cādence®

PSpice for TI: Getting Started

Product Version 17.40 - 2020
June 2020

Contents

1

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.

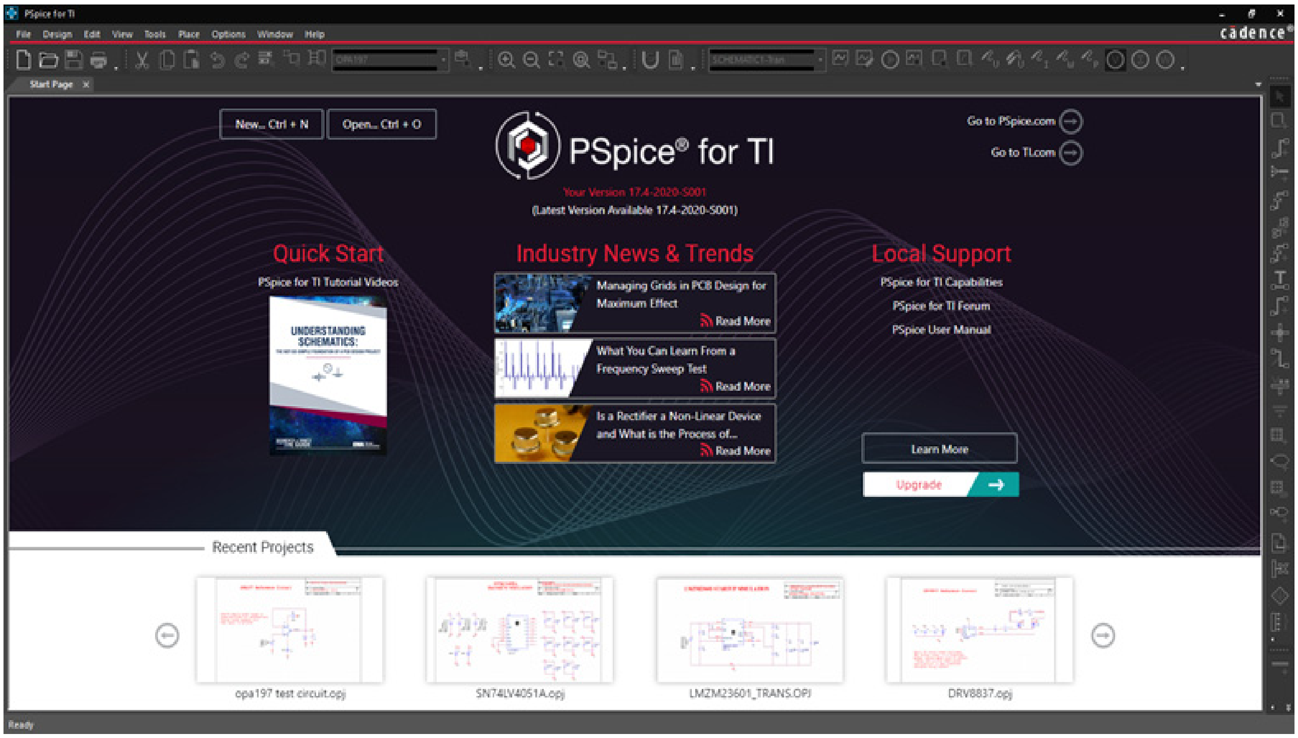
Opening Sample Reference Project

To open a sample reference project, do the following:

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

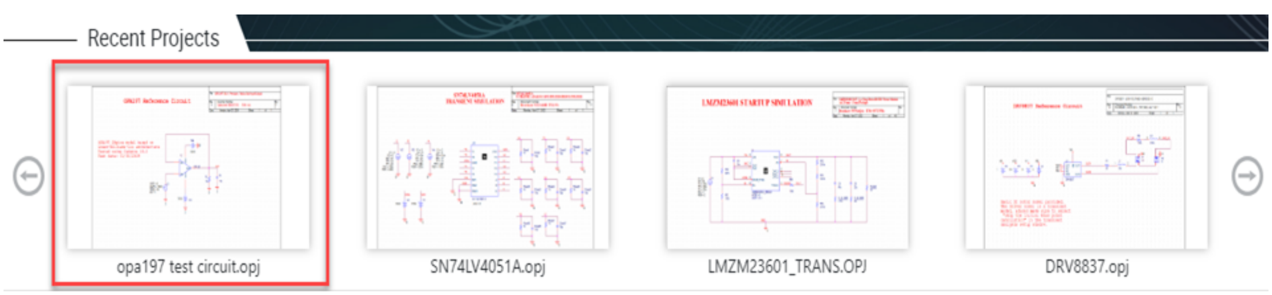
   Sample projects will be seen on the first invocation of PSpice for TI. You can choose sample projects from the *Recent Projects* section and whatever is your last opened project, it will start showing up in this section.

Figure 1-1 PSpice for TI start page



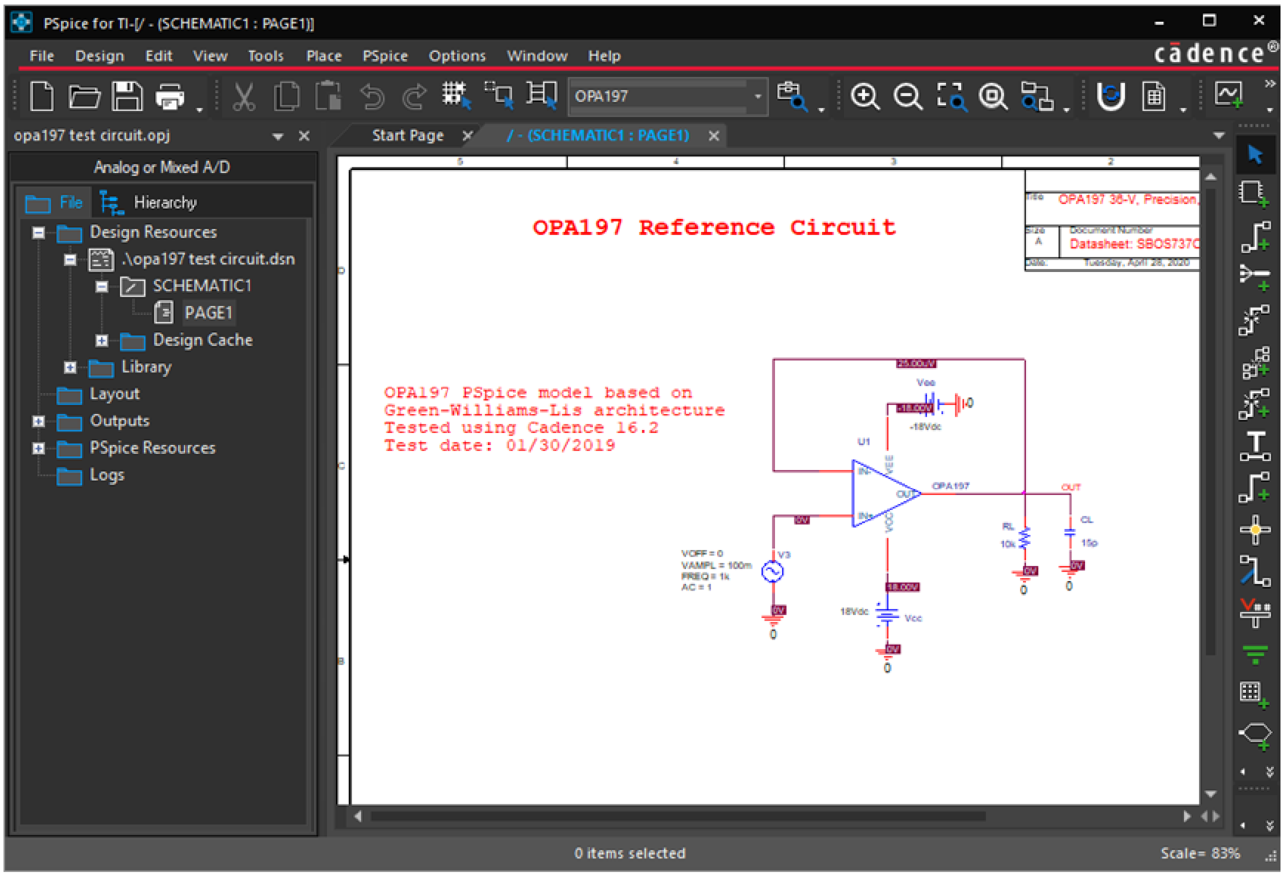
2. Click *opa197 test circuit.opj* from *Recent Projects* section to open the project.

Figure 1-2 Recent Projects List



3. Open `PAGE1` under `SCHEMATIC1`.

Figure 1-3 OPA197 Reference Circuit schematic page



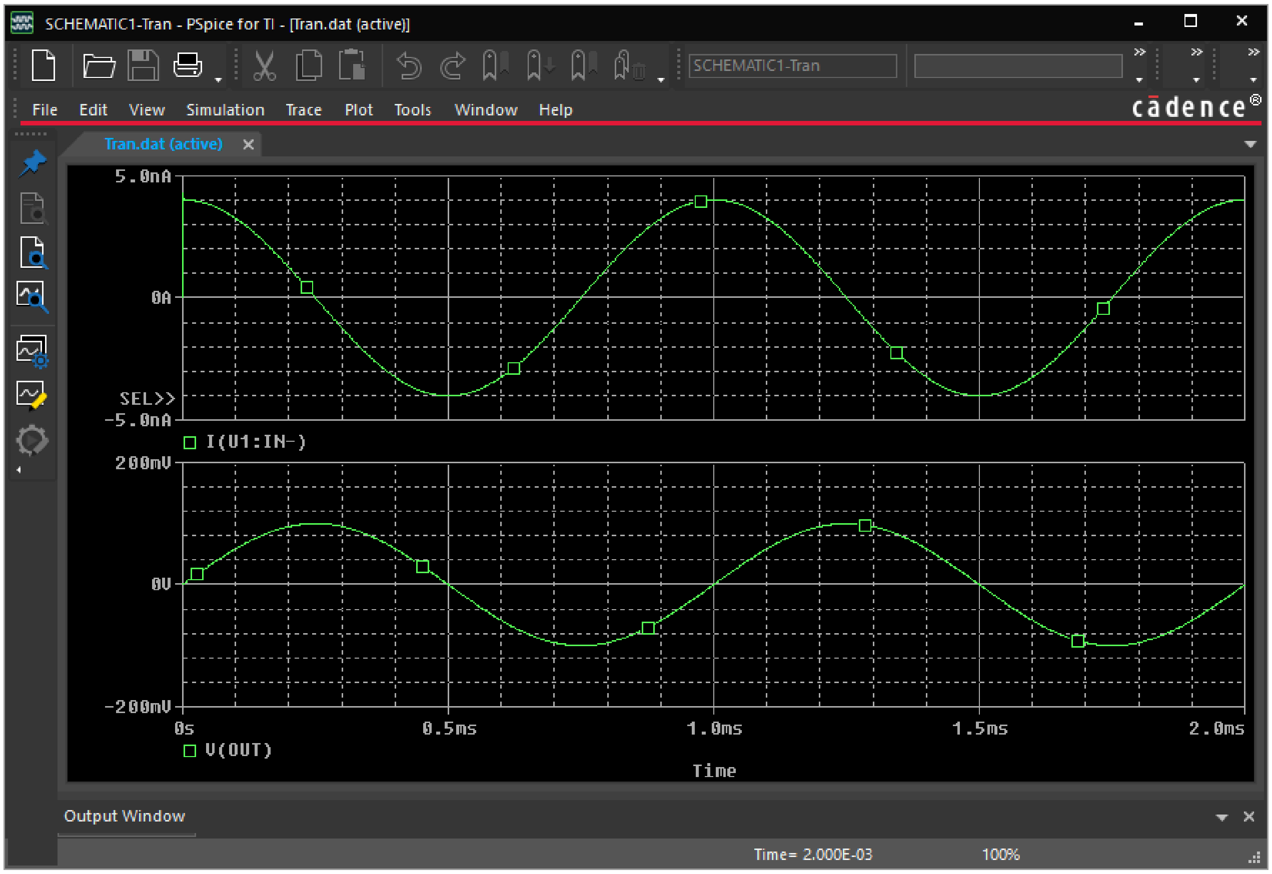
Simulating a Design

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.

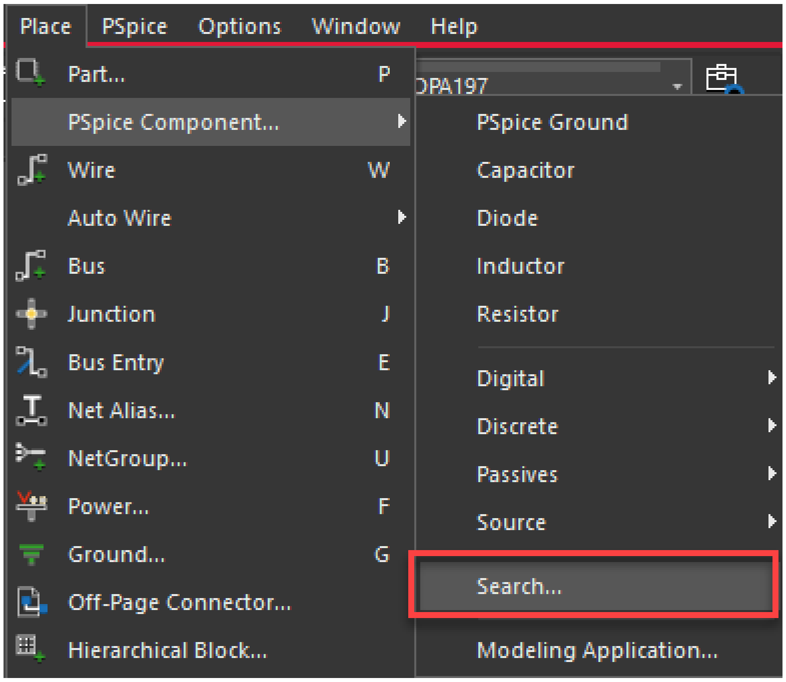
Figure 1-4 Output Waveform - Probe window

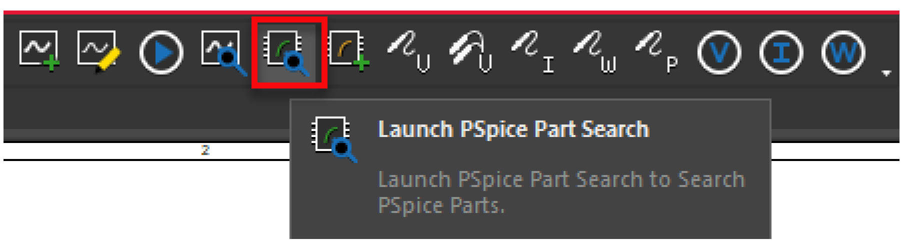


Searching for a TI Part

To search for a TI part, do the following:

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

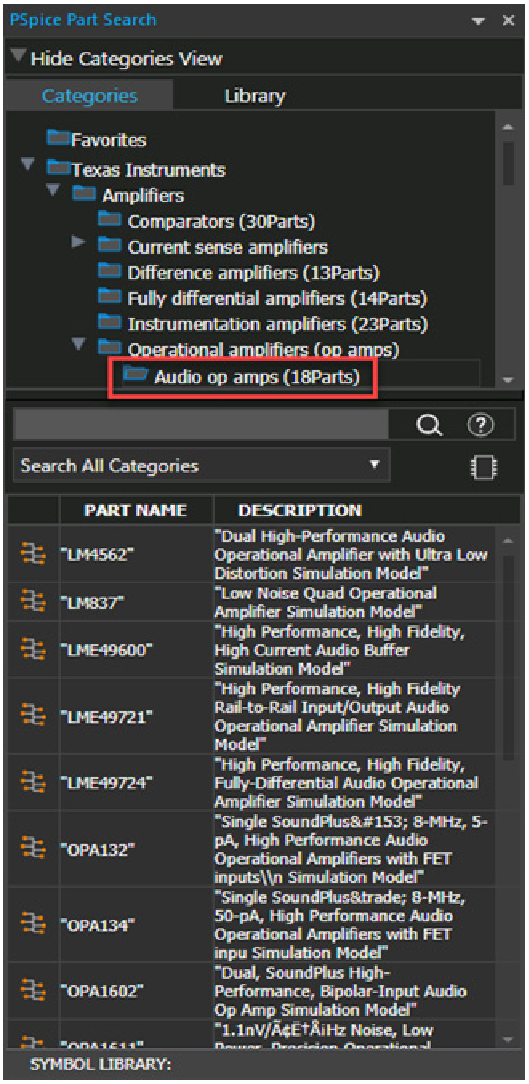




The *PSpice Part Search* panel opens. You can browse for parts under various categories as per design needs.

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.

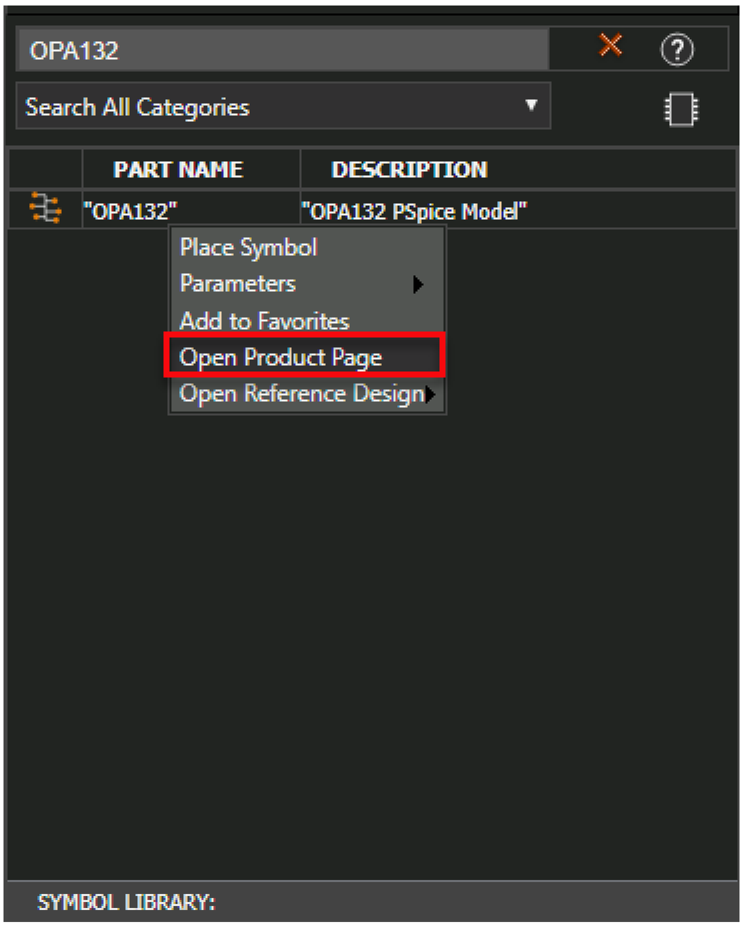
Figure 1-5 PSpice Part Search window



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

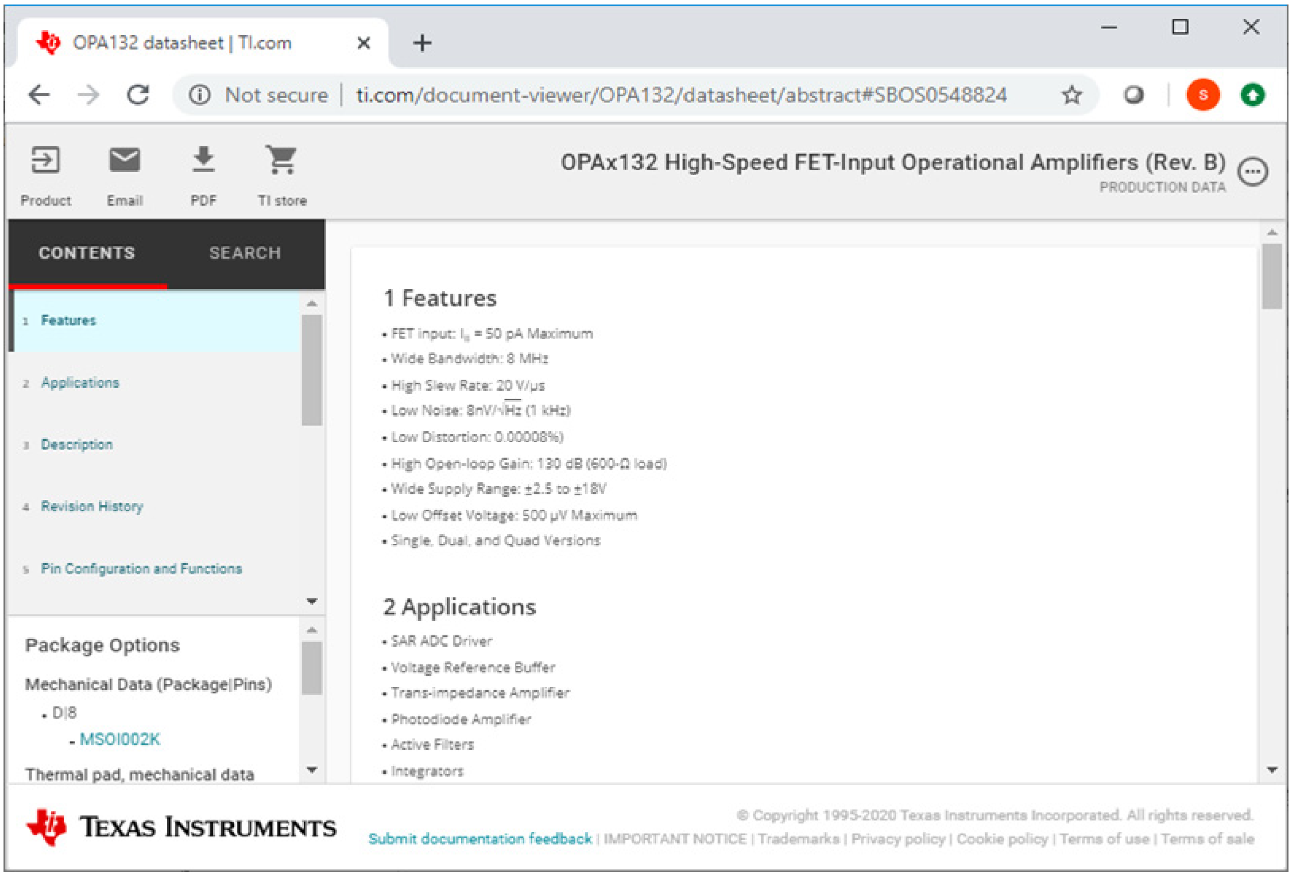
Accessing Design Resources for a Particular TI Part

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



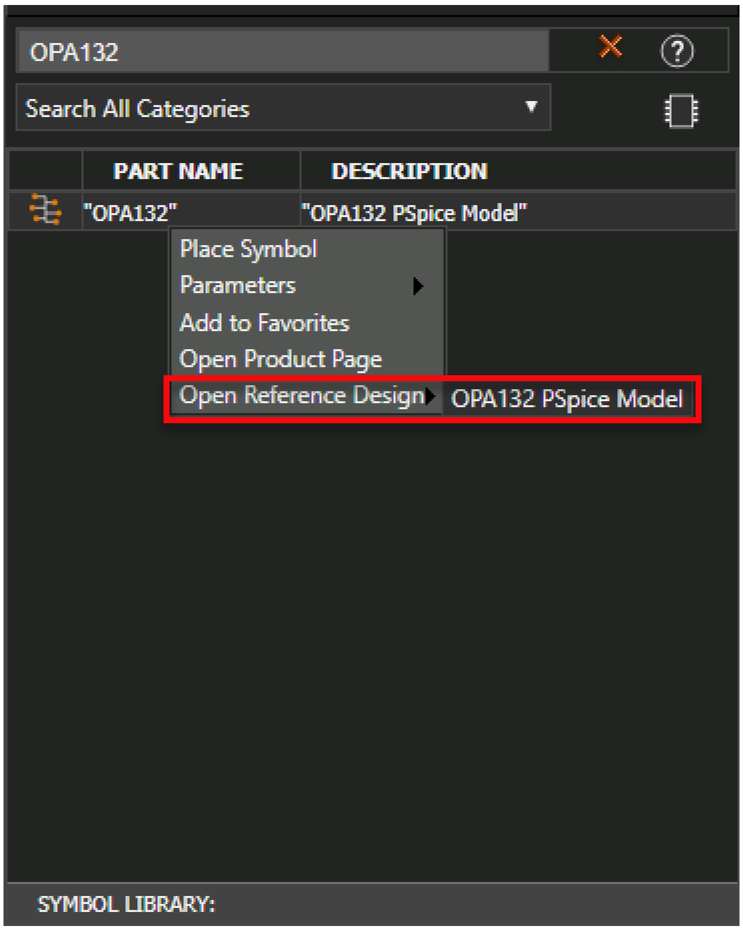
The OPA132 product page will open from `ti.com` on your web browser.

Figure 1-6 OPA132 Product Page



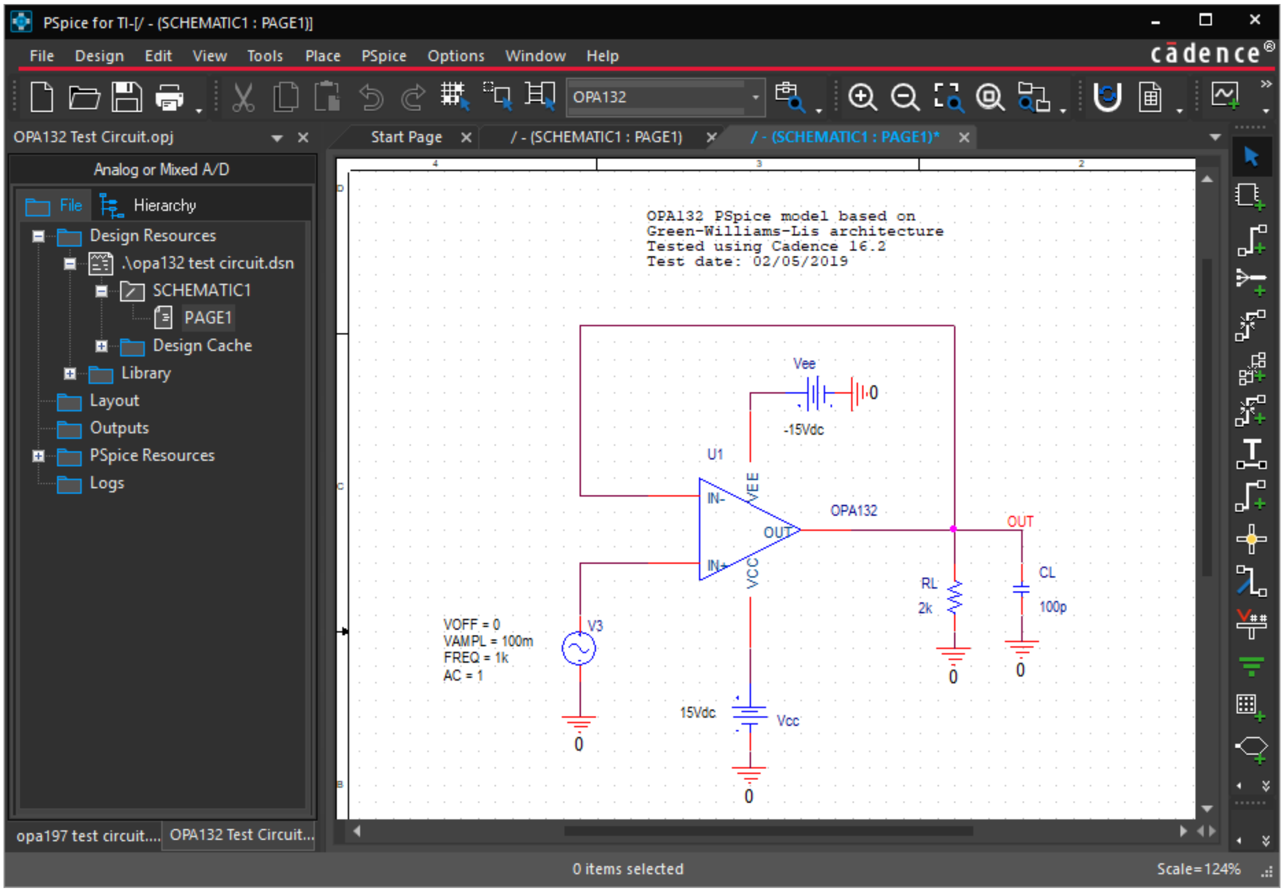
Opening a Reference Design

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.

Figure 1-7 OPA132 Test Circuit schematic page

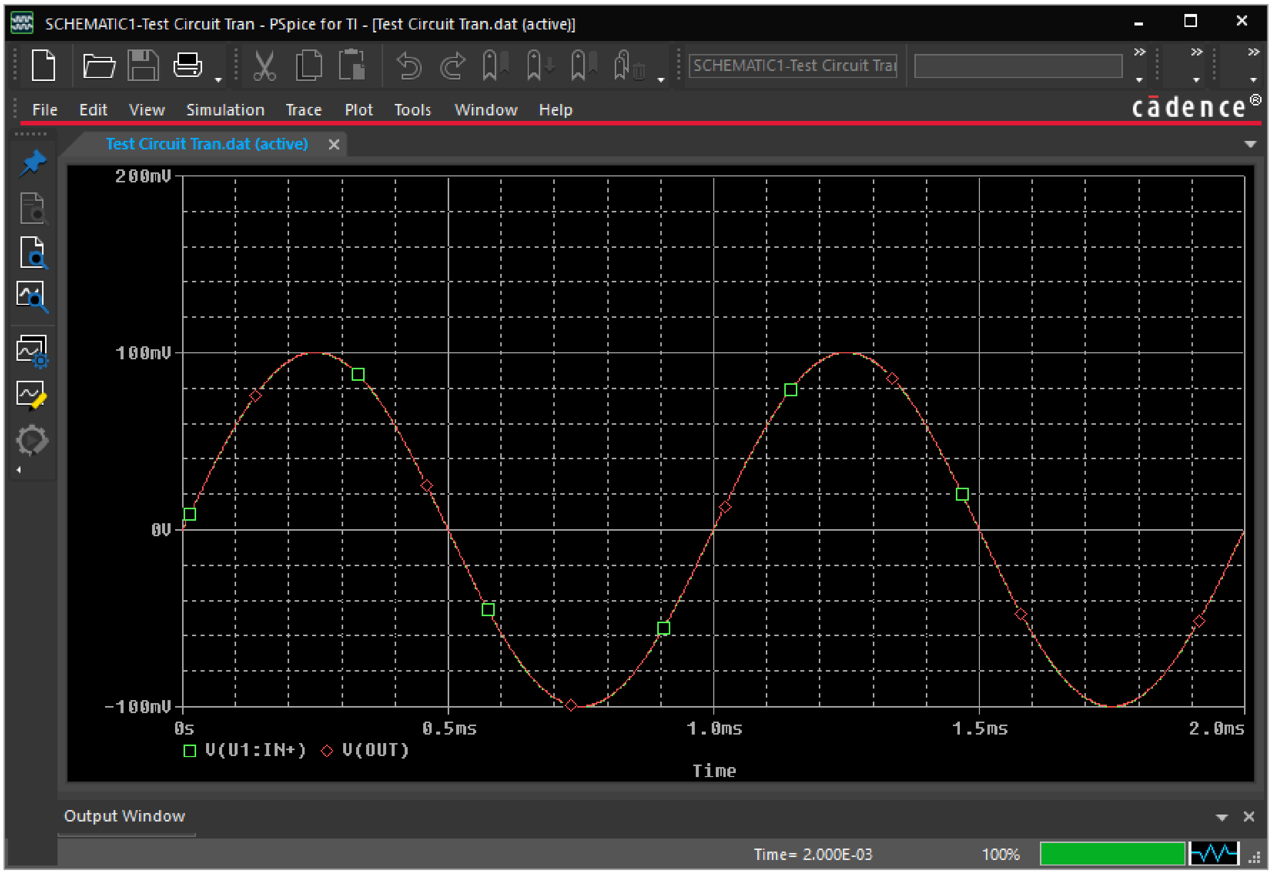


Simulating a Reference Design

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.

Figure 1-8 Output Waveform - Probe window



2

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.

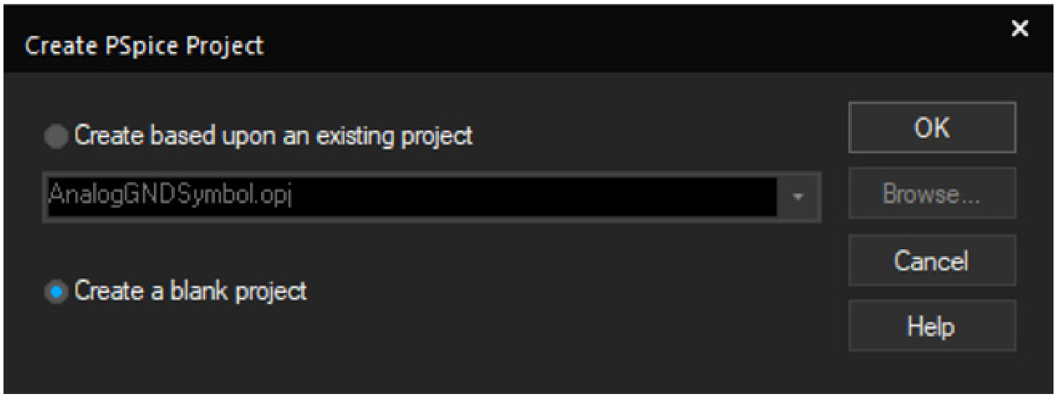
To create a new project, do the following:

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*.

Figure 2-1 Create PSpice Project window

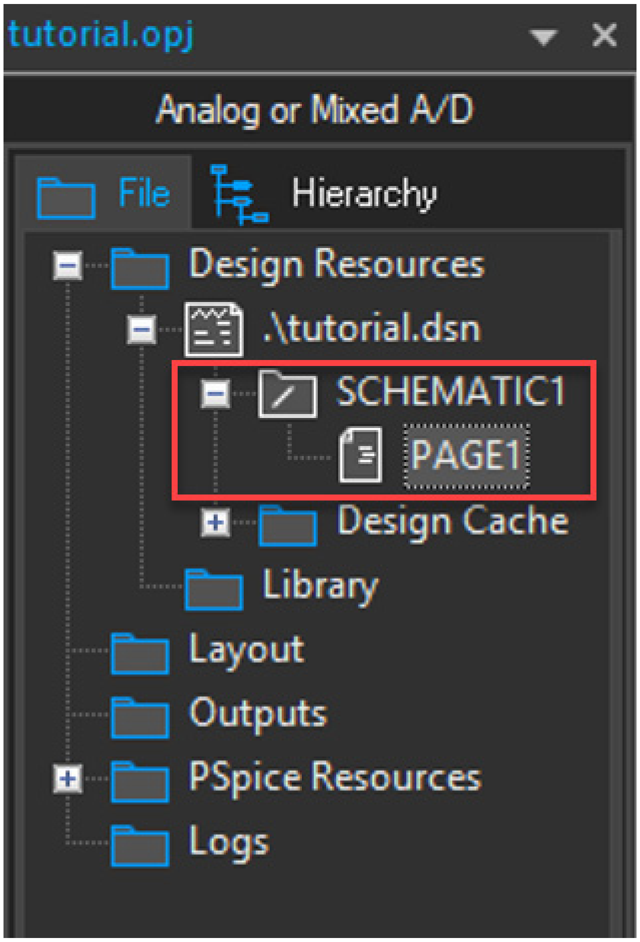


6. Click *OK*.

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`.

Figure 2-2 Project Manager window

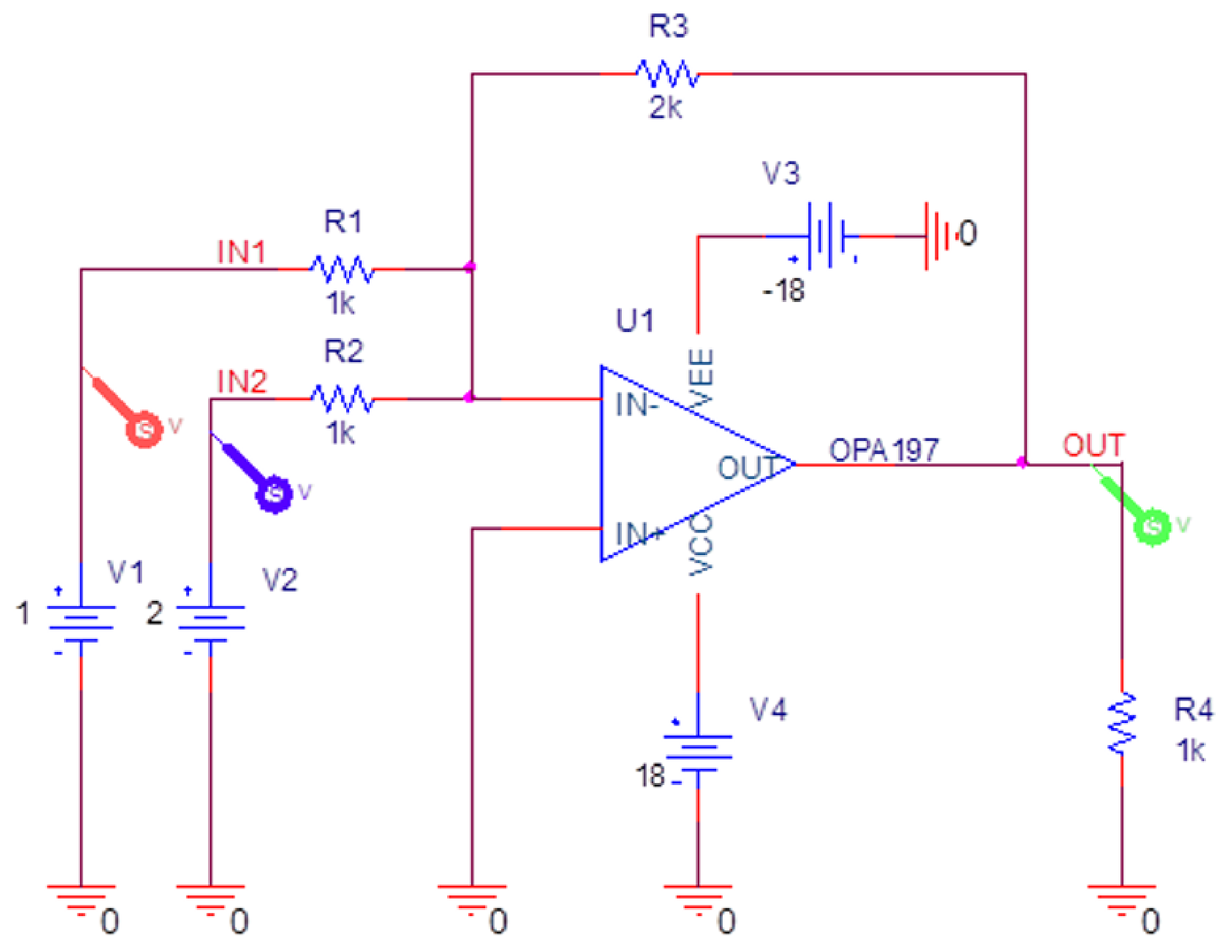
.



Creating a Design

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

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

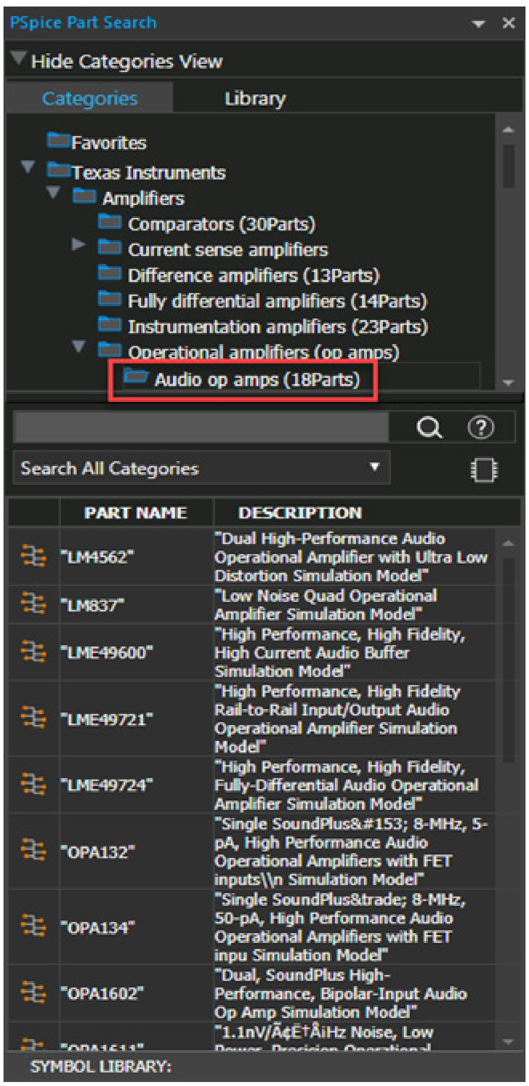
Placing the Op-Amp

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.

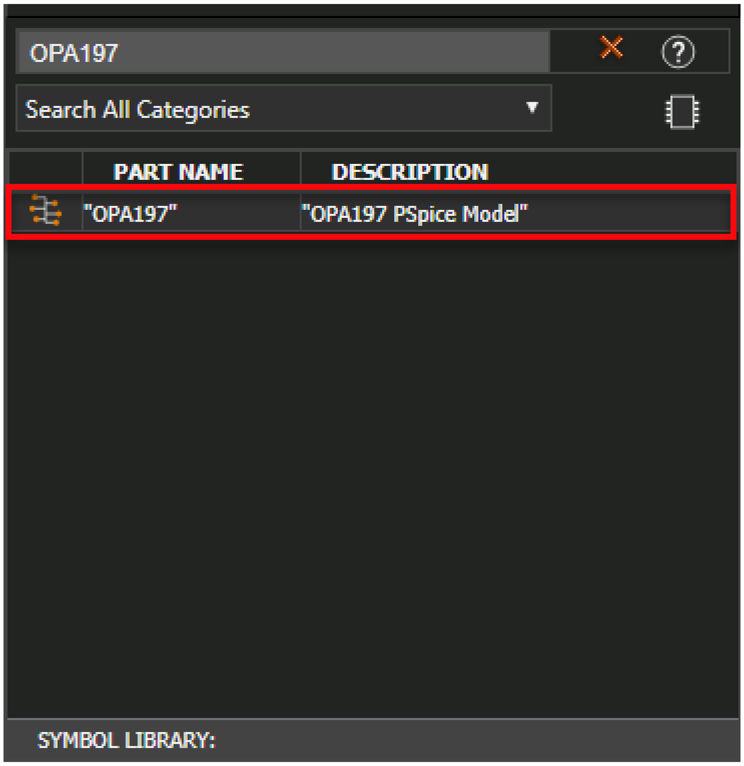
   Figure 2-4 PSpice Part Search window



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

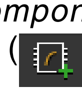
   OPA197 appears in the search result.

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



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

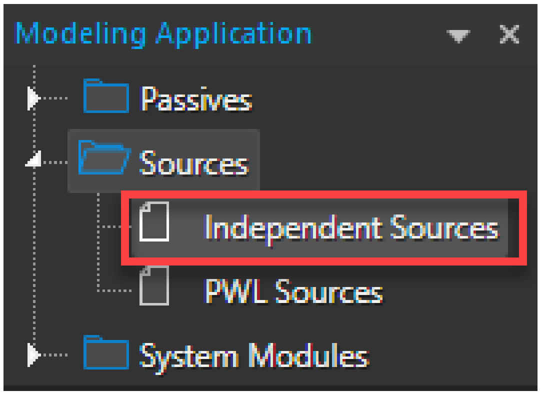
Placing DC Voltage Sources using Modeling Application

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.*

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.

Figure 2-6 Independent Sources window

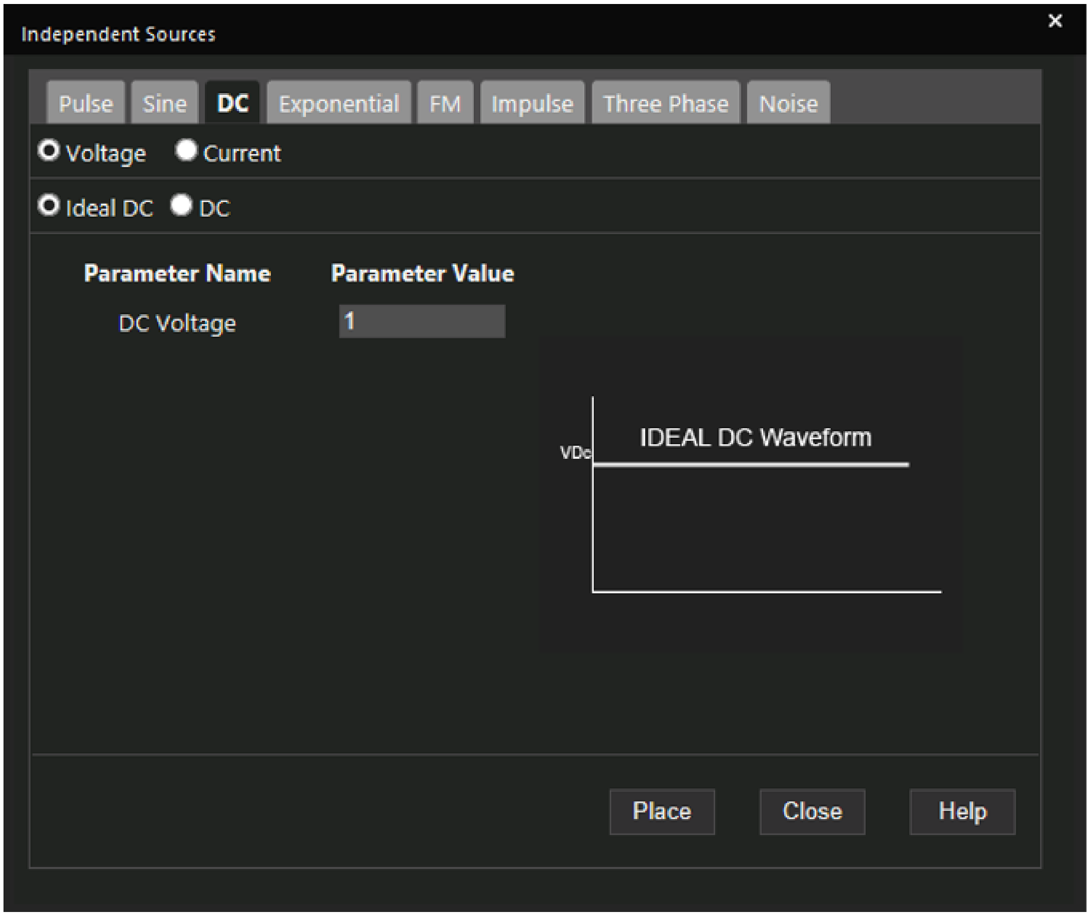
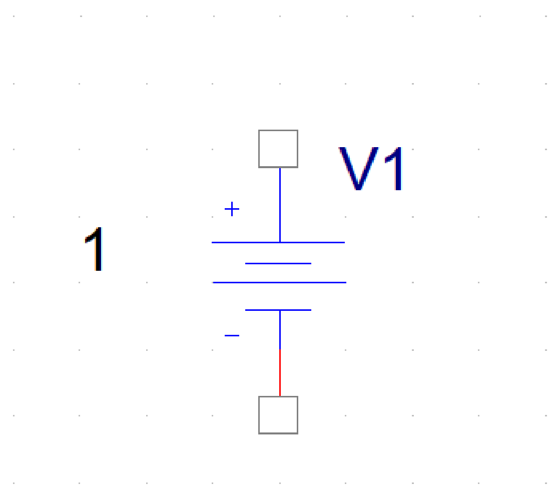


Figure 2-7 Voltage V1 on the schematic page



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.

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

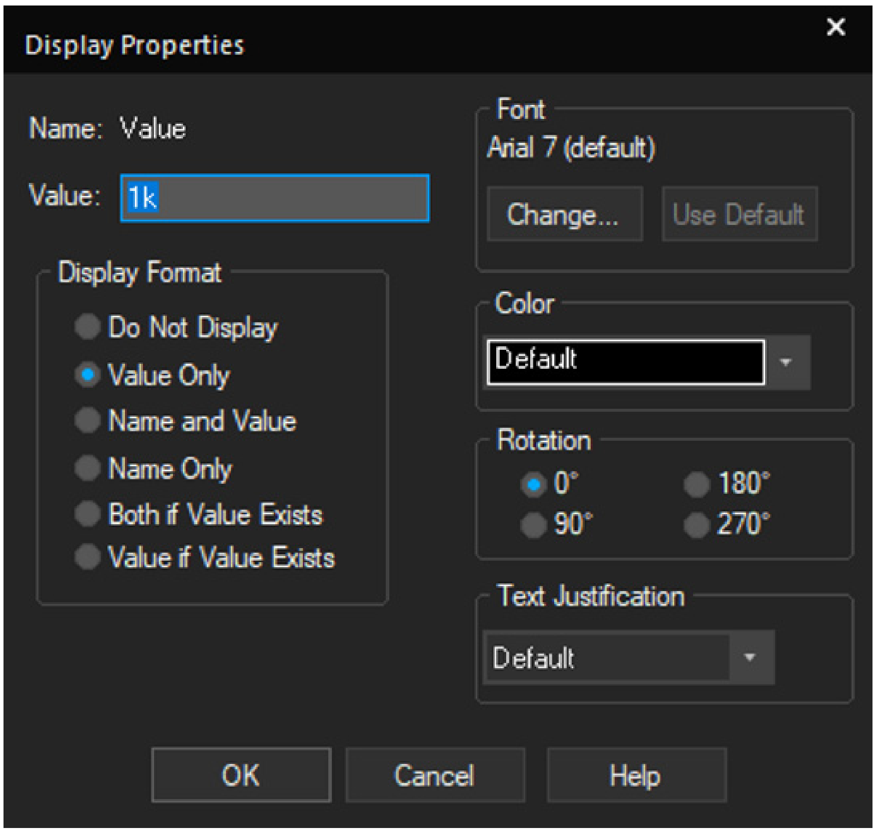
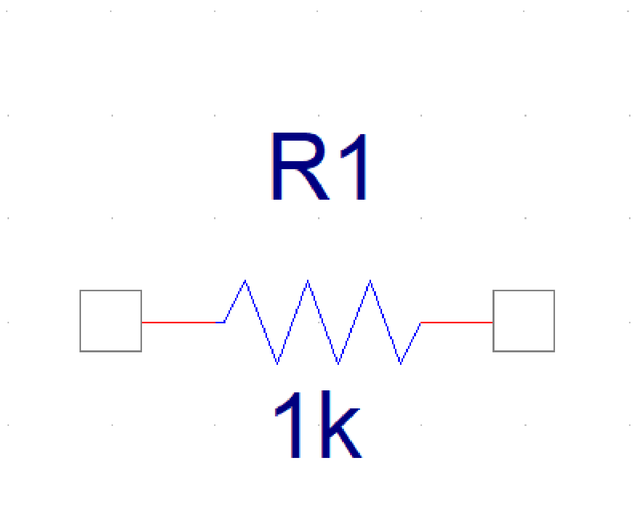


Figure 2-9 Resistance R1 on the schematic page

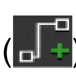


5. Similarly, place 3 more resistances, and specify the value shown in the following figure.

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

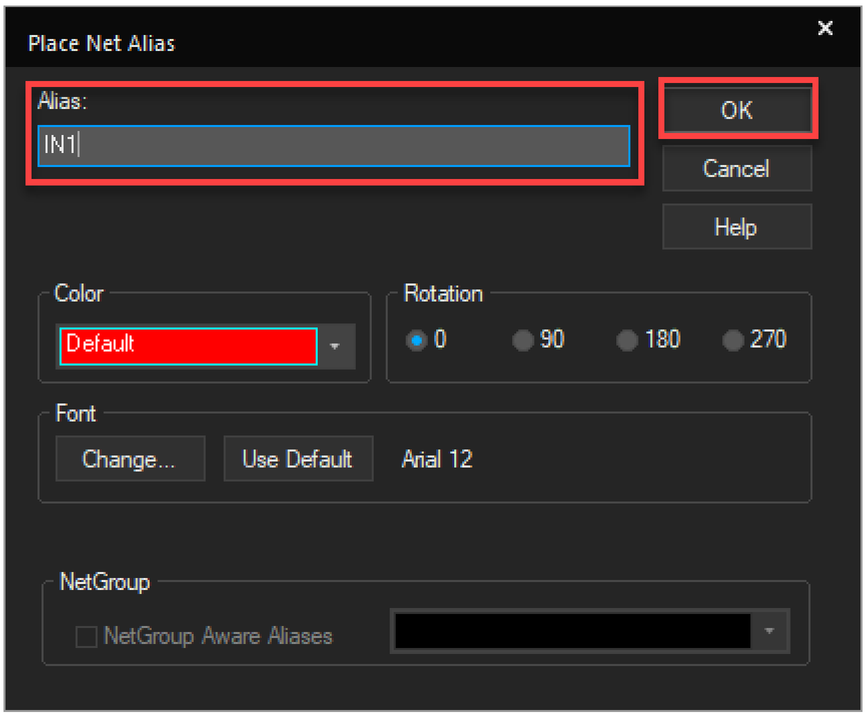
A net is not required to have an alias, but by using an alias, you can establish connectivity. To create a net alias, do the following;

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*.

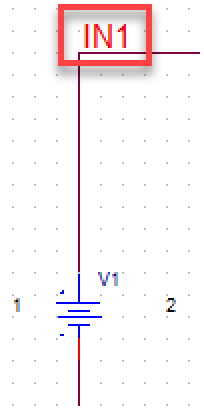
   Figure 2-10 Place Net Alias window

   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.

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

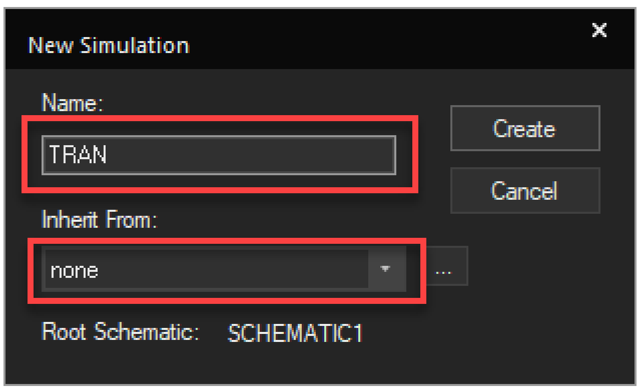
1. Select *PSpice – New Simulation Profile* or click the *New Simulation Profile* icon () on the PSpice toolbar.

   The New Simulation dialog box opens

2. Specify the name of the new simulation profile as `TRAN`.

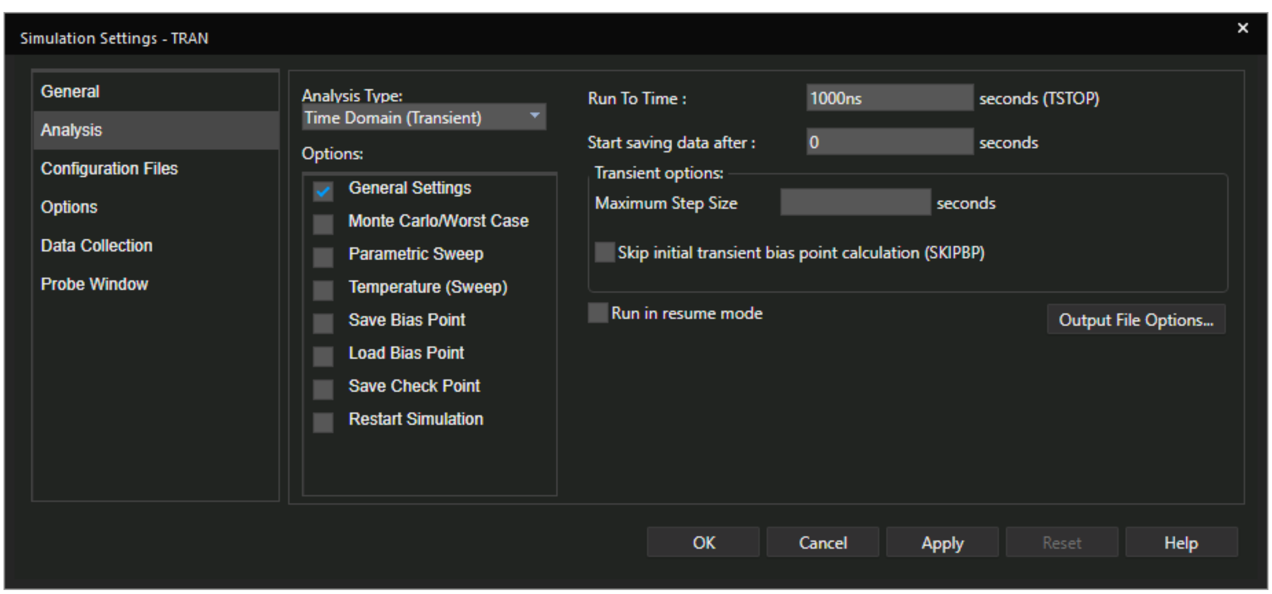
3. In the *Inherit From* drop-down list, ensure that `none` is selected and click *Create*.

Figure 2-11 New Simulation window



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

Figure 2-12 Simulation Settings - Analysis tab



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:

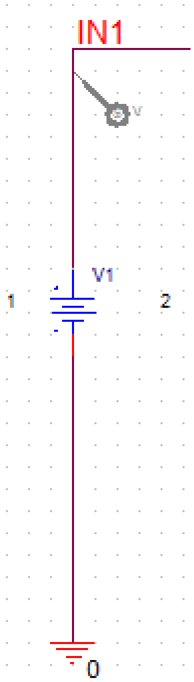
- Choose *PSpice – Markers* or use the icons provided on the PSpice toolbar.

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

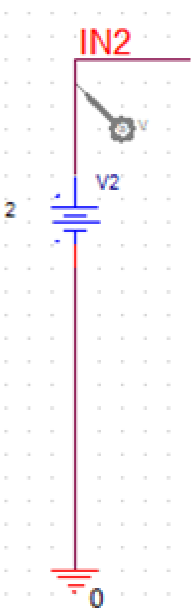
You will now add Voltage markers to view the output waveforms in the Probe window. To do so:

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

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



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



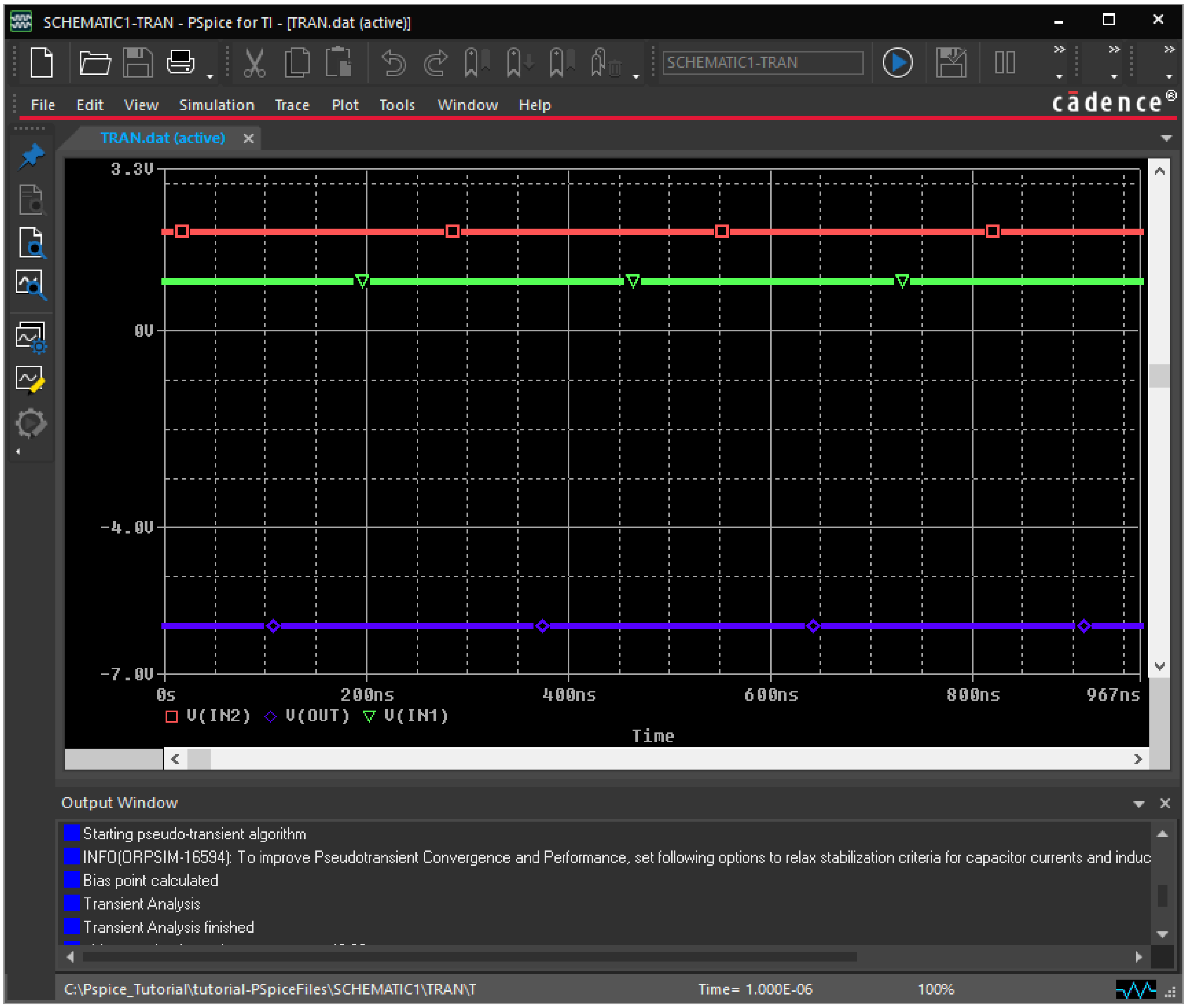
4. Place the third marker at the output of the op-amp.

Running the simulation

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

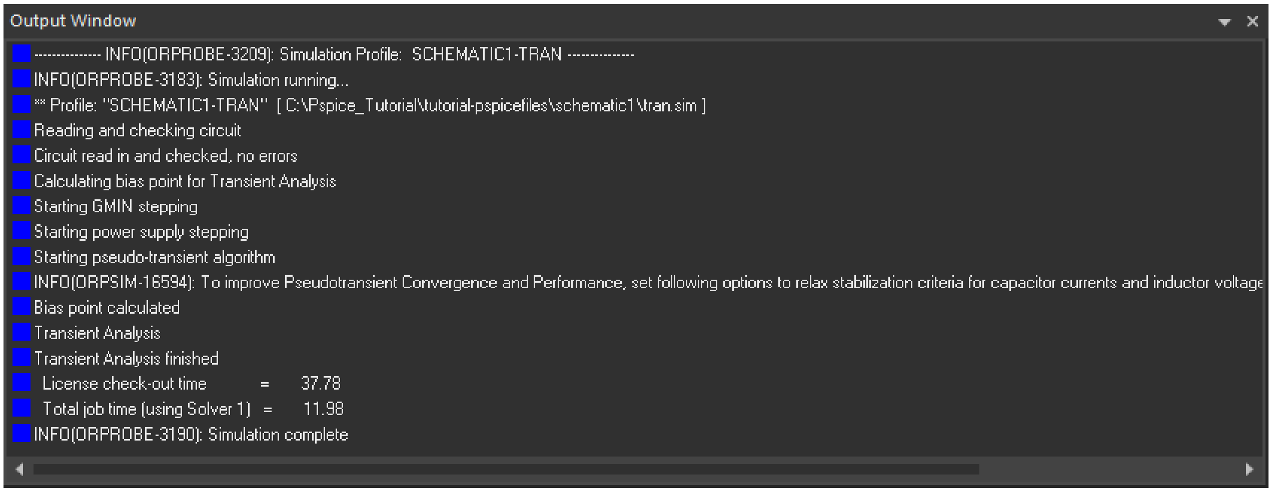
The plot window opens with simulation result.

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:

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

Trace Color	Trace Name	Y1	Y2	Y1 - Y2		Y1(Cursor1) - Y2(Cursor2)		0.000		
	X Values	0.000	0.000	0.000		Y1 - Y1(Cursor1)	Y2 - Y2(Cursor2)	Max Y	Min Y	Avg Y
CURSOR 1,2	V(IN2)	2.0000	2.0000	0.000		0.000	0.000	2.0000	2.0000	2.0000
	V(OUT)	-5.9999	-5.9999	0.000		-7.9999	-7.9999	-5.9999	-5.9999	-5.9999
	V(IN1)	1.0000	1.0000	0.000		-1.0000	-1.0000	1.0000	1.0000	1.0000

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:

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