KWIK CIRCUIT FAQ

SPICE Model for a Full Bridge Strain Gauge Sensor – by Tim Green

FAQ: Full Bridge Strain Gauge Sensor Spice Model

Introduction

This KWIK (Know-how With Integrated Knowledge) Circuit application note offers a step by step guide to address a specific design challenge. For a given set of application circuit requirements, it illustrates how these are addressed using generic formulae and makes them easily scalable to other similar application specifications. This Full Bridge Strain Gauge Sensor Model enables SPICE simulation of the electrical and physical properties of a bridge strain gauge. The SPICE model uses parameters which are taken from the strain gauge datasheet. Real-world Wheatstone bridge resistances are used in the model which translates physical strain into electrical voltages at the bridge output. Real-world strain is modeled as a voltage, Vforce.

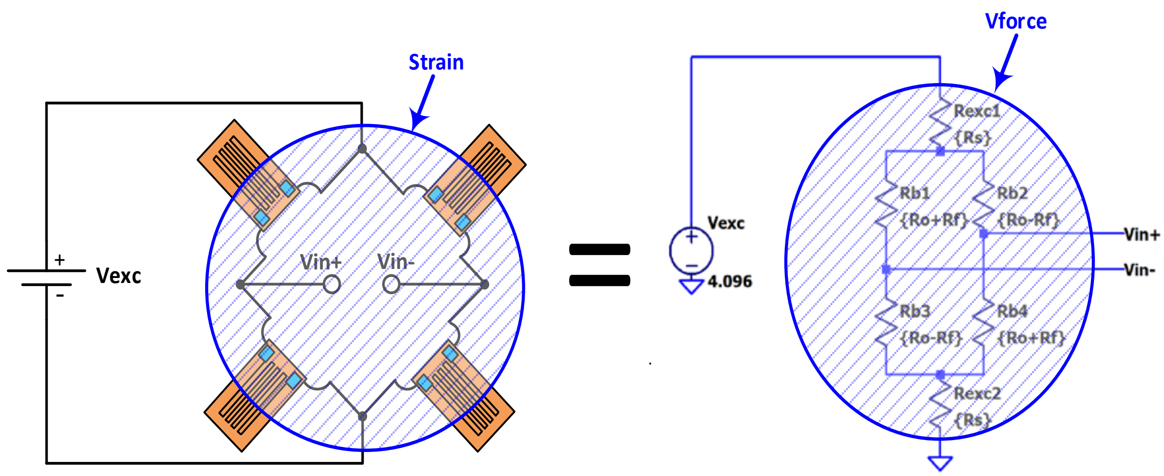

Figure 1. – Strain Gauge Bridge and Equivalent Electrical Model

Design Specifications Example

The key datasheet parameters used in the strain gauge model are shown in the Table 1. These map to the equivalent SPICE model.

Table 1. Datasheet specifications

Parameter	Datasheet value	Spice Model Symbol Value
Capacity	1 lb.	Vforce_max, 1lb = 1V
Rated Output (RO)	1.5mV/V nom	Vin_Span, 1.5m (V/V)
Excitation (VDC or VAC)	10V max	Vexc, 4.096V
Bridge Resistance	350Ω (nominal)	Rb1, Rb2, Rb3, Rb4 = Ro = 350 Ω
*Excitation Resistance	Not Applicable	Rexc1 = Rexc2 = 1fΩ (femto-ohms)

*Excitation Resistance reduces Vexc across bridge sensor for application specific scaling. Rexc1, Rexc2 = 1 fΩ implies a short-circuit

Design Description

This Strain Gauge Model (Figure 2) is simulated using LTSPICE but is also PSPICE compatible. The model enables users to simulate sensor loading of a reference excitation voltage and to connect signal conditioning circuitry to the bridge. This enables them to simulate all common mode, differential, and source impedance effects. The model assumes that all bridge components (Rb1, Rb2, Rb3, Rb4) change in response to applied strain. Only nominal sensor specifications are modeled.

Design Tips / Considerations

1. Load the excitation-circuit (shown as Vexc) by connecting it to the sensor model.
2. Connect the bridge sensor output to any signal conditioning circuitry being used for common mode, differential, full range, and accuracy simulations.
3. Use SPICE parameter stepping (. step param) with a DC Analysis (. op) to sweep from minimum to maximum force applied to the sensor model.

Design Procedure

1. Map the sensor model to the datasheet parameters listed in Table 1.

2. Run a DC Analysis SPICE simulation (using the sweep parameter) and confirm that Vin matches the expected output for the respective strain input. Note that Vin = (Vin+) – (Vin-)

3. Connect the sensor model to an excitation and signal conditioning circuit to simulate the complete application.

Design Simulations

Vin = (Vin+) – (Vin-),

Vin = (Vexc)*(Vin_Span) * (Vin_Force)

Where

Vexc = 4.096V,

Vin_Span = 1.5mV/V,

Vin_Force: 1V is equates to 1lb of physical force on the strain gauge.

A plot of these results is shown in Figure 3

Table 2. Design Goal v's Simulation

Strain	Vin_Force	(Vin+) – (Vin-) Ideal	(Vin+) – (Vin-) Simulation
0.1lb	0.1V	0.6144mV	0.61440468mV
0.5lb	0.5V	3.072mV	3.0720234mV
0.7lb	0.7V	4.3008mV	4.3008327mV
1.0lb	1.0V	6.144mV	6.1438084mV

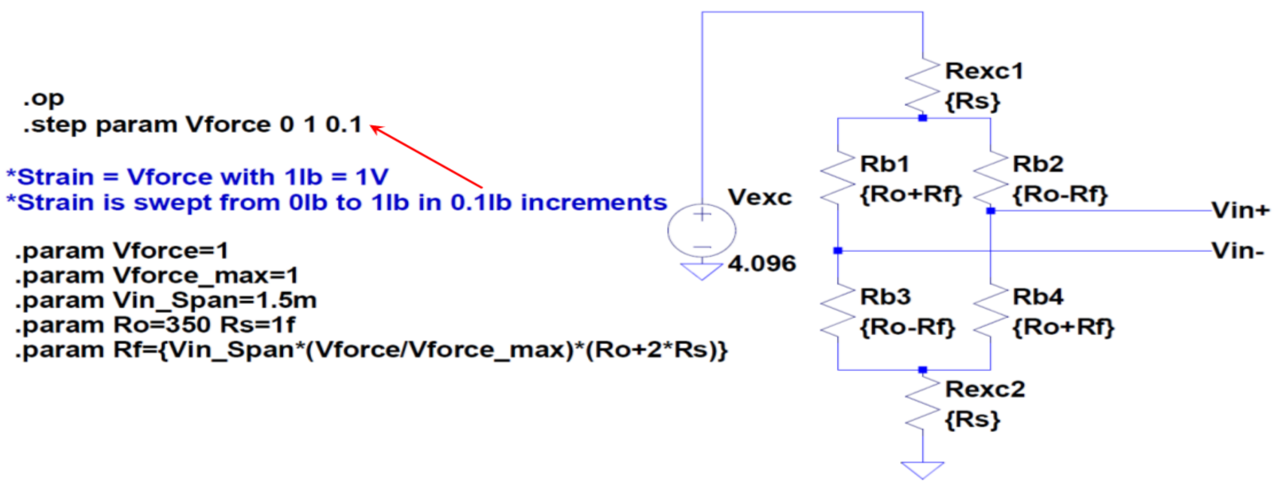

Figure 2. Schematic showing Strain Gauge Model and Simulation Parameters

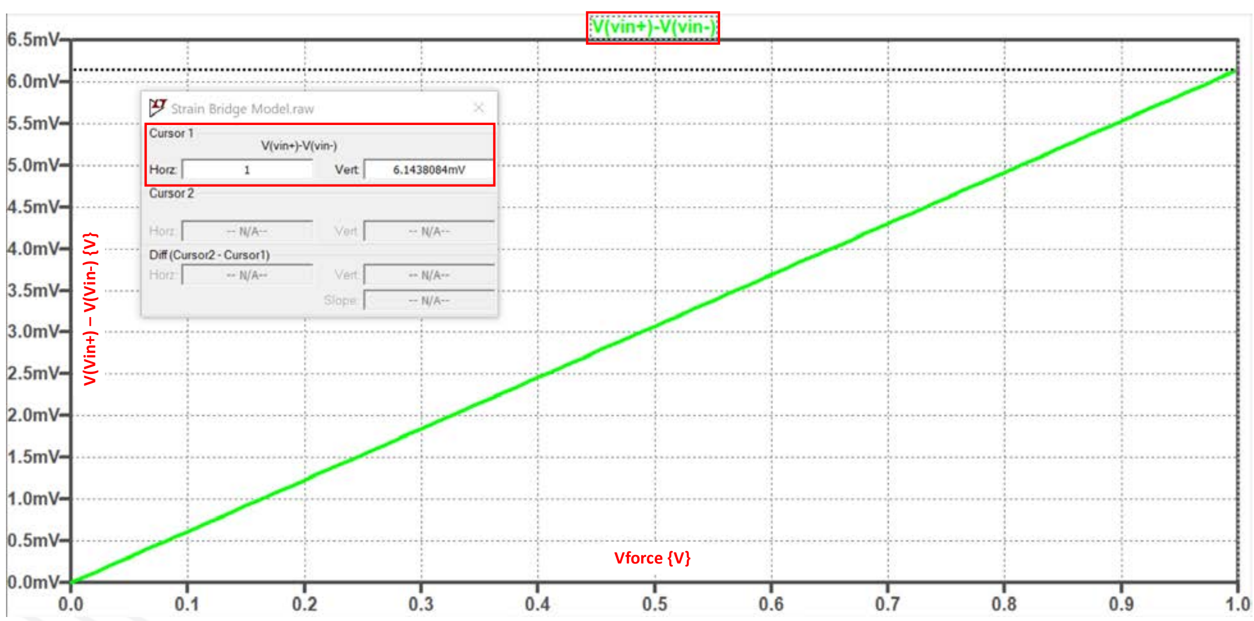

Figure 3. Plot of Simulated Force versus Input Voltage for Strain Gauge Model

Typical Circuit Application

A typical application circuit for the sensor model is shown in Figure 4. It is first necessary to amplify and offset the small differential input bridge voltage Vin (0V to 6.144mV) to be within the input range of the ADC for optimal scaling and resolution. The AD8237 single supply instrumentation amplifier is used for this purpose and it also rejects large common mode voltages of approximately half the reference voltage level(4.096V/2 = 2.048V). The Voffset_dac source used in the schematic represents a low impedance voltage output DAC which is used to adjust upwards the output voltage of the AD8237 offset to be 100mV for zero input strain. Voffset_dac also compensates for the input offset voltage of the AD8237 which is amplified by a factor of 781.25x at Vout. The LT1461-4.096 is a precision 4.096V series voltage reference which provides the required excitation current, Iexc, calculated using the following formula:

Iexc = Vref/Ro = 4.096V/350ohms = 11.7mA

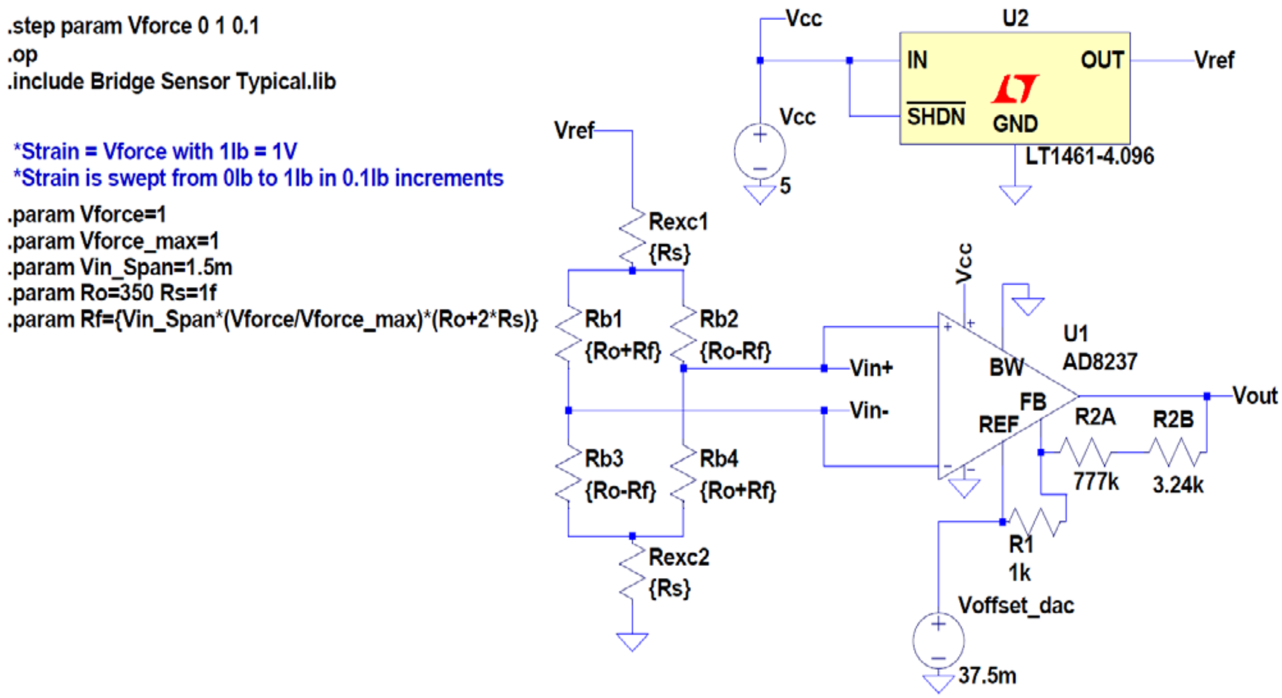

Figure 4. Conditioning the Output Voltage of the Sensor Model

Given the desired output voltage range and the input voltage range and span for the strain gauge, the gain and offset for the signal conditioning circuitry can be computed as follows:

Vout_max=4.096V, Vout_min=0.1V
Vforce_max=1V, Vforce_min=0V
Vin_span=1.5mV/V, Vin=(Vin+)-(V-)

$$Vin\_max = \frac{Vforce\_max}{Vforce\_max} * Vin\_span * Vref$$

$$Vin\_max = \frac{1V}{1V} * \frac{1.5mV}{V} * 4.096V = 6.144mV$$

$$Vin\_min = \frac{Vforce\_min}{Vforce\_max} * Vin\_span * Vref$$

$$Vin\_min = \frac{0}{1} * \frac{1.5mV}{V} * 4.096V = 0V$$

$$Vout\_max - Vout\_min = Gain * (Vin\_max - Vin\_min)$$

$$4.9V - 0.1V = Gain * (6.144mV - 0) \rightarrow Gain = 781.25$$

$$Vout = Gain * (Vin\_min) + Voffset\_dac$$

$$0.1V = 781.25(0V) + Voffset\_dac \rightarrow Voffset\_dac = 0.1V$$

Measurements on the model for the AD8237 showed it to have a positive input offset voltage of 80uV. When this is amplified by the calculated gain factor of 781.25, it results in an output voltage Vout = 62.5mV when Vin = 0V and Voffset_dac = 0V. To achieve ideal scaling where Vout = 100mV for Vin = 0V, Voffset_dac must be set to (100mV-62.5mV) = 37.5mV. Simulation results for this application circuit are shown in Figure 5 and Figure 6.

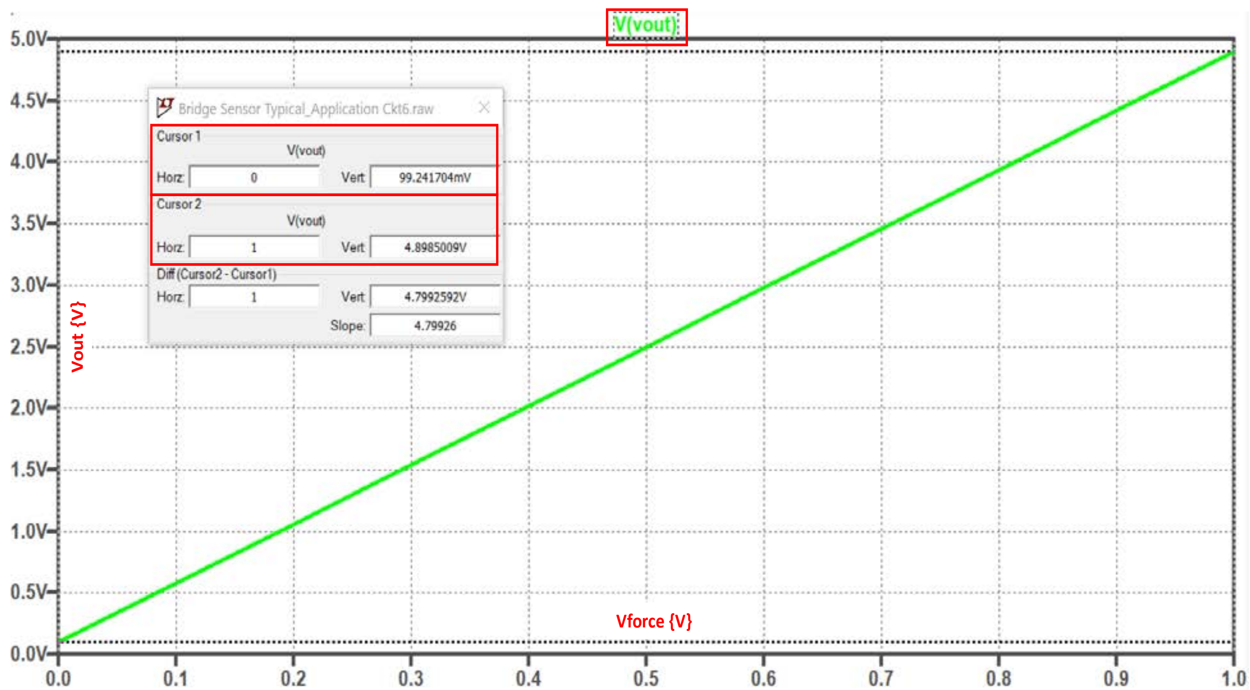

Figure 5. Plot of Simulated Output Voltage versus Input Force for Strain Gauge Application Circuit

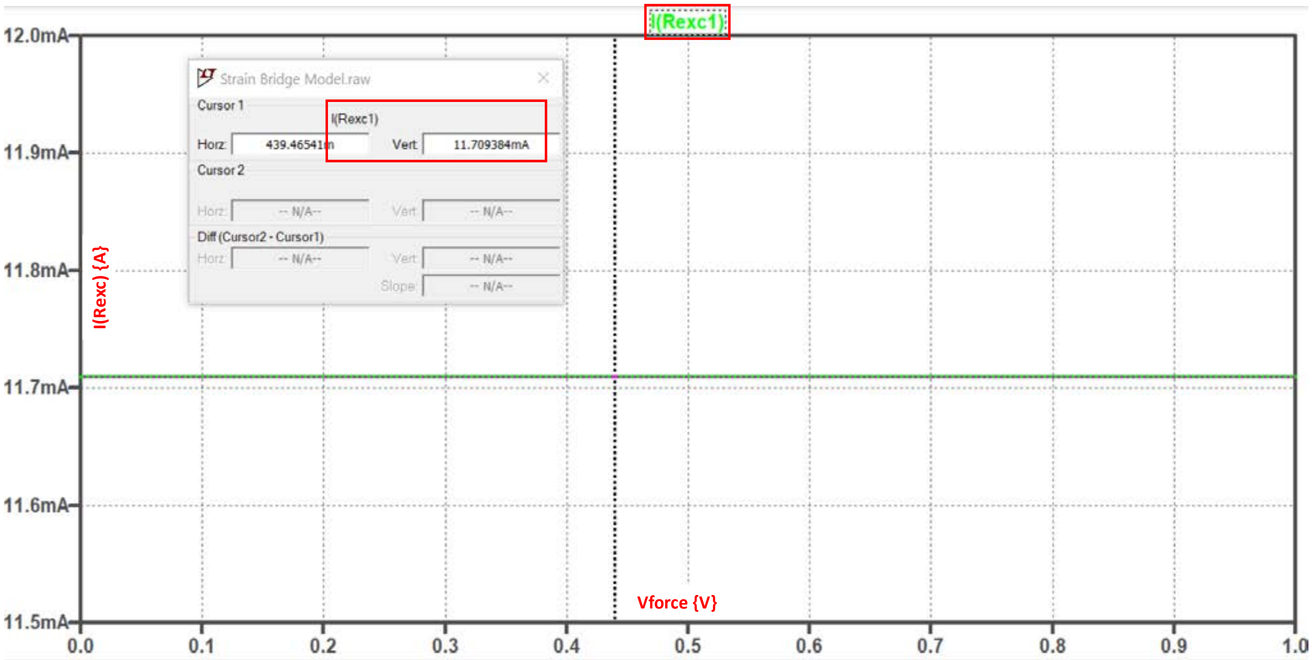

Figure 6. Plot of Simulated Excitation Current for Strain Gauge Application Circuit

Design Devices

Table 3. Series Voltage References

Part Number	Vout (V) typ	Initial Accuracy (%) max	Vout Tempco (ppm/V) max	Vnoise (Vp-p) typ	Iout Sourcing (A) max	Vs+ (V) min/max
LT1461ACS8-4	4.096	0.04	3	32u	50m	4.06/20

Table 4. Instrumentation Amplifiers

Part Number	Vos (V) max	Ibias (A) max	Gain (V/V) min/max	BW Low Gain (Hz) typ	Vnoise (V/rt-Hz) typ	Vs span (V) min/max
AD8237	75u	650p	1/1000	200k	68n	1.8/5.5

Table 5. Op Amps (for Reference & DAC Output Buffers, as needed)

Part Number	Vos (V) max	Ibias (A) max	GBP (Hz) typ	Vnoise (V/rt-Hz) typ	Iq/Amp (A) typ	Vs span (V) min/max
AD8538	13u	25p	430k	50n	180u	2.7/5.0

References

"Analog Bits - Modeling a Load Cell in SPICE"
Created by Alec Schmidt, last modified on Jun 20, 2017
Modeling a Load Cell in SPICE

"How to Stay Out of Deep Water when Designing with Bridge Sensors"
by Gustavo Castro and Scott Hunt
Analog-dialogue/how-to-stay-out-of-deep-water

"Bridge-Type Sensor Measurements are Enhanced by Autozeroed Instrumentation Amplifiers with Digitally Programmable Gain and Output Offset"
by Reza Moghimi
Analog-dialogue/bridge-type-sensor-measurements.html

"Practical Design Techniques for Sensor Signal Conditioning"
Edited by Walt Kester, Analog Devices, 1999, ISBN-0-916550-20-6
Education-library/practical-design-techniques-sensor-signal-conditioning.html

Instrumentation Amplifier Diamond Plot Tool
The Diamond Plot Tool is a web application that generates a configuration-specific Output Voltage Range vs. Input Common-Mode Voltage graph, also known as the Diamond Plot, for Analog Devices Instrumentation Amplifiers.

LTspice
LTspice® is a high-performance SPICE III simulator, schematic capture and waveform viewer with enhancements and models for easing the simulation of switching regulator, linear, and signal chain circuits