Changing the Temperature in PSPICE Simulations

Overview

Circuit behavior often has strong temperature dependence. For example, in bipolar circuits the current depends on $I_S \exp(V_{BE}/V_T)$ where the thermal voltage $V_T = k_B T/q$ (k_B = Boltzmann constant, T= absolute temperature and q=electronic charge). When the temperature changes, so does V_T. In addition, the scale current I_S depends on temperature. In PSPICE, for example, I_S is modeled as[1]

EQ. 1

$$I_S(T) = I_S\left(\frac{T}{T_{NOM}}\right)^{X_{TI}} e^{((T/T_{NOM}-1)(E_G/V_T))}$$

where I_S is the scale current at the "nominal" temperature T_{NOM} (set at 300K by default)[2] and T is the operating temperature of the transistor (both in degrees Kelvin). Other parameters appear in EQ. 1 with values that are set in the •model statement for the particular type of transistor under consideration (for example, for the bipolar type Q2N2222, X_{TI} = 3 and E_G = 1.11V).

Next is a discussion of how to control temperature in PSPICE, including setting individual temperatures for each device in the circuit.

Setting T_{NOM}

The default value[3] for parameter T_{NOM} is set by selecting PSPICE/EDIT PROFILE/SIMULATION SETTINGS and using the OPTIONS tab to get the menu in Figure 1.

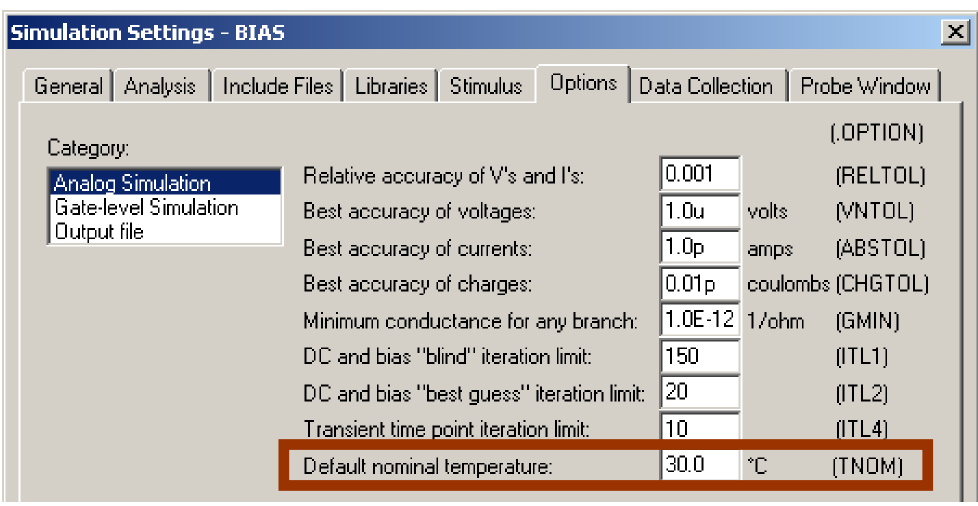

FIGURE 1

Setting the value of T_{NOM}

Changing T_{NOM} results in a notation in the OUTPUT file as shown below:

*Analysis directives:
•OP

[1] See the on-line PSPICE REFERENCE GUIDE, p. 212

[2] T_{NOM} is the temperature at which the model parameters have been fitted to get good *I-V* curves.

[3] This value can be <u>overridden</u> by specifying the variable T_MEASURED in the •model statement of the device.

•TEMP 27
•OPTIONS TNOM= 30.0

The entry •OPTIONS TNOM= 30.0 shows that TNOM has been set to 30ºC. The output file listing •TEMP 27 indicates that the <u>circuit</u> temperature is set at 27ºC. In other words, the menu of Figure 1 and the variable T_{NOM} are about the selection of the <u>model parameters</u>, and have nothing to do with the <u>circuit</u> temperature.

It is recommended that you leave this menu at the default value 27º C because this is the temperature where the model parameters were set up.

Changing Circuit Temperature T

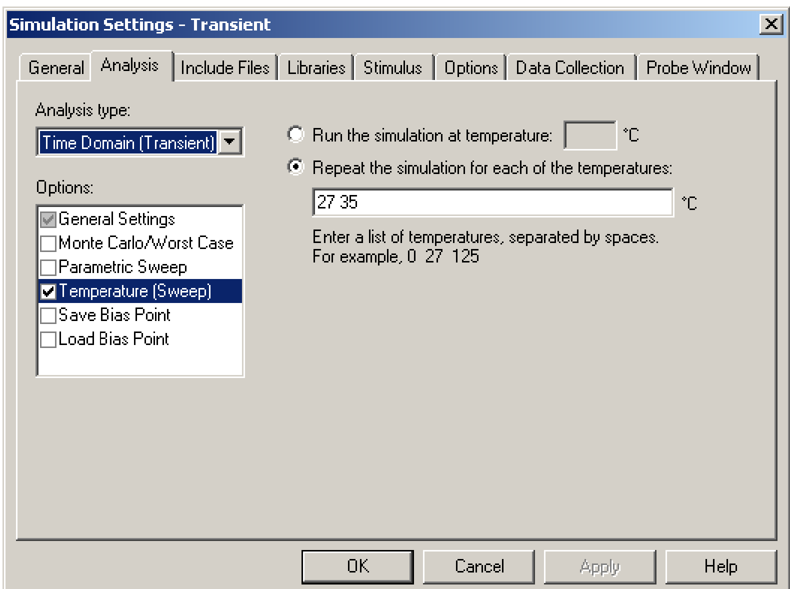

FIGURE 2

Menu for setting the circuit temperature (the "global" temperature) in the TRANSIENT simulation; in this case a request is made for <u>two</u> runs at the temperatures T=27ºC and T=35ºC.

The temperature of the entire circuit is set using the SIMULATION SETTINGS. For example, for a DC Bias simulation, we can follow the steps described by Herniter in his on-line supplement to his book. By selecting PSPICE/EDIT SIMULATION PROFILE and clicking on TEMPERATURE (SWEEP) we obtain the menu of Figure 2. In this menu, the temperature can be filled in, and checking the TEMPERATURE (SWEEP) box activates your choices.Changing the temperature leads to an entry in the OUTPUT FILE when the simulation is run like that below

*Analysis directives:
•OP
•TEMP 150

The line •TEMP sets the so-called "global temperature specification" of the simulation. That is, all the devices in the circuit are set to this temperature.

This approach works with other types of simulation as well. For example, for a TRANSIENT SIMULATION we can click on the ANALYSIS tab to obtain the menu shown in Figure 2. Here two temperatures have been selected, the default 27ºC and also 35ºC.

An example output waveform for a common emitter amplifier is shown in Figure 3. The corresponding added lines in the OUTPUT file are

*Analysis directives:
•TRAN/OP 0 2m 0 1u
•TEMP 27 35

In addition, before the listing for each simulation the OUTPUT file lists the temperature as shown below for the 35ºC run:

****   INITIAL TRANSIENT SOLUTION   TEMPERATURE =   35.000 DEG C

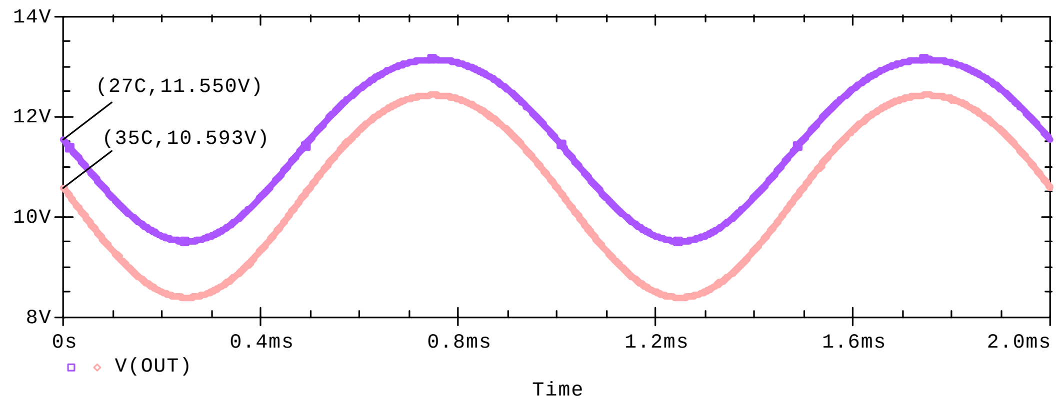

FIGURE 3
Example output waveform for two temperatures

Temperature adjusted •model parameters

If there are devices in your circuit, you can look at the •model parameters. For example, in the circuit of Figure 4 the temperature has been set using the menu of Figure 2 at T=127ºC. When the BIAS simulation is run, the output file shows the • model parameters of Figure 5.

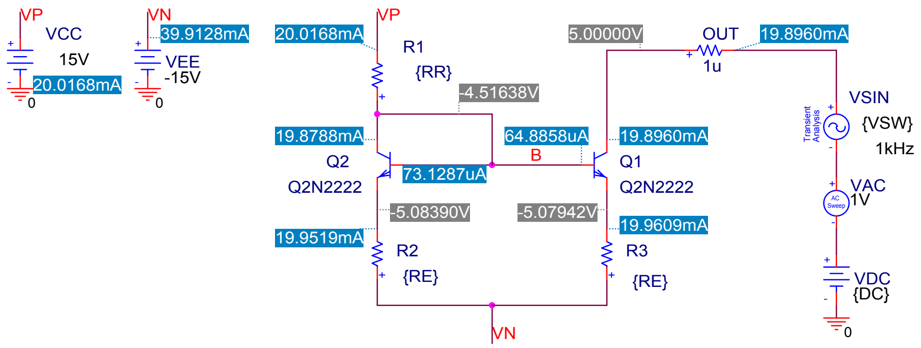

FIGURE 4
Example current-mirror circuit where the temperature has been set at T = 127ºC

Q2N2222		ISE	1.43E-14	MJE	0.377	TR	4.69E-08
	NPN	NE	1.307	CJC	7.31E-12	XTB	1.5
IS	1.43E-14	BR	6.09E+00	MJC	0.3416	CN	2.42E+00
BF	255.9	NR	1	TF	4.11E-10	D	0.87
NF	1.00E+00	RB	10	XTF	3		
VAF	74.03	RC	1	VTF	1.70E+00		
IKF	0.2847	CJE	2.20E-11	ITF	0.6		

FIGURE 5

Output file listing of •model parameters for Figure 4 at T_{NOM}

However, in addition, the output file shows the temperature-adjusted model parameters of Figure 6. Note, in particular, that the scale current has changed in accord with EQ. 1, and that BF has changed as well. Most other parameters also have changed somewhat, indicating that the detailed temperature dependence is too complex for hand analysis.

Q2N2222		RB	10	**IS**	**1.545E-09**	VTF	1.7
		BR	9.377	ISS	0	ITF	0.6
BF	**393.9**	ISC	0	VJS	0.569	TR	4.691E-08
ISE	6.602E-11	VJC	0.569	CJS	0	XTB	1.5
VJE	0.569	CJC	8.008E-12	GAMMA	1E-11	CN	2.42
CJE	2.434E-11	RC	1	RCO	0	D	0.87
RE	0	RBM	10	VO	10		

FIGURE 6

Output file listing for temperature-adjusted •model parameters at T=127ºC

Individual device temperatures

The above approach is fine if you want all the devices at the same temperature. However, in some circuits the operation is greatly affected when different devices have different temperatures. An example is the differential amplifier shown in Figure 7[4]. This circuit depends on both transistors being matched, that is, both having the same model parameters.

In Figure 7 the transistor temperatures relative to the global temperature are controlled by the parameters Q_TEMP1 and Q_TEMP2. As seen in Figure 7, both output voltages OUT1 and OUT2 are the same in this case where both transistors are at the same temperature. However, as shown in Figure 8, when the relative temperature of one transistor is increased relative to the global temperature by changing Q_TEMP2, the two outputs are no longer the same.

How is this individual control of temperatures accomplished?

- First a Q2N2222 is pasted on the schematic. By clicking on the transistor to highlight it and selecting EDIT/PSPICE MODEL we put it into the •model editor for this device.
- We copy the •model statement for the Q2N2222, and close the model editor. Delete the Q2N2222 transistor from the schematic.

[4] The AC source plays no part in the Q-point simulation, so both bases are at DC ground in Figure 7 and Figure 8.

- Next, paste a BREAKOUT transistor QbreakN into the circuit[5]. QbreakN is a blank n-transistor, that is, a transistor with only default model parameters.

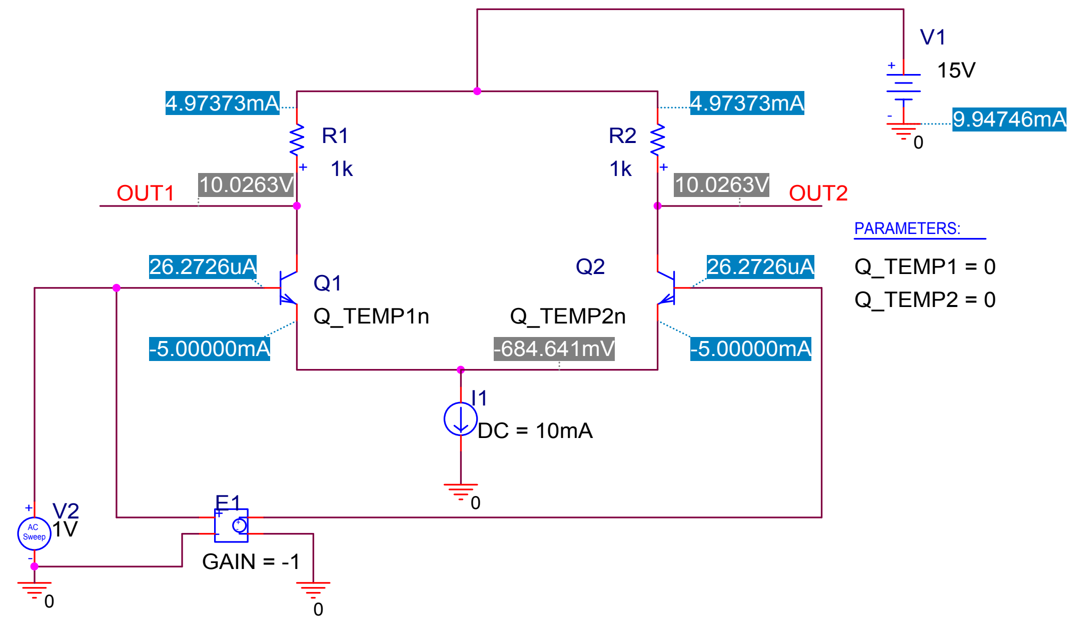

FIGURE 7
A differential amplifier allowing individual control of device temperatures; both transistors have the same temperature, 27ºC

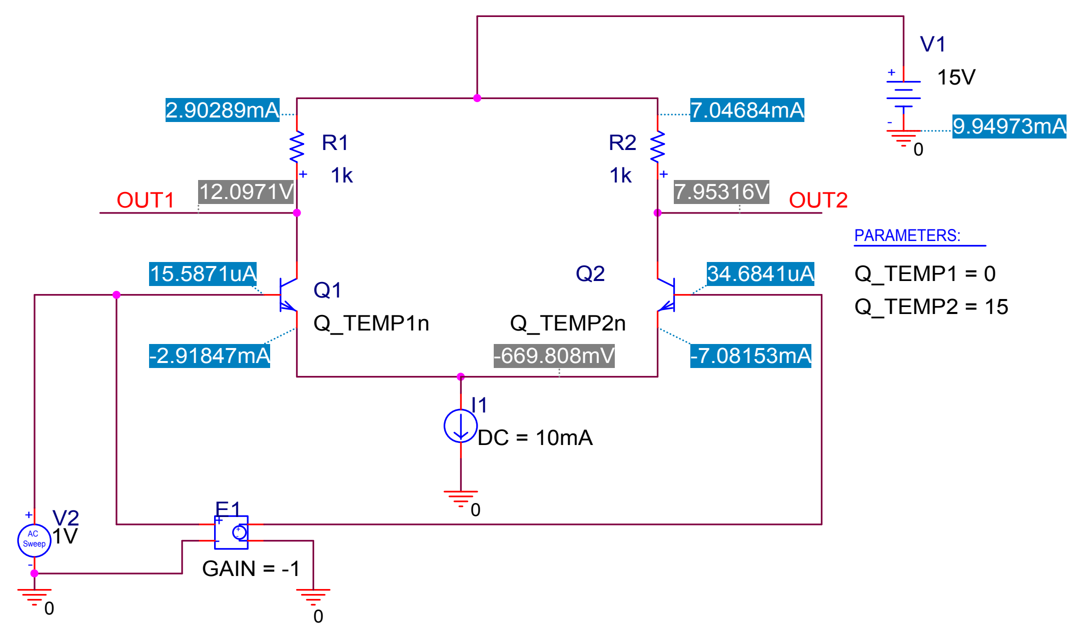

FIGURE 8
The same amplifier as Figure 7, but with transistor Q2 running 15ºC hotter than Q1

[5] The use of a breakout transistor allows us to choose our own name for the transistor model, which helps as a reminder several days or weeks from now in reconstructing what we did with this circuit.

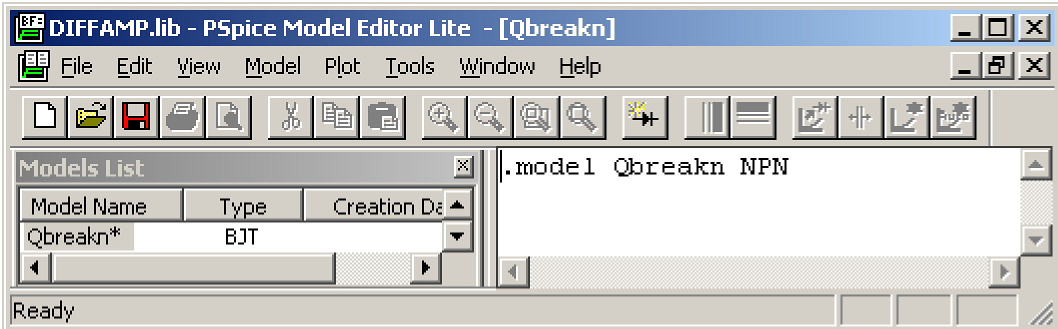

FIGURE 9

The listing for QbreakN as it is originally pasted onto the schematic

- Clicking on the transistor to highlight it and selecting EDIT/PSPICE MODEL we put this transistor into the •model editor as shown in Figure 9.
- Next, we paste the copied •model statement for the Q2N2222 into the QbreakN window to make the QbreakN transistor the same as the Q2N2222. Then, just to keep track of the model, we change the name to Q_TEMP1n.
- Next is the key step for temperature control: we specify the parameter T_REL_GLOBAL, which sets the device temperature relative to the global temperature. Normally this parameter has the default value of zero, but here we set it to a parameter Q_TEMP1 that PSPICE can control directly, and we indicate that it is a parameter by enclosing {Q_TEMP1} in curly braces. See Figure 10.

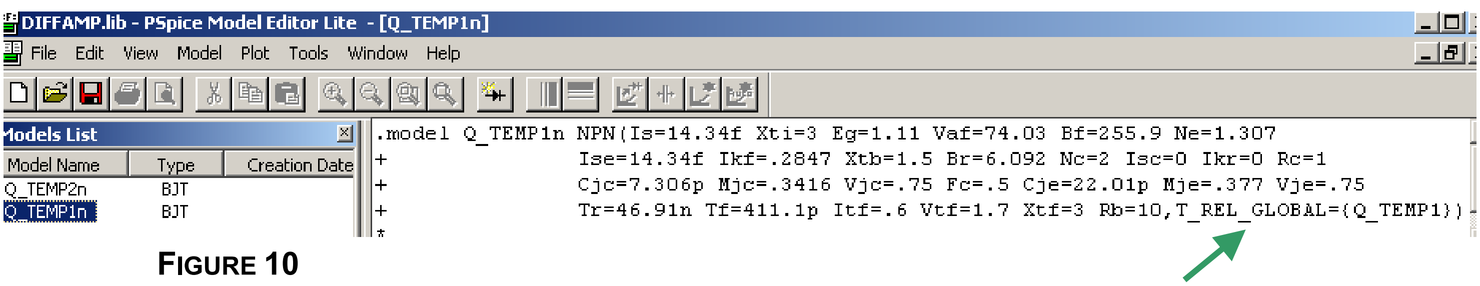

FIGURE 10

The •model statement for the device Q_TEMP1n setting the parameter T_REL_GLOBAL to {Q_TEMP1}

The same procedure is used to set up another QbreakN, named Q_TEMP2n, as a Q2N2222 with T_REL_GLOBAL = {Q_TEMP2}, a second temperature parameter. Now any type of simulation (BIAS, or TRANSIENT or whatever) can be run with these two device temperatures set as we wish relative to the global temperature. If we run the BIAS simulation for the case in Figure 8, we find in the OUTPUT file the listing in Figure 11 for the model parameters. The variables T_Measured and T_Current are identified in the next section. T_Current does tell you the relative temperatures of the various devices in the circuit, but that aside, normally T_Measured and T_Current can be ignored.

	Q_TEMP1n NPN	Q_TEMP2n NPN
T_Measured	27	27
T_Current	27	42
IS	1.43E-14	1.43E-14
BF	255.9	255.9
NF	1	1
VAF	74.03	74.03

FIGURE 11

Part of the BJT MODEL PARAMETERS listing for the simulation of Figure 8

Unlike the case using the menu of Figure 2, the output file for individually set temperatures does not list the temperature-adjusted •model parameters. The listing of temperature-adjusted parameters can be activated, however, by setting the device temperatures and also using Figure 2 to set the global temperature to a value other than the default value of 27ºC, say to the value T=27.01ºC. An example is shown in Figure 12. Comparison of BF and IS for the device at 27º C with Figure 5 shows they are nearly the same. Comparison of BF and IS for the device at 127º C with Figure 6 shows they also are close to the same.

	Q_TEMP1n	Q_TEMP2n
BF	**255.9**	**393.9**
ISE	1.436E-14	6.606E-11
VJE	0.75	0.569
CJE	2.201E-11	2.434E-11
RE	0	0
RB	10	10

BR	6.092	9.378
ISC	0	0
VJC	0.75	0.569
CJC	7.306E-12	8.008E-12
RC	1	1
RBM	10	10
IS	**1.436E-14**	**1.547E-09**

ISS	0	0
VJS	0.75	0.569
CJS	0	0
GAMMA	1E-11	1E-11
RCO	0	0
VO	10	10

FIGURE 12

Temperature-adjusted parameters for two transistors, with Q_TEMP1n at 27.01ºC (T_REL_GLOBAL=0) and Q_TEMP2n at 127.01ºC (T_REL_GLOBAL=100) and the global temperature set using Figure 2 to T = 27.01ºC.

Although we have changed the temperature for device Q2, the global temperature remains 27.01ºC, as indicated in the OUTPUT file by the following line that appears before the listing of the simulated Q-point values:

```
****    OPERATING POINT INFORMATION      TEMPERATURE =   27.010 DEG C
```

Again, you can keep track of what is happening by reading the OUTPUT file.

Summary

Just to summarize the many temperatures:

1. GLOBAL temperature = temperature of circuit as a whole. Set using PSPICE/EDIT SIMULATION PROFILE, and selecting the SIMULATION SETTINGS/ANALYSIS tab to obtain the TEMPERATURE (SWEEP) menu. Corresponds to the •TEMP <VALUE> command in the ANALYSIS DIRECTIVES of the OUTPUT FILE, and to the value listed as TEMPERATURE in the OUTPUT FILE.

2. T_REL_GLOBAL = temperature of device relative to the global (circuit) temperature. Set in the •model statement of the device by adding T_REL_GLOBAL= <VALUE>, where <VALUE> is the operating temperature of the device <u>relative to</u> the global temperature of the rest of the circuit. See the on-line Reference Manual, p. 51.

3. T_MEASURED = temperature where the device parameters were measured and fitted to *I-V* curves. Set in the •model statement of the device by adding the variable T_MEASURED = <VALUE>, where <VALUE> is the temperature you want. Normally should be left alone, at the default value of 27ºC.

4. TNOM = <u>default</u> value of T_MEASURED. Set on the SIMULATIONS SETTINGS/ OPTION tab as T_{NOM} and overridden by T_MEASURED. Normally should be left alone, at the default value of 27ºC. Corresponds to the •OPTIONS TNOM=<VALUE> command in the ANALYSIS DIRECTIVES of the OUTPUT FILE.

5. T_CURRENT = T_MEASURED+T_REL_GLOBAL. This temperature is listed in the output file, but has no bearing on the calculation. Don't worry about it.

Caution

It should be borne in mind that all transistor parameters vary from one device to another, and this is true of the parameters affecting the temperature dependence as well. Therefore, the PSPICE results for temperature dependence are only so-called "typical" results, and your results in the lab very likely will differ.