



A Multi-Simulator NTC Thermistor SPICE Model With Temperature Driven by a Voltage

By Alain Stas, Product Marketing Engineer for Non-Linear Resistors

To help designers who need to evaluate the response of a sensor to a defined dynamic temperature profile, Vishay now offers a multi-software simulation of an NTC thermistor whose temperature is driven by an external voltage. The same principles of simulation have been tested with success in no less than six of the most popular electronic simulators. The SPICE model syntax and structure have been adapted to fit each software package.

In the classic spice model of an NTC thermistor, the temperature is simulated by the embedded TEMP variable. This is ideal for the examination of the general response of a circuit when external ambient temperature is swept. Such PSpice and LTspice models are available for Vishay thermistors. However, such modeling is not convenient when you want to evaluate the response of a sensor to a defined dynamic temperature profile. To address this issue, the new model presented here uses the thermistor temperature at a third virtual pin for simulation purpose only, which must be connected to an external voltage. This external voltage is thus the dynamic temperature of the thermistor according to the user's application needs.

For our simulation, the NTC is placed via the two physical pins into a voltage divider with a fixed resistor and a DC source. The voltage between these two pins follows the temperature variation applied to the third virtual pin T_i of the NTC. The eventual self-heating (due to the current) is taken into account in the model U1, but the reaction time of the thermistor is not part of the model and must be built separately, as in the circuit in Fig. 1.

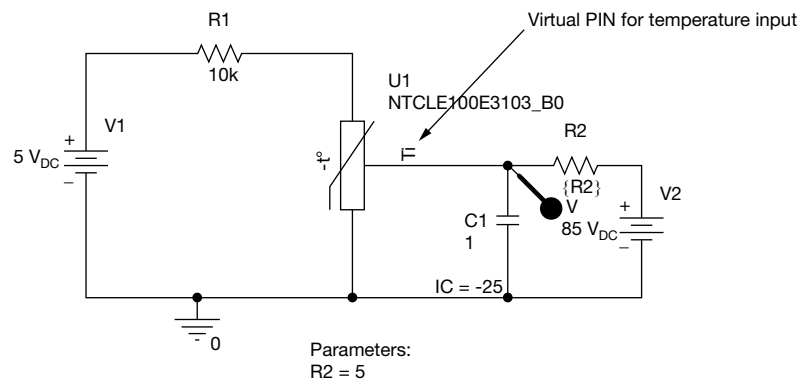


Fig. 1 - Base circuit

In the simple example in Fig. 1, the thermistor's response to an external temperature step change is simulated by a R2C1 circuit, with capacitor C1 (initial state 25 V, as the thermistor is at 25 °C) being charged by a voltage source V2 of 85 V (final temperature of 85 °C) through a resistor R2, whose value is in fact the response time of the thermistor in the application conditions. The RC circuit thus simulates the thermal inertia of the sensor to an external temperature change.

This circuit is thus the base of a temperature-sensing circuit, where the temperature can follow the application conditions with complete flexibility. The user can perform a fitting of the practical response time of their sensor with the value of R2, or, for more complex situations, even replace the RC circuit and DC source V2 by a programmable PWL source with a pre-recorded profile describing the thermal response of the sensor exactly.

The same fundamental simulation is made directly available in six simulators. To see the differentiation of the SPICE syntaxes of these software, Vishay has run through the import procedures for each of the chosen programs and resolved any problems. For the design engineer, this will save time as they can let the simulation run as received and concentrate on their own application and not on the usual warning messages like "line too long: limit is 132 characters," "function not recognized," and "too many parameters for sub-circuit type," to name but a few.

A Multi-Simulator NTC Thermistor SPICE Model With Temperature Driven by a Voltage

Here is the selected software listed in alphabetic order, together with a Vishay tested thermistor and the availability of the full NTC library:

- **Altium Designer 16** with the NTCLE101E3C90173 (no library)
- **Cadence OrCAD PSpice 16.6** with the NTCLE100E3103_B0 (complete library available)
- **LTspice IV (or LTspice XVII 64 bits)** with the NTCLE203E3202_B0 (complete library available)
- **NI Multisim Designer 14.0** with the NTCLE101E3C90173 (no library)
- **SIMetrix** with the NTCLE101E3C90173 (no library)
- **Tina TI** with the NTCS0805E3472_MT (library available: netlist text must be copy / pasted)

When the library is available, the part number of the thermistor can be modified in order to select other resistance temperature characteristics. When a library is not available, the electrical parameters must then be manually changed in the netlist associated with the model.

The screen captures resulting from the performed simulations for each software are shown below. The graphs in Fig. 2 through Fig. 7 show the component temperature rising exponentially from 25 °C to 85 °C, as well as the corresponding negative voltage variation in the divider bridge. In the LTspice and PSpice simulations (Fig. 3 and Fig. 4), the response time is even swept in a range via the resistance R2, showing the possible variations in the speed of a sensor's response.

On this basis, designers will be able to further develop their own applications in the software of their choice.

For the design engineers specifically using LTspice, an extended model (including Monte Carlo tolerances and a more complex transfer function) already exists and is presented at the following address:

www.digikey.com/en/ptm/v/vishay-bc-components/advanced-lt-spice-modeling-features/tutorial.

However, the LTspice model functionalities are not directly compatible with the other software.

The multi-software, voltage-driven thermistor model is available on request at the following address: edesign.ntc@vishay.com.

Please indicate the software name, and if you already have an idea of the thermistor type that you want to use. A Vishay engineer will get back to you in a timely manner to help in developing your application.

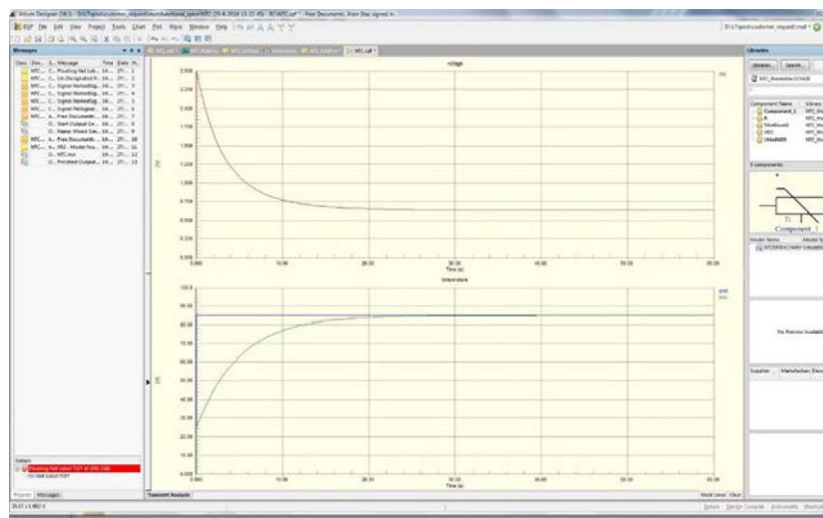


Fig. 2 - Screen capture in Altium Designer 16

A Multi-Simulator NTC Thermistor SPICE Model With Temperature Driven by a Voltage

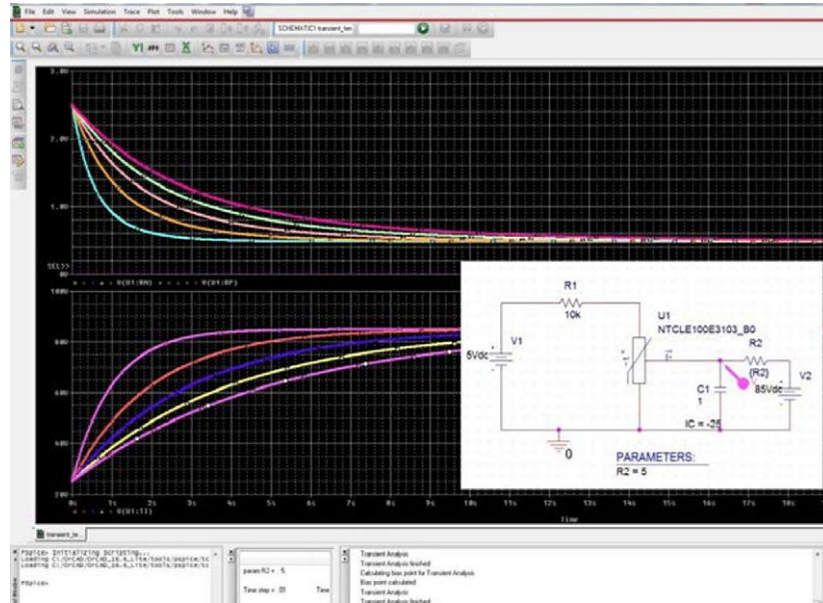


Fig. 3 - Screen capture in PSpice 16

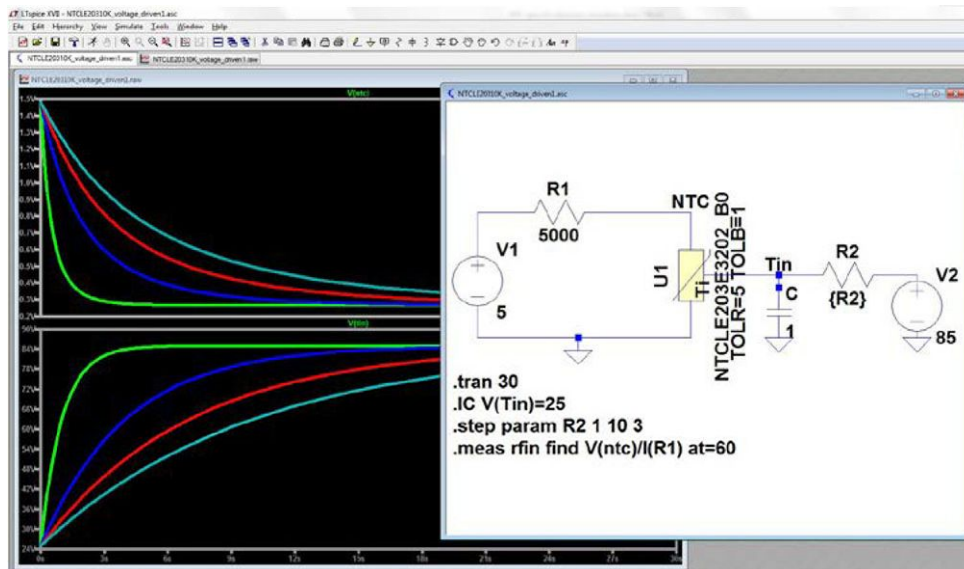


Fig. 4 - Screen capture in LTSpice IV

A Multi-Simulator NTC Thermistor SPICE Model With Temperature Driven by a Voltage

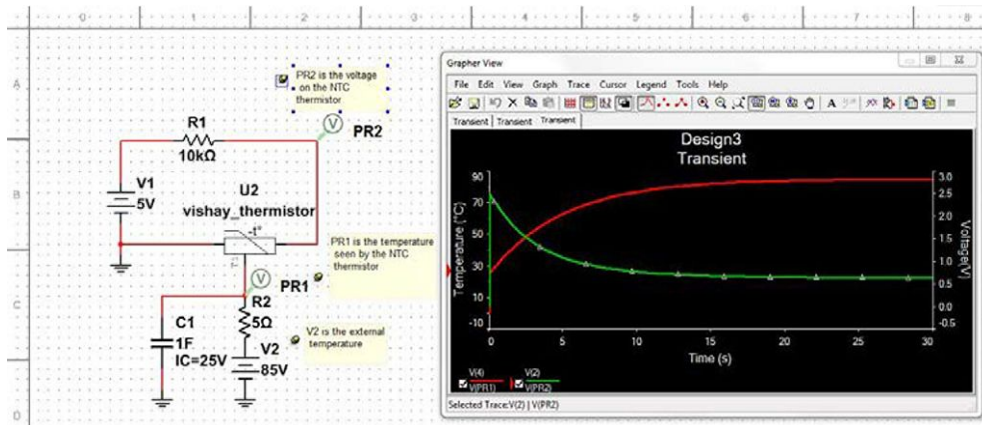


Fig. 5 - Screen capture in Multisim Designer 14.0

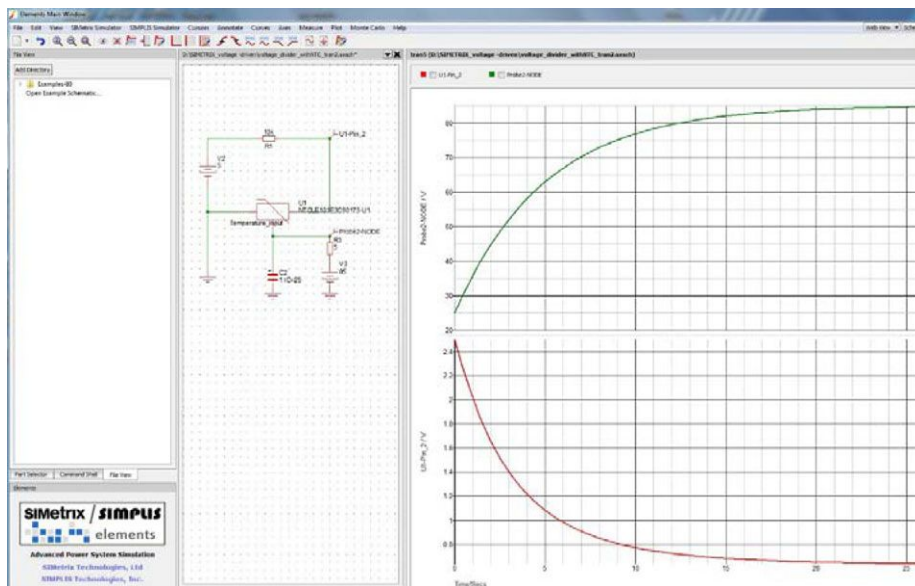


Fig. 6 - Screen capture in SIMetrix

A Multi-Simulator NTC Thermistor SPICE Model With Temperature Driven by a Voltage

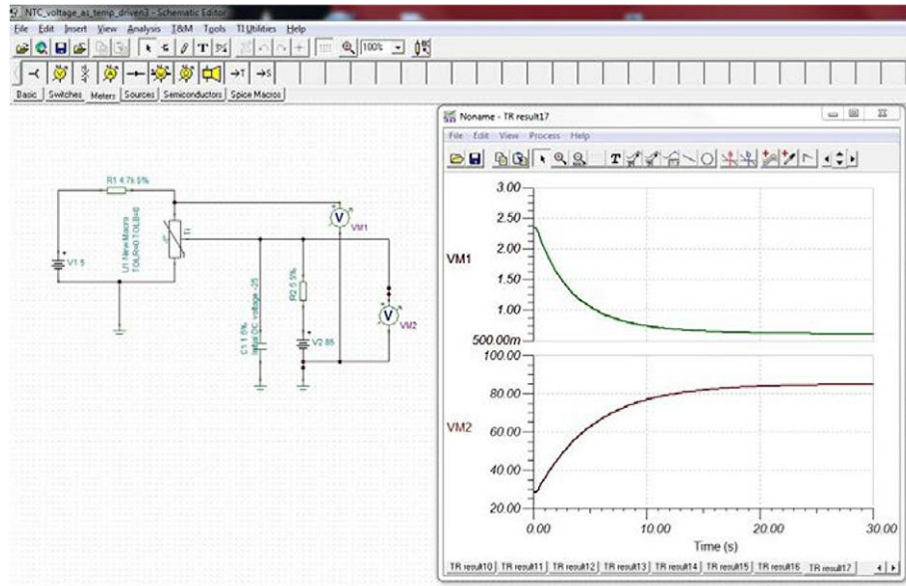


Fig. 7 - Screen capture in Tina TI