

# Introduction to Simulation Environment

**Targeted simulator: SIMetrix**

## About this document

### Scope and purpose

The current document provides guidance on how to get started with the attached behavioral model using the SIMetrix 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

	<b>About this document</b> .....	1
	<b>Table of contents</b> .....	2
<b>1</b>	<b>Resources</b> .....	3
<b>2</b>	<b>Starting a new project</b> .....	4
2.1	Create a new project .....	4
2.2	Import of an existing model .....	5
2.2.1	Install the model .....	5
2.2.2	Associate model with symbol .....	7
2.3	Set up and run a transient analysis .....	11
2.4	Set up global parameters .....	14
2.5	Recommended simulation settings .....	17
<b>3</b>	<b>Opening an existing project</b> .....	19
3.1	Transient simulation considerations .....	19
	<b>Revision history</b> .....	24
	<b>Disclaimer</b> .....	25

### 1 Resources

## 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
- **.sxslb** : schematic symbol view for graphical user interface
- **.xsch** : project file which contains one/several application setup(s)

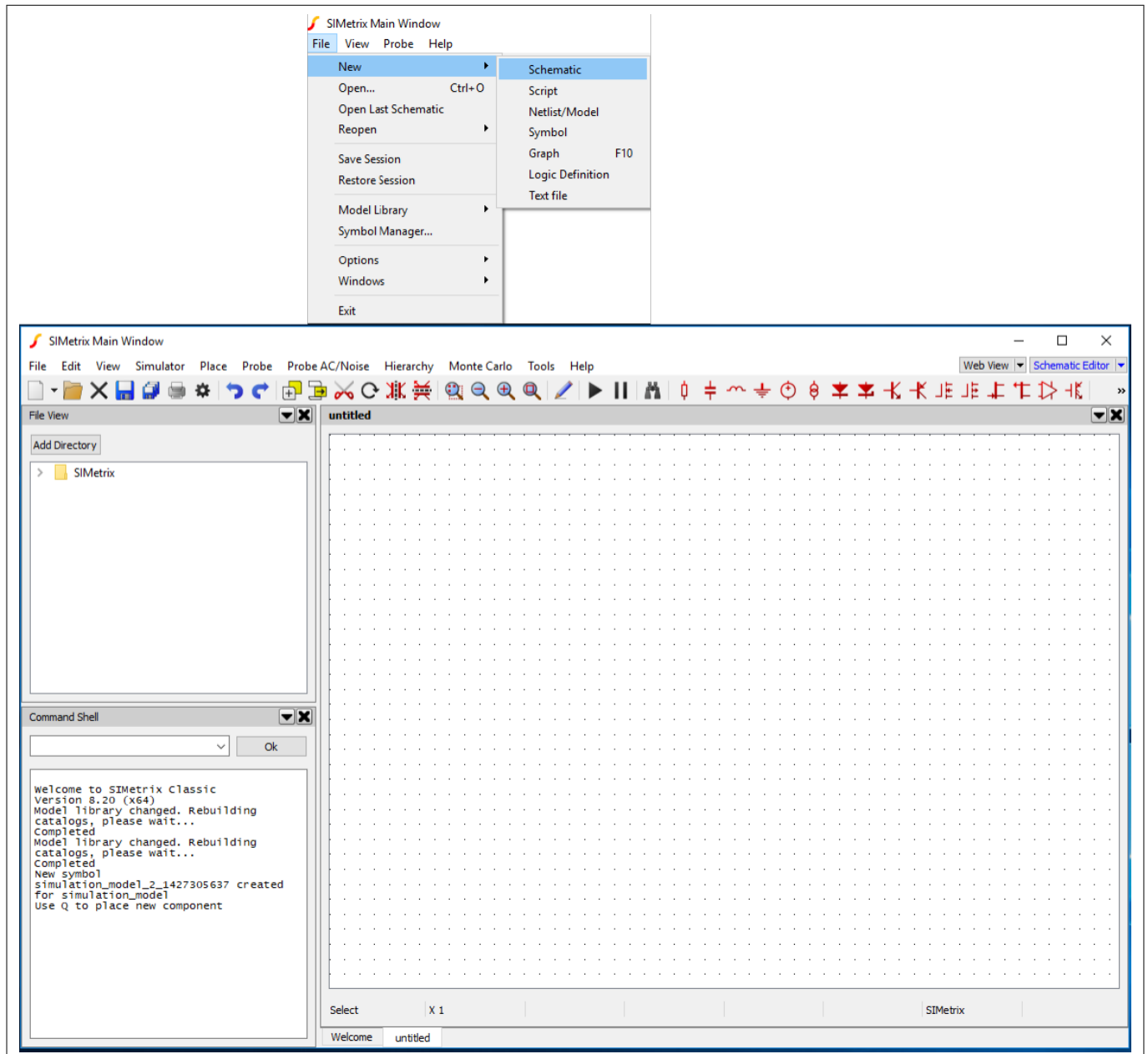
## 2 Starting a new project

## 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 **SIMetrix** application. Select **File** menu→**New**→**Schematic** and the window will display a new schematic panel, similar to that below:



**Figure 1** Create new Schematic

## 2 Starting a new project

### 2.2 Import of an existing model

Before setting up a simulation, the model libraries of interest must be integrated in the simulator tool. This section gives an introduction about the installation of the simulation model files in **SIMetrix**.

All simulation models are installed using a common, two-step procedure:

- Install the model
- Associate model with symbol

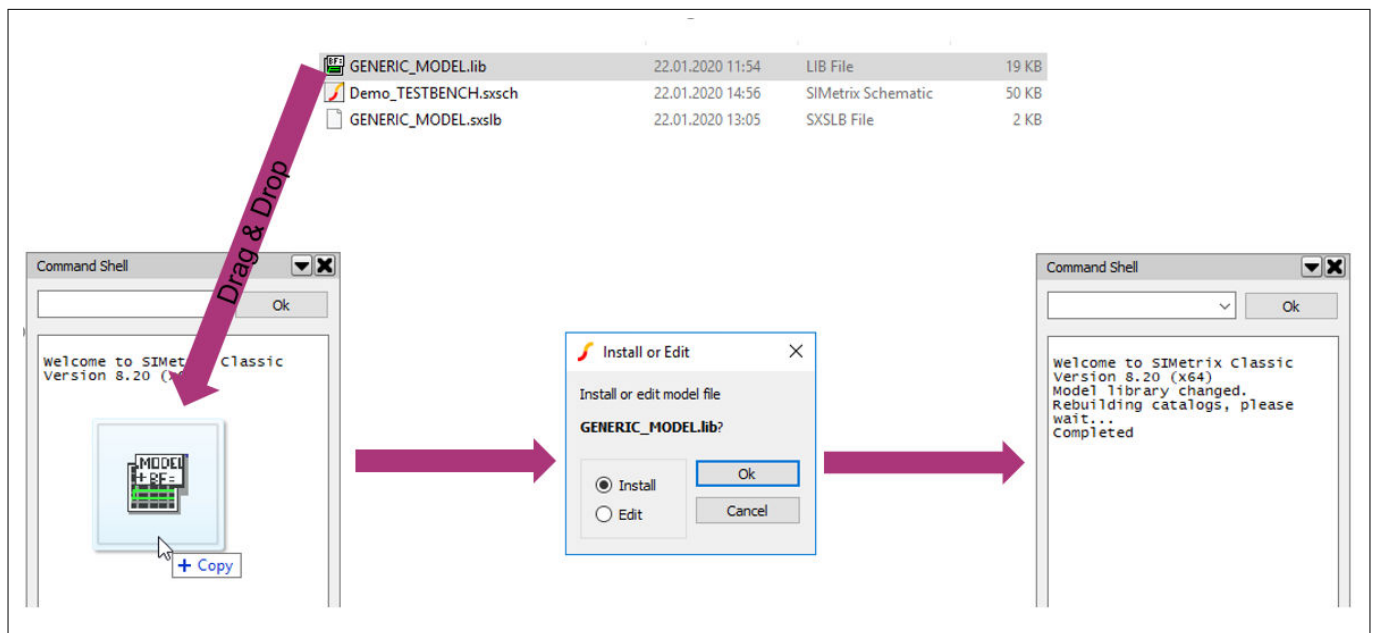
#### 2.2.1 Install the model

SIMetrix present two ways of model installation: “drag and drop method” and “local installation method”.

##### Drag and Drop method

To install a model select and drag the code source file (**.lib**), e.g. GENERIC\_MODEL.lib and the symbol file (**.xsxlb**), e.g. simulation\_model.xsxlb and drop them in the **SIMetrix** Command Shell

- Open windows explorer and locate the device models
- Select items to be installed
- Make sure that the SIMetrix **command shell** is in focus. If not, select a schematic or graph and press the spacebar
- Select the files in windows explorer, then drag and drop them into the **command shell**
- For files which are processed individually, a message box will pop up. Once confirmed the simulation model will be installed



**Figure 2** Drag & Drop

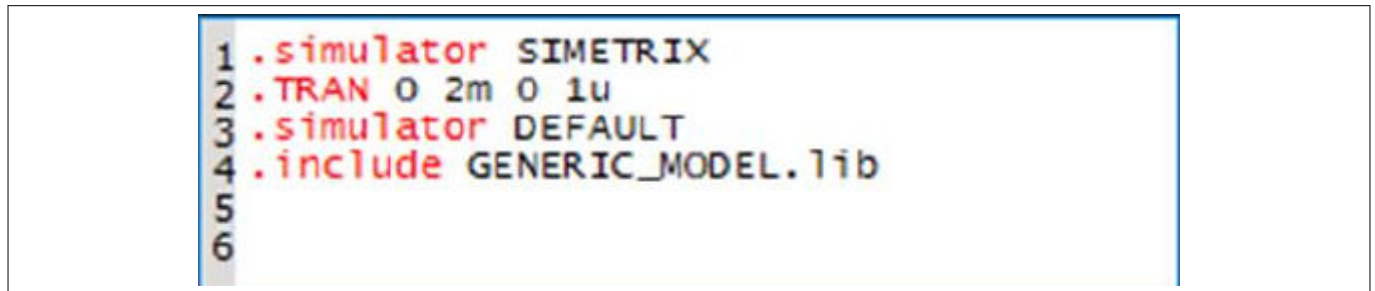
### 2 Starting a new project

#### Local Installation method

A model (.lib file) can also be enabled for a certain schematic only – by including a reference in the Command Window.

Steps to follow:

- Open the schematic sheet, press **F11** on keyboard to toggle **Command Window**
- Use **.include** command followed by library name. The model library file (.lib) must be place in the same folder with the schematic



```
1 .simulator SIMETRIX
2 .TRAN 0 2m 0 1u
3 .simulator DEFAULT
4 .include GENERIC_MODEL.lib
5
6
```

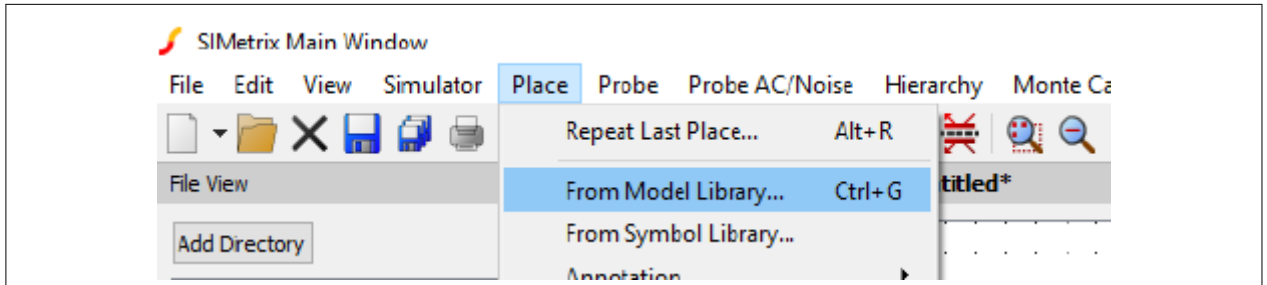
**Figure 3**                      **Local model**

## 2 Starting a new project

### 2.2.2 Associate model with symbol

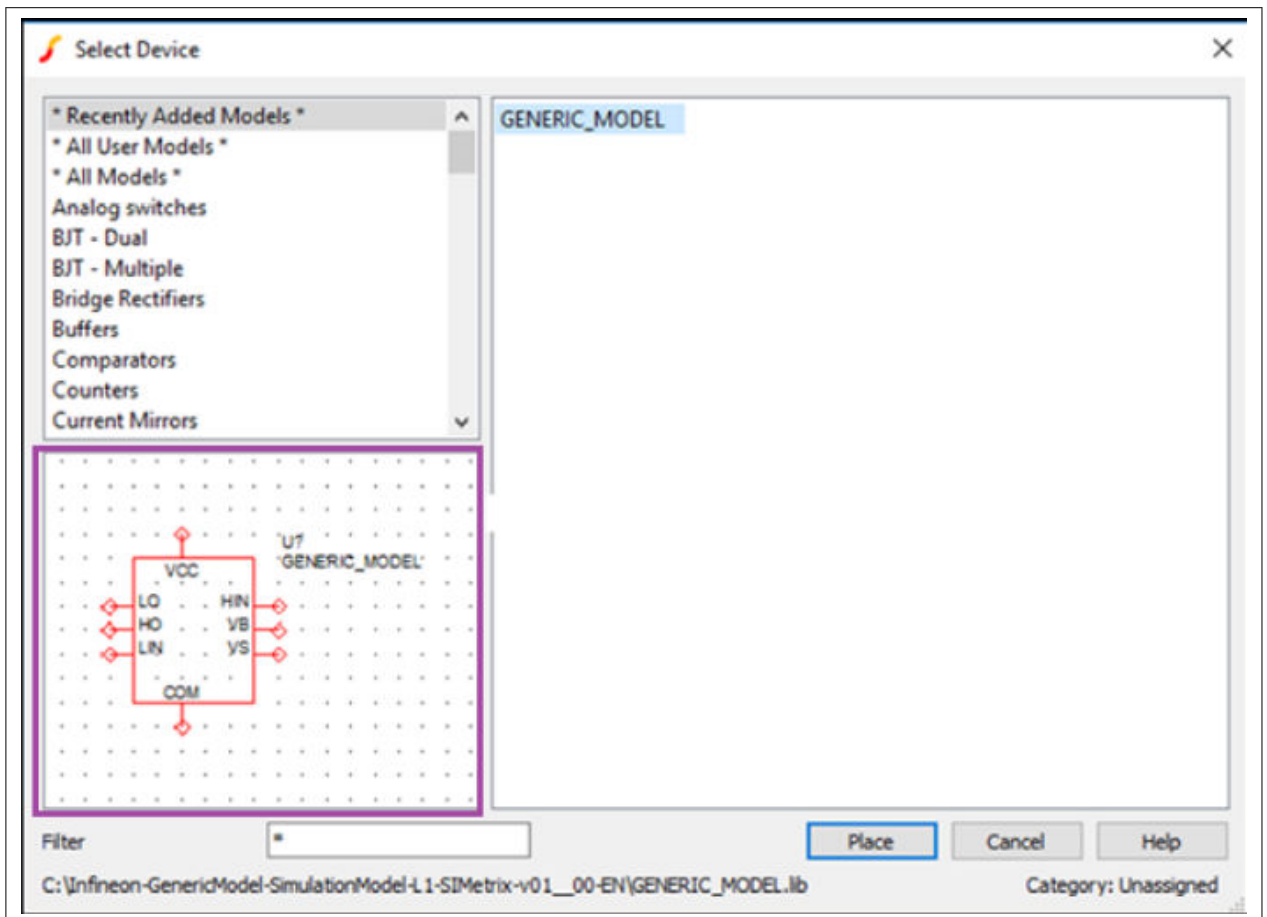
Once the simulation model has been installed, the symbol (**.xsxlib**) and the code source file (**.lib**) can be placed on the schematic and start a simulation:

- To associate a symbol to a new installed library, go to **Place → From Model Library**



**Figure 4 Associate symbol with model**

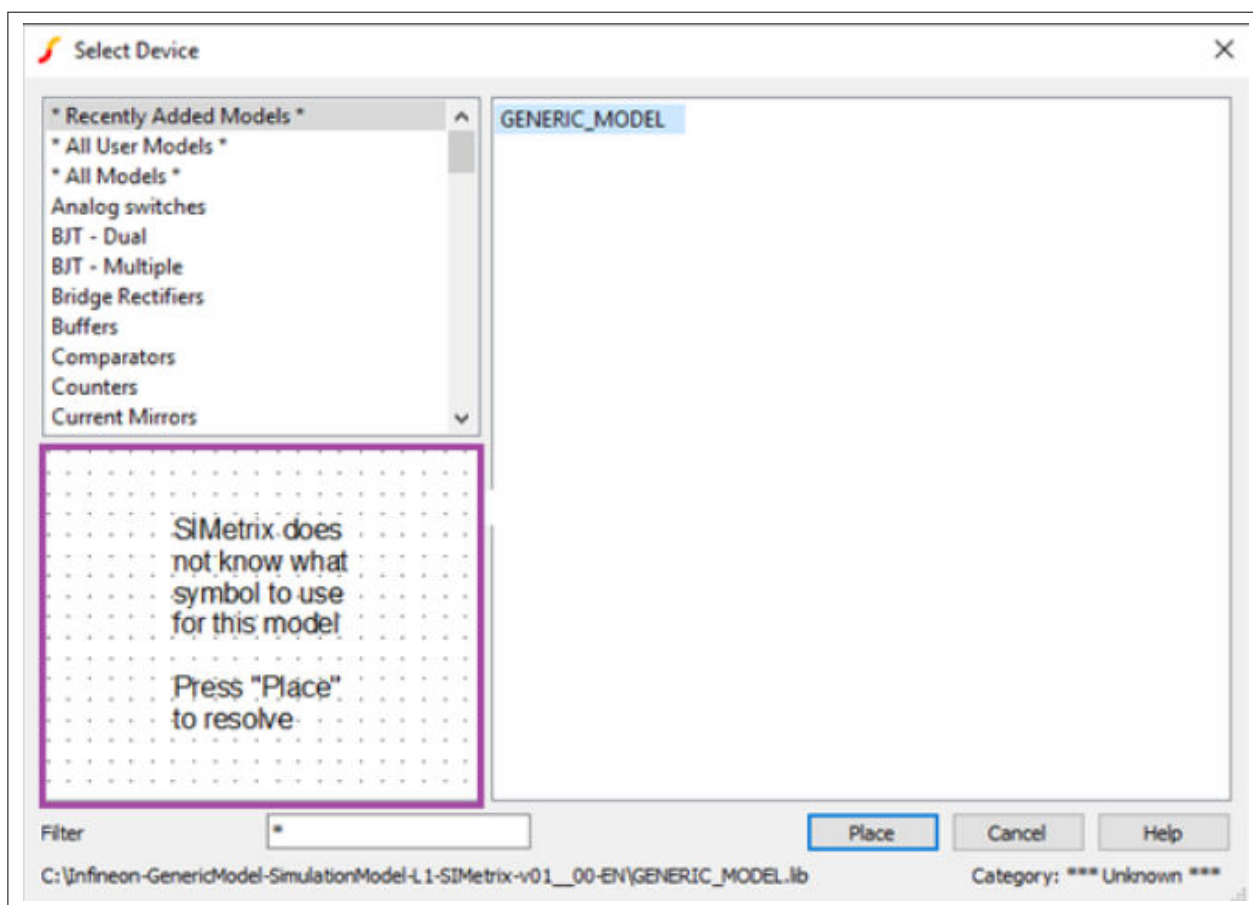
- In the **“Recently Added Models”** section select the chosen part, e.g. **GENERIC\_MODEL** and a schematic symbol will appear in the preview window, like in **Figure 5**



**Figure 5 Associate Symbol with Model**

- In case **SIMetrix** display the following message: **“SIMetrix does not know what symbol to use for this model”** like in **Figure 6**, press **Place**, select **Auto Create Symbol** (Figure 7) and a default rectangular shape will be associated.

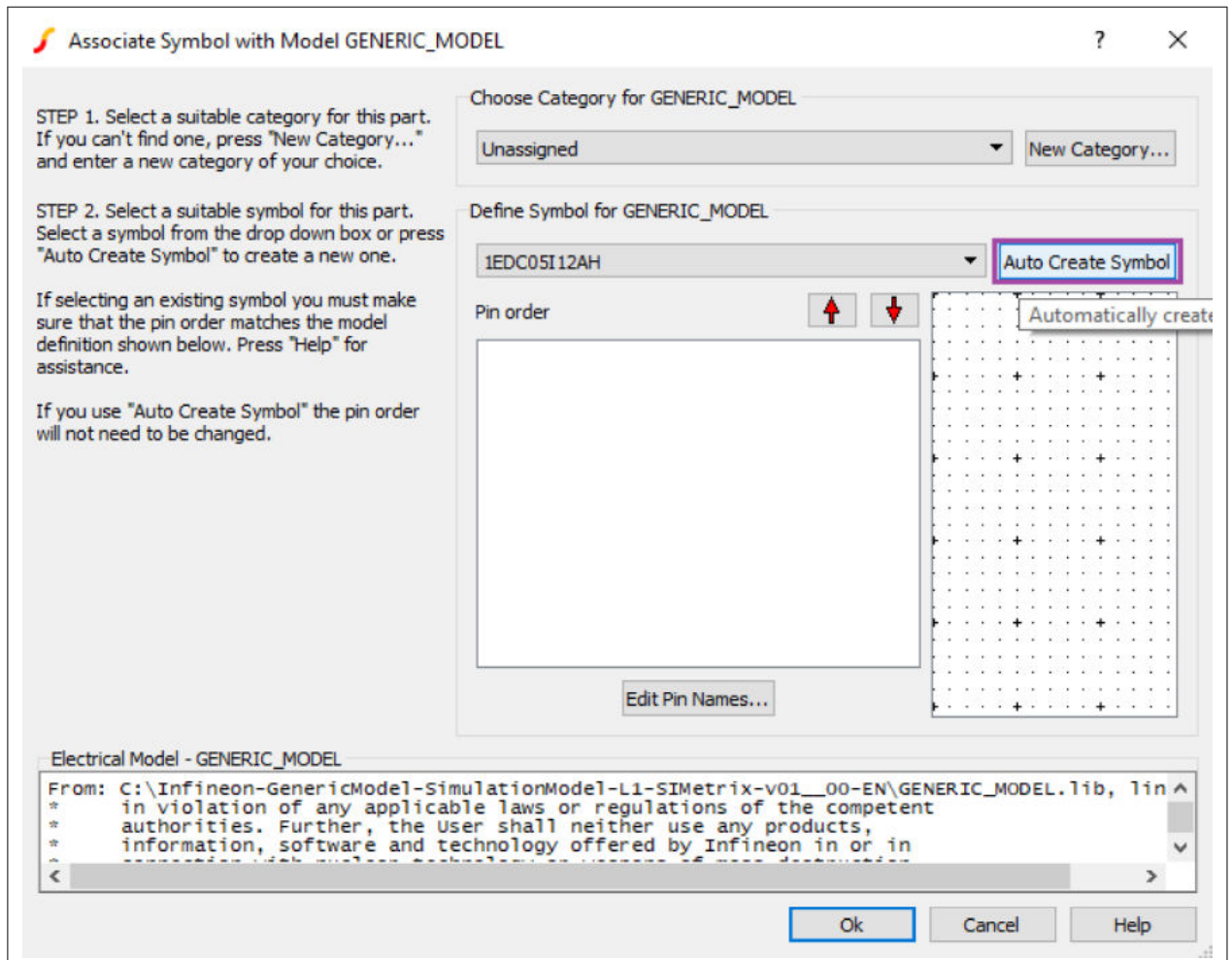
## 2 Starting a new project



**Figure 6** Associate symbol with model for unknown part



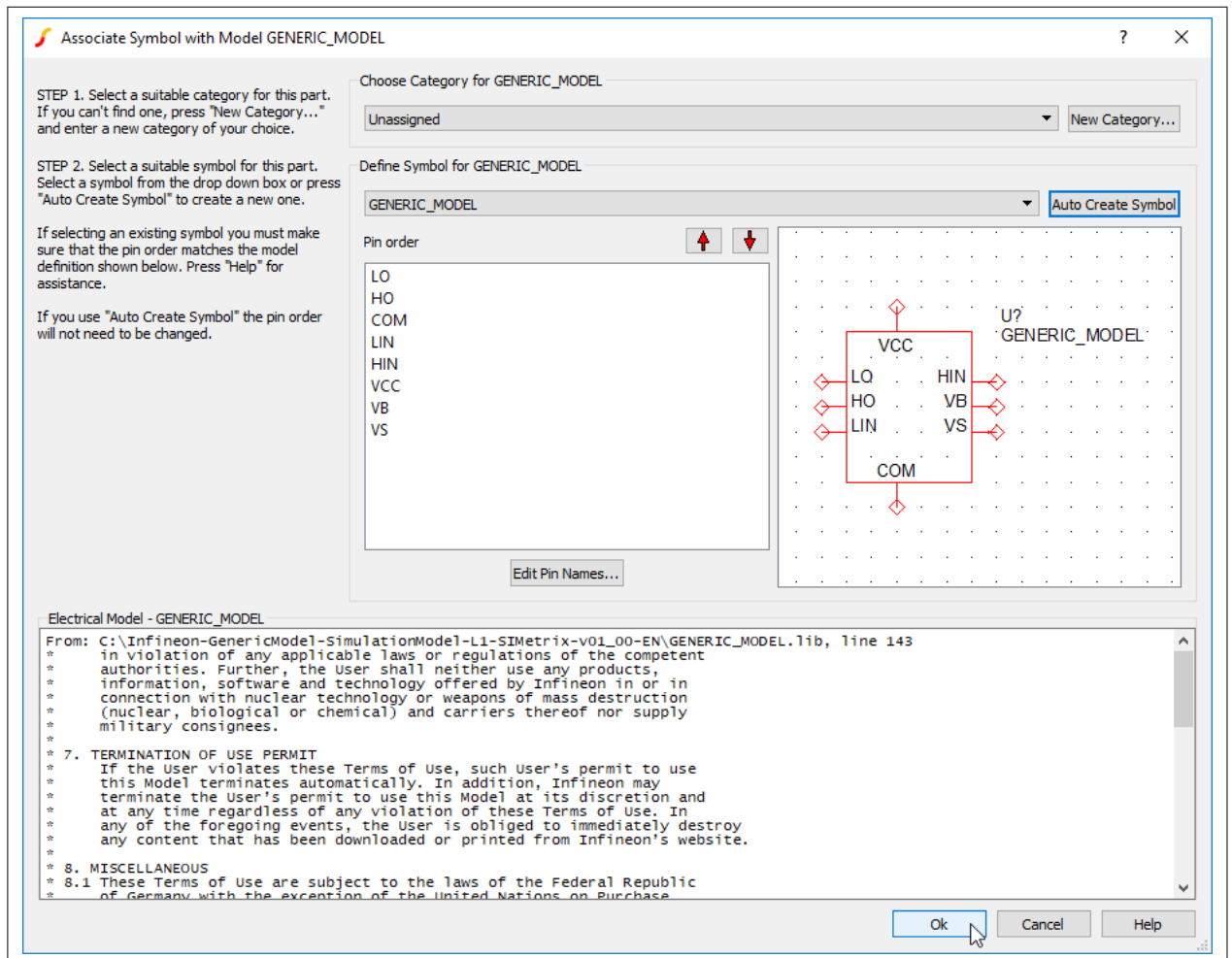
### 2 Starting a new project



**Figure 7 Associate symbol with model - Auto create symbol**

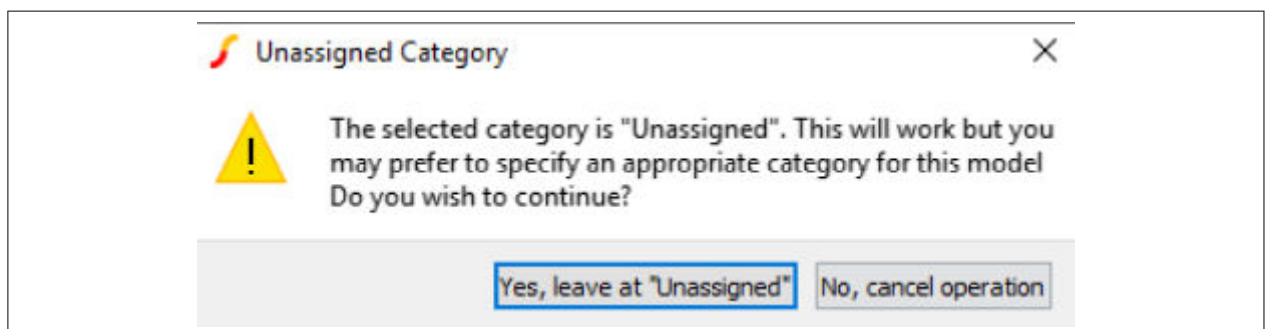
- A preview of the model can be observed in the **Electrical Model** window.

## 2 Starting a new project



**Figure 8 Associate symbol with model – Auto create symbol**

- If the part will be placed in the **"Unassigned"** category, press OK and select **Yes, leave at "Unassigned"** (Figure 9), otherwise select a proper category or create a new one.



**Figure 9 Associate Symbol with Model – Auto Create Symbol**

The symbol is now ready and could be place in the schematic.

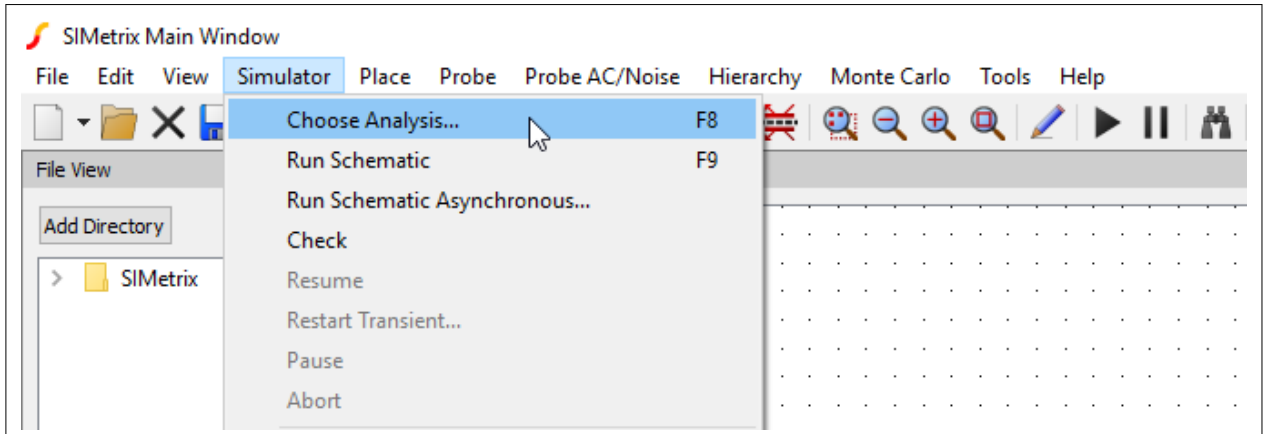
## 2 Starting a new project

### 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.

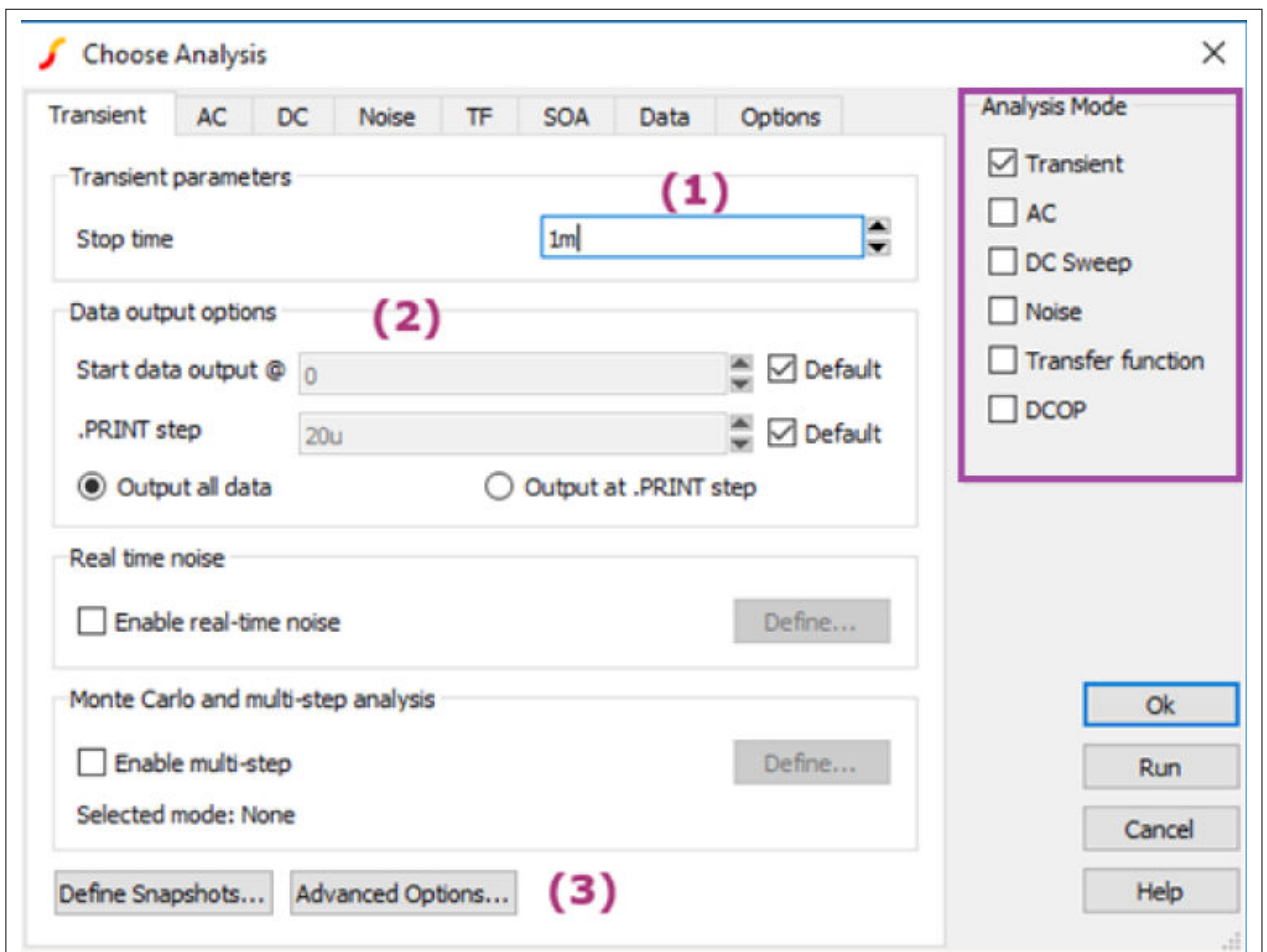
To perform this analysis follow the next steps:

- From the main menu select **Simulator → Choose Analysis**



**Figure 10 Choose Analysis**

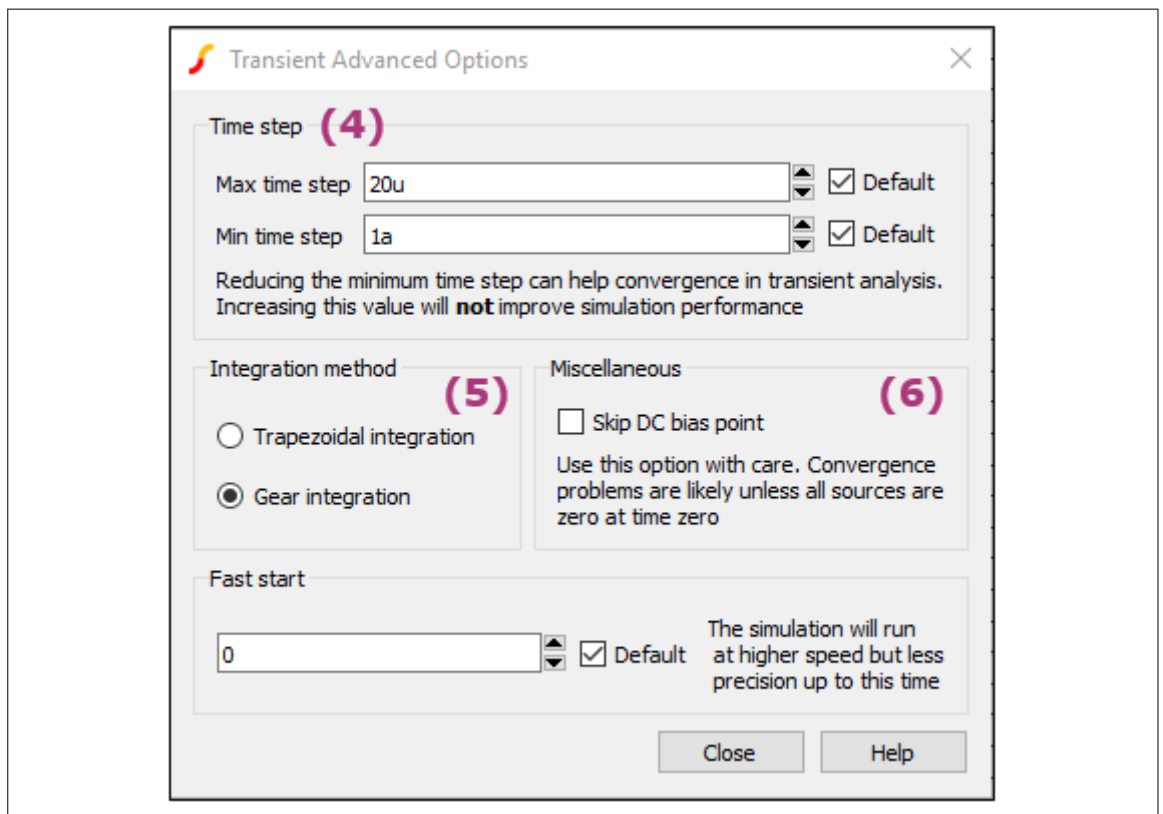
- From „**Choose Analysis**” dialog box select **Transient** check box situated on the right under „Analysis Mode”.



**Figure 11 Simulation settings**

### 2 Starting a new project

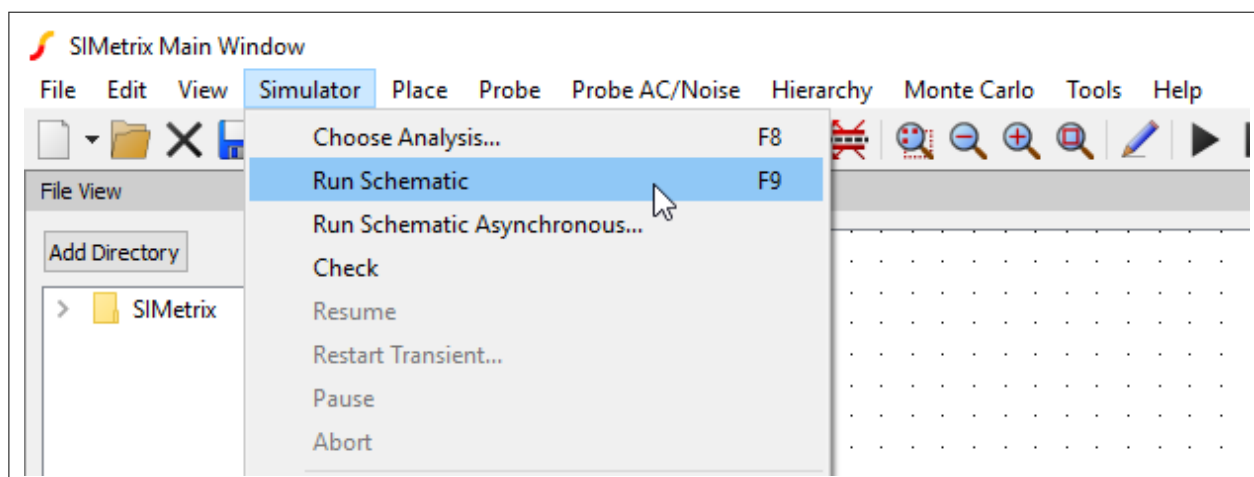
- Select **Transient** tab at the top and enter the simulation parameters as described below:
  - **Stop time (1)** = total simulation time. Note that the simulation can be paused before the **Stoptime** is reached, allowing the results obtained so far to be examined
  - **Data output option (2)** = Sometimes it is desirable to restrict the amount of data being generated by the simulator which in some situations can be very large. It can be specified that data output does not begin until after a certain duration. It can also be specified a certain time interval for the data to be displayed
  - **Advanced option (3)** = press on the button and will opens the dialog box (Figure 12 Transient Advanced Options)



**Figure 12 Simulation settings**

- **Time Step (4)** = defines the maximum internal step size, which is dependent on the specified Stop Time parameter
  - **Integration Method (5)** = it is recommended to set this to **Gear integration** because in most cases it works better and is less noisy than Trapezoidal integration
  - **Skip DC bias point (6)** = the simulation will start with all nodes at zero volts (transient analysis convergence problems are very likely if this is not the case as the circuit will be starting from an unknown state)
- From the main menu, select **Simulator → Run Schematic** to perform the analysis

### 2 Starting a new project



**Figure 13** Run the simulation

## 2 Starting a new project

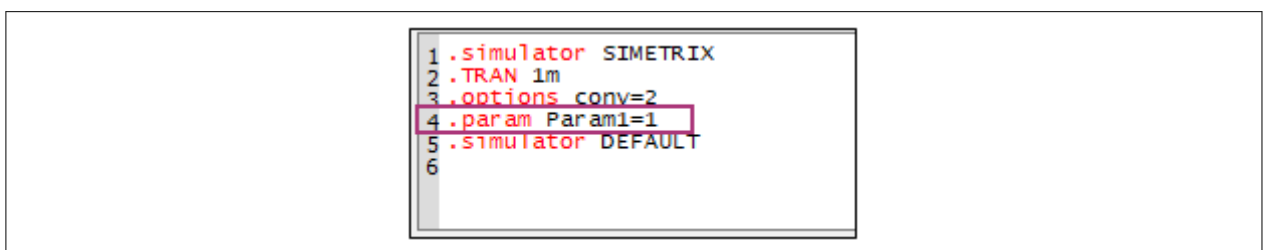
### 2.4 Set up global parameters

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 **SIMetrix**:

- Open the schematic sheet
- Press **F11** on keyboard to toggle **Command Window**;
- Use **.PARAM** command followed by parameter name and value

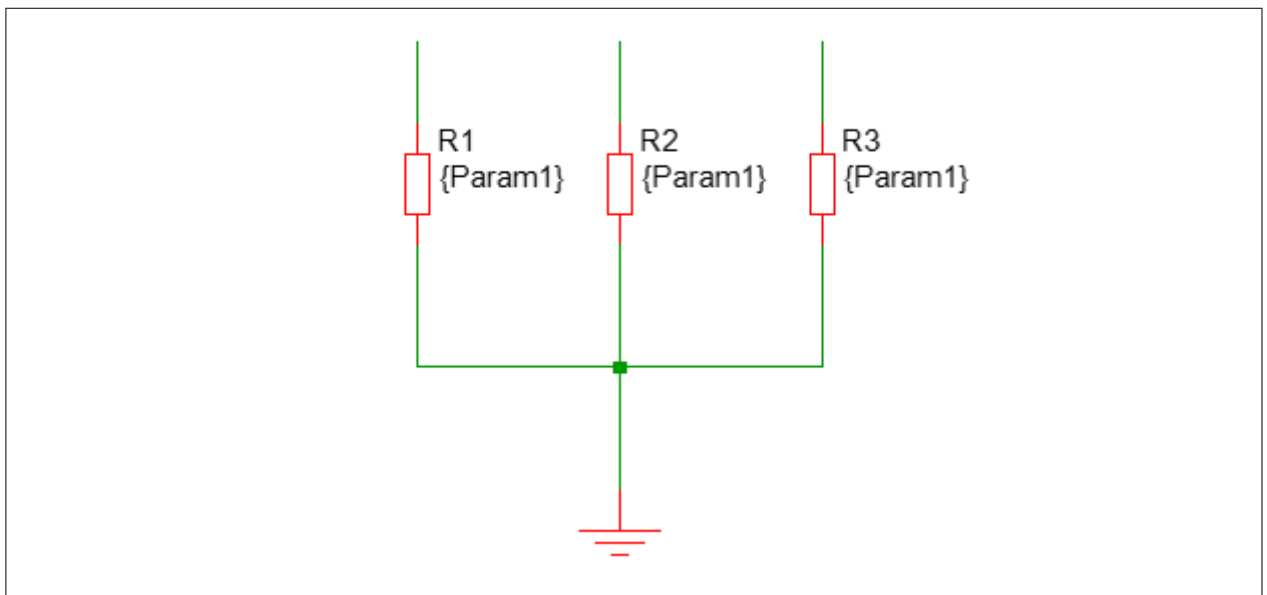


```
1 .simulator SIMETRIX
2 .TRAN 1m
3 .options conv=2
4 .param Param1=1
5 .simulator DEFAULT
6
```

**Figure 14** Define global parameter

Two practical examples are shown below:

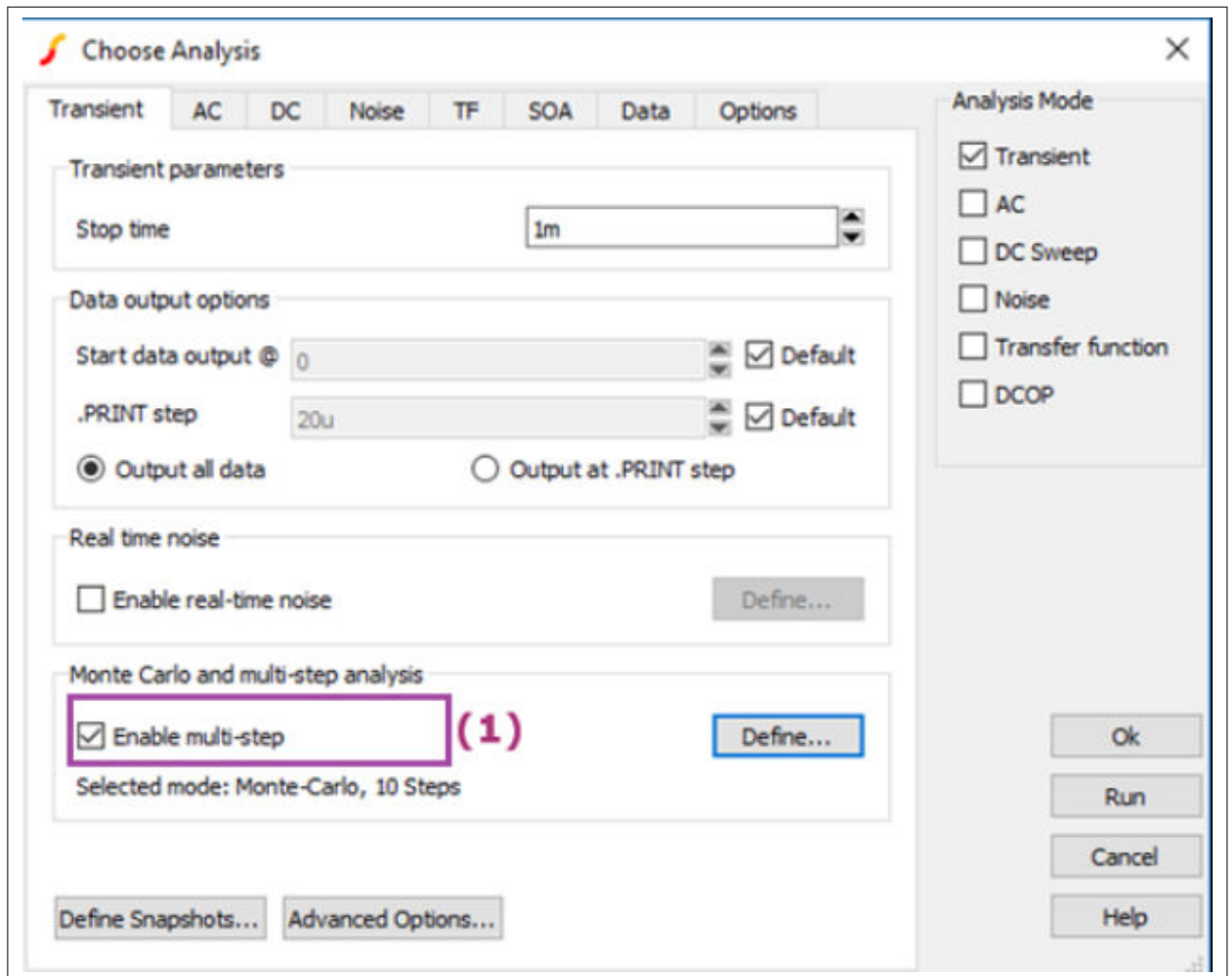
- Apply the same value to multiple instances: in the value field of any component use the parameter name between curly brackets



**Figure 15** 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 16**
- In “**Chose Analysis**” menu select “**Enable multi-step**” (1) and click on “**Define...**”

## 2 Starting a new project



**Figure 16** Set up a parametric simulation

- Select “**Parameter**” from “**Sweep mode**” menu (3), enter the “**Parameter name**” (4) and define the “**Step parameters**” (5)

## 2 Starting a new project

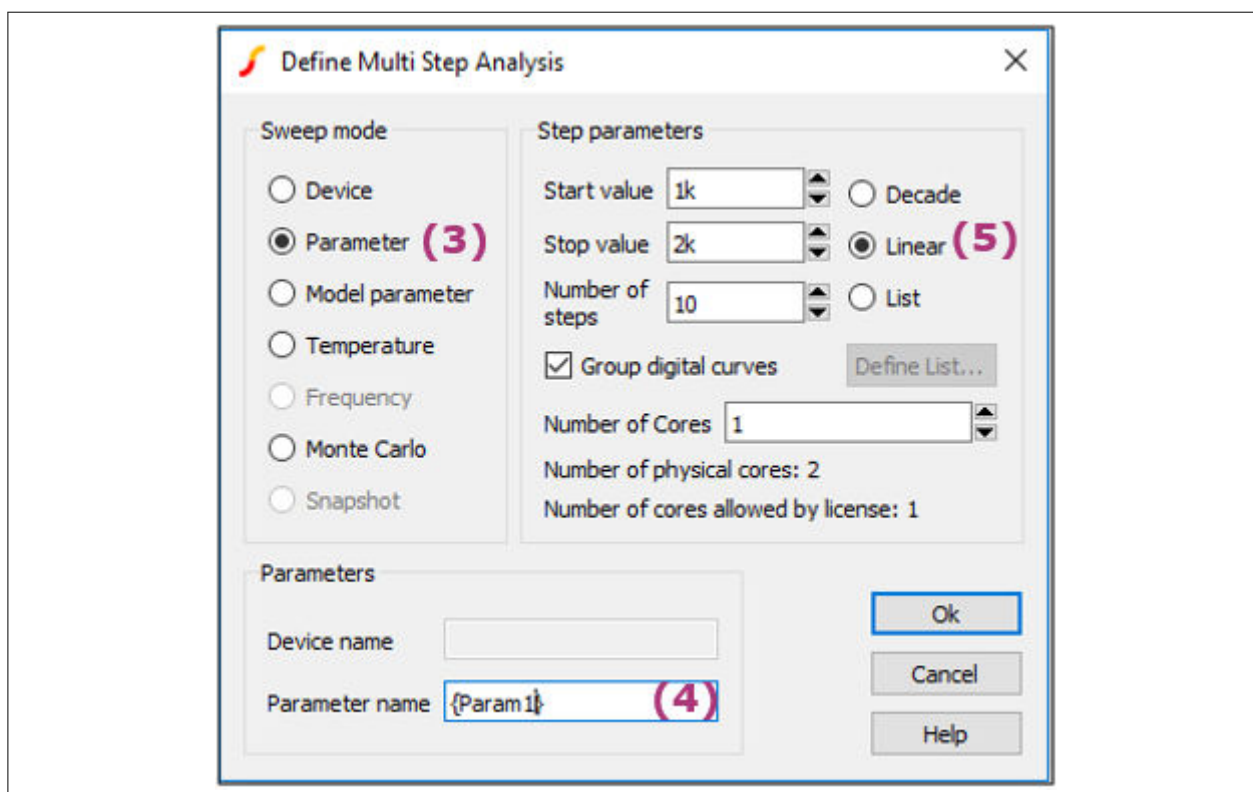


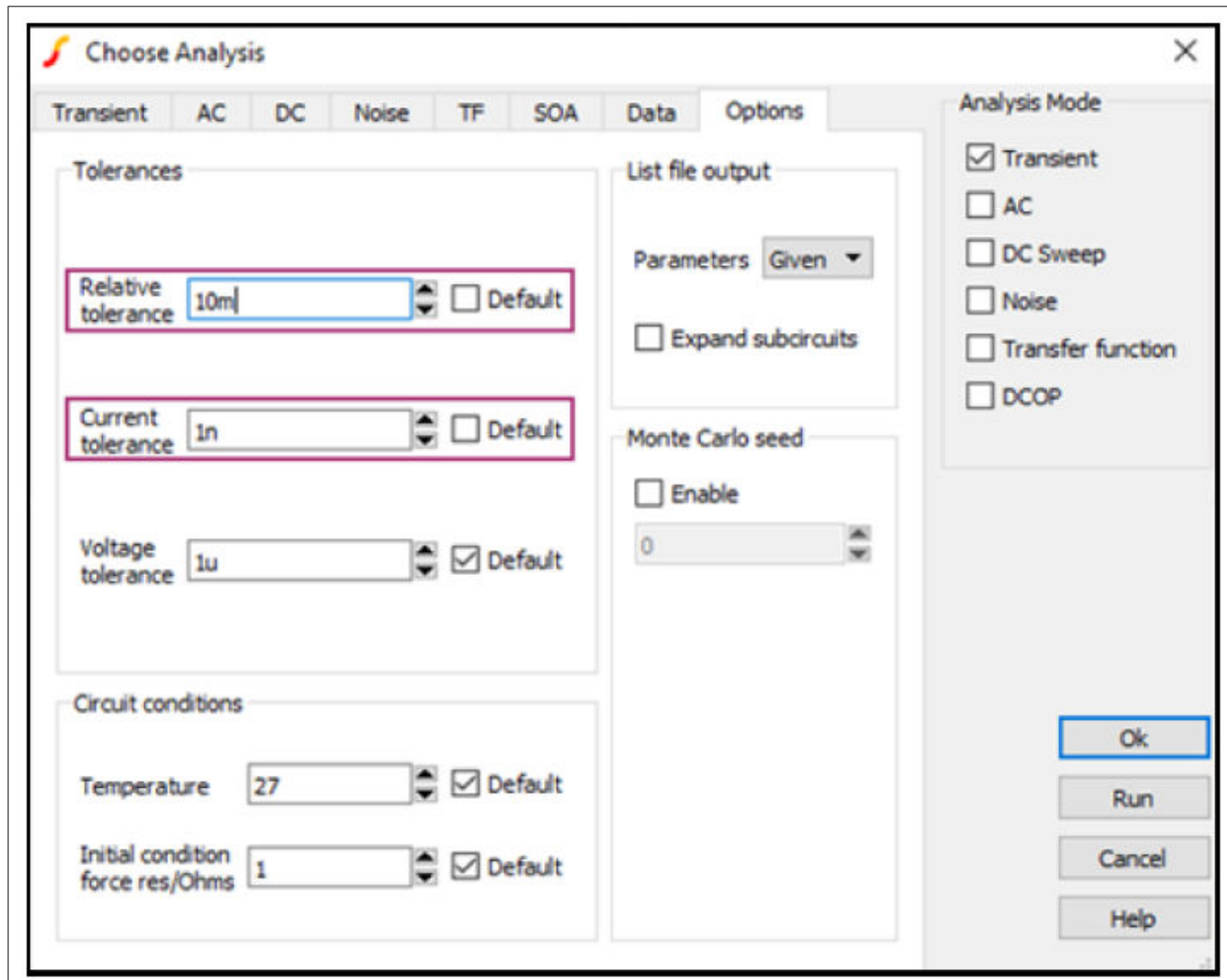
Figure 17 Set up a parametric simulation (b)



## 2 Starting a new project

### 2.5 Recommended simulation settings

As **Spice** was not originally designed for power electronics and highly non-linear components, the simulation parameters (**Simulator** → **Choose Analysis** → **Options**) may be optimized for a specific design.



**Figure 18** Simulation settings

The following recommended values facilitate convergence:

**Table 1** Recommended values for simulation parameters

Relative tolerance=10m	relative accuracy of voltage and currents
Current tolerance=1n	maximum current accuracy
Integration method=Gear	
Skip DC bias point	
.options noOplter	skip the initial attempt at solving the DC operating point; can be define in the schematic's <b>F11</b> window

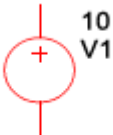
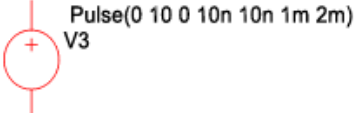
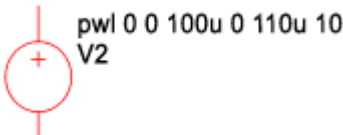



In general, the delivered models need to detect a rising edge starting from 0V on Inputs and Supply voltages to properly for a proper start-up sequence. It is recommended to use **VPWL** or **VPULSE** instead of **VDC** component.

### 2 Starting a new project

*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
		
		

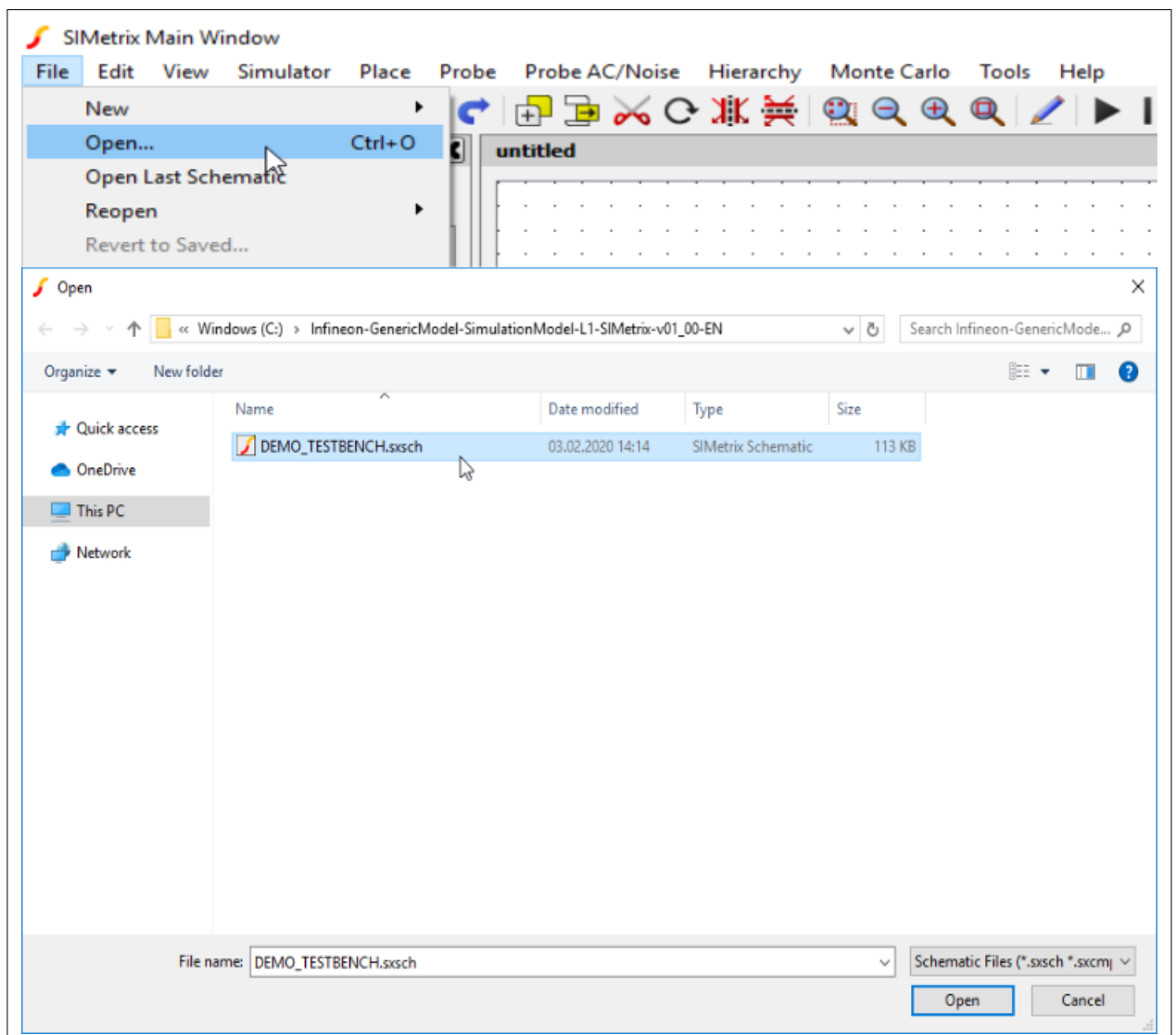
### 3 Opening an existing project

## 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

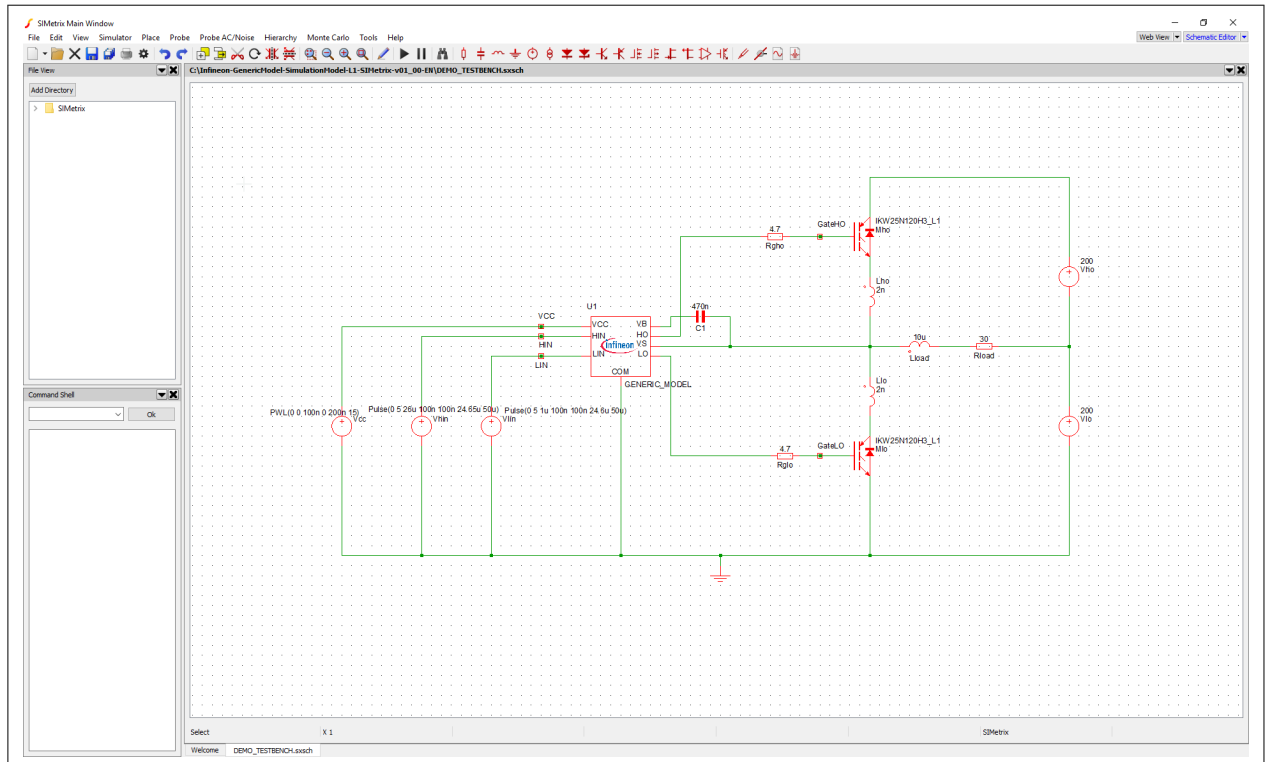
- Open the **SIMetrix** application and from the **File → Open**
- In the browse window locate and open the **schematic file** (.xsch), e.g. “DEMO\_TESTBENCH.xsch” file from the Infineon-GenericModel-SimulationModel-L1-SIMetrix-v01\_00-EN package directory. For example:



**Figure 19** Open an existing project

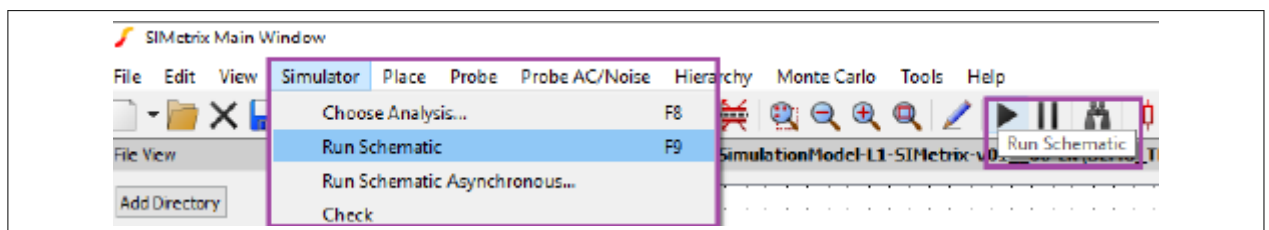
- The window will display the application testbench schematic, similar to that below:

### 3 Opening an existing project



**Figure 20** Open the demo\_testbench

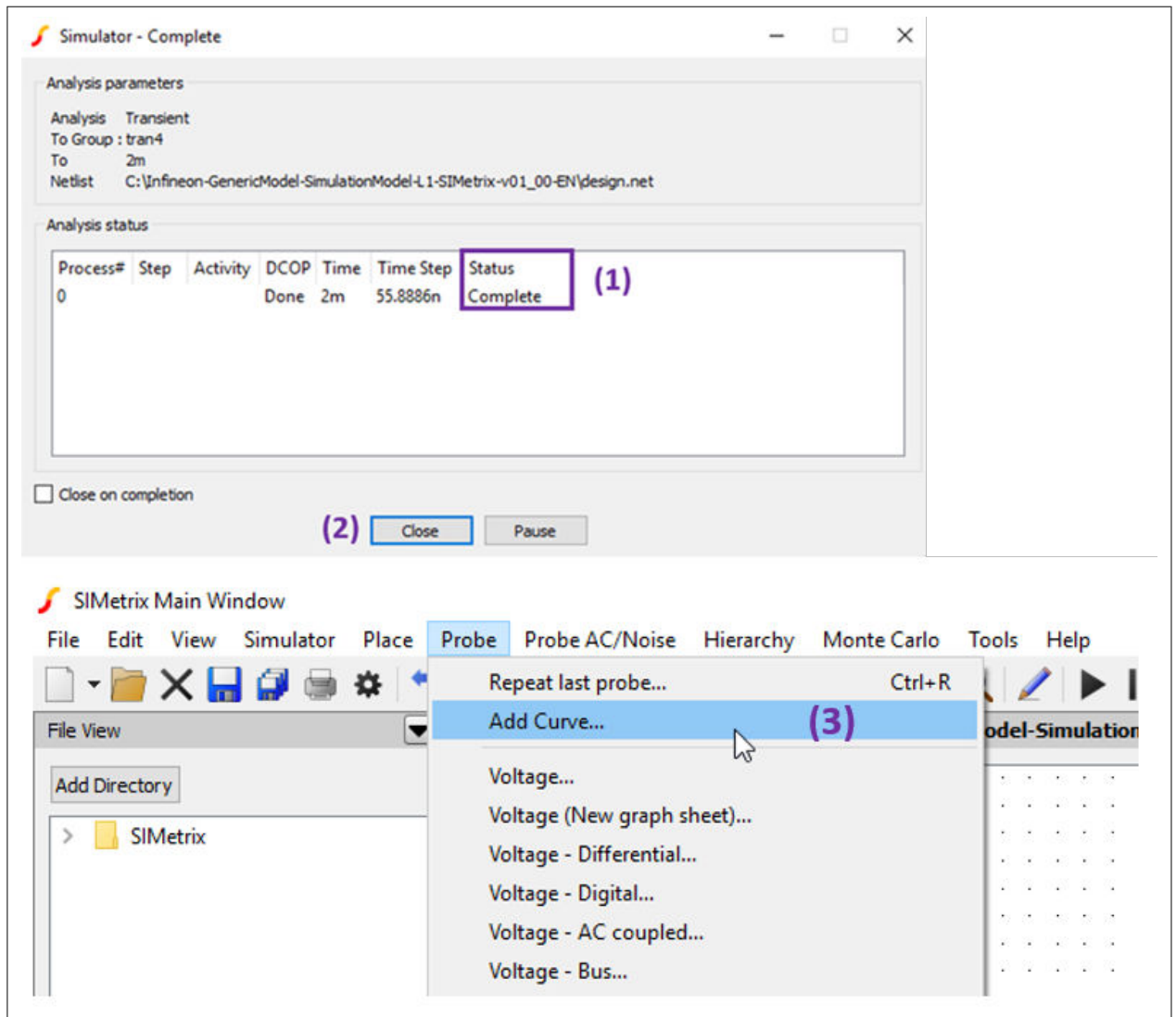
- To start the simulation, press “**F9**” on the keyboard or open the **Simulator** → **Run Schematic**. Alternatively, the play button on the **SIMetrix** toolbar may be used



**Figure 21** Start a simulation

- Once the netlist has been generated, the “**Simulator**” will open and the simulation will automatically start
- After the **Simulator Status** will show Complete (1) the signals can be plotted. To do this, close the Simulator window (2) and from the main menu select **Probe** → **Add curve** (3)

### 3 Opening an existing project



**Figure 22 Plot signals**

- To plot the signals please select them from the **Available vectors (4)** then press “OK” button **(5)**

*Note: Filtering of the desired signals can be done from “Edit Filter” menu (6), using the check boxes in the right of the window by their type (e.g. voltages, currents (7))*

## 3 Opening an existing project

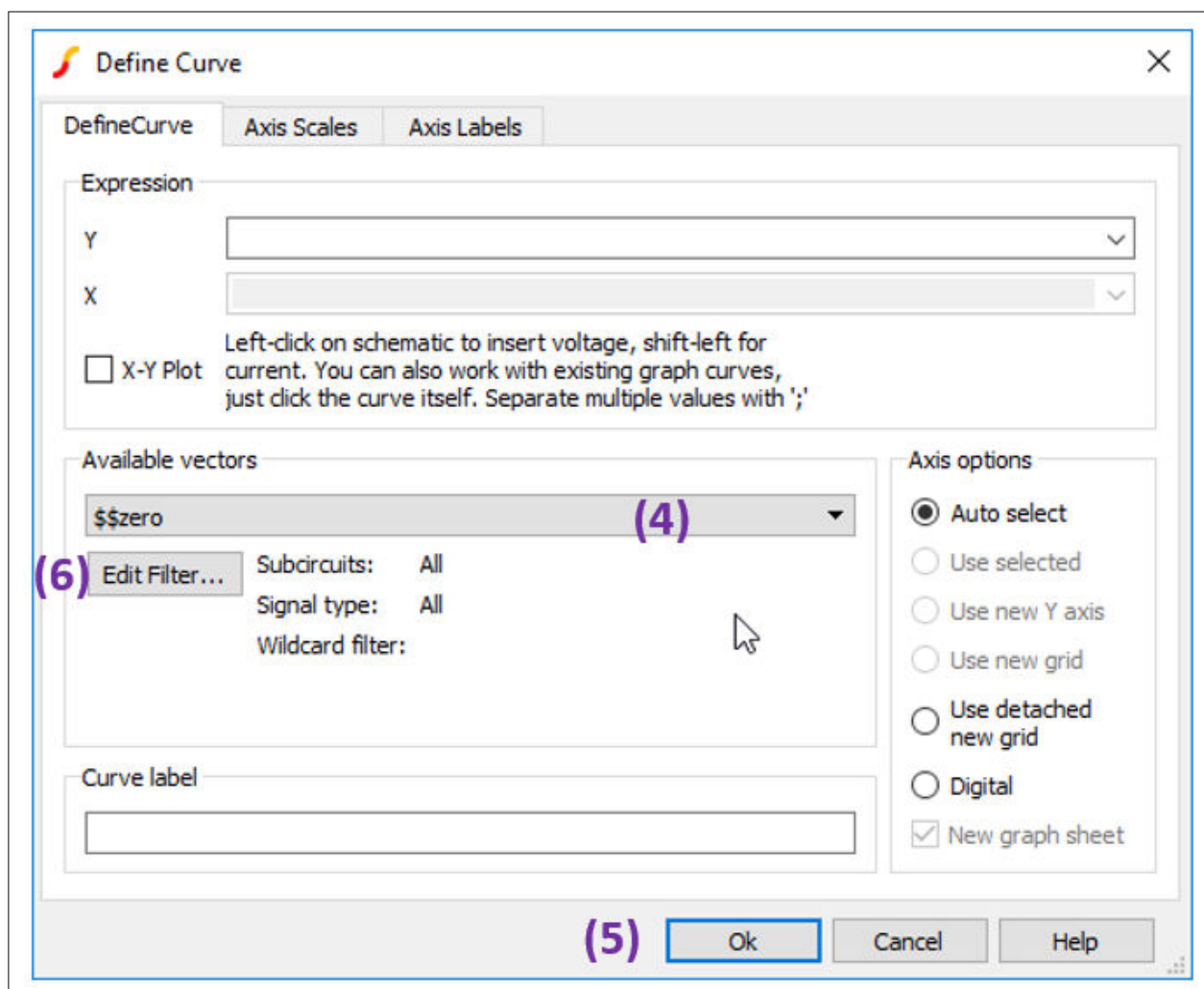
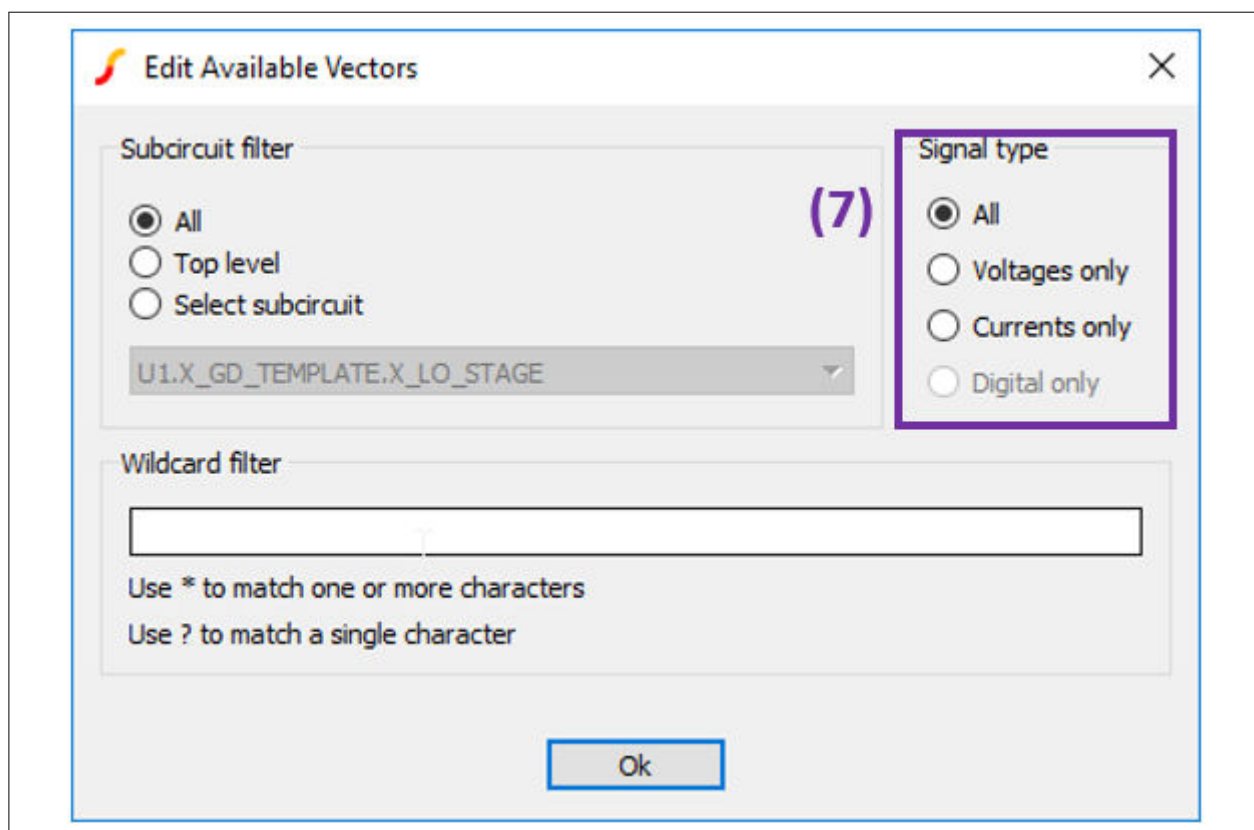


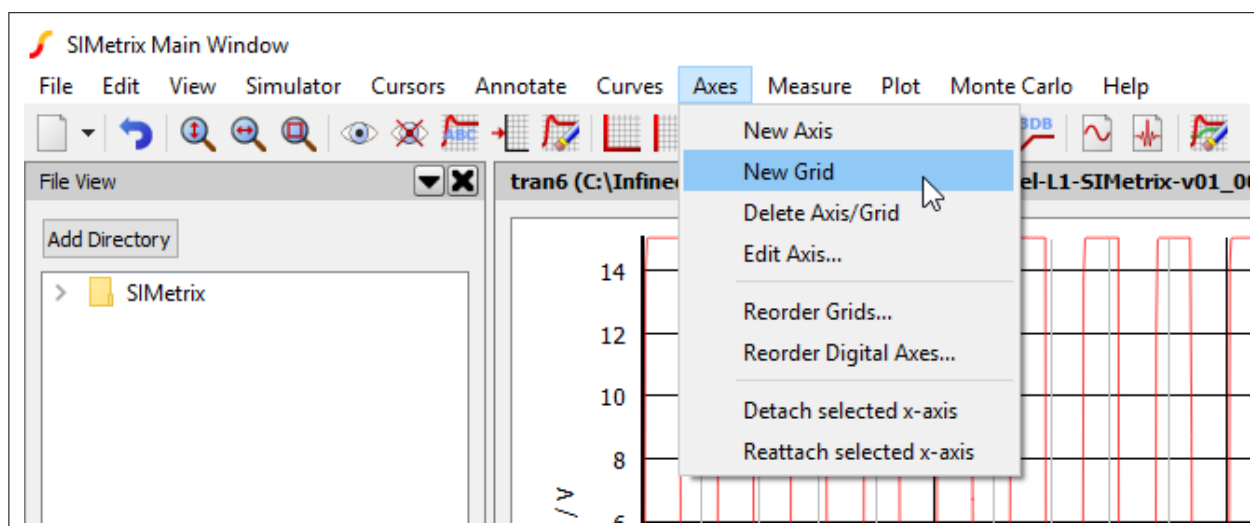
Figure 23 Plot signals

### 3 Opening an existing project



**Figure 24** Plot signals

- To add other windows, select from the main menu **Axes → New Grid**



**Figure 25** Plot signals

## **Revision history**

<b>Document version</b>	<b>Date of release</b>	<b>Description of changes</b>
V1.0	2020-01-20	GettingStarted released



## Trademarks

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

**Edition 2021-12-14**

**Published by**

**Infineon Technologies AG**  
**81726 Munich, Germany**

**© 2021 Infineon Technologies AG**  
**All Rights Reserved.**

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

**Email: [erratum@infineon.com](mailto:erratum@infineon.com)**

**Document reference**  
**IFX-cwf1637752464530**

## 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.

## 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.