GENERAL

analogue design and simulation using orcad capture and pspice

Published: September 19, 2026 | Index ID: #-

Analogue Design and Simulation Using OrCAD Capture and PSpice

Analog circuit design is the backbone of countless electronic systems, from low power sensor interfaces to high performance audio amplifiers. While digital design has become highly automated, analog engineers still rely on meticulous modeling, component selection, and iterative simulation to achieve reliable performance. One of the most powerful toolchains for this purpose is the combination of OrCAD Capture for schematic capture and PSpice for circuit simulation. This article walks you through every stage of the analog design workflow, from initial concept to final verification, while highlighting best practices, common pitfalls, and advanced techniques that will help you master OrCAD Capture and PSpice.

Why OrCAD Capture and PSpice? A Quick Overview

OrCAD Capture is a versatile schematic editor that allows engineers to build complex analog and mixed signal circuits quickly. Its component library, intelligent auto routing, and seamless integration with simulation tools make it a favorite among professional designers. PSpice, on the other hand, is a robust simulation engine that supports a wide range of analysis types—DC, AC, transient, noise, Monte Carlo, and more—ensuring that every aspect of your design behaves as expected.

When combined, these tools provide a full design environment: capture the schematic, run simulations, tweak component values, and iterate until the circuit meets specifications. Moreover, OrCAD Capture's ability to export netlists directly to PSpice eliminates the need for manual netlist creation, reducing errors and saving valuable time.

Getting Started: Setting Up Your Project

Before you dive into drawing schematics, you must set up a clean project structure. This not only keeps files organized but also streamlines collaboration with colleagues.

1. Create a New Project – Launch OrCAD Capture and select File > New > Project. Choose a descriptive name (e.g., AudioAmp_V1) and a location on your file system.
2. Define Project Properties – In the Project Properties dialog, set the default simulation settings: choose PSpice as the simulation engine, specify the default simulation language (SPICE or PSpice), and set the library search path.
3. Organize Sub Projects – For large designs, use sub projects for each subsystem (e.g., PowerSupply, Amplifier, FeedbackLoop). This modular approach simplifies navigation and version control.
4. Version Control Integration – If you use Git, Subversion, or another VCS, add the project folder to the repository. OrCAD Capture can automatically track changes to schematic files, making it easier to roll back mistakes.

Building the Schematic in OrCAD Capture

Choosing the Right Components

Analog design starts with component selection. While OrCAD Capture offers a massive library, it's crucial to use

verified models that match your target process and supply voltages.

- Transistors – Use models from reputable sources (e.g., TI, Analog Devices, or manufacturer datasheets). Verify that the model includes the appropriate temperature range and process corners.
- Passive Elements – For resistors and capacitors, select models that include tolerance, temperature coefficient, and ESR (for capacitors).
- Specialized Components – For op amps, voltage references, and current mirrors, choose models that provide accurate offset, bias current, and noise figures.

Component Placement and Netting

Once components are selected, follow these best practices to ensure a clean, simulation ready schematic:

1. Logical Grouping – Place related components together (e.g., all biasing resistors near the transistor). This aids readability and future modifications.
2. Use Subcircuits – For repeated structures (e.g., a current mirror), create a subcircuit. This reduces schematic clutter and allows you to test the subcircuit independently.
3. Label Nets Clearly – Use descriptive net names (e.g., VDD, VSS, Vout) instead of generic numbers. This becomes invaluable during simulation debugging.
4. Avoid Unnecessary Crossings – While schematic editors handle crossings automatically, try to minimize them for clarity.

Adding Simulation Annotations

OrCAD Capture can automatically generate netlists for PSpice, but you need to annotate the schematic with simulation directives:

- Simulation Settings – Right click on the schematic and choose Simulation Settings. Select the analysis type (e.g., Transient), set the simulation time, and specify the time step.
- Voltage Sources – Use the V component to insert DC or AC sources. Label them appropriately and set their values.
- Probe Points – Add probes (e.g., V(out)) to capture the signals you care about during simulation.
- Model Overrides – If you need a custom model or parameter, right click the component, choose Properties, and edit the model reference.

Running Simulations in PSpice

Initial DC Analysis

Before you perform transient or AC simulations, verify that the circuit reaches a steady state under the given bias conditions.

1. Choose DC Sweep – In the simulation settings, select DC Sweep and define the sweep variable (e.g., Vin from 0 /V to 5 /V).
2. Set Convergence Criteria – Ensure that the simulation converges by checking the Convergence tab. If the solver fails, try adjusting the initial conditions or using the Newton method.
3. Analyze Results – Look at key node voltages and currents to confirm that the circuit behaves as expected. Pay attention to any unrealistic values that might indicate a wiring error.

Transient Simulation for Time Domain Performance

Transient analysis is essential for verifying waveforms, rise/fall times, and stability.

- Define Input Waveforms – Use Voltage Source components with Pulse or Sine parameters to model realistic signals.

- Set Simulation Time – Choose a duration that captures several periods of the input signal plus settling time (e.g., 5 /ms for a 1 /kHz sine wave).
- Plot Key Outputs – Add wave traces for Vout, Vout Vref, and any intermediate nodes to monitor signal integrity.
- Check for Oscillations – Look for unintended oscillations that may indicate insufficient feedback or parasitic capacitance.

AC Analysis for Frequency Response

AC analysis reveals the gain, bandwidth, and phase margin of your amplifier.

1. Set Frequency Sweep – Choose a logarithmic sweep from 10 /Hz to 10 /MHz (or higher if needed).
2. Use Small Signal Models – Ensure that the components have accurate small signal models; otherwise, the AC result may be misleading.
3. Plot Bode Plots – Generate magnitude and phase plots for the transfer function (e.g., Vout/Vin).
4. Determine Key Metrics – Extract the gain bandwidth product, unity gain frequency, and phase margin from the plots.

Noise Analysis for Signal to Noise Ratio

In many analog applications, especially audio and sensor interfaces, noise performance is critical.

- Choose Noise Analysis – In PSpice, select Noise Analysis and define the frequency range of interest.
- Identify Noise Sources – Look at the contributions from resistors, transistors, and other components.
- Calculate SNR – Use the noise plot to compute the signal to noise ratio (SNR) at the output.
- Optimize – Replace noisy components with low noise alternatives or adjust biasing to reduce noise.

Monte Carlo Analysis for Process Variations

Manufacturing tolerances can shift circuit performance. Monte Carlo simulation helps predict the spread of key parameters.

1. Set Variation Parameters – Define tolerance percentages for resistors, capacitors, and transistor parameters.
2. Run Multiple Trials – Execute 100 or more simulation runs to generate a statistical distribution.
3. Analyze Results – Plot histograms for output voltage, gain, and offset. Ensure that the design meets the worst case specifications.
4. Apply Design Margins – If the distribution is too wide, consider adding trimming resistors or using a different topology.

Parametric Sweep for Design Optimization

Parametric sweeps let you explore how variations in component values affect performance.

- Define Sweep Variables – Choose one or more components (e.g., Rbias, Cload) and set their sweep ranges.
- Automate Trade Off Analysis – Use the Plot 2 D feature to see how output gain varies with Rbias.
- Identify Optimal Points – Select component values that balance performance, power consumption, and cost.

Common Pitfalls and How to Avoid Them

Incorrect Component Models

Using generic or outdated models can lead to simulation results that diverge from reality. Always download the latest SPICE models from the manufacturer and verify their validity for your supply voltage and temperature range.

Overlooking Parasitics

Parasitic capacitances and inductances can drastically affect high frequency performance. If you notice unexpected oscillations, add parasitic elements or use a more detailed layout level simulation.

Ignoring Convergence Issues

PSpice may fail to converge due to large DC biases or highly nonlinear components. To resolve this, try the following:

- Use the Initial Conditions option to provide a starting point.
- Adjust the Maximum Step Size or switch to the Newton solver.
- Simplify the circuit temporarily to isolate the problematic section.

Inadequate Probe Placement

Baca Juga (Artikel Terkait)

- **advanced engineering electromagnetics solution manual edition hardcover by balanis constantine a published by wiley**
  URL: http://djjsearch.com/advanced-engineering-electromagnetics-solution-manual-edition-hardcover-by-balanis-constantine-a-published-by-wiley.pdf
- **advanced engineering electromagnetics solutions manual**
  URL: http://djjsearch.com/advanced-engineering-electromagnetics-solutions-manual.pdf
- **advanced engineering electromagnetics wiley 1989 grading**
  URL: http://djjsearch.com/advanced-engineering-electromagnetics-wiley-1989-grading.pdf
- **advanced engineering fluid mechanics by biswas**
  URL: http://djjsearch.com/advanced-engineering-fluid-mechanics-by-biswas.pdf