

## Introduction

PAC-Designer software, a Windows-based design tool from Lattice Semiconductor gives users the capability to graphically design analog filters and other analog functions for ispPAC devices. ispPAC devices are In-System-Programmable (ISP<sup>™</sup>) Analog Circuits. The user describes the properties or parameters in the GUI on a schematic or entries in the Filter Table. The software then allows the user to simulate the filter response in the simulation window. Once the design properties are confirmed, the ispPAC devices can be programmed or reprogrammed while soldered to the PC board. The configuration information for the filter and other analog functions are stored in non-volatile E<sup>2</sup>CMOS<sup>®</sup> directly on the chip. This technology allows the user to make changes to the analog circuit without changing external components. The use of ispPAC technology increases the integration of components, speeds the design prototype turn cycle and allows a designer the capability to easily make design changes.

## PSpice Support

For systems that require further simulation with other components on the board, PAC-Designer supports PSpice simulation tools. PAC-Designer software version 1.2.1 exports PSpice models for devices such as ispPAC10 and ispPAC80 devices.

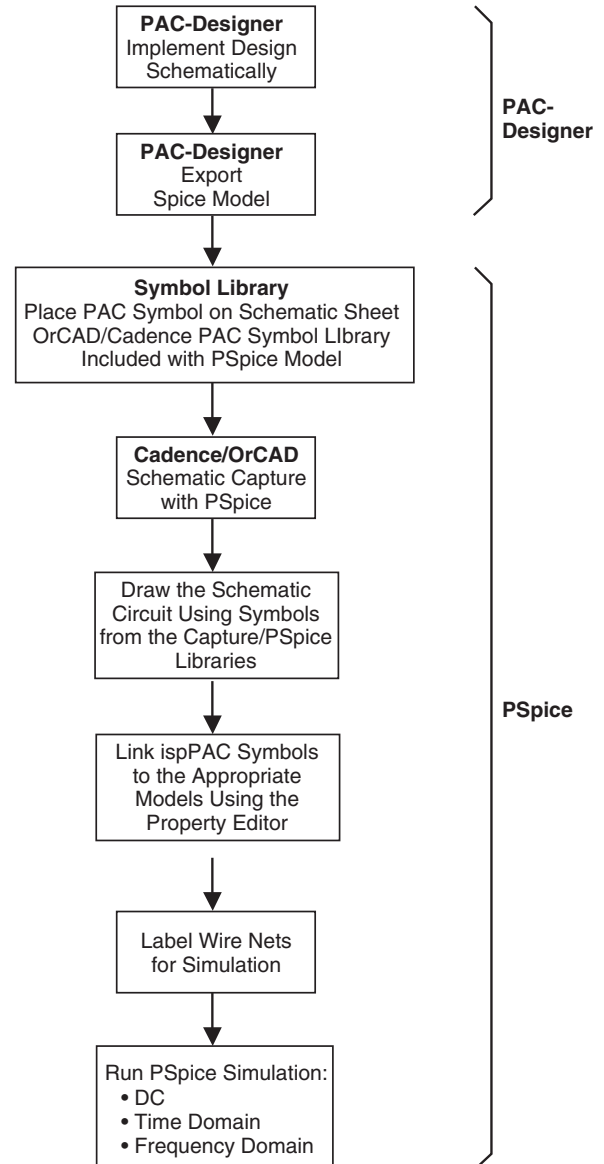
The support capabilities allow the user to perform Time-Domain or Frequency-Domain simulations for ispPAC devices in combination with other analog components, both active and passive. The user must have a version of Orcad PSpice and Orcad Capture tools to perform schematic capture and simulation. PAC-Designer exports the required model that is used in the PSpice simulation. Each circuit that is configured by the user in PAC-Designer will have a unique corresponding model, which can be exported after any design changes are performed.

### Requirements:

- Lattice PAC-Designer Software version 1.2.1
- Orcad Schematic Capture
- Orcad PSpice (Full Version 9.2)
- PC Computer with Windows 95/98/2000/NT

The following information outlines the steps necessary to perform a PSpice model export from PAC-Designer and

Figure 1. Model Simulation Flow



to perform a PSpice simulation using Orcad PSpice on a PC platform. For further information on the use of Orcad PSpice or Schematic Capture tools, please refer to the appropriate user manuals from the manufacturer. Orcad is a Cadence product family.

Once the ispPAC symbol is set up on the schematic and linked to the appropriate library, the simulation is performed completely by the Orcad PSpice tool set. The Orcad schematic capture tool allows the user to place multiple ispPAC devices on the schematic along with any

# PSpice Simulation Using ispPAC SPICE Models and PAC-Designer

other valid library components. This allows users to simulate the functionality of a complete board or system.

## Signal Paths and Functions

The analog signal paths for the ispPAC devices are supported in the model libraries. For the ispPAC80 and the ispPAC10, the analog functions that are exported to the Spice model netlist include the filter or analog function signal path from the ispPAC analog inputs to the ispPAC analog outputs. The 2.5V reference pin  $VREF_{OUT}$  is also supported in the simulation model and can be used to reference input pins to 2.5V or be used for other circuit reference voltages or biasing circuits. The JTAG programming pin functions for reprogramming the device are not supported in the simulation; the configuration of the device comes from the model that was exported by PAC-Designer. For the ispPAC80, the model supports the filter configuration that is stored in the “A” configuration registers. The SPI functions and other digital functions are not supported in the model. To change configurations for simulation purposes, simply export a new Spice Model with PAC-Designer and link the path to the ispPAC schematic symbol.

## PAC-Designer Export Capabilities

PAC-Designer Version 1.2.1 includes the PSpice export option. After the design is complete in the PAC-Designer Schematic Window, save the file and export the SPICE model for PSpice. PAC-Designer will write a PSpice

compatible model that relates the current ispPAC configuration to the netlist for simulation.

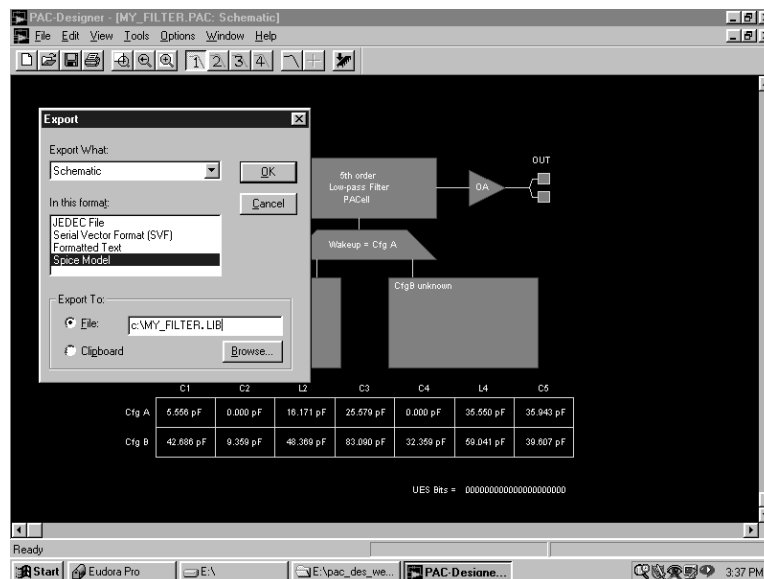
## Exporting the PSpice Netlist from PAC-Designer

Under the “File” menu on the Top Tool Bar, select the “Export” options. Select the “Spice Model” option to export a model that corresponds to the active PAC-Designer schematic (Figure 2). If changes are made to the schematic in PAC-Designer, the user must export a new model so the changes are reflected in the PSpice netlist and the PSpice simulation.

## Adding Libraries to the Schematic Design in ORCAD/Cadence Capture

To Add a Library to the Project, choose the “Place” command on the top Tool Bar or click on the Part icon on the right Schematic Tool Bar. This will bring up a window that will allow you to add Libraries to your design. Once the Libraries are listed in the lower section, individual components can be added to the schematic (Figure 3). Connect the necessary components for the schematic and label Net names on Inputs and Outputs for simulation. All Libraries that are active in the design will show up in the lower Library. Components within a given Library will be listed in the upper part window. The symbol will be shown for the component that is selected from the menu.

Figure 2. Exporting the Spice Model from PAC-Designer



# PSpice Simulation Using ispPAC SPICE Models and PAC-Designer

Figure 3. Adding Libraries to the Project

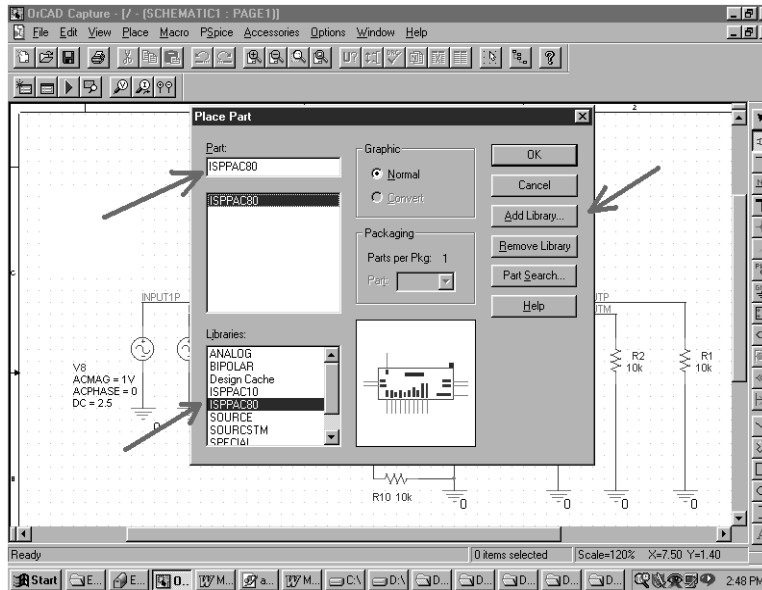
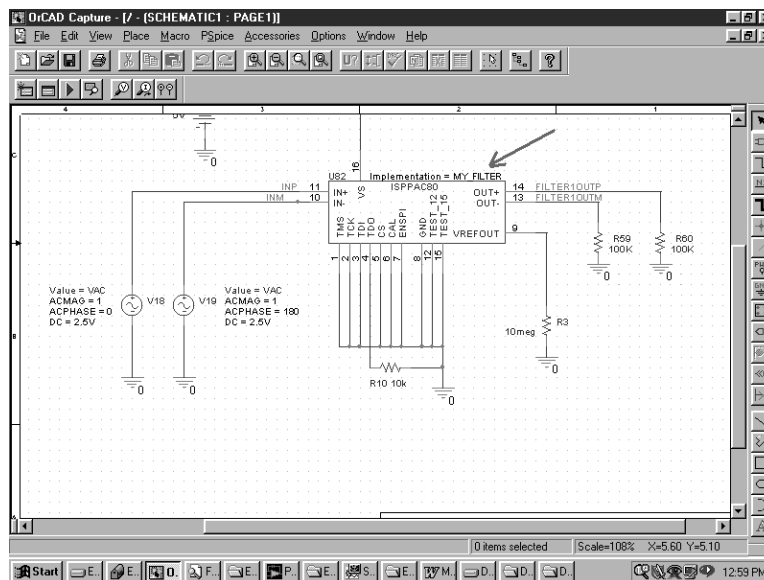


Figure 4. Implementation Property



The Directory location for the Libraries is up to the user. Make sure the ispPAC libraries and the symbols are in an easy to locate local directory. The file name extension for a Library is .LIB. The Orcad/Cadence schematic capture symbols that are included with PAC-Designer have an .OLB extension. These are located in the Spice subdirectory of the PAC-Designer installation directory.

## Editing the Implementation Property

Once the components are connected to the ispPAC device, the simulation can be run for either Time-Domain or Frequency-Domain. Each type of PSpice simulation

requires different setup parameters and different sources for the particular sweep that is being run. In the schematic shown, VAC sources were chosen for the Frequency-Domain simulation. Shown above the ispPAC80 symbol, is the "Implementation" Property. The Implementation Property is the name of the PSpice netlist that is associated with the ispPAC80 device symbol. The schematic can be drawn with the default netlist and then the Implementation Property can be edited for each ispPAC device on the schematic. The Implementation Property is the same as the filename.lib, of the PSpice Netlist file that the user exported from PAC-Designer software, shown as MY\_FILTER in this example (Figure 4).

# PSpice Simulation Using ispPAC SPICE Models and PAC-Designer

## Editing Properties for Symbols on the Schematic

The Property Editor controls schematic Symbol Properties. Any Symbol Property can be edited by double-clicking on the symbol to open the Property Editor. If Properties are already shown on the schematic as text, they can be edited individually by double-clicking and changing the value. Not all properties are visible on the

schematic sheet or need to be visible to be active. The Property Editor allows the user to display or hide any Property. Select the individual Property to edit and then click the Display button to activate the Display Menu.

The Implementation Property is shown as “MY-FILTER”. This represents a SPICE model that was exported from PAC-Designer. To point the symbol to another filter, change the Implementation Property (Figure 5).

Figure 5. Editing Properties

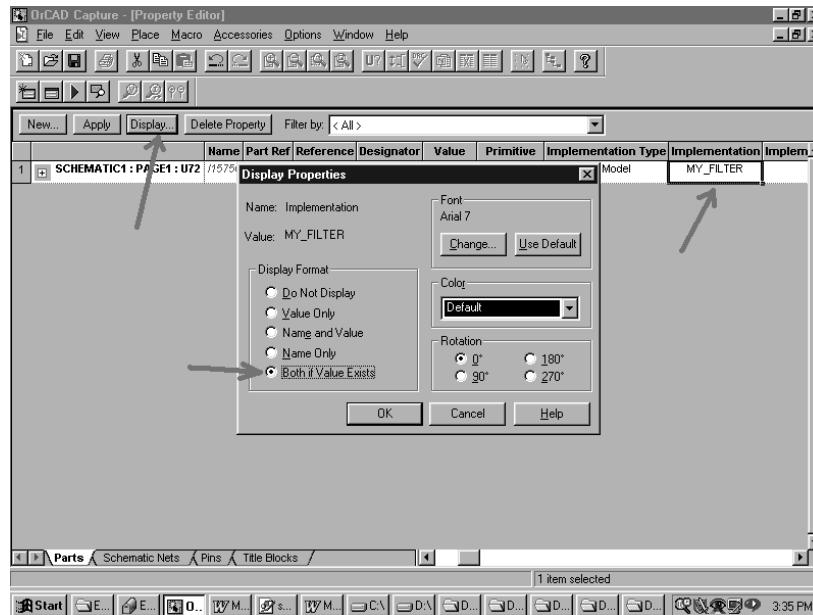
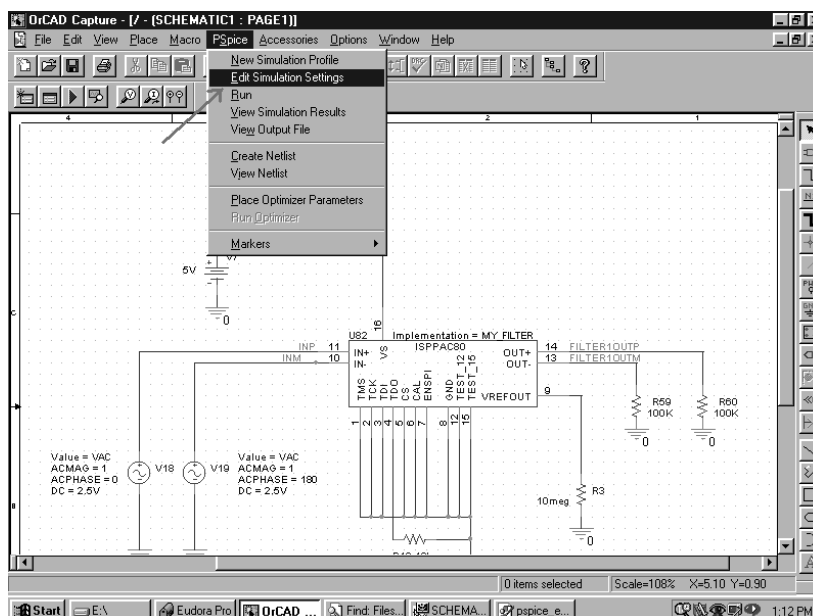


Figure 6. Editing PSpice Simulation Settings



# PSpice Simulation Using ispPAC SPICE Models and PAC-Designer

Figure 7. Simulation Settings – Library Paths

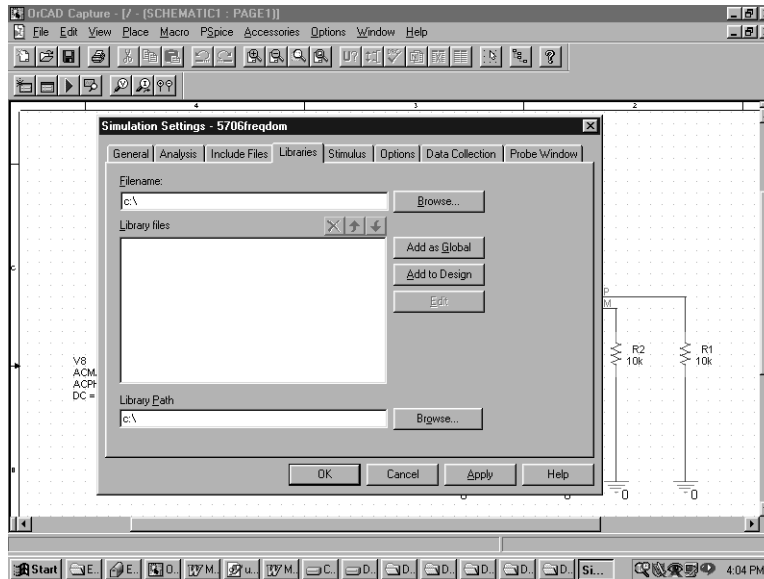
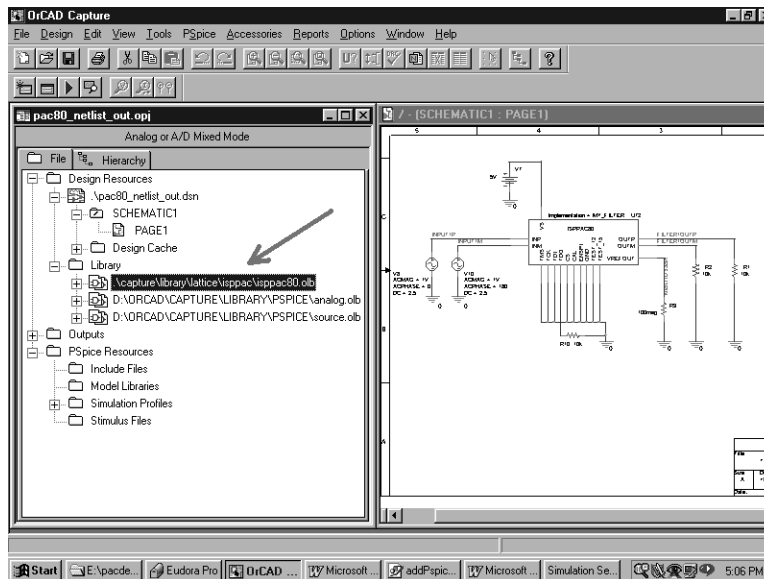


Figure 8. Project Directory Structure



## PSpice Simulation

To set up the schematic for simulation, PSpice will need to know where the Libraries are located. On the top tool bar, select “PSpice”, “Edit Simulation Settings”. Choose the “Library” tab and select the path where the Libraries are located. The Libraries can be Local or Global for the design. Once the PSpice Libraries are set up, the simulation parameters can be set in the “Analysis” section of the same window (Figures 6 and 7).

## Project Directory Structure

The window on the left shows the Project Directory Structure. The Project consists of Schematics, Libraries and Simulation profiles along with Output files and Cache files. After The Libraries are added using the “PSpice” “Edit Simulation Settings,” they will show up as part of the Project and be included in the directory structure. Project files can be added or deleted from the Project directories using the directory tree shown on the left side of the window (Figure 8).

# ***PSpice Simulation Using ispPAC SPICE Models and PAC-Designer***

---

Once the Libraries are in place and the schematic is ready, the simulation can be carried out as a standard PSpice operation and flow.

## **Technical Support Assistance**

Toll Free Hotline: 1-800-LATTICE (Domestic)  
International: 1-408-826-6002  
E-mail: [ispPACs@latticesemi.com](mailto:ispPACs@latticesemi.com)  
Internet: <http://www.latticesemi.com>