

---

# Getting started

---

# 1

This chapter describes how to start OrCAD Capture.

## Starting Capture

The OrCAD Release 9 installation process puts Capture in the `\PROGRAM FILES\ORCAD\CAPTURE` folder, and adds “Pspice Student” to the Programs menu (available from the Start button).

### To start Capture

- 1 From the Start menu, choose Programs. The Programs menu displays.
- 2 From the Pspice Student menu item, choose Capture.

# The Capture session frame

Once you start Capture, you see the Capture *session frame*. You do all your schematic design and processing within this window.



Figure 1 *Capture's session frame*

The minimized Session Log icon in the lower left portion of the Capture session frame is the session log. The session log provides information about everything you have done in the current Capture session. Detailed information about this window—and the other windows in Capture—is given in [Chapter 2, The Capture work environment](#).

In Capture, each design that you open is in a separate project manager window. If you need to work simultaneously with several designs, you can open them all, and each will have its own project manager window.

Depending on which type of window you have *active* (an active window is one whose title bar is highlighted), certain buttons on the toolbar and certain items on the menus may be unavailable, since you perform tasks and use tools based upon the type of window that is active. Also, the menus and menu choices vary, depending on which type of window is active. The available menus and menu choices will also vary depending upon the type of project you are working with.

# Creating a design for PSpice A/D simulation

When creating designs for PSpice A/D simulation, use Capture's Analog or Mixed-Mode Project Wizard to assist you in setting up your project. Only use parts from libraries specifically created for PSpice, or your own libraries. These libraries can be found in your `\PROGRAM FILES\ORCAD\CAPTURE\LIBRARY\PSPICE` directory.

For analog circuits, use the "0" ground symbol found in `SOURCE.OLB`, or use one of the other ground symbols and rename it to "0."

## To create a new PSpice A/D design

- 1 From the File menu, choose New and then Project. The New Project dialog box appears.
- 2 Enter a name for your new PSpice project. This name will also serve as the design's name.
- 3 Click the Browse button to specify the new location for your project and design.
- 4 Select Analog or Mixed-Signal Circuit Wizard.
- 5 Click OK. The Analog Mixed-Mode Project Wizard appears.
- 6 Select the PSpice part symbol libraries as desired. Add them by clicking the Add button.
- 7 Click the Finish button. Capture creates a new project and design.

# The schematic page editor

The schematic page editor is used to display and edit schematic pages. You can place parts, wires, buses, and draw graphics. The schematic page editor has a tool palette that you can use to draw and place everything you need to create a schematic page. You can print from within the schematic page editor, or from the project manager window.

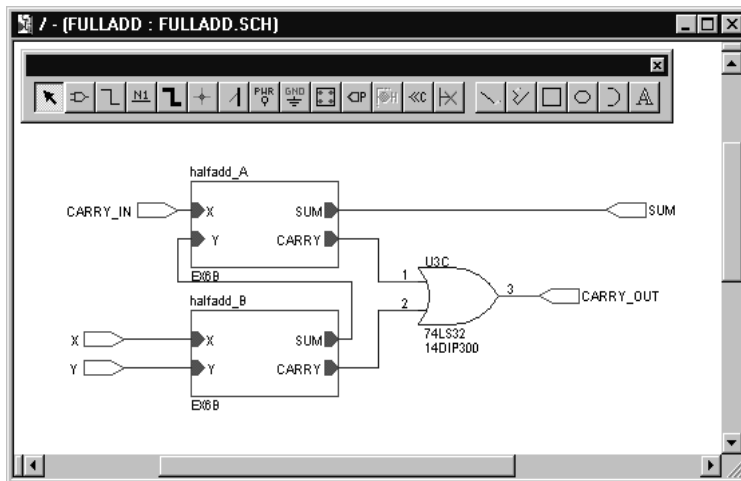


Figure 5 Schematic page editor

# Analyses you can run with PSpice A/D

See Chapter 2, *Simulation examples*, for introductory examples showing how to run each type of analysis.

See Part three, *Setting Up and Running Analyses*, for a more detailed discussion of each type of analysis and how to set it up.

## Basic analyses

### DC sweep & other DC calculations

These DC analyses evaluate circuit performance in response to a direct current source. Table 1 summarizes what PSpice A/D calculates for each DC analysis type.

**Table 1** *DC analysis types*

For this DC analysis...	PSpice A/D computes this...
DC sweep	Steady-state voltages, currents, and digital states when sweeping a source, a model parameter, or temperature over a range of values.
Bias point detail	Bias point data in addition to what is automatically computed in any simulation.
DC sensitivity	Sensitivity of a net or part voltage as a function of bias point.
Small-signal DC transfer	Small-signal DC gain, input resistance, and output resistance as a function of bias point.

## AC sweep and noise

These AC analyses evaluate circuit performance in response to a small-signal alternating current source. Table 2 summarizes what PSpice A/D calculates for each AC analysis type.

**Table 2** *AC analysis types*

For this AC analysis...	PSpice A/D computes this...
AC sweep	Small-signal response of the circuit (linearized around the bias point) when sweeping one or more sources over a range of frequencies. Outputs include voltages and currents with magnitude and phase; you can use this information to obtain Bode plots.
Noise	For each frequency specified in the AC analysis: <ul style="list-style-type: none"><li>• Propagated noise contributions at an output net from every noise generator in the circuit.</li><li>• RMS sum of the noise contributions at the output.</li><li>• Equivalent input noise.</li></ul>

---

**Note** *To run a noise analysis, you must also run an AC sweep analysis.*

## Transient and Fourier

These time-based analyses evaluate circuit performance in response to time-varying sources. Table 3 summarizes what PSpice A/D calculates for each time-based analysis type.

**Table 3** *Time-based analysis types*

For this time-based analysis...	PSpice A/D computes this...
Transient	<p>Voltages, currents, and digital states tracked over time.</p> <p>For digital devices, you can set the propagation delays to minimum, typical, and maximum. If you have enabled digital worst-case timing analysis, then PSpice A/D considers all possible combinations of propagation delays within the minimum and maximum range.</p>
Fourier	DC and Fourier components of the transient analysis results.

**Note** *To run a Fourier analysis, you must also run a transient analysis.*

## Advanced multi-run analyses

The multi-run analyses—parametric, temperature, Monte Carlo, and sensitivity/worst-case—result in a series of DC sweep, AC sweep, or transient analyses depending on which basic analyses you enabled.

### Parametric and temperature

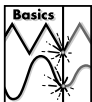
For parametric and temperature analyses, PSpice A/D steps a circuit value in a sequence that you specify and runs a simulation for each value.

Table 4 shows the circuit values that you can step for each kind of analysis.

**Table 4** *Parametric and temperature analysis types*

For this analysis...	You can step one of these...
Parametric	global parameter model parameter component value DC source operational temperature
Temperature	operational temperature

**Note** *Parametric analysis is not supported in PSpice A/D Basics.*



## Files that PSpice A/D generates

After reading the circuit file, netlist file, model libraries, and any other required inputs, PSpice A/D starts the simulation. As simulation progresses, PSpice A/D saves results to two files—the data file and the PSpice output file.

For a description of how to display simulation results, see Part four, *Viewing results*.

For a description of the waveform analyzer program, see *What is waveform analysis?* on page 46.

There are two ways to add waveforms to the display:

- From within PSpice A/D, by specifying trace expressions.
- From within Capture, by cross-probing.

Example: Each instance of a VPRINT1 symbol placed in your schematic causes PSpice A/D to generate a table of voltage values for the connecting net, and to write the table to the PSpice output file.

### Waveform data file

The data file contains simulation results that can be displayed graphically. PSpice A/D reads this file automatically and displays waveforms reflecting circuit response at nets, pins, and parts that you marked in your schematic (cross-probing). You can set up your design so PSpice A/D displays the results as the simulation progresses or after the simulation completes.

After PSpice A/D has read the data file and displays the initial set of results, you can add more waveforms and perform post-simulation analysis of the data.

### PSpice output file

The PSpice output file is an ASCII text file that contains:

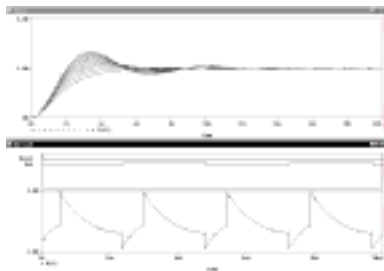
- the netlist representation of the circuit,
- the PSpice command syntax for simulation commands and options (like the enabled analyses),
- simulation results, and
- warning and error messages for problems encountered during read-in or simulation.

Its content is determined by:

- the types of analyses you run,
- the options you select for running PSpice A/D, and
- the simulation control symbols (like VPRINT1 and VPLOT1) that you place and connect to nets in your design.

# Analyzing waveforms with PSpice A/D

Taken together, simulation and waveform analysis is an iterative process. After analyzing simulation results, you can refine your design and simulation settings and then perform a new simulation and waveform analysis.



## What is waveform analysis?

After completing the simulation, PSpice A/D plots the waveform results so you can visualize the circuit's behavior and determine the validity of your design.

Perform post-simulation analysis of the results. This means you can plot additional information derived from the waveforms. What you can plot depends on the types of analyses you run. Bode plots, phase margin, derivatives for small-signal characteristics, waveform families, and histograms are only a few of the possibilities. You can also plot other waveform characteristics such as rise time versus temperature, or percent overshoot versus component value.

Pinpoint design errors in digital circuits. When PSpice A/D detects setup and hold violations, race conditions, or timing hazards, a detailed message appears along with corresponding waveforms. PSpice A/D also helps you locate the problem in your design.