Platinum Thin Film Resistor SPICE Models Sweep Temperatures in Service of IC Electronic Simulation

by Alain Stas, Product Marketing Engineer, Non-Linear Resistors, Vishay Intertechnology Inc.

With the advent of IoT applications, increase in production of electric and hybrid electric vehicles (EV / HEV), and expanding industrial automation market, there is a growing need for accurate temperature sensing simulations.

Complex mechatronics problems need powerful computation software, creating a need for efficient electronic component models — including passive components.

Even though SPICE models for active components have been available for decades, it is clear that realistic models are still difficult to obtain from passive component manufacturers. So generic models are used, and the results, while qualitatively correct, are subject to optimization.

This article will describe a case of a general IC electronic design, including one central passive component temperature sensor, where a new precise model designed by the manufacturer brings added value to the application.

We will be talking here about the LTC2063 with a low 2 µA supply current, presented in 2017 by Analog Devices (Reference 1).

When this IC is used in a temperature sensor application, the schematics can be depicted as in Figure 1.

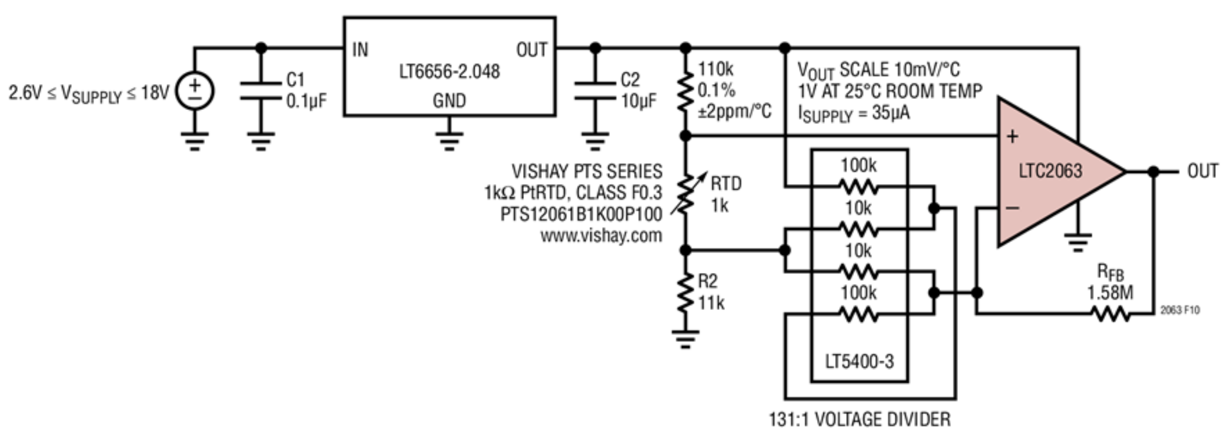

Figure 1

The input signal for temperature sensing is provided by a surface-mount platinum temperature sensor manufactured, among other passive elements, by Vishay (PTS1206 class 1B sensor in Reference 2). This kind of linear temperature sensor has become more and more popular in automotive applications since its qualification to AEC-Q200. For example, it presents a good alternative solution to surface-mount NTC devices when ultimate stability at high temperatures is requested. Linearity of electrical response of the PTS all along the temperature range is also a strong advantage when compared to NTCs. While more sensitive than RTDs, the NTCs are not capable of providing the same nearly ideal linearity between -40 °C and 85 °C, for example, even after some linearization work.

Of course, Analog Devices offers a very useful LTspice model (Reference 3) for this circuit, where the PTS sensor is presented as a variable resistor, as shown in Figure 2.

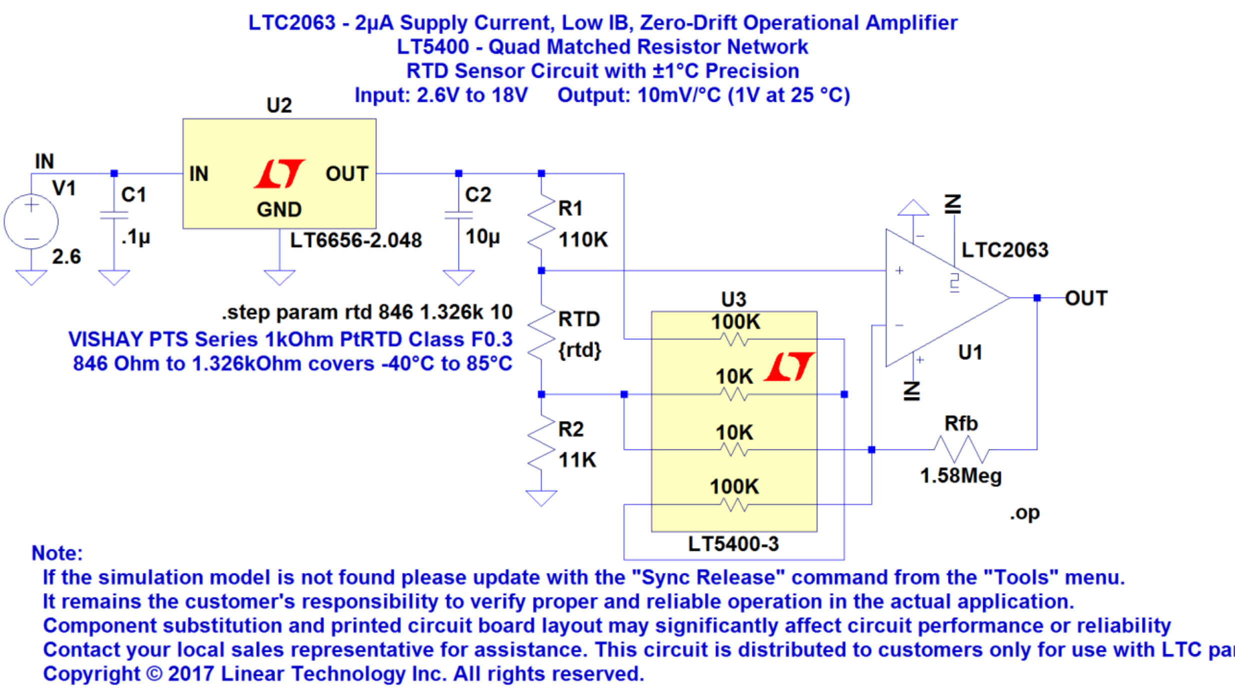

Figure 2

One detail will jump out to component engineers if they have some knowledge about simulations. While we deal here with a temperature measuring circuit with an announced ±1 °C overall temperature precision, the variable temperature (the global ambient temperature) does not appear anywhere in the SPICE directives or in the definition of the Vishay PTS.

So in this particular case I decided to explicitly introduce the DC temperature sweep SPICE model into the simulation to avoid making the user of the LTC2063 dig down in the PTS datasheet. That leads us to Figure 3, where we are now able to:

- Sweep the temperature variable

- Visualize the influence of the tolerances in temperature for the PTS

- Perform a fine tuning on the value of the feedback resistor

- Test the circuit with Monte Carlo tolerances on all passive elements (fixed resistors, PTS)

- Compute the effective accuracy of the LTC2063 output in °C (see Figure 4)

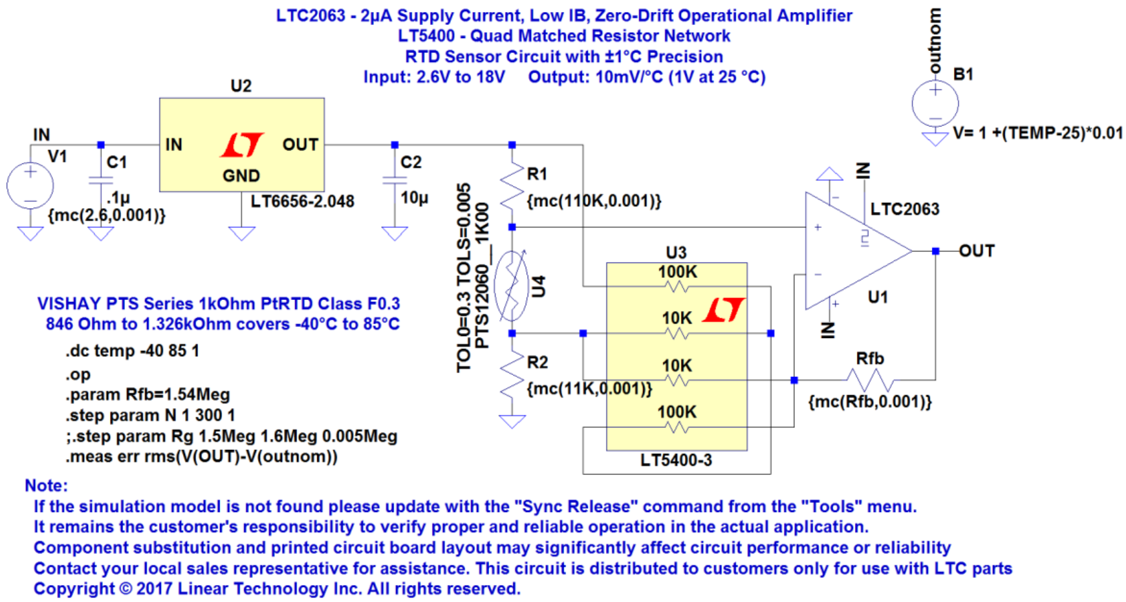

Figure 3

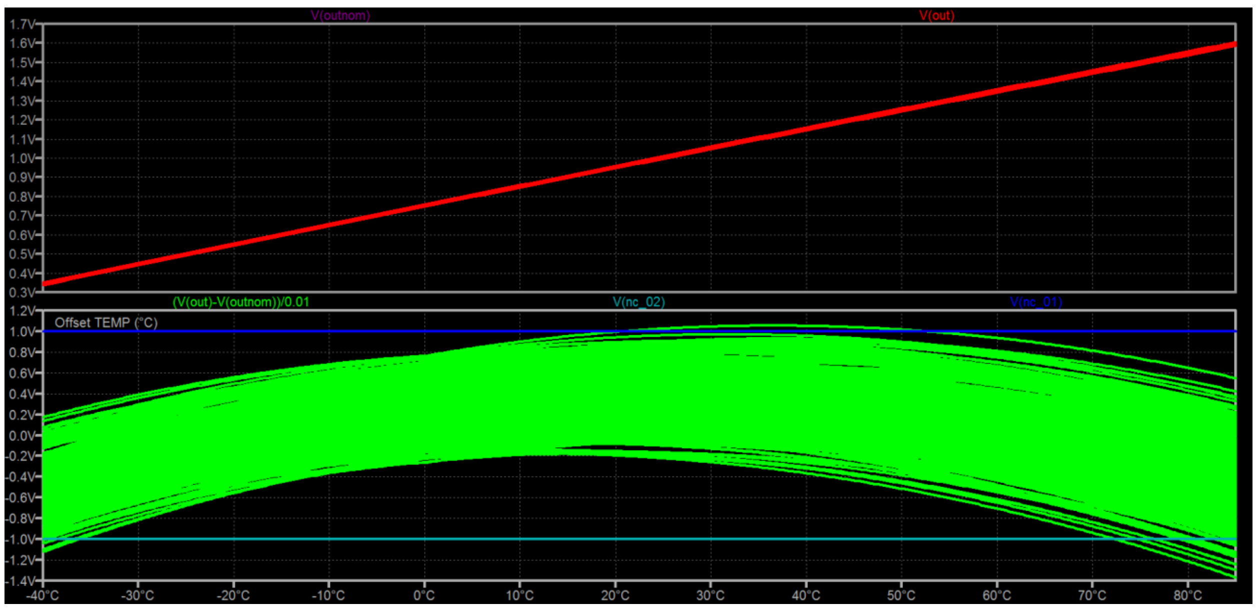

Figure 4

Higher pane: the linear output voltage of the LTC2063 as a function of the temperature.

Lower pane: the offset temperature readout linearity deviation from the output voltage (considering an output at 25 °C of 1 V and a sensitivity of 10 mV/°C).

The results of Figure 4 show that even with all the tolerances, this circuit provides a linear output as a function of temperature (higher pane) with an overall accuracy mainly within the ± 1 °C (lower pane) QED.

We could even go further in the analysis and introduce a dynamic variation of temperature over time for the PTS sensor, which would require another type of SPICE model for the sensor.

We could then demonstrate the very important effects on the application of the time delay in the time response of the sensor: this would lead us into the testing sensors in smaller case sizes like 0805, 0603, or even smaller.

In any case we have here a clear example where a SPICE model created directly by the sensor manufacturer provides a nice complement to the simulation of ICs themselves.

This example also shows that there are numerous possiblities for development in the temperature sensing simulation field. In anticipation of these exciting developments , the reader can always visualize the simulation described in this article at the following URL:.

http://www.vishay.com/videos/resistors/hands-on-electronic-simulation-of-an-optimized-linear-output-temperature-sensing-circuit.html

References:

1) web : https://www.analog.com/en/products/ltc2063.html

2) web: https://www.vishay.com/docs/28899/ptsat.pdf

3) web: https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html