

Getting started

Targeted Simulator: LTspice

About this document

Scope and purpose

The current document provides guidance on how to get started with the attached simulation model using the **LTspice** 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 schematic	3
2.2 Import an existing model	4
2.2.1 Local installation.....	4
2.2.2 Global installation.....	5
2.3 Set up and run a transient analysis	6
2.4 Set up global parameters.....	7
2.5 Recommended simulation settings.....	9
3 Opening an existing project.....	11
3.1 Steps to run the simulation:.....	11
3.2 Simulation results and measurements	12
4 Disclaimer	14
4.1 Scope of use.....	14
4.2 Important notice	14
4.3 Confidential information	14
4.4 Warranty	14
4.5 Liability	14
4.6 Export regulations.....	15
4.7 Termination of use permit	15
4.8 Miscellaneous	15
Revision history.....	16

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)
- **.asy** : schematic symbol view for graphical user interface
- **.asc** : 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 LTspice.

2.1 Create a new schematic

Begin by opening LTspice application. From **File → New → Schematic**, start a new project. A new schematic panel will be displayed like in **Figure 1**.

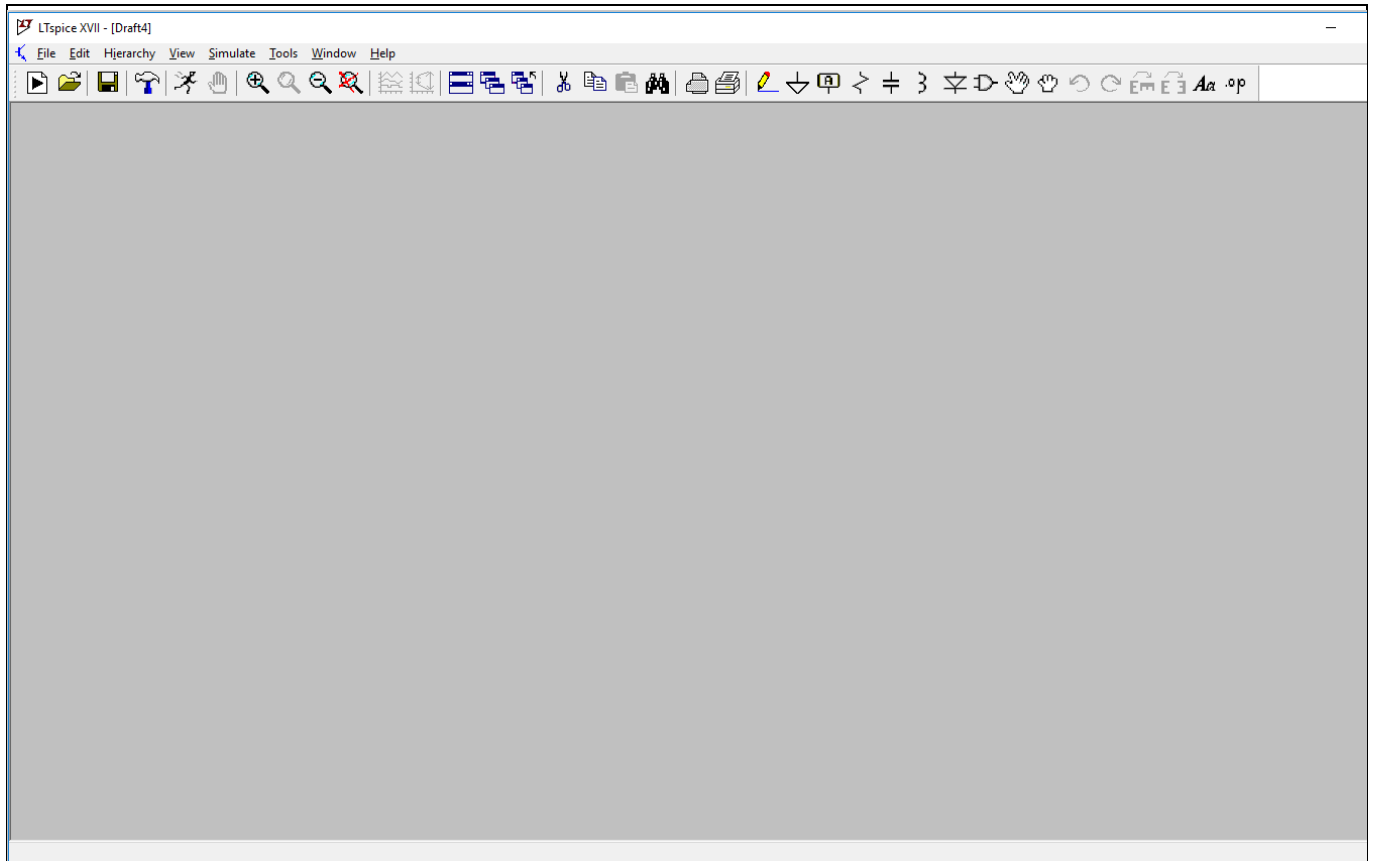


Figure 1 Create new schematic

Getting started

Targeted Simulator: LTspice

Starting a new project

2.2 Import an existing model

*Note: Before setting up a simulation, model libraries of interest must be integrated in **LTspice** tool. This section describes how to import the simulation model files, using one of the following methods: Local installation, Global installation*

2.2.1 Local installation

This method requires that the code source (**.lib**) and the schematic symbol view (**.asy**) to be placed in the same folder where the schematic file was created as in [Figure 2](#).




	simulation_model	13.01.2020 11:35	LTspice Schematic
	simulation_model	10.01.2020 14:49	LTspice Symbol
	simulation_model	10.01.2020 14:30	LIB File

Figure 2 Local installation

[Figure 3](#) shows how to place a symbol on schematic:

- Go to **Edit** → **Component** (1)
- Chose schematic folder from the dropdown list **Top Directory** (2)
- Symbol should be display in the below list
- Double click to place the symbol on schematic (3)

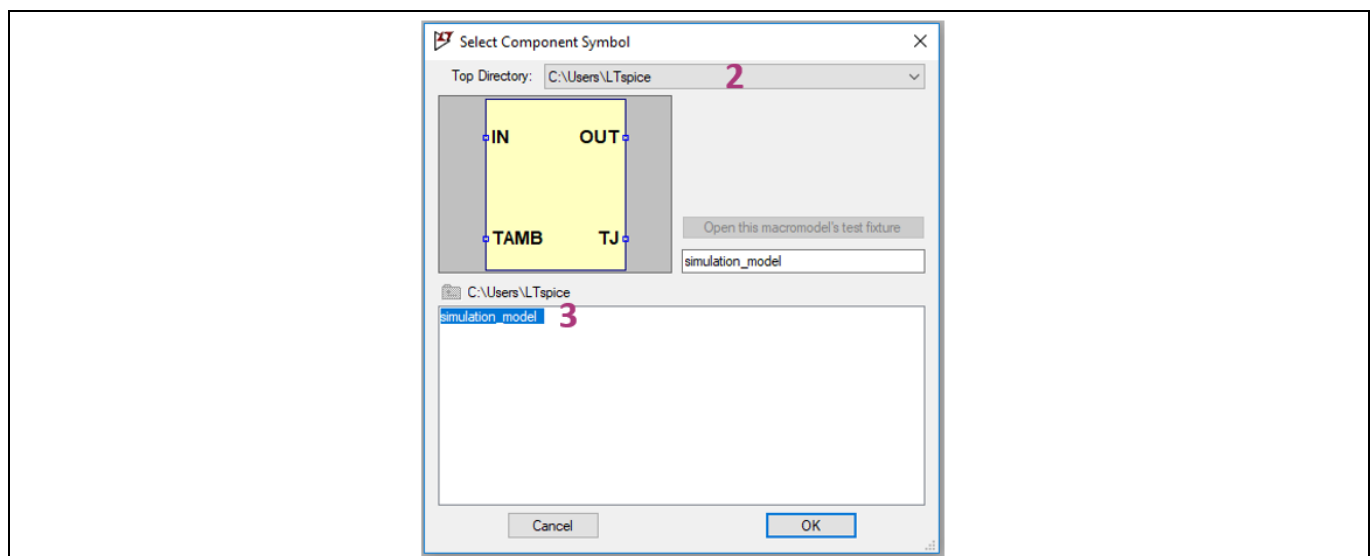


Figure 3 Place part from folder

Getting started

Targeted Simulator: LTspice

Starting a new project

2.2.2 Global installation

A more structured approach is to create a user library into **LTspice** library structure:

- **/LTC/LTspice_version/lib/sub/myLib** (for code source file **.lib**)
- **/LTC/LTspice_version/lib/sym/myLib** (for schematic symbol view file **.asy**)

Copy simulation models' files to appropriate folders and then place the part on the schematic:

- Go to **Edit** → **Component**
- Double click on **myLib** as in **Figure 4**

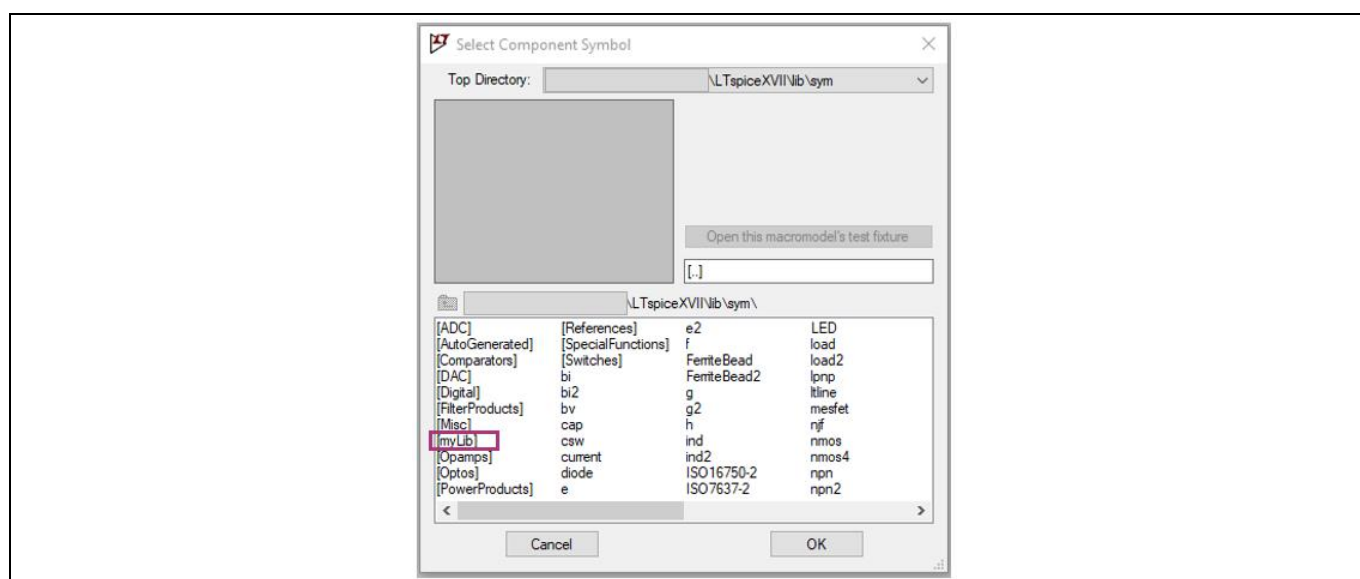


Figure 4 User library

- Double click to place the symbol on schematic page as shown in **Figure 5**

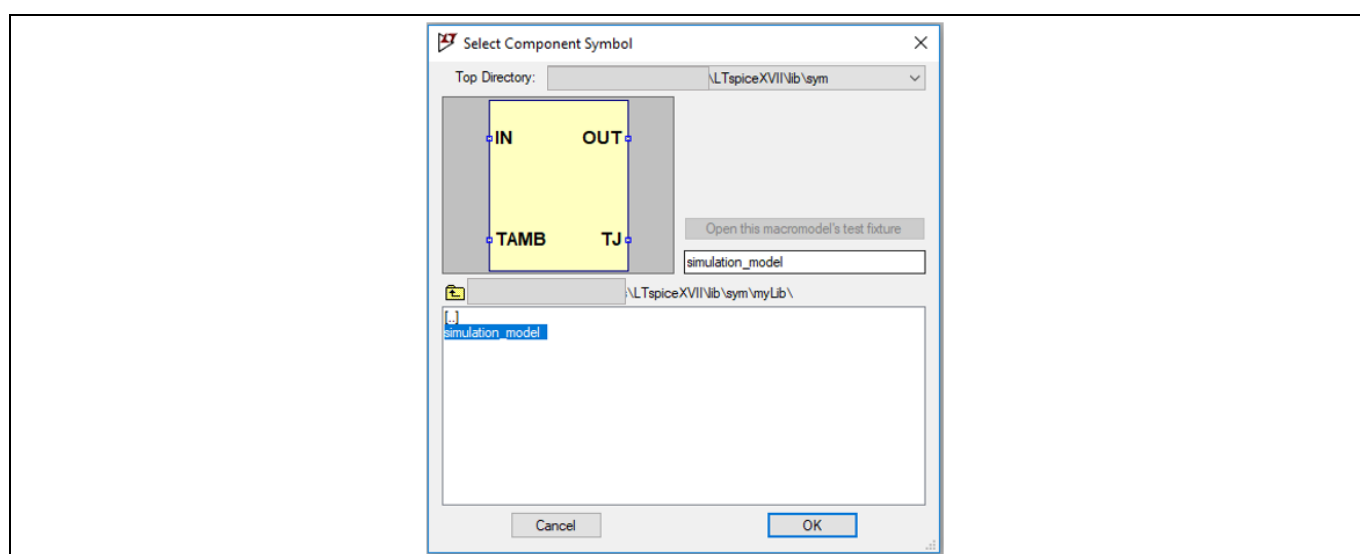


Figure 5 Place part from user library

In case of LTspice software update, there is the possibility that user created libraries may be erased or rewritten.

2.3 Set up and run a transient analysis

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

- From main menu select **Simulate** and choose **Edit Simulation Command**
- **Edit Simulation Command** dialog box appears like in [Figure 6](#)

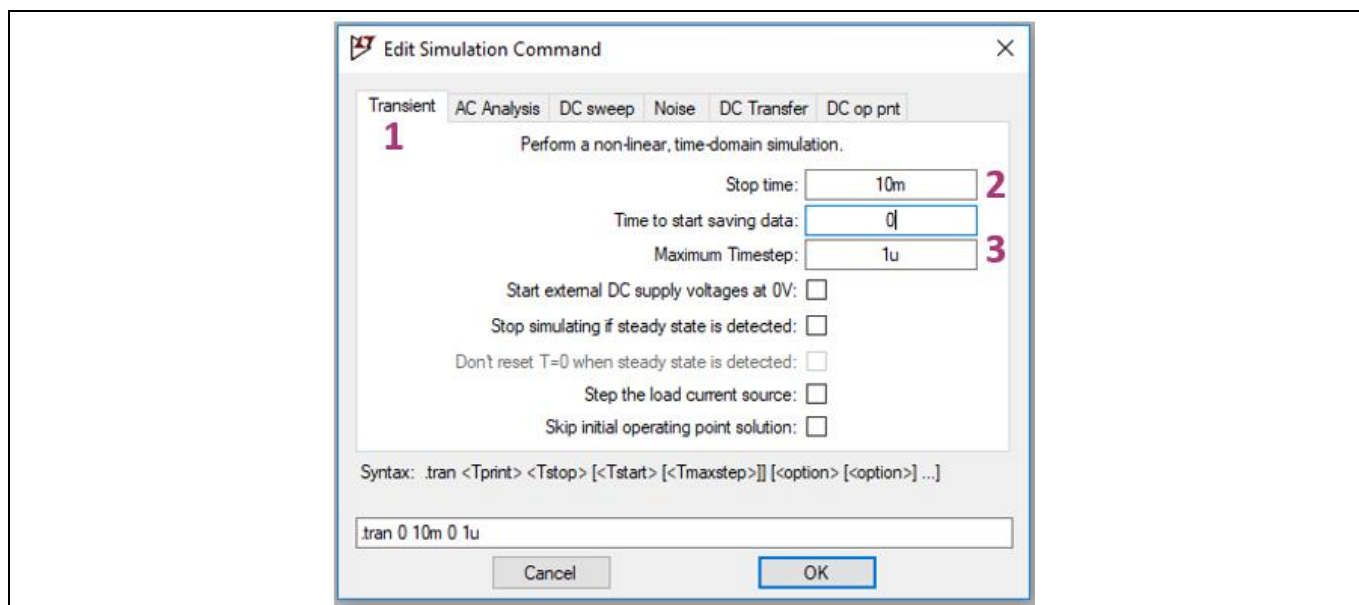


Figure 6 Simulation settings

- From top list choose **Transient (1)** and specify required parameters:
 - **Stop time (2)** = total simulation time
 - **Maximum Timestep (3)** = maximum internal step size, which is dependent on the specified **TSTOP** parameter
- Click **OK** and place **.tran** command on schematic
- From **Simulate** menu, press **Run** to perform the analysis

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 7 shows how to define a global parameter in **LTspice**. Follow the instructions given below:

- Open schematic sheet
- From **Edit** menu choose **.op Spice Directive's'**
- Use **.PARAM** command followed by parameter name and value

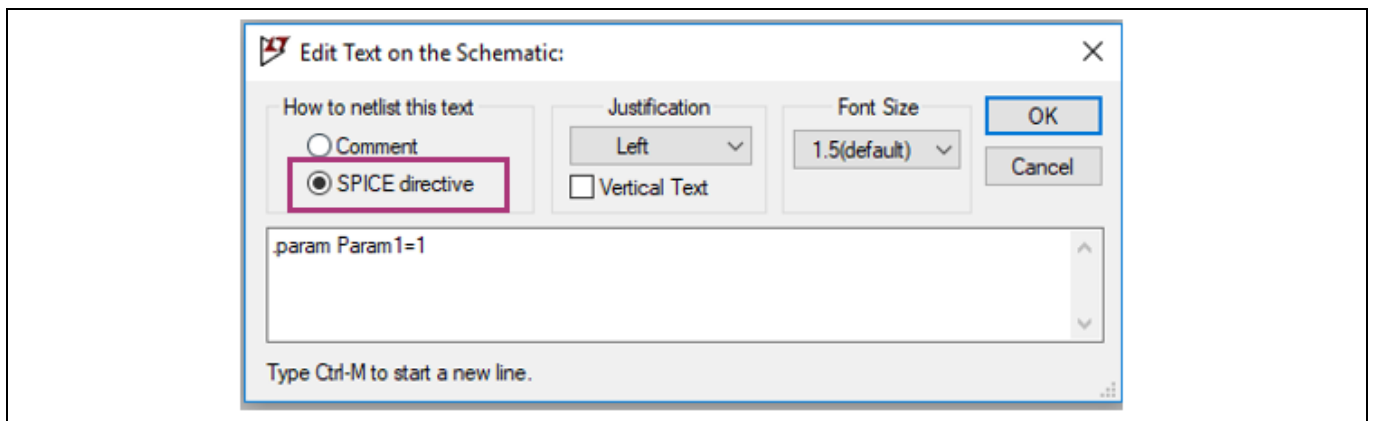


Figure 7 Define new parameter

- Make sure that **SPICE directive** is selected
- Click **OK** and place command on the schematic

Two practical examples are shown below:

Figure 8 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.

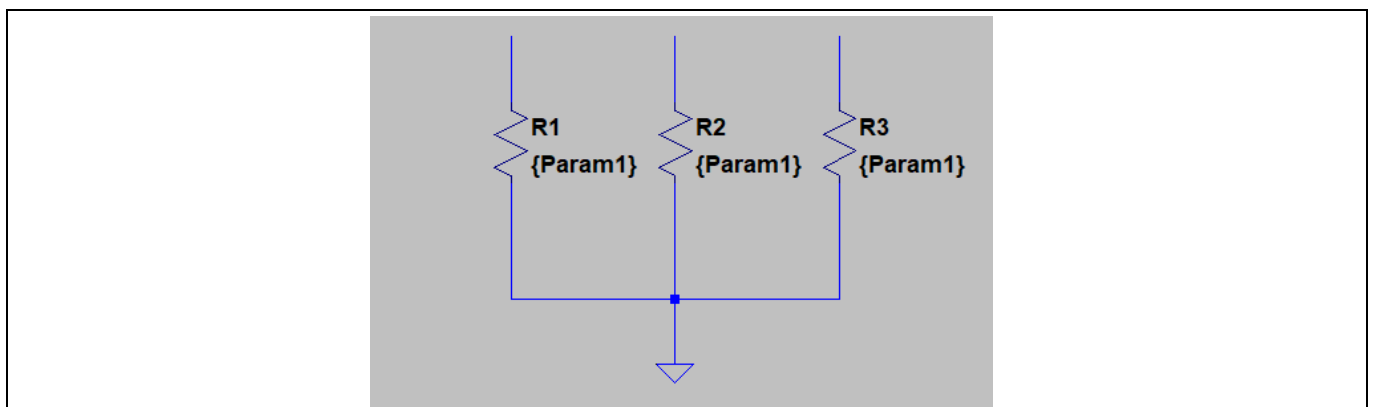


Figure 8 Global parameter for multiple instances

Getting started

Targeted Simulator: LTspice

Starting a new project

Figure 9 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.

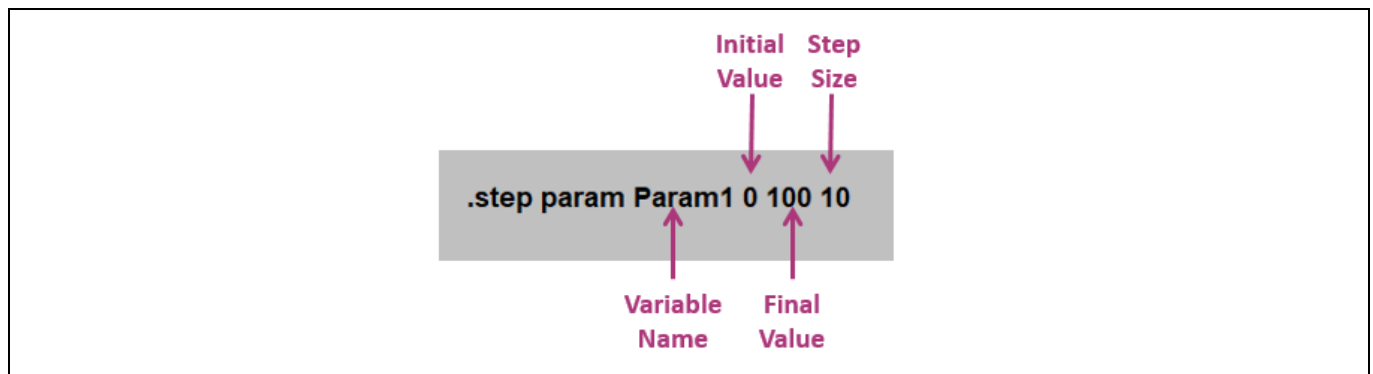


Figure 9 Set up a parametric simulation

Getting started

Targeted Simulator: LTspice

Starting a new project

2.5 Recommended simulation settings

As **LTspice** was not originally designed for power electronics and highly non-linear components, the simulation parameters (**Simulate** → **Control Panel** → **SPICE**) can be optimized for a specific design. **Figure 10** illustrates an example.

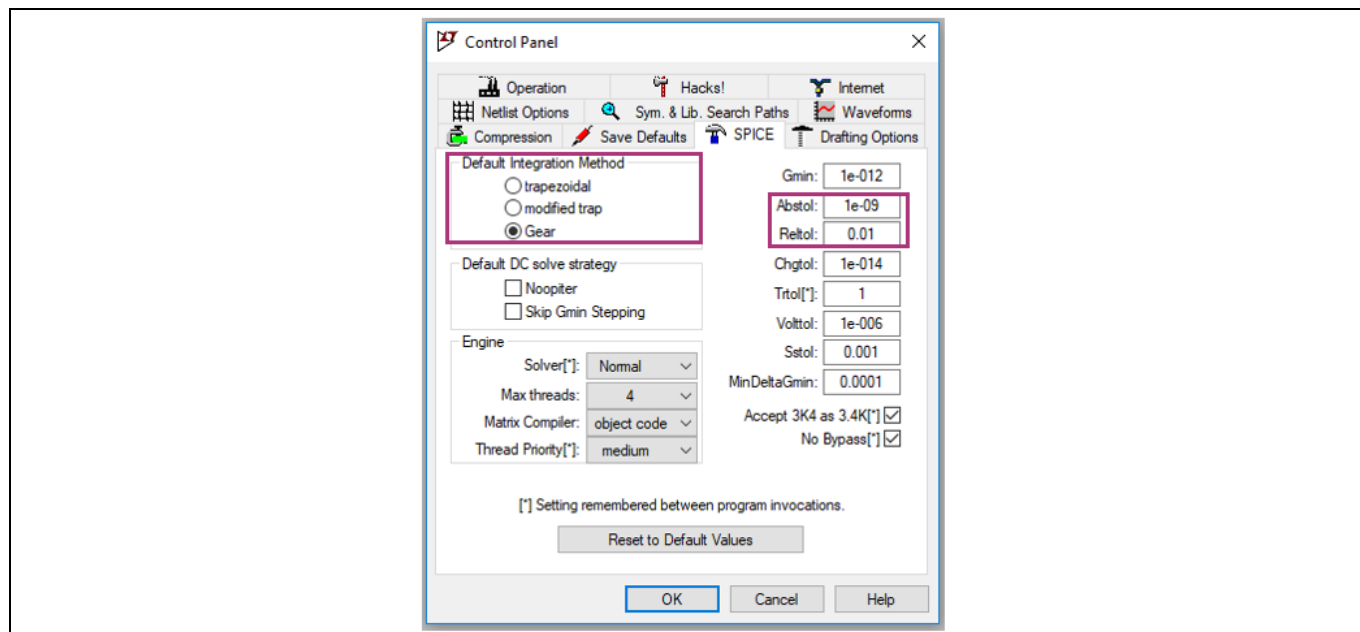


Figure 10 Simulation parameters

Table 1 below provides the recommended values to facilitate convergence.

Table 1 Recommended values for simulation parameters

Reltol=0.01	relative accuracy of voltage and currents
Abstol=1nA	maximum current accuracy
Integration method=Gear	this method is numerically more stable

The delivered models need to detect a rising edge starting from 0V 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.


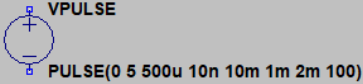
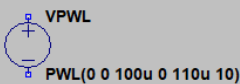



Getting started

Targeted Simulator: LTspice

Starting a new project

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
		
		

Getting started

Targeted Simulator: LTspice

Opening an existing project

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: Extract the archive files.

As shown in **Figure 11**, open **LTspice** application and from **File** menu click **Open**. In the browse window locate and open the **“.asc”** file.

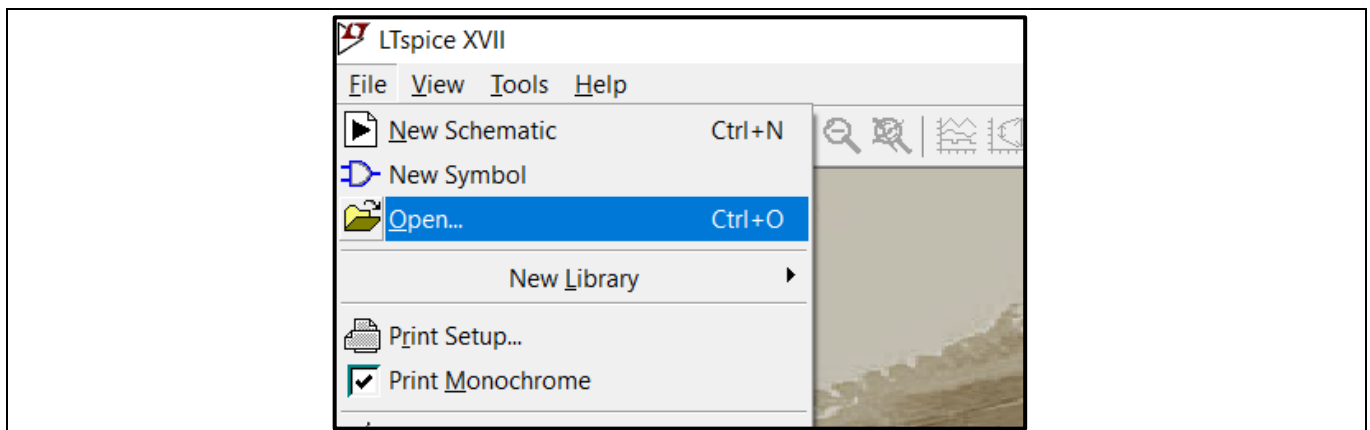


Figure 11 Open an existing project

To start the simulation, open **Simulate** menu and click **Run**. Alternatively, the **Run** button on the LTspice toolbar can be used, as illustrated in **Figure 12**.

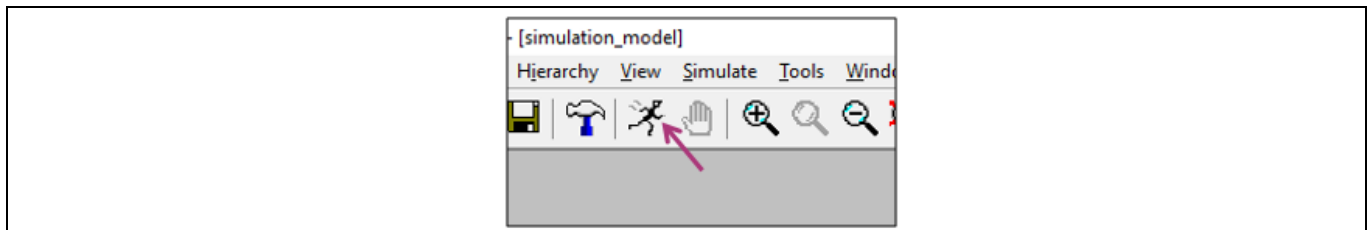


Figure 12 Start the simulation

As **Figure 13** shows, the simulation progress can be monitored in the bottom left corner of the schematic window.

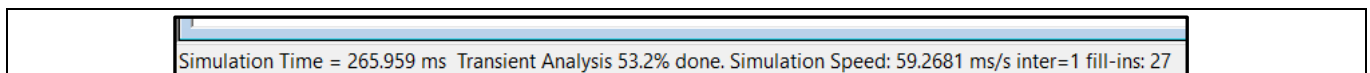


Figure 13 Simulation progress

Getting started

Targeted Simulator: LTspice

Opening an existing project

3.2 Simulation results and measurements

When the simulation starts, the waveform viewer appears automatically and signals can be plotted during and after the simulation stops.

In order to plot the voltage signals, left click on any wire as in [Figure 14](#).

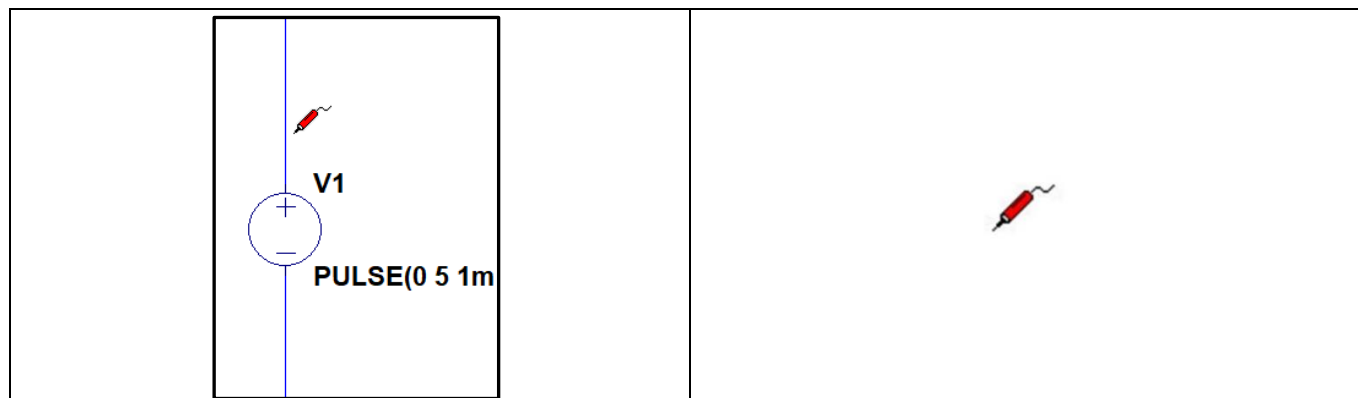


Figure 14 Voltage probe

For current probing, left click on component body or pin as in [Figure 15](#).

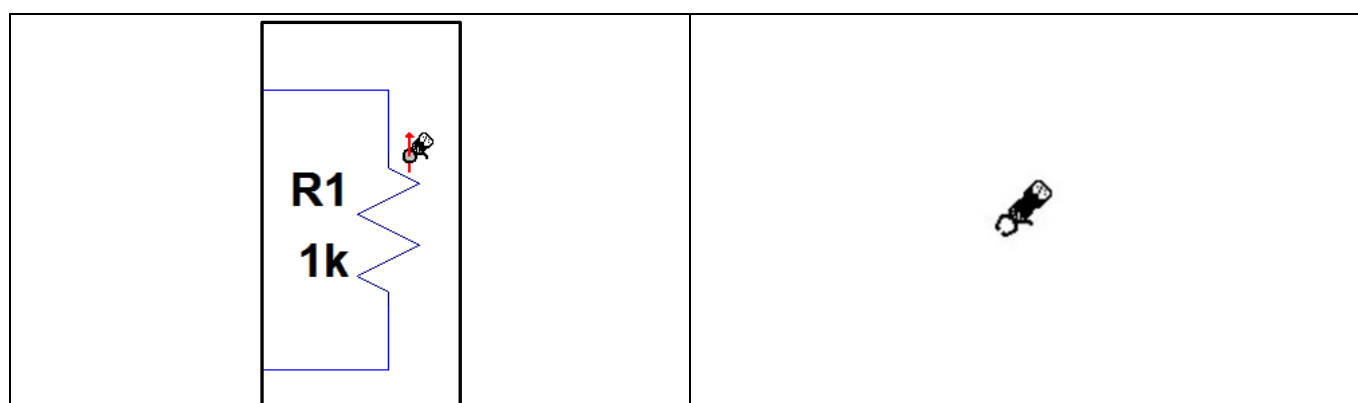


Figure 15 Current probe

[Figure 16](#) shows how to add a new plot window from toolbar: **Plot Settings** → **Add Plot Pane**. Signals can be also added to waveform viewer from toolbar: **Plot Settings** → **Add Trace**.

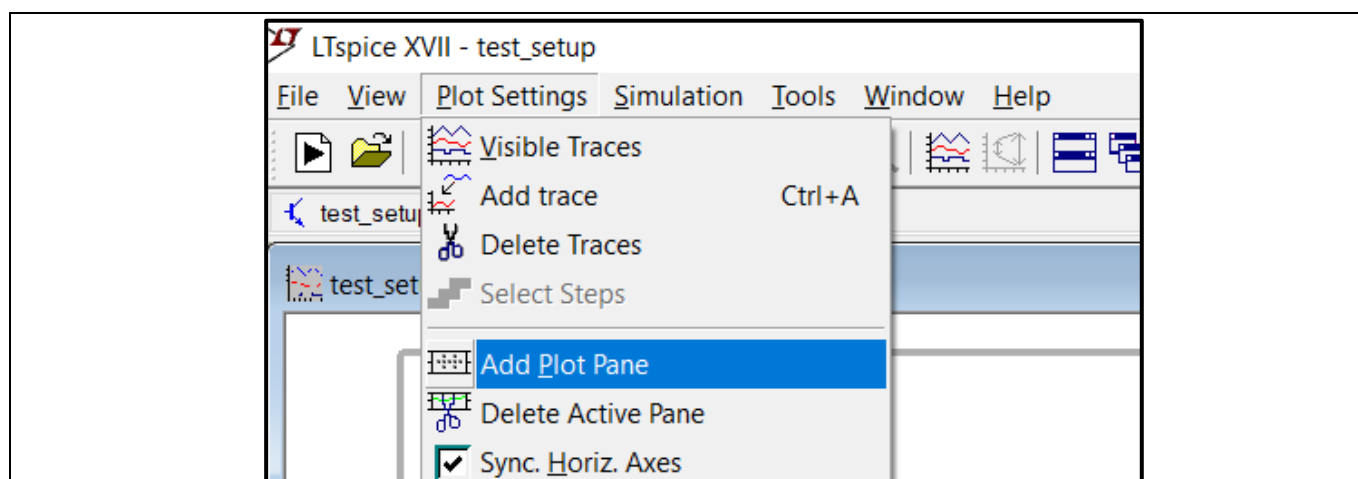


Figure 16 Add a new plot

Getting started

Targeted Simulator: LTspice

Opening an existing project

Add cursors for signal value measurement by right clicking on the desired trace name. **Figure 17** shows an example of a table measurement window.

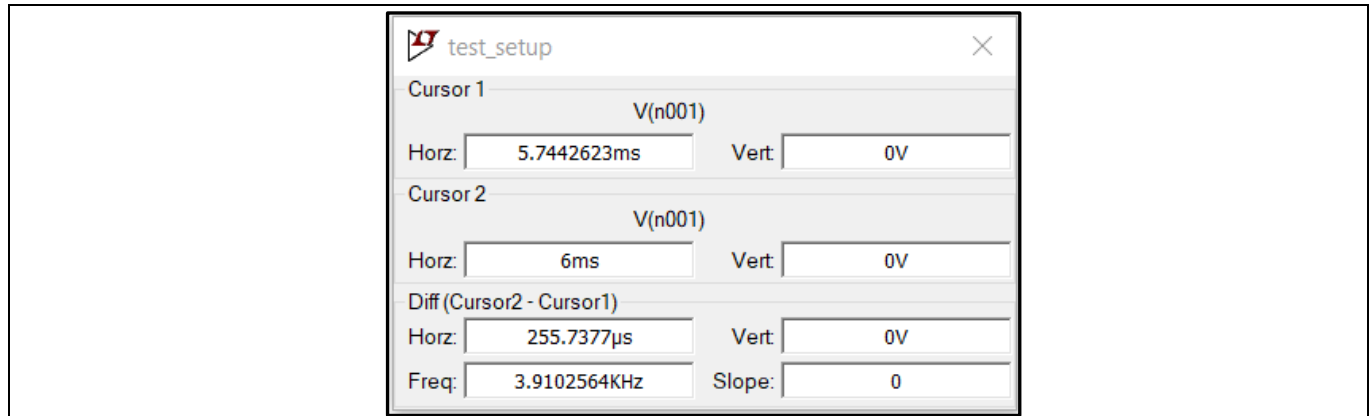


Figure 17 Add cursors

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

Disclaimer

- 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	2021-10-19	Initial version created

Trademarks

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

Edition 2021-10-19

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

Z8F80218330

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.