CHAPTER 10

Monte Carlo Analysis

Chapter Outline

Monte Carlo analysis is essentially a statistical analysis that calculates the response of a circuit when device model parameters are randomly varied between specified tolerance limits according to a specified statistical distribution. For example, all the circuits encountered so far have been simulated using fixed component values. However, discrete real components such as resistors, inductors and capacitors all have a specified tolerance, so that when you select, for example, a 10 kΩ ± 1% resistor, you can expect the actual measured resistor value to be somewhere between 9900 and 10 100 Ω. Other discrete components and semiconductors in a circuit will also have tolerances and so the combined effect of all the component tolerances may result in a significant deviation from the expected circuit response. This is especially the case in filter designs where applied component tolerances may result in a deviation from the required filter response.

What the Monte Carlo analysis does it to provide statistical data predicting the effect of randomly varying model parameters or component values (variance) within specified tolerance limits. The generated values follow a statistically defined distribution. The circuit analysis (DC, AC or transient) is repeated a number of specified times with each Monte Carlo run generating a new set of randomly derived component or model parameter values. The greater the number of runs, the greater the chances that every component value within its tolerance range will be used for simulation. It is not uncommon to perform hundreds or even thousands of Monte Carlo runs in order to cover as many possible component values within their tolerance limits. Monte Carlo, in effect,

Analog Design and Simulation using OrCAD Capture and PSpice. DOI: 10.1016/B978-0-08-097095-0.00010-6

predicts the robustness or yield of a circuit by varying component or model parameter values up to their specified tolerance limits.

Although the results of a Monte Carlo analysis can be seen as a spread of waveforms in the PSpice waveform viewer (Probe), a **Performance Analysis** can be used to generate and display histograms for the statistical data together with a summary of the statistical data. This provides a more visual representation of the statistical results of a Monte Carlo analysis.

10.1 SIMULATION SETTINGS

A Monte Carlo analysis is run in conjunction with another analysis, AC, DC or transient analysis. Tolerances are applied to parts in the schematic via the **Property Editor** and the required analysis is created in the simulation profile. For the band pass filter in Figure 10.1, component tolerances have been added to the resistors and capacitors and displayed on the circuit. The circuit response is analyzed by performing an AC sweep from 10 Hz to 100 kHz. Monte Carlo will run an initial analysis with all nominal values being used and then run subsequent analysis using randomly generated component values up to the number of Monte Carlo runs specified.

Figure 10.2 shows the simulation profile for an AC sweep running a Monte Carlo analysis.

OUTPUT VARIABLE

You need to specify a node voltage or an independent current or voltage source. In this example, the output variable is V(out).

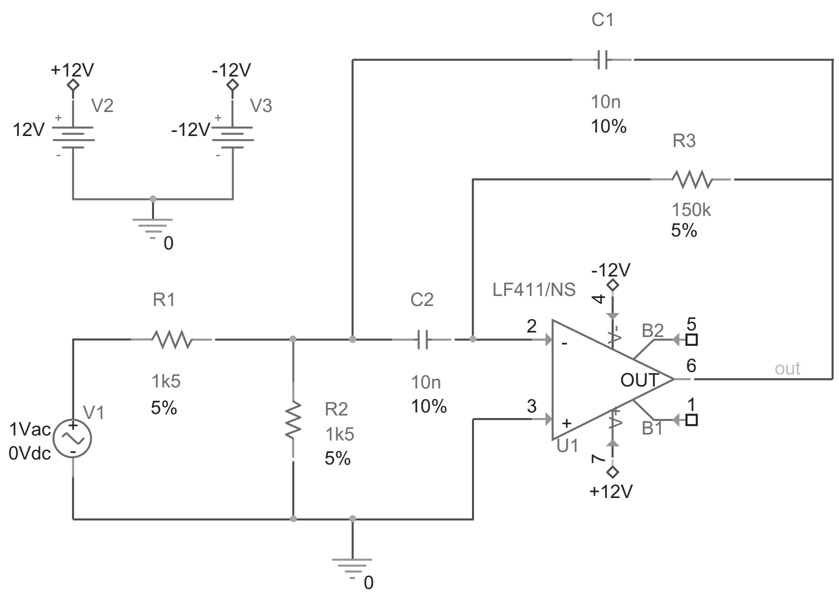


FIGURE 10.1
1500 Hz band pass filter.

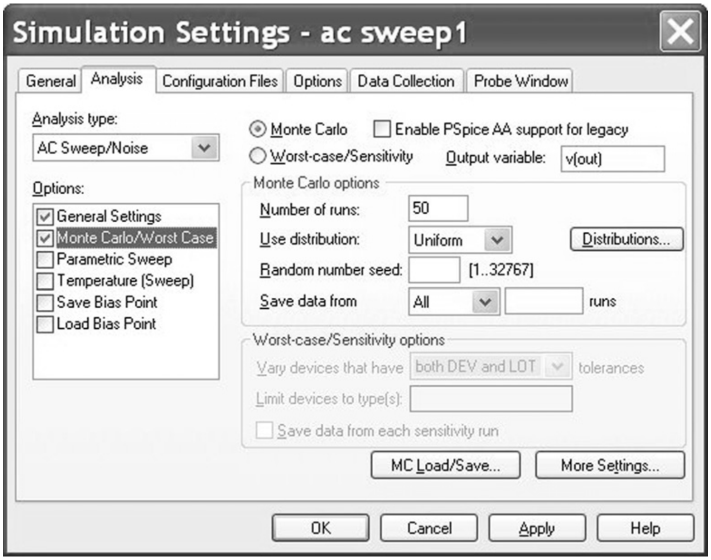


FIGURE 10.2 Monte Carlo simulation settings.

NUMBER OF RUNS

This is the number of times the AC, DC or transient analysis is run. The maximum number of waveforms that can be displayed in Probe is 400. However, for printed results, the maximum number of runs has now been increased from 2000 to 10 000. The first run is the nominal run where no tolerances are applied.

USE DISTRIBUTION

The model parameter deviations from the nominal values up to the tolerance limits are determined by a probability distribution curve. By default, the distribution curve is uniform; that is, each value has an equal chance of being used. The other option is the Gaussian distribution, which is the familiar bell-shaped curve commonly used in manufacturing. Component values are more likely to take on values found near the center of the distribution compared to the outer edges of the tolerance limits.

You can also define your own probability distribution curves using coordinate pairs specifying the deviation and associated probability, where the deviation is in the range −1 to +1 and the probability is in the range from 0 to 1. More information can be found in the PSpice A/D Reference Guide in <install dir> doc\pspcref\pspcref.pdf.

RANDOM NUMBER SEED

As with most random number generators, an initial seed value is required to generate a set of random numbers. This value must be an odd integer number from 1 to 32 767. If no seed number is specified, the default value of 17 533 is used.

SAVE DATA FROM

This allows you to save data from selected runs. For example, if you just want to see the circuit response from the nominal run, select none. If you want to save all the runs, select **All.** If you want to select every third run from the nominal run, i.e. the fourth, seventh, tenth, etc., select **Every,** then enter 3 in the **runs** box. If you want to save data from the first three runs, select **First** and then enter 3 in the **runs** box. If you want to save data from the third, fifth, seventh and tenth run, select **Runs (list)** and enter 3, 5, 7, 10 in the **runs** box. Saved runs will be displayed in Probe.

MC LOAD/SAVE

PSpice now has history support which allows you to save randomly generated model parameters or component values from a Monte Carlo run in a file, which can be used for subsequent analysis.

MORE SETTINGS

This option allows you to specify Collating Functions which are applied to the output waveforms returning a single resulting value. For example, the MAX function searches for and returns the maximum value of a waveform. The YMAX function returns the value corresponding to the greatest difference between the nominal run waveform and the current waveform. The RISE and FALL functions search for the first occurrence of a waveform crossing above or below a set threshold value. For each function, you can specify the range over which you want to apply these functions. Figure 10.3 shows the available collating functions.

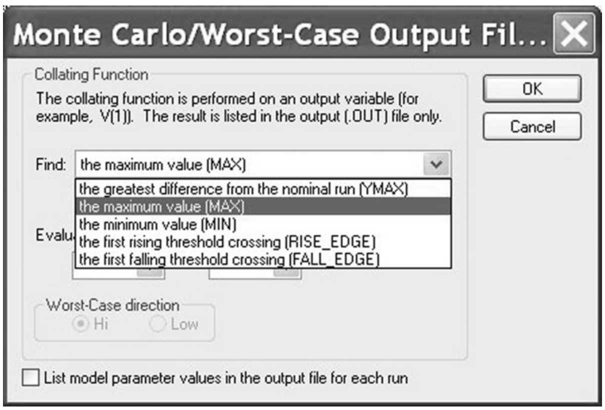


FIGURE 10.3
Collating functions.

The collating functions are summarized as:

YMAX: Find the greatest difference from the nominal run
MAX: Find the maximum value
MIN: Find the minimum value
RISE_EDGE: Find the first rising threshold crossing
FALL_EDGE: Find the first falling threshold crossing

10.2 ADDING TOLERANCE VALUES

Tolerances can now be added to the discrete R, L and C parts in the Property Editor as these parts now include a tolerance property. You must make sure that you include the % symbol when entering the tolerance value i.e. 10%.

In previous OrCAD versions the only way to add tolerances to discrete parts was to use generic **Breakout** parts, which allowed you to edit the PSpice model. For example, to add a tolerance to a resistor, you had to use an Rbreak part from the **breakout** library, edit the model (rmb > Edit PSpice Model) and add tolerance statements to the PSpice model. The default model definition for an Rbreak is given by:

```
.model Rbreak RES R=1
```

where Rbreak is the model name that can be changed and appears on the schematic, RES is the PSpice model type and R is a resistance multiplier. There are two types of tolerance you can add, **dev** and **lot**, as shown below:

```
.model Rmc1 RES R=1 lot = 2% dev=5%
```

Here, the model name has been changed to Rmc1 and two tolerance types have been added to the model statement. The dev tolerance causes the tolerance values of devices that share the same model name (Rmc1 in this example) to vary independently from each other, while the lot tolerance will cause the tolerance values of devices, with the same model name, to track together.

Dev is the same as applying a standard 5% tolerance to all the resistors, where each resistor with the same model name will be assigned its own random resistance value independent from all the other resistors with the same model name.

Lot is where all the resistors with the same model name will be treated as one group and will track together by as much as $\pm$ 2%. This was mainly used in integrated circuit (IC) design where a change in temperature was equally applied to groups of components. The combined total tolerance for Rmc1 can therefore be as much as $\pm$ 7%.

One example in which you would use both lot and dev is for a single in-line (SIL) or dual in-line (DIL) resistor pack; each resistor will have a dev tolerance defined, and therefore each randomly generated resistance value will have a different value from the others. If there is a rise in temperature, the lot tolerance will ensure that all resistance values increase together by the same percentage.

As mentioned above, only discrete parts have an attached tolerance property which can be edited in the Property Editor. If you want to add for example a specified manufacturer's tolerance to the B_f of a transistor to see its effect on a circuit's performance, you will have to edit the transistor model and add, for example, the dev tolerance to the B_f model parameter as:

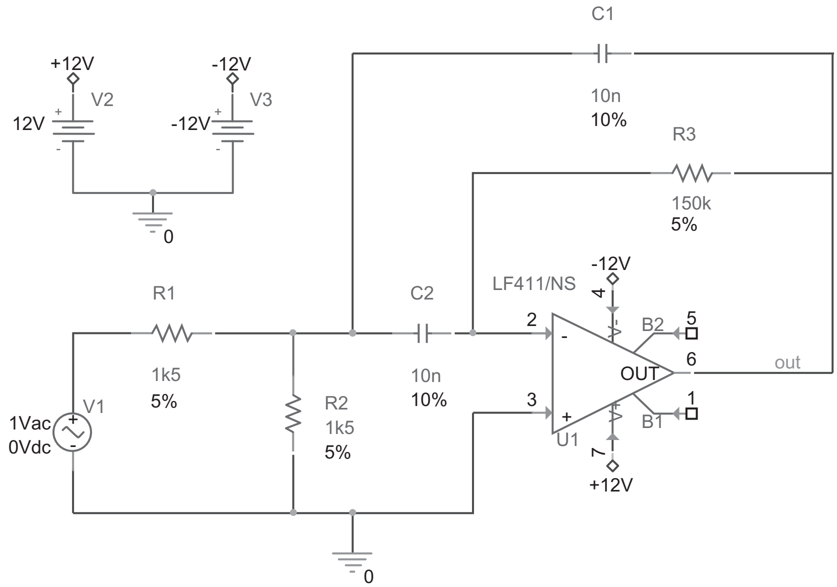
```
.model Q2N3906 PNP (Is=1.41f Xti=3 Eg=1.11 Vaf=18.7
                Bf=180.7 dev=50% Ne=1.5 Ise=0
```

In this example, a dev tolerance of 50% with a default uniform distribution has been added to the B_f of the transistor. However, the distribution for the B_f is more likely to be Gaussian, so you would add dev/gauss=12.5% as PSpice limits a Gaussian distribution to $\pm 4\sigma$:

```
.model Q2N3906 PNP (Is=1.41f Xti=3 Eg=1.11 Vaf=18.7
                Bf=180.7 dev/gauss=12.5% Ne=1.5 Ise=0
```

10.3 EXERCISES

FIGURE 10.4
1500 Hz band pass filter.



Exercise 1

Figure 10.4 shows a Sallen and Key 1500 Hz band pass filter circuit. Tolerances will be added to the resistors and capacitors and a Monte Carlo analysis will be performed to predict the statistical variation of the band pass frequency.

1. Draw the circuit in Figure 10.4. If you are using the demo CD, use the ua741 opamps, which can be found in the **eval** library; otherwise, the LF411 opamp can be found in the **opamp** library. Performing a **Part Search** will result in

the LF411 opamp being sourced from different manufacturers. It does not matter which one you use. V1 is a V_{AC} source from the **source** library, used for an AC analysis, and the power symbols are VCC_CIRCLE symbols from **Place > Power**, renamed +12 V and −12 V, respectively.

NOTE

When running a Part Search for the LF411, make sure the search part is pointing to the <install dir> \Tools\ Capture > Library > PSpice library (see Chapter 5, Exercise 3).

---

2. You need to add a 5% tolerance to all the resistors. Hold down the control key and select the resistors R1, R2 and R3, then **rmb > Edit Properties**. In the **Property Editor** highlight the entire **Tolerance** Row (or column) as shown in Figure 10.5 and **rmb > Edit**. In the **Edit Property Values** box (Figure 10.6), type in **5%** and click on OK.

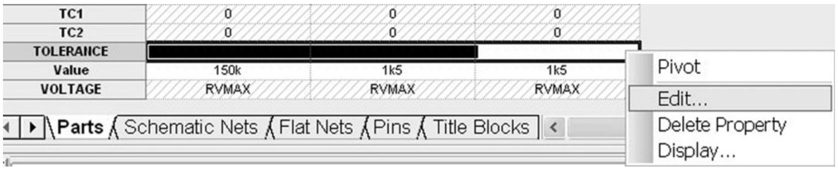


FIGURE 10.5
Select entire TOLERANCE row and select **Edit**.

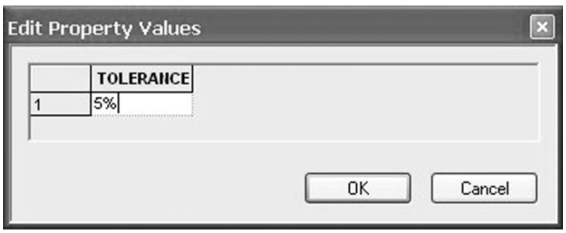


FIGURE 10.6
Adding a 5% tolerance to the resistors.

3. To display the component tolerances on the schematic, select the entire TOLERANCE row as in Figure 10.5, **rmb > Display** and in **Display Properties** (Figure 10.7), select **Value Only** and click on OK.

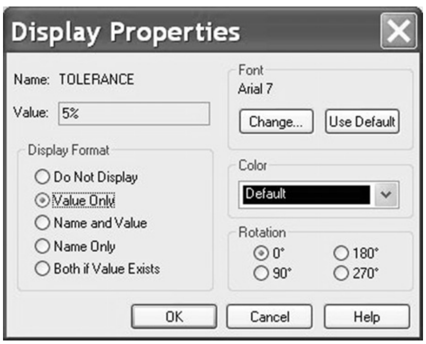


FIGURE 10.7
Displaying tolerance values.

4. Repeat Step 2 by assigning a 10% tolerance to capacitors C1 and C2.
5. Create a PSpice simulation profile called AC Sweep for an AC logarithmic sweep from 100 to 10 kHz with 50 points per decade (Figure 10.8). Select **Monte Carlo/Worst Case** in the **Options** box.

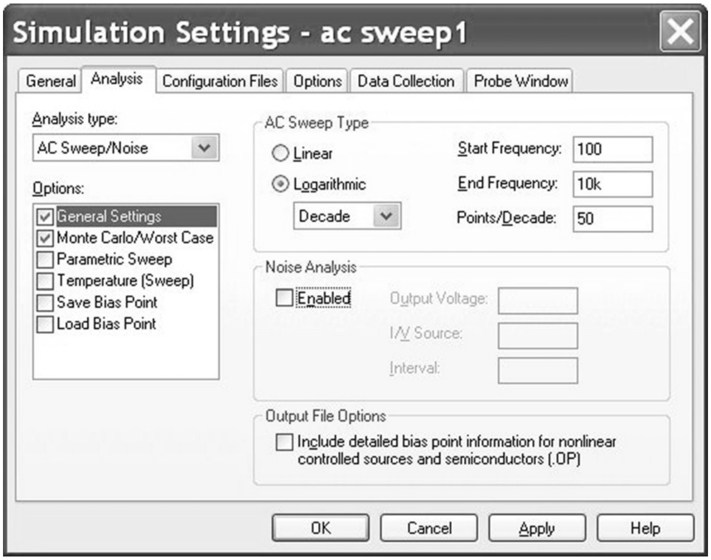


FIGURE 10.8 Simulation profile for an AC sweep.

The output variable is **V(out)** and the number of runs is 50. The distribution used will be uniform and you will use the default random seed setting by leaving the Random number seed box empty. Your Monte Carlo simulation settings should be as shown in Figure 10.9.

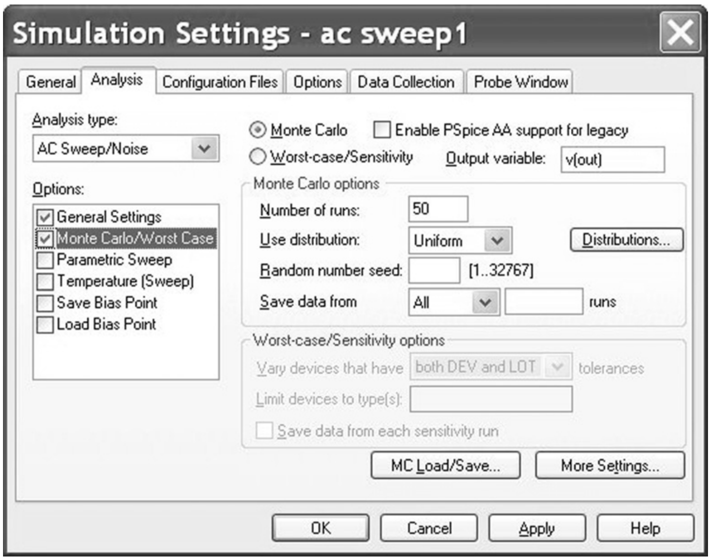


FIGURE 10.9 Monte Carlo simulation settings.

6. Place a dB voltage marker on node **out** (**PSpice > Markers > Advanced > dB Magnitude of Voltage**) and run the simulation.

NOTE

Monte Carlo runs can produce large data files. Since we are only interested in the output node, the simulation can be set up to collect data for display waveforms only for markers placed on the schematic. In the simulation profile, select the **Probe Window** tab and make sure that **All markers on open schematic** is selected.

---

7. When PSpice launches, a list of available runs is shown (Figure 10.10). Select **All** and click on OK.

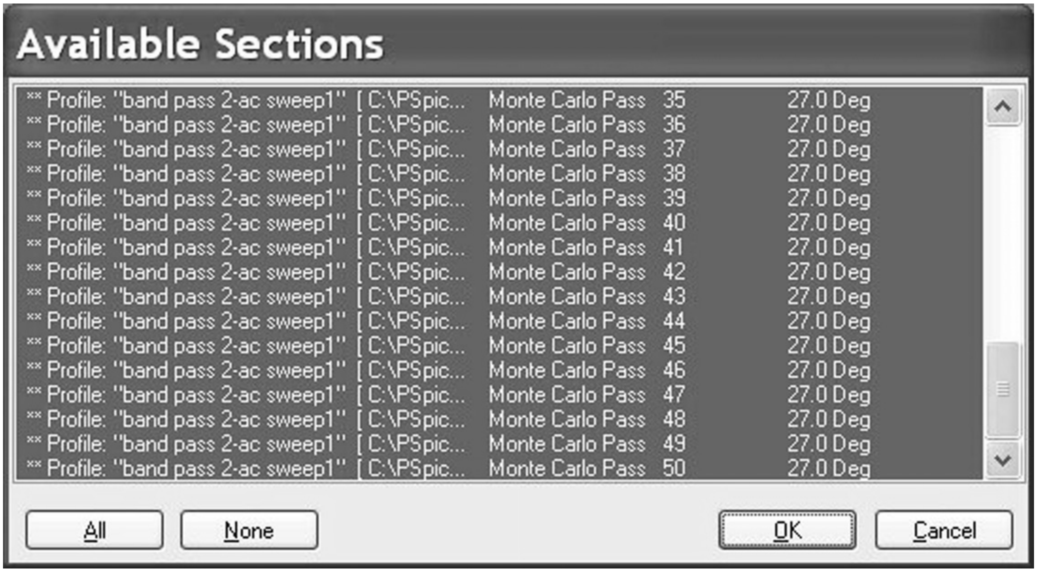


FIGURE 10.10
Available sections.

8. Figure 10.11 shows the band pass filter response showing a spread in the center frequency. Open the output file, **View > Output File**{ **XE "Output File"** } and scroll down to see the results for each Monte Carlo run. At the bottom of the file, you will see a summary of the statistical results and the deviation from the nominal for each run.
However, to get a better picture of the spread of the center frequency, you can use a **Performance Analysis** to display histograms representing the statistically generated data. Performance analysis will be covered in more detail in Chapter 12, but for now, the steps required for a performance analysis for the bandpass filter are introduced in Exercise 2.

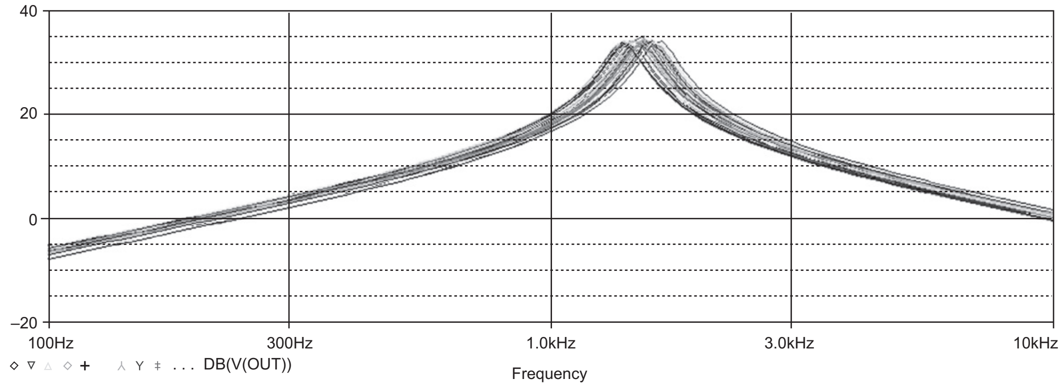
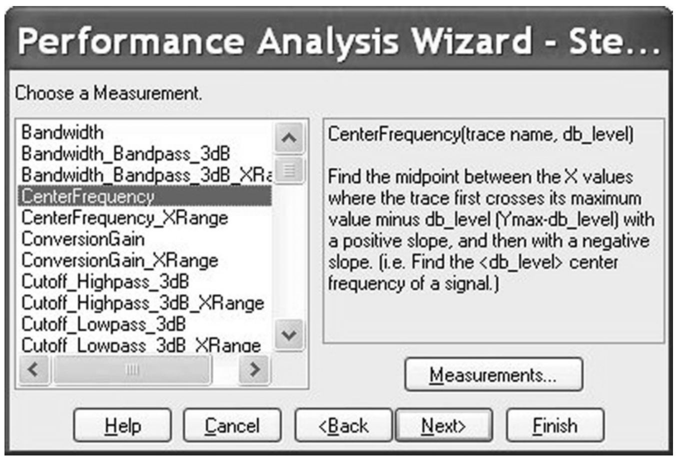


FIGURE 10.11
Monte Carlo band pass filter response.

Exercise 2

1. In Probe, from the top toolbar, select **Trace > Performance Analysis.** At the bottom of the **Performance Analysis** window, click on **Wizard,** then in the next window click on **Next.** You will be presented with a list of measurements as shown in Figure 10.12.

FIGURE 10.12
Available measurements.



2. Select the **CenterFrequency** measurement and click on **Next**.
3. In the **Measurement Expression** window (Figure 10.13), enter the name of the trace to search as **V(out)**. Alternatively, you can click on the **Name of trace to search** icon and search for the trace name, V(out).
   Enter a value of 3 for the **db level down for measurement** as shown. The center frequency of the filter will be measured 3 dB down either side of the maximum value for the Monte Carlo nominal run (no component tolerances applied). Click on **Next** in the **Performance Analysis Wizard**.

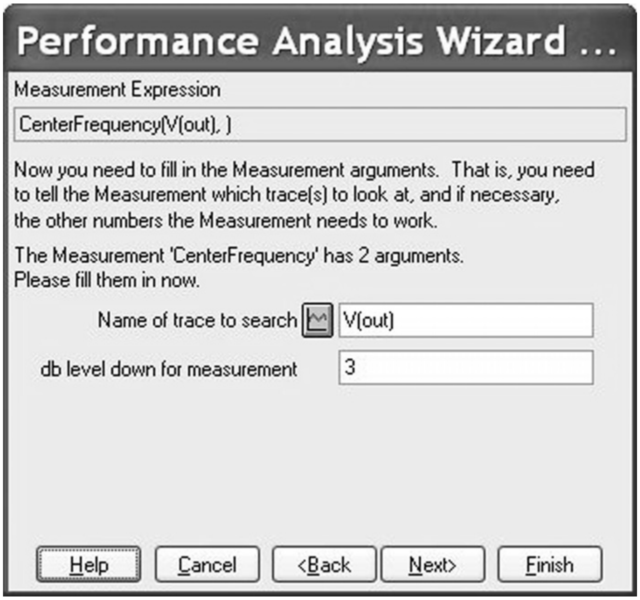


FIGURE 10.13
V(out) is selected as the trace variable with a −3 dB measurement down either side.

4. The nominal trace waveform of the band pass circuit will be displayed (Figure 10.14). This is the simulation response using the component nominal values. The trace shows two points where the measurements from the waveform will be taken. In this case, the center frequency is defined at 3 dB down from the maximum value. This is shown so that you can confirm whether you are seeing the correct circuit response and that the measurements are taken at the correct points on the waveform.

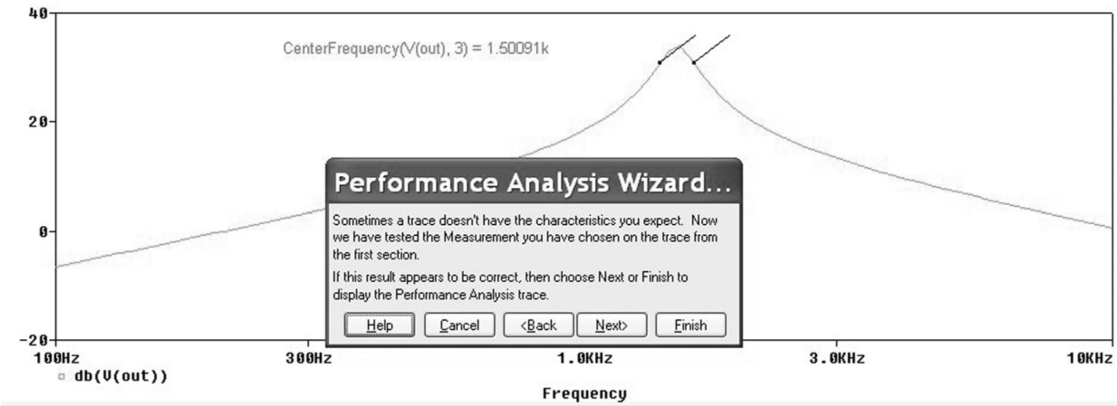


FIGURE 10.14
Nominal trace showing where the −3 dB measurement will be taken on the nominal circuit response waveform.

5. Click on **Next** in the **Performance Analysis Wizard**.
6. Histograms will now be displayed (Figure 10.15) representing the generated statistical data together with a summary of statistical data.

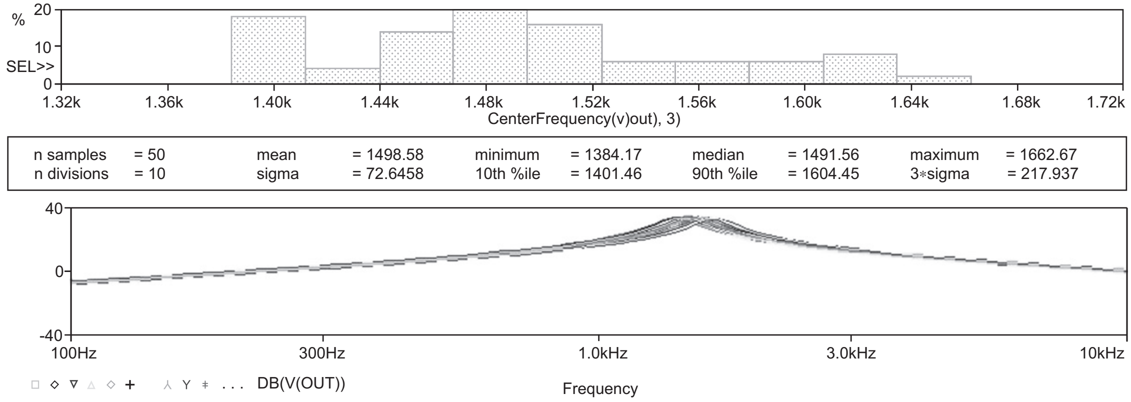


FIGURE 10.15
Histograms shows the possible spread in center frequency.

7. Change the resistor tolerances to 1% and the capacitor tolerances to 5% and re-run the simulation. Run a performance analysis as before using the center frequency measurement as above. The results are shown in Figure 10.16.

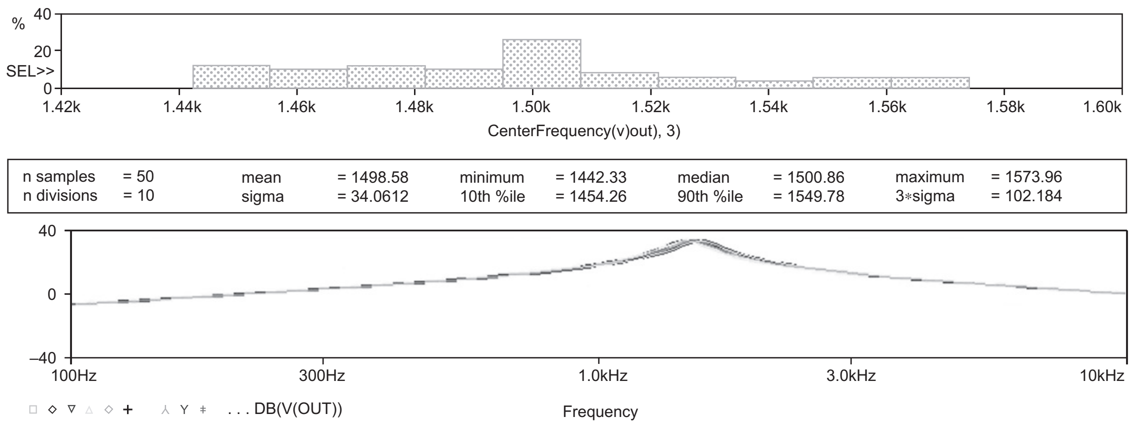


FIGURE 10.16
Statistical data for the center frequency with tighter component tolerances.

NOTE

Because the tolerances are displayed on the schematic, there is no need to use the Property Editor to change the tolerance values. Just double click on the tolerance values in the schematic and enter the new tolerance value.

---

8. Run another Performance analysis but this time, use the bandwidth measurement.

NOTE
The number of displayed histograms can be changed in Probe. Select **Tools > Options > Histogram Divisions**.

FILTER SPECIFICATIONS

The gain of the filter is given by:

$$\begin{aligned} |G| &= \frac{R3}{2R1} \\ |G| &= \frac{150 \times 10^3}{2 \times 1.5 \times 10^3} = 50 \end{aligned} \tag{10.1}$$

or in dB:

$$|G|_{\text{dB}} = 20log_{10}50 = 34 \text{ dB}$$

The band pass center frequency is given by:

$$f = \frac{1}{2\pi C\sqrt{\left(\frac{R1R2}{R1+R2}\right)R3}} \tag{10.2}$$

where C1 = C2 = C

$$f = \frac{1}{2\pi \times 10 \times 10^{-9}\sqrt{\left(\frac{1.5 \times 10^3 \times 1.5 \times 10^3}{1.5 \times 10^3 + 1.5 \times 10^3}\right) \times 150 \times 10^3}}$$

$$f = 1501 \text{ Hz}$$

The −3 dB bandwidth is given by:

$$\begin{aligned} f &= \frac{1}{\pi\, C\, R3} \\ f &= \frac{1}{\pi \times 10 \times 10^{-9} \times 150 \times 10^3} = 212.2 \text{ Hz} \end{aligned} \tag{10.3}$$

This page intentionally left blank