cadence®

PSpice A/D Simulations in Allegro® X System Capture

Product Version 23.1
September 2023

© 2023 Cadence Design Systems, Inc. All rights reserved.
Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Product PSpice contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

All other trademarks are the property of their respective holders.

Restricted Permission: This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information. Cadence is committed to using respectful language in our code and communications. We are also active in the removal and/or replacement of inappropriate language from existing content. This product documentation may however contain material that is no longer considered appropriate but still reflects long-standing industry terminology. Such content will be addressed at a time when the related software can be updated without end-user impact.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

Contents

1
Simulating System Capture Designs 5
Setting Up Simulation Environment 6
Set Up PSpice Libraries 6
Enabling PSpice in System Capture 7
Creating New Project 8
Placing Components 10
Placement Checklist for PSpice-Compatible Designs 10
Creating Parts Based on Model Parameters 12
Setting Up Simulation Profiles 14
Modifying a Simulation Profile 14
Creating a New Simulation Profile 15
Simulating the Design 16
Viewing Output Waveforms 17
Enabling Bias Display 19

2
Simulation Models and Stimulus 21
Associating PSpice Model 21
Editing Models 24
Adding Circuit Stimulus 24
Using Stimulus Editor 24
Creating Custom Digital Power Supplies 26

3
Simulating Hierarchical Design 29
Simulating a Block Schematic 30
Using Alternate Simulation Profiles 32

@ 2023 All Rights Reserved.

4
Running Analysis 35
Parametric Sweep 35
Advanced Analysis 37

@ 2023 All Rights Reserved.

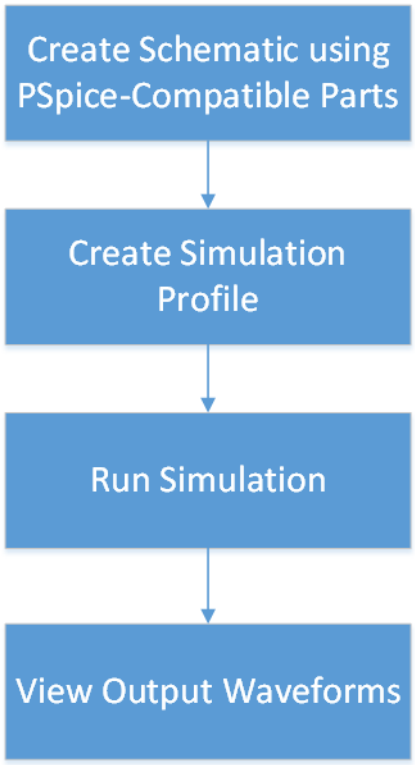
1

Simulating System Capture Designs

With the integrated System Capture – PSpice flow, schematic designs created in System Capture can be easily simulated in PSpice A/D. The integration provides seamless access to PSpice features from within System Capture.

Important

System Capture - PSpice flow is available only on Windows.



System Capture – PSpice Flow

@ 2023
All Rights Reserved.

To simulate a design in PSpice, components must have an associated PSpice model.You can use components from PSpice-compatible Cadence-supplied libraries, or create new components for the available models.

Setting Up Simulation Environment

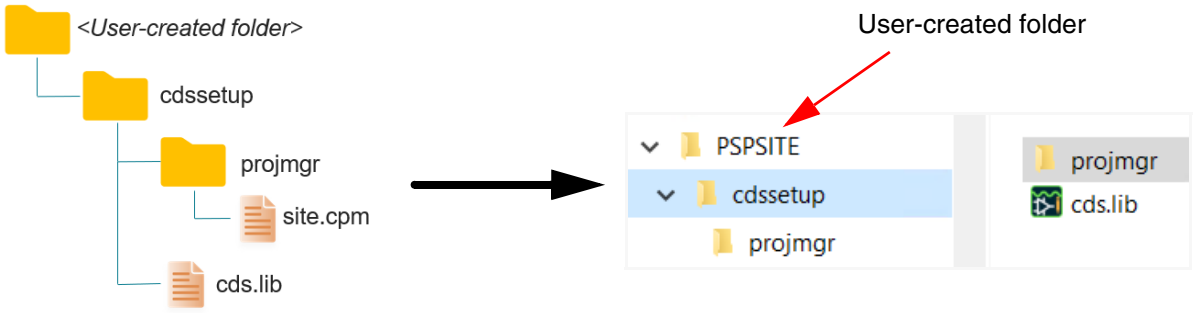
Before you can create a System Capture schematic that can be simulated using PSpice, you need to complete the following tasks:

- Set Up PSpice Libraries
- Enabling PSpice in System Capture

Set Up PSpice Libraries

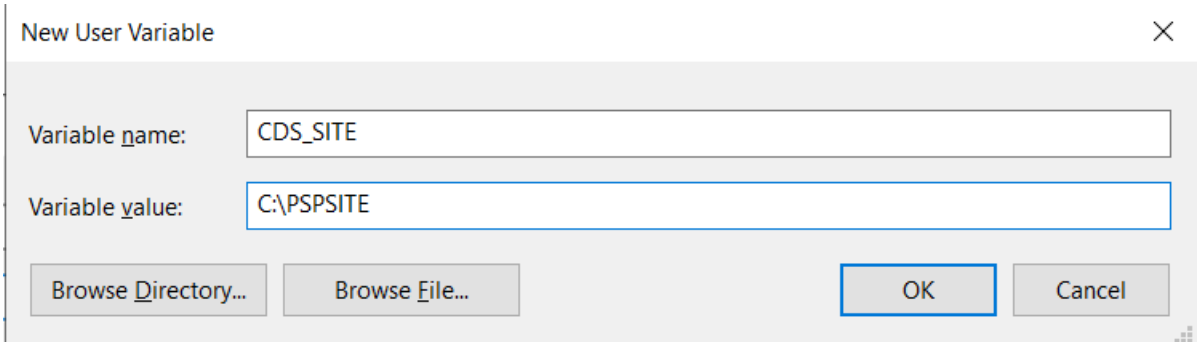
Cadence-supplied libraries are shipped as a self-extracting executable in the installation. To make these component libraries available in System Capture, complete the following tasks:

1. Navigate to `<install_dir>/share/canvaslibrary/pspice`, and double-click `pspicecanvaslib.exe` to extract the PSpice libraries.
2. Create a new folder with the following folder structure.



@ 2023
All Rights Reserved.

3. Set the `CDS_SITE` variable to point to the folder created in step 2



   Setting the `CDS_SITE` variable ensures that parts from the Cadence-supplied libraries can be instantiated in System Capture.

4. Modify `cds.lib` to include the Cadence-supplied PSpice libraries, the standard library, and any other libraries that you need to use during the schematic design process.

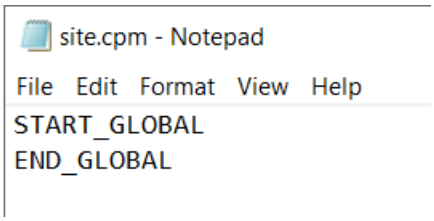
cds.lib - Notepad

File Edit Format View Help

```
DEFINE standard <Installation_hierarchy>/share/library/standard
   ## standard library is must for System Capture

INCLUDE <Installation_hierarchy>share/canvaslibrary/pspice/canvaspsp_cds.lib
   ## Path to the folder containing libraries extracted from pspicecanvaslib.exe
```

5. Under the *projmgr* folder, create a `site.cpm` file with statements `START_GLOBAL` and `END_GLOBAL`.



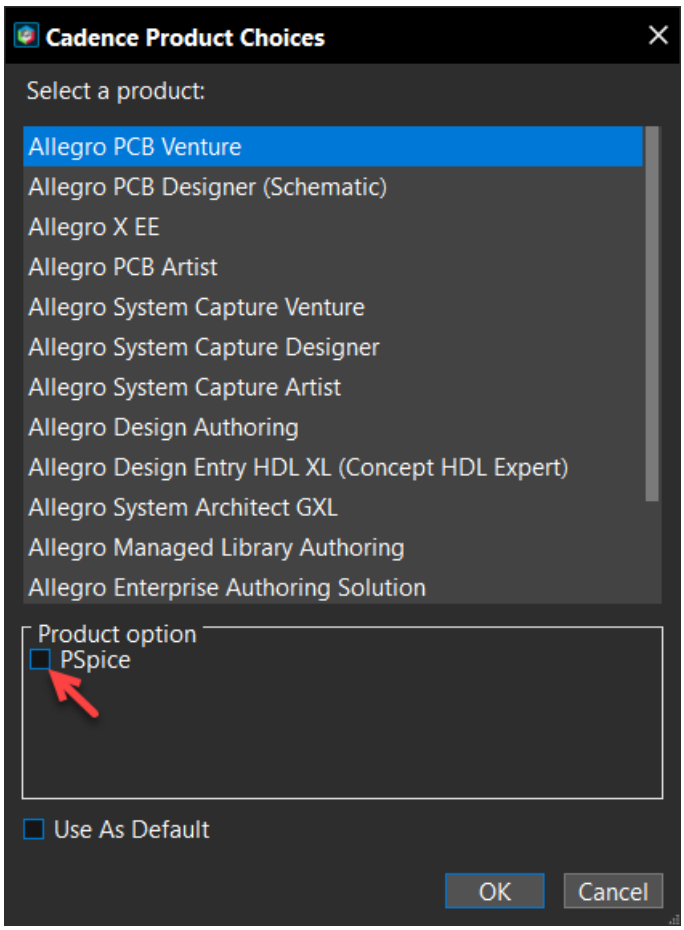
   The `site.cpm` file is required to ensure that the PSpice library components shown under the *Library* tab of Unified Search panel in System Capture.

Enabling PSpice in System Capture

To enable the System Capture - PSpice integration flow, you need an enterprise license, such as *PS2200* or *SDA 200*.

@ 2023 All Rights Reserved.

➡ When launching System Capture, select the *PSpice* product option in addition to the System Capture license, in the Cadence Product Choices dialog box.



Note: Selecting the option enables PSpice toolbar and menus in System Capture.

System Capture starts and a dialog box listing the available PSpice licenses is displayed.

Creating New Project

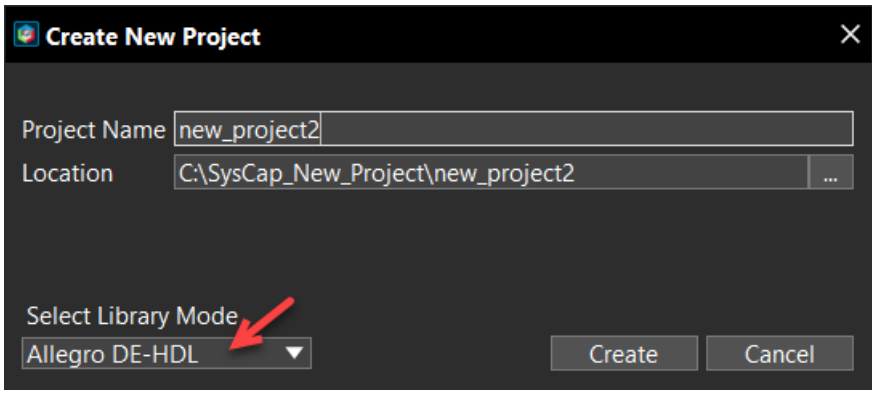
System Capture-PSpice flow is supported only for the designs that are based on Design Entry HDL libraries.

To create a new project using Design Entry HDL libraries, do the following steps:

1. Choose *File – New Project*.
2. Specify the project name and location.

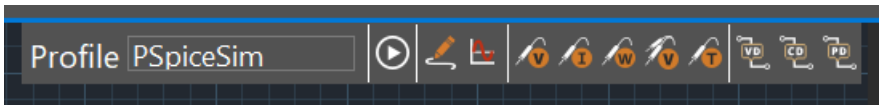
@ 2023
All Rights Reserved.

3. From the *Select Library Mode* list, select *Allegro DE-HDL* and click *Create*.

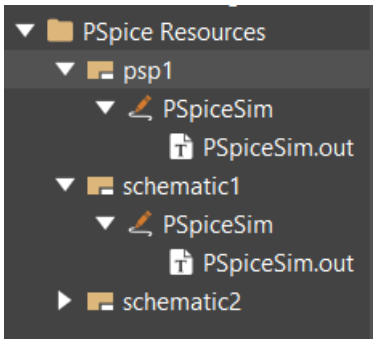


A project is created, and a blank canvas opens in System Capture.

A new menu—*PSpice*—is added to the menu bar. In addition, a new floating toolbar is added on the upper-right corner of the canvas. The toolbar contains icons to run simulations, edit simulation profiles, view simulation results, and add markers.



The Project Explorer panel displays a new entry, *PSpice Resources*. Under this node, PSpice resources for the entire design are listed. If a design has multiple blocks, block-specific resources are listed under each block name.



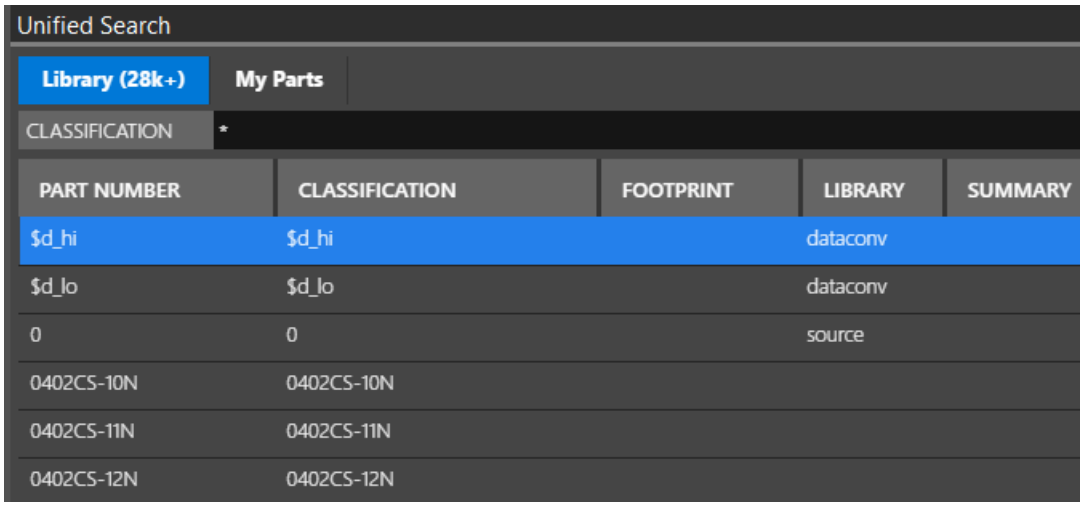
@ 2023
All Rights Reserved.

Placing Components

Parts from the libraries included in the project are available for placement from the Unified Search panel.

To place a part, do the following:

1. Search for the component in the Unified Search panel.
2. Right-click the component and choose *Place Part*.



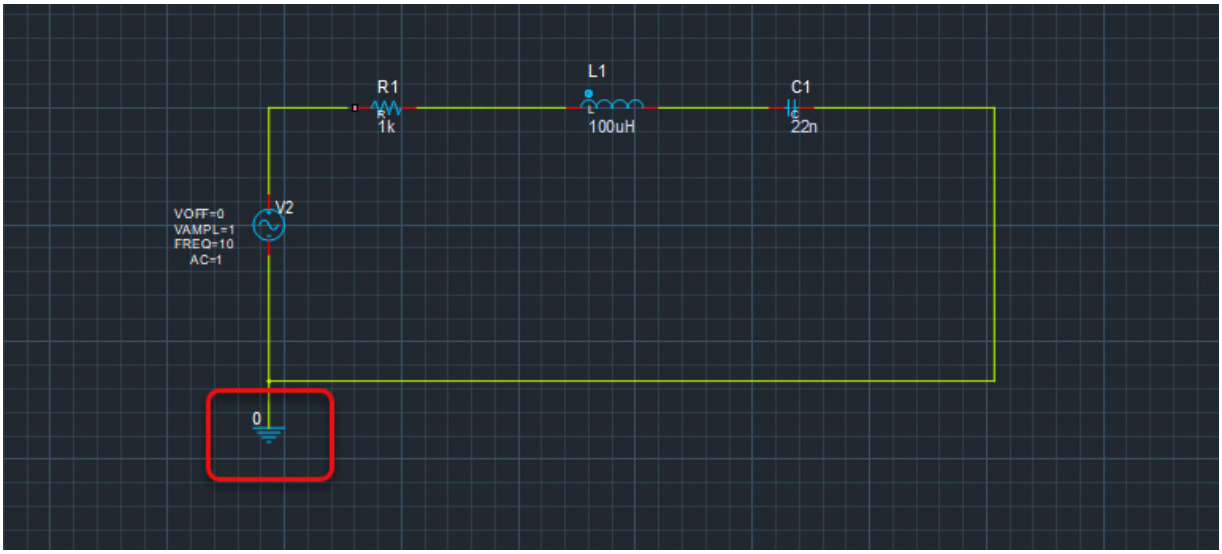
Placement Checklist for PSpice-Compatible Designs

To ensure that schematic designs created in System Capture can be simulated in PSpice, consider the following points:

- To run an analog simulation, the design must have at least one ground symbol.

@ 2023
All Rights Reserved.

Use the PSpice ground symbol, ‘0’, instead of GND. In the Unified Search panel, search for 0 and place the ground symbol. By default, this symbol is available in the source library.



- To identify unconnected pins in the design, instead of using the *No Connect* symbol, use the PSpice pin property, *FLOAT.* This is required to ensure that the generated PSpice netlist is correct. Valid values for the FLOAT property are:

e (Error)	If FLOAT property is set to e or ERROR, the pin is not netlisted. An error message is returned when the PSpice simulation netlist is generated. Use this value when you want to be reminded that this pin is a "no connect" and should be treated in a special way. Error is the default value.
r (RtoGND)	Adds a large virtual resistor to the pin. The second pin of the virtual resistor is grounded by connecting it to 0 symbol in the netlist. The resistor has a value of 1/GMIN. With this value, the simulation netlist can be created and PSpice analysis can be run. The virtual resistor is not processed as part of the layout netlist nor listed in the BOM.
n (UniqueNet)	A unique node is attached to the unconnected pin in the generated PSpice simulation netlist. Use this value when you want the pin to remain unconnected but correspond to the Probe data associated with its part.

@ 2023
All Rights Reserved.

u (Unmodeled)	Use of this value is recommended for unmodeled component pins. Unmodeled pins are non-functional for a simulation , and not present in a PSpice template and the model file.

Creating Parts Based on Model Parameters

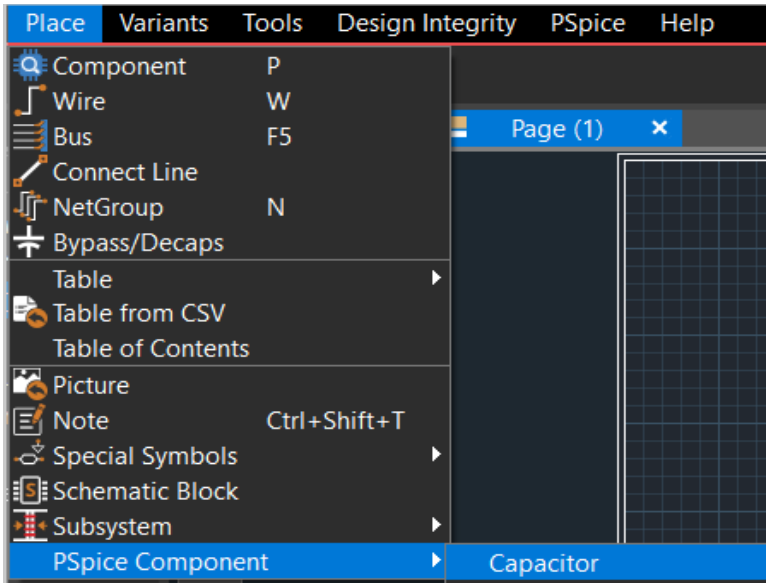
If you do not have the required part in an existing library, you can easily create part in System Capture by specifying the values of the electrical parameters from the datasheets to the model definition. Using this approach, you can create parts for the following component models:

- Capacitor
- Inductor
- Independent Sources
- Power Diode
- Power MOSET
- Switch
- Transformer
- TVS (Transient-Voltage-Suppression diode)
- VCO (Voltage Controlled Oscillators)
- Zener

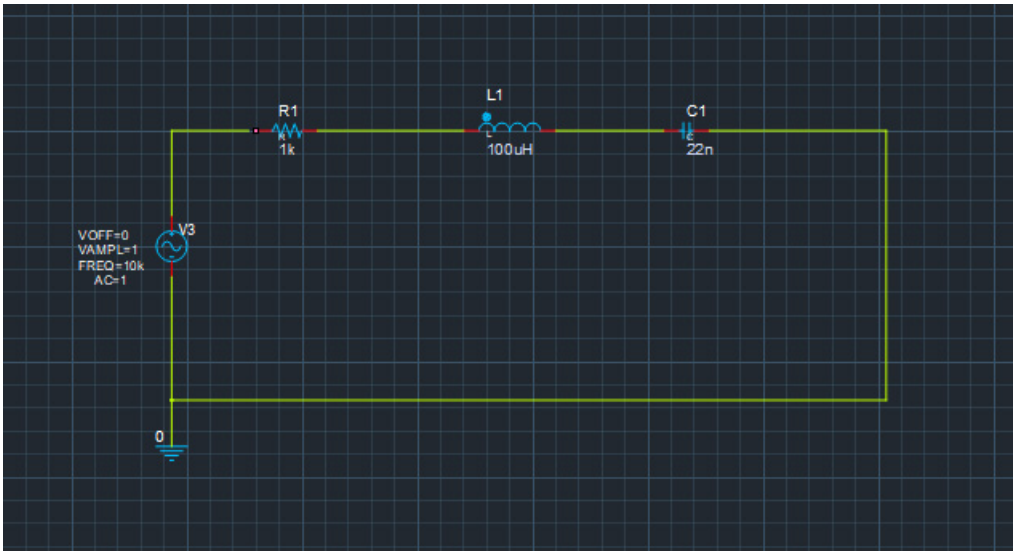
@ 2023 All Rights Reserved.

To model new parts based on electrical parameter values, do the following:

1. Choose *Place – PSpice Component*.



2. From the submenu select the model to create the component.
3. Specify the component parameter values.
4. Place a component instance on the schematic and complete the connectivity.



Related Topics

- Connecting Components

@ 2023
All Rights Reserved.

Setting Up Simulation Profiles

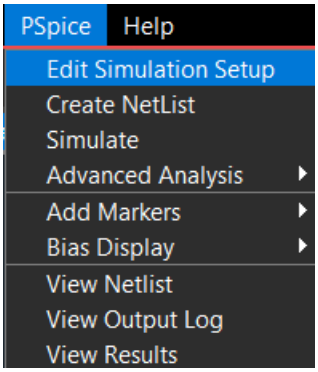
Inputs and instructions to the PSpice simulator are specified using simulation profiles. In a PSpice-enabled project, a default simulation profile is created for the schematic design. You can modify the default profile, or create a new one.

You can create multiple simulation profiles. You can have a different profile for different analysis type. This makes it quick and easy to run different types of analysis on a design.

Modifying a Simulation Profile

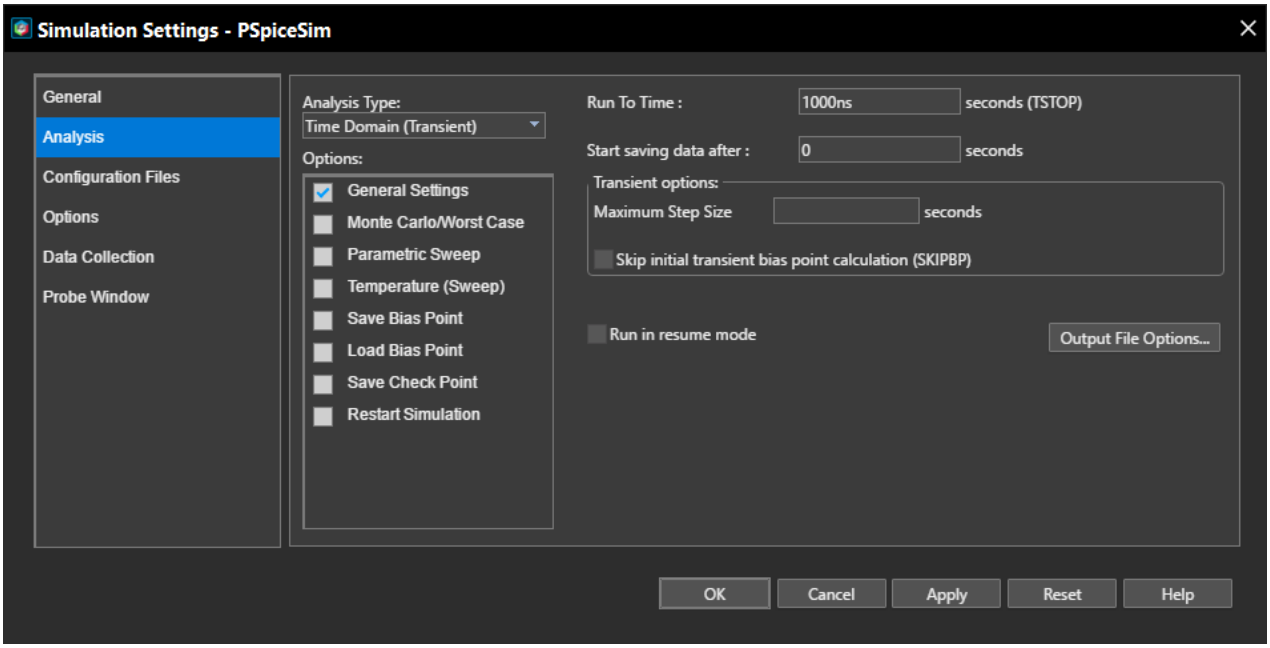
To edit an existing profile, do the following steps:

1. Choose *PSpice – Edit Simulation Setup.*



@ 2023
All Rights Reserved.

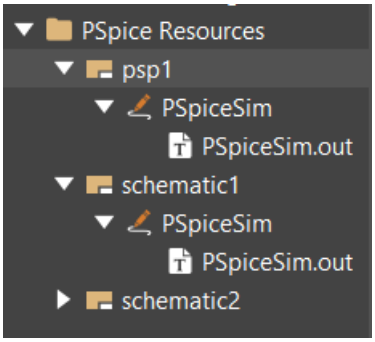
2. In the *Simulation Settings* dialog, modify the simulation parameters as required, and click *OK* to update the profile.



Creating a New Simulation Profile

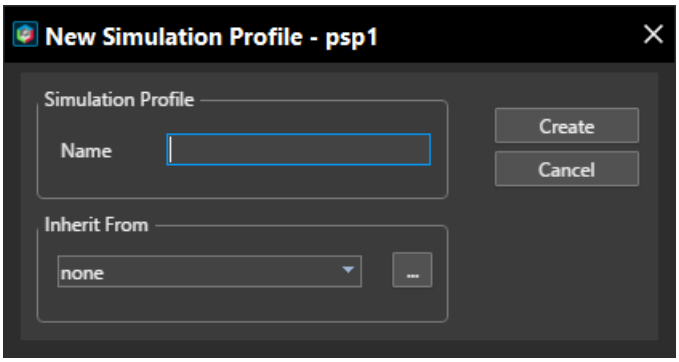
To create a new simulation profile, do the following steps:

1. In the Project Explorer panel, open *PSpice Resources*.



@ 2023
All Rights Reserved.

2. Right-click the project name and choose *New Simulation Profile*.



3. Specify a new profile name and click *Create*.

   In the Simulation Settings dialog box is displayed for the new profile.

4. Specify simulation details, such as the analysis type, simulator run time, maximum step size, and so on, and click *OK*.

The new simulation profile is created and set as the current simulation profile.

Related Topics

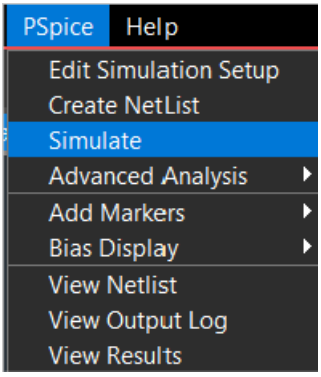
- Analysis settings for simulation profiles

Simulating the Design

You can run simulation for the design you created and display the simulation results in the PSpice Simulator Probe window.

To simulate the design, use one of the following methods:

- Choose *PSpice – Simulate*.



@ 2023
All Rights Reserved.

- Click *Run Simulation* from the PSpice toolbar.
- Right-click on the simulation profile for the design in the Project Explorer panel and choose *Simulate*.

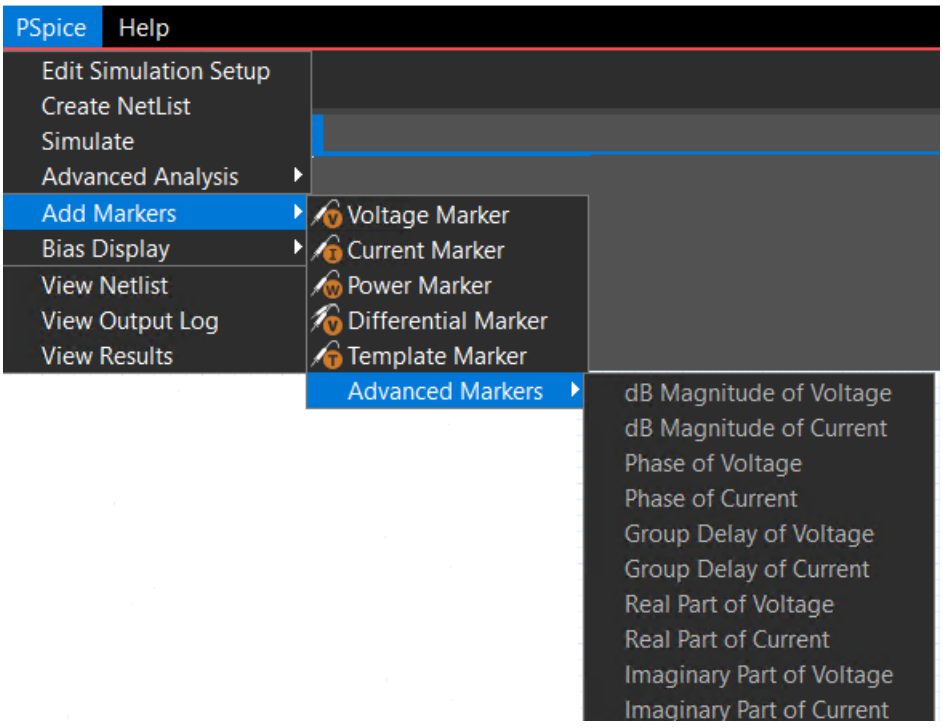
As the simulation starts, the PSpice Waveform Viewer window is displayed in the background.

Viewing Output Waveforms

To display output waveforms in PSpice, you need to place markers in your design to indicate the points where you want to see the output waveforms.

1. From the PSpice toolbar, select the required markers and place them in the design to display traces in the PSpice Waveform Viewer window.

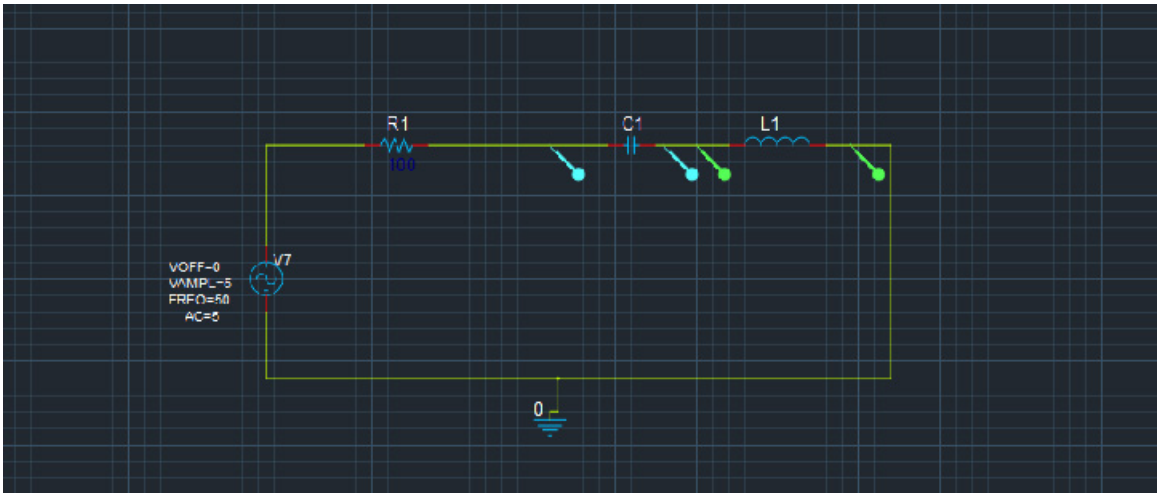
You can also use the *PSpice – Add Markers* menu to add markers.



2. Select the marker and place it in the design to display the traces in PSpice Waveform Viewer window.

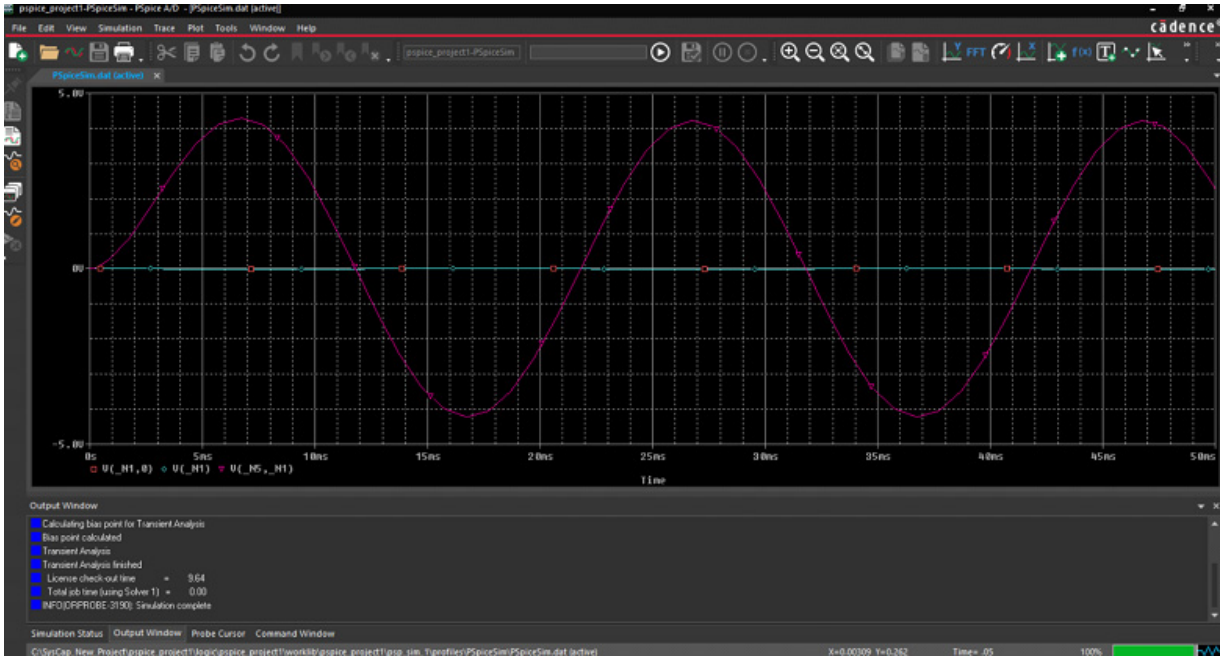
@ 2023
All Rights Reserved.

Note: In System Capture, you can place multiple voltage markers on the same net in a same or a different location.



The simulation output is displayed as waveform in the Waveform Viewer window.

3. You can also add Measurements in the Probe window.



Related Topics

- Compose Measurements
- Add Traces

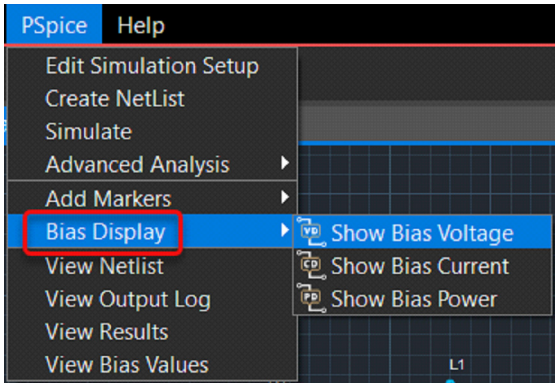
@ 2023
All Rights Reserved.

Enabling Bias Display

After simulating the design, you can display bias point information, such as bias voltage, bias current, and bias power, on your schematic to identify potential problem areas in your design.

To enable bias display in your design, do the following in System Capture:

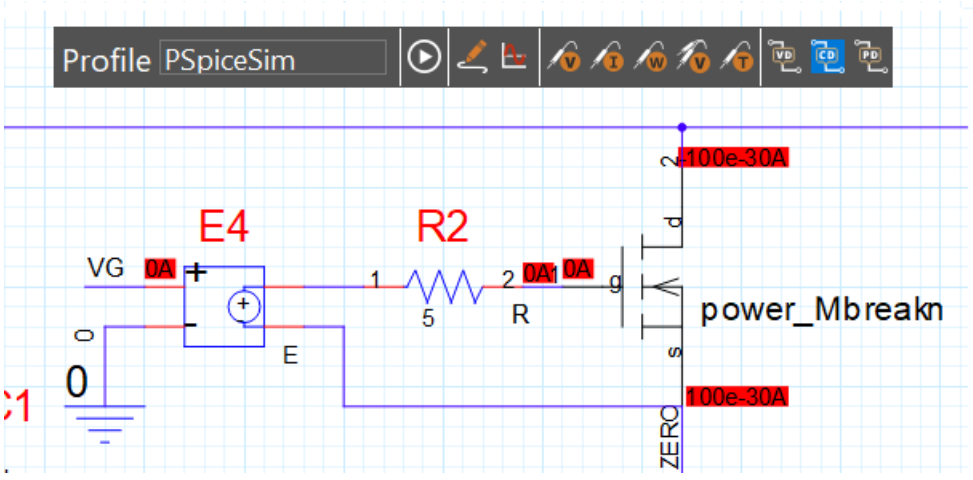
1. Choose *PSpice – Bias Display*. The available bias display options can be selected from the submenu.



2. Choose the required option to display the corresponding bias point.

   Alternatively, use the PSpice toolbar to enable bias point display. Check the Bias displays in the toolbar that are marked as enabled.

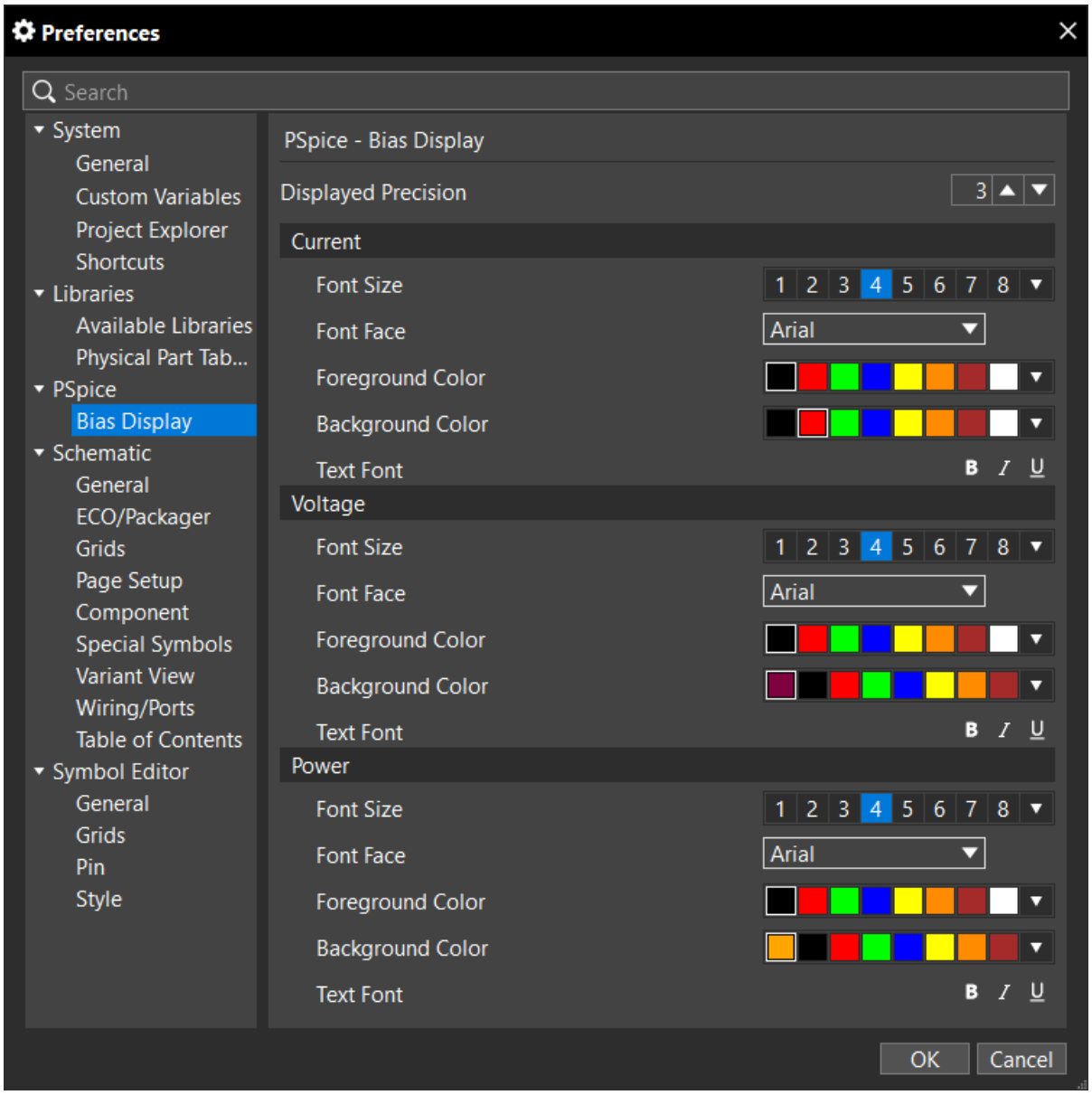
Selected bias point data is displayed on the schematic.



Note: Display settings of the bias point data, such as font size and color can be

@ 2023
All Rights Reserved.

customized using the Bias Display page of the *Preferences* dialog box.



@ 2023
All Rights Reserved.

2

Simulation Models and Stimulus

To simulate a design in PSpice, the components must have associated electrical models. In addition, inputs to the design, such as power supplies or stimulus for the circuit, also need to be specified.

Tip

When you copy or archive a PSpice-enabled System Capture project, PSpice related information is retained including simulation profiles, marker information, and simulation outputs.

Associating PSpice Model

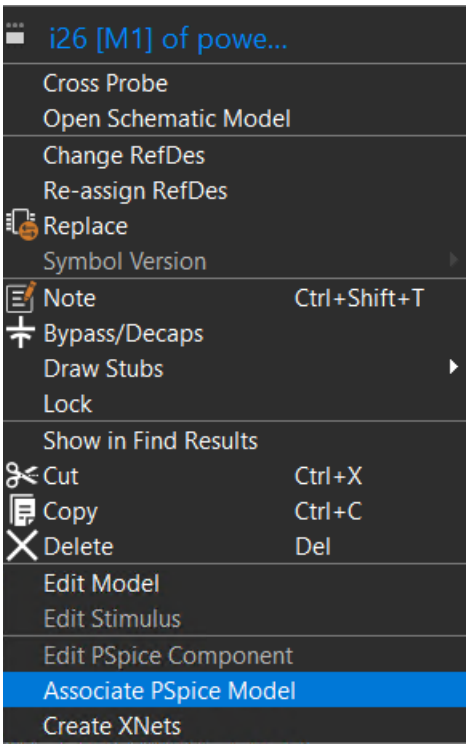
If a design component is not a Cadence-supplied PSpice compatible part, it can still be simulated in PSpice after associating a spice model to it.

PSpice supports associating custom or downloaded models to a component symbol on the schematic. To associate a PSpice model to a component instance, do the following steps:

1. Right-click the component.

@ 2023 All Rights Reserved.

2. Choose *Associate PSpice Model*.

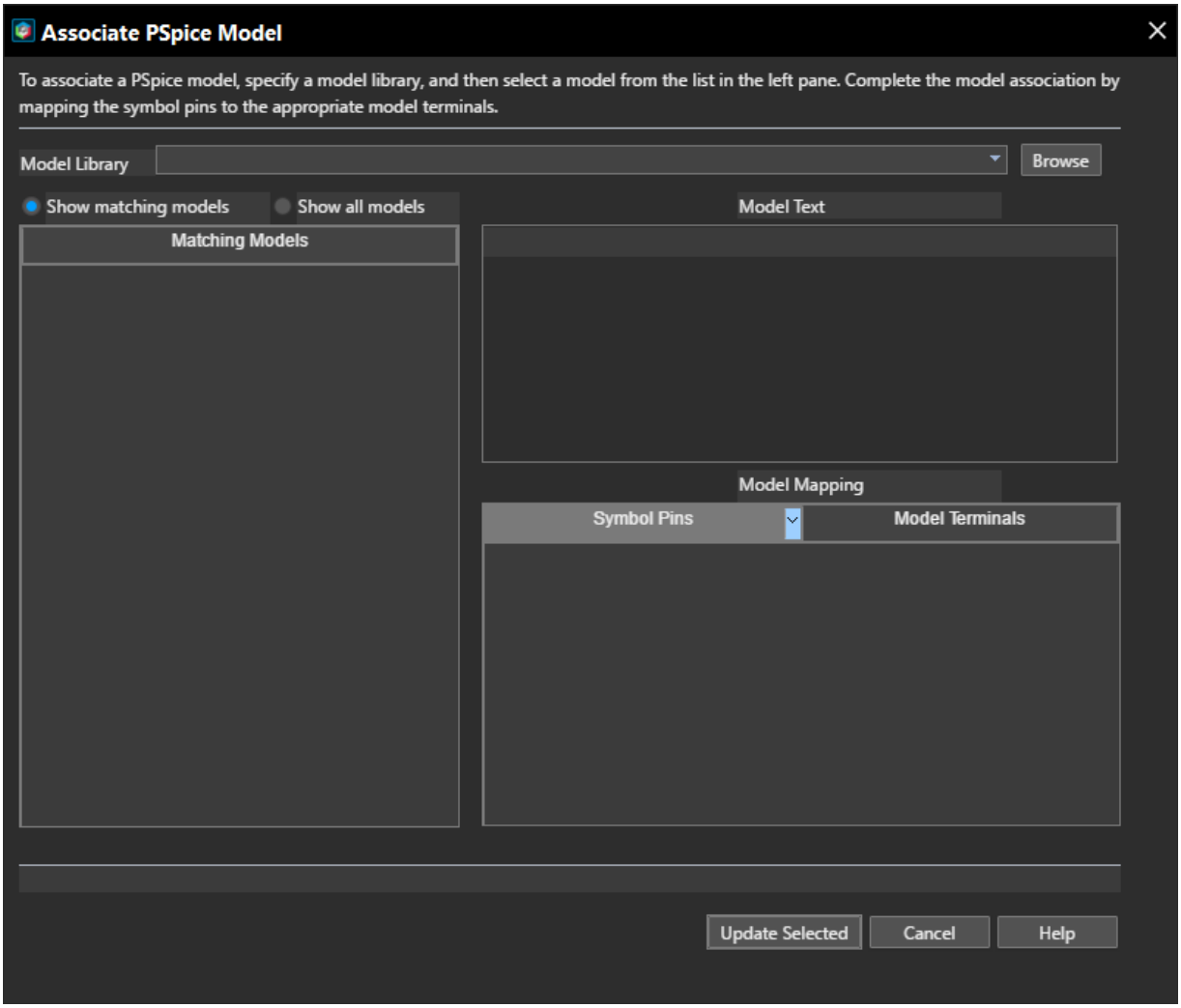


   If the selected part already has a PSpice model attached, a message appears to confirm if you want to overwrite this implementation.

3. To proceed, click *Yes*.

@ 2023
All Rights Reserved.

The *Associate PSpice Model* dialog box is displayed.



4. In the Model Library field, specify the location of the library containing the PSpice model to be associated with the component.

   The Matching Models section is populated with the list of models that can be associated with the selected component. The Model Text section displays the model definition of the model selected in the list.

5. In the Model Mapping section, map the symbol pins to the correct model terminal. For this, select a port from the *Model Terminal* for each symbol pin.

Tip

See the model text to identify the model terminal to be mapped to each symbol pin.

6. Click *Update Selected*.

@ 2023
All Rights Reserved.

The instantiated component instance has the PSpice model associated with it.

Editing Models

You can edit models associated with an instantiated component. To modify the model parameter values of a placed component, do the following steps:

1. Right-click the component and choose *Edit Model*.

   Model definition opens in Model Editor.

2. Modify the parameter values or add new parameter, as required.
3. Save the changes.

Adding Circuit Stimulus

To run analysis on your design, you need to add stimuli defining inputs for the circuit. For this, you can instantiate parts from the Cadence-supplied libraries, *sourcestm and source.* Depending on the part used, stimulus information is added either as part property, or attached as a stimulus file, or specified using Stimulus Editor.

For example, if you use parts `STIM1`, `STIM4`, `STIM8`, or `STIM16` from the *source* library, stimulus information is specified as part property.

Parts, such as `VPWL_F_RE_FOREVER`, `IPWL_F_RE_FOREVER`, `VPWL_F_N_TIMES`, and `IPWL_F_N_TIMES`, use file-based stimulus. The stimulus information specified using PSpice netlist syntax is saved in a file attached to the part.

For parts such as VSTIM and ISTIM, stimulus information is specified using Stimulus Editor.

Using Stimulus Editor

The `VSTIM` and `ISTIM` parts are used to define time-based input signals. Parts such as `DIGSTIMn` are used to specify digital stimulus to a circuit. When using these components, stimulus information is specified using Stimulus Editor.

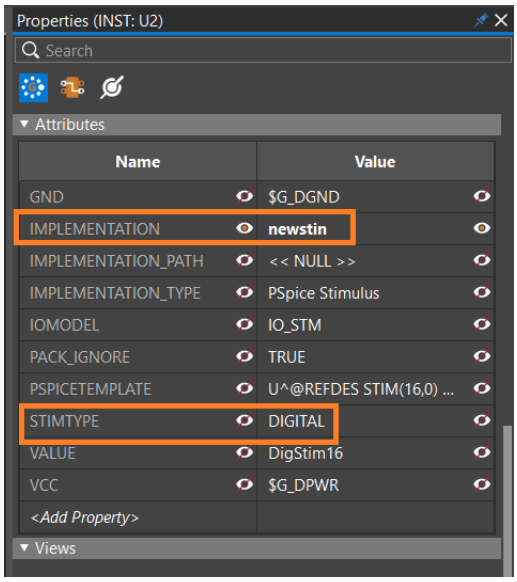
Tip

The value of the *STIMTYPE* property indicates if the stimulus is analog or digital.

To add stimulus to a circuit, do the following steps:

@ 2023 All Rights Reserved.

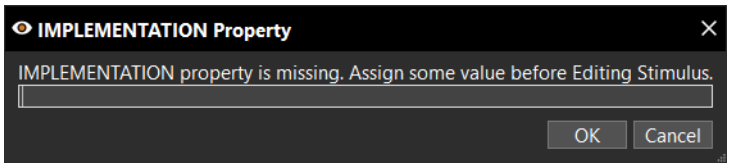
1. Place a stimulus part on the schematic.
2. For the part, specify the value of the *IMPLEMENTATION* property in the *Attributes* section of the *Properties* panel.



This value is used as the stimulus name.

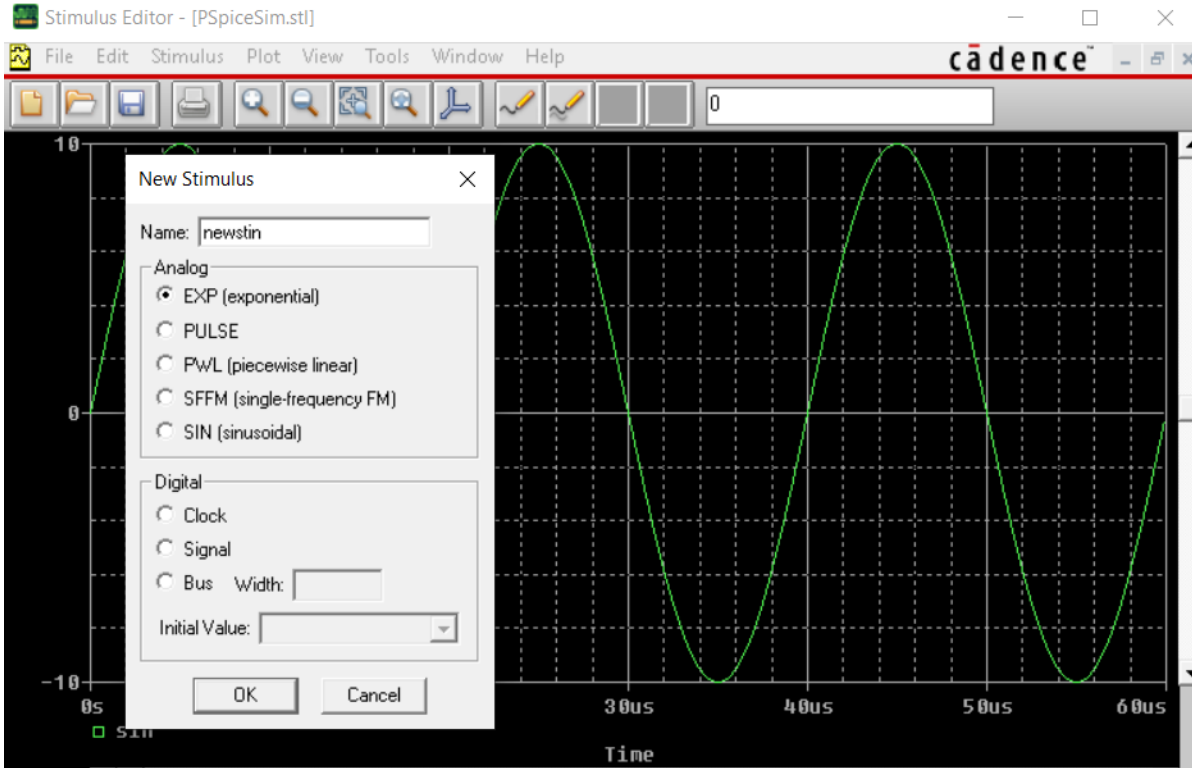
3. Right-click the placed part and choose *Edit Stimulus*.

Note: If the IMPLEMENTATION property is not specified, the following dialog is displayed on selecting *Edit Stimulus*.



@ 2023
All Rights Reserved.

The Stimulus Editor window opens and the *New Stimulus* dialog box is displayed.



4. Select the stimulus type and specify details in the subsequent dialog boxes.

   Depending on the stimulus details you specify, a stimulus waveform is displayed in the Stimulus Editor window.

5. Save the waveform.

   A message box prompts you to update the schematic.

6. Click *Yes*, to update the schematic with the stimulus information.

Creating Custom Digital Power Supplies

Digital power supplies are required if a design contains digital power and ground nodes. To define a custom digital power supply for the schematic, use parts from the `special` library. The following customizable digital power supply parts are available in this library:

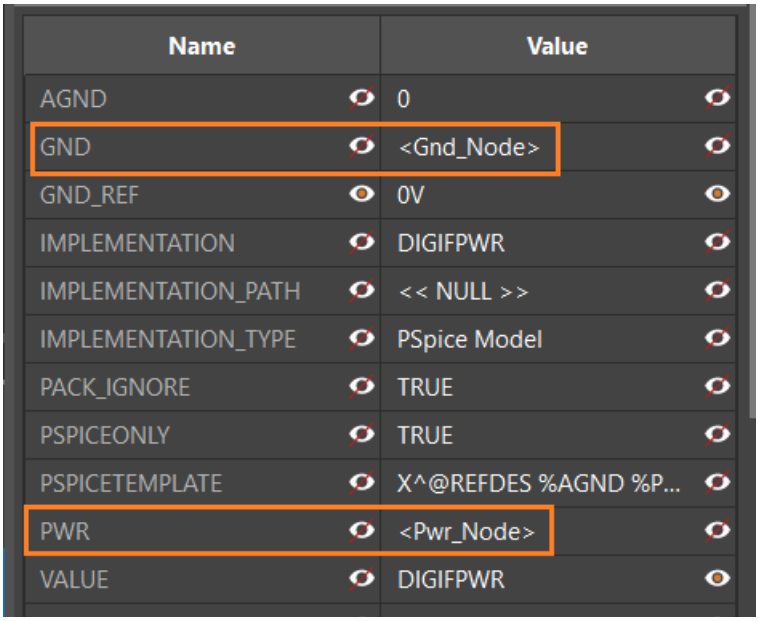
- CD4000_PWR
- DIGIFPWR (TTL power supply)

@ 2023 All Rights Reserved.

- ECL_10K_PWR
- ECL_100K_PWR

To create a custom digital power supply, do the following steps.

1. Place the required power supply part in your design.
2. To specify power and ground voltages, assign values to the PWR and GND component properties, respectively.



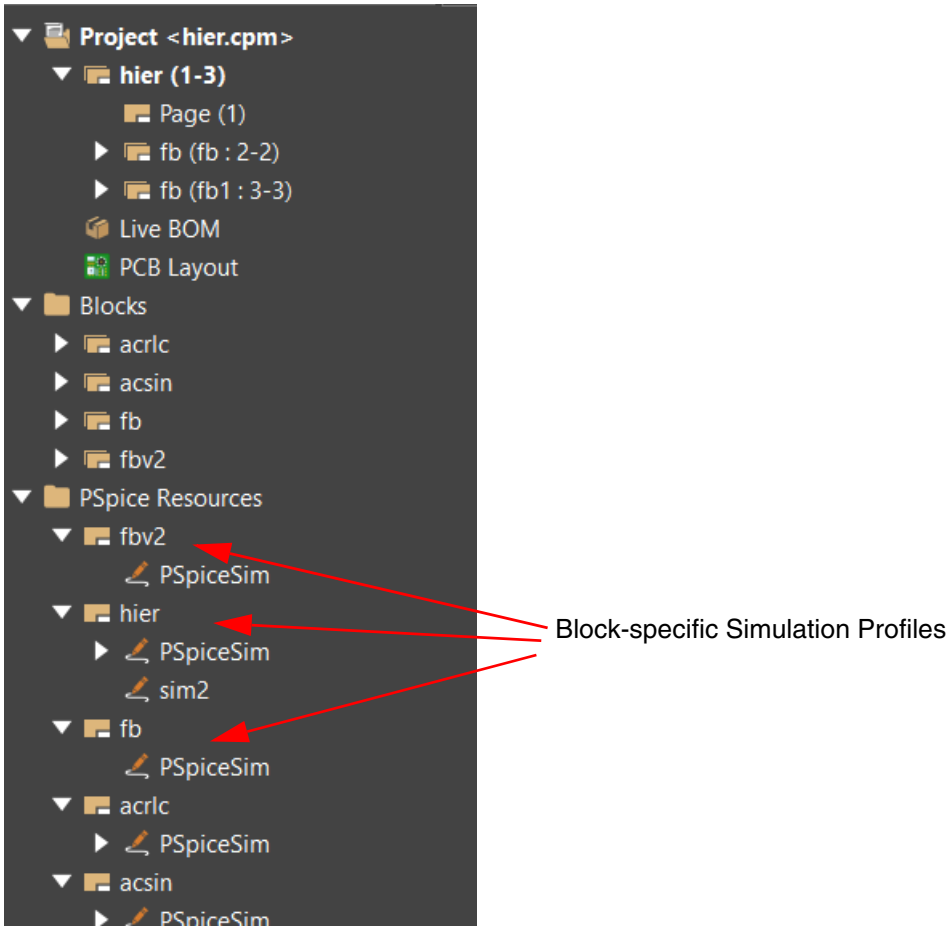
@ 2023
All Rights Reserved.

@ 2023 All Rights Reserved.

3

Simulating Hierarchical Design

The hierarchical designs created in System Capture can be simulated using PSpice. On creating a hierarchical schematic in PSpice-enabled System Capture, a new simulation profile is created for each schematic block. Using the block-specific simulation profile, each block can be simulated independently in PSpice A/D.



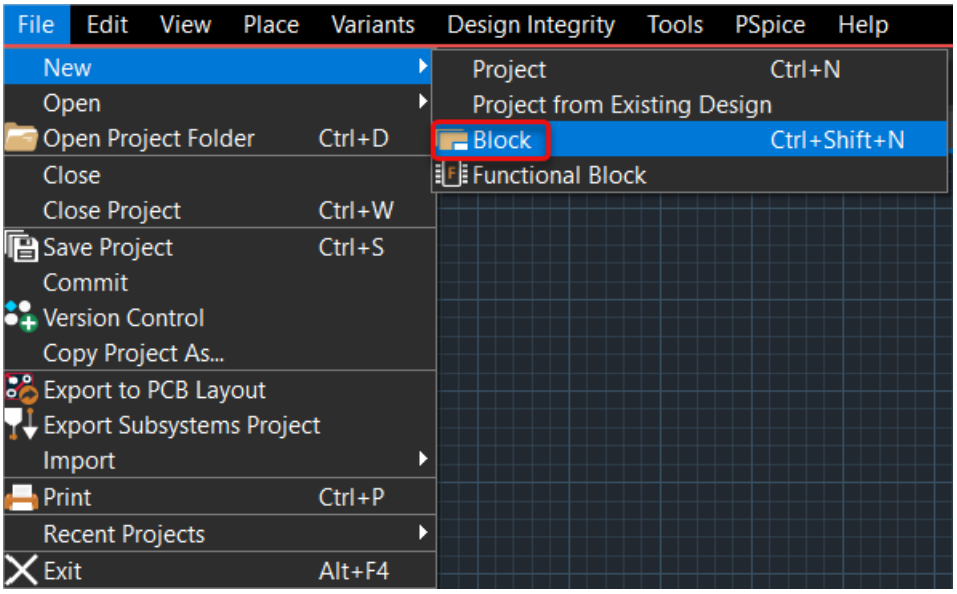
For simulating the complete design, the simulation profile of the root design is used.

@ 2023
All Rights Reserved.

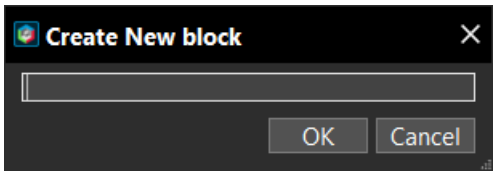
Simulating a Block Schematic

To create and simulate a block in System Capture, do the following steps:

1. Choose *File – New – Block*.

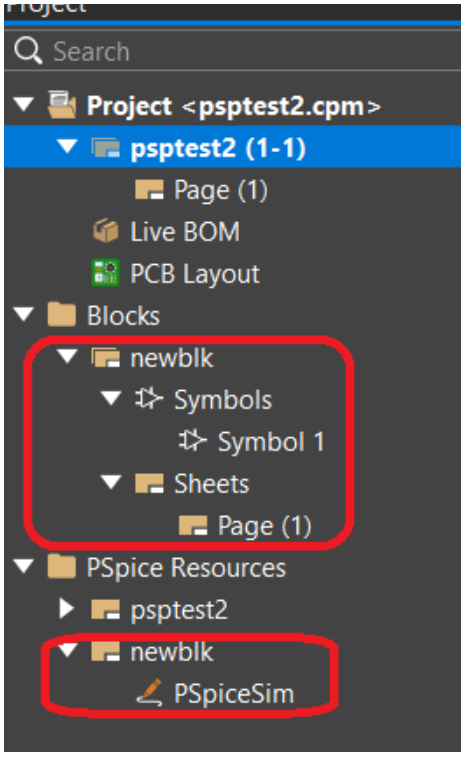


2. In the *Create New block* dialog box, specify the new block name and click *OK*.



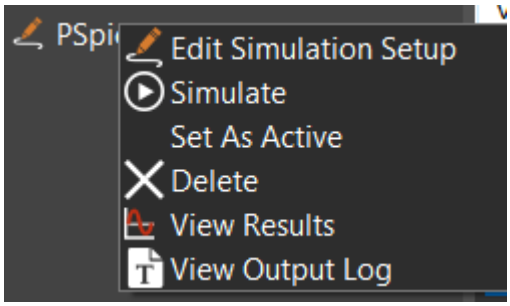
@ 2023
All Rights Reserved.

A new block is created under the *Blocks* node with a blank schematic page and a zero-pin symbol. A default simulation profile is also created and is visible under the *PSpice Resources* node.



Block-specific Simulation

3. Create the block schematic and update the simulation profile as required.
4. To simulate the block, right-click the simulation profile for the block under the PSpice Resources node, and choose *Simulate*.

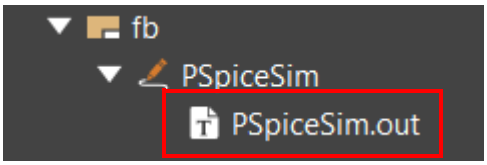


@ 2023
All Rights Reserved.

Tip

If the shortcut menu commands appear disabled, the block symbol might be open for editing. Close the symbol view to enable the block simulation commands. Alternatively, open the block schematic, and click the *Simulate* icon from the PSpice toolbar on the schematic page.

When the simulation completes, the simulation output file for the block is also listed under the *PSpice Resources* node.



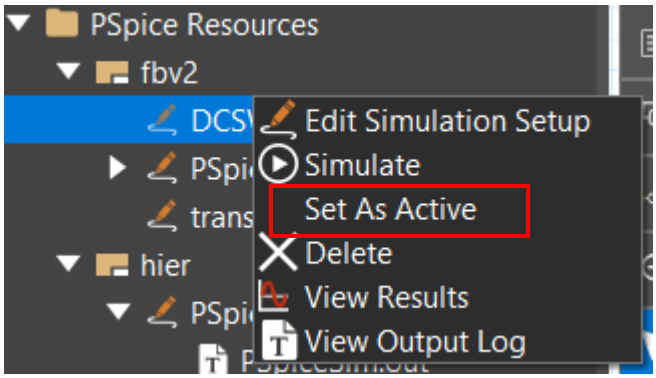
Using Alternate Simulation Profiles

When you create a block, a new simulation profile, *PSpiceSim,* is created by default for each schematic block. This default profile is displayed under the block name in the *PSpice Resources* node, and the PSpice toolbar. When you simulate the design from the *PSpice – Simulate* menu, or by clicking the *Simulate* icon, the default simulation profile is used.

Alternate simulation profiles created for a block are also listed under the block name in the *PSpice Resources* node. To simulate a block design using an alternate profile, use one of the following methods:

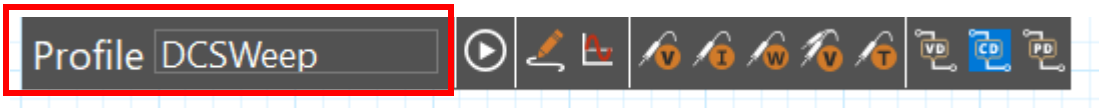
- Right-click the profile name to be used, and choose *Simulate*.
- Modify the default profile, and simulate. For this, do the following:

  a. Right-click the profile name to be used and choose *Set As Active*.



@ 2023
All Rights Reserved.

The active simulation profile name is reflected in the PSpice toolbar for the block schematic.



b. To simulate the block schematic, click the Simulate button ( ) from the PSpice toolbar.

After a profile is set as the active simulation profile, it becomes the default profile for subsequent simulations of that schematic.

Tip

Schematic blocks can be used for creating and simulating different versions of a schematic. To do this, open the new page of a block and copy an existing schematic. You can create variations by adding a new component, replacing an existing component, or by modifying component parameters.

@ 2023
All Rights Reserved.

@ 2023 All Rights Reserved.

4

Running Analysis

With PSpice integration you can run Advanced Analysis tools on your System Capture design. PSpice Advanced Analysis includes Sensitivity analysis, Optimizer runs, Monte Carlo analysis, Smoke analysis, and Parametric Plotter.

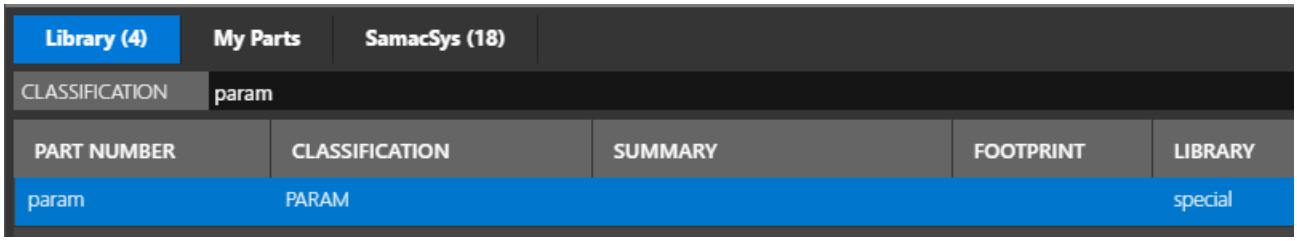
Parametric Sweep

Parametric Sweep is a multi-run simulation analysis where circuit behavior is tested by varying the specified parameter values. Parametric Sweep is available with AC Sweep, DC Sweep, and Transient analysis.

To run a Parametric Sweep analysis, do the following tasks:

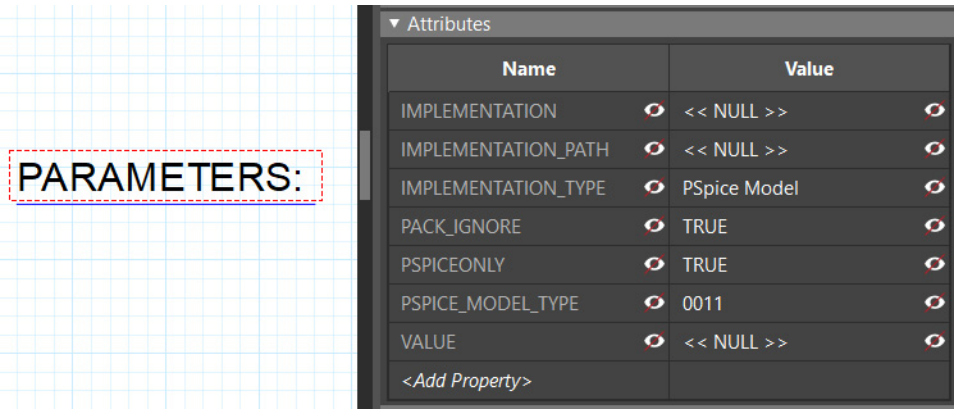
1. Add the component for which parameter values are to be changed.

    a. Using the Unified Search panel, place an instance of the *param* component from the *special* library on the schematic.



@ 2023
All Rights Reserved.

b. Select the instances and in the Attributes section click *<Add Property>*.



c. Add the name and the value of the parameter to be modified.

2. Specify the simulation settings.

a. From the PSpice toolbar, select the *Edit Simulation Setup* icon.

b. Select the *Parametric Sweep* check box.

c. In the Sweep Variable section, select the *Global Parameter* option.

d. In the Parameter name field, specify the parameter name entered in the param component instance (step c of 1).

e. In the Sweep Type section, select the option to indicate how parameter values are to be modified.

f. For the selected Sweep type, specify the required field values.

g. Click OK to save the changes to the simulation profile.

3. Run the simulation.

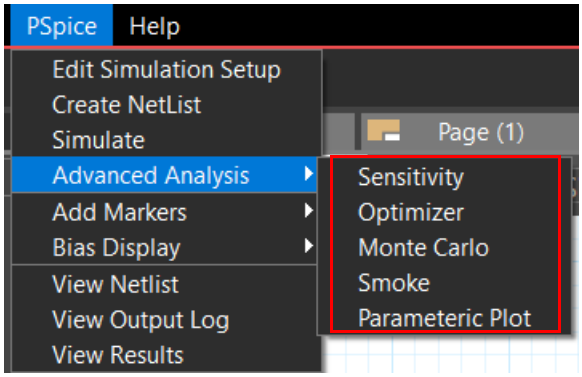
After the simulation run completes, place the required markers on the schematic to view the output waveform.

@ 2023
All Rights Reserved.

Advanced Analysis

Besides the Transient, AC and DC Sweep analysis, you can also run PSpice Advanced Analysis simulations to optimize your designs, improve design reliability and yield, to reduce overall cost. These Advanced Analysis checks examine the circuit performance against specified goals defined through various measurement expressions created for the design.

➡ To run Advanced Analysis checks on a System Capture design, choose *PSpice – Advanced Analysis.*



From the submenu, choose the analysis to be run.

Advance Analysis Options

Option	Function
Sensitivity	Identifies key components that impact the design performance
Optimizer	Calculates optimum component values
Monte Carlo	Predicts production yield
Smoke	Identifies components most likely to fail in the PCB
Parametric Plotter	Records circuit behavior within the tolerance range

Related Topics

- Measurement Expressions

@ 2023
All Rights Reserved.

@ 2023 All Rights Reserved.