Getting started

Targeted simulator: OrCAD PSpice

About this document

Scope and purpose

The current document provides guidance on how to get started with the attached simulation model using the **OrCAD PSpice** simulator either by creating a new project from scratch or based on an existing test bench like the one attached to the zipped archive.

Intended audience

This document has been created for general purposes and has not been tailored to any specific product. Therefore, the naming of symbols, test benches and other files will differ from the ones included in the delivery package.

Table of contents

About this document **1**
Table of contents **1**
1 Resources **2**
2 Starting a new project **3**
2.1 Create a new project 3
2.2 Import of an existing model 4
2.2.1 Importing a .lib file 4
2.2.2 Importing an .olb file 5
2.3 Set up and run a transient analysis 6
2.4 Set up global parameters 8
2.5 Recommended simulation settings 11
3 Opening an existing project **13**
3.1 Steps to run the simulation: 13
3.2 Simulation results and measurements 14
4 Disclaimer **17**
4.1 Scope of use 17
4.2 Important notice 17
4.3 Confidential information 17
4.4 Warranty 17
4.5 Liability 17
4.6 Export regulations 18
4.7 Termination of use permit 18
4.8 Miscellaneous 18
Revision history **19**

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 which contains one/several application setup(s)

2 Starting a new project

Instructions presented below assume basic knowledge of the simulation environment such as opening a schematic and running a simulation since this document is not a comprehensive instruction manual for PSpice.

2.1 Create a new project

Begin by opening the **OrCAD Capture CIS** application. From **File** → **New** → **Project**, select the project type as **PSpice Analog Mixed A/D** to enable the simulation. Then specify the name of the project and a directory as storage placeholder. **Figure 1** below shows an example.

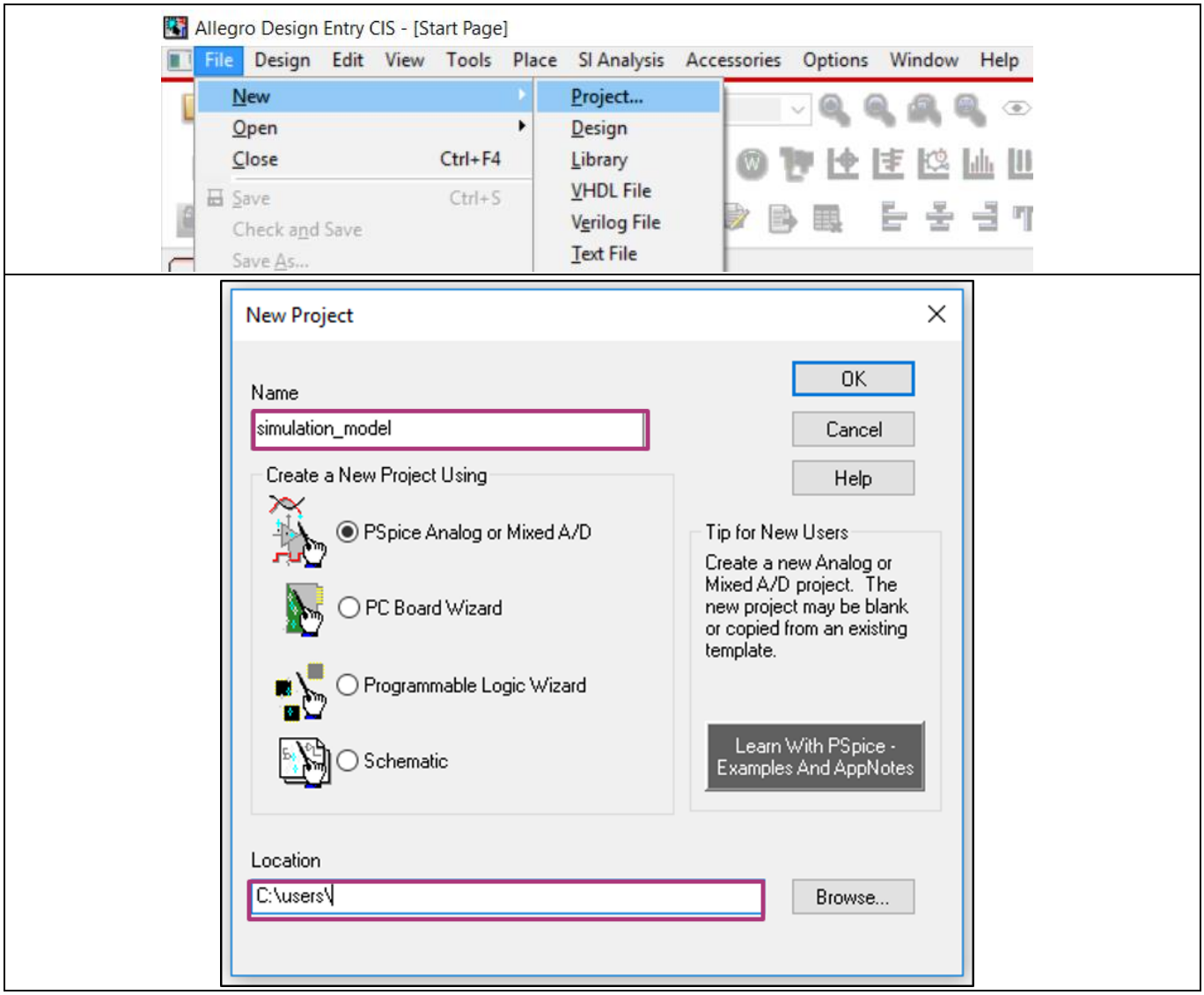


Figure 1 Create new project

After pressing **OK**, select the **Create a blank project** option in the **Create PSpice Project** pop-up and confirm by clicking **OK**, as shown in **Figure 2**.

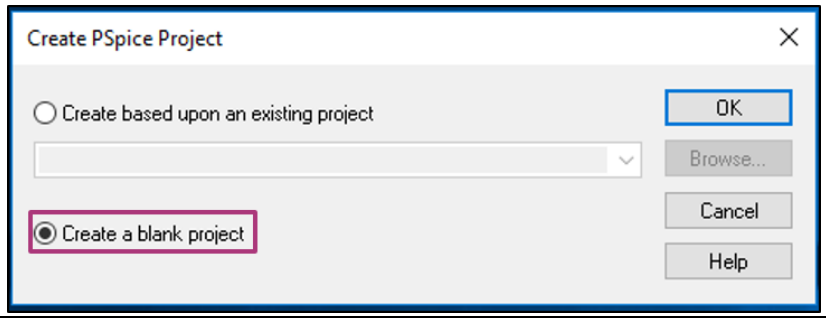


Figure 2 Blank project

A schematic page will be created in the project automatically. In order to create a new schematic page, right click on the design file (**.dsn**), select **New Schematic**, choose a name and press **OK**. To continue, right-click on the newly created schematic folder and choose **New page**. A schematic page will appear.

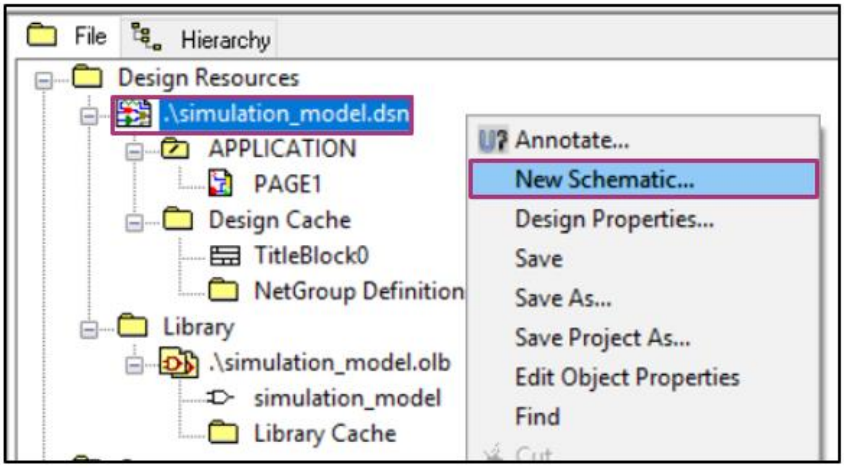


Figure 3 New Schematic

2.2 Import of an existing model

*Note: To simulate a model in schematic mode, **Cadence OrCAD** 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

Figure 4 describes the following steps. From **PSpice** menu choose **Edit Simulation Profile**. Once the **Simulation Settings** window pops up navigate to the **Configuration Files** (1) tab. Under **Category** select **Library** (2). Click **Browse** (3) next to the **Filename** location bar and select the model ***.lib** file. After validating the selection, choose one of the three modes of library assignment (4):

- **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** (5).*

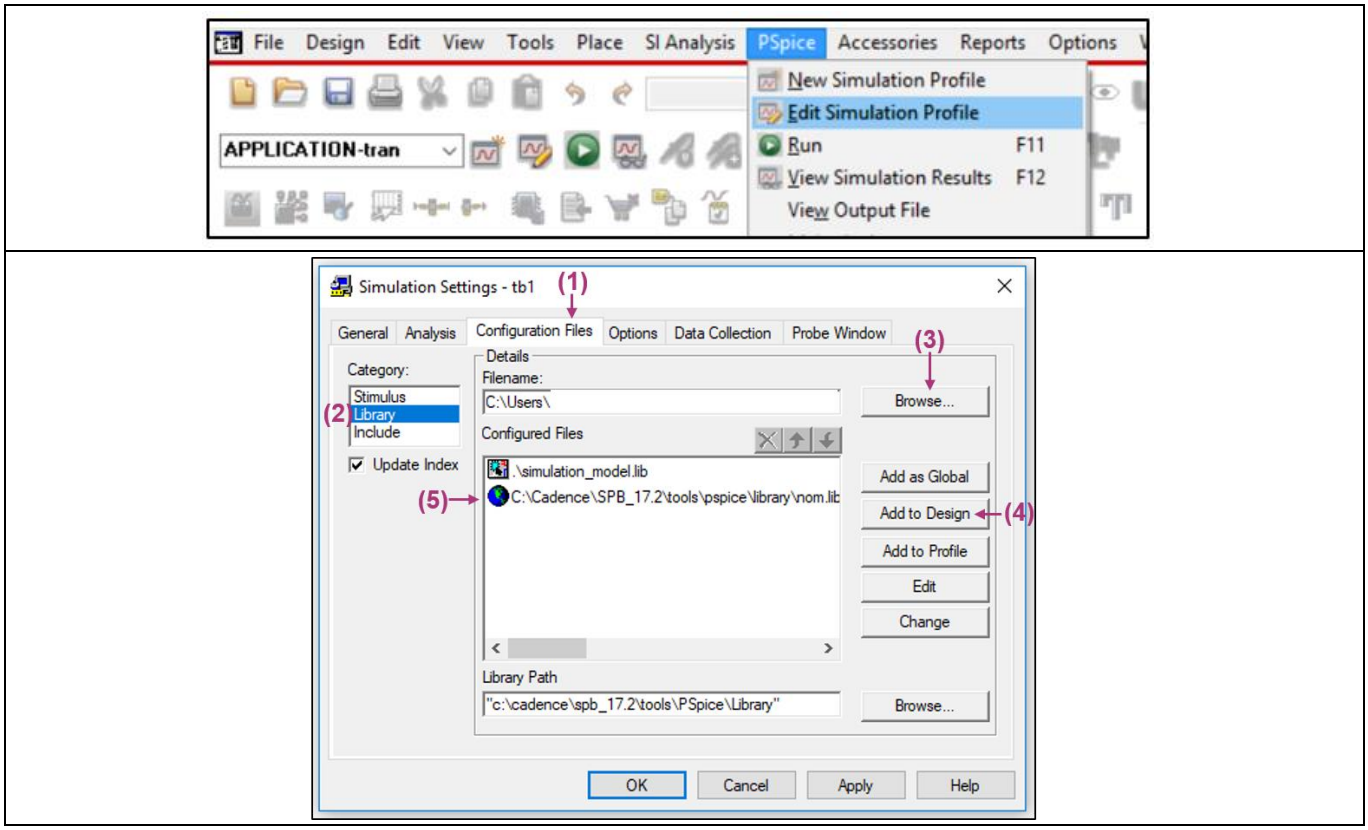


Figure 4 Import .lib file

2.2.2 Importing an .olb file

Figure 5 describes the following steps. From the **Project** tab, under **Design Resources**, right-click on **Library** (1) and choose **Add File** (2). In the pop-up window, browse for the **.olb** file (3)

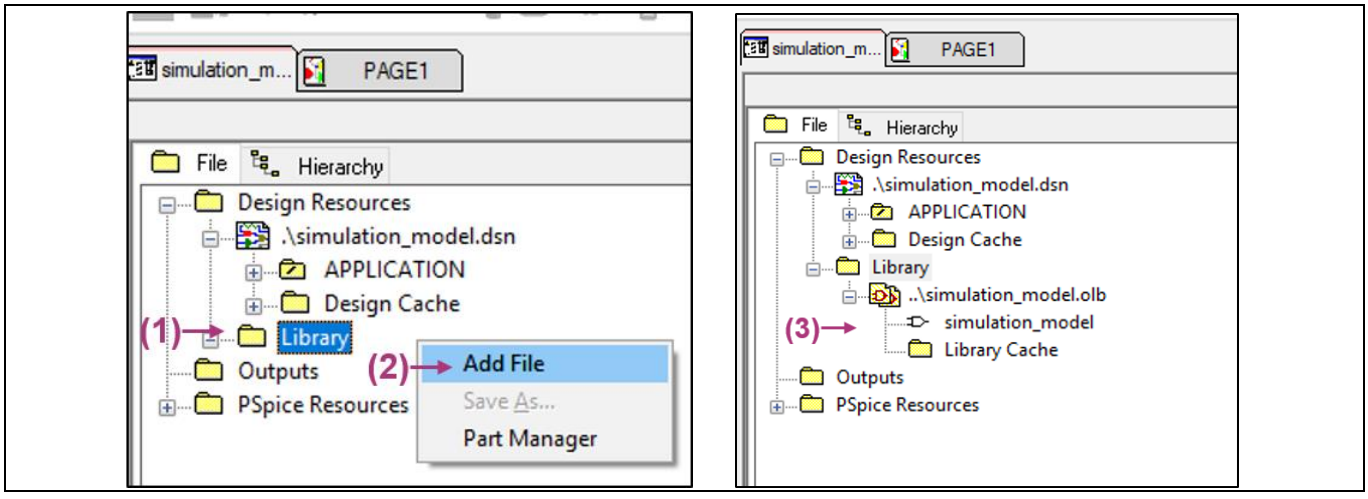


Figure 5 Import .olb file

Search in the **Place Part** sidebar (or press **"P"** on the keyboard) and place the symbol in the desired schematic (4), as shown in **Figure 6**.

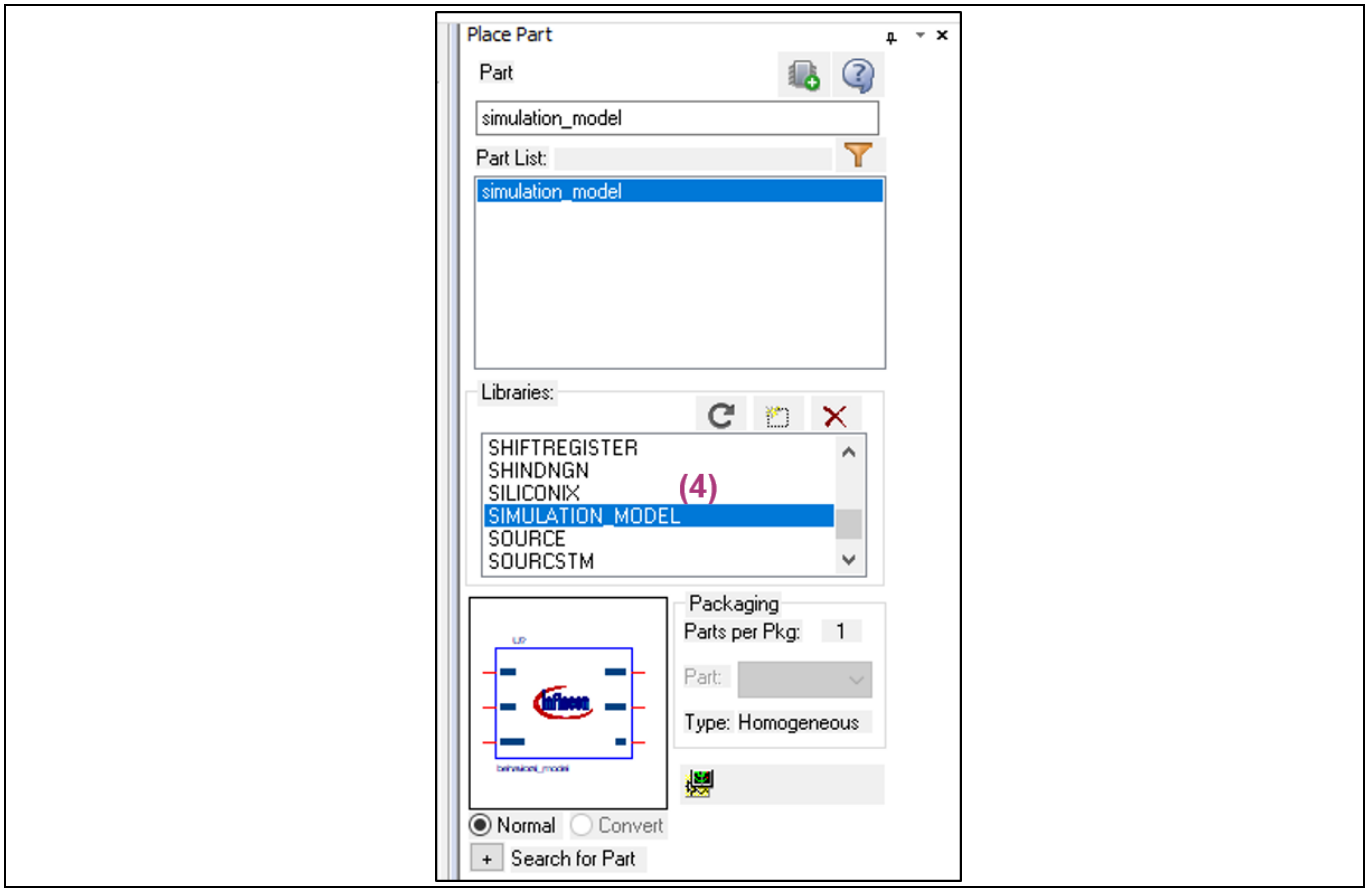


Figure 6 Place part

2.3 Set up and run a transient analysis

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

As shown in **Figure 7**, from the main menu select **PSpice**, choose **New Simulation Profile**, set the name of the simulation profile and click on **Create (1)**.

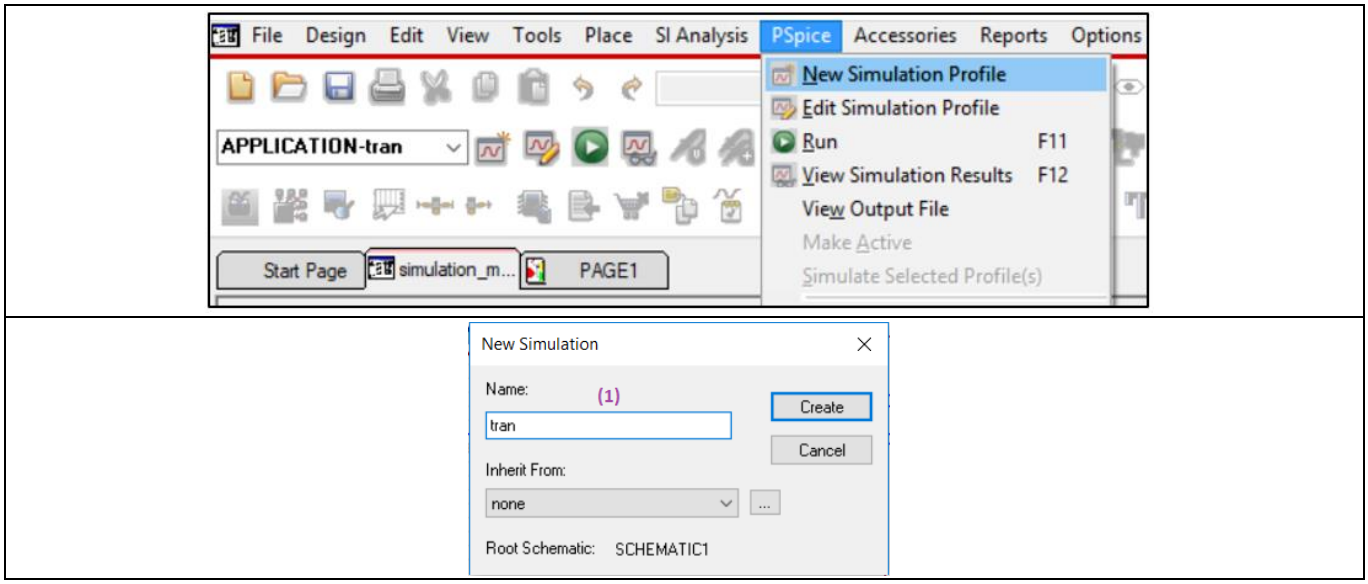


Figure 7 New simulation profile

Figure 8 shows the **Simulation Settings** dialog box that appears. From the **Analysis Type** list, select **Time Domain (Transient)** (2) and specify the required parameters:

- **TSTOP** (3): total simulation time
- **Maximum step size** (4): defines the maximum internal step size, which is dependent on the specified **TSTOP** parameter

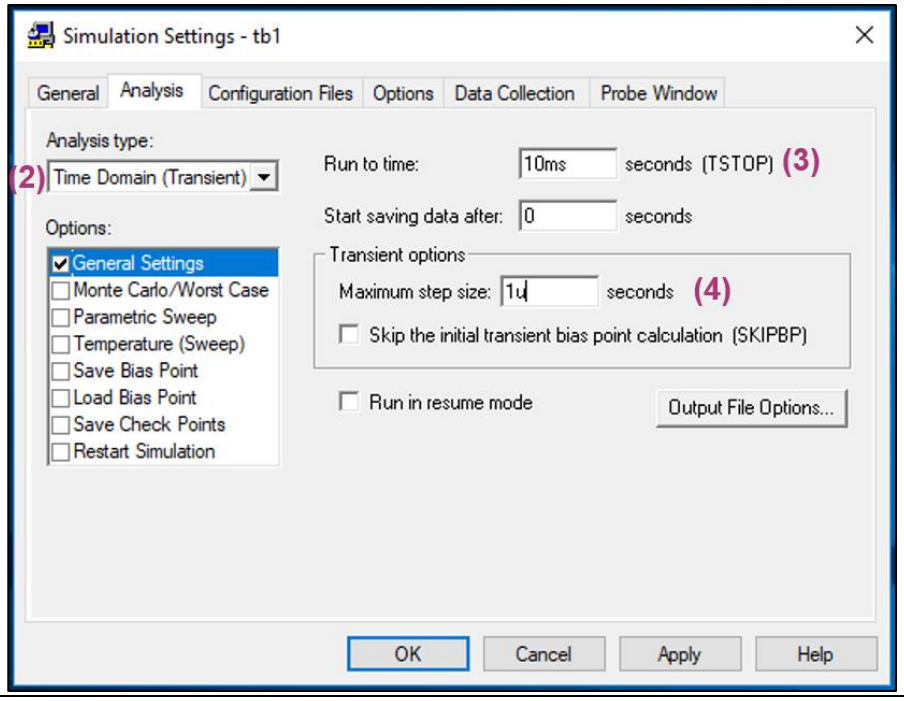


Figure 8 Analysis settings

As shown in **Figure 9**, under **Simulation Settings** go to **Options** (5) tab, select the category **Gate-level Simulation** (6) and then make sure that "**0**" is selected in the dropdown list of the **DIGINITSTATE** (7).

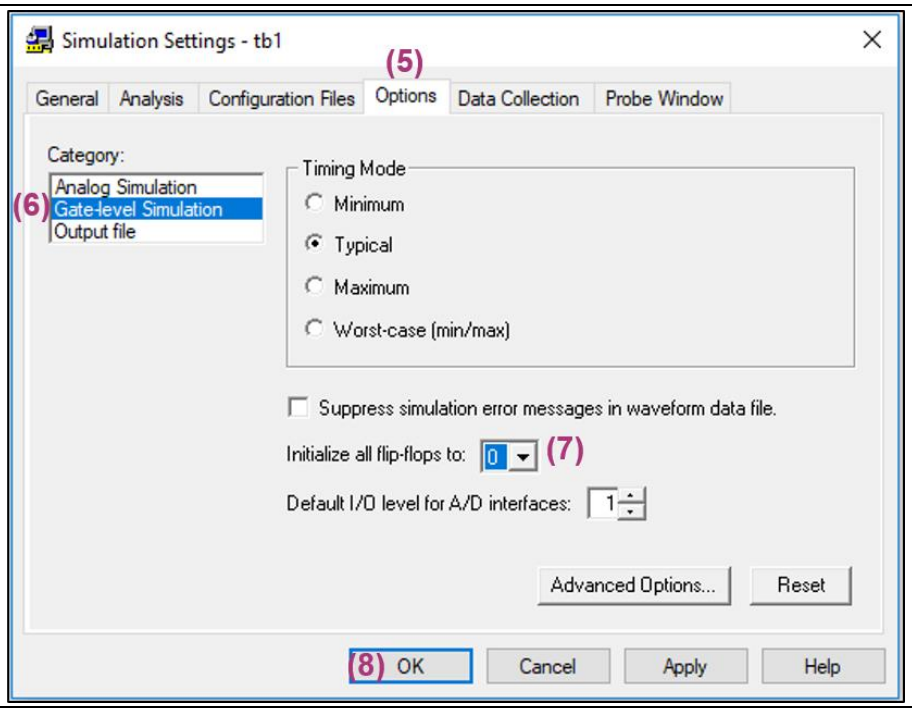


Figure 9 Options settings

Click **OK** to apply the settings and close the **Simulation Settings** dialog box **(8)**. To perform the analysis, press **Run** in the **PSpice** menu, as shown in **Figure 10**.

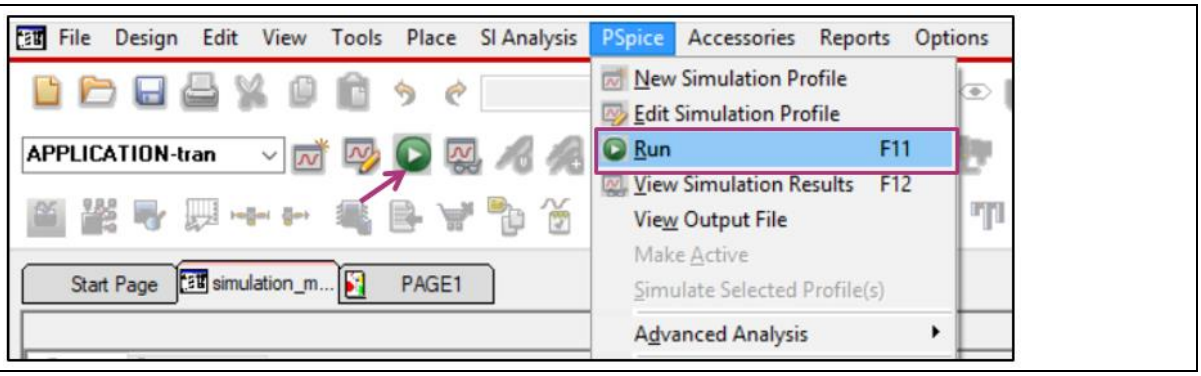


Figure 10 Run the simulation

2.4 Set up global parameters

A model parameter allows 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 11 shows how to define a global parameter in **Cadence OrCAD**. Follow the instructions given below:

- In the **Place Part** sidebar, find the instance **PARAM(1)**
- Select **PARAM** (from the library named "**SPECIAL**"), double click on it and place it on the schematic page **(2)**

*Note: If the "**SPECIAL**" library is missing, load it from **C:\Cadence\SPB_17.2\tools\capture\library\pspice\special.olb** as show in Chapter **2.2.2**.*

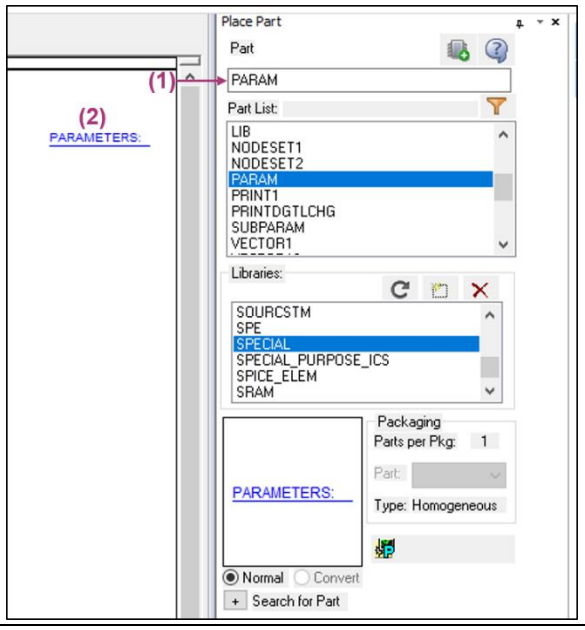


Figure 11 Global parameter

- Double click on **PARAMETERS** to start the spreadsheet editor
- As shown in **Figure 12**, click **New Property** (3), type a **Name** and **Value** and press **OK** (4)

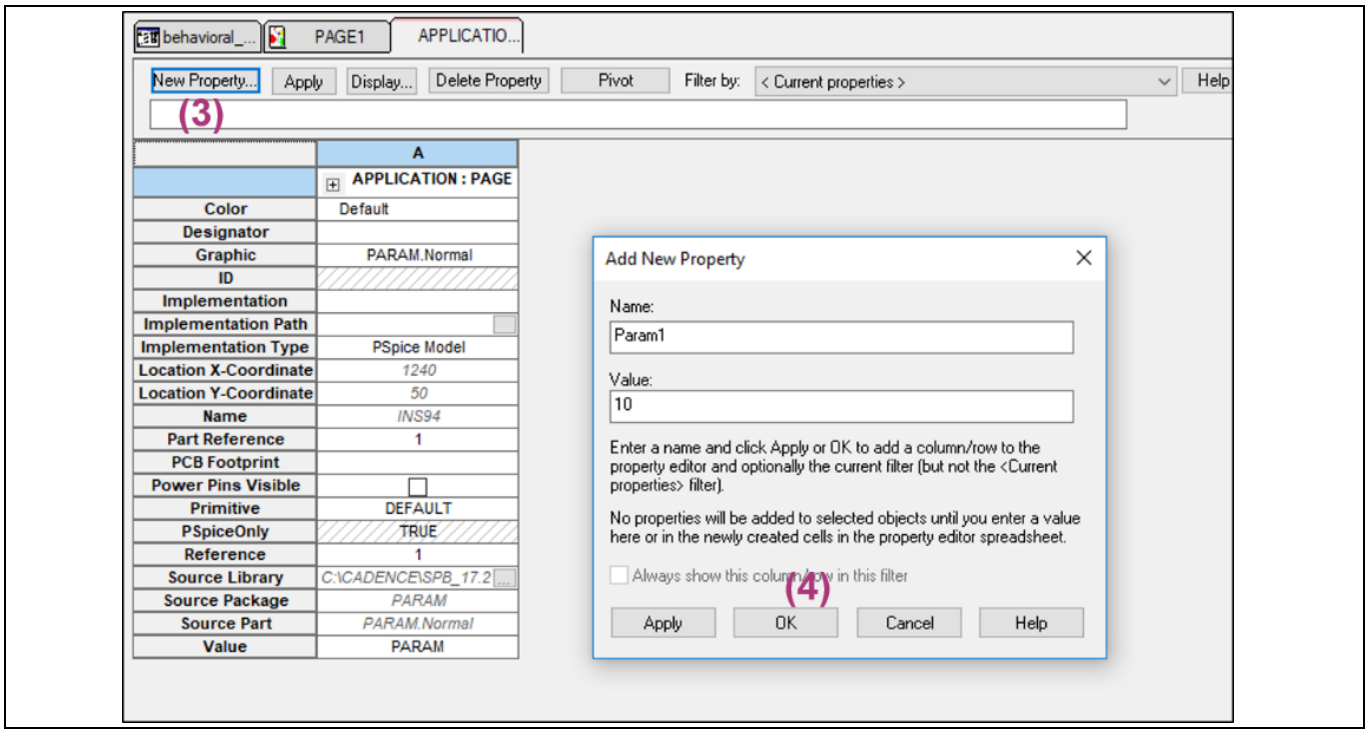


Figure 12 Add new parameter

- Right-click on **Param1** and, as illustrated in **Figure 13** choose the **Display option** (5). Select **Name and Value** (6) and click **OK** (7). This will result in showing the recently created global parameter on the test bench

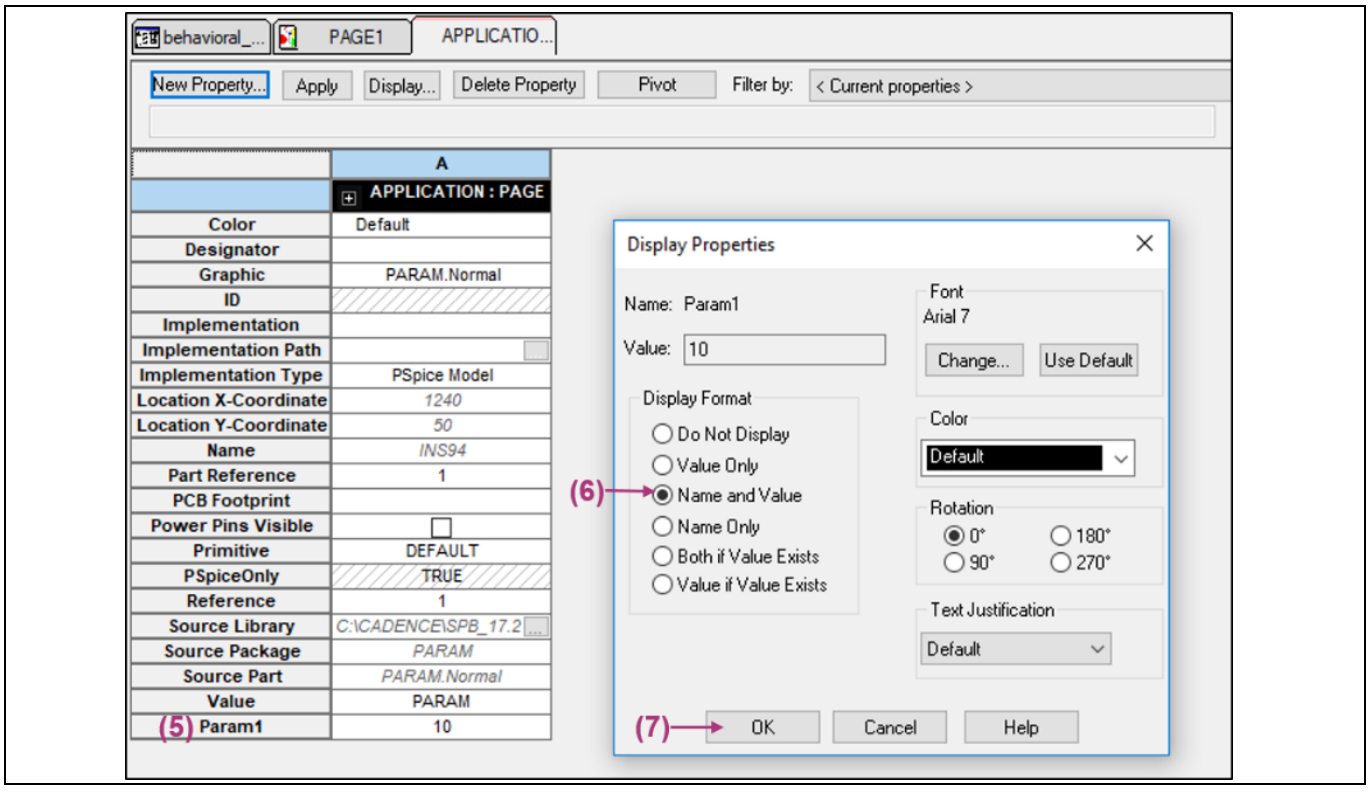


Figure 13 Display a global parameter

Two practical examples are shown below:

Figure 14 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.

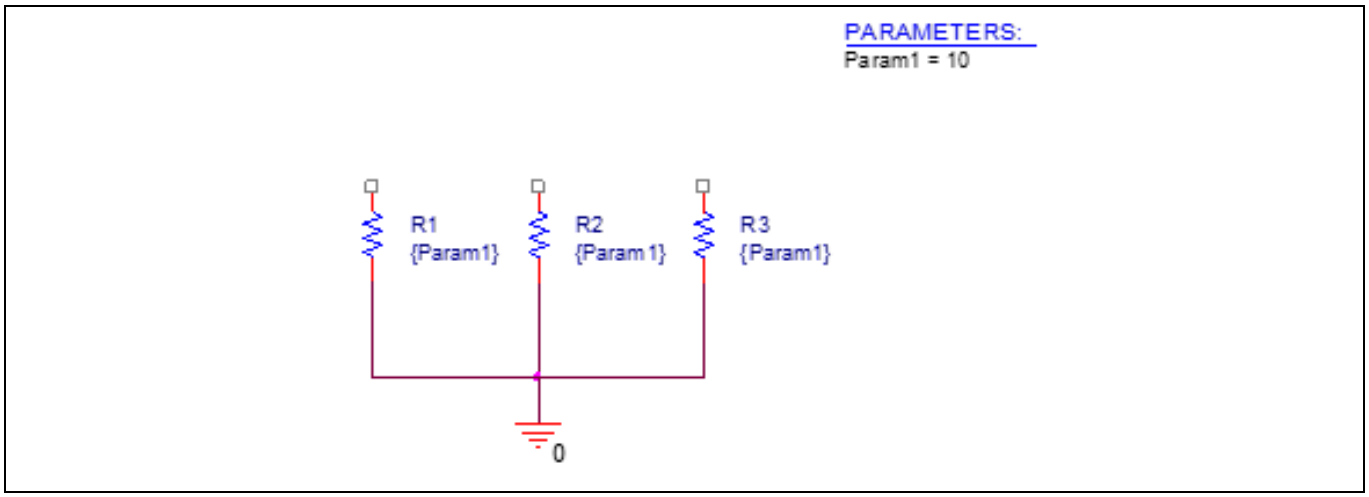


Figure 14 Global parameter for multiple instances

Figure 15 shows the steps required to set up and run a parametric simulation. Perform an analysis to sweep a 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.

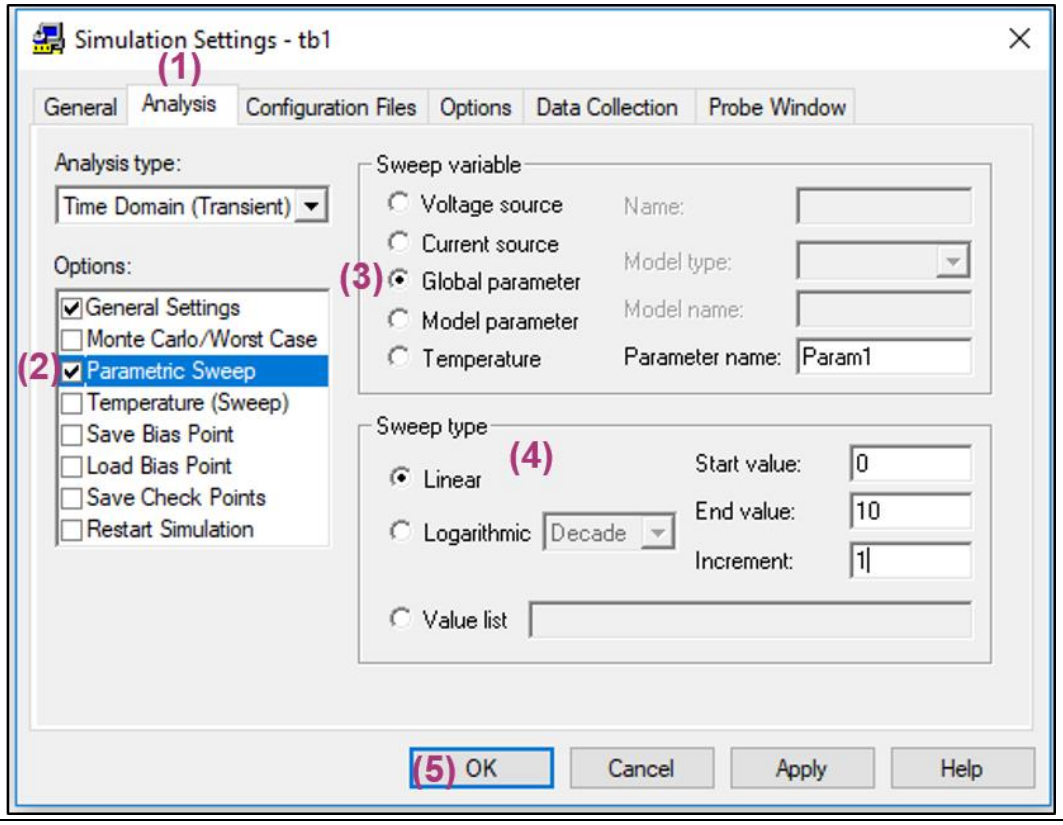


Figure 15 Set up a parametric simulation

2.5 Recommended simulation settings

As **PSpice** was not originally designed for power electronics and highly non-linear components, the simulation parameters (**PSpice** → **Edit Simulation Profile**: **Options**) can be optimized for a specific design. **Figure 16** illustrates an example.

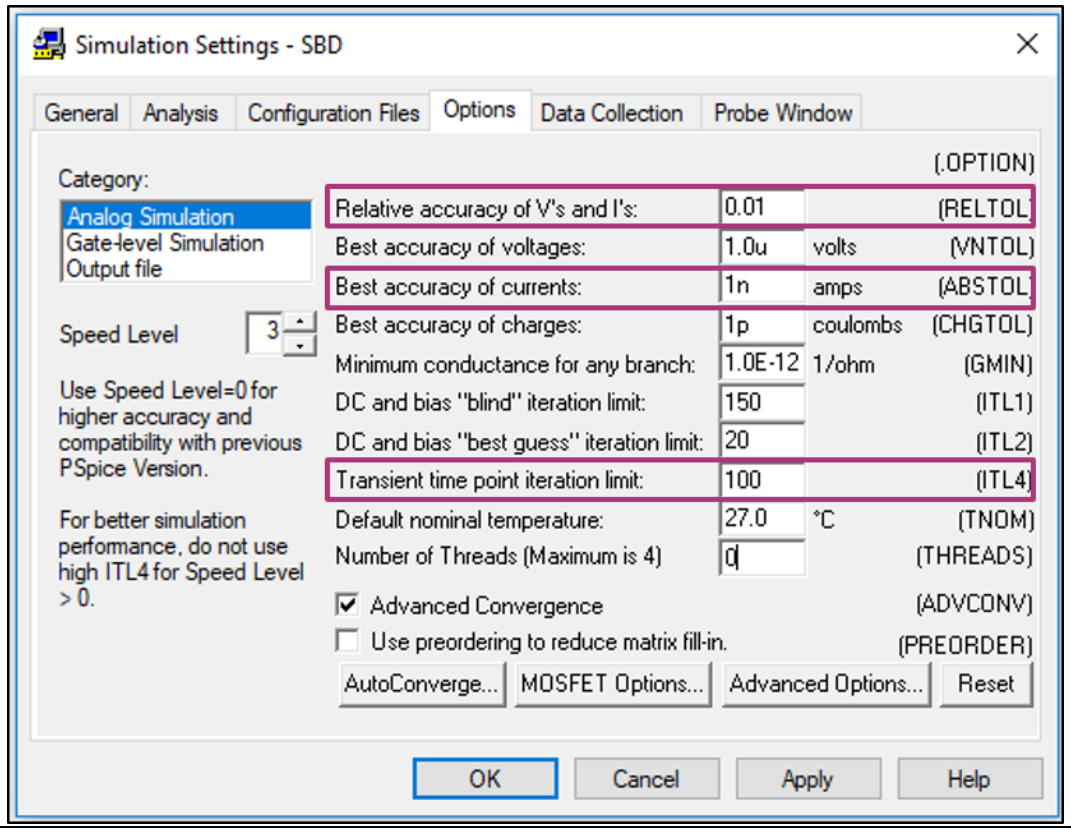


Figure 16 Simulation parameters

Table 1 below provides the recommended values to facilitate convergence.

Table 1 Recommended values to facilitate convergence

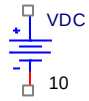
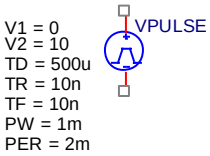
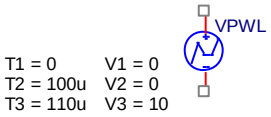
RELTOL=0.01	Relative accuracy of voltage and currents
ABSTOL=1nA	Maximum current accuracy
ITL4=100	Maximum number of iterations for transient analysis

The delivered models need to detect a rising edge starting from 0 V on inputs and supply voltages for a proper start-up sequence. It is recommended to use **VPWL** or **VPULSE** instead of **VDC** component.

Note: In order to prevent potential simulator convergence issues, avoid using voltage sources such as VDC 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 **VPULSE** or **VPWL**.

Table 2 Input and supply voltage sources

VDC	VPULSE	VPWL
		
		

3 Opening an existing project

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

3.1 Steps to run the simulation:

Note: *A fully licensed version of **Cadence OrCAD** for Windows is required to be installed. Extract the files to C:\ drive.*

As shown in **Figure 17**, open the **OrCAD Capture CIS** application and from the **File** menu select **Open** → **Project.** In the opened window locate and open the **.opj** file package directory.

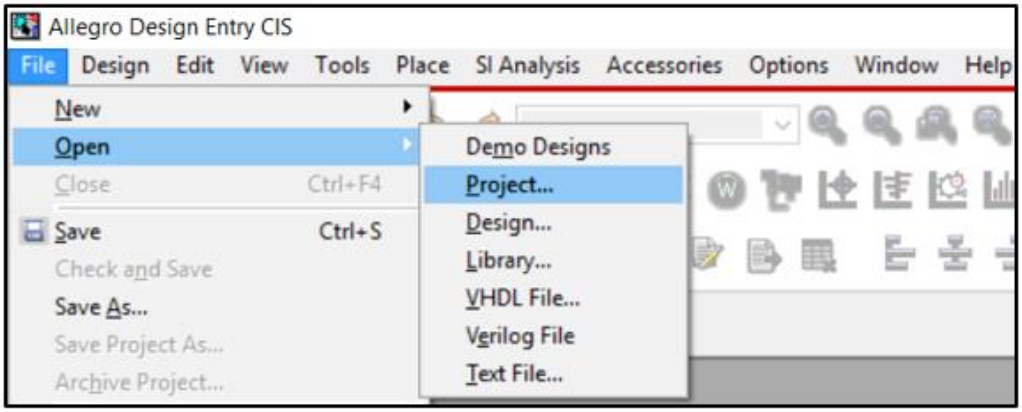


Figure 17 Open an existing project

Figure 18 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 (**(1)** and **(2)**), part libraries **(3)**, netlists, simulation models, simulation profiles **(4)**, stimulus files, and any other related information.

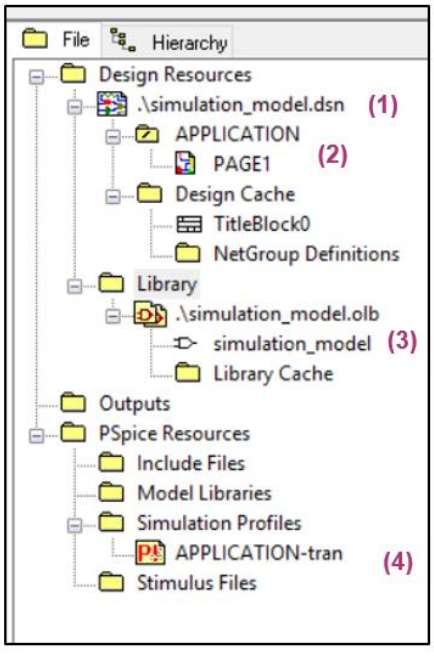


Figure 18 Project manager

As visible on **Figure 19**, Users can select from already available test benches (with included simulation profile and predefined waveforms probes).

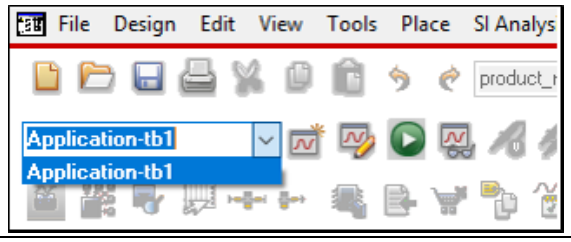


Figure 19 Select the application test bench

The simulation is started by pressing "F11" on the keyboard or by opening the **PSpice** menu and clicking **Run**. Alternatively, the **Play** button on the PSpice toolbar can be used, as illustrated in **Figure 20**.

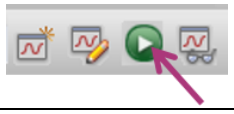


Figure 20 Start the simulation

3.2 Simulation results and measurements

Once the netlist has been generated the **"Allegro AMS Simulator"** (PSpice) will open and the simulation will automatically start. As **Figure 21** shows, a progress bar will be displayed in the bottom right corner **(5)**.

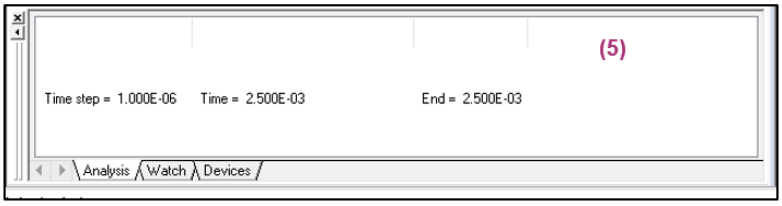


Figure 21 Simulation progress

After the simulation reaches 100%, the signals can be plotted. As shown in the **Figure 22** below, click on the **Trace** menu → **Add trace (6)**. Alternatively, it is possible to press the **Add trace** button (toolbox section) or **Insert** button (keyboard). As visible in **Figure 23**, the respective signals need to be selected from the list to plot them. After the selection is done press the **OK** button **(7)**.

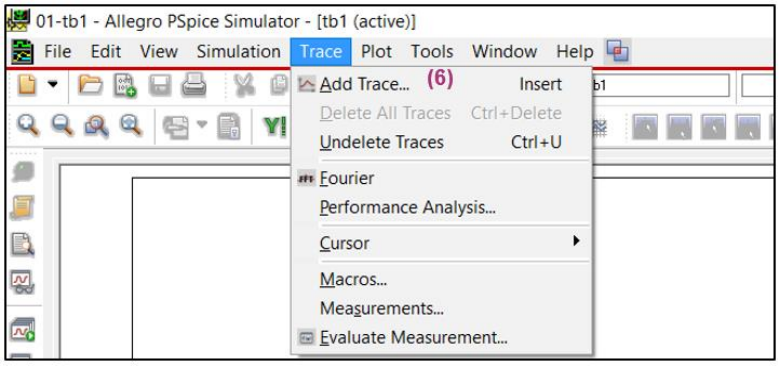


Figure 22 Plot signals

*Note: The desired signals can be filtered by using the check boxes in the center of the window according to their type (e.g. voltages, currents, power, etc.). **(8)**). Signals can also be composed by selecting the appropriate functions available in the right list "**Functions or Macros**" **(9)**.*

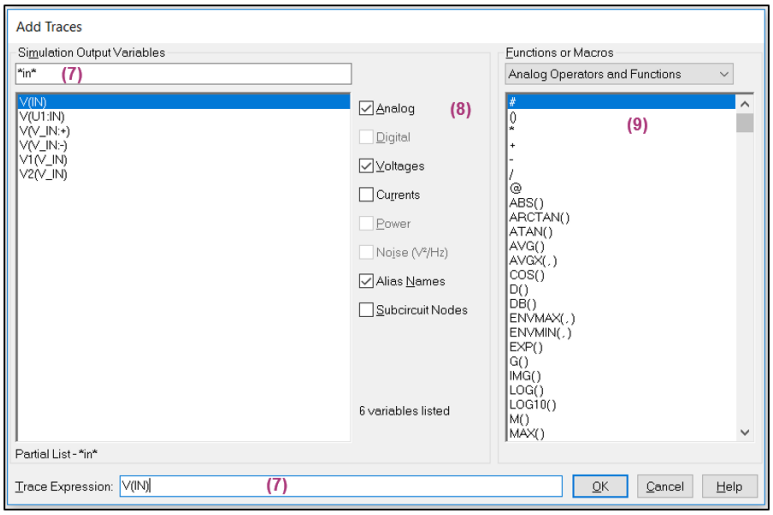


Figure 23 Plot signals

Figure 24 below explains how to add a new plot by clicking **Plot** → **Add Plot to Window**.

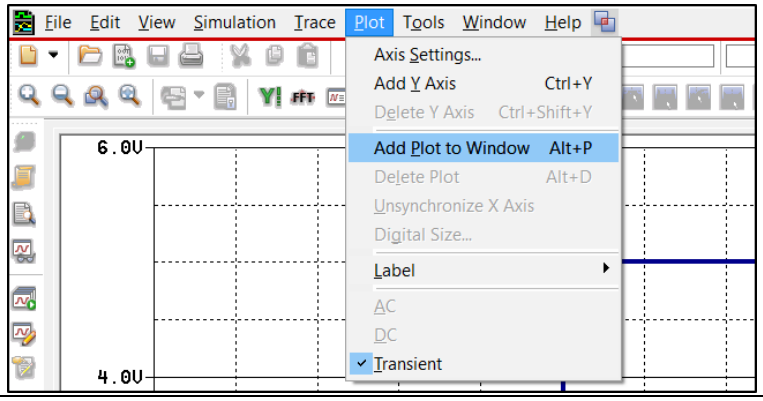


Figure 24 Add a new plot

See **Figure 25** for adding of cursors for signal value measurement by choosing **Trace** → **Cursor** →**Display**. Alternatively, this can be done by a left-click onto the **Display Cursors** button (toolbox).

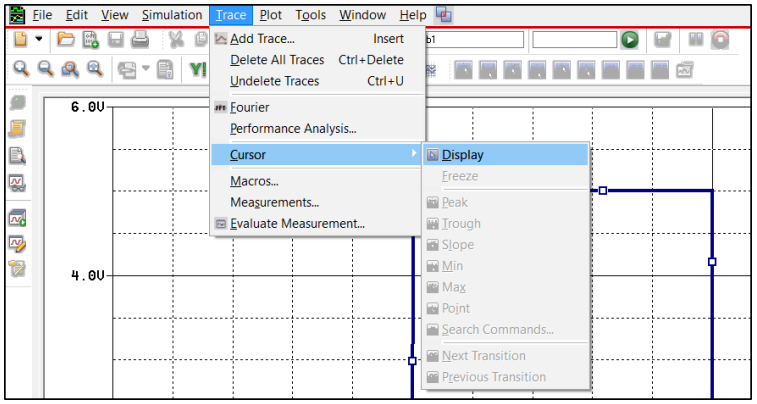
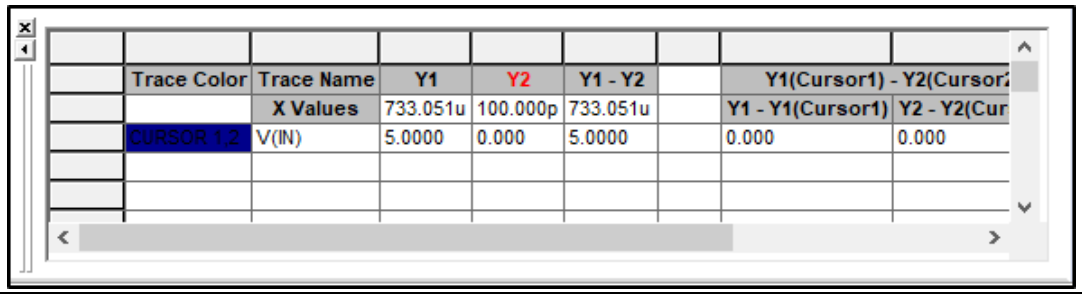


Figure 25 Add cursors

Figure 26 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 26 Measurement values

4 Disclaimer

By downloading and/or using this Infineon model (“Model”), the user (including you) agrees to be bound by the Terms of Use here stated. If user does not agree to all terms of use here stated, user shall not download, use or copy the Model but immediately delete it (to the extent that it was downloaded already).

4.1 Scope of use

a) Any use of this Model provided by Infineon Technologies AG is subject to these Terms of Use

b) This Model, provided by Infineon, does not fully represent all of the specifications and operating characteristics of the product to which this Model relates

c) This Model only describes the characteristics of a typical product. In all cases, the current datasheet information for a given product is the final design guideline and the only actual performance specification. Although this Model can be a useful tool to evaluate the product performance, it cannot simulate the exact product performance under all conditions and it is also not intended to replace bread-boarding for final verification

4.2 Important notice

d) Infineon Technologies AG (“Infineon”) is not and cannot be aware of the specific application of the Infineon’s Model by User. However, Model may from time to time be used by User in potentially harmful and/or life-endangering applications such as traffic, logistic, medical, nuclear or military applications or in other applications where failure of the Model may predictably cause damage to persons’ life or health or to property (hereinafter "Critical Applications")

e) User acknowledges that Infineon has not specifically designed or qualified the Model for Critical Applications that the Model may contain errors and bugs and that User is required to qualify the Model for Critical Applications pursuant to the applicable local quality, safety and legal requirements before permitting or giving access to any such use

4.3 Confidential information

f) User shall treat ideas, concepts and information incorporated in the Model, the documentation and the content of these Terms of Use (together hereinafter "Confidential Information") as confidential, not disclose it to any third party unless otherwise agreed in writing between User and Infineon, not use it for any other purposes than using the Model for simulation and testing purposes only

4.4 Warranty

g) User acknowledges that the Model is provided by Infineon under these Terms of Use and is provided free of charge and "AS IS" without any warranty or liability of any kind and Infineon hereby expressly disclaims any warranties or representations, whether express, implied, statutory or otherwise, including but not limited to warranties of workmanship, merchantability, fitness for a particular purpose, defects in the Model, or non-infringement of third parties intellectual property rights

h) Infineon reserves the right to make corrections, deletions, modifications, enhancements, improvements and other changes to the Model at any time or to move or discontinue any Model without notice

4.5 Liability

i) Nothing in these Terms of Use shall limit or exclude Infineon's liability under mandatory liability laws, for injuries to life, body or health, for fraudulent concealment of defects in the software, or in cases of Infineon's intentional misconduct or gross negligence

j) Without prejudice to Section 4.5.i), in cases of Infineon's slight negligent breach of obligations that restrict essential rights or duties arising from the nature of these Terms of Use in a way that there is a risk of non-achievement of the purpose of these Terms of Use, or of an obligation whose observance User regularly may trust in and whereas compliance with, only makes proper execution of these Terms of Use possible, Infineon's liability shall be limited to the typically, foreseeable damage

k) Without prejudice to Sections 4.8 n)and Sections 4.8 o), Infineon's liability under this Agreement shall be excluded in all other cases

4.6 Export regulations

l) The User shall comply with all applicable national and international laws and regulations, in particular the applicable export control regulations and sanction programs. The User also agrees not to export, re-export or transfer any software or technology developed with or using information, software or technology offered by Infineon, in violation of any applicable laws or regulations of the competent authorities. Further, the User shall neither use any products, information, software and technology offered by Infineon in or in connection with nuclear technology or weapons of mass destruction (nuclear, biological or chemical) and carriers thereof nor supply military consignees

4.7 Termination of use permit

m) If the User violates these Terms of Use, such User’s permit to use this Model terminates automatically. In addition, Infineon may terminate the User’s permit to use this Model at its discretion and at any time regardless of any violation of these Terms of Use. In any of the foregoing events, the User is obliged to immediately destroy any content that has been downloaded or printed from Infineon’s website

4.8 Miscellaneous

n) These Terms of Use are subject to the laws of the Federal Republic of Germany with the exception of the United Nations on Purchase Contracts on the International Sale of Goods dated April 11, 1980 (CISG). The exclusive place of jurisdiction is Munich, Germany

o) Should any provision in these Terms of Use be or become invalid, the validity of all other provisions or agreements shall hereby remain unaffected

Revision history

Document version	Date of release	Description of changes
01.00	2021-08-04	Initial release

Trademarks
All referenced product or service names and trademarks ar2021s.

Edition 2021-08-04

Published by

Infineon Technologies AG

81726 Munich, Germany

© 2021 Infineon Technologies AG. All Rights Reserved.

Do you have a question about this document?

Email: erratum@infineon.com

Document reference
Z8F80183059

IMPORTANT NOTICE

The information contained in this application note is given as a hint for the implementation of the product only and shall in no event be regarded as a description or warranty of a certain functionality, condition or quality of the product. Before implementation of the product, the recipient of this application note must verify any function and other technical information given herein in the real application. 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) with respect to any and all information given in this application note.

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.

For further information on the product, technology, delivery terms and conditions and prices please contact your nearest Infineon Technologies office (**www.infineon.com**).

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.