Getting started with Infineon simulation model

Targeted simulator: OrCAD® and PSpice® from Cadence

About this document

Scope and purpose

This user guide provides instructions for getting started with the attached simulation model using the OrCAD® and PSpice® simulator from Cadence, either by creating a new project from scratch or by using an existing test bench, such as the one included in the zipped archive.

Intended audience

This document is intended for general use and is not tailored to any specific product. Therefore, the naming of symbols, test benches, and other files may differ from those included in the delivery package.

Table of contents

About this document . . . 1
Table of contents . . . 2
1 Resources . . . 3
2 Starting a new project . . . 4
2.1 Creating a new project . . . 4
2.2 Importing an existing model . . . 5
2.2.1 Importing a .lib file . . . 6
2.2.2 Importing an .olb file . . . 7
2.3 Setting up and running a transient analysis . . . 9
2.4 Setting up global parameters . . . 11
2.5 Recommended simulation settings . . . 15
3 Opening an existing project . . . 18
3.1 Running the simulation . . . 18
3.2 Simulation results and measurements . . . 20
Revision history . . . 23
Disclaimer . . . 24

1 Resources

The minimum number of files required to follow the steps from this document can be found in the zipped archive with the following extensions:

- `.lib`: Unencrypted or encrypted model code (to protect Infineon's IP)
- `.olb`: Schematic symbol view for graphical user interface
- `.opj`: Project file that contains one or several application setup(s)

2 Starting a new project

The subsequent sections of this chapter provide basic information about the simulation environment including how to open a schematic and run a simulation. This is because the document is not intended to serve as a comprehensive instruction manual for PSpice®.

2.1 Creating a new project

Follow these steps to create a new schematic:

1. Open OrCAD® Capture CIS application
2. Navigate to **File** > **New** > **Project** and start a new project, as shown in Figure 1

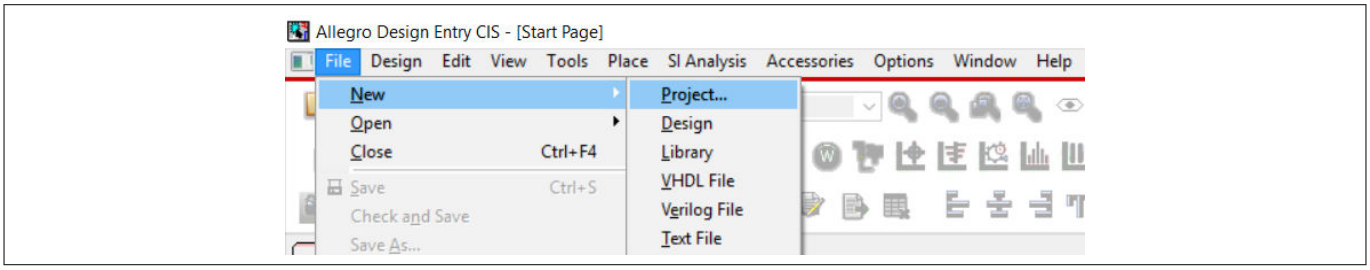


Figure 1 Creating a new project

A new project panel will be displayed, as shown in Figure 2.

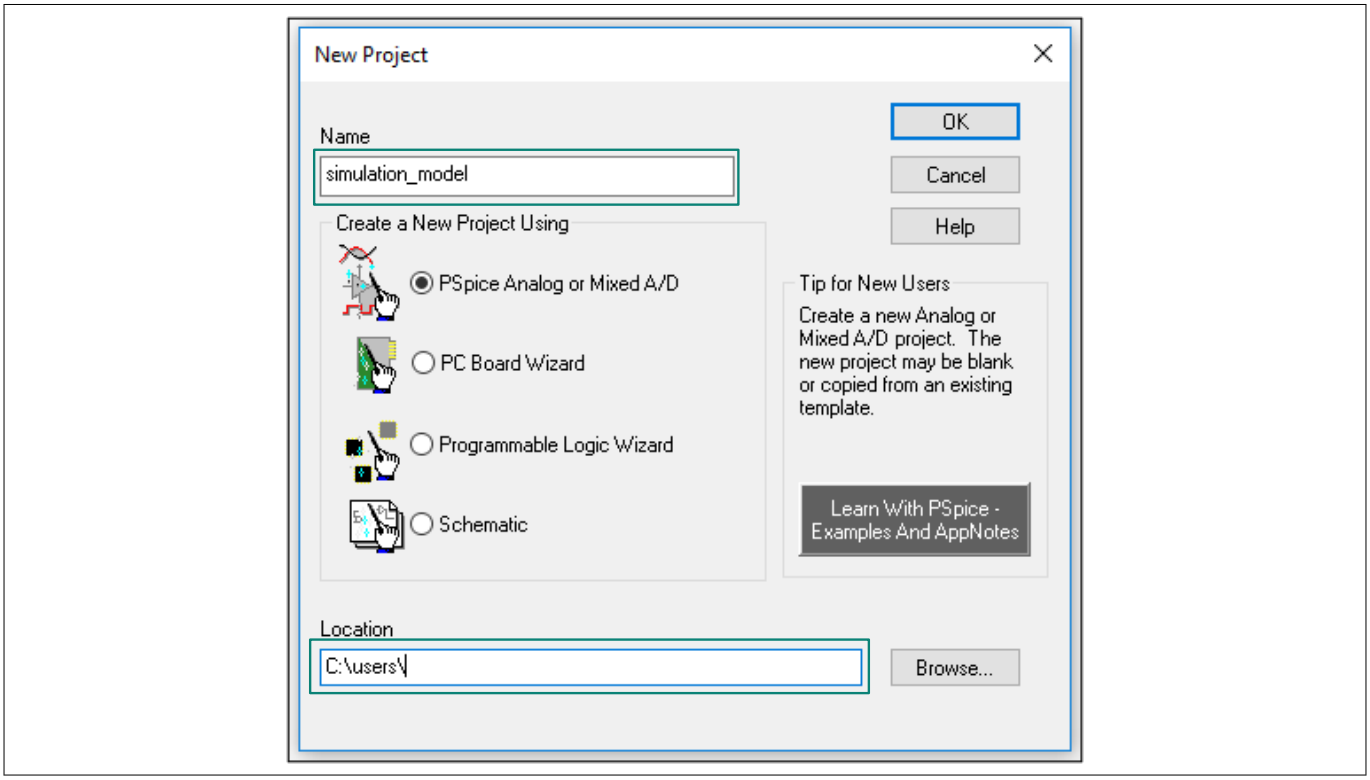


Figure 2 New project

3. Select the project type as **PSpice Analog or Mixed A/D** to enable the simulation
4. Specify the **Name** of the project and a directory as storage placeholder and click **OK**
5. Select **Create a blank project** option in the **Create PSpice Project** pop-up and confirm by clicking **OK**, as shown in Figure 3

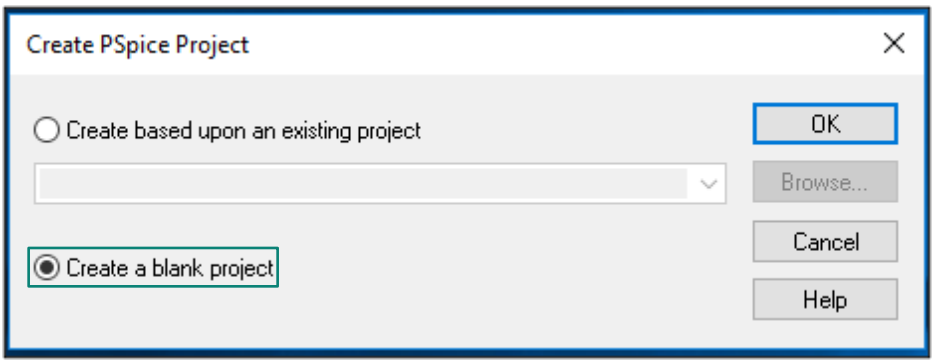


Figure 3 Blank project

A schematic page will be created in the project automatically. In order to create a new schematic page, follow these steps:

1. Right click on the design file (**.dsn**)
2. Select **New Schematic**, choose a name and click **OK**
3. To continue, right click on the newly created schematic folder and choose **New page**.
   A schematic page will appear

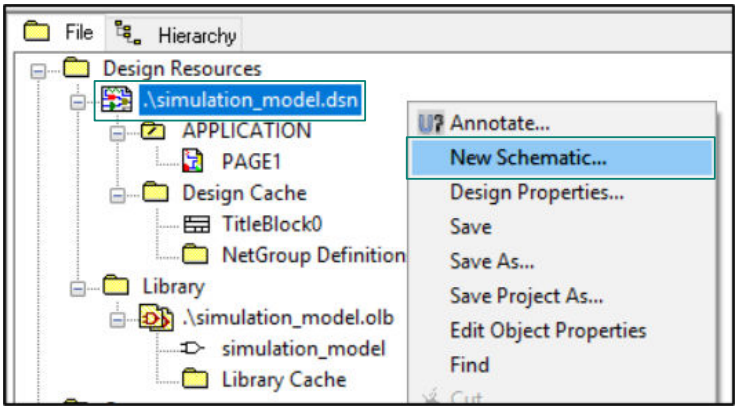


Figure 4 New schematic

2.2 Importing an existing model

***Note**: To simulate a model in schematic mode, OrCAD® from Cadence requires two types of input files: `simulation_model.lib` (device behavior source code) and `simulation_model.olb` (schematic symbol view with device pinout). These two types of files are generally referred to as a library.*

2.2.1 Importing a .lib file

Follow these steps to import a `.lib` file:

1. Navigate to **PSpice** > **Edit Simulation Profile**

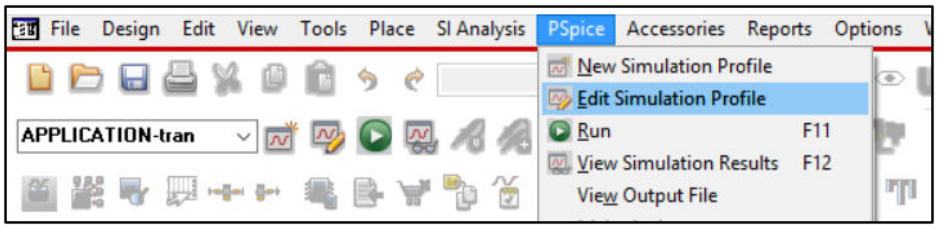


Figure 5 Importing .lib file

2. In the **Simulation Settings** window, go to **Configuration Files** tab. From **Category** list, select **Library**
3. Click **Browse** next to the **Filename** and select the model `*.lib` file
4. After validating the selection, choose one of the three modes of library assignment:
   - **Add as Global**: To all designs
   - **Add to Design**: For current design only (recommended)
   - **Add to Profile**: Local to the current test-bench inside the current design

***Note**: If the "nom" library is not automatically listed in the simulation profile, it must be added from: `C:\Cadence\SPB_17.2\tools\Pspice\Library\nom.lib`.*

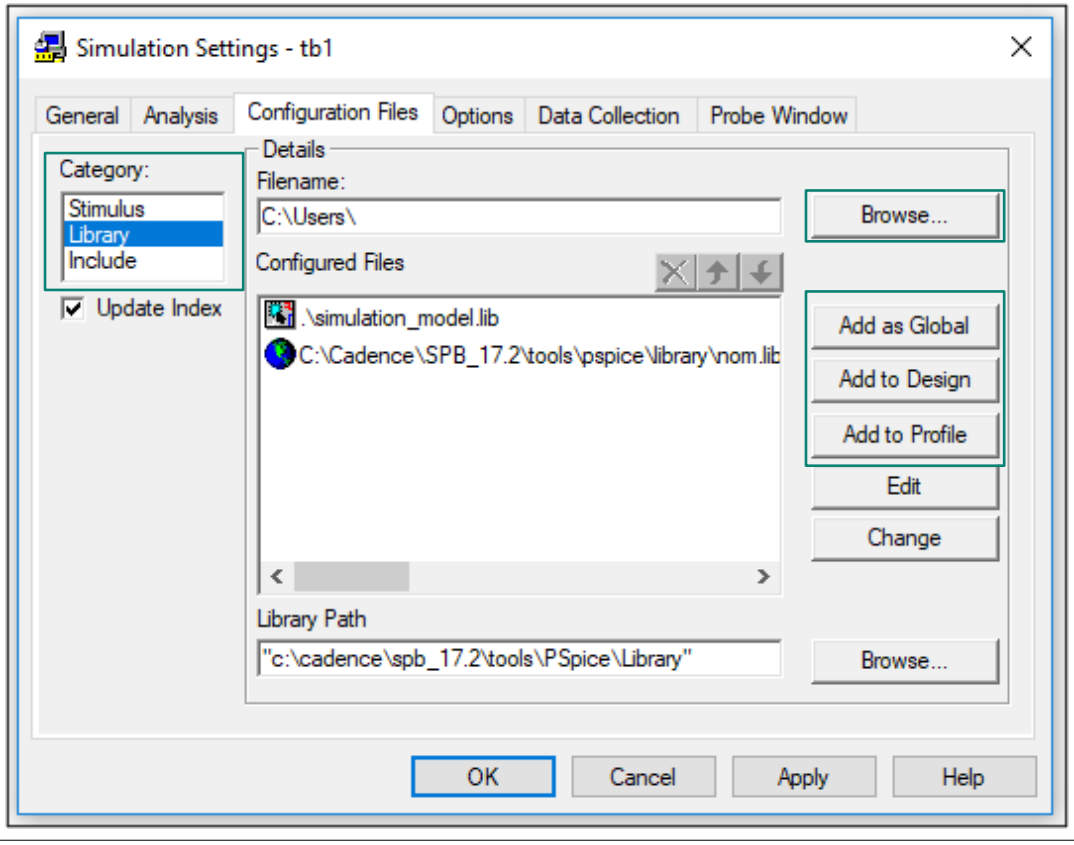


Figure 6 Simulation settings

2.2.2 Importing an .olb file

Follow these steps to import an `.olb` file:

1. From the **Project** tab, under **Design Resources**, right-click on **Library** and choose **Add File**

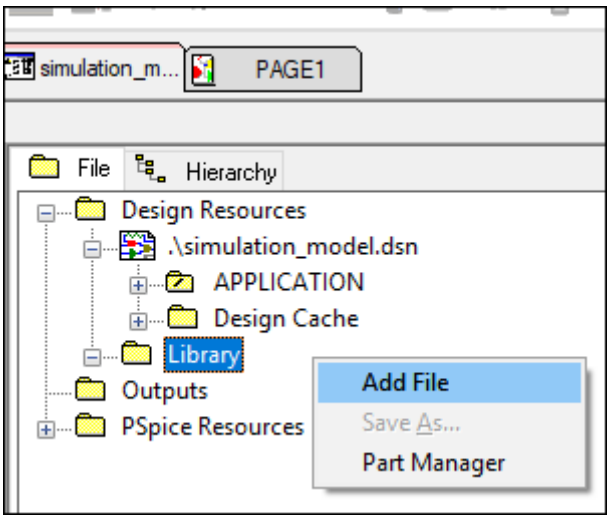


Figure 7 Importing an .olb file

2. In the pop-up window, browse for the `.olb` file

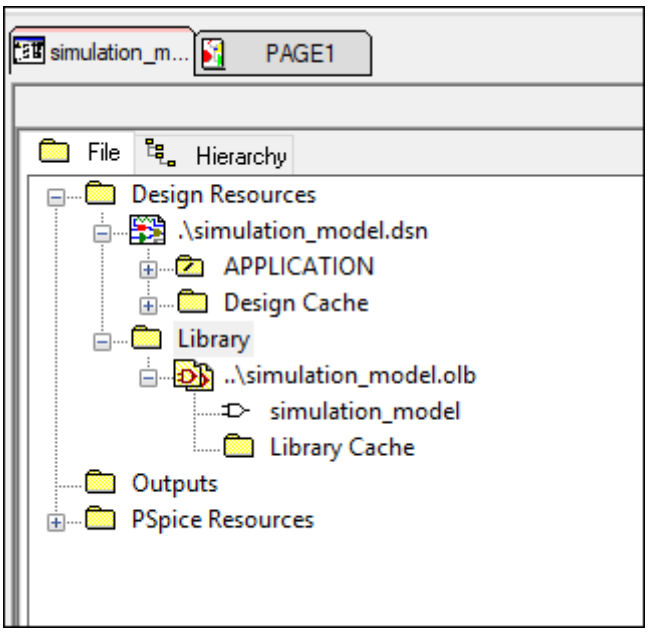


Figure 8 Browse for the .olb file

3. Search in the **Place Part** sidebar (or press **P** on the keyboard) and place the symbol in the desired schematic, as shown in Figure 9

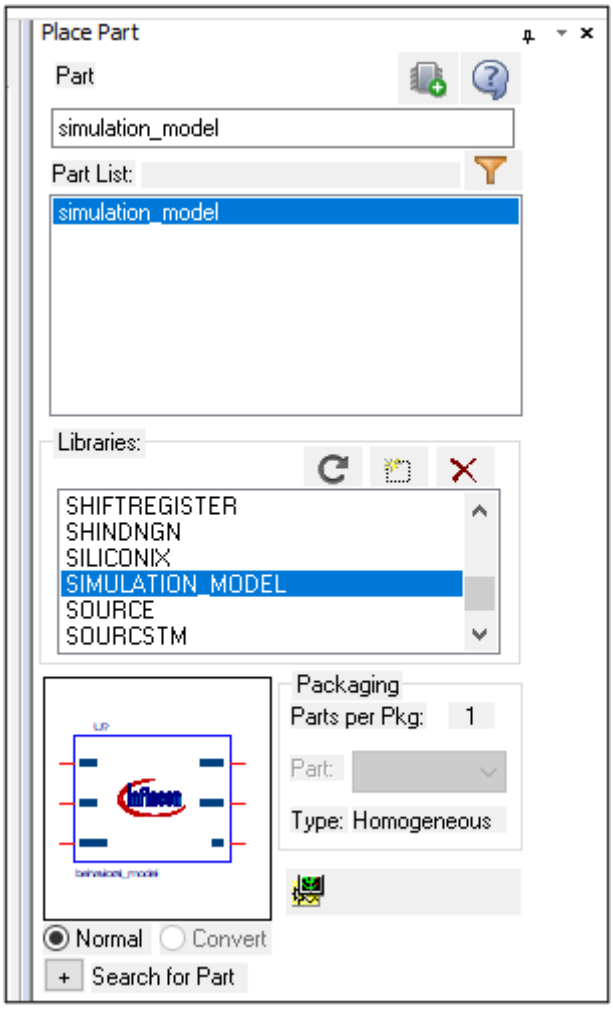


Figure 9 Place part

2.3 Setting up and running a transient analysis

A transient analysis evaluates the time-domain behavior of the circuit, similar to what an oscilloscope displays in the lab. To perform this analysis, follow these steps:

1. From the main menu, select **PSpice** and choose **New Simulation Profile**, as shown in Figure 10

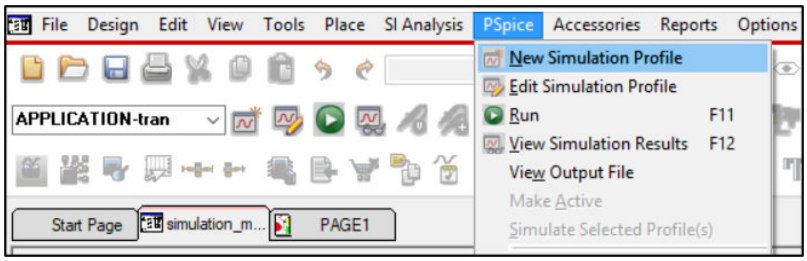


Figure 10 New simulation profile

2. Set the **Name** of the simulation profile and click **Create**, as shown in Figure 11

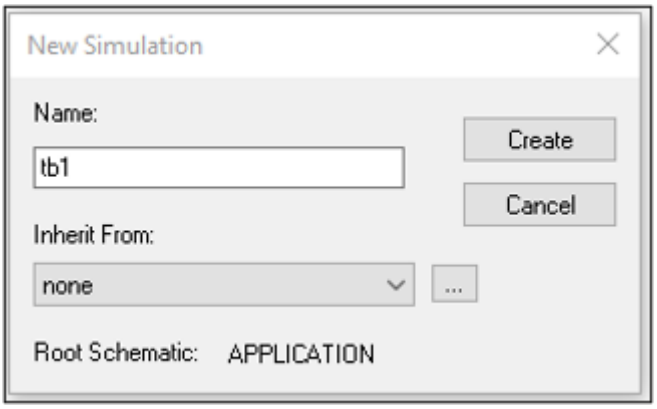


Figure 11 New simulation

The **Simulation Settings** dialog box appears, as shown in Figure 12

3. From the **Analysis Type** list, select **Time Domain (Transient)** and specify the required parameters:
   - **TSTOP** = The total simulation duration
   - **Maximum step size** = The maximum internal step size, which depends on the specified *TSTOP* parameter

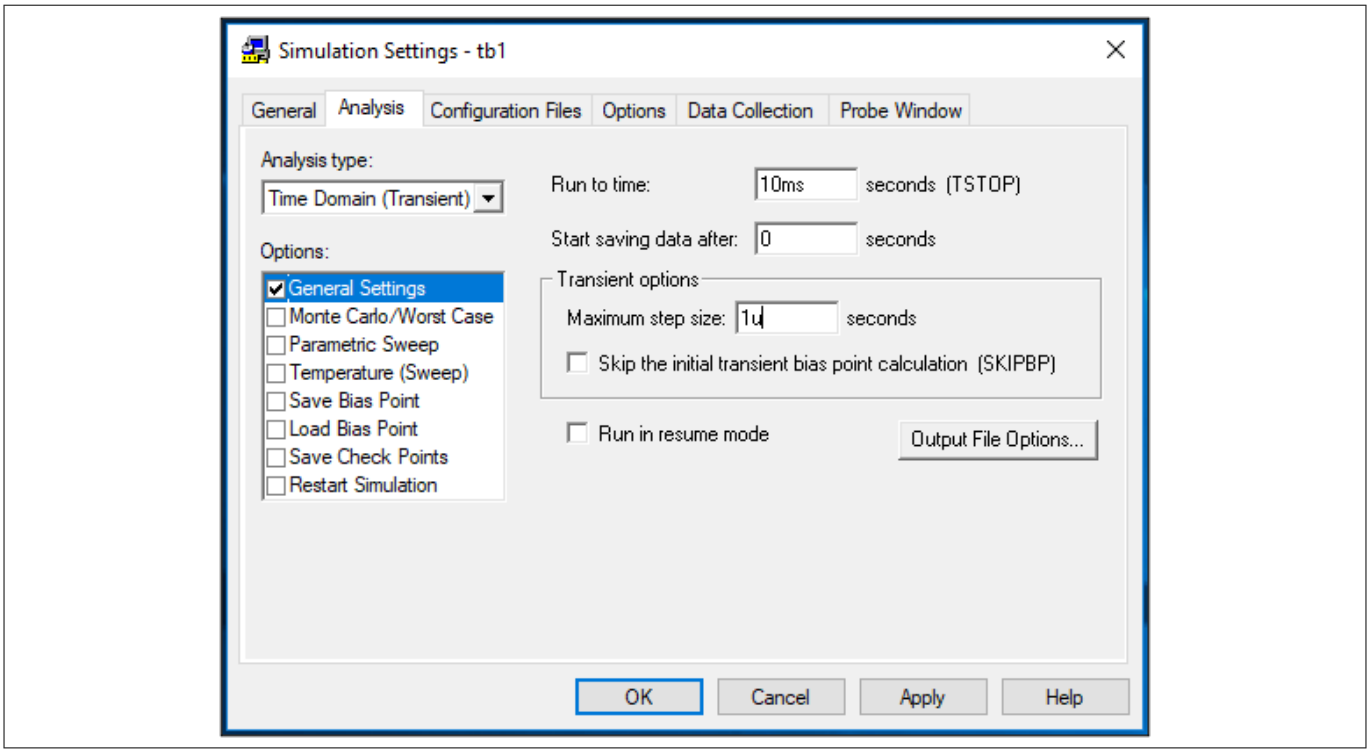


Figure 12 Analysis settings

4. Under **Simulation Settings**, go to **Options** tab, as shown in Figure 13
5. Select the category as **Gate-Level Simulation**. Ensure that the **Initialize all flip-flops to** is set to '0'

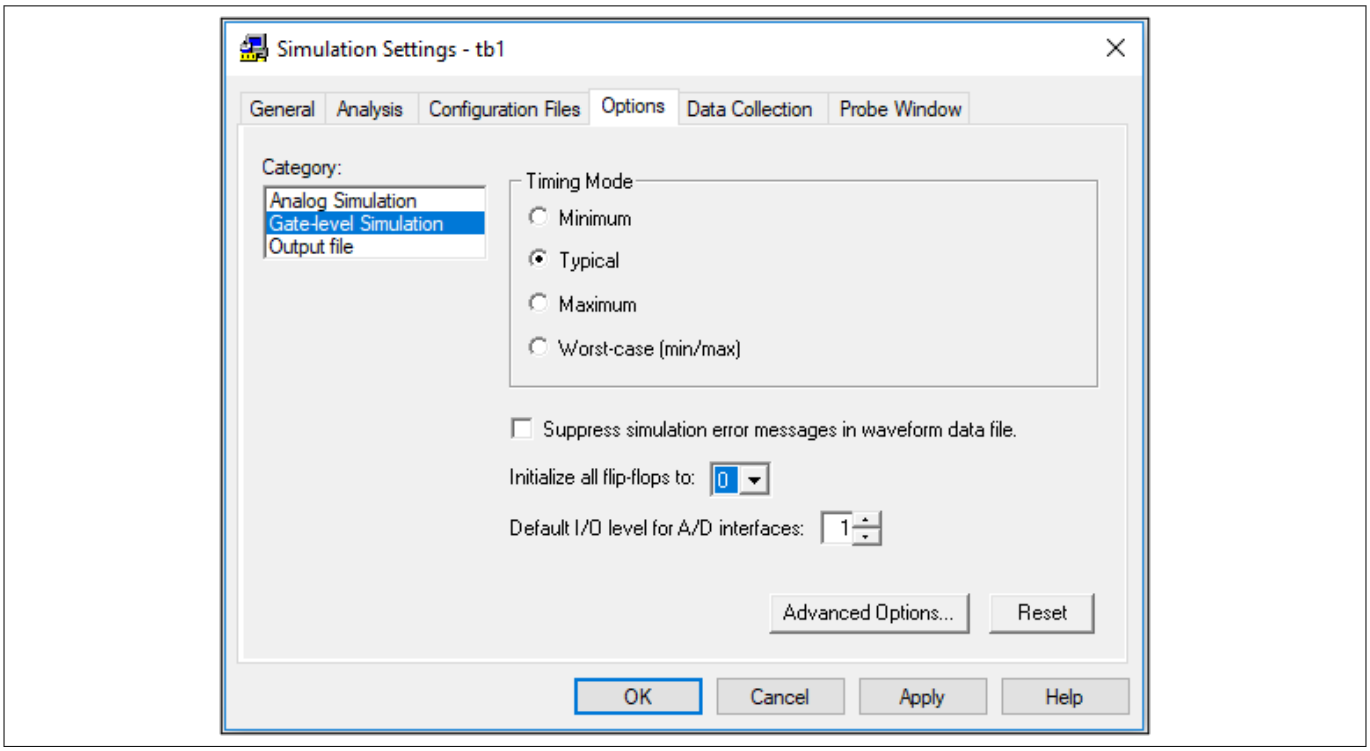


Figure 13 Options settings

6. Click **OK** to apply the settings and close the **Simulation Settings** dialog box
7. From the **PSpice** menu, click **Run** to perform the analysis, as shown in Figure 14

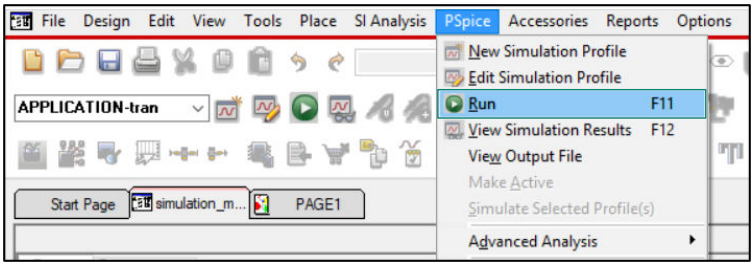


Figure 14 Run the simulation

2.4 Setting up global parameters

A model parameter allows the users to directly control specific components within any hierarchical level of the circuit, serving the following purposes:

- To apply the same value to multiple part instances
- To set up an analysis that sweeps a variable through a range of values (for example, DC sweep or parametric analysis)

Figure 15 shows how to define a global parameter in OrCAD® from Cadence. Follow the instructions below:

1. In the **Place Part** sidebar, find the instance **PARAM**
2. Under **Libraries**, select **SPECIAL**. Select and double-click the **PARAM** from the **Part List** and place it on the schematic page

***Note**: If the **SPECIAL** library is missing, load it from `C:\Cadence\SPB_17.2\tools\capture\library\pspice\special.olb` as show in* Importing an .olb file.

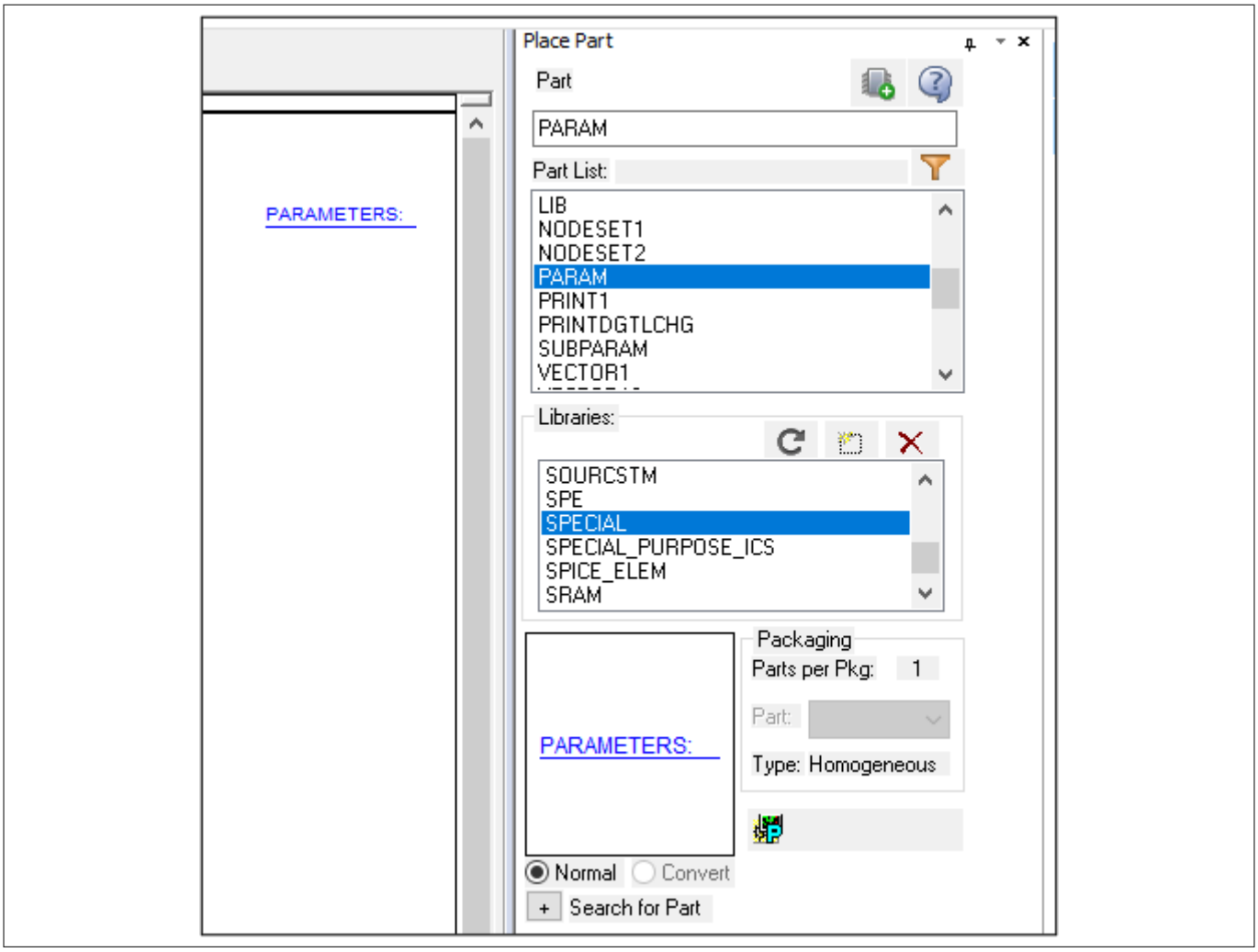


Figure 15 Global parameter

3. Double click on **PARAMETERS** to start the spreadsheet editor
4. click **New Property**, type a **Name** and **Value** and click **OK**, as shown in Figure 16

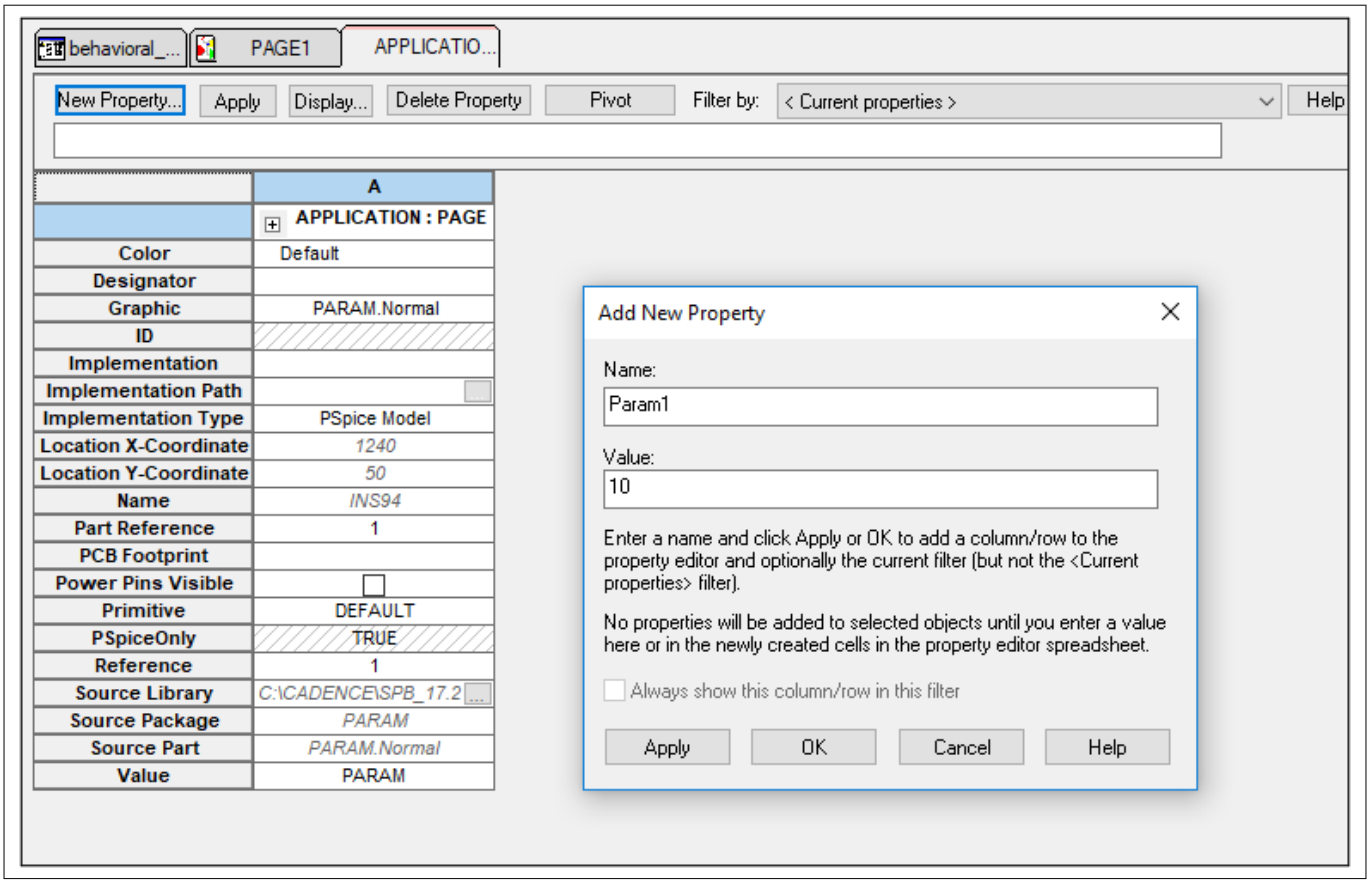


Figure 16 Add new parameter

5. Select **Param1** in spreadsheet editor and choose the **Display** option. In **Display Properties** window, set **Display Format** to **Name and Value**, then click **OK**, as shown in Figure 17. This shows the recently created global parameter on the test bench

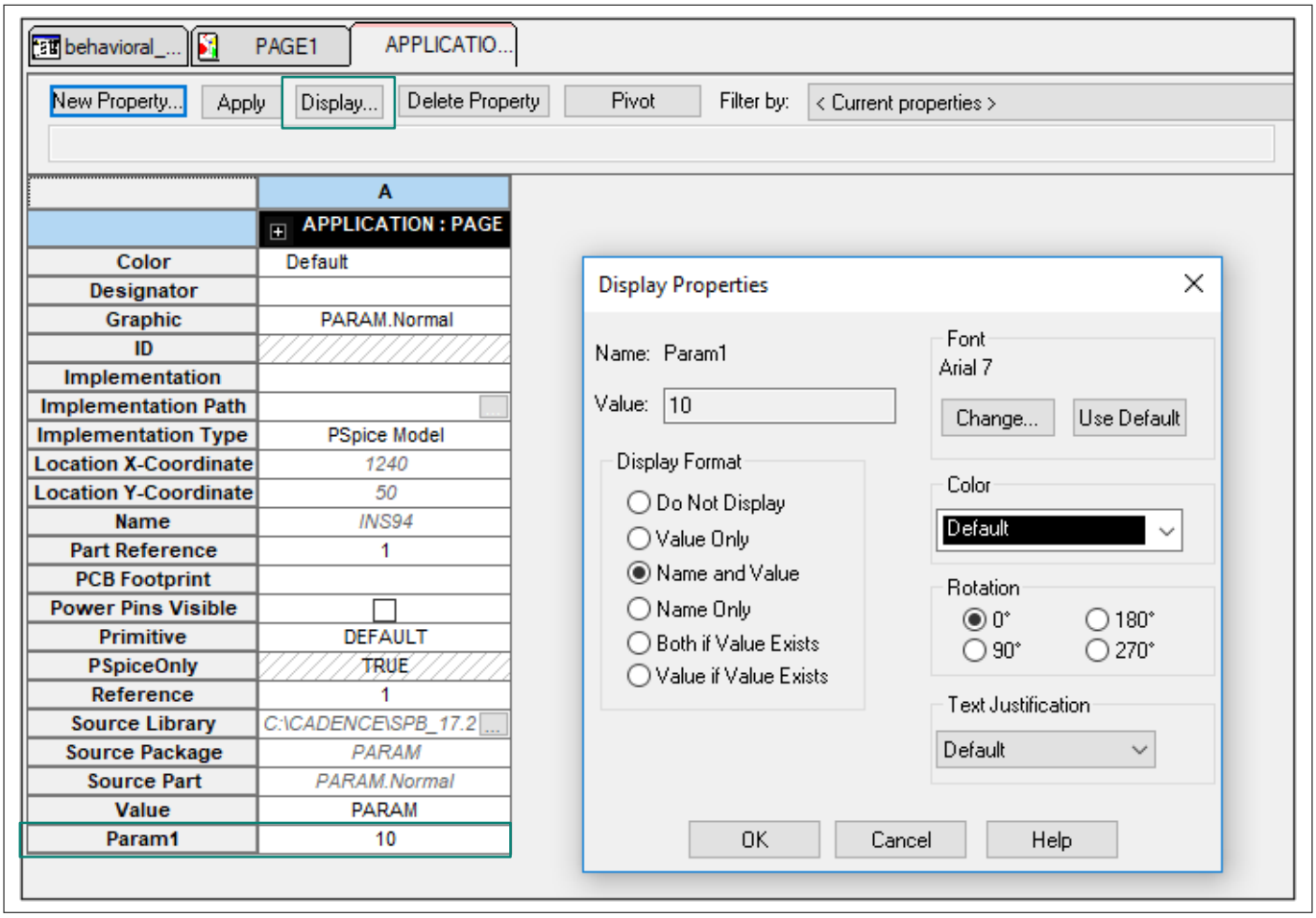


Figure 17 Display a global parameter

Two practical examples are shown below:

Figure 18 shows the use of global parameter for multiple instances. To apply the same value to multiple instances, use the parameter name between curly brackets within the value field of any component.

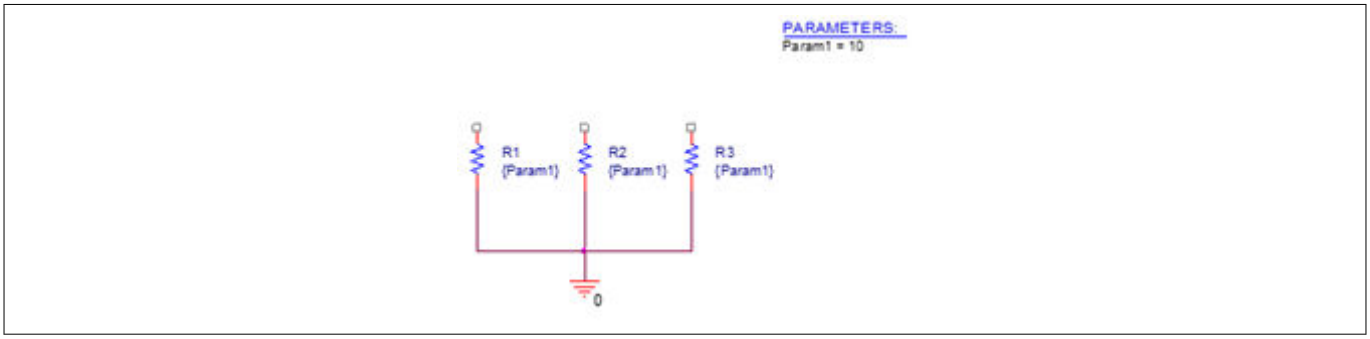


Figure 18 Global parameter for multiple instances

Figure 19 shows the steps required to set up and run a parametric simulation. Perform an analysis to sweep variable through a range of values. The effect is the same as running the circuit several times, once for each value of the swept variable.

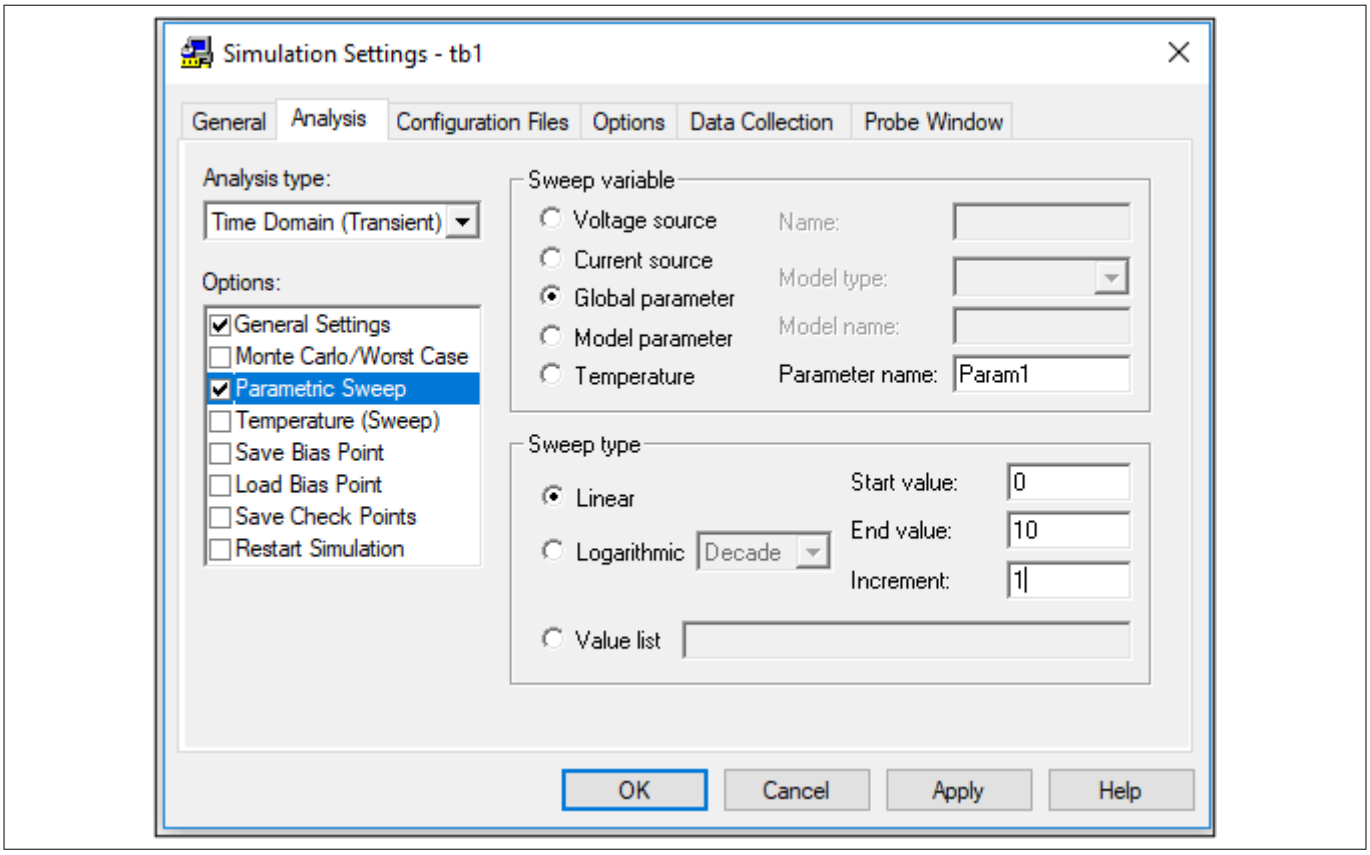


Figure 19 Set up a parametric simulation

2.5 Recommended simulation settings

Because PSpice® was not originally designed specifically for power electronics and highly non-linear components, the simulation parameters (**PSpice** > **Edit Simulation Profile** > **Options**) can be optimized for a given design. Figure 20 illustrates an example.

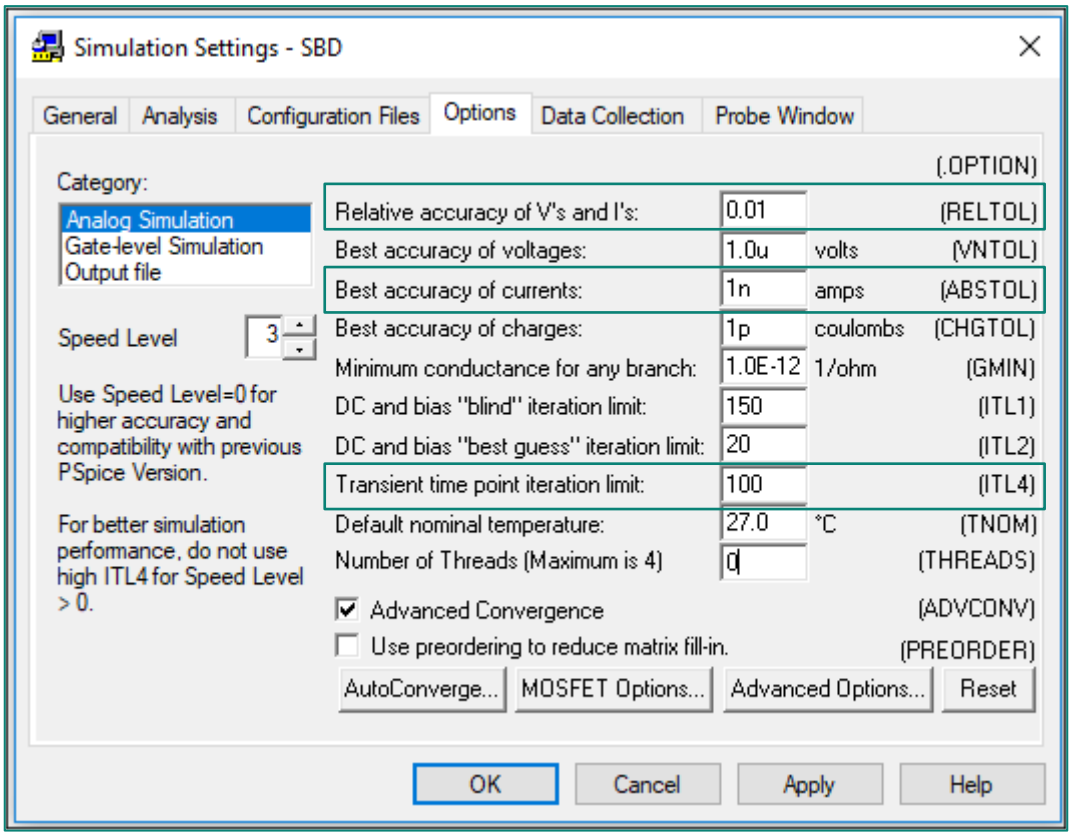


Figure 20 Simulation parameters

Table 1 provides the recommended values to facilitate convergence.

Table 1 Recommended values to facilitate convergence

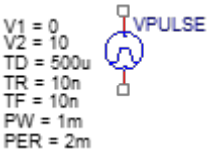
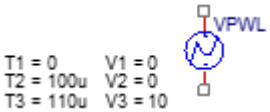
Parameter	Value	Description
RELTOL	0.01	Relative accuracy of voltage and currents
ABSTOL	1 nA	Maximum current accuracy
ITL4	100	Maximum number of iterations for transient analysis

The delivered models must detect a rising edge starting from 0 V on both input and supply voltages to ensure the correct start-up sequence. Therefore, it is recommended to use V_{PWL} or V_{PULSE} instead of V_{DC} component.

***Note**: To prevent potential simulator convergence issues, avoid using voltage sources such as V_{DC} components, as most models require a power-up pattern that starts from a zero-volt supply for at least a few microseconds.*

Table 2 shows the recommended input and supply voltage sources. It is best-practice to use sources like V_{PULSE} or V_{PWL}.

Table 2 Input and supply voltage sources

V_{DC}	V_{PULSE}	V_{PWL}
		
⊗	✓	✓

3 Opening an existing project

A simulation model is delivered with an attached test bench already configured and ready for use.

3.1 Running the simulation

Follow these steps to run the simulation:

***Note**: For windows, the user must install a fully licensed version of OrCAD® from Cadence. Extract the files to `c:\` drive.*

1. Open OrCAD® Capture CIS application
2. Navigate to **File** > **Open** > **Project**, as shown in Figure 21
3. In the browse window, locate and open the `.opj` file package directory

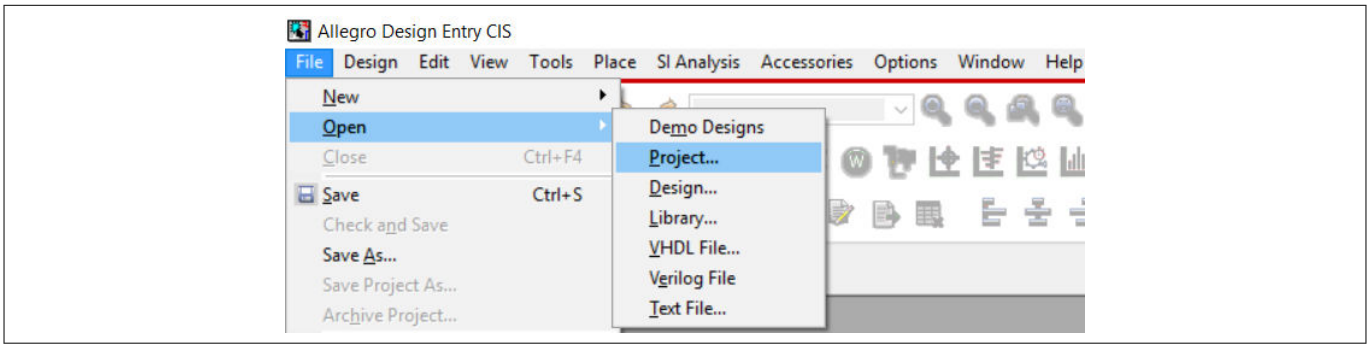


Figure 21 Opening an existing project

Figure 22 shows the project manager that appears in the PSpice® session frame whenever you open or create a project. Use the project file manager to collect and organize all the resources needed throughout the design flow. These resources include schematic design files, part libraries, netlists, simulation models, simulation profiles, stimulus files, and any other related information.

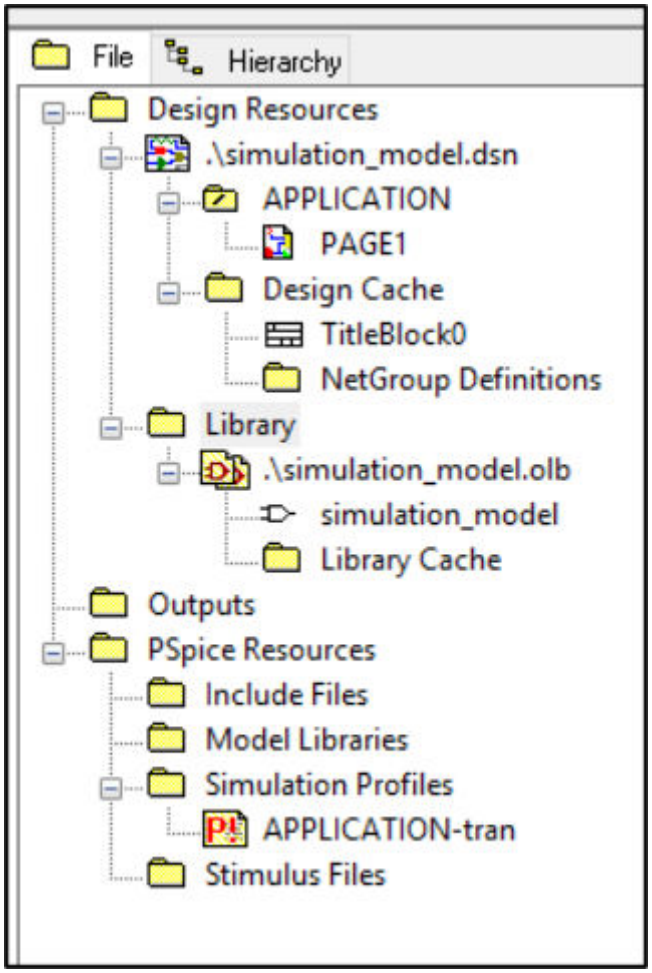


Figure 22 Project manager

Users can select from already available test benches (with included simulation profile and predefined waveforms probes), as shown in Figure 23.

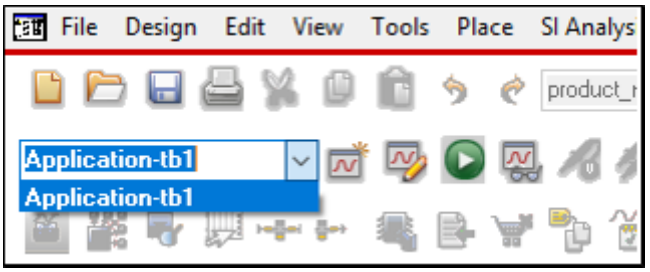


Figure 23 Select the application test bench

4. To start the simulation, open **PSpice** menu and click **Run** or press **F11** on the keyboard. Alternatively, the **Play** button on the PSpice® toolbar can be used, as shown in Figure 24



Figure 24 Start the simulation

3.2 Simulation results and measurements

Once the netlist has been generated the Allegro AMS Simulator (PSpice®) will open and the simulation automatically starts.

A progress bar will be displayed in the bottom right corner, as shown in Figure 25.

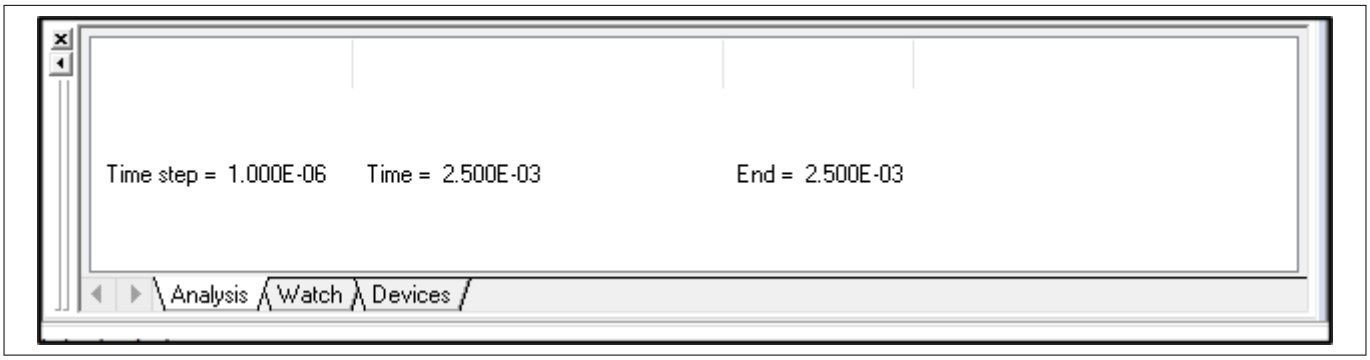


Figure 25 Simulation progress

After the simulation reaches 100%, the signals can be plotted. Navigate to **Trace** > **Add Trace**, as shown in Figure 26. Alternatively, it is possible to press the **Add Trace** button (toolbox section) or **Insert** button (keyboard). Figure 27 shows the respective signals that must be selected from the list to plot them. After the selection, click **OK** button.

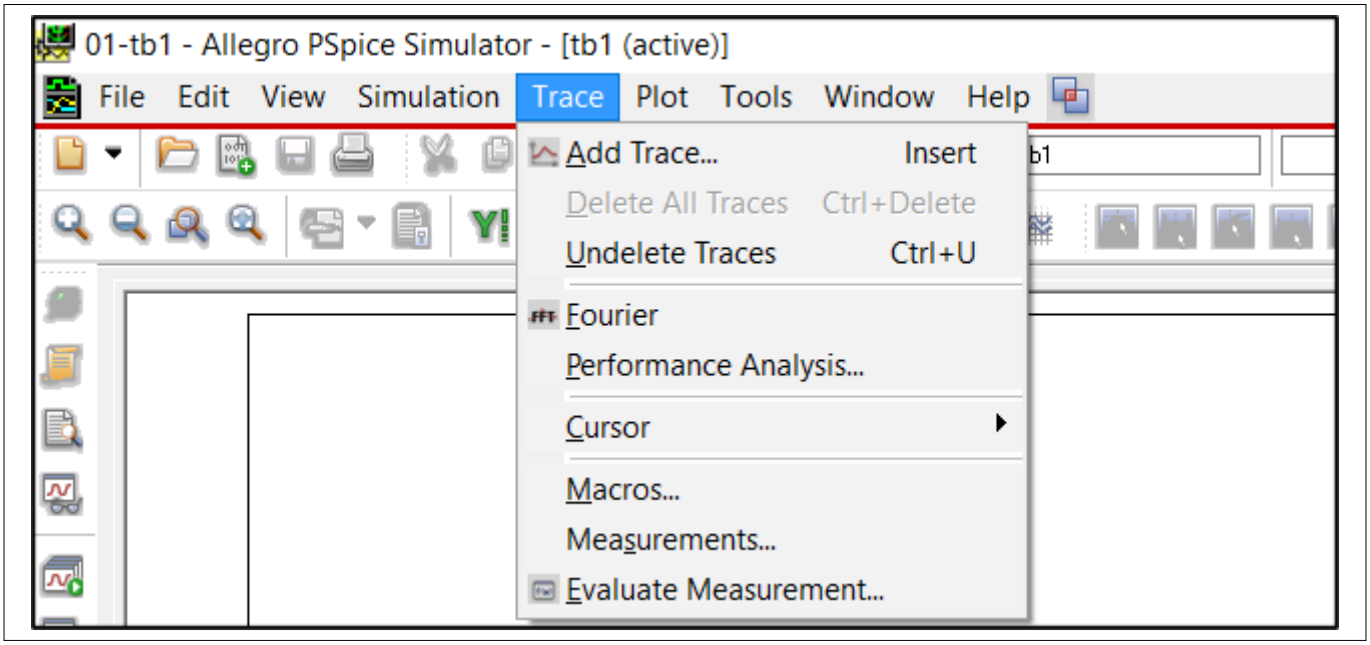


Figure 26 Plot signals

***Note**: The desired signals can be filtered by using the check boxes in the center of the window according to their type (for example, voltages, currents, power, and so on). Signals can also be composed by selecting the appropriate functions available in the right list **Functions or Macros**.*

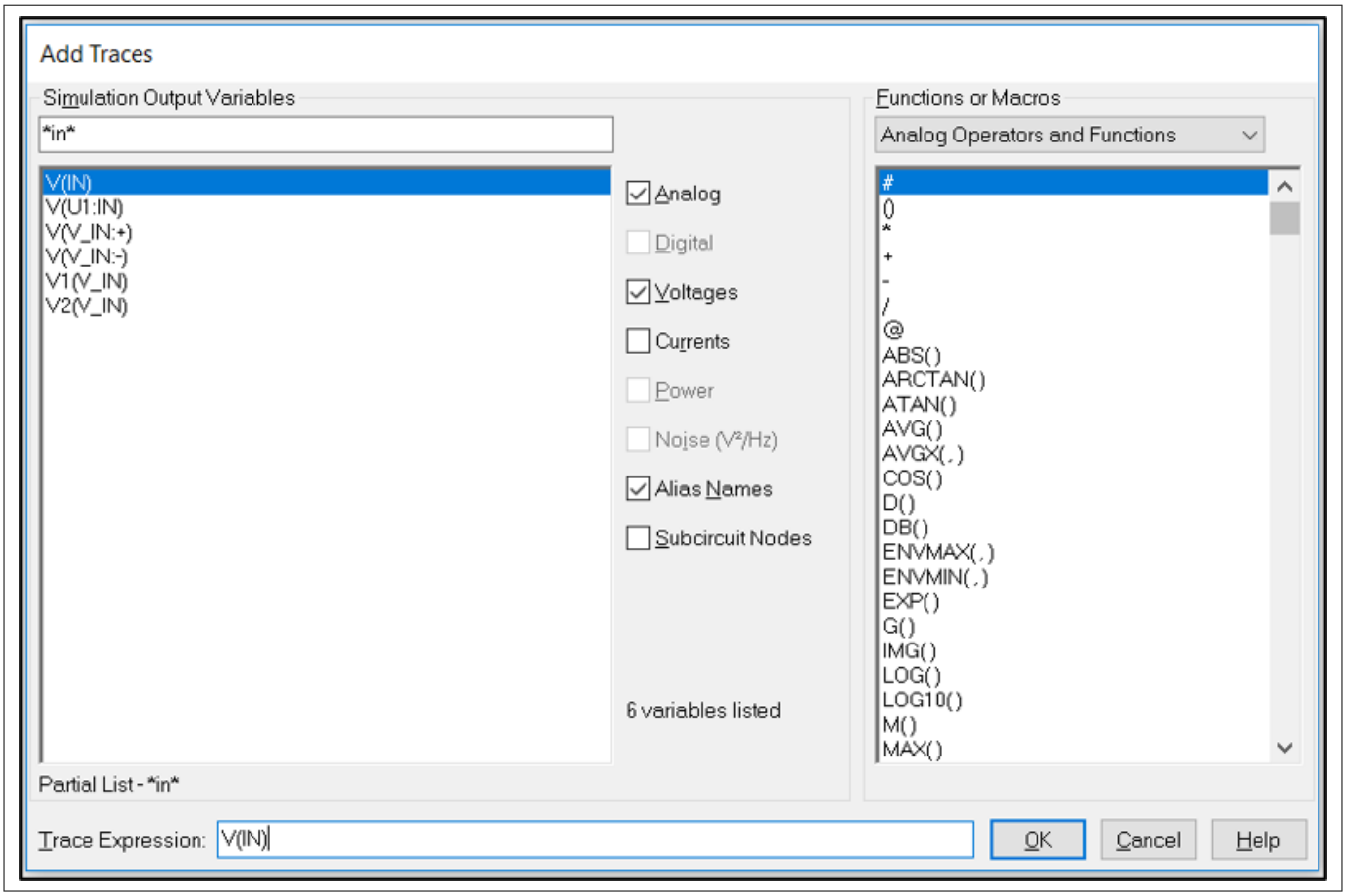


Figure 27 Add traces

Figure 28 shows how to add a new plot: Go to **Plot** > **Add Plot to Window**.

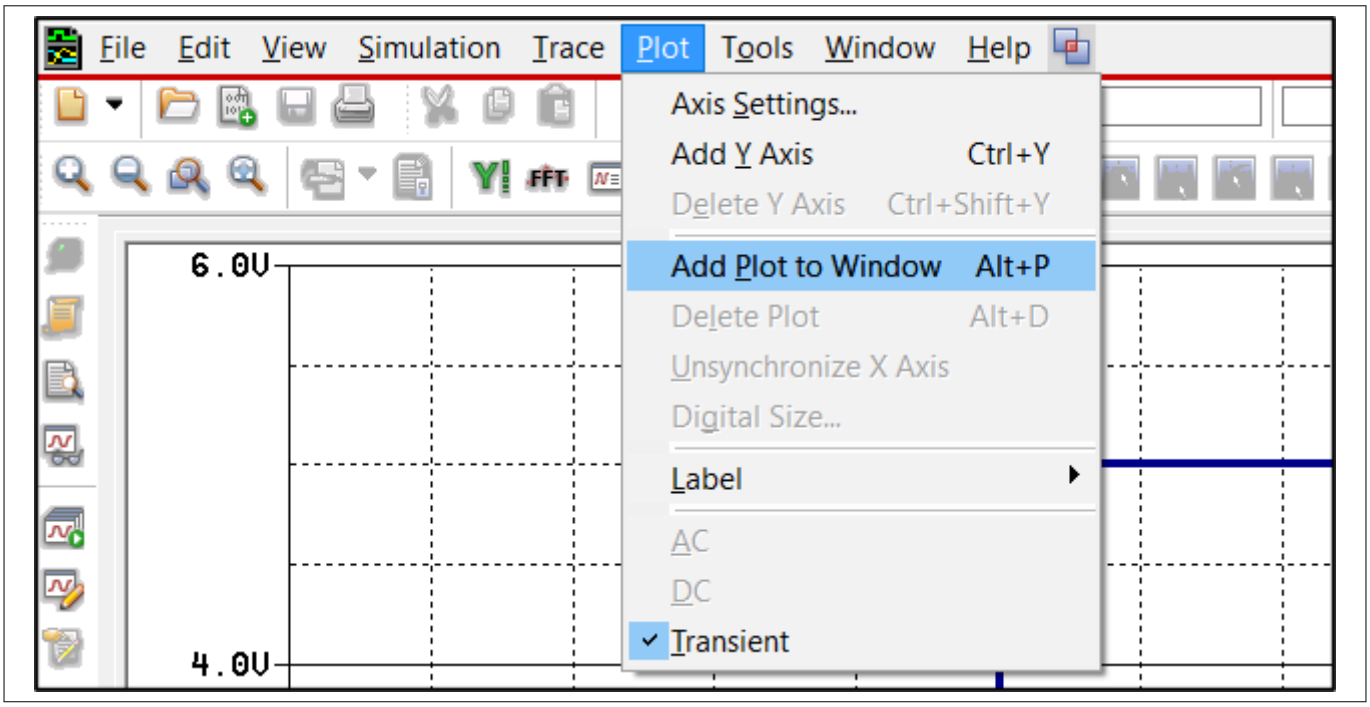


Figure 28 Add a new plot

To add cursors for the signal value measurement, go to **Trace** > **Cursor** > **Display**, as shown in Figure 29. Alternatively, this can be done by a left-click onto the **Display Cursors** button (toolbox).

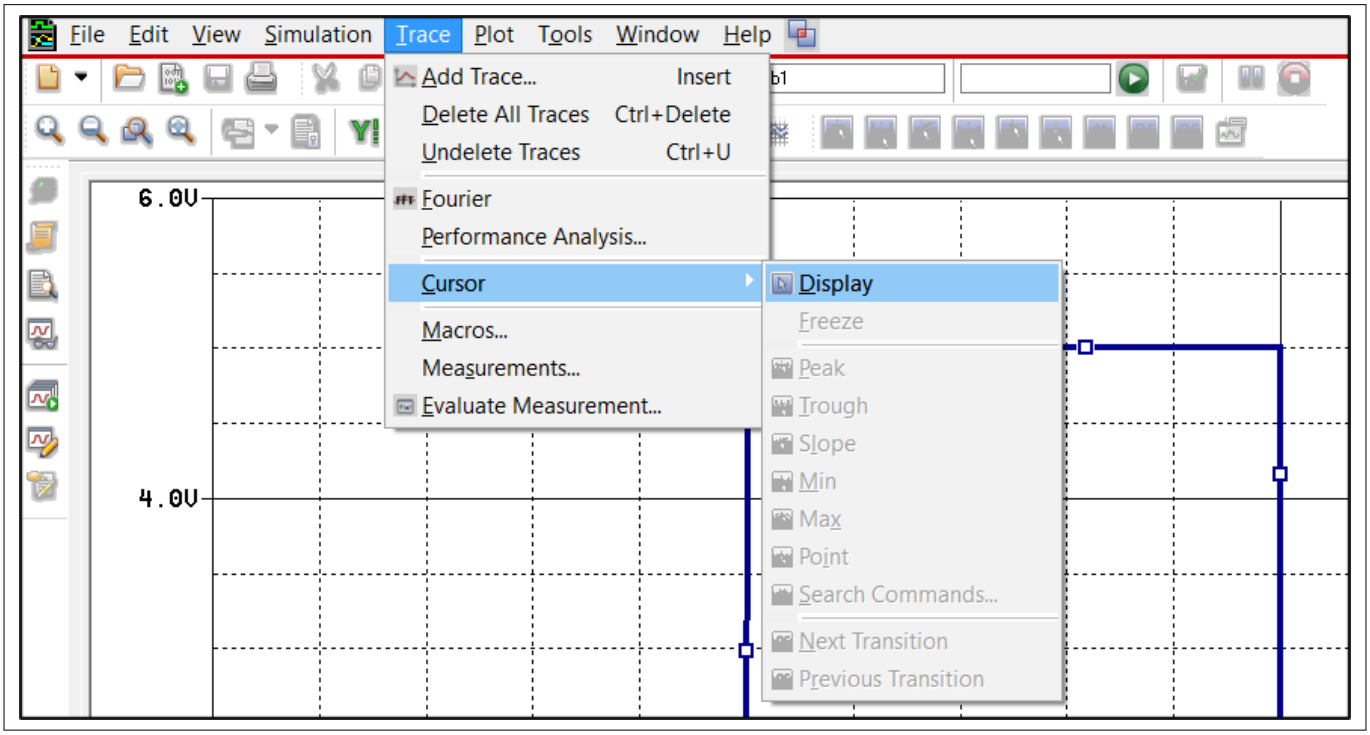
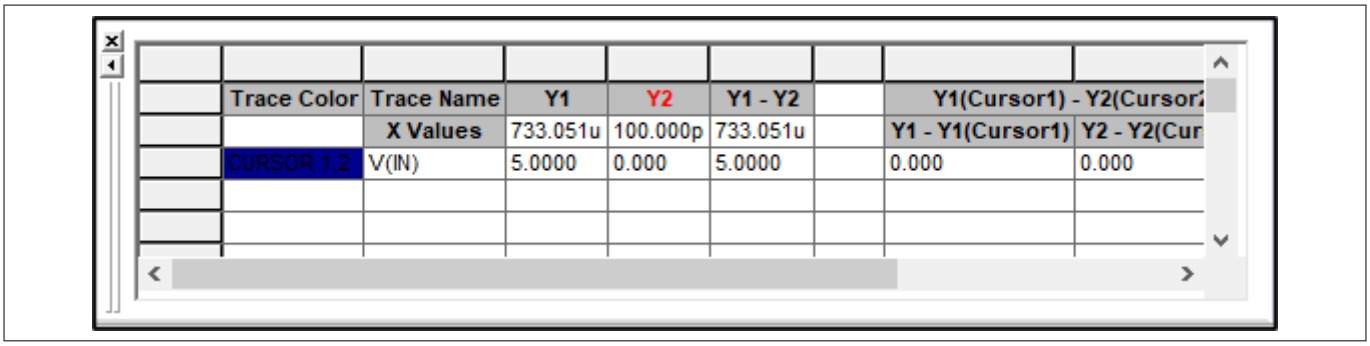


Figure 29 Add cursors

Figure 30 shows an example of a table measurement menu, which is observable in the bottom right corner of the GUI.



Trace Color	Trace Name	Y1	Y2	Y1 - Y2		Y1(Cursor1) - Y2(Cursor2	
	X Values	733.051u	100.000p	733.051u		Y1 - Y1(Cursor1)	Y2 - Y2(Cur
CURSOR 1,2	V(IN)	5.0000	0.000	5.0000		0.000	0.000

Figure 30 Measurement values

Revision history

Document revision	Date	Description of changes
01.00	2021-10-19	Initial version created
02.00	2026-03-18	Updated the document structure

Trademarks

All referenced product or service names and trademarks are the property of their respective owners.

Edition 2026-03-18

Published by

Infineon Technologies AG

81726 Munich, Germany

© 2026 Infineon Technologies AG

All Rights Reserved.

Do you have a question about any aspect of this document?

Email: erratum@infineon.com

Document reference
IFX-sxf1773857671033

Important notice

The information given in this document shall in no event be regarded as a guarantee of conditions or characteristics ("Beschaffenheitsgarantie").

With respect to any examples, hints or any typical values stated herein and/or any information regarding the application of the product, Infineon Technologies hereby disclaims any and all warranties and liabilities of any kind, including without limitation warranties of non-infringement of intellectual property rights of any third party.

In addition, any information given in this document is subject to customer's compliance with its obligations stated in this document and any applicable legal requirements, norms and standards concerning customer's products and any use of the product of Infineon Technologies in customer's applications.

The data contained in this document is exclusively intended for technically trained staff. It is the responsibility of customer's technical departments to evaluate the suitability of the product for the intended application and the completeness of the product information given in this document with respect to such application.

Warnings

Due to technical requirements products may contain dangerous substances. For information on the types in question please contact your nearest Infineon Technologies office.

Except as otherwise explicitly approved by Infineon Technologies in a written document signed by authorized representatives of Infineon Technologies, Infineon Technologies' products may not be used in any applications where a failure of the product or any consequences of the use thereof can reasonably be expected to result in personal injury.