Digital Simulation of Temperature Control Circuits With QSPICE™ and Vishay Thermistors

By Alain Stas

The article “The Art of Simulating Electronics with Thermistors and RTDs” [1] inspired the following remark from an assiduous reader: “Many temperature control circuits are now digital and microcontroller-based.” So, how can we reconcile the principles described in that article and its predecessors [2][3] with modern digital electronics?

At first glance, it seemed an impossible task. However, after thinking twice, we also realized that the control processes used in electronics have not changed much during the last 50 years - we still have on / off temperature control, pulse width modulation (PMW), and PI / PID processes. It is only the concrete form in which these processes are brought together that has changed. For the most part, the whole circuit is now digitized, with analog transistors or ICs being replaced by digital modules.

In contrast, in many applications the sensor part remains analog, and will probably remain so for years. The past has proven that the most reliable, precise, rugged, and cheap solutions for temperature sensing and control are negative temperature coefficient (NTC) thermistors and resistance temperature detectors (RTD) like thin film Ni and Pt components, especially in automotive applications. So, SPICE thermistor models that include all the component features like tolerance, self-heating, and response time, are still of use, as the sensing part remains analog. However, in theory they need to be rebuilt to be compatible with modern electronics.

There is a variety of software that allows for digital simulations (SYNOPSYS Saber RD, SIEMENS EDA’s Xpedition AMS with VHDL or VERILOG -AMS, or Simulink Simscape from The Mathworks) and they are perfect when you can afford to pay for them. However, most of them currently use generic components, especially for the passives. The author of this article knows very well that the introduction of complete, precise passive elements models in all this software has only begun. (See reference [4] for an introduction to real thermistor models in SIMULINK / SIMSCAPE)

While all these software variations are actively used, it is a fact that SPICE analysis remains the most popular way to analyze an electronic circuit. And in the future, standardization of all hardware description languages is simply not in the cards, as every company is fighting for their piece of the pie.

Until recently, one could only dream of a free equivalent to software like LTspice (first introduced by Linear Technologies, now part of Analog Devices) [5] that could include some digital hardware to describe digital electronics control devices. Fortunately, that dream has now become a reality with QSPICE™ software from Qorvo [6].

QSPICE appeared in the first half of 2023 and its creator is no other than Mike Engelhardt, who was involved in the creation of LTspice. It’s beyond the scope of this article to explain the many differences between QSPICE and LTspice. For our purposes, it’s enough to know that QSPICE supports VERILOG and C++. So, in the same way you import your SPICE netlist into the software, you can now describe digital electronics with circuit blocks and interface the new described logic hardware with the analog sensing and control parts of the circuit. This feature is the missing link between the SPICE of yesterday and the SPICE of tomorrow. The rest of this article will be dedicated to exploring cases of temperature control circuits being described into VERILOG modules.

The first case is an on / off temperature control circuit (Figure 1) using a bistable RS latch. It features a safety threshold that prevents the circuit from heating indefinitely in the event of an open circuit. The VERILOG module is the gray box marked RS_latchFS (FS stands for failsafe). An advantage of using the VERILOG module to build the micro controller is that the failsafe can be implemented only by programming the ADC conversion of the temperature signal.

Digital Simulation of Temperature Control Circuits With QSPICE™ and Vishay Thermistors

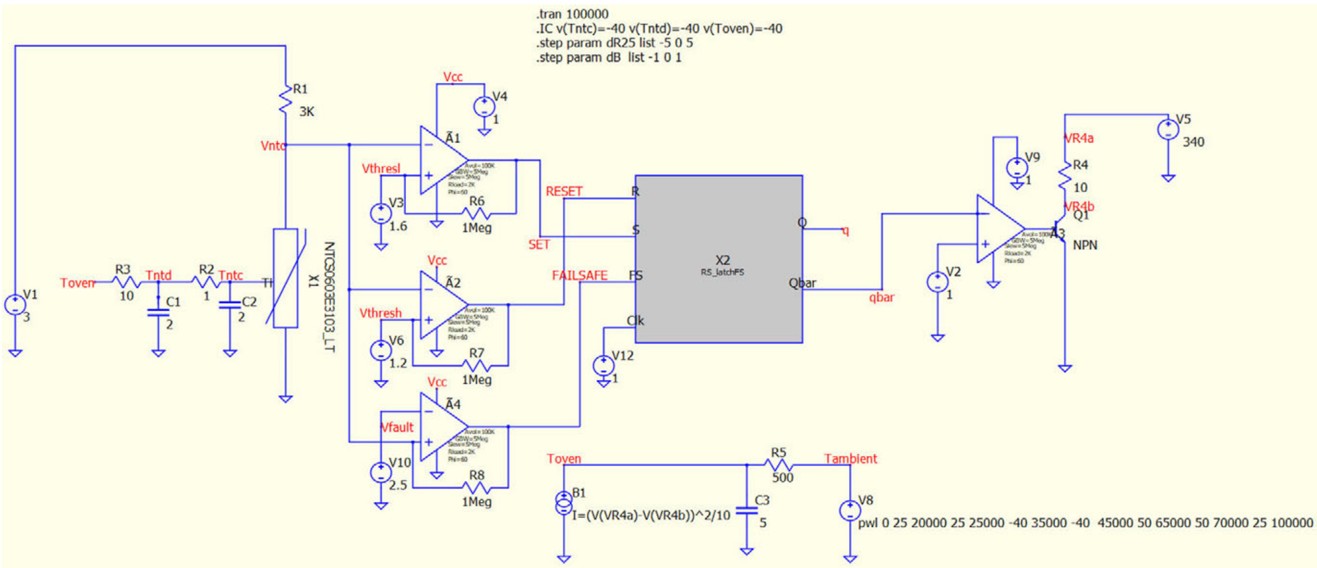

Fig. 1 - Thermal control circuit of an oven with RS latch logic, including failsafe

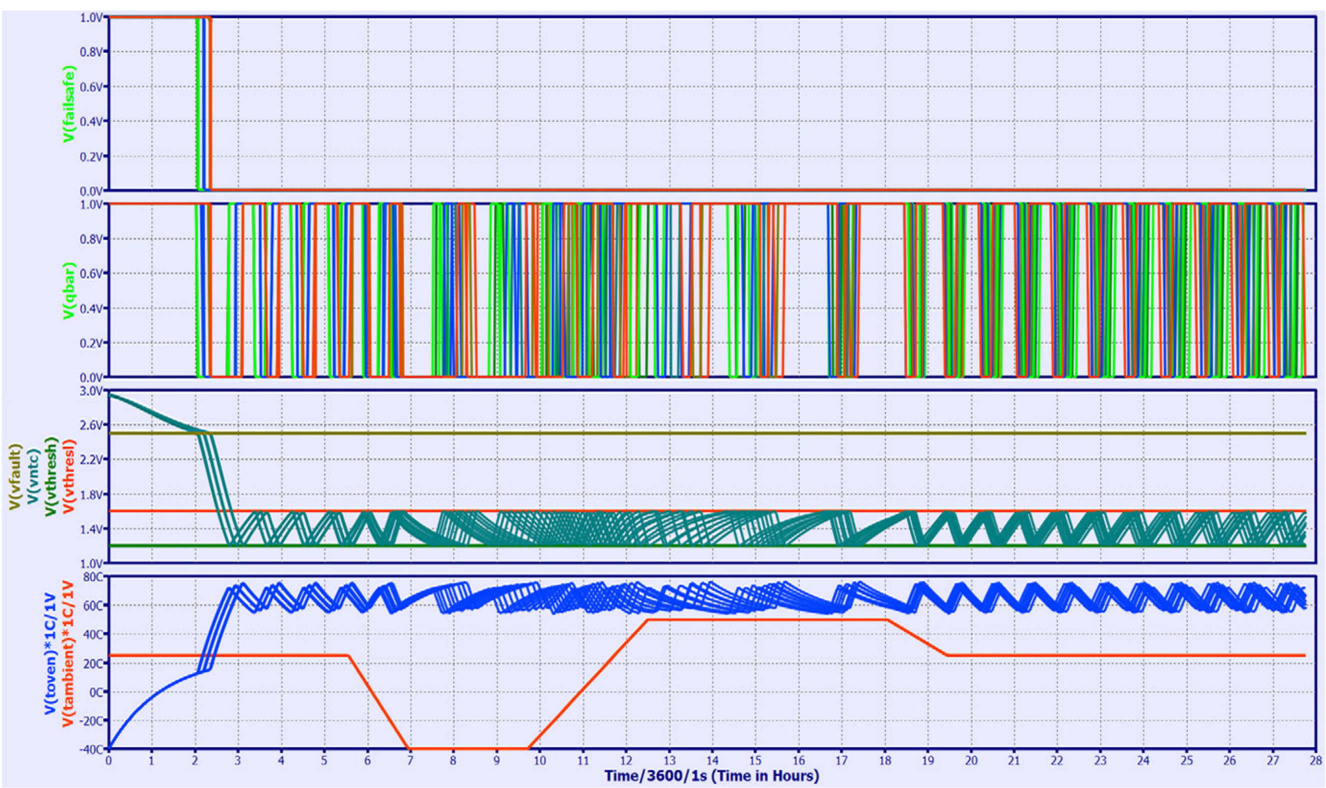

Fig. 2 - High pane: failsafe voltage; second pane: output RS; third pane: different voltage; low pane: oven and ambient temperatures

Digital Simulation of Temperature Control Circuits With QSPICE™ and Vishay Thermistors

We can see from the screen captures in Figure 2 that the oven temperature (V(oven) lower pane), which has been intentionally initialized at -40 °C, induces the failsafe of the RS-latch to be switched on (V(failsafe) is equal to 1 V in the upper pane from time 0 h to 2 h). The heating that should happen is prevented, but the oven temperature then tends to increase to get back naturally to the ambient temperature of 25 °C. Once the oven has reached the limit threshold lower temperature (10 °C) after 2 h, corresponding to the failsafe voltage of 2.5 V, the heating then takes place until the oven temperature reaches the target temperature (near 80 °C). Then the heating stops and the oven cools down until the lower threshold (near 60 °C), and the cycle starts up again. As the ambient temperature changes, the duration of the limit cycle is adapted. The total elapsed time for the simulation is 1 s for a real application time of 28 h. Of course, the model's parameters are customizable, so anyone can adapt it to their own application.

QSPICE allows the user to sweep the NTC parameters dR25 (± 5 %) and dB (± 1 %), which provides the bundle of curves showing the limit cases.

The second example of a complete VERILOG module exemplifies the PWM excitation of a Vishay VOT8125 optotriac triggering another triac, the L01030ME (Figure 3). This last triac is commuting an AC main voltage onto a load, which produces the current used to heat an oven (Toven). The oven temperature is measured by an NTCS0603E3103FJT NTC thermistor (10 kΩ, ± 5 %), with some response time delay fixed by the R11C2 coupler. The oven temperature-dependent voltage generated by the thermistor is converted by the controller's internal ADC, while a sawtooth signal is produced by a simple counter to generate the PWM. All of this is included in the VERILOG module.

When the output of the controller is low, the optotriac is switched on. At the startup of the heating, the duty cycle will be high (Figure 4a), and as we approach the steady state, it will be low (Figure 4b).

The temperature of the oven increases from the initial ambient to the steady-state temperature fixed by R7 (6000 Ω). Four extreme cases are represented and depend upon the thermistor's tolerance (Figure 5).

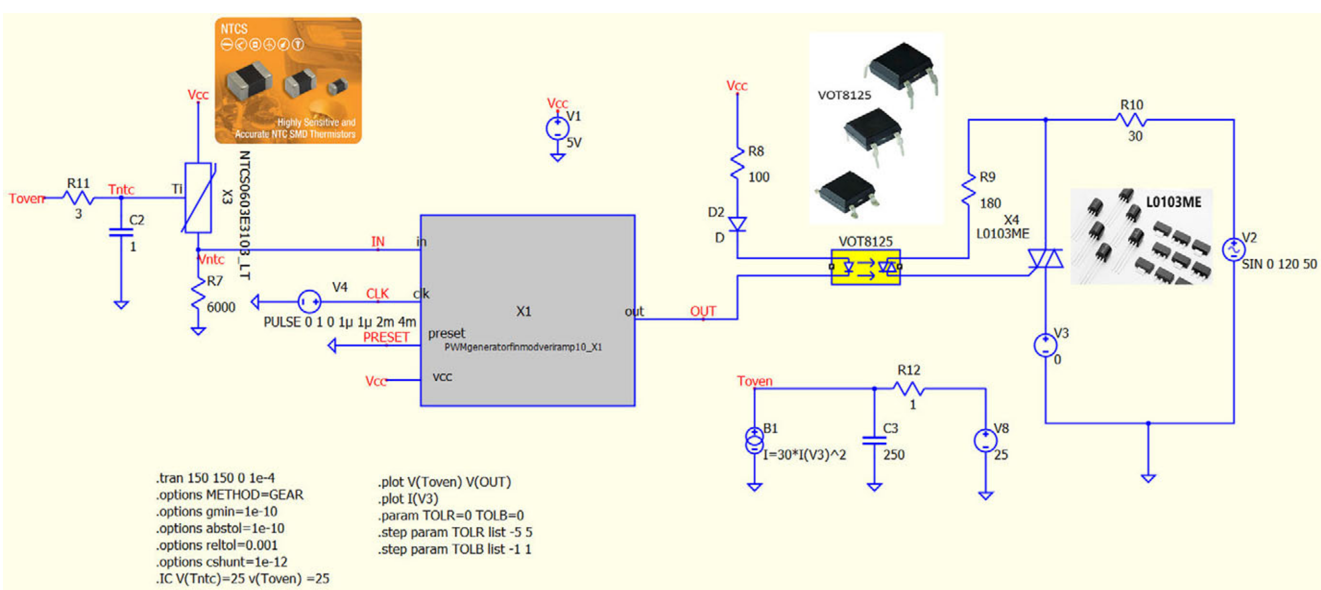

Fig. 3 - Thermal control of an oven with optotriac and PWM controller

Digital Simulation of Temperature Control Circuits With QSPICE™ and Vishay Thermistors

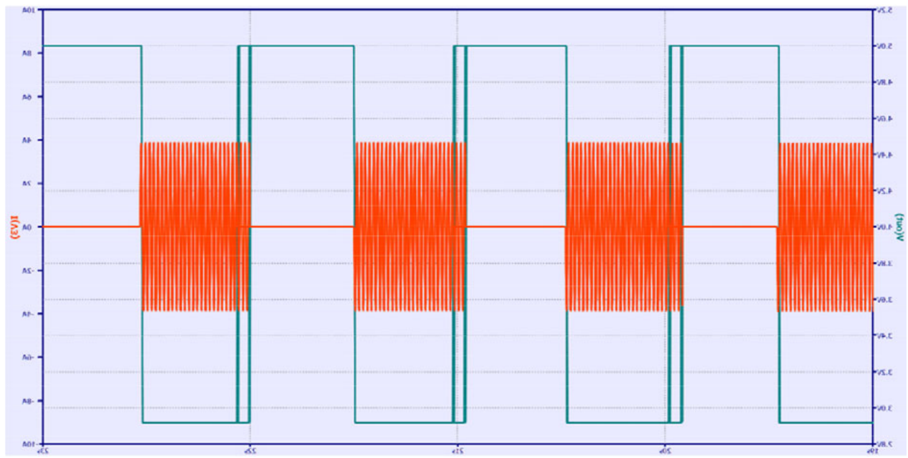

Fig. 4a - Mid duty cycle

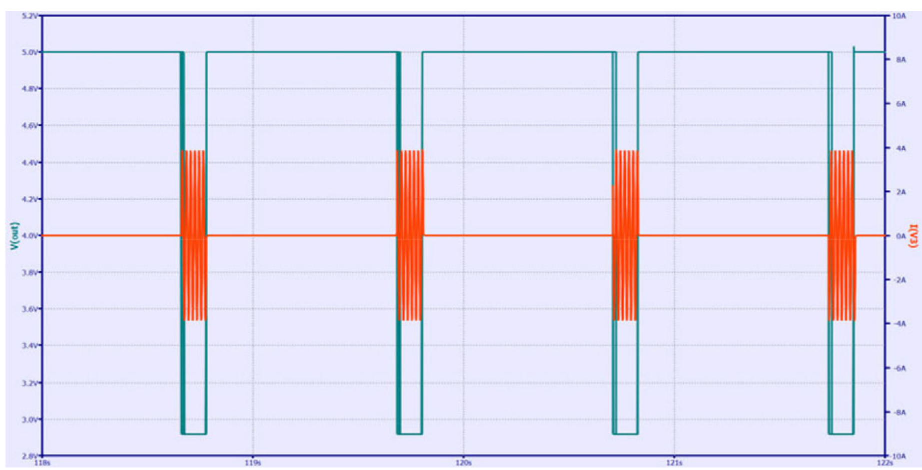

Fig. 4b - Short duty cycle

Digital Simulation of Temperature Control Circuits With QSPICE™ and Vishay Thermistors

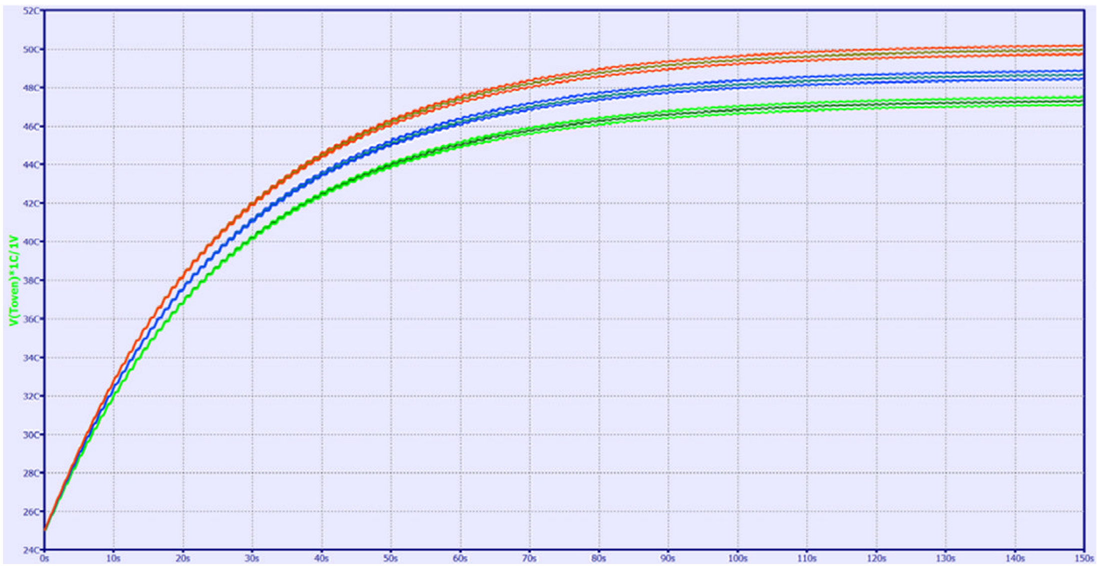

Fig. 5 - Oven temperature resulting from the circuit in Fig. 3

The third and last example uses a more evolved controller to tune a PWM current in a Peltier thermoelectric element, which cools down a heat source represented by a pulse current source of 50 W with 5 pulses every 100 s (Figure 7 third pane). A first real NTC thermistor measures the cold side of the Peltier, and another thermistor (virtual this time) is used to indicate the reference temperature to be kept for the cold side (here 25 °C.) We can see on the screen captures that the temperature control works effectively, as each time a heat pulse is generated, the PWM works actively (Figure 7, second pane). Also, the PWM duty time is adapted to the ambient temperature, changing slowly in time.

The strength of PWM modulation can be tuned up or down with the EFF parameter and two values are illustrated in the high pain of Figure 7 (blue: strong PWM; magenta: lower PWM). With the stronger PWM, it is visible in the second pane that the portion of time with I(R1) blocked at maximum value is higher.

Digital Simulation of Temperature Control Circuits With QSPICE™ and Vishay Thermistors

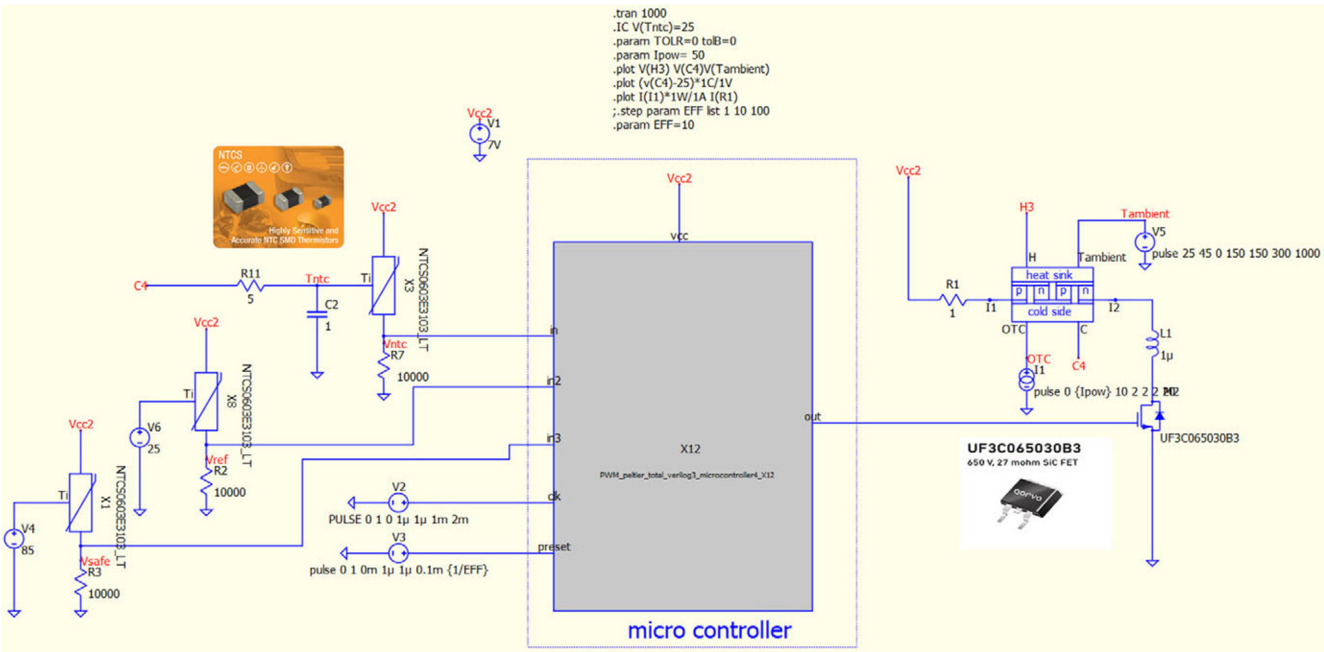

Fig. 6 - Temperature control circuit with Peltier thermoelectric element and PWM microcontroller

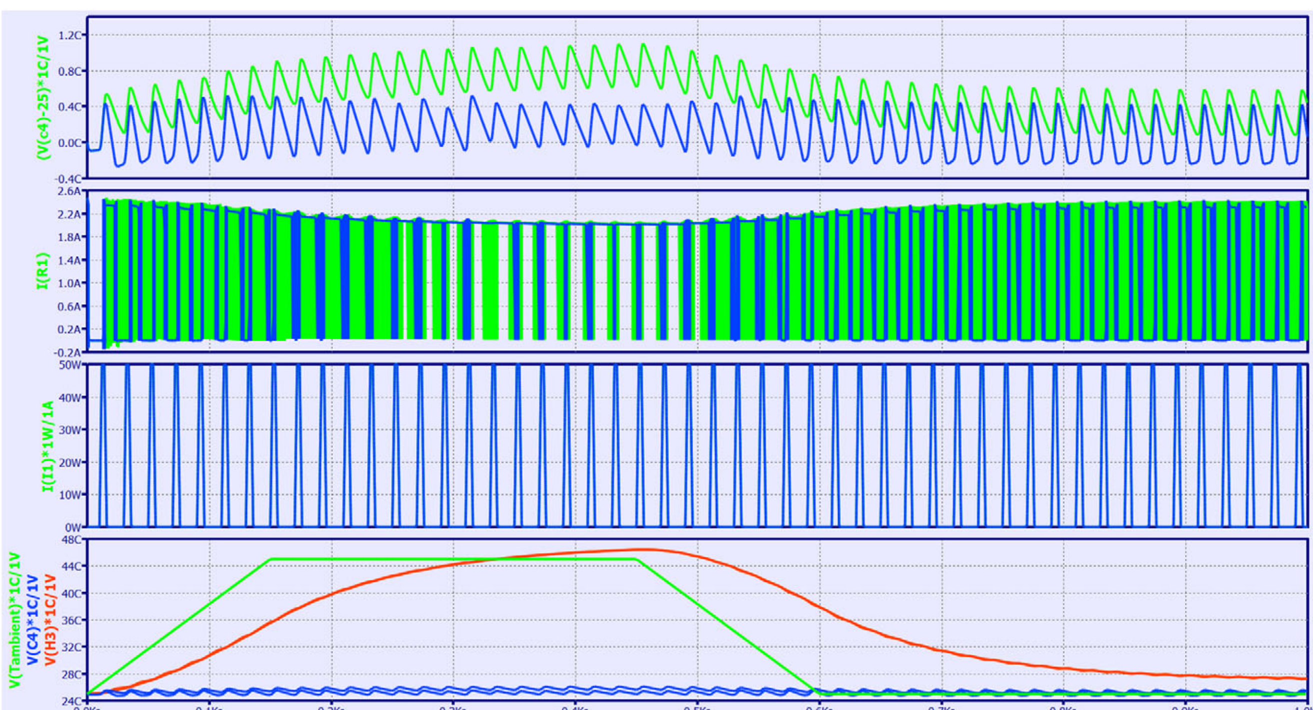

Fig. 7 - High pane: offset real controlled temperature vs target temp; second pane: heating current; third pane: heat source OTC power peaks; lower pane: temperatures of cold side C4 / hot side H3 and ambient temperature Tambient

Digital Simulation of Temperature Control Circuits With QSPICE™ and Vishay Thermistors

With these three examples of simulation in QSPICE involving VERILOG modules, we hope to have brought a new brick to the building dedicated to the great subject of digital temperature control simulation. By keeping our dynamic SPICE models of Vishay NTC thermistors with all their original features, and using other advanced devices (like Vishay opto-triacs, Qorvo SiC transistors, and Peltier elements) in conjunction with simple QSPICE-compatible VERILOG modules, it has been possible to demonstrate the optimization of digital temperature control simulations in order to prepare for faster future experimentation. The three examples of QSPICE simulations can be freely downloaded on the Hackster.io website [7][8][9].

REFERENCES

[1] Die Kunst der (simulierten) Elektronik (mit Thermistoren und RTDs)
SPICE-Simulationen für Thermistoren und RTDs | DigiKey

[2] 2021: A Simulation Odyssey for Thermistors, Part 2 - Planet Analog

[3] Old-School Analog Temperature Control Circuits Solved with Modern LTSpice Thermistor Dynamic Models, Part 3 - Planet Analog

[4] Thermistors (NTC and RTD) from Vishay Modeled in Simscape - File Exchange - MATLAB Central (mathworks.com)

[5] LTspice Information Center | Analog Devices

[6] QSPICE™ Simulator - Qorvo

[7] QSPICE™ ON /OFF Temperature Control Simulation with NTC - Hackster.io

[8] PWM Controller VERILOG in QSPICE™ (with ADC and Counter) - Hackster.io

[9] A Simple PELTIER Temperature Controller in VERILOG/QSPICE™ - Hackster.io