Perfect Timing

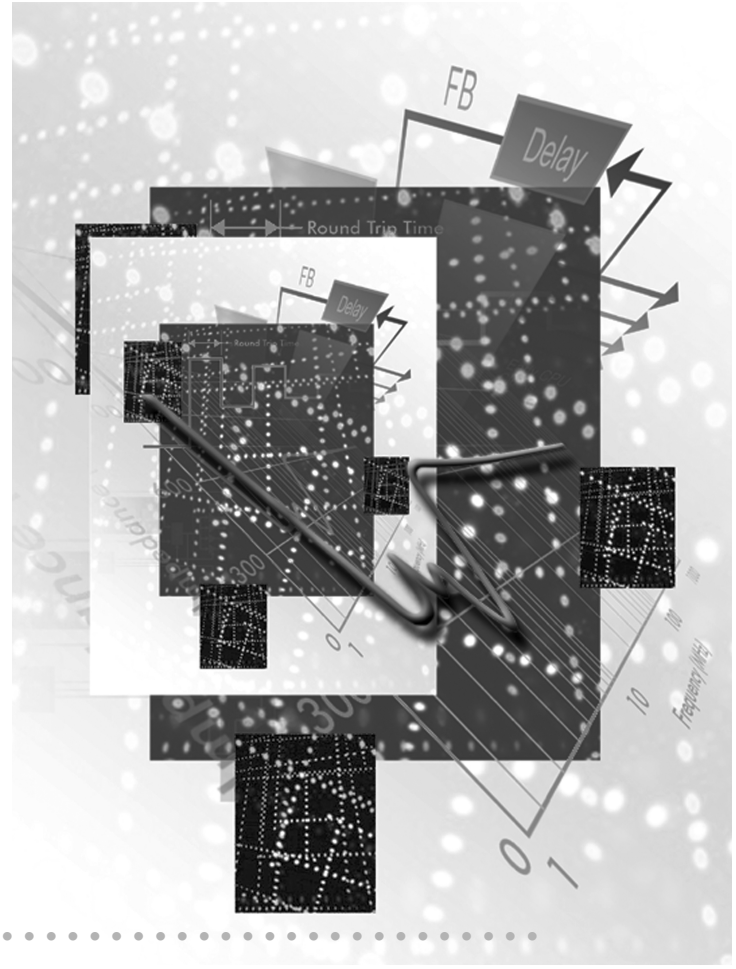
A Design Guide for Clock Generation and Distribution

CYPRESS

Driving the Communications Revolution™

Perfect Timing

A Design Guide for Clock Generation and Distribution



CYPRESS

Driving the Communications Revolution™

Table of Contents

1 Introduction

Have you designed a clock circuit where you did something because it was done that way in the past? Or have you ever use a particular part without knowing why purely for the reason it was on another design? This happens quite often with today's designs particularly with clock generation and distribution circuits.

This book has been written for those designers who want to design clock circuitry with the best methods. It is written for engineers by engineers. Its focus is on the practical implementation of clock generation and distribution circuits for use in high-speed digital designs. The material is from many time-tested designs as well as new techniques learned to address the needs for faster clock frequencies.

The ultimate goal is to have clean, solid clocks. Many companies now employ full departments dedicated to signal integrity. They perform simulations, design reviews, and a variety of analysis to ensure the best operation of the clock. There are several factors that affect the clock waveform the designer needs to know. This book will address many of the key issues for clocks.

Clock generators play a key role in designs today. In the pursuit of high-speed, many systems have adopted synchronous design styles. With this methodology comes the need for a variety of frequencies, and many copies of the same clock. In most systems, these clocks need to be in phase with one another. If they are not, precious cycle time is lost. Skew between clocks becomes very important in keeping all of the devices operating at their peak rates. Specialized clock buffers have led the way in providing clean, accurate clock signals.

Delay between clocks have also been minimized with the use of PLLs. These devices give designers the flexibility to align clock edges or allow them to be moved either ahead or back in time to increase their data valid windows. They can also compensate for trace length delays and unique chip timings. Clock buffers can really aid an engineer in creating the best possible designs.

Chapter 2 discusses clock drivers with low skew outputs and zero delay buffers. This chapter discusses their principles of operation and highlights some typical applications. These two types of devices are the cornerstones of clock distribution and are designed to specifically generate clock signals with clean, symmetric edges.

With synchronous system, timing budgets are key to successfully meeting set-up and hold times. Chapter 3 discusses the many factors that need to be addressed when calculating timing margins. Jitter, skew, phase error, and a host of other variations can steal valuable time in a given cycle. But, how do these contributors add together and interact? This chapter addresses the necessary methods for analyzing the different factors. It also introduces a new method called Total Timing Budget. This value aids the designer in determining the contributors to the cycle time without over margining the numbers.

Jitter is an often-used term when designing with clocks, however, it does have many misunderstood definitions. Chapter 4 discusses the origin of jitter, its definitions, and unravels the mysteries sometimes associated with datasheet labels. Each type of jitter is discussed and how its measurements are taken. This allows the designer to assess the types of jitter that are really of a concern to the design. It also highlights the characteristics of jitter to aid in the isolation of the source that may cause excessive jitter problems.

There are many factors that can affect a clocks performance. Among them is the penetration of noise into the power system. Noise has the effect of adding jitter and delay to clock buffers and PLLs. In Chapter 5, the effects of noise are discussed and how it can be minimized. By using bypass capacitors and ferrite beads, clean, solid power can be achieved. This chapter shows a variety of results through extensive testing of clock systems.

Also vital in providing solid, clean power to the clocking components is the layout of the power planes. Chapter 6 discusses the techniques for printed circuit board design that provides solid power performance. From planes to cutouts, from VIAs to bypass capacitor wiring, this chapter gives detailed explanations of the rules with which to follow. Also addressed are the layout rules for the individual clock signals. In order to provide error free waveforms, a variety of signal layout considerations are given. Crosstalk, impedance imbalances, VIAs, and line widths all play key roles in signal integrity and need to be considered when designing a clock circuit.

As digital clocks increase in speed, they become transmission lines. A variety of reflections can occur that will cause false triggers if they are not properly terminated. Chapter 7 discusses the many aspects of line termination for clock signals. There are several techniques available to the designer for terminating signals and therefore a variety of the methods are discussed. With the popularity of differential signal in today's high-speed world, both LVPECL and LVDS differential signal termination is shown.

When working with clock buffers and PLLs, engineers often ask "How many PLLS can I put in series?" Chapter 8 addresses this very question. It details the nature of PLLs and their loop bandwidths, which is key to understanding the answer to that question.

Electromagnetic interference, or EMI for short, is a very important factor in those devices used by the general public. Many systems must pass rigorous testing standards before the product is released. The designer is often faced with fixing an EMI problem just prior to the final phase of the design. But on many occasions, it is not well understood what caused the EMI and worse yet, what can be done about it. Chapter 9 discusses where EMI originates and how it is measured. There are some unique properties of clocking such as harmonics that are important to understand when determining the effects of EMI. This chapter explains these relationships and discusses a variety of ways to suppress these unwanted signals.

Before releasing a design for manufacture, it is recommended to simulate the clock signals to ensure adequate operation. The popular tools used today for signal integrity are IBIS and SPICE model simulators. Chapter 10 shows these models and what information about the clock component is contained inside. This gives the engineers incite on what the simulation really means.

Once the design is complete and boards are in test, how does an engineer validate the clock signal for proper operation? Can any probe or oscilloscope be used? Chapter 11 discusses the common pitfalls associated with probing high-speed digital clocks with inadequate equipment. It also outlines the methods than can be used to measure the actual clock signal.

And finally, Chapter 12 discusses the clock generator. The clock generator is an extension of the zero delay buffer but with many added features and benefits. It allows for the generation of seemingly unrelated frequencies as needed for the many clocks in today's synchronous systems. This chapter addresses how these devices work and some of the key attributes that make clock generators indispensable.

Hopefully, you will find the information in this book clear and concise, and above all, beneficial. It is meant for you, the designer, as a reference handbook. If you have any comments or suggestions on new information that should be incorporated, we would like to hear from you. Please send an email to: perfect_timing@cypress.com

2

Clock Buffer Basics

Clocks are the basic building blocks for all electronics today. For every data transition in a synchronous digital system, there is a clock that controls a register. Most systems use Crystals, Frequency Timing Generator (FTGs), or inexpensive resonators to generate precision clocks for their synchronous systems. Additionally, clock buffers are used to create multiple copies, multiply and divide clock frequencies, and even move clock edges forwards or backwards in time. Many clock-buffering solutions have been created over the past few years to address the many challenges required by today's high-speed logic systems. Some of these challenges include: High Operating and Output Frequencies, Propagation Delays from Input to Output, Output to Output Skew between pins, Cycle-to-Cycle and Long Term Jitter, Spread Spectrum, Output Drive Strength, I/O Voltage Standards, and Redundancy. Because Clocks are the fastest signals in a system and are usually under the heaviest loads, special consideration must be given when creating clocking trees. In this chapter, we outline the basic functions of Non-PLL and PLL based buffers and show how these devices can be used to address the high-speed logic design challenges.

In today's typical synchronous designs, multiple clock signals are often needed to drive a variety of components. To create the required number of copies, a clock tree is constructed. The tree begins with a clock source such as an oscillator or an external signal and drives one or more buffers. The number of buffers is typically dependant on the number and placement of the target devices. Figure 2.1 illustrates the concept of the clock tree.

In years past, generic logic components were used as clock buffers. These were adequate at the time, but they did little to maintain the signal integrity of the clock. In fact, they actually were a detriment to the circuit. As clock trees increased in speed and timing margins reduced, propagation delay and output skew became increasing important. In the next several sections, we discuss the older devices and why they are inadequate to meets the needs of today's designs. The definitions of the common terms associated with modern buffers follow. Finally, we address the attributes of the modern clock buffer with and without a PLL. The frequency timing generator (FTG) that is often used as a clock source is a special type of PLL clock buffer. A discussion of these devices can be found in Chapter 12.

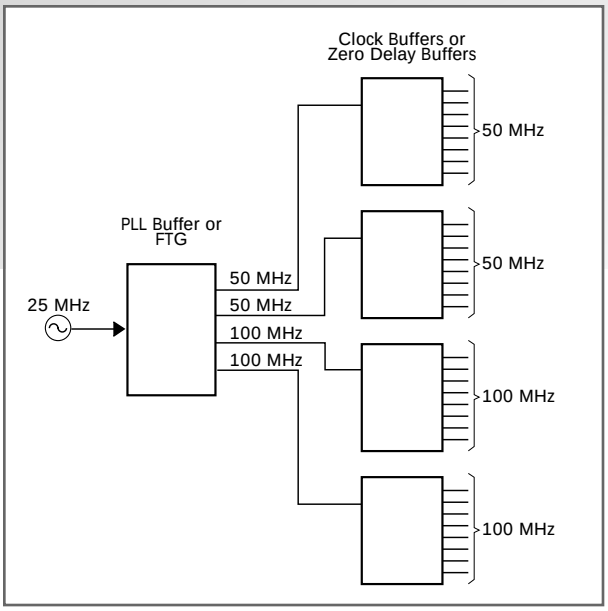


Figure 2.1 Typical Clock Tree

Early Buffers

A clock buffer is a device in which the output waveform follows the input waveform. The input signal propagates through the device and is re-driven by the output buffers. Hence, such devices have a propagation delay associated with them. In addition, due to differences between the propagation delay through the device on each input-output path, skew will exist between the outputs. An example of a non-PLL based clock buffer is the 74F244 that is available from several manufacturers. These devices have been available for many years and were suitable for designs where frequencies were below 20MHz. Designers would bring in a clock and fan it out to multiple synchronous devices on a circuit card. With these slow frequencies and associated rise times, designers had suitable margins with which to meet setup and hold times for their synchronous interfaces. However, these buffers are not optimal for today's high speed clocking requirements. The 74F244 suffers from a long propagation delay (3-5 ns) and long output-to-output skew delays. Non-PLL based Clock Buffers have improved in recent years and use more advanced I/O design techniques to improve the output-to-output skew. As the clock period gets shorter, the uncertainty or skew in the clock distribution system becomes more of a factor. Since clocks are used to drive the processors and to synchronize the transfer of data between system components, the clock distribution system is an essential part of the system design. A clock distribution system design that does not take skew into consideration may result in a system with degraded performance and reliability.

Skew

Skew is the variation in the arrival time of two signals specified to occur at the same time. Skew is composed of the output skew of the driving device and variation in the board delays caused by the layout variation of the board traces. Since the clock signal drives many components of the system, and since all of these components should receive their clock signal at precisely the same time in order to be synchronized, any variation in the arrival of the clock signal at its destination will directly impact system performance. Skew directly affects system margins by altering the arrival of a clock edge. Because elements in a synchronized system require clock signals to arrive at the same time, clock skew reduces the cycle time within which information can be passed from one device to the next.

As system speeds increase, clock skew becomes an increasingly large portion of the total cycle time. When cycle times were 50 ns, clock skew was rarely a design priority. It could be as much as 20% of the cycle time and presented no problem. As cycle times dropped to 15 ns and less, clock skew requires an ever-increasing amount of design resource. Typically, these high-speed systems can have only 10% of their timing budget dedicated to clock skew, so obviously, it must be reduced.

There are two types of clock skew that affect system performance. The clock driver causes intrinsic skew and the PCB layout and design is referred to as extrinsic skew. Extrinsic Skew and layout procedures for clock trees will be discussed later in this book.

$$t_{SKEW\_INTRINSIC} = \textit{Device Induced Skew}$$

$$t_{SKEW\_EXTRINSIC} = \textit{PCB + Layout + Operating Environment Induced Skew}$$

$$t_{SKEW} = t_{SKEW\_INTRINSIC} + t_{SKEW\_EXTRINSIC}$$

Intrinsic clock skew is the amount of skew caused by the clock driver or buffer by itself. Board layout or any other design issues except for the specification stated on the clock driver data sheet do not cause intrinsic skew.

Output Skew

Output skew is also referred to as pin-to-pin skew. Output skew is the difference between delays of any two outputs on the same device at identical transitions. JEDEC defines output skew as the skew between specified outputs of a single device with all driving inputs connected together and the outputs switching in the same direction while driving identical specified loads. Figures 2.2 and 2.3 shows a clock buffer with common input Cin driving outputs Co1_1 through Co1_n. The absolute maximum difference between rising edges of the outputs will be specified as Output Skew (t_{SK}). Typical output skew in today's high performance clock buffers is around 200ps.

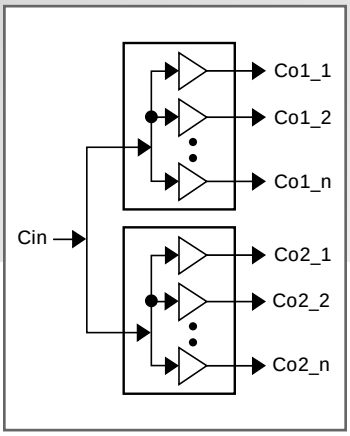


Figure 2.2 Clock Buffer Model

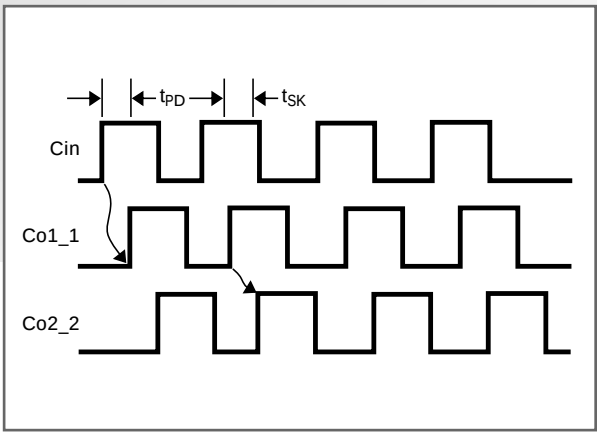


Figure 2.3 Clock Buffer Skew

Part-to-Part Skew

Part-to-part skew is also known as package skew and device-to-device skew. Part-to-part skew is similar to output skew except that it applies to two or more identical devices. Part-to-part skew is defined as the magnitude of the difference in propagation delays between any specified outputs of two separate devices operating at identical conditions. The devices must have the same input signal, supply voltage, ambient temperature, package, load, environment, etc. Figure 2.4 illustrates t_{DSK} from the preceding example. Typical Part-to-Part Skew for today's high performance buffers is around 500ps.

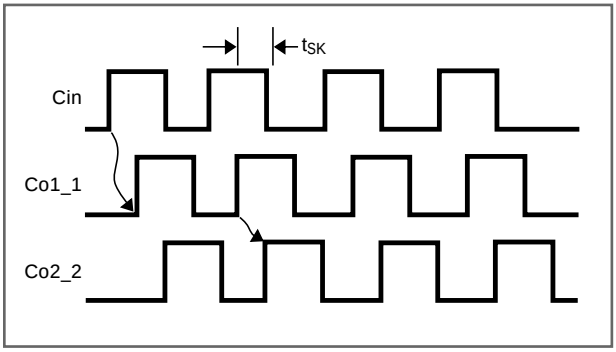


Figure 2.4 Part-to-Part Skew

Propagation Delay

Propagation delay (t_{pd}) is the time between specified reference points on the input and output voltage waveforms with the output changing from one defined level (low) to the other (low). Propagation delay is illustrated in Figure 2.3. Non-PLL based devices in today's high performance devices ranges from 3 to 7 ns. PLL based buffers are able to zero out this propagation delay with the aid of Phase Detectors, Loop Filters and VCOs.

Uneven Loading

When using a high-speed clock buffer or PLL, care must be taken to equally load the outputs of the device to ensure that tight skew tolerances are maintained. Inherent in each output of the clock driver is an output impedance that is mostly resistive in nature (along with some inductance and capacitance). When each of these resistive outputs is equally loaded, the tight skew specification of the clock driver is preserved. If the loads become unbalanced, the RC time constants of the various outputs would be different, and the skew would be directly proportional to the variation in the loading.

Input Threshold Variation

After the low skew clock signals have been distributed, the clock receivers must accept the clock input with minimal variations. If the input threshold levels of the receivers are not uniform, the clock receivers will respond to the clock signals at different times creating clock skew. If one load device has a threshold of 1.2V and another load device has a threshold of 1.7V and the rising edge rate is 1V/ns, there will be 500 ps of skew caused by the point at which the load device switches based on the input signal. Most manufacturers center the input threshold level of their devices near 1.5 volts nominal for TTL input devices. This input threshold will vary slightly from manufacturer to manufacturer especially as conditions (such as voltage and temperature) change. The TTL specification for the input threshold level is guaranteed to be a logic high when the input voltage is above 2.0 volts and a logic low when the input voltage level is below 0.8 volts. This leaves a 1.2-volt window over voltage and temperature. Components with CMOS rail swing inputs have a typical input threshold of $V_{CC/2}$ or about 2.5 volts, which is much higher than the TTL level. If the threshold levels are not uniform, clock skew will develop between components because of these variations. There are many I/O standards which have emerged and all must be taken into consideration when providing clocks to different subsystems. A table of some of the more prevalent standard is listed below which lists the standard along with the input threshold voltages.

I/O Standard	$V_{CCI/O}$	V_{REF}	$V_{IH(min)}$	$V_{IH(max)}$	$V_{IL(min)}$	$V_{IL(max)}$
1.8V Interface	1.8V	—	1.08	2.25	–0.3	0.682
LVCMOS2	2.5V	—	1.7	3.0	–0.3	0.7
LVCMOS	3.3V	—	2.0	3.9	–0.3	0.8
LVTTL	3/3.3V	—	2.0	3.9	–0.3	0.8
GTL+	—	1.0V	1.1	1.3	–0.3	0.9
HSTL Class I	1.5V	0.75V	0.8	1.9	–0.3	0.8
HSTL Class II	1.5V	0.75V	0.8	1.9	–0.3	0.8
HSTL Class III	1.5V	0.9V	0.78	1.9	–0.3	0.8
HSTL Class IV	1.5V	0.9V	0.78	1.9	–0.3	0.8
SSTL2 Class I	2.5V	1.25V	1.33	3.0	–0.3	1.17
SSTL2 Class II	2.5V	1.25V	1.33	3.0	–0.3	1.17
SSTL3 Class I	3.3V	1.5V	1.5	3.9	–0.3	1.5
SSTL3 Class II	3.3V	1.5V	1.5	3.9	–0.3	1.5

Non-PLL Based Clock Drivers

There are two main types of modern clock driver architectures: a buffer-type device (non-PLL) and a feedback-type device (PLL).

In a buffer-style (non-PLL) clock driver, the input wave propagates through the device and is “re-driven” by the output buffers. This output signal directly follows the input signal and has a delay (t_{PD}) that ranges from 5 ns to over 15 ns. These devices differ from the buffers in the past such as the 74F244 in that they are designed specifically for clock signals. On a 74F244, there are eight inputs and eight outputs. To create a one to eight buffer, all eight inputs are tied together. This causes excess loading at the inputs on the driving signal. A one to eight clock buffer has only one input and hence only one load. The output rise and fall times are also equally matched and therefore do not contribute to duty cycle error. With their improved I/O structure, the pin-to-pin skew is kept to a minimum. The output skew of this device, if it is not listed on the datasheet, can be calculated by subtracting the minimum propagation delay from the maximum propagation delay.

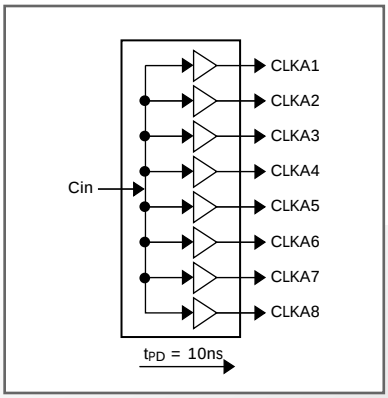


Figure 2.5 Basic Buffer

The 10 ns t_{PD} clock driver delay shown in Figure 2.5 does not take into account the affects of the board layout and design. These types of devices are excellent for buffering source signals such as oscillators where the output phase does not need to match the input. A variety of the non-PLL based buffers are available on the market today and typically range from as few as 4 outputs to as many as 30. Some devices also include configurable I/O and internal registers to divide the output frequencies.

Among the highest performance non-PLL based LVCMOS clock buffers available today is the B9940L. The B9940L is a low voltage clock distribution buffer with the capability to select either a differential LVPECL or a LVCMOS/LVTTL compatible input clock. The two clock sources can be used to provide for a test clock as well as the primary system clock. All other control inputs are LVCMOS/LVTTL compatible. The eighteen outputs are 2.5V or 3.3V compatible and can drive two series terminated 50-Ohm transmission lines. With this capability, the B9940L has an effective fan-out of 1:36. Low output-to-output skews of 150ps, a device to device skew of 750ps, and a high-end operating frequency of 200 MHz, makes the B9940L an ideal clock distribution buffer for nested clock trees in synchronous systems.

These devices still face the problems of device propagation delay. The propagation delay through these devices is about 5 ns. This delay will cause skew in systems where both the reference clock to the buffer and the outputs of the buffer need to be aligned. These devices also have the drawback that the output waveform is directly based on the input waveform. If the input waveform is a non-50% duty-cycle clock, the output waveform will also have a less-than-ideal duty cycle. Expensive crystal oscillators with tight tolerances are needed when using this type of buffer in systems requiring near 50/50 outputs.

These devices also lack the ability to phase adjust or frequency multiply their outputs. Phase adjustment allows the clock driver to compensate for trace propagation delay mismatches and set-up and hold time differences, and frequency multiplication allows the distribution of high and low frequency clocks from the same common reference. Expensive components and time-consuming board routing techniques must be used to compensate for the functional shortcomings of these buffer-style clock driver devices. PLL based devices have been incorporated to address all of these shortcomings.

PLL Based Clock Drivers

The second type of clock distribution device uses a feedback input that is a function of one of its outputs. The feedback input can be connected internally or externally to the part. If its an external feedback, a trace is used to connect an output pin to the feedback pin. This type of device is usually based upon one or more phase-locked loops (PLLs) that are used to align the phase and frequency of the feedback input and the reference input. Since the feedback input is a reflecion of an output pin, the propagation delay is effectively eliminated. In addition to very low device propagation delay, this type of architecture enables output signals to be phase shifted to compensate for board-level trace-length mismatches. Outputs can be selectively divided, multiplied, or inverted while still

maintaining very low output skew. PLLs have a number of desirable properties that include the ability to multiply clock frequencies, correct clock duty cycles and cancel out clock distribution delays. Many PLL Based Clock Buffers have been brought to market in recent years to aid clock tree designs that require zero propagation delay from the input signal to the output. A completely integrated PLL allows alignment in both the phase and the frequency of the reference with an output. We will look at some of the more prevalent PLL based Clock buffers and their features in the following sections.

What is a PLL?

The basic PLL is a feedback system that receives an incoming oscillating signal and generates an output waveform that oscillates at the same frequency as the input signal. It is comprised of a phase/frequency detector, a filter, and a voltage-controlled oscillator as shown in Figure 2.6. In order for the PLL to align the REF input with an output, the output must be fed back to the input of the PLL. This feedback input (FB) is used as the alignment signal on which all other outputs are based.

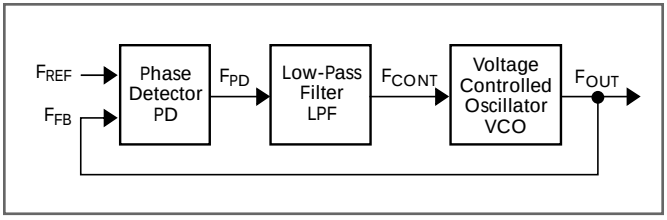


Figure 2.6 PLL Block Diagram

The Phase Frequency Detector (PD) evaluates the rising edge of the REF input with respect to the FB input. If the REF input occurs before the FB input indicating that the Voltage Controlled Oscillator (VCO) is running too slowly, the PD produces a Pump Up signal that lasts until the rising edge of the FB input. If the FB input occurs before the REF input, the PD produces a Pump Down signal that is triggered on the rising edge of the FB input and lasts until the rising edge of REF. This Pump Down pulse forces the VCO to run slower. In this way, the PD forces the VCO to run faster or slower based on the relationship of the REF and FB inputs. The output of the VCO is the internally generated oscillator waveform. The input voltage that controls the frequency of the VCO is a measure of the input frequency — as the input frequency changes then so does this voltage. The PLL is designed to operate within a limited band of frequencies. If the input frequency is outside this band then the circuit will not lock-on to the input signal and FREF and FOUT will be different. As long as FREF remains within the tracking range of the circuit then FOUT = FREF. However, if FREF moves out of range then the circuit goes out of lock, and once again the input and internal frequencies will be different. In the absence of a REF input, the condition of the output is device specific. For instance, with the loss of a reference input, the Cypress CY2308 ZDB will tri-state all outputs. However, the outputs of the CY7B991V operate at the devices

slowest speed while the outputs of the CY7B994V will run at their highest frequency. Therefore, the specifics of the device need to be known if the design will be placed in this condition. There are now buffers that support dual clock inputs if the loss of an input clock is expected.) The Filter converts these Pump Up and Pump Down signals into a single control voltage (FCONT) and its magnitude is dependent on the number of previous Pump Up and Pump Down pulses that have occurred. The range of the voltage produced by the filter is guaranteed to force the VCO into any frequency within the selected frequency range.

The Zero Delay Buffer

A zero delay buffer is a device that can fanout one clock signal into multiple clock signals with zero delay and very low skew between the outputs. This device is well suited for a variety of clock distribution applications requiring tight input-output and output-output skews. A simplified diagram of a zero delay buffer is shown in Figure 2.7. A zero delay buffer is built with a PLL that uses a reference input and a feedback input. The feedback input is driven by one of the outputs. The phase detector adjusts the output frequency of the VCO so that its two inputs have no phase or frequency difference. Since the PLL control loop includes one of the outputs and its load, it will dynamically compensate for the load placed on that output. This means that it will have zero delay from the input to the output that drives feedback independent of the loading on that output. Note that this is only the case for the output being monitored by the Feedback input and all other outputs have an input to output delay that is affected by the differences in the output loads. Please see the section "Lead or Lag Adjustment" for a discussion of this topic.

The Cypress Semiconductor CY2308 is a dual bank, general purpose zero delay buffer (ZDB) providing eight copies of a single input clock with zero delay from input to output and low skew between outputs. This popular buffer is designed for use in a variety of clock distribution applications and will be used throughout this book as the typical Zero Delay, PLL based buffer. The capability to externally connect the feedback path on the device provides skew-control and opens up opportunities for some interesting applications.

Lead or Lag Adjustments

Lead can be defined as the output of the buffer transitioning earlier in time than the input reference signal. It can also be viewed as negative delay. Lag, on the other hand, is the output clock transitioning later in time than the input and is a positive delay. To adjust the lead or lag of the outputs on the CY2308, we must understand the relationships between REF and FBK, and the relationship between the output driving FBK and the other outputs. First, we need to understand a few properties of Phase Locked Loops. The PLL senses the phase of the FBK pin at a threshold of $V_{DD/2}$ and compares it to the REF pin at the same $V_{DD/2}$ threshold. All the outputs start their transition at the same time (including the output driving FBK).

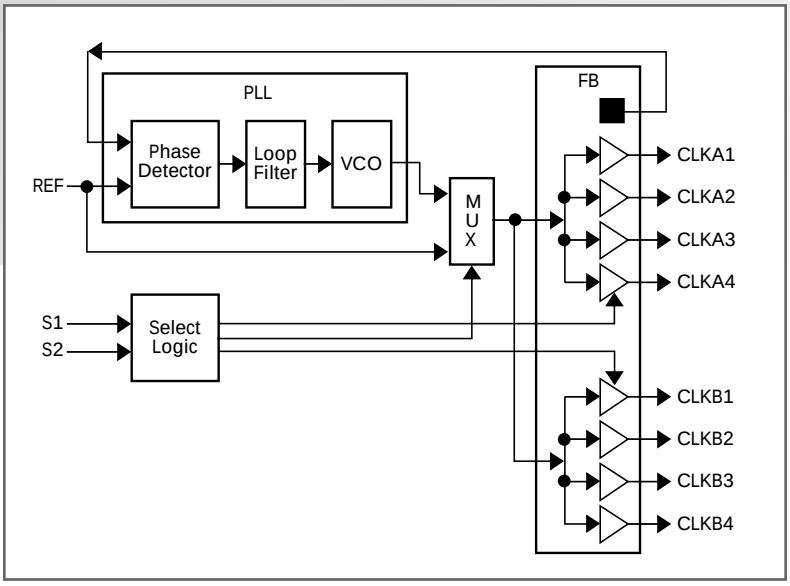


Figure 2.7 CY2308 PLL Clock Buffer

Changing the load on an output changes its rise time and therefore how long it takes the output to get to the $V_{DD/2}$ threshold. Using these properties to our advantage, we can then adjust the time when the outputs reach the $V_{DD/2}$ threshold relative to when the REF input reaches the $V_{DD/2}$ threshold. The output driving FBK however cannot be adjusted: it will always have zero delay from the REF input at $V_{DD/2}$. Loading the output used for the feedback more heavily can advance in time the other outputs. The other outputs can also be delayed in time by loading the output for the feedback more lightly than the other outputs. Figure 2.8 shows how many picoseconds the outputs are moved versus the difference in the loading between the feedback output and the other outputs. As a rough guideline, the adjustment is 50 ps/pF of loading difference. Note that the zero delay buffer will always adjust itself to keep the $V_{DD/2}$ point of the output at zero delay from the $V_{DD/2}$ point of the reference. If the application requires the outputs of the zero delay buffer to have zero delay from another output of the reference clock chip, the output of the clock chip that is driving the zero delay buffer must be loaded the same as the other outputs of the clock chip or the outputs of the zero delay buffer will be advanced/ delayed with reference to those other outputs.

Adding additional capacitance beyond 30pF is not suggested due to the possibility of degrading the clock edges and adding more jitter to the outputs.

Adjusting the lead or lag of the output skew with a capacitor has its benefits. However, it does have imperfections due to variations in the capacitor itself. For small delay adjustments, it is more precise to use trace delay that matches the needed lead or lag times. For larger amounts of delay, a programmable skew device such as the CY7B994V should be considered.

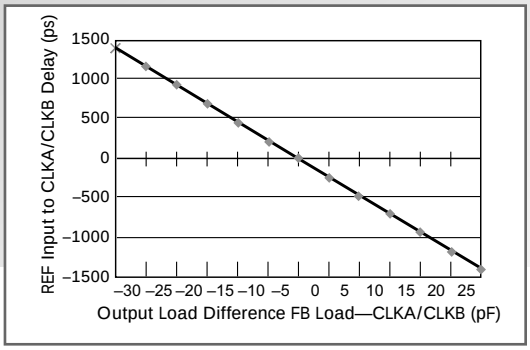


Figure 2.8 Lead/Lag Adjustments

Using External Feedback

Many ZDBs have an open external feedback path that is simply closed by driving any output into the FB pin for zero delay buffer operation. However, the feedback path can be used for other interesting applications. Using a discrete delay element in the feedback path will generate outputs that lead the input signal. Sometimes designs require some copies of a clock that are early compared to the remaining copies of the input clock. Figure 2.9 shows a circuit implementation to generate such early clocks using a Zero Delay Buffer.

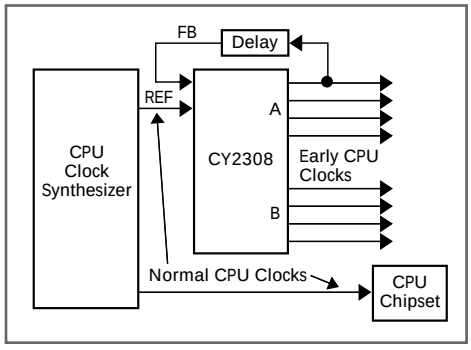


Figure 2.9 Early Clocks

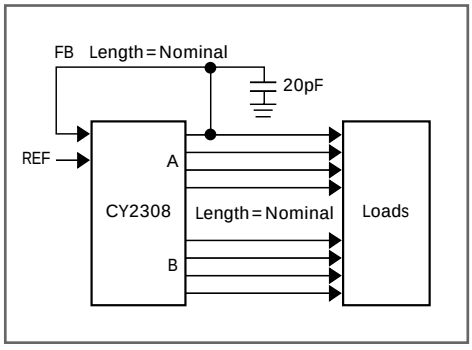


Figure 2.10 Trace Skew Adjustment

Another simple approach to lead or lag output clocks is to insert trace delay into the feedback path. The outputs of the buffer will lead the input by the amount of trace delay added in the feedback path. This approach provides a precise method for delay adjustment. Some designers will embed a very long trace into the board from an output pin to the feedback pin. At the ends of each trace segment, the designer places pads for zero ohm resistors. This allows for incremental additional of delay into the feedback path to align the outputs to the precise phase. Figure 2.10 shows an example where the feedback path is 5 inches shorter than the other output traces. This initially allows the remaining outputs to be later in time than the input. By placing a capacitor on the feedback line, the outputs can be moved forward in time.

Frequency Multiplier

By using an external divider in the feedback path we can also create a frequency multiplier out of a simple zero delay buffer. As shown in Figure 2.11, a /N divider in the feedback path will cause all outputs to run at a frequency which is N times the input frequency. Whatever the multiplication factor, input and output frequencies must be in the range of the PLL.

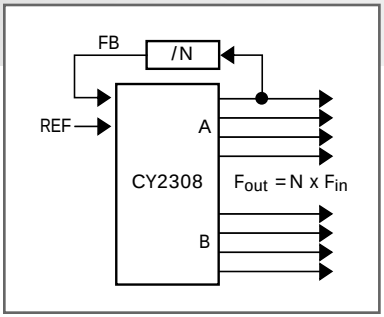


Figure 2.11 Frequency Multiplier

Specialty Application Clock Buffers

This chapter has focused primarily on general purpose clock drivers and zero delay buffers. However, there are a number of clock buffers available that have been designed to fit the requirements of particular subsystems that require precision clocking. For example, there are devices with frequencies to meet specific standards such as MPEG video, T1 communication, and a variety of other specifications. There are also clock components designed to address a particular application. An example of this is the SDRAM clock buffer.

DDR SDRAMs uses double data rate architecture to achieve high-speed operation. The double data rate architecture is essentially a 2n-prefetch architecture with an interface designed to transfer two data words per clock cycle at the I/O pins. A single read or write access for the DDR SDRAM consists of a single 2n-bit wide, one-clock-cycle data transfer at the internal DRAM core and two corresponding n-bit wide, one-half-clock-cycle data transfers at the I/O pins. Very specific clocking requirements have been specified by JEDEC to ensure proper operation of these DDR SDRAMs. DDR SDRAMs require high-speed SSTL-2 clocking solutions and the v857 has emerged to support these requirements. The v857 is offered by many vendors and is a high performance, low-skew, low-jitter phase-lock loop clock driver. It takes one pair of differential input signals and fans out to 10 pairs of differential output with low skew and low jitter at SSTL-2 Voltage levels. All data inputs and outputs are SSTL_2 level compatible with JEDEC standard for SSTL_2 and can drive up to 14 DDR SDRAM loads.

Differential Clocks

There are two common electrical methods to transmit data from a source to a destination. One method uses a “single-ended” signaling concept that makes use of two conductors between the transmitter and receiver. It uses a dedicated signal-line to send the signal from

transmitter to receiver and a common ground return shared by all signals. The other method is a differential signaling where true and complement forms of the signal are sent from the transmitter to the receiver. While this also uses two conductors between the transmitter and receiver, they now both carry active signals and neither is shared with other signals. Differential signaling uses twice as many signal lines as a single-ended signaling.

Differential signaling has a number of important advantages over the single-ended due to the ability of the differential receiver to reject any signal that is common to both lines. This ability is commonly referred to as common-mode noise rejection. The common mode rejection occurs because the receiver is only sensitive to a difference between the two inputs. When the two differential paths are closely linked, the noise will be apparent on both signals and rejected at the destination. With a single ended approach, the external noise will be apparent on the signal line itself. Because the differential signaling method rejects the common- mode noise signals, lower voltage levels can be used for reliable transmission of serial data. An additional benefit from the lower voltages used for differential signaling is reduced comparable power levels.

Differential PECL, LVPECL and LVDS clocks have become quite popular means of clocking high-speed logic in recent years.

PECL Buffers

As clock speeds increase above 100 MHz, noise immunity is of particular concern. PECL and LVPECL Clocks are particularly good for clocking high-speed devices. More and more devices are starting to require PECL clock inputs to drive their logic. Framers, SERDES, Switch Fabrics and FPGAs are among the latest devices supporting PECL and LVPECL inputs.

LVPECL stems from ECL (emitter coupled logic) but uses a positive rather than a negative supply voltage. It also uses 3.3 volt power supply rather than 5V. The ECL V_{DD} and V_{EE} pins have traditionally been powered from a –5.2V supply, V_{DD} being grounded and V_{EE} set at –5.2V where the intent is to achieve the lowest V_{DD} noise by grounding the V_{DD} pins. In more recent designs, however, ECL is often used with +5.0V instead of –5.2V and is commonly referred as PECL (V_{DD} set to +5.0V and V_{EE} tied to ground). Since V_{DD} noise is not a major concern, this permits the use of a standard logic supply.

As clock speeds rise beyond 100 MHz, the advantages of using ECL and PECL become more obvious. Most of these advantages involve the use of differential signal transmission. Differential signals are less susceptible to ground noise problems as all noise becomes common-mode. Single-ended CMOS is much more susceptible, since ground bounce and other noise affect logic thresholds, degrading noise immunity. Logic levels are less critical in differential signaling as the threshold can tolerate significant signal attenuation. Differential circuits also tend to generate less noise in the power supply. ECL is designed with termination resistors that allow high-frequency signals to propagate with minimal overshoot and reflection.

A new series of Zero Delay Buffers are becoming available that support LVPECL. The LVPECL differential driver is designed for low-voltage, high frequency operation to over 400 MHz. It significantly reduces the transient switching noise and power dissipation when compared to conventional single-ended drivers.

LVDS Buffers

LVDS (Low Voltage Differential Signaling) has become a popular means of transporting binary data across boards and backplanes in recent years as numerous low cost data buffers and FPGAs have begun to support this transmission scheme. To achieve high data rates and keep power requirements low, LVDS uses a differential voltage swing of only 350 mV (typical, in point-to-point applications). Furthermore, the LVDS CMOS current-mode driver design greatly reduces quiescent power supply requirements. LVDS Data and Clock Buffers have rapidly emerged to support this standard as defined in TIA/EIA-644.

Spread Aware Buffers

The use of Spread Spectrum Timing (SST) technology has been popular in the motherboard and printer markets for some time. It is being used in virtually all motherboard designs using chipsets that support a greater than 100-MHz busses. Spread spectrum timing signals are used in a variety of applications including PCI, CPU, and memory buses. Nearly all motherboard chipset vendors are designing their parts to work with spread spectrum timing signals. While the fast-paced motherboard market has quickly adopted the technology, it has also been embraced by other markets. The technology was developed solely for the purpose of reducing peak EMI.

Spread spectrum timing is a very effective tool for reducing peak EMI and may be easily integrated into many different systems without affecting other circuit elements. The one type of circuit element that may cause a timing problem when driven by a Spread Spectrum timing signal is a downstream PLL. A downstream PLL is a device that receives a reference timing signal from another PLL-based device, including those that utilize SST technology. “Downstream” may also apply to PLLs within Clock and Data Recovery (CDR) circuits. In a CDR application, the ability to properly track a modulated serial datastream is critical for clock extraction. For this reason, tracking skew is very important in downstream PLL applications. A zero delay buffer used on a memory module to buffer the clock signal and provide the correct timing to latch the data could be considered as a likely downstream PLL. Although these devices may work properly with very stable reference inputs, if they cannot track a dynamically changing input signal then the output timing signals will not be synchronized to the system timing. Clock Buffers that can track the variations of a Spread Spectrum input are said to be “Spread Aware”.

Spread Aware Zero Delay Buffers are specifically designed to receive a spread spectrum modulated input signal. The PLL characteristics of Spread Aware devices will track the frequency modulation on the input signal with minimal accumulated tracking skew. Therefore, spread spectrum modulation present on the ZDB input signal would also be

present on the output signals. This will reduce the EMI emissions of the system. In addition, since the PLL tracking skew has been minimized, the system designer will have the benefit of the greatest possible timing margins.

Which One to Choose?

The clock circuit is usually critical to the operation of the system. If the clock circuit fails then the system fails. Because of this, the proper selection of the clock driver/buffer is usually critical to the success or failure of the design. When selecting the clock driver/buffer there are several parameters and characteristics for which the designer can watch to ensure the correct and reliable operation of the system. Since clock drivers tend to operate at high frequencies, it is important to ensure that the clock driver has low power dissipation. Unlike a buffer or latch that changes state only when one of the inputs changes, every output on the clock driver changes state every clock cycle at the fastest rate available in the system. This means that the clock driver is most likely switching more power in a smaller package than any other component in the system. While heat sinks and other cooling methods will help, it is best to start with a clock buffer/driver that inherently dissipates low power.

Choosing between a clock buffer or PLL is usually dependent upon the need for zero delay in the driver or the need to increase the clock frequency. If zero delay or increased frequency is needed, a PLL is the obvious choice. An advantage of a PLL over a buffer is the ability to correct the duty cycle in the clock line if there is distortion.

Conclusion

As system clocking speeds increase, the issues of skew and noise begin to receive primary consideration. Increasing the clock frequencies and requiring tighter tolerances in the clocking circuits will achieve future gains in system performance. Low skew clock buffers and PLL clock drivers will assist the designer in meeting the system requirements for speed, skew, and noise but, the clock circuit must be designed as a clocking system with consideration given to all aspects of the clock distribution network including driver, receivers, transmission lines, and signal routing. If the designer is aware of the problems that can develop, the difficulties can be avoided. The best method to identify the clock tolerances is to create a timing budget. The timing budget identifies the effects that each element of the tree has on the clock signals. The next Chapter discusses timing budget in detail.

3
Timing Budget

In order to ensure proper operation of a synchronous system, all of the timing elements must fit within the given cycle. When the sum of the timing elements exceeds the available time, the system will malfunction. It is therefore necessary to create a timing budget that identifies all of the factors that require a portion of the clock period and evaluate their influence. This allows the designer to understand how fast the design can operate and how much margin there is in the system. Chapter three focuses on the elements that contribute to the timing budget and highlights a newer, more precise way to accumulate the timing.

There are a variety of factors that add into the timing budget. Many of these are obvious but some are a bit more subtle. To illustrate, we will build a timing model and add the associated timing elements piece by piece. Figure 3.1 shows a simplified design with the basic components of a synchronous system. Register A drives an output QA on every rising edge of CLOCK. QA attaches to a trace with a delay of $t_{FLIGHT1}$ and is the input to some amount of combinatorial logic. The delay associated with this logic is represented as t_{LOGIC} and can be thought of as the "work" that takes place in the cycle. The result of this logic again travels through a trace with a delay of $t_{FLIGHT2}$ and drives the resulting register. Register B is also clocked with the same CLOCK as Register A. The data from the combinatorial logic must arrive prior to the clock (t_{SETUP}) and must be held past the clock (t_{HOLD}) in order for the register to operate properly.

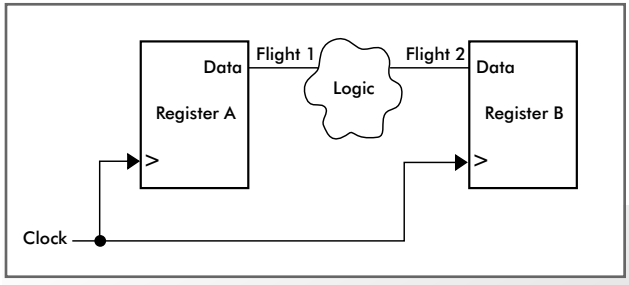


Figure 3.1 Simple Synchronous System

A Simple Timing Budget

Using this simplified model, we can begin to create a simple timing budget. The timing budget uses the following components:

- t_{CLK} to Q: Delay from CLOCK rising to Q_A valid
- $t_{FLIGHT1}$: Time taken to send Q_A output to Logic input on a printed circuit board
- t_{LOGIC}: Time taken to perform logic operations
- $t_{FLIGHT2}$: Time taken to send logic output to Register B input on a printed circuit board
- t_{SETUP}: Time when Data at Register B must be valid before CLOCK rising
- t_{HOLD}: Time when Data at Register B must be valid after CLOCK rising

This breakdown of timing elements above assumes the clock signal has no jitter and that it reliably makes a constant period. However, most clock sources contain some amount of jitter. This not only includes clock generators but applies to oscillators as well. Therefore, in addition to the above timing elements, the timing budget analysis must be concerned with the variance of the clock signal from one cycle to the next.

For example, if the first clock edge arrives later than nominal but the second clock edge arrives earlier than nominal, the result shortens the effective time by which all activities must complete. To allow for the worst case in such a scenario designers must allow for the maximum cycle-to-cycle jitter of the clock within the timing budget. Therefore, the following parameter is added to the simplified model above.

- $t_{REFJitter}$: Maximum Cycle-to-Cycle Jitter on Reference Clock

Distributed Clocks

It is often necessary to operate a large number of logic blocks, or subsystems, in the same clock domain. In order not to overload the clock signal, designers must create replicas of the reference clock using a buffer.

As discussed in the previous chapter, there are two main categories of clock buffers:

1. Clock Buffers
2. Zero Delay Buffers (ZDBs)

Figure 3.2 shows the simplified system that uses a fan-out clock buffer to distribute its reference clock to four elements in the same clock domain. Although the buffer creates four copies of its input signal, the four outputs do not appear at the signal pins at precisely the same time. Because of different lengths of bond wires in a package and variances in silicon processes, some outputs in a buffer will arrive at their output pins sooner than others. The difference in the output timing is pin-to-pin skew. By inserting the fan-out buffer to the design, additional timing elements must be added to complete the timing budget.

t_{PD}: Maximum propagation delay for the input clock signal to make it through the buffer

t_{SKEW}: Maximum pin-to-pin skew of the buffer

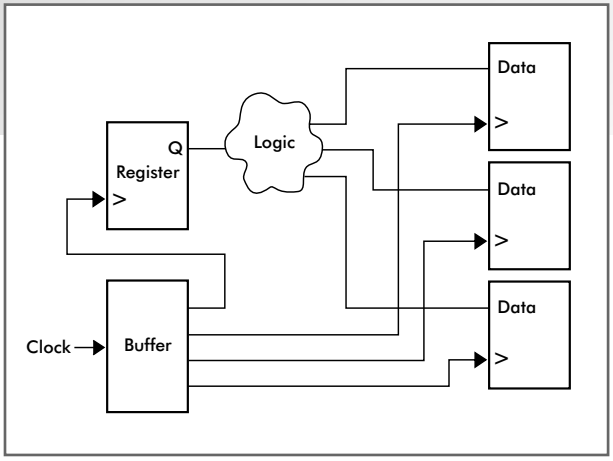


Figure 3.2 Clock Buffer Distribution

Zero Delay Buffers

The consequence of using a fan-out buffer is that the propagation delay (t_{PD}) can be very long and may leave insufficient time to complete all of the other tasks within a time period. Thus, designers can use zero delay buffers (ZDBs) when they need to duplicate and send the reference clock source to more than one destination.

A ZDB uses phase-locked loop (PLL) technology to replicate an input signal. It can also generate an output signal that is an integer multiple or dividend of the input while maintaining phase alignment. For example, the input reference could be 33.3 MHz and the resulting output could be 100 MHz or conversely, the input could be 100 MHz and the output 33.3 MHz.

By maintaining phase alignment with the input, ZDBs alleviate the large propagation delay of a fan-out buffer. However, they bring a new set of considerations in meeting the timing budget.

First, the design must consider the cycle-to-cycle jitter of the ZDB instead of the reference input. Cycle-to-cycle jitter has very high frequency components and most ZDBs will filter out input jitter that is higher than a few megahertz. Consequently, $t_{REFJitter}$ no longer affects the timing budget and is replaced by the cycle-to-cycle jitter of the ZDB ($t_{Jitterc-c}$).

Second, Zero Delay Buffers do not offer exactly zero offset between the input and the output. That difference is called either phase error or input-to-output skew. For ZDBs that offer an external feedback, this parameter is partially under the designer's control. By adjusting the trace length of the feedback signal, designers can affect the timing relations

between input and output. The delay will still be represented as t_{PD} in the timing budget, but it will have a smaller variation than a non-PLL based buffer. The delay can also be positive or negative and is discussed in more detail in the next sections.

Clock Timing Budget

When constructing the overall timing budget, the arrival of the clock edges is a key component. It is often the most complex portion as well. To properly understand the margins of the clock circuit, a window needs to be created that shows the earliest arriving clock and the latest arriving clock. These values are the most negative and most positive phase difference with respect to the reference clock. Figure 3.3 illustrates the timing window. The window is marked by the edge of the earliest clock (T_{FIRST}) and the edge of the latest clock (T_{LAST}).

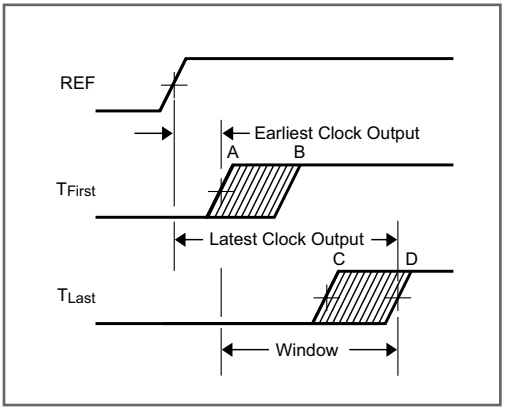


Figure 3.3 Timing Window

With this window, we can determine the margins at the device that is to be clocked. To illustrate this in a system environment, suppose we have a circuit that is shown in Figure 3.4. A 25 MHz oscillator sources a CY26114 clock generator. The generator creates two copies of a 100 MHz clock and two copies of a 50 MHz clock and each clock drives a zero delay buffer. A CY2509 ZDB is used for the 100 MHz signals while a CY2309 is used for the 50 MHz clocks. This is a typical system where different types of ZDBs may be used to match speed and function.

Although half of the outputs are 50 MHz and the other half are 100 MHz, this particular system requires that all outputs be phase aligned to the each other. A reference clock is used to evaluate how far in time each clock phase occurs. In this case, the reference is the output of the clock generator since any delays and skew prior to this point have no effect on the outputs of the ZDBs. To create the timing window, the parameters for skew, delay, and jitter that were identified earlier need to be added as shown in Figure 3.5.

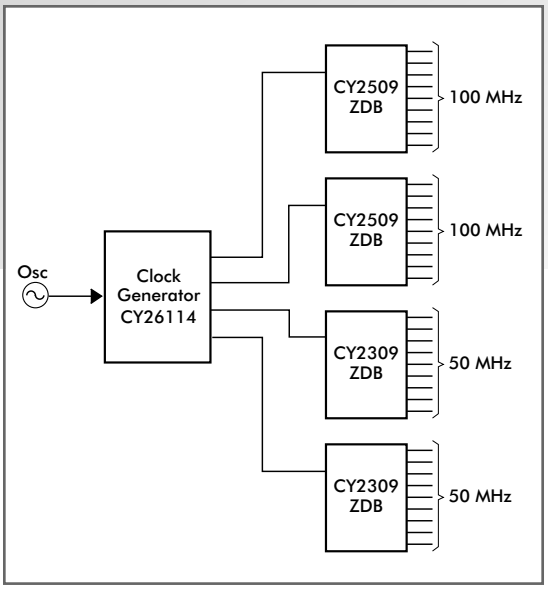


Figure 3.4 Typical Clock Tree

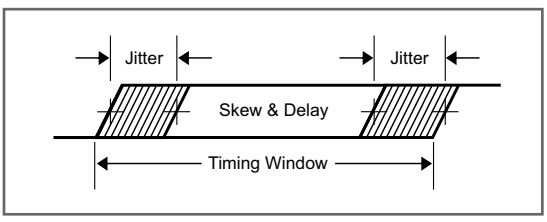


Figure 3.5 Timing Window Components

For the components in this design, there are several parameters that are required from their respective datasheets.

CY26114	CY2509	CY2309
Jitterc-c: 200 ps	Jitterc-c: 100 ps	Jitterc-c: 200 ps
Skew: 250 ps	Skew: 250 ps	Skew: 250 ps
Delay: N/A	Delay: ± 350 ps	Delay: ± 350 ps

The timing window measures the earliest clock to the latest clock at the destination. For each of the outputs, the window is generated as in Figure 3.6. The window represents the earliest clock to the latest clock at all of the destination devices. To create the window, all of the timing elements from the reference clock to the destination clocks need to be added. A window should be created for each part and the widest opening should be chosen. In this case, the CY2309 has the largest jitter, delay and skew components so we will show the window for those outputs. Begin by adding all of the fixed skews in the clock paths such as pin-to-pin skew. Some datasheets will specify a bank-to-bank skew and a part-to-

part skew. Use whichever applies to the clock tree configuration. Notice that the window will widen if there is trace skew in the path between the buffers and the loads or between the clock generator and the buffers. This model assumes they are equally matched in length and there is no variation. The value of 0 ps is used in the window. Then add any other forms of delay associated with the components. This includes any input to output variations. Although PLLs "zero" the part delay, there is a delay tolerance and it needs to be included. Finally, the jitter for all components is included at both ends of the window.

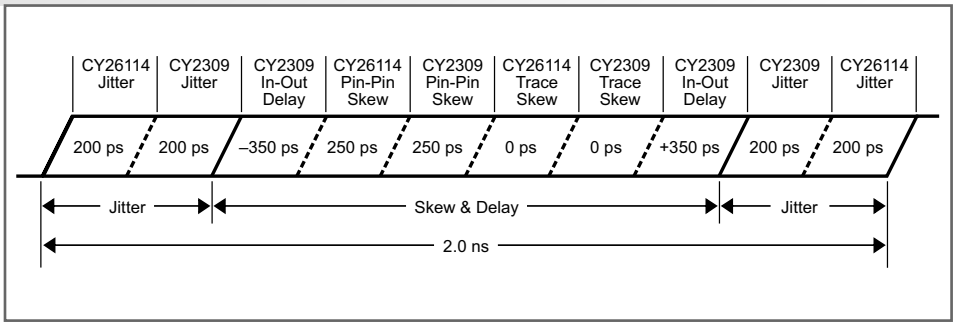


Figure 3.6 Datasheet Timing Window

From this example, the window is shown to be 2.0 ns. With the system operating at 100 MHz, the window represents a large variation. However, these are all worst-case numbers and there are hefty margins associated with each value.

Measured Data

To evaluate how datasheet values match against actual components, the design in Figure 3.4 was fabricated and tested. Each clock output was measured for skew, delay, and jitter. The maximum positive phase delay, or in other terms the latest clock with respect to the reference was measured to be +571 ps. The maximum negative phase delay, that is the earliest arriving clock is measured as –346 ps. These numbers include the maximum jitter measured of 120 ps. Figure 3.7 shows the variations at each clock buffer and the total timing window. Each box in this figure represents the ZDBs used in the clock tree of figure 3.4. Each output of the ZDB is measured with respect to the reference clock (from the CY26114) and the earliest and latest delay is recorded for each buffer. Notice that some of the clocks precede the reference while others are later in time. The earliest and latest clock on any of the outputs bound the timing window.

The measured timing window yields a result of 917 ps while the calculated timing window using datasheet parameters was 2 ns! This is a significant difference. Because the simple summation of datasheet values produces a very conservative result, some clock buffer manufacturers are including a new parameter that combines the jitter, delay, and skew. One of the terms that is used for this value is Total Timing Budget (TTB™).

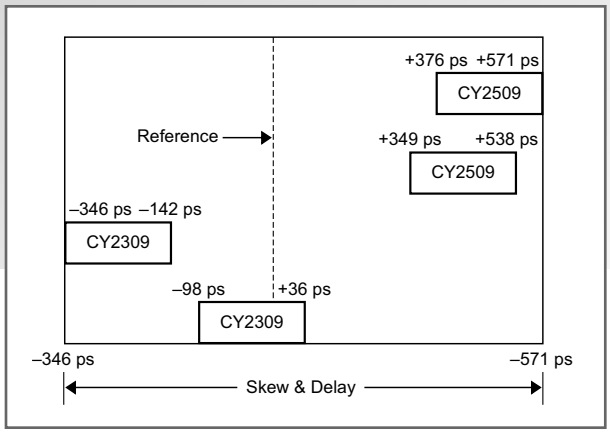


Figure 3.7 Measured Timing Window

Total Timing Budget (TTB™) Window

Since phase locked loop devices became commercially available over ten years ago, the method by which data sheets report jitter, skew, and other performance parameters have been the same. In the meantime, operating frequencies have increased considerably. As seen in the previous example, the estimated budget using these numbers can become very large.

Minimizing the Window

The TTB parameter includes the delay, skew and jitter components specific to a clock buffer and provides one number with which to work. TTB can be defined as the maximum window that all transitions of any output clock of a device will occur with respect to its input reference clock. This substantially reduces the task of calculating the timing budget numbers. It also avoids the compounding of guard bands associated with the individual numbers. It should be noted that TTB doesn't replace the traditional numbers as they may define other key aspects needed during the circuit design.

The TTB parameter is guaranteed over the device's datasheet temperature, voltage, frequency, and input clock's rise time range. It also includes the manufacturer's process variables and hence is an absolute maximum guaranteed window for all conditions that are stated in the data sheet.

Some datasheets provide TTB values for worst-case conditions and frequencies. However, this too may add more guardband than necessary for a design operating at a specific frequency whereas the specification will include the complete operating range. Thus, TTB values may be published not only for the maximum frequency but also for specific and common operating rates. For instance, a Zero Delay Buffer that operates up to 200 MHz may have a published TTB number for the 200 MHz, and for 100 MHz, 66.6 MHz and 33.3 MHz, as these are common frequencies used in designs. This provides more accurate information to the designer for the timing budget.

To illustrate the benefit of TTB, let's analyze the parameters associated with a ZDB that is used in a timing budget. The three factors are pin-to-pin skew, input-to-output-delay (phase error), and cycle-to-cycle jitter. Figure 3.8 shows these three parameters for a typical ZDB.

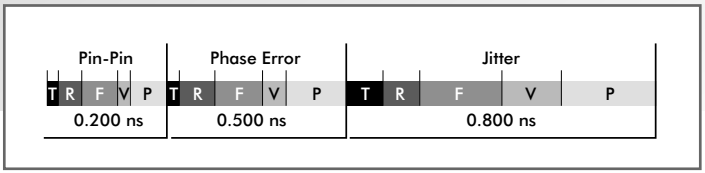


Figure 3.8 ZDB Traditional Timing Budget

In the figure, T represents the contribution due to temperature, R is the input rise time variation, F is the frequency of operation, V is the voltage fluctuation and P is the process variation. The amount of time associated with these parameters in this figure is 1.5 ns, which includes a guard band for each individual parameter.

For the same component, the TTB window is much smaller as shown in Figure 3.9.

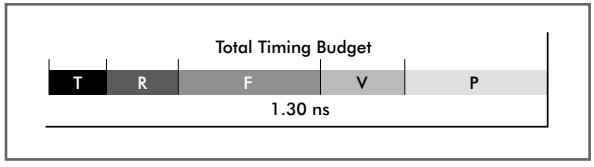


Figure 3.9 ZDB TTB Timing Budget

The TTB number, which includes all of the variations as before, specifies 1.3 ns instead of the 1.5 ns. This gives 200 ps back to the designer for timing margin. This TTB number is for the full operating frequency range of the buffer. However, if the system is operating at a fixed rate of 133 MHz, a different, more precise TTB number may be provided. This allows the designer to use a specific TTB number for 133 MHz to further enhance the budget. Figure 3.10 shows this time line. Notice that the value F does not appear as it represents the frequency variation that is now fixed.

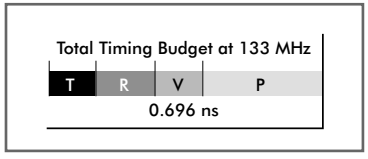


Figure 3.10 ZDB TTB Timing Budget at Frequency

This has dramatically improved the variables used in the timing budget numbers. By using the TTB at 133 MHz, the timing budget parameters for the clock buffer enhanced the budget from 1.5 ns to 696 ps, which is a savings of over 800 ps. This is a significant savings for a system with a cycle time of 7.5 ns.

TTB Characterization Data

Creating a TTB number for a component follows the normal path of part characterization. The difference between a TTB characterization and a standard (non-TTB) characterization is that TTB requires additional measurements under a variety of conditions. Components from each of the process corners are tested in a variety of conditions that include: temperature, voltage, input clock condition, feedback clock condition and frequency. The numbers are tabulated and the TTB parameter is generated.

The following four figures show the effects temperature, voltage, frequency, and rise time of the feedback signal have on the budget windows. Notice in particular the delay difference in the frequency range. Using the delays associated with the exact system frequency can yield much better timing margins. This is true using the TTB method or using the traditional timing budget method. Note that these graphs illustrate the effect these parameters have on delay, however, the amount of variation is device specific.

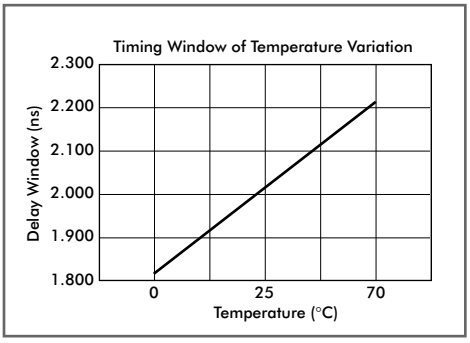


Figure 3.11 Temperature Variation

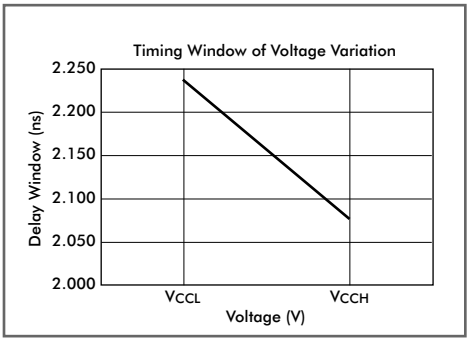


Figure 3.12 Voltage Variation

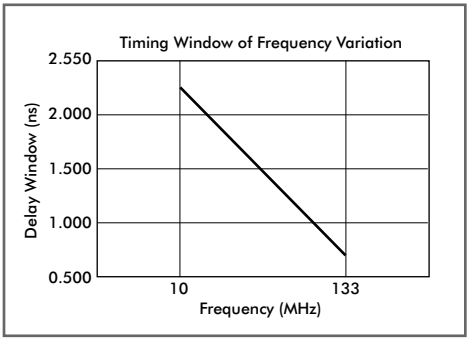


Figure 3.13 Frequency Variation

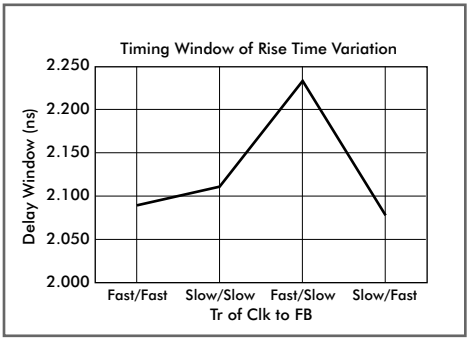


Figure 3.14 Feedback Rise-Time Variation

Conclusion

Creating a timing topology and allocating the timing budget is one of the fundamental efforts in system architecture for complex synchronous systems. This chapter illustrated some of the basic considerations on how selection of clock distribution components can impact timing budget considerations. Delay, skew, and jitter are all key elements that must be addressed

Datasheets that list the timing parameters individually are not sufficient to meet a very tight timing budget. Instead, a parameter that represents the part's total timing budget should be used. This number better reflects the operation of the device without excessive guard banding. In the absence of this number, designers should work closely with their timing solutions supplier to understand the projected performance of each clock device.

4

Clock Jitter

Jitter plays an important role when designing clock generation and distribution circuits. This is especially true when using PLL based buffers. Excess jitter can rob a design of its precious cycle time or cause data to be latched incorrectly. Understanding the concepts of jitter and knowing its definition can avoid these problems. There are also different ways to measure jitter and often they are confused with one another. This can lead to erroneous calculations and cause a system to malfunction. To further confuse the matter, jitter can be expressed in many different ways. Component datasheets often specify jitter in terms of absolute, peak-to-peak, RMS, one Sigma, and UI. This chapter focuses on the definitions of the different types of jitter and how they are measured.

Fundamentally, clock jitter can be defined as the deviation in a clock's output transition from its ideal position. Before discussing the specifics of jitter in clocks for high-speed digital designs, its sources in general should be identified. Jitter can be divided into two essential types: Random Jitter and non-random or Deterministic Jitter. Total jitter is equal to the sum of Deterministic Jitter and Random Jitter. But these two types of jitter have very different behaviors. It is helpful to understand the characteristic of each when evaluating jitter within a system.

Deterministic Jitter

Deterministic Jitter (DJ) is non random and bounded. This type of jitter can be traced to a specific source. Deterministic jitter is due to signal noise, crosstalk, power supplies, and other similar sources. High values of this type of jitter are usually an indication of a problem within the design. Deterministic Jitter is typically non-Gaussian in nature and tends to grow by fixed amounts. There are four categories that contribute to deterministic jitter: duty cycle distortion, data dependent, sinusoidal, and uncorrelated bounded. Some systems, when describing jitter, will specify the sinusoidal jitter (SJ) component separately.

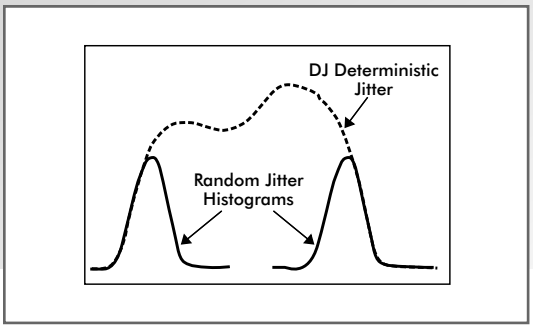


Figure 4.1 Total Jitter

Duty Cycle Distortion and Pulse Width Distortion are different names for the same thing. This type of jitter is also referred as half-period jitter. It is the difference in the width of the pulse with its output low versus the width of the pulse when the output is high. Ideally, these two widths are equal. When they are not, the crossing points for signal transitions (low-to-high and high-to-low) can be different. This is most evident when viewing a signal with an eye diagram.

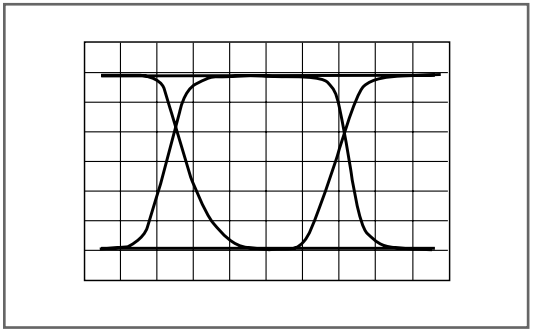


Figure 4.2 Duty Cycle Distortions

Data Dependent Jitter and Inter-Symbol Interference describe the same condition but viewed from different perspectives. Data Dependent jitter is measured in the time domain while Inter-Symbol Interference is analyzed in the frequency domain. Data Dependant jitter appears in a system when a pattern changes from a fixed stream of pulses to a unique bit position. The rising and falling edges are altered from one position to another based on a data stream. An example of this condition is a regular clocking waveform changing into double width pulses and back again. From the frequency domain perspective, its can be viewed as pulse spreading due to bandwidth limitations. This category of jitter is normally found in datastreams and is not typical in digital discrete clocks.

Sinusoidal Jitter alters the position of the rising and falling edges of the waveform by varying amounts. The amount of the timing displacement follows a regular sinusoidal pattern.

Uncorrelated bounded jitter comprises the remaining deterministic jitter not categorized by any of the three previous types. It is uncorrelated in that there is no bit pattern dependency but bounded as it has a fixed limit. Power supply noise and crosstalk are contributors of this type of jitter.

Random Jitter

Random jitter (RJ) is often due to a large number of forces in nature and each of which follows uniform statistical probability. Random jitter follows Gaussian probability densities and its limits are unbounded. Because it is statistical, it forms a bell shaped curve with the tails extending towards infinity. In theory, its maximum values can be limitless and therefore Random Jitter is often bounded using standard deviations. In practice, arbitrary limits are imposed and peak-to-peak values are available. Random jitter normally comes from elements within the environment. It can be caused by thermal noise, radiation, semiconductor crystalline structures, and a variety of other indistinguishable components.

Gaussian Statistics

When many independent random factors interact and accumulate, data will follow a distribution called the Gaussian probability distribution function. This distribution is also called a Normal distribution. The Gaussian distribution has some special mathematical properties that form the basis of many statistical measurements.

A Gaussian curve follows a bell shaped distribution. The mean of the samples are aligned in the middle and hence the highest point in the curve. This is also the point at which the horizontal axis marks its zero origin. As mentioned earlier, the tails of the curve extend towards infinity. Since the data is unbounded in theory, a set of windows is defined to represent various amounts of data.

The standard deviation, or one sigma σ, is defined as containing 68.26% of all measurements on one side of the mean. Two sigma 2σ contains 95.4% and four sigma contains 99.994% of all measurements on a side of the mean. For pure Gaussian mathematics, all measurements are possible. Although, theoretically Gaussian jitter has no bounds, the probability of an event occurring very far from the median becomes extremely remote. In practice, there are bounds that do not exceed fourteen sigma in Gaussian systems. In general, plus or minus six sigma (contains 99.9999999% of all sampled data) is the maximum deviation from the mean to use for calculating peak-to-peak jitter for a Gaussian distribution of jitter. But there are some specific applications that use a higher sigma value.

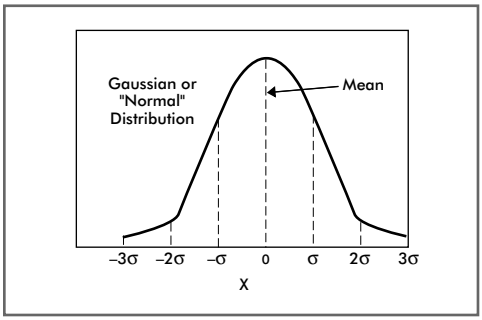


Figure 4.3 Gaussian Distribution

Jitter Mathematics

The Gaussian distribution shows the samples graphically but there are several mathematical equations that need to be highlighted. Although test measurement equipment will provide these values, it is helpful to understand how they are computed and their meaning. This can be crucial when extracting jitter parameters from a device datasheet and relating it to the design requirements.

Since jitter in a clock is measured on an edge, or event in general terms, it can be represented as E. The time at which the signal rises is therefore t_E. The clock signal will have many rising edges and hence any occurrence of n events can be represented as E [n], which will occur at times $t_E[n]$. Using these representations, instantaneous jitter between an ideal edge and the actual measured edge can be expressed as:

$$J[n] = t_E[n]_{IDEAL} - t_E[n]_{ACTUAL}$$

The IDEAL edge of a signal can easily be calculated from the period of the clock. This can be an adjacent edge or n edges away in time. Using n=0 for the reference edge, the IDEAL edge at time n is equal to the following:

$$tE[n]IDEAL = t_E[0] + n/fCLOCK$$

With the Gaussian distribution, sigma (or root-mean-square average) is often used to represent a portion of the data. This parameter is often specified in device datasheets as $Jitter_{RMS}$. To calculate the RMS value, the mean must first be derived. Given N as the total number of samples in the measurement, the mean jitter amount can be expressed as:

$$J_{MEAN} = \frac{1}{N} \sum_{N=1}^{N} j[n]$$

Using the mean, the one sigma, or RMS value from the mean is computed.

$$J_{ONE\ SIGMA}\sigma = \sqrt{\frac{1}{N-1} \sum_{N=1}^{N} (j[n] - J_{MEAN})^2}$$

Another common parameter specified in device datasheets is the peak-to-peak jitter. Graphically, it is the width of the base of the distribution. Mathematically, peak-to-peak jitter is derived from subtracting the minimum jitter in the sample set from the maximum jitter.

$$J_{PEAK\text{-}TO\text{-}PEAK} = max\{J[n]\} - min\{J[n]\}$$

Measuring the Jitter

Due to the speeds of the high-speed digital signals, it is important to measure jitter with the correct equipment. Often, engineers will use oscilloscopes with persistence to measure jitter. Although this provides a rough measurement it is very inaccurate and suffers from trigger instability.

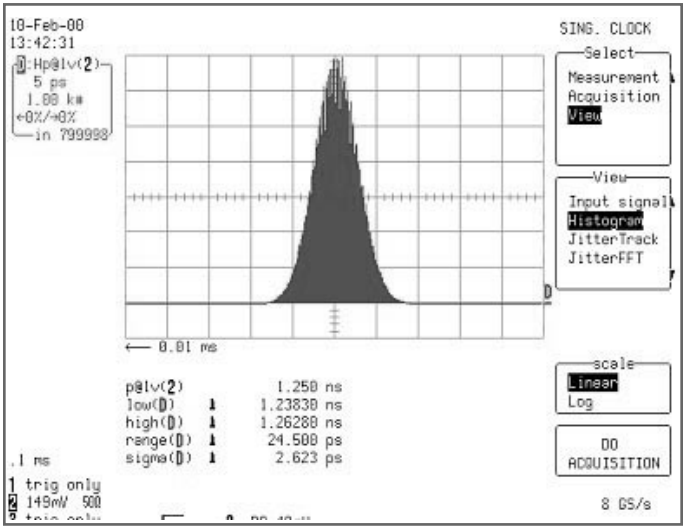


Figure 4.4 Jitter Histogram

It is, therefore, required that specialized equipment be used to measure jitter. These are called Timing Interval Analyzers (TIAs) and specialized Digital Storage Oscilloscopes. A variety of jitter test equipment is available from companies such as LeCroy, Tektronix, and Agilent. These instruments not only allow the precise capture of data, they incorporate the mathematical analysis and graphical displays that easily extract jitter values.

Jitter is often displayed with the use of histograms. The vertical axis is the number of samples and the horizontal axis is time. The display shown in Figure 4.4 follows a Gaussian distribution and therefore indicates Random Jitter is the primary source. Notice that the tails do not extend to infinity and have fixed values because the sample size is finite. The mean jitter value is in the middle of the sample set and standard deviation is used as a measure of the dispersion about the mean. RMS, one sigma, and standard deviation all have equivalent meanings. The peak-to-peak jitter is the difference between the highest and lowest value captured and is represented as the width of the base.

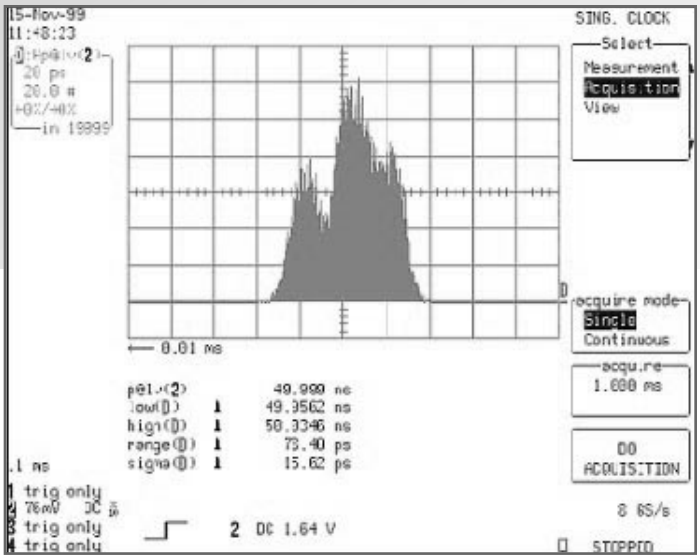


Figure 4.5 Bimodal Distribution

When Deterministic jitter is coupled with the random jitter component, a non-Gaussian distribution with multiple nodes is created. Depending on the components, it may be bimodal, meaning two peaks, or it can be multi-modal and thus containing several peaks. Each peak will form a Gaussian distribution in itself.

Application Jitter

These basic principles can be applied further to specify and quantify jitter in high-speed digital clocks. Jitter, in its application sense, can be defined in three ways: Cycle-to-cycle jitter, Period jitter, and Long Term jitter.

Cycle-to-Cycle Jitter

Cycle-to-cycle jitter is the change in a clock's output transition from its corresponding position in the previous cycle. This is also sometimes referred to as short term jitter. This type of jitter is the most difficult to measure and usually requires a Timing Interval Analyzer (TIA) or some other specialized jitter test equipment. Figure 4.6 shows a graphical representation of cycle-to-cycle jitter. J_1 is the jitter value measured by subtracting the cycle time t_2 from cycle time t_1. The next jitter value, J_2, is the value measured by subtracting the cycle time t_3 from cycle time t_2. The maximum of the J values measured over time, typically ten thousand cycles, is the maximum cycle-to-cycle jitter.

When designing clock and distribution circuits, for synchronous systems or processor based systems where a PLL may be downstream, cycle-to-cycle jitter is important. PLLs may not be able to track this high frequency jitter and timing skew may result.

There are several different definitions used to describe cycle-to-cycle jitter that may be seen in different datasheets. Cycle-to-cycle jitter is also known as adjacent cycle jitter. Often,

cycle-to-cycle jitter will be provided as a single number that correlates to the largest J value recorded from all samples. This is the maximum variation from one cycle to the next and can lead or lag in time from the previous cycle. Therefore, this number is sometimes expressed in +/- units.

Sometimes a device's datasheet will specify a peak-to-peak value for the cycle-to-cycle jitter. This represents the difference of the minimum cycle and the maximum cycle throughout the sample set. Therefore, this number is greater than the normal cycle-to-cycle jitter and is larger than what is expected between adjacent cycles.

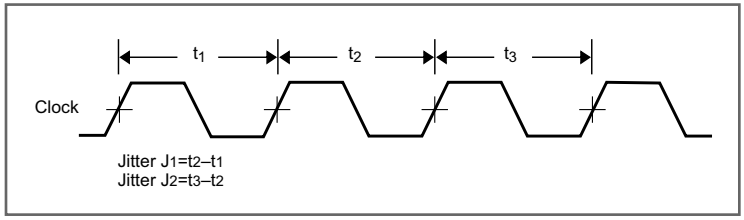


Figure 4.6 Cycle-to-Cycle Jitter

Period Jitter

Period jitter measures the maximum change in a clock's output transition from its ideal position. Figure 4.7 shows a clock waveform where the first rising edge is fixed and the next rising edge is measured. This second edge may lead or lag the ideal cycle width. The maximum of this value measured over time, typically ten thousand cycles, is the maximum period jitter.

Period jitter can impact the performance of a synchronous system by cutting into the overall timing margin budget. It can affect cycle time as well as data set-up and hold times. There are several ways to describe period jitter.

Many datasheets will also label Period jitter as Absolute jitter. These values and peak-to-peak Jitter all describe the same measurement. The number is the difference between the earliest and the latest signal. Referring to a Gaussain distribution, this is measuring the width of the base of the peak. It can also be thought of, referring to Figure 4.7, the latest t_1 minus the earliest t_1 recorded in the sample set.

Another method for indicating Period jitter is using a form of an average. The methods of root-mean-square (RMS), standard deviation, and one sigma all describe a subset of the peak-to-peak period jitter and mean the same thing. Again, from the Gaussain distribution, one sigma, or one standard deviation, is 68.26% of the sample size. Jitter expressed in any of these terms uses the jitter measured from this subset of data collected. By definition, this value will be less than the peak-to-peak amount.

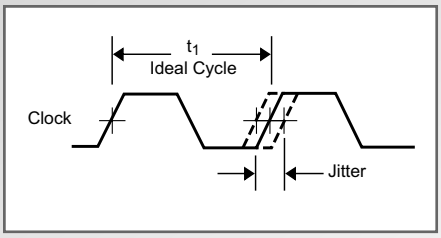


Figure 4.7 Period Jitter

Long-Term Jitter

Long-Term jitter measures the maximum change in a clock's output transition from its ideal position over many cycles. This type of jitter measurement generally applies to a few specific applications. The term "many" depends on the application and the frequency of the clock. For PC motherboards and graphics applications, "many" usually refers to a period of 10-20 microseconds. For other applications it will be different. Figure 4.8 shows a graphical description of long-term jitter. Cycle 0 begins with the rising edge of the clock. Long term jitter is the amount of deviation in cycle N of the ideal clock edge. Cycle N's actual clock edge may lead or lag the ideal clock edge.

A classic example of a system affected by long-term jitter is a graphics card driving a display. Assume that a pixel of data is meant for a pixel at coordinates (10,24) on the screen. Because of a clock with excessive long-term jitter, this data may be driven at a pixel with coordinates (11,28) on the screen. Over an extended period of time, the data meant for pixel (10,24) may be driving a pixel far away from its ideal position. Since this shift is normally consistent over all pixels, the overall effect of the jittery clock is to cause an image shift from its ideal display position. This effect is sometimes called "running" of the screen.

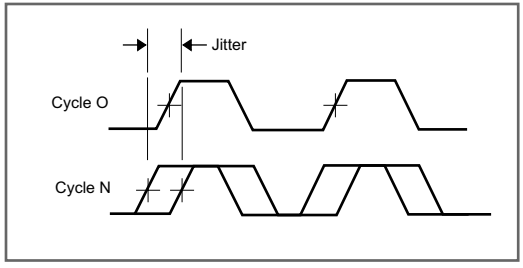


Figure 4.8 Long-Term Jitter

In most general applications using clock buffers and generators, jitter is expressed in term of time such as picoseconds. In some applications, often in the data communications sector, jitter is expressed in terms of UI (unit interval). UI normalizes the amount of jitter to the period of the clock. One UI is equal to the period and therefore jitter UI can be expressed as:

$$JitterUI = Jitter\ (sec)/1\ UI\ (sec)$$

Diagnostic Techniques

As seen from this chapter, there are a variety of sources that can contribute to excessive jitter. Power supply noise is a large contributor as outlined in that section of this book but other sources may be possible. The following is a list of the leading factors that cause excessive jitter in clock buffer and PLLs.

The power supply pin decoupling capacitance or layout may be ineffective. Check the board layout against the PLL recommended layout guideline for capacitor value and connections for minimum inductance

The power supply noise is reaching the PLL supply pin. It is often easiest and fastest to verify this fact by supplying the PLL with an external power supply. Then check for any improvement. Although it can be cumbersome to lift the VDD supply pins, it provides positive feedback that the power supply caused the jitter. Care must be taken to maintain VDD pin decoupling capacitance. This technique avoids trying to characterize the supply noise and then trying to obtain power supply rejection curves that correspond to the frequencies of interest.

The input reference clock may have excessive noise or modulation within the PLL loop bandwidth. Connect the input reference clock to a spectrum analyzer and check the level of sidebands below the loop bandwidth.

The output trace has crosstalk to other switching signals on the board layout. The length and spacing of signal traces next to the clock trace should be checked. It is preferable to have a ground trace next to the clock lines or extra spacing between the other signals.

The measurement equipment or method is not appropriate and is providing false readings. Measuring jitter from an oscilloscope screen is not precise because of scope trigger jitter and retrace jitter. It is preferred to use a high sampling rate scope with dedicated jitter analysis firmware or software. This uses the scope-sampling clock that is more precise.

Verify that the component isn't defective and that the PLL chosen meets the jitter design requirements.

Conclusion

Jitter is a common parameter associated with clock generation and distribution. Controlling excessive jitter is necessary for achieving the most efficient and error free design. By understanding the terms and knowing the measurement techniques, the engineer can design and verify stable clock waveforms that offer solid performance.

5 Power Supply Filtering

The power supply is the foundation to providing optimal clock signal performance. As the quality of the voltage supply degrades, so does signal integrity. Noise in the supply will affect jitter and skew in clock buffers. The designer can choose whether the amount of timing degradation is acceptable to the total budget or incorporate filtering and layout techniques to minimize it. This chapter will address the application of ferrite bead filtering and decoupling capacitors.

Most designers know that decoupling caps are required in any high-speed digital design. They know this because all previous designs used them. However, it is often unclear which value to use and how many to populate. Ferrite beads offer another level of filtering to produce the absolute best quality in signals. But choosing the correct ferrite bead can be quite difficult. The specifications for beads often do not provide the information needed to select the right component.

In order evaluate the performance of power plane filters, a test set-up is needed that allows a controlled injection of "noise" into the supply. The effect on a variety of clock buffers and PLLs in terms of jitter and skew can then be analyzed.

Test Fixture

Figure 5.1 shows the test fixture used for the jitter and skew evaluations. The component in the middle of the diagram is the Device Under Test (DUT), which will be populated with various clock PLLs and buffers. A pulse generator supplies a controlled input waveform into the device. To view and capture the output signal, a Time Interval Analyzer (TIA) is used. A waveform generator is also attached to the input voltage of the DUT that will be used to create the noise component.

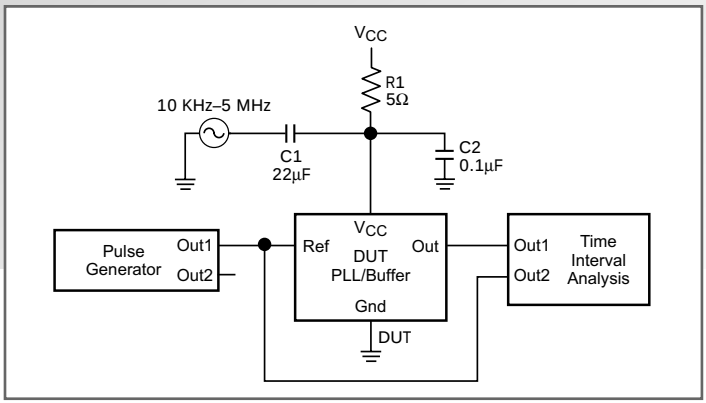


Figure 5.1 Supply Noise Test Fixture

For the jitter contribution versus frequency plots, the entire supply plane is modulated by a sine wave source. A frequency is swept across the region of test (10 kHz–5 MHz) to simulate the effects of noise propagating through the power plane. Every test sweeps through this range, as noise isn't limited to a single frequency. The 0.1 µF V_{CC} decoupling capacitors are left connected to each V_{CC} pin that allows the DUT to operate in a similar manner to an actual operating environment where decoupling caps are populated. The noise signal passes through a 22 µF bulk capacitor and overdrives the 0.1 µF decoupling caps. The main supply will drive a slightly higher DC voltage that normal to compensate for the voltage drop across R1.

PLL Test

In the first evaluation, a PLL-based Zero Delay Buffer is the device under test. For a PLL, phase offset jitter is the critical parameter that needs to be measured. Figure 5.2 shows a simplified drawing where the reference input (REFIN) is connected to the function generator and is also used as the trigger for the test equipment. One of the outputs is connected to the feedback path in the PLL and is the point where the jitter is measured.

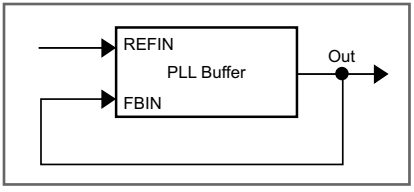


Figure 5.2 PLL Jitter Test

In Figure 5.3, the scope trace from the TIA shows the input signal and the resulting signal through the device. By injecting a 300 mV noise signal into the power supply, the PLL outputs are affected in both amplitude and phase offset. The PLL input is a well-defined signal while the output varies considerably. The additional jitter measured on the output is a direct function of the noise and can expressed with the following equation.

Additional Jitter (ns) = Noise (V)/Slew Rate (V/ns)

The Noise value in the equation is simply the amplitude of the noise present on the power supply pin of the part. The Slew Rate is the transition rate of the output driver independent of any noise. Hence, additional jitter is reduced as the rise time of the signal increases given the same noise level

The phase offset jitter added by the supply noise has a bimodal distribution as shown in the histogram. It is deterministic and bounded jitter because it is injected by a finite noise source with a known amplitude. The source can be traced to the filter of the VCO and the trip point variation of the input reference signal. The VCO is directly affected by the varying voltage inputs due to noise. This translates into jitter on the outputs. However, some PLL designs are more tolerant than others due to the use of better noise filtering circuits.

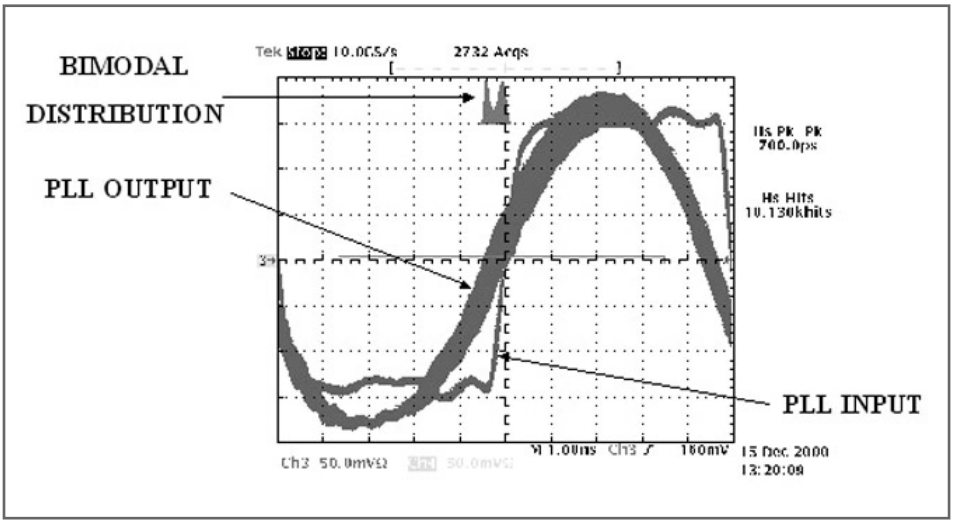


Figure 5.3 Power Noise in PLL

PLL Noise Rejection

Because some PLLs reject noise better than others, they react differently to the same level of noise on their power supplies. In Figures 5.4 and 5.5, two different PLLs are tested for their jitter component with the injection of noise. Each received 100 mV, 50mV, and 25 mV noise signals sweeping through the frequency range. Both exhibited higher amounts of jitter as the noise amplitude increased. However, PLL2 in Figure 5.5 shows jitter of 100 ps at 200 KHz whereas PLL1 in Figure 5.4 exceeds 350 ps jitter at this same frequency. At the 25 mV noise level, PLL2 peaked at 50 ps whereas PLL1 exhibited nearly 100 ps of jitter. Additionally, the frequency at which PLL1 had its peak value at 200KHz while PLL2 was at 5 MHz. Clearly, there is a difference in the performance of these two parts. Unfortunately, data to determine noise rejection is not typically listed on a standard datasheet and isn't obvious to the designer.

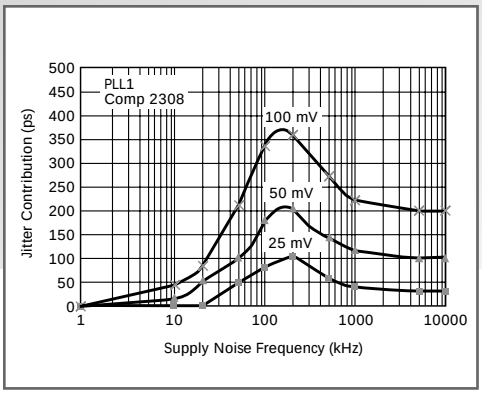


Figure 5.4 Jitter in PLL1

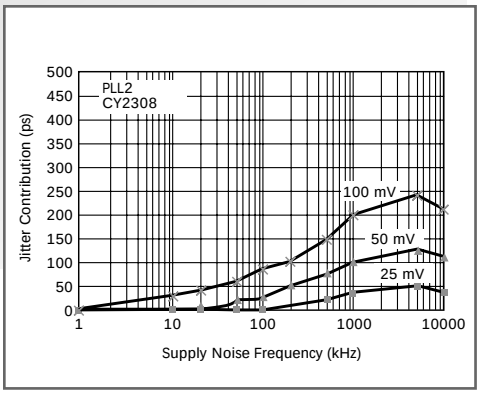


Figure 5.5 Jitter in PLL2

Non-PLL Buffer

The delay (T_{PD}) and output skew can be critical elements in a non-PLL based clock buffer. Using the same test methodology, the effect that power supply noise has on these parameters can be determined. Similarly, a waveform function generator drives the input to the buffer and is also the signal used to trigger the TIA. The output of the buffer is analyzed for increased delay.

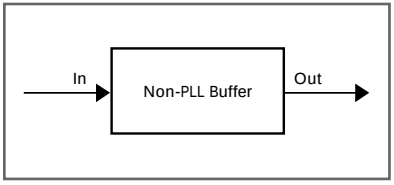


Figure 5.6 Buffer Delay Test

The buffer driver is also affected by the supply voltage noise. The scope trace in Figure 5.7 shows a variation in the delay through the device as the noise sweeps through the frequency range. In typical CMOS buffers, the supply voltage affects the triggering of the part by changing the gate switch point on the input. More significantly, it changes the internal propagation time of the buffer. The noise causes variations in the voltage supply and hence varies the T_{PD}. The net effect is the output signal will exhibit jitter due to the delay variation caused by the noise.

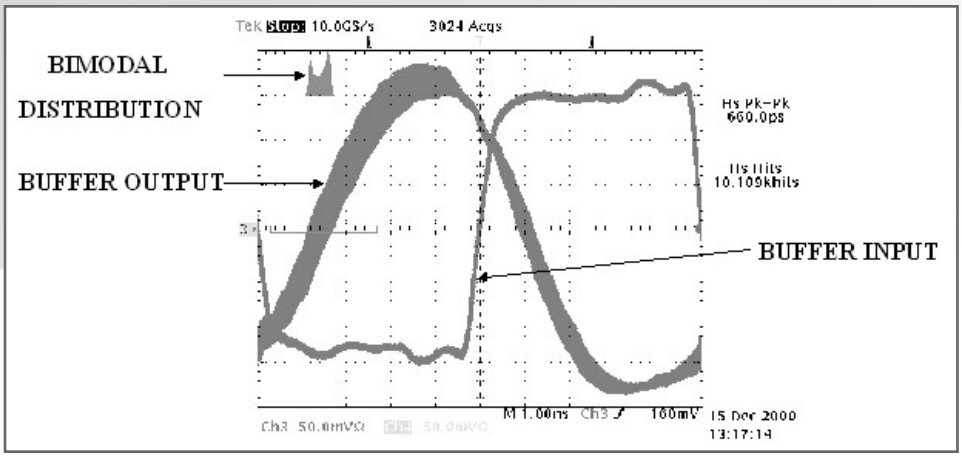


Figure 5.7 Power Noise in Buffer

The increase in jitter, which is really input-to-output skew, affects all output pins of the buffer equally. Therefore, for simple buffers, the output-to-output skew remains constant. However, if the buffer contains complex features and the delay paths internally vary, the output-to-output skew could change.

Filtering the Noise

Noisy supply planes are a result of a variety of sources. Power supply switching circuits, digital components with busses, and even clocks can contribute to the noise. Many digital synchronous designs have buffers switching many outputs simultaneously causing current spikes large enough to generate power supply noise. To control noise in the power plane from affecting the clock circuitry, filters such as ferrite beads can be used.

The ferrite bead provides AC isolation between the power plane and the PLL (or buffer). By using beads, noise from the power plane is attenuated before reaching the PLL. Likewise, digital noise from a PLL or buffer with many simultaneous switching outputs is also attenuated before reaching the power plane.

The ferrite bead is chosen over a plain inductor because it provides better AC isolation. It does this by storing part of its created magnetic field in the ferrite core material. The ferrite bead also has a lower DC resistance than an inductor for an equivalent impedance value. This is important because the voltage drop is usually undesirable when attempting to meet the margins required for component power. The ferrite bead equivalent circuit is shown in Figure 5.8.

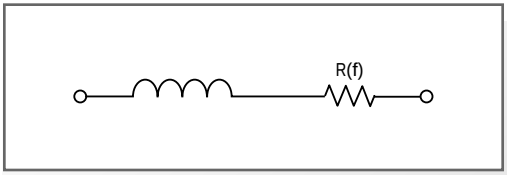


Figure 5.8 Equivalent Ferrite Bead Circuit

The ferrite bead is represented as an inductor and resistor in series. The resistance however, is dependant on the frequency of the signal and is therefore labeled as R(f). The ferrite bead appears resistive at high frequencies but is best analyzed with a plot showing impedance versus frequency. The characteristic behavior of a typical ferrite bead is shown in Figure 5.9.

When designing filters, it is preferred to minimize the DC resistance (R) so that the voltage drop is kept small. At the same time, the impedance (Z) needs to be as large as possible at the frequency that is to be filtered. In actuality, it was found through testing that choosing the lowest DC resistance is not always best choice. And finally, the inductive (X) portion of the bead should also be small. A large X will distort the filter's effectiveness and there is a likelihood of a resonance condition occurring.

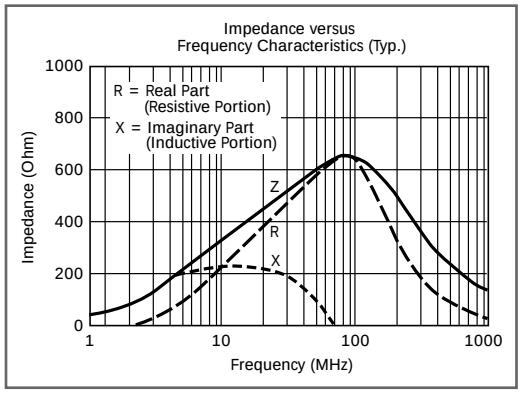


Figure 5.9 Ferrite Bead Impedance

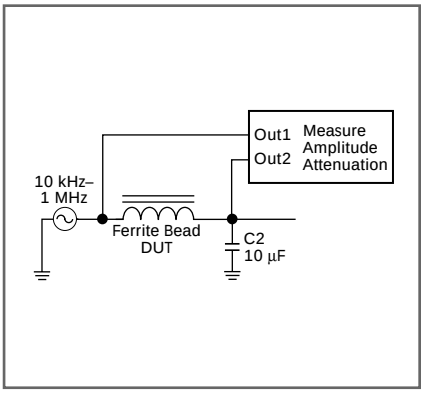


Figure 5.10 Ferrite Bead Measurement

To evaluate how well noise is suppressed using ferrite beads with varying characteristics, a different test jig is used. Figure 5.10 shows a noise generator that will sweep a frequency range of 10 kHz to 1 MHz through a test ferrite bead. The bead is used with a bulk capacitor to complete the filter design. A 10 µF ceramic surface mount capacitor with a low effective series makes the filter more efficient. This becomes especially important at the lower frequencies when the bead impedance drops off.

Channel 1 (CH1) of the oscilloscope measures the amplitude of the voltage (V_{OUT}) through the filter under test. Channel 2 (CH2) measures the noise voltage (V_{IN}) injected into the filter. As the input signal sweeps through the frequency range, a comparison of the voltage ratio can be plotted. This test was performed on five different ferrite beads and the results are show in Figure 5.11. The plot shows the voltage attenuation in dB that can be expressed as:

$$Attenuation\ (dB) = 20*Log\ (V_{OUT}/V_{IN})$$

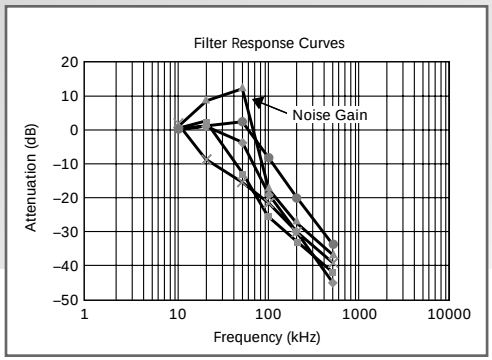


Figure 5.11 Ferrite Bead Results

One particular bead stands out among the others. Due to resonance near the 100 kHz frequency, the bead actually increased the noise ratio rather than attenuating it. Therefore, this one is not recommended for the clock filter design.

Knowing how the beads react in the noise test system, the impedance-frequency plots can be analyzed for correlation. Figures 5.12 and 5.13 show the plots for two different ferrite beads.

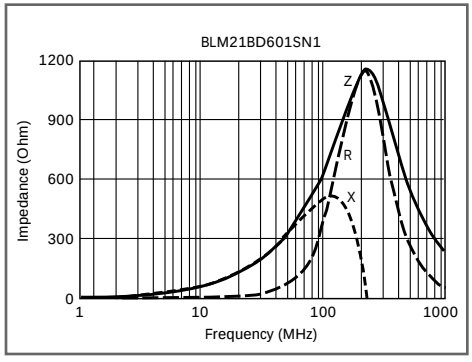


Figure 5.12 Recommended Bead

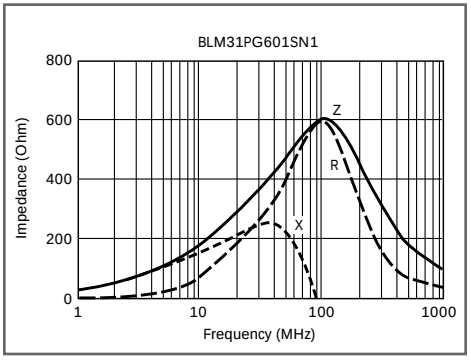


Figure 5.13 Not Recommended Bead

It is not obvious from these manufacturer datasheet curves which bead is suitable for a filter as the variations are subtle. The bead that is “not recommended” for clock noise filtering has a characteristic of a long inductive tail extending to lower frequencies. It’s not that this is a bad bead design, it merely means that this particular one isn’t suited for this application.

Measured Impact on PLL

The bead test fixture provides the ability to verify that noise in the power supply is attenuated. The beads that pass this test can be placed into the jitter test fixture and validated with clock PLLs and buffers. Using the same Zero Delay Buffers as before (Figure 5.4 and 5.5 show the results without the ferrite bead), a new set of tests are executed with the ferrite beads in place.

Four of the ferrite beads used in the bead test are subjected to 50 mV of noise and the PLL output is measured for jitter. The first observation is the overall jitter has dropped considerably as seen in Figure 5.14 and 5.15. Note that these curves are the same scale as Figures 5.4 and 5.5. However, the attenuation of jitter varied slightly based on the specific bead in the circuit. PLL1 shows peak jitter ranging from 125 ps to less than 25 ps depending on the bead. With no bead in place, the jitter peaked at 200 ps at the 50 mV level.

PLL2 also shows a marked improvement. With no bead filtering the noise, the jitter peaked at 150 ps. With any of the four ferrite beads in place, the jitter is extremely small and inconsequential. This is due to the fact this particular PLL contains additional internal noise rejection capabilities

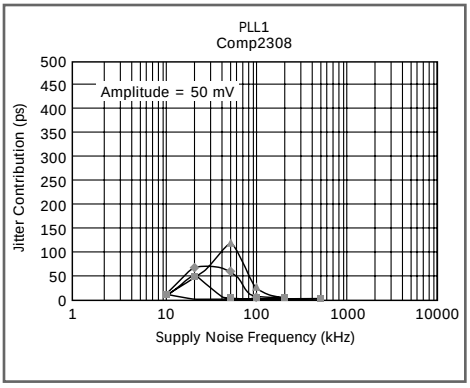


Figure 5.14 PLL1 Jitter with Filter

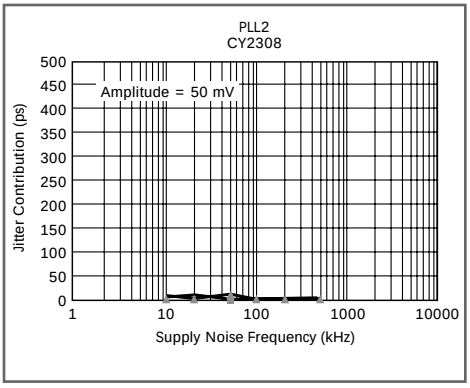


Figure 5.15 PLL2 Jitter with Filter

Non-Sinusoidal Noise

The analysis has so far used a sine wave noise source. In order to properly validate the filtering capability of the beads, non-sinusoidal noise needs to be injected. By using noise generated by a switching power supply, noise with dominant peaks and many harmonics can be tested. The operating frequency of the switching power supply is 250 kHz and exhibits lower frequency modulation when under load. Within each switcher period, there is an initial step where the voltage changes abruptly and contains harmonics into the MHz region. This represents a real-world circuit board with power plane noise.

The spectrum analyzer plots in Figure 5.16 display the spectral purity of the PLL output at a fixed frequency of 66 MHz. The system board is under heavy load to maximize the switcher noise. The goal is to have the peak as sharply defined as possible whereas a wider curve indicates jitter. The plots show the result of two different ferrite beads populated in the system where the one on the left has a sharp, well-defined peak and the plot on the right shows a broader, less-defined peak. Therefore, the left plot shows a recommended ferrite bead.

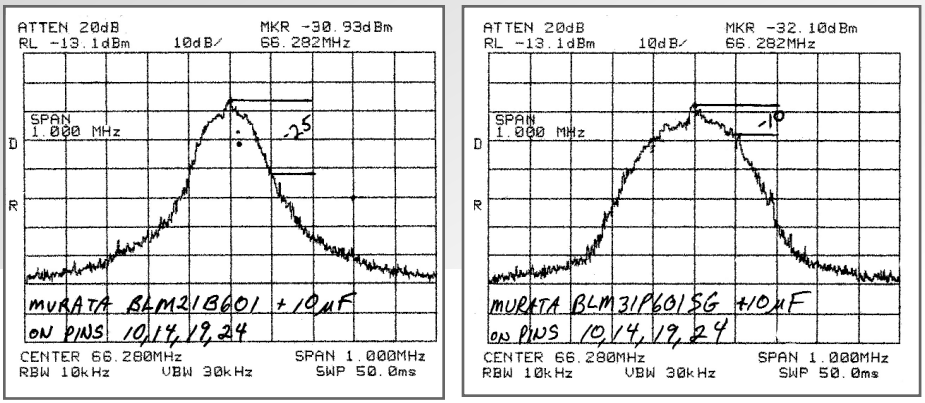


Figure 5.16 Spectrum Analyzer Plots

Ferrite Bead Recommendations

It is very difficult to determine which ferrite bead will perform the best based on the manufacturers specifications. By using data from a test environment, a bead that will filter the noise can be selected. Based on these tests, the following beads seem to perform well in filtering power supply noise for clock buffers. They are by no means the only beads that will work.

1. Murata BLM21BD601SN1, BLM18BD421SN1, BLM18BD601SN1
2. Vishay-Dale ILB1206-300
3. TDK ACB2012M-300, ACB2012L-120
4. Steward HZ0805C202R
5. Fair-Rite 2512063017Y0

The beads that were tested and found not to work well had several common characteristics. Beads with the following should be avoided for filter design:

1. Not characterized at lower frequencies such as 10 kHz
2. Very low DC resistance (< 0.1 ohm).
3. Long inductive “tail” extending to low frequencies.

Resistive Filters

Some PLL components have a separate VDD pin (VDDA) that powers the PLL core while the other VDD pins power the output drivers. This usually gives more flexibility in the design of an external supply filter. For example, the PLL manufacturer datasheet may specify IDDA (for VDDA pin) drawing 15 mA max. With the current this low, it becomes possible to use a resistor instead of a ferrite bead. Choosing a value such as 8 ohm results in only 0.12 volt drop and may be within operating specifications. For example, if using a 3.3V supply, the voltage drops to 3.18V at the VDDA pin. PLL manufacturer datasheets may contain specific filter recommendations or layout guidelines that include a suggested filter. The advantage of using a resistor instead of a ferrite bead is that the lower frequencies are better filtered. This can be important for applications where long-term jitter is important. Long-term jitter

is the edge stability measured many clock cycles away, often 20 µs or more. Applications that may be affected by this include graphics DOT clock generation and serial link communication clocks.

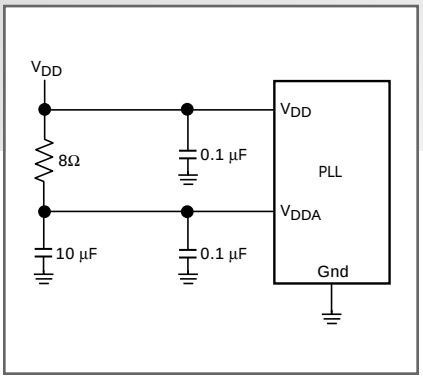


Figure 5.17 Resistive Filter

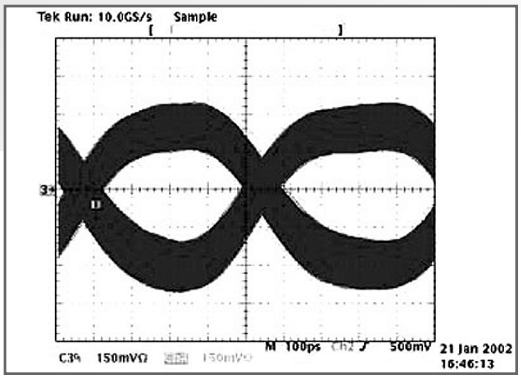


Figure 5.18 Poor Method to View LVPECL Driver

Supply Noise In Differential Signaling

The previous analysis had focused on single ended interface signals. As operating frequencies have climbed, designers are using differential signaling. In this next example, a clock driver with an LVPECL interface is analyzed. The component is a non-PLL fanout buffer operating at 1 GHz. For these tests, the same modulation is applied to the VDD pins as was previously described. Since these are differential drivers, it is generally expected that it is more immune to supply noise

Figure 5.18 shows an oscilloscope view of the output driver's differential pair. The measurement shows the amplitude and offset change with supply noise levels. LVPECL is referenced to the voltage supply so it is expected the common mode voltage moves with the modulation on VDD. However, the largest concern is with the signal crossing point perturbation and not the common mode voltage. It becomes difficult to check with this view how much the crossing points have actually been affected in time and a high performance differential probe or a differential-capable time interval analyzer needs to be used.

Shown in Figure 5.19 is the jitter contribution of the LVPECL clock driver operating at 1 GHz captured by a timing interval analyzer. In this series of tests, noise with amplitudes ranging from 25 mV to 300 mV was injected into the system. The graph shows that noise does affect a differential driver by moving its crossing point. At lower amplitude levels, the jitter was measured to be less than 5 ps. At the highest level of noise, 300 mV, the jitter reached nearly 6 ps. Although the amount of jitter is small, it can represent a significant portion of a period when clock speeds reach high frequencies.

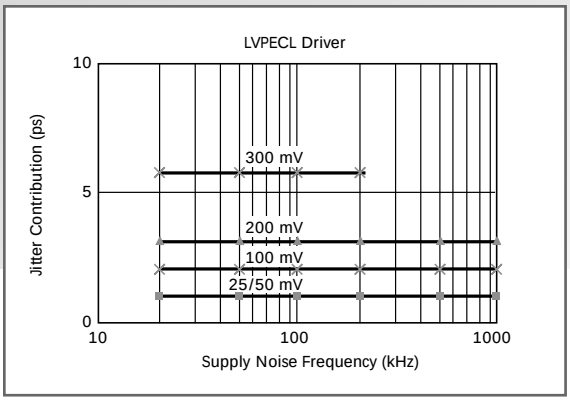


Figure 5.19 LVPECL Supply Noise

Bypass Capacitors

Bypass capacitors, also commonly referred to as decoupling caps, are often the most confusing component in a high-speed digital design. Engineers know that capacitors need to be in the design, but the purpose and operation of the part is many times unclear. Many designs use the same decoupling methodology that was used in the past. However, with the increase in the speed of signals, the implementation needs to be updated.

The bypass capacitor has three major purposes. It is used to prevent voltage droop on the V_{DD} pins, it provides a low impedance path from the power plane to the ground plane, and it provides a signal return path between the power and ground planes. Droop and filtering are addressed in the following sections and the signal return paths are discussed in Chapter 6, PCB Layout Considerations.

Voltage Droop

The initial capacitance value that is chosen for a bypass capacitor is done to provide a minimum current need. When a clock buffer's outputs switch, the power terminals will sag due to the die to PCB voltage drop. This drop, or droop as it refers to a temporary condition, occurs because of the inductance and resistance of the device lead frame and bonding wire. The bypass capacitor function is to supply this momentary need in current with its stored charge. To do this, it must store a minimum amount of energy. The buffer loading determines this energy value and the amount stored is given by the formula:

$$Q = CV$$

Where : Q = Charge
V = applied voltage
C = capacitance in farads

Differentiating this equation yields:

$$I(t) = dQ/dt = C * dV/dt$$

$$C = I\ dt/dV$$

This equation allows us to calculate the capacitance required to prevent voltage droop due to the switching clock outputs. For example, suppose we have a 3.3V clock buffer with eight outputs driving a 50-ohm trace with a rise/fall time of 2 ns. Let's also assume that a droop of 50 mV is the maximum amount of voltage droop allowable to the component.

First we need to calculate the output current during the rise of the clocks. Assuming the outputs reach 3V, each output requires 3/50 ohm = 60 mA. Because the buffer has eight outputs, it requires a total of 480 mA.

Solving for C yields:

$$C = (480\ mA * 2\ ns)/50\ mV$$

$$C = 0.0192\ \mu F$$

This suggests that this amount of total capacitance is required on the component to maintain less than 50 mV of droop. Often, 0.1 µF capacitors are used for bypassing. Another way to view this situation is that a 0.1 µF capacitor supplies 480 mA of instantaneous current in 2 ns with only 9.6 mV of voltage droop across the capacitor.

This example assumed a worst-case voltage swing on the outputs of 3 volts. Depending on the termination method used, this swing could be less than this amount. However, since this is used only as a guide, its best to over estimate the value.

Capacitor Filtering

Similar to the filters discussed earlier in the chapter, decoupling capacitors block unwanted AC variations (noise) in two directions. They prevent noise from entering the device from the power plane and they also suppress noise from the device to the power plane.

Decoupling caps are more than a capacitor. They are in effect a capacitor in series with an inductor and a resistor as shown in Figure 5.20.

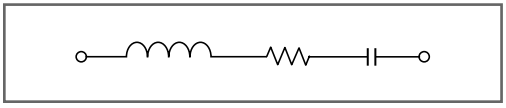


Figure 5.20 Capacitor Model

This is important in understanding what happens during the operation of a capacitor. Because it is filtering AC signals, the characteristics of the capacitor are always changing. To understand the operation over frequency, curves are provided for each value, package type, and composition. Figure 5.21 shows the impedance versus frequency curve for a 0.1 µF capacitor packaged in a 0603 body.

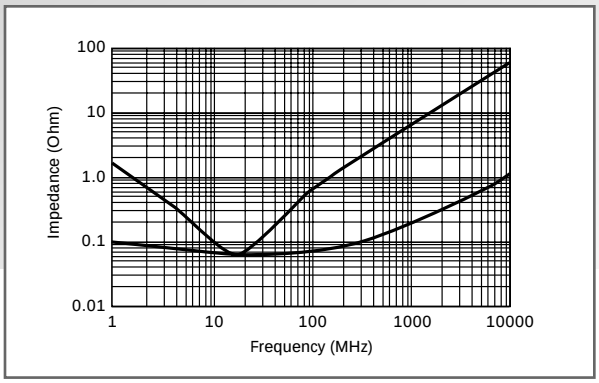


Figure 5.21 A 0.1 µF cap in 0603 package.

The curve shows this capacitor resonates at approximately 11 MHz which is the point where the impedance is lowest. On either side of this frequency, the amount of impedance increases. The ideal capacitor has zero ohms at those frequencies that need bypassing.

The 0.1 µF capacitor has been used for decoupling for many years. Even when clock rates were as slow as 100 kHz, the 0.1 µF was used. As the clock frequencies increased, it was thought the decoupling capacitor needed to decrease in value in order to address the higher frequencies. Many designs started to use 0.01 µF capacitors instead of the 0.1 µF. Figure 5.22 shows a 0.01 µF capacitor curve in the same 0603 package as the 0.1 µF.

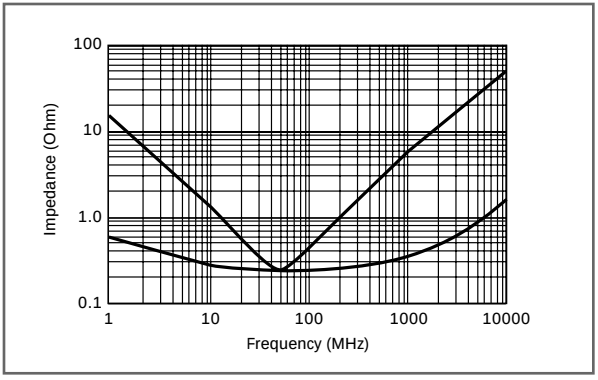


Figure 5.22 A 0.01 µF cap in 0603 package.

Although the resonant frequency of this device has slightly increased, the impedance has not changed in the higher frequencies. With the exception of the narrow frequency band near resonance, this capacitor did not provide any substantial benefit.

But why didn't the smaller capacitor provide lower impedance at the higher frequencies? The answer is inductance. Associated with the package is a fair amount of inductance and is the limiting factor at the higher frequencies. Inductance is driven by the size of the metal material for the leads. In order to reduce inductance, the package size needs to decrease. This not only affects the 0.01 µF cap, but also benefits the 0.1 µF cap.

Many designers add two different values of capacitors into the design. The thought is by using a larger capacitor in parallel with a smaller capacitor, a larger area of the frequency range will benefit from lower impedance. However, when the two different capacitors are place together, interesting interactions occur.

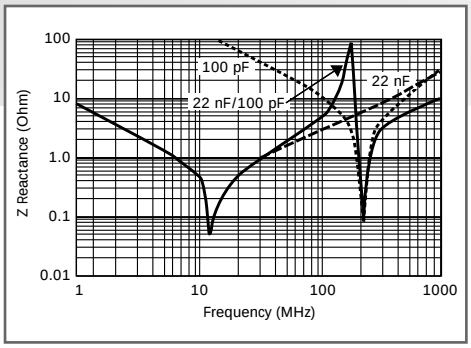


Figure 5.23 Two Different Capacitors

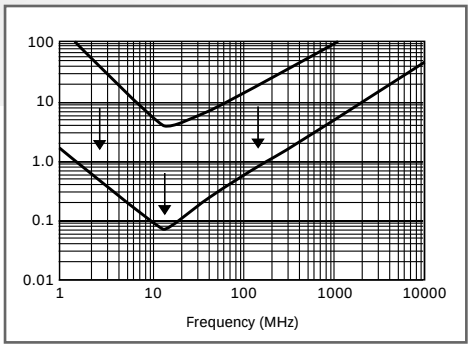


Figure 5.24 Two Capacitors in Parallel

Figure 5.23 shows the results with two capacitors of different values in parallel. Notice the spike in impedance just above 100 MHz. The goal of obtaining lower impedance across a wider frequency is not achieved. On the contrary, the impedance has actually increased in some areas. Therefore, this practice is not recommended.

But, there is a method that will help to lower the impedance across all frequencies. When two capacitors of the same value and package size are in parallel with each other, no peak increase in impedance is realized. However, they do have the effect of halving the inductance. Since inductance is the limiting factor, this causes the impedance curve to drop across all frequencies as illustrated in Figure 5.24.

What Frequencies to Bypass

In is a common misconception that the faster clock rates lead to the need for higher frequency decoupling. This is only partly true. Because the frequencies have increased, the technology for high-speed clocks has been driven to provide faster rise and fall times. Therefore, a signal whose frequency is 1 MHz will need the same decoupling as a clock running at 50 MHz using the same high-speed buffers. The difference may be there is more timing margin afforded in the slower speed system. A rough generalization suggests that frequencies up to one-half the fastest signal transition rate (rise or fall) frequency need to be bypassed.

$$f_{BYPASS} = 0.5/\text{Transition Rate}$$

It should be noted that decoupling is really a four-part system. The four parts are comprised of: the power supply, the bulk capacitors, the bypass capacitors, and the intrinsic capacitance in the board. Each provides decoupling in it respective frequency bands with the power supply addressing the very low frequencies and the intrinsic board capacitance

addressing the high frequencies. The difference between the capacitance is large enough between each part that the large spiking effect with multiple capacitor value interaction does not occur. However, between each frequency band, there will be small peaks. These peaks will always exist but they can be moved slightly.

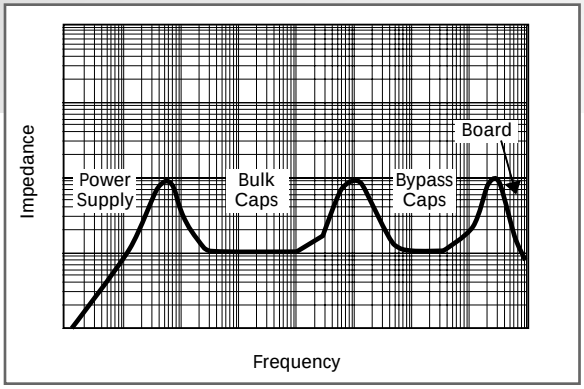


Figure 5.25 Four-Part System

Generally, engineers have limited ability to alter the characteristics of the power supply and board plane capacitance. But it is possible to vary the bulk and bypass capacitors values. The goal is to have the four-part system provide a low impedance path (less than 1 ohm) throughout the frequency range. By adjusting the values of the bypass capacitors, the small impedance peaks can be shifted to allow the lowest impedance to match the frequencies of concern.

Capacitor Types

There are many different types of capacitors available for decoupling. The ceramic type capacitors are the most efficient and should be used for bypassing high-speed digital designs. Capacitors can be affected by temperature, voltage and time. Classes are used to rate the properties associated with ceramic capacitors.

Class 1 capacitors, or temperature compensating capacitors, have predictable temperature coefficients and in general do not exhibit any aging characteristics. Thus, they are the most stable capacitor available. The most popular Class 1 multilayer ceramic capacitors are C0G (NP0) temperature compensating capacitors. EIA Class 2 and 3 capacitors provide a wider range of capacitance values and temperature stability. The most commonly used Class 2 dielectric is X7R. X7R provides intermediate capacitance values that vary only ±15% over the temperature range of –55°C to 125°C.

Class 3 dielectrics are the Y5V and Z5U series. The Y5V provides the highest capacitance values per physical volume and is used where limited temperature changes are expected. The capacitance value for Y5V can vary from 22% to –82% over the –30°C to 85°C temperature range. The Z5U dielectric falls between X7R and Y5V in both stability and capacitance range.

Variations in voltage have little effect on Class 1 dielectrics but it does affect the capacitance Class 2 dielectrics. In class 2 capacitors, the application of DC voltage reduces the capacitance while the application of an AC voltage within a reasonable range tends to increase capacitance. Figure 5.26 shows the effects of AC voltage on an X7R capacitor.

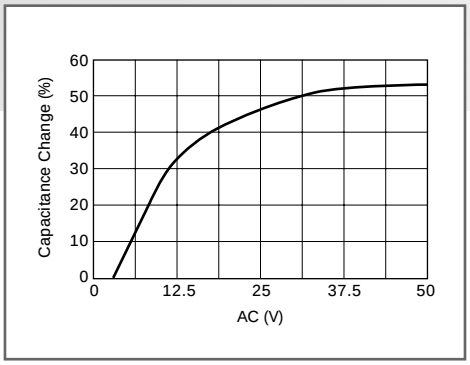


Figure 5.26 AC Voltage on X7R Capacitor

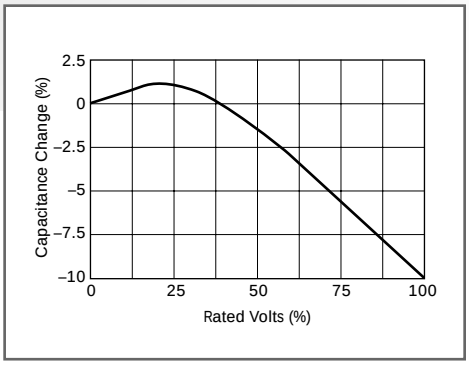


Figure 5.27 Capacitance vs. DC Voltage

The capacitance of bypass capacitors also changes as a function of the actual applied DC voltage. The effect of the DC voltage is shown in Figure 5.27 at 20ºC.

Another factor that affects the capacitance of a specific device is time. Class 2 ceramic capacitors change capacitance with time as well as temperature, voltage and frequency. This change with time is known as aging. Aging is seen as an exponential loss in capacitance over time. A typical curve of an aging rate for several different ceramic capacitors is shown in Figure 5.28.

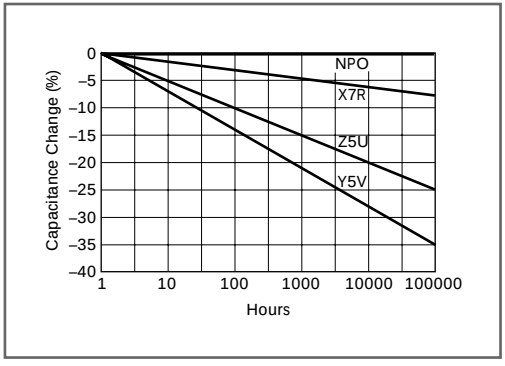


Figure 5.28 Capacitor Aging

If a Class 2 ceramic capacitors has been sitting idle for a period of time and begins to age, it can be "reset" by heating the device. By raising the temperature above its curie point (125°C for 4 hours or 150°C for $^1/_2$ hour) the part will reverse it's aging factor and return to its initial capacitance value. Because the capacitance changes rapidly after de-aging, the

capacitance measurements include a time period for the process. Various manufacturers use different time bases but the most popular one is twenty-four hours after "last heat." The aging curve can be altered depending on voltage and other stresses.

The Bypass Rules

As seen by these series of plots, the package size and value of decoupling makes a difference. How the capacitors are connected plays a key role in how well the buffer or PLL performs. The inductance of the leads must also be minimized. A short, fat trace provides the lowest inductance and should be used whenever possible. Long, narrow traces should be avoided as they increase inductance. Passing through a via is acceptable provided that the path is lower inductance than an alternative longer trace. The following is a list of general rules when designing with bypass caps:

1. Select the Largest Value Capacitor in the Smallest Available Package
2. Ensure the Capacitor Value Meets the Voltage Droop Requirement
3. Use Only One Value of Bypass Capacitor for the Component
4. Keep Bypass Capacitors Close to Component
5. Minimize the Lead Inductance
6. Place Capacitors in Parallel to Improve Response
7. Consider Multi-cap Packages
8. Use Good PCB Layout Techniques (See Chapter 6)

Conclusion

Noise does affect the performance of clock buffers and PLLs. Having a clean, solid power source is fundamental in providing good clock waveforms. All high-speed digital designs of today need decoupling capacitors to ensure good signal quality. For the jitter critical designs, ferrite beads should be considered. Although it is difficult to select which bead will perform best, the analysis and recommendations presented in this chapter provides the necessary guideline information.

6

PCB Layout Considerations

The Printed Circuit Boards (PCB) provides two significant functions in an electrical design. It provides the mechanical locations for the components that reside on the board and it provides the connectivity between the components. These connections play an important role in providing a solid pathway to the power system and to ensure the highest degree in signal integrity. This chapter addresses the many aspects of PCB traces for clocks.

The traces and planes can be separated into two categories. There are the pathways for the power source and there are traces required to carry the actual signals. In the first part of this chapter, PCB layout as it applies to the power planes is discussed. Later in the chapter, layout considerations for signal traces are addressed.

Power & Ground Planes

Power planes are large sheets of a conductive material that typically reside on entire PCB layers. They provide four primary functions to the circuit. They provide:

1. A low impedance path for power from its source to the components on the PCB.
2. A physical channel to vent and move heat from the components.
3. Electrostatic shielding between the electromagnetic fields of signal traces that run on both sides of the planes.
4. A sheet capacitance for the ground plane that exist on other layers of the PCB. This in turn provides additional AC bypassing within the power circuitry of the PCB.

The first and foremost functionality of a power plane is to reduce the resistance that causes a voltage drop between the component and the power source. The thickest power plane available will provide the best results. For example, using a two-ounce copper power plane instead of a one-ounce will cut in half any point-to-point power path resistance. It can be thought of as having two resistors (and inductors) in parallel. The increased plane thickness reduces both the DC resistance and the AC inductance drops. The drop in the DC resistance allows the power supply to reach the component cleanly, while the reduction in AC inductance provides a low impedance path for the signal return currents.

As a secondary benefit, the thicker plane also increases the ability to sink heat out of the component. The bond wires and lead frames are a major thermal path in non-heatsinked components.

The planes also aid in the reduction of Electro Magnetic Interference (EMI). They provide a lower impedance path across which the EMI develops and a larger faraday shield to short out these radiated fields.

The sheet capacitance that the power plane provides is proportional to its size, its distance from the ground plane, and the dielectric constant of the material between them. It has the benefit of providing bypass capacitance particularly at the high frequencies. While it is far from being sufficient to provide all of the bypassing needs of a high-speed logic design it should be utilized to its maximum. The capacitance of the planes can be calculated by the following equations:

$$C = 0.0885 E_R[(N-1)*A)]/t$$

Where: E_R = The Relative Dielectric Constant
N = Number of Plates
A = Area of one side of one Plate in square centimeters
t = Thickness (separation of plates) in centimeters

$$C = 0.225 E_R[(N-1)*A')]/t'$$

Where: E_R = The Relative Dielectric Constant
N = Number of Plates
A' = Area of one side of one Plate in square inches
t' = Thickness (separation of plates) in inches

For example, using a 10-inch by 10-inch FR-4 board with an E_R of 4.1 and a 0.005-inch separation between the power and ground plane, the capacitance is calculated as:

$$C= 0.225(4.1)[(2-1)*100]/0.005 = 18,450 pF$$

This is equivalent to 184 pF per square inch. Table 6-1 shows the dielectric constants for several common materials used in PCB design today. It is always advisable to consult your fabricator for the precise ER value as different epoxies are used when constructing a PCB. Dielectric constants of PCB also change with frequency as shown in the table.

Table 6-1 Dielectric Constants

Material	E_R @ 1 MHz	E_R @ 300 MHz
FR-4, Tetra Functional	4.2 – 4.6	4.0 – 4.3
FR-4, Hi-Grade Multifunctional	4.2 – 4.6	4.1 – 4.4
Polyimide	4.2 – 4.6	4.1 – 4.3
GETEK	3.9 – 4.1	3.9 – 4.0
BT	3.6 – 4.1	3.55 – 4.0
CE	3.6 – 4.0	3.6 – 4.0

Ground Island

With clock generators and buffers, it is recommended to create a ground island directly beneath and on the same layer as the device. Connections are then made from that island using short traces to the ground pins as shown in Figure 6.1. These traces should be as wide as possible.

The ground island that is then present should be stapled with vias to the inner ground plane or planes. The general rule is one via per ground pin as close to where the components ground trace meets the island. Adding more vias in parallel reduces the effective impedance. Beyond two vias per ground pin becomes increasingly less effective. Also, the more vias that are added, the ground plane impedance where they attach can be impacted. The PCB fabrication process usually has a guideline on the via requirements.

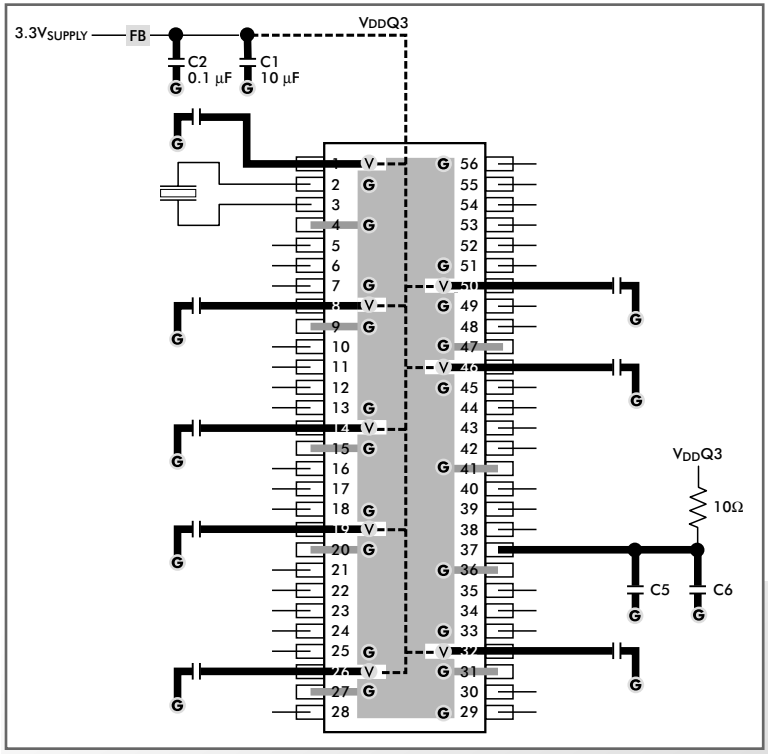


Figure 6.1 Ground Island

On smaller buffers, a large ground island may not be practical or necessary. In this case, a short, wide trace may be used to supply the power to the VDD pin and the VSS pin connected to the ground plane with a via. Figure 6.2 shows a small clock buffer with these types of connections. The VDD trace is filtered with a ferrite bead prior to the via connection to the power plane. The ferrite bead and its associated capacitors should be as close as possible to the clock buffer.

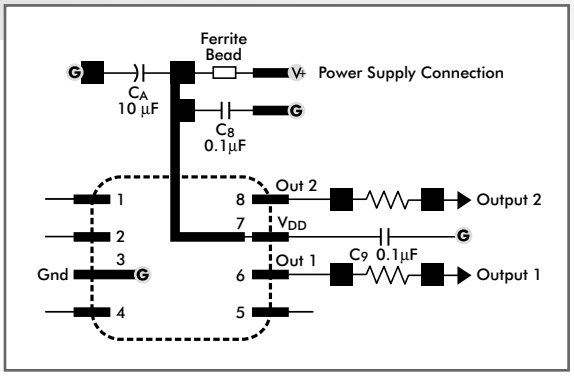


Figure 6.2 Small Buffer Layout

Vias

Vias are commonly used to connect the power plane to the power traces that ultimately attach to the power pins of the components. They can also have an impact on the power signal quality.

First, vias produce a higher resistance than a copper trace. As noted earlier, the higher the resistance, the greater the voltage drop. This is due to the material typically used in the fabrication of the via, which is plated granular copper. The resistance of a via changes based on the thickness of the copper that is plated in the via hole. Therefore, larger vias have a lower DC resistance and hence develop less of a DC voltage drop for any given current.

Vias also add inductance to the power trace. This inductance causes high frequency noise that is present on the power plane to stay on the plane (which is good) but it also isolates the capacitance effect of the power plane from the components on the other ends of the vias. Filling vias with solder, using heavy plating, enlarging their size and using multiple vias per power connection are the preferred methods of lowering both the resistive and inductive parasitic effects they have on power connections.

Power Traces

Similar to vias, a trace also has some amount of resistance, capacitance and inductance. The overall resistance must be kept to a minimum to avoid voltage drop on the trace. For the power plane to reach the power trace and ultimately the component, a via is also required.

If the connections to the device are made in the correct sequence, the resistance and inductance of the trace and the via will isolate the component's noise from passing into the power plane. This effect increases as the frequency of the noise rises. To achieve this isolation, the power trace must pass from the via, to the bypass capacitors pad and then to the component. This order is important to create an island of protected trace between the bypass capacitor and the component.

It is important to highlight that the trace between the component and bypass capacitor be as short as possible. The goal is to keep the inductance to a between the devices to a minimum. A short wide trace will produce better results than a long narrow trace.

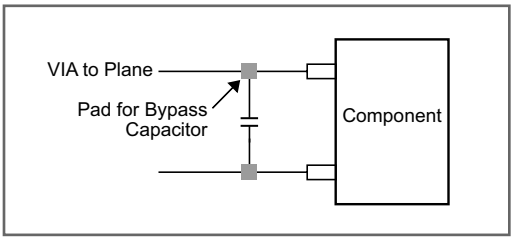


Figure 6.3 Power Trace to Component

Signal Return Paths

The power planes play a key role in the returns currents of high-speed digital clocks. At very slow speeds, the return current follows the path of least resistance. But at higher speeds, the return currents follow the path of least inductance. This can be either on the power plane or the ground plane directly below the signal trace. Normally, the return path is on the ground plane in standard PCB designs. When the return current reaches the driving component, the bypass capacitors provide the bridge to the proper voltage plane.

In order to maintain good signal integrity and to minimize crosstalk, a clean, unobstructed return path needs to be provided. For example, if a return signal encounters a 5-ohm trace within the ground plane, the integrity of the normal signal on the 50 ohm trace can be compromised.

There are a few simple rules to follow for planes. First, it should be as continuous as possible. By placing excessive clearances around holes that pass through it may make fabrication easier but it can cause the return current to travel through a non-optimal path. The effect of the clearance holes should be analyzed. Second, cutouts in the ground plane must be avoided. Similar to the clearance holes for the vias, a ground plane cutout will force the return path current around the cutout if the clock signal trace crossed above the cutout. This will increase the inductance of the path and will decrease the rise time of the clock on the signal trace. It will also increase the potential for crosstalk.

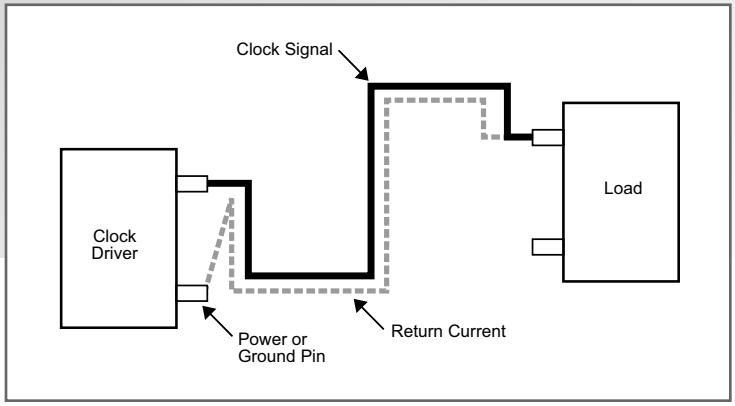


Figure 6.4 Signal Return Path

Signals

The second area with which we need to be concerned is the routing of clock signals. The goal, obviously, is to provide a pathway for the clock that has no noise, has uniform impedance, and does not produce any EMI. The next section discusses the affects board layout has on these attributes.

Crosstalk

Crosstalk can exist between traces on a PCB. If this occurs on a clock signal with sufficiently strong amplitude, a false trigger can occur. Therefore, crosstalk needs to be minimized in the design.

As a signal travels through a trace, it creates a magnetic field. It also reacts to other magnetic fields within its path. Therefore, the trace acts as both a field generator and as an antenna. The voltages that external fields cause are proportional to the strength of the external fields and the length of trace that is exposed to the field. Often, the trace that causes the crosstalk is termed the Source or Aggressor and the trace that is affected by the first is called the Victim.

Crosstalk between traces is a function of both mutual inductance and mutual capacitance. A model of the inductance and capacitance between traces is shown in Figure 6.5. Its magnitude is proportional to the distance from the source trace, the speed of the signal edge rate, and the impedance of the victim trace. In digital systems, crosstalk caused by mutual inductance is typically equal to or larger than the crosstalk associated with mutual capacitance.

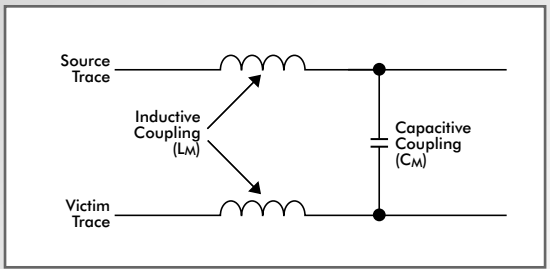


Figure 6.5 Mutual Coupling Between Traces

To illustrate the effects of spacing, the mutual inductance LM can be calculated with the following equation:

$$L_M = L/(1 + (s/h)^2)$$

Where: L = Inductance of the Wire
s = separation between the wires
h = height above the plane

This shows that moving the traces away (value S) from each other or by moving the traces closer (value H) to the plane, the mutual inductance is reduced by the square of the change. And since crosstalk is proportional to the mutual inductance, the magnitude of the crosstalk is also reduced. Figure 6.6 illustrates this model.

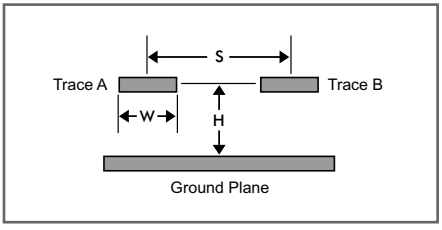


Figure 6.6 Trace and Ground Plane

The mutual capacitance CM injects current IM into the victim trace and can be calculated as:

$$I_M = C_M \, dVs/dt$$

Where: Vs is the source voltage.

Figure 6.7 illustrates the effects of trace impedance on crosstalk coupling. The higher the impedance of the victim trace, the more susceptible it is to noise from crosstalk. Figure 6.8 shows a similar effect but with varying the width of the trace line. Wider traces produce less crosstalk coupling. Therefore, minimum coupling is created with maximum spacing, maximum trace widths, and minimum impedance.

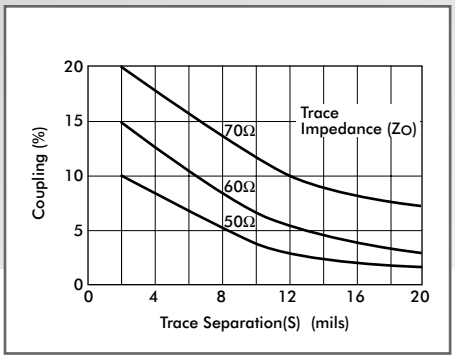


Figure 6.7 Crosstalk vs. Impedance and Spacing

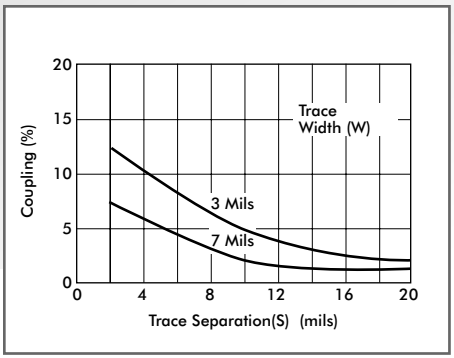


Figure 6.8 Crosstalk vs. Width and Spacing

To minimize the effects of magnetic field coupling, three basic rules should be followed.

First, separate the traces with more distance. The effect that is seen is directly proportional to the square of the distance of the elements and therefore doubling the distance will reduce the coupling by a factor of four.

The second method of decreasing the coupling is to shield the target trace. Either routing it on another layer or placing protective or guard traces between the two does this. In this way the magnetic lines will have a portion of their energy developed in the protective traces, usually at ground potential, and the field that cuts across the trace being shielded is less.

A third way to reduce the impact of crosstalk is to use differential signaling. This type of signal rejects noise from crosstalk if both the true and complement signal are equally effected. This is commonly called common mode noise rejection. However, if the crosstalk affects one trace of the differential pair and not the other, the noise coupling can be a factor.

Guard Traces

In some instances, victim traces need to be shielded or guarded from source traces to prevent crosstalk. This is not often the case as increasing the signal separation will solve the crosstalk issues. But in stubborn cases or when the absolutely lowest noise environment must be obtained, guard traces should be considered. In their simplest form, a guard trace consists of a single grounded trace between a source signal trace and the victim clock trace. The guard trace should be grounded on both ends.

While not prevalent in digital logic designs, these structures have found much use in the area if guarding sensitive analog signals. However, with reducing signaling levels due to higher frequency clocks, the effective crosstalk noise becomes an increasing concern. As the signaling level decreases, the percentage that this noise represents (signal to noise ratio) becomes worse. Noise whose magnitude is 100 mV riding in a 3.3-volt clock is only

3%, or a 33:1 S/N ratio. If this 100 mV signal exists in a differential clock that only swings 1.0 volt, then it becomes 10% noise or a 10:1 S/N ratio problem. This can easily cause an unwanted clock. Therefore, care must be taken when designing with smaller signal swings to avoid crosstalk.

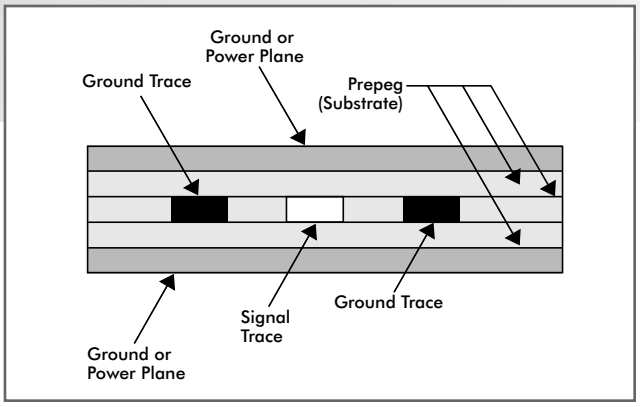


Figure 6.9 Guard Traces

Layers

In order to keep noise at a minimum and to achieve the best in signal integrity, control of the PCB layers is required.

As was mentioned before, spacing the power and ground planes as close together gains additional bypassing capacitance. Of more importance is to create a reference plane that provides constant trace impedance. The reference plane is usually the ground plane. The thickness of the substrate between controlled impedance traces and the ground plane must be selected to be both manufacterable and able to keep the desired characteristic trace impedance within acceptable limits. As the spacing of layers becomes less, the ability to hold tight, repeatable spacing values decreases. A 1 mil variation on a 10 mil spacing is 10% and on a 20 mil spacing it is only 5%. You should always consult with your fabricator on the tolerances.

When power planes are correctly bypassed, they are at the same AC potential as the ground planes. Therefore, they can be used as a reference for controlled impedance traces. Using both power and ground planes however, is unwise as the transition between the two has a high potential of causing noticeable impedance discontinuities in the traces impedance and this is a source of unwanted reflections.

Vias in Traces

When routing a clock trace, the signal should remain on the same signal plan. Using a via to route to other plane diminishes the signal quality of the clock. Vias should therefore be avoided.

Vias exhibit resistance, capacitive reactance and inductive reactance. Because they are composed of granular copper they exhibit more resistance than an equal length of PCB trace. The largest signal degradation that vias cause to the clocks that pass through them is due to their inductance and it comes from the granular copper that is plated into the via holes. Inductance is a function of frequency and is greater with faster speed clocks. Adding a via has a similar effect as adding a resistor in the path whose value increases with a higher rate clock.

If vias cannot be avoided and have to be used in a clock trace, the following guidelines should be considered:

1. Make vias as large as possible (more area equals less inductance and resistance).
2. Plate external layers with the maximum thickness of copper during fabrication.
3. Plating or filling the via holes solid.

EMI Generation

PCB traces act as small antennas to the clock signals on the traces. In practical terms, the goal is to provide a "poor" antenna to prevent unwanted EMI.

There are a few trace configurations that should be avoided. If a trace, otherwise an antenna, is created whose length is 1/4, 1/2, 1 or any other integer multiple of one times the electrical length of the fundamental frequency of the clock, then a close to ideal radiating source will be created. This trace will have along its length a standing voltage wave of maximum (or near maximum) voltage magnitude. This produces the optimal configuration to cause the maximum Electro-magnetic coupling to the surrounding area. If the trace is on the surface of a PCB then the area on three sides is air. It can also couple with other radiating traces and hence raise the level of EMI. (See Chapter 9 for more details on EMI.)

Differential Clock Traces

Differential clocks require some special attention to achieve the most benefit from its inherent properties. This type of clock signal is often used to eliminate the effects of noise in the system but this effect can be voided with poor signal routing. When using differential clocks, the following rules should be followed:

1. The traces (true and complement) must be physically equal in length. To construct them otherwise will cause a phase lead or lag condition. This will degrade the duty cycle or clock-to-clock period and cause a bimodal clocking condition. In this situation there are two distinct half cycle periods created in the resulting clock.
2. Treat the signals alike. If you must place a via to transition between layers, place the vias electrically at the same point (within a minimum of 0.005 inch occurrence in each trace).

3. Keep the traces constantly spaced. It is important that they maintain this spacing as they round corners.
4. Keep the two traces (true and complement) close to each other. Because they are differential does not mean that they are immune to external magnetic noise fields (crosstalk). If one trace is closer to the source than the other it will have a highest voltage induced in it. Keeping the trace pairs as close as possible will minimize this effect.

Conclusion

PCB layout is an important aspect of design especially when implementing high-speed digital clocks. The layout can affect the signal response, termination, EMI, and a host of other factors. Care needs to be taken in both the layout of the power planes as well as the signal layout. It is important to remember that traces, vias, and planes have different characteristics when high-speed speed signals are present. They no longer act the same when a DC signal is applied. With the higher frequencies, these materials act as inductors and can attenuate the signal. This can severely impact the operation of the design. Layout can also affect other factors such as crosstalk and EMI. Excess noise can be coupled into signals on the board or into the external environment causing unwanted inference. By understanding how these signals operate and influence each other, and by following a few ground rules, a reliable, quite design can be achieved.

7 Clock Termination

A primary goal of any timing generation and distribution circuit is to have clean, reliable clock edges at the destination of the trace. Without proper termination, impedance mismatches will occur and reflections will be evident. The resulting reflected pulses may be large enough to falsely trigger a device input. This can lead to disastrous consequences especially when the signal is a clock.

There are several methods that can be used to terminate clock traces each with a variety of tradeoffs. There are choices that impact signal integrity, power, and board area. We will discuss each alternative and address its tradeoffs. Before analyzing the specifics of terminations, let's first look at what happens when an electrical wave propagates down a transmission line with no termination. Figure 7.1 shows an ideal line with a clock source (driver) and some device (load) at the end. This is an ideal model because no parasitic elements are assumed.

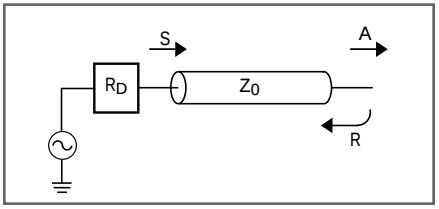


Figure 7.1 Ideal Transmission Line

Where: S is the Signal into the Transmission Line
R_D is the Impedance exhibited by the Driver Source
Z_O is the Trace Impedance
A is the Signal Accepted into the Load
R is the Signal Reflected from the Load

When the clock driver output is a signal whose voltage is V, then signal strength S is equal to: $V * Z_O/(Z_O + R_D)$. This wave will continue to the load end of the transmission line where part of it will be accepted and the remaining part will be reflected. The accepted portion into the Load can be calculated as $A_L = 2 * Z_L/(Z_L + Z_O)$ where Z_L is the load impedance and the portion reflected back toward the beginning of the line is determined by: $R_L = (Z_L - Z_O)/(Z_L + Z_O)$. The wave of magnitude R_L will continue propagating until it encounters the next impedance change, which is at the source. Again, there will be an accepted and a reflected function. The reflected function from the source is $R_S = (R_D - Z_O)/(R_D + Z_O)$. And this wave will propagate towards the load end of the trace.

Notice that a wave will continually bounce back and forth on the trace until it eventually dampens out. If the reflected wave is substantial and is near the threshold value of the load component, a false clock can be realized. Also note that the equations allow the signal strength to be either positive or negative and therefore satisfy the situations where a signal can be over-damped as well as under-damped.

Low Impedance Driver

Let's look at a situation where the source impedance is less than the trace impedance. This is typical of a standard or high drive clock buffer into a typical trace. We will assume that the output impedance of our clock driver is 17 ohms and the trace impedance is 50 ohms. We will also assume that the input impedance of our load Z_L is very high and representative of typical loads. If the output of the clock drives a voltage of V then the signal into the transmission line is:

$$S = 50 \text{ ohms}/(50 \text{ ohms} + 17 \text{ ohms}) * V = 3/4\ V$$

This equation tells us that we have $3/4$ V propagating down our trace. When this signal reaches the end of the trace, it will have two components: the accepted signal and the reflected signal. The accepted signal (A_L) is the voltage seen by the load device and the reflected signal (R_L) is sent back to the source.

$$A_L = 2 * 1M \text{ ohm}/(1 \text{ M ohm} + 50) * 3/4\ V = 1\ 1/2\ V$$

$$R_L = (1 \text{ M ohm} - 50 \text{ ohm})/(1 \text{ M ohm} + 50 \text{ ohm}) * 3/4\ V = 3/4\ V$$

In this situation, the device at the end of the trace actually sees $1\ 1/2$ V at its input and $3/4$ V is transmitted back to the source. This is the case of a clock signal that exhibits overshoot. Figure 7.2 illustrates the effect of this impedance mismatch. The Round Trip Time is the propagation delay of the trace times two thus equally the time it takes for the signal to travel down the trace and back.

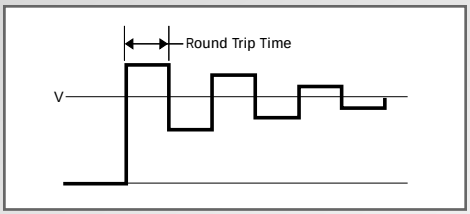


Figure 7.2 Under-dampened Clock Trace

The reflected wave ($^3/_4$ V) will propagate back to the source until it encounters the 17 ohm mismatch. It then will have the accepted and reflection components as we described for the end of the trace. The reflected wave can be calculated as:

$$Rs = (17 \text{ ohms} - 50 \text{ ohms})/(17 \text{ ohms} + 50 \text{ ohms}) * {}^3/_4 \text{ V}$$
$$= -{}^1/_2 * {}^3/_4 \text{ V}$$
$$= -0.375 \text{ V}$$

Notice two key factors in the reflected wave: its magnitude is still of considerable value and the signal is negative. This negative signal will propagate back towards the load end of the trace and again exhibit accepted and reflected properties. The load will see a signal alternating above and below the desired voltage V while approaching V. The width of the pulses is equal to two times the trace delay.

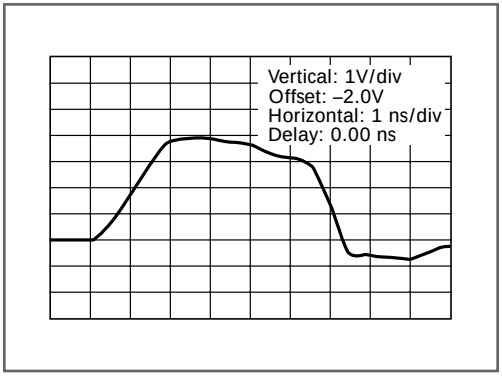


Figure 7.3 Unterminated Clock Trace
Single Clock Driver (19.8 Output Impedance), 6-inch Long Trace, 51.7 ohm Stripline (Effective), Single Load

High Impedance Driver

Now let's look at the opposite case where the driver has a higher value than the trace impedance. This depicts a situation where the clock driver may be very weak and has a slow rise time.

We will assume that the output impedance of our clock driver is 150 ohms and the trace impedance is 50 ohms. And again, we will also assume that the input impedance of our

load is sufficiently high. If the output of the clock drives a voltage of V, then the signal into the transmission line is:

$$S = 50 \text{ ohms}/(50 \text{ ohms} + 150 \text{ ohms}) * V = 1/4\ V$$

This indicates that $1/4$ V propagates down the transmission line. When this signal reaches the end of the trace, it will again have two components: the accepted signal and the reflected signal. As before, the accepted signal is the voltage seen by the load device and the reflected wave is sent back to the source.

$$A_L = 2 * 1 \text{ M ohm}/(1 \text{ M ohm} + 150) * 1/4\ V = 1/2\ V$$

$$R_L = (1 \text{ M ohm} - 150 \text{ ohm})/(1 \text{ M ohm} + 150 \text{ ohm}) * 1/4\ V = 1/4\ V$$

In this situation, the device at the end of the trace only sees $1/2$ V at its input and $1/4$ V is transmitted back to the source. This is a case of undershoot on the clock signal. Figure 7.3 illustrates this condition.

The reflected wave ($1/4$ V) will propagate back to the source until it encounters the 150 ohm mismatch. It then will have the accepted and reflected components as we described for the load end of the trace. The reflected wave can be calculated as:

$$R_S = (150 \text{ ohms} - 50 \text{ ohms})/(150 \text{ ohms} + 50 \text{ ohms}) * 1/4\ V = 1/2 * 1/4\ V = 0.125\ V$$

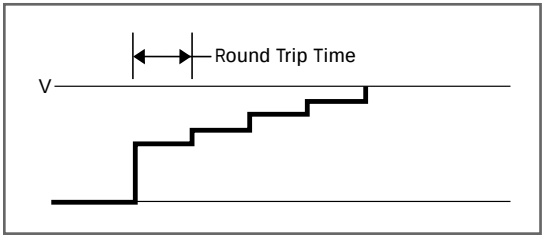


Figure 7.4 Over-dampened Clock Trace

This positive signal will propagate back toward the end of the trace and again exhibits accepted and reflected properties. The effect it ultimately has is a stair-stepping signal approaching the value V. The width of each stair is equal to twice the signal trace delay.

Notice that in both the low impedance driver and the high impedance driver cases, the signal at the load end is quite different than the signal at the source end. Since the load is the device that will accept and react to the clock signal, we want to optimize the signal at this point. Ideally, we want to have a single point-to-point connection for each clock on our board to achieve the best signal quality. This can generally be accomplished using clock fanout buffers. However, there are times when we may want to drive multiple loads from a single driver.

When to Terminate

Before we address how to terminate, let's first answer the question of when to terminate. For clock signals, it is best practice to always terminate the trace. This will prevent any unwanted pulses from occurring and prevent unwanted false triggers. If circumstances prevent proper termination then the trace delay must be short as compared to the edge rate of the clock. A general guideline is the trace delay must be at least six times faster than the rise time of the clock. Any trace longer than that ratio must be terminated or reflections will become a factor. All clock traces should be simulated either with IBIS or Spice model simulation to ensure proper termination.

Source Termination

By far, Source Termination is the most popular form of termination for the typical clock circuit. Source termination, or more commonly known as Series Termination, uses a resistor placed in series with the trace as close to the source as possible. The intent of the resistor is to match the output impedance of the clock driver to the impedance of the trace. This allows the reflected wave to be absorbed when returning.

There are several key advantages to Series Termination. First, it is simple to implement and requires little board area. Second, it does not create a constant DC current flow and therefore doesn't consume excessive power. And third, it is relatively easy to calculate the necessary resistor values.

However, there are a few drawbacks for Series Termination. Series Termination can only be used if the load devices are at the end of the trace. We will see why when we discuss the wave propagation. The other drawback is it may be difficult to achieve a perfect termination since we are trying to match the output impedance of a clock driver and the output driver impedance changes throughout its I-V curve and therefore the impedance value is variable.

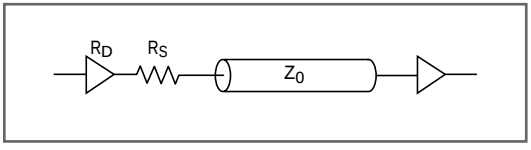


Figure 7.5 Series Terminated Trace

Since we want to match the impedance of the trace to the impedance of the driver, we add the resistor RS to the trace. We want to set the value such that the output impedance of the driver (RD) plus the Series resistor (RS) is equal to the trace impedance (ZO). This can be expressed in the following equation:

$$R_D + R_S = Z_O$$

Clock drivers have some amount of impedance, and in general, the lower the output impedance, the faster the signal will transition. To determine RS we need to know RD. Some

component datasheets will specify the value to be used for R_D for a particular driver. It can also be derived from the IBIS model of the device. Many IBIS model simulators extract the value for easy access or you can browse the model itself to get the value.

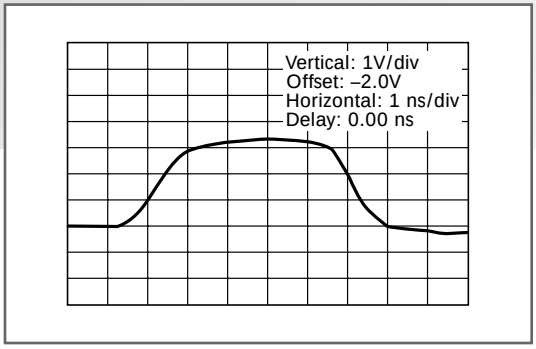


Figure 7.6 Series Terminated Trace

Single Clock Driver (19.8 Output Impedance), 31.9-ohm Series Resistor, 6-inch Long Trace, 51.7 ohm Stripline (Effective), Single Load

When the output buffer drives the clock signal, the wave will propagate along the trace. Since there is an impedance mismatch at the point of R_S and Z_O, we have both reflected and accepted properties. Since R_S was chosen such that R_S plus R_D is equal to Z_O, the transmitted voltage will be $^1/_2$ that of the output driver – a voltage divider. The half wave will propagate down the trace to the end where it will experience its reflected and accepted functions. The amount that will be accepted and reflected is:

$$A_L = 2 * 1\ M\ ohm/(1\ M\ ohm + Zo) * {}^1/_2\ V = V$$

$$R_L = (1\ M\ ohm - Zo)/(1\ M\ ohm + Zo) * {}^1/_2\ V = {}^1/_2\ V$$

The input of the load device will have the full voltage V and the $^1/_2$ V is propagated back to R_S.

Notice we have several different impedance changes in our model. There is the output driver impedance, the series resistor, and the trace impedance. Reflections and acceptances will occur at every impedance boundary. It is therefore imperative that R_S is sufficiently close to the output driver or a complex set of reflecting waves will occur. The distance that is tolerable is dependant on the rise time of the signal. The faster the rise time, the closer the resistor needs to be to the source. In order to have the reflections become a non-factor, the trace propagation time between the source and R_S should the one-tenth of the signal rise time. For example, if we have a clock signal that has a 2 ns rise time then the flight time between the driver and the resistor should be no greater than 200 ps. If the trace characteristics are such that the propagation delay is 175 ps/inch, then the maximum distance R_S should be from the driver is 1.14 inches.

In a Series Terminated trace, the receiver "sees" driver impedance equal to the trace impedance Z_O. Knowing this relationship, we can determine the rise time of the signal at the receiver based on the RC time constant.

$$tr = 2.2\ Z_O\ C_L$$

where: tr is the rise time of the signal from 10% to 90%
Z_O is the trace impedance
C_L is the load capacitance

This equation gives us the fastest rise time possible at the receiver with a series terminated trace. Further, we can add the rise time of the driver (t_D) to provide us with the actual rise time at the receiver (t):

$$t = (t_D^2 + t_r^2)^{1/2}$$

The power dissipated by series termination is small compared to other types of termination. However, it isn't easily derived. In order to calculate the power dissipated by R_T , our series termination resistor, we need to determine its voltage drop. The difficulties are due to the fact that the voltage across the resistor is changing based on several factors. The first reason is the obvious: V_{OH} and V_{OL} are changing. However, since the series termination resistor acts as a voltage divider when the signal first propagates down the line and remains in that condition until the wave returns, we have a dependency on the length of the trace. Once the reflected pulse returns, we no longer have the same voltage drop and hence a changed power dissipation. In conjunction with the length of the trace, clock frequency is also of importance. For every clock cycle, a voltage drop is presented across the resistor. We can then express the power dissipated as:

$$P_T = f * 2T * (V_{OH} - V_{OL})^2/2R_T$$

Where: PT is the power dissipation in the termination resistor
f is the clock frequency
T the is the trace delay time
R_T is the value of the termination resistor

End Termination

There are several types of end termination: the Split Termination (also known as Thevenin equivalent), Parallel Termination, AC Termination, and several others. Most of these methods use resistors to match the trace impedance at the end of the trace instead of the source. We will focus on these three and how they apply to clock signals.

End termination differs from Series termination in that the trace is terminated at the end of the trace. This results in little to no reflections returning back to the source. This allows for populating additional devices along the trace (which may cause signal reflections and introduce other termination issues).

Thevenin Equivalent

Thevenin Equivalent uses a pull-up/pull-down pair of resistors at the end of the trace. The parallel combination of the two resistors is set equal to the impedance of the trace. However, care must be taken as to not exceed the high and low drive currents (I_{OL}, I_{OH}) of the driver. In Figure 7.8, a simulation of a Thevenin terminated trace is shown. Notice that the signal does not reach the voltage rails due to the termination.

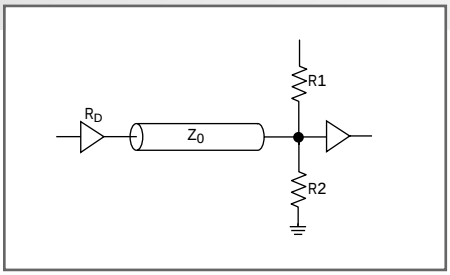


Figure 7.7 Thevenin Equivalent Termination

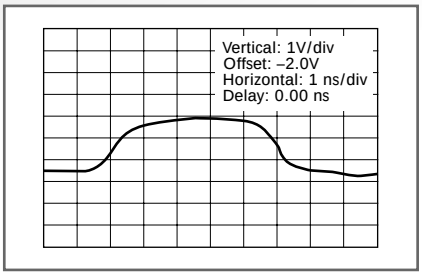


Figure 7.8 Thevenin Terminated Trace
Single Clock Driver (19.8 Output Impedance), 103 ohm Thevenin Resistors, 6-inch Long Trace, 51.7 ohm Stripline (Effective), Single Load

In order to prevent any reflections, the junction of R1 and R2 must be equal to the value of the trace impedance. The values of R1 and R2 can be expressed with the following formula:

$$\frac{1}{R1} + \frac{1}{R2} = \frac{1}{Zo}$$

If the output driver is symmetrical and provides equal drive for high as well as low, then R1 = R2. If they are not equal, then the values need to be adjusted to prevent overloading the driver. We can express the I_{OH} and I_{OL} with the following two equations:

$$I_{OH} < (V_{CC} - V_{OH})\ 1/R1 - (V_{OH})\ 1/R2$$

$$I_{OL} > (V_{CC} - V_{OL})\ 1/R1 - (V_{OL})\ 1/R2$$

Notice that the expression for I_{OH} typically results in negative numbers on either side of the inequality since the driver is sourcing the current.

To determine the rise time at the receiver in a Thevenin terminated trace, we use an R_C time constant as we did with series termination. However, there are a few differences. Instead of Z_O, the receiver "sees" the trace impedance Z_O in parallel with the thevenin equivalent of Z_O and hence the result is $Z_O/2$ ohms. Applying this to our R_C equation we notice that the rise time of the signal can be twice as fast as the series terminated trace.

$$tr = 2.2\ (Z_O\ /2)Cr = 1.1Z_O\ Cr$$

Further, we can add the rise time of the driver (t_D) to provide us with the actual rise time at the receiver (t):

$$t = (t_D^2 + tr^2)^{1/2}$$

The Thevenin Equivalent termination resistors are always consuming power. For clock signals, we can safely assume a fifty percent duty cycle and hence half the time the output is V_{OH} and the other half its V_{OL}. We can then determine the power dissipated by each resistor.

$$P(R1) = ((V_{CC} - V_{OH})^2 + (V_{CC} - V_{OL})^2)/2*R1$$

$$P(R2) = (V_{OH})^2 + (V_{OL})^2/2*R2$$

Parallel End Termination

Parallel End Termination uses only a pull-down resistor at the end of the trace. The value of this resistor is equal to the trace impedance Z_O to terminate the signal. The advantage this method has over the Thevenin equivalent circuit is the constant consumption of power is eliminated. However, care again must be taken to ensure the output driver is not overloaded.

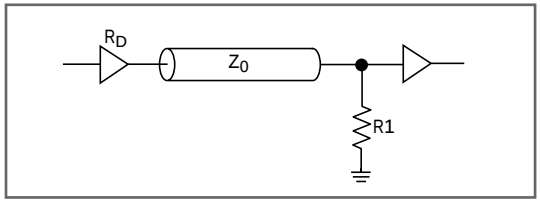


Figure 7.9 Parallel End Terminator

In this case, I_{OH} (which is negative) must be able to supply the necessary current through the terminating resistor based on the following equation:

$$I_{OH} < - V_{OH}/R1$$

The rise time at the receiver for the parallel termination is the same as the Thevein equivalent termination method.

$$tr = 2.2 (Z_O /2)Cr = 1.1Z_O Cr$$

Further, we can add the rise time of the driver (t_D) to provide us with the actual rise time at the receiver (t):

$$t = (t_D^2 + tr^2)^{1/2}$$

The power dissipated for the pulldown circuit is only one component of the Thevenin Equivalent power calculation. Since we only have one resistor to ground then the power is equal to the follow equation:

$$P(R1) = [(V_{OH})^2 + (V_{OL})^2]/2R1$$

AC Termination

AC Termination is a variation of the parallel termination and addresses the constant dissipation of power. AC Termination adds a capacitor between ground and the termination resistor to prevent the constant flow of DC current. This is accomplished by charging the capacitor to a voltage halfway between the driver's high and low voltage. The voltage across the termination resistor will be cut in half and thus saves power. In order for the capacitor to charge correctly, the signal must be DC balanced meaning the signal must be transitioning equal periods of high states as well as low states. This is easily satisfied for our typical clock signal.

The value of R is the same as in the Parallel Termination method and is set to equal Z_O. In order to select C, we first must know the period of the clock present on the trace. We need to ensure that the capacitor doesn't significantly discharge during the high or low portions of our clock cycle. We therefore set C such that the RC time constant is much greater than that of the period of the clock. Although this method sounds simple enough, it does not work well with long traces. It is also difficult to simulate with many IBIS model simulators since several clock cycles need to occur in order to charge the capacitor. For this reason, series termination or one of the other end termination methods may be a better choice.

AC Termination Part II

Some designers will use a variation of the AC termination described above. Instead of using the capacitor to provide a biasing voltage for the period, the capacitor is used block the flow of DC current. During the transition of a clock signal, the capacitor provides a path to ground that allows the termination resistor to operate properly. However, when the signal reaches the high level, there will be no current flow through the termination resistor to ground because of the capacitor. This prevents the dissipation of power during the high (and low) states of the clock. Note that the driver must be able to supply the needed output drive current during the signal transition through the resistor to ground. The value of the capacitor is small and typically 10 pF. However, the signal waveform should be measured to ensure proper operation.

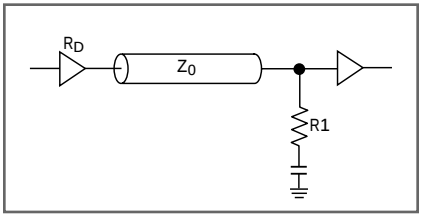


Figure 7.10 AC Termination

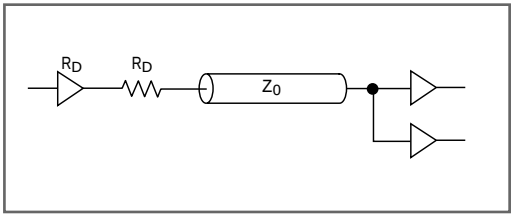


Figure 7.11 Multiple Loaded Trace

Multiple Loads

We've covered the point-to-point single ended clock cases with the most common termination techniques, but what about multiple destinations? Ideally, clock signals should have a single source for every load device, but there are times when two or more loads may be required. We will look at several of the common scenarios and address the areas of concern.

Suppose we have two devices that we want to drive and both are very close physically to each other. There is only one clock driver output available and no option to add a larger buffer or second component. We also assume the trace is sufficiently long and therefore acts as a transmission line. We then have a single driver and a single trace with multiple loads as show in Figure 7.11.

If we use the Series termination method, Rs is calculated as the difference between the output impedance of the driver and the effective trace impedance. To prevent unwanted reflections from occurring due to the multiple loads, we must first analyze the stub length (which is the short trace segment between two separate destination loads) and signal rise time.

If the distance between the two loads is short, then the two loads will act as one capacitive lumped load. However, if they are not, then we will have an un-terminated stub and our signal will reflect at each point and result in multiple accepted and transmitted waves — basically, a poorly terminated trace. So the key question is how close do the loads need to be? That answer depends solely on the rise time of the driving signal. A good rule of thumb is: the length delay of the trace stub must be six times faster than the rise time of the signal. This guideline allows the transitioning signal to propagate up and down the stub rapidly enough to appear as a single pulse.

The rise time at this load will be different than a single load. If you recall from the earlier section with the single load, the rise time can be expressed as:

$$t_R = 2.2\ Z^*C$$

Our capacitance has increased because there are two capacitive loads in parallel and thus doubling the value of C. There is also additional trace capacitance due to the small stub connecting the two loads. Multiple loads at the end of the trace will work provided the stub is short and the rise time is still suitable for your application. However, it is always recommended to simulate your design.

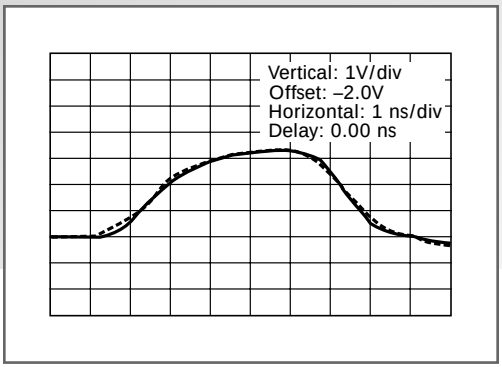


Figure 7.12 Multiple Loaded Trace

Single Clock Driver (19.8 Output Impedance), 31.9-ohm Series Resistor, 6-inch Long Trace, 51.7 ohm Stripline (Effective), Two Loads 1-inch apart

Another common way to implement series termination driving two loads is to use two termination resistors and two separate traces.

This allows the loads to be physically far apart from each other with each segment terminated. The calculation for determining Rs is a bit different than the single series termination trace. In this case, we have the source driving a series resistor and trace impedance in parallel with another series resistor and trace. We can express this as:

$$R_D + RS1 \,||\, RS2 = Z_O \,||\, Z_O$$

Notice when we add the second load, the trace impedance is halved since it is viewed in parallel. This results in the values of Rs to be less than the single trace termination. Note that if the parallel trace impedance is less than the output driver impedance, the trace cannot be properly terminated. Let's take a moment to review the wave propagation in the dual trace scenario. The driver will launch a signal with amplitude V into the trace. At the load side of the series termination resistors, $^1/_2$ V will begin to propagate down the trace.

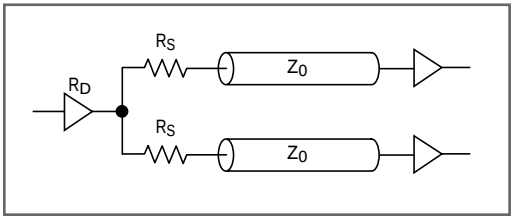


Figure 7.13 Dual Traces

When the signal encounters the end, it will reflect back (the load will see V) propagating back towards the series termination resistor. Once encountered, some signal portion will be reflected and some will propagate beyond the resistor. In order to minimize continuing reflections, the overall trace lengths need to be equal in length. Also, the termination

resistors must be close to the source as given by the guideline previously in series terminated traces. If the trace lengths to the resistors have a flight time slower than one-tenth the rise time of the signal, this segment will act as a transmission line by itself. This will cause a reflection when the propagating signal encounters the resistors. This scenario is known as a bifurcated trace and should be avoided.

Care also needs to be exercised when selecting the clock driver. As each trace is added in parallel, the effective impedance is lowered. If this value is smaller than the output impedance of the clock driver, then no value of Rs will be able to properly terminate the trace. Therefore, select a driver with output impedance that will allow for termination. As in the case before, it is advisable to simulate the design with an IBIS or Spice simulator to verify the trace will perform as expected.

Differential Signals

Differential signaling is gaining popularity in timing circuits as a result of ever increasing clock rates. Differential signaling becomes a real benefit when distributing a clock along a backplane from one card to another due to its ability to tolerate ground voltage shifts. However, there are several different types of differential drivers and to complicate the situation, many published methods for terminating.

For distributing clocks within a system, LVDS, LVPECL, and Differential LVCMOS are the most popular drivers used today. We will focus on these styles and address the proper termination.

Differential signaling propagates two signals along the board traces. The second signal is equal in amplitude and opposite to the polarity of the first and the receiving input switches based on their difference. Two major benefits as a result. First, if noise occurs within the system and it affects both signals, it is canceled out as a result of the difference of the signals at the receiver input. This noise is known as common mode noise in that it is common to both signals. The second major benefit is that the ground reference between the two devices can vary without causing a false trigger. This again is due to the fact that the receiver is comparing the two input signals with each other independent of a voltage reference. The degree of noise rejection by the receiver is dependant of the type of signaling and the termination method used.

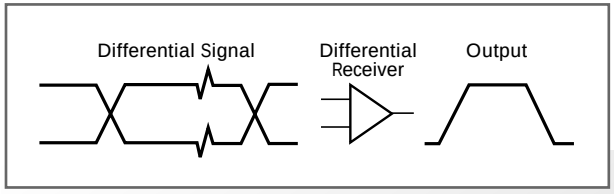


Figure 7.14 Differential Signals

LVDS

Low Voltage Differential Signalling (LVDS) is defined by two industry standards: the IEEE-Scalable Coherent Interface (SCI) and the ANSI/TIA/EIA-644 specification. The EIA-644 defines the electrical characteristics of the signaling and is generally followed for most LVDS applications.

The output stage of an LVDS driver is a defined as a current source. Therefore, termination is not only required to eliminate unwanted reflections, it is also necessary to provide a voltage differential for the receiver inputs.

The voltage swing range of LVDS is from 250 mV to 450 mV with 350 mV as the typical value for the driver. Due to its small voltage swing, LVDS is able to reach faster transition speeds than that of LVCMOS or LVTTL. The specification allows for 622 Mbps but a variety of devices have exceeded these speeds. Another characteristic of LVDS is that it centers it voltage swing at 1.2V and is known as its common mode voltage V_{CM}. This provides an additional margin for noise originating on the power and ground planes.

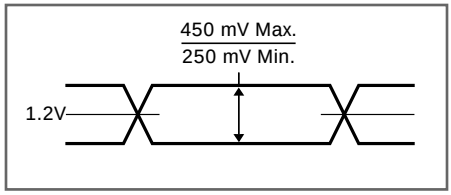


Figure 7.15 LVDS Signal Levels

There are many published methods for terminating LVDS circuits that include no components to as many as half a dozen. We will examine many of them and highlight the benefits of each.

The first type of termination is no external termination. This assumes that the chosen LVDS receiver has an internal resistor to provide the necessary differential voltage for the comparator. The range of value for the internal resistor is typically from 100 ohms to 120 ohms. The benefit of this configuration is obvious: no components necessary. With a fixed termination value inside the receiver, the proper value cannot be chosen to match that of the impedance of the trace. Thus, reflections may occur on each of the signals.

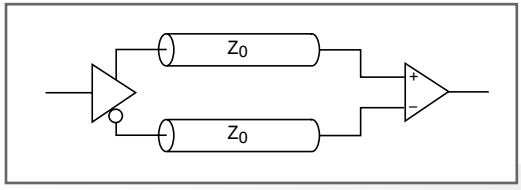


Figure 7.16 LVDS Internal Termination

The most common termination found on LVDS traces is the single resistor terminator. This resistor provides the necessary differential voltage for the input of the receiver and also terminates the two signals to each other. The value of the resistor is chosen based on the impedance of the trace:

$$R_T = 2 * Z_O$$

In can be thought of as having two resistors equal to $^1/_2$ R_T in series and the center tap as having a floating ground. For LVDS, this is actually 1.2 volts. The signals will propagate down the trace having equal but opposite voltages centered on 1.2V. When the signals encounter the termination network, it appears as equal to the trace impedance and hence no reflection occurs. As the two signals collide, they cancel each other and hence do not continue to propagate down the opposite trace.

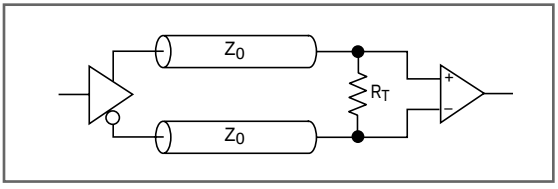


Figure 7.17 LVDS Single Termination

But let's examine the case where the two signals are not exactly aligned. This can be a result of mismatched trace lengths, variations in board material, output driver skew, or a combination of these fluctuations as well as other sources. In this case, the two signals propagate down their respective traces but they are separated in time. As they encounter the termination resistor no reflections occur however, they do not cancel each other out. The signal that arrives early will begin its journey back to the source on the other trace until it encounters the other signal. It will partially cancel but not fully. The signal arriving later will still reach the termination resistor slightly attenuated because of the collision of the other signal, but it will not completely dissipate in the resistor network. It will also begin to propagate back to the source but on the other trace. The magnitude of the two signals is a function of the time difference T as it relates to the rise/fall time of the pulse. As each signal arrives at the source, it will encounter a low impedance driver and thus reflect back down the line. Depending on the magnitude, this could be substantial. If the trace length delay aligns with the clock period, these pulses could be become quite large. Therefore, special attention should be afforded to signal skew especially for LVDS clock signals. Alternative solutions may also be used to dampen the unwanted pulses.

One method used to address the skew between the differential traces is to add a capacitor. The resistor R_T can be split in two each being $^1/_2$ it's original size and equal to Z_O. A capacitor is then added at the mid point. If the differential signals are truly equal and opposite, only a DC value of V_{CM} will be realized at this junction. However, if the signals are skewed, an AC signal will appear. The capacitor is used to terminate these extraneous signals. The size of C is chosen such that the RC time constant is slightly larger than skew between the signals.

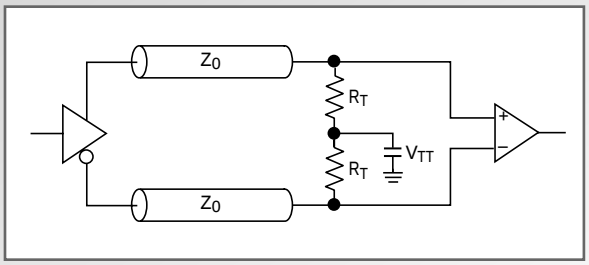


Figure 7.18 LVDS Termination

In order to provide superior LVDS receiver operation, the inputs are ideally referenced to 1.2V. If we use two resistors instead of one, we are also able to improve the characteristics of the termination. The connection at the two resistors is connected to VTT, the termination voltage that is set equal to the VCM. Note that VTT isn't connected to ground and doing so would cause the receiver to malfunction.

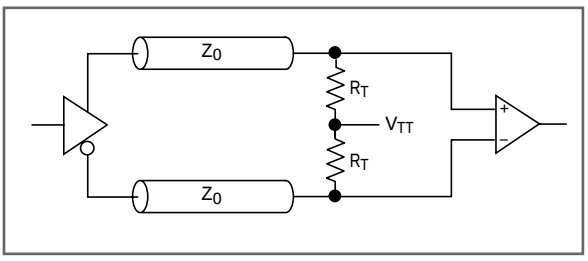


Figure 7.19 LVDS With Voltage Termination

LVPECL

LVPECL (Low Voltage Positive Emitter-Coupled Logic) is a variation of ECL but uses positive supply voltage instead of negative voltages. LVPECL outputs have specific loading requirements to ensure proper operation. Because of its open-emitter output design, it can source current but it cannot sink current. To allow the output to switch, some form of pull-down is required as part of the termination.

The voltage swings range on LVPECL larger than LVDS with typical VOL at 1.6V and VOH at 2.35. Similarly due to its small voltage swing, LVPECL is able to reach faster transition speeds than that of LVCMOS or LVTTL.

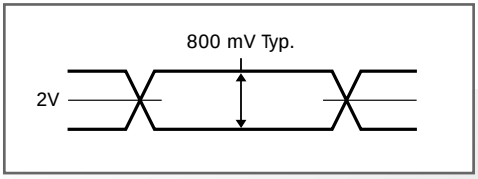


Figure 7.20 LVPECL Signal Levels

As with other types of drivers, LVPECL also needs to be terminated into a matching impedance transmission line to avoid unwanted reflections. It is differential so we must ensure all modes are terminated as with LVDS. There are several common ways to terminate LVPECL traces.

One method is Thevenin Equivalent termination. This is the same as the termination we discussed for single ended clock signals and is used for both signals. We also need to take into account biasing and therefore must select the proper resistor values for impedance matching as well as the voltage levels.

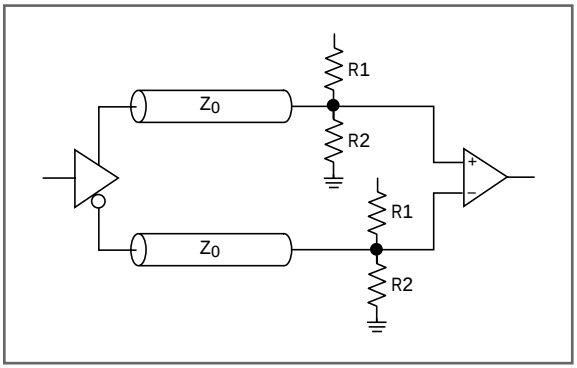


Figure 7.21 LVPECL Thevenin Termination

Because LVPECL does not swing rail to rail and we must provide a pulldown path, a termination voltage VTT is required. This value is usually specified as VCC – 2V, which for a 3.3V system yields VTT to be 1.3V. This value is slightly less than VOL and thus provides the required pulldown path. Knowing VTT, we can now calculate the values of R1 and R2 for the termination.

$$R1 = (Vcc * Zo)/VTT$$

$$R2 = (Vcc * Zo)/(Vcc - VTT)$$

For example, if we have a 3.3V system with a 50-ohm trace, the values of R1 and R2 are computed to be:

$$R1 = (3.3 * 50)/1.3 = 127 \text{ ohms}$$

$$R2 = (3.3 * 50)/(3.3 - 1.3) = 82.5 \text{ ohms}$$

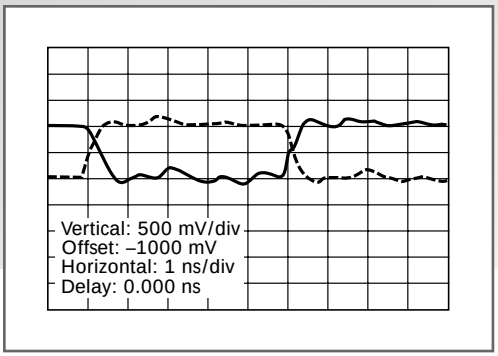


Figure 7.22 LVPECL Signal

Differential Clock Driver (12.3 Output Impedance), 50-ohm Resistors to 1.3v Vtt, 4-inch Long Trace, 49.1 ohm Stripline (Effective)

The Thevenin Equivalent termination design will yield excellent results for clock signals since all modes are terminated. No reflections will occur between each signal or back through any of the resistors to the other signal path. As in the case with the single ended signals, the termination resistors need to be near the receiver. The ideal position is just past the receiver along the trace.

The disadvantage to the Thevenin Equivalent termination is the amount of power dissipated. Since it provides a resistive path from V_{CC} to ground for each signal, current is constantly flowing. The power dissipated for a 3.3-volt system in this configuration is nearly 52 mW for each signal or 104 mW for the differential pair.

$$Power = V^2/R = (3.3)^2/(127 + 82.5) = 52\ mW$$

An alternative to achieve lower power is the Parallel Termination method or also known as Shunt Bias termination. This method uses a pulldown resistor equal to Z_O on each signal to the termination voltage V_{TT}. This type of biasing requires an additional voltage supply of 1.3V to provide the V_{TT} level. The least amount of power is dissipated using this method. If we use V_{OH} and V_{OL} as 2.35V and 1.6V respectively for a 50% duty cycle clock, then the power is about 11 mW for a 50-ohm trace.

$$Power = {}^1/_2 * (V_{OH}–V_{TT})^2/R + {}^1/_2 *(V_{OH}–V_{TT})^2/R$$

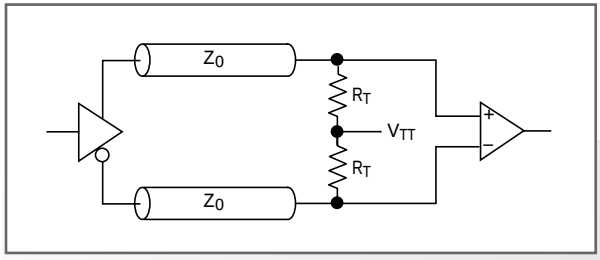


Figure 7.23 LVPECL Shunt Bias Termination

In some systems, V_{TT} of 1.3V may be difficult to obtain and a termination method that doesn't require a unique voltage can be beneficial. The Y-Bias configuration connects both of the termination resistors for each differential signal to a single load resistor. Since the net sum of the current in the two signals remains constant, the voltage drop across this load also remains constant. R_L , the load resistor, is selected to provide adequate biasing.

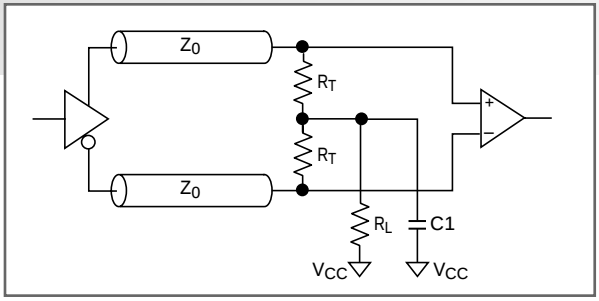


Figure 7.24 LVPECL Y-Bias Termination

R_L is determined by summing the currents of a logic 1 and a logic 0 and then calculating the resistance to provide an equivalent V_{TT}.

$$I_{OH} = (V_{OH} - V_{TT})/R_T$$

$$I_{OL} = (V_{OL} - V_{TT})/R_T$$

$$R_L = V_{TT}/(I_{OH} + I_{OL})$$

For a 3.3V system, we can simplify and combine these equations.

$$R_L = R_T$$

Therefore, the value of R_L should be set to the termination value. At DC and low frequencies, the power supply is assumed to be a short circuit. However, at very high frequencies, the power supply appears as near infinite impedance. A 0.1 µF to 0.01 µF capacitor is connected to V_{CC} to create an AC short at high frequencies.

Shunt Bias with Pulldown

Probably the most versatile and easy to implement termination network for LVPECL buffers is the Shunt Bias with Pulldown. This method starts with the standard Shunt bias resistors whose R_T value is equal to the trace impedance. Instead of a separate power supply to source V_{TT}, a voltage divider resistor network is used. Capacitor C1 is used to provide an AC path to ground in the event the differential signals have skew. This value typically ranges from 0.01 µF to 0.1 µF. The final piece of this circuit is a pair of 150-ohm pulldown resistors at the source. Because LVPECL drivers are open emitter and do not sink current, some ringing may occur in the high-to-low transition. These resistors will help prevent this

condition. Also, for designs where the signals are driven through a backplane to another card, they will provide the proper pulldown necessary for the driver if the receiving card is unpopulated.

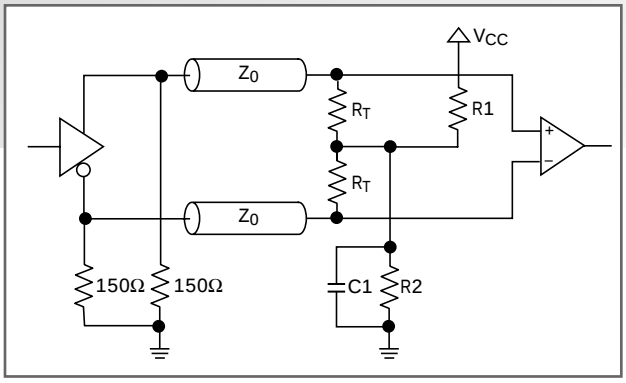


Figure 7.25 Shunt Bias with Pulldown

In Figure 7.26, the simulation of this design is shown. The DC signal on the scope trace just above 2 volts is the voltage divider of R1 and R2. By adjusting this bias voltage, the signal levels can be optimized for the specific LVPECL receiver in the design. The VTT bias voltage applied is slightly higher than the Shunt Bias method due to the 150 ohms pulldown resistors. The other two signals are the inputs to the differential receiver.

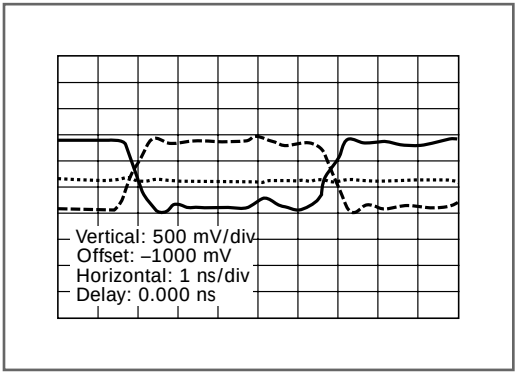


Figure 7.26 LVPECL Shunt with Pulldown
Differential Clock Driver (12.3 Output Impedance), 50-ohm Resistors to bias of 2v Vtt, 4-inch Long Trace, 49.1 ohm Stripline (Effective), two 50-ohm pulldown resistors.

Terminating Unused Outputs

Clock buffer devices may have more outputs than needed for a design. We have addressed the various termination techniques for loaded outputs so now let's focus on what should be done with the unused pins. For devices whose output structure requires termination in order to transition such as LVPECL, unused outputs can be left unattached. For all other types of output drivers we may need additional termination.

The two areas of concern with unused outputs are EMI and clock skew. Additional EMI results from the unused pin acting as a small antenna and radiating the transitioning signal. This can be of consequence if the pin size is large or if there are many pins together that create a small grid. However, most modern package styles have geometrically small pins and a few unconnected outputs have no appreciable impact on EMI.

For applications where pin-to-pin skew is critical, then un-terminated unused outputs may be of consequence. There are two different factors than can affect the output skew of the device. For very fast signals, those that are operating at less than 1 ns rise times, frequency harmonics can feedback into the device. This causes unwanted noise within the component similar to power supply noise. The result is excessive skew on the outputs. A second factor than affects the output skew is a delta in the output driver current. However, this is very dependent on the internal design of the component. If the clock generator or buffer has a single bank output, meaning all outputs have the same internal voltage feed, there is no difference in the current due to unloaded outputs. But with devices with multiple banks and a difference in current between the two banks, skew between the banks (and their respective outputs) can increase. This is a result of one bank with more loads than its neighboring bank due to unterminated outputs. This will cause the output driver banks to draw different amounts of current and thus increase the skew. The magnitude of the skew is small and typically is less than 30 ps.

When small amounts of EMI or slight variations in skew needs to be avoided, unused outputs should be addressed. Many clock buffers have output driver enables either controlled by dedicated inputs or through a serial control bus. These conditions can be managed by disabling unused outputs. Without driver enables, the other option is to terminate. Termination can be accomplished with either a capacitor or a resistor. By using a small capacitor to ground, generally 5–10 pF, the output is terminated into a lumped load. The capacitor also slows the edge rate to keep EMI to a minimum. Instead of the capacitor, a resistor to ground can also be used to terminate the output. The value of the resistor is chosen large enough as to not exceed the I_{OL} of the driver. The disadvantage of the resistor design is that it continuously consumes power whereas the capacitor design does not.

A good practical approach to unused outputs is to place pads for either the capacitor or resistor termination in the design. Keep the trace lengths very short on the buffer output pins to prevent unwanted EMI. During the initial testing of the design, analyze the output skew of the clock buffer to ensure it is within tolerance. If there is excess skew observed then populate the termination components.

Conclusion

Clock signals are the foundation of any high-speed digital design. In order to have the circuit operate without fail, signal integrity is required. With either single ended or differential signals, the goals are the same: eliminate unwanted reflections. Reflections caused by changes in impedance can be managed by selecting a suitable termination technique. After the termination technique has been chosen and the circuit sketched, it is always recommended to simulate the design.

8

Cascading PLLs

Engineers are often faced with questions surrounding the use of cascaded phase-locked loops. *Can I cascade two PLLs to generate a frequency and distribute it to my ASIC or microprocessor? How many PLLs can I cascade before I run into problems? What kind of problems will I run into when I put several PLLs in series?* Unfortunately, the answers to these questions can be very complicated and they are highly dependant on the design of the PLLs chosen. To make matters even more difficult, digital engineers who typically are the ones implementing the clock circuits are rarely exposed to the types of analog interactions these components exhibit. This chapter will address these issues but will remain practical by avoiding complex equations and mathematical analysis. The focus will be on demonstrating the interaction of phase-locked loops, understanding key terms, and assessing the risks of cascading PLLs.

A Cascaded PLL

A cascaded PLL is simply two or more phase-locked loops in series that produces an output clock. A simplified diagram of a cascaded PLL design is shown in Figure 8.1 with two phase-locked loops (PLL) where one PLL performs a frequency multiplication and the other performs zero-delay clock distribution of the multiplied clock.

Phase-locked loops of the past were normally found as standalone components. In the 1990's, in a quest for faster signal processing, PLLs started to become embedded in a variety of devices. They are now used in clock generators, microprocessors, microcontrollers, ASICs, digital signal processors, oscillators (some programmable oscillators use embedded PLLs), and many other components. PLLs dramatically improve the data valid windows by adjusting clock skew and therefore accommodate the needs of short cycle time clocks.

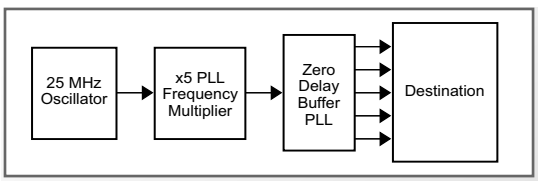


Figure 8.1 Cascaded PLLs

They also allow slower frequencies to be multiplied to achieve must faster clock rates. This has led to the huge popularity of PLLs in high-speed digital designs. However, when adding a PLL into a design, care must be taken to ensure tolerances are not violated at a system level. When more than one PLL is used, the interaction of the PLLs can become complex and the analysis is at times difficult. PLLs have a closed-loop response that has an impact on the jitter transfer characteristics. They can actually amplify noise or jitter both can be source both internally and externally. The largest gain experienced in the frequency region is commonly called jitter peaking. When cascading like PLLs, jitter peaking grows exponentially from stage to stage. The effects of cascaded PLLs and jitter peaking on clock performance may range from not having any effect on system operation to rendering the system non-functional.

Acquisition and Tracking

PLLs are characterized as having two states: phase-locked or acquiring lock. Tracking is synonymous with the locked condition and simply describes the extent to which the loop can follow variations in the input clock frequency. PLLs operate on the phase of signals and therefore are susceptible to changes in the clock edges on the inputs. The transient response of a PLL is generally a very complex, non-linear process. In general terms, the PLL will follow the presence of a slowly occurring signal at the input and does not react to rapidly occurring transitions (frequencies outside of the PLL's loop bandwidth). Understanding how the noise spectrum propagates through the PLL is addressed in our discussions of jitter accumulation, jitter peaking, tracking skew, and phase noise.

Acquisition is the process the PLL undergoes to align the output to the input phase. Although most designs have wide margins for acquisition time there are situations where this period is important. When a PLL operates near the point of instability, the PLL lock time can become excessive. It should also be noted that noise associated with the voltage supply also has an impact on both the acquisition and tracking of the PLL.

Jitter Accumulation

The accumulation of jitter is typically measured as Long-Term Jitter (refer to Chapter 4 for a complete discussion on jitter). A consequence of cascading PLLs is jitter accumulation. Every phase-locked loop has the inherent characteristic of transferring a spectrum of noise from the input to the output. Unfortunately, it also has the inherent characteristic of adding noise. These two characteristics are referred to as Jitter Transfer and Jitter Generation, respectively. Whether a PLL adds jitter or removes jitter from a clocking path is dependent upon whether the device eliminates more noise than it generates.

Devices that attenuate jitter have loop filtering configurations that eliminate unwanted clock jitter from the input without creating excess jitter in the process. These devices are typically marketed as low-noise or low-jitter PLLs. Their primary function is to eliminate high frequency noise. However, they tend to create unwanted noise at lower frequencies. The characteristic measurement that is the most meaningful is the plot of the closed loop gain for a PLL. This is referred to as the jitter transfer curve.

In Figure 8.2, a PLL has been plotted for its jitter transfer characteristic. The result is a measurement of the amount of jitter transferred from the input to the output along with the jitter generated in the process. PLLs eliminate high frequency jitter, as demonstrated by the roll-off beyond 2 MHz. For this particular PLL, the loop bandwidth is approximately 2.5 MHz where the output is –3dB. At frequencies within the loop bandwidth the jitter is magnified from 0 dB to nearly 2dB. The highest point of magnification is the jitter peak.

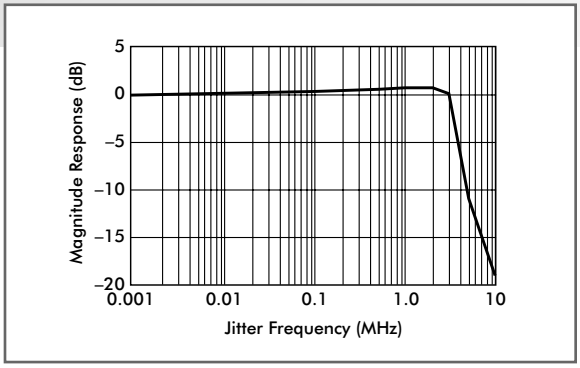


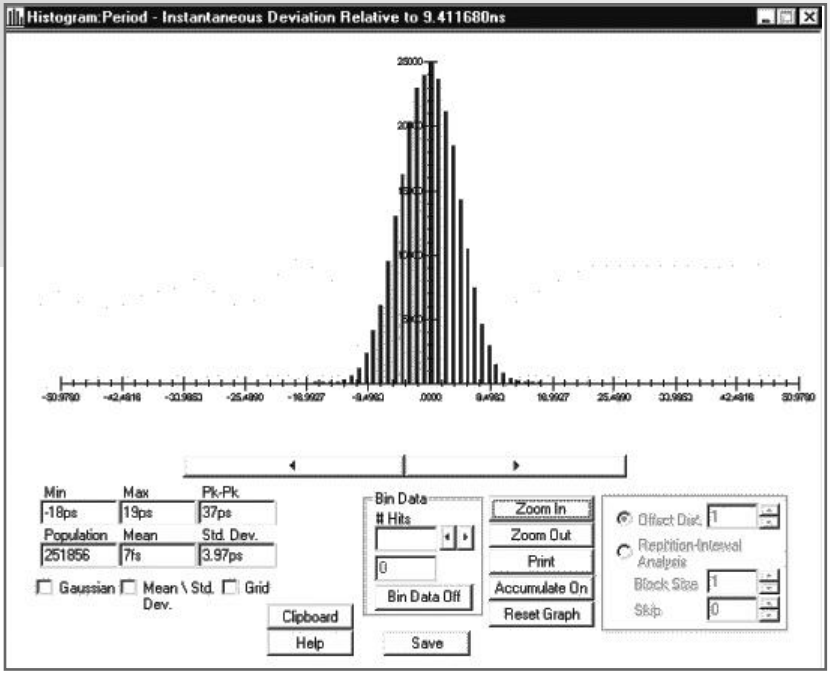
Figure 8.2 Jitter Transfer Curve

Figure 8.3 shows a set of histograms with jitter accumulation through a series of phase-locked loops. These are generated with a Time Interval Analyzer (TIA) that measures cycle-to-cycle jitter over an extended period of time. The first of the four plots shows the jitter performance of a 106.25 MHz clock source. The clock oscillator provides an output with a peak-to-peak jitter of 37 ps. As the signal progresses through a series of three Cypress Semiconductor CY2308 Zero Delay Buffers, the cycle-to-cycle jitter expands from 37 ps to 55 ps through the first PLL, to 59 ps after the second PLL, and finally to 68 ps after the third PLL. This shows there is a moderate accumulation of short term, cycle-to-cycle jitter through each of the PLLs. By placing three zero delay buffers in series, the cycle-to-cycle jitter increased from 37 ps to 68 ps. For most systems today, this amount of increase is well within the margins of a system. However, we must also analyze the long-term jitter.

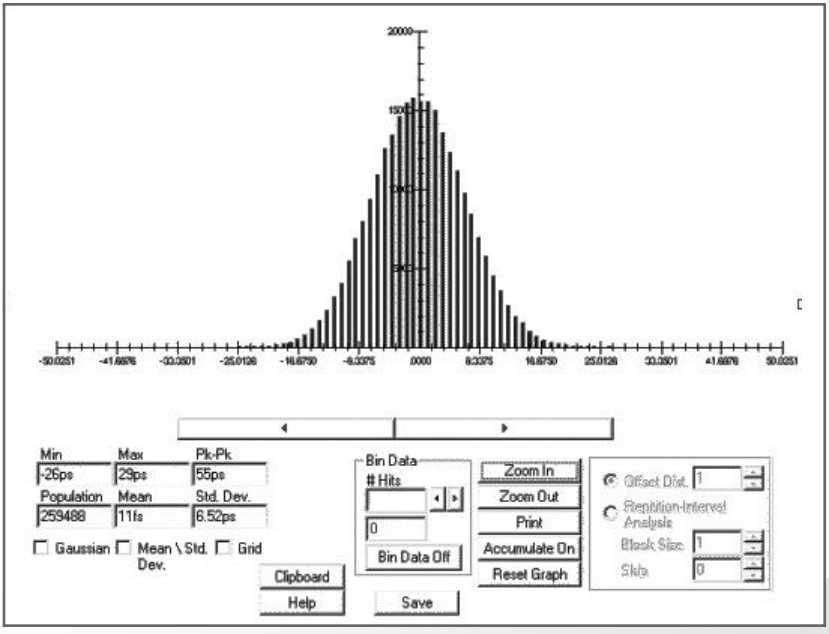
Phase Noise

While the TIA technique is typically used to measure clock jitter, a Phase Noise measurement can better demonstrate the impacts of jitter accumulation and jitter attenuation over varying portions of the frequency spectrum

Phase noise is measured in the frequency domain and is expressed as a ratio of signal power to noise power measured in a 1 Hz bandwidth at a given offset from the desired signal. A series of four phase noise plots in figure 8.4 shows the effect of cascaded PLLs in the frequency domain.

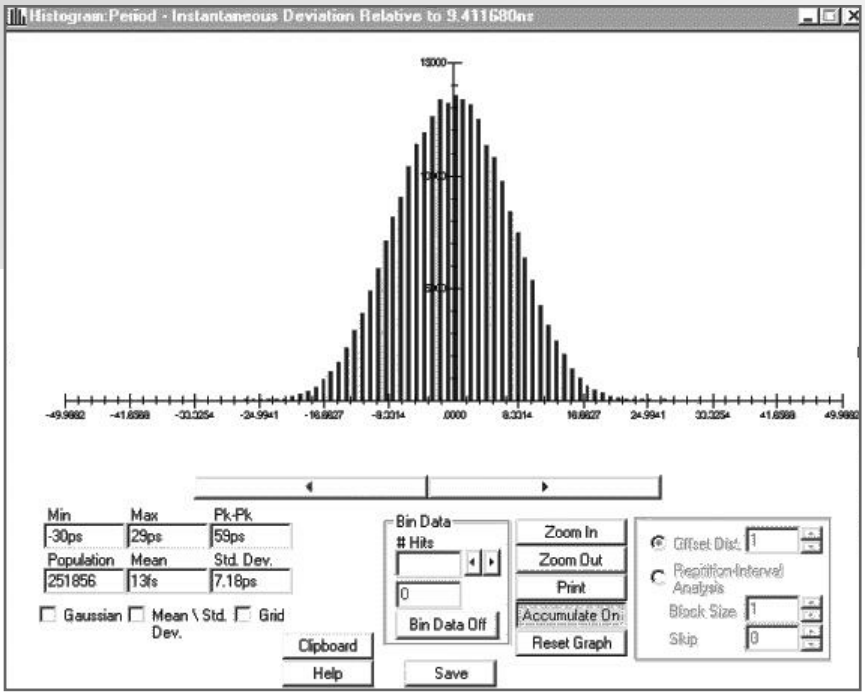


Plot 1: Clock Oscillator

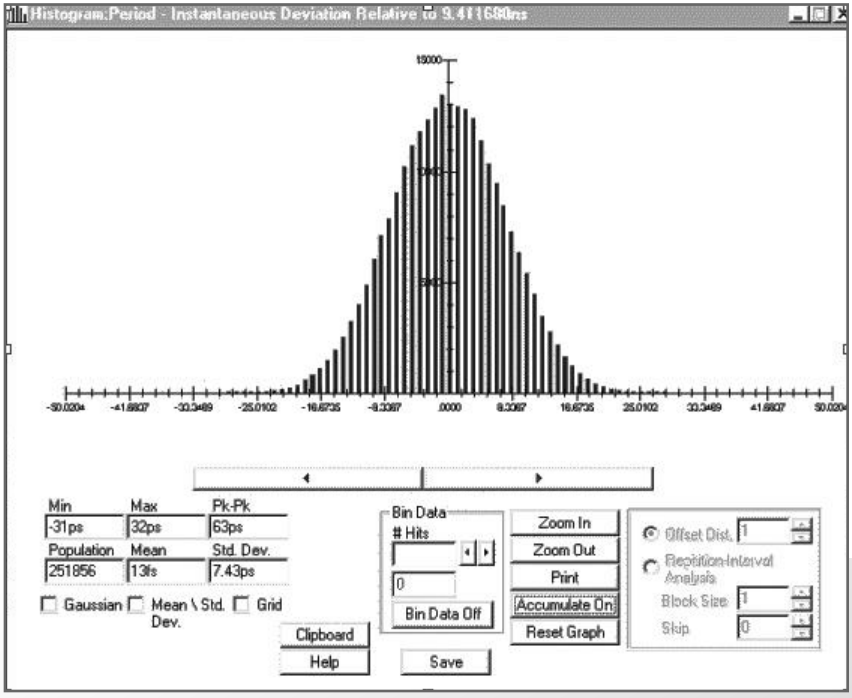


Plot 2: PLL #1

Figure 8.3 Cascaded PLL Jitter

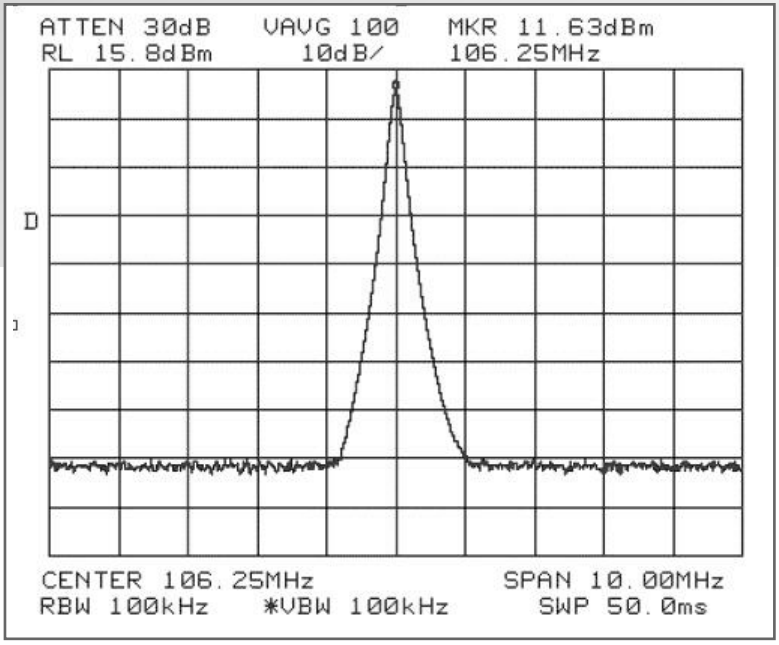


Plot 3: PLL #2

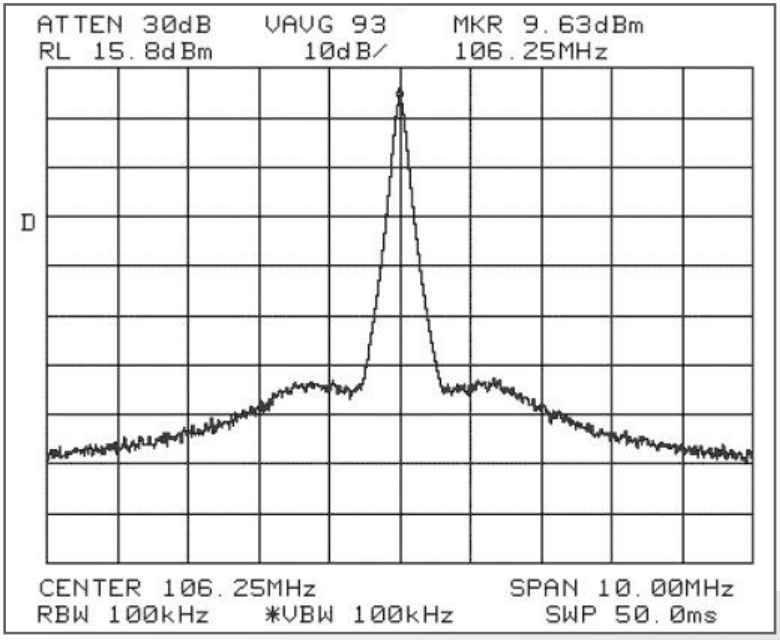


Plot 4: PLL #3

Figure 8.3 Cascaded PLL Jitter—cont'd.

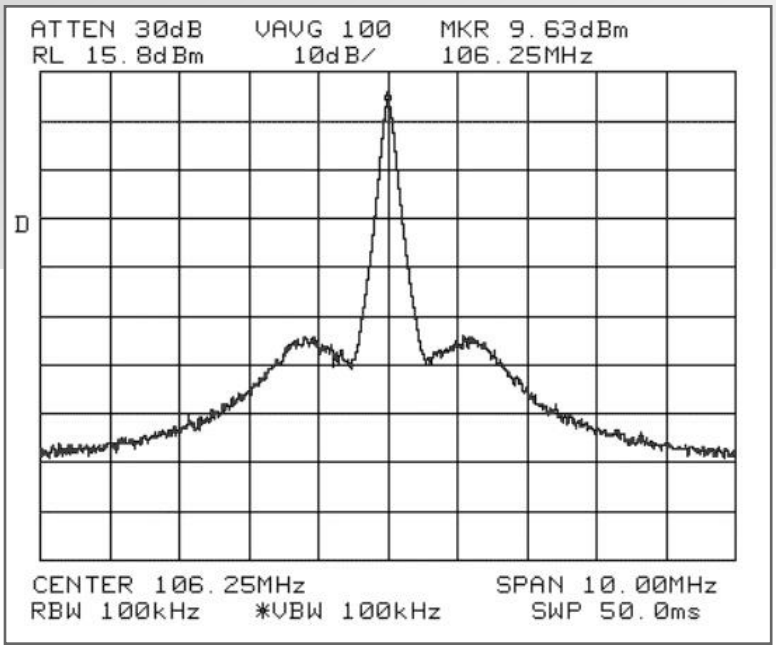


Plot 1: Clock Oscillator

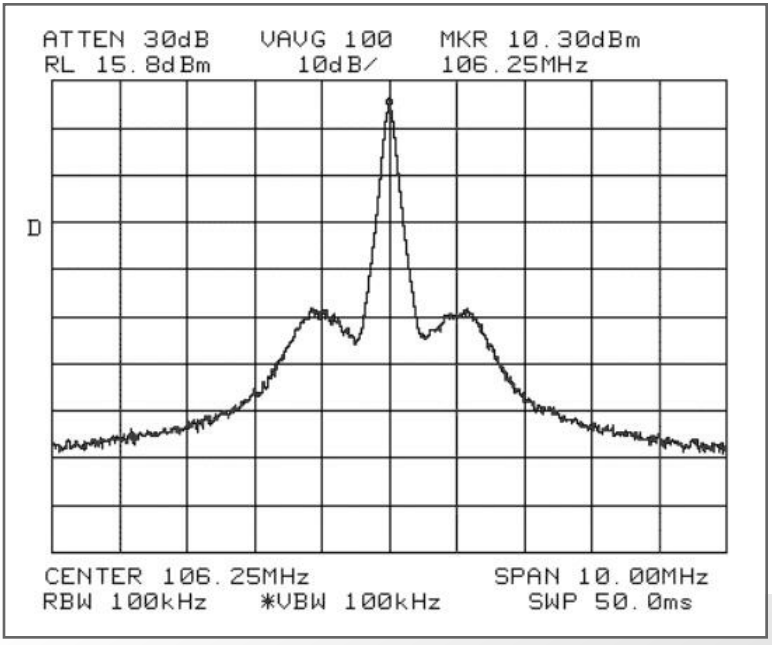


Plot 2: PLL #1

Figure 8.4 Cascaded PLL Phase Noise



Plot 3: PLL #2



Plot 4: PLL #5

Figure 8.4 Cascaded PLL Phase Noise—cont'd.

Plot 1 is the spectrally pure output of the 106.25 MHz clock oscillator. Note that the gain of the frequencies away from the center is flat. In plot 2, after the first PLL, the gain no longer is flat. The noise nearest the peak is the low frequency noise or long-term jitter. The noise at the tails is the high frequency component of the clock and is the cycle-to-cycle jitter.

The rise in the "shoulders" of the plot near the center frequency is a characteristic of the PLL. These shoulders appear at approximately 1 MHz from the 106.25 MHz carrier of the CY2308 zero delay buffer as highlighted in figure 8.5, which is the loop bandwidth of the PLL in this device. As the signal progresses through the second PLL in plot 3, the noise component again increases throughout the frequency range. In plot 4, after cascading through five PLLs, the low frequency gain has increased to nearly 30dB. Although the cycle-to-cycle jitter has a small amount of gain, the long-term jitter amplification is substantial.

One of the characteristics of PLLs is the ability to multiply. While it is not evident in the shoulders of these phase noise plots with non-multiplied outputs, there is a phase comparison that occurs during frequency multiplication that may cause a spur within the frequency domain. Multiplication also amplifies the input phase noise. The noise increases by a multiple approximately equivalent to the multiplication factor; however, it is only within the –3dB loop bandwidth of the PLL.

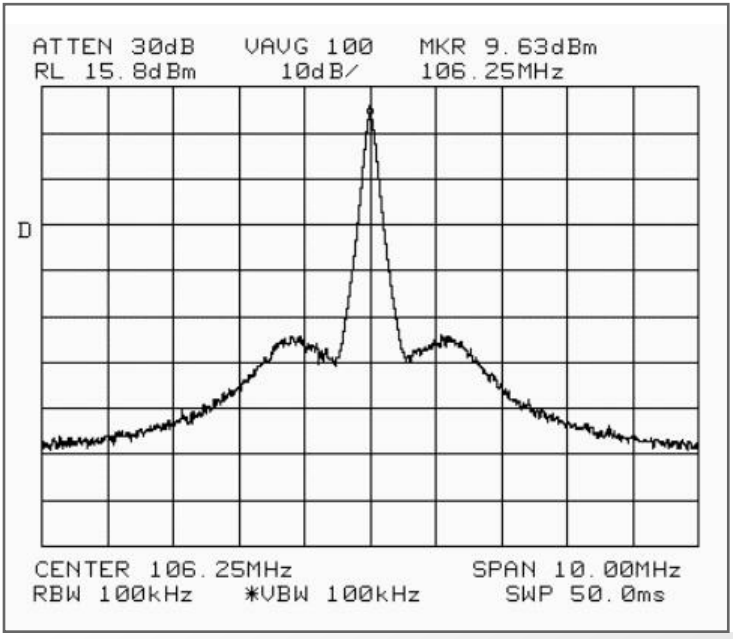


Figure 8.5 1 MHz Phase Noise Shoulders

Tracking Skew

Tracking skew is the deviation of the output of the PLL from its input. Cascaded PLLs can have an adverse effect on the amount of skew exhibited. Modulations and excessive input noise that are sometimes created by jitter peaking, can lead a phase-locked loop into a condition of instability that results in less than optimal output conditions.

To demonstrate the impact of cascading PLLs and tracking skew, the cascade of PLLs in Figure 8.4 was extended to five Zero Delay Buffer PLLs. If we analyze the output of the fifth PLL with a TIA, we can observe a very interesting phenomenon. In Figure 8.6, the output of the fifth PLL shows a peak-to-peak jitter of 71 ps. However, the spectral distribution is no longer Gaussian. There is now a dual peak distribution that indicates that the PLL is unable to track the excess phase noise being presented at the input to this fifth PLL.

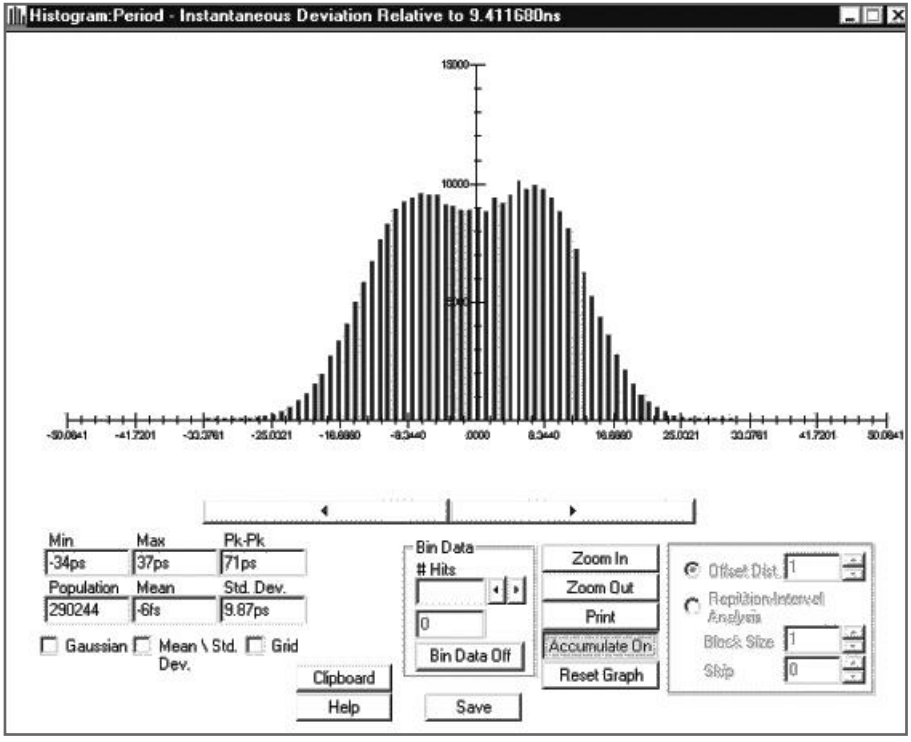


Figure 8.6 Tracking Skew in Time Domain

It should be noted that PLLs are normally capable of tracking Long-Term Jitter. PLLs, by design, are incapable of tracking cycle-to-cycle jitter, because the PLL response time is typically slow. When the modulation occurs at a rate and level that is too difficult for a PLL to track, the PLL may give a 'best-effort' tracking which we refer to as tracking skew.

Selecting PLLs

There are a variety of options when designing a clock circuit. Some include the use of PLLs and some do not. If your design requires the use of multiple PLLs, then some analysis should be performed to ensure their proper operation.

The first step is to determine if a PLL really needs to be used. If the design requires low skew outputs but no phase relationship or no multiplication is required, then a non-PLL buffer may be the best option. If its determined that a PLL device needs to be used, then determine how many PLLs are cascaded together. Keep in mind that some ASICs and microprocessors may have internal PLLs.

After determining that PLLs exists in your design, identify the intent of their use. They can be used to allow the propagation of jitter as in the case of a spread spectrum clock or they may be used to attenuate incoming period jitter. A designer should not incorporate a spread spectrum device in a design that requires low noise at the destination. Spread spectrum technology should be used for those designs where spreading the frequency distribution is useful in reducing the overall peak-power emissions (EMI) from the design.

Determining the loop bandwidth of a PLL device may be the most important consideration when cascading PLLs. The loop bandwidth of a commercially available device typically has been considered proprietary information to the device developer. However, as PLL circuit interaction has become more critical to understand, suppliers have been willing to provide loop bandwidth information. Although loop bandwidth is deterministic, it can be complicated to characterize with devices that have programmability. For instance, the Cypress Semiconductor CY22392 clock generator incorporates programmability that changes the loop bandwidth for a given output configuration.

If a jitter transfer curve is available for the chosen PLL device. The jitter transfer curve will show the jitter peaking as well as the loop bandwidth of the PLL. Figure 8.2 earlier in the chapter shows a clean curve of a PLL clock buffer where Figure 8.7 shows a PLL with some deficiencies. This particular jitter transfer curve indicates excess amounts of jitter peaking at high frequencies.

Cascading PLLs

When cascading several PLLs it is important to either determine the loop bandwidth or to have the jitter transfer curve. Because PLLs exhibit jitter peaking it is advisable to avoid cascading several devices that have similar jitter transfer curves. Unless there is a specific frequency spectrum that needs to be avoided, select devices that limit the overlap of jitter gain in their jitter transfer characteristic.

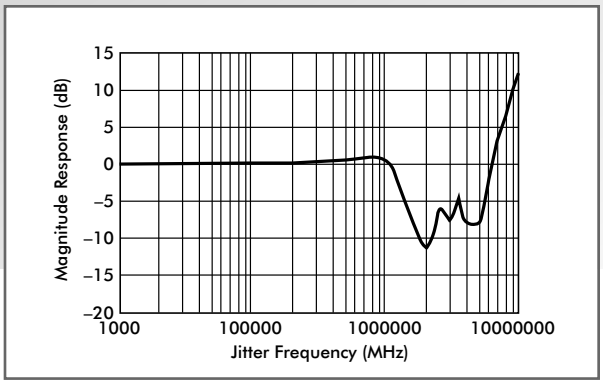


Figure 8.7 A Poor Jitter Transfer Curve

If the input reference is a spread spectrum signal then the intentional frequency modulation must not be rolled-off by the downstream PLLs. The loop bandwidth of the subsequent stages should be significantly greater than the spread spectrum modulation rate. If it is not, the spread spectrum modulation will be filtered out or the subsequent PLL outputs may have excess tracking skew. There are PLL clock buffers now available that are specifically design to pass spread spectrum signals. These devices should be considered when buffering these types of signals.

If the input is a noisy reference and requires jitter attenuation, begin with a jitter attenuating PLL that has a close-in loop bandwidth and a sharp roll-off. This will avoid the unnecessary propagation and further peaking of the jitter through successive PLLs and the power supply system. All subsequent PLLs should once again avoid significant overlap in the gain section of the jitter transfer frequency curve.

Because PLLs are in so many devices, it is sometimes difficult to obtain either the loop bandwidth or the jitter transfer curves. In this circumstance, it is necessary to measure the short term and long-term jitter produced by the cascaded PLLs to ensure proper operation. In all cases, it is advisable to perform jitter measurements.

Conclusion

While there are many complex interactions that occur with cascaded PLLs, there are those who want a simple rule of thumb. Generally, more than three cascaded PLLs should be avoided. If the design requires more than three, obtain the jitter transfer curves and arrange the devices so the jitter is attenuated. This will allow cascading six or more PLLs without creating excess jitter. For designs with critical timing margins and jitter sensitive components, measurements should be taken of the circuit to ensure proper operation.

9

Electro-Magnetic Interference

Any device capable of generating signals with frequencies in the RF range is a potential source of Electro-Magnetic Interference (EMI). These signals can cause interference in the normal operation of electronic devices such as radios, televisions, cell phones and other types of equipment. The primary sources of EMI in most systems are the clock generation and distribution circuits. In this chapter, we will discuss how EMI is generated, how to measure it, and how to minimize its impact.

Interference is caused by electro-magnetic waves that are produced by charged particles moving in an electric field. This condition occurs wherever electric signals exist. There are regulatory agencies that require devices that produce EMI to adhere to a certain set of rules and regulations. Among these rules and regulations is a requirement that the source of radiation not be greater than a pre-determined level at a certain distance from the source within a fixed frequency range. In the United States, the regulatory agency that governs the control of EMI is the Federal Communications Commission (FCC).

But what happens when the radiation sources are too great? How does the clock driving a processor affect a radio or telephone? FM radio stations operate in a frequency range of 88 MHz to 108 MHz. A 200 kHz guard band separates the stations from each other. In order to have good clarity and to receive stations from far away, radio receivers use high gain amplifiers to pick up weak signals.

A typical frequency in synchronous devices is 33.3 MHz and is often used as the clock source for PCI busses, ASICs, FPGAs, and processors. Associated with the 33 MHz is a series of harmonic frequencies. (Harmonics are integer multiples of the fundamental frequency and are discussed later in this chapter.) The 3rd harmonic of 33.3 MHz is 99.9 MHz and thus a board containing 33 MHz can cause distortion on a radio that is tuned to 99.90 MHz.

Causes of EMI

Clock sources can contribute to EMI in two ways. EMI can be produced through the repetitive nature of a synchronous clock and from an improperly terminated trace. The energy from the clocks radiates into a field through an antenna. An antenna might be in the form of: PCB traces, PCB rework wires, components with insufficient shielding, connectors, cables (shielded or unshielded), and improperly grounded equipment.

In high-speed digital devices, fixed frequency clocks are the primary source of EMI. This is due to the fact that they are always operating at a constant frequency that allows energy to increase to higher levels. Signals that are non-repetitive or asynchronous will not generate as much EMI.

As the need for higher throughput has driven faster clock frequencies, signal transition rates have also increased. But with the faster rise and fall times comes an even larger increase in the energy level of the radiated signal. Figure 9.1 shows two signals that have the same frequency, amplitude, duty cycle, and phase. However, they differ in the signal transition rate. The clock with the faster rise time will have a measurably higher amount of radiated energy than the slower transitioning signal. The second factor that contributes to EMI is an improperly terminated trace. As discussed in Chapter 7 on termination, a trace can exhibit overshoot and undershoot when there is an impedance mismatch. When this condition occurs, radiated energy will increase. Depending on the severity of the overshoot and undershoot levels, this could represent as much as 3 to 4 dB of EMI at a particular signal, or node in EMI terms. If there are ten to twenty nodes with severe overshoot then passing the FCC compliance test is in jeopardy. Figure 9.2 shows an improperly terminated trace where overshoots are evident. The same circuit with series termination contains no overshoots as shown in Figure 9.3. This clock will also exhibit less EMI.

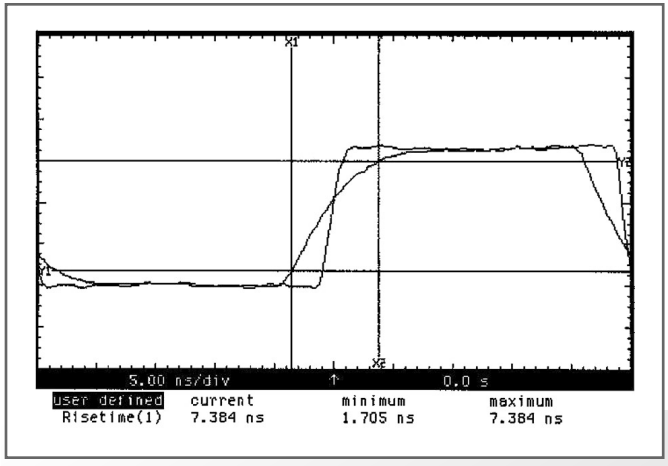


Figure 9.1

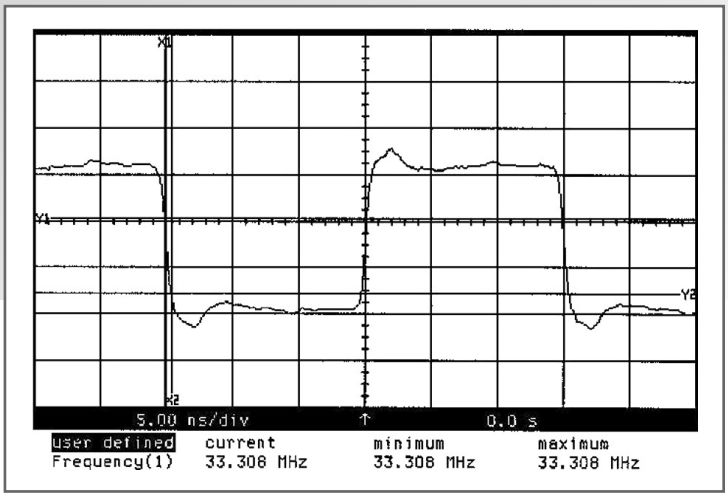


Figure 9.2 Overshoot in Signal

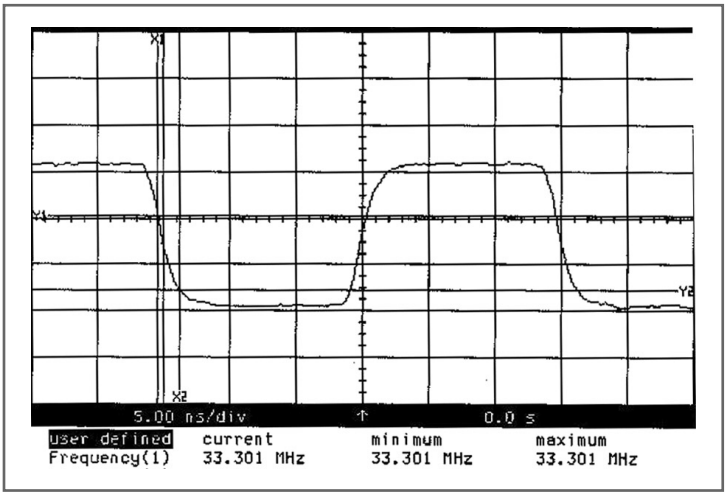


Figure 9.3 Properly Terminated Trace

Measuring EMI

Measuring EMI is accomplished by placing a radiating device in a known environment such as an Anechoic chamber or Open Area Test Site (OATS). Within this environment, the level of energy coming from the radiating source is measured and compared to the limits specified by the FCC. In 1975, Part 15 was established that relates to equipment that does not intentionally generate RF energy. The FCC specifies a maximum energy level that can be radiated at a particular frequency at a specified distance.

There are other groups responsible for regulating radio interference besides the FCC. Two noteworthy organizations are CISPR and VCCI. CISPR (International Special Committee On Radio Interference) was established in 1934 by a group of international organizations to address radio interference. It is a non-governmental group composed of National Committees of the International Electrotechnical Commission (IEC) as well as other international organizations. CISR is the standard typically followed in Europe.

The VCCI (Voluntary Control Council for Interference by Information Technology Equipment) was formed in December 1985 by four Japanese industry associations in response to a government request that electronics manufacturers participate in the control of EMI. VCCI is considered the standard for Japan.

The FCC has two main classes of radiation levels: Class A and Class B. A Class A digital device is: "A digital device marketed for use in a commercial, industrial, or business environment and not intended for use by the general public or in the home." A Class B device is defined as: "A digital device marketed for use in the home, although it could be used elsewhere." Class B levels are more difficult to meet than class A.

The following chart shows the voltage and dB levels allowed under the FCC Rules and Regulations, Part 15, for Class A and B Digital Devices.

Table 9.1 FCC Class Limits

	Class "A" (10 meters)		Class "B" (3 meters)	
Frequency (MHz)	µV/m	dB(µV/m)	µV/m	dB(µV/m)
30 – 88	90	39	100	40
88 – 216	150	43.5	150	43.5
216 – 960	210	46.5	200	46
> 960	300	49.5	500	54

When a testing lab measures a particular device, a report is generated containing a detailed chart and graph of the levels of energy measured at all frequencies from 30 to 1000 MHz. If the device being measured is compliant, the graph will show that the peak energy levels of any frequency are below the values in Table 9.1. However, if peak values exceed these limits, a change in the design is necessary to reduce the EMI.

Many IBIS simulation tools also provide the capability to analyze potential EMI hazards. While these models are not exact, they can pinpoint areas of concern during the design phase of the project. This provides more time to make the necessary corrections.

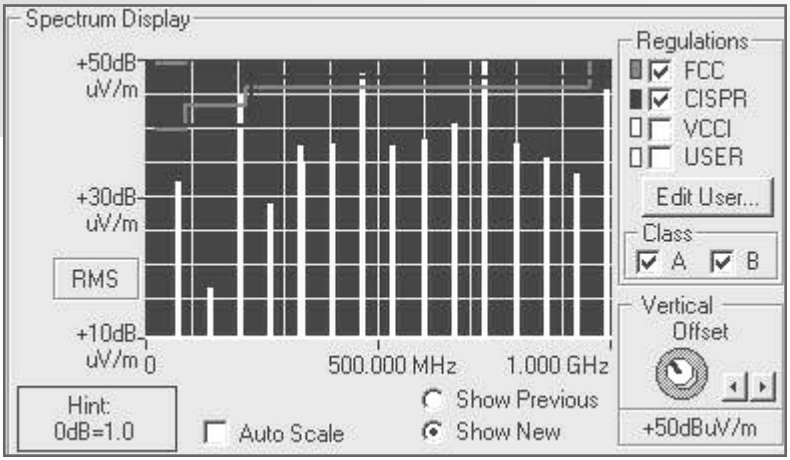


Figure 9.4 IBIS Simulation Showing EMI Prediction

Reducing EMI

There are several methods with which to solve an EMI problem in a digital system. The designer could choose to shield the design, filter a signal, or to remove the energy from the offending source. These methods could be used individually or in conjunction with others.

The first method, shielding, is not an electrical solution but a mechanical implementation. Shielding uses metallic packaging to keep the EMI from escaping the unit. This method has been used often in the past but it can sometimes be a costly solution. It also doesn't lend itself to an easy fix when an EMI problem is found shortly before a product release.

The remaining methods, filtering and energy removal, isolates the trace that is radiating the EMI. To identify which trace (or traces) is causing the problem, a test in the anechoic chamber or an EMI simulation should be performed. From this testing, an emission report will identify which frequencies exceed the specified limits. These particular frequencies are typically called hot spots. By knowing the frequencies (including the harmonics) the clock trace can be identified. The sections later in this chapter discuss frequency harmonics and how it relates to the fundamental frequency.

Since poorly terminated signals can cause hot spots, the first solution is to ensure all signals are properly terminated. The signals that are causing the EMI should be simulated and the traces should be analyzed for overshoot and undershoot. If there are exceptional amounts, then adjust the termination values to create a better waveform.

If all the signals are properly terminated and little to no overshoot is present, then the transition rate of the clock needs to be addressed. A substitution of a slower speed buffer may supply the answer. Many clock buffers have an option for high-speed or low-speed outputs. Often, these parts are either pin-for-pin replacements or the device has a programmable slew rate. If the lower drive is acceptable for the system, this may be the best solution. This method directly addresses the clock trace that is causing the problem and typically there is no additional cost to implement.

If a slower device is not available, filtering is a common way to slow the edge rate of a signal. This usually involves adding a capacitor to the signal that will soften the edge rate based on an RC time constant. The values of the capacitors are generally in the range of 5 to 15 pF. Often designers will include these capacitors, which need to be placed near the source, in their schematics but not populate them unless an EMI problem is exposed. If the clock trace uses series termination, the capacitor can be placed on either side of the resistor to reduce EMI. However, for optimal termination and signal integrity, the capacitor needs to be placed between the driver and the resistor as shown in Figure 9.5.

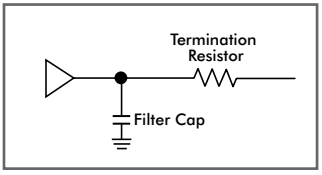


Figure 9.5 EMI Reducing Caps

Although this method reduces EMI, it does however, degrade the signal integrity of the clock. Instead of sharp, clean edges suitable for high-speed clocks, the edges become rounded. Also, capacitors may need to be added for every clock copy in the design.

Clock Modulation

Another method to address signals that emit excessive radiation is clock modulation. Clock modulation, also known as Spread Spectrum, began to appear in computer system designs as early as 1995. Most personal computers designed today use Spread Spectrum technology to reduce EMI. Spread Spectrum provides a cost effective method to keep EMI low.

Clock modulation is a technique whereby an input reference clock at some frequency is modulated such that the frequency of the output clock varies slightly. For example, a 40 MHz reference clock with Spread Spectrum applied can produce an output swing from 39.60 MHz to 40.40 MHz. This would represent a Spread Spectrum clock that has a 2% bandwidth centered on the reference frequency of 40 MHz. The purpose of modulating the frequency of a clock is to distribute the energy of a single or narrow band over a much wider band of frequencies. This reduces the amount of peak energy at any one frequency in the spectrum. The amount of EMI reduction is affected by the modulation profile, the percentage of frequency variation (bandwidth), and the modulation rate.

When a modulated clock changes from its minimum frequency to its maximum frequency, the Spread Spectrum logic applies a specific profile to the envelope that will best reduce EMI. This profile can be seen using several different types of equipment. The most effective piece of equipment to view the profile of a modulated clock is the modulation domain analyzer, which displays frequency over time. As the clock generator is increasing in frequency, the Spread Spectrum logic will cause the frequency to change only at predetermined times in the profile. The waveform of this profile is important in producing the maximum amount of dB reduction. Figure 9.6 illustrates the profile of a Spread Spectrum clock generator using the Lexmark profile, which is commonly referred to as a Hershey Kiss as it resembles the shape of the chocolate candy. Other profiles, such as a linear ramp are sometimes used to reduce EMI. However, they are found not to be as effective across the frequency spectrum as the Lexmark profile.

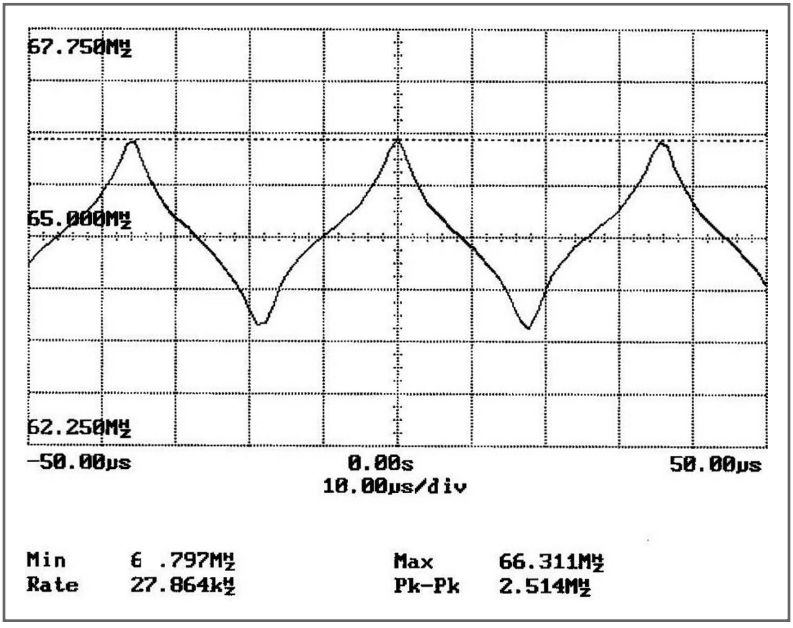


Figure 9.6. Lexmark Modulation Profile

In this particular profile example, Spread Spectrum is applied to a 65 MHz clock. The output frequency is sweeping from a minimum of 63.797 MHz to a maximum of 66.311 MHz. This is known as center spread because the 65 MHz clock is spread equally above and below its frequency. Notice that the rate of change in frequency is faster at the minimum and maximum peak frequencies and changing more slowly at the center of the frequency spectrum due to the profile.

The total amount of spread is called the bandwidth of the clock and is calculated by subtracting the maximum frequency from the minimum frequency.

$$BW = F_{MAX} - F_{MIN}$$

The amount of spread of a modulated clock is most commonly represented in terms of a percentage of the reference frequency. The bandwidth, in percent, can be calculated by dividing the total spread amount by the reference frequency times 100.

$$BW\% = (BW/F_{REF}) * 100$$

For the example in Figure 9.6, the BW and BW% are calculated as:

$$BW = 66.311\ MHz - 63.797\ MHz = 2.514\ MHz$$

$$BW\% = (2.514\ MHz/65\ MHz) * 100 = 3.87\%$$

This same 65 MHz clock can be observed on a spectrum analyzer for the purposes of determining dB reduction in the EMI of this clock. Figure 9.7 is a spectrum analyzer display of the 65 MHz clock with and without Spread Spectrum applied. The clock with no Spread Spectrum has a very narrow frequency range centered on 65 MHz. The energy peak is also higher than the other waveform. The wider scan is the clock with Spread Spectrum active. The center frequency is still 65 MHz as it is using a center spread technique. From this display, the amount of EMI dB reduction can be determined by measuring the difference between the peak energy in each of these clocks. This view shows a dB reduction of 6.48 dB at the fundamental frequency of 65 MHz. The one parameter in a Spread Spectrum clock that has the largest impact on dB reduction is the bandwidth of the modulated clock. If the bandwidth of this clock were increased, the dB reduction would increase.

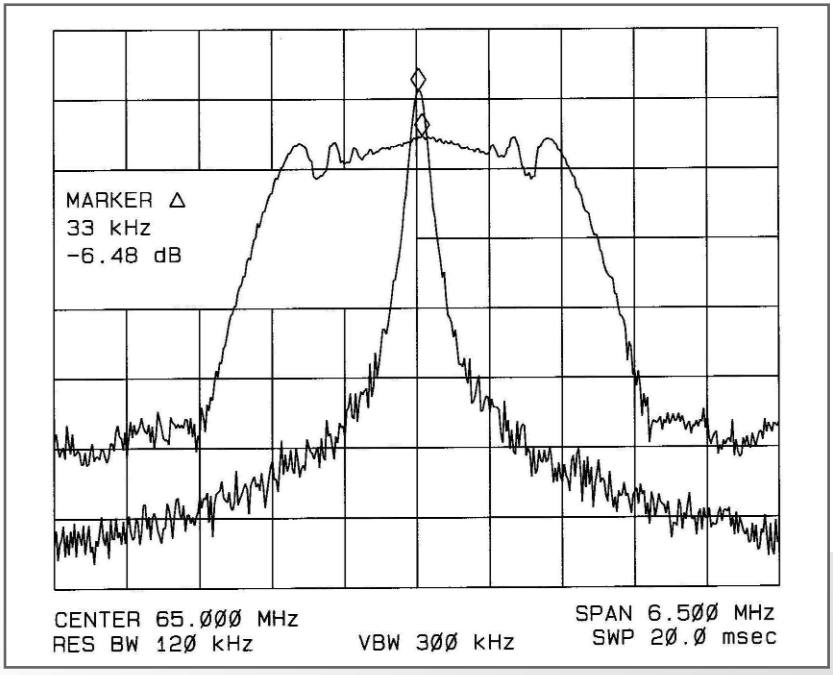


Figure 9.7 Spectrum Analysis of 65 MHz

Another key aspect with a modulated clock is the modulation rate. This is the rate at which the modulation profile repeats itself. The modulation rate of the clock in Figure 9.6 is 27.86 kHz. There are several reasons why this low frequency modulation rate is important. If the modulation rate is below 20 kHz, it is possible to generate audible noise in the system. If the modulation rate is to high, in excess of 200 kHz, the effect of modulation might be defeated by the loop bandwidth of the filters used in downstream PLL's.

The Spectrum analyzer scan in Figure 9.7 is of the fundamental frequency of this 65.00 MHz clock. Most high-speed digital designs have problems complying with EMI regulations at harmonic frequencies rather than at the fundamental frequency. Since this 65.00 MHz clock, like most of the high-speed digital clocks, is a 50/50 duty cycle clock, the odd harmonics will contain higher energy levels than the even harmonics. Performing a Fourier transform on the relative energy level of digital clocks with duty cycles from 0 to 50% can prove this fact. Figures 9.8 through 9.12 show this analysis in graph form on the first, second, third and fourth harmonics of a frequency. These graphs plot relative energy against the duty cycle of the clock.

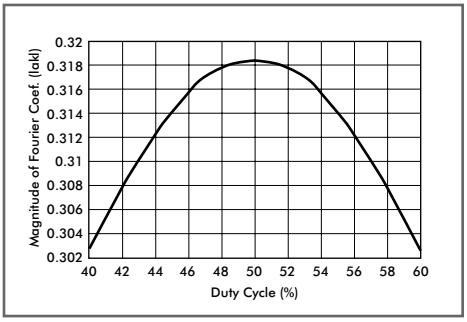


Figure 9.8 1st Harmonic

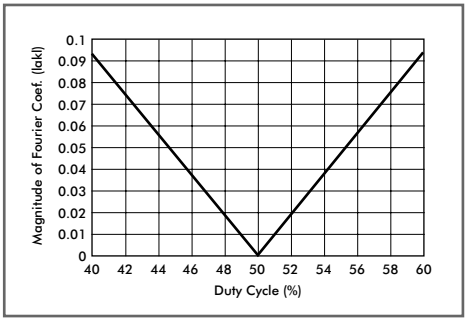


Figure 9.9 2nd Harmonic

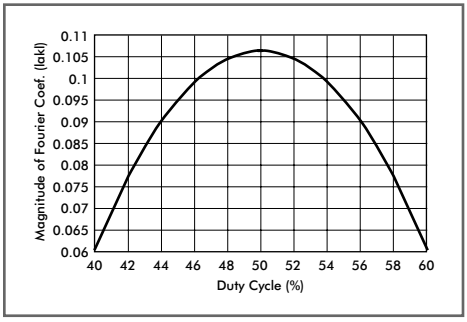


Figure 9.10 3rd Harmonic

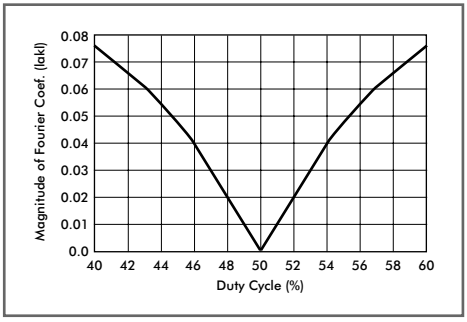


Figure 9.11 4th Harmonic

As can be seen in Figures 9.8 and 9.10, the odd harmonics have maximum energy levels at the 50% duty cycle and diminish as the duty cycle changes. However, as seen in Figures 9.9 and 9.11, the even harmonics have minimum energy levels when the duty cycle is exactly 50%.

Spread Spectrum provides a clear difference in the peak energy of a 65 MHz digital clock at the fundamental frequency. However, the problems faced by many digital systems often occur at the higher harmonic frequencies.

Harmonic Frequencies

Modulation of the 65.00 MHz clock at a total spread of 3.87 %, or 2.514 MHz has yielded a dB reduction of 6.48 dB. The harmonic frequency of a clock is an integer multiple of the fundamental frequency. The frequency of the 3rd harmonic is 3 x 65 MHz, which is 195 MHz, the 5th harmonic is 325 MHz, and the 7th harmonic is 455 MHz. Every frequency generated by the Spread Spectrum clock will also be a multiple meaning, for example that the bandwidth of the modulated clock at the 5th harmonic will be 2.514 MHz x 5 equaling 12.57 MHz. The bandwidth percent will not change but the actual frequency domain that energy is spread over will increase by a factor of five. This will cause the dB reduction of peak energy at the 5th harmonic to be much larger than at the fundamental frequency. The dB reduction does not multiply by the same factor since it's a logarithmic scale. Figures 9.12 through 9.14 show the dB reductions of the 65 MHz clock at the higher harmonics. The reduction in energy ranges from 9.57 dB for the 3rd harmonic to 14.15 dB for the 9th harmonic.

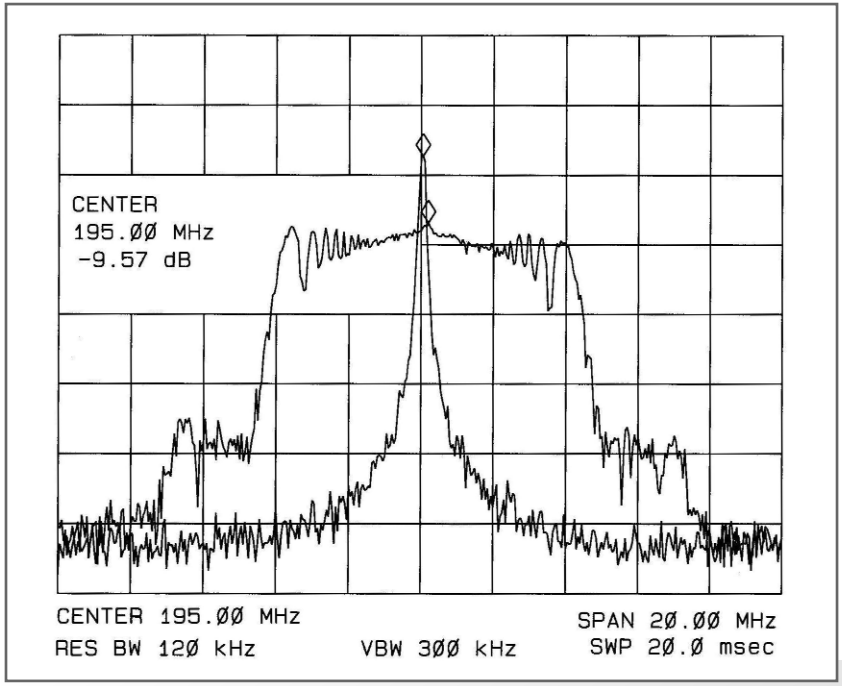


Figure 9.12. 65 MHz 3rd Harmonic

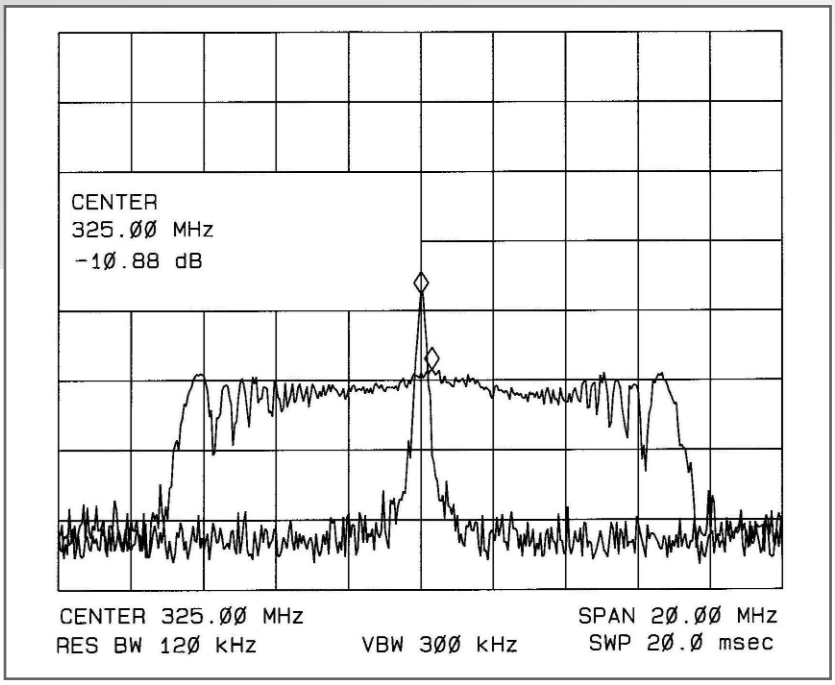


Figure 9.13. 65 MHz 5th Harmonic

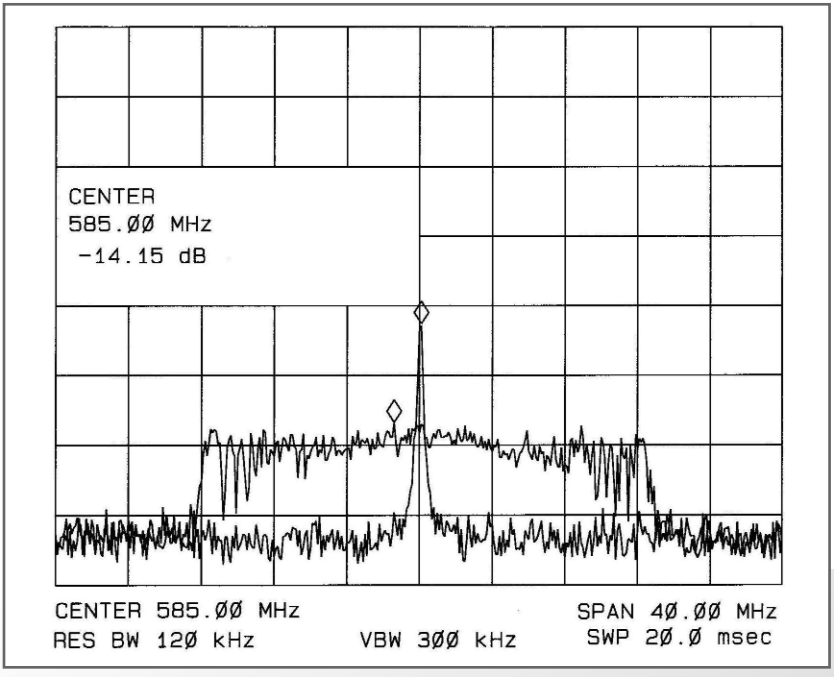


Figure 9.14 65 MHz 9th Harmonic

Each of these scans are taken from a very small four layer board with no other signal or circuit nodes to influence the results of these measurements. Actual results will vary based on the specifics of the design. However, these are representative of the type of data one should expect.

Bandwidth Requirements

As explained earlier, bandwidth is the amount of frequency modulation or spread applied to a particular clock. In terms of bandwidth percent spread, a typical application will use anywhere from one to four percent. A higher reduction in dB will result with a wider spread. However, at lower reference frequencies, a greater spread percentage is required to attain the same amount of reduction of a higher frequency reference. For example, compare a 10 MHz clock with a 28 MHz clock each with a 2% spread. The 28 MHz clock will be modulated over a 560 kHz range while the 10 MHz covers only a 200 kHz range. To see how this affects the amount of reduction, Figures 9.15 to 9.18 show the energy levels for each.

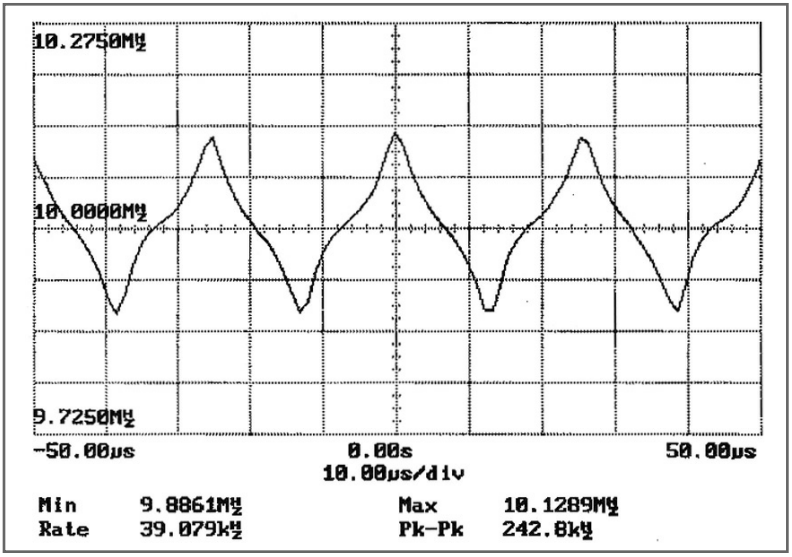


Figure 9.15. 10 MHz SSCG Profile.

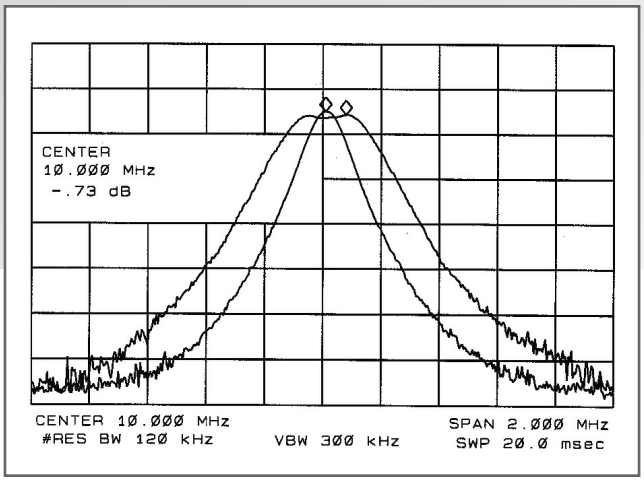


Figure 9.16. 10 MHz, Small dB Reduction.

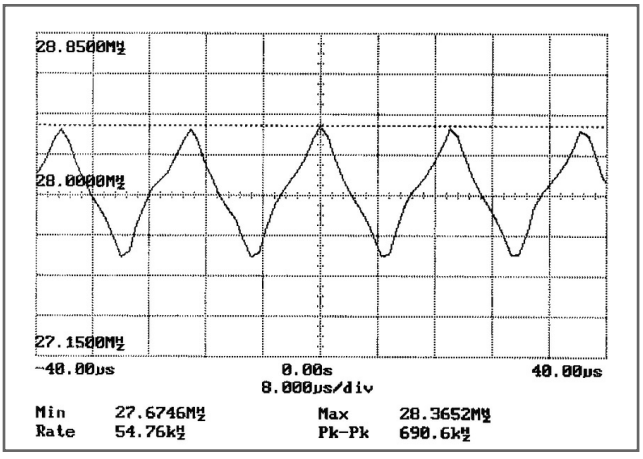


Figure 9.17. 28 MHz SSCG Profile

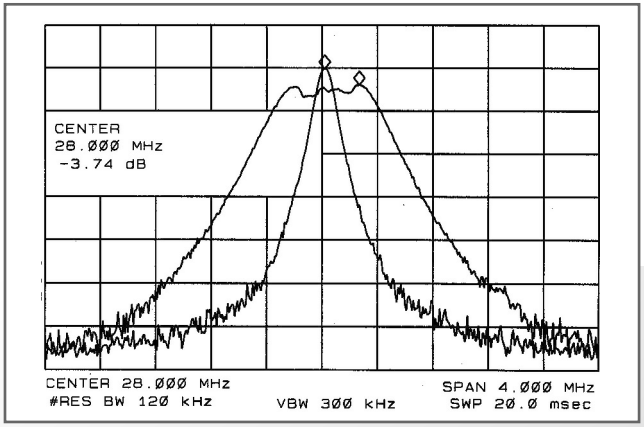


Figure 9.18 28 MHz Larger dB Reduction.

Figure 9.15 shows the modulation profile for the 10 MHz clock. From this view, the bandwidth of the spread can be verified to be 242.8 kHz. By dividing by the reference frequency, the bandwidth percent is 2.42%. Figure 9.16 shows the energy from the 10 MHz clock without Spread Spectrum and with Spread Spectrum active. Notice that there is only a small reduction in the energy peak.

Figure 9.17 show the modulation profile for the 28 MHz clock. Again, from this view, the bandwidth of the spread can be verified to be 690.6 kHz and produces a percentage of 2.46%. Figure 9.18 shows the energy from the 28 MHz clock without Spread Spectrum and with Spread Spectrum active. This shows a larger reduction in energy.

Both references had Spread Spectrum applied using identical profiles. The percent of spread was also the same: 2.42 % versus 2.46%. However, there was only a 0.73 dB reduction EMI on the 10 MHz while the 28 MHz realized a 3.74 dB reduction. The slower frequency needs to have a larger spread percentage to yield the same results.

Down Spread and Center Spread

Certain applications are frequency sensitive and are already operating at the maximum rate they can tolerate. A Spread Spectrum clock with a center-spread modulation will exceed the limits of this type of design. For this reason, Spread Spectrum also performs down spread. Down spread limits the modulated frequency to the reference clock. For example, a 40 MHz reference clock with an applied 2% down-spread Spread Spectrum signal will range from 39.2 MHz to 40 MHz.

One disadvantage of down-spread clocks is the effective center frequency of the clock is reduced by one-half of the bandwidth. With a Spread Spectrum clock providing a clean and symmetrical down-spread profile, the effective center frequency is calculated as:

$$F_{EFFCENT} = F_{MAX} - {}^{1}/_{2} \ (F_{MAX}-F_{MIN})$$

Applying this to the 40 MHz example above, the effective center frequency yields:

$$F_{EFFCENT} = 40 - {}^{1}/_{2} \ (40 - 39.2) = 39.6 \text{ MHz}$$

$$F_{MIN} = 39.20 \text{ MHz}$$

A center-spread Spread Spectrum clock is one that is modulating the frequency of the output clock symmetrically about the reference frequency. This means that for a clean modulation profile, the output frequency will increase and decrease the same amount above and below the reference frequency. The effective center frequency (FEFFCENT) of a center-spread Spread Spectrum clock is the reference frequency itself. In Figure 9.19, the scan shows a center Spread Spectrum clock with a bandwidth of 2.42%. From the data in this scan, we see that the maximum frequency is 10.1289 MHz and the minimum frequency is 9.8861 MHz. Calculating for the effective center frequency yields the expected result.

$$F_{EFFCENT} = 10.1289 \text{ MHz} - {}^{1}/_{2} \ (10.1289 - 9.8861) = 10.0075 \text{ MHz}$$

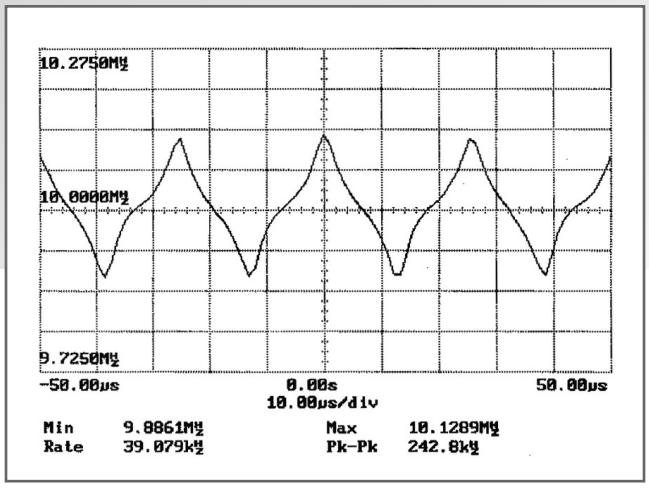


Figure 9.19. 10 MHz SSCG modulation profile.

Modulation Rate

The modulation rate to obtain a low EMI clock profile is very important to the design engineer. This is the rate at which the clock sweeps from the minimum to maximum and back to minimum frequency. One complete cycle of the profile is the modulation rate. Devices that generate Spread Spectrum have different modulation rates and are specified in the datasheet for a particular manufacturers part. This frequency can be as low as 20 kHz and as high as 150 kHz, depending on the design of the particular component.

Spread Spectrum clocks often drive other components that contain a PLL. It is important that the downstream PLL has a loop bandwidth that allows the modulation frequency to pass. If the bandwidth is too low, the effect of Spread Spectrum on EMI will be diminished. Downstream PLLs should be specifically rated to accept Spread Spectrum clock signals. More information on multiple PLLs and loop filters can be found in the Cascading PLL chapter.

Tracking Spread Spectrum

Spread Spectrum can be an effective method for reducing peak EMI and may easily be integrated into a clock design. However, in those designs where a PLL may be "downstream", that is a PLL device that is driven by the Spread Spectrum clock, extra care needs to be taken. Since Spread Spectrum purposely modulates the clock, the downstream PLL must be able to track the frequency change in order to pass the modulated clock. If the PLL's bandwidth is too low, tracking skew will occur. This will have the effect of added jitter into the system. The designer needs to ensure that any downstream PLLs will track the modulated signal. If the downstream PLLs are part of a clock buffer circuit, Spread Aware™ type devices should be used. These devices are specifically designed and tested to operate with Spread Spectrum clock signals.

Conclusion

The generation and distribution of high-speed digital clocks can produce unwanted EMI. When the energy levels exceed allowable thresholds, the offending signals must be addressed. Both mechanical and electrical solutions exist, however a few circuit design changes may be the most cost effective. Clocks that are improperly terminated or have fast transition rates can lead to excessive emissions. To control these emissions, consider these guidelines:

- Properly Terminate all Clock Signals
- Use Slow Rise Time Buffers When Possible
- Consider Filter Capacitors on Selected Signals
- Use Spread Spectrum Clocks for High-Speed

By applying a few simple techniques, the radiated waves can be controlled. Proper termination, capacitor filters, and spread spectrum technology can solve many of these problems.

10

IBIS & SPICE Models

As the frequency of digital clocks increases so does the difficulty in creating robust clock trees having quality signals. In previous chapters, we have discussed the effects of fast edge rates and how to properly terminate traces. However, simulation is necessary to ensure signal integrity. There are two types of models used to simulate the signal characteristics commonly used in today's design environments: IBIS and SPICE. Each has the ability to simulate and analyze a circuit and give feedback on the performance of the design. Many of the available simulation tools include a scope trace viewer that allows the designer to assess the performance of their design. The critical need is to know how a signal looks as its propagates down a trace and encounters impedance changes, terminations, stubs, and other devices that work to change the signal characteristics. Although some companies have dedicated signal integrity departments that specialize in simulation, the design engineer can gain critical information by understanding the effects certain aspects of the circuit have on signal quality. This provides confidence that the circuit will work when the first prototypes arrive. This chapter provides an overview of IBIS and SPICE simulation models and the information contained within.

SPICE Models

SPICE (Simulation Program with Integrated Circuit Emphasis) was created through research at the University of California at Berkeley (UCB) in the early 1970's. It was originally written in FORTRAN and was run on mainframe computers. SPICE 2G6 was the last Fortran version and later replaced with SPICE3 that is a C-language version of 2G6. SPICE3 is still maintained by the UCB as public domain software. Other versions of SPICE have been developed and marketed as proprietary commercial software including PSPICE, HSPICE, and AllSPICE. Most variants of SPICE have maintained similar syntax and algorithms to SPICE3, although they often add some proprietary functionality such as enhanced transistor models. SPICE simulators have been written for most platforms including UNIX workstations and Intel based personal computers.

SPICE is a structural modeling language. It simulates the behavior of devices by running a numerical analysis on a structural model of the device. This structural model is created using basic components such as diodes, transistors, capacitors, resistors, and dependent

current and voltage sources. The components are combined into a circuit by a file called the netlist. Figure 10.2 shows a SPICE netlist for the circuit shown in Figure 10.1. The numbers after the component names in the netlist are arbitrary numbers that correspond to the nodes of the circuit. Many implementations of SPICE contain schematic capture programs and will create these netlist files directly. SPICE uses many physical parameters to accurately model semiconductor devices such as diodes and transistors. These parameters are primarily related to the manufacturing process used to create the devices. Any parameters not specified in a model will be assigned default values.

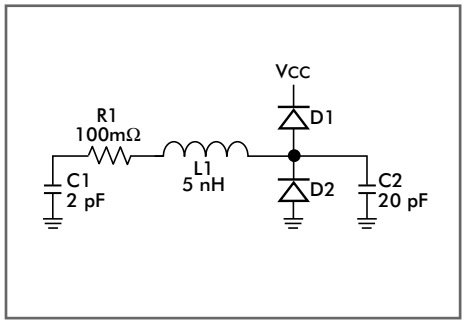


Figure 10.1 CMOS Circuit

```
C1      1       0       2p
R1      1       2       100m
L1      2       3       5n
VCC     4       3       DC 3.3
D1      3       4       DCLAMP
D2      0       3       DCLAMP
C2      3       0       20p
.MODEL DCLAMP D (IS=10N N=1 BV=1200 IBV=10E-3
```

Figure 10.2 SPICE Netlist

When a SPICE simulation is run, the circuit is “solved” using current and voltage equations. Many aspects of the circuit can be examined such as transient analysis (behavior over time) and frequency sweep (behavior with respect to a sweeping input frequency). More complicated analysis can also be run such as a Monte Carlo analysis in which all components are varied per their individual tolerances. A Set of curves is then plotted to show the behavior of the circuit as a function of the statistics of tolerance combinations. From this, you can see how the waveform might vary from device to device over a large number of test samples.

Models for devices such as integrated circuits can be put into a construction known as a sub-circuit that can then be distributed as a library file and called from SPICE to be included in a simulation. These sub-circuits comprise a netlist of standard components connected

and specified in such a way as to describe the whole device. A single sub-circuit for a device such as an operational amplifier can contain thousands of components in its netlist. Fortunately, the sub-circuit can be included as one component in your netlist.

XSPICE is an extension to Berkeley SPICE that was developed by the Georgia Tech Research Institute in 1992. XSPICE allows code-level modeling using the C-language to add new models directly to the SPICE core. This modeling capability expands the capability of SPICE simulators to include 12-state digital modeling, system level modeling, and the ability to include truly arbitrary behavior within models.

Signal integrity simulations can be performed with SPICE by modeling the buffers and by using transient analysis. The transmission line model is used to model board traces, cables, and other interconnection parts. SPICE includes both a lossy and lossless transmission line model. SPICE also includes coupled transmission line models for crosstalk simulation.

IBIS Models

Although SPICE is a comprehensive modeling and simulation environment, there are a few problems from a designer's perspective. What may be the biggest drawback is the reluctance of integrated circuit manufacturers to release SPICE models for their devices. Due to its structural nature, a SPICE model must reveal many details of a device that manufacturers consider proprietary. The lack of readily available models often makes it difficult to use SPICE to validate circuit designs.

IBIS, which stands for the Input/Output Buffer Information Specification, was created in 1993 by an industry-wide group of simulation experts to provide a standard method to exchange electrical component modeling data between integrated circuit manufacturers, simulation software vendors, and design engineers. This standard was developed and is maintained by regular "IBIS Open Forum" meetings that are held by the group. The latest approved version of the standard as of this writing is version 3.2 and is an official ANSI/EIA-656 and IEC 62014-1 standard. The official web site of the group is located at **http://www.eigroup.org/IBIS/**.

In contrast to the structural models of SPICE, an IBIS model is a behavioral model. It is not defined by the structure of the component to be modeled but rather by the way it behaves. IBIS models contain data based on how the component acts or reacts within a circuit as it is tested or operated. This data takes the form of current-voltage (I-V) tables, edge waveforms, lumped capacitance, inductance, and resistances. This data can come from actual measurements or full circuit simulation. Since the model is based on how the device behaves and not mathematical formulas, non-linear effects can also be included in the model. Although IBIS includes many analog characteristics, it is really a digital simulation tool. The stimulation performed by an IBIS driver is a logic transition with a rising edges, falling edges, or a series involving both edges.

The IBIS specification actually takes the form of a documented model file for a sample component. A model file is an ASCII text file that contains the models for all the drivers and

receivers for a single device or family of devices. Figure 10.3 shows the general input structure for an IBIS model. This includes the lumped package parasitics (capacitance, resistance, and inductance) and the ESD clamping structures and the capacitance of the input stage. The parasitics are specified as values while the ESD clamps are specified as I-V tables. Figure 10.4 shows the basic IBIS output structure and it contains the same basic structure as the input model with the addition of the behavior of the driver itself. These are specified as I-V tables and as voltage-time tables for rise-time /fall-time information.

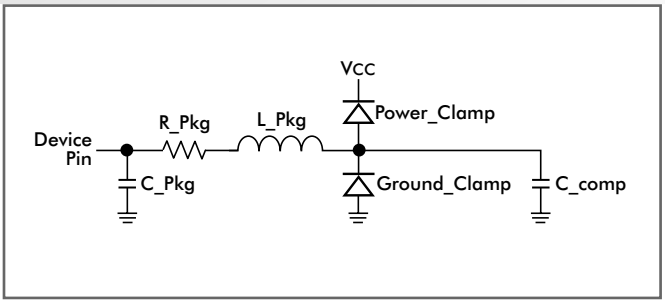


Figure 10.3 IBIS Input Structure

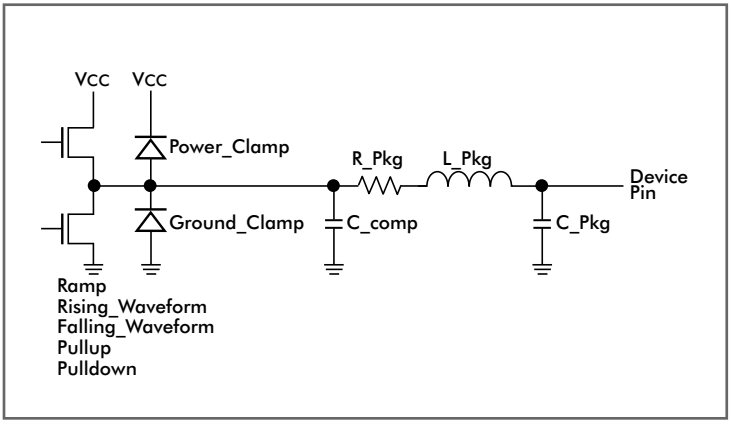


Figure 10.4 IBIS Output Structure

There are three sets of possible values within an IBIS model. The values have different meanings depending on the context. In general, the “min.” values describe slow, weak behavior while the “max.” values describe fast, strong behavior. The “typ.” values refer to typical operating conditions. This behavior is determined by temperature and voltage extremes as well as manufacturing process tolerances. The conditions that cause a behavior may vary for different device families. For instance, a CMOS device will exhibit fast, strong behavior at low temperatures, while a bipolar device will exhibit this behavior at high temperatures. These conditions are also given in the model file.

Version 1.0 of the IBIS specification defines a baseline model to which all IBIS generated models must be backward compatible. Version 1.1 is a refined revision and is generally accepted as the first major release. Version 1.1 defines the input and output structures given in Figures 10.3 and 10.4. Parameters GND_Clamp, Power_Clamp and Pullup/ Pulldown information are provided in tables. Figure 10.5 shows the first few lines of the Pulldown table for an IBIS model.

Pulldown Voltage	I (typ.)	I (min.)	I (max.)
–3.30	–2.09A	–2.12A	–2.06A
–3.10	–1.93A	–1.96A	–1.90A
–2.90	–1.76A	–1.79A	–1.73A
–2.70	–1.59A	–1.63A	–1.56A
–2.50	–1.43A	–1.46A	–1.40A
–2.30	–1.26A	–1.30A	–1.23A
–2.10	–1.10A	–1.13A	–1.07A
–1.90	–0.93A	–0.97A	–0.90A

Figure 10.5 IBIS Pulldown Table

This is an I-V table that shows the current characteristics of the output stage of the device when the Pulldown is active and a specific voltage is applied to the output. The current in an I-V table is positive when the direction of the current is into the component. For the Pulldown and GND_Clamp, the table voltage is the actual voltage. Therefore, the first entry in table 10.5 signifies a 2.09A current flow out of the device when a voltage of –3.3V is externally applied to the output pin. For the Pullup and Power_Clamp values, the table voltage is V_{CC} relative meaning that Voutput = $V_{CC} – V_{table}$. For instance, an entry of -3.3V in a Pulldown table on a device with a V_{CC} of 3.3V indicates an applied voltage of 6.6V on the output pin. All devices must be characterized for an input voltage ranging from $–V_{CC}$ to twice V_{CC} although some table types are not required to span that entire range. This range is ideal for determining the effects of transient overshoots and undershoots.

Another important feature included in the first IBIS release is the Ramp specification. Ramp values show the rise and fall times for an output buffer. An example is shown in Figure 10.6.

Ramp Variable	Typ.	Min.	Max.
dV/dt__r	1.75/1.45n	1.56/1.38n	1.95/1.49n
dV/dt__r	1.78/1.53n	1.59/1.44n	1.98/1.59n
R__load=50.00			

Figure 10.6 IBIS Ramp Values

In the Ramp table, the dV/dt_r is the rising edge and the dV/dt_f is the falling edge of the signal. The number above the fraction is the voltage delta from 20% to 80% of the total voltage swing. The lower number is the time this transition takes. As mentioned earlier, the

min is the slow, weak behavior while the max is the fast, strong behavior. The resistive load for which this behavior is observed is also given in this section as the R_load parameter. The Ramp values include the intrinsic component capacitance but do not take into account the package parasitics.

IBIS Version 2.1 Update

Version 2.1 was the second major release and expanded the CMOS/TTL centric v1.1 to include other types if I/O, such as ECL, PECL, and other differential signals. There were many other extensions introduced in 2.1, but one of the most important was the Rising_Waveform and Falling_Waveform specification. The Ramp values specify the rise and fall times for a buffer and implies a linear transition. The Rising_Waveform and Falling_Waveform specifications describe the signal transitions using a voltage-time (V-T) table allowing an accurate representation of outputs with non-linear transitions such as those that employ transition control and waveform shaping. Figure 10.7 shows the test fixture for measurement of the V-T tables for the rising and falling waveforms. This level of detail allows a much more accurate description of the transitions than the simple fixture in the Ramp specification. Components are allowed, if not encouraged, to have multiple waveform sections with different test fixtures to provide the simulator more information for an accurate simulation.

Version 3.2 contains many enhancements to effectively model advanced buffer behavior such as bus hold and multi-stage buffers. A transit time specification was also added to model stored charge in the clamp diodes. A very important addition to version 3.2 is electronic board descriptions and package models. Both of these add controlled impedance trace descriptors to allow the modeling of traces within a package or on a board.

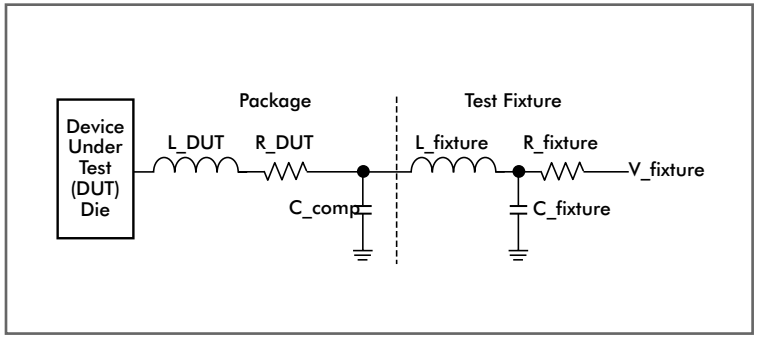


Figure 10.7 Rising_Waveform and Falling_Waveform Test Fixture

Electronic board descriptions (EBDs) are most commonly used for single in-line memory modules (SIMMs) & dual in-line memory modules (DIMMs) and are a description of board level interconnect. These descriptions are intended to reside in a separate .EBD file. Figure 10.8 shows the EBD path description that describes a board that is pictured in Figure 10.9. Several keywords are used to describe the circuit including the Fork construct

to define branching of the traces, L for inductance, C for capacitance, and R for the trace impedance. The EBD can also define discrete and external components such as the 2 nH inductor shown in the example. A discrete R, L, or C component is specified by simply setting the length property to zero.

```
(Path Description  PassThru1
Pin B5
Len=0  L=2.0n/
Len=1.0  L=6.0n  C=2.0p/
Fork
Len=0.8L=1.0n C=2.0p/
Node µ23.16
Endfork
Len=2.1L=6.0n C=2.0p/
Pin A5
```

Figure 10.8 EBD Path Description

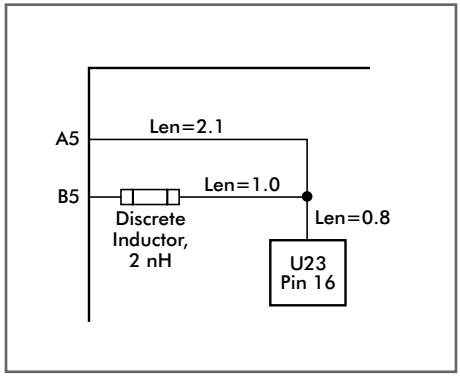


Figure 10.9 EBD Path Description Layout

The length and the L, R, and C values of the path descriptions are given in terms of unit length. The mathematics work out so that any unit of length can be used because the units fall out of the equations for both characteristic impedance Z_0 ($Z_0=\sqrt{L/C}$) and propagation delay T_D ($T_D=\sqrt{LC}\cdot Length$). The only firm requirement for unit length is that it is consistent with all the parameters for a given trace.

Package models allow more elaborate descriptions of a component package. Like EBD files, they include path descriptions to model traces, bond wires, and other methods of interconnection. More complicated behavior is also modeled such as pin-to-pin coupling. A package model can exist separately in a PKG file or it can reside within an IBIS file between the [Define Package Model] and [End Package Model] keywords.

IBIS Simulation

IBIS simulation can show much about how a signal will behave as it propagates through a circuit. IBIS simulation can be performed both pre-layout and post-layout design depending on the phase of your design cycle. Pre-layout simulation tools involve simulating a single net with varying configurations and allows for immediate feedback on the effects of those configurations. This "what-if" analysis can be done with factors such as trace length, trace impedance, placement of buffers, strength of buffers, and different terminations. Figure 10.10 shows part of the screen for Innoveda's Hyperlynx LineSim pre-layout simulation tool. With this tool, you can graphically build a net using building blocks of drivers, receivers, transmission lines, and terminations. "Scope Probes" can be placed on (shown by the arrows next to the buffers) to analyze the voltage behavior exhibited by a pulsing driver.

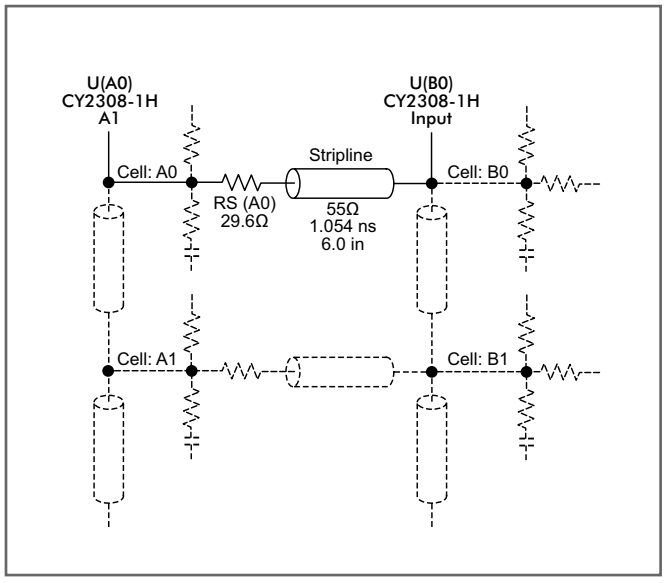


Figure 10.10 Hyperlynx LineSim Entry

Post-layout simulations use a PCB layout as the information for line characteristics instead of an entry tool. While you can usually simulate individual nets as in the pre-layout simulation, the real power in the post-layout simulation comes from the ability to simulate the entire board. You can simulate a whole board and generate a listing of problems with overshoot, settling time, logic thresholds, or skew. Constraints can be defined, and any net violating the constraints can be flagged. Traces in close proximity to one another can be analyzed for crosstalk coupling effects between each other. Some simulators can even estimate electromagnetic interference (EMI) from the geometry of a trace and the signals on that trace.

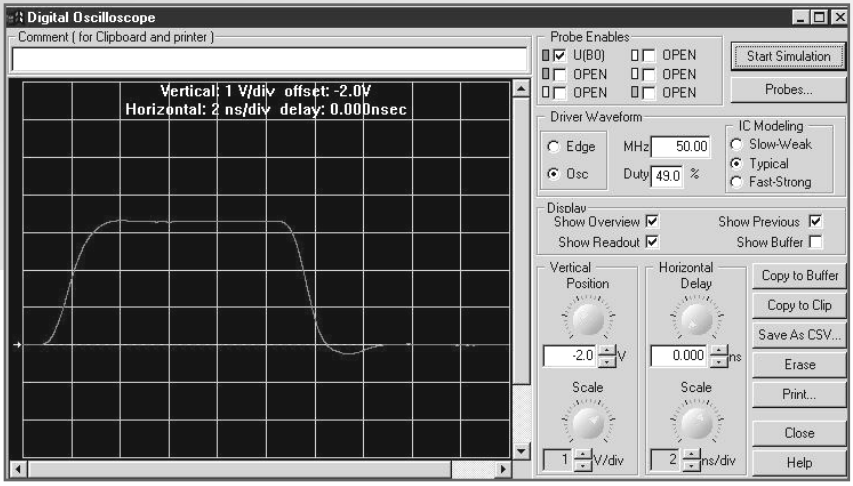


Figure 10.11: Hyperlynx LineSim Scope Trace

IBIS Tips and Tricks

You will most likely encounter errors with IBIS models at least some point when using IBIS simulators. If this happens, and the error messages from the simulator seem cryptic and nondescript, you may want to try running the model through the IBIS "golden parser". The IBIS open forum manages this program for the purpose of validating IBIS models. The program accepts a file, parses through the model, and checks for any errors, inconsistencies, or discrepancies with the IBIS specification. The IBIS Open Forum maintains the source code for the parser and executables for various operating systems are available and can be found on the IBIS Open Forum web site.

There is another important resource when working with IBIS models and its specification. The IBIS open forum maintains an email reflector among its members specifically to address questions and issues from end users. One or more of the open forum experts usually answers questions quickly.

As mentioned earlier, IBIS simulation tools often extract the output impedance of the clock driver from the IBIS model. This value is needed to calculate the resistor value for series termination (See Chapter 7 on Clock Termination). However, if you don't have a simulator and if the output impedance value isn't listed on the component datasheet, it can be derived by hand from the IBIS model.

There are several methods for extracting the output impedance. Because the value isn't the same throughout the output voltage swing, the impedance is an approximation. Here are two of the most common ways to obtain the value. First, open the IBIS model file and locate the I-V tables for the output driver. The top of the file will list which model name represents each pin of the component. In the Pullup IV table, the current (I) is listed for a voltage (V) level in the rising edge of the clock driver.

The first method is a rough extraction, but relatively simple. Locate the voltage value for $V_{CC}/2$, which is a typical threshold value for an input device, and its corresponding current.

The table lists typical, minimum and maximum values. Calculate the output impedance by dividing the voltage by the current. For this number to be valid the driver must still be in its active region and the current value hasn't reached saturation. If the current "flattens out" then the impedance calculation will not show the correct value.

A better method is to use two I-V points from the table and use the delta V and delta I values. From the Pullup table, locate a voltage (V) near VCC/2 but where the current (I) is still in its active region. Locate a second lower voltage and its corresponding current (again, still in the active region). The output impedance is equal to the difference in the two voltages divided by the difference in their corresponding currents.

For example, Figure 10.12 shows a portion of a Pullup I-V table for a clock buffer. Since the driver saturates at 1.3 volts for the typical case, use the value of 1.2 volts and its corresponding 95.71 mA current. A second point is arbitrarily chosen at 0.1 volts (but before the point at which the current starts to drop rapidly) and its current of 10.00 mA. The impedance is calculated with the following equation:

Output Impedance = (V2 - V1)/(I2 - I1)

Output Impedance = (1.2 - 0.1)/(0.09571 - 0.01000)

Output Impedance = 12 ohms

For this device, the estimated output impedance is 12 ohms. Notice that the output impedance will vary depending on the points chosen. However, this will provide an initial value from which to calculate a series termination resistor.

[Pullup] voltage	I(typ.)	I(min.)	I(max.)
0.10	–10.00 mA	–6.93 mA	–13.38 mA
0.20	–19.66 mA	–13.59 mA	–26.37 mA
0.30	–28.97 mA	–19.97 mA	–38.96 mA
0.40	–37.93 mA	–26.06 mA	–51.14 mA
0.50	–46.53 mA	–31.85 mA	–62.90 mA
0.60	–54.75 mA	–37.34 mA	–74.24 mA
0.70	–62.59 mA	–42.52 mA	–85.15 mA
0.80	–70.04 mA	–47.37 mA	–95.61 mA
0.90	–77.09 mA	–51.89 mA	–0.11A
1.00	–83.73 mA	–56.06 mA	–0.12A
1.10	–89.94 mA	–59.88 mA	–0.12A
1.20	–95.71 mA	–63.33 mA	–0.13A
1.30	–0.10A	–66.41 mA	–0.14A
1.40	–0.11A	–69.10 mA	–0.15A
1.50	–0.11A	–71.39 mA	–0.16A
1.60	–0.11A	–73.28 mA	–0.16A
1.70	–0.12A	–74.74 mA	–0.17A
1.80	–0.12A	–75.81 mA	–0.17A
1.90	–0.12A	–76.74 mA	–0.18A

Figure 10.12 Clock Driver I-V Table

IBIS Vs. SPICE Simulation

The debate on whether IBIS or SPICE simulation is better has been a fierce one and may even be a wasted argument. Most designers who simulate circuits will tell you that they use both methods. Each method has its own strengths and weaknesses.

Since SPICE is a numerical structural simulation, a SPICE model can be made to be more accurate than an IBIS model. Although extensions are being added to IBIS to allow a greater range of expression, it still does not have the flexibility that SPICE has to add components to model the simulation closer to reality. IBIS is limited to the constructions of the IBIS specification. Also, not all simulators implement all the extensions and therefore some simulators can produce different results. Another clear advantage of SPICE is the greater number of simulation tools. This is because SPICE has been available longer than IBIS and it is more widely known by design engineers. Most engineering curricula have been using SPICE in circuits courses for many years. Of course, don't forget that IBIS was created to model digital buffers only. SPICE is the only choice when working with analog circuits involving functions such as amplification. However, many companies require Non-Disclosure agreements before the SPICE models can be released as they detail the inner working of the device.

The main advantage of IBIS is its simplicity and convenience. In general, an IBIS model is ready to use in any simulator with no modifications. Because an IBIS simulator is often made for that purpose only, the learning curve typically associated with the process and settings of SPICE simulators is greatly diminished. One complicated issue with SPICE is convergence problems with certain combinations of models. This has to do with the underlying mathematics of the simulation and is often tricky to overcome. This problem does not exist with IBIS models.

Another advantage of IBIS is in the simulation time of large designs. Running a worst-case overshoot analysis on an entire complex board can be performed relatively quickly with IBIS models. Some IBIS tools allow you to look at multiple boards that are connected together. Generally, a complete board simulation would take much longer with SPICE. SPICE has to convert every component in a model to a mathematical relationship and then run iterative calculations to determine a solution. In a IBIS behavioral model, no such iterations are necessary.

There are also ways to get some of the advantages from each simulation method. There are converters available that convert IBIS models to SPICE models and vice versa. These converters aren't always easy to use but they can be handy as a starting point when you must use SPICE to customize a simulation and only have IBIS models for a device. There are also some SPICE simulators that will read in IBIS models and convert them behind the scenes to integrate with the SPICE simulation. For example, the popular Pspice program from Cadence has a built-in IBIS translator for V1.1 models.

Conclusion

Most companies that offer timing technology products also have IBIS models available for their devices. IBIS simulation is easy to perform and can provide much insight into the operation of your design. This simulation is very powerful and allows for the many different types of board stack-ups, components, connectors, and discretes. SPICE modeling also provides an excellent environment and includes functional as well as signal simulation. However, this simulation can be more complex and the models for each component may be difficult to obtain. Using either tool, you can also quickly gain insight into signal integrity problems that arise both in existing circuits and in current designs.

11

Probing High-Speed Clocks

To this point, we have discussed proper clock design techniques and have showed some simulations of the signals. But when actual hardware is analyzed, we need to understand exactly what limitations exist with the chosen test equipment. In many situations, engineers will probe high-speed clocks and the waveform will appear very different than expected. It is often thought that the design is not working properly but in many cases, the engineer exceeded the bandwidth of the equipment.

In the past, oscilloscopes were primarily analog in design with a couple of input channels. They also used one inch round binding posts as the input connection and twisted pair wires were needed to prevent outside interference. The bandwidths of these scopes were also quite limited. Oscilloscopes have progressed dramatically over the years with an abundant of signal capture features and storage capabilities added. They have also increased their bandwidths to meet the needs of probing high-speed digital clocks. In this chapter, we will examine the techniques needed to proper probe a high-speed clock. We will also answer the fundamental question: “Will my 1 GHz Oscilloscope correctly show my 100 MHz clock signal?”

There are two areas of importance when probing high-speed clocks:

1. Oscilloscope Bandwidth
2. Probe and Connection

We will examine both one of these and how it affects the resulting waveform display.

Oscilloscope Bandwidth

Of key importance, obviously, is the instrument being used. To properly measure a high-speed clock, the bandwidth of the scope must be wide enough to accept the signal accurately. With the use of digital storage scopes, we must also ensure the sample rate and sample size is equally sufficient.

The scope bandwidth is defined as the frequency above which a sine wave’s amplitude is degraded by 3 dB or more. Typically a scope (the frame or mainframe) has a bandwidth

and each plug-in or channel has it's own bandwidth. This is most important when channels differ or when plug-in technology can alter the capability of the scope.

In a signal such as a clock, there are two events that need to be considered when measuring. There is the continuous, repetitive stream of waves and there is the single-shot pulse due to the rising and falling edges. There are times when each need to be measured and their respective bandwidth requirements can differ. To view repetitive waves, the bandwidth of the scope needs to be at least three times the highest frequency measured. This ensures the complete waveform will be captured without distortion. To view repetitive waves, the bandwidth of the scope needs to be at least three times the highest frequency measured. This ensures the complete waveform will be captured without distortion.

The single-shot pulse bandwidth is probably the most important when considering high-speed signals as this gives the limits for the clock edge rates. We need to ensure the scope will accurately capture the fast rise and fall times of our signal. Design engineers use time to measure rising and falling edges however; oscilloscopes are specified in terms of bandwidth. Therefore, a conversion equation is required to compare the equipment to the signal. As mentioned earlier, the scope bandwidth is measure to the –3 dB point. To convert to an approximate rise time, we use the following equation:

$$t = 0.338/F_{3dB}$$

This allows us to relate the scope's response to the signal that we need to measure. Some oscilloscopes use a slightly different method for measuring bandwidth and are listed as RMS bandwidth. To convert this measurement into time, use this equation as an approximation:

$$t = 0.361/F_{RMS}$$

This suggests that a 1 GHz scope, at best, can accurately view a 340 ps rise time in a clock signal. But, before we can conclusively say this is our scope's limit, we need to analyze a few other factors

Sample Rate

Since many of the modern oscilloscope are digital and samples the incoming signal, we need to consider how fast they can capture the pulse. Sample rate is simply the number of measured points per second within a waveform.

When a digital storage scope samples the incoming waveform, a dot is used to represent its voltage and time. The faster the sample rate, the more dots are used to represent the signal. Interpolation is used to curve fit to the signal by connecting the dots and showing a contiguous waveform on the display. The sample rate, as compared to the bandwidth, should be at least four to one. An 8 or 16 to 1 ratio will provide excellent waveforms and with values greater than 10, minimal interpolation is necessary.

After the signal is sampled, memory is required to store the dots. The amount should be large enough to allow capture of the complete signal at the given sample rate. When storage is limited, the amount of time viewed on the display can be affected.

Probe and Connection

A scope is only one element that affects the quality of the displayed signal. The probes and connecting cables are easy to forget. However, they are as significant as the scope. Further, they offer the most chance to provide error into the test system.

Similar to the scope, the probe also has a specified bandwidth. The signal that we wish to measure must first pass through the probe before it reaches the scope. This means the bandwidth of the probes must be considered in the same way as the scope. We can convert the edge rate response of the probe using the same formulas that we specified earlier for the scope.

To determine if the signal we are to measure can be properly displayed, a simple formula can be used as an approximation.

$$t_{DISPLAY} = \sqrt{(t^2_{SIGNAL} + t^2_{PROBE} + t^2_{SCOPE})}$$

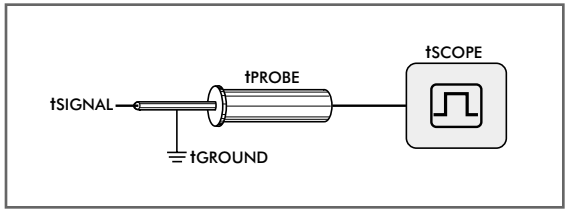


Figure 11.1 Scope Connections

tSIGNAL is the fastest rise (or fall) time of the signal we need to measure. tPROBE is the bandwidth of the probe converted into it's rise time equivalent and tSCOPE is the bandwidth of the scope converted into it's rise time equivalent.

Knowing these relationships, let's do an example. Suppose we have a 100 MHz signal with 1 ns edge rates driven by a clock buffer. We use an oscilloscope with a 400 MHz bandwidth and a probe with a 400 MHz bandwidth. To the casual observer, it appears that the test equipment will display the signal. Or will it?

First, we need to convert the bandwidths into equivalent edge rates.

$$t_{SCOPE} = 0.338/400 = 840\ ps$$

$$t_{PROBE} = 0.338/400 = 840\ ps$$

Now we can calculate what the display will show as the rise time of our signal.

$$t_{DISPLAY} = (1^2 \text{ ns} + 0.84^2 \text{ ns} + 0.84^2 \text{ ns})^{1/2}$$

$$t_{DISPLAY} = 1.5 \text{ ns}$$

The display will show the rise time of the 1 ns signal as 1.5 ns! Notice in these equations, the frequency of the signal does not matter. It is the transition time of the signal that is of importance. Furthermore, even though the scope and the probes individually could handle the signal edge rate, they did not when coupled together. Higher bandwidth scope and probes are needed to properly measure this signal.

Ground Leads

In addition to the selecting the proper bandwidth of the equipment, we also need to be concerned with the probe attachment to the signal. Many probes have a plastic neck with a ground connector at the base. A common method for attaching a ground is to clip a 4-inch wire to the probe. An alligator clip at the other end usually connects to a ground somewhere on the board. Little consideration is given to the effects of this wire.

But, the ground wire plays a key role in accurately displaying the clock signal. When the signal is being measure, current flows into the probe, through the wire, and back into the board ground plane. This can be view as a circuit represented by an R, C and L.

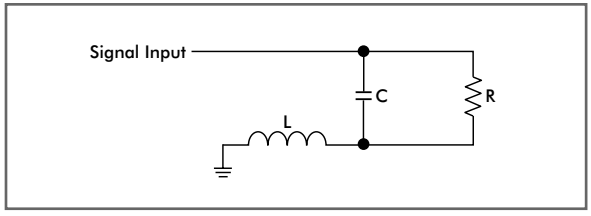


Figure 11.2 Scope Probe Circuit

There are also several types of probes available and include passive and active (FET) voltage varieties. Probes specify their input impedance R and can range from less than 20K to over 10M ohms. Input capacitance C can be as high as 15 pF and down to less than 1pF loading. Oscilloscope manufactures typically list in their documentation the application for which a particular probe was designed. The inductance L is a function of the wire ground loop. By making a few assumptions, we can calculate an LC time constant that will show the limit of the signal response on the probe.

If we assume the ground wire is rectangular in nature, we can use the standard formula for calculating inductance.

$$L \text{ (nH)} = 10.16 * [l * \ln[\frac{2 * w}{d}] + w * \ln[\frac{2 * l}{d}]$$

Where: L is the resulting inductance of the wire (nH)
l is the length of the wire (inches)
w is the width of the wire (inches)
d is the diameter of the wire (inches)
ln is the Natural Log

We can calculate the inductance of the ground loop by applying the physical dimensions of the wire. For example, a wire with a diameter of 0.02 inches, a ground wire with the length of 3 inches, and a ground connection on the board about 0.5 inch (width) from the signal, yields approximately 150 nH. Figure 11.3 shows a wire rectangle and its dimensions.

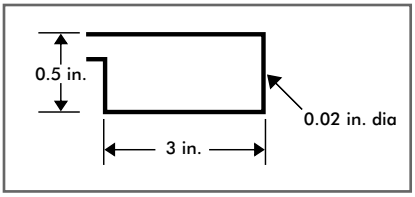


Figure 11.3 Wire Loop Inductance

By assuming the ground on the probe to be similar to the rectangular wire, the same equation can be used to estimate the inductance of the ground strap.

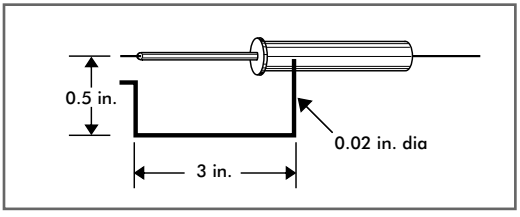


Figure 11.4 Probe Wire Loop Inductance

By using this value of L and the C specified for the probe, we can estimate the rise time limit of the ground wire on our signal. If we choose a typical 5 pF probe, we can estimate the rise time with the following equation:

$$t_{GROUND} = 3.4 * (L * C)^{1/2}$$

$$t_{GROUND} = 3.4 * (150 * 0.005)^{1/2} = 2.9\ ns$$

This yields a value for t_{GROUND} to be 2.9 ns! This is an incredibly large impact on our ability to measure the signal. We now know that our original $t_{DISPLAY}$ equation needs to include the ground loop parameter.

$$t_{DISPLAY} = \sqrt{(t^2_{SIGNAL} + t^2_{PROBE} + t^2_{SCOPE} + t^2_{GROUND})}$$

If we apply the t_{GROUND} parameter to the example we used earlier, the error on the display of the scope is even larger. With the 100 MHz clock with the 1 ns rise time in the previous example and using the scope and probe just described, we can estimate what the display will show.

$$t_{DISPLAY} = (1^2 \text{ ns} + 0.84^2 \text{ ns} + 0.84^2 \text{ ns} + 2.9^2 \text{ ns})^{1/2}$$

$$t_{DISPLAY} = 3.3 \text{ ns}$$

The 3.3 ns rise time on the scope is very different that the 1 ns clock on the board.

When probing high-speed clocks, a key goal is to minimize the t_{GROUND} component of this equation. By doing so, we maximize the use of the bandwidth of the scope and probe. As seen in the previous equations, the two attributes that cause the delay are the capacitance of the probe and the inductance of the ground wire. Since C is a fixed value associated with the probe, select a probe with the smallest input capacitance as possible. Inductance is a function of the geometric size of the ground wire.

As seen in the example, a long wire is insufficient to provide the necessary grounding for high-speed clocks. Changing the shape of the ground strap minimizes the inductance. By selecting the shortest and widest wire for the ground, inductance is minimized. Many probes today have a small spring wire that is short, but very thin. This method generally works, however it is not optimal. For very high-speed clocks, use a short, flat blade for the ground connection. The ground connection needs to be close to the signal being probed. There are test connectors available to which signals can be wired and have collars connected to ground. This is an ideal way to attach the probe to the signal as the probe tip slides into the collar. Care should be taken that the connector doesn't add any stubs to the trace.

Conclusion

Having the proper test equipment is essential when probing high-speed clocks. Engineers often take for granted the waveforms on an oscilloscope as correct, however they can be easily be misled. To view an accurate picture of the signal, the proper bandwidth of the scope and probe must be used and equally as important, a low inductance ground connection. By following a few simple rules, the scope can display the real clock signal. Now you know the answer to the question "Will my 1 GHz Oscilloscope correctly show my 100 MHz clock signal?".

12

Clock Generators

When people speak of clock generators, they often are referring to a piece of test equipment that will allow the generation of a particular frequency and waveform to stimulate a device under test. This general idea can be taken a step further and used as a clock source embedded in a design. Many designers are familiar with oscillators that provide a fixed clock, however, a variety of frequencies can be produced using a clock generator. This chapter will discuss this function and how it applies to an integrated circuit (IC) that can perform that same function on a printed circuit board (PCB) within your system.

Before discussing the aspects of clock generators, Resonators and Crystals should be examined, as they are the basic building blocks. Typically when someone talks about a resonator, they are referring to a ceramic resonator. When someone discusses a crystal, they are talking about a crystal resonator. Both of these devices provide similar functions but with different base substances and differing levels of accuracy.

Resonators & Crystals

Ceramic resonators (“Resonators”) are piezoelectric ceramic devices that are designed to oscillate at a particular frequency. The material from which they are made has the resonant property induced during the manufacturing process. Since this is under the control of manufacturing tolerances, and the quality factor of this material is far less than quartz, resonators cannot match the frequency stability of a crystal resonator (Crystal). This quality factor, also referred to as "Q" is the ratio of energy stored to energy dissipated, which is also defined as the ratio of reactance to series resistance at the resonant frequency. Resonators are typically used in cost sensitive, lower performance applications. They tend to cost half that of a Crystal and available in a smaller physical size. The drawbacks to Resonators are that they lack the frequency and temperature stability of their crystal counterparts.

Crystals are similar in nature to a Resonators but use a piece of cut quartz for its resonant element. Quartz has a naturally high “Q” value allowing it to maintain high accuracy and

frequency stability over its entire operating range of temperature and voltage. The remainder of the discussion will use the Crystal as the clock source for a clock generator.

When someone refers to a Crystal, they are typically not referring to a piece of quartz alone, but rather one that has been packaged and tuned to resonate at a given frequency. A Crystal, when properly stimulated, will act as a narrow band filter at its given frequency. Typically, a Crystal is connected to a device that has an excitation and detection circuit. The gain amplifier in this circuit presents broadband noise to the Crystal which filters out all but the energy present in the Crystal bandwidth for the given Crystal center frequency. This energy is allowed to grow through amplification and a portion thereof is fed back through the Crystal. A square wave is created from the sinusoid input as it reaches its limits when forced through a high-gain system. There are many such devices that contain this type of excitation circuitry including Digital Signal Processors, microcontrollers, and microprocessors.

Crystals come in two basic types: series and parallel resonant. These two types are physically identical but vary in the manner in which they are tuned to their stated frequency. A parallel resonant Crystal is adjusted to a given frequency with a specified value of load capacitance, whereas a series resonant Crystal requires no load capacitance. The load capacitance is what the Crystal "sees" at its pins to the excitation circuit. If a series resonant Crystal is configured with a load capacitance, it will operate at a frequency higher than its stated value. Likewise, if a parallel resonant Crystal is used without any load capacitance it will operate at a frequency lower than its stated value. Most excitation circuits will have a rated load input capacitance.

The parallel resonant configuration is the most common and is shown in Figure 12.1.

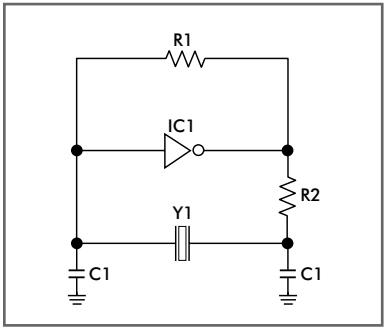


Figure 12.1 Parallel Resonant Crystal Circuit

In order to calculate the load capacitors needed, the following formula can be used:

$$C_L=((C1 * C2)/(C1 + C2)) + C_{STRAY}$$

C_{STRAY} is the stray capacitance in the circuit and is typically between 2 and 5 pF. If using a parallel resonant Crystal with a C_L of 20 pF, C1 and C2 will be approximately 27–33 pF, each depending on the level of stray capacitance. If after measuring the output of the circuit it is found that the frequency is too low, the values of C1 and C2 should be

decreased in order to increase the oscillation frequency. The opposite would be true if the frequency was too high. To make this whole process much easier, many clock generators are optimized for a particular crystal load. Additionally, some clock generators allow the user to change the load capacitance which helps eliminate the variance of the stray capacitance found on the board from the above equation.

Oscillators

Taking the building block approach one step further, the oscillator is the next piece. An oscillator is a device that has the Crystal, excitation, and detection circuitry in one single package. In this case, the output of the oscillator is a square wave that can be used by other devices that require a clock directly. This is typically the easiest clock source to use. The drawback is that a separate oscillator must be purchased for each required frequency because the frequencies are fixed.

Programmable Oscillators

The Programmable Oscillator is a hybrid device that consists of a Crystal and a clock generator in the same package. It performs a function similar to a dedicated oscillator. It offers the advantage of being "configurable", allowing the user to select a given frequency after, rather than before the purchase. The possible drawback is that the phase noise of this device may be higher than the dedicated oscillator, which may be of concern in the design in which it is used. The concept of programmability, which allows a variety of frequencies to be selected will be discussed next.

Clock Generators

A Clock generator, also known as a frequency timing generator (FTG) or frequency synthesizer, is a device that uses a Crystal, an oscillator or a given clock as an input, and can create a wide range of frequencies as an output. As an example: a 16 MHz crystal can be used as an input into the FTG and it in- turn generates an output of 14.31818 MHz. Even though the output frequency seems to have little in common with the input frequency, it can be typically be created within an FTG with no deviation in frequency.

The benefit to an FTG, as discussed above, is allowing the user to configure the part to have numerous output configurations. This can be done just prior to installing it on the board as well as on-board, allowing the utmost in versatility. All of these benefits do however come with some drawbacks. The largest being increased phase noise or jitter. This is mostly an issue in communications standards, such as SONET, FibreChannel and Gigabit Ethernet, whose extremely high speeds dictate an extremely pure clock source. An additional item to note is whether the output must be in phase with the input frequency. Some FTG's offer phase alignment for particular configurations only, while others do not offer this at all.

A Frequency Timing Generator is made up of divider (Q), phase detector (PD), charge pump (CP), voltage controlled oscillator (VCO), a multiplier (P) and a post divider (N), as shown in Figure 12.2. This is essentially a PLL as was discussed in Chapter 2 with the P, Q, and N dividers added.

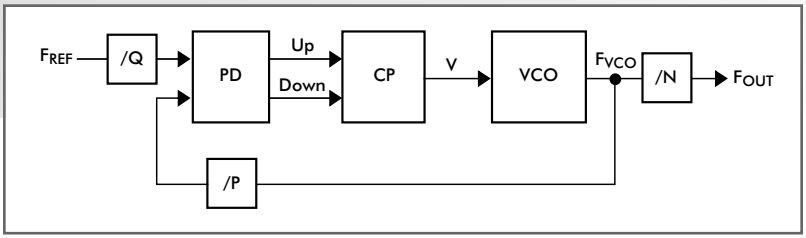


Figure 12.2 Clock Generator Block Diagram

The Q counter takes the reference clock input and divides it prior to sending it into one of the phase detector inputs. The P counter takes the output of the VCO and divides it down before it is given as the feedback (FB) into the second phase detector input (effectively causing multiplication). The Phase Detector determines which input signal, the reference/Q (REF) or the feedback/P (FB), occurs first. It sends an up or down signal to the charge pump to align the two signals. If the feedback is slow, it will send an "up" to the charge pump. The charge pump monitors the Phase Detector output and either increases or decreases its reference voltage that it supplies to the VCO. The VCO looks at the control voltage from the charge pump and creates the corresponding frequency as its output. Increasing or decreasing the control voltage will increase or decrease the VCO frequency. The N counter (post divider) divides the output of the VCO prior to its output of the device. By adjusting the P, Q, and N values, the output is created from the input frequency. Both Q and N are clock dividers (pre and post-divide) while P is the multiplication factor.

In the simplest case, assume that the P, Q and N dividers are all set to one. This device then functions as a zero delay buffer (Figure 12.3). The reference clock drives the REF input of the phase detector. The phase detector follows the rising edge of the REF signal and causes the charge pump to speed up or slow down the VCO such that FB signal at its other input matches every REF rising edge with a corresponding FB rising edge. Since this is actually the VCO output, the VCO runs at the same frequency as the reference input. By matching the FB and output buffer path lengths within the device, the output of the device has zero skew (propagation delay) relative to its input.

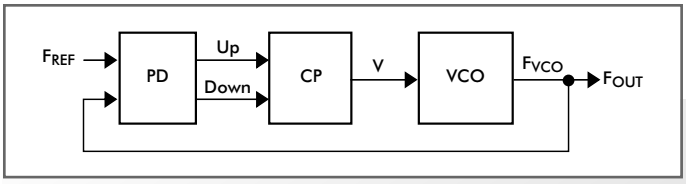


Figure 12.3 Zero Delay Buffer (ZDB) Block Diagram

A more complex example is to generate the 14.31818 MHz output from a 16 MHz input. The values for P, Q and N are not one in this case and need to be calculated to get the desired result while not exceeding the limits of the VCO. The output frequency is a function of the VCO divided by N. The VCO frequency is a function of the input clock and the P and Q ratio. These two relationships can be expressed with the following equations.

$$Output\ frequency = VCO\ frequency/N$$

$$Input\ frequency/Q = VCO\ frequency/P$$

Combining them, the output frequency and ratios can be expressed with the following equations:

$$Output\ frequency = (Input\ Frequency/Q) * (P/N)$$

$$Output\ frequency/Input\ Frequency = P/(N*Q)$$

For the example of a 16 MHz input generating a 14.318182 output, there is the following substitution:

$$14.31818/16.000000 = P/(N * Q)$$

$$0.89488652 = P/(N * Q)$$

By using the values of Q set to 44, P equal to 315 and an N equal to 8, the phase detector has an input frequency of 45.45454 kHz and the VCO has a frequency of 114.545455 MHz. Both of these frequencies are within their operating limits of the particular device chosen for this example. The methods of how to calculate these numbers by hand as well as through software algorithms are discussed next.

Calculating P, Q and Post Dividers

When calculating the values that are needed for the P and Q counters, there are several methods that can be used. The "by hand" approach is to factor the input and output frequencies into a multiple of primes.

$$16 \times 10^6 = 2^{10} * 5^6$$

$$14.31818 \times 10^6 = (2^5 * 3^2 * 5^7 * 7)/11$$

Therefore, 16 MHz must be multiplied by (315/352) to get 14.31818 as an output.

In order to do this within a clock generator, the minimum and maximum requirements of the VCO as well as the phase detector must be maintained. To further restrict the possible values of P, Q and N, the clock generator must have enough bits of resolution in the P, Q and N counters to achieve the divide values needed.

By using a tool developed for this purpose, it is much easier to calculate what the counter values are needed as well as what values are within the bounds of the phase detector and VCO. One such program is called CyClocksRT, which can be found at **http://www.cypress.com** in the Timing Technology section. The user has a simple GUI interface specifying the input frequency, the output frequencies, and some internal dividers. The prior example using this GUI is shown in Figure 12.4.

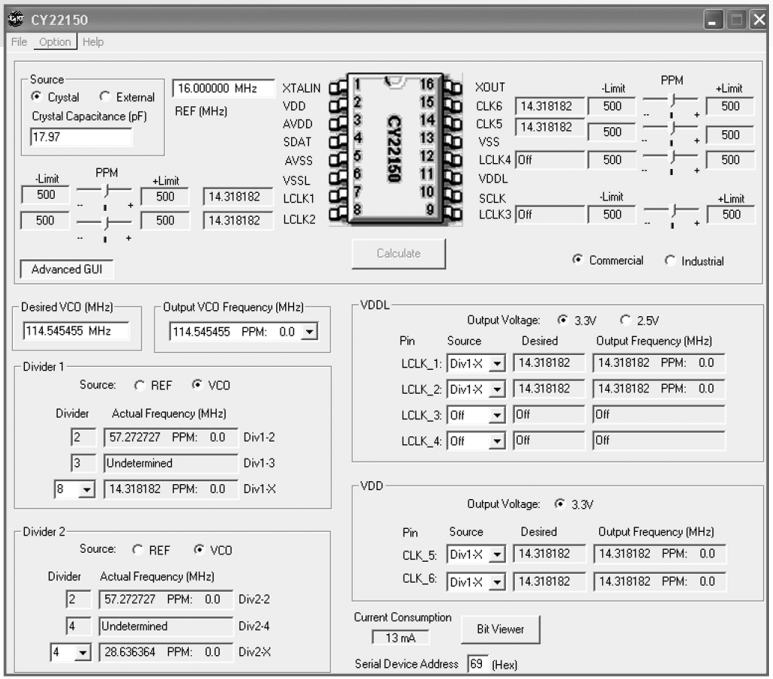


Figure 12.4 CyClocksRT GUI

Parts Per Million (PPM)

When the ratio of P, Q and N do not allow the output to operate at the desired frequency, the deviation is expressed in terms of PPM, or Parts per Million. In the case of a 16 MHz clock with an error of 5 PPM, the actual error would be 5 (parts)* 16 (millions of Hz)= 80 Hz. The 16 MHz frequency with a 5 PPM error rate is therefore 16.000080 MHz.

As seen, a single digit PPM is an extremely small amount of inaccuracy. Many clock inputs can tolerate a range of 50 to 200 PPM and still operate properly. Often, clock generators with high resolution P, Q and N counters, can provide single digit PPM levels with virtually any input/output combination.

The PPM that a clock generator specifies assumes the input has 0 PPM. If the input clock (oscillator, crystal, or clock source) varies in frequency, the output of the clock generator will also vary. For example, a 25 MHz 0 PPM clock driving a clock generator may produce a 48 MHz output with 0 PPM. If the 25 MHz clock has a 25 PPM error rate, the 48 MHz output will also exhibit a 25 PPM deviation.

Fractional N

Some generators incorporate a feature called Fractional N. Fractional N is a means of bouncing between multiple P/Q values in order to achieve a lower PPM or reduce the effective jitter when high P & Q values are used. As an example, choose an output frequency that cannot be exactly met and the closest result is 5 PPM. In order to get to the desired output frequency with 0 PPM, Fractional N can bounce between 2 different P/Q ratios, one at +5 PPM for 66.667% of the time, and the other at -10 PPM for 33.333% of the time. The average, which is filtered by the FTG loop bandwidth to prevent jitter, is the desired frequency at 0 PPM. Using a Fractional N technique requires the FTG to have special circuitry to allow the "jumping" between various P/Q ratios. This is typically done by providing a table of multiple P & Q values and using a counter or state machine to cycle through several of them at a programmable rate. This same technique can be applied to a spread spectrum output.

Spread spectrum is the varying of an output clock within a given band. An example would be to have a 100 MHz clock that varies from 99.5 to 100.5 MHz at a 40 kHz rate. This can be accomplished using the same basic hardware as that used for Fractional N. The benefits and uses of this technique are covered in much greater detail in Chapter 9 of this book.

Part Configuration

Once the P, Q, and N values are selected, a part needs to be programmed. Depending on the technology of the device, this can be accomplished in several different ways.

Mask programmed parts are hard coded at the die level. The appropriate connections are shorted or left open during the mask process of the wafer manufacturing process. This type of configuration may yield a very high number of die per wafer allowing the cost per chip to be very low. The downside of this technology is that once the "mask" is made, the device can only have the given configuration.

EPROM based designs allow for more flexibility in the configuration process. They use an EPROM cell that is left "blank" at the wafer manufacturing process. This allows the device to be programmed with the desired configuration. Although this is a one time programmable (OTP) device, it still offers much greater flexibility than the mask based part.

In order to get around the "program once" limitation, some FTG devices are SRAM based. This allows the devices to be configured on board through a serial programming stream. The benefit is that the device can be configured an unlimited number of times, allowing for maximum flexibility of output frequencies. However, the power-up default is set to an initial configuration based on the original design criteria of the part and is fixed.

The latest technology is a hybrid approach. It uses a FLASH cell to store the configuration data thus allowing the user to select the power-up default. This same part also has SRAM—based registers that allow the user to re-configure the part an unlimited number of times while it is on the board. This hybrid technology offers the most flexibility.

Multiple PLLs

Multiple PLLs in a single package offer many benefits as well as a few detractors. The benefits of multiple PLLs within a part allow the user to generate multiple, seemingly unrelated, frequencies within a single device. The inherent problem with multiple PLLs is that each PLL may interact with the others, allowing additional phase noise (jitter) to be coupled into the VCO and out onto its respective output. This is typically not an issue for pure digital designs where phase noise is of less concern. The areas that are typically affected are items such as A/D devices as well as down-stream PLL based devices.

Conclusion

With the multitude of frequencies required for many synchronous systems, timing generators can offer a flexible solution. They can generate a variety of clocks from a single input source. A large benefit is the ability to change the frequencies as the design progresses without changing the physical part.

References

"High-Speed Digital Design, A Handbook of Black Magic" *By Howard Johnson and Martin Graham;* Prentice Hall.

"Chaotic Dynamics, an Introduction" *by G.L. Baker and J.P. Gollub*; Cambridge University Press, Cambridge

"Introduction to Microwaves" *by Wheeler*; Prentice Hall

"T11.2 / Project 1316-DT", A proposed draft of ASC of NCITS. *Reference work done by and for Secretariat National Committee for Information Technology Standardization (NCITS)*

"Synchronous Optical Network (SONET) Transport Systems: Common Generic Criteria", TR-253-CORE, Issue 1, December 1994; *Bell Communications Research, Inc (Bellcore)*

"Jitter & Timing Analysis", *Len Mooney, LeCroy Corporation Inc.*, Chestnut Ridge, New York 10977-6499

"MOS/LSI Design and Application", Texas Instruments Electronics Series, *by Dr. William Carr, Dr. Jack Mize*, McGraw-Hill Book Company

"Designing with TTL Integrated Circuits", Texas Instruments Electronics Series, *by the IC Applications Staff,* McGraw-Hill Book Company

"IEEE Std 802.3, 2000 Edition—Carrier sense multiple access with collision detection (CSMA/CD) access method and physical layer specifications: Adopted by the ISO/IEC and redesignated as ISO/IEC 8802-3:2000(E)", The Institute of Electrical and Electronics Engineers, Inc. *by its members for the computer society*.

"Transistor Circuit Design", *by Laurence G. Cowles*, Prentice-Hall, Inc.

Spread Aware™ Application Note, *Cypress Semiconductor*, June 29, 1999

"A Semidigital Dual Delay-Locked Loop", IEEE Journal of Solid-State Circuits, Vol. 32, No. 11, November 1997 1683, *Stefanos Sidiropoulos, Student Member, IEEE, and Mark A. Horowitz, Senior Member, IEEE*

Lin L., et al, “A 1.6GHz Differential Low-Noise CMOS Frequency Synthesizer using a Wideband PLL Architecture”, Digest of Technical Papers, International Solid-State Circuit Conference, pp. xxx-xxx, Feb. 2000.

“TP 12.4: A 900-MHz Local Oscillator using a DLL-based Frequency Multiplier Technique for PCS Applications”, George Chien, Paul R. Gray; University of California, Berkeley, CA.

“Printed Circuit Board Design Techniques for EMC Compliance”, By Mark I Montrose, IEEE Press1996.

Glossary

Word	Definition
Anechoic	Without echo
Anechoic chamber	A test room designed to have minimal reflections by using highly absorbent materials. Used as the facility to detect and measure electromagnetic emission
ANSI	American National Standards Institute
Attenuation	Process to reduce the amplitude of a signal.
Bimodal	Having two modes. In statistics refers to having two peaks in a distribution
Blind VIA	A plated through-hole connecting an outer layer to one or more internal layers of a multilayer printed circuit board. It does not extend fully through all of the layers of base material of the board
Buried Microstrip	Type of PCB construction; The trace signal is on an inner layer surrounded by a dielectric with a reference plane below and air above.
Buried VIA	A plated through-hole connecting an inner layer to one or more internal layers of a multilayer printed circuit board.
CISPR	International Special Committee On Radio Interference; EMI standards body for Europe
CMR	Common Mode Rejection
Common Mode	Voltage or current that is common to both sides of a differential pair signal
dB	Decibel; a unit for representing the ratio of the magnitudes of two voltages or currents equal to 20 times the logarithm of the voltage or current ratio
Dielectric	An insulating material that lies between two or more conducting materials

Dielectric Constant	Sometimes called relative permittivity; A constant that is the ratio of a material's permittivity to that of a vacuum
Differential Mode	Voltage or current that is unique to one signal of a differential pair signal
Droop	A momentary drop in supply voltage typically due to switching signals
DSO	Digital Storage Oscilloscope; Test Equipment
DUT	Device Under Test; Refers to any element on which a measurement is taken
Duty Cycle	A ratio of the high time and the low time of a digital signal. Usually expressed in term of a percentage. A 50% duty cycle waveform has one-half the period high and one-half low.
EBD	Electronic Board Description
EIA	Electronic Industries Alliance
EMC	Electromagnetic Compatibility; The ability a device can exist within its environment
EMI	Electromagnetic Interference; The process by which energy is transmitted
ESR	Effective Series Resistance; the total resistance of a capacitor including the DC and AC components
Extrinsic Skew	Skew generated outside a device usually from layout or design parameters
FCC	Federal Communications Commission
FET	Field Effect Transistor
FTG	Frequency Timing Generator; A Device that generates seemingly unrelated output frequencies
Gaussian Distribution	A bell shaped curve representing a statistical distribution
GTL	Gunning Transceiver Logic; A low-level, high-speed interface standard for digital integrated circuits
Histogram	A graph representing the frequency of an occurrence
HSTL	High Speed Transceiver Logic; A JEDEC specification for electrical interface
IBIS	Input/Output Buffer Information Specification; A behavioral model of a device representing it input and output structures.

IEC	National Committees of the International Electro technical Commissions
Impedance	Similar to resistance, but a measurement of the AC opposition to current flow.
Intrinsic Skew	Skew belonging to the device by its very nature
JEDEC	(Joint Electron Device Engineering Council) — the semiconductor standardization body of the EIA
Jitter	The deviation of a clock's output from its ideal position
Jitter Generation	The effect of a PLL added noise to the output clock signal
Jitter Transfer	Amount of jitter passed from the input to the output of a device or PLL
LVCMOS	Low Voltage Complementary Metal Oxide Semiconductor
LVDS	Low Voltage Differential Signaling
LVPECL	Low Voltage Positive Emitter Coupled Logic
LVTTL	Low Voltage Transistor Transistor Logic
Mean	For statistics, it is short for arithmetic mean. Also referred to as average.
Micron	Unit of measure equal to one millionth of a meter (metric)
Microstrip	Type of PCB construction; The trace signal is on an outer layer with a dielectric separating the trace from a reference plane.
Mil	Unit of length equal to 0.001 of an inch
Modulate	To vary the amplitude, frequency, phase, or intensity of a signal with another signal
MOSFET	Metal Oxide Semiconductor Field Effect Transistor
Multimodal	Having more than one mode. In statistics; refers to having more than one peak in a distribution
Normal Distribution	*See Gaussian Distribution*
PCB	Printed Circuit Board.
Peak-to-peak	Includes measurement data for the maximum values. Used in measuring jitter.
Phase Error	Skew between the edges of two clock signals
Phase Noise	Short term frequency fluctuation of a signal

PLL	Phase Locked Loop; a function that locks a reference frequency to the phase of an input.
Polyimide	Thermoplastic used with glass to produce printed circuit laminates
Prepeg	Material used in PCBs consisting of a base material impregnated with a synthetic resin, such as epoxy or Polyimide
PWB	Printed Wiring Board
RF	Radio Frequency; The frequency range, roughly from 10 kHz to 100 GHz, used in communications
RMS	Root Mean Square
Shunt	To divert part or all of a current by connecting a circuit in parallel with another
Sigma	See Standard Deviation
Skew	The variation in the arrival time of two signals specified to occur at the same time
Slew Rate	The rate of change of a signal
SPICE	Simulation Program with Integrated Circuit Emphasis; Detailed simulation model of a circuit or device
Spread Spectrum	A technique that spreads a signal bandwidth over a wide range of frequencies
SST	Spread Spectrum Technology; Also known as Clock Modulation
Standard Deviation	Measurement of the spread of data about the mean value. One Standard Deviation contains 68.26% of the data on one side of the mean.
Stripline	Type of PCB construction; The trace signal is on an inner layer surrounded by a dielectric with a reference plane above and below.
Thevenin	Theorem by M.L. Thevenin, a French Engineer, to rearrange any linear circuit into two networks connected together. Often used to describe a termination technique that is rearranged to provide an equivalent circuit.
TIA	Timing Interval Analyzer; Test Equipment
Tracking	Ability for a PLL to follow the phase variations of an input signal
TTB	Total Timing Budget;
UI	Unit Interval; Equal to one period of a clock waveform.

VCC	Collector supply voltage used in Bipolar technology. Often misused for VDD.
VCCI	Voluntary Control Council for Interference by Information Technology Equipment; EMI standards body for Japan
VCO	Voltage Controlled Oscillator; Frequency output is proportional to the value of a control voltage
VDD	Drain supply voltage; power supply (VD is the drain voltage of a FET)
VIA	A plated through-hole connecting an outer layer to one or more internal layers of a multilayer printed circuit board.
VSS	Source Supply Voltage — Usually 0 volts (ground) in digital circuits (Vs is the source voltage of a FET)
VTT	Termination Voltage
ZDB	Zero Delay Buffer; A PLL based clock buffer that has a nominal zero delay from the input reference clock to the output clock.

Cypress Semiconductor Corp.

3901 North First Street

San Jose, CA 95134

www.cypress.com

Printed in U.S.A.

1-0702TTDMAN