

# **Getting Started with Allegro® X System Capture – PSpice A/D**

**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.

Allegro PCB Router 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.ulv.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. vtkQt, © 2000-2005, Matthias Koenig. 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

<u>Simulating a System Capture Design in PSpice A/D</u> .....	1
<u>Component Specifications</u> .....	2
<u>How to Use This Tutorial</u> .....	2
<u>Software Requirements</u> .....	3
<u>Creating a Project</u> .....	3
<u>Running Simulation in PSpice A/D</u> .....	10
<u>Setting Up Simulation Profiles</u> .....	10
<u>Running Simulation</u> .....	11
<u>Placing Markers</u> .....	12
<u>Creating Measurements in PSpice</u> .....	14
<u>Running Parametric Sweep</u> .....	17
<u>Running AC Analysis</u> .....	21

2

<u>Running Advanced Analysis</u> .....	27
<u>Specifying Tolerances</u> .....	27
<u>Running Sensitivity Analysis</u> .....	28
<u>Running Optimizer Analysis</u> .....	31
<u>Running Monte Carlo Analysis</u> .....	35
<u>Running Parametric Plotter</u> .....	38

3

<u>Modifying a Component Model</u> .....	45
<u>Associating PSpice Models</u> .....	45
<u>Adding Stimulus to Circuit</u> .....	50

A

<u>Setting up PSpice Libraries</u> .....	55
--	----

# **Getting Started with Allegro X System Capture - PSpice A/D**

---

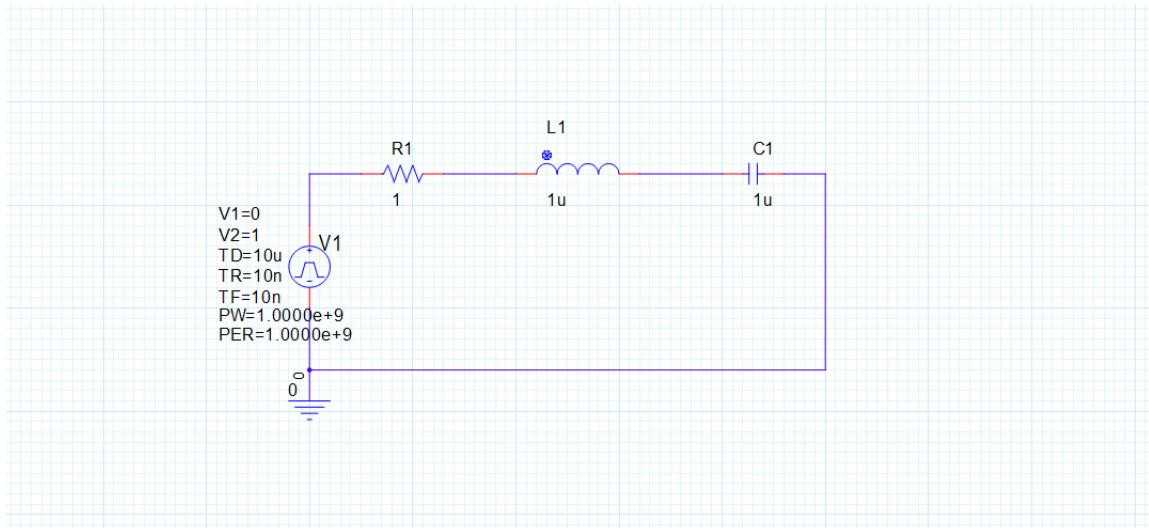
---

# Simulating a System Capture Design in PSpice A/D

---

This tutorial uses a simple RLC circuit to walk you through the steps required to create a schematic in Allegro X System Capture and simulate it in PSpice A/D. The flow covered in this tutorial is compatible only with Windows because PSpice is available only on Windows.

A simple RLC circuit for this tutorial is shown in the following figure. We will create this design in System Capture and simulate it in PSpice A/D to analyze the results.



RLC Circuit for the Tutorial

## Component Specifications

In this tutorial, we will create a design with the specifications listed in the following table:

COMPONENTS	DETAILS
Resistor, R	A resistor with value 1 ohm
Inductor, L	An inductor with value 1 uH
Capacitor, C	A capacitor with value 1 uF
Step Voltage Source	A Step voltage source and the parameters are set as follows: <ul style="list-style-type: none"><li>■ V1 is 0 V</li><li>■ V2 is 1 V</li><li>■ Delay is 10 us</li><li>■ Rise time, <i>TR</i> is 10 ns</li><li>■ AC is 1 V</li><li>■ DC is 0 V</li></ul>
Ground	Place 0 as Ground Symbol

## How to Use This Tutorial

Before you start, create a working directory in your system to store the generated design files.

We will refer to this directory as <your\_work\_area> and for this tutorial, <your\_work\_area> is set to the C : \<SysCap\_New\_Project> folder.

Cadence-supplied library parts are available with the installation.

For more information about setting up library for this tutorial, refer [Appendix A, “Setting up PSpice Libraries”](#).

## Software Requirements

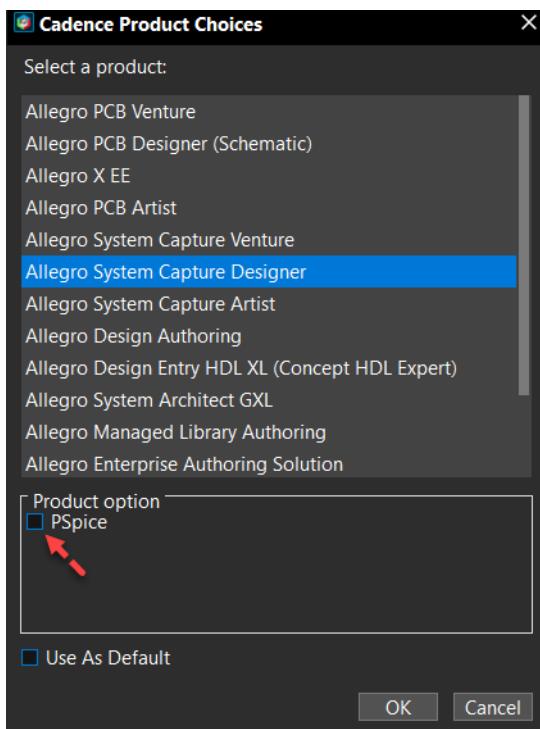
- Allegro X System Capture and PSpice installation – 17.4-2019 HotFix 028 or higher.
- Licenses required:
  - a. Allegro X System Capture – all licenses.
  - b. Allegro PSpice(R) System Designer (PS2200)
  - c. Allegro Enterprise System Design Authoring (SDA200) which includes both the above licenses as a single package.

## Creating a Project

To create an RLC circuit design in System Capture, you will create a new project using Cadence-provided PSpice-compatible parts.

To create a new PSpice-enabled project in System Capture, do the following:

1. Choose the System Capture license and select the product option *PSpice*.
2. Click *OK*



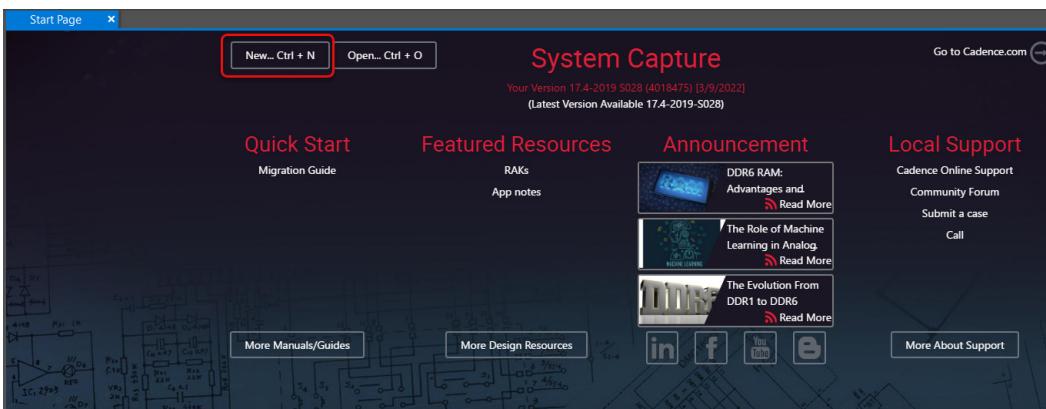
# Getting Started with Allegro X System Capture - PSpice A/D

## Simulating a System Capture Design in PSpice A/D

The Allegro X System Capture main window opens with the Start Page.



### 3. Click *New... Ctrl+N*.



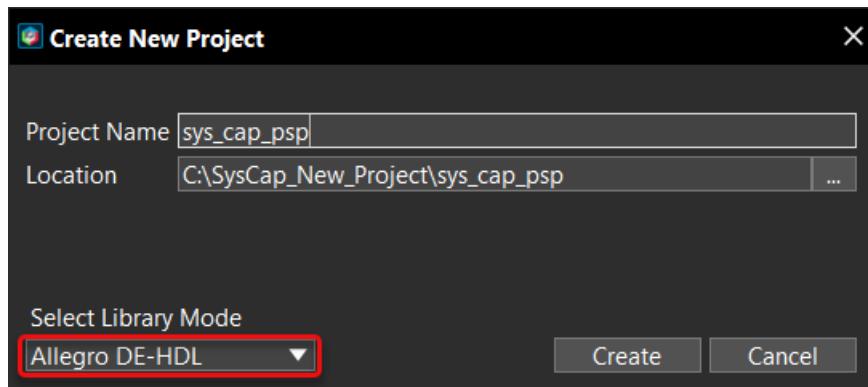
Alternatively, choose *File – New – Project*.

### 4. Specify *sys\_cap\_psp* in the *Project Name* field and navigate to the following location C : \<working\_directory>\sys\_cap\_psp.

## Getting Started with Allegro X System Capture - PSpice A/D

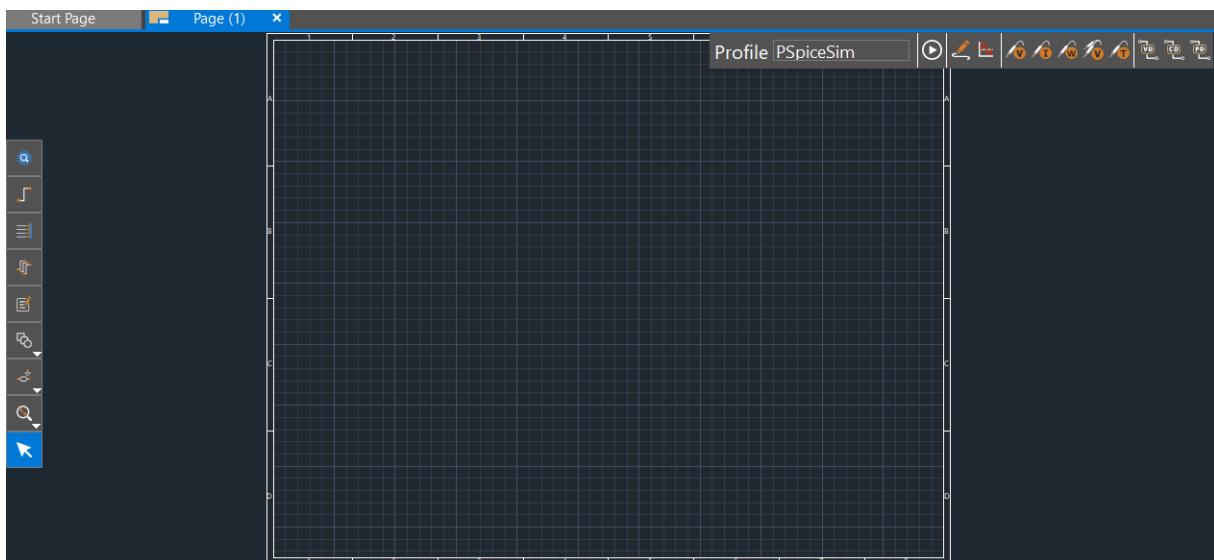
### Simulating a System Capture Design in PSpice A/D

5. Select *Allegro DE-HDL* from the *Select Library Mode* list, and click *Create*.



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

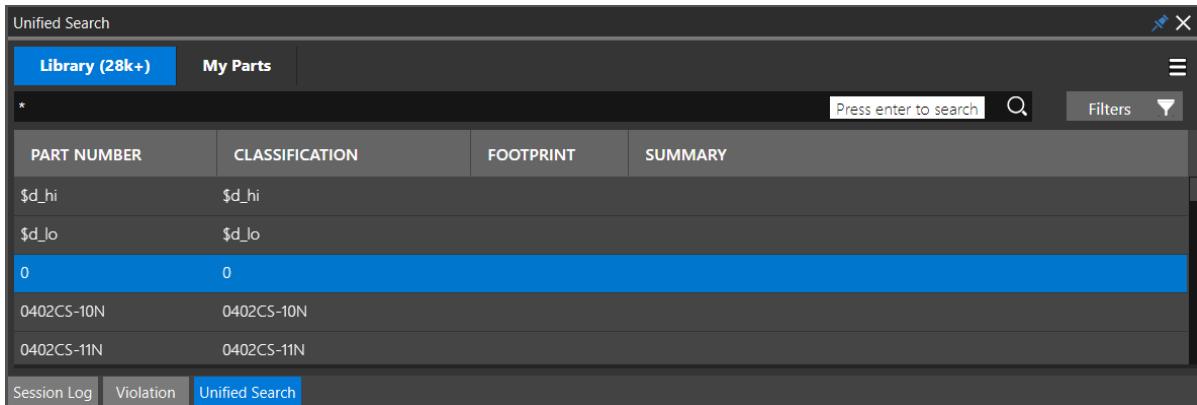
A new project is created, and blank canvas is displayed.



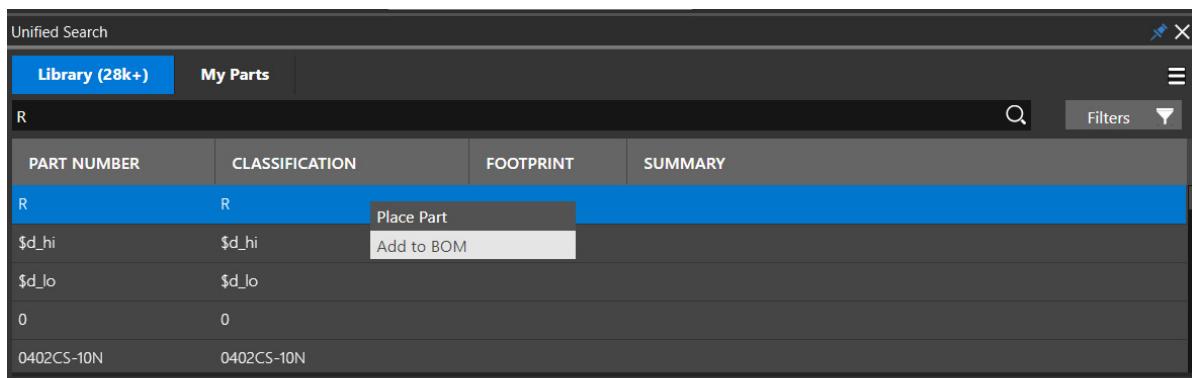
## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

To place the components, use the *Unified Search* window of System Capture. Cadence-supplied library parts are accessible from the *Library* tab in the *Unified Search* window.



6. To create an RLC circuit, place the parts as stated in the [Component Specifications](#) table.
7. In the *Search* bar, type *R* and press the *Enter* key.
8. Right-click the first row with part number *R*, and choose *Place Part* to place component *R1* on the canvas.

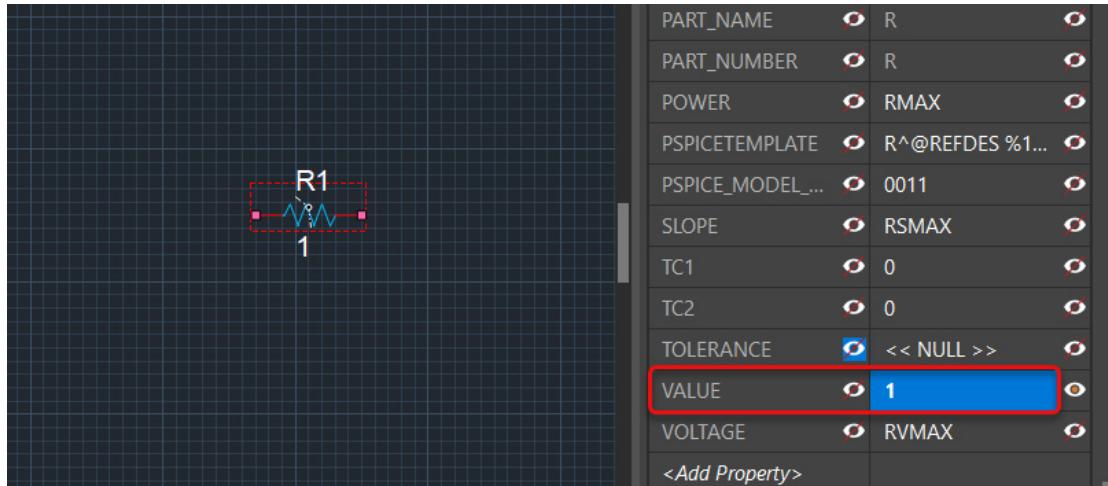


9. Search and place an inductor *L1*, and a capacitor *C1* on the canvas.

## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

10. Select the component  $R1$  and set its value as 1 in the *Attributes* pane.



11. Select  $L1$  and  $C1$  and set the value as 1 uH and 1 uF respectively in the *Attribute* pane.

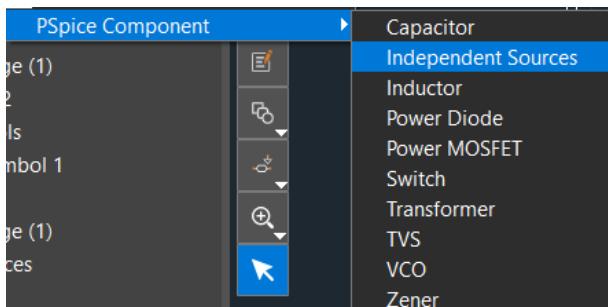


We will use the *PSpice Components* menu to add a step voltage source.

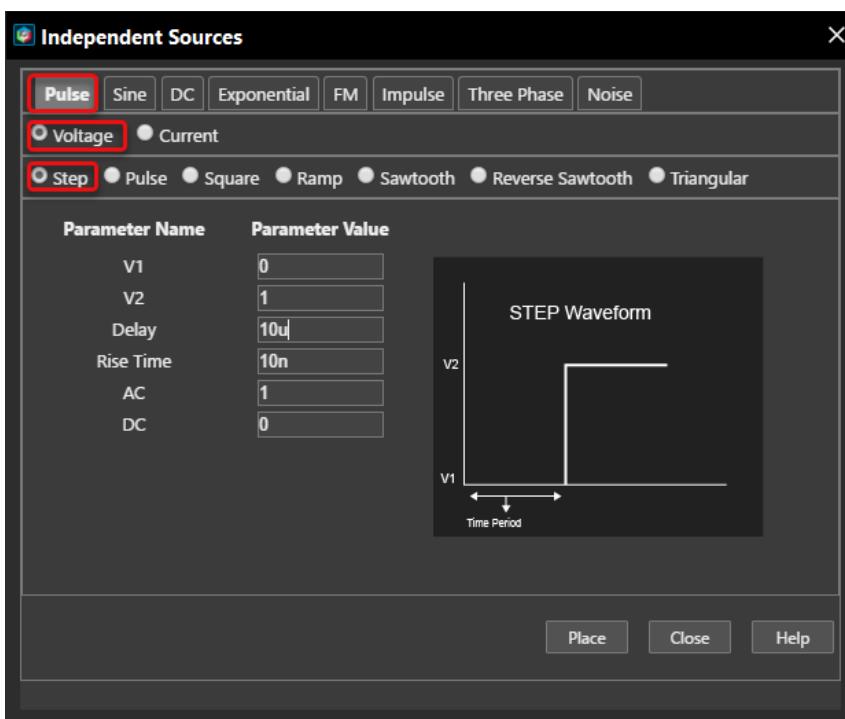
## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

12. To place the voltage source, choose *Place – PSpice Component – Independent Sources*.



13. In the *Independent Sources* dialog box, select the *Pulse* tab and choose *Step* as the voltage source.
14. Specify the electrical parameter values as shown in the Component Specifications table and click *Place* to place the voltage source on the schematic.



## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

---

15. To connect the components, select the *Draw Wire* icon from the functional tool bar and click the connecting end point at the input of the circuit.



Alternatively, use the shortcut key `W` to draw the wire.

Follow these steps to create the circuit:

- a. Join the positive terminal of the voltage source to pin1 of the input resistor  $R1$ .
- b. Connect pin 2 of  $R1$  to pin1 of inductor  $L$ .
- c. Connect pin 2 of  $L$  to pin1 of the output capacitor  $C$ .

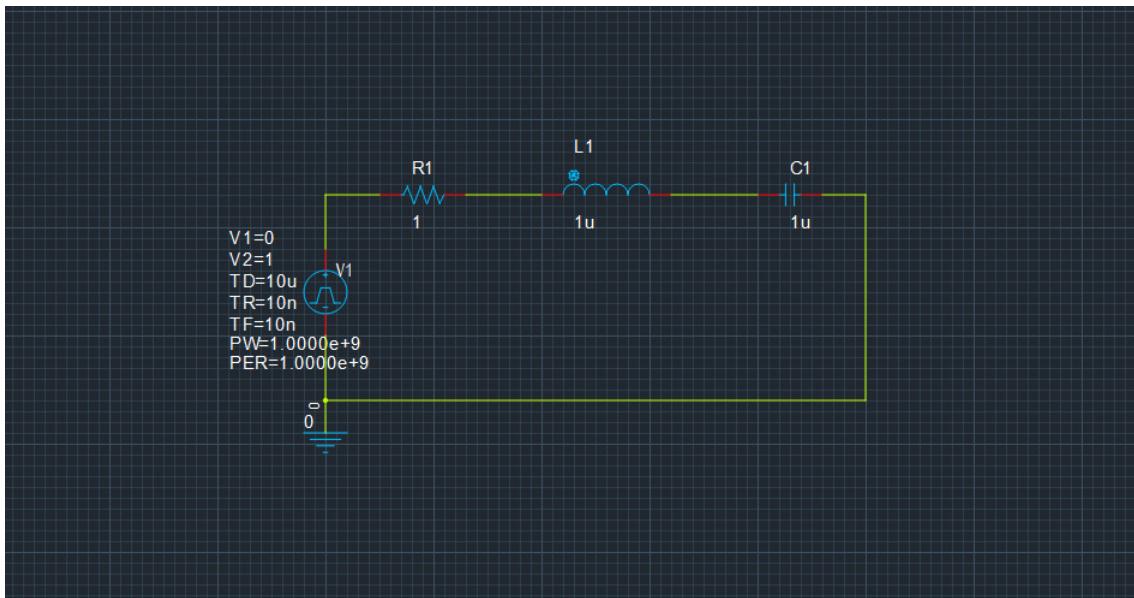
To simulate a design in PSpice, you need a ground symbol added to the design.

16. To place a ground symbol, search for `0` in the *Unified Search* window and place the ground symbol.

## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

Complete the circuit as shown in the following figure:



17. Save the design.

## Running Simulation in PSpice A/D

To simulate the design in PSpice A/D, you need to set up the simulation profile and run the simulation.

### Setting Up Simulation Profiles

By default, a simulation profile *PSpiceSim* is available for all the designs. In this tutorial, we will modify the existing simulation profile for our design.

To modify the simulation profile, do the following:

1. Click the *Edit Simulation Setup* icon on the PSpice Simulation toolbar.

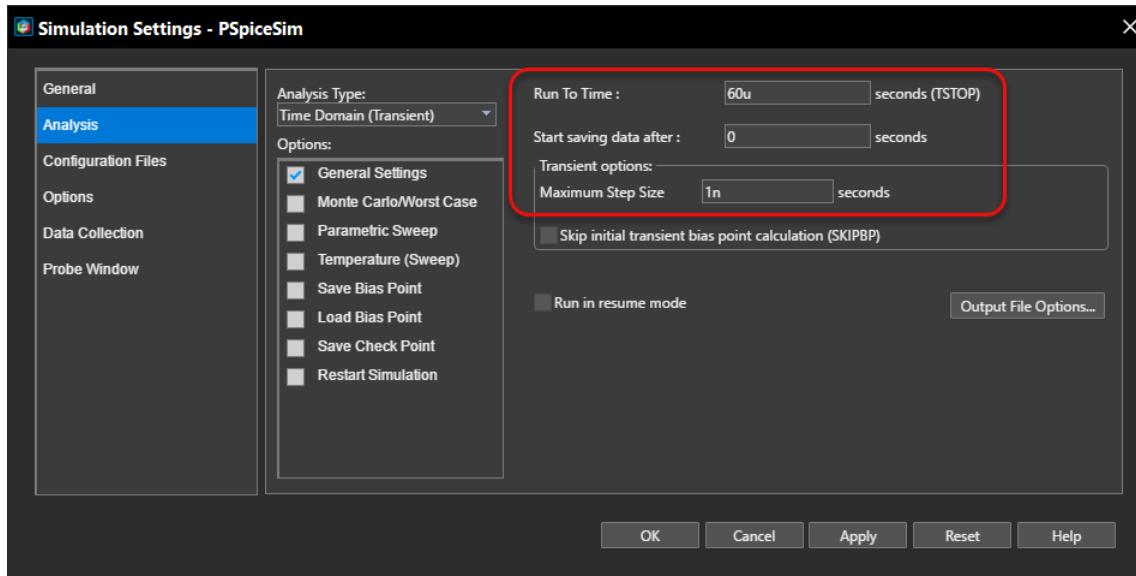


2. Specify 60 us in the *Run to Time* field and, 1 ns in the *Maximum Step Size* field.

## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

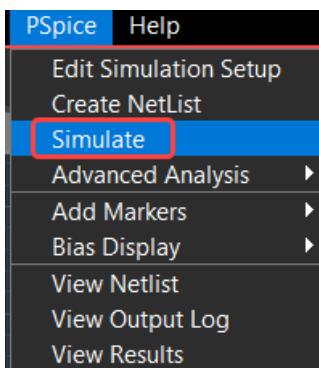
3. Click *OK* to update the profile.



## Running Simulation

After setting up or updating the profile, you will run the simulation.

- To run the simulation, choose *PSpice – Simulate*.



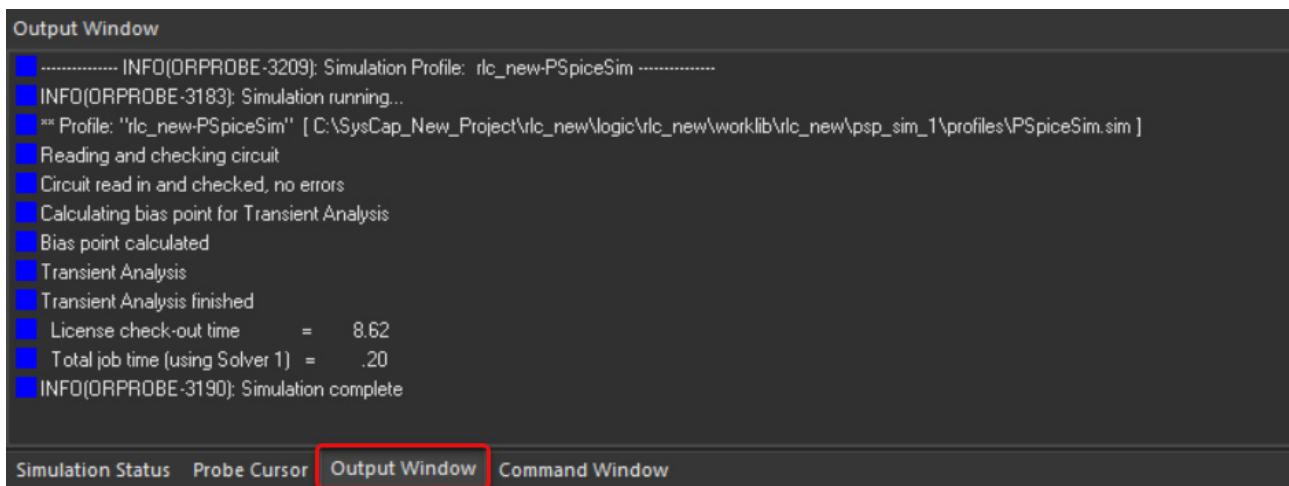
Alternatively, you can click the *Run Simulation* icon in the PSpice Simulation toolbar.



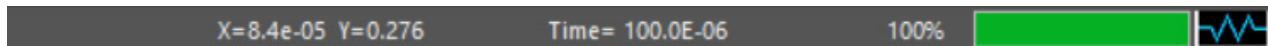
## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

After the simulation starts, the PSpice A/D application is displayed with the *PSpice Probe* window. PSpice A/D calculates bias point voltages and currents in the background, and the simulation status is displayed in *Output Window*.



A progress bar at the lower-right corner indicates that the simulation is successfully completed.



After the completion of the simulation, place markers to display the traces in the output waveform and analyze the simulation result.

### Placing Markers

To view the output in the *PSpice Simulation* window, place the marker in your design.

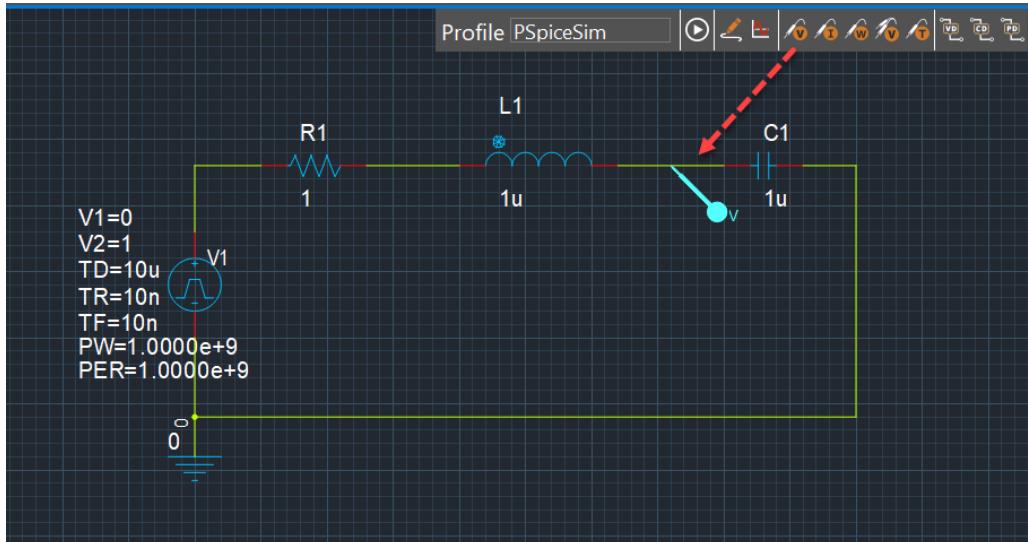
To place a voltage marker across L1 in your design, do the following:

1. Select the *Voltage Marker* icon in the PSpice Simulation toolbar to analyze the output voltage taken across L1 in the design.

## Getting Started with Allegro X System Capture - PSpice A/D

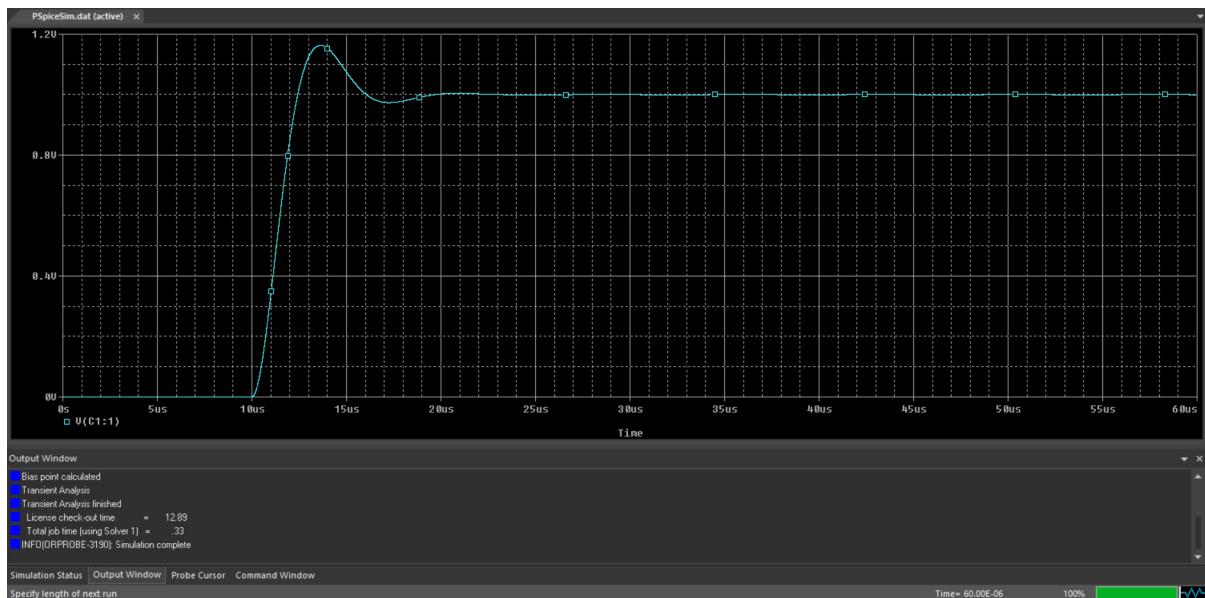
### Simulating a System Capture Design in PSpice A/D

2. Click to place the marker between  $L_1$  and  $C_1$ .



3. Press `Esc` to exit the marker mode.

The output waveform is displayed in the *PSpice Probe* window of PSpice A/D.



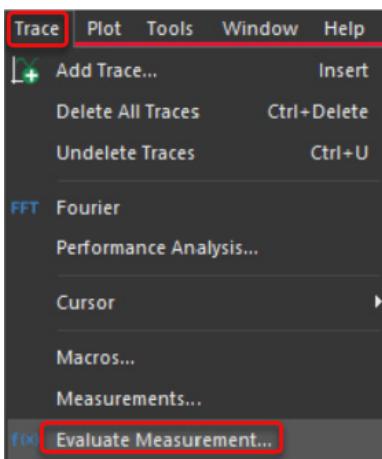
To analyze the output waveform, we need to create certain measurements.

## Creating Measurements in PSpice

To evaluate the characteristics of the waveform generated using PSpice Simulator, create measurements in PSpice.

To create measurements in the active simulation profile *PSpiceSim*, do the following:

1. In the *PSpice* window, choose *Trace – Evaluate Measurement*.



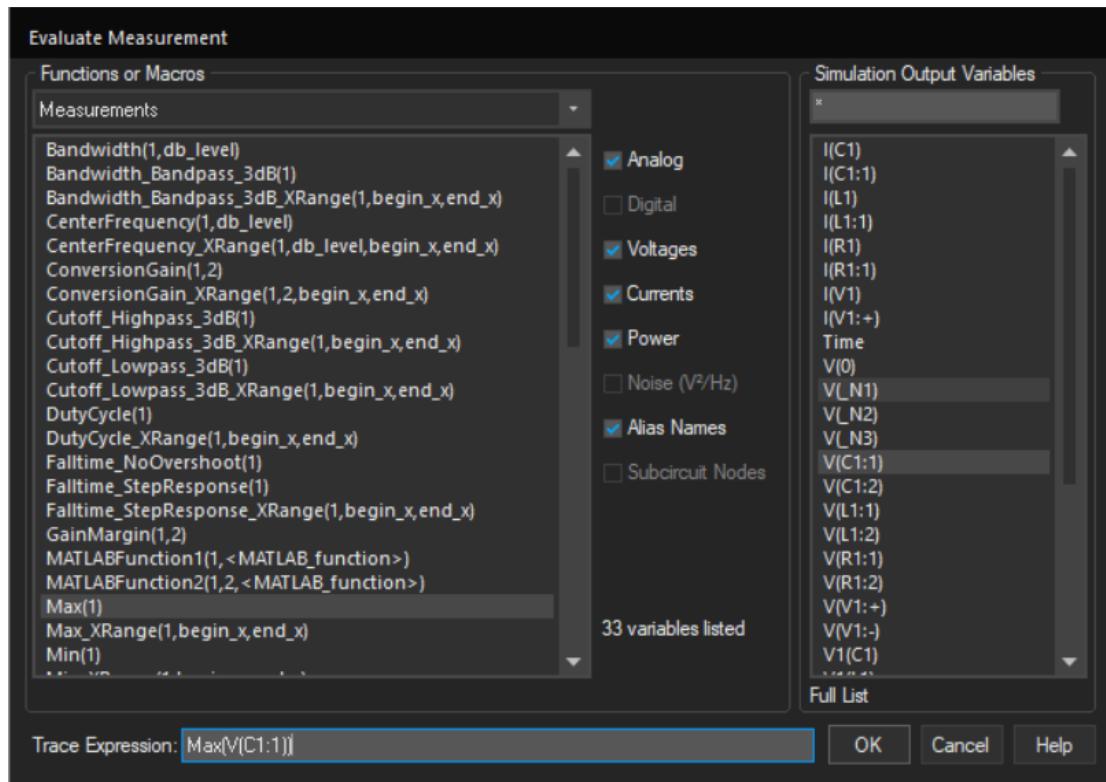
The *Evaluate Measurement* dialog box is displayed.

2. Choose *Function* as *Max(1)* and *Simulation Output Variable* as *V(C1:1)* to trace the maximum voltage of capacitor C1.

## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

- Click **OK** to list the new measurement in the *Measurement Results* table.

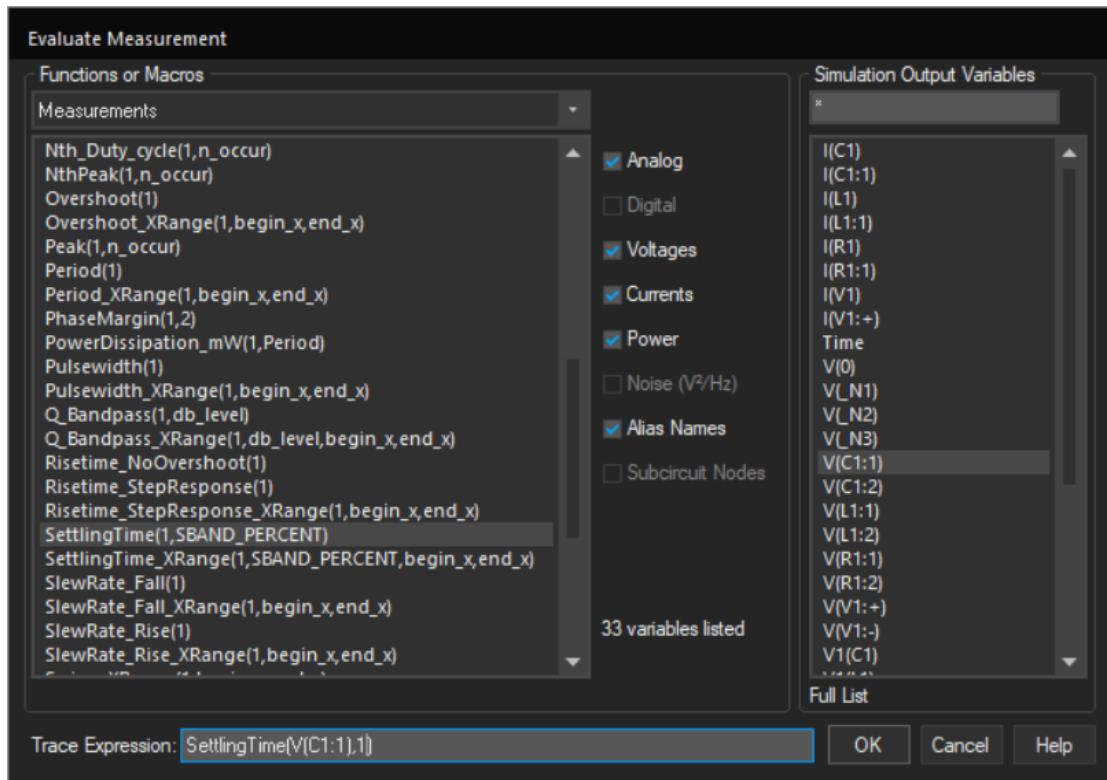


Let us add another measurement to trace the settling time of V(C1).

## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

4. To add another measurement, choose *Function* as *SettlingTime(1,SBAND\_PERCENT)* and *Simulation Output Variable* as *V(C1:1),1*.



The new measurements are displayed with their corresponding values in the *PSpice Simulation* window.

Measurement Results			
Evaluate	Measurement	Value	
<input checked="" type="checkbox"/>	Max(V(C1:1))	1.16303	
<input checked="" type="checkbox"/>	SettlingTime(V(C1:1),1)	18.78557u	Click here to evaluate all

The measurement result shows that the maximum value of  $V(C1:1)$  is  $1.16303\text{ V}$  and the waveform settles at  $1\text{ V}$  after  $18.785\text{ }\mu\text{s}$  within  $1\%$  variation.

## Running Parametric Sweep

To test the circuit response for a changing value of circuit resistance, we will run parametric sweep analysis.

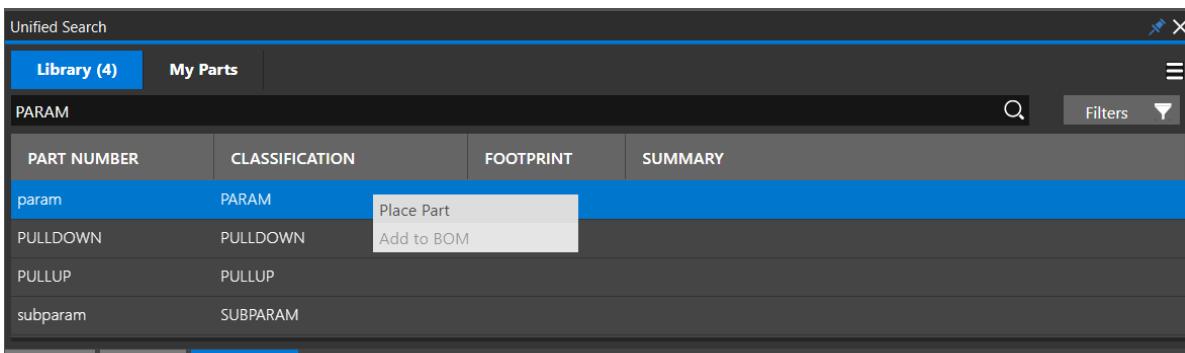
Before you run a parametric sweep, define global parameters with a default value, and use them to sweep for different values for the design in a single simulation.

To define global parameters, we will use the PARAM component. These parameters can then be defined as variable for parametric sweep in simulation settings.

The PARAM component is visible on the schematic and the component values can be edited from the schematic itself.

To run a Parametric sweep simulation on your schematic design, do the following:

1. Search for *param* in the *Unified Search* window.
2. Right-click param and choose *Place Part* to place the PARAM component.



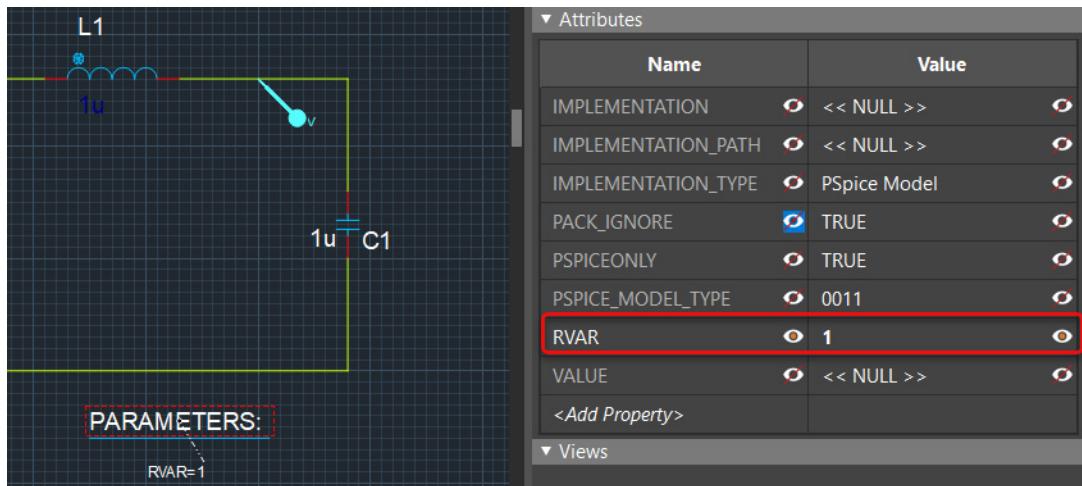
3. Select *param* and click <Add Property> in the *Attributes* pane.

To sweep the value of the resistor, let us set the value of R1 to {RVAR}. The value of RVAR has to be specified within {}.

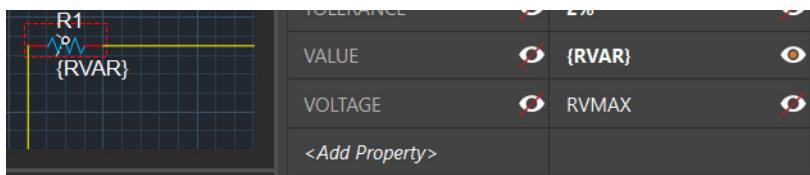
## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

4. Select the PARAMETER to specify  $RVAR$  as the name of the property and  $1$  as its value.

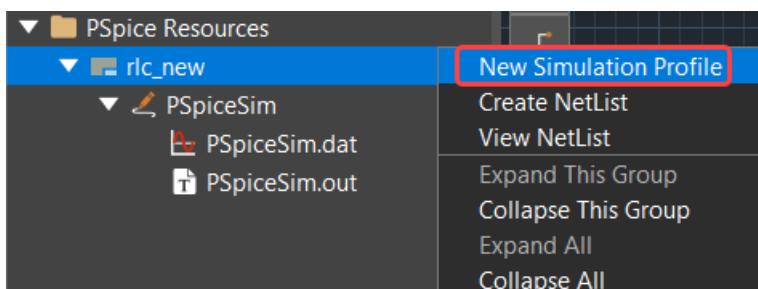


5. Click the resistor  $R1$  and specify  $\{RVAR\}$  as its value in the *Attributes* pane.



Next, modify the simulation profile to specify simulation parameters. In this tutorial, we will use a new stimulation profile to specify the parameters and run parametric sweep.

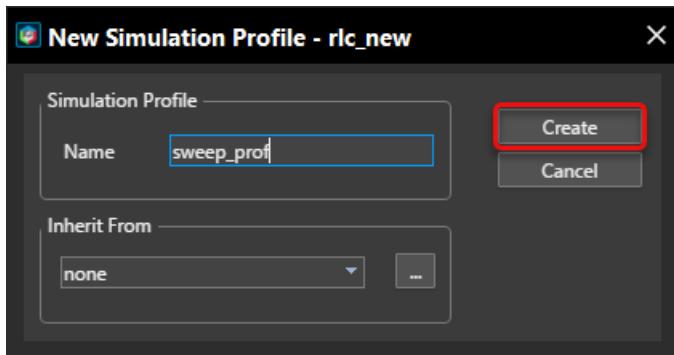
6. In the *Project Explorer* panel under *PSpice Resources*, right-click the block name and choose *New Simulation Profile*.



## Getting Started with Allegro X System Capture - PSpice A/D

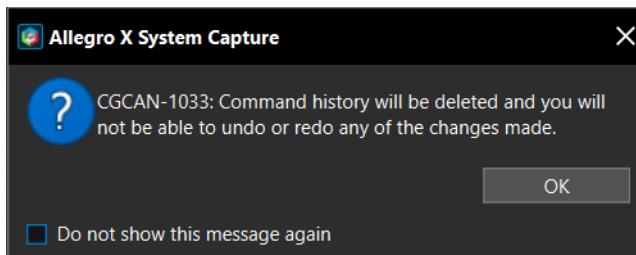
### Simulating a System Capture Design in PSpice A/D

7. Specify the profile name as *sweep\_prof* and click the *Create* button.



A message prompts for your approval to delete the command history.

8. Click *OK* to proceed.



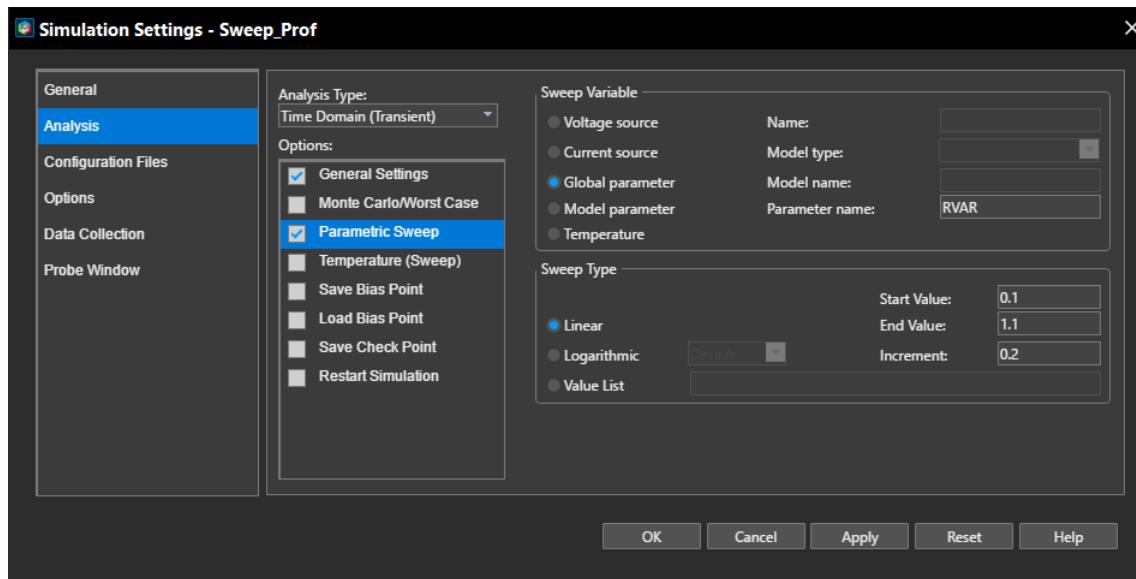
The *Simulation Settings* dialog box is displayed.

9. Select the *Parametric Sweep* check box.
10. In the *Sweep Variable* section, select the *Global Parameter* option and specify the parameter name as *RVAR* in the *Parameter name* field.
11. In the *Sweep Type* section, select the option *Linear* and specify the *Start Value* as *0.1*, *End Value* as *1.1*, and *Increment* as *0.2* in the respective fields.
12. In *General Settings*, specify *60 us* in the *Run to Time* field and, *1 ns* in the *Maximum Step Size* field.

## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

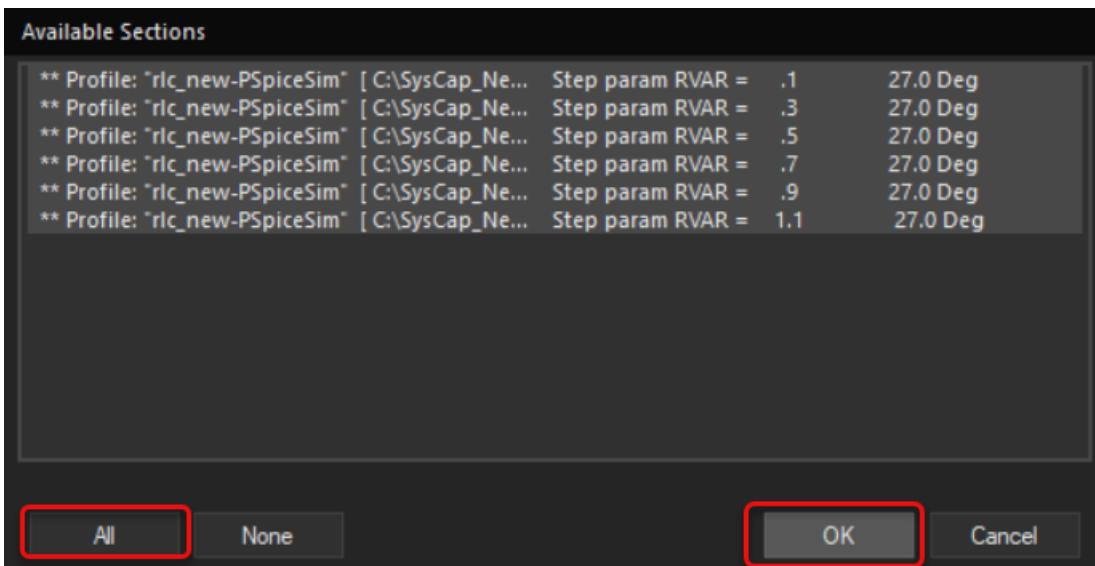
13. Click *OK* to save the changes to the simulation profile.



14. Run the simulation.

A selection window is displayed in PSpice where the simulation runs for all the varied values of R1 are listed. You can click any specific simulation run to view the corresponding output waveform.

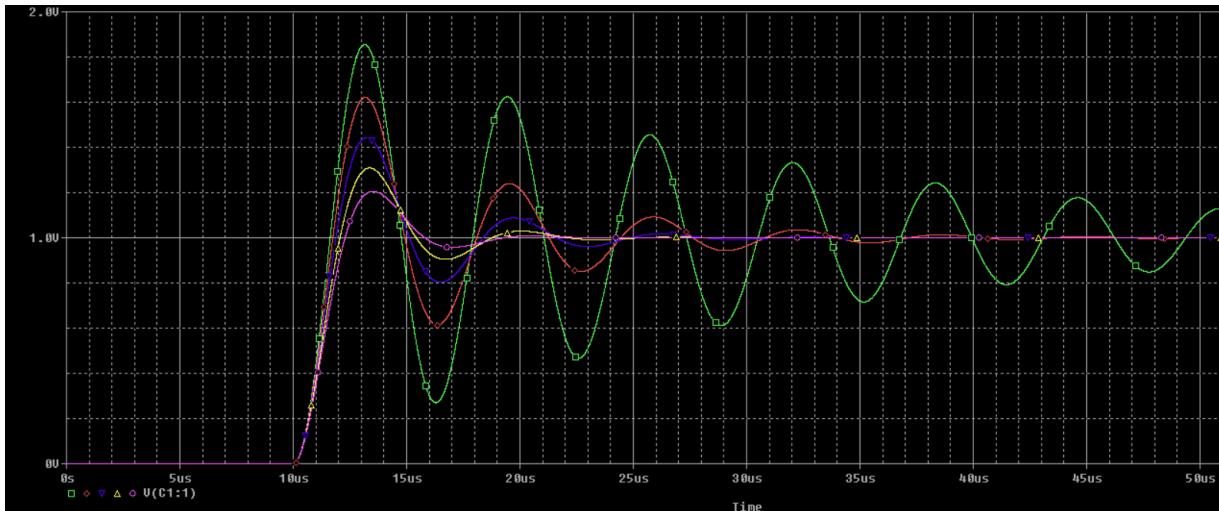
15. To view the waveforms of all the simulation runs, click *All* and then click *OK*.



## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

As we selected all the simulation runs, the resultant waveform is displayed in PSpice A/D after the simulation is completed for all the varied values of R1.

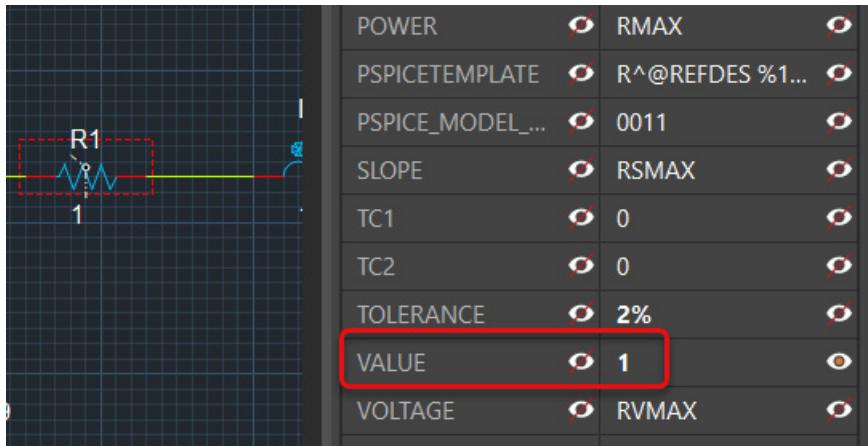


The result shows the response when parameter R1 varies from 0.1 to 1.1.

## Running AC Analysis

To run AC analysis on the design, we will create a new simulation profile and specify the following simulation settings.

1. Before you start, select the resistor *R1* in the System Capture design and change its value to 1.



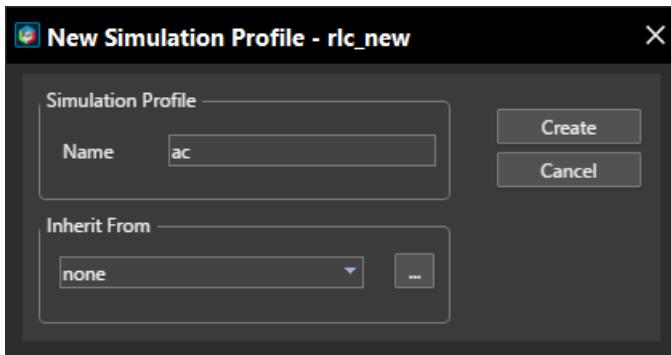
Let us create a new simulation profile and perform AC analysis.

## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

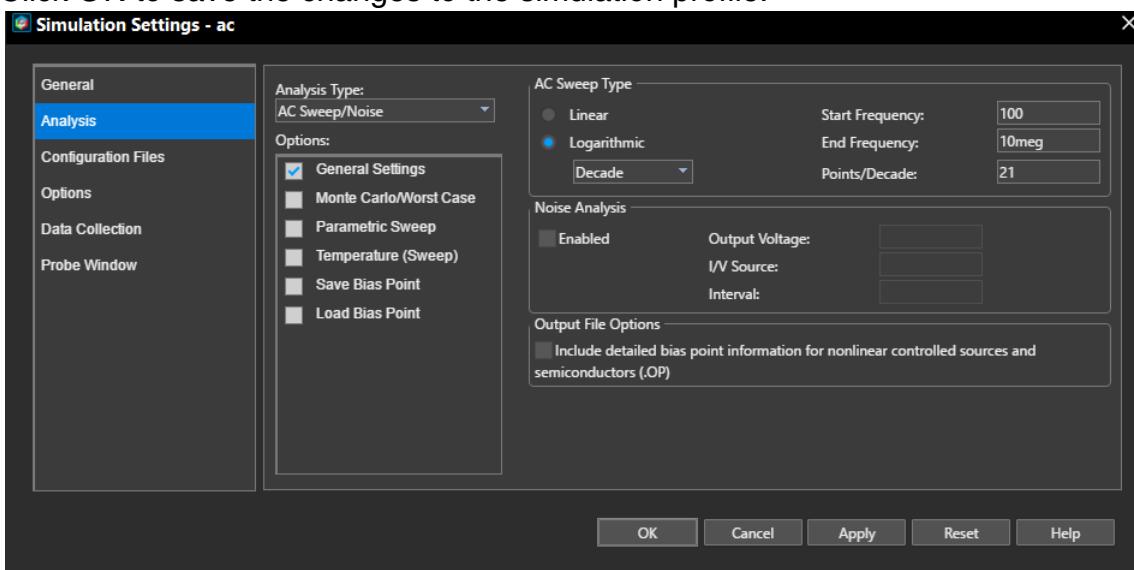
To create a new simulation profile for AC analysis, do the following:

1. In the *Project Explorer* panel under *PSpice Resources*, right-click the block name and choose New Simulation Profile.
2. Specify the profile name as *ac* and click the *Create* button.



The *Simulation Settings* dialog box is displayed.

3. Select the *Analysis Type* as *AC Sweep/Noise*.
4. In the *AC Sweep/Noise Type* section, select the option *Logarithmic – Decade* and specify the *Start Frequency* as 100, *End Frequency* as 10 meg, and *Points/Decade* as 21 in the respective fields.
5. Click *OK* to save the changes to the simulation profile.

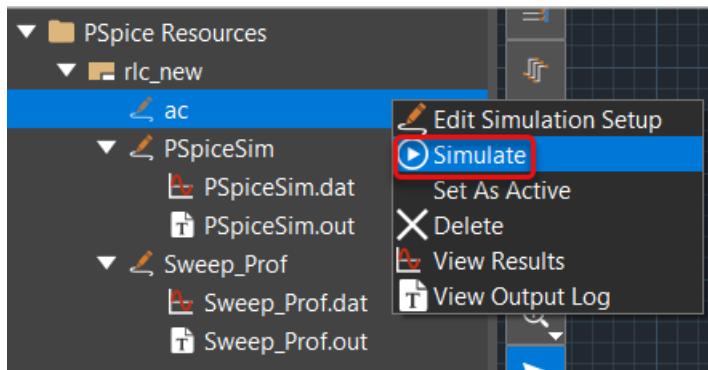


6. Click the *Simulation icon* in the PSpice Simulation toolbar to run the *ac* simulation.

## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

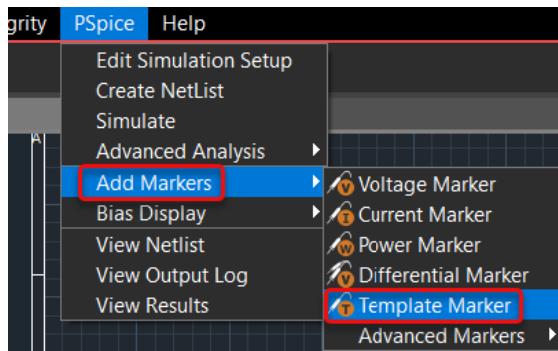
Alternatively, you can right-click *ac* profile and click *Simulate* to run the simulation.



To view the trace of the resultant waveform, let us plot Bode plot using template markers across C1 and see the output waveform.

To place a template marker, do the following:

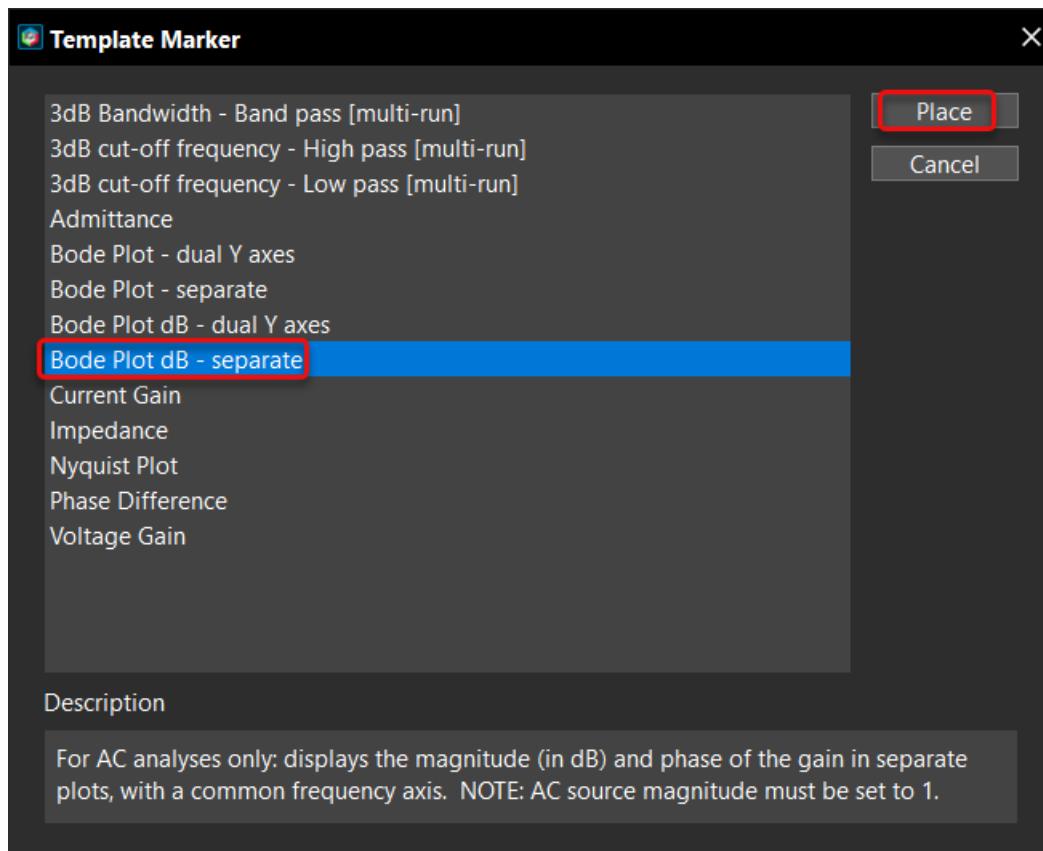
7. In the System Capture, choose *PSpice – Add Markers – Template Marker*.



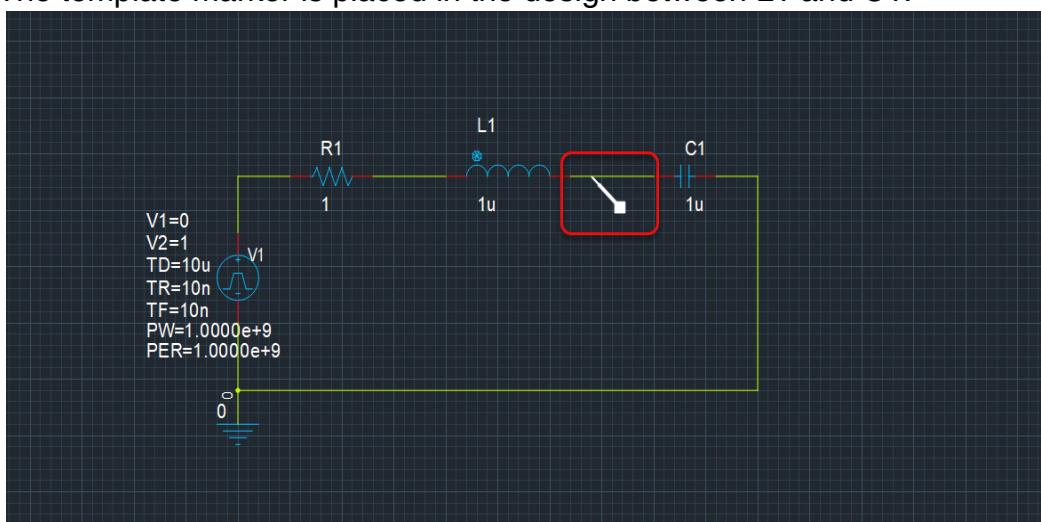
## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

8. In the *Template Marker* dialog box, select *Bode Plot db – separate* and click *Place*.



The template marker is placed in the design between L1 and C1.

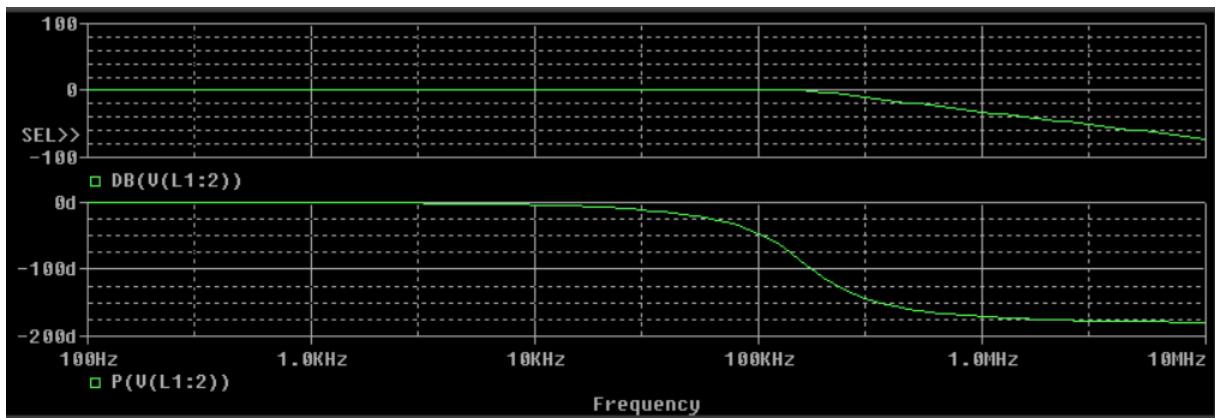


9. Run the simulation and view the output in PSpice A/D.

## Getting Started with Allegro X System Capture - PSpice A/D

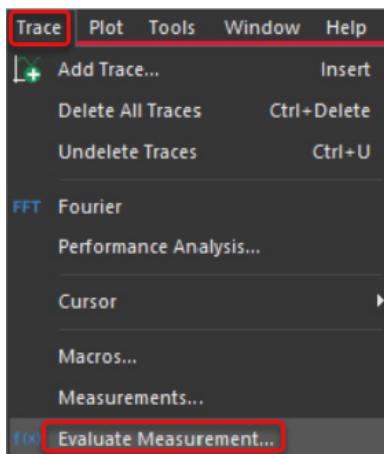
### Simulating a System Capture Design in PSpice A/D

The Bode plot is displayed in the *PSpice Probe* window.



Earlier we added two measurements for transient analysis in the *PSpiceSim* profile. We will add a new measurement for the AC analysis in the active simulation profile *ac*.

10. In the *PSpice Probe* window, choose *Trace – Evaluate Measurement*.

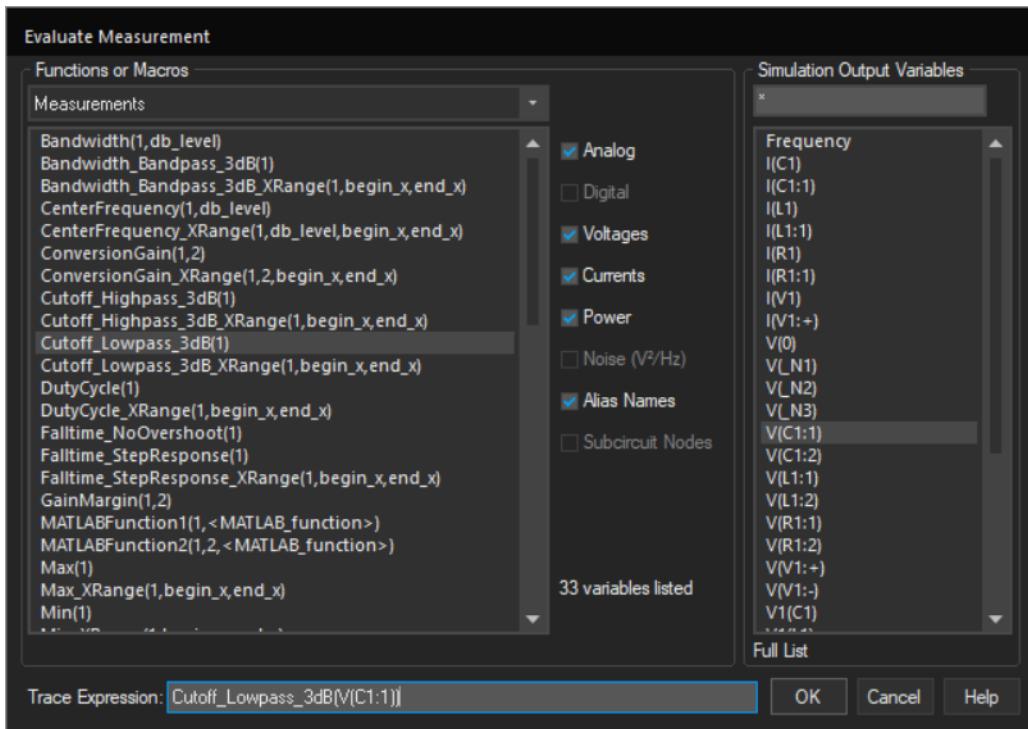


The *Evaluate Measurement* dialog box is displayed.

## Getting Started with Allegro X System Capture - PSpice A/D

### Simulating a System Capture Design in PSpice A/D

11. Choose Function as *Cutoff\_Lowpass\_3dB* and Simulation Output Variable as *V(C1:1)* and click *OK*.



A new measurement is displayed with its corresponding value in the *PSpice Simulation* window.

Measurement Results		
Evaluate	Measurement	Value
<input checked="" type="checkbox"/>	Cutoff_Lowpass_3dB(V(C1:1))	185.59714k
Click here to evaluate a new measurement...		

## Running Advanced Analysis

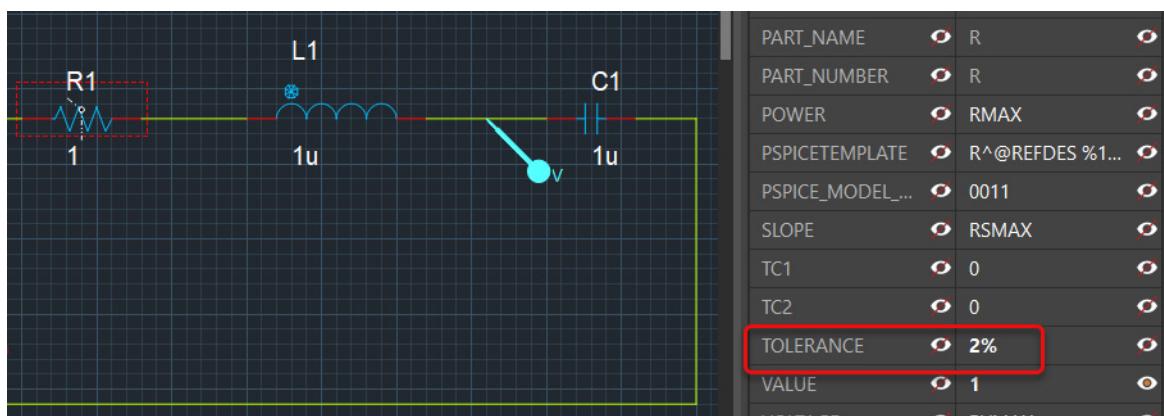
PSpice Advanced Analysis includes Sensitivity analysis, Optimizer, Monte Carlo analysis, Smoke analysis, and Parametric Plotter. After you have verified the design by running the PSpice simulation, use PSpice Advance Analysis to maximize design performance, yield, and reliability.

### Specifying Tolerances

To make the design ready for advance analysis, you need to specify tolerances for the components  $R1$ ,  $L1$ , and  $C1$  respectively.

To set tolerance value for the resistor  $R1$ , do the following:

1. In System Capture, select  $R1$ .
2. In the *Attributes* pane, specify the tolerance value for  $R1$  as 2%.

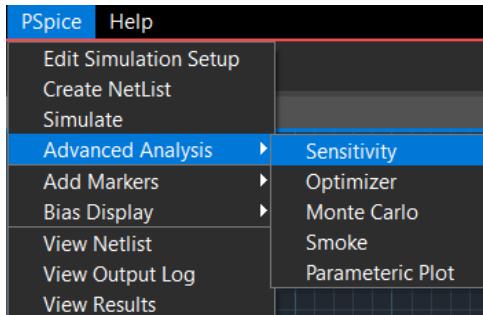


3. Select  $L1$  and  $C1$  and set the tolerance value as 1% for both.
4. After setting tolerances, run the simulation again for both transient and AC simulation profiles.

# Running Sensitivity Analysis

Sensitivity identifies the parameters that are critical to the measurement goals. Let us now perform sensitivity analysis.

- To perform sensitivity analysis, choose *PSpice – Advanced Analysis – Sensitivity*.



PSpice Advanced Analysis is launched with the sensitivity tool activated where components with the parameter values are listed.

# Importing Measurements

To analyze the sensitivity of components against the measurements created in PSpice, we will now import the added measurements.

To import the measurements, do the following:

1. In the *Specifications* table, click the row with the text *Click here to import a measurement created within PSpice*.

The *Import Measurement(s)* dialog box is displayed.

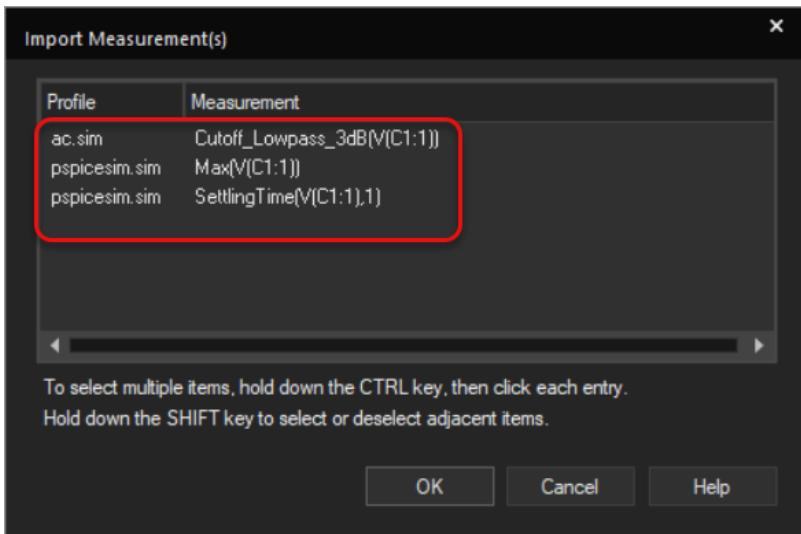
- 2.** Select measurements listed in the *Import Measurement(s)* window:

- ❑  $Cutoff\_Lowpass\_3dB(V(C1:1))$
  - ❑  $Max(V(C1:1))$
  - ❑  $Settling\ Time(V(C1:1), 1)$

## Getting Started with Allegro X System Capture - PSpice A/D

### Running Advanced Analysis

#### 3. Click OK.



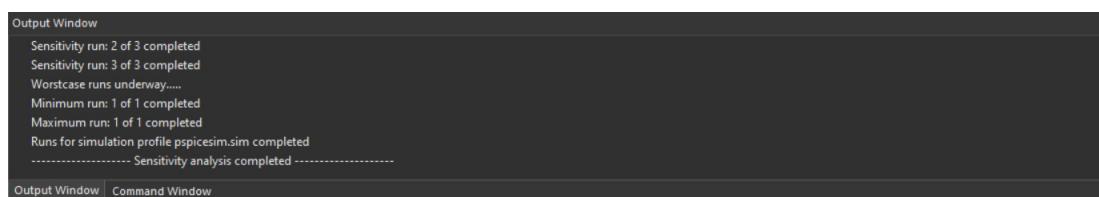
The imported measurements are listed in the *Specifications* table.

	On/Off	Profile	Measurement	Original	Min	Max	Specifications
▶	✓	ac.sim	Cutoff_Lowpass_3...				
▶	✓	pspicesim.sim	Max(V(C1:1))				
▶	✓	pspicesim.sim	SettlingTime(V(C1:1),1)				
							Click here to import a measurement created within PSpice...

#### 4. Click the Start icon on the toolbar to run Sensitivity analysis.



The sensitivity analysis starts. You can check the status in *Output Window*.



The result of the selected measurement is displayed in the *Parameter* table. This table lists all the components for which tolerance is defined.

# Getting Started with Allegro X System Capture - PSpice A/D

## Running Advanced Analysis

The result shows L1 and C1 as sensitive components.

Sensitivity Component Filter = [ ]						
	Component	Parameter	Original	@Min	@Max	Abs Sensitivity
▶	C1	VALUE	1u	1.01...	990n	-196.6664k
◀	L1	VALUE	1u	990n	1.01...	196.8274k
◀	R1	VALUE	1	1.0200	980m	-392.0479m

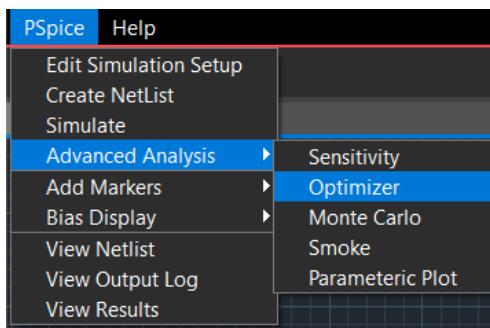
The measurement values are displayed in the *Specifications* table.

	On/Off	Profile	Measurement	Original	Min	Max	Specifications
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	ac.sim	Cutoff_Lowpass_3...	185.5971k	183.1889k	188.1857k	
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	pspicesim.sim	Max(V(C1:1))	1.1630	1.1514	1.1750	
<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	pspicesim.sim	SettlingTime(V(C1:1))	18.7856u	18.6819u	18.8829u	

## Running Optimizer Analysis

In our design, we use optimizer to meet certain measurement goals. To run Optimizer analysis for our design, do the following:

1. Choose *PSpice – Advanced Analysis – Optimizer*.



The *Optimizer* tool window is displayed.

2. In the *Parameter [Next Run]* table, click the cell with the text *Click here to import a parameter from the design property map*.
3. Click *Ctrl1* and select *Component R1, L1, and C1* for the *Parameter* named *VALUE*.

Parameters Selection Component Filter [*]				
Component	Parameter	Original	Min	Max
C1	TC2	0	0	0
C1	VALUE	1u	100n	10u
C1	VC1	0	0	0
C1	VC2	0	0	0
L1	IL1	0	0	0
L1	IL2	0	0	0
L1	TC1	0	0	0
L1	TC2	0	0	0
L1	VALUE	1u	100n	10u
PARAM	RVAR	1	100m	10
R1	TC1	0	0	0
R1	TC2	0	0	0
R1	VALUE	1	100m	10
V1	AC	1	100m	10
V1	ACPHASE	1	100m	10
V1	DC	0	0	0
V1	DC	0	0	0

To select multiple items, hold down the CTRL key, then click each entry.  
Hold down the SHIFT key to select or deselect adjacent items.

OK Cancel

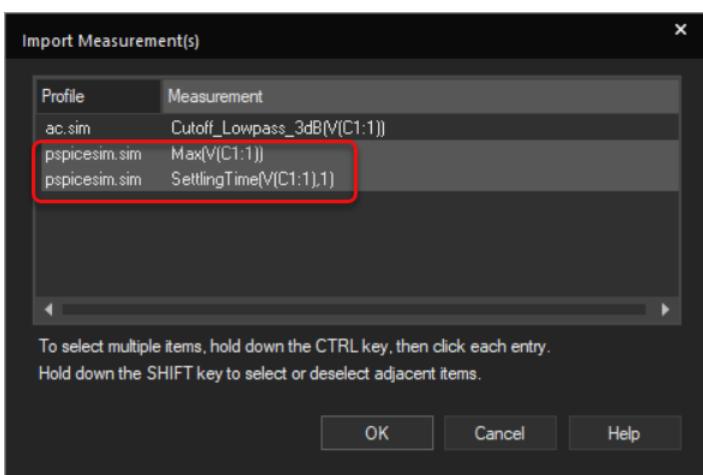
# Getting Started with Allegro X System Capture - PSpice A/D

## Running Advanced Analysis

- #### **4. Click *OK*.**

The components are added with their corresponding values.

5. Click the cell with the text *Click here to import a measurement created within PSpice* to import the measurement *Max(V(C1:1))* and *SettlingTime(V(C1:1))* in the *Specifications [Next Run]* section.



To meet the specific measurement goals, you need to specify goals in the *Advanced Analysis – Optimizer* window.

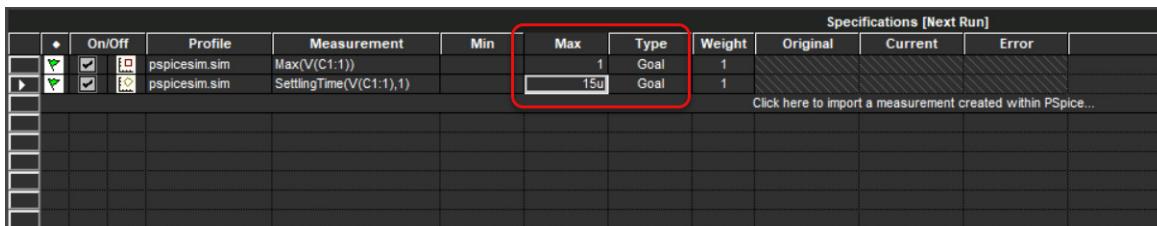
6. In the *Specifications [Next Run]* table, select the *Goal* function for both the imported measurements.

## Getting Started with Allegro X System Capture - PSpice A/D

### Running Advanced Analysis

---

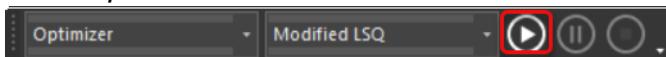
7. Specify the maximum measurement values of  $\text{Max}(V(C1:1))$  as 1 and  $\text{SettlingTime}(V(C1:1), 1)$  as  $15 \mu$ , respectively.



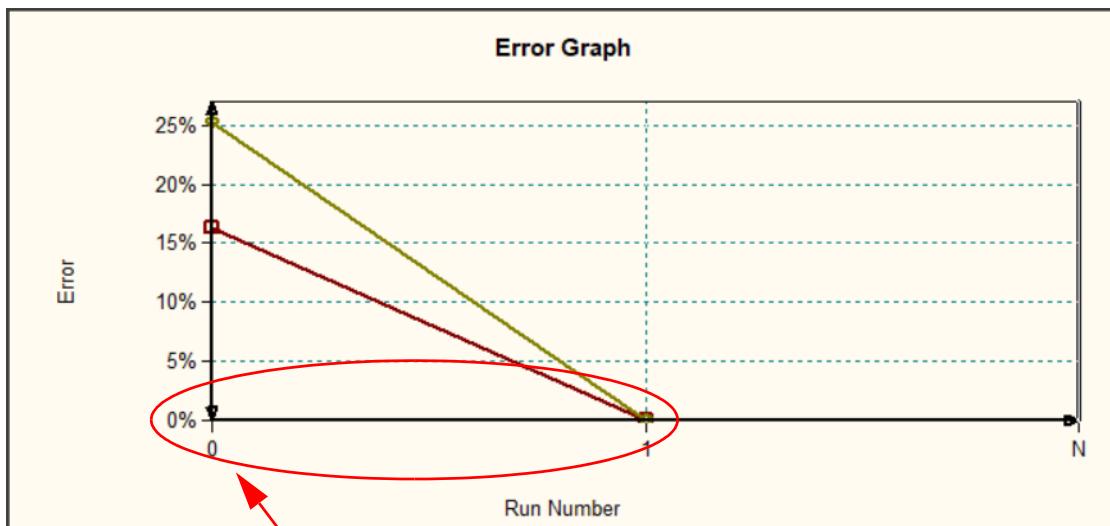
	On/Off	Profile	Measurement	Min	Max	Type	Weight	Original	Current	Error
	<input checked="" type="checkbox"/>	pspicesim.sim	Max(V(C1:1))		1	Goal	1			
	<input checked="" type="checkbox"/>	pspicesim.sim	SettlingTime(V(C1:1), 1)		15u	Goal	1			

Click here to import a measurement created within PSpice...

8. Run Optimizer.

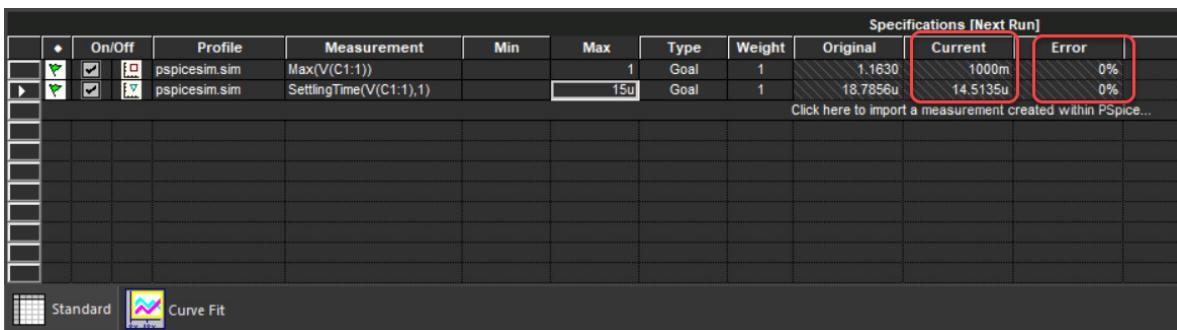


The resultant graph is displayed that shows the error% becomes zero at the optimization run 1.



Error% becomes zero from optimization run 0 to 1

The current values are optimized within the specified goals for added measurements.



	On/Off	Profile	Measurement	Min	Max	Type	Weight	Original	Current	Error
	<input checked="" type="checkbox"/>	pspicesim.sim	Max(V(C1:1))		1	Goal	1	1.1630	1000m	0%
	<input checked="" type="checkbox"/>	pspicesim.sim	SettlingTime(V(C1:1), 1)		15u	Goal	1	18.7856u	14.5135u	0%

Click here to import a measurement created within PSpice...

Standard    Curve Fit

# Getting Started with Allegro X System Capture - PSpice A/D

## Running Advanced Analysis

The optimized values for each of the selected components is shown in the *Parameters [Next Run]* table.

Optimized value within the defined goals

				Parameters [Next Run]				
	On/Off	Component	Parameter	Original	Min	Max	Current	
	<input checked="" type="checkbox"/>	R1	VALUE	1	100m	10	1.5306	
	<input checked="" type="checkbox"/>	L1	VALUE	1u	100n	10u	190n	
	<input checked="" type="checkbox"/>	C1	VALUE	1u	100n	10u	710.9872n	

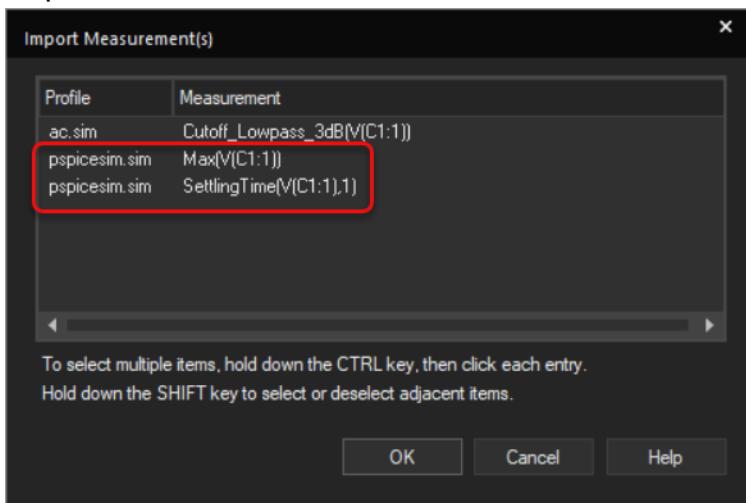
Click here to import a parameter from the design property map...

## Running Monte Carlo Analysis

Monte Carlo analysis checks the production yield running on a circuit and estimates statistical behavior. For the Monte Carlo analysis, tolerances are required to be specified. This is already done for the design.

To run the Monte Carlo analysis on the tutorial design, do the following:

1. In System Capture, choose *PSpice – Advanced Analysis – Monte Carlo*.  
The Advanced Analysis Monte Carlo tool is displayed in the PSpice window.
2. Import the measurement *Max(V(C1:1))* and *Settling Time(V(C1:1))* created within PSpice.

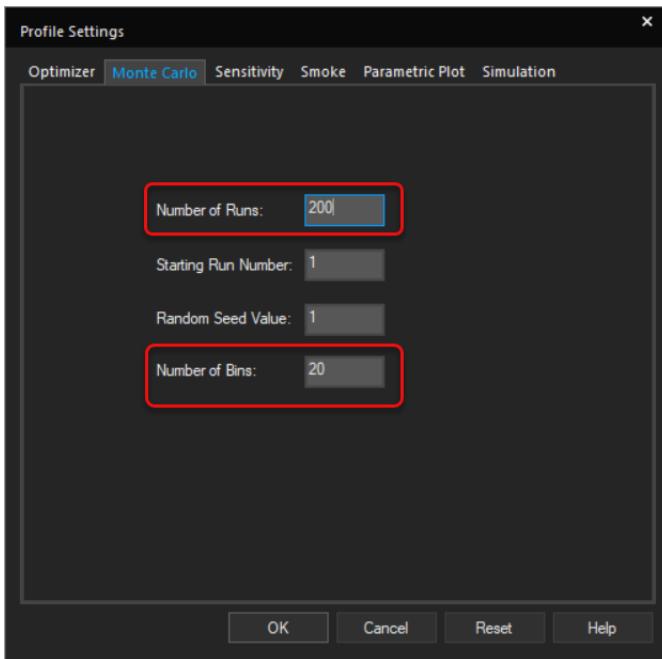


3. Click *OK*.
4. In the *Monte Carlo analysis* window, select *Edit – Profile Settings*.
5. Specify the value of *Number of Runs* as 200 and *Number of Bins* as 20 .

## Getting Started with Allegro X System Capture - PSpice A/D

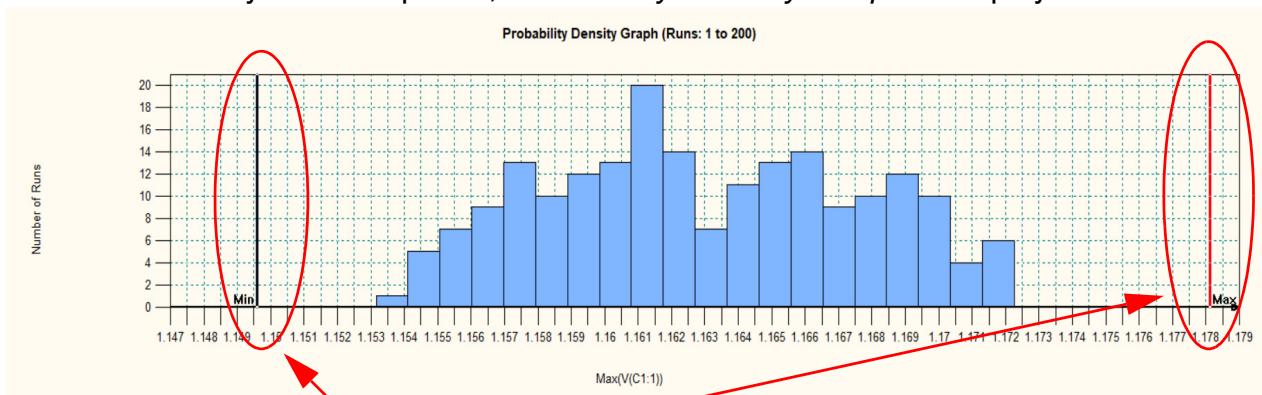
### Running Advanced Analysis

The Number of Bins indicates the number of divisions.



6. Click the *Start icon* to run the Monte Carlo analysis.

After the analysis is completed, *Probability Density Graph* is displayed.



Run within the Cursor Minimum and Cursor Maximum values indicating the yield of the design is 100%

The Monte Carlo analysis randomly varies all the model parameters of your design including R1, L1, and C1 for which tolerance is defined and determines the overall variation on products through graphical representation.

# Getting Started with Allegro X System Capture - PSpice A/D

## Running Advanced Analysis

All the parameter values for the selected measurements are shown in the *Statistical Information* table.

## Running Parametric Plotter

The Parametric Plotter analysis provides flexibility to sweep multiple parameters and records circuit behavior to ensure the stability of the design.

To run Parametric Plotter analysis, do the following:

1. Choose *PSpice – Advanced Analysis – Parametric Plot*.

The Parametric Plotter tab is displayed in the PSpice Advanced analysis window.

2. In the *Sweep Parameters* section of the *Parametric Plot* window, click the cell with the text *Click here to import a parameter from the design property map*.

For our design, we sweep the value of the components R1 and L1 and observe the impact of variations on the output result.

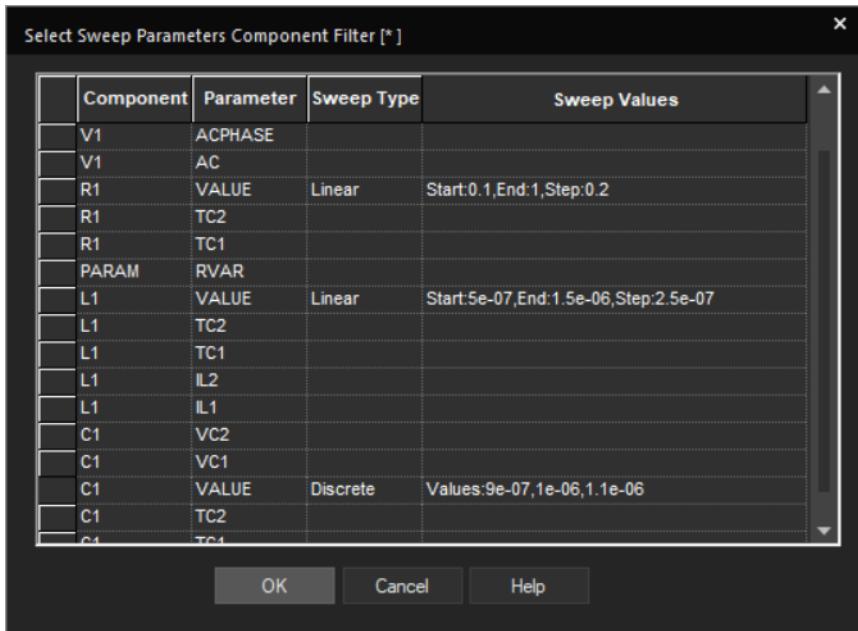
3. In the *Sweep Parametric Components* dialog box, specify the following sweep parameters:

- a. For the component *R1*, specify the *Parameter* as *Value* and *Sweep Type* as *Linear*.
- b. Specify the *Start Value* as *0 . 1*, *End Value* as *1*, and *Step Value* as *0 . 2*.
- c. For the component *L1*, specify the *Parameter* as *Value* and *Sweep Type* as *Linear*.
- d. Specify the *Start Value* as *5e-07*, *End Value* as *1.5e-06*, and *Step Value* as *2.5e-07*.
- e. For the component *C1*, specify the *Parameter* as *Value* and *Sweep Type* as *Discrete*.
- f. Specify the *Start Value* as *9e-07*, *End Value* as *1e-06*, and *Step Value* as *1.1e-06*.

## Getting Started with Allegro X System Capture - PSpice A/D

### Running Advanced Analysis

#### 4. Click OK.



The selected parameters are added in the *Sweep Parameter* window. After adding the parameters, a sweep variable is automatically assigned to each of the parameters.

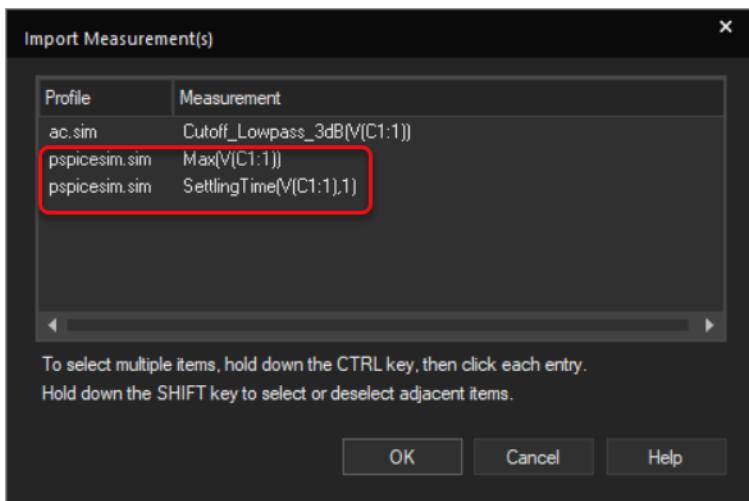
The value of the sweep variable defines how the parameter *value* varies during sweep analysis.

	On/Off	Component	Parameter	Sweep Variable	Sweep Type	Sweep Values	Number of Steps
▶	✓	r1	value	outer	Linear	Start:0.1,End:1,Step:0.2	5
▶	✓	l1	value	inner1	Linear	Start:5e-07,End:1.5e-06,S...	5
▶	✓	c1	value	inner2	Discrete	Values:9e-07,1e-06,1.1e-06	3

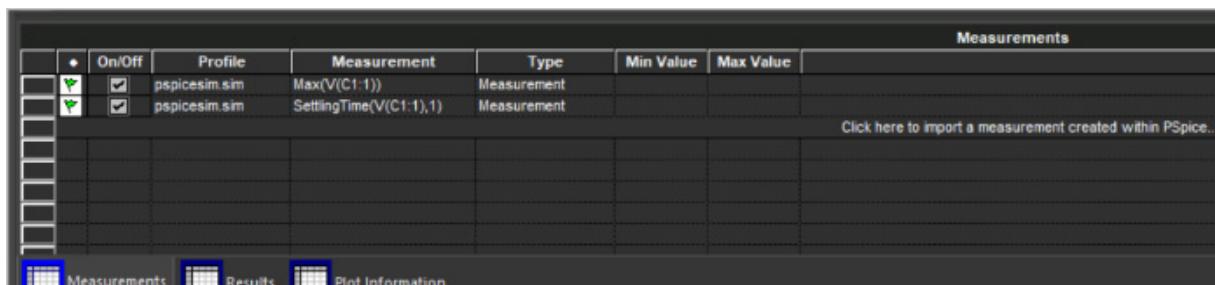
## Getting Started with Allegro X System Capture - PSpice A/D

### Running Advanced Analysis

- To simulate and observe variations for different values, click the cell with text *Click here to import a measurement created within PSpice*, and import the measurements as shown in the following figure:



The imported measurement is displayed in the *Measurements* section.



- To run the analysis, choose *Run – Start Parametric Plotter* or click the *Run* button.



- To view the measurement results, click the *Results* tab.

## Getting Started with Allegro X System Capture - PSpice A/D

### Running Advanced Analysis

The *Results* tab lists the values of the parameters and the measurement results for each value.

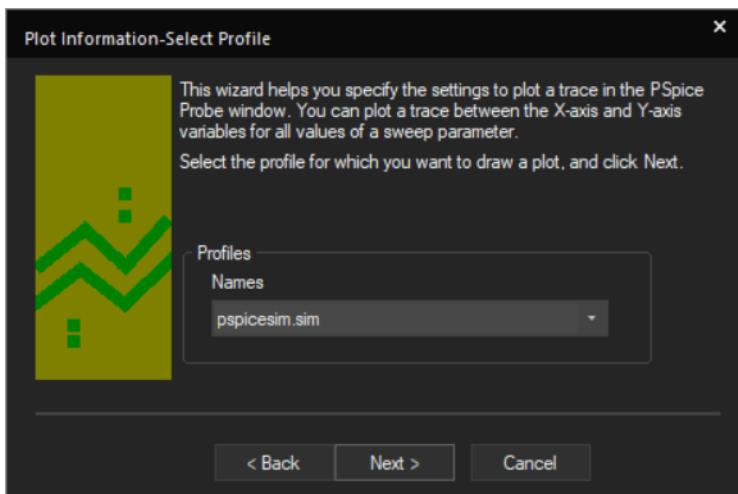
r1:value	i1:value	c1:value	Results pspicesim:
0.1	5e-07	9e-07	1.809588041219
0.1	5e-07	1e-06	1.800346788731
0.1	5e-07	1.1e-06	1.791648461985
0.1	7.5e-07	9e-07	1.841695371337
0.1	7.5e-07	1e-06	1.833860869504
0.1	7.5e-07	1.1e-06	1.826473265434
0.1	1e-06	9e-07	1.861404191249
0.1	1e-06	1e-06	1.854464283357
0.1	1e-06	1.1e-06	1.847912906563
0.1	1.25e-06	9e-07	1.875105819288
0.1	1.25e-06	1e-06	1.868801677612
0.1	1.25e-06	1.1e-06	1.862845874828
0.1	1.5e-06	9e-07	1.885353864145
0.1	1.5e-06	1e-06	1.87953245188
0.1	1.5e-06	1.1e-06	1.87402958756

8. To view the graph of the results, click the *Plot Information* tab.

9. Click the cell with the text *Click here to add plot*.

The *Plot Information* tab is displayed.

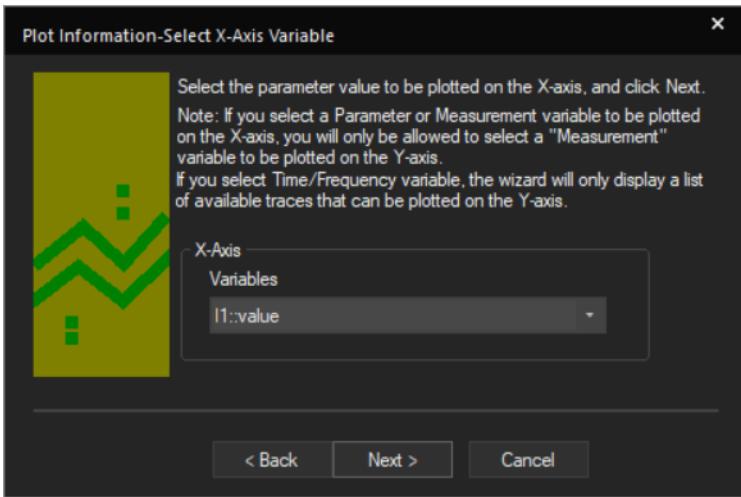
10. Select the profile *PSpicesim.sim* and click *Next*.



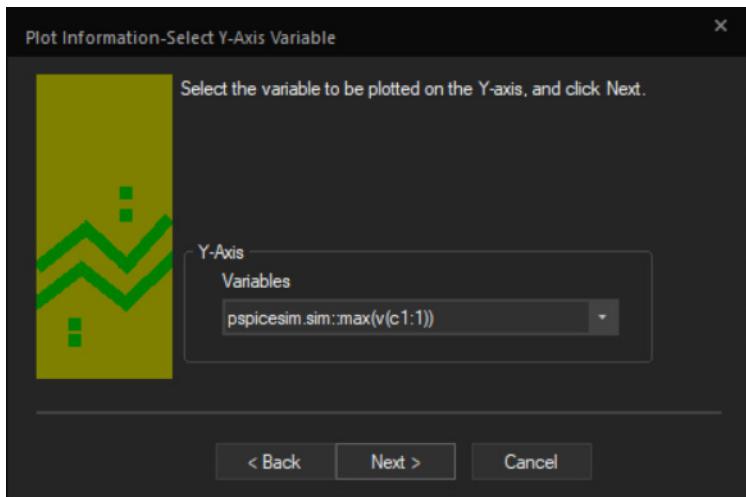
## Getting Started with Allegro X System Capture - PSpice A/D

### Running Advanced Analysis

11. Choose *I1::value* as the X-Axis variable and click *Next*.



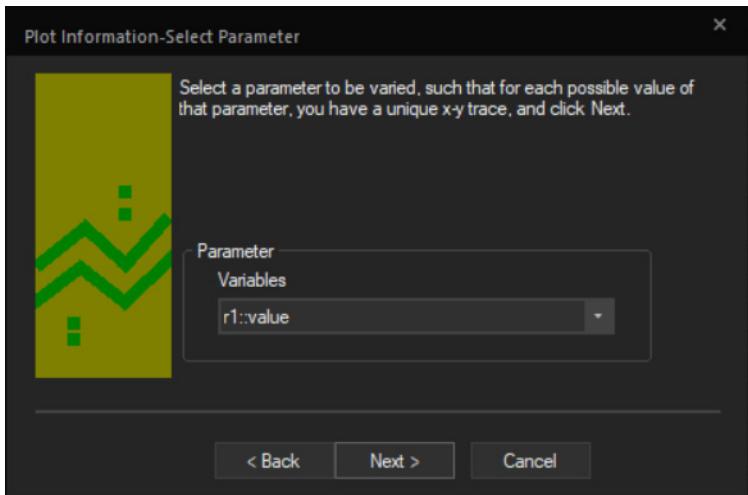
12. Choose *pSpicesim.sim::max(V(C1:1))* as the Y-Axis variable and click *Next*.



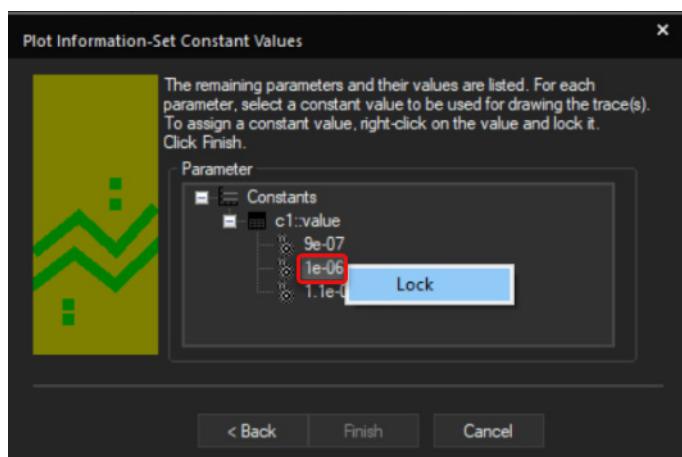
## Getting Started with Allegro X System Capture - PSpice A/D

### Running Advanced Analysis

13. Select *r1:value* in the *Variables* field and click *Next*.



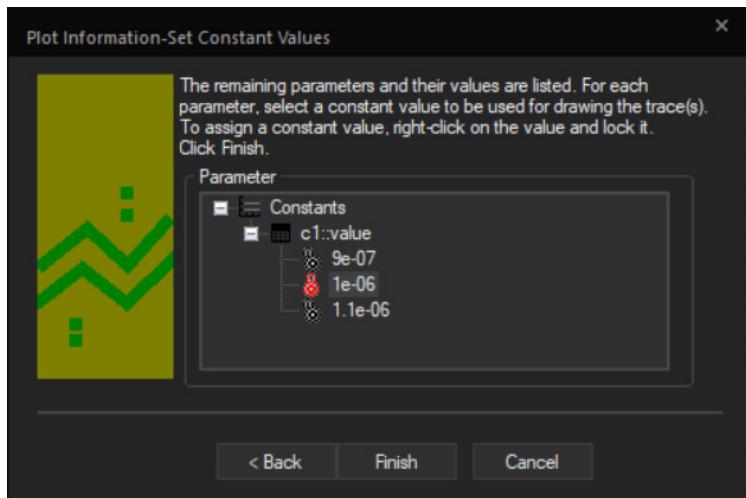
14. Right-click the value *1e-06* and choose *Lock*.



## Getting Started with Allegro X System Capture - PSpice A/D

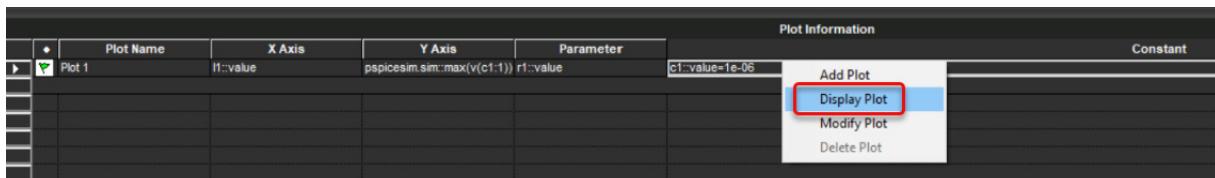
### Running Advanced Analysis

15. Click *Finish*.

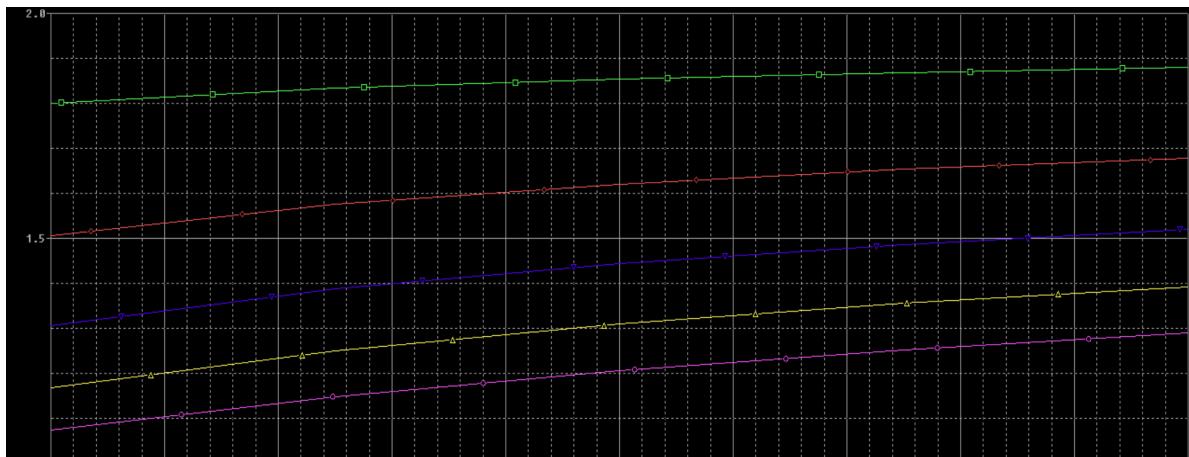


The plot is displayed in the *Plot Information* section of the *Parametric Plotter* window.

16. Right-click and select *Display Plot*.



The result of the Parametric Plotter is displayed in PSpice A/D.



For more information about various advanced analysis options, such as, Monte Carlo Analysis, Parametric Plot Analysis, Optimizer Analysis, and Sensitivity Analysis, refer to [PSpice A/D User Guide](#).

---

## Modifying a Component Model

---

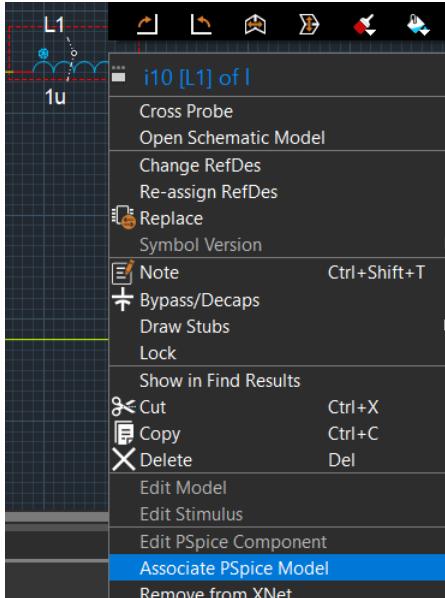
If you want to attach PSpice models available from different vendors, you can associate a PSpice simulation model with the design using *Associate PSpice Model* function. You can also edit the model associated with the design.

### Associating PSpice Models

Let us now associate a PSpice model to the inductor L1 from a model library.

To associate a PSpice model to the component L1, do the following:

1. Right-click the inductor *L1* and choose *Associate PSpice Model*.

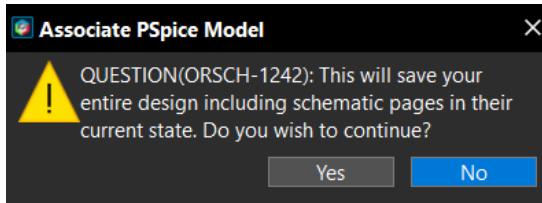


## Getting Started with Allegro X System Capture - PSpice A/D

### Modifying a Component Model

---

A message prompts you to confirm saving the design.



2. Click Yes to proceed.

The *Associate PSpice Model* dialog box is displayed.

In this tutorial, we will associate the *0402CS-1N0 inductor* model from the *coilcraft* library shipped with the installation.

3. In the *Model Library* field, click the *Browse* button and navigate to  
`<install_dir>tools/pspice/library`

The *Matching Models* section displays the list of models in the specified model library matching the selected component.

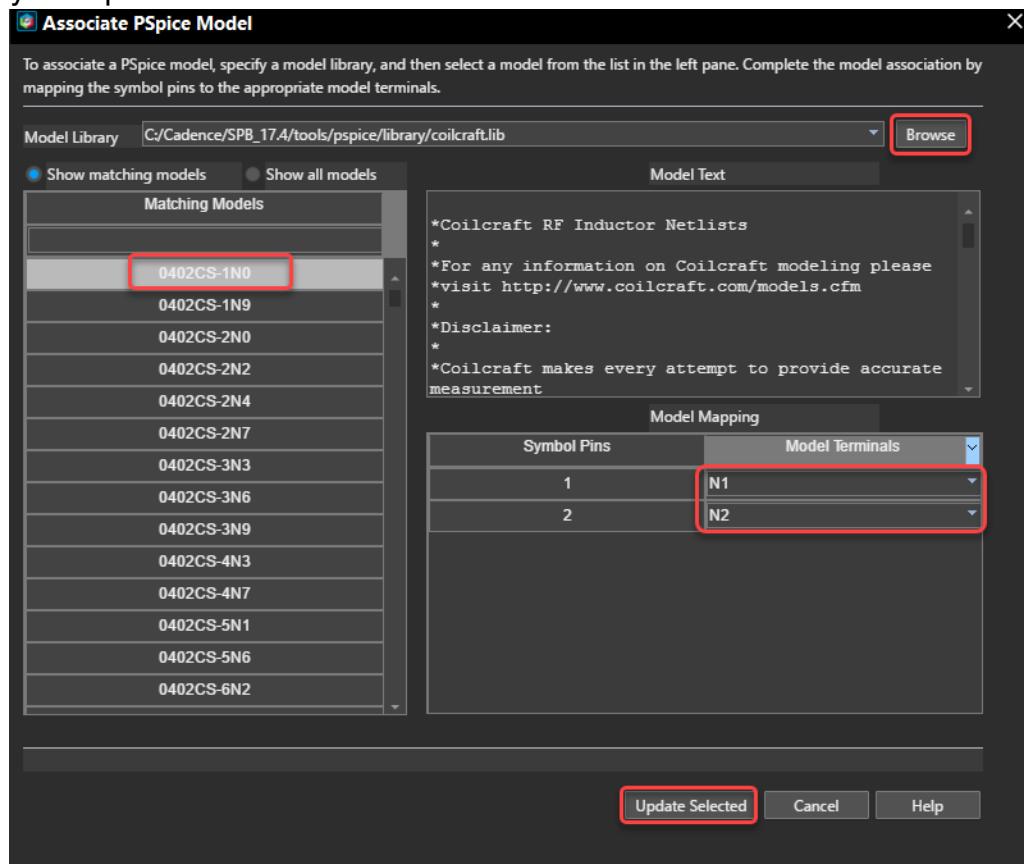
4. Select the model *0402CS-1N0* from the *Matching Model* list.

The *Model Text* section displays the model definition in text.

## Getting Started with Allegro X System Capture - PSpice A/D

### Modifying a Component Model

5. In the *Model Mapping* section, map the symbol pin 1 to the model terminal *N1* and the symbol pin 2 to the model terminal *N2*.



6. Click *Update Selected*.

A confirmation message is displayed indicating that the PSpice model is now associated with the inductor instance, *L1*.

7. Click *OK*.

## Getting Started with Allegro X System Capture - PSpice A/D

### Modifying a Component Model

With the new model association, the attributes of the component have also changed. You can check the associated PSpice model details in the *Attributes* pane of the *Properties* panel.

▼ Attributes			
Name	Value		
CURRENT	LMAX		
DIELECTRIC	DSMAX		
DIST	FLAT		
IC	<< NULL >>		
IL1	0		
IL2	0		
IMPLEMENTATION	0402CS-1N0		
IMPLEMENTATION_PA...	<< NULL >>		
IMPLEMENTATION_TY...	PSpice Model		
K1	3.40e-06		
PART_NAME	L		
PART_NUMBER	L		
PINMAPPING	{"N1":"1","N2":"2"}		
PSPICELIB	D:/Cadence/...		
PSPICETEMPLATE	X^@REFDES %1 %2...		
PSPICE_MODEL_TYPE	0011		
TC1	0		
TC2	0		
TOLERANCE	1%		

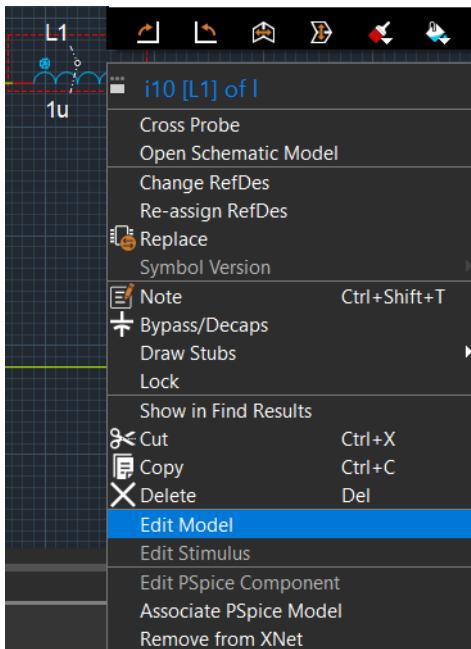
Properties Details

If required, you can view and edit the PSpice model associated with components in the design.

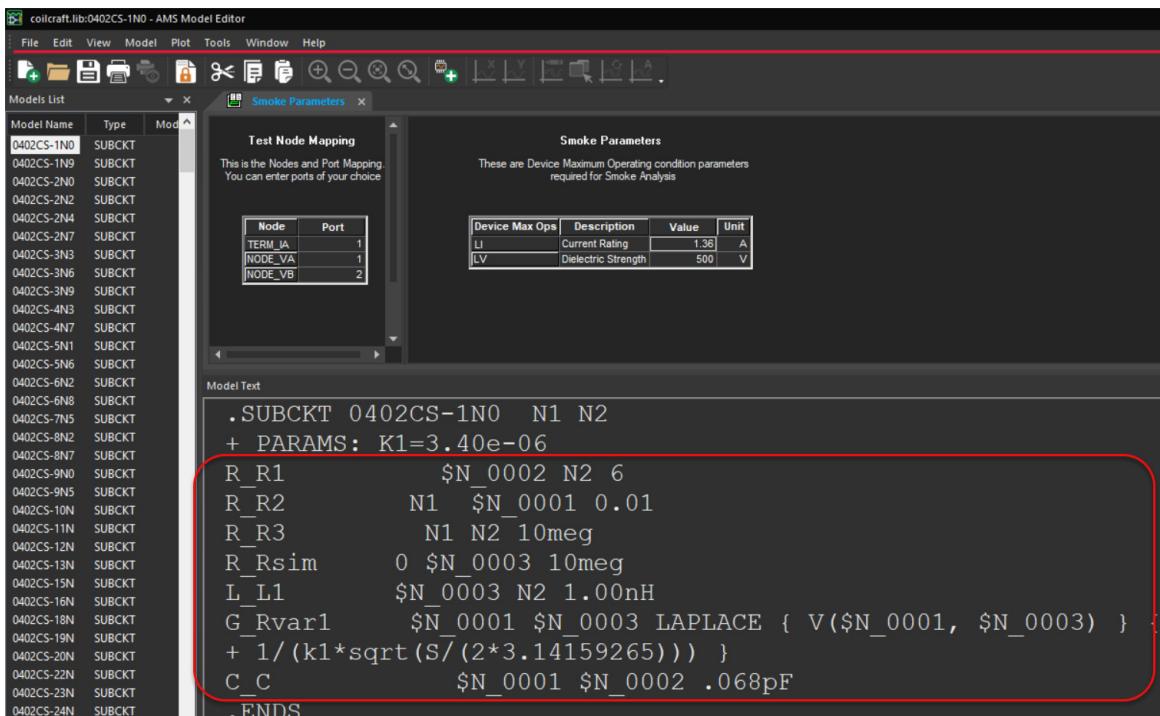
# Getting Started with Allegro X System Capture - PSpice A/D

## Modifying a Component Model

- To edit the inductor model, right-click the component *L1* choose *Edit Model*.



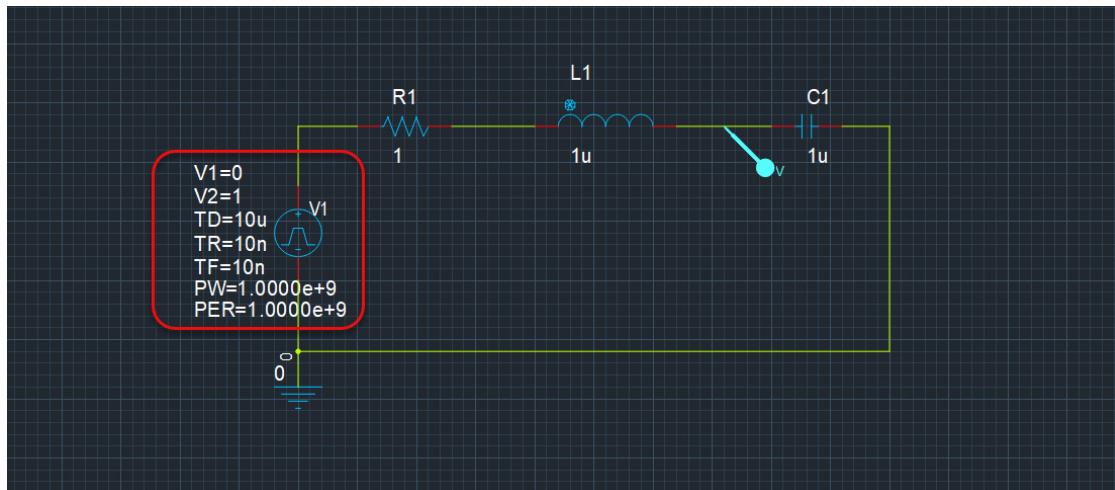
*Model Editor* window is displayed. It shows the model definition where you can modify the component values.



## Adding Stimulus to Circuit

To check how the circuit responds to a stimulus, we will create a custom stimulus and use it in our design.

Before adding an input stimulus source to the design, remove the instantiated voltage source from the circuit that is shown in the following figure:



To delete the voltage source, do the following:

1. Click the voltage source and press the *Delete* button.
2. Save the changes.
3. Set the default profile *PSpiceSim* as an active profile.

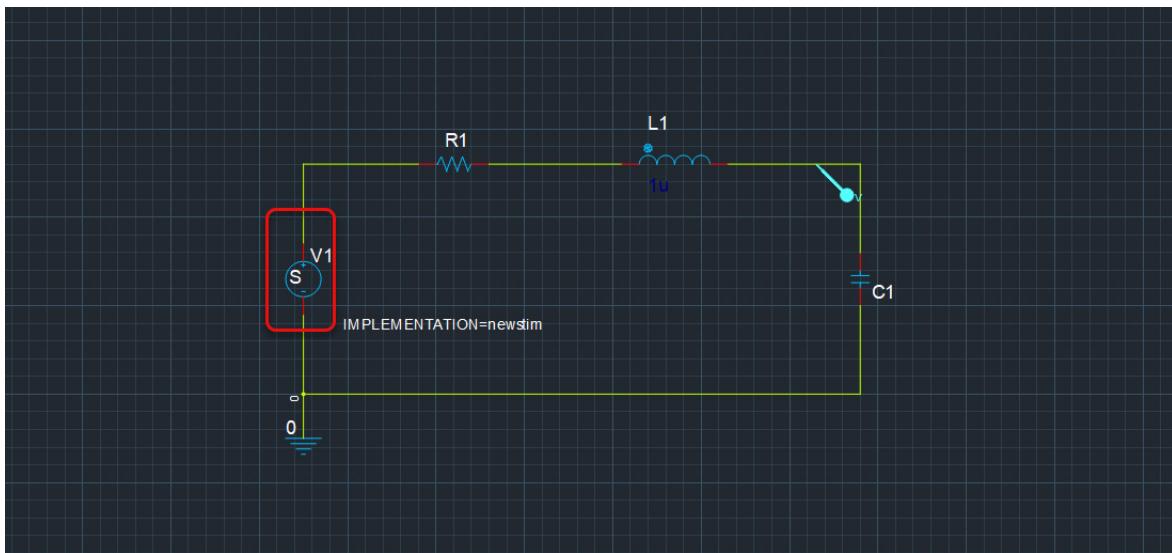
Now, to add new stimuli to your circuit, follow these steps:

4. Search for *VSTIM* in the *Unified Search window* and place the part as an input to the circuit.

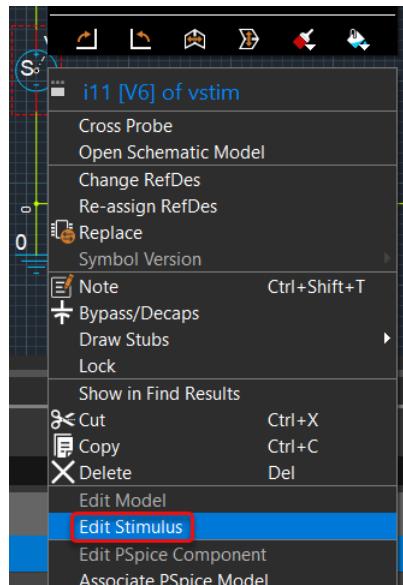
## Getting Started with Allegro X System Capture - PSpice A/D

### Modifying a Component Model

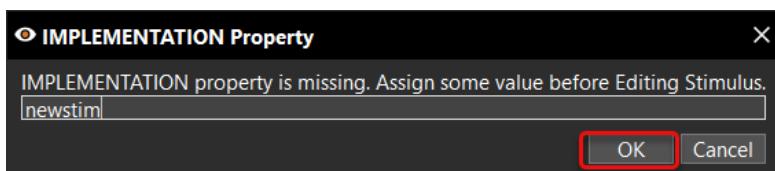
The added stimulus source is shown in the following figure:



5. Right-click *VSTIM* and choose *Edit Stimulus*.



6. In the *IMPLEMENTATION* Property dialog box, assign the value of the instantiated VSTIM as *newstim* and click *OK*.



## Getting Started with Allegro X System Capture - PSpice A/D

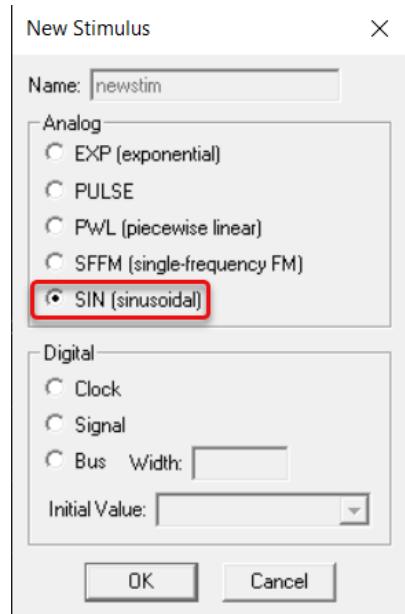
### Modifying a Component Model

The VSTIM value is automatically added as *newstim* in the *Attributes* pane.

▼ Attributes	
Name	Value
AC	<< NULL >>
DC	<< NULL >>
IMPLEMENTATION	newstim
IMPLEMENTATION_PATH	<< NULL >>
IMPLEMENTATION_TYPE	PSpice Stimulus
PACK_IGNORE	TRUE
PSPICEONLY	TRUE
PSPICETEMPLATE	V^@REFDES %+ %- ?D...
STIMTYPE	ANALOG
VALUE	<< NULL >>
<Add Property>	

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

7. To check the circuit behavior for a sinusoidal input voltage, select *Sin (Sinusoidal)*.



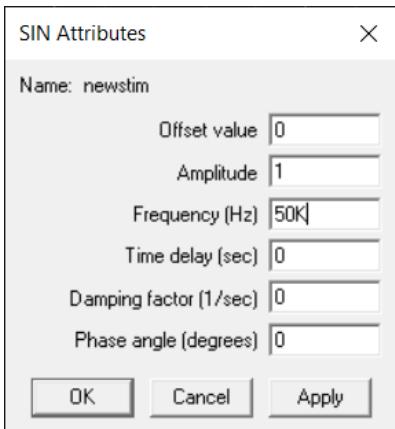
8. Specify the parameter values as follows:

- Offset Value* as 0
- Amplitude* as 1

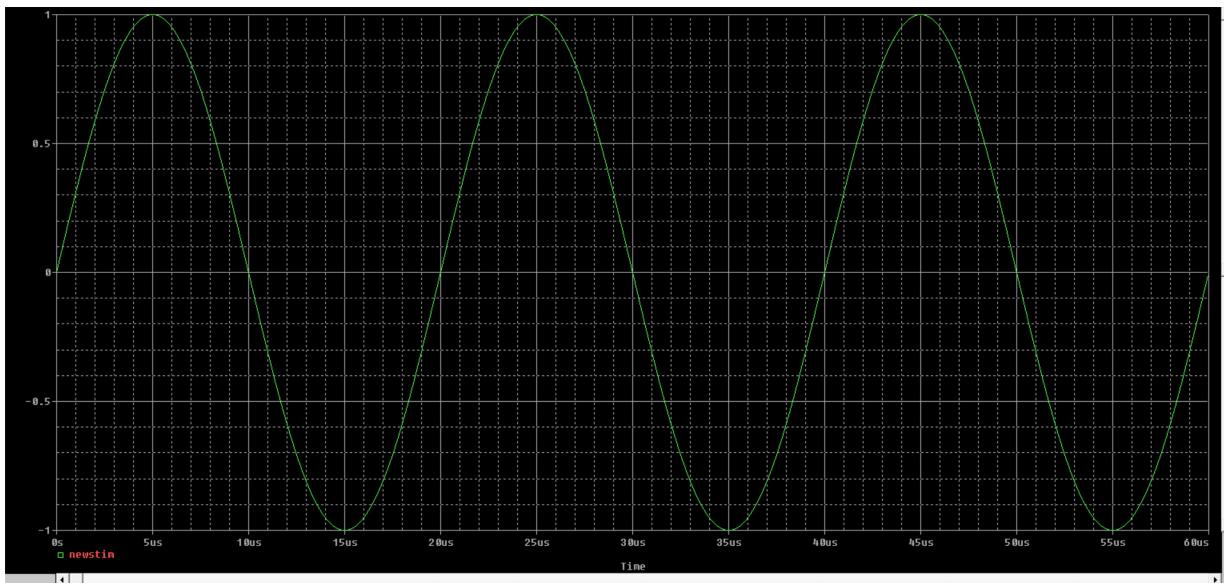
## Getting Started with Allegro X System Capture - PSpice A/D

### Modifying a Component Model

#### Frequency as 50 K

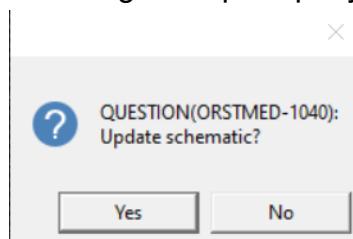


9. Click *OK* and the sinusoidal input waveform is displayed.



10. Choose *File – Save* to save the waveform.

A message box prompts you to update the schematic.



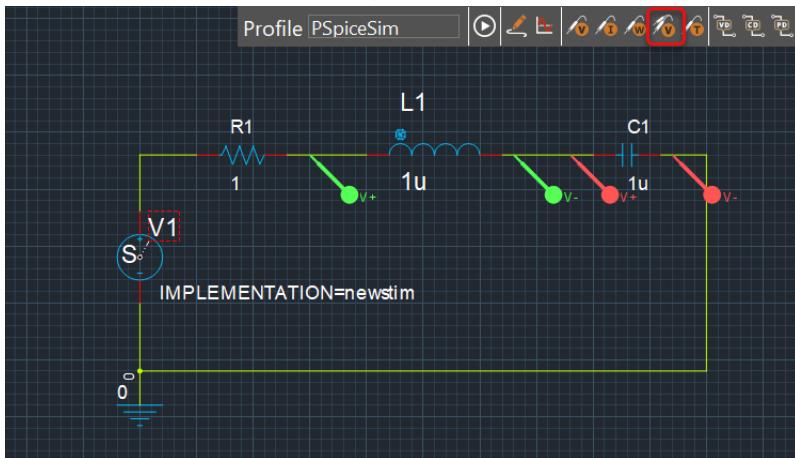
11. Click *Yes*, to update the schematic with the stimulus information.

## Getting Started with Allegro X System Capture - PSpice A/D

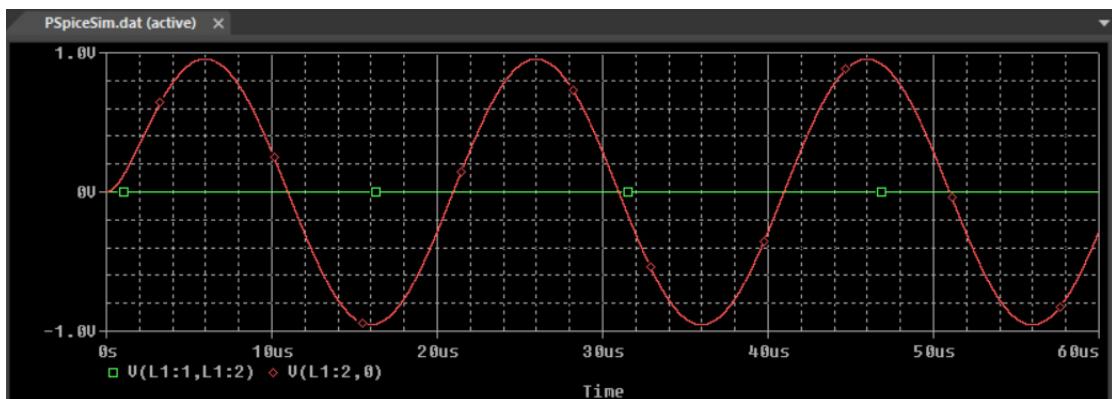
### Modifying a Component Model

12. Run the simulation.

13. Place the differential voltage marker across  $L_1$  and  $C_1$ .



The output waveform is displayed as shown in the following figure:

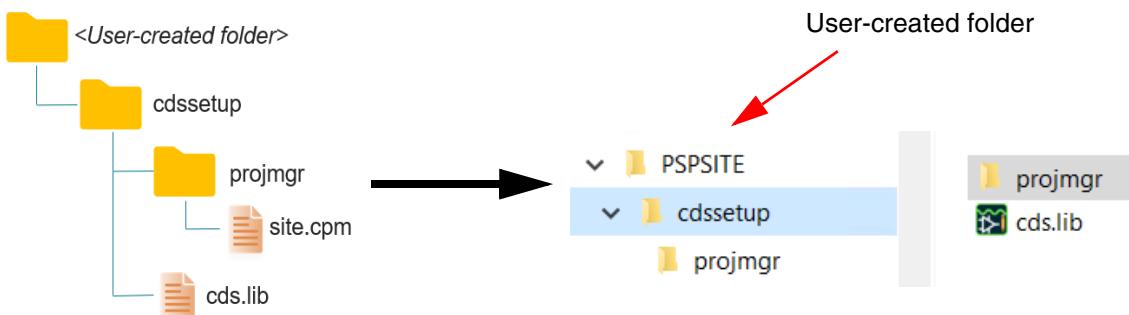


It shows the voltage across the inductor  $L_1$ , and capacitor  $C_1$  for the stimulus input.

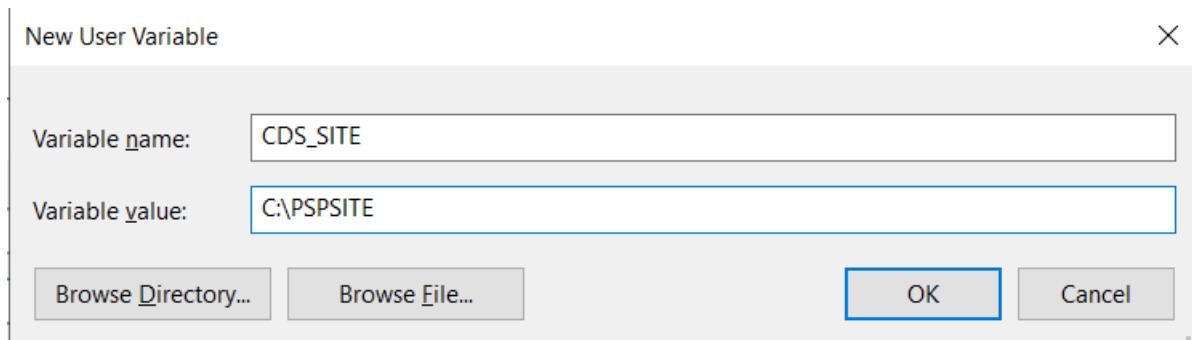
## Setting up PSpice Libraries

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.



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.

## Getting Started with Allegro X System Capture - PSpice A/D

### Setting up PSpice Libraries

---

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`.

 site.cpm - Notepad

File Edit Format View Help

```
START_GLOBAL
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.