Performing a Monte Carlo Analysis

Prepared By: Jason Grieve, Aaron Hunnewell, Lei Wang, and RW Hendricks
Date: February 10, 2007
Revision: 1.0 (original release)
Application: PSpice 9.2 and above

A Monte Carlo calculation is one in which component values are randomly replaced by values within their specified tolerance and the circuit is solved repetitively, perhaps many hundreds of times. A histogram is then generated that shows the frequency with which various node voltages or branch currents are found. The minimum and/or maximum node voltages and branch currents may also be found.

A Monte Carlo calculation may be performed as follows:
1. Complete your circuit diagram in PSpice.
2. Renumber the nodes of interest if desired.¹
3. Run a Bias Point simulation to be sure your circuit is set up correctly.
4. Double click on a resistor, capacitor, or inductor to access its properties.
5. At the top of the Property Editor screen, if necessary, change the “Filter by” list to “Current Properties” to view more properties of the device.
6. Scroll to the right and enter a tolerance percentage (i.e. 5%) as shown in Figure 1. Be sure to enter the per cent symbol. Then close the Properties Editor window.

![Property Editor](image)

Figure 1: Set the tolerance percentage of a resistor.

Performing a Monte Carlo Analysis

Figure 2: Create a new simulation.

Figure 3: Primary sweep settings window.
7. Repeat for each resistor, capacitor, or inductor entering the appropriate values.

8. Create a new simulation profile by clicking the “New Simulation Profile”. Enter a name for the simulation as shown in Figure 2. Then click “Create”.

9. When the “New Simulation” window closes, the “Simulations Settings” window opens as shown in Figure 3.

10. Select “DC Sweep” for the Analysis type and highlight the “Primary Sweep”. Enter the name of a voltage source or a current source in your model (it does not matter which). For the circuit shown in Figure 2, V1 is used.

11. Enter the same starting and ending value for the sweep. The increment size is irrelevant. For this example, we set the starting and ending values to 9 V.

12. Now check the “Monte-Carlos/Worst Case” box. The “Simulations Settings – Monte Carlo simulation” window appears as shown in Figure 4.

13. Check the Monte Carlo radio button and enter the parameter for which you wish to perform the Monte Carlo calculation. In Figure 4, we have selected voltage at node N2 as shown in the circuit in Figure 2, which is given by $V(N2)$.

14. Next, enter a number of runs; 500 is usually more than adequate. This is the number of times the circuit will be solved for, in this case, $V(N2)$. You can explore how many runs need to be made by checking the results, to be discussed below, for any significant variation with number of runs.
15. The choice of distribution is important. Select the Uniform distribution if you do not know anything about the distribution, but know the Tolerance of the device. However, if you have measurements of the distribution of device properties, you may enter them here.

16. For the Random Number Seed, a random number between 1 and 32767.

17. Click the “Apply” button and then the “OK” button.

18. Return to the main PSpice toolbar and “Run” the simulation.

19. Click “OK” on the resulting “Available Sections” window. A blank histogram screen will appear.

20. Select the “Schematic” window and place a voltage probe on any node of interest. The histogram window will immediately show a histogram of the voltages at the node as computed from the multiple solutions of the circuit. Note that below the histogram various statistics of the distribution are also shown and include the minimum, maximum, mean, median, and standard error of the distribution.

21. Return to the schematic and delete the probe. Place the probe on a different node and see the histogram of results for that node. Or, place a current probe on a branch (you must place the probe on the pin of a device) and see a histogram of the current distribution in the circuit.