Parametric and temperature analysis

Chapter overview

This chapter describes how to set up parametric and temperature analyses. Parametric and temperature are both simple multi-run analysis types.

This chapter includes the following sections:

- Parametric analysis on page 362
- Temperature analysis on page 371
Parametric analysis

Minimum requirements to run a parametric analysis

Minimum circuit design requirements

- Set up the circuit according to the swept variable type as listed in Table 32.
- Set up a DC sweep, AC sweep, or transient analysis.

Table 32  Parametric analysis circuit design requirements

<table>
<thead>
<tr>
<th>Swept variable type</th>
<th>Requirement</th>
</tr>
</thead>
<tbody>
<tr>
<td>voltage source</td>
<td>voltage source with a DC specification (VDC, for example)</td>
</tr>
<tr>
<td>temperature</td>
<td>none</td>
</tr>
<tr>
<td>current source</td>
<td>current source with a DC specification (IDC, for example)</td>
</tr>
<tr>
<td>model parameter</td>
<td>PSpice A/D model</td>
</tr>
<tr>
<td>global parameter</td>
<td>global parameter defined with a parameter block (PARAM)</td>
</tr>
</tbody>
</table>

Minimum program setup requirements

1. In the Simulation Settings dialog box, from the Analysis type list box, select Time Domain (Transient).
2. Under Options, select Parametric Sweep if it is not already enabled.
3. Specify the required parameters for the sweep.

Note  Parametric analysis is not supported in PSpice A/D Basics.
4 Click OK to save the simulation profile.

5 From the PSpice menu, choose Run to start the simulation.

**Note** Do not specify a DC sweep and a parametric analysis for the same variable.

**Overview of parametric analysis**

Parametric analysis performs multiple iterations of a specified standard analysis while varying a global parameter, model parameter, component value, or operational temperature. The effect is the same as running the circuit several times, once for each value of the swept variable.

See *Parametric analysis* on page 80 for a description of how to set up a parametric analysis.
RLC filter example

This example shows how to perform a parametric sweep and analyze the results with performance analysis.

Use performance analysis to derive values from a series of simulator runs and plot these values versus a parameter that varies between the simulator runs.

For this example, the derived values are the overshoot and the rise time versus the damping resistance of the filter.

Entering the design

The schematic representation for the RLC filter (RLCFILT.OPJ) is shown in Figure 69.

![RLCFILT.OPJ schematic](image)

**Figure 69** Passive filter schematic.

This series of PSpice A/D runs varies the value of resistor R1 from 0.5 to 1.5 ohms in 0.1 ohm steps. Since the time-constant of the circuit is about one second, perform a transient analysis of approximately 20 seconds.

Create the circuit in OrCAD Capture by placing a piecewise linear independent current source (IPWL from SOURCE.OLB). Set the current source properties as follows:

- \( AC = 1a \)
- \( T1 = 0s \)
- \( I1 = 0a \)
- \( T2 = 10ms \)
I_2 = 0a
T_3 = 10.1ms
I_3 = 1a

Place an instance of a resistor and set its VALUE property to the expression, \( R \). To define \( R \) as a global parameter, place a PARAM pseudocomponent and use the Property Editor to create a new property \( R \) and set its value to 0.5. Place an inductor and set its value to 1H, place a capacitor and set its value to 1, and place an analog ground symbol (0 from SOURCE.OLB). Wire the schematic symbols together as shown in Figure 69.

**Running the simulation**

Run PSpice A/D with the following analyses enabled:

- **transient**
  - print step: 100ms
  - final time: 20s
- **parametric**
  - swept var. type: global parameter
  - sweep type: linear
  - name: \( R \)
  - start value: 0.5
  - end value: 1.5
  - increment: 0.1

After setting up the analyses, start the simulation by choosing Run from the PSpice menu.

**Using performance analysis to plot overshoot and rise time**

After performing the simulation that creates the data file RLCFILT.DAT, you can calculate the specified performance analysis goal functions.

When the simulation is finished, a list appears containing all of the sections (runs) in the data file produced by PSpice A/D. To use the data from every run, select All and click OK in the Available Selections dialog box. In the case of Figure 70, the trace \( I(L1) \) from the ninth section was added by specifying the following in the Add Traces dialog box:

\[ I(L1)@9 \]
Chapter 12  
Parametric and temperature analysis

Figure 70  
Current of L1 when R1 is 1.5 ohms.

To run performance analysis

1  From the Trace menu, choose Performance Analysis.
2  Click OK.

PSpice resets the X-axis variable for the graph to be the parameter that changed between PSpice A/D runs. In the example, this is the R parameter.

To see the rise time for the current through the inductor L1, click the Add Trace toolbar button and then enter:

```
genrise(I(L1))
```

Figure 71 shows how the rise time decreases as the damping resistance increases for the filter.

Another Y axis can be added to the plot for the overshoot of the current through L1 by selecting Add Y Axis from the Plot menu. The Y axis is immediately added. Now click the Add Trace toolbar button and enter:

```
overshoot(I(L1))
```

Figure 71 shows how the overshoot increases with increasing resistance.

Troubleshooting tip

More than one PSpice A/D run or data section is required for performance analysis. Because one data value is derived for each waveform in a related set of waveforms, at least two data points are required to produce a trace.

Use Eval Goal Function (from the Trace menu) to evaluate a goal function on a single waveform and produce a single data point result.

The genrise and overshoot goal functions are contained in the file PSPICE.PRB in the OrCAD directory.

The image shows a graph of current through L1 with R1 set at 1.5 ohms, illustrating the performance analysis process.
Figure 71  Rise time and overshoot vs. damping resistance.
Example: frequency response vs. arbitrary parameter

You can view a plot of the linear response of a circuit at a specific frequency as one of the circuit parameters varies (such as the output of a band pass filter at its center frequency vs. an inductor value).

In this example, the value of a nonlinear capacitance is measured using a 10 kHz AC signal and plotted versus its bias voltage. The capacitance is in parallel with a resistor, so a trace expression is used to calculate the capacitance from the complex admittance of the R-C pair.

Setting up the circuit

Enter the circuit in Capture as shown in Figure 72

To create the capacitor model in the schematic editor:

1. Place a CBREAK part.
2. Select it so that it is highlighted.
3. From the Edit menu, choose PSpice Model.
4. In the Model Text frame, enter the following:
   \[ \text{.model Cnln CAP(C=1 VC1=-0.01 VC2=0.05)} \]
5. From the File menu, choose Save.

Set up the circuit for a parametric AC analysis (sweep Vbias), and run PSpice A/D. Include only the frequency of interest in the AC sweep.
To display the results

Use PSpice to display the capacitance calculated at the frequency of interest versus the stepped parameter.

1. Simulate the circuit.
2. Load all AC analysis sections.
3. From the Trace menu, choose Add Trace or click the Add Trace toolbar button.
4. Add the following trace expression:
   \[ \text{IMG}(-\text{I(Vin)}/\text{V(1,0)})/(2\times3.1416\times\text{Frequency}) \]
   Or add the expression:
   \[ \text{CvF}(-\text{I(Vin)}/\text{V(1,0)}) \]

Where \( \text{CvF} \) is a macro which measures the effective capacitance in a complex conductance. Macros are defined using the Macros command on the Trace menu. The \( \text{CvF} \) macro should be defined as:

\[ \text{CvF}(G) = \frac{\text{IMG}(G)}{2\times3.1416\times\text{Frequency}} \]

**Note** - \( -\text{I(Vin)}/\text{V(1)} \) is the complex admittance of the R-C branch; the minus sign is required for correct polarity.

To use performance analysis to plot capacitance vs. bias voltage

1. From the Trace menu, choose Performance Analysis.
2. Click Wizard.
3. Click Next>.
4. Click YatX in the Choose a Goal Function list, and then click Next>.
5. In the Name of Trace text box, type the following:
   \[ \text{CvF}(-\text{I(Vin)}/\text{V(1)}) \]
6 In the X value to get Y value at text box, type 10K.

7 Click Next. The wizard displays the gain trace for the first run to text the goal function (YatX).

8 Click Finish. The resultant plot is shown in Figure 73.

Figure 73 Plot of capacitance versus bias voltage.
Temperature analysis

Minimum requirements to run a temperature analysis

Minimum circuit design requirements

None.

Minimum program setup requirements

1. In the Simulation Settings dialog box, from the Analysis type list box, select Time Domain (Transient).

2. Under Options, select Temperature Sweep if it is not already enabled.

3. Specify the required parameters for the sweep.

4. Click OK to save the simulation profile.

5. From the PSpice menu, choose Run to start the simulation.
Overview of temperature analysis

For a temperature analysis, PSpice A/D reruns standard analyses set in the Simulation Settings dialog box at different temperatures.

You can specify zero or more temperatures. If no temperature is specified, the circuit is run at 27°C. If more than one temperature is listed, the simulation runs once for each temperature in the list.

Setting the temperature to a value other than the default results in recalculating the values of temperature-dependent devices. In EXAMPLE.OPJ (see Figure 74), the temperature for all of the analyses is set to 35°C. The values for resistors RC1 and RC2 are recomputed based upon the CRES model which has parameters TC1 and TC2 reflecting linear and quadratic temperature dependencies.

Likewise, the Q3 and Q4 device values are recomputed using the Q2N2222 model which also has temperature-dependent parameters. In the simulation output file, these recomputed device values are reported in the section labeled TEMPERATURE ADJUSTED VALUES.

The example circuit EXAMPLE.OPJ is provided with the OrCAD program installation.

Figure 74 Example schematic EXAMPLE.OPJ.