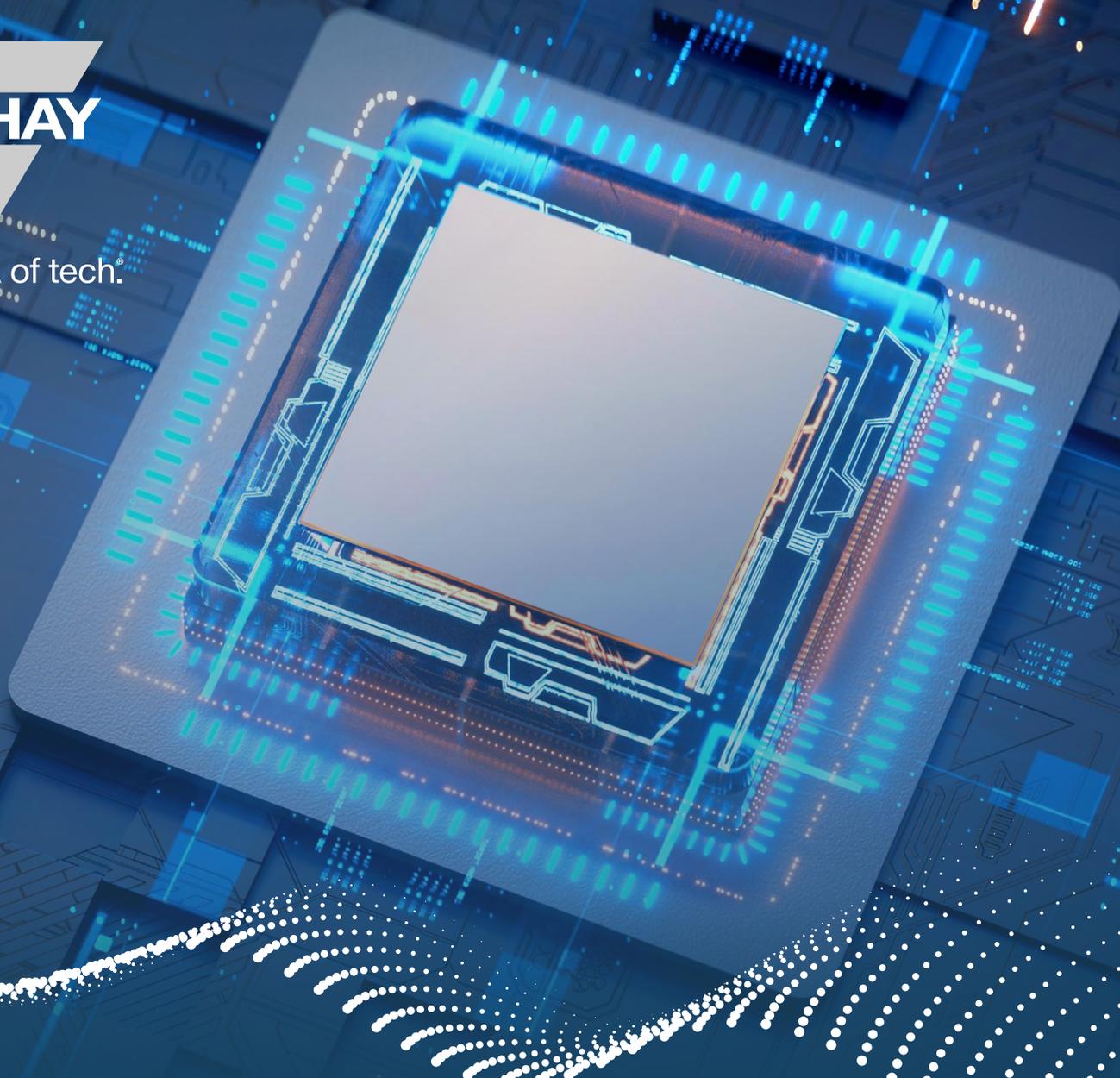




The DNA of tech.®



ELECTRONIC SIMULATION TOOLKIT NON-LINEAR RESISTORS

www.vishay.com



Vishay Non-Linear Resistor (NLR) Products

This book is conceived as a kit and cumulates some exemplative information for a proper conceptual analog circuit design for the following Vishay NLR products: the latest leaded through-hole **NTCLE350** and **NTCLE317** NTC thermistor; **SMD PTS** platinum thin film resistors; **PTCEL** ceramic PTC thermistors; and bare die **NTCC200 / NTCC201**, in addition to the more mature SMD **NTCS**; other leaded **NTCL** thermistors; and nickel thin film SMD **TFPTs**; **PTCSL03** temperature sensing components; and **NTCALUG** and **VDR** NTC assemblies. Links to datasheets, infographics, videos, and simulation tools are provided (all links are given for free downloadable simulations, with reprints of the website-associated text files). There is a focus on analog conception of temperature measurement or compensation circuits with SPICE software like LTspice® XVII [1], online VHDL-AMS analog circuit design tools like PartQuest™ Explore from Siemens EDA [2], or other simulations tools, such as SIMetrix [3]. For a complete overview of electronic simulations developed by Vishay, consult the Appendix (page 63). All the videos can also be found in reference [4]. All the presented simulations files can be downloaded from reference [5].

- [1] LTspice XVII is provided by ADI at [LTspice Simulator | Analog Devices](#)
- [2] PartQuest Explore is an online tool located at [PartQuest™ Explore | Design, Modeling, Simulation and Analysis](#)
- [3] [SIMetrix](#), from SIMetrix Technologies at [SIMetrix Circuit Design and Simulation](#)
- [4] [Vishay Thermistors Electronic Simulation](#)
- [5] [Hackster.io - The community dedicated to learning hardware](#)

The section about PTCELS covers overcurrent limitations, while the TFPT0402 exemplifies airflow detection. Jump directly to the page of interest by clicking on the links below.

- 1. [NTCLE350 Series](#) 2
- 2. [NTCLE317 Series](#) 4
- 3. [PTS Series](#) 6
- 4. [PTCEL Series](#) 13
- 5. [NTCC200 / NTCC201 Series](#) 21
- 6. [NTCS Series](#) 23
- 7. [NTCLE213 Series](#) 30
- 8. [NTCLE100..B0 Series](#) 43
- 9. [TFPT Series](#) 45
- 10. [NTCLE203...SB0 Series](#) 47
- 11. [PTCSL03 Series](#) 54
- 12. [NTCALUG Series](#) 56
- 13. [VDRS, VDRUS, and MLV Varistors Series](#) 60
- 14. [All Simulations With Thermistors / RTDs](#) 63

1. NTCLE350 SERIES FOR DEMANDING APPLICATIONS (RELATIVE WATERPROOFNESS, RESISTANCE TO FLUIDS)

Links

- Datasheet: www.vishay.com/doc?29218
- Product presentation video: [Vishay NTCLE350 Series Thermistors - YouTube](#)
- Product page at Vishay website for further info: [Vishay NTCLE350 Series Thermistors](#)
- Simulation: [Oil Temperature Measurement Simulation With Vishay NTCLE350 - Hackster.io](#)

Oil Temperature Measurement Simulation With Vishay NTCLE350

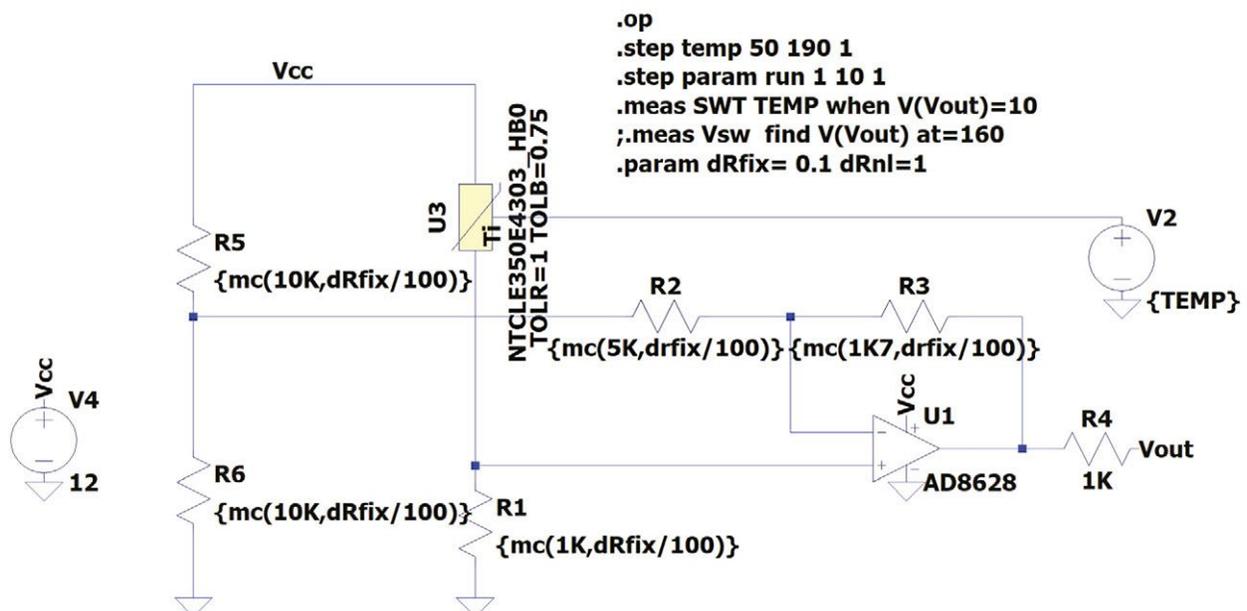
An oil temperature measurement circuit is simulated in LTspice for a Wheatstone bridge with a thermistor for temperatures between +50 °C and +185 °C.

Hardware Components Used in This Project

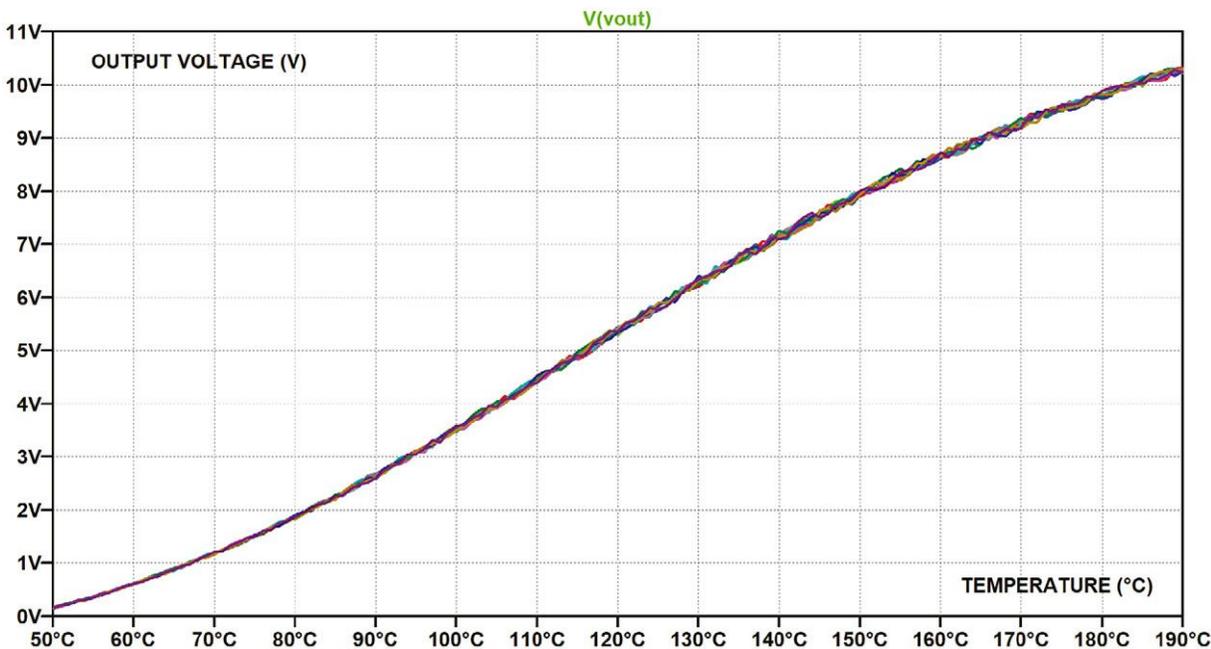
- 1 x Vishay CRCW
- 1 x Analog Devices AD8628
- 1 x [Vishay NTCLE350](#) - insulated extended temperature, through-hole

Story

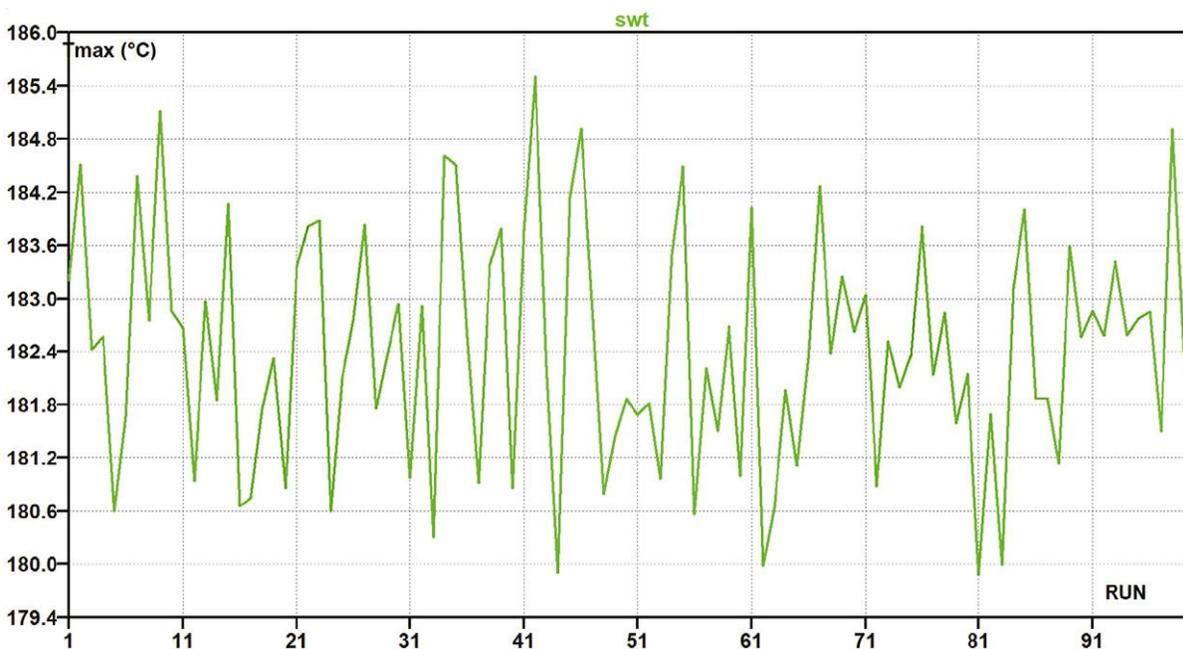
This is a simulation in LTspice XVII (presently offered by ADI) of a project for an oil temperature sensor for temperature going from +50 °C up to +185 °C. The base element is a Vishay NTCLE350 thermistor (PEEK-insulated, leaded, NiFe 0.2 mm wire, 30 kΩ, ± 1 %). The whole LTspice model can be downloaded from the attachments for this project. The particularity of this simulation is that all tolerance for the NTC thermistor and all other passive elements are modeled according to a simple Monte Carlo function. The circuit is presented in the following picture:



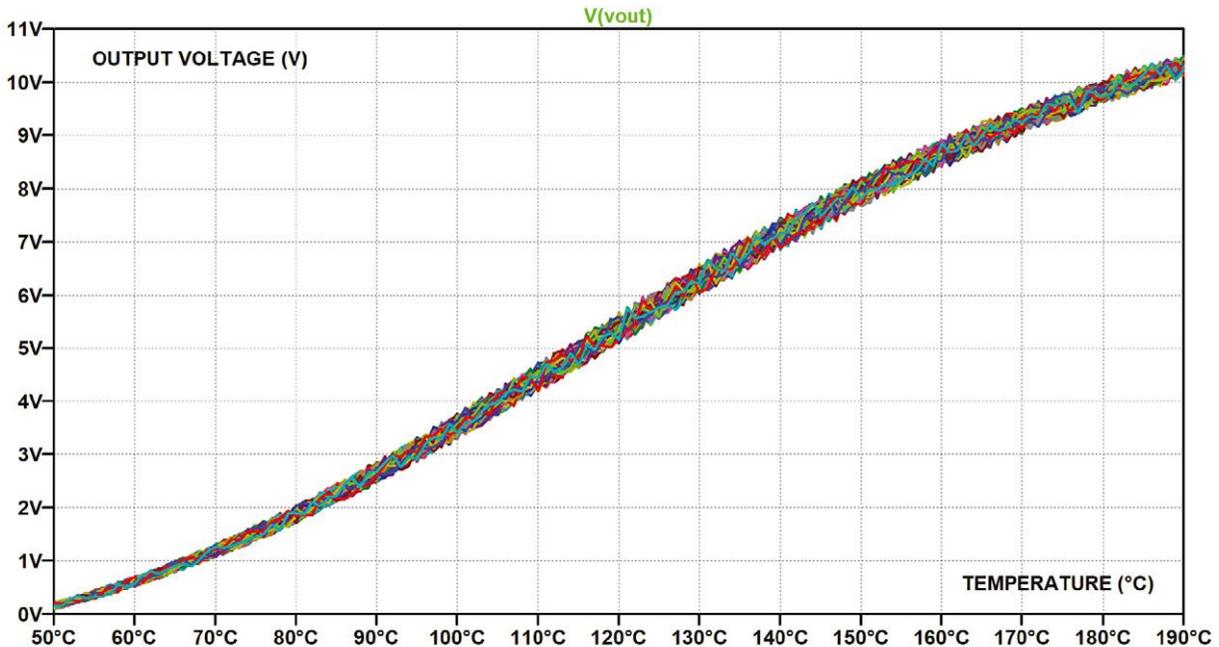
The simulation of the output voltage as a function of the temperature TEMP gives the following graph:



It is then possible with some SPICE directives to determine the accuracy of the V_{OUT} measurement: at +185 °C, V_{OUT} is equal to 10.06 V nominally. Introducing tolerances for the circuit elements, we can then compute at which temperature span the V_{OUT} voltage will cross 10.06 V. The SPICE error log file allows for visualization of this span, which will swing between +183 °C and +187 °C, thus +185 °C \pm 2 °C, values near the specification.



One of the advantages of the simulations is that the user can program other tolerances at will and visualize the results immediately. Here are the results with $dR_{25} = 5\%$ and $dB_{25/85} = 1\%$.



In the SPICE error log file, the temperature span corresponding to a real +185 °C temperature can be evaluated around +182 °C to +188 °C. The user can thus evaluate if the decreased tolerance (price increase) is worth the value of the improved accuracy.

2. NTCLE317 SERIES FOR LESS AGGRESSIVE APPLICATIONS (FEWER FLUIDS, ACIDS, AND ADBLUE) BUT VERY FAST REACTIONS REQUESTED

Links

- Datasheet: www.vishay.com/doc?29216
- Product presentation video: [NTCLE317E4103SBA NTC Thermistors. Long Insulated Leads 150 °C With Very Low Thermal Gradient | Vishay](#)
- Product page at Vishay website for further info: [NTCLE317E4103SBA NTC Thermistors. Long Insulated Leads 150 °C With Very Low Thermal Gradient | Vishay](#)
- Simulation: [Simulate Accurately Your NTCLE317E4103SBA Temp. Accuracy - Hackster.io](#)

Accurately Simulate Your NTCLE317E4103SBA's Temperature Accuracy

This LTspice simulation shows how to model this NTCLE317 ± 0.5 °C accuracy thermistor used in a simple ADC circuit.

Hardware Components Used in This Project

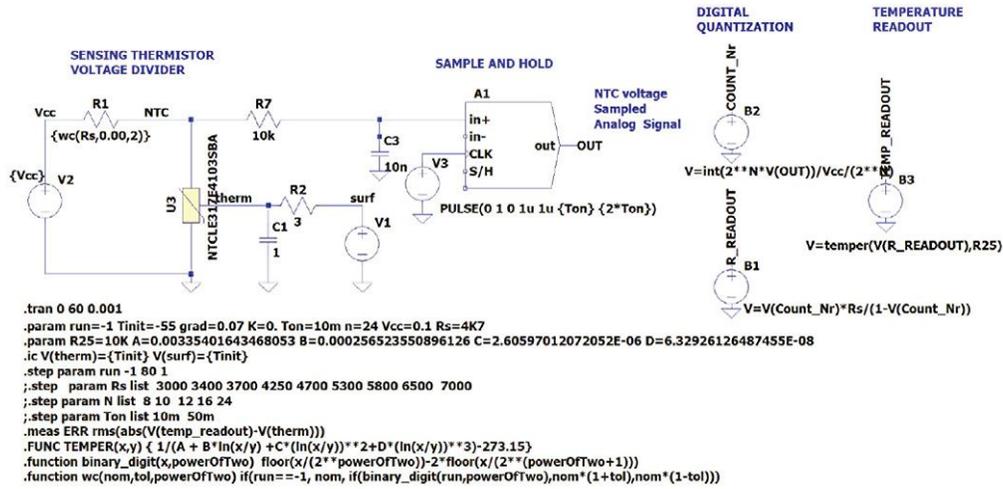
- 1 x Vishay CRCW
- 1 x [Vishay NTCLE317 10 k \$\Omega\$, 0.5 °C Accuracy](#)

Story

The Vishay NTCLE317E4103SBA two-point sensor thermistor (accuracy of ± 0.5 °C between +25 °C and +85 °C) is simulated in LTspice XVII. The polynomial curve of Steinhart and Hart as well as the afore-mentioned accuracy are modeled as precisely as can be.

The presented circuit intends to provide a temperature reading from a thermistor voltage after sampling and digital quantization. This temperature reading is then compared to the real temperature and the (simulated) sensor accuracy is then represented graphically.

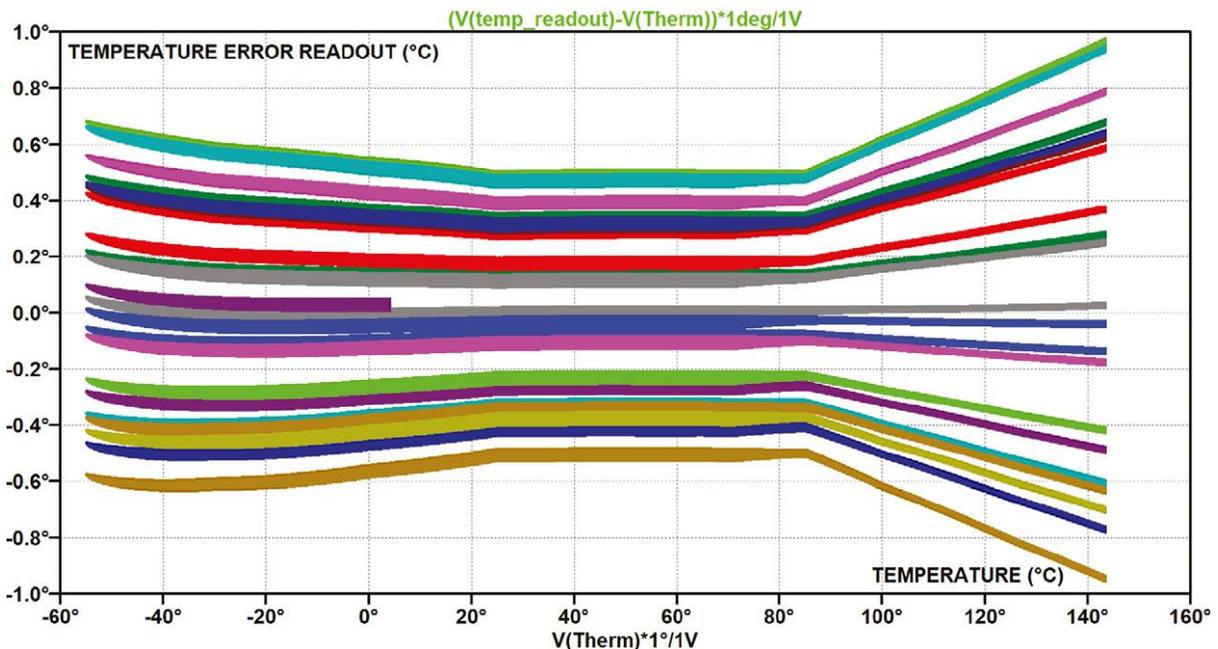
Here is the circuit:



We present the results corresponding to a sample time period of 10 ms and we assume that the ADC bit number is 24. The measured temperature is represented by the voltage source V1, which swings between low and high temperature. The thermistor voltage $V(NTC)$ is then sampled and converted to a digital signal and then transformed into a temperature knowing the Steinhart and Hart coefficients (A, B, C, and D) of the NTCLE317E4103SBA thermistor.

LTspice is the ideal software to perform such a simulation, as the thermistor characteristics tolerances can be easily expressed in temperature terms via flexible arbitrary behaviour sources (ABS). The netlist of the thermistor modelling can be found in the software section of the project. We have adopted a Monte Carlo distribution: this means that each simulation run picks up one tolerance of the sensor randomly within the tolerance range. We analyze here the sensor accuracy alone, but in a further simulation every other circuit element could be studied with its own tolerances.

Once thirty runs are completed, we plot the temperature measurement error as a function of the thermistor temperature V_{therm} (transformed in °C), and we get the following graph.



This accurately reproduces the two-point sensor's precision (± 0.5 °C between +25 °C and +85 °C, and then increasing up to ± 1 °C around +140 °C).



3. PTS1206 SERIES

Links

- Datasheet: www.vishay.com/doc?28762
- Product page at Vishay website for further info: [PTS Series Temperature Dependent Platinum Thin Film Chip Resistor \(RTD\) Vishay](#)
- Simulation: [Linear Output Temperature Sensing Circuit With Vishay PTS - Hackster.io](#)



PTS SURFACE-MOUNT PT THIN FILM RTD

LINEAR POSITIVE TEMPERATURE COEFFICIENT ELEMENT



PTS Series
(Standard and AT)

COMPLIANCE TO STANDARDS

- AEC-Q200 qualification
- ESD HBM 0.5 kV (0603);
1 kV (0805 / 1206)
- RoHS-compliant
- Halogen-free
- Whisker-tested per JEDEC 22A121.01

MULTIPLE DESIGN TOOLS AVAILABLE

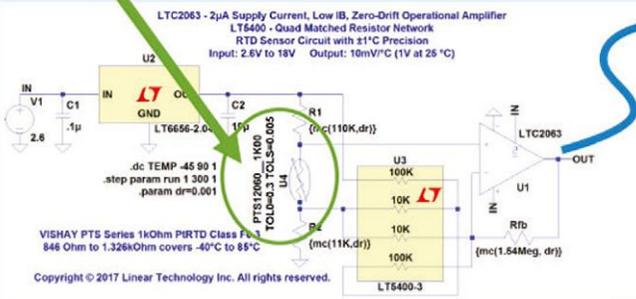
- LTspice® models with Monte Carlo tolerances available (online)
- PSpice® models available (online)
- VHDL-AMS models available (on SystemVision® Cloud website)

TEMPERATURE SENSING / CONTROL / PROTECTION / COMPENSATION

APPLICATIONS

- LED over current limitation
- Optoelectronics
- Portable instrumentation systems
- Car infotainment
- Motor vehicles
- Automotive under the hood applications
- Medical equipment

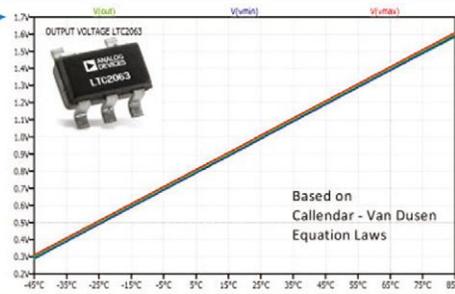
PTS1206 Class F0.3



LTC2063 - 2µA Supply Current, Low IB, Zero-Drift Operational Amplifier
 LT6400 - Quad Matched Resistor Network
 RTD Sensor Circuit with ±1°C Precision
 Input: 2.8V to 18V Output: 10mV/°C (1V at 25 °C)

VISHAY PTS Series 1kOhm PIRTD Class F0.3
 846 Ohm to 1.326kOhm covers -40°C to 85°C

Copyright © 2017 Linear Technology Inc. All rights reserved.



Based on Callendar - Van Dusen Equation Laws

For Technical Questions: nlr@vishay.com
 © 2019 VISHAY INTERTECHNOLOGY, INC. ALL RIGHTS RESERVED

3.1 LINEAR OUTPUT TEMPERATURE SENSING CIRCUIT WITH VISHAY PTS

An optimized linear output temperature sensing circuit based on Vishay PTS platinum thin film RTD and an Analog Devices LTC2063.

Hardware Components Used in This Project

- 1 x Analog Devices LTC2063
- 1 x Vishay PTS1206
- 1 x Analog Devices LT5400-3

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

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, 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 μ A supply current, presented in 2017 by Analog Devices [1].

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

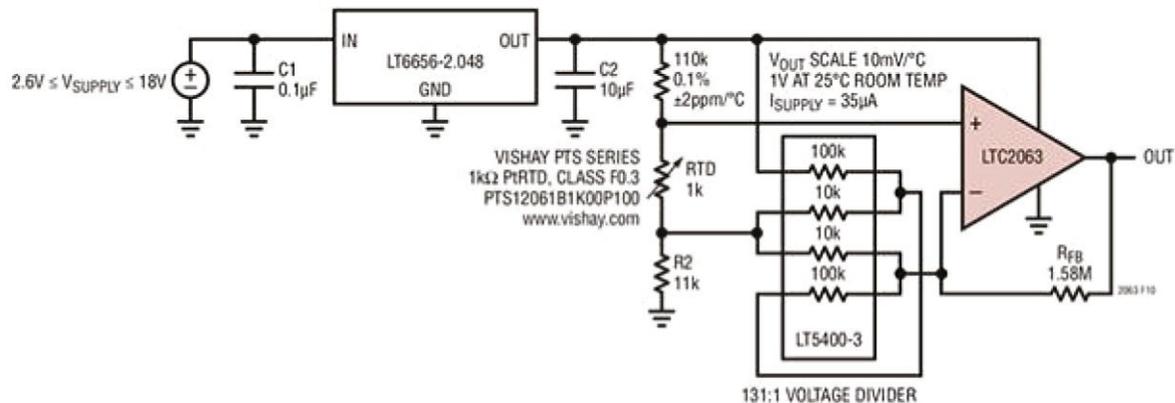


Fig. 1

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 [2]). This 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. The 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 ideal linearity between -40 °C and +85 °C, for example, even after some linearization work.

Of course, Analog Devices offers a particularly useful LTspice model [3] for this circuit, where the PTS sensor is presented as a variable resistor, as shown in Fig. 2.

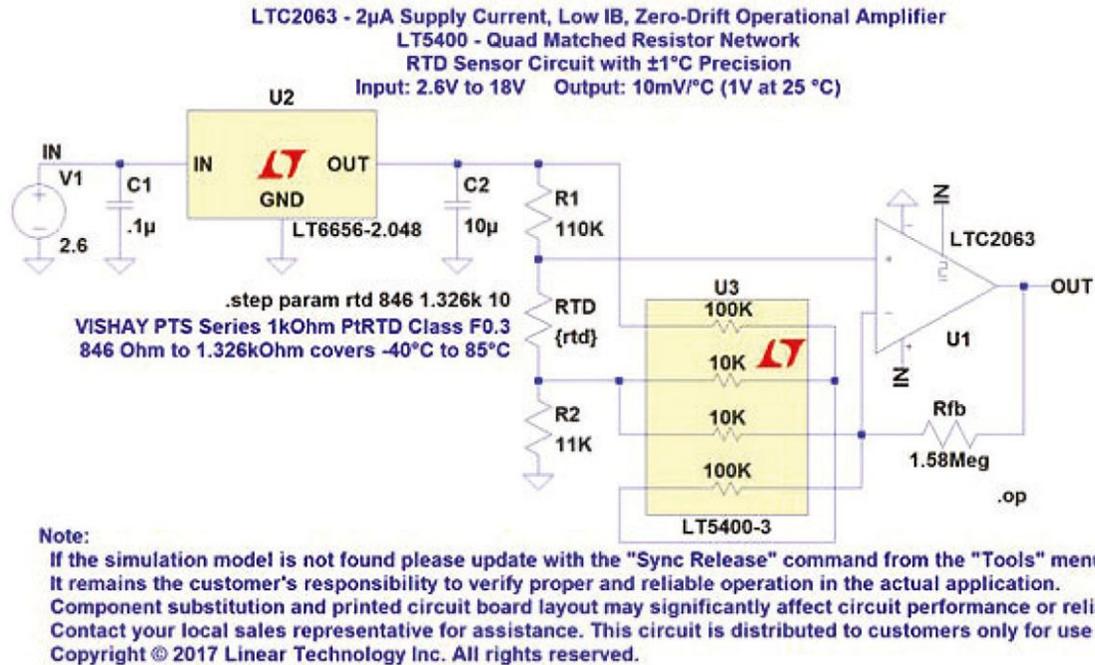


Fig. 2

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 $\pm 1^\circ\text{C}$ 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 case the DC temperature sweep SPICE model has been introduced into the simulation to avoid making the user of the LTC2063 dig down in the PTS datasheet. That leads us to Fig. 3, where we are now able to:

- Sweep the temperature variable
- Visualize the influence of the tolerances in temperatures 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 $^\circ\text{C}$ (see Fig. 4)

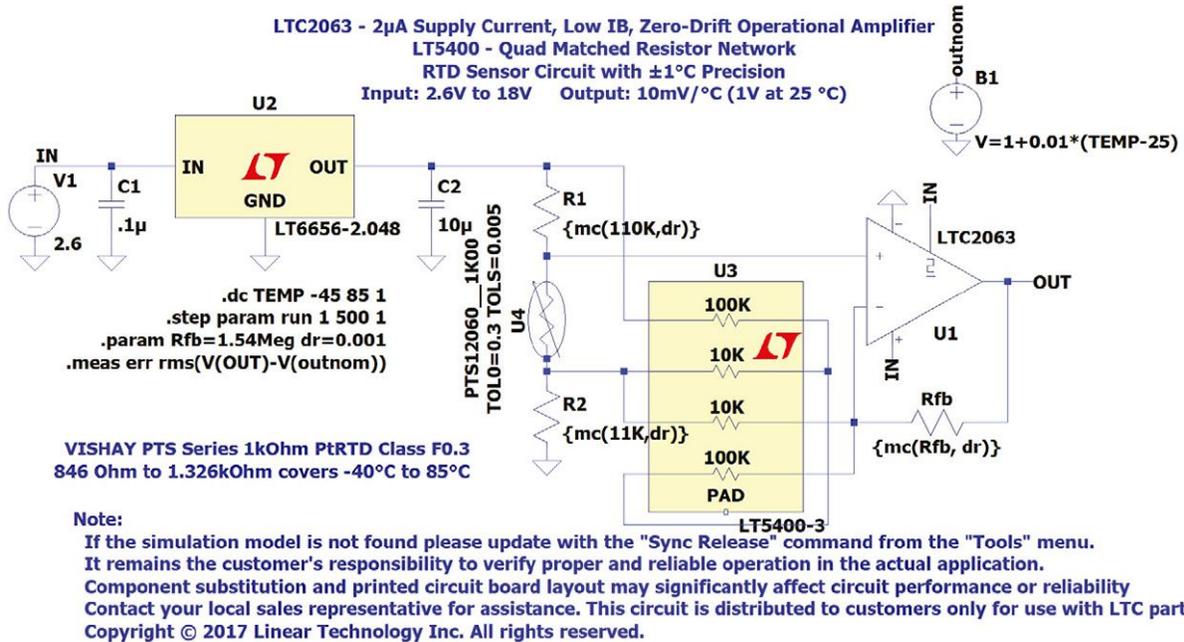


Fig. 3

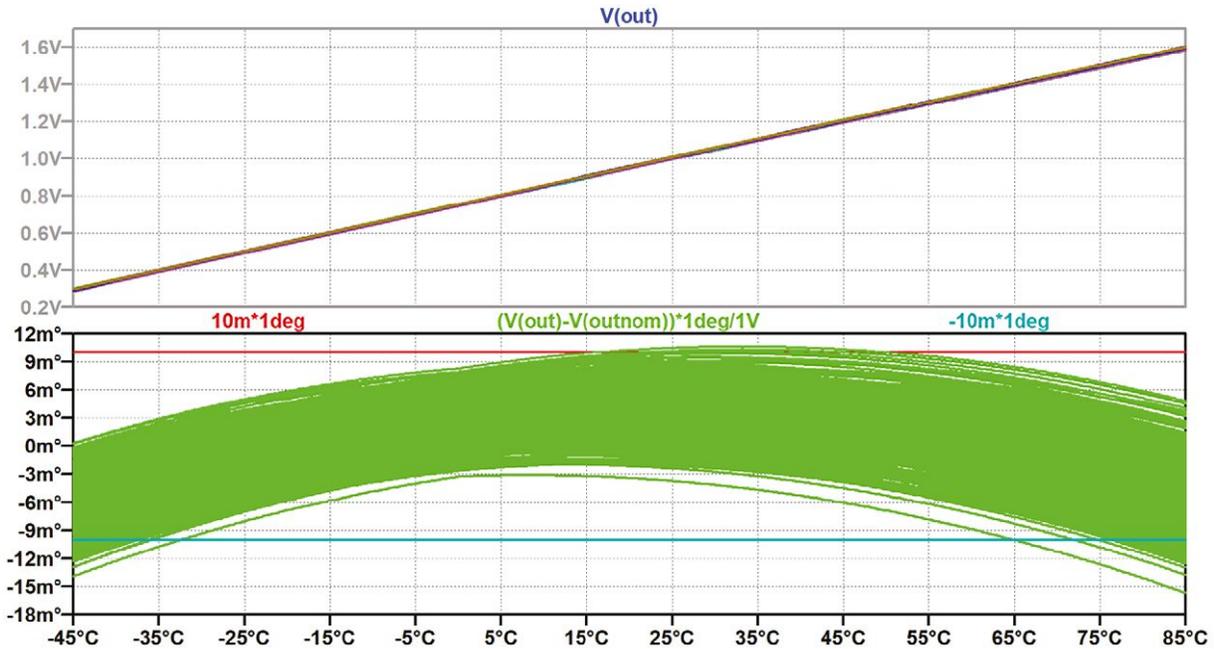


Fig. 4

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 Fig. 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 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 possibilities 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] PTS - Analog Devices: <https://www.analog.com/en/products/ltc2063.html>
- [2] PTS AT series for automotive use: <https://www.vishay.com/doc?28899>
- [3] Analog Devices: <https://www.analog.com/en/design-center/design-tools-and-calculators/LTspice-simulator.html>

3.2 ANALOG TEMPERATURE COMPENSATION WITH PTS

Link

- [Simulation of RTD Temperature Compensation for Log Converter - Hackster.io](#)

Simulation of RTD Temperature Compensation for Log Converter

A classical analog logarithmic converter is simulated with LTspice with emphasis on drift compensation in temperature with a PTS RTD.

Hardware Components Used in This Project

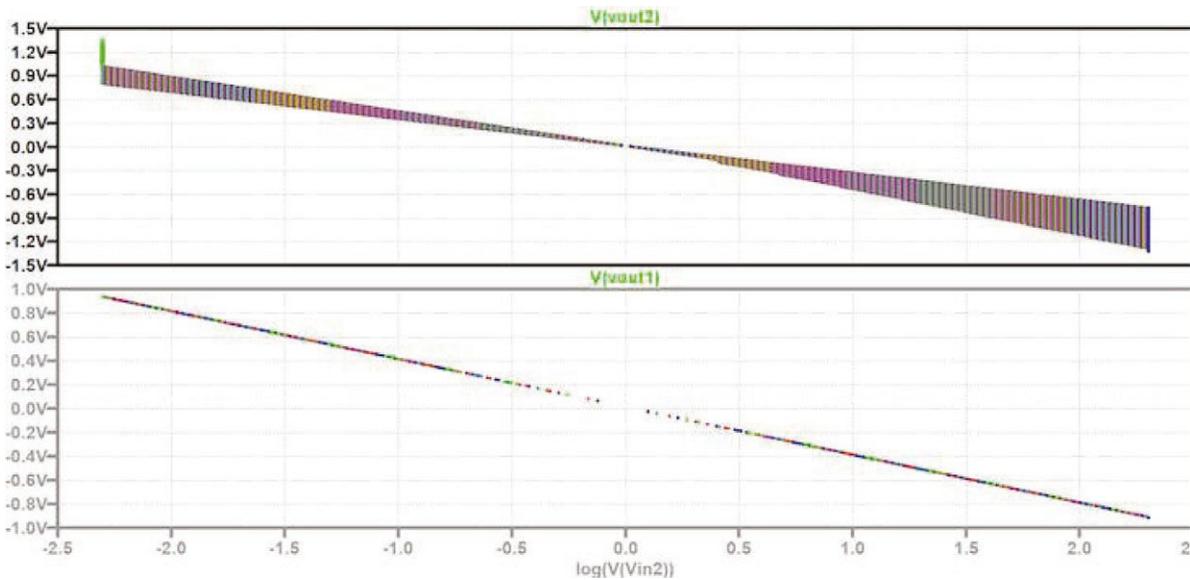
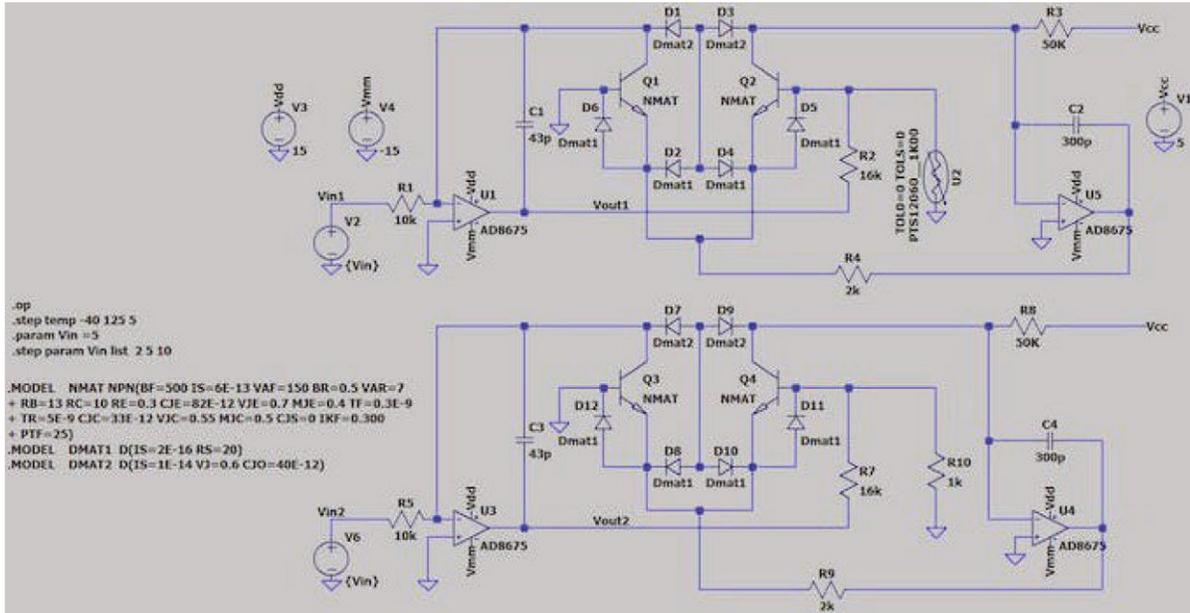
- 1 x Vishay PTS12060__1K00
- 1 x Analog Devices SSM2212
- 1 x Analog Devices AD8675

Story: Temperature Compensation With a Linear RTD From Vishay for a Logarithmic Converter

Back in my youth, I used to contemplate the circuit ideas in the wonderful book “The Art of Electronics” by Horowitz and Hill (ISBN 0-521-37095-7).

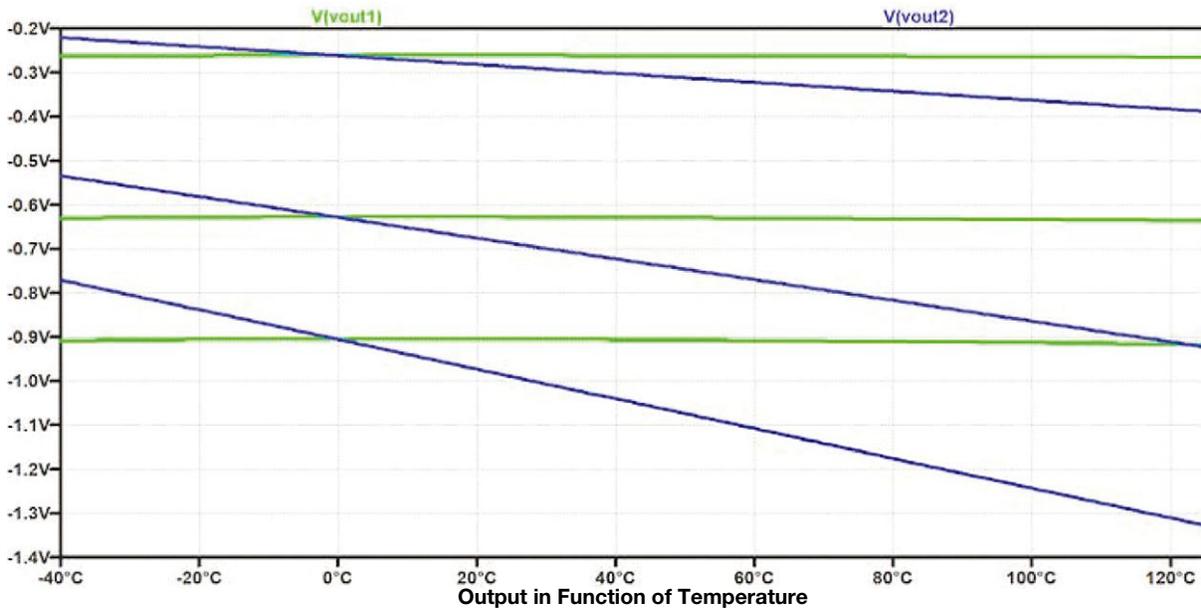
Among the multitude of exposed circuits, I found the fundamental chapter on opamps on page 255 a proposed schematic for a fast logarithmic converter with a peculiar temperature compensation. The idea has come to me recently to simulate this circuit with SPICE to fully visualize what such a circuit does. I have developed full SPICE models for such temperature-dependent resistances (PTS surface-mount from Vishay), and I could not wait to reproduce the original circuit idea in simulation. Such a log converter was important for in all sorts of computations for dB conversion, etc. The conversion is based on the fact that the base emitter voltage of a npn transistor is proportional to the logarithm of the collector current. But at the same time, it depends on temperature. And that is the reason for the presence of an RTD, which is a temperature linearly depending resistance. Here, there are two circuits: one with RTD (circuit above) and its equivalent with a fixed resistor (under).

We apply different V_{IN} levels and record the output voltage V_{OUT1} (with compensation) and V_{OUT2} (without compensation) for three temperatures (-40 °C, 25 °C, and 125 °C).

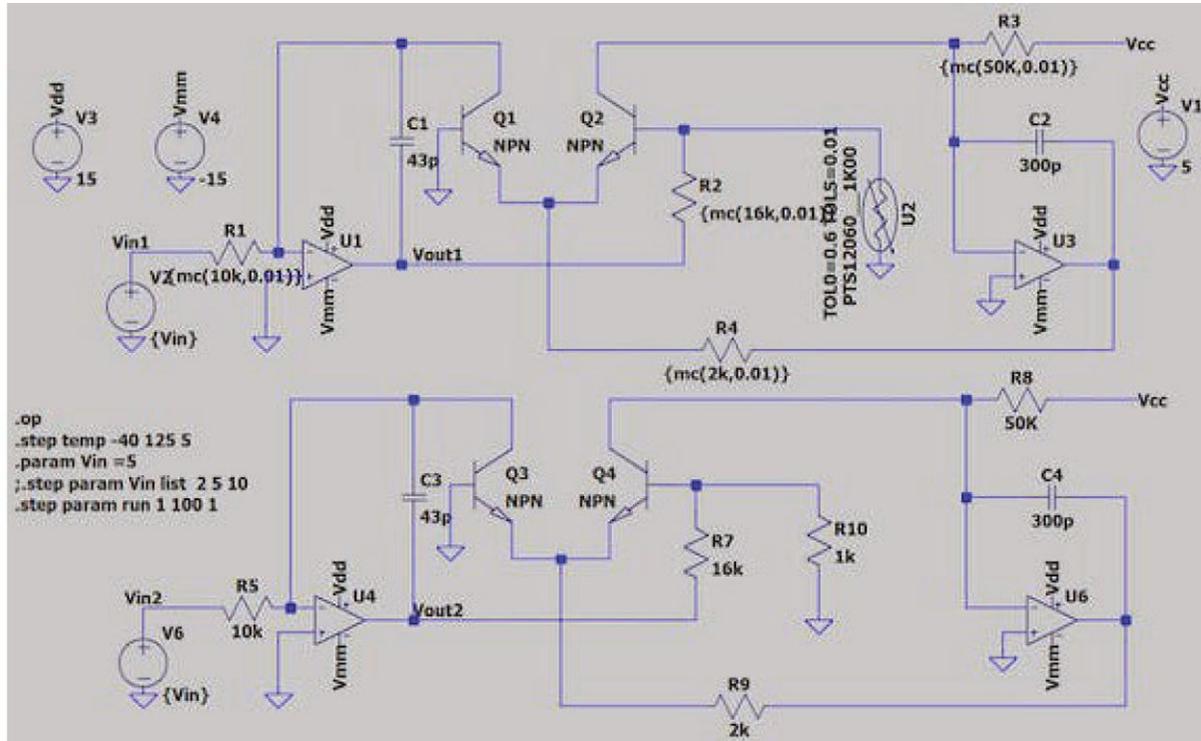


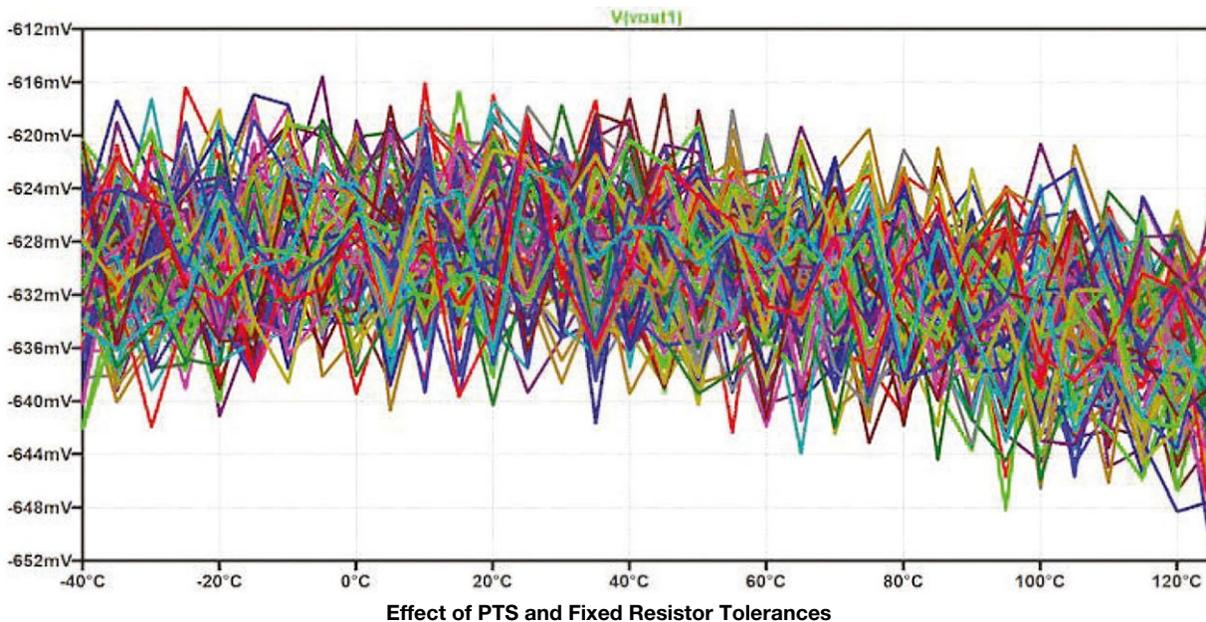
We see that V_{OUT1} (lower pane) is perfectly stabilized in temperature, and changes linearly as a function of the log of the input voltage. On the contrary, V_{OUT2} presents a spread in temperature for any fixed V_{IN} .

The sweeping in temperature for different input voltages shows even better the compensation effect (green compensated against blue non-compensated).



If we introduce the PTS tolerances ($TOL_0 = 0.6$ and $TOL_S = 0.01$) we can rerun the simulation, and we see the influence of these tolerances on the compensation: for $V_{IN} = 5$ V, all the fixed resistors have $\pm 1\%$ tolerance.





4. PTCEL: OVERCURRENT SUPPRESSION IN PRE-CHARGE / DISCHARGE CIRCUITS

Links

- Datasheet: www.vishay.com/doc?29165
- Product page at Vishay Website for further info: [PTCEL PTC Thermistors, Inrush Current Limiter and Energy Load-Dump | Vishay](#)
- Infographics: www.vishay.com/doc?48615
- Product video: [PTCEL PTC Thermistors, Inrush Current Limiter and Energy Load-Dump | Vishay](#)
- Simulation: [SPICE Simulation of Overcurrent Protection With Vishay PTC - Hackster.io](#)

4.1 SPICE SIMULATION OF OVERCURRENT PROTECTION WITH VISHAY PTC

An LTSpice simulation shows how a PTCEL network protects an SMPS circuit, allowing the smoothing capacitor to charge completely.

Hardware Components Used in This Project

- 1 x Vishay PTCEL17R121NBE

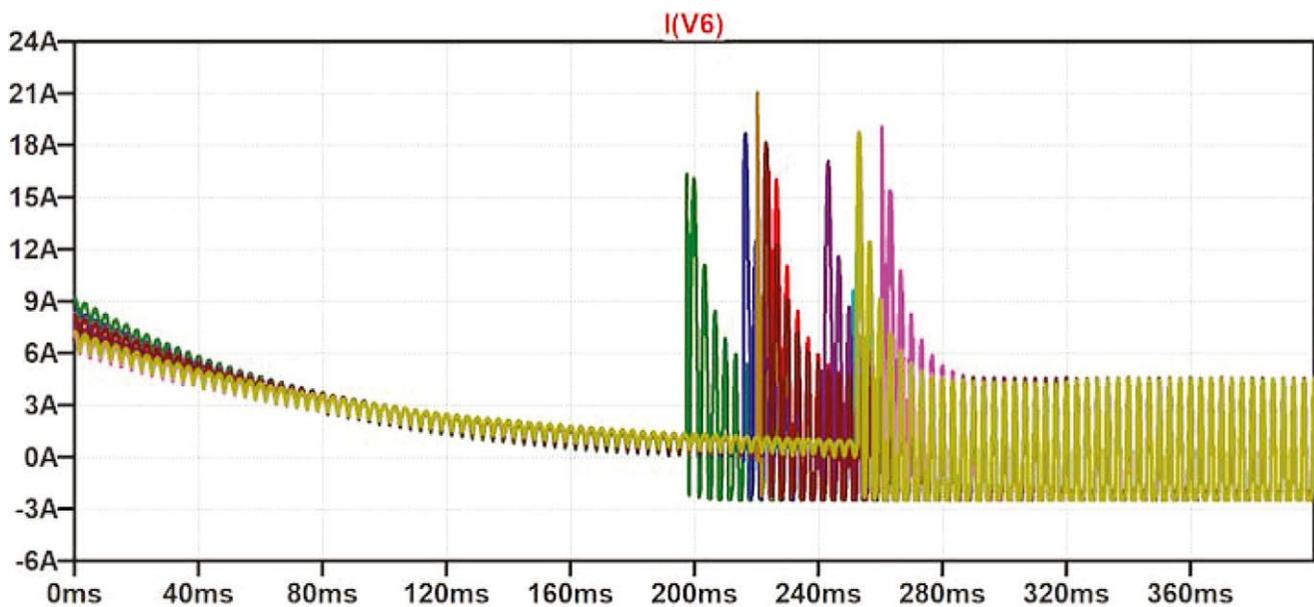
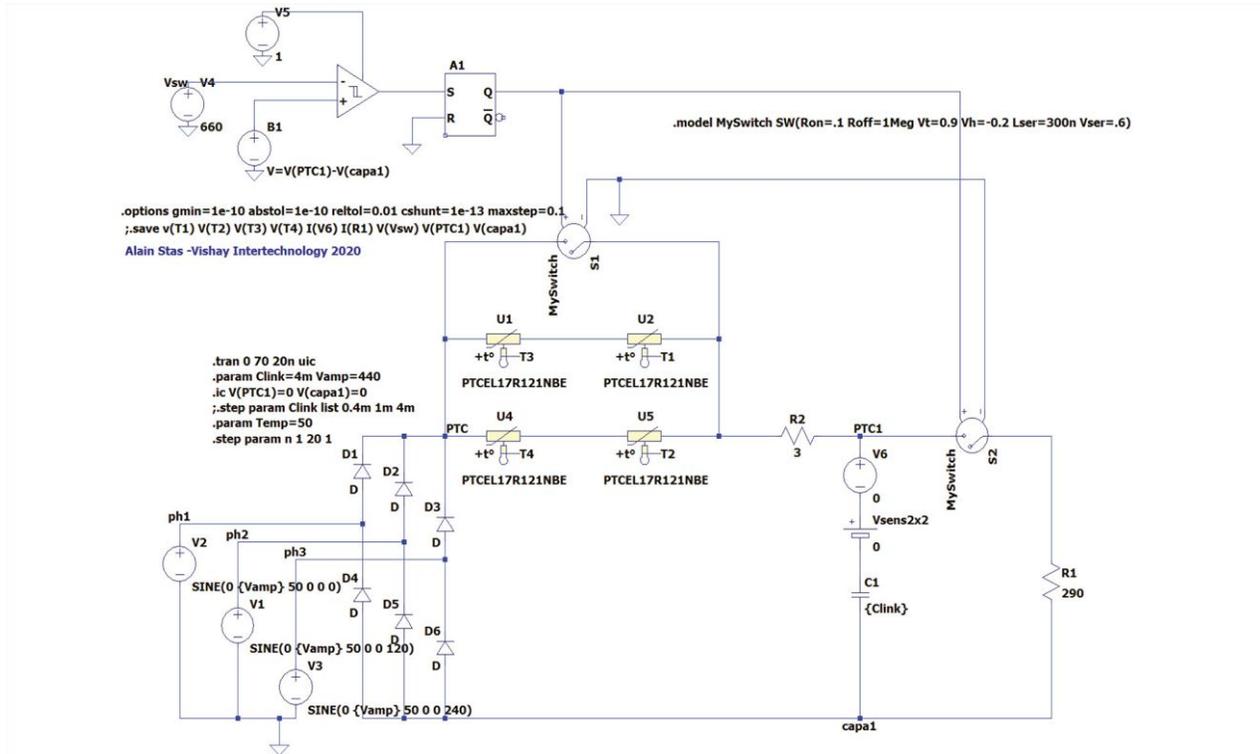
Story: Overcurrent Protection of SMPS With a Network of Vishay PTC Thermistors (PTCEL)

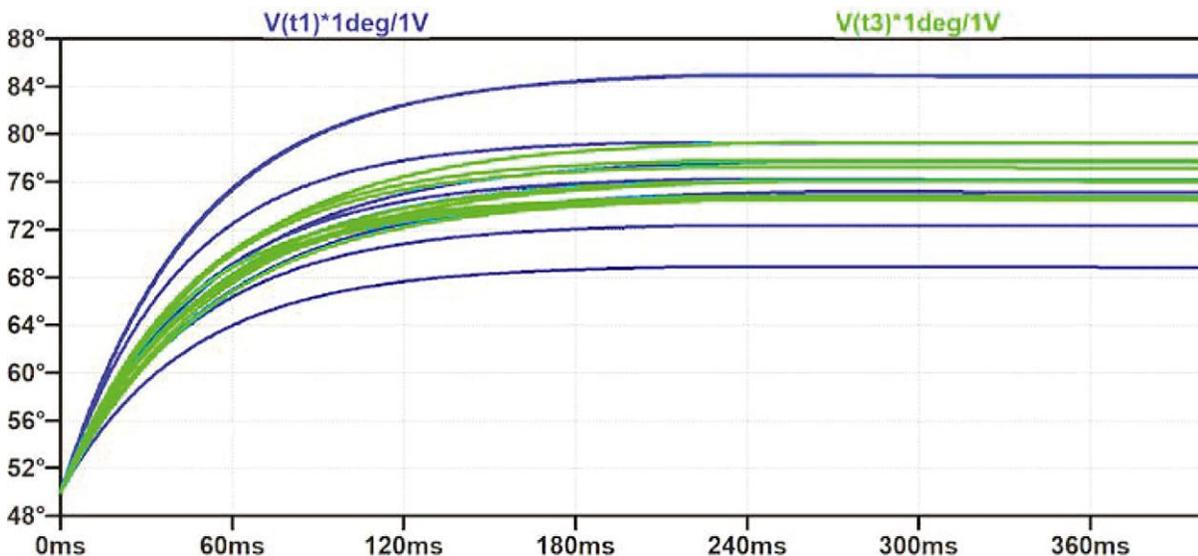
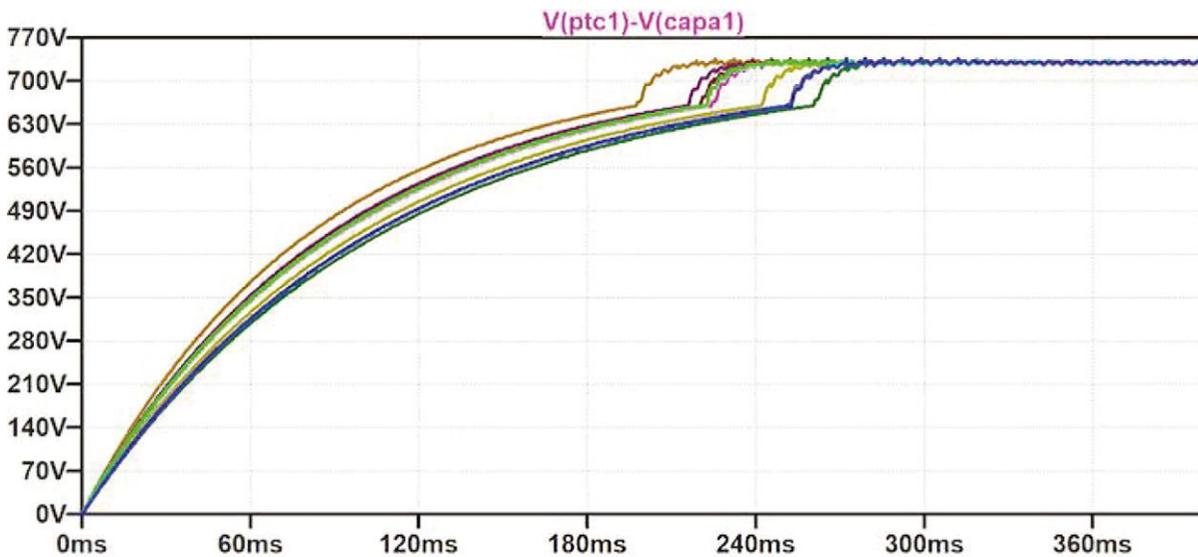
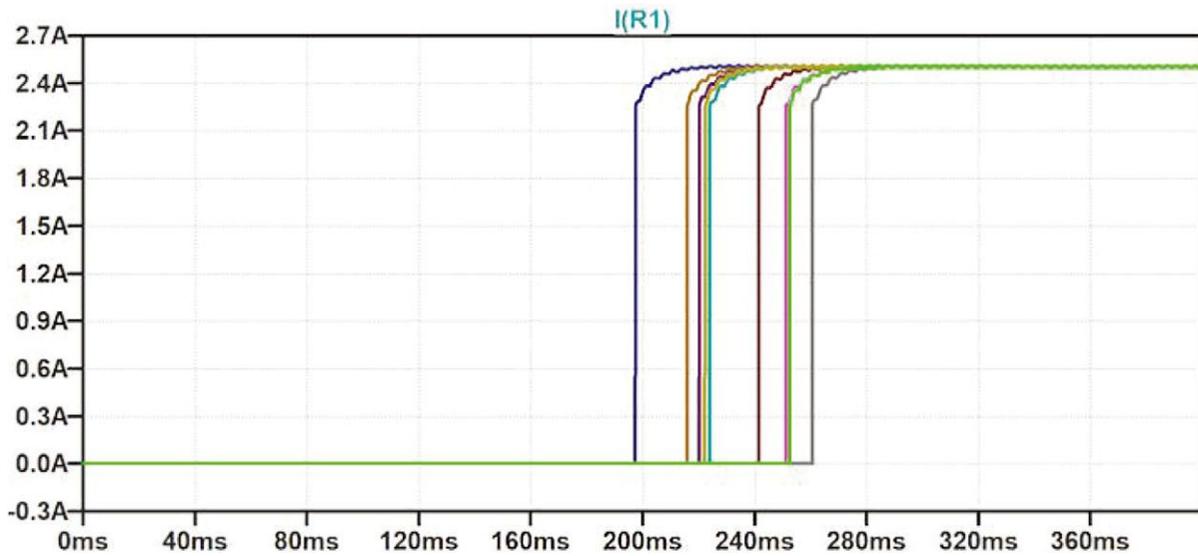
The simulated circuit is an SMPS at switch-on. A smoothing capacitor is charging and pumps a lot of current. A PTC network protects the diode bridge against the surge. If overloads really happen, the PTCs will heat up and switch to high resistances. When the capacitance has charged to a given level, a switch disconnects the PTC network and switches on the SMPS load. The two situations simulated hereunder correspond to:

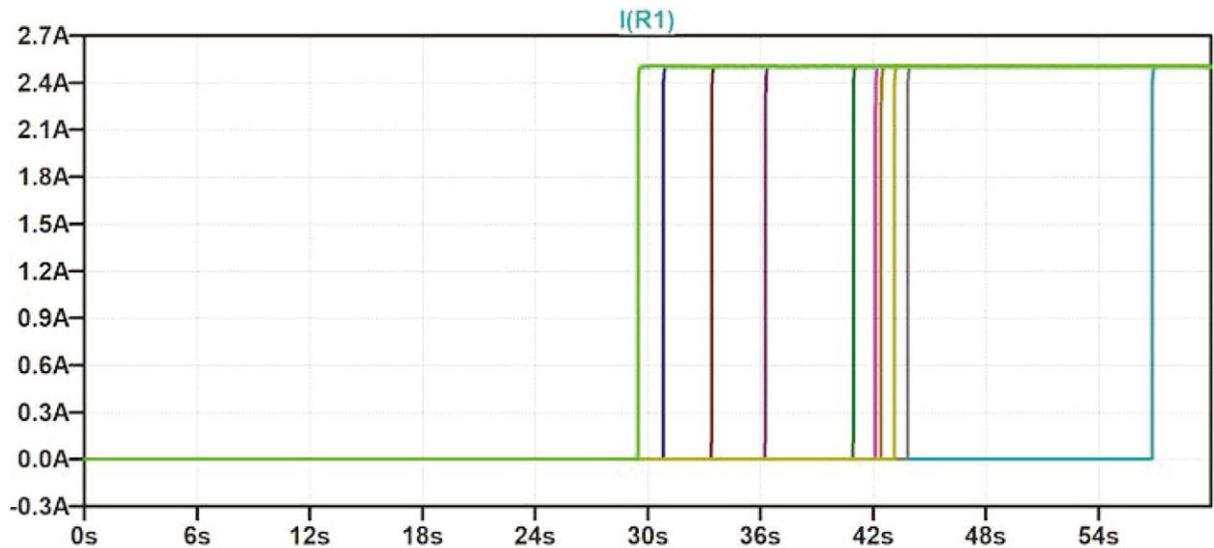
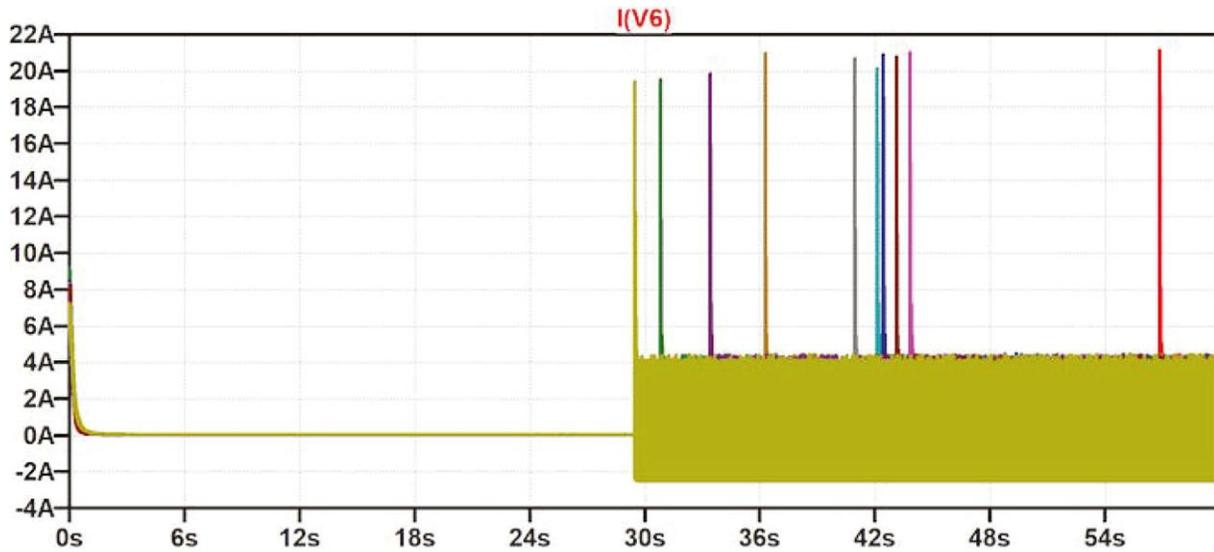
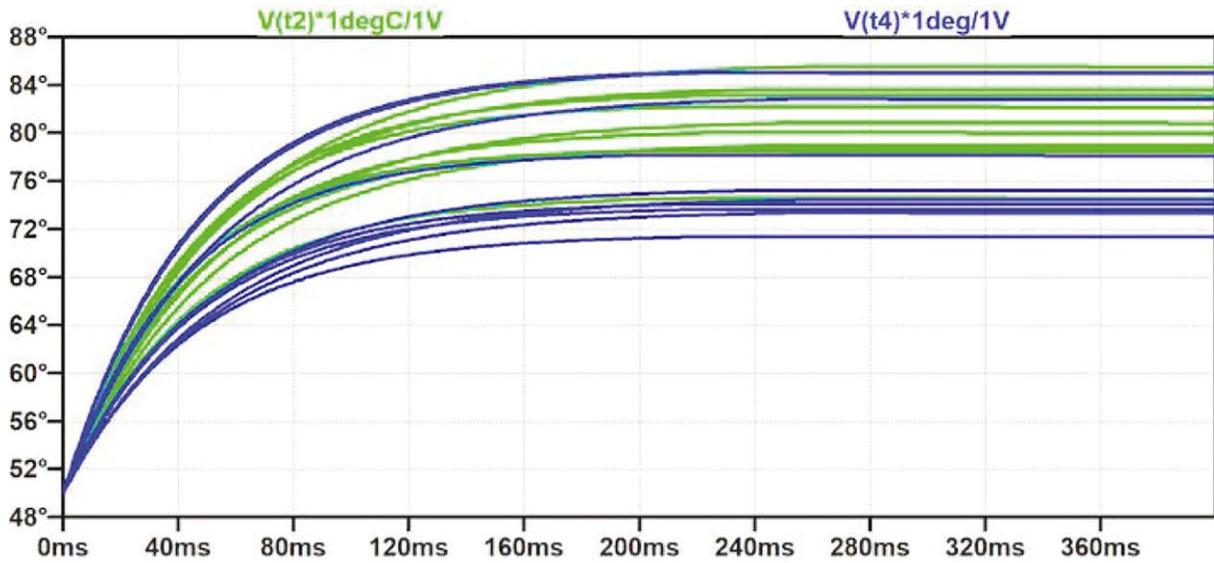
1. A normal circuit behavior (the switch happens between 180 ms and 300 ms)
2. A case where the capacitance is too high (the switch will happen much later, between 30 s and 60 s). In this case, PTC 2 goes to a high value first and after when the current has dropped down. As the PTC cools down, the capacitance ends up charging at the switch level, albeit after a very long time.

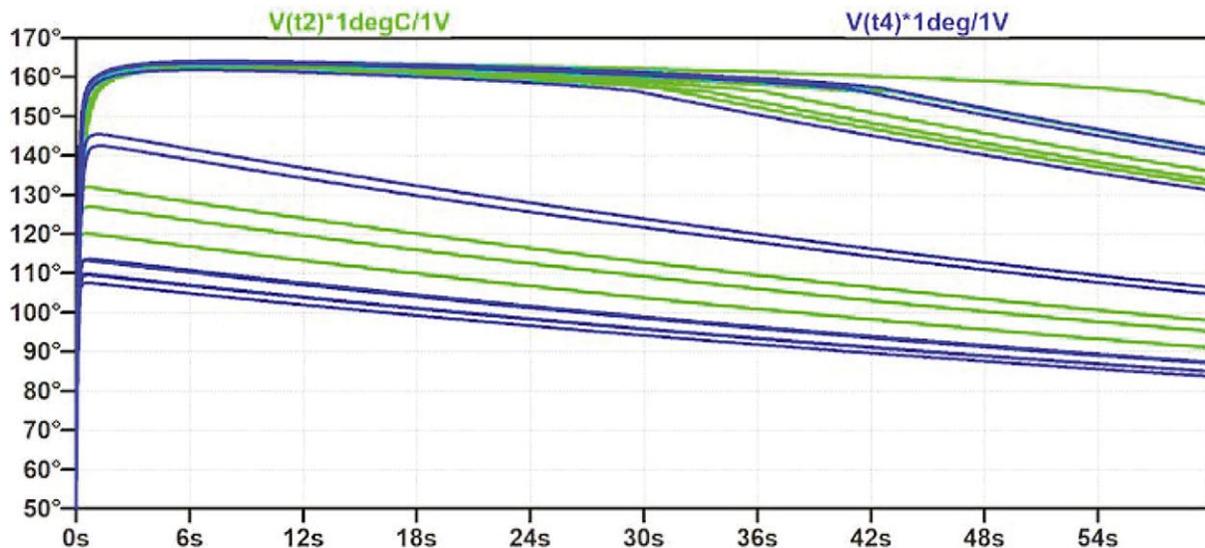
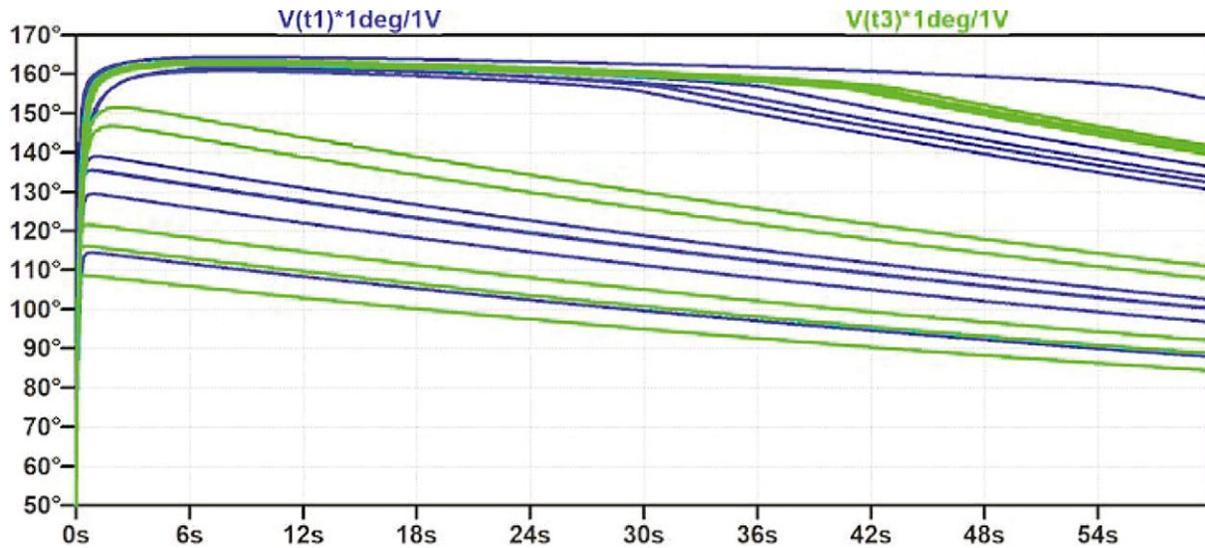
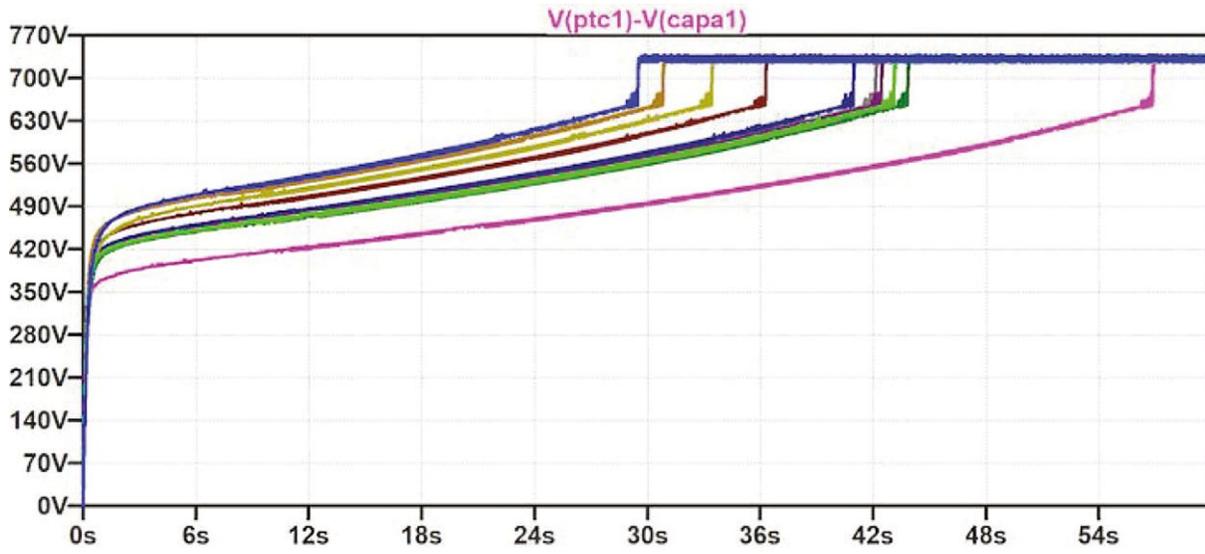
The PTCEL models have three pins: two electrical and one readout for their temperature. The virtual oscilloscope allows for monitoring all PTC temperatures individually.

The R_{25} of the PTCs are provided with a tolerance of $\pm 30\%$. Repetition of the simulation visualizes this tolerance.



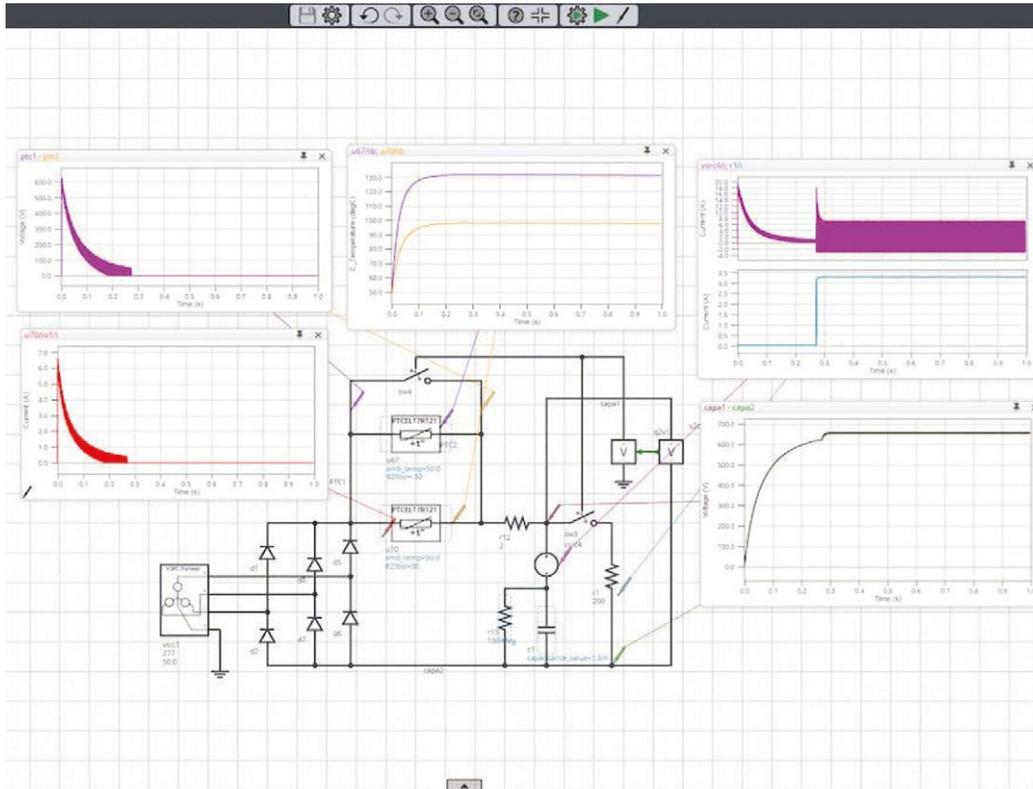






4.2 SIMULATION IN PartQuest™ Explore - PTC OVERVOLTAGE PROTECTION

During the smoothing capacitor charging at SMPS switch-on, a PTC protects a circuit against over-current.



Hardware Components Used in This Project

- 1 x Vishay PTCEL series

Story

A live tunable simulation (VHDL AMS) of a basic SMPS current protection on the PartQuest Explore website, where different parameters can be tuned (smoothing capacitor value, ambient temperature, PTC tolerance, etc.).

This simulation can be exercised at:

- [PartQuest™ Explore | Design, Modeling, Simulation and Analysis](#)
- [3 phase AC SMPS With PTC Network Protection - trip/non trip 070619 | PartQuest™ Explore](#)
- Other models of PTC (inrush current protection) are available there

4.3 PTCEL CURRENT LIMITER (MODELED IN SIMetrix SIMULATOR)

Link

- [PTCEL Current Limiter \(Modeled in SIMETRIX Simulator\) - Hackster.io](#)

The Vishay leaded inrush current limiting PTCEL component can be simulated in SIMetrix / Simplis software, with internal temperature display.

Hardware Components Used in This Project

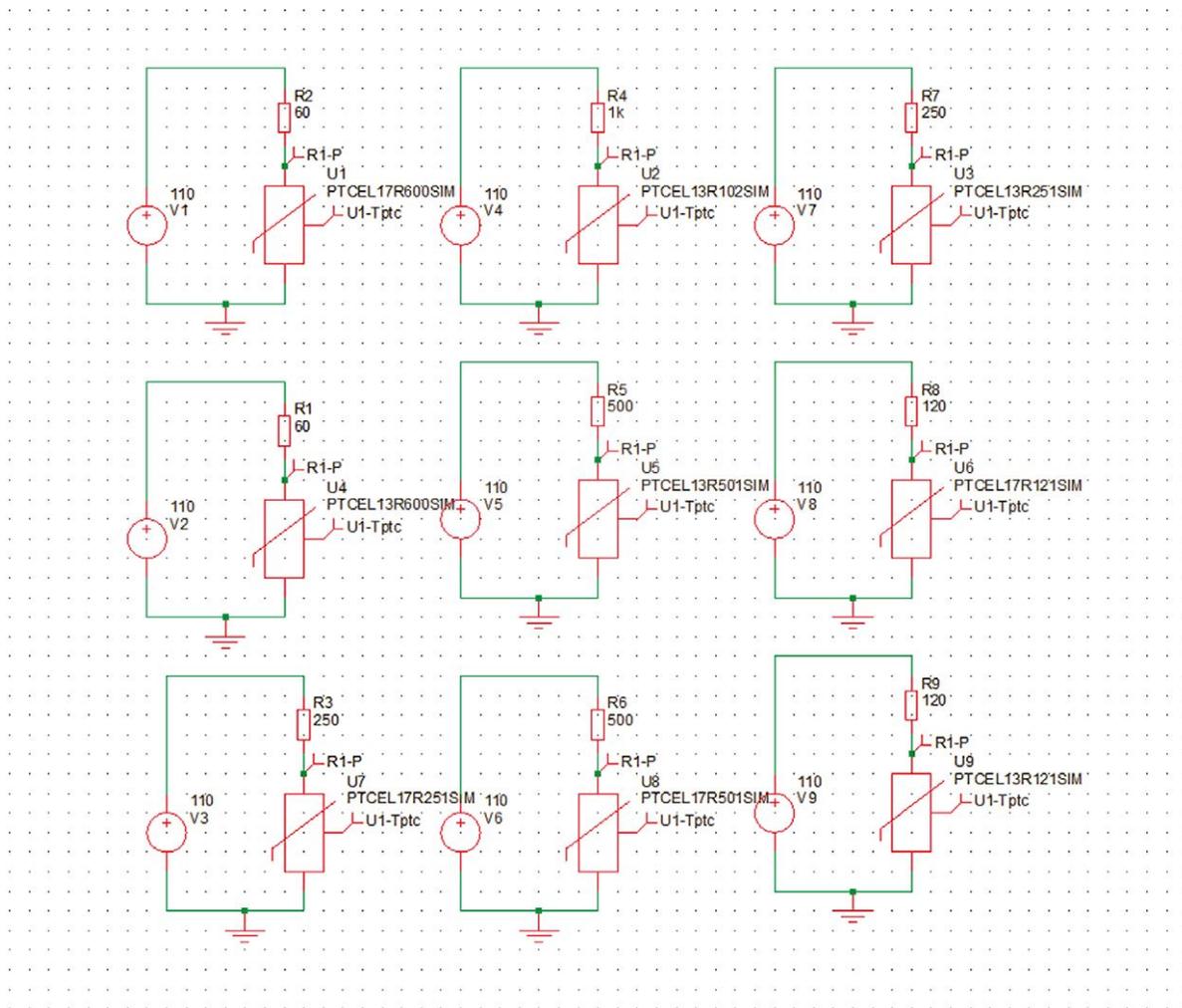
- 1 x [Vishay PTCEL series](#)

Story

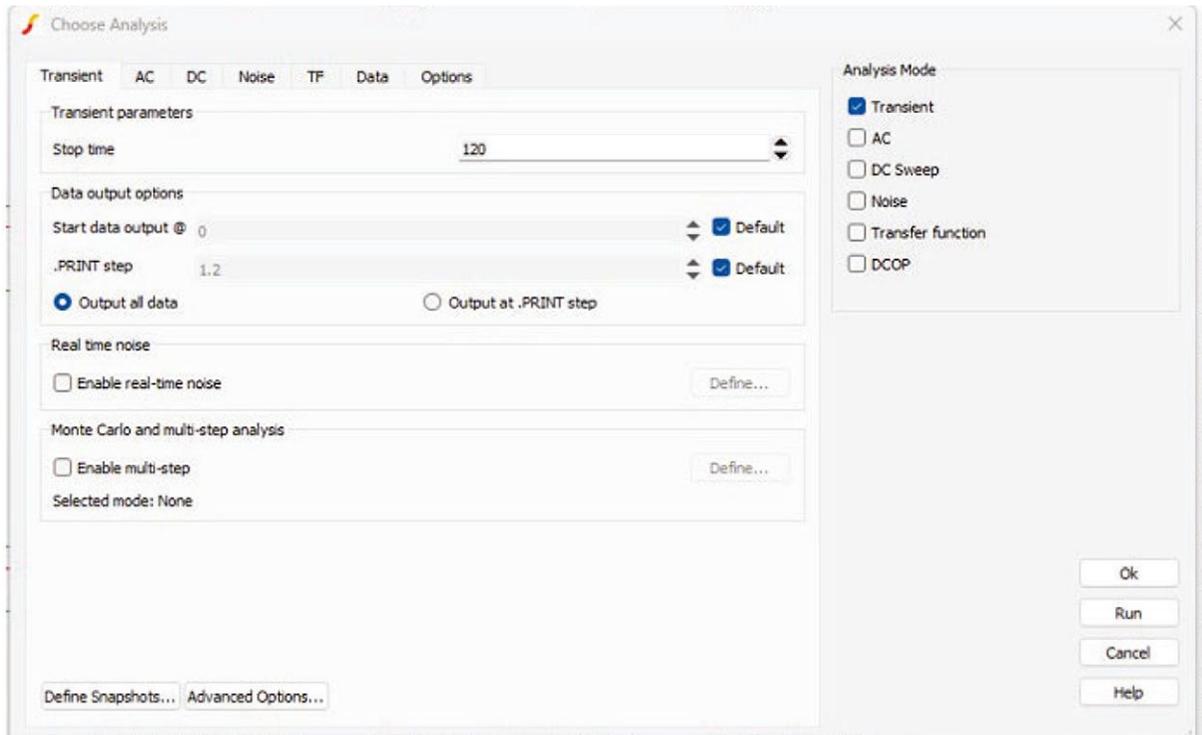
SIMetrix modelling of PTCEL leaded through-hole components, inrush current limiting PTCs. The principle of simulation is explained in the following video.

[Vishay Thermistors. Electronic Simulation Part 3: SIMetrix SPICE Models for PTCEL - YouTube](#)

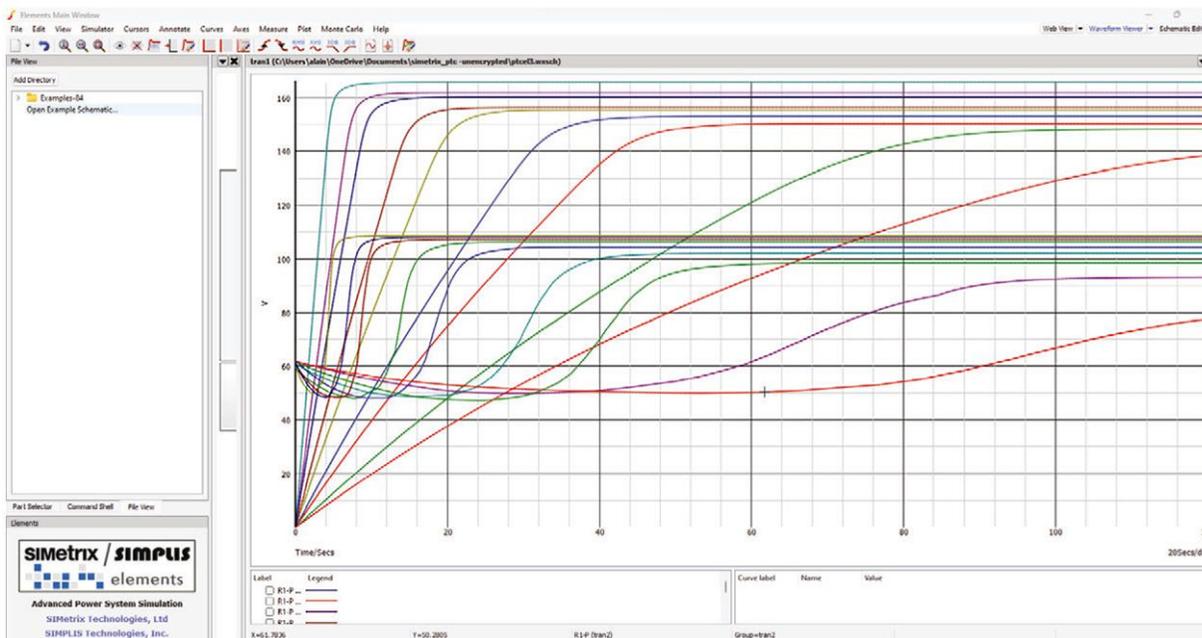
The following circuits can be built in SIMetrix



The simulation parameters are as follows:



and the results as follows:



The models and symbols are encrypted and consequently can only be used with the full version of SIMetrix / Simplis.



5. NTCC200 / NTCC201

Links

- Product page on Vishay website for further info: [NTCC200E4, NTCC300E4 Leadless NTC Thermistor Die Suitable for Wire Bonding | Vishay](#)
- Product page on Vishay website for further info: [NTCC201E4 Enhanced Leadless NTC Thermistor Die Suitable for Wire Bonding | Vishay](#)
- Presentation video: [Vishay - Vishay Enhanced Wire Bondable Die NTCC201 Thermistors](#)
- Simulation: [LTspice Inverter Simulation With Thermal Effects - Hackster.io](#)

LTspice Inverter Simulation With Thermal Effects

An LTspice simulation of a full bridge IGBT schematic with NTC thermistor temperature control and derating above a defined temperature.

Hardware Components Used in This Project

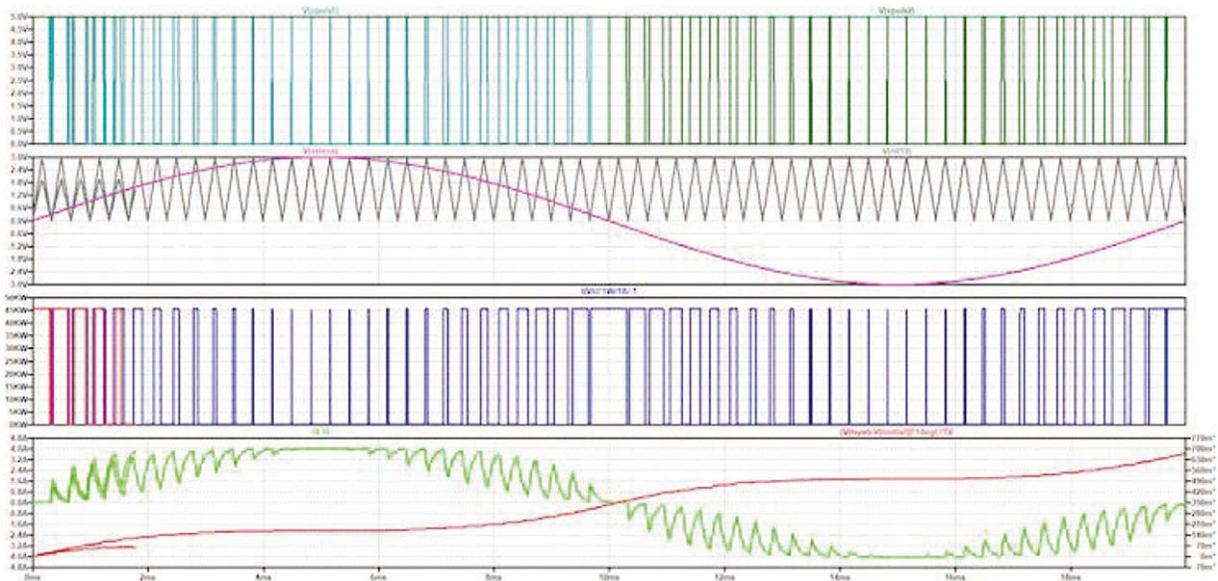
- 1 x Vishay NTCC200E4472FT
- 1 x Infineon IRG4PC50WPBF
- 1 x Analog Devices ADUM4120

Story

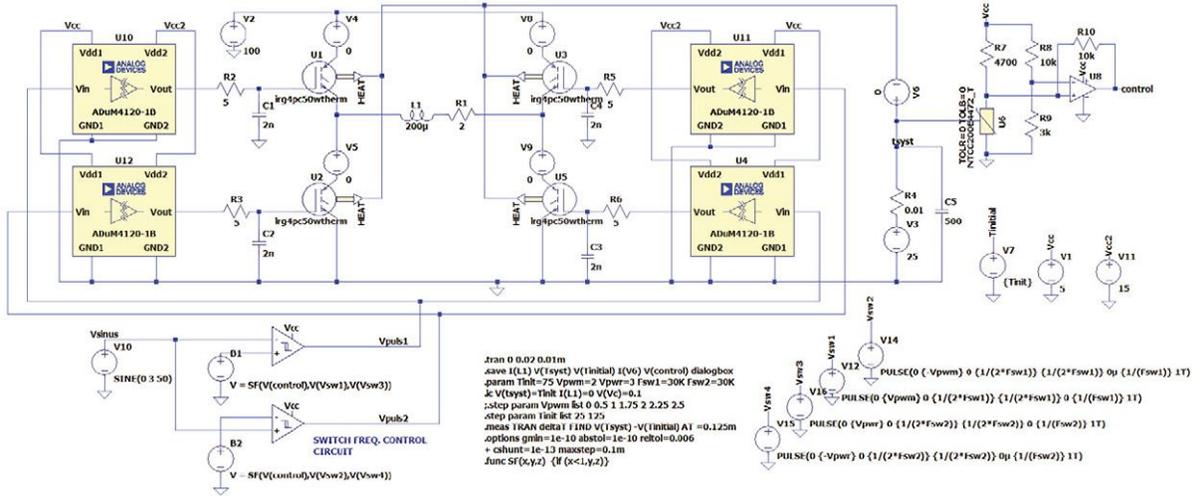
Warning: the following simulation intends only to show how a full IGBT bridge can be simulated with gate drivers and taking account of thermal effects. It is thus intended as an educational example only, without assurances on practical results. The primary goal is to show that when proper models are available, very realistic results of a relatively complex schematic can be obtained within a short time. The strategy exposed for the derating in temperature works on “paper”, but has not been tested in practice and is only intended to develop engineers, intuition.

The models used belong to their respective owners (Analog Devices for the gate drivers; for the Infineon IGBT, consult the netlist provided in the Appendix. The thermistor is a Vishay model).

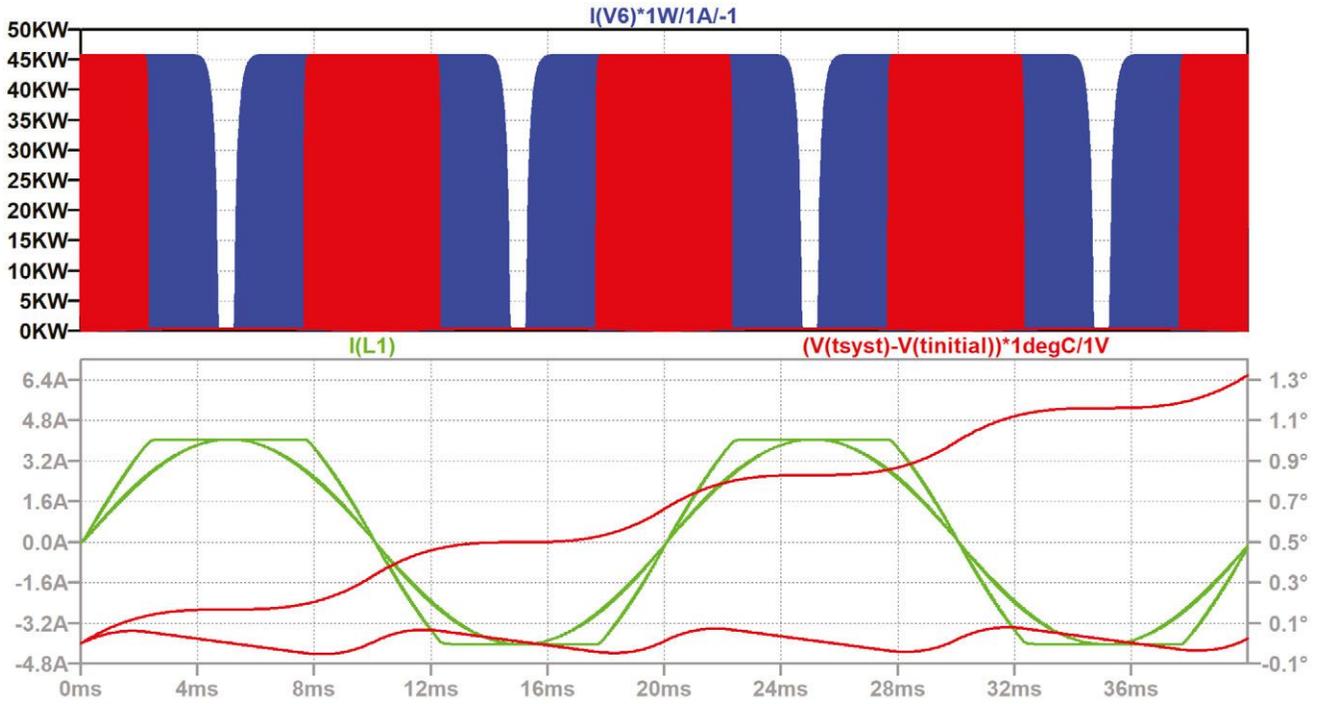
Like most simulations, the results are a bit self-explanatory. The gates are pulsed with a signal resulting from a 50 Hz signal compared to a 30 kHz sawtooth signal. This PWM is better seen at a lower frequency, as presented in the following graph (3 kHz).



At 30 kHz, the modulation is high enough to produce a nearly 50 Hz sinusoidal current into the load.



The IGBT models have been replaced by thermal models that show a supplementary pin, which is the pure heat source produced by the devices. The thermal pins of the IGBTs are all connected to the heatsink and to a voltage source, which is the ambient temperature.



At high temperature, the duty period is reduced: the IGBT switching is derated, which induces less heat into the heatsink and the temperature stabilizes. More information about the simulation can be obtained at edesign.ntc@vishay.com

6. NTCS VISHAY NTC SMD THERMISTORS

Links:

Product page on Vishay website for further info

- [NTCS....E3...SMT SMD NTC Thermistors With Enhanced Stability | Vishay](#)
- [NTCS0402E3.....T SMD 0402, Glass Protected NTC Thermistors | Vishay](#)
- [NTCS0603E3.....T SMD 0603, Glass Protected NTC Thermistors | Vishay](#)
- [NTCS0805E3.....T SMD 0805, Glass Protected NTC Thermistors | Vishay](#)

NTCS SURFACE-MOUNT NTC THERMISTORS

GLASS PROTECTED / NI BARRIER - SN TERMINATIONS

SNAPSHOT

COMPLIANCE TO STANDARDS

- AEC-Q200 qualified
- Sustaining ESD air discharge: 25 kV (HBM)
- UL 1434 certified (File E148885)
- Sulfur-resistant to ASTM B809-95
- RoHS-compliant
- Whisker testing according to JESD 201 A

MULTIPLE DESIGN TOOLS AVAILABLE

- Resistance / temperature curve in "My Vishay NTC Curve" (©Excel)
- Resistor / thermistor network output computation (online)
- SPICE multi-simulator models (PSpice / LTspice / TINA / Multisim / Altium / Simetrix / SaberRD...)

TEMPERATURE SENSING / CONTROL / PROTECTION

NTCS0805

APPLICATIONS

- LED over-current limitation
- Li-ion and Li-polymer over-temperature detection
- Power modules (EV, HEV,...)
- Optoelectronics
- Car infotainment
- Medical equipment

(VERY) STABLE IN CLIMATIC TESTING (OPERATIONAL LIFE)

www.vishay.com

© 2018 VISHAY INTERTECHNOLOGY, INC. ALL RIGHTS RESERVED

thermistor1@vishay.com
MS7613-1805

6.1 LTspice SIMULATION OF HIGH TEMPERATURE LED CURRENT CONTROL WITH ANALOG DEVICES LT3474 USING A RESISTOR / THERMISTOR NETWORK

Links

- Video: [Vishay - LTspice Simulation of LED Current Control with Temperature Sensing by NTC](#)
- Simulation: [LTspice Simulation of LED Current Control - Hackster.io](#)

Hardware Components Used in This Project

- 1 x Vishay NTCS0805E3223JHT
- 1 x Analog Devices LT3474

Story: How to Compensate LED Current Variation With an NTC / Resistor Network

We are going to present the principles of LED current control using an LT3474 [3], which is a step-down 1 A current driver, and a Vishay surface-mount NTC thermistor. The LED thermal SPICE model is like the thermal models presented in the first series of articles [1], completed by the SPICE model principles developed in practical lightning design with LEDs [2]. The whole schematic is detailed in Fig. 5. In fact, it is an extension of the PWM-sourced LTspice LT3474 Macromodel test fixture [3], with several new features. The thermal modeling of the LED (including the heatsink) is made with a resistor R6 and a capacitor C4, to which the power dissipated in the LED is applied (the B1 analog behavior model source), as well as a voltage source V4 equivalent to the ambient temperature. The voltage at the node THERM represents the LED system's temperature and is sensed by a thermistor (a 22 kΩ surface-mount Vishay NTC). This thermistor is a part of a resistor network [4] connected to the ADJ pin of the LT3474 and will tune down the pulse current into the LED. To complete the LED modeling, we use two other analog behavior sources:

- B2: a negative voltage source representing the decrease in forward voltage of 3 mV/°C in series with the LED
- B3: models the optical output of the LED (as defined in *Practical Lighting Design with LEDs* [2], but not directly used in the thermal simulation of this article)

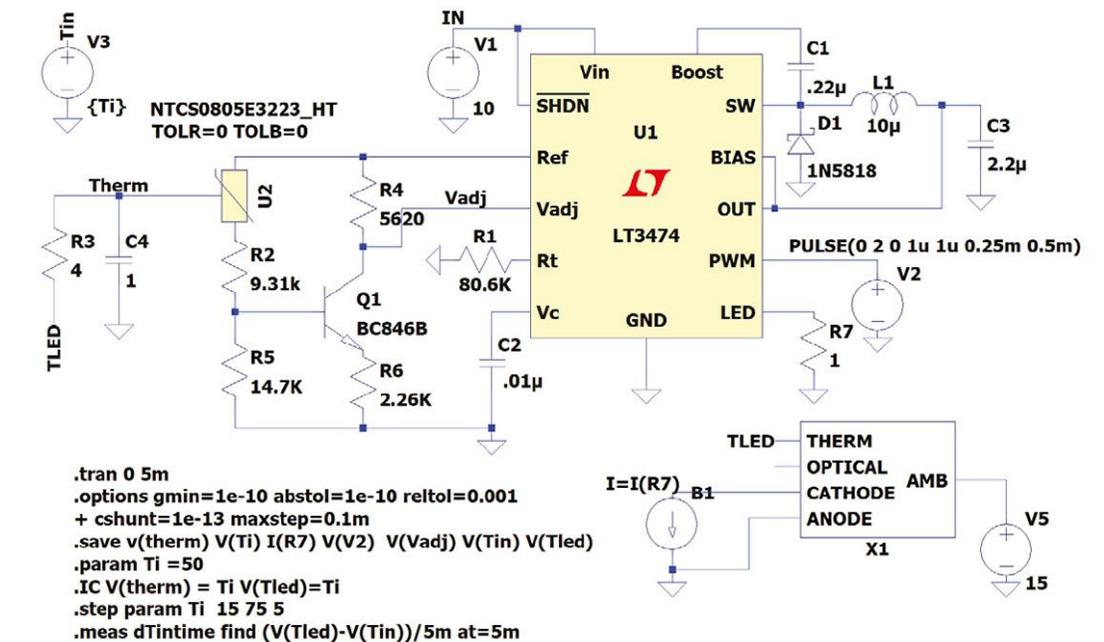


Fig. 5

Now a simulation of a 2 kHz PWM source in the circuit of Fig. 5 is performed with a ratio (real time / computation time) of 50 μ s/s, and it could take as long as 24 hours for a device like an LED to reach a steady state after a few seconds of real time. For memory, most simulations should have a practical duration of a few seconds for computation. That is what makes simulation interesting before testing hardware. If you need to spend hours staring at your computer's screen (for one simulation), the whole thing will not make sense anymore.

Getting results within a reasonable time frame was accomplished as follows: instead of fixing the initial temperature of the LED system at +25 °C and waiting forever, I swept the initial temperature of the LED through a given temperature range (+25 °C to +100 °C). After 2 ms of switching on the PWM source, we looked to see if the temperature was still increasing, remained stable, or eventually decreased. The temperature's initial rate of change after 2 ms is represented in Fig. 6.

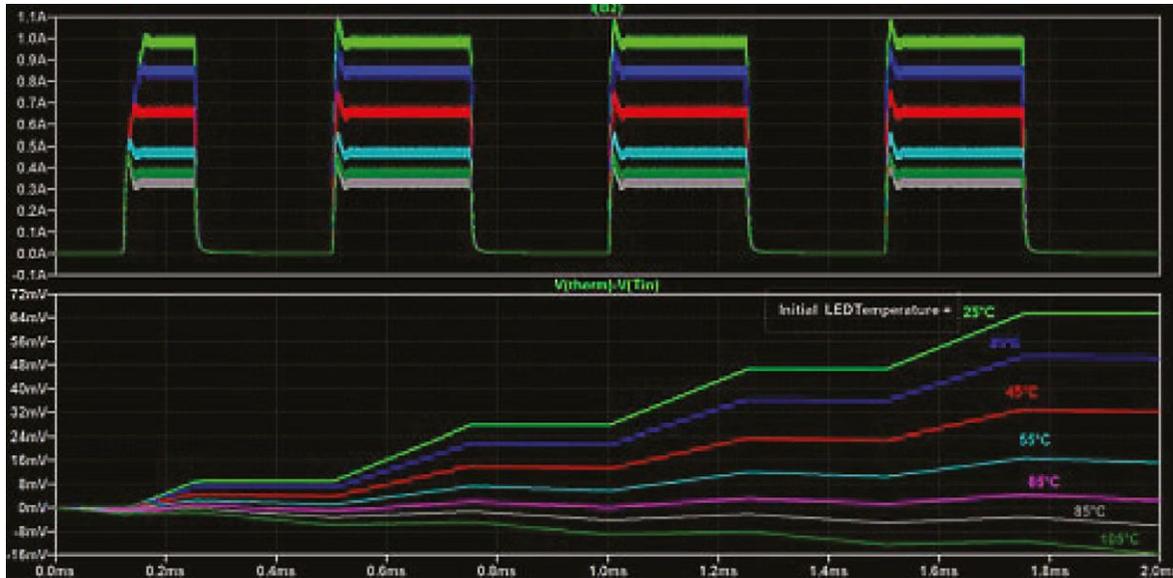
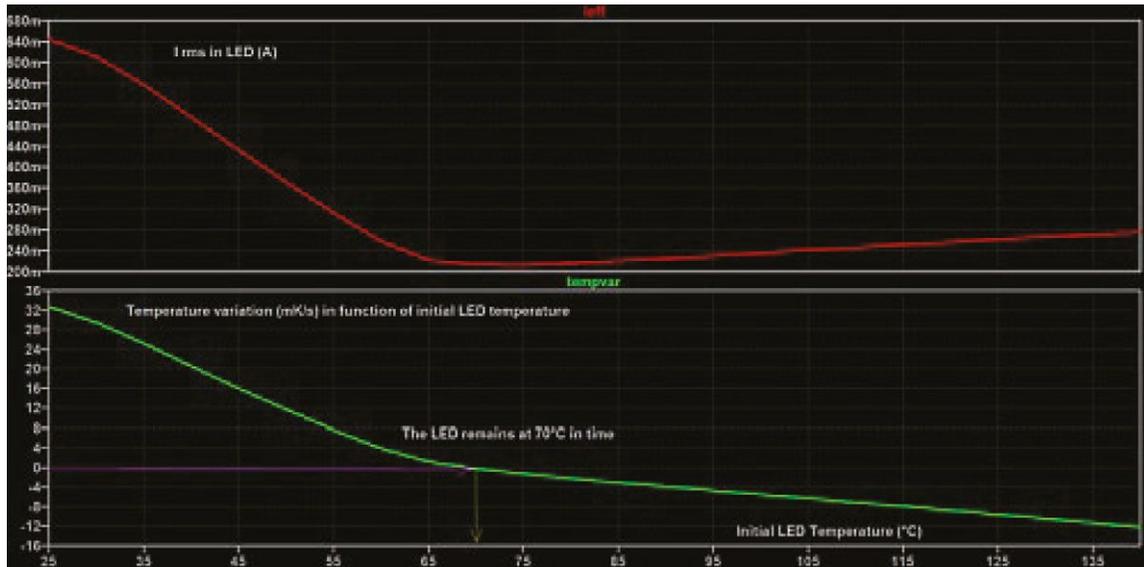


Fig. 6

For an initial temperature of +25 °C, the pulsed current amplitude is 1 A and the LED begins to warm up (Fig. 6: green curve of lower pane). When the fixed initial temperature rises (+35 °C in blue, then +45 °C in red, etc.), the current is progressively tuned down and the initial temperature slope decreases.

Around +65 °C (curve in magenta), the temperature remains flat after 2 ms. Above +65 °C, the LED begins to cool down, leading to a re-increasing of the current. Therefore, we know that we have detected an equilibrium point there, which we can determine precisely by measuring this initial temperature rate of rise, extracting it from the "SPICE Log File", and plotting it (as shown in Fig. 7). We measure the RMS value of the current in the LED as well.


Fig. 7

We see that a zero rate of temperature rise is reached for a temperature of +70 °C. At this temperature, the current into the LED reaches a minimum. This is thus the steady state of the system.

How can we apply the principles of this simulation in the real world? In practice, the thermal parameters of the LED and the heatsink must be adapted. Therefore, the user will need some preliminary measurements. It is possible to apply a standard current of 1 A DC, for example, and to measure the speed of temperature rise. This will give the response time (R6C4 constant) of the LED system. The power dissipated divided by the temperature increase during this 1 A application will give the dissipation coefficient.

Then the user can fix the maximum temperature / power at which the LED can work and adapt the fixed resistor / thermistor network to achieve this goal. Fig. 8 shows that it is possible to adapt the steady state of one defined LED from +40 °C to +70 °C by changing the value of R3.

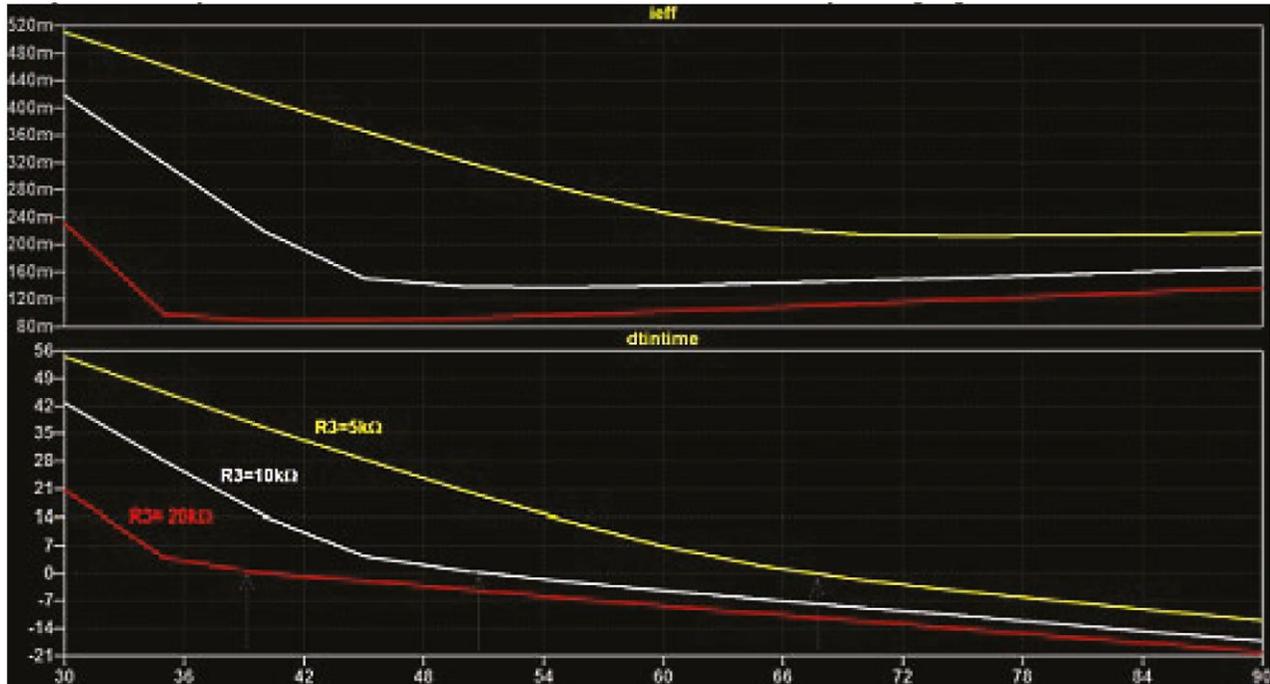


Fig. 8 - Higher Pane: I_{RMS} in LED / Lower Pane: Temperature Increase (mK/s) as a Function of Initial LED Temperature ($^{\circ}C$)

In a similar way, the user will now be able to study the influence of the other fixed resistors, the ambient tolerance, and more before finalizing the design.

To see a demo video of this simulation, please visit [this link](#).

LTspice is a registered trademark of Analog Devices, Inc.

References

- [1] [Old-School Analog Temperature Control Circuits Solved With Modern LTspice Thermistor Dynamic Models. Part 2](#)
- [2] [Practical Lightning Design with LEDs 1st Edition](#) by Ron Lenk (Author), Carl Lenk (Author) ISBN-13: 978-0470612798
- [3] Linear Technology: [LT3474](#) Step Down 1A LED DRIVER, web, Aug 2017
- [4] Analog Circuit Design Volume Three: Design Note Collection by Bob Dobkin (Author), John Hamburger (Author) ISBN-13: 978-0128000014

6.2. NTC SMD FOR STRAIN GAGE COMPENSATION

Links

- Video: [Vishay - Vishay Thermistors Electronic Simulation Part 5: Temp Compensation of Strain Gage Resistive Sensor](#)
- Simulation: [Piezoelectric Strain Gage Temperature Compensation With NTC - Hackster.io](#)

LTSpice simulation of the temperature compensation of a strain gage bridge with a fixed Vishay resistor / NTC thermistor network.

Hardware Components Used in This Project

- 1 x [Vishay NTCS0603 SMD thermistor 10 kΩ, ± 1 %](#)
- 1 x Vishay TNPW06031K2
- 1 x Vishay TNPW0603100R

Story: LTSpice Simulation of Full-Bridge Piezoelectric Resistive Strain Gage Sensors With Vishay NTC Thermistors / Resistor Network Compensation

A simple SPICE model of such a gage was found at the following URL:

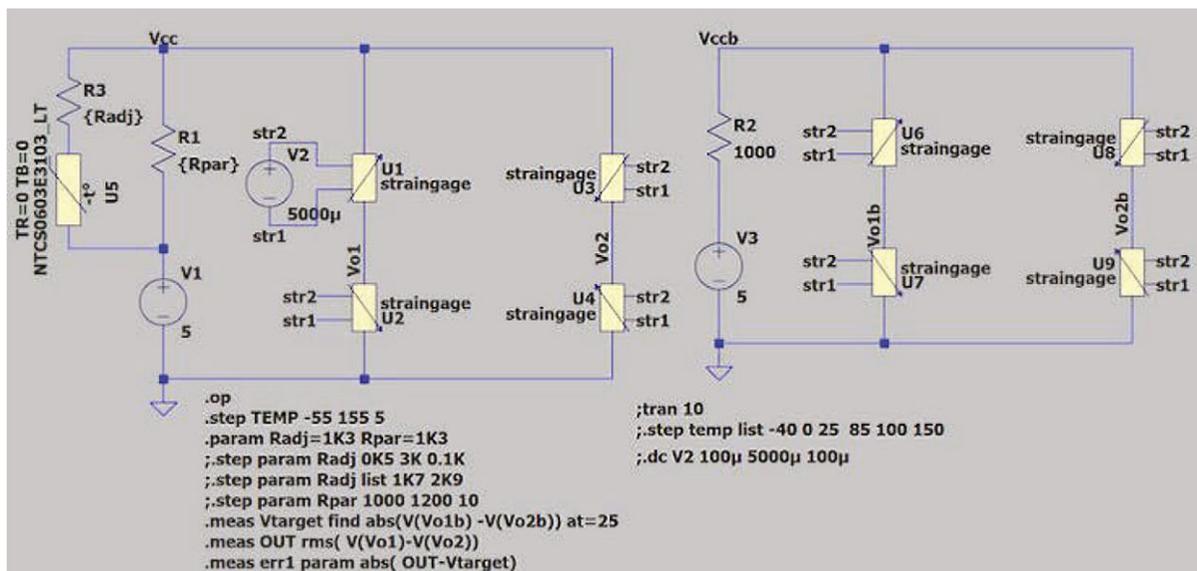
[\(PDF\) A SPICE model of piezoresistive strain gages using engineering parameters for signal conditioning circuits design \(researchgate.net\)](#)

This SPICE model is scalable to all resistive piezoelectric sensors by adapting resistance, TCR (temperature coefficient of resistance), GF (gage factor), and TCGF (temperature coefficient of gage factor). The next lines are a snippet of the beginning of the subckt netlist (available as an Appendix to this project).

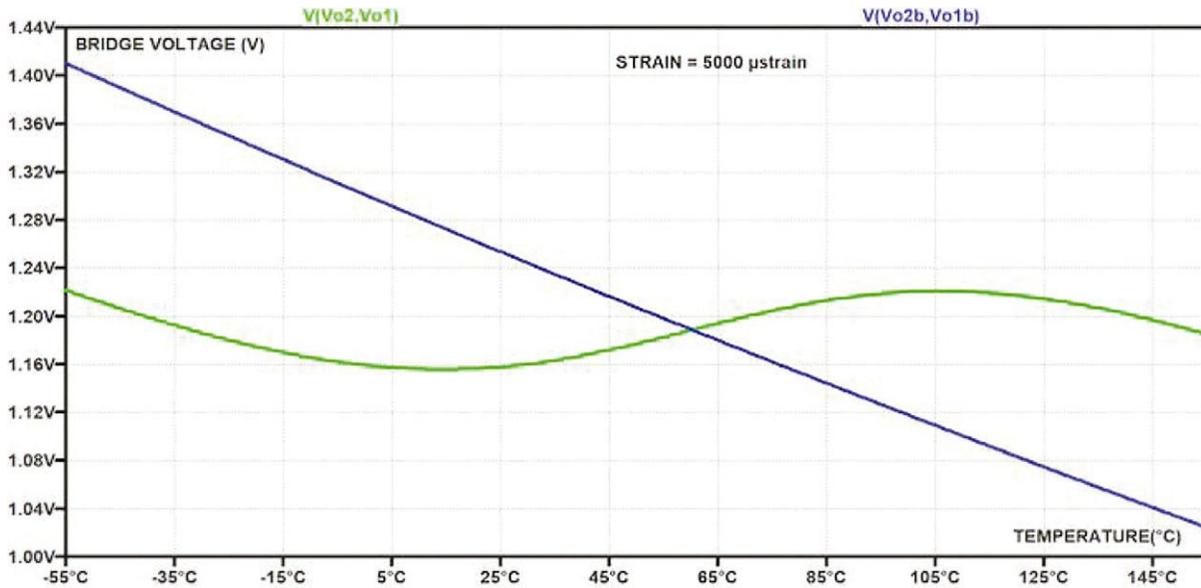
- Simple model for piezoresistive strain gages
- The subcircuit strain gage simulates the behavior of a strain gage
- Input controls (1 micro strain = 1 microvolt)
- Output: floating impedance (pseudo-component)

.SUBCKT strain gage 1 2 8 9 params: R0 = 1000, TCGF = -2.0E-3, TCR = 1e-3 GF0 = 100

So it is now possible to place a full piezoresistive bridge into a bias circuit, where the bridge voltage will be tuned to compensate nearly exactly the output in a constant way as a function of the temperature.

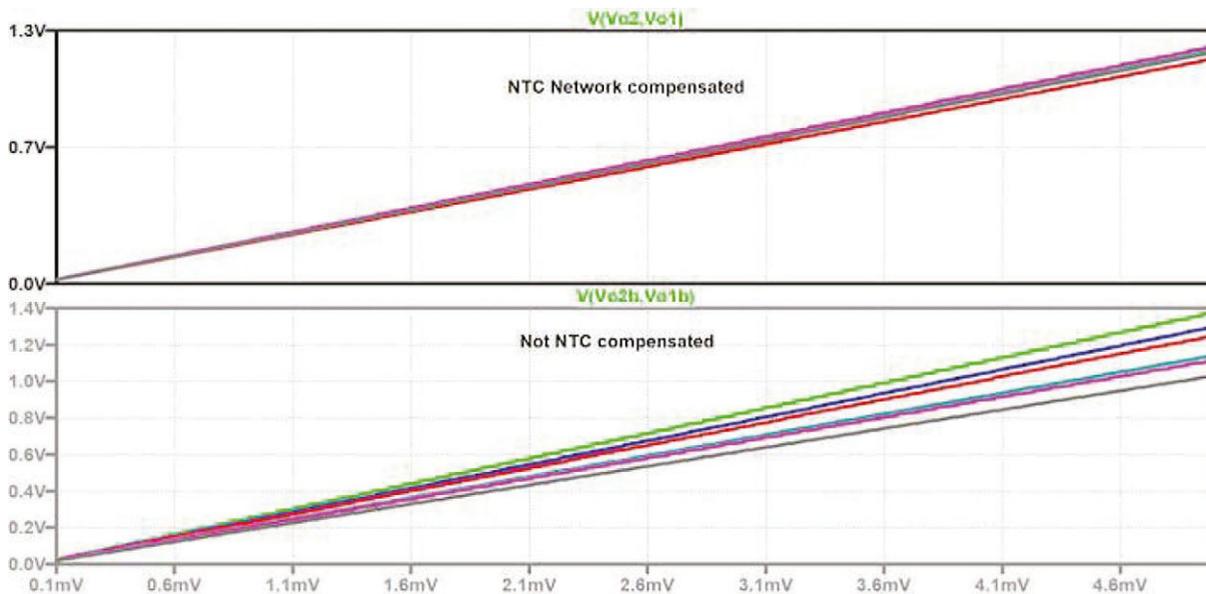


Schematics



Bridge Voltage Output vs. Temperature

By choosing the most appropriate device among the different possible Vishay thermistors and sweeping the values of the resistors of the compensation network, it is possible to stabilize the bridge voltage for a given strain.



Bridge Output vs. Strain (NTC Compensated Above) and NOT (Under)

7. MINI CHIP LEADED THROUGH-HOLE NTCLE213

Product page on Vishay website for further info: [NTCLE213E3 NTC Thermistor, Epoxy Coated Mini Sensor | Vishay](#)

7.1 APPLICATION SIMULATION: THERMOELECTRIC ELEMENT CONTROL WITH ADI ADN8834

Link

- [ADN8834 simulation in action with TEC and thermistor - Hackster.io](#)

LTspice simulation of a thermoelectric element control with Analog Devices ADN8834 controller and Vishay thermistor.

Hardware components used in this project

- 1 x Analog Devices ADN8834
- 1 x [Vishay Thermistor NTCLE213E3103](#)

Story

An LTspice simulation of laser module cooling a Peltier element with the help of the Analog Devices ADN8834 controller and with input given by Vishay’s NTCLE203 / NTCLE213 10 kΩ thermistor.

The total application simulation circuit is reprinted in Fig. 9.

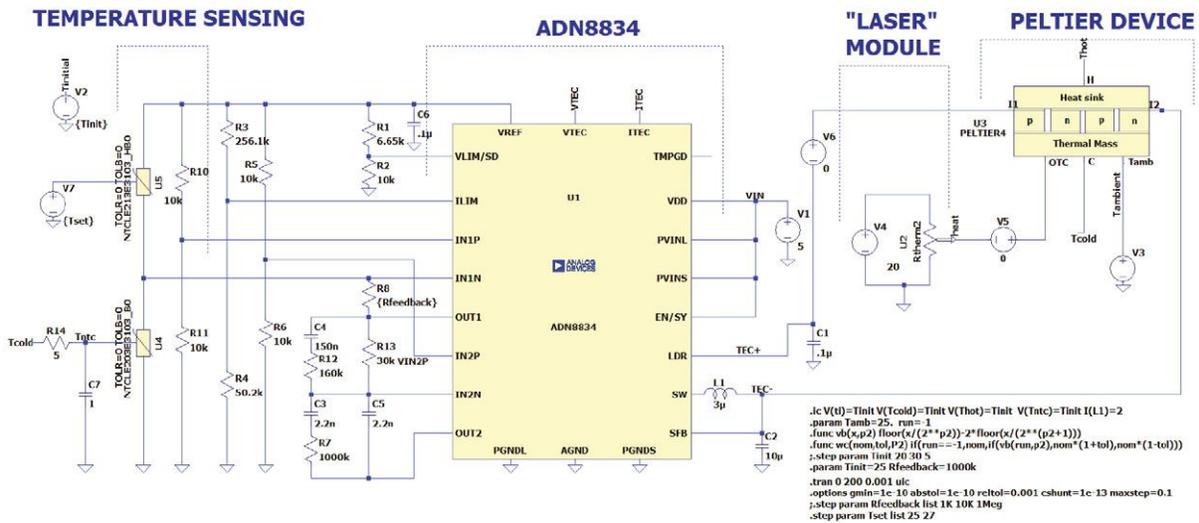


Fig. 9

We are going to detail the different models of these circuits.

1) Internal Structure of Peltier

The model of this project is copied from the following reference:

SPICE model of thermoelectric elements including thermal effects

February 2000 Conference Record - IEEE Instrumentation and Measurement Technology Conference 2: 1019 - 1023 vol. 2

DOI: 10.1109/IMTC.2000.848895 Source: IEEE Xplore

Conference: Instrumentation and Measurement Technology Conference, 2000. IMTC 2000.

Proceedings of the 17th IEEE Volume: 2

The supplied netlist for the Peltier has been recreated in LTspice and can be seen in Fig. 10. You will find in this simulation (present in the archive to download with this project) all the parameters that influence the thermal behavior of the TE element. This will allow the user to calibrate the Peltier parameters (thermal mass, heatsink capacity) by comparison with cold and hot temperature measurements.

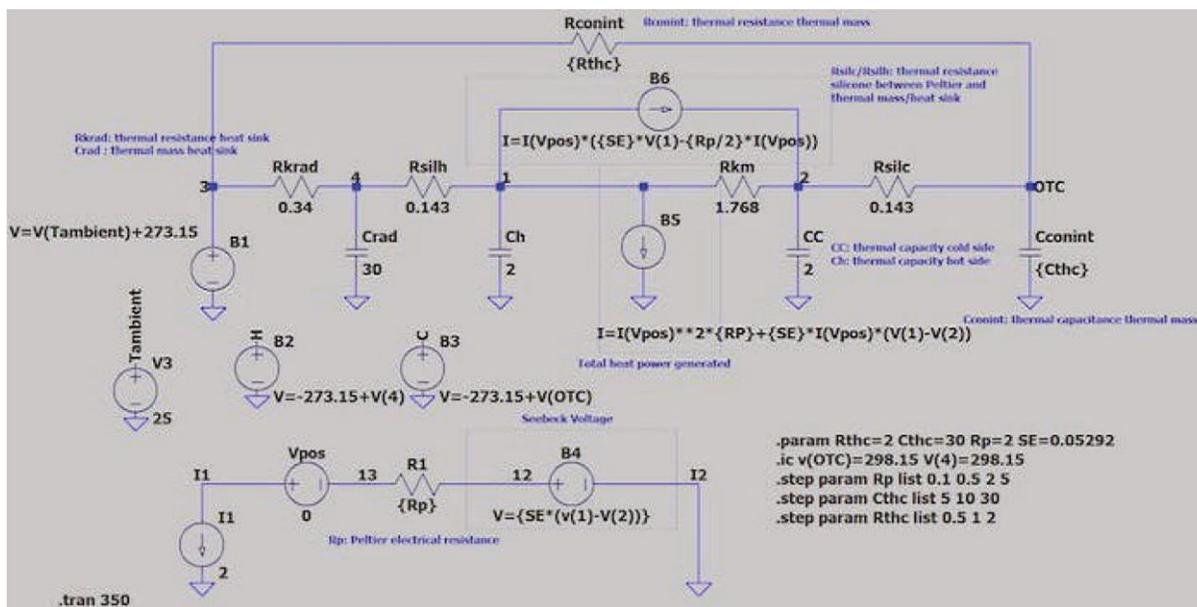
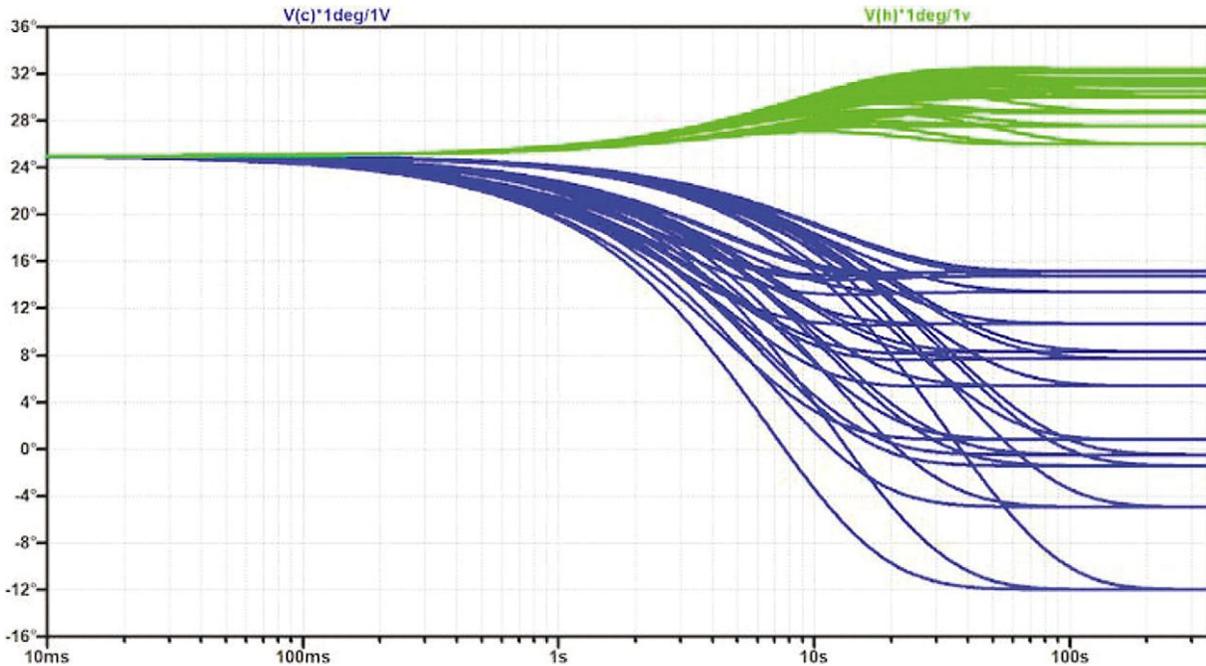


Fig. 10


Fig. 11

2) The Thermistor NTCLE213 / NTCLE203

The precise thermistor characteristics are defined in the file.

The parameter that is application-dependent is the response time, which depends upon the used medium (air) and if air is stirred or not (NTCL213:5s in stirred air / 9 s in still air for example).

3) The Laser Source

This is in fact a heat source directly communicated to the thermal mass of the Peltier. You will need to evaluate how many watts are produced and adjust the current in the source V5 to be equal to this power (equivalence W and I).

4) The Total Simulation With the ADN8834

Fortunately, a SPICE model is available directly from ADI LTspice XVII and the application circuit is reprinted from the datasheet. The advantage is that we are going to be able to test the influence of any parameter of any element, in order to develop the intuition of the application.

In Fig. 12, we perform a simulation where the initial temperature is either lower or higher than the set temperature. We see that in any case, the reached temperature always converges toward the set temperature.

In Fig. 13 / Fig. 14, we sweep the feedback resistor value (R_{feedback}) to visualize the influence on the thermal stability. The higher the value of this resistor, the lower the temperature swing is.

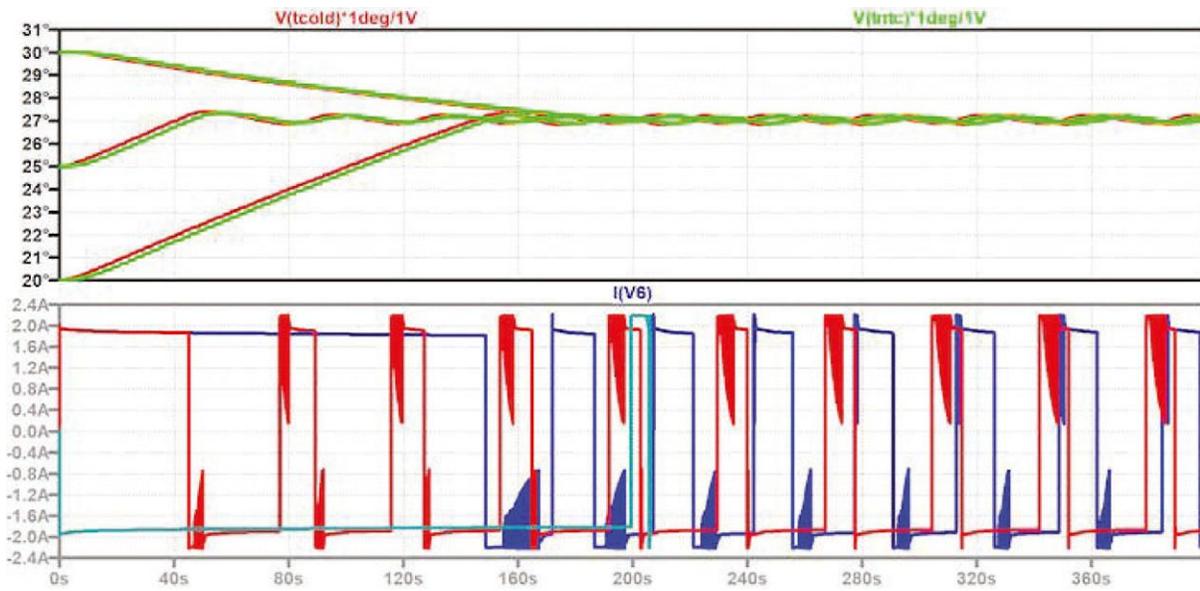


Fig. 12

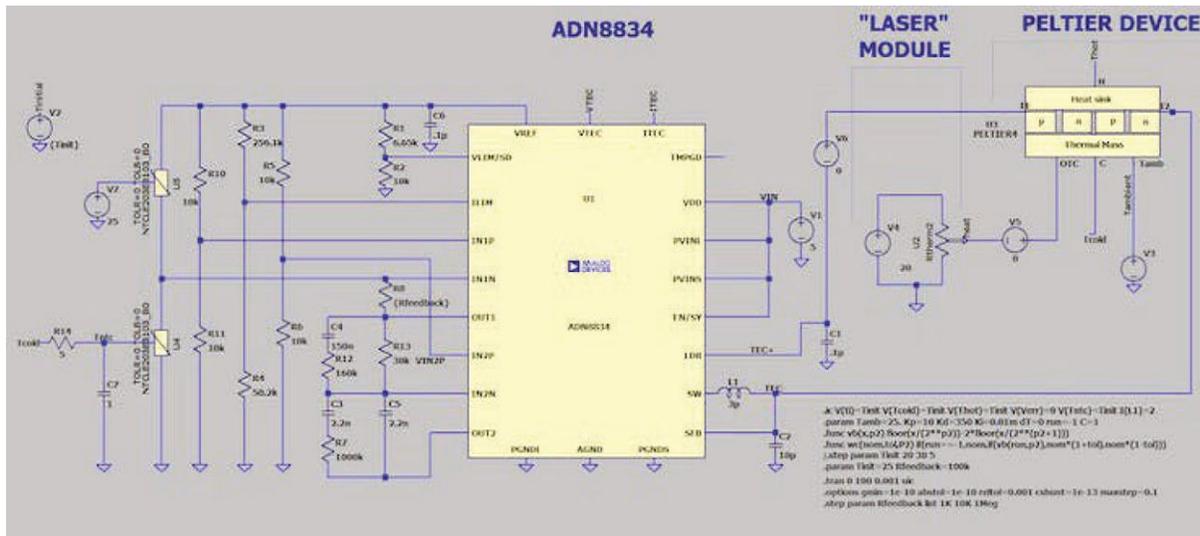
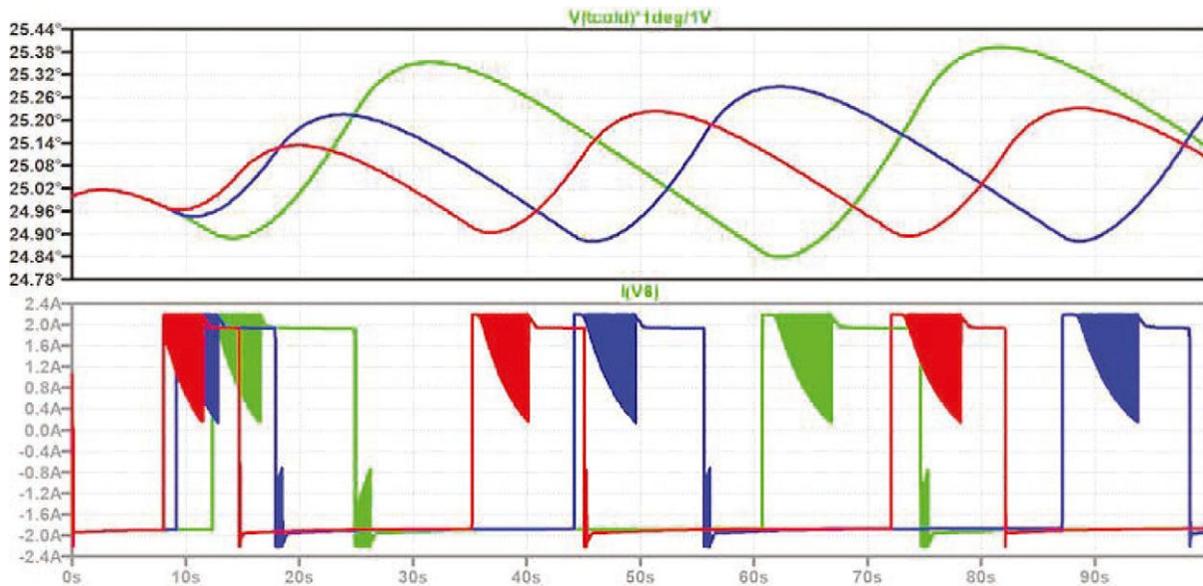


Fig. 13


Fig. 14

In Fig. 15 and Fig. 16, we visualize the influence of set temperature value T_{set} (+25 °C or +27 °C).

You can fix this value by means of a virtual voltage fed to the reference thermistor: the ADN8834 will flawlessly keep the thermal mass at the targeted T_{set} .

In Fig. 17, the influence of ambient temperature changes per dwells: we see that the switching frequency is inversely proportional to the ambient temperature.

In Fig. 18, we see what happens to the thermal loop when the thermistor response time decreases to very low values (temperature swing decreases slightly).

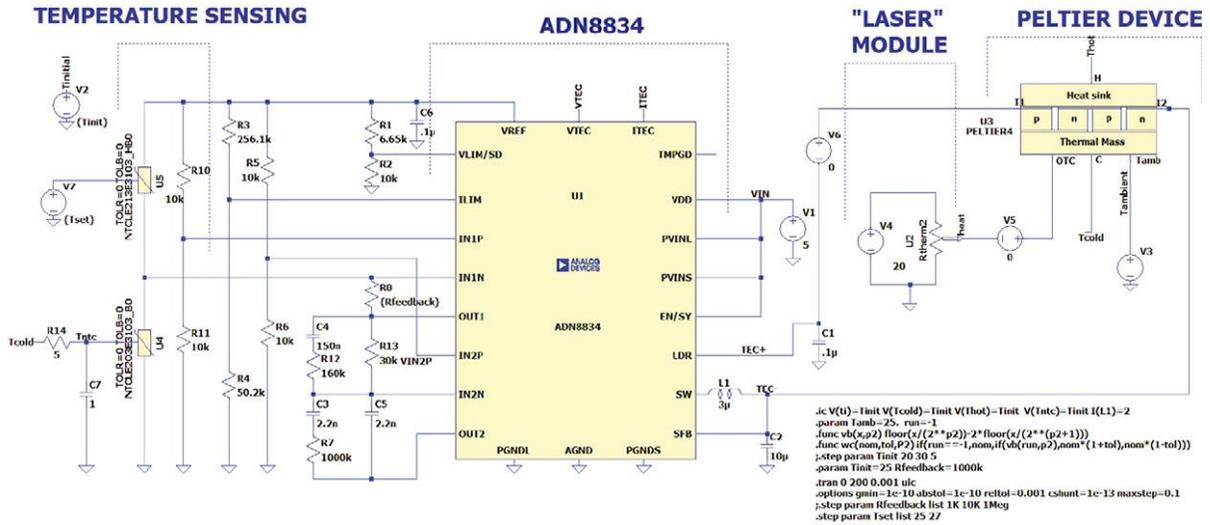


Fig. 15

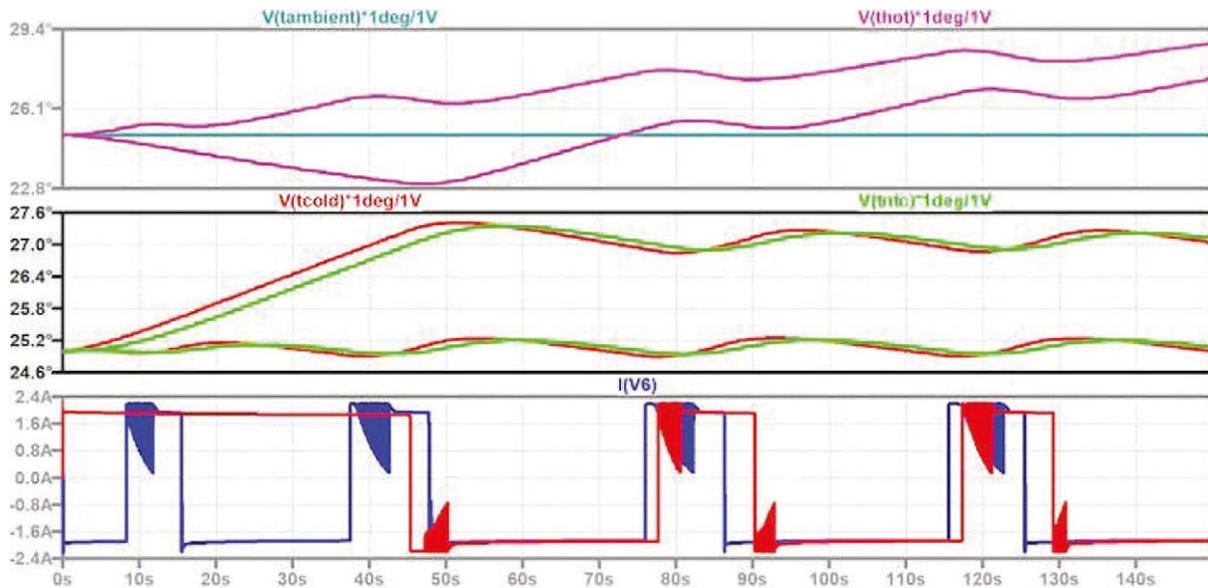


Fig. 16

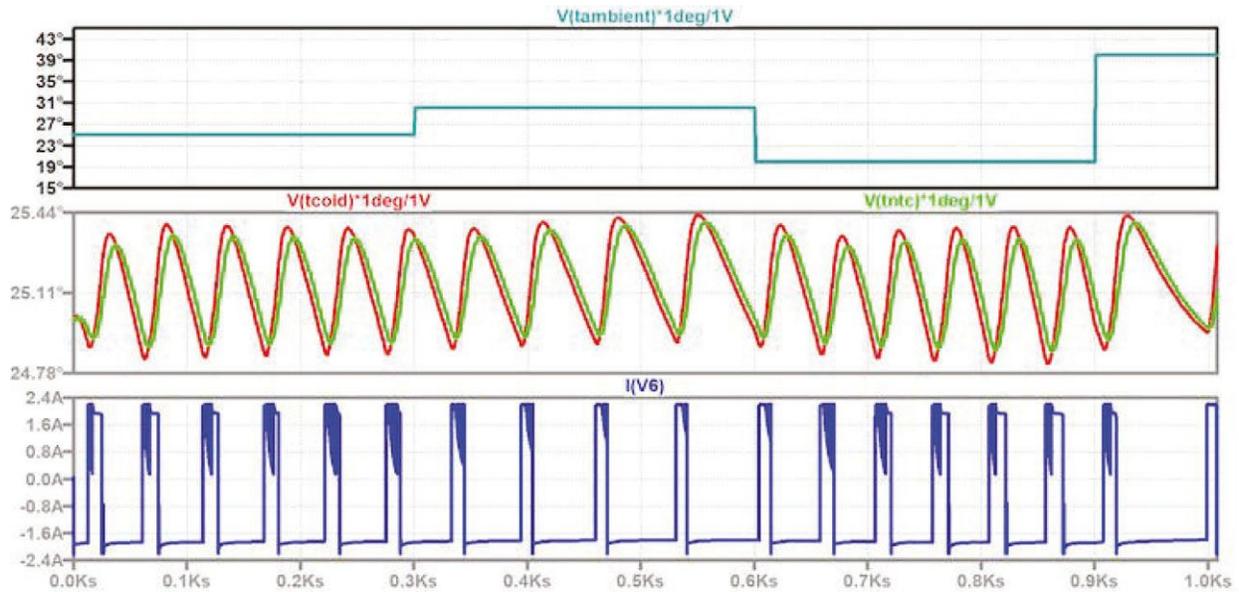
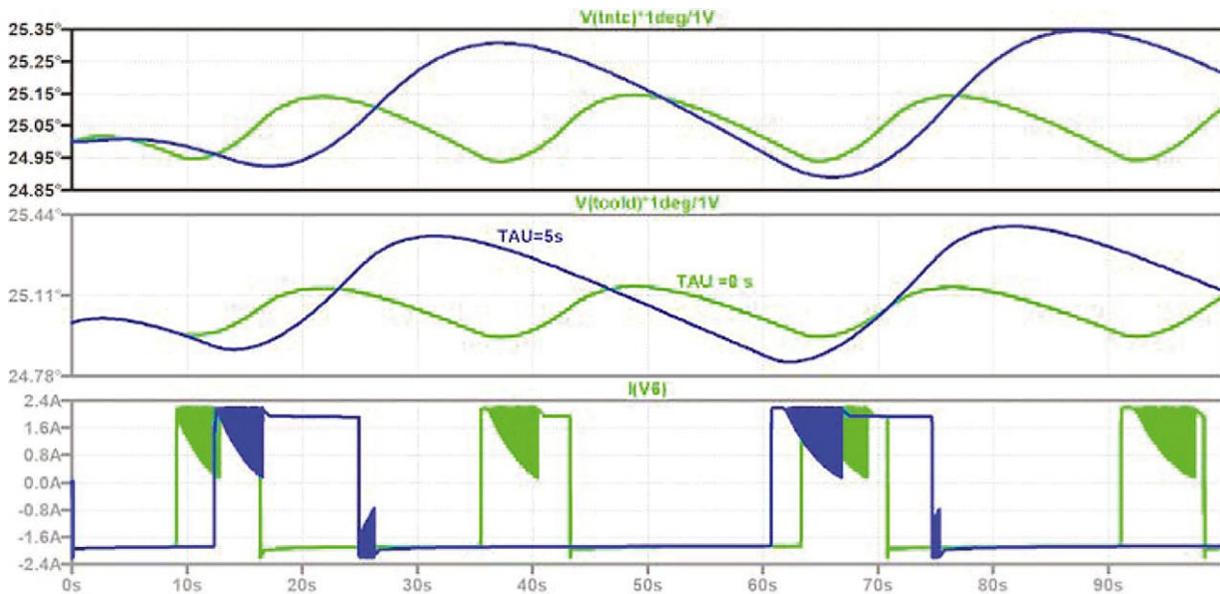

Fig. 17

Fig. 18

Fig. 19 / Fig. 20 show the influence of the heat power of the laser on the circuit behavior, together with preceding ambient temperature changes.

With laser power: above 1000 s, when the heat of the laser is on, and the ambient is too high, the Peltier is always on but cannot cool down the thermal mass to the target.

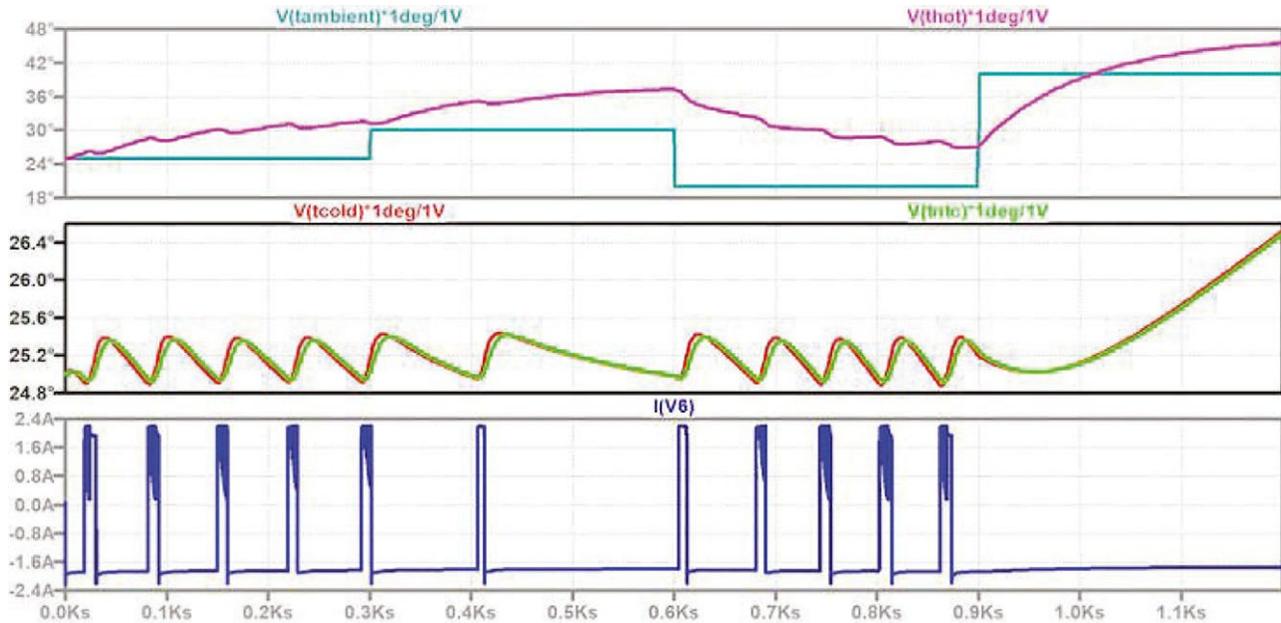


Fig. 19

Without laser power

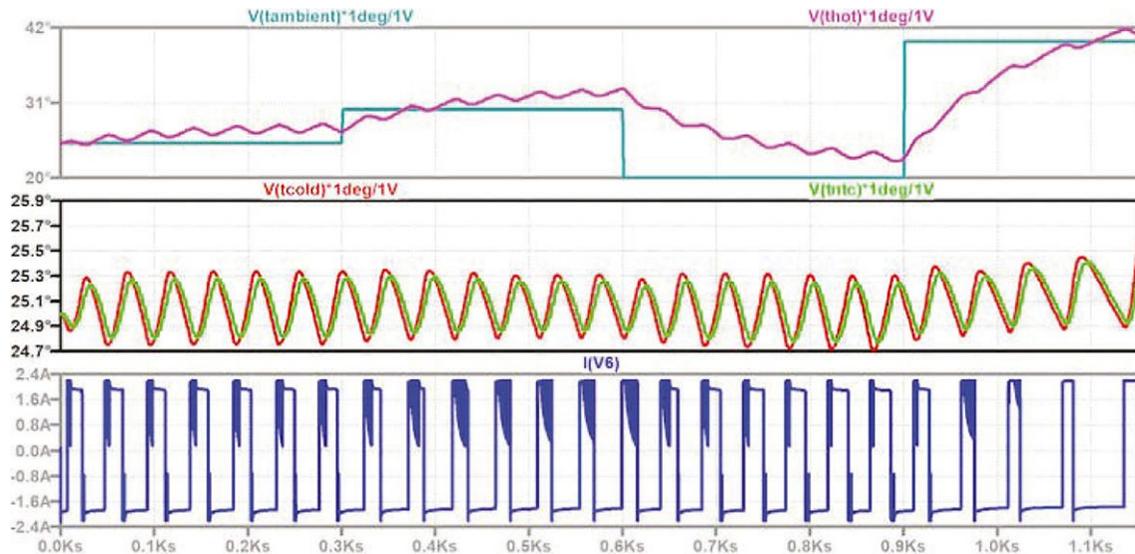


Fig. 20

Conclusion of this project: such a simulation is the ideal (and inexpensive) way to prepare experiments and will give many results to be compared later with practical results.

7.2. SIMULATION OF TEMPERATURE CONTROL WITH LTC1040

Links

- [Temperature Control With LTC1040 - Hackster.io](#)
- Video: www.youtube.com/watch?v=ozGk2PGVSOs&t=21s

Story

The first circuit will use an LTC1040 electronic thermostat with an NTC temperature sensor (NTCLE203 or NTCLE213). The circuit is shown in Fig. 21.

Complete Heating/Cooling Automatic Thermostat with LTC1040 and NTC Thermistor (driven by voltage as temperature)

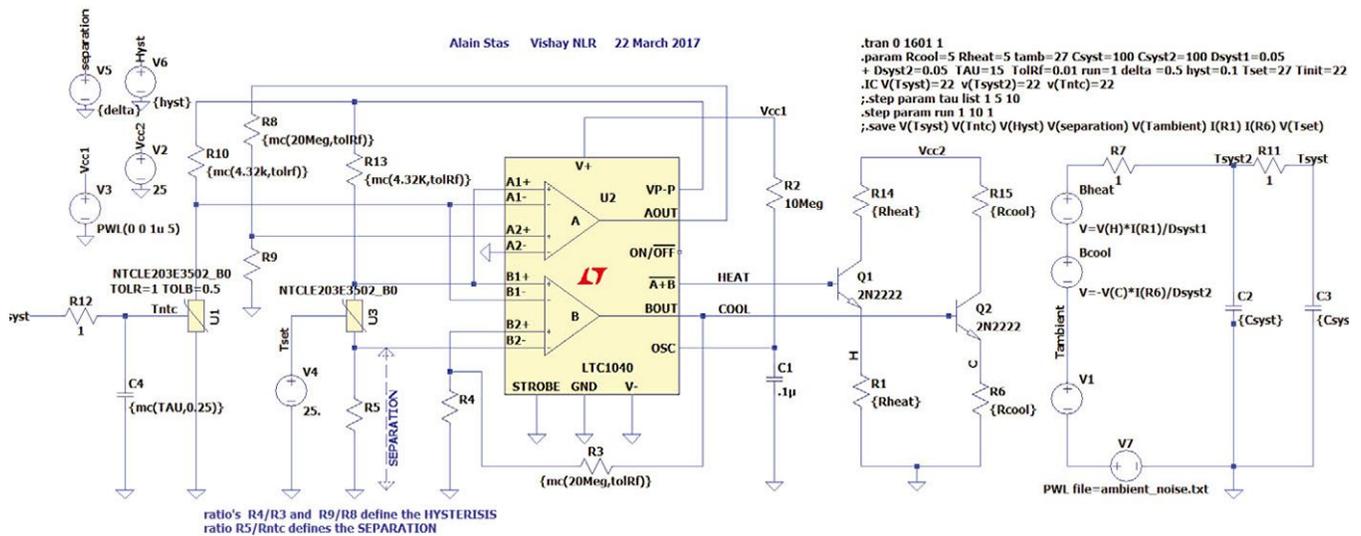
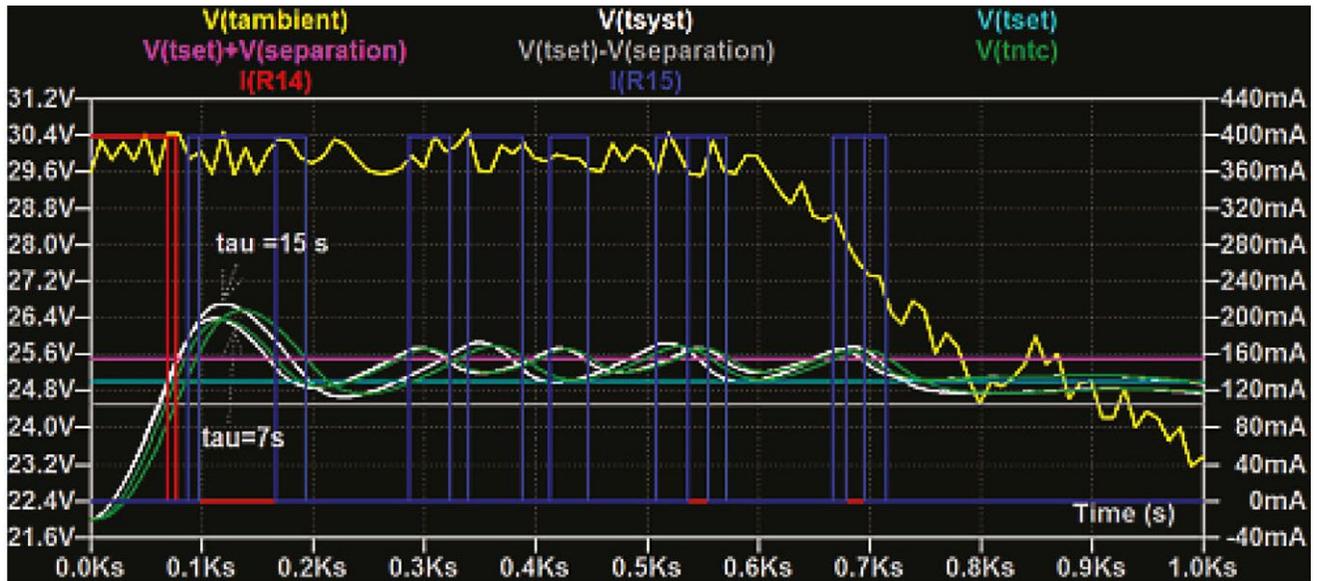


Fig. 21

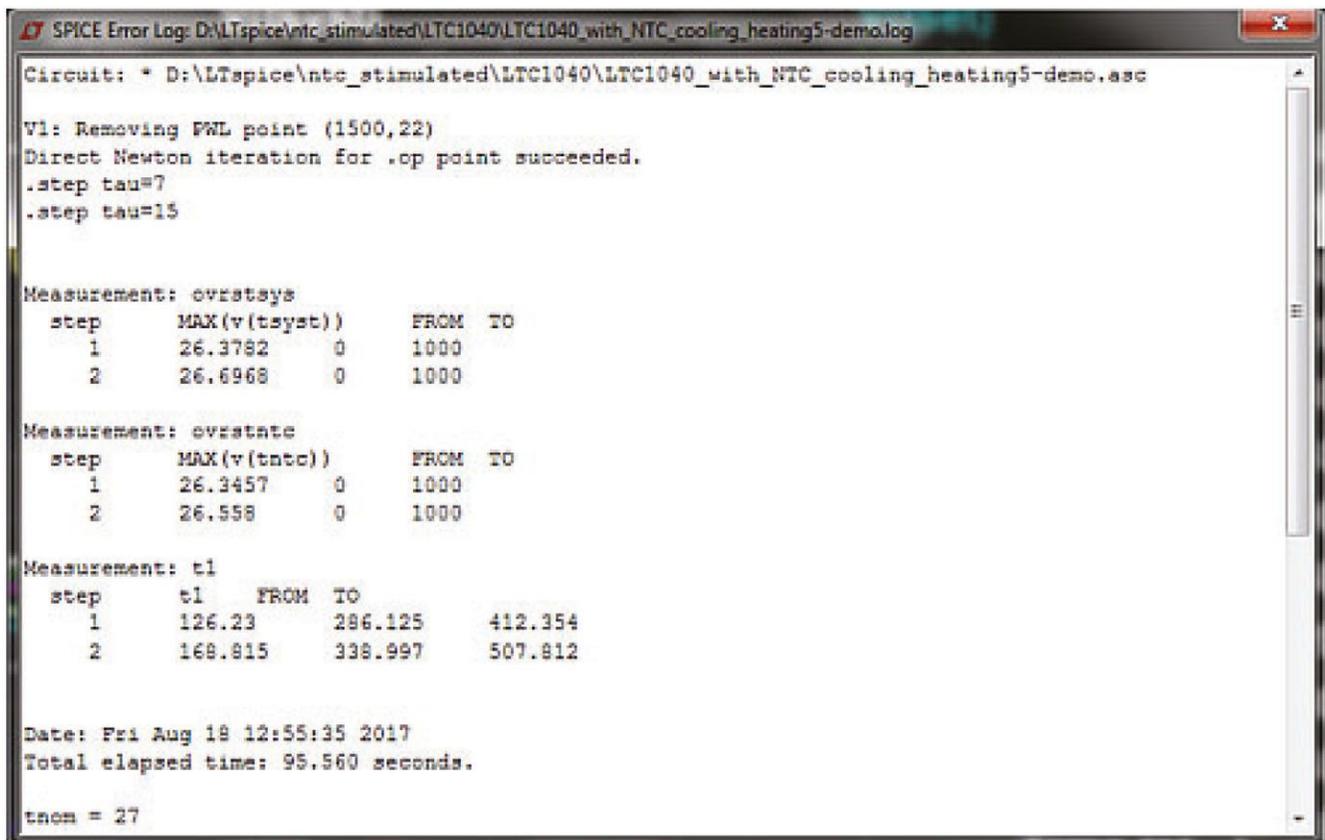
If you want to see the live emulation of this circuit simulation, a five-minute video can be viewed [here](#).

In a nutshell, the simulation (performed according to a Monte Carlo analysis) shows how the LTC1040 will regulate the temperature of a system between two separate temperatures (+24.5 °C for the heating and around +25.5 °C, with a hysteresis of 0.25 °C around these two temperatures), for an external temperatures first above, and then under, the target temperature. To avoid duplicating the information available online, I will provide some complementary simulations not performed in the video. Take, for example, the effect of the sensor response time. In stirred air, the response time of the thermistor of the NTCLE213 is 5 s, for NTCLE203 it is 7 s. But if you build a housing around these components, the thermal response could easily be 15 s or more.

In Fig. 22, the response of the system in these two simulations - different overshoots and duty times - are visible.


Fig. 22

With the help of measurements recorded in the “spice log file” (Fig. 23), one can compute an increase in overshoot of about 0.32 °C when going from 7 s to 15 s. Also, the cooler will be on for 126 s for $\tau = 7$ s and 169 s for $\tau = 15$ s.


Fig. 23



7.3. SIMULATION OF PWM TEMPERATURE CONTROL WITH OPTOTRIAC

[For a Few Simulations More - Part 7 - Thermistor, Optotriac and PWM, Hackster.io](#)

This project is a complete thermostat based on an optotriac controlled with a pulse width modulation (PWM) process and an NTC thermistor.

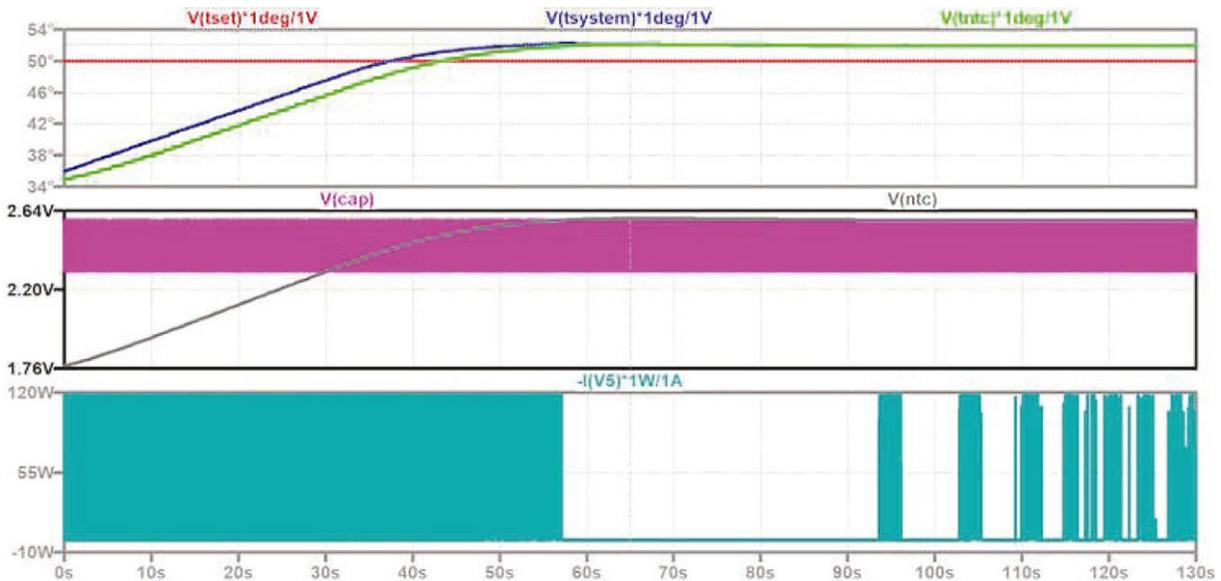
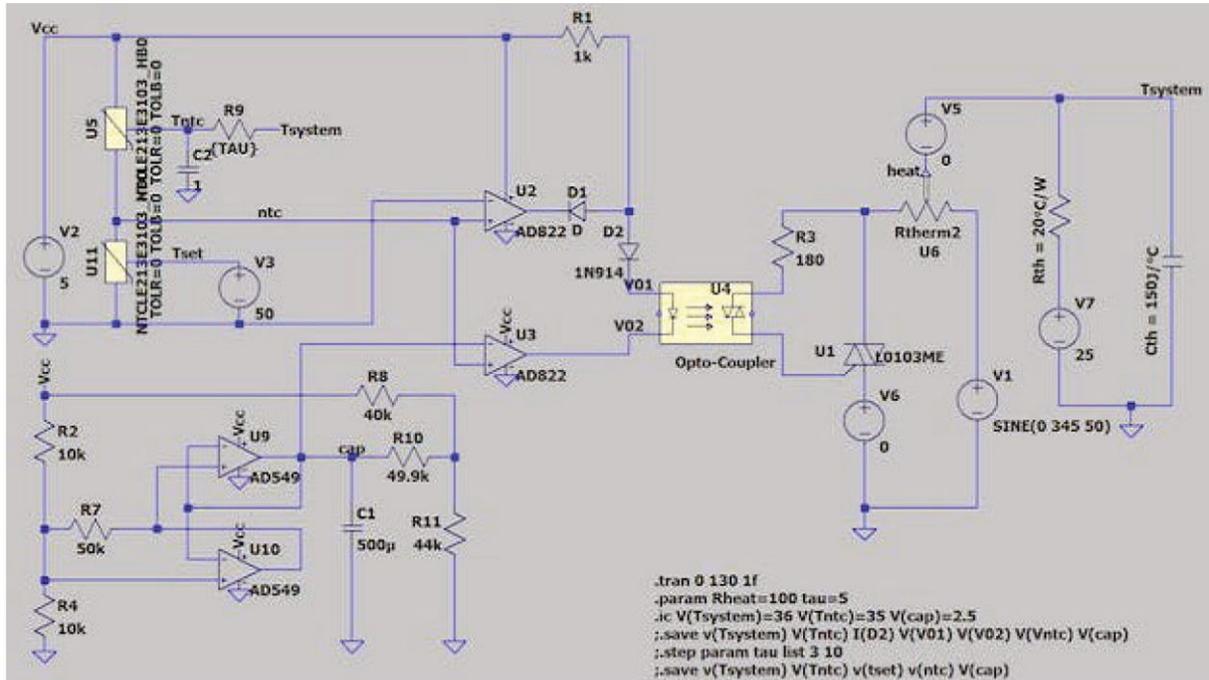
Hardware Components Used in This Project

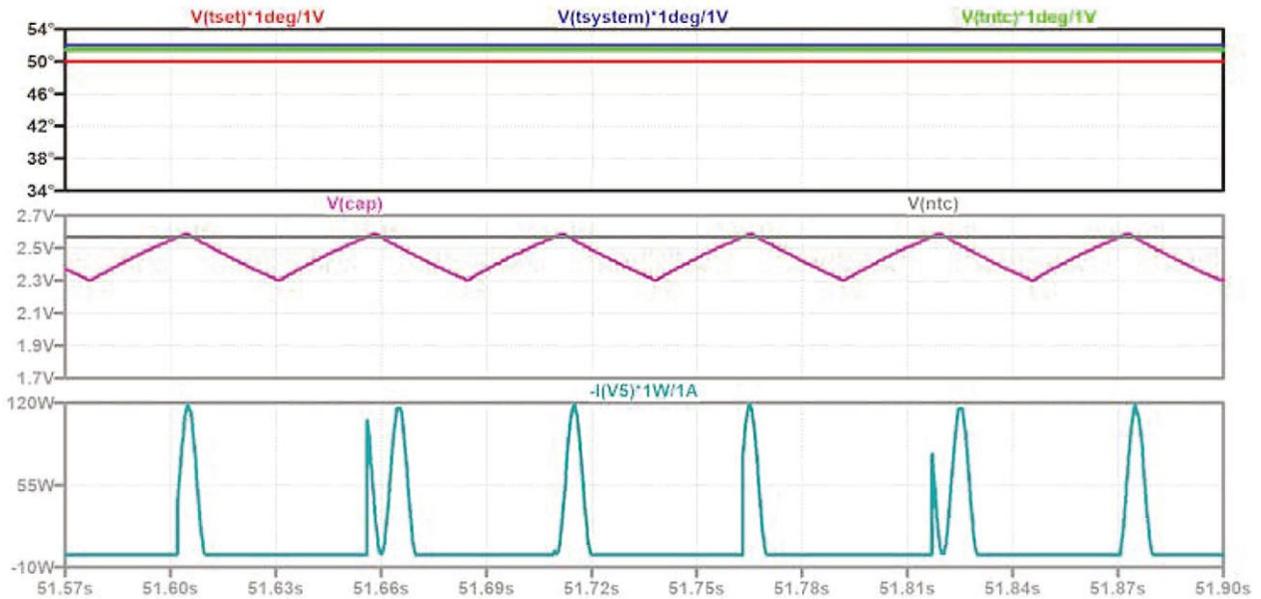
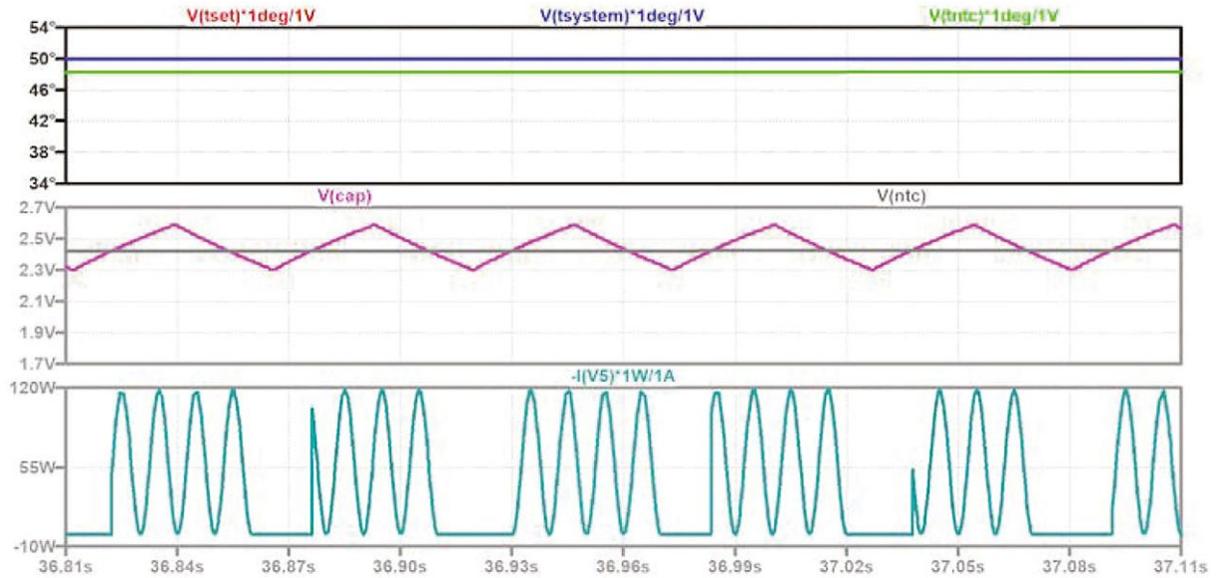
- 1 x [Vishay Thermistor NTCLE213E3103](#)
- 1 x Vishay VOT8125
- 1 x Analog Devices AD822
- 1 x Analog Devices AD549

Story

In this project, a defined type of temperature process control simulation is performed, called pulse width modulation (PWM). A sawtooth signal (at node CAP) is produced, and a temperature-dependent signal (node NTC) is compared to it. Under a defined temperature threshold, the opto triac is constantly on and the full heating power is applied to the system.

Above this threshold, the opto triac begins to be switched on / off with a duty cycle proportional with the difference between the measured temperature and the target temperature. Above the target temperature, the optotriac is completely off. The steady state is thus reached just under this last limit, with a periodic burst of current pulses into the heating element.







8. LEADED THROUGH-HOLE NTCLE100..B0 SERIES

Links

- Datasheet: www.vishay.com/doc?29049
- Product page on Vishay webpage for further info: [NTCLE100E3 NTC Thermistors, Radial Leaded, Standard Precision | Vishay](#)
- Simulation of oven temperature control: [For a Few Simulations More - Part 3 - Oven Control - Hackster.io](#)

For a Few Simulations More - Part 3 - Oven Control

Another simulation of a complete oven temperature control with the help of a Vishay thermistor.

Hardware Components Used in This Project

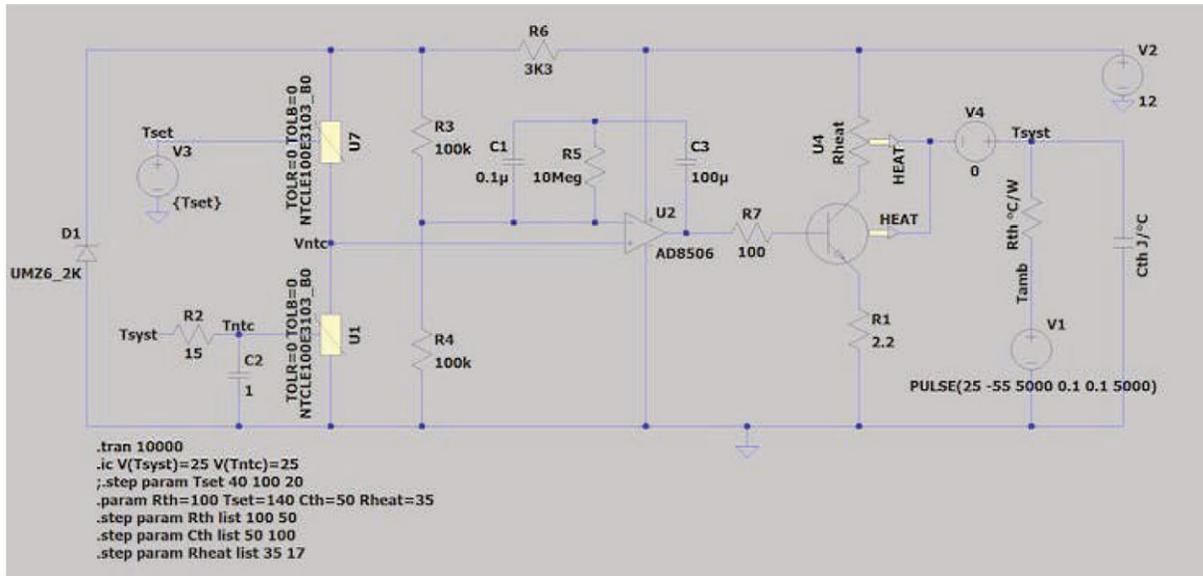
- 1 x Analog Devices AD8664 opamp
- 1 x [Vishay NTC through-hole thermistor 10 kΩ](#)
- 1 x Onsemi 2N3055
- 1 x Rohm UMZ6

Story

For A Few Simulations More - Part 3 - another temperature controller at:

- [Oven Controllers \(techlib.com\)](#)
- [Barometer Experiments \(techlib.com\)](#)

It is in fact a proportional-integrative controller (not an on / off) because the feedback branch is connected to the inverting input of the AD8664 opamp. The originality of this simulation is that the heat produced by the heating elements is modeled, together with the heat produced by the npn transistor. This heat is injected into a thermal circuit (R_{th} C_{th}), which is modeling the oven, and which is connected to a (temperature) voltage source modeling the ambient external temperature. The reached temperature T_{sys} of the oven is fed back to the Vishay NTC thermistor to drive the proportional temperature control.



The parameters C_{th} and R_{th} (respectively, the thermal capacity and resistance of the oven) can be swept, and the ambient temperature profile can be changed at will. The results show full heating when under the set temperature, moderated heating to keep the temperature constant around the set temperature, and the bias change of the npn transistor when ambient temperature decreases. Eventually, when the heat loss is too high and when ambient temperature is very low, the set temperature cannot be held any more. In practice, you will need to evaluate the values of C_{th} and R_{th} for your case.

9. TFPT: TEMPERATURE CONTROL (0603 / 0805 / 1206) AND FLOW DETECTION (0402)

Links

- Datasheet: www.vishay.com/doc?33017
- Product page at Vishay website for further info: [TFPT SMD PTC - Nickel Thin Film Linear Thermistors | Vishay](https://www.vishay.com/Products/TFPT-SMD-PTC-Nickel-Thin-Film-Linear-Thermistors)

TFPT SURFACE-MOUNT NI THIN FILM RTD

LINEAR POSITIVE TEMPERATURE COEFFICIENT ELEMENT

TFPT0603 / TFPT0805 / TFPT1206

Thin Film Flat Chip RTDs

COMPLIANCE TO STANDARDS

- AEC-Q200 qualification
- ESD HBM 0.5 kV (0603); 1 kV (0805 / 1206)
- RoHS-compliant
- Halogen-free
- Whisker-tested per JEDEC® 22A121.01

MULTIPLE DESIGN TOOLS AVAILABLE

- LTspice® models with Monte Carlo tolerances ([online](#))
- Cadence® PSpice models ([online](#))
- VHDL-AMS models (on [SystemVision® Cloud](#))

TEMPERATURE SENSING / CONTROL / PROTECTION / COMPENSATION

[SystemVision® Cloud](#)

- Laser temperature regulator using Peltier TEC

APPLICATIONS

- LED current control
- Portable instrumentation systems
- Peltier cooling elements
- CTA flow sensors
- BLDC drivers
- Optoelectronics
- Car infotainment
- Medical equipment

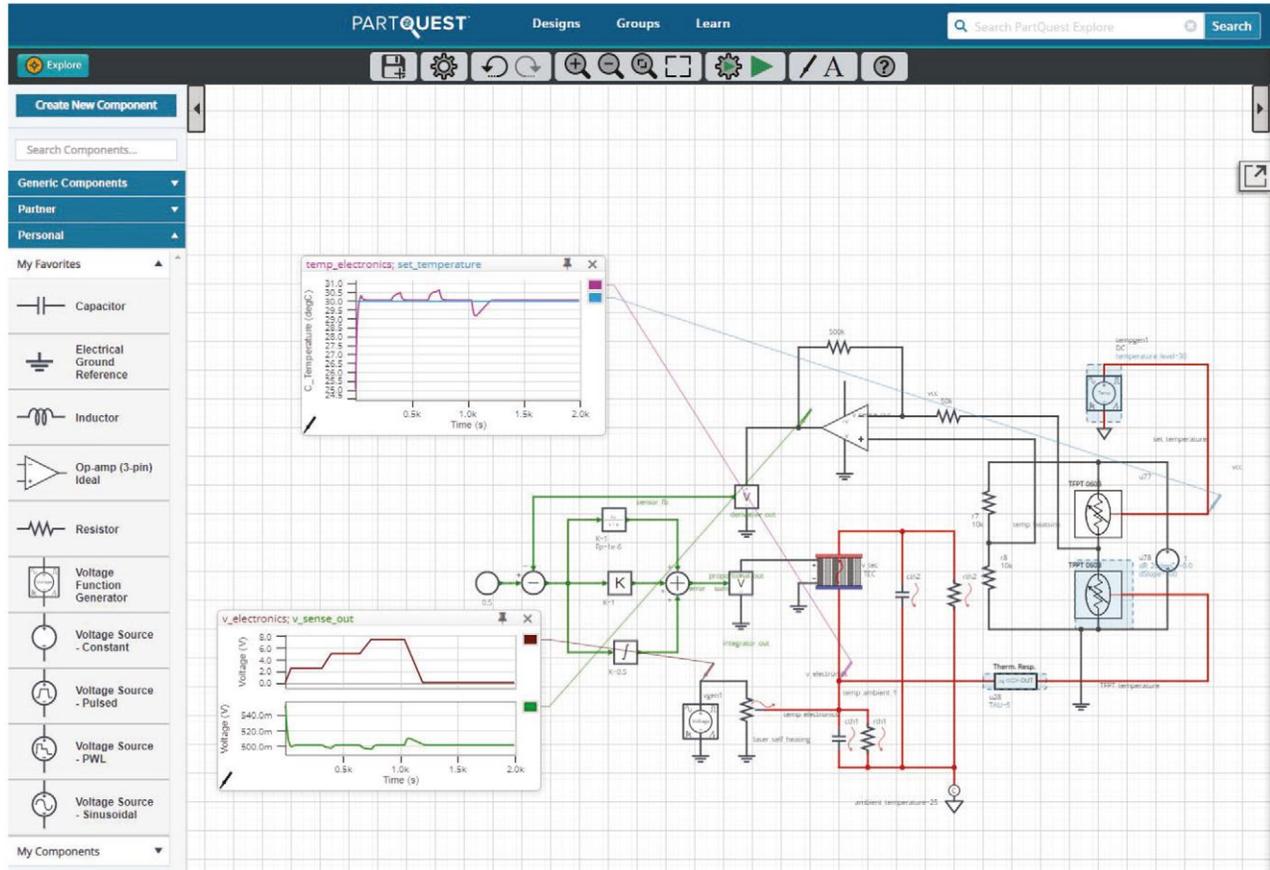
www.vishay.com

For Technical Questions: nlr@vishay.com

IGI2168156-1910

9.1 SIMULATION OF THERMOELECTRIC ELEMENT TEMPERATURE CONTROL

[Laser Temperature Regulator Using Peltier TEC With Vishay TFPT Temperature Sensor | PartQuest™ Explore](#)



The initial simulation and description is by Mike Donnelly (Siemens EDA). The generic thermistor has been replaced by a 1 kΩ Vishay TFPT0603 in the 0603 case. The sensor element (now positive temperature coefficient) has been switched in the opposite arm of the Wheatstone bridge. In this live tunable design, the target temperature level of the laser device can be chosen between 5 °C and 30 °C. The temperature regulation will work accordingly. The tolerances of the TFPT sensor (R_{25} in % and slope ppm/k) can be chosen inside the specification (± 0.5 % and ± 400 ppm/K).

9.2 VIDEO ON SIMULATION FOR FLOW CONTROL WITH 0402 CASE

[Vishay - Vishay Thermistors Electronic Simulation Part 6: Air Flow Detection With SMD 0402 TFPT: 5Ω RTD](#)

[Vishay Thermistors Electronic Simulation Part 6: Air Flow Detection With SMD 0402 TFPT: 5Ω RTD - YouTube](#)

Simulation files: not publicly published; available on demand at: edesign.ntc@vishay.com

10. LEADED THROUGH-HOLE NTCLE203...SB0

Links

- Datasheet: www.vishay.com/doc?29118
- Product page at Vishay website for further info: [NTCLE203E3...SB0 NTC Thermistors, 2-Point Radial Leaded, Automotive Grade | Vishay](#)

10.1. SIMULATION OF ANALOG PID TEMPERATURE CONTROL WITH THERMISTOR

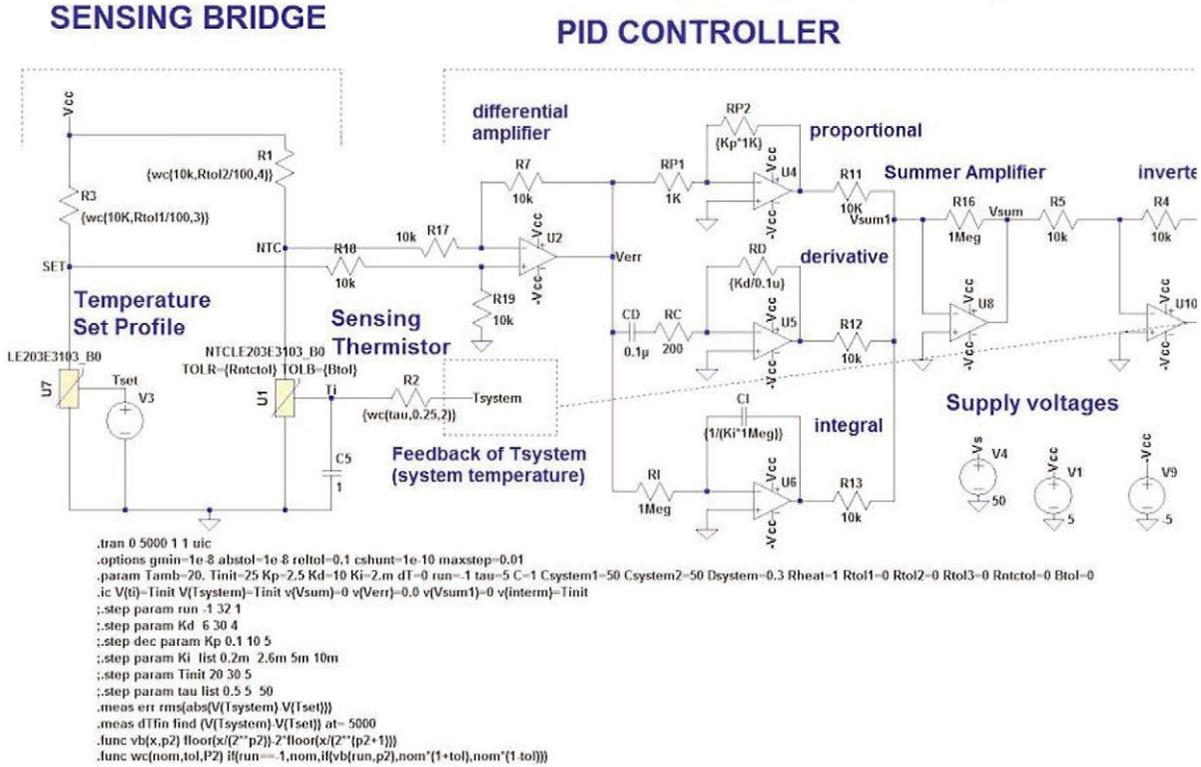
Link

- [Simulation of Analog PID Temperature Control With Thermistor - Hackster.io](#)

An LTSpice simulation of an PID temperature control circuit with a Vishay thermistor temperature sensing and heat flow modeling.

[Beginner](#), Protip, 1 hour

PID TEMPERATURE CONTROLLER WITH NTC (TEMPERATURE/ VOLTAGE DRIVEN) and TEM SENSING BRIDGE



Hardware Components Used in This Project

- 1 x Vishay NTCLE203E3103SB0
- 1 x Vishay CRCW

Story

This project is the presentation of a SPICE simulation (LTspice) of a PID temperature control with a Vishay thermistor as a temperature probe.

The conception of an electronic thermostat can be done in the cadre of one EDA, in this occurrence LTspice XVII. The versatility of this software is such that it is possible to create new special models of thermal transistors and resistances, with the heat flux created by those devices directly injected into thermal circuits modeling heatsinks. We thus take advantage of the equivalence of RC electrical and thermal circuits. It is the best of both worlds, as we model the electronic part of the circuit in a SPICE environment (devices are the real thing), and the thermal part is a bit like modeling in Simulink of Mentor's System Vision Cloud.

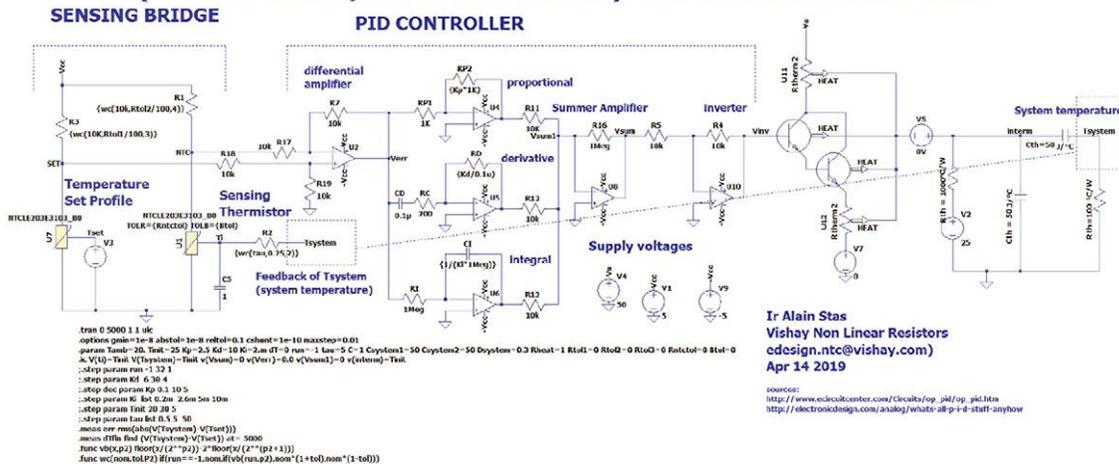
The following design of this temperature control PID based on a thermistor bridge has been published in detail elsewhere in a more rudimentary form:

<https://www.planetanalog.com/old-school-analog-temperature-control-circuits-solved-with-modern-lt-spice-thermistor-dynamic-models-part-2/#>

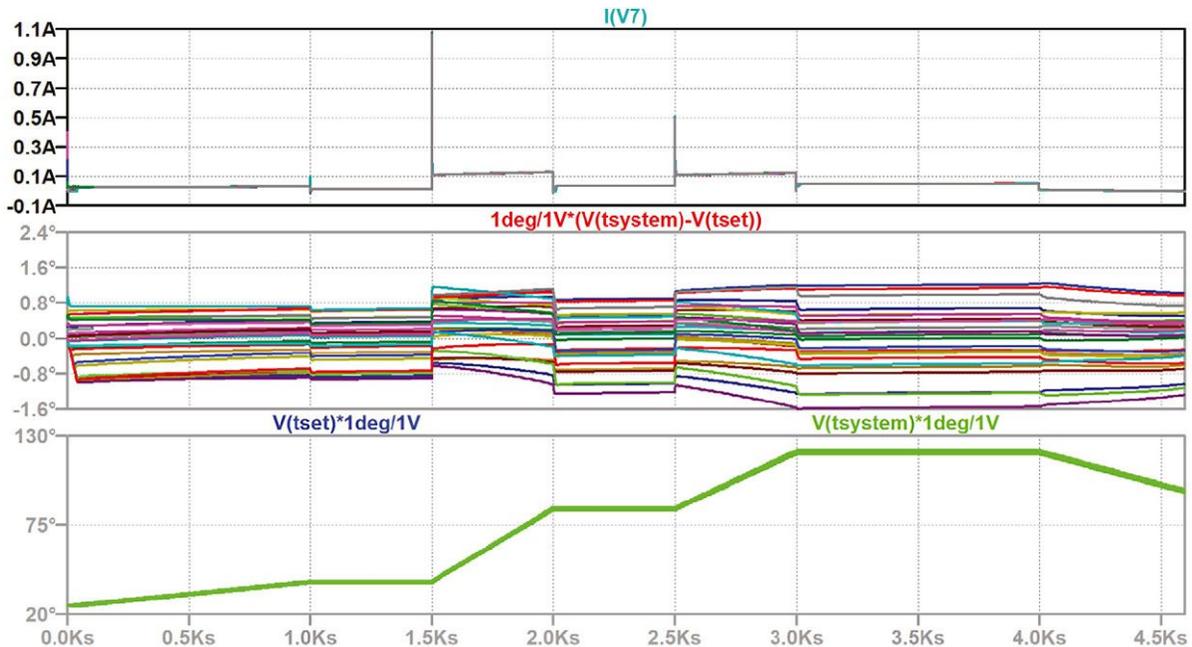
We send the reader to this article for the details.

We will emphasize here the modifications with respect to the original design. The transistors and warming element resistor models are now explicitly drawn into the design. And the heat flux generated by these devices is directly injected into an RC thermal circuit, modeling the system object to heat up (heatsink / oven etc.).

PID TEMPERATURE CONTROLLER WITH NTC THERMISTOR (TEMPERATURE/ VOLTAGE DRIVEN) and TEMPERATURE NOISE



After optimization of the PID, it is thus possible to simulate the temperature variation of the “system”, compared to the target profile (included in the simulation).



The analysis can be done as a worst-case analysis or Monte Carlo random, or here a combination of both distributions. It is possible to visualize temperatures in LTspice XVII by multiplying the corresponding voltages by a factor of ($1^\circ / 1 \text{ V}$). The scale is then transformed magically into $^\circ\text{C}$.

One can see in the above graphs (mid pane) that we can control the target temperature within the $\pm 2^\circ\text{C}$, taking account of all tolerances of all passive elements (thermistor / fixed resistors/ response time etc.).

The package to be downloaded with this project is self-contained (all the models are included). Keep all models in one directory and perform the simulation.

10.2. LTspice SIMULATION ON / OFF TEMPERATURE CONTROL VIA VOT8125

Link

- [LTspice Simulation On / Off Temperature Control via VOT8125 and NTC - Hackster.io](https://hackster.io/projects/1028125)

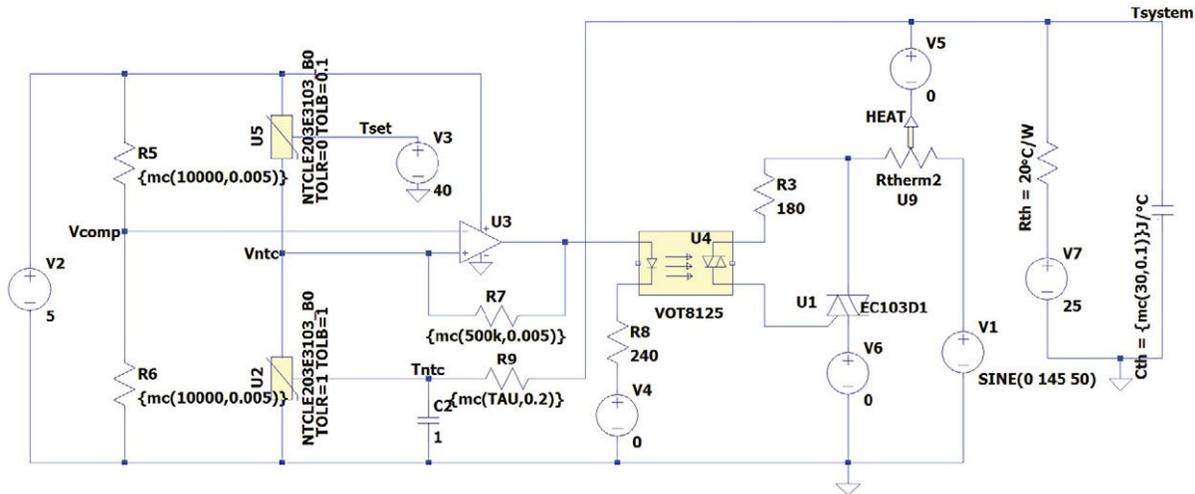
A simulation is shown in LTspice of a on / off temperature regulator with an optotriac and an NTC thermistor from Vishay.

Hardware Components Used in This Project

- 1 x Vishay VOT8125
- 1 x [Vishay NTC Through-Hole Thermistor 10 k \$\Omega\$](#)
- 1 x Vishay NTCLE203E3103SB0
- 1 x Analog Devices LT1721

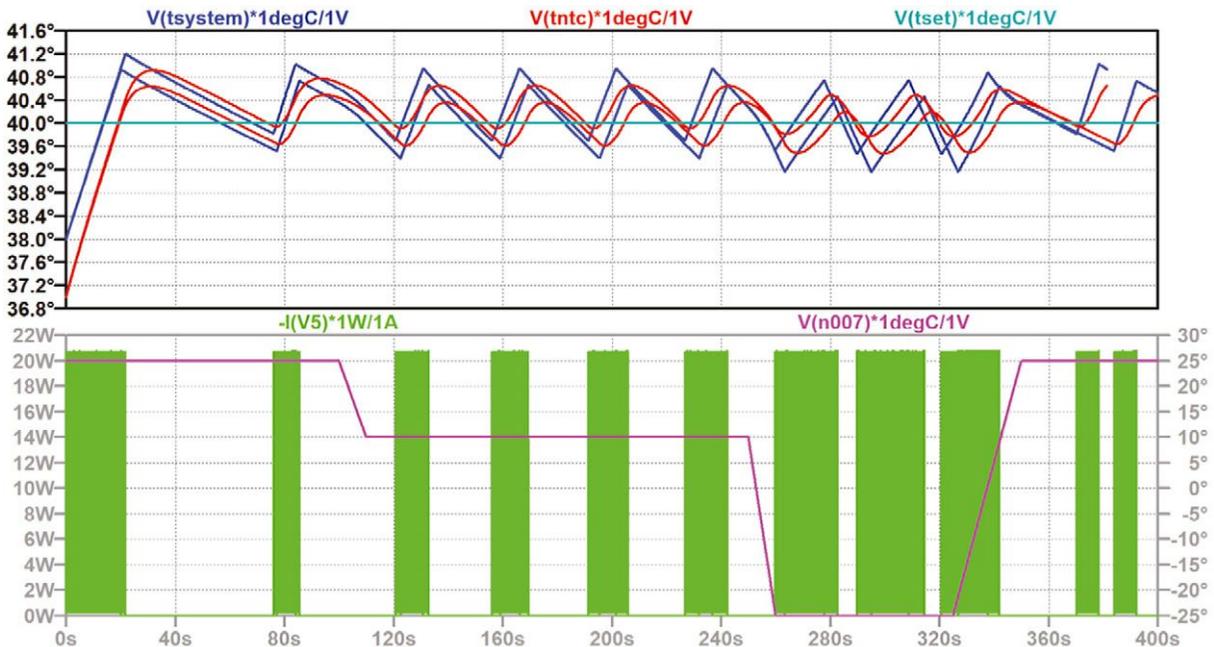
Story

We present an on / off temperature control simulation in LTspice XVII. The temperature sensing is performed by an NTC thermistor (from Vishay). The insulation between the low voltage sensing bridge and the high voltage AC heater is ensured by a Vishay opto triac VOT8125. The heating element is modeled as a thermal model with three pins, the third one being the heat source (as a current) injected into a thermal RC circuit modeling the oven or the device to heat. The voltage signal T_{system} is the device / oven temperature and is directly fed back to the thermistor via another RC network, which represents the response time of the thermistor.



```
.tran 0 125 1f 0.1m
.step param run 1 3 1
.param run=1 Rheat=100 Tamb=18 t1=90 t2=150 T3=200 T4=250 tau=5 dT1=5 dT2=5 dT3=5
.ic V(Tsystem)=38 V(Tntc)=37
```

In the thermal circuit of the device / oven we have introduced the ambient external temperature, changing in time from +25 °C to lower temperature. At low temperatures, the duty cycle of the triac on state increases.



One can visualize the different response of the system depending upon the different element tolerances (Monte Carlo distribution).

The zip file contains all the models for all devices of the simulations, including the special model for a heat emitting resistor, the

Vishay NTC thermistors, and the VOT8125.

10.3. ANOTHER TEMPERATURE CONTROL WITH 6N136 OPTOCOUPLER

Link

- [For a Few Simulations More - Part 6 - Return on Optocouplers - Hackster.io](#)

Another thermostat based on the 6N136 optocoupler and a thermistor from Vishay

Hardware Components Used in This Project

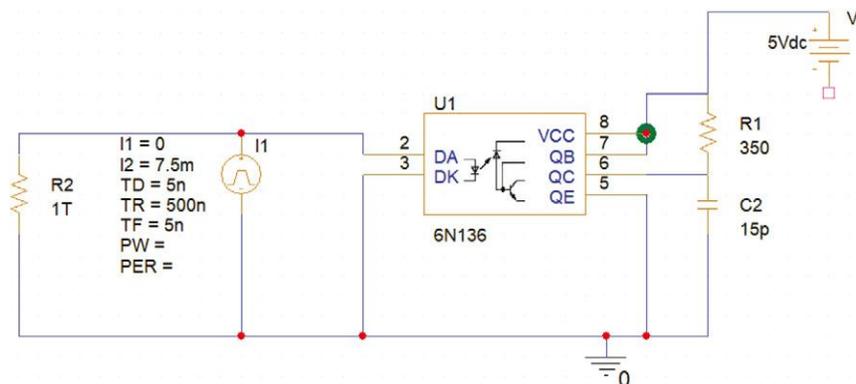
- 1 x [Vishay NTC Through-Hole Thermistor 10 k \$\Omega\$](#)
- 1 x Vishay 6N136
- 1 x Analog Devices OP77

Story

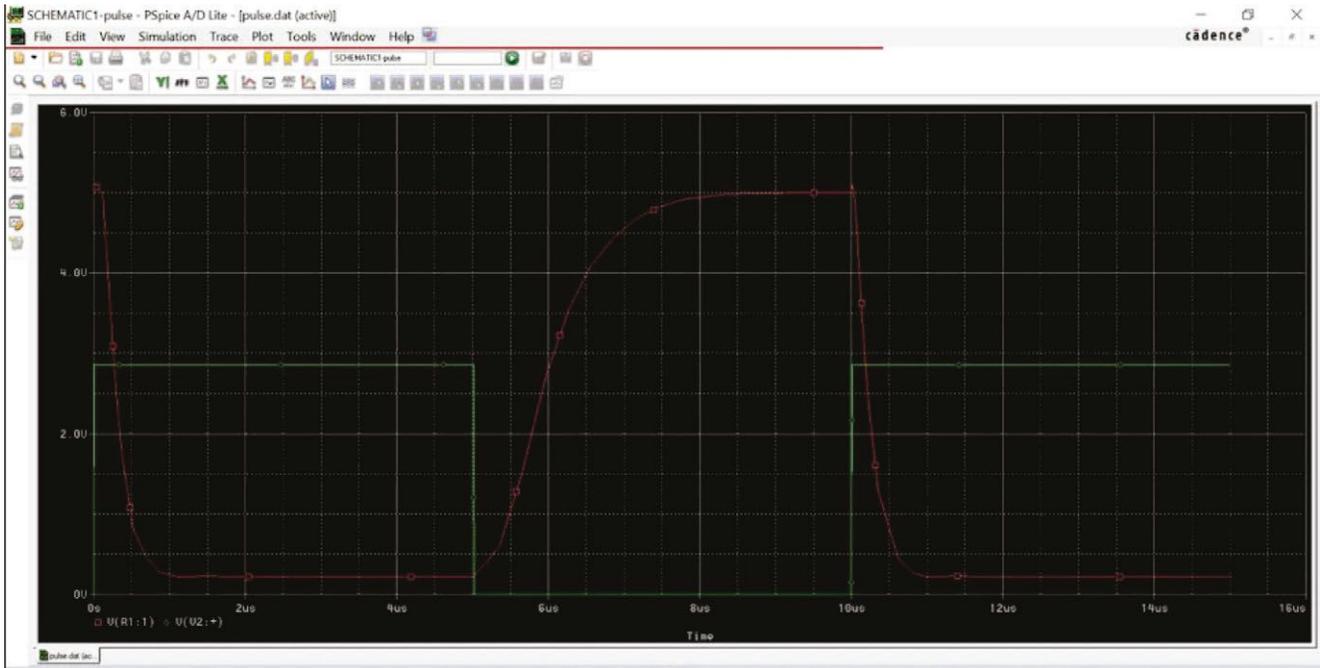
For a Few Simulations More - Part 6: a Thermostat With the Optocoupler 6N136 and Thermistor NTCLE100

Those who have read Part 5 might have asked themselves why a 6N137 high speed optocoupler with logic output was used for the commutation, and they would be right: there is no real need for such a fast optocoupler. At least we have solved the problem of the importation of the PSpice model into LTspice XVII with verifications. If we take the same circuit and replace 6N137 (with logical circuitry) with 6N136 (with transistor output), the functionality of the circuit should be the same. That is what I would like to simulate at least.

In order to prepare the simulation, we need to import the PSpice model ([6N135, 6N136 High Speed Optocoupler, 1 MBd, Photodiode with Transistor Output | Vishay](#)) into LTspice, and this time the importation process is flawless (the new file and the symbol are joined in the zip file appendix). This time I was also able to verify that the model is working in PSpice Orcad capture light. The image below captures the simulation. Note that the model will not work in PSpice for TI. for the same reasons evoked in Part 5.

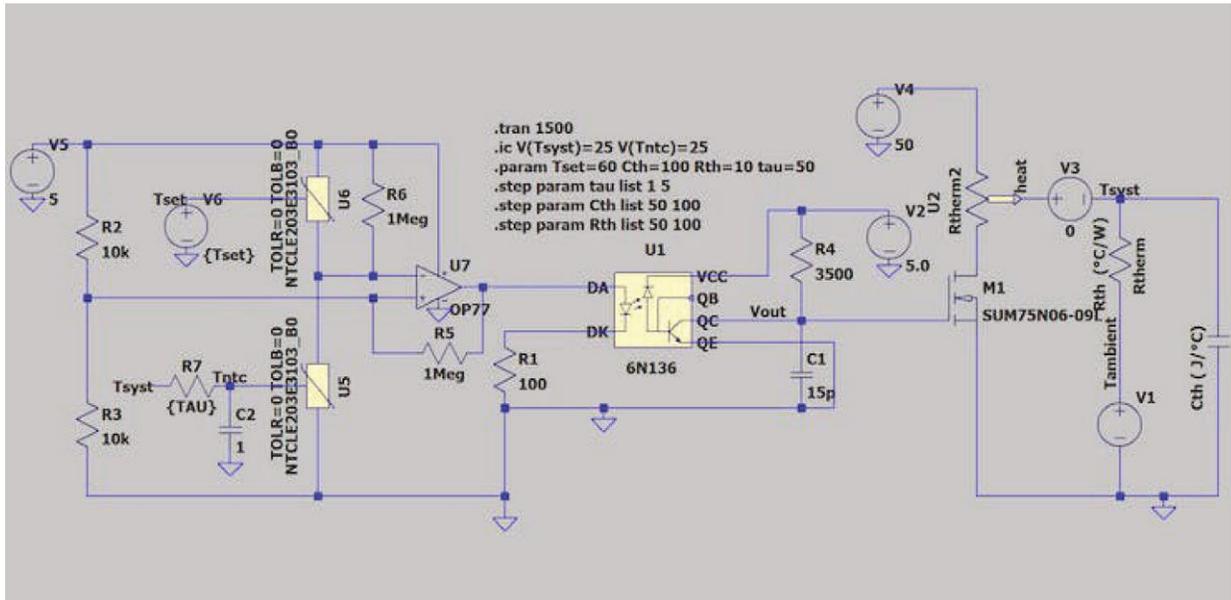


PSpice Simulation of 6N136 Functionality

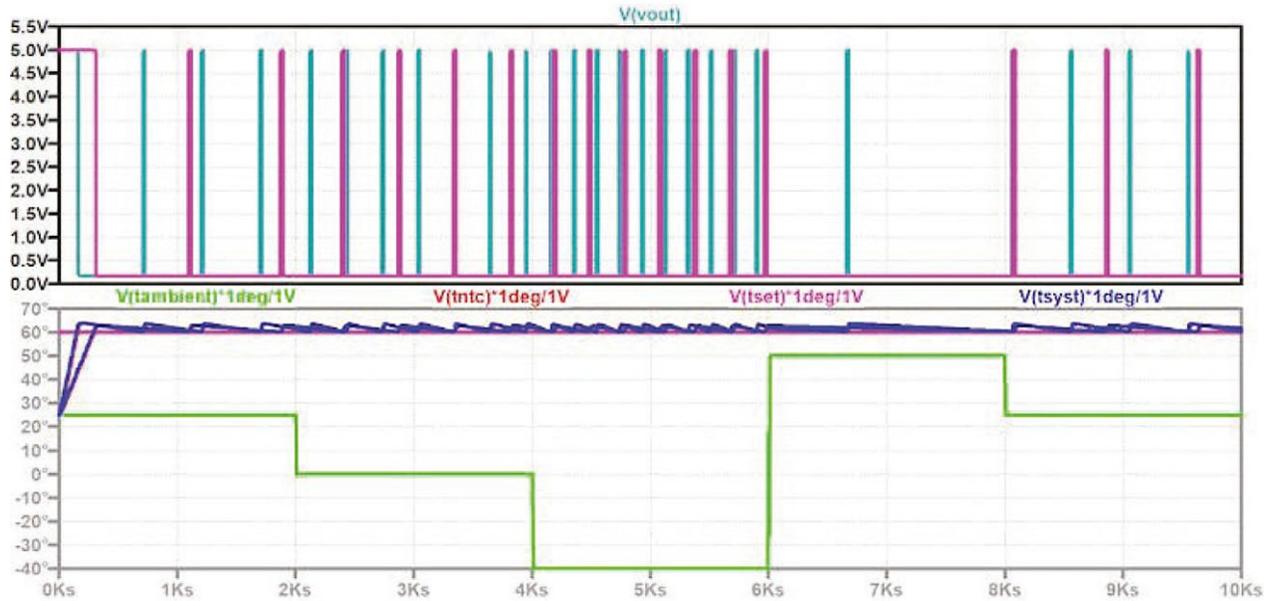


Waveforms of the Above Circuit Simulation

This leads us to the new thermostat of Part 6. Basically, it is the same circuit as in Part 5, except the 6N136 optocoupler is a 6N137. As expected, everything seems to work well.

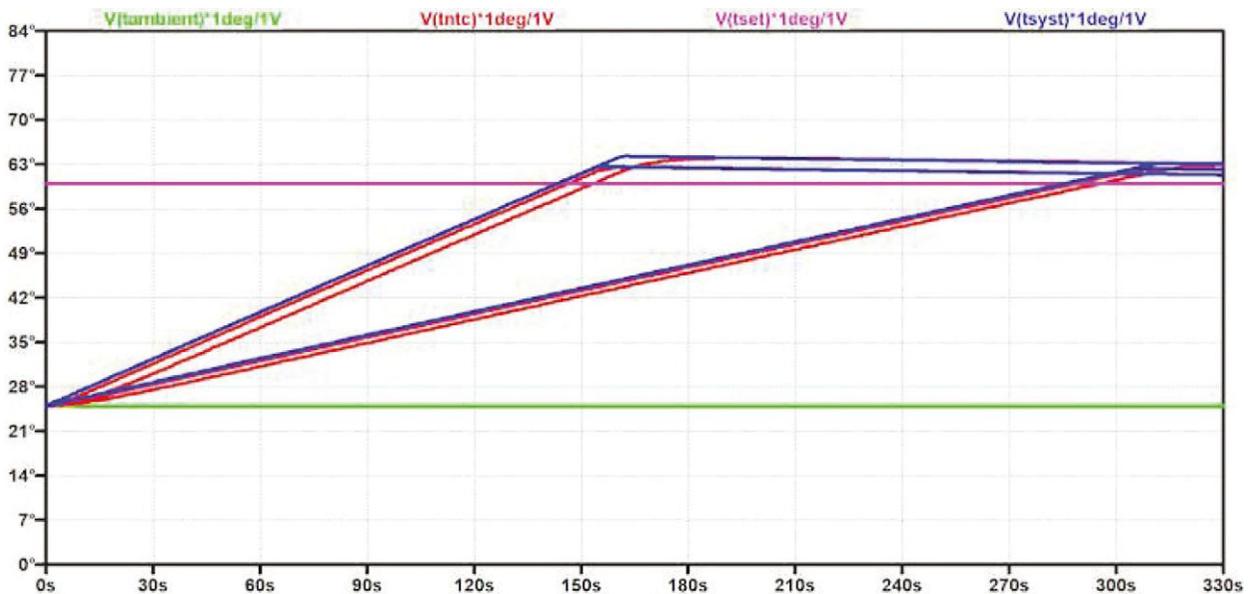


Thermostat Based on 6N136 and Thermistor



Waveforms in Time

The system temperature stabilizes around the T_{set} temperature with a hysteresis defined by R5. The transient time going from ambient up to T_{set} (60 °C) depends upon the thermal capacity of the system C_{th} . The frequency of V_{OUT} pulsing also depends on the external temperature T_{amb} , via thermal resistance R_{th} . The user can sweep all the R_{th} , C_{th} values that they want within the same simulation. The τ parameter (response time of the thermistor) heavily influences the first overshoot of temperature. The lower the response is, the smaller the overshoot.



Blow Up the Waveforms

11. PTCSL03 SERIES

Links

- Datasheet: www.vishay.com/doc?29160
- Simulation tools:
 - SPICE model: [PTCSL03 PTC Thermistors, Mini Radial Leaded for Over-Temperature Protection | Vishay](#)
 - Example in LTspice XVII: [SPICE Simulation of Power Semiconductor Protection With PTC - Hackster.io](#)

SPICE Simulation of Power Semiconductor Protection With PTC

An LTspice simulation of over-temperature protection of a power BJT by a Vishay PTC sensing through-hole component.

Hardware Components Used in This Project

- 1 x Rohm 2SCR552P
- 1 x Vishay PTCSL03T141

Story

This simulation is about PTC protection of a power semiconductor (BJT, MOSFET or IGBT). In a circuit like the one in Fig. 24, a lot of heat is generated by both fixed resistors and bipolar transistors.

This heat is transferred to a heatsink on which all elements are fixed, and the final heatsink temperature is then measured by Vishay PTCSL03 leaded through-hole components to strongly reduce the current passing through the transistor by strongly decreasing the base voltage.

The heat generation is modeled via the supplementary pins created for a simple fixed resistor and a npn transistor. Both netlists will be found in the appendixes of this project.

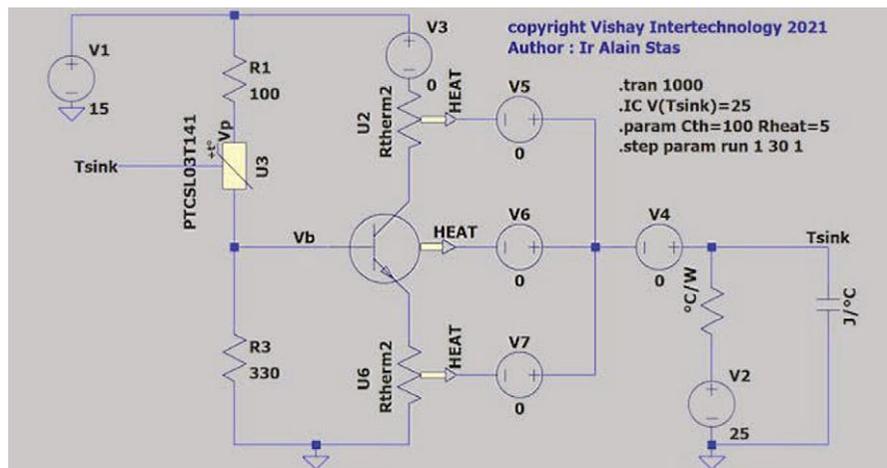


Fig. 24

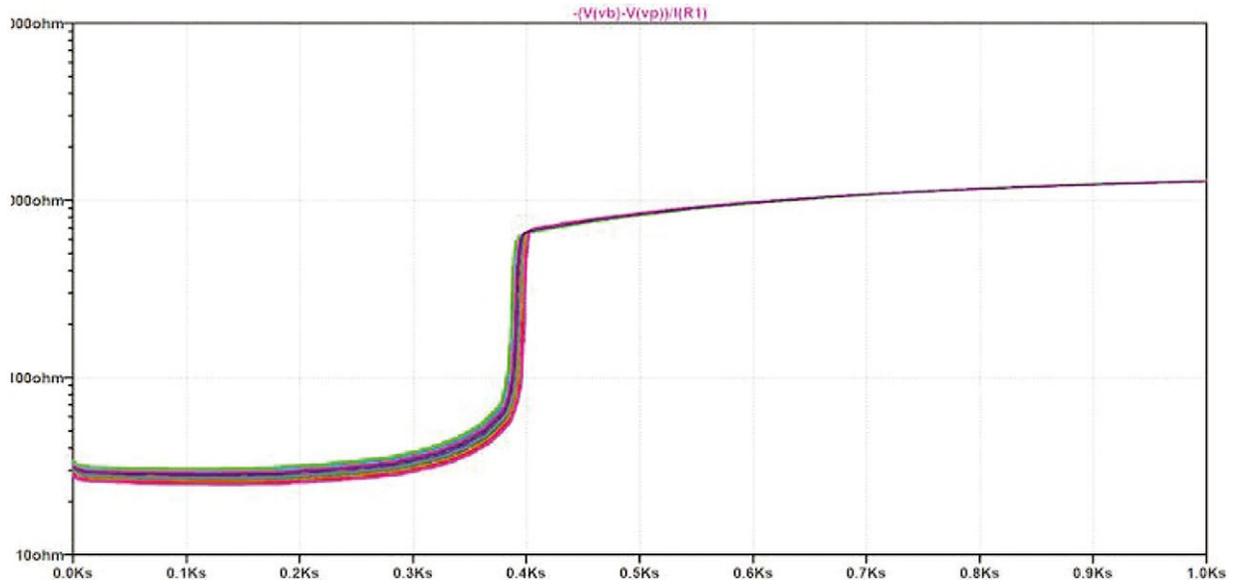


Fig. 25

Fig. 25 represents the PTC resistance variation in time showing the strong resistance increase at a defined temperature (here between +115 °C and +120 °C) (Fig. 26).

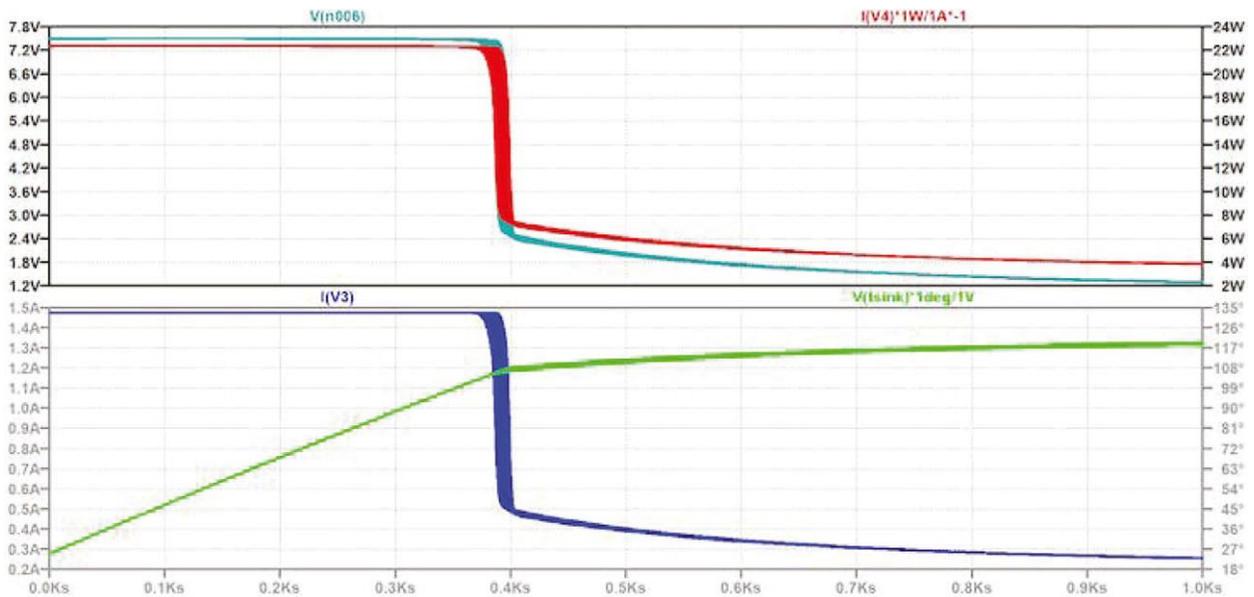


Fig. 26

Above this temperature, the current through the load (U2) will be strongly reduced.

12. NTCALUG

Links

- Product page on Vishay website for further info: [NTCALUG...Custom Capabilities NTC LUG Ring Tongue | Vishay](#)
- Application example in an article at “Planet Analog”. The article excerpt is below.

[A New Trilogy of LTspice Circuits With NTC Thermistors - Part 2: LTspice Simulation of PID Temperature Control of a Peltier Thermoelectric Element With a NTC Thermistor - Planet Analog.](#)

Although it is common knowledge that their efficiency is poor, Peltier thermoelectric elements are considered the most versatile cooling elements; they are used in a wide range of applications, such as a laser diode cooling systems or portable fridge coolers, as well as automotive thermoelectric generators.

The recent emergence of SPICE models realistically describing this element type [1][2] makes it possible to integrate them into temperature control circuit SPICE simulations. This article describes a straightforward current driving circuit for the cooling of such a Peltier element, with the cold mass temperature measured by a NTC thermistor.

Before digging into the total application, I would like to focus a bit on the sensor device that makes the temperature control possible in the first place; in this case, it is an NTCALUG03A ring tongue sensor, one of the smallest devices on the market. The picture in Fig. 27 compares it to a more common larger device. The SPICE electro-thermal model for the NTCALUG was also made available very recently [4].

The first very simple simulation I will present is the transient behavior of the NTCALUG03 sensor when screwed / pressed onto a metal plate (as presented in Fig. 27).

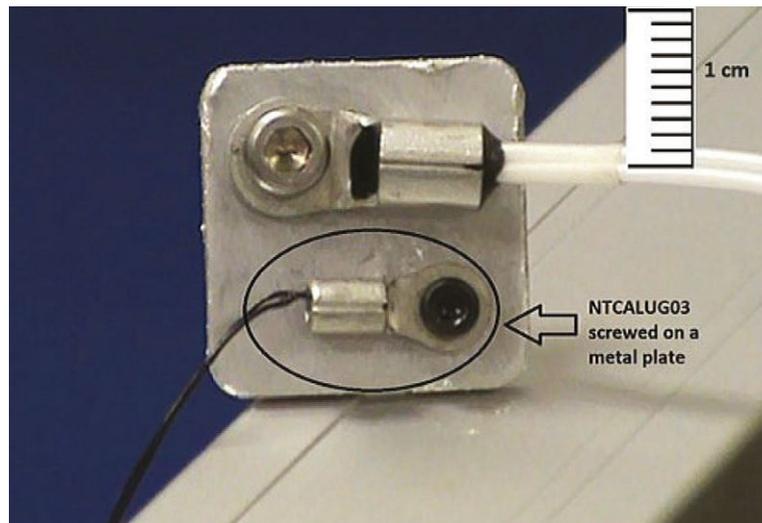
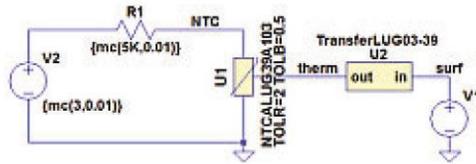


Fig. 27

The circuit of the NTCALUG03 macro-model is presented in Fig. 28, where the NTC is placed in a classic voltage divider. Fig. 29 shows the transient temperature evolution, where the plate on which the ring is screwed or pressed is going through a series of temperature changes. We see the short latency reaction time just after the changes, and at steady state see the gradient between the plate and thermistor element (to read further [the remainder of the article can be consulted at the above URL on the Planet Analog website](#)).

Macromodel Test fixture File of the VISHAY NTCALUG03A /NTCALUG39A LUG type thermistors :

- TolR and tolB are modelled according to a LT spice {mc(nom, tol in %)} Monte Carlo function
- The NTCALUG models are temperature driven by TI Pin (therm) which is connected to out pin of TransferLUG03-09 which models the thermal response of the LUG to the temperature (V1 source)of the surface on which the LUG is screwed on.
- Tinit is the ambient temperature and also the initial temperature.
- Grad is the ratio between the difference (surface/thermistor) and the difference (surface/ambient) at steady state It is here equal to 70 mK/K but can be adapted to fit other application conditions (may NDT be zero, use 1f for zero)



```
.tran 120
.param run=0 Tinit=25 grad=0.07 Tinit and Grad MUST be defined in every simulation
.IC V(therm)= Tinit V(surf)=Tinit Initial state of thermistor must be defined in every simulation
;step param run 1 100 1
.meas Rc find -V(ntc)/R1 when V(therm)=40
.meas Ra find -V(ntc)/R1 when V(therm)= 25
.meas Rh find -V(ntc)/R1 when V(therm)=125
```

Fig. 28

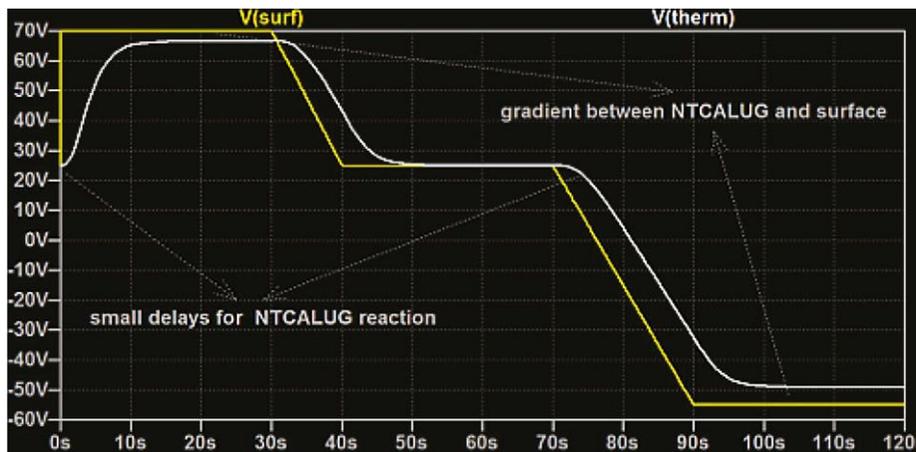


Fig. 29

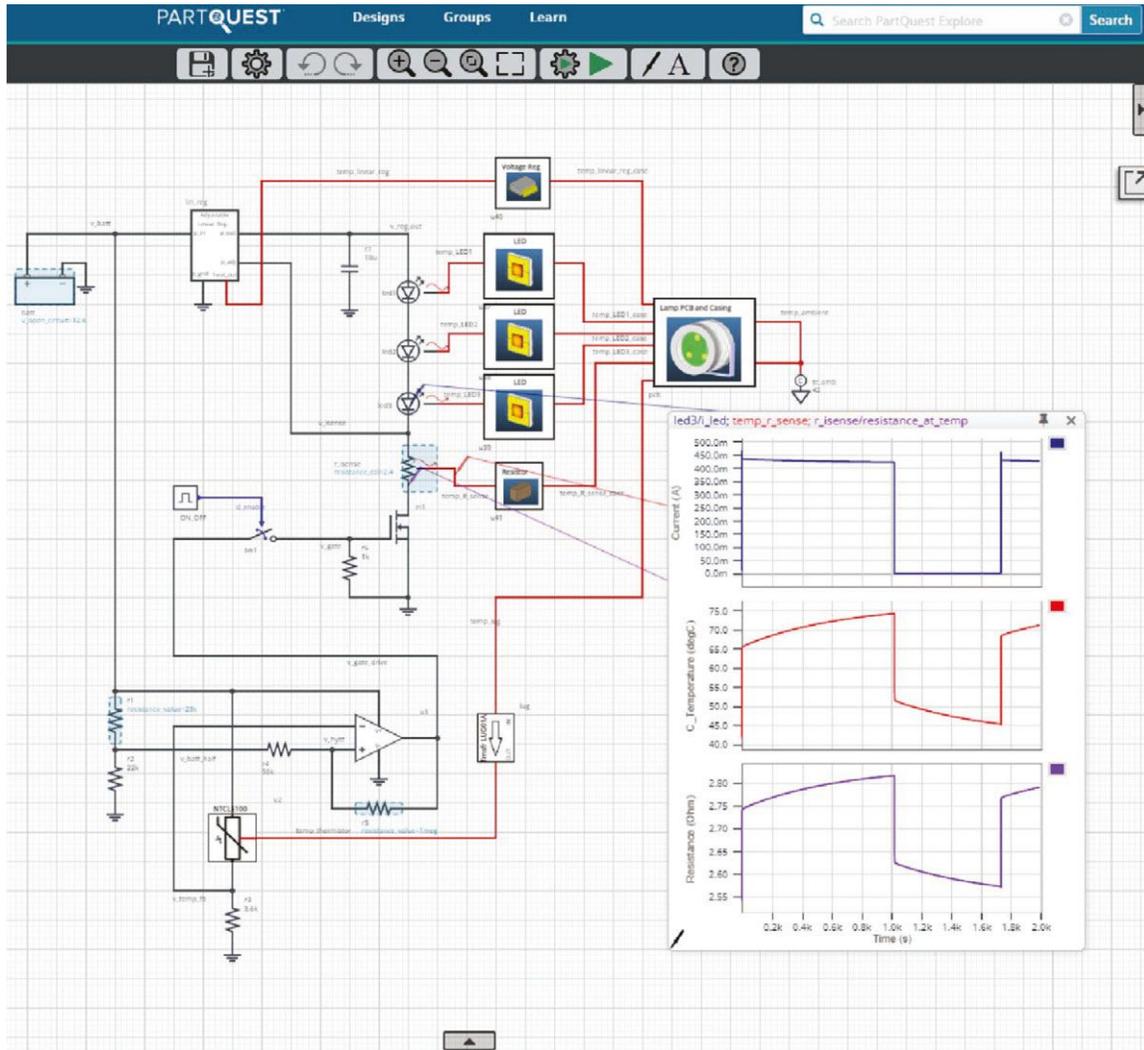
In yellow: the temperature of the plate surface; in white: the response of the NTCALUG (pressed against the plate at time = 0 s)

References

- [1] [Improved SPICE Modeling and Analysis of a Thermoelectric Module](#)
- [2] Chakib Alaoui in International Journal of Engineering (IJE), Volume (5): Issue (1): 2011
- [3] [Old-School Analog Temperature Control Circuits Solved With Modern LTspice Thermistor Dynamic Models, Part 2](#)
- [4] [LTspice](#)

Simulation at PartQuest Explore: Analog LED Driver With Thermal Protection Using FloTHERM BCI ROM (With NTC Temperature Control)

[Analog LED Driver With Thermal Protection Using FloTHERM BCI ROM | PartQuest™ Explore](#)



This LED spotlight example demonstrates the value of simulating both the electrical and thermal aspects of power dissipating circuits together, simultaneously. In this design, a Vishay NTCLE100 thermistor is used in a detection circuit to monitor the enclosure temperature. It is used for thermal shut-down protection, to keep the enclosure temperature well below the “ T_g ” (glass transition temperature) of the spotlight’s Nylon 6 polymer lens. This is particularly helpful when operating at higher external ambient temperatures.

The thermal side of the circuit consists of connected BCI-ROMs generated by Simcenter FloTHERM in the IEEE Standard VHDL-AMS format. Thermal BCI-ROMs for the linear regulator, three LEDs, and the current sense resistor are connected to electrical models of these devices. The thermal environment is represented by a BCI-ROM of the rest of the system: PCB, lamp casing, and the thermistor.

The ability to include an accurate model of the thermal environment is key to having “thermally aware” circuit functional design and board layout processes.

- Vishay Design Tool: LTspice models for the different NTCALUG series

NTCALUG01A: www.vishay.com/docs/29178/LTspice_volt_as_temp_ntcalug01a.zip

NTCALUG02: www.vishay.com/docs/29177/LTspice_volt_as_temp_ntcalug02a.zip

NTCALUG03: www.vishay.com/docs/29176/LTspice_volt_as_temp_ntcalug03-39.zip

Each macro model NTCALUG01/02/03 offers a simulation presenting the product in dynamic behavior, with its adjustable characteristics needed for fitting the real measurements (particularly the gradient between the surface temperature and the temperature seen by the thermosensitive element in steady state). By default, these characteristics will take the values determined by Vishay in their standard conditions.

Macromodel Test fixture File of the VISHAY NTCALUG01T LUG type thermistors :

- ToIR and toIB are modelled according to a LT spice {mc(nom, tol in %)} Monte Carlo function
- The NTCALUG models are temperature driven by Ti Pin (therm) which is connected to out pin of TransfertLUG01T which models the thermal response of the LUG to the temperature (V1 source)of the surface on which the LUG is screwed on.
- Tinit is the ambient temperature and also the initial temperature.
- Grad is the ratio between the difference (surface/thermistor) and the difference (surface/ambient) at steady state It is here equal to 150 mK/K but can be adapted to fit other application conditions

V1 is the surface temperature on which the NTCALUG01 is screwed

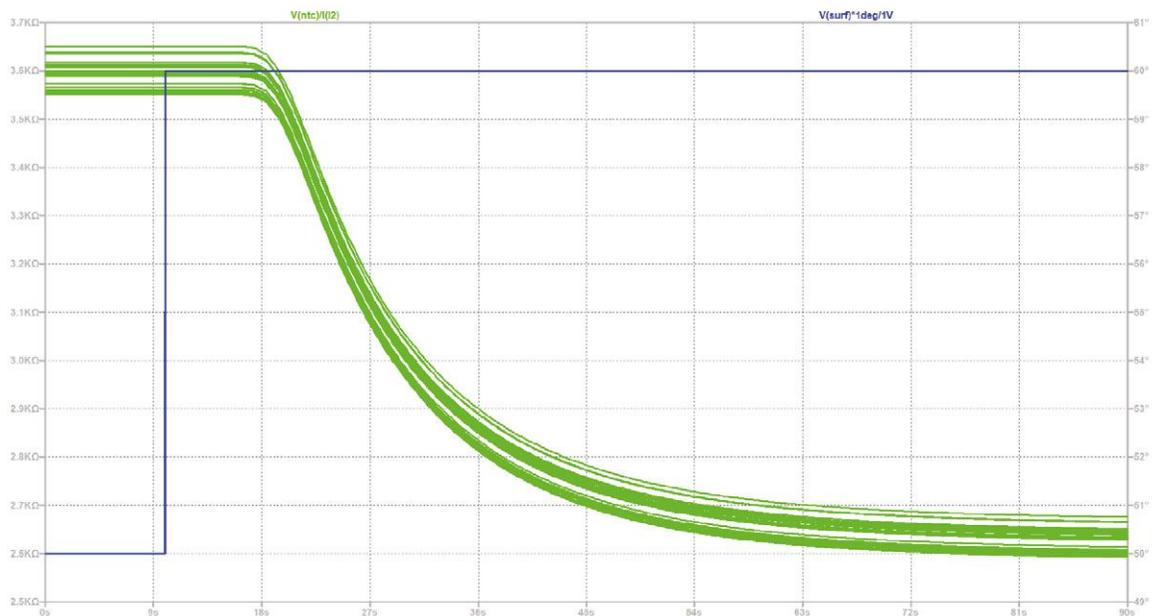
```

PULSE(50 60 10 1f 1f 500 1000 3)
    
```

```

.tran 0 90 0 0.01
.ic V(therm)=50
.param run=0 Tinit=50. grad=0.15
.step param run 1 10 1
.meas TRAN gradient FIND (V(Therm)-85)/60 AT 300
.meas TAU2 find time when V(therm)=25+63.2/100*60
.meas TAU param TAU2
    
```

ThermistorTemperature must be defined at time 0.
grad and Tinit must be defined in every simulation

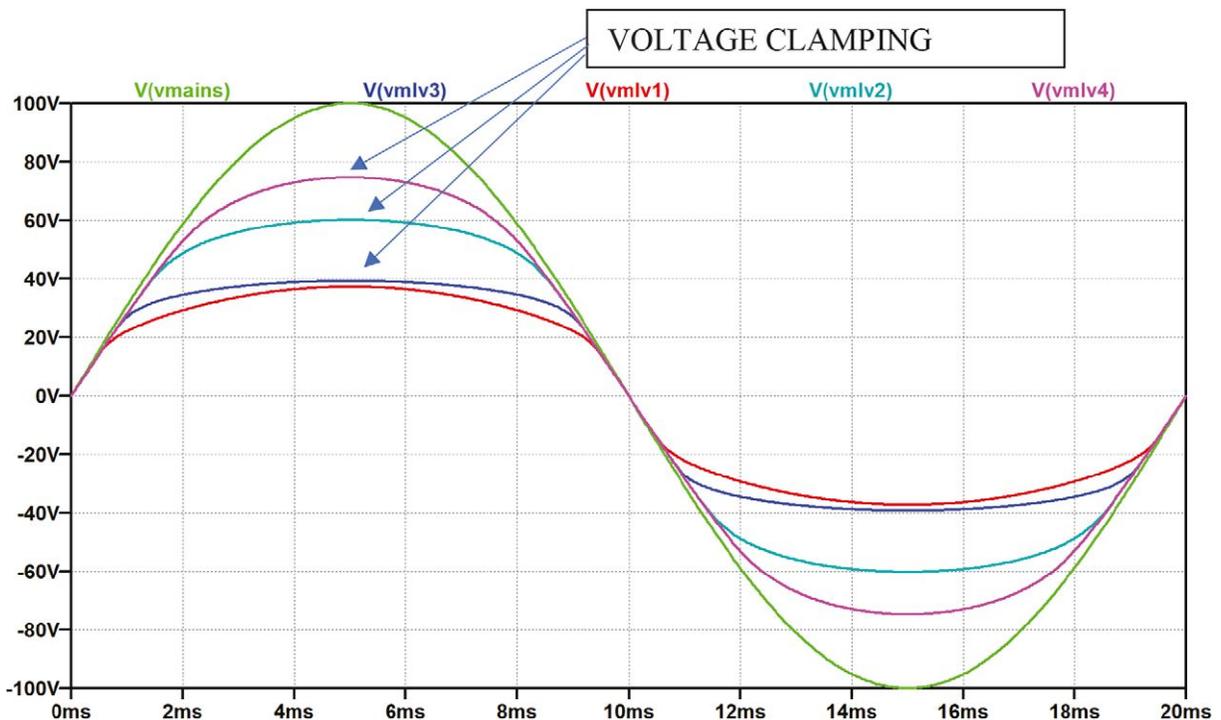
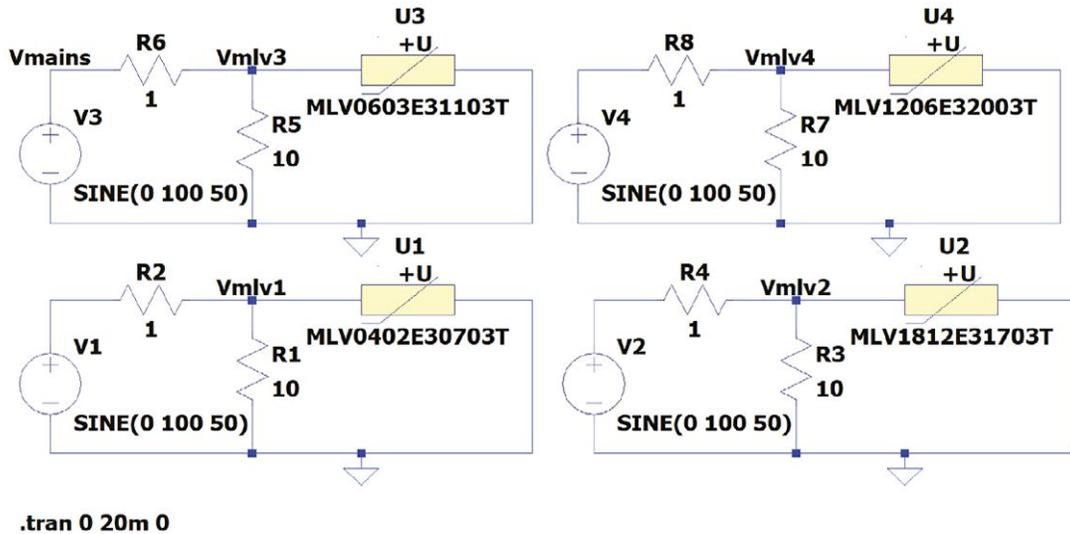


13. VDRS, VDRUS, AND MLV VARISTORS SERIES

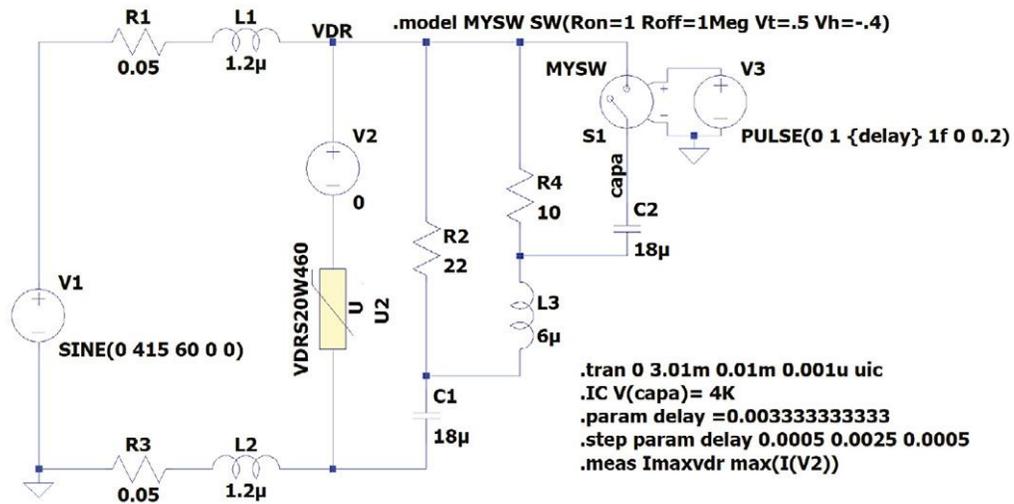
Links

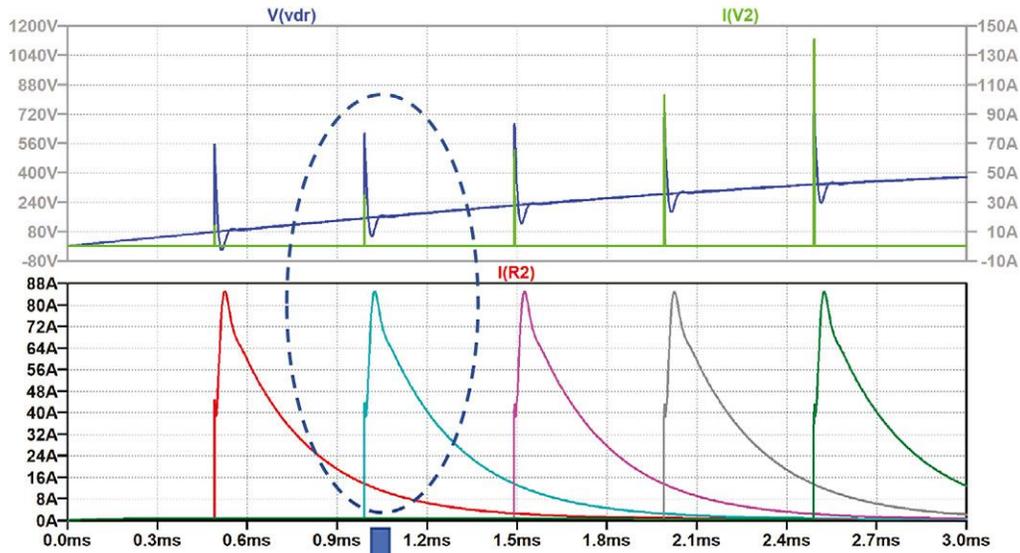
- For MLV [Keyword and Part Number Search Results | Vishay](#)
- For VDRS: [VDRS Series VDR Metal Oxide Varistors Standard | Vishay](#)
- For VDRUS: [VDRUS Series VDR Metal Oxide Varistors Ultra Surge High Operating Temperature 125 °C | Vishay](#)

Vishay provides simulations (to be found within the LTspice zip files) showing voltage clamping as depicted below.

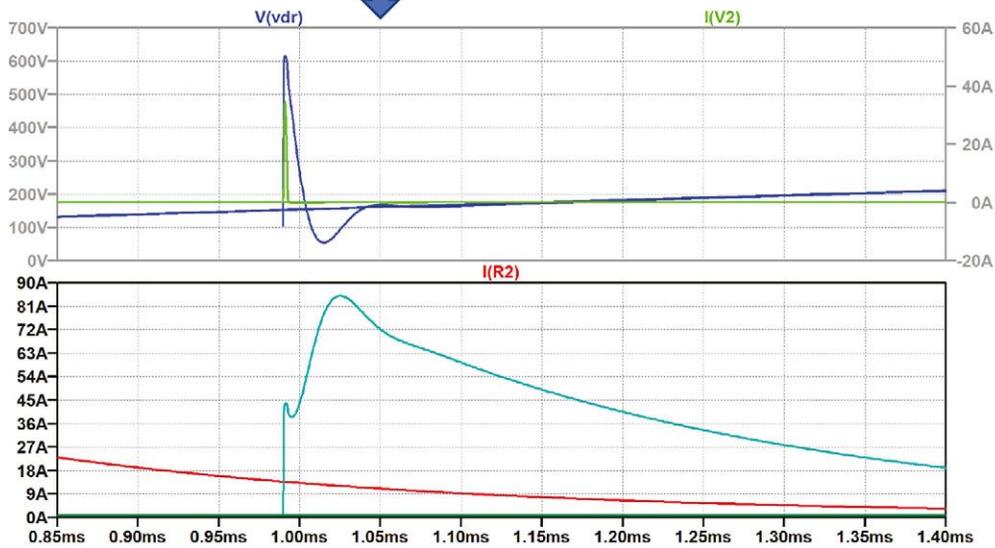


Simulations can be built (here in LTspice XVII) in order to visually show the effect of overvoltage spikes (like below) on the overcurrents in load (R2) or passing through the VDR (V2).





Close up



14. APPENDIX

LIST OF ALL SIMULATIONS FOR THERMISTORS / RTDs AVAILABLE ON HACKSTER.IO	
SUBJECT	SOFTWARE
Altium Designer Simulation Model for Vishay NTC Thermistor	Altium designer
ADN8834 Simulation in Action With TEC and Thermistor 🏆	LTspice XVII
Analog Amplitude to Frequency Conversion for NTC Thermistor	LTspice XVII
Another Fire Alarm SPICE Simulation With Thermistor	LTspice XVII
Another Proportional Controller for Thermoelectric Peltier	LTspice XVII
Another Simulated Buzzer Alarm System With NTC Thermistor	LTspice XVII
Battery Temperature Protection With LTC4079 and NTC	LTspice XVII
Double 555 and NTC Thermistor Based Buzzer	LTspice XVII
For a Few Simulations More - Part 2 - Thermostats With NTC	LTspice XVII
For a Few Simulations More - Part 7 - Thermistor, Optotriac, and PWM	LTspice XVII
For a Few Simulations More - (Thermostats With Vishay NTC)	LTspice XVII
For a Few Simulations More - Part 3 - Oven Control	LTspice XVII
For a Few Simulations More - Part 5 - NTC and Optocoupler	LTspice XVII
For a Few Simulations More - Part 4 - Thermostats With NTC	LTspice XVII
For a Few Simulations More - Part 6 - Return on Optocouplers	LTspice XVII
Heater - Cooler Simulation: Return to Basics	LTspice XVII
Limit Cycle Temperature Control With Vishay NTC Thermistor	LTspice XVII
Linear Output Temperature Sensing Circuit With Vishay PTS	LTspice XVII
LTspice for PID Control of a Thermoelectric Cooling Element 🏆🏆	LTspice XVII
LTspice Inverter Simulation With Thermal Effects 🏆🏆	LTspice XVII
LTspice Simulation of a Fire Detector (With Vishay NTC) 🏆	LTspice XVII
LTspice Simulation of Cold Temperature Alarm Circuit	LTspice XVII
LTspice Simulation of Heat Alarm With NTC and Transistors	LTspice XVII
LTspice Simulation of Improved Fire Detector (Rate of Rise)	LTspice XVII
LTspice Simulation of LED Current Control	LTspice XVII
LTspice Simulation of PWM Heating With Thermistor 🏆	LTspice XVII
LTspice Simulation of Thermostat With 555 and Vishay NTC	LTspice XVII
LTspice Simulation of Thermostat With Transistors and NTC	LTspice XVII
LTspice Simulation ON / OFF Temperature Control via VOT8125 and NTC	LTspice XVII
Maximal Linearity for Temperature Sensing With NTC (LTspice)	LTspice XVII
Oil Temperature Measurement Simulation With Vishay NTCLE350 🏆	LTspice XVII
Piezoelectric Strain Gage Temperature Compensation With NTC	LTspice XVII
Simulate Accurately Your NTCLE317E4103SBA Temperature Accuracy	LTspice XVII
Simulation of Analog PID Temperature Control With Thermistor	LTspice XVII
Simulation of RTD Temperature Compensation for Log Converter	LTspice XVII
SPICE Modelling of a Peltier Cooled "Laser" With Thermistor	LTspice XVII
SPICE Simulation of 4001 Controlled NTC Thermostat	LTspice XVII
SPICE Simulation of Cooling / Heating Peltier With Thermistor 🏆	LTspice XVII
SPICE Simulation of LED Dimming With 555 and NTC	LTspice XVII
SPICE Simulation of Overcurrent Protection With Vishay PTC 🏆	LTspice XVII
SPICE Simulation of Power Semiconductor Protection With PTC	LTspice XVII
SPICE Simulation of Temperature Derating of Boost Converter	LTspice XVII
SPICE Simulation of Temperature Detector With Buzzer and NTC	LTspice XVII



LIST OF ALL SIMULATIONS FOR THERMISTORS / RTDs AVAILABLE ON HACKSTER.IO	
SUBJECT	SOFTWARE
SPICE Simulation of Temperature Stabilized Amplifier Stage	LTspice XVII
TC621- TC4469 and NTC Thermistor Circuit in LTspice	LTspice XVII
Temperature Control With LTC1040	LTspice XVII
Dynamic Modeling of a Vishay NTC Thermistor in Micro-Cap 	Micro-Cap
Multisim Model for a Vishay NTC Thermistor  	Multisim
Multisim Simulation of Strain Gage Bridge Compensation	Multisim
Simulation of PTC Overvoltage Protection	PartQuest Explore
PSpice Simulation With NTC With Temperature Driven as Voltage	PSpice
Vishay Thermistors in PSpice for TI	PSpice for TI
SABER RD Version of a Dynamic Vishay NTC Thermistor	Saber RD
Electronic Simulation of NTC Thermistor in SIMetrix	SIMetrix
PTCEL Current Limiter (Modeled in SIMetrix Simulator)	SIMetrix
Dynamic RTD -PTS - Pt Thin Film Model Simulation in TINA TI	TINA TI
Electronic Simulation of Dynamic Thermistor in TINA-TI	TINA TI

Altium Designer:

[Altium Designer - PCB Design Software](#)

LTspice XVII:

[LTspice Information Center | Analog Devices](#)

Micro-Cap:

[Spectrum Software - Micro-Cap 12. Analog Simulation, Mixed Mode Simulation, and Digital Simulation Software. SPICE and PSpice® Compatible Circuit Simulator. \(spectrumsoft.com\)](#)

Multisim:

[Multisim Live Online Circuit Simulator](#)

PSpice:

[Electronic Circuit Optimization & Simulation | Cadence PSpice | PSpice](#)

PSpice for TI:

[PSpice-FOR-TI Simulation Tool | TI.com](#)

Saber RD:

[SaberRD \(synopsys.com\)](#)

SIMetrix:

[SIMetrix Circuit Design and Simulation](#)

Tina TI:

[TINA-TI Simulation Tool | TI.com](#)