

Low Latency Measurement

Precision Signal Chain

LTspice® Simulation Guide



Introduction

This guide offers a walkthrough on simulating the low latency measurement signal chain from the precision narrow bandwidth signal chain platform using the accompanying LTspice schematic (.asc) file. AC, transient and noise simulations are covered. A basic level of familiarity with LTspice is assumed.

LTspice® is a high performance SPICE simulation software, schematic capture and waveform viewer with enhancements and models for easing the simulation of analog circuits. Included in the download of LTspice are macromodels for a majority of Analog Devices switching regulators, amplifiers, as well as a library of devices for general circuit simulation. It can be downloaded at analog.com/ltspice.

The included file, PNB0001 Adaptable Measurement Low Latency.asc, is a simulation of a signal chain and can be opened with LTspice. For tutorials and more information about LTspice visit www.analog.com/ltspice.

Signal chain presentation

We can see a block diagram of the signal chain in Figure 1

This simulation is an example of how to design a signal chain with low latency from a sensor through an ADC. It has input protection from the ADG5421F, and variable gain from the LTC6373 an amplifier with programmable gain. The amplifier is followed by a filter and then sampled

by the AD4630-24 a 24 bit, 2Msps ADC. The LTC6655LN-5.0 is used as a reference.

Component models

LTspice models for all electronic components in this signal chain are included in LTspice's library, so no external files are needed. Simulation models for components might not always cover the full behavior of the real component. Therefore, it is important to be aware of what is included in the models so that we can properly judge results and be aware of the limits of our simulations.

For this signal chain, here's a list of the included components and their limitations:

- ▶ ADG5421F.
 - Modelled: Input protection, switchable inputs
- ▶ LTC6373.
 - Modelled: Noise, distortion, digital gain selection
- ▶ LTC6655LN-5.0
 - Modelled: Noise
- ▶ AD4630-24
 - Modelled: Noise, distortion
 - Not Modelled: Latency through AD4630-24, digital code output, reference voltage range

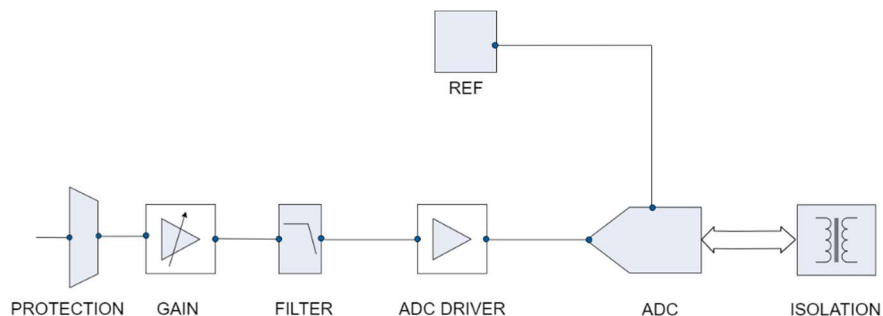


Figure 1. – Signal chain block diagram

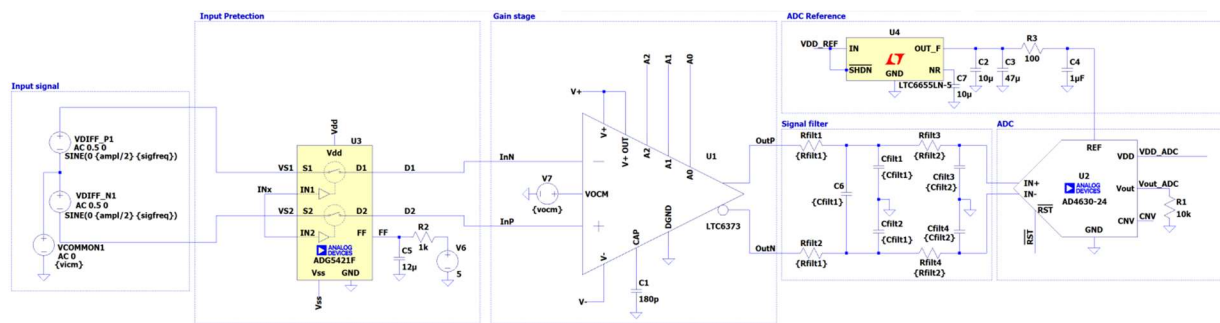


Figure 2 Overall LTspice simulation circuit

Overview

In Figure 2 we can see a screenshot of the simulated signal chain in LTspice.

Simulation file usage

The accompanying LTspice simulation file is divided in boxed sections for readability. Some of these sections are:

Simulation commands

You will find LTspice simulation commands for three kinds of simulation in this block, namely, transient, AC, and noise. Note that LTspice requires that only one simulation command exists in the schematic, so the most convenient use of this block is to comment-out all commands except for the one we intend to run. Commenting a command out can be quickly done by right clicking on the text box containing

it, pressing escape to dismiss the simulation dialog, and then, in the appearing, choosing “Comment” on the “How to netlist this text” control.

Parameters

Most variable elements in this signal chain can be controlled by modifying the parameters defined in this section.

Parameters available in this section are:

Input signal: Amplitude, Frequency and Common mode voltage

Signal filter: Resistors and capacitors that define the signal filter

ADC: Change the sample frequency of the AD4630-24

PGA: Set the gain and common mode voltage of the LTC6373

Supplies: Control the supply voltages of the LTC6737, AD4630, LTC6655LN-5.0, and ADG5412F

Transient simulations

Transient simulation allows us to observe the signals in our circuit in the time domain; it is done through the .TRAN command.

Transient simulation example: verifying circuit gain

In this example, we will use a transient simulation to check that the gain set in the simulation parameters for the amplifier stage is applied properly to an input signal.

We start out by commenting out all simulation commands except for the .TRAN one:

```
Simulation commands
.tran 25m uic                --> Transient Simulation. Default Stop time 25ms
;AC dec 100 1 1MEG          --> AC Simulation. Default range 1Hz to 1MHz
;noise V(Vout_ADC) VDIFF_P1 dec 100 1 2MEG --> Noise Simulation. Default output Vout_ADC
```

Figure 3 Transient simulation enabled

The command requires time length to be simulated, while other parameters are optional.

Use the parameter PGA_gain in the simulation to set the desired gain for the LTC6373.

The convert signal for the ADC is delayed by 20ms to allow for the reference to settle.

The latency through the AD4630-24 is not modeled in the model for the AD4630-24. The Vout pin of the ADC updates at the rising edge of the CNV pulse.

The physical AD4630-24 presents its digital output data 282ns after the rising edge of the CNV pulse.

The 282ns period is the typical ADC conversion time. Please consult the data sheet for further details.

The Vout pin of the ADC is automatically scaled with respect to the differential analog input with a scale factor of 1V/V. This is very convenient for the AC and Noise analysis. For transient analysis, you should keep in mind that this automatic rescaling means that Vout will match the differential analog input with the 1V/V factor, regardless of the voltage at the REF input pin. During transient simulation, the voltage at the REF pin defines the full-scale limits of the ADC as specified in the data sheet from $-1/128 \times V_{REF}$ to $129/128 \times V_{REF}$. Any voltages presented in transient simulation to the input outside this range will limit the Vout pin to the positive or negative full-scale values.

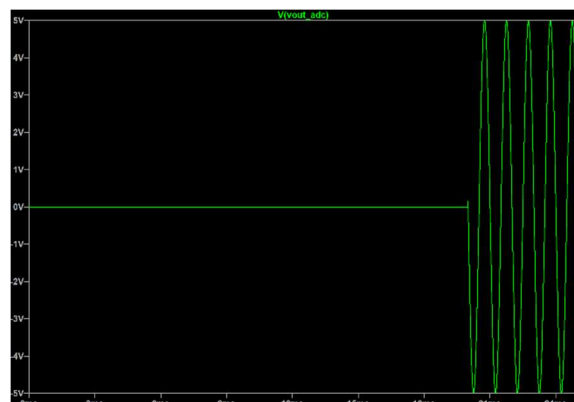


Figure 4: Transient simulation ADC output expressed in voltage

AC simulations

The AC simulation generates the circuit response over the frequency domain. In this case, we must comment the transient command and mark the ac as a Spice command. Both commands are in the same box on the top right corner.

LTspice Simulation Guide

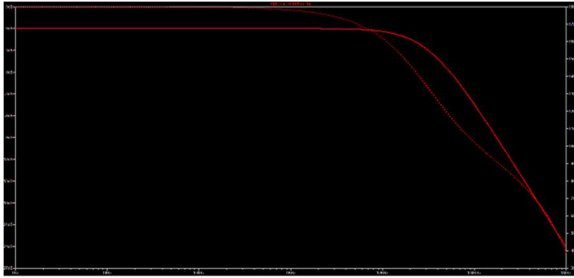


Figure 5: AC response at the input of the AD4630-24

AC simulation example: gain stage and signal filter bandwidths

In this example, we will use an AC simulation to check the resulting bandwidths of the amplifier stage and the signal filter stage.

Commence by enabling the AC simulation command only:

```
Simulation commands
;tran 25m uic --> Transient Simulation. Default Stop time 25ms
.AC dec 100 1 1MEG --> AC Simulation. Default range 1Hz to 1MHz
.noise V(Vout_ADC) VDIFF_P1 dec 100 1 2MEG --> Noise Simulation. Default output Vout_ADC
```

Figure 6 AC simulation enabled

Noise simulations

A noise simulation performed through the .NOISE command allows to extract the noise spectral density of our circuit. It is concerned with random noise (thermal, flicker, shot) generated by the components that comprise the circuit, meaning it has nothing to do with any kind of coupled interference, out of band signal components, crosstalk, power supply harmonics and such. It is important to note that this kind of analysis is a particular case of small-signal AC analysis independent from TRAN and AC ones. That is, even though our noise analysis will reveal the existence of noise voltage in our circuit, those voltages can't be observed in the time domain through a TRAN simulation or have any effect on an .AC simulation.

Noise simulation example: total signal chain noise

In this example, we will use the NOISE simulation to check overall noise performance of the complete signal chain.

We will start by commenting out all simulation commands except for the .NOISE one:

```
Simulation commands
;tran 25m uic --> Transient Simulation. Default Stop time 25ms
;AC dec 100 1 1MEG --> AC Simulation. Default range 1Hz to 1MHz
.noise V(Vout_ADC) VDIFF_P1 dec 1000 1 2GHz --> Noise Simulation. Default output Vout_ADC
```

Figure 7. Noise simulation enabled

Note that the simulation command requires to specify what is the circuit node where we want to measure the noise (in this case the output of the ADC) as well as the input signal source. The result we get in the waveform viewer after running the simulation and left clicking on the output node is the following:

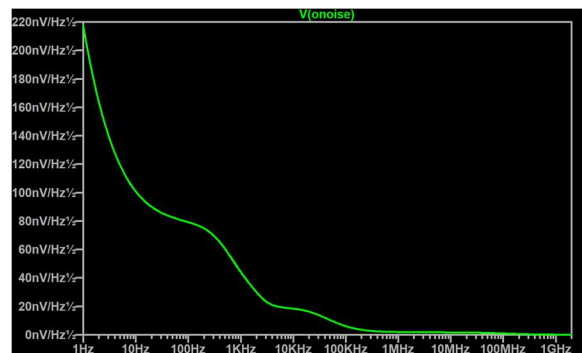


Figure 8: Noise versus frequency at the output of the AD4630-24

The waveform viewer can perform for us the integration of the displayed noise power spectral density by pressing CTRL+left click on the trace label:

V(noise)

Interval Start:	1000mHz
Interval End:	2GHz
Total RMS noise:	18.156μV

Figure 9. Displaying integrated RMS noise

As we can see, with the simulated configuration this signal chain will exhibit 18.156 μV_{rms} of noise at its output.

Note that the result is the integrated RMS noise of the displayed waveform. This means the result will change if modify the frequency interval over which the analysis is performed (in the simulation command), and it will also vary if we zoom-in on the x axis.

References

- [1] Analog Devices, "AD4630-24 Datasheet," [Online]. Available: <https://www.analog.com/media/en/technical-documentation/data-sheets/ad4630-24.pdf>.
- [2] Analog Devices, "LTC6373 Datasheet," [Online]. Available: <https://www.analog.com/media/en/technical-documentation/data-sheets/ltc6373.pdf>.
- [3] G. Alonso, "LTspice: Generating Triangular & Sawtooth Waveforms," [Online]. Available: <https://www.analog.com/en/technical-articles/ltspice-generating-triangular-sawtooth-waveforms.html>.
- [4] G. Alonso, "LTspice: Using the .STEP Command to Perform Repeated Analysis," [Online]. Available: <https://www.analog.com/en/technical-articles/ltspice-using-the-step-command-to-perform-repeated-analysis.html>.

LTSPICE® is a high-performance SPICE III simulator, schematic capture and waveform viewer with