

Developing A Precise Pt100 RTD Simulator For SPICE

by Thomas Kuehl, Senior Applications Engineer, Texas Instruments Incorporated

It is fairly common practice for engineers to pre-design and test their analog circuits using a circuit simulation program before an actual prototype is assembled. Although a variety of SPICE-based circuit simulators are available for purchase, several semiconductor companies are now providing their customers with a free, abbreviated version of the full-featured program. When compared, these simulation programs may provide fewer analysis options, fewer convenience features and sometimes node and device limitations. However, TINA-TI (TI's offering to designers) is powerful enough to handle almost any circuit simulation involving signal conditioning elements and sensors.

The resistance temperature detector (RTD) is a popular sensor commonly employed in temperature measurements. Its resistance changes in a near linear fashion, increasing with temperature. Although some RTDs are available for the full -200°C to $+850^{\circ}\text{C}$ operational range, more often they are specified for a narrower range such as -75°C to $+250^{\circ}\text{C}$, or -200°C to $+650^{\circ}\text{C}$. The particular range is dependent on the application. RTDs are resistive elements comprised of different metal and metal alloys such as platinum, copper, nickel, nickel-iron and molybdenum. Each type is intended for a particular temperature range, resistance and accuracy requirements. The element resistance range extends from about $25\ \Omega$ to $1\ \text{k}\Omega$, for commonly-available RTDs.

The Pt100, a $100\ \Omega$ platinum RTD, is popular because of its well-established behavior over temperature, wide temperature range, reasonable cost and overall availability. It can be obtained with a 0°C accuracy of $\pm 0.1^{\circ}$ or better, and is available in two-, three- and four-wire arrangements for Kelvin sense connections.

Although SPICE simulator libraries include a wide assortment of electronic components, RTD simulator models may not be there. Despite this, you can readily create one from other common circuit elements available within the library. Since the RTD is a precise resistor with well-established temperature coefficients, the simplest RTD model may be based upon the basic SPICE resistor model.

The SPICE resistor mathematical model has the form and takes the value equal to:

$$\langle \text{Value} \rangle \quad R \left(1 + TC1 \cdot (T - T_{nom}) + TC2 \cdot (T - T_{nom})^2 \right) \quad \text{Eq. 1}$$

where, R is the resistance multiplier, $TC1$ is the linear temperature coefficient $^{\circ}\text{C}^{-1}$, $TC2$ is the quadratic temperature coefficient $^{\circ}\text{C}^{-2}$

The highest coefficient in the equation is of the second degree. This is important to note because this limits how accurately the RTD response can be modeled with the SPICE resistor model. The precise RTD mathematical model has a quartic resistance response over temperature (IEC751 Standard, -200°C to $+850^{\circ}\text{C}$), which is based upon the Callendar-Van Duesen equation:

$$R(t) = R_{nom} \left(1 + a \cdot t + b \cdot t^2 + c \cdot t^3 (100 - t) \right) \quad \text{Eq.2}$$

For the Pt100:

$$R_{nom} = 100 \Omega$$

$$a = 3.90830 \times 10^{-3}$$

$$b = -5.77500 \times 10^{-7}$$

$$c = 0 \text{ for } 0^\circ\text{C} \leq t < 850^\circ\text{C}$$

$$c = -4.18301 \times 10^{-12} \text{ for } -200^\circ\text{C} < t < 0^\circ\text{C}$$

where, t is the temperature in $^\circ\text{C}$, R_{nom} is the RTD specified resistance (usually at 0°C). The standardized polynomial coefficients for the RTD are a , b and c . These coefficients may differ slightly depending on the standard referenced such as IEC751, DIN 43760, JIS C1604, etc.

When the temperature is 0°C , the product terms equate to zero and the nominal RTD resistance results.

If the RTD temperature range is limited to 0°C and above, then coefficient $c = 0$ and the equation simplifies to a second-degree polynomial:

$$R(t) = R_{nom} (1 + a \cdot t + b \cdot t^2) \quad \text{Eq. 3}$$

This fits nicely with Eq. 1. Then it is a simple matter of inserting the a and b coefficients in Eq. 3 to obtain the RTD resistance within a limited temperature range. Applying Eq. 3 to the Pt100 with the coefficients listed above results in:

$$R(t) = 100 [1 + 3.90830 \cdot 10^{-3} \cdot t + (-5.77500 \cdot 10^{-7} \cdot t^2)] \quad \text{Eq. 4}$$

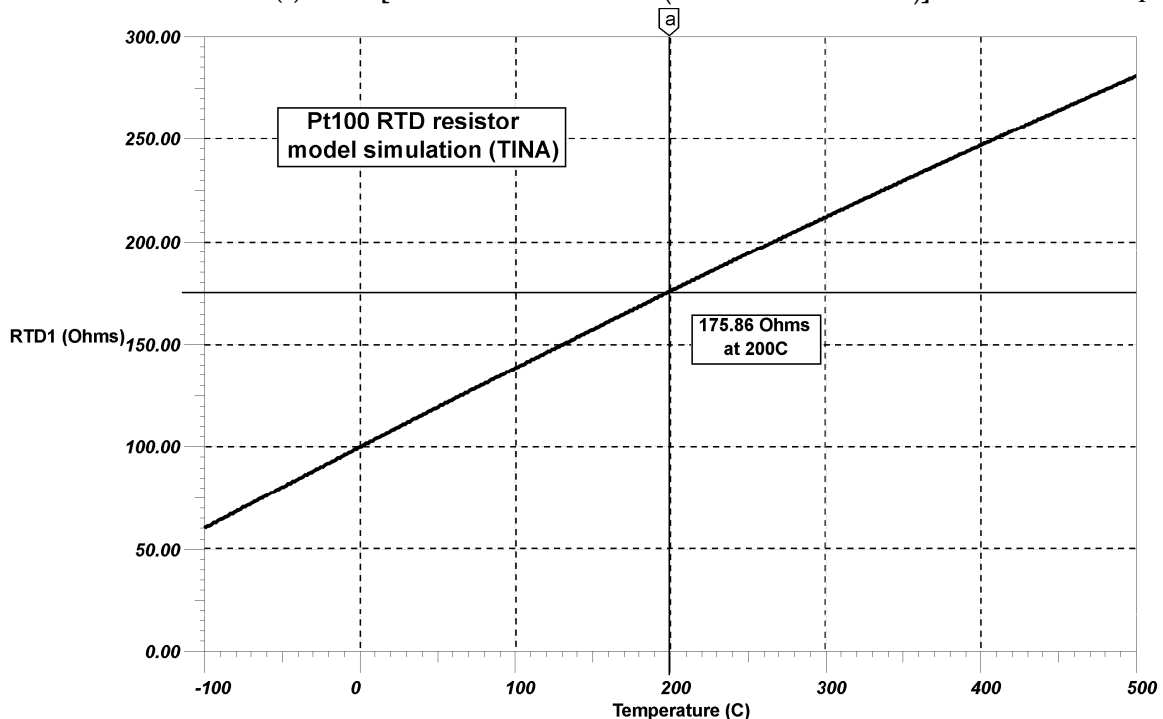


Fig. 1: Pt100 RTD Resistor Simulation Response

A Pt100 simulation, over temperature, was accomplished for the resistor model defined by Eq. 4. It will be referred to as RTD1. The response is shown in Fig. 1. This simple resistor-based RTD model is useful for applications where the simulation temperatures are restricted to zero degrees minimum, and the maximum temperature limits of the simulation software, or RTD itself.

If the resistor model is used for sub-zero temperatures, the RTD resistance exhibits an error of about $+1\ \Omega$ as a temperature of -200°C is approached. On the surface, this appears to be a small error. However, it is a large error relative to the ideal $18.508\ \Omega$ RTD resistance attained at -200°C . A somewhat more developed model with a bit more complexity is required to attain maximum accuracy at the coldest temperatures.

When unique circuit elements are needed for simulation purposes, a common practice is to develop a SPICE subcircuit, often referred to as a macromodel, often comprising common SPICE circuit elements such as passive devices, transistors and independent sources. It also may include controlled sources such as the voltage-controlled voltage source (VCVS) and voltage-controlled current-source (VCCS), and others. Combined, they can provide an element that closely simulates the electrical performance characteristics of a much more complex SPICE model. It often provides much faster simulation times and can be easily inserted into, and removed from, within the overall circuit. But before developing an RTD macromodel, discussion about the simulation program temperature characteristics is necessary.

When using a SPICE-based simulator for wide temperature, you need to understand the program's minimum and maximum simulation temperature range. For example, it is -100°C to $+500^\circ\text{C}$ for TINA. If the full RTD temperature range is of interest, then another approach to simulating the temperature range will be required.

Another point to consider is that the simulation temperature may be global and relative to all circuit elements. When that is the case a simulation made over a wide temperature expanse results not only in the RTD, but all components being subjected to the simulation temperatures. Passive and active components within TINA-TI default to this *relative* temperature mode, but many can be set to an *absolute* temperature as well. The absolute setting establishes the component at a *fixed* temperature and maintains their electrical characteristics at that temperature. Resistors, capacitors, diodes, transistors are among the components that may have their temperature specified as either relative or absolute.

Active circuits that rely on their own complex macromodel, such as op amps and instrumentation amplifiers, may not provide an absolute temperature fixing option. They are deliberately allowed to drift across temperature providing a means for assessing the circuit's dc and ac performances over the temperature span of interest. And although you may desire to drift just the RTD over temperature, the macromodel circuit will be drifted right along with it which may have not been the original intention.

Op amps and other macromodels are often designed to mimic the device performance over the temperature range specified in the product's data sheet. For example, macromodels for many TI op amps have a specified temperature range of -40°C to $+125^\circ\text{C}$. If the entire RTD circuit is drifted over TINA-TI maximum simulation range of -100°C to $+500^\circ\text{C}$, the op amp macromodel may produce unreliable electrical results once outside its specified temperature range. These responses may or may not accurately reflect the true performance at the extended temperatures. Even if they do, for physical or thermal reasons, doing so would not be practical for the actual product.

Often, in a simulation circuit incorporating a temperature sensor such as the RTD, the intent is to hold the interface electronics at a constant temperature and subject only the sensor to the temperature excursion. Alternatively, one may wish to hold the temperature sensor at a constant temperature, then drift the interface circuit and observe its behavior over a limited temperature range. The former requires some method that will allow all of the temperature sensor interface electronics to remain at the constant temperature, and the sensor alone to be drifted over temperature over a range that may be beyond that of the simulator software.

One approach to overcoming the simulator temperature range limitations is to design the RTD macromodel such that it responds to a different stimulus. For instance, a voltage or current unit could be equated to a temperature unit such as 1 V or 1 mA could be made equivalent to 1°C. The temperature range then becomes fundamentally limitless from a simulation standpoint. This unit transformation may be accomplished by employing a voltage- or current-controlled resistor as the basis for the RTD macromodel. A voltage-controlled resistor would be ideally suited for this purpose.

An excellent SPICE resource is the eCircuit Center (<http://www.ecircuitcenter.com/>). It provides information on a wide variety of SPICE topics and models. Among the many model topics listed is one for a voltage-controlled resistor (VCR). The VCR is based on a *resistorless* model relying on the basic Ohm's Law relationship of $V = I \times R$. The R in this application is an electrical equivalent of a resistance. The I is a sensed current through this resistance. An ammeter function is handled in SPICE with a zero volt (0 V) voltage source. The voltage (V) across the resistor is based on the sensed current times the equivalent resistor value:

$$Value = \{I(V_{Sense}) \cdot R\}$$

This ability to describe the output voltage in terms of an equation is exploited in the development of the RTD macromodel.

Model development flexibility is achieved by employing the SPICE analog behavioral modeling (ABM) option. Simply, it provides you the ability to create a controlled voltage and current source whose value is dependent on a mathematical expression. The value can be the solution of a simple linear relationship, or something more complex like the polynomial response associated with an RTD. Here's an example of a SPICE VCVS model that incorporates an ABM expression. Eq. 3 from above will be used in the model:

$$Eth \ 1 \ 3 \ value = \{ I(Vsense) * R_{nom} * (1+(A*V(4,5) + (B*PWR(4,5),2))))\}$$

Eth is the VCVS designator. A voltage applied to nodes 4 and 5 controls the VCVS output. R_{NOM} and coefficients A and B are from Eq. 3. Vsense is a 0 V independent source that senses the current through R_{nom} . Additional statements are needed to build the complete macromodel. Also, a separate TINA program operation is required to create the macromodel symbol, although that procedure will not be described here.

Fig. 2 (overleaf) shows the schematic diagram for a prototype RTD macromodel.

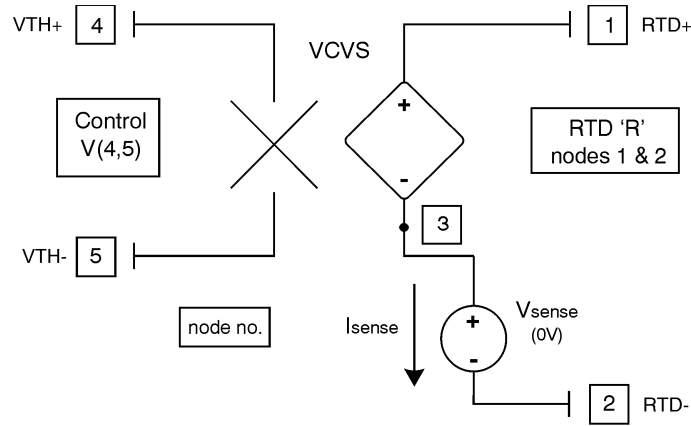


Fig. 2: Prototype SPICE RTD Simulator Model

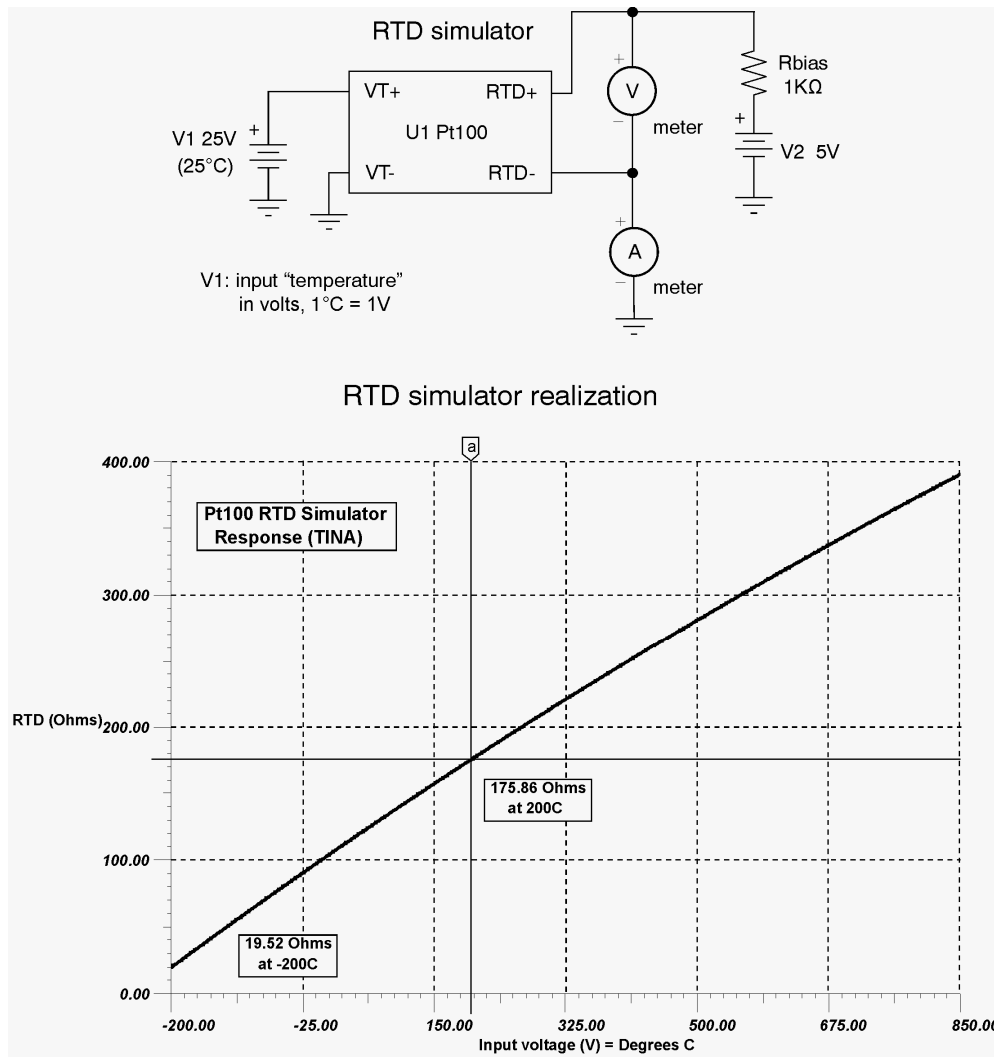


Fig. 3: RTD Simulator Response

Platinum RTDs are available with different nominal resistances and have correspondingly unique coefficients. Therefore, provisions can be included within the Netlist to change their values

easily. This is readily accomplished using the `<.param>` keyword or *parameter* expression for variables. A test circuit that includes the prototype RTD macromodel and the simulated response are provided in Fig. 3. The control voltage that corresponds with temperature is swept over a temperature range from $-100\text{ }^{\circ}\text{C}$ to $+850\text{ }^{\circ}\text{C}$. Then the corresponding RTD across temperature is calculated and plotted. This plot is developed using a TINA post-processing analysis tool.

The objective of equating volts to degrees Celsius is accomplished by the prototype RTD macromodel. This is useful over the entire Pt100 temperature range. However, because of its simplicity, inaccuracy increases as the temperature moves ever further below 0°C .

This is a good time to mention that the prototype RTD macromodel may be used to accurately simulate a different $100\ \Omega$ standard RTD. The SAMA RC-4-1966 is a US RTD standard that uses a slightly dissimilar platinum alloy content than the Pt100. It is specified with a resistance of $98.129\ \Omega$ (0°C) and has polynomial coefficients that differ slightly from the Pt100. Unlike the Pt100, it doesn't require correction below 0°C . The prototype RTD macromodel can be used to this RTD minimum specified temperature. Simply substitute R_{nom} and new coefficients within the prototype RTD Netlist allow the model to be used with this particular RTD.

Attaining precise performance from the Pt100 RTD macromodel requires the third- and fourth-order coefficient terms to be included for temperatures below 0°C . This could be accomplished using curve fitting techniques, but doing so requires further analysis and modification of the response equation and even higher order polynomial coefficients might be required. Some of the outcomes may include unacceptable errors within some temperature spans and/or endpoints. A bit more brute-force, but nonetheless precise alternative, is to switch in a second VCVS that becomes active only below 0°C . The basic RTD macromodel can be modified to add a second VCVS (VCVSB) into circuit where it will be summed with VCVSA at sub-zero temperatures.

Connecting VCVSB into the path with VCVSA can be accomplished with a voltage-controlled switch model available in SPICE. Here the goal is for the switch to transparently connect VCVSB into the circuit as the temperature, or equivalent voltage in this case, passes through 0°C . SPICE provides both voltage- and current-controlled switches and in this application the voltage option is the easiest to apply. Here the VCVSA control voltage serves a convenient voltage source used to activate the voltage-controlled switch connecting VCVSB into the circuit.

This is a straightforward concept, but because the voltage-controlled switch (VSWITCH) model includes non-ideal characteristics, they behave a little differently than an ideal switch. These non-ideal characteristics include R_{ON} and R_{OFF} resistances and switch voltage thresholds. During switching, the voltage-controlled switch actually ramps between R_{OFF} and R_{ON} as the switch is closed, and visa versa. Otherwise, the instantaneous ON/OFF discontinuity plays havoc with the circuit, and convergence can be difficult to achieve during a simulation. The SPICE manual recommends that the ratio of R_{ON} to R_{OFF} be kept less than 10^{12} due to SPICE dynamic range limitations. R_{ON} was set to $0.1\ \Omega$ and R_{OFF} to $1\ \text{M}\Omega$ in the RTD simulator model.

Controlling the switch state is a matter of selecting the switch V_{ON} and V_{OFF} voltages. When the control voltage is below V_{OFF} , the switch is OFF and it exhibits a resistance R_{OFF} . Similarly, with the control voltage above V_{ON} , the switch resistance now becomes R_{ON} . The transition region

between V_{OFF} and V_{ON} is an area where the switch develops gain. The narrower this range is made, the higher the gain. Note that SPICE cautions against making this region too narrow. Experimentation is sometimes required to get this right.

A SPDT switch is required for the VCVS switching function. This is realized using two SPST switches. The switches require that the ON/OFF thresholds be carefully set so their switch transitions overlap at zero. If not, a large glitch will result at the switching point.

When the model was first tested all appeared to be functioning correctly. But then a problem arose with the RTD resistance value when the temperature dropped below 0°C . A close examination of the model revealed that the $0.1\ \Omega$ switch resistance R_{ON} was an overlooked factor that comes into play when the switch is ON. Adding a uniquely configured VCVS to the circuit provided compensation for the ON resistance. It subtracts a voltage equal to the voltage drop created by current I_{SENSE} flowing through R_{ON} . This VCVS, labeled E_{RON} , relies upon an ABM value statement, where its voltage is a function of the current I_{SENSE} .

Lastly, an additional voltage-controlled, voltage-source, VCVSC is added to provide a direct reading meter of the RTD resistance. Its output is a voltage that is proportional to the RTD resistance at a given temperature, where $1\ \text{V}$ of output voltage represents $1\ \Omega$ of RTD resistance. The resistance is a function of the voltage developed across the RTD^{+} and RTD^{-} terminals and the current through the RTD circuit, I_{SENSE} . This meter is added as a convenience. Placing the voltmeter across the monitor terminals is optional and may be removed providing the SPICE simulator employed can tolerate open terminals.

Fig. 4 shows the complete schematic diagram for the full-temperature range Pt100 macromodel. The final circuit result is an RTD simulator that smoothly switches in VCVSB as the temperature transitions through 0°C .

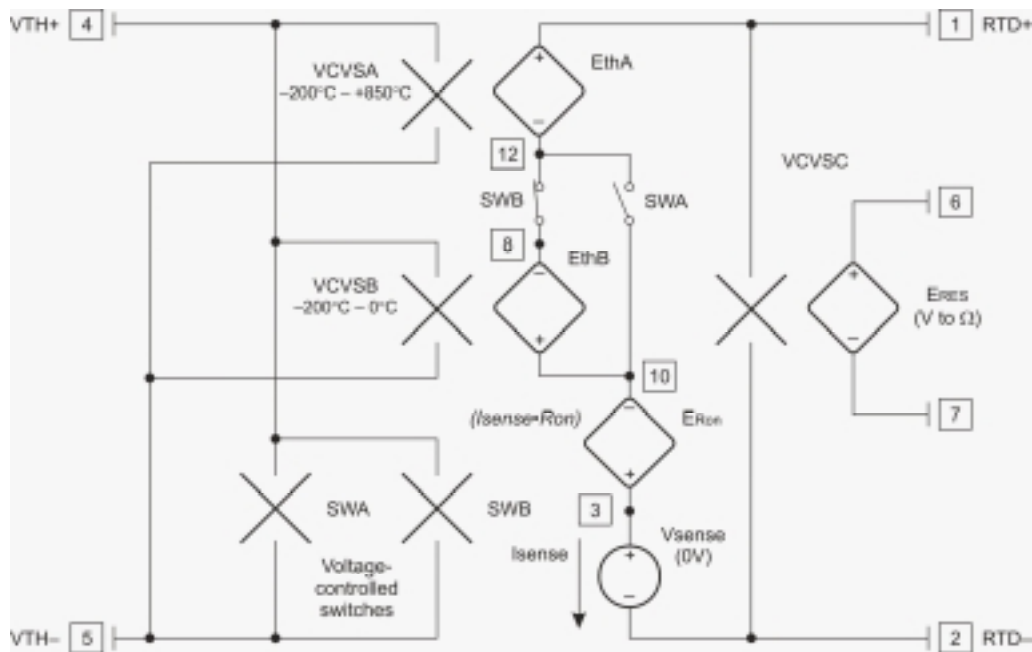


Fig. 4: Complete RTD Simulator Model

A SPICE Netlist for the complete Pt100 RTD macromodel is provided in Appendix 1. The syntax listed is compatible with most versions of Cadence PSPICE. If your simulation program relies on a SPICE-based engine, you should be able to convert the syntax without much difficulty. The Netlist includes commenting, making it easy to alter the parameters for a particular RTD. \

The RTD simulator allows the temperature corresponding input voltage to be changed to any desired value, but be sure to use a range applicable to the particular RTD. When the macromodel was tested from -200°C to $+850^{\circ}\text{C}$, the resistance mimicked the Pt100 polynomial equation to at least four places past the decimal. This included testing temperatures within ± 0.1 of 0°C , where the SWA and SWB switch state. When using the macromodel, be sure to keep the RTD current within the recommended operating range for the actual RTD device. The RTD macromodel does not include self-heating effects.

The RTD macromodel, or RTD simulator as it might best be called now, by itself has limited usefulness. However, when coupled with an RTD interface circuit, more meaningful circuit simulations can be conducted. Fig. 5 illustrates an applications circuit where the RTD simulator is connected with an INA326 instrumentation amplifier. The INA326 provides voltage gain and signal conditioning. The reference pin voltage was selected such that the output voltage swings from nearly 0 V when the RTD's temperature is -200°C , and 4.096 V at $+850^{\circ}\text{C}$. This output voltage range fits nicely with the input range of single-supply ADCs.

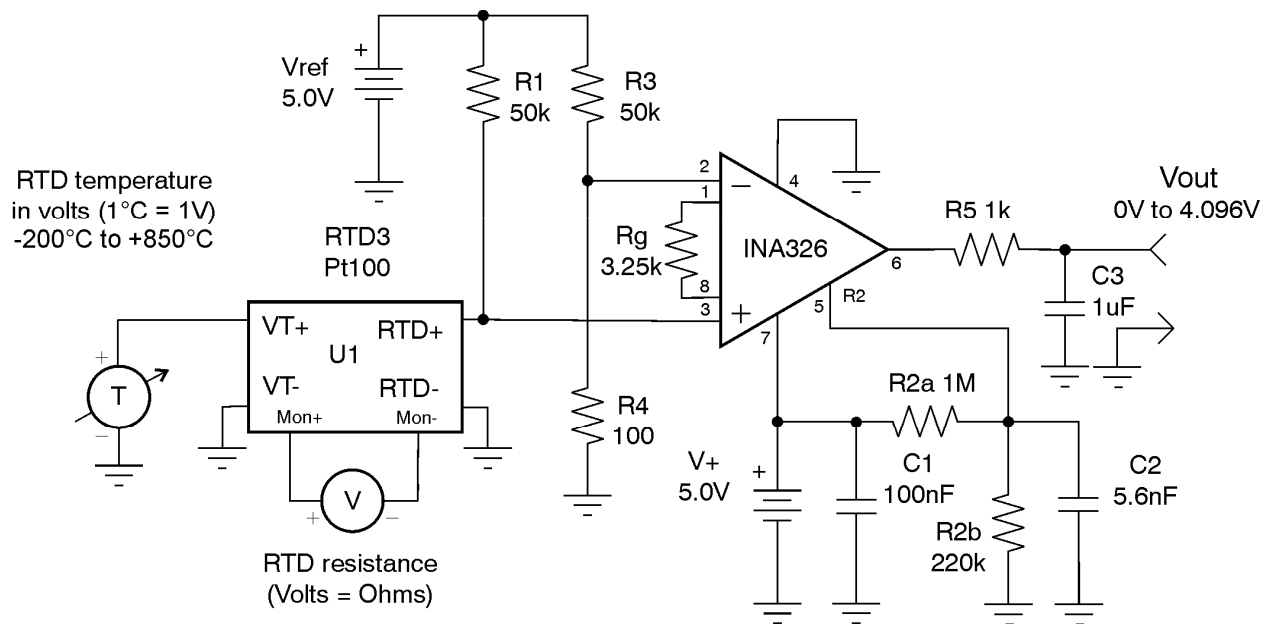


Fig. 5: Pt100 RTD Simulator With INA326 Amplifier

A good ADC choice for this application is the Texas Instruments, 12-bit, ADS7829, which has a 0 V to 4.096 V input range. Operating the INA326 from split supply rails allows the output to readily swing to 0 V and below. Since most RTDs are slow responding sensors, the INA326 instrumentation amplifier is bandwidth-limited at 100 Hz. Bandwidth limiting the amplifier allows one to take advantage of the ADC high SNR. Some applications may allow you to use an

even lower cutoff frequency. Here the -3 dB bandwidth is set by a second-order, low-pass function, comprised of RC networks connected at the output and R2 pins of the INA326. Information on selecting the component values can be found in the datasheet. Fig. 6 shows the simulated RTD resistance and INA326 dc output levels across the full temperature range.

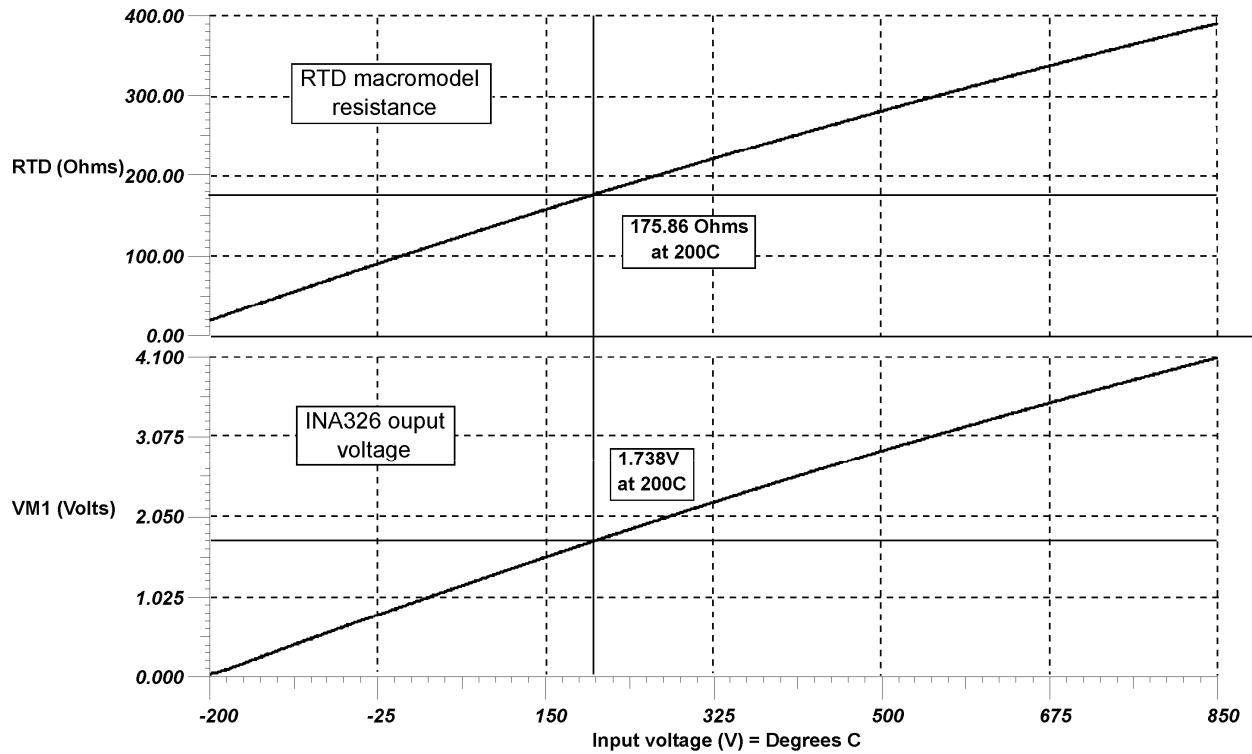


Fig. 6: Pt100 Simulator Resistance And INA326 Voltage Response Across Temperature

The RTD macromodel, RTD3, is available for download at the Texas Instruments TINA-TI website <http://www.ti.com>

Acknowledgements

The author would like to thank his colleagues Tim Green and Neil Albaugh (retired) of the Precision Analog, Linear Applications Engineering group at Texas Instruments. Their analog circuit expertise and modeling suggestions have proven invaluable. I also would like to thank Bill Sands, of *Analog & RF Models*, who shared some insights on using curve fitting techniques as a way of modeling the RTD. And lastly, Rick Faehnrich of eCircuit Center, for providing an informative SPICE resource site to the engineering community. It proved to be of great help during the RTD macromodel development.

About The Author

Thomas Kuehl is a Senior Applications Engineer in the High-Performance Linear group of Texas Instruments. Before joining the applications group, he spent the first quarter-century of his career in product engineering. His external interests include practicing the guitar, amateur radio (AC7A) and enjoying the outdoors. Thomas has published several articles on communications antennas. He can be reached at: ti_tomkuehl@ti.com

Appendix 1: RTD Simulator Netlist Using PSPICE Syntax

```
* Wide temperature RTD simulator with monitor
* Revision A, T E Kuehl 07-27-2006
* RTD temperature range is -200°C to +850°C (IEC)
* This model includes sub-zero (t<0C), third- and fourth-order terms
* RTD+ pos RTD connection
* RTD- neg RTD connection
* VT+ pos voltage corresponding to measured temperature
* VT- neg voltage corresponding to measured temperature
* Mon+ VT+ external meter connection
* Mon- VT- external meter connection

*** Note if simulation errors occur with TINA 7.0 versions (TINA-TI 7.0)
*** select the "Analysis" tab, then "Set Analysis Parameters" > click on the
*** "hand symbol" > "view all." Find and change Shunt Conductance [S]
*** from 0 to 1e-12.

*Connections VT+ VT- Mon+ Mon- RTD+ RTD-
.subckt RTD3 4 5 6 7 1 2

*** RTD coefficients
*** Pt100
.param Rnom = 100
.param A = 3.9083e-3
.param B = -5.775e-7
.param C = -4.23225e-12

*** Voltage controlled, voltage sources

EthA 1 12 value = {I(Vsense)*(Rnom*(1+(A*V(4,5))+(B*PWR(V(4,5),2))))}
EthB 10 8 value = {I(Vsense)*(Rnom*(C*PWR(V(4,5),3))*(V(4,5)-100))}

*** Voltage controlled switches

*** +SW -SW +Con -Con
SWA 12 10 5 4 SWTEMPA
```

```
.Model SWTEMPA VSWITCH (Ron= 0.1 Roff=1e6 Von=50u Voff=100u)
SWB 12 8 5 4 SWTEMPB
.Model SWTEMPB VSWITCH (Ron= 0.1 Roff=1e6 Von=25u Voff=0)
```

```
*** Correction for SWA/SWB "on" resistance, Ron = 0.1 Ohms
ERon 3 10 value = {I(Vsense)*0.1}
```

```
*** RTD Resistance Value
Eres 6 7 value = {V(1,2)/I(Vsense)}
```

```
*** Current meter
Vsense 3 2 DC 0
```

```
.ends RTD3
```

