

PSpice for TI: Getting Started

**Product Version 17.40 - 2020
June 2020**

© 2020 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.

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

<u>Working With TI Sample Designs</u>	1
<u>Opening Sample Reference Project</u>	1
<u>Simulating a Design</u>	3
<u>Searching for a TI Part</u>	4
<u>Accessing Design Resources for a Particular TI Part</u>	7
<u>Opening a Reference Design</u>	9
<u>Simulating a Reference Design</u>	10
<u>Working With a New Design</u>	13
<u>Creating a New Project</u>	13
<u>Creating a Design</u>	15
<u>Placing the Op-Amp</u>	17
<u>Placing DC Voltage Sources using Modeling Application</u>	18
<u>Placing Ground Symbols</u>	21
<u>Placing the Resistors</u>	21
<u>Connecting the Components</u>	23
<u>Assigning Net Aliases</u>	23
<u>Creating a New Simulation Profile</u>	26
<u>Placing Markers</u>	27
<u>Running the simulation</u>	29

PSpice for TI: Getting Started

Working With TI Sample Designs

This chapter demonstrates how you can use sample designs shipped with PSpice for TI. You will learn how to simulate and modify the sample design.

Opening Sample Reference Project

To open a sample reference project, do the following:

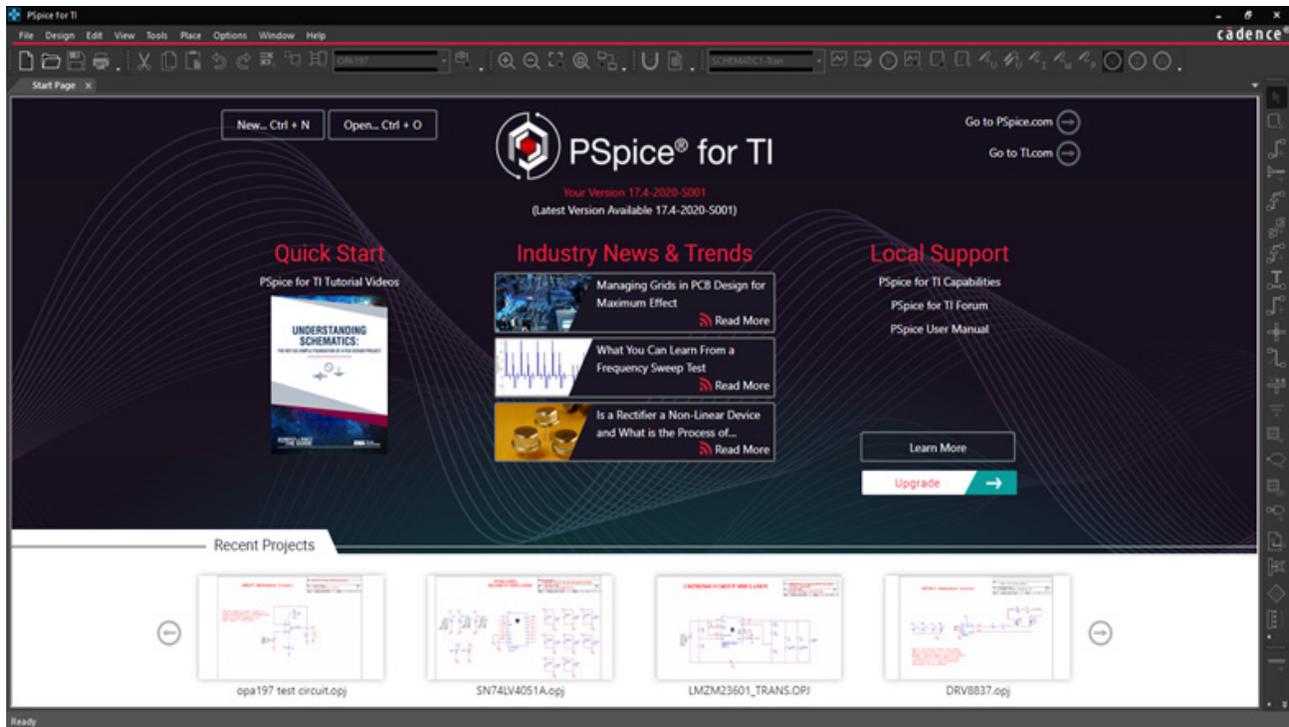
1. From the start menu, select *PSpice for TI 2020*.

Sample projects will be seen on the first invocation of PSpice for TI. You can choose sample projects from the *Recent Projects* section and whatever is your last opened project, it will start showing up in this section.

PSPice for TI: Getting Started

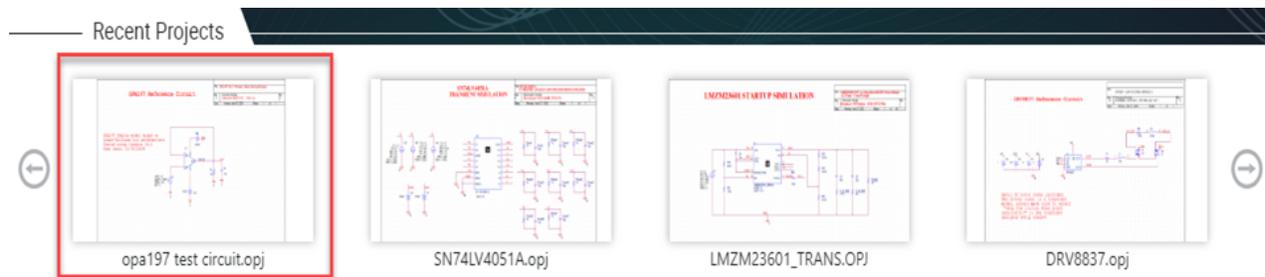
Working With TI Sample Designs

Figure 1-1 PSPice for TI start page



2. Click *opa197 test circuit.opj* from *Recent Projects* section to open the project.

Figure 1-2 Recent Projects List

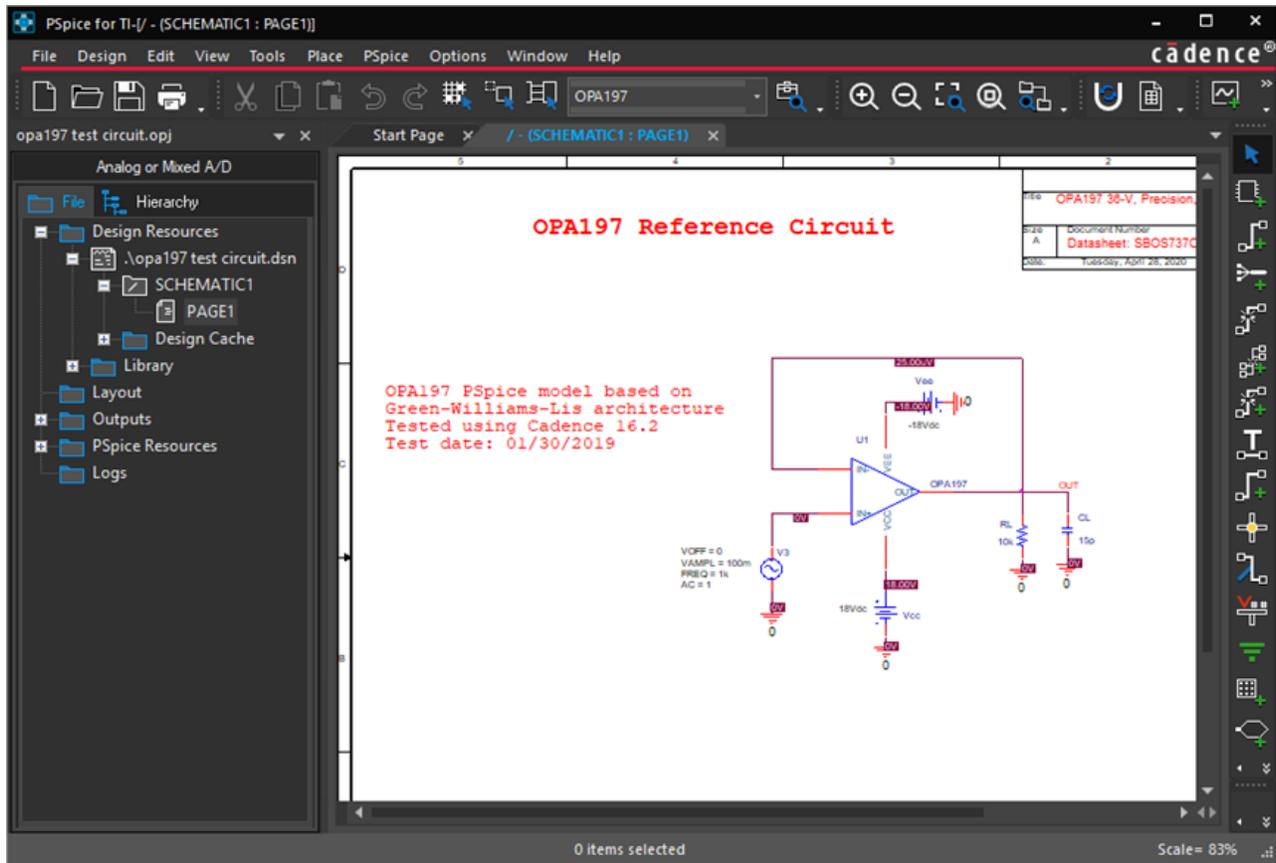


3. Open PAGE1 under SCHEMATIC1.

PSpice for TI: Getting Started

Working With TI Sample Designs

Figure 1-3 OPA197 Reference Circuit schematic page



Simulating a Design

When the simulation is run from the schematic, the simulator reads the PSPICE models connectivity information (netlist) from the design files, and the analysis type from the simulation profile. All sample designs are complete in connectivity and have predefined simulation profiles.

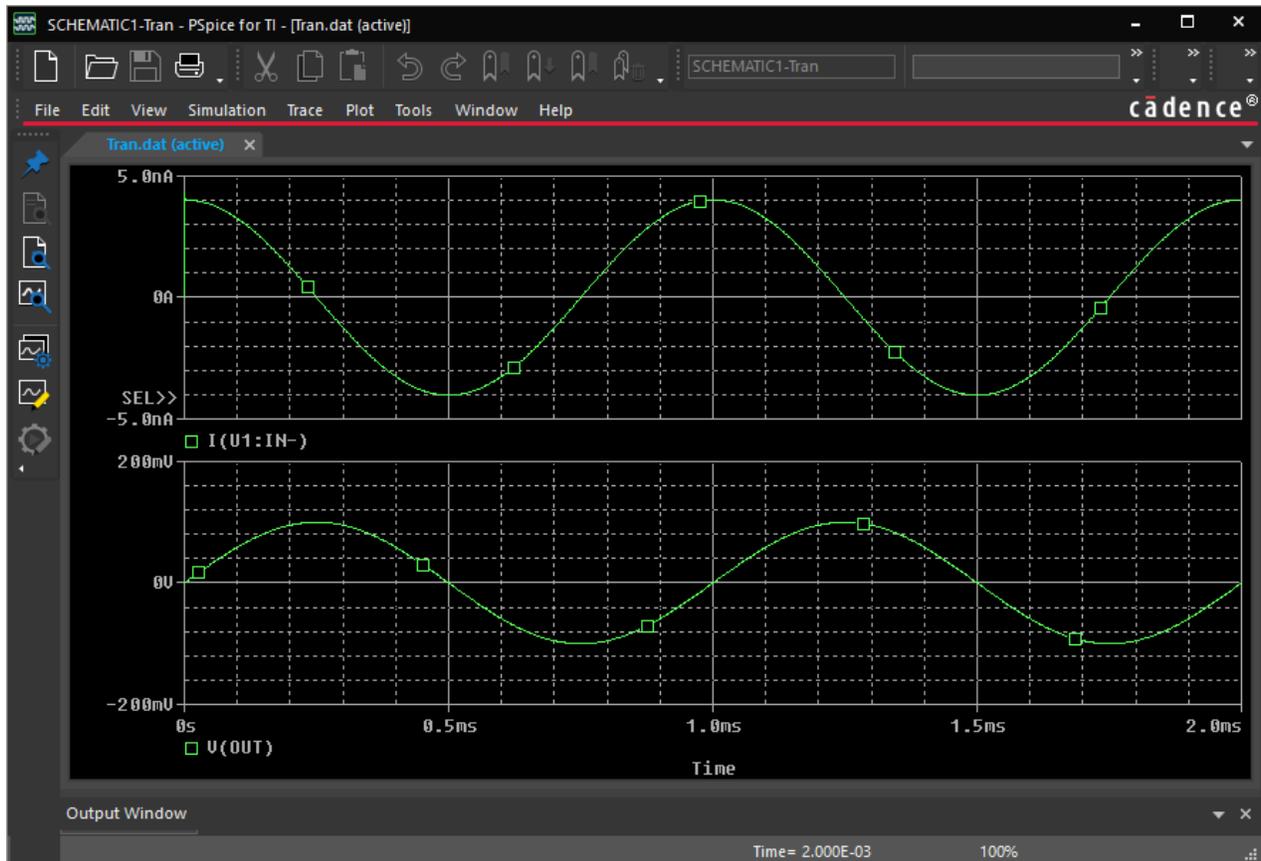
To simulate the design, choose *PSpice – Run* or click the *Run PSpice* icon () on the PSpice toolbar.

The output waveforms will appear in the PSpice plot window.

PSpice for TI: Getting Started

Working With TI Sample Designs

Figure 1-4 Output Waveform - Probe window



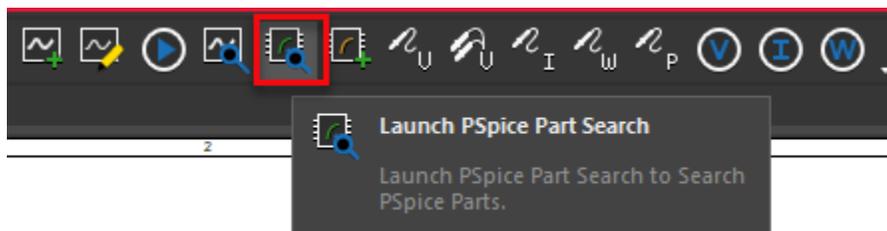
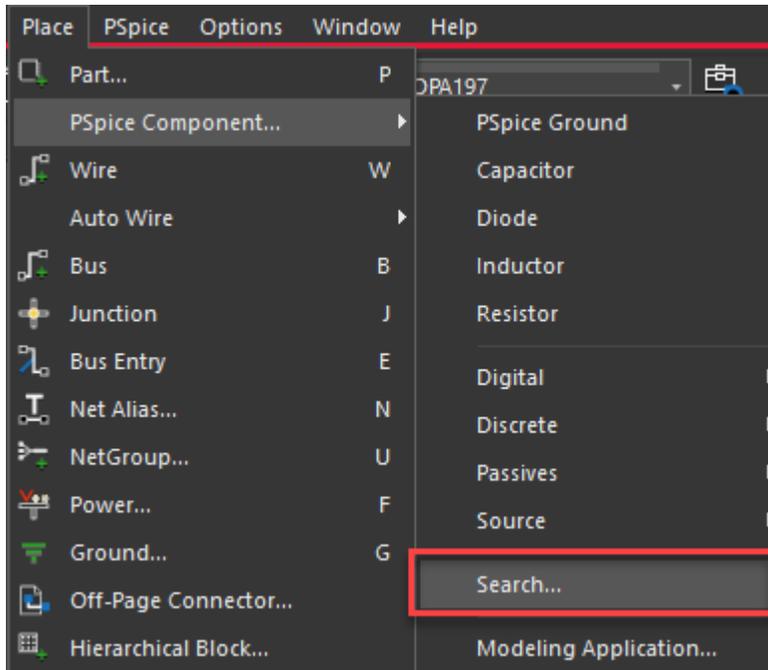
Searching for a TI Part

To search for a TI part, do the following:

PSpice for TI: Getting Started

Working With TI Sample Designs

1. Select *Place – PSpice Component – Search* or click the *Launch PSpice Part Search* icon () on the PSpice toolbar.



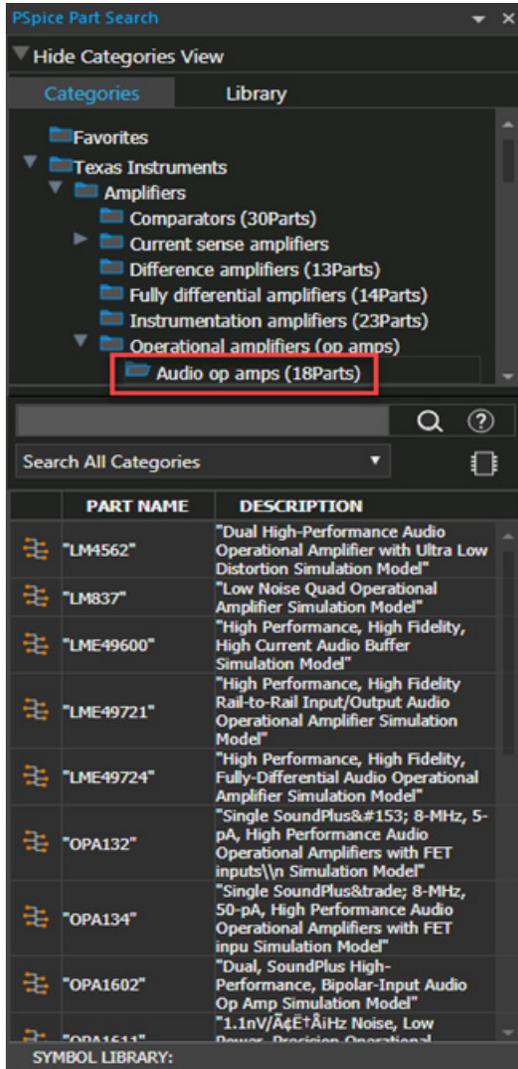
The *PSpice Part Search* panel opens. You can browse for parts under various categories as per design needs.

2. For example, to view audio operational amplifier, select *Texas Instrument - Amplifiers - Operational amplifiers (op amps) - Audio op amps* to display various op-amps available.

PSpice for TI: Getting Started

Working With TI Sample Designs

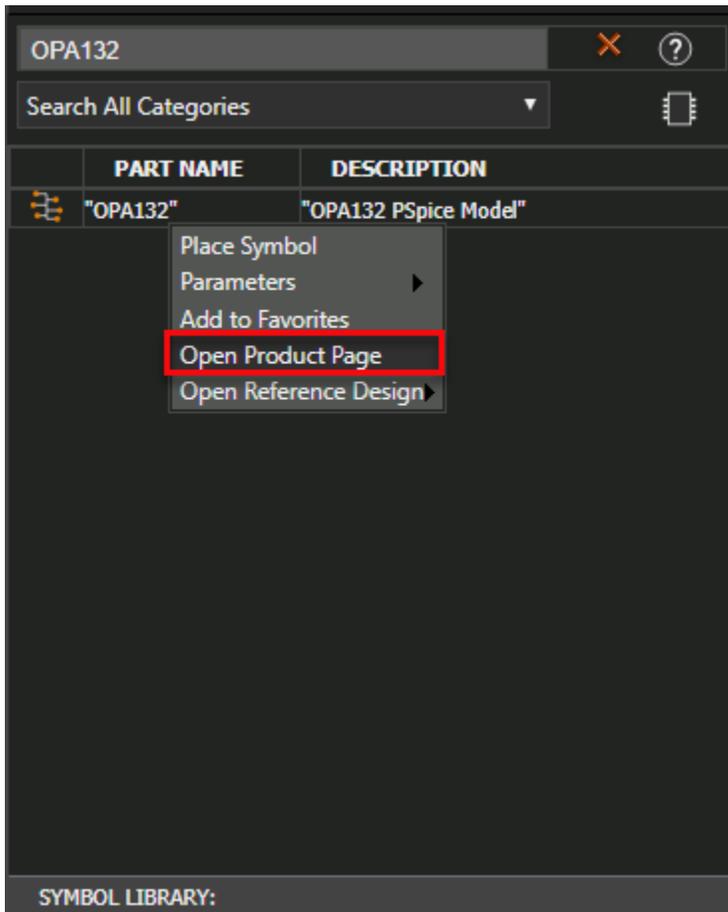
Figure 1-5 PSpice Part Search window



3. To search for a particular part (say *OPA132*), enter *OPA132* in the part search field and click the  icon.

Accessing Design Resources for a Particular TI Part

- To view the product page of a particular part, right-click the part name in the part search field and click *Open Product Page*.



The OPA132 product page will open from ti.com on your web browser .

PSPICE for TI: Getting Started

Working With TI Sample Designs

Figure 1-6 OPA132 Product Page

The screenshot displays the OPA132 product page on the Texas Instruments website. The browser address bar shows the URL: ti.com/document-viewer/OPA132/datasheet/abstract#SBOS0548824. The page title is "OPAx132 High-Speed FET-Input Operational Amplifiers (Rev. B)".

The page layout includes a navigation bar with icons for Product, Email, PDF, and TI store. Below this is a "CONTENTS" sidebar with a search bar and a list of sections: 1 Features, 2 Applications, 3 Description, 4 Revision History, and 5 Pin Configuration and Functions. The "Features" section is currently selected and highlighted in light blue.

The main content area is divided into two sections:

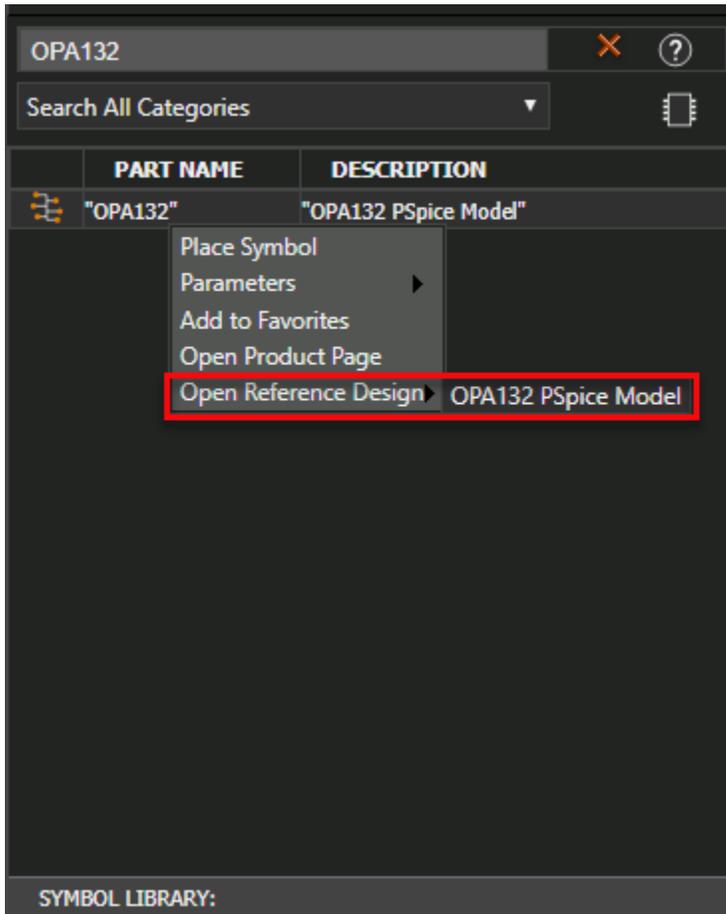
- 1 Features**
 - FET input: I_{i1} = 50 pA Maximum
 - Wide Bandwidth: 8 MHz
 - High Slew Rate: 20 V/ μ s
 - Low Noise: 8nV/ $\sqrt{\text{Hz}}$ (1 kHz)
 - Low Distortion: 0.00008%
 - High Open-loop Gain: 130 dB (600- Ω load)
 - Wide Supply Range: ± 2.5 to ± 18 V
 - Low Offset Voltage: 500 μ V Maximum
 - Single, Dual, and Quad Versions
- 2 Applications**
 - SAR ADC Driver
 - Voltage Reference Buffer
 - Trans-impedance Amplifier
 - Photodiode Amplifier
 - Active Filters
 - Integrators

Below the main content, there are sections for "Package Options", "Mechanical Data (Package|Pins)" (listing D|8 and MSOI002K), and "Thermal pad, mechanical data".

The footer of the page includes the Texas Instruments logo, a copyright notice: "© Copyright 1995-2020 Texas Instruments Incorporated. All rights reserved.", and a link to "Submit documentation feedback". Other links include "IMPORTANT NOTICE", "Trademarks", "Privacy policy", "Cookie policy", "Terms of use", and "Terms of sale".

Opening a Reference Design

1. To open a reference design, right-click on the part and select *Open Reference Design* menu to see a list of reference designs available.

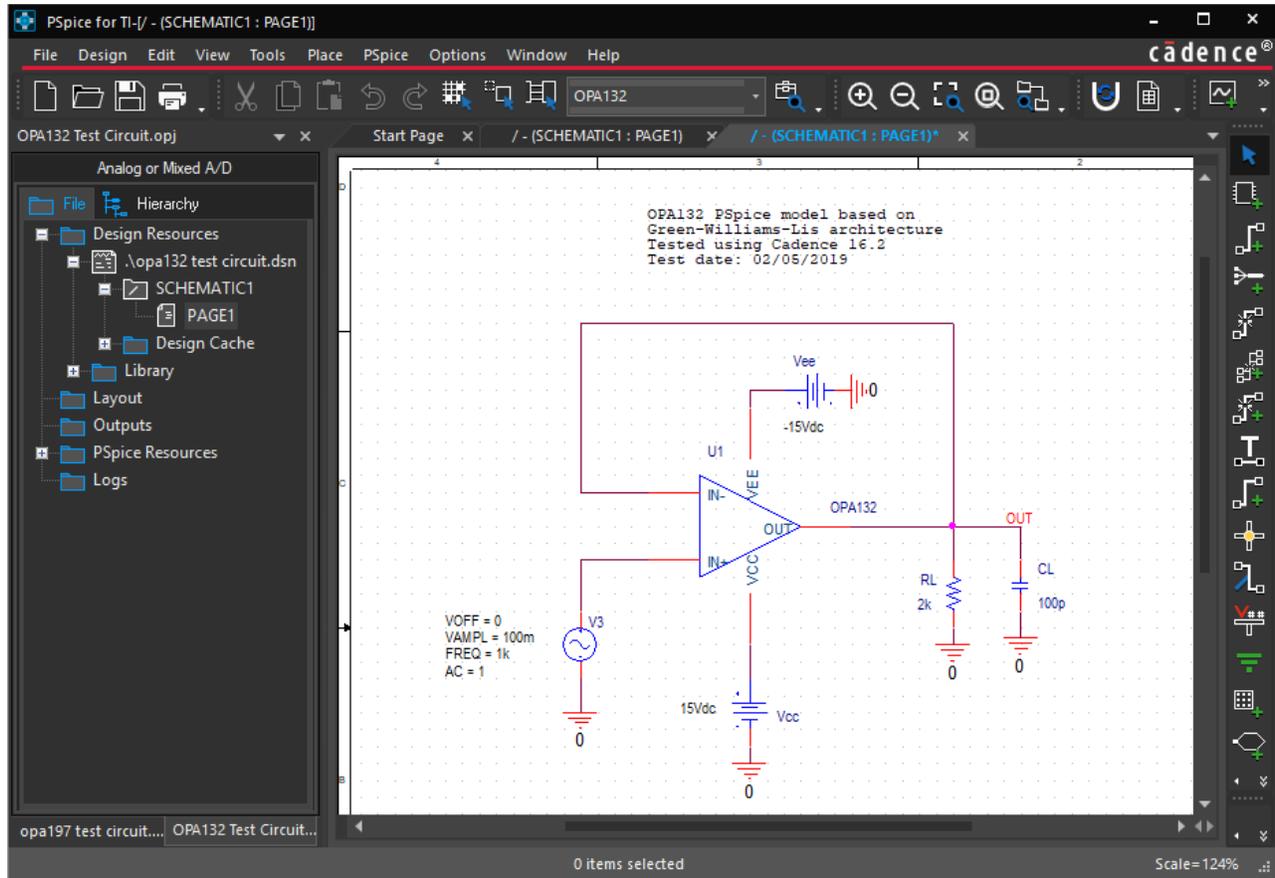


2. Click on the reference design name to open the corresponding reference design.

PSpice for TI: Getting Started

Working With TI Sample Designs

Figure 1-7 OPA132 Test Circuit schematic page



Simulating a Reference Design

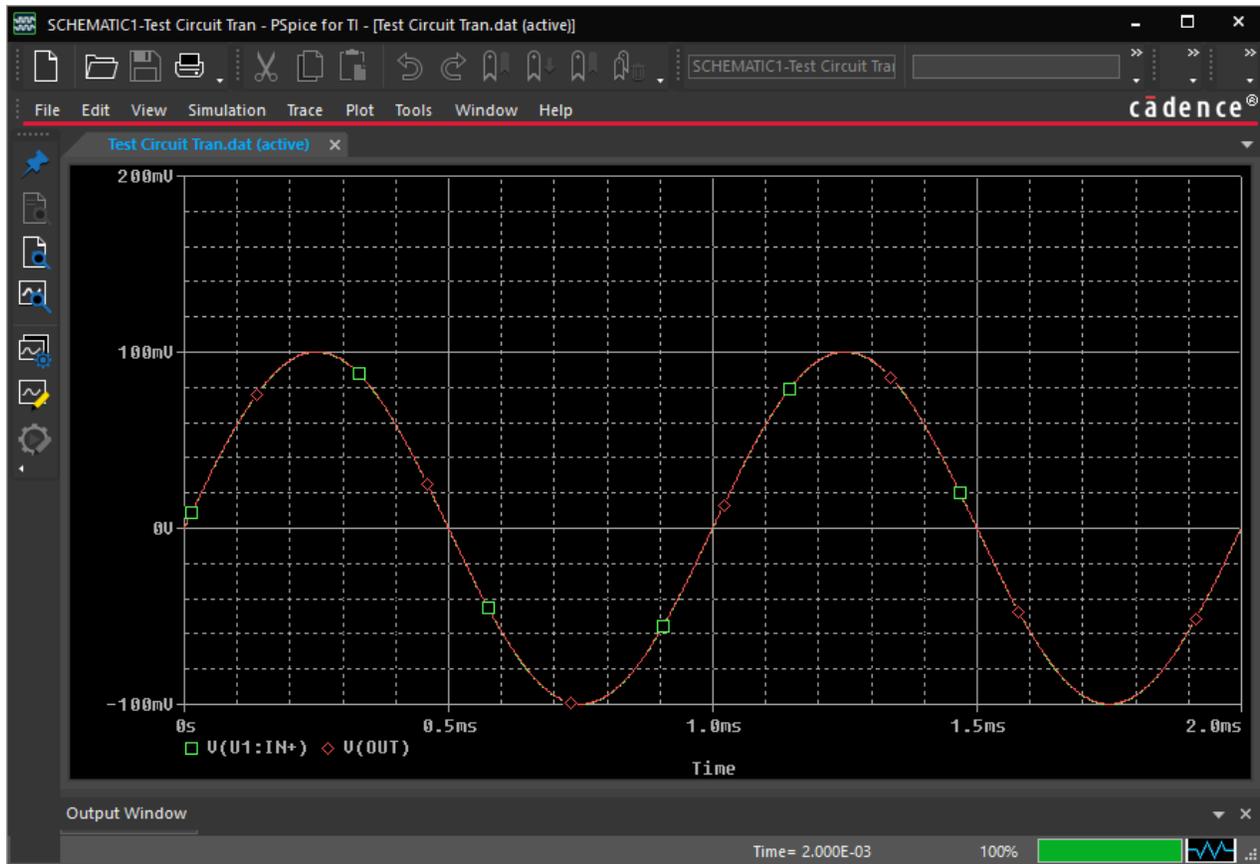
1. To simulate a reference design, choose *PSpice – Run* or click the Run PSpice icon (🎮) on the PSpice toolbar.

The output waveform will appear in the PSpice plot window.

PSpice for TI: Getting Started

Working With TI Sample Designs

Figure 1-8 Output Waveform - Probe window



PSpice for TI: Getting Started

Working With TI Sample Designs

Working With a New Design

This chapter demonstrates how to create and simulate a new design. You will learn the basic tasks to perform in order to create a new schematic design, and running the simulation.

Creating a New Project

A project file (.OPJ) is a container for the design file (.DSN). In addition, a project file also includes simulation profile.

To create a new project, do the following:

1. Select *File – New – Project*.
2. In the *New Project* dialog box, specify the project name as `tutorial`.
3. Specify the location where you want the project files to be created.

For this tutorial, specify the location as: `C:\TI_PSpice_Tutorial`.

4. Click *OK*.

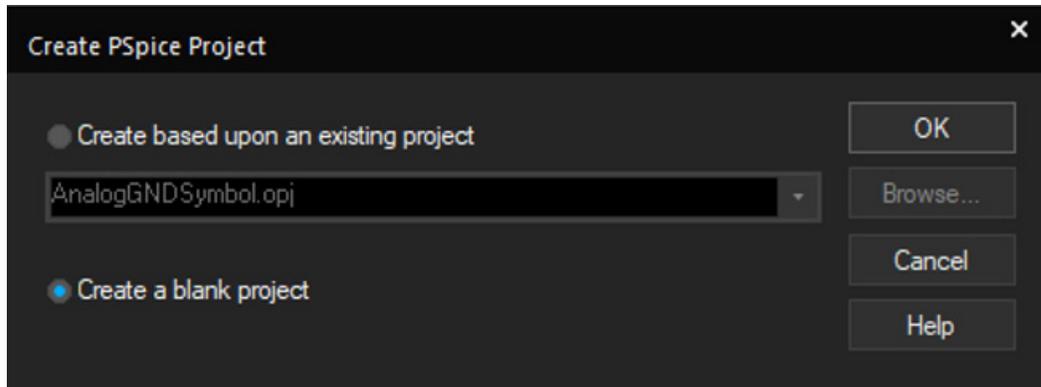
The *Create PSPice Project* dialog box appears.

5. Select *Create a blank project*.

PSpice for TI: Getting Started

Working With a New Design

Figure 2-1 Create PSpice Project window



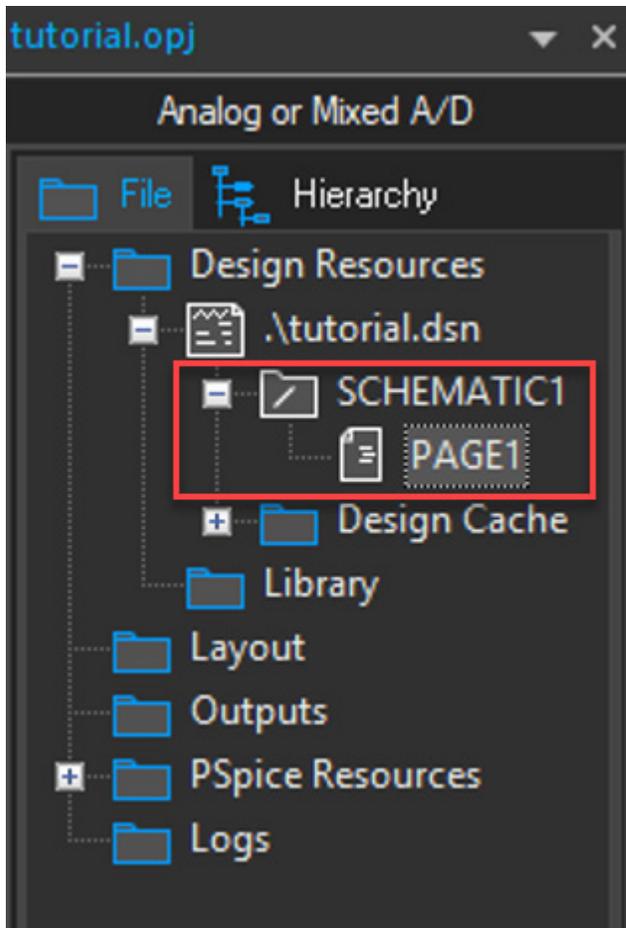
6. Click *OK*.

The tutorial project is created. In the project manager window, a design file, `tutorial.dsn` is created. Below the design file, a schematic folder with the name `SCHEMATIC1` is created. The folder has a schematic page named `PAGE1`.

PSpice for TI: Getting Started

Working With a New Design

Figure 2-2 Project Manager window



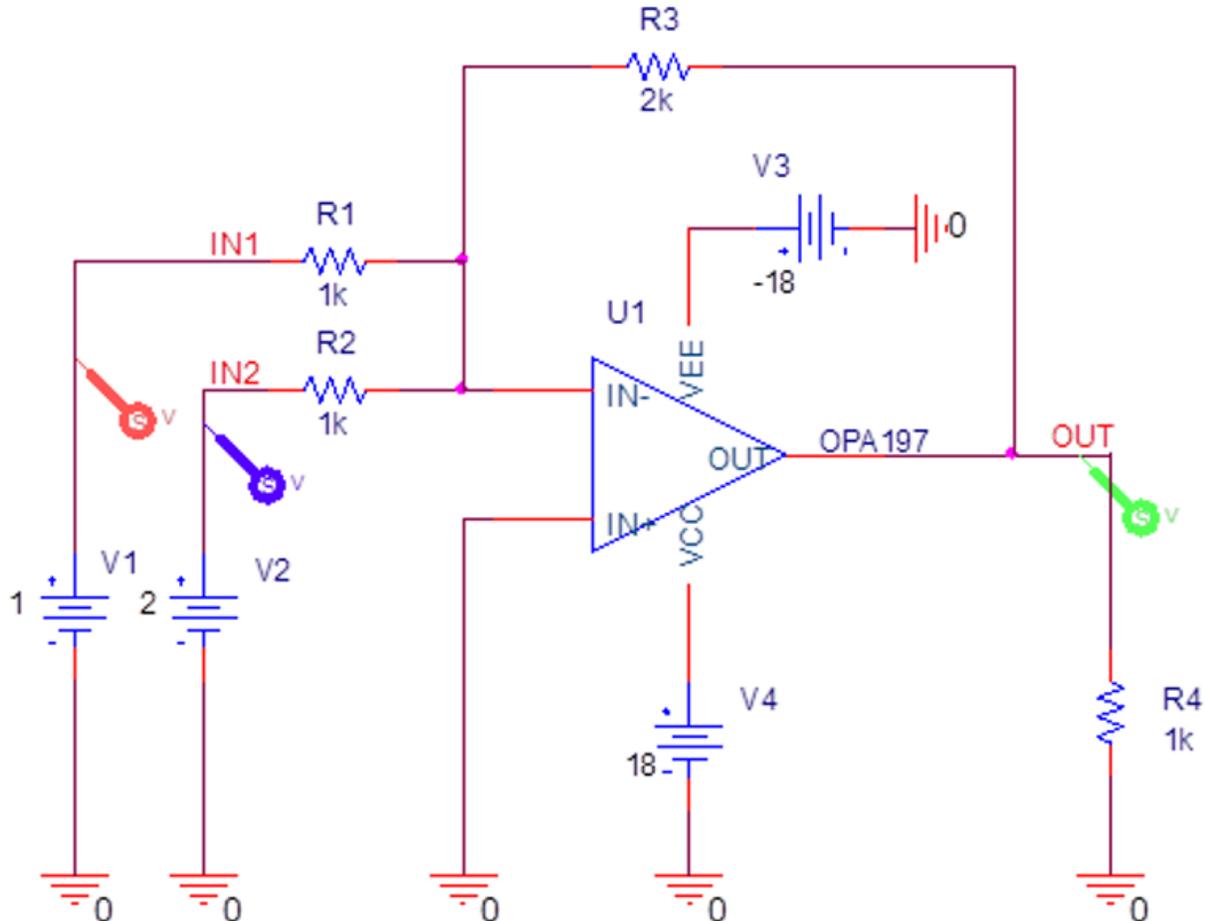
Creating a Design

You will now create the design of the Inverting Summing Amplifier as shown in the following figure.

PSpice for TI: Getting Started

Working With a New Design

Figure 2-3 Schematic design for the inverting summing amplifier



You will also perform the following tasks in order to create the schematic design as shown above:

- [Placing the Op-Amp](#)
- [Placing DC Voltage Sources using Modeling Application](#)
- [Placing Ground Symbols](#)
- [Placing the Resistors](#)
- [Connecting the Components](#)
- [Assigning Net Aliases](#)
- [Placing Markers](#)

PSpice for TI: Getting Started

Working With a New Design

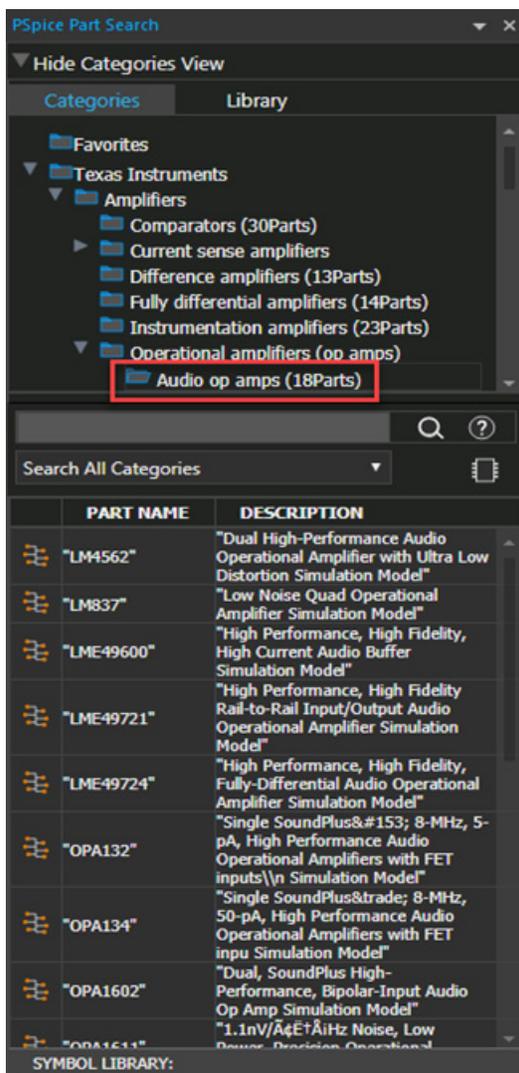
Placing the Op-Amp

To place *OPA197*, do the following:

1. Select *Place – PSpice Component – Search* or click the *Launch PSpice Part Search* icon () on PSpice toolbar.

The *PSpice Part Search* window opens.

Figure 2-4 PSpice Part Search window



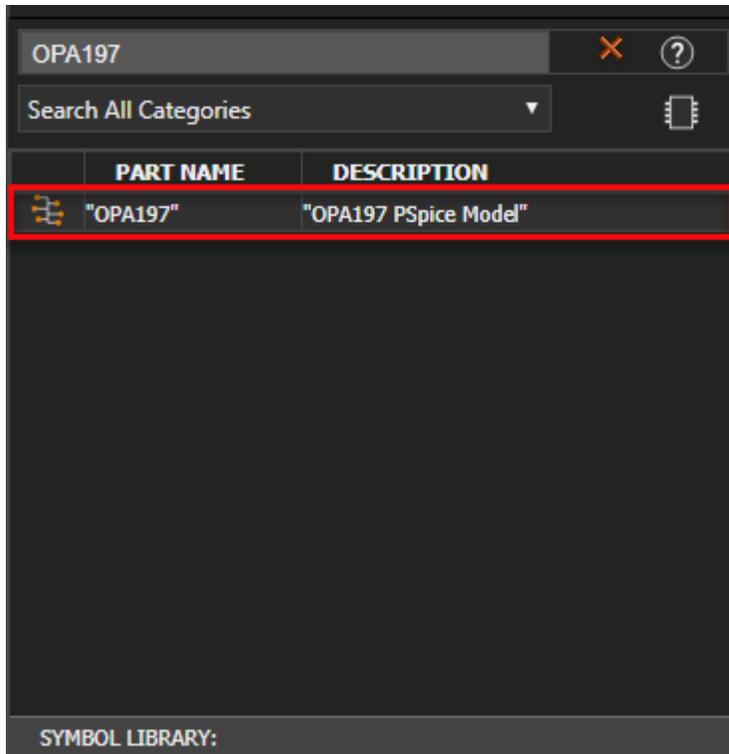
2. Search for *OPA197* in the part search field, and click  icon.

OPA197 appears in the search result.

PSpice for TI: Getting Started

Working With a New Design

3. Double-click *OPA197* and click the schematic page to place the part.



4. Right-click and select *End Mode* or press *Esc*.

Placing DC Voltage Sources using Modeling Application

1. Select *Place – PSpice Component – Modeling Application* or click the *Launch Modeling Application* icon () on the PSpice toolbar.

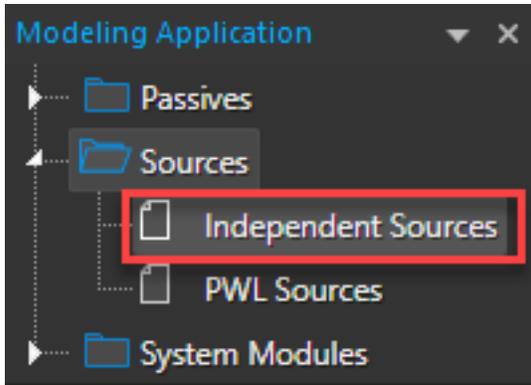
The *Modeling Application* pane opens.

2. Select *Sources – Independent Sources*.

PSPICE for TI: Getting Started

Working With a New Design

Figure 2-5 Modeling Application window



3. Click *DC*.
4. Select *Voltage* and *Ideal DC*.
5. Select *DC Voltage* as 1 and click *Place* to place the voltage source.

PSpice for TI: Getting Started

Working With a New Design

Figure 2-6 Independent Sources window

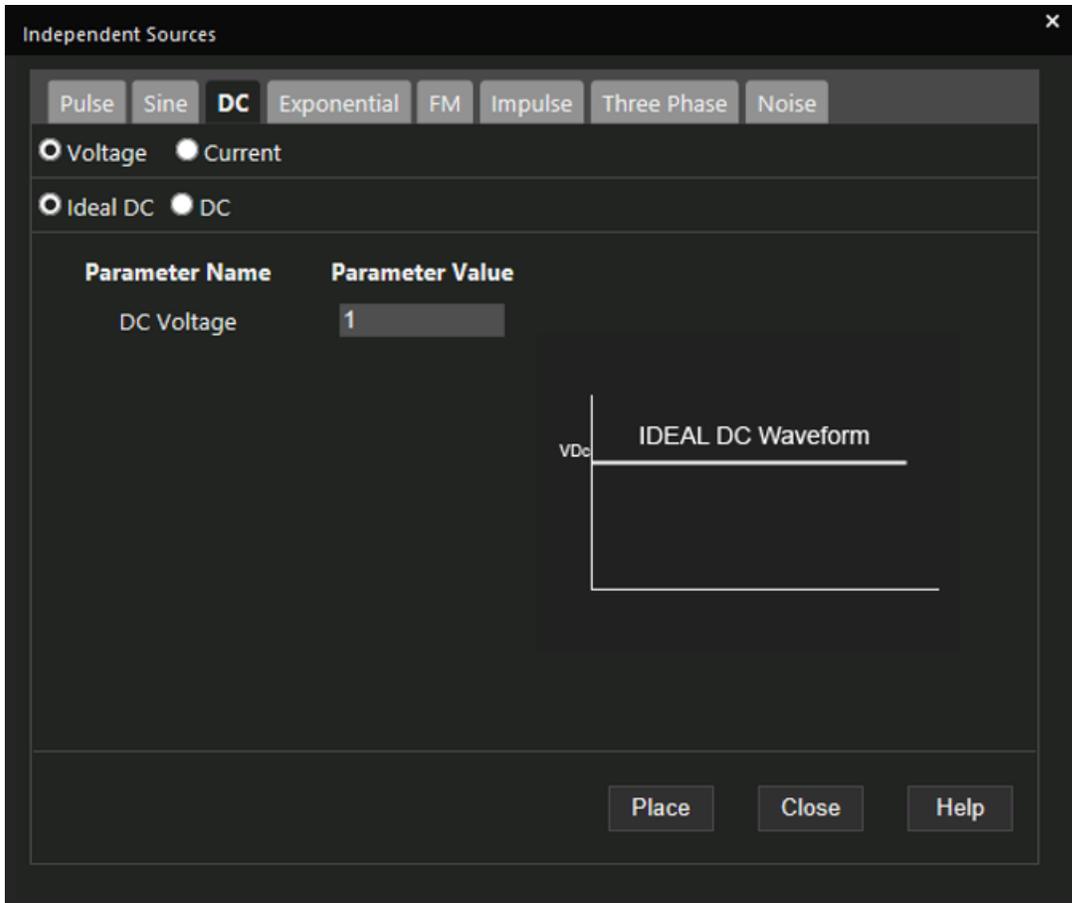
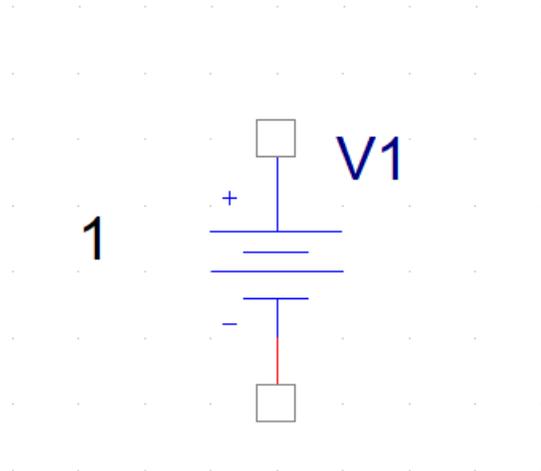


Figure 2-7 Voltage V1 on the schematic page



6. Similarly, place 3 more DC voltage sources with the following voltage values:

Voltage Name	Voltage Value
V2	2V
V3	-18V
V4	18V

7. To save the design, select *File – Save* or press *CTRL+S*.

Placing Ground Symbols

1. Select *Place – PSpice Component – PSpice Ground*.
2. Click the schematic page to place the part.
3. Right-click and select *End Mode* or press *Esc*.

Placing the Resistors

1. Select *Place – PSpice Component – Resistor*.

The part symbol is attached to a pointer.

2. Click the schematic page to place the resistor.

PSpice for TI: Getting Started

Working With a New Design

3. To stop placing the resistor, right-click and select *End Mode* or press *Esc*.
4. To change the value, double-click the value of the resistor.

The *Display Properties* window opens.

Figure 2-8 Display Properties window

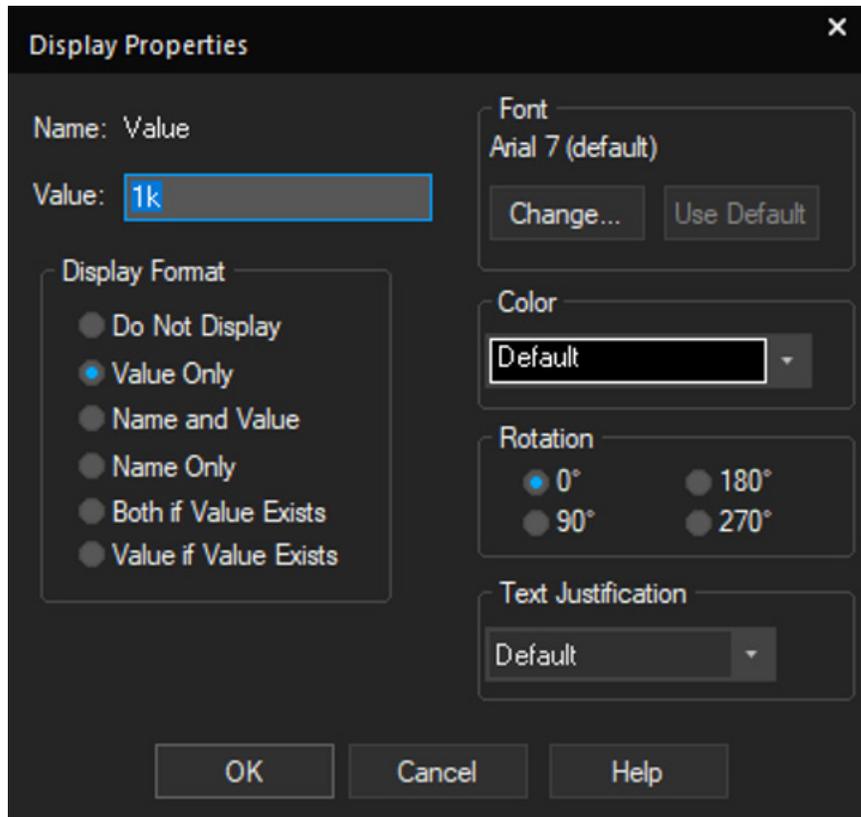
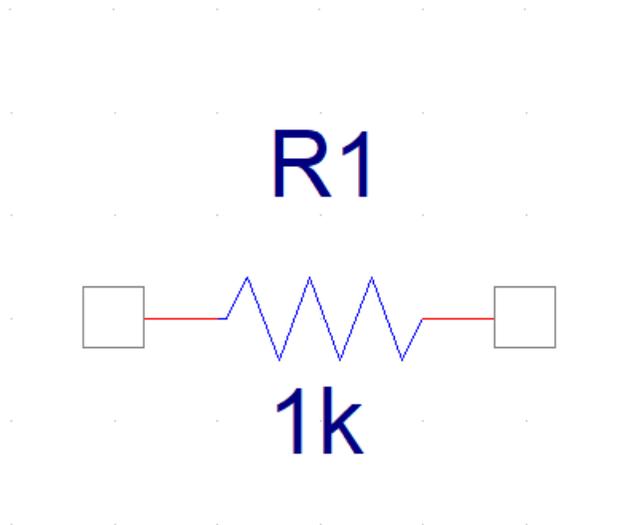


Figure 2-9 Resistance R1 on the schematic page



5. Similarly, place 3 more resistances, and specify the value shown in the following figure.

Resistance Name	Resistance Value
R2	1K
R3	2K
R4	1K

6. Click *OK*.

Connecting the Components

1. Select *Place – Wire*, press *W*, or click the Place wire icon () on the *Draw Electrical* toolbar.

The pointer changes to a crosshair.

2. Click any valid connection point to end the wire.

Assigning Net Aliases

A net is not required to have an alias, but by using an alias, you can establish connectivity. To create a net alias, do the following;

PSPICE for TI: Getting Started

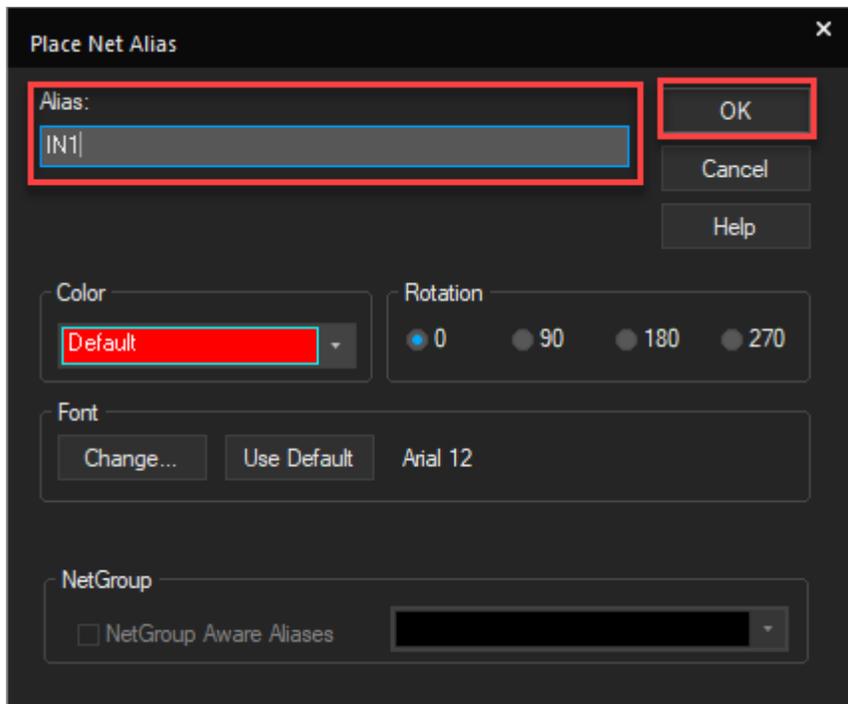
Working With a New Design

1. Select *Place - Net Alias*.

The *Place Net Alias* dialog opens.

2. Enter the net alias text in the *Alias* dialog box and click *OK*.

Figure 2-10 Place Net Alias window



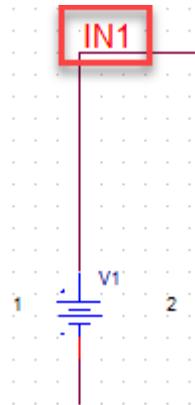
A rectangle representing the net alias is attached to the pointer.

3. Use the mouse to move the net alias and click the left mouse button on the wire to place the net alias.

PSpice for TI: Getting Started

Working With a New Design

The net alias appears in the selection color.

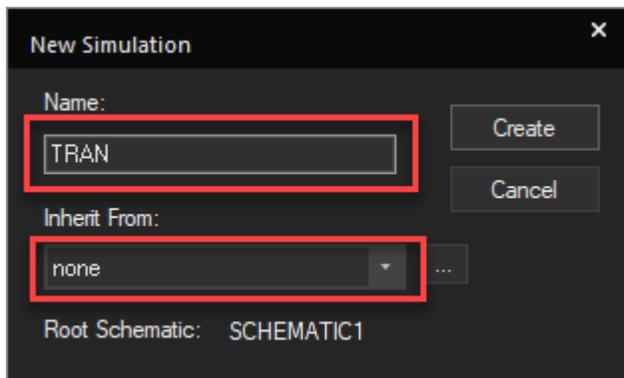


4. Similarly, place the net aliases `IN2` and `OUT`.
5. Press *Esc* to dismiss the net alias tool.

Creating a New Simulation Profile

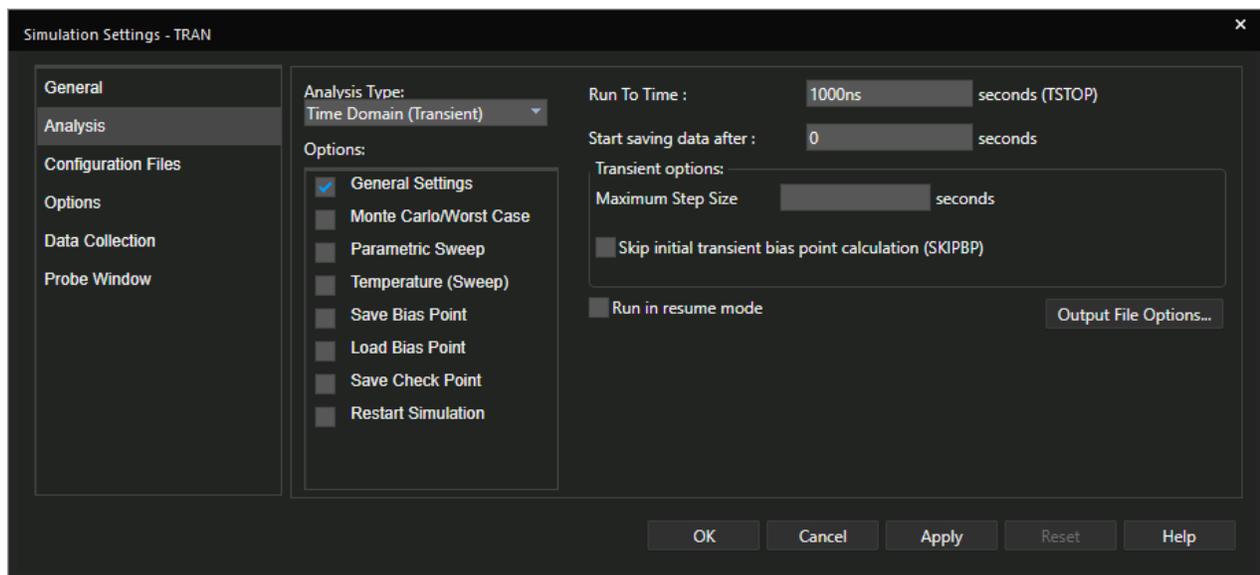
1. Select *PSpice – New Simulation Profile* or click the *New Simulation Profile* icon () on the PSpice toolbar.
The New Simulation dialog box opens
2. Specify the name of the new simulation profile as `TRAN`.
3. In the *Inherit From* drop-down list, ensure that `none` is selected and click *Create*.

Figure 2-11 New Simulation window



The *Simulation Setting* dialog box appears with the *Analysis* tab selected.

Figure 2-12 Simulation Settings - Analysis tab



PSpice for TI: Getting Started

Working With a New Design

4. Retain all the default settings and click *OK* to close the dialog box.

Placing Markers

To visualize the circuit behavior and determine the validity of your circuit design, you can plot the output waveforms in the Probe window. By analyzing the output waveforms, you can evaluate your circuit for performance.

For PSpice for TI to display output waveforms in the Probe window, you need to place markers in your circuit design in Capture to indicate the points where you want to see simulation waveforms displayed in PSpice for TI.

Markers can be placed:

- before simulation to limit results written to the waveform data file, and automatically display those traces in the active Probe window.
- during or after simulation, to automatically display traces in the active Probe window.

To add markers:

- Choose *PSpice – Markers* or use the icons provided on the PSpice toolbar.

Note: To view the markers in the simulation results, the schematic design must be open.

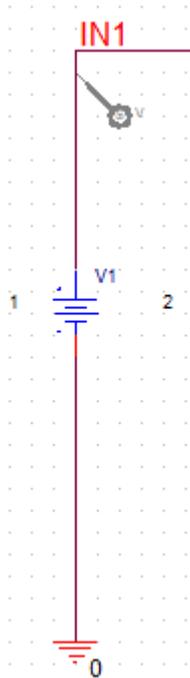
You will now add Voltage markers to view the output waveforms in the Probe window. To do so:

1. Select *PSpice – Markers – Voltage Level* or click the *Voltage/Level Marker* icon () on the PSpice toolbar.

PSpice for TI: Getting Started

Working With a New Design

2. Place the first marker at the 1 Volt power supply V1.



3. Place the second marker at the 2 Volt power supply V2.



PSpice for TI: Getting Started

Working With a New Design

- Place the third marker at the output of the op-amp.



Running the simulation

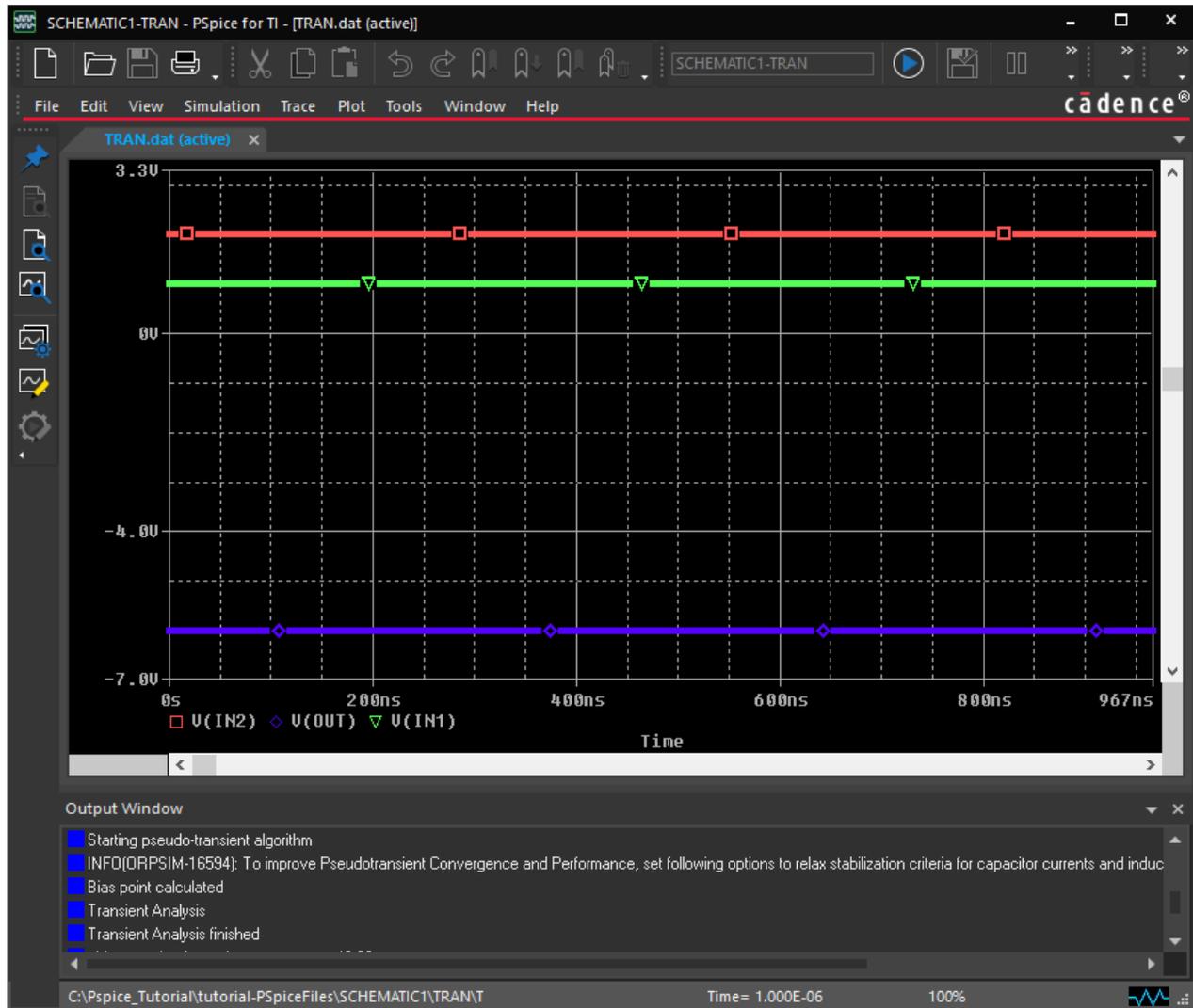
To simulate the design, choose *PSpice – Run* or click the *Run PSpice* icon () on the PSpice toolbar.

The plot window opens with simulation result.

PSpice for TI: Getting Started

Working With a New Design

Figure 2-13 Output Waveform - Probe window

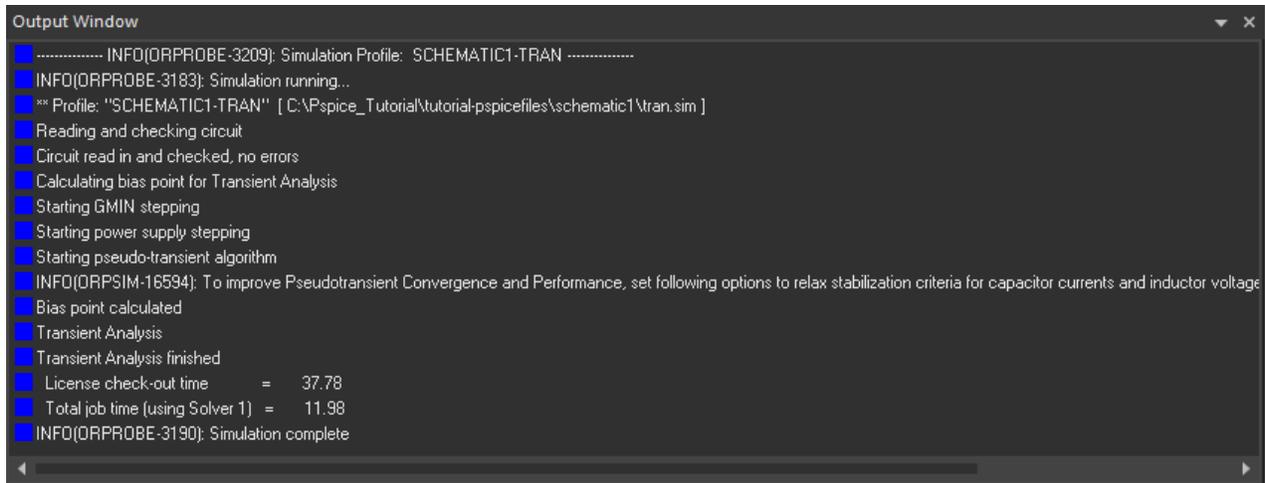


By default, you will see the output window as shown in the following figure with simulation complete status:

PSpice for TI: Getting Started

Working With a New Design

Figure 2-14 Output Window



To view the probe cursor window, select *View – Cursor Window* or click the *Toggle Cursor* icon () on the *Probe Window* toolbar.

The *Probe Cursor* window opens.

Figure 2-15 Probe Cursor window



The *Probe Cursor* window shows the value of the output voltage corresponding to the values of the input voltages and resistances according to the formula of the inverting summing amplifier:

$$V_{out} = -R_3 \left(\frac{V_1}{R_1} + \frac{V_2}{R_2} \right)$$