CIRCLE 521

MODEL THERMISTOR WITH SPICE

JIM HAGERMAN
5137 Camino Playa Malago,
San Diego, CA 92124; (619) 931-5012.

With this Spice subcircuit, a typical resistive component can be replaced by a two-terminal thermistor in Spice simulations *(Fig. 1)*.

Most Spice-based simulators permit a temperature-dependent model of a resistor. However, the model is limited to a second-order polynomial that may not result in an accurate model of the actual component's characteristic, especially over a wide temperature range.

To overcome this problem, the subcircuit uses the high-order polynomial description of the nonlinear voltage-controlled current source. In this way, any polynomial (of nth degree) that describes the thermistor's resistance as a function of temperature can be applied directly in Spice. There are several known methods for generating these polynomials, and numerous easy-to-use software packages can simplify this task.

The model's accuracy is mostly limited by the accuracy of the polynomial. Using TableCurve 2.0 from AISN Software, Grants Pass, Ore., two polynomials were generated to model the data listed for a Dale Electronics negative-temperature coefficient (NTC) curve #1 from 0 to 100°C. The results for the two polynomials indicate good correlation with the expected data *(Figs. 2a and 2b)*.□

Reference:

Tuinenga, Paul, *Spice, A Guide to Circuit Simulation and Analysis Using PSpice*. Englewood Cliffs, NJ: Prentice-Hall, 1988.

```
*   NTC.SUB     Jim Hagerman 7/5/90
*
*   This is a model of a 10k Ohm thermistor with characteristics of a Dale Electronics NTC
    curve #1.
*
*   The nominal value of resistance at 25C is Rref.
*
*   The characteristics R(T) is used for Gout. Note that if Celsius is used TNOM is set to
    zero.

.options tnom=0
.subckt ntc 1 2
   Eout 1 3 poly(2) (5,0) (4,0) 0 0 0 0 1.0
   Vsense 3 2 dc 0.0
   Fout 0 4 Vsense 1.0
   Rref 4 0 10k
   Gout 0 5 poly(1) 6 0 3.266
   +              -0.16633619
   +               0.0046450693
   +              -8.6856965e-5
   +               1.017213e-6
   +              -3.8668603e-9
   +              -8.8615615e-11
   +               1.678045e-12
   +              -1.3013017e-14
   +               4.8617031e-17
   +              -6.8866237e-20
   Ro 5 0 1.0
   Itemp 0 6 dc 1.0
   Rt 6 0 Rtemp 0.001
   .model Rtemp res(r=1 tc1=1000)
.ends ntc
```

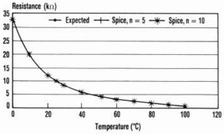
1. THIS SPICE SUBCIRCUIT serves as a model of a 10-kΩ thermistor. When simulating with Spice, the model can be used in place of a normal resistive component.

VOTE!

Read all the Ideas for Design in this issue, select your favorite, and circle the appropriate number on the Reader Service Card. The winner receives a $150 Best-of-Issue award and becomes eligible for a $1,500 Idea-of-the-Year award.

T	Resistance (kΩ)		
	Expected	Spice	
		n = 5	n = 10
0	32.66	32.63	32.66
10	19.90	19.98	19.90
20	12.49	12.45	12.49
25	10.00	9.949	10.00
30	8.058	8.025	8.058
40	5.326	5.360	5.326
50	3.602	3.652	3.602
60	2.488	2.488	2.488
70	1.751	1.701	1.751
80	1.256	1.229	1.256
90	0.9164	0.9751	0.9164
100	0.6792	0.6597	0.6793

(a)



(b)

2. ACTUAL SPICE SIMULATION results for two different values of n are compared with the expected values (a). A plot of the same data illustrates the model's resistance-temperature curve (b).