On the other hand, the second proposal achieves a DC gain over 50 dB while reduces PVT variations maintaining the required specifications. The problem of reduced gain was overcome by using self-cascode transistors which also reduced the gain dependence with voltage supply while the process variation was reduced implementing sums of the transconductance. As a result, a robust to PVT OTA with relatively high DC gain is obtained.

Comparison between Miller amplifier and the proposed amplifier by implementing the whole modulator show that an improvement over 9 dB in SNDR is achieved with the proposed amplifier. In simulation results is clearly observed that IBN increases as the amplifier gain decreases which degrades the modulator performance.

Using feedback amplifiers, different buffers was proposed. First proposal show a fully differential buffer that has wide input and output voltage excursion but has output impedance of 2 kΩ. To reduce this values, single ended buffers was accomplished where the open loop was improved. Final proposal has wide input and output voltage excursion with output impedance of 55 Ω and reduced PVT variations. Again, self-cascode transistor was used to improve DC gain.

The study of the technology used is very important to accomplish a successful design. SOI technology has advantage over Bulk-CMOS technology related to body capacitances and SCE reduction. However, floating body effects imposes strong issues like the history effects and threshold voltage variability.

Circuit's non-idealities are issues that must be taken into account to perform design of a CT $\Sigma\Delta$ modulator. Modeling of these non-idealities is pretty important in order to estimate if a given system accomplishes the required specifications. As a consequence, a successful circuit implementation depends widely on correct modeling and simulations at system level.

The influence of OpAmp's DC gain over the whole CT $\Sigma\Delta$ modulator is clearly shown in this work. Minimum gain specification is necessary to ensure the required performance of modulator. Therefore, the study of the effects of PVT variations on amplifiers is helpful to identify the maximum resolution achieved by a modulator.

5.1 Future works

Mismatch analysis must be performed to the proposed circuits in order to study its robustness to statistical variations on its parameter characteristics.

Besides doing mismatch analysis, it is necessary to do the layout to obtain simulations considering parasitic elements to validate the design.

Despite having a less significant effect on the modulator, ADC is other important building block which must be reviewed with the purpose of achieving a more efficient modulator implementation.

DACs are other circuits that impose several limitations in CT $\Sigma\Delta$ modulator. Therefore, it is necessary to analyze DAC implementations and their PVT variations in order to have a robust implementation of a modulator.

Amplifiers must be designed to improve linearity at voltages below the proposed nominal voltage in this work.

Bibliography

[1] L. Breems and J. Huijsing, *Continuous-Time Sigma-Delta Modulation for A/D Conversion in Radio Receivers*. Kluwer Academic Publishers, 2001.

[2] S. Gambini and J. Rabaey, "A 100KS/s 65dB DR $\Sigma\Delta$ ADC with 0.65V supply voltage," in *Solid State Circuits Conference, 2007. ESSCIRC 2007. 33rd European*, sept. 2007, pp. 202 –205.

[3] M. Ortmanns and F. Gerfers, *Continuous-Time Sigma-Delta A/D Conversion*. Springer, 2009.

[4] A. Chandrakasan, W. Bowhill, and F. Fox, *Design of High-Performance up Circuits*. Wiley, 2000.

[5] J. Cherry and W. Snelgrove, *Continuous-Time Sigma-Delta Modulators for High-Speed A/D Conversion*. Kluwer Academic Publishers, 2002.

[6] C.-Y. Lu, "Calibrated Continuous-Time Sigma-Delta Modulators," Ph.D. dissertation, Texas A&M University, 2010.

[7] A. Marshall and S. Natarajan., *SOI Design Analog, Memory and Digital Techniques*. Kluwer, 2002.

[8] A. M. Takayasu Sakurai and T. Douseki., *Fully-Depleted SOI CMOS Circuits and Technology for Ultralow-Power Applications*. Springer, 2006.

[9] K. Bernstein and N. Rohrer, *SOI Circuit Design Concepts*. Springer, 2007.

[10] R. Schoofs, M. Steyaert, and W. Sansen, "A 1 GHz continuous-time sigma-delta A/D converter in 90 nm standard CMOS," in *Microwave Symposium Digest, 2005 IEEE MTT-S International*, june 2005, p. 4 pp.

[11] G. Mitteregger, C. Ebner, S. Mechnig, T. Blon, C. Holuigue, and E. Romani, "A 20-mW 640-MHz CMOS Continuous-Time ADC With 20-MHz Signal Bandwidth, 80-dB Dynamic Range and 12-bit ENOB," *Solid-State Circuits, IEEE Journal of*, vol. 41, no. 12, pp. 2641 –2649, dec. 2006.

[12] K. Matsukawa, Y. Mitani, M. Takayama, K. Obata, S. Dosho, and A. Matsuzawa, "A Fifth-Order Continuous-Time Delta-Sigma Modulator With Single-Opamp Resonator," *Solid-State Circuits, IEEE Journal of*, vol. 45, no. 4, pp. 697 –706, april 2010.

[13] K.-C. Hong and H. Chiueh, "A 36-mW 320-MHz CMOS continuous-time sigma-delta modulator with 10-MHz bandwidth and 12-bit resolution," in *Circuits and Systems (MWSCAS), 2010 53rd IEEE International Midwest Symposium on*, aug. 2010, pp. 725 –728.

[14] S. Paton, A. Di Giandomenico, L. Hernandez, A. Wiesbauer, T. Potscher, and M. Clara, "A 70-mW 300-MHz CMOS continuous-time $\Sigma\Delta$ ADC with 15-MHz bandwidth and 11 bits of resolution," *Solid-State Circuits, IEEE Journal of*, vol. 39, no. 7, pp. 1056 – 1063, july 2004.

[15] L. Dorrer, F. Kuttner, A. Santner, C. Kropf, T. Puaschitz, T. Hartig, and M. Punzenberger, "A Continuous $\Sigma\Delta$ ADC for Voice Coding with 92dB DR in 45nm CMOS," in *Solid-State Circuits Conference, 2008. ISSCC 2008. Digest of Technical Papers. IEEE International*, feb. 2008, pp. 502 –631.

[16] W. Yang, W. Schofield, H. Shibata, S. Korrapati, A. Shaikh, N. Abaskharoun, and D. Ribner, "A 100mW 10MHz-BW CT $\Sigma\Delta$ Modulator with 87dB DR and 91dBc IMD," in *Solid-State Circuits Conference, 2008. ISSCC 2008. Digest of Technical Papers. IEEE International*, feb. 2008, pp. 498 –631.

[17] Y.-S. Shu, B.-S. Song, and K. Bacrania, "A 65nm CMOS CT $\Sigma\Delta$ Modulator with 81dB DR and 8MHz BW Auto-Tuned by Pulse Injection," in *Solid-State Circuits Conference, 2008. ISSCC 2008. Digest of Technical Papers. IEEE International*, feb. 2008, pp. 500 –631.

[18] Y.-C. Lin, W.-H. Hsieh, and C.-C. Hung, "A continuous-time delta-sigma modulator using feedback resistors," in *VLSI Design, Automation and Test, 2009. VLSI-DAT '09. International Symposium on*, april 2009, pp. 243 –246.

[19] M. Straayer and M. Perrott, "A 12-Bit, 10-MHz Bandwidth, Continuous-Time $\Sigma\Delta$ ADC With a 5-Bit, 950-MS/s VCO-Based Quantizer," *Solid-State Circuits, IEEE Journal of*, vol. 43, no. 4, pp. 805 –814, april 2008.

[20] C.-Y. Lu, M. Onabajo, V. Gadde, Y.-C. Lo, H.-P. Chen, V. Periasamy, and J. Silva-Martinez, "A 25 MHz Bandwidth 5th-Order Continuous-Time Low-Pass Sigma-Delta Modulator With 67.7 dB SNDR Using Time-Domain Quantization and Feedback," *Solid-State Circuits, IEEE Journal of*, vol. 45, no. 9, pp. 1795 –1808, sept. 2010.

[21] Y. Chen and K. pang Pun, "A 0.5V 2-1 cascaded continuous-time Delta-Sigma modulator synthesized with a new method," in *Circuits and Systems (MWSCAS), 2010 53rd IEEE International Midwest Symposium on*, aug. 2010, pp. 1165 –1168.

[22] F. Gerfers, M. Ortmanns, and Y. Manoli, "A 1.5-V 12-bit power-efficient continuous-time third-order Sigma; Delta; modulator," *Solid-State Circuits, IEEE Journal of*, vol. 38, no. 8, pp. 1343 – 1352, aug. 2003.

[23] M. Ortmanns, F. Gerfers, and Y. Manoli, "Compensation of finite gain-bandwidth induced errors in continuous-time sigma-delta modulators," *Circuits and Systems I: Regular Papers, IEEE Transactions on*, vol. 51, no. 6, pp. 1088 – 1099, june 2004.

[24] S. Brigati, F. Francesconi, P. Malcovati, D. Tonietto, A. Baschirotto, and F. Maloberti, "Modeling sigma-delta modulator non-idealities in SIMULINK(R)," in *Circuits and Systems, 1999. ISCAS '99. Proceedings of the 1999 IEEE International Symposium on*, vol. 2, jul 1999, pp. 384 –387 vol.2.

[25] R. Lyons, *Understandig Digital Signal Processing*. Prentice Hall, 2004.

[26] O. Belotti and F. Maloberti, "Design of Continuous-Time $\Sigma\Delta$ modulators with real analog blocks," in *Ph.D. Research in Microelectronics and Electronics (PRIME), 2010 Conference on*, july 2010, pp. 1 –4.

[27] D. Johns and K. Martin, *Analog Integrated Circuit Design*. John Wiley & Sons, Inc, 1997.

[28] M. S. L. Yao and W. Sansen, *Low power, Low voltage Sigma Delta Modulators in Nanometer CMOS*. Springer, 2006.

[29] R. J. Baker, *CMOS Circuit Design, Layout and Simulation*. IEEE Press, 2005.

[30] J. Zhang, L. Yao, and Y. Lian, "A 1.2-V 2.7-mW 160MHz continuous-time delta-sigma modulator with input-feedforward structure," in *Custom Integrated Circuits Conference, 2009. CICC '09. IEEE*, sept. 2009, pp. 475 –478.

[31] I. Padilla, "QUIESCENT CURRENT CONTROL CIRCUIT FOR CLASS AB AMPLIFIERS," Ph.D. dissertation, New Mexico State University, 2007.