Getting started with Infineon simulation model

Targeted simulator: SIMetrix

About this document

Scope and purpose

This user guide provides instructions for getting started with the attached simulation model using the SIMetrix simulator, 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 schematic . . . 4
2.2 Importing an existing model . . . 5
2.2.1 Installing the model . . . 5
2.2.2 Associating the model with symbol . . . 6
2.3 Setting up and running a transient analysis . . . 9
2.4 Setting up global parameters . . . 10
2.5 Recommended simulation settings . . . 13
3 Opening an existing project . . . 15
3.1 Running the simulation . . . 15
3.2 Simulation results and measurements . . . 16
Revision history . . . 21
Disclaimer . . . 22

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)
- `.sxslb`: Schematic symbol view for graphical user interface
- `.sxsch`: 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 spice.

2.1 Creating a new schematic

Follow these steps to create a new schematic:

1. Open the SIMetrix application
2. Navigate to **File** > **New** > **Schematic** and start a new project, as shown in Figure 1

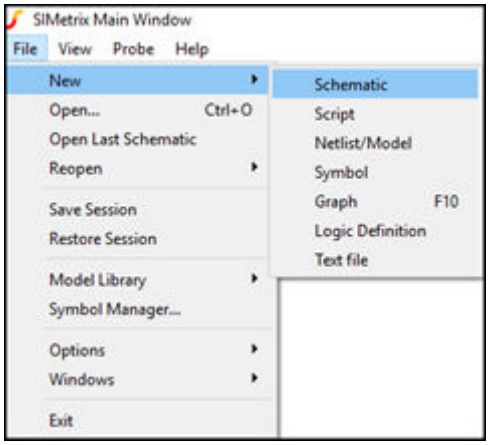


Figure 1 Creating new schematic

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

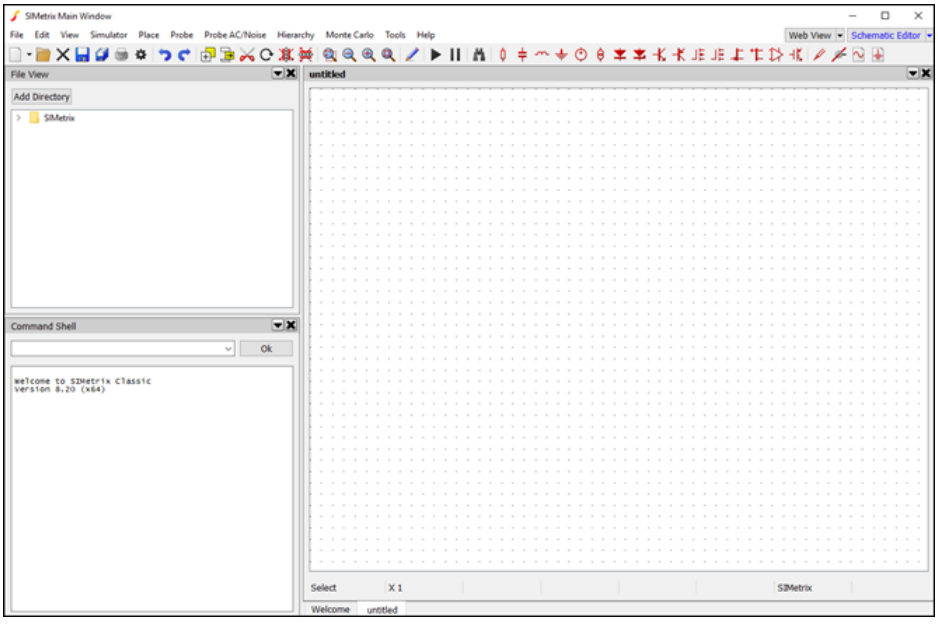


Figure 2 New schematic panel

2.2 Importing an existing model

***Note**: Before setting up a simulation, the required model libraries must be integrated into the simulator tool. This section describes how to import the simulation model files in SIMetrix.*

All simulation models are installed using two-step procedure:

- Install the model
- Associate model with symbol

2.2.1 Installing the model

There are several ways to install simulation models in SIMetrix, with the two most common methods being:

- Drag-and-drop method
- Local installation method

Drag-and-drop method

To install a model, select and drag the code source file (`.lib`)—for example, `simulation_model.lib`— and the symbol file (`.sxslb`)—for example `simulation_model.sxslb`—into the SIMetrix Command Shell, as shown in see Figure 3.

Steps:

1. Open Windows Explorer and locate the device models
2. Select items you want to install
3. Ensure that the SIMetrix Command Shell is in focus.
   If it is not, select a schematic or graph and press the spacebar
4. Drag the selected files from Windows Explorer and drop them into the Command Shell
5. For files that are processed individually, a confirmation message box will appear.
   Once confirmed, the simulation model will be installed

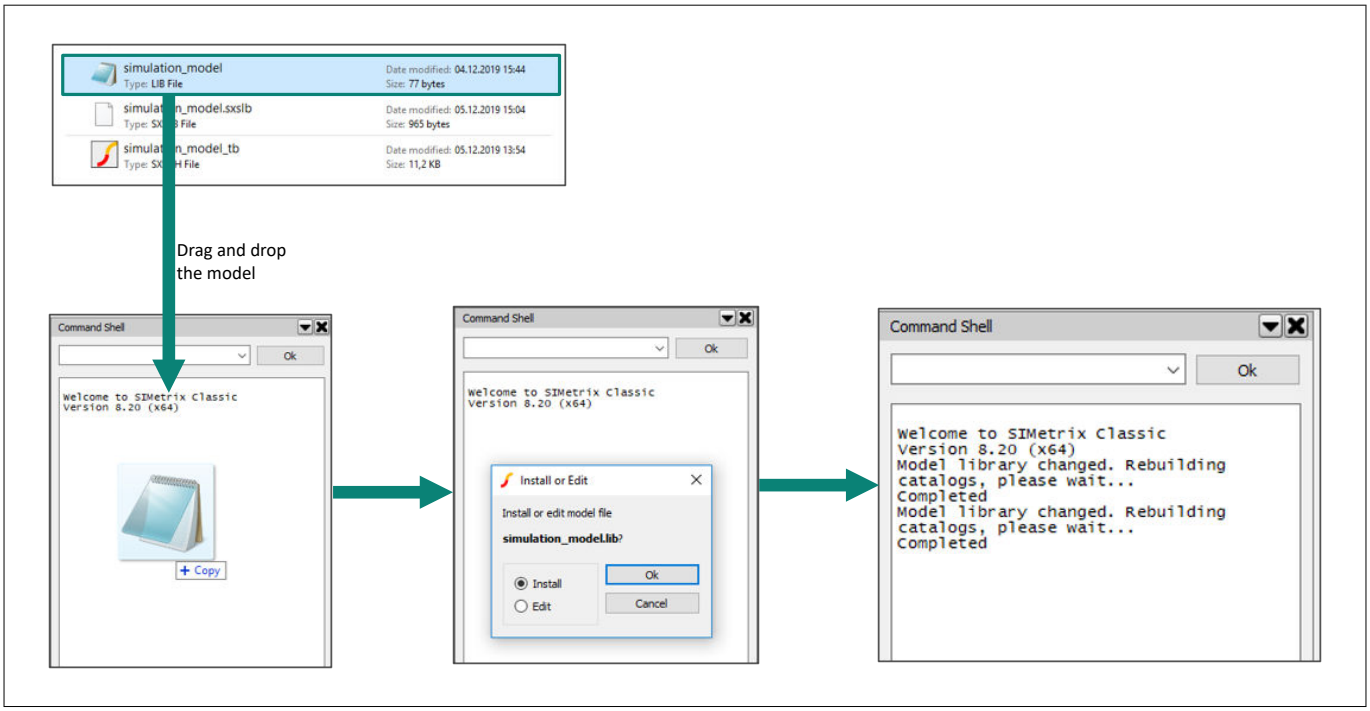


Figure 3 Drag and drop

Local installation method

A model (`.lib` file) can also be enabled for a specific schematic only by including a reference in the Command Window, as shown in Figure 4.

Steps:

1. Open the schematic sheet and press **F11** on keyboard to toggle Command Window
2. Use `.include` command followed by library name.
   Ensure that the model library file (`.lib`) is placed in the same folder as the schematic

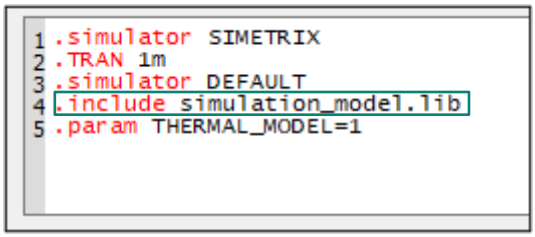


Figure 4 Local model

2.2.2 Associating the model with symbol

After the simulation files have been installed, Figure 5 and Figure 6 illustrate how to place a symbol on the schematic.

Follow these steps to associate the model with symbol:

1. From main menu, select **Place** > **From Model Library**

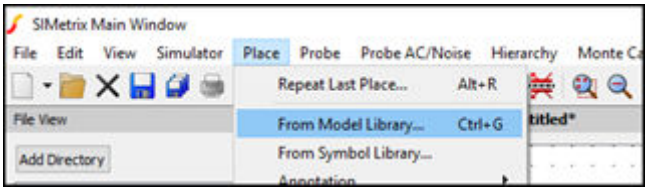


Figure 5 Place symbol

2. On the left-hand side, select the *** Recently Added Models *** category.
   The installed simulation models will be listed on the right

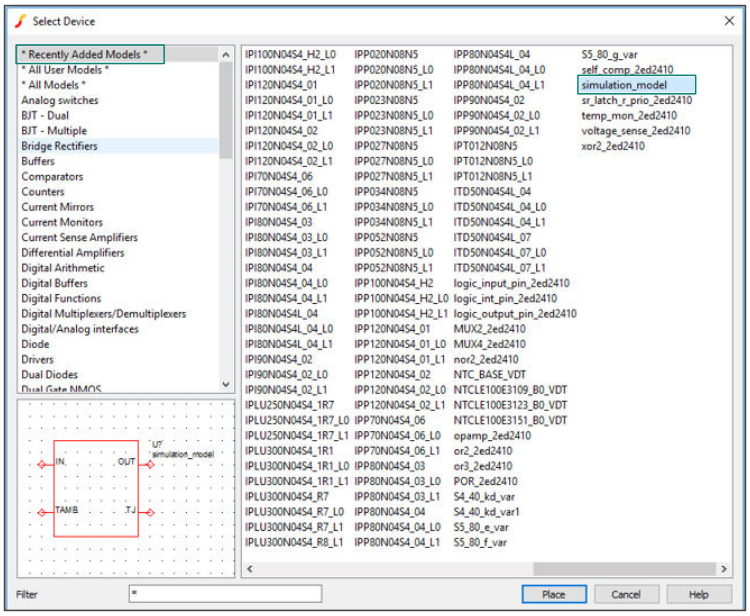


Figure 6 Recently added models

3. Select the appropriate part; its schematic symbol will appear in the preview window
4. If SIMetrix displays the following message: `SIMetrix does not know what symbol to use for this model` as shown in the Figure 7, click **Place**

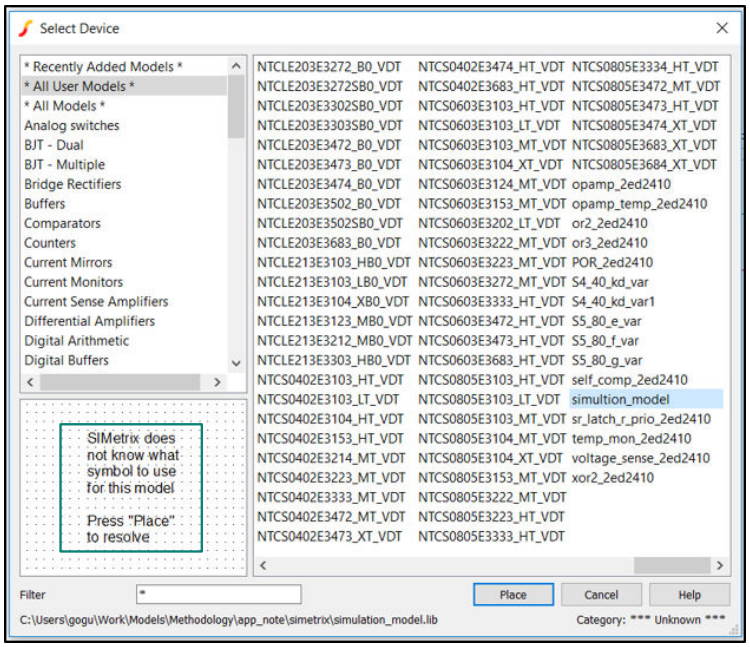


Figure 7 Resolving the message

A new window will appear, as shown in Figure 8.

5. Select **Auto Create Symbol**. A default rectangular shape will be associated, as shown in Figure 9

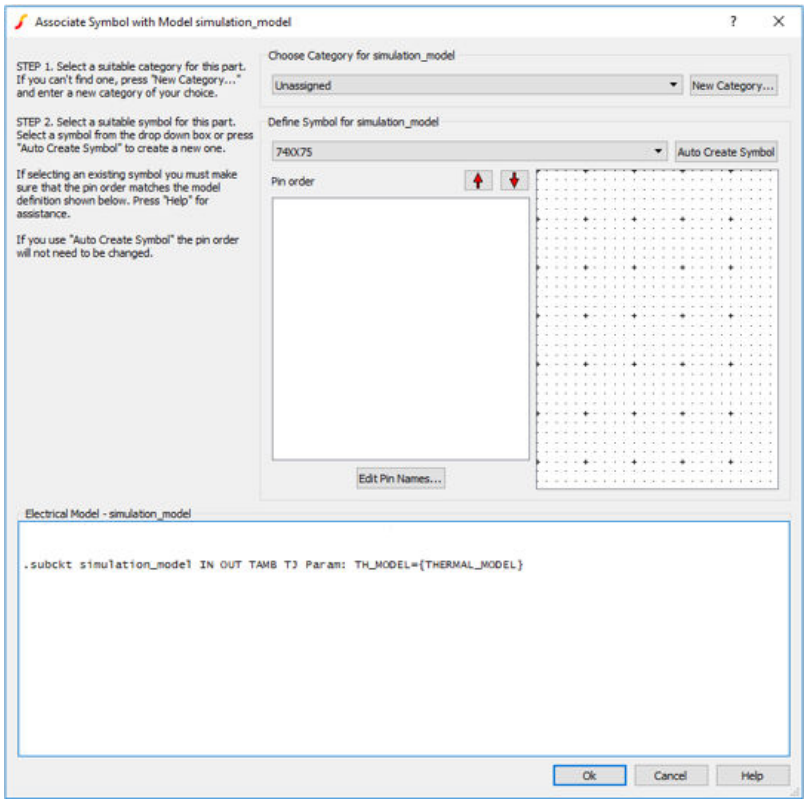


Figure 8 Auto create symbol

A preview of the model will be displayed in the **Electrical Model** window, as shown in Figure 9.

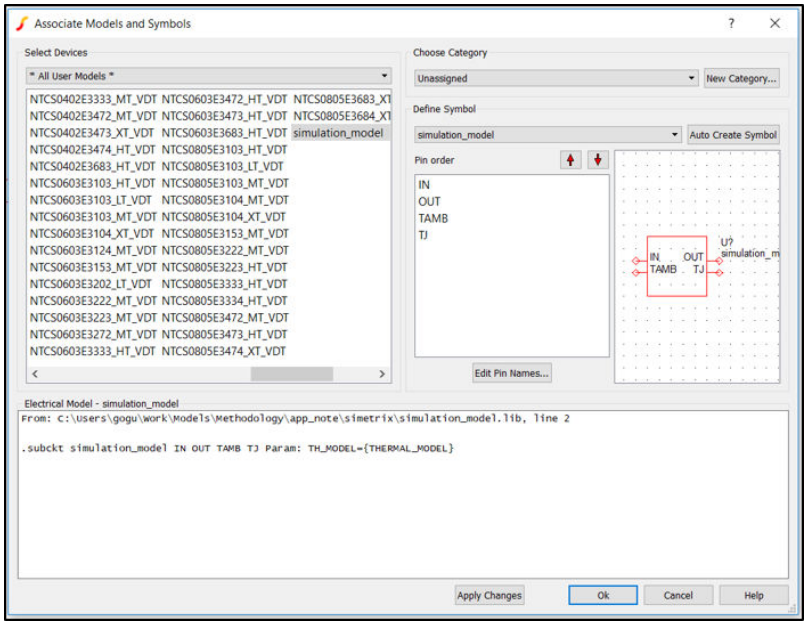


Figure 9 Associate symbol with model

6. If the part is to remain under the **Unassigned** category, click **OK** and confirm with **Yes, leave at "Unassigned"**, as shown in Figure 10. Otherwise, select an appropriate category or create a new one

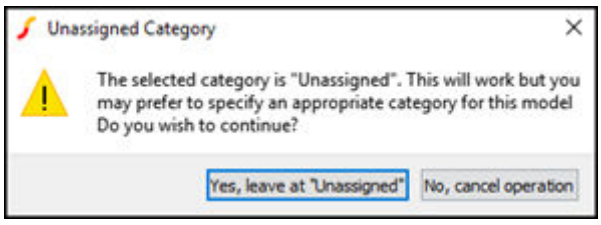


Figure 10 Leaving the part unassigned

The symbol is now ready and can be placed on the schematic.

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 **Simulator** > **Choose analysis**, as shown in Figure 11

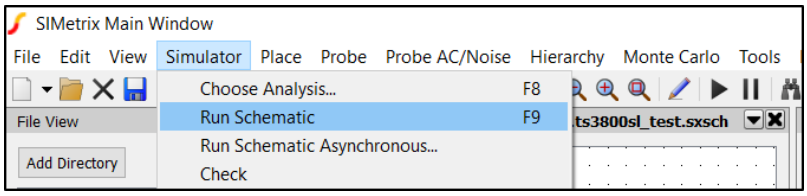


Figure 11 Choose analysis

2. The **Choose Analysis** dialog box appears, as shown in Figure 12

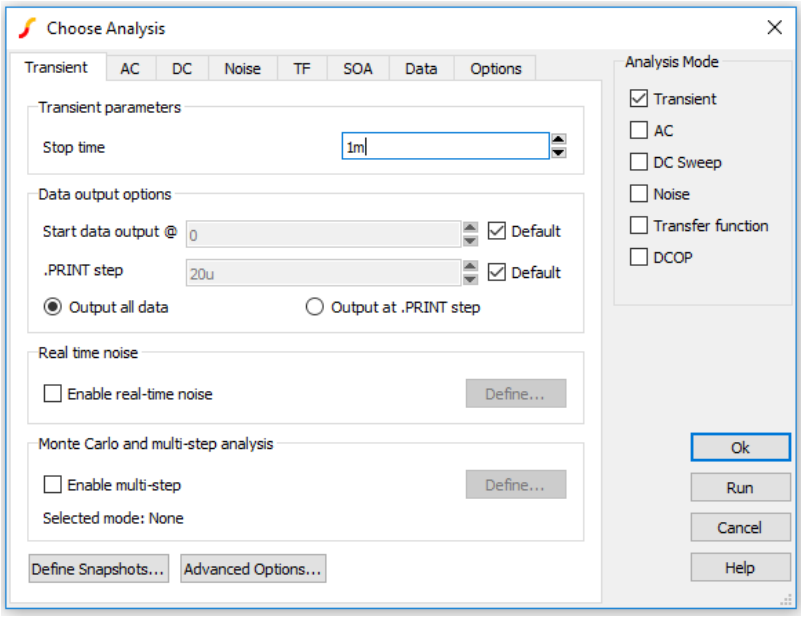


Figure 12 Analysis settings

3. From the **Choose Analysis** window, choose transient and specify the required parameters:
   - **Stop time** = The total simulation duration.

Note that the simulation can be paused before the **Stop time** is reached allowing the results obtained so far to be examined

- **Data output option** = The restriction of the amount of data generated by the simulator

4. Click **Advanced option**. A dialog box appears, as shown in Figure 13

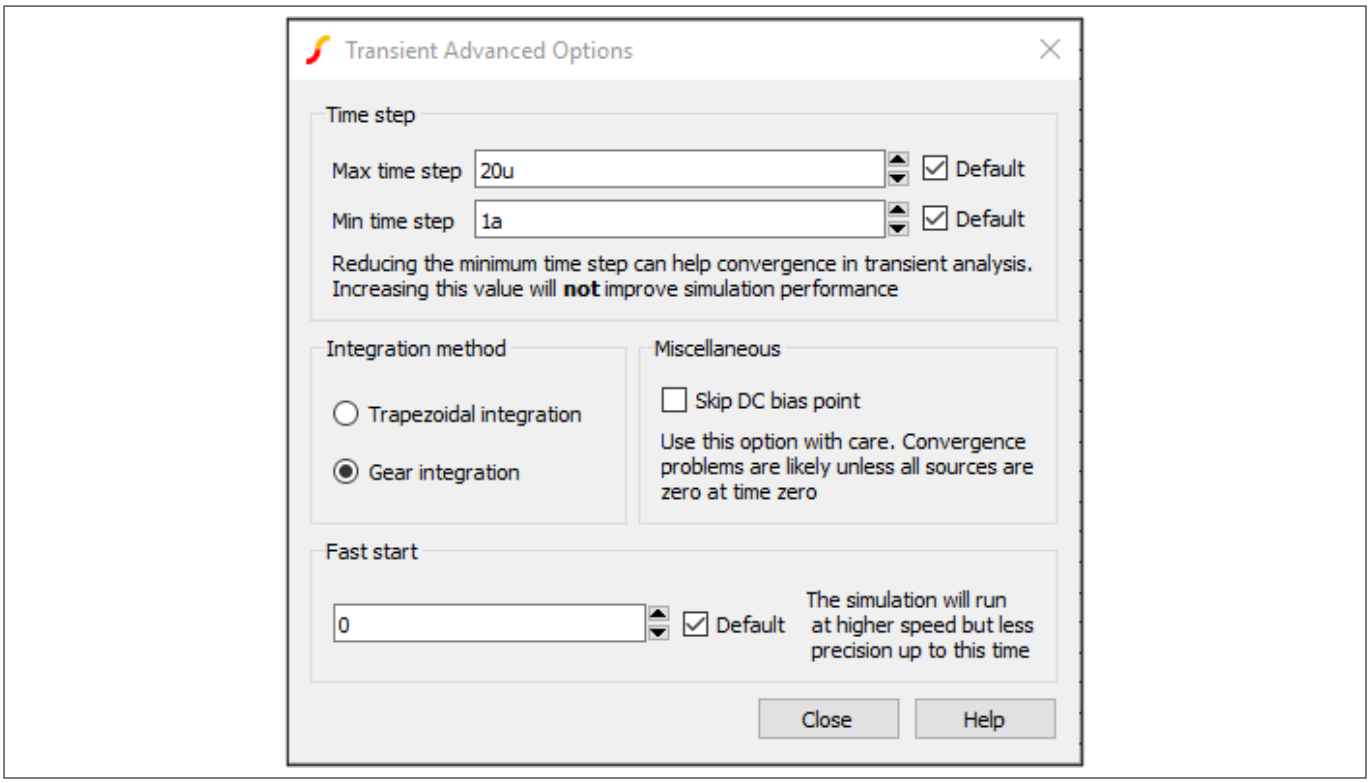


Figure 13 Transient advanced options

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:

- 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 define a global parameter in SIMetrix, follow the instructions below:

1. Open the schematic sheet
2. Press **F11** on keyboard to toggle **Command Window**
3. Use `.PARAM` command followed by parameter name and value, as shown in Figure 14

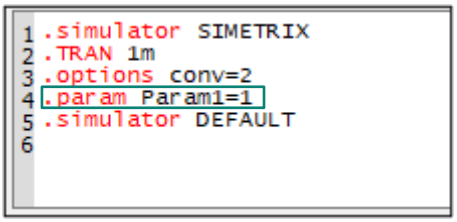


Figure 14 Define global parameter

Two practical examples are shown below:

Figure 15 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.

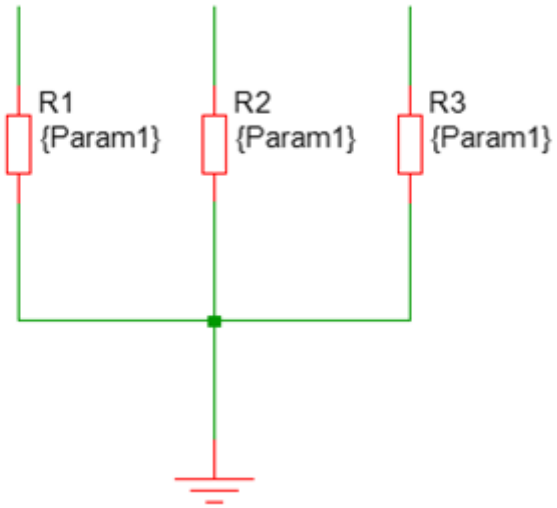


Figure 15 Global parameter for multiple instances

Figure 16 and Figure 17 show 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.

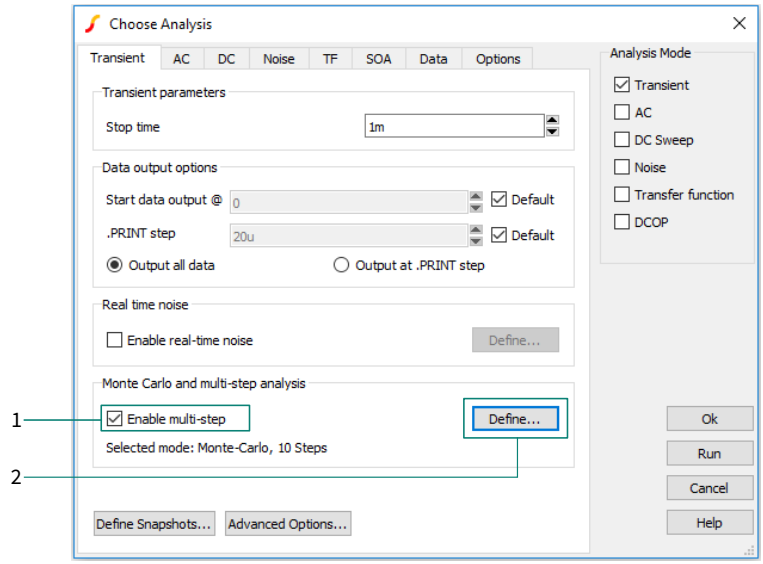


Figure 16 **Set up a parametric simulation (1)**

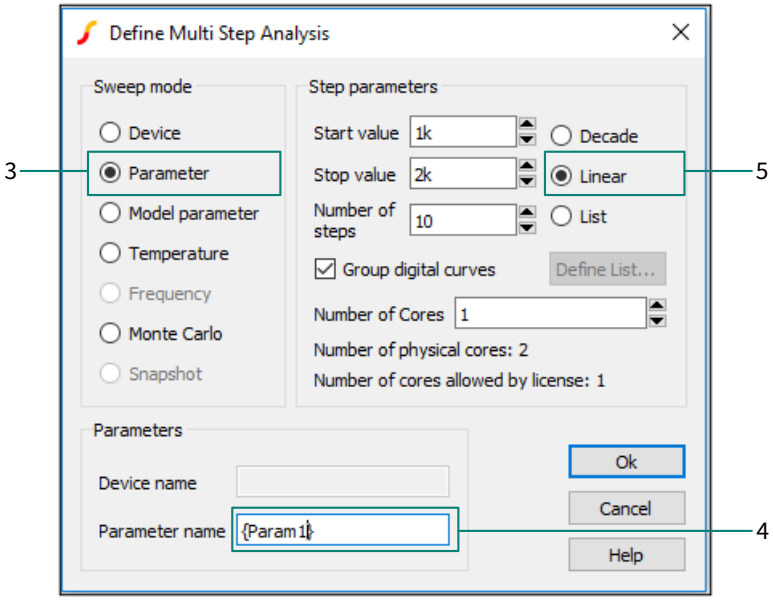


Figure 17 **Set up a parametric simulation (2)**

2.5 Recommended simulation settings

Because spice was not originally designed specifically for power electronics and highly non-linear components, the simulation parameters (**Simulator** > **Choose Analysis** > **Options**) can be optimized for a given design.

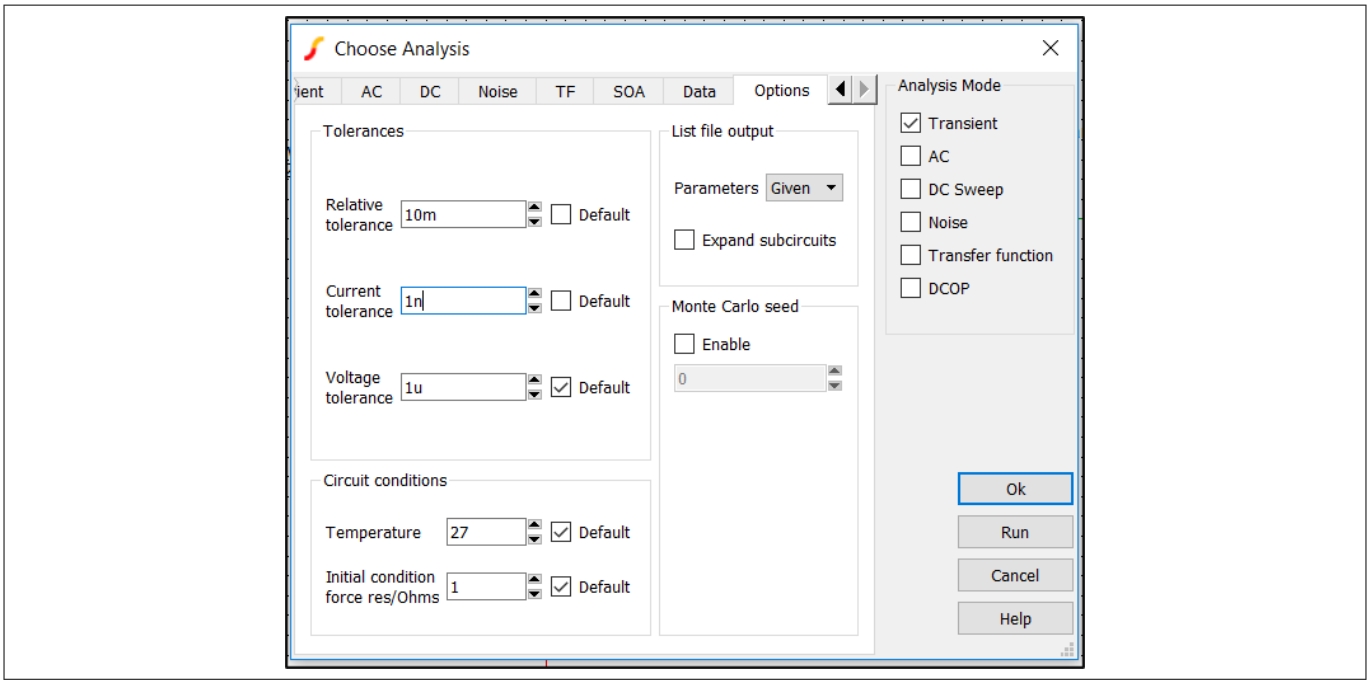


Figure 18 Simulation parameters

Table 1 provides the recommended values to facilitate convergence.

Table 1 Recommended values to facilitate convergence

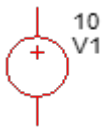
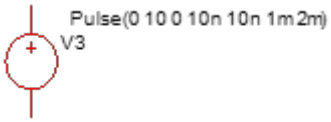
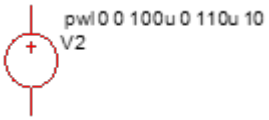
Parameter	Value	Description
Relative tolerance	10m	The relative accuracy of voltage and currents.
Current tolerance	1n	The maximum current accuracy.
Default Integration Method	Gear	Refer to Setting up and running a transient analysis.
Skip DC bias point	—	Refer to Setting up and running a transient analysis.
.options noOpIter	—	Skip the initial attempt at solving the DC operating point; can be define in the schematic's F11 window.

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 such as V_{PULSE} or V_{PWL}.

Table 2 Recommended 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**: A fully licensed version of SIMetrix for Windows is required to be installed. Extract the files to `c:\` drive.*

1. Open the SIMetrix application
2. From the main menu, select **File** > **Open**, as shown in Figure 19
3. In the browse window, locate and open the schematic file (`.sxsch`)

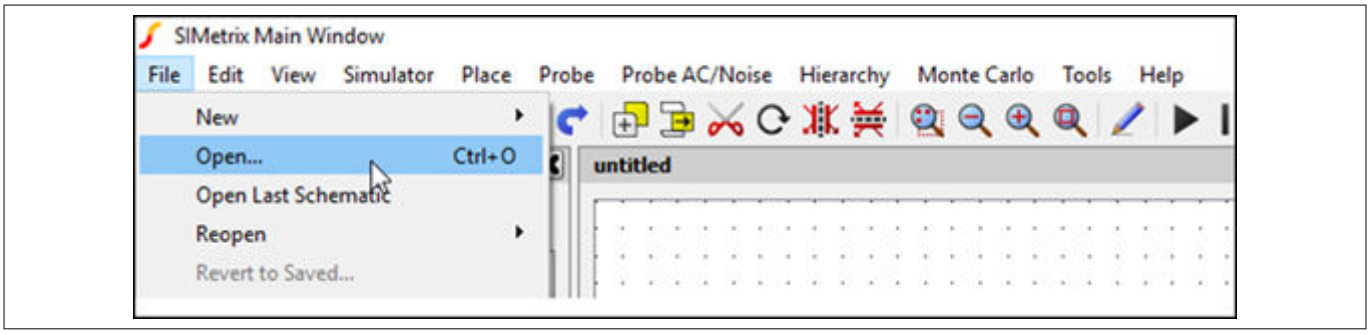


Figure 19 Opening an existing project

4. To start the simulation, press **F9** on the keyboard or go to **Simulator** > **Run Schematic**. Alternatively, click on play button on the SIMetrix toolbar

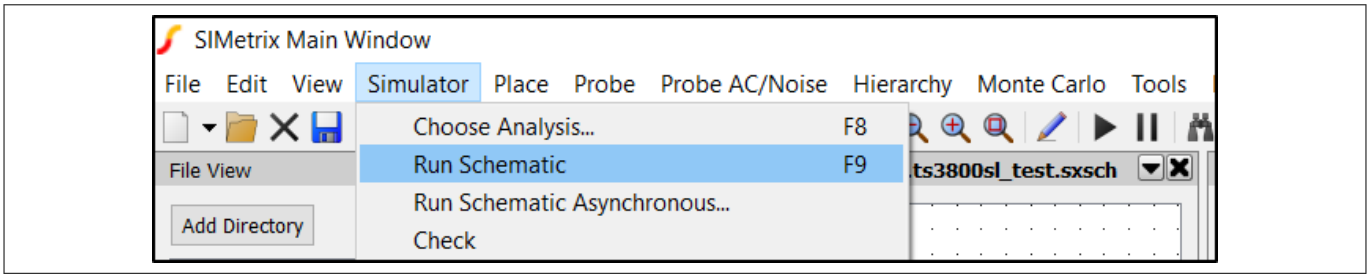


Figure 20 Start the simulation

A simulation progress window will appear, as shown in Figure 21.

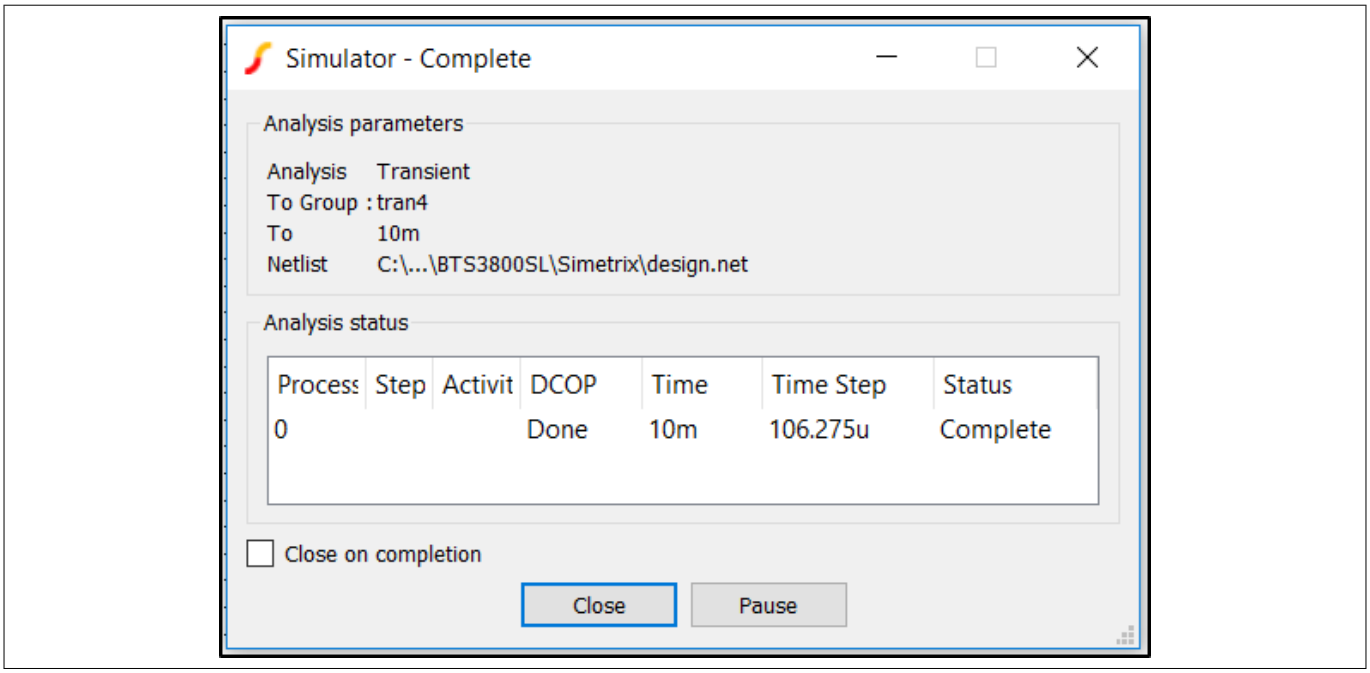


Figure 21 Simulation status

3.2 Simulation results and measurements

After the simulation status is complete as shown in Figure 21, close the window and plot the signals. To do this, go to SIMetrix main window and select **Probe** > **Add curve**, as shown in Figure 22.

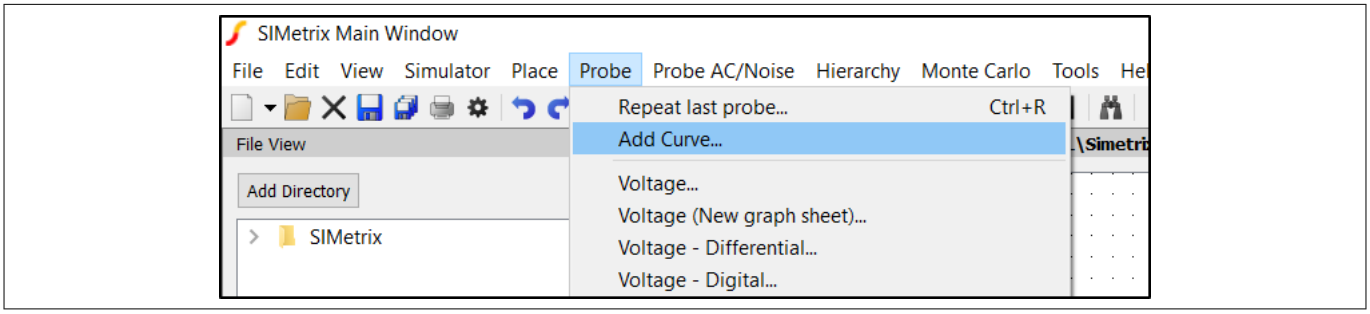


Figure 22 Plot signals

To plot the signals, select them from the **Available vectors** and then click **OK**, as shown in Figure 23.

***Note**: You can filter the desired signals by type (for example, voltages, currents) from **Edit Filter** option under **Available vectors**. Select the required signal type on the right side of the **Edit Available Vectors** window, as shown in Figure 24.*

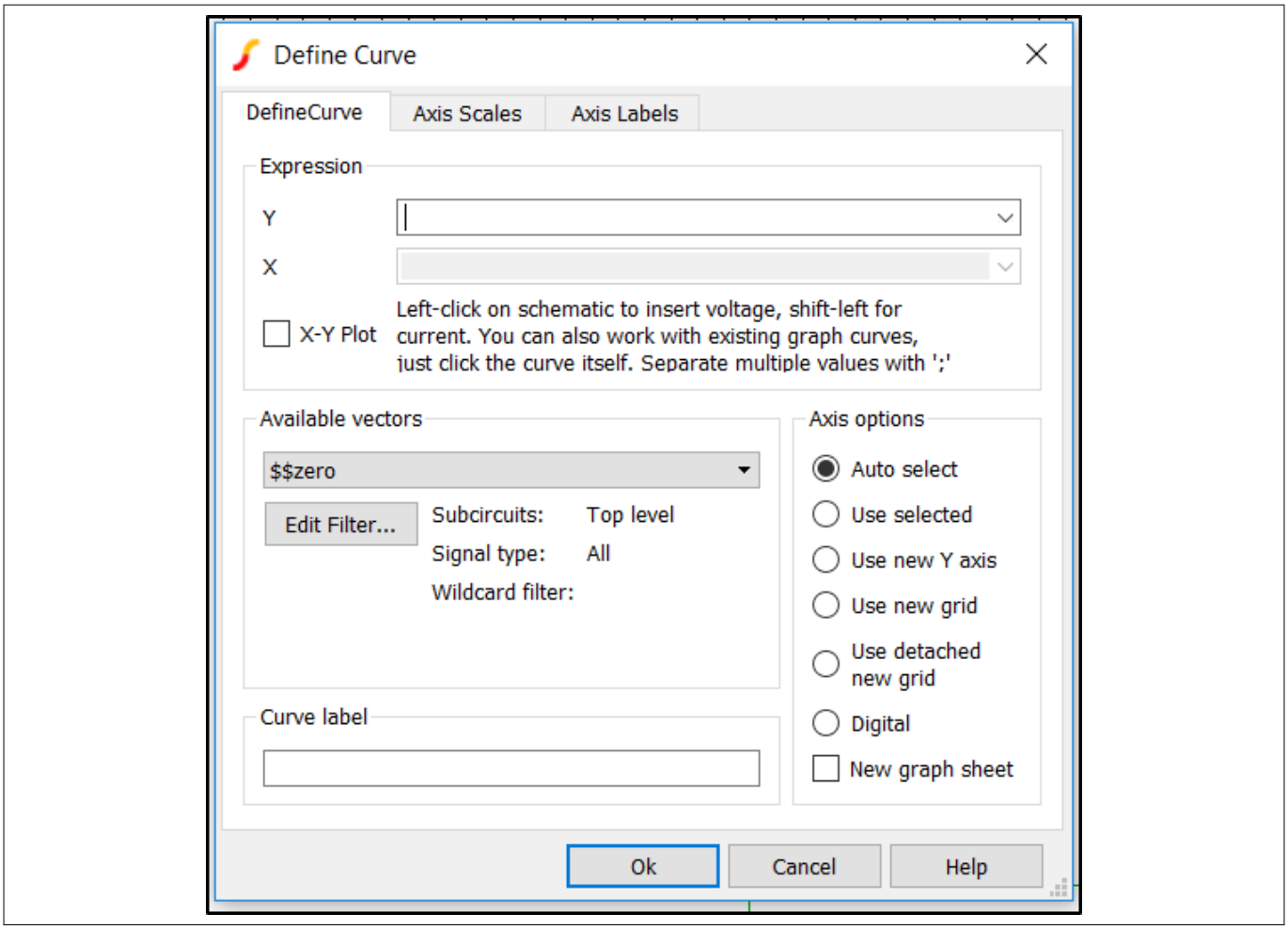


Figure 23 Define a new curve

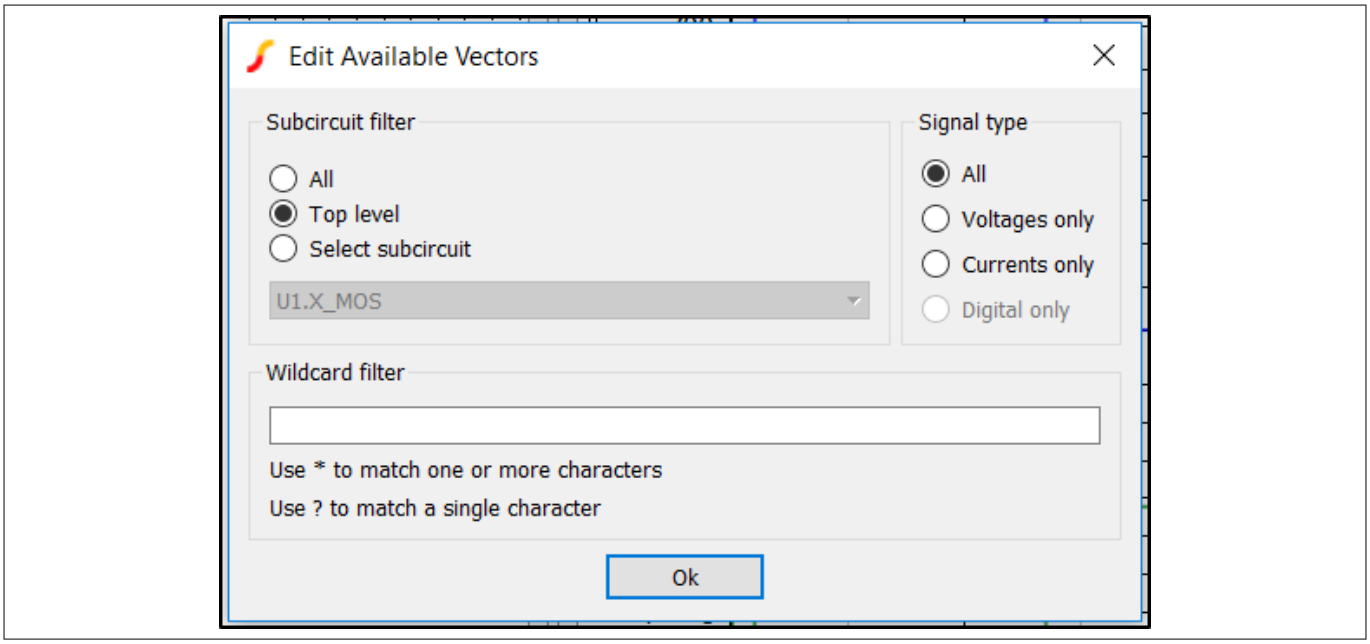


Figure 24 Filtering the available signals

To add other window, select **Axes** > **New Grid** from the main menu, as shown in Figure 25.

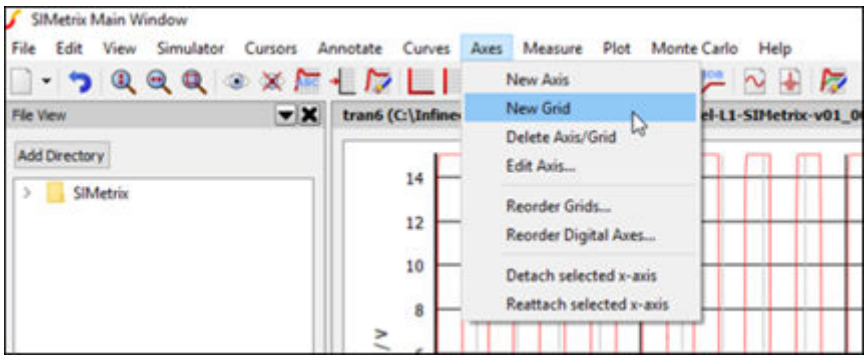


Figure 25 New Grid

To keep the probes from one simulation run to another, there are fixed probes available for both voltages and currents that can be placed directly on schematic components, as shown in Figure 26.

- To place a fixed voltage probe, press **B** on keyboard and place it on a wire
- To place a fixed current probe, press **U** on keyboard and place it on a device or component pin
- To edit the probe, double-click on it
- A separate grid can be used with a given predefined order of appearance in the list of signals by specifying a letter (for example, D for the 4th position), as shown in Figure 27

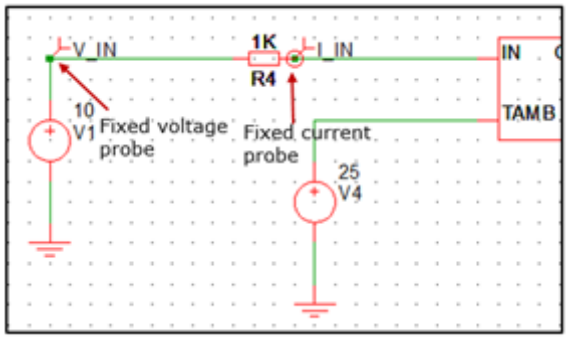


Figure 26 Fixed probes

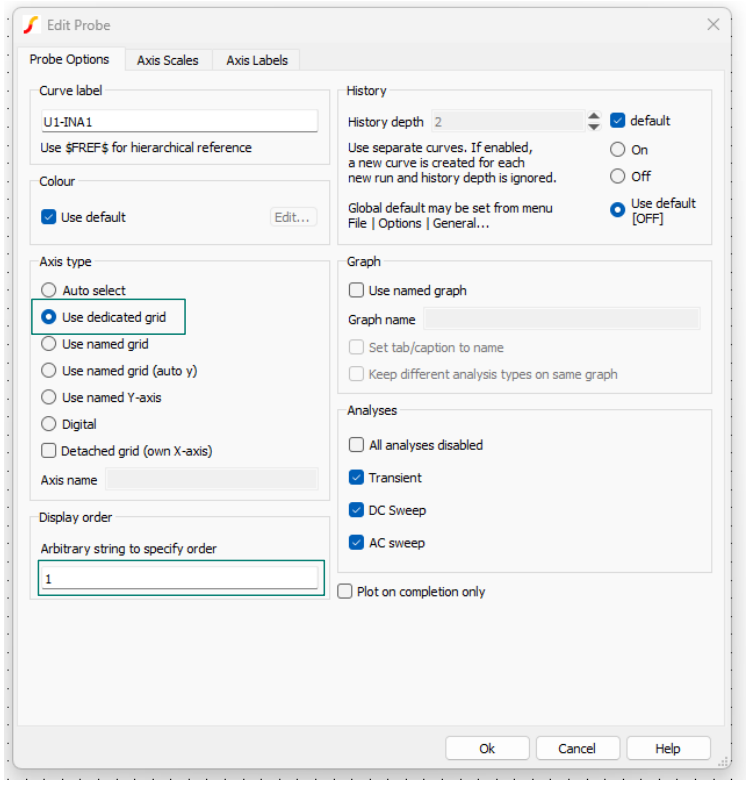


Figure 27 Edit a fixed probe

Navigate to **Cursor** > **Toggle On/Off** to add cursors for signal value measurement, as shown in Figure 28.

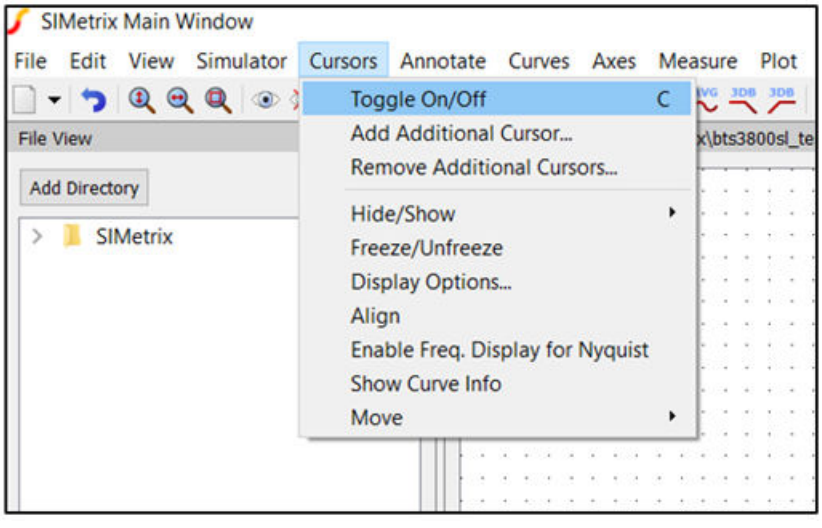


Figure 28 Add cursors

Figure 29 shows an example of a table measurement menu, which is observable in the bottom right corner of the GUI.

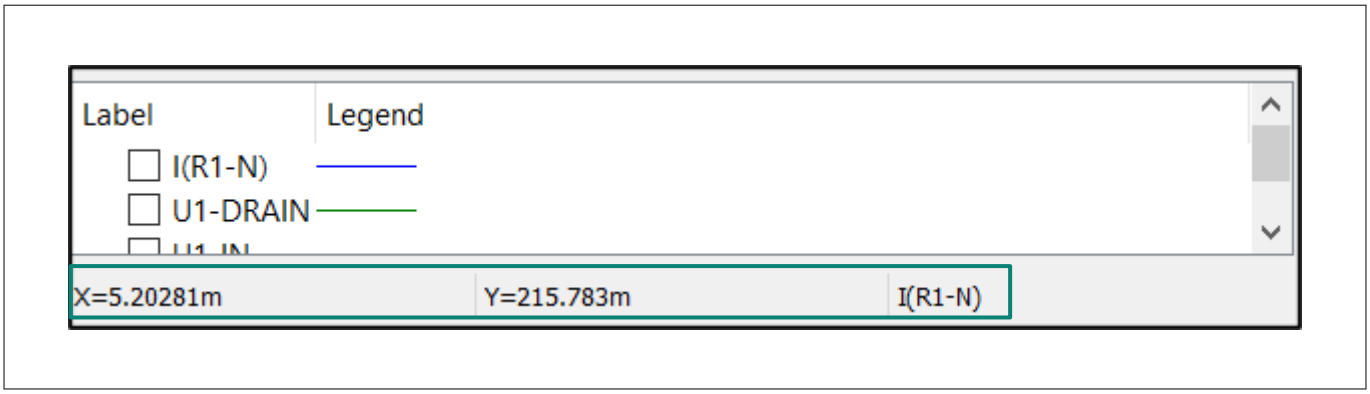


Figure 29 Measurement values

Revision history

Document revision	Date	Description of changes
01.00	2022-03-18	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-lsc1772528337527

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.