

Mixed-Signal Blockset™

Reference



MATLAB® & SIMULINK®

R2022b



How to Contact MathWorks



Latest news: www.mathworks.com
Sales and services: www.mathworks.com/sales_and_services
User community: www.mathworks.com/matlabcentral
Technical support: www.mathworks.com/support/contact_us



Phone: 508-647-7000



The MathWorks, Inc.
1 Apple Hill Drive
Natick, MA 01760-2098

Mixed-Signal Blockset™ Reference

© COPYRIGHT 2019–2022 by The MathWorks, Inc.

The software described in this document is furnished under a license agreement. The software may be used or copied only under the terms of the license agreement. No part of this manual may be photocopied or reproduced in any form without prior written consent from The MathWorks, Inc.

FEDERAL ACQUISITION: This provision applies to all acquisitions of the Program and Documentation by, for, or through the federal government of the United States. By accepting delivery of the Program or Documentation, the government hereby agrees that this software or documentation qualifies as commercial computer software or commercial computer software documentation as such terms are used or defined in FAR 12.212, DFARS Part 227.72, and DFARS 252.227-7014. Accordingly, the terms and conditions of this Agreement and only those rights specified in this Agreement, shall pertain to and govern the use, modification, reproduction, release, performance, display, and disclosure of the Program and Documentation by the federal government (or other entity acquiring for or through the federal government) and shall supersede any conflicting contractual terms or conditions. If this License fails to meet the government's needs or is inconsistent in any respect with federal procurement law, the government agrees to return the Program and Documentation, unused, to The MathWorks, Inc.

Trademarks

MATLAB and Simulink are registered trademarks of The MathWorks, Inc. See www.mathworks.com/trademarks for a list of additional trademarks. Other product or brand names may be trademarks or registered trademarks of their respective holders.

Patents

MathWorks products are protected by one or more U.S. patents. Please see www.mathworks.com/patents for more information.

Revision History

March 2019	Online only	New for Version 1.0 (Release 2019a)
September 2019	Online only	Revised for Version 1.1 (Release 2019b)
March 2020	Online only	Revised for Version 1.2 (Release 2020a)
September 2020	Online only	Revised for Version 1.3 (Release 2020b)
March 2021	Online only	Revised for Version 2.0 (Release 2021a)
September 2021	Online only	Revised for Version 2.1 (Release 2021b)
March 2022	Online only	Revised for Version 2.2 (Release 2022a)
September 2022	Online only	Revised for Version 2.3 (Release 2022b)

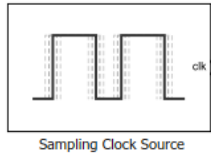
1	Blocks: Data Converters
2	Blocks: PLL
3	Blocks: Utilities
4	Functions: Data Converters
5	Functions: Phase-Locked Loop
6	Functions: Utilities

Blocks: Data Converters

Sampling Clock Source

Generate clock signal with aperture jitter

Library: Mixed-Signal Blockset / ADC / Building Blocks



Description

The Sampling Clock Source block generates either a sine wave or square wave clock with aperture jitter impairments.

Ports

Output

clk — Output clock with aperture jitter

scalar

Output clock signal with aperture jitter, returned as a scalar. The clock is a periodic pulse train that can be a sine wave or a square wave, based on the **Clock type** parameter.

Data Types: double

Parameters

Clock type — Output clock signal shape

Square wave (default) | Sine wave

Shape of the output clock signal.

- Square wave — Returns a square wave of 1 volts amplitude and specified frequency.
- Sine wave — Returns a sine wave of 1 volts amplitude and specified frequency.

Programmatic Use

Block parameter: SignalType

Type: character vector

Values: Square wave | Sine wave

Default: Square wave

Clock frequency (Hz) — Desired frequency of output clock signal

1e6 (default) | scalar

Desired frequency of the output clock signal, specified as a scalar in hertz.

Programmatic Use

Block parameter: Freq

Type: character vector

Values: scalar

Default: 1e6

Min (V) — Minimum clock output voltage

0 (default) | real scalar

Minimum voltage of the output clock signal, specified as a real scalar in volts.

Programmatic Use

Block parameter: Min

Type: character vector

Values: real scalar

Default: 0

Max (V) — Maximum clock output voltage

1 (default) | real scalar

Maximum voltage of the output clock signal, specified as a real scalar in volts.

Programmatic Use

Block parameter: Max

Type: character vector

Values: real scalar

Default: 1

RMS aperture jitter (s) — Standard deviation of clock edge locations

1e-12 (default) | real nonnegative scalar

Standard deviation of clock edge locations, generated by an impaired clock with respect to an ideal clock. **RMS aperture jitter** is specified as a real nonnegative scalar in seconds.

Aperture delay is the delay between sampling clock signal and the actual instant when the sample is taken. Aperture jitter is the sample to sample variation between aperture delay. Aperture jitter results in an error voltage in ADC that is proportional to the magnitude of the jitter and the slew rate of the input signal to ADC.

Programmatic Use

Block parameter: RMSJitt

Type: character vector

Values: nonnegative real scalar

Default: 1e-12

Advanced

Specify custom seed — Specify seed

off (default) | on

Select to specify a custom seed for generating the clock signal. By default, this option is deselected.

Seed — Seed for generating clock signal

0 (default) | nonnegative real scalar

Seed for generating the clock signal, specified as a nonnegative real scalar.

Programmatic Use

Block parameter: Seed

Type: character vector
Values: nonnegative real scalar
Default: 0

Dependencies

To enable this parameter, select **Specify custom seed** in the **Advanced** tab.

Specify custom initial output – Specify initial output

off (default) | on

Select to specify a custom initial output for the sampling clock source. By default, this option is deselected.

Initial output – Initial output of sampling clock

1 (default) | real scalar

The initial output of the sampling clock source, specified as a real scalar.

Programmatic Use

Block parameter: InitialOutput

Type: character vector

Values: real scalar

Default: 0

Dependencies

To enable this parameter, select **Specify custom initial output** in the **Advanced** tab.

Enable increased buffer size – Enable increased buffer size

off (default) | on

Select to enable increased buffer size during simulation. This increases the buffer size of the Variable Pulse Delay block inside the Sampling Clock Source block. By default, this option is deselected.

Buffer size – Number of samples of the input buffering available during simulation

50 (default) | positive integer scalar

Number of samples of the input buffering available during simulation, specified as a positive integer scalar. This sets the buffer size of the Variable Pulse Delay block inside the Sampling Clock Source block.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

To enable this parameter, select **Enable increased buffer size** in the **Advanced** tab.

Programmatic Use

Block parameter: NBuffer

Type: character vector

Values: positive integer scalar

Default: 50

Version History

Introduced in R2019a

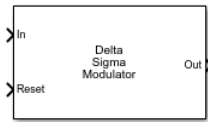
See Also

SAR ADC | Flash ADC

Delta Sigma Modulator

Model a discrete delta sigma modulator based ADC

Library: Mixed-Signal Blockset / ADC / Building Blocks



Description

Use the Delta Sigma Modulator block to model a discrete delta sigma modulator based ADC using a set of different architectures. The block supports these architectures:

- Cascade of feedback resonators.
- Cascade of feedback integrators.
- Cascade of feed-forward resonators.
- Cascade of feed-forward integrators.

You can model an ADC of orders from two to six. You can also model circuit based noise.

Ports

Input

In — Analog input signal

scalar

Analog input signal, specified as a scalar.

Data Types: double

Reset — Reset Delta Sigma Modulator

scalar

Reset signal for Delta Sigma Modulator block, specified as a scalar.

Data Types: double

Output

Out — Digital output signal

scalar

Digital output signal, returned as a scalar.

Data Types: fixed point | single | double | int8 | int16 | int32 | uint8 | uint16 | uint32 | Boolean

Parameters

Delta Sigma Modulator architecture — Architecture of delta sigma modulator

CIFB (default) | CRFB | CIFF | CRFF

Architecture of the delta sigma modulator, specified as:

- CIFB — cascade of feedback integrators.
- CRFB — cascade of feedback resonators.
- CIFF — cascade of feed-forward integrators.
- CRFF — cascade of feed-forward resonators.

Programmatic Use

Block parameter: dsmArchitecture

Type: character vector

Values: CIFB | CRFB | CIFF | CRFF

Default: CIFB

Delta Sigma Modulator order — Order of delta sigma modulator

2nd order (default) | 3rd order | 4th order | 5th order | 6th order

Order of delta sigma modulator. You can choose between orders two to six.

Programmatic Use

Block parameter: dsmOrder

Type: character vector

Values: 2nd order | 3rd order | 4th order | 5th order | 6th order

Default: 2nd order

Input Parameters

Sampling frequency (Hz) — Sampling frequency of delta sigma modulator

128e3 (default) | positive real scalar

Sampling frequency of the delta sigma modulator, specified as a positive real scalar in Hz.

Programmatic Use

Block parameter: SamplingFrequency

Type: character vector

Values: positive real scalar

Default: 128e3

Quantizer levels — Number of quantizer levels in delta sigma modulator

2 (default) | positive real scalar

Number of quantizer levels in delta sigma modulator, specified as a positive real scalar.

Programmatic Use

Block parameter: NumberLevels

Type: character vector

Values: positive real scalar

Default: 2

Quantizer output — Range of quantizer output

[-1;1] (default) | real valued vector

Range of the quantizer output in the delta sigma modulator architecture, specified as a vector with real elements.

Programmatic Use**Block parameter:** QuantizerOutput**Type:** character vector**Values:** real valued vector**Default:** [-1;1]**Coefficient 'a' — Feedback/feed-forward coefficients from/to the quantizer**

[0.1601 0.6515] (default) | real valued vector

Feedback coefficients from the quantizer or feed-forward coefficients to the quantizer, specified as a vector with real elements.

Programmatic Use**Block parameter:** a**Type:** character vector**Values:** real valued vector**Default:** [0.1601 0.6515]**Coefficient 'g' — Resonator coefficients**

[0.0] (default) | real valued vector

Resonator coefficients, specified as a vector with real elements.

Programmatic Use**Block parameter:** g**Type:** character vector**Values:** real valued vector**Default:** [0.0]**Coefficient 'b' — Feed-in coefficients from the modulator input to each integrator**

[0.1601 0.6515 1] (default) | real valued vector

Feed-in coefficients from the modulator input to each integrator, specified as a vector with real elements.

Programmatic Use**Block parameter:** b**Type:** character vector**Values:** real valued vector**Default:** [0.1601 0.6515 1]**Coefficient 'c' — Integrator inter-stage coefficients**

[1 1] (default) | real valued vector

Integrator inter-stage coefficients, specified as a vector with real elements.

Programmatic Use**Block parameter:** c**Type:** character vector**Values:** real valued vector**Default:** [1 1]

Noise**Enable Noise Impairment — Enable noise calculations in delta sigma modulator ADC**

on (default) | off

Enable noise calculations in the delta sigma modulator ADC.

Signal to noise ratio (dB) — Ratio of signal power to noise power

75 (default) | nonnegative real scalar

Ratio of signal power to noise power, specified as a nonnegative real scalar in dB.

Programmatic Use**Block parameter:** SNR**Type:** character vector**Values:** nonnegative real scalar**Default:** 75**Input Signal Power (W) — Power of input signal**

0.125 (default) | real scalar

Power of the signal at the input of the delta sigma modulator, specified as a real scalar in watts.

Programmatic Use**Block parameter:** InputPower**Type:** character vector**Values:** real scalar**Default:** 0.125**System Bandwidth (Hz) — Bandwidth of delta sigma modulator system**

1000 (default) | nonnegative real scalar

Bandwidth of the delta sigma modulator system,

Programmatic Use**Block parameter:** BandWidth**Type:** character vector**Values:** nonnegative real scalar**Default:** 1000

Version History

Introduced in R2021b**See Also**

SAR ADC | Flash ADC | ADC Testbench

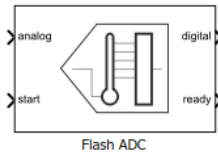
Topics

"Delta Sigma Modulator Data Converter with Half-Band Filter for Decimation"

Flash ADC

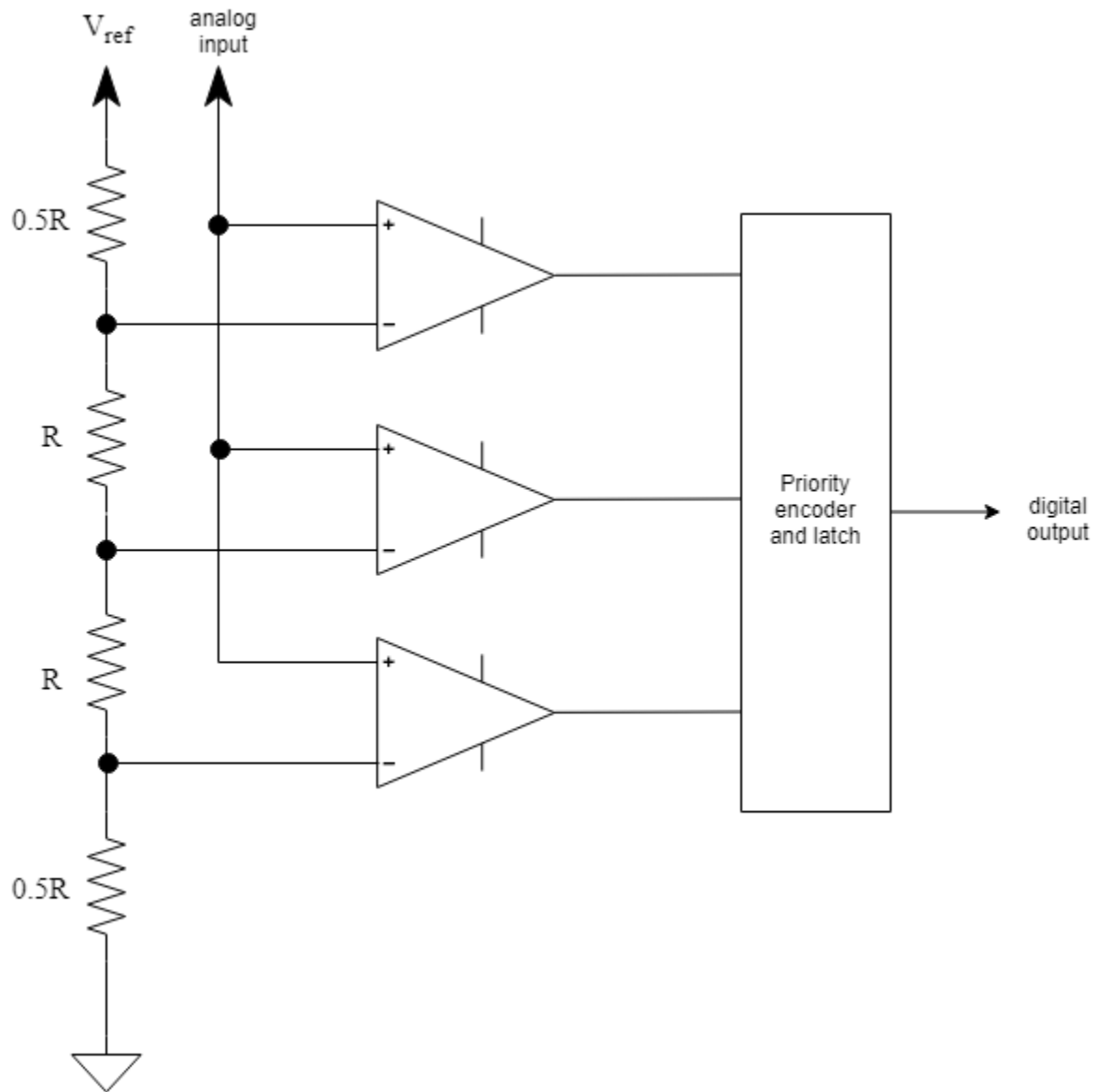
N-bit ADC with flash architecture

Library: Mixed-Signal Blockset / ADC / Architectures



Description

An N -bit flash ADC comprises of a resistive ladder that contains 2^N resistors and 2^N-1 comparators.



The reference voltage of each comparator is 1 least significant bit (LSB) higher than the one below it in the ladder. As a result, all comparators below a certain point will have input voltage greater than the reference voltage, and a logic 1 output. All comparators above that point will have input voltage smaller than the reference voltage, and a logic 0 output. The output of 2^N-1 comparators are passed through a priority encoder to produce the digital output. This encoding scheme is called thermometer encoding.

Since the analog input is applied to all the comparators at once, the flash ADC architecture is very fast. But the ADC has low resolution and high power requirements due to a large number of resistors required to implement the architecture.

Ports

Input

analog — Analog input signal

scalar

Analog input signal, specified as a scalar.

Data Types: double

start — External clock to start conversion

scalar

External clock to start conversion, specified as a scalar. The analog to digital conversion process starts at the rising edge of the signal at the **start** port.

Data Types: double

Output

digital — Converted digital output signal

scalar

Converted digital output signal, returned as scalar.

Data Types: fixed point | single | double | int8 | int16 | int32 | uint8 | uint16 | uint32 | Boolean

ready — Determines whether analog to digital conversion is complete

scalar

Determines whether the analog to digital conversion is complete, returned as a scalar.

Data Types: double

Parameters

Configuration

Number of bits — Number of physical output bits

10 (default) | positive real integer in the range [1, 26]

Number of physical output bits, specified as a unitless positive real integer in the range [1, 26]. **Number of bits** determines the resolution of the ADC.

Programmatic Use

- Use `get_param(gcb, 'NBits')` to view the current **Number of bits**.
- Use `set_param(gcb, 'NBits', value)` to set **Number of bits** to a specific value.

Data Types: double

Input range (V) — ADC dynamic range

[-1 1] (default) | 2-element row vector

ADC dynamic range, specified as a 2-element row vector in volts.

Programmatic Use

- Use `get_param(gcb, 'InputRange')` to view the current **Input range (V)**.
- Use `set_param(gcb, 'InputRange', value)` to set **Input range (V)** to a specific value.

Data Types: double

Use external start clock – Connect to external start conversion clock

on (default) | off

Select to connect to an external start conversion clock. By default, this option is selected. If you deselect this option, a Sampling Clock Source block inside the Flash ADC is used to generate the start conversion clock.

Conversion start frequency (Hz) – Frequency of internal start conversion clock

1e6 (default) | positive real scalar

Frequency of internal start conversion clock, specified as a positive real scalar in hertz. **Conversion start frequency (Hz)** determines the rate of the ADC.

Dependencies

This parameter is only available when **Use external start clock** is not selected.

Programmatic Use

- Use `get_param(gcb, 'StartFreq')` to view the current value of **Conversion start frequency (Hz)**.
- Use `set_param(gcb, 'StartFreq', value)` to set **Conversion start frequency (Hz)** to a specific value.

Data Types: double

RMS aperture jitter (s) – RMS aperture jitter added to the start conversion clock

1e-12 (default) | real nonnegative scalar

RMS aperture jitter added as an impairment to the start conversion clock, specified as a real nonnegative scalar in seconds. Set **RMS aperture jitter** value to zero if you want a clean clock signal.

Dependencies

This parameter is only available when **Use external start clock** is not selected.

Programmatic Use

- Use `get_param(gcb, 'StartClkJitter')` to view the current value of **RMS aperture jitter (s)**.
- Use `set_param(gcb, 'StartClkJitter', value)` to set **RMS aperture jitter (s)** to a specific value.

Data Types: double

Edge trigger type – Clock edge type that triggers the output update

Rising edge (default) | Falling edge | Either edge

Clock edge type that triggers the output update:

- **Rising edge** — the output is updated with the rising edge of the clock signal.
- **Falling edge** — the output is updated with the falling edge of the clock signal.
- **Either edge** — the output is updated with both the rising and the falling edge of the clock signal.

Programmatic Use

- Use `get_param(gcb, 'Trigger')` to view the current **Edge trigger type**.
- Use `set_param(gcb, 'Trigger', value)` to set **Edge trigger type** to a specific value.

Match input scale — Inherit output polarity and data type from input`off (default) | on`

Inherit the output polarity and data type from the analog input signal to the ADC. When this option is selected, it forces the ADC to output a scalar double matched to the input's scale of the ADC.

Output polarity — Defines ADC output polarity`Auto (default) | Bipolar | Unipolar`

Defines the ADC output data polarity.

If **Output polarity** is set to `Auto`, the minimum and maximum values of the output are determined by the polarity of the **Input range**.

If **Output polarity** is set to `Bipolar`, the outputs are between $-2^{N_{\text{bits}}-1}$ and $2^{N_{\text{bits}}-1}-1$.

If **Output polarity** is set to `Unipolar`, the outputs are between 0 and $2^{N_{\text{bits}}}-1$.

Dependencies

This parameter is only editable when **Match input scale** option is deselected.

Programmatic Use

- Use `get_param(gcb, 'OutputPolarity')` to view the current **Output polarity**.
- Use `set_param(gcb, 'OutputPolarity', value)` to set **Output polarity** to a specific value.

Output data type — Defines ADC output data type`fixdt(1, Nbits) (default) | fixdt(0, Nbits) | double | single | int8 | int16 | int32 | uint8 | uint16 | uint32 | Inherit: Inherit via back propagation`

Defines ADC output data type.

Unsigned integers and fixed-point types (`fixdt(0, Nbits)`) are not available when the **Output polarity** is set to `Bipolar` or `Auto`.

Signed integers and fixed-point types (`fixdt(1, Nbits)`) are not available when the **Output polarity** is set to `Unipolar`.

Dependencies

This parameter is only editable when **Match input scale** option is deselected.

Programmatic Use

- Use `get_param(gcb, 'OutDataType')` to view the current **Output data type**.

- Use `set_param(gcb, 'OutDataType', value)` to set **Output data type** to a specific value.

Impairments

Enable impairments – Enable impairments in ADC simulation

off (default) | on

Select to enable impairments such as offset error and gain error in ADC simulation. By default, this option is deselected.

Offset error – Shifts quantization steps by specific value

3 LSB (default) | real scalar

Shifts quantization steps by specific value, specified as a scalar in %FS, FS, or LSB.

Note The full scale range of the converter is defined as the difference between the last and first code on the +0.5 LSB compensated transfer curve. In a +0.5 LSB compensated transfer curve, first code is 0.5 LSB wide while the last code is 1.5 LSB wide. The input values must be considered within the full scale range of the converter.

Note LSB is calculated by the equation $LSB = \frac{\text{Full scale range of converter}}{2^{Nbits}}$.

Dependencies

To enable this parameter, select **Enable impairments** in the **Impairments** tab.

Programmatic Use

Block parameter: OffsetError

Type: character vector

Values: real scalar

Default: 3 LSB

Data Types: double

Gain error – Error on slope of ADC transfer curve

2 LSB (default) | real scalar

Error on the slope of the straight line interpolating ADC transfer curve, specified as a real scalar in %FS, FS, or LSB.

Note The full scale range of the converter is defined as the difference between the last and first code on the +0.5 LSB compensated transfer curve. In a +0.5 LSB compensated transfer curve, first code is 0.5 LSB wide while the last code is 1.5 LSB wide. The input values must be considered within the full scale range of the converter.

Note LSB is calculated by the equation $LSB = \frac{\text{Full scale range of converter}}{2^{Nbits}}$.

Dependencies

To enable this parameter, select **Enable impairments** in the **Impairments** tab.

Programmatic Use

Block parameter: GainError

Type: character vector

Values: real scalar

Default: 2 LSB

Data Types: double

Missing codes — Position of the failed comparators

[] (default) | row vector with positive real values

Position of the failed comparators, specified as a row vector with positive real values.

Dependencies

This parameter is only available when **Enable impairments** is selected.

Programmatic Use

- Use `get_param(gcb, 'Bubbles')` to view the current **Missing codes**.
- Use `set_param(gcb, 'Bubbles', value)` to set **Missing codes** to a specific value.

Data Types: double

Version History

Introduced in R2019a

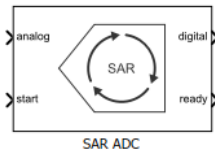
See Also

SAR ADC | Aperture Jitter Measurement | ADC DC Measurement | ADC AC Measurement | ADC Testbench

SAR ADC

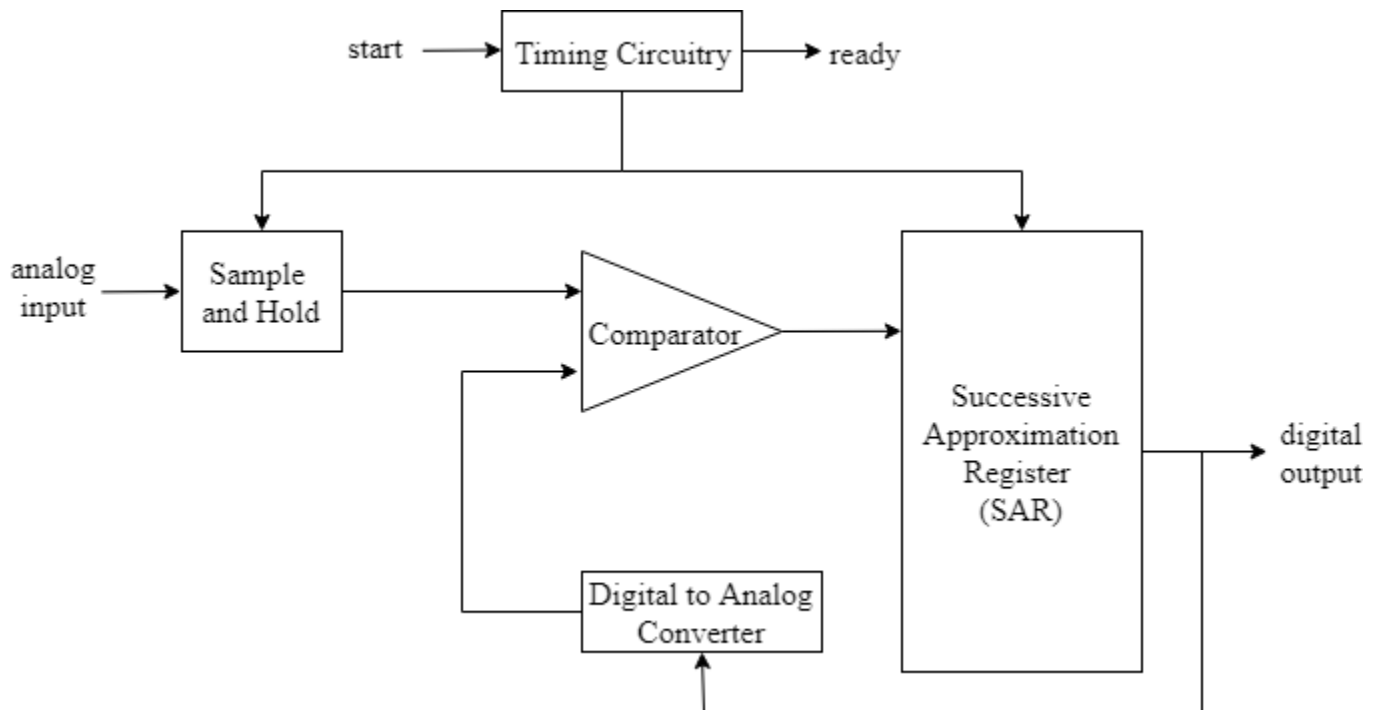
N-bit successive approximation register (SAR) based ADC

Library: Mixed-Signal Blockset / ADC / Architectures



Description

Successive Approximation Register (SAR) based ADC consists of a sample and hold circuit (SHA), a comparator, an internal digital to analog converter (DAC), and a successive approximation register.



When the ADC receives the **start** command, SHA is placed in hold mode. The most significant bit (MSB) of the SAR is set to logic 1, and all other bits are set to logic 0.

The output of the SAR is fed back to a DAC, whose output is compared with the incoming input signal. If the DAC output is greater than the analog input, MSB is reset, otherwise it is left set. The next MSB is now set to 1, and the process is repeated until every bit the SAR is compared. The final value of the SAR at the end of this process corresponds to the analog input value. The end of the conversion process is indicated by the **ready** signal.

Ports

Input

analog — Analog input signal

scalar

Analog input signal, specified as a scalar.

Data Types: double

start — External clock to start conversion

scalar

External clock to start conversion, specified as a scalar. The analog to digital conversion process starts at the rising edge of the signal at the **start** port.

Data Types: double

Output

digital — Converted digital output signal

scalar

Converted digital output signal, returned as a scalar.

Data Types: fixed point | single | double | int8 | int16 | int32 | uint8 | uint16 | uint32 | Boolean

ready — Determines whether analog to digital conversion is complete

scalar

Determines whether the analog to digital conversion is complete, returned as a scalar.

Data Types: double

Parameters

Configuration

Number of bits — Number of physical output bits

8 (default) | positive real integer in the range [1, 26]

Number of physical output bits, specified as a unitless positive real integer in the range [1, 26]. **Number of bits** determines the resolution of the ADC.

Programmatic Use

- Use `get_param(gcb, 'NBits')` to view the current **Number of bits**.
- Use `set_param(gcb, 'NBits', value)` to set **Number of bits** to a specific value.

Data Types: double

Input range (V) — ADC dynamic range

[-1 1] (default) | 2-element row vector

ADC dynamic range, specified as a 2-element row vector in volts.

Programmatic Use

- Use `get_param(gcb, 'InputRange')` to view the current **Input range (V)**.
- Use `set_param(gcb, 'InputRange', value)` to set **Input range (V)** to a specific value.

Data Types: double

Use external start clock – Connect to external start conversion clock

on (default) | off

Select to connect to an external start conversion clock. By default, this option is selected. If you deselect this option, a Sampling Clock Source block inside the SAR ADC is used to generate the start conversion clock.

Conversion start frequency (Hz) – Frequency of internal start conversion clock

10e3 (default) | positive real scalar

Frequency of internal start conversion clock, specified as a positive real scalar in Hz. **Conversion start frequency** determines the rate of the ADC.

Dependencies

This parameter is only available when **Use external start clock** is not selected.

Programmatic Use

- Use `get_param(gcb, 'StartFreq')` to view the current value of **Conversion start frequency (Hz)**.
- Use `set_param(gcb, 'StartFreq', value)` to set **Conversion start frequency (Hz)** to a specific value.

Data Types: double

RMS aperture jitter (s) – RMS aperture jitter added to the start conversion clock

0 (default) | real nonnegative scalar

RMS aperture jitter added as an impairment to the start conversion clock, specified as a real nonnegative scalar in s. Set **RMS aperture jitter** value to zero if you want a clean clock signal.

Dependencies

This parameter is only available when **Use external start clock** is not selected.

Programmatic Use

- Use `get_param(gcb, 'StartClkJitter')` to view the current value of **RMS aperture jitter (s)**.
- Use `set_param(gcb, 'StartClkJitter', value)` to set **RMS aperture jitter (s)** to a specific value.

Data Types: double

SAR Frequency (Hz) – Frequency of SAR clock

2e7 (default) | real scalar

Frequency of the SAR clock, specified as a real scalar in Hz. **SAR Frequency (Hz)** must be high enough to allow the ADC to perform *Nbits* comparison, where *Nbits* is the **Number of bits** of the

ADC. The block has one cycle overhead due to algebraic loop removal. So, the clock must run for one additional cycle before the output is ready. So, the **SAR Frequency (Hz)** (f_{SAR}) is given by the equation $f_{\text{SAR}} \geq (N_{\text{bits}} + 1)f_{\text{start}}$, where f_{start} is the **Conversion start frequency**.

Programmatic Use

- Use `get_param(gcb, 'SARFreq')` to view the current value of **SAR Frequency (Hz)**.
- Use `set_param(gcb, 'SARFreq', value)` to set **SAR Frequency (Hz)** to a specific value.

Match input scale — Inherit output polarity and data type from input

`off (default) | on`

Inherit the output polarity and data type from the analog input signal to the ADC. When this option is selected, it forces the ADC to output a scalar double matched to the input's scale of the ADC.

Output polarity — Defines ADC output polarity

`Auto (default) | Bipolar | Unipolar`

Defines the ADC output data polarity.

If **Output polarity** is set to `Auto`, the minimum and maximum values of the output are determined by the polarity of the **Input range**.

If **Output polarity** is set to `Bipolar`, the outputs are between $-2^{N_{\text{bits}}-1}$ and $2^{N_{\text{bits}}-1}-1$.

If **Output polarity** is set to `Unipolar`, the outputs are between 0 and $2^{N_{\text{bits}}}-1$.

Dependencies

This parameter is only editable when **Match input scale** option is deselected.

Programmatic Use

- Use `get_param(gcb, 'OutputPolarity')` to view the current **Output polarity**.
- Use `set_param(gcb, 'OutputPolarity', value)` to set **Output polarity** to a specific value.

Output data type — Defines ADC output data type

`fixdt(1, Nbits) (default) | fixdt(0, Nbits) | double | single | int8 | int16 | int32 | uint8 | uint16 | uint32 | Inherit: Inherit via back propagation`

Defines ADC output data type.

Unsigned integers and fixed-point types (`fixdt(0, Nbits)`) are not available when the **Output polarity** is set to `Bipolar` or `Auto`.

Signed integers and fixed-point types (`fixdt(1, Nbits)`) are not available when the **Output polarity** is set to `Unipolar`.

Dependencies

This parameter is only editable when **Match input scale** option is deselected.

Programmatic Use

- Use `get_param(gcb, 'OutDataType')` to view the current **Output data type**.
- Use `set_param(gcb, 'OutDataType', value)` to set **Output data type** to a specific value.

Impairments

Enable impairments — Enable impairments in ADC simulation

off (default) | on

Select to enable impairments such as offset error and gain error in ADC simulation. By default, this option is deselected.

Offset error — Shifts quantization steps by specific value

1 LSB (default) | real scalar

Shifts quantization steps by specific value, specified as a scalar in least significant bit (LSB) or %.

Note The full scale range of the converter is defined as the difference between the last and first code on the +0.5 LSB compensated transfer curve. In a +0.5 LSB compensated transfer curve, first code is 0.5 LSB wide while the last code is 1.5 LSB wide. The input values must be considered within the full scale range of the converter.

Note LSB is calculated by the equation $LSB = \frac{\text{Full scale range of converter}}{2^{Nbits}}$.

Dependencies

This parameter is only available when **Enable impairments** is selected.

Programmatic Use

- Use `get_param(gcb, 'OffsetError')` to view the current value of **Offset error (LSB)**.
- Use `set_param(gcb, 'OffsetError', value)` to set **Offset error (LSB)** to a specific value.

Data Types: double

Gain error — Error on slope of ADC transfer curve

2 LSB (default) | real scalar

Error on the slope of the straight line interpolating ADC transfer curve, specified as a real scalar in %FS, FS, or LSB.

Note The full scale range of the converter is defined as the difference between the last and first code on the +0.5 LSB compensated transfer curve. In a +0.5 LSB compensated transfer curve, first code is 0.5 LSB wide while the last code is 1.5 LSB wide. The input values must be considered within the full scale range of the converter.

Note LSB is calculated by the equation $LSB = \frac{\text{Full scale range of converter}}{2^{Nbits}}$.

Dependencies

To enable this parameter, select **Enable impairments** in the **Impairments** tab.

Programmatic Use

Block parameter: GainError

Type: character vector

Values: real scalar

Default: 2 LSB

Data Types: double

Version History

Introduced in R2019a

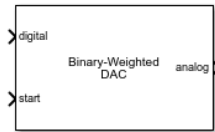
See Also

Flash ADC | Aperture Jitter Measurement | ADC DC Measurement | ADC AC Measurement | ADC Testbench

Binary Weighted DAC

N-bit DAC based on R-2R weighted resistor architecture

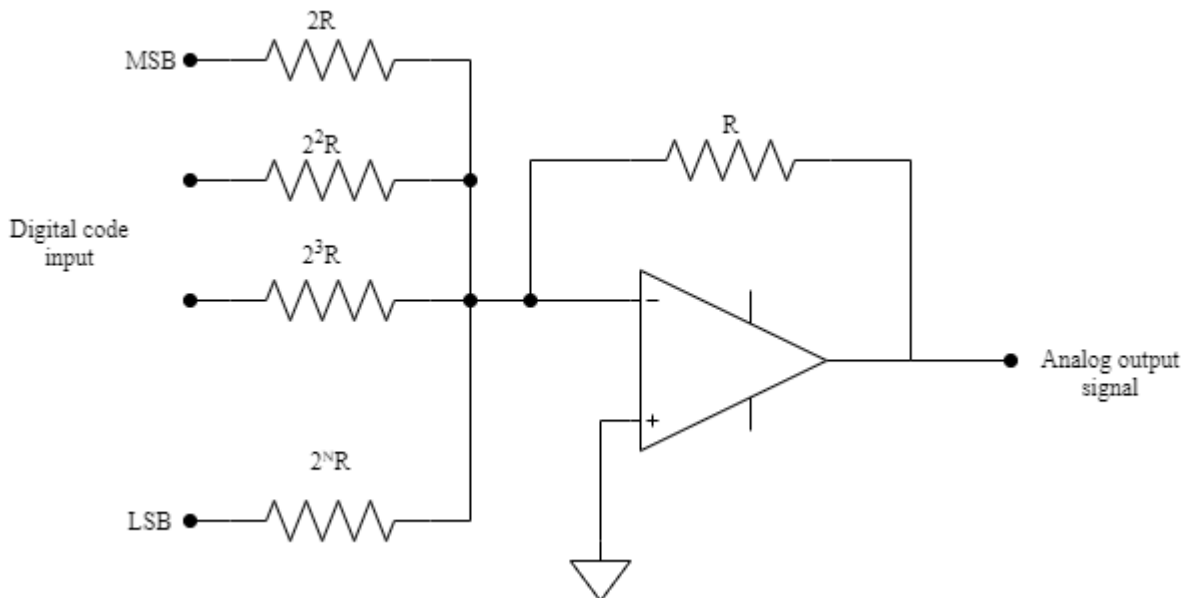
Library: Mixed-Signal Blockset / DAC / Architectures



Description

The R-2R DAC is one of the most common types of Binary-Weighted DACs. It consists of a parallel binary-weighted resistor bank. Each digital level is converted to an equivalent analog signal by the resistor bank.

The input/output transfer curve of the binary weighted DAC can be nonmonotonic, which means that the transfer curve can reverse its direction.



The R-2R DAC architecture is low resolution and consumes more power due to the large number of resistors required to implement the architecture.

Ports

Input

digital – Digital input signal to DAC

integer

Digital input signal to DAC, specified as an integer.

If the **Input polarity** parameter is set to **Bipolar**, the allowed range of the signal is $[-2^{NBits-1}, 2^{NBits-1}]$.

If the **Input polarity** parameter is set to **Unipolar**, the allowed range of the signal is $[0, 2^{NBits-1}]$.

Data Types: `single` | `double` | `int8` | `int16` | `int32` | `uint8` | `uint16` | `uint32` | `fixed point`

start — External clock to start conversion

scalar

External clock to start conversion, specified as a scalar. The digital-to-analog conversion process starts at the rising edge of the signal at the **start** port.

Dependencies

To enable this port, select **Use external start clock** in the **General** tab.

Data Types: `double`

Output

analog — Converted analog output signal

scalar

Converted analog output signal, returned as a scalar.

Data Types: `double`

ready — Indicates whether digital-to-analog conversion is complete

scalar

Indicates whether the digital-to-analog conversion is complete, returned as a scalar.

Dependencies

To enable this port, select **Show ready port** in the **General** tab.

Data Types: `double`

Parameters

General

Number of bits — Number of bits in input word

5 (default) | positive real integer

Number of bits in the input word, specified as a unitless positive real integer. **Number of bits** determines the resolution of the DAC.

Programmatic Use

Block parameter: `NBits`

Type: character vector

Values: positive real integer

Default: 5

Data Types: `double`

Input polarity — Polarity of input signal to DAC

Bipolar (default) | Unipolar

Polarity of the input signal to the DAC.

Programmatic Use**Block parameter:** Polarity**Type:** character vector**Values:** Bipolar|Unipolar**Default:** Bipolar**Use external start clock — Connect to external start conversion clock**

on (default) | off

Select to connect to an external start conversion clock. By default, this option is selected. If you deselect this option, a Sampling Clock Source block inside the Segmented DAC is used to generate the start conversion clock

Conversion start frequency (Hz) — Frequency of internal start conversion clock

1e6 (default) | positive real scalar

Frequency of the internal start conversion clock, specified as a real scalar in Hz. The **Conversion start frequency** parameter determines the conversion rate at the start of conversion.

Dependencies

To enable this parameter, deselect **Use external start clock**.

Programmatic Use**Block parameter:** StartFreq**Type:** character vector**Values:** positive real scalar**Default:** 1e6

Data Types: double

Reference (V) — Reference voltage

2 (default) | real scalar

Reference voltage of the DAC, specified as a real scalar in volts. **Reference (V)** helps determine the output from the input digital code, **Number of bits**, and **Bias (V)** using the equation:

$$\text{DAC output} = \left(\left(\frac{\text{Digital input code}}{2^{\text{Number of bits}}} \right) \text{Reference} \right) + \text{Bias}.$$

Programmatic Use**Block parameter:** Ref**Type:** character vector**Values:** real scalar**Default:** 2

Data Types: double

Bias (V) — Bias voltage added to output

0 (default) | real scalar

Bias voltage added to the output of the DAC, specified as a real scalar in volts. **Bias (V)** helps determine the output from the input digital code, **Number of bits**, and **Reference (V)** using the equation:

$$\text{DAC output} = \left(\left(\frac{\text{Digital input code}}{2^{\text{Number of bits}}} \right) \text{Reference} \right) + \text{Bias}.$$

Programmatic Use**Block parameter:** Bias**Type:** character vector**Values:** real scalar**Default:** 0

Data Types: double

Show ready port — Enable ready port on block

off (default) | on

Select to enable the **ready** port on the block. This option is deselected by default.

Impairments**Enable linearity impairments — Enable offset and gain errors in DAC simulation**

on (default) | off

Select to enable impairments such as offset error and gain error in DAC simulation. This parameter is selected. by default.

Offset error — Shifts quantization steps by specific value

0 LSB (default) | real scalar

Shifts quantization steps by a specific value, specified as a scalar in %FS (percentage full scale), FS (full scale), or LSB (least significant bit).

Offset error is applied before **Reference (V)** and **Bias (V)**.

Note The full scale range of the converter is defined as the difference between the last and first code on the +0.5 LSB compensated transfer curve. In a +0.5 LSB compensated transfer curve, first code is 0.5 LSB wide while the last code is 1.5 LSB wide. The input values must be considered within the full scale range of the converter.

Note LSB is calculated by the equation $\text{LSB} = \frac{\text{Full scale range of converter}}{2^{N_{\text{bits}}}}$.

Dependencies

To enable this parameter, select **Enable linearity impairments** in the **Impairments** tab.

Programmatic Use**Block parameter:** OffsetError**Type:** character vector**Values:** real scalar

Default: 0 LSB

Data Types: double

Gain error – Error in slope of DAC transfer curve

0 LSB (default) | real scalar

Error in the slope of the straight line interpolating the DAC transfer curve, specified as a real scalar in %FS (percentage full scale), FS (full scale), or LSB (least significant bit).

Gain error is applied before **Reference (V)** and **Bias (V)**.

Note The full scale range of the converter is defined as the difference between the last and first code on the +0.5 LSB compensated transfer curve. In a +0.5 LSB compensated transfer curve, first code is 0.5 LSB wide while the last code is 1.5 LSB wide. The input values must be considered within the full scale range of the converter.

Note LSB is calculated by the equation $LSB = \frac{\text{Full scale range of converter}}{2^{Nbits}}$.

Dependencies

To enable this parameter, select **Enable linearity impairments** in the **Impairments** tab.

Programmatic Use

Block parameter: GainError

Type: character vector

Values: real scalar

Default: 0 LSB

Data Types: double

Enable timing impairments – Enable timing impairments in DAC simulation

on (default) | off

Select to enable timing impairments such as settling time or slew rate in DAC simulation. This parameter is selected. by default.

Specify switch timing using – Specify how DAC calculates switch timing

Settling time (default) | Slew rate

Specify whether the Binary Weighted DAC calculates switch timing using the settling time parameters or the slew rate parameters.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Impairments** tab.

Settling time (s) – Time required for output to settle

2e-7 (default) | nonnegative real scalar

The time required for the output of the DAC to settle to within some fraction of its final value, specified as a nonnegative real scalar in seconds.

Dependencies

To enable this parameter, select **Enable timing impairments** and set **Specify switch timing using** to **Settling time** in the **Impairments** tab.

Programmatic Use

Block parameter: SettlingTime

Type: character vector

Values: real scalar

Default: 2e-7

Data Types: double

Settling time tolerance (LSB) — Tolerance for calculating settling time

0.5 (default) | positive real scalar

The tolerance allowed for calculating settling time, specified as a positive real scalar in LSB. The output of the DAC must settle within the **Settling time tolerance (LSB)** by **Settling time (s)**.

Dependencies

To enable this parameter, select **Enable timing impairments** and set **Specify switch timing using** to **Settling time** in the **Impairments** tab.

Programmatic Use

Block parameter: SettlingTimeTolerance

Type: character vector

Values: positive real scalar

Default: 0.5

Data Types: double

Rising slew rate — Switch rising slew rate for DAC

5015625 (default) | positive real scalar | positive real vector

Switch the rising slew rate for the DAC, specified as a positive real scalar or vector. If **Rising slew rate** is scalar, it specifies the same slew rate for all the switches. If **Rising slew rate** is a vector of length *Nbits*, it specifies the slew rate for each individual switch.

Dependencies

To enable this parameter, select **Enable timing impairments** and set **Specify switch timing using** to **Slew rate** in the **Impairments** tab.

Programmatic Use

Block parameter: RisingSlewRate

Type: character vector

Values: positive real scalar | positive real vector

Default: 5015625

Falling slew rate — Switch falling slew rate for DAC

-5015625 (default) | negative real scalar | negative real vector

Switch the falling slew rate for the DAC, specified as a positive real scalar or vector. If **Falling slew rate** is scalar, it specifies the same slew rate for all the switches. If **Falling slew rate** is a vector of length *Nbits*, it specifies the slew rate for each individual switch.

Dependencies

To enable this parameter, select **Enable timing impairments** and set **Specify switch timing using** to **Slew rate** in the **Impairments** tab.

Programmatic Use

Block parameter: FallingSlewRate

Type: character vector

Values: negative real scalar | negative real vector

Default: -5015625

Version History

Introduced in R2020a

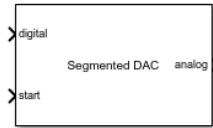
See Also

DAC Testbench | DAC DC measurement | DAC AC measurement | in1dn1

Segmented DAC

Convert large digital input to analog signal using arrangement of smaller DACs

Library: Mixed-Signal Blockset / DAC / Architectures



Description

The Segmented DAC block converts a large digital signal into analog output by splitting it over several smaller DACs. The Segmented DAC block supports up to five binary-weighted segmented DACs.

Ports

Input

digital — Digital input signal

integer

Digital input signal to DAC, specified as an integer.

If the **Input polarity** parameter is set to **Bipolar**, the allowed range of the signal is $[-2^{NBits-1}, 2^{NBits-1}]$.

If the **Input polarity** parameter is set to **Unipolar**, the allowed range of the signal is $[0, 2^{NBits-1}]$.

Data Types: `single` | `double` | `int8` | `int16` | `int32` | `uint8` | `uint16` | `uint32` | `fixed point`

start — External clock to start conversion

scalar

External clock to start conversion, specified as a scalar. The digital to-analog conversion process starts at the rising edge of the signal at the **start** port.

Dependencies

To enable this port, select **Use external start clock**.

Data Types: `double`

Output

analog — Converted analog output signal

scalar

Converted analog output signal, returned as a scalar.

Data Types: `double`

Parameters

Input polarity — Define DAC input data polarity

Unipolar (default) | Bipolar

Define the polarity of the DAC input data. Set **Input polarity** to:

- Unipolar when the digital input can only be positive.
- Bipolar when the digital input can be both positive and negative.

Programmatic Use

Block parameter: Polarity

Type: character vector

Values: Unipolar | Bipolar

Default: Unipolar

Use external start clock — Connect to external start conversion clock

on (default) | off

Select to connect to an external start conversion clock. This option is selected by default. If you deselect this option, a Sampling Clock Source block inside the Segmented DAC block is used to generate the start conversion clock.

Conversion start frequency (Hz) — Frequency of internal start conversion clock

1e6 (default) | real scalar

Frequency of the internal start conversion clock, specified as a real scalar in Hz. The **Conversion start frequency** parameter determines the conversion rate at the start of conversion.

Dependencies

To enable this parameter, deselect **Use external start clock**.

Programmatic Use

Block parameter: StartFreq

Type: character vector

Values: real scalar

Default: 1e6

Reference (V) — DAC output reference magnitude

0.5 (default) | real scalar

The reference magnitude of the DAC output, specified as a real scalar in volts. **Reference (V)** is one least significant bit (LSB) greater than the maximum achievable output.

Programmatic Use

Block parameter: Ref

Type: character vector

Values: real scalar

Default: 0.5

Bias (V) — Difference between analog output for code zero and analog zero

0 (default) | real scalar

The difference between the analog output for code zero and analog zero in an unimpaired DAC, specified as a real scalar in volts.

Programmatic Use**Block parameter:** Bias**Type:** character vector**Values:** real scalar**Default:** 0**Settling time (s) — Time required for output to settle**

2e-7 (default) | nonnegative real scalar

The time required for the output of the DAC to settle to within some fraction of its final value, specified as a nonnegative real scalar in seconds.

Programmatic Use**Block parameter:** SettlingTime**Type:** character vector**Values:** real scalar**Default:** 2e-7**Settling time tolerance (LSB) — Tolerance for calculating settling time**

0.5 (default) | positive real scalar

The tolerance allowed for calculating settling time, specified as a positive real scalar in LSB. The output of the DAC must settle within the **Settling time tolerance (LSB)** by **Settling time (s)**.

Programmatic Use**Block parameter:** SettlingTimeTolerance**Type:** character vector**Values:** positive real scalar**Default:** 0.5**Segment settings****Topology — Topology of base DAC segment**

Binary Weighted (default)

The topology of the base DAC for the segment. You can only use a Binary Weighted DAC.

Bits — Number of physical input bits for DAC segment

real scalar greater than 2

Number of physical input bits for the DAC segment, specified as a real scalar greater than 2. For the first two DAC segments, the default value of **Bits** is 4. For subsequent segments, the default value is 2.

Offset error — Shift quantization steps

0 (default) | real scalar

Shift quantization steps by the value you provide in **Offset error** parameter, specified as a real scalar.

Offset error unit — Unit of offset error

LSB (default) | FS | %FS

Unit of offset error, specified as LSB, full scale (FS), or percentage full scale (%FS).

Note The full scale range of the converter is defined as the difference between the last and first code on the +0.5 LSB compensated transfer curve. In a +0.5 LSB compensated transfer curve, first code is

0.5 LSB wide while the last code is 1.5 LSB wide. The input values must be considered within the full scale range of the converter.

Note LSB is calculated by the equation $LSB = \frac{\text{Full scale range of converter}}{2^{Nbits}}$.

Gain error – Error in slope of DAC transfer curve

0 (default) | real scalar

Error in the slope of the DAC transfer curve, specified as a real scalar.

Gain error unit – Unit of gain error

LSB (default) | FS | %FS

Unit of gain error, specified as LSB, full scale (FS), or percentage full scale (%FS).

Note The full scale range of the converter is defined as the difference between the last and first code on the +0.5 LSB compensated transfer curve. In a +0.5 LSB compensated transfer curve, first code is 0.5 LSB wide while the last code is 1.5 LSB wide. The input values must be considered within the full scale range of the converter.

Note LSB is calculated by the equation $LSB = \frac{\text{Full scale range of converter}}{2^{Nbits}}$.

New segment – Add new DAC segment

button

Click to add a new DAC segment with default values. Currently you can add up to five DAC segments to the Segmented DAC block.

Duplicate selected – Duplicate selected segment

button

Click to add a new DAC segment by duplicating a selected segment. Currently you can add up to five segments to the Segmented DAC block.

Delete selected – Delete selected segment

button

Click to delete the selected DAC segment from the Segmented DAC block.

Version History

Introduced in R2021a

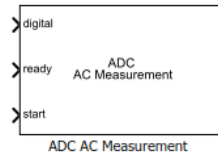
See Also

DAC Testbench | Binary Weighted DAC

ADC AC Measurement

Measure AC performance metrics of ADC output

Library: Mixed-Signal Blockset / ADC / Measurements & Testbenches



Description

The ADC AC Measurement block measures ADC AC performance metrics such as signal to noise ratio (SNR), signal to noise and distortion ratio (SINAD), spurious free dynamic range (SFDR), effective number of bits (ENOB), noise floor, and conversion delay. You can use ADC AC Measurement block to validate the ADC architectural models provided in Mixed-Signal Blockset, or you can use an ADC of your own implementation

Ports

Input

digital — Converted digital signal from ADC

scalar

Converted digital signal from an ADC, specified as a scalar.

Data Types: `fixed_point` | `single` | `double` | `int8` | `int16` | `int32` | `uint8` | `uint16` | `uint32`

ready — Indicates whether analog to digital conversion is complete

scalar

Indicates whether the analog to digital conversion is complete, specified as a scalar.

Data Types: `double`

start — External conversion start clock

scalar

External conversion start clock, specified as a scalar. The analog to digital conversion process starts at the rising edge of the signal at the **start** port.

Data Types: `double`

Parameters

Distortion measurement type — Type of distortion to measure

Harmonic (default) | Intermodulation

Type of distortion the ADC AC Measurement block is set to measure, specified as Harmonic or Intermodulation.

Programmatic Use**Block parameter:** DistortionMeasurement**Type:** character vector**Values:** Harmonic | Intermodulation**Default:** Harmonic**Analog stimulus frequency (Hz) — Frequency of the analog input signal to ADC**

positive real scalar | positive real valued vector

Frequency of the analog input signal to an ADC block, specified as a positive real scalar in hertz.

Analog stimulus frequency must match the input frequency to the ADC device under test.

- When the ADC AC Measurement is set to measure the Harmonic distortion, the default value of **Analog stimulus frequency** is 10000.
- When the ADC AC Measurement is set to measure the Intermodulation distortion, the default value of **Analog stimulus frequency** is [9000, 11000].

Analog stimulus frequency needs to satisfy two requirements:

- All the output codes of the ADC must be activated.
- The **Analog stimulus frequency** must not share any common multiples other than 1 with the **Start conversion frequency**.

To satisfy both the conditions, use the equation $f_{\text{analog}} = \frac{J}{M}f_{\text{start}}$ [2],

where:

f_{analog} is the analog signal frequency,

f_{start} is the start conversion frequency,

$M > 2^{N_{\text{bits}}} \cdot \pi$, where N_{bits} is the number of bits of the ADC,

and J is an integer with no common factors with M .

Programmatic Use**Block parameter:** InputFrequency**Type:** character vector**Values:** positive real scalar | positive real valued vector**Default:** 10000**Resolution bandwidth (Hz) — Resolution bandwidth**

positive real scalar

Resolution bandwidth, specified as a positive real scalar in hertz. This parameter defines the smallest positive frequency that can be resolved. By default, this parameter is calculated automatically. You can deselect **Set automatically** to customize the value.

- When the ADC AC Measurement is set to measure the Harmonic distortion, the default value of **Resolution bandwidth (Hz)** is 1000.
- When the ADC AC Measurement is set to measure the Intermodulation distortion, the default value of **Resolution bandwidth (Hz)** is 900.

Programmatic Use**Block parameter:** RBW**Type:** character vector**Values:** positive real scalar**Default:** 1000**Number of bits — Number of physical bits in ADC**

5 (default) | positive real integer

Number of physical bits in ADC, specified as a unitless positive real integer. **Number of bits** must match the resolution specified in the ADC block.

Programmatic Use**Block parameter:** NBits**Type:** character vector**Values:** positive real integer**Default:** 5**Start conversion frequency (Hz) — Frequency of the start conversion clock of the ADC**

10e6 (default) | positive real scalar

Frequency of the start conversion clock of the ADC, specified as a positive real scalar in hertz. **Start conversion frequency** must match the frequency of the start conversion clock of the ADC block.

Programmatic Use**Block parameter:** Frequency**Type:** character vector**Values:** positive real scalar**Default:** 10e6**Hold off time (s) — Delays measurement analysis to avoid corruption by transients**

0 (default) | nonnegative real scalar

Delays measurement analysis to avoid corruption by transients, specified as a nonnegative real scalar in seconds.

Programmatic Use**Block parameter:** HoldOffTime**Type:** character vector**Values:** nonnegative real scalar**Default:** 0**Recommended min. simulation stop time (s) — Minimum time simulation must run for meaningful result**

0.009 (default) | positive real scalar

Minimum time the simulation must run to obtain meaningful results, specified as a positive real scalar in seconds.

For AC measurement, the simulation must run so that the ADC can generate 6 spectral updates of the ADC output. The time to generate one spectral output based on Welch's method [1] on page 1-38 is given by:

$$t = \frac{1.5 \cdot \text{SamplingFrequency}}{RBW}$$

where *SamplingFrequency* and *RBW* are the sampling frequency and resolution bandwidth of the spectrum estimator inside the ADC Testbench block.

This parameter is only reported by the testbench and is not editable.

Data Types: `double`

Set as model stop time – Automatically set recommended min. simulation stop time as model stop time

button

Click to automatically set the **Recommended min. simulation stop time (s)** as the stop time of the Simulink® model.

Output result to base workspace – Store detailed test results to base workspace

off (default) | on

Store detailed test results in the base workspace for further processing at the end of simulation. By default, this option is not selected.

Workspace variable name – Name of the variable that stores detailed test results

`adc_ac_out` (default) | character string

Name of the variable that stores detailed test results, specified as a character string.

Dependencies

This parameter is only available when **Output result to base workspace** is selected.

Programmatic Use

Block parameter: `VariableName`

Type: character vector

Values: character string

Default: `adc_ac_out`

Show spectrum analyzer during simulation – Displays spectrum analyzer during simulation

off (default) | on

Displays spectrum analyzer during simulation. By default, this option is not selected.

More About

SNR

Signal to noise ratio or SNR is the ratio of the RMS signal amplitude to the mean value of the root-sum-squares of all other spectral components, excluding the DC and first five harmonics.

SINAD

Signal to noise and distortion ratio, or SINAD is the ratio of the RMS signal amplitude to the mean value of the root-sum-squares of all other spectral components and harmonics, excluding DC.

SFDR

Spurious free dynamic range or SFDR is the ratio of the RMS signal amplitude to the RMS value of the peak spurious content, measured over the entire first Nyquist zone (DC to half of sampling frequency).

ENOB

Effective number of bits or ENOB represents the actual resolution of an ADC after considering internal noise and errors. It is given by $ENOB = \frac{SINAD - 1.76}{6.02}$.

Version History

Introduced in R2019a

References

- [1] Spectrum Analyzer, DSP System Toolbox, MathWorks Documentation.
- [2] IEEE Std 1241-2010. "IEEE Standard for Terminology and Test Methods for Analog-to-Digital Converters," pp. 29-30, 14 January 2011.

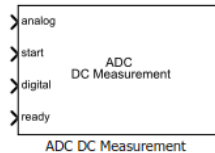
See Also

SAR ADC | Flash ADC | ADC Testbench | ADC DC Measurement

ADC DC Measurement

Measure DC performance metrics of ADC output

Library: Mixed-Signal Blockset / ADC / Measurements & Testbenches



Description

The ADC DC Measurement block measures ADC DC performance metrics such as offset error, gain error, integral nonlinearity (INL), and differential nonlinearity (DNL). You can use ADC DC Measurement block to validate the ADC architectural models provided in Mixed-Signal Blockset, or you can use an ADC of your own implementation.

Ports

Input

analog — Analog input signal to ADC

scalar

Analog input signal to ADC block, specified as a scalar.

Data Types: double

start — External conversion start clock

scalar

External conversion start clock, specified as a scalar. The analog to digital conversion process starts at the rising edge of the signal at the **start** port.

Data Types: double

digital — Converted digital signal from ADC

scalar

Converted digital signal from an ADC, specified as a scalar.

Data Types: fixed point | single | double | int8 | int16 | int32 | uint8 | uint16 | uint32

ready — Indicates whether analog to digital conversion is complete

scalar

Indicates whether the analog to digital conversion is complete, specified as a scalar.

Data Types: double

Parameters

Number of bits — Number of physical bits in ADC

5 (default) | positive real integer

Number of physical bits in ADC, specified as a unitless positive real integer. **Number of bits** must match the resolution specified in the ADC block.

Programmatic Use

- Use `get_param(gcb, 'NBits')` to view the current **Number of bits**.
- Use `set_param(gcb, 'NBits', value)` to set **Number of bits** to a specific value.

Start conversion frequency (Hz) — Frequency of the start conversion clock of ADC

10e6 (default) | positive real scalar

Frequency of the start conversion clock of the ADC, specified as a positive real scalar in hertz. **Start conversion frequency** must match the frequency of the start conversion clock of the ADC block. This parameter is used to calculate **Recommended simulation stop time**.

Programmatic Use

- Use `get_param(gcb, 'Frequency')` to view the current value of **Start conversion frequency**.
- Use `set_param(gcb, 'Frequency', value)` to set **Start conversion frequency** to a specific value.

Input range (V) — Dynamic range of ADC

[-1 1] (default) | 2-element vector

Dynamic range of the ADC, specified as a 2-element vector in V. The two vector elements represent the minimum and maximum values of the dynamic range, from left to right.

Programmatic Use

- Use `get_param(gcb, 'InputRange')` to view the current value of **Input range**.
- Use `set_param(gcb, 'InputRange', value)` to set **Input range** to a specific value.

Hold off time (s) — Delays measurement analysis to avoid corruption by transients

0 (default) | nonnegative real scalar

Delays measurement analysis to avoid corruption by transients, specified as a nonnegative real scalar in seconds.

Programmatic Use

- Use `get_param(gcb, 'HoldOffTime')` to view the current value of **Hold off time**.
- Use `set_param(gcb, 'HoldOffTime', value)` to set **Hold off time** to a specific value.

Recommended min. simulation stop time (s) — Minimum time simulation must run for meaningful result

6.4e-5 (default) | positive real scalar

Minimum time the simulation must run to obtain meaningful results, specified as a positive real scalar in seconds.

For DC measurement, the simulation must run so that ADC can sample each digital code 10 times with the default error tolerance of 0.1, assuming a ramp input that traverses the full scale range of the ADC over the period of simulation. Based on this assumption, the analog input frequency (f_{analog}), generated by the ADC Testbench block for the sawtooth waveform is set as:

$$f_{\text{analog}} = \frac{\text{StartFreq} \cdot \text{ErrorTolerance}}{2^{(N\text{bits} + 1)}}$$

where *StartFreq* is the frequency of the conversion start clock and *Nbits* is the resolution of the ADC.

So, the **Recommended min. simulation stop time (s)** (*T*) is calculated by using the formula:

$$T = \frac{1}{f_{\text{analog}}} + \text{HoldOffTime}.$$

Data Types: double

Set as model stop time — Automatically set recommended min. simulation stop time as model stop time

button

Click to automatically set the **Recommended min. simulation stop time (s)** as the stop time of the Simulink model.

Endpoint — Measure DNL, INL using endpoint method

on (default) | off

Measure the differential nonlinearity (DNL) error and integral nonlinearity (INL) error using the endpoint method. This method uses the end points of the actual transfer function to measure the DNL and INL error.

Best fit — Measure DNL, INL using best fit method

on (default) | off

Measure the differential nonlinearity (DNL) error and integral nonlinearity (INL) error using the best fit method. This method uses a standard curve fitting technique to find the best fit to measure the DNL and INL error.

Output result to base workspace — Store detailed test results to base workspace

off (default) | on

Store detailed test results to a `struct` in the base workspace for further processing. By default, this option is not selected.

Workspace variable name — Name of the variable that stores detailed test results

`adc_dc_out` (default) | character string

Name of the variable that stores detailed test results, specified as a character string.

Dependencies

This parameter is only available when **Output result to base workspace** is selected

Programmatic Use

- Use `get_param(gcb, 'VariableName')` to view the current value of **Workspace variable name**.

- Use `set_param(gcb, 'VariableName', value)` to set **Workspace variable name** to a specific value.

Plot — Plot measurement results

button

Click to plot measurement result for further analysis.

More About

Offset Error

Offset error represents the offset of the ADC transfer function curve from its ideal value at a single point.

Gain Error

Gain error represents the deviation of the slope of the ADC transfer function curve from its ideal value.

INL Error

Integral nonlinearity (INL) error, also termed as relative accuracy, is the maximum deviation of the measured transfer function from a straight line. The straight line can be a best fit using standard curve fitting technique, or drawn between the end points of the actual transfer function after gain adjustment.

The best fit method gives a better prediction of distortion in AC applications, and a lower value of linearity error. The endpoint method is mostly used in measurement application of data converters, since the error budget depends on the actual deviation from ideal transfer function.

DNL Error

Differential nonlinearity (DNL) is the deviation from the ideal difference (1 LSB) between analog input levels that trigger any two successive digital output levels. The DNL error is the maximum value of DNL found at any transition.

Version History

Introduced in R2019a

See Also

SAR ADC | Flash ADC | ADC Testbench | ADC AC Measurement

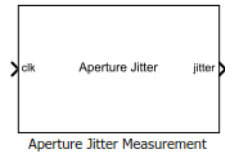
Topics

“Measuring Offset and Gain Errors in ADC”

Aperture Jitter Measurement

Measure aperture jitter of periodic signals

Library: Mixed-Signal Blockset / ADC / Measurements & Testbenches



Description

The Aperture Jitter Measurement measure the aperture jitter of periodic signals. In practical data converters, there is a delay between the sampling edge of the sample clock signal and when the sample is actually taken. This delay is known as aperture delay. Aperture jitter is the sample to sample variation between aperture delay.

Ports

Input

clk — Clock signal input

scalar

Clock signal input, specified as a scalar in volts.

Data Types: double

Output

jitter — Jitter output

scalar

Aperture jitter output, returned as a scalar in seconds.

Data Types: double

Parameters

Frequency — Input clock frequency

1e6 (default) | real positive scalar

Input clock frequency, specified as a real positive scalar in Hz.

Programmatic Use

- Use `get_param(gcb, 'Frequency')` to view the current value of **Frequency**.
- Use `set_param(gcb, 'Frequency', value)` to set **Frequency** to a specific value.

Recommended min. simulation stop time — Minimum time the simulation must run for meaningful result

9.901e-05 (default) | scalar

Minimum time the simulation must run for meaningful result, specified as a scalar in seconds. This is calculated using the JEDEC Standards for measuring aperture jitter.

Signal range

Min — Minimum value of clock signal

0 (default) | real scalar

Minimum value of clock signal, specified as a real scalar in volts.

Programmatic Use

- Use `get_param(gcb, 'InputMin')` to view the current value of **Min**.
- Use `set_param(gcb, 'InputMin', value)` to set **Min** to a specific value.

Max — Maximum value of clock signal

1 (default) | real scalar

Maximum value of clock signal, specified as a real scalar in volts.

Programmatic Use

- Use `get_param(gcb, 'InputMax')` to view the current value of **Max**.
- Use `set_param(gcb, 'InputMax', value)` to set **Max** to a specific value.

Hold off time (s) — Delay before measurement analysis

0 (default) | nonnegative real scalar

Delays measurement analysis to avoid corruption by transients, specified as a nonnegative real scalar in seconds.

Programmatic Use

- Use `get_param(gcb, 'HoldOffTime')` to view the current value of **Hold off time (s)**.
- Use `set_param(gcb, 'HoldOffTime', value)` to set **Hold off time (s)** to a specific value.

Data Types: double

Version History

Introduced in R2019a

See Also

Flash ADC | SAR ADC

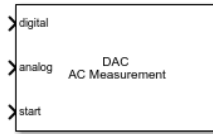
External Websites

<https://www.jedec.org/>

DAC AC Measurement

Measure AC performance metrics of DAC output

Library: Mixed-Signal Blockset / DAC / Measurements & Testbenches



Description

The DAC AC Measurement block measures DAC AC performance metrics such as signal-to-noise ratio (SNR), signal to noise and distortion ratio (SINAD), spurious-free dynamic range (SFDR), effective number of bits (ENOB), and noise floor. You can use DAC AC Measurement block to validate the DAC architecture models provided in Mixed-Signal Blockset, or you can use a DAC of your own implementation

Ports

Input

digital — Digital input signal from DAC

scalar

Digital input signal from a DAC, specified as a scalar.

Data Types: `fixed_point` | `single` | `double` | `int8` | `int16` | `int32` | `uint8` | `uint16` | `uint32`

analog — Converted analog signal from DAC

scalar

Converted analog signal from a DAC, specified as a scalar.

Data Types: `double`

start — External clock to start conversion

scalar

External clock to start conversion, specified as a scalar. This port determines when digital-to-analog conversion process starts.

Data Types: `double`

Parameters

Digital signal frequency (Hz) — Frequency of digital input signal to DAC

1e3 (default) | positive real scalar

Frequency of the digital input signal to the DAC block, specified as a positive real scalar in hertz.

Digital signal frequency (Hz) must match the input frequency of the DAC device under test.

Digital input frequency (Hz) needs to satisfy two requirements:

- All the output codes of the DAC must be activated.
- The **Digital signal frequency (Hz)** must not share any common multiples other than 1 with the **Conversion start frequency (Hz)**.

Programmatic Use

Block parameter: InputFrequency

Type: character vector

Values: positive real scalar

Default: 1e3

Data Types: double

Start conversion frequency (Hz) — Frequency of internal start conversion clock

1e6 (default) | positive real scalar

Frequency of the internal start conversion clock, specified as a real scalar in Hz. The **Start conversion frequency** parameter determines the conversion rate at the start of conversion.

Programmatic Use

Block parameter: StartFreq

Type: character vector

Values: positive real scalar

Default: 1e6

Data Types: double

Settling time tolerance (LSB) — Tolerance for calculating settling time

0.5 (default) | positive real scalar

The tolerance allowed for calculating settling time, specified as a positive real scalar in LSB. The output of the DAC must settle within the **Settling time tolerance (LSB)** by **Settling time (s)**.

Programmatic Use

Block parameter: SettlingTimeTolerance

Type: character vector

Values: positive real scalar

Default: 0.5

Data Types: double

Hold off time (s) — Delay before measurement analysis to avoid corruption by transients

0 (default) | nonnegative real scalar

Delay before measurement analysis to avoid corruption by transients, specified as a nonnegative real scalar in seconds.

Programmatic Use

Block parameter: HoldOffTime

Type: character vector

Values: nonnegative real scalar

Default: 0

Recommended simulation stop time (s) — Minimum time simulation must run for meaningful result

0.09 (default) | positive real scalar

Minimum time the simulation must run to obtain meaningful results, specified as a positive real scalar in seconds.

To measure AC performance, the simulation must run so that the DAC can generate six spectral updates of the DAC output. So, the **Recommended simulation stop time (s)** T is given by [1]:

$$T = 6 \left(\frac{1.5}{RBW} + \text{Hold off time} \right),$$

where RBW is the resolution bandwidth of the spectrum estimator inside the DAC Testbench block and is given by the equation: $RBW = [\min(\text{Input frequency})0.1]$.

This parameter is only reported by the block and is not editable.

Data Types: double

Output result to base workspace — Store detailed test results to base workspace

off (default) | on

Store detailed test results to a struct in the base workspace for further processing at the end of simulation. By default, this parameter is deselected.

Workspace variable name — Name of the variable that stores detailed test results

dac_ac_out (default) | character string

Name of the variable that stores detailed test results, specified as a character string.

Dependencies

To enable this parameter, select **Output result to base workspace** parameter.

Programmatic Use

Block parameter: VariableName

Type: character vector

Values: character string

Default: dac_ac_out

Show spectrum analyzer during simulation — Displays Spectrum Analyzer during simulation

on (default) | off

Select this parameter to display the Spectrum Analyzer window during simulation. By default, this parameter is selected.

More About

SNR

Signal-to-noise ratio or SNR is the ratio of the RMS (root-mean-square) signal amplitude to the mean value of the root-sum-squares (RSS) of all other spectral components, excluding the DC and first five harmonics.

SINAD

Signal to noise and distortion ratio or SINAD is the ratio of the RMS signal amplitude to the mean value of the root-sum-squares of all other spectral components and harmonics, excluding DC.

SFDR

Spurious free dynamic range or SFDR is the ratio of the RMS signal amplitude to the RMS value of the peak spurious content, measured over the entire first Nyquist zone (DC to half of sampling frequency).

ENOB

Effective number of bits or ENOB represents the actual resolution of a DAC after considering internal noise and errors. It is given by $ENOB = \frac{SINAD - 1.76}{6.02}$.

Version History

Introduced in R2020a

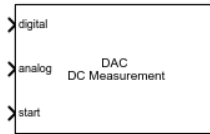
See Also

DAC DC measurement | Binary Weighted DAC | DAC Testbench

DAC DC Measurement

Measure DC performance metrics of DAC output

Library: Mixed-Signal Blockset / DAC / Measurements & Testbenches



Description

The DAC DC Measurement block measures DAC DC performance metrics such as offset error, gain error, integral nonlinearity (INL), and differential nonlinearity (DNL) errors. You can use the DAC DC Measurement block to validate the DAC architecture models provided in Mixed-Signal Blockset, or you can use a DAC of your own implementation.

Ports

Input

digital — Digital input signal from DAC

scalar

Digital signal from a DAC, specified as a scalar.

Data Types: `fixed point` | `single` | `double` | `int8` | `int16` | `int32` | `uint8` | `uint16` | `uint32`

analog — Converted analog signal from DAC

scalar

Converted analog signal from a DAC, specified as a scalar.

Data Types: `double`

start — External clock to start conversion

scalar

External clock to start conversion, specified as a scalar. This port determines when digital-to-analog conversion process starts.

Data Types: `double`

Parameters

Input polarity — Polarity of input signal to DAC

`Bipolar` (default) | `Unipolar`

Polarity of the input signal to the DAC.

Programmatic Use

Block parameter: Polarity

Type: character vector
Values: Bipolar|Unipolar
Default: Bipolar

Reference (V) — Reference voltage

2 (default) | real scalar

Reference voltage of the DAC, specified as a real scalar in volts. **Reference (V)** helps determine the output from the input digital code, **Number of bits**, and **Bias (V)** using the equation:

$$\text{DAC output} = \left(\left(\frac{\text{Digital input code}}{2^{\text{Number of bits}}} \right) \text{Reference} \right) + \text{Bias}.$$

Programmatic Use

Block parameter: Ref
Type: character vector
Values: real scalar
Default: 2

Data Types: double

Bias (V) — Bias voltage added to output

0 (default) | real scalar

Bias voltage added to the output of the DAC, specified as a real scalar in volts. **Bias (V)** helps determine the output from the input digital code, **Number of bits**, and **Reference (V)** using the equation:

$$\text{DAC output} = \left(\left(\frac{\text{Digital input code}}{2^{\text{Number of bits}}} \right) \text{Reference} \right) + \text{Bias}.$$

Programmatic Use

Block parameter: Bias
Type: character vector
Values: real scalar
Default: 0

Data Types: double

Settling time (s) — Time required for output to settle

3e-7 (default) | nonnegative real scalar

The time required for the output of the DAC to settle to within some fraction of its final value, specified as a nonnegative real scalar in seconds.

Programmatic Use

Block parameter: SettlingTime
Type: character vector
Values: real scalar
Default: 3e-7

Data Types: double

Hold off time (s) — Delay before measurement analysis

1e-3 (default) | nonnegative real scalar

Delay before measurement analysis to avoid corruption by transients, specified as a nonnegative real scalar in seconds.

Programmatic Use

Block parameter: HoldOffTime

Type: character vector

Values: nonnegative real scalar

Default: 1e-3

Data Types: double

Number of bits – Number of bits in input word

10 (default) | positive real integer

Number of bits in the input word, specified as a unitless positive real integer. **Number of bits** determines the resolution of the DAC.

Programmatic Use

Block parameter: NBits

Type: character vector

Values: positive real integer

Default: 10

Data Types: double

Start conversion frequency (Hz) – Frequency of internal start conversion clock

1e6 (default) | positive real scalar

Frequency of the internal start conversion clock, specified as a real scalar in Hz. The **Start conversion frequency** parameter determines the conversion rate at the start of conversion.

Programmatic Use

Block parameter: StartFreq

Type: character vector

Values: positive real scalar

Default: 1e6

Data Types: double

Recommended simulation stop time (s) – Minimum time simulation must run for meaningful result

0.02148 (default) | positive real scalar

Minimum time the simulation must run to obtain meaningful results, specified as a positive real scalar in seconds.

To measure DC performance, the simulation must run so that the DAC can sample each digital code 20 times. Based on this assumption, the **Recommended simulation stop time (s)** T is given by:

$$T = \frac{\text{Samples per bit}}{\left(\text{StartFreq}/2^{N\text{bits} + 1}\right)} + \text{Hold off time},$$

where StartFreq is the frequency of the conversion start clock and $N\text{bits}$ is the resolution of the DAC.

The number of samples per bit is calculated using the equation:

$$\text{Samples per bit} = \max\left(\frac{1}{\text{Error tolerance}}, 10\right).$$

This parameter is only reported by the block and is not editable.

Data Types: `double`

Endpoint — Measure DNL, INL using endpoint method

`on` (default) | `off`

Measure the differential nonlinearity (DNL) error and integral nonlinearity (INL) error using the endpoint method. This method uses the endpoints of the actual transfer function to measure the DNL and INL errors.

Best fit — Measure DNL, INL using best fit method

`on` (default) | `off`

Measure the differential nonlinearity (DNL) error and integral nonlinearity (INL) error using the best fit method. This method uses a standard curve-fitting technique to find the best fit to measure the DNL and INL errors.

Output result to base workspace — Store detailed test results to base workspace

`off` (default) | `on`

Select to store detailed test results to a `struct` in the base workspace for further processing at the end of simulation. By default, this parameter is deselected.

Workspace variable name — Name of the variable that stores detailed test results

`dac_dc_out` (default) | character string

Name of the variable that stores detailed test results, specified as a character string.

Dependencies

To enable this parameter, select **Output result to base workspace** parameter.

Programmatic Use

Block parameter: `VariableName`

Type: character vector

Values: character string

Default: `dac_dc_out`

Plot — Plot measurement results

`button`

Click to plot measurement result for further analysis.

More About

Offset Error

Offset error represents the offset of the DAC transfer function curve from its ideal value at a single point.

Gain Error

Gain error represents the deviation of the slope of the DAC transfer function curve from its ideal value.

INL Error

Integral nonlinearity (INL) error, also termed as relative accuracy, is the maximum deviation of the measured transfer function from a straight line. The straight line can either be a best fit using standard-curve fitting technique, or be drawn between the endpoints of the actual transfer function after gain adjustment.

The best fit method gives a better prediction of distortion in AC applications, and a lower value of linearity error. The endpoint method is mostly used in the measurement applications of data converters, since the error budget depends on actual deviation from the ideal transfer function.

DNL Error

Differential nonlinearity (DNL) is the deviation from the ideal difference (1 LSB) between analog input levels that trigger any two successive digital output levels. The DNL error is the maximum value of DNL found at any transition.

Version History

Introduced in R2020a

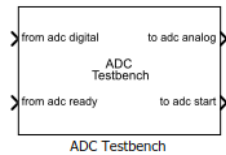
See Also

Binary Weighted DAC | DAC AC measurement | DAC Testbench

ADC Testbench

Measures DC and AC performance metrics of ADC output

Library: Mixed-Signal Blockset / ADC / Measurements & Testbenches



Description

The ADC Testbench block measures both DC and AC performance metrics. DC performance metrics include offset error and gain error. AC performance metrics include signal to noise ratio (SNR), signal to noise and distortion ratio (SINAD), spurious free dynamic range (SFDR), effective number of bits (ENOB), noise floor, and conversion delay.

The ADC Testbench block generates the stimulus to drive the device under test (DUT) from the **Stimulus** tab. The setup parameters for validating the DUT are defined in the **Setup** tab and the target validation metrics are defined in the **Target Metric** tab.

You can use the ADC Testbench block to validate the ADC architectural models provided in Mixed-Signal Blockset, or you can validate an ADC of your own implementation.

Ports

Input

from adc digital – Digital signal from ADC output

scalar

Digital input signal from the ADC output, specified as a scalar.

Data Types: fixed point | single | double | int8 | int16 | int32 | uint8 | uint16 | uint32 | Boolean

from adc ready – Conversion ready signal from ADC output

scalar

Conversion ready signal from the ADC output, specified as a scalar. This signal indicates when the ADC conversion process is complete.

Data Types: double

Output

to adc analog – Analog stimulus signal for ADC input

scalar

Analog output stimulus signal for the ADC input, returned as a scalar.

Data Types: double

to_adc_start — External start conversion clock for ADC

scalar

External start conversion clock for ADC, returned as a scalar. The rising edge of this signal starts the conversion process in ADC block.

Data Types: double

Parameters**Measurement — Select whether to measure DC or AC performance metrics**

DC (default) | AC

Select whether to measure static (DC) or dynamic (AC) performance metrics.

- Select DC to measure offset error and gain error.
- Select AC to measure SNR, SINAD, SFDR, ENOB, noise floor, and conversion delay.

Recommended min. simulation stop time (s) — Minimum time simulation must run for meaningful result

2.048e-02 (default) | positive real scalar

Minimum time for which the simulation must run to obtain meaningful results, specified as a positive real scalar in seconds.

- To measure DC performance, the simulation must run so that ADC can sample each digital code 10 times, assuming a ramp input that traverses the full scale range of the ADC over the period of simulation. Based on this assumption and considering that the maximum allowed error tolerance is 0.1, the analog input frequency (f_{analog}), generated by the ADC Testbench block for the sawtooth waveform is set as:

$$f_{\text{analog}} = \frac{\text{StartFreq} \cdot \min(\text{ErrorTolerance}, 0.1)}{2^{(N\text{bits} + 1)}}$$

where *StartFreq* is the frequency of the conversion start clock and *Nbits* is the resolution of the ADC.

So, the **Recommended min. simulation stop time (s)** *T* is calculated by using the formula:

$$T = \frac{1}{f_{\text{analog}}} + \text{HoldOffTime}.$$

- To measure AC performance, the simulation must run so that the ADC can generate 6 spectral updates of the ADC output. The time to generate one spectral output based on Welch's method [1] on page 1-38 is given by:

$$t = \frac{1.5 \cdot \text{SamplingFrequency}}{\text{RBW}}$$

where *SamplingFrequency* and *RBW* are the sampling frequency and resolution bandwidth of the spectrum estimator inside the ADC Testbench block.

This parameter is only reported by the testbench and is not editable.

Data Types: double

Set as model stop time — Automatically set recommended min. simulation stop time as model stop time

button

Click to automatically set the **Recommended min. simulation stop time (s)** as the stop time of the Simulink model.

Endpoint — Measure DNL, INL using endpoint method

on (default) | off

Measure the differential nonlinearity (DNL) error and integral nonlinearity (INL) error using the endpoint method. This method uses the end points of the actual transfer function to measure the DNL and INL error.

Best fit — Measure DNL, INL using best fit method

on (default) | off

Measure the differential nonlinearity (DNL) error and integral nonlinearity (INL) error using the best fit method. This method uses a standard curve fitting technique to find the best fit to measure the DNL and INL error.

Plot DC analysis result — Plot DC analysis results

button

Click to plot DC analysis result for further analysis. To perform a complete DC analysis including integral nonlinearity (INL) and differential nonlinearity (DNL), use the ADC DC Measurement block.

Dependencies

This parameter is only available when **Measurement** option is set to DC.

Export measurement result — Store detailed test results to base workspace

button

Click to store detailed test results to a spreadsheet (XLS file) or as comma-separated values (CSV file) for further processing.

Stimulus**Distortion measurement type — Type of distortion to measure**

Harmonic (default) | Intermodulation

Type of distortion the ADC Testbench block is set to measure, specified as Harmonic or Intermodulation.

Dependencies

To enable this parameter, set **Measurement** option as AC.

Programmatic Use**Block parameter:** DistortionMeasurement**Type:** character vector**Values:** Harmonic | Intermodulation**Default:** Harmonic**Analog stimulus frequency (Hz) — Frequency of the analog input signal to ADC**

positive real scalar | positive real valued vector

Frequency of the analog input signal to an ADC block, specified as a positive real scalar in hertz.

Analog stimulus frequency must match the input frequency to the ADC device under test. By default, this parameter is calculated automatically. You can deselect **Set automatically** to customize the value.

- When the ADC Testbench is set to measure the Harmonic distortion, the default value of **Analog stimulus frequency** is 976.563.
- When the ADC Testbench is set to measure the Intermodulation distortion, the default value of **Analog stimulus frequency** is [878.907, 1074.22].

Analog stimulus frequency needs to satisfy two requirements:

- All the output codes of the ADC must be activated.
- The **Analog stimulus frequency** must not share any common multiples other than 1 with the **Start conversion frequency**.

To satisfy both the conditions, use the equation $f_{\text{analog}} = \frac{J}{M}f_{\text{start}}$ [2],

where:

f_{analog} is the analog signal frequency,

f_{start} is the start conversion frequency,

$M > 2^{N_{\text{bits}}} \cdot \pi$, where N_{bits} is the number of bits of the ADC,

and J is an integer with no common factors with M .

Dependencies

To enable this parameter, set **Measurement** option as AC.

Programmatic Use

Block parameter: InputFrequency

Type: character vector

Values: positive real scalar | positive real valued vector

Default: 976.563

Resolution bandwidth (Hz) — Resolution bandwidth

positive real scalar

Resolution bandwidth, specified as a positive real scalar in hertz. This parameter defines the smallest positive frequency that can be resolved. By default, this parameter is calculated automatically. You can deselect **Set automatically** to customize the value.

- When the ADC AC Measurement is set to measure the Harmonic distortion, the default value of **Resolution bandwidth (Hz)** is 97.6563.
- When the ADC AC Measurement is set to measure the Intermodulation distortion, the default value of **Resolution bandwidth (Hz)** is 87.8907.

Dependencies

To enable this parameter, set **Measurement** option as AC.

Programmatic Use

Block parameter: RBW

Type: character vector

Values: positive real scalar

Default: 97.6563

Start conversion frequency (Hz) — Frequency of the start conversion clock of the ADC
1e6 (default) | positive real scalar

Frequency of the start conversion clock of the ADC, specified as a positive real scalar in Hz. **Start conversion frequency (Hz)** must match the frequency of the start conversion clock of the ADC block.

Programmatic Use

Block parameter: StartFreq

Type: character vector

Values: positive real scalar

Default: 1e6

Data Types: double

RMS aperture jitter (s) — RMS aperture jitter to be added by the start conversion clock
40e-12 (default) | positive real scalar

RMS aperture jitter to be added by the start conversion clock, specified as a positive real scalar in seconds.

Programmatic Use

Block parameter: RMSJitt

Type: character vector

Values: positive real scalar

Default: 40e-12

Data Types: double

Error tolerance (LSB) — Maximum difference between successive samples of analog signal

0.1 (default) | positive scalar in the range (0, 1]

Maximum allowed difference in the amplitude of the successive samples of the analog input signal, specified as positive real scalar in least significant bit (LSB).

Dependencies

To enable this parameter, set **Measurement** option as DC.

Data Types: double

Setup

Autofill setup parameter — Automatically propagate setup parameters from ADC
button

Click to automatically propagate setup parameters from the ADC.

Dependencies

This parameter only works when the ADC is a Flash ADC or a SAR ADC from the Mixed-Signal Blockset.

Number of bits — Number of physical output bits

10 (default) | positive real integer in the range [1, 26]

Number of physical output bits, specified as a unitless positive real integer in the range [1, 26]. **Number of bits** determines the resolution of the ADC.

Programmatic Use

- Use `get_param(gcb, 'NBits')` to view the current **Number of bits**.
- Use `set_param(gcb, 'NBits', value)` to set **Number of bits** to a specific value.

Data Types: double

Input range (V) — ADC dynamic range

[-1 1] (default) | 2-element row vector

ADC dynamic range, specified as a 2-element row vector in volts.

Programmatic Use

- Use `get_param(gcb, 'InputRange')` to view the current **Input range (V)**.
- Use `set_param(gcb, 'InputRange', value)` to set **Input range (V)** to a specific value.

Data Types: double

Hold off time (s) — Delay before measurement analysis

0 (default) | nonnegative real scalar

Delays measurement analysis to avoid corruption by transients, specified as a nonnegative real scalar in seconds.

Programmatic Use

Block parameter: HoldOffTime

Type: character vector

Values: nonnegative real scalar

Default: 0

Data Types: double

Show spectrum analyzer during simulation — Displays spectrum analyzer during simulation

off (default) | on

Displays spectrum analyzer during simulation. By default, this option is deselected.

Dependencies

This parameter is only available when **Measurement** option is set to AC.

Enable increased buffer size — Enable increased buffer size

off (default) | on

Select to enable increased buffer size during simulation. By default, this option is deselected.

Buffer size – Number of samples of the input buffering available during simulation

5 (default) | positive integer scalar

Number of samples of the input buffering available during simulation, specified as a positive integer scalar.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size** to a large enough value that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when **Enable increased buffer size** option is selected in the **Configuration** tab.

Programmatic Use**Block parameter:** NBuffer**Type:** character vector**Values:** positive integer scalar**Default:** 5

Data Types: double

Target Metric**Autofill target metric – Automatically propagate target metrics from ADC**

button

Click to automatically propagate target metrics from ADC.

Dependencies

- To enable this parameter, set **Measurement** option to DC.
- This parameter only works when the ADC is a Flash ADC or a SAR ADC from the Mixed-Signal Blockset.

Offset error – Shifts quantization steps by specific value

1.5 LSB (default) | real scalar

Shifts quantization steps by specific value, specified as a positive real scalar in %FS, FS, or LSB.

Note The full scale range of the converter is defined as the difference between the last and first code on the +0.5 LSB compensated transfer curve. In a +0.5 LSB compensated transfer curve, first code is 0.5 LSB wide while the last code is 1.5 LSB wide. The input values must be considered within the full scale range of the converter.

Note LSB is calculated by the equation $LSB = \frac{\text{Full scale range of converter}}{2^{Nbits}}$.

Dependencies

To enable this parameter, set **Measurement** option to DC.

Programmatic Use**Block parameter:** TargetOffsetError**Type:** character vector**Values:** real scalar**Default:** 1.5 LSB

Data Types: double

Gain error – Error on the slope of ADC transfer curve

1 LSB (default) | real scalar

Error on the slope of the straight line interpolating ADC transfer curve, specified as a positive real scalar in least significant bit %FS, FS, or LSB.

Note The full scale range of the converter is defined as the difference between the last and first code on the +0.5 LSB compensated transfer curve. In a +0.5 LSB compensated transfer curve, first code is 0.5 LSB wide while the last code is 1.5 LSB wide. The input values must be considered within the full scale range of the converter.

Note LSB is calculated by the equation $LSB = \frac{\text{Full scale range of converter}}{2^{Nbits}}$.

Dependencies

To enable this parameter, set **Measurement** option to DC.

Programmatic Use**Block parameter:** TargetGainError**Type:** character vector**Values:** real scalar**Default:** 1 LSB

Data Types: double

Version History**Introduced in R2019a****References**

[1] Spectrum Analyzer

[2] IEEE Std 1241-2010. "IEEE Standard for Terminology and Test Methods for Analog-to-Digital Converters," pp. 29-30, 14 January 2011.

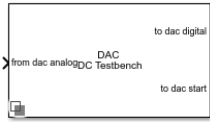
See Also

ADC DC Measurement | ADC AC Measurement | Flash ADC | SAR ADC

DAC Testbench

Measure DC and AC performance metrics of DAC output

Library: Mixed-Signal Blockset / DAC / Measurements & Testbenches



Description

The DAC Testbench block measures both DC and AC performance metrics of a DAC (digital to analog converter). DC performance metrics include offset error and gain error. AC performance metrics include signal-to-noise ratio (SNR), signal to noise and distortion ratio (SINAD), spurious-free dynamic range (SFDR), effective number of bits (ENOB), and noise floor.

The DAC Testbench block generates the stimulus to drive the device under test (DUT) from the **Stimulus** tab. The setup parameters for validating the DUT are defined on the **Setup** tab. The target validation metrics are defined on the **Target Metric** tab.

You can use the DAC Testbench block to validate the DAC architecture models provided in Mixed-Signal Blockset, or you can validate a DAC of your own implementation.

Ports

Input

from dac analog — Analog signal from DAC output

scalar

Analog input signal from the DAC output, specified as a scalar.

Data Types: double

Output

to dac digital — Digital stimulus signal for DAC input

scalar

Digital output stimulus signal for the DAC input, returned as a scalar.

Data Types: single | double | int8 | int16 | int32 | uint8 | uint16 | uint32 | fixed point

to dac start — External start conversion clock for DAC

scalar

External start conversion clock for DAC, returned as a scalar. The rising edge of this signal starts the conversion process in the DAC block.

Data Types: double

Parameters

Measurement — Select whether to measure DC or AC performance metrics

DC (default) | AC

Select whether to measure static (DC) or dynamic (AC) performance metrics:

- Select DC to measure offset error and gain error.
- Select AC to measure SNR, SINAD, SFDR, ENOB, and noise floor.

Recommended min. simulation stop time (s) — Minimum time simulation must run for meaningful result

2.048e-02 (default) | positive real scalar

Minimum time for which the simulation must run to obtain meaningful results, specified as a positive real scalar in seconds.

- To measure DC performance, the simulation must run so that the DAC can sample each digital code 20 times. Based on this assumption, the **Recommended min. simulation stop time (s)** T is given by:

$$T = \frac{\text{Samples per bit}}{\left(\text{StartFreq}/2^{Nbits+1}\right)} + \text{Hold off time},$$

where StartFreq is the frequency of the conversion-start clock and $Nbits$ is the resolution of the DAC.

The number of samples per bit is calculated using the equation:

$$\text{Samples per bit} = \max\left(\frac{1}{\text{Error tolerance}}, 10\right).$$

- To measure AC performance, the simulation must run so that the DAC can generate six spectral updates of the DAC output. So, the **Recommended min. simulation stop time (s)** T is given by [1]:

$$T = 6\left(\frac{1.5}{RBW} + \text{Hold off time}\right),$$

where RBW is the resolution bandwidth of the spectrum estimator inside the DAC Testbench block and is given by the equation: $RBW = [\min(\text{Input frequency})0.1]$.

This parameter is only reported by the testbench and is not editable.

Data Types: double

Set as model stop time — Automatically set recommended min. simulation stop time as model stop time

button

Click to automatically set the **Recommended min. simulation stop time (s)** as the stop time of the Simulink model.

Endpoint — Measure DNL, INL using endpoint method

on (default) | off

Measure the differential nonlinearity (DNL) error and integral nonlinearity (INL) error using the endpoint method. This method uses the endpoints of the actual transfer function to measure the DNL and INL errors.

Dependencies

To enable this parameter, set **Measurement** to DC.

Best fit – Measure DNL, INL using best fit method

on (default) | off

Measure the differential nonlinearity (DNL) error and integral nonlinearity (INL) error using the best fit method. This method uses a standard curve-fitting technique to find the best fit to measure the DNL and INL errors.

Dependencies

To enable this parameter, set **Measurement** to DC.

Plot DC analysis result – Plot DC analysis result

button

Click to plot DC analysis result for further analysis. To perform a complete DC analysis including integral nonlinearity (INL) and differential nonlinearity (DNL), use the DAC DC Measurement block.

Dependencies

To enable this parameter, set **Measurement** to DC.

Export measurement result – Store detailed test results to base workspace

button

Click to store detailed test results to a spreadsheet (XLS file) or as comma-separated values (CSV file) for further processing.

Stimulus**Digital input frequency (Hz) – Frequency of digital input signal to DAC**

1e4 (default) | positive real scalar

Frequency of the digital input signal to the DAC block, specified as a positive real scalar in hertz.

Digital input frequency (Hz) must match the input frequency of the DAC device under test.

Digital input frequency (Hz) needs to satisfy two requirements:

- All the output codes of the DAC must be activated.
- The **Digital input frequency (Hz)** must not share any common multiples other than 1 with the **Start conversion frequency (Hz)**.

Dependencies

To enable this parameter, set **Measurement** to AC.

Programmatic Use

Block parameter: InputFrequency

Type: character vector

Values: positive real scalar

Default: 1e3

Data Types: double

Start conversion frequency (Hz) — Frequency of internal start-conversion clock

1e6 (default) | positive real scalar

Frequency of internal start-conversion clock, specified as a positive real scalar in Hz. **Start conversion frequency (Hz)** determines the rate of the DAC.

Programmatic Use

Block parameter: StartFreq

Type: character vector

Values: positive real scalar

Default: 1e6

Data Types: double

Error tolerance (LSB) — Maximum difference between successive samples of digital signal

0.1 (default) | positive scalar in the range (0, 1]

Maximum allowed difference in the amplitude of successive samples of the digital input signal, specified as a positive real scalar in least significant bit (LSB).

Dependencies

To enable this parameter, set **Measurement** to DC.

Programmatic Use

Block parameter: ErrorTolerance

Type: character vector

Values: positive scalar in the range (0, 1]

Default: 0.1

Data Types: double

Setup

Autofill setup parameters — Automatically propagate setup parameters from DAC button

Click to automatically propagate setup parameters from the DAC.

Dependencies

The DAC must be a Binary Weighted DAC from the Mixed-Signal Blockset.

Number of bits — Number of bits in input word

10 (default) | positive real integer

Number of bits in the input word, specified as a unitless positive real integer. **Number of bits** determines the resolution of the DAC.

Programmatic Use

Block parameter: NBits

Type: character vector

Values: positive real integer

Default: 10

Data Types: double

Input polarity — Polarity of input signal to DAC

Bipolar (default) | Unipolar

Polarity of the input signal to the DAC.

Programmatic Use

Block parameter: Polarity

Type: character vector

Values: Bipolar|Unipolar

Default: Bipolar

Reference (V) — Reference voltage

1 (default) | real scalar

Reference voltage of the DAC, specified as a real scalar in volts. **Reference (V)** helps determine the output from the input digital code, **Number of bits**, and **Bias (V)** using the equation:

$$\text{DAC output} = \left(\left(\frac{\text{Digital input code}}{2^{\text{Number of bits}}} \right) \text{Reference} \right) + \text{Bias}.$$

Dependencies

To enable this parameter, set **Measurement** to DC.

Programmatic Use

Block parameter: Ref

Type: character vector

Values: real scalar

Default: 1

Data Types: double

Bias (V) — Bias voltage added to output

0 (default) | real scalar

Bias voltage added to the output of the DAC, specified as a real scalar in volts. **Bias (V)** helps determine the output from the input digital code, **Number of bits**, and **Reference (V)** using the equation:

$$\text{DAC output} = \left(\left(\frac{\text{Digital input code}}{2^{\text{Number of bits}}} \right) \text{Reference} \right) + \text{Bias}$$

Dependencies

To enable this parameter, set **Measurement** to DC.

Programmatic Use

Block parameter: Bias

Type: character vector

Values: real scalar

Default: 0

Data Types: double

Settling time (s) — Time required for output to settle

0.25/1e-6 (default) | nonnegative real scalar

The time required for the output of the DAC to settle to within some fraction of its final value, specified as a nonnegative real scalar in seconds.

Dependencies

To enable this parameter, set **Measurement** to DC.

Programmatic Use

Block parameter: SettlingTime

Type: character vector

Values: real scalar

Default: 0.25/1e-6

Data Types: double

Settling time tolerance (LSB) — Tolerance for calculating settling time

0.5 (default) | positive real scalar

The tolerance allowed for calculating settling time, specified as a positive real scalar in LSB. The output of the DAC must settle within the **Settling time tolerance (LSB)** by **Settling time (s)**.

Dependencies

To enable this parameter, set **Measurement** to AC.

Programmatic Use

Block parameter: SettlingTimeTolerance

Type: character vector

Values: positive real scalar

Default: 0.5

Data Types: double

Hold off time (s) — Delay before measurement analysis

0 (default) | nonnegative real scalar

Delay before measurement analysis to avoid corruption by transients, specified as a nonnegative real scalar in seconds.

Programmatic Use

- Use `get_param(gcb, 'HoldOffTime')` to view the current value of **Hold off time (s)**.
- Use `set_param(gcb, 'HoldOffTime', value)` to set **Hold off time (s)** to a specific value.

Data Types: double

Show spectrum analyzer during simulation — Displays Spectrum Analyzer during simulation

off (default) | on

Select this parameter to display the Spectrum Analyzer window during simulation. By default, this parameter is deselected.

Dependencies

To enable this parameter, set **Measurement** to AC.

Target Metric

Autofill target metric – Automatically propagate target metrics from DAC
button

Click to automatically propagate target metrics from the DAC.

Dependencies

- To enable this parameter, set **Measurement** to DC.
- The DAC must be a Binary Weighted DAC from the Mixed-Signal Blockset.

Offset error – Shifts quantization steps by specific value

0 LSB (default) | real scalar

Shifts quantization steps by a specific value, specified as a real scalar in %FS (percentage of full scale), FS (full scale), or LSB (least significant bit).

Note The full scale range of the converter is defined as the difference between the last and first code on the +0.5 LSB compensated transfer curve. In a +0.5 LSB compensated transfer curve, first code is 0.5 LSB wide while the last code is 1.5 LSB wide. The input values must be considered within the full scale range of the converter.

Note LSB is calculated by the equation $LSB = \frac{\text{Full scale range of converter}}{2^{Nbits}}$.

Dependencies

To enable this parameter, set **Measurement** to DC.

Programmatic Use

Block parameter: TargetOffsetError

Type: character vector

Values: real scalar

Default: 0 LSB

Data Types: double

Gain error – Error in slope of DAC transfer curve

0 LSB (default) | real scalar

Error in the slope of the straight line interpolating the DAC transfer curve, specified as a real scalar in %FS (percentage of full scale), FS (full scale), or LSB (least significant bit).

Note The full scale range of the converter is defined as the difference between the last and first code on the +0.5 LSB compensated transfer curve. In a +0.5 LSB compensated transfer curve, first code is 0.5 LSB wide while the last code is 1.5 LSB wide. The input values must be considered within the full scale range of the converter.

Note LSB is calculated by the equation $LSB = \frac{\text{Full scale range of converter}}{2^{Nbits}}$.

Dependencies

To enable this parameter, set **Measurement** to DC.

Programmatic Use

Programmatic Use

Block parameter: TargetGainError

Type: character vector

Values: real scalar

Default: 0 LSB

Data Types: double

Version History

Introduced in R2020a

References

[1] Spectrum Analyzer

See Also

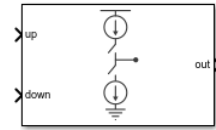
Binary Weighted DAC | DAC AC measurement | DAC DC measurement

Blocks: PLL

Charge Pump

Output a current proportional to the difference in duty cycle between two input ports

Library: Mixed-Signal Blockset / PLL / Building Blocks



Description

The Charge Pump block produces an output current which is proportional to the difference in duty cycles between the signals at its **up** and **down** input ports. In a phase-locked loop (PLL) system, the Charge Pump block converts the phase error as represented by the two outputs of the PFD block into a single current at the input to the Loop Filter.

Ports

Input

up — Input port

scalar

Input port, connected to the **up** output of a PFD in a PLL system.

Data Types: double

down — Input port

scalar

Input port, connected to the **down** output of PFD in a PLL system.

Data Types: double

Output

out — Output port

scalar

Output port, connected to the Loop Filter block in a PLL system. **out** delivers current proportional to the difference in duty cycles between **up** and **down** input ports.

Data Types: double

Parameters

Configuration

Output current (A) — Design output current

1e-3 (default) | positive real scalar

Full scale magnitude of design output current, specified as a positive real scalar in amperes.

Programmatic Use

- Use `get_param(gcb, 'OutputCurrent')` to view the current value of **Output current (A)**.
- Use `set_param(gcb, 'OutputCurrent', value)` to set **Output current (A)** to a specific value.

Input threshold (V) — Logic switching threshold at input ports

0.5 (default) | real scalar

Logic switching threshold at input ports, specified as a real scalar in volts.

Programmatic Use

- Use `get_param(gcb, 'InputThreshold')` to view the current value of **Input threshold (V)**.
- Use `set_param(gcb, 'InputThreshold', value)` to set **Input threshold (V)** to a specific value.

Enable increased buffer size — Enable increased buffer size

off (default) | on

Select to enable increased buffer size during simulation. This increases the buffer size of the Logic Decision and Slew Rate inside the Charge Pump block. By default, this option is deselected.

Buffer size — Number of samples of the input buffering available during simulation

10 (default) | positive integer scalar

Number of samples of the input buffering available during simulation, specified as a positive integer scalar. This sets the buffer size of the Logic Decision and Slew Rate inside the Charge Pump block.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when **Enable increased buffer size** option is selected in the **Configuration** tab.

Programmatic Use

- Use `get_param(gcb, 'NBuffer')` to view the current value of **Buffer size**.
- Use `set_param(gcb, 'NBuffer', value)` to set **Buffer size** to a specific value.

Impairments**Enable current impairments — Add current impairments to simulation**

on (default) | off

Select to add current impairments such as current imbalance and leakage current to simulation. By default, this option is selected.

Current imbalance (A) — Difference between full scale positive and negative current

1e-7 (default) | positive real scalar

Difference between full scale positive and negative current, specified as a positive real scalar in amperes.

Dependencies

To enable this parameter, select **Enable current impairments** in the **Impairments** tab.

Programmatic Use

- Use `get_param(gcb, 'CurrentImbalance')` to view the current value of **Current imbalance (A)**.
- Use `set_param(gcb, 'CurrentImbalance', value)` to set **Current imbalance (A)** to a specific value.

Data Types: double

Leakage current (A) – Output current without any input

1e-8 (default) | nonnegative real scalar

Output current when both inputs are at logic zero, specified as a nonnegative real scalar in amperes.

Dependencies

To enable this parameter, select **Enable current impairments** in the **Impairments** tab.

Programmatic Use

- Use `get_param(gcb, 'LeakageCurrent')` to view the current value of **Leakage current (A)**.
- Use `set_param(gcb, 'LeakageCurrent', value)` to set **Leakage current (A)** to a specific value.

Data Types: double

Enable timing impairments – Add timing impairments to simulation

on (default) | off

Select to add timing impairments such as rise/fall time and propagation delay to simulation. By default, this option is selected.

Output step size calculation – Determine how output step size is calculated

Default (default) | Advanced

Determine how output step size is calculated:

- Select **Default** to calculate output step size from rise/fall time. Output step size (ΔT) is given by
$$\Delta T = \frac{(\text{Rise/fall time})^2}{6 \cdot 0.22}$$
.
- Select **Advanced** to calculate output step size from maximum frequency of interest. Output step size (ΔT) is given by
$$\Delta T = \frac{\text{Rise/fall time}}{6 \cdot \text{Maximum frequency of interest}}$$
.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Impairments** tab.

Maximum frequency of interest (Hz) – Maximum frequency of interest at output

10e9 (default) | positive real scalar

Maximum frequency of interest at the output, specified as a positive real scalar in Hz.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Impairments** tab and choose **Advanced** for **Output step size calculation**.

Programmatic Use

- Use `get_param(gcb, 'MaxFreqInterest')` to view the current value of **Maximum frequency of interest (Hz)**.
- Use `set_param(gcb, 'MaxFreqInterest', value)` to set **Maximum frequency of interest (Hz)** to a specific value.

Data Types: double

up

Rise/fall time (s) — 20%-80% rise/fall time for up input port

5e-9 (default) | positive real scalar

20%-80% rise/fall time for **up** input port, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable timing Impairments** in the **Impairments** tab.

Programmatic Use

- Use `get_param(gcb, 'RiseFallUp')` to view the current value of **up Rise/fall time**.
- Use `set_param(gcb, 'RiseFallUp', value)` to set **up Rise/fall time** to a specific value.

Up propagation delay — Total propagation delay for up input port

6e-9 (default) | positive real scalar

Total propagation delay for **up** input port, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable Impairments** in the **Impairments** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelayUp')` to view the current value of **Up propagation delay**.
- Use `set_param(gcb, 'PropDelayUp', value)` to set **Up propagation delay** to a specific value.

down

Rise/fall time — 20%-80% rise/fall time for down input port

2e-9 (default) | positive real scalar

20%-80% rise/fall time for **down** input port, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable Impairments** in the **Impairments** tab.

Programmatic Use

- Use `get_param(gcb, 'RiseFallDown')` to view the current value of **down Rise/fall time**.

- Use `set_param(gcb, 'RiseFallDown', value)` to set **down Rise/fall time** to a specific value.

Down propagation delay — Total propagation delay for down input port

4e-9 (default) | positive real scalar

Total propagation delay for **down** input port, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable Impairments** in the **Impairments** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelayDown')` to view the current value of **Down propagation delay**.
- Use `set_param(gcb, 'PropDelayDown', value)` to set **Down propagation delay** to a specific value.

More About

Inside the Charge Pump

The Charge Pump block converts the two outputs of the PFD block into a single output. It consists of two current branches: one Up and one down. The difference between these two branches is summed to the leakage current impairment, if enabled.

Each current branch consists of a Logic Decision block, an Impairments subsystem, and a gain block. The Logic Decision block compares the incoming signal to the **Input Threshold**. The Impairments subsystem incorporates the effect of the charge pump impairments. The gain block multiplies the output of the Impairments subsystem to produce the current level defined in the **Output current** parameter.

Version History

Introduced in R2019a

References

- [1] Banerjee, Dean. *PLL Performance, Simulation and Design*. Indianapolis, IN: Dog Ear Publishing, 2006.
- [2] Gardner, Floyd M. *Phaselock Techniques*. Hoboken, NJ: John Wiley & Sons, Inc. 2005.

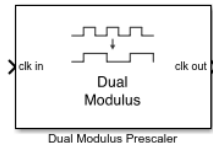
See Also

PFD | Loop Filter

Dual Modulus Prescaler

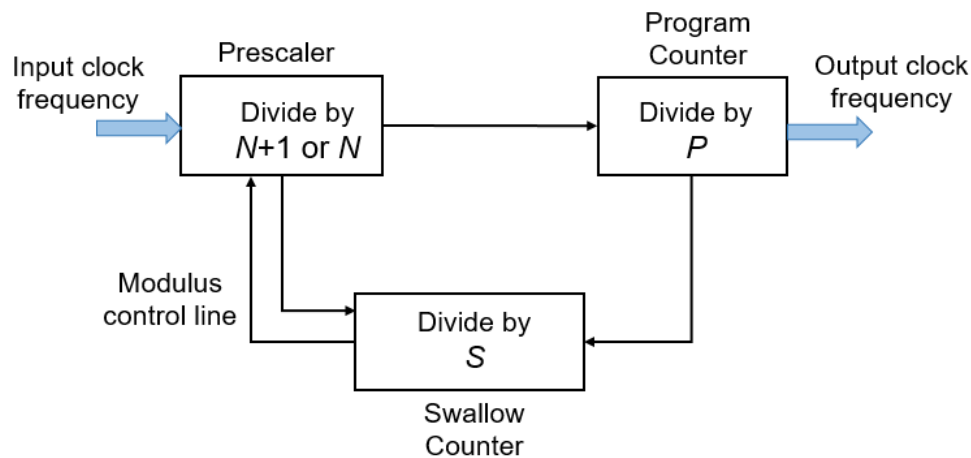
Integer clock divider with two divider ratios

Library: Mixed-Signal Blockset / PLL / Building Blocks



Description

The Dual Modulus Prescaler subsystem block consists of a program counter, a swallow counter and a prescaler.



When the block first receives an input signal, the pulse swallow function is activated. The prescaler divides the input signal frequency by $(N+1)$, where N is defined by the **Prescaler divider value (N)** parameter. Both the program and swallow counters start counting. The swallow counter resets after counting to S pulses, or $(N+1)S$ cycles, where S is defined by the **Swallow counter value (S)** parameter. Then, the pulse swallow function is deactivated, and the prescaler divides the input frequency by N .

Since the program counter has already sensed S pulses, it requires $(P-S)$ more pulses, or $(P-S)N$ cycles to reach overflow, where P is defined by the **Program counter value (P)** parameter. The cycle repeats after both counters are reset.

The effective divider value of the dual modulus prescaler is the ratio of the input frequency to the output frequency:

$$\frac{f_{\text{in}}}{f_{\text{out}}} = (N+1)S + N(P-S) = NP + S$$

Note To prevent the program counter and prescaler from resetting prematurely before the swallow counter finishes counting, the condition $P \geq S$ must be met.

The dual modulus prescaler is also known as pulse swallow divider.

Ports

Input

clk in – Input clock frequency

scalar

Input clock frequency, specified as a scalar. In a phase-locked loop (PLL) system, the **clk in** port is connected to the output port of a VCO block.

Data Types: double

Output

clk out – Output clock frequency

scalar

Output clock frequency, specified as a scalar. In a PLL system, the **clk out** port is connected to the feedback input port of a PFD block. The output at the **clk out** port is a square pulse train of 1 V amplitude.

Data Types: double

Parameters

Program counter value, P – Maximum value of program counter

12 (default) | scalar integer

Maximum value of the program counter, specified as a scalar integer. The counter resets after P cycles.

Programmatic Use

- Use `get_param(gcb, 'P')` to view the current **Program counter value** value.
- Use `set_param(gcb, 'P', value)` to set **Program counter value** to a specific value.

Prescaler divider value, N – Prescaler divider value

4 (default) | scalar integer

Prescaler divider value, specified as a scalar integer. A $N/(N+1)$ dual modulus prescaler divides the input frequency by either N or $N+1$, depending on the logical state of modulus control line.

Programmatic Use

- Use `get_param(gcb, 'N')` to view the current **Prescaler divider value**.
- Use `set_param(gcb, 'N', value)` to set **Prescaler divider value** to a specific value.

Swallow counter value, S – Maximum value of swallow counter

2 (default) | scalar integer

Maximum value of the swallow counter, specified as a scalar integer. When the swallow counter resets after S cycles, the pulse swallow function is deactivated.

Programmatic Use

- Use `get_param(gcb, 'S')` to view the current **Swallow counter value** value.
- Use `set_param(gcb, 'S', value)` to set **Swallow counter value** to a specific value.

Enable increased buffer size — Enable increased buffer size

off (default) | on

Select to enable increased buffer size during simulation. This increases the buffer size of the Logic Decision inside the Dual Modulus Prescaler block. By default, this option is deselected.

Buffer size — Number of samples of the input buffering available during simulation

1 (default) | positive integer scalar

Number of samples of the input buffering available during simulation, specified as a positive integer scalar. This sets the buffer size of the Logic Decision inside the Dual Modulus Prescaler block.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when **Enable increased buffer size** option is selected in the Block Parameters dialog box.

Programmatic Use

- Use `get_param(gcb, 'NBuffer')` to view the current value of **Buffer size**.
- Use `set_param(gcb, 'NBuffer', value)` to set **Buffer size** to a specific value.

More About**Inside the Mask**

The Dual Modulus Prescaler subsystem block consists of three different subsystems that implement the three main parts of the dual modulus prescaler. The Prescaler divides the input frequency by either N or $N+1$, depending on the logical state of modulus control line. The Program Counter subsystem always divides the prescaler output frequency by P .

The Swallow Counter divides the prescaler output by S . S depends on the digital input and can vary from 1 to maximum number of channels. S also determines the logic state of the modulus control line.

Version History

Introduced in R2019a

References

[1] Razavi, Behzad. *RF Microelectronics*. Upper Saddle River, NJ: Prentice Hall PTR, 1998.

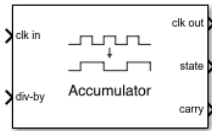
See Also

PFD | VCO | Single Modulus Prescaler | Fractional Clock Divider with Accumulator | Fractional Clock Divider with DSM

Fractional Clock Divider with Accumulator

Clock divider that divides frequency of input signal by fractional number

Library: Mixed-Signal Blockset / PLL / Building Blocks



Description

The Fractional Clock Divider with Accumulator block divides the frequency of the input signal by a tunable fractional value ($N.FF$). When compared to the Single Modulus Prescaler block, the Fractional Clock Divider with Accumulator block helps to achieve a narrow channel spacing that can be less than the reference frequency of a phase-locked loop (PLL) system.

Ports

Input

clk in — Input clock frequency

scalar

Input clock frequency, specified as a scalar. In a PLL system, the **clk in** port is connected to the output port of a VCO block.

Data Types: double

div-by — Ratio of output to input clock frequency

fractional scalar

Ratio of output to input clock frequency, specified as a fractional scalar.

The value at the **div-by** port is split into two parts: the integer part (N) and the fractional part ($.FF$).

Data Types: double

Output

clk out — Output clock frequency

scalar

Output clock frequency, specified as a scalar. In a PLL system, the **clk out** port is connected to the feedback input port of a PFD block. The output at the **clk out** port is a square pulse train of 1 V amplitude.

Data Types: double

state — Missing fractional pulse storage

scalar

The fractional missing pulse storage. The value of the **state** port goes up by F with each rising edge of the **clk out** value of the previous cycle. Whenever the **state** port value goes over 1, the value overflows and sets the **carry** port value to 1.

Data Types: `double`

carry — Activates the pulse swallow function when state port overflows

0 (default) | 1

Output port that activates the pulse swallow function when **state** port overflows. The pulse removal is analogous to dividing the input frequency by $N+1$ instead of N .

Data Types: `Boolean`

Parameters

Enable increased buffer size — Enable increased buffer size

off (default) | on

Select to enable increased buffer size during simulation. This increases the buffer size of the Logic Decision inside the Fractional Clock Divider with Accumulator block. By default, this option is deselected.

Buffer size — Number of samples of the input buffering available during simulation

1 (default) | positive integer scalar

Number of samples of the input buffering available during simulation, specified as a positive integer scalar. This sets the buffer size of the Logic Decision inside the Fractional Clock Divider with Accumulator block.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when **Enable increased buffer size** option is selected in the Block Parameters dialog box.

Programmatic Use

- Use `get_param(gcb, 'NBuffer')` to view the current value of **Buffer size**.
- Use `set_param(gcb, 'NBuffer', value)` to set **Buffer size** to a specific value.

More About

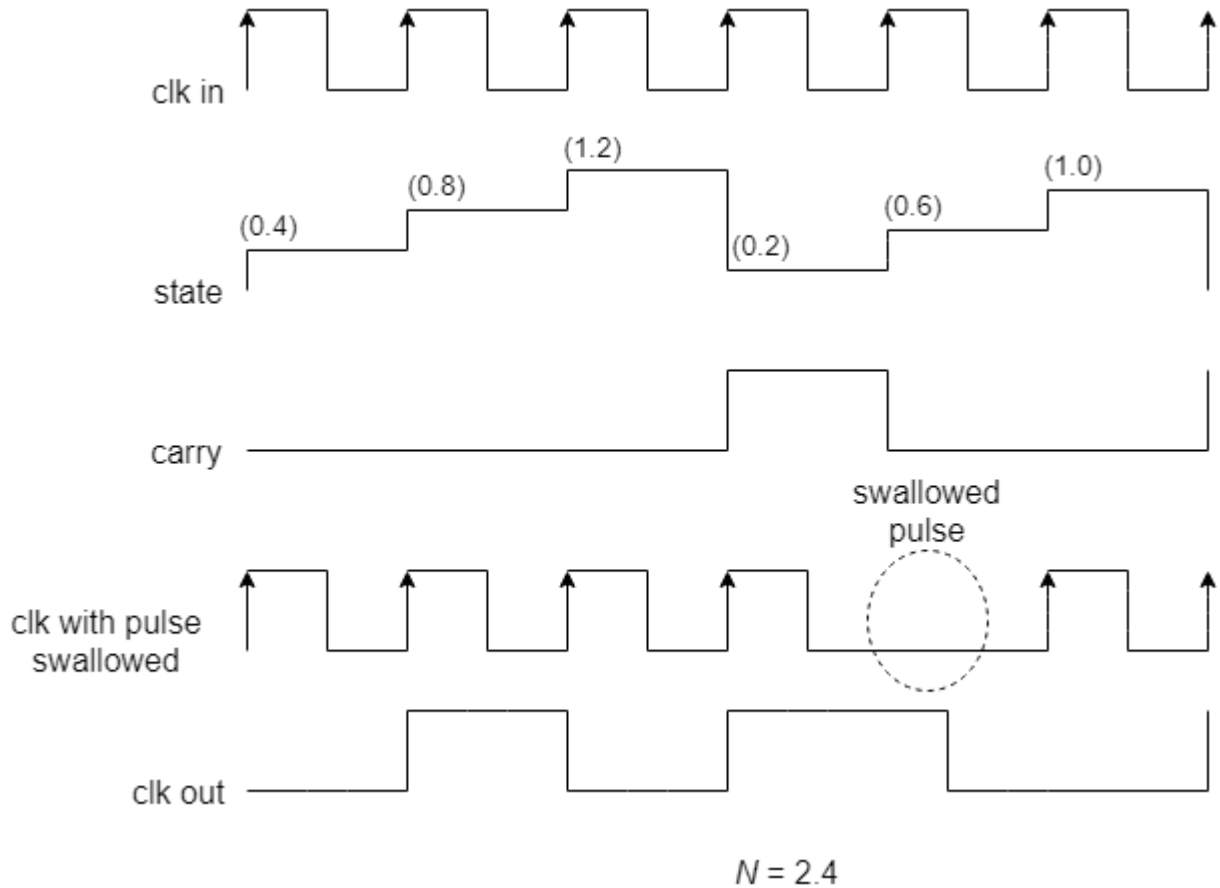
Inside the Mask

The Fractional Clock Divider with Accumulator consists of three subsystems: a Fractional Accumulator, a Pulse Swallower, and a Single Modulus Prescaler block.

When the block first receives an input signal, the Single Modulus Prescaler block divides the input signal by the integer part (N) of the value of the **div-by** port. The fractional part ($.FF$) is stored in the **state** port of the Fractional Accumulator subsystem.

The Fractional Accumulator subsystem updates the **state** port value with each rising edge received by the **clk out** port in the previous cycle. When the value of the **state** port goes over 1, the value overflows and changes the value emitted by the **carry** port to 1.

The **carry** port activates the Pulse Swallower subsystem, and removes one pulse from the **clk in** signal. It is like dividing the input signal by a factor of $N+1$, achieving the overall fractional division.



Version History

Introduced in R2019a

References

[1] Best, Roland E. *Phase-Locked Loop*. New York, NY: Tata McGraw-Hill Companies Inc., 2003.

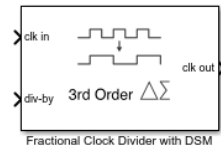
See Also

PFD | VCO | Fractional Clock Divider with DSM | Single Modulus Prescaler | Dual Modulus Prescaler

Fractional Clock Divider with DSM

Delta Sigma Modulator based fractional clock divider

Library: Mixed-Signal Blockset / PLL / Building Blocks



Description

Using delta sigma (Δ - Σ) modulation technique, a Fractional Clock Divider with DSM reduces the primary fractional spurs by spreading out the range over which the **div-by** value is varied. This block allows delta sigma modulation of up to 4th order.

Ports

Input

clk in — Input clock frequency

scalar

Input clock frequency that needs to be divided, specified as a scalar. In a phase-locked loop (PLL) system, the **clk in** port is connected to the output of a VCO block.

Data Types: double

div-by — Ratio of output to input clock frequency

fractional scalar

Ratio of output to input clock frequency, specified as a fractional scalar. The value at the **div-by** port, $N.FF$, is split into two parts: the integer part (N) and the fractional part ($.FF$).

For an n th-order delta sigma modulator, the value at the **div-by** port is achieved by varying N between 2^n different integer values.

Note For an n th order delta sigma modulator, use a value $\geq 2^n$ at the **div-by** port.

Data Types: double

Output

clk out — Output clock frequency

scalar

Output clock frequency, specified as a scalar. In a PLL system, the **clk out** port is connected to the feedback input port of a PFD block. The output at the **clk out** port is a square pulse train of 1 V amplitude.

Data Types: double

Parameters

Delta Sigma Modulator order — Order of the Delta Sigma Modulator

3rd order (default) | 1st order | 2nd order | 4th order

The order of the delta sigma modulator.

For an n th-order of the delta sigma modulator, the value at the **div-by** port is achieved by varying the N counter value between 2^n different values. Modulator order defines the range of values by which the signal at the **clk in** port will be divided, providing a division effect similar to $N.FF$ value at the **div-by** port.

Programmatic Use

- Use `get_param(gcb, 'dsm')` to view the current **Delta Sigma Modulator order**.
- Use `set_param(gcb, 'dsm', value)` to set **Delta Sigma Modulator order** to a specific value.

Enable increased buffer size — Enable increased buffer size

off (default) | on

Select to enable increased buffer size during simulation. This increases the buffer size of the Logic Decision inside the Fractional Clock Divider with DSM block. By default, this option is deselected.

Buffer size — Number of samples of the input buffering available during simulation

1 (default) | positive integer scalar

Number of samples of the input buffering available during simulation, specified as a positive integer scalar. This sets the buffer size of the Logic Decision inside the Fractional Clock Divider with DSM block.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when **Enable increased buffer size** option is selected in the Block Parameters dialog box.

Programmatic Use

- Use `get_param(gcb, 'NBuffer')` to view the current value of **Buffer size**.
- Use `set_param(gcb, 'NBuffer', value)` to set **Buffer size** to a specific value.

More About

Inside the Mask

The Fractional Clock Divider with DSM subsystem block consists of four delta sigma modulators of orders one to four encapsulated inside the DSM Selector variant subsystem. The output of the DSM selector drives a Single Modulus Prescaler block. Given the **Delta Sigma Modulator order**, corresponding delta sigma modulator gets activated.

The modulator order defines the range over which the N counter value is varied. For an n th-order delta sigma modulator, N is varied over 2^n different values. This variation is achieved by integrating

the changes in the fractional part (.FF) from the previous cycle and quantizing the differential changes.

The general form of the transfer function for an nth order delta sigma modulator is:

$$Y(z) = X(z) + E(z) \cdot (1 - z^{-1})^n$$

where

- $Y(z)$ = Output of the modulator
- $X(z)$ = Input the modulator
- $E(z)$ = Quantization error

$E(z)$ is calculated by subtracting the value of input $X(z)$ in the present cycle from its value in the previous cycle. In other words, $E(z)$ is a form of a digital highpass filtering.

The higher-order modulators reduce the primary fractional spurs by alternating N over a larger range of integer values. As a result, the fractional spurs are pushed to higher frequencies in the frequency spectrum and can be filtered more effectively by the loop filter in a PLL system.

For example, if the third-order delta sigma modulator is activated, N is varied over 8 different values, which can range from $(N-3)$ to $(N+4)$.

Delta Sigma Modulator Sequence

Modulator Order	Range	DSM Sequence
1st	0, 1	$N, N+1$
2nd	-1, 0, 1, 2	$N-1, N, N+1, N+2$
3rd	-3, -2, -1, 0, 1, 2, 3, 4	$N-3, N-2, \dots, N+4$
4th	-7, -6, ..., 7, 8	$N-7, N-6, \dots, N+8$

Version History

Introduced in R2019a

References

- [1] Miller, B. and Conley, R.J., *A Multiple Modulator Fractional Divider*. IEEE Transactions on Instrumentation and Measurement, vol. 40, no. 3, 1991, pp. 578-583.

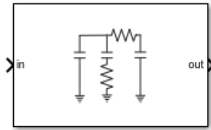
See Also

PFD | VCO | Fractional Clock Divider with Accumulator | Single Modulus Prescaler | Dual Modulus Prescaler

Loop Filter

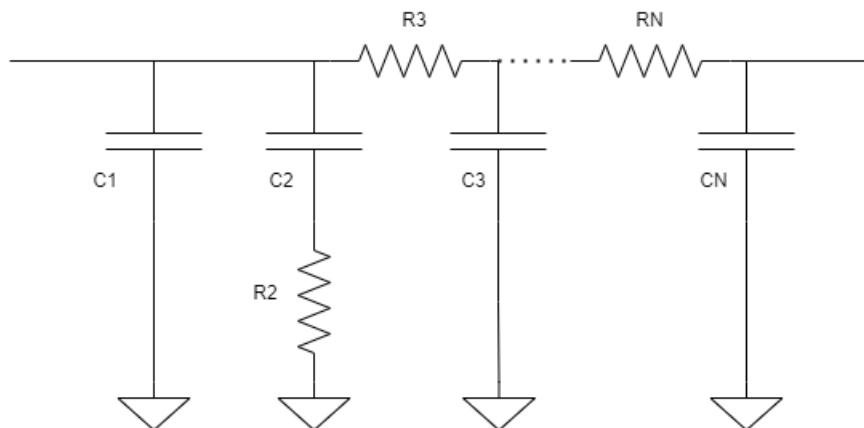
Model second-, third-, or fourth-order passive loop filter

Library: Mixed-Signal Blockset / PLL / Building Blocks



Description

The Loop Filter subsystem block is a passive filter whose order can vary from two to four. In a phase-locked loop (PLL) system, the subsystem filters the output of Charge Pump block and delivers the control voltage to a VCO block to generate required frequency signal.



nth Order Passive Loop Filter

Ports

Input

in – Input current

scalar

Input current, specified as a scalar. In a phase-locked loop (PLL) system, the **in** port is connected to the output of a Charge Pump block, which provides the current value.

Data Types: double

Output

out – Output voltage

scalar

Output voltage, specified as a scalar. In a PLL system, the **out** port is connected to the input port of a VCO block and provides the control voltage to VCO.

Data Types: double

Parameters

Configuration

Loop filter type – Order of the loop filter

3rd Order passive (default) | 2nd Order passive | 4th Order passive

Order of the loop filter. Simulates a second-, third-, or fourth-order passive RC loop filter.

Programmatic Use

- Use `get_param(gcb, 'FilterType')` to view the current order of **Loop filter type**.
- Use `set_param(gcb, 'FilterType', value)` to set **Loop filter type** to a specific order.

C1 (F) – Capacitance 1

14.5661e-15 (default) | positive real scalar

Capacitor value C1, specified as a positive real scalar in farads.

Programmatic Use

- Use `get_param(gcb, 'C1')` to view the current value of **C1 (F)**.
- Use `set_param(gcb, 'C1', value)` to set **C1 (F)** to a specific value.

Data Types: double

C2 (F) – Capacitance 2

160.276e-15 (default) | positive real scalar

Capacitor value C2, specified as a positive real scalar in farads.

Programmatic Use

- Use `get_param(gcb, 'C2')` to view the current value of **C2 (F)**.
- Use `set_param(gcb, 'C2', value)` to set **C2 (F)** to a specific value.

Data Types: double

C3 (F) – Capacitance 3

1.0452e-15 (default) | positive real scalar

Capacitor value C3, specified as a positive real scalar in farads.

Dependencies

To enable this parameter, select 3rd Order passive or 4th Order passive in **Loop filter type**.

Programmatic Use

- Use `get_param(gcb, 'C3')` to view the current value of **C3 (F)**.
- Use `set_param(gcb, 'C3', value)` to set **C3 (F)** to a specific value.

Data Types: double

C4 (F) — Capacitance 4

1e-12 (default) | positive real scalar

Capacitor value C4, specified as a positive real scalar in farads.

Dependencies

To enable this parameter, select 4th Order passive in **Loop filter type**.

Programmatic Use

- Use `get_param(gcb, 'C4')` to view the current value of **C4 (F)**.
- Use `set_param(gcb, 'C4', value)` to set **C4 (F)** to a specific value.

Data Types: double

R2 (ohms) — Resistance 2

3.9955e6 (default) | positive real scalar

Resistor value R2, specified as a positive real scalar in ohms.

Programmatic Use

- Use `get_param(gcb, 'R2')` to view the current value of **R2 (ohms)**.
- Use `set_param(gcb, 'R2', value)` to set **R2 (ohms)** to a specific value.

Data Types: double

R3 (ohms) — Resistance 3

51.0435e6 (default) | positive real scalar

Resistor value R3, specified as a positive real scalar in ohms.

Dependencies

To enable this parameter, select 3rd Order passive or 4th Order passive in **Loop filter type**.

Programmatic Use

- Use `get_param(gcb, 'R3')` to view the current value of **R3 (ohms)**.
- Use `set_param(gcb, 'R3', value)` to set **R3 (ohms)** to a specific value.

Data Types: double

R4 (ohms) — Resistance 4

12e3 (default) | positive real scalar

Resistor value R4, specified as a positive real scalar in ohms.

Dependencies

To enable this parameter, select 4th Order passive in **Loop filter type**.

Programmatic Use

- Use `get_param(gcb, 'R4')` to view the current value of **R4 (ohms)**.
- Use `set_param(gcb, 'R4', value)` to set **R4 (ohms)** to a specific value.

Data Types: double

Enable increased buffer size — Enable increased buffer size

off (default) | on

Select to enable increased buffer size during simulation. This increases the buffer size of the Convert Sample Time subsystem inside the Loop Filter block. By default, this option is deselected.

Buffer size — Number of samples of the input buffering available during simulation

1000 (default) | positive integer scalar

Number of samples of the input buffering available during simulation, specified as a positive integer scalar. This sets the buffer size of the Convert Sample Time subsystem inside the Loop Filter block.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when **Enable increased buffer size** option is selected in the **Configuration** tab.

Programmatic Use

- Use `get_param(gcb, 'NBuffer')` to view the current value of **Buffer size**.
- Use `set_param(gcb, 'NBuffer', value)` to set **Buffer size** to a specific value.

Data Types: double

Impairments

Enable impairments — Add circuit impairments to simulation

off (default) | on

Select to add circuit impairments to simulation. By default, this option is disabled.

Operating temperature (°C) — Temperature to determine the level of thermal noise

30 (default) | scalar

Temperature of the resistor, specified as a scalar in degree Celsius. **Operating temperature (°C)** determines the level of thermal (Johnson) noise.

Dependencies

To enable this parameter, select **Enable impairments** in the **Impairments** tab.

Programmatic Use

- Use `get_param(gcb, 'Temperature')` to view the current value of **Operating temperature (°C)**.
- Use `set_param(gcb, 'Temperature', value)` to set **Operating temperature (°C)** to a specific value.

Data Types: double

More About

Inside the Mask

The Loop Filter subsystem block consists of four parts: Convert Sample Time, Main Filter, Extra Poles, and Resistor Thermal Noise. The Main Filter and Extra Pole are implemented using Biquad IIR filters, and generate the transfer function based on the filter order selected. Convert Sample Time is used to convert the discrete output of PFD to a continuous signal. Resistor Thermal Noise incorporates the thermal noise based on the operating temperature of loop filter.

Loop Filter Transfer Function

Transfer function of second order passive loop filter:

$$Z(s) = \frac{R2 \cdot C2 \cdot s + 1}{A2 \cdot s^2 + A1 \cdot s}$$

Transfer function of third order passive loop filter:

$$Z(s) = \frac{R2 \cdot C2 \cdot s + 1}{A3 \cdot s^3 + A2 \cdot s^2 + A1 \cdot s}$$

Transfer function of fourth order passive loop filter:

$$Z(s) = \frac{R2 \cdot C2 \cdot s + 1}{A4 \cdot s^4 + A3 \cdot s^3 + A2 \cdot s^2 + A1 \cdot s}$$

where, A1, A2, A3, and A4 are the loop filter coefficients.

Loop Filter Coefficients

Filter Order	A1	A2	A3	A4
2nd	$C1+C2$	$C1 \cdot C2 \cdot R2$	N/A	N/A
3rd	$C1+C2+C3$	$(R2 \cdot C2 \cdot C3) + (R2 \cdot C1 \cdot C2) + (R3 \cdot C3 \cdot C1) + (R3 \cdot C3 \cdot C2)$	$C1 \cdot C2 \cdot C3 \cdot R2 \cdot R3$	N/A
4th	$C1+C2+C3+C4$	$C2 \cdot R2(C1+C3+C4) + R3(C1+C2)(C3+C4) + C4 \cdot R4(C1+C2+C3)$	$((R2 \cdot C2 \cdot C3) + (R2 \cdot C1 \cdot C2) + (R3 \cdot C3 \cdot C1) + (R3 \cdot C3 \cdot C2))R4 \cdot C4 + C1 \cdot C2 \cdot R2 \cdot R3(C3 + C4)$	$C1 \cdot C2 \cdot C3 \cdot C4 \cdot R2 \cdot R3 \cdot R4$

Version History

Introduced in R2019a

References

- [1] Banerjee, Dean. *PLL Performance, Simulation and Design*. Indianapolis, IN: Dog Ear Publishing, 2006.

[2] Bleany, B.I and Bleany B. *Electricity and Magnetism*. Oxford, UK: Oxford University Press, 1976.

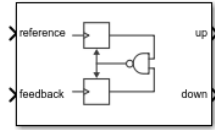
See Also

Charge Pump | VCO

PFD

Phase/frequency detector that compares phase and frequency between two signals

Library: Mixed-Signal Blockset / PLL / Building Blocks



Description

The PFD block produces two output pulses that differ in duty cycle. The difference in the duty cycle is proportional to the phase difference between input signals. In frequency synthesizer circuits, such as phase-locked loops (PLL), the PFD block compares the phase and frequency between the reference signal and signal generated by the VCO block and determines the phase error.

Ports

Input

reference — Reference frequency

scalar

Input port that transmits reference frequency to determine phase error.

Data Types: double

feedback — Feedback frequency

scalar

Output port that transmits the feedback frequency to determine the phase error. In a PLL system, the output of the VCO is fed back through **feedback** port to PFD after passing through a clock divider.

Data Types: double

Output

up — Transmits reference frequency

scalar

Output port that transmits reference frequency to Charge Pump to convert the phase error into current. The difference in the duty cycle of signals in **up** and **down** ports is proportional to the phase difference between the signals in **reference** and **feedback** ports.

Data Types: double

down — Transmits feedback frequency

scalar

Output port that transmits feedback frequency to Charge Pump to convert the phase error into current. The difference in the duty cycle of signals in **up** and **down** ports is proportional to the phase difference between the signals in **reference** and **feedback** ports.

Data Types: double

Parameters

Configuration

Deadband compensation (s) — Delay added for active output near zero phase offset
30e-12 (default) | positive real scalar

Delay added for active output near zero phase offset, specified as a positive real scalar in seconds. Deadband is the phase offset band near zero phase offset for which the PFD output is negligible.

Programmatic Use

- Use `get_param(gcb, 'DeadbandCompensation')` to view the current value of **Deadband compensation**.
- Use `set_param(gcb, 'DeadbandCompensation', value)` to set **up Rise/fall time** to a specific value.

Data Types: double

Enable increased buffer size — Enable increased buffer size
off (default) | on

Select to enable increased buffer size during simulation. This increases the buffer size of the Variable Pulse Delay, Logic Decision, and Slew Rate blocks inside the PFD block. By default, this option is deselected.

Buffer size — Number of samples of the input buffering available during simulation
10 (default) | positive integer scalar

Number of samples of the input buffering available during simulation, specified as a positive integer scalar. This sets the buffer size of the Variable Pulse Delay, Logic Decision, and Slew Rate blocks inside the PFD block.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when **Enable increased buffer size** option is selected in the **Configuration** tab.

Programmatic Use

- Use `get_param(gcb, 'NBuffer')` to view the current value of **Buffer size**.
- Use `set_param(gcb, 'NBuffer', value)` to set **Buffer size** to a specific value.

Data Types: double

Impairments

Enable impairments — Add circuit impairments to simulation
on (default) | off

Select to add circuit impairments such as rise/fall time and propagation delay to the simulation. By default, this option is selected.

Output step size calculation — Determine how output step size is calculated

Default (default) | Advanced

Determine how output step size is calculated:

- Select **Default** to calculate output step size from rise/fall time. Output step size (ΔT) is given by
$$\Delta T = \frac{(\text{Rise/fall time})^2}{6 \cdot 0.22}$$
.
- Select **Advanced** to calculate output step size from maximum frequency of interest. Output step size (ΔT) is given by
$$\Delta T = \frac{\text{Rise/fall time}}{6 \cdot \text{Maximum frequency of interest}}$$
.

Dependencies

To enable this parameter, select **Enable Impairments** in the **Impairments** tab.

Maximum frequency of interest (Hz) — Maximum frequency of interest at output

10e9 (default) | positive real scalar

Maximum frequency of interest at the output, specified as a positive real scalar in Hz.

Dependencies

To enable this parameter, select **Enable Impairments** in the **Impairments** tab and choose **Advanced** for **Output step size calculation**.

Programmatic Use

- Use `get_param(gcb, 'MaxFreqInterest')` to view the current value of **Maximum frequency of interest (Hz)**.
- Use `set_param(gcb, 'MaxFreqInterest', value)` to set **Maximum frequency of interest (Hz)** to a specific value.

Data Types: double

Rise/fall time (s) — 20% - 80% rise/fall time for up output port of PFD

3e-11 (default) | positive real scalar

20% - 80% rise/fall time for the up output port of the PFD, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable Impairments** in the **Impairments** tab.

Programmatic Use

- Use `get_param(gcb, 'RiseFallTime')` to view the current value of **Rise/fall time (s)**.
- Use `set_param(gcb, 'RiseFallTime', value)` to set **Impairments** to a specific value.

Data Types: double

Propagation Delay (s) — Delay from input port to output port of PFD

50e-12 (default) | positive real scalar

Delay from the input port to output port of the PFD, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable Impairments** in the **Impairments** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelay')` to view the current value of **Propagation Delay (s)**.
- Use `set_param(gcb, 'PropDelay', value)` to set **Propagation Delay (s)** to a specific value.

Data Types: `double`

More About

Inside the Mask

PFD consists of two synchronous D flip-flops. The reference and feedback signals received at the corresponding ports act as the trigger. The outputs of the two flip-flops pass through a NAND gate, which acts as the reset signal. A pulse delay is introduced after the NAND gate using Variable Pulse Delay block to compensate for deadband.

The impairments are contained in variant subsystems and activated when impairments are enabled. The impairment subsystem utilized Slew Rate block to implement rise/fall time and propagation delay.

Version History

Introduced in R2019a

References

[1] Banerjee, Dean. *PLL Performance, Simulation and Design*. Indianapolis, IN: Dog Ear Publishing, 2006.

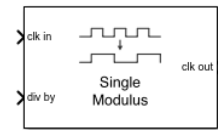
See Also

Charge Pump | Fractional Clock Divider with Accumulator | Fractional Clock Divider with DSM | Single Modulus Prescaler | Dual Modulus Prescaler

Single Modulus Prescaler

Integer clock divider that divides frequency of input signal

Library: Mixed-Signal Blockset / PLL / Building Blocks



Description

The Single Modulus Prescaler subsystem block divides the frequency of the input signal by a tunable integer value, N , passed to the **div-by** port. In frequency synthesizer circuits, such as a phase-locked loop (PLL) system, these prescalers divide the VCO output frequency by an integer value. The resulting lower frequency at the prescaler output port is comparable to the reference input at the PFD block. The Single Modulus Prescaler is also termed as integer clock divider.

Ports

Input

clk in — Input clock frequency

scalar

Input clock frequency, specified as a scalar. In a PLL system, the **clk in** port is connected to the output port of a VCO block.

Data Types: double

div-by — Ratio of output to input clock frequency

scalar integer

Ratio of output to input clock frequency, expressed as a scalar integer.

Data Types: double

Output

clk out — Output clock frequency

scalar

Output clock frequency, expressed as a scalar. In a PLL system, the **clk out** port is connected to the feedback input port of a PFD block. The output at the **clk out** port is a square pulse train of 1 V amplitude.

Data Types: double

Parameters

Enable increased buffer size — Enable increased buffer size

off (default) | on

Select to enable increased buffer size during simulation. This increases the buffer size of the Logic Decision inside the Single Modulus Prescaler block. By default, this option is deselected.

Buffer size — Number of samples of the input buffering available during simulation

1 (default) | positive integer scalar

Number of samples of the input buffering available during simulation, specified as a positive integer scalar. This sets the buffer size of the Logic Decision inside the Single Modulus Prescaler block.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when **Enable increased buffer size** option is selected in the Block Parameters dialog box.

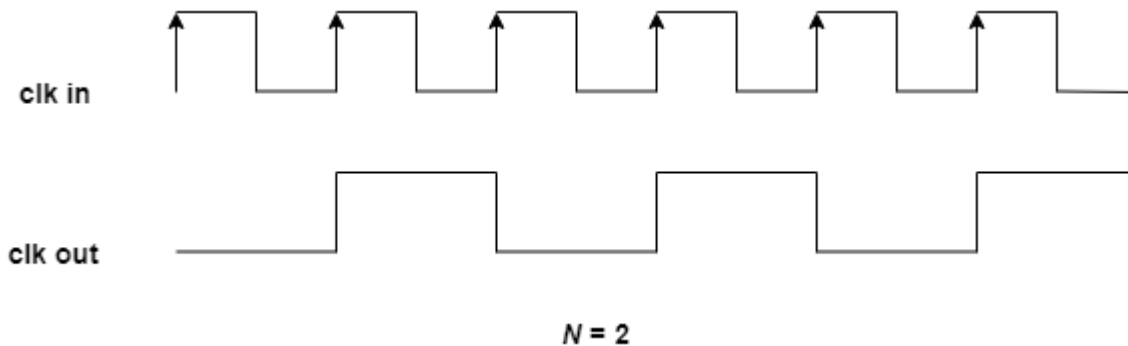
Programmatic Use

- Use `get_param(gcb, 'NBuffer')` to view the current value of **Buffer size**.
- Use `set_param(gcb, 'NBuffer', value)` to set **Buffer size** to a specific value.

More About

Inside the Mask

The Single Modulus Prescaler block contains the integer clock divider subsystem. Inside the subsystem, a trigger port tracks the rising edges of the input clock signal received at **clk in** port. The output sends a pulse only after N cycles have been detected. As a result, the input clock frequency reduces by a factor of N .



Version History

Introduced in R2019a

References

- [1] Razavi, Behzad. *RF Microelectronics*. Upper Saddle River, NJ: Prentice Hall PTR, 1998.

See Also

PFD | VCO | Dual Modulus Prescaler | Fractional Clock Divider with Accumulator | Fractional Clock Divider with DSM

VCO

Model voltage controlled oscillator

Library: Mixed-Signal Blockset / PLL / Building Blocks



Description

VCO or voltage controlled oscillator is a voltage to frequency converter. It produces an output square wave signal whose frequency is controlled by the voltage at the input **vctrl** port. The frequency of the output signal, F is determined either by:

$$F = (K_{VCO} \cdot V_{ctrl}) + F_o$$

where:

- K_{vco} = voltage sensitivity (in Hz/V)
- V_{ctrl} = control voltage (in V)
- F_o = free running frequency (in Hz)

or from linear interpolation using the mapping:

$$F = \text{interp}(F_{out}(V_{ctrl}))$$

where:

- V_{ctrl} = vector of control voltages (in V)
- F_{out} = vector of corresponding output frequencies (in Hz)

Ports

Input

vctrl — Voltage used to control VCO output frequency

scalar | vector

VCO control voltage used to control the output frequency of the VCO. In a phase-locked loop (PLL) system, **vctrl** is the output of the Loop Filter that contains the phase error information.

Data Types: double

Output

vco out — Output square wave signal determined by vctrl port

scalar

Output square wave signal of VCO. In a PLL system, **vco out** is the output clock generated by the PLL. It is also fed back to the PFD block through a clock divider to complete the control loop.

Data Types: double

Parameters

Parameters

Specify using — Define how VCO output frequency is specified

Voltage sensitivity (default) | Output frequency vs. control voltage

Define how VCO output frequency is specified:

- Select Voltage sensitivity to specify output frequency from **Voltage sensitivity (Hz/V)** and **Free running frequency (Hz)**.
- Select Output frequency vs. control voltage to interpolate output frequency from **Control voltage (V)** vector versus **Output frequency (Hz)** vector.

Programmatic Use

Block parameter: SpecifyUsing

Type: character vector

Values: Voltage sensitivity | Output frequency vs. control voltage

Default: Voltage sensitivity

Voltage sensitivity (Hz/V) — Measure of change in output frequency of VCO

100e6 (default) | positive real scalar

Measure of change in output frequency for input voltage change, specified as a positive real scalar with units in Hz/V. This parameter is also reported as **VCO voltage sensitivity** in the **Loop Filter** tab and is used to automatically calculate the filter component values of the loop filter.

Dependencies

To enable this parameter, select Voltage sensitivity in **Specify using** in the **Parameters** tab.

Programmatic Use

Block parameter: Kvco

Type: character vector

Values: positive real scalar

Default: 100e6

Data Types: double

Free running frequency (Hz) — VCO output frequency without control voltage

2.5e9 (default) | positive real scalar

Frequency of the VCO without any control voltage input (0 V), or the quiescent frequency, specified as a positive real scalar in Hz.

Dependencies

To enable this parameter, select Voltage sensitivity in **Specify using** in the **Parameters** tab.

Programmatic Use

Block parameter: Fo

Type: character vector

Values: positive real scalar

Default: 2.5e9

Data Types: double

Control voltage (V) — Control voltage values

[-5 0 5] (default) | real valued vector

Control voltage values of the VCO, specified as a real valued vector in volts.

Dependencies

To enable this parameter, select Output frequency vs. control voltage in **Specify using** in the **Parameters** tab.

Programmatic Use

Block parameter: ControlVoltage

Type: character vector

Values: real valued vector

Default: [-5 0 5]

Data Types: double

Output frequency (Hz) — VCO output frequency values

[2e9 2.5e9 3e9] (default) | positive real valued vector

Output frequency of the values of the VCO, corresponding to the **Control voltage (V)** vector, specified in Hz.

Dependencies

To enable this parameter, select Output frequency vs. control voltage in **Specify using** in the **Parameters** tab.

Programmatic Use

Block parameter: OutputFrequency

Type: character vector

Values: positive real valued vector

Default: [2e9 2.5e9 3e9]

Data Types: double

Output amplitude (V) — Maximum amplitude of the VCO output voltage

1 (default) | positive real scalar

Maximum amplitude of the VCO output voltage, specified as a positive real scalar.

Programmatic Use

Block parameter: Amplitude

Type: character vector

Values: positive real scalar

Default: 1

Data Types: double

Enable increased buffer size — Enable increased buffer size

off (default) | on

Select to enable increased buffer size during simulation. This increases the buffer size of the Variable Pulse Delay block inside the VCO block. By default, this option is deselected.

Buffer size — Number of samples of the input buffering available during simulation

10 (default) | positive integer scalar

Number of samples of the input buffering available during simulation, specified as a positive integer scalar. This sets the buffer size of the Variable Pulse Delay block inside the VCO block.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

To enable this parameter, select **Enable increased buffer size**.

Programmatic Use**Block parameter:** NBuffer**Type:** character vector**Values:** positive integer scalar**Default:** 10

Data Types: double

Impairments**Add phase noise — Add phase noise as a function of frequency**

on (default) | off

Select to introduce phase noise as a function of frequency to the VCO. By default, this option is selected.

Phase noise frequency offset (Hz) — Frequency offsets of phase noise from carrier frequency

[30e3 100e3 1e6 3e6 10e6] (default) | positive real valued vector

The frequency offsets of phase noise from the carrier frequency specified as a positive real valued vector in Hz.

Dependencies

To enable this parameter, select **Add phase noise** in the **Impairments** tab.

Programmatic Use**Block parameter:** Foffset**Type:** character vector**Values:** positive real valued vector**Default:** [30e3 100e3 1e6 3e6 10e6]

Data Types: double

Phase noise level (dBc/Hz) — Phase noise power at specified frequency offsets relative to the carrier

[-56 -106 -132 -143 -152] (default) | negative real valued vector

The phase noise power in a 1 Hz bandwidth centered at the specified frequency offsets relative to the carrier specified as a negative real valued vector in dBc/Hz. The elements of **Phase noise level** corresponds to relative elements in the **Phase noise frequency offset**.

Dependencies

To enable this parameter, select **Add phase noise** in the **Impairments** tab.

Programmatic Use

Block parameter: PhaseNoise

Type: character vector

Values: negative real valued vector

Default: [-56 -106 -132 -143 -152]

Data Types: double

More About

VCO Subsystem

The VCO subsystem block consists of two subsystems, Ideal VCO and Real VCO encapsulated under one variant subsystem.

If **Add phase noise** impairment is disabled, then the Ideal VCO subsystem gets active. This produces the following two orthogonal output signals, without any phase noise impairment and hence the name Ideal VCO:

$$y_1(t) = A \cos \int (2\pi K_{\text{vco}} * V_{\text{ctrl}} + 2\pi F_{\text{out}}) dt$$

$$y_2(t) = -A \sin \int (2\pi K_{\text{vco}} * V_{\text{ctrl}} + 2\pi F_{\text{out}}) dt$$

Out of the two orthogonal outputs, only the real part of the signal, $y_1(t)$ is connected to the output port of VCO.

When the **Add phase noise** impairment is enabled, the Real VCO block becomes active which introduces phase noise as a function of frequency to the ideal VCO output. The subsystem consists of the Ideal VCO block with a phase noise generator block. The latter adds phase noise impairment to the input signal.

Version History

Introduced in R2019a

References

[1] Banerjee, Dean. *PLL Performance, Simulation and Design*. Indianapolis, IN: Dog Ear Publishing, 2006.

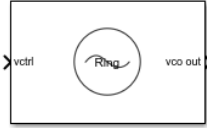
See Also

Loop Filter | PFD | VCO Testbench | Ring Oscillator VCO

Ring Oscillator VCO

Model ring oscillator VCO

Library: Mixed-Signal Blockset / PLL / Building Blocks



Description

The Ring Oscillator VCO block models the output signal, frequency control, period jitter, and flicker noise of a VCO (voltage controlled oscillator) such as a bias controlled ring oscillator circuit. This block generates the phase noise using a mathematical description of the phase noise of ring oscillators. This allows faster computation of simulation results during both startup and the subsequent simulations. You can also control the phase noise profile by selecting the Gaussian noise level, corner frequency, and flicker exponent. The phase noise spectrum is limited to the spectra that can be produced by the physical model of a ring oscillator.

You can choose the coefficients for the mathematical description of the phase noise. You can provide a specific phase noise spectral density from a data sheet and compare that to the phase noise spectral density that the mathematical coefficients produce. You can then adjust the coefficients to fit the specified phase noise in a way that makes the most sense physically.

Note If the flicker noise corner frequency is set to zero, the Ring Oscillator VCO block can also be used to model a tank-tuned VCO.

Ports

Input

vctrl — Voltage used to control VCO output frequency

scalar | vector

VCO control voltage used to control the output frequency of the ring oscillator VCO. In a phase-locked loop (PLL) system, **vctrl** is the output of the Loop Filter that contains the phase error information.

Data Types: `double`

Output

vco out — Output signal determined by **vctrl** port

scalar

Output signal of the ring oscillator VCO. In a PLL system, **vco out** is the output clock generated by the PLL. It is also fed back to the PFD block through a clock divider to complete the control loop.

Data Types: `double`

Parameters

Parameters

Specify using — Define how VCO output frequency is specified

Voltage sensitivity (default) | Output frequency vs. control voltage

Define how the VCO output frequency is specified:

- Select **Voltage sensitivity** to specify output frequency from the **Voltage sensitivity (Hz/V)** and **Free running frequency (Hz)** parameters.
- Select **Output frequency vs. control voltage** to interpolate output frequency from the **Control voltage (V)** vector versus **Output frequency (Hz)** vector.

Programmatic Use

Block parameter: SpecifyUsing

Type: character vector

Values: Voltage sensitivity | Output frequency vs. control voltage

Default: Voltage sensitivity

Voltage sensitivity (Hz/V) — Measure of change in output frequency of VCO

100e6 (default) | positive real scalar

Measure of change in output frequency for input voltage change, specified as a positive real scalar with units in Hz/V. This parameter is also reported as **VCO voltage sensitivity** in the **Loop Filter** tab and is used to automatically calculate the filter component values of the loop filter.

Dependencies

To enable this parameter, in the **Parameters** tab, set **Specify using** to **Voltage sensitivity**.

Programmatic Use

Block parameter: Kvco

Type: character vector

Values: positive real scalar

Default: 100e6

Data Types: double

Free running frequency (Hz) — VCO output frequency without control voltage

2.5e9 (default) | positive real scalar

Frequency of the VCO without any control voltage input (0 V) or the quiescent frequency, specified as a positive real scalar in hertz.

Dependencies

To enable this parameter, in the **Parameters** tab, set **Specify using** to **Voltage sensitivity**.

Programmatic Use

Block parameter: Fo

Type: character vector

Values: positive real scalar

Default: 2.5e9

Data Types: double

Control voltage (V) – Control voltage values

[-5 0 5] (default) | real valued vector

Control voltage values of the VCO, specified as a real valued vector in volts.

Dependencies

To enable this parameter, in the **Parameters** tab, set **Specify using** to Output frequency vs. control voltage.

Programmatic Use**Block parameter:** ControlVoltage**Type:** character vector**Values:** real valued vector**Default:** [-5 0 5]

Data Types: double

Output frequency (Hz) – VCO output frequency values

[2e9 2.5e9 3e9] (default) | positive real valued vector

Output frequency of the VCO corresponding to the **Control voltage (V)** vector, specified in hertz.

Dependencies

To enable this parameter, in the **Parameters** tab, set **Specify using** to Output frequency vs. control voltage.

Programmatic Use**Block parameter:** OutputFrequency**Type:** character vector**Values:** positive real valued vector**Default:** [2e9 2.5e9 3e9]

Data Types: double

Output amplitude (V) – Maximum amplitude of VCO output voltage

1 (default) | positive real scalar

Maximum amplitude of the VCO output voltage, specified as a positive real scalar.

Programmatic Use**Block parameter:** Amplitude**Type:** character vector**Values:** positive real scalar**Default:** 1

Data Types: double

Enable increased buffer size – Enable increased buffer size

off (default) | on

Select to enable increased buffer size during simulation. This increases the buffer size of the Variable Pulse Delay block inside the Ring Oscillator VCO block. By default, this option is deselected.

Buffer size – Number of samples of input buffering available during simulation

10 (default) | positive integer scalar

Number of samples of input buffering available during simulation, specified as a positive integer scalar. This sets the buffer size of the Variable Pulse Delay block inside the Ring Oscillator VCO block.

Selecting a different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

To enable this parameter, select **Enable increased buffer size**.

Programmatic Use

Block parameter: NBuffer

Type: character vector

Values: positive integer scalar

Default: 10

Data Types: double

Impairments

Add phase noise — Add phase noise as a function of frequency

on (default) | off

Select to introduce phase noise as a function of frequency to the VCO. By default, this option is selected.

Phase noise frequency offset (Hz) — Frequency offsets of specified phase noise from carrier frequency

[30e3 100e3 1e6 3e6 10e6] (default) | positive real valued vector

The frequency offsets of the specified phase noise from the carrier frequency, specified as a positive real valued vector in hertz.

Dependencies

To enable this parameter, select **Add phase noise** in the **Impairments** tab.

Programmatic Use

Block parameter: Foffset

Type: character vector

Values: positive real valued vector

Default: [30e3 100e3 1e6 3e6 10e6]

Data Types: double

Phase noise level (dBc/Hz) — Specified phase noise power at phase noise frequency offsets relative to the carrier

[-56 -106 -132 -143 -152] (default) | negative real valued vector

The specified phase noise power in a 1 Hz bandwidth centered at the phase noise frequency offsets relative to the carrier, specified as a negative real valued vector in dBc/Hz. The elements of **Phase noise level** correspond to relative elements in the **Phase noise frequency offset (Hz)** parameter.

Dependencies

To enable this parameter, select **Add phase noise** in the **Impairments** tab.

Programmatic Use**Block parameter:** PhaseNoise**Type:** character vector**Values:** negative real valued vector**Default:** [-56 -106 -132 -143 -152]

Data Types: double

Estimate phase noise parameters — Set noise parameters intended to match specified noise spectrum

button

Click to set the noise parameters to an initial estimate intended to match the specified noise spectrum.

Period jitter (S) — Standard deviation of period jitter

1.7e-15 (default) | positive real scalar

Standard deviation of the period jitter, specified as a positive real scalar in seconds. Period jitter is the deviation in cycle time of a clock signal with respect to the ideal period.

Programmatic Use**Block parameter:** PeriodJitter**Type:** character vector**Values:** positive real scalar**Default:** 1.7e-15**Flicker corner frequency (Hz) — Corner frequency of flicker noise**

5e5 (default) | scalar

Corner frequency of the flicker noise, specified as a scalar in hertz. **Flicker corner frequency (Hz)** is defined as the frequency at which the phase noise transitions from $1/f^2$ to $1/f^3$ due to flicker noise. At this frequency, the spectral densities of period jitter and flicker noise are equal.

Programmatic Use**Block parameter:** CornerFrequency**Type:** character vector**Values:** scalar**Default:** 5e5**Customize flicker exponent (Advanced feature) — Customize flicker noise power spectral distribution**

off (default) | on

Select this parameter to customize the power spectral distribution of the flicker noise. Traditionally, flicker noise is defined as the $1/f$ noise, but it can vary as $1/f^V$, where $0.8 < V < 1.5$.

Flicker exponent — Flicker noise power exponent

1.0 (default) | 0.8 | 0.9 | 1.1 | 1.2 | 1.3 | 1.4 | 1.5

Flicker noise power exponent, specified between 0.8 to 1.5.

Programmatic Use**Block parameter:** FlickerExponent**Type:** character vector

Values: 1.0 | 0.8 | 0.9 | 1.1 | 1.2 | 1.3 | 1.4 | 1.5

Default: 1.0

Plot fit — Plot specified and expected output phase noise density

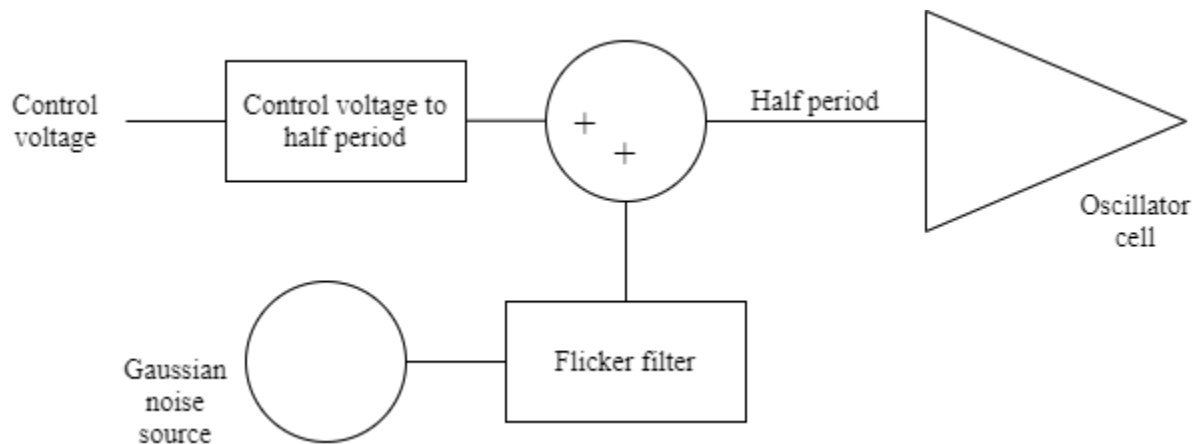
button

Click to plot the specified phase noise density and expected output phase noise density.

More About

Phase Noise in Ring Oscillator VCO

The Ring Oscillator VCO block generates the phase noise with the help of a Gaussian noise source and a flicker filter.



When enable the phase noise, the phase noise is calculated from the variance of the period offset stochastic process from the spectral density at a single frequency. Given the oscillation frequency f_0 , offset frequency f , and single sideband spectral density $\mathcal{L}(f)$, the variance of the period offset is:

$$\sigma_T^2 = \mathcal{L}(f) \frac{f^2}{f_0^3}$$

Thus, except for flicker noise, you need a period offset derived from an uncorrelated random process with a constant variance. This period offset is generated by the block's Gaussian noise source.

To model flicker noise, the flicker filter introduces additional gain at low frequencies, down to four orders of magnitude below the flicker corner frequency. To increase the energy spectral density from $1/f^2$ to $1/f^3$, the flicker filter must introduce a voltage gain of $1/\sqrt{f}$ below the corner frequency while maintaining unity gain above the corner frequency. To achieve this, the flicker filter is a recursive digital filter with an alternating sequence of four poles and four zeros. The lowest frequency zero is a constant factor higher than the lowest frequency pole, the next higher frequency pole is the same constant factor higher than the lowest frequency zero, and that pattern of alternating poles and zeros is maintained through the remainder of the sequence. The constant factor is a function of the flicker filter exponent, with a factor of the square root of ten for the nominal case of $1/f$ flicker noise.

To minimize numerical noise in the flicker filter, the sample rate of the Gaussian noise source and flicker filter is limited to 20 times the highest phase noise specification offset frequency, unless that

sample rate is comparable to or greater than twice the frequency of oscillation. For higher phase specification noise offset frequencies, the sample rate is limited to twice the frequency of oscillation.

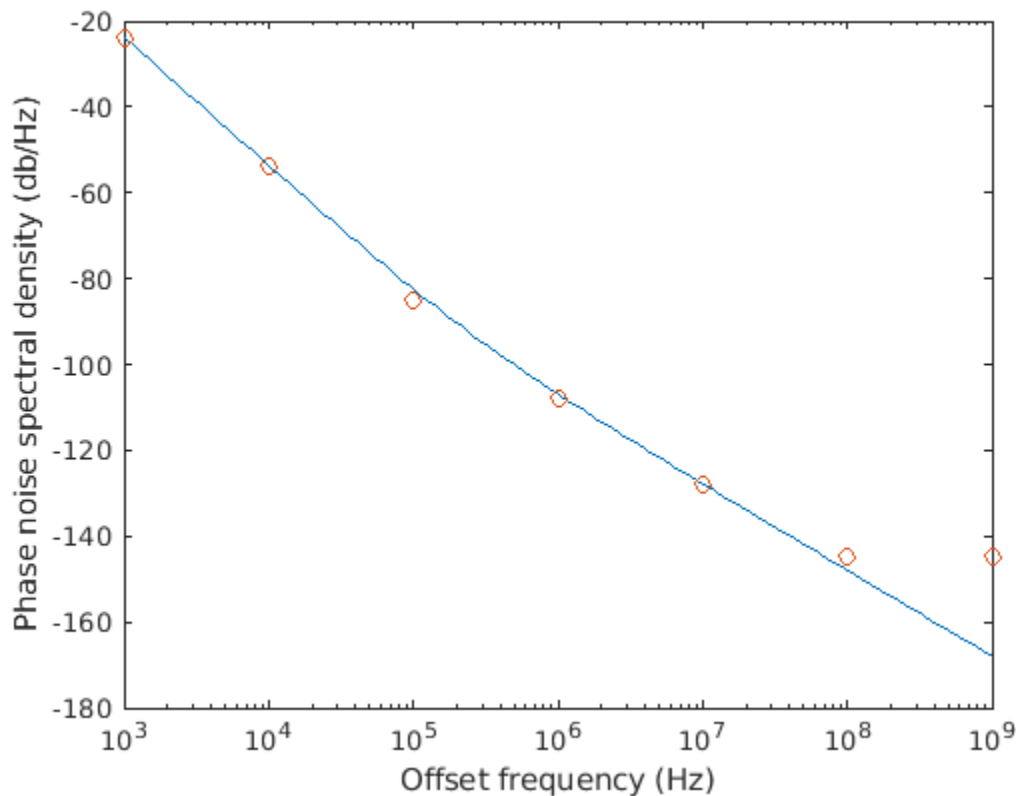
The variance of the Gaussian noise source is adjusted to compensate for the difference in sample rate between the noise source and the oscillator's frequency of oscillation.

When the phase noise is disabled, the variance of the Gaussian noise is set to zero.

Matching Measured Phase Noise to Physical Model

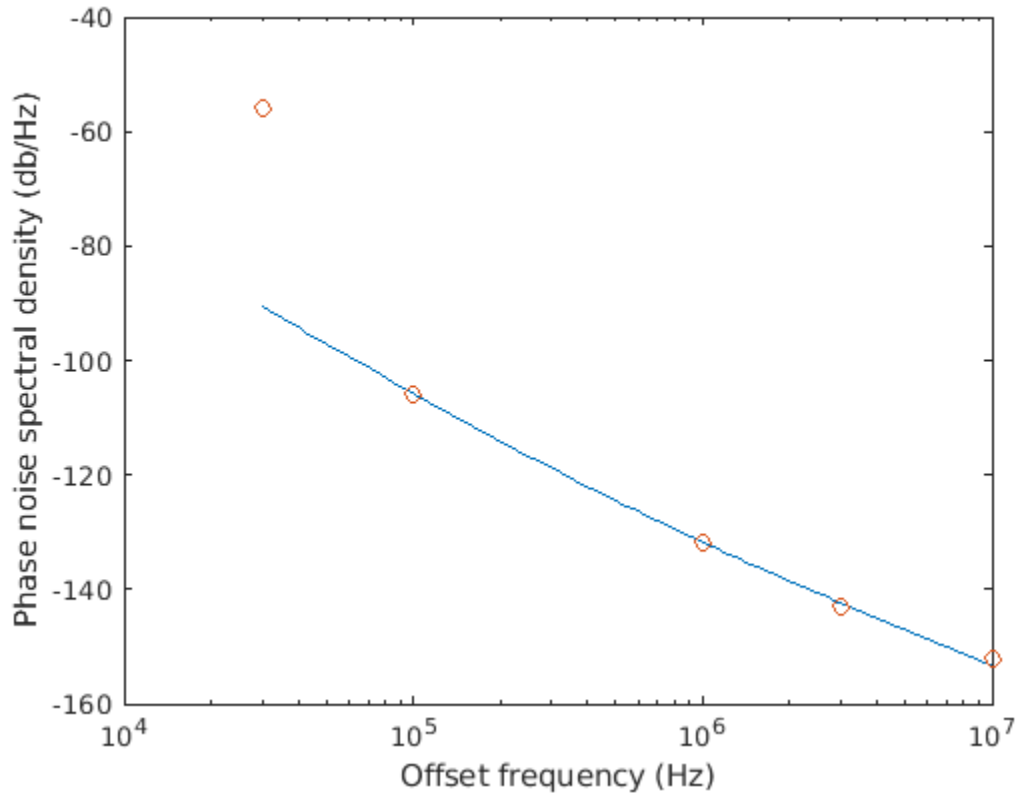
Specified phase noise spectra often include measurement artifacts that should not be included in the model of the VCO itself. While the parameter values produced by the **Estimate phase noise parameters** button will often come close to creating an appropriate physical model, there will be cases that require your judgement. Two common problems are noise floor and resolution bandwidth.

Sometimes, the measured phase noise appears to match a physical model up to frequencies for which the phase noise may be below the noise floor of the measurement.



It appears that there is a measurement noise floor at -140 dBc/Hz for the specification dataset, and the phase noise for the device under test is probably below that noise floor for offset frequencies above 100 MHz. In this case, a set of parameter values that fit best to the data at lower frequency offsets is likely to produce a more accurate model.

Sometimes, the measured phase noise appears to match a physical model at all levels except the lowest frequency offset.



The most likely cause for a result such as this is that the resolution bandwidth used to make the measurement was too large to produce an accurate measurement at the lowest frequency offset, 30 kHz in this case. Even if the carrier was outside the passband of the measurement filter, the carrier energy passed through the limited stopband rejection of the measurement filter was much larger than the energy in the passband. In this case, a set of parameter values that fit best to the data at higher frequency offsets is likely to produce a more accurate model.

Version History

Introduced in R2021a

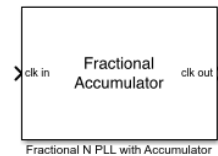
See Also

Loop Filter | PFD | VCO Testbench | VCO

Fractional N PLL with Accumulator

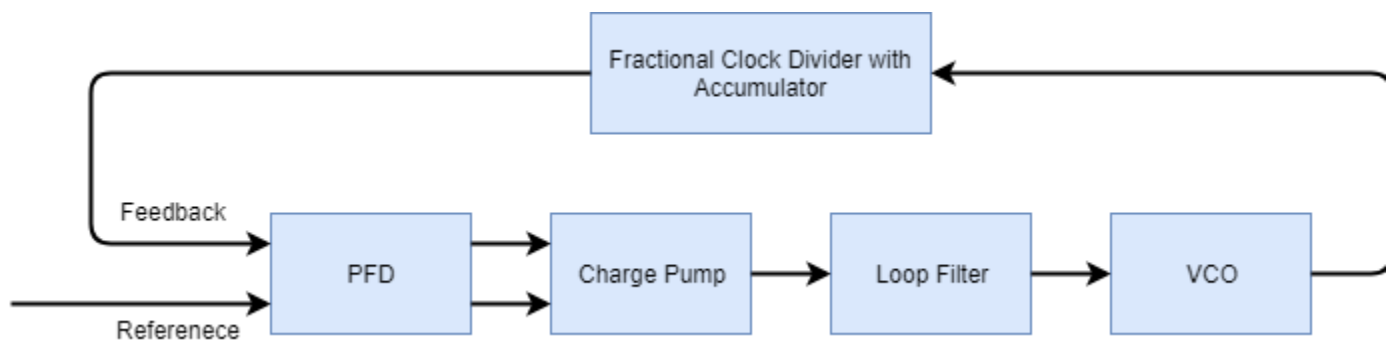
Frequency synthesizer with accumulator based fractional N PLL architecture

Library: Mixed-Signal Blockset / PLL / Architectures



Description

The Fractional N PLL with Accumulator reference architecture uses a Fractional Clock Divider with Accumulator block as the frequency divider in a PLL system. The frequency divider divides the frequency of the VCO output signal by a fractional value to make it comparable to a PFD reference signal frequency.



Ports

Input

clk in – Input clock signal

scalar

Input clock signal, specified as a scalar. The signal at the **clk in** port is used as the reference signal for the PFD block in a PLL system.

Data Types: double

Output

clk out – Output clock signal

scalar

Output clock signal, specified as a scalar. The signal at the **clk out** port is the output of the VCO block in a PLL system.

Data Types: double

Parameters

Enable increased buffer size — Enable increased buffer size

button

Select to enable increased buffer size during the simulation. This increases the buffer size of all the building blocks in the PLL model that belong to the Mixed-Signal Blockset/PLL/Building Blocks Simulink library. The building blocks are PFD, Charge Pump, Loop Filter, VCO, and Fractional Clock Divider with Accumulator. By default, this option is deselected.

Buffer size for loop filter — Buffer size for loop filter

1000 (default) | positive integer scalar

Buffer size for the loop filter, specified as a positive integer scalar. This sets the number of extra buffer samples available during the simulation to the Convert Sample Time subsystem inside the loop filter.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size for loop filter** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when the **Enable increased buffer size** option is selected.

Programmatic Use

- Use `get_param(gcb, 'NBufferFilter')` to view the current value of **Buffer size for loop filter**.
- Use `set_param(gcb, 'NBufferFilter', value)` to set **Buffer size for loop filter** to a specific value.

Buffer size for PFD, charge pump, VCO, prescaler — Buffer size for PFD, charge pump, VCO, and prescaler

10 (default) | positive integer scalar

Buffer size for the PFD, charge pump, VCO, and prescaler, specified as a positive integer scalar. This sets the buffer size of the PFD, Charge Pump, VCO, and Fractional Clock Divider with Accumulator blocks inside the PLL model.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size for PFD, charge pump, VCO, prescaler** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when the **Enable increased buffer size** option is selected.

Programmatic Use

- Use `get_param(gcb, 'NBuffer')` to view the current value of **Buffer size for PFD, charge pump, VCO, prescaler**.
- Use `set_param(gcb, 'NBuffer', value)` to set **Buffer size for PFD, charge pump, VCO, prescaler** to a specific value.

PFD

Configuration

Deadband compensation (s) — Delay added for active output near zero phase offset

40e-12 (default) | positive real scalar

Delay added for active output near zero phase offset, specified as a positive real scalar in seconds. Deadband is the phase offset band near zero phase offset for which the PFD output is negligible.

Programmatic Use

- Use `get_param(gcb, 'DeadbandCompensation')` to view the current value of **Deadband compensation (s)**.
- Use `set_param(gcb, 'DeadbandCompensation', value)` to set **Deadband compensation (s)** to a specific value.

Data Types: double

Impairments

Enable impairments — Add circuit impairments to simulation

off (default) | on

Select to add circuit impairments such as rise/fall time and propagation delay to simulation. By default, this option is deselected.

Output step size calculation — Determine how output step size is calculated

Default (default) | Advanced

Determine how output step size is calculated:

- Select **Default** to calculate output step size from rise/fall time. Output step size (ΔT) is given by

$$\Delta T = \frac{(\text{Rise/fall time})^2}{6 \cdot 0.22}$$
- Select **Advanced** to calculate output step size from maximum frequency of interest. Output step size (ΔT) is given by $\Delta T = \frac{\text{Rise/fall time}}{6 \cdot \text{Maximum frequency of interest}}$

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab.

Maximum frequency of interest (Hz) — Maximum frequency of interest at output

10e9 (default) | positive real scalar

Maximum frequency of interest at the output, specified as a positive real scalar in Hz.

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab and choose **Advanced** for **Output step size calculation**.

Programmatic Use

- Use `get_param(gcb, 'MaxFreqInterest')` to view the current value of **Maximum frequency of interest (Hz)**.

- Use `set_param(gcb, 'MaxFreqInterest', value)` to set **Maximum frequency of interest (Hz)** to a specific value.

Data Types: double

Rise/fall time (s) — 20% - 80% rise/fall time for up output port of PFD

3e-11 (default) | positive real scalar

20% - 80% rise/fall time for the up output port of the PFD, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab.

Programmatic Use

- Use `get_param(gcb, 'RiseFallTime')` to view the current value of **Rise/fall time (s)**.
- Use `set_param(gcb, 'RiseFallTime', value)` to set **Rise/fall time (s)** to a specific value.

Data Types: double

Propagation Delay (s) — Delay from input port to output port of PFD

50e-12 (default) | positive real scalar

Delay from the input port to output port of the PFD, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelay')` to view the current value of **Propagation Delay (s)**.
- Use `set_param(gcb, 'PropDelay', value)` to set **Propagation Delay (s)** to a specific value.

Data Types: double

Charge pump**Configuration****Output current (A) — Design output current**

1e-3 (default) | positive real scalar

Full scale magnitude of design output current, specified as a positive real scalar in amperes. This parameter is also reported as **Charge pump current** in the **Loop Filter** tab and is used to automatically calculate the filter component values of the loop filter.

Programmatic Use

- Use `get_param(gcb, 'OutputCurrent')` to view the current value of **Output current (A)**.
- Use `set_param(gcb, 'OutputCurrent', value)` to set **Output current (A)** to a specific value.

Data Types: double

Input threshold (V) — Logic switching threshold at input ports

0.5 (default) | real scalar

Logic switching threshold at input ports, specified as a real scalar in volts.

Programmatic Use

- Use `get_param(gcb, 'InputThreshold')` to view the current value of **Input threshold (V)**.
- Use `set_param(gcb, 'InputThreshold', value)` to set **Input threshold (V)** to a specific value.

Data Types: double

Impairments

Enable current impairments — Add current impairments to simulation

off (default) | on

Select to add current impairments such as current imbalance and leakage current to simulation. By default, this option is deselected.

Current imbalance (A) — Difference between full scale positive and negative current

1e-7 (default) | positive real scalar

Difference between full scale positive and negative current, specified as a positive real scalar in amperes.

Dependencies

To enable this parameter, select **Enable current impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'CurrentImbalance')` to view the current value of **Current imbalance (A)**.
- Use `set_param(gcb, 'CurrentImbalance', value)` to set **Current imbalance (A)** to a specific value.

Data Types: double

Leakage current (A) — Output current without any input

1e-8 (default) | nonnegative real scalar

Output current when both inputs are at logic zero, specified as a nonnegative real scalar in amperes.

Dependencies

To enable this parameter, select **Enable current impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'LeakageCurrent')` to view the current value of **Leakage current (A)**.
- Use `set_param(gcb, 'LeakageCurrent', value)` to set **Leakage current (A)** to a specific value.

Data Types: double

Enable timing impairments — Add timing impairments to simulation

off (default) | on

Select to add timing impairments such as rise/fall time and propagation delay to simulation. By default, this option is deselected.

Output step size calculation – Determine how output step size is calculated

Default (default) | Advanced

Determine how output step size is calculated:

- Select **Default** to calculate output step size from rise/fall time. Output step size (ΔT) is given by
$$\Delta T = \frac{(\text{Rise/fall time})^2}{6 \cdot 0.22}$$
.
- Select **Advanced** to calculate output step size from maximum frequency of interest. Output step size (ΔT) is given by
$$\Delta T = \frac{\text{Rise/fall time}}{6 \cdot \text{Maximum frequency of interest}}$$
.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge Pump** tab.

Maximum frequency of interest (Hz) – Maximum frequency of interest at output

10e9 (default) | positive real scalar

Maximum frequency of interest at the output, specified as a positive real scalar in Hz.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge Pump** tab and choose **Advanced** for **Output step size calculation**.

Programmatic Use

- Use `get_param(gcb, 'MaxFreqInterestCp')` to view the current value of **Maximum frequency of interest (Hz)**.
- Use `set_param(gcb, 'MaxFreqInterestCp', value)` to set **Maximum frequency of interest (Hz)** to a specific value.

Data Types: double

Up

Rise/fall time (s) – 20% - 80% rise/fall time for up input port

5e-9 (default) | positive real scalar

20% - 80% rise/fall time for the up input port of the charge pump, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'RiseFallUp')` to view the current value of **Up Rise/fall time (s)**.
- Use `set_param(gcb, 'RiseFallUp', value)` to set **Up Rise/fall time (s)** to a specific value.

Data Types: double

Propagation delay (s) — Total propagation delay from up input port to output port of charge pump

6e-9 (default) | positive real scalar

Total propagation delay from the up input port to output port of the charge pump, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelayUp')` to view the current value of **Up Propagation delay (s)**.
- Use `set_param(gcb, 'PropDelayUp', value)` to set **Up Propagation delay (s)** to a specific value.

Data Types: double

Down**Rise/fall time — 20% - 80% rise/fall time for down input port**

2e-9 (default) | scalar

20% - 80% rise/fall time for down input port of charge pump.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'RiseFallDown')` to view the current value of **Down Rise/fall time (s)**.
- Use `set_param(gcb, 'RiseFallDown', value)` to set **Down Rise/fall time (s)** to a specific value.

Data Types: double

Propagation delay (s) — Total propagation delay from up input port to output port of charge pump

4e-9 (default) | positive real scalar

Total propagation delay from the up input port to output port of the charge pump, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelayUp')` to view the current value of **Down Propagation delay (s)**.
- Use `set_param(gcb, 'PropDelayUp', value)` to set **Down Propagation delay (s)** to a specific value.

Data Types: double

VCO

Specify using — Define how VCO output frequency is specified

Voltage sensitivity (default) | Output frequency vs. control voltage

Define how VCO output frequency is specified:

- Select **Voltage sensitivity** to specify output frequency from **Voltage sensitivity (Hz/V)** and **Free running frequency (Hz)**.
- Select **Output frequency vs. control voltage** to interpolate output frequency from **Control voltage (V)** vector versus **Output frequency (Hz)** vector.

Programmatic Use

- Use `set_param(gcb, 'SpecifyUsing', 'Voltage sensitivity')` to set **Specify using** to **Voltage sensitivity**.
- Use `set_param(gcb, 'SpecifyUsing', 'Output frequency vs. control voltage')` to set **Specify using** to **Output frequency vs. control voltage**.

Voltage sensitivity (Hz/V) — Measure of change in output frequency of VCO

100e6 (default) | positive real scalar

Measure of change in output frequency for input voltage change, specified as a positive real scalar with units in Hz/V. This parameter is also reported as **VCO voltage sensitivity** in the **Loop Filter** tab and is used to automatically calculate the filter component values of the loop filter.

Dependencies

To enable this parameter, select **Voltage sensitivity** in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'Kvco')` to view the current **Voltage sensitivity (Hz/V)** value.
- Use `set_param(gcb, 'Kvco', value)` to set **Voltage sensitivity (Hz/V)** to a specific value.

Data Types: double

Free running frequency (Hz) — VCO output frequency without control voltage

1.8e9 (default) | positive real scalar

Frequency of the VCO without any control voltage input (0 V), or the quiescent frequency, specified as a positive real scalar in Hz.

Dependencies

To enable this parameter, select **Voltage sensitivity** in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'Fo')` to view current **Free running frequency (Hz)** value.
- Use `set_param(gcb, 'Fo', value)` to set **Free running frequency (Hz)** to a specific value.

Data Types: double

Control voltage (V) — Control voltage values

[-5 0 5] (default) | real valued vector

Control voltage values of the VCO, specified as a real valued vector in volts.

Dependencies

To enable this parameter, select Output frequency vs. control voltage in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'ControlVoltage')` to view current **Control voltage (V)** value.
- Use `set_param(gcb, 'ControlVoltage', value)` to set **Control voltage (V)** to a specific value.

Data Types: double

Output frequency (Hz) — VCO output frequency values

[2e9 2.5e9 3e9] (default) | real valued vector

Output frequency of the values of the VCO, corresponding to the **Control voltage (V)** vector, specified in Hz.

Dependencies

To enable this parameter, select Output frequency vs. control voltage in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'OutputFrequency')` to view current **Output frequency (Hz)** value.
- Use `set_param(gcb, 'OutputFrequency', value)` to set **Output frequency (Hz)** to a specific value.

Data Types: double

Output amplitude gain — Ratio of VCO output voltage to input voltage

1 (default) | positive real scalar

Ratio of VCO output voltage to input voltage, specified as a positive real scalar. The input voltage has a nontunable value of 1 V.

Programmatic Use

- Use `get_param(gcb, 'Amplitude')` to view current **Output amplitude gain** value.
- Use `set_param(gcb, 'Amplitude', value)` to set **Output amplitude gain** to a specific value.

Data Types: double

Impairment

Add Phase-noise — Add phase noise as function of frequency

off (default) | on

Select to introduce phase noise as a function of frequency to the VCO. By default, this option is deselected.

Phase noise frequency offset (Hz) — Frequency offsets of phase noise from carrier frequency

[30e3 100e3 1e6 3e6 10e6] (default) | real valued vector

Frequency offsets of the phase noise from the carrier frequency, specified as a real valued vector in Hz.

Dependencies

To enable this parameter, select **Add phase noise** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'Foffset')` to view the current **Phase noise frequency offset (Hz)** metric.
- Use `set_param(gcb, 'Foffset', value)` to set **Phase noise frequency offset (Hz)** to a specific metric.

Data Types: `double`

Phase noise level (dBc/Hz) — Phase noise power at specified frequency offsets relative to the carrier

`[-56 -106 -132 -143 -152]` (default) | real valued vector

Real valued vector specifying the phase noise power in a 1 Hz bandwidth centered at the specified frequency offsets relative to the carrier. The value is specified in dBc/Hz.

Dependencies

To enable this parameter, select **Add phase noise** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'PhaseNoise')` to view the current **Phase noise level (dBc/Hz)** metric.
- Use `set_param(gcb, 'PhaseNoise', value)` to set **Phase noise level (dBc/Hz)** to a specific metric.

Data Types: `double`

Prescaler

Fractional clock divider value — Value by which the clock divider divides the input frequency

`70.20` (default) | positive real scalar

Value by which the clock divider divides the input frequency, specified as a positive real scalar.

Programmatic Use

- Use `get_param(gcb, 'N')` to view the current value of **Fractional clock divider value**.
- Use `set_param(gcb, 'N', value)` to set **Fractional clock divider value** to a specific value.

Min clock divider value — Minimum value by which clock divider can divide input frequency

`70` (default) | positive real scalar

Minimum value by which the clock divider can divide input frequency, specified as a positive real scalar. This parameter is also reported in the **Loop Filter** tab and is used to automatically calculate the filter component values of the loop filter.

Programmatic Use

- Use `get_param(gcb, 'Nmin')` to view the current value of **Min clock divider value**.
- Use `set_param(gcb, 'Nmin', value)` to set **Min clock divider value** to a specific value.

Loop Filter**Filter component values — Determines how filter components are computed**

Automatic (default) | Manual

Select how filter components for the loop filter are computed:

- Select **Automatic** to automatically compute filter components from system specifications. Resistance and capacitance edit boxes in the **Loop Filter** tab are not editable if this option is selected. Rather, the filter component values are calculated from **Loop bandwidth (Hz)**, **Phase margin (degrees)**, **VCO voltage sensitivity**, **Charge pump current**, and **Min clock divider value**. By default, this option is selected.
- Select **Manual** to manually enter the resistance and capacitance values to design a customized loop filter.

Loop bandwidth (Hz) — Frequency at which magnitude of open loop transfer function becomes 1

0.5e6 (default) | positive real scalar

Frequency at which the magnitude of the open loop transfer function becomes 1, specified as a positive real scalar in Hz. Lower values of **Loop bandwidth (Hz)** result in reduced phase noise and reference spurs at the expense of longer lock time and less phase margin.

Dependencies

This parameter is only available when **Automatic** is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'Fc')` to view the current value of **Loop bandwidth (Hz)**.
- Use `set_param(gcb, 'Fc', value)` to set **Loop bandwidth (Hz)** to a specific value.

Phase margin (degrees) — Phase of open loop transfer function at loop bandwidth subtracted from 180°

45 (default) | positive real scalar

Phase of the open loop transfer function at the loop bandwidth subtracted from 180°, specified as a positive real scalar in degrees. For optimum lock time, select a phase margin between 40° and 55°.

Dependencies

This parameter is only available when **Automatic** is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'Phi')` to view the current value of **Phase margin (degrees)**.
- Use `set_param(gcb, 'Phi', value)` to set **Phase margin (degrees)** to a specific value.

Data Types: double

Loop filter type – Order of the loop filter

3rd Order Passive (default) | 2nd Order Passive | 4th Order Passive

Order of the loop filter. Applies a second-, third-, or fourth-order passive RC loop filter in the PLL system.

C1 (F) – Capacitance 1

3e-14 (default) | positive real scalar

Capacitor value C1, specified as a positive real scalar in farad.

Dependencies

This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C1')` to view the current value of **C1 (F)**.
- Use `set_param(gcb, 'C1', value)` to set **C1 (F)** to a specific value.

Data Types: double

C2 (F) – Capacitance 2

3.3e-13 (default) | positive real scalar

Capacitor value C2, specified as a positive real scalar in farad.

Dependencies

This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C2')` to view the current value of **C2 (F)**.
- Use `set_param(gcb, 'C2', value)` to set **C2 (F)** to a specific value.

Data Types: double

C3 (F) – Capacitance 3

2.15e-15 (default) | positive real scalar

Capacitor value C3, specified as a positive real scalar in farad.

Dependencies

- To enable this parameter, select 3rd Order Passive or 4th Order Passive in **Loop filter type**.
- This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C3')` to view the current value of **C3 (F)**.
- Use `set_param(gcb, 'C3', value)` to set **C3 (F)** to a specific value.

Data Types: double

C4 (F) – Capacitance 4

1e-12 (default) | positive real scalar

Capacitor value C4, specified as a positive real scalar in farad.

Dependencies

- To enable this parameter, select 4th Order Passive in **Loop filter type**.
- This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C4')` to view the current value of **C4 (F)**.
- Use `set_param(gcb, 'C4', value)` to set **C4 (F)** to a specific value.

Data Types: double

R2 (ohms) – Resistance 2

2.33e+06 (default) | positive real scalar

Resistor value R2, specified as a positive real scalar in ohms.

Dependencies

This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'R2')` to view the current value of **R2 (ohms)**.
- Use `set_param(gcb, 'R2', value)` to set **R2 (ohms)** to a specific value.

Data Types: double

R3 (ohms) – Resistance 3

2.98e+07 (default) | positive real scalar

Resistor value R3, specified as a positive real scalar in ohms.

Dependencies

- To enable this parameter, select 3rd Order Passive or 4th Order Passive in **Loop filter type**.
- This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'R3')` to view the current value of **R3 (ohms)**.
- Use `set_param(gcb, 'R3', value)` to set **R3 (ohms)** to a specific value.

Data Types: double

R4 (ohms) – Resistance 4

12e3 (default) | positive real scalar

Resistor value R4, specified as a positive real scalar in ohms.

Dependencies

- To enable this parameter, select 4th Order Passive in **Loop filter type**.
- This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'R4')` to view the current value of **R4 (ohms)**.
- Use `set_param(gcb, 'R4', value)` to set **R4 (ohms)** to a specific value.

Data Types: double

Enable impairments – Add circuit impairments to simulation

off (default) | on

Select to add circuit impairments such as operating temperature to determine thermal noise to simulation. By default, this option is deselected.

Operating temperature (°C) – Temperature to determine the level of thermal noise

30 (default) | real scalar

Temperature of the resistor, specified as a real scalar in °C. **Operating temperature** determines the level of thermal (Johnson) noise.

Dependencies

To enable this parameter, select **Enable impairments** in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'Temperature')` to view the current value of **Operating temperature**.
- Use `set_param(gcb, 'Temperature', value)` to set **Operating temperature** to a specific value.

Data Types: double

Export Loop Filter Component Values – Export loop filter component values

button

Click to export loop filter component values to a spreadsheet (XLS file) or as comma-separated values (CSV file).

Probe**PFD up and PFD down (pfd_up and pfd_down) – Select to probe PFD outputs**

off (default) | on

Select to probe the PFD output wires (pfd_up and pfd_down) to view the response of the PFD.

Charge pump output (cp_out) – Select to probe charge pump output

off (default) | on

Select to probe the charge pump output wire (cp_out) to view the response of the Charge Pump.

Loop filter output (lf_out) — Select to probe loop filter output

off (default) | on

Select to probe loop filter output wire (lf_out) to view the response of the Loop Filter. The loop filter output provides the control voltage to the VCO.

Prescaler output (ps_out) — Select to probe prescaler output

off (default) | on

Select to probe the prescaler output wire (ps_out) to view the response of the Fractional Clock Divider with Accumulator.

Analysis

Open Loop Analysis — Plot the presimulation open loop analysis

on (default) | off

Select to plot the gain margin and phase margin of the PLL system before simulation. By default, this option is selected.

Closed Loop Analysis — Plot the presimulation closed loop analysis

off (default) | on

Select to plot the pole-zero map, loop bandwidth, step response, and impulse response of the PLL system before simulation. You must have a license to Control System Toolbox™ to plot the step response and impulse response of the PLL system. By default, this option is deselected.

Plot Loop Dynamics — Plot loop dynamics of PLL system

button

Click to plot the presimulation loop dynamics of the PLL system.

Version History

Introduced in R2019a

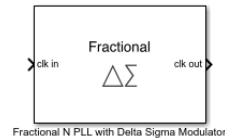
See Also

PFD | Charge Pump | Loop Filter | Fractional Clock Divider with Accumulator | VCO

Fractional N PLL with Delta Sigma Modulator

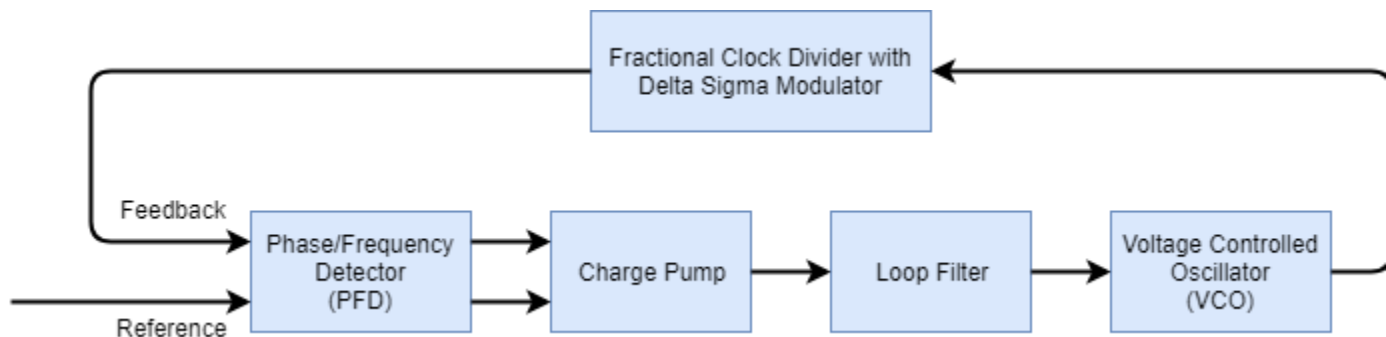
Frequency synthesizer with delta sigma modulator based fractional N PLL architecture

Library: Mixed-Signal Blockset / PLL / Architectures



Description

The Fractional N PLL with Delta Sigma Modulator reference architecture uses a Fractional Clock Divider with DSM block as the frequency divider in a PLL system. The frequency divider divides the frequency of the VCO output signal by a fractional value using the delta sigma modulation technique to make it comparable to a PFD reference signal frequency.



Ports

Input

clk in – Input clock signal

scalar

Input clock signal, specified as a scalar. The signal at the **clk in** port is used as the reference signal for the PFD block in a PLL system.

Data Types: double

Output

clk out – Output clock signal

scalar

Output clock signal, specified as a scalar. The signal at the **clk out** port is the output of the VCO block in a PLL system.

Data Types: double

Parameters

Enable increased buffer size – Enable increased buffer size

button

Select to enable increased buffer size during the simulation. This increases the buffer size of all the building blocks in the PLL model that belong to the Mixed-Signal Blockset/PLL/Building Blocks Simulink library. The building blocks are PFD, Charge Pump, Loop Filter, VCO, and Fractional Clock Divider with DSM. By default, this option is deselected.

Buffer size for loop filter – Buffer size for loop filter

1000 (default) | positive integer scalar

Buffer size for the loop filter, specified as a positive integer scalar. This sets the number of extra buffer samples available during the simulation to the Convert Sample Time subsystem inside the loop filter.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size for loop filter** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when the **Enable increased buffer size** option is selected.

Programmatic Use

- Use `get_param(gcb, 'NBufferFilter')` to view the current value of **Buffer size for loop filter**.
- Use `set_param(gcb, 'NBufferFilter', value)` to set **Buffer size for loop filter** to a specific value.

Buffer size for PFD, charge pump, VCO, prescaler – Buffer size for PFD, charge pump, VCO, and prescaler

10 (default) | positive integer scalar

Buffer size for the PFD, charge pump, VCO, and prescaler, specified as a positive integer scalar. This sets the buffer size of the PFD, Charge Pump, VCO, and Fractional Clock Divider with DSM blocks inside the PLL model.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size for PFD, charge pump, VCO, prescaler** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when the **Enable increased buffer size** option is selected.

Programmatic Use

- Use `get_param(gcb, 'NBuffer')` to view the current value of **Buffer size for PFD, charge pump, VCO, prescaler**.
- Use `set_param(gcb, 'NBuffer', value)` to set **Buffer size for PFD, charge pump, VCO, prescaler** to a specific value.

PFD**Configuration****Deadband compensation (s) – Delay added for active output near zero phase offset**

40e-12 (default) | positive real scalar

Delay added for active output near zero phase offset, specified as a positive real scalar in seconds. Deadband is the phase offset band near zero phase offset for which the PFD output is negligible.

Programmatic Use

- Use `get_param(gcb, 'DeadbandCompensation')` to view the current value of **Deadband compensation (s)**.
- Use `set_param(gcb, 'DeadbandCompensation', value)` to set **Deadband compensation (s)** to a specific value.

Data Types: double

Impairments**Enable impairments – Add circuit impairments to simulation**

off (default) | on

Select to add circuit impairments such as rise/fall time and propagation delay to simulation. By default, this option is deselected.

Output step size calculation – Determine how output step size is calculated

Default (default) | Advanced

Determine how output step size is calculated:

- Select **Default** to calculate output step size from rise/fall time. Output step size (ΔT) is given by

$$\Delta T = \frac{(\text{Rise/fall time})^2}{6 \cdot 0.22}$$
- Select **Advanced** to calculate output step size from maximum frequency of interest. Output step size (ΔT) is given by $\Delta T = \frac{\text{Rise/fall time}}{6 \cdot \text{Maximum frequency of interest}}$

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab.

Maximum frequency of interest (Hz) – Maximum frequency of interest at output

10e9 (default) | positive real scalar

Maximum frequency of interest at the output, specified as a positive real scalar in Hz.

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab and choose **Advanced** for **Output step size calculation**.

Programmatic Use

- Use `get_param(gcb, 'MaxFreqInterest')` to view the current value of **Maximum frequency of interest (Hz)**.

- Use `set_param(gcb, 'MaxFreqInterest', value)` to set **Maximum frequency of interest (Hz)** to a specific value.

Data Types: double

Rise/fall time (s) — 20% - 80% rise/fall time for up output port of PFD

3e-11 (default) | positive real scalar

20% - 80% rise/fall time for the up output port of the PFD, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab.

Programmatic Use

- Use `get_param(gcb, 'RiseFallTime')` to view the current value of **Rise/fall time (s)**.
- Use `set_param(gcb, 'RiseFallTime', value)` to set **Rise/fall time (s)** to a specific value.

Data Types: double

Propagation Delay (s) — Delay from input port to output port of PFD

50e-12 (default) | positive real scalar

Delay from the input port to output port of the PFD, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelay')` to view the current value of **Propagation Delay (s)**.
- Use `set_param(gcb, 'PropDelay', value)` to set **Propagation Delay (s)** to a specific value.

Data Types: double

Charge pump

Configuration

Output current (A) — Design output current

1e-3 (default) | positive real scalar

Full scale magnitude of design output current, specified as a positive real scalar in amperes. This parameter is also reported as **Charge pump current** in the **Loop Filter** tab and is used to automatically calculate the filter component values of the loop filter.

Programmatic Use

- Use `get_param(gcb, 'OutputCurrent')` to view the current value of **Output current (A)**.
- Use `set_param(gcb, 'OutputCurrent', value)` to set **Output current (A)** to a specific value.

Data Types: double

Input threshold (V) — Logic switching threshold at input ports

0.5 (default) | real scalar

Logic switching threshold at input ports, specified as a real scalar in volts.

Programmatic Use

- Use `get_param(gcb, 'InputThreshold')` to view the current value of **Input threshold (V)**.
- Use `set_param(gcb, 'InputThreshold', value)` to set **Input threshold (V)** to a specific value.

Data Types: double

Impairments**Enable current impairments — Add current impairments to simulation**

off (default) | on

Select to add current impairments such as current imbalance and leakage current to simulation. By default, this option is deselected.

Current imbalance (A) — Difference between full scale positive and negative current

1e-7 (default) | positive real scalar

Difference between full scale positive and negative current, specified as a positive real scalar in amperes.

Dependencies

To enable this parameter, select **Enable current impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'CurrentImbalance')` to view the current value of **Current imbalance (A)**.
- Use `set_param(gcb, 'CurrentImbalance', value)` to set **Current imbalance (A)** to a specific value.

Data Types: double

Leakage current (A) — Output current without any input

1e-8 (default) | nonnegative real scalar

Output current when both inputs are at logic zero, specified as a nonnegative real scalar in amperes.

Dependencies

To enable this parameter, select **Enable current impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'LeakageCurrent')` to view the current value of **Leakage current (A)**.
- Use `set_param(gcb, 'LeakageCurrent', value)` to set **Leakage current (A)** to a specific value.

Data Types: double

Enable timing impairments — Add timing impairments to simulation

off (default) | on

Select to add timing impairments such as rise/fall time and propagation delay to simulation. By default, this option is deselected.

Output step size calculation – Determine how output step size is calculated

Default (default) | Advanced

Determine how output step size is calculated:

- Select **Default** to calculate output step size from rise/fall time. Output step size (ΔT) is given by
$$\Delta T = \frac{(\text{Rise/fall time})^2}{6 \cdot 0.22}$$
.
- Select **Advanced** to calculate output step size from maximum frequency of interest. Output step size (ΔT) is given by
$$\Delta T = \frac{\text{Rise/fall time}}{6 \cdot \text{Maximum frequency of interest}}$$
.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge Pump** tab.

Maximum frequency of interest (Hz) – Maximum frequency of interest at output

10e9 (default) | positive real scalar

Maximum frequency of interest at the output, specified as a positive real scalar in Hz.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge Pump** tab and choose **Advanced** for **Output step size calculation**.

Programmatic Use

- Use `get_param(gcb, 'MaxFreqInterestCp')` to view the current value of **Maximum frequency of interest (Hz)**.
- Use `set_param(gcb, 'MaxFreqInterestCp', value)` to set **Maximum frequency of interest (Hz)** to a specific value.

Data Types: double

Up

Rise/fall time (s) – 20% - 80% rise/fall time for up input port

5e-9 (default) | positive real scalar

20% - 80% rise/fall time for the up input port of the charge pump, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'RiseFallUp')` to view the current value of **Up Rise/fall time (s)**.
- Use `set_param(gcb, 'RiseFallUp', value)` to set **Up Rise/fall time (s)** to a specific value.

Data Types: double

Propagation delay (s) — Total propagation delay from up input port to output port of charge pump

6e-9 (default) | positive real scalar

Total propagation delay from the up input port to output port of the charge pump, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelayUp')` to view the current value of **Up Propagation delay (s)**.
- Use `set_param(gcb, 'PropDelayUp', value)` to set **Up Propagation delay (s)** to a specific value.

Data Types: double

Down**Rise/fall time — 20% - 80% rise/fall time for down input port**

2e-9 (default) | scalar

20% - 80% rise/fall time for down input port of charge pump.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'RiseFallDown')` to view the current value of **Down Rise/fall time (s)**.
- Use `set_param(gcb, 'RiseFallDown', value)` to set **Down Rise/fall time (s)** to a specific value.

Data Types: double

Propagation delay (s) — Total propagation delay from up input port to output port of charge pump

4e-9 (default) | positive real scalar

Total propagation delay from the up input port to output port of the charge pump, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelayUp')` to view the current value of **Down Propagation delay (s)**.
- Use `set_param(gcb, 'PropDelayUp', value)` to set **Down Propagation delay (s)** to a specific value.

Data Types: double

VCO

Specify using — Define how VCO output frequency is specified

Voltage sensitivity (default) | Output frequency vs. control voltage

Define how VCO output frequency is specified:

- Select **Voltage sensitivity** to specify output frequency from **Voltage sensitivity (Hz/V)** and **Free running frequency (Hz)**.
- Select **Output frequency vs. control voltage** to interpolate output frequency from **Control voltage (V)** vector versus **Output frequency (Hz)** vector.

Programmatic Use

- Use `set_param(gcb, 'SpecifyUsing', 'Voltage sensitivity')` to set **Specify using** to **Voltage sensitivity**.
- Use `set_param(gcb, 'SpecifyUsing', 'Output frequency vs. control voltage')` to set **Specify using** to **Output frequency vs. control voltage**.

Voltage sensitivity (Hz/V) — Measure of change in output frequency of VCO

100e6 (default) | positive real scalar

Measure of change in output frequency for input voltage change, specified as a positive real scalar with units in Hz/V. This parameter is also reported as **VCO voltage sensitivity** in the **Loop Filter** tab and is used to automatically calculate the filter component values of the loop filter.

Dependencies

To enable this parameter, select **Voltage sensitivity** in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'Kvco')` to view the current **Voltage sensitivity (Hz/V)** value.
- Use `set_param(gcb, 'Kvco', value)` to set **Voltage sensitivity (Hz/V)** to a specific value.

Data Types: double

Free running frequency (Hz) — VCO output frequency without control voltage

1.8e9 (default) | positive real scalar

Frequency of the VCO without any control voltage input (0 V), or the quiescent frequency, specified as a positive real scalar in Hz.

Dependencies

To enable this parameter, select **Voltage sensitivity** in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'Fo')` to view current **Free running frequency (Hz)** value.
- Use `set_param(gcb, 'Fo', value)` to set **Free running frequency (Hz)** to a specific value.

Data Types: double

Control voltage (V) — Control voltage values

[-5 0 5] (default) | real valued vector

Control voltage values of the VCO, specified as a real valued vector in volts.

Dependencies

To enable this parameter, select Output frequency vs. control voltage in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'ControlVoltage')` to view current **Control voltage (V)** value.
- Use `set_param(gcb, 'ControlVoltage', value)` to set **Control voltage (V)** to a specific value.

Data Types: double

Output frequency (Hz) — VCO output frequency values

[2e9 2.5e9 3e9] (default) | real valued vector

Output frequency of the values of the VCO, corresponding to the **Control voltage (V)** vector, specified in Hz.

Dependencies

To enable this parameter, select Output frequency vs. control voltage in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'OutputFrequency')` to view current **Output frequency (Hz)** value.
- Use `set_param(gcb, 'OutputFrequency', value)` to set **Output frequency (Hz)** to a specific value.

Data Types: double

Output amplitude gain — Ratio of VCO output voltage to input voltage

1 (default) | positive real scalar

Ratio of VCO output voltage to input voltage, specified as a positive real scalar. The input voltage has a nontunable value of 1 V.

Programmatic Use

- Use `get_param(gcb, 'Amplitude')` to view current **Output amplitude gain** value.
- Use `set_param(gcb, 'Amplitude', value)` to set **Output amplitude gain** to a specific value.

Data Types: double

Impairment

Add Phase-noise — Add phase noise as function of frequency

off (default) | on

Select to introduce phase noise as a function of frequency to the VCO. By default, this option is deselected.

Phase noise frequency offset (Hz) — Frequency offsets of phase noise from carrier frequency

[30e3 100e3 1e6 3e6 10e6] (default) | real valued vector

Frequency offsets of the phase noise from the carrier frequency, specified as a real valued vector in Hz.

Dependencies

To enable this parameter, select **Add phase noise** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'Foffset')` to view the current **Phase noise frequency offset (Hz)** metric.
- Use `set_param(gcb, 'Foffset', value)` to set **Phase noise frequency offset (Hz)** to a specific metric.

Data Types: double

Phase noise level (dBc/Hz) — Phase noise power at specified frequency offsets relative to the carrier

[-56 -106 -132 -143 -152] (default) | real valued vector

Real valued vector specifying the phase noise power in a 1 Hz bandwidth centered at the specified frequency offsets relative to the carrier. The value is specified in dBc/Hz.

Dependencies

To enable this parameter, select **Add phase noise** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'PhaseNoise')` to view the current **Phase noise level (dBc/Hz)** metric.
- Use `set_param(gcb, 'PhaseNoise', value)` to set **Phase noise level (dBc/Hz)** to a specific metric.

Data Types: double

Prescaler

Fractional clock divider value — Value by which the clock divider divides the input frequency

70.20 (default) | positive real scalar

Value by which the clock divider divides the input frequency, specified as a positive real scalar.

Programmatic Use

- Use `get_param(gcb, 'N')` to view the current value of **Fractional clock divider value**.
- Use `set_param(gcb, 'N', value)` to set **Fractional clock divider value** to a specific value.

Delta Sigma Modulator order — Order of the Delta Sigma Modulator

3rd order (default) | 1st order | 2nd order | 4th order

The order of the delta sigma modulator. For more information, see Fractional Clock Divider with DSM.

Programmatic Use

- Use `get_param(gcb, 'dsm')` to view the current **Delta Sigma Modulator order**.

- Use `set_param(gcb, 'dsm', value)` to set **Delta Sigma Modulator order** to a specific value.

Min clock divider value — Minimum value by which clock divider can divide input frequency

70 (default) | positive real scalar

Minimum value by which the clock divider can divide input frequency, specified as a positive real scalar. This parameter is also reported in the **Loop Filter** tab and is used to automatically calculate the filter component values of the loop filter.

Programmatic Use

- Use `get_param(gcb, 'Nmin')` to view the current value of **Min clock divider value**.
- Use `set_param(gcb, 'Nmin', value)` to set **Min clock divider value** to a specific value.

Loop Filter

Filter component values — Determines how filter components are computed

Automatic (default) | Manual

Select how filter components for the loop filter are computed:

- Select **Automatic** to automatically compute filter components from system specifications. Resistance and capacitance edit boxes in the **Loop Filter** tab are not editable if this option is selected. Rather, the filter component values are calculated from **Loop bandwidth (Hz)**, **Phase margin (degrees)**, **VCO voltage sensitivity**, **Charge pump current**, and **Min clock divider value**. By default, this option is selected.
- Select **Manual** to manually enter the resistance and capacitance values to design a customized loop filter.

Loop bandwidth (Hz) — Frequency at which magnitude of open loop transfer function becomes 1

0.5e6 (default) | positive real scalar

Frequency at which the magnitude of the open loop transfer function becomes 1, specified as a positive real scalar in Hz. Lower values of **Loop bandwidth (Hz)** result in reduced phase noise and reference spurs at the expense of longer lock time and less phase margin.

Dependencies

This parameter is only available when **Automatic** is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'Fc')` to view the current value of **Loop bandwidth (Hz)**.
- Use `set_param(gcb, 'Fc', value)` to set **Loop bandwidth (Hz)** to a specific value.

Phase margin (degrees) — Phase of open loop transfer function at loop bandwidth subtracted from 180°

45 (default) | positive real scalar

Phase of the open loop transfer function at the loop bandwidth subtracted from 180°, specified as a positive real scalar in degrees. For optimum lock time, select a phase margin between 40° and 55°.

Dependencies

This parameter is only available when Automatic is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'Phi')` to view the current value of **Phase margin (degrees)**.
- Use `set_param(gcb, 'Phi', value)` to set **Phase margin (degrees)** to a specific value.

Data Types: double

Loop filter type – Order of the loop filter

3rd Order Passive (default) | 2nd Order Passive | 4th Order Passive

Order of the loop filter. Applies a second-, third-, or fourth-order passive RC loop filter in the PLL system.

C1 (F) – Capacitance 1

3e-14 (default) | positive real scalar

Capacitor value C1, specified as a positive real scalar in farad.

Dependencies

This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C1')` to view the current value of **C1 (F)**.
- Use `set_param(gcb, 'C1', value)` to set **C1 (F)** to a specific value.

Data Types: double

C2 (F) – Capacitance 2

3.3e-13 (default) | positive real scalar

Capacitor value C2, specified as a positive real scalar in farad.

Dependencies

This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C2')` to view the current value of **C2 (F)**.
- Use `set_param(gcb, 'C2', value)` to set **C2 (F)** to a specific value.

Data Types: double

C3 (F) – Capacitance 3

2.15e-15 (default) | positive real scalar

Capacitor value C3, specified as a positive real scalar in farad.

Dependencies

- To enable this parameter, select 3rd Order Passive or 4th Order Passive in **Loop filter type**.
- This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C3')` to view the current value of **C3 (F)**.
- Use `set_param(gcb, 'C3', value)` to set **C3 (F)** to a specific value.

Data Types: double

C4 (F) – Capacitance 4

1e-12 (default) | positive real scalar

Capacitor value C4, specified as a positive real scalar in farad.

Dependencies

- To enable this parameter, select 4th Order Passive in **Loop filter type**.
- This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C4')` to view the current value of **C4 (F)**.
- Use `set_param(gcb, 'C4', value)` to set **C4 (F)** to a specific value.

Data Types: double

R2 (ohms) – Resistance 2

2.33e+06 (default) | positive real scalar

Resistor value R2, specified as a positive real scalar in ohms.

Dependencies

This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'R2')` to view the current value of **R2 (ohms)**.
- Use `set_param(gcb, 'R2', value)` to set **R2 (ohms)** to a specific value.

Data Types: double

R3 (ohms) – Resistance 3

2.98e+07 (default) | positive real scalar

Resistor value R3, specified as a positive real scalar in ohms.

Dependencies

- To enable this parameter, select 3rd Order Passive or 4th Order Passive in **Loop filter type**.
- This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'R3')` to view the current value of **R3 (ohms)**.
- Use `set_param(gcb, 'R3', value)` to set **R3 (ohms)** to a specific value.

Data Types: double

R4 (ohms) – Resistance 4

12e3 (default) | positive real scalar

Resistor value R4, specified as a positive real scalar in ohms.

Dependencies

- To enable this parameter, select 4th Order Passive in **Loop filter type**.
- This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'R4')` to view the current value of **R4 (ohms)**.
- Use `set_param(gcb, 'R4', value)` to set **R4 (ohms)** to a specific value.

Data Types: double

Enable impairments – Add circuit impairments to simulation

off (default) | on

Select to add circuit impairments such as operating temperature to determine thermal noise to simulation. By default, this option is deselected.

Operating temperature (°C) – Temperature to determine the level of thermal noise

30 (default) | real scalar

Temperature of the resistor, specified as a real scalar in °C. **Operating temperature** determines the level of thermal (Johnson) noise.**Dependencies**To enable this parameter, select **Enable impairments** in the **Loop Filter** tab.**Programmatic Use**

- Use `get_param(gcb, 'Temperature')` to view the current value of **Operating temperature**.
- Use `set_param(gcb, 'Temperature', value)` to set **Operating temperature** to a specific value.

Data Types: double

Export Loop Filter Component Values — Export loop filter component values

button

Click to export loop filter component values to a spreadsheet (XLS file) or as comma-separated values (CSV file).

Probe**PFD up and PFD down (pfd_up and pfd_down) — Select to probe PFD outputs**

off (default) | on

Select to probe the PFD output wires (pfd_up and pfd_down) to view the response of the PFD.

Charge pump output (cp_out) — Select to probe charge pump output

off (default) | on

Select to probe the charge pump output wire (cp_out) to view the response of the Charge Pump.

Loop filter output (lf_out) — Select to probe loop filter output

off (default) | on

Select to probe loop filter output wire (lf_out) to view the response of the Loop Filter. The loop filter output provides the control voltage to the VCO.

Prescaler output (ps_out) — Select to probe prescaler output

off (default) | on

Select to probe the prescaler output wire (ps_out) to view the response of the Fractional Clock Divider with Accumulator.

Analysis**Open Loop Analysis — Plot the presimulation open loop analysis**

on (default) | off

Select to plot the gain margin and phase margin of the PLL system before simulation. By default, this option is selected.

Closed Loop Analysis — Plot the presimulation closed loop analysis

off (default) | on

Select to plot the pole-zero map, loop bandwidth, step response, and impulse response of the PLL system before simulation. You must have a license to Control System Toolbox to plot the step response and impulse response of the PLL system. By default, this option is deselected.

Plot Loop Dynamics — Plot loop dynamics of PLL system

button

Click to plot the presimulation loop dynamics of the PLL system.

Version History**Introduced in R2019a**

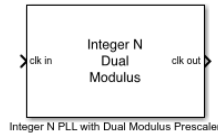
See Also

PFD | Charge Pump | Loop Filter | Fractional Clock Divider with DSM | VCO

Integer N PLL with Dual Modulus Prescaler

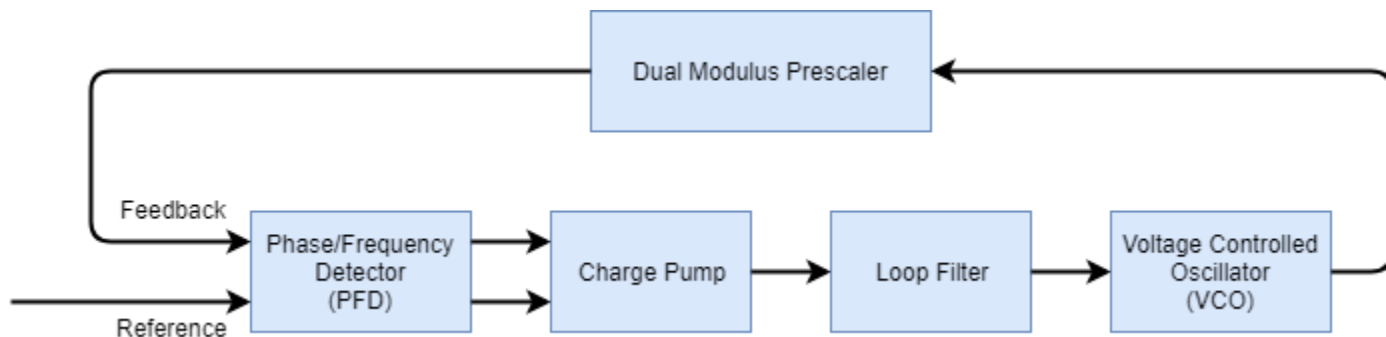
Frequency synthesizer with dual modulus prescaler based integer N PLL architecture

Library: Mixed-Signal Blockset / PLL / Architectures



Description

The Integer N PLL with Dual Modulus Prescaler reference architecture uses a Dual Modulus Prescaler block as the frequency divider in a PLL system. The frequency divider divides the frequency of the VCO output signal by an integer value to make it comparable to a PFD reference signal frequency.



Ports

Input

clk in — Input clock signal

scalar

Input clock signal, specified as a scalar. The signal at the **clk in** port is used as the reference signal for the PFD block in a PLL system.

Data Types: double

Output

clk out — Output clock signal

scalar

Output clock signal, specified as a scalar. The signal at the **clk out** port is the output of the VCO block in a PLL system.

Data Types: double

Parameters

Enable increased buffer size — Enable increased buffer size

button

Select to enable increased buffer size during the simulation. This increases the buffer size of all the building blocks in the PLL model that belong to the Mixed-Signal Blockset/PLL/Building Blocks Simulink library. The building blocks are PFD, Charge Pump, Loop Filter, VCO, and Dual Modulus Prescaler. By default, this option is deselected.

Buffer size for loop filter — Buffer size for loop filter

1000 (default) | positive integer scalar

Buffer size for the loop filter, specified as a positive integer scalar. This sets the number of extra buffer samples available during the simulation to the Convert Sample Time subsystem inside the loop filter.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size for loop filter** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when the **Enable increased buffer size** option is selected.

Programmatic Use

- Use `get_param(gcb, 'NBufferFilter')` to view the current value of **Buffer size for loop filter**.
- Use `set_param(gcb, 'NBufferFilter', value)` to set **Buffer size for loop filter** to a specific value.

Buffer size for PFD, charge pump, VCO, prescaler — Buffer size for PFD, charge pump, VCO, and prescaler

10 (default) | positive integer scalar

Buffer size for the PFD, charge pump, VCO, and prescaler, specified as a positive integer scalar. This sets the buffer size of the PFD, Charge Pump, VCO, and Dual Modulus Prescaler blocks inside the PLL model.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size for PFD, charge pump, VCO, prescaler** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when the **Enable increased buffer size** option is selected.

Programmatic Use

- Use `get_param(gcb, 'NBuffer')` to view the current value of **Buffer size for PFD, charge pump, VCO, prescaler**.
- Use `set_param(gcb, 'NBuffer', value)` to set **Buffer size for PFD, charge pump, VCO, prescaler** to a specific value.

PFD**Configuration****Deadband compensation (s) — Delay added for active output near zero phase offset**

40e-12 (default) | positive real scalar

Delay added for active output near zero phase offset, specified as a positive real scalar in seconds. Deadband is the phase offset band near zero phase offset for which the PFD output is negligible.

Programmatic Use

- Use `get_param(gcb, 'DeadbandCompensation')` to view the current value of **Deadband compensation (s)**.
- Use `set_param(gcb, 'DeadbandCompensation', value)` to set **Deadband compensation (s)** to a specific value.

Data Types: double

Impairments**Enable impairments — Add circuit impairments to simulation**

off (default) | on

Select to add circuit impairments such as rise/fall time and propagation delay to simulation. By default, this option is deselected.

Output step size calculation — Determine how output step size is calculated

Default (default) | Advanced

Determine how output step size is calculated:

- Select **Default** to calculate output step size from rise/fall time. Output step size (ΔT) is given by

$$\Delta T = \frac{(\text{Rise/fall time})^2}{6 \cdot 0.22}$$
- Select **Advanced** to calculate output step size from maximum frequency of interest. Output step size (ΔT) is given by $\Delta T = \frac{\text{Rise/fall time}}{6 \cdot \text{Maximum frequency of interest}}$

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab.

Maximum frequency of interest (Hz) — Maximum frequency of interest at output

10e9 (default) | positive real scalar

Maximum frequency of interest at the output, specified as a positive real scalar in Hz.

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab and choose **Advanced** for **Output step size calculation**.

Programmatic Use

- Use `get_param(gcb, 'MaxFreqInterest')` to view the current value of **Maximum frequency of interest (Hz)**.

- Use `set_param(gcb, 'MaxFreqInterest', value)` to set **Maximum frequency of interest (Hz)** to a specific value.

Data Types: double

Rise/fall time (s) — 20% - 80% rise/fall time for up output port of PFD

3e-11 (default) | positive real scalar

20% - 80% rise/fall time for the up output port of the PFD, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab.

Programmatic Use

- Use `get_param(gcb, 'RiseFallTime')` to view the current value of **Rise/fall time (s)**.
- Use `set_param(gcb, 'RiseFallTime', value)` to set **Rise/fall time (s)** to a specific value.

Data Types: double

Propagation Delay (s) — Delay from input port to output port of PFD

50e-12 (default) | positive real scalar

Delay from the input port to output port of the PFD, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelay')` to view the current value of **Propagation Delay (s)**.
- Use `set_param(gcb, 'PropDelay', value)` to set **Propagation Delay (s)** to a specific value.

Data Types: double

Charge pump

Configuration

Output current (A) — Design output current

1e-3 (default) | positive real scalar

Full scale magnitude of design output current, specified as a positive real scalar in amperes. This parameter is also reported as **Charge pump current** in the **Loop Filter** tab and is used to automatically calculate the filter component values of the loop filter.

Programmatic Use

- Use `get_param(gcb, 'OutputCurrent')` to view the current value of **Output current (A)**.
- Use `set_param(gcb, 'OutputCurrent', value)` to set **Output current (A)** to a specific value.

Data Types: double

Input threshold (V) — Logic switching threshold at input ports

0.5 (default) | real scalar

Logic switching threshold at input ports, specified as a real scalar in volts.

Programmatic Use

- Use `get_param(gcb, 'InputThreshold')` to view the current value of **Input threshold (V)**.
- Use `set_param(gcb, 'InputThreshold', value)` to set **Input threshold (V)** to a specific value.

Data Types: double

Impairments**Enable current impairments — Add current impairments to simulation**

off (default) | on

Select to add current impairments such as current imbalance and leakage current to simulation. By default, this option is deselected.

Current imbalance (A) — Difference between full scale positive and negative current

1e-7 (default) | positive real scalar

Difference between full scale positive and negative current, specified as a positive real scalar in amperes.

Dependencies

To enable this parameter, select **Enable current impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'CurrentImbalance')` to view the current value of **Current imbalance (A)**.
- Use `set_param(gcb, 'CurrentImbalance', value)` to set **Current imbalance (A)** to a specific value.

Data Types: double

Leakage current (A) — Output current without any input

1e-8 (default) | nonnegative real scalar

Output current when both inputs are at logic zero, specified as a nonnegative real scalar in amperes.

Dependencies

To enable this parameter, select **Enable current impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'LeakageCurrent')` to view the current value of **Leakage current (A)**.
- Use `set_param(gcb, 'LeakageCurrent', value)` to set **Leakage current (A)** to a specific value.

Data Types: double

Enable timing impairments — Add timing impairments to simulation

off (default) | on

Select to add timing impairments such as rise/fall time and propagation delay to simulation. By default, this option is deselected.

Output step size calculation – Determine how output step size is calculated

Default (default) | Advanced

Determine how output step size is calculated:

- Select **Default** to calculate output step size from rise/fall time. Output step size (ΔT) is given by
$$\Delta T = \frac{(\text{Rise/fall time})^2}{6 \cdot 0.22}$$
.
- Select **Advanced** to calculate output step size from maximum frequency of interest. Output step size (ΔT) is given by
$$\Delta T = \frac{\text{Rise/fall time}}{6 \cdot \text{Maximum frequency of interest}}$$
.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge Pump** tab.

Maximum frequency of interest (Hz) – Maximum frequency of interest at output

10e9 (default) | positive real scalar

Maximum frequency of interest at the output, specified as a positive real scalar in Hz.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge Pump** tab and choose **Advanced** for **Output step size calculation**.

Programmatic Use

- Use `get_param(gcb, 'MaxFreqInterestCp')` to view the current value of **Maximum frequency of interest (Hz)**.
- Use `set_param(gcb, 'MaxFreqInterestCp', value)` to set **Maximum frequency of interest (Hz)** to a specific value.

Data Types: double

Up

Rise/fall time (s) – 20% - 80% rise/fall time for up input port

5e-9 (default) | positive real scalar

20% - 80% rise/fall time for the up input port of the charge pump, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'RiseFallUp')` to view the current value of **Up Rise/fall time (s)**.
- Use `set_param(gcb, 'RiseFallUp', value)` to set **Up Rise/fall time (s)** to a specific value.

Data Types: double

Propagation delay (s) — Total propagation delay from up input port to output port of charge pump

6e-9 (default) | positive real scalar

Total propagation delay from the up input port to output port of the charge pump, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelayUp')` to view the current value of **Up Propagation delay (s)**.
- Use `set_param(gcb, 'PropDelayUp', value)` to set **Up Propagation delay (s)** to a specific value.

Data Types: double

Down**Rise/fall time — 20% - 80% rise/fall time for down input port**

2e-9 (default) | scalar

20% - 80% rise/fall time for down input port of charge pump.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'RiseFallDown')` to view the current value of **Down Rise/fall time (s)**.
- Use `set_param(gcb, 'RiseFallDown', value)` to set **Down Rise/fall time (s)** to a specific value.

Data Types: double

Propagation delay (s) — Total propagation delay from up input port to output port of charge pump

4e-9 (default) | positive real scalar

Total propagation delay from the up input port to output port of the charge pump, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelayUp')` to view the current value of **Down Propagation delay (s)**.
- Use `set_param(gcb, 'PropDelayUp', value)` to set **Down Propagation delay (s)** to a specific value.

Data Types: double

VCO

Specify using — Define how VCO output frequency is specified

Voltage sensitivity (default) | Output frequency vs. control voltage

Define how VCO output frequency is specified:

- Select **Voltage sensitivity** to specify output frequency from **Voltage sensitivity (Hz/V)** and **Free running frequency (Hz)**.
- Select **Output frequency vs. control voltage** to interpolate output frequency from **Control voltage (V)** vector versus **Output frequency (Hz)** vector.

Programmatic Use

- Use `set_param(gcb, 'SpecifyUsing', 'Voltage sensitivity')` to set **Specify using** to **Voltage sensitivity**.
- Use `set_param(gcb, 'SpecifyUsing', 'Output frequency vs. control voltage')` to set **Specify using** to **Output frequency vs. control voltage**.

Voltage sensitivity (Hz/V) — Measure of change in output frequency of VCO

100e6 (default) | positive real scalar

Measure of change in output frequency for input voltage change, specified as a positive real scalar with units in Hz/V. This parameter is also reported as **VCO voltage sensitivity** in the **Loop Filter** tab and is used to automatically calculate the filter component values of the loop filter.

Dependencies

To enable this parameter, select **Voltage sensitivity** in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'Kvco')` to view the current **Voltage sensitivity (Hz/V)** value.
- Use `set_param(gcb, 'Kvco', value)` to set **Voltage sensitivity (Hz/V)** to a specific value.

Data Types: double

Free running frequency (Hz) — VCO output frequency without control voltage

1.8e9 (default) | positive real scalar

Frequency of the VCO without any control voltage input (0 V), or the quiescent frequency, specified as a positive real scalar in Hz.

Dependencies

To enable this parameter, select **Voltage sensitivity** in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'Fo')` to view current **Free running frequency (Hz)** value.
- Use `set_param(gcb, 'Fo', value)` to set **Free running frequency (Hz)** to a specific value.

Data Types: double

Control voltage (V) — Control voltage values

[-5 0 5] (default) | real valued vector

Control voltage values of the VCO, specified as a real valued vector in volts.

Dependencies

To enable this parameter, select Output frequency vs. control voltage in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'ControlVoltage')` to view current **Control voltage (V)** value.
- Use `set_param(gcb, 'ControlVoltage', value)` to set **Control voltage (V)** to a specific value.

Data Types: double

Output frequency (Hz) — VCO output frequency values

[2e9 2.5e9 3e9] (default) | real valued vector

Output frequency of the values of the VCO, corresponding to the **Control voltage (V)** vector, specified in Hz.

Dependencies

To enable this parameter, select Output frequency vs. control voltage in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'OutputFrequency')` to view current **Output frequency (Hz)** value.
- Use `set_param(gcb, 'OutputFrequency', value)` to set **Output frequency (Hz)** to a specific value.

Data Types: double

Output amplitude gain — Ratio of VCO output voltage to input voltage

1 (default) | positive real scalar

Ratio of VCO output voltage to input voltage, specified as a positive real scalar. The input voltage has a nontunable value of 1 V.

Programmatic Use

- Use `get_param(gcb, 'Amplitude')` to view current **Output amplitude gain** value.
- Use `set_param(gcb, 'Amplitude', value)` to set **Output amplitude gain** to a specific value.

Data Types: double

Impairment**Add Phase-noise — Add phase noise as function of frequency**

off (default) | on

Select to introduce phase noise as a function of frequency to the VCO. By default, this option is deselected.

Phase noise frequency offset (Hz) — Frequency offsets of phase noise from carrier frequency

[30e3 100e3 1e6 3e6 10e6] (default) | real valued vector

Frequency offsets of the phase noise from the carrier frequency, specified as a real valued vector in Hz.

Dependencies

To enable this parameter, select **Add Phase-noise** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'Foffset')` to view the current **Phase noise frequency offset (Hz)** metric.
- Use `set_param(gcb, 'Foffset', value)` to set **Phase noise frequency offset (Hz)** to a specific metric.

Data Types: `double`

Phase noise level (dBc/Hz) – Phase noise power at specified frequency offsets relative to the carrier

`[-56 -106 -132 -143 -152]` (default) | real valued vector

Real valued vector specifying the phase noise power in a 1 Hz bandwidth centered at the specified frequency offsets relative to the carrier. The value is specified in dBc/Hz.

Dependencies

To enable this parameter, select **Add Phase-noise** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'PhaseNoise')` to view the current **Phase noise level (dBc/Hz)** metric.
- Use `set_param(gcb, 'PhaseNoise', value)` to set **Phase noise level (dBc/Hz)** to a specific metric.

Data Types: `double`

Prescaler

Program counter value, P – Value of the program counter inside dual modulus prescaler

`18` (default) | positive real scalar

Value of the program counter inside the dual modulus prescaler, specified as a positive real scalar.

Program counter value, P is used to calculate the effective divider value. For more information, see Dual Modulus Prescaler.

Programmatic Use

- Use `get_param(gcb, 'ProgramCounter')` to view the current **Program counter value, P**.
- Use `set_param(gcb, 'ProgramCounter', value)` to set **Program counter value, P** to a specific value.

Data Types: `double`

Prescaler divider value, N – Value of the prescaler divider inside dual modulus prescaler

`5` (default) | positive real scalar

Value of the prescaler divider inside the dual modulus prescaler, specified as a positive real scalar. **Prescaler divider value, N** is used to calculate the effective divider value. For more information, see Dual Modulus Prescaler.

Programmatic Use

- Use `get_param(gcb, 'PrescalerDivider')` to view the current **Prescaler divider value, N**.
- Use `set_param(gcb, 'PrescalerDivider', value)` to set **Prescaler divider value, N** to a specific value.

Data Types: double

Swallow counter value, S — Value of the swallow counter inside dual modulus prescaler
10 (default) | positive real scalar

Value of the swallow counter inside the dual modulus prescaler, specified as a positive real scalar. **Swallow counter value, S** is used to calculate the effective divider value. For more information, see Dual Modulus Prescaler.

Programmatic Use

- Use `get_param(gcb, 'SwallowCounter')` to view the current **Swallow counter value, S**.
- Use `set_param(gcb, 'SwallowCounter', value)` to set **Swallow counter value, S** to a specific value.

Data Types: double

Min clock divider value — Minimum value by which clock divider can divide input frequency
100 (default) | positive real scalar

Minimum value by which the clock divider can divide input frequency, specified as a positive real scalar. This parameter is also reported in the **Loop Filter** tab and is used to automatically calculate the filter component values of the loop filter.

Programmatic Use

- Use `get_param(gcb, 'Nmin')` to view the current value of **Min clock divider value**.
- Use `set_param(gcb, 'Nmin', value)` to set **Min clock divider value** to a specific value.

Data Types: double

Loop Filter

Filter component values — Determines how filter components are computed
Automatic (default) | Manual

Select how filter components for the loop filter are computed:

- Select **Automatic** to automatically compute filter components from system specifications. Resistance and capacitance edit boxes in the **Loop Filter** tab are not editable if this option is selected. Rather, the filter component values are calculated from **Loop bandwidth (Hz)**, **Phase margin (degrees)**, **VCO voltage sensitivity**, **Charge pump current**, and **Min clock divider value**. By default, this option is selected.
- Select **Manual** to manually enter the resistance and capacitance values to design a customized loop filter.

Loop bandwidth (Hz) — Frequency at which magnitude of open loop transfer function becomes 1

1e6 (default) | positive real scalar

Frequency at which the magnitude of the open loop transfer function becomes 1, specified as a positive real scalar in Hz. Lower values of **Loop bandwidth (Hz)** result in reduced phase noise and reference spurs at the expense of longer lock time and less phase margin.

Dependencies

This parameter is only available when Automatic is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'Fc')` to view the current value of **Loop bandwidth (Hz)**.
- Use `set_param(gcb, 'Fc', value)` to set **Loop bandwidth (Hz)** to a specific value.

Data Types: double

Phase margin (degrees) — Phase of open loop transfer function at loop bandwidth subtracted from 180°

45 (default) | positive real scalar

Phase of the open loop transfer function at the loop bandwidth subtracted from 180°, specified as a positive real scalar in degrees. For optimum lock time, select a phase margin between 40° and 55°.

Dependencies

This parameter is only available when Automatic is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'Phi')` to view the current value of **Phase margin (degrees)**.
- Use `set_param(gcb, 'Phi', value)` to set **Phase margin (degrees)** to a specific value.

Data Types: double

Loop filter type — Order of the loop filter

3rd Order Passive (default) | 2nd Order Passive | 4th Order Passive

Order of the loop filter. Applies a second-, third-, or fourth-order passive RC loop filter in the PLL system.

C1 (F) — Capacitance 1

5.24e-15 (default) | positive real scalar

Capacitor value C1, specified as a positive real scalar in farad.

Dependencies

This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C1')` to view the current value of **C1 (F)**.
- Use `set_param(gcb, 'C1', value)` to set **C1 (F)** to a specific value.

Data Types: double

C2 (F) – Capacitance 2

5.77e-14 (default) | positive real scalar

Capacitor value C2, specified as a positive real scalar in farad.

Dependencies

This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C2')` to view the current value of **C2 (F)**.
- Use `set_param(gcb, 'C2', value)` to set **C2 (F)** to a specific value.

Data Types: double

C3 (F) – Capacitance 3

3.76e-16 (default) | positive real scalar

Capacitor value C3, specified as a positive real scalar in farad.

Dependencies

- To enable this parameter, select 3rd Order Passive or 4th Order Passive in **Loop filter type**.
- This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C3')` to view the current value of **C3 (F)**.
- Use `set_param(gcb, 'C3', value)` to set **C3 (F)** to a specific value.

Data Types: double

C4 (F) – Capacitance 4

1e-12 (default) | positive real scalar

Capacitor value C4, specified as a positive real scalar in farad.

Dependencies

- To enable this parameter, select 4th Order Passive in **Loop filter type**.
- This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C4')` to view the current value of **C4 (F)**.
- Use `set_param(gcb, 'C4', value)` to set **C4 (F)** to a specific value.

Data Types: double

R2 (ohms) – Resistance 2

6.66e+06 (default) | positive real scalar

Resistor value R2, specified as a positive real scalar in ohms.

Dependencies

This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'R2')` to view the current value of **R2 (ohms)**.
- Use `set_param(gcb, 'R2', value)` to set **R2 (ohms)** to a specific value.

Data Types: double

R3 (ohms) – Resistance 3

8.51e+07 (default) | positive real scalar

Resistor value R3, specified as a positive real scalar in ohms.

Dependencies

- To enable this parameter, select 3rd Order Passive or 4th Order Passive in **Loop filter type**.
- This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'R3')` to view the current value of **R3 (ohms)**.
- Use `set_param(gcb, 'R3', value)` to set **R3 (ohms)** to a specific value.

Data Types: double

R4 (ohms) – Resistance 4

12e3 (default) | positive real scalar

Resistor value R4, specified as a positive real scalar in ohms.

Dependencies

- To enable this parameter, select 4th Order Passive in **Loop filter type**.
- This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'R4')` to view the current value of **R4 (ohms)**.
- Use `set_param(gcb, 'R4', value)` to set **R4 (ohms)** to a specific value.

Data Types: double

Enable impairments — Add circuit impairments to simulation

off (default) | on

Select to add circuit impairments such as operating temperature to determine thermal noise to simulation. By default, this option is deselected.

Operating temperature (°C) — Temperature to determine the level of thermal noise

30 (default) | real scalar

Temperature of the resistor, specified as a real scalar in °C. **Operating temperature** determines the level of thermal (Johnson) noise.

Dependencies

To enable this parameter, select **Enable impairments** in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'Temperature')` to view the current value of **Operating temperature**.
- Use `set_param(gcb, 'Temperature', value)` to set **Operating temperature** to a specific value.

Data Types: double

Export Loop Filter Component Values — Export loop filter component values

button

Click to export loop filter component values to a spreadsheet (XLS file) or as comma-separated values (CSV file).

Probe**PFD up and PFD down (pfd_up and pfd_down) — Select to probe PFD outputs**

off (default) | on

Select to probe the PFD output wires (pfd_up and pfd_down) to view the response of the PFD.

Charge pump output (cp_out) — Select to probe charge pump output

off (default) | on

Select to probe the charge pump output wire (cp_out) to view the response of the Charge Pump.

Loop filter output (lf_out) — Select to probe loop filter output

off (default) | on

Select to probe loop filter output wire (lf_out) to view the response of the Loop Filter. The loop filter output provides the control voltage to the VCO.

Prescaler output (ps_out) — Select to probe prescaler output

off (default) | on

Select to probe the prescaler output wire (ps_out) to view the response of the Fractional Clock Divider with Accumulator.

Analysis

Open Loop Analysis — Plot the presimulation open loop analysis

on (default) | off

Select to plot the gain margin and phase margin of the PLL system before simulation. By default, this option is selected.

Closed Loop Analysis — Plot the presimulation closed loop analysis

off (default) | on

Select to plot the pole-zero map, loop bandwidth, step response, and impulse response of the PLL system before simulation. You must have a license to Control System Toolbox to plot the step response and impulse response of the PLL system. By default, this option is deselected.

Plot Loop Dynamics — Plot loop dynamics of PLL system

button

Click to plot the presimulation loop dynamics of the PLL system.

Version History

Introduced in R2019a

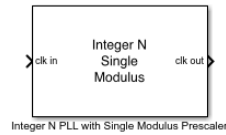
See Also

PFD | Charge Pump | Loop Filter | Dual Modulus Prescaler | VCO

Integer N PLL with Single Modulus Prescaler

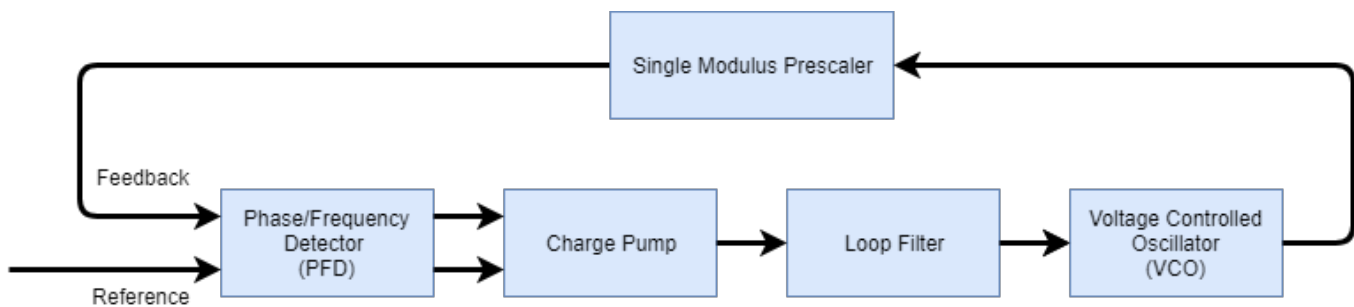
Frequency synthesizer with single modulus prescaler based integer N PLL architecture

Library: Mixed-Signal Blockset / PLL / Architectures



Description

The Integer N PLL with Single Modulus Prescaler reference architecture uses a Single Modulus Prescaler block as the frequency divider in a PLL system. The frequency divider divides the frequency of the VCO output signal by an integer value to make it comparable to a PFD reference signal frequency.



Ports

Input

clk in – Input clock signal

scalar

Input clock signal, specified as a scalar. The signal at the **clk in** port is used as the reference signal for the PFD block in a PLL system.

Data Types: double

Output

clk out – Output clock signal

scalar

Output clock signal, specified as a scalar. The signal at the **clk out** port is the output of the VCO block in a PLL system.

Data Types: double

Parameters

Enable increased buffer size – Enable increased buffer size

button

Select to enable increased buffer size during the simulation. This increases the buffer size of all the building blocks in the PLL model that belong to the Mixed-Signal Blockset/PLL/Building Blocks Simulink library. The building blocks are PFD, Charge Pump, Loop Filter, VCO, and Single Modulus Prescaler. By default, this option is deselected.

Buffer size for loop filter – Buffer size for loop filter

1000 (default) | positive integer scalar

Buffer size for the loop filter, specified as a positive integer scalar. This sets the number of extra buffer samples available during the simulation to the Convert Sample Time subsystem inside the loop filter.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size for loop filter** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when the **Enable increased buffer size** option is selected.

Programmatic Use

- Use `get_param(gcb, 'NBufferFilter')` to view the current value of **Buffer size for loop filter**.
- Use `set_param(gcb, 'NBufferFilter', value)` to set **Buffer size for loop filter** to a specific value.

Buffer size for PFD, charge pump, VCO, prescaler – Buffer size for PFD, charge pump, VCO, and prescaler

10 (default) | positive integer scalar

Buffer size for the PFD, charge pump, VCO, and prescaler, specified as a positive integer scalar. This sets the buffer size of the PFD, Charge Pump, VCO, and Single Modulus Prescaler blocks inside the PLL model.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size for PFD, charge pump, VCO, prescaler** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when the **Enable increased buffer size** option is selected.

Programmatic Use

- Use `get_param(gcb, 'NBuffer')` to view the current value of **Buffer size for PFD, charge pump, VCO, prescaler**.
- Use `set_param(gcb, 'NBuffer', value)` to set **Buffer size for PFD, charge pump, VCO, prescaler** to a specific value.

PFD**Configuration****Deadband compensation (s) – Delay added for active output near zero phase offset**

30e-12 (default) | positive real scalar

Delay added for active output near zero phase offset, specified as a positive real scalar in seconds. Deadband is the phase offset band near zero phase offset for which the PFD output is negligible.

Programmatic Use

- Use `get_param(gcb, 'DeadbandCompensation')` to view the current value of **Deadband compensation (s)**.
- Use `set_param(gcb, 'DeadbandCompensation', value)` to set **Deadband compensation (s)** to a specific value.

Data Types: double

Impairments**Enable impairments – Add circuit impairments to simulation**

off (default) | on

Select to add circuit impairments such as rise/fall time and propagation delay to simulation. By default, this option is deselected.

Output step size calculation – Determine how output step size is calculated

Default (default) | Advanced

Determine how output step size is calculated:

- Select **Default** to calculate output step size from rise/fall time. Output step size (ΔT) is given by

$$\Delta T = \frac{(\text{Rise/fall time})^2}{6 \cdot 0.22}$$
- Select **Advanced** to calculate output step size from maximum frequency of interest. Output step size (ΔT) is given by $\Delta T = \frac{\text{Rise/fall time}}{6 \cdot \text{Maximum frequency of interest}}$

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab.

Maximum frequency of interest (Hz) – Maximum frequency of interest at output

10e9 (default) | positive real scalar

Maximum frequency of interest at the output, specified as a positive real scalar in Hz.

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab and choose **Advanced** for **Output step size calculation**.

Programmatic Use

- Use `get_param(gcb, 'MaxFreqInterest')` to view the current value of **Maximum frequency of interest (Hz)**.

- Use `set_param(gcb, 'MaxFreqInterest', value)` to set **Maximum frequency of interest (Hz)** to a specific value.

Data Types: double

Rise/fall time (s) — 20% - 80% rise/fall time for up output port of PFD

30e-12 (default) | positive real scalar

20% - 80% rise/fall time for the up output port of the PFD, specified as a real positive scalar in seconds.

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab.

Programmatic Use

- Use `get_param(gcb, 'RiseFallTime')` to view the current value of **Rise/fall time (s)**.
- Use `set_param(gcb, 'RiseFallTime', value)` to set **Rise/fall time (s)** to a specific value.

Propagation Delay (s) — Delay from input port to output port of PFD

50e-12 (default) | positive real scalar

Delay from the input port to output port of the PFD, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable Impairments** in the **PFD** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelay')` to view the current value of **Propagation Delay (s)**.
- Use `set_param(gcb, 'PropDelay', value)` to set **Propagation Delay (s)** to a specific value.

Data Types: double

Charge pump

Configuration

Output current (A) — Design output current

1e-3 (default) | positive real scalar

Full scale magnitude of design output current, specified as a positive real scalar in amperes. This parameter is also reported as **Charge pump current** in the **Loop Filter** tab and is used to automatically calculate the filter component values of the loop filter.

Programmatic Use

- Use `get_param(gcb, 'OutputCurrent')` to view the current value of **Output current (A)**.
- Use `set_param(gcb, 'OutputCurrent', value)` to set **Output current (A)** to a specific value.

Data Types: double

Input threshold (V) — Logic switching threshold at input ports

0.5 (default) | real scalar

Logic switching threshold at input ports, specified as a real scalar in volts.

Programmatic Use

- Use `get_param(gcb, 'InputThreshold')` to view the current value of **Input threshold (V)**.
- Use `set_param(gcb, 'InputThreshold', value)` to set **Input threshold (V)** to a specific value.

Data Types: double

Impairments**Enable current impairments — Add current impairments to simulation**

off (default) | on

Select to add current impairments such as current imbalance and leakage current to simulation. By default, this option is deselected.

Current imbalance (A) — Difference between full scale positive and negative current

1e-7 (default) | positive real scalar

Difference between full scale positive and negative current, specified as a positive real scalar in amperes.

Dependencies

To enable this parameter, select **Enable current impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'CurrentImbalance')` to view the current value of **Current imbalance (A)**.
- Use `set_param(gcb, 'CurrentImbalance', value)` to set **Current imbalance (A)** to a specific value.

Data Types: double

Leakage current (A) — Output current without any input

1e-8 (default) | nonnegative real scalar

Output current when both inputs are at logic zero, specified as a nonnegative real scalar in amperes.

Dependencies

To enable this parameter, select **Enable current impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'LeakageCurrent')` to view the current value of **Leakage current (A)**.
- Use `set_param(gcb, 'LeakageCurrent', value)` to set **Leakage current (A)** to a specific value.

Data Types: double

Enable timing impairments — Add timing impairments to simulation

off (default) | on

Select to add timing impairments such as rise/fall time and propagation delay to simulation. By default, this option is deselected.

Output step size calculation – Determine how output step size is calculated

Default (default) | Advanced

Determine how output step size is calculated:

- Select **Default** to calculate output step size from rise/fall time. Output step size (ΔT) is given by
$$\Delta T = \frac{(\text{Rise/fall time})^2}{6 \cdot 0.22}$$
.
- Select **Advanced** to calculate output step size from maximum frequency of interest. Output step size (ΔT) is given by
$$\Delta T = \frac{\text{Rise/fall time}}{6 \cdot \text{Maximum frequency of interest}}$$
.

DependenciesTo enable this parameter, select **Enable timing impairments** in the **Charge Pump** tab.**Maximum frequency of interest (Hz) – Maximum frequency of interest at output**

10e9 (default) | positive real scalar

Maximum frequency of interest at the output, specified as a positive real scalar in Hz.

DependenciesTo enable this parameter, select **Enable timing impairments** in the **Charge Pump** tab and choose **Advanced** for **Output step size calculation**.**Programmatic Use**

- Use `get_param(gcb, 'MaxFreqInterestCp')` to view the current value of **Maximum frequency of interest (Hz)**.
- Use `set_param(gcb, 'MaxFreqInterestCp', value)` to set **Maximum frequency of interest (Hz)** to a specific value.

Data Types: double

Up**Rise/fall time (s) – 20% - 80% rise/fall time for up input port**

5e-9 (default) | positive real scalar

20% - 80% rise/fall time for the up input port of the charge pump, specified as a positive real scalar in seconds.

DependenciesTo enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.**Programmatic Use**

- Use `get_param(gcb, 'RiseFallUp')` to view the current value of **Up Rise/fall time (s)**.
- Use `set_param(gcb, 'RiseFallUp', value)` to set **Up Rise/fall time (s)** to a specific value.

Data Types: double

Propagation delay (s) – Total propagation delay from up input port to output port of charge pump

6e-9 (default) | positive real scalar

Total propagation delay from the up input port to output port of the charge pump, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelayUp')` to view the current value of **Up Propagation delay (s)**.
- Use `set_param(gcb, 'PropDelayUp', value)` to set **Up Propagation delay (s)** to a specific value.

Data Types: double

Down**Rise/fall time — 20% - 80% rise/fall time for down input port**

2e-9 (default) | scalar

20% - 80% rise/fall time for down input port of charge pump.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'RiseFallDown')` to view the current value of **Down Rise/fall time (s)**.
- Use `set_param(gcb, 'RiseFallDown', value)` to set **Down Rise/fall time (s)** to a specific value.

Data Types: double

Propagation delay (s) — Total propagation delay from up input port to output port of charge pump

4e-9 (default) | positive real scalar

Total propagation delay from the up input port to output port of the charge pump, specified as a positive real scalar in seconds.

Dependencies

To enable this parameter, select **Enable timing impairments** in the **Charge pump** tab.

Programmatic Use

- Use `get_param(gcb, 'PropDelayUp')` to view the current value of **Down Propagation delay (s)**.
- Use `set_param(gcb, 'PropDelayUp', value)` to set **Down Propagation delay (s)** to a specific value.

Data Types: double

VCO**Specify using — Define how VCO output frequency is specified**

Voltage sensitivity (default) | Output frequency vs. control voltage

Define how VCO output frequency is specified:

- Select `Voltage sensitivity` to specify output frequency from **Voltage sensitivity (Hz/V)** and **Free running frequency (Hz)**.
- Select `Output frequency vs. control voltage` to interpolate output frequency from **Control voltage (V)** vector versus **Output frequency (Hz)** vector.

Programmatic Use

- Use `set_param(gcb, 'SpecifyUsing', 'Voltage sensitivity')` to set **Specify using** to `Voltage sensitivity`.
- Use `set_param(gcb, 'SpecifyUsing', 'Output frequency vs. control voltage')` to set **Specify using** to `Output frequency vs. control voltage`.

Voltage sensitivity (Hz/V) — Measure of change in output frequency of VCO

100e6 (default) | positive real scalar

Measure of change in output frequency for input voltage change, specified as a positive real scalar with units in Hz/V. This parameter is also reported as **VCO voltage sensitivity** in the **Loop Filter** tab and is used to automatically calculate the filter component values of the loop filter.

Dependencies

To enable this parameter, select `Voltage sensitivity` in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'Kvco')` to view the current **Voltage sensitivity (Hz/V)** value.
- Use `set_param(gcb, 'Kvco', value)` to set **Voltage sensitivity (Hz/V)** to a specific value.

Data Types: `double`

Free running frequency (Hz) — VCO output frequency without control voltage

1.8e9 (default) | positive real scalar

Frequency of the VCO without any control voltage input (0 V), or the quiescent frequency, specified as a positive real scalar in Hz.

Dependencies

To enable this parameter, select `Voltage sensitivity` in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'Fo')` to view current **Free running frequency (Hz)** value.
- Use `set_param(gcb, 'Fo', value)` to set **Free running frequency (Hz)** to a specific value.

Data Types: `double`

Control voltage (V) — Control voltage values

[-5 0 5] (default) | real valued vector

Control voltage values of the VCO, specified as a real valued vector in volts.

Dependencies

To enable this parameter, select Output frequency vs. control voltage in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'ControlVoltage')` to view current **Control voltage (V)** value.
- Use `set_param(gcb, 'ControlVoltage', value)` to set **Control voltage (V)** to a specific value.

Data Types: double

Output frequency (Hz) – VCO output frequency values

[2e9 2.5e9 3e9] (default) | real valued vector

Output frequency of the values of the VCO, corresponding to the **Control voltage (V)** vector, specified in Hz.

Dependencies

To enable this parameter, select Output frequency vs. control voltage in **Specify using** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'OutputFrequency')` to view current **Output frequency (Hz)** value.
- Use `set_param(gcb, 'OutputFrequency', value)` to set **Output frequency (Hz)** to a specific value.

Data Types: double

Output amplitude gain – Ratio of VCO output voltage to input voltage

1 (default) | positive real scalar

Ratio of VCO output voltage to input voltage, specified as a positive real scalar. The input voltage has a nontunable value of 1 V.

Programmatic Use

- Use `get_param(gcb, 'Amplitude')` to view current **Output amplitude gain** value.
- Use `set_param(gcb, 'Amplitude', value)` to set **Output amplitude gain** to a specific value.

Data Types: double

Impairment**Add Phase-noise – Add phase noise as function of frequency**

off (default) | on

Select to introduce phase noise as a function of frequency to the VCO. By default, this option is deselected.

Phase noise frequency offset (Hz) – Frequency offsets of phase noise from carrier frequency

[30e3 100e3 1e6 3e6 10e6] (default) | real valued vector

Frequency offsets of the phase noise from the carrier frequency, specified as a real valued vector in Hz.

Dependencies

To enable this parameter, select **Add Phase-noise** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'Foffset')` to view the current **Phase noise frequency offset (Hz)** metric.
- Use `set_param(gcb, 'Foffset', value)` to set **Phase noise frequency offset (Hz)** to a specific metric.

Data Types: double

Phase noise level (dBc/Hz) — Phase noise power at specified frequency offsets relative to the carrier

[-56 -106 -132 -143 -152] (default) | real valued vector

Real valued vector specifying the phase noise power in a 1 Hz bandwidth centered at the specified frequency offsets relative to the carrier. The value is specified in dBc/Hz.

Dependencies

To enable this parameter, select **Add Phase-noise** in the **VCO** tab.

Programmatic Use

- Use `get_param(gcb, 'PhaseNoise')` to view the current **Phase noise level (dBc/Hz)** metric.
- Use `set_param(gcb, 'PhaseNoise', value)` to set **Phase noise level (dBc/Hz)** to a specific metric.

Data Types: double

Prescaler

Clock divider value — Value by which the clock divider divides the input frequency

100 (default) | real positive scalar

Value by which the clock divider divides the input frequency, specified as a real positive scalar.

Programmatic Use

- Use `get_param(gcb, 'N')` to view the current value of **Clock divider value**.
- Use `set_param(gcb, 'N', value)` to set **Clock divider value** to a specific value.

Data Types: double

Min clock divider value — Minimum value by which clock divider can divide input frequency

100 (default) | real positive scalar

Minimum value by which the clock divider can divide input frequency, specified as a real positive scalar. This parameter is also reported in the **Loop Filter** tab and is used to automatically calculate the filter component values of the loop filter.

Programmatic Use

- Use `get_param(gcb, 'Nmin')` to view the current value of **Min clock divider value**.
- Use `set_param(gcb, 'Nmin', value)` to set **Min clock divider value** to a specific value.

Data Types: double

Loop Filter**Filter component values — Determines how filter components are computed**

Automatic (default) | Manual

Select how filter components for the loop filter are computed:

- Select `Automatic` to automatically compute filter components from system specifications. Resistance and capacitance edit boxes in the **Loop Filter** tab are not editable if this option is selected. Rather, the filter component values are calculated from **Loop bandwidth (Hz)**, **Phase margin (degrees)**, **VCO voltage sensitivity**, **Charge pump current**, and **Min clock divider value**. By default, this option is selected.
- Select `Manual` to manually enter the resistance and capacitance values to design a customized loop filter.

Loop bandwidth (Hz) — Frequency at which magnitude of open loop transfer function becomes 1

2e6 (default) | positive real scalar

Frequency at which the magnitude of the open loop transfer function becomes 1, specified as a positive real scalar in Hz. Lower values of **Loop bandwidth (Hz)** result in reduced phase noise and reference spurs at the expense of longer lock time and less phase margin.

Dependencies

This parameter is only available when `Automatic` is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'Fc')` to view the current value of **Loop bandwidth (Hz)**.
- Use `set_param(gcb, 'Fc', value)` to set **Loop bandwidth (Hz)** to a specific value.

Data Types: double

Phase margin (degrees) — Phase of open loop transfer function at loop bandwidth subtracted from 180°

45 (default) | positive real scalar

Phase of the open loop transfer function at the loop bandwidth subtracted from 180°, specified as a positive real scalar in degrees. For optimum lock time, select a phase margin between 40° and 55°.

Dependencies

This parameter is only available when `Automatic` is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'Phi')` to view the current value of **Phase margin (degrees)**.
- Use `set_param(gcb, 'Phi', value)` to set **Phase margin (degrees)** to a specific value.

Data Types: double

Loop filter type — Order of the loop filter

3rd Order Passive (default) | 2nd Order Passive | 4th Order Passive

Order of the loop filter. Applies a second-, third-, or fourth-order passive RC loop filter in the PLL system.

C1 (F) — Capacitance 1

1.31e-15 (default) | positive real scalar

Capacitor value C1, specified as a positive real scalar in farad.

Dependencies

This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C1')` to view the current value of **C1 (F)**.
- Use `set_param(gcb, 'C1', value)` to set **C1 (F)** to a specific value.

Data Types: double

C2 (F) — Capacitance 2

1.44e-14 (default) | positive real scalar

Capacitor value C2, specified as a positive real scalar in farad.

Dependencies

This parameter is only editable when Manual is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C2')` to view the current value of **C2 (F)**.
- Use `set_param(gcb, 'C2', value)` to set **C2 (F)** to a specific value.

Data Types: double

C3 (F) — Capacitance 3

9.41e-17 (default) | positive real scalar

Capacitor value C3, specified as a positive real scalar in farad.

Dependencies

- To enable this parameter, select 3rd Order Passive or 4th Order Passive in **Loop filter type**.

- This parameter is only editable when `Manual` is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C3')` to view the current value of **C3 (F)**.
- Use `set_param(gcb, 'C3', value)` to set **C3 (F)** to a specific value.

Data Types: `double`

C4 (F) — Capacitance 4

331.5752 (default) | positive real scalar

Capacitor value C4, specified as a positive real scalar in farad.

Dependencies

- To enable this parameter, select `4th Order Passive` in **Loop filter type**.
- This parameter is only editable when `Manual` is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'C4')` to view the current value of **C4 (F)**.
- Use `set_param(gcb, 'C4', value)` to set **C4 (F)** to a specific value.

Data Types: `double`

R2 (ohms) — Resistance 2

1.33e+07 (default) | positive real scalar

Resistor value R2, specified as a positive real scalar in ohms.

Dependencies

This parameter is only editable when `Manual` is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'R2')` to view the current value of **R2 (ohms)**.
- Use `set_param(gcb, 'R2', value)` to set **R2 (ohms)** to a specific value.

Data Types: `double`

R3 (ohms) — Resistance 3

1.7e+08 (default) | positive real scalar

Resistor value R3, specified as a positive real scalar in ohms.

Dependencies

- To enable this parameter, select `3rd Order Passive` or `4th Order Passive` in **Loop filter type**.
- This parameter is only editable when `Manual` is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'R3')` to view the current value of **R3 (ohms)**.
- Use `set_param(gcb, 'R3', value)` to set **R3 (ohms)** to a specific value.

Data Types: double

R4 (ohms) — Resistance 4

28.1695 (default) | positive real scalar

Resistor value R4, specified as a positive real scalar in ohms.

Dependencies

- To enable this parameter, select **4th Order Passive** in **Loop filter type**.
- This parameter is only editable when **Manual** is selected for the **Filter Component values** parameter in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'R4')` to view the current value of **R4 (ohms)**.
- Use `set_param(gcb, 'R4', value)` to set **R4 (ohms)** to a specific value.

Data Types: double

Enable impairments — Add circuit impairments to simulation

off (default) | on

Select to add circuit impairments such as operating temperature to determine thermal noise to simulation. By default, this option is deselected.

Operating temperature (°C) — Temperature to determine the level of thermal noise

30 (default) | real scalar

Temperature of the resistor, specified as a real scalar in °C. **Operating temperature** determines the level of thermal (Johnson) noise.

Dependencies

To enable this parameter, select **Enable impairments** in the **Loop Filter** tab.

Programmatic Use

- Use `get_param(gcb, 'Temperature')` to view the current value of **Operating temperature**.
- Use `set_param(gcb, 'Temperature', value)` to set **Operating temperature** to a specific value.

Data Types: double

Export Loop Filter Component Values — Export loop filter component values

button

Click to export loop filter component values to a spreadsheet (XLS file) or as comma-separated values (CSV file).

Probe

PFD up and PFD down (pfd_up and pfd_down) – Select to probe PFD outputs

off (default) | on

Select to probe the PFD output wires (pfd_up and pfd_down) to view the response of the PFD.

Charge pump output (cp_out) – Select to probe charge pump output

off (default) | on

Select to probe the charge pump output wire (cp_out) to view the response of the Charge Pump.

Loop filter output (lf_out) – Select to probe loop filter output

off (default) | on

Select to probe loop filter output wire (lf_out) to view the response of the Loop Filter. The loop filter output provides the control voltage to the VCO.

Prescaler output (ps_out) – Select to probe prescaler output

off (default) | on

Select to probe the prescaler output wire (ps_out) to view the response of the Fractional Clock Divider with Accumulator.

Analysis

Open Loop Analysis – Plot the presimulation open loop analysis

on (default) | off

Select to plot the gain margin and phase margin of the PLL system before simulation. By default, this option is selected.

Closed Loop Analysis – Plot the presimulation closed loop analysis

off (default) | on

Select to plot the pole-zero map, loop bandwidth, step response, and impulse response of the PLL system before simulation. You must have a license to Control System Toolbox to plot the step response and impulse response of the PLL system. By default, this option is deselected.

Plot Loop Dynamics – Plot loop dynamics of PLL system

button

Click to plot the presimulation loop dynamics of the PLL system.

Version History

Introduced in R2019a

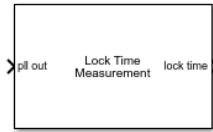
See Also

PFD | Charge Pump | Loop Filter | Single Modulus Prescaler | VCO

Lock Time Measurement

Measure time a PLL takes to reach target frequency within given tolerance

Library: Mixed-Signal Blockset / PLL / Measurements & Testbenches



Description

The Lock Time Measurement block measures the lock time of a phase-locked loop (PLL) system. Lock time is the time required by the PLL to reach the target frequency within the given error tolerance.

Ports

Input

pll out — Input clock signal

scalar

Input clock signal to the Lock Time Measurement block, specified as a scalar. The **pll out** port is connected to the output of a PLL system.

Data Types: double

Output

lock time — Time required to achieve lock in PLL

real positive scalar

Time at which locking takes place in a PLL system, returned as a real positive scalar in seconds.

Data Types: double

Parameters

Target frequency (Hz) — Target operating frequency of PLL to measure lock time

2.1e9 (default) | real positive scalar

Target operating frequency of the PLL device under test (DUT) to measure the lock time of the PLL, specified as a real positive scalar in Hz.

Programmatic Use

- Use `get_param(gcb, 'TargetFreq')` to view the current value of **Target frequency**.
- Use `set_param(gcb, 'TargetFreq', value)` to set **Target frequency** to a specific value.

Error tolerance (Hz) — Error tolerance for lock time measurement

1e5 (default) | real positive scalar

Error tolerance for lock time measurement, specified as a real positive scalar in Hz.

Programmatic Use

- Use `get_param(gcb, 'FreqErrorTol')` to view the current value of **Error tolerance**.
- Use `set_param(gcb, 'FreqErrorTol', value)` to set **Error tolerance** to a specific value.

Version History

Introduced in R2019a

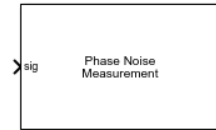
See Also

Fractional N PLL with Analog Compensation | Fractional N PLL with Delta Sigma Modulator | Integer N PLL with Dual Modulus Prescaler | Integer N PLL with Single Modulus Prescaler | PLL Testbench

Phase Noise Measurement

Compute phase noise at specific frequency offset vectors

Library: Mixed-Signal Blockset / PLL / Measurements & Testbenches



Description

The Phase Noise Measurement block measures and plots the phase noise profile for a time domain signal. It also displays the phase noise values for the specified frequency offset vector on the icon of the block.

Ports

Input

sig — Input signal

scalar

Input time domain signal to the Phase Noise Measurement block, specified as a scalar.

Data Types: double

Parameters

Resolution bandwidth (Hz) — Smallest positive frequency that can be resolved

30e3 (default) | real positive scalar

Smallest positive frequency that can be resolved, specified as a real positive scalar. The **Resolution bandwidth (Hz)** is used to determine window length for spectral analysis using the Welch method. For more information, see “Spectrum Estimation — Welch's Method”.

In general, **Resolution bandwidth (Hz)** should be less than or equal to the lowest offset frequency value.

Programmatic Use

- Use `get_param(gcb, 'ResBandwidth')` to view the current value of **Resolution bandwidth (Hz)**.
- Use `set_param(gcb, 'ResBandwidth', value)` to set **Resolution bandwidth (Hz)** to a specific value.

No. of spectral averages — Number of spectral averages

4 (default) | positive integer scalar

Number of spectral averages, specified as a positive integer scalar. The `dsp.SpectrumEstimator` System object™ used by the Phase Noise Measurement block computes the current power spectrum

or power density spectrum estimate by averaging over the number specified by **No. of spectral averages**.

Programmatic Use

- Use `get_param(gcb, 'SpectralAverages')` to view the current value of **No. of spectral averages**.
- Use `set_param(gcb, 'SpectralAverages', value)` to set **No. of spectral averages** to a specific value.

Hold off time (s) – Delays measurement analysis to avoid transients

0 (default) | real nonnegative scalar

Delays measurement analysis by the specified amount of time to avoid corruption by transients, specified as a real nonnegative scalar in s.

Programmatic Use

- Use `get_param(gcb, 'HoldOffTime')` to view the current value of **Hold off time (s)**.
- Use `set_param(gcb, 'HoldOffTime', value)` to set **Hold off time (s)** to a specific value.

Frequency offset vector (Hz) – Frequency points relative to fundamental frequency where phase noise is calculated

[30e3 100e3 1e6 3e6 10e6] (default) | real valued vector

Frequency points relative to fundamental frequency where phase noise is calculated, specified as a real valued vector in Hz.

Programmatic Use

- Use `get_param(gcb, 'PhaseNoiseFreqOffset')` to view the current value of **Frequency offset vector (Hz)**.
- Use `set_param(gcb, 'PhaseNoiseFreqOffset', value)` to set **Frequency offset vector (Hz)** to a specific value.

Specify target – Specify target phase noise vector

off (default) | on

Specify the target phase noise vector for a given frequency offset vector. By default, this option is deselected

Phase noise vector (dBc/Hz) – Target phase noise profile for given frequency offset vector

[-300 -300 -300 -300 -300] (default) | real valued vector

Target phase noise profile for given frequency offset vector, specified as a real valued vector in dBc/Hz. **Phase noise vector (dBc/Hz)** is the phase noise power in a 1 Hz bandwidth centered at the specified frequency offsets relative to the carrier

Dependencies

To enable this parameter, select the **Specify target** parameter.

Programmatic Use

- Use `get_param(gcb, 'TargetPhaseNoise')` to view the current value of **Phase noise vector (dBc/Hz)**.

- Use `set_param(gcb, 'TargetPhaseNoise', value)` to set **Phase noise vector (dBc/Hz)** to a specific value.

Set stop time – Set recommended minimum simulation stop time as model stop time
button

Click to set the **Recommended min. simulation stop time (s)** reported by the Phase Noise Measurement block as the model stop time.

Plot phase noise – Plot phase noise profile of input signal
button

Click to plot the measured phase noise profile of the input signal. Target phase noise profile is also overlaid if the **Specify target** option is selected. You can plot the phase noise profile any time during simulation.

Export measurement results – Export measurement results after simulation
button

Click to export measurement results to an excel spreadsheet (XLS) or as comma-separated values (CSV) after the simulation is complete.

Algorithms

The Phase Noise Measurement block uses the zero crossing points of a signal to measure the phase noise. From the zero crossing points, the phase error (φ) is extracted both at the rising and falling edge of the signal. The captured phase error is used to generate a periodic signal ($\sin(\varphi)$). This periodic signal is interpolated using a fixed time step proportional to the maximum phase noise frequency offset. The power density of the modified phase noise signal, using the `dsp.SpectrumEstimator` System object, directly provides the phase noise profile of the signal of interest.

Version History

Introduced in R2019a

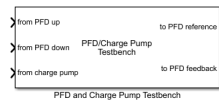
See Also

Fractional N PLL with Analog Compensation | Fractional N PLL with Delta Sigma Modulator | Integer N PLL with Dual Modulus Prescaler | Integer N PLL with Single Modulus Prescaler | PLL Testbench

PFD and Charge Pump Testbench

Generic test environment for phase/frequency detectors and charge pumps

Library: Mixed-Signal Blockset / PLL / Measurements & Testbenches



Description

The PFD and Charge Pump Testbench block evaluates the behavioral model of a PFD and charge pump. A single stimulus generator determines whether the PFD is operating in the phase offset mode or frequency offset mode.

The PFD and Charge Pump Testbench block generates the stimulus to drive the device under test (DUT) from the **Stimulus** tab. The setup parameters for validating the DUT are defined in the **Setup** tab and the target validation metrics are defined in the **Target Metrics** tab.

The testbench measure PFD performance metrics such as deadband, linear range, and timing impairments. It also measures charge pump performance metrics such as sensitivity, phase offset, and spur current.

Ports

Input

from PFD up — Measures PFD reference frequency
scalar

Measures the reference frequency of the PFD block.

Data Types: double

from PFD down — Measures PFD feedback frequency
scalar

Measures the feedback frequency of the PFD block.

Data Types: double

from charge pump — Measures charge pump output current
scalar

Measures the output current of the Charge Pump block.

Data Types: double

Output

to PFD reference — Provides reference frequency to PFD
scalar

Provides reference frequency to the PFD to determine phase error.

Data Types: double

to PFD feedback — Provides feedback frequency to PFD

scalar

Provides feedback frequency to the PFD block.

Data Types: double

Parameters

Stimulus

Phase sweep (°) — Maximum phase excursion from phase offset

300 (default) | real positive scalar

Maximum phase excursion from phase offset, specified as a real positive scalar in degrees.

Programmatic Use

- Use `get_param(gcb, 'PhaseSweep')` to view the current value of **Phase sweep**.
- Use `set_param(gcb, 'PhaseSweep', value)` to set **Phase sweep** to a specific value.

Data Types: double

Phase offset (°) — Relative phase value at center of phase sweep

0 (default) | real scalar

Relative phase value at the center of the phase sweep, specified as a real scalar in degrees.

Programmatic Use

- Use `get_param(gcb, 'PhaseOffset')` to view the current value of **Phase offset**.
- Use `set_param(gcb, 'PhaseOffset', value)` to set **Phase offset** to a specific value.

Data Types: double

Clock frequency (Hz) — Desired reference clock frequency

2.5e9 Hz (default) | real positive scalar

Desired clock frequency for the reference counter output, specified as a real positive scalar in Hz.

Programmatic Use

- Use `get_param(gcb, 'ClockFrequency')` to view the current value of **Clock frequency**.
- Use `set_param(gcb, 'ClockFrequency', value)` to set **Clock frequency** to a specific value.

Data Types: double

Number of phases in sweep — Total number of evenly spaced phase offsets to simulate

2000 (default) | real positive scalar

The number of evenly spaced phase offsets in a sweep of phase offset, specified as a real positive scalar.

Programmatic Use

- Use `get_param(gcb, 'NPhases')` to view the current value of **Number of phases in sweep**.
- Use `set_param(gcb, 'NPhases', value)` to set **Number of phases in sweep** to a specific value.

Data Types: double

Duty cycle (%) – Duty cycle of stimulus clock

50 (default) | real positive scalar

Duty cycle for the stimulus clock at both reference and feedback ports, specified as real positive scalar.

Programmatic Use

- Use `get_param(gcb, 'DutyCycle')` to view the current value of **Duty cycle**.
- Use `set_param(gcb, 'NPhases', value)` to set **Duty cycle** to a specific value.

Data Types: double

Setup**Plot figures on top after simulation – Plot figures on top after simulation**

on (default) | off

Select to plot the figures on the top of all other windows after simulation. By default, this option is selected.

Report PFD metrics – Report PFD metrics

on (default) | off

Select to display the PFD metrics (Deadband, Linear Range, and Propagation delay) on the icon of the PFD and Charge Pump Testbench. By default, this option is selected

Data Types: double

Report Charge Pump metrics – Report Charge Pump metrics

on (default) | off

Select to display the Charge Pump metrics (Sensitivity, Phase offset, and Spur current) on the icon of the PFD and Charge Pump Testbench. By default, this option is selected

Data Types: double

Logic Threshold (V) – Switching threshold at input of charge pump

0.5 (default) | real scalar

Switching threshold at the input of a charge pump, specified as a real scalar in V. It is the voltage at which the timing of rising and falling edges is measured.

Programmatic Use

- Use `get_param(gcb, 'VSwitch')` to view the current value of **Logic Threshold**.
- Use `set_param(gcb, 'VSwitch', value)` to set **Logic Threshold** to a specific value.

Data Types: double

Enable increased buffer size — Enable increased buffer size

off (default) | on

Select to enable increased buffer size during simulation. This increases the buffer size of the Variable Pulse Delay and Logic Decision blocks inside the PFD and Charge Pump Testbench. By default, this option is deselected.

Buffer size — Number of samples of the input buffering available during simulation

5 (default) | positive integer scalar

Number of samples of the input buffering available during simulation, specified as a positive integer scalar. This sets the Variable Pulse Delay and Logic Decision inside the PFD and Charge Pump Testbench.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when **Enable increased buffer size** option is selected in the **Configuration** tab.

Programmatic Use

- Use `get_param(gcb, 'NBuffer')` to view the current value of **Buffer size**.
- Use `set_param(gcb, 'NBuffer', value)` to set **Buffer size** to a specific value.

Target Metrics**PFD Metrics****Target deadband (°) — Maximum acceptable size of reduced sensitivity region near zero phase offset**

0 (default) | real nonnegative scalar

Maximum acceptable size of reduced sensitivity region near zero phase offset, specified as a real nonnegative scalar in degrees. It refers to the size of the deadband region.

Programmatic Use

- Use `get_param(gcb, 'TgtDeadband')` to view the current value of **Target deadband**.
- Use `set_param(gcb, 'TgtDeadband', value)` to set **Target deadband** to a specific value.

Data Types: double

Target linear range (°) — Maximum phase offset at which the output remains approximately equal to the input offset

290 (default) | real positive scalar

Maximum phase offset at which the output remains approximately equal to the input offset, specified as a real positive scalar in degrees.

Programmatic Use

- Use `get_param(gcb, 'TgtRange')` to view the current value of **Target linear range**.

- Use `set_param(gcb, 'TgtRange', value)` to set **Target linear range** to a specific value.

Data Types: double

Target propagation delay (s) — Maximum acceptable delay from input to output

60e-12 (default) | real positive scalar

Maximum acceptable delay from the input to output, specified as a real positive scalar in s.

Programmatic Use

- Use `get_param(gcb, 'TgtPropDelay')` to view the current value of **Target propagation delay**.
- Use `set_param(gcb, 'TgtPropDelay', value)` to set **Target propagation delay** to a specific value.

Data Types: double

Target rise/fall time (s) — Maximum acceptable 20% - 80% rise/fall time

30e-12 (default) | real positive scalar

Maximum acceptable 20% - 80% rise/fall time, specified as a real positive scalar in s.

Programmatic Use

- Use `get_param(gcb, 'TgtRiseFall')` to view the current value of **Target rise/fall time**.
- Use `set_param(gcb, 'TgtRiseFall', value)` to set **Target rise/fall time** to a specific value.

Data Types: double

Charge Pump Metrics

Target sensitivity (A/°) — Maximum acceptable charge pump sensitivity

1e-9 (default) | real positive scalar

Maximum acceptable charge pump sensitivity, specified as a real positive scalar in A/°.

Programmatic Use

- Use `get_param(gcb, 'TgtSensitivity')` to view the current value of **Target sensitivity**.
- Use `set_param(gcb, 'TgtSensitivity', value)` to set **Target sensitivity** to a specific value.

Data Types: double

Target phase offset (°) — Maximum acceptable phase offset at output of charge pump

10 (default) | real positive scalar

Maximum acceptable phase offset at the output of a charge pump, specified as a real positive scalar in degrees.

Programmatic Use

- Use `get_param(gcb, 'TgtOffset')` to view the current value of **Target phase offset**.
- Use `set_param(gcb, 'TgtOffset', value)` to set **Target phase offset** to a specific value.

Data Types: double

Target spur current (A) – Charge pump output current magnitude

1e-7 (default)

Magnitude of the output current of the charge pump at the reference frequency.

Programmatic Use

- Use `get_param(gcb, 'TgtSpurCurrent')` to view the current value of **Target spur current**.
- Use `set_param(gcb, 'TgtSpurCurrent', value)` to set **Target spur current** to a specific value.

Data Types: double

Version History

Introduced in R2019a

References

[1] Banerjee, Dean. *PLL Performance, Simulation and Design*. Indianapolis, IN: Dog Ear Publishing, 2006.

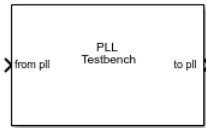
See Also

PFD | Charge Pump

PLL Testbench

Validate PLL system by measuring operating frequency, lock time, and phase noise

Library: Mixed-Signal Blockset / PLL / Measurements & Testbenches



Description

The PLL Testbench block provides input stimulus in the form of a clock signal to a phase-locked loop (PLL) system. The testbench also validates the performance of the PLL system by comparing the operating frequency, lock time, and phase noise against the target metrics.

The PLL Testbench block generates the stimulus to drive the device under test (DUT) from the **Stimulus** tab. The setup parameters for validating the DUT are defined in the **Setup** tab and the target validation metrics are defined in the **Target Metric** tab.

Ports

Input

from pll — Input clock signal

scalar

Input clock signal to the PLL Testbench, specified as a scalar. The **from pll** port is connected to the output of a PLL system.

Data Types: double

Output

to pll — Output clock signal

scalar

Output clock signal, returned as a sine or square wave as specified in the **Signal type** parameter. The signal at the **to pll** port provides the stimulus to a PLL system.

Data Types: double

Parameters

Set stop time — Set recommended minimum simulation stop time as model stop time

button

Click to set the **Recommended min. simulation stop time (s)** reported by the PLL Testbench block as the model stop time.

Dependencies

This button is only available when **Phase noise** measurement option is selected in the **Setup** tab.

Plot phase noise profile — Plot phase noise profile of the PLL DUT

button

Click to plot the phase noise profile of the PLL device (DUT) and to compare it with the user-defined phase noise profile after simulation is complete.

Dependencies

This button is only available when **Phase noise** measurement option is selected in the **Setup** tab.

Export measurement results — Export measurement results after simulation

button

Click to export measurement results to an excel spreadsheet (XLS) or as comma-separated values (CSV) after the simulation is complete.

Stimulus**Signal type — Shape of clock signal going to PLL DUT**

Square (default) | Sine

Shape of the clock signal going to the input of the PLL device under test (DUT). Choose between a Sine or a Square wave.

Programmatic Use

- Use `get_param(gcb, 'SignalType')` to view the current value of **Signal type**.
- Use `set_param(gcb, 'SignalType', value)` to set **Signal type** to a specific value.

Amplitude (V) — Maximum value of stimulus signal at PLL input

1 (default) | real positive scalar

Maximum value of the stimulus signal at PLL input, specified as a real positive scalar.

Programmatic Use

- Use `get_param(gcb, 'InputAmplitude')` to view the current value of **Amplitude (V)**.
- Use `set_param(gcb, 'InputAmplitude', value)` to set **Amplitude (V)** to a specific value.

Frequency (Hz) — Frequency of stimulus signal at PLL input

30e6 (default) | real positive scalar

Frequency of the stimulus signal at PLL input, specified as a real positive scalar.

Programmatic Use

- Use `get_param(gcb, 'ClkFreq')` to view the current value of **Frequency (Hz)**.
- Use `set_param(gcb, 'ClkFreq', value)` to set **Frequency (Hz)** to a specific value.

Setup**Frequency of operation — Measure operating frequency of PLL DUT**

on (default) | off

Select to measure the operating frequency of the PLL DUT. By default, this option is selected.

Lock time — Measure locking time of PLL DUT

off (default) | on

Select to measure the locking time of the PLL DUT with user-specified error tolerance. By default, this option is deselected.

Target frequency of operation (Hz) — Target operating frequency of PLL DUT to calculate lock time

2.1e9 (default) | real positive scalar

Target operating frequency of the PLL DUT to calculate the lock time of the PLL, specified as a real positive scalar in Hz.

Dependencies

To enable this parameter, select the **Lock time** measurement option in the **Setup** tab.

Programmatic Use

- Use `get_param(gcb, 'ExpectedFreq')` to view the current value of **Target frequency of operation(Hz)**.
- Use `set_param(gcb, 'ExpectedFreq', value)` to set **Target frequency of operation (Hz)** to a specific value.

Error tolerance (Hz) — Error tolerance for lock time measurement

1e6 (default) | real positive scalar

Error tolerance for lock time measurement, specified as a real positive scalar in Hz.

Dependencies

To enable this parameter, select the **Lock time** measurement option in the **Setup** tab.

Programmatic Use

- Use `get_param(gcb, 'FreqErrorTol')` to view the current value of **Error tolerance (Hz)**.
- Use `set_param(gcb, 'FreqErrorTol', value)` to set **Error tolerance (Hz)** to a specific value.

Phase noise — Measure phase noise level of PLL DUT

off (default) | on

Select to measure the phase noise level of the PLL DUT at user defined frequency offset points. By default, this option is deselected.

Resolution bandwidth (Hz) — Smallest positive frequency that can be resolved

200e3 (default) | real positive scalar

Smallest positive frequency that can be resolved, specified as a real positive scalar. The **Resolution bandwidth (Hz)** is used to determine window length for spectral analysis using the Welch method. For more information, see “Spectrum Estimation — Welch's Method”.

In general, **Resolution bandwidth (Hz)** should be less than or equal to the lowest offset frequency value.

Dependencies

To enable this parameter, select the **Phase noise** measurement option in the **Setup** tab.

Programmatic Use

- Use `get_param(gcb, 'ResBandwidth')` to view the current value of **Resolution bandwidth (Hz)**.
- Use `set_param(gcb, 'ResBandwidth', value)` to set **Resolution bandwidth (Hz)** to a specific value.

No. of spectral averages — Number of spectral averages

4 (default) | positive integer scalar

Number of spectral averages, specified as a positive integer scalar. The `dsp.SpectrumEstimator` System object used by the Phase Noise Measurement subsystem inside the PLL Testbench block computes the current power spectrum or power density spectrum estimate by averaging over the number specified by **No. of spectral averages**.

Dependencies

To enable this parameter, select the **Phase noise** measurement option in the **Setup** tab.

Programmatic Use

- Use `get_param(gcb, 'SpectralAverages')` to view the current value of **No. of spectral averages**.
- Use `set_param(gcb, 'SpectralAverages', value)` to set **No. of spectral averages** to a specific value.

Hold off time (s) — Delays measurement analysis to avoid transients

0 (default) | real nonnegative scalar

Delays measurement analysis by the specified amount of time to avoid corruption by transients, specified as a real nonnegative scalar in s.

To enable this parameter, select the **Phase noise** measurement option in the **Setup** tab.

Programmatic Use

- Use `get_param(gcb, 'HoldOffTime')` to view the current value of **Hold off time (s)**.
- Use `set_param(gcb, 'HoldOffTime', value)` to set **Hold off time (s)** to a specific value.

Frequency offset vector (Hz) — Frequency points relative to fundamental frequency where phase noise is calculated

[300e3 1e6 3e6 10e6] (default) | real valued vector

Frequency points relative to fundamental frequency where phase noise is calculated, specified as a real valued vector in Hz. This values are also reported in the **Target Metrics** tab as **Phase noise frequency offset (Hz)**.

To enable this parameter, select the **Phase noise** measurement option in the **Setup** tab.

Programmatic Use

- Use `get_param(gcb, 'PhaseNoiseFreqOffset')` to view the current value of **Frequency offset vector (Hz)**.

- Use `set_param(gcb, 'PhaseNoiseFreqOffset', value)` to set **Frequency offset vector (Hz)** to a specific value.

Target Metrics

Frequency of operation — Target operating frequency at which PLL needs to lock

2.1e9 (default) | real positive scalar

Target operating frequency at which PLL DUT needs to lock, specified as a real positive scalar in Hz.

If you select the **Lock time** as a measurement option, the **Frequency of operation** is reported from the **Target frequency of operation (Hz)** parameter.

Dependencies

To enable this parameter, select the **Frequency of operation** measurement option in the **Setup** tab.

Programmatic Use

- Use `get_param(gcb, 'TargetFreq')` to view the current value of **Frequency of operation**.
- Use `set_param(gcb, 'TargetFreq', value)` to set **Frequency of operation** to a specific value.

Lock time (s) — Maximum time in which PLL DUT needs to get locked

3e-6 (default) | real nonnegative scalar

Maximum time in which PLL DUT needs to get locked, specified as a real nonnegative scalar in s.

Dependencies

To enable this parameter, select the **Lock time** measurement option in the **Setup** tab.

Programmatic Use

- Use `get_param(gcb, 'TargetLockTime')` to view the current value of **Lock time**.
- Use `set_param(gcb, 'TargetLockTime', value)` to set **Lock time** to a specific value.

Phase noise (dBc/Hz) — Target noise power corresponding to frequency offset vector

[-85, -125, -150, -180] (default) | real valued vector

Target noise power level relative to the carrier in a 1 Hz frequency bandwidth centered at the frequencies specified in the **Frequency offset vector (Hz)** parameter, specified as a real valued vector in dBc/Hz.

Dependencies

To enable this parameter, select the **Phase noise** measurement option in the **Setup** tab.

Programmatic Use

- Use `get_param(gcb, 'TargetPhaseNoiseVector')` to view the current value of **Phase noise**.
- Use `set_param(gcb, 'TargetPhaseNoiseVector', value)` to set **Phase noise** to a specific value.

Version History

Introduced in R2019a

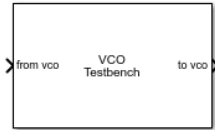
See Also

Fractional N PLL with Analog Compensation | Fractional N PLL with Delta Sigma Modulator | Integer N PLL with Dual Modulus Prescaler | Integer N PLL with Single Modulus Prescaler

VCO Testbench

Validate voltage controlled oscillator (VCO) by measuring phase noise metrics or VCO characteristics

Library: Mixed-Signal Blockset / PLL / Measurements & Testbenches



Description

The VCO Testbench block validates the VCO device under test (DUT) by measuring one of the two target metrics: phase noise, or voltage sensitivity and quiescent frequency. You can use the testbench to validate a VCO of your own implementation, or you can use the VCO block from the Mixed-Signal Blockset.

The VCO Testbench block generates the stimulus (control voltage) to drive the device under test (DUT) from the **Stimulus** tab. The setup parameters for validating the DUT are defined in the **Setup** tab and the target validation metrics are defined in the **Target Metric** tab.

To take the full advantage of the VCO testbench capabilities by using autofill parameters options, use only two blocks, the VCO DUT and the VCO Testbench in the Simulink model.

Ports

Input

from vco — Input signal

scalar

Input signal, which flows from the output of the VCO DUT. This input signal is used to calculate either the phase noise metric, or the voltage sensitivity and free running frequency of the VCO, depending on the **Measurement** option chosen.

Data Types: double

Output

to vco — Output signal

scalar

Output signal, which provides control voltage stimulus to the input of VCO DUT.

Data Types: double

Parameters

Measurement — Target metric to be measured

Phase noise (default) | Kvco and Fo

Determines which of the two given target metrics is being measured.

Select **Phase noise** if you want the testbench to measure and compare phase noise to a target phase noise profile.

Select **Kvco** and **Fo** if you want the testbench to compute voltage sensitivity (K_{vco}) and quiescent frequency (F_o) using a range of control voltages.

Programmatic Use

- Use `get_param(gcb, 'MeasurementOptions')` to view the current **Measurement** option.
- Use `set_param(gcb, 'MeasurementOptions', value)` to set **Measurement** to a specific option.

Set stop time – Set recommended minimum simulation stop time as model stop time button

Click to set the **Recommended min. simulation stop time (s)** reported by the VCO Testbench block as the model stop time.

Dependencies

This button is only available when you select **Phase noise** as the **Measurement** option.

Plot measurement – Plot measurements button

Plots the relevant VCO metrics based on the **Measurement** options.

Selecting **Phase noise** in **Measurement** plots phase noise profile of VCO.

Selecting **Kvco** and **Fo** in **Measurement** plots the VCO characteristics and K_{vco} .

Export measurement results – Export measurement results button

Exports the relevant VCO metrics based on the **Measurement** options to an excel file.

Selecting **Phase noise** in **Measurement** exports the phase noise profile of VCO.

Selecting **Kvco** and **Fo** in **Measurement** exports the VCO characteristics and K_{vco} .

Stimulus

Control voltage (V) – VCO control voltage 2 (default) | scalar

Control voltage provided by VCO Testbench, expressed as a scalar in V. The value specified in **Control Voltage (V)** flows through the **to vco** port that provides the input of VCO.

Dependencies

To enable this parameter, select **Phase noise** as the **Measurement** option.

Programmatic Use

- Use `get_param(gcb, 'ControlVoltage')` to view current **Control Voltage (V)** value.

- Use `set_param(gcb, 'ControlVoltage', value)` to set **Control Voltage (V)** to a specific value.

Data Types: double

Range of control voltage (V) — Range of VCO control voltages

[2 7] (default) | two-element row vector

Control voltage provided by VCO Testbench, expressed as a two-element row vector in V. This parameter specifies the minimum and maximum values of control voltage, which is used to generate ten control voltage value points, including the provided values. These ten values of control voltage are sent to VCO input to measure K_{vco} and F_o .

Dependencies

To enable this parameter, select `Kvco` and `Fo` as the **Measurement** option.

Programmatic Use

- Use `get_param(gcb, 'ControlVoltageRange')` to view the current **Range of control voltage (V)** values.
- Use `set_param(gcb, 'ControlVoltage', value)` to set **Range of control voltage (V)** to specific values.

Data Types: double

Setup

Autofill setup parameters — Automatically calculate setup parameters for phase noise measurement

button

Click this button to automatically populate setup parameters (**Resolution bandwidth (Hz)** and **No. of spectral averages**) for phase noise measurement.

If the DUT is a VCO from Mixed-Signal Blockset, the `dsp.SpectrumEstimator` System object used by the VCO Testbench block for phase noise measurement automatically calculates setup parameters based on the VCO specifications.

Dependencies

- This button only works if the VCO DUT is a VCO block from Mixed-Signal Blockset.
- This button is only available when you select `Phase noise` as the **Measurement** option.

Resolution bandwidth (Hz) — Smallest positive frequency that can be resolved

30e3 (default) | real positive scalar

Smallest positive frequency that can be resolved, specified as a real positive scalar. The **Resolution bandwidth (Hz)** is used to determine window length for spectral analysis using the Welch method. For more information, see “Spectrum Estimation — Welch's Method”.

In general, **Resolution bandwidth (Hz)** should be less than or equal to the lowest offset frequency value.

If the DUT is a VCO block from the Mixed-Signal Blockset library, you can use the **Autofill setup parameters** button to automatically calculate **Resolution bandwidth (Hz)**.

Dependencies

To enable this parameter, select Phase noise as the **Measurement** option.

Programmatic Use

- Use `get_param(gcb, 'ResBandwidth')` to view current **Resolution bandwidth (Hz)** value.
- Use `set_param(gcb, 'ResBandwidth', value)` to set **Resolution bandwidth (Hz)** to a specific value.

Data Types: double

No. of spectral averages — Number of spectral averages

8 (default) | positive integer scalar

Number of spectral averages, specified as a positive integer scalar. The `dsp.SpectrumEstimator` System object used by the Phase Noise Measurement subsystem inside the VCO Testbench block computes the current power spectrum or power density spectrum estimate by averaging over the number specified by **No. of spectral averages**.

If the DUT is a VCO block from the Mixed-Signal Blockset library, you can use the **Autofill setup parameters** button to automatically calculate number of spectral averages.

Dependencies

To enable this parameter, select Phase noise as the **Measurement** option.

Programmatic Use

- Use `get_param(gcb, 'SpectralAverages')` to view current **No. of spectral averages** value.
- Use `set_param(gcb, 'SpectralAverages', value)` to set **No. of spectral averages** to a specific value.

Data Types: double

Hold-off time (s) — Delays measurement analysis to avoid transients

0 (default)

Hold-off period, specified as a nonnegative scalar in s. **Hold-off time** delays measurement analysis by the specified amount of time to avoid corrupting simulation results due to transients.

Programmatic Use

- Use `get_param(gcb, 'HoldOffTime')` to view the current **Hold-off time (s)** value.
- Use `set_param(gcb, 'HoldOffTime', value)` to set **Hold-off time (s)** to a specific value.

Target Metric

Autofill target metric — Automatically populate target phase noise metric
button

Click this button to automatically populate the target phase noise metric from VCO specifications.

Dependencies

- This button will only work if the VCO DUT is a VCO block from Mixed-Signal Blockset.

- This button is only available when you select Phase noise as **Measurement** option.

Phase noise frequency offset vector (Hz) — Phase noise frequency offset vector
[30e3 100e3 1e6 3e6 10e6] (default) | real valued vector

The frequency offsets of phase noise from the carrier frequency, collected from the data sheet, specified as a real valued vector in Hz.

If the DUT is a VCO block from the Mixed-Signal Blockset library, you can use the **Autofill target metric** button to automatically transfer VCO phase noise frequency offset vector values to the VCO Testbench.

Dependencies

To enable this parameter, select Phase noise as the **Measurement** option.

Programmatic Use

- Use `get_param(gcb, 'PhaseNoiseFreqOffset')` to view the current **Phase noise frequency offset vector (Hz)** value.
- Use `set_param(gcb, 'PhaseNoiseFreqOffset', value)` to set **Phase noise frequency offset vector (Hz)** to a specific value.

Data Types: double

Phase noise vector (dBc/Hz) — Phase noise vector
[-45 -55 -65 -75 -100] (default) | real valued vector

The phase noise power in a 1 Hz bandwidth centered at the specified frequency offsets relative to the carrier, collected from the data sheet, specified as a real valued vector in dBc/Hz. The elements of **Phase noise vector (dBc/Hz)** corresponds to relative elements in the **Phase noise frequency offset vector (Hz)**.

If the DUT is a VCO block from the Mixed-Signal Blockset library, you can use the **Autofill target metric** button to automatically transfer VCO phase noise vector values to the testbench.

Dependencies

To enable this parameter, select Phase noise as the **Measurement** option.

Programmatic Use

- Use `get_param(gcb, 'PhaseNoiseVector')` to view the current **Phase noise vector (dBc/Hz)** value.
- Use `set_param(gcb, 'PhaseNoiseVector', value)` to set **Phase noise vector (dBc/Hz)** to a specific value.

More About

Inside VCO Testbench

The VCO Testbench subsystem block consists of a Step block and a subsystem that contains two variants: a Phase-noise measurement subsystem and a Frequency detector subsystem. The variant subsystems are enabled based on the selection of the **Measurement** option in the block parameters dialog box.

Inside the Step block, the **Step time** is set to **Hold-off time (s)** value and delays the measurement analysis by the specified time.

Phase Noise Measurement

The Phase-noise subsystem is activated when Phase noise is selected as the **Measurement** option. The subsystem uses the zero crossing points of a signal to measure the phase noise. From the zero crossing points, the phase error (ϕ) is extracted both at the rising and falling edge of the signal. The captured phase error is used to generate a periodic signal ($\sin(\phi)$). This periodic signal is interpolated using a fixed time step proportional to the maximum phase noise frequency offset. The power density of the modified phase noise signal, using the `dsp.SpectrumEstimator` System object, directly provides the phase noise profile of the signal of interest.

Voltage Sensitivity and Free Running Frequency Measurement

The Frequency detector subsystem is activated when K_{vco} and F_o is selected as the **Measurement** option. The subsystem spreads the range of control voltage over 10 points using their minimum and maximum values. A VCO characteristics curve is generated for each point. Voltage sensitivity (K_{vco}) is calculated by taking the average of slopes between each points. Quiescent frequency (F_o) is extrapolated at control voltage zero.

Version History

Introduced in R2019a

References

[1] Banerjee, Dean. *PLL Performance, Simulation and Design*. Indianapolis, IN: Dog Ear Publishing, 2006.

See Also

VCO

Blocks: Utilities

Linear Circuit Wizard

Generate or modify linear circuit

Library: Mixed-Signal Blockset / Utilities



Description

Use the Linear Circuit Wizard block to create or modify a linear, time-invariant circuit such as a custom-design filter or a circuit with extracted parasitics. Using this block, you can parse a SPICE netlist file that describes the circuit elements and the circuit nodes. You can then use the block parameter dialog box to refine the input and output ports and to model device noise. You can review the port and device noise definitions from a base workspace output structure.

The Linear Circuit Wizard block sets up and solves linear circuit equations to produce a set of Laplace domain transfer functions. You can review these transfer functions either through magnitude vs. frequency plots or through pole-zero location reports.

The Linear Circuit Wizard block creates and configures a MATLAB System block to represent your circuit as a new block, which is independent of the Linear Circuit Wizard block, in a Simulink model. This MATLAB System block is configured using the port and device noise definitions and the results of the circuit analysis. When you start the simulation, the MATLAB System block uses fixed-step discrete sample time to convert the Laplace domain transfer functions into recursive digital filter coefficients for execution during the simulation.

Parameters

Circuit design name — Name of circuit displayed on block icon

'Linear Circuit' (default) | character vector | string scalar

Name of the circuit that is displayed on the block icon, specified as a character vector or string scalar.

Programmatic Use

Block parameter: CircuitDesignName

Type: character vector

Value: character string

Default: 'Linear Circuit'

Block name — Instance name displayed under block icon

'Linear Circuit' (default) | character vector | string scalar

Instance name to be displayed under the block icon, specified as a character vector or string scalar.

Programmatic Use

Block parameter: BlockName

Type: character vector

Value: character string

Default: 'Linear Circuit'

Netlist file name — Name of netlist file that defines circuit

character vector | string scalar

Name of the netlist file (SPICE syntax) that defines the linear circuit, specified as a character vector or string scalar.

Programmatic Use

Block parameter: NetlistFileName

Type: character vector

Value: character string

Parse netlist file and redefine ports — Parse named netlist file

button

Click to parse the named netlist file. This action redefines the existing ports according to the new SPICE syntax. For more information, see “Model Linear Circuit Response from SPICE Netlist”.

Temperature (°C) — Operating temperature of device

100 (default) | real scalar

Operating temperature of the linear circuit device, specified as a real scalar in °C. **Temperature (°C)** is used to model device noise generated by the linear circuit. For more information about device noise, see “Define Device Noise Using Linear Circuit Wizard”.

Programmatic Use

Block parameter: Temperature

Type: character vector

Values: real scalar | double

Default: 100

More About

SPICE Netlist

The Linear Circuit Wizard block solves a linear, time-invariant circuit described by a SPICE netlist. The block supports a limited set of SPICE circuit elements and netlist statement syntaxes. For more information about supported circuit elements and netlist statement syntaxes, see “Model Linear Circuit Response from SPICE Netlist”.

Ports

The ports in the Linear Circuit Wizard block are initially parsed from the SPICE netlist. Once the ports are defined, you can add, delete, or modify the order of the ports from the Linear Circuit Wizard block parameter dialog.. The supported ports can either be input or output, and either be voltage or current.

Note The signals at both the input and output ports must use a fixed step discrete sample time equal to the block’s sample time.

The most common interface definition for an analog circuit is a set of voltages and currents defined at circuit nodes. To create the interface to digital logic and behavioral models from the interface to

analog circuits requires some conversion. The Linear Circuit Wizard block constructs the conversion between ports and analog circuit nodes based on the ports definitions that you supply. This enables the Linear Circuit Wizard block to produce a block in your Simulink model that can be connected directly to other types of blocks. For more information, see “Ports Supported in Linear Circuit Wizard”.

Device Noise

The Linear Circuit Wizard block can model the noise generated by the transistors and resistors in the analog circuit. The spectral density of the device noise can include accurate modeling of flicker noise. The circuit model includes the transfer function from the device noise source to the circuit ports. For more information, see “Define Device Noise Using Linear Circuit Wizard”.

Results

The Linear Circuit Wizard block outputs a linear circuit block that can be added to the Simulink model. This block is a MATLAB System block with all the input and output ports defined in the Linear Circuit Wizard block parameter dialog box. The generated block requires a fixed-step discrete sample time, which the block either inherits from the surrounding model or defines for itself. For more information, see “MATLAB Systems Generated from Linear Circuit Wizard”.

The Linear Circuit Wizard block can generate multiple MATLAB System blocks that represent different linear circuit blocks. Once generated, the MATLAB System blocks are independent of their respective Linear Circuit Wizard blocks.

Independent of the generation of MATLAB System blocks, the Linear Circuit Wizard block can also generate a report for ports of the generated block, a report for poles and zeroes of the entire linear circuit, and plot the transfer functions between the input and output ports. For more information, see “Verify MATLAB System Block Configuration”.

Version History

Introduced in R2020b

See Also

Topics

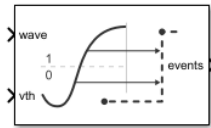
“Circuit Design Details Affect PLL Performance”

“Analyze T-Coil Circuit”

Logic Decision

Threshold crossing detector at input to binary process

Library: Mixed-Signal Blockset / Utilities



Description

Logic Decision block produces an output event at a fixed delay from almost exactly the time the input signal crosses a decision threshold. This block helps transitioning from a uniformly sampled input waveform to an event driven digital logic subsystem.

Ports

Input

wave — Input signal

fixed step sampling | variable step sampling | scalar

Input signal, specified as a floating point scalar. The input signal can either be inherited, or defined by the Logic Decision block as a fixed step discrete sample time.

Data Types: double | Boolean

vth — Decision threshold value

scalar

Decision threshold value, specified as a scalar. The value at **vth** port determines when the input signal is delayed by a fixed amount.

Data Types: floating point

Output

events — Output signal

0 | 1

Output signal, returned as either 0 or 1.

Data Types: double | Boolean

Parameters

Sample time — Source of sample time

Inherited (default) | Fixed

Source of sample time.

- Select **Inherited** to inherit sample time from previous block.
- Select **Fixed** to set discrete sample time to a fixed value.

Programmatic Use

- Use `get_param(gcb, 'SampleTimeSource')` to view the current source of **Sample time**.
- Use `set_param(gcb, 'SampleTimeSource', value)` to set **Sample time** to a specific value.

Sample time value – Actual value of sample time

20e-12 (default) | real scalar excluding zero

Actual value of sample time, specified as a real scalar excluding zero.

Dependencies

To enable this parameter, select **Fixed** in **Sample time** parameter.

Programmatic Use

- Use `get_param(gcb, 'SampleTimeIn')` to view the current **Sample time value**.
- Use `set_param(gcb, 'SampleTimeIn', value)` to set **Sample time value** to a specific value.

Minimum delay value – Minimum propagation delay value

1e-15 (default) | positive scalar

The minimum propagation delay for the block, specified as a positive scalar. For a fixed discrete input sample time, the actual delay is the maximum of this parameter value and the fixed step size.

Programmatic Use

- Use `get_param(gcb, 'Delay')` to view the current **Minimum delay value**.
- Use `set_param(gcb, 'Delay', value)` to set **Minimum delay value** to a specific value.

Buffer size – Number of threshold crossings to buffer

1 (default) | positive integer scalar

The number of pending output events that can be stored in the block, specified as a positive integer scalar.

Programmatic Use

- Use `get_param(gcb, 'BufferSizeIn')` to view the current **Buffer size**.
- Use `set_param(gcb, 'BufferSizeIn', value)` to set **Buffer size** to a specific value.

Version History

Introduced in R2019a

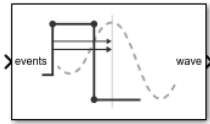
See Also

Slew Rate | Variable Pulse Delay

Slew Rate

Model amplitude, rise and fall times, and propagation delay of logic gates

Library: Mixed-Signal Blockset / Utilities



Description

The Slew Rate block converts a logical signal to a signal with user-defined finite slew rate and propagation delay.

Ports

Input

events — Input signal

scalar

Variable step, event driven input signal, specified as a scalar. The signal at **events** port comes from the output of a logic gate such as Variable Pulse Delay block.

Data Types: single | double | int8 | int16 | int32 | int64 | uint8 | uint16 | uint32 | uint64 | Boolean

Output

wave — Output signal

scalar

Fixed step, uniformly sampled output signal, returned as a scalar. You can define the finite slew rate and propagation delay for the signal at the **wave** port.

Data Types: double

Parameters

Output step size calculation — Defines how to calculate output step size

Default (default) | Advanced

Defines how to calculate output step size.

- Choose Default to calculate **Output step size calculation** based on rise/fall time. The **Output sample interval** (ΔT) is given by $\Delta T = \frac{(\text{Rise/fall time})^2}{6 \cdot 0.22}$.
- Choose Advanced to calculate **Output step size calculation** based on frequency of interest. The **Output sample interval** (ΔT) is given by $\Delta T = \frac{\text{Rise/fall time}}{6 \cdot \text{Maximum frequency of interest}}$.

Programmatic Use

- Use `get_param(gcb, 'DefaultOrAdvanced')` to determine how the output step size is being calculated.

Maximum frequency of interest (Hz) — Maximum frequency of interest at output

11e9 (default) | scalar

Maximum frequency of interest at output, specified as a scalar in Hz. **Maximum frequency of interest** is used to calculate **Output sample interval** and **Minimum 20%-80% rise/fall time**.

Dependencies

This parameter is only available when Advanced is selected for **Output step size calculation**.

Programmatic Use

- Use `get_param(gcb, 'MaxFreqInterest')` to view the current value of **Maximum frequency of interest**.
- Use `set_param(gcb, 'MaxFreqInterest', value)` to set **Maximum frequency of interest** to a specific value.

Output sample interval — Output sample interval

23ps (default) | scalar

Output sample interval, specified as a scalar in s. This parameter is nontunable.

The **Output sample interval** (ΔT) is given by $\Delta T = \frac{\text{Rise/fall time}}{6 \cdot \text{Maximum frequency of interest}}$.

Dependencies

This parameter is only available when Advanced is selected for **Output step size calculation**.

Programmatic Use

- Use `get_param(gcb, 'OutputSampleTime')` to view the current value of **Output sample interval**.
- Use `set_param(gcb, 'OutputSampleTime', value)` to set **Output sample interval** to a specific value.

Minimum 20%-80% rise/fall time — Minimum rise/fall time required at the output for meaningful simulation

23ps (default) | scalar

Minimum rise/fall time required at the output for meaningful simulation, specified as a scalar in ps. This is a nontunable parameter.

Dependencies

This parameter is only available when Advanced is selected for **Output step size calculation**.

Programmatic Use

- Use `get_param(gcb, 'ConversionRiseFall')` to view the current value of **Minimum 20%-80% rise/fall time**.

- Use `set_param(gcb, 'ConversionRiseFall', value)` to set **Minimum 20%-80% rise/fall time** to a specific value.

Rise/fall time (s) — 20%-80% rise time at the output

30e-12 (default) | scalar

Time required for signal to change from 20% to 80% in a full amplitude edge at the output, specified as a scalar.

Programmatic Use

- Use `get_param(gcb, 'RiseTime')` to view the current value of **Rise/fall time**.
- Use `set_param(gcb, 'RiseTime', value)` to set **Rise/fall time** to a specific value.

Slew rate for 20%-80% edge of unit amplitude signal — Slew rate for the 20%-80% edge at the output

56GHz (default) | scalar

Slew rate for the 20%-80% edge at the output, specified as a scalar. This parameter is nontunable.

Programmatic Use

- Use `get_param(gcb, 'RisingSlewRate')` to view the current value of **Slew rate for 20%-80% edge of unit amplitude signal**.
- Use `set_param(gcb, 'RisingSlewRate', value)` to set **Slew rate for 20%-80% edge of unit amplitude signal** to a specific value.

Minimum propagation delay — Minimum propagation delay for meaningful simulation

48ps (default) | scalar

Minimum propagation delay for meaningful simulation, specified as a scalar. This parameter is nontunable.

Minimum propagation delay is calculated from **Maximum frequency of interest** where

$$\text{Minimum propagation delay} = \frac{0.269}{\text{Maximum frequency of interest}}$$

Programmatic Use

- Use `get_param(gcb, 'ConversionDelay')` to view the current value of **Minimum propagation delay**.
- Use `set_param(gcb, 'ConversionDelay', value)` to set **Minimum propagation delay** to a specific value.

Propagation delay (s) — Propagation delay of a rising edge

48e-12 (default) | scalar

Propagation delay of a rising edge, specified as a scalar in s. **Propagation delay** is measured at a threshold equal to the half of the amplitude.

Programmatic Use

- Use `get_param(gcb, 'RisepropDelay')` to view the current value of **Propagation delay**.
- Use `set_param(gcb, 'RisePropDelay', value)` to set **Propagation delay** to a specific value.

Enable increased buffer size – Enable increased buffer size

button

Select to enable increased buffer size during the simulation. This increases the buffer size of all the blocks in the PLL model that belong to the Mixed-Signal Blockset/Utilities Simulink library. By default, this option is deselected.

Buffer size – Buffer size of the blocks

10 (default) | positive integer scaler

Buffer size of all the blocks in the PLL model that belong to the Mixed-Signal Blockset/Utilities Simulink library.

Selecting different simulation solver or sampling strategies can change the number of input samples needed to produce an accurate output sample. Set the **Buffer size** to a large enough value so that the input buffer contains all the input samples required.

Dependencies

This parameter is only available when the **Enable increased buffer size** option is selected.

Programmatic Use

- Use `get_param(gcb, 'NBuffer')` to view the current value of **Buffer size**.
- Use `set_param(gcb, 'NBuffer', value)` to set **Buffer size** to a specific value.

Version History

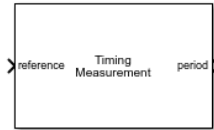
Introduced in R2019a**See Also**

Logic Decision | Variable Pulse Delay

Timing Measurement

Measure period, frequency, duty cycle, rise time, fall time, and delay of a signal

Library: Mixed-Signal Blockset / Utilities



Description

The Timing Measurement block measures the basic timing metrics such as period, frequency, duty cycle, rise time, fall time, and delay of a signal.

Ports

Input

reference — Input signal

scalar

Input signal whose timing metrics are being measured, specified as a scalar.

Data Types: `double`

test — Test signal

scalar

Test signal against which the delay of the input signal is measured, specified as a scalar.

Dependencies

To enable this port, select **Delay** as a measurement option in the **Selection** tab.

Data Types: `double`

Output

period — Period of input signal

scalar

Period of the input signal, returned as a scalar.

Data Types: `double`

frequency — Frequency of input signal

scalar

Frequency of the input signal, returned as a scalar.

Dependencies

To enable this port, select **Frequency** as a measurement option in the **Selection** tab.

Data Types: double

rise time — Rise time of input signal

scalar

Rise time of the input signal, returned as a scalar.

Dependencies

To enable this port, select **Rise Time** as a measurement option in the **Selection** tab.

Data Types: double

fall time — Fall time of input signal

scalar

Fall time of the input signal, returned as a scalar.

Dependencies

To enable this port, select **Fall time** as a measurement option in the **Selection** tab.

Data Types: double

duty cycle — Duty cycle of input signal

scalar

Duty cycle of the input signal, returned as a scalar.

Dependencies

To enable this port, select **Duty Cycle** as a measurement option in the **Selection** tab.

Data Types: double

delay — Delay of input signal

scalar

Delay of the input signal, returned as a scalar.

Dependencies

To enable this port, select **Delay** as a measurement option in the **Selection** tab.

Data Types: double

Parameters

Selection

Period — Measure period of input signal

on (default) | off

Select to measure the period of the input signal.

Period is defined as the time required to complete one full cycle of a periodic signal, specified in seconds.

Frequency — Measure frequency of input signal

off (default) | on

Select to measure the frequency of the input signal.

Frequency is defined as the number of cycles completed per unit of time by a periodic signal, specified in hertz. It is the inverse of **Period**.

Rise Time — Measure rise time of input signal

off (default) | on

Select to measure the rise time of the input signal.

Rise time is defined as the time required by the rising edge of the signal to reach from 10% to 90% (or 20% to 80%) of its steady state value, specified in seconds.

Fall Time — Measure fall time of input signal

off (default) | on

Select to measure the fall time of the input signal.

Fall time is defined as the time required by the falling edge of the signal to go from 90% to 10% (or 80% to 20%) of its steady state value, specified in seconds.

Duty Cycle — Measure duty cycle of input signal

off (default) | on

Select to measure the duty cycle of the input signal.

Duty cycle is defined as the ratio of positive pulse width to period, specified as a fraction.

Delay — Measure delay of input signal

off (default) | on

Select to measure the delay of the input signal.

Delay is defined as the time difference between the threshold crossing points between the signal of interest against a test signal, specified in seconds.

Configuration**Use same reference threshold for all measurements — Use the same reference threshold for measuring all timing metrics**

on (default) | off

Select to use the same reference threshold for measuring all timing metrics.

Reference threshold (V) — Reference threshold used for measuring timing metrics

0 (default) | scalar

Reference threshold voltage used for measuring all timing metrics, specified as a scalar in volts.

Dependencies

To enable this parameter, select **Use same reference threshold for all measurements** in the **Configuration** tab.

Programmatic Use

- Use `get_param(gcb, 'SameRefThreshold')` to view the current value of **Reference threshold (V)**.
- Use `set_param(gcb, 'SameRefThreshold', value)` to set **Reference threshold (V)** to a specific value.

Period/Frequency**Reference threshold (V) — Reference threshold voltage used to measure period and/or frequency**`0 (default) | scalar`

Reference threshold voltage used to measure the period and/or the frequency of the signal of interest, specified as a scalar in volts.

Dependencies

To enable this parameter, deselect **Use same reference threshold for all measurements** in the **Configuration** tab and select **Period** and/or **Frequency** in the **Selection** tab.

Programmatic Use

- Use `get_param(gcb, 'PeriodRefThreshold')` to view the current value of **Reference threshold (V)**.
- Use `set_param(gcb, 'PeriodRefThreshold', value)` to set **Reference threshold (V)** to a specific value.

Rise/Fall Time**Input range (V) — Peak-to-peak value of input signal**`[-1 1] (default) | 2-element row vector`

Peak-to-peak value of the input signal, specified as a 2-element row vector in volts. The first element defines the notch value of the input signal and the second element defines the peak value. **Input range (V)** is used to calculate the upper and lower threshold levels to calculate the rise/fall times.

Dependencies

To enable this parameter, select **Rise Time** and/or **Fall Time** in the **Selection** tab.

Programmatic Use

- Use `get_param(gcb, 'Range')` to view the current value of **Reference threshold (V)**.
- Use `set_param(gcb, 'Range', value)` to set **Reference threshold (V)** to a specific value.

Type — Input threshold levels to measure rise/fall times`80%/20% (default) | 90%/10%`

Determines the input threshold levels to measure the rise/fall times, specified as either 80%/20% or 90%/10%.

Dependencies

To enable this parameter, select **Rise Time** and/or **Fall Time** in the **Selection** tab.

Programmatic Use

- Use `get_param(gcb, 'MeasurementType')` to view the current value of **Reference threshold (V)**.
- Use `set_param(gcb, 'MeasurementType', value)` to set **Reference threshold (V)** to a specific value.

Duty Cycle**Reference threshold (V) — Reference threshold voltage used to measure duty cycle**

0 (default) | scalar

Reference threshold voltage used to measure the duty cycle of the signal of interest, specified as a scalar in volts.

Dependencies

To enable this parameter, deselect **Use same reference threshold for all measurements** in the **Configuration** tab and select **Duty Cycle** in the **Selection** tab.

Programmatic Use

- Use `get_param(gcb, 'DcRefThreshold')` to view the current value of **Reference threshold (V)**.
- Use `set_param(gcb, 'DcRefThreshold', value)` to set **Reference threshold (V)** to a specific value.

Delay**Reference threshold (V) — Threshold voltage level in signal of interest used to measure delay**

0 (default) | scalar

Threshold voltage level in reference signal of interest whose delay is being measured, specified as a scalar in volts.

Dependencies

To enable this parameter, deselect **Use same reference threshold for all measurements** in the **Configuration** tab and select **Delay** in the **Selection** tab.

Programmatic Use

- Use `get_param(gcb, 'DelayRefThreshold')` to view the current value of **Reference threshold (V)**.
- Use `set_param(gcb, 'DelayRefThreshold', value)` to set **Reference threshold (V)** to a specific value.

Test threshold (V) — Threshold voltage level in test signal used to measure delay

0 (default) | scalar

Threshold voltage level in test signal against which the delay of the reference signal is being measured, specified as a scalar in volts.

Dependencies

To enable this parameter, deselect **Use same reference threshold for all measurements** in the **Configuration** tab and select **Delay** in the **Selection** tab.

Programmatic Use

- Use `get_param(gcb, 'DelayTstThreshold')` to view the current value of **Reference threshold (V)**.
- Use `set_param(gcb, 'DelayTstThreshold', value)` to set **Reference threshold (V)** to a specific value.

Version History

Introduced in R2020b

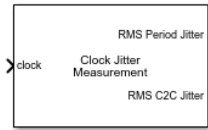
See Also

`timeDomainSignal2RiseTime` | `timeDomainSignal2FallTime` |
`timeDomainSignal2DutyCycle`

Clock Jitter Measurement

Measure jitter in periodic signals

Library: Mixed-Signal Blockset / Utilities



Description

Use the Clock Jitter Measurement block to measure the RMS (root mean squared) periodic jitter in clock signals. You can also measure cycle-to-cycle (C2C) jitter and duty cycle distortion (DCD).

Ports

Input

clock — Input clock signal

scalar

Input clock signal, specified as a scalar. The input signal must have only one threshold crossing per period.

Data Types: double

Output

Period Jitter — Running RMS value of period jitter

scalar

Running RMS value of the period jitter calculated up to the current time step, returned as a scalar. Period jitter is the deviation in the cycle time of a clock signal with respect to the ideal period.

Data Types: double

C2C — Running RMS value of cycle-to-cycle jitter

scalar

Running RMS value of the cycle-to-cycle jitter calculated up to the current time step, returned as a scalar. Cycle-to-cycle jitter is the difference in the period between the two consecutive cycles of the clock.

Dependencies

To enable this port, select **Cycle-2-Cycle (C2C) Jitter**.

Data Types: double

DCD — Difference between real duty cycle and 50% value

scalar

Difference between the real duty cycle in percentage and the ideal 50% value, returned as a scalar.

Dependencies

To enable this port, select **Duty Cycle Distortion (DCD)**.

Data Types: double

Parameters**Clock Frequency — Frequency of input clock signal**

1e6 (default) | positive real scalar

Frequency of the input clock signal, specified as a positive real scalar in Hz. **Clock Frequency** is used to calculate the ideal value of the period of the input signal.

Programmatic Use

Block parameter: Frequency

Type: character vector

Values: positive real scalar

Default: 1e6

Threshold — Threshold signal level

0 (default) | real scalar

Threshold signal level to calculate the rising and falling edge of the signal, specified as a real scalar.

Programmatic Use

Block parameter: Threshold

Type: character vector

Values: real scalar

Default: 0

Period Jitter — RMS period jitter

on (default) | off

Select to calculate the RMS period jitter. This option is selected by default.

Period jitter is the deviation in the cycle time of a clock signal with respect to the ideal period.

Cycle-2-Cycle (C2C) Jitter — RMS cycle-to-cycle jitter

off (default) | on

Select to calculate the RMS cycle-to-cycle (C2C) jitter. This option is deselected by default.

Cycle-to-cycle jitter is the difference in the period between the two consecutive cycles of the clock.

Duty Cycle Distortion (DCD) — Duty cycle distortion for each cycle

off (default) | on

Select to calculate the duty cycle distortion for each cycle. This option is deselected by default.

Duty cycle distortion is the difference between the real duty cycle in percentage and the ideal 50% value.

Simulate using — Select simulation mode

Code generation (default) | Interpreted execution

Select the simulation mode. This choice affects the simulation performance.

Simulating the model using the **Code generation** method requires additional startup time, but the subsequent simulations run faster. Simulating the model using the **Interpreted execution** method may reduce the startup time, but the subsequent simulations run slower. For more information, see “Simulation Modes”.

Version History

Introduced in R2021a

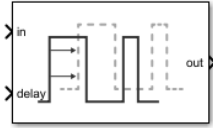
See Also

Aperture Jitter Measurement

Variable Pulse Delay

Delay samples by controlled, continuously variable amount

Library: Mixed-Signal Blockset / Utilities



Description

Variable Pulse Delay block introduces a controllable delay in signal samples. Each sample at the **in** port is delayed by the value at the **delay** port at the time the input sample arrived. The delayed samples at the **out** port must maintain the same order as at the **in** port.

At the beginning of the simulation, the **out** port is set to the value of the **Initial Input** parameter.

Ports

Input

in — Input sample data

fixed step discrete sample | variable step discrete sample

Input sample data, whose type and width are inherited from the signal source. The input port supports data bus operation, but does not support framed inputs.

Data Types: int8 | int16 | int32 | int64 | uint8 | uint16 | uint32 | uint64 | Boolean | floating point

delay — Introduces delay to input signal

nonnegative scalar

Introduces delay to the input signal, specified as a nonnegative scalar. The value at the **delay** port at the time of the arrival of input signal determines the amount of delay introduced.

Data Types: floating point

Output

out — Delayed output sample

fixed step discrete sample | variable step discrete sample

Delayed output sample data, whose type and width are the same as the input signal. The value at the **delay** port at the time of the arrival of input signal determines the amount of delay introduced at the **out** port. The input and output signals must maintain the same order.

Data Types: int8 | int16 | int32 | int64 | uint8 | uint16 | uint32 | uint64 | Boolean | floating point

Parameters

Initial Input — The value at the output port before simulation

0 (default) | scalar

The value at the output port before simulation, specified as a scalar.

At the beginning of the simulation, the value at the **out** port is equal to the value set by the **Initial Input** parameter. If the value at the **in** port is not equal to the value of the **Initial Input** parameter, then the output will transition to the value at the **in** port after a delay equal to the value at the **delay** port.

Programmatic Use

- Use `get_param(gcb, 'InitialOutput')` to view the current value of **Initial Input**.
- Use `set_param(gcb, 'InitialOutput', value)` to set **Initial Input** to a specific value.

Buffer Size — Number of samples of the input buffering available during simulation

1 (default) | positive scalar integer

Number of samples of the input buffering available during simulation, specified as a positive scalar integer.

Programmatic Use

- Use `get_param(gcb, 'BufferSize')` to view the current value of **Buffer Size**.
- Use `set_param(gcb, 'BufferSize', value)` to set **Buffer Size** to a specific value.

Version History

Introduced in R2019a

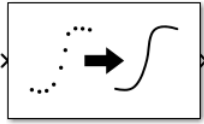
See Also

Logic Decision | Slew Rate

Lowpass Resampler

Convert signal from one sample time to another

Library: Mixed-Signal Blockset / Utilities



Description

The Lowpass Resampler block converts either a fixed-step discrete or a variable-step discrete sample time at its input to a different sample time at its output. To calculate the output sample values, the block uses a lowpass filtering interpolation algorithm. The algorithm minimizes frequency aliasing at the output with respect to an output rise/fall time.

If the output rise/fall time is inherited from a fixed-step discrete input, the cutoff frequency is the Nyquist rate of the input. Otherwise, the cutoff frequency is the Nyquist rate associated with a sample interval obtained by scaling the specified 20%–80% output rise/fall time to a value for 0%–100% rise/fall time.

Ports

Input

in — Discrete time input signal

scalar

Discrete time input signal, specified as fixed-step or variable-step scalar.

Data Types: `single` | `double` | `int8` | `int16` | `int32` | `uint8` | `uint16` | `uint32` | `Boolean`

Output

out — Continuous time output signal

scalar

Continuous time output signal, returned as a fixed-step or variable-step scalar.

Data Types: `single` | `double`

Parameters

Inherit output rise/fall time — Inherit output rise/fall time from fixed-step input sample time

`on` (default) | `off`

Select to inherit the output rise/fall time from the fixed-step input sample time.

Note In case of variable step input, there is no rise/fall time to inherit.

Output rise/fall time – 20%–80% output rise/fall time

1e-10 (default) | positive real scalar

20%–80% output rise/fall time, specified as a positive real scalar.

Dependencies

To enable this parameter, deselect the **Inherit output rise/fall time** parameter.

Programmatic Use**Block parameter:** OutputRiseFall**Type:** character vector**Values:** positive real scalar**Default:** 1e-10**Number of samples of delay – Number of samples of propagation delay for fixed-step operation**

1 (default) | positive real scalar

Number of samples of propagation delay for fixed-step operation, specified as a positive real scalar.

If the Lowpass Resampler block inherits **Output rise/fall time** in fixed-step mode, the resampler conversion delay is given by $NDelay \cdot \tau$, where $NDelay$ is the **Number of samples of delay** parameter and τ is the inherited input sample interval.

In the variable-step input mode, the resampler conversion delay is given by $0.6\tau_v$ when $NDelay$ equals to one, and is $(NDelay-0.5) \cdot \tau_v$ when $NDelay$ is greater than one. τ_v is 5/3 times the **Output rise/fall time** parameter.

There is a tradeoff between the delay of the conversion and the suppression of out-of-band numerical artifacts, with greater delay producing better fidelity in band and greater suppression out-of-band.

Note If you need anti-aliasing filter rejection, set **Number of samples of delay** to 5 or higher.

Dependencies

To enable this parameter, deselect the **Inherit output rise/fall time** parameter.

Programmatic Use**Block parameter:** NDelay**Type:** character vector**Values:** positive real scalar**Default:** 1**Output sample time – Type of output sample time**

Inherited (default) | Fixed step discrete | Variable step discrete

Type of the output sample time to be used by downstream blocks, specified as either fixed step discrete or variable step discrete. The output sampling rate must be higher than the input sampling rate. For more information, see “Variable Step Sampling Scheme” on page 3-26.

Programmatic Use**Block parameter:** OutputTsType**Type:** character vector

Values: Fixed step discrete | Variable step discrete

Default: Inherited

Samples out per rise/fall time — Number of output samples in single output rise/fall time

5 (default) | positive real scalar

Number of output samples in a single output rise/fall time, specified as a positive real scalar.

Dependencies

To enable this parameter, select the option Fixed step discrete or Variable step discrete in the **Output sample time** parameter.

Programmatic Use

Block parameter: OutputTsRatio

Type: character vector

Values: positive real scalar

Default: 5

Enable large buffer — Enable extra buffer samples

on (default) | off

Select to enable extra buffer samples. This option is enabled by default.

Number of extra buffer samples — Number of extra buffer samples to support sample delays

10 (default) | positive real scalar

Number of extra buffer samples needed to support the number of samples of delay, specified as a positive real scalar.

Programmatic Use

Block parameter: NBuffer

Type: character vector

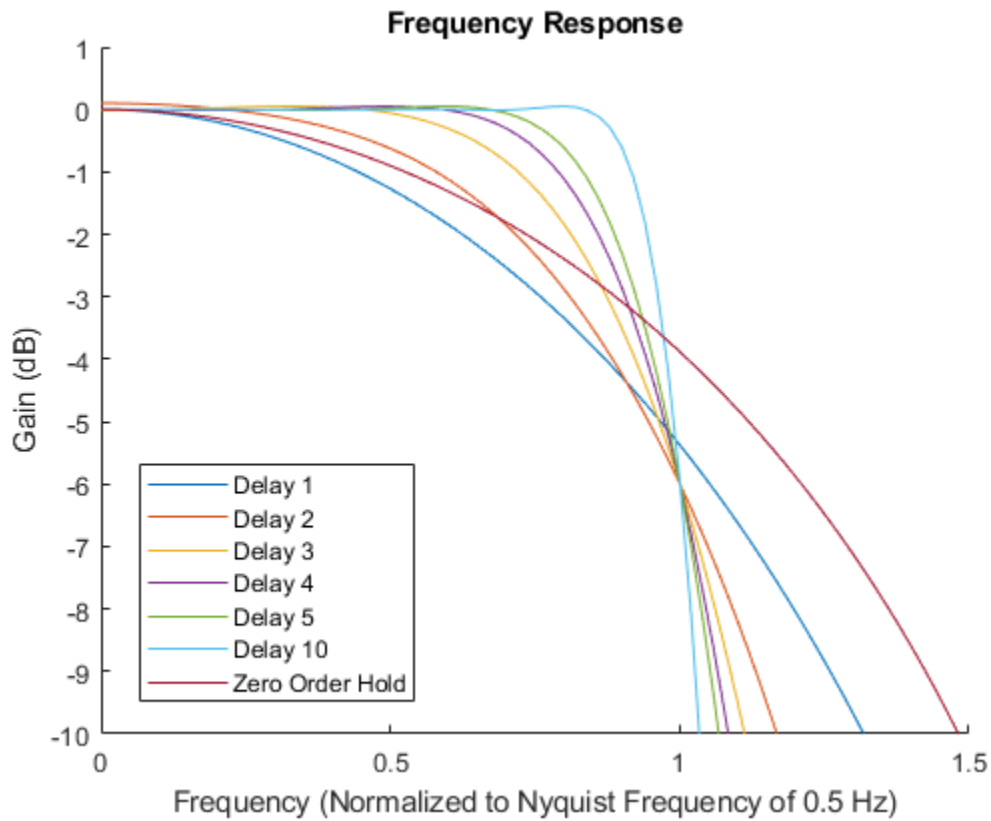
Values: positive real scalar

Default: 10

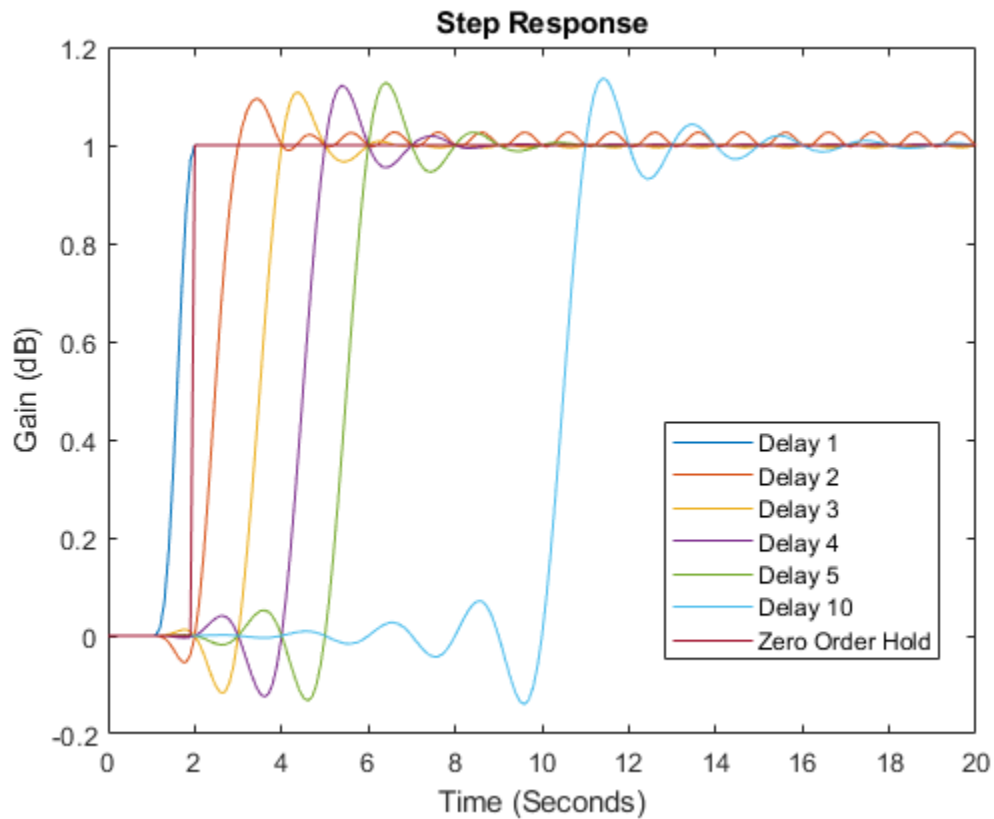
More About

Frequency and Step Response

The Lowpass Resampler block interpolates the incoming discrete signal using an anti-aliasing filter. The anti-aliasing filter introduces some delay and the suppression of outputs above the specified signal bandwidth is not perfect. Introducing larger delays produces better fidelity in band and greater suppression out-of-band at the cost of more delay inserted in the signal path. In all cases, the interpolation filter is designed to produce negligible group delay distortion in-band.



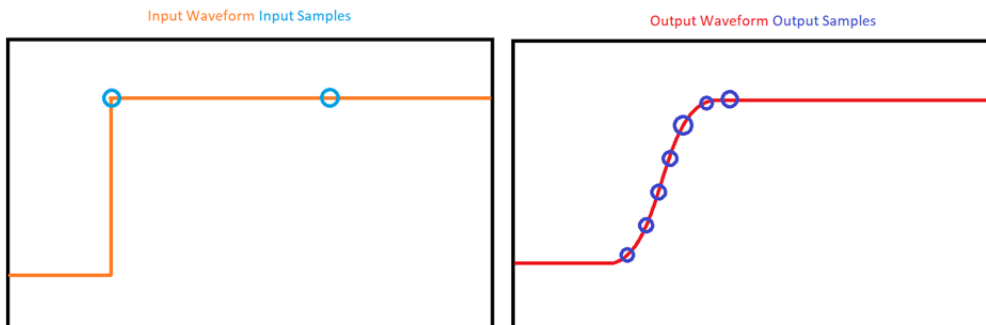
In the time domain, the interpolation does introduce some ringing in the step response of the anti-aliasing filter, which could affect some digital switching applications. However, the choice of **Number of samples of delay** parameter of 1 and **Output rise/fall time** parameter greater than zero is specifically designed to produce a smooth waveform transition with no ringing.



Variable Step Sampling Scheme

In some cases, it might be necessary for the Lowpass Resampler block to define its own sample time. This is particularly true if the block receives a unspecified output sample time at its input.

If the Lowpass Resampler block receives a signal that changes its value at the input, the block generates some output samples to show the transition. The number of samples are defined by the **Samples out per rise/fall time** parameter.



Version History

Introduced in R2021a

Updated in R2022a

Lowpass Resampler can define its own output sample times.

Introduced in R2021a

Use to change the sampling time of a signal.

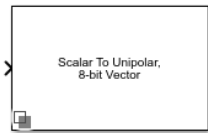
See Also

Logic Decision | Variable Pulse Delay | `lowpassResamplesim(model)`

Binary Vector Conversion

Convert scalar integer to binary logic vector and vice versa

Library: Mixed-Signal Blockset / Utilities



Description

The Binary Vector Conversion block encodes, decodes, and manipulates binary coded vectors. You can convert a scalar input to a logical (Boolean) vector signal using binary magnitude or two's complement and vice versa. You can also resize and reverse the bit index arrangement of the binary logic vectors.

Ports

Input

in — Input signal

scalar | logical vector

Input signal, specified as a scalar or logical vector.

- When encoding, the input signal is a scalar.
- When decoding, the input signal is a logical (Boolean) vector.

Note When converting to a scalar value from a bit stream, any input greater than 1 is considered as logical '1'. Similarly any input bit is less than 0 is considered logical '0'.

Data Types: single | double | uint8 | uint16 | uint32 | Boolean | fixdt(0,16)

Output

out — Output signal

scalar | logical vector

Output signal, specified as a scalar or logical vector.

- When encoding, the output signal is a logical (Boolean) vector.
- When decoding, the output signal is a scalar.

Data Types: single | double | uint8 | uint16 | uint32 | Boolean | fixdt(0,16)

Parameters

Convert — Type of conversion operation

Scalar to binary-coded vector (default) | Binary-coded vector to scalar | Resize binary-coded vector | Reverse binary-coded vector

Type of conversion, specified as one of these:

- Scalar to binary-coded vector — Encode a scalar input signal to logical vector.
- Binary-coded vector to scalar — Decode a logical vector signal to scalar.
- Resize binary-coded vector — Resize a logical vector while preserving the sign using two's complement method.
- Reverse binary-coded vector — Reverse the bit index arrangement of a logical vector.

Note When converting to a scalar value from a bit stream we will consider any input greater than 1 as a logical '1'. Similarly if the input bit is less than 0 (i.e. any -ve number) then it will be considered logical '0'.

Programmatic Use

Block parameter: Convert

Type: character vector

Values: Scalar to binary-coded vector | Binary-coded vector to scalar | Resize binary-coded vector | Reverse binary-coded vector

Default: Scalar to binary-coded vector

Logical vector encoding — Method of encoding logical vector

Unipolar (magnitude) (default) | Bipolar (2's complement)

Method of encoding logical vector, specified as either Unipolar (magnitude) or Bipolar (2's complement).

Dependencies

To enable this parameter, set **Convert** to Scalar to binary-coded vector, Binary-coded vector to scalar, or Resize binary-coded vector.

Programmatic Use

Block parameter: Encoding

Type: character vector

Values: Unipolar (magnitude) | Bipolar (2's complement)

Default: Unipolar (magnitude)

Input vector length — Number of vector elements in input vector

8 (default) | positive real scalar

Number of vector elements in the input vector, specified as a positive real scalar.

By default, the least significant bit (LSB) is the first element of the vector (index 1). But you can also set the most significant bit (MSB) as the first element of the vector.

Dependencies

To enable this parameter, set **Convert** to Binary-coded vector to scalar, Resize binary-coded vector, or Reverse binary-coded vector.

Programmatic Use

Block parameter: InputLength

Type: character vector

Values: positive real scalar

Default: 8

Output vector length — Number of vector elements in output vector

8 (default) | positive real scalar

Number of vector elements in the output vector, specified as a positive real scalar.

By default, the least significant bit (LSB) is the first element of the vector (index 1). But you can also set the most significant bit (MSB) as the first element of the vector.

Dependencies

To enable this parameter, set **Convert** to Scalar to binary-coded vector or Resize binary-coded vector.

Programmatic Use

Block parameter: OutputLength

Type: character vector

Values: positive real scalar

Default: 8

Output data type — Datatype of output scalar

double (default) | Inherit: Inherit via back propagation | single | uint8 | uint16 | uint32 | fixdt(0,16)

Datatype of the output scalar. You can choose to inherit the datatype, specify directly, or express as a datatype object.

Dependencies

To enable this parameter, set **Convert** to Binary-coded vector to scalar.

Programmatic Use

Block parameter: DataType

Type: character vector

Values: Inherit: Inherit via back propagation | double | single | uint8 | uint16 | uint32 | fixdt(0,16)

Default: double

Version History

Introduced in R2021a

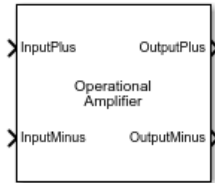
See Also

Segmented DAC

Operational Amplifier

Model operational amplifier with two or more poles

Library: Mixed-Signal Blockset / Utilities



Description

Use the Operational Amplifier block to model an operational amplifier with two or more poles. You can create a double pole amplifier from the important circuit parameters or a multiple pole amplifier from transfer function. The operational amplifier model can be used to characterize the operational amplifier performance as part of a larger circuit system.

Ports

Input

InputPlus — Positive input terminal

scalar

Positive input terminal of the operational amplifier.

Data Types: `single` | `double`

InputMinus — Negative input terminal

scalar

Negative input terminal of the operational amplifier.

Data Types: `single` | `double`

Output

OutputPlus — Positive output terminal

scalar

Positive output terminal of the operational amplifier.

Data Types: `single` | `double`

OutputMinus — Negative output terminal

scalar

Negative output terminal of the operational amplifier.

Data Types: `single` | `double`

Parameters

Operational Amplifier Circuit — Type of operational amplifier circuit

Double Pole Circuit (default) | Multiple Pole Circuit

Type of the operational amplifier circuit.

- Double Pole Circuit — Use circuit parameters such as open loop gain, unity gain bandwidth, maximum tail current, and slew rate to define a simple double pole circuit.
- Multiple Pole Circuit — Use transfer function poles and zeros to define a multiple pole circuit.

Programmatic Use

Block parameter: CircuitType

Type: character vector

Values: Double Pole Circuit | Multiple Pole Circuit

Default: Double Pole Circuit

Signal Parameters

Supply rail high (V) — Maximum supply voltage

5 (default) | real scalar

Maximum supply voltage provided to the Operational Amplifier block, specified as a real scalar in volts.

Programmatic Use

Block parameter: VoltageSupplyPlus

Type: character vector

Values: real scalar

Default: 5

Supply rail low (V) — Minimum supply voltage

-5 (default) | real scalar

Minimum supply voltage provided to the Operational Amplifier block, specified as a real scalar in volts.

Programmatic Use

Block parameter: VoltageSupplyMinus

Type: character vector

Values: real scalar

Default: -5

Input offset voltage (V) — Offset voltage at operational amplifier input

0 (default) | scalar

Offset voltage at the input of the operational amplifier, specified as a scalar in volts. Input offset voltage is applied to obtain a zero voltage at the output of the operational amplifier.

Programmatic Use

Block parameter: OffsetVoltage

Type: character vector

Values: scalar

Default: 0

Output Resistance (Ohms) — Resistance at operational amplifier output

80 (default) | positive real scalar

Resistance at the output terminals of the operational amplifier, specified as a positive real scalar in ohms.

Programmatic Use**Block parameter:** OutputResistance**Type:** character vector**Values:** positive real scalar**Default:** 80**Advanced Parameters****Open loop gain (V/V) — Operational amplifier gain without feedback**

855e3 (default) | positive real scalar

The gain of the operational amplifier without any positive or negative feedback, specified as a unitless positive real scalar.

Programmatic Use**Block parameter:** Gain**Type:** character vector**Values:** positive real scalar**Default:** 855e3**Unity Gain Bandwidth (Hz) — Frequency at which operational amplifier gain becomes unity**

1e8 (default) | positive real scalar

The frequency at which the open loop gain of the operational amplifier becomes unity, specified as a positive real scalar in hertz.

Programmatic Use**Block parameter:** FrequencyUnityGain**Type:** character vector**Values:** positive real scalar**Default:** 1e8**Maximum Tail Current (A) — Maximum current passing through tail MOSFET in operational amplifier circuit**

100e-6 (default) | scalar

Maximum value of the current passing through the tail MOSFET in the operational amplifier circuit, specified as a scalar in amperes.

Programmatic Use**Block parameter:** InputCurrentMax**Type:** character vector**Values:** scalar**Default:** 100e-6**Slew Rate (V/s) — Rate of change of operational amplifier output voltage with time**

0.5e6 (default) | positive real scalar

The rate of the change of the output voltage of the operational amplifier with time, specified as a positive real scalar.

Programmatic Use**Block parameter:** SlewRate**Type:** character vector**Values:** positive real scalar**Default:** 0.5e6**Transfer function poles — Poles in operational transfer function in Laplace domain**

[-2*pi*122; -2*pi*2.1e6; -2*pi*32e6] (default) | column vector

Poles in the transfer function of the operational amplifier in the Laplace domain, specified as a column vector.

Programmatic Use**Block parameter:** Poles**Type:** character vector**Values:** column vector**Default:** 80**Transfer function zeroes — Zeros in operational amplifier transfer function in Laplace domain**

[2*pi*3.3e6; 2*pi*29e6] (default) | column vector

Zeros in the transfer function of the operational amplifier in the Laplace domain, specified as a column vector.

Programmatic Use**Block parameter:** Zeros**Type:** character vector**Values:** column vector**Default:** 80**Sample interval to be used in MATLAB analyses — Sample interval used in MATLAB analyses**

1e-8 (default) | scalar

Sample interval used in MATLAB analyses, specified as a scalar in seconds.

Programmatic Use**Block parameter:** SampleInterval**Type:** character vector**Values:** scalar**Default:** 1e-8**Simulate using — Select simulation mode**

Interpreted execution (default) | Code generation

Select the simulation mode. This choice affects the simulation performance.

Simulating the model using the Code generation method requires additional startup time, but the subsequent simulations run faster. Simulating the model using the Interpreted execution method may reduce the startup time, but the subsequent simulations run slower. For more information, see “Simulation Modes”.

Programmatic Use**Block parameter:** SimulateUsing

Type: character vector

Values: Code generation| Interpreted execution

Default: Interpreted execution

Version History

Introduced in R2021b

See Also

Linear Circuit Wizard

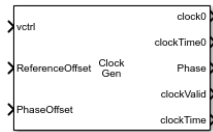
Topics

“Design Inverting Amplifier”

Clock Generator

Generate clock signal with one or more phases

Library: Mixed-Signal Blockset / Utilities



Description

The Clock Generator block generates clock signal with multiple output phases and detailed phase noise modeling. Each output is modeled using two ports: a saturated clock signal and a clock transition time. The block allows you to model any clock recovery loop that uses a voltage controlled oscillator (VCO). The block consistently provides clock times for which the noise floor is below -150 dBC/Hz. Using the block, you can also include phase noise that matches a physical model.

Ports

Input

vctrl — Voltage to control output frequency

scalar | vector

Voltage to control the output frequency, specified as a scalar or vector.

Data Types: double

ReferenceOffset — Reference frequency offset

scalar

Reference frequency offset to be used, specified as a scalar. If **ReferenceOffsetPort** is selected, the **ReferenceOffset** port accepts the reference offset value from an external block.

Data Types: double

PhaseOffset — Duty cycle phase offset

scalar

Duty cycle phase offset to be used, specified as a scalar. If **PhaseOffsetPort** is selected, the **PhaseOffset** port accepts the phase offset value from an external block.

Data Types: double

Output

clockN — Saturated output clock signal

scalar

The saturated output clock signal, returned as a scalar. There is one **clockN** output port for each output phase. The label *N* indicates the clock phase and is a nonnegative scalar. The output is a square wave whose amplitude is defined by the **Output amplitude** parameter.

Data Types: double

clockTimeN — Simulation time at which last clock transition occurred

scalar

The simulation time at which the last clock transition occurred, returned as a scalar. There is one **clockTimeN** output port for each output phase. The label *N* indicates the clock phase and is a nonnegative scalar.

Data Types: double

Phase — Most recent clock phase of fundamental clock

scalar

The most recent clock phase of the fundamental clock, returned as a scalar.

Data Types: double

clockValid — Variable step discrete sampled clock

scalar

Variable step discrete sampled clock, specified as a scalar. The edge sample times of the signal in the **clockValid** port exactly match the values in the clock time signal

Data Types: double

clockTime — Absolute clock time of most recent clock change

scalar

Absolute clock time of the most recent clock change in the **clockValid** port.

Data Types: double

Parameters

Main

Specify using — Define how control sensitivity of clock generator is specified

Voltage sensitivity (default) | Output frequency vs. control voltage | Period offset

Define how the control sensitivity of the clock generator is specified:

- Select **Voltage sensitivity** to specify output frequency from the **Voltage sensitivity (Hz/V)** and **Free running frequency (Hz)** parameters.
- Select **Output frequency vs. control voltage** to interpolate output frequency from the **Control voltage (V)** vector versus **Output frequency (Hz)** vector.
- Select **Period offset** to adjust the cycle time by a fraction of a nominal clock period. The fraction is indicated by the **Control voltage (V)** parameter. The **Voltage sensitivity (Hz/V)** and the **Free running frequency (Hz)** are determined from **Symbol time (s)**.

Programmatic Use

Block parameter: SpecifyUsing

Type: character vector

Values: Voltage sensitivity | Output frequency vs. control voltage | Period offset

Default: Voltage sensitivity

Voltage sensitivity (Hz/V) – Measure of change in output frequency

100e6 (default) | positive real scalar

Measure of change in output frequency for input voltage change, specified as a positive real scalar with units in Hz/V.

Programmatic Use

Block parameter: Kvco

Type: character vector

Values: positive real scalar

Default: 100e6

Data Types: double

Free running frequency (Hz) – Output frequency without control voltage

10e9 (default) | positive real scalar

Frequency of the clock generator without any control voltage input (0 V) or the quiescent frequency, specified as a positive real scalar in hertz.

Programmatic Use

Block parameter: Fo

Type: character vector

Values: positive real scalar

Default: 10e9

Data Types: double

Control voltage (V) – Control voltage values

[-5 0 5] (default) | real valued vector

Control voltage values of the clock generator, specified as a real valued vector in volts.

Programmatic Use

Block parameter: ControlVoltage

Type: character vector

Values: real valued vector

Default: [-5 0 5]

Data Types: double

Output frequency (Hz) – Clock generator output frequency values

[9.9e9 10e9 10.5e9] (default) | positive real valued vector

Output frequency of the clock generator corresponding to the **Control voltage (V)** vector, specified in hertz.

Programmatic Use

Block parameter: OutputFrequency

Type: character vector

Values: positive real valued vector

Default: [9.9e9 10e9 10.5e9]

Data Types: double

Output amplitude (V) — Maximum amplitude of clock generator output voltage

1 (default) | positive real scalar

Maximum amplitude of the clock generator output voltage, specified as a positive real scalar.

Programmatic Use**Block parameter:** Amplitude**Type:** character vector**Values:** positive real scalar**Default:** 1

Data Types: double

Phase unit — Units of output phase and duty cycle

Degrees (default) | Fraction of a clock cycle

The units of the output phase and duty cycle, specified as Degrees or Fraction of a clock cycle.

Programmatic Use**Block parameter:** PhaseUnits**Type:** character vector**Values:** Degrees | Fraction of a clock cycle**Default:** Degrees**Output phase — Phases of output clock signals**

[0] (default) | nonnegative real scalar | nonnegative real valued vector

The phases of the output clock signals, specified as a scalar vector. If specified as a vector, each element defines one output.

Programmatic Use**Block parameter:** OutputPhase**Type:** character vector**Values:** nonnegative real scalar | nonnegative real valued vector**Default:** [0]**Output duty cycle — Duty cycle of the output clock signals**

[180] (default) | positive real scalar | positive real valued vector

The duty cycles of the output clock signals, specified as a scalar or a vector. The clock output phase refers to the rising edge of the output clock.

If specified as a vector, each element defines one output. Missing or empty elements of the vector are given the default duty cycle. Extra elements are ignored.

The duty cycle must be greater than $\frac{\text{Sample interval}}{\text{Symbol time}}$ but less than $1 - \frac{\text{Sample interval}}{\text{Symbol time}}$, defined as a fraction of clock cycle.

Programmatic Use**Block parameter:** OutputDutyCycle**Type:** character vector**Values:** positive real scalar | positive real valued vector**Default:** [180]

PhaseOffsetPort — Define duty cycle phase offset through an input port

on (default) | off

Define the duty cycle phase offset through **PhaseOffset** input port from an external block. If you deselect the **PhaseOffsetPort** parameter, it is removed from the AMI files. This effectively hard-codes phase offset to the value defined by the **.Phase offset (symbol time)** parameter.

Phase offset (symbol time) — Duty cycle phase offset

0 (default) | scalar in the range [-0.5,0.5]

Duty cycle phase offset, specified as a scalar in the range [-0.5, 0.5] in fraction of symbol time. Phase offset manually shifts clock probability distribution function (PDF) for better bit error rate (BER)..

Dependencies

To enable this parameter, deselect **PhaseOffsetPort**.

Programmatic Use**Block parameter:** PhaseOffset**Type:** character vector**Values:** scalar**Default:** 0**ReferenceOffsetPort — Define reference frequency offset through an input port**

on (default) | off

Define the reference frequency offset through **ReferenceOffset** input port from an external block. If you deselect the **ReferenceOffsetPort** parameter, it is removed from the AMI files. This effectively hard-codes **Reference offset** to the value defined by the **.Reference clock frequency offset (ppm)** parameter.

Reference clock frequency offset (ppm) — Factor by which nominal output frequency is to be offset

0 (default) | scalar in the range of [-300,300]

The factor by which the nominal output frequency is to be offset from $\frac{1}{\text{Symbol time}}$, specified as a scalar in the range of [-300,300] in unit of parts per million. It is the deviation between the transmitter oscillator frequency and the receiver oscillator frequency

Dependencies

To enable this parameter, deselect **ReferenceOffsetPort**.

Programmatic Use**Block parameter:** ReferenceOffset**Type:** character vector**Values:** scalar in the range of [-300,300]**Default:** 0**Simulate using — Select simulation mode**

Code generation (default) | Interpreted execution

Select the simulation mode. This choice affects the simulation performance.

Simulating the model using the Code generation method requires additional startup time, but the subsequent simulations run faster. Simulating the model using the Interpreted execution

method may reduce the startup time, but the subsequent simulations run slower. For more information, see “Simulation Modes”.

Programmatic Use

Block parameter: SimulateUsing

Type: character vector

Values: Code generation| Interpreted execution

Default: Code generation

Phase Noise

Add phase noise — Add phase noise as a function of frequency

on (default) | off

Select to introduce phase noise as a function of frequency. By default, this option is selected.

Phase noise frequency offset (Hz) — Frequency offsets of specified phase noise from carrier frequency

[30e3 100e3 1e6 3e6 10e6] (default) | positive real valued vector

The frequency offsets of the specified phase noise from the carrier frequency, specified as a positive real valued vector in hertz.

Dependencies

To enable this parameter, select **Add phase noise** in the **Impairments** tab.

Programmatic Use

Block parameter: Foffset

Type: character vector

Values: positive real valued vector

Default: [30e3 100e3 1e6 3e6 10e6]

Data Types: double

Phase noise level (dBc/Hz) — Specified phase noise power at phase noise frequency offsets relative to the carrier

[-56 -106 -132 -143 -152] (default) | negative real valued vector

The specified phase noise power in a 1 Hz bandwidth centered at the phase noise frequency offsets relative to the carrier, specified as a negative real valued vector in dBc/Hz. The elements of **Phase noise level** correspond to relative elements in the **Phase noise frequency offset (Hz)** parameter.

Dependencies

To enable this parameter, select **Add phase noise** in the **Impairments** tab.

Programmatic Use

Block parameter: PhaseNoise

Type: character vector

Values: negative real valued vector

Default: [-56 -106 -132 -143 -152]

Data Types: double

Estimate phase noise parameters — Estimate phase noise parameters from measured phase noise data

button

Click to estimate the phase noise parameters from the phase noise measured phase noise data. This calculates the **Period jitter (S)** and **Flicker corner frequency (Hz)** parameters from the **Phase noise frequency offset (Hz)** and **Phase noise level (dBc/Hz)** parameters. As a result, the phase noise profile matches a physical model.

Period jitter (S) – Standard deviation of period jitter

1.7e-15 (default) | positive real scalar

Standard deviation of the period jitter, specified as a positive real scalar in seconds. Period jitter is the deviation in cycle time of a clock signal with respect to the ideal period.

Programmatic Use

Block parameter: PeriodJitter

Type: character vector

Values: positive real scalar

Default: 1.7e-15

Flicker corner frequency (Hz) – Corner frequency of flicker noise

0 (default) | scalar

Corner frequency of the flicker noise, specified as a scalar in hertz. **Flicker corner frequency (Hz)** is defined as the frequency at which the phase noise transitions from $1/f^2$ to $1/f^3$ due to flicker noise. At this frequency, the spectral densities of period jitter and flicker noise are equal.

Programmatic Use

Block parameter: CornerFrequency

Type: character vector

Values: scalar

Default: 0

Customize flicker exponent (Advanced feature) – Customize flicker noise power spectral distribution

off (default) | on

Select this parameter to customize the power spectral distribution of the flicker noise. Traditionally, flicker noise is defined as the $1/f$ noise, but it can vary as $1/f^V$, where $0.8 < V < 1.5$.

Flicker exponent – Flicker noise power exponent

1.0 (default) | 0.8 | 0.9 | 1.1 | 1.2 | 1.3 | 1.4 | 1.5

Flicker noise power exponent, specified between 0.8 to 1.5.

Programmatic Use

Block parameter: FlickerExponent

Type: character vector

Values: 1.0 | 0.8 | 0.9 | 1.1 | 1.2 | 1.3 | 1.4 | 1.5

Default: 1.0

Plot fit – Plot fitted phase noise profile

button

Click to plot the fitted phase noise profile. This allows you how the fitted model matches the specified phase noise data.

Advanced**Symbol time (s) — Time of single symbol duration**

100e-12 (default) | real positive scalar

Time of a single symbol duration, specified as a real positive scalar in seconds.

Programmatic Use**Block parameter:** SymbolTime**Type:** character vector**Values:** real positive scalar**Default:** 100e-12

Data Types: double

Sample interval (s) — Uniform time step of waveform

6.25e-12 (default) | real positive scalar

Uniform time step of the waveform, specified as a real positive scalar in seconds.

Programmatic Use**Block parameter:** SampleInterval**Type:** character vector**Values:** real positive scalar**Default:** 6.25e-12

Data Types: double

Modulation (2:NRZ, 3:PAM3, 4:PAM4) — Modulation scheme

2 (default) | 3 | 4

Number of logic levels in the modulation scheme:

- Select 2 if the modulation scheme is NRZ(non-return to zero).
- Select 3 if the modulation scheme PAM3 (pulse amplitude modulation level 3).
- Select 4 if the modulation scheme PAM4 (pulse amplitude modulation level 4).

Programmatic Use**Block parameter:** Modulation**Type:** character vector**Values:** 2 | 3 | 4**Default:** 2

Data Types: char

Input waveform type — Type of input waveform

Sample (default) | Impulse

Type of input waveform, either a sample by sample signal or an impulse response signal.

Programmatic Use**Block parameter:** WaveType**Type:** character vector**Values:** Sample | Impulse**Default:** Sample

More About

Clock and Data Recovery

To optimize clock and data recovery operation, specify the control sensitivity of the Clock Generator block using the `Period offset` option in the “Specify using” on page 3-0 parameter. This maps the period offset input from the `vctrl` port to the center frequency and control sensitivity configuration. The voltage sensitivity is calculated for the currently configured symbol rate.

IBIS AMI Parameters

The reference offset and phase offset parameters are supported as IBIS-AMI parameters. You can define how they are included in the AMI files by using the “PhaseOffsetPort” on page 3-0 and “ReferenceOffsetPort” on page 3-0 .

Version History

Introduced in R2022a

See Also

Signal Sampler | Ring Oscillator VCO | CDR

Topics

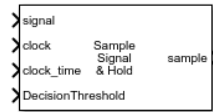
“Clock and Data Recovery in SerDes System” (SerDes Toolbox)

“Model Clock Recovery Loops in SerDes Toolbox” (SerDes Toolbox)

Signal Sampler

Sample incoming signal at the edge of incoming clock

Library: Mixed-Signal Blockset / Utilities



Description

The Signal Sampler block estimates the value of an incoming fixed step discrete sampled signal at a specific time that is typically between two sample times. The block obtains this estimate using an accurate input value for the time and linear interpolation for the signal.

The Signal Sampler block calculates the time interval from the last sample to the exact time at which the clock transition occurred as the modular residue with respect to the sample interval. The block uses linear interpolation to calculate the new output sample value.

Ports

Input

signal — Fixed step discrete input signal

scalar

Fixed step discrete input signal to be sampled, specified as a scalar.

Data Types: `single` | `double` | `int8` | `int16` | `int32` | `uint8` | `uint16` | `uint32` | `fixed point`

clock — Saturated but sampled clock waveform

scalar

Saturated but sampled clock waveform, specified as a scalar. The input at the **clock** port, along with the input at the **clock_time** port, is used to define the time at which to sample the input signal.

Data Types: `double`

clock_time — Exact time of clock transition

scalar

Exact time of the clock transition, specified as a scalar. The input at the **clock_time** port, along with the input at the **clock** port, is used to define the time at which to sample the input signal.

DecisionThreshold — Voltage at which latched output switches polarity

scalar

The voltage at which latched output switches polarity, specified as a scalar.

Data Types: `double`

Output

sample — Sample output value at clock edge

scalar | -1 | 1

Sample output value at the clock edge, returned as a scalar.

The output has two modes:

- **Continuous** — the output is exactly the value of the input signal at the time of the clock edge.
- **Latched** — the output is binary, with values -1 or 1.

Data Types: single | double | int8 | int16 | int32 | int64 | uint8 | uint16 | uint32 | uint64 | Boolean | fixed point

Parameters

Main

Clock edge — Determine when input signal is sampled

Rising (default) | Falling

Determine when the input signal is sampled, either the rising or falling edge of the clock signal.

Programmatic Use

Block parameter: ClockEdge

Type: character vector

Values: Rising | Falling

Default: Rising

Output mode — Determine output sample mode

Continuous (default) | Latched

Determine the output sample mode, either Continuous or Latched.

In Continuous mode, the output is exactly the value of the input signal at the time of the clock edge.

In Latched mode, the output is binary, with values -1 or 1.

Programmatic Use

Block parameter: OutputMode

Type: character vector

Values: Continuous | Latched

Default: Continuous

Simulate using — Select simulation mode

Code generation (default) | Interpreted execution

Select the simulation mode. This choice affects the simulation performance.

Simulating the model using the Code generation method requires additional startup time, but the subsequent simulations run faster. Simulating the model using the Interpreted execution method may reduce the startup time, but the subsequent simulations run slower. For more information, see “Simulation Modes”.

Programmatic Use**Block parameter:** SimulateUsing**Type:** character vector**Values:** Code generation| Interpreted execution**Default:** Code generation**Latched Output****DecisionThresholdPort — Turn on decision threshold port**

on (default) | off

Select to turn on the decision threshold port. When turned on, the voltage at the **DecisionThreshold** port is used to decide when to switch polarity for latched output. By default, this option is selected.

Decision threshold is supported as an AMI parameter.

Programmatic Use**Block parameter:** DecisionThresholdPort**Type:** character vector**Values:** on| off**Default:** on**Latch decision threshold — Voltage at which latched output switches polarity**

0 (default) | scalar

The voltage at which latched output switches polarity, specified as a scalar. Use this parameter to define the decision threshold for latched output in the block itself.

Decision threshold is supported as an AMI parameter.

Dependencies

To enable this parameter, deselect the **DecisionThresholdPort** parameter.

Programmatic Use**Block parameter:** DecisionThreshold**Type:** character vector**Values:** scalar**Default:** 0**Latch sensitivity — Voltage above or below decision threshold needed to assure that latched output switches to correct polarity**

0 (default) | scalar

The voltage above or below the decision threshold needed to assure that the latched output switches to the correct polarity. This parameter is used to model the metastable region of the latch. The behavior is modeled as the worst case, in which the latched output remains unchanged unless the sampled voltage is outside the metastable region. If the data sample voltage lies within the region (\pm **Latch sensitivity**), there is a 50% probability of bit error.

Latch sensitivity is supported as an AMI parameter.

Programmatic Use**Block parameter:** Sensitivity**Type:** character vector**Values:** scalar

Default: 0

Advanced

Symbol time (s) — Time of single symbol duration

100e-12 (default) | real positive scalar

Time of a single symbol duration, specified as a real positive scalar in seconds.

Programmatic Use

Block parameter: SymbolTime

Type: character vector

Values: real positive scalar

Default: 100e-12

Data Types: double

Sample interval (s) — Uniform time step of waveform

6.25e-12 (default) | real positive scalar

Uniform time step of the waveform, specified as a real positive scalar in seconds.

Programmatic Use

Block parameter: SampleInterval

Type: character vector

Values: real positive scalar

Default: 6.25e-12

Data Types: double

Modulation (2:NRZ, 3:PAM3, 4:PAM4) — Modulation scheme

2 (default) | 3 | 4

Number of logic levels in the modulation scheme:

- Select 2 if the modulation scheme is NRZ(non-return to zero).
- Select 3 if the modulation scheme PAM3 (pulse amplitude modulation level 3).
- Select 4 if the modulation scheme PAM4 (pulse amplitude modulation level 4).

Programmatic Use

Block parameter: Modulation

Type: character vector

Values: 2 | 3 | 4

Default: 2

Data Types: char

Input waveform type — Type of input waveform

Sample (default) | Impulse

Type of input waveform, either a sample by sample signal or an impulse response signal.

Programmatic Use

Block parameter: WaveType

Type: character vector

Values: Sample | Impulse

Default: Sample

Version History

Introduced in R2022a

See Also

Clock Generator | CDR

Topics

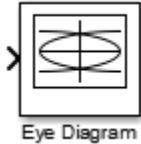
“Clock and Data Recovery in SerDes System” (SerDes Toolbox)

“Model Clock Recovery Loops in SerDes Toolbox” (SerDes Toolbox)

Eye Diagram

Display eye diagram of time-domain signal

Library: Communications Toolbox / Comm Sinks
 Communications Toolbox HDL Support / Comm Sinks
 Mixed-Signal Blockset / Utilities
 SerDes Toolbox / Utilities




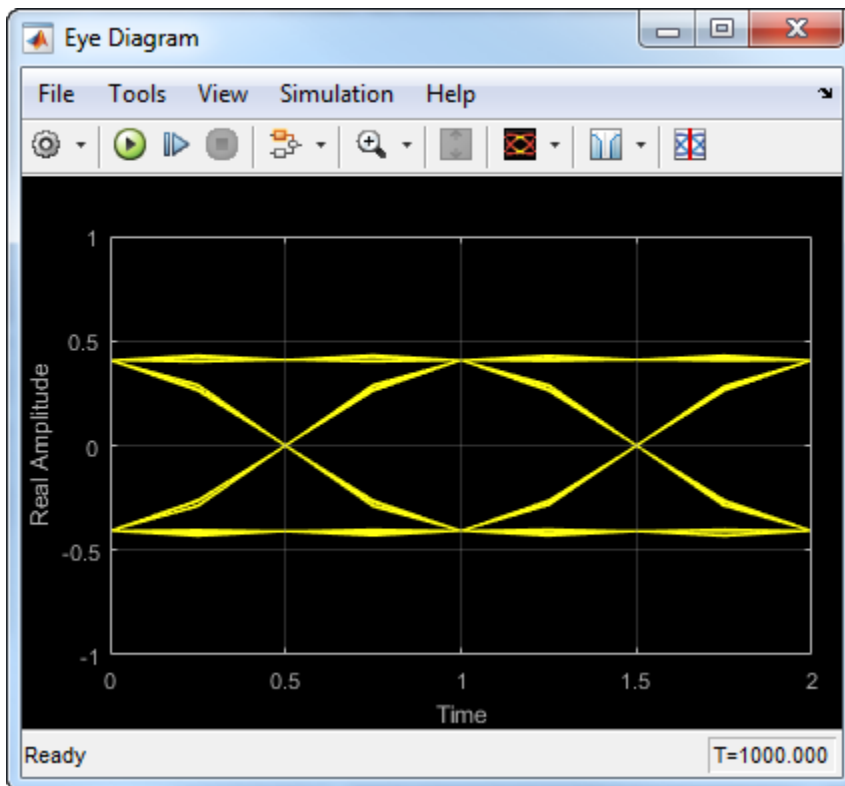
Description

The Eye Diagram block displays multiple traces of a modulated signal to produce an eye diagram. You can use the block to reveal the modulation characteristics of the signal, such as the effects of pulse shaping or channel distortions. .

The Eye Diagram block has one input port. This block accepts a column vector or scalar input signal. The block accepts a signal with the following data types: double, single, base integer, and fixed point. All data types are cast as double before the block displays results.

To modify the eye diagram display, select **View > Configuration Properties** or click the

Configuration Properties button (). Then select the **Main**, **2D color histogram**, **Axes**, or **Export** tabs and modify the settings.



Ports

Input

In — Input signal

scalar | column vector

Input signal, specified as a scalar or column vector.

Data Types: double

Parameters

Main Tab

Display mode — Display mode

Line plot (default) | 2D color Histogram

Display mode of the eye diagram, specified as Line plot or 2D color histogram. Selecting 2D color histogram makes the histogram tab available.

Tunable: Yes

Enable measurements — Enable measurements

off (default) | on

Select this check box to enable eye measurements of the input signal.

Show horizontal (jitter) histogram — Display jitter histogram

off (default) | on

Select this radio button to display the jitter histogram. This can also be accessed by using the histogram button drop down on the toolbar.

Dependencies

This parameter is available when **Display mode** is 2D color histogram and **Enable measurements** is selected.

Show vertical (noise) histogram — Display noise histogram

off (default) | on

Select this radio button to display the noise histogram. This can also be accessed by using the histogram button drop down on the toolbar.

Dependencies

This parameter is available when **Display mode** is 2D color histogram and **Enable measurements** is selected.

Do not show horizontal or vertical histogram — Do not show horizontal or vertical histogram

on (default) | off

Select this radio button to display neither the histogram noise nor the histogram jitter.

Dependencies

This parameter is available when **Display mode** is 2D color histogram and **Enable measurements** is selected.

Show horizontal bathtub curve — Show horizontal bathtub curve

off (default) | on

Select this check box to display the horizontal bathtub curve. This can also be accessed by using the bathtub curve button on the toolbar.

Dependencies

This parameter is available when **Enable measurements** is selected.

Show vertical bathtub curve — Show vertical bathtub curve

off (default) | on

Select this check box to display the vertical bathtub curve. This can also be accessed by using the bathtub curve button on the toolbar.

Dependencies

This parameter is available when **Enable measurements** is selected.

Eye diagram to display — Eye diagram to display

Real only (default) | Real and imaginary

Select either **Real only** or **Real and imaginary** to display one or both eye diagrams. To make eye measurements, this parameter must be **Real only**.

Tunable: Yes**Color fading — Color fading**

off (default) | on

Select this check box to fade the points in the display as the interval of time after they are first plotted increases.

Tunable: Yes**Dependencies**

This parameter is available only when the **Display mode** is **Line plot**.

Samples per symbol — Samples per symbol

8 (default) | positive integer

Number of samples per symbol, specified as a positive integer. Use with **Symbols per trace** to determine the number of samples per trace.

Tunable: Yes**Sample offset — Sample offset**

0 (default) | nonnegative integer

Sample offset, specified as a nonnegative integer smaller than the product of **Samples per symbol** and **Symbols per trace**. The offset provides the number of samples to omit before plotting the first point.

Tunable: Yes**Symbols per trace — Symbols per trace**

2 (default) | positive integer

Number of symbols plotted per trace, specified as a positive integer.

Tunable: Yes**Traces to display — Number of traces to display**

40 (default) | positive integer

Number of traces plotted, specified as a positive integer.

Tunable: Yes

Dependencies

This parameter is available only when the **Display mode** is Line plot

Axes Tab

Title — Title label

None (default)

Label that appears above the eye diagram plot.

Tunable: Yes

Show grid — Toggle scope grid

on (default) | off

Toggle this check box to turn the grid on and off.

Tunable: Yes

Y-limits (Minimum) — Lower limit of y-axis

-1.1 (default) | scalar

Minimum value of the y-axis.

Tunable: Yes

Y-limits (Maximum) — Upper limit of y-axis

1.1 (default) | scalar

Maximum value of the y-axis.

Tunable: Yes

Real axis label — Real axis label

Real Amplitude (default)

Text that the scope displays along the real axis.

Tunable: Yes

Imaginary axis label — Imaginary axis label

Imaginary Amplitude (default)

Text that the scope displays along the imaginary axis.

Tunable: Yes

2D Histogram Tab

The 2D histogram tab is available when you click the histogram button or when the display mode is set to 2D color histogram.

Oversampling method — Oversampling method

None (default) | Input interpolation | Histogram interpolation

Oversampling method, specified as None, Input interpolation, or Histogram interpolation.

To plot eye diagrams as quickly as possible, set the **Oversampling method** to None. The drawback to not oversampling is that the plots look pixelated when the number of samples per trace is small. To create smoother, less-pixelated plots using a small number of samples per trace, set the **Oversampling method** to Input interpolation or Histogram interpolation. Input interpolation is the faster of the two interpolation methods and produces good results when the signal-to-noise ratio (SNR) is high. With a lower SNR, this oversampling method is not recommended because it introduces a bias to the centers of the histogram ranges. Histogram interpolation is not as fast as the other techniques, but it provides good results even when the SNR is low.


Tunable: Yes

Color scale — Color scale

Linear (default) | Logarithmic

Color scale of the histogram plot, specified as either Linear or Logarithmic. Set **Color scale** to Logarithmic if certain areas of the eye diagram include a disproportionate number of points.

Tunable: Yes

The toolbar contains a histogram reset button , which resets the internal histogram buffers and clears the display. This button is not available when the display mode is set to Line plot.

Export Tab

Export measurements, histograms and bathtub curves — Export measurements, histograms and bathtub curves

Off (default) | off

Select this check box export the eye diagram measurements to the MATLAB workspace.

Tunable: Yes

Variable name — Variable name

EyeData (default)

Specify the name of the variable to which the eye diagram measurements are saved. The data is saved as a structure having these fields:

- MeasurementSettings

- Measurements
- JitterHistogram
- NoiseHistogram
- HorizontalBathtub
- VerticalBathtub
- BlockName

Tunable: Yes

Style Dialog Box

In the **Style** dialog box, you can customize the style of the active display. You can change the color of the figure containing the displays, the background and foreground colors of display axes, and properties of lines in a display. To open this dialog box, select **View > Style**.

Figure color — Figure color

black (default)

Specify the background color of the scope figure.

Axes colors — Axes colors

black | gray (default)

Specify the fill and line colors for the axes.

Line — Line style, thickness and color for line plots

continuous | 0.5 | yellow (default)

Specify the line style, line width, and line color for the displayed signal.

Dependencies

This parameter is available only when the **Display mode** is Line plot.

Marker — Data point marker

None (default) | ...

Data point marker for the selected signal, specified as one of the choices in this table data point markers. This parameter is similar to the Marker property for MATLAB Handle Graphics® plot objects.

Specifier	Marker Type
none	No marker (default)
○	Circle
□	Square
×	Cross
●	Point
+	Plus sign
*	Asterisk

Specifier	Marker Type
◇	Diamond
▽	Downward-pointing triangle
△	Upward-pointing triangle
◁	Left-pointing triangle
▷	Right-pointing triangle
☆	Five-pointed star (pentagram)
☆☆	Six-pointed star (hexagram)

Dependencies

This parameter is available only when the **Display mode** is `Line plot`.

Colormap — Colormap for histograms

Hot (default) | Parula | Jet | HSV | Cool | SpringSummer | Autumn | Winter | Gray | Bone | Copper | Pink | Lines | Custom

Specify the colormap of the histogram plots as one of these schemes: Parula, Jet, HSV, Hot, Cool, Spring, Summer, Autumn, Winter, Gray, Bone, Copper, Pink, Lines, or Custom. If you select Custom, a dialog box pops up from which you can enter code to specify your own colormap.

Dependencies

This parameter is available only when the **Display mode** is `2D color histogram`.

Measurement Settings Pane

To change measurement settings, first select **Enable measurements**. Then, in the **Eye Measurements** pane, click the arrow next to **Settings**. You can control these measurement settings.

Eye level boundaries — Time range for calculating eye levels

[40 60] (default) | two-element vector

Time range for calculating eye levels, specified as a two-element vector. These values are expressed as a percentage of the symbol duration.

Tunable: Yes

Decision boundary — Amplitude level threshold

0 (default) | scalar

Amplitude level threshold in V, specified as a scalar. This parameter separates the different signaling regions for horizontal (jitter) histograms. This parameter is tunable, but the jitter histograms reset when the parameter changes.

For non-return-to-zero (NRZ) signals, set **Decision boundary** to 0. For return-to-zero (RZ) signals, set **Decision boundary** to half the maximum amplitude.

Tunable: Yes

Rise/Fall thresholds — Amplitude levels of the rise and fall transitions

[10 90] (default) | two-element vector

Amplitude levels of the rise and fall transitions, specified as a two-element vector. These values are expressed as a percentage of the eye amplitude. This parameter is tunable, but the crossing histograms of the rise and fall thresholds reset when the parameter changes.

Tunable: Yes

Hysteresis — Amplitude tolerance of the horizontal crossings

0 (default) | scalar

Amplitude tolerance of the horizontal crossings in V, specified as a scalar. Increase hysteresis to provide more tolerance to spurious crossings due to noise. This parameter is tunable, but the jitter and the rise and fall histograms reset when the parameter changes.

Tunable: Yes

BER threshold — BER used for eye measurements

1e-12 (default) | nonnegative scalar from 0 to 0.5

BER used for eye measurements, specified as a nonnegative scalar from 0 to 0.5. The value is used to make measurements of random jitter, total jitter, horizontal eye openings, and vertical eye openings.

Tunable: Yes

Bathtub BERs — BER values used to calculate openings of bathtub curves

[0.5 0.1 0.01 0.001 0.0001 1e-05 1e-06 1e-07 1e-08 1e-09 1e-10 1e-11 1e-12] (default) | vector

BER values used to calculate openings of bathtub curves, specified as a vector whose elements range from 0 to 0.5. Horizontal and vertical eye openings are calculated for each of the values specified by this parameter.

Tunable: Yes

Dependencies

To enable this parameter, select **Show horizontal bathtub curve**, **Show vertical bathtub curve**, or both.

Measurement delay — Duration of initial data discarded from measurements

0 (default) | nonnegative scalar

Duration of initial data discarded from measurements, in seconds, specified as a nonnegative scalar.

Block Characteristics

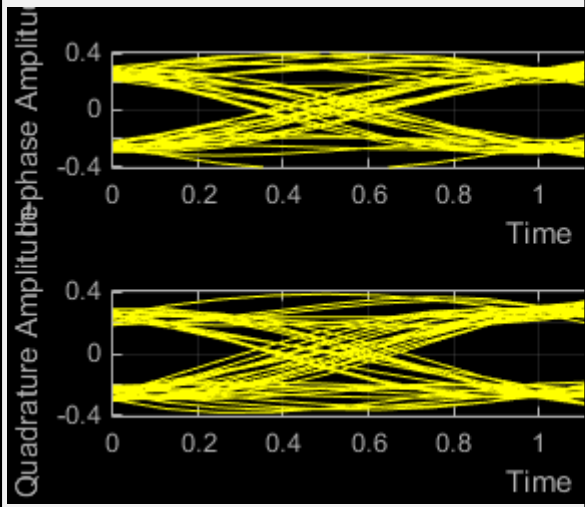
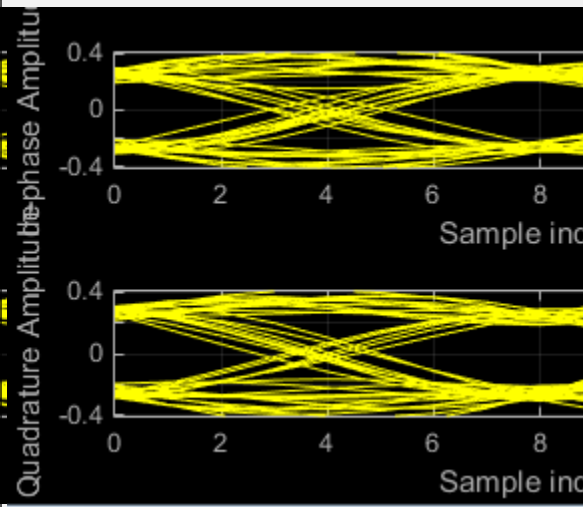
Data Types	Boolean double enumerated fixed point integer single
Direct Feedthrough	no
Multidimensional Signals	no
Variable-Size Signals	no
Zero-Crossing Detection	no

More About

Using Eye Diagram in Conditionally Executed Subsystems

When an Eye Diagram block is placed in a conditionally executed subsystem, for example in a triggered or enabled subsystem:

- Input size must be an integer multiple of `SamplesPerSymbol * SymbolsPerTrace`
- Sample offset must be zero
- The rightmost part of the display is intentionally omitted. This figure compares typical eye diagram display when placed in a normal system versus one placed in a conditionally executed subsystem.

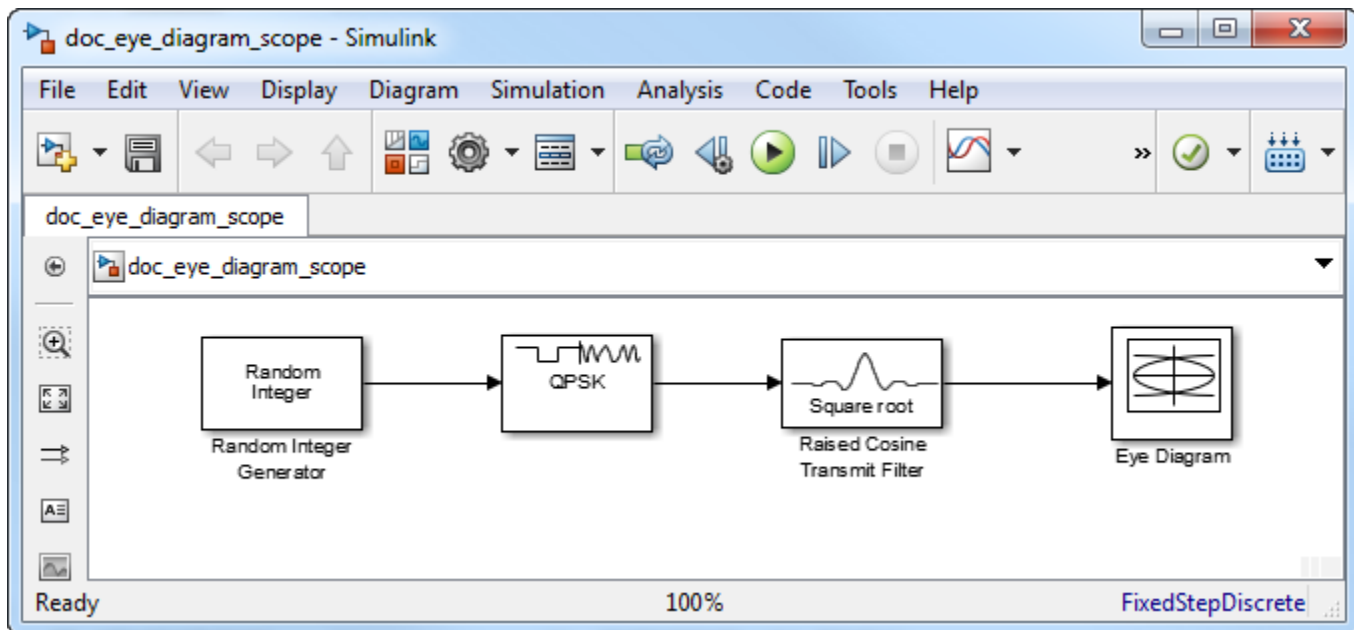
Eye Diagram Plot in Normal System	Eye Diagram Plot in Conditionally Executed Subsystem
	
<p>In a regular Eye Diagram, the rightmost part is a line between the last sample of a trace and the first sample of the next trace.</p>	<p>In conditionally executed subsystems, these traces may be non-contiguous, thus this rightmost segment could corrupt the display and is omitted.</p>

View Eye Diagram

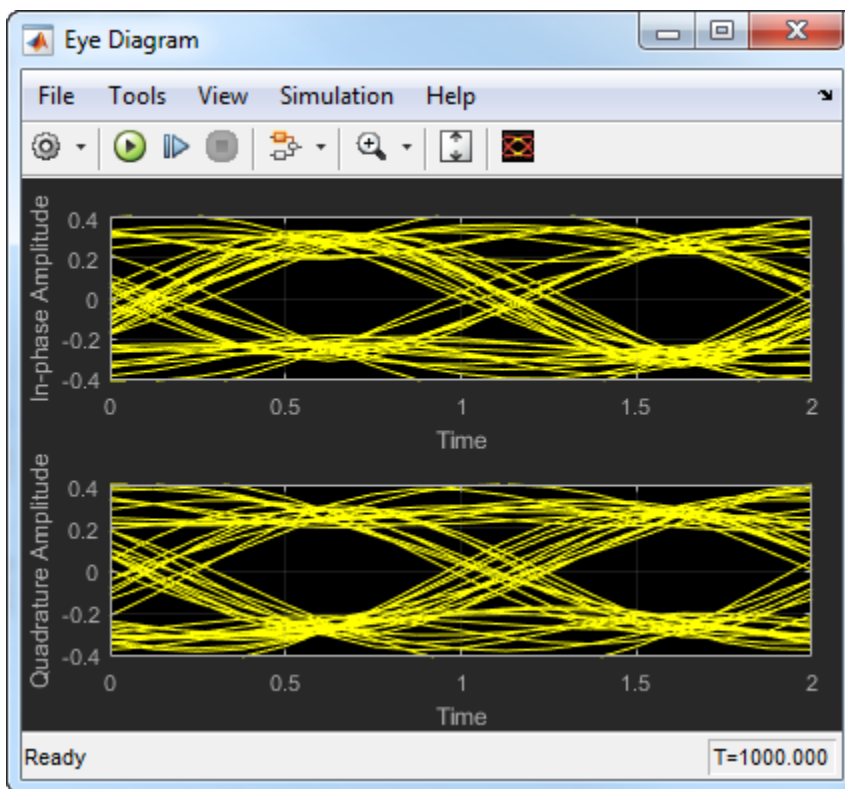
Display the eye diagram of a filtered QPSK signal using the Eye Diagram block.

Load the `doc_eye_diagram_scope` model from the MATLAB command prompt.

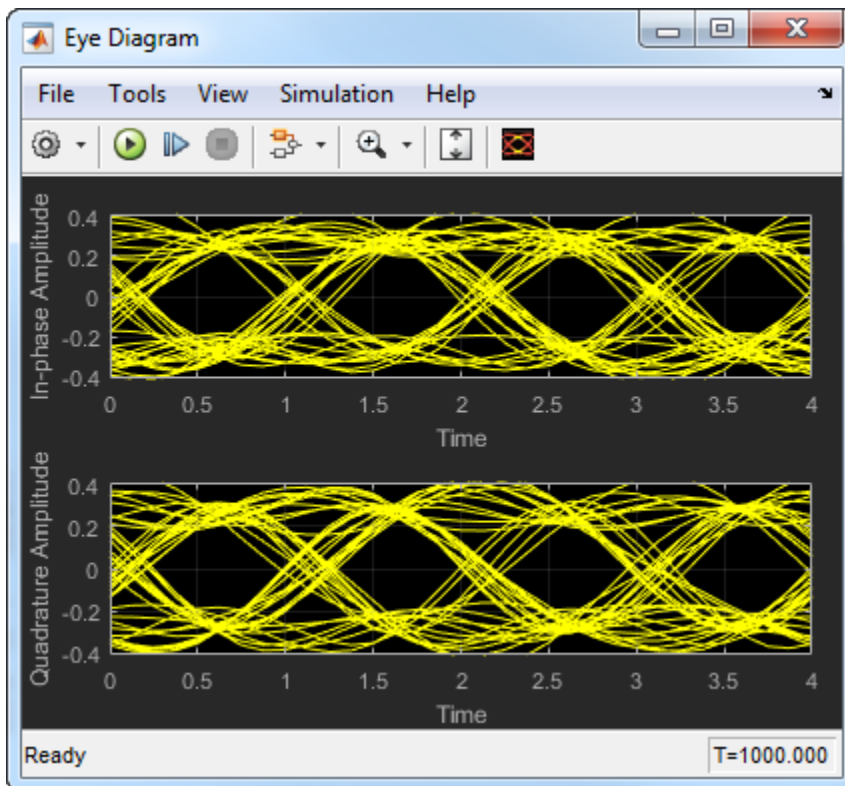
```
doc_eye_diagram_scope
```



Run the model and observe that two symbols are displayed.



Open the configuration parameters dialog box. Change the **Symbols per trace** parameter to 4. Run the simulation and observe that four symbols are displayed.



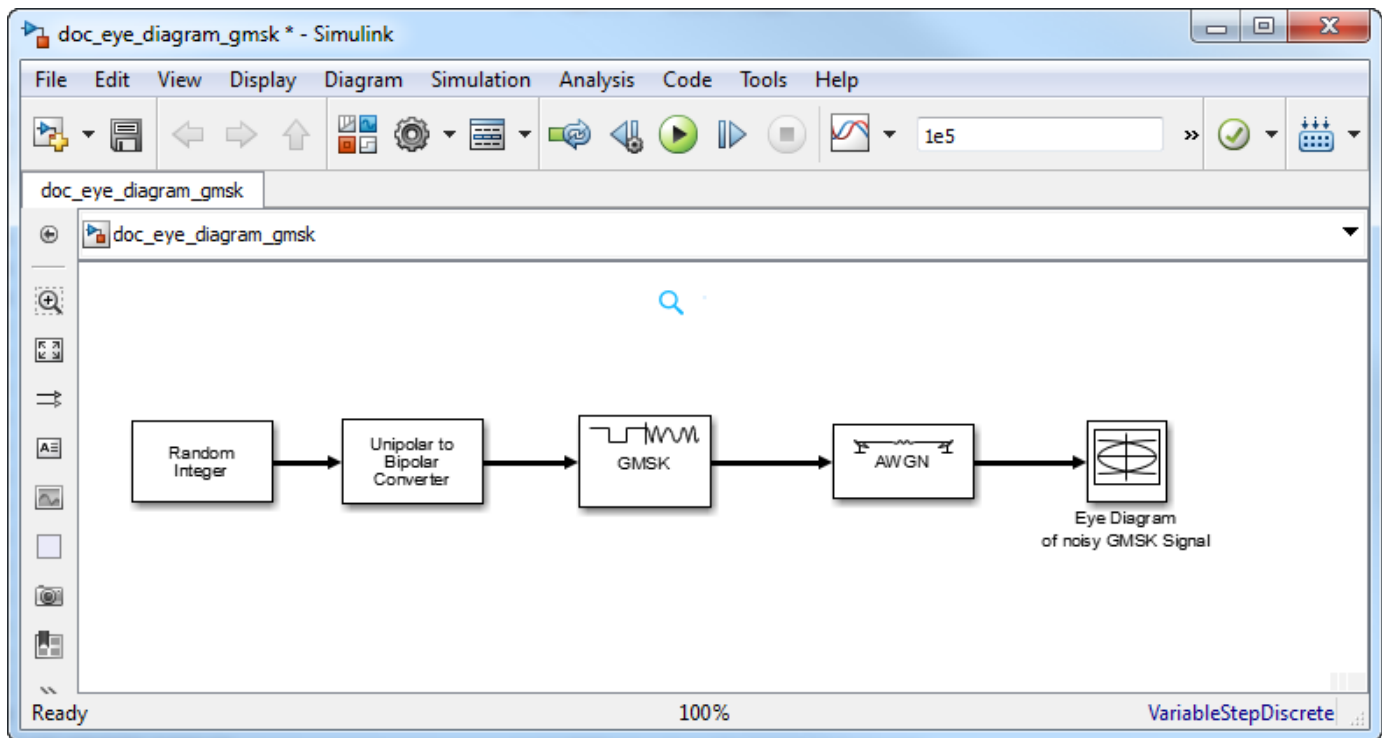
Try changing the Raised Cosine Transmit Filter parameters or changing additional Eye Diagram parameters to see their effects on the eye diagram.

Histogram Plots

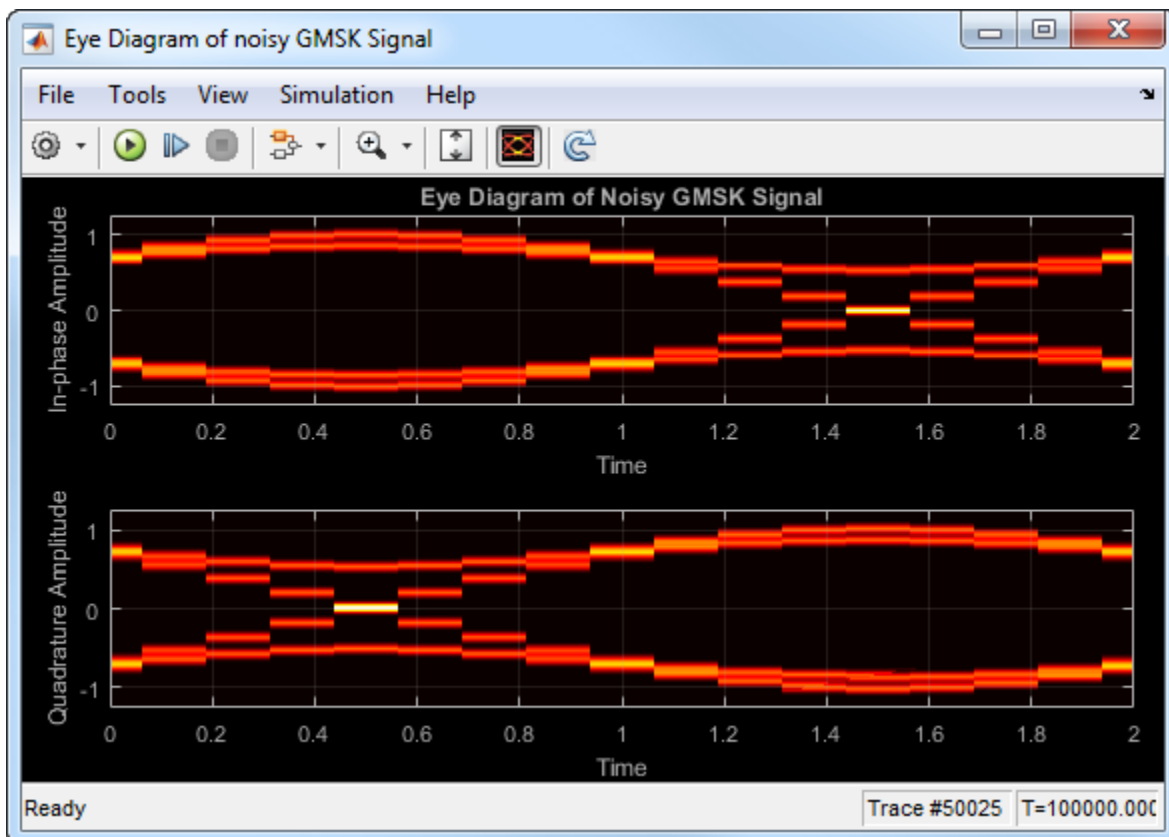
Display histogram plots of a noisy GMSK signal.

Load the `doc_eye_diagram_gmsk` model from the MATLAB command prompt.

```
doc_eye_diagram_gmsk
```

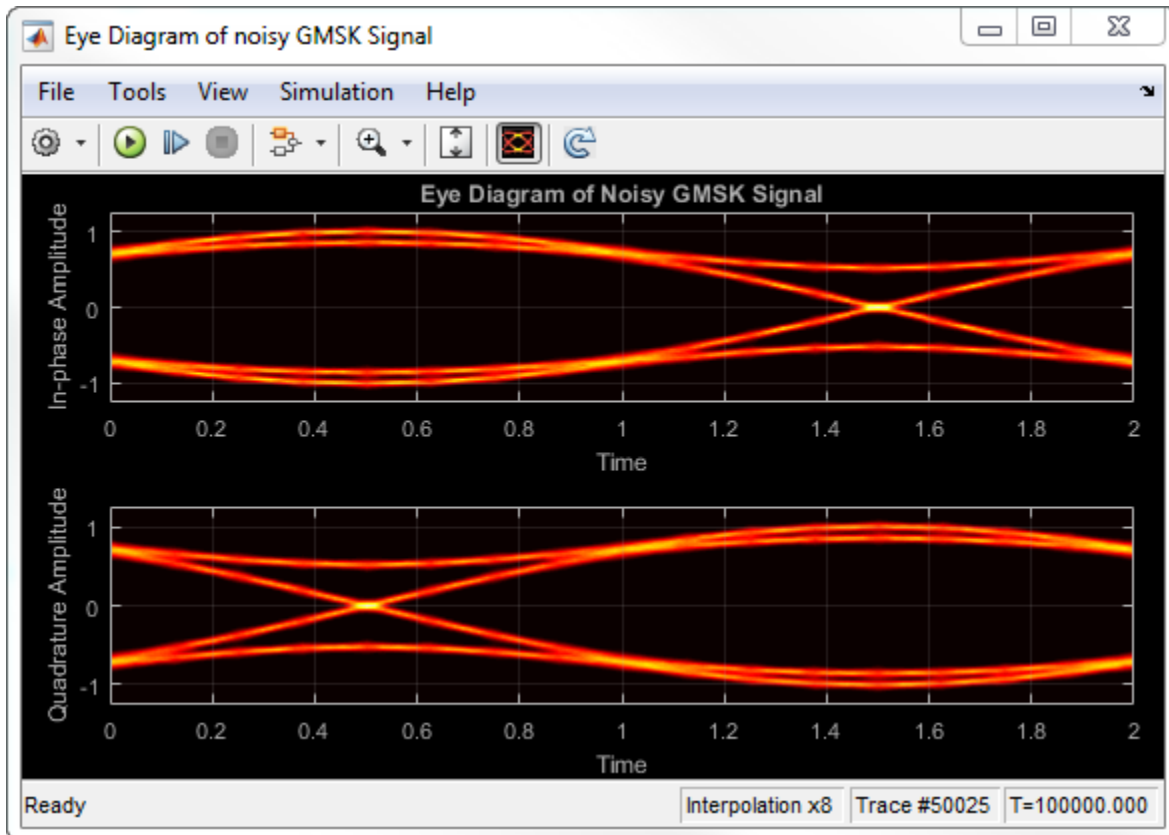


Run the model. The eye diagram is configured to show a histogram without interpolation.



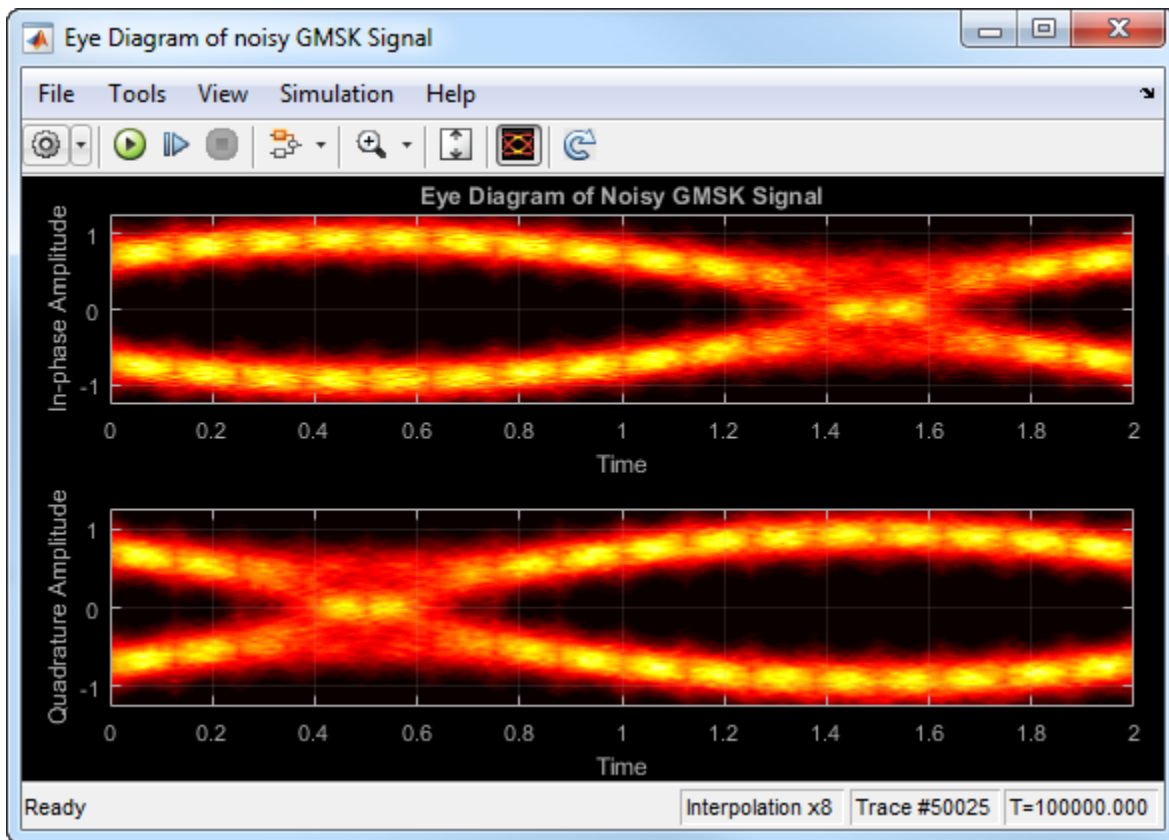
The lack of interpolation results in a plot having piecewise-continuous behavior.

Open the **2D Histogram** tab of the Configuration Properties dialog box. Set the **Oversampling method** to Input interpolation. Run the model.



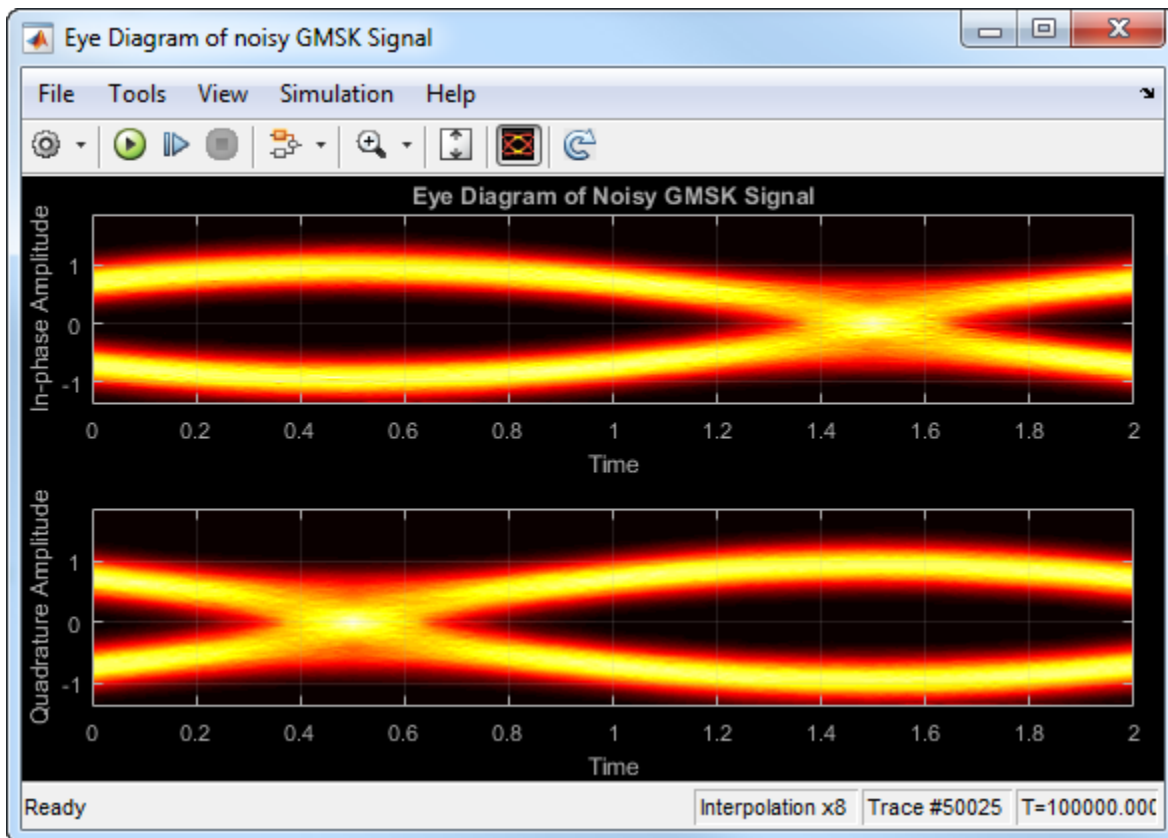
The interpolation smooths the eye diagram.

On the AWGN Channel block, change **SNR (dB)** from 25 to 10. Run the model.



Observe that vertical striping is present in the eye diagram. This striping is the result of input interpolation, which has limited accuracy in low-SNR conditions.

Set the **Oversampling method** to Histogram interpolation. Run the model.



The eye diagram plot now renders accurately because the histogram interpolation method works for all SNR values. This method is not as fast as the other techniques and results in increased execution time.

Version History

Introduced in R2014a

Extended Capabilities

C/C++ Code Generation

Generate C and C++ code using Simulink® Coder™.

This block is excluded from the generated code when code generation is performed on a system containing this block.

HDL Code Generation

Generate Verilog and VHDL code for FPGA and ASIC designs using HDL Coder™.

This block can be used for simulation visibility in subsystems that generate HDL code, but is not included in the hardware implementation.

Functions: Data Converters

inldnl

Integral nonlinearity (INL) and differential nonlinearity (DNL) of data converters

Syntax

```
s = inldnl(analog,digital,range,type)
s = inldnl( ____,Name,Value)
```

Description

`s = inldnl(analog,digital,range,type)` calculates the integral nonlinearity (INL) and differential nonlinearity (DNL) errors of ADCs and DACs. The function calculates INL and DNL using the analog and digital input output data and the nominal analog dynamic range of the converter. The function can calculate INL and DNL either using the endpoint method, or the best fit method, or using both methods.

The `inldnl` function only analyzes converters with a finite number of bits. That means ADCs must have saturation and quantization. The function ignores any digital value pairs that contain NaN values.

`s = inldnl(____,Name,Value)` calculates the INL and DNL errors of ADCs and DACs using one or more name-value pair arguments in addition to the input arguments in the previous syntax. Enclose each argument name in quotes. Unspecified arguments take default values.

Note Initial conditions and other anomalous data can cause this function to behave erratically. This function can analyze nonmonotonic converters, but it cannot handle multiple distinct occurrences of the same code in one transfer function.

Examples

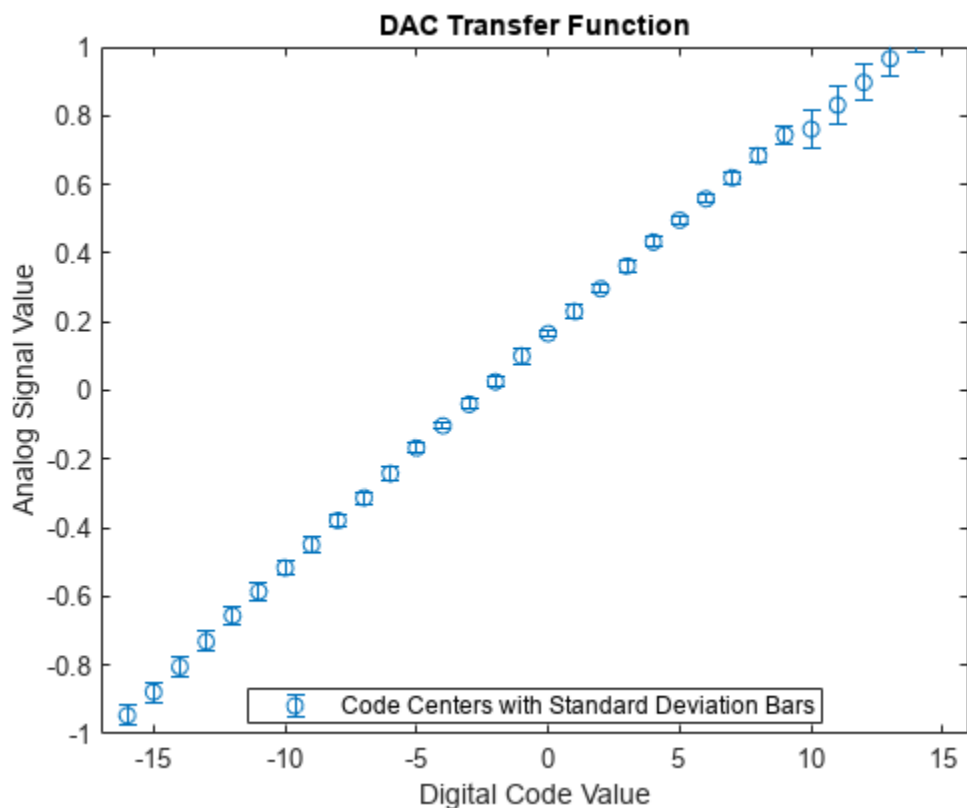
Calculate INL and DNL of DAC

Load the digital input and the analog output of a DAC from MAT-files.

```
load 'digital.mat'
load 'analog.mat'
```

The nominal analog dynamic range of the DAC is $[-1, 1]$. Turn on plotting for the output converter threshold. Calculate INL and DNL using both best fit and endpoint methods.

```
inldnl(a,d,[-1 1], 'DAC', 'GenPlotData', 'on', 'INLMethod', 'All', 'DNLMethod', 'All')
```



```
ans = struct with fields:
    Type: 'DAC'
    NBits: 5
    LSB: 0.0625
    MissingCodes: [0x1 double]
    Codes: [-16 -15 -14 -13 -12 -11 -10 -9 -8 -7 -6 -5 -4 -3 ... ]
    IdealCodeCenters: [-1 -0.9375 -0.8750 -0.8125 -0.7500 -0.6875 ... ]
    CodeCenters: [-0.9465 -0.8780 -0.8048 -0.7310 -0.6557 -0.5862 ... ]
    CodeCenterStd: [0.0282 0.0292 0.0296 0.0287 0.0254 0.0237 0.0207 ... ]
    EndpointINL: [1.7764e-15 0.0465 0.1679 0.2985 0.4547 0.5169 ... ]
    BestFitINL: [-0.5244 -0.4713 -0.3432 -0.2060 -0.0432 0.0257 ... ]
    EndpointDNL: [0.0465 0.1215 0.1305 0.1562 0.0623 0.0654 0.0331 ... ]
    BestFitDNL: [0.0531 0.1281 0.1371 0.1628 0.0689 0.0720 0.0397 ... ]
    BestFitPoly: [0.0652 0.1293]
    OffsetError: 0.8562
    GainError: 1.5393
    GainErrorUnit: 'LSB'
    TCNominal: [32x2 double]
    TCMeasured: [32x2 double]
```

Input Arguments

analog — Analog input to or output from device
vector

- If the device under test (DUT) is an ADC, analog input to the ADC, specified as a vector.
- If the DUT is a DAC, analog output from the DAC, specified as a vector.

Data Types: `double`

digital — Digital output from or input to device

integer vector

- If the device under test (DUT) is an ADC, digital output from the ADC, specified as a vector of integers.
- If the DUT is a DAC, digital input to the DAC, specified as a vector with integer values.

Data Types: `fi` | `single` | `double` | `int8` | `int16` | `int32` | `uint8` | `uint16` | `uint32`

range — Nominal analog dynamic range of device

2-element vector

Nominal analog dynamic range of the ADC or DAC, specified as a 2-element vector.

Data Types: `double`

type — Type of device

`Auto` | `ADC` | `DAC`

Type of the device under test, specified as `Auto`, `ADC`, or `DAC`. The type determines whether to analyze the data as an ADC or DAC.

If The `type` is set to `Auto` and if the transfer function is discrete, the `inldnl` function analyzes the data as a DAC. The transfer function is considered as discrete if the analog data is less than half of the digital code width for each digital code.

If The `type` is set to `Auto` and if the transfer function is continuous, the `inldnl` function analyzes the data as an ADC.

Data Types: `string`

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1, . . . , NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Before R2021a, use commas to separate each name and value, and enclose Name in quotes.

Example: `inldnl(a,d,[-1 1], 'DAC', 'INLMethod', 'All', 'DNLMMethod', 'All')` calculates the INL and DNL of a DAC using both endpoint and best fit method.

OffsetErrorUnit — Unit of reported offset error

`LSB` (default) | `%FS` | `FS` | `All`

Unit of reported offset error, specified as `LSB` (least significant bit), `%FS` (percentage full scale), `FS` (full scale), or `all`.

Note The full scale range of the converter is defined as the difference between the last and first code on the +0.5 LSB compensated transfer curve. In a +0.5 LSB compensated transfer curve, first code is

0.5 LSB wide while the last code is 1.5 LSB wide. The input values must be considered within the full scale range of the converter.

Note LSB is calculated by the equation $LSB = \frac{\text{Full scale range of converter}}{2^{Nbits}}$.

Data Types: string

GainErrorUnit — Unit of reported gain error

LSB (default) | %FS | FS | All

Unit of reported gain error, specified as LSB (least significant bit), %FS (percentage full scale), FS (full scale), or all.

Note The full scale range of the converter is defined as the difference between the last and first code on the +0.5 LSB compensated transfer curve. In a +0.5 LSB compensated transfer curve, first code is 0.5 LSB wide while the last code is 1.5 LSB wide. The input values must be considered within the full scale range of the converter.

Note LSB is calculated by the equation $LSB = \frac{\text{Full scale range of converter}}{2^{Nbits}}$.

Data Types: string

GenPlotData — Send output data vectors to output data structure

off (default) | on

Send the output data vectors of the inldnl function to the output data structure *s*, specified as off or on. If GenPlotData is set to on, the output data structure contains the output data vectors. The output data vectors can then be picked up by the DAC DC measurement, DAC Testbench, ADC DC Measurement, or ADC Testbench blocks to plot the DC analysis results.

Data Types: string

INLMethod — Method to calculate INL

Endpoint (default) | BestFit | All

Method to calculate INL, specified as Endpoint, BestFit, or All.

- If INLMethod is set to Endpoint, the inldnl function compares each threshold's position to the threshold position of an ideal converter, as determined by a line from the first code transition to the last code transition.
- If INLMethod is set to BestFit, the inldnl function first takes the best linear fit of the ADC or DAC transfer curve. Then the function proceeds to calculate the INL using the same steps as the Endpoint method.

Data Types: string

DNLMethod — Method to calculate DNL

Endpoint (default) | BestFit | All

Method to calculate DNL, specified as `Endpoint`, `BestFit`, or `All`.

- If `DNLMethod` is set to `Endpoint`, the `inldnl` function compares each threshold's position to the threshold position of an ideal converter, as determined by a line from the first code transition to the last code transition to find the INL. The DNL is calculated from the difference between the elements of the INL vector.
- If `DNLMethod` is set to `BestFit`, the `inldnl` function first takes the best linear fit of the ADC or DAC transfer curve. Then the function proceeds to calculate the DNL using the same steps as the `Endpoint` method.

Data Types: `string`

AbsoluteError — Return absolute error and full scale DNL for testing

`off` (default) | `on`

Return absolute error and full scale DNL for testing, specified as `on` or `off`. Absolute error is the total uncompensated error including offset error, gain error, and nonlinearities. In simulation, to specifically test that the measurements match the impairments, absolute error can be used instead of INL. This is because absolute error describes the entire transfer curve in a single vector.

Data Types: `string`

Output Arguments

s — Output device information

structure

Output information of the `inldnl` function, returned as a structure. The output contains information about the device under test in these fields:

Name	Values	Description	Data Types
Type	ADC or DAC	Type of the device under test (DUT)	string
Nbits	positive real integer	Resolution of the ADC or DAC DUT	double
LSB	positive real scalar	Least significant bit value of the DUT. LSB is the smallest level the ADC can convert or the smallest increment of the DAC output.	double
MissingCodes	vector	Missing codes in DUT.	double
Codes	column vector	Digital code	double
IdealCodeCenters	column vector	Ideal code center of the digital code	double
CodeCenters	column vector	Calculated code center of the digital code	double

Name	Values	Description	Data Types
CodeCenterStd	column vector	Standard deviation of the code center from the ideal value	double
EndpointINL	column vector	INL using Endpoint method	double
BestFitINL	column vector	INL using BestFit method	double
EndPointDNL	column vector	DNL using Endpoint method	double
BestFitDNL	column vector	DNL using BestFit method	double
BestFitPoly	vector	Polynomial describing the best fit using standard curve-fitting technique.	double
OffsetError	real scalar	Offset error of DUT	double
GainError	real scalar	Gain error of DUT	double
OffsetErrorUnit	LSB, %FS, or FS	Unit of reported offset error	string
GainErrorUnit	LSB, %FS, or FS	Unit of reported gain error	string
TCNominal	vector	Nominal transfer curve of the DUT	double
TCMeasured	vector	Measured transfer curve of the DUT	double

If you do not assign an output variable, the `inldnl` function also plots the transfer function of the device under test in the active figure.

Data Types: `struct`

More About

Offset Error

Offset error represents the offset of the data converter transfer function curve from its ideal value at a single point. For more information, see “Measuring Offset and Gain Errors in ADC”.

Gain Error

Gain error represents the deviation of the slope of the data converter transfer function curve from its ideal value. For more information, see “Measuring Offset and Gain Errors in ADC”.

INL Error

Integral nonlinearity (INL) error, also termed as relative accuracy, is the maximum deviation of the measured transfer function from a straight line. The straight line can either be a best fit using

standard curve-fitting technique, or be drawn between the endpoints of the actual transfer function after gain adjustment.

The best fit method gives a better prediction of distortion in AC applications, and a lower value of linearity error. The endpoint method is mostly used in the measurement applications of data converters, since the error budget depends on actual deviation from the ideal transfer function.

DNL Error

Differential nonlinearity (DNL) is the deviation from the ideal difference (1 LSB) between analog input levels that trigger any two successive digital output levels. The DNL error is the maximum value of DNL found at any transition.

Version History

Introduced in R2020a

See Also

SAR ADC | ADC DC Measurement | Flash ADC

Topics

“Measuring Offset and Gain Errors in ADC”

Functions: Phase-Locked Loop

phaseNoiseMeasure

Measure and plot phase noise profile of time or frequency-domain signal

Syntax

```
PNMeasure = phaseNoiseMeasure(Xin,Yin,RBW,FrOffset,PlotOption,tag,Name,Value)
PNMeasure = phaseNoiseMeasure(oooooooooooooooo1` ____,PNTarget,Name,Value)
[PNMeasure,GenFrOffset,GenPN] = phaseNoiseMeasure( ____ )
```

Description

`PNMeasure = phaseNoiseMeasure(Xin,Yin,RBW,FrOffset,PlotOption,tag,Name,Value)` measures the phase noise levels of either a time or frequency-domain signal at the specified frequency offset points. The function also plots phase noise profile at the specified frequency offset points when you specify the `PlotOption` argument as 'on'. If you specify the Name-Value pair argument, enclose each argument name in quotes. Unspecified arguments take default values.

`PNMeasure = phaseNoiseMeasure(oooooooooooooooo1` ____,PNTarget,Name,Value)` in addition to the input arguments in the previous syntax compares phase noise levels at the specified frequency offsets to a target phase noise profile. Set the `PlotOption` argument to 'on' to plot and compare the measured phase noise profile to the target profile.

`[PNMeasure,GenFrOffset,GenPN] = phaseNoiseMeasure(____)` additionally returns the phase noise waveform data represented by the frequency offset vector and the corresponding phase noise vector.

Examples

Phase Noise Profile from Frequency and Power Vectors

Load the power spectrum data (frequency and power vectors) of the signal obtained from spectrum analysis.

```
load frequency.mat;
load corresponding_power.mat;
```

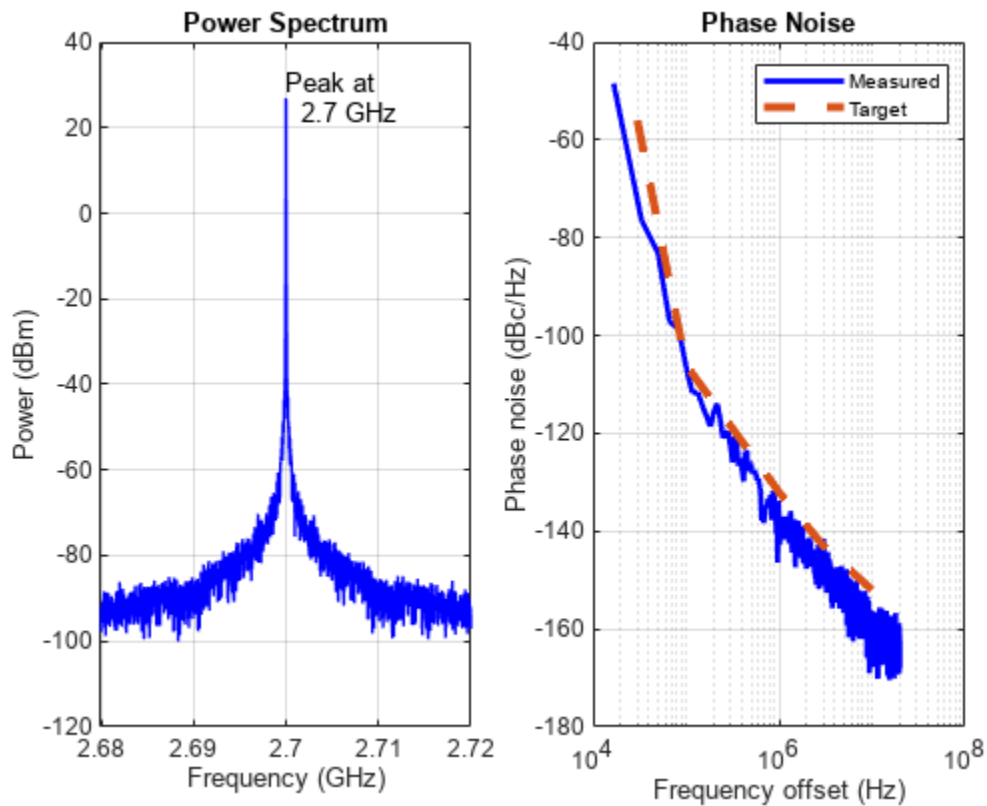
Set the resolution bandwidth of the spectrum analysis to 25 kHz. The frequency offset points are 30 kHz, 100 kHz, 1 MHz, 3 MHz, and 10 MHz. The target phase noise profile corresponding to these frequency offset points is:

- -56 dBc/Hz at 30 kHz
- -106 dBc/Hz at 100 kHz
- -132 dBc/Hz at 1 MHz
- -143 dBc/Hz at 3 MHz
- -152 dBc/Hz at 10 MHz

```
rbw = 25e3;
FrOffset = [30e3 100e3 1e6 3e6 10e6];
PNTarget = [-56 -106 -132 -143 -152];
```

Use the phaseNoiseMeasure function to measure and plot the phase noise profile.

```
PNMeasure = phaseNoiseMeasure(f1,p1,rbw,FrOffset,'on','Phase noise', PNTarget)
```



```
PNMeasure = 5×1
```

```
-70.8795
-106.2594
-136.6468
-147.3779
-157.0967
```

Phase Noise Profile from Time Domain Signal

Load the time domain signal represented by the time and signal value vectors.

```
load time_1.mat;
load signal_1.mat;
```

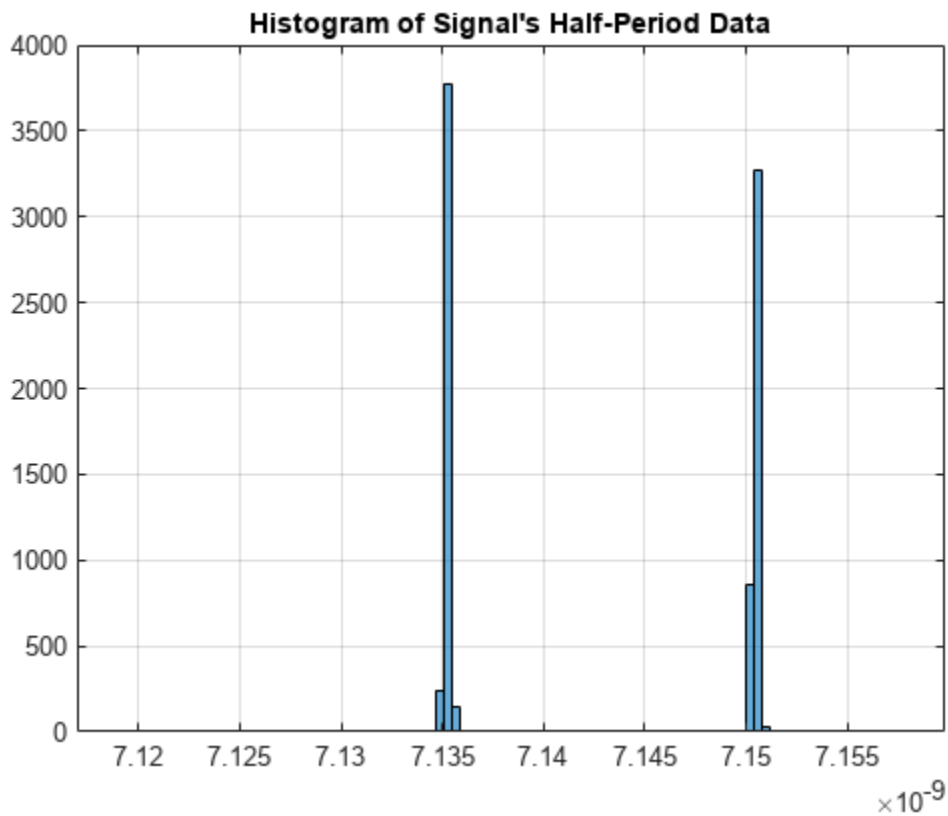
Set the resolution bandwidth of the spectrum analysis to 75 kHz. The frequency offset points are 100 kHz, 300 kHz, 500kHz, 1 MHz, 3 MHz, and 10 MHz. The target phase noise profile corresponding to these frequency offset points is:

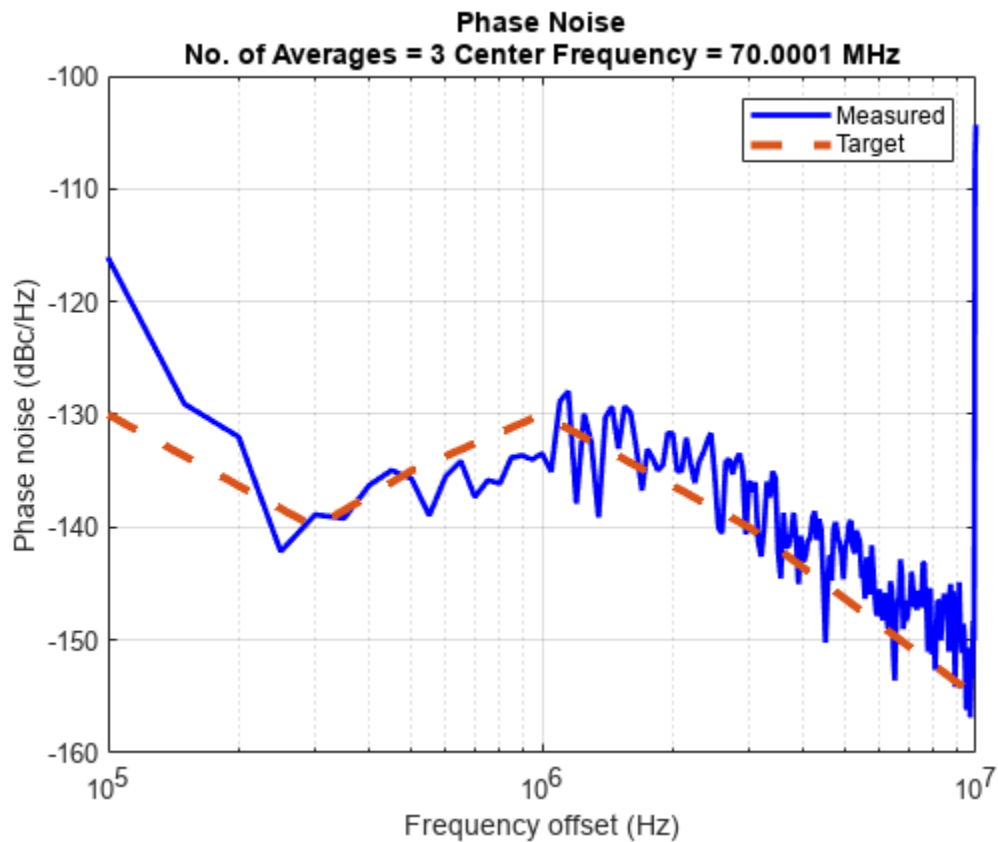
- -130 dBc/Hz at 100 kHz
- -140 dBc/Hz at 300 kHz
- -135 dBc/Hz at 500 kHz
- -130 dBc/Hz at 1 MHz
- -140 dBc/Hz at 3 MHz
- -155 dBc/Hz at 10 MHz

```
rbw = 75e3;
FrOffset = [100e3 300e3 500e3 1e6 3e6 10e6];
PNTarget = [-130 -140 -135 -130 -140 -155];
```

Use the phaseNoiseMeasure function to measure and plot the phase noise profile.

```
[PNMeasure] = phaseNoiseMeasure(t,x,rbw,FrOffset,'on','Phase noise',PNTarget,'Type','Time','Hist
```





PNMeasure = 6x1

```
-116.0967
-138.8853
-135.6704
-133.4544
-135.8667
-104.3244
```

Input Arguments

Xin — Time or frequency vector

positive real vector

Time or frequency vector, specified as a positive real vector. If you specify a time-domain signal or 'Type' name-value pair as 'Time', Xin is a time vector in seconds. If you specify a frequency-domain signal or 'Type' name-value pair as 'Frequency', Xin is a frequency vector in hertz.

Data Types: double

Yin — Signal value or power vector

real vector

Signal value or power vector, specified as a real vector. If you specify a time-domain signal or 'Type' name-value pair as 'Time', Yin is a signal value vector in volts. If you specify a frequency-domain signal or 'Type' name-value pair as 'Frequency', Yin is a power vector in dBm.

Data Types: double

RBW — Resolution bandwidth used in spectrum analysis

positive real scalar

Resolution bandwidth used in the spectrum analysis, specified as a positive real scalar in hertz. RBW defines the smallest positive frequency at which the frequency components of a signal can be resolved.

Data Types: double

FrOffset — Frequency offset points to measure phase noise

positive real vector

Frequency offset points at which phase noise levels are calculated, specified as a positive real vector in hertz.

Data Types: double

PlotOption — Plot phase noise analysis results

'on' | 'off'

Plot the phase noise analysis results in a figure, specified as on or off. Set PlotOption to 'on' to view the power spectrum and phase noise profile plots. If the Name-Value pair argument 'Type' is specified as 'Time', you can only plot the phase noise profile of the time-domain signal.

Data Types: char

tag — Figure identifier

string scalar | character vector

Figure identifier, specified as a string scalar or a character vector. Tag figures to keep multiple plots open as you change the simulation parameters.

Data Types: char | string

PNTarget — Target phase noise levels

real vector

Target phase noise levels corresponding to the frequency offset points defined in FrOffset, specified as real vector in dBc/Hz. To compare PNTarget with PNMeasure, set PlotOption argument to on to view the phase noise comparison plot.

Data Types: double

Name-Value Pair Arguments

Specify optional pairs of arguments as Name1=Value1, . . . , NameN=ValueN, where Name is the argument name and Value is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Before R2021a, use commas to separate each name and value, and enclose Name in quotes.

Example: `PNMeasure = phaseNoiseMeasure(f1,p1,rbw,FrOffset,'on','Phase noise',PNTarget,'Type','Frequency','Histogram','on')`

Type — Type of input signal

'Frequency' (default) | 'Time'

Type of the input signal, specified as the comma-separated consisting of 'Type' and one of the following:

- 'Frequency' if the input signal is power spectrum data.
- 'Time' if the input signal is time domain data.

Histogram — Plot histogram of half-period information for time-domain signal

'off' (default) | 'on'

Plot the histogram of the half-period information for the time-domain signal, specified as the comma-separated pair consisting of 'Histogram' and 'off' or 'on'. You can plot the histogram only when 'Type' is specified as 'Time'.

Output Arguments

PNMeasure — Measured phase noise levels

real vector

Measured phase noise levels corresponding to frequency offset points defined in `FrOffset`, returned as a real vector in dBc/Hz. You can compare `PNMeasure` with the target phase noise levels with `PNTarget` defined in the function.

Data Types: `double`

GenFrOffset — Frequency offset points used to plot phase noise profile

vector

Frequency offset points generated by the `phaseNoiseMeasure` function that are used to plot the phase noise profile, returned as a vector.

Data Types: `double`

GenPN — Phase noise values used to plot phase noise profile

vector

Phase noise values generated by the `phaseNoiseMeasure` function that are used to plot the phase noise profile, returned as a vector. Each element in `GenPN` represents the phase noise at the corresponding frequency offset point represented in `GenFrOffset`.

Data Types: `double`

Version History

Introduced in R2020a

See Also

Phase Noise Measurement | VCO | Fractional N PLL with Accumulator | Fractional N PLL with Delta Sigma Modulator | Integer N PLL with Dual Modulus Prescaler | Integer N PLL with Single Modulus Prescaler | VCO Testbench | PLL Testbench

phaseNoiseToJitter

Measure RMS phase jitter from phase noise data

Syntax

```
[Jrms_rad,Jrms_deg] = phaseNoiseToJitter(PNFreq,PNPow)
[Jrms_rad,Jrms_deg, Jrms_s] = phaseNoiseToJitter( ____, 'Frequency', frequency)
```

Description

[Jrms_rad,Jrms_deg] = phaseNoiseToJitter(PNFreq,PNPow) returns the effective RMS phase jitter in radians and degrees from the phase noise frequency and power levels.

[Jrms_rad,Jrms_deg, Jrms_s] = phaseNoiseToJitter(____, 'Frequency', frequency) also returns the effective RMS phase jitter in seconds when you specify the signal frequency in addition to the input arguments in the previous syntax.

Examples

Measure Effective RMS Phase Jitter from Phase Noise Profile

Use a signal of 100 MHz frequency. The phase noise profile is:

- -125 dBc/Hz at 100 Hz
- -150 dBc/Hz at 1 kHz
- -174 dBc/Hz at 10 kHz
- -174 dBc/Hz at 200 MHz

Calculate the effective RMS phase jitter in radian, degree and second.

```
PNFreq = [100,1e3,1e4,200e6];
PNPow = [-125,-150,-174,-174];
[Jrms_rad Jrms_deg Jrms_sec]=phaseNoiseToJitter(PNFreq,PNPow, 'Frequency',100e6)

Jrms_rad = 4.0430e-05
Jrms_deg = 0.0023
Jrms_sec = 6.4346e-14
```

Input Arguments

PNFreq — Frequency points relative to fundamental frequency at which phase noise is calculated

real-valued vector

Frequency points relative to the fundamental frequency to which phase noise is calculated, specified as a real-valued vector in hertz.

Data Types: double

PNPow — Phase noise power at specified frequency offsets relative to fundamental frequency

real-valued vector

Phase noise power in 1-Hz bandwidth centered at the specified frequency offsets relative to the fundamental frequency, specified as a real-valued vector in dBc/Hz. The elements of `PNPow` correspond to the elements of `PNFreq`.

Data Types: double

frequency — Signal frequency

100e6 (default) | scalar

Signal frequency, specified as a scalar in hertz. Signal frequency is used to calculate the RMS phase jitter in seconds.

Data Types: double

Output Arguments

Jrms_rad — Effective RMS phase jitter in radians

scalar

Effective RMS phase jitter, returned as a scalar in radians.

Jrms_deg — Effective RMS phase jitter in degrees

scalar

Effective RMS phase jitter, returned as a scalar in degrees.

Jrms_s — Effective RMS phase jitter in seconds

scalar

Effective RMS phase jitter, returned as a scalar in seconds. To calculate `Jrms_s`, define the signal frequency using Name-Value pair arguments.

Version History

Introduced in R2020b

See Also

`phaseNoiseMeasure`

timeDomainSignal2DutyCycle

Measure duty cycle of time-domain signal

Syntax

```
DutyCycle = timeDomainSignal2DutyCycle(Tin, Vin)
```

Description

`DutyCycle = timeDomainSignal2DutyCycle(Tin, Vin)` returns the duty cycle of the time-domain input signal represented by `[Tin, Vin]`.

Examples

Duty Cycle of Time-Domain Signal

Determine the duty cycle of a time-domain signal.

```
load tin.mat;  
load xin.mat;  
dutyCycle = timeDomainSignal2DutyCycle(t,x)
```

```
dutyCycle = 4×1
```

```
    0.5000  
    0.5000  
    0.5000  
    0.5000
```

Input Arguments

Tin — Sample time points in signal

vector

Sample time points in the signal of interest, specified as a vector of the same length as `Vin`.

Data Types: double

Vin — Signal values at sample time points

vector

Signal values at the sample time points defined in `Tin`, specified as a vector of the same length as `Tin`.

Data Types: double

Output Arguments

DutyCycle — Ratio of positive pulse width to pulse period

vector

Ratio of the positive pulse width to pulse period, returned as a vector of the same length as the number of pulse periods present in the signal. Since pulse width cannot exceed pulse period, `DutyCycle` has a range of $(0, 1)$.

Version History

Introduced in R2020b

See Also

`timeDomainSignal2RiseTime` | `timeDomainSignal2FallTime` | `dutycycle`

timeDomainSignal2FallTime

Measure fall time of time domain signal

Syntax

```
Tf = timeDomainSignal2FallTime(Tin,Vin,RefLevel)
```

Description

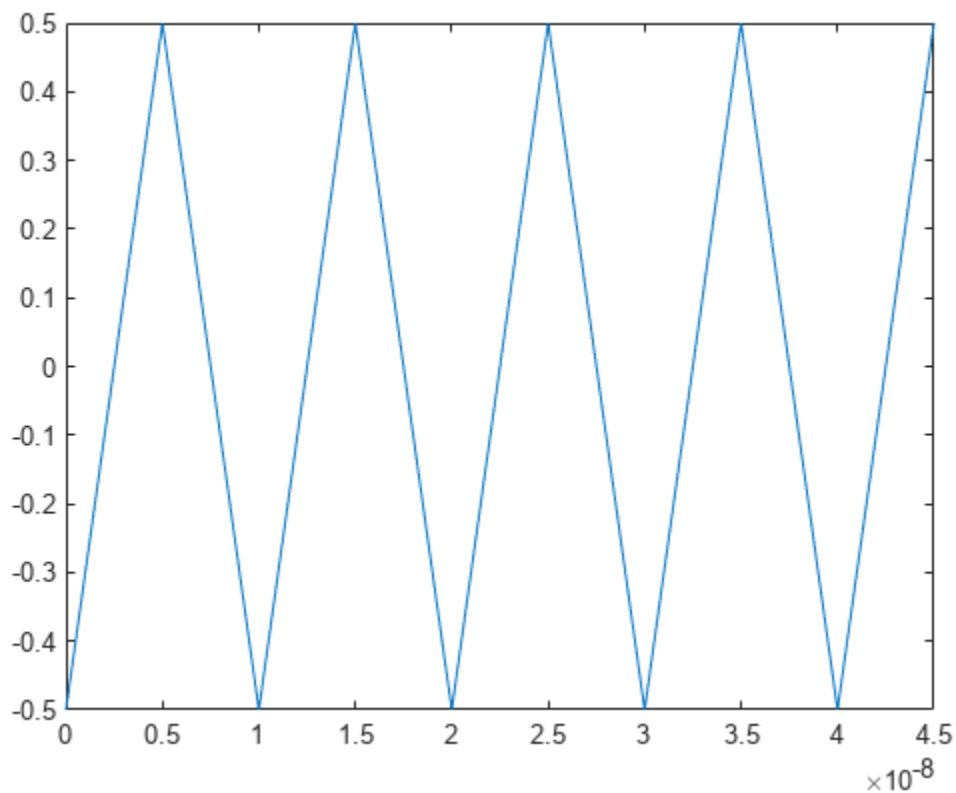
`Tf = timeDomainSignal2FallTime(Tin,Vin,RefLevel)` returns the fall time of the time domain input signal represented by `[Tin, Vin]`.

Examples

Fall Time of Time Domain Signal

Determine the fall time of a time domain signal.

```
load tin.mat;  
load xin.mat;  
plot (t,x)
```



```
RefLevel = [20 80];  
Tf = timeDomainSignal2FallTime(t,x,RefLevel)
```

```
Tf = 4×1  
10-8 ×  
  
    0.2970  
    0.2970  
    0.2970  
    0.2970
```

Input Arguments

Tin — Sample time points in signal

vector

Sample time points in the signal of interest, specified as a vector of the same length as **Vin**.

Data Types: `double`

Vin — Signal values at sample time points

vector

Signal values at the sample time points defined in **Tin**, specified as a vector of the same length as **Tin**.

Data Types: `double`

RefLevel — Reference levels as percentage of waveform amplitude

2-element positive row vector

Reference levels to calculate fall time as a percentage of waveform amplitude, specified as a 2-element row vector of positive values. The reference levels must be in increasing order.

Data Types: `double`

Output Arguments

Tf — Fall time of input signal

vector

Fall time of the input signal, returned as a vector of the same length as the number of falling edges in the signal.

Version History

Introduced in R2020b

See Also

`timeDomainSignal2RiseTime` | `timeDomainSignal2DutyCycle` | `falltime`

timeDomainSignal2RiseTime

Measure rise time of time domain signal

Syntax

```
Tr = timeDomainSignal2RiseTime(Tin,Vin,RefLevel)
```

Description

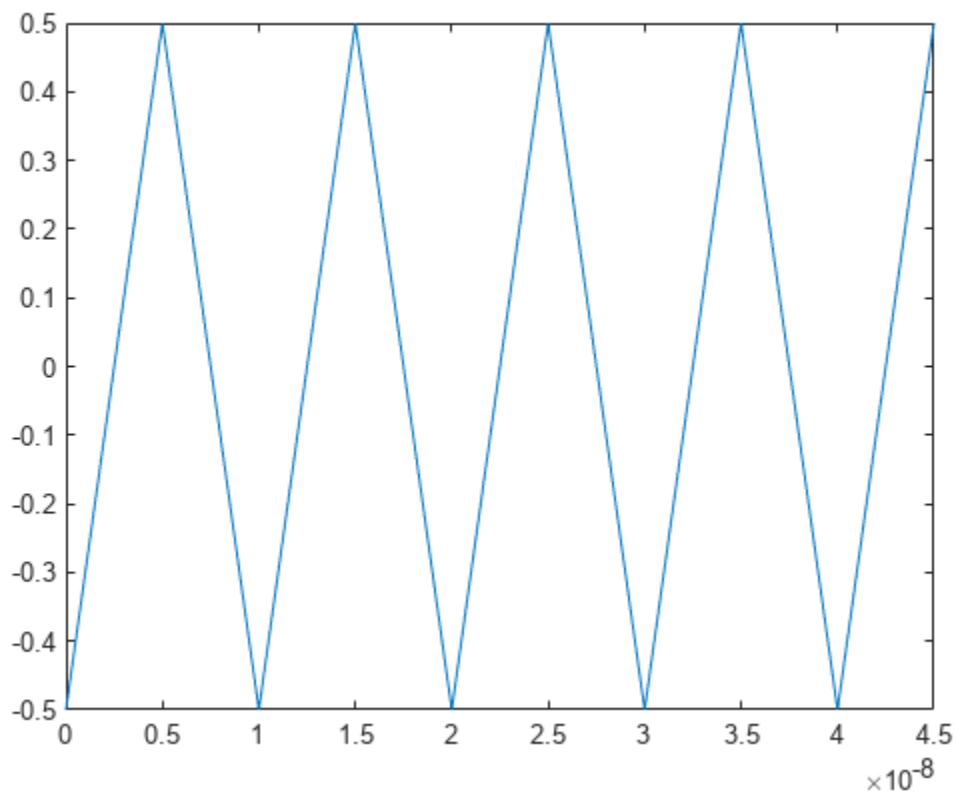
`Tr = timeDomainSignal2RiseTime(Tin,Vin,RefLevel)` returns the rise time of the time-domain input signal represented by `[Tin, Vin]`.

Examples

Rise Time of Time-Domain Signal

Determine the rise time of a time-domain signal.

```
load tin.mat;  
load xin.mat;  
plot(t,x)
```



```
RefLevel = [20 80];  
Tr = timeDomainSignal2RiseTime(t,x,RefLevel)
```

```
Tr = 5×1  
10-8 ×
```

```
0.2970  
0.2970  
0.2970  
0.2970  
0.2970
```

Input Arguments

Tin — Sample time points in signal

vector

Sample time points in the signal of interest, specified as a vector of the same length as **Vin**.

Data Types: `double`

Vin — Signal values at sample time points

vector

Signal values at the sample time points defined in **Tin**, specified as a vector of the same length as **Tin**.

Data Types: `double`

RefLevel — Reference levels as percentage of waveform amplitude

2-element positive row vector

Reference levels to calculate rise time as a percentage of waveform amplitude, specified as a 2-element row vector of positive values. The reference levels must be in increasing order.

Data Types: `double`

Output Arguments

Tr — Rise time of input signal

vector

Rise time of the input signal, returned as a vector of the same length as the number of rising edges in the signal.

Version History

Introduced in R2020b

See Also

`timeDomainSignal2FallTime` | `timeDomainSignal2DutyCycle` | `risetime`

Functions: Utilities

lowpassResample

Convert signal from one sample time to another

Syntax

```
[vq,vdq] = lowpassResample(t,v,tq,config)
```

Description

`[vq,vdq] = lowpassResample(t,v,tq,config)` returns an interpolated vector of samples `vq` and their derivatives `vdq` from input sample times `t`, input sample values `v`, and output sample times `tq`. The interpolation process is defined by the parameters in `config`.

Examples

Resample Discrete Sine wave

Sample a sine wave at π samples per cycle.

```
t = (0:20)*2;  
v = sin(t);
```

Define the interpolation sample times.

```
tq = (0:400)*0.1;
```

Define the interpolation configuration.

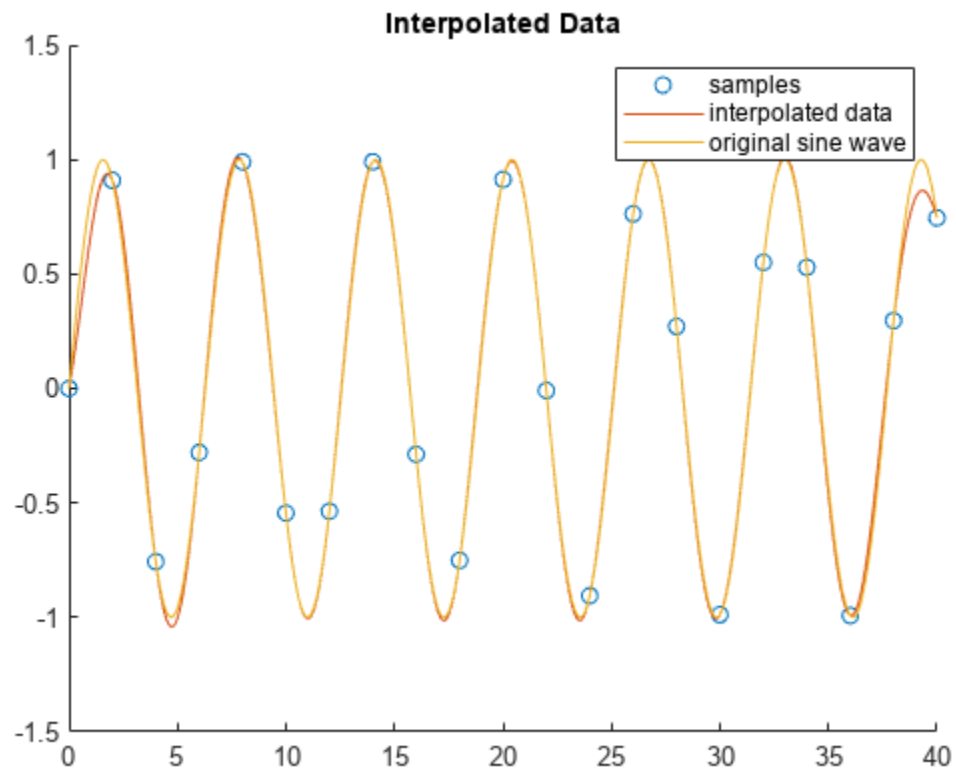
```
config.OutputRiseFall = 2; %Fixed step sample interval  
config.NDelay = 5;  
config.SampleMode = 'fixed';  
config.CausalMode = 'off';
```

Perform the interpolation.

```
[vq,vdq]=lowpassResample(t,v,tq,config);
```

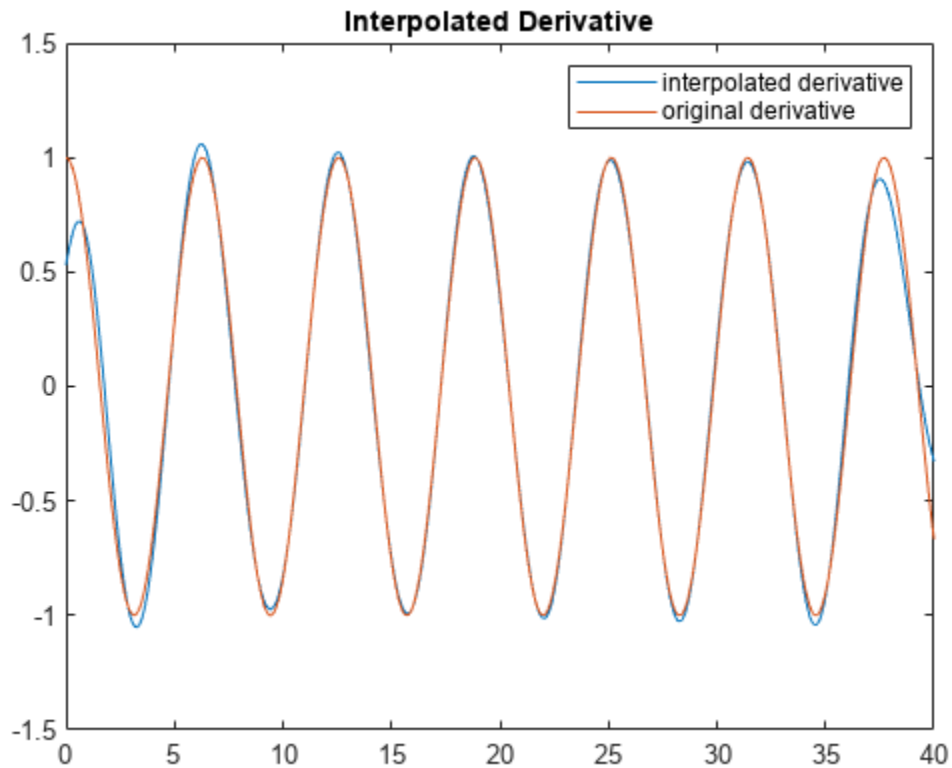
Scatter plot the samples, plot the interpolated data, and plot the original sine wave.

```
scatter(t,v);  
hold on;  
plot(tq,vq);  
plot(tq,sin(tq));  
hold off;  
title('Interpolated Data');  
legend('samples','interpolated data','original sine wave');
```



Plot the interpolated derivative and the original derivative.

```
plot(tq,vdq);  
hold on;  
plot(tq,cos(tq));  
hold off;  
title('Interpolated Derivative');  
legend('interpolated derivative','original derivative');
```



Input Arguments

t — Input sample times

vector

Input sample times, specified as a fixed-step or variable-step vector.

Data Types: `single` | `double` | `int8` | `int16` | `int32` | `uint8` | `uint16` | `uint32` | `logical`

v — Input sample values

vector

Input sample values corresponding to the input sample times defined in **t**, specified as a vector.

Data Types: `single` | `double`

tq — Output sample times

vector

Output sample times, specified as a fixed-step or variable-step vector.

Data Types: `single` | `double`

config — Interpolation parameters

structure

Interpolation parameters, specified as a structure with fields.

Field	Description	Value	Default
OutputRiseFall	The 0%-100% rise/fall time of the interpolated output.	Positive real scalar	1e-10
NDelay	Number of rise/fall times by which the interpolated output will be delayed with respect to the input.	Positive real integer	1
SampleMode	Input sampling mode, either fixed-step discrete time or variable-step discrete time.	fixed, variable	variable
CausalMode	Determine whether you want the interpolation process to introduce delay. Select <code>CausalMode</code> to introduce enough delay between input and output samples in the interpolation process so that the process is strictly causal.	off, on	off

Data Types: struct

Output Arguments

vq — Interpolated samples

vector

Interpolated samples, returned as a vector.

Data Types: single | double

vdq — Derivative of interpolated samples

vector

Derivative of interpolated samples corresponding to `vq`, returned as a vector.

Data Types: single | double

More About

Delay in Interpolated Output

You can define the number of rise/fall times by which the interpolated output will be delayed with respect to the input using the `config.NDelay` parameter.

The default value of `config.NDelay` of 1 produces an interpolation that has no ringing due to the Gibbs phenomenon and also has modest rejected out of band numerical artifacts.

Setting `config.NDelay` to 5 introduces enough anti-aliasing filtering to satisfy most applications, but introduces ringing due to the Gibbs phenomenon.

Setting `config.NDelay` to 10 introduces enough anti-aliasing filtering to satisfy demanding applications, but at the cost of additional delay and computation.

Setting `config.NDelay` to greater than 10 is supported, but is not required normally.

Input Sampling Mode

You can define the input sampling mode using the `config.SampleMode` parameter.

The default value of `config.SampleMode` of `variable` assumes that the input values are the result of a zero order hold process. In this case, the signal value is always equal to the value of the most recent sample. This choice is appropriate for saturated signals for which the transition times are the most important consideration.

Setting `config.SampleMode` to `fixed` assumes that the input values are instantaneous samples of a mathematically continuous signal at uniformly spaced sample times. Use this option for signals that are subject to a significant amount of analog filtering.

Causality of Interpolation Process

You can define the interpolation process as causal or not using the `config.SampleMode` parameter.

The default value of `config.CausalMode` of `off` aligns the time scale of the interpolation with the time scale of the input. This is appropriate when all necessary input values are available in a single vector.

Setting `config.CausalMode` to `on` introduces enough delay so that the sample values are available before performing an interpolation. The interpolated sample times are therefore always delayed by a constant value with respect to the input sample times. The required additional delay in this mode is `config.OutputRiseFall` times `config.NDelay`. This behavior mimics the behavior of the Lowpass Resampler block.

Version History

Introduced in R2021a

Extended Capabilities

C/C++ Code Generation

Generate C and C++ code using MATLAB® Coder™.

See Also

Logic Decision | Variable Pulse Delay | Lowpass Resampler

adeinfo2msa

Extract simulation data from Cadence to Mixed Signal Analyzer app

Syntax

```
adeinfo2msa
```

Description

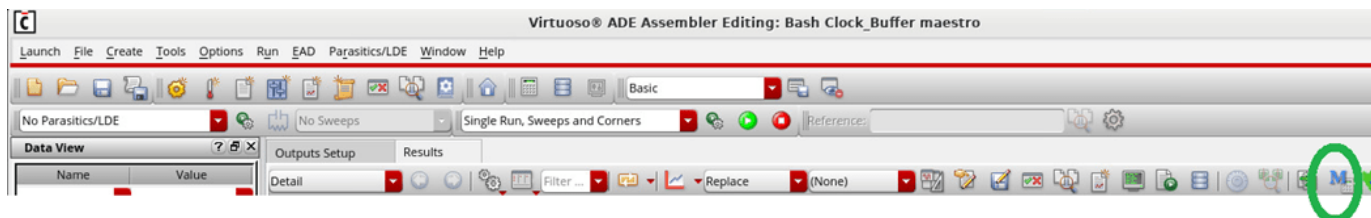
adeinfo2msa extracts simulation data from Cadence ADE simulation runs to analyze them using the **Mixed Signal Analyzer** app.

You can use the name-value pair arguments to specify which simulation runs and tests to import from the Cadence ADE Assembler/Maestro. If no name-value arguments are defined, the function by default creates a .mat file where only metrics data from the latest Cadence simulation run is saved. The function then launches the **Mixed Signal Analyzer** app and loads the generated .mat file.

Examples

Export Cadence® Simulation Data to MAT File

You can export the Cadence® simulation run results to a .mat file after an interactive run result is created. The .mat file can then be imported to the **Mixed-Signal Analyzer** app. To export the data, launch MATLAB® from Cadence ADE by clicking the **M** button in the toolbar.



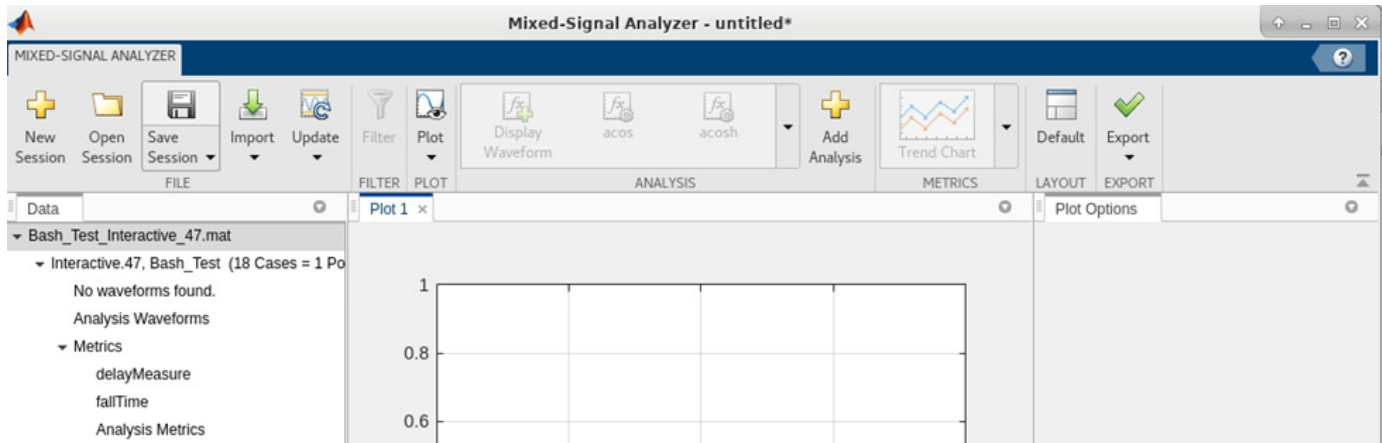
The MATLAB session that opens should have adeInfo in the base workspace.



Run the function `adeinfo2msa`. Since no name-value arguments are specified, the function creates a .mat file with only the metrics data from the latest Cadence simulation run. The .mat file is saved in the present working directory.

```
>> adeinfo2msa
Extracting simulation data...
mat file (Bash_Test_Interactive.125_metricsOnly.mat) created with only metrics data extracted
```

Once the .mat file is generated, the app automatically launches with the imported .mat file.



You can also specify the simulation run type, run name, and test name. Use the name-value pair argument `import2msa` to determine whether to launch the app with the generated .mat file or just to save the .mat file in the present working directory. Use the name-value pair argument `metricsOnly` to import only the simulation metrics data for faster performance.

Input Arguments

Name-Value Pair Arguments

Specify optional pairs of arguments as `Name1=Value1, ..., NameN=ValueN`, where `Name` is the argument name and `Value` is the corresponding value. Name-value arguments must appear after other arguments, but the order of the pairs does not matter.

Before R2021a, use commas to separate each name and value, and enclose Name in quotes.

Example: `adeinfo2msa(runName='Interactive.70',import2msa=false)` imports the simulation data of the Cadence interactive run number 70, but does not launch the **Mixed-Signal Analyzer** app.

import2msa — Launch Mixed-Signal Analyzer app after importing simulation data

`true` (default) | `false`

Determine whether to launch the **Mixed-Signal Analyzer** app after importing simulation data from Cadence. Setting it to `true` launches the app as well as creating the MAT file and saving it to the present working directory. Setting it to `false` only creates the MAT file and saves it to the present working directory, but does not launch the app.

metricsOnly — Only import metrics from simulation data

`true` (default) | `false`

Determine whether to import only the metrics or everything from the Cadence simulation data. Setting it to `false` can import redundant signals and waveforms, making the process time consuming.

testName — Test name mentioned in the Cadence ADE Maestro view

array of strings

The test name as mentioned in Cadence ADE Maestro view. This is the test whose simulation data you want to import. You can import multiple tests using a cell array of strings.

runName — Cadence simulation run name

array of strings

The Cadence simulation run name. You can find it in the Cadence interactive run results history as Interactive.<runNumber>. You can import data for multiple runs using a cell array of strings.

runType — Specify type of Cadence

Interactive (default) | Ocean

Specify whether processing Interactive or Ocean simulation run results from Cadence.

fileName — Specify name of MAT file where simulation results is to be stored

array of strings

Specify the name of the MAT file where the extracted Cadence simulation data is to be stored.

More About

Generated Simulation Data

The `adeinfo2msa` generates a `.mat` file to save the data from the Cadence simulation run and saves it in the current working directory. The Cadence simulation run is pointed by an `adeInfo` object in the MATLAB base workspace.

The simulation data available from the generated `.mat` file are:

Simulation Data	Description
<code>dbTables</code>	Table containing results of ADE simulation.
<code>totalCorners</code>	Structure of tables containing corner parameter definitions from ADE.
<code>paramTable</code>	Structure of tables containing parameter definitions from ADE.
<code>paramConditionTable</code>	Structure of tables containing parameter conditions from ADE.
<code>wfResults</code>	Index entries of all results that have waveform.
<code>wfCorners</code>	Structure of corner labels that contain the saved waveforms.
<code>wfOutput</code>	Structure of containing net names for which waveforms exist.
<code>WaveformDB</code>	Structure of waveform objects from ADE run number.
<code>simDBS</code>	Structure of simulation database containing single data point.

Version History

Introduced in R2022b

See Also

Mixed Signal Analyzer

