Introduction to Simulation Environment

Targeted simulator: OrCAD

About this document

Scope and purpose:

The current document provides guidance on how to get started with the attached behavioral model using the OrCAD Simulator either by creating a new project from scratch or based on an existing testbench like the one attached to the current archive.

Important note:

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

Table of contents

1. Resources 3
2. Starting a new project 4
2.1. Create a new project 4
2.2. Import of an existing model 5
2.2.1. Importing a .lib file 5
2.2.2. Importing an .olb file 6
2.2.3. Generating an .olb file 7
2.3. Set up and run a transient analysis 9
2.4. Set up global parameters 10
2.5. Recommended simulation settings 13
3. Opening an existing project 15
3.1. Transient simulation considerations 15
4. INFINEON'S SIMULATION MODEL TERMS OF USE 19
Revision history 21

1. Resources

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

- **.lib :** the model code – to be found either as encrypted (to protect Infineon's IP) or unencrypted format
- **.olb :** schematic symbol view for graphical user interface
- **.opj** : project file which contains one/several application setup(s)

2. Starting a new project

The instructions presented below assume basic knowledge like opening a schematic and running a simulation. If necessary, please get familiar with simulation environment before proceeding.

2.1. Create a new project

Begin by opening the OrCAD Capture CIS application. From **File** menu→**New**→**Project**, select the project type as **PSpice Analog Mixed A/D** to enable simulation and then specify the name of the project and a directory as storage placeholder. For example:

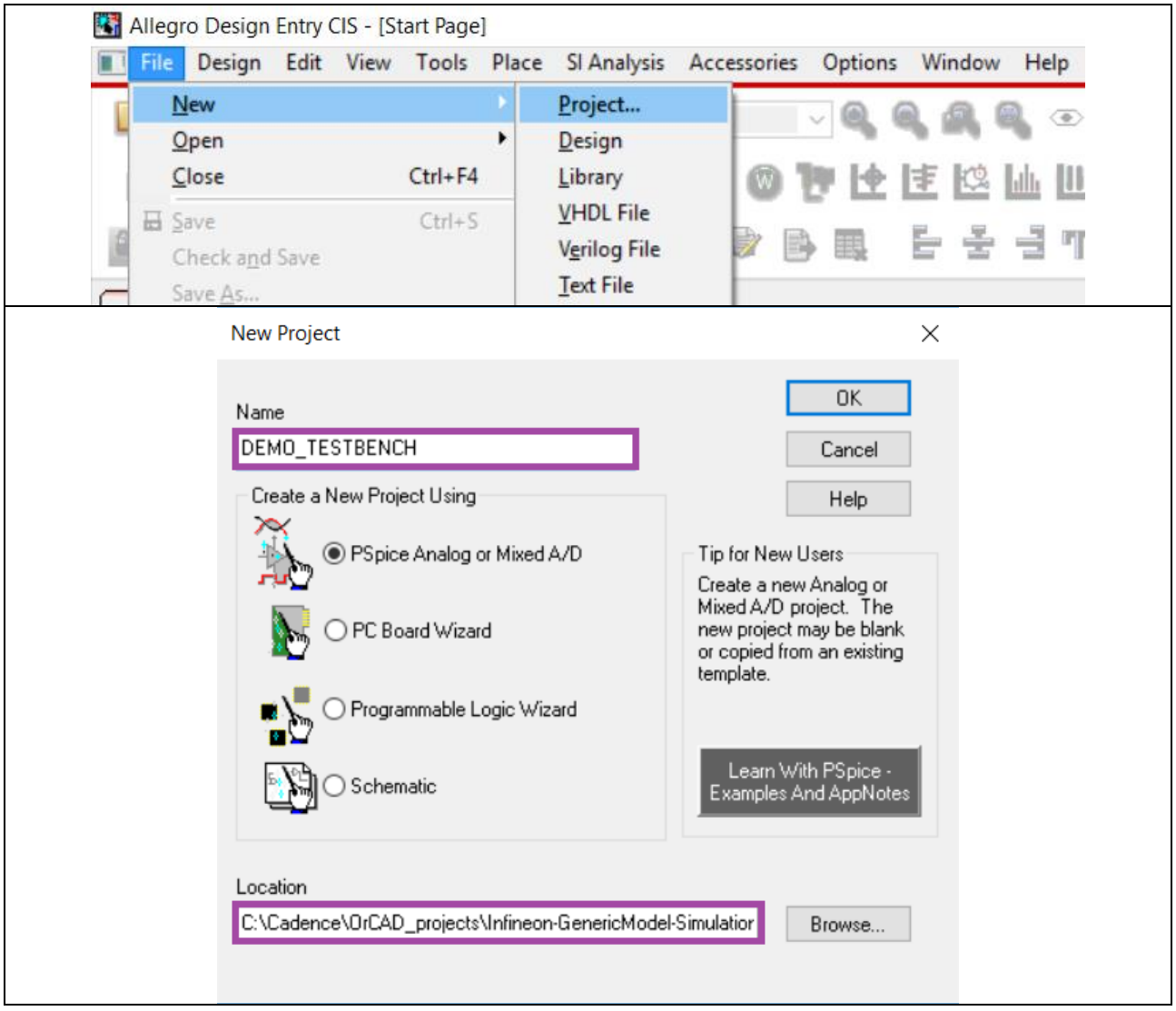


Figure 1 Create new project

- After pressing OK, select the **Create a blank project** option in the **Create PSpice Project** pop-up and confirm with OK (Figure 2)

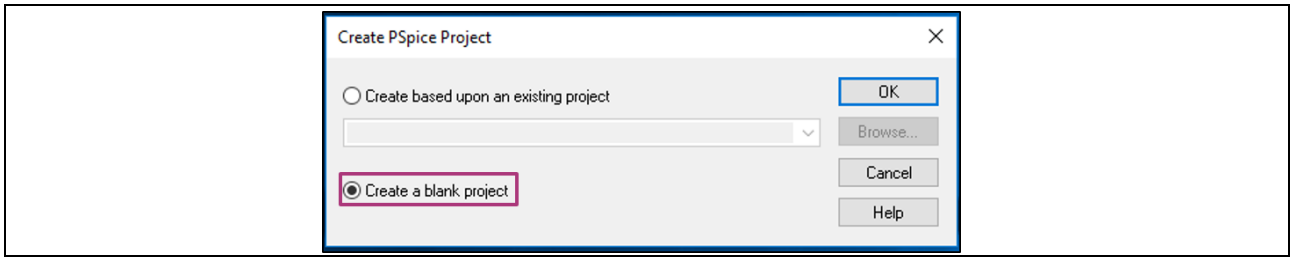


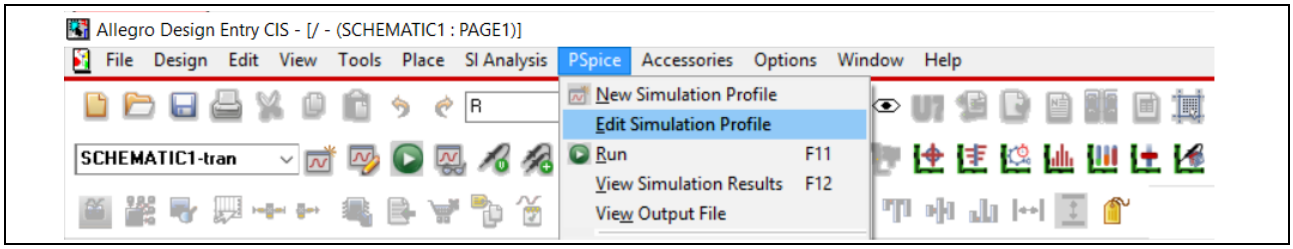
Figure 2 Blank project

2.2. Import of an existing model

To simulate a model in schematic mode, **Cadence OrCAD** requires two types of input files: **"GENERIC_MODEL.LIB"** (device behavior source code) and **"GENERIC_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

- From **PSpice** menu choose **Edit Simulation Profile**.
- Once the **Simulation Settings** window pops up navigate to the **Configuration Files (1)** tab and under **Category** select **Library (2)** (Figure 3)
- 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:
  - **Add as Global (4)** - to all designs
  - **Add to Design (4)** - for current design only (recommended)
  - **Add to Profile (4)** - local to the current test-bench inside the current design
- 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)**



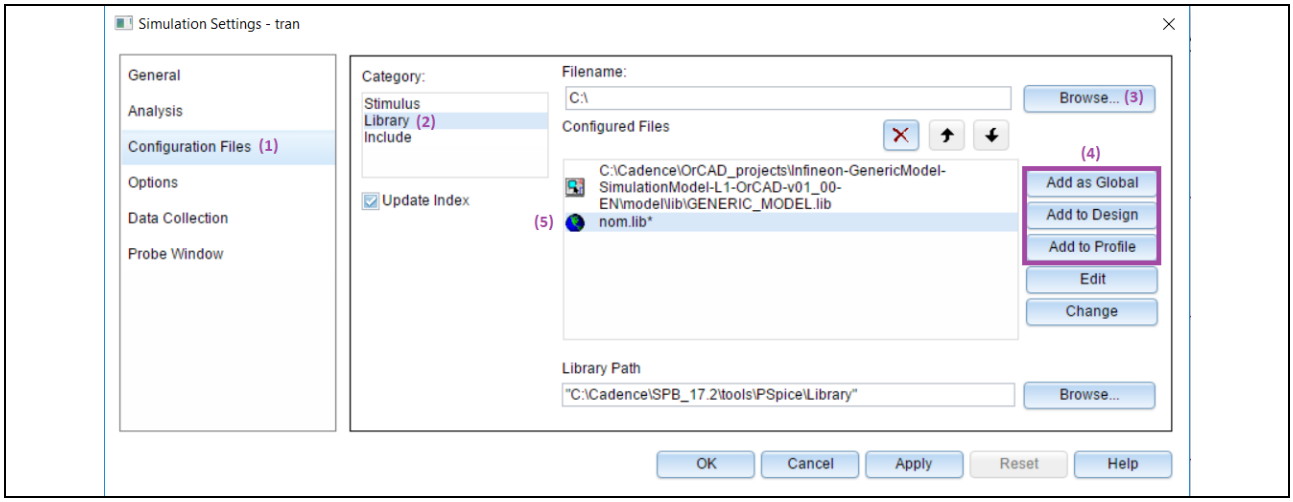


Figure 3 Import .lib file

2.2.2. Importing an .olb file

- 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)**

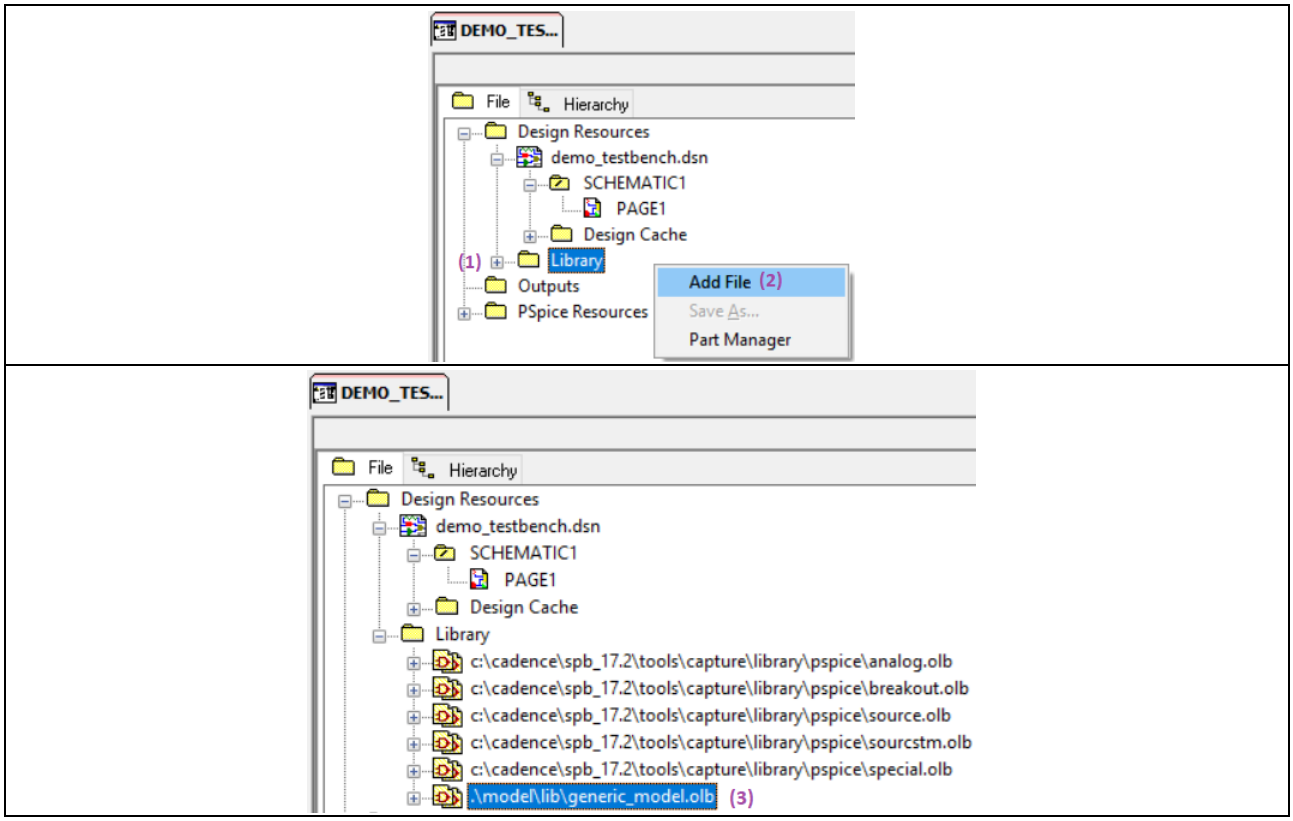


Figure 4 Import .olb file

- Search in the Place Part sidebar or press “P” on the keyboard and place the symbol in the desired schematic **(4)**

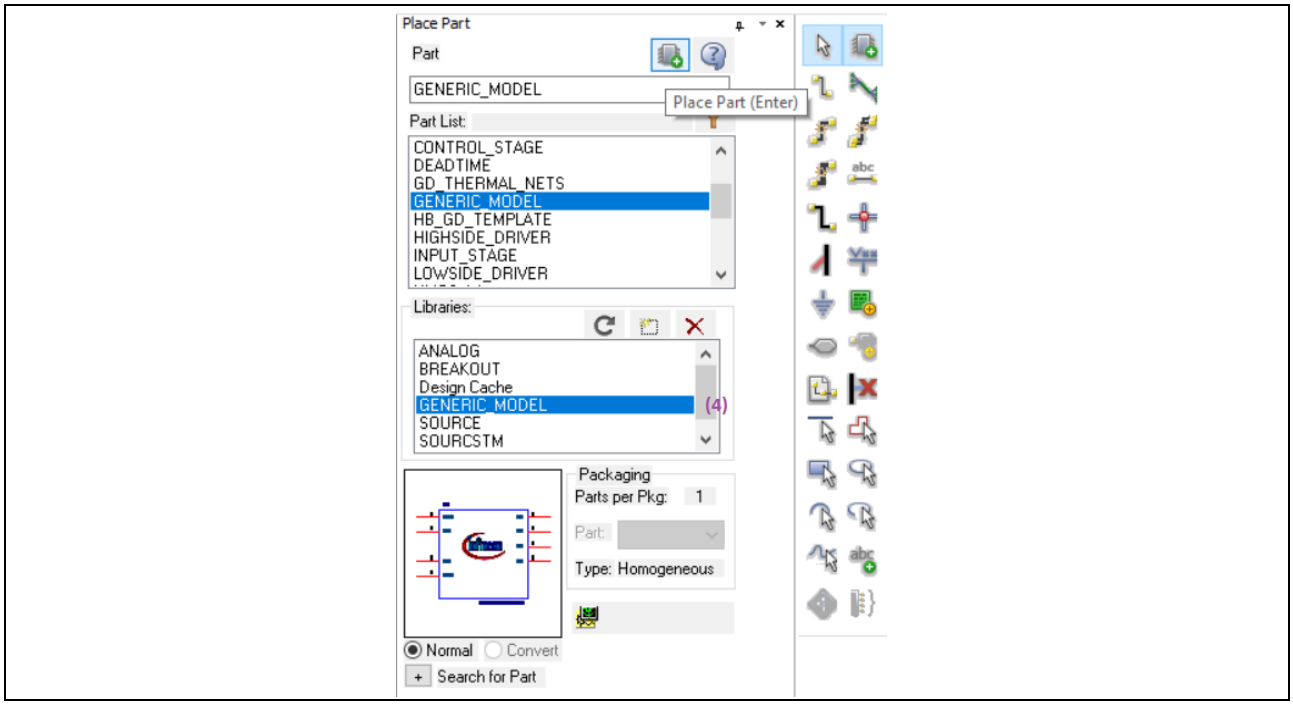


Figure 5 Place part

2.2.3. Generating an .olb file

- Open the AMS Model Editor tool from the Cadence suite.

NOTE: Please note that the provided symbol files are compatible with all major releases of OrCAD starting with version 16.2 onward.

- From **File** menu→**Open**, **(1)** navigate to the "lib" directory in the package and select the "GENERIC_MODEL.lib" file. **(2)** (Figure 6)

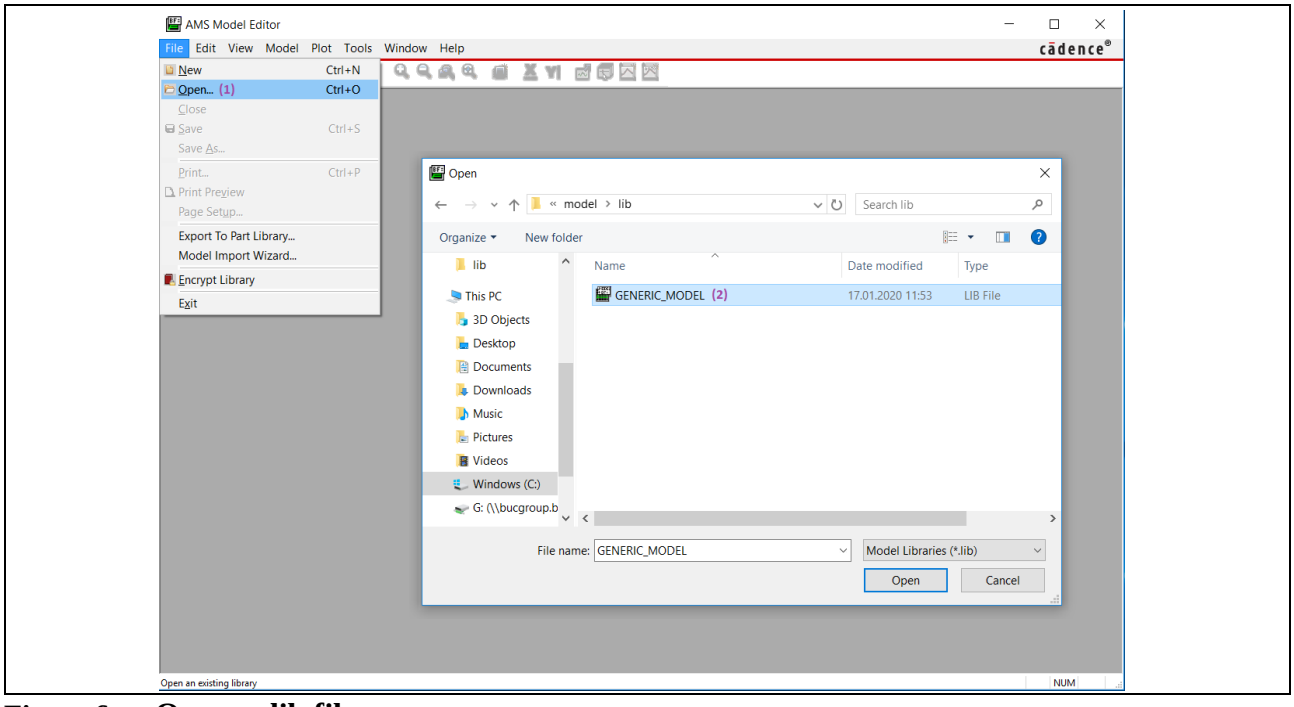


Figure 6 Open a .lib file

- From **File** menu→**Export to Capture Part Library…** **(3)** press the lower **Browse** button next to the "Enter Output Part Library:" location bar **(4),** if you want another location for the symbol and select the new desired save location for the symbol file; otherwise, after **(3)** just click OK (Figure 7)

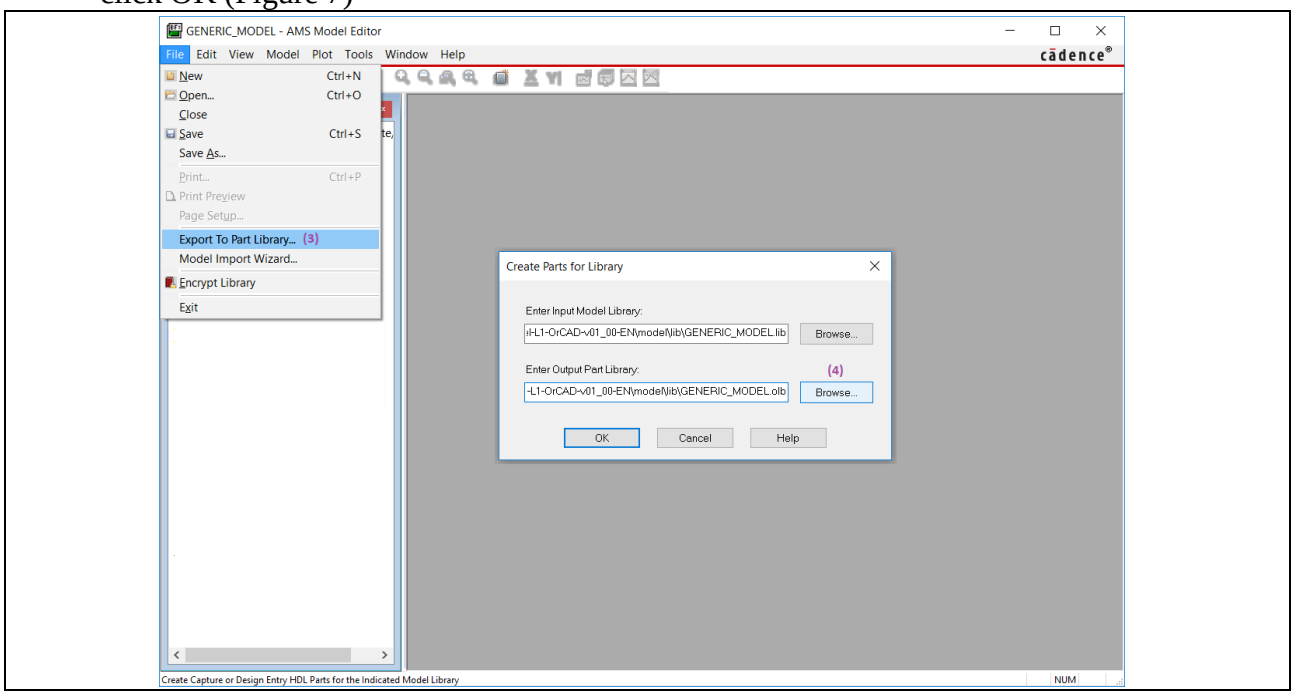


Figure 7 Generate a new symbol

- The report will confirm the creation of the symbol with 0 error and 0 warning messages

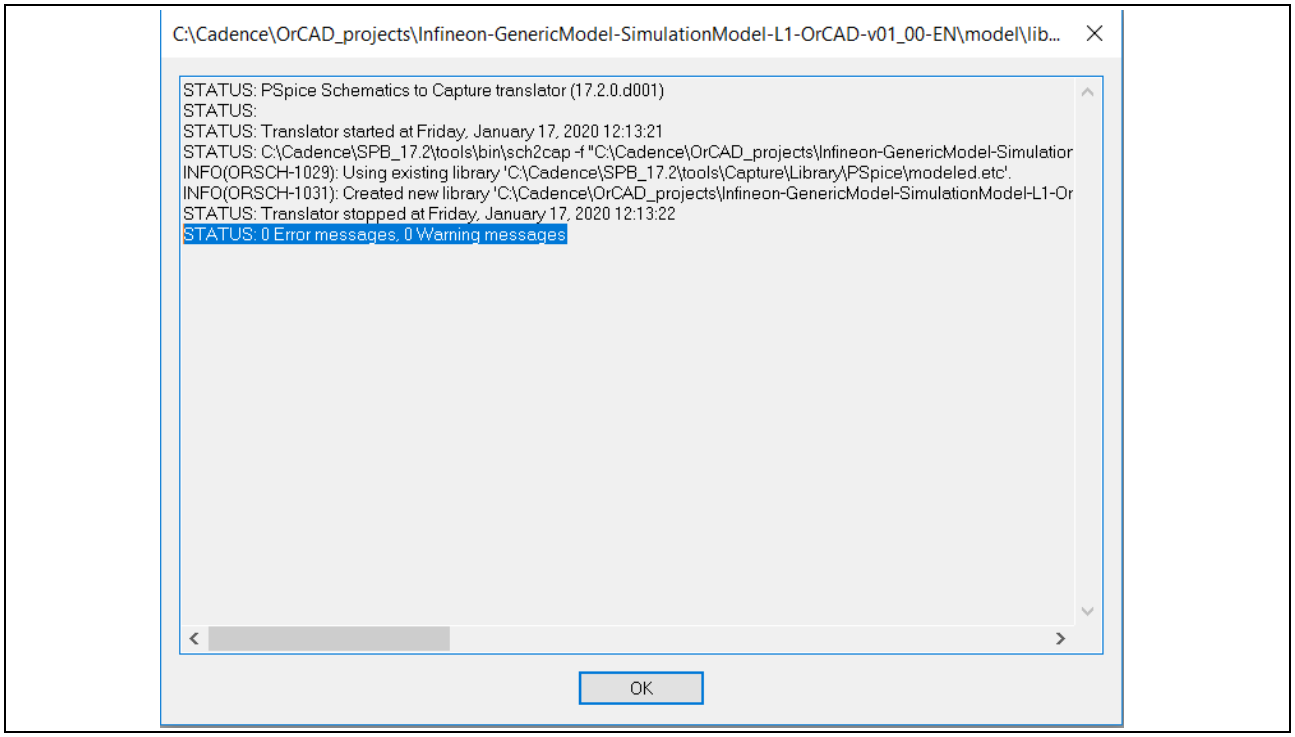


Figure 8 Successful message

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:

- From the main menu select **PSpice** and choose **New Simulation Profile**
- Set the name of the simulation profile and click on **Create** (1)

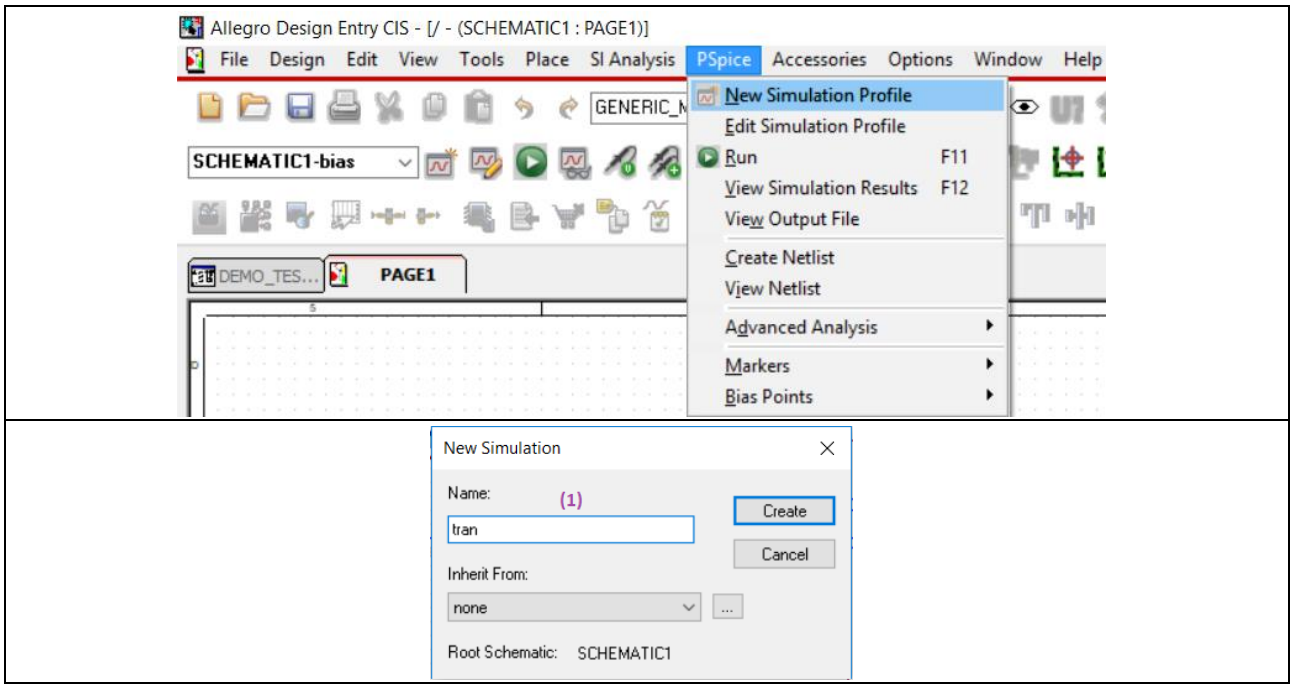


Figure 9 New simulation profile

- **Simulation Setting** dialog box appears

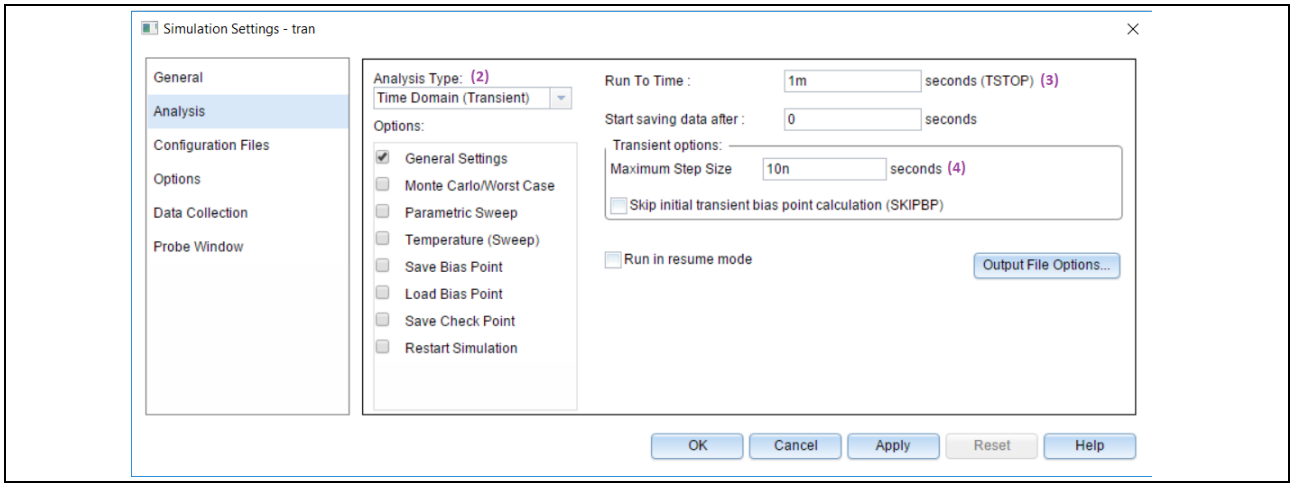


Figure 10 Simulation settings

- 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

- Under **Simulation Settings** go to **Options (5)** tab, select the Category **Gate Level Simulation (6)** and then make sure that **"0"** is selected from the dropdown list of the **DIGINITSTATE (7)**:

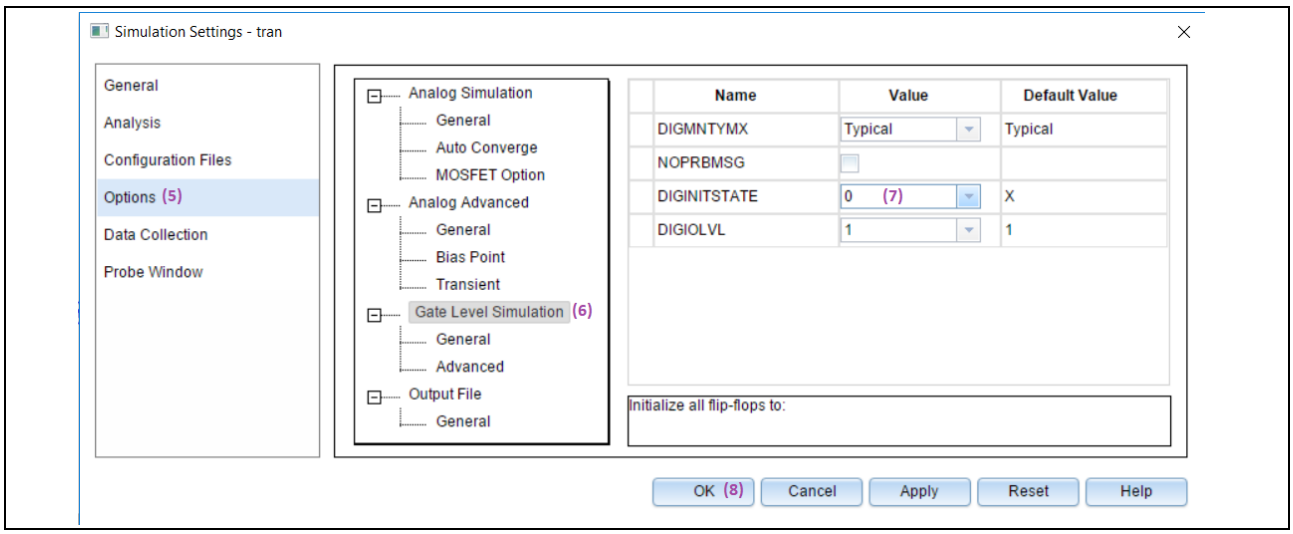


Figure 11 Simulation settings

- Click **OK** to close **Simulation Setting** dialog box **(8)**
- From **Pspice** menu, press **Run** to perform the analysis

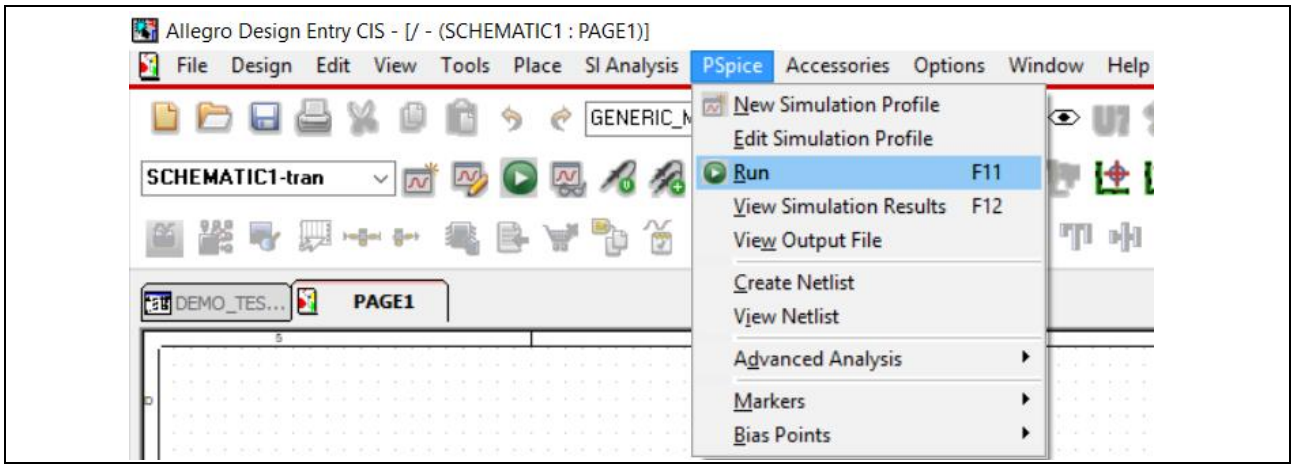


Figure 12 Run the simulation

2.4. Set up global parameters

A model parameter allows the user to have a direct control over specific components inside the circuit at any hierarchical level with the following purposes:

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

To declare a global parameter in Cadence OrCAD:

- Search in **Place Part** sidebar the instance **PARAM (1)**
- Select **PARAM** (from the library named **"SPECIAL"**), double click on it and place it on your schematic page **(2)**
- If the **"SPECIAL"** library is missing, load it from: **C:\Cadence\SPB_17.2\tools\capture\library\pspice\special.olb** as show in the section above – **2.2.2. Importing an .olb file**

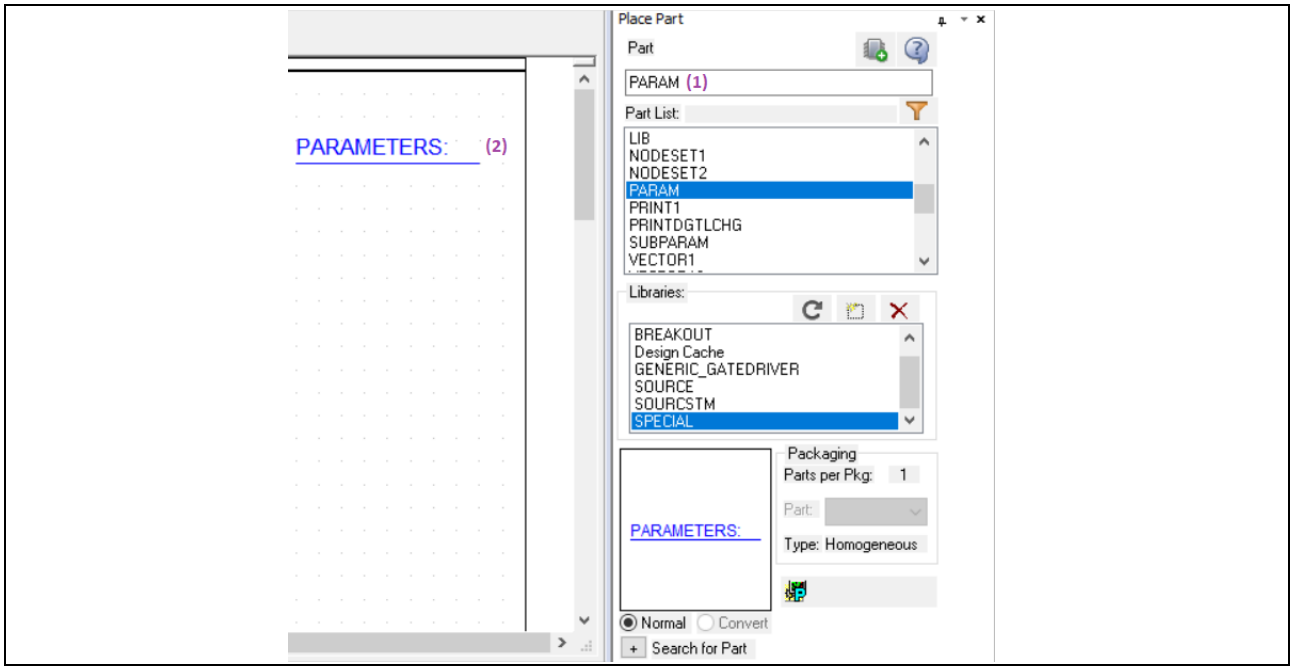


Figure 13 Global parameter

- Double click it to start the spreadsheet editor
- Click New Property **(3)**
- Type the Name and Value and press OK **(4)**

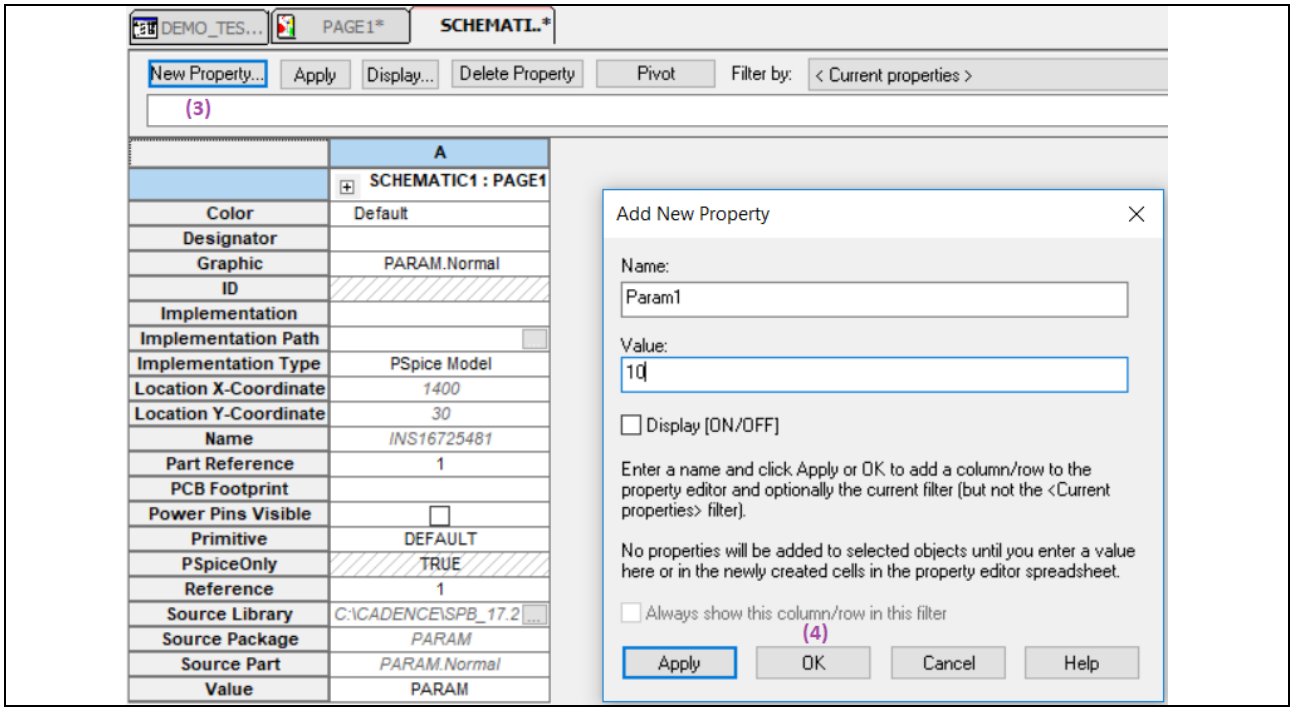


Figure 14 Add new parameter

- Right-click on Param1 on the **Display option (5)**, select **Name and Value (6)**, then click **OK (7)** (it will show up on the testbench)

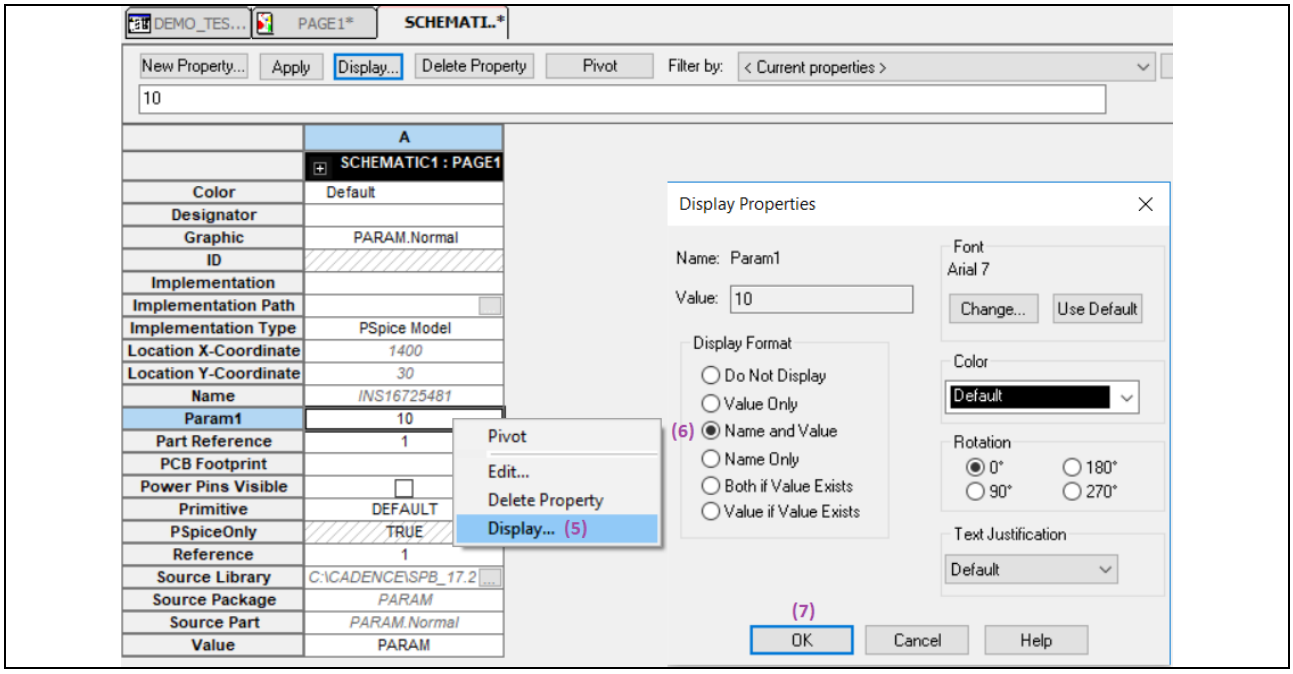


Figure 15 Display a global parameter

Two practical examples are shown below:

- Apply the same value to multiple instances: in the value field of any component use the parameter name between curly brackets

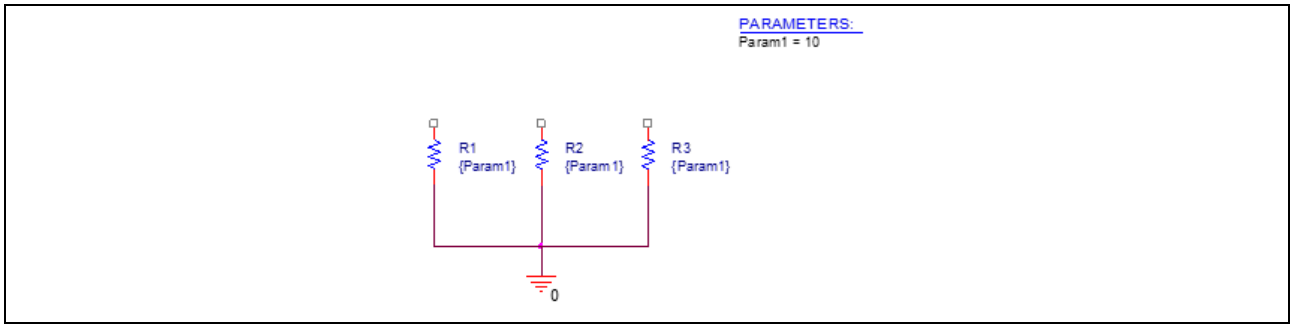


Figure 16 Global parameter for multiple instances

- Set up 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. To perform this analysis follow the steps from **Figure 17**.

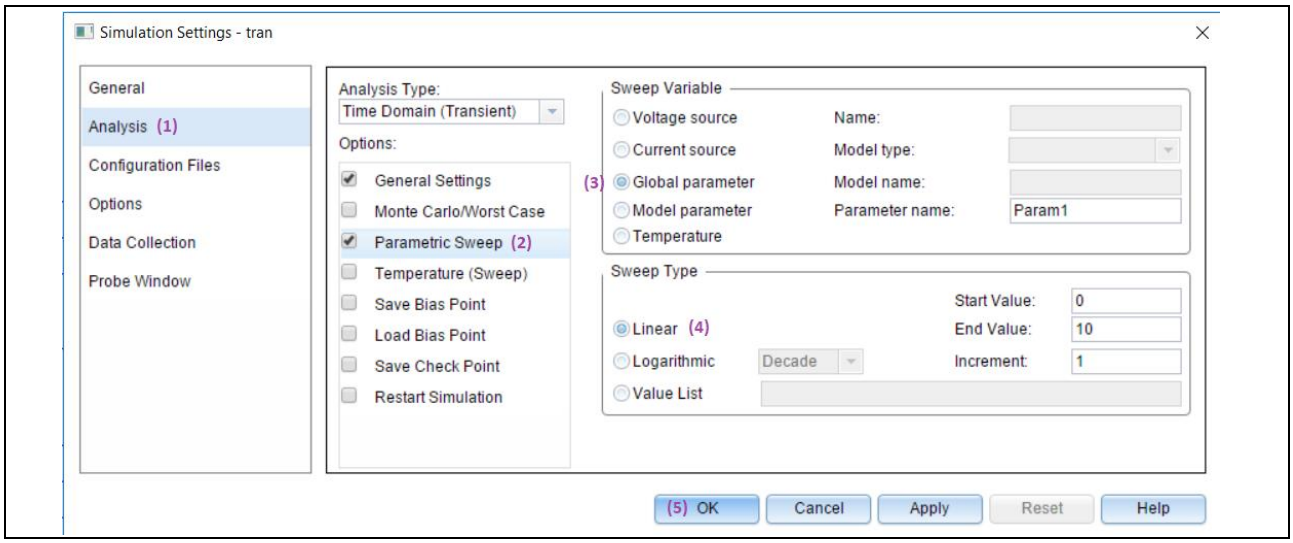


Figure 17 Set up a parametric simulation

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**) may be optimized for a specific design.

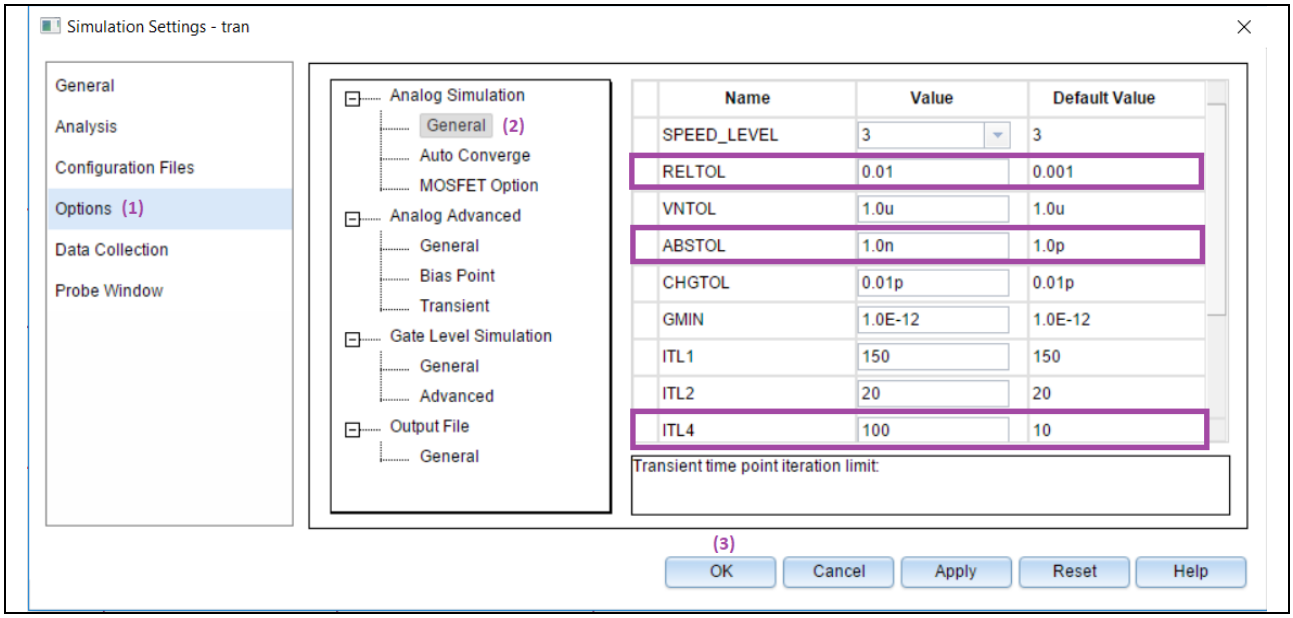


Figure 18 Simulation parameters

The following recommended values facilitate convergence:

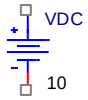
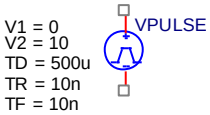
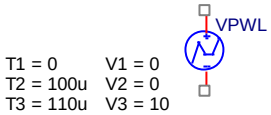
Table 1 Recommended values for simulation parameters

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

In general, the delivered models need to detect a rising edge starting from 0V on Inputs and Supply voltages to properly for a proper start-up sequence. It is recommended to use **VPWL** or **VPULSE** instead of **VDC** component.

TIP: In order to avoid the simulator's potential convergence issues please avoid using voltage sources such as VDC components as most of the models require a power-up pattern starting for at least a couple of micro-seconds from a zero volts supply. Therefore, it is best-practice to use sources like VPULSE or VPWL as shown below:

Table 2 Table A

VDC	VPULSE	VPWL
		
		

3. Opening an existing project

Typically, the model is delivered with an attached demo_testbench which is already configured and is ready for use.

3.1. Transient simulation considerations

Open the OrCAD Capture CIS application and from the **File** menu select **Open→Project.** In the browse window locate and open the "DEMO_TESTBENCH.opj" file from the Infineon-GenericModel-SimulationModel-L1-OrCAD-v01_00-EN package directory. For example:

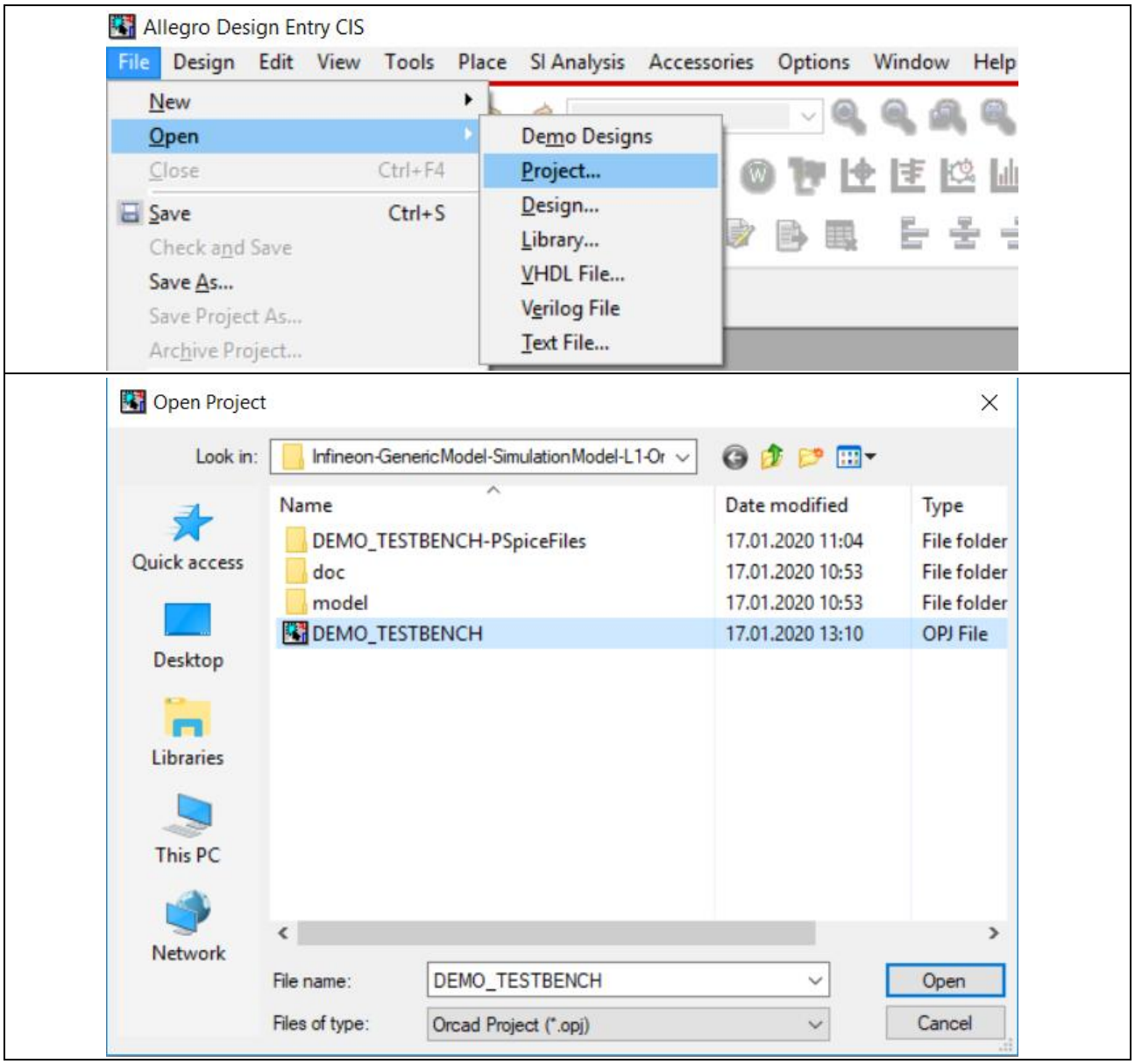


Figure 19 Open an existing project

- Navigate to **PAGE1** of **SCHEMATIC1** in the project tree **(1)** and right click on it→**Edit Page** to open the schematic. **(2)** In this section, one or more demo testbenches can be found with all the components and their values.

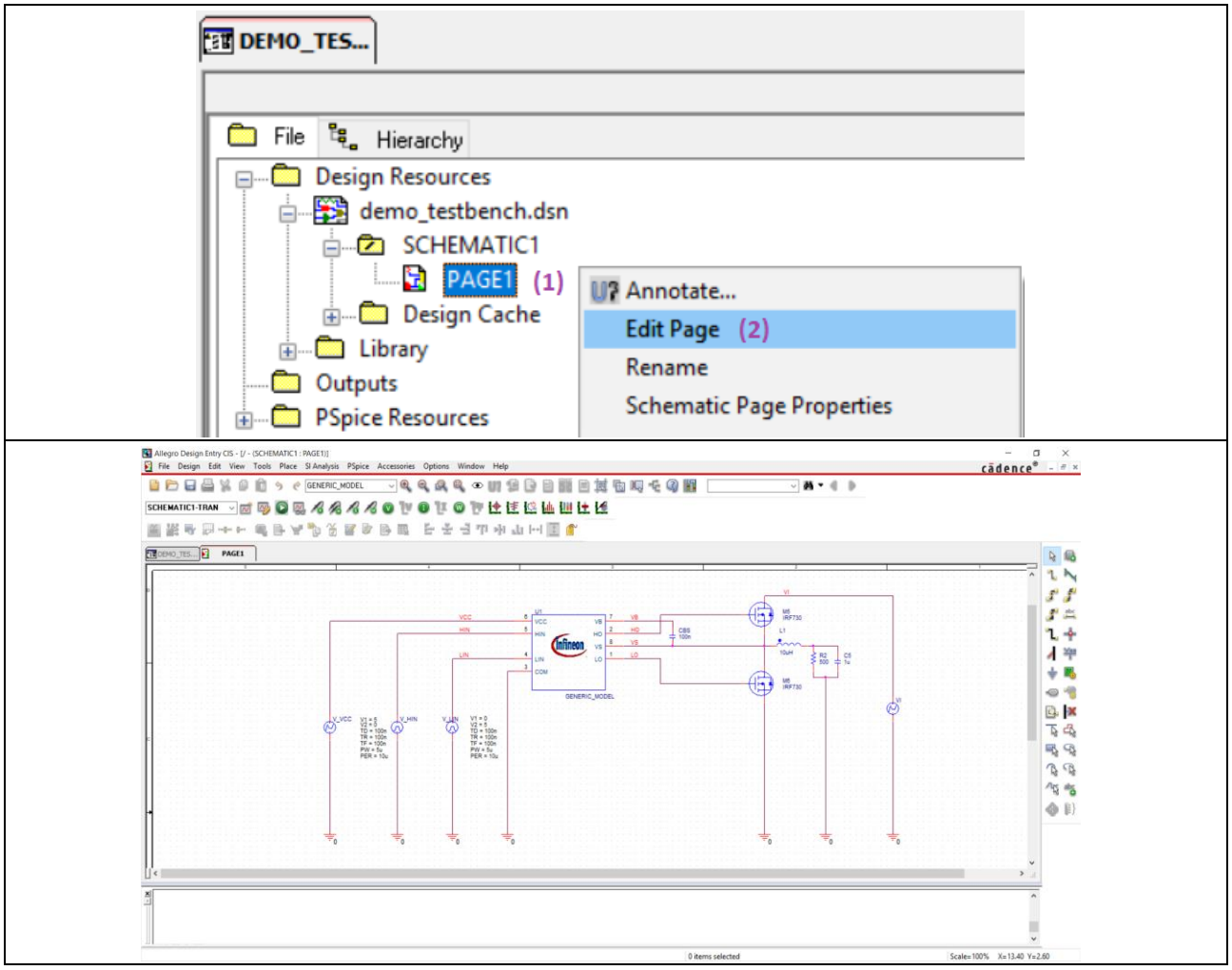


Figure 20 Open the demo_testbench

- To start the simulation, press “F11” on the keyboard or open the **PSpice** menu and click **Run**. Alternatively, the play button on the PSpice toolbar can be used
- Once the netlist has been generated the “Allegro AMS Simulator” (PSpice) will open; and the simulation will automatically start. A progress bar will be shown in the bottom right corner **(3)**
- After the simulation reaches 100% the signals can be plotted. To do this, click on the **Trace** menu→**Add trace** or press **Insert** button on the keyboard or press the **Add trace** pictogram from the toolbox **(4)**
- To plot the signals please select them from the list then press “OK” button **(5)**

Note: Filtering of the desired signals can be done using the check boxes in the middle of the window by their type (e.g. voltages, currents, power, etc. **(6)**)
Signals can also be composed with the functions available in the right side list “Functions or macros” **(7)**.

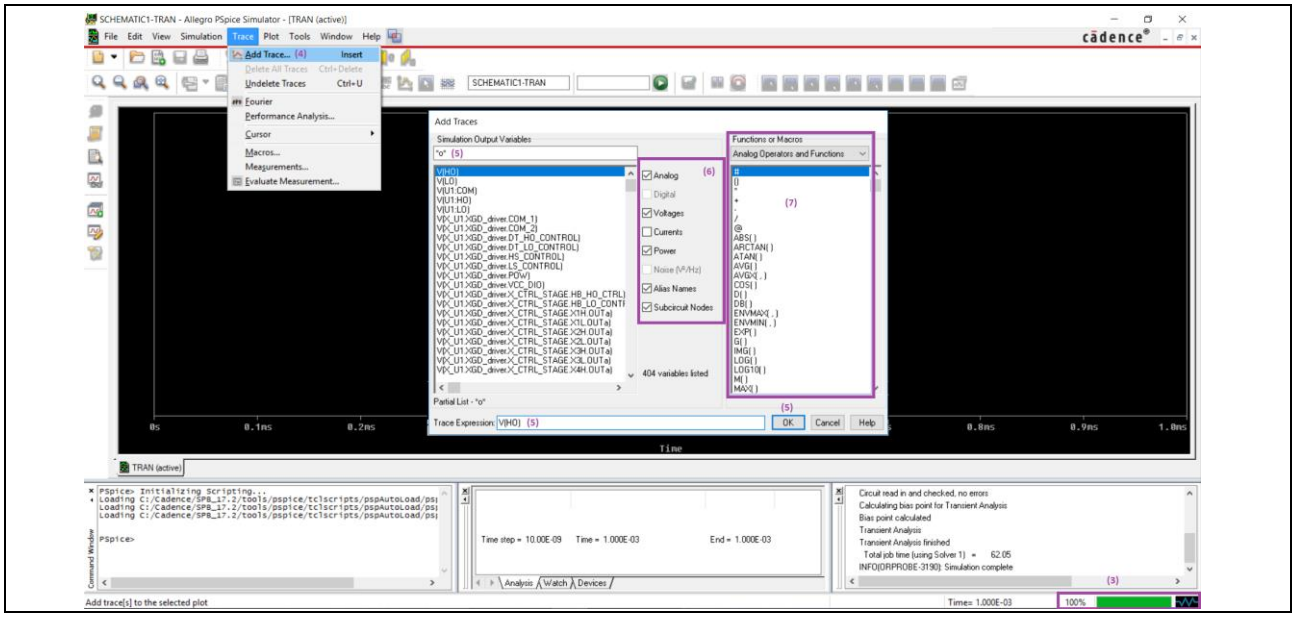


Figure 21 Plot signals

- To add other windows, click on **Plot** menu→**Add Plot to Window**

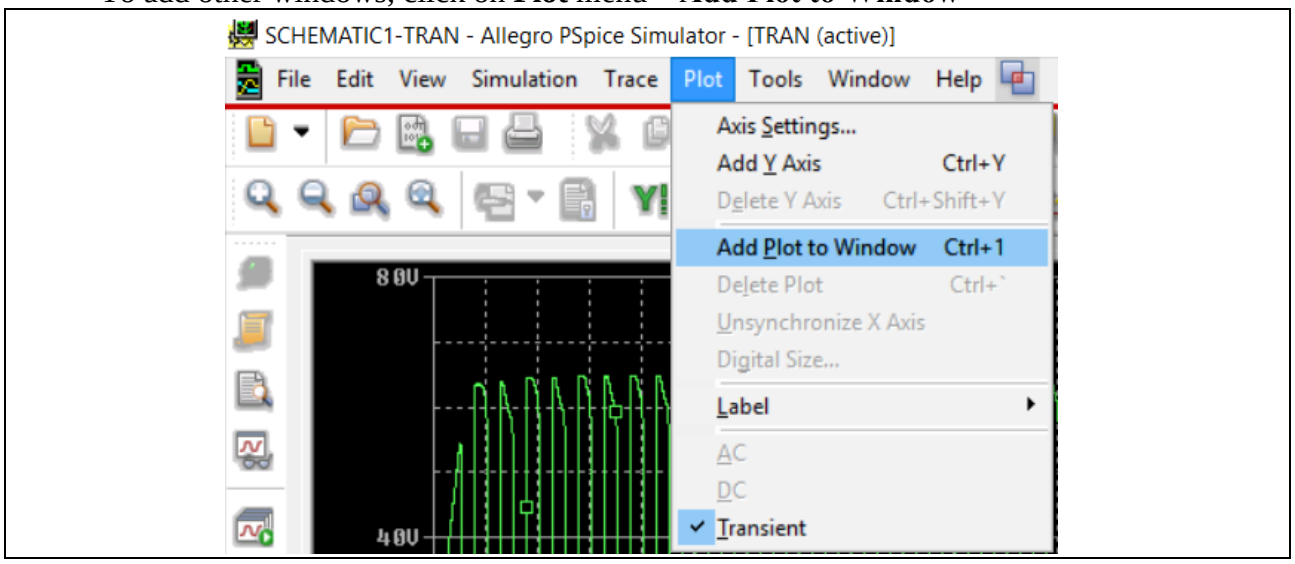


Figure 22 Plot a new window

- Cursors for signal value measurement can be added from the **Trace** menu→**Cursor→Display** or by left clicking the "Display Cursors" pictogram on the toolbox (Figure 23)

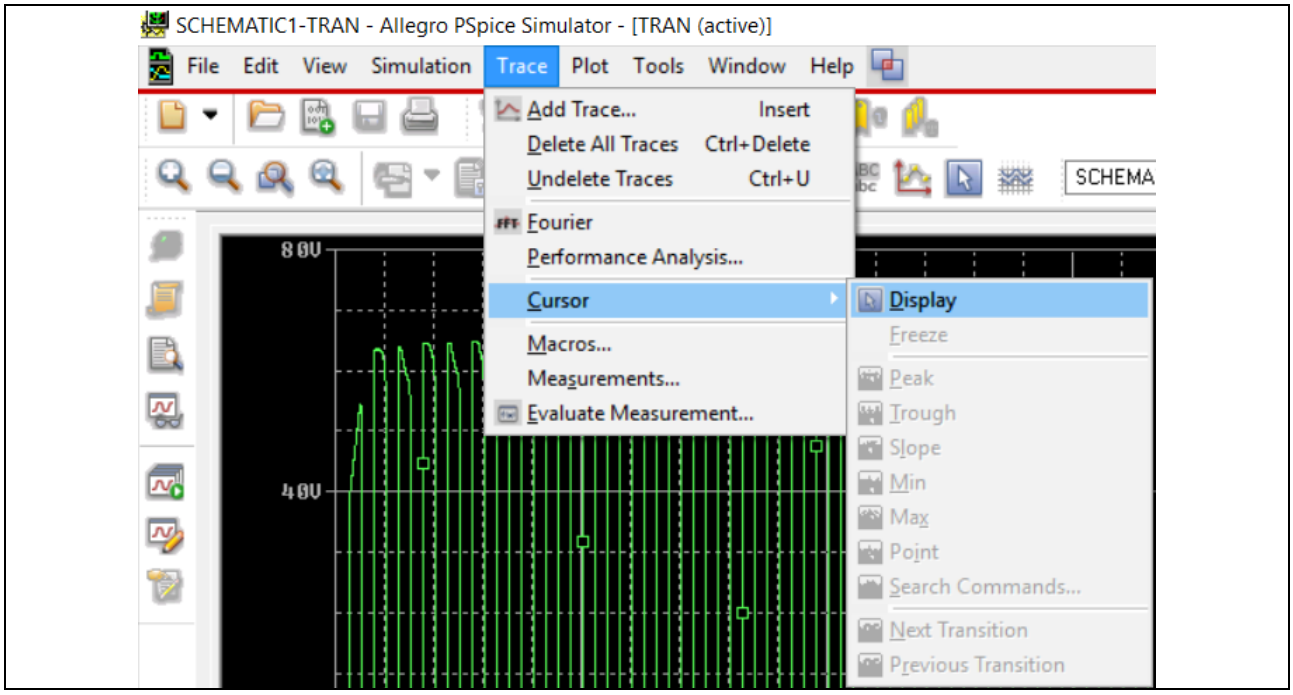


Figure 23 Add a cursor

- On the bottom right corner a table measurement menu can be observed

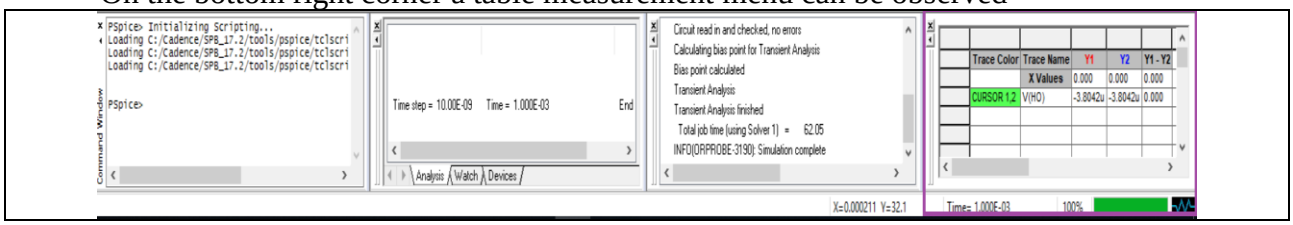


Figure 24 Measurement values

4. INFINEON'S SIMULATION MODEL TERMS OF USE

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).

1. SCOPE OF USE

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

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

1.3 This Model only describes the characteristics of a typical product. In all cases, the current data sheet 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.

2. IMPORTANT NOTICE

2.1 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").

2.2 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.

3. CONFIDENTIAL INFORMATION

User shall treat ideas, concepts and information incorporated in the Model, the documentation and the content of this Terms of Use (together hereinafter "Confidential Information") 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. WARRANTY

4.1 User acknowledges that the Model is provided by Infineon under this Terms of Use 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.

4.2 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.

5. LIABILITY

5.1 Nothing in this 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.

5.2 Without prejudice to Sections 5.1, in cases of Infineon's slight negligent breach of obligations that restrict essential rights or duties arising from the nature of this Terms of Use in a way that there is a risk of non-achievement of the purpose of this Terms of Use or of an obligation whose observance

User regularly may trust in and whereas compliance with only makes proper execution of this Terms of Use possible, Infineon's liability shall be limited to the typically, foreseeable damage.

5.3 Without prejudice to Sections 8.1 and 8.2, Infineon's liability under this Agreement shall be excluded in all other cases.

6. EXPORT REGULATIONS

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.

7. TERMINATION OF USE PERMIT

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.

8. MISCELLANEOUS

8.1 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.

8.2 Should any provision in these Terms of Use be or become invalid, the validity of all other provisions or agreements shall remain unaffected thereby.

Revision history

Document version	Date of release	Description of changes
Rev 1.0	2020-01-20	GettingStarted released